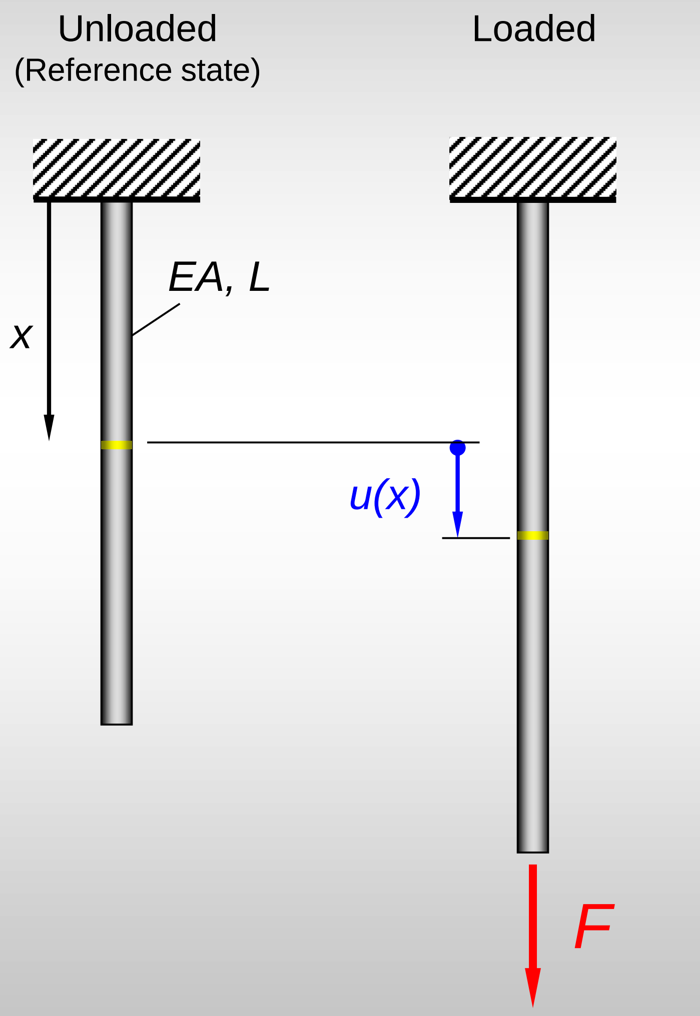
Theory of the Finite Element Method

Using a “Super Simple” Example

Dr. Ulrich Simon

www.UZWR.de

Example: Tensile Rod



Given:

Rod with …

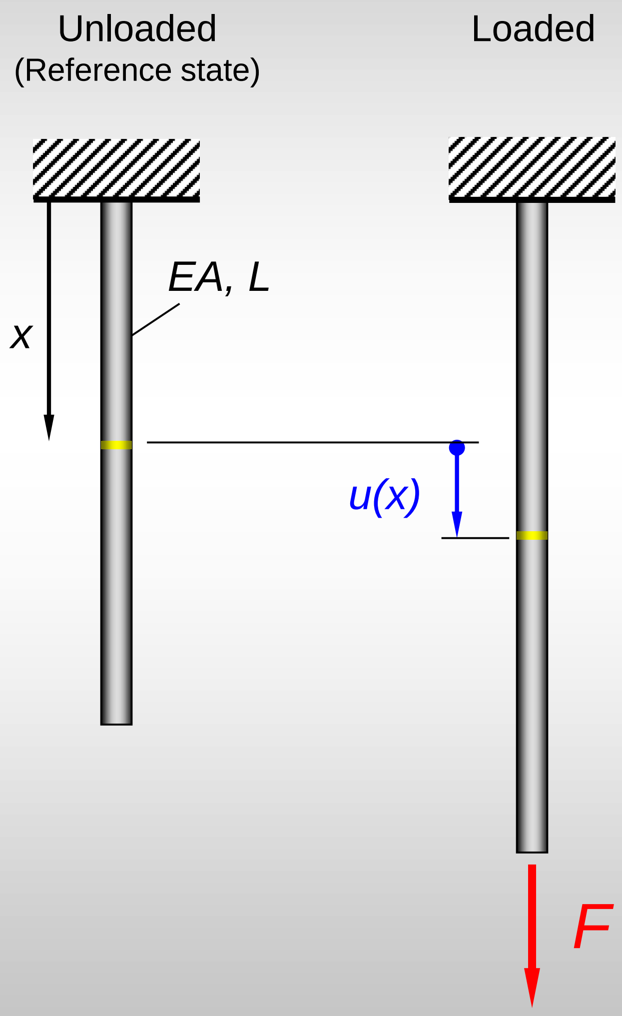
- Length L
- Cross-section A (constant)
- E-modulus E (constant)
- Force F (axial)
- Upper end fixed

