Trusses
Element Formulation by Virtual Work

- Use virtual work to derive element stiffness matrix based on assumed displacements
  - Principle of virtual work states that if a general structure that is in equilibrium with its applied forces deforms due to a set of small compatible virtual displacements, the virtual work done is equal to its virtual strain energy of internal stresses.
At element level, $\delta U_e = \delta W_e$
- $\delta U_e$ = virtual strain energy of internal stresses
- $\delta W_e$ = virtual work of external forces acting through virtual displacements
We now assume a simple displacement function to define the displacement of every material point in the element.

Usually use low order polynomials

Here

\[ u = a_1 + a_2x \]

– \( u \) is axial displacement
– \( a_1, a_2 \) are constants to be determined
– \( x \) is local coordinate along member
The constants are found by imposing the known nodal displacements $u_i$, $u_j$ at nodes $i$ and $j$

\begin{align*}
u_i &= a_1 + a_2 x_i \\
u_j &= a_1 + a_2 x_j
\end{align*}

- $u_i$, $u_j$ are nodal displacements
- $x_i$, $x_j$ are nodal coordinates
letting $x_i = 0$, $x_j = L$, we get

- $a_1 = u_i$
- $a_2 = (u_j - u_i)/L$

We can write

$$u = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = [N]\{d\}$$

- $[N] = \text{matrix of element shape functions or interpolation functions}$
- $\{d\} = \text{nodal displacements}$
\([ N ] = \begin{bmatrix} N_1 & N_2 \end{bmatrix} \]

\[ N_1 = 1 - \frac{x}{L}, \]

\[ N_2 = \frac{x}{L} \]

Properties

\( N_i = 1 \) at node \( i \) and zero at all other nodes

\( N_i = 1 \)

i.e. at any point in the element \( N_1 + N_2 = 1 \)
Strain is given by

\[ \varepsilon = \frac{du}{dx} = \frac{d}{dx} \{N\} \{d\} = [B] \{d\} \]

\[ [B] = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix} \]

where \([B]\) is a matrix relating strain to nodal displacement (matrix of derivatives of shape function)
Now

\[ \sigma = E(\varepsilon - \varepsilon_o) = E[B]\{d\} - E\varepsilon_o \]

- Stress and strain are constant in a member

Define internal virtual strain energy for a set of virtual displacements \{\delta d\} to be

\[ \delta U_e = v(\delta \varepsilon)^T \sigma dV \]
\[ - \delta \varepsilon = \text{virtual strain} \]
\[ - \sigma = \text{stress level at equilibrium} \]
\[ - dV = \text{volume} \]

\[ \text{Virtual work of nodal forces is} \]
\[ \delta W_e = \{\delta d\}^T \{f\} \]

\[ \text{Then, virtual work is given by} \]
\[ (\delta \varepsilon)^T \sigma dv = \{\delta d\}^T \{f\} \]
Substituting and rearranging gives

\[ \nu ( [ B ] \{ \delta d \} )^T ( E [ B ] \{ d \} - E \varepsilon_o ) dV = \{ \delta d \}^T \{ f \} \]

\[ \{ \delta d \}^T \nu [ B ]^T E [ B ] \{ d \} dV = \{ \delta d \}^T \nu [ B ]^T E \varepsilon_o dV + \{ \delta d \}^T \{ f \} \]

Canceling \( \{ \delta d \}^T \) gives \([k]\){d} = \{F\} \]

where

\[ [k] = \nu [ B ]^T E [ B ] dV \]

\[ \{ F \} = \{ f \} + EA \varepsilon_o \begin{cases} -1 \\ 1 \end{cases} \]

For thermal problem \( \varepsilon_o = \alpha T \)

2/4/02
for a truss we get

\[
[k] = \frac{EA}{L} \begin{vmatrix}
1 & 1 \\
-1 & 1 \\
\end{vmatrix}
\]

this formulation method also applies to 2-d and 3-d elements
Procedure for Direct Stiffness Method (Displacement Method)

1. Discretize into finite elements, Identify nodes, elements and number them in order.

2. Develop element stiffness matrices \([K_e]\) for all the elements.

3. Assemble element stiffness matrices to get the global stiffness matrix \([K_G] = \Sigma [K_e]\). The size of global stiffness matrix = total d.o.f of the structure including at boundary nodes. Assembly is done by matching element displacement with global displacements. Also develop appropriate force vector (by adding element force vectors) such that equation of the type \([K_G] \{u\} = \{F\}\) is obtained.
Procedure for Direct Stiffness Method

4. Apply kinematic boundary conditions. Without applying boundary conditions, $[K_G]$ will be singular. (minimum number of boundary conditions required is to arrest ‘Rigid Body’ displacements).

