Table of Contents

1 CHAPTER 1 – INTRODUCTION TO THE FINITE ELEMENT METHOD AND BAR ELEMENTS

Cook : 2.1, 2.2, 2.4 (for bars), 2.5 and 2.6

1.1 Simple springs

To establish the systematic elastic behaviour of bar elements we shall determine equilibrium equations for a bar element by two separate methodologies. First we assume the bar is uniform and behaves like a spring, which is equivalent to the most fundamentally simple expression of a uniform and uniformly loaded bar.

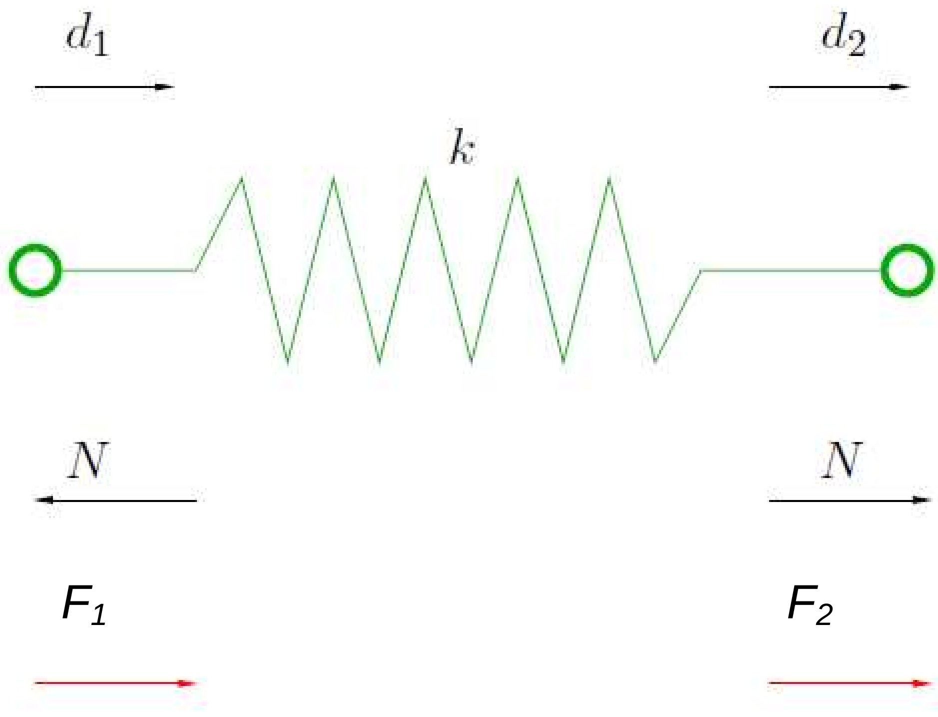


Figure 1-1 - Forces and degrees of freedom in a spring

Displacement is denoted *d*, forces are denoted *F*, and the spring stiffness is denoted *k*. From Figure 1-1 we find that force equilibrium may be described according to equation **(1)**.

$$F_1 + F_2 = 0 \quad \textbf{(1)}$$

Furthermore we find that the displacements d_1 and d_2 may be expressed according to the forces F_1 and F_2 as described in equations **(2)**;

$$k(d_1 - d_2) = F_1 \quad \textbf{(2)}$$
$$k(d_2 - d_1) = F_2$$

If we organise the system of equations **(2)** in a matrix equation, we find the following relation **(3)**;

$$k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} d_1 \\ d_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \qquad \textbf{(3)}$$

On element level it is common to generalise equation **(3)** with the following notation **(4)**;

$$\mathbf{kd} = \mathbf{r}\text{, where} \qquad \textbf{(4)}$$

$$\mathbf{k} = k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix},$$

$$\mathbf{d} = \begin{bmatrix} d_1 \\ d_2 \end{bmatrix},$$

$$\mathbf{r} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

In equation **(4)**, **k** is known as the element stiffness matrix, **d** is the element displacement vector and **r** is the element load vector.

The system of equations **(3)** makes little sense on its own, since the equilibrium equations **(1)** show that F_1 and F_2 are oppositely equal forces. Consequently d_1 and d_2 are also oppositely equal displacements if we only consider the case shown in Figure 1-1. Therefore it is initially no real reason to introduce the matrix form of the equilibrium equations since we could more easily calculate the elongation of the spring by the following expression **(5)**;

$$2d_2 = Nk \qquad \textbf{(5)}$$

Furthermore, if we tried to solve equation **(3)** we would find that the matrix equation did not have a unique solution in its current state. The explanation is that equation **(3)** only has a solution if consistent boundary conditions are applied, which is an issue we will come back to later in this chapter. The interesting question is thus; why are we organising our equations in matrix form, as stated in equation **(3)**? For single springs, there is no real reason to use a matrix equation in order to calculate displacements. However, for systems of springs we may utilise the general matrix equilibrium formulation systematically to achieve a single matrix equation for the entire system of springs. The reason we are looking at the matrix formulations is at the centre of the finite element method. This entire course is dedicated to explaining why and how we shall utilise such matrix formulations in order to establish simple linear systems of equations for larger structures composed of bars, beams and membranes.

Now we shall interchange the spring with a bar and include more details in section 1.2.

1.2 Bar elements

In Figure 1-2, a bar element is shown. At each end of the bar the position is denoted as **nodes**. When elements are subject to boundary conditions or elements are connected to one another, the applications of boundary conditions or connections are always in the nodes of the element. Interactions between elements may in specialised cases be applied between nodes, but for the purposes of this course, boundary conditions and connections between elements will always be placed in nodes. For bar elements the nodes are always at either end of the bar.

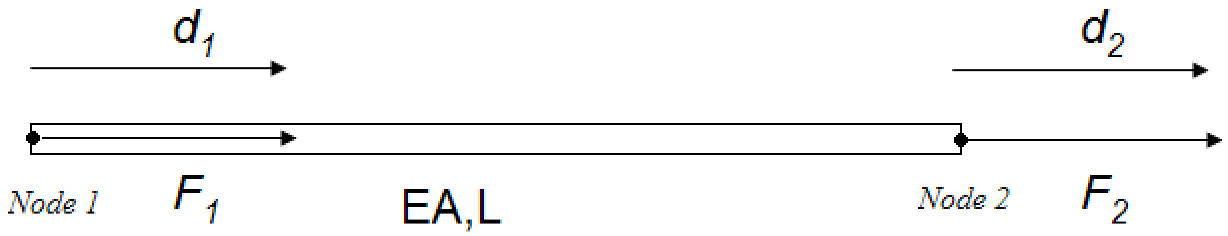


Figure 1-2 – A bar element

Displacements and forces are denoted *d* and *F* respectively. It is a convention in structural mechanics that;

> *nodal forces and nodal displacements within an element are always defined as positive in the same direction.*

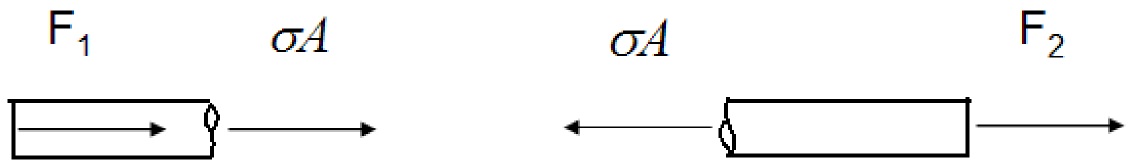


Figure 1-3 – Sectional equilibrium for the bar element

With reference to Figure 1-2 and Figure 1-3 we will establish equilibrium much in the same manner as for the spring in section 1.1. Equilibrium is deduced in equations **(6)** and **(7)**.

$$\begin{aligned} &F_1 + \sigma A = 0 \\ &F_2 - \sigma A = 0 \\ &\sigma = E\varepsilon \\ &\varepsilon = \frac{d_2 - d_1}{L} \end{aligned} \qquad \textbf{(6)}$$

If we substitute strain for stress into equations **(6)** we find two equilibrium equations for the bar **(7)**;

$$\left.\begin{aligned} F_1 + \frac{EA}{L}(d_2 - d_1) = 0 \\ F_2 - \frac{EA}{L}(d_2 - d_1) = 0 \end{aligned}\right\} \Rightarrow \begin{aligned} F_1 = \frac{EA}{L}(d_1 - d_2) \\ F_2 = \frac{EA}{L}(d_2 - d_1) \end{aligned} \qquad \textbf{(7)}$$

As for the spring element, we may organise the equilibrium equations in matrix form

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} d_1 \\ d_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \qquad \textbf{(8)}$$

If we substitute the tensile stiffness of the bar $\frac{EA}{L}$ with the spring stiffness k as given in section 1.1, we find that equations **(8)** and **(3)** are in fact the same equation. As for the spring element, it is customary to generalise equation **(8)** with the following terminology

$$\mathbf{kd} = \mathbf{r}\text{, where} \qquad \textbf{(9)}$$

$$\mathbf{k} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix},$$

$$\mathbf{d} = \begin{bmatrix} d_1 \\ d_2 \end{bmatrix},$$

$$\mathbf{r} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

k is the element stiffness matrix, **d** is the element displacement vector and **r** is the element load vector.

From equation **(9)** we should also make the following important observation;

> *A column of* **k** *is the vector of loads that must be applied to an element at its nodes to maintain a deformation state in which the corresponding nodal degree of freedom has unit value while all other nodal degrees of freedom are zero.*

For bar elements, each node has only one degree of freedom, which makes the above observation fairly obvious. However, the result is in fact general for all element formulations, and plays an important part in how kinematic compatibility is achieved when using the finite element method. We shall return to this observation at a later stage when we discuss beam elements.

1.3 System analysis

In sections 1.1 and 1.2 we looked at the elastic properties of springs and bars, and we found matrix equations for equilibrium of single elements, assuming linear elastic material properties. The system analysis is concerned with connecting elements to one another, and to apply boundary conditions and loading. For any structural mechanics problem the following conditions must apply;

- Kinematic compatibility
- Equilibrium
- A material law

In sections 1.1 and 1.2 the two latter bullets were considered. Boundary conditions and continuity in the structure is covered under kinematic compatibility, and furthermore loading adds further consideration on equilibrium.

To illustrate how systems are combined in the finite element method, we will look at a simple system of two connected bars.

