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.

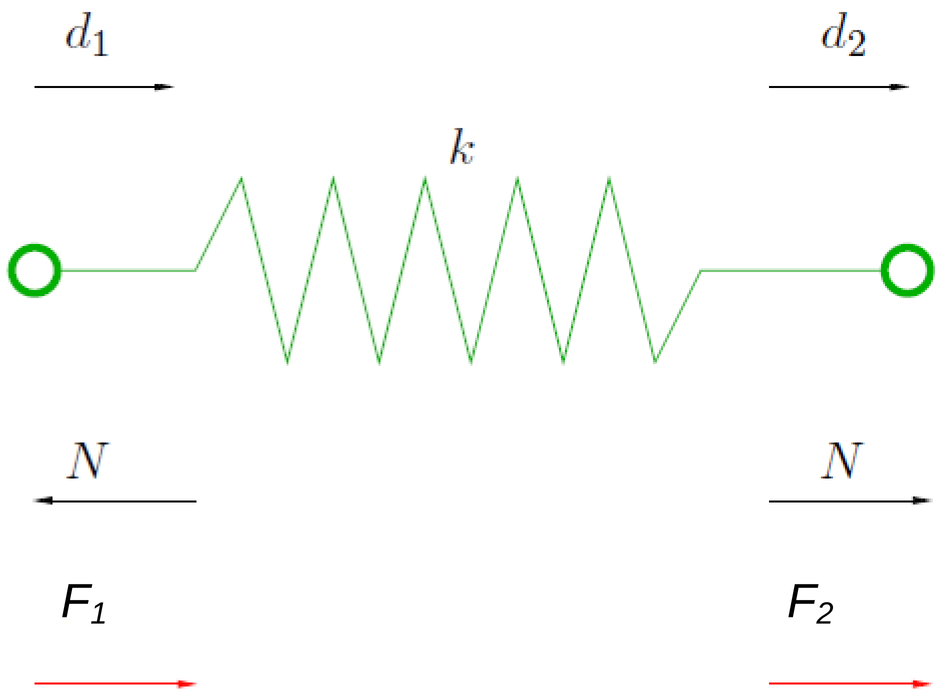


Figure 1.1-1 - Forces and degrees of freedom in a spring

Displacement is denoted *d*, forces are denoted *F*, and the spring stiffness is denoted *k*. From Figure 1.1-1 we find that force equilibrium may be described according to equation **(1)**.

$$F_1 + F_2 = 0 \tag{1}$$

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)**;

$$\begin{aligned} k(d_1 - d_2) &= F_1 \\ k(d_2 - d_1) &= F_2 \end{aligned} \tag{2}$$

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

$$k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} d_1 \\ d_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \tag{3}$$

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

$$\mathbf{kd} = \mathbf{r}\text{, where} \tag{4}$$

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

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

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

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 1.1-1. 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)**;

$$2d_2 = N/k \tag{5}$$

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 1.2-1, 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.

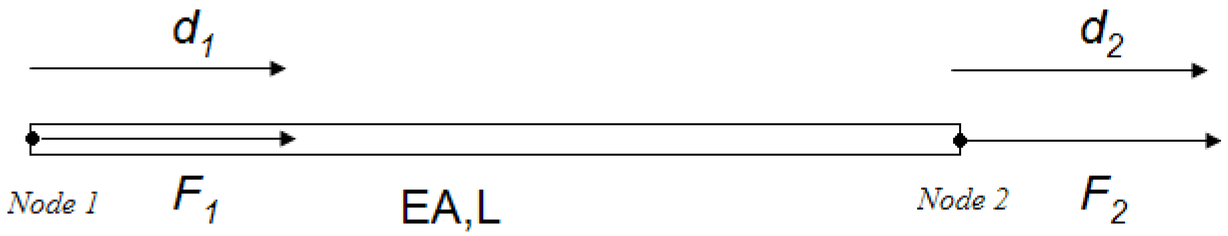


Figure 1.2-1 – A bar element

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.*

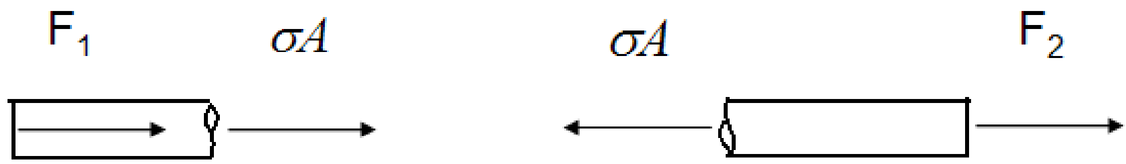


Figure 1.2-2 – Sectional equilibrium for the bar element

With reference to Figure 1.2-1 and Figure 1.2-2 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)**.

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

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

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

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

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

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

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

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

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

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

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.(Given that the stiffness matrix is subject to consistent boundary conditions)*

For bar elements, each node has only one degree of freedom, which makes the above observation normally 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

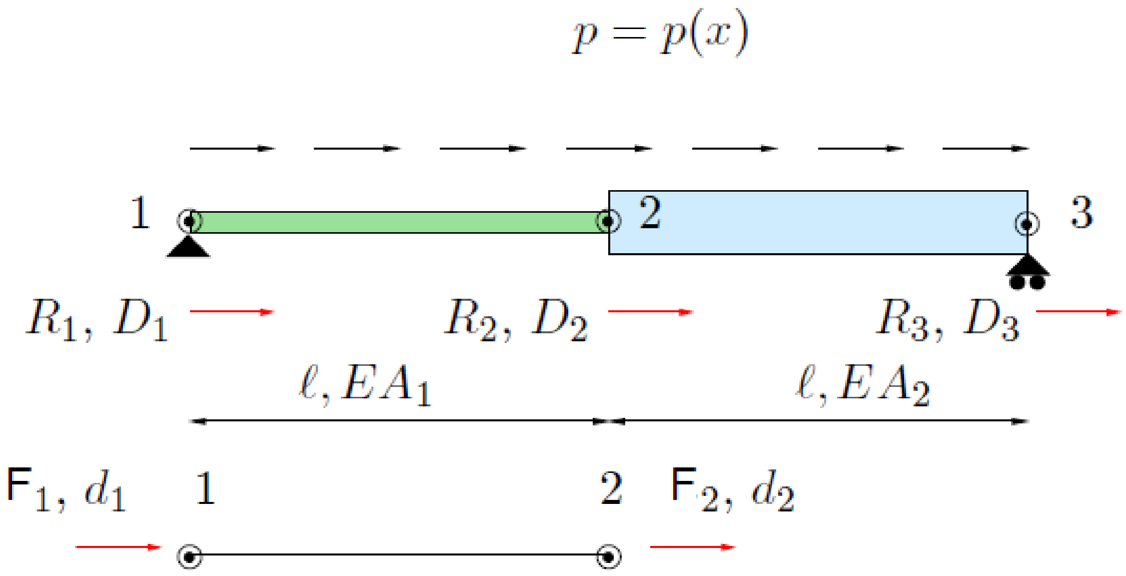


Figure 1.3-1 – Axially loaded, simply supported bar with varying cross-section

In Figure 1.3-1, 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 1.3-1.

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

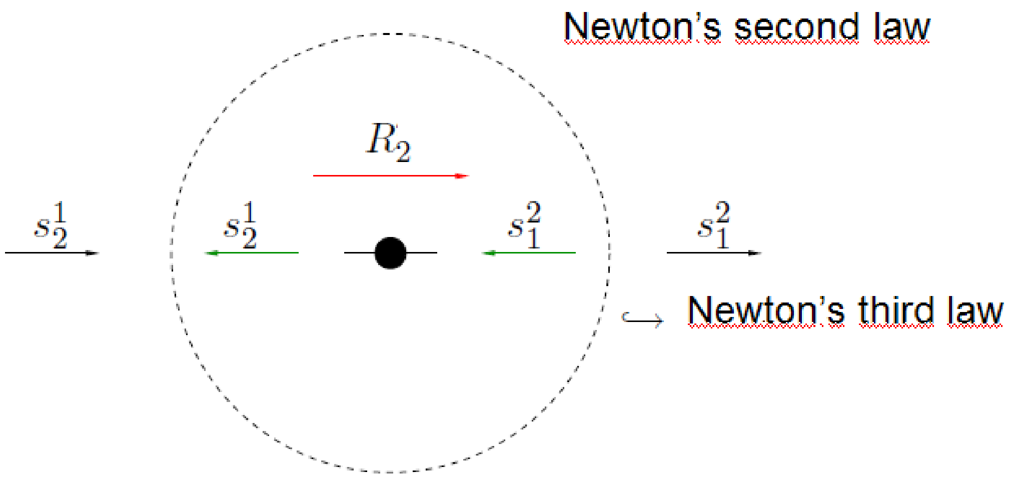


Figure 1.3-2 – Nodal force equilibrium

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)**;

$$R_i = \sum_{e=1}^{m} s_i^e \qquad \textbf{(10)}$$

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;

$$\mathbf{KD} = \mathbf{R} \Rightarrow \frac{E}{l}\begin{bmatrix} A_1 & -A_1 & 0 \\ -A_1 & A_1 + A_2 & -A_2 \\ 0 & -A_2 & A_2 \end{bmatrix}\begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \end{bmatrix}$$

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;

$$\mathbf{D} = \mathbf{K}^{-1}\mathbf{R} \tag{11}$$

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;

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

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;

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

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;

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

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. A thorough explanation of why the stiffness matrix is positive definite will be given chapter 4.

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 1.5-1.

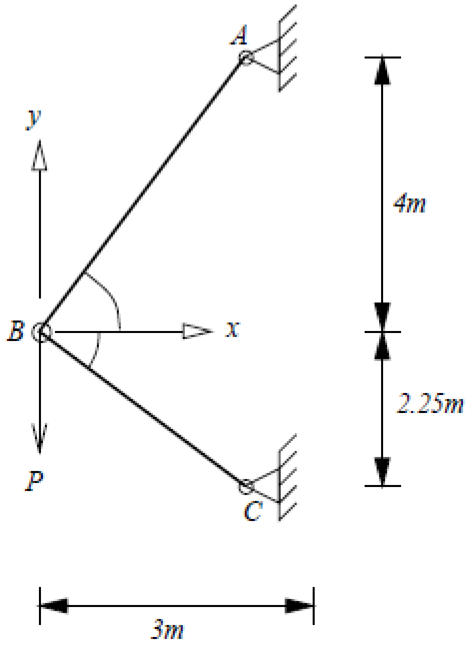


Figure 1.5-1 – Bars rotated relative to the global coordinate system

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;

$$\mathbf{r}_a^T \mathbf{d}_a = \mathbf{r}_b^T \mathbf{d}_b \tag{12}$$

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;

$$\mathbf{r}_a^T \mathbf{d}_a = \mathbf{r}_b^T \mathbf{T} \mathbf{d}_a \tag{13}$$

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;

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

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 1.5-2 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.

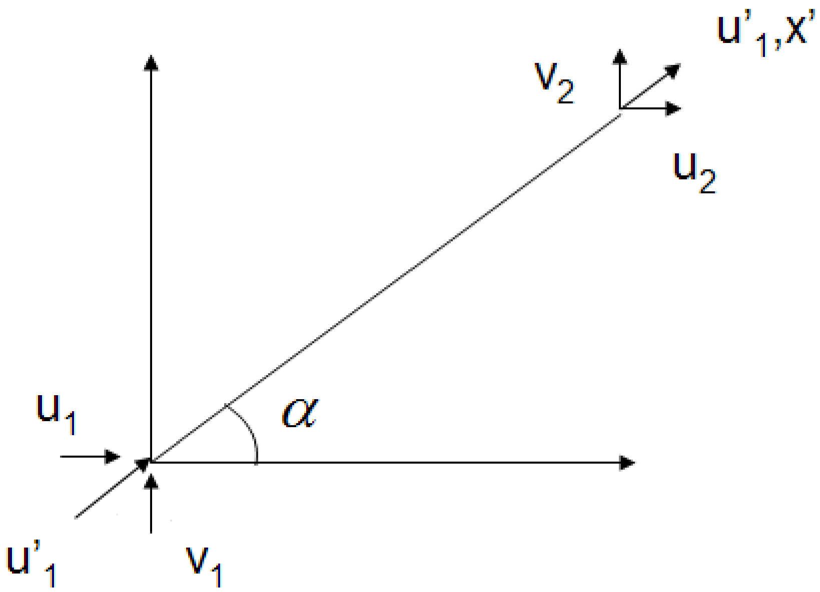


Figure 1.5-2 – Bar element in local and rotated coordinate systems

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 1.6-1. 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

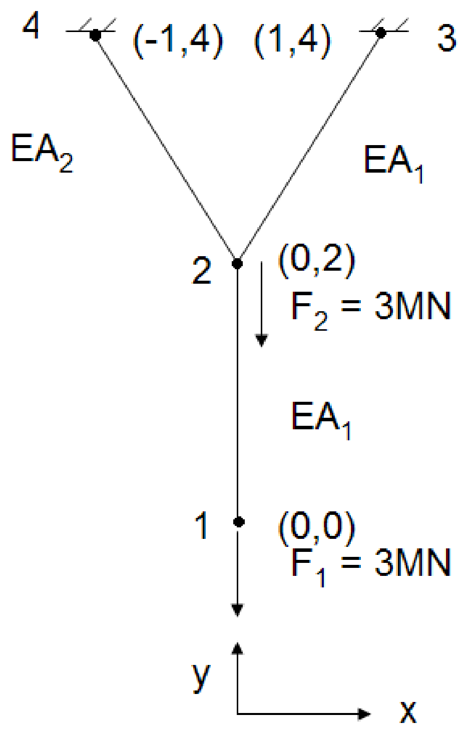


Figure 1.6-1 – Frame of three bars

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 1.5-2), 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

In this 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, explain why the solutions are still consistent.
2. Move node 2 from position (0,2) to position (0, 1), Apply a horizontal load in positive x-direction, F_x, in node 2 equal to 2MN, 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 ! Løsningsprossessoren
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 $\mathbf{kd} = \mathbf{r}$. Specifically, energy methods are the most efficient manner of determining both the stiffness matrix $\mathbf{k}$, and the only general manner of determining a consistent load vector $\mathbf{r}$.

2.1.1 Potential energy in a bar

If we examine the axially loaded, simply supported bar in Figure 2.1-1, 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.

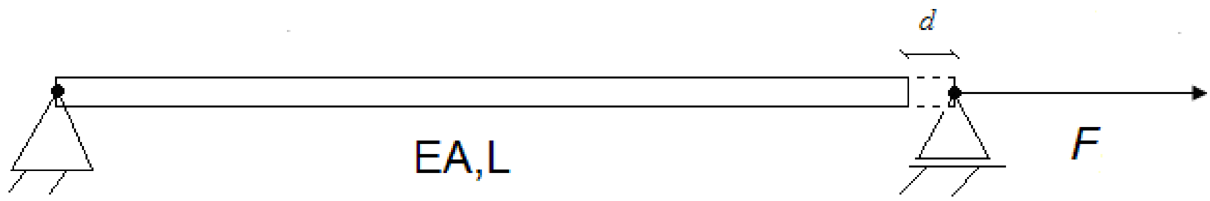


Figure 2.1-1 – Axially loaded, simply supported bar

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 2.1-2)

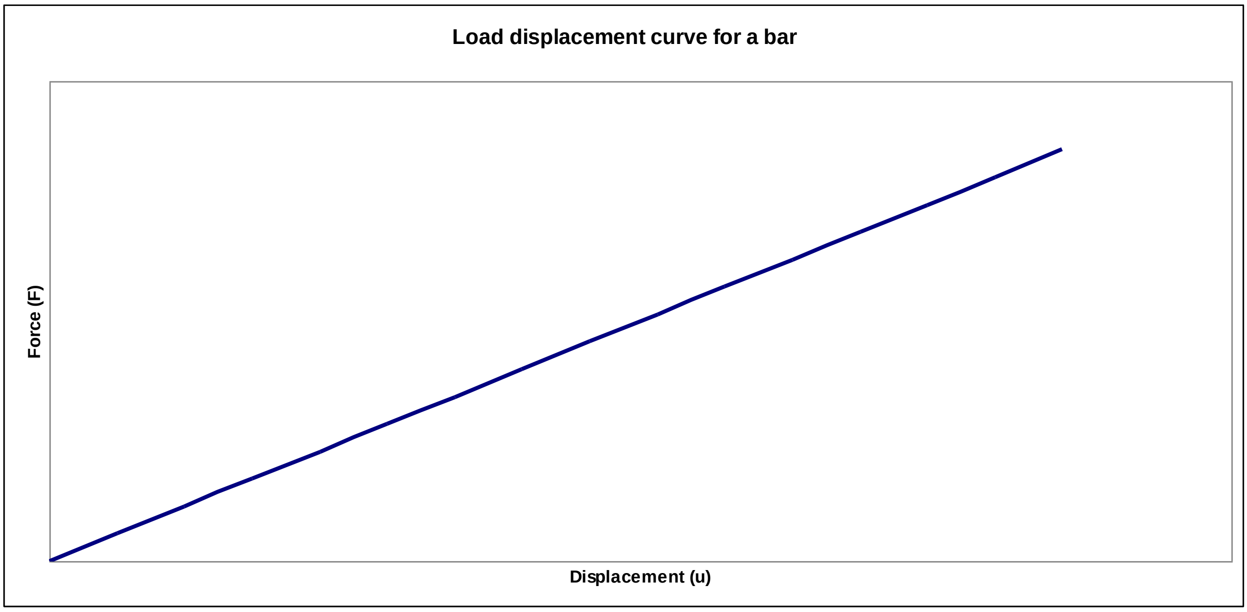


Figure 2.1-2 – Load displacement curve for an axially loaded bar

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 total 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 1.2-1, given again here as Figure 2.1-3 for ease of reference.

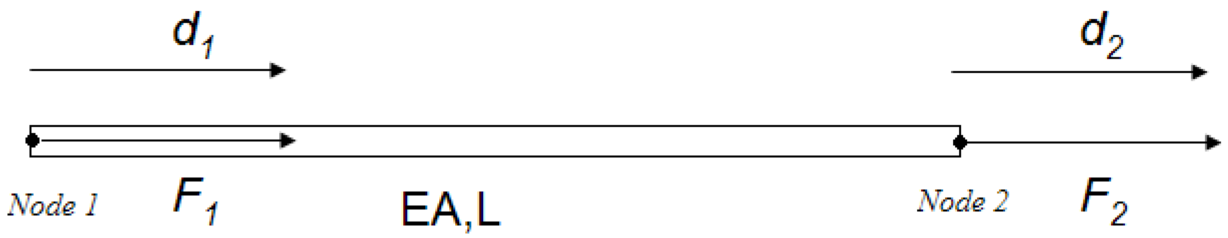


Figure 2.1-3 – A bar element

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 2.1-4.

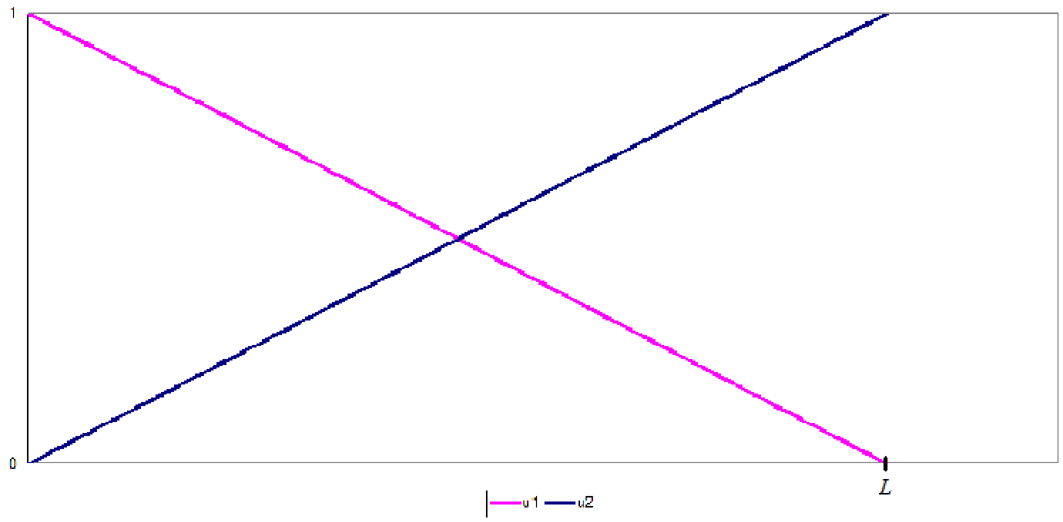


Figure 2.1-4 Shape functions u_1 and u_2

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 long as the material is linearly elastic, and large deformations are not

considered), 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)**;

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

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;

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

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;

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

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 2.1-5

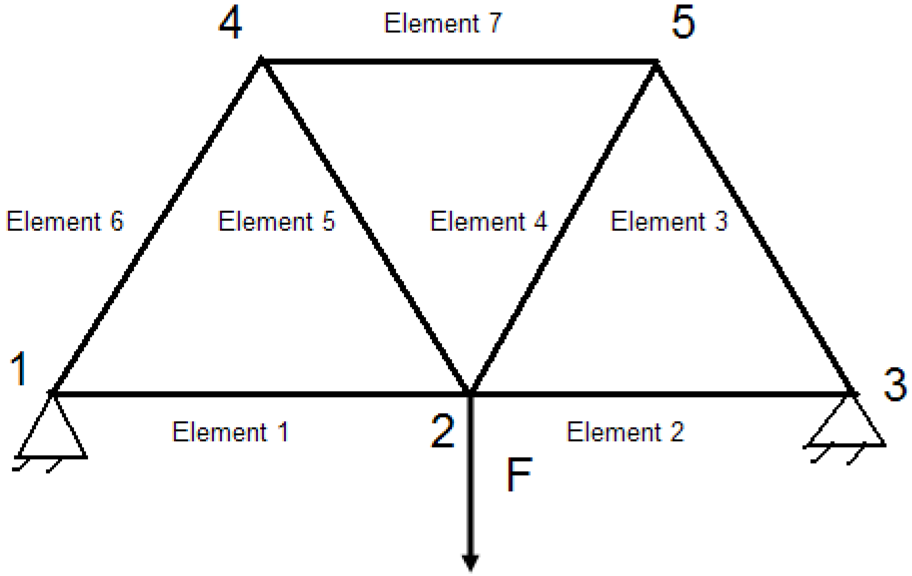


Figure 2.1-5 – A vertically loaded frame of bars

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 2.1-5 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)**;

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

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.

<u>Augmentation of element stiffness matrix for element 1</u>

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;

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

<u>Augmentation of element stiffness matrix for element 2</u>

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

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

<u>Augmentation of element stiffness matrix for element 3</u>

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;

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

Augmentation of element stiffness matrix for element 7

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.

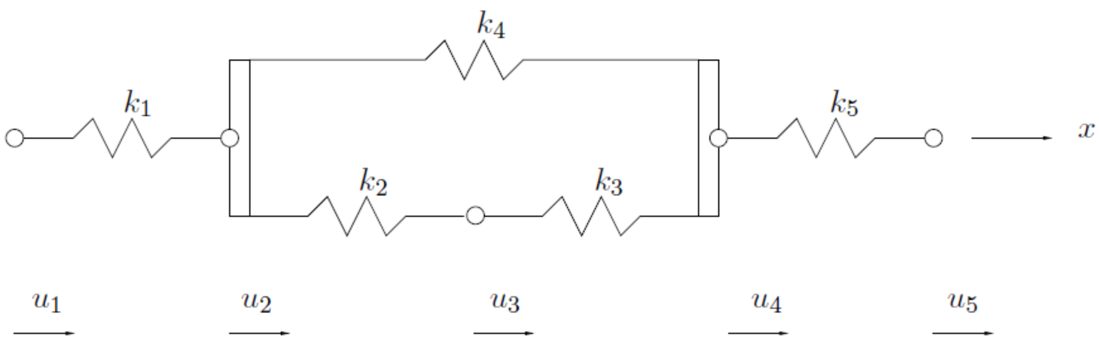
b) Apply cross-sectional area *A*=0.015 and Young modulus E=207 GPa. Substitute the force F with ---. Recalculate the nodal displacements and reaction forces.

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).

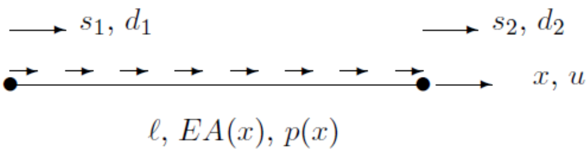
2.2.2

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 BASIC ELASTICITY THEORY

Curriculum – These notes, in addition to Cook et. al. chapters 4.1, 4.2, 4.3, 4.4, 4.7 and 4.9

This chapter concerns itself with the basic conditions of elasticity theory. The equations apply for linear static load and response. Dynamic effects of inertia are thus not considered. The chapter covers the following topics;

- Equilibrium
- Kinematics – displacements and strains
- Material law
- Boundary conditions

The combined considerations of the four above mentioned topics allows for consistent static analyses. We shall also use to types of notation;

- Vector notation, and
- Tensor notation

Vector notation is known from linear algebra. Tensor notation is typical for continuum mechanics theory, but not all students know this notation. The course will therefore only include this notation when absolutely necessary.

