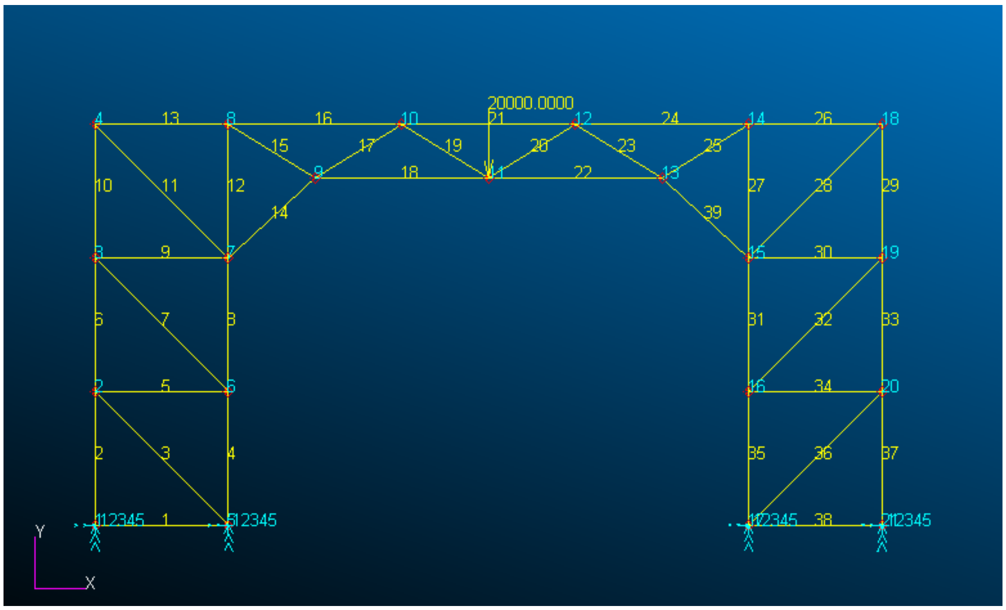
Homework 8 Stress Report



Tiger Sievers

Instructor: Dr.Richard Hale

Department of Aerospace Engineering

May 30, 2025

Executive Summary

This report investigates the structural performance of a two-dimensional truss frame subjected to vertical loading. The geometry, adapted from Curtis [1], was modeled in MSC PATRAN [2] and consists of 39 bar elements connected by pin joints, forming a symmetric frame with pinned supports. The objective is to evaluate member performance under axial loads, considering both material and buckling failure.

Two primary failure criteria were used:

- Maximum allowable axial stress: 300 MPa
- Buckling limit based on Euler's critical load for compressive members

An initial analysis was conducted with a 1 N vertical load at Node 11 to determine the most critical member. Element 21 was identified as the limiting component, governed by compressive stress and potential buckling failure. The structure was then evaluated under an increased load of 20 kN. At this load level, Element 21 reaches a stress corresponding to a margin of safety of 1.5, as shown in Table 0.0.1 , assuming the member retains its original radius of 0.01 m.

To take a new load of 20 kN A redesign was implemented by increasing the radius of Element 21. With a new radius of 0.0473 m, the margin of safety remains at 1.5 under the new loading condition, indicating successful mitigation of buckling risk through geometric modification.

These results confirm that Element 21 governs overall structural performance, with buckling being the controlling failure mode. The structure can safely support larger loads with appropriate resizing of the critical member.

Scenario	Applied Load (N)	Margin of Safety	Bar Radius (m)
Max Load (9.17 kN Load)	9,170	1.5	0.010
Resized for 20 kN	20,000	1.5	0.0473

Table 0.0.1 Summary of Critical Member Analysis (Element 21)

Contents

List of Figures

List of Tables

1. Introduction

This report documents the structural stress analysis of a two-dimensional truss frame using MSC PATRAN[2] for finite element modeling and using NASTRAN for simulation. The primary objective is to determine the maximum vertical load the structure can support without exceeding material or buckling limits, as well as to identify the minimum allowable rod diameter required to support a 20 kN central load.

