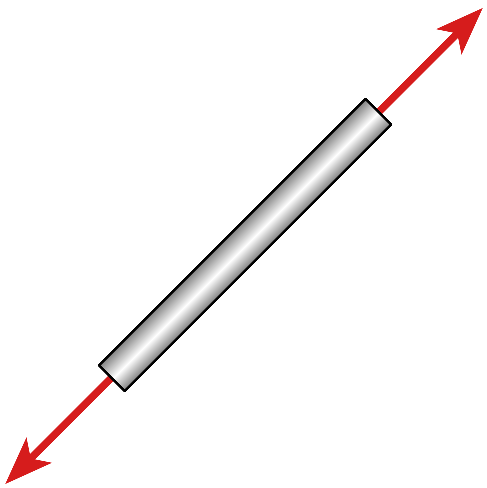
CEE321—Structural Analysis and Design

Structural Analysis with the Direct Stiffness Method

Modeling Structural Systems as Springs



Arizona State University

Brian Chevalier

Updated: April 20, 2020

Contents

1 Truss Direct Stiffness Method

The *Direct Stiffness Method* is a *Finite Element Method* of analysis that models structural elements as springs. We will begin with deriving the axial stiffness of a structural element.

1.1 Axial Stiffness

Members in a truss are modeled as springs with only axial deformation. The axial force in the spring is modeled by Hooke's law:

$$N = ku \tag{1}$$

where N is the axial force, k is the axial stiffness, and u is the axial deformation. To define the axial stiffness we will look at a member of constant cross-sectional area with only elastic deformation. The stress in such a member is defined by the stress-strain relationship:

$$\sigma = E\varepsilon \tag{2}$$

where σ is the stress, E is the modulus of elasticity, and ε is the strain. Multiplying both sides of the equation by the cross-sectional area, A, will convert the stress into a force.

$$\sigma A = E\varepsilon A \tag{3}$$

$$N = E\varepsilon A \tag{4}$$

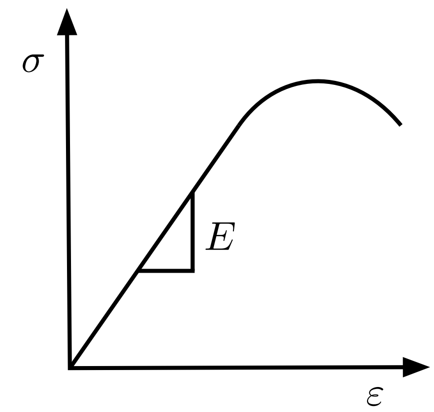


Figure 1: Stress-Strain Relationship

The engineering definition of strain will be used here. Strain is defined as:

$$\varepsilon = \frac{\Delta L}{L_0} = \frac{u}{L} \tag{5}$$

where L is the original length of the member and ΔL is the change in length of the member. Then substituting in the definition of engineering strain into the previous equation:

$$N = EA\frac{u}{L} \tag{6}$$

Therefore the axial stiffness from the equation is $\frac{AE}{L}$ for any structural member. The axial force of a structural element can be rewritten as:

$$N = \frac{AE}{L} u \tag{7}$$

1.2 Local Element Stiffness

Each element in a structural system will have its own local element stiffness equations. The goal of this section will be to develop the local element stiffness equations for a member in matrix form. Taking the nodal equilibrium from figure 2 becomes:

$$-N = ku_{1x'}$$
$$-N = ku_{2x'}$$

Next we will put these equations in matrix form in terms of the local coordinate system. The equation will take the form:

$$\mathbf{N} = \mathbf{ku} \tag{8}$$

$$\underbrace{\begin{Bmatrix} -N \\ 0 \\ \hline N \\ 0 \end{Bmatrix}}_{\mathbf{N'}} = \underbrace{\frac{AE}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}}_{\mathbf{k}_{local}} \underbrace{\begin{Bmatrix} u_{1x'} \\ u_{1y'} \\ \hline u_{2x'} \\ u_{2y'} \end{Bmatrix}}_{\mathbf{u'}} \tag{9}$$

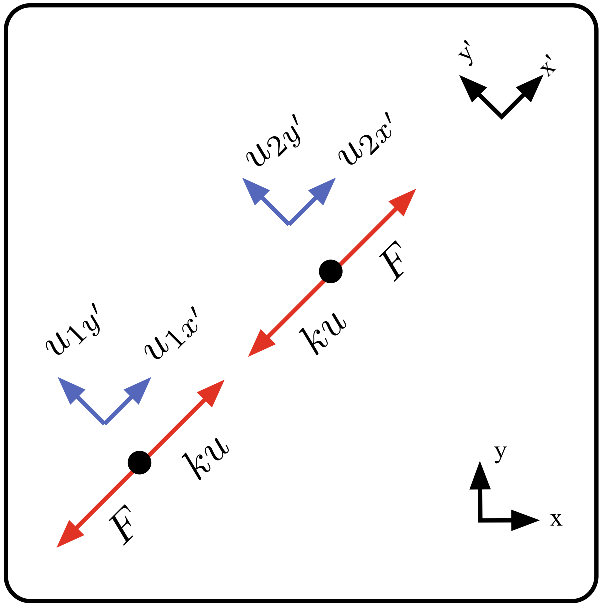


Figure 2: Local Member FBDs

Next, we must take the local element stiffness matrix and derive the global element stiffness matrix.

