Lezione 3 - FEA & FEA (Practical Applications)
(Just theory)

Different Approaches to Practical Analysis

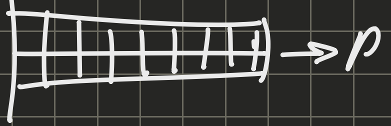
- ↳ Analytical
  - ↳ exact solutions → only approach that does so
  - ↳ very few solved cases
  - ↳ valid for classes of problems.

$$\sigma = \frac{P}{A}$$

- ↳ Experimental
  - ↳ direct link to physical objects
  - ↳ expensive (€ & time)
  - ↳ non transferable to other cases.

from strain gauge
↓
$\varepsilon \to \sigma = E\varepsilon$

- ↳ Numerical → approximation and widening of cases → cheapest approach to solve many cases
  - ↳ versatile
  - ↳ approximated → can be a problem based on case.
  - ↳ nontransferable to similar cases
    - ↳ the results are valid just for that object.



$$\{u\} = [K]^{-1}\{F\} \to \{\varepsilon\} \to \{\sigma\}$$

More steps more approximation error.

- ↳ What is used in FEM

More numerical methods for mechanical design falls entirely within the context of computer aided approach

FEA / FEM falls within CAE (computer aided engineering)
↳ tools for calculation and checking.

In CAE there is more than just FEA but also things like CFD, MBD and CAO.

FEA is a method for solving the analytical models in engineering
↳ Everything that we do to allow us to apply FEM.

FEA → setup FEM → the algorithm

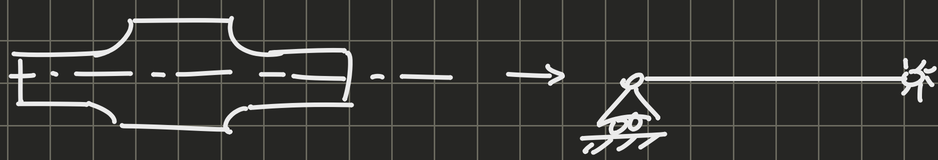
FEM was born in the 40s (before computers), but was only picked up in the 60s and 70s (thanks to computers) due to high numbers of repetitive calculations.
↳ It was developed by engineers through intuitive bases rather that mathematical, now it has become mathematical.

We will be using FEA/FEM to solve problems of solid mechanics, but its usable in problems where we have field quantities.

FEM is the simulation of a physical phenomena via numerical process based on piecewise polynomial interpolation.

General Overview of FEA (for solid mechanics)

1. Defining a suitable mathematical model (abstract schematisation) of the real physical problem.

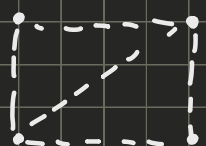


2. since there is no continuity, we divide the object in parts, finite elements. We discretize. The elements are connected to each other a nodes. The resulting grid is called the mesh. Two elements talk to another through nodes.

The advantage for discretization is that the mathematics are easier, since we go from a differential problem to a system of linear/non-linear equation

Each equation of the system defines what is happening at one specific node.

more specifically at one d.o.f for one node, so we need one equation for each d.o.f.



→ we need at least 4 equations, more depending on the degrees of freedom

When we solve the system we know the displacement of each node, but not what happens in between, so we will have to solve this.

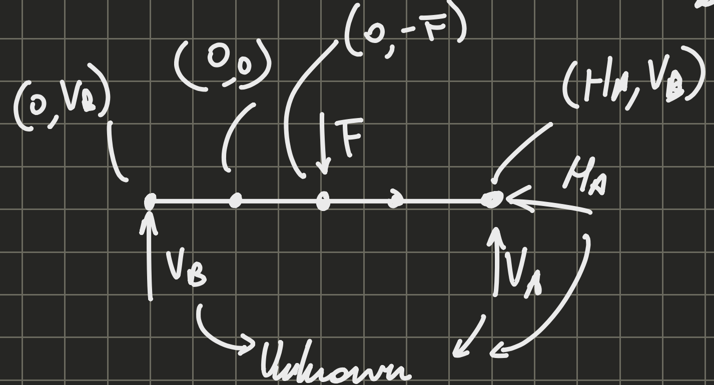
3. The system of equations is solved after imposing the appropriate boundary conditions at nodes

$$[k]\{u\} = \{F\}$$

$\{F\}$ → Load applied at each node.

$\{u\}$ → d.o.f freedom of a node

$[k]$ → Stiffness matrix → stiffness is a function of the geometry and the material.



We will have knowns and unknowns in $\{u\}$ and $\{F\}$, depending on the configuration.

