3. Direct Approach of Finite Element Formulation

3.1 Spring and Bars

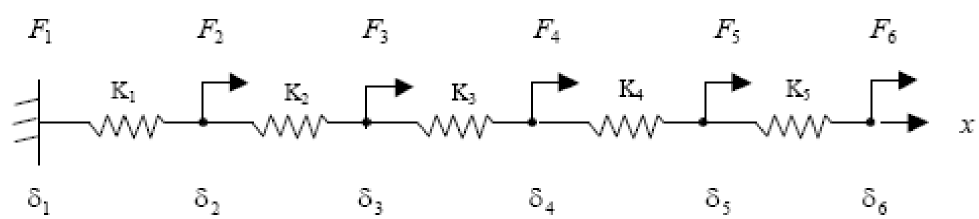
Direct approach has the following features:

- It *applies physical concept* (e.g. force equilibrium, energy conservation, mass conservation, etc.) *directly* to *discretized elements*.
  It is *easy* in its physical interpretation.
- It *does not need elaborate*, sophisticated mathematical manipulation or concept.
- Its applicability is ***limited*** to certain problems for which equilibrium or conservation law can be easily stated in terms of physical quantities one wants to obtain. In most cases, discretized elements are self obvious in the physical sense.

There are several examples of direct approaches as illustrated in the following. Using the first example, important features of FEM will be discussed in detail.

Example 1: Force Balance (Linear Spring System) [Bathe P.79, Ex.3.1]

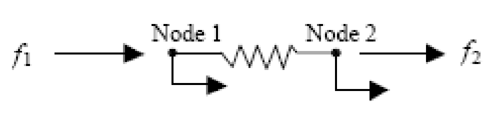
The problem of a linear spring system is depicted in the following figure.



In this particular problem, let us assume that $\delta_1, F_2, \cdots, F_6$ are specified. Solve the displacements at the nodes, and the reaction force at the node number 1.

One can typically follow several steps as an FEM procedure described below:

One element:



Equilibrium:

$$f_1 = k(\delta_1 - \delta_2)$$
$$f_2 = k(\delta_2 - \delta_1),$$

(Note that $f_1 = -f_2$ for force equilibrium.)

Element Matrix Equation for one element:

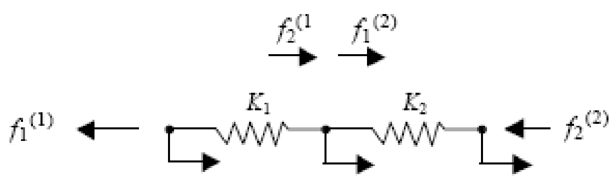
$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \quad \Leftrightarrow \quad [K]^e \{\phi\}^e = \{f\}^e$$

Global matrix equation by Assembly:

$$[K]\{\delta\} = \{F\}$$

where $[K]$ is called 'Stiffness Matrix' and
$\{F\}$ is called 'Resultant Nodal Force Matrix'.

The physical meaning of assembly procedure can be found in the force balance as described below:



$$\underbrace{(-k_1\delta_1 + k_1\delta_2)}_{f_2^{(1)}} + \underbrace{(k_2\delta_2 - k_2\delta_3)}_{f_1^{(2)}} = \quad f_2^{(1)} \quad + \quad f_1^{(2)} \quad = F_2$$

from element 1 from element 2

External nodal force applied at node 2

The typical assembly can be done as shown below:

$$\begin{bmatrix} k_1 & -k_1 & & & & \\ -k_1 & k_1+k_2 & -k_2 & & 0 & \\ & -k_2 & k_2+k_3 & -k_3 & & \\ & & -k_3 & k_3+k_4 & -k_4 & \\ & 0 & & -k_4 & k_4+k_5 & -k_5 \\ & & & & -k_5 & k_5 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \\ \delta_6 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} \tag{1}$$

banded and symmetric matrix

Now, let us pay attention to the boundary condition in this particular case.

