



# Progettazione Meccanica CAD/CAE Integrata

A.A. 2025/2026

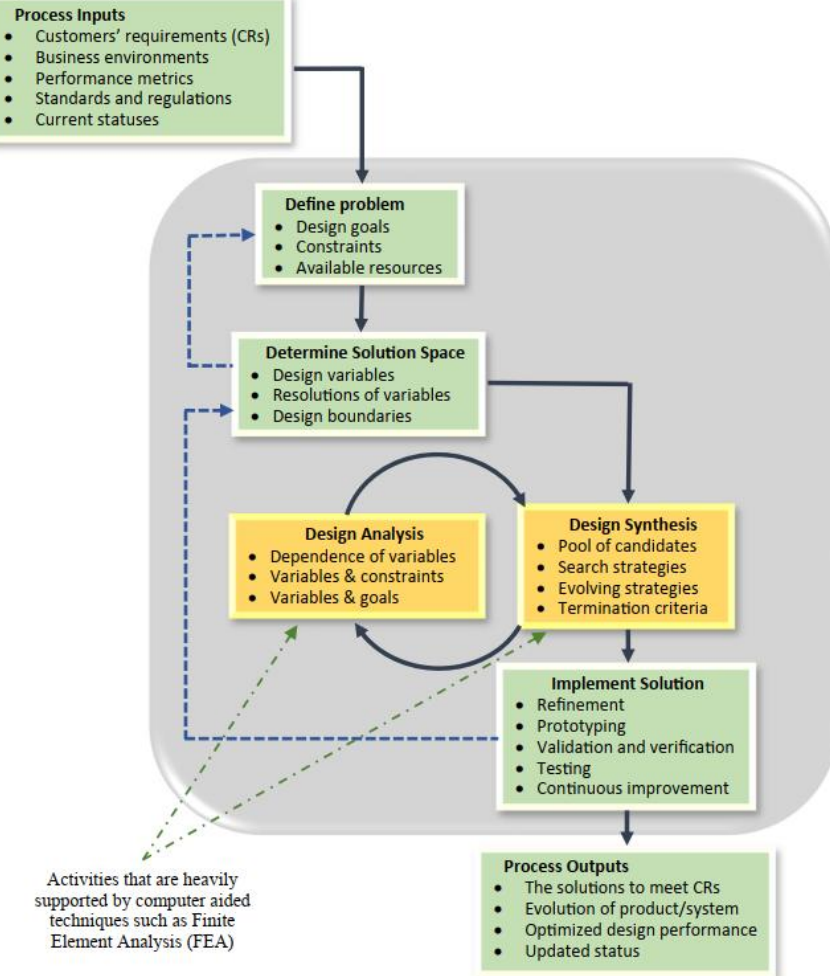
## Analisi strutturale in ambiente virtuale Introduzione

Ph.D. Eng. Domenico Marzullo



**UNIVERSITÀ  
DEGLI STUDI  
DI TRIESTE**

**Dipartimento di Ingegneria e Architettura  
Università degli Studi di Trieste**



Engineering design process:

Set of steps that a designer follows to obtain the **solution** to a formulated problem.

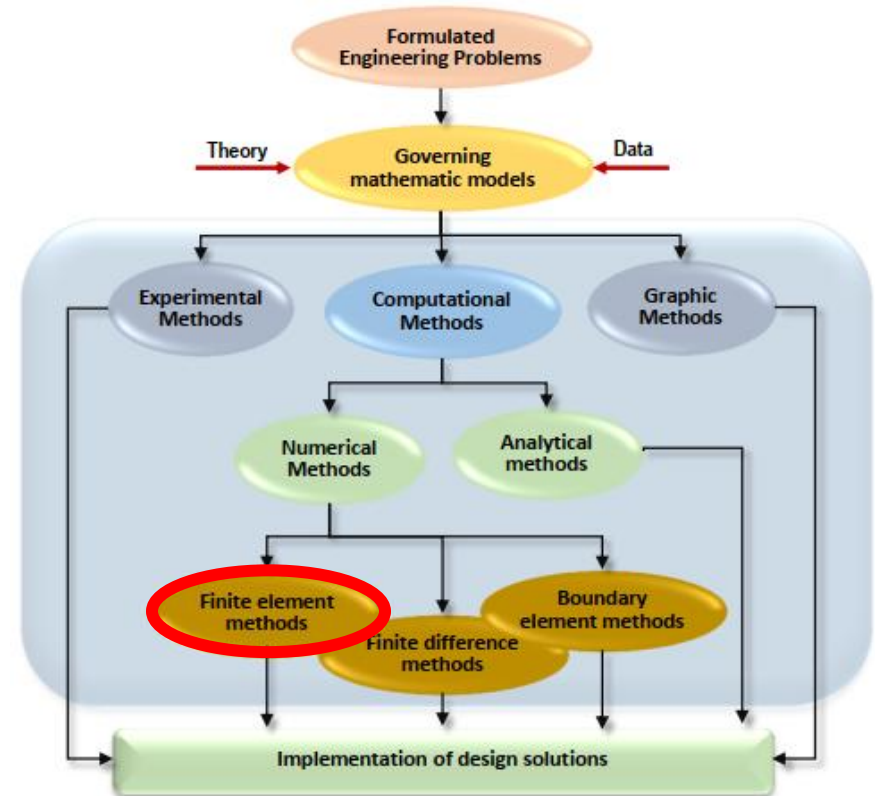
The problem formulation is strictly affected by the process inputs (customers' needs, requirements, standards, etc.).

Computer Aided techniques support designers through the design process.

An engineering problem is formulated as a mathematical model.

Numerical methods are the most widely used computational tools in engineering practices.

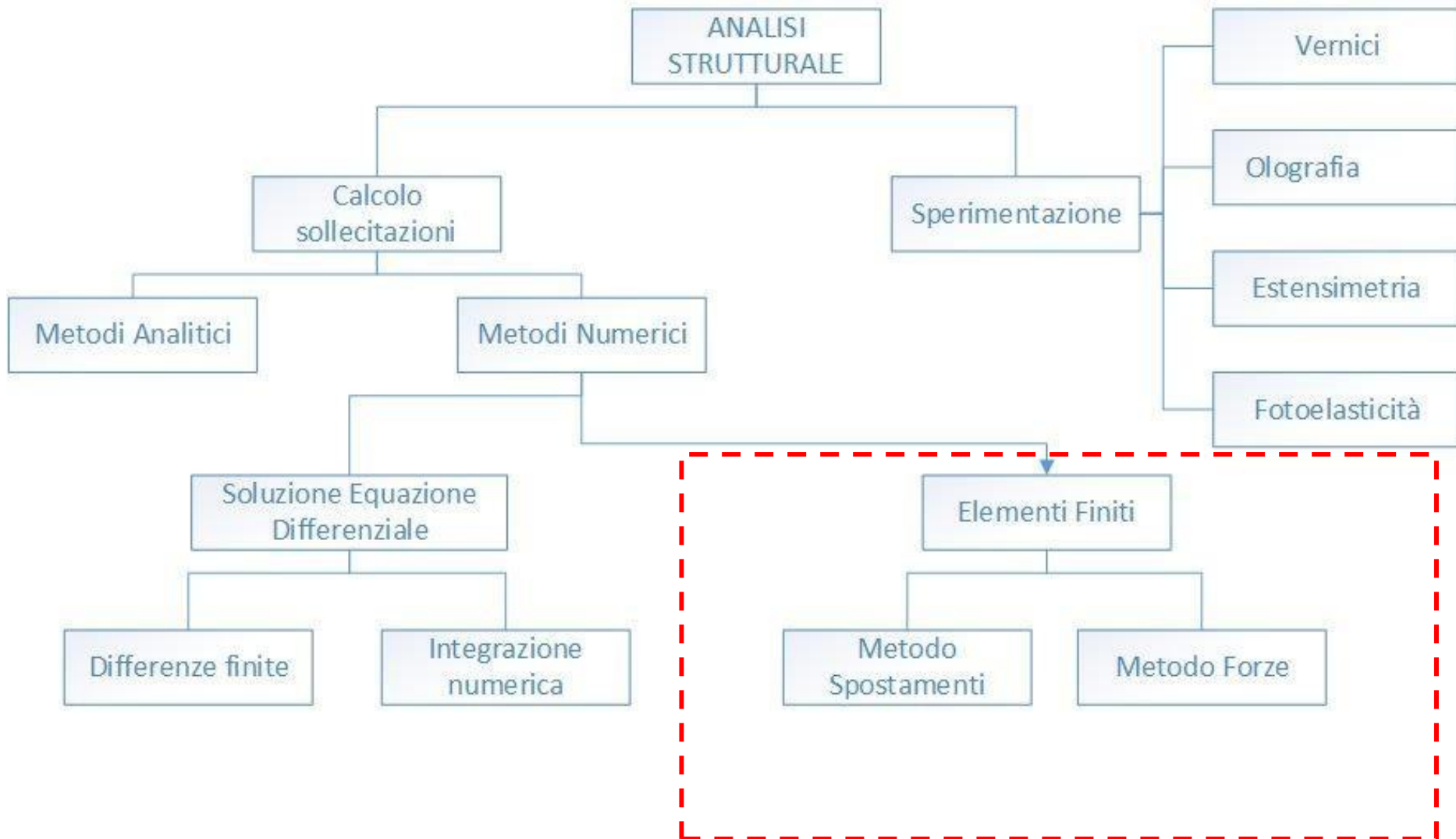
FEM is a **numerical method** for finding approximate solutions to an engineering problem described by a set of equations.



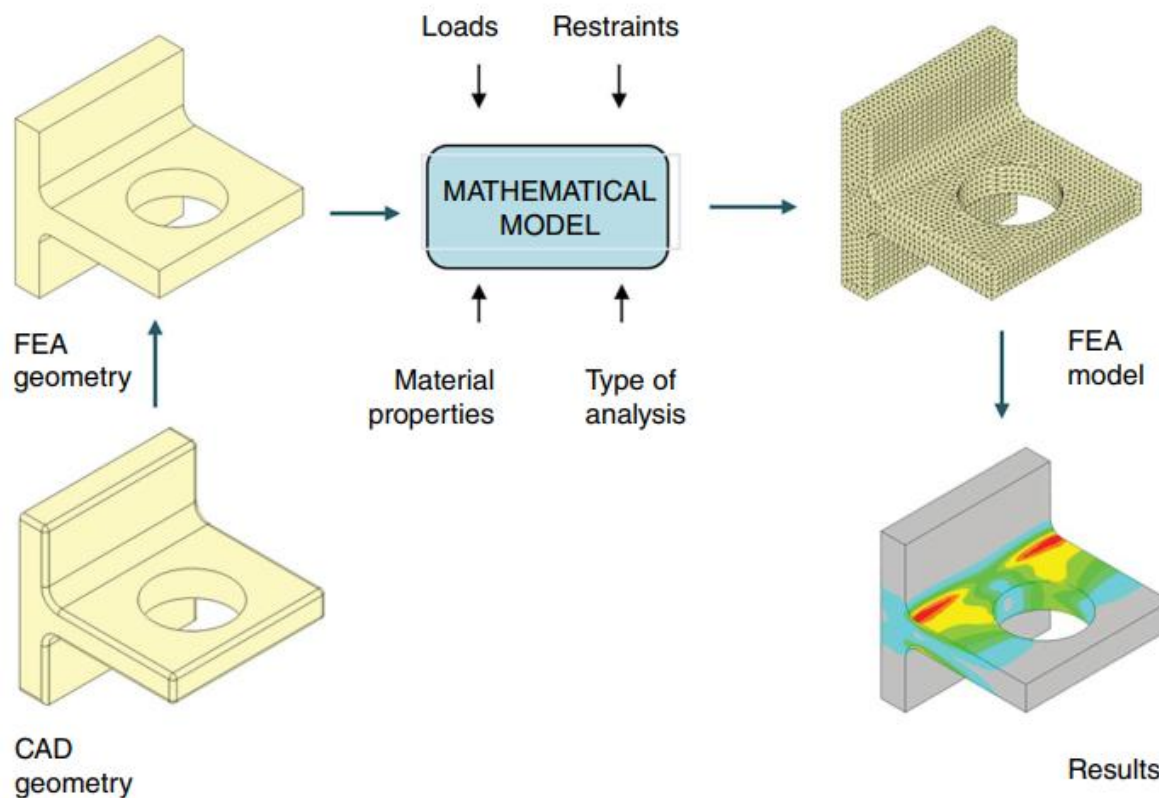
## ANALISI STRUTTURALE - Generalità

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

# ANALISI STRUTTURALE - Generalità



FEA uses continuous digital models that are discretised in subregions (**finite elements**) to analyse the **behaviour of a system** under defined **boundary and/or initial conditions** (loads, restraints, materials, temperatures, etc.).

