

Introduzione al Metodo agli Elementi Finiti

Finite Element Method, **FEM**

oppure

Finite Element Analysis, **FEA** o **FE**

**Applicazione all'analisi statica strutturale elastica lineare**

Ing. **Ciro Santus**

<http://people.unipi.it/static/ciro.santus/>

# Metodo agli Elementi Finiti

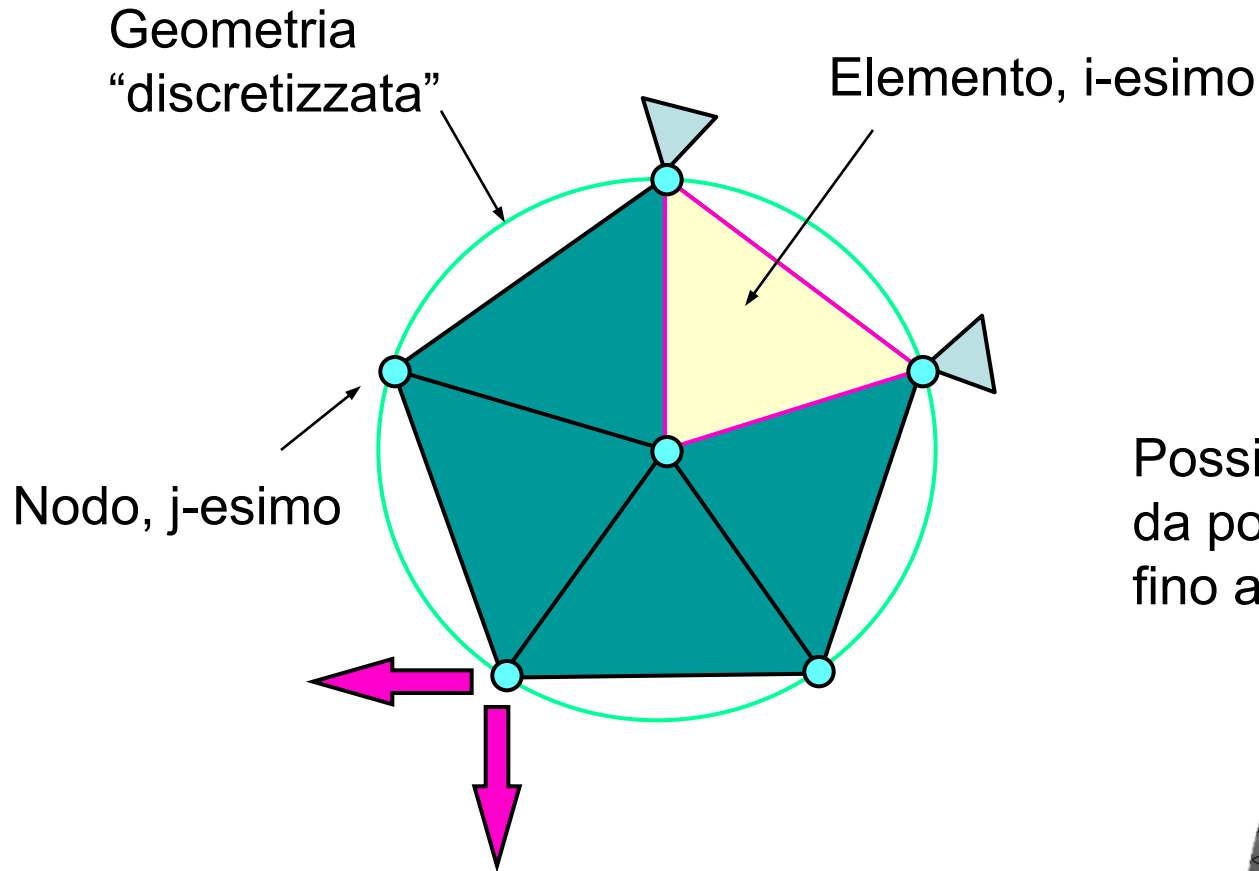
Risoluzione numerica di problemi fisici  
(equazioni alle derivate parziali)

Esistono altri metodi alternativi, es.: BEM, metodo delle differenze

Ormai si può considerare il metodo *standard* per risolvere problemi strutturali, termici, fluidodinamici, ecc.

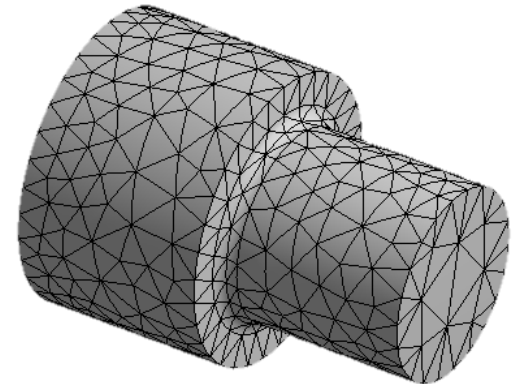
Evoluzione del metodo a partire dagli anni '50. Attualmente esistono importanti SW commerciali, Es.: Ansys, Abaqus