1.3.1 Example – Axially loaded bars with varying cross-sections

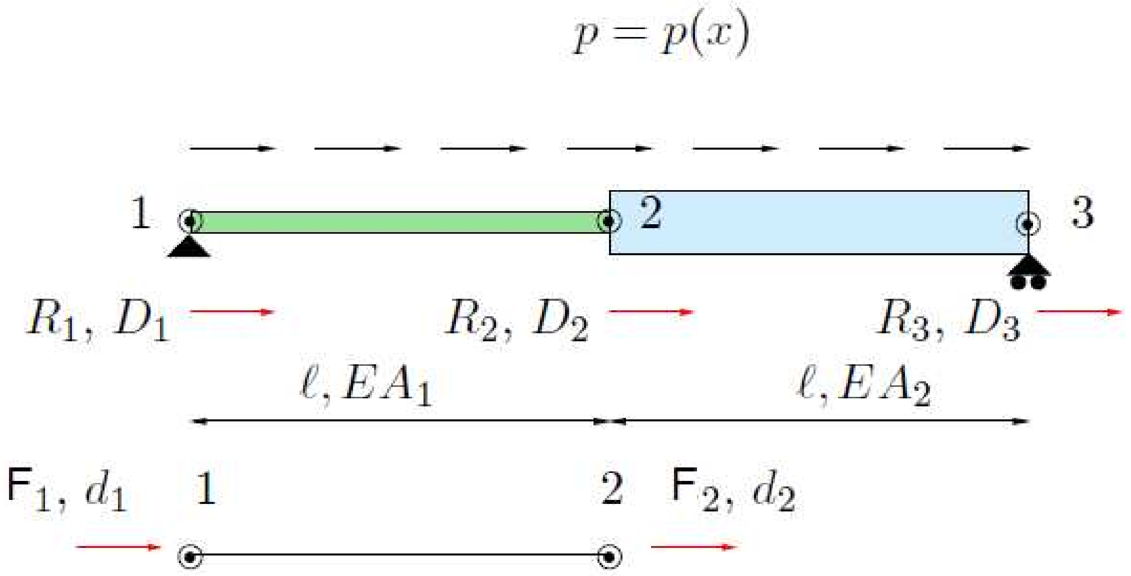


Figure 1-4 – Axially loaded, simply supported bar with varying cross-section

In Figure 1-4, there are three nodes, 1, 2 and 3. Note that node number 2 is shared by both bar elements. From section 1.2 we know that the element equilibrium equation for a single bar element is given by equation **(9)**. The two individual element stiffness relations may therefore be described by the following equations;

$$1 \rightarrow \frac{EA_1}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} d_1 \\ d_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

$$2 \rightarrow \frac{EA_2}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} d_1 \\ d_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

However, in the global system we choose suffixes for the global node numbers, and not the individual element. In the global system we have three nodes, and we may reformulate the equilibrium equations for each individual element;

$$\text{Element } 1 \rightarrow \frac{EA_1}{l}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix}$$

$$\text{Element 2} \rightarrow \frac{EA_2}{l}\begin{bmatrix} 0 & 0 & 0 \\ 0 & 1 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix}$$

D_i denotes displacements in the global system, and the suffix i marks the relevant node. From the above equations it should be noted that the redundant displacements at node 3 for element 1 and node 1 for element 3 have been included in the stiffness relations. The nodes are not connected to the relevant elements so there is no relation between force and displacement. This has been indicated by rows and columns of zeroes. Since we have included all the redundant equilibrium relations in the individual element stiffness matrices, they may now easily be combined into a global stiffness matrix since all the element stiffness matrices have the same dimension;

$$\mathbf{k}_1 + \mathbf{k}_2 = \frac{E}{l}\begin{bmatrix} A_1 & -A_1 & 0 \\ -A_1 & A_1 + A_2 & -A_2 \\ 0 & -A_2 & A_2 \end{bmatrix} = \mathbf{K}$$

Note that the element stiffness matrices are denoted by small letter boldfaced **k**, with suffix equal to the element number. The global stiffness matrix is denoted by a capital boldfaced **K.**

Now we have established the stiffness matrix for the two connected bars. The global displacement vector is trivially given as;

$$\mathbf{D} = \begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix}$$

Note that the global displacement vector is denoted by a capital boldfaced **D**. Since we have established the stiffness and the displacement, we are left with applying the forces to the system. In this case, we are faced with a distributed load. This is a problem we shall return to at a later stage in the course. For the purposes of this lecture we shall only assume that there are two concentrated loads, R_2 and R_3 at nodes 2 and 3 both directed axially and to the right relative to Figure 1-4.

In general, the forces must be balanced at each individual node.

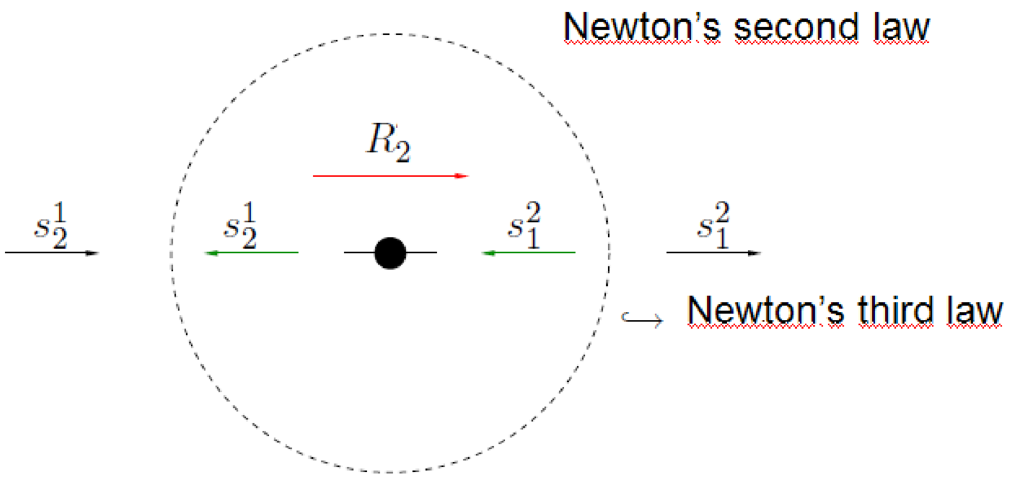


Figure 1-5 – Nodal force equilibrium

The nodal force R_i for a node $_i$ must balance all the forces s_i^j entering the node from neighbouring elements. Formally the relation may be expressed as follows **(10)**;

$$R_i = \sum_{e=1}^{m} s_i^e \tag{10}$$

R denotes the nodal force and *m* is the number of elements with a boundary to node *i*.

With the loading conditions in place, we are able to complete the expression for the global equilibrium equation;

$$\mathbf{KD} = \mathbf{R} \Rightarrow \frac{E}{l}\begin{bmatrix} A_1 & -A_1 & 0 \\ -A_1 & A_1 + A_2 & -A_2 \\ 0 & -A_2 & A_2 \end{bmatrix}\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} 0 \\ R_2 \\ R_3 \end{bmatrix}$$

In the process of establishing the equilibrium equations for the global system of two axially loaded bars, we have completed the following main steps of the algorithm we shall know as the finite element method;

- Meshing (we created two elements from a system)
- Establish local stiffness matrices
- Assemble local elements for a global stiffness matrix
- Apply loading

The next step in the algorithm is to include boundary conditions. In our case, there is a boundary condition at node no. 1 which requires that the displacement at node no. 1 is zero. This can easily be included in our equilibrium equation by simply demanding that D_1 is zero. We perform this by zeroing out the rows and columns in the global stiffness matrix which are governed by the displacement D_1.

$$\mathbf{KD} = \frac{EA}{l}\begin{bmatrix} A_1 + A_2 & -A_2 \\ -A_2 & A_2 \end{bmatrix}\begin{bmatrix} D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} R_2 \\ R_3 \end{bmatrix}$$

Now we are ready to solve our system. We invert our global stiffness matrix and multiply with the load vector. This process yields the global displacements;

$$\mathbf{D} = \mathbf{K}^{-1}\mathbf{R} \qquad \textbf{(11)}$$

For larger systems it may be more or less impossible to compute $\mathbf{K}^{-1}$ by hand, so for the remainder of this course we shall use computational aids in Matlab and ANSYS in order to compute the actual solutions. Only for very simplified systems will there be a need for manual calculation of inverses.

1.4 Properties of the stiffness matrix

The global stiffness matrix of a structure has several useful properties which are relevant both for computational efficiency in solving large systems as well as properties which are important in order to prove existence and uniqueness of solutions. The global stiffness matrix **K** is/has;

- Sparse
- Symmetric
- Only positive diagonal elements
- Positive definite
- Singular

1.4.1 Sparsity

The global stiffness matrix is almost always sparse, since only contributions from neighbouring elements are included for individual element entries in the matrix. This has little theoretical value, but a huge importance for effective solution methodologies.

1.4.2 Symmetry

Symmetry of the global stiffness matrix follows from Betti-Maxwells theorem;

> *If two sets of loads act on a linearly elastic structure then work done by the first set of loads in acting through displacements produced by the second set of loads is equal to the work done by the second set in acting through displacements produced by the first set.*

More precisely, if loads $\mathbf{R}_1$ and $\mathbf{R}_2$ are produce displacements $\mathbf{D}_1$ and $\mathbf{D}_2$, then;

$$\mathbf{R}_1^T\mathbf{D}_2 = \mathbf{R}_2^T\mathbf{D}_1$$

If we substitute for the load vectors $\mathbf{R}_1$ and $\mathbf{R}_2$ we find;

$$(\mathbf{K}\mathbf{D}_1)^T\mathbf{D}_2 = (\mathbf{K}\mathbf{D}_2)^T\mathbf{D}_1 \Rightarrow \mathbf{D}_1^T\mathbf{K}\mathbf{D}_2 = \mathbf{D}_2^T\mathbf{K}\mathbf{D}_1$$

K is an n x n matrix, and the displacement vectors $\mathbf{D}_1$ and $\mathbf{D}_2$ are n x 1 vectors. Thus both products on either side of the equation are scalar quantities. Since they are scalars they may be transposed without disturbing the equality;

$$\mathbf{D}_1^T\mathbf{K}\mathbf{D}_2 = \mathbf{D}_2^T\mathbf{K}\mathbf{D}_1 \Rightarrow \mathbf{D}_2^T\mathbf{K}^T\mathbf{D}_1 - \mathbf{D}_2^T\mathbf{K}^T\mathbf{D}_1 = 0 \Rightarrow \mathbf{D}_2^T\left(\mathbf{K}^T - \mathbf{K}\right)\mathbf{D}_1 = 0$$

Since neither $\mathbf{D}_1$ nor $\mathbf{D}_2$ are zero vectors (as that would mean zero loading) the expression inside the parentheses must vanish. This concludes the proof that $\mathbf{K} = \mathbf{K}^T$

1.4.3 Only positive diagonal elements

It is physically obvious that diagonal elements must be positive. If all degrees of freedom except one (arbitrarily chosen one) is constrained, a negative diagonal element would imply a negative displacement for a positive force. This is of course impossible.

