Aalto University
School of Engineering

GEO – E1050
Finite Element Method in Geoengineering

Lecture 8-9. Advanced FEM
Errors, error estimates, finite element refinement, nonlinear FEM, time dimension and more

Refresh

1. What element shape functions are for?

Refresh

1. What element shape functions are for?

$$\overbrace{\Delta\mathbf{u}}^{\substack{\text{Vector containing}\\ \text{displacement}\\ \text{at some point}\\ \text{within the}\\ \text{finite element}}} = \overbrace{\mathbf{N}}^{\substack{\text{Matrix containing}\\ \text{shape functions values}\\ \text{at some point within}\\ \text{the finite element}}} \overbrace{\Delta\mathbf{d}}^{\substack{\text{Vector containing}\\ \text{displacements}\\ \text{of element nodes}}}{}_{E}$$

Refresh

2. How to compute strains within finite element, knowing nodal displacements?

Refresh

2. How to compute strains within finite element, knowing nodal displacements?

$$\varepsilon_{ij} = 0.5\left(-\frac{\partial u_j}{\partial x_i} - \frac{\partial u_i}{\partial x_j} \right), \qquad i, j = 1,2,3$$

$$\Delta\boldsymbol{\varepsilon} = \begin{bmatrix} d\varepsilon_{11} \\ d\varepsilon_{22} \\ d\varepsilon_{12} \end{bmatrix} = \mathbf{B}\,\Delta\mathbf{d}_E = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \dots & \dots & \frac{\partial N_n}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & \dots & \dots & 0 & \frac{\partial N_n}{\partial y} \\ 0.5\frac{\partial N_1}{\partial y} & 0.5\frac{\partial N_1}{\partial x} & 0.5\frac{\partial N_2}{\partial y} & 0.5\frac{\partial N_2}{\partial x} & \dots & \dots & 0.5\frac{\partial N_n}{\partial y} & 0.5\frac{\partial N_n}{\partial x} \end{bmatrix} \overbrace{\begin{bmatrix} d_{x1} \\ d_{y1} \\ d_{x2} \\ d_{y2} \\ d_{x3} \\ d_{y3} \end{bmatrix}}^{\Delta\mathbf{d}_E}$$

Refresh

3. What is element stiffness matrix for?

Refresh

3. What is element stiffness matrix for?

$$\overbrace{\mathbf{K}_E}^{\begin{array}{c}\text{Element stiffnes}\\\text{marix}\end{array}} \overbrace{\Delta\mathbf{d}_E}^{\begin{array}{c}\text{Vector containing}\\\text{displacements increments}\\\text{of element nodes}\end{array}} = \overbrace{\Delta\mathbf{R}_E}^{\begin{array}{c}\text{Vector containing}\\\text{forces at element nodes}\\\text{(and increments of them)}\end{array}}$$

Refresh

4. What is global stiffness matrix?

Refresh

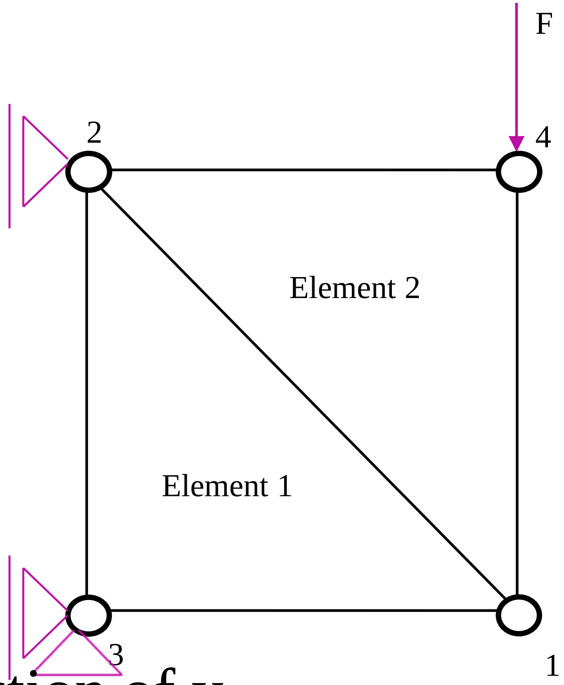
4. What is global stiffness matrix?

$$\overbrace{\mathbf{K}_G}^{\substack{\text{Global stiffnes}\\ \text{marix}}}\;\overbrace{\Delta\mathbf{d}_G}^{\substack{\text{Vector containing}\\ \text{displacement increments}\\ \text{of all element nodes}}} = \overbrace{\Delta\mathbf{R}_G}^{\substack{\text{Vector containing forces}\\ \text{and their increments}\\ \text{at all element nodes}}}$$

Refresh

5. How we apply boundary conditions?

Refresh



5. How we apply boundary conditions?

element number

stiffness of
node number 1
in direction of x

K^{1}_{12xy}

due to movement of node number 2 in direction of y

$$\mathbf{K}_G \Delta\mathbf{d}_G = \overbrace{\begin{bmatrix} K^1_{11xx}+K^2_{11xx} & K^1_{11xy}+K^2_{11xy} & K^1_{12xx}+K^2_{12xx} & K^1_{12xy}+K^2_{12xy} & K^1_{13xx}+0 & K^1_{13xy}+0 & K^2_{14xx}+0 & K^2_{14xy}+0 \\ & K^1_{11yy}+K^2_{11yy} & K^1_{12yx}+K^2_{12yx} & K^1_{12yy}+K^2_{12yy} & K^1_{13yx}+0 & K^1_{13yy}+0 & K^2_{14yx}+0 & K^2_{14yy}+0 \\ & & K^1_{22xx}+K^2_{22xx} & K^1_{22xy}+K^2_{22xy} & K^1_{23xx}+0 & K^1_{23xy}+0 & K^2_{24xx}+0 & K^2_{24xy}+0 \\ & & & K^1_{22yy}+K^2_{22yy} & K^1_{23yx}+0 & K^1_{23yy}+0 & K^2_{24yx}+0 & K^2_{24yy}+0 \\ & & & & K^1_{33xx}+0 & K^1_{33xy}+0 & 0 & 0 \\ & & & & & K^1_{33yy}+0 & 0 & 0 \\ & & & & & & K^2_{44xx}+0 & K^2_{44xy}+0 \\ . & & & & & & & K^2_{44yy}+0 \end{bmatrix}}^{\mathbf{K}_G} \overbrace{\begin{bmatrix} d_{x1} \\ d_{y1} \\ 0 \\ d_{y2} \\ 0 \\ 0 \\ d_{x4} \\ d_{y4} \end{bmatrix}}^{\Delta\mathbf{d}_G} = \overbrace{\begin{bmatrix} 0 \\ 0 \\ \Delta R_{x2} \\ 0 \\ \Delta R_{x3} \\ \Delta R_{y3} \\ 0 \\ F \end{bmatrix}}^{\Delta\mathbf{R}_G} = \Delta\mathbf{R}_G$$

Refresh

6. Having calculated all nodal displacements, how do we get strains and stresses?

$$\Delta\boldsymbol{\varepsilon}=\begin{bmatrix} d\varepsilon_{11}\\ d\varepsilon_{22}\\ d\varepsilon_{12}\end{bmatrix}=\mathbf{B}\,\Delta\mathbf{d}_E=\begin{bmatrix}\frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \dots & \dots & \frac{\partial N_n}{\partial x} & 0\\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & \dots & \dots & 0 & \frac{\partial N_n}{\partial y}\\ 0.5\frac{\partial N_1}{\partial y} & 0.5\frac{\partial N_1}{\partial x} & 0.5\frac{\partial N_2}{\partial y} & 0.5\frac{\partial N_2}{\partial x} & \dots & \dots & 0.5\frac{\partial N_n}{\partial y} & 0.5\frac{\partial N_n}{\partial x}\end{bmatrix}\overbrace{\begin{bmatrix} d_{x1}\\ d_{y1}\\ d_{x2}\\ d_{y2}\\ d_{x3}\\ d_{y3}\end{bmatrix}}^{\Delta\mathbf{d}_E}$$

$$\left\{\overbrace{\begin{bmatrix}\Delta\sigma_{11}\\ \Delta\sigma_{22}\\ \Delta\sigma_{33}\\ \Delta\sigma_{12}\end{bmatrix}}^{\Delta\boldsymbol{\sigma}}=\overbrace{\frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix}1-\nu & \nu & 0\\ \nu & 1-\nu & 0\\ \nu & \nu & 0\\ 0 & 0 & 1-2\nu\end{bmatrix}}^{\mathbf{D}}\overbrace{\begin{bmatrix}\Delta\varepsilon_{11}\\ \Delta\varepsilon_{22}\\ \Delta\varepsilon_{12}\end{bmatrix}}^{\Delta\boldsymbol{\varepsilon}}\right.$$

Error control and mesh refinement

Where is the approximation of Finite Element Algorithm?

How to make a method which would be accurate?

Error control and mesh refinement

Where is the approximation of Finite Element Algorithm?

$$dE = 0.5\int_V d\boldsymbol{\varepsilon}^T d\boldsymbol{\sigma} dV = 0.5\int_V d\boldsymbol{\varepsilon}^T \mathbf{D} d\boldsymbol{\varepsilon} dV = \ldots$$

Approximation: substituting strain field by the nodal displacements and some functions when creating the element stiffness matrices!

$$= 0.5\int_V \left(\mathbf{B}d\mathbf{d}_E\right)^T \mathbf{D}\mathbf{B}d\mathbf{d}_E dV = 0.5\int_V d\mathbf{d}_E^T \mathbf{B}^T \mathbf{D}\mathbf{B}d\mathbf{d}_E dV$$

Another approximation: numerical integration

Error control and mesh refinement

Typically we define error in strain energy, that is:

$$E=\sqrt{\int_V (\boldsymbol{\varepsilon}-\hat{\boldsymbol{\varepsilon}})^T \mathbf{D}(\boldsymbol{\varepsilon}-\hat{\boldsymbol{\varepsilon}})dV}$$

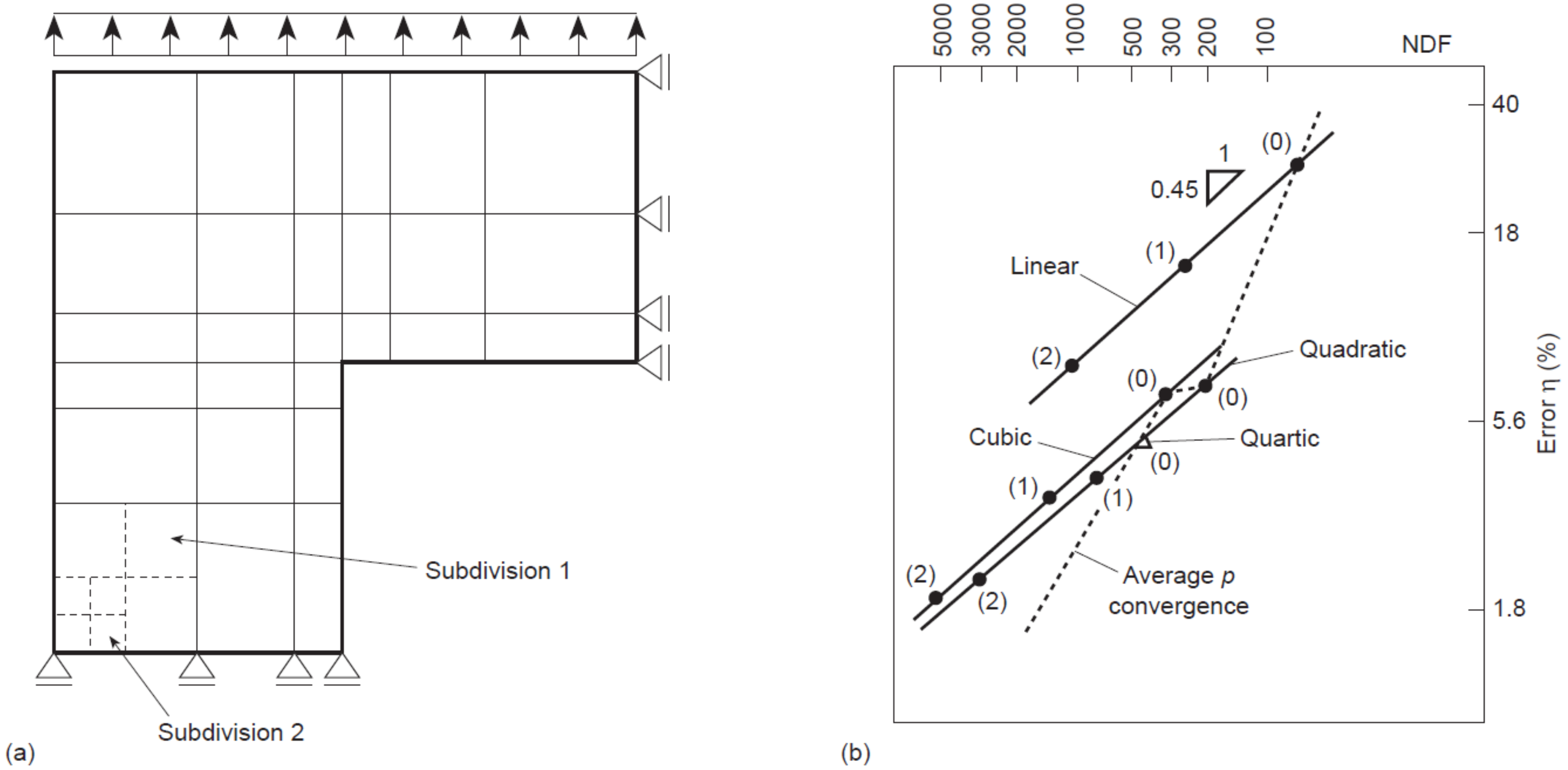
And the strain energy as:

$$E_{Tot}=\sqrt{\int_V \boldsymbol{\varepsilon}^T \mathbf{D}\boldsymbol{\varepsilon} dV}$$

The relative error will be:

