

PROTOTIPAZIONE VIRTUALE

Analisi strutturale in ambiente virtuale

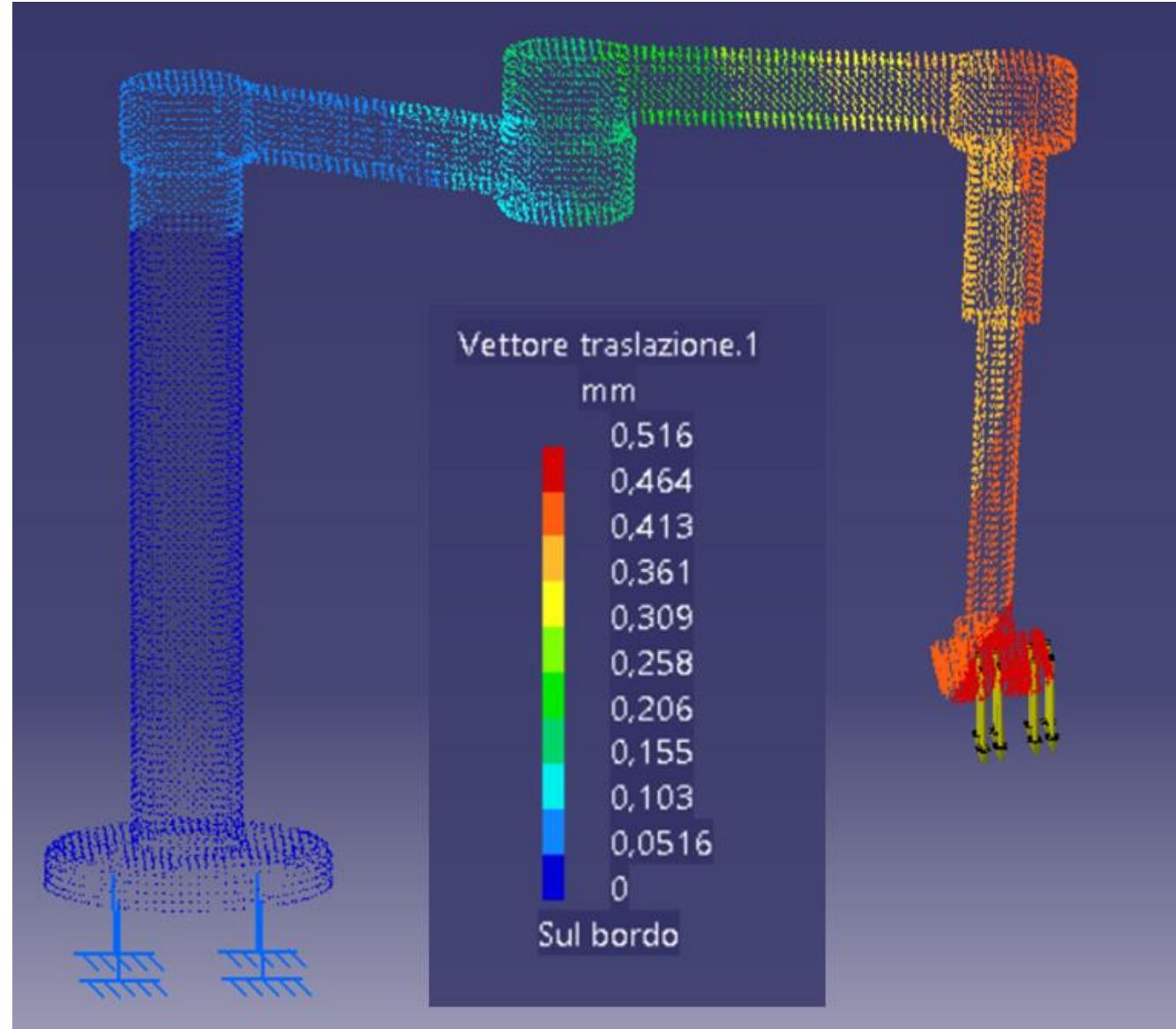
Giuseppe Di Gironimo

University of Naples Federico II



Introduction

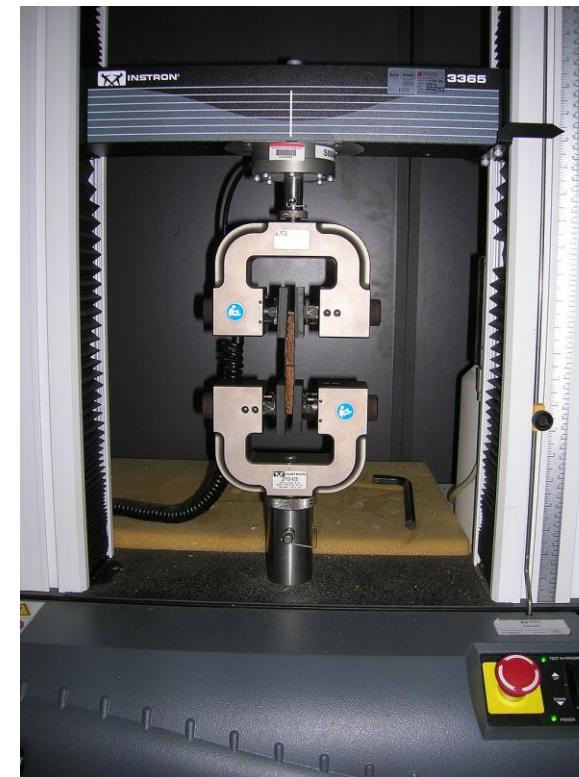
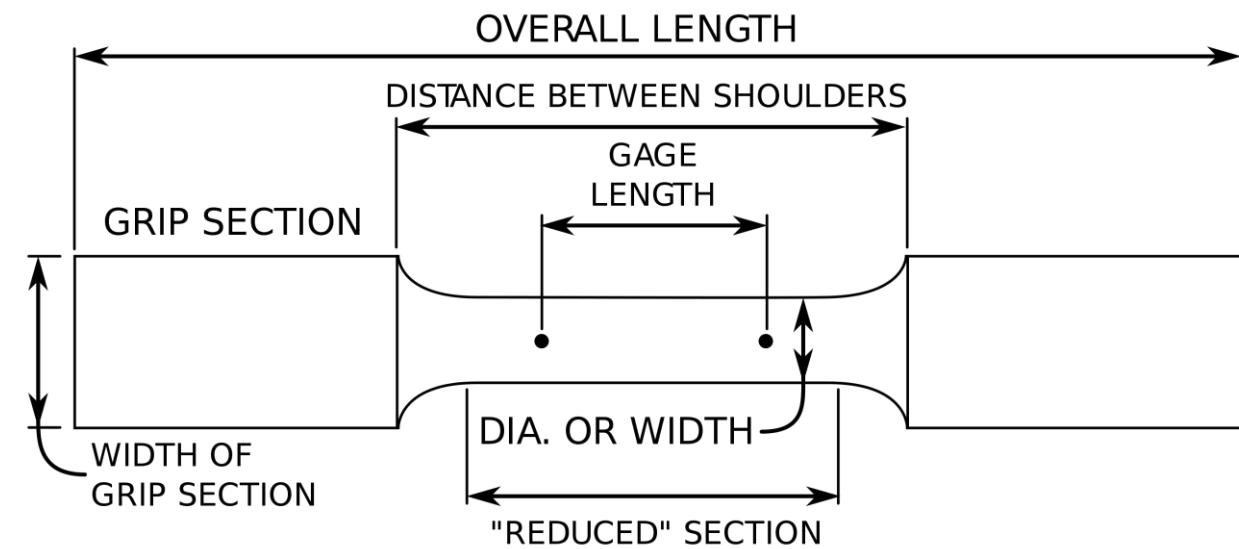
- Structural Analysis deals essentially with the determination of stress and displacement distributions under prescribed loads, temperatures and constraints, both under static and dynamic conditions.



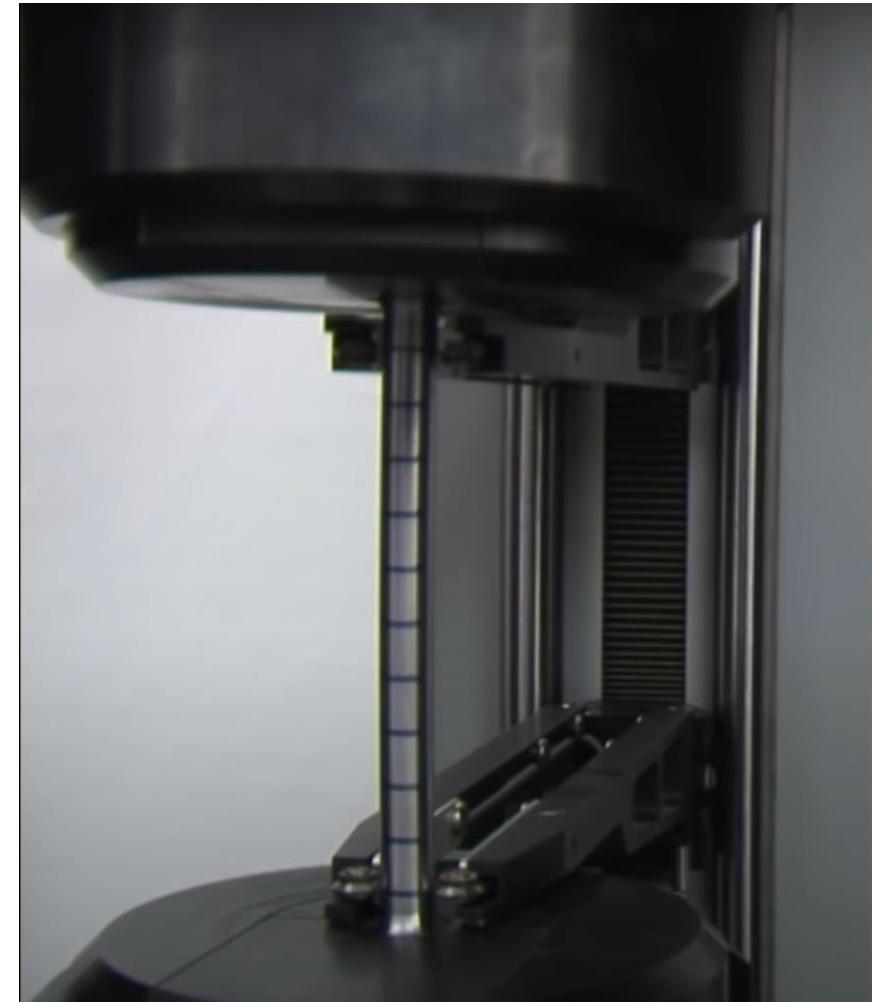
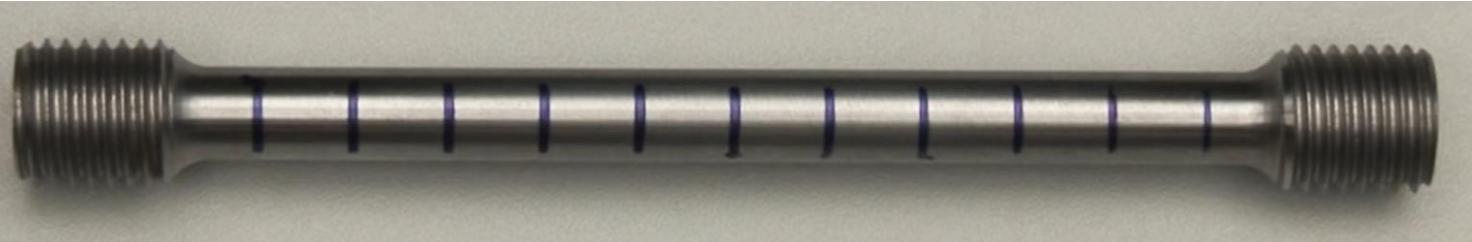
2