1.4.4 Positive definite

The global stiffness matrix is positive definite, which by definition means that;

$$\mathbf{x}^T \mathbf{K} \mathbf{x} > 0, \forall \mathbf{x} \in \mathbf{R}^n \neq \mathbf{0}$$

The consequence of a positive definite stiffness matrix is that it is possible to use various types of matrix factorisations on **K**, which is highly useful when extracting Eigen-values and for optimised algorithms.

1.4.5 Singularity

The stiffness matrix is singular before boundary conditions are applied. When no boundary conditions are applied, the system is a mechanism rather than a static system in equilibrium. Therefore the deformations are undetermined, and the consequence is that the stiffness matrix is singular.

1.5 Arbitrary orientation of the stiffness matrix

The stiffness matrix is normally established in the local coordinate system of an element, but when applied to a structure, the stiffness matrix must often be rotated in order to match the global coordinate system, see for instance Figure 1-6.

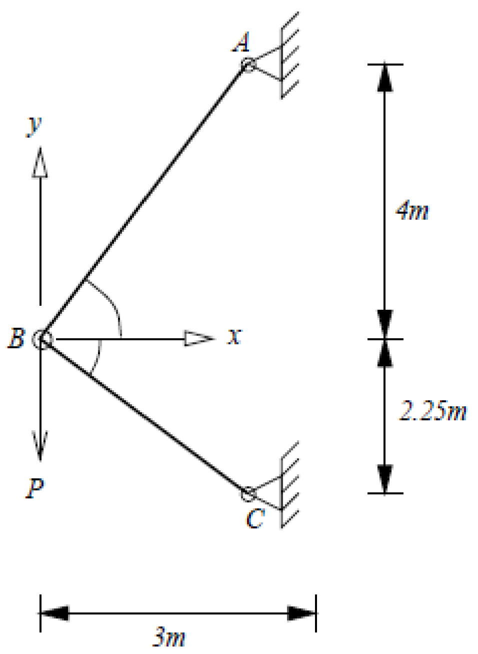


Figure 1-6 – Bars rotated relative to the global coordinate system

If we assume a set of forces **r** and a corresponding set of displacements **d,** we may express the forces and displacements in an arbitrary coordinate system. If we assume two consistent coordinate systems (which means they must be complete and have a basis) *a* and *b*, we assemble the load and displacement vectors in the two coordinate systems respectively; $\mathbf{r}_a$, $\mathbf{d}_a$ and $\mathbf{r}_b$, $\mathbf{d}_b$. The work done by the loading **r** is not dependent on the coordinate system in which it is expressed. Therefore;

$$\mathbf{r}_a^T \mathbf{d}_a = \mathbf{r}_b^T \mathbf{d}_b \qquad \textbf{(12)}$$

If we assume that there exists a linear transformation **T** which transforms a vector in one coordinate system to a vector in the other;

$$\mathbf{x}_b = \mathbf{T}_{ab} \mathbf{x}_a$$

If we apply the transformation **T** on the displacement vector $\mathbf{d}_a$ we find;

$$\mathbf{d}_b = \mathbf{T} \mathbf{d}_a$$

If this relation is inserted into equation **(12)** we find;

$$\mathbf{r}_a^T \mathbf{d}_a = \mathbf{r}_b^T \mathbf{T} \mathbf{d}_a \qquad \textbf{(13)}$$

Since the displacement and loading is arbitrarily chosen, this equation must apply for any $\mathbf{d}_a$, which leads to;

$$\mathbf{r}_a = \mathbf{T}^T \mathbf{r}_b$$

If we insert our new found relations into the equilibrium equation for an element we find the following;

$$\mathbf{r} = \mathbf{k}\mathbf{d} + \mathbf{r}^e$$

$$\mathbf{r}_a = \mathbf{T}^T \mathbf{r}_b = \mathbf{T}^T \left(\mathbf{k}_b \mathbf{d}_b + \mathbf{r}_b^e \right) = \mathbf{T}^T \left(\mathbf{k}_b \mathbf{T} \mathbf{d}_a + \mathbf{r}_b^e \right) = \mathbf{k}_a \mathbf{d}_a + \mathbf{r}_a^e$$

Since the displacements and loads were chosen arbitrarily the relation must be valid for any choice of ; $\mathbf{r}_a$, $\mathbf{d}_a$ and $\mathbf{r}_b$, $\mathbf{d}_b$, we find that;

$$\mathbf{k}_a = \mathbf{T}^T \mathbf{k}_b \mathbf{T}, \mathbf{r}_a = \mathbf{T} \mathbf{r}_b \qquad \textbf{(14)}$$

1.5.1 Application to bars

The stiffness matrix found in sections 1.1and 1.2 is deduced based on the notion that the displacement is only axial, and we have placed the coordinate system with the x-axis in the axial direction. Thus the deformation has only one component at each end. In two dimensional space the displacement component in axial direction for the bar does not necessarily align itself with one of the axes of the coordinate system. In that case, the displacement is axial along an arbitrary line in two dimensional space. Displacement along such a line has components in both axes of the coordinate system, and therefore displacement components for a bar in two dimensional space will generally increase to four rather than two compared to the one-dimensional case.

In Figure 1-7 a bar is expressed in two separate coordinate systems. The coordinate system marked by an asterisk (x’) is the oriented along the axis of the bar. The other coordinate system is a simple Cartesian coordinate system with an origin at the first node.

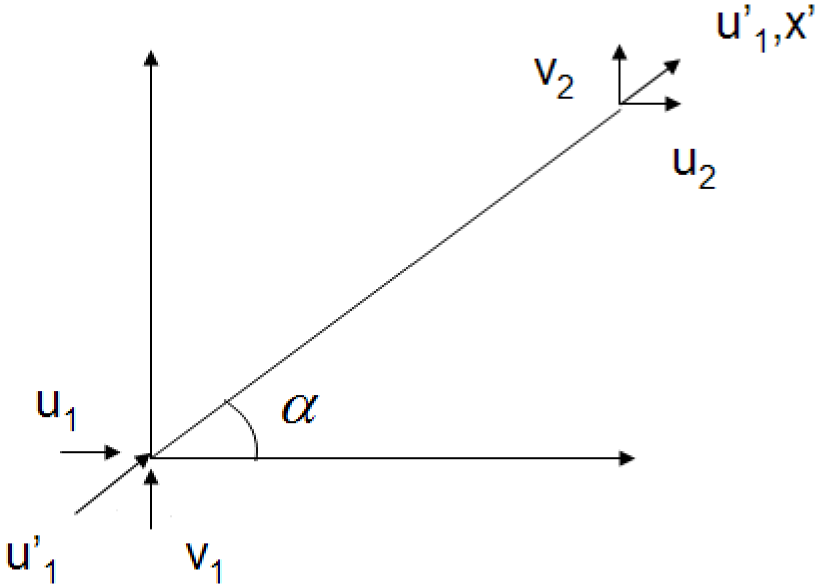


Figure 1-7 – Bar element in local and rotated coordinate systems

We want to express the equilibrium equation for the bar in the Cartesian coordinate system, such that we may combine it with other bars in the same coordinate system. In order to achieve this, we must find the displacement components of u' in x and y. This is done by simple trigonometry;

u_1 is the x-component of u_1'

v_1 is the y-component of u_1'

u_2 is the x-component of u_2'

v_2 is the y-component of u_2'

The transformation matrix is thus;

$$\mathbf{T} = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \end{bmatrix}$$

From equation **(14)** we find that the stiffness matrix and the load vector may be described as follows **(15)**;

$$\mathbf{k} = \mathbf{T}^T \mathbf{k}' \mathbf{T} \qquad \textbf{(15)}$$

$$\mathbf{r} = \mathbf{T}^T \mathbf{r}'$$

1.6 Extended example

A frame of bars consisting of three members is shown in Figure 1-8. In this exercise we shall solve for displacements in the three bars based on analytical calculations.‘

The material and structural parameters are found in the list below;

- E = 207 GPa (Typical for hardened steel)
- A1 = 0.0025 m2 (5cm x 5cm rectangular cross-section)
- A2 = 0.0015 m2 (3cm x 5cm rectangular cross-section)

We already know the element stiffness relations from equation **(9)**. What we need to do may be summarized by the following list;

1. Rotate the element stiffness matrices such that they are all represented in the same coordinate system
2. Augment the individual element stiffness matrices such that they may be summed to a global stiffness matrix
3. Implement boundary conditions and eliminate all rows and columns in the global stiffness matrix related to constrained degrees of freedom
4. Establish a load vector
5. Invert the global stiffness matrix and solve for displacements

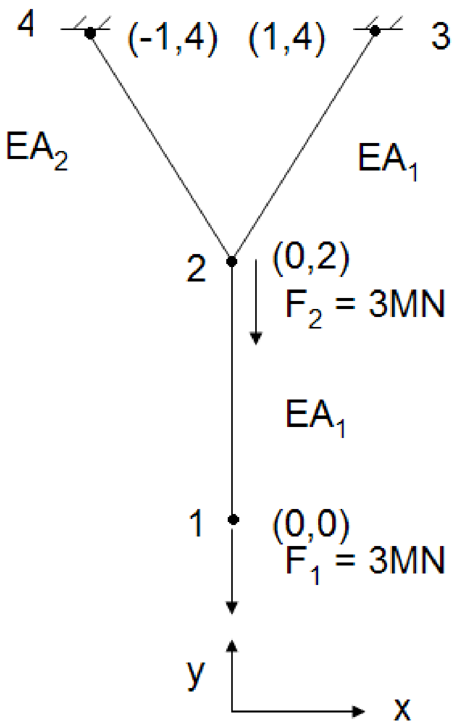


Figure 1-8 – Frame of three bars

1.6.1 Step 1 – Rotate the element stiffness matrices

1.6.1.1 Element 1

Element 1 is defined between nodes 1 and 2, and is rotated 90 degrees relative to the x-axis. Thus the cosine is zero and the sine is 1. From section 1.5.1, we know the form of the transformation matrix, and thus we find;

$$\mathbf{T}_1 = \begin{bmatrix} 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