3.1 Equilibrium

The starting point for our analyses is the principle of conservation of momentum, which in the static case reduces to a consideration of a balance of forces and moments. If we ignore inertia forces, this is simply Newton's second law. We consider a linearly elastic body *B*, which covers part of the plane $V \in \mathbf{R}^2$. The body is circumscribed by a smooth boundary *S*, with unit normal vector **n**, which points out from the body. The body is loaded by a set of static loads:

- Φ, which is a set of tractions which affect the boundary of the body, and
- **F**, which constitute volume forces which affect the interior of the body. Typical volume forces are gravitational forces

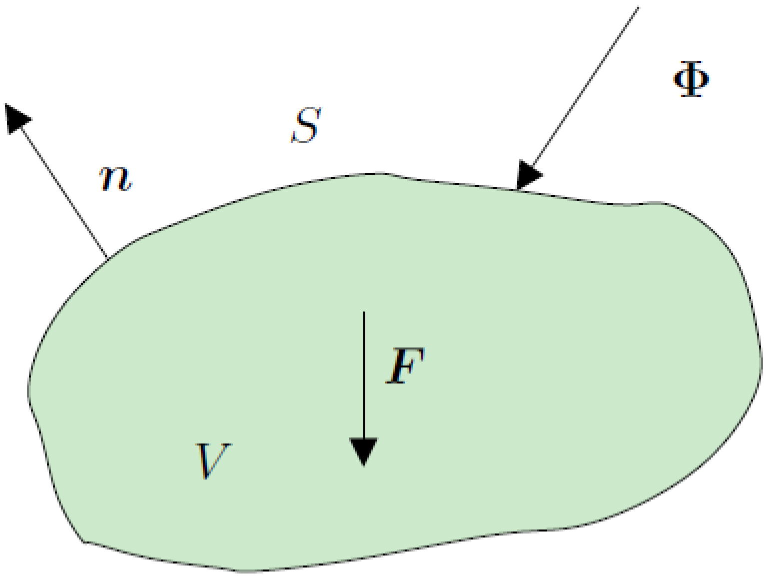


Figure 3.1-1 – The body *B* with a volume *V*, boundary *S* exposed to tractions and forces

Typically we know tractions as distributed loads. A more rigorous definition may be found in equation **(23)**;

$$\Phi = \lim_{\Delta A \to 0} \frac{\Delta f}{\Delta A} \qquad \textbf{(23)}$$

Continuum mechanics holds three main principles;

1. Conservation of mass
2. Conservation of force equilibrium and moment equilibrium
3. Conservation of energy

The sum of forces is expressed according to the following integral;

$$\sum F = 0 \Rightarrow \int_V F dV + \int_S \Phi dS = 0 \qquad \textbf{(24)}$$

Tractions and body forces may be decomposed along unit vectors. In a Cartesian coordinate system for instance, the tractions and body forces may be expressed by their individual components **(25)**.

$$\Phi = \begin{Bmatrix} \phi_x \\ \phi_y \end{Bmatrix}, \mathbf{F} = \begin{Bmatrix} F_x \\ F_y \end{Bmatrix} \qquad \textbf{(25)}$$

In the definition of traction, a surface is involved. Traction is force per unit of surface, (which incidentally is the same definition as stress). The surface is defined by the surface normal **n**. The same type of definition is made for stress, and the term stress tensor is introduced such that we may link the stress to an arbitrary surface, just like the surface tractions.

A tensor is a generalisation of the vector concept. While a vector has two components in 2 dimensions, a tensor may have many. The stress tensor, for instance, has 4 components in 2 dimensions. The 2 dimensional stress tensor may be written on the following form;

$$\sigma = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{bmatrix} \tag{26}$$

The stress tensor consists of four stresses, each of which relate to a surface in the plane, and a direction on that surface. The physical interpretation is listed as follows;

- The stress is classed by surface and direction. The second index gives the unit normal of the surface.
- The first index gives the direction of the stress on that surface

To establish the equilibrium equations on local form, we must find a relation between tractions Φ which depend on the surfaces, and the stresses σ. In order to establish this relation, we will perform equilibrium calculations on a small differential element as shown in Figure 3.1-2.

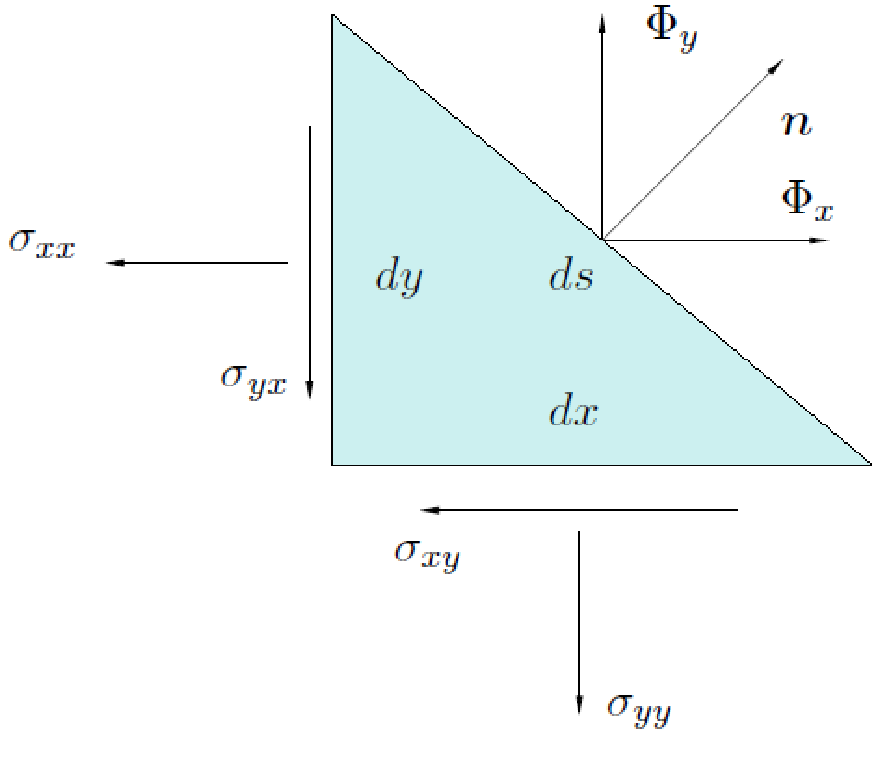


Figure 3.1-2 – Stresses and tractions on a small differential element

Cauchy’s law may be formulated based on equilibrium calculations

$$-\sigma_{xx}dy - \sigma_{xy}dx + \Phi_x ds = 0 \qquad \textbf{(27)}$$
$$-\sigma_{yx}dy - \sigma_{yy}dx + \Phi_y ds = 0$$

To establish the relation in terms of Cartesian coordinates we use the following relations;

$$dy = n_x ds \qquad \textbf{(28)}$$
$$dx = n_y ds$$

If we combine equations **(27)** and **(28)** we find a more convenient expression for equilibrium of the small element;

$$\begin{bmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{bmatrix} \begin{Bmatrix} n_x \\ n_y \end{Bmatrix} = \begin{Bmatrix} \Phi_x \\ \Phi_y \end{Bmatrix} \qquad \textbf{(29)}$$

This expression may be written on total form or by index notation and Einstein's summation rule;

$$\sigma \cdot \mathbf{n} = \Phi \text{ or } \sigma_{ij} n_j = \Phi_i$$

By applying equations **(24)** and **(29)** we find the differential equation for equilibrium of an elastic body through use of the divergence theorem;

$$\int_V \mathbf{F} dV + \int_S \Phi dS = 0 \qquad \textbf{(30)}$$
$$\int_V \mathbf{F} dV + \int_S \sigma \cdot \mathbf{n} dS = 0$$
$$\int_V \mathbf{F} + \nabla \cdot \sigma dV = 0$$

We have developed the differential equation for an arbitrary volume of solid, which in turn means the equation must be valid for any volume. This implies **(31)**

$$\mathbf{F} + \nabla \cdot \sigma = 0 \qquad \textbf{(31)}$$

On component form, for the two-dimensional problem, equation **(31)** may be rewritten **(32)**;

$$\sigma_{xx,x} + \sigma_{xy,y} - F_x = 0 \qquad \textbf{(32)}$$
$$\sigma_{yx,x} + \sigma_{yy,y} - F_y = 0$$

Moment equilibrium may be expressed similarly and leads to the useful conclusion that the stress tensor is symmetric, i.e.;

$$\sigma = \sigma^T$$

Since the stress tensor is symmetric, we do not need to know all the elements in order to uniquely describe its contents. In two dimensions we only need three components, since the fourth component follows from symmetry. Generally the stress tensor is written as a vector in finite element contexts, where only the necessary components to completely describe the stress tensor are included. In two dimensions the following stresses are relevant;

$$\sigma = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}$$

The equilibrium equation in two dimensions may thereby be rewritten as well;

$$\underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}}_{\boldsymbol{\partial}^T \boldsymbol{\sigma}} + \underbrace{\begin{Bmatrix} F_x \\ F_y \end{Bmatrix}}_{+\boldsymbol{F}} = \underbrace{\begin{Bmatrix} 0 \\ 0 \end{Bmatrix}}_{=\boldsymbol{0}}$$

3.2 Kinematics

When a system is exposed to loading it will deform. The relation between displacement and strains is essential to allow us to use equilibrium to balance forces and kinematics (displacements) through a material law. We divide strains into two categories, illustrated in Figure 3.2-1;

- Volumetric strains, and
- Shape changing strains (i.e. shear strains)

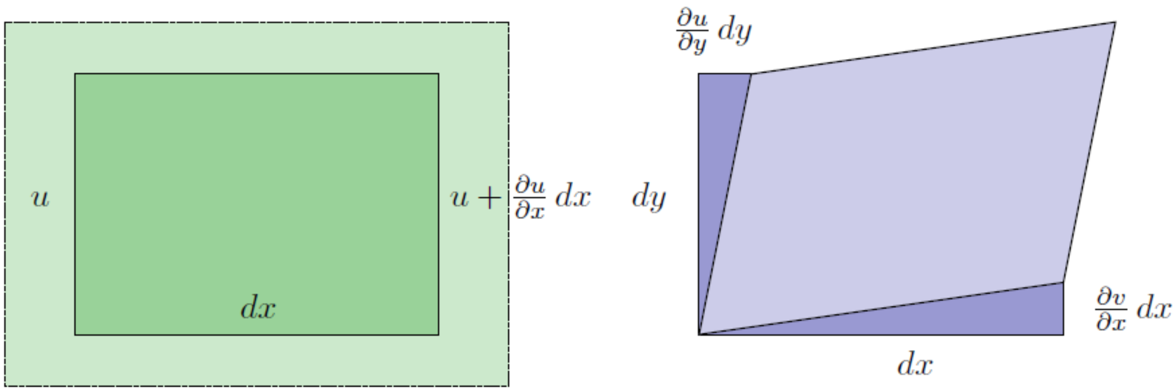


Figure 3.2-1 – Volumetric strain to the left and shear strain to the right

Change in length yields the axial strain;

$$\varepsilon_{xx} = \frac{u + \frac{\partial u}{\partial x} dx - u}{dx} = \frac{\partial u}{\partial x}$$

Change in angle yields the shear strain. Note that we assume small displacements. From vector calculus we can calculate the angle between two vectors;

$$\cos\alpha = \frac{\mathbf{a} \cdot \mathbf{b}}{|\mathbf{a}| \cdot |\mathbf{b}|} = \frac{\frac{\partial v}{\partial x} dxdy + \frac{\partial u}{\partial y} dxdy}{dxdy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

This yields the expression for the shear strains if the following assumptions are made;

$$\gamma_{xy} = \frac{\pi}{2} - \alpha \approx \sin\left(\frac{\pi}{2} - \alpha\right) = \cos\alpha = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

The general expression for the strain tensor is;

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Note that the strain tensor is symmetric. Like the stresses, strains may also be expressed on vector form, and the symmetry of the strain tensor is also taken advantage of;

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

$$\varepsilon = \partial u$$

Note that $\gamma_{xy} = 2\varepsilon_{xy}$. This is the difference between tensorial and engineering strains, and ha no physical significance. It is only a matter of convenience in terms of interpretation and expression of strains.

3.3 Material law

In this course we shall only consider linear elastic materials. If we use the vectorised forms of stress and strain we thus have a linear relation called the generalised Hooke's law;

$$\sigma = \mathbf{E}\varepsilon$$

We may also define an expanded Hooke's law, where initial stresses and strains are considered;

$$\sigma = \mathbf{E}(\varepsilon - \varepsilon_0) + \sigma_0$$

In both cases, **E** is a matrix/tensor.

3.3.1 Example - bar

In one-dimensional analyses, say for bars, there is only one displacement component, which is the axial one. Thus there is only 1 coordinate in the expressions for stresses and strains;

$$\mathbf{u} = u$$

To find the strains, we insert into the definition;

$$\varepsilon_{xx} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) = \frac{1}{2}\left(\frac{du}{dx} + \frac{du}{dx}\right) = \frac{du}{dx}$$

3.3.2 Example – Membrane

A membrane has two displacement components;

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix}$$

The strains are therefore described by two directions, and the stress tensor has 4 components. Since the stress tensor is symmetric, only three of them are independent;

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

$$\varepsilon = \partial u$$

For plane stress we get the generalised Young modulus;

$$\mathbf{E} = \frac{E}{1-v^2} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1-v}{2} \end{bmatrix}$$

For plane strain we get a different Young modulus;

$$\mathbf{E} = \frac{E}{(1-v)(1-2v)} \begin{bmatrix} 1-v & v & 0 \\ v & 1-v & 0 \\ 0 & 0 & \frac{1-2v}{2} \end{bmatrix}$$

3.4 Graphic representation

In Figure 3.4-1, the relation between displacements, forces, stresses, tractions and strains are shown.

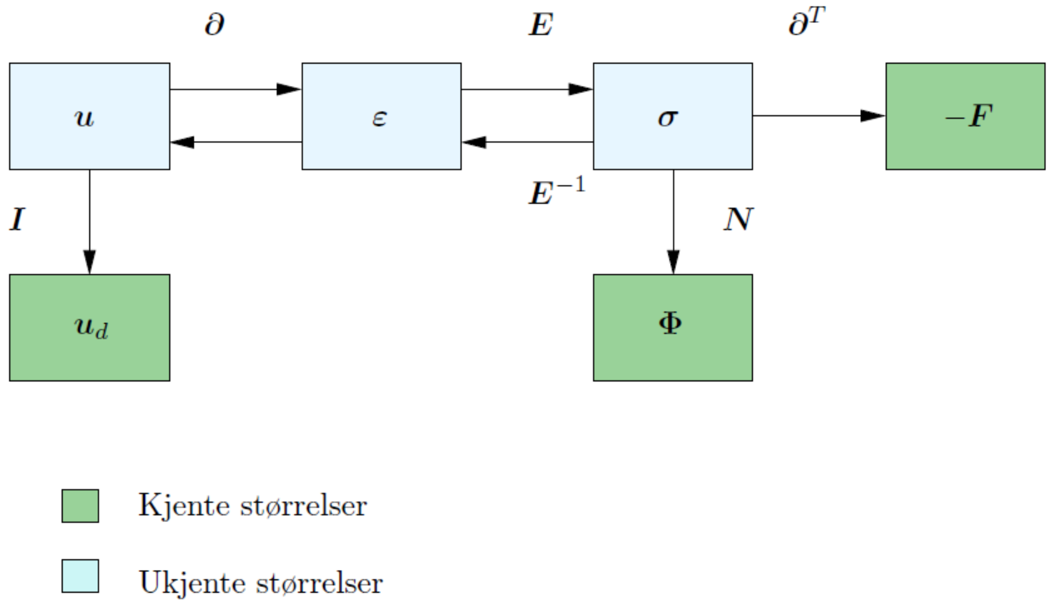


Figure 3.4-1 – Relation between known and unknown entities in a finite element solution

3.5 Weak form

Previously, in this chapter, we investigated the basic equations for a linearly elastic material under static loading and response. All equations were on strong form (explicit differential equations). In their strong form the equations apply for all interior points in the domain of the unknown functions. The equations we have summarized are;

1. Strain-displacement relations
2. Stress-strain relations
3. Equilibrium on the interior of an elastic body
4. Equilibrium on the boundary of an elastic body
5. Continuity on the boundary

In the element method we shall not apply all these equations directly, since this is inconvenient computationally. We transfer these equations into a weak form.

We shall derive the governing equations from a different standpoint, which we introduced in Chapter 2, using the principle of minimum potential energy. Then the equilibrium equations are a consequence of a stationary value of a functional which expresses the energy in an elastic system. This is a variational formulation. The variational formulation is also an integral formulation, where, if we choose, we can require that the governing equations hold true over an integral of a selection of the domain of the equations. In that case, the solution is accurate as an average value over a finite element, rather than over the entire domain as they would be on a strong form.

Further details on variational principles may be found in Cook et. al, Chapters 4.1, 4.2, 4.3, 4.4, 4.7 and 4.9.

When structuring a variational principle, we must follow certain rules;

1. Choice of primary variable
2. Choice of weak and strong formulations
3. Choice of variational principle/rule

3.5.1 Choice of primary variable

In the finite element, the displacements are the primary variable, and stresses and strains are immediately following from the displacements. Thus all solutions we shall be solving for are displacements, from which stresses, strains, reaction forces etc. will be deduced afterwards.

3.5.2 Weak and strong formulations

We want to establish the equilibrium equations on weak form. The weak equations are thus alternative formulations for;

1. Equilibrium inside the body
2. Equilibrium on the boundary

The strong formulations are all the equations that remain;

- Strain- displacement relations

- Stress-strain relations
- Continuity on the boundary

The main difference between the weak and the strong formulations are;

- Strong formulations are satisfied pointwise on the domain of the functions
- Weak formulations are satisfied as integrals, or mean values, piece-wise over the domain

3.5.3 Variational principle

Basically we have two standard choices (among others, but these are dominating);

- The principle of virtual displacements
- The principle of minimum potential energy

Virtual displacements are often intuitive, and students often have a basic understanding of the principle. However, the principle of minimum potential energy is governing in the field of the finite element method, as the most literature, be it instruction manuals or textbooks, are formulated based on the PMPE. Thus this course shall apply the principle of minimum potential energy even though the principle of virtual displacements is more intuitive for some, and equally applicable.

3.6 The principle of minimum potential energy

The principle of minimum potential energy may be discussed in further detail than we did in Chapter 2. Now we introduce the strain energy density **(33)**;

$$u = \frac{1}{2}\sigma^T \varepsilon \qquad \textbf{(33)}$$

The strain energy density is the product of stress and strain for any material point in the continuum. The concept of strain energy density will be illustrated with the examples of volumetric strain, and shape changing strain, i.e. shear strain.

Strain energy density is the strain energy stored in an elastic body per unit of volume. Thus when we integrate the strain energy density over the volume, as will be done later in this Section, the total strain energy in the body is the result. Thereby:

u = energy/volume

We start by showing that equation **(33)** applies for volumetric strains. In Figure 3.6-1, a cubic volume of an elastic material exposed to an axial force F_x is shown, where the sides of the volume have length a, and the resulting displacement is δ.

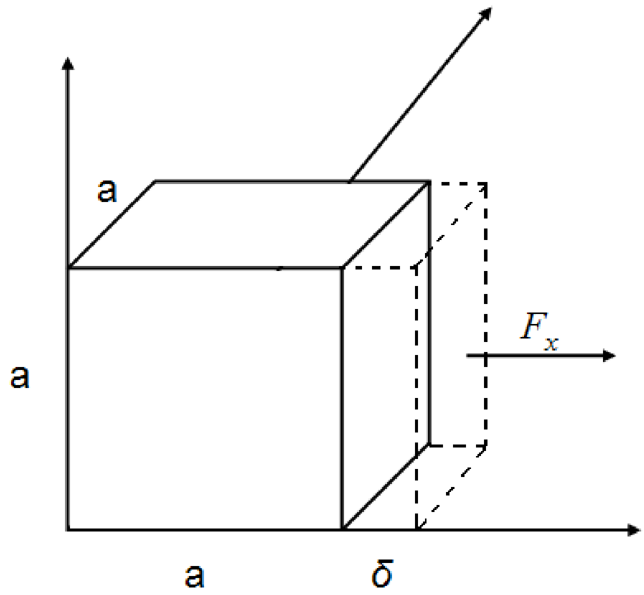


Figure 3.6-1 – Volume of elastic material subject to a force F_x

The stress on the surface where F_x is applied is trivially:

$$\sigma_{xx} = \frac{F_x}{a^2} \Rightarrow F_x = \sigma_{xx} a^2$$

The strain in the volume is found from the displacement δ:

$$\varepsilon_{xx} = \frac{\delta}{a} \Rightarrow \delta = \varepsilon_{xx} a$$

The elastic strain energy U stored in the volume follows from follows from Figure 2.1-2, and is simply the area under the load displacement curve. This is expressed as follows:

$$U = \frac{1}{2} F_x \delta$$

If we insert for the stress and strain relations, we find the stored elastic strain energy as a function of stresses and strains:

$$U = \frac{1}{2} a^3 \sigma_{xx} \varepsilon_{xx}$$

The strain energy density is defined as the energy per unit of volume, so the strain energy density may therefore be found:

$$u = \frac{U}{V} = \frac{1}{2} \frac{a^3 \sigma_{xx} \varepsilon_{xx}}{a^3} = \frac{1}{2} \sigma_{xx} \varepsilon_{xx}$$

The strain energy density thus conforms to equation **(33)** for volumetric strains. The same argument also applies to strain energy density for shear strains. In Figure 3.6-2 a cubic volume of elastic material is subject to shear stresses τ_{xy}.

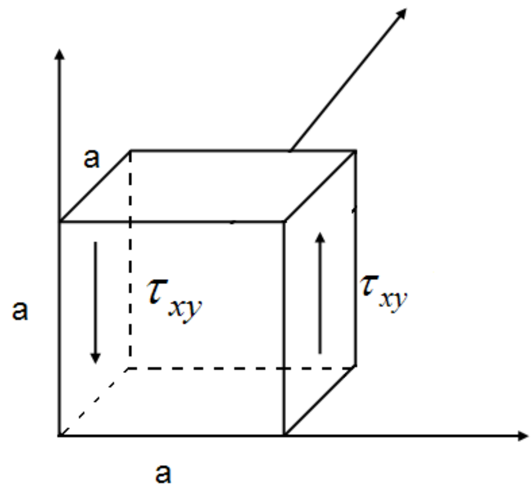


Figure 3.6-2 – Cubic volume of elastic material subject to a shear stress τ_{xy} in un-deformed configuration

The deformed configuration for the volume in Figure 3.6-2 is shown in Figure 3.6-3:

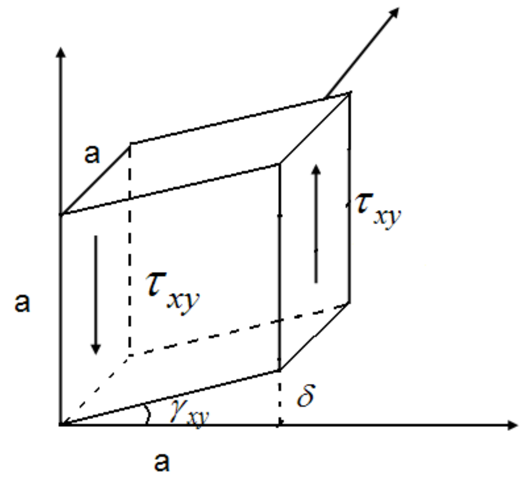


Figure 3.6-3 – Cubic volume of elastic material subject to shear stress in deformed configuration

Based on Figure 3.6-3, and the assumption of small displacements, the displacement may be calculated as follows:

$$\delta = a\gamma_{xy}$$

The force acting on the surface of the shear stress is simply the stress multiplied with the area on which it acts:

$$F_{xy} = \tau_{xy}a^2$$

The elastic energy may be calculated:

$$U = \frac{1}{2}F_{xy}\delta = \frac{1}{2}a^3\gamma_{xy}\tau_{xy}$$

Finding the strain energy density only leaves dividing the total elastic energy with the volume of the cube:

$$u = \frac{U}{V} = \frac{1}{2}\tau_{xy}\gamma_{xy}$$

The strain energy density thus conforms to equation **(33)** for shear strains also.

If we integrate the strain energy density over the volume of the body, we necessarily achieve the strain energy elastically stored in the body. The total strain energy is defined in Chapter 2 and termed *U*;

$$U = \int_V \frac{1}{2}\sigma^T \varepsilon dV \left(= \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV \right) \tag{34}$$

If we also include initial stresses and strains to the equation we may now reformulate the total potential energy functional;

$$\Pi = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon + \varepsilon^T (\sigma_0 - \mathbf{E}\varepsilon_0) dV - \int_V \mathbf{u}^T \cdot \mathbf{F} dV - \int_S \mathbf{u}^T \cdot \Phi dS \tag{35}$$

Equilibrium equations may now be found by determining the stationary value of this functional **(35)**.

The process of determining the stationary value of the energy functional may be understood in the following manner; If we have an elastic system we may se what happens to the potential energy close to an equilibrium position. This is illustrated in Figure 3.6-4.

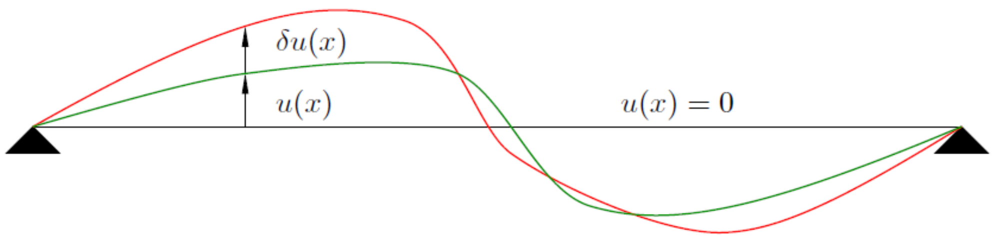


Figure 3.6-4 – Variation about an equilibrium position

Thus, if you wish to find how heavy something is, try to lift it. We se what happens to the potential energy when we change the displacement field in a small domain close to the equilibrium position.