Displacements: $\delta_1 = 0$: specified $\delta_2, \delta_3, \delta_4, \delta_5, \delta_6$: unknown

Geometric condition, ***Essential*** boundary condition

Forces: F_2, F_3, F_4, F_5, F_6 :specified F_1 : unkown reaction force

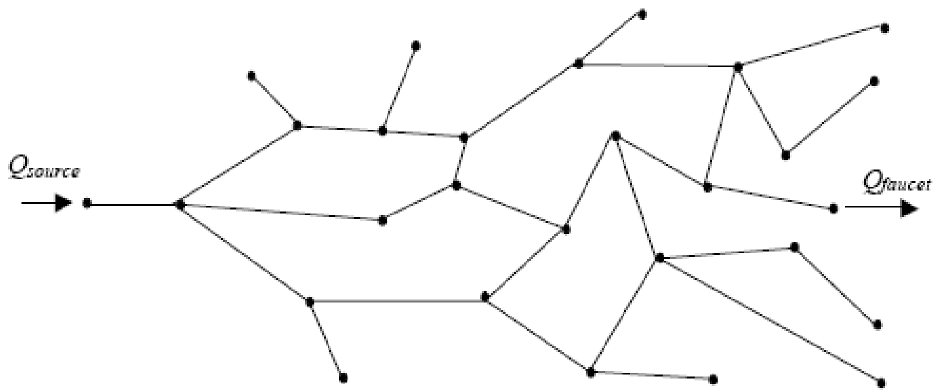
Force condition, ***Natural*** boundary condition

One should recognize that for each node *only one of the displacement and force can be specified as a boundary condition. Nature does not allow to specify both the displacement and force simultaneously at any node. If none of the two is known, then the problem is not well posed*, in other words, one does not have a problem to solve.

It should also be noted that if there is no geometry constraint at all, there is no unique solution. One can get a solution only up to a constant. (In other words, the linear spring system can be moved in x-axis without further deformation.) In this case, the stiffness matrix becomes a singular matrix. You will easily understand that equation (1) is singular since the summation of six rows becomes null. Think about the physical meaning of the fact that the summation of six rows becomes null. It just indicates that the force applied on the system is in balance! In this regard, it is obvious that *at least one geometry constraint should be assigned* in order to get a unique solution. Later, we will discuss the methods of introducing boundary conditions into equation (1).

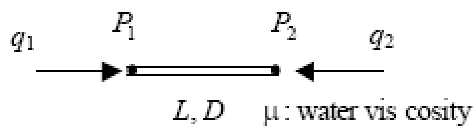
Example 3: Mass Conservation (Flow network, Electric network)

[Bathe P.82, Ex.3.3]



There is a water flow network as depicted above. The problem is to find water flow rate in the pipes and faucets given the water flow rate from the reservoir. The nature of the problem is almost identical to the previous two examples. Therefore, only the summary will be described below.

One element:



Mass conservation:

Fluid mechanics: $(P_1 - P_2) = \dfrac{128 q_1 L \mu}{\pi D^4}$

$$q_1 = k(P_1 - P_2)$$

$$\text{with } k \equiv \frac{\pi D^4}{128 L \mu}$$

$q_1 = k(P_1 - P_2)$: mass flow rate entering the element th rough node 1

$q_2 = k(P_2 - P_1)$: mass flow rate entering the element th rough node 2

(Note that $q_1 = -q_2$ for mass conservation.)

Element Matrix Equation for one element:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix} \qquad \Leftrightarrow \qquad [K]^e \{\phi\}^e = \{f\}^e$$

Global matrix equation by Assembly:

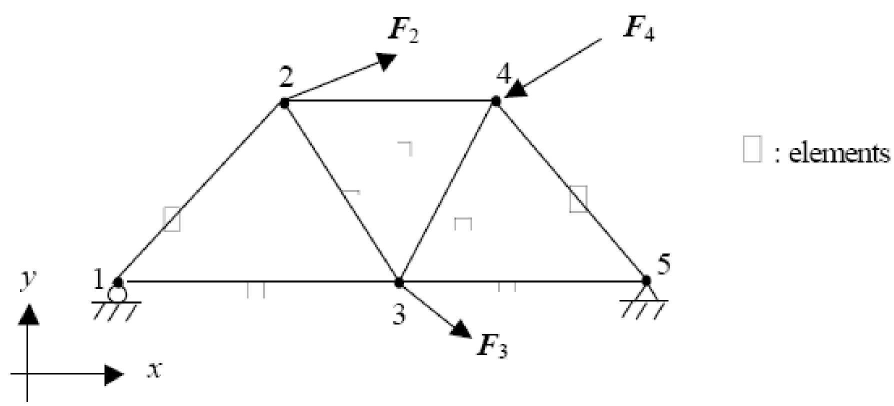
$$[K]\{P\} = \{Q\}$$

Now consider the assembly procedure. The assembly procedure is identical to the previous two examples. You are requested to find the physical meaning of the assembly procedure yourself.