# Nodi & Elementi

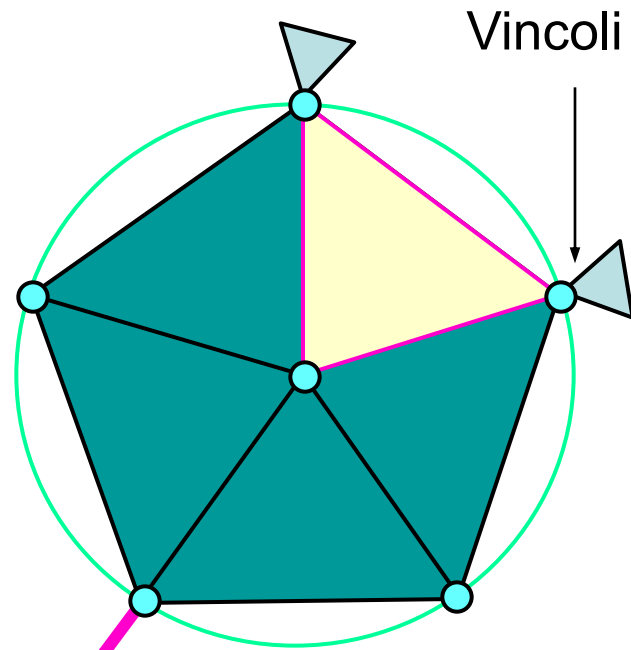


Gradi di libertà del singolo nodo:  
Spostamenti nelle direzioni x,y,z

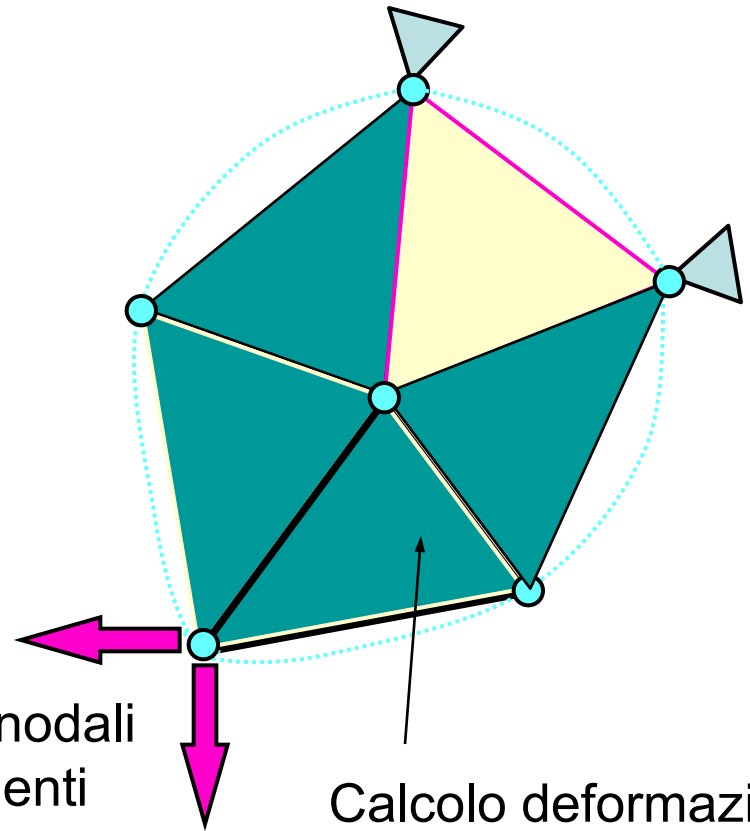
Possibilità di gestire modelli  
da poche migliaia di elementi,  
fino a 1 000 000 elementi



# Vincoli e carichi



Forze esterne applicate, su alcuni nodi



Calcolo nodali spostamenti deformata (prima incognita)

Calcolo deformazioni e tensioni, in ogni punto a partire dagli spost. nodali (Funzioni di forma)

# Soluzione del modello agli Elementi Finiti

Il sistema di equazioni differenziali alle derivate parziali, si “riduce” ad un sistema lineare, in cui le incognite sono gli spostamenti nodali.

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



Numero molto elevato di incognite, comunque finito, condizione ideale per l'implementazione al calcolatore

La soluzione del modello consiste nella risoluzione di questo sistema nell'incognita  $\{U\}$  degli spostamenti nodali.

# Scelta del tipo elemento

Geometria

2D

3D

Linee

Elemento trave 2D  
(beam)

El. trave 3D  
(beam)

Aree

El. piano  
(plane strain/stress)

El. Guscio  
(shell)

Volume

-----

Elemento  
solido (brick)

# Scelta del tipo elemento

## Elementi Trave (Beam)

Il nodo rappresenta una sezione

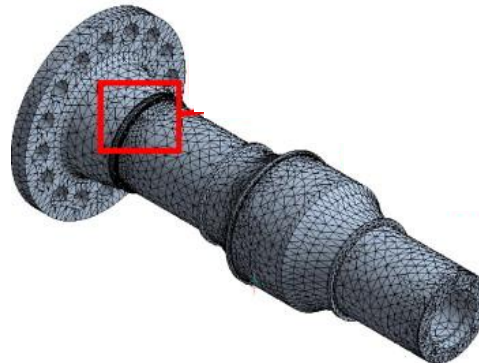
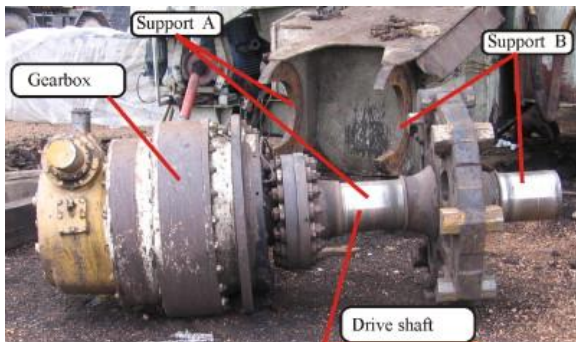


## Elementi Guscio (Shell)

Il nodo rappresenta uno spessore

## Elementi Solidi (Brick)

Il nodo rappresenta un punto solido



Introduzione al codice  
agli Elementi Finiti

**Ansys**

**Ansys**

```
graph TD; A[Ansys] --> B[Ansys APDL o "Classic"]; A --> C[Ansys Workbench];
```

**Ansys**

APDL o "Classic"

**Ansys**

Workbench

**BEGIN Level**



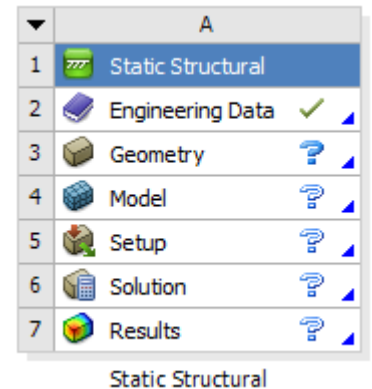
**PREP7**

**SOLUTION**

**POST1**

APDL

Workbench



# ANSYS APDL

**PREP7**

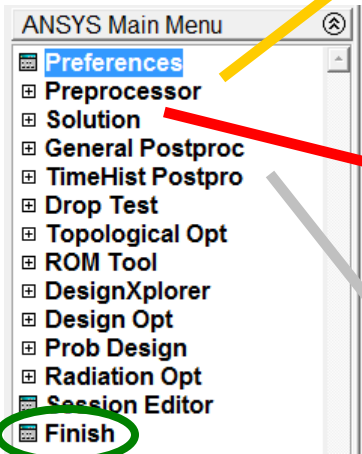
- definizione ELEMENT TYPE
- definizione REAL CONSTANTS
- definizione MATERIAL PROPERTIES
- definizione GEOMETRIA MODELLO
- definizione MESH del modello
- applicazione di VINCOLI e CARICHI