The rotated element stiffness matrix is found according to equation **(15)**;

$$\mathbf{k}_1^e = \mathbf{T}^T \mathbf{k} \mathbf{T} = \begin{bmatrix} 0 & 0 \\ 1 & 0 \\ 0 & 0 \\ 0 & 1 \end{bmatrix} \frac{EA_1}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} = \frac{EA}{L} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

1.6.1.2 Element 2

Element 2 is defined between nodes 2 and 3, and is rotated relative to the x-axis. We could establish the angle of rotation before we transform the stiffness matrix, but it is easier to compute sines and cosines directly from the triangle. The sine of the angle is the height of the element, which is 2 *m*, divided by the length which is $\sqrt{5}m$. Similarly we find the cosine;

$$\sin\alpha = \frac{2\sqrt{5}}{5}$$

$$\cos\alpha = \frac{\sqrt{5}}{5}$$

This leaves us with a transformation matrix;

$$\mathbf{T} = \frac{\sqrt{5}}{5} \begin{bmatrix} 1 & 2 & 0 & 0 \\ 0 & 0 & 1 & 2 \end{bmatrix}$$

Again we employ equation **(15)**;

$$\mathbf{k}_2^e = \mathbf{T}^T \mathbf{k} \mathbf{T} = \frac{5}{25} \begin{bmatrix} 1 & 0 \\ 2 & 0 \\ 0 & 1 \\ 0 & 2 \end{bmatrix} \frac{EA_1}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} 1 & 2 & 0 & 0 \\ 0 & 0 & 1 & 2 \end{bmatrix} = \frac{EA_1}{5L} \begin{bmatrix} 1 & 2 & -1 & -2 \\ 2 & 4 & -2 & -4 \\ -1 & -2 & 1 & 2 \\ -2 & -4 & 2 & 4 \end{bmatrix}$$

1.6.1.3 Element 3

Element 3 is rotated relative to the x-axis and we perform practically the same calculation as for element 2. Note that the cosine is now negative, since the angle to the x-axis is greater than 90 degrees.

$$\sin\alpha = \frac{2\sqrt{5}}{5}$$

$$\cos\alpha = -\frac{\sqrt{5}}{5}$$

The trigonometric calculations give us the transformation matrix;

$$\mathbf{T} = \frac{\sqrt{5}}{5}\begin{bmatrix} -1 & 2 & 0 & 0 \\ 0 & 0 & -1 & 2 \end{bmatrix}$$

The transformation matrix allows us to use equation **(15)**;

$$\mathbf{k}_3^e = \mathbf{T}^T\mathbf{k}\mathbf{T} = \frac{5}{25}\begin{bmatrix} -1 & 0 \\ 2 & 0 \\ 0 & -1 \\ 0 & 2 \end{bmatrix}\frac{EA_2}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} -1 & 2 & 0 & 0 \\ 0 & 0 & -1 & 2 \end{bmatrix} = \frac{EA_2}{5L}\begin{bmatrix} 1 & -2 & -1 & 2 \\ -2 & 4 & -2 & -4 \\ -1 & -2 & 1 & -2 \\ 2 & -4 & -2 & 4 \end{bmatrix}$$

1.6.2 Step 2 - Augment the individual element stiffness matrices

We have two degrees of freedom in each node (Ref. Figure 1-7), displacement in x-direction and y-direction respectively. When we rotated the stiffness matrix, using the transformation matrix **T**, we committed to using the same displacements as we used when we developed the transformation matrix. Thus the new element displacement and element load vectors should read;

$$\mathbf{d} = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}, \mathbf{r} = \begin{bmatrix} F_{x,1} \\ F_{y,1} \\ F_{x,2} \\ F_{y,2} \end{bmatrix}$$

We observe that we have two degrees of freedom for each node, and we have four nodes in our system. Thus we need to have 8 degrees of freedom in our global system (before boundary conditions, where some of these will be eliminated). When we have 8 degrees of freedom, we need an 8x8 global stiffness matrix, an 8x1 displacement vector and an 8x1 load vector.

1.6.2.1 Augmentation of element stiffness matrix for element 1

The first element is connected to the first 4 degrees of freedom, lateral and vertical displacements in nodes 1 and 2 respectively. The remaining 4 degrees of freedom are however not included for the first element, so for these degrees of freedom the element stiffness matrix should have entries of zero;

$$\mathbf{k}_1 = \frac{EA_1}{L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

1.6.2.2 Augmentation of element stiffness matrix for element 2

The second element is connected to nodes 2 and 3, which results in a relation to degrees of freedom 3 through 6 (as 1 and 2 are related to node 1 and 7 and 8 are related to node 4). The augmented stiffness matrix becomes the following;

$$\mathbf{k}_2 = \frac{EA_2}{5L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 2 & -1 & -2 & 0 & 0 \\ 0 & 0 & 2 & 4 & -2 & -4 & 0 & 0 \\ 0 & 0 & -1 & -2 & 1 & 2 & 0 & 0 \\ 0 & 0 & -2 & -4 & 2 & 4 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

1.6.2.3 Augmentation of element stiffness matrix for element 3

Element 3 is connected to nodes 2 and 4, which relates element 3 to degrees of freedom 3, 4, 7 and 8. The augmented stiffness matrix becomes the following;

$$\mathbf{k}_3 = \frac{EA_2}{5L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & -2 & 0 & 0 & -1 & 2 \\ 0 & 0 & -2 & 4 & 0 & 0 & 2 & -4 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 2 & 0 & 0 & 1 & -2 \\ 0 & 0 & 2 & -4 & 0 & 0 & -2 & 4 \end{bmatrix}$$

1.6.3 Step 3 – Implement boundary conditions

Nodes 3 and 4 are constrained, which means that the 4 degrees of freedom 5 through 8 are constrained. This implies that the rows 5 through 8 and columns 5 through 8 in the element stiffness matrices may be eliminated. The resulting global stiffness matrix may be found by the following expression;

$$\mathbf{K} = \mathbf{k}_1^* + \mathbf{k}_2^* + \mathbf{k}_3^* = \frac{EA_1}{L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} + \frac{EA_1}{5L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 2 \\ 0 & 0 & 2 & 4 \end{bmatrix} + \frac{EA_2}{5L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & -2 \\ 0 & 0 & -2 & 4 \end{bmatrix}$$

$$\mathbf{K} = E\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & \frac{A_1}{2} & 0 & \frac{-A_1}{2} \\ 0 & 0 & \frac{A_1 + A_2}{5\sqrt{5}} & \frac{2A_1 - 2A_2}{5\sqrt{5}} \\ 0 & \frac{-A_1}{2} & \frac{2A_1 - 2A_2}{5\sqrt{5}} & \frac{A_1}{2} + 4\frac{A_1 + A_2}{5\sqrt{5}} \end{bmatrix}$$

Note that the global element stiffness matrix has zero entries in its first row and its first column. This happens since the bar has no stiffness in x-direction (we do not consider bending). Since there are only zero entries in the relevant row and column, the stiffness matrix is not invertible, and the solution for this degree of freedom is irrelevant. Therefore we must eliminate the first row and first column in the global stiffness matrix. The displacement vector is included in the demonstration in order to avoid confusion on which degrees of freedom are solved for and which are not.

$$\mathbf{KD} = E\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & \frac{A_1}{2} & 0 & \frac{-A_1}{2} \\ 0 & 0 & \frac{A_1 + A_2}{5\sqrt{5}} & \frac{2A_1 - 2A_2}{5\sqrt{5}} \\ 0 & \frac{-A_1}{2} & \frac{2A_1 - 2A_2}{5\sqrt{5}} & \frac{A_1}{2} + 4\frac{A_1 + A_2}{5\sqrt{5}} \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} = E\begin{bmatrix} \frac{A_1}{2} & 0 & \frac{-A_1}{2} \\ 0 & \frac{A_1 + A_2}{5\sqrt{5}} & \frac{2A_1 - 2A_2}{5\sqrt{5}} \\ \frac{-A_1}{2} & \frac{2A_1 - 2A_2}{5\sqrt{5}} & \frac{A_1}{2} + 4\frac{A_1 + A_2}{5\sqrt{5}} \end{bmatrix}\begin{Bmatrix} v_1 \\ u_2 \\ v_2 \end{Bmatrix}$$

1.6.4 Step 4 – Establish the load vector

We have vertical loading equal to 3 MN in nodes 1 and 2. The direction is opposite to the y-axis, so the load vector must invert the values for the loads;

$$\mathbf{R} = \begin{bmatrix} -F_1 \\ 0 \\ -F_2 \end{bmatrix}$$

1.6.5 Solve for displacements

If we insert the values for areas, Young modulus and forces we achieve the following system of equations;

$$10^8\begin{bmatrix} 2.5875 & 0 & -2.5875 \\ 0 & 0.7406 & 0.3703 \\ -2.5875 & 0.3703 & 5.5498 \end{bmatrix}\begin{Bmatrix} v_1 \\ u_2 \\ v_2 \end{Bmatrix} = 10^6\begin{Bmatrix} -1 \\ 0 \\ -1 \end{Bmatrix}$$

If we invert the stiffness matrix and solve for the displacements we find the following solution;

$$\begin{Bmatrix} v_1 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} -0.0332 \\ 0.0108 \\ -0.0216 \end{Bmatrix}$$

1.7 Exercises – Mandatory assignment

In this mandatory exercise we shall redo the work done in section 1.6. The following tasks shall be performed;

1. With the below found matlab script and the ansys code, confirm that the solution in section 1.6 is correct. Note that ANSYS gives a vertical displacement in node 1, please explain why the solutions are still consistent.
2. Move node 2 from position (0,2) to position (0.5, 2) and redo the example in section 1.6. When you are finished, confirm your results using ANSYS and Matlab

