Project 1 – Finite Element Analysis and Design of a Plane Truss

Problem Statement:

The plane truss in Figure 1 is analyzed using finite element analysis (FEA) for three load cases:

A) Axial load: $F_{x13} = F_{x14} = 10{,}000\ N$
B) Shear load: $F_{y13} = F_{y14} = 1{,}000\ N$
C) Bending moment: $F_{x13} = -F_{x14} = 10{,}000\ N$

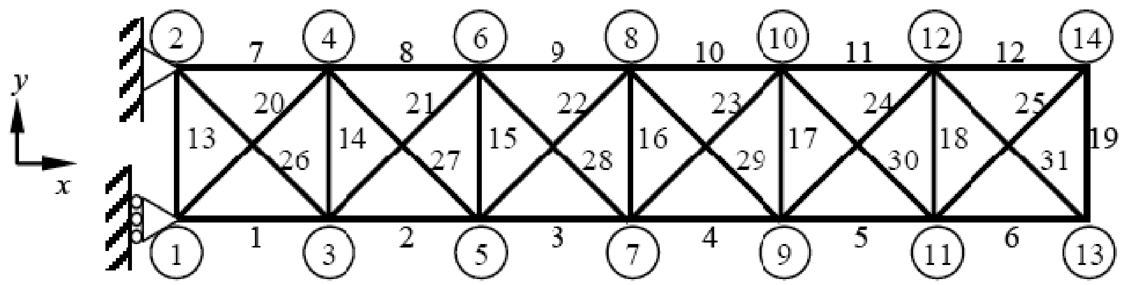

Figure 1. Schematic of plane truss under consideration.

The horizontal and vertical members have length L, while inclined members have length $\sqrt{2}L$. Assume the Young's modulus $E = 100\ GPa$, density $\rho = 7{,}830\,\frac{kg}{m^3}$, cross-sectional area $A = 1.0\ cm^2$, and $L = 0.3\ m$.

An FEA software package (student version of ABAQUS) is used to calculate the nodal displacements, with post-processing performed in MATLAB. The finite element (FE) model is verified by relating the FE model characteristics to that of a cantilever beam. Finally, a fully-stressed design (FSD) optimization is performed to minimize the total weight of the frame. Each element is constrained to have a minimum gage area of $0.1\ cm^2$, and the maximum allowable stress in each element after considering a safety factor is $100\ MPa$.

Part 1: Model Development and Verification

The model geometry, boundary conditions, and loads for each load case were defined in an input file. The nodal displacements for each load case were written to a separate report file by ABAQUS. MATLAB was used to calculate the element forces from the nodal displacements written to the ABAQUS report file. A screenshot from the ABAQUS Visualization window for each load case shows the element stresses superimposed on the displaced frame model are shown in Figure 2-4, below. The prescribed boundary conditions and loads are also superimposed on each figure.

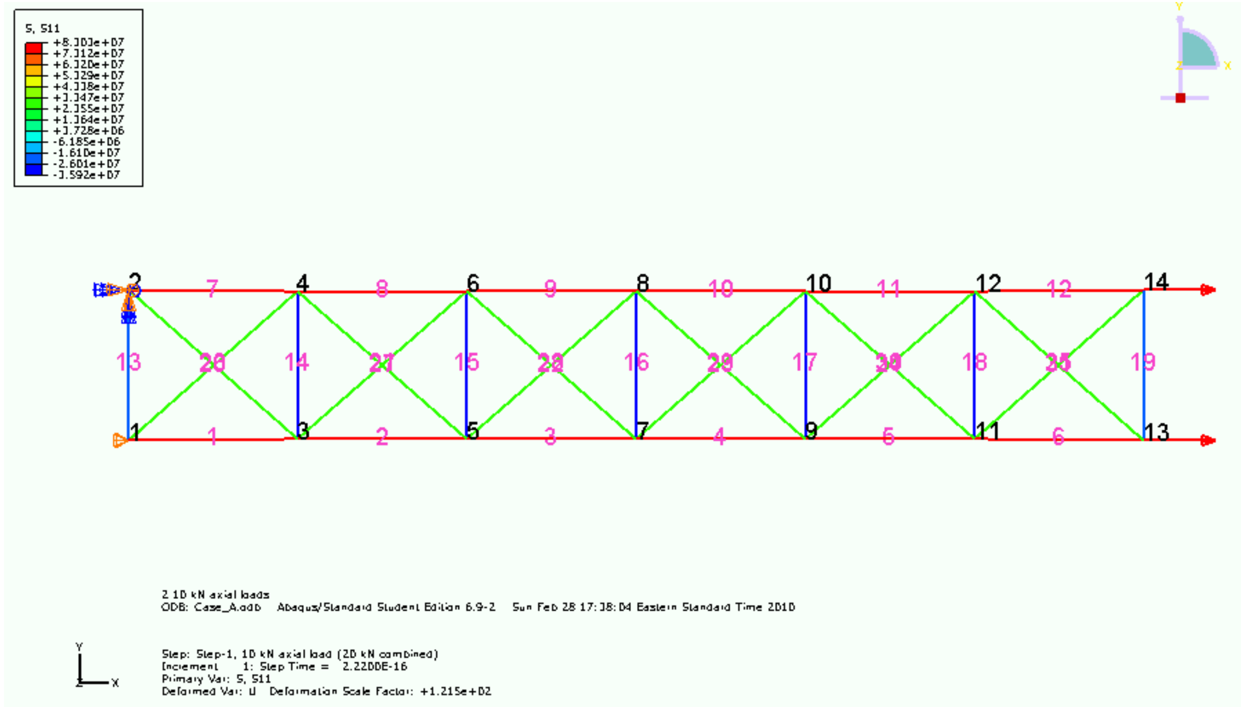

Figure 2. Plot of deformed truss for the axial load conditions for load case A. A 10 kN load is applied in the x-direction at nodes 13 and 14. The essential boundary conditions at nodes 1 and 2 are also shown. The elemental axial stress contours are plotted for the frame.

