LINEAR HOMOGENEOUS TRUSS

1 Truss structures

A truss is a mechanical element whose dimension in one direction – the truss axis – is much larger than the dimensions in each direction perpendicular to the axis. A truss structure is an assembly of trusses, which are connected mutually and to the suroundings with hinges. The truss can transfer only axial forces along its axis, so bending is not possible, and the axis must be and remain straight.

In this chapter, we first consider small elongation and rotation of a truss. The material behaves linearly elastic and the resulting equilibrium equation is linear. The finite element method is used to model truss structures and to solve the resulting set of equilibrium equations.

Large elongations and rotations lead to a set of nonlinear equations. Moreover, the material behavior is likely to be nonlinear as well. Solution of the set of equations must be done iteratively. Implementation of various material models in the finite element software is the subject of the next chapter.

1.1 Homogeneous truss

We consider a truss to be oriented with its axis along the global x-axis. Its undeformed length is l_0. The undeformed cross-sectional area has a uniform value A_0. It is assumed that the material of the truss is isotropic and homogeneous.

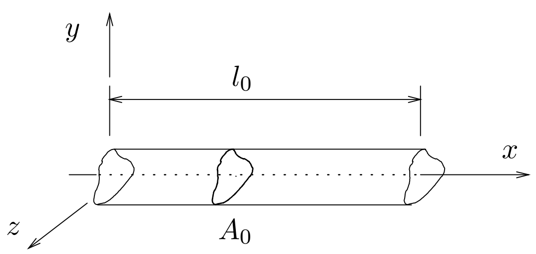


Fig. 1.1 : *Homogeneous truss*

1.1.1 Elongation and contraction

In the deformed state the length of the truss is l and its cross-sectional area is A. The elongation is described by the axial elongation factor λ. The change in cross-sectional area is described by the contraction μ. It is assumed that the load, which provokes the deformation, is such that the deformation is homogeneous. This means that λ and μ are the same in each point of the truss. The volume change is described by the volume ratio J.

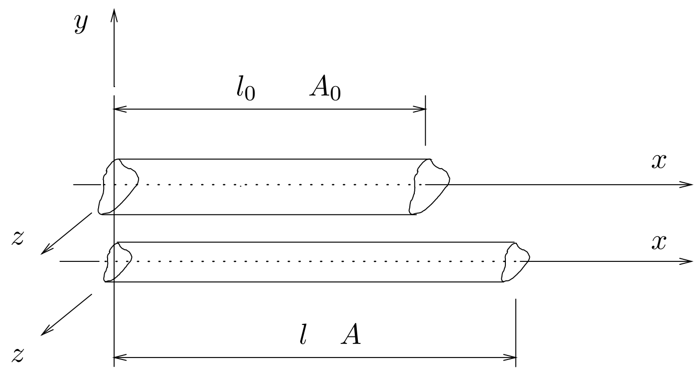


Fig. 1.2 : *Deformation of a homogeneous truss*

elongation factor	$\lambda = \dfrac{l}{l_0} = \dfrac{l_0 + \Delta l}{l_0} = 1 + \dfrac{\Delta l}{l_0}$
contraction	$\mu = \sqrt{\dfrac{A}{A_0}}$
volume change	$J = \dfrac{lA}{l_0 A_0} = \lambda \mu^2$

1.1.2 Stress

The deformation of the truss is caused by an external axial force N. In each cross-section (x) of the truss, an internal axial force $N(x)$ exists. With no volume load, the cross-sectional load will be the same in each cross-section. If a volume load is applied, this is not the case, but we will not consider such loading here.

The axial load is such that it causes only axial deformation and no bending. In the absence of a volume load the deformation will be homogeneous.

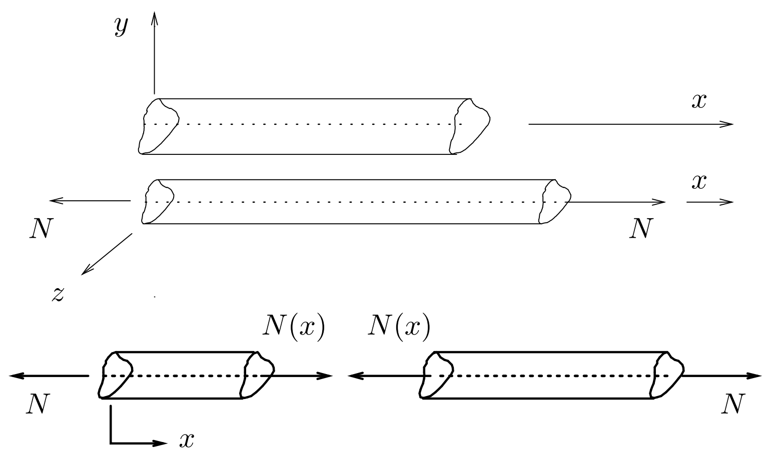


Fig. 1.3 : *Axial loading of a homogeneous truss*

The cross-sectional force is the result of the axial cross-sectional stress. For a homogeneous material with no volume loads, the stress is uniform over the cross-section. This leads to the definition of *true stress*, being the axial force divided by the deformed (= real) cross-sectional area. In many (engineering) applications the *engineering* or *nominal stress* is used, defined as the ratio of the axial force and the undeformed cross-sectional area. True stress and engineering stress, are related by the contraction μ.

In literature a truss is sometimes called a *tie* when it carries a tensile force and a *strut* when it is loaded in compression.