1.7.1 Matlab script

```
% Declaration of stiffness, forces and geometric parameters


E=207000000000;
A1=0.0025;
A2=0.0015;
L1=2;
L2=sqrt(5);
F1=3000000;
F2=3000000;



% Calculation of element stiffness matrices
%------------------------------------------


% Local stiffness matrix
kloc=[1 -1; -1 1]


% --- Element 1 ---


T=[0 1 0 0; 0 0 0 1];
k_e_1=T'*kloc*T


% --- Element 2 ---
```

```
T=(sqrt(5)/5)*[1 2 0 0; 0 0 1 2];
k_e_2=T'*kloc*T

% --- Element 3 ---

T=(sqrt(5)/5)*[-1 2 0 0; 0 0 -1 2];
k_e_3=T'*kloc*T

% Augmentation of element stiffness matrices to prepare for assembly

% --- Element 1 ---

k1=zeros(8,8); % Create 8x8 matrix of only zero entries
k1(1:4,1:4)=k_e_1; % The first 4x4 matrix in the upper quadrant of k1 is substituted for the local rotated stiffness matrix

% --- Element 2 ---

k2=zeros(8,8); % Create another 8x8 matrix of only zero entries
k2(3:6,3:6)=k_e_2; % row 3 to row 6 and column 3 to column 6 is a 4x4 matrix which relates to degrees of freedom 3 to 6, which in turn relate to nodes 2 and 3

% --- Element 3 ---

k3=zeros(8,8);
k3(3:4,3:4)=k_e_3(1:2,1:2);
k3(3:4,7:8)=k_e_3(1:2,3:4);
k3(7:8,3:4)=k_e_3(3:4,1:2);
k3(7:8,7:8)=k_e_3(3:4,3:4)

% Assembly of stiffness matrices

K_tot=E*A1/L1*k1+E*A1/L2*k2+E*A2/L2*k3;

% Boundary conditions - We know that the degrees of freedom in nodes 3 and
```

```
% 4 are constrained. This means that degrees of freedom 5 through 8 are
% zero.

K=zeros(3,3);
K(1:3,1:3)=K_tot(2:4,2:4);

% Forces are acting oppositely to the y-axis and therefore they must be
% inverted in the global load vector.

R=[-F1; 0; -F2];

D=inv(K)*R
```

1.7.2 ANSYS script

```
/BATCH,LIST
/FILNAM,ex411
/TITLE, LineÃ¦r statisk analyse rett stav
/PREP7
ET,1,1 ! LINK1 elementer
R,1,0.0025 ! Tverrsnitts areal til staven
R,2,0.0015
MP,EX,1,207e9 ! E-modulen
!Geometri ("solid modelling")
K,1,0,0 ! Punkt A er origo
K,2,0,2 ! Punkt B er i x=0, y=2
K,3,1,4 ! Punkt C i x=1, y=4
K,4,-1,4 ! Punkt D i x=-1, y=4
L,1,2 ! Linje AB
L,2,3 ! Linje BC
L,2,4 ! Linje BD
!Inndeling i elementer
LESIZE,1,,,1 ! Deklarer at linje AB skal inndeles i ett element
LESIZE,2,,,1 ! Deklarerer at linje BC skal inndeles i ett element
LESIZE,3,,,1 ! Deklarerer at linje BD skal inndeles i ett element
REAL,1 ! Bruk tverrsnittsareal nr. 1 for inndelingen (neste to linjer)
```

```
LMESH,1 ! Inndeling av linje 1
LMESH,2 ! Inndeling av linje 2
REAL,2 ! Bruk tverrsnittsareal nr. 2 for inndelingen (neste linje)
LMESH,3 !Inndeling av linje 3
FINISH ! Ut av Preprossessoren
/SOLU ! Læøsningsprossessoren
ANTYPE, STATIC ! Statisk analyse (default)
DK,3,all ! Ingen forskyvninger i punkt C
DK,4,all ! Ingen forskyvninger i punkt D
FK,1,fy,-3e6 ! Belastning i punkt A
FK,2,fy,-3e6  ! Belastning i punkt B
DTRAN ! Overfører grensebetetingelser til elementmodell
SBCTRAN ! og belastningen
SOLVE ! Løsningsprosedyren
FINISH ! Ut av Løsningsprossessoren
/POST1 ! Postprossessoren
SET ! Last inn analyseresultatene
PLDISP,1 ! Deformert konstruksjon
PRNSOL,U,COMP ! Utskrift av forskyvningene (global akse)
LOCAL,11,0,,,,53.1301 ! Lokalt aksesystem
RSYS,11 ! aktiveres og brukes til å lese
PRNSOL,U,COMP ! forskyvningene og
PRESOL, F ! kreftene
PRESOL,ELEM ! Ta ut tilgjengelige elementresultater (aksialkrefter)
```

2 CHAPTER 2 –AN INTRODUCTION TO ENERGY METHODS

Cook : 4.1, 4.2, 4.3, 4.4 (parts of section 4.4, will be revisited in chapters 4 and 5 of this compendium)

2.1 An introduction to energy methods

In classic solid mechanics it is common to deduce equations of motion by equilibrium calculations, particularly for bars, beams and plates. In a finite element context it is however not common to use equilibrium to deduce equations of motion. The finite element method is based on assuming displacement functions between nodes. The nature of the finite element method will therefore render energy methods much more applicable, since energy methods are also based on assuming a set of deformations in whichever directions are relevant (i.e. axial direction for bars, vertical and axial direction for beams, all directions for solids etc.). When we assume a set of displacement functions in the element method, we may use energy methods to use these assumed displacements in order to determine element equilibrium equations on the form **kd** = **r**. Specifically, energy methods are the most efficient manner of determining both the stiffness matrix **k**, and the only general manner of determining a consistent load vector **r**.

2.1.1 Potential energy in a bar

If we examine the axially loaded, simply supported bar in **Error! Reference source not found.**, we find that the applied axial force is *F*, the Young modulus is E, the cross-sectional area is A and the length of the bar is L.

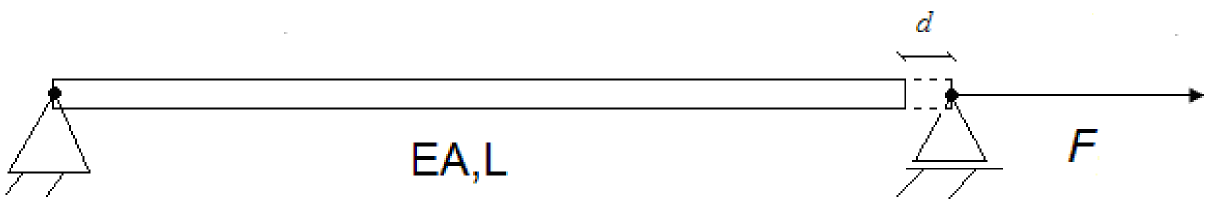


Figure 2-1 – Axially loaded, simply supported bar

We know from Section 1.2 that the displacement *d* of the bar is linear in *F*, and we may express the force displacement in terms of a function (Ref.Figure 2-2)

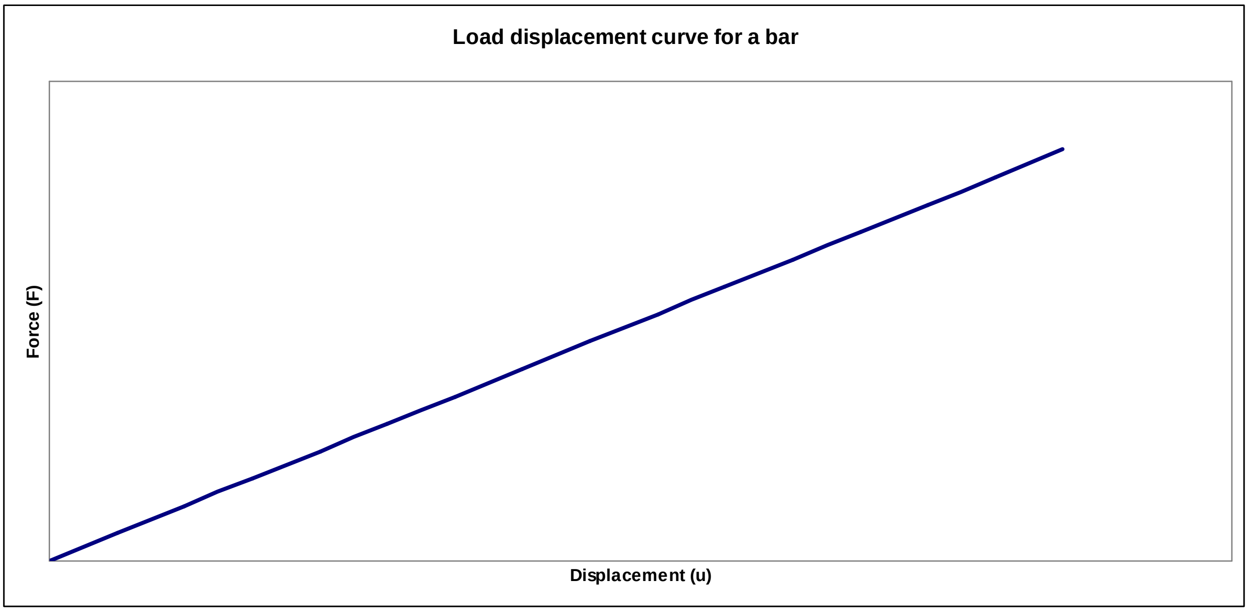


Figure 2-2 – Load displacement curve for an axially loaded bar

If we want to determine the potential energy in the bar, we can integrate the load displacement curve **(16)**;

$$F = \frac{EA}{L}u \Rightarrow u(F) = \frac{L}{EA}F \tag{16}$$

$$U = \int_0^F \frac{L}{EA} u du = \frac{L}{2EA} u^2 \Big|_{F=0}^{F=F} = \frac{L}{2EA}F^2 = \frac{EA}{2L}u^2$$

In the above equation, *U* has been defined as the total potential energy. The work done by the axial force is simply the force times the displacement **(17)**.

$$\Omega = Fd \tag{17}$$

In the book by Cook et. al., the work done by external forces has been given the symbol Ω. It is also common to use the symbol *H*, which may be found in several other references, however since Cook et. al. is our main reference, we shall use Ω.

The difference between the internal energy, which in this case is stored elastic energy, and the external work done is called the potential energy functional;

$$\Pi = U - \Omega \tag{18}$$

The total potential energy is given the symbol Π.

2.1.2 The principle of minimum potential energy

The principle of minimum potential energy may be stated as the following;

> *From all admissible deformations, the system which fulfils the equilibrium equations of a conservative system is the system which has the least potential energy*

By a conservative system it is meant that the potential energy of an arbitrary deformation configuration is path independent, i.e. that the potential energy does not depend on the load deformation history. A linearly elastic deformation of a bar is an example of a conservative system. A plastically deformed bar is an example of a non-conservative system, since a deformation may be achieved by either elastic or inelastic load displacement history. A permanent plastic deformation has different energy than a linear elastic deformation, and these two deformations may be equal. The work done in these two cases is obviously different, and thus load deformation is path dependent.