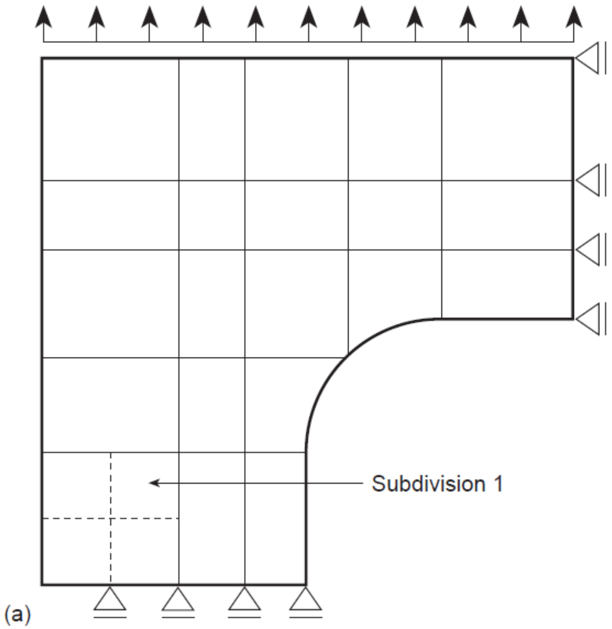
$$e=\eta=\frac{E}{E_{Tot}}$$

Error control and mesh refinement

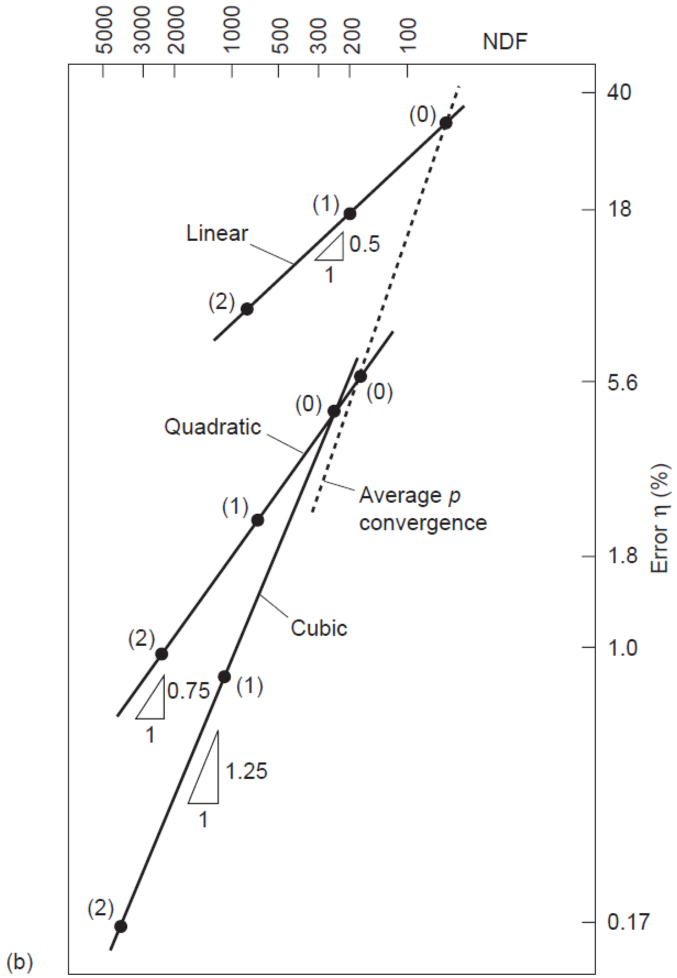


© Zienkiewicz Taylor

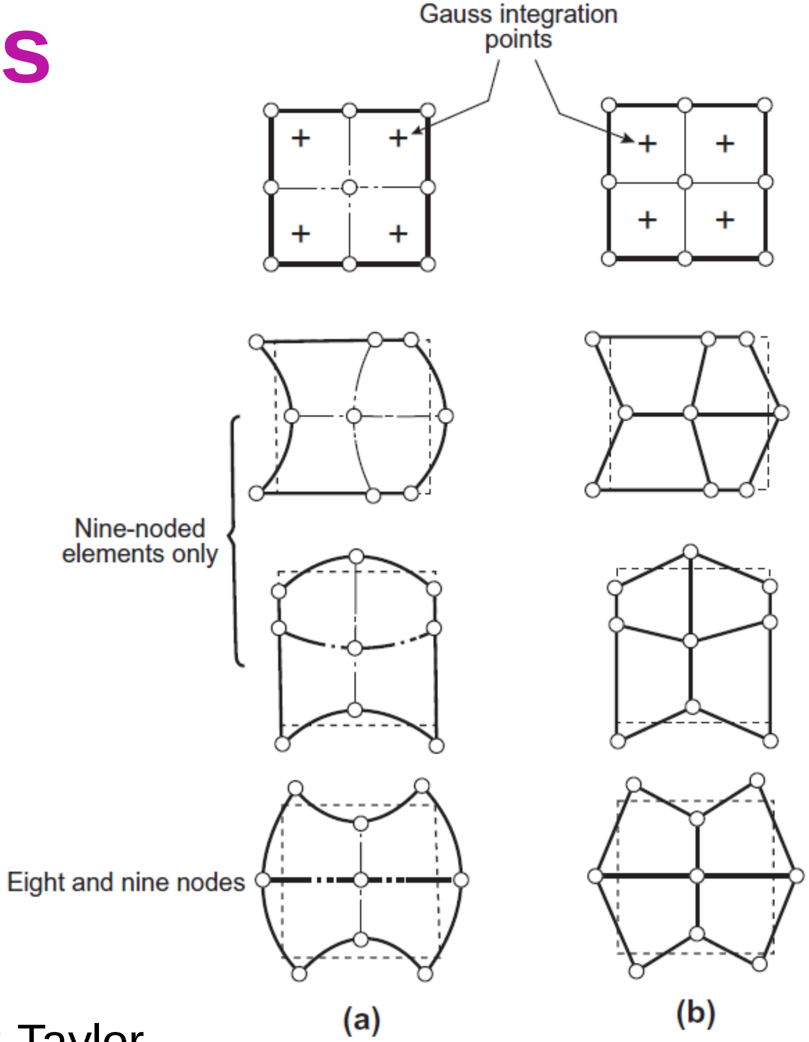
Error control and mesh refinement



© Zienkiewicz Taylor



Zero energy modes



Zienkiewicz Taylor

FIGURE 8.5

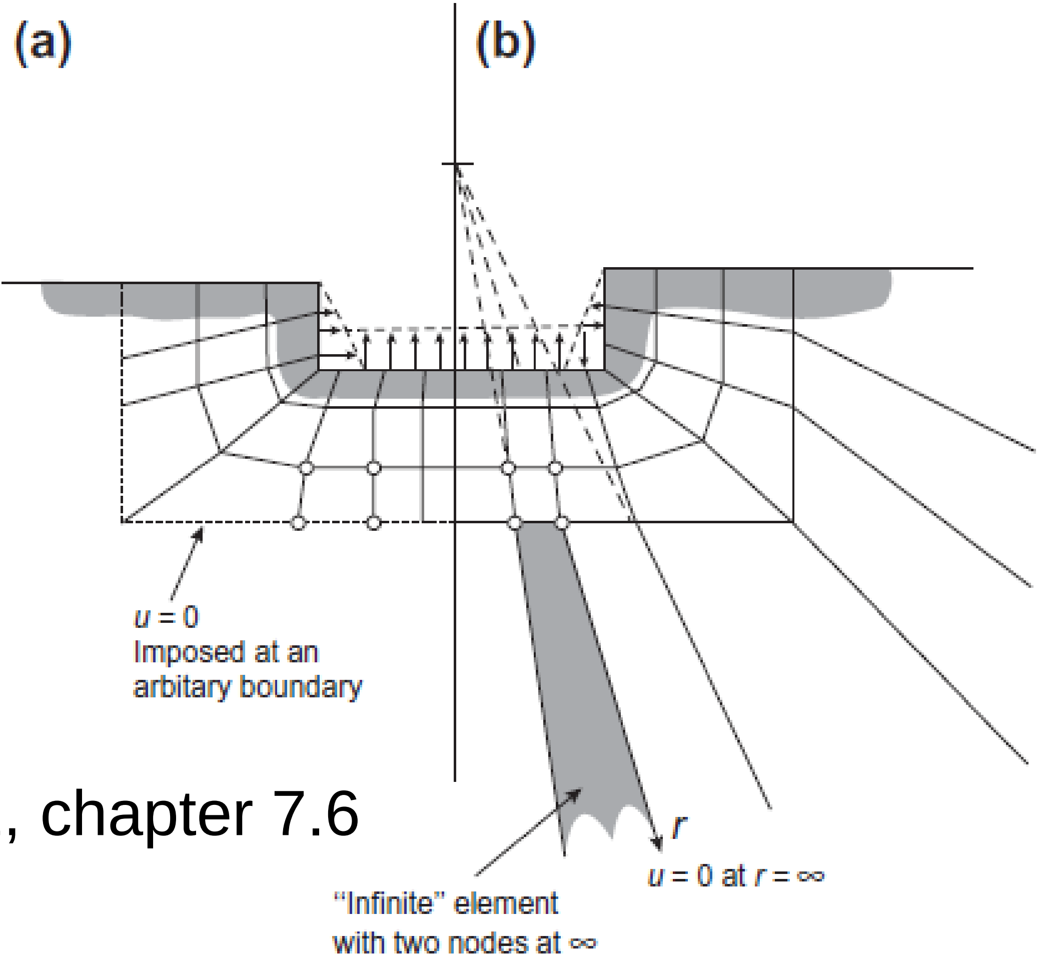
(a) Zero-energy (singular) modes for eight- and nine-node quadratic elements and (b) for a patch of bilinear elements with single integration points.

Aalto University
School of Engineering

GEO – E1050
Finite Element Method in Geoengineering

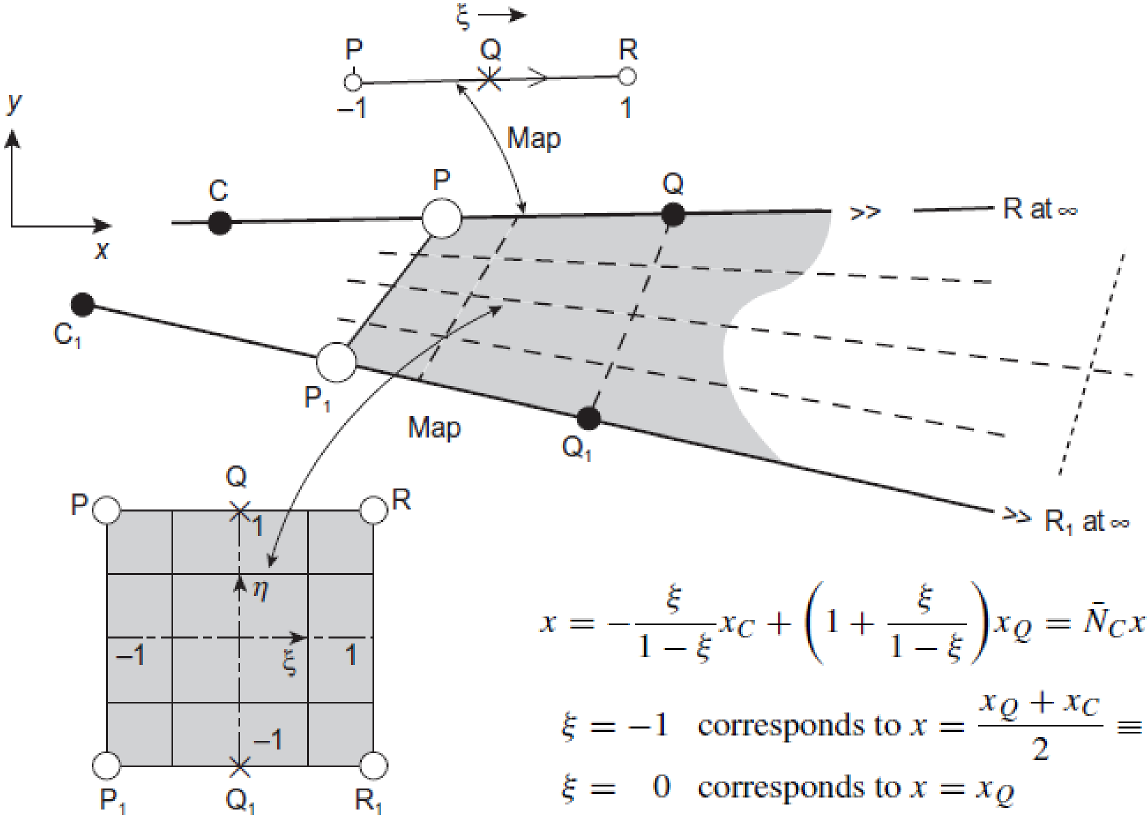
Special elements

Infinite elements



Zienkiewicz, chapter 7.6

Infinite elements



$$x = -\frac{\xi}{1-\xi}x_C + \left(1 + \frac{\xi}{1-\xi}\right)x_Q = \bar{N}_C x_C + \bar{N}_Q x_Q$$

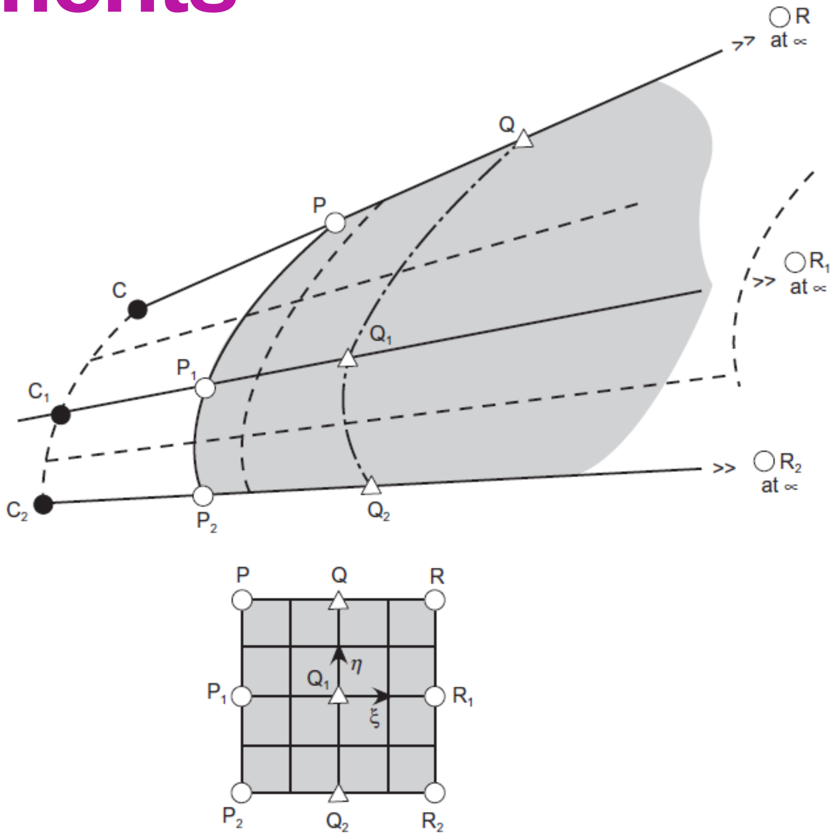
$$\xi = -1 \quad \text{corresponds to } x = \frac{x_Q + x_C}{2} \equiv x_P$$

$$\xi = \quad 0 \quad \text{corresponds to } x = x_Q$$

$$\xi = \quad 1 \quad \text{corresponds to } x = \infty$$

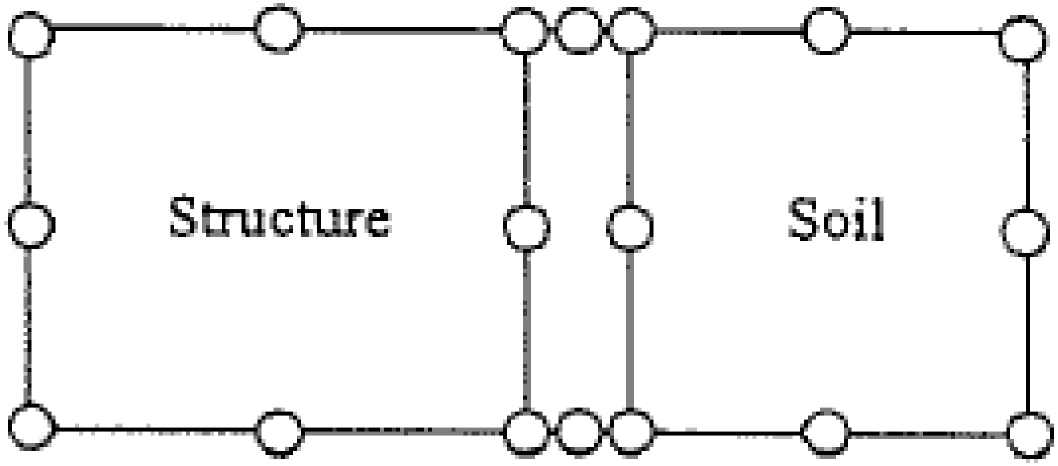
Zienkiewicz, chapter 7.6, linear interpolation

Infinite elements



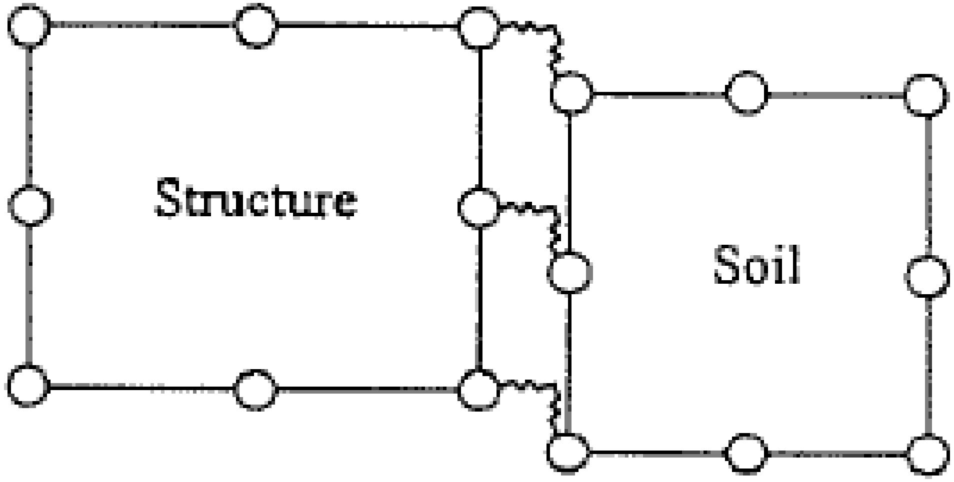
Zienkiewicz, chapter 7.6, quadratic interpolation

Interfaces: to allow for slippage & discontinuity



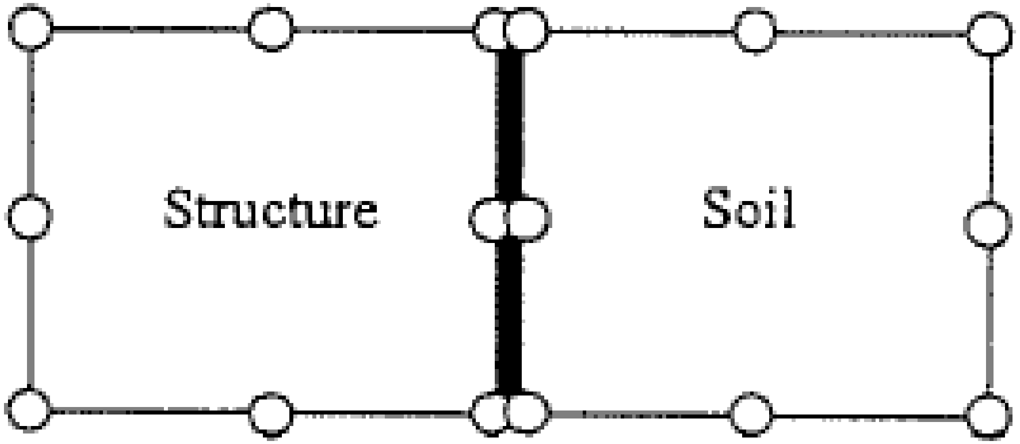
Reduced stiffness element

Interfaces



Springs…

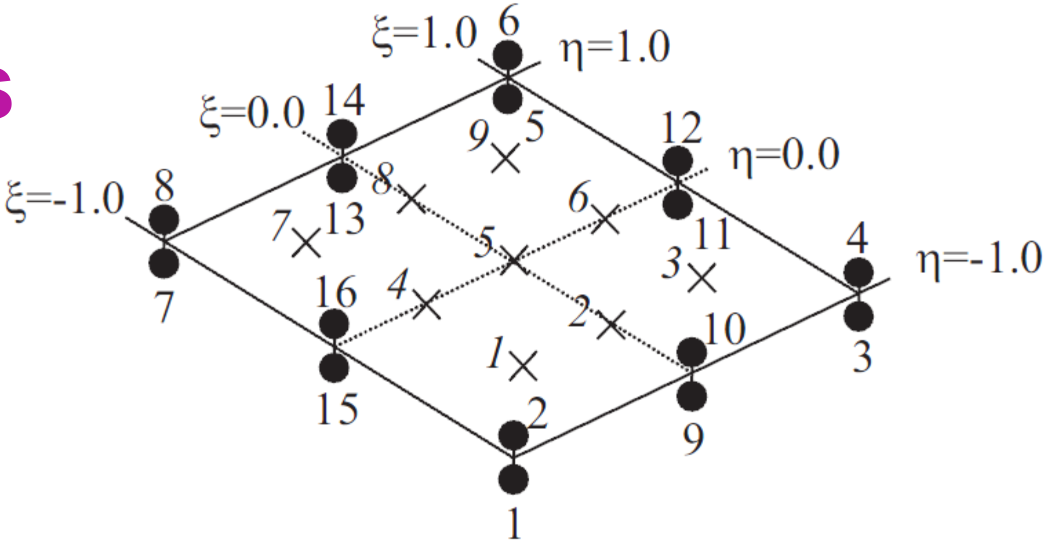
Interfaces



Interface elements…

Always check theory manual !!!

Interfaces: Plaxis



- Initial distance between nodal pairs is zero
- Each node 3 degrees of freedom
- Allows for creation of gaps, slippage etc.
- Friction coefficient can be defined, as well as static/dynamic friction coefficients