5. Solve for unknown displacements $\{u\}$ ($\{u\} = [K_G]^{-1}\{F\}$).

6. Once displacements are determined find

(a) reactions by picking up appropriate rows from the equation $\{F\} = [K_G] \{u\}$, (b) Find element forces $\{f\} = [K_e] \{u_e\}$, (c) Element stresses given by $\{\sigma_e\} = [D][B]\{u_e\}$. 
**Example 1**

![Diagram showing a bar assembly with boundary conditions](image)

**Problem:** Find the stresses in the two bar assembly which is loaded with force $P$, and constrained at the two ends, as shown in the figure.

**Solution:** Use two 1-D bar elements.

Boundary Conditions

$u_1 = 0, u_2 = 0$
Imagine a frictionless pin at node 2, which connects the two elements. We can assemble the global FE equation as follows,

\[
\begin{bmatrix}
  2 & -2 \\
-2 & 2+1 & -1 \\
-1 & 1
\end{bmatrix}
\]
Load and boundary conditions (BC) are,

\[ u_1 = u_3 = 0, \quad F_2 = P \]

FE equation becomes,

\[
\begin{bmatrix}
0 & 0 & 0 \\
0 & 2 & -2 \\
0 & -1 & 3
\end{bmatrix}
\begin{bmatrix}
0 \\
u_2 \\
0
\end{bmatrix} =
\begin{bmatrix}
F_1 \\
P \\
F_3
\end{bmatrix}
\]

Deleting the 1st row and column, and the 3rd row and column, we obtain,

\[
\frac{EA}{L} \begin{bmatrix} 3 \end{bmatrix} \{u_2\} = \{P\}
\]

Thus,

\[ u_2 = \frac{PL}{3EA} \]
Deleting the 1\textsuperscript{st} row and column, and the 3\textsuperscript{rd} row and column, we obtain,
\[
E_A \begin{bmatrix} 3 \end{bmatrix} \{ u_2 \} = \{ P \}
\]

Thus,
\[
u_2 = \frac{PL}{3EA}
\]

and
\[
\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \frac{PL}{3EA} \begin{bmatrix} 0 \\ 1 \\ 0 \end{bmatrix}
\]

Reactions
\[
\{ F_1 \} = \frac{AE}{L} \begin{bmatrix} 2 & -2 & 0 \end{bmatrix} \frac{PL}{3AE} \begin{bmatrix} 0 \\ 1 \\ 0 \end{bmatrix} = -\frac{2P}{3}
\]

\[
\{ F_3 \} = \frac{AE}{L} \begin{bmatrix} 0 & -1 & 1 \end{bmatrix} \frac{PL}{3AE} \begin{bmatrix} 0 \\ 1 \\ 0 \end{bmatrix} = -\frac{P}{3}
\]
Element Forces

Element 1
\[
\begin{bmatrix}
 f_1 \\
 f_2
\end{bmatrix} = \frac{2AE}{L} \begin{bmatrix}
 1 & -1 \\
 -1 & 1
\end{bmatrix} \begin{bmatrix}
 u_1 \\
 u_2
\end{bmatrix}
\]
\[
= \frac{2AE}{L} \begin{bmatrix}
 1 & -1 \\
 -1 & 1
\end{bmatrix} \frac{PL}{3AE} \begin{bmatrix}
 0 \\
 1
\end{bmatrix} = \begin{bmatrix}
 -2p/3 \\
 2p/3
\end{bmatrix}
\]