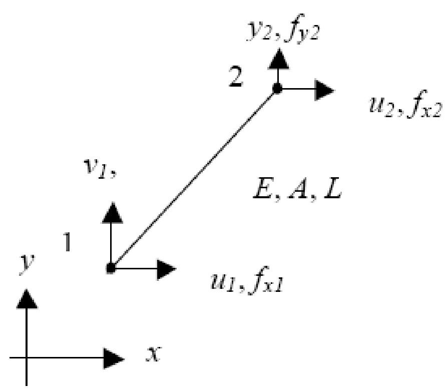
Also think about the boundary condition types. ***Which physical quantities are corresponding to essential and natural boundary condition? Can you make sure the pair characteristics of the boundary condition types for each node? Do you expect to obtain a singular matrix after the assembly? Why is it so?*** *What condition constitutes the well-posed problem?* You will definitely find the similarities among the three examples.

3.2 Truss Element

Next example is concerned about a two-dimensional analysis of truss structures with the so-called truss elements. Note that truss elements are joined by pin joint so that a truss element cannot bear the bending moment and shear force in contrast to a beam element. It can bear the tensile/compressive force (i.e. longitudinal force) only. Consider the following schematic diagram for this example.



One element:



Force-Deformation Law:

Elastic Elongation: $F = \dfrac{AE}{L}\Delta L$

which will result in the final global matrix equation of the following form:

$$[K]\{\delta\} = \{F\}$$

You are requested to find the physical meaning of the assembly procedure yourself.

Again think about the boundary condition types. ***Which physical quantities are corresponding to essential and natural boundary condition, respectively? Can you make sure the pair characteristics of the boundary condition types for each node? Do you expect to obtain a singular matrix after the assembly? Why is it so?*** *What condition constitutes the well-posed problem?* You will definitely find the similarity to the previous examples.

Note: The global stiffness matrix has three rank deficiency. For a well posed problem, one has to remove three equations (rows) or replace them with appropriate equations associated with boundary conditions. Think about the origin of the rank deficiency yourself.

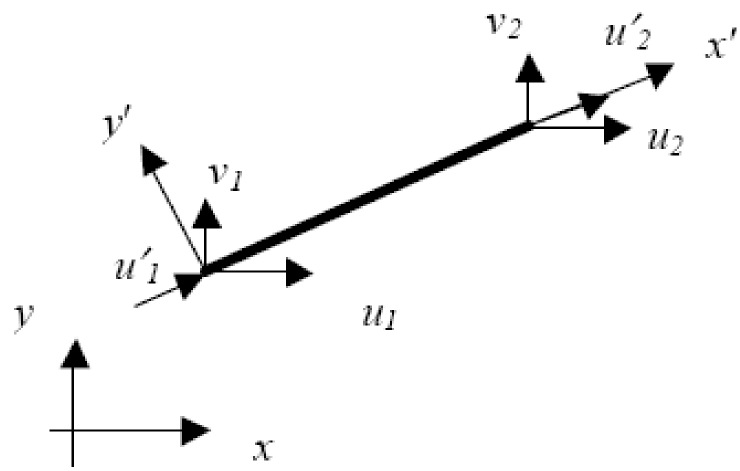
For the deformation problems, there are two kinds of approaches depending on which variable is considered unknown to be solved for.

i) $[K]\{x\}=\{F\}$ with $[K]$ being **stiffness** matrix

2.2 Coordinate Transformation

In many cases, one can introduce a local coordinate system associated with each element in addition to a global coordinate system. A local coordinate system can be defined in many cases in a self-obvious way inherent to the element itself. It is much easier to determine the stiffness matrix with respect to the local coordinate system of an element than with respect to the global coordinate system. The stiffness matrix with respect to the local coordinate system is to be transformed to that with respect to the global coordinate system before the assembly procedure.

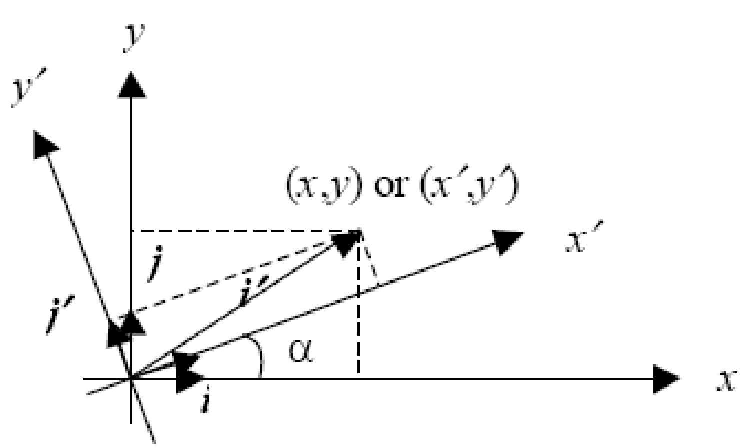
Example:



$$\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} \qquad \Rightarrow \qquad \begin{Bmatrix} u_1' \\ u_2' \end{Bmatrix}$$