**SOLUTION**

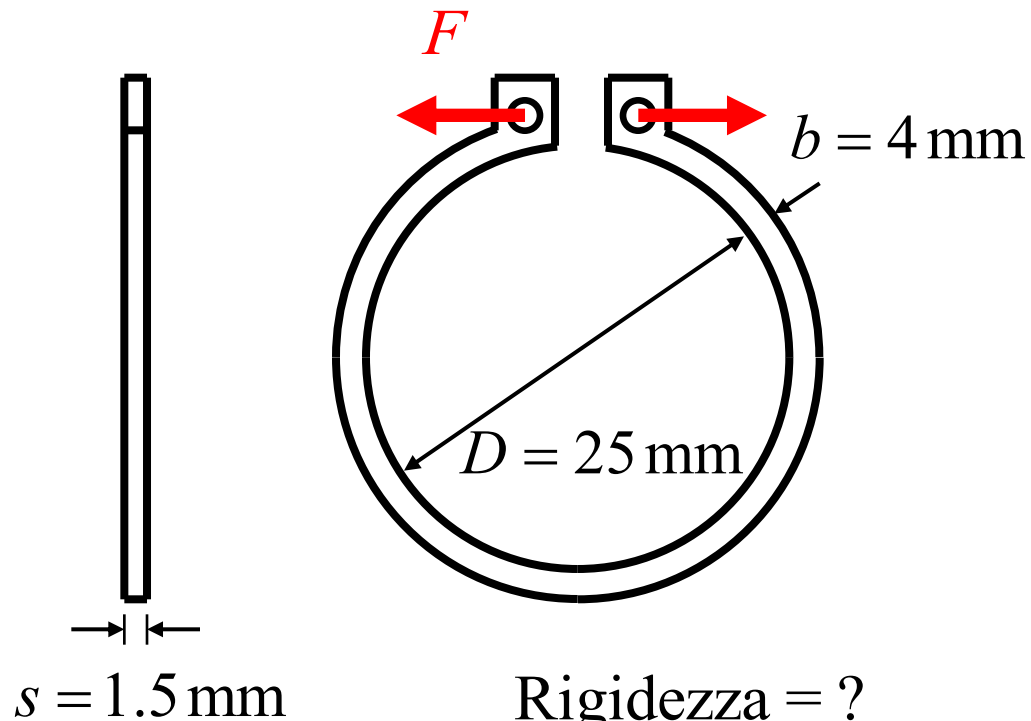
*Soluzione FEM*

**POST1**

- PLOT visualizzazione grafica dei risultati
- LIST risultati in forma numerica



# Es.: Modellazione solida: Anello elastico (plane stress)



Rigidezza = ?

Stato di tensione = ?

# Definizione elementi

**Definizione elementi**

Defined Element Types:  
Type 1 PLANE182

Library of Element Types

Library of Element Types

Structural Mass  
Link  
Beam  
Pipe  
Solid  
Shell  
Solid-Shell

Quad 4 node 182  
8 node 183  
Brick 8 node 185  
20node 186  
concret 65

Quad 4 node 182

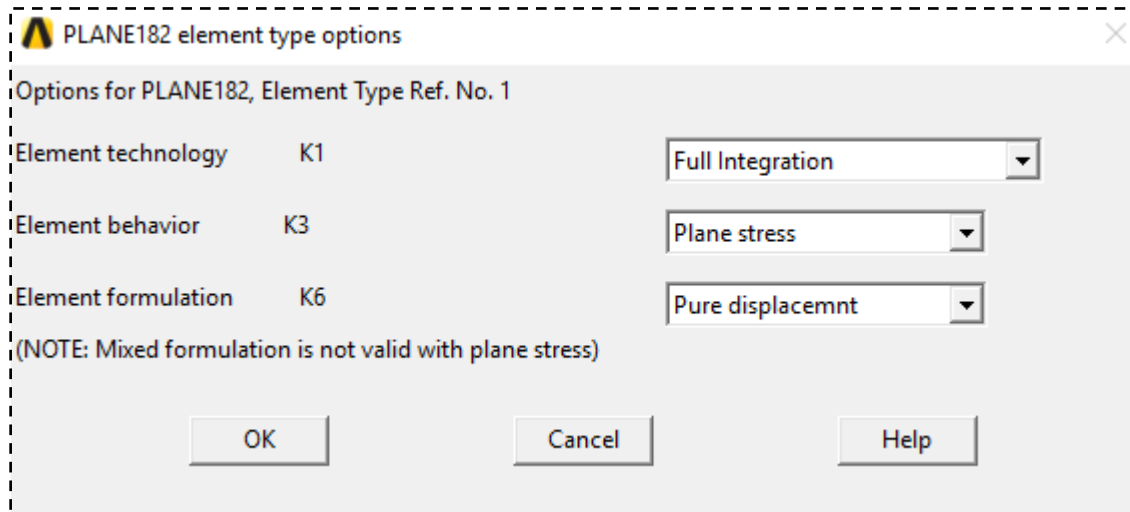
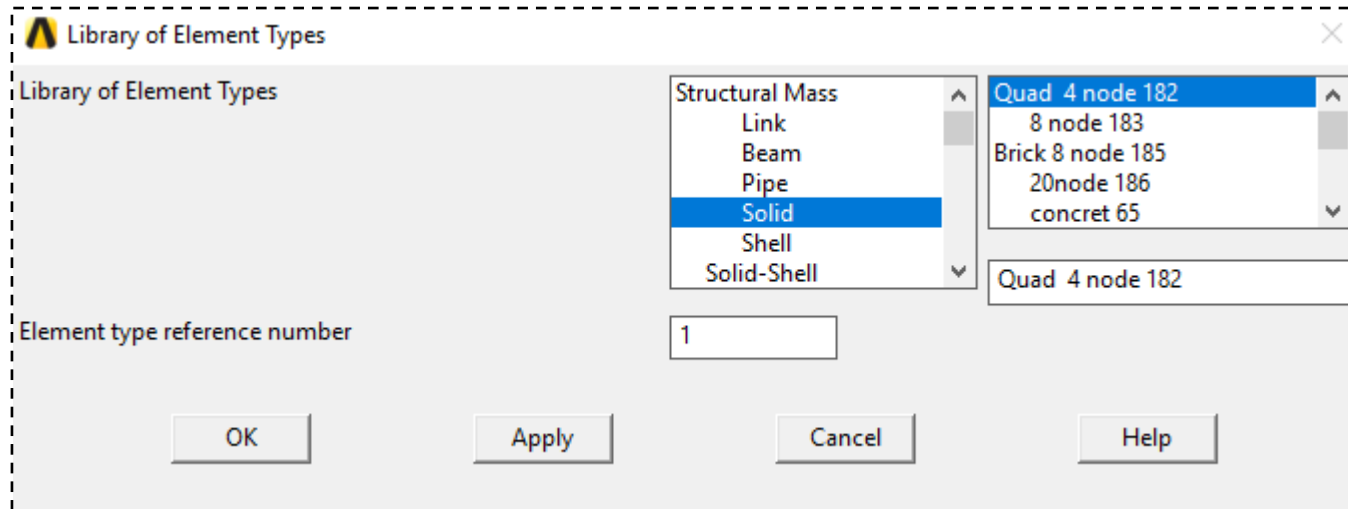
Element type reference number: 1