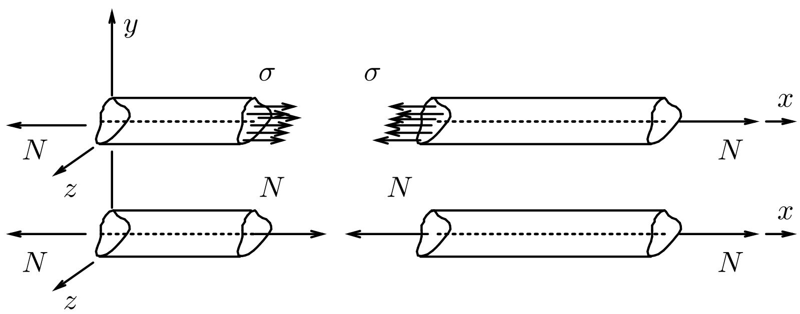


Fig. 1.4 : *Cross-sectional stress in an axially loaded homogeneous truss*

axial stress	$\sigma = \sigma(y,z)$
cross-sectional force	$N(x) = N = \int_A \sigma(y,z)\, dA$
stress uniform in cross-section	$N = \int_A \sigma \, dA = \sigma A$
true stress	$\sigma = \dfrac{N}{A}$
engineering stress	$\sigma_n = \dfrac{N}{A_0}$
relation	$\sigma = \dfrac{N}{A} = \dfrac{A_0}{A}\dfrac{N}{A_0} = \dfrac{1}{\mu^2}\,\sigma_n$

1.2 Linear deformation

When the elongation of the truss is very small, the contraction is even smaller so that the deformed cross-sectional area can be taken to be equal to the initial cross-sectional area. Consequently there is no difference between the true stress and the engineering stress.

1.2.1 Linear strain

Elongation is generally described by the strain ε. For small elongation and rotation, the linear strain is used. For the elongation, this strain is related to the elongation facor λ and for the contraction to the contraction μ.

linear strain $\qquad \varepsilon = \varepsilon_l = \lambda - 1$

contractive linear strain $\qquad \varepsilon_d = \mu - 1$

1.2.2 Linear elastic behavior

The linear elastic material behavior is characterized by two matarial constants: Young's modulus and Poisson's ratio. Young's modulus relates the axial stress to the axial strain. Poisson's ratio relates the contractive strain to the axial strain. For most materials Poisson's ratio is about 0.3. For small elongations this value is constant. For small deformation the volume change factor J can be expressed in the linear strain. For incompressible material $J = 1$ implying $\nu = \frac{1}{2}$.

axial stress $\sim$ strain $\qquad \sigma = E\varepsilon$

contraction strain $\qquad \varepsilon = \lambda - 1 \quad \rightarrow \quad \varepsilon_d = \mu - 1 = -\nu\varepsilon = -\nu(\lambda - 1)$

volume change $\qquad J = (\varepsilon + 1)(-\nu\varepsilon + 1)^2 \approx \varepsilon(1 - 2\nu) + 1$

The table lists values of Young's modulus and Poisson's ratio for some materials.

material	E [GPa]	ν [-]	material	E [GPa]	ν [-]
Aluminum	69 - 79	0.31 - 0.34	Copper	105 - 150	0.33 - 0.35
Cast iron	105 - 150	0.21 - 0.30	Steel	200	0.33
Stainless steel	190 - 200	0.28	Lead	14	0.43
Magnesium	41 - 45	0.29 - 0.35	Nickel	180 - 215	0.31
Titanium	80 - 130	0.31 - 0.34	Tungsten	400	0.27
Diamond	820 - 1050	-	Graphite	240 - 390	-
Glass	70 - 80	0.24	Epoxy	3.5 - 17	0.34
Nylon	1.4 - 2.8	0.32 - 0.40	Rubber	0.01 - 0.1	0.5

1.2.3 Equilibrium

We consider a truss with length l_0 and cross-sectional area A_0 with its axis along the global x-axis. One end ($x = 0$) is fixed and the other ($x = l_0$) can be displaced in x-direction only. The elongation of the truss equals this displacement u. The displacement is caused by an *external axial force* f_e. In the deformed state the length of the truss is $l = l_0 + u$ and its cross-sectional area is A. The material of the truss is homogeneous.

When the external axial force f_e is prescribed, the elongation $\Delta l = u$ of the truss can be determined by solving the equilibrium equation in point P, which states that the internal force must be equal to the external force. The *internal force* f_i is a function of the elongation, a relation which is determined by the material behavior. It represents the resistance of the truss against elongation.

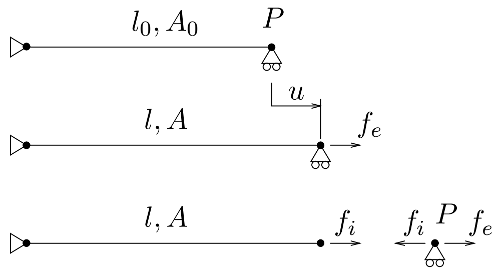


Fig. 1.5 : *Equilibrium of external and internal axial force*

external force	f_e
internal force	$f_i = f_i(u)$
equilibrium of point P	$f_i(u) = f_e$

When the deformation (= elongation) is very small, there is virtually no difference between the undeformed and the deformed geometry. Such deformation is referred to as being *geometrically linear*. The true axial stress $\sigma = N/A$ approximately equals the engineering stress $\sigma_n = N/A_0$, where N is the axial force.

When, moreover, the material behavior is not influenced by the deformation, as is the case for linear elastic behavior, – this is referred to as *physical linearity* – the total deformation is linear and the internal force f_i can be linearly related to the displacement u.

The equilibrium equation can be solved directly, yielding the displacement u.