Global system   Local system

) Vector Transformation in 2-D



$$\mathbf{V} = x\mathbf{i} + y\mathbf{j} = x'\mathbf{i}' + y'\mathbf{j}'$$

$$\begin{aligned} x' &= \cos\alpha \cdot x + \sin\alpha \cdot y \\ y' &= -\sin\alpha \cdot x + \cos\alpha \cdot y \end{aligned} \qquad \Rightarrow \qquad \begin{Bmatrix} x' \\ y' \end{Bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \begin{Bmatrix} x \\ y \end{Bmatrix}$$

$$\text{or} \qquad \begin{Bmatrix} x \\ y \end{Bmatrix} = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix} \begin{Bmatrix} x' \\ y' \end{Bmatrix}$$

ii) Transformation of stiffness matrix

The element matrix equation can be generally represented in terms of the local coordinate system. In the discussion below, we are interested in coordinate transformation of stiffness matrix associated with vectors such as displacement and force. Suppose the element stiffness matrix is represented by the following equation:

$$[K']^e \{x'\}^e = \{b'\}^e \tag{1}$$

The vector transformation of $\{x'\}^e$ and $\{y'\}^e$ between the local and global coordinate system might be

$$\{x'\}^e = [\Phi]\{x\}^e \tag{2}$$

and $$\{b'\}^e = [\Phi]\{b\}^e \tag{3}$$

where $\{x\}^e$ and $\{b\}^e$ are referenced to the global coordinate system. Then, equations (1)-(3) yields

$$[K']^e[\Phi]\{x\}^e = [\Phi]\{b\}^e \tag{4}$$

To get a matrix equation in the global coordinate system in terms of

$$[K]^e \{x\}^e = \{b\}^e \tag{5}$$

pre-multiply Eq. (4) by $[\Phi]^{-1}$ (a generalized inverse matrix of $[\Phi]$).

Then

$$[\Phi]^{-1}[K'][\Phi]\{x\}^e = [\Phi]^{-1}[\Phi]\{b\}^e = [I]\{b\}^e = \{b\}^e \tag{6}$$

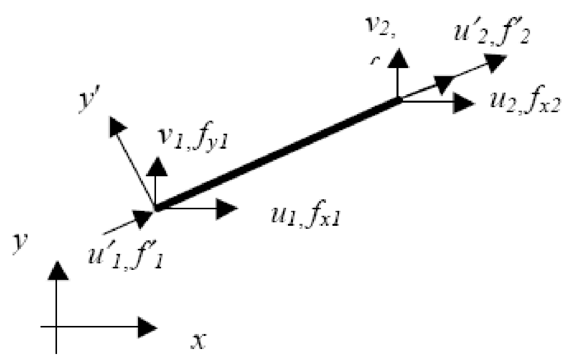
By comparison between equation (6) and (5), one can find

$$[K]^e = [\Phi]^{-1}[K']^e[\Phi] \tag{7}$$

Note:

1. For an orthogonal coordinate system, the transformation matrix has the characteristics of $[\Phi]^{-1} = [\Phi]^T$. If $[K']^e$ is symmetric, then $[K]^e$ remains symmetric!
2. If degree of freedom in the local coordinate system is different from that in the global coordinate system, $[\Phi]$ is not a square matrix.

Example: transformation in Truss Element



$\Rightarrow$

$$\sigma = E\varepsilon,\ \ \varepsilon = \frac{\Delta L}{L} = \frac{u_2' - u_1'}{L}$$

$$f_1' = -\sigma A = -\frac{EA}{L}(u_2' - u_1')$$
$$f_2' = \sigma A = -f_1'$$

i.e.

$$\begin{Bmatrix} f_1' \\ f_2' \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1' \\ u_2' \end{Bmatrix} \qquad \Leftrightarrow \qquad [K']^e = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{bmatrix} \cos\alpha & 0 \\ \sin\alpha & 0 \\ 0 & \cos\alpha \\ 0 & \sin\alpha \end{bmatrix} \begin{Bmatrix} u_1' \\ u_2' \end{Bmatrix} \qquad \Rightarrow \; [\Phi]^{-1}$$

$$\begin{Bmatrix} u_1' \\ u_2' \end{Bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} \qquad \Rightarrow \; [\Phi]$$