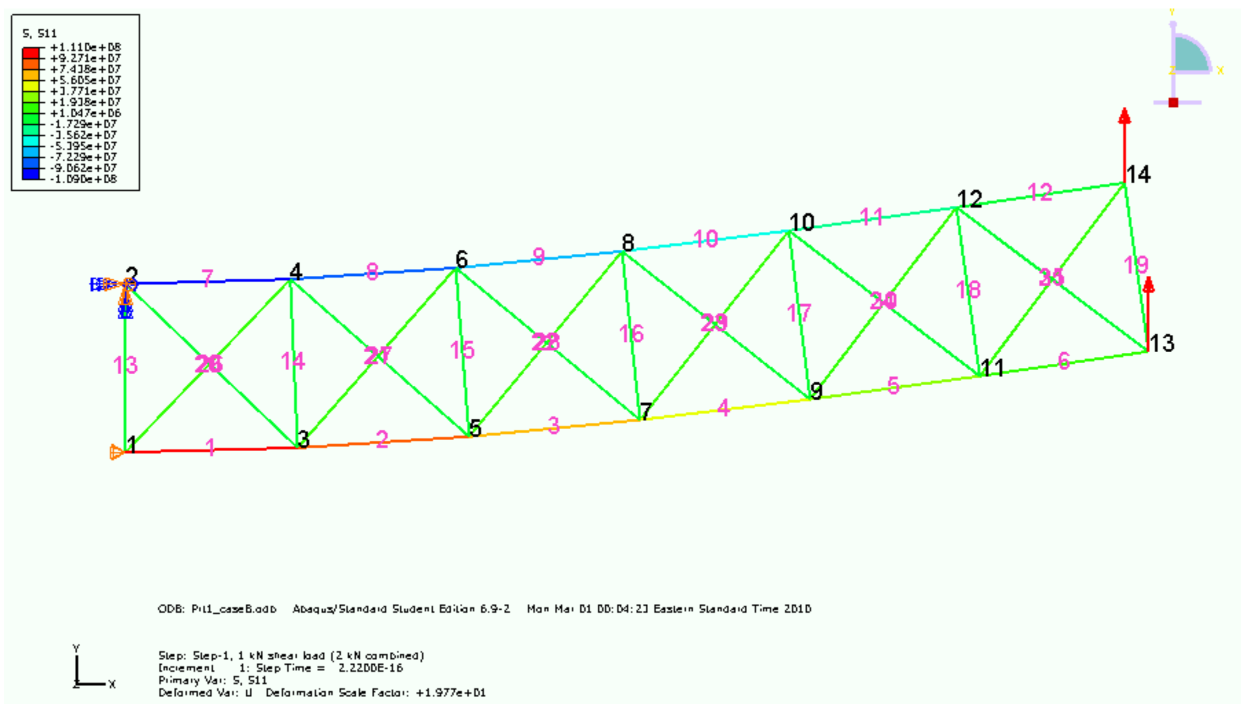

Figure 3. Plot of deformed truss for the shear load conditions for load case B. A 1 kN load is applied in the y-direction at nodes 13 and 14. The essential boundary conditions at nodes 1 and 2 are also shown. The elemental axial stress contours are plotted for the frame.

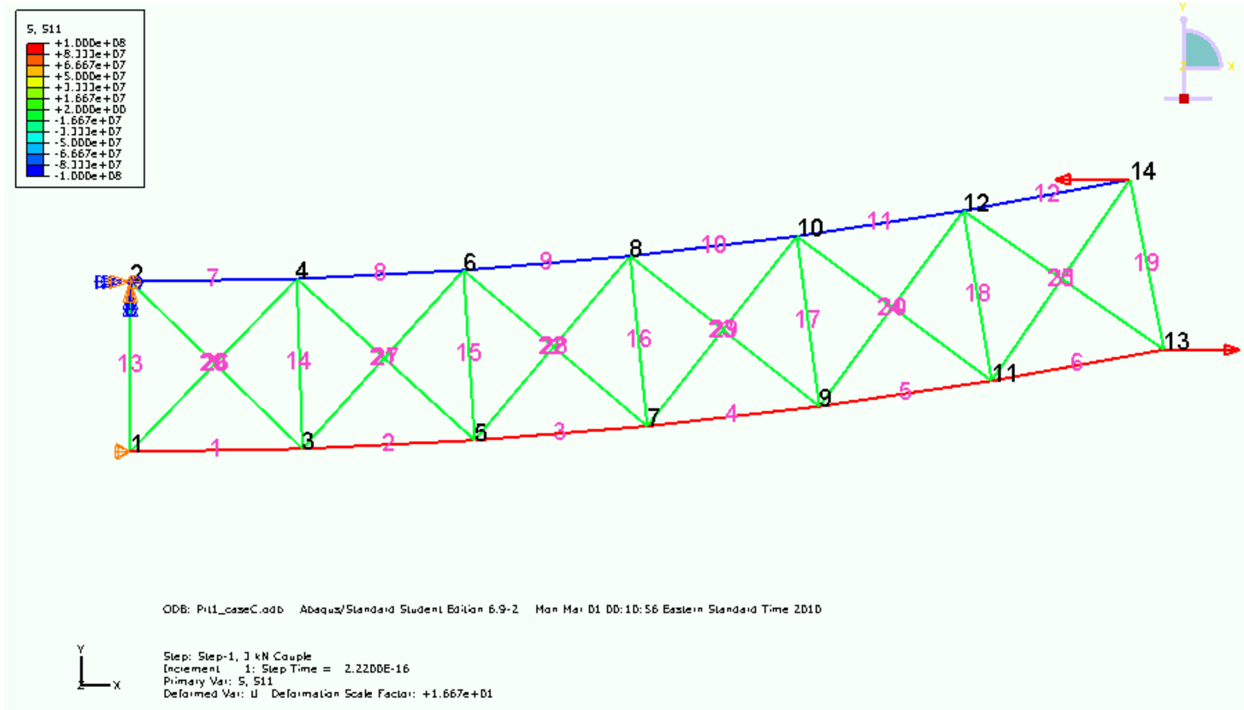

Figure 4. Plot of deformed truss for the shear load conditions for load case C. A 10,000 kN load is applied in the positive and negative x-direction at nodes 13 and 14, respectively. The essential boundary conditions at nodes 1 and 2 are also shown. The elemental axial stress contours are plotted for the frame.