1.2.4 Solution procedure

Because the relation between the external force f_e and the axial displacement u is linear, the latter can be solved directly from the equilibrium equation $f_i = f_e$, yielding the exact solution $u_{exact} = u_s$. The stiffness K of the truss depends on the Young's modulus E and on the initial geometry (A_0 and l_0).

$$f_i = \sigma_n A_0 = E\varepsilon A_0 = \frac{EA_0}{l_0}\, u = Ku$$

$$f_i = f_e \quad \rightarrow \quad Ku = f_e \quad \rightarrow \quad u = u_s = \frac{f_e}{K} = \frac{l_0}{EA_0}\, f_e$$

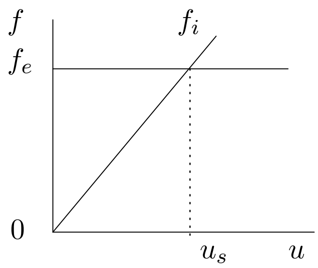


Fig. 1.6 : *Solution of linear equilibrium equation*

Proportionality and superposition

Two important characteristics hold for linear problems :

- the deformation is *proportional* to the load : when the external force f_e is multiplied by a factor, say α, the elongation u is also multiplied by α.
- *superposition* holds : when we determine the elongation u_1 and u_2 for two separate forces, f_{e1} and f_{e2} respectively, the elongation for the combined loading $f_{e1} + f_{e2}$ is the sum of the separate elongations : $u_1 + u_2$.

1.3 Finite element method for linear truss

When a truss structure is loaded by external forces or prescribed displacements, its deformation can sometimes be calculated analytically, especially when the structure is statically determinate. When the structure is statically indeterminate, this is only possible for very simple cases. Practical problems can be solved numerically, using the *finite element method*.

When the trusses in the structure show small elongation and rotation, and when moreover their material behavior is linearly elastic, the whole problem is linear and the finite element method can be explained rather straightforwardly.

In the following we restrict ourselves to two-dimensional structures.

Truss element

A truss element e with two nodal points is oriented with its axis in the 1-direction of a two-dimensional coordinate system. Both nodes move in this direction – being by definition positive – , leading to an elongation of the truss : its initial length l_0^e becomes l^e.

This elongation is resisted by the material of the truss, leading to reaction forces in both nodes : the *internal nodal point forces*, again defined to be positive in the positive 1-direction. In absence of distributed axial load the axial force N in the truss is constant and a function of the elongation.

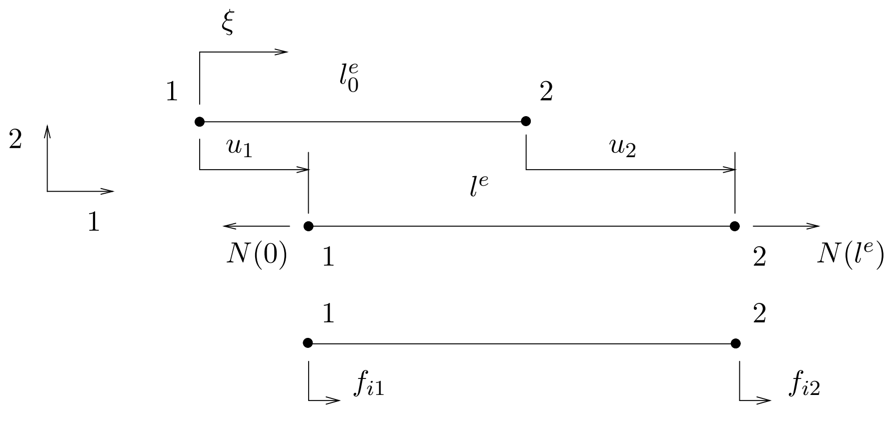


Fig. 1.7 : *One-dimensional truss element*

$$\underset{\sim}{u}^e = \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} u(0) \\ u(l^e) \end{bmatrix}$$

$$\underset{\sim}{f}_i^e = \begin{bmatrix} f_{i1} \\ f_{i2} \end{bmatrix} = \begin{bmatrix} -N(0) \\ N(l^e) \end{bmatrix} = \begin{bmatrix} -k(u_2 - u_1) \\ k(u_2 - u_1) \end{bmatrix} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

In the initial situation an angle α_0 may exist between the axis of a two-dimensional truss element and the 1-direction of the coordinate system. The displacements and forces of/in the nodal points have two components, and are defined positive in the positive coordinate directions. Due to the deformation, the current angle of the axis is α. For small deformations and rotations we have $\alpha \approx \alpha_0$.

The internal force components can be expressed in the axial force N and the *cosine* and *sine* of the angle α. For a linear element the nodal forces $\underset{\sim}{f}_i^e$ can be related to the elongation, expressed in the nodal displacements in the direction of the element axis, denoted as u_i^L, which are related to the displacement components of the nodal points $\underset{\sim}{u}^e$. This relation is expressed by the *element stiffness matrix* $\underline{K}^e$.

