Ten-bar Truss Problem

Kuei-Yuan Chan

Contents

Chapter 1

Truss Introduction

By definition, a truss is a structure in which multiple long bars are connected to each other at their ends. Because the specifications such as the number and size of the bars can be changed, the geometric forms that can be combined are very changeable. It is easy to apply to various engineering application fields, so the truss engineering is widely used. It is a very common structure in buildings such as houses and bridges.

The physical model applied when dealing with truss problems usually uses the following assumptions：

- Both external and internal forces act on the ends of the bar.
- The connection between the bars is simplified as a pin connection.
- The influence of the weight of the bar itself is ignored, so each bar is regarded as a two-force member.
- Define the positive direction of the force of each bar as tension, and the negative direction as compression.

Next chapter will take a two-dimensional ten-bar truss as an example, and use the finite element method (FEM) to treat each bar as an individual element and calculate its response to external forces.

Chapter 2

Ten-bar Truss

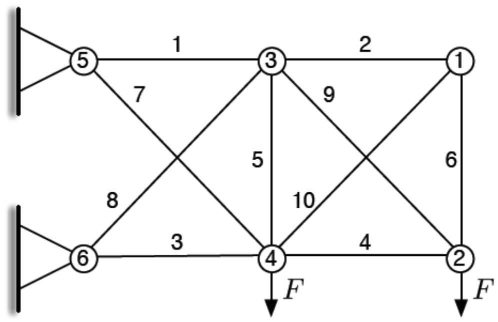

Figure 2.1: Ten bar truss structure diagram

A ten-bar truss is one of the typical truss structures, as shown in Figure 2.1. This chapter uses a ten-bar truss example to demonstrate how to use the finite element method to solve the problem of the truss.

2.1 Problem Definition

Under the following known conditions, given the section radius of the bar, try to find the **displacement**, **stress** and **reaction force** of each bar:

- The overall structure is in static equilibrium.
- All bar cross-sections are circular.
- The material is steel, Young's modulus $E = 200$ GPa, density $\rho = 7860$ kg/m^3, yield strength $\sigma_y = 250$ MPa.
- Both parallel and vertical bars (bars 1 to 6) have lengths of 9.14 m.
- The section radii of bars 1 to 6 are the same as r_1, and the section radii of bars 7 to 10 are the same as r_2.
- All bar radii are optimized between 0.001 and 0.5 m.
- The load F on both nodes 2 and 4 is 1.0×10^7 N down.

2.2 Modeling

The finite element method is applied to the truss, and each bar is regarded as an element, and the connection between elements is a node. The solution process is shown in Figure 2.2. Firstly, the element table is established, and the stiffness matrix is calculated by using the data in the table. Then use the relationship between force, stiffness and displacement to find the required displacement. Finally, stresses and reaction forces can also be calculated from displacements.

The steps for the finite element method are provided below. The calculation process can be assisted by MATLAB software, and this method can be written into a program for execution.

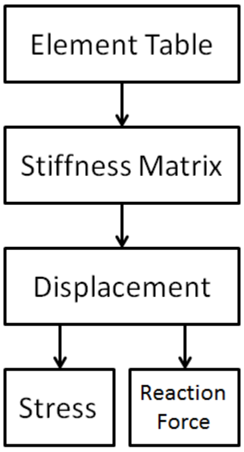

Figure 2.2: Finite Element Method Process

Step 1 Element Table

After numbering each element and node (this section is based on the number in the Figure 2.1), first record the coordinates of each node, which is convenient for calculating the element table:

Table 2.1: Nodal Coordinates

node	x	y
1	18.28	9.14
2	18.28	0
3	9.14	9.14
4	9.14	0
5	0	9.14
6	0	0

First fill in the nodes at both ends of each element in the table, and then calculate the area, length, angle and other information:

The coefficient of elasticity E is known, and the cross-sectional area is calculated by the radius r_1 or r_2 through the circle area formula:

$$A = \pi r^2 \tag{2.1}$$

Table 2.2: Element Table

Element	node i	node j	E	A	L	cos	sin
1	3	5					
2	1	3					
3	4	6					
4	2	4					
5	3	4					
6	1	2					
7	4	5					
8	3	6					
9	2	3					
10	1	4					

Using the node coordinates at both ends of each element, calculate the bar length with Pythagorean theorem:

$$L = \sqrt{(x_{\text{node j}} - x_{\text{node i}})^2 + (y_{\text{node j}} - y_{\text{node i}})^2} \tag{2.2}$$