(note: $[\Phi]^{-1} = [\Phi]^T$)

To get

$$[K]^e \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \end{Bmatrix}$$

make use of the transformation rule:

$$[K]^e = [\Phi]^{-1}[K']^e[\Phi] = \begin{bmatrix} \cos\alpha & 0 \\ \sin\alpha & 0 \\ 0 & \cos\alpha \\ 0 & \sin\alpha \end{bmatrix} \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \end{bmatrix}$$

Finally one obtains the stiffness matrix which is exactly identical to the previously derived one:

$$[K]^e = \frac{AE}{L} \begin{bmatrix} \cos^2\alpha & \sin\alpha\cos\alpha & -\cos^2\alpha & -\sin\alpha\cos\alpha \\ \sin\alpha\cos\alpha & \sin^2\alpha & -\sin\alpha\cos\alpha & -\sin^2\alpha \\ -\cos^2\alpha & -\sin\alpha\cos\alpha & \cos^2\alpha & \sin\alpha\cos\alpha \\ -\sin\alpha\cos\alpha & -\sin^2\alpha & \sin\alpha\cos\alpha & \sin^2\alpha \end{bmatrix}$$

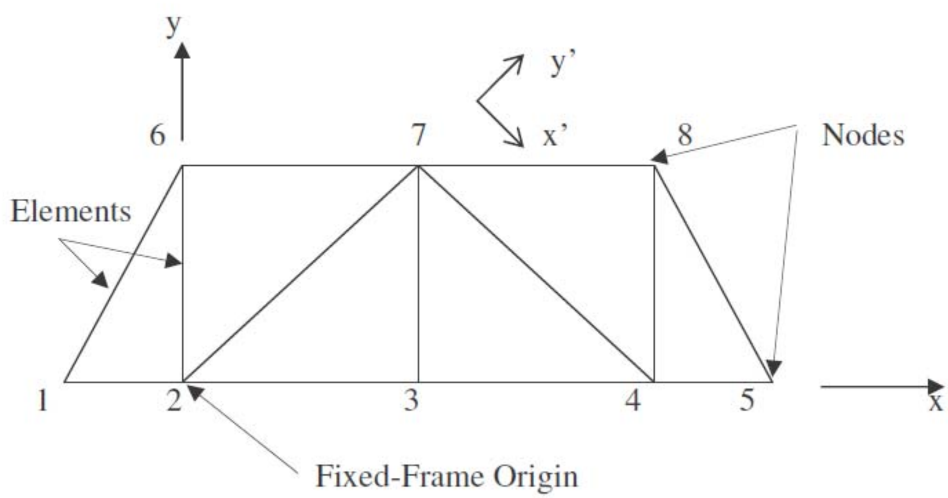
The reason for using the coordinate transformation is well demonstrated via this example. It is much easier to evaluate $[K']^e$ in the local coordinate system than $[K]^e$ in the global coordinate system. Note, however, that one has to calculate $[K]^e$ via the transformation rule before the assembly (element by element).

Note: In this particular example, the coordinate transformation matrix [Φ] is is not a square matrix because of the difference in the degree of freedom between the local and global coordinate systems. Think about which part of the derivation is affected by this difference. You have to recognize the following facts:

$$[\Phi][\Phi]^{-1} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} = [I]_{2\,x\,2}$$

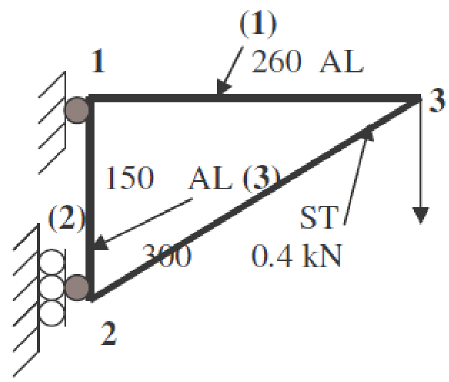
but
$$[\Phi]^{-1}[\Phi] = \begin{bmatrix} \cos^2\alpha & \cos\alpha\sin\alpha & 0 & 0 \\ \cos\alpha\sin\alpha & \sin^2\alpha & 0 & 0 \\ 0 & 0 & \cos^2\alpha & \cos\alpha\sin\alpha \\ 0 & 0 & \cos\alpha\sin\alpha & \sin^2\alpha \end{bmatrix} \neq [I]_{4\,x\,4}$$

which is not expected in the derivation of equation (6), i.e., $[\Phi]^{-1}[\Phi]\{b\} = [I]\{b\} = \{b\}$.