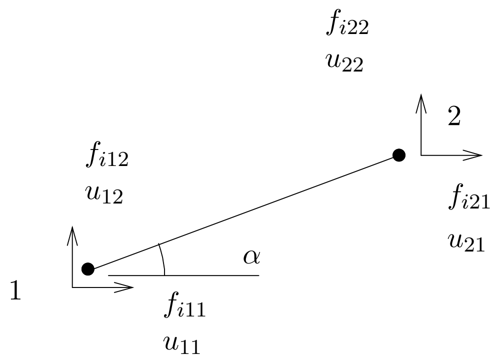


Fig. 1.8 : *Two-dimensional truss element*

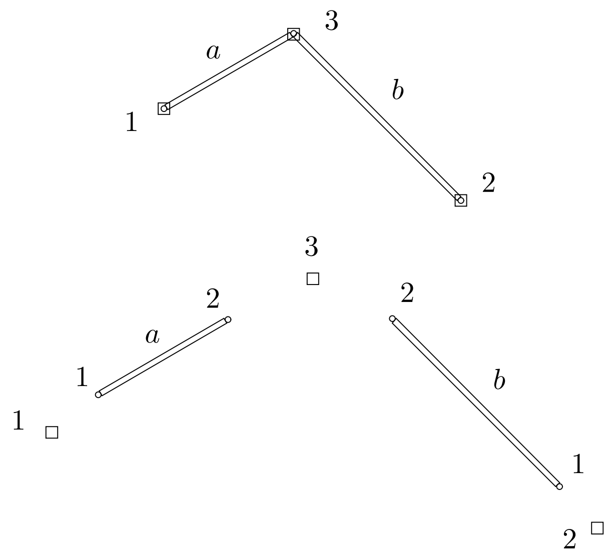
$$\begin{bmatrix} f_{i11} \\ f_{i12} \\ f_{i21} \\ f_{i22} \end{bmatrix} = \begin{bmatrix} cf_{i1}^L \\ sf_{i1}^L \\ cf_{i2}^L \\ sf_{i2}^L \end{bmatrix} = k(u_2^L - u_1^L) \begin{bmatrix} -c \\ -s \\ c \\ s \end{bmatrix} = k(u_{21}c + u_{22}s - u_{11}c - u_{12}s) \begin{bmatrix} -c \\ -s \\ c \\ s \end{bmatrix}$$

$$= k \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{bmatrix} u_{11} \\ u_{12} \\ u_{21} \\ u_{22} \end{bmatrix} = \underline{K}^e \, \underset{\sim}{u}^e \qquad \begin{cases} c = \cos(\alpha) \\ s = \sin(\alpha) \end{cases}$$

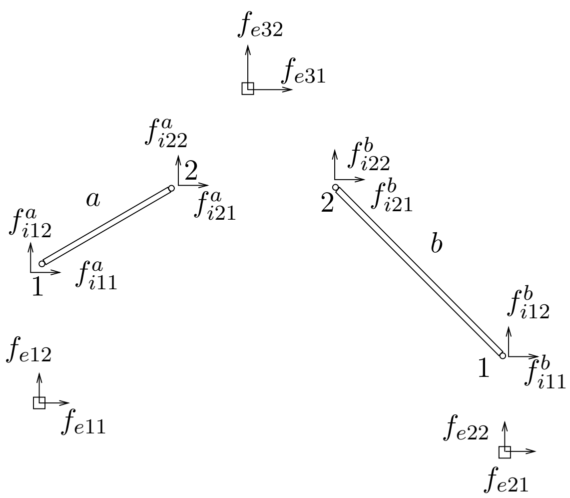
Assembling

For the truss structure, the internal nodal forces are stored in a column $\underset{\sim}{f}_i$ and are related to the nodal displacement components in the column $\underset{\sim}{u}$. This relation is expressed by the *structural* or *global stiffness matrix* $\underline{K}$. The contributions of the individual element stiffnesses to the structural stiffness are added in the assembling procedure.

The example shows two truss elements connected in one system node.



In each system node the internal nodal forces of all the connected elements must be added.



$$\begin{bmatrix} f_{e11} \\ f_{e12} \\ f_{e21} \\ f_{e22} \\ f_{e31} \\ f_{e32} \end{bmatrix} = \begin{bmatrix} f_{i11} \\ f_{i12} \\ f_{i21} \\ f_{i22} \\ f_{i31} \\ f_{i32} \end{bmatrix} = \begin{bmatrix} f^a_{i11} \\ f^a_{i12} \\ f^b_{i11} \\ f^b_{i12} \\ f^a_{i21} + f^b_{i21} \\ f^a_{i22} + f^b_{i22} \end{bmatrix} = \begin{bmatrix} f^a_{i11} \\ f^a_{i12} \\ 0 \\ 0 \\ f^a_{i21} \\ f^b_{i22} \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ f^b_{i11} \\ f^b_{i12} \\ f^b_{i21} \\ f^b_{i22} \end{bmatrix}$$

The displacements of element nodes which are connected to one and the same system node, must be equal to assure continuity.

$$\begin{bmatrix} f_{i11} \\ f_{i12} \\ f_{i21} \\ f_{i22} \\ f_{i31} \\ f_{i32} \end{bmatrix} = \begin{bmatrix} k^a c^2 & k^a cs & 0 & 0 & -k^a c^2 & -k^a cs \\ k^a cs & k^a s^2 & 0 & 0 & -k^a cs & -k^a s^2 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -k^a c^2 & -k^a cs & 0 & 0 & k^a c^2 & k^a cs \\ -k^a cs & -k^a s^2 & 0 & 0 & k^a cs & k^a s^2 \end{bmatrix} \begin{bmatrix} u_{11} \\ u_{12} \\ u_{21} \\ u_{22} \\ u_{31} \\ u_{32} \end{bmatrix} +$$