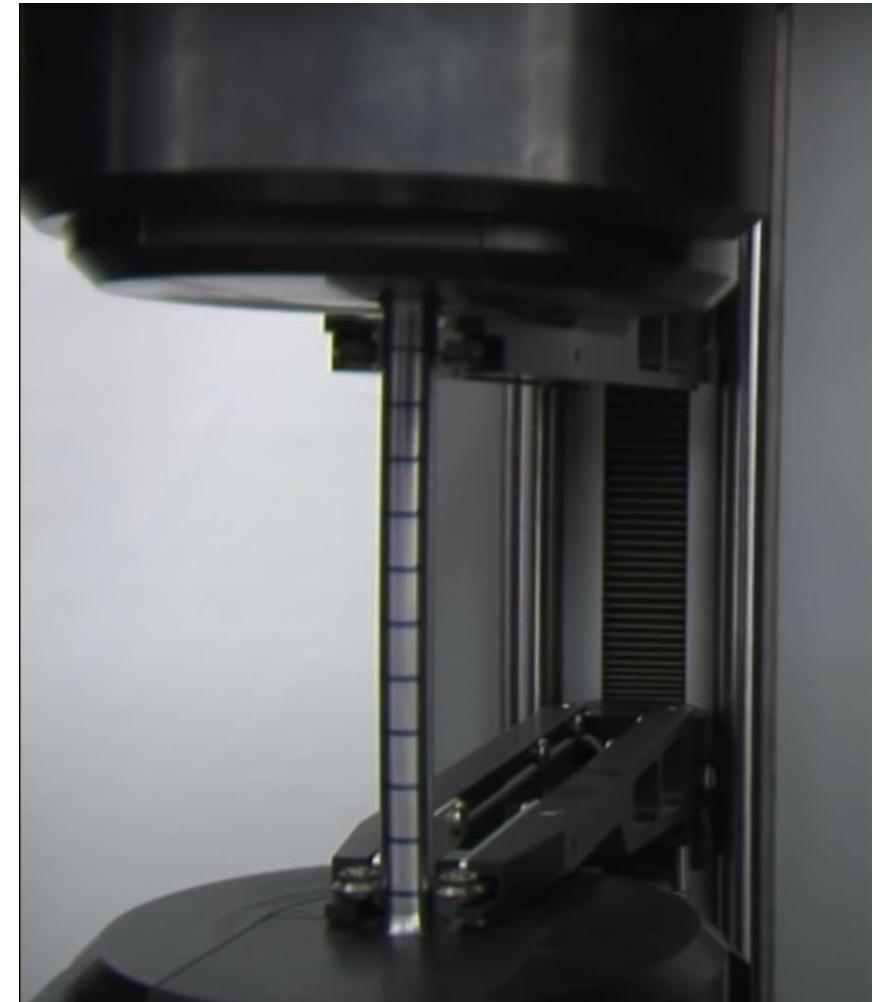
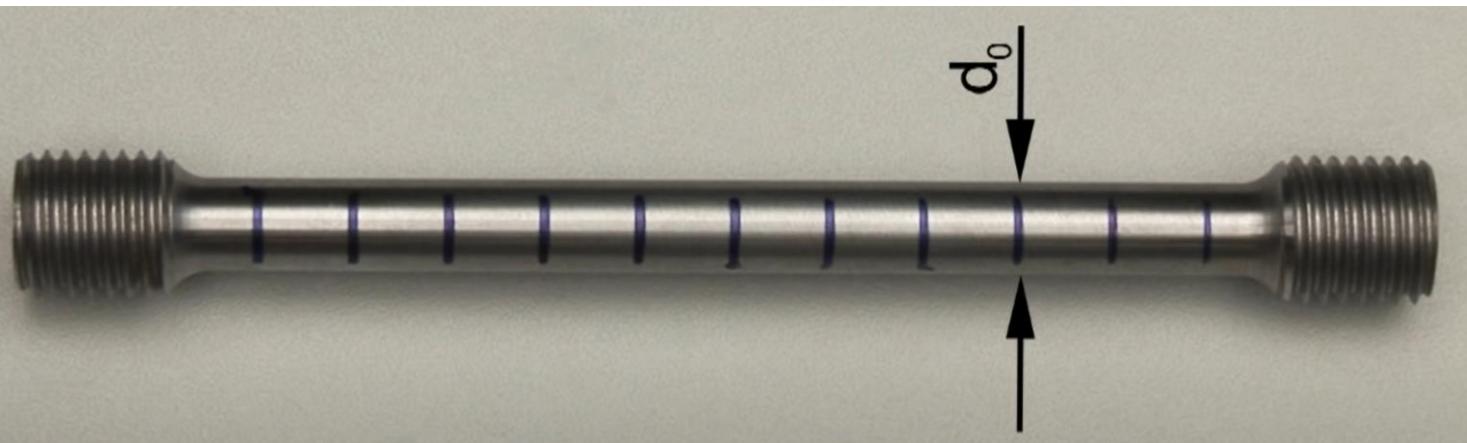
Tensile testing



Tensile testing



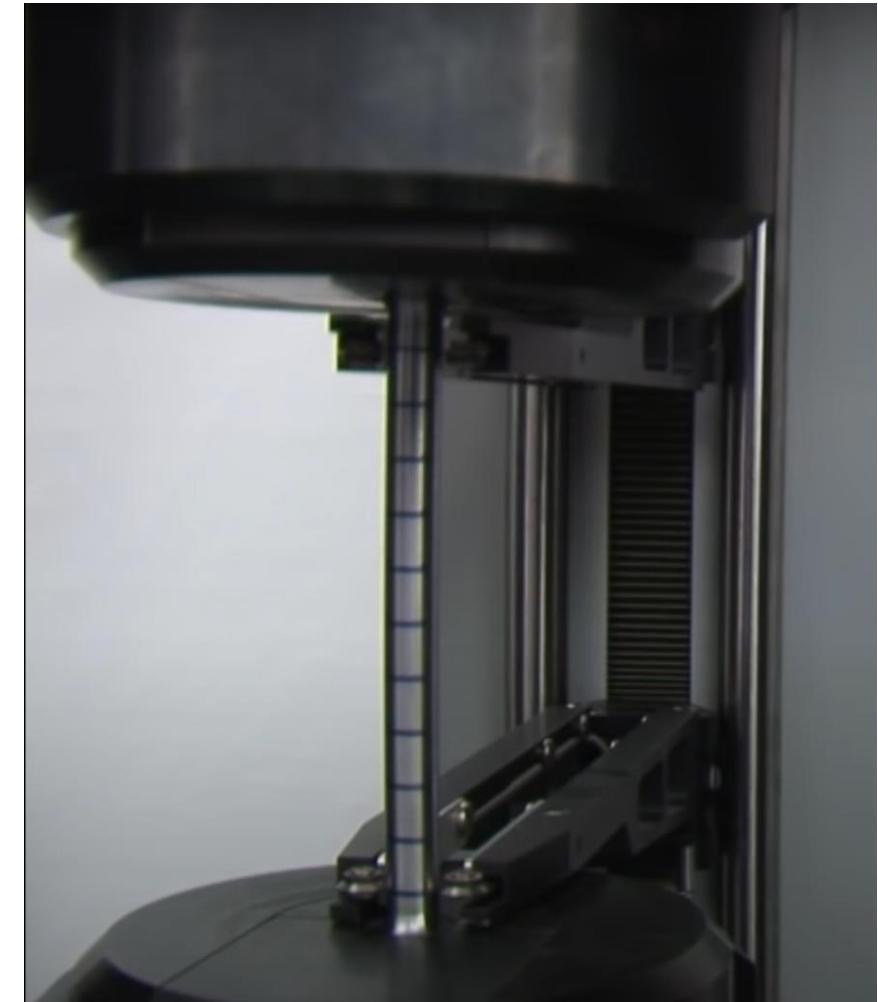
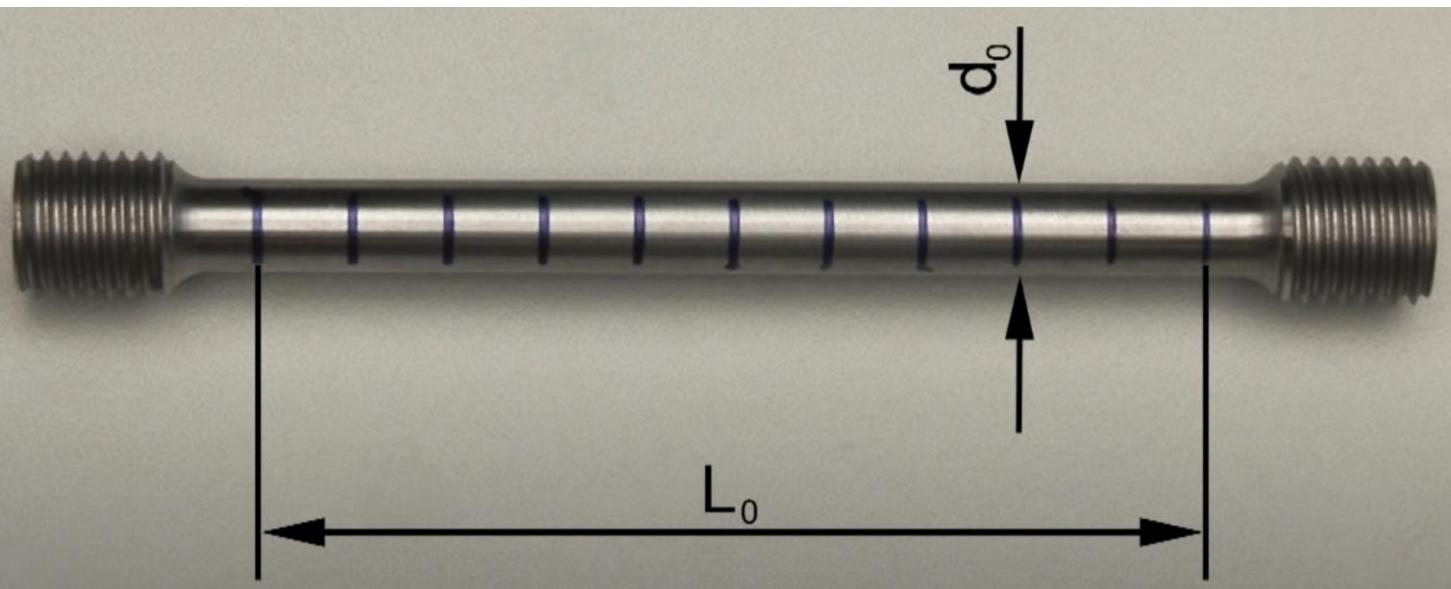
Tensile testing



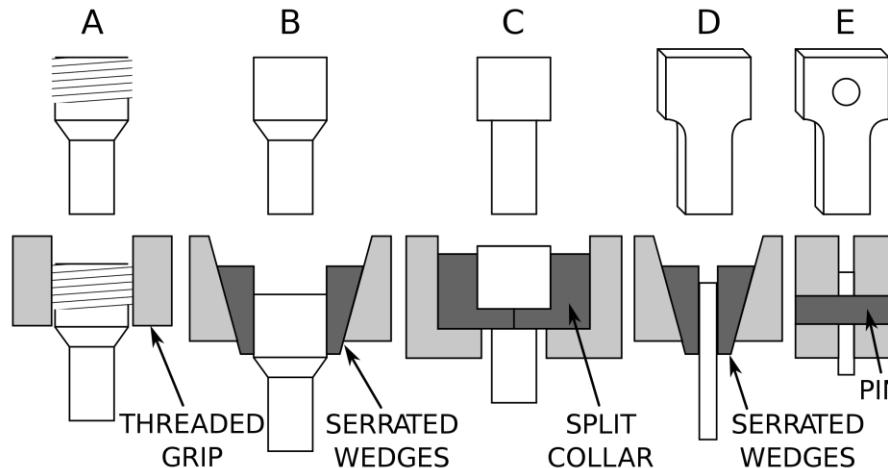
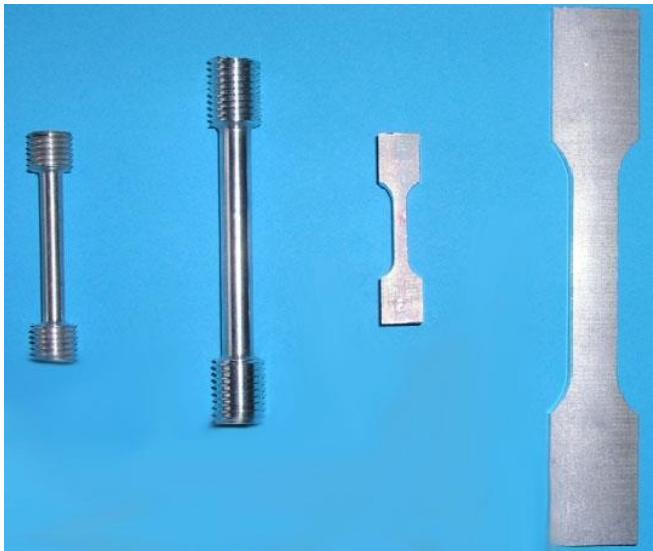
<https://www.youtube.com/watch?v=D8U4G5kpcM>



Tensile testing



Tensile testing



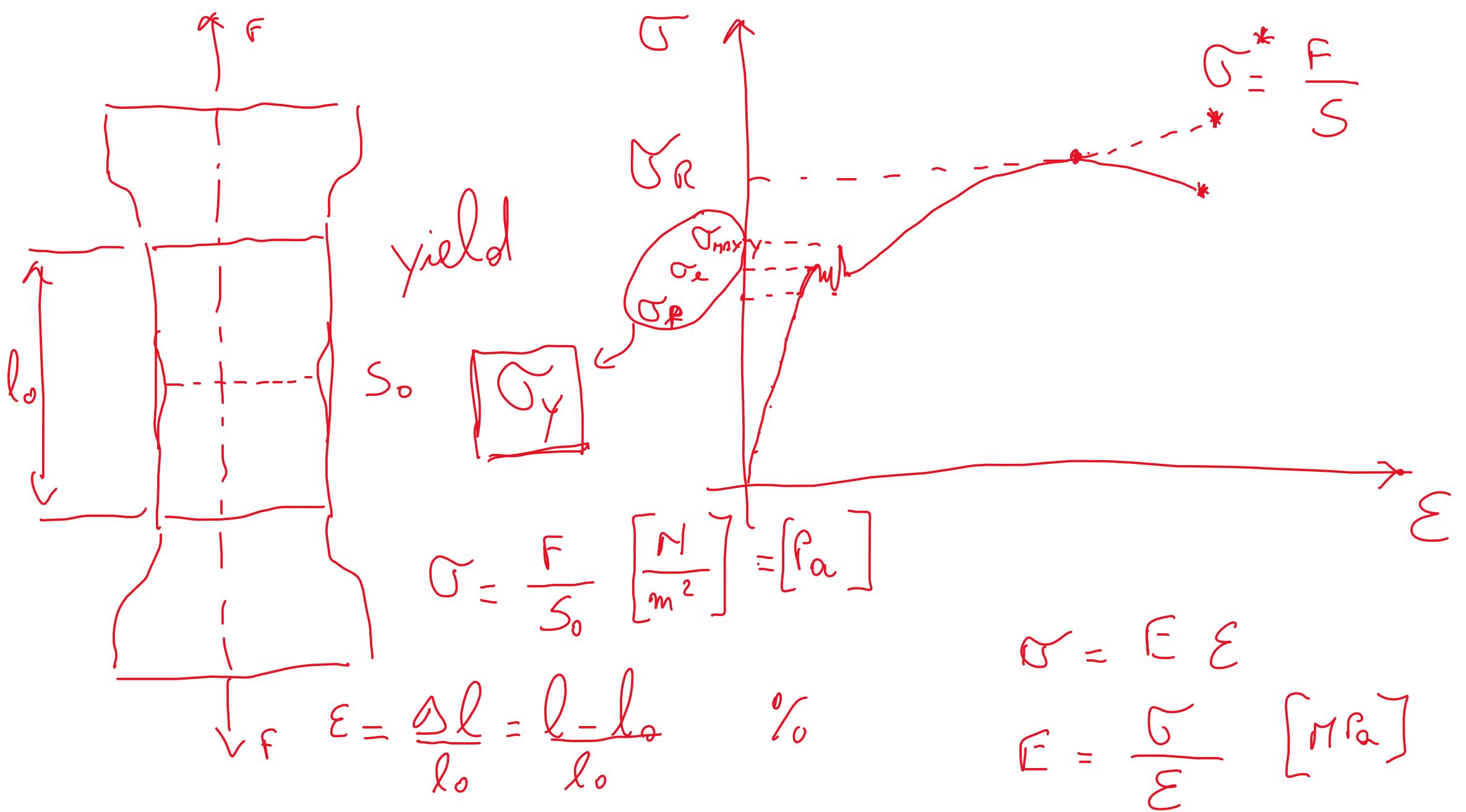
A schematic of various shoulder styles for tensile specimens.