Obtained results are known within the limits.

What is the physical meaning behind $[k]$?

The components of the matrix are integrals

Though an energetic approach we get:

$$[k] = \int_V [B]^T \underbrace{[E][B]}_{\text{If linear this is the equivalent to Hooke's law.}} dV$$

$[E]$ is the materials property matrix

$[B]$ is the strain displacement matrix $\Rightarrow \{\varepsilon\} = [B]\{u\}$

From the elastic deformation energy we get $[K]$:

$$U_{el} = \frac{1}{2}\int_V \{\varepsilon\}^T \underbrace{[E]\{\varepsilon\}}_{\{\sigma\}} dV =$$ → Discretized representation of continuum mechanics

$$= \frac{1}{2}\{u\}^T \int_V [B]^T [E][B]\, dV \{u\} = \underbrace{\frac{1}{2}\{u\}^T [K]\{u\}}$$

$$\left.\begin{array}{l} \varepsilon_x = \dfrac{du}{dx} \\ \varepsilon_y = \dfrac{dv}{dy} \end{array} \quad \gamma_{xy} = \frac{dv}{dx} + \frac{du}{dy} \right\}$$ → Components of $[B]$

we are effectively saying that each elements in a discrete structure acts like a spring, so the structure is a superposition of a system of linear elastic springs.

→ 4. Approximation of the displacement field between nodes

Since we now know the displacement of each node, we now can interpolate between elements to find an estimate of the position, this can be linear or polynomial (excluding linear)

Using second order elements (parabolas) to interpolate is better in terms of accuracy but is more computationally expensive. The smaller the elements the better the result but the slower the calculation.

Linear elements can be useful but if we are unlucky it can be disastrous.

5. Known displacement, stress and strain can be computed.

There is a trade-off between time and computational accuracy.

Software for FEM

Pre-processor → graphical environment for defining the problem

↓

Solver → carrying out calculations

↓

Post-Processor → graphical environment in which we get the results and manipulate them.

The worst-case scenario is when we get the wrong results that seem reasonable.

Types of Analyses

- Linear
  - → simplest
  - → most common
  - → fastest
  - → assumptions can be verified by hand.
- Non-linear
  - → nonlinearities: geometry (large displacements), material, loads, contact problems

→ in general, iterative and requires more computational time

→ numerical convergence can become an issue

→ require experience on both linear and non-linear physics.

Types of finite elements

→ There are infinite, but some are more common than other

pg. 15 lost

cubes are the elements with the most nodes, and can be difficult to divide.

(Strain = deformazione: ε)

Common Types of Finite Elements

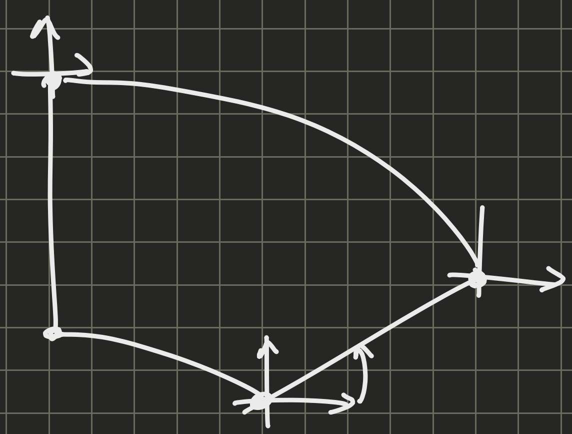
Finite Elements for (solid) continuum mechanics

$$
\begin{array}{ccc}
\text{Displacements} & & \text{Forces (Volumes)} \\
\downarrow & & \uparrow \\
\text{Strains} & \longleftrightarrow & \text{Stress}
\end{array}
$$

Finite Elements suitable for this kind of problem are independent on the dimension of the element.

An approach based on displacements is adopted: the primary unknown are the translation displacements.

other quantities are derived through the appropriate relations.



The rotations are dependent on the displacements with respect to the starting condition.

Deriving the rotation from other things, reduces the degrees of freedom, allowing us to reduce the calculations.
↳ If they are not solid this is not true.

To use non-linear elements we need to get another node for each element.

1D elements still work in 3D space.

Important shapes are:
↳ linear triangles and squares, as well as non-linear triangles and squares.
↳ There are comparable 3D forms.

Day 2 - FEA & FEM

↳ We commonly use solid elements which function the same based on their dimensions.

Finite Elements for structural mechanicals

↳ In structural mechanical we work with kinematics (rotations and deflections). Typically only 1D and 2D are used, like beams, plates and shells.