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & k^b c^2 & k^b cs & -k^b c^2 & -k^b cs \\ 0 & 0 & k^b cs & k^b s^2 & -k^b cs & -k^b s^2 \\ 0 & 0 & -k^b c^2 & -k^b cs & k^b c^2 & k^b cs \\ 0 & 0 & -k^b cs & -k^b s^2 & k^b cs & k^b s^2 \end{bmatrix} \begin{bmatrix} u_{11} \\ u_{12} \\ u_{21} \\ u_{22} \\ u_{31} \\ u_{32} \end{bmatrix} =$$

$$\begin{bmatrix} k^a c^2 & k^a cs & 0 & 0 & -k^a c^2 & -k^a cs \\ k^a cs & k^a s^2 & 0 & 0 & -k^a cs & -k^a s^2 \\ 0 & 0 & k^b c^2 & k^b cs & -k^b c^2 & -k^b cs \\ 0 & 0 & k^b cs & k^b s^2 & -k^b cs & -k^b s^2 \\ -k^a c^2 & -k^a cs & -k^b c^2 & -k^b cs & k^a c^2 + k^b c^2 & k^a cs + k^b cs \\ -k^a cs & -k^a s^2 & -k^b cs & -k^b s^2 & k^a cs + k^b cs & k^a s^2 + k^b s^2 \end{bmatrix} \begin{bmatrix} u_{11} \\ u_{12} \\ u_{21} \\ u_{22} \\ u_{31} \\ u_{32} \end{bmatrix}$$

Assembled system equations $\underset{\sim}{f}_i = \underline{K}\underset{\sim}{u}$

Equilibrium of the truss structure requires that the internal nodal point forces $\underset{\sim}{f}_i$ are equal to the external nodal point forces $\underset{\sim}{f}_e$. This leads to a linear system of equations for the nodal displacement components $\underset{\sim}{u}$. It is, however, not yet possible to solve this set of equations, because some essential boundary conditions have to be incorporated first.

$$\underset{\sim}{f}_i = \underset{\sim}{f}_e \qquad \rightarrow \qquad \underline{K}\underset{\sim}{u} = \underset{\sim}{f}_e = \underset{\sim}{f}$$

Boundary conditions

The equilibrium equations can only be solved uniquely, when proper boundary conditions are prescribed. These boundary conditions are suppressed displacements, prescribed displacements and prescribed forces.

It is always needed to prevent rigid body motions, because otherwise no (unique) solution can be determined. The algebraic system of equations $\underline{K}\,\underset{\sim}{u} = \underset{\sim}{f}$ has to be solved to determine the nodal displacements $\underset{\sim}{u}$. However, the stiffness matrix $\underline{K}$ is singular and cannot be inverted to solve $\underset{\sim}{u}$ when $\underset{\sim}{f}$ is known. This singularity is obvious as we must conclude that for a rigid body translation $\underset{\sim}{u} = \underset{\sim}{a} \neq \underset{\sim}{0}$ the nodal forces are zero.

rigid translation	$\underset{\sim}{u} = \underset{\sim}{a}$
no forces needed	$\underline{K}\,\underset{\sim}{a} = \underset{\sim}{0} \quad \text{with} \quad \underset{\sim}{a} \neq \underset{\sim}{0} \quad \rightarrow \quad \underline{K}$ singular

To get a non-singular matrix we have to suppress the rigid body movement of the construction, by prescribing enough nodal displacements. Besides boundary conditions to suppress rigid body motion, some more nodal displacements may be prescribed, as well as some nodal forces. When in a node a displacement component is prescribed, the associated external force component is unknown and vice versa.

Prescribed nodal displacement components are often denoted as *kinematic* boundary conditions and prescribed nodal forces as *dynamic* boundary conditions.

The *p*rescribed degrees of freedom $\underset{\sim}{u}_p$ are associated with unknown force components $\underset{\sim}{f}_p$. The *u*nknown degrees of freedom $\underset{\sim}{u}_u$ are associated with the known (prescribed or zero) force components $\underset{\sim}{f}_u$. The components of $\underset{\sim}{u}$ and $\underset{\sim}{f}$ are reorganized.

reorganizing
$$\underset{\sim}{u} = \begin{bmatrix} \underset{\sim}{u}_u \\ \underset{\sim}{u}_p \end{bmatrix} \quad ; \quad \underset{\sim}{f} = \begin{bmatrix} \underset{\sim}{f}_u \\ \underset{\sim}{f}_p \end{bmatrix}$$

Reorganizing components of $\underset{\sim}{u}$ implies that *columns* of $\underline{K}$ have to be reorganized in the same way. Reorganizing components of $\underset{\sim}{f}$ implies that *rows* of $\underline{K}$ have to be reorganized in the same way. The components associated with the various parts of $\underset{\sim}{u}$ and $\underset{\sim}{f}$ can be placed in sub-matrices of the resulting matrix $\underline{K}$. The reorganization of columns and matrix described above is called *partitioning.*

As we can see, this partitioning leads to two sets of equations. Only the first set is relevant for the calculation of the unknown $\underset{\sim}{u}_u$. After determination of these unknowns, the second set is used to calculate the unknown reaction forces $\underset{\sim}{f}_p$.

equilibrium
$$\underline{K}\,\underset{\sim}{u} = \underset{\sim}{f}$$

partitioning
$$\begin{bmatrix} \underline{K}_{uu} & \underline{K}_{up} \\ \underline{K}_{pu} & \underline{K}_{pp} \end{bmatrix} \begin{bmatrix} \underset{\sim}{u}_u \\ \underset{\sim}{u}_p \end{bmatrix} = \begin{bmatrix} \underset{\sim}{f}_u \\ \underset{\sim}{f}_p \end{bmatrix} \quad \rightarrow \quad \left. \begin{array}{l} \underline{K}_{uu}\underset{\sim}{u}_u + \underline{K}_{up}\underset{\sim}{u}_p = \underset{\sim}{f}_u \\ \underline{K}_{pu}\underset{\sim}{u}_u + \underline{K}_{pp}\underset{\sim}{u}_p = \underset{\sim}{f}_p \end{array} \right\}$$

solving $\underset{\sim}{u}_u$
$$\underline{K}_{uu}\underset{\sim}{u}_u = \underset{\sim}{f}_u - \underline{K}_{up}\underset{\sim}{u}_p \quad \rightarrow \quad \underset{\sim}{u}_u = \underline{K}_{uu}^{-1}(\underset{\sim}{f}_u - \underline{K}_{up}\underset{\sim}{u}_p)$$

calculating $\underset{\sim}{f}_p$
$$\underset{\sim}{f}_p = \underline{K}_{pu}\underset{\sim}{u}_u + \underline{K}_{pp}\underset{\sim}{u}_p$$

Links

Links (or tyings) are relations between some of the components of $\underset{\sim}{u}$. In these relations we make a difference between independent and dependent components. Dependent or *linked components* can be calculated from the independent ones after these have been solved. The linked components are removed from the equation system, as will be seen later. The independent components are not, so they are called *retained components*. Components which are not part of link relations are simply denoted as *free*. To identify the various components we use the indices l (linked), r (retained) and f (free).