Keys A through C are for round specimens,

keys D and E are for flat specimens.

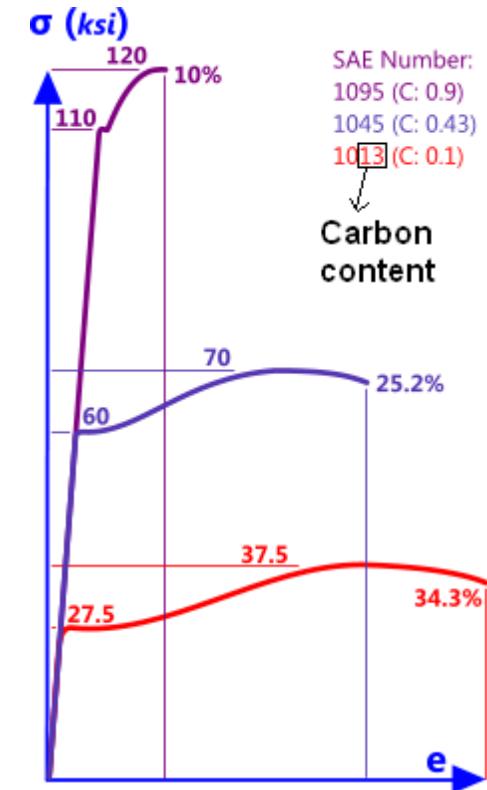
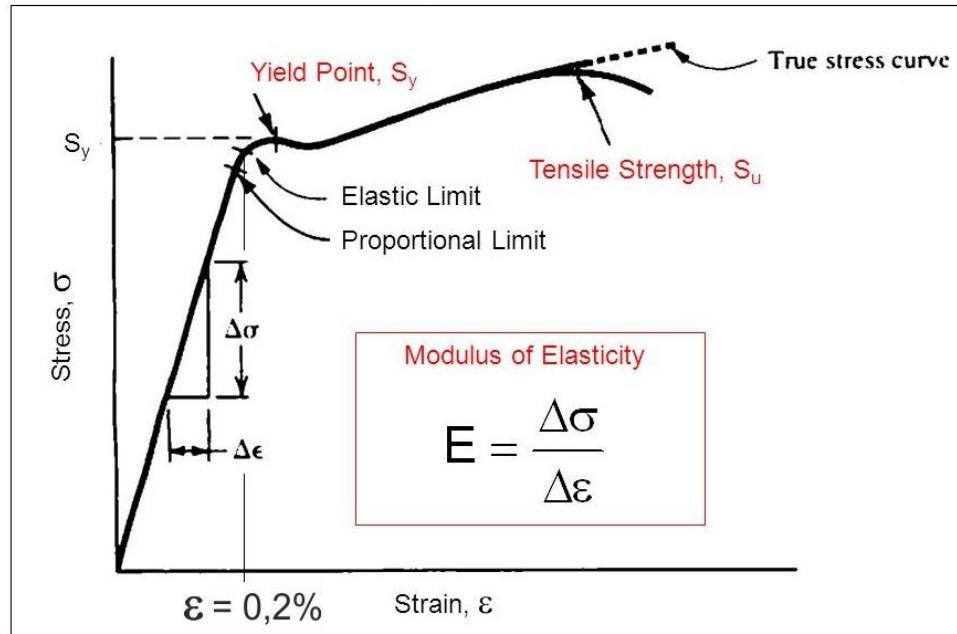
- A. A threaded shoulder for use with a threaded grip
- B. A round shoulder for use with serrated grips
- C. A butt end shoulder for use with a split collar
- D. A flat shoulder for used with serrated grips
- E. A flat shoulder with a through hole for a pinned grip

Based on an image from Tensile testing by Joseph R. Davis



Tensile testing

Stress-Strain Curve for Steel



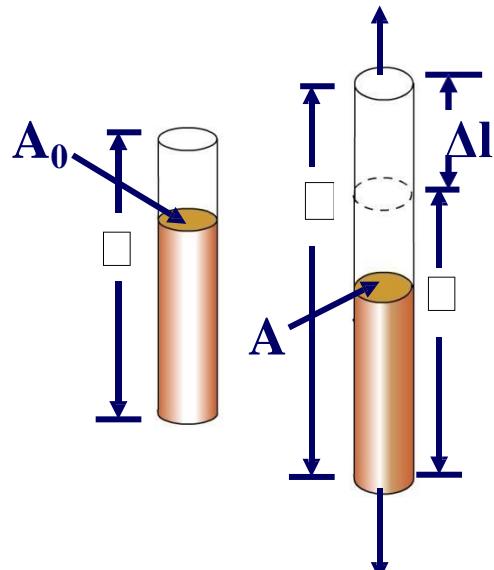
SFORZO E DEFORMAZIONE NOMINALI

TENSIONE NOMINALE $\rightarrow \sigma = \frac{F}{A_0}$

forza a trazione assiale media
superficie resistente iniziale

DEFORMAZIONE NOMINALE $\rightarrow \epsilon = \frac{\Delta l}{l_0}$

variazione di lunghezza del provino ad un certo istante della prova
lunghezza del tratto utile del provino



Unità della tensione: PSI (pound per square inch) or N/m²(Pascal) 1 PSI = 6.89×10^3 Pa

$$1 \text{ MPa} = 10^6 \text{ Pa} = 1 \text{ N/mm}^2$$

$$1 \text{ GPa} = 10^9 \text{ Pa} = 10^3 \text{ MPa}$$

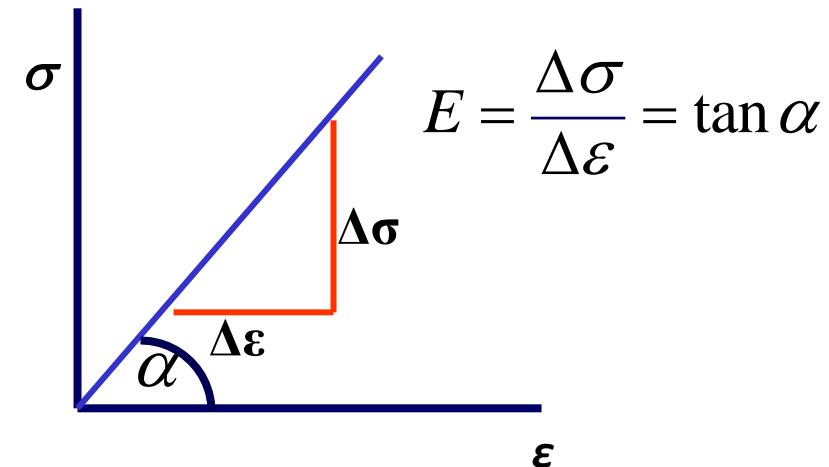


STRESS – STRAIN RELATION

Stress and strain have a linear relationship in the elastic field (Hooke's law)

Elastic Modul (Young) (E) :

$$E = \frac{\sigma \text{ (stress)}}{\varepsilon \text{ (strain)}}$$

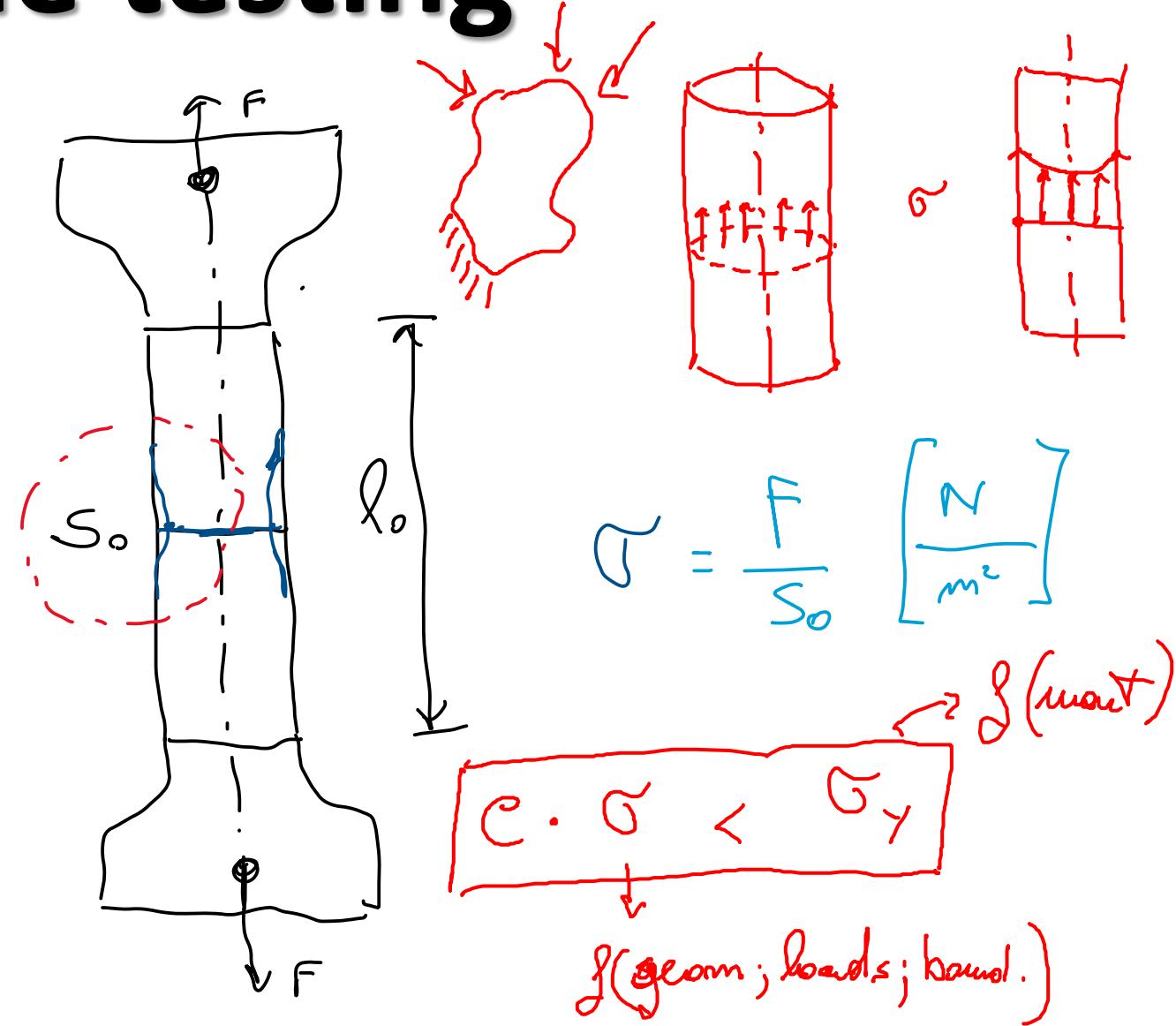
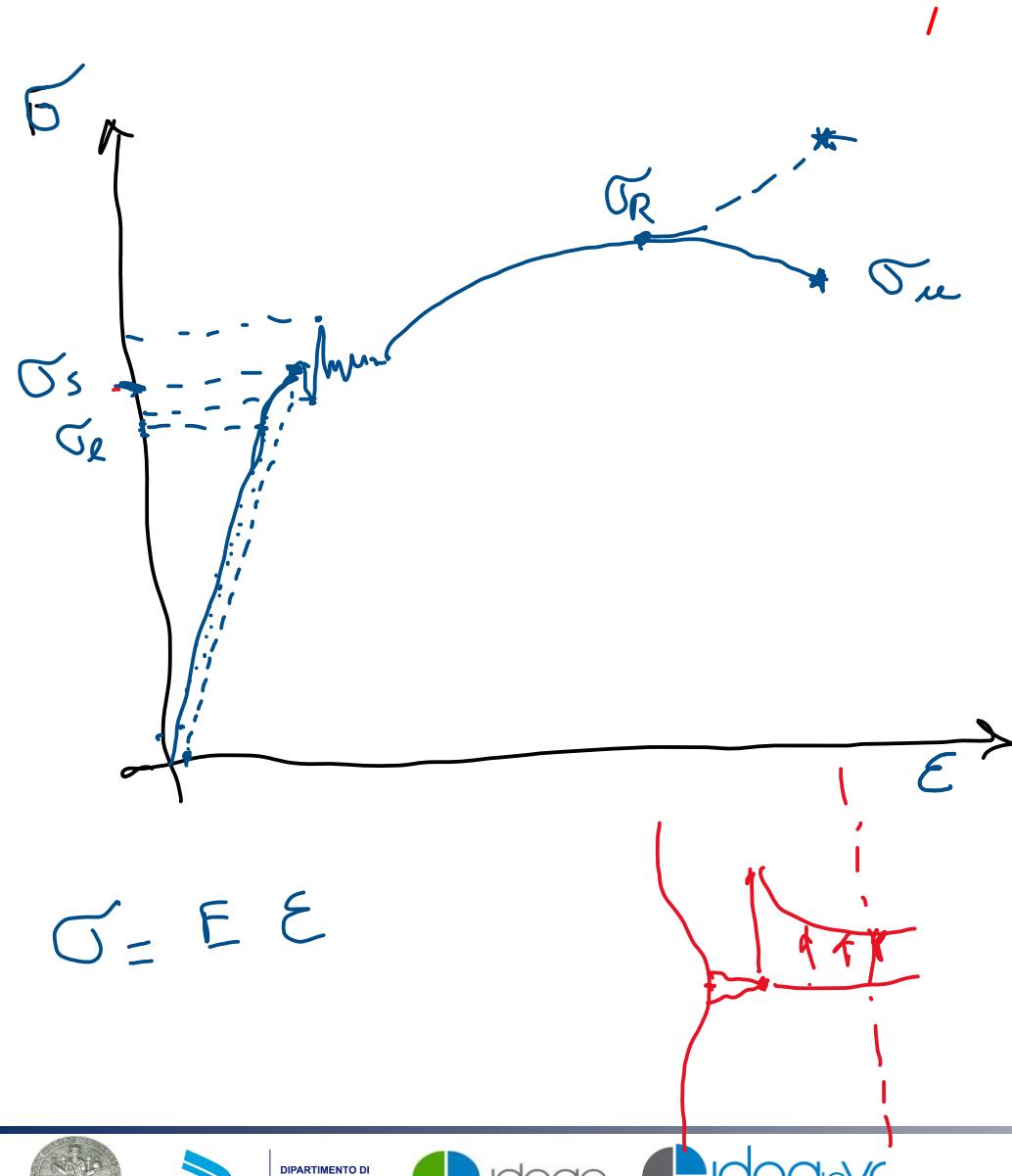