The structure consists of a symmetric truss with a single vertical force applied at the center of the top joint. All truss members are constructed from an aluminum alloy with a Young's modulus of $E = 70$ GPa and a circular cross-sectional geometry. The original configuration assumes rod diameters of 2 cm. The analysis considers both axial stress and buckling behavior. The allowable axial stress in any member is limited to 300 MPa in either tension or compression. Additionally, compressive members are constrained by Euler buckling, with the critical buckling load defined by:

$$P_{\text{cr}} = \frac{\pi^2 EI}{L^2}$$

where I is the moment of inertia of the circular cross section and L is the member length. The model geometry, material properties, boundary conditions, and loading configuration were defined within MSC PATRAN [2]. The simulation results from NASTRAN were post-processed to evaluate member stresses and determine failure modes. Results from both cases—the maximum allowable load and the minimum diameter for a 20 kN load—are presented and interpreted to ensure the structural integrity of the system under the defined constraints.

2. Geometry

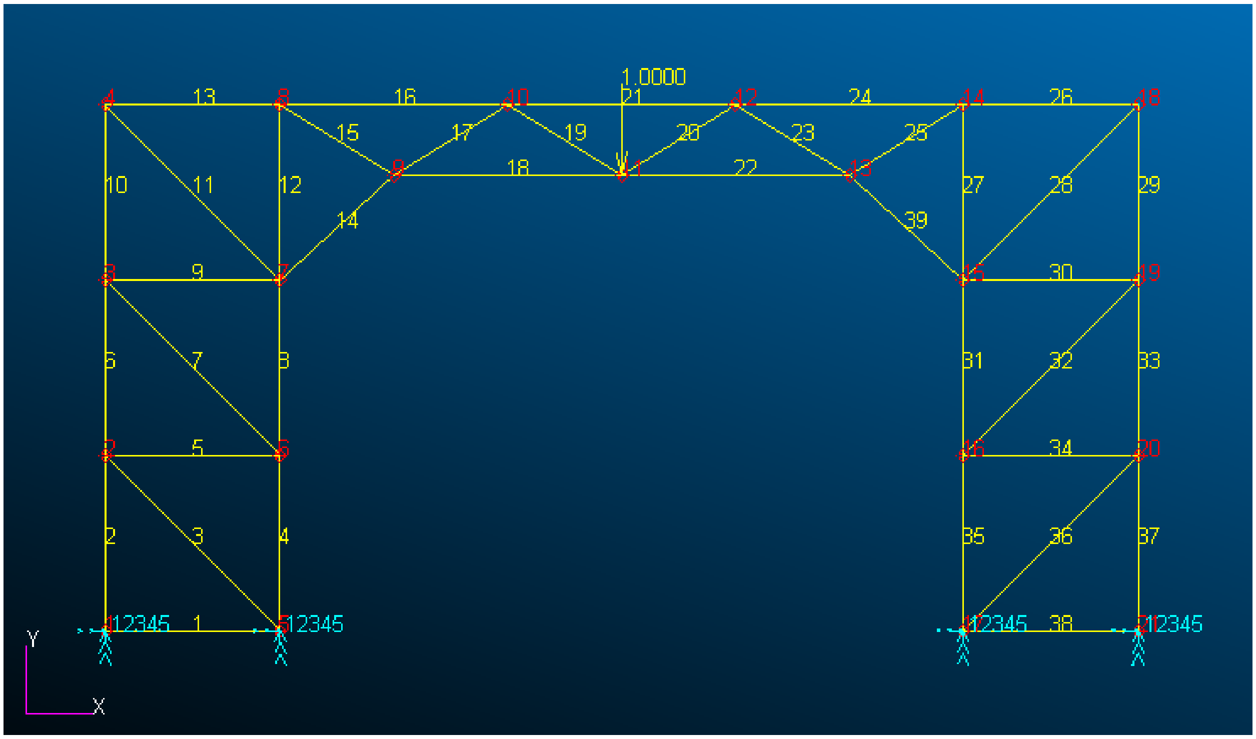


Fig. 2.0.0.1 Problem 10.16 Geometry

The structural geometry, originally adapted from Curtis [1], was manually constructed within MSC PATRAN [2]. The model consists of a symmetric, two-dimensional steel frame with straight truss elements (bar members) connected by pin joints. The system includes two vertical towers supporting an arched roof structure composed of diagonally braced members. The frame features pinned supports at the bases of both vertical towers. Each tower consists of four vertical segments subdivided by three intermediate horizontal members and two sets of diagonal bracing.