If we insert the two expressions we have deduced, for internal potential energy **(16)** and work done by external forces **(17)** respectively into the equation for the total potential energy **(18)**, we get an algebraic expression for the total potential energy;

$$\Pi = \frac{EA}{2L}u^2 - Fu \tag{19}$$

We observe that the total potential energy (functional) is a second order equation in *u*, which means it has a global minimum as the second order term is positive. According to the principle of minimum potential energy, we must find the configuration with minimum potential energy in order to find the system which fulfils the equilibrium equations. Obviously, the function has a global minimum, and we may generally find that global minimum by finding the stationary value of the derivative (Ref. **(20)**);

$$d\Pi = \frac{EA}{2L}udu - Fdu = \left(\frac{\mathrm{EA}}{\mathrm{L}}u - F\right)du = 0 \Rightarrow u = \frac{FL}{EA} \tag{20}$$

In equation **(19)** we have the benefit that the function is a second degree polynomial in *u*, which means we know that there is only one stationary value, and since the second order term is positive, that stationary value is a global minimum. For a more general system, it is not obvious however that the stationary value is unique, nor that it is a global minimum. For the purposes of this course however, all systems we shall investigate will only have one unique stationary value for the potential energy functional, and that stationary value shall be the a global minimum, which means the principle of minimum potential energy applies, and we may find the stationary value by a differentiation/*variation* operation. Note that this does not apply for nonlinear systems, and in those cases we need to sort through different possible solutions in order to find the physically relevant one. Nonlinear analyses will be investigated in module 2 of this course.

2.1.3 Equilibrium of a bar revisited

The equilibrium for a simply supported bar was established in equation **(20)**. If we do not have any boundary conditions, and we wish to establish the potential energy in a generally supported and generally loaded bar, we may simply wait to impose boundary conditions until after we have established the equilibrium equations. Consider the bar from Figure 1-2, given again here as Figure 2-3 for ease of reference.

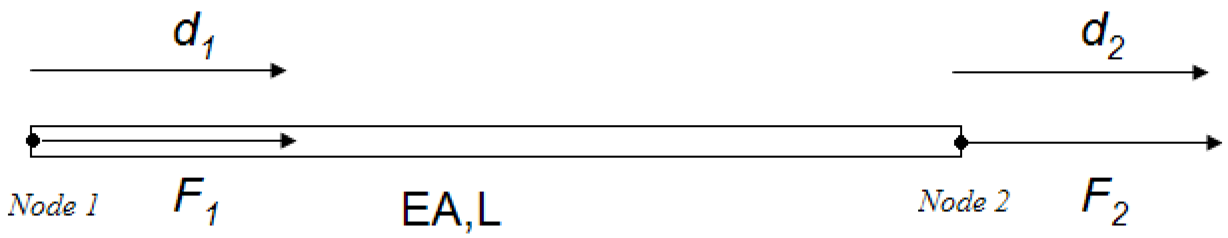


Figure 2-3 – A bar element

We choose two displacement functions **(21)**;

$$u_1 = 1 - \frac{x}{L}$$
$$u_2 = \frac{x}{L} \quad \textbf{(21)}$$

The functions have the properties that u_1 is unity at node 1 and zero at node 2. u_2 is unity at node 2 and zero at node 1. The functions are plotted in Figure 2-4.

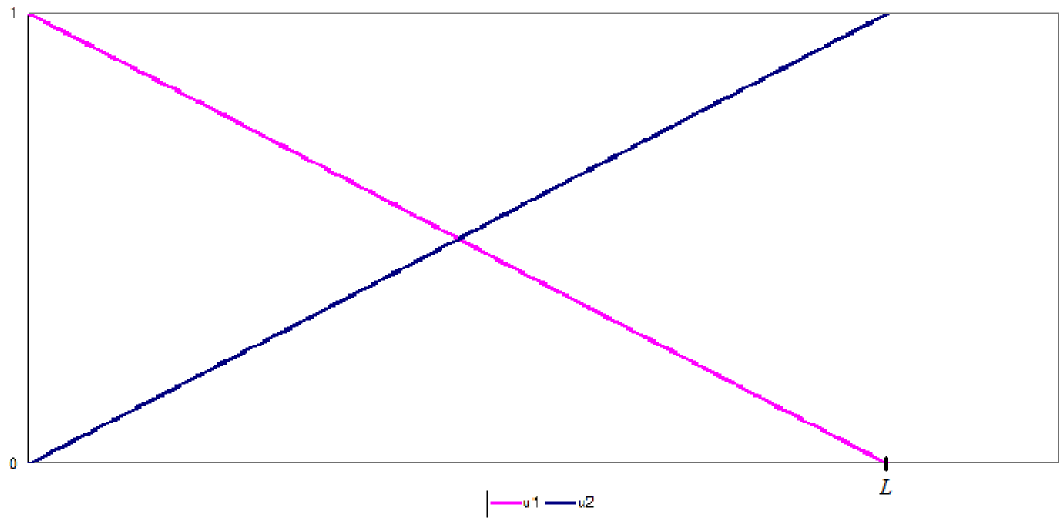


Figure 2-4 Shape functions u_1 and u_2

The importance of choosing the given displacement functions cannot be overstated. The benefits of choosing shape functions in this manner are the following;

- If u_1 or u_2 are set to zero, either a boundary condition at node 1 or node 2 of zero displacement will be automatically fulfilled for any end displacement on the other node. This is a formal requirement and a necessary condition for a shape function if it is implemented using the principle of minimum potential energy.
- Linear functions are chosen for a bar, since the displacement of a uniform bar is always linear, as such they are capable of returning exact deformations relative to the theory applied

- Continuity at each node is assured
- Rotational continuity is not assured, but bar theory does not require rotational continuity between individual members. (For beams for instance, we cannot use linear functions since rotational continuity is required).

Before we start to introduce the assumed displacement functions, we shall revisit the deduction for equilibrium of a bar element using the principle of minimum potential energy. The work done by external forces and the stored potential energy in the resulting displacements according to the assumed displacement functions may be calculated based on equation **(18)**;

$$U = \frac{EA}{2L}(u_2 - u_1)^2$$
$$\Omega = F_1 u_1 + F_2 u_2$$
$$\Pi = \frac{EA}{2L}(u_2 - u_1)^2 - F_1 u_1 - F_2 u_2$$

As we have found the total potential energy function, we may invoke the principle of minimum potential energy and differentiate to find the stationary value;

$$d\Pi = \frac{EA}{L}(u_2 - u_1)(du_2 - du_1) - F_1 du_1 - F_2 du_2 = 0$$
$$\Rightarrow \left(\frac{EA}{L}(u_1 - u_2) - F_1\right)du_1 + \left(\frac{EA}{L}(u_2 - u_1) - F_2\right)du_2 = 0$$

Since displacements at either end of the bar are admissible, the sum of the two differentials du_1 and du_2 can only be zero for arbitrarily chosen du_1 and du_2 if du_1 and du_2 are zero individually (Later we shall discover that this statement is called the fundamental theorem of variational calculus). This in turn means we can deduce two equations from the differential of the total potential energy functional;

$$\left.\begin{aligned}\frac{EA}{L}(u_1 - u_2) = F_1\\ \frac{EA}{L}(u_2 - u_1) = F_2\end{aligned}\right\} \Rightarrow \frac{EA}{L}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}u_1\\ u_2\end{bmatrix} = \begin{bmatrix}F_1\\ F_2\end{bmatrix}$$

We conclude that the equilibrium equation for a bar element may be deduced using the principle of minimum potential energy.

When we introduce shape functions, we can no longer view the displacement related to each node. We must instead integrate the stored elastic energy along the length of the element, since any combination of displacement functions could theoretically be applied. The expression for the potential energy stored as elastic energy in a bar may be expressed on the following form;

$$U = \frac{1}{2}\int_0^L EA\left(\frac{du}{dx}\right)^2 dx$$

If we introduce our assumed displacement functions we may integrate in x, assuming that E and A are constant along the length of the bar.

$$U = \frac{1}{2}\int_0^L EA\left(d_1\left(\frac{-1}{L}\right) + d_2\frac{1}{L}\right)^2 dx = \frac{1}{2}\int_0^L EA\left(d_1^2\frac{1}{L^2} - d_1 d_2\frac{1}{L^2} + d_2^2\frac{1}{L^2}\right)dx$$

$$U = \frac{EA}{2}\left[d_1^2\frac{x}{L^2} - 2d_1 d_2\frac{x}{L^2} + d_2^2\frac{x}{L^2}\right]_{x=0}^{x=L} = \frac{EA}{2L}\left(d_1^2 - 2d_1 d_2 + d_2^2\right)$$

Now we have our total potential energy expressed in terms of two undetermined coefficients which give the amplitudes of our assumed displacement polynomials. The remaining issue is to determine the work done by external forces;

$$\Omega = F_1 d_1 + F_2 d_2$$

In order to determine these coefficients we invoke the principle of minimum potential energy;

$$d\Pi = \frac{EA}{2L}\left(2d_1 dd_1 - 2d_2 dd_1 - 2d_1 dd_2 + 2d_2 dd_2\right) - F_1 dd_1 - F_2 dd_2 = 0$$

Since the above equation is valid for admissible d_1 and d_2, we may individually set dd_1 and dd_2 to zero;

$$\left.\begin{array}{l} \frac{EA}{L}\left(d_1 - d_2\right) = F_1 \\ \frac{EA}{L}\left(-d_1 + d_2\right) = F_2 \end{array}\right\} \Rightarrow \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

To repeat the process of assembly and a practical approach to applying boundary conditions we shall have another extended example of a bar frame.