Example: YOUNG MODUL (STEEL): 207 GPa

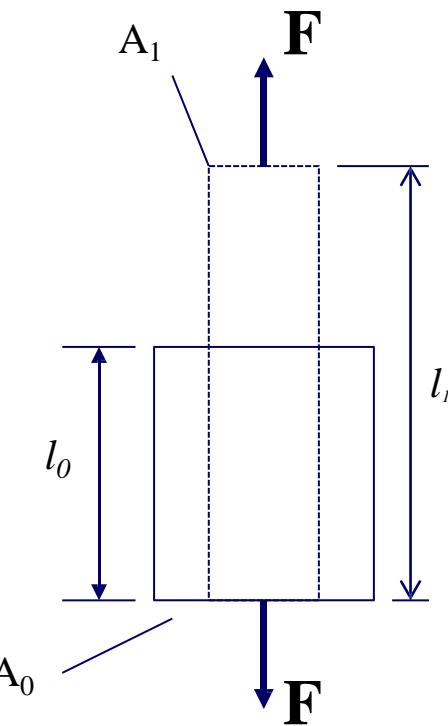
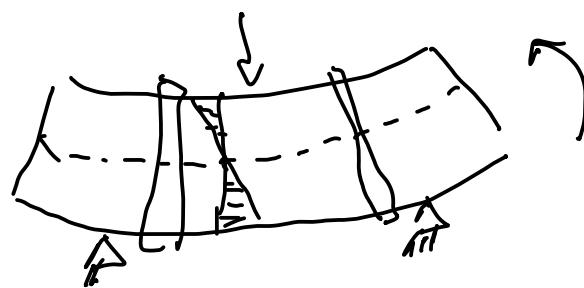
YOUNG MODUL (ALUMINUM): 76 GPa



Tensile testing

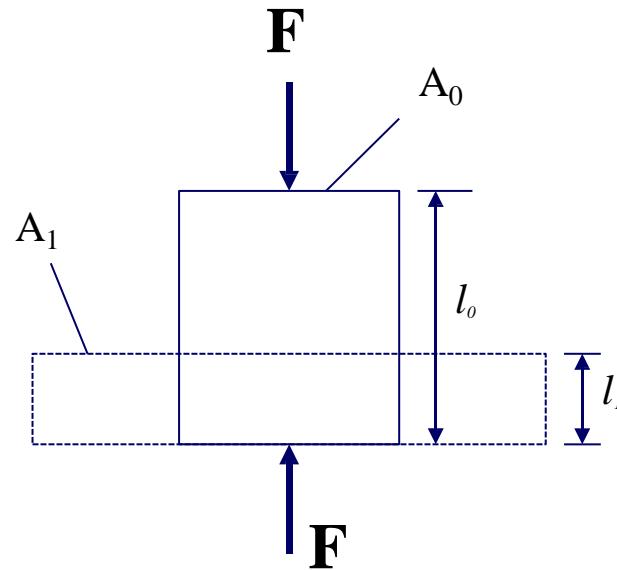


Elementary stresses

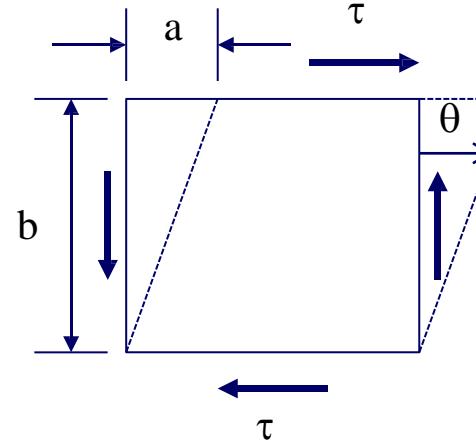
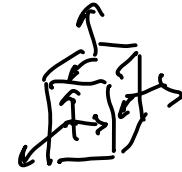


Uniform traction

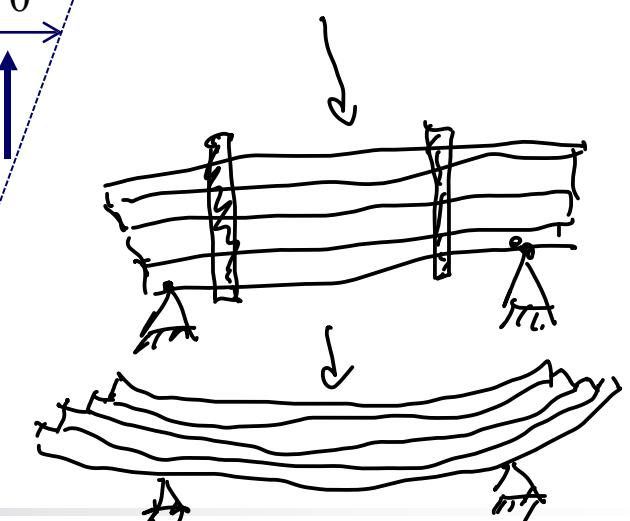
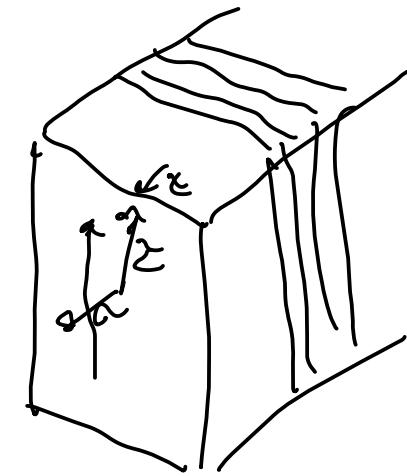
- A_0, l_0 = initial section and length
- A_1, l_1 = final section and length
- A, l = instantaneous section and length



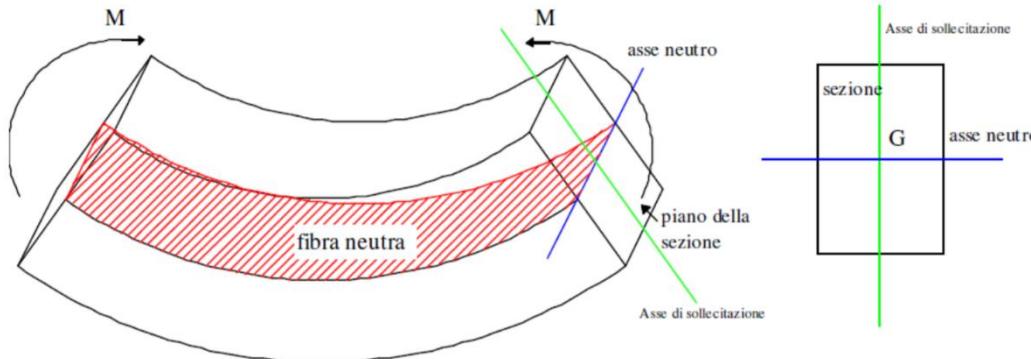
Uniform compression



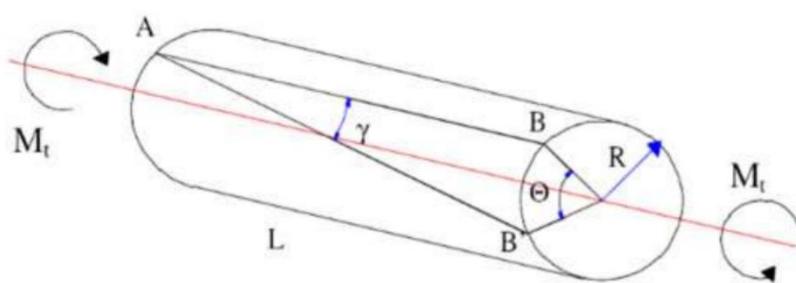
Shear



Elementary stresses



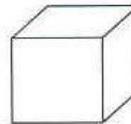
Bending



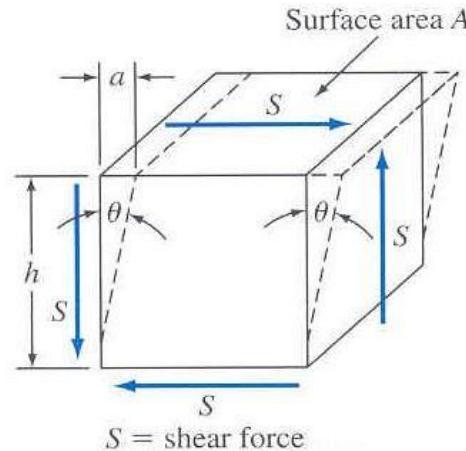
Torsion

SFORZO E DEFORMAZIONE DI TAGLIO

$$\tau = \frac{S \text{ (forza di taglio)}}{A \text{ (superficie di applicazione della forza di taglio)}}$$



(a) Unstressed body



$$\gamma = \frac{\text{spostamento a}}{\text{distanza "h" sulla quale agisce lo sforzo}}$$

$$\text{Modulo di taglio } G = \tau / \gamma$$



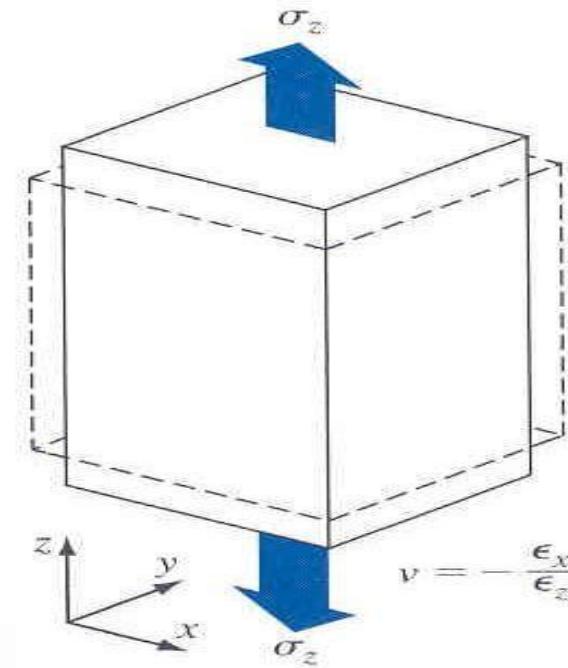
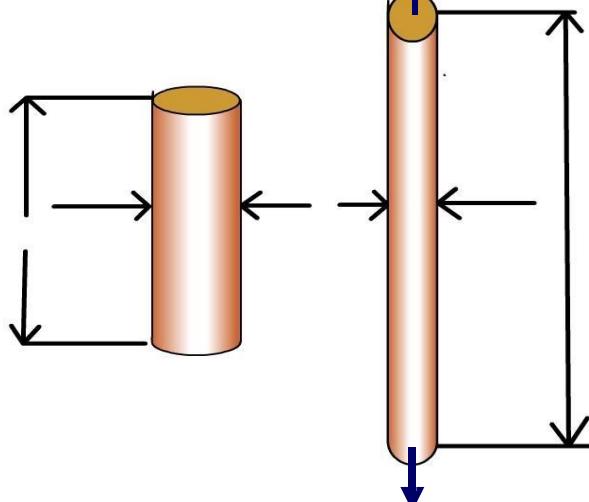
POISSON'S RATIO

The deformation (expansion or contraction) of a material in directions perpendicular to the direction of loading.