The need to solve differential equations may be avoided by applying the Rayleigh Ritz method on a functional such as Π. With this method, we establish an approximation to the unknown functions which in the potential energy functional are displacements, **u**. The result is a problem with a finite number of unknowns, and is described by algebraic equations rather than differential equations. The element method is a special form of the Rayleigh-Ritz method, which we shall discuss in detail in the next chapter.

Much time in MEK4550 shall be devoted to determine expressions for *U*, since the strain energy functional is the basis for development of stiffness matrices. The strong and weak formulations are illustrated in Figure 3.6-5

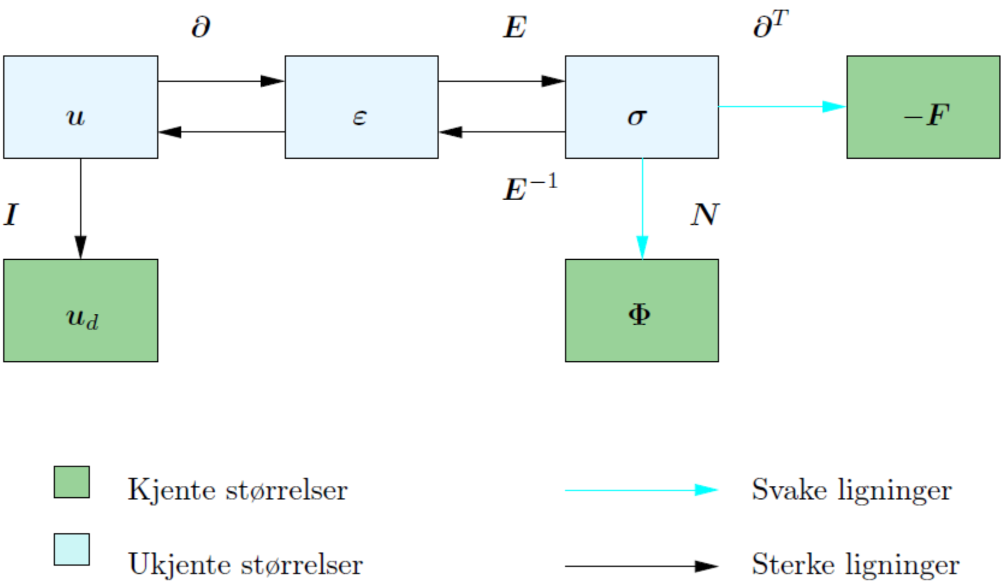


Figure 3.6-5 - Weak and strong equations

3.7 Examples

In this section we shall employ the principle of minimum potential energy according to the Rayleigh method. The Rayleigh method is based on assuming that the displacements of a system may be described by a single function. We shall make three examples of this. But first we shall use the principle of minimum potential energy to deduce the differential equation for a beam subject to distributed loading.

3.7.1 The differential equation for a beam under distributed loading

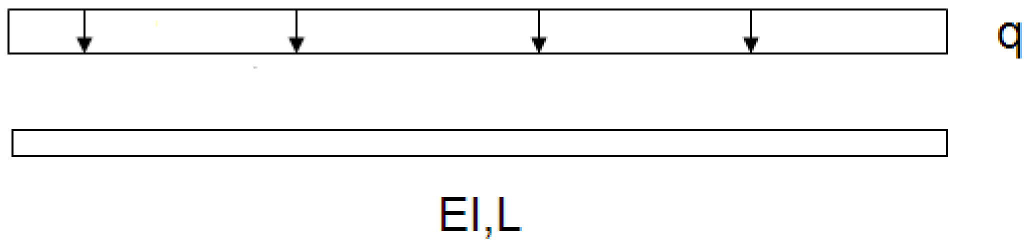


Figure 3.7-1 – A finite length beam with uniformly distributed load and uniform cross-section, made from an isotropic material

When we have a beam of uniform cross-section with a uniformly distributed load, if the displacements are small and shear is negligible we may assume Euler-Bernoulli beam theory. With linear theory and small displacements, the displacement field across a beam may be described according to the following equations;

$$u = -z\frac{dw}{dx}$$
$$w = w(x)$$

For beams we only consider axial strains, thus;

$$\varepsilon_{xx} = \frac{du}{dx} = -z\frac{d^2w}{dx^2}$$

We insert the strains into the total potential energy functional;

$$\Pi = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV - \int_V \mathbf{u}^T \cdot \mathbf{F} dV - \int_S \mathbf{u}^T \cdot \Phi dS$$
$$U = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV = \int_V \varepsilon_{xx}^2 E dV = \int_V E z^2 \left(\frac{d^2w}{dx^2}\right)^2 dV$$

When we integrate over the volume, we assume that the beam axis goes is positioned in the centre of area for the beam cross-section, thus the function w is not dependent on the y and z coordinates. Therefore we may isolate the integration;

$$\int_A z^2 dA = I$$

This allows us to simplify the expression for the internal strain energy;

$$U = \frac{1}{2}\int_V Ez^2\left(\frac{d^2w}{dx^2}\right)^2 dV = \frac{1}{2}\int_L EI\left(\frac{d^2w}{dx^2}\right)^2 dx$$

We find the variation of the energy functional, but we shall vary the internal strain energy separately for simplicity.

$$\delta U = \frac{1}{2}\int_V \delta\left(Ez^2\left(\frac{d^2w}{dx^2}\right)^2\right)dV = \int_L EI\left(\frac{d^2w}{dx^2}\right)\delta w_{,xx}dx$$

When the variation includes derivatives of the displacements, we use integration by parts to transform the varied entities to variations in the functions themselves and not their derivatives. We perform integration by parts twice on the internal strain energy;

$$\int_L EI\left(\frac{d^2w}{dx^2}\right)\delta w_{,xx}dx = \left(\frac{d^2w}{dx^2}\right)\delta w_{,x}\bigg|_0^L - \int_L EI\left(\frac{d^3w}{dx^3}\right)\delta w_{,x}dx$$

$$= \left(\frac{d^2w}{dx^2}\right)\delta w_{,x}\bigg|_0^L - \left(\frac{d^3w}{dx^3}\right)\delta w\bigg|_0^L + \int_L EI\left(\frac{d^4w}{dx^4}\right)\delta w dx$$

Variation of a functional at a fixed point is equivalent to a constant in terms of differentiation, i.e. there is no potential for change in a function that has a fixed evaluation point.

$$\delta U = \int_L EI\left(\frac{d^4w}{dx^4}\right)\delta w dx$$

We complete the expression for the total potential energy functional, and use the principle of minimum potential energy;

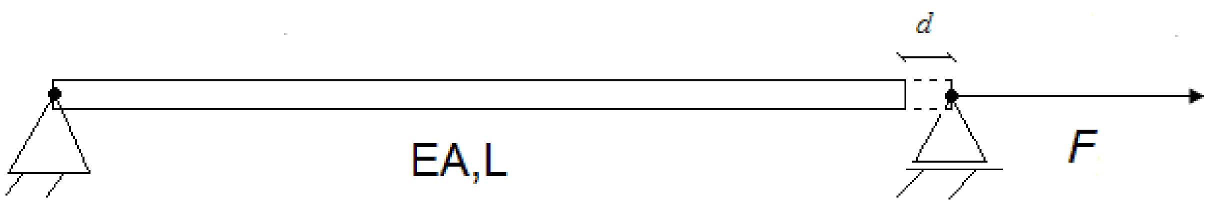
$$\Pi = \frac{1}{2}\int_L EI\left(\frac{d^2w}{dx^2}\right)^2 dx - \int_L qwdx$$

$$\delta\Pi = \int_L EI\left(\frac{d^4w}{dx^4}\right)\delta w dx - \int_L q\delta w dx = 0$$

$$\Rightarrow \int_L\left(EI\frac{d^4w}{dx^4} - q\right)\delta w dx = 0 \Rightarrow \frac{d^4w}{dx^4} = \frac{q}{EI}$$

We recognize the differential equation for a transversely loaded beam.

3.7.2 Displacement of a simply supported bar



We assume that the bar starts at the origin and that the *x*-coordinate is parallel to the bar axis. We further assume that the lateral displacement may be expressed according to the following function;

$$u(x) = d\frac{x}{L}$$

The axial strain is found by differentiating the displacement with regard to *x*. $\varepsilon_{xx} = \frac{du}{dx} = \frac{d}{L}$ We observe that the displacement is zero at the origin, hence the boundary conditions for the system are fulfilled by the assumed displacement function. We insert the assumed displacement function into the expression for the total potential energy;

$$\Pi = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV - \int_S \mathbf{u}^T \Phi dS = \frac{1}{2}\int_V E\left(\frac{d}{L}\right)^2 dV - dF$$

$$\Pi = \frac{1}{2}\int_L EA\left(\frac{d}{L}\right)^2 dx - dF$$

$$\delta\Pi = \frac{1}{2}\int_L 2\delta d\left(EA\frac{d}{L^2}\right)dx - F\delta d = \frac{EA}{L}d - F = 0$$

$$d = \frac{FL}{EA}$$

We observe that the assumed displacement function yields the exact result. This is simply because the chosen displacement function is the exact solution.

3.7.3 Transverse deflection of a cantilever beam

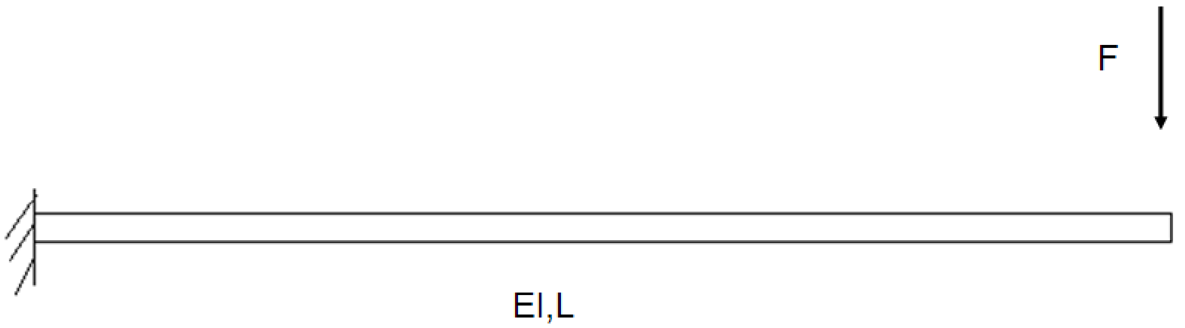


Figure 3.7-2 – Cantilever beam with end load

The boundary conditions for a cantilever beam are that both the displacement and rotation at the origin is zero. A simple function which fulfils both of the boundary conditions is x^2. If we assume that the displacement function $w(x) = c\frac{x^2}{L^2}$, we find that the total potential energy may be expressed as follows;

$$\Pi = \int_L M\kappa dx - cF = \int_L EI\left(\frac{d^2w}{dx^2}\right)^2 dx - cF = \frac{1}{2}\int_L 4EIc^2 \frac{1}{L^4} dx - cF = \frac{2EIc^2}{L^3} - cF$$

$$\delta\Pi = \delta c\left(\frac{4EIc}{L^3} - F\right) = 0 \Rightarrow c = \frac{FL^3}{4EI}$$

We find that the end deflection is fairly well approximated since the exact solution is $\frac{FL^3}{3EI}$, thus the displacement is underestimated by 25%. We shall understand in following chapters that the Rayleigh method (and the Rayleigh-Ritz method) underestimates displacements in load controlled conditions in general. One important observation is that the displacements are estimated with fairly good accuracy. The moments on the other hand are not. When the displacements are second order functions, the moments are constant functions according to the differential equation for a beam. Thus the moments are;

$$M = \frac{FL}{2}$$

The exact solution for the moment distribution is *F(x-L)*, which is only a match to our solution in the middle of the beam. The important lesson is that the displacements are more accurately predicted than the moments.

3.7.4 Displacement of a simply supported beam

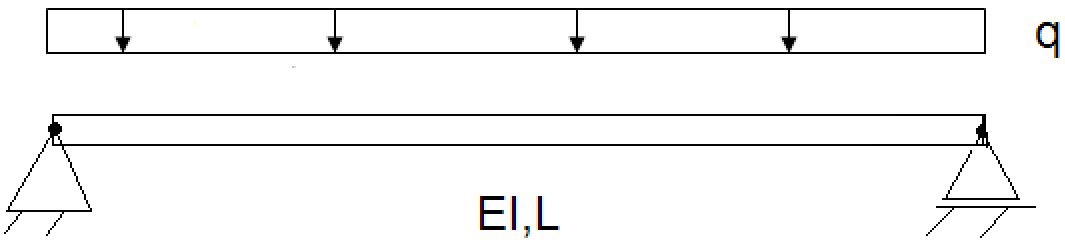


Figure 3.7-3 – Simply supported beam with distributed load

The boundary conditions for the simply supported beam is zero displacements at the origin and at position $x = L$. A function which fulfils the boundary conditions is $w(x) = c\sin\left(\frac{\pi x}{L}\right)$. If we insert this displacement assumption into the total potential energy functional, we achieve the following result;

$$\Pi = \int_L M\kappa dx - \int_L qwdx = \int_L EI\left(\frac{d^2w}{dx^2}\right)^2 dx - \int_L qwdx$$

$$= \frac{1}{2}\int_L c^2 EI\frac{\pi^4}{L^4}\sin^2\left(\frac{\pi x}{L}\right)dx - \int_L qc\sin\left(\frac{\pi x}{L}\right)dx = c^2 EI\frac{\pi^4}{4L^3} - c\frac{2qL}{\pi}$$

$$\delta\Pi = \delta c\left(EI\frac{\pi^4}{2L^3} - \frac{2qL}{\pi}\right) = 0 \Rightarrow c = \frac{4qL^4}{EI\pi^5}$$

Compared to the exact solution, the mid displacement deviates by less than 1%. The exact solution for the mid displacement is $\frac{5qL^4}{384EI}$.

3.8 Exercises

Øving 4.1

Spenningene i et plant problem kan tilnærmes ved:

$$\sigma_{xx} = a_0 + a_1 x + a_2 y \qquad \sigma_{yy} = b_0 + b_1 x + b_2 y \qquad \sigma_{xy} = c_0 + c_1 x + c_2 y$$

Hva må sammenhengen mellom a_i, b_i og c_i være for at likevekt skal være tilfredsstilt når vi ikke har volumkrefter?

Øving 4.2

Transformasjon av *spenninger* og *tøyninger* følger reglene for transformasjon av tensorer (andre ordens tensorer)

$$\bar{\varepsilon} = \boldsymbol{T}\varepsilon\boldsymbol{T}^T \qquad \text{og} \qquad \bar{\boldsymbol{\sigma}} = \boldsymbol{T}\boldsymbol{\sigma}\boldsymbol{T}^T$$

Dersom en vektor transformerer som

$$\bar{\boldsymbol{u}} = \begin{Bmatrix} \bar{u}_x \\ \bar{u}_y \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta \\ -\sin\theta & \cos\theta \end{bmatrix} \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} = \boldsymbol{T}\boldsymbol{u}$$

hvordan transformerer da spenningene uttrykt på matrise/vektor form

$$\bar{\boldsymbol{\sigma}} = \boldsymbol{T}_2\boldsymbol{\sigma} \qquad \text{hvor} \qquad \boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}$$

Finn $\boldsymbol{T}_2$.

Øving 4.3

I vanlig plan bjelketeori, hvor vi ikke tar med skjærdeformasjoner på tverrs, kan alle deformasjoner og tøyninger uttrykkes ved forskyvningen på tverrs av bjelkeaksen, $u_z = w$. Rotasjonen antas å være liten, plane tverrsnitt forblir plane og normalt på den deformerte bjelkeaksen.

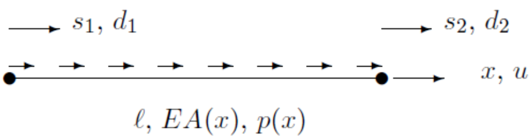
Aksialforskyvningene kan derfor skrives som:

$$u_x = u = -z\theta = -z\frac{dw}{dx}$$

Finn uttrykket for tøyningen i bjelkeproblemet. Identifiser tøyningsoperatoren, $\boldsymbol{\partial}$.

Øving 4.4

Figuren under viser stavproblemet fra Øving 1.1.



a) Sett opputtrykket for *tøyningenergitetthetsfunksjonen*, $\mathcal{U}(u)$, for stavproblemet.

b) Benytt $\mathcal{U}(u)$ fra a) og sett opp uttykket for potensiell energi, $\Pi(u)$, for stavproblemet.

c) Finn tilbake til uttykket for differensialligningen (og naturlige randkrav) for stavelementet ved å finne stasjonærverdien (første deriverte er lik null) for potensiell energi:

   i) Finn først:
   $$\delta\Pi = \frac{\partial\Pi}{\partial u}\delta u + \frac{\partial\Pi}{\partial u_{,x}}\delta u_{,x}$$

   hvor $u_{,x}$ benevner u derivert med hensyn på x.

   ii) Benytt delvis integrasjon på uttrykket som inneholder $\delta u_{,x}$ for å etablere et uttrykk kun inneholdene δu:
   $$\delta\Pi = \frac{\partial\Pi}{\partial u}\delta u$$

   iii) Benytt at $\delta u = 0$ på S_u og etabler ligningene for indre likevekt og likevekt på randen.

d) Anta at u er gitt av (sett $\ell = 1$)
$$u = (1 - \frac{x}{\ell})u_1 + \frac{x}{\ell}u_2$$
Benytt dette til å etablere et tilnærmet uttrykk for *tøyningenergien*, $U(u)$, for stavproblemet.
$$U(u) = \int_\ell \mathcal{U}(u)\,dx$$

e) Sett utrykket fra d) ved å benytte matrise-vektor operasjoner
$$U(u) = \frac{1}{2}\boldsymbol{d}^T\boldsymbol{k}\boldsymbol{d}$$

hvor $\boldsymbol{d}^T = \{u_1, u_2\}$.

Sammenlign resultatene for $\boldsymbol{k}$ med
$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$
fra Øving 1.1.

4 THE RAYLEIGH-RITZ METHOD AND BEAM ELEMENTS

From the previous three chapters in this compendium, we have become acquainted with bars, the stiffness matrix for a bar and we have studied variational principles, specifically the principle of minimum potential energy. In this chapter we shall adopt the principle of minimum potential energy on bars and beams, and we shall introduce new notation and new means of formulating displacement functions in order to find stiffness matrices efficiently.

4.1 Generalised methodology to establish the stiffness matrix

In section 2.1.3 we found the stiffness matrix for a bar element using two assumed displacement functions. This led to fairly lengthy calculations involving the nodal displacements d_1 and d_2, as well as their respective differentials. For more complex elements, the calculations are even more lengthy, and in the end so lengthy that they are inconvenient. To establish a methodology for generation of stiffness matrices for general elements, a standard notation which generalises the variational operation is necessary. This methodology and the associated notation will be introduced in this section.

We start with the equation for the total potential energy of a system without initial stresses or strains;

$$\Pi = U - \Omega = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV - \int_V \mathbf{u}^T \mathbf{F} dV - \int_S \mathbf{u}^T \Phi dS$$

We know from section 3.2 that the relation between displacements and strains could be expressed according to the following formula;

$$\varepsilon = \partial \mathbf{u}$$

The finite element method is based on assuming a displacement field **u**, after which undetermined coefficients in the displacement field are established using the principle of minimum potential energy. In section 3.7 we introduced the Rayleigh method, where we assumed a single displacement function as a solution to various systems. In this chapter we generalise this methodology to include more than one displacement function and the new/augmented method is called the Rayleigh-Ritz method. For the purposes of the finite element method in structural mechanics, the displacement field is always based on polynomials. Generally these are polynomials in three dimensions, i.e. dependent on *x*, *y* and *z*, however we shall consider only polynomials in one dimension in this chapter, i.e. polynomials in *x*. In later chapters, we will introduce additional degrees of freedom to our polynomials. When the displacement field is a polynomial, we may express it generally;

$$\mathbf{u} = c_0 + c_1 x + c_2 x^2 + \ldots + c_n x^n$$

In FE problems, we reorganise the general displacement field to associate individual polynomials with nodal degrees of freedom;

$$\mathbf{u} = d_1 \cdot P_1(1, x, x^2..) + d_2 \cdot P_2(1, x, x^2..) + .. + d_n \cdot P_n(1, x, x^2..)$$

If we express **u** as a vector equation, we may place the individual polynomials in one vector and the nodal degrees of freedom in another;

$$\mathbf{u} = \begin{bmatrix} P_1 & P_2 & \dots & P_n \end{bmatrix} \begin{bmatrix} d_1 \\ d_2 \\ \cdot \\ d_n \end{bmatrix}$$

We name the vector of polynomials **N**, and the polynomials express shape functions. The nodal displacement vector is named **d**. Thus;

$$\mathbf{u} = \mathbf{N}\mathbf{d},$$
$$\mathbf{N} = \begin{bmatrix} P_1 & P_2 & \dots & P_n \end{bmatrix},$$
$$\mathbf{d} = \begin{bmatrix} d_1 \\ d_2 \\ \cdot \\ d_n \end{bmatrix}$$

It is not readily explained how a general assumption that the displacements may be expressed in terms of polynomials leads to the conclusion that the individual elements of the undetermined coefficients express nodal degrees of freedom. For now, we shall simply state that this is the case. The explanation lies with completeness and continuity, but also convenience. The details of how and why the polynomials are adapted to represent individual nodal degrees of freedom shall be the subject of much attention in this course.

The shape functions **N** is a row vector (when we introduce more variables in the polynomial assumptions it is convenient to turn **N** into a matrix) while the nodal displacement vector **d** is a column vector. The product **Nd** is therefore a scalar quantity. If we use the relation between strain and displacements we find the following relation;

$$\varepsilon = \partial\mathbf{u} = \partial\mathbf{N}\mathbf{d} \qquad \textbf{(36)}$$

The nodal displacement vector does not depend on *x*, and therefore we only need to differentiate the shape functions in order to find the strains. The unit displacement strain matrix **B**, may thus be expressed as;

$$\mathbf{B} = \partial\mathbf{N} \Rightarrow \varepsilon = \mathbf{B}\mathbf{d} \qquad \textbf{(37)}$$

If we substitute the strain expressions in the total potential energy functional, we find;

$$\Pi = \frac{1}{2}\int_V \mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\,dV - \int_V \mathbf{d}^T\mathbf{N}^T\mathbf{F}\,dV - \int_S \mathbf{d}^T\mathbf{N}^T\Phi\,dS$$

Now we shall attempt to develop a general expression for the stiffness matrix **k** and the load vector **r**. In order to do so, we have to use one general result on the symmetry of the strain energy density as well as a result from calculus in linear algebra. The strain energy density is only a number for each volumetric point;

$$\sigma^T\varepsilon = \mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d} = f(x, y, z)$$

This entails that the strain energy density is a scalar function (as opposed to a vector function). This is physically obvious, since the strain energy in a point cannot have a direction. Since the strain energy density is a scalar function, it is equal to its own inverse;

$$\left(\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\right) = \left(\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\right)^T$$

We also need the following result from linear algebra;

$$\delta\left(\mathbf{x}^T\mathbf{A}\mathbf{x}\right) = \delta\mathbf{x}^T\mathbf{A}\mathbf{x} + \mathbf{x}^T\mathbf{A}\delta\mathbf{x}$$

Now we invoke the principle of minimum potential energy;

$$\delta\Pi = \frac{1}{2}\delta\int_V \mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\,dV - \delta\int_V \mathbf{d}^T\mathbf{N}^T\mathbf{F}dV - \delta\int_S \mathbf{d}^T\mathbf{N}^T\Phi dS = 0$$

The variation operation should relate to the undetermined coefficients **d**, since **B**, **N** and **E** are matrices of known functions. The variation operation may therefore be expressed as follows;

$$\delta\Pi = \frac{1}{2}\int_V \delta\left(\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\right)dV - \int_V \delta\left(\mathbf{d}^T\mathbf{N}^T\mathbf{F}\right)dV - \int_S \delta\left(\mathbf{d}^T\mathbf{N}^T\Phi\right)dS$$

$$= \frac{1}{2}\int_V \delta\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d} + \mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\,\delta\mathbf{d}dV - \int_V \delta\mathbf{d}^T\mathbf{N}^T\mathbf{F}dV - \int_S \delta\mathbf{d}^T\mathbf{N}^T\Phi dS$$

We use the relation $\left(\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\,\delta\mathbf{d}\right) = \left(\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\,\delta\mathbf{d}\right)^T$;

$$\delta\Pi = \frac{1}{2}\int_V \left(\delta\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d} + \delta\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}\right)dV - \int_V \delta\mathbf{d}^T\mathbf{N}^T\mathbf{F}dV - \int_S \delta\mathbf{d}^T\mathbf{N}^T\Phi dS$$

$$= \int_V \delta\mathbf{d}^T\mathbf{B}^T\mathbf{E}\mathbf{B}\mathbf{d}dV - \int_V \delta\mathbf{d}^T\mathbf{N}^T\mathbf{F}dV - \int_S \delta\mathbf{d}^T\mathbf{N}^T\Phi dS$$

$$= \delta\mathbf{d}^T\left(\int_V \mathbf{B}^T\mathbf{E}\mathbf{B}dV\mathbf{d} - \int_V \mathbf{N}^T\mathbf{F}dV - \int_S \mathbf{N}^T\Phi dS\right) = 0$$

By the fundamental theorem of variational calculus, we conclude that the variation is admissible, and therefore we may set the content of the parenthesis equal to zero;

$$\int_V \mathbf{B}^T\mathbf{E}\mathbf{B}dV\mathbf{d} - \int_V \mathbf{N}^T\mathbf{F}dV - \int_S \mathbf{N}^T\Phi dS = 0$$

We may group the terms according to physical interpretation;

$$\int_V \mathbf{B}^T\mathbf{E}\mathbf{B}dV = \mathbf{k}$$

$$\int_V \mathbf{N}^T\mathbf{F}dV + \int_S \mathbf{N}^T\Phi dS = \mathbf{r}$$

Thus the integral over the strain energy density is interpreted as the element stiffness matrix and the accumulated work done by volume forces and tractions give rise to the element load vector. The variational operation may thus be concluded by the following equation;

$$\mathbf{k}\mathbf{d} = \mathbf{r},$$

which is the familiar equilibrium equation. We have established element equilibrium equations by means of the Rayleigh-Ritz method. We have assumed a set of displacement polynomials and applied the principle of minimum potential energy to establish equilibrium. There are different means of establishing element equilibrium, one example is weighted residuals methods (introduced in Cook et. al. in chapter 5), others are direct physical derivation and the method of virtual displacements. For structural mechanics problems, the Rayleigh-Ritz method covers all necessary applications since the principle of minimum potential energy is sufficient to describe static problems. Thus we shall devote ourselves to this method in the following parts of this course.