The nodal displacements for each load case are tabulated in Table 1 and the element forces are summarized in Table 2. MATLAB was used to calculate the element forces from the nodal displacements written to the output report from ABAQUS. The element stresses were also output from ABAQUS to compare with the element stresses calculated in MATLAB. MATLAB was used for the element force and stress calculations to make it easier to verify the FE model and perform the FSD optimization in the proceeding sections.

For the axial load case A, the nodal displacements should be mainly in the x-direction, since that is the direction of the applied loads. Also, due to the zero-displacement boundary conditions in the x-direction at x=0, it is expected that the displacement in the x-direction gradually increases to its maximum displacement at the end of the frame where the loads are applied. The loads are also symmetric about the y-axis, so the displacements should also be symmetric about the y-axis. All of these criteria are met for load case A.

The shear load case should be asymmetric about the y-axis since the applied loads are not symmetric about this axis. Also, since the loads are applied in the positive y-direction, frame should deflect in the y-direction with the magnitude of the y-displacements increasing approaching the frame tip where the loads are applied. Looking at the nodal displacement for load case B in Table 1, the model behaves as expected.

Load case C, an applied couple at the free end of the frame, should be antisymmetric about the y-axis to match the antisymmetric loading about this axis. Also, for a pure moment, there should not be any strain generated in the y-direction since there are no forces (i.e. no stress) acting in that direction. Therefore, the nodal displacements in the y-direction should be the same at any given x location. The nodal displacements for this load case in Table 1 show the expected behavior.

Table 1. Nodal displacements for the three load cases. Note that numerical errors resulted in non-zero x-displacements at nodes 1 and 2, and the y-displacement at node 2. However, the magnitude of these calculated displacements is negligible.

Node #	Load Case A		Load Case B		Load Case C	
	U_x [m]	U_y [m]	U_x [m]	U_y [m]	U_x [m]	U_y [m]
1	1.00e-32	5.69e-05	1.20e-32	2.69e-05	1.00e-32	6.64e-19
2	1.00e-32	0.00	-1.20e-32	2.00e-33	-1.00e-32	0.00
3	2.43e-4	8.23e-05	3.33e-4	4.27e-4	3.00e-4	3.00e-4
4	2.43e-4	-2.55e-05	-3.27e-4	4.30e-4	-3.00e-4	3.00e-4
5	4.92e-4	7.96e-05	6.03e-4	1.44e-3	6.00e-4	1.20e-3
6	4.92e-4	-2.28e-05	-5.97e-4	1.44e-3	-6.00e-4	1.20e-3
7	7.41e-4	8.00e-05	8.13e-4	2.94e-3	9.00e-4	2.70e-3
8	7.41e-4	-2.31e-05	-8.07e-4	2.94e-3	-9.00e-4	2.70e-3
9	9.89e-4	7.96e-05	9.63e-4	4.79e-3	1.20e-3	4.80e-3
10	9.89e-4	-2.28e-05	-9.57e-4	4.79e-3	-1.20e-3	4.80e-3
11	1.24e-3	8.23e-05	1.05e-3	6.89e-3	1.50e-3	7.50e-3
12	1.24e-3	-2.55e-05	-1.05e-3	6.89e-3	-1.50e-3	7.50e-3
13	1.48e-3	5.69e-05	1.08e-3	9.10e-3	1.80e-3	1.08e-2
14	1.48e-3	3.43e-12	-1.08e-3	9.10e-3	-1.80e-3	1.08e-2

Table 2. Element forces for each load case. Negative forces indicate that the element is in compression and positive forces indicate the element is in tension.

Element #	Load Case A [kN]	Load Case B [kN]	Load Case C [kN]
1	8.10	11.1	10.0
2	8.30	8.99	10.0
3	8.28	7.00	10.0
4	8.28	5.00	10.0
5	8.30	3.00	10.0
6	8.10	1.00	10.0
7	8.10	-10.9	-10.0
8	8.30	-9.01	-10.0
9	8.28	-7.00	-10.0
10	8.28	-5.00	-10.0
11	8.30	-3.00	-10.0
12	8.10	-1.00	-10.0
13	-1.90	-0.90	-2.21e-11
14	-3.59	0.09	0.00
15	-3.41	-0.01	0.00
16	-3.43	0.00	0.00
17	-3.41	0.00	0.00
18	-3.59	0.00	0.00
19	-1.90	0.00	0.00
20	2.68	1.27	-1.17e-8
21	2.40	1.43	0.00
22	2.43	1.41	7.28e-9
23	2.43	1.41	-1.46e-8
24	2.40	1.41	-1.46e-8
25	2.68	1.41	0.00
26	2.68	-1.56	6.39e-10
27	2.40	-1.40	0.00
28	2.43	-1.42	-7.28e-9
29	2.43	-1.41	1.46e-8
30	2.40	-1.41	1.46e-8
31	2.68	-1.41	0.00

The FE model is verified by comparing the tip deflection of the frame to the tip deflection of a cantilever whose dimensions match the frame. The tip deflection of the cantilever beam for each of the applied loads can beam calculated from shear -deformable beam theory:

$$u_{tip} = \frac{F_a l}{(EA)_{eq}}$$

$$v_{tip} = \frac{F_s l^3}{3(EI)_{eq}} + \frac{F_s l}{(GA)_{eq}}$$

$$v_{tip} = \frac{Cl^2}{2(EI)_{eq}}$$

Where $F_a = 20{,}000\ N$ applied in load case A, $F_s = 2{,}000\ N$ applied in load case B, and $C = 10{,}000\ N \cdot m$ applied in load case C. The beam length for these calculations is the total length of the FE model, which in this case is 1.8 m. The FEA results can be used to calculate the equivalent section properties, $(EA)_{eq}, (EI)_{eq}$, and $(GA)_{eq}$. The tip deflections used in the equations are computed from the average tip deflection between nodes 13 and 14 of the FE model.