The value of Poisson's ratio is the negative of the ratio of transverse strain to axial strain.

For small values of these changes, is the amount of transversal elongation divided by the amount of axial compression.

$$\nu = -\frac{\epsilon_{\text{lateral}}}{\epsilon_{\text{longitudinal}}} = -\frac{\epsilon_y}{\epsilon_z} = -\frac{\epsilon_x}{\epsilon_z}$$



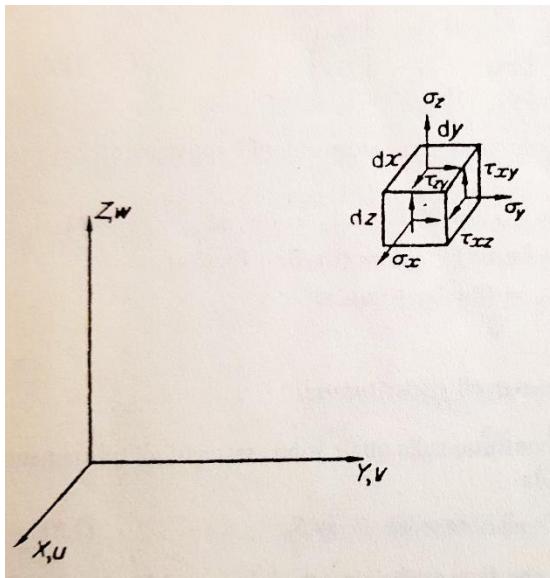
isotropic materials

$$-\epsilon_x = -\epsilon_y$$

Example: Stainless steel $\rightarrow 0.28$
copper $\rightarrow 0.33$

Most materials have Poisson's ratio values ranging between 0.0 and 0.5.

STATO DI SFORZO



In un punto interno di un corpo lo stato di tensione è definito dal tensore degli sforzi

$$\begin{pmatrix} \sigma_x & \tau_{yx} & \tau_{zx} \\ \tau_{xy} & \sigma_y & \tau_{zy} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{pmatrix}$$

dove

$$\tau_{xy} = \tau_{yx} \quad \tau_{xz} = \tau_{zx} \quad \tau_{yz} = \tau_{zy}$$

Le componenti del tensore degli sforzi devono soddisfare in tutto il corpo le seguenti equazioni di equilibrio

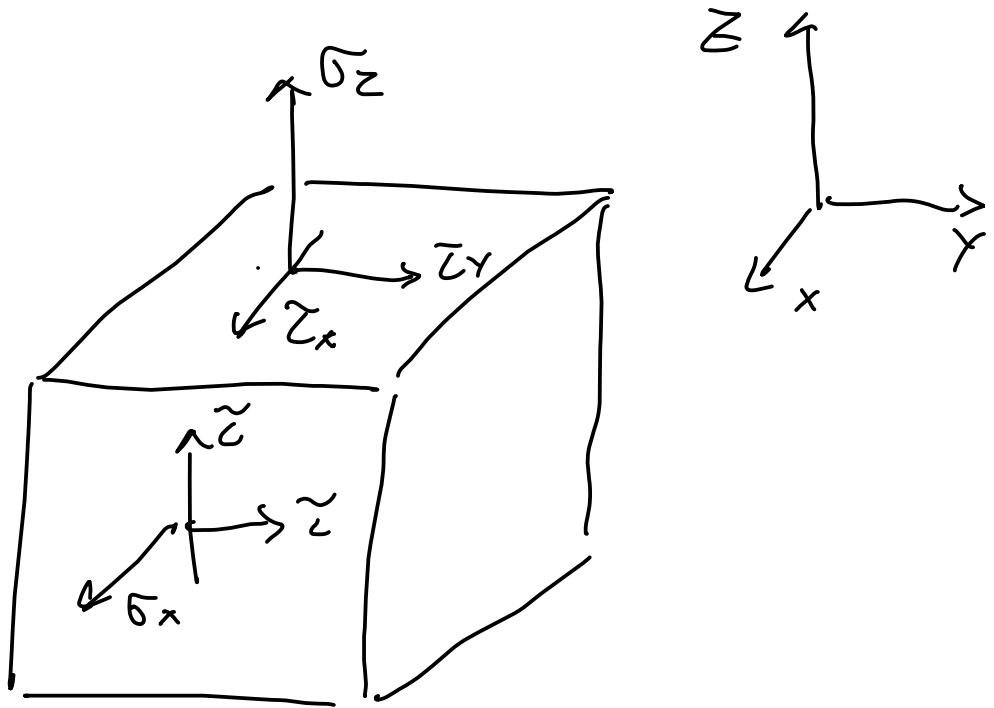
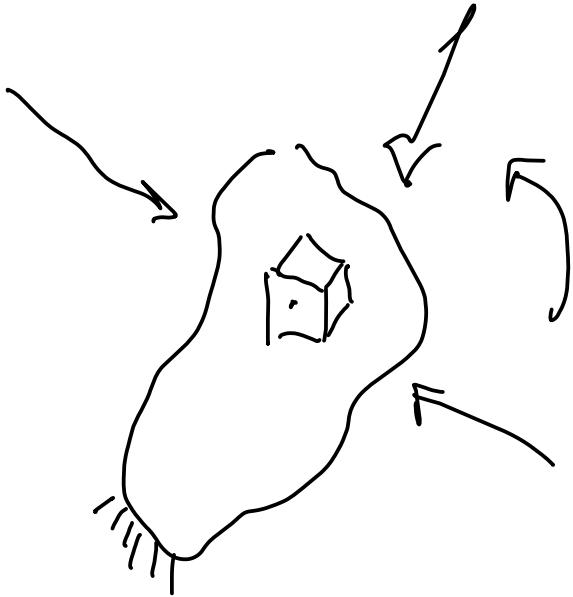
$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + b_x = 0$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + b_y = 0$$

$$\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + b_z = 0$$

dove \$b_x\$, \$b_y\$ e \$b_z\$ sono forze di volume





STRESS – STRAIN RELATION

Stress-strain general relation for an elastic-linear material can be written in the form (Navier-Cauchy equations)

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & d_{13} & d_{14} & d_{15} & d_{16} \\ d_{21} & d_{22} & d_{23} & d_{24} & d_{25} & d_{26} \\ d_{31} & d_{32} & d_{33} & d_{34} & d_{35} & d_{36} \\ d_{41} & d_{42} & d_{43} & d_{44} & d_{45} & d_{46} \\ d_{51} & d_{52} & d_{53} & d_{54} & d_{55} & d_{56} \\ d_{61} & d_{62} & d_{63} & d_{64} & d_{65} & d_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$$

simm.

Which for a generic material has 21 independent parameters.

For an isotropic material there are only two independent parameters, and the stiffness matrix is particularized in:

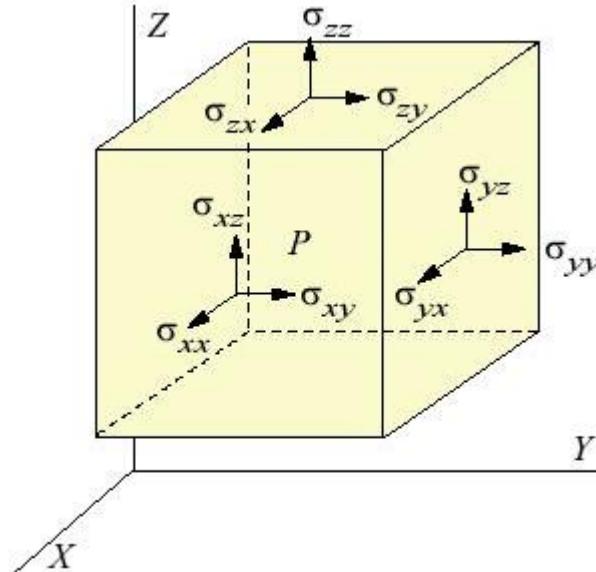
$$\mathbf{D} = \frac{E}{2(1+\nu)} \begin{bmatrix} \frac{2(1-\nu)}{1-2\nu} & \frac{2\nu}{1-2\nu} & \frac{2\nu}{1-2\nu} & 0 \\ \frac{2\nu}{1-2\nu} & \frac{2(1-\nu)}{1-2\nu} & \frac{2\nu}{1-2\nu} & 0 \\ \frac{2\nu}{1-2\nu} & \frac{2\nu}{1-2\nu} & \frac{2(1-\nu)}{1-2\nu} & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ &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

simm.

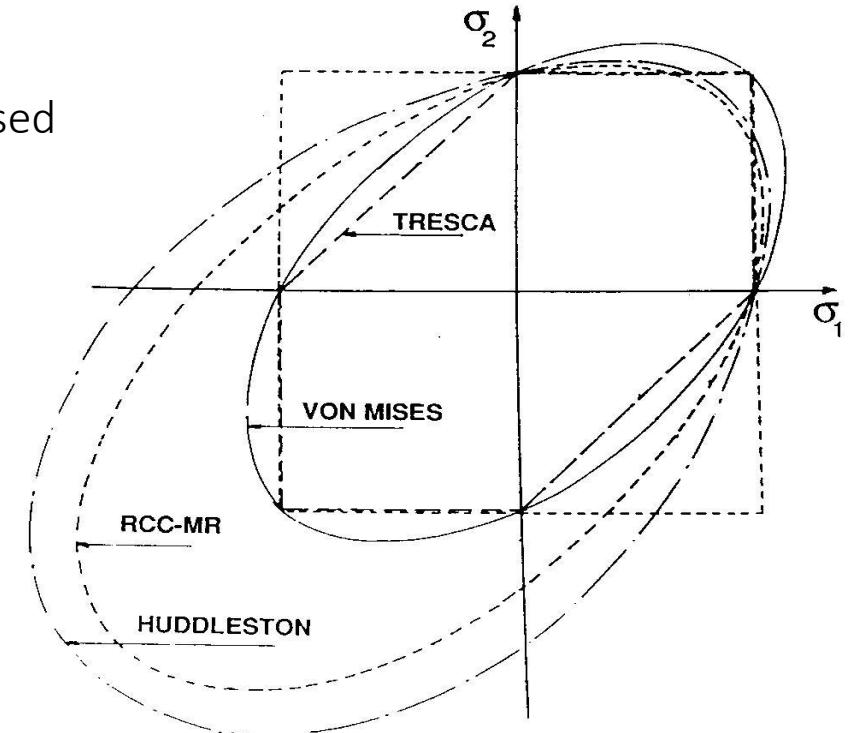


STRESS INTENSITY

To verify a structure, it is necessary to compare the stress state in the most stressed point of a structure with an admissible stress value



$$\sigma_{\text{von mises}} = \sqrt{\frac{1}{2} \left[(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2 \right] + 3(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)}$$



TRESCA : $\bar{\sigma}_{\text{TR}} = \text{Max} [|\sigma_2 - \sigma_1|, |\sigma_1 - \sigma_3|, |\sigma_3 - \sigma_2|]$

VON MISES : $\bar{\sigma}_{\text{VM}} = (1/\sqrt{2}) \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$

RCC-MR, addenda Nov.87 : $\bar{\sigma}_{\text{RCC}} = 0.867 \bar{\sigma}_{\text{VM}} + 0.133 (\sigma_1 + \sigma_2 + \sigma_3)$
(Austenitic steels)

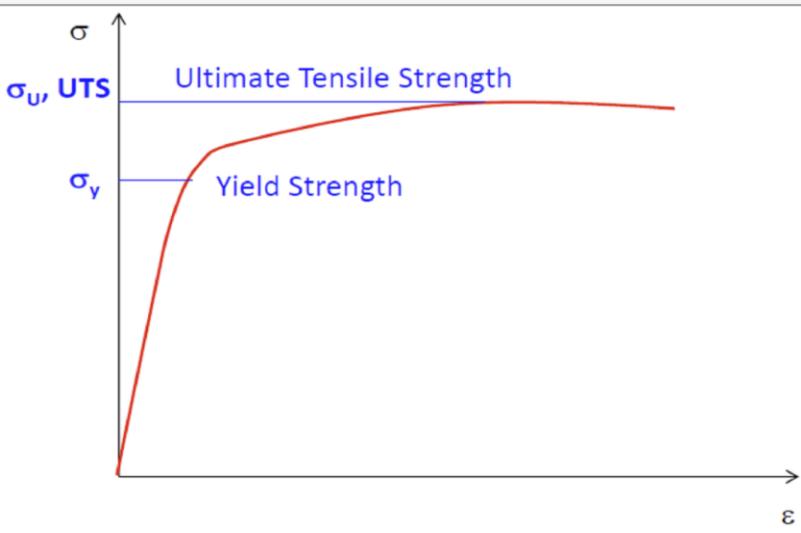


TENSIONE AMMISSIBILE

$$\sigma_{eq} = f(\sigma_t, \sigma_s, \chi_t, \chi_{tan}) < \sigma_a = \frac{\sigma_y}{1.5}$$

↓
cond. carico e vincoli
geometria

$\sigma_a > \sigma_a$ → materiali
certo d'ogni



Norma	Modello	Tensione ammissibile
Asme VIII Divisione 1.	Dbf	$s_m = \min\left(\frac{UTS}{3,5}, \frac{\sigma_y}{1,5}\right)$
Asme VIII Divisione 2.	Dbf + Dba	$s_m = \min\left(\frac{UTS}{2,4}, \frac{\sigma_y}{1,5}\right)$
En 13445.	Dbf	$f_d = \min\left(\frac{UTS}{2,4}, \frac{\sigma_y}{1,5}\right)$
En 13445.	Dba	$f_d = \min\left(\frac{UTS}{1,875}, \frac{\sigma_y}{1,5}\right)$



TENSIONE AMMISSIBILE

Design by Formulas (DBF)

il dimensionamento e la verifica sono basati su relazioni pre-confezionate (formulas) ideate per coprire, con adeguati coefficienti di sicurezza, tutte le principali situazioni che si è soliti incontrare nel progetto di un recipiente in pressione; le formule sono solitamente basate su modelli semplici o semi-empirici non molto accurati, per cui i coefficienti di sicurezza tendono ad essere più elevati.