Divide the x and y coordinate variables of the two end nodes by the length to get the cos and sin values of the angle, respectively:

$$\cos\theta_e = \frac{(x_{\text{node j}} - x_{\text{node i}})}{L} \tag{2.3}$$

$$\sin\theta_e = \frac{(\text{y}_{\text{node j}} - y_{\text{node i}})}{L} \tag{2.4}$$

The complete element table can be obtained by integrating the dimensions and positions of all elements calculated from the formulas 2.1 to 2.4 into the Table 2.2. This table contains all the data needed to calculate the stiffness matrix in the next step.

Step 2 Stiffness Matrix

Each element is connected to two nodes at the end, and the nodes have degrees of freedom

(DOF) in two directions of x and y respectively. The stiffness matrix of 4×4 (represented by $\mathbf{k}$ in the mathematical formula) represents the relationship between displacement and force on the 4 degrees of freedom in the element:

$$\mathbf{k}^e = \frac{EA_e}{L_e} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{2.5}$$

where c represents $\cos\theta_e$; s represents $\sin\theta_e$.

There are 6 nodes in the overall structure of the truss, for a total of $2 \times 6 = 12$ degrees of freedom. Take node1 in x direction as DOF1, node1 in y direction as DOF2, node2 in x direction as DOF3, etc. The 12 degrees of freedom are numbered in the order of node1 to node6 and in the order of x and y. Substituting the data shown in the element table into the formula 2.5 to create the stiffness matrix of each element. Take elements 2 and 6 as an example:

$$\mathbf{k}^2 = \frac{EA_2}{9.14} \begin{array}{c} \begin{array}{cccc} 1 & 2 & 5 & 6 \end{array} \quad \leftarrow\downarrow \text{Global DOF} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 5 \\ 6 \end{array} \end{array} \tag{2.6}$$

$$\mathbf{k}^6 = \frac{EA_6}{9.14} \begin{array}{c} \begin{array}{cccc} 1 & 2 & 3 & 4 \end{array} \quad \leftarrow\downarrow \text{Global DOF} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array} \end{array} \tag{2.7}$$

All 10 elements are substituted into formula 2.5 with reference to the element table. After deriving the stiffness matrix, according to the corresponding degrees of freedom, it is integrated into the overall stiffness matrix $\mathbf{K}$ of 12×12:

$$\mathbf{K} \leftarrow \sum_e \mathbf{k}^e \tag{2.8}$$

Step 3 Displacement

In order to use the relationship between force, stiffness and displacement:

$$\mathbf{F} = \mathbf{KQ} \tag{2.9}$$

According to Step2, set the 12 degrees of freedom of the overall system. Creates a 12×1 force vector and a 12×1 displacement vector corresponding to the 12 degrees of freedom.

$$\mathbf{F} = \begin{bmatrix} F_1 & F_2 & ... & F_{12} \end{bmatrix}^T \tag{2.10}$$

$$\mathbf{Q} = \begin{bmatrix} Q_1 & Q_2 & ... & Q_{12} \end{bmatrix}^T \tag{2.11}$$

Boundary conditions are applied, node5 and node6 are fixed ends, and there is no displacement in x and y directions:

$$Q_9 = Q_{10} = Q_{11} = Q_{12} = 0 \tag{2.12}$$

Since there is no displacement from DOF9 to DOF12, the corresponding rows and columns in the stiffness matrix $\mathbf{K}$, force vector $\mathbf{F}$, and displacement vector $\mathbf{Q}$ can be eliminated. The stiffness matrix $\mathbf{K}$ intercepts the 1st row to the 8th row, the 1st column to the 8th column, and simplifies it into $\mathbf{K}_{reduced}$ of 8×8; the force vector $\mathbf{F}$ and displacement vector $\mathbf{Q}$ only take the first 8, and simplify it into $\mathbf{F}_{reduced}$ and $\mathbf{Q}_{reduced}$ of 8×1:

$$\mathbf{K}_{\text{reduced}} = \begin{bmatrix} K_{1,1} & K_{1,2} & \cdots & K_{1,8} \\ K_{2,1} & K_{2,2} & \cdots & K_{2,8} \\ \vdots & \vdots & \ddots & \vdots \\ K_{4,1} & K_{8,2} & \cdots & K_{8,8} \end{bmatrix} \tag{2.13}$$

$$\mathbf{F}_{\text{reduced}} = \begin{bmatrix} F_1 & F_2 & ... & F_8 \end{bmatrix}^T \tag{2.14}$$

$$\mathbf{Q}_{\text{reduced}} = \begin{bmatrix} Q_1 & Q_2 & ... & Q_8 \end{bmatrix}^T \tag{2.15}$$