From the FEA results in Table 1, the average tip deflection is:

$$u_{tip} = \frac{1}{2}(u_{13} + u_{14}) = \frac{1}{2}(1.48 \times 10^{-3} + 1.48 \times 10^{-3}) = 1.48 \times 10^{-3}\ m$$

Computing the equivalent axial rigidity $(EA)_{eq}$ from the above equation:

$$(EA)_{eq} = \frac{F_a l}{u_{tip}} = \frac{(20000)(1.8)}{1.48 \times 10^{-3}} = 2.43 \times 10^7\ N$$

The average tip deflection for load case C is:

$$v_{tip} = \frac{1}{2}(v_{13} + v_{14}) = \frac{1}{2}(1.08 \times 10^{-2} + 1.08 \times 10^{-2}) = 1.08 \times 10^{-2}\ m$$

The equivalent flexural rigidity $(EI)_{eq}$ is computed from the equation above:

$$(EI)_{eq} = \frac{Cl^2}{2v_{tip}} = \frac{.3(10000)(1.8^2)}{2(1.08 \times 10^{-2})} = 4.50 \times 10^5\ N \cdot m^2$$

The average tip deflection for load case B is:

$$v_{tip} = \frac{1}{2}(v_{13} + v_{14}) = \frac{1}{2}(9.10 \times 10^{-2} + 9.10 \times 10^{-2}) = 9.10 \times 10^{-2}\ m$$

The equivalent shear rigidity $(GA)_{eq}$ is calculated from the above equation and with the previous result for the equivalent flexural rigidity $(EI)_{eq} = 1.50 \times 10^6 N \cdot m^2$:

$$(GA)_{eq} = \frac{F_s l}{v_{tip} - \frac{F_s l^3}{3(EI)_{eq}}} = \frac{(2000)(1.8)}{(9.10 \times 10^{-2}) - \frac{(2000)(1.8^3)}{3(4.50 \times 10^5)}} = 7.78 \times 10^6\ N$$

$$\boxed{\begin{aligned} (EA)_{eq} &= 2.43 \times 10^7\ N \\ \therefore (EI)_{eq} &= 4.50 \times 10^5\ N \cdot m^2 \\ (GA)_{eq} &= 7.78 \times 10^6\ N \end{aligned}}$$

To verify the FE model, two additional truss bays are added to the previous truss previously analyzed. The same load cases A though C were applied and the average tip deflections were calculated. These deflections are then compared to the expected tip displacements calculated from beam theory using the equivalent axial rigidity, flexural rigidity and shear rigidity calculated in the previous step. The comparison of the FEA results and the beam theory calculations are summarized in Table 3, below.

Table 3. Comparison of tip deflections calculated by the extended 8-bay truss FE model and beam deflection equations using the equivalent beam properties calculated from the original 6-bay truss model.

	Load Case A		Load Case B		Load Case C	
	FEA Model	Beam Theory	FEA Model	Beam Theory	FEA Model	Beam Theory
u_{tip}	1.98e-3	1.98e-3	-	-	-	-
v_{tip}	-	-	21.1e-3	21.1e-3	19.2e-3	19.2e-3

Therefore, the FE model accurately predicts the tip deflections for the equivalent cantilever beam.

Part 2: FSD Optimization

The goal of the design optimization is to minimize the total weight of the truss by employing an FSD optimization to minimize the cross sectional area of each truss element. The stresses in the beams were calculated along with the data in Table 1 and 2. Assuming that the load distribution of the frame is not a function of the cross-sectional area of the elements, then the FSD can be determined by minimizing the cross-sectional area of each element until the stress or cross-sectional area is constrained. The new cross section area is calculated using:

$$A_{new} = A_{old}\left|\frac{\sigma}{\sigma_{allowable}}\right|$$

$$\text{such that } A_{new} \geq A_{min}$$

$$\text{where, } A_{min} = 0.1\ cm^2 \text{ and } \sigma_{allowable} = 100\ MPa$$

For each load case the element areas are optimized and the final design will choose the largest area determined among all three load cases. The “old” area for each truss member is 1.0 cm^2. Table 4 below lists the stress and FSD optimized member area. The weight of the original truss is calculated by summing the weights of each individual truss element:

$$truss\ mass = \sum_{i=1}^{31} m_i = \sum_{i=1}^{31} \rho A_i L_i = 8.45\ kg$$

Where the material density $\rho = 7{,}830\ kg/m^3$, and the area and length for each element are defined in the problem statement.

Table 4. Element stresses for each load case and recalculated areas after applying the FSD area calculation for the first design iteration. The critical load cases for each area are highlighted in red.