Associated with the linked degrees of freedom are nodal forces, which ensure the relationship. They are calculated by requiring that the total virtual energy, associated with the links, is zero.

The column $\underset{\sim}{u}$ is reorganized such that free, retained and linked components are grouped in columns $\underset{\sim}{u}_f$, $\underset{\sim}{u}_r$ and $\underset{\sim}{u}_l$. The right-hand column $\underset{\sim}{f}$ is reorganized in the same way. The matrix $\underline{K}$ is adapted accordingly by moving rows and columns.

$$\begin{bmatrix} \underline{K}_{ff} & \underline{K}_{fr} & \underline{K}_{fl} \\ \underline{K}_{rf} & \underline{K}_{rr} & \underline{K}_{rl} \\ \underline{K}_{lf} & \underline{K}_{lr} & \underline{K}_{ll} \end{bmatrix} \begin{bmatrix} \underset{\sim}{u}_f \\ \underset{\sim}{u}_r \\ \underset{\sim}{u}_l \end{bmatrix} = \begin{bmatrix} \underset{\sim}{f}_f \\ \underset{\sim}{f}_r + \bar{\underset{\sim}{f}}_r \\ \underset{\sim}{f}_l + \bar{\underset{\sim}{f}}_l \end{bmatrix} \quad \rightarrow \quad \begin{array}{l} \underline{K}_{ff}\underset{\sim}{u}_f + \underline{K}_{fr}\underset{\sim}{u}_r + \underline{K}_{fl}\underset{\sim}{u}_l = \underset{\sim}{f}_f \\ \underline{K}_{rf}\underset{\sim}{u}_f + \underline{K}_{rr}\underset{\sim}{u}_r + \underline{K}_{rl}\underset{\sim}{u}_l = \underset{\sim}{f}_r + \bar{\underset{\sim}{f}}_r \\ \underline{K}_{lf}\underset{\sim}{u}_f + \underline{K}_{lr}\underset{\sim}{u}_r + \underline{K}_{ll}\underset{\sim}{u}_l = \underset{\sim}{f}_l + \bar{\underset{\sim}{f}}_l \end{array}$$

The relation between $\underset{\sim}{u}_l$ and $\underset{\sim}{u}_r$ is denoted with a matrix $\underline{L}_{lr}$. Imposing the link relations will result in a change of the corresponding components of $\underset{\sim}{f}$. In a mechanical system $\bar{\underset{\sim}{f}}_r$ and $\bar{\underset{\sim}{f}}_l$ may be seen as forces which are needed to realize the links between $\underset{\sim}{u}_r$ and $\underset{\sim}{u}_l$. The resulting work of these forces at a random change in $\underset{\sim}{u}_r$ and $\underset{\sim}{u}_l$ must be zero, which results in a relation between $\bar{\underset{\sim}{f}}_r$ and $\bar{\underset{\sim}{f}}_l$.

$$\begin{array}{l} \underset{\sim}{u}_l = \underline{L}_{lr}\underset{\sim}{u}_r \\ \bar{\underset{\sim}{f}}_l^T \delta\underset{\sim}{u}_l + \bar{\underset{\sim}{f}}_r^T \delta\underset{\sim}{u}_r = 0 \quad \forall \quad \{\delta\underset{\sim}{u}_l, \delta\underset{\sim}{u}_r\} \qquad \rightarrow \\ \bar{\underset{\sim}{f}}_l^T \underline{L}_{lr} + \bar{\underset{\sim}{f}}_r^T = \underset{\sim}{0}^T \quad \rightarrow \quad \underline{L}_{lr}^T \bar{\underset{\sim}{f}}_l + \bar{\underset{\sim}{f}}_r = \underset{\sim}{0} \quad \rightarrow \quad \bar{\underset{\sim}{f}}_r = -\underline{L}_{lr}^T \bar{\underset{\sim}{f}}_l = -\underline{L}_{rl}\bar{\underset{\sim}{f}}_l \end{array}$$

Implementation of the link relations results in two systems of algebraic equations from which $\underset{\sim}{u}_r$ and $\underset{\sim}{u}_l$ can be solved.