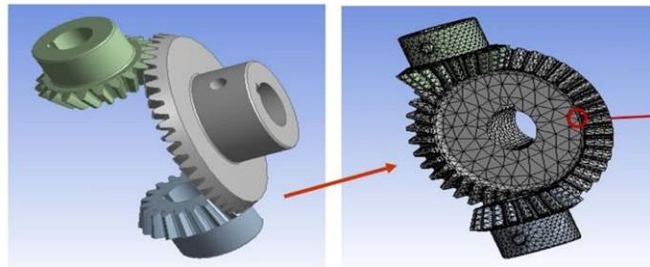


Reference: Paul M. Kurowski. Finite Element Analysis for Design Engineers. 3<sup>rd</sup> Edition. SAE International, 2023.

FEA is based on the **module-based design**: a complex problem can be treated as a set of simple subproblems. The solution to the complex problem can be found by assembling the solutions of the subproblems.

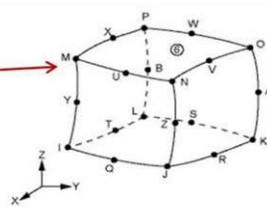
FEA uses the Finite Element Method (**FEM**) to solve engineering problems.

FEA breaks down a large problem into smaller problems (**finite elements**) represented by one or a set of **partial differential equations** (PDEs) in respective physical disciplines. The whole problem is represented by a **large system of PDEs** as the set of all the simple equations that govern each finite element.

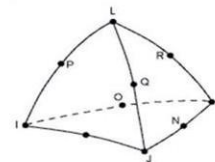


3D Solids

→ 3D Element



Hex Element



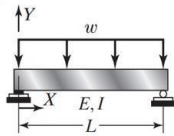
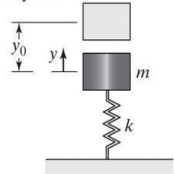
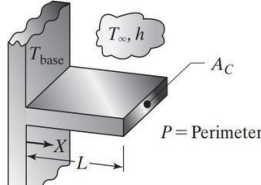
Tet Element

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

PDEs

In general, engineering problems are mathematical models of physical situations. Mathematical models of many engineering problems are differential equations with a set of corresponding boundary and/or initial conditions. The differential equations are derived by applying the fundamental laws and principles of nature to a system or a control volume. These governing equations represent balance of mass, force, or energy.

Examples of governing differential equations, boundary conditions, initial conditions, and exact solutions for some engineering problems

Problem Type	Governing Equation, Boundary Conditions, or Initial Conditions	Solution
A beam: 	$EI \frac{d^2 Y}{dX^2} = \frac{wX(L - X)}{2}$ Boundary conditions: at $X = 0, Y = 0$ and at $X = L, Y = 0$	Deflection of the beam $Y$ as the function of distance $X$ : $Y = \frac{w}{24EI} (-X^4 + 2LX^3 - L^3X)$
An elastic system: 	$\frac{d^2 y}{dt^2} + \omega_n^2 y = 0$ where $\omega_n^2 = \frac{k}{m}$ Initial conditions: at time $t = 0, y = y_0$ and at time $t = 0, \frac{dy}{dt} = 0$	The position of the mass $y$ as the function of time: $y(t) = y_0 \cos \omega_n t$
A fin: 	$\frac{d^2 T}{dX^2} - \frac{hp}{kA_c} (T - T_\infty) = 0$ Boundary conditions: at $X = 0, T = T_{base}$ as $L \rightarrow \infty, T = T_\infty$	Temperature distribution along the fin as the function of $X$ : $T = T_\infty + (T_{base} - T_\infty) e^{-\sqrt{\frac{hp}{kA_c}} X}$

In any given engineering problem, there are two sets of design parameters that influence the way in which a system behaves. First, there are those parameters that provide information regarding the natural behaviour of a given system. These parameters include material and geometric properties such as modulus of elasticity, thermal conductivity, viscosity, and area, and second moment of area.

Problem Type	Examples of Parameters That Characterize a System
<b>Solid Mechanics Examples</b>	
<p>A truss</p>	Modulus of elasticity, $E$ ; member length, $L$ ; cross-sectional area, $A$
<p>An elastic plate</p>	Modulus of elasticity, $E$ ; length, $L$ ; cross-sectional area, $A$
<p>A beam</p>	Modulus of elasticity, $E$ ; member length, $L$ ; second moment of area, $I$
<p>A shaft</p>	Modulus of rigidity, $G$ ; member length, $L$ ; polar moment of inertia of the area, $J$
<b>Heat Transfer Examples</b>	
<p>A wall</p>	Thermal conductivity, $K$ ; thickness, $L$ ; area, $A$

On the other hand, there are parameters that produce **disturbances** in a system.

Examples of these parameters include external forces, moments, temperature difference across a medium, and pressure difference in fluid flow. The **system characteristics** dictate the **natural behaviour of a system**, and they always appear in the **homogenous part** of the solution of a governing differential equation. In contrast, the **parameters that cause the disturbances** appear in the **particular solution**.

Problem Type	Examples of Parameters that Produce Disturbances in a System
Solid Mechanics	External forces and moments; support excitation
Heat Transfer	Temperature difference; heat input
Fluid Flow and Pipe Networks	Pressure difference; rate of flow
Electrical Network	Voltage difference

There are many practical engineering problems for which we cannot obtain exact solutions. This inability to obtain an exact solution may be attributed to either the complex nature of governing differential equations or the difficulties that arise from dealing with the boundary and initial conditions.

To deal with such problems, we resort to **numerical approximations**. In contrast to analytical solutions, which show the exact behavior of a system at any point within the system, numerical solutions approximate exact solutions only at discrete points, called nodes. The first step of any numerical procedure is **discretization**. This process divides the medium of interest into a number of small subregions (elements) and nodes. There are two common classes of numerical methods:

1. finite difference methods
2. finite element methods

There are many practical engineering problems for which we **cannot** obtain **exact solutions**. This inability to obtain an exact solution may be due to either the **complex** nature of the governing differential equations or the **difficulties** in dealing with the boundary and initial conditions.

To deal with such problems, we resort to **numerical approximations**. In contrast to analytical solutions, which show the exact behaviour of a system at any point within the system, numerical solutions **approximate exact solutions** only at **discrete points**, called nodes

Moreover, physical phenomena are **continuous** (analog) and we use computers (digital) for solving continuous problems



problems shall be **discretised** -> **Finite Element Method (FEM)**.

The result of the discretisation is a **mesh** composed of **nodes** connected with entities called **elements**.

The calculations are basically done at the nodes, and the **results are interpolated** through the elements.

The discretization resolution (number of nodes) affects the **accuracy of the results**.

Let's calculate the area of a circle with  $r=1 \rightarrow A = \pi r^2 = \pi$

Circle approximated by 3 elements (triangle)

$$\overline{AO} = r$$

$$\overline{AD} = \overline{AO} * \cos(30^\circ) = r * \cos(30^\circ) = r * \frac{\sqrt{3}}{2}$$

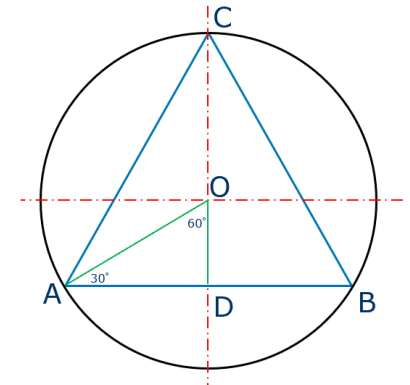
$$b = \overline{AB} = 2 * \overline{AD} = 2 * r * \frac{\sqrt{3}}{2} = r * \sqrt{3}$$

$$\left. \begin{array}{l} b = r * \sqrt{3} = 1,732 \\ h = \frac{3}{2}r = 1,5 \end{array} \right\}$$

$$\rightarrow A_{tr} = \frac{b * h}{2} = \frac{1,732 * 1,5}{2} = 1,299$$

$$\overline{OD} = r * \cos(60^\circ) = r * \frac{1}{2}$$

$$h = \overline{CD} = \overline{OD} + \overline{OC} = \frac{r}{2} + r = \frac{3}{2}r$$



$$err_{tr} = 1 - \frac{A_{tr}}{A} = 1 - \frac{1,299}{\pi} = 58,7\%$$

Let's calculate the area of a circle with  $r=1 \rightarrow A = \pi r^2 = \pi$

Circle approximated by 3 elements (triangle)

$$b = r * \sqrt{3} = 1,732 \quad h = \frac{3r}{2} = 1,5 \quad A_{tr} = \frac{b*h}{2} = \frac{1,732*1,5}{2} = 1,299$$

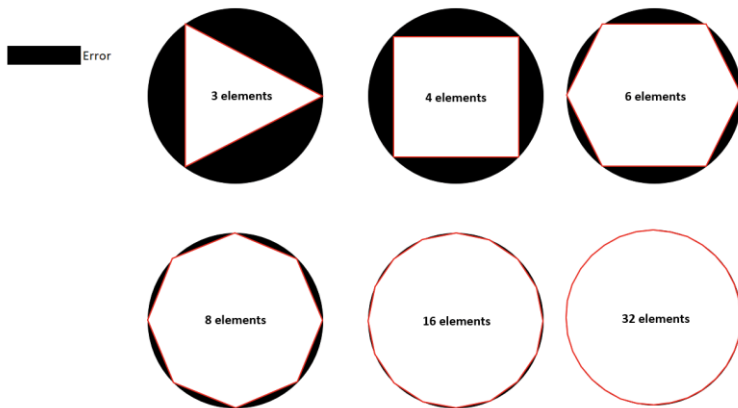
$$err_{tr} = 1 - \frac{A_{tr}}{A} = 1 - \frac{1,299}{\pi} = 58,7\%$$

Circle approximated by 4 elements (square)