Interfaces: Optum in limit analysis

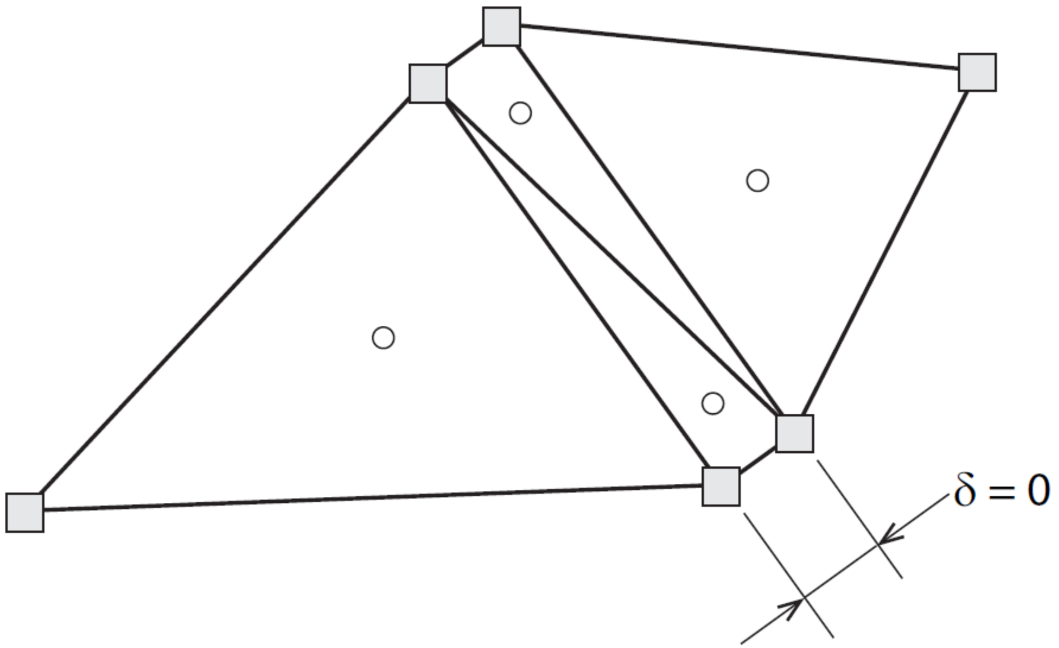
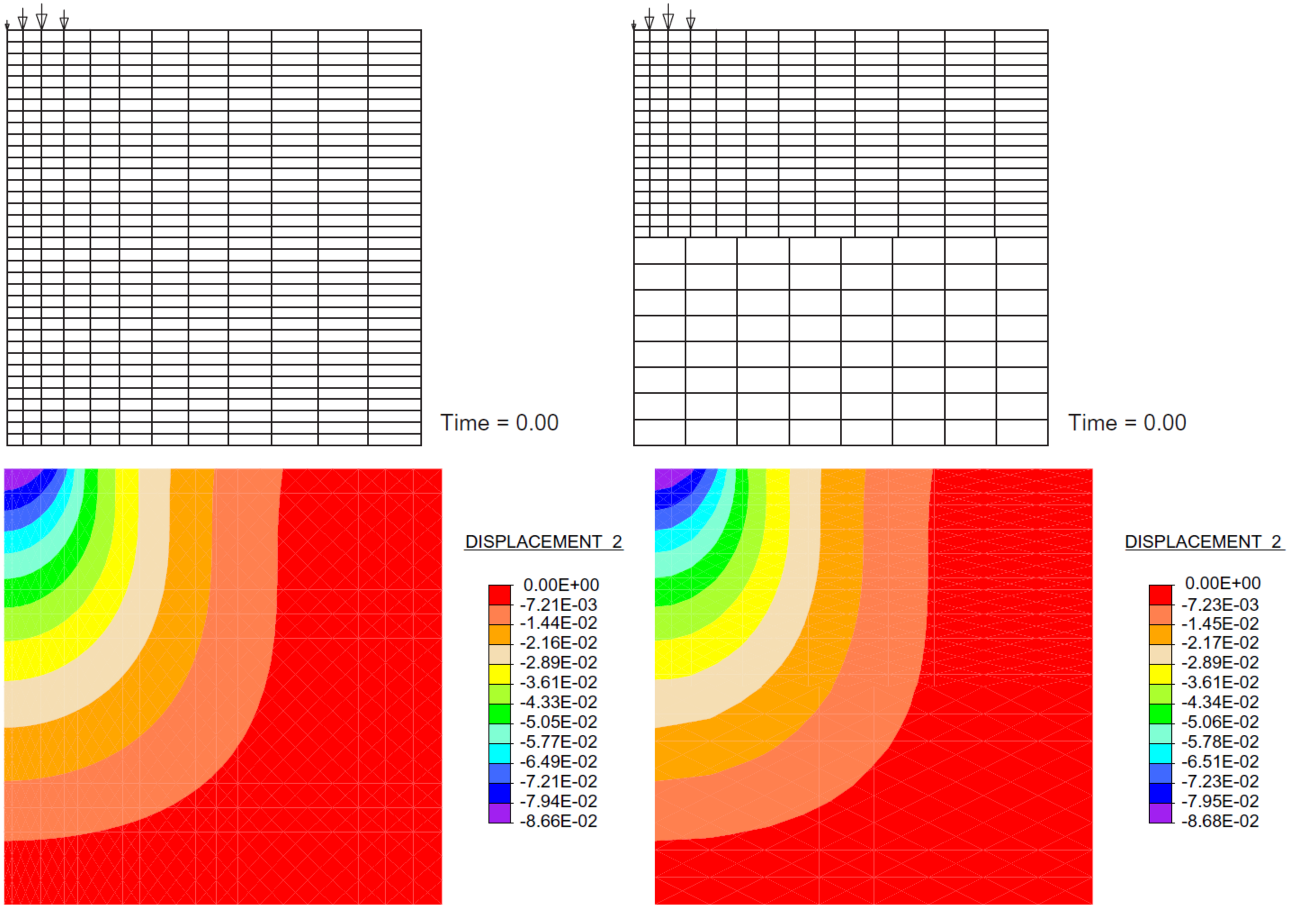


Figure 13.2: Lower bound elements joined by two zero-thickness elements to produce a statically admissible stress discontinuity.

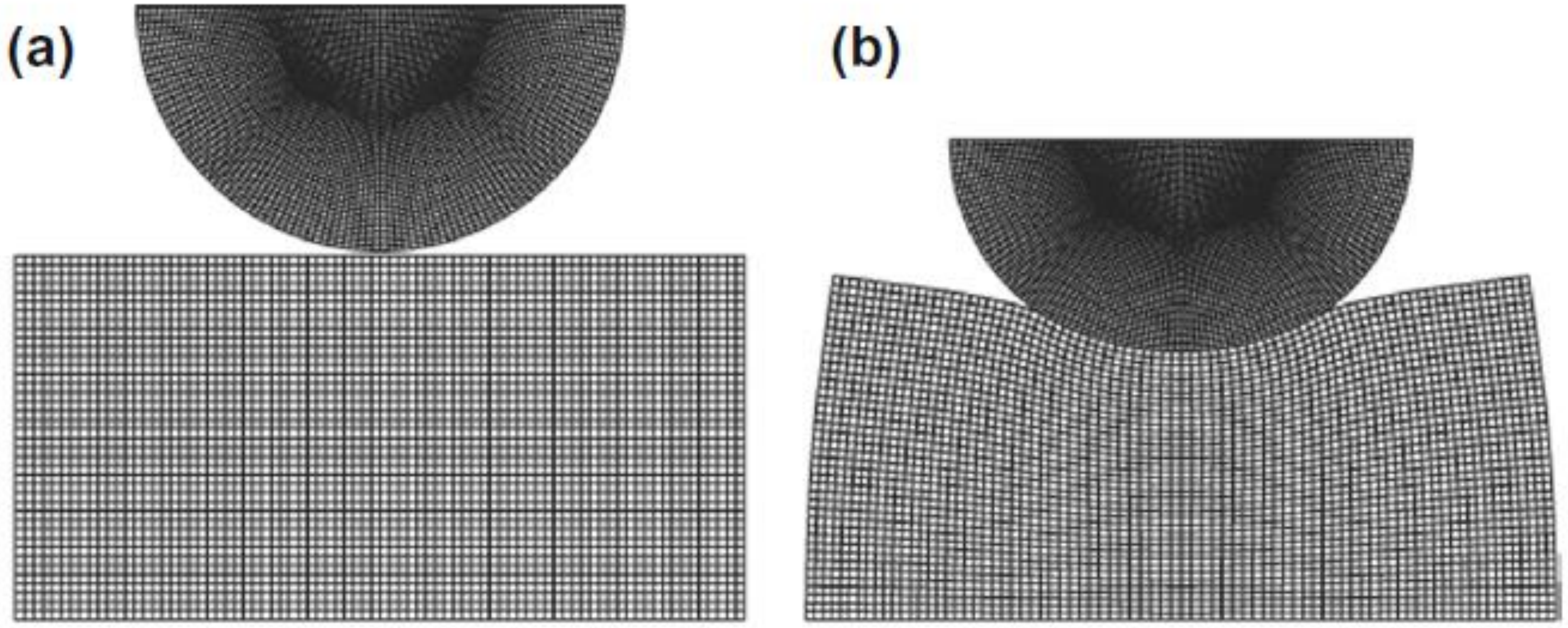
Contact between domains

‘fake’ boundary condition, using Lagrange multipliers (Zienkiewicz Chapter 11)



Contact between domains

If we can do that, we also can do slip / Hertzian contact!
Algorithms still under development…



(Zienkiewicz Vol 2 Chapter 8).

Aalto University
School of Engineering

GEO – E1050
Finite Element Method in Geoengineering

Nonlinear Finite Element Method in Geoengineering

To learn today…

The lecture should give you overview of how non-linear problems in Finite Element Method in Geoengineering are solved.

1. Understand the problem: why non-linear problems require special treatment
2. Understand solutions and their limitations:
    - Initial (tangent) stiffness technique
    - Newton-Raphson
    - Modified Newton-Raphson
3. Understand idea behind the arc-length method
    - full derivation of arc-length method is not required
4. Bonus – geometric non-linearity

Element Stifness matrix

$$0.25h^2\mathbf{B}^T\mathbf{D}\mathbf{B}d\mathbf{d}_E = \Delta\mathbf{R}$$

$$\left\{ \overbrace{\begin{bmatrix} \Delta\sigma_{11} \\ \Delta\sigma_{22} \\ \Delta\sigma_{33} \\ \Delta\sigma_{12} \end{bmatrix}}^{\Delta\boldsymbol{\sigma}} = \overbrace{\frac{E}{(1+v)(1-2v)}\begin{bmatrix} 1-v & v & 0 \\ v & 1-v & 0 \\ v & v & 0 \\ 0 & 0 & 1-2v \end{bmatrix}}^{\mathbf{D}} \overbrace{\begin{bmatrix} \Delta\varepsilon_{11} \\ \Delta\varepsilon_{22} \\ \Delta\varepsilon_{12} \end{bmatrix}}^{\Delta\boldsymbol{\varepsilon}} \right.$$

$$\mathbf{K}_E = 0.25h^2 \overbrace{\begin{bmatrix} -1/h & 0 & 0 & 0 & 1/h & 0 \\ 0 & 0 & 0 & -1/h & 0 & 1/h \\ 0 & -0.5/h & -0.5/h & 0 & 0.5/h & 0.5/h \end{bmatrix}}^{\mathbf{B}}{}^{T} \cdot \overbrace{\frac{E}{(1+v)(1-2v)}\begin{bmatrix} 1-v & v & 0 \\ v & 1-v & 0 \\ v & v & 0 \\ 0 & 0 & 1-2v \end{bmatrix}}^{\mathbf{D}}$$

$$\cdot \overbrace{\begin{bmatrix} -1/h & 0 & 0 & 0 & 1/h & 0 \\ 0 & 0 & 0 & -1/h & 0 & 1/h \\ 0 & -0.5/h & -0.5/h & 0 & 0.5/h & 0.5/h \end{bmatrix}}^{\mathbf{B}}$$

Global Stifness matrix

$$\overbrace{\mathbf{K}_E}^{\substack{\text{Element stiffnes}\\ \text{marix}}} \quad \overbrace{\Delta\mathbf{d}_E}^{\substack{\text{Vector containing}\\ \text{displacements increments}\\ \text{of element nodes}}} \quad = \quad \overbrace{\Delta\mathbf{R}_E}^{\substack{\text{Vector containing}\\ \text{forces at element nodes}\\ \text{(and increments of them)}}}$$

$$\overbrace{\mathbf{K}_G}^{\substack{\text{Global stiffnes}\\ \text{marix}}} \quad \overbrace{\Delta\mathbf{d}_G}^{\substack{\text{Vector containing}\\ \text{displacement increments}\\ \text{of all element nodes}}} \quad = \quad \overbrace{\Delta\mathbf{R}_G}^{\substack{\text{Vector containing forces}\\ \text{and their increments}\\ \text{at all element nodes}}}$$

$$\mathbf{K}_E = 0.25\frac{E}{(1+v)(1-2v)}\overbrace{\begin{bmatrix} -1 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 \\ 0 & -0.5 & -0.5 & 0 & 0.5 & 0.5 \end{bmatrix}}^{\mathbf{B}}{}^{T} \cdot \overbrace{\begin{bmatrix} 1-v & v & 0 \\ v & 1-v & 0 \\ v & v & 0 \\ 0 & 0 & 1-2v \end{bmatrix}}^{\mathbf{D}}$$

$$\cdot \overbrace{\begin{bmatrix} -1 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 \\ 0 & -0.5 & -0.5 & 0 & 0.5 & 0.5 \end{bmatrix}}^{\mathbf{B}}$$

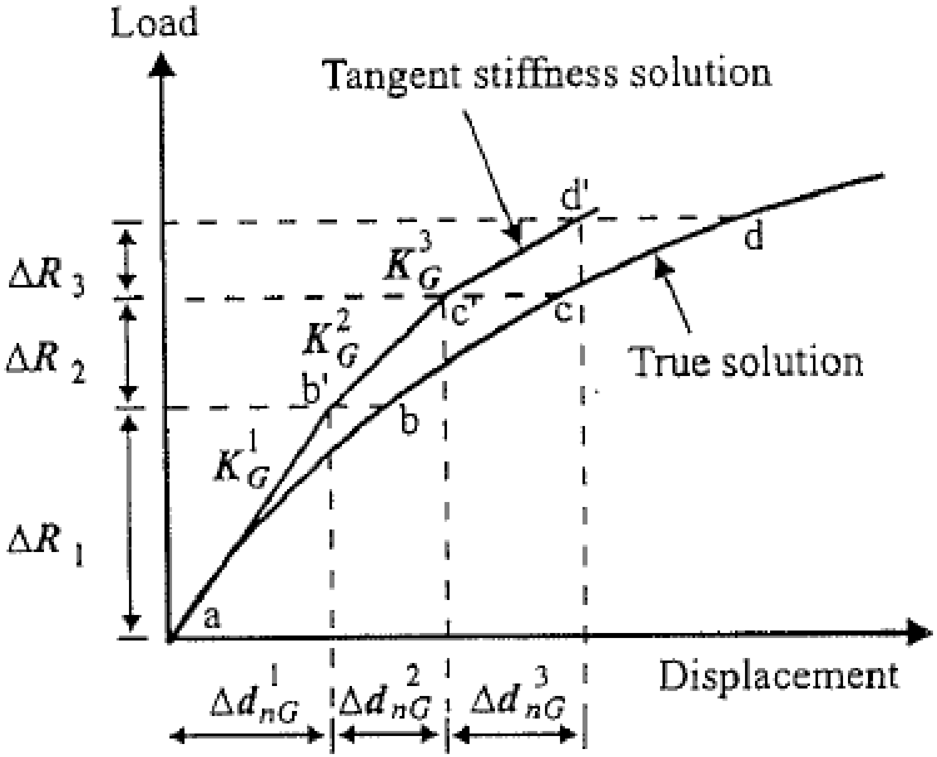
Initial (tangent) stiffness / incremental solution

In the incremental solution we divide the load into increments $\Delta P = \lambda P$ where λ is also known as a load factor and apply a repeated solution of

$$\Delta w = K_T^{-1} \Delta P$$

Basically we divide the load into substeps and treat each as linear – but that is usually not accurate enough and inefficient

Initial (tangent) stiffness / incremental solution



Basically we divide the load into substeps and treat each as linear – but that is usually not accurate enough and inefficient

Initial (tangent) stiffness / incremental solution

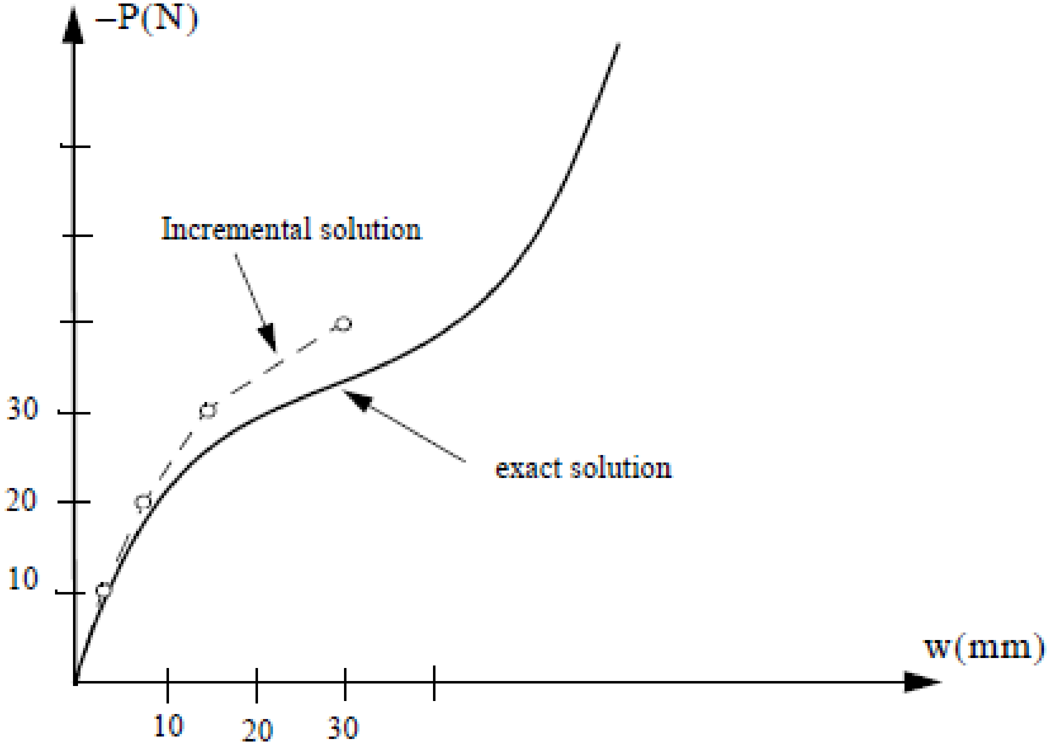
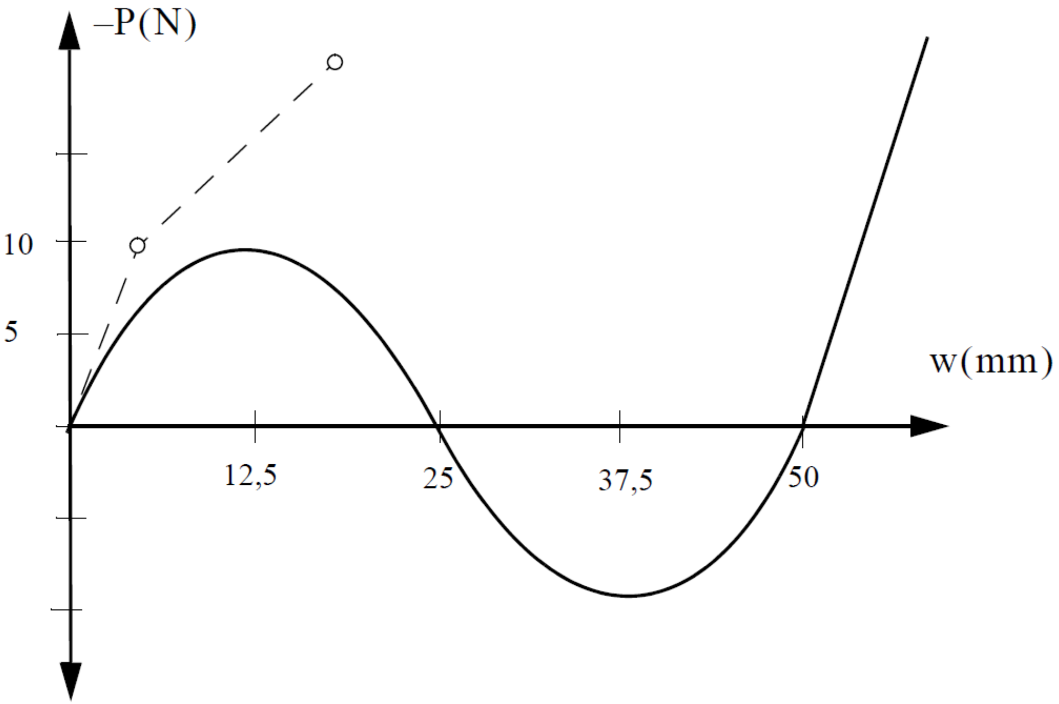


Fig. 2.2 Incremental solution of the problem 1

Initial (tangent) stiffness / incremental solution



Basically we divide the load into substeps and treat each as linear – but that is usually not accurate enough and inefficient

Recalculate stresses in the elements

$$\overbrace{\mathbf{K}_G}^{\substack{\text{Global stiffnes}\\ \text{marix}}} \overbrace{\Delta\mathbf{d}_G}^{\substack{\text{Vector containing}\\ \text{displacement increments}\\ \text{of all element nodes}}} = \overbrace{\Delta\mathbf{R}_G}^{\substack{\text{Vector containing forces}\\ \text{and their increments}\\ \text{at all element nodes}}}$$

$\uparrow$

Depends on stress and strain state… Hence it is not constant.