4.2 Example – Deduction of the stiffness matrix for a bar element

We have deduced the stiffness matrix for a bar element several times during this course. We shall perform this operation one last time, in order to show that the new notation and the new result for the generalisation of the variational operation results in a smooth and simple general method to establish stiffness matrices. We assume the familiar displacement functions;

$$\mathbf{u} = d_1\left(1 - \frac{x}{L}\right) + d_2\frac{x}{L}$$

The displacement functions may be expressed in our new notation;

$$\mathbf{N} = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix}$$

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

The only strains we are concerned with are axial strains, and they are assumed to be constant over any perpendicular cross-section. The strains are thus simply $\varepsilon_{xx} = \frac{du}{dx}$. The differential operator is therefore $\partial = \left\{ \frac{d}{dx} \right\}$. We calculate the **B**-matrix by multiplying the differential operator with the shape functions;

$$\mathbf{B} = \partial\mathbf{N} = \frac{d}{dx}\begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

When we know the **B**-matrix we may deduce the stiffness matrix;

$$\mathbf{k} = \int_V \mathbf{B}^T\mathbf{E}\mathbf{B}\,dV = \int_L\int_A \begin{bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{bmatrix} E \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} dA dx = EA\int_L \begin{bmatrix} \frac{1}{L^2} & -\frac{1}{L^2} \\ -\frac{1}{L^2} & \frac{1}{L^2} \end{bmatrix} dx$$

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

We find that the new notation and technique to establish the stiffness matrix is convenient and much simpler than previous efforts.

4.3 Limitations to the Rayleigh-Ritz method

We have seen how to use the Rayleigh method and the Rayleigh-Ritz method to approximate displacement behaviour by assuming displacement functions and calculating their amplitude(s) by means of the principle of minimum potential energy. However, this method introduces some limitations and some important problems which should be understood when using the method.

When we introduce shape functions to dictate the displacement behaviour of a system we constrain its natural freedom of displacement which generally is total. We force the solution to exist in a solution space which we have confined by the extent/applicability of our assumed displacement functions. Whenever we constrain our system, we must re-evaluate the principle of minimum potential energy. The displacements in a natural unconstrained system will find a state of equilibrium for which the internal strain energy in the system is minimized. If we constrain the solution space, we will always, at least to some extent, reduce the systems inherent ability to find the least complicated state of equilibrium, where as little strain energy as possible is stored. Thus the consequence of restraining the solution space is to heighten the stiffness of our systems. When we heighten the stiffness of our systems this has two essential effects;

- In load controlled conditions we generally underestimate the displacements
- In displacement conditions we generally overestimate the induced stresses

Another important aspect is the requirement that the displacement functions must fulfil the boundary conditions of our physical problems. Otherwise, strain energy will be stored where it is not supposed to.

4.4 The stiffness matrix for a beam element

In order to establish the stiffness matrix for a beam, we need displacement functions related to vertical displacements and rotations at both nodes of an element. The displacement functions for beams are called the beam equations, and they are listed as follows;

$$\mathbf{N} = \left\{ 1 - 3\left(\frac{x}{L}\right)^2 + 2\left(\frac{x}{L}\right)^3 \quad -L\left[\frac{x}{L} - 2\left(\frac{x}{L}\right)^2 + \left(\frac{x}{L}\right)^3\right] \quad 3\left(\frac{x}{L}\right)^2 - 2\left(\frac{x}{L}\right)^3 \quad L\left[\left(\frac{x}{L}\right)^2 - \left(\frac{x}{L}\right)^3\right] \right\}$$

A graphic representation is given in Figure 4.4-1.

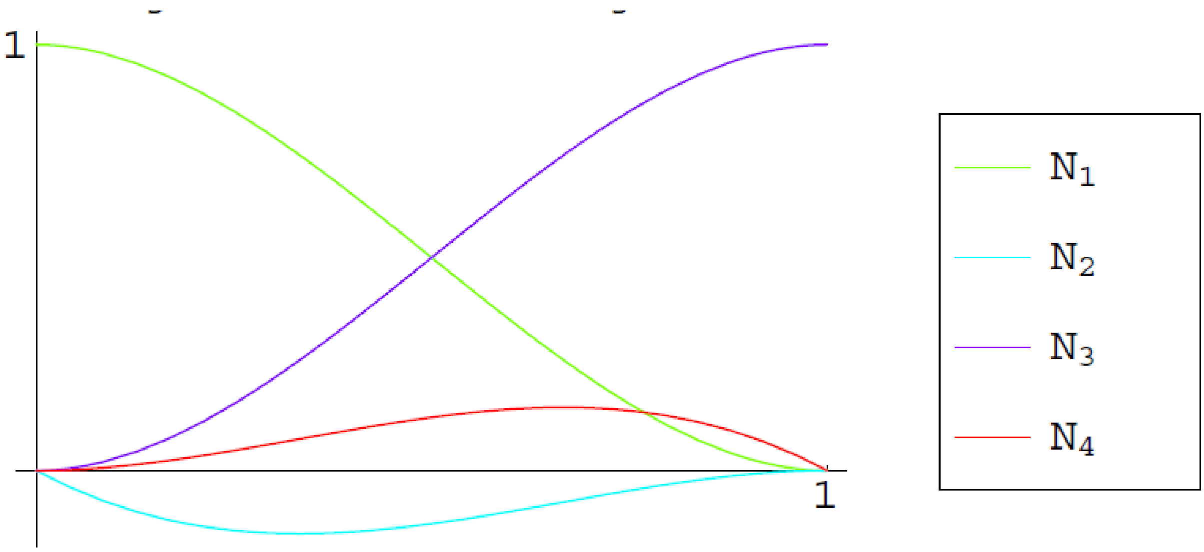


Figure 4.4-1 – Shape functions for beam elements (the beam equations)

We note that by constraining either function we impose a boundary condition of fixed displacement and/or rotation at the relevant node in a beam element. This is how boundary conditions are introduced practically – we impose them by eliminating the assumed displacement functions which relate to the relevant degree of freedom.

From section 3.7.1 we know that the strain energy in a beam may be expressed as the product of moment and curvature integrated over the length of the beam;

$$U = \int_L M\kappa dx = \int_L EI\left(\frac{d^2w}{dx^2}\right)^2 dx$$

If we assume that the second moment of inertia and the Young modulus is constant over the length of the beam, we may find a differential operator for a beam element;

$$\partial = \frac{d^2}{dx^2}$$

Using the relation $\mathbf{B} = \partial\mathbf{N}$, we find;

$$\mathbf{B} = \partial\mathbf{N}$$
$$= \frac{d^2}{dx^2}\left\{1-3\left(\frac{x}{L}\right)^2+2\left(\frac{x}{L}\right)^3 \quad -L\left[\frac{x}{L}-2\left(\frac{x}{L}\right)^2+\left(\frac{x}{L}\right)^3\right] \quad 3\left(\frac{x}{L}\right)^2-2\left(\frac{x}{L}\right)^3 \quad L\left[\left(\frac{x}{L}\right)^2-\left(\frac{x}{L}\right)^3\right]\right\}$$
$$= \frac{1}{L^2}\left\{-6+12\frac{x}{L} \quad -L\left(6\frac{x}{L}-4\right) \quad 6-12\frac{x}{L} \quad L\left(2-6\frac{x}{L}\right)\right\}$$

Now we may insert into the expression for the general calculation of the stiffness matrix;

$$\mathbf{k} = \int_L EI\mathbf{B}^T\mathbf{B}dx = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & -6L \\ -6L & 4L^2 & 6L & 2L^2 \\ -12 & 6L & 12 & 6L \\ -6L & 2L^2 & 6L & 4L^2 \end{bmatrix} \qquad \textbf{(38)}$$

Nodal loads are coupled to the relevant nodal displacements. Nodal moments are coupled to the relevant nodal rotations. Distributed loads are transformed to nodal loads and nodal moments by integration of the displacement functions. Example by distributed load for beam element;

$$\mathbf{r} = \int_L q\mathbf{N}^T dx = \begin{Bmatrix} \frac{qL}{2} \\ \frac{qL^2}{12} \\ \frac{qL}{2} \\ -\frac{qL^2}{12} \end{Bmatrix} \tag{39}$$

4.5 Extended example – Beam frame

In this Section we shall thoroughly investigate a beam frame, inspired by an example given by Bell, (Bell, K. “Matrisestatikk”, Tapir Forlag, 1994). This example is also intended to demonstrate the process of a finite element analysis by dividing the analysis into a local element analysis and a system analysis. The local element analysis is characterized by the following steps:

- Uncoupling (in nodes)/ Meshing
- Localisation (apply local coordinate systems)
- Element formulation (Stiffness relation / Material law)
- Element load vector formulations

The system analysis applies all the local element analyses to combine into a global stiffness relation by the following steps:

- Globalisation (Global coordinate system)
- Assembly (Combine the individual local element stiffness matrices to a global stiffness matrix)
- Boundary conditions
- Solution (Determine nodal displacements)
- Derived solutions (Stresses, strains, reaction forces and/or force resultants and moment resultants etc.)

To make the process of transforming local element analyses to the system analysis smooth, we set two requirements:

- The system degrees of freedom and the element degrees of freedom are expressed in the same coordinate system
- The system degrees of freedom have a one-to-one relation with the element degrees of freedom

4.5.1 The problem

In Figure 4.5-1, the beam frame which we shall solve for is shown.

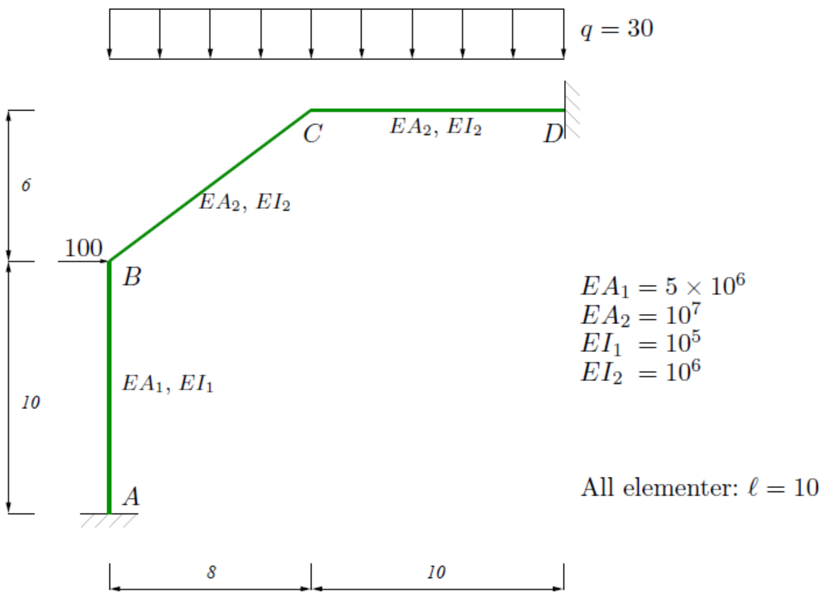


Figure 4.5-1 – Beam frame with a point load and a distributed load

The first task in the local analysis is to decouple the elements in nodes, or to mesh the system.

4.5.2 Meshing

The meshing may be done in an arbitrary number of ways, as we could divide each individual member of the frame into any number of elements. For the present analyses we shall choose three elements, and we shall introduce nodes at positions A, B, C, and D. This leads to the following elements:

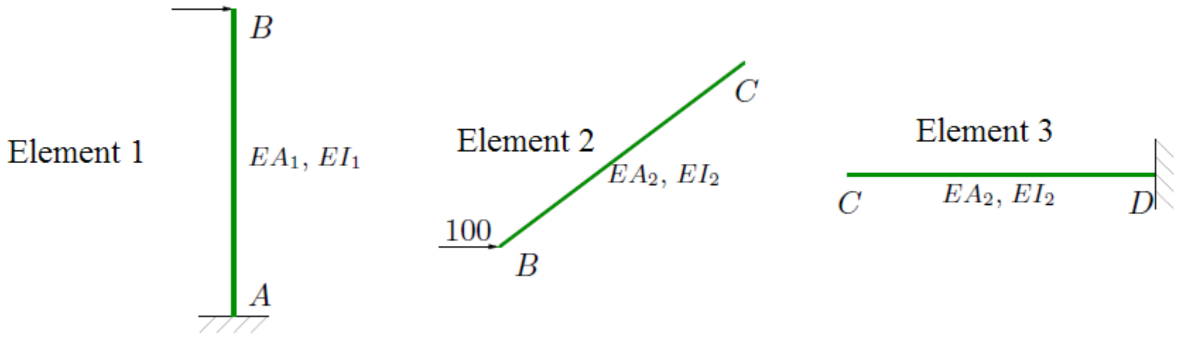


Figure 4.5-2 – Meshing of the system into 3 elements

The following global node numbering is chosen:

- Node 1 in point A
- Node 2 in point B
- Node 3 in point C
- Node 4 in point D

4.5.3 Localisation

The element nodal enumeration and corresponding directionalities of degrees of freedom are given as follows:

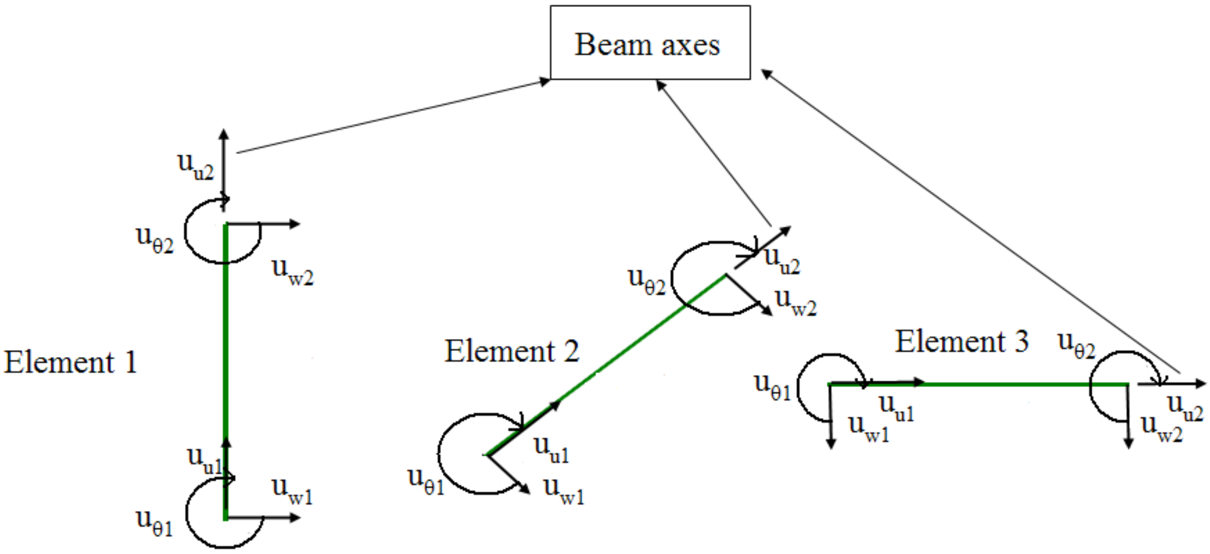


Figure 4.5-3 – Local degrees of freedom and orientation of beam axes for the three decoupled elements

In Figure 4.5-3, the displacements u_{wi} are transverse nodal displacements perpendicular to the beam axis, u_{ui} are axial displacements parallel to the beam axis and $u_{\theta i}$ are nodal rotations. Each beam element has its own beam axis, and orients its local coordinate system using the beam axis as the x-axis.

4.5.4 Element formulation

The stiffness matrix for a beam element where axial displacements are disregarded is given in equation **(40)**. In the present case, however, we shall include axial displacements also. Axial displacements of beams are governed by the same behaviour as axial displacements of bars. This will be shown in Section 7, when a more rigorous deduction of the element stiffness matrix for beams shall be performed. Combining the element stiffness relations of bars and beams results as follows:

$$\mathbf{kd} = \mathbf{r} \Rightarrow \frac{E}{L^3}\begin{bmatrix} AL^2 & 0 & 0 & -AL^2 & 0 & 0 \\ 0 & 12I & 6IL & 0 & -12I & -6IL \\ 0 & 6IL & 4IL^2 & 0 & 6IL & 2IL^2 \\ -AL^2 & 0 & 0 & AL^2 & 0 & 0 \\ 0 & -12I & 6IL & 0 & 12I & 6IL \\ 0 & -6IL & 2IL^2 & 0 & 6IL & 4IL^2 \end{bmatrix}\begin{bmatrix} u_{u1} \\ u_{w1} \\ u_{\theta 1} \\ u_{u2} \\ u_{w2} \\ u_{\theta 2} \end{bmatrix} = \begin{bmatrix} r_{u1} \\ r_{w1} \\ r_{\theta 1} \\ r_{u2} \\ r_{w2} \\ r_{\theta 2} \end{bmatrix}$$

The element stiffness relation shown above is simply the linear combination of the stiffness relations for beams and bars, assuming that bar and beam behaviours are linearly independent.

As the stiffness relation is determined, the remaining part of the localisation process is to determine the local nodal load vectors for the individual elements.

4.5.5 Formulation of element load vectors

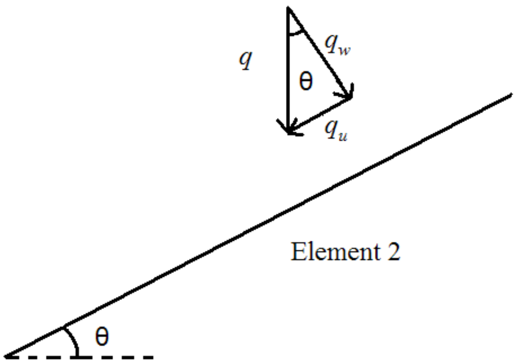
Load vector for element 1:

Element 1 is exposed only to a nodal load at node 2, in transverse direction relative to the beam axis. The load vector for the element is thus:

$$\mathbf{r}_1^e = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 100 \\ 0 \end{bmatrix}$$

Load vector for element 2:

Element 2 is exposed to a nodal load at global node 2, which is the local node number 1 for the element. This load also applies to element 1, which leaves us with the choice of whether we should apply the load in the element load vector for element 1 or 2. We have included it for element 1, and therefore we cannot apply it for element 2 as well. The other load acting on element 2 is the uniformly distributed load q. This load acts at an angle to the beam axis, different from the perpendicular angle, and may therefore be decomposed into two separate directions.



In the figure above, the angle of attack of q is perpendicular to the dotted line, and q_w is perpendicular to element 2. Thus the angle between these lines must be equal. Based on the dimensions of element 2, shown in Figure 4.5-1, we find the following:

$$\sin\theta = \frac{6}{10}$$

$$\cos\theta = \frac{8}{10}$$

The perpendicular and parallel components of the loading may therefore be expressed as functions of q:

$$q_w = q\cos\theta$$

$$q_u = q\sin\theta$$

The perpendicular component q_w is a distributed load corresponding to the beam behaviour of element 2. The element loads for this load is described in equation **(41)**. The result is therefore:

$$\mathbf{r}_{qw} = \frac{1}{12} q \cos\theta L \begin{bmatrix} 0 \\ 6 \\ L \\ 0 \\ 6 \\ -L \end{bmatrix} = \begin{bmatrix} 0 \\ -120 \\ 200 \\ 0 \\ -120 \\ -200 \end{bmatrix}$$

Note that the transverse nodal loads are negative, since the positive y-axis is directed in the opposite direction of the loading. For the horizontal component q_u, the load must be calculated according to the below expression:

$$\int_V \mathbf{N}^T \mathbf{F} dV + \int_S \mathbf{N}^T \Phi dS = \mathbf{r}$$

The assumed displacement functions for bars are given in Section 4.2:

$$\mathbf{N} = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix}$$

Combining the two equations allows a calculation of the corresponding load vector:

$$\mathbf{r}_{qu} = \int_0^L -q \cos\theta \begin{bmatrix} 1 - \frac{x}{L} \\ \frac{x}{L} \end{bmatrix} dx = -\frac{1}{2} qL \cos\theta \begin{bmatrix} 1 \\ 1 \end{bmatrix} = -\begin{bmatrix} 90 \\ 90 \end{bmatrix}$$

Note that the axial loading vector is set negative, since the direction of q_u is the opposite of the positive beam axis. Expanding the $\mathbf{r}_{qu}$ vector to the right dimension, and combining with $\mathbf{r}_{qw}$ yields the element load vector:

$$\mathbf{r}_2^e = \mathbf{r}_{qu}^e + \mathbf{r}_{qw} = \begin{bmatrix} -90 \\ -120 \\ 200 \\ -90 \\ -120 \\ -200 \end{bmatrix}$$

<u>Load vector for element 3:</u>

Element 3 is only subject to an uniformly distributed load, perpendicular to the beam axis, which means that equation **(41)** yields a consistent load vector:

$$\mathbf{r}_3^e = \frac{1}{12} qL \begin{bmatrix} 0 \\ 6 \\ L \\ 0 \\ 6 \\ -L \end{bmatrix} = \begin{bmatrix} 0 \\ -150 \\ 250 \\ 0 \\ -150 \\ -250 \end{bmatrix}$$

4.5.6 Globalisation

At the present stage in the analyses, all elements and load vectors are represented in their own local coordinate system. The next step in the analysis is to express all the local element stiffness matrices and the element load vectors in a common global coordinate system. For this process we need first to choose a global coordinate system, and secondly to find a transformation matrix for each of the elements. For the present analysis, the global coordinate system is chosen to have its origin at position A, and the y-axis is parallel to element 1 while the x-axis is parallel to element 3.

The next step is to find transformation matrices for each individual element. The transformation of axial and transverse displacement components was illustrated in Section 1.5.1 for bars. The same approach applies for beams in terms of rotation of translational degrees of freedom. Rotations, however, are rotationally invariant and therefore the rotations are not subject to transformation when the coordinate system changes. The resulting transformation matrix is found below:

$$\mathbf{T} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & \sin\theta & 0 \\ 0 & 0 & 0 & -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Now it only remains to transform the element stiffness matrices and the load vectors.

<u>Transformation of element 1</u>

Element 1 has its beam axis 90 degrees rotated from the x-axis. The resulting transformation matrix is given:

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

$$\mathbf{k}_1 = \mathbf{T}_1^T \mathbf{k} \mathbf{T}_1 = 10^3 \begin{bmatrix} 1.2 & 0 & 6 & -1.2 & 0 & 6 \\ 0 & 500 & 0 & 0 & -500 & 0 \\ 6 & 0 & 40 & -6 & 0 & 20 \\ -1.2 & 0 & -6 & 1.2 & 0 & -6 \\ 0 & -500 & 0 & 0 & 500 & 0 \\ 6 & 0 & 20 & -6 & 0 & 40 \end{bmatrix}$$

<u>Transformation of element 2</u>

Element 2 has its beam axis at an angle θ relative to the x-axis. The resulting transformation matrix is given:

$$\mathbf{T}_2 = \begin{bmatrix} 0.8 & 0.6 & 0 & 0 & 0 & 0 \\ -0.6 & 0.8 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.8 & 0.6 & 0 \\ 0 & 0 & 0 & -0.6 & 0.8 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

$$\mathbf{k}_2 = \mathbf{T}_2^T \mathbf{k} \mathbf{T}_2 = 10^3 \begin{bmatrix} 644 & 474 & 36 & -644 & -474 & 36 \\ 474 & 368 & -48 & -474 & -368 & -48 \\ 36 & -48 & 400 & -36 & 48 & 200 \\ -644 & -474 & -36 & 644 & 474 & -36 \\ -474 & -368 & 48 & 474 & 368 & 48 \\ 36 & -48 & 200 & -36 & 48 & 400 \end{bmatrix}$$

The load vector is also rotated at an angle θ relative to the x-axis, so the loading vector must also be rotated:

$$\mathbf{r}_2 = \mathbf{T}_2^T \mathbf{r}_2^e = \begin{bmatrix} 0 \\ -160 \\ 200 \\ 0 \\ -160 \\ -200 \end{bmatrix}$$

<u>Transformation of element 3</u>

Element 3 is parallel to the x-axis, and therefore there is no need to rotate neither the stiffness matrix nor the load vector:

$$\mathbf{k}_3 = \mathbf{k} = 10^3 \begin{bmatrix} 500 & 0 & 0 & -500 & 0 & 0 \\ 0 & 1.2 & -6 & 0 & -1.2 & -6 \\ 0 & -6 & 40 & 0 & 6 & 20 \\ -500 & 0 & 0 & 500 & 0 & 0 \\ 0 & -1.2 & 6 & 0 & 1.2 & 6 \\ 0 & -6 & 20 & 0 & 6 & 40 \end{bmatrix}$$

$$\mathbf{r}_3 = \mathbf{r}_3^e$$

4.5.7 Assembly

Prior to any application of boundary conditions, the number of degrees of freedom is a simple function of the number of nodes. In each node there are three degrees of freedom, consisting of two translations and a rotation. In the entire system we have chosen 4 nodes, which result in a total of 12 degrees of freedom. The global displacement vector may therefore be formulated as follows:

$$\mathbf{D}_g^T = [u_1 \quad v_1 \quad \theta_1 \quad u_2 \quad v_2 \quad \theta_2 \quad u_3 \quad v_3 \quad \theta_3 \quad u_4 \quad v_4 \quad \theta_4]$$

Since the system has 12 degrees of freedom, the global stiffness matrix must be a 12x12 matrix. This means we must augment all the local element stiffness matrices to 12x12 matrices, and all local element load vectors to 12x1 vectors. This process is best performed for each individual element, and will be discussed in this fashion.