To determine:

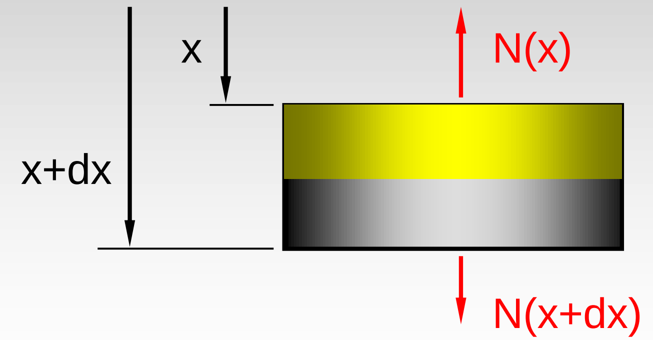
Deformation of the loaded rod:

Displacement function $u(x)$

A) Classical Solution (Method of „infinite“ Elements)



Differential Element (infinitesimale heigth dx)



Generate the Differential Equation

1. Kinematics: $\varepsilon = u'$
2. Material: $\sigma = E\varepsilon \Rightarrow N = EAu'$
3. Equilibrium: $N' = 0$

$$\boxed{\Rightarrow \text{ODE:} \quad (EA\,u')' = 0}$$

If EA = const then

$$\boxed{u'' = 0}$$

A) Classical Solution (Method of „infinite“ Elements)

Solve the Differential Equation

$$u''(x) = 0$$

Integrate 2 times: $u'(x) = C_1$

$u(x) = C_1 \cdot x + C_2$ (General Solution)

Adjust to Boundary Conditions

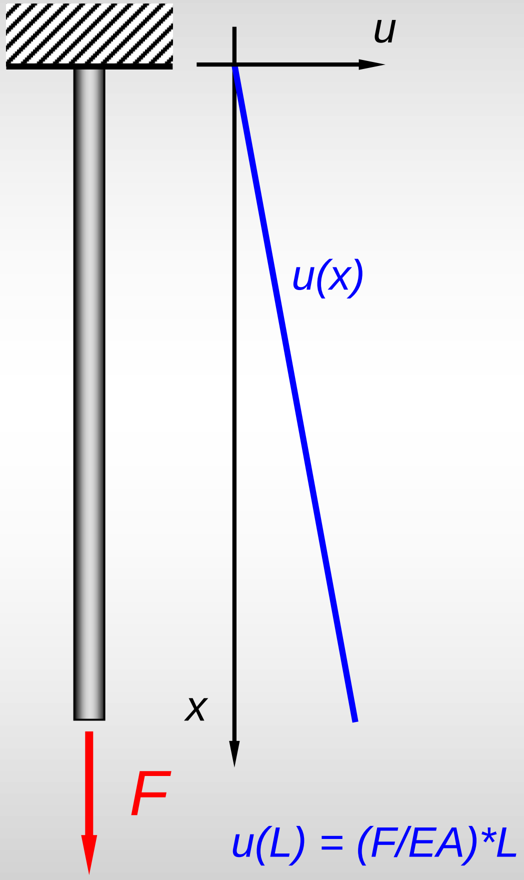
Top (fixation): $u(0) = 0 \Rightarrow C_2 = 0$