Element 2
\[
\begin{bmatrix}
 f_1 \\
 f_2
\end{bmatrix} = \frac{AE}{L} \begin{bmatrix}
 1 & -1 \\
 -1 & 1
\end{bmatrix} \begin{bmatrix}
 u_2 \\
 u_3
\end{bmatrix}
\]
\[
= \frac{AE}{L} \begin{bmatrix}
 1 & -1 \\
 -1 & 1
\end{bmatrix} \frac{PL}{3AE} \begin{bmatrix}
 1 \\
 0
\end{bmatrix} = \begin{bmatrix}
 p/3 \\
 -p/3
\end{bmatrix}
\]
Stress in element 1 is

\[ \sigma_1 = E \epsilon_1 = E B_1 u_1 = E \begin{bmatrix} -1/L & 1/L \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \]

\[ = E \frac{u_2 - u_1}{L} = E \left( \frac{PL}{3EA} - 0 \right) = \frac{P}{3A} \]

(member is in tension)

Similarly, stress in element 2 is

\[ \sigma_2 = E \epsilon_2 = E B_2 u_2 = E \begin{bmatrix} -1/L & 1/L \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} \]

\[ = E \frac{u_3 - u_2}{L} = E \left( 0 - \frac{PL}{3EA} \right) = -\frac{P}{3A} \]

which indicates that bar 2 is in compression.
Notes:

- In this case, the calculated stresses in elements 1 and 2 are exact within the linear theory for 1-D bar structures. It will not help if we further divide element 1 or 2 into smaller finite elements.

- For tapered bars, averaged values of the cross-sectional areas should be used for the elements.

- We need to find the displacements first in order to find the stresses, since we are using the displacement based FEM.
Example 2.

Problem: Determine the support reaction forces at the two ends of the bar shown above, given the following,

\[ P = 6.0 \times 10^4 \text{ N}, \quad E = 2.0 \times 10^4 \text{ N} / \text{mm}^2, \]
\[ A = 250 \text{ mm}^2, \quad L = 150 \text{ mm}, \quad \Delta = 1.2 \text{ mm} \]
Solution

We first check to see if or not the contact of the bar with the wall on the right will occur. To do this, we imagine the wall on the right is removed and calculate the displacement at the right end,

\[ \Delta_0 = \frac{PL}{EA} = \frac{(6.0 \times 10^4)(150)}{(2.0 \times 10^4)(250)} = 1.8 \text{ mm} > \Delta = 1.2 \text{ mm} \]

Thus, contact occurs.

Element 1                     Element 2

\[
\begin{bmatrix}
  u_1 & u_2 \\
  u_2 & u_3 \\
\end{bmatrix}
\]

\[
\begin{bmatrix}
  1 & -1 \\
  -1 & 1 \\
\end{bmatrix}
\]

\[
\begin{bmatrix}
  u_2 & u_3 \\
  -1 & 1 \\
\end{bmatrix}
\]

\[
\begin{bmatrix}
  1 & -1 \\
  -1 & 1 \\
\end{bmatrix}
\]
The global FE equation is found to be,

\[
\frac{EA}{L} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix}
\]

The load and boundary conditions are,

\( F_2 = P = 6.0 \times 10^4 \text{ N}, \quad u_1 = 0, \quad u_3 = \Delta = 1.2 \text{ mm} \)

The FE equation becomes,

\[
\frac{EA}{L} \begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 1 & \Delta \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ P \\ F_3 \end{bmatrix}
\]
The 2nd equation gives,

\[ \frac{EA}{L} \begin{bmatrix} 2 & -1 \end{bmatrix} \begin{bmatrix} u_2 \\ \Delta \end{bmatrix} = \{P\} \]

that is,

\[ \frac{EA}{L} [2] \{u_2\} = \left\{ P + \frac{EA}{L} \Delta \right\} \]

Solving this, we obtain \( u_2 = \frac{1}{2} \left( \frac{PL}{EA} + \Delta \right) = 1.5 \text{ mm} \)

and

\[ \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 1.5 \\ 1.2 \end{bmatrix} \text{(mm)} \]
To calculate the support reaction forces, we apply the 1\textsuperscript{st} and 3\textsuperscript{rd} equations in the global FE equation.