Element #	Load Case A		Load Case B		Load Case C		Optimal FSD Area
	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	
1	81.0	0.810	111	1.11	100	1.00	1.11
2	83.0	0.830	89.9	0.899	100	1.00	1.00
3	82.8	0.828	70.0	0.700	100	1.00	1.00
4	82.8	0.828	50.0	0.500	100	1.00	1.00
5	83.0	0.830	30.0	0.300	100	1.00	1.00
6	81.0	0.810	10.0	0.100	100	1.00	1.00
7	81.0	0.810	-109	1.09	-100	1.00	1.09
8	83.0	0.830	-90.1	0.901	-100	1.00	1.00
9	82.8	0.828	-70.0	0.700	-100	1.00	1.00
10	82.8	0.828	-50.0	0.500	-100	1.00	1.00
11	83.0	0.830	-30.0	0.300	-100	1.00	1.00
12	81.0	0.810	-10.0	0.100	-100	1.00	1.00
13	-19.0	0.190	-8.95	0.100	0	0.100	0.190
14	-35.9	0.359	0.937	0.100	0	0.100	0.359
15	-34.1	0.341	-96.7e-3	0.100	0	0.100	0.341
16	-34.3	0.343	10.0e-3	0.100	0	0.100	0.343
17	-34.1	0.341	0	0.100	0	0.100	0.341
18	-35.9	0.359	0	0.100	0	0.100	0.359
19	-19.0	0.190	0	0.100	0	0.100	0.190
20	26.8	0.268	12.7	0.127	0	0.100	0.268
21	24.0	0.240	14.3	0.143	0	0.100	0.240
22	24.3	0.243	14.1	0.141	0	0.100	0.243
23	24.3	0.243	14.1	0.141	0	0.100	0.243
24	24.0	0.240	14.1	0.141	0	0.100	0.240
25	26.8	0.268	14.1	0.141	0	0.100	0.268
26	26.8	0.268	-15.6	0.156	0	0.100	0.268
27	24.0	0.240	-14.0	0.140	0	0.100	0.240
28	24.3	0.243	-14.2	0.142	0	0.100	0.243
29	24.3	0.243	-14.1	0.141	0	0.100	0.243
30	24.0	0.240	-14.1	0.141	0	0.100	0.240
31	26.8	0.268	-14.1	0.141	0	0.100	0.268

The new optimal areas are applied to the truss and the element stresses are recalculated for the same 3 load cases using ABAQUS. The resulting element stresses are given in Table 5, below.

Table 5. Element stresses for each load case for the FSD optimization results for the first design iteration. The actively constrained stresses and gage areas are highlighted in red.

Element #	Cross-sectional Area [cm^2].	Element Stresses [MPa]		
		Case A	Case B	Case C
1	1.11	85.2	100	90.1
2	1.00	94.6	89.8	100
3	1.00	94.6	70.0	100
4	1.00	94.6	50.0	100
5	1.00	94.6	30.0	100
6	1.00	94.1	10.0	100
7	1.09	86.8	-99.3	-91.7
8	1.00	94.6	-90.2	-100
9	1.00	94.6	-70.0	-100
10	1.00	94.6	-50.0	-100
11	1.00	94.6	-30.0	-100
12	1.00	94.1	-10.0	-100
13	0.190	-28.3	-43.3	0.283
14	0.359	-30.1	4.49	0.136
15	0.341	-31.7	-0.423	-13.3e-3
16	0.343	-31.6	40.0e-3	0
17	0.341	-31.6	-3.33	-3.33e-3
18	0.359	-31.5	0	0
19	0.190	-31.3	0	0
20	0.268	28.4	43.4	-0.284
21	0.240	31.9	59.9	28.2e-3
22	0.243	31.5	58.1	-2.50e-3
23	0.243	31.5	58.2	-0.500e-3
24	0.240	31.6	58.9	0
25	0.268	31.3	52.8	0
26	0.268	28.4	-62.1	-0.284
27	0.249	31.9	-58.0	28.3e-3
28	0.243	31.5	-58.3	-3.50e-3
29	0.243	31.5	-58.2	-1.00e-3
30	0.240	31.6	-58.9	-1.67e-3
31	0.268	31.3	-52.8	0

The first iteration design is clearly not optimized because neither stress nor area constraints are active for elements 13 through 31. The reason the optimal design was not found in the first iteration is because the assumption that changing the area of an element does not affect the load distribution of the entire truss is not accurate. The element areas will result in a non-linear change in the stiffness matrix, and therefore produce an intermediate, non-optimal design. Therefore, a second iteration of the FSD is performed by calculating new cross-sectional areas for each member. These results are tabulated in Table 6, below.

Table 6. Element stresses for each load case and recalculated areas after applying the FSD area calculation for the second design iteration. The critical load cases for each area are highlighted in red.

Element #	Load Case A		Load Case B		Load Case C		Optimal FSD Area
	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	
1	85.2	0.946	100	1.12	90.1	1.00	1.12
2	94.6	0.946	89.8	0.898	100	1.00	1.00
3	94.6	0.946	70.0	0.700	100	1.00	1.00
4	94.6	0.946	50.0	0.500	100	1.00	1.00
5	94.6	0.946	30.0	0.300	100	1.00	1.00
6	94.1	0.941	10.0	0.100	100	1.00	1.00
7	86.8	0.946	-99.3	1.08	-91.7	1.00	1.08
8	94.6	0.946	-90.2	0.902	-100	1.00	1.00
9	94.6	0.946	-70.0	0.700	-100	1.00	1.00
10	94.6	0.946	-50.0	0.500	-100	1.00	1.00
11	94.6	0.946	-30.0	0.300	-100	1.00	1.00
12	94.1	0.941	-10.0	0.100	-100	1.00	1.00
13	-28.3	0.100	-43.3	0.100	0.283	0.100	0.100
14	-30.1	0.108	4.49	0.100	0.136	0.100	0.108
15	-31.7	0.108	-0.423	0.100	-13.3e-3	0.100	0.108
16	-31.6	0.108	40.0e-3	0.100	0	0.100	0.108
17	-31.6	0.108	-3.33	0.100	-3.33e-3	0.100	0.108
18	-31.5	0.113	0	0.100	0	0.100	0.113
19	-31.3	0.100	0	0.100	0	0.100	0.100
20	28.4	0.100	43.4	0.116	-0.284	0.100	0.116
21	31.9	0.100	59.9	0.144	28.2e-3	0.100	0.144
22	31.5	0.100	58.1	0.141	-2.50e-3	0.100	0.141
23	31.5	0.100	58.2	0.141	-0.500e-3	0.100	0.141
24	31.6	0.100	58.9	0.141	0	0.100	0.141
25	31.3	0.100	52.8	0.141	0	0.100	0.141
26	28.4	0.100	-62.1	0.166	-0.284	0.100	0.166
27	31.9	0.100	-58.0	0.139	28.3e-3	0.100	0.139
28	31.5	0.100	-58.3	0.142	-3.50e-3	0.100	0.142
29	31.5	0.100	-58.2	0.141	-1.00e-3	0.100	0.141
30	31.6	0.100	-58.9	0.141	-1.67e-3	0.100	0.141
31	31.3	0.100	-52.8	0.141	0	0.100	0.141