OK Apply Cancel Help

Add Close Help

**Elemento solido piano  
es. Plane 182**

# Definizione elementi



## Definizione keyoptions

es.:

- plane stress
- plane strain
- axisymmetric

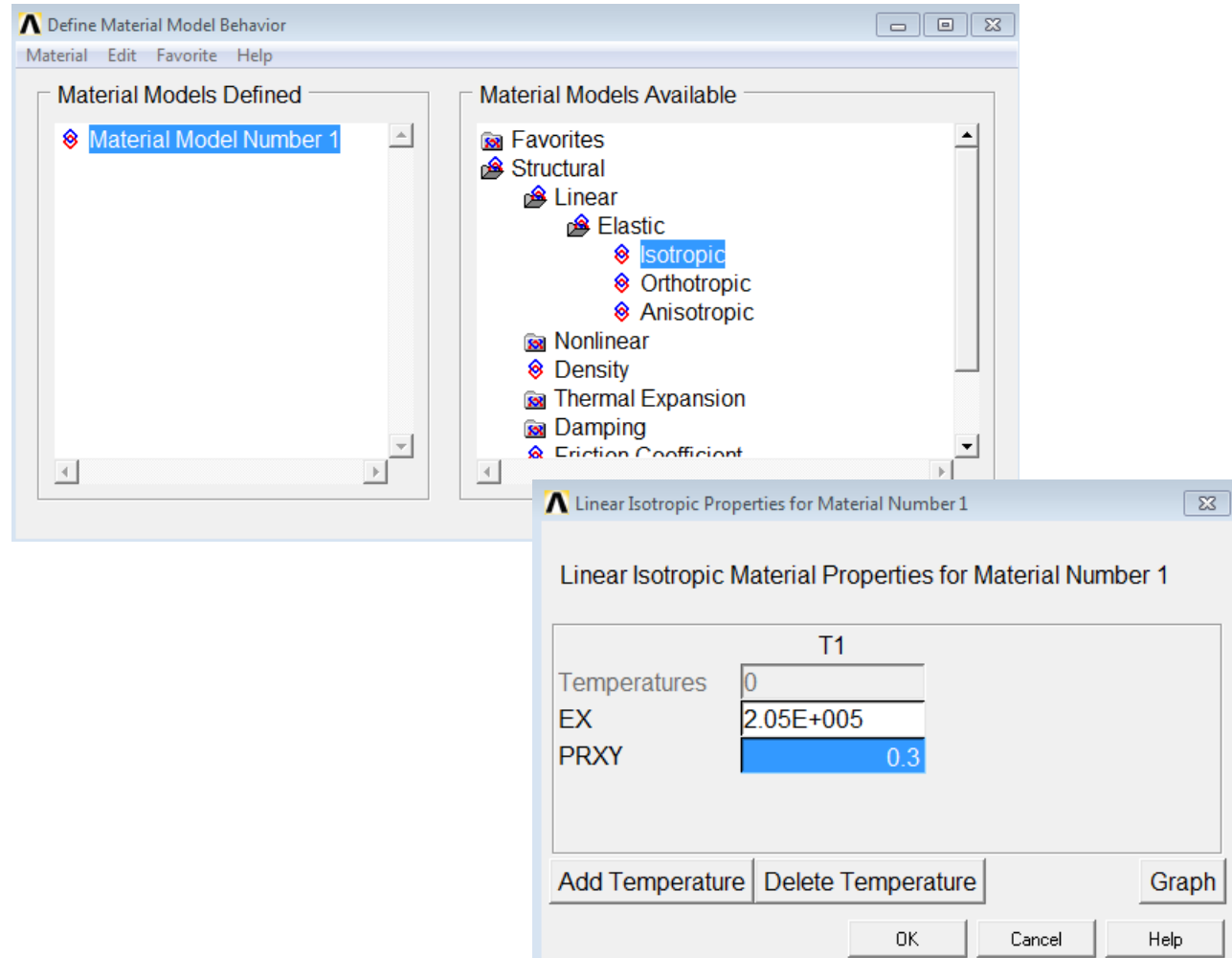
# Definizione proprietà dei materiali

Materiale:

Elastico Lineare Isotropo Omogeneo

Moduli di Young e Poisson

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
    - Material Library
    - Temperature Units
    - Electromag Units
    - Material Models
    - Convert ALPx
    - Change Mat Num
    - Failure Criteria
    - Write to File
    - Read from File
  - Sections
  - Modeling
  - Meshing
  - Checking Ctrl
  - Numbering Ctrl
  - Archive Model
  - Coupling / Ceqn
  - Multi-field Set Up
  - Loads
  - Physics
  - Path Operations
- Solution
  - General Postproc
  - TimeHist Postpro
  - Drop Test
  - Topological Opt
  - ROM Tool
  - DesignXplorer
  - Design Opt
  - Prob Design
  - Radiation Opt
- Session Editor
- Finish