The 1\textsuperscript{st} equation gives,

\[ F_1 = \frac{EA}{L} \begin{bmatrix} 1 & -1 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{EA}{L} (-u_2) = -5.0 \times 10^4 \text{ N} \]

and the 3\textsuperscript{rd} equation gives,

\[ F_3 = \frac{EA}{L} \begin{bmatrix} 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{EA}{L} (-u_2 + u_3) \]

\[ = -1.0 \times 10^4 \text{ N} \]
Stress in element 1

\[ \sigma_1 = E\varepsilon_1 = EB \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = E \begin{bmatrix} -1/L \\ 1/L \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \]

\[ = E \frac{u_2 - u_1}{L} = 2.0 \times 10^4 \frac{1.5 - 0}{150} = 200 \text{N} / \text{mm}^2 \]

Stress in element 2

\[ \sigma_2 = E\varepsilon_2 = EB \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = E \begin{bmatrix} -1/L \\ 1/L \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} \]

\[ = E \frac{u_3 - u_2}{L} = 2.0 \times 10^4 \frac{1.2 - 1.5}{150} = -40 \text{N} / \text{mm}^2 \]
Direct Element Formulation

- truss element acts like 1-d spring
  - $l >>$ transverse dimensions
  - pinned connection to other members (only axial loading).
  - usually constant cross section and modulus of elasticity
\[ k = \frac{AE}{L} \]

» \( A \) = cross section area

» \( E \) = modulus of elasticity

» \( L \) = length
Assume displacements are much smaller than overall geometry
  - vertical displacements of horizontal member produce no vertical force
Stiffness matrix is written in local element coordinates aligned along element axis
want stiffness matrix for arbitrary orientation
◆ rotate coordinate systems using rotation matrix \([R]\) 
◆ displacement components in global coordinates are related to displacement components in local coordinates by 
\[
\{d'\} = [R] \{d\}
\]
  – \(\{d\} = \) displacement in global coordinates
  – \(\{d'\} = \) displacement in local element coordinates
\[
\begin{bmatrix}
1^{st} \text{ column}
\end{bmatrix} = \begin{bmatrix}
k \\
0 \\
-k \\
0
\end{bmatrix}
\]

\[
\begin{bmatrix}
3^{rd} \text{ column}
\end{bmatrix} = \begin{bmatrix}
-k \\
0 \\
k \\
0
\end{bmatrix}
\]
\[ \left[ 4^{th\ column} \right] = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \]

\[ \left[ 2^{nd\ column} \right] = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \]
start with member on $x$ axis, element equations are

$$\begin{bmatrix} k & 0 & -k & 0 \\ 0 & 0 & 0 & 0 \\ -k & 0 & k & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{bmatrix} = \begin{bmatrix} p'_i \\ q'_i \\ p'_j \\ q'_j \end{bmatrix}$$

or \( \{k'\} \{d'\} = \{f'\} \)

Note that $y$ equations are all zero
at node $i$

$u'_i = u_i \cos(\theta) + v_i \sin(\theta)$  \hspace{1cm} $p'_i = p_i \cos(\theta) + q_i \sin(\theta)$

$v'_i = -u_i \sin(\theta) + v_i \cos(\theta)$  \hspace{1cm} $q'_i = -q_i \sin(\theta) + q_i \cos(\theta)$
At node i

\[
\begin{align*}
\begin{bmatrix} u'_i \\ v'_i \end{bmatrix} &= \begin{bmatrix} \cos \theta & \sin \theta \\ -\sin \theta & \cos \theta \end{bmatrix} \begin{bmatrix} u_i \\ v_i \end{bmatrix} \\
\begin{bmatrix} p'_i \\ q'_i \end{bmatrix} &= \begin{bmatrix} \cos \theta & \sin \theta \\ -\sin \theta & \cos \theta \end{bmatrix} \begin{bmatrix} p_i \\ q_i \end{bmatrix}
\end{align*}
\]

A similar matrix can be obtained at node j

\[
\begin{align*}
\begin{bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{bmatrix} &= \begin{bmatrix} \cos \theta & \sin \theta & 0 & 0 & 0 \\ -\sin \theta & \cos \theta & 0 & 0 & 0 \\ 0 & 0 & \cos \theta & \sin \theta & 0 \\ 0 & 0 & -\sin \theta & \cos \theta & 0 \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{bmatrix}
\end{align*}
\]
Matrix $[R]$ is:

\[
\begin{bmatrix}
\cos \theta & \sin \theta & 0 & 0 \\
-\sin \theta & \cos \theta & 0 & 0 \\
0 & 0 & \cos \theta & \sin \theta \\
0 & 0 & -\sin \theta & \cos \theta
\end{bmatrix}
= \begin{bmatrix}
c & s & 0 & 0 \\
-s & c & 0 & 0 \\
0 & 0 & c & s \\
0 & 0 & -s & c
\end{bmatrix}
\]
Similarly, force components are related by
\[ \{f'\} = [R]\{f\} \]