Each node and element is numbered for finite element reference, and the entire assembly is symmetric across the y-axis at Node 11.

3. Loading and Boundary Conditions

The truss structure is subjected to a single vertical point load and multiple support constraints to simulate realistic loading behavior. A downward point load P is applied at node 11, acting in the negative y-direction. This load represents the central vertical force used to evaluate both the maximum supported load and the minimum rod diameter required for a 20 kN loading scenario.

Boundary conditions are defined at nodes 1, 8, 17, and 21, where pinned supports are applied. These supports restrict translational degrees of freedom in both the x,y and z directions. Rotational constraints are applied in the x and y axis to constrict the model to two dimesions,and allowing free rotation on the z-axis.

This configuration ensures that the structure's response is governed by axial forces in the members, aligning with classical truss assumptions and allowing for accurate evaluation of internal stresses and buckling criteria using linear static analysis.

4. Materials

The structure is composed of an aluminum alloy modeled as a linear elastic material. It has a Young's modulus of 70 GPa and a Poisson's ratio of 0.3, consistent with typical aluminum behavior. This material assumption enables the use of superposition and linear finite element analysis to evaluate axial stress and buckling performance in truss elements. Yielding is not considered in this analysis, as failure is assumed to occur due to instability (buckling) in compressive members.

5. Finite Element Model

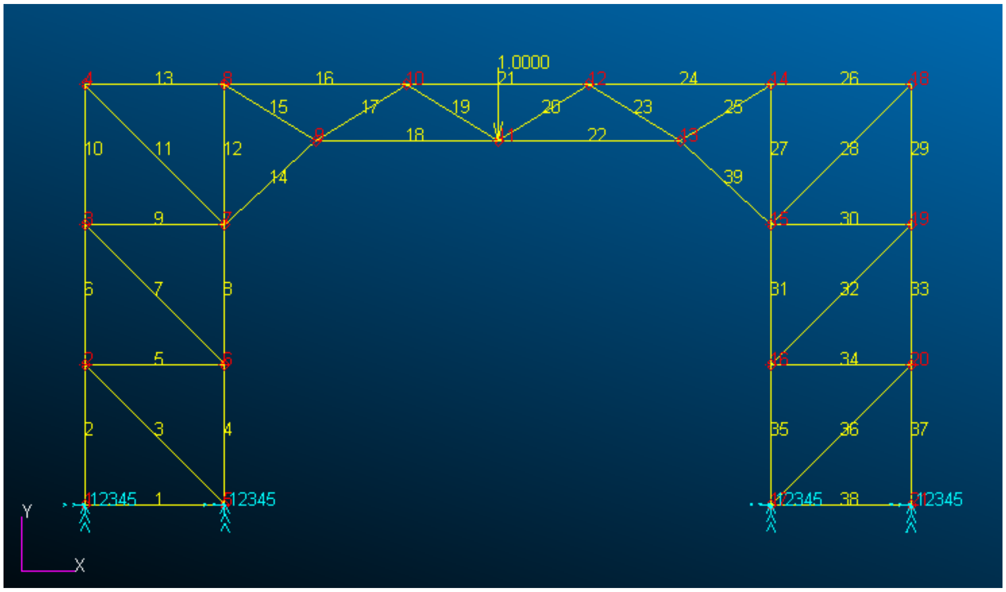


Fig. 5.0.0.1 PATRAN Model

The structural geometry, adapted from Curtis [1], was constructed within MSC PATRAN [2]. The model consists of a symmetric, two-dimensional aluminum frame with straight truss elements (bar members) connected by pin joints. The system includes two vertical towers supporting an arched roof structure composed of diagonally braced members.

The frame features pinned supports at the bases of both vertical towers. Each tower consists of four vertical segments subdivided by three intermediate horizontal members and two sets of diagonal bracing, forming a repetitive X-pattern for stability. The arched roof spans between the tops of the towers and includes a central vertical drop and multiple diagonal braces, forming a series of interconnected triangular units to simulate load path distribution under vertical loads. Each node and element is numbered for finite element reference, and the entire assembly is symmetric horizontally about Node 11.