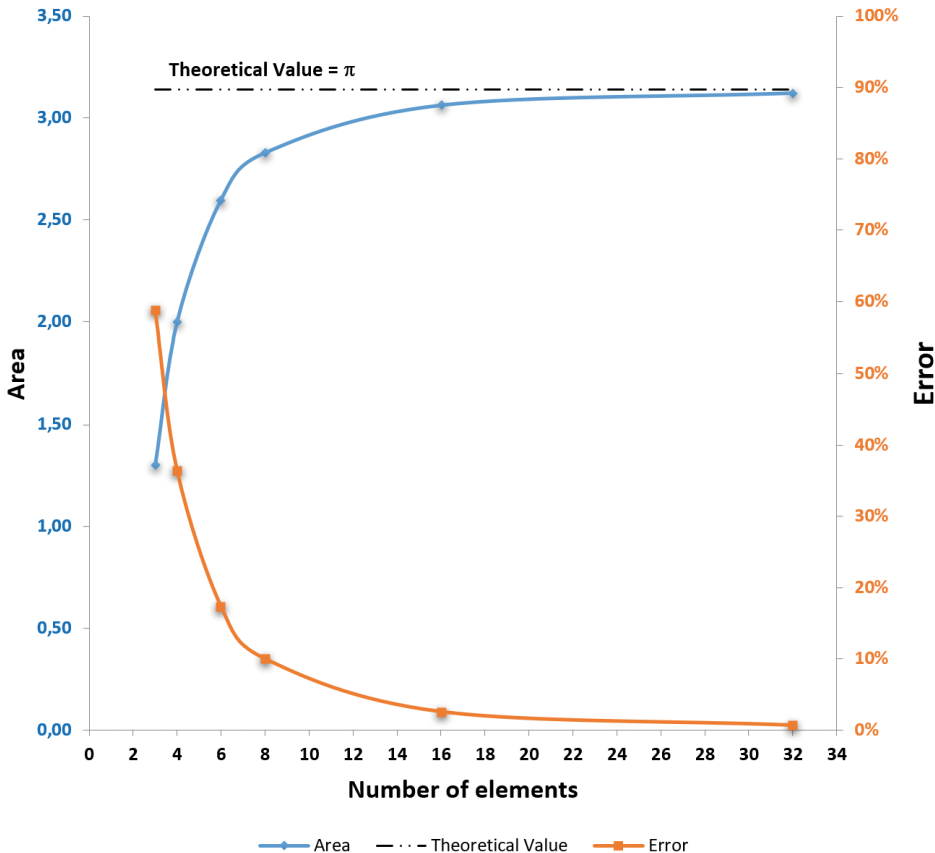
$$b = r * \sqrt{2} = 1,414$$

$$A_{sq} = b^2 = 1,414^2 = 2$$

$$err_{sq} = 1 - \frac{A_{sq}}{A} = 1 - \frac{2}{\pi} = 36,3\%$$



MATCHING SHAPE	NUMBER OF ELEMENTS	b EDGE LENGTH	h APOTHEM LENGTH	AREA (WHITE PORTION)		ERROR (BLACK PORTION)
TRIANGLE	3	1.7320	1.5	$(b*h)/2$	1.2990	59%
SQUARE	4	1.4142	—	$b^2$	1.9999	36%
HEXAGON	6	1.0000	0.8660	$6*(b*h)/2$	2.5980	17%
OCTAGON	8	0.7653	0.6238	$8*(b*h)/2$	2.8279	10%
HEXADECAGON	16	0.3902	0.9807	$16*(b*h)/2$	3.0611	3%
TRIACONTADIGON	32	0.1960	0.9952	$32*(b*h)/2$	3.1208	1%

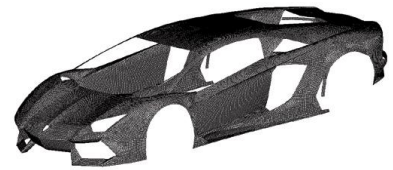
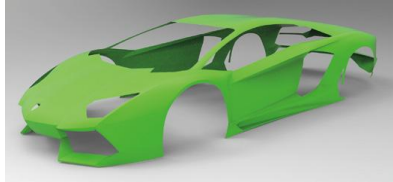


From 16 elements to 32 elements, the error decreases by 2%

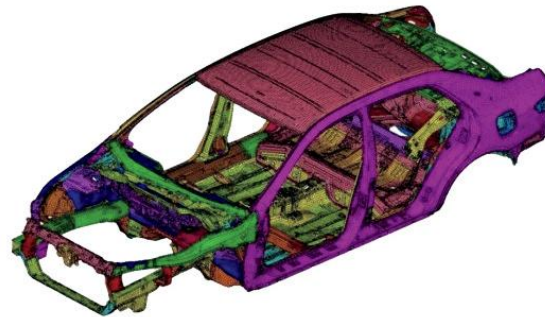
The number of nodes and elements shall be the minimum required to achieve the desired level of accuracy within the acceptable computation time.

**Balancing** accuracy, computation time, and mesh density (number of nodes and elements).

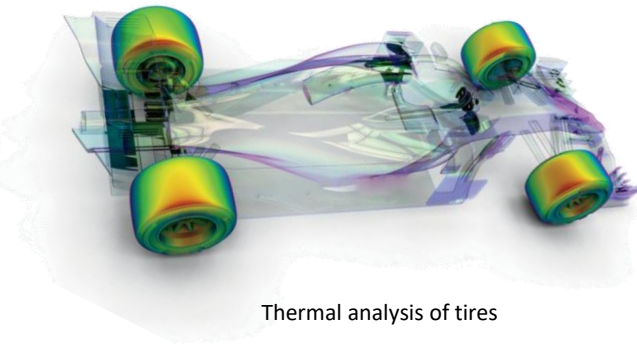
Reference: Dominique Madier. Practical Finite Element Analysis for Mechanical Engineers. FEA Academy, 2020.



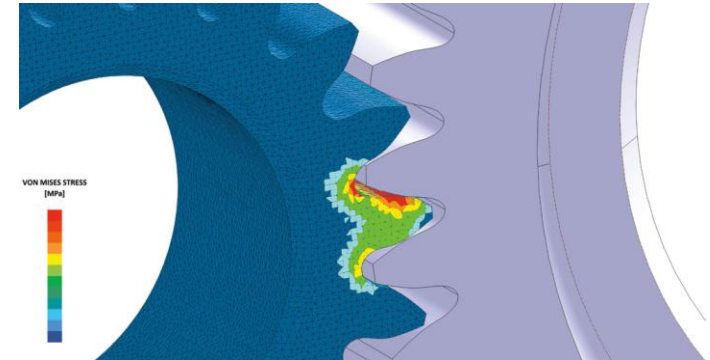
Modal analysis of the body of a Lamborghini Aventador



Stress analysis of a car chassis



Thermal analysis of tires

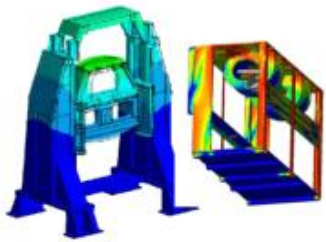


Contact analysis in a gear system

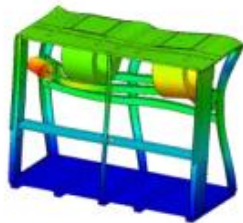


Study of the In-Flight Wing Up Bending of a Commercial Aircraft

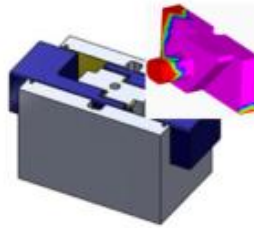
Reference: Dominique Madier. Practical Finite Element Analysis for Mechanical Engineers. FEA Academy, 2020.



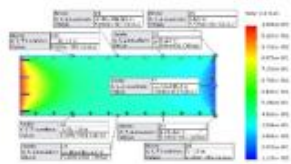
(1) Structural analysis



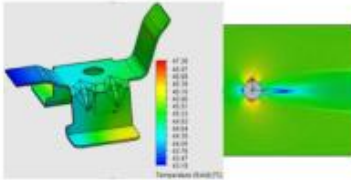
(2) Modal analysis



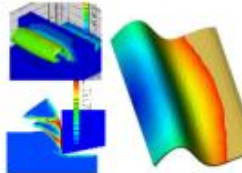
(3) Fatigue analysis



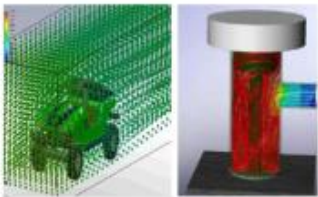
(4) Heat transfer



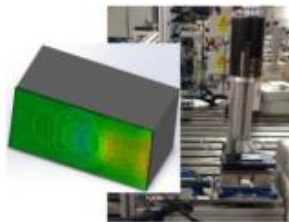
(5) Conjugate heat transfer



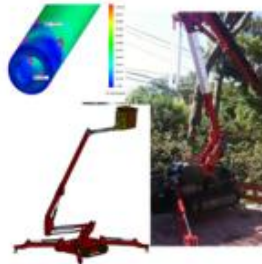
(6) Manufacturing processes



(7) Flow simulation



(8) Fluid structural interaction (FSI)



(9) Failure diagnosing

## Structural Analysis

determines the displacement and stress under static loading conditions

## Modal analysis

evaluate the natural frequencies and mode shapes of a structure

## Fatigue analysis

predicts the effect of the cyclic loading on the part/assembly. Fatigue is responsible for approximately 80% of all structural failures.



# FEA process



The FEA process is based on three main steps:

## **Pre-processing**

- Defining geometry
- Assigning material properties
- Applying loads
- Defining restraints
- Selecting mesh elements
- Meshing

## **Simulation**

- Computing numerical solution

## **Post-processing**

- Reading results
- Checking the validity of results



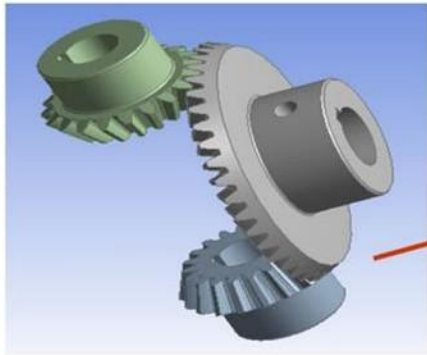
# Metodo degli Elementi Finiti

# METODO DEGLI ELEMENTI FINITI

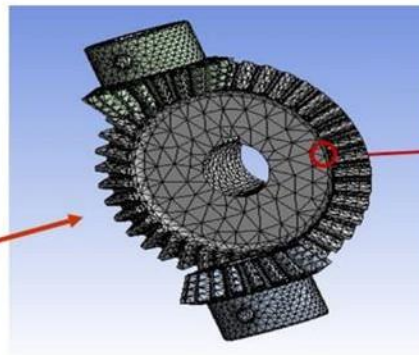
the finite element method uses integral formulations rather than difference equations to create a system of algebraic equations. Moreover, a continuous function is assumed to represent the approximate solution for each element. The complete solution is then generated by connecting or assembling the individual solutions, allowing for continuity at the inter-elemental boundaries.