Local force displacement relation is
\[ [k']\{d'\} = \{f'\} \]

Global force displacement relation is
\[ [k][R]\{d\} = [R]\{f\} \]

Using fact that \([R]^{-1} = [R]^T\), we get
\[ [R]^T[k][R]\{d\} = \{f\} \]
then \([k] = \text{stiffness matrix in global coordinates is } [R]^{T}[k'][R]\)

\[
[k] = k \begin{bmatrix}
c^2 & cs & -c^2 & -cs \\
cs & s^2 & -cs & -s^2 \\
-c^2 & -cs & c^2 & cs \\
-cs & -s^2 & cs & s^2 \\
\end{bmatrix}
\]
Structure equation is $[k] \{D\} = \{F\}$
- $[k]$ = structure stiffness matrix
- $\{D\}$ = nodal displacement vector
- $\{F\}$ = applied load vector

$$\sigma = DB\{u_i', u_j'\} \quad \text{note } u_i' = u_i \cos(\theta) + v_i \sin(\theta)$$

$$\sigma = E \begin{bmatrix} -1 & 1 \\ L & L \end{bmatrix} \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{bmatrix} = \frac{E}{L} \begin{bmatrix} -c & -s & c & s \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{bmatrix}$$
Example 3

A simple plane truss is made of two identical bars (with $E$, $A$, and $L$), and loaded as shown in the figure. Find

1) displacement of node 2;

2) stress in each bar.

Solution:

This simple structure is used here to demonstrate the assembly and solution process using the bar element in 2-D space.
<table>
<thead>
<tr>
<th>Element</th>
<th>i-node</th>
<th>Coordinate</th>
<th>j-node</th>
<th>Coordinate</th>
<th>Length</th>
<th>C</th>
<th>S</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>x</td>
<td>y</td>
<td>x</td>
<td>y</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>Lcos45</td>
<td>Lsin45</td>
<td>L</td>
<td>cos45</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0</td>
<td>2Lsin45</td>
<td>Lcos45</td>
<td>Lsin45</td>
<td>L</td>
<td>cos45</td>
</tr>
</tbody>
</table>

\[
[k'] = k \begin{bmatrix}
  c^2 & cs & -c^2 & -cs \\
  cs & s^2 & -cs & -s^2 \\
  -c^2 & -cs & c^2 & cs \\
  -cs & -s^2 & cs & s^2 \\
\end{bmatrix}
\]

\[
C = \frac{x_j - x_i}{L}, \quad m = \frac{y_j - y_i}{L} \\
L = \sqrt{(x_j - x_i)^2 + (y_j - y_i)^2}
\]
**Element 1:**

\[ k_1 = \frac{EA}{2L} \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} \]

**Element 2:**

\[ k_2 = \frac{EA}{2L} \begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \]
Assemble the structure FE equation,

\[
\begin{bmatrix}
  u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \\
  1 & 1 & -1 & -1 & 0 & 0 \\
  1 & 1 & -1 & -1 & 0 & 0 \\
 -1 & -1 & 2 & 0 & -1 & 1 \\
 -1 & -1 & 0 & 2 & 1 & -1 \\
 0 & 0 & -1 & 1 & 1 & -1 \\
 0 & 0 & 1 & -1 & -1 & 1 \\
\end{bmatrix}
\begin{bmatrix}
  u_1 \\
  v_1 \\
  u_2 \\
  v_2 \\
  u_3 \\
  v_3 \\
\end{bmatrix}
= \begin{bmatrix}
  F_{1x} \\
  F_{1y} \\
  F_{2x} \\
  F_{2y} \\
  F_{3x} \\
  F_{3y} \\
\end{bmatrix}
\]
Load and boundary conditions (BC):

\[ u_1 = v_1 = u_3 = v_3 = 0, \quad F_{2X} = P_1, \quad F_{2Y} = P_2 \]