We reduce the number of degrees of freedom, meaning that the mathematical equations are different.

2 for nodes in 1D, 3 in 2D.

Comparative

$v_c = \overbrace{1}^{\text{Due to Bending}} + \overbrace{0{,}031}^{\text{Due to shear}}$

$\sigma_B = 300$ MPa

In this case FEA is not really needed but when comparing to different kinds of elements.

With the single linear element gives us the exact solution. On the other hand other 2D forms are worse at estimating (in this case) at varying degrees.

The problem with using 1D elements is that for continuum mechanics they are not good, as they don't consider the notch effect which has a substantial impact on the result.

Triangular and square linear are bad at measuring constant strain.

If we consider elements with linear strain the performance is improved, with linear we need more parts, same for quadratic elements.

Triangles are generally worse that squares.

Distorsions in the shape of elements introduces errors in the case.

We should always try to use the best applicable case.

Distorsion

If the geometric domain which we need to make measurement is regular, there is no issue.
If its distorted it introduces errors.

Types:
- For squares if $a/b > 10$ its distorted
- $\beta > 45$, $\alpha > 135$ its distorsion.

Distorsions come in many forms.
- Distorsions come from the setup no the calculation parts.

While distorsions are not favored, they are many times not available, so the best we can do is to try our best to reduce it.

Set up of FEA: steps:

- mathematical models for FEA work on idealization of the real physical model.
- The most appropriate idealization can be defined a fine-tuned to understand the physical nature of the problem, This idealization occurs in our brain out.

GARBAGE IN = GARBAGE OUT

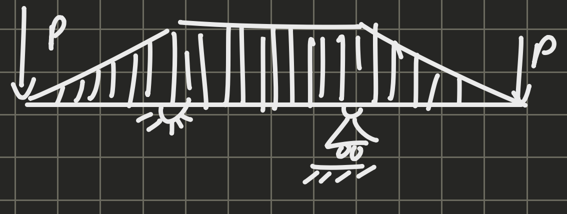
- To idealise we need to remove all superfluous details while keeping the needed, reducing time and keeping the accuracy.

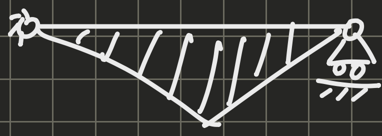
To choose the appropriate elements, it is not possible:

- without knowing the theory
- knowing the elements (what they can and cannot do)

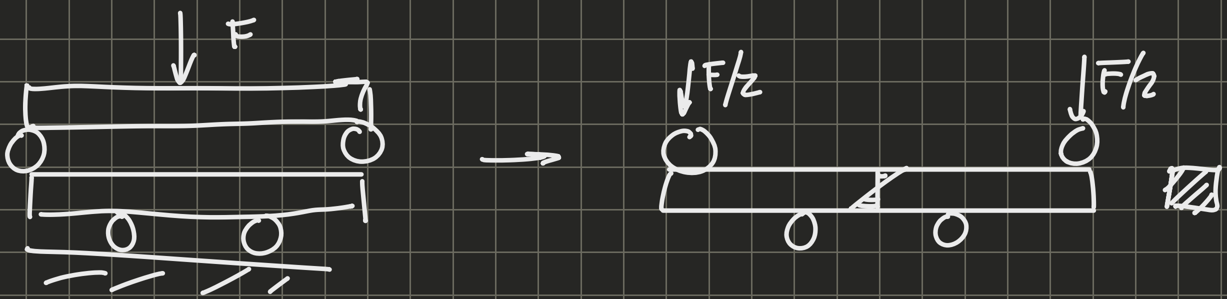
Example:

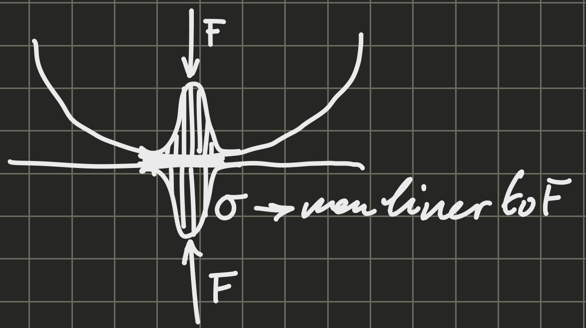
4 point bending





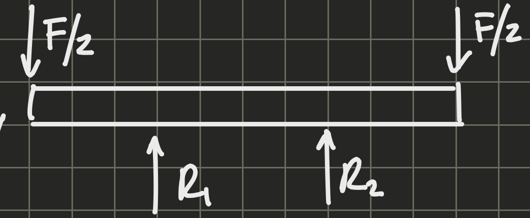
Rollering points are contact points, since contact points are non-linear, the analysis will immediately be non-linear.