The new design is analyzed in ABAQUS and the element forces are again computed. The element stresses for the new truss design are tabulated in Table 7, below.

Table 7. Element stresses for each load case for the FSD optimization results for the second design iteration. The actively constrained stresses and gage areas are highlighted in red.

Element #	Cross-sectional Area [cm^2].	Element Stresses [MPa]		
		Case A	Case B	Case C
1	1.12	87.0	101	89.3
2	1.00	97.5	89.4	100
3	1.00	97.5	70.1	100
4	1.00	97.5	50.0	100
5	1.00	97.5	30.0	100
6	1.00	97.1	10.0	100
7	1.08	90.2	-99.1	-92.5
8	1.00	97.5	-90.6	-100
9	1.00	97.5	-69.9	-100
10	1.00	97.5	-50.0	-100
11	1.00	97.5	-30.0	-100
12	1.00	97.1	-10.0	-100
13	0.100	-25.4	-70.8	0.542
14	0.108	-46.9	21.6	0.423
15	0.108	-46.9	-4.32	-66.7e-3
16	0.108	-46.9	0.900	10.0e-3
17	0.108	-46.6	-0.143	-3.33e-3
18	0.113	-48.0	23.3e-3	0
19	0.100	-29.2	0	0
20	0.116	30.9	86.3	-0.660
21	0.144	24.8	104	84.0e-3
22	0.141	25.4	99.1	-14.3e-3
23	0.141	25.4	100	1.50e-3
24	0.141	25.1	100	-1.67e-3
25	0.141	29.3	100	-6.67e-3
26	0.166	21.6	-110	-0.461
27	0.139	25.7	-95.8	85.7e-3
28	0.142	25.2	-101	-12.7e-3
29	0.141	25.4	-100	1.50e-3
30	0.141	25.1	-100	0
31	0.141	29.3	-100	6.67e-3

The second iteration increased the number of actively constrained elements 12 to 22, but there are still 9 elements that are not constrained. The FSD technique is again used to calculate new element areas to use in a third iteration. The calculations are tabulated in Table 8, below.

Table 8. Element stresses for each load case and recalculated areas after applying the FSD area calculation for the third design iteration. The critical load cases for each area are highlighted in red.

Element #	Load Case A		Load Case B		Load Case C		Optimal FSD Area
	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	
1	87.0	0.975	101	1.13	89.3	1.00	1.13
2	97.5	0.975	89.4	0.894	100	1.00	1.00
3	97.5	0.975	70.1	0.701	100	1.00	1.00
4	97.5	0.975	50.0	0.500	100	1.00	1.00
5	97.5	0.975	30.0	0.300	100	1.00	1.00
6	97.1	0.971	10.0	0.100	100	1.00	1.00
7	90.2	0.975	-99.1	1.07	-92.5	1.00	1.07
8	97.5	0.975	-90.6	0.906	-100	1.00	1.00
9	97.5	0.975	-69.9	0.699	-100	1.00	1.00
10	97.5	0.975	-50.0	0.500	-100	1.00	1.00
11	97.5	0.975	-30.0	0.300	-100	1.00	1.00
12	97.1	0.971	-10.0	0.100	-100	1.00	1.00
13	-25.4	0.100	-70.8	0.100	0.542	0.100	0.100
14	-46.9	0.100	21.6	0.100	0.423	0.100	0.100
15	-46.9	0.100	-4.32	0.100	-66.7e-3	0.100	0.100
16	-46.9	0.100	0.900	0.100	10.0e-3	0.100	0.100
17	-46.6	0.100	-0.143	0.100	-3.33e-3	0.100	0.100
18	-48.0	0.100	23.3e-3	0.100	0	0.100	0.100
19	-29.2	0.100	0	0.100	0	0.100	0.100
20	30.9	0.100	86.3	0.100	-0.660	0.100	0.100
21	24.8	0.100	104	0.145	84.0e-3	0.100	0.145
22	25.4	0.100	99.1	0.140	-14.3e-3	0.100	0.140
23	25.4	0.100	100	0.142	1.50e-3	0.100	0.142
24	25.1	0.100	100	0.141	-1.67e-3	0.100	0.141
25	29.3	0.100	100	0.141	-6.67e-3	0.100	0.141
26	21.6	0.100	-110	0.183	-0.461	0.100	0.183
27	25.7	0.100	-95.8	0.133	85.7e-3	0.100	0.133
28	25.2	0.100	-101	0.143	-12.7e-3	0.100	0.143
29	25.4	0.100	-100	0.141	1.50e-3	0.100	0.141
30	25.1	0.100	-100	0.141	0	0.100	0.141
31	29.3	0.100	-100	0.141	6.67e-3	0.100	0.141

After running the FEA analysis in ABAQUS, the element stresses are again calculated. The resulting stresses for the third iteration of the frame design are tabulated in Table 9 for each load case.