\[
\begin{bmatrix}
\frac{EA}{2L} & \begin{bmatrix}
1 & 1 & -1 & -1 & 0 & 0 \\
1 & 1 & -1 & -1 & 0 & 0 \\
-1 & -1 & 2 & 0 & -1 & 1 \\
-1 & -1 & 0 & 2 & 1 & -1 \\
0 & 0 & -1 & 1 & 1 & -1 \\
0 & 0 & 1 & -1 & -1 & 1 \\
\end{bmatrix}
\end{bmatrix}
\begin{bmatrix}
u_1 \\
u_2 \\
u_3 \\
F_{1X} \\
F_{1Y} \\
F_{2X} \\
F_{2Y} \\
F_{3X} \\
F_{3Y} \\
\end{bmatrix} = 0
\]
Condensed FE equation,
\[
\frac{EA}{2L} \begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix} \begin{bmatrix} u_2 \\ v_2 \end{bmatrix} = \begin{bmatrix} P_1 \\ P_2 \end{bmatrix}
\]

Solving this, we obtain the displacement of node 2,
\[
\begin{bmatrix} u_2 \\ v_2 \end{bmatrix} = \frac{L}{EA} \begin{bmatrix} P_1 \\ P_2 \end{bmatrix}
\]

stresses in the two bars,
\[
\sigma_1 = \frac{E}{L} \frac{\sqrt{2}}{2} \begin{bmatrix} -1 & -1 & 1 & 1 \end{bmatrix} \frac{L}{EA} \begin{bmatrix} 0 \\ 0 \\ P_1 \\ P_2 \end{bmatrix} = \frac{\sqrt{2}}{2A} (P_1 + P_2)
\]

\[
\sigma_2 = \frac{E}{L} \frac{\sqrt{2}}{2} \begin{bmatrix} 1 & -1 & -1 & 1 \end{bmatrix} \frac{L}{EA} \begin{bmatrix} P_1 \\ P_2 \\ 0 \\ 0 \end{bmatrix} = \frac{\sqrt{2}}{2A} (P_1 - P_2)
\]
Example 4 (Multipoint Constraint)

For the plane truss shown above,

\[ P = 1000 \text{ kN}, \quad L = 1 \text{ m}, \quad E = 210 \text{ GPa}, \]
\[ A = 6.0 \times 10^{-4} \text{ m}^2 \quad \text{for elements 1 and 2}, \]
\[ A = 6\sqrt{2} \times 10^{-4} \text{ m}^2 \quad \text{for element 3}. \]

Determine the displacements and reaction forces.
Solution:

We have an inclined roller at node 3, which needs special attention in the FE solution. We first assemble the global FE equation for the truss.

Element 1: \( \theta = 90^\circ, \ l = 0, \ m = 1 \)

\[
\mathbf{k}_1 = \frac{(210 \times 10^9) (6.0 \times 10^{-4})}{1} \begin{bmatrix}
0 & 0 & 0 & 0 \\
0 & 1 & 0 & -1 \\
0 & 0 & 0 & 0 \\
0 & -1 & 0 & 1 \\
\end{bmatrix} \quad (N/m)
\]
Element 2: \( \theta = 0^\circ, \quad l = 1, \quad m = 0 \)

\[
k_2 = \frac{(210 \times 10^9)(6.0 \times 10^{-4})}{1}
\begin{bmatrix}
1 & 0 & -1 & 0 \\
0 & 0 & 0 & 0 \\
-1 & 0 & 1 & 0 \\
0 & 0 & 0 & 0
\end{bmatrix} \text{ (N / m)}
\]

Element 3

\( \theta = 45^\circ, \quad l = \frac{1}{\sqrt{2}}, \quad m = \frac{1}{\sqrt{2}} \)

\[
k_3 = \frac{(210 \times 10^9)(6\sqrt{2} \times 10^{-4})}{\sqrt{2}}
\begin{bmatrix}
0.5 & 0.5 & -0.5 & -0.5 \\
0.5 & 0.5 & -0.5 & -0.5 \\
-0.5 & -0.5 & 0.5 & 0.5 \\
-0.5 & -0.5 & 0.5 & 0.5
\end{bmatrix} \text{ (N / m)}
\]
The global FE equation is,

\[
\begin{bmatrix}
0.5 & 0.5 & 0 & 0 & -0.5 & -0.5 \\
1.5 & 0 & -1 & -0.5 & -0.5 \\
1 & 0 & -1 & 0 \\
1 & 0 & 0 & \end{bmatrix}
\begin{align*}
\begin{bmatrix}
u_1 \\
u_2 \\
u_3 \\
v_4 
\end{bmatrix}
&= \\
\begin{bmatrix}
F_{1x} \\
F_{1y} \\
F_{2x} \\
F_{2y} \\
F_{3x} \\
F_{3y}
\end{bmatrix}
\end{align*}
\]