In elasto-plasticity most often – material softens which means that the predicted displacements are too small (we assume that the material is “too stiff”)

Recalculate stresses in the elements

$$\overbrace{\mathbf{K}_G}^{\substack{\text{Global stiffnes}\\ \text{marix}}} \; \overbrace{\Delta\mathbf{d}_G}^{\substack{\text{Vector containing}\\ \text{displacement increments}\\ \text{of all element nodes}}} = \overbrace{\Delta\mathbf{R}_G}^{\substack{\text{Vector containing forces}\\ \text{and their increments}\\ \text{at all element nodes}}}$$

For each element: $\Delta\boldsymbol{\varepsilon} = \mathbf{B}\,\Delta\mathbf{d}_E$

Having strain increment, we calculate stress increment accurately (various algorithms exist) $\Delta\boldsymbol{\sigma} = \int \mathbf{D}^{ep}(...)d\boldsymbol{\varepsilon}$

As material most often softens, the stresses are lower than what we assumed initially.

Having stresses, one can compute unbalanced forces in the system – and use those to compute next iteration $\Delta\mathbf{R}_i$

Newton - Raphson

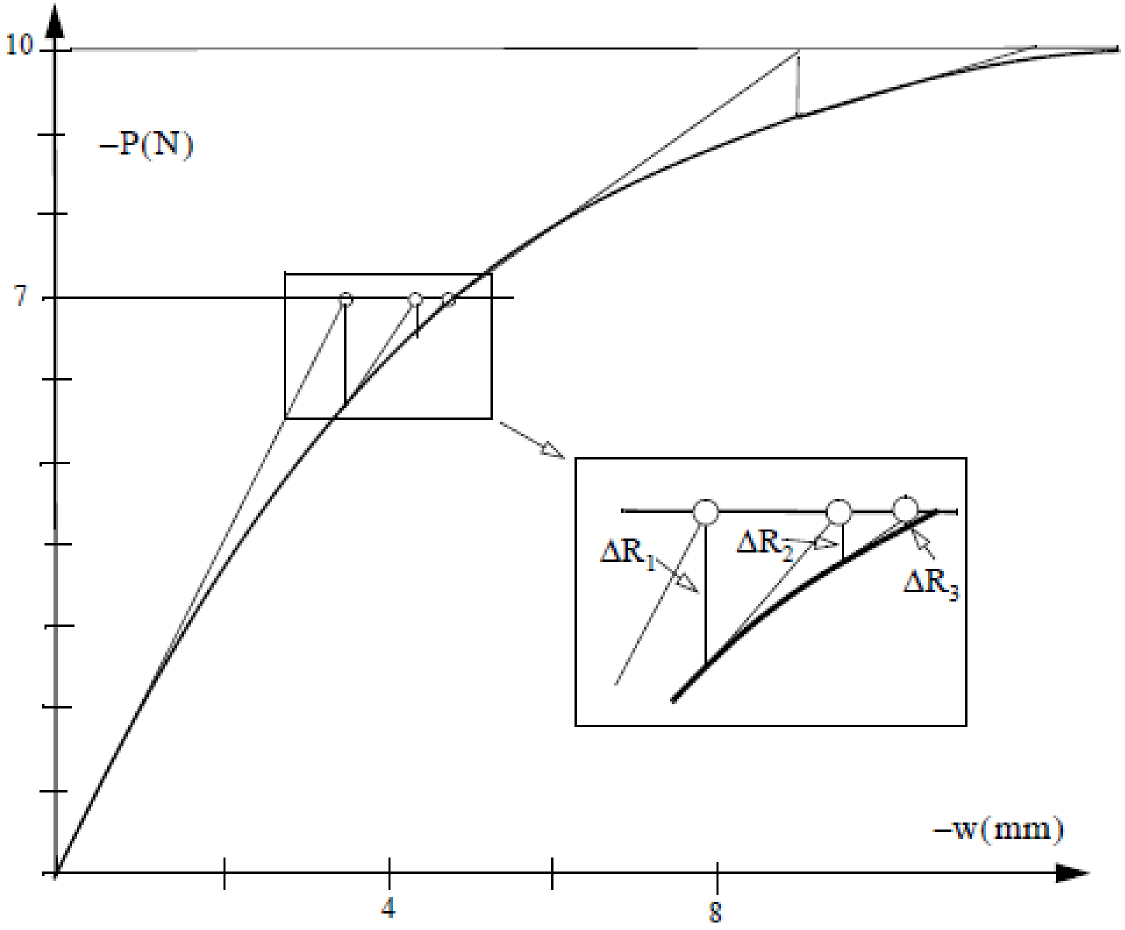
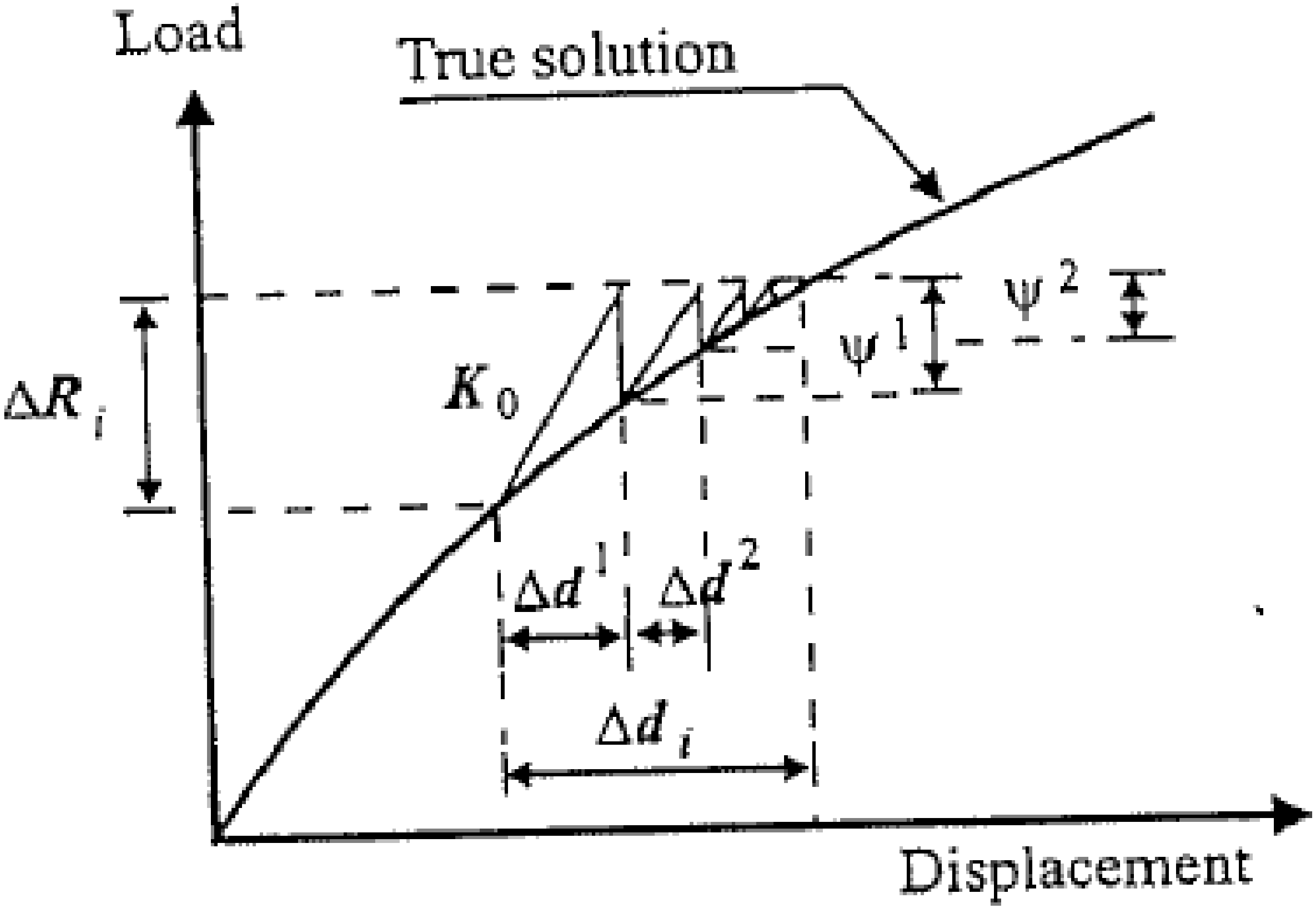
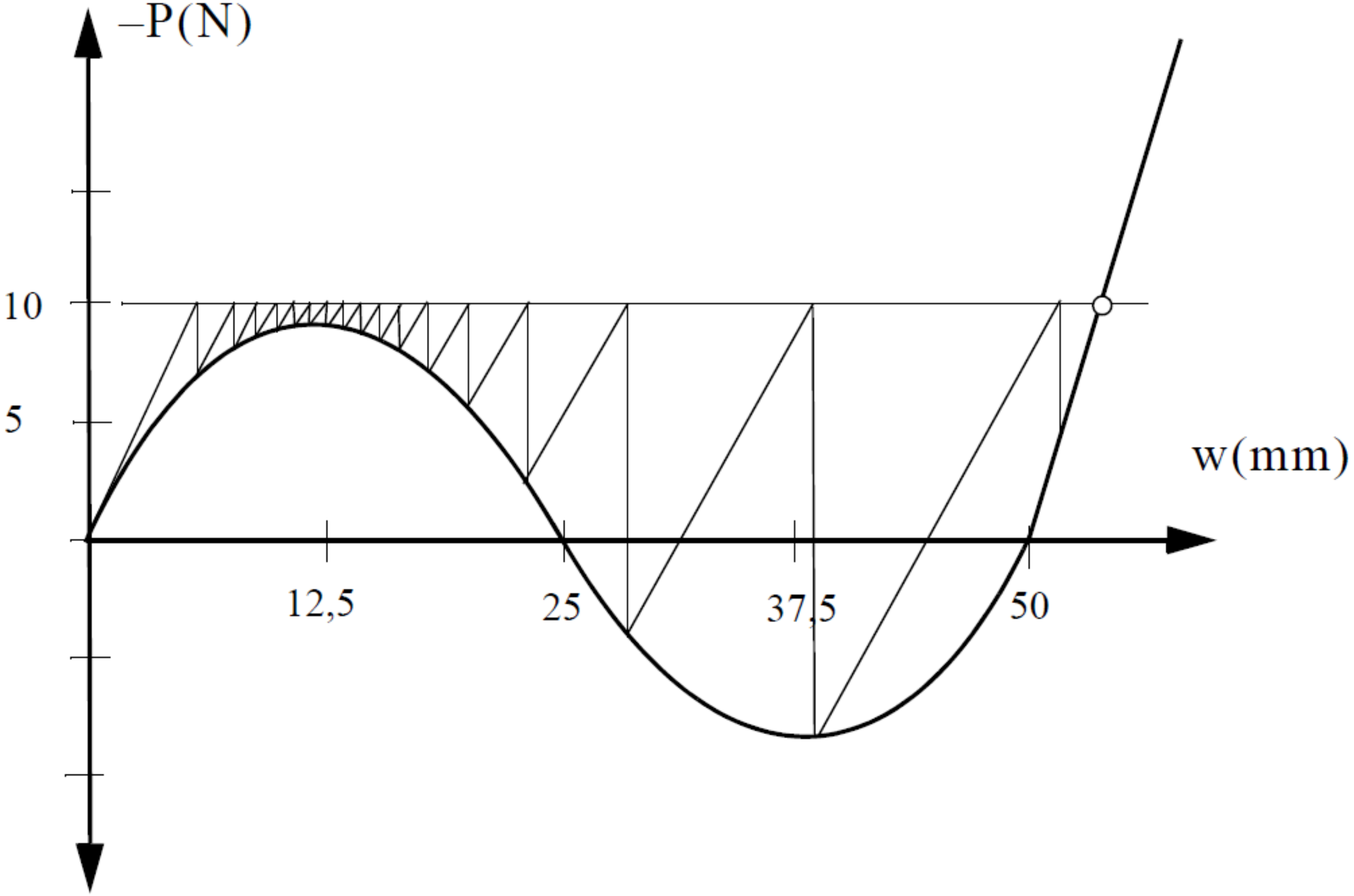


Fig. 2.6 Graphical representation of the Newton-Raphson method

Modified Newton Raphson



Modified Newton Raphson



Arc-length method

Idea: do not solve the exact load step, but solve as much as easily done and than proceed to solve the next bit…

So we have sort of automatic load – stepping (can be achieved differently too)

We choose the arc-length ∆l and converge to the load-displacement curve…

Arc-length more and more commonly used (e.g. now it is used in Plaxis)

Deals with softening well!

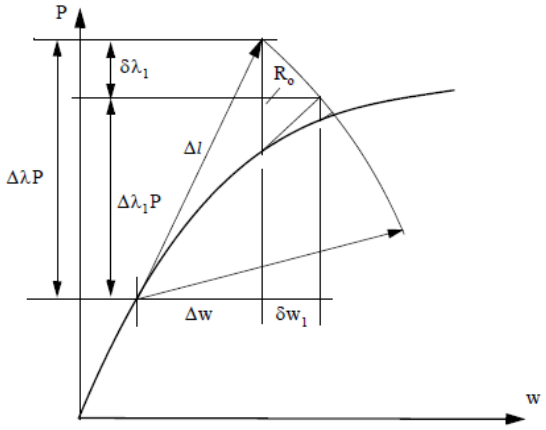


Fig. 2.8 Explanation of the arc length method

Arc-length method

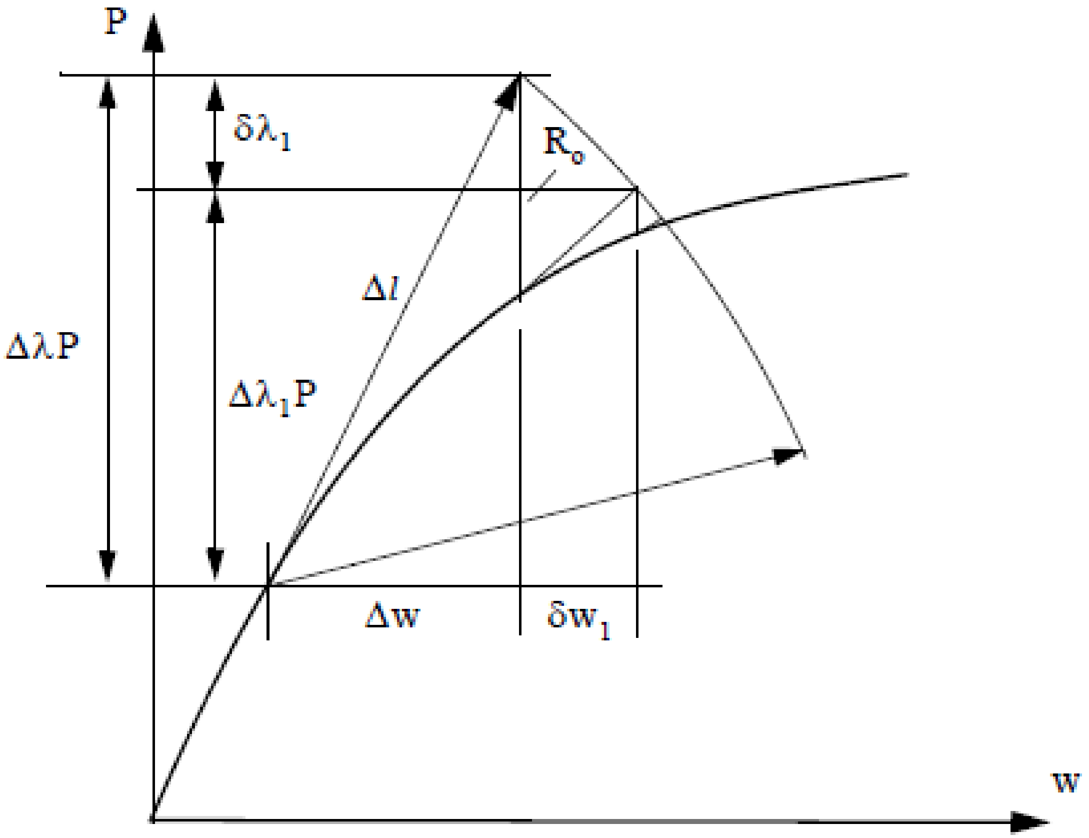


Fig. 2.8 Explanation of the arc length method

Arc-length method

$$R(w, \lambda) = -\lambda P + P_{int} = 0$$

$$\Delta l^2 = \Delta w^2 + (\Delta\lambda\psi P)^2$$

$$a = \Delta w^2 + (\Delta\lambda\psi P)^2 - \Delta l^2 = 0$$

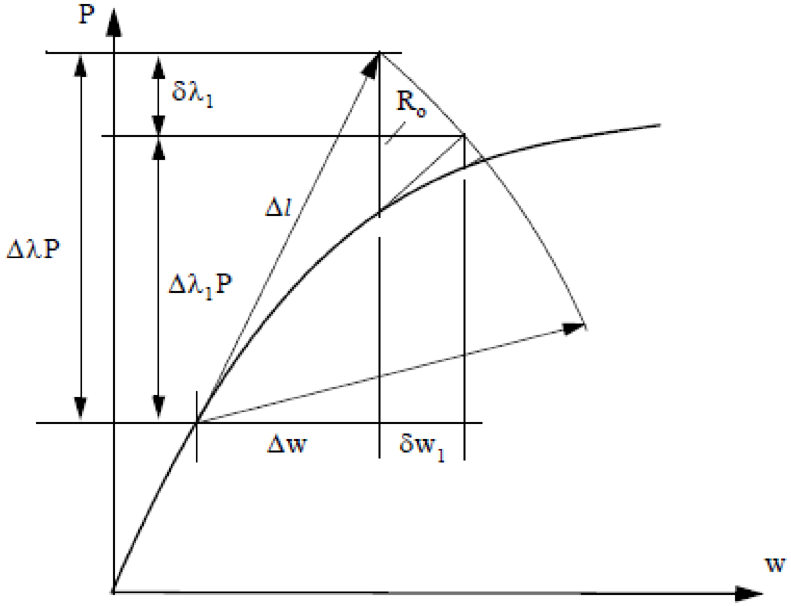


Fig. 2.8 Explanation of the arc length method

ψ - factor – as x & y axes have different scales…

Arc-length method

Using a Taylor series expansion we can write for the equilibrium equation:

$$\delta R(w, \lambda) = \frac{\partial R}{\partial w}\delta w + \frac{\partial R}{\partial \lambda}\delta\lambda = 0$$

with

$$\frac{\partial R}{\partial w} = K_T \quad , \quad \frac{\partial R}{\partial \lambda} = -P$$

$$\delta a = 2\Delta w \delta w + 2\Delta\lambda(\psi P)^2\delta\lambda = 0$$

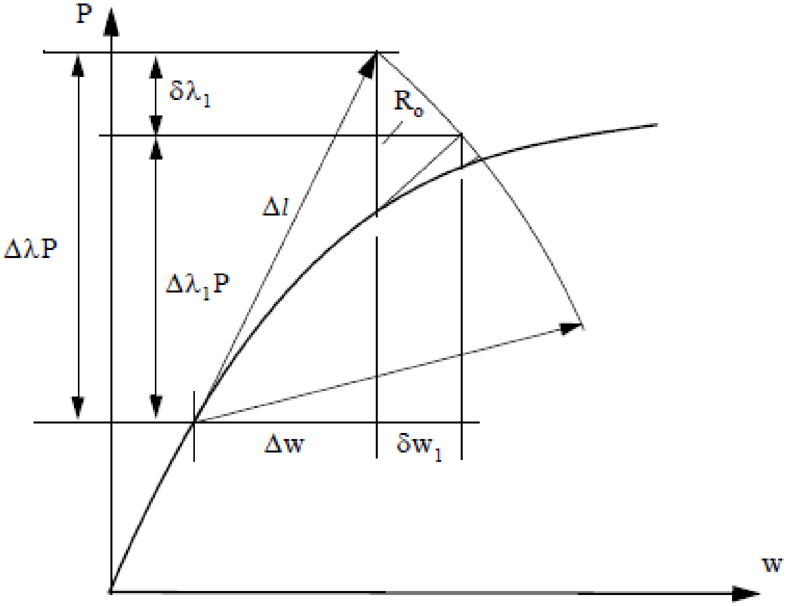


Fig. 2.8 Explanation of the arc length method

Arc-length method

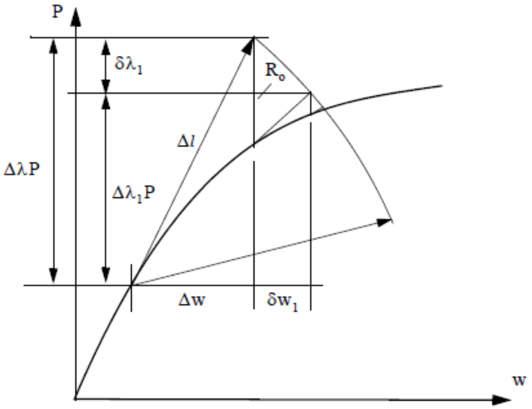


Fig. 2.8 Explanation of the arc length method

After substitution of

$$\delta R = R_n - R_o \quad , \quad \delta a = a_n - a_o$$

where the subscripts *n* and *o* mean new and old value, we obtain a system of equations.

$$\begin{bmatrix} R_o \\ a_o \end{bmatrix} = \begin{bmatrix} K_T & -P \\ 2\Delta w \ , & 2\Delta\lambda(\psi P)^2 \end{bmatrix} \begin{bmatrix} \delta w \\ \delta\lambda \end{bmatrix}$$

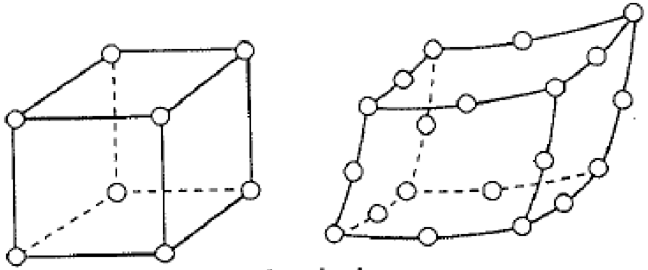
which can be solved for the change in displacement δw and change in load factor $\delta\lambda$.