6. Analysis

6.1 Maximum Load Estimation and Buckling Analysis

To estimate the maximum load the structure can carry before failure, a unit load of 1 N was applied vertically at Node 11. The resulting axial stresses in each member were computed using MSC PATRAN [2]. Results are shown in Table 6.1.1 . Failure is governed by two criteria: an allowable stress limit of 300 MPa (tension or compression) and Euler buckling for compressive members.

#	Axial Stress (Pa)	#	Axial Stress (Pa)
1	0.0	2	-6.654×10^2
3	1.054×10^3	4	-1.672×10^3
5	-7.454×10^2	6	8.007×10^1
7	1.054×10^3	8	-2.418×10^3
9	-7.454×10^2	10	8.255×10^2
11	-1.167×10^3	12	-1.423×10^2
13	8.255×10^2	14	-2.138×10^3
15	2.715×10^2	16	5.943×10^2
17	-3.038×10^3	18	1.248×10^3
19	3.038×10^3	20	3.038×10^3
21	-4.581×10^3	22	1.248×10^3
23	-3.038×10^3	24	5.943×10^2
25	2.715×10^2	26	8.255×10^2
27	-1.423×10^2	28	-1.167×10^3
29	8.255×10^2	30	-7.454×10^2
31	-2.418×10^3	32	1.054×10^3
33	8.007×10^1	34	-7.454×10^2
35	-1.672×10^3	36	1.054×10^3
37	-6.654×10^2	38	0.0
39	-2.138×10^3		

Table 6.1.1 Axial stresses with a 1 N applied load at Node 11

Critical Member – Element 21 Element 21, located at the center of the arch, experiences the highest compressive stress at -4.581×10^3 Pa. As it is in compression, both material and buckling failure must be considered.

Buckling Analysis Assuming a pinned-pinned condition and a circular cross-section of radius $r = 0.01$ m:

$$I = \frac{1}{4}\pi r^4 = 7.854 \times 10^{-9} \text{ m}^4$$
$$A = \pi r^2 = 3.14 \times 10^{-4} \text{ m}^2$$

Euler's critical buckling load is:

$$P_{cr} = \frac{\pi^2 EI}{L^2} \tag{1}$$

Using $E = 70 \times 10^9$ Pa and $L = 1.3$ m:

$$P_{cr} = \frac{\pi^2(70\times10^9)(7.854\times10^{-9})}{(1.3)^2} = 1.38\times10^4\ \text{N}$$
$$\sigma_{cr} = \frac{P_{cr}}{A} = \frac{1.38\times10^4}{3.14\times10^{-4}} = 4.38\times10^7\ \text{Pa}$$

Maximum Load Before Failure Assuming a linear relationship between applied load and internal stress:

$$\frac{P}{1\ \text{N}} = \frac{\sigma_{cr}}{|\sigma_{21}|} = \frac{4.38\times10^7}{4.581\times10^3}$$
$$P = 9.56\times10^3\ \text{N}$$

Thus, the estimated maximum load before buckling occurs in Element 21 is approximately 9.56 kN.

Margin of Safety Adding a margin of safety of 1.5 the calculated maximum load becomes:

$$M.S. = \frac{\sigma_{cr}}{\sigma_{9.17\text{kN}}} - 1$$
$$\Rightarrow M.S. = \frac{4.38\times10^7\ \text{Pa}}{2.92\times10^7\ \text{Pa}} - 1 = 1.5$$
$$\Rightarrow P_{\max} = 9.17\times10^3\ \text{N}$$

6.2 Element Sizing with a Given Load

The applied force was increased to 20 kN at Node 11 to assess the structure under a higher load. The member cross-sectional area remained at $A = 3.14\times10^{-4}\ \text{m}^2$. The resulting axial stresses are shown in Table 6.2.1 .