# Modellazione solida, anello elastico, plane stress

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
  - Modeling
    - Create
      - Keypoints
      - Lines
      - Areas
        - Arbitrary
        - Rectangle
        - Circle
          - Solid Circle
          - Annulus
          - Partial Annulus
          - By End Points
          - By Dimensions
        - Polygon
        - Area Fillet
      - Volumes
      - Nodes
      - Elements
      - Contact Pair
      - Circuit
      - Transducers
    - Operate
      - Move / Modify
      - Copy
      - Reflect
      - Check Geom
      - Delete
      - Cyclic Sector
      - CMS
      - Genl plane strn
      - Update Geom

Part Annular Circ Area

Pick  Unpick

---

WP X =  
Y =

Global X =  
Y =  
Z =

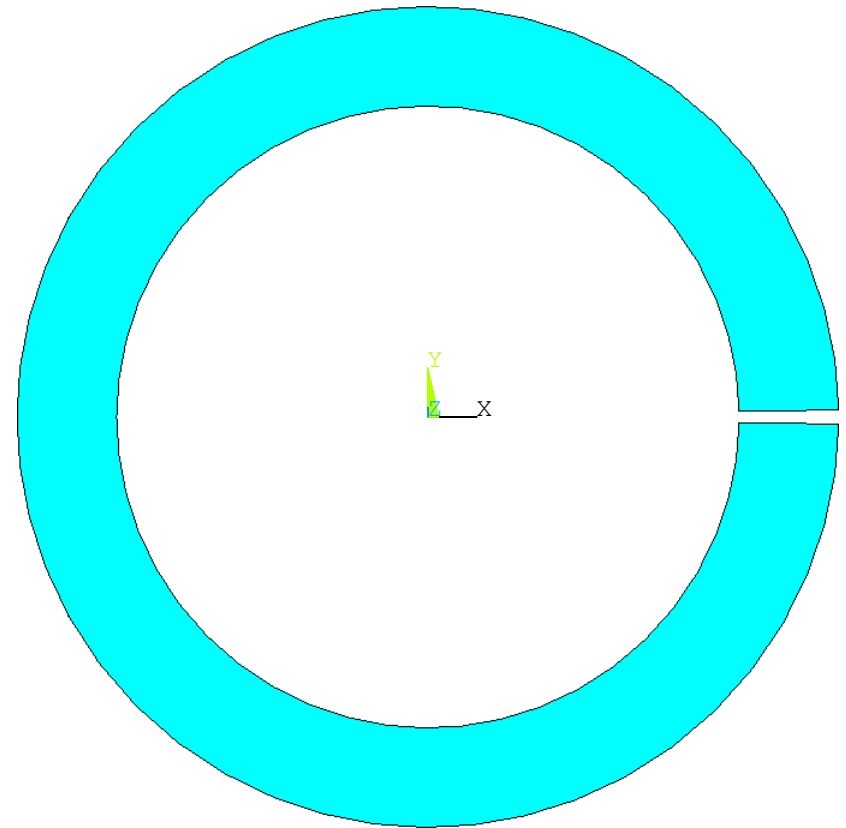
---

WP X	0
WP Y	0
Rad-1	25/2
Theta-1	1
Rad-2	25/2+4
Theta-2	359

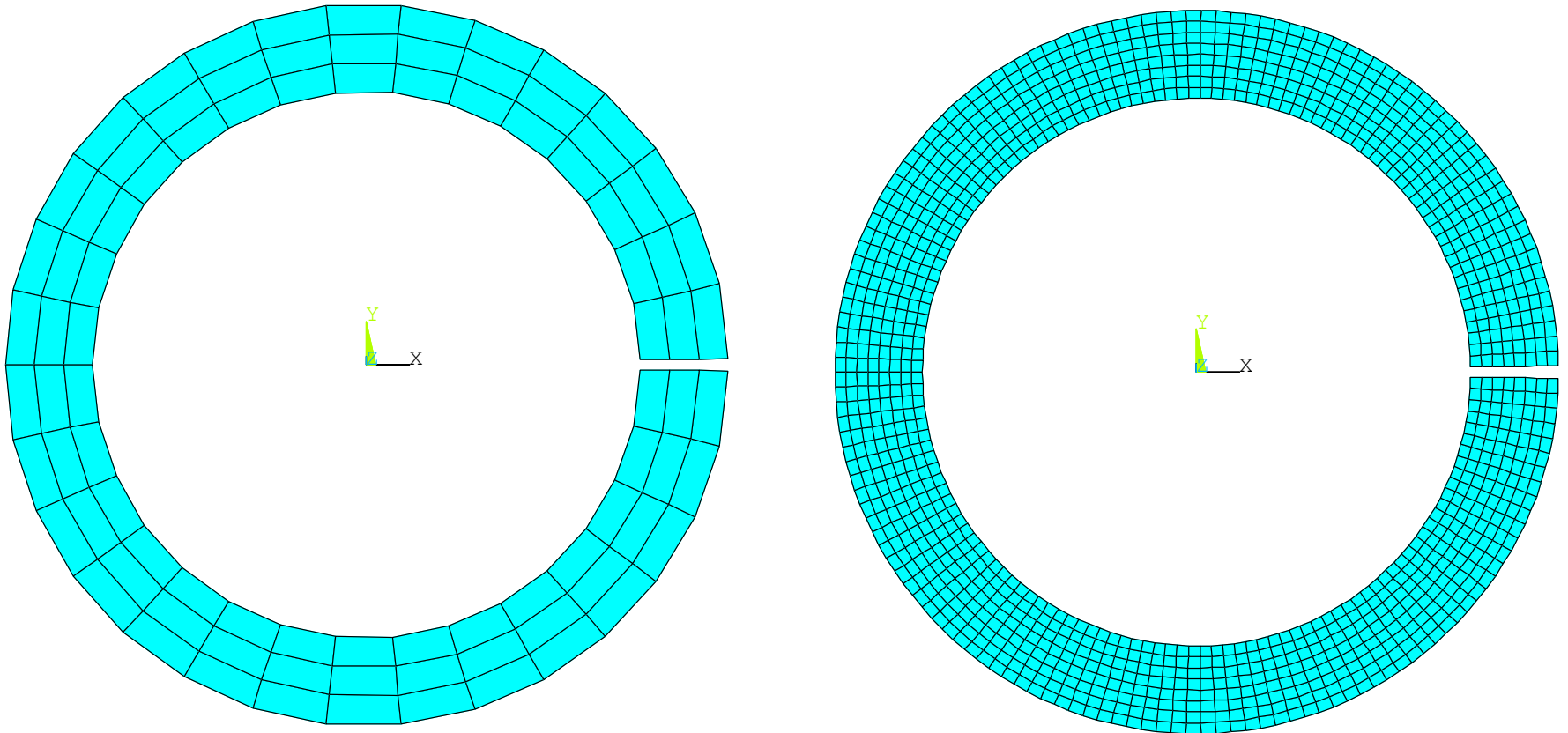
OK Apply

Reset Cancel

Help



# “Mesh”: nodi ed elementi



**“Infittimento della Mesh”**

# Condizioni di vincolo

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
  - Modeling
  - Meshing