Load and boundary conditions (BC):

\[
u_1 = v_1 = v_2 = 0, \quad \text{and} \quad v_3 = 0,
\]

\[
F_{2x} = P, \quad F_{3x'} = 0.
\]
From the transformation relation and the BC, we have

\[ v'_3 = \begin{bmatrix} -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix} \{u_3\} = \frac{\sqrt{2}}{2} (-u_3 + v_3) = 0, \]

that is, \( u_3 - v_3 = 0 \)

This is a *multipoint constraint* (MPC).

Similarly, we have a relation for the force at node 3,

\[ F_{3x'} = \begin{bmatrix} \sqrt{2} & \sqrt{2} \\ \sqrt{2} & \sqrt{2} \end{bmatrix} \{F_{3X} \} = \frac{\sqrt{2}}{2} (F_{3X} + F_{3Y}) = 0, \]

that is, \( F_{3X} + F_{3Y} = 0 \)
Applying the load and BC’s in the structure FE equation by ‘deleting’ 1\(^{st}\), 2\(^{nd}\) and 4\(^{th}\) rows and columns, we have

\[
1260 \times 10^5 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} P \\ F_{3X} \\ F_{3Y} \end{bmatrix}
\]

Further, from the MPC and the force relation at node 3, the equation becomes,

\[
1260 \times 10^5 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} P \\ F_{3X} \\ -F_{3X} \end{bmatrix}
\]
which is

\[ 1260 \times 10^5 \begin{bmatrix} 1 & -1 \\ -1 & 2 \\ 0 & 1 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} P \\ F_{3x} \\ -F_{3x} \end{bmatrix} \]

The 3\textsuperscript{rd} equation yields,

\[ F_{3x} = -1260 \times 10^5 u_3 \]

Substituting this into the 2\textsuperscript{nd} equation and rearranging, we have

\[ 1260 \times 10^5 \begin{bmatrix} 1 & -1 \\ -1 & 3 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} P \\ 0 \end{bmatrix} \]
Solving the previous equation, displacements are

\[
\begin{bmatrix}
u_2 \\ u_3
\end{bmatrix} = \frac{1}{2520 \times 10^5} \begin{bmatrix} 3P \\ P \end{bmatrix} = \begin{bmatrix} 0.01191 \\ 0.003968 \end{bmatrix} \quad \text{(m)}
\]

From the global FE equation, we can calculate the reaction forces,

\[
\begin{bmatrix}
F_{1x} \\
F_{1y} \\
F_{2y} \\
F_{3x} \\
F_{3y}
\end{bmatrix} = 1260 \times 10^5 \begin{bmatrix}
0 & -0.5 & -0.5 \\
0 & -0.5 & -0.5 \\
0 & 0 & 0 \\
-1 & 1.5 & 0.5 \\
0 & 0.5 & 0.5
\end{bmatrix} \begin{bmatrix}
u_2 \\ u_3 \\ v_3
\end{bmatrix} = \begin{bmatrix} -500 \\ -500 \\ 0.0 \\ -500 \\ 500 \end{bmatrix} \quad \text{(kN)}
\]
Finite Element Model

- usually use existing codes to solve problems
- user responsible for
  - creating the model
  - executing the program
  - interpreting the results
arrangement of nodes and elements is known as the mesh

plan to make the mesh model the structure as accurately as possible
for a truss

- each member is modeled as 1 truss element
- truss members or elements are connected at nodes
- node connections behave like pin joints
- truss element behaves in exact agreement with assumptions
- no need to divide a member into more than 1 element
– such subdivision will cause execution to fail
  » due to zero stiffness against lateral force at the node connection where 2 members are in axial alignment

bad {...}
there is geometric symmetry
  – often possible to reduce the size of problem by using symmetry
  – need loading symmetry as well
Fig. 3-5 and 3-6 show symmetric loads and the reduced model