#	Axial Stress (Pa)	#	Axial Stress (Pa)
1	0.0	2	-1.331×10^7
3	2.108×10^7	4	-3.345×10^7
5	-1.491×10^7	6	1.601×10^6
7	2.108×10^7	8	-4.836×10^7
9	-1.491×10^7	10	1.651×10^7
11	-2.335×10^7	12	-2.846×10^6
13	1.651×10^7	14	-4.276×10^7
15	5.430×10^6	16	1.189×10^7
17	-6.077×10^7	18	2.496×10^7
19	6.077×10^7	20	6.077×10^7
21	-9.162×10^7	22	2.496×10^7
23	-6.077×10^7	24	1.189×10^7
25	5.430×10^6	26	1.651×10^7
27	-2.846×10^6	28	-2.335×10^7
29	1.651×10^7	30	-1.491×10^7
31	-4.836×10^7	32	2.108×10^7
33	1.601×10^6	34	-1.491×10^7
35	-3.345×10^7	36	2.108×10^7
37	-1.331×10^7	38	0.0
39	-4.276×10^7		

Table 6.2.1 Axial stresses with a 20 kN applied load at Node 11

Sizing Element 21 for Buckling Resistance Element 21 reaches compressive stress of -9.162×10^7 Pa. The resulting axial force is:

$$P = \sigma A = (9.162 \times 10^7)(3.14 \times 10^{-4}) = 2.88 \times 10^4 \text{ N}$$

We now solve for the minimum required radius r to avoid buckling. Starting with:

$$\begin{aligned} P_{cr} &= \frac{\pi^2 EI}{L^2} = \left(\frac{\pi^3 E}{4L^2}\right) r^4 \\ \sigma_{cr} &= \frac{P_{cr}}{A} = \left(\frac{\pi^2 E}{4L^2}\right) r^2 = (1.022 \times 10^{11}) r^2 \end{aligned}$$

Setting $\sigma_{cr} = \frac{P}{A} = \frac{2.88 \times 10^4}{\pi r^2}$:

$$\begin{aligned} 1.022 \times 10^{11} r^2 &= \frac{2.88 \times 10^4}{\pi r^2} \\ \Rightarrow r^4 &= \frac{2.88 \times 10^4}{\pi \cdot 1.022 \times 10^{11}} \\ \Rightarrow r = \left(\frac{2.88 \times 10^4}{3.210 \times 10^{11}}\right)^{1/4} &= 0.0173 \text{ m} \end{aligned}$$

To prevent buckling under a 20 kN applied load, Element 21 must have a minimum cross-sectional radius of $r = 0.0173$ m.

Margin of Safety Adding margin of safety of 1.5 the minimum allowable rod radius can be calculated as:

$$\begin{aligned} M.S. &= \frac{\sigma_{cr}}{\sigma_{20\text{kN}}} - 1 \\ \Rightarrow M.S. &= \frac{(1.022 \times 10^{11} r^2) \text{ Pa}}{9.162 \times 10^7 \text{ Pa}} - 1 = 1.5 \\ &\Rightarrow r = 0.0473 \text{ m} \end{aligned}$$

7. Summary

This report presents the structural analysis of a two-dimensional truss frame modeled in MSC PATRAN [2], based on geometry adapted from Curtis [1]. The structure consists of a symmetric frame composed of straight truss members connected by pin joints, including two vertical towers and an arched roof with diagonal bracing. All supports are pinned, and each member is assumed to carry only axial force.

Two primary analyses were performed to assess structural performance:

- **Maximum Load Estimation:** A unit vertical load of 1 N was applied at Node 11 to determine internal axial forces throughout the structure. The resulting stresses were used to identify the critical member (Element 21), which experienced the largest compressive stress of -4.581×10^3 Pa. Using Euler's buckling formula and the known cross-section of radius $r = 0.01$ m, the critical buckling load was calculated as 1.38×10^4 N, corresponding to a critical stress of 4.38×10^7 Pa. Scaling the applied load linearly, the estimated maximum allowable external load before buckling of Element 21 is approximately 9.56 kN.

- **Element Sizing for Given Load:** The analysis was repeated with an applied load of 20 kN at Node 11. Element 21 again experienced the highest compressive stress of -9.162×10^7 Pa. The corresponding internal axial force of 2.88×10^4 N was used to back-calculate the minimum required radius to avoid buckling. Using the relationship between cross-sectional geometry and critical buckling stress, the required radius was found to be 0.0173 m but should be raised to 0.0366 m to meet an increased safety factor.

Overall, the analysis shows that the original member sizing is insufficient to prevent buckling under higher applied loads. For the structure to remain stable under a 20 kN load, critical members such as Element 21 must be resized to meet Euler buckling requirements.

References

[1] Curtis, H. D., *Fundemaentals of Aircraft Structures Analysis*, 1999, Chap. 10.

[2] “MSC Patran,” , 2024.