Aalto University
School of Engineering

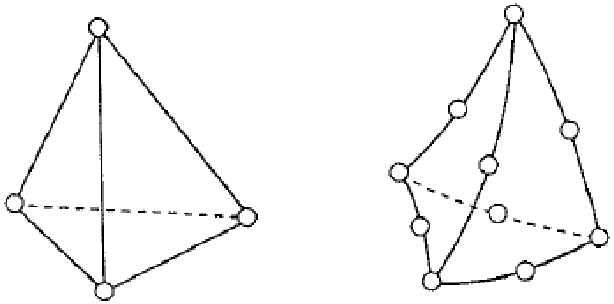
GEO – E1050
Finite Element Method in Geoengineering

3D FEM

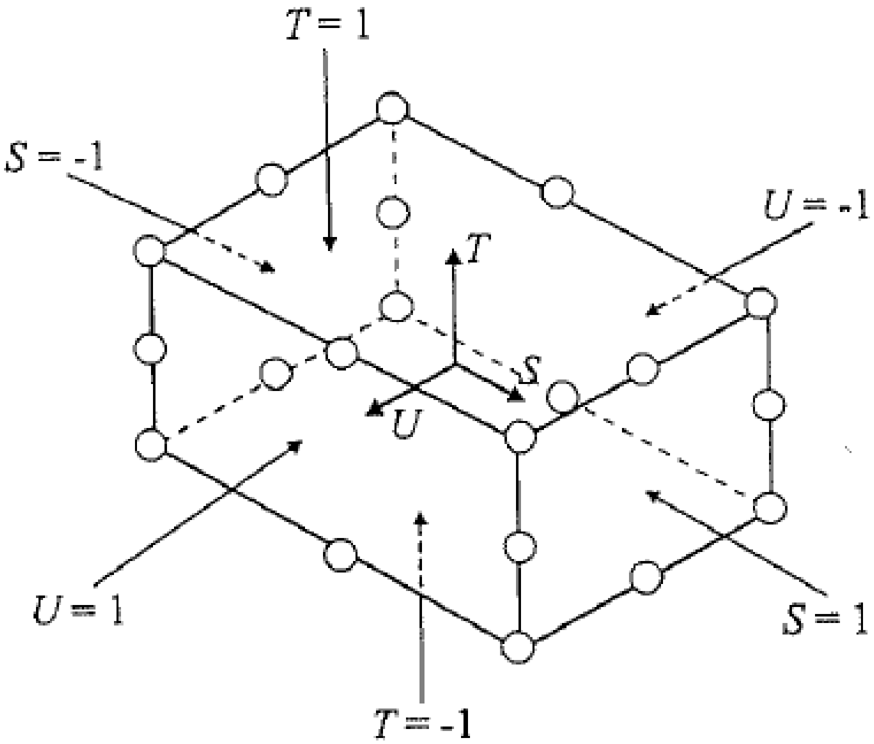
3D FEM



a) Hexahedron



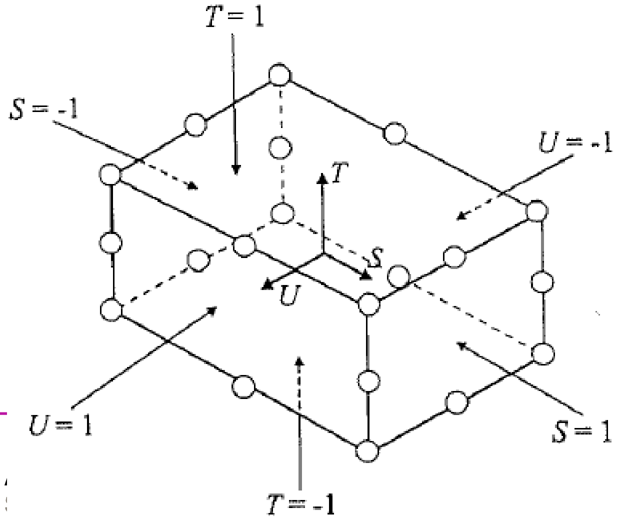
b) Tetrahedron



3D FEM

Corner nodes:

$$
\begin{aligned}
N_1 &= 1/8(1-S)(1-T)(1-U) - 1/2(N_9+N_{12}+N_{13})\\
N_2 &= 1/8(1+S)(1-T)(1-U) - 1/2(N_9+N_{10}+N_{14})\\
N_3 &= 1/8(1+S)(1+T)(1-U) - 1/2(N_{10}+N_{11}+N_{15})\\
N_4 &= 1/8(1-S)(1+T)(1-U) - 1/2(N_{11}+N_{12}+N_{16})\\
N_5 &= 1/8(1-S)(1-T)(1+U) - 1/2(N_{13}+N_{17}+N_{20})\\
N_6 &= 1/8(1+S)(1-T)(1+U) - 1/2(N_{14}+N_{17}+N_{18})\\
N_7 &= 1/8(1+S)(1+T)(1+U) - 1/2(N_{15}+N_{18}+N_{19})\\
N_8 &= 1/8(1-S)(1+T)(1+U) - 1/2(N_{16}+N_{19}+N_{20})
\end{aligned}
$$



Mid - side nodes:

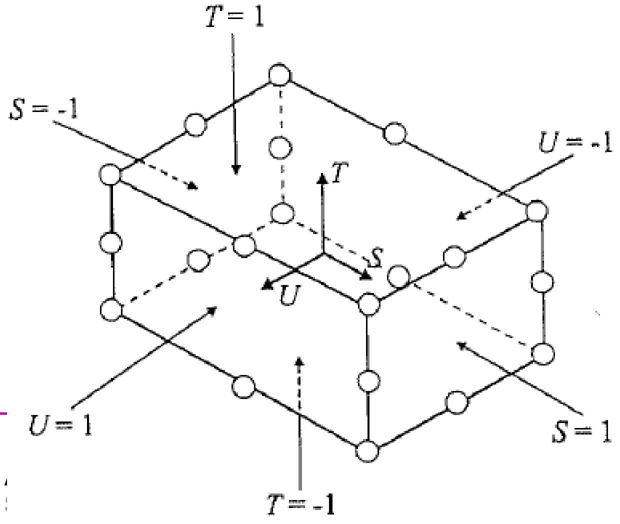
$$
\begin{aligned}
N_9 &= 1/4(1-S^2)(1-T)(1-U)\\
N_{10} &= 1/4(1-T^2)(1+S)(1-U)\\
N_{11} &= 1/4(1-S^2)(1+T)(1-U)\\
N_{12} &= 1/4(1-T^2)(1-S)(1-U)\\
N_{13} &= 1/4(1-U^2)(1-S)(1-T)\\
N_{14} &= 1/4(1-U^2)(1+S)(1-T)\\
N_{15} &= 1/4(1-U^2)(1+S)(1+T)\\
N_{16} &= 1/4(1-U^2)(1-S)(1+T)\\
N_{17} &= 1/4(1-S^2)(1-T)(1+U)\\
N_{18} &= 1/4(1-T^2)(1+S)(1+U)\\
N_{19} &= 1/4(1-S^2)(1+T)(1+U)\\
N_{20} &= 1/4(1-T^2)(1-S)(1+U)
\end{aligned}
$$

$$
\begin{Bmatrix} \Delta u \\ \Delta v \\ \Delta w \end{Bmatrix} = [N] \begin{Bmatrix} \Delta u \\ \Delta v \\ \Delta w \end{Bmatrix}_{nodes}
$$

3D FEM

Corner nodes:

$$\begin{aligned}
N_1 &= \tfrac{1}{8}(1-S)(1-T)(1-U) - \tfrac{1}{2}(N_9+N_{12}+N_{13}) \\
N_2 &= \tfrac{1}{8}(1+S)(1-T)(1-U) - \tfrac{1}{2}(N_9+N_{10}+N_{14}) \\
N_3 &= \tfrac{1}{8}(1+S)(1+T)(1-U) - \tfrac{1}{2}(N_{10}+N_{11}+N_{15}) \\
N_4 &= \tfrac{1}{8}(1-S)(1+T)(1-U) - \tfrac{1}{2}(N_{11}+N_{12}+N_{16}) \\
N_5 &= \tfrac{1}{8}(1-S)(1-T)(1+U) - \tfrac{1}{2}(N_{13}+N_{17}+N_{20}) \\
N_6 &= \tfrac{1}{8}(1+S)(1-T)(1+U) - \tfrac{1}{2}(N_{14}+N_{17}+N_{18}) \\
N_7 &= \tfrac{1}{8}(1+S)(1+T)(1+U) - \tfrac{1}{2}(N_{15}+N_{18}+N_{19}) \\
N_8 &= \tfrac{1}{8}(1-S)(1+T)(1+U) - \tfrac{1}{2}(N_{16}+N_{19}+N_{20})
\end{aligned}$$



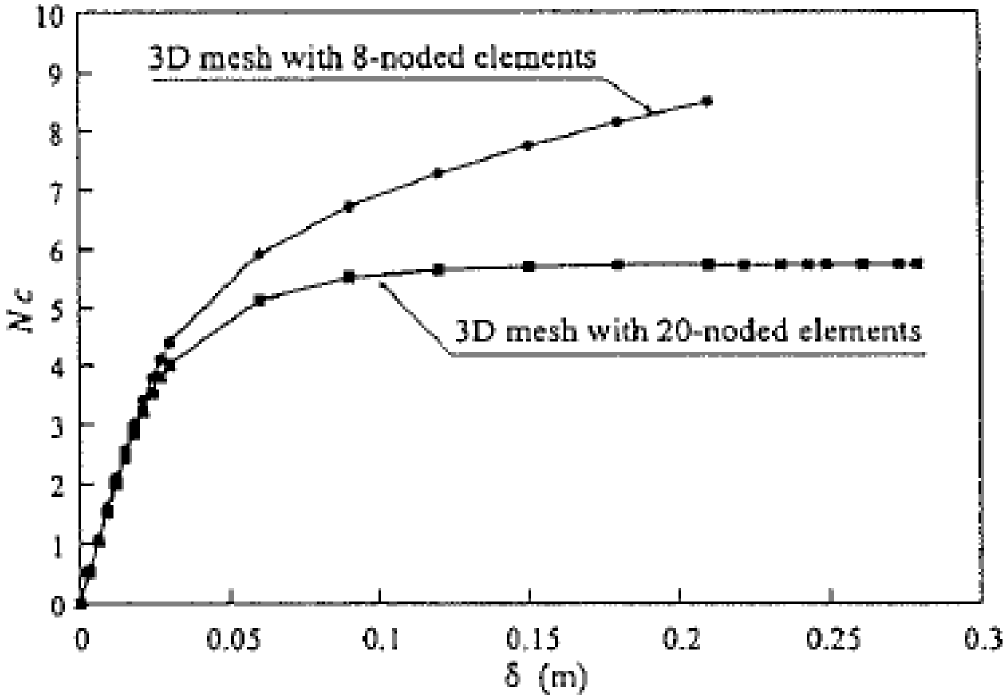
Mid - side nodes:

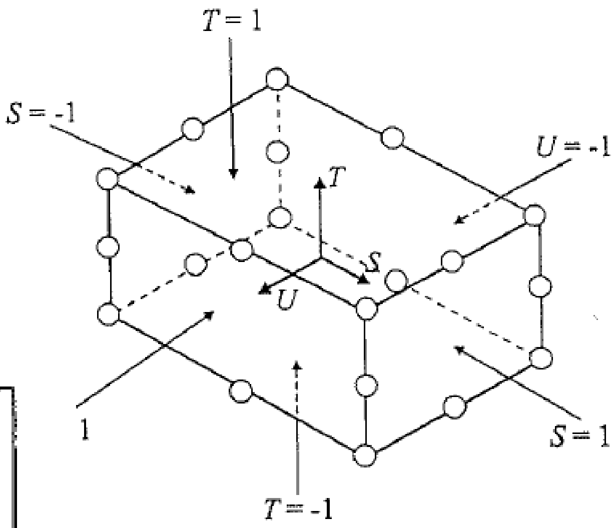
$$\begin{aligned}
N_9 &= \tfrac{1}{4}(1-S^2)(1-T)(1-U) \\
N_{10} &= \tfrac{1}{4}(1-T^2)(1+S)(1-U) \\
N_{11} &= \tfrac{1}{4}(1-S^2)(1+T)(1-U) \\
N_{12} &= \tfrac{1}{4}(1-T^2)(1-S)(1-U) \\
N_{13} &= \tfrac{1}{4}(1-U^2)(1-S)(1-T) \\
N_{14} &= \tfrac{1}{4}(1-U^2)(1+S)(1-T) \\
N_{15} &= \tfrac{1}{4}(1-U^2)(1+S)(1+T) \\
N_{16} &= \tfrac{1}{4}(1-U^2)(1-S)(1+T) \\
N_{17} &= \tfrac{1}{4}(1-S^2)(1-T)(1+U) \\
N_{18} &= \tfrac{1}{4}(1-T^2)(1+S)(1+U) \\
N_{19} &= \tfrac{1}{4}(1-S^2)(1+T)(1+U) \\
N_{20} &= \tfrac{1}{4}(1-T^2)(1-S)(1+U)
\end{aligned}$$

$$\begin{Bmatrix} \Delta u \\ \Delta v \\ \Delta w \end{Bmatrix} = [N] \begin{Bmatrix} \Delta u \\ \Delta v \\ \Delta w \end{Bmatrix}_{nodes}$$

3D FEM

$$[K_G]\{\Delta d\}_{nG} = \{\Delta R_G\}$$





Global matrix K_G can be very big, thus usually iterative solutions are used, not direct inverse; direct inversion still may be quicker for highly non-linear problems

Aalto University
School of Engineering

GEO – E1050
Finite Element Method in Geoengineering

Geometric non-linearity

Geometric non-linearity

1.3.1 Rigid bar with rotational spring - vertical load

The structural system in Fig. 1.4 consists of a rigid bar and a spring of stiffness k and is loaded by a vertical force at the end. It is clear that as the deflection at the end increases, the geometry change during loading can no longer be neglected.

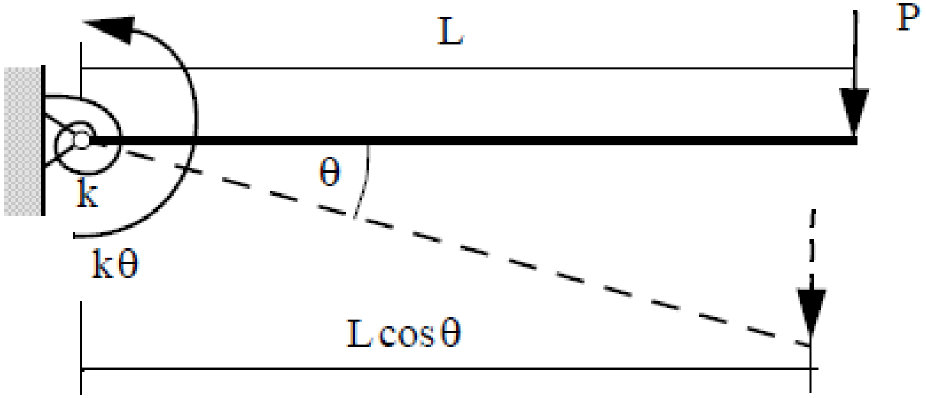


Fig. 1.4 Example of geometrically nonlinear bahviour

The Equilibrium condition is written as:

$$PL\cos\theta = k\theta$$

Geometric non-linearity

The relationship between applied load and rotation is

$$P = \frac{k\theta}{L\cos\theta}$$

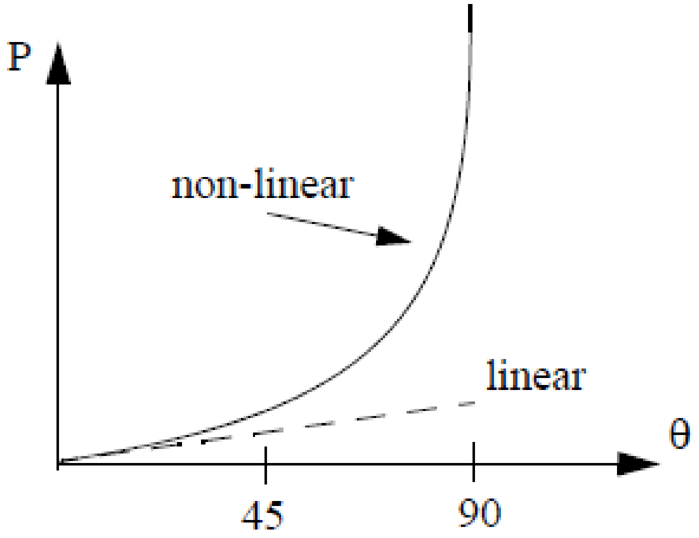


Fig. 1.5 Linear and non-linear equilibrium paths of the spring beam system

Geometric non-linearity

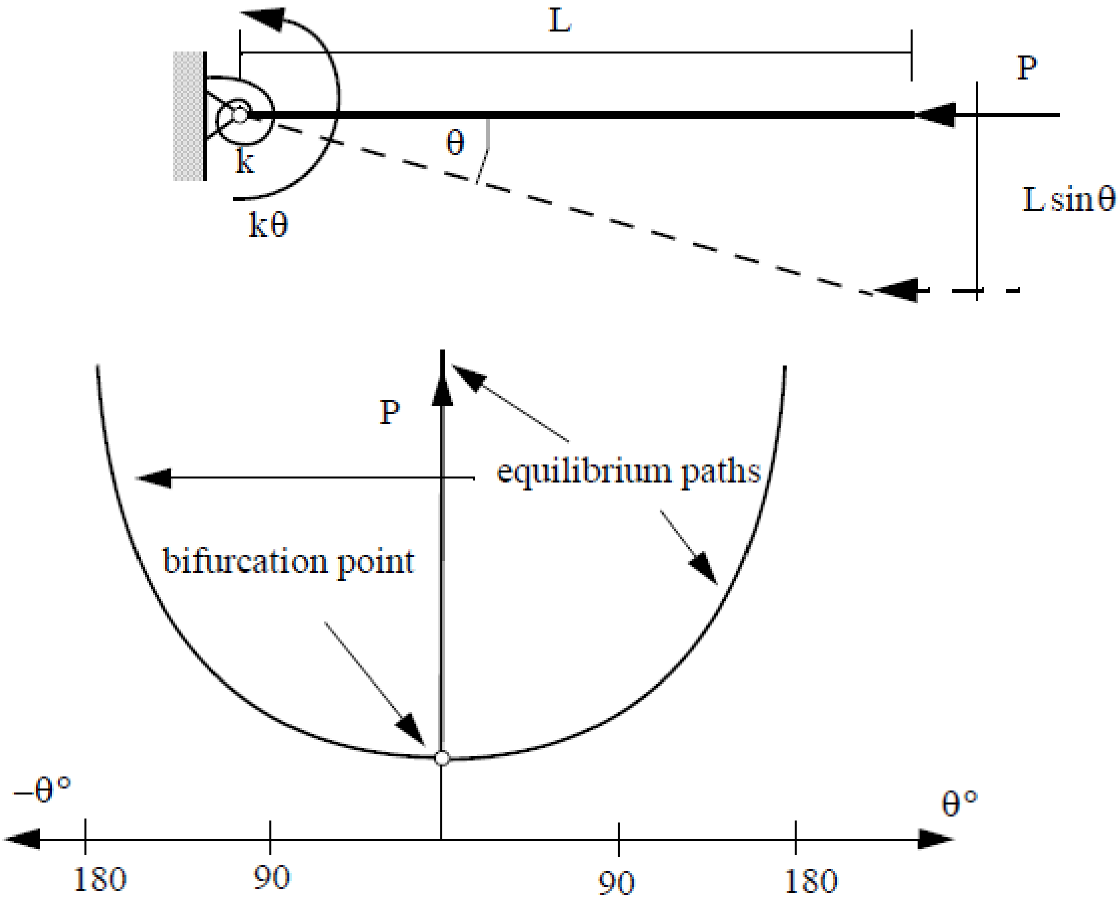


Fig. 1.7 Equilibrium paths for beam-spring example with horiz. loading

Aalto University
School of Engineering

GEO – E1050
Errors and error control

To learn today…

How automatic re-meshing works

- **elasticity**
- **elasto-plasticity**

Limitations and pitfalls in numerical analyses:

- discretisation errors
- boundary conditions
- selection of elements
- contact detection / no contact
- input parameters consistency

How to avoid errors…

Automatic re-meshing

Linear Elasticity:

- **we discretise the domain into elements**
- **internal work = external work**

$$dE = 0.5\int_V d\boldsymbol{\varepsilon}^T d\boldsymbol{\sigma} dV \qquad dL = \int_S d\mathbf{d}_E^T \mathbf{T} dS = d\mathbf{d}_E^T \Delta\mathbf{R}$$

1) **Error minimised when internal work in each element is the same**… So we reduce / increase the size of elements as long as we get constant internal work…
2) Alternative: each element has same average strain

In an "optimal mesh" it is desirable that the distribution of element energy norm error should be equal for all elements.

Automatic re-meshing

Elasto - plasticity

$$dE = 0.5\int_V d\boldsymbol{\varepsilon}^T d\boldsymbol{\sigma} dV$$

Error minimised when internal work in each element is the same… So we reduce / increase the size of elements as long as we get constant internal work…

we need to take into account plastic work

AND WE FIND THAT THE IDEA DOES NOT ALWAYS WORK

> **why**: element with very high elastic stress with little plastic strain will lead to higher work than an element with lots of plasticity and deformations, but low stress

so – error related to the amount of work – but also to more general deformations and plastic deformation

Automatic re-meshing elasto - plasticity

Options:

- shear dissipation

+ assumes that most errors and displacements in soils are in the shear zones

+ / - related to plastic work

- poor if material has points with high stress and little shearing and actually errors are somewhere else

- superb for limit analysis as in limit analysis the only dissipation is shear dissipation

$$D_s = (\boldsymbol{\sigma} - \mathbf{m}p)^{\mathrm{T}}(\varepsilon - \mathbf{m}\varepsilon_v)$$

Automatic re-meshing elasto - plasticity

Options:

- total dissipation
  - + / - related to plastic work
  - - poor if material has points with high stress and little shearing and actually errors are somewhere else

- not well suitable for materials where we have really dissipation due to shearing (e.g. Mohr – Coulomb)

$$D = \boldsymbol{\sigma}^{\mathsf{T}} \varepsilon.$$

Automatic re-meshing elasto - plasticity

Options:

- Total strain $||\varepsilon||$
  - not related to plastic work
  - can be useful, but we forgo idea of work totally…

- Plastic multiplier λ
  - related to plastic strain
  - similarly, we let go the idea of work…

Limitations & pitfalls

We will concentrate on issues related to numerical simulation mostly

Observational error & conceptual models are all the same in all design

(though in FEM it is especially tempting to make complex conceptual model)

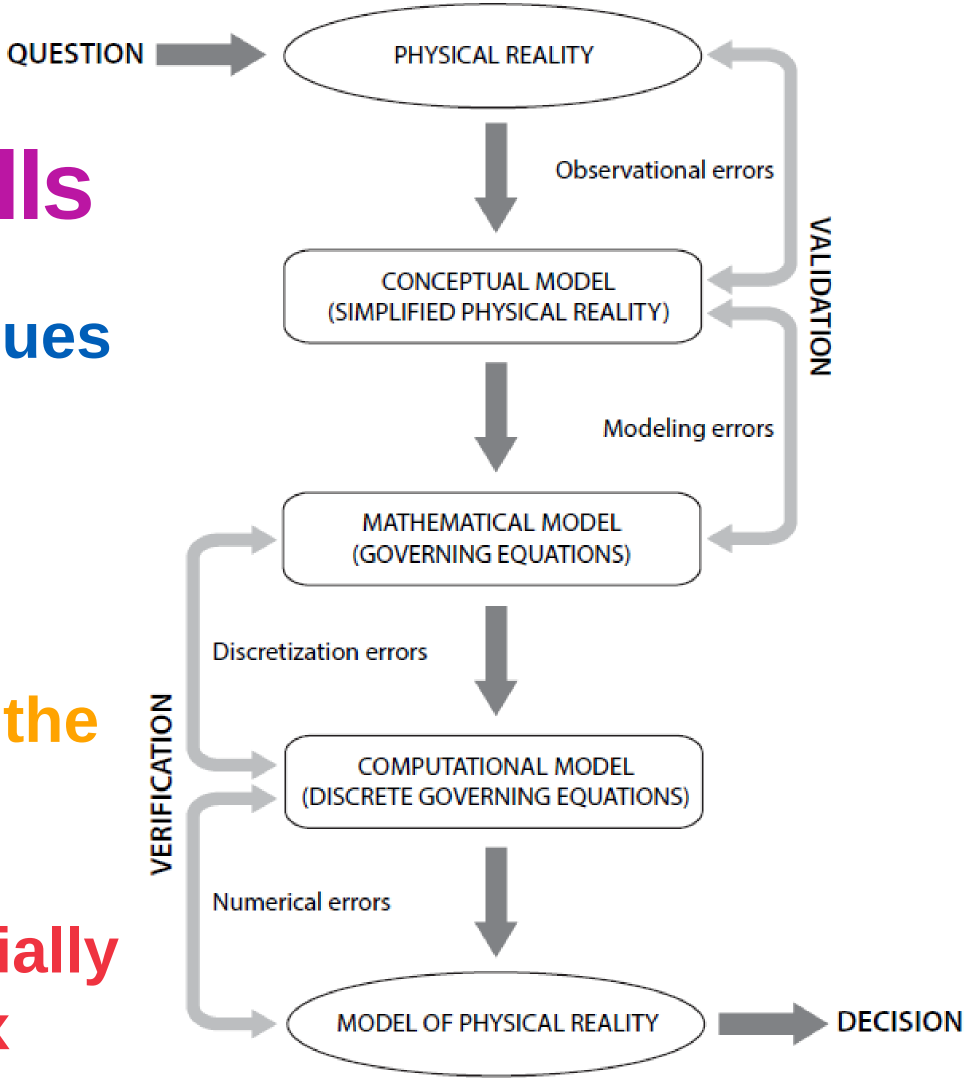


Figure 15.1: From question to decision using numerical analysis.

Limitations & pitfalls

Mathematical model:

- constitutive model for material – THE issue of
Advanced Soil Mechanics course

- other errors – model for water flow, thermal conduction
- the differential equations we discretise have rather limited intended area of use
    - generally related to very slow processes which are steady state
- Correct choice of parameters for them is **tough**

Numerical Methods in Geotechnics course

Limitations & pitfalls

Discretization errors

wrong input data – we just defined the problem correctly

Usually we can see it during input…

If not, we can see it when checking results

But sometimes things are difficult to detect

Especially when we do not have clear idea what the results should be, both qualitatively and quantitatively (roughly)

Limitations & pitfalls

Mars Orbiter
lost (1999)

Discretization errors: wrong input

units… kPa vs MPa…

SI vs imperial

zero level definition

element defined twice in the same place
- especially easy for 1D elements in 2D simulation etc.

Laufenburg
bridge (2003)

wrong material assigned

Countless errors, in FEM and outside of FEM

Limitations & pitfalls

Discretization errors: element selection

Element order:

tempting to lower it because simulation ‘works’
negative jacobian errors, shape distortion…

Limitations & pitfalls

Discretization errors: element selection

Different type of elements in the same simulation (now rarely allowed, apart of 1D elements)

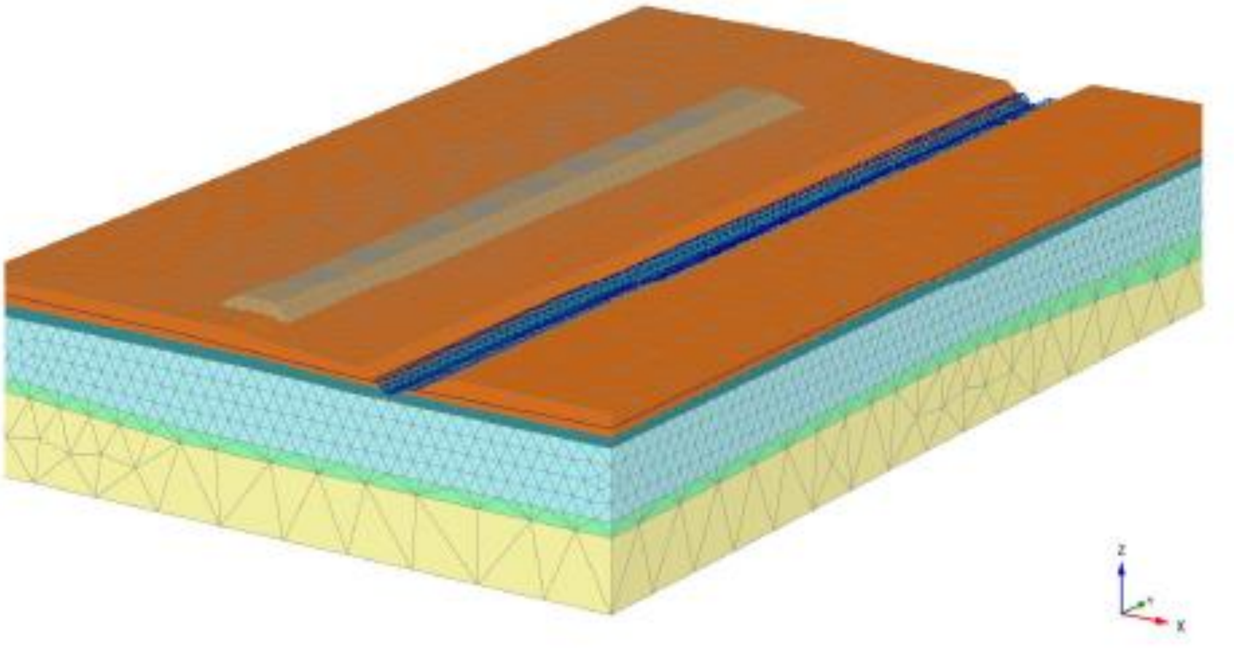
In very large models, useful to check parts of them for correctness…

Limitations & pitfalls

Discretization errors: element selection

Quality of mesh may lead to ‘game breaking errors’ in 3D

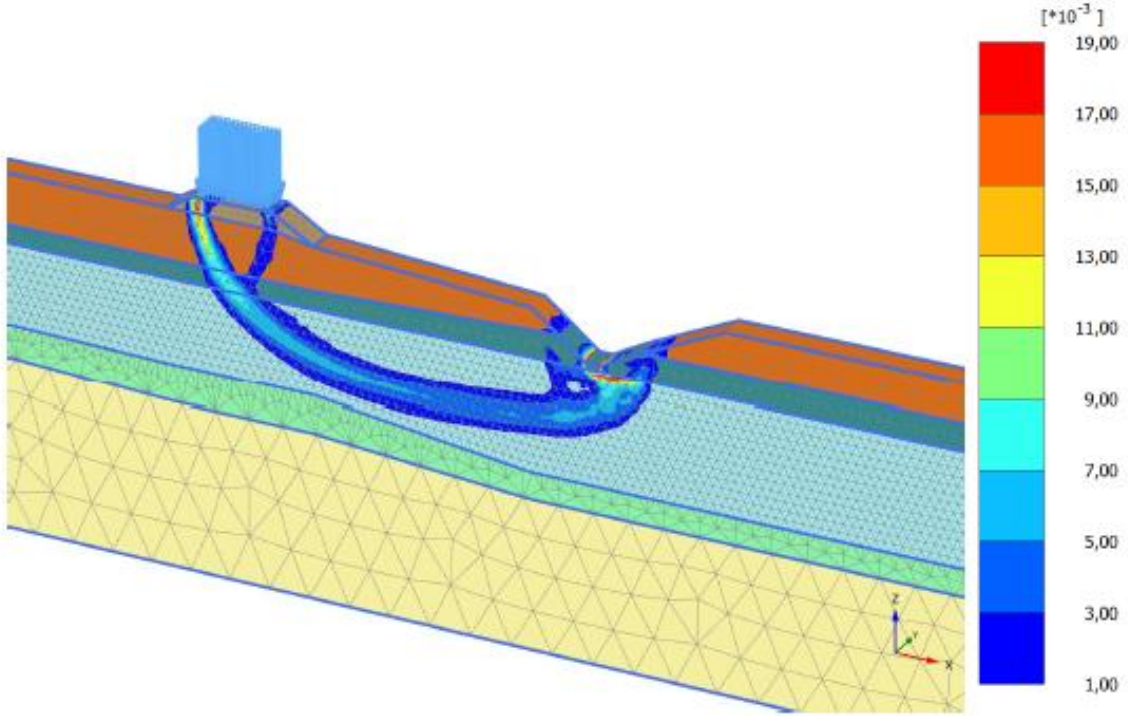
Limitations & pitfalls



© Mansikkamäki 2015

3D geometry, 240 000 element (10 node tetrahedral)

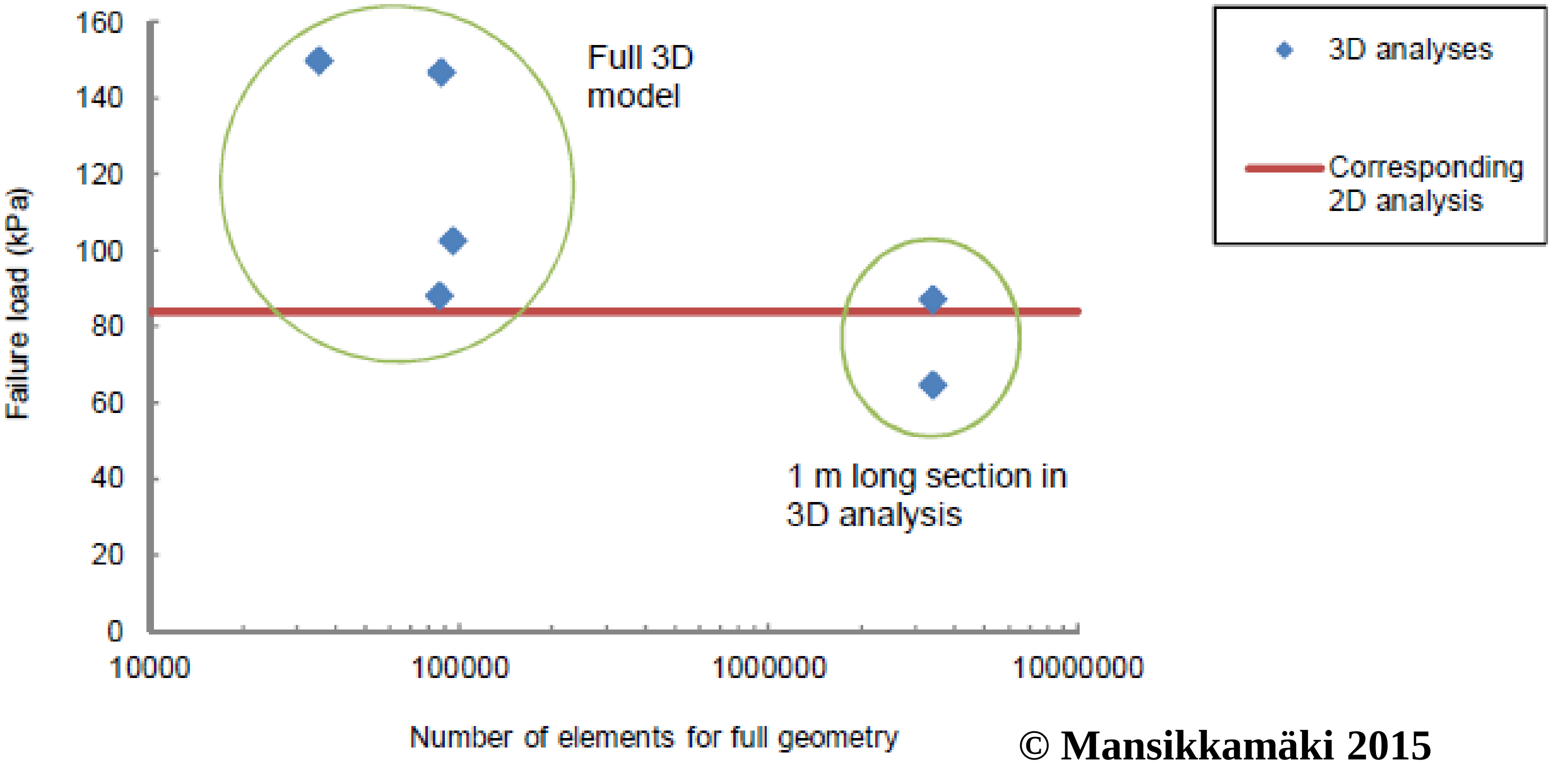
Limitations & pitfalls



© Mansikkamäki 2015

1 m long section of an embankment… More elements, same 10 node tetrahedral

Limitations & pitfalls



© Mansikkamäki 2015

For smaller models – differences due to convergence criteria set (remember your Comsol exercise)

Limitations & pitfalls

Discretization errors: contact

Contact is expensive, does not always work

But otherwise elements not in contact initially will not be in contact at any point in analysis

No easy solution – but always check when in doubt

Divide load into two – before contact occurs
... and after contact occurs...

e.g. apply corrective loads to the right elements...