1.3 Global Element Stiffness

Transforming local coordinates to global coordinates is described by the following equation:

$$\begin{Bmatrix} u_x \\ u_y \end{Bmatrix} = \underbrace{\begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix}}_{\mathbf{R}} \begin{Bmatrix} u'_x \\ u'_y \end{Bmatrix} \tag{10}$$

where $\mathbf{R}$ is the general rotation matrix. The particular rotation matrix for the two degree of freedom system takes the form:

$$\mathbf{T} = \begin{bmatrix} \mathbf{R} & \mathbf{0} \\ \mathbf{0} & \mathbf{R} \end{bmatrix} \tag{11}$$

Note that the values of $\cos\theta$ and $\sin\theta$ can be determined by solving for the x and y component of the element unit vector, respectively. The unit vector can be calculated by:

$$\mathbf{n} = \frac{\mathbf{x_2} - \mathbf{x_1}}{||\mathbf{x_2} - \mathbf{x_1}||} \tag{12}$$

where $\mathbf{n}$ is the unit vector along the element, and $\mathbf{x}_2$ and $\mathbf{x}_1$ are the coordinates of node 1 and node 2, respectively. Now, we take equation 9, and multiply both sides of the equation by the local to global transformation matrix, then solve for the axial forces $\mathbf{N}'$. Note that the global deformation, $\mathbf{u}$ is equal to $\mathbf{T}^\intercal\mathbf{u}$

$$\mathbf{N}' = \mathbf{k}\mathbf{u}' \tag{13}$$

$$\mathbf{T}\mathbf{N}' = \mathbf{k}\mathbf{T}^\intercal\mathbf{u} \tag{14}$$

$$\mathbf{N}' = \underbrace{\mathbf{T}\mathbf{k}\mathbf{T}^\intercal}_{\mathbf{k}_{global}}\mathbf{u} \tag{15}$$

This gives us the relationship between the global deformation of nodes to the axial force. Enforcing the global nodal boundary conditions will allow solving the system of equations for the member forces. Expanding the equation for $\mathbf{k}_{global}$ looks like:

$$\mathbf{k}_{global} = \frac{AE}{L}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \tag{16}$$

where l and m are $\cos\theta$ and $\sin\theta$, respectively. Notice that the $\mathbf{k}_{global}$ matrix is symetric about the main diagonal.

1.4 Global Structural Stiffness

Deriving the global structural stiffness matrix for a truss is a matter of combining the global element stiffness matricies.

Each node has the ability to translate in the x direction and translate in the y direction. Each of these nodes is said to have 2 degrees of freedom. In order to assemble the global structural stiffness matrix, each degree of freedom must be numbered at each node. The degree of freedom number is a function of the node number. Table 1 builds the relationship between the nodal number and the associated DoF number for the x and y direction.

Table 1: Degree of Freedom Counting

Node Number	x	y
0	0	1
1	2	3
2	4	5
i	$2i$	$2i+1$

Assembly. Now that the DoF counting has been determined, the rows and columns of the global element stiffness matrix are labeled with the associated start node and end node DoF numbers. This is shown in the following equation:

$$\mathbf{k}_{global} = \begin{array}{cccc|l} 2i & 2i+1 & 2j & 2j+1 & \\ \end{array}$$

$$\mathbf{k}_{global} = \begin{array}{c} \\ \end{array} \begin{array}{c} \begin{array}{cccc} 2i & 2i+1 & 2j & 2j+1 \end{array} \\ \begin{bmatrix} \bullet & \bullet & \bullet & \bullet \\ \bullet & \bullet & \bullet & \bullet \\ \bullet & \bullet & \bullet & \bullet \\ \bullet & \bullet & \bullet & \bullet \end{bmatrix} \end{array} \begin{array}{l} \\ 2i \\ 2i+1 \\ 2j \\ 2j+1 \end{array} \tag{17}$$

where i is the start node number, and j is the end node number. Now, to build the structure stiffness matrix, the contributions of each element stiffness matrix are added to the matching elements of the global matrix. For instance, element $\mathbf{k}_{global(0,0)}$ will be added to element $(2i, 2i)$ in the global matrix.