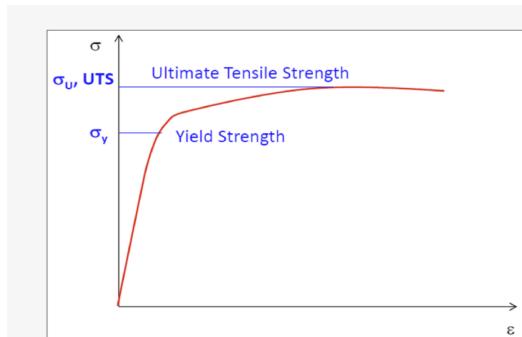
Design by Analysis (DBA)

il dimensionamento e la verifica sono basati su analisi accurate dell'effettivo stato di tensione, solitamente ottenibile solo con modelli basati sul Finite Element Method (FEM).

L'approccio DBA si rende necessario per i casi non coperti dalle relazioni relative al metodo DBF, ma viene impiegato anche in alternativa a quest'ultimo, salvo i casi in cui i modelli analitici semplici.

Fidando sulla maggiore accuratezza

dell'analisi, i coefficienti di sicurezza impiegati tendono ad essere più bassi.



Norma	Modello	Tensione ammissibile
Asme VIII Divisione 1.	Dbf	$S_m = \min\left(\frac{UTS}{3,5} \frac{\sigma_y}{1,5}\right)$
Asme VIII Divisione 2.	Dbf + Dba	$S_m = \min\left(\frac{UTS}{2,4} \frac{\sigma_y}{1,5}\right)$
En 13445.	Dbf	$f_d = \min\left(\frac{UTS}{2,4} \frac{\sigma_y}{1,5}\right)$
En 13445.	Dba	$f_d = \min\left(\frac{UTS}{1,875} \frac{\sigma_y}{1,5}\right)$



ANALISI STRUTTURALE

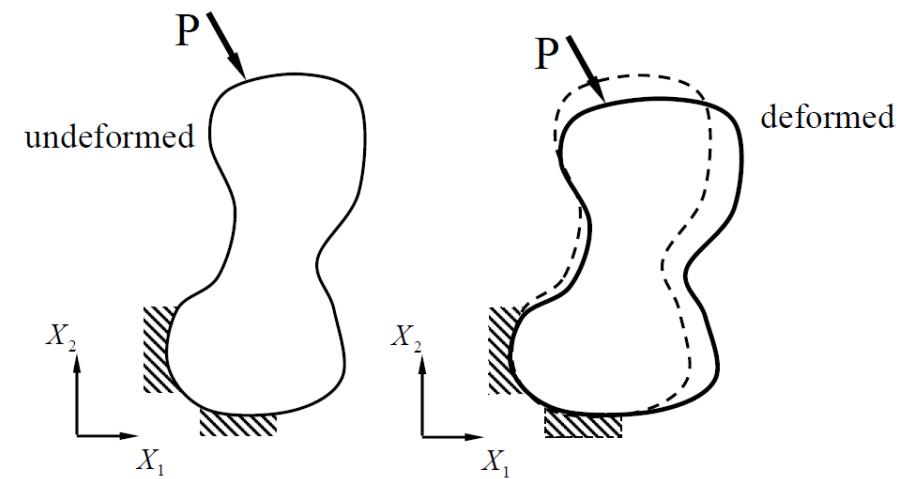
Problema fondamentale

Assegnata

- la regione entro la quale si vuole considerare il campo (la superficie che racchiude la geometria) (*Boundary*)
- la natura dei materiali contenuti entro la regione (*material properties*)
- le condizioni al contorno della regione e la posizione e l'intensità delle sorgenti (*boundary conditions*)

Vogliamo conoscere in ogni punto del materiale il valore di:

- spostamenti
- sforzi
- deformazioni



Geometry
Material properties
Boundary conditions

Stress, strains, displacements
at each material point



DESIGN CRITERIA

- Design criteria guarantee the material's integrity. (Not meeting the design criteria does not mean the component will actually fail!)
- Design Codes or Standards are provided (and are mandatory) by a government authority for common types of structures to ensure safe component operation and prevent harm, e.g. the EN13445 Standard for *unfired pressure vessels* is provided by the European committee for Standardization (CEN)
- Different types of components → different design criteria
- Design criteria define:
 - Material properties
 - Fabrication requirements
 - Inspection and testing requirements
 - Verification methods (design by testing/design by formulae – DBF / design by analysis - DBA). In case of design by analysis:
 - Load factors for different types of loads
 - Stress/strain limits

Finite element methods

The behavior of a continuous system is governed by partial differential equations.

$$\begin{cases} \nabla^2 u + \frac{1}{1-2\nu} \cdot \frac{\partial}{\partial x} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + \frac{X}{G} = 0 \\ \nabla^2 v + \frac{1}{1-2\nu} \cdot \frac{\partial}{\partial y} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + \frac{Y}{G} = 0 \\ \nabla^2 w + \frac{1}{1-2\nu} \cdot \frac{\partial}{\partial z} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + \frac{Z}{G} = 0 \end{cases}$$

Finite element methods are techniques suitable for approximating these differential equations with a system of algebraic equations in a finite number of unknown's parameters



Small Displacement Theory & Rotations

- Small displacement theory:

$$\sin(\Phi) = \tan(\Phi) = \Phi$$

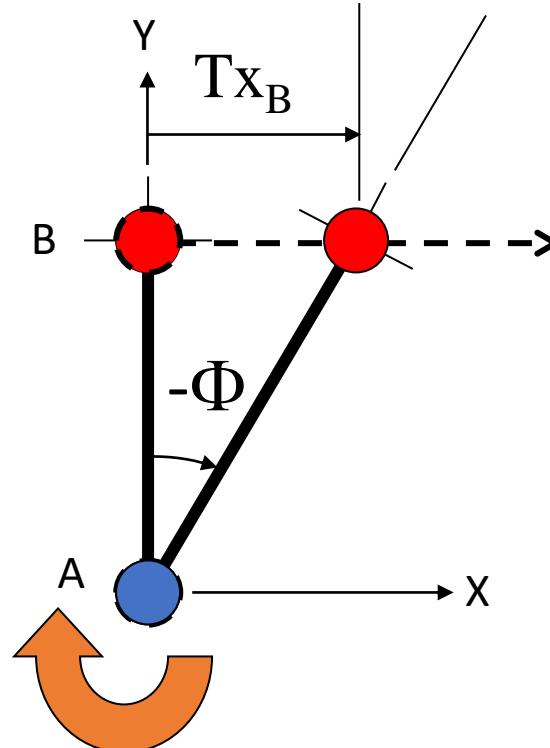
$$\cos(\Phi) = 1$$

- For Rz @ A

$$Rz_B = Rz_A = \Phi$$

$$Tx_B = (-\Phi) * L_{AB}$$

$$Ty_B = 0$$



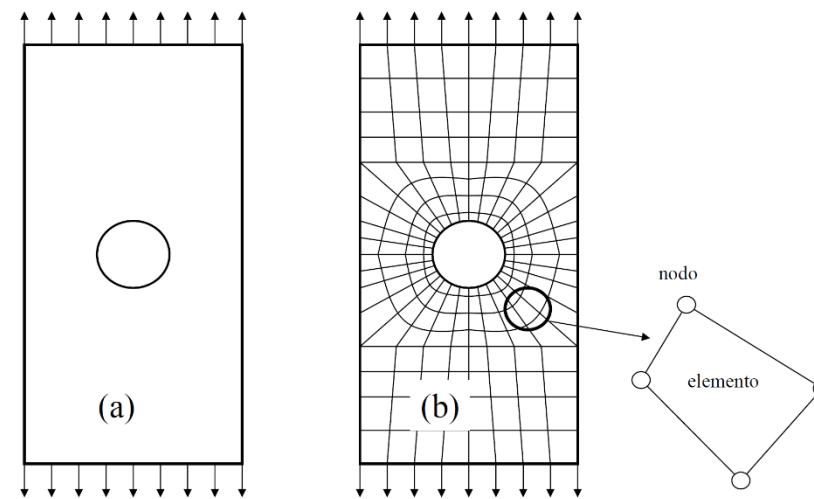
METODO DEGLI ELEMENTI FINITI

- Metodo per la risoluzione numerica di una equazione differenziale, sia essa alle derivate totali o parziali
- Più precisamente si tratta di un metodo per approssimare una equazione differenziale con un sistema di equazioni algebriche

Il metodo è fondato sulla suddivisione del dominio di partenza in sottodomini che sono elementi di forma e comportamento tipici.

Descrizione

- Divide la struttura in diversi elementi (parti di struttura)
- Riconnette gli elementi ai nodi
- Tale processo porta ad un set di equazioni algebriche risolvibili simultaneamente



In questo metodo si discretizza il continuo, che ha infiniti gradi di libertà, con un insieme di elementi di dimensioni finite, tra loro interconnessi in punti predefiniti (nodi).



METODO DEGLI ELEMENTI FINITI

Tutti i fenomeni ingegneristici sono descrivibili in un set di equazioni di governo e di condizioni vincolari, che con il metodo agli elementi finiti si riducono ad un set di equazioni algebriche risolvibili simultaneamente.

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



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

$[K]$: proprietà

$\{u\}$: comportamento

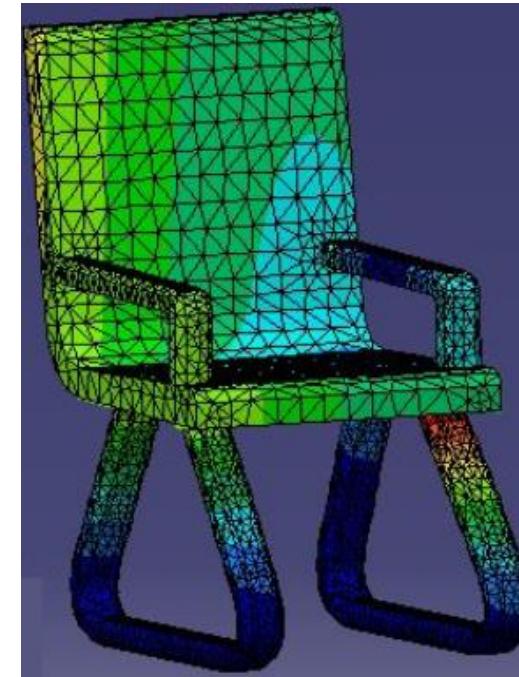
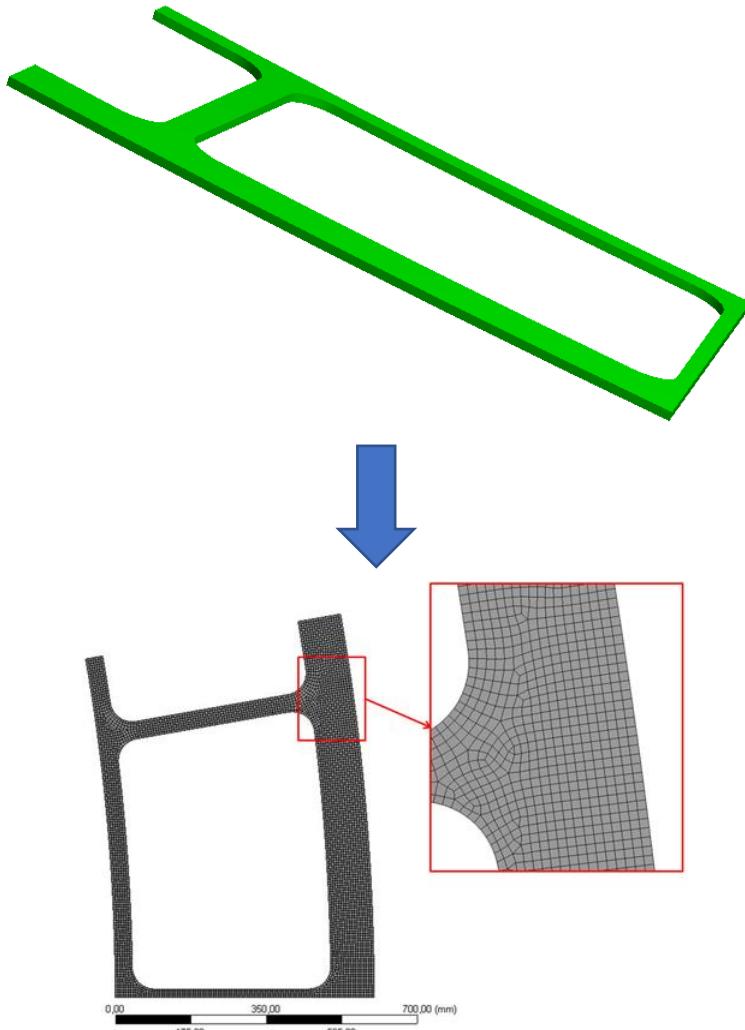
$\{F\}$: azione