Element 1

Element 1 has global nodes 1 and 2, and since there are three degrees of freedom for each node, the first 6 degrees of freedom are associated with element 1. Thus the contribution from element 1 to the global stiffness matrix is given as follows:

$$\mathbf{k}_1^g = 10^3 \begin{bmatrix} 1.2 & 0 & 6 & -1.2 & 0 & 6 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 500 & 0 & 0 & -500 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 6 & 0 & 40 & -6 & 0 & 20 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1.2 & 0 & -6 & 1.2 & 0 & -6 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -500 & 0 & 0 & 500 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 6 & 0 & 20 & -6 & 0 & 40 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

The load vector is associated with the same degrees of freedom and becomes as follows:

$$\left(\mathbf{r}_1^e\right)^T = [0 \quad 0 \quad 0 \quad 0 \quad 100 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0 \quad 0]$$

Element 2

Element 2 has global nodes 2 and 3, and since there are three degrees of freedom for each node, degrees of freedom 4 through 9 are associated with element 1. Thus the contribution from element 1 to the global stiffness matrix is given as follows:

$$
\mathbf{k}_2^g = 10^3 \begin{bmatrix}
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 644 & 474 & 36 & -644 & -474 & 36 & 0 & 0 & 0 \\
0 & 0 & 0 & 474 & 368 & -48 & -474 & -368 & -48 & 0 & 0 & 0 \\
0 & 0 & 0 & 36 & -48 & 400 & -36 & 48 & 200 & 0 & 0 & 0 \\
0 & 0 & 0 & -644 & -474 & -36 & 644 & 474 & -36 & 0 & 0 & 0 \\
0 & 0 & 0 & -474 & -368 & 48 & 474 & 368 & 48 & 0 & 0 & 0 \\
0 & 0 & 0 & 36 & -48 & 200 & -36 & 48 & 400 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
$$

The load vector is associated with the same degrees of freedom and becomes as follows:

$$
\left(\mathbf{r}_2^e\right)^T = \begin{bmatrix} 0 & 0 & 0 & 0 & -150 & 200 & 0 & -150 & -200 & 0 & 0 & 0 \end{bmatrix}
$$

<u>Element 3</u>

Element 3 has global nodes 3 and 4, and since there are three degrees of freedom for each node, the last 6 degrees of freedom are associated with element 3. Thus the contribution from element 3 to the global stiffness matrix is given as follows:

$$
\mathbf{k}_3^g = 10^3 \begin{bmatrix}
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 500 & 0 & 0 & -500 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 1.2 & -6 & 0 & -1.2-6 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -6 & 40 & 0 & 6 & 20 \\
0 & 0 & 0 & 0 & 0 & 0 & -500 & 0 & 0 & 500 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -1.2 & 6 & 0 & 1.2 & 6 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -6 & 20 & 0 & 6 & 40
\end{bmatrix}
$$

The load vector is associated with the same degrees of freedom and becomes as follows:

$$
\left(\mathbf{r}_3^e\right)^T = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & -150 & 250 & 0 & -150 & -250 \end{bmatrix}
$$

Having determined the element stiffness matrices and load vectors in the global coordinate system, and expanded their dimensions, the next step is to assemble the global stiffness matrix and the global load vector:

$$\mathbf{K}_g = \mathbf{k}_1^g + \mathbf{k}_2^g + \mathbf{k}_3^g$$

$$\mathbf{K}_g = 10^3 \begin{bmatrix} 1.2 & 0 & 6 & -1.2 & 0 & 6 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 500 & 0 & 0 & -500 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 6 & 0 & 40 & -6 & 0 & 20 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1.2 & 0 & -6 & 646 & 474 & 30 & -644 & -474 & 36 & 0 & 0 & 0 \\ 0 & -500 & 0 & 474 & 868 & -48 & -474 & -368 & -48 & 0 & 0 & 0 \\ 6 & 0 & 20 & 30 & -48 & 440 & -36 & 48 & 200 & 0 & 0 & 0 \\ 0 & 0 & 0 & -644 & -474 & -36 & 1644 & 474 & -36 & -500 & 0 & 0 \\ 0 & 0 & 0 & -474 & -368 & 48 & 474 & 380 & -12 & 0 & -1.2 & -6 \\ 0 & 0 & 0 & 36 & -48 & 200 & -36 & -12 & 800 & 0 & 6 & 20 \\ 0 & 0 & 0 & 0 & 0 & 0 & -500 & 0 & 0 & 500 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1.2 & 6 & 0 & 1.2 & 6 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -6 & 20 & 0 & 6 & 40 \end{bmatrix}$$

We observe that the global stiffness matrix is symmetric, and that all the diagonal elements are positive. The global load vector is calculated next:

$$\mathbf{R}_g^T = \left(\mathbf{r}_1^g + \mathbf{r}_2^g + \mathbf{r}_3^g\right)^T = \begin{bmatrix} 0 & 0 & 0 & 100 & -150 & 200 & 0 & -300 & 50 & 0 & -150 & -250 \end{bmatrix}$$

4.5.8 Boundary conditions

We have determined all the components of the global stiffness relation;

$$\mathbf{K}_g \mathbf{D}_g = \mathbf{R}_g$$

$$10^3 \begin{bmatrix} 1.2 & 0 & 6 & -1.2 & 0 & 6 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 500 & 0 & 0 & -500 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 6 & 0 & 40 & -6 & 0 & 20 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1.2 & 0 & -6 & 646 & 474 & 30 & -644 & -474 & 36 & 0 & 0 & 0 \\ 0 & -500 & 0 & 474 & 868 & -48 & -474 & -368 & -48 & 0 & 0 & 0 \\ 6 & 0 & 20 & 30 & -48 & 440 & -36 & 48 & 200 & 0 & 0 & 0 \\ 0 & 0 & 0 & -644 & -474 & -36 & 1644 & 474 & -36 & -500 & 0 & 0 \\ 0 & 0 & 0 & -474 & -368 & 48 & 474 & 380 & -12 & 0 & -1.2 & -6 \\ 0 & 0 & 0 & 36 & -48 & 200 & -36 & -12 & 800 & 0 & 6 & 20 \\ 0 & 0 & 0 & 0 & 0 & 0 & -500 & 0 & 0 & 500 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1.2 & 6 & 0 & 1.2 & 6 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -6 & 20 & 0 & 6 & 40 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \\ u_3 \\ v_3 \\ \theta_3 \\ u_4 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 100 \\ -150 \\ 200 \\ 0 \\ -300 \\ 50 \\ 0 \\ -150 \\ -250 \end{bmatrix}$$

The beam fram is fixed in nodes 1 and 4 which means that degrees of freedom 1, 2, 3, 10, 11 and 12 must be fixed. Thus we eliminate the relevant rows and columns:

$$
\begin{bmatrix}
1.2 & 0 & 6 & -1.2 & 0 & 6 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 500 & 0 & 0 & -500 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
6 & 0 & 40 & 6 & 0 & 20 & 0 & 0 & 0 & 0 & 0 & 0 \\
-1.2 & 0 & -6 & 646 & 474 & 30 & -644 & -474 & 36 & 0 & 0 & 0 \\
0 & -500 & 0 & 474 & 868 & -48 & -474 & -368 & -48 & 0 & 0 & 0 \\
6 & 0 & 20 & 30 & -48 & 440 & -36 & 48 & 200 & 0 & 0 & 0 \\
0 & 0 & 0 & -644 & -474 & -36 & 1644 & 474 & -36 & -500 & 0 & 0 \\
0 & 0 & 0 & -474 & -368 & 48 & 474 & 380 & -12 & 0 & -1.2 & -6 \\
0 & 0 & 0 & 36 & -48 & 200 & -36 & -12 & 800 & 0 & 6 & 20 \\
0 & 0 & 0 & 0 & 0 & 0 & -500 & 0 & 0 & 500 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -1.2 & 6 & 0 & 1.2 & 6 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -6 & 20 & 0 & 6 & 40
\end{bmatrix}
\begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \\ u_3 \\ v_3 \\ \theta_3 \\ u_4 \\ v_4 \\ \theta_4 \end{bmatrix}
=
\begin{bmatrix} 0 \\ 0 \\ 0 \\ 100 \\ -150 \\ 200 \\ 0 \\ -300 \\ 50 \\ 0 \\ -150 \\ -250 \end{bmatrix}
$$

The remaining system is therefore:

$$
\mathbf{KD} = \mathbf{R} \Rightarrow = 10^3
\begin{bmatrix}
645 & 474 & 30 & -644 & -474 & 36 \\
474 & 868 & -48 & -474 & -368 & -48 \\
30 & -48 & 440 & -36 & 48 & 200 \\
-644 & -474 & -36 & 1644 & 474 & -36 \\
-474 & -368 & 48 & 474 & 368 & -12 \\
36 & -48 & 200 & -36 & -12 & 800
\end{bmatrix}
\begin{bmatrix} u_2 \\ v_2 \\ \theta_2 \\ u_3 \\ v_3 \\ \theta_3 \end{bmatrix}
=
\begin{bmatrix} 100 \\ -150 \\ 200 \\ 0 \\ -300 \\ 50 \end{bmatrix}
$$

4.5.9 Solution

Inverting the above relation, and solving for the displacements yields the following global displacement vector:

$$
\mathbf{D} = 10^{-3}
\begin{bmatrix} -9.25 \\ -0.628 \\ 2.69 \\ 0.127 \\ -13.46 \\ -0.428 \end{bmatrix}
$$

The exact same solution may be determined using the following Matlab script:

```
EA1=5*10^6;
EA2=10^7;
EI1=10^5;
EI2=10^6;
L=10;

a=EA1/L;
b=EI1/L^3;
```

```
k=[a 0 0 -a 0 0; 0 12*b -6*b*L 0 -12*b -6*b*L; 0 -6*b*L 4*b*L^2 0 6*b*L
2*b*L^2; -a 0 0 a 0 0; 0 -12*b 6*b*L 0 12*b 6*b*L; 0 -6*b*L 2*b*L^2 0 6*b*L
4*b*L^2];
a=EA2/L;
b=EI2/L^3;
k2=[a 0 0 -a 0 0; 0 12*b -6*b*L 0 -12*b -6*b*L; 0 -6*b*L 4*b*L^2 0 6*b*L
2*b*L^2; -a 0 0 a 0 0; 0 -12*b 6*b*L 0 12*b 6*b*L; 0 -6*b*L 2*b*L^2 0 6*b*L
4*b*L^2];
T1=[0 1 0 0 0 0; -1 0 0 0 0 0; 0 0 1 0 0 0; 0 0 0 0 1 0; 0 0 0 -1 0 0; 0 0 0 0
0 1];
k1r=T1'*k*T1;
T2=[0.8 0.6 0 0 0 0; -0.6 0.8 0 0 0 0; 0 0 1 0 0 0; 0 0 0 0.8 0.6 0; 0 0 0 -
0.6 0.8 0; 0 0 0 0 0 1];
k2r=T2'*k2*T2;

K=k2r;
K(1:3,1:3)=K(1:3,1:3)+k1r(4:6,4:6);
K(4:6,4:6)=K(4:6,4:6)+k2(1:3,1:3);

R=[100 -150 200 0 -300 50]';
inv(K)*R*1000
```

The solution has also been implemented in ANSYS, resulting also in the exact same displacements:

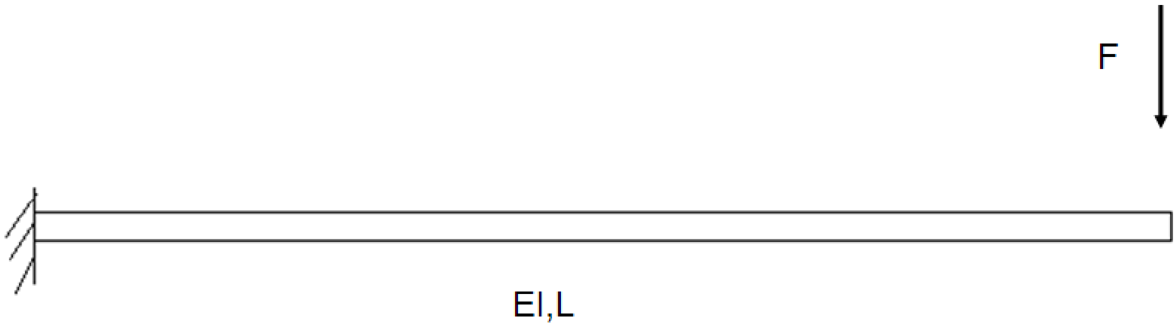
```
/BATCH,LIST
/FILNAM,Example4_5
/TITLE, Linear static beam frame analysis
/PREP7 ! Preprossessoren
ET,1,3 ! BEAM3 elementer
R,1,2.5e-5,5e-7,0.1 ! Height, I-value and E-value for element 1
R,2,5e-5,5e-6,0.1 !Height, I-value and E-value for elements 2 and 3
MP,EX,1,2e11 ! E-modulen
! Define all 4 nodes
N,1,0,0,0
N,2,0,10,0
N,3,8,16,0
N,4,18,16,0
! Set the real constant set for element 1
REAL,1
! Define Element 1 from node 1 to node 2
E,1,2
! Set the real constant set for element 2 and 3
REAL,2
!Define elements 2 and 3 from node 2 to 3, and 3 to 4 respectively
E,2,3
E,3,4
! Boundary conditions in node 1
D,1,ux
D,1,uy
D,1,rotz
! Boundary conditions in node 4
```

```
D,4,ux
D,4,uy
D,4,rotz
! Applied loading in node 2
! Note that the default direction of the z-axis is out of the plane of the screen. Thus the moments
! must be negative rather than positive.
F,2,fx,100
F,2,fy,-150
f,2,mz,-200
! Applied loading in node 3, comment on moment direction from node 2 applies here also
f,3,fy,-300
f,3,mz,-50
! Meshing, element definitions, boundary conditions and loading has been completed.
FINISH
/SOLU
SOLVE
```

4.6 Exercises

4.6.1 Mandatory assignment 2 in MEK4550

4.6.1.1 Exercise 1

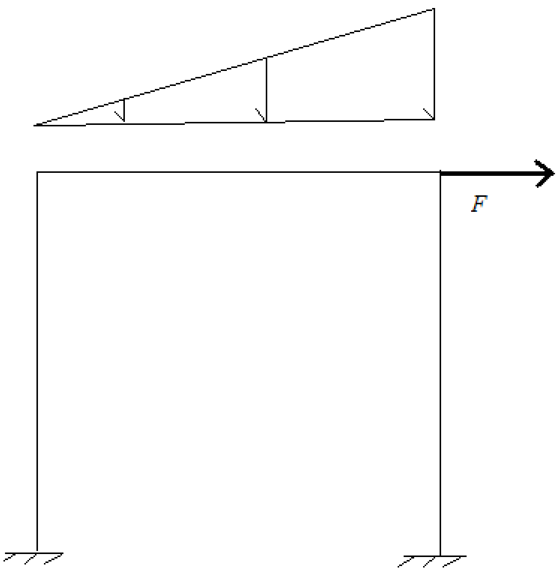


In this exercise we shall look at an end loaded cantilever beam. We will use the Rayleigh-Ritz method to develop a solution to the displacements. Generally we shall assume;

$$w = \sum_{n=2}^{m} c_n \left(\frac{x}{L}\right)^n$$

a) Show that the displacement assumption fulfils the boundary conditions for the beam
b) Vectorise the displacement assumption, and give it on the form **u**=**Nd**.
c) Find the load vector of the system
d) Find the stiffness matrix of the system
e) Implement the stiffness matrix and the load vector for general *m* in Matlab
f) Assume EI=1000, L=10, F=1. Use the analytical solution for comparison; How many terms must we include to achieve less than 1% error on the displacements? How accurately may we describe the moments?
g) We have created an “element” with particular boundary conditions and a set of displacement polynomials to achieve 1% accuracy on displacements. May this “element” be combined with other elements, if yes, how may continuity and kinematic compatability be ensured?
h) Would you characterise the proposed solution as a good solution?
i) Recalculate the load vector instead with a constant distributed load *q* over the length of the cantilever
j) How many terms are necessary now, in order to achieve a good solution (1% accuracy) on the displacements?
k) How did the change of load condition influence the accuracy of the moment calculations?

4.6.1.2 Exercise 2



EI=1000, L=10, A=1

We assume that bending stiffness, length and area is equal for all three elements.

a) The maximum load from the distributed load is *q*. Determine the load vector for the beam using the beam equations.
b) Establish the global displacement vector
c) Introduce the boundary conditions and re-establish the displacement vector
d) Establish the stiffness matrix for the system with boundary conditions readily implemented.
e) Assume q=1 and F=1 and solve for displacemenets
f) Use ANSYS to verify your results with BEAM3 elements
g) Use matlab to calculate the reaction forces in the two constraints and use ANSYS to confirm the results.

5 LINEAR TRIANGLE AND BILINEAR RECTANGLE

Cook – Sections 3.2, 3.4, 3.8 and 3.9

The elements we shall work with in this chapter are called membrane elements. Membrane behaviour is classified by pure in-plane behaviour. This means a membrane only deforms within its plane. An analogy to beams and bars would be that bars are to beams what membranes are to plates. A bar considers only axial deformation while the beam also includes rotation about the axis. A membrane only considers axial and in-plane shear deformation within its plane, but no rotations about any line in the membrane's plane.

We shall start by investigating the two most fundamental membrane elements. The linear triangle element, in the literature often referred as the constant strain triangle, and the bilinear rectangle. First we shall look at a mathematical description of the membrane behaviour.

5.1 Displacement assumption

Membrane elements only consider in-plane deformations, and all other deformations are considered negligible. Thus we have a two-dimensional problem. The displacement assumption is given in equation **(40)**;

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{Bmatrix} u_0(x,y) \\ v_0(x,y) \end{Bmatrix} \tag{40}$$

The strain field is based on plane stress or plane strain. Thus we only include in-plane strains in our stress-strain relation. The strain field is given in equation **(41)**;

$$\varepsilon = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \tag{41}$$

The relation between the displacement and the strain fields is given by the general differential operator ∂;

$$\partial = \begin{bmatrix} \dfrac{\partial}{\partial x} & 0 \\ 0 & \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial x} \end{bmatrix} \tag{42}$$

Before we make assumptions on the displacement field, i.e. insert polynomial assumptions for the displacements, we need to understand which requirements we place on the displacement polynomials;

- Convergence requirement: The displacements and their derivatives are continuous within the element borders. On the boundary the displacements are continuous but the strains are not. The general level of continuity is therefore $\mathbf{C}^{m-1}$ where m is the order of the polynomials in the polynomial assumptions.

- Completeness – The element must be capable of describing rigid body displacements and rigid body rotations as well as a constant strain field exactly. In order to achieve complete polynomials we apply Pascal's triangle:

Pascal's triangle for polynomials:

$$
\begin{array}{ccccccc}
 & & & 1 & & & \\
 & & x & & y & & \\
 & x^2 & & xy & & y^2 & \\
x^3 & & x^2 y & & xy^2 & & y^3
\end{array}
$$

5.2 Coordinate systems and coordinate transformations

Our study on the fundamental linear membrane elements changes the perspective of coordinate systems relative to bars and beams. When we study uni-axial elements, we only need to understand the orientation of the element relative to the x-axis, and to understand the length of each element. Thereby we may assemble global stiffness matrices fairly simply. When we take the step into a two-dimensional reality, we may no longer apply this simpler approach. A two-dimensional element may take any size and within the confines of its definition also shape. Thus the stiffness matrix for an individual element may vary greatly from other elements. There is a vast array of methods to handle the issues of coordinate systems and indeed how to determine stiffness matrices and the relation between their local and global coordinates. In this chapter we shall investigate two methods, which are among the simpler approaches. As an introduction to coordinate systems and coordinate transformations in two dimensions we will study:

- Generalised coordinate systems, and
- Natural coordinates

The systems we introduce are general and applicable to most element configurations. They are not confined to two dimensions either. For the constant strain triangle we shall apply generalised coordinates, and for the bi-linear rectangle we shall apply natural coordinates. Natural coordinates are not readily applicable to triangular elements, but generalised coordinates apply to any kind of element geometry. Applying generalised coordinates to the bi-linear rectangle will be the subject of exercises.

5.3 Constant strain triangle

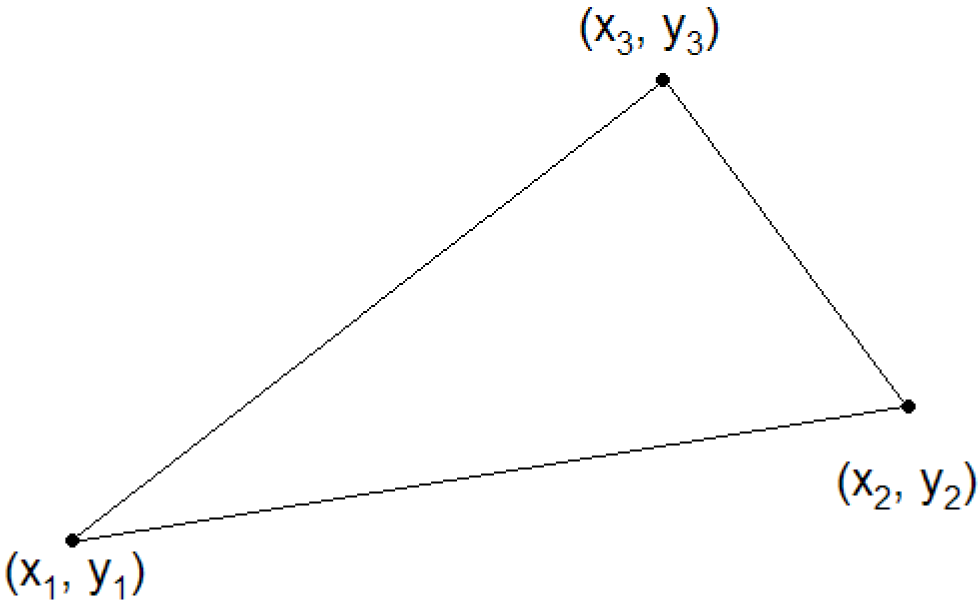


Figure 5.3-1 – Three nodes with respective x- and y coordinates

The number of degrees of freedom for the constant shear triangle is six. We have two displacements in each node, which means six linearly independent displacements. In each direction however, we only have three degrees of freedom. With three degrees of freedom we need three displacement polynomials. Based on Pascal's triangle, we find the three first polynomials to use as a displacement assumption for the constant strain triangle.

$$P = P(1, x, y)$$

For the constant strain triangle we shall apply generalised coordinates. The principle of generalised coordinates is illustrated by example applied to the constant strain triangle.

First we write our displacement assumption on vector matrix form:

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \mathbf{N_{q0}} & 0 \\ 0 & \mathbf{N_{q0}} \end{bmatrix} \begin{Bmatrix} \mathbf{q}_u \\ \mathbf{q}_v \end{Bmatrix} \qquad \textbf{(43)}$$

The quantities $\mathbf{N_{q0}}$ and $\mathbf{q}$ are described by the following equations:

$$\mathbf{q}_u = \begin{Bmatrix} q_{u1} \\ q_{u2} \\ q_{u3} \end{Bmatrix}, \mathbf{q}_v = \begin{Bmatrix} q_{v1} \\ q_{v2} \\ q_{v3} \end{Bmatrix}$$

$$\mathbf{N_{q0}} = \{1 \quad x \quad y\}$$

In equation **(43)** we have thus rewritten the displacement assumption on a vector matrix form.