1.5 Boundary Conditions

A node can be restrained in the global frame of reference either in the x or y direction. Table 2 shows the possible truss supports. The number 1 indicates the support is free to translate in direction, and a 0 indicates the support is restrained in that direction.

Table 2: Truss Support Types

Support	x	y
Pin	1	1
Free	0	0
Horizontal Roller	0	1
Vertical Roller	1	0

Each node in the truss is assigned boundary conditions.

2 Pseudocode

The programming style used here is a style that mimics the physical node, element and truss objects. These objects have a set of properties, and derived properties that we can take advantage of. For instance, once an element 'knows' it's start and end node, it can compute a unit vector along itself, it can compute its own length, and it can compute a unit vector pointing in the direction of itself. In addition, it is able to take the DoF counting of it's own start and end node and combine them when building the structure stiffness matrix.

The following outlines the classes used for analyzing trusses with the direct stiffness method.

Node

- Properties
  - x: The x position
  - y: The y position
  - fixity: Boundary condition as a string ('pin', 'free', etc.)
  - id: An integer counter for each node. Starts out undefined
  - nDoF: The number of degrees of freedom at each node
- Derived Properties
  - BC: The boundary conditions as boolean values looked up based on the fixity
  - DoF: The degree of freedom counting computed from the node number

Element

- Properties
  - SN: Start Node
  - EN: End Node
  - material: Some material with a Young's Modulus
  - cross_section: Some cross section with an area
- Derived Properties
  - vector: The vector along the element
  - length: The length (norm) of the vector
  - unit_vector: The unit vector pointing in the direction of the element
  - kglobal: The global element stiffness matrix
  - DoF: The degree of freedom numbering of both the start node and end node
  - index_grid: The DoF numbering as a grid. The elements of the structure stiffness matrix where the element stiffness matrix contributes to.

Truss

- Properties
  - nodes
  - elements
- Derived Properties
  - K: Structural stiffness matrix
  - BC: Structure Boundary condition array, (combines all nodal boundary conditions)
- Functions
  - solve: Solve for the displacements of each node given some loading
  - plot: plot the undeformed structure

This seems like quite a bit of work when compared to how this code can be structured in a more procedural way. However, the benefit to this is there is greater clarity in the overall purpose of the code. The code written in this way can also be directly shared when writing a frame analysis program, because many of the properties and computed properties are computed in the exact same way.

This means reducing the total amount of code, and being able to reuse the same code verification strategies. It also means reducing the amount of code that needs to be copied and pasted between projects that rely on local variables.

Furthermore, when using the code to test many structures, their behaviors, and reproducing results the exact details of how to procedurally solve a system are hidden and higher level tasks can be accomplished.

3 Beam Direct Stiffness Method

A beam element has four degrees of freedom. Two degrees of freedom at each node which includes the rotation, θ, and the deflection or transverse displacement, w.

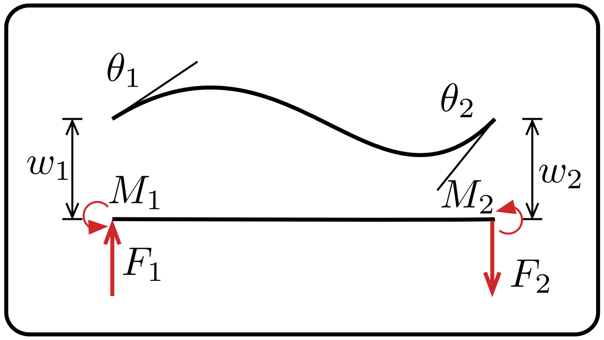


Figure 3: Degrees of Freedom

3.1 Bending Stiffness

We will isolate each degree of freedom by applying boundary conditions at each node, such that each beam will only have one unrestrained kinematic unknown. Figures 4 and 5 show the various boundary conditions that will be used to determine the beam bending stiffness. This will allow us to derive the force-displacement relationship necessary to solve the beam.

$$w_0 = 0 \;\; \theta_0 = ?$$
$$w_L = 0 \;\; \theta_L = 0$$

The following variables will be used for notational convenience when defining the beam stiffness matrix.

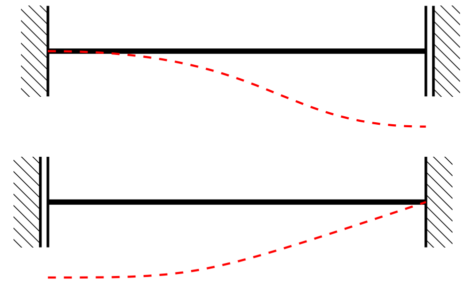


Figure 4: Isolation of Deflection

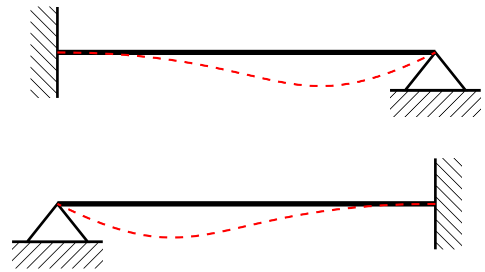


Figure 5: Isolation of Rotation

$$a = \tfrac{12EI}{L^3} \quad b = \tfrac{6EI}{L^2} \quad c = \tfrac{4EI}{L} \quad d = \tfrac{2EI}{L}$$