– need to impose extra conditions along the line of symmetry
  » displacement constraints: nodes along the line of symmetry must always move along that line
  » changed loads: the load at the line of symmetry is split in two
Computer input assistance

- a preprocessor is used to assist user input
- required inputs are
  - data to locate nodes in space
  - definition of elements by node numbers
  - type of analysis to be done
  - material properties
  - displacement conditions
  - applied loads
interactive preprocessors are preferable

- you can see each node as it is created
- elements are displayed as they are created
- symbols are given for displacement and load conditions
- usually allow mesh generation by replication or interpolation of an existing mesh
- allow inserting nodes along lines
- allow entering a grid by minimum and maximum positions plus a grid spacing
- Truss element consists of 2 node numbers that connect to form element
- Other information for truss is
  - Modulus of elasticity
  - Cross sectional area
- Data can form a material table
- Assign element data by reference to the table
◆ boundary or displacement conditions are set by selecting a node and setting its displacement

✝ do not over constrain a structure by prescribing zero displacements where there is no physical support
loading conditions are set by selecting nodes and specifying force or moment components

check model carefully at this point
Analysis Step

- mostly transparent to user
- small truss models have enough accuracy and performance for an accurate solution
- a large model has a large number of elements and nodes
numerical solution may not be accurate if there are full matrices

get better accuracy if the nonzero terms are close to the diagonal
  – reduces the number of operations and round off error (banded matrix)
in FE model, element or node numbering can affect bandwidth

- good numbering pattern can minimize bandwidth
- different methods based on node or element numbering
- to minimize, plan numbering pattern so nodes that connect through an element have their equations assembled close together
In Fig. 3-7, node numbers are considered, X’s show nonzero terms.

Figure 3-7. Bandwidth of Simple Stiffness Matrices
In Fig. 3-8, node numbers are considered.

**Figure 3-8. Wavefront of Simple Stiffness Matrices**

<table>
<thead>
<tr>
<th></th>
<th>[K]</th>
<th>{D}</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[ X \ X \ 0 \ 0 \ 0 ]</td>
<td>{ u_1 }</td>
</tr>
<tr>
<td></td>
<td>[ X \ X \ 0 \ 0 \ 0 ]</td>
<td>{ u_2 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_3 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_4 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_5 }</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>[K]</th>
<th>{D}</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[ X \ X \ 0 \ 0 \ 0 ]</td>
<td>{ u_1 }</td>
</tr>
<tr>
<td></td>
<td>[ X \ X \ 0 \ 0 \ 0 ]</td>
<td>{ u_2 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_4 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_5 }</td>
</tr>
<tr>
<td></td>
<td>[ 0 \ 0 \ 0 \ 0 ]</td>
<td>{ u_3 }</td>
</tr>
</tbody>
</table>
many programs have bandwidth or wavefront minimizers available
most programs will keep original numbering for display but use the minimized number scheme
• numerical algorithms, numerical range of the computer affect solution
• relative stiffness of members can influence results
  – problems when members of high and low stiffness connect
  – can exceed precision of computer
  – physical situation is usually undesirable
Approximation error for truss is zero

Most common error messages (errors) come from
  – incorrect definition of elements
  – incorrect application of displacement boundary conditions
– may get non-positive definite structure stiffness matrix from not enough boundary conditions to prevent rigid body motion
  » two elements connect in-line zero lateral stiffness
  » truss structure not kinematically stable (linkage)
next look at stress components
  – in continua, stress components are related to averaged quantities at the nodes
  – trusses have a stress in each member (not easy to plot)
truss model is exact so it does not usually need refinement
Output Processing and Evaluation

- Get numerical results with input data followed by all nodal displacements and element stresses
- First graphic to look at is the deformed shape of the structure
  - Nodal displacements are exaggerated to show structure deformation
  - Check to ensure model behaves as expected
linear elastic analysis, failure is by
  - overstressing
  - buckling (have to find members with significant compression and use Euler's buckling equation)
Final Remarks

- few situations where a truss element is the right element for modeling behavior