# METODO DEGLI ELEMENTI FINITI

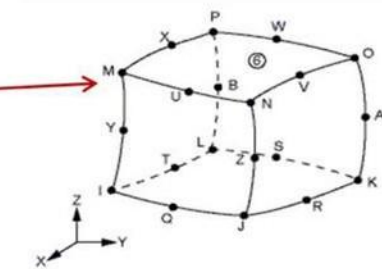
Con il metodo degli elementi finiti la risoluzione di un problema complesso viene ricondotto alla soluzione di molti problemi più semplici. Il modello virtuale è suddiviso in piccole parti di forma semplice denominate "elementi finiti"



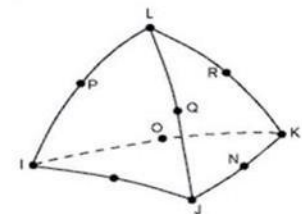
3D Solids



3D Element



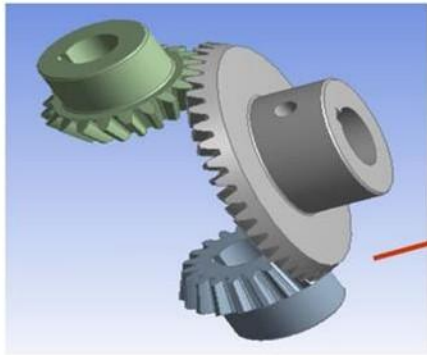
Hex Element



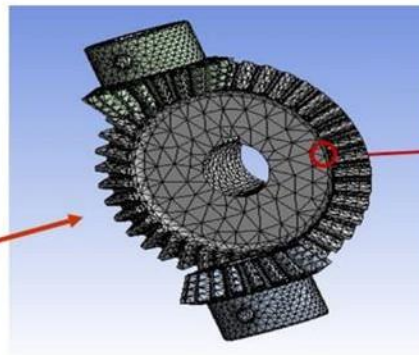
Tet Element

# METODO DEGLI ELEMENTI FINITI

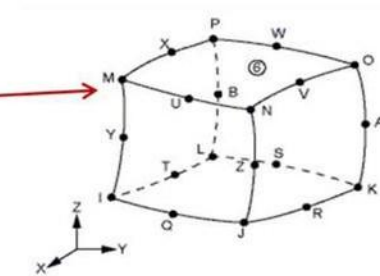
Gli elementi, detti appunto elementi finiti, sono caratterizzati dalla presenza dei nodi. Le forze che sollecitano il modello virtuale così discretizzato si trasmettono da un elemento all'altro solo attraverso questi nodi che collegano tra loro gli elementi stessi



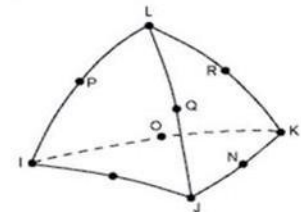
3D Solids



3D Element



Hex Element



Tet Element

# METODO DEGLI ELEMENTI FINITI

Il comportamento di questi elementi, in termini di deformazione indotta dallo spostamento dei nodi provocato in generale dalle azioni applicate all'elemento i esimo dagli elementi adiacenti è stato molto ben studiato ed è ben noto

Il sistema di forze esterne che sollecita il sistema reale è quindi applicato al modello agli elementi finiti attraverso i nodi degli elementi stessi. E' quindi calcolato lo stato di tensione/deformazione nei singoli elementi che approssima quanto accadrebbe nel sistema fisico reale

Lo spostamento di ciascun nodo è quantificato in termini di traslazione lungo direzioni  $x$ ,  $y$  e  $z$  di un sistema cartesiano ortogonale. Queste traslazioni definiscono i gradi di libertà del nodo (DOF, Degrees of Freedom). Ogni nodo, evidentemente assimilato ad un punto nello spazio, ha 3 DOF

# METODO DEGLI ELEMENTI FINITI

Nei software di analisi strutturale sono programmate le equazioni di equilibrio di ciascun nodo rispettando nel contempo anche la connettività tra tutti gli elementi finiti in cui si è discretizzato il modello solido

Nell'analisi di sollecitazione le equazioni di equilibrio dei nodi contengono le incognite, ovvero gli spostamenti dei nodi stessi

Tutti i coefficienti noti delle equazioni di equilibrio sono calcolati e/o definiti in funzione delle proprietà del materiale, dei vincoli, dei carichi e della geometria del sistema

*I programmi di analisi strutturale FEM costruiscono quindi un sistema algebrico lineare costituito dalle suddette equazioni di equilibrio e lo risolvono nelle incognite rappresentate dagli spostamenti dei nodi*

# METODO DEGLI ELEMENTI FINITI

Noti gli spostamenti dei nodi è possibile risalire con facilità alle sollecitazioni che si instaurano nei nodi e anche all' interno di un qualunque punto del singolo elemento finito.

Il numero delle citate equazioni, con le corrispondenti incognite può essere anche molto elevato (possono definirsi centinaia di migliaia o milioni di equazioni) e corrisponde, formalmente, al numero complessivo dei DOF relativo a tutti i nodi con cui si è modellizzato il sistema reale

# METODO DEGLI ELEMENTI FINITI

Nell'analisi statica più semplice il solutore risolve il sistema lineare di equazioni calcolando per ogni nodo **le componenti dello spostamento nelle tre direzioni x, y e z**

Quando questi spostamenti sono noti i programmi di analisi strutturale calcolano **le deformazioni relative e lungo varie direzioni per ciascun elemento finito**

Solo a questo punto il programma calcola **le sollecitazioni in ciascun elemento finito**

# METODO DEGLI ELEMENTI FINITI

## Analisi Lineare Elastica

E' il tipo più comune di analisi agli elementi finiti. Solitamente si suppone che il materiale segua la legge di Hooke si considerano unicamente forze esterne costanti nel tempo

Si opera quindi in regime sforzi-deformazioni lineare. Tutte le deformazioni sono considerate perfettamente elastiche per cui si suppone il sistema riprenda la forma originaria indeformata non appena i carichi sono rimossi

Nell' analisi statica i software strutturali, calcolano gli spostamenti, le deformazioni relative (opportunamente definite), le sollecitazioni e le forze di reazione

Le forze di reazione sono sempre presenti perché equilibrano il sistema di forze esterne. I software strutturali prevedono sempre un vincolamento almeno isostatico del sistema sollecitato per evitare la labilità dello stesso

# ANALISI STRUTTURALE

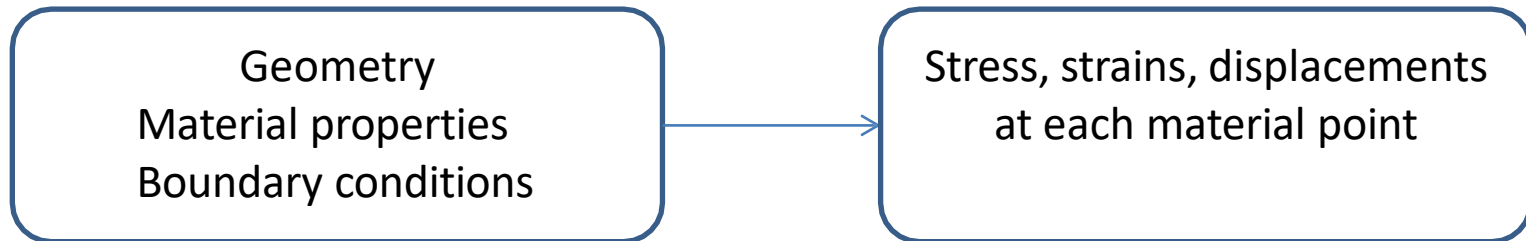
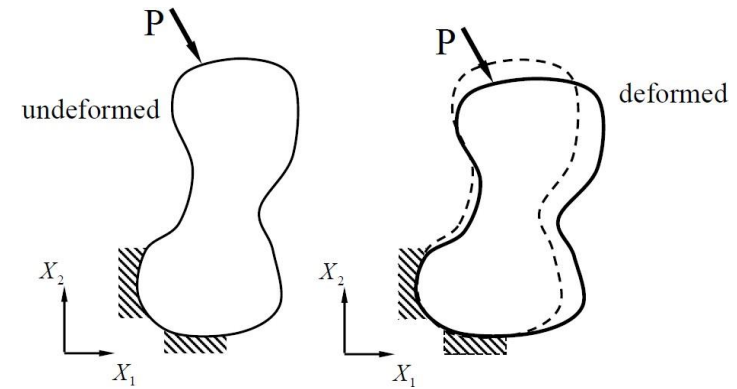
Problema fondamentale

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



# 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:

1. Material properties
2. Fabrication requirements
3. Inspection and testing requirements
4. 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

# EQUAZIONI DI GOVERNO

Il comportamento di un sistema continuo è governato da equazioni differenziali alle derivate parziali.

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

I metodi degli elementi finiti sono tecniche atte ad approssimare tali equazioni differenziali con un sistema di equazioni algebriche in un numero finito di incognite.

# Small Displacement Theory & Rotations

- Small displacement theory:

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

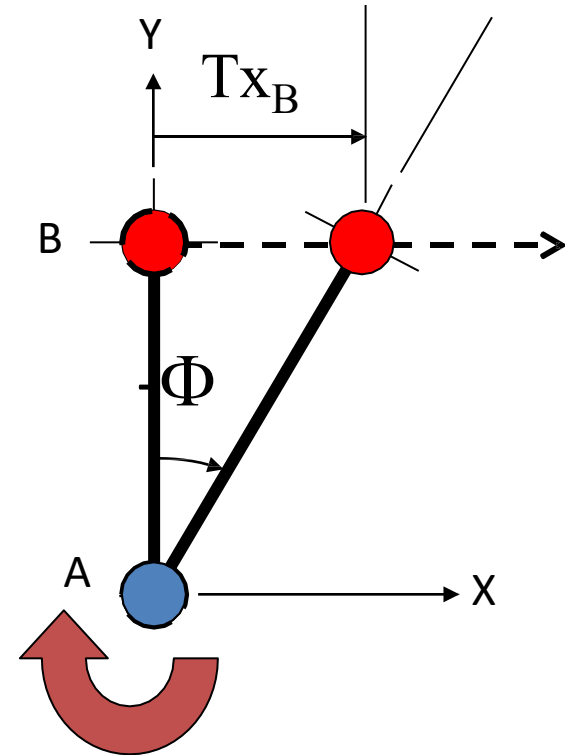
$$\cos(\Phi) = 1$$

- For  $R_z @ A$

$$R_{z_B} = R_{z_A} = \Phi$$

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

$$T_{Y_B} = 0$$



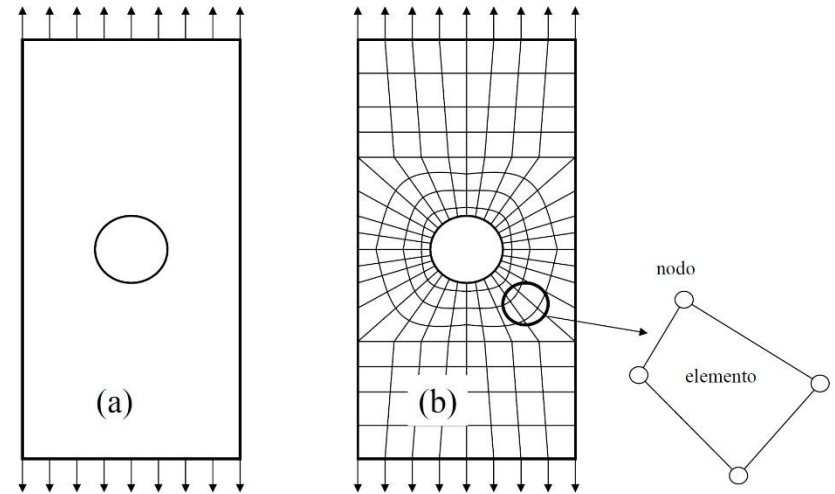
# METODO DEGLI ELEMENTI FINITI

- 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

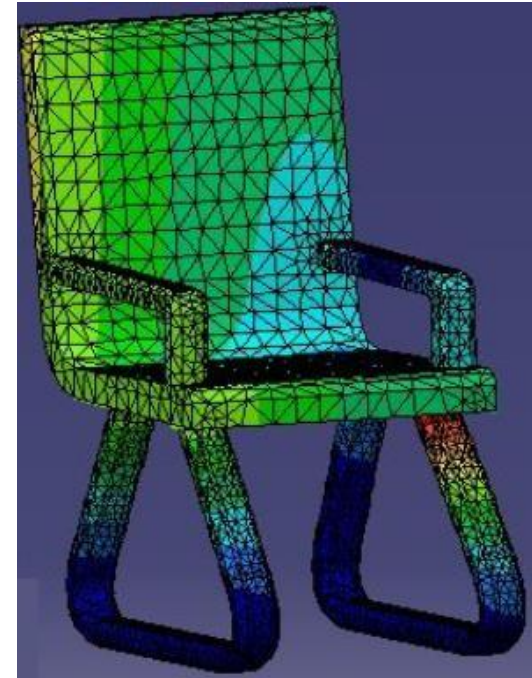
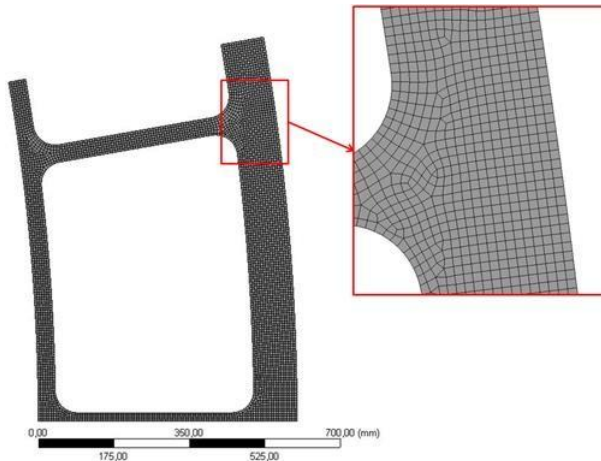
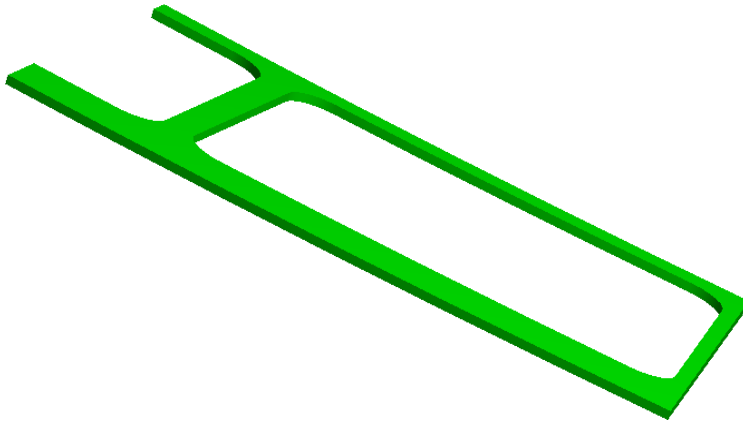
$\{u\}$ : comportamento

$\{F\}$ : azione

	Proprietà	Comportamento	Azione
Elastico	Rigidità	Deformazioni	Forze
Termico	Conduttività	Temperatura	Sorgenti di calore
Fluidodinamica	Viscosità	Velocità	Forze volumetriche

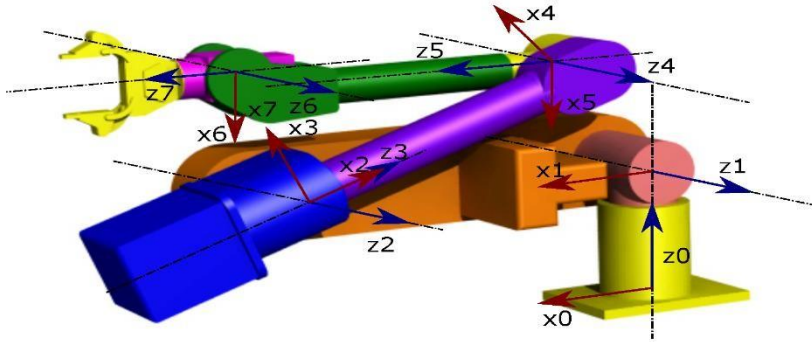
# METODO DEGLI ELEMENTI FINITI

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

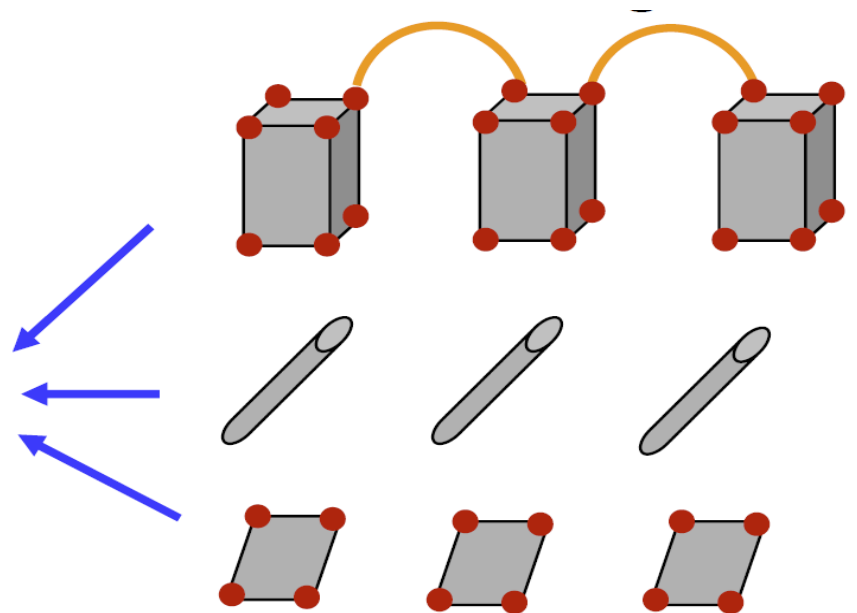
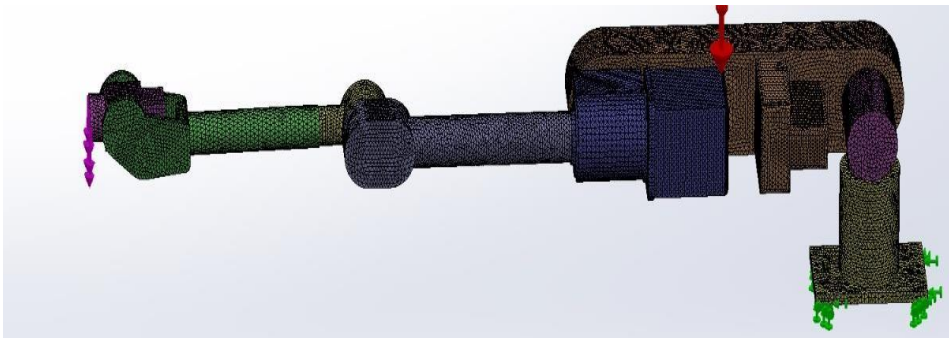


1. Per ogni nodo si calcolano Forze e Spostamenti  $\rightarrow$  reazioni vincolari
2. Per ogni elemento deformazioni( $\epsilon$ ), stress ( $\sigma$ )

# METODO DEGLI ELEMENTI FINITI

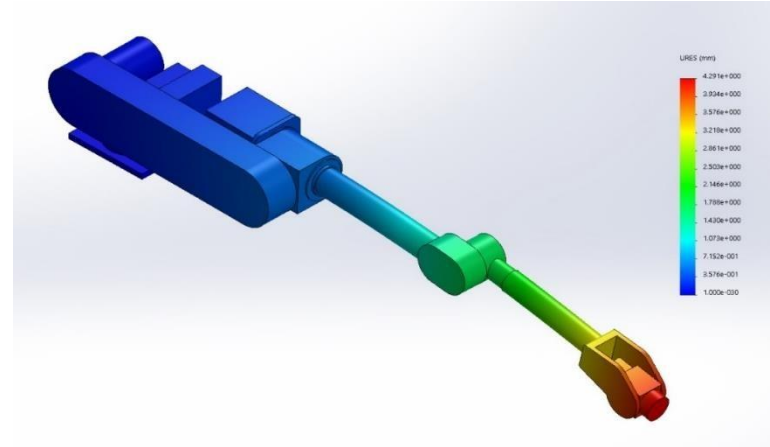
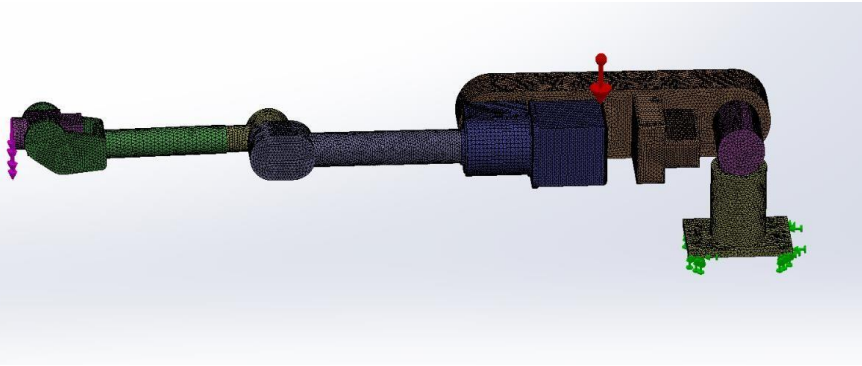


I nodi degli elementi adiacenti trasmettono vincoli e carichi tra gli elementi

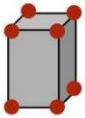


# METODO DEGLI ELEMENTI FINITI

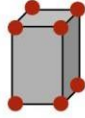
Le equazioni algebriche per ogni elemento sono ricavate in maniera semplice e combinate per ottenere le variabili incognite ai nodi