  - Checking Ctrl
  - Numbering Ctrl
  - Archive Model
  - Coupling / Ceqn
  - Multi-field Set Up
  - Loads
    - Analysis Type
    - Define Loads
      - Settings
      - Apply
        - Structural
          - Displacement
            - On Lines
            - On Areas
            - On Keypoints
            - On Nodes
            - On Node Components
              - Symmetry B.C.
              - Antisymm B.C.
            - Force/Moment
            - Pressure
            - Temperature
            - Inertia
            - Pretnsn Sectn
            - Gen Plane Strain
            - Other
          - Field Volume Intr
          - Initial Condit'n
          - Load Vector
          - Functions

Apply U,ROT on Lines

Pick     Unpick

---

Single     Box

Polygon     Circle

Loop

---

Count = 1

Maximum = 4

Minimum = 1

Line No. = 2

---

List of Items

Min, Max, Inc

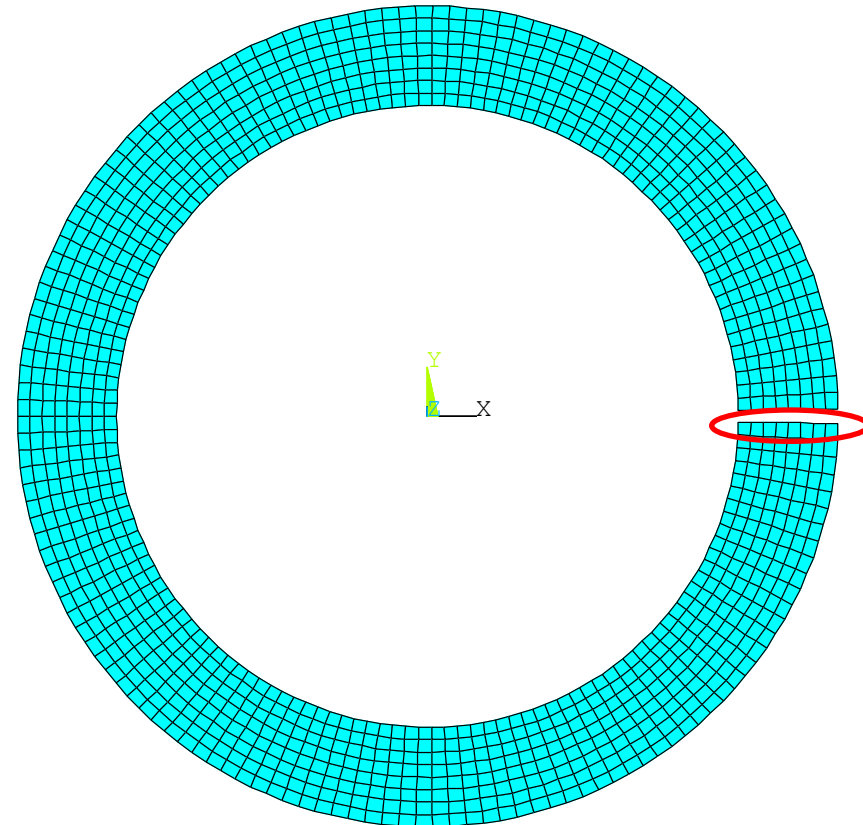
\_\_\_\_\_

---

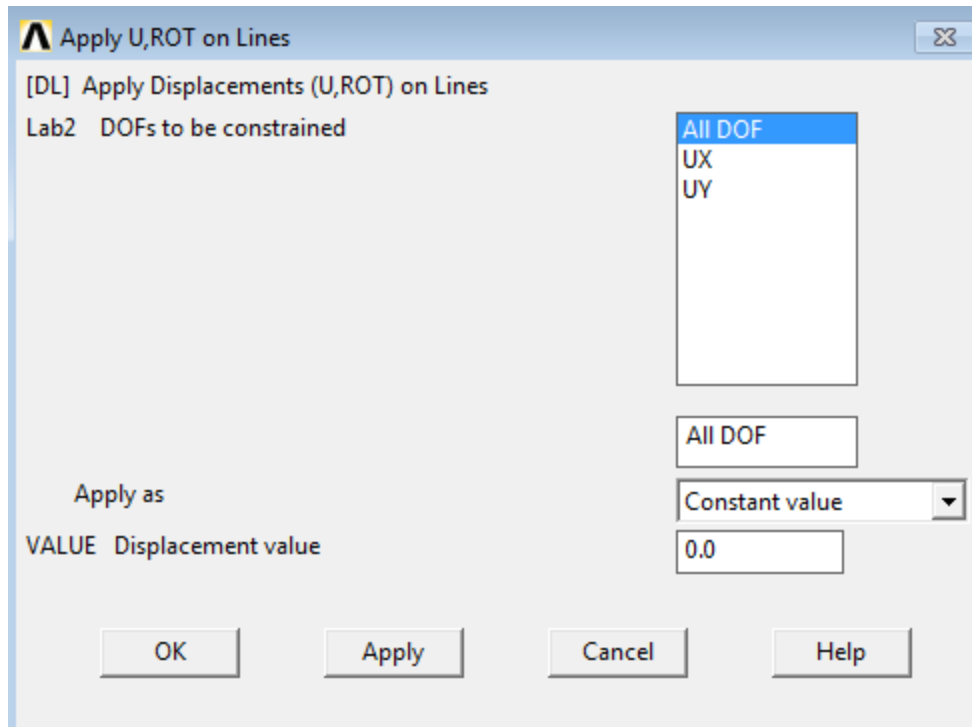
OK    Apply

Reset    Cancel

Pick All    Help



# Condizioni di vincolo



Spostamento imposto su tutti i gradi di libertà = incastro.