Using the formula 2.9, after multiplying $\mathbf{K}^{-1}_{reduced}$ together, the displacement of each node in the x and y directions can be calculated:

$$\begin{aligned} \mathbf{K}_{\text{reduced}}\mathbf{Q}_{\text{reduced}} &= \mathbf{F}_{\text{reduced}} \\ \mathbf{Q}_{\text{reduced}} &= \mathbf{K}^{-1}_{\text{reduced}}\mathbf{F}_{\text{reduced}} \end{aligned} \tag{2.16}$$

In summary, Q_1 to Q_8 are obtained by the formula 2.16, and Q_9 to Q_{12} are zero.

Step 4 Stress

The relationship between stress and strain is as follows:

$$\sigma = E \times \varepsilon \tag{2.17}$$

where σ is stress; ε is strain; E is Young's modulus.

Strain is defined as:

$$\varepsilon = \frac{\delta}{L} \tag{2.18}$$

where δ is the length variable.

The length variable of each element can be obtained by transforming the displacement of the node in each direction with a trigonometric function:

$$\boldsymbol{\sigma} = \frac{E_e}{l_e} \begin{bmatrix} -c & -s & c & s \end{bmatrix} \mathbf{Q} \tag{2.19}$$

Step 5 Reaction Force

In this structure, the reaction force will be generated at the fixed ends node5 and node6, and the corresponding degrees of freedom are DOF9 to DOF12, which can be calculated through the displacement of all nodes in the system. Therefore, we take columns 9 to 12 in the stiffness matrix:

$$\mathbf{K}_{\text{reaction}} = \begin{bmatrix} K_{9,1} & K_{9,2} & \cdots & K_{9,12} \\ K_{10,1} & K_{10,2} & \cdots & K_{10,12} \\ K_{11,1} & K_{11,2} & \cdots & K_{11,12} \\ K_{12,1} & K_{12,2} & \cdots & K_{12,12} \end{bmatrix} \tag{2.20}$$

The reaction force can be obtained by multiplying the matrix of the formula 2.20 by the displacement matrix $\mathbf{Q}$:

$$\begin{Bmatrix} R_9 \\ R_{10} \\ R_{11} \\ R_{12} \end{Bmatrix} = \mathbf{K}_{\text{reaction}} \mathbf{Q} \tag{2.21}$$

The above is the flow of the finite element method for the ten-bar truss. Under the conditions of given design variables r_1 and r_2, the displacement, stress and reaction force of each bar can be calculated through the formula provided in 5 steps. The displacement from the formula 2.16, the stress from the formula 2.19 and the reaction force from formula 2.21 are the desired.

2.3 Problem Formulation

The objective function of the problem is set to minimize the weight of the structure. The stress is constrained to $\sigma_y = 250$ MPa and only the displacement at node 2 are restricted to less than 0.02 m. Let the design variables be the section radii of all 10 bars.

Mathematical expression of optimization problem:

$$
\begin{aligned}
\min_{r_1,\, r_2} \quad & f(r_1, r_2) = \sum_{i=1}^{6} m_i(r_1) + \sum_{i=7}^{10} m_i(r_2) \\
\text{subject to} \quad & |\boldsymbol{\sigma}_i| \leq \sigma_y \\
& \Delta s_2 \leq 0.02 \\
\text{where} \quad & f : \text{mass of all bars} \\
& \Delta s_2 : \text{displacement of node 2} \\
& \sigma_y : \text{yield stress} \\
& \boldsymbol{\sigma}_i : \text{stress in each bar}
\end{aligned}
$$

2.4 Problem Solution

After optimization using `fmincon`, its best value and best solution can be obtained:

$$(r_1, r_2) = (0.3, 0.2663) \qquad\qquad f = 212410$$

Its design space, feasible solution space and objective function value are represented as shown in Figure 2.3. From Figure 2.3, it is obvious that the active constraint is the one about displacement, which is the 11^{th} constraint in the program.