2.1.4 Extended example – Bar Frame II

We consider a vertically loaded bar frame, shown in Figure 2-5

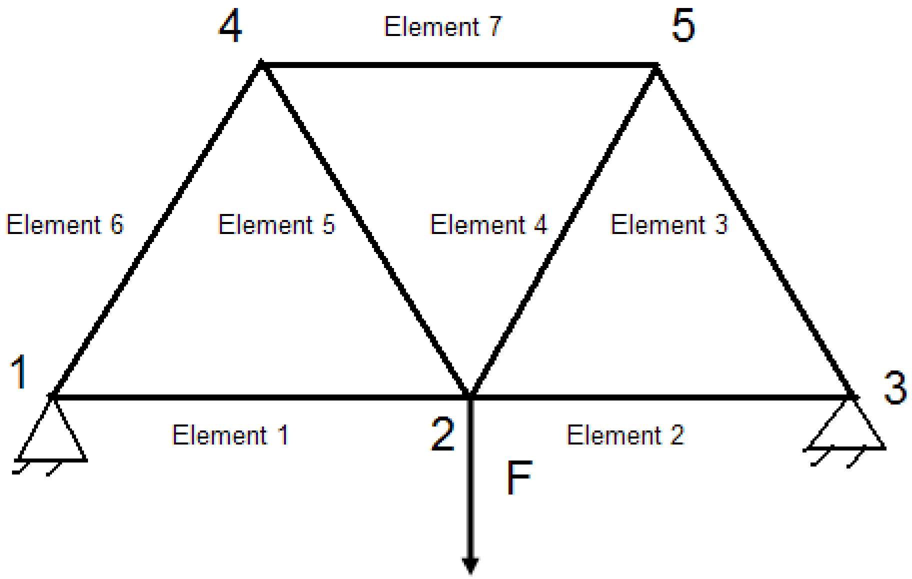


Figure 2-5 – A vertically loaded frame of bars

The nodal coordinates are given in the following list, along with the material properties and cross-sectional area of the bars;

- E=1
- A=1
- F=0.1
- Node 1: (0,0)
- Node 2: (0,2)
- Node 3: (0,4)
- Node 4: (1,1)
- Node 5: (3,1)

We follow the same procedure as we chose for the bar frame in the extended example of section 1.6;

1. Rotate the element stiffness matrices such that they are all represented in the same coordinate system
2. Augment the individual element stiffness matrices such that they may be summed to a global stiffness matrix
3. Implement boundary conditions and eliminate all rows and columns in the global stiffness matrix related to constrained degrees of freedom
4. Establish a load vector
5. Invert the global stiffness matrix and solve for displacements

2.1.4.1 Rotation of element stiffness matrices

The basic stiffness matrix for a bar element is;

$$\mathbf{k} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Elements 1, 2 and 7 have the same orientation, length and axial stiffness (EA). Thus they have identical element stiffness matrices. The direction of each element is parallel to the x-axis, which means the sine is zero and the direction cosine is 1. Thus the transformation matrix is simply;

$$\mathbf{T}_1 = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \end{bmatrix}$$

We find the element stiffness matrix expressed in the global coordinate system;

$$\mathbf{k}_1^e = \mathbf{T}_1^T \mathbf{k} \mathbf{T}_1 = \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$\mathbf{k}_2^e = \mathbf{k}_7^e = \mathbf{k}_1^e$$

Elements 6 and 4 have the same angle to the global x-axis and the same length, and therefore they as well have the same element stiffness matrix (and thus the same transformation matrix).

$$\mathbf{T}_6 = \frac{\sqrt{2}}{2}\begin{bmatrix} 1 & 1 & 0 & 0 \\ 0 & 0 & 1 & 1 \end{bmatrix}$$

$$\mathbf{k}_6^e = \mathbf{T}_6^T \mathbf{k} \mathbf{T}_6 = \frac{1}{2}\begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix}$$

$$\mathbf{k}_4^e = \mathbf{k}_6^e$$

Elements 3 and 5 have the same angle to the global x-axis and the same length. This means the third and final element stiffness matrix configuration may be calculated for element 3 and 5 both;

$$\mathbf{T}_5 = \frac{\sqrt{2}}{2}\begin{bmatrix} -1 & 1 & 0 & 0 \\ 0 & 0 & -1 & 1 \end{bmatrix}$$

$$\mathbf{k}_5^e = \mathbf{T}_5^T \mathbf{k} \mathbf{T}_5 = \frac{1}{2}\begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix}$$

$$\mathbf{k}_3^e = \mathbf{k}_5^e$$

2.1.4.2 Augmentation of the element stiffness matrix to a global system

The system in Figure 2-5 is a two-dimensional frame of bars, which means each node has two degrees of freedom. The total number of degrees of freedom in the system (including those constrained by boundary conditions) is 10. The global displacement vector may therefore be written as follows **(22)**;

$$\mathbf{D}_a = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \\ u_5 \\ v_5 \end{bmatrix} \quad \textbf{(22)}$$

The small suffix *a* on **D** has been included to show that this is the augmented global displacement vector, to distinguish it from the full global displacement vector in which we have included boundary conditions. The sub indices on *u* and *v* indicate node number and *u* indicates displacement in x-direction as v indicates displacement in y-direction. Since $\mathbf{D}_a$ is a 10x1 vector, $\mathbf{k}_i^a$ (i.e. the augmented element stiffness matrices) have dimension 10x10.

Augmentation of element stiffness matrix for element 1

Element 1 has two degrees of freedom in nodes 1 and 2. this means Element 1 relates to degrees of freedom 1 through 4. The augmented element stiffness matrix becomes;

$$\mathbf{k}_1^a = \frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Augmentation of element stiffness matrix for element 2

Element 2 relates to degrees of freedom 3 through 6 via nodes 2 and 3;

$$\mathbf{k}_2^a = \frac{EA}{L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Augmentation of element stiffness matrix for element 3

Element 3 has degrees of freedom in nodes 3 and 5, which means element 3 relates to degrees of freedom 5, 6, 9 and 10;

$$\mathbf{k}_3^a = \frac{EA}{2L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 & 0 & 0 & -1 & 1 \\ 0 & 0 & 0 & 0 & -1 & 1 & 0 & 0 & 1 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 1 & 0 & 0 & 1 & -1 \\ 0 & 0 & 0 & 0 & 1 & -1 & 0 & 0 & -1 & 1 \end{bmatrix}$$

<u>Augmentation of element stiffness matrix for element 4</u>

Element 4 has degrees of freedom in nodes 2 and 5, which means degrees of freedom 3, 4, 9 and 10 are related to the augmented stiffness matrix;

$$\mathbf{k}_4^a = \frac{EA}{2L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 1 & 0 & 0 & 0 & 0 & -1 & -1 \\ 0 & 0 & 1 & 1 & 0 & 0 & 0 & 0 & -1 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & -1 & 0 & 0 & 0 & 0 & 1 & 1 \\ 0 & 0 & -1 & -1 & 0 & 0 & 0 & 0 & 1 & 1 \end{bmatrix}$$

<u>Augmentation of element stiffness matrix for element 5</u>

Element 5 is related to nodes 2 and 4, which means element 5 is related to degrees of freedom 3, 4, 7 and 8;

$$\mathbf{k}_5^a = \frac{EA}{2L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & -1 & 0 & 0 & -1 & 1 & 0 & 0 \\ 0 & 0 & -1 & 1 & 0 & 0 & 1 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 1 & 0 & 0 & 1 & -1 & 0 & 0 \\ 0 & 0 & 1 & -1 & 0 & 0 & -1 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

<u>Augmentation of element stiffness matrix for element 6</u>

Element 6 is related to nodes 1 and 4, which means element 6 is related to degrees of freedom 1, 2, 7 and 8;

$$\mathbf{k}_6^a = \frac{EA}{2L}\begin{bmatrix} 1 & 1 & 0 & 0 & 0 & 0 & -1 & -1 & 0 & 0 \\ 1 & 1 & 0 & 0 & 0 & 0 & -1 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & -1 & 0 & 0 & 0 & 0 & 1 & 1 & 0 & 0 \\ -1 & -1 & 0 & 0 & 0 & 0 & 1 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

<u>Augmentation of element stiffness matrix for element 7</u>

Element 7 has degrees of freedom in nodes 4 and 5 which relates to degrees of freedom 7 through 10.

$$\mathbf{k}_7^a = \frac{EA}{L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

2.1.4.3 Summation to global stiffness matrix

Note that when the matrices have been summed, it has been included that the element lengths for the lateral elements are 2 whereas the element lengths for the angled elements is $\sqrt{2}$.

$$\mathbf{K}_a = \frac{1}{2}\begin{bmatrix} \frac{2+\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -1 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 \\ \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 \\ -1 & 0 & 2+\sqrt{2} & 0 & -1 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \\ 0 & 0 & 0 & \sqrt{2} & 0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\ 0 & 0 & -1 & 0 & \frac{2+\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \\ 0 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\ -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & 1+\sqrt{2} & 0 & -1 & 0 \\ -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & 0 & \sqrt{2} & 0 & 0 \\ 0 & 0 & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -1 & 0 & 1+\sqrt{2} & 0 \\ 0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & 0 & \sqrt{2} \end{bmatrix}$$

2.1.4.4 Implementation of boundary conditions

From the constraint in node 1 we must fix degrees of freedom 1 and 2. From the constraint in node 3 we must fix degrees of freedom 5 and 6. We achieve this by eliminating rows and columns 1, 2, 5 and 6 from the global stiffness matrix, and entries 1, 2, 5 and 6 from the global load vector;

$$
\mathbf{K}_a = \frac{1}{2}\begin{bmatrix}
\frac{2+\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -1 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 \\
\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 \\
-1 & 0 & 2+\sqrt{2} & 0 & -1 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \\
0 & 0 & 0 & \sqrt{2} & 0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\
0 & 0 & -1 & 0 & \frac{2+\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \\
0 & 0 & 0 & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\
-\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 0 & 0 & 1+\sqrt{2} & 0 & -1 & 0 \\
-\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & 0 & \sqrt{2} & 0 & 0 \\
0 & 0 & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -1 & 0 & 1+\sqrt{2} & 0 \\
0 & 0 & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & 0 & \sqrt{2}
\end{bmatrix}
$$

The resulting global stiffness matrix becomes;

$$
\mathbf{K} = \frac{1}{2}\begin{bmatrix}
2+\sqrt{2} & 0 & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \\
0 & \sqrt{2} & \frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\
-\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & 1+\sqrt{2} & 0 & -1 & 0 \\
\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & \sqrt{2} & 0 & 0 \\
-\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & -1 & 0 & 1+\sqrt{2} & 0 \\
\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} & 0 & 0 & 0 & \sqrt{2}
\end{bmatrix}
$$

The relevant degrees of freedom are the initial augmented global displacement vector, where degrees of freedom 1, 2, 5 and 6 have been excluded;

$$\mathbf{D} = \begin{bmatrix} u_2 \\ v_2 \\ u_4 \\ v_4 \\ u_5 \\ v_5 \end{bmatrix}$$

2.1.4.5 Establish a load vector

The global load vector consists of a single point load in node 2, with direction along the y-axis. This is related to displacement v_2 and must therefore have the same place in the loading vector as the displacement v_2. There are no other loads, and therefore the global load vector may be written as follows;

$$\mathbf{R} = \begin{bmatrix} 0 \\ -0.1 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