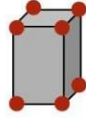
$$[K^E]\{u^E\} = \{F^E\}$$



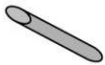
$$[K^E]\{u^E\} = \{F^E\}$$



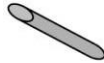
$$[K^E]\{u^E\} = \{F^E\}$$



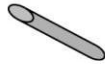
$$[K^E]\{u^E\} = \{F^E\}$$



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



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



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



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



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

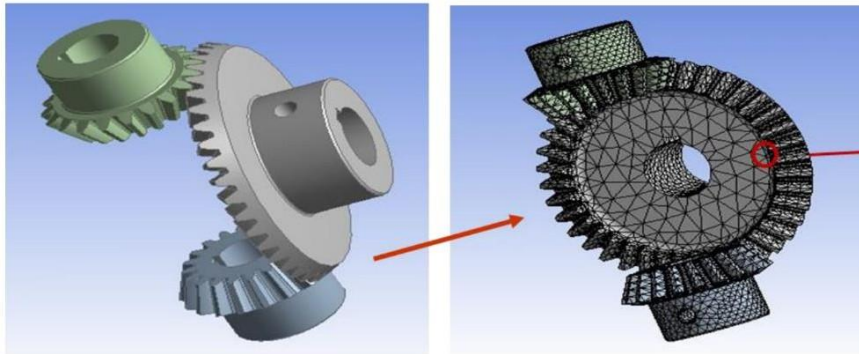


$$\{u\} = [K]^{-1}\{F\}$$

# PREPROCESSING

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

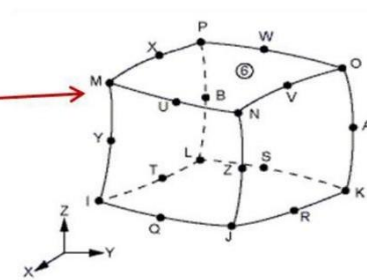
## ELEMENTI SOLIDI - BRICK



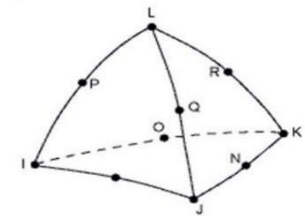
3D Solids



3D Element

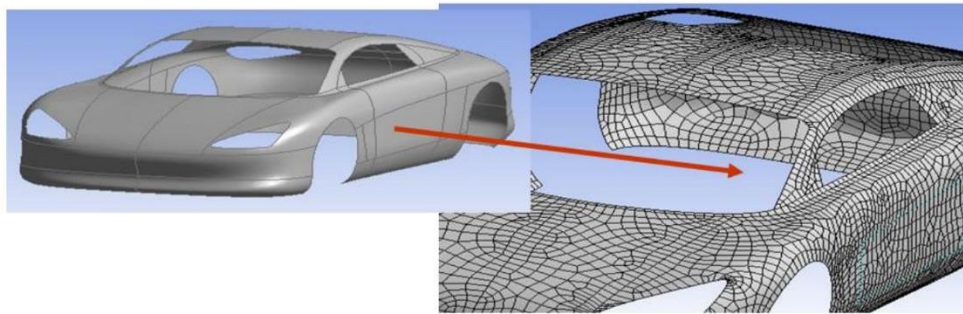
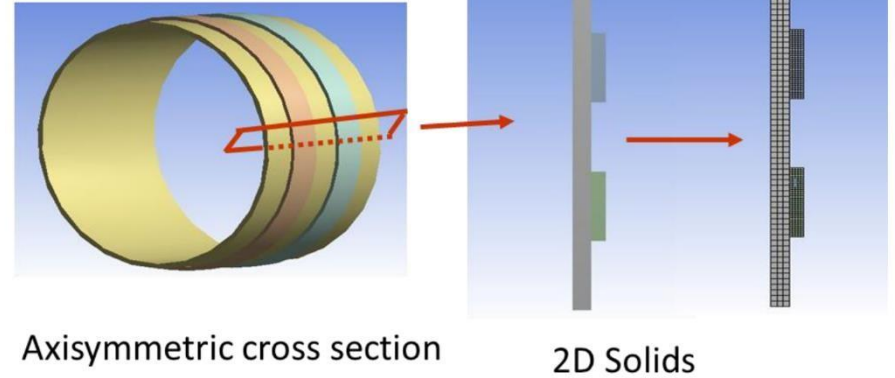
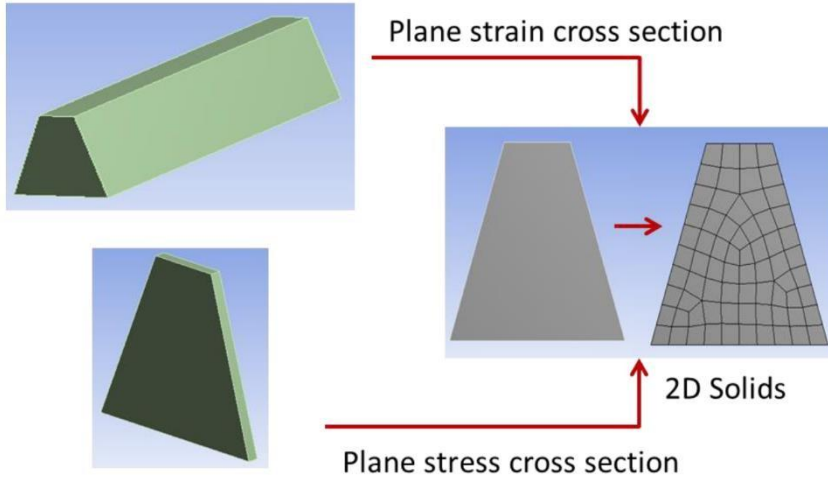


Hex Element

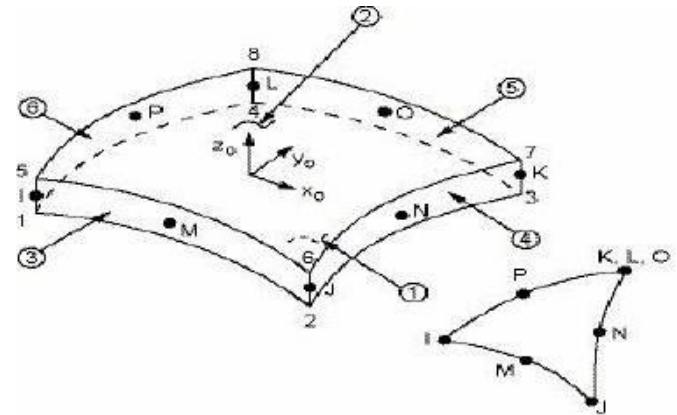


Tet Element

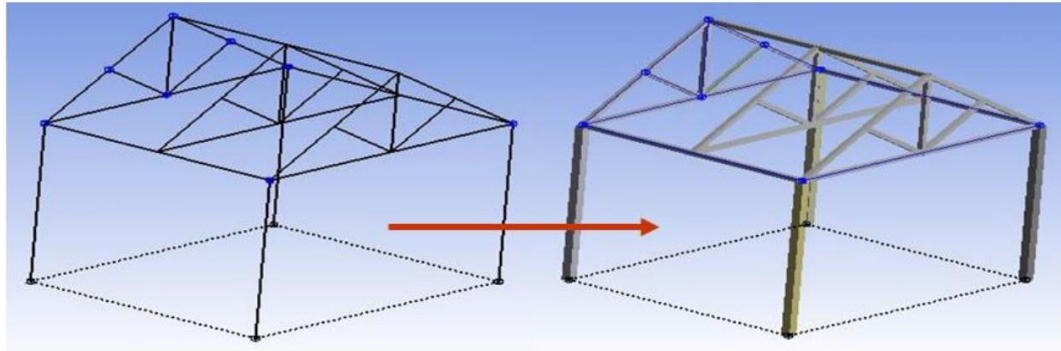
# ELEMENTI DI SUPERFICIE - SHELL



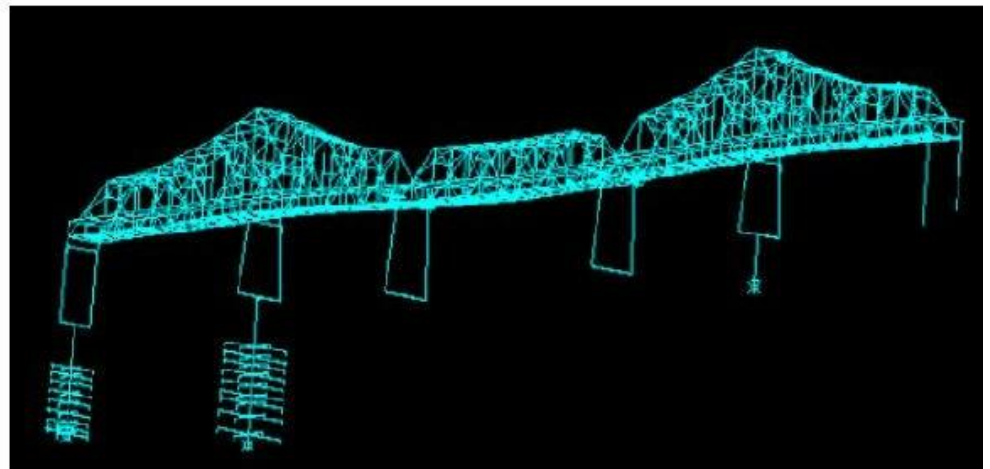
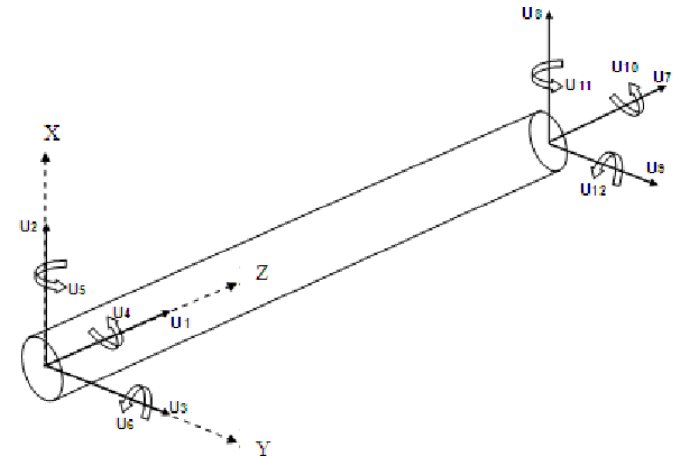
Surface Body



# ELEMENTI MONODIMENSIONALI - BEAM



Line Body

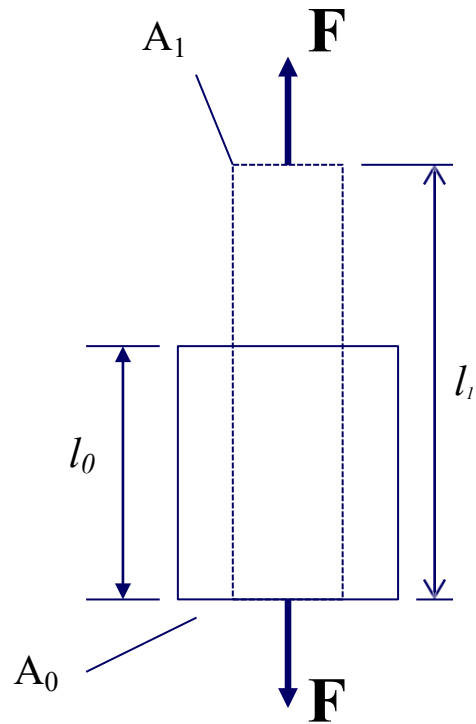


**RICHIAMI**

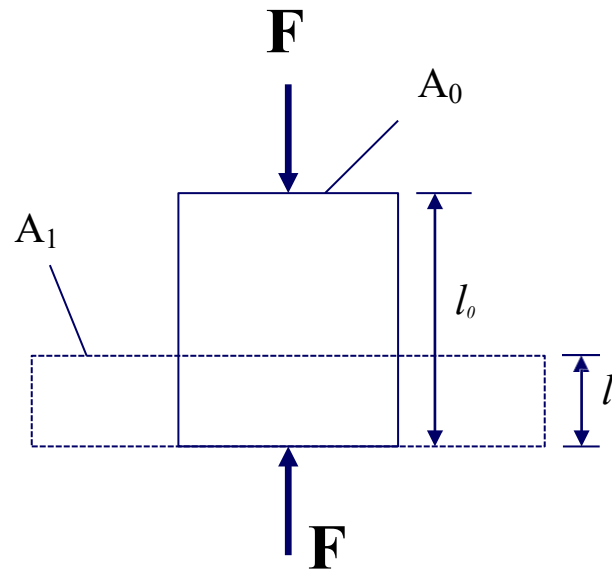
**SOLLECITAZIONI ELEMENTARI E  
CRITERI DI RESISTENZA**

# SOLLECITAZIONI ELEMENTARI

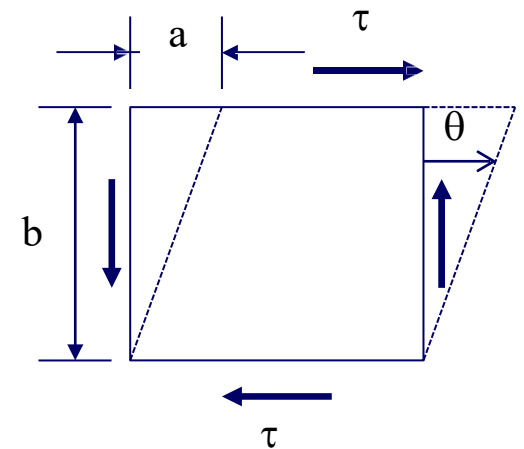
- $A_0, l_0$  = sezione e lunghezza iniziali
- $A_1, l_1$  = sezione e lunghezza finali
- $A, l$  = sezione e lunghezza istantanee