# Condizioni di carico: pressione sulla linea

## Preferences

### Preprocessor

- Element Type
- Real Constants
- Material Props
- Sections
- Modeling
- Meshing
- Checking Ctrl
- Numbering Ctrl
- Archive Model
- Coupling / Ceqn
- Multi-field Set Up
- Loads

### Analysis Type

#### Define Loads

##### Settings

##### Apply

##### Structural

##### Displacement

##### Force/Moment

##### Pressure

##### On Lines

##### On Areas

##### On Nodes

##### On Node Components

##### On Elements

##### On Element Components

##### From Fluid Analy

##### On Beams

##### Temperature

##### Inertia

##### Pretnsn Sectn

##### Gen Plane Strain

##### Other

##### Field Volume Intr

##### Initial Condit'n

##### Load Vector

##### Functions

### Apply PRES on Lines

Pick  Unpick

Single  Box

Polygon  Circle

Loop

Count = 0

Maximum = 4

Minimum = 1

Line No. =

List of Items

Min, Max, Inc

OK

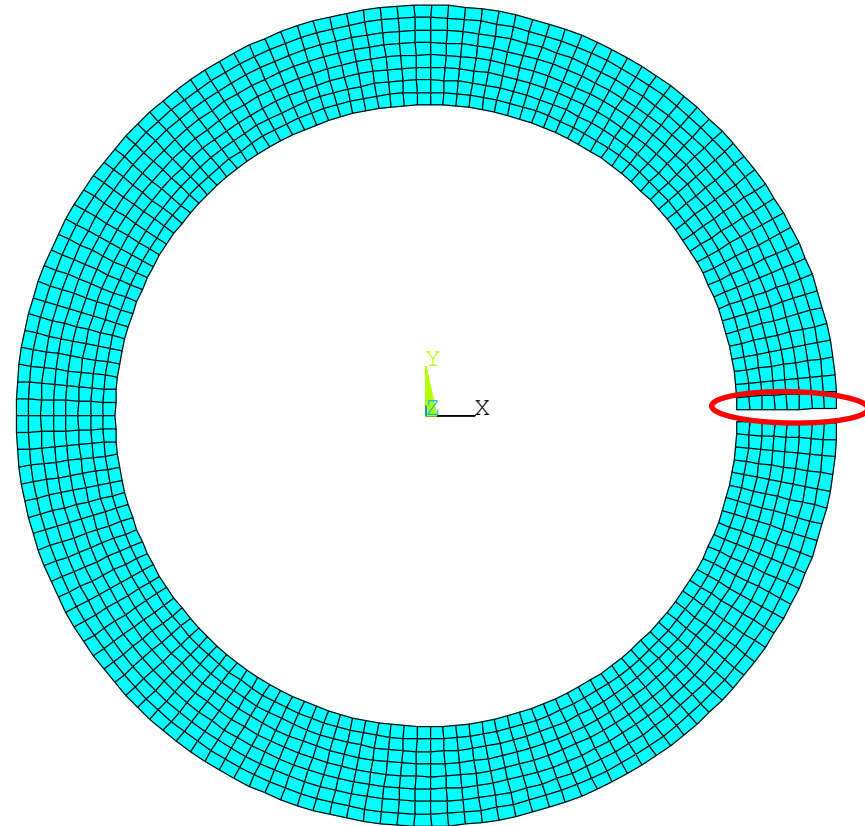
Apply

Reset

Cancel

Pick All

Help



# Condizioni di carico: pressione sulla linea uniforme

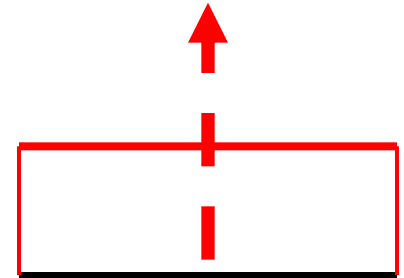
Apply PRES on lines

[SFL] Apply PRES on lines as a

If Constant value then:  
VALUE Load PRES value

If Constant value then:  
Optional PRES values at end J of line  
(leave blank for uniform PRES)  
Value