Bottom (open, force): $N(L) = F \Rightarrow u'(L) = F/(EA)$

$\Rightarrow C_1 = F/EA$

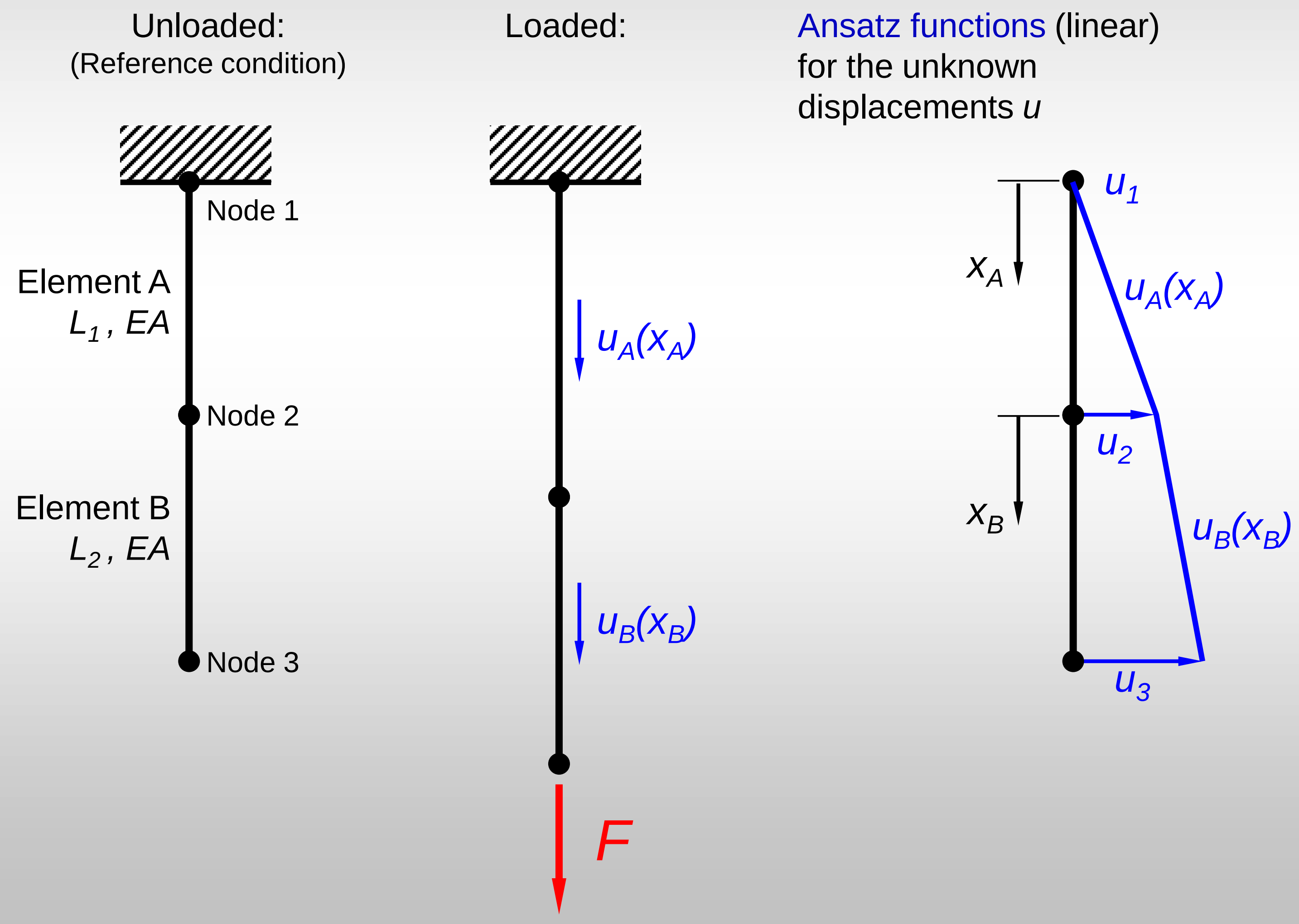
Adjusted Solution

$$\mathbf{u(x) = (F/EA) \cdot x}$$



B) Solution with FEM

Discretization: We divide the rod into (only) two finite (= not infinitesimal small) **Elements.** The Elements are connected at their **nodes**.