**Trazione uniforme**

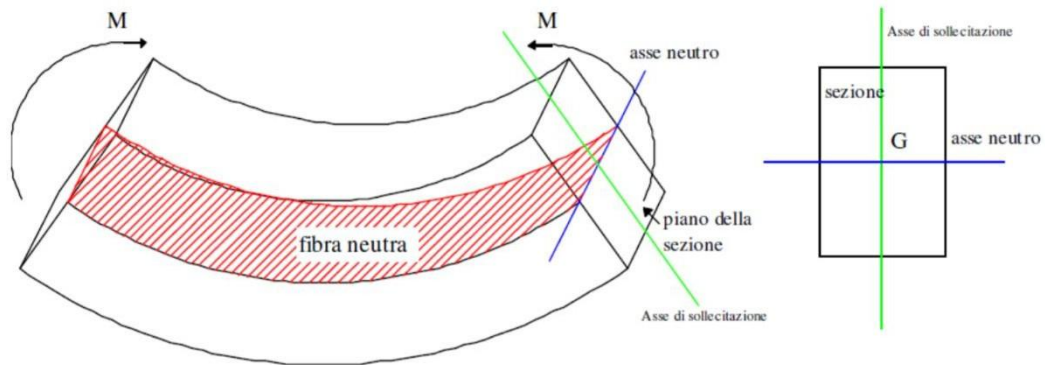


**Compressione uniforme**

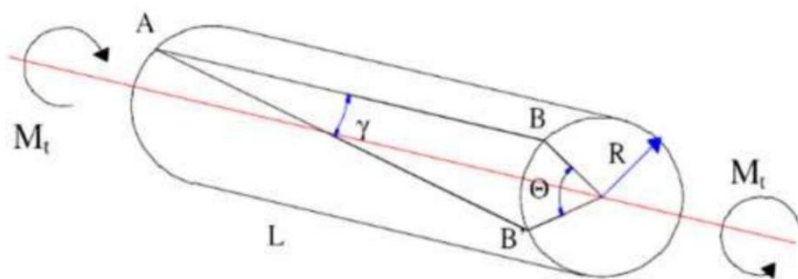


**Taglio**

# SOLLECITAZIONI ELEMENTARI



**Flessione**



**Torsione**

# SFORZO E DEFORMAZIONE NOMINALI

TENSIONE  
NOMINALE

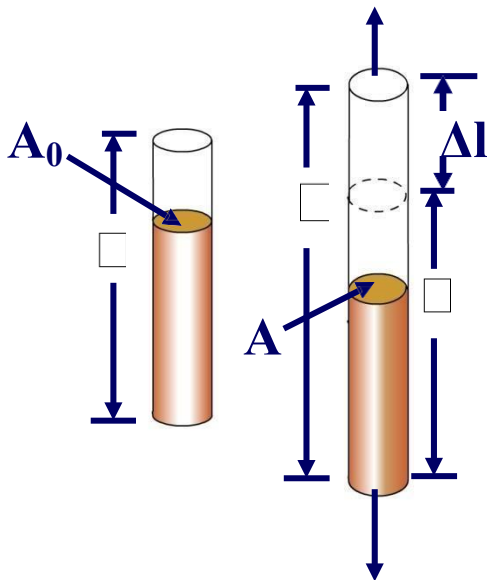
$$\sigma = \frac{F}{A_0}$$

← forza a trazione assiale media  
← superficie resistente iniziale

DEFORMAZIONE  
NOMINALE

$$\varepsilon = \frac{\Delta l}{l_0}$$

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



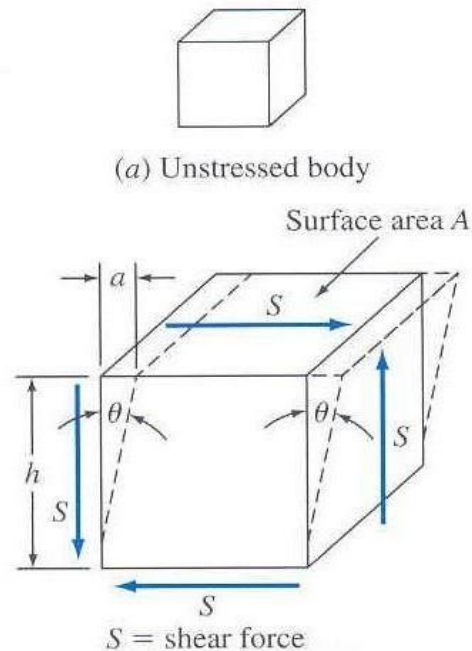
**Unità della tensione: PSI (pound per square inch) or N/m<sup>2</sup> (Pascal) 1 PSI = 6.89 x 10<sup>3</sup> Pa**

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

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

# SFORZO E DEFORMAZIONE DI TAGLIO

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

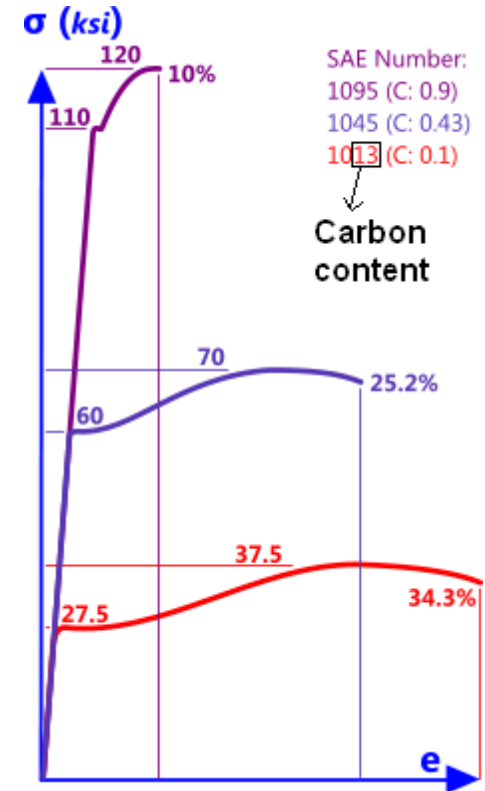
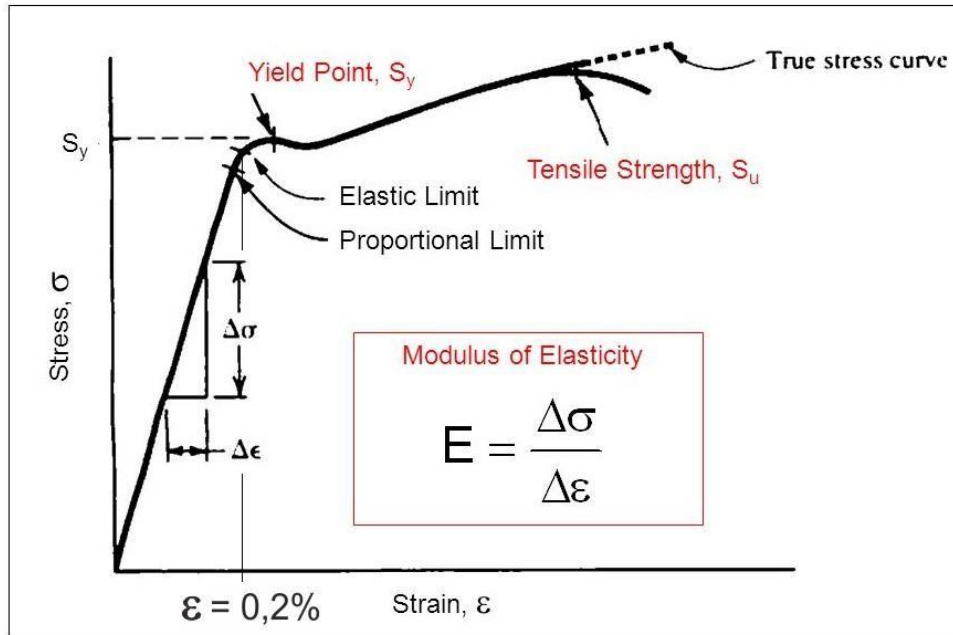


$$\gamma = \frac{\text{spostamento a}}{\text{distanza "h" sulla quale agisce lo sforzo}}$$

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

# CURVA TENSIONI NOMINALI – DEFORMAZIONI NOMINALI

## Stress-Strain Curve for Steel

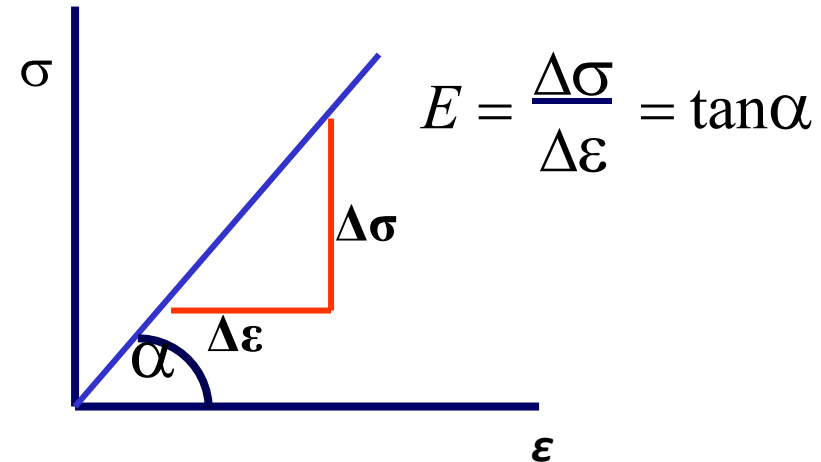


# RELAZIONE SFORZO - DEFORMAZIONE

Lo sforzo e la deformazione hanno una relazione lineare nel campo elastico (Legge di Hooke)