	Proprietà	Comportamento	Azione
Elastico	Rigidezza	Deformazioni	Forze
Termico	Conduttività	Temperatura	Sorgenti di calore
Fluido-dinamica	Viscosità	Velocità	Forze volumetriche



METODO DEGLI ELEMENTI FINITI

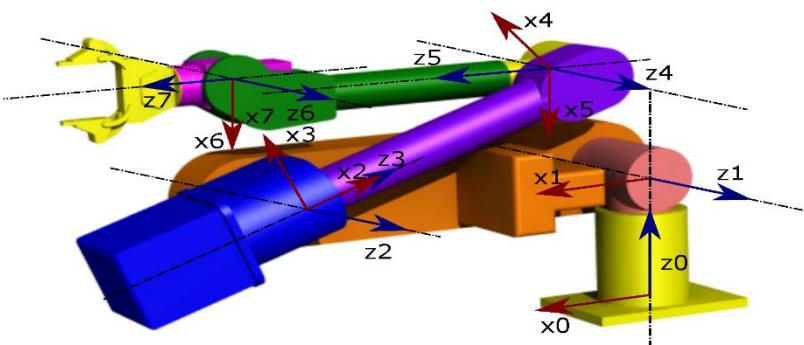
Il dominio è suddiviso in piccoli elementi semplici che condividono i vincoli e i carichi attraverso i nodi



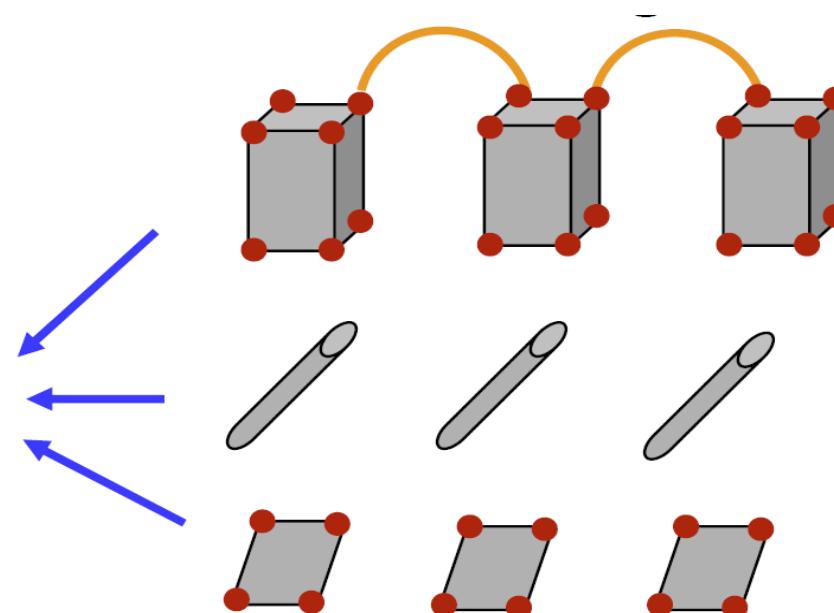
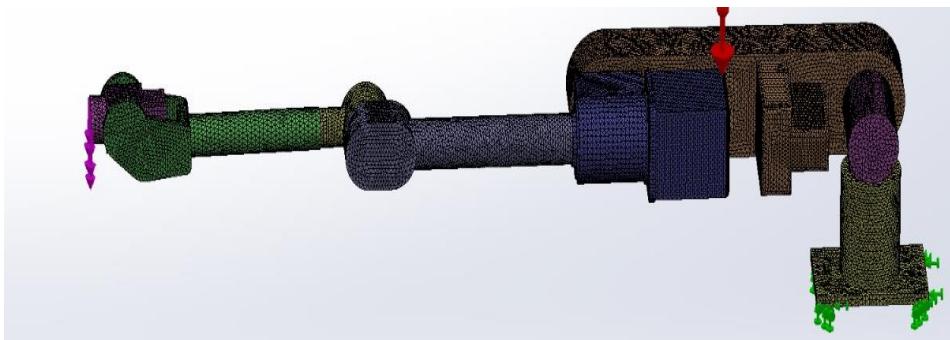
1. Per ogni nodo si calcolano Forze e Spostamenti → reazioni vincolari
2. Per ogni elemento deformazioni(ε), stress (σ)



METODO DEGLI ELEMENTI FINITI

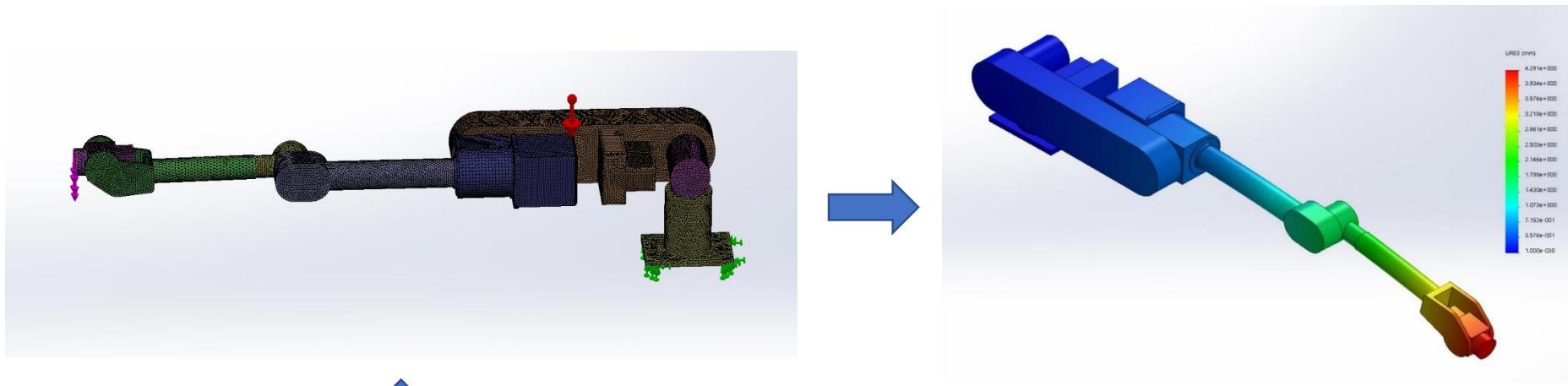


I nodi degli elementi adiacenti trasmettono vincoli e carichi tra gli elementi

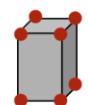


METODO DEGLI ELEMENTI FINITI

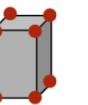
Le equazioni algebriche per ogni elemento sono ricavate in maniera semplice e combinate per ottenere le variabili incognite ai nodi



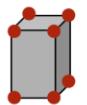
$$[K^E]\{u^E\} = \{F^E\}$$



$$[K^E]\{u^E\} = \{F^E\}$$



$$[K^E]\{u^E\} = \{F^E\}$$



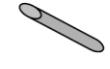
$$[K^E]\{u^E\} = \{F^E\}$$



$$[K^E]\{u^E\} = \{F^E\}$$



$$[K^E]\{u^E\} = \{F^E\}$$



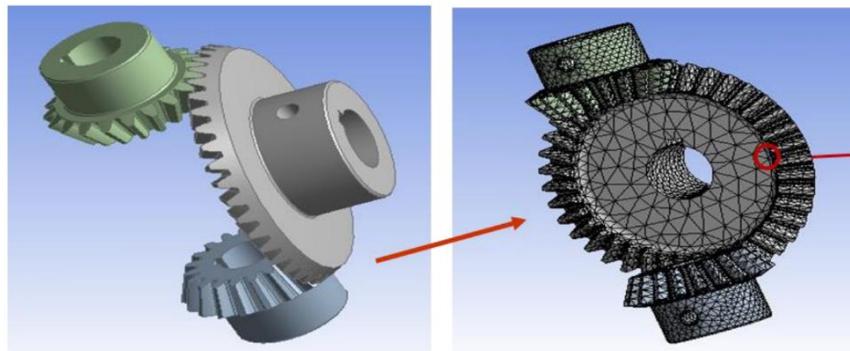
$$\{u\} = [K]^{-1}\{F\}$$



PREPROCESSING

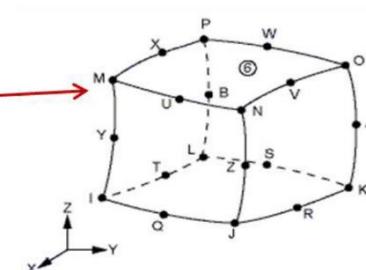
- Definisco la Geometria
- Assegno le proprietà dei materiali
- Definisco il tipo di elemento da utilizzare
- Genero la mesh della struttura

ELEMENTI SOLIDI - BRICK

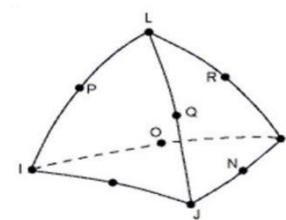


3D Solids

→ 3D Element



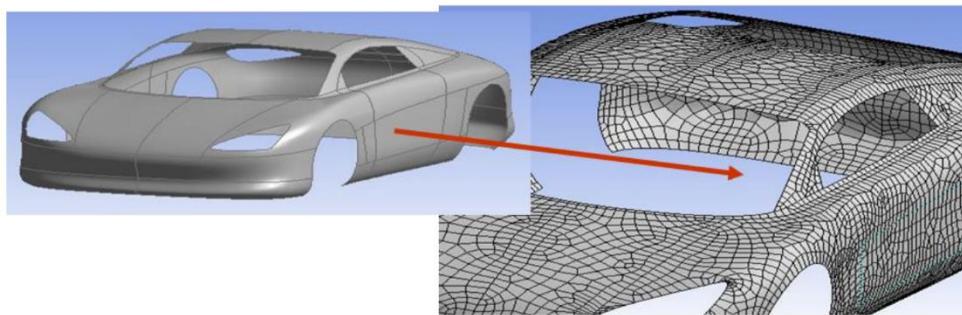
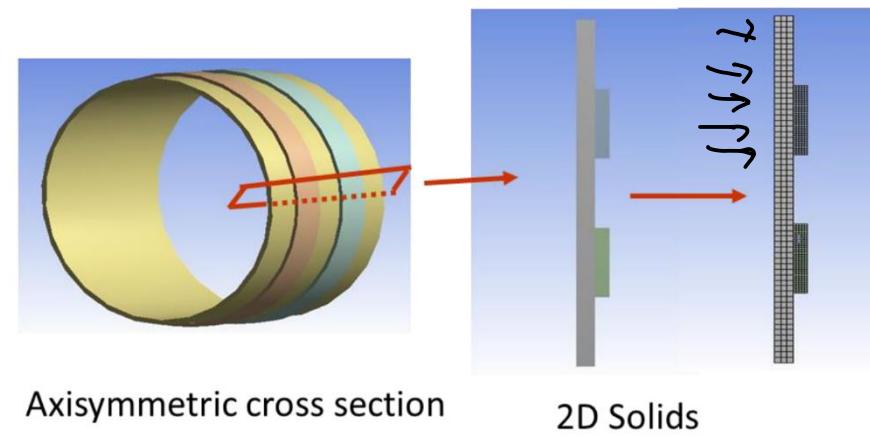
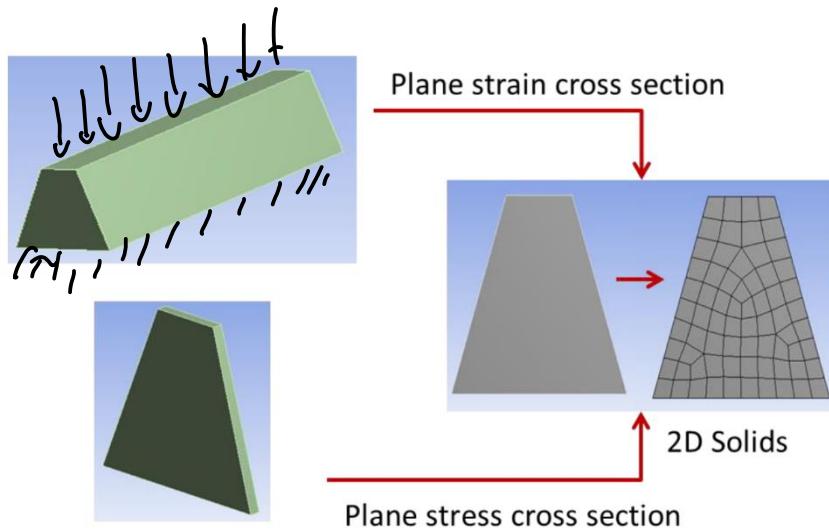
Hex Element