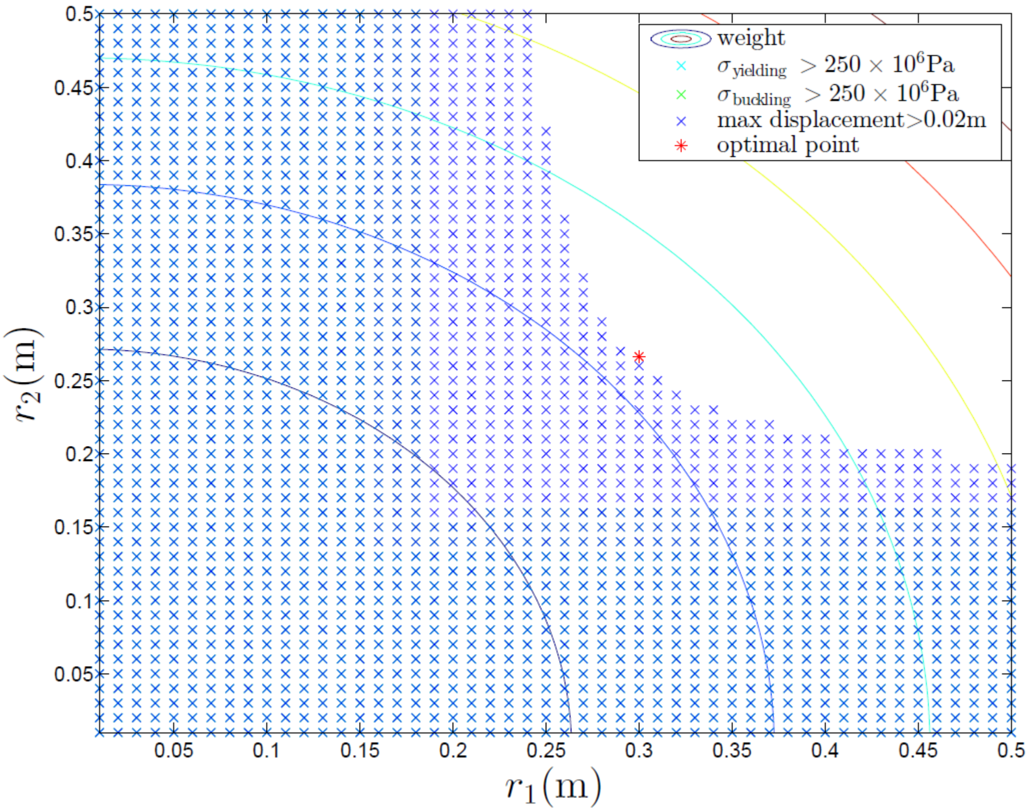

Figure 2.3: 10-bar truss design space

2.5 Uncertainty Analysis

Due to manufacturing variations, all bars r_i follows Gaussian distribution as $N(\mu, 0.0052)$. Due to material variation, $E \sim N(200, 20)$ GPa. What are the probability of violating active constraints when uncertainties in manufacturing and materials are considered?

We use Monte-Carlo with 1 million samples to calculate the probability. Therefore, considering the manufacturing variations and material variation, we generate multivariate normal random numbers (mvnrnd) of all bars radii and Young's modulus to analyze.

The uncertainty analysis shows that at the optima we found above, five of the constraints have more than 1% probability to be violated under the uncertainties. The five constraints are stress of element 1, stress of element 3, stress of element 7, stress of element 8, and displacement of node 2 (active in previous problem), with failure probabilities of 2.37%, 2.81%, 3.48%, 3.25% and 58.73%, respectively.

2.6 Design under Uncertainty

The following part is using Monte-Carlo to solve the probabilistic design problem, and use Monte-Carlo to verify the failure probability at the optimal. Constraints have to be satisfied 99% of the time.

It is necessary that creat a new constraint function with the consideration of the effects of random variability. Therefore, the new constraints are that the probability of violating constraints needs to be less than 0.01, which means 99% reliability.

However, does `fmincon` with Monte-Carlo or `ga` with Monte-Carlo work? The answer to the question is "NO" or "DIFFICULT" because we need large enough samples used in Monte-Carol so that the solution can be found. But if the samples are too large, the simulation time is too long!!! Thus, we need to use another method, FOSM, to deal with the problem.

Bibliography

[1] Tirupathi R. Chandrupatla, Ashok D. Belegundu, Introduction to Finite Elements in Engineering, 3rd ed. Pearson Education, 2002.