If the roller is much stiffer than the beam, then the area of application can be loosely approximated to a point where the force is applied.

This means we can approximate to:

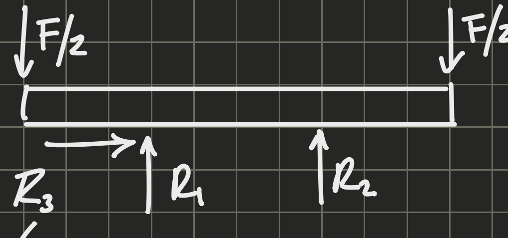


 → we can also simplify this, by just taking a slice, assuming a properly taken plate stress consideration and assuming our 2D object can simulate the 3D object.

$$[k]\{u\} = \{F\}$$

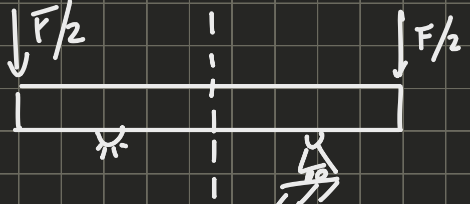
Since $[k]$ is sdp, the system must be isostatic or hyperstatic

With how its currently drawn its not isostatic since we have not put the horizontal constraint on the left; it means that it will then be isostatic.



↳ The force still needs to be added, even though it comes out to be 0.

We can also simplify by realising the symmetry of a system which we are looking at.

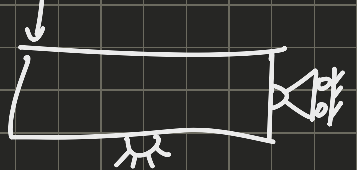


The roller can be seen as the same as the hinge since the horizontal is 0.

With symmetry we either use half the elements or use the same number of elements for half the space so more accuracy.

By using symmetry we need to add constraints to remake the system isostatic. Since nodes of solid object only have 2 d.o.f. since they can't rotate, we only can constrain in 2 possible ways.

To prevent our structure from compenetrating we need to fix the movement in the horizontal direction



While the vertical moment won't get constrained since we want to allow the movement to be the same as the original.

It's possibile to have more than one plain of constraint, since we want to reduce calculations, we still need to keep the symmetry constraints.

We need to make sure each simmetry is the exact same.

If the aim was to to find the deflection we could think about 1D elements. Since we change the dimension, we must change the constraint at the axis of symmetry, since we now have to also consider the rotation we need to block the deformation so it's the same as the original system. we need to constrain like before, so the constraint becomes a slide.

Discretization (after idealization)

↳ The mesh is not a given, but an unknown of the problem.

In accordance with de Saint-Venant, independent on how we apply the load, the stresses will be the same.

↳ This explains why we can approximate the Ω in a roller to a force in a single point.

When we place a load in a point, the model generates stresses around the area which in reality does not exist, so the stresses have no physical meaning and so we can ignore it.

Constraints:

- set $= 0$ certain degrees of freedom.
- since in reality ideal constraints do not exist, we can state the compliance to these constraints.

Multi-Point Constraints

- constraints which relate a specific node d.o.f to another node d.o.f

Master node ... Slave node

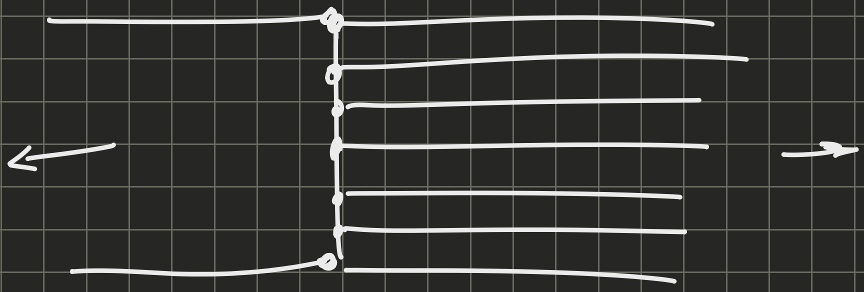
$$C_1 + C_2 u_n + C_3 v_n + C_4 w_n = u_i$$

- Example case.
- Horizontal displacement of i, is a linear combination of the displacement in all direction for node n.

Practical Implementation

- Pins (Welding together elements)
- Tieing elements
- Coupling
- Slider

Sliders when we don't have enough computational elements.
Welding together elements of different sizes.