The unknown displacement function of the entire rod is described with a series of simple (linear) **ansatz functions** (see figure). This is the basic concept of FEM.

$$u_A(x_A) = \hat{u}_1 + (\hat{u}_2 - \hat{u}_1)\frac{x_A}{L_A} \quad = \hat{u}_1\left(1 - \frac{x_A}{L_A}\right) + \hat{u}_2\frac{x_A}{L_A}$$

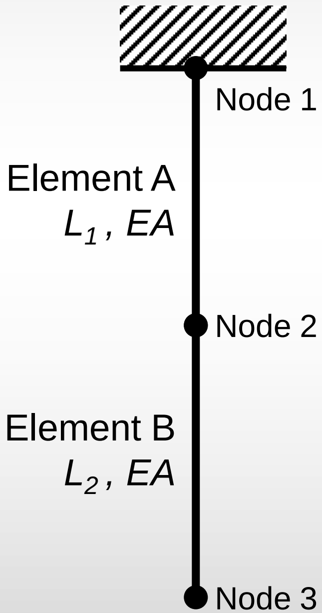
$$u_B(x_B) = \hat{u}_2 + (\hat{u}_3 - \hat{u}_2)\frac{x_B}{L_B} \quad = \hat{u}_2\left(1 - \frac{x_B}{L_B}\right) + \hat{u}_3\frac{x_B}{L_B}$$

The remaining unknowns are the three "nodal displacements" $\hat{u}_1$, $\hat{u}_2$, $\hat{u}_3$ and a no longer a whole function $u(x)$. Now we introduce the so-called "**virtual displacements (VD)".** These are additional, virtual, small, arbitrary displacements $\delta\hat{u}_1$, $\delta\hat{u}_2$, $\delta\hat{u}_3$, consistent with BC. Basically: we "waggle" the nodes a bit.

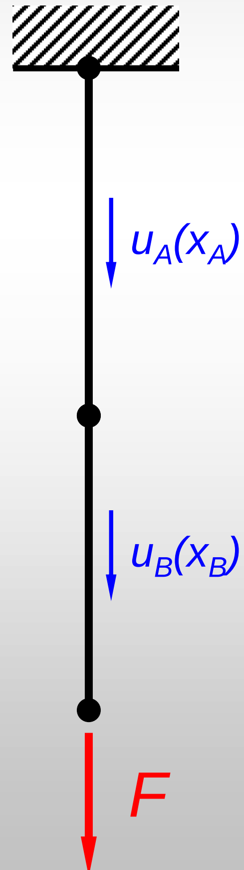
Now the **Principle of Virtual Displacements (PVD)** applies: A mechanical system is in equilibrium when the total work (i.e. elastic minus external work) due to the virtual displacements consequently disappears.

$$\delta W = 0 \quad \Rightarrow \quad \delta W_{el} - \delta W_a = 0$$

Unloaded:
(Reference condition)

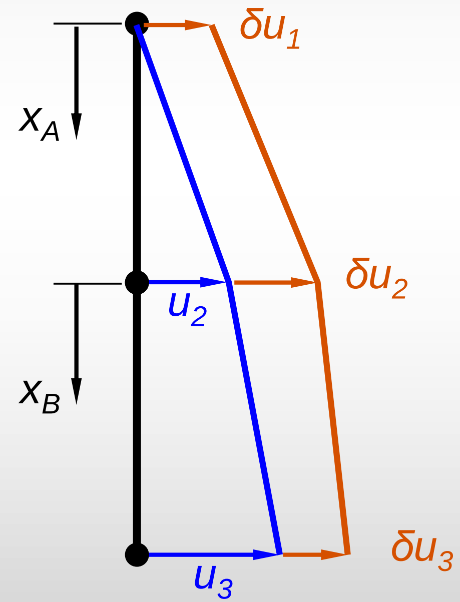


Loaded:



Ansatz functions (linear) for the unknown displacements u

Virtual displacements at all nodal displace- ments



For our simple example we can apply:

Virt. elastic work = normal force N times VD

Virt. external work = external force F times VD

The normal force N can be replaced by the expression EA/L times the element elongation. Element elongation again can be expressed by a difference of the nodal displacements:

$$\delta W = N_A(\delta\hat{u}_2 - \delta\hat{u}_1) + N_B(\delta\hat{u}_3 - \delta\hat{u}_2) - F\delta\hat{u}_3$$
$$= \frac{EA}{L_A}(\hat{u}_2 - \hat{u}_1)(\delta\hat{u}_2 - \delta\hat{u}_1) + \frac{EA}{L_B}(\hat{u}_3 - \hat{u}_2)(\delta\hat{u}_3 - \delta\hat{u}_2) - F\delta\hat{u}_3$$

$$\begin{aligned}\delta W = \delta\hat{u}_1&\left(+\frac{EA}{L_A}\hat{u}_1 - \frac{EA}{L_A}\hat{u}_2 \right)\\ +\delta\hat{u}_2&\left(-\frac{EA}{L_A}\hat{u}_1 + \frac{EA}{L_A}\hat{u}_2 + \frac{EA}{L_B}\hat{u}_2 - \frac{EA}{L_B}\hat{u}_3 \right)\\ +\delta\hat{u}_3&\left(-\frac{EA}{L_B}\hat{u}_2 + \frac{EA}{L_B}\hat{u}_3 - F \right) = 0\end{aligned}$$

With this principle we unfortunately have only **<u>one</u> equation for the <u>three</u> unknown** displacements $\hat{u}_1$, $\hat{u}_2$, $\hat{u}_3$. What a shame! However, there is a trick…

Abbreviated we write:

$$\delta\hat{u}_1(\ldots)_1 + \delta\hat{u}_2(\ldots)_2 + \delta\hat{u}_3(\ldots)_3 = 0$$

The **virtual displacements can be chosen independently** of one another. For instance all except one can be zero. Then the term within the bracket next to this not zero VD has to be zero, in order to fulfill the equation. However, as we can chose the VD we want and also another VD could be chosen as the only non-zero value, consequently all three brackets must individually be zero. **We get three equations**. Juhu!

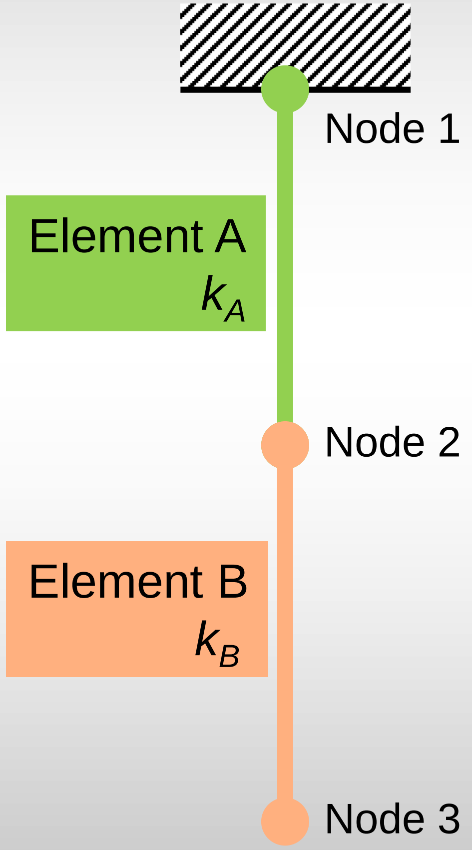
$$(\ldots)_1 = 0; \quad (\ldots)_2 = 0; \quad (\ldots)_3 = 0$$