OK Apply Cancel Help



$$p = 2 \text{ MPa}, b = 4 \text{ mm}$$

$$P = p b = 8 \text{ N/mm}$$

$$F = P s = 12 \text{ N}$$

Alternativamente si può dare come input la forza  $F$

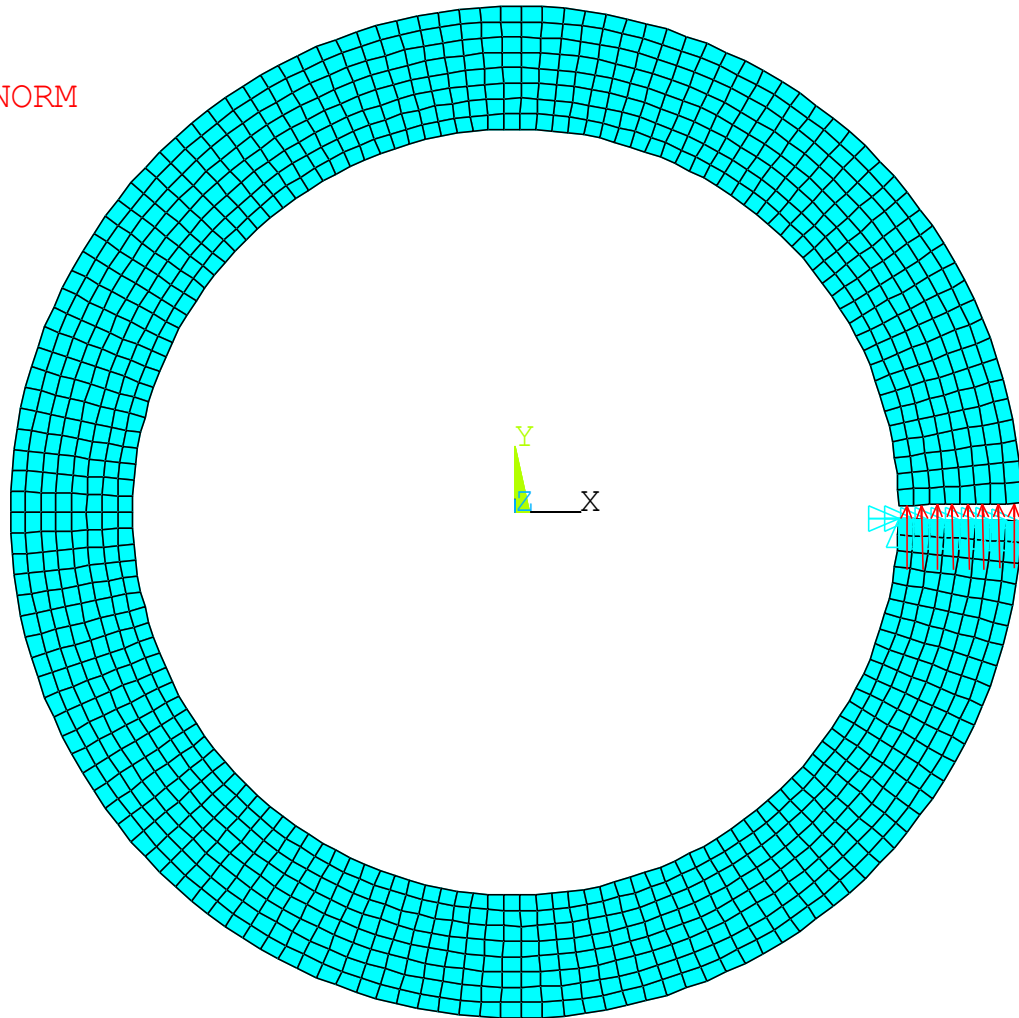
# Condizione di vincolo: incastro

## Condizioni di carico: pressione

U

PRES-NORM

2



*Piccolo errore nella  
direzione della forza*

# Solution

- [-] Solution
  - [+] Analysis Type
  - [+] Define Loads
  - [+] Load Step Opts
  - [+] SE Management (CMS)
  - [+] Results Tracking
  - [-] Solve
    - [+] Current LS → Calcola la soluzione
    - [+] From LS Files
    - [+] Partial Solu
  - [+] Manual Rezoning
  - [+] Multi-field Set Up
  - [+] ADAMS Connection
  - [+] Diagnostics
  - [+] Unabridged Menu

# Postprocessing: POST1

Rappresentazione **deformata**

**Listato numerico** dei risultati

**Plot grafico** dei risultati

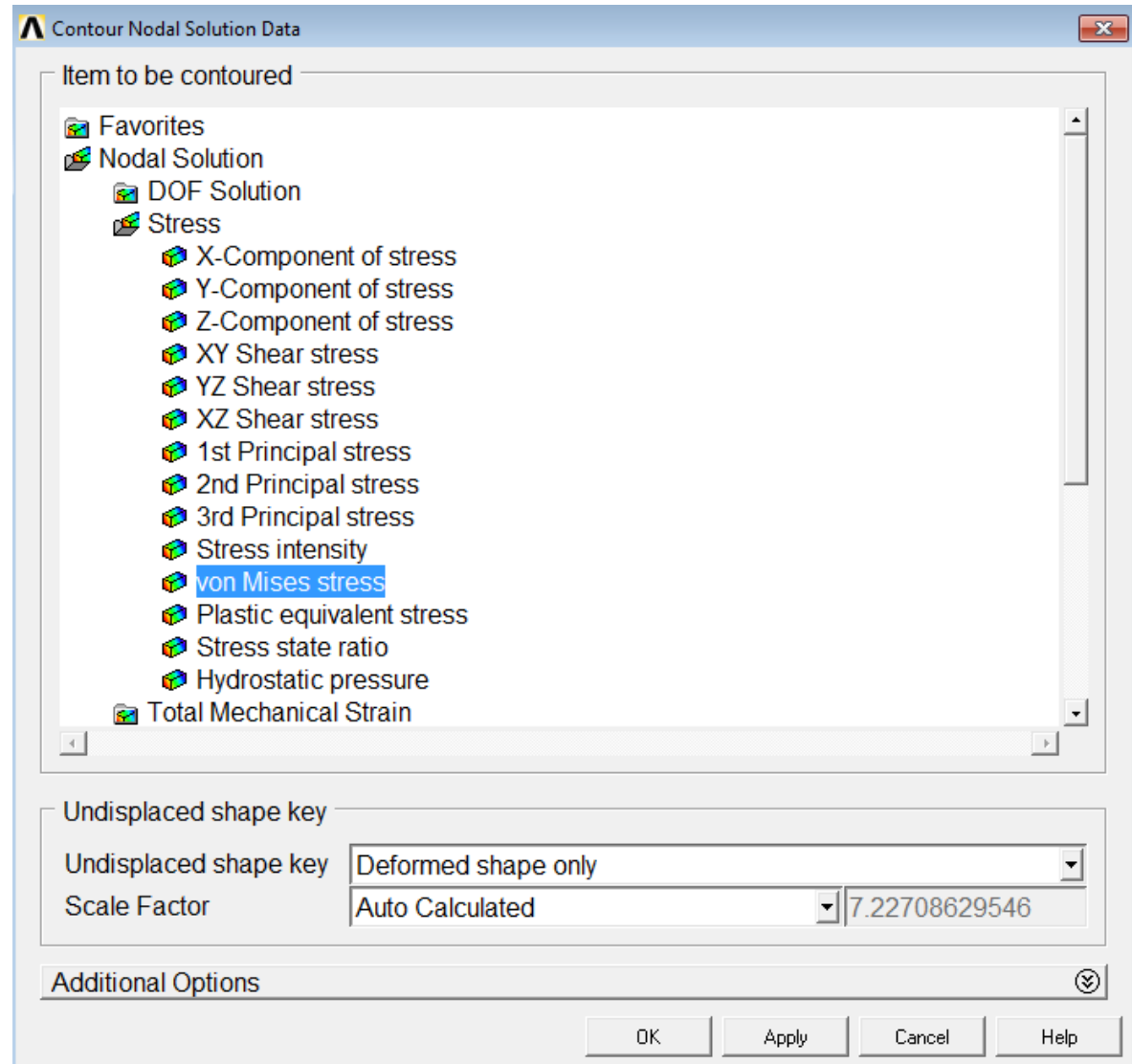
(tensioni eqv., tensioni principali, ecc.)

Grafici dell'andamento dei risultati su *path* definiti sul modello