$$\left. \begin{array}{l} \underline{K}_{ff}\underset{\sim}{u}_f + (\underline{K}_{fr} + \underline{K}_{fl}\underline{L}_{lr})\underset{\sim}{u}_r = \underset{\sim}{f}_f \\ \underline{K}_{rf}\underset{\sim}{u}_f + (\underline{K}_{rr} + \underline{K}_{rl}\underline{L}_{lr})\underset{\sim}{u}_r = \underset{\sim}{f}_r - \underline{L}_{rl}\bar{\underset{\sim}{f}}_l \\ \underline{K}_{lf}\underset{\sim}{u}_f + (\underline{K}_{lr} + \underline{K}_{ll}\underline{L}_{lr})\underset{\sim}{u}_r = \underset{\sim}{f}_l + \bar{\underset{\sim}{f}}_l \\ \\ \text{elimination of } \bar{\underset{\sim}{f}}_l \end{array} \right\} \quad \rightarrow$$

$$\left. \begin{array}{l} \underline{K}_{ff}\underset{\sim}{u}_f + (\underline{K}_{fr} + \underline{K}_{fl}\underline{L}_{lr})\underset{\sim}{u}_r = \underset{\sim}{f}_f \\ (\underline{K}_{rf} + \underline{L}_{rl}\underline{K}_{lf})\underset{\sim}{u}_f + (\underline{K}_{rr} + \underline{K}_{rl}\underline{L}_{lr} + \underline{L}_{rl}\underline{K}_{lr} + \underline{L}_{rl}\underline{K}_{ll}\underline{L}_{lr})\underset{\sim}{u}_r = \underset{\sim}{f}_r + \underline{L}_{rl}\underset{\sim}{f}_l \end{array} \right\} \quad \rightarrow$$

$$\begin{bmatrix} \underline{K}_{ff} & \underline{K}_{fr} + \underline{K}_{fl}\underline{L}_{rl} \\ \underline{K}_{rf} + \underline{L}_{rl}\underline{K}_{lf} & \underline{K}_{rr} + \underline{K}_{rl}\underline{L}_{lr} + \underline{L}_{rl}\underline{K}_{lr} + \underline{L}_{rl}\underline{K}_{ll}\underline{L}_{lr} \end{bmatrix} \begin{bmatrix} \underset{\sim}{u}_f \\ \underset{\sim}{u}_r \end{bmatrix} = \begin{bmatrix} \underset{\sim}{f}_f \\ \underset{\sim}{f}_r + \underline{L}_{rl}\underset{\sim}{f}_l \end{bmatrix} \quad \rightarrow$$

$$\underline{K}\underset{\sim}{u} = \underset{\sim}{f}$$

Program structure

A finite element program starts with reading data from an input file and initialization of variables and databases.

Subsquently, a loop over all elements is started to calculate $\underline{K}^e$ for each individual element and place it at the proper location in the structural matrix $\underline{K}$ (assembly). After taking into account the link relations and boundary conditions, the unknown nodal displacements are calculated.

Subsequently, another loop over all elements is entered to calculate the element strains, stresses and internal nodal forces $\underset{\sim}{f}_i^e$. The latter are assembled in the column $\underset{\sim}{f}_i$, which then contains the reaction forces of the system.

Finally some calculated values are stored for post-processing.

```
read input data from input file
calculate additional variables from input data
initialize values and arrays

for all elements
   calculate initial element stiffness matrix
   assemble global stiffness matrix
end element loop

determine external load from input

take tyings into account
take boundary conditions into account

calculate nodal displacements

for all elements
   calculate stresses from material behavior
   calculate element internal nodal forces
   assemble global internal load column
end element loop

store data for post-processing
```

1.3.1 FE program `tr2dL`

The Matlab program **`tr2dL`** is used to model and analyze two-dimensional truss structures. The input data, which must be provided by the user, are a.o. the coordinates of the nodes, the location of the trusses between nodes, element material data, link relations and prescribed nodal displacements and forces.

In this section, a few examples of two-dimensional truss structures are shown, which are modelled and analyzed with the program.

Simple two-dimensional truss structure

A simple truss structure is shown in the left figure below. The length of the horizontal truss [1] is 100 mm and the length of truss [2] is $200/\sqrt{3}$ mm. Cross-sectional areas are 10 and 20 mm^2, respectively. Young's modulus is 200 and 150 GPa and Poisson's ratio is $\nu = 0.3$. The prescribed force $F = -100$ N leads to the deformation $\{u_{2x}, u_{2y}\} = \{-0.0071, -0.0222\}$ mm, which is shown in the right part of the figure. The real deformation is very small, which is in accordance with the theory, so it is enlarged 1000 times.

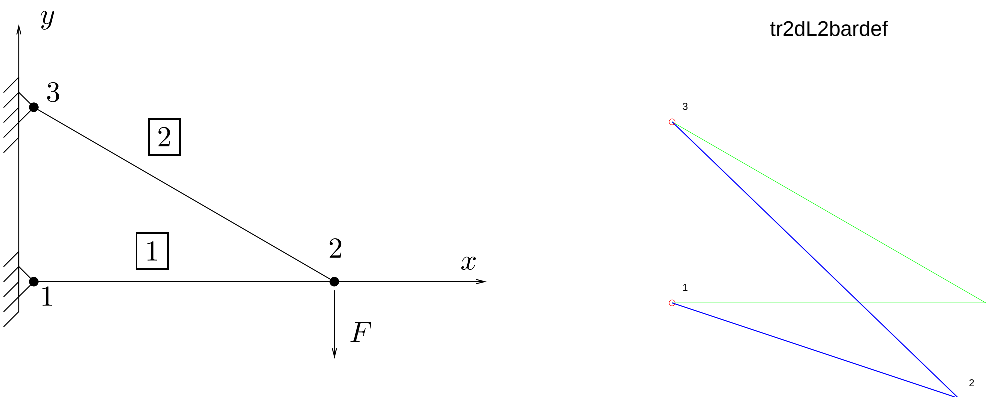


Fig. 1.9 : *Deformation of a truss structure (× 1000).*

Transformation of nodal coordinate system

It is possible to prescribe nodal displacements and/or forces in a local nodal coordinate system, which is rotated over an angle w.r.t. the global system. An example is shown below, where in node 2, the local coordinate axes are rotated over 45^o w.r.t. the xy-axes. The length of the horizontal truss is 100 mm and the length of the vertical truss is 50 mm. Cross-sectional areas are 1 mm^2. Young's moduli and Poisson's ratios are 100 GPa and 0.25. The external load is $F = 100$ N.