2.1.4.6 Solution

The displacement vector may be found as by the following equation;

$$\mathbf{D} = \mathbf{K}^{-1}\mathbf{R} = \begin{bmatrix} 0 \\ -0.4828 \\ 0.1 \\ -0.2414 \\ -0.1 \\ -0.2414 \end{bmatrix}$$

A Matlab script for the solution of this exercise is given below;

```
E=1;
A=1;
L_straight=2;
L_angled=sqrt(2);

T6=[sqrt(2)/2 sqrt(2)/2 0 0; 0 0 sqrt(2)/2 sqrt(2)/2];
T5=[-sqrt(2)/2 sqrt(2)/2 0 0; 0 0 -sqrt(2)/2 sqrt(2)/2];
```

```
k1e=(E*A/L_straight)*[1 0 -1 0; 0 0 0 0; -1 0 1 0; 0 0 0 0];
k2e=k1e;
k7e=k1e;

k=[1 -1; -1 1];

k6e=(E*A/L_angled)*T6'*k*T6;
k5e=(E*A/L_angled)*T5'*k*T5;
k4e=k6e;
k3e=k5e;

K=zeros(10,10);

% ELEMENT 1
K(1:4,1:4)=k1e;

% ELEMENT 2
K(3:6,3:6)=K(3:6,3:6)+k2e;

% ELEMENT 3
K(5:6,5:6)=K(5:6,5:6)+k3e(1:2,1:2);
K(5:6,9:10)=K(5:6,9:10)+k3e(1:2,3:4);
K(9:10,5:6)=K(9:10,5:6)+k3e(3:4,1:2);
K(9:10,9:10)=K(9:10,9:10)+k3e(3:4,3:4);

% ELEMENT 4
K(3:4,3:4)=K(3:4,3:4)+k4e(1:2,1:2);
K(3:4,9:10)=K(3:4,9:10)+k4e(1:2,3:4);
K(9:10,3:4)=K(9:10,3:4)+k4e(3:4,1:2);
K(9:10,9:10)=K(9:10,9:10)+k4e(3:4,3:4);

% ELEMENT 5
K(3:4,3:4)=K(3:4,3:4)+k5e(1:2,1:2);
K(3:4,7:8)=K(3:4,7:8)+k5e(1:2,3:4);
```

```
K(7:8,3:4)=K(7:8,3:4)+k5e(3:4,1:2);
K(7:8,7:8)=K(7:8,7:8)+k5e(3:4,3:4);

% ELEMENT 6
K(1:2,1:2)=K(1:2,1:2)+k6e(1:2,1:2);
K(1:2,7:8)=K(1:2,7:8)+k6e(1:2,3:4);
K(7:8,1:2)=K(7:8,1:2)+k6e(3:4,1:2);
K(7:8,7:8)=K(7:8,7:8)+k6e(3:4,3:4);

% ELEMENT 7
K(7:8,7:8)=K(7:8,7:8)+k7e(1:2,1:2);
K(7:8,9:10)=K(7:8,9:10)+k7e(1:2,3:4);
K(9:10,7:8)=K(9:10,7:8)+k7e(3:4,1:2);
K(9:10,9:10)=K(9:10,9:10)+k7e(3:4,3:4);

K_glob=zeros(6,6);
K_glob(1:2,1:2)=K(3:4,3:4);
K_glob(1:2,3:6)=K(3:4,7:10);
K_glob(3:6,1:2)=K(7:10,3:4);
K_glob(3:6,3:6)=K(7:10,7:10);

R=zeros(6,1);
R(2)=-0.1;
D=inv(K_glob)*R
```

An Ansys script is given below for the solution of this exercise;

```
/BATCH,LIST
/FILNAM,ex411
/TITLE, LineÃ¦r statisk analyse rett stav
/PREP7
ET,1,1 ! LINK1 elementer
R,1,1 ! Tverrsnitts areal til staven
MP,EX,1,1 ! E-modulen
!Geometri ("solid modelling")
K,1,0,0 ! Punkt A er i origo
K,2,2,0 ! Punkt B er i x=2, y=0
K,3,4,0 ! Punkt C i x=4, y=0
K,4,1,1 ! Punkt D i x=1, y=1
K,5,3,1 ! Punkt E i x=3, y=1
L,1,2 ! Linje 12
L,2,3 ! Linje 23
L,3,5 ! Linje 35
L,2,5 ! Linje 25
L,2,4 ! Linje 24
L,1,4 ! Linje 14
L,4,5 ! Linje 45
!Inndeling i elementer
LESIZE,ALL,,,1 ! Deklarer at alle linjer skal inndeles i ett element
REAL,1 ! Bruk tverrsnittsareal nr. 1 for inndelingen (neste to linjer)
LMESH,ALL ! Inndeling av alle linjer
FINISH ! Ut av Preprossessoren
/SOLU ! LÃ¸sningsprossessoren
ANTYPE, STATIC ! Statisk analyse (default)
DK,1,all ! Ingen forskyvninger i punkt A
DK,3,all ! Ingen forskyvninger i punkt C
FK,2,fy,-0.1 ! Belastning i punkt B
DTRAN ! OverfÃ¸rer grensebetetingelser til elementmodell
SBCTRAN ! og belastningen
SOLVE ! LÃ¸sningsprosedyren
```

```
FINISH ! Ut av LÃ¸sningsprossessoren
/POST1 ! Postprossessoren
SET ! Last inn analyseresultatene
PLDISP,1 ! Deformert konstruksjon
PRNSOL,U,COMP ! Utskrift av forskyvningene (global akse)
LOCAL,11,0,,,,53.1301 ! Lokalt aksesystem
PRESOL,ELEM ! Ta ut tilgjengelige elementresultater (aksialkrefter)
```

2.2 Exercises

2.2.1 Continuation of extended example 2.1.4

a) Before boundary conditions are applied to the global stiffness matrix, constrained degrees of freedom are still present in the stiffness matrix. If we multiply the global stiffness matrix with the nodal displacements we get the forces in each node, which allows us to find the reaction forces in nodes 1 and 3. Find the reaction forces.

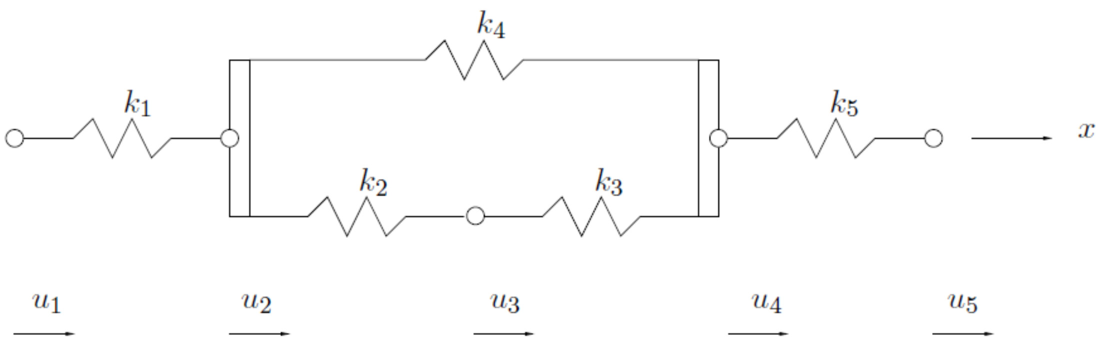
b) Apply cross-sectional area *A*=0.015 and Young modulus E=207 GPa. Substitute the force F with ---. Recalculate the nodal displacements and reaction forces.

c) The steel is of type X65, which has a specified minimum yield stress of 450 MPa. However, bars are inaccurate (no bending for instance), and therefore the allowed axial stress is well below the yield limit. The allowable axial stress is 250 MPa. Find the maximum load F which keeps the axial stress below 250 MPa anywhere in the structure

d) Use ANSYS to experiment on the consequences of allowing lateral displacement in node no. 3 (i.e. change the boundary condition to only constrain the vertical degree of freedom in node 3).

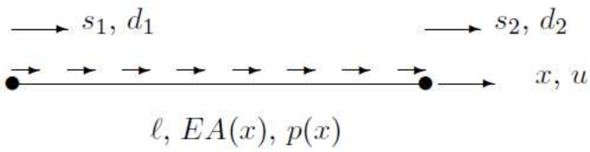
2.2.2

In the figure below a system of springs is shown. Find the global stiffness matrix for the system of springs;



2.2.3

I denne oppgaven skal vi benytte *direkte* oppsetting av elementsivhetsmatrisen for et stavelement basert på løsningen av en *ordinær differensialligning.*



a) Differentialligningen for et stavelement er gitt ved utrykket:

$$-\frac{d}{dx}\left(EA(x)\frac{du(x)}{dx}\right) = p(x)$$

hvor $EA(x)$ er aksiell stivhet for staven, $u(x)$ er forskvninger og $p(x)$ er en aksiell jevnt fordelt last langs stavelementet.

Set $EA(x) = 1$ konstant og $p(x) = 0$ langs stav aksen. Set opp uttrykket for den nye differentialligningen.

b) Finn det generelle utrykket for løsningen av den nye differentialligningen.

c) Elementets lengde er $\ell = 1$. Benytt randkravene

$$u(x=0) = d_1 \quad \text{og} \quad u(x=1) = d_2$$

til å finne den spesielle løsningen for dette problemet.

d) Spenningene i staven kan finnes ved uttrykket

$$\sigma = E\frac{du}{dx}$$

E er elastisitetsmodulen til materialet (materiallov), og $\varepsilon = \frac{du}{dx}$ er aksialtøyningen i staven. Kreftene i staven kan nå finnes fra uttrykket

$$N = A\sigma$$

hvor A er stavens tverrsnittsareal. Benytt dette til å finne en sammenheng mellom endekraft og endeforskyvning for de fire tilfellene:

i) Finn $N_1 = N(x = 0)$ for $d_1 = 1$ og $d_2 = 0$.

ii) Finn $N_1 = N(x = 0)$ for $d_1 = 0$ og $d_2 = 1$.

iii) Finn $N_2 = N(x = 1)$ for $d_1 = 1$ og $d_2 = 0$.

iv) Finn $N_2 = N(x = 1)$ for $d_1 = 0$ og $d_2 = 1$.

e) Benytt svarene i oppgave d) (og definisjonen av at s_1 er rettet motsatt vei av N) til å etablere relasjonen

$$\begin{Bmatrix} s_1 \\ s_2 \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix}$$

f) Finn samme relasjon som i e) men med aksiell stivhet $EA = EA$ konstant, og stavens lengde satt lik $\ell = \ell$.

3 REFERENCES

/1/