What to do...

Check, check and check again

Are the results looking all right?

- stress concentrations where they should occur?
- **stress, strain, displacement fields continous?**
- no areas which may be in contact?
- sharp edges in elasticity or Poisson equation problems (steady flow, thermal conduction, diffusion, ...)
  - if so, maybe needed small rounding

What to do...

Check, check and check again

Are the results are **quantitatively sensible?**

- values similar to those from hand made calculations?
- no outragous large or too small values?
- you only do calculations to confirm what you know and make sure that you are not wrong, but if the **expectations** and **calculations** do not match, it is a **red light**

What to do...

Check, check and check again

Are the **numerics correct?**

- elastic energy > 0?

- error norms sensible? Less than say 5%?

- if you make better mesh and recalculate, are the results only **negligibly different?**

What to do...

Check, check and check again

When you are experienced FEM modeller, you can calculate results which are close to reality...

But you can easily fudge the calculations so it will be invisible to most, in order to get any results your boss wants

- especially easy in 3D
- be careful...
- don’t yield to pressure...

Aalto University
School of Engineering

GEO – E1050
Finite Element Method in Geoengineering

Dynamic problems

Dynamic problems: require time in the equation…

We add time dependent parts to the equation:

$$\mathbf{M}\ddot{\tilde{\mathbf{u}}} + \mathbf{C}\dot{\tilde{\mathbf{u}}} + \mathbf{K}\tilde{\mathbf{u}} - \mathbf{f} = \mathbf{0}$$

$$\dot{\tilde{\mathbf{u}}} = \frac{\mathrm{d}\tilde{\mathbf{u}}}{\mathrm{d}t} \quad \text{and} \quad \ddot{\tilde{\mathbf{u}}} = \frac{\mathrm{d}^2\tilde{\mathbf{u}}}{\mathrm{d}t^2}$$

Dynamic problems…

We add time dependent parts to the equation:

$$\mathbf{M}\ddot{\tilde{\mathbf{u}}} + \mathbf{C}\dot{\tilde{\mathbf{u}}} + \mathbf{K}\tilde{\mathbf{u}} - \mathbf{f} = \mathbf{0}$$

$$\dot{\tilde{\mathbf{u}}} = \frac{\mathrm{d}\tilde{\mathbf{u}}}{\mathrm{d}t} \qquad \text{and} \qquad \ddot{\tilde{\mathbf{u}}} = \frac{\mathrm{d}^2\tilde{\mathbf{u}}}{\mathrm{d}t^2}$$

$$\mathbf{K}^e = \mathbf{H}^e = \int_{\Omega_e} \mathbf{b}^T \mathbf{k} \mathbf{b} \, \mathrm{d}\Omega + \int_{\Gamma_{qe}} \mathbf{N}^T H \mathbf{N} \, \mathrm{d}\Gamma$$

$$\mathbf{C}^e = \int_{\Omega_e} \mathbf{N}^T c \mathbf{N} \, \mathrm{d}\Omega$$

$$\mathbf{M}^e = \int_{\Omega_e} \mathbf{N}^T \rho \mathbf{N} \, \mathrm{d}\Omega$$

$$\mathbf{f}^e = \mathbf{s}^e = \int_{\Omega_e} \mathbf{N}^{\mathrm{T}} Q \, \mathrm{d}\Omega + \int_{\Gamma_{qe}} \mathbf{N}^T (\bar{q} - H\phi_0) \, \mathrm{d}\Gamma$$

Dynamic problems: require time…

We add inertial parts to the equation:

$$\mathbf{K}^e = \mathbf{H}^e = \int_{\Omega_e} \mathbf{b}^T \mathbf{k} \mathbf{b} \, \mathrm{d}\Omega + \int_{\Gamma_{qe}} \mathbf{N}^T H \mathbf{N} \, \mathrm{d}\Gamma$$

$$\mathbf{C}^e = \int_{\Omega_e} \mathbf{N}^T c \mathbf{N} \, \mathrm{d}\Omega \longrightarrow$$

viscous forces

$$\mathbf{M}^e = \int_{\Omega_e} \mathbf{N}^T \rho \mathbf{N} \, \mathrm{d}\Omega \longrightarrow$$

inertial forces

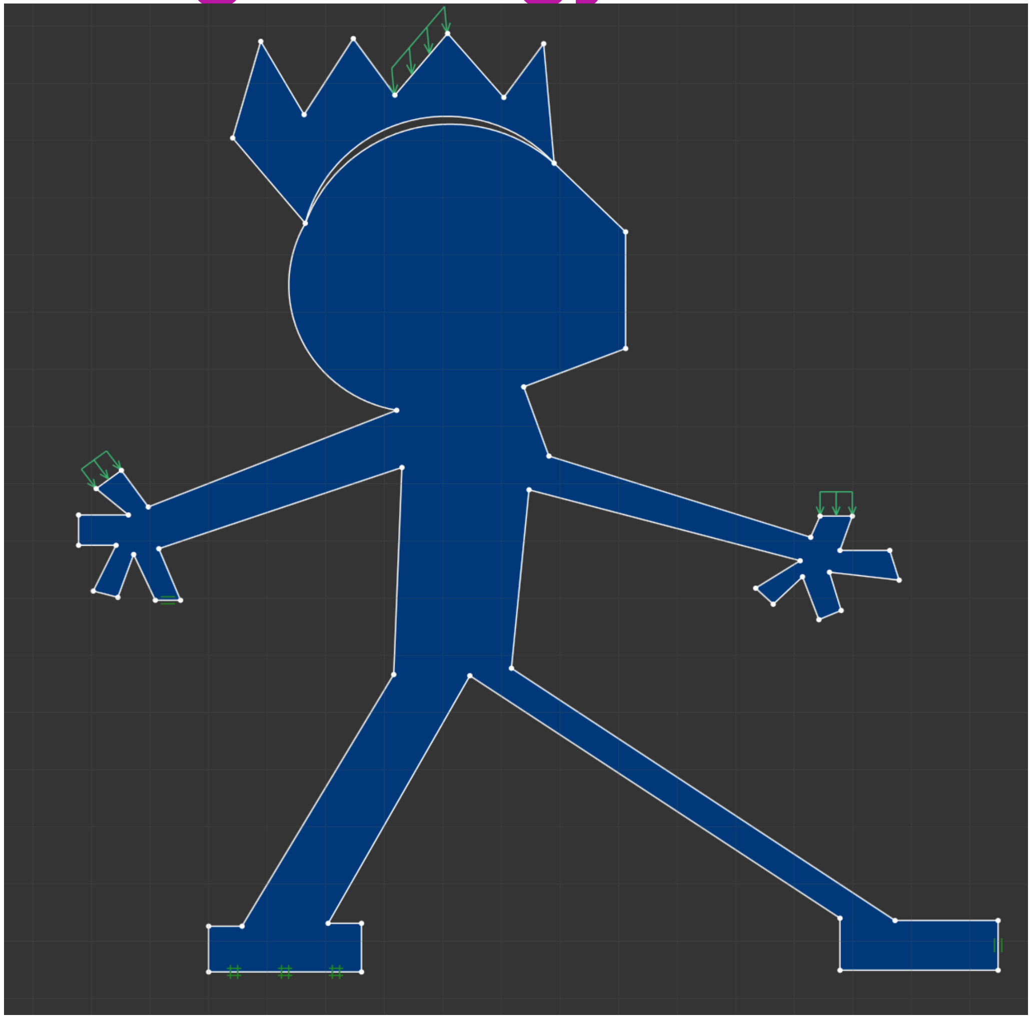
$$\mathbf{f}^e = \mathbf{s}^e = \int_{\Omega_e} \mathbf{N}^{\mathrm{T}} Q \, \mathrm{d}\Omega + \int_{\Gamma_{qe}} \mathbf{N}^T (\bar{q} - H\phi_0) \, \mathrm{d}\Gamma$$

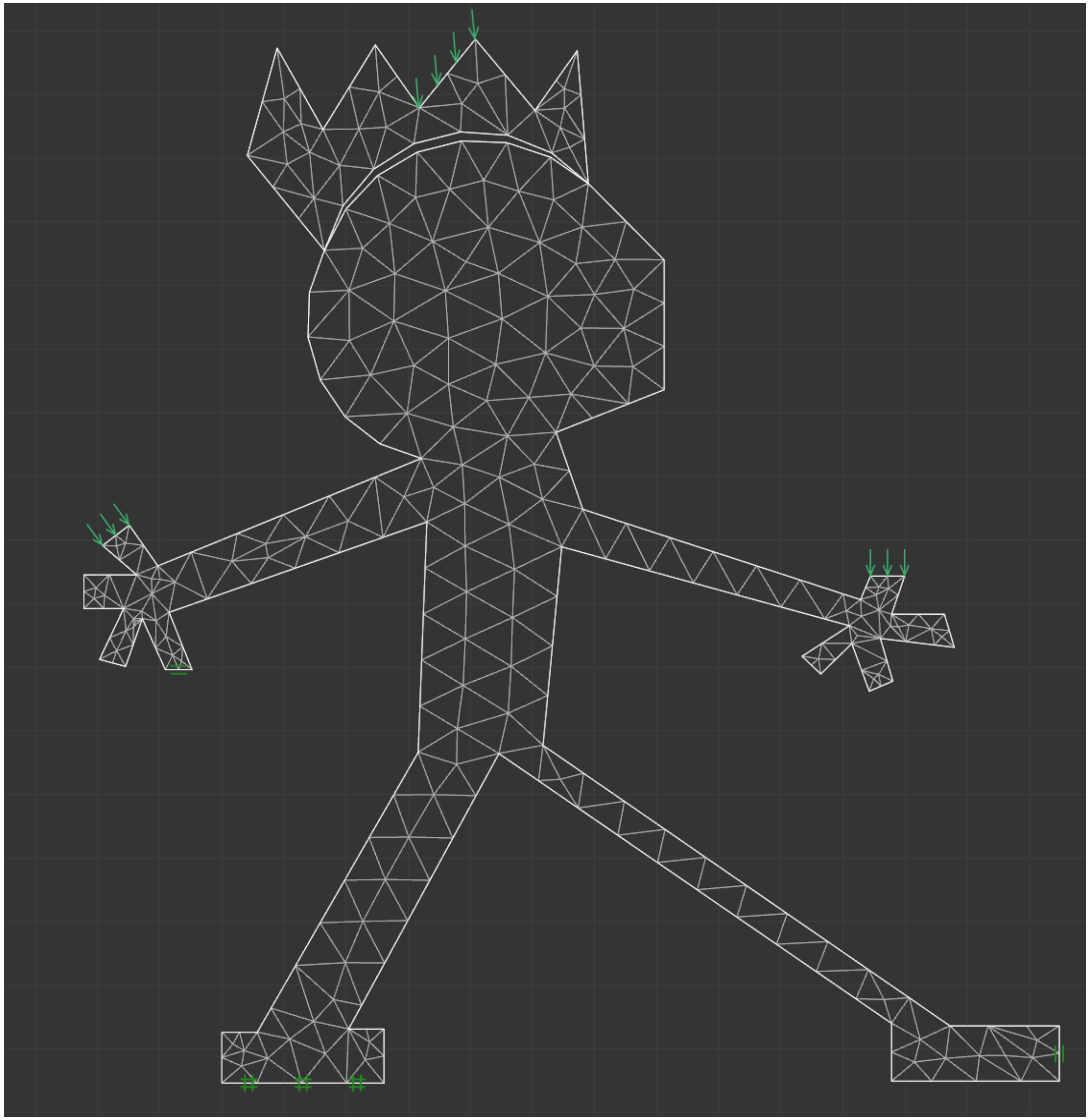
Aalto University
School of Engineering

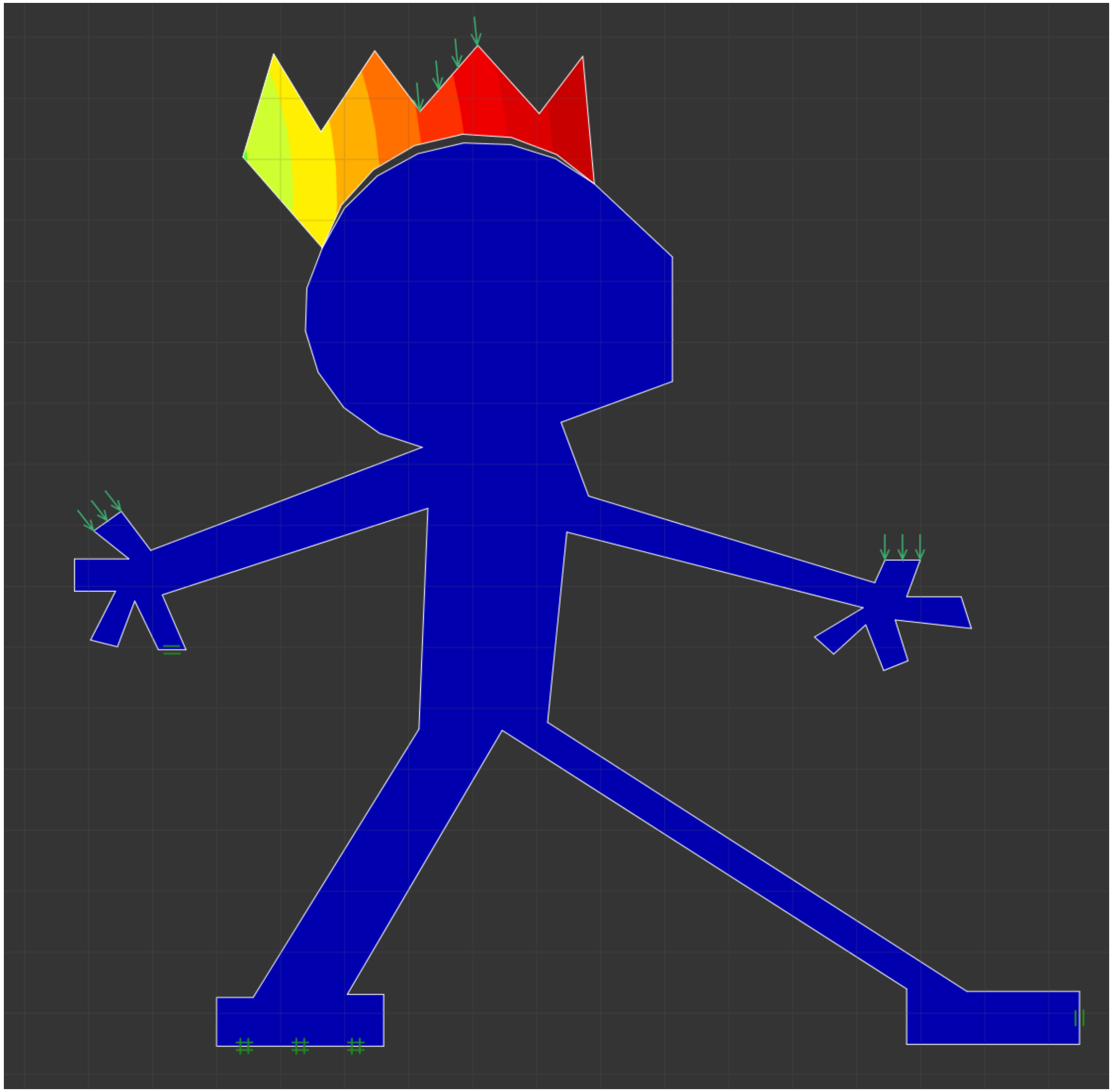
GEO – E1050
Finite Element Method in Geoengineering