Generally, we want the undetermined coefficients of each of the polynomials to represent the magnitude of a specific nodal displacement rather than being general coefficients of a polynomial, in order to conveniently apply boundary conditions and kinematic compatibility. Thus we want the shape of our equation on the form:

$$\mathbf{u} = \mathbf{N}\mathbf{d} \tag{44}$$

Currently we have an equation on the form;

$$\mathbf{u} = \mathbf{N_q}\mathbf{q} \tag{45}$$

In order to transfer the equation over to a form we want, we have to perform some transformation of the product $\mathbf{N_q}\mathbf{q}$. The transformation is achieved by simply evaluating the displacement polynomials at the nodes of the element:

$$\mathbf{d} = \begin{Bmatrix} \mathbf{d}_u \\ \mathbf{d}_v \end{Bmatrix} = \begin{bmatrix} \mathbf{A}_0 & 0 \\ 0 & \mathbf{A}_0 \end{bmatrix} \begin{Bmatrix} \mathbf{q}_u \\ \mathbf{q}_v \end{Bmatrix} = \mathbf{A}\mathbf{q}$$

$$\mathbf{A}_0 = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}$$

By inverting the above relation, we find an expression for **q**;

$$\mathbf{q} = \mathbf{A}^{-1}\mathbf{d}$$

By combining equations **(44)** and **(45)** with the expression for **q**, we find;

$$\mathbf{u} = \mathbf{N}\mathbf{d} = \mathbf{N_q}\mathbf{q} = \mathbf{N_q}\mathbf{A}^{-1}\mathbf{d}$$

This leads to our conclusion for **N**:

$$\mathbf{N} = \mathbf{N_q}\mathbf{A}^{-1} \tag{46}$$

Since $\mathbf{q}_u$ and $\mathbf{q}_v$ are ordered in a specific fashion, we have introduced a restriction on **d**, which defines the ordering of the nodal degrees of freedom. The **d** vector is thus:

$$\mathbf{d}^T = \{d_{u1} \quad d_{u2} \quad d_{u3} \quad d_{v1} \quad d_{v2} \quad d_{v3}\} \tag{47}$$

Assuming that the inverse of **A** exists, which is not always the case, it is straightforward to determine the expression for the stiffness matrix. The **B**-matrix may be found by applying the differential operator defined in equation **(42)** to the expression for **N** as shown in equation **(46)**.

$$\mathbf{B} = \partial\mathbf{N} = \partial\left(\mathbf{N_q}\mathbf{A}^{-1}\right)$$

On block-matrix form, we may clarify further how the B-matrix is caculated:

$$\mathbf{B} = \partial\left(\mathbf{N_q}\mathbf{A}^{-1}\right) = \begin{bmatrix} \dfrac{\partial}{\partial x} & 0 \\ 0 & \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} \mathbf{N}_{q0} & \mathbf{0} \\ \mathbf{0} & \mathbf{N}_{q0} \end{bmatrix} \begin{bmatrix} \mathbf{A}_0^{-1} & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_0^{-1} \end{bmatrix} = \begin{bmatrix} \mathbf{N}_{q0,x}\mathbf{A}_0^{-1} & 0 \\ 0 & \mathbf{N}_{q0,y}\mathbf{A}_0^{-1} \\ \mathbf{N}_{q0,y}\mathbf{A}_0^{-1} & \mathbf{N}_{q0,x}\mathbf{A}_0^{-1} \end{bmatrix} \tag{48}$$

The derivatives of $\mathbf{N}_{q0}$ are easily determined:

$$\mathbf{N}_{q0,x} = \{0 \quad 1 \quad 0\}$$
$$\mathbf{N}_{q0,y} = \{0 \quad 0 \quad 1\}$$

As discussed in Section 3.3.2, the material law for a membrane is either plane stress or plane strain. The generalised Young's modulus **E**, found in the expression for the stiffness matrix will in this case thus be either that of plane stress or plane strain. Applying the expression for the **B**-matrix with a suitable material law yields the final expression for the element stiffness matrix;

$$\int_V \mathbf{B}^T \mathbf{E} \mathbf{B} dV = \mathbf{k}$$

Since the differentiation operation on the assumed polynomial displacements result in constant values for the **B**-matrix, the integration over the volume of the triangle simply results in a factor on the product $\mathbf{B}^T\mathbf{E}\mathbf{B}$ which is the volume, assuming that the thickness of the membrane element is constant over the area. Thus:

$$\mathbf{k} = Ah\mathbf{B}^T\mathbf{E}\mathbf{B} \tag{49}$$

In equation **(49)**, A is the area of the triangle and h is the assumed constant thickness of the triangle. From linear algebra, we may apply a general result to determine the area A of the triangle:

$$A = \frac{1}{2}\left|\det \mathbf{A}_0\right|$$

Thus we arrive at the final expression for **k**:

$$\mathbf{k} = \frac{h \cdot \left|\det \mathbf{A}_0\right|}{2}\mathbf{B}^T\mathbf{E}\mathbf{B} \tag{50}$$

5.3.1 Example – Stiffness matrix for a constant strain triangle element

We shall find the stiffness matrix for a triangle with the following properties:

- E = 207 GPa
- $v = 0.3$
- Uniform thickness of membrane – 10 *mm*
- Plane strain shall be assumed

The same element will be reinvestigated in section 5.3.3, and in that example subject to static analysis. The triangle is shown in Figure 5.3-2.

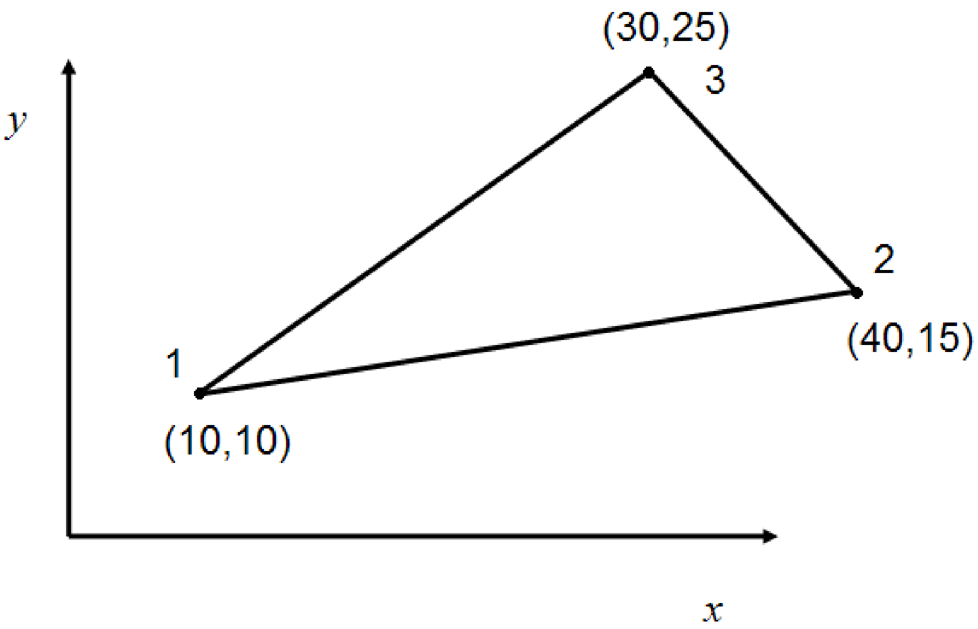


Figure 5.3-2 – A three node triangular element, units are in millimetres

First we compute $\mathbf{A}_0$:

$$\mathbf{A}_0 = \begin{bmatrix} 1 & 0.01 & 0.01 \\ 1 & 0.04 & 0.015 \\ 1 & 0.03 & 0.025 \end{bmatrix}$$

Note that millimetres have been transformed to metres. This has been done since the Young's modulus is defined in terms of force per square metre. From equation **(48)** we recognize that the inverse of $\mathbf{A}_0$ is necessary to compute:

$$\mathbf{A}_0^{-1} = \begin{bmatrix} 1.571429 & 0.142857 & -0.71429 \\ -28.5714 & 42.85714 & -14.2857 \\ -28.5714 & -57.1429 & 85.71429 \end{bmatrix}$$

The products $\mathbf{N}_{q0,x}\mathbf{A}_0^{-1}$ and $\mathbf{N}_{q0,y}\mathbf{A}_0^{-1}$ are computed:

$$\mathbf{N}_{q0,x}\mathbf{A}_0^{-1} = \{-28.5714 \quad 42.85714 \quad -14.2857\}$$
$$\mathbf{N}_{q0,y}\mathbf{A}_0^{-1} = \{-28.5714 \quad -57.1429 \quad 85.71429\}$$

Based on the above calculations, the **B**-matrix is found:

$$\mathbf{B} = \begin{bmatrix} -28.5714 & 42.86714 & -14.2857 & 0 & 0 & 0 \\ 0 & 0 & 0 & -28.5714 & -57.1429 & 85.71429 \\ -28.5714 & -57.1429 & 85.71429 & -28.5714 & 42.86714 & -14.2857 \end{bmatrix}$$

The remaining part is to compute the product of **B**-matrices and integrate over the volume to determine the stiffness matrix **k**.

$$\mathbf{k} = \frac{h \cdot \left|\det \mathbf{A}_0\right|}{2} \mathbf{B}^T \mathbf{E} \mathbf{B}$$

The product $\mathbf{B}^T\mathbf{E}\mathbf{B}$ is computed:

$$\mathbf{B}^T\mathbf{E}\mathbf{B} = \frac{E}{0.91} \begin{bmatrix} -28.57 & 0 & -28.57 \\ 42.87 & 0 & -57.14 \\ -14.29 & 0 & 85.71 \\ 0 & -28.57 & -28.57 \\ 0 & -57.14 & 42.87 \\ 0 & 85.714 & -14.29 \end{bmatrix} \begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix} \begin{bmatrix} -28.57 & 42.87 & -14.29 & 0 & 0 & 0 \\ 0 & 0 & 0 & -28.57 & -57.14 & 85.71 \\ -28.57 & -57.14 & 85.71 & -28.57 & 42.87 & -14.29 \end{bmatrix}$$

$$\mathbf{B}^T\mathbf{E}\mathbf{B} = E \begin{bmatrix} 1211 & -718 & -439 & 583 & 67 & -650 \\ -718 & 3274 & -2557 & 224 & -1749 & 1525 \\ -439 & -2557 & 3050 & 807 & 1682 & -875 \\ 583 & 224 & 807 & 1211 & 1323 & -2534 \\ 67 & -1279 & 1682 & 1323 & 4285 & -5618 \\ -650 & 1525 & -875 & -2534 & -5618 & 8152 \end{bmatrix}$$

Finally we may calculate **k** by multiplying the remainder of equation **(50)**:

$$\mathbf{k} = \frac{h \cdot \det \mathbf{A}_0}{2} \mathbf{B}^T\mathbf{E}\mathbf{B} = 362250 \cdot \begin{bmatrix} 1211 & -718 & -439 & 583 & 67 & -650 \\ -718 & 3274 & -2557 & 224 & -1749 & 1525 \\ -439 & -2557 & 3050 & 807 & 1682 & -875 \\ 583 & 224 & 807 & 1211 & 1323 & -2534 \\ 67 & -1279 & 1682 & 1323 & 4285 & -5618 \\ -650 & 1525 & -875 & -2534 & -5618 & 8152 \end{bmatrix}$$

Since lengths were transformed to metres, the unit of the stiffness matrix is N/m. Question: If rotational degrees of freedom are involved, will all the entries of the stiffness matrix still have the same unit?

5.3.2 A different approach to determine the stiffness matrix of a constant strain triangle element

As a note, an alternative calculation procedure for the stiffness matrix will be shown. The two methodologies, i.e. the methodology described previously in this chapter and the method which will be shown in the following, do not result in any particular differences in terms of computational efficiency or computational stability, so the choice of methodology is generally a choice of convenience.

Instead of transforming the $\mathbf{N}_q$-matrix to to an **N**-matrix via the **A**-matrix, the $\mathbf{B}_q$-matrix may be calculated instead:

$$\mathbf{B}_q = \partial \mathbf{N}_q = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} \mathbf{N}_{q0} & \mathbf{0} \\ \mathbf{0} & \mathbf{N}_{q0} \end{bmatrix} = \begin{bmatrix} \mathbf{N}_{q0,x} & 0 \\ 0 & \mathbf{N}_{q0,y} \\ \mathbf{N}_{q0,y} & \mathbf{N}_{q0,x} \end{bmatrix}$$

Based on the $\mathbf{B}_q$-matrix, the stiffness matrix is calculated in two steps:

$$\mathbf{k}_q = \int_V \mathbf{B}_q^T \mathbf{E} \mathbf{B}_q dV = \frac{h|\det \mathbf{A}_0|}{2} \mathbf{B}_q^T \mathbf{E} \mathbf{B}_q \qquad \textbf{(51)}$$

From equation **(51)** the $\mathbf{k}_q$ matrix is transformed to the correct global coordinate system via the **A**-matrix:

$$\mathbf{k} = \mathbf{A}^{-T} \mathbf{k}_q \mathbf{A}^{-1} \qquad \textbf{(52)}$$

To prove the relation between the **k** and the $\mathbf{k_q}$ matrix is left to the exercises.

5.3.3 Example – Simplified analysis of a triangular stiffener

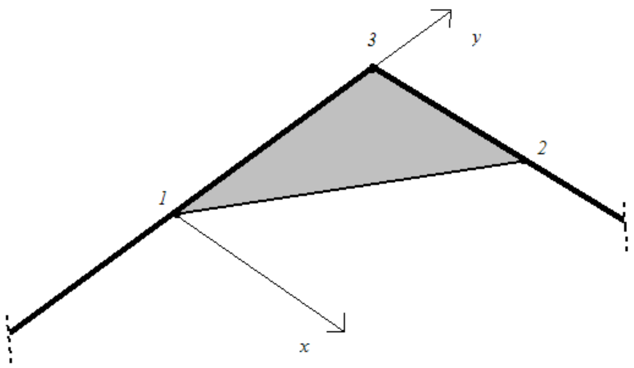


Figure 5.3-3 – Part of a larger structure, containing a stiffener

The triangle shown in Figure 5.3-3 is a steel stiffener in a larger structure. The structure is fixed in nodes 2 and 3, while the arm along the y-axis is flexible. The steel arms are conservatively considered to be so flexible that the stiffener absorbs all the loading.

The loading along the y-axis shall be simplified to include only point loads in node 1. F_x = 120 kN, F_y = 80 kN. The specifications for the stiffener are listed as follows:

- E = 207 GPa, v = 0.3
- h = 6 mm
- $x_i = \{0,0.4,0\}, y_i = \{0.0,0.4,0.4\}$.
- Plane stress shall be assumed
- Maximum allowable local principal stress is 420 MPa

The example shall determine if the stress in the triangular stiffener is within the allowable limit.

First the **A**-matrix is determined:

$$\mathbf{A} = \begin{bmatrix} \mathbf{A}_0 & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_0 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 1 & 0.4 & 0.4 & 0 & 0 & 0 \\ 1 & 0 & 0.4 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0.4 & 0.4 \\ 0 & 0 & 0 & 1 & 0 & 0.4 \end{bmatrix}$$

The inverse of **A** is found by simple calculation:

$$\mathbf{A}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 2.5 & -2.5 & 0 & 0 & 0 \\ -2.5 & 0 & 2.5 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2.5 & -2.5 \\ 0 & 0 & 0 & -2.5 & 0 & 2.5 \end{bmatrix}$$

$\mathbf{N_{q0,x}}$ and $\mathbf{N_{q0,y}}$ are found by differentiation of $\mathbf{N}_{q0}$:

$$\mathbf{N}_{q0,x} = \{0 \quad 1 \quad 0\}$$

$$\mathbf{N}_{q0,y} = \{0 \quad 0 \quad 1\}$$

The $\mathbf{B}_q$-matrix is simply a function of $\mathbf{N_{q0,x}}$ and $\mathbf{N_{q0,y}}$:

$$\mathbf{B}_q = \begin{bmatrix} \mathbf{N}_{q0,x} & 0 \\ 0 & \mathbf{N}_{q0,y} \\ \mathbf{N}_{q0,y} & \mathbf{N}_{q0,x} \end{bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix}$$

The stiffness matrix in generalised coordinates, $\mathbf{k}_q$ may now be calculated based on equation **(51)**:

$$\mathbf{k}_q = \frac{h \det \mathbf{A}_0}{2} \mathbf{B}_q^T \mathbf{E} \mathbf{B}_q = 10^8 \cdot \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1.0919 & 0 & 0 & 0 & 0.3276 \\ 0 & 0 & 0.3822 & 0 & 0.3822 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0.3822 & 0 & 0.3822 & 0 \\ 0 & 0.3276 & 0 & 0 & 0 & 1.0919 \end{bmatrix}$$

From equation **(52)**, the element stiffness matrix in global coordinates may be found:

$$\mathbf{k} = \mathbf{A}^{-T} \mathbf{k}_q \mathbf{A}^{-1} = 10^8 \cdot \begin{bmatrix} 2.3885 & 0 & -2.3885 & 0 & -2.3885 & 2.3885 \\ 0 & 6.8242 & -6.8242 & -2.0473 & 0 & 2.0473 \\ -2.3885 & -6.8242 & 9.2126 & 2.0473 & 2.3885 & -4.4357 \\ 0 & -2.0473 & 2.0473 & 6.8242 & 0 & -6.8242 \\ -2.3885 & 0 & 2.3885 & 0 & 2.3885 & -2.3885 \\ 2.3885 & 2.0437 & -4.4357 & -6.8242 & -2.3885 & 9.2126 \end{bmatrix}$$

We now know the element stiffness matrix for the stiffener. The next step is naturally to implement the boundary conditions. The element is fixed in nodes 2 and 3, which leaves only the degrees of freedom d_{u1} and d_{v1}. These are entries 1 and 4 in the nodal displacement vector, and therefore the global stiffness matrix, and global displacement vector may be found as follows:

$$\mathbf{K} = \begin{bmatrix} k_{11} & k_{14} \\ k_{41} & k_{44} \end{bmatrix} = 10^8 \cdot \begin{bmatrix} 2.3385 & 0 \\ 0 & 6.8242 \end{bmatrix}$$

$$\mathbf{D} = \begin{Bmatrix} d_{u1} \\ d_{v1} \end{Bmatrix}$$

The loads are point loads, and therefore we may apply them directly in the global load vector:

$$\mathbf{R} = \begin{Bmatrix} F_x \\ F_y \end{Bmatrix} = \begin{Bmatrix} 120000 \\ 80000 \end{Bmatrix}$$

To determine the nodal displacements in node 1 is now only a matter of algebra:

$$\mathbf{D} = \mathbf{K}^{-1}\mathbf{R} = 10^{-3} \begin{Bmatrix} 0.5024 \\ 0.1172 \end{Bmatrix}$$

In order to determine the element stress vector, it is easiest to determine the strains first. From equation **(37)**, we know that **ε** = **Bd**. Using the same methodology as in Example 5.3.1, the **B** matrix is found:

$$\mathbf{B} = \begin{bmatrix} 0 & 2.5 & -2.5 & 0 & 0 & 0 \\ 0 & 0 & 0 & -2.5 & 0 & 2.5 \\ -2.5 & 0 & 2.5 & 0 & 2.5 & 2.5 \end{bmatrix}$$

The nodal displacement vector is the result of the global displacement vector inserted into the element displacement vector:

$$\mathbf{d}^T = 10^{-3} \{0.5024 \quad 0 \quad 0 \quad 0.1172 \quad 0 \quad 0\}$$

The product of these quantities yields the strain:

$$\varepsilon = \mathbf{Bd} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = 10^{-3} \begin{Bmatrix} 0 \\ -0.2931 \\ -1.256 \end{Bmatrix}$$

Note that the compressive strain in x-direction is zero. Since the strains are known, it is a trivial matter to insert the strains into the material law to determine the stresses:

$$\sigma = \mathbf{E}\varepsilon = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix} = -\begin{Bmatrix} 20 \\ 67 \\ 100 \end{Bmatrix} MPa$$

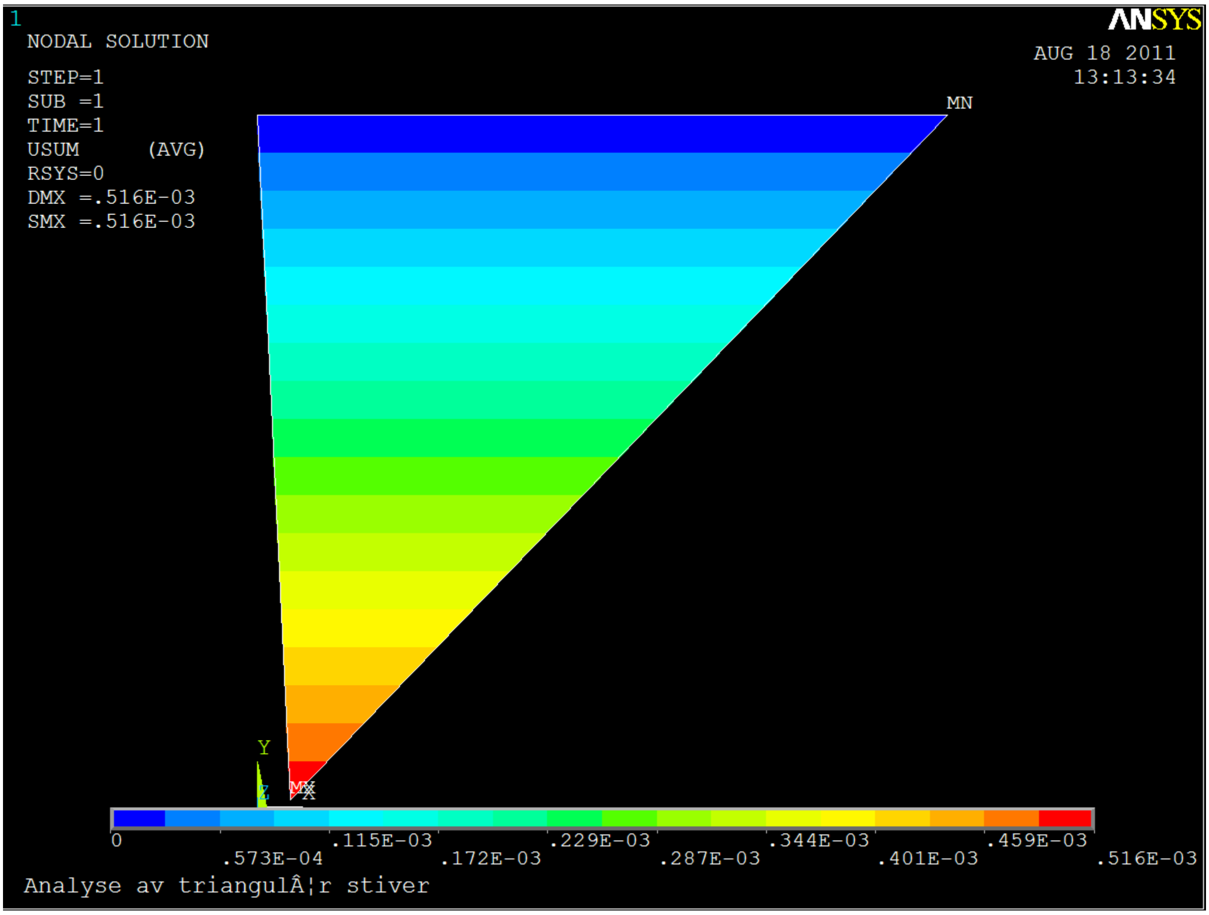
Question: How can the compressive stress in x-direction be non-zero when the strain is zero?

The acceptance criterion was based on the principal stresses. The two principal stresses are the Eigen-values of the stress tensor. The stress tensor is stated for ease of reference:

$$\sigma_{ij} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{bmatrix}$$

The Eigen-values of the stress tensor are in this case -146 MPa and 59 MPa respectively. The 1st principal stress is therefore much lower than the acceptable 420 MPa reference, and thus the stiffener is not exposed to excessive loading based on an analysis with only 1 element. In the exercises, the consequences of increasing the number of elements will be investigated.

The validity of the analyses is checked using ANSYS, with a linear triangular element called PLANE42. The displacement field is shown in the figure below:



The displacement field in the figure above is the vector sum of the lateral and transverse displacements. With the following ANSYS code, an exact match with the analytical results is found:

```
/BATCH, LIST
/FILENAME, TEST
/TITLE, Analyse av triangulær stiver
/PREP7
! Element type PLANE42
! Keyopts 1=0 (global coordinate system), 2=1 (no extra displacement shapes), 3=3 (Plane stress
! with thickness input), 5=0, 6=0 (No refinement on stress solution).
ET,1,42,0,1,3,,0,0
! Real constant set 1, is the thickness of the plate
```

```
R,1,0.006
! Assigning the real constants to element type 1
REAL,1
! Material properties
MP,EX,1,207E9
MP,EY,1,207E9
MP,NUXY,1,0.3
! Definition of nodes
N,1,0,0,0
N,2,0.4,0.4,0
N,3,0,0.4,0
! Defining an element with nodes 1, 2 and 3
E,1,2,3
! Boundary conditions
D,2,UX,0
D,2,UY,0
D,3,UX,0
D,3,UY,0
! Application of loads
F,1,FX,120000
F,1,FY,80000
! Finish preprocessor
FINISH
! Solution processor
/SOLU
SOLVE
```

The Matlab code for the present example is given below:

```matlab
% Establish constants
Emod=207*10^9;
v=0.3;
h=0.006;
Fx=120000;
Fy=80000;
% Plane stress material law
E=Emod/(1-v^2)*[1 v 0; v 1 0; 0 0 (1-v)/2];

%-------------------------------------------
% Calculate stiffness matrix
%-------------------------------------------

% A-matrix
A0=[1 0 0; 1 0.4 0.4; 1 0 0.4];
A(1:3,1:3)=A0;
A(4:6,4:6)=A0;
% Determine inverse of A and A0
Ai=inv(A);
A0i=inv(A0);
% Differentiated Nq-vectors
Nqx=[0 1 0];
```

```
Nqy=[0 0 1];
% Determine Bq-matrix
Bq=[Nqx 0 0 0; 0 0 0 Nqy; Nqy Nqx];
% kq-matrix
kq=0.5*det(A0)*h*Bq'*E*Bq
% Transform kq to k
k=Ai'*kq*Ai;
% Calculate B-matrix for later use with
% Stress calculations
NqxA0i=Nqx*A0i;
NqyA0i=Nqy*A0i;
B=[NqxA0i 0 0 0; 0 0 0 NqyA0i; NqyA0i NqxA0i];

%-------------------------------------------
% Determine global stiffness matrix
% and global load vector
%-------------------------------------------

K=zeros(2,2);
K(1,1)=k(1,1);
K(1,2)=k(1,4);
K(2,1)=k(4,1);
K(2,2)=k(4,4);
R=[Fx; Fy];

%-------------------------------------------
% Solve for the displacements
%-------------------------------------------

D=inv(K)*R

%-------------------------------------------
% Determine stresses
%-------------------------------------------

d=zeros(6,1);
d(1,1)=D(1,1);
d(4,1)=D(2,1);

Epsilon=B*d;
Sigma=E*Epsilon
Sigma_Tensor=[Sigma(1,1) Sigma(3,1); Sigma(3,1) Sigma(2,1)];

% Principal stresses
eigs(Sigma_Tensor)
```

5.4 The bi-linear rectangle

In Figure 5.4-1, a rectangular membrane element with 8 degrees of freedom is shown. The degrees of freedom are (u_i, v_i) and they are defined in each of the 4 corners of the rectangle.