**Modulo di Elasticità (E) :**

$$E = \frac{\sigma \text{ (sforzo)}}{\varepsilon \text{ (deformazione)}}$$

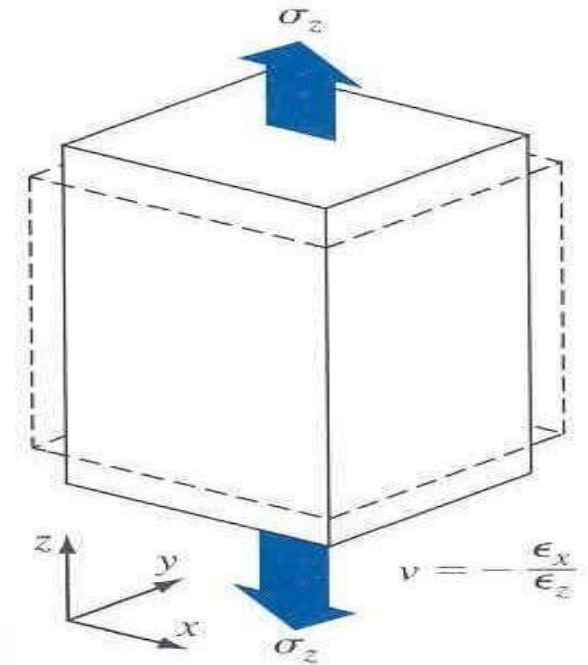
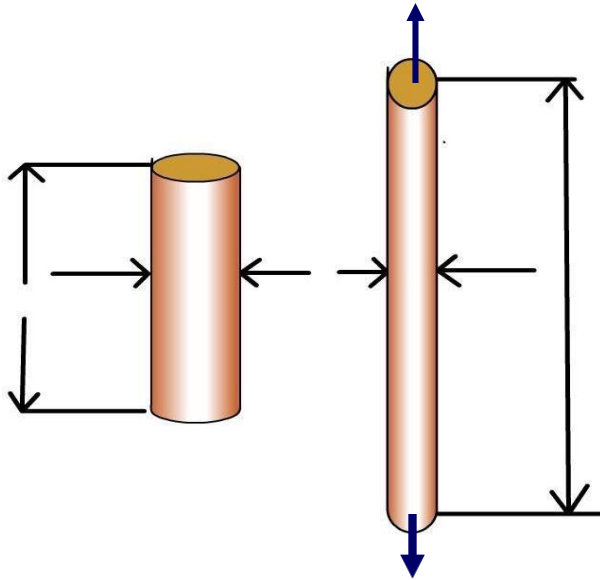
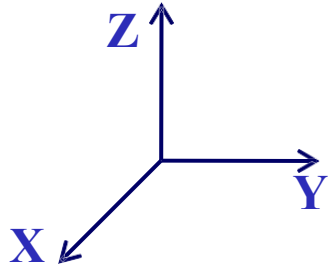


Esempi: Modulo di elasticità dell'acciaio: 207 GPa

Modulo di elasticità dell'alluminio: 76 GPa

# RAPPORTO DI POISSON

$$\nu = -\frac{\varepsilon(\text{laterale})}{\varepsilon(\text{longitudinale})} = -\frac{\varepsilon_y}{\varepsilon_z} = -\frac{\varepsilon_x}{\varepsilon_z}$$



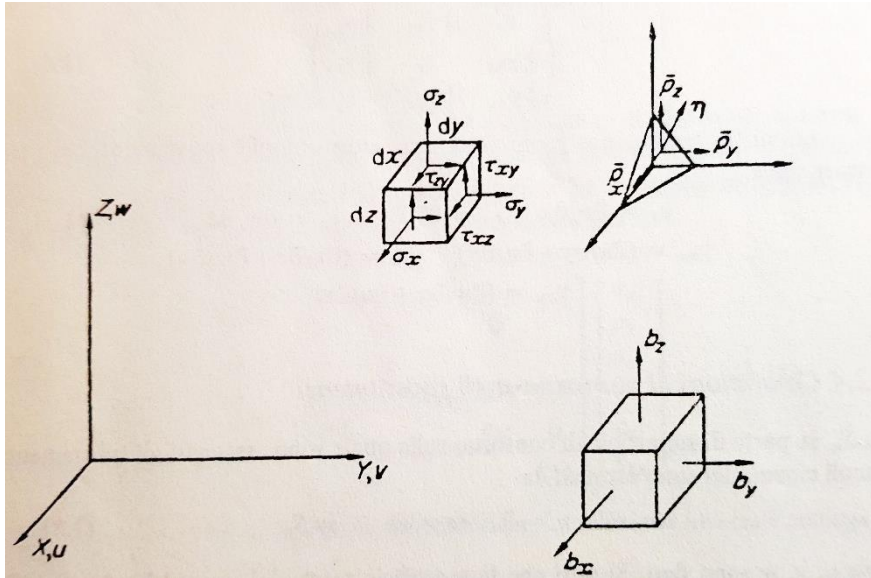
Esempi: Acciaio inossidabile  $\rightarrow$  0.28

Rame  $\rightarrow$  0.33

Per materiali isotropi

$$-\varepsilon_x = -\varepsilon_y$$

# STATO DI SFORZO



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

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

dove

$$r_{xy} = r_{yx}$$

$$r_{xz} = r_{zx}$$

$$r_{yz} = r_{zy}$$

Le componenti di tensore di sforzo devono soddisfare in tutto il corpo le seguenti equazioni di equilibrio

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

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

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

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

# LEGAME TRA SFORZI E DEFORMAZIONI

I legami sforzo-deformazione per un materiale elastico-lineare si possono scrivere nella forma (Equazioni di Navier-Cauchy)

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & d_{13} & d_{14} & d_{15} & d_{16} \\ & d_{22} & d_{23} & d_{24} & d_{25} & d_{26} \\ & & d_{33} & d_{34} & d_{35} & d_{36} \\ & & & d_{44} & d_{45} & d_{46} \\ & \text{simm.} & & & d_{55} & d_{56} \\ & & & & & d_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$$

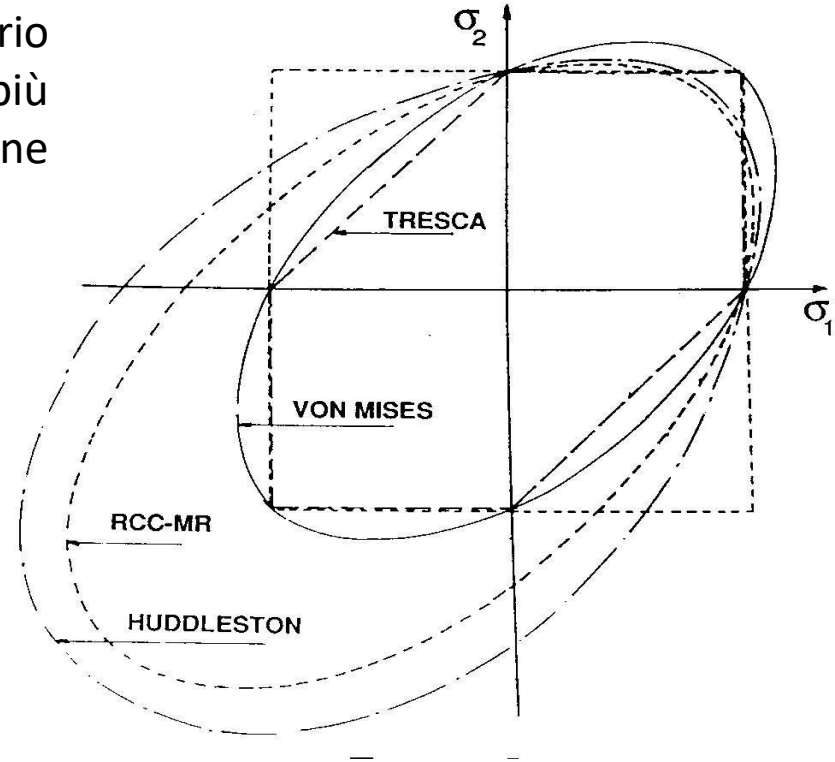
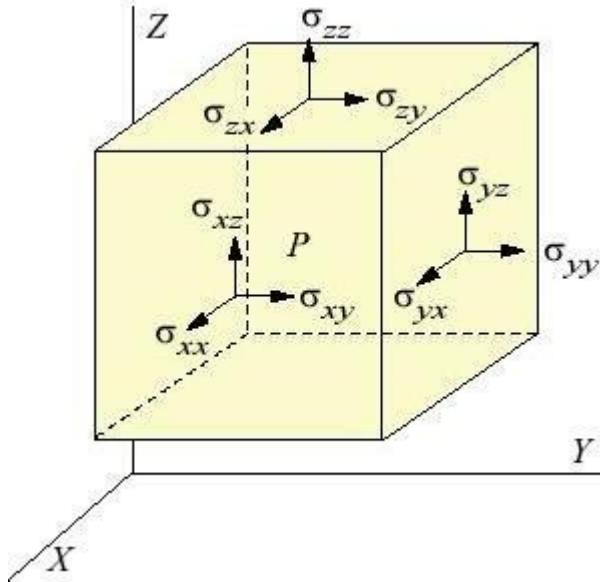
Che per un generico materiale presenta 21 costanti indipendenti.

Per un materiale isotropo le costanti indipendenti sono soltanto due, e la matrice di rigidezza si particolarizza nella:

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

# STRESS INTENSITY

Per effettuare la verifica di una struttura è necessario confrontare lo stato tensionale nel punto più sollecitato di una struttura con un valore di tensione ammissibile



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

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

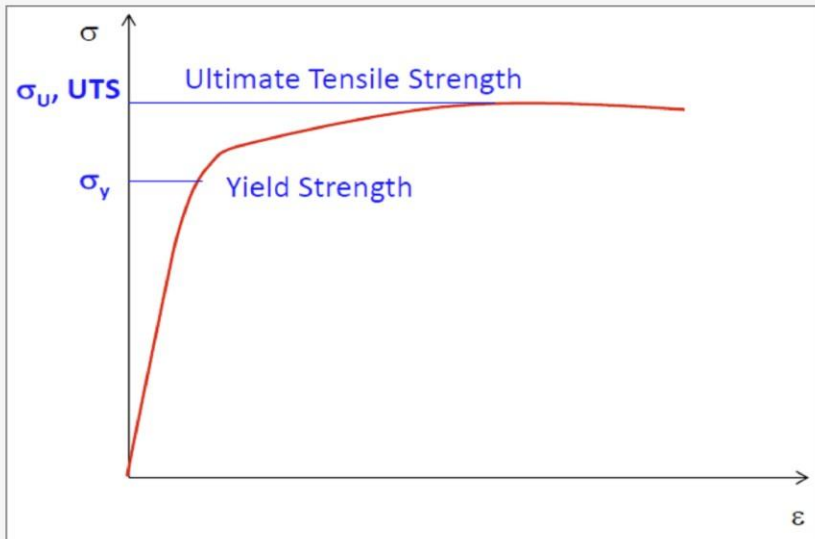
$$\text{RCC-MR, addenda Nov.87 : } \bar{\sigma}_{RCC} = 0.867 \bar{\sigma}_{VM} + 0.133 (\sigma_1 + \sigma_2 + \sigma_3)$$

(Austenitic steels)

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

# TENSIONE AMMISSIBILE

$$\sigma_a = \frac{\sigma_y}{1.5}$$



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	$f_d = \min\left(\frac{UTS}{1,875} \frac{\sigma_y}{1,5}\right)$