Problem solving strategy

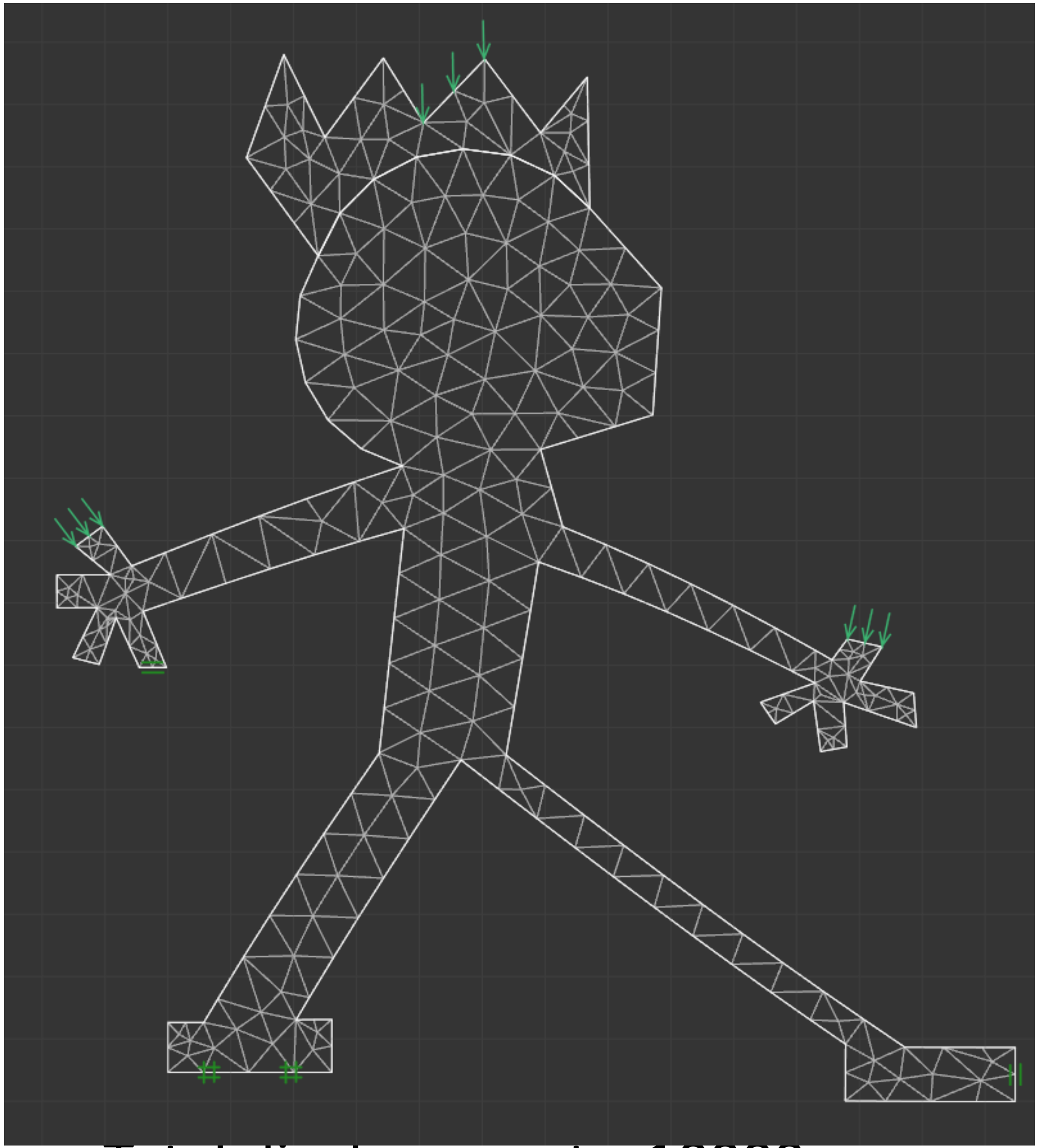
Problem solving strategy



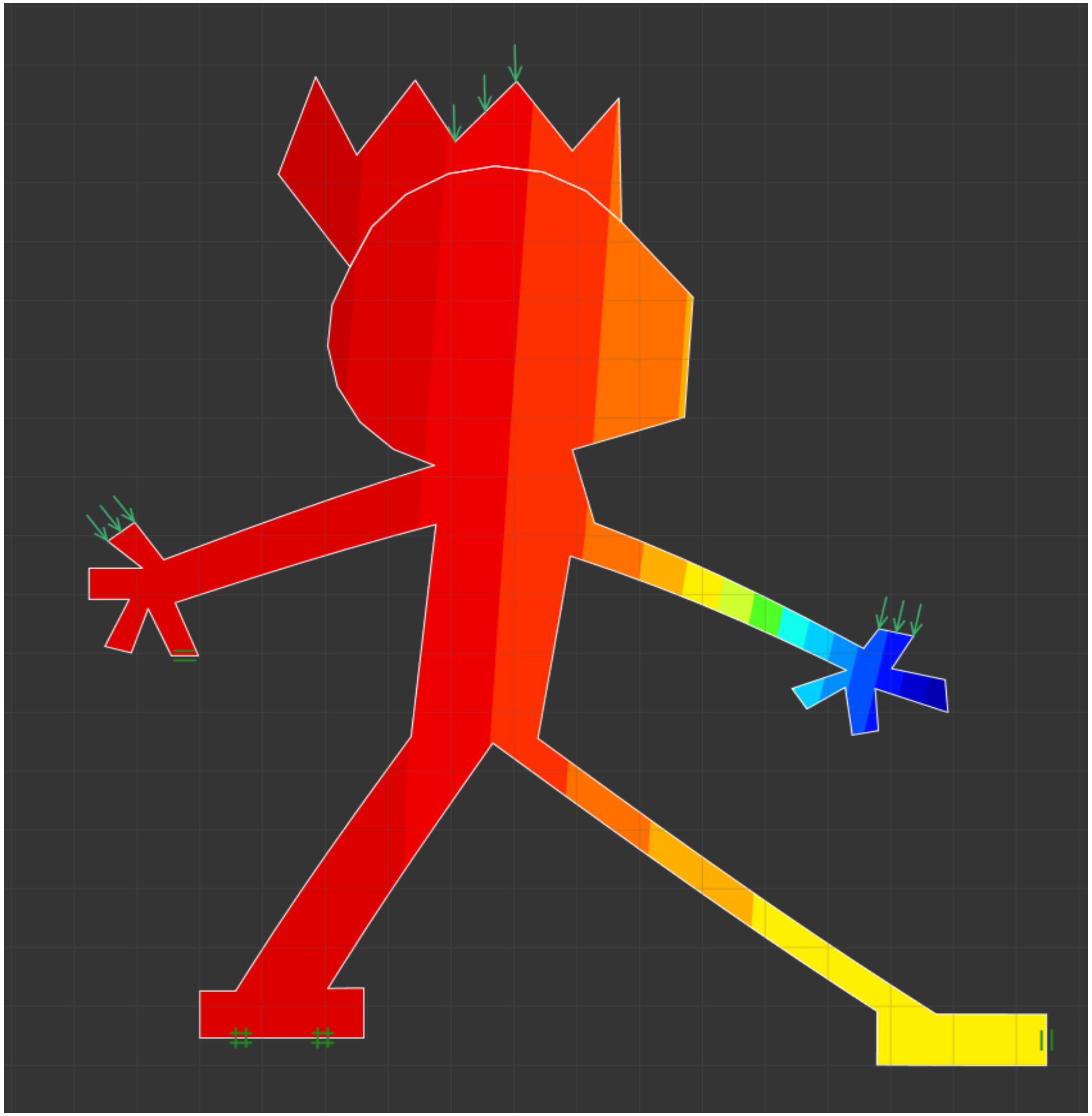




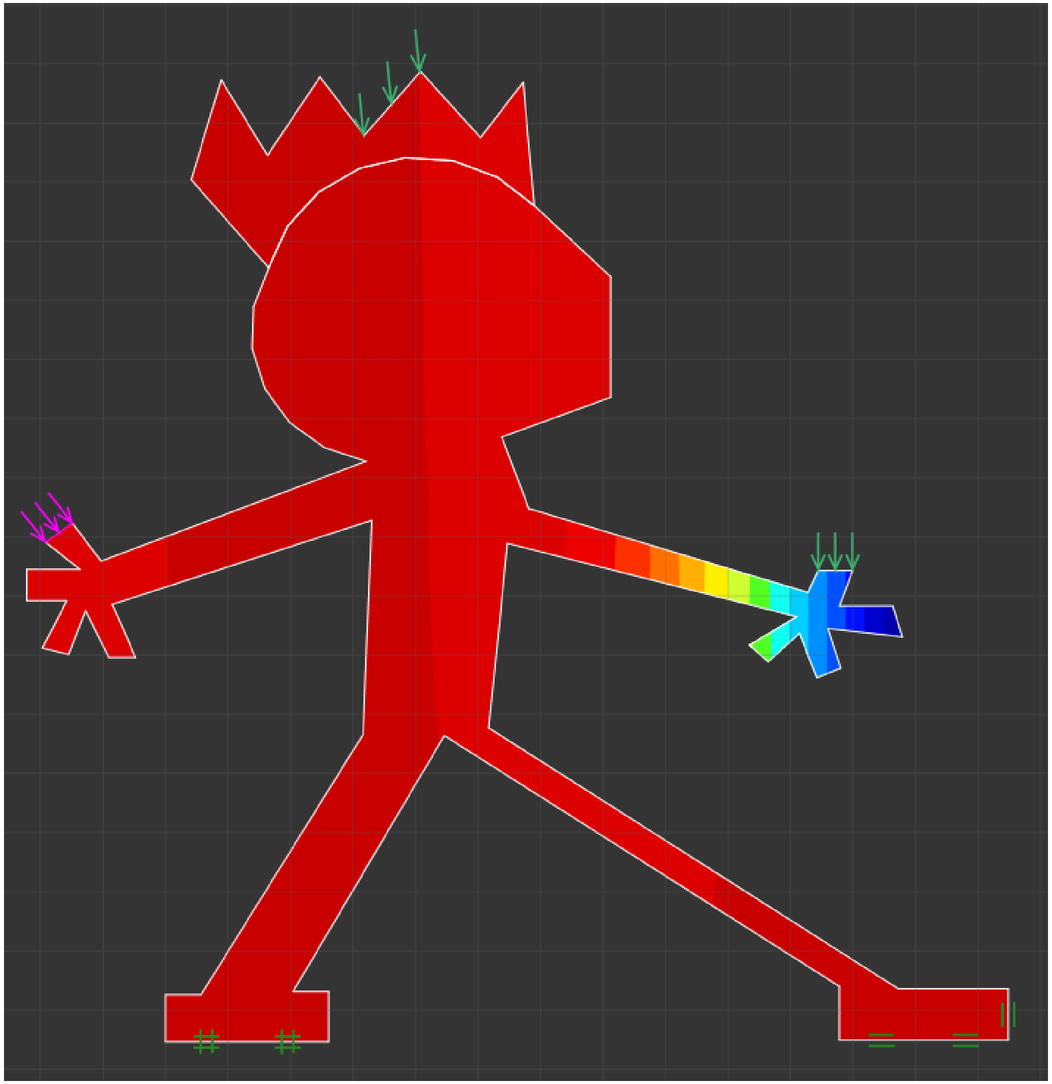
Total displacements: 10000 m



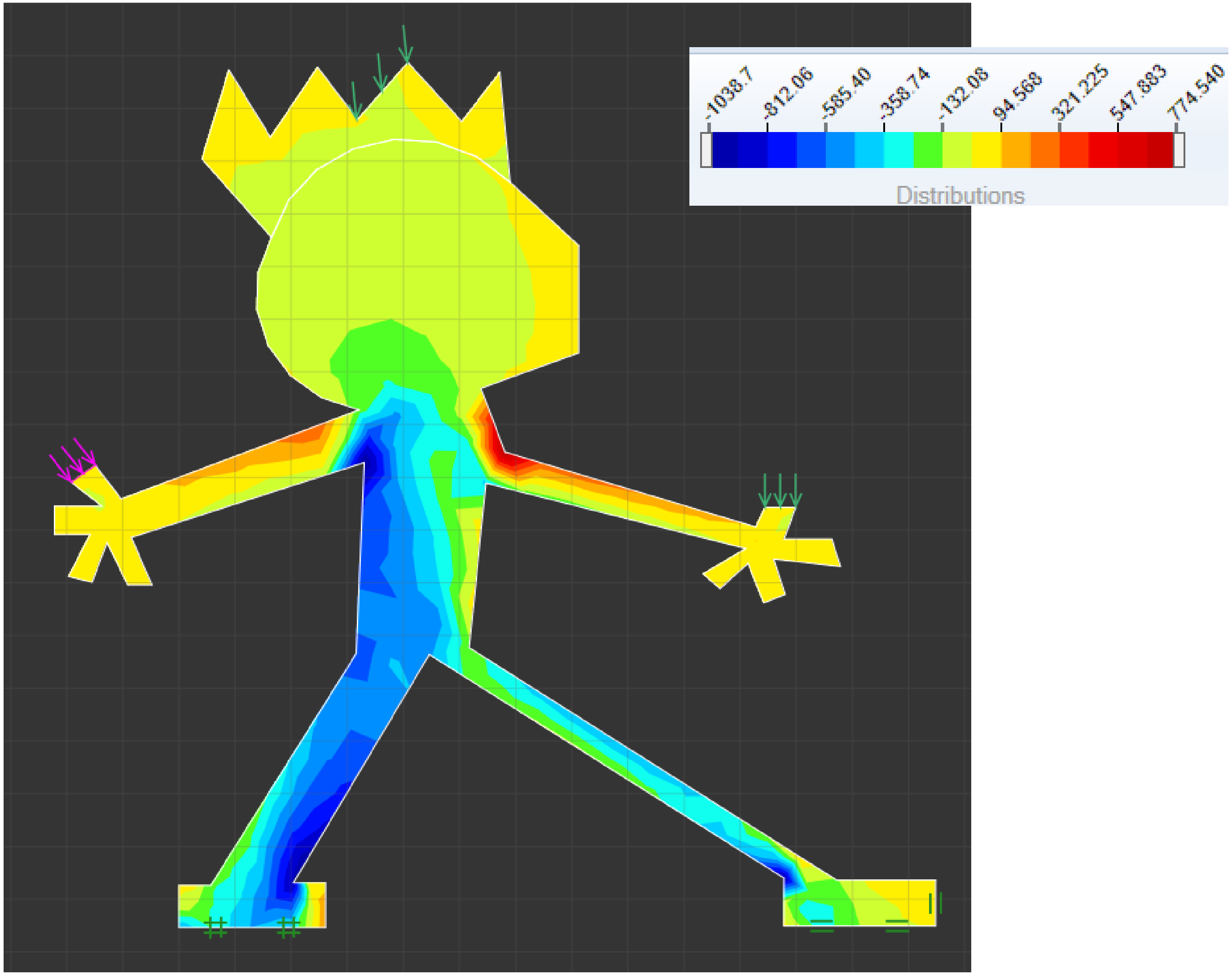
Total displacements: 10000 m



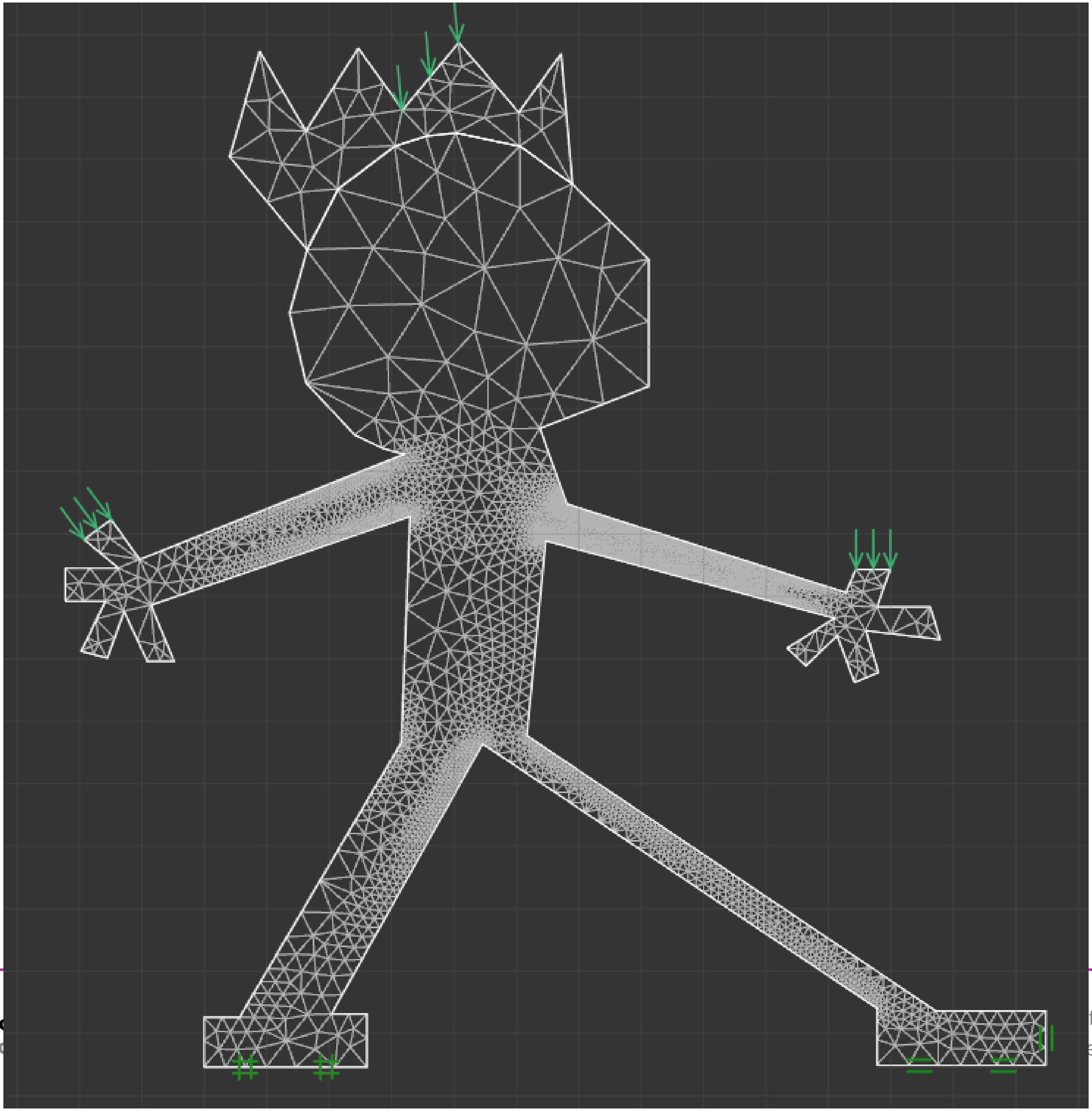
Total displacements: 10000 m

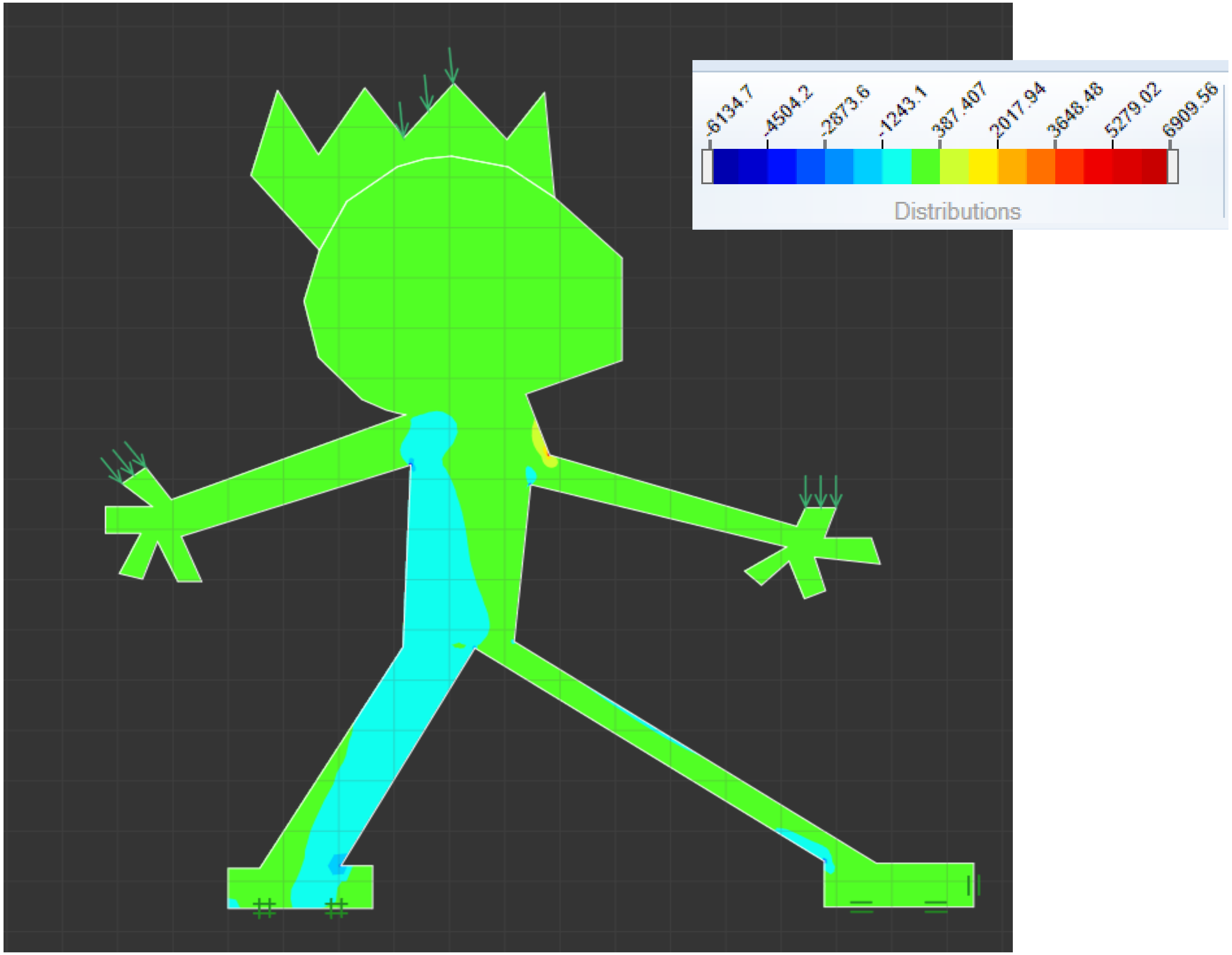


Total displacements: 0.08 m


-1038.7
-812.06
-585.40
-358.74
-132.08
94.568
321.225
547.883
774.540
Distributions

Y stress




-6134.7
-4504.2
-2873.6
-1243.1
387.407
2017.94
3648.48
5279.02
6909.56
Distributions

Y stress

Thank you