Finite Element Analysis (FEA)

In its most simplistic form, finite element analysis capitalizes upon this fundamental equation:

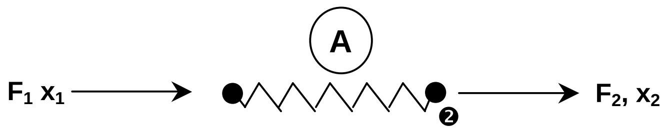
$$k\,x = F \qquad \textbf{[1]}$$

where:	k	spring stiffness	[units of force/units of displacement]
	x	deflection	[units of displacement]
	F	force	[units of force]

In finite element analysis, k is referred to as "stiffness" and instead of the equation being written in scalar form, as it is in equation **[1]**, it is written as a system of equations in matrix form, as shown in **[2]**.

$$[k]\{x\} = \{F\} \qquad \textbf{[2]}$$

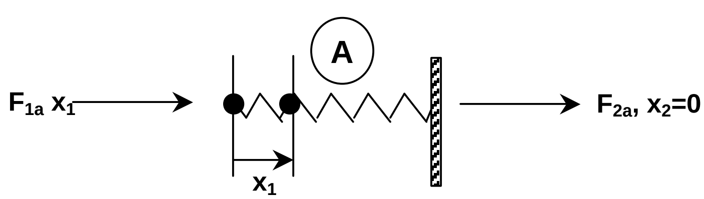
An example of how to form this system of equations is shown below. Consider this:



What is shown in this picture, is an **element**, A, with **nodes**, and ❷; each node being subjected to a displacement x_1 or x_2 resulting in nodal forces, $\mathbf{F_1}$ and $\mathbf{F_2}$, respectively. The system of equations representing this situation are given below:

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{11} \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \qquad \textbf{[3]}$$

Now, let's conduct an experiment on the spring. Hold node 2 fixed so that it cannot deflect.

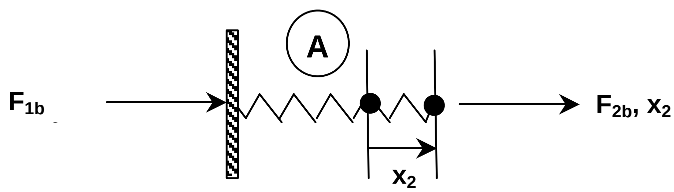


Summing forces:

$$\mathbf{F_{1a} + F_{2a} \quad = \quad 0}$$

$$\mathbf{F_{2a} = -F_{1a}}$$
$$\mathbf{F_{1a} = k\,x_1}$$
$$\mathbf{F_{2a} = -k\,x_1}$$

Now, hold node 1 stationary and apply a force at node 2.



Summing forces

$$\mathbf{F_{1b} + F_{2b} = 0}$$
$$\mathbf{F_{2b} = -F_{1b}}$$
$$\mathbf{F_{2b} = k\,x_2}$$
$$\mathbf{F_{1b} = -k\,x_2}$$

The fact that this system is linear (we assume a linear spring rate) enables us to utilize **superposition**, i.e., we can add the effects of each of these cases to determine the overall effect:

$$F_1 = F_{1a} + F_{1b} = k\,x_1 - k\,x_2$$
$$F_2 = F_{2a} + F_{2b} = -k\,x_1 + k\,x_2$$

These equations say:

The **total force** at node 1, $\mathbf{F_1}$, is the amount of force on node 1, caused by holding node 2 fixed + the amount of force on node 1 caused by holding node1 fixed.

The **total force** at node 2, $\mathbf{F_2}$, is the amount of force on node 2, caused by holding node 1 fixed + the amount of force on node 2 caused by holding node 2 fixed

Now, the system of equations representing this element and its nodal forces and displacements may be written,

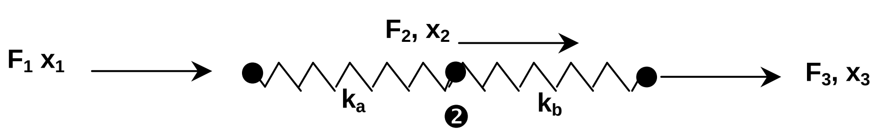
$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \qquad \textbf{[4]}$$

Notice a couple of features about the "stiffness matrix":

1. It is symmetric (i.e., row 1 is the same as column 1, and row 2 is the same as column 2)
2. The determinant of this matrix is zero, i.e., the matrix is singular

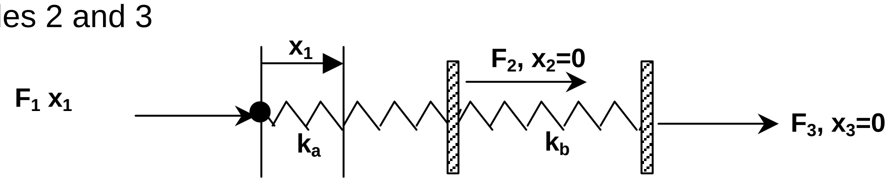
---

The example given above represents a singular element with two nodes. The whole idea behind finite element analysis is to **discretize** a component into several hundreds or even several thousands of elements. The next example shows you how multiple elements are assembled into a system.