Table 9. Element stresses for each load case for the FSD optimization results for the third design iteration. The actively constrained stresses and gage areas are highlighted in red.

Element #	Cross-sectional Area [cm^2].	Element Stresses [MPa]		
		Case A	Case B	Case C
1	1.13	86.4	101	88.6
2	1.00	97.6	89.1	100
3	1.00	97.6	70.2	100
4	1.00	97.6	49.9	100
5	1.00	97.6	30.0	100
6	1.00	97.2	10.0	100
7	1.07	91.2	-99.3	-93.4
8	1.00	97.6	-90.9	-100
9	1.00	97.6	-69.8	-100
10	1.00	97.6	-50.1	-100
11	1.00	97.6	-30.0	-100
12	1.00	97.2	-10.0	-100
13	0.100	-24.1	-63.0	0.776
14	0.100	-48.2	28.0	0.649
15	0.100	-48.6	-6.82	-0.107
16	0.100	-48.9	1.64	20.0e-3
17	0.100	-48.2	-0.507	-3.33e-3
18	0.100	-52.1	83.3e-3	0
19	0.100	-28.4	-33.3e-3	0
20	0.100	34.0	89.0	-1.10
21	0.145	23.5	106	0.123
22	0.140	24.7	98.7	-20.e-3
23	0.142	24.4	100	3.17e-3
24	0.141	23.8	100	-2.58e-13
25	0.141	28.5	100	3.33e-3
26	0.183	18.6	-106	-0.600
27	0.133	25.7	-96.7	0.135
28	0.143	24.1	-101	-20.8e-3
29	0.141	24.6	-99.7	4.67e-3
30	0.141	23.8	-100	-1.67e-3
31	0.141	28.5	-100	-3.33e-3

The third iteration further improved the design by actively constraining all but 3 elements (elements 21, 26, and 27). The FSD technique is applied again to calculate new cross sectional areas which are presented in Table 10, below.

Table 10. Element stresses for each load case and recalculated areas after applying the FSD area calculation for the fourth design iteration. The critical load cases for each area are highlighted in red.

Element #	Load Case A		Load Case B		Load Case C		Optimal FSD Area
	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	
1	86.4	0.976	101	1.14	88.6	1.00	1.14
2	97.6	0.976	89.1	0.891	100	1.00	1.00
3	97.6	0.976	70.2	0.702	100	1.00	1.00
4	97.6	0.976	49.9	0.500	100	1.00	1.00
5	97.6	0.976	30.0	0.300	100	1.00	1.00
6	97.2	0.972	10.0	0.100	100	1.00	1.00
7	91.2	0.976	-99.3	1.06	-93.4	1.00	1.06
8	97.6	0.976	-90.9	0.910	-100	1.00	1.00
9	97.6	0.976	-69.8	0.698	-100	1.00	1.00
10	97.6	0.976	-50.1	0.501	-100	1.00	1.00
11	97.6	0.976	-30.0	0.300	-100	1.00	1.00
12	97.2	0.972	-10.0	0.100	-100	1.00	1.00
13	-24.1	0.100	-63.0	0.100	0.776	0.100	0.100
14	-48.2	0.100	28.0	0.100	0.649	0.100	0.100
15	-48.6	0.100	-6.82	0.100	-0.107	0.100	0.100
16	-48.9	0.100	1.64	0.100	20.0e-3	0.100	0.100
17	-48.2	0.100	-0.507	0.100	-3.33e-3	0.100	0.100
18	-52.1	0.100	83.3e-3	0.100	0	0.100	0.100
19	-28.4	0.100	-33.3e-3	0.100	0	0.100	0.100
20	34.0	0.100	89.0	0.100	-1.10	0.100	0.100
21	23.5	0.100	106	0.154	0.123	0.100	0.154
22	24.7	0.100	98.7	0.138	-20.e-3	0.100	0.138
23	24.4	0.100	100	0.142	3.17e-3	0.100	0.142
24	23.8	0.100	100	0.141	-2.58e-13	0.100	0.141
25	28.5	0.100	100	0.141	3.33e-3	0.100	0.141
26	18.6	0.100	-106	0.194	-0.600	0.100	0.194
27	25.7	0.100	-96.7	0.129	0.135	0.100	0.129
28	24.1	0.100	-101	0.145	-20.8e-3	0.100	0.145
29	24.6	0.100	-99.7	0.141	4.67e-3	0.100	0.141
30	23.8	0.100	-100	0.142	-1.67e-3	0.100	0.142
31	28.5	0.100	-100	0.141	-3.33e-3	0.100	0.141

Using the new cross-sectional areas for the elements calculated in Table 10, the element stresses were recomputed in ABAQUS. The results are shown in Table 11.

Table 11. Element stresses for each load case for the FSD optimization results for the fourth design iteration. The actively constrained stresses and gage areas are highlighted in red.

Element #	Cross-sectional Area [cm^2].	Element Stresses [MPa]		
		Case A	Case B	Case C
1	1.14	85.6	100	87.8
2	1.00	97.6	88.7	100
3	1.00	97.6	70.4	100
4	1.00	97.6	49.9	100
5	1.00	97.6	30.0	100
6	1.00	97.2	9.99	100
7	1.06	92.0	-100	-94.2
8	1.00	97.6	-91.3	-100
9	1.00	97.6	-69.6	-100
10	1.00	97.5	-50.1	-100
11	1.00	97.6	-30.0	-100
12	1.00	97.2	-10.0	-100
13	0.100	-24.4	-60.3	1.05
14	0.100	-48.7	27.0	0.877
15	0.100	-48.7	-8.91	-0.143
16	0.100	-48.9	2.89	23.3e-3
17	0.100	-48.2	-0.517	-3.33e-3
18	0.100	-52.1	0.323	0
19	0.100	-28.4	-66.7e-3	0
20	0.100	34.5	85.2	-1.49
21	0.154	22.3	103	0.159
22	0.138	25.0	98.6	-29.5e-3
23	0.142	24.4	100	4.83e-3
24	0.141	23.8	100	-1.67e-3
25	0.141	28.5	100	-5.00e-3
26	0.194	17.8	-102	-0.766
27	0.129	26.6	-95.7	0.191
28	0.145	23.8	-101.2	-27.5e-3
29	0.141	24.6	-99.4	4.17e-3
30	0.142	23.6	-100	2.56e-13
31	0.141	28.5	-100	5.00e-3

The resulting element stresses are constrained within the requested tolerance for every element, except elements 21 and 27. The cross sectional areas are recomputed using FSD and are shown in Table 12.