… which we can also write down in matrix form:

$$\begin{bmatrix} \frac{EA}{L_A} & -\frac{EA}{L_A} & 0 \\ -\frac{EA}{L_A} & \frac{EA}{L_A} + \frac{EA}{L_B} & -\frac{EA}{L_B} \\ 0 & -\frac{EA}{L_B} & \frac{EA}{L_B} \end{bmatrix} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ F \end{bmatrix}$$

Entire Stiffness Matrix

$$\begin{array}{cc} & \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{array}{c} 1 \\ 2 \\ 3 \end{array} & \begin{bmatrix} k_A & -k_A & 0 \\ -k_A & k_A + k_B & -k_B \\ 0 & -k_B & k_B \end{bmatrix} \end{array} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ F \end{bmatrix}$$



Element Stiffness Matrix of Element i

$$\begin{bmatrix} k_i & -k_i \\ -k_i & k_i \end{bmatrix}$$

$$\begin{bmatrix} k_A & -k_A & 0 \\ -k_A & k_A + k_B & -k_B \\ 0 & -k_B & k_B \end{bmatrix} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ F \end{bmatrix}$$

Or in short:

$$\underline{\underline{K}}\,\underline{\hat{u}} = \underline{\underline{F}}$$

$\underline{\underline{K}}$ - Stiffness matrix
$\underline{\hat{u}}$ - Vector of the unknown nodal displacement
$\underline{F}$ - Vector of the nodal forces

This is the classical fundamental equation of a structural mechanics, linear FE-analysis. A **linear system of equations** for the unknown nodal displacements

We still have to account for the **boundary conditions**. The rod is fixed at the top end. As a consequence node 1 cannot be displaced:

$$\hat{u}_1 = 0$$

Because the virtual displacements also have to fulfill the boundary conditions

$\delta\hat{u}_1 = 0.$

we need to eliminate the first line in the system of equations, as this equation does no longer need to be fulfilled. The first column of the matrix can also be removed, as these elements are in any case multiplied by zero. So it becomes …

$$\begin{bmatrix} k_A + k_B & -k_B \\ -k_B & k_B \end{bmatrix} \begin{bmatrix} \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ F \end{bmatrix}$$

We **solve** the system of equations and obtain the nodal displacements

$$\hat{u}_2 = \frac{L_A}{EA} F \quad \text{und} \quad \hat{u}_3 = \frac{L_A + L_B}{EA} F$$

<u>Analytical Solution</u>:

u(x) = (F/EA)*x

Here the **FE-solution** corresponds exactly with the (existing) analytical solution. In a more complex example this would not be the case.
Generally, it applies that the convergence of the numerical solution with the exact solution continually improves with an increasing number of finite elements. For extremely complicated problems there is no longer an analytical solution; for such cases one needs FEM!

From the nodal displacements one can also determine **strains and stresses** in a subsequent calculation [Nachlaufrechnung]. In our example strains and stresses stay constant within the elements.

$$\varepsilon_A(x_A) = \frac{\hat{u}_2 - \hat{u}_1}{L_A}$$

$$\varepsilon_B(x_B) = \frac{\hat{u}_3 - \hat{u}_2}{L_B}$$

Strains

$$\sigma_A(x_A) = E\varepsilon_A(x_A)$$

$$\sigma_B(x_B) = E\varepsilon_B(x_B)$$

Stresses

Finished!

Summary

The **essential steps** and ideas of FEM are thus:

- Discretization: Division of the spatial domain into finite elements
- Choose simple Ansatz functions (polynomials) for the unknown variables within the elements. This reduces the problem to a finite number of unknowns.
- Write up a mechanical principle (e.g. PVD, the mathematician says “weak formulation” of the PDE) and
- From this derive a system of equations for the unknown nodal variables
- Solve the system of equations

Many of these steps will no longer be apparent when using a commercial FE program. With the selection of an analysis and an element type the underlying PDE and the Ansatz functions are implicitly already chosen. The mechanical principle was only being used during the development of the program code in order to determine the template structure of the stiffness matrix. During the solution run the program first creates the (big) linear system of equations based on that known template structure, than solves the system in terms of nodal displacements, and finally calculating strains and than stresses.