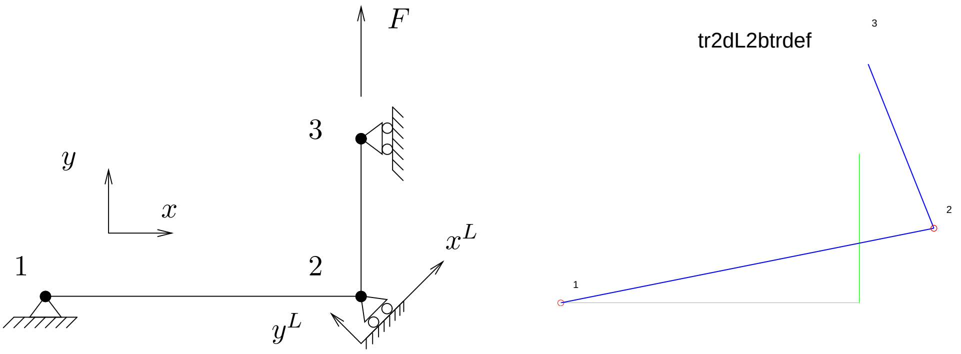


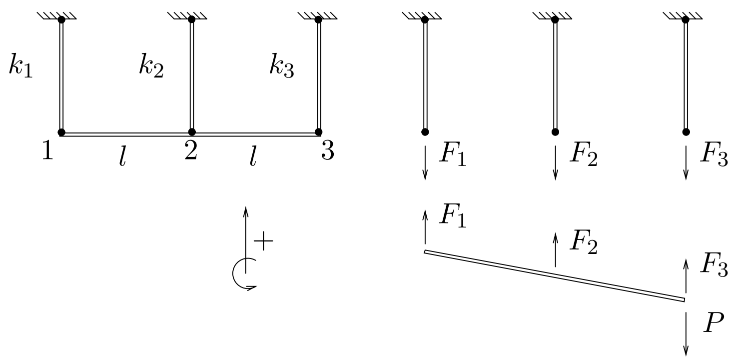
Fig. 1.10 : *Deformation of a truss structure (× 250), with a transformed nodal coordinate system.*

Tyings

The figure shows a rigid beam hanging from three trusses, which have equal stiffness k. The load P will cause an elongation of the trusses, which can be calculated, using link relations.

First the governing equations will be presented and solved analytically. Subsequently the solution of the finite element program will be presented.

The two equilibrium relations are not sufficient to solve the problem. Deformation and thus material behavior (= stiffness k) has to be taken into account. Still the final set of equations cannot be solved.



truss stiffness $\quad k_1 = k_2 = k_3 = k$

equilibrium $\quad F_1 + F_2 + F_3 - P = 0 \quad ; \quad -F_1 2l - F_2 l = 0$

deformation $\quad v_1 = -\dfrac{F_1}{k} \quad ; \quad v_2 = -\dfrac{F_2}{k} \quad ; \quad v_3 = -\dfrac{F_3}{k}$

equilibrium equations in displacements

$$-kv_1 - kv_2 - kv_3 - P = 0 \qquad ; \qquad 2lkv_1 + lkv_2 = 0$$

Due to the rigidity of the beam, the displacements v_1, v_2 and v_3 are not independent. The dependency represents a link relation. Displacement v_2 is linked to the displacements v_1 and v_3. Displacement v_2 is eliminated from the equation system and v_1 and v_3 are retained.

link relation $\qquad v_2 = \frac{1}{2}(v_1 + v_3) \quad \rightarrow \quad v_2 = \begin{bmatrix} \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix}$

elimination of v_2 $\quad \rightarrow \quad$ equation for retained displacements

$$\left.\begin{array}{l} -\frac{3}{2}kv_1 - \frac{3}{2}kv_3 - P = 0 \\ \frac{5}{2}lkv_1 + \frac{1}{2}lkv_3 = 0 \quad \rightarrow \quad v_1 = -\frac{1}{5}v_3 \end{array}\right\} \rightarrow$$

$$\frac{3}{10}kv_3 - \frac{3}{2}kv_3 - P = 0 \quad \rightarrow \quad -\frac{6}{5}kv_3 - P = 0 \quad \rightarrow$$

$$v_3 = -\frac{5}{6}\frac{P}{k} \quad \rightarrow \quad v_1 = \frac{1}{6}\frac{P}{k}$$

$$\text{link} \quad \rightarrow \quad v_2 = -\frac{1}{3}\frac{P}{k}$$

The finite element solution is calculated. The undeformed and deformed structure is shown in the figure below. Both for the analytic and the numerical calculation, we find the next values for the nodal displacements, when setting $k = 100$ N/mm and $P = -10000$ N.

$$v_1 = \frac{100}{6} = 16.66mm \quad ; \quad v_2 = -\frac{100}{3} = -33.33mm \quad ; \quad v_3 = -\frac{500}{6} = -83.33mm$$

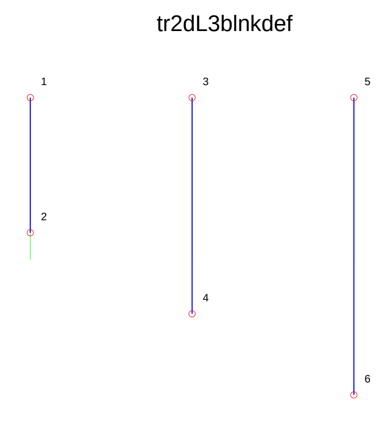


Fig. 1.11 : *Deformation of a truss structure with applied links (× 10).*