Table 12. Element stresses for each load case and recalculated areas after applying the FSD area calculation for the fifth design iteration. The critical load cases for each area are highlighted in red.

Element #	Load Case A		Load Case B		Load Case C		Optimal FSD Area
	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	$\sigma\ [MPa]$	$A_{new}\ [cm^2]$	
1	85.6	0.976	100	1.14	87.8	1.00	1.14
2	97.6	0.976	88.7	0.887	100	1.00	1.00
3	97.6	0.976	70.4	0.704	100	1.00	1.00
4	97.6	0.976	49.9	0.500	100	1.00	1.00
5	97.6	0.976	30.0	0.300	100	1.00	1.00
6	97.2	0.972	9.99	0.100	100	1.00	1.00
7	92.0	0.976	-100	1.06	-94.2	1.00	1.06
8	97.6	0.976	-91.3	0.913	-100	1.00	1.00
9	97.6	0.976	-69.6	0.696	-100	1.00	1.00
10	97.5	0.975	-50.1	0.501	-100	1.00	1.00
11	97.6	0.976	-30.0	0.300	-100	1.00	1.00
12	97.2	0.972	-10.0	0.100	-100	1.00	1.00
13	-24.4	0.100	-60.3	0.100	1.05	0.100	0.100
14	-48.7	0.100	27.0	0.100	0.877	0.100	0.100
15	-48.7	0.100	-8.91	0.100	-0.143	0.100	0.100
16	-48.9	0.100	2.89	0.100	23.3e-3	0.100	0.100
17	-48.2	0.100	-0.517	0.100	-3.33e-3	0.100	0.100
18	-52.1	0.100	0.323	0.100	0	0.100	0.100
19	-28.4	0.100	-66.7e-3	0.100	0	0.100	0.100
20	34.5	0.100	85.2	0.100	-1.49	0.100	0.100
21	22.3	0.100	103	0.159	0.159	0.100	0.159
22	25.0	0.100	98.6	0.136	-29.5e-3	0.100	0.136
23	24.4	0.100	100	0.143	4.83e-3	0.100	0.143
24	23.8	0.100	100	0.141	-1.67e-3	0.100	0.141
25	28.5	0.100	100	0.142	-5.00e-3	0.100	0.142
26	17.8	0.100	-102	0.198	-0.766	0.100	0.198
27	26.6	0.100	-95.7	0.123	0.191	0.100	0.123
28	23.8	0.100	-101.2	0.147	-27.5e-3	0.100	0.147
29	24.6	0.100	-99.4	0.140	4.17e-3	0.100	0.140
30	23.6	0.100	-100	0.142	2.56e-13	0.100	0.142
31	28.5	0.100	-100	0.141	5.00e-3	0.100	0.141

The new areas are entered into the ABAQUS model and the element stresses are computed for the new design. Table 13 summarizes the resulting element stresses.

Table 13. Element stresses for each load case for the FSD optimization results for the fifth design iteration. The actively constrained stresses and gage areas are highlighted in red.

Element #	Cross-sectional Area [cm^2].	Element Stresses [MPa]		
		Case A	Case B	Case C
1	1.14	97.6	100	87.8
2	1.00	97.6	88.4	100
3	1.00	97.6	70.5	100
4	1.00	97.6	49.8	100
5	1.00	97.2	30.1	100
6	1.00	92.0	9.97	100
7	1.06	97.6	-99.9	-94.2
8	1.00	97.6	-91.6	-100
9	1.00	97.5	-69.5	-100
10	1.00	97.6	-50.2	-100
11	1.00	97.2	-29.9	-100
12	1.00	-24.5	-10.0	-100
13	0.100	-48.6	-59.2	1.05
14	0.100	-48.5	25.2	0.882
15	0.100	-48.9	-10.4	-0.143
16	0.100	-48.2	3.67	26.7e-3
17	0.100	-52.2	-1.09	-3.33e-3
18	0.100	-28.4	0.223	0
19	0.100	34.7	-0.333	0
20	0.100	21.4	83.7	-1.49
21	0.154	25.4	102	0.154
22	0.138	24.2	98.5	-29.5e-3
23	0.142	23.8	101	4.67e-3
24	0.141	28.3	99.8	-3.87e-13
25	0.141	17.5	99.9	-1.67e-3
26	0.194	27.7	-101	-0.754
27	0.129	23.5	-98.0	0.198
28	0.145	24.7	-101	-26.0e-3
29	0.141	23.6	-99.4	5.50e-3
30	0.142	28.5	-100	2.56e-13
31	0.141	97.6	-100	1.67e-3

Table 13 shows the final results after 5 iterations of the FSD optimization technique. All elements are actively constrained within the requested stress tolerance of 2 MPa. The final mass of the FSD optimized truss is calculated to be 3.60 kg. Relative to the initial truss mass of 8.54 kg, the FSD optimization results is an overall decrease of 57.9% in the mass of the frame. Table 13 shows the final element areas and the stresses resulting from the three load cases. The load cases where the stress is critical or the element has the minimum allowable gage area are highlighted in red.