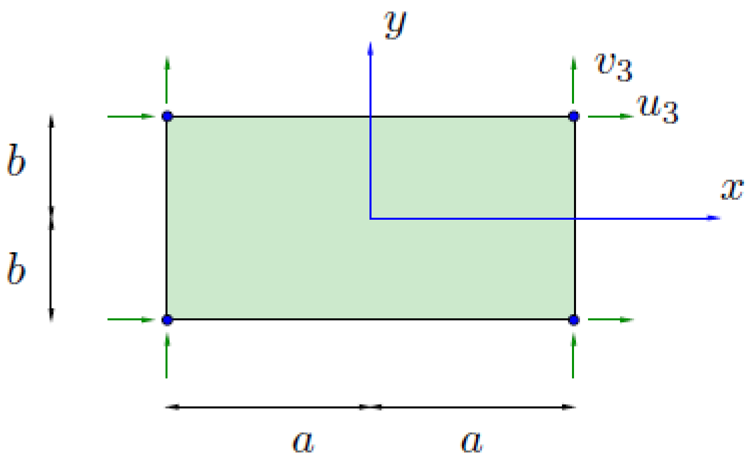


Figure 5.4-1 – A rectangular membrane with sides 2*a* and 2*b*

If we wish to define the stiffness matrix for general, rectangular membrane elements, there are an infinite number of possibilities to do so, since the dimensions of the rectangles have infinite possibilities for variation. We shall introduce natural coordinates to express all possible rectangles in the same unit type formulation, such that the stiffness matrix, and load vector expressions, may be expressed generally. The other alternative is to determine the stiffness matrix individually for any rectangular membrane element as part of the numerical process in the finite element solution methodology, but this is costly in terms of computational efficiency and therefore it is desirable to determine general expressions. The natural coordinates allow for a general methodology for rectangular elements.

5.4.1 Natural coordinates

We introduce two new coordinates:

$$\xi = \frac{x}{a}$$

$$\eta = \frac{y}{b}$$

The two new coordinates will take the intervals $\xi \in [-1,1], \eta \in [-1,1]$ if defined on the element and on its boundaries. It is also convenient to define the relation between the standard Cartesian coordinate representation and the natural coordinates on vector-matrix form:

$$\begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} a & 0 \\ 0 & b \end{bmatrix} \begin{bmatrix} \xi \\ \eta \end{bmatrix}, \begin{bmatrix} a & 0 \\ 0 & b \end{bmatrix} = \mathbf{A}$$

The coordinate transformation matrix is most often called the Jacobi-matrix, and generally denoted **J** in standard mathematical texts. The inverse relation is trivially found by inverting the coordinate transformation matrix:

$$\begin{bmatrix} \xi \\ \eta \end{bmatrix} = \begin{bmatrix} \frac{1}{a} & 0 \\ 0 & \frac{1}{b} \end{bmatrix} \begin{bmatrix} x \\ y \end{bmatrix}$$

To determine the stiffness matrix we must apply the general differential operator ∂, which means we must differentiate displacement assumptions. If we express our displacement assumptions in terms of natural coordinates it is therefore necessary to define differentiation of functions expressed in natural coordinates. This is, however, a simple matter of application of the chain rule:

$$\frac{\partial}{\partial x}=\frac{\partial}{\partial \xi}\frac{\partial \xi}{\partial x}+\frac{\partial}{\partial \eta}\frac{\partial \eta}{\partial x}=\frac{1}{a}\frac{\partial}{\partial \xi}$$
$$\frac{\partial}{\partial y}=\frac{\partial}{\partial \xi}\frac{\partial \xi}{\partial y}+\frac{\partial}{\partial \eta}\frac{\partial \eta}{\partial y}=\frac{1}{b}\frac{\partial}{\partial \eta}$$

This formulation also allows for a vector-matrix representation of differentiation:

$$\begin{bmatrix}\dfrac{\partial}{\partial x}\\ \dfrac{\partial}{\partial y}\end{bmatrix}=\mathbf{A}^{-1}\begin{bmatrix}\dfrac{\partial}{\partial \xi}\\ \dfrac{\partial}{\partial \eta}\end{bmatrix} \tag{53}$$

In addition to differentiation, we must also integrate functions depending on the natural coordinates, which means we must express the relation between integration in Cartesian coordinates and natural coordinates:

$$\int_{-b}^{b}\int_{-a}^{a} f(x,y)dxdy=\int_{-1}^{1}\int_{-1}^{1} g(\xi,\eta)\cdot abd\xi d\eta$$

For standard coordinate transformations, the factor separating $dxdy$ from $d\xi d\eta$ is the determinant of the coordinate transformation matrix, i.e. the determinant of the Jacobi matrix. This factor represents the difference in the area for an incremental part of the integration domains for the two coordinate representations.

5.4.2 Choice of displacement functions

For the triangular element we had three nodes, and three polynomial functions to choose from when determining the assumed displacement field. For the rectangle, we have 4 nodes, and subsequently we must apply 4 polynomial functions. We choose the basic form of the functions from Pascal's triangle:

$$\begin{array}{ccc} & 1 & \\ x & & y \\ & xy & \end{array}$$

Figure 5.4-2 – Four displacement functions chosen from Pascal's triangle

Note that either x^2 or y^2 could have been chosen instead of xy as the fourth term, as both of these functions are represented in the same row of Pascal's triangle. Either of the functions would generally work, but empirically it is found that the best general results are achieved if the functions are chosen such that the balance between x- and y-terms is as even as possible. We would for instance favour either x- or y-terms if one of the quadratic x^2 or y^2 terms were chosen instead. Physically this would be interpreted as having a dominance of second order displacement behaviour in only one of the coordinate directions.

In order to fulfil kinematic compatibility and to ease the implementation of boundary conditions, our displacement field must consist of functions which have unit value at one node and zero at the remaining nodes. As for the triangular elements, we shall use the same displacement polynomials in order to approximate displacements in u- and v-directions both. Thus we wish to determine four polynomials which have the properties shown in Figure 5.4-3.

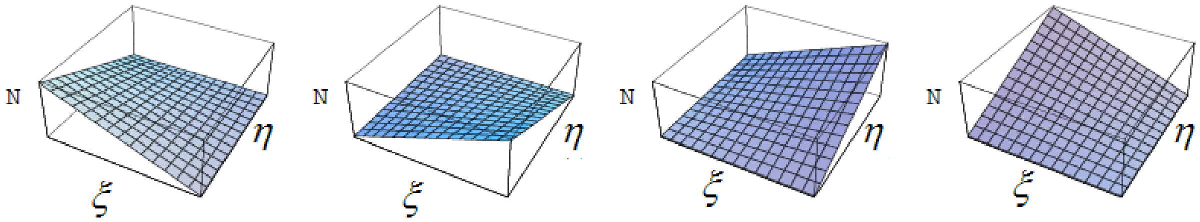


Figure 5.4-3 – Displacement polynomials for a bi-linear rectangle

In Section 7, interpolation techniques and methods to derive such polynomials will be discussed in detail. For now, the polynomial expressions will simply be presented:

$$N_i = \frac{1}{4}(1+\xi_i\xi)(1+\eta_i\eta) \tag{54}$$

In the above equation, ξ_i is the coordinate value of ξ in the i-th node, and correspondingly η_i is the coordinate value of η in the i-th node. If we enumerate the nodes according to Figure 5.4-4, we may find expressions for the displacement polynomials.

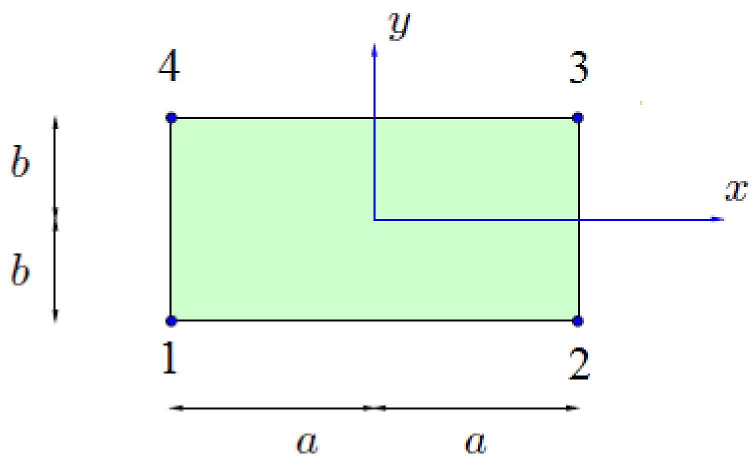


Figure 5.4-4 – Enumeration of nodes for the bi-linear rectangle

For node 2, for instance, the ξ-coordinate is 1 and the η-coordinate is -1. Thus the displacement polynomial becomes:

$$N_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

We also observe that the polynomial has the correct form according to the displacement polynomials shown in Figure 5.4-2.

5.4.3 Development of the stiffness matrix

For the development of the stiffness matrix it is necessary to formalise the displacement assumption. The displacement polynomials derived in the previous sub-section are applied for displacements in both *u*- and *v*-directions. Thus the displacement assumption may be written as follows:

$$\mathbf{u} = \mathbf{N}\mathbf{d} = \begin{bmatrix} \mathbf{N}_0 & \mathbf{0} \\ \mathbf{0} & \mathbf{N}_0 \end{bmatrix} \begin{bmatrix} \mathbf{d}_u \\ \mathbf{d}_v \end{bmatrix} \tag{55}$$

In equation **(55)**, the variables may be described as follows:

$$\mathbf{d}_u = d_{u,i}$$
$$\mathbf{d}_v = d_{v,i}$$
$$\mathbf{N}_0 = N_i$$
$$i \in \{1,2,3,4\}$$

N_i are taken from equation **(54)**, and i represents the relevant node number. The full expression for $\mathbf{N}_0$ is given below:

$$\mathbf{N}_0 = \frac{1}{4}\left[(1-\xi)(1-\eta) \quad (1+\xi)(1-\eta) \quad (1+\xi)(1+\eta) \quad (1-\xi)(1+\eta)\right]$$

Based on the expression for **N**, we may derive the unit displacement strain matrix **B**:

$$\mathbf{B} = \partial\mathbf{N}$$

However, the displacement polynomials are expressed in natural coordinates, so the generalised differential operator ∂ must be re-assessed:

$$\partial = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{56}$$

Combining equations **(53)** and **(56)** yields the following expression for the differential operator:

$$\partial = \begin{bmatrix} \frac{1}{a}\frac{\partial}{\partial \xi} & 0 \\ 0 & \frac{1}{b}\frac{\partial}{\partial \eta} \\ \frac{1}{b}\frac{\partial}{\partial \eta} & \frac{1}{a}\frac{\partial}{\partial \xi} \end{bmatrix}$$

Applying the differential operator to the displacement polynomials yields the **B**-matrix:

$$\mathbf{B} = \partial \mathbf{N} = \begin{bmatrix} \frac{1}{a}\frac{\partial}{\partial \xi} & 0 \\ 0 & \frac{1}{b}\frac{\partial}{\partial \eta} \\ \frac{1}{b}\frac{\partial}{\partial \eta} & \frac{1}{a}\frac{\partial}{\partial \xi} \end{bmatrix} \begin{bmatrix} \mathbf{N}_0 & \mathbf{0} \\ \mathbf{0} & \mathbf{N}_0 \end{bmatrix} = \begin{bmatrix} \frac{1}{a}\mathbf{N}_{0,\xi} & \mathbf{0} \\ \mathbf{0} & \frac{1}{b}\mathbf{N}_{0,\eta} \\ \frac{1}{b}\mathbf{N}_{0,\eta} & \frac{1}{a}\mathbf{N}_{0,\xi} \end{bmatrix}$$

The partial derivatives of the **N**0-matrix are calculated as follows:

$$\mathbf{N}_{0,\xi} = \frac{1}{4}\left[-(1-\eta) \quad (1-\eta) \quad (1+\eta) \quad -(1+\eta) \right]$$

$$\mathbf{N}_{0,\eta} = \frac{1}{4}\left[-(1-\xi) \quad -(1+\xi) \quad (1+\xi) \quad (1-\xi) \right]$$

The remaining part is compute the element stiffness matrix, based on the expression for the **B**-matrix, assuming that the thickness of the element is uniform:

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B} dV = h \int_{-1}^{1} \int_{-1}^{1} \mathbf{B}^T \mathbf{E} \mathbf{B} ab d\xi d\eta$$

The result is an 8x8 stiffness matrix, and each term is fairly lengthy. For instance, assuming plane stress:

$$k_{11} = Eh \frac{a^2(1-v) + 2b^2}{6ab(1-v^2)}$$

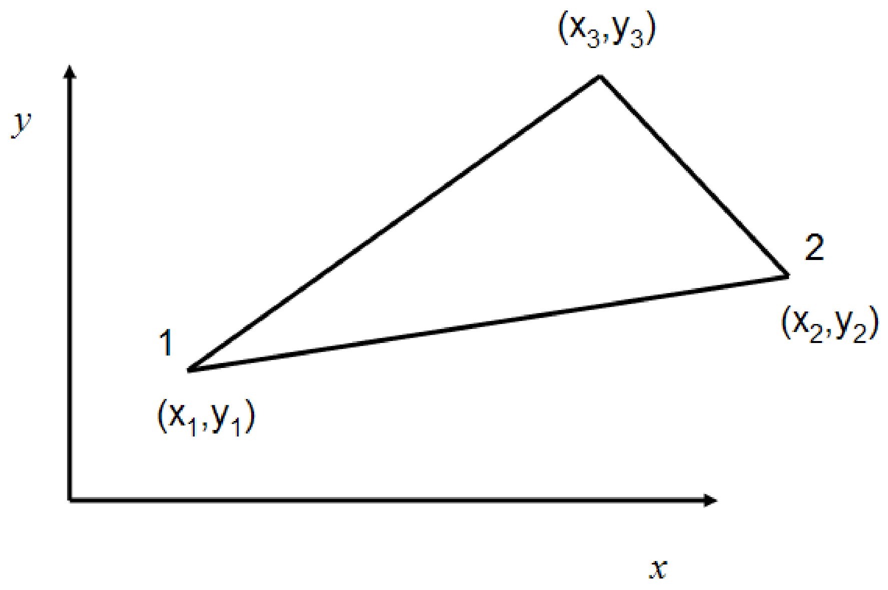
5.5 Exercises

5.5.1 Exercise 1

Use the relation between **N** and $\mathbf{N_q}$ in equation **(46)** to prove that the transformation in equation **(52)** yields the correct relation between **k** and $\mathbf{k_q}$.

5.5.2 Exercise 2

A three node triangular element is shown in the figure below;



Assume that $x_i = \{1.0, 2.1, 1.5\}, y_i = \{1.0, 1.5, 2.5\}$.

a) Determine **N**

b) Evaluate the displacement polynomials in all three nodes, i.e. determine that N_1 has unit value in node 1, and zero in nodes 2 and 3 etc. for all three displacement polynomials.

c) Determine **k**

Assume $u_1 = u_2 = v_2 = 0$, and point lodes in node 3: $F_y = F_x = 1$. Convert the Matlab example in Section 5.3.3 to:

d) Determine the nodal displacements

e) Determine the stress and the strain tensors for the element

f) Convert the ANSYS file in Section 5.3.3 to confirm the Matlab results

g) Using Ansys, what is the relative difference in nodal displacements if you apply 4, 16 or 64 elements instead of just one, compared to the results from d)?

h) Using Ansys, what is the relative difference in maximum stress in the triangle if you apply 4, 16 or 64 elements instead of just one, compared to the results from e)?

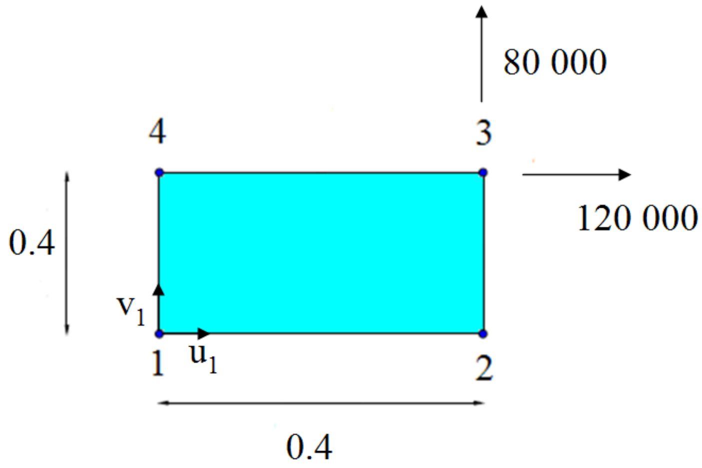
i) Should inaccuracies in displacements theoretically be smaller or larger than inaccuracies in stresses and strains? Does this fit with the findings from g) and h)? What do these findings tell you about the results in Example 5.3.3?

5.5.3 Exercise 3

The 4-node bi-linear rectangle shown in Figure 5.4-4 will be studied in this exercise. Use Matlab to determine an analytical expression for the stiffness matrix. Confirm that k_{11} becomes the same expression as shown in Section 5.4.3.

5.5.4 Exercise 4

(Note – Exercise 3 must be solved first). Based on the stiffness matrix developed in Exercise 3, the problem shown in the Figure below shall be investigated:



The rectangle is fixed in node 1, and in node 2 v_2 is fixed. The material properties are given as follows:

- $E = 207 \cdot 10^9$
- $v = 0.3$

a) Solve for nodal displacements using Matlab, and confirm using ANSYS with a 4-node PLANE42 element.

b) Find the strain vector using Matlab, and determine the maximum principal strain on the domain of the element and confirm against the ANSYS solution.

5.5.5 Exercise 5

For a 4-node bi-linear rectangle, place node 1 at the origin of a Cartesian coordinate system. Let the length be 0.4, and the width be 0.4 for the element.

a) Apply generalised coordinates to determine the stiffness matrix for the element (using Matlab for the differentiations and integrations)

b) Compare the stiffness matrix to the element stiffness matrix determined in Exercise 4, to show that the two different methodologies produce the same element stiffness matrix

6 ASSEMBLY

When implementing solutions for a single element, or systems composed of only a few elements, implementation of boundary conditions and subsequently assembling the global stiffness matrix and global load vector is fairly simple. The systems we have analysed so far in this course indeed are only composed of one or just a few elements. In this Section we shall deduce a formal methodology for the implementation of boundary conditions and the subsequent assembly process. The formal methodology allows us to expand our scope such that we may analyse larger systems of elements.

It should be noted that the methodologies deduced in this Section are not the same as the methods applied in professional finite element software, since the methods deduced here are unfavourable in terms of computational efficiency. From a mathematical perspective, however, the methods are instructive and transparent.

6.1 Background

In Section 3, we found the expression for the total potential energy functional:

$$\Pi(\mathbf{u}) = U - \Omega = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV - \int_V \mathbf{u}^T \mathbf{F} dV - \int_S \mathbf{u}^T \Phi dS$$

The total potential energy functional is defined over the entire volume V of the elastic body, and the tractions act on the closed surface S which circumscribes the volume. For the purpose of the finite element method, we choose to partition the volume V into a set of n elements such that:

$$V = \bigcup_{e=1}^{n} V_e$$

In other words, the volume V may be partitioned into a set of element volumes V_e such that the union of all the element volumes becomes the total volume. An important feature of the partition is that it must satisfy the condition that all the individual element volumes are disjoint, i.e. no part of the volume is included in more than one element of the partition. More precisely:

$$V_i \cap V_j = \emptyset, i \neq j \text{, when } i, j \in \{1, \ldots, n\}$$

Each individual part of the partition is called an element. A subset of the partition includes elements which have a surface boundary included in S. This implies that:

$$S = \bigcup_{e=1}^{n} S_e$$

In the above equation, S_e is zero when the element does not have a surface contact with the outer surface boundary, and when the element has a surface boundary included in S, S_e is the area of that surface boundary.

Since integration is additive we may reformulate the expression for the total potential energy functional:

$$\Pi(\mathbf{u}) = \sum_{e=1}^{n} \frac{1}{2}\int_{V_e} \varepsilon^T \mathbf{E}\varepsilon dV_e - \int_{V_e} \mathbf{u}^T \mathbf{F} dV_e - \int_{S_e} \mathbf{u}^T \Phi dS_e$$

In other words, the integration over the total potential energy functional may be performed individually for each element, and the sum must be equal to the integral over the entire volume.

In Section 4.1, the concept of element displacement vectors and element displacement polynomials was introduced, and introducing these concepts to the present discussion yields:

$$\Pi = \sum_{e=1}^{n} \frac{1}{2} \int_{V_e} \mathbf{d}_e{}^T \mathbf{B}_e{}^T \mathbf{E} \mathbf{B}_e \mathbf{d}_e dV_e - \int_{V_e} \mathbf{d}_e{}^T \mathbf{N}_e{}^T \mathbf{F} dV_e - \int_{S_e} \mathbf{d}_e{}^T \mathbf{N}_e{}^T \Phi dS_e \tag{57}$$

In equation **(57)**, $\mathbf{d}_e$ is the element displacement vector, $\mathbf{N}_e$ is the matrix of element displacement polynomials and $\mathbf{B}_e$ is the element unit displacement strain matrix.

6.2 Topology matrices

Topology matrices are generally introduced in order to augment stiffness matrices and to sort local from global degrees of freedom. Simply put, there exists a matrix $\mathbf{a}_e$ such that:

$$\mathbf{d}_e = \mathbf{a}_e \mathbf{D} \tag{58}$$

In equation **(58)**, **D** is the global displacement vector and **R** is the global load vector. Generally speaking, if an element has *k* degrees of freedom, and the total number of global degrees of freedom is *m*, then $\mathbf{a}_e$ is a *k* x *m* matrix. If the element degrees of freedom correspond to global degrees of freedom (GDOF) according to the following sequence:

$$GDOF_i = \{a_1, a_2, \ldots, a_k\}$$

then $\mathbf{a}_e$ is a matrix of zeros with the exception of $\{k_{1,a1}, k_{2,a2}, \ldots k_{k,ak}\}$ which have the value 1. The generation of $\mathbf{a}_e$ is illustrated by an example below.

6.2.1 Example of a topology matrix

Say we have a 8 DOF system, and an element with 4 DOF. If the element degrees of freedom correspond to the global degrees of freedom 2, 5, 8 and 7, then $\mathbf{a}_e$ is a 4 x 8 matrix and it becomes:

$$\mathbf{a}_e = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix}$$

By direct calculation this can easily be verified:

$$\begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \\ d_7 \\ d_8 \end{bmatrix} = \begin{bmatrix} d_2 \\ d_5 \\ d_8 \\ d_7 \end{bmatrix}$$

6.3 Augmentation of the stiffness matrix

The concept of the topology matrix may be introduced to the general deduction of the stiffness matrix and load vector, by simple alteration of the deduction in Section 4.1. The deduction will be given in part, and the remaining arithmetic is left to the exercises. Inserting for equation **(58)** into **(57)** leaves:

$$\Pi = \sum_{e=1}^{n} \frac{1}{2} \int_{V_e} \mathbf{D}^T \mathbf{a}_e^T \mathbf{B}_e{}^T \mathbf{E} \mathbf{B}_e \mathbf{a}_e \mathbf{D} dV_e - \int_{V_e} \mathbf{D}^T \mathbf{a}_e^T \mathbf{N}_e{}^T \mathbf{F} dV_e - \int_{S_e} \mathbf{D}^T \mathbf{a}_e^T \mathbf{N}_e{}^T \Phi dS_e \tag{59}$$

Performing the variation operation and applying the principle of minimum potential energy results in the following expression:

$$= \delta \mathbf{D}^T \sum_{e=1}^{n} \left(\mathbf{a}_e^T \int_{V_e} \mathbf{B}_e{}^T \mathbf{E} \mathbf{B}_e dV_e \mathbf{a}_e \right) \mathbf{D} = \sum_{e=1}^{n} \left(\mathbf{a}_e^T \int_{V_e} \mathbf{N}_e{}^T \mathbf{F} dV_e + \mathbf{a}_e^T \int_{S_e} \mathbf{N}_e{}^T \Phi dS_e \right) \tag{60}$$

From Section 4.1 we know the following identities:

$$\begin{gathered} \int_{V_e} \mathbf{B}_e{}^T \mathbf{E} \mathbf{B}_e dV_e = \mathbf{k}_e \\ \int_{V_e} \mathbf{N}_e{}^T \mathbf{F} dV_e + \int_{S_e} \mathbf{N}_e{}^T \Phi dS_e = \mathbf{r}_e \end{gathered} \tag{61}$$

Combining equations **(60)** and **(61)** yields the expression for augmentation of local stiffness matrices and summation into a global stiffness matrix as well as an expression for augmentation of element load vectors and summation into a global load vector:

$$\begin{gathered} \mathbf{K} = \sum_{e=1}^{n} \mathbf{a}_e^T \mathbf{k}_e \mathbf{a}_e \\ \mathbf{R} = \sum_{e=1}^{n} \mathbf{a}_e^T \mathbf{r}_e + \sum_{k} \mathbf{R}_k \end{gathered} \tag{62}$$

In equation **(62)**, the global force vector has been added to $\sum_k \mathbf{R}_k$. $\mathbf{R}_k$ are nodal loads/moments.