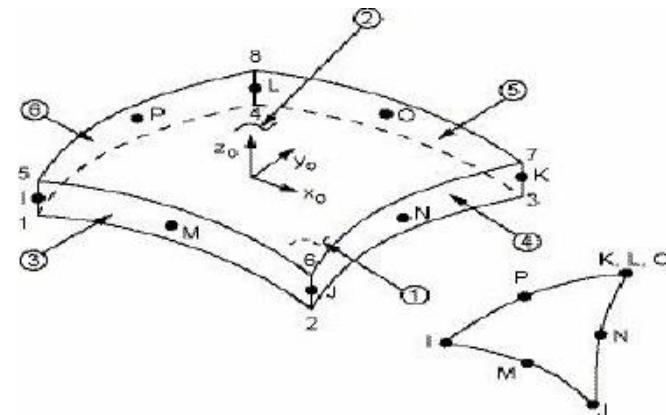
Tet Element



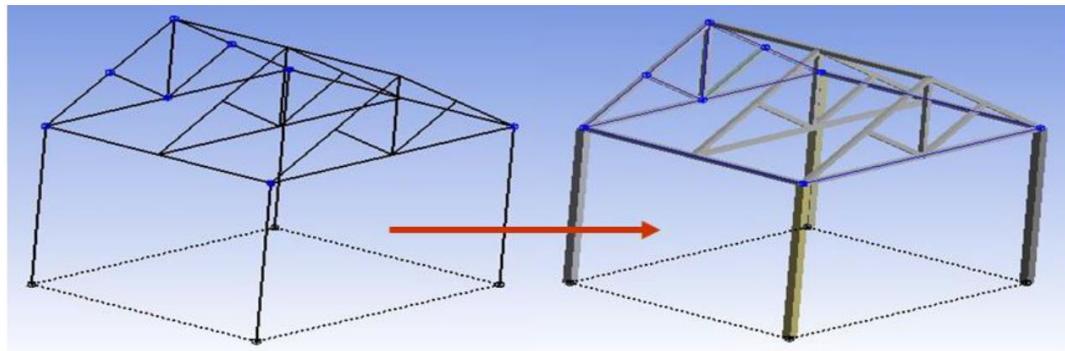
SHELL ELEMENTS



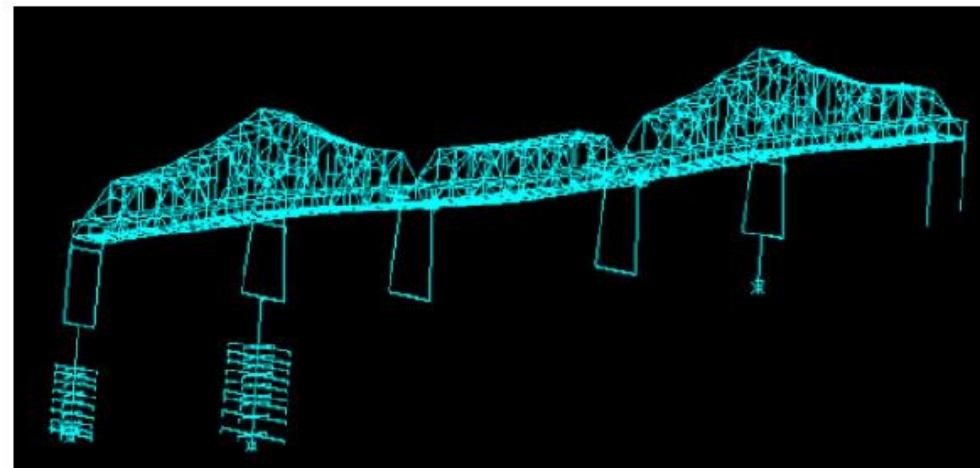
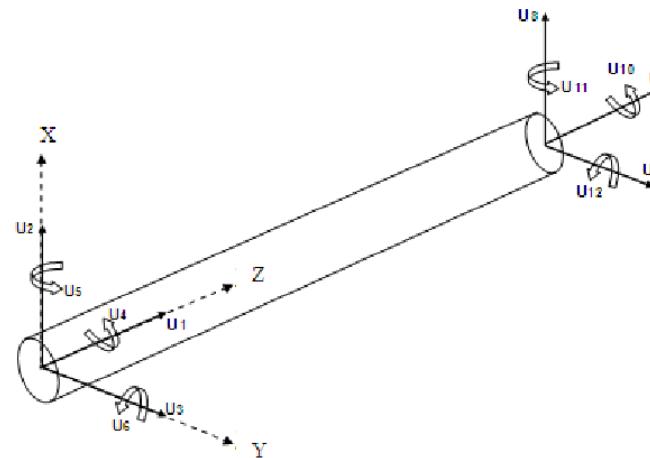
Surface Body



MONODIMENSIONAL ELEMENTS - BEAM



Line Body

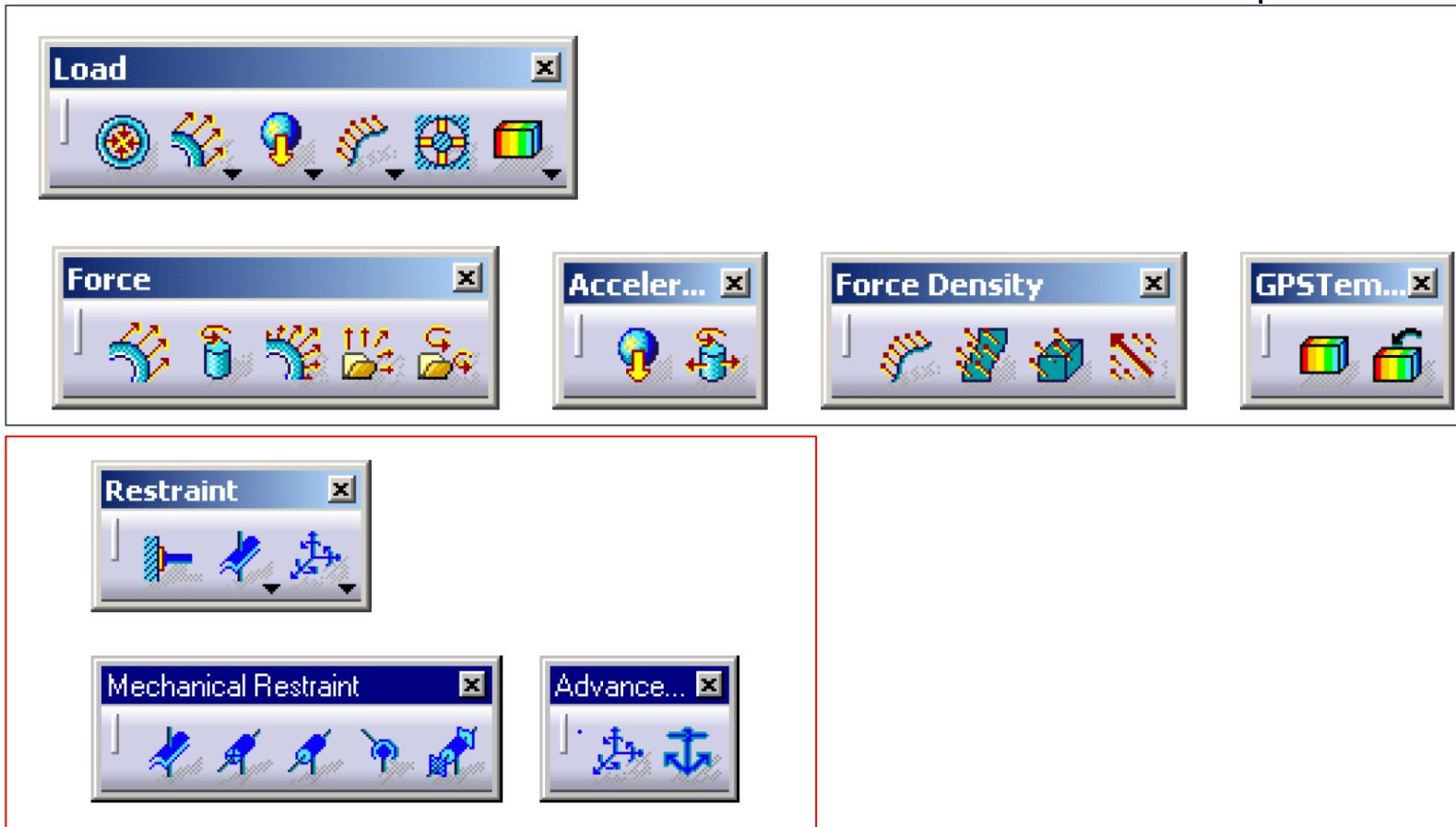


SOLUTION

- Application of loads
- Application of Restraints
- Calculation of results



Pressures
Distributed loads
Accelerations
Force density
Temperature



POST-PROCESSING

Analisi dei risultati e confronto con le tensioni ammissibili



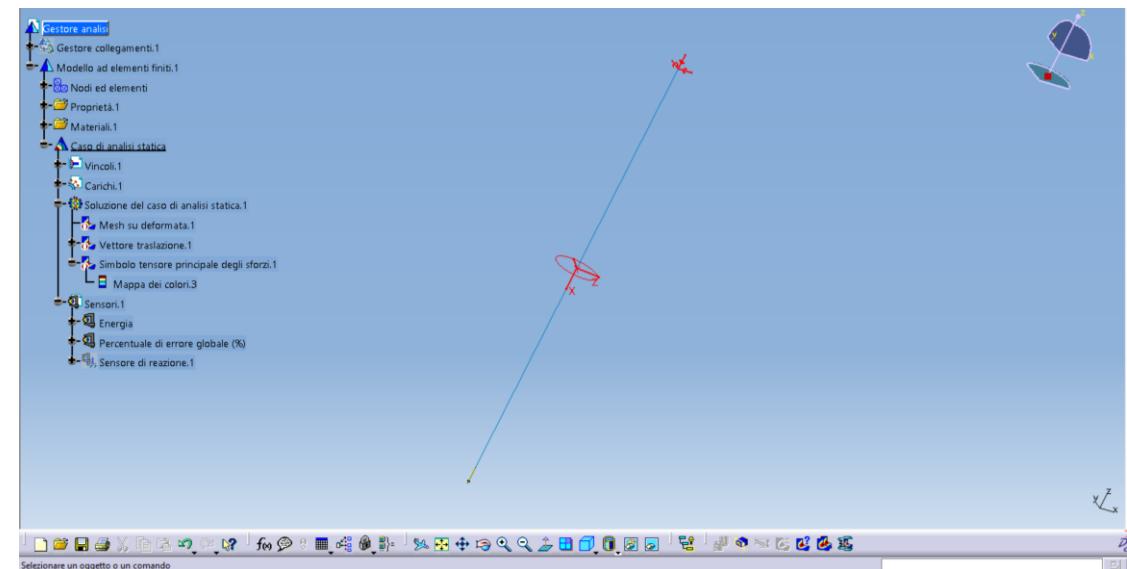
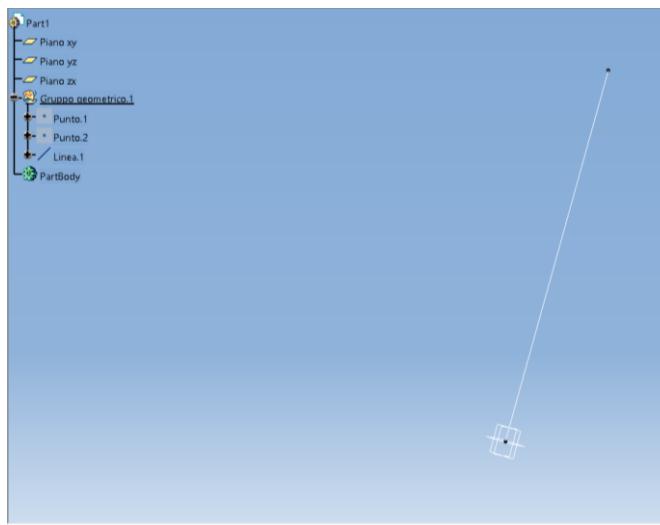
Per una prima verifica della correttezza del modello è buona pratica verificare l'equilibrio della struttura alla traslazione e alla rotazione, calcolando le reazioni vincolari mediante la creazione di un sensore.

ESEMPIO – 1

Elementi monodimensionali

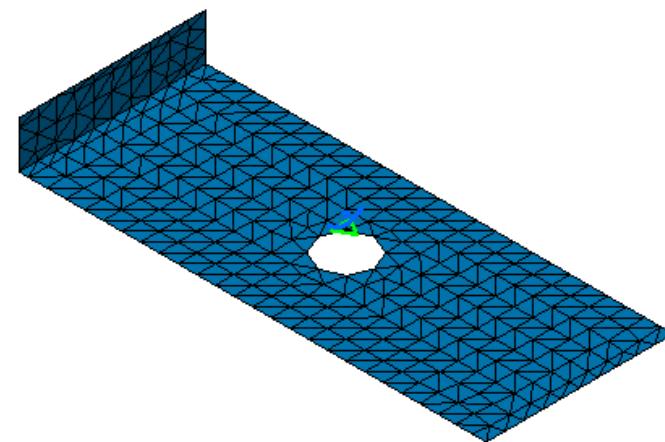
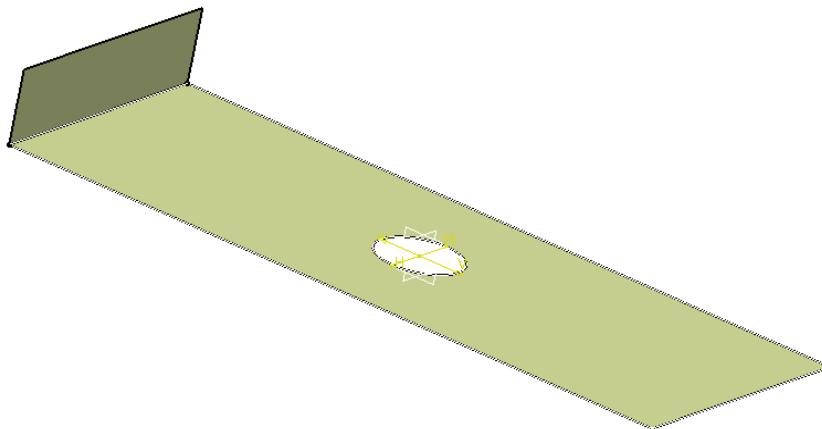


- You cannot select a sketch geometry.
- You cannot mesh 1D body belonging to hybrid body.



ESEMPIO – 2

Elementi shell



ESEMPIO – 3 Elementi solidi