Local and Global Coordinates

Examples



For the truss structure shown:

Find displacements of joints 2 and 3
Find stress, strain, & internal forces
in each member.

$A_{AL} = 200\ mm^2$, $A_{ST} = 100\ mm^2$

All other dimensions are in mm.

Solution

Let the following node pairs form the elements:

Element	Node Pair
(1)	1-3
(2)	2-1
(3)	2-3

Using Shigley's Machine Design book for yield strength values, we have,

$S_{y\ (AL)} = 0.0375 kN/mm^2$ (375 Mpa)

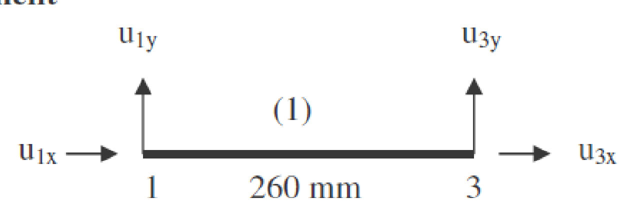
$S_{y\ (ST)} = 0.0586 kN/mm^2$ (586 Mpa)

$E_{(AL)} = 69 kN/mm^2$, $E_{(ST)} = 207 kN/mm^2$

$A^{(1)} = A^{(2)} = 200 mm^2$, $A^{(3)} = 100 mm^2$

Find the stiffness matrix for each element

Element (1)



$L^{(1)} = 260$ mm,
$E^{(1)} = 69 kN/mm^2$,
$A^{(1)} = 200 mm^2$

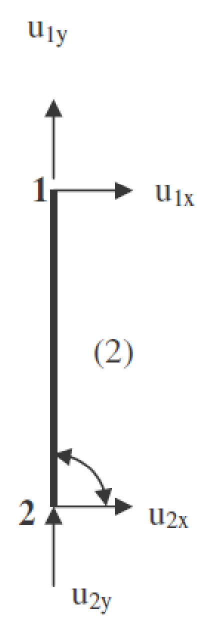
$\theta = 0$
$c = \cos\theta = 1$, $c^2 = 1$
$s = \sin\theta = 0$, $s^2 = 0$
$cs = 0$

$EA/L = 69 \text{ kN/mm}^2 \times 200 \text{ mm}^2 \times 1/(260\text{mm}) = 53.1 \text{ kN/mm}$

$$[K_g]^{(1)} = (AE/L) \times \begin{pmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{pmatrix}$$

$$[K_g]^{(1)} = (53.1) \times \begin{pmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix}$$

Element 2



$\theta = 90^0$
$c = \cos 90^0 = 0, \ c^2 = 0$

$s = \sin 90^0 = \cos 0^0 = 1, \ s^2 = 1$

$cs = 0$

$EA/L = 69 \times 200 \times (1/150) = 92 \text{ kN/mm}$

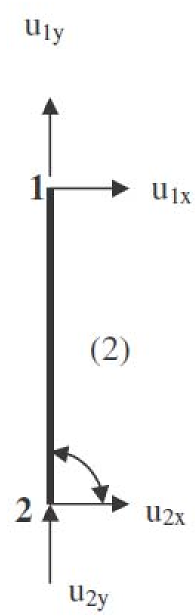
$$[k_g]^{(2)} = (92) \begin{array}{c} \begin{array}{cccc} u_{2x} & u_{2y} & u_{1x} & u_{1y} \end{array} \\ \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{pmatrix} \end{array} \begin{array}{c} \\ u_{2x} \\ u_{2y} \\ u_{1x} \\ u_{1y} \end{array}$$

$EA/L = 69\ kN/mm^2 \times 200\ mm^2 \times 1/(260mm) = 53.1\ kN/mm$

$$[K_g]^{(1)} = (AE/L) \times \begin{pmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{pmatrix}$$

$$[K_g]^{(1)} = (53.1) \times \begin{pmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix}$$