Case 1:

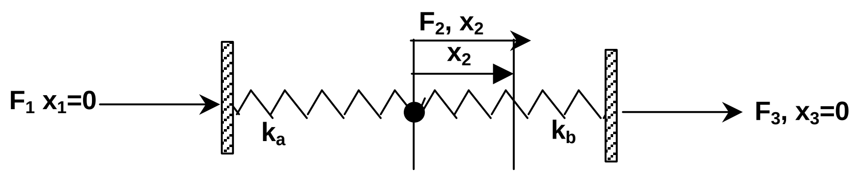
Fix nodes 2 and 3



$\mathbf{F_1 = k_a x_1}$
$\mathbf{F_2 = - F_1}$
$\mathbf{F_3 = 0}$

Case 2:

Fix nodes 1 and 3

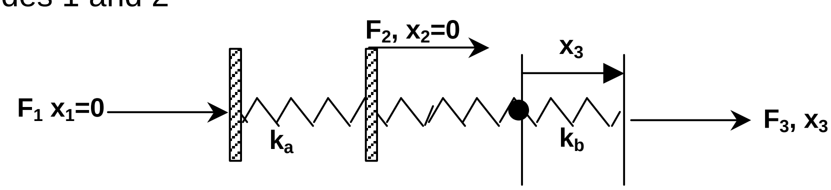


If the displacements at both ends are zero, then the middle node will have nodal force F_2 equal to $k_a\, x_2 + k_b\, x_2$ (the first term, $k_a\, x_2$, extends the first spring, the second term, $k_b\, x_2$, compressess the second spring)

The force at node 1, F_1 in this case will be $-k_a\, x_2$
The force at node 2, F_3 in this case will be $-k_b\, x_2$

Case 3:

Fix nodes 1 and 2



$F_3 = k_b\, x_3$
$F_2 = -k_b\, x_3$
$F_1 = 0$

Using superposition

Case 1:	$F_1 = k_a\, x_1$	$F_2 = -k_a\, x_1$	$F_3 = 0$
Case 2:	$F_1 = -k_a x_2$	$F_2 = (k_a+k_b)x_2$	$F_3 = -k_b x_2$
Case 3:	$F_1 = 0$	$F_2 = -k_b\, x_3$	$F_3 = k_b\, x_3$

$F_1 = k_a\, x_1 - k_a x_2$
$F_2 = -k_a\, x_1 + (k_a+k_b)x_2 - k_b\, x_3$
$F_3 = -k_b x_2 + k_b\, x_3$

In matrix form:

$$\begin{bmatrix} k_a & -k_a & 0 \\ -k_a & k_a + k_b & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \qquad \textbf{[5]}$$

For two or three elements, examining combinations of possibilites of nodal displacements and resulting nodal forces is not prohibitively complex. However, imagine trying to determine all the possibilities for even a slightly more complex system, say 10 or 15 nodes. Finite element analysis using even hundreds of nodes, instead of thousands, would be highly inaccurate and would yield undependable results. So how is the process of assembling thousands of elements and thousands of nodes into a system of equations handled efficiently?

Generally, the stiffness matrix for each element is computed, and then the thousands of matrices required for all the elements are assembled into one large system matrix--a **global stiffness** matrix. For example, if there are 3 nodes on two elements, 2 elemental matrices are computed, and then a global stiffness matrix is assembled by "superimposing" the elemental matrices. Let's look at the previous example, form two elemental matrices and superimpose them to get a single 3 x 3 global stiffness matrix.

The first element consists of nodes 1 and 2. The elemental stiffness matrix for element 1 is:

$$\left[K^{e_1}\right] = \begin{bmatrix} k_a & -k_a \\ -k_a & k_a \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$

The second element consits of nodes 2 and 3. The elemental stiffness matrix for element 2 is:

$$\left[K^{e_2}\right] = \begin{bmatrix} k_b & -k_b \\ -k_b & k_b \end{bmatrix} = \begin{bmatrix} k_{22} & k_{23} \\ k_{32} & k_{33} \end{bmatrix}$$

Superimposing the two elemental matrices results in the following global matrix

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & k_a + k_b & -k_b \\ 0 & -k_b & k_b \end{bmatrix}$$

The overlap occurs in the global matrix where elements share nodes. For example, node 2 is shared by elements 1 and 2, so elemental stiffness matrices are added together beginning at global element member k_{22}.

To test your understanding of assembling a global stiffness matrix, try assembling the following elemental matrices into a single global stiffness matrix. Also, draw each element and label nodes and nodal forces.

$$\left[K^{e_1}\right]=\begin{bmatrix} k_a & -k_a \\ -k_a & k_a \end{bmatrix}=\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$

$$\left[K^{e_2}\right]=\begin{bmatrix} k_b & -k_b \\ -k_b & k_b \end{bmatrix}=\begin{bmatrix} k_{22} & k_{23} \\ k_{32} & k_{33} \end{bmatrix}$$

$$\left[K^{e_3}\right]=\begin{bmatrix} k_c & -k_c \\ -k_c & k_c \end{bmatrix}=\begin{bmatrix} k_{33} & k_{34} \\ k_{43} & k_{44} \end{bmatrix}$$