The problem with this is that the nodes don't have reactions so if we don't tie we have problem. With the tie we are able to do it.

Evaluation of results

↳ We don't have many tools to do so, other than trust in ourselves.

Useful tools:

↳ checking reaction forces → if errors ~ 5-10% the first steps are good. → don't then later on → if first step is wrong, the later steps will definitely be wrong.

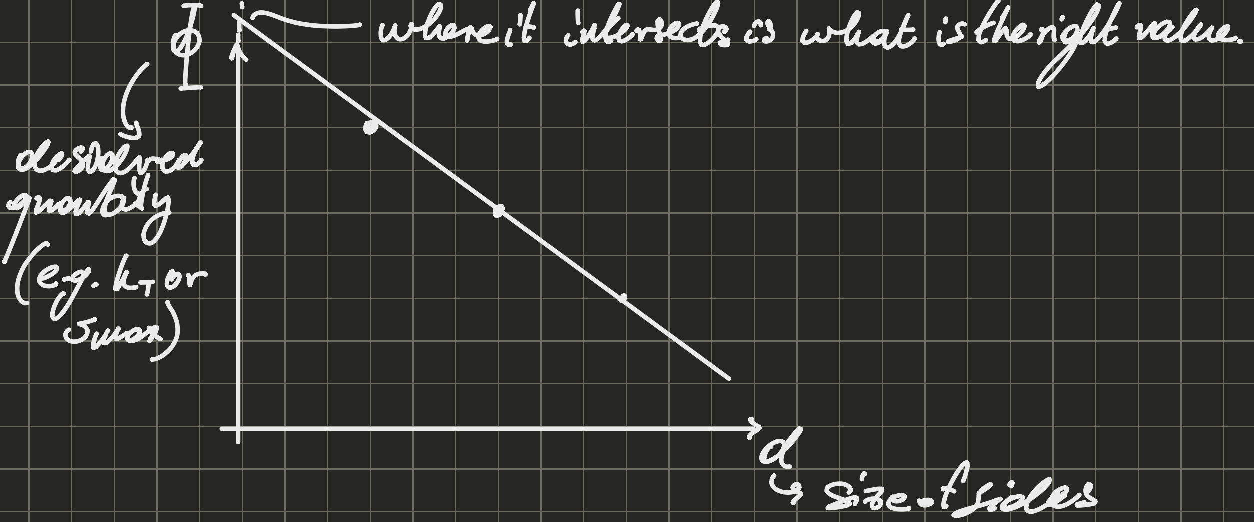
↳ observing the discontinuities of the calculated stress field.

↳ carrying out a convergence analysis.

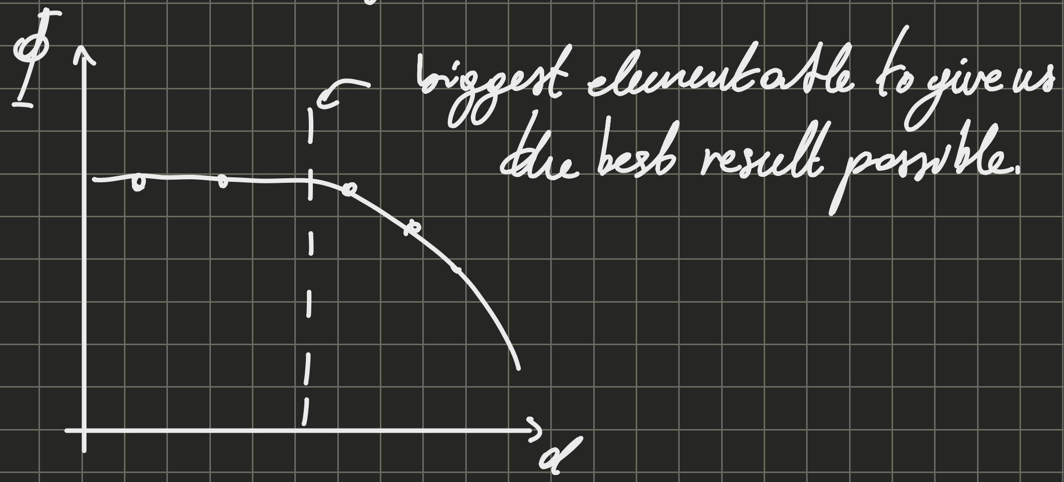
→ many programs automatically calculate the average, we have to turn it off, since it's terrible.

→ Not numerical convergence

↳ Analysing our mesh with the size of elements in our solution.



Other times it goes like:



We should always apply it to have an idea of the best size.