Element 2



$\theta = 90^0$

$c = \cos 90^0 = 0,\ c^2 = 0$

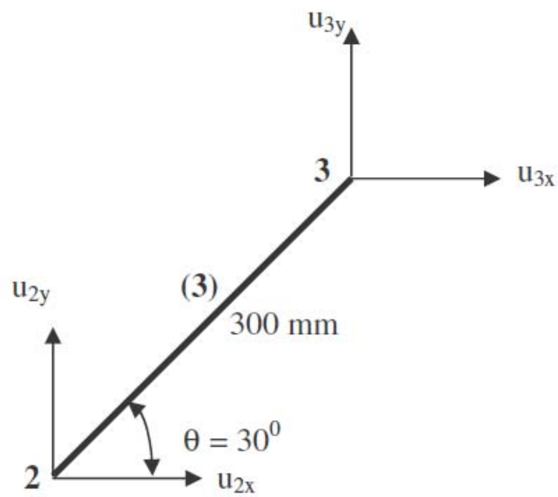
$s = \sin 90^0 = \cos 0^0 = 1,\ s^2 = 1$

$cs = 0$

$EA/L = 69 \times 200 \times (1/150) = 92\ kN/mm$

$$[k_g]^{(2)} = (92) \begin{array}{c} \begin{array}{cccc} u_{2x} & u_{2y} & u_{1x} & u_{1y} \end{array} \\ \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{pmatrix} \end{array} \begin{array}{c} \\ u_{2x} \\ u_{2y} \\ u_{1x} \\ u_{1y} \end{array}$$

Element 3



$\theta = 30^0$
$c = \cos 30^0 = 0.866,\ c^2 = 0.75$

$s = \cos 60^0 = .5,\ s^2 = 0.25$

$cs = 0.433$

EA/L = 207 x 100 x (1/300) = 69 kN/mm

$$[k_g]^{(3)} = \begin{array}{c|cccc} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \\ \hline u_{2x} & .75 & .433 & -.75 & -.433 \\ u_{2y} & -.433 & .25 & -.433 & -.25 \\ u_{3x} & -.75 & -.433 & .75 & .433 \\ u_{3y} & -.433 & -.25 & .433 & .25 \end{array} \quad (69)$$

Assembling the stiffness matrices

Since there are 6 deflections (or DOF), u_1 through u_6, the matrix is 6 x 6. Now, we will place the individual matrix element from the element stiffness matrices into the global matrix according to their position of row and column members.

Element [1]

$$\begin{array}{c|cccccc} & u_{1x} & u_{1y} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \\ \hline u_{1x} & 53.1 & & & & -53.1 & \\ u_{1y} & & & & & & \\ u_{2x} & & & & & & \\ u_{2y} & & & & & & \\ u_{3x} & -53.1 & & & & 53.1 & \\ u_{3y} & & & & & & \end{array}$$

The blank spaces in the matrix have a zero value.

Element [2]

$$\begin{array}{cc} & \begin{array}{cccccc} u_{1x} & u_{1y} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \end{array} \\ \begin{array}{c} u_{1x} \\ u_{1y} \\ u_{2x} \\ u_{2y} \\ u_{3x} \\ u_{3y} \end{array} & \left(\begin{array}{cccccc} & & & & & \\ & 92 & & -92 & & \\ & & & & & \\ & -92 & & 92 & & \\ & & & & & \\ & & & & & \end{array}\right) \end{array}$$

Element [3]

$$\begin{array}{cc} & \begin{array}{cccccc} u_{1x} & u_{1y} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \end{array} \\ \begin{array}{c} u_{1x} \\ u_{1y} \\ u_{2x} \\ u_{2y} \\ u_{3x} \\ u_{3y} \end{array} & \left(\begin{array}{cccccc} & & & & & \\ & & & & & \\ & & 51.7 & 29.9 & -51.7 & -29.9 \\ & & 29.9 & 17.2 & -29.9 & -17.2 \\ & & -51.7 & -29.9 & 51.7 & 29.9 \\ & & -29.9 & -17.2 & 29.9 & 17.2 \end{array}\right) \end{array}$$

Assembling all the terms for elements [1] , [2] and [3], we get the complete matrix equation of the structure.

$$
\begin{array}{cccccc}
u_{1x} & u_{1y} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \\
\end{array}
$$

$$
\begin{pmatrix}
53.1 & 0 & 0 & 0 & -53.1 & 0 \\
0 & 92 & 0 & -92 & 0 & 0 \\
0 & 0 & 51.7 & 29.9 & -51.7 & -29.9 \\
0 & -92 & 29.9 & 109.2 & -29.9 & -17.2 \\
-53.1 & 0 & -51.7 & -29.9 & 104.8 & 29.9 \\
0 & 0 & -29.9 & -17.2 & 29.9 & 17.2
\end{pmatrix}
\begin{pmatrix} u_{1x} \\ u_{1y} \\ u_{2x} \\ u_{2y} \\ u_{3x} \\ u_{3y} \end{pmatrix}
=
\begin{pmatrix} F_1 \\ F_1 \\ F_1 \\ F_1 \\ F_1 \\ F_1 \end{pmatrix}
$$

Boundary conditions

Node 1 is fixed in both x and y directions, where as, node 2 is fixed in x-direction only and free to move in the y-direction. Thus,

$u_{1x} = u_{1y} = u_{2x} = 0.$

Therefore, all the columns and rows containing these elements should be set to zero. The reduced matrix is:

$$
\begin{bmatrix}
109.2 & -29.9 & -17.2 \\
-29.9 & 104.8 & 29.9 \\
-17.2 & 29.9 & 17.2
\end{bmatrix}
\begin{Bmatrix} u_{2y} \\ u_{3x} \\ u_{3y} \end{Bmatrix}
=
\begin{Bmatrix} 0 \\ 0 \\ -0.4 \end{Bmatrix}
$$

Writing the matrix equation into algebraic linear equations, we get,

$$
\begin{aligned}
29.9u_{2y} - 29.9\, u_{3x} - 17.2u_{3y} &= 0 \\
-29.9u_{2y} + 104\, u_{3x} + 29.9u_{3y} &= 0 \\
-17.2u_{2y} + 29.9u_{3x} + 17.2u_{3y} &= -0.4
\end{aligned}
$$

solving, we get $u_{2y} = -0.0043$

$u_{3x} = 0.0131$

$u_{3y} = -0.0502$

Sress, Strain and deflections

Element (1)

Note that u_{1x}, u_{1y}, u_{2x}, etc. are not coordinates, they are actual displacements.

$\Delta L = u_{3x} = 0.0131$

$\varepsilon = \Delta L/L = 0.0131/260 = 5.02 \text{ x } 10^{-5}$ mm/mm

$\sigma = E\,\varepsilon = 69 \text{ x } 5.02 \text{ x } 10^{-5} = 0.00347$ kN/mm^2

Reaction $R = \sigma\, A = 0.00347$ kN

Element (2)

$\Delta L = u_{2y} = 0.0043$

$\varepsilon = \Delta L/L = 0.0043/150 = 2.87 \text{ x } 10^{-5}$ mm/mm

$\sigma = E\,\varepsilon = 69 \text{ x } 2.87 \text{ x } 10^{-5} = 1.9803$ kN/mm^2

Reaction $R = \sigma\, A = (1.9803 \text{ x } 10^{-3})\ (200)\ = 0.396$ kN

Element (3)

Since element (3) is at an angle 30^0, the change in the length is found by adding the displacement components of nodes 2 and 3 along the element (at 30^0). Thus,

$\Delta L = u_{3x} \cos 30^0 + u_{3y} \sin 30^0 - u_{2y} \cos 60^0$

$= 0.0131 \cos 30^0\ -0.0502 \sin 30^0 + 0.0043 \cos 60^0$

$= -0.0116$

$\varepsilon = \Delta L/L = -0.0116/300 = -3.87 \text{ x } 10^{-5} = 3.87 \text{ x } 10^{-5}$ mm/mm

$\sigma = E\,\varepsilon = 207 \text{ x } -3.87.87 \text{ x } 10^{-5} = -.0080$ kN/mm^2

Reaction $R = \sigma\, A = (-0.0087)\ (100)\ = 0.-0.800$ kN

Factor of Safety
Factor of safety '*n*' is the ratio of yield strength to the actual stress found in the part.

$$Element(1)\ \ n = \frac{S_y}{\sigma} = \frac{0.0375}{0.00347} = 10.8$$
$$Element(2)\ \ n = \frac{S_y}{\sigma} = \frac{0.0375}{0.00198} = 18.9$$
$$Element(3)\ \ n = \frac{S_y}{\sigma} = \frac{0.0586}{0.0080} = 7.325$$

The lowest factor of safety is found in element (3), and therefore, the steel bar is the most likely to fail before the aluminum bar does.

Final Notes

- The example presented gives an insight into how the element analysis works. The example problem is too simple to need a computer based solution; however, it gives the insight into the actual FEA procedure. In a commercial FEA package, solution of a typical problem generates a very large stiffness matrix, which will require a computer assisted solution.

- In an FEA software, the node and element numbers will have variable subscripts so that they will be compatible with a computer-solution

- Direct or equilibrium method is the earliest FEA method.