The local element stiffness matrix for a beam is therefore:

$$\left\{\begin{array}{c} V_0 \\ M_0 \\ \hline V_L \\ M_L \end{array}\right\} = \underbrace{\left[\begin{array}{cc|cc} a & b & -a & b \\ b & c & -b & d \\ \hline -a & -b & a & -b \\ b & d & -b & c \end{array}\right]}_{\mathbf{k_{local}}} \left\{\begin{array}{c} w_0 \\ \theta_0 \\ \hline w_L \\ \theta_L \end{array}\right\} \tag{18}$$

The global element stiffness matrix will not be discussed since a beam only has elements along one coordinate system.

4 Frame Direct Stiffness Method

The *Direct Stiffness Method* extends solving statically indeterminate frames. Each node in a frame has three degrees of freedom in comparison to two degrees of freedom for a truss. The additional degree of freedom is the rotation at each node.

4.1 Global Element Stiffness

The global transformation matrix is:

$$\begin{Bmatrix} u_x \\ u_y \\ \theta \end{Bmatrix} = \begin{bmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u'_x \\ u'_y \\ \theta' \end{Bmatrix} \tag{19}$$

Note that this transformation matrix has no effect on the rotation since rotating the whole coordinate system does not change the relative rotation at a node.

Combining the *local* element stiffness matricies for both a beam element and truss element yields:

$$\left\{\begin{array}{c} N_0 \\ V_0 \\ M_0 \\ \hline N_L \\ V_L \\ M_L \end{array}\right\} = \underbrace{\left[\begin{array}{c|cc|c|cc} e & 0 & 0 & -e & 0 & 0 \\ \hline 0 & a & b & 0 & -a & b \\ 0 & b & c & 0 & -b & d \\ \hline -e & 0 & 0 & e & 0 & 0 \\ \hline 0 & -a & -b & 0 & a & -b \\ 0 & b & d & 0 & -b & c \end{array}\right]}_{\mathbf{k}_{local}} \left\{\begin{array}{c} u_0 \\ w_0 \\ \theta_0 \\ \hline u_L \\ w_L \\ \theta_L \end{array}\right\} \tag{20}$$

4.2 Global Structural Stiffness

Table 3: Degree of Freedom Counting

Node Number	x	y	θ
0	0	1	2
1	3	4	5
2	6	7	8
i	$3i$	$3i+1$	$3i+2$

4.3 Boundary Conditions

Table 4 shows a table of the possible boundary conditions for a frame node support.

Table 4: Frame Support Types

Support	x	y	θ
Pin	1	1	1
Free	0	0	1
Horizontal Roller	0	1	1
Vertical Roller	1	0	1
Fixed	0	0	0
Slide	0	1	0

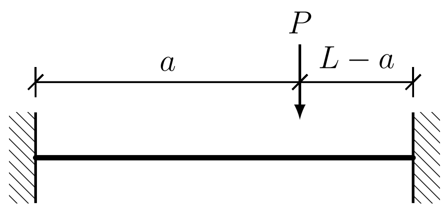
4.4 Equivalent Nodal Loading

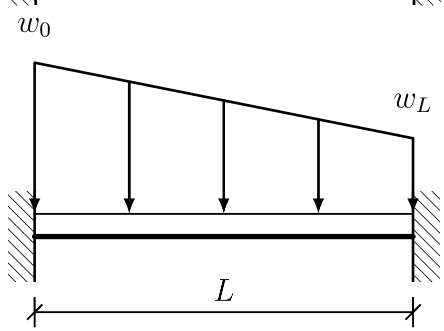
To solve the deformations with the governing equations, the equivalent nodal loading must be known (i.e. N_0, V_0, etc).

The equivalent nodal loading for a frame element with an applied point load

$$\mathbf{q}' = \left\{0, \frac{Pb^2(L+2a)}{L^3}, \frac{Pab^2}{L^2}, 0, \frac{Pa^2(L+2b)}{L^3}, \frac{-Pa^2b}{L^2}\right\} \qquad (21)$$

$$\begin{aligned}\mathbf{q}' = \Bigg\{&0, \frac{L(7w_0+3w_L)}{20}, \frac{L^2(3w_0+2w_L)}{60}, \\ &0, \frac{L(3w_0+7w_L)}{20}, -\frac{L^2(2w_0+3w_L)}{60}\Bigg\}\end{aligned} \qquad (22)$$





References