6.3.1 Extended example – Revisit of the triangular stiffener

In example 5.3.3, a triangular stiffener exposed to a point load was studied. In this example, the same stiffener will be analysed, only this time with 4 membrane elements rather than 1. The triangle is shown again, in a meshed state:

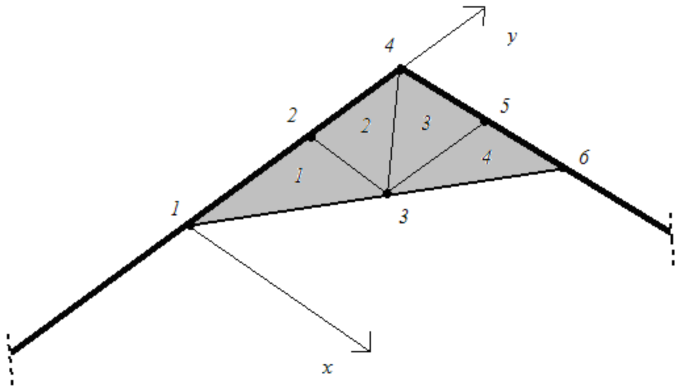


Figure 6.3-1 – Part of a larger structure, containing a stiffener

In Figure 6.3-1, the nodes are given outside of the structure and the element enumeration is shown in the interior of each element. Thus, the meshed structure has 6 nodes and 4 elements.

The structure is fixed in nodes 4 and 5 and 6, while the arm along the y-axis is flexible. The steel arms are conservatively considered to be so flexible that the stiffener absorbs all the loading.

The loading along the y-axis shall be simplified to include only point loads in node 1. F_x = 120 kN, F_y = 80 kN. The specifications for the stiffener are listed as follows:

- E = 207 GPa, v = 0.3
- $h = 6\ mm$
- $x_i = \{0,0,0.2,0.0,0.2,0.4\}, y_i = \{0.0,0.2,0.2,0.4,0.4,0.4\}$.
- Plane stress shall be assumed
- Maximum allowable local principal stress is 420 MPa

The example shall determine if the stress in the triangular stiffener is within the allowable limit.

6.3.1.1 Element stiffness matrices

Generalised coordinates have been applied.

Element 1

The global and local nodal enumerations have been chosen as equal.

$$\mathbf{A}_0 = \begin{bmatrix} 1 & 0 & 0 \\ 1 & 0 & 0.2 \\ 1 & 0.2 & 0.2 \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} \mathbf{N}_{q0,x}\mathbf{A}_0^{-1} & \mathbf{0} \\ \mathbf{0} & \mathbf{N}_{q0,y}\mathbf{A}_0^{-1} \\ \mathbf{N}_{q0,y}\mathbf{A}_0^{-1} & \mathbf{N}_{q0,x}\mathbf{A}_0^{-1} \end{bmatrix}$$

$$\mathbf{k}_1 = \frac{h\left|\det(\mathbf{A}_0)\right|}{2}\mathbf{B}^T\mathbf{E}\mathbf{B}$$

<u>Element 2</u>

Element nodes 1, 2 and 3 have been chosen as global nodes 2, 3 and 4 respectively.

$$\mathbf{A}_0 = \begin{bmatrix} 1 & 0 & 0.2 \\ 1 & 0.2 & 0.2 \\ 1 & 0 & 0.4 \end{bmatrix}$$

$$\mathbf{k}_2 = \frac{h\left|\det(\mathbf{A}_0)\right|}{2}\mathbf{B}^T\mathbf{E}\mathbf{B}$$

<u>Element 3</u>

Element nodes 1, 2 and 3 have been chosen as global nodes 3, 4 and 5 respectively

$$\mathbf{A}_0 = \begin{bmatrix} 1 & 0 & 0.2 \\ 1 & 0.2 & 0.2 \\ 1 & 0 & 0.4 \end{bmatrix}$$

$$\mathbf{k}_3 = \frac{h\left|\det(\mathbf{A}_0)\right|}{2}\mathbf{B}^T\mathbf{E}\mathbf{B}$$

<u>Element 4</u>

Element nodes 1, 2 and 3 have been chosen as global nodes 3, 5 and 6 respectively

$$\mathbf{A}_0 = \begin{bmatrix} 1 & 0.2 & 0.2 \\ 1 & 0.2 & 0.4 \\ 1 & 0.4 & 0.4 \end{bmatrix}$$

$$\mathbf{k}_4 = \frac{h\left|\det(\mathbf{A}_0)\right|}{2}\mathbf{B}^T\mathbf{E}\mathbf{B}$$

6.3.1.2 Boundary conditions

The global displacement vector is chosen as:

$$\mathbf{D}_g^T = \begin{bmatrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & u_4 & v_4 & u_5 & v_5 & u_6 & v_6 \end{bmatrix}$$

Applying a fixed condition in global nodes 4, 5 and 6 reduces the global displacement vector to:

$$\mathbf{D}^T = \begin{bmatrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{bmatrix}$$

6.3.1.3 Assembly

The topology matrices are determined for each individual element:

<u>Element 1</u>

6 element degrees of freedom shall match with 6 global degrees of freedom. The topology matrix becomes:

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

Since the **a**-matrices are large, and full of zeroes, they will not be shown in full for the remainder of the example. Instead, the entries of the relevant **a**-matrix that are non-zero, i.e. 1, will be given on index form. So for **a**1, the following index pairs yields entries of 1:

$$\{(1,1) \quad (2,3) \quad (3,5) \quad (4,2) \quad (5,4) \quad (6,6)\}$$

<u>Element 2</u>

The following index pairs yields entries of 1:

$$\{(1,3) \quad (2,5) \quad (3,7) \quad (4,4) \quad (5,6) \quad (6,8)\}$$

<u>Element 3</u>

The following index pairs yields entries of 1:

$$\{(1,5) \quad (2,7) \quad (3,9) \quad (4,6) \quad (5,8) \quad (6,10)\}$$

<u>Element 4</u>

The following index pairs yields entries of 1:

$$\{(1,5) \quad (2,9) \quad (3,11) \quad (4,6) \quad (5,10) \quad (6,12)\}$$

<u>Summing up</u>

As all the element stiffness matrices are determined and augmented, the global stiffness matrix may be assembled:

$$\mathbf{K} = \mathbf{a}_1^T\mathbf{k}_1\mathbf{a}_1 + \mathbf{a}_2^T\mathbf{k}_2\mathbf{a}_2 + \mathbf{a}_3^T\mathbf{k}_3\mathbf{a}_3 + \mathbf{a}_4^T\mathbf{k}_4\mathbf{a}_4$$

Applying the boundary condition simply consists of eliminating rows and columns 4, 5 and 6. The final result is given below:

$$\mathbf{K} = 10^9 \begin{bmatrix} 0.2388 & 0 & -0.2388 & 0.2388 & 0 & -0.2388 \\ 0 & 0.6824 & 0.2047 & -0.6824 & -0.2047 & 0 \\ -0.2388 & 0.2047 & 1.8425 & 0 & -1.3648 & 0 \\ 0.2388 & -0.6824 & 0 & 1.8425 & 0 & -0.4777 \\ 0 & -0.2047 & -1.3648 & 0 & 1.8425 & 0 \\ -0.2388 & 0 & 0 & -0.4777 & 0 & 1.8425 \end{bmatrix}$$

There is little work in establishing the load vector, since we only have point loads:

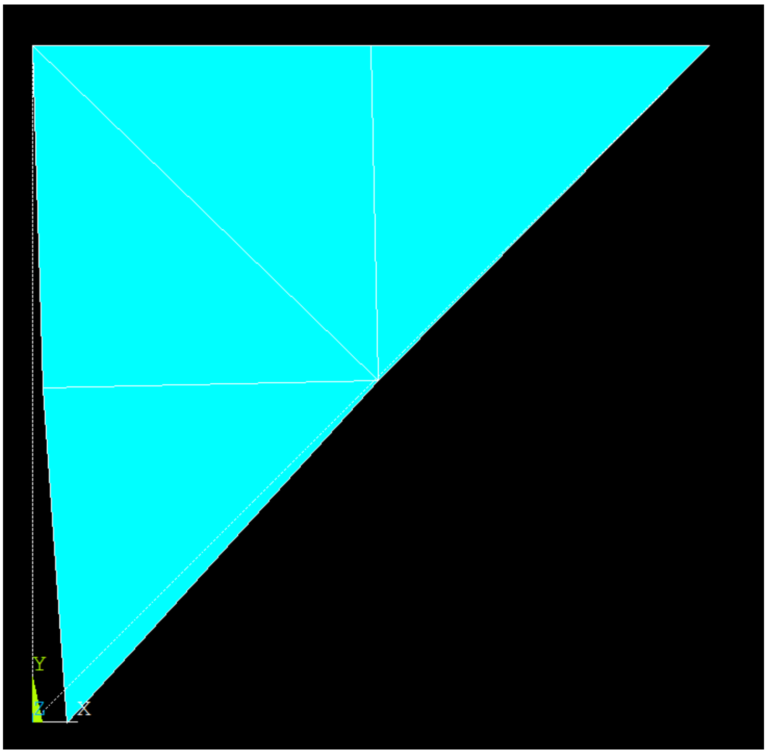
$$\mathbf{R}^T = [120000 \quad 80000 \quad 0 \quad 0 \quad 0 \quad 0]$$

6.3.1.4 Solution

Solving for the displacements yields the following displacement vector:

$$\mathbf{D} = \mathbf{K}^{-1}\mathbf{R} = 10^{-3}[1.0015 \quad -0.0146 \quad 0.2886 \quad -0.1089 \quad 0.2122 \quad 0.1016]$$

With the below ANSYS code (in Section 6.3.1.6) , this displacement vector is replicated exactly. The displacement vector yields the following displaced structure (depicted in ANSYS):



From the figure it is observed that the linear displacement along the y-axis in Example 5.3.3 is replaced with a non-linear one, and that the larger part of the displacements are found in node 1. Compared to Example 5.3.3, u_1 in node 1 is also doubled, i.e. the system of four elements is significantly less stiff than the system of 1 element.

Calculation of stresses

6.3.1.5 Matlab code

```
% Establish constants
Emod=207*10^9;
v=0.3;
h=0.006;
Fx=120000;
Fy=80000;
% Plane stress material law
E=Emod/(1-v^2)*[1 v 0; v 1 0; 0 0 (1-v)/2];

%-------------------------------------------
```

```
% Calculate stiffness matrix
%--------------------------------------------

%--------------------------------------------
% Element 1
%--------------------------------------------

% A-matrix
A0=[1 0 0; 1 0 0.2; 1 0.2 0.2];
A(1:3,1:3)=A0;
A(4:6,4:6)=A0;
% Determine inverse of A and A0
Ai=inv(A);
A0i=inv(A0);
% Differentiated Nq-vectors
Nqx=[0 1 0];
Nqy=[0 0 1];
% Determine Bq-matrix
Bq=[Nqx 0 0 0; 0 0 0 Nqy; Nqy Nqx];
% kq-matrix
kq=0.5*abs(det(A0))*h*Bq'*E*Bq;
% Transform kq to k
k1=Ai'*kq*Ai;
% Calculate B-matrix for later use with
% Stress calculations
NqxA0i=Nqx*A0i;
NqyA0i=Nqy*A0i;
B1=[NqxA0i 0 0 0; 0 0 0 NqyA0i; NqyA0i NqxA0i];

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

% A-matrix
A0=[1 0 0.2; 1 0.2 0.2; 1 0 0.4];
A(1:3,1:3)=A0;
A(4:6,4:6)=A0;
% Determine inverse of A and A0
Ai=inv(A);
A0i=inv(A0);
% kq-matrix
kq=0.5*abs(det(A0))*h*Bq'*E*Bq;
% Transform kq to k
k2=Ai'*kq*Ai;
% Calculate B-matrix for later use with
% Stress calculations
NqxA0i=Nqx*A0i;
NqyA0i=Nqy*A0i;
B2=[NqxA0i 0 0 0; 0 0 0 NqyA0i; NqyA0i NqxA0i];

%--------------------------------------------
% Element 3
%--------------------------------------------

% A-matrix
A0=[1 0.2 0.2; 1 0 0.4; 1 0.2 0.4];
A(1:3,1:3)=A0;
A(4:6,4:6)=A0;
```

```
% Determine inverse of A and A0
Ai=inv(A);
A0i=inv(A0);
% kq-matrix
kq=0.5*abs(det(A0))*h*Bq'*E*Bq;
% Transform kq to k
k3=Ai'*kq*Ai;
% Calculate B-matrix for later use with
% Stress calculations
NqxA0i=Nqx*A0i;
NqyA0i=Nqy*A0i;
B3=[NqxA0i 0 0 0; 0 0 0 NqyA0i; NqyA0i NqxA0i];

%--------------------------------------------
% Element 4
%--------------------------------------------

% A-matrix
A0=[1 0.2 0.2; 1 0.2 0.4; 1 0.4 0.4];
A(1:3,1:3)=A0;
A(4:6,4:6)=A0;
% Determine inverse of A and A0
Ai=inv(A);
A0i=inv(A0);
% kq-matrix
kq=0.5*abs(det(A0))*h*Bq'*E*Bq;
% Transform kq to k
k4=Ai'*kq*Ai;
% Calculate B-matrix for later use with
% Stress calculations
NqxA0i=Nqx*A0i;
NqyA0i=Nqy*A0i;
B4=[NqxA0i 0 0 0; 0 0 0 NqyA0i; NqyA0i NqxA0i];

%-------------------------------------------
% Augment the element stiffness matrices
%-------------------------------------------

a1=zeros(6,12);
a2=a1;
a3=a1;
a4=a1;

a1(1,1)=1;
a1(2,3)=1;
a1(3,5)=1;
a1(4,2)=1;
a1(5,4)=1;
a1(6,6)=1;

a2(1,3)=1;
a2(2,5)=1;
a2(3,7)=1;
a2(4,4)=1;
a2(5,6)=1;
a2(6,8)=1;

a3(1,5)=1;
```

```
a3(2,7)=1;
a3(3,9)=1;
a3(4,6)=1;
a3(5,8)=1;
a3(5,10)=1;

a4(1,5)=1;
a3(2,9)=1;
a4(3,11)=1;
a4(4,6)=1;
a4(5,10)=1;
a4(6,12)=1;

k1a=a1'*k1*a1;
k2a=a2'*k2*a2;
k3a=a3'*k3*a3;
k4a=a4'*k4*a4;

%-------------------------------------------
% Calculate the global stiffness matrix
%-------------------------------------------

Kg=k1a+k2a+k3a+k4a;

%-------------------------------------------
% Implement boundary conditions
%-------------------------------------------

K=Kg(1:6,1:6);

%-------------------------------------------
% Determine the load vector
%-------------------------------------------

R=[120000 80000 0 0 0 0]';

%-------------------------------------------
% Solve for displacements
%-------------------------------------------

D=inv(K)*R
```

6.3.1.6 ANSYS code

```
/BATCH, LIST
/FILENAME, TEST
/TITLE, Analyse av triangulær stiver
/PREP7
! Element type PLANE42
! Keyopts 1=0 (global coordinate system), 2=1 (no extra displacement shapes), 3=3 (Plane stress
! with thickness input), 5=0, 6=0 (No refinement on stress solution).
ET,1,42,0,1,3,,0,0
! Real constant set 1, is the thickness of the plate
```

```
R,1,0.006
! Assigning the real constants to element type 1
REAL,1
! Material properties
MP,EX,1,207E9
MP,EY,1,207E9
MP,NUXY,1,0.3
! Definition of nodes
N,1,0,0,0
N,2,0.0,0.2,0
N,3,0.2,0.2,0
N,4,0,0.4,0
N,5,0.2,0.4,0
N,6,0.4,0.4,0
! Defining an element with nodes 1, 2 and 3
E,1,2,3
E,2,3,4
E,3,4,5
E,3,5,6
! Boundary conditions
D,4,UX,0
D,4,UY,0
D,5,UX,0
D,5,UY,0
D,6,UX,0
D,6,UY,0
! Application of loads
F,1,FX,120000
F,1,FY,80000
! Finish preprocessor
FINISH
! Solution processor
/SOLU
SOLVE
```

6.4 Exercises

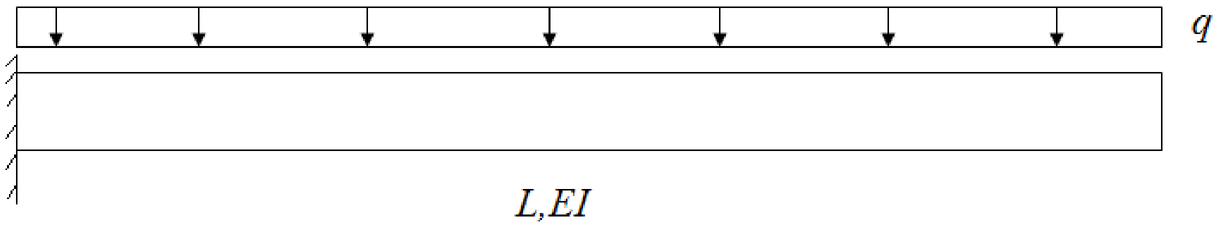
The exercises in this Section constitute mandatory assignment no. 3.

6.4.1 Exercise 1

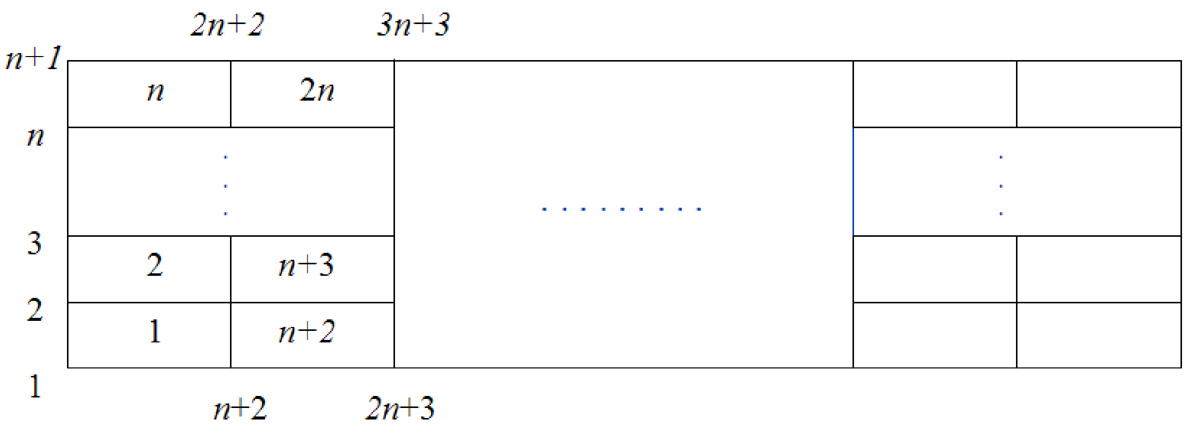
Complete the proof in Section 6.3 to prove equation **(62)**

6.4.2 Exercise 2

In this exercise we shall study a cantilever beam with membrane elements.



The cantilever is loaded with a constant distributed load q=40 000, $E = 207 \cdot 10^6$, v = 0.3 and L = 5. The cross-section is rectangular. The height is 0.5 and the thickness (into the paper plane) is 1. The element mesh and nodal enumeration shall be used as shown in the figure below:

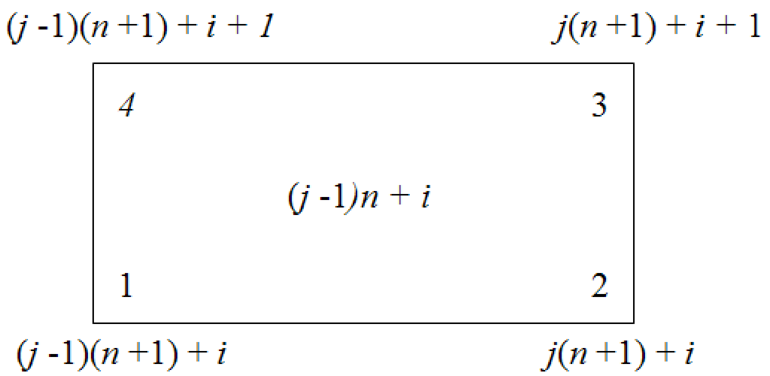


a) In exercise 5.3.3 the stiffness matrix for a bi-linear rectangular membrane element was developed. The element mesh shown in the figure above has n elements over the height of the beam and m elements along the length of the beam. Find the stiffness matrix for the elements in the figure above.

b) The number of nodes over the height is n +1 and over the length the number of nodes is m +1. Complete the figure above with element and nodal enumeration for the right side of the beam.

c) How many nodes are there in the system? How many total global degrees of freedom are there before we impose boundary conditions?

d) If the index i runs from 1 to n and the index j runs from 1 to m, the position ij is element positioned at column i and row j in the matrix of elements which constitutes our mesh. Show that the element at position ij has element and node numbers as given in the figure below:



The local node numbers for the element is given on the interior of the element, i.e. nodes 1 through 4.

e) What is the dimension of the topology matrix **a** for the elements used to model the cantilever?

f) Find the entries of the topology matrix, for the element at position ij, which have the value 1.

g) Determine the load vector for the elements which have a boundary to the distributed load.

h) Use the intermediate results gained from exercises a)-g) to complete the following Matlab script. Note – input marked with capital letters must be inserted by you.

```
n=NUMBER OF ELEMENTS OVER HEIGHT;
% Aspect ratio of 1:2 is used, i.e. the height of the elements is half the
% length.
m=n*5;

E=207000000;
v=0.3;
L=5;
h=0.5
t=1;
b=INSERTED BY USER;
a=INSERTED BY USER;

k_elem=FROM EXERCISE 5.5.3

k=zeros(8,8);
K_glob=zeros(NUMBER OF GDOF, NUMBER OF GDOF)

for i=1:n
    for j=1:m
```

```
        a=zeros(DIMENSION OF a-matrix);
        a(1,2*((j-1)*(n+1)+i)-1)=1;
        .
        . [INSERT REMAINING ENTRIES OF a-MATRIX WITH VALUE 1
        .

        k=TRANSFORM WITH a-MATRIX;

        K_glob=K_glob+k;

    end
end

% IMPOSE BOUNDARY CONDIITON

K=K_glob(RELEVANT PART OF K_glob);

% DETERMINE LOAD VECTOR

.
.
.

% INVERT AND SOLVE

.
.
.
```

i) Using Matlab, model the cantilever with only 1 element, and compare the solution to ANSYS and also an analytical solution using Navier beam theory.

j) Using the above script, use 3 elements over the height and 15 elements over the length. Solve for the displacements and compare to ANSYS (with the below ansys script) for the maximum vertical displacement. How does this compare to analytical beam theory?

/BATCH, LIST

/FILENAME, TEST

/TITLE, Analysis of cantilever beam

/PREP7

! Element type PLANE42

! Keyopts 1=0 (global coordinate system), 2=1 (no extra displacement shapes), 3=3 (Plane stress

! with thickness input), 5=0, 6=0 (No refinement on stress solution).

ET,1,42,0,1,3,,0,0

! Real constant set 1, is the thickness of the plate

R,1,1

```
! Assigning the real constants to element type 1
REAL,1
! Material properties
MP,EX,1,207E6
MP,EY,1,207E6
MP,NUXY,1,0.3
RECTNG,0,5,0,0.5
! MESHING - For simplicity we just choose a default mesh from ansys
AMESH,all
! Choose all nodes at location x = 0
NSEL,S,LOC,X,0
! Apply boundary conditions for these nodes
D,all,UX,0
D,all,UY,0
! Choose all nodes, so they are available for load application
NSEL,all
! Apply load
F,17,FY,-10000
! Exit pre-processor and solve
FINISH
/SOL
ALLSEL
SOLVE
FINISH
```

k) Experiment with increasing the number of elements until the result converges. How many elements are necessary for 3 significant digits of convergence on the displacements?

l) For the converged solution, which positions in the mesh should be compared to beam theory in terms of vertical displacements, and furthermore, how does this solution compare to beam theory? Discuss potential differences and explain why Navier beam theory and membrane theory should result in different displacements.

7 INTERPOLATION AND CHOICE OF DISPLACEMENT FUNCTIONS

In this Section we shall introduce a set of interpolation techniques and how to develop suitable displacement assumption polynomials in finite element contexts.

8 BOUNDARY CONDITIONS AND KINEMATIC COMPATIBILITY

9 ISO-PARAMETRIC ELEMENTS AND GEOMETRIC MAPPING

10 STATIC CONDENSATION

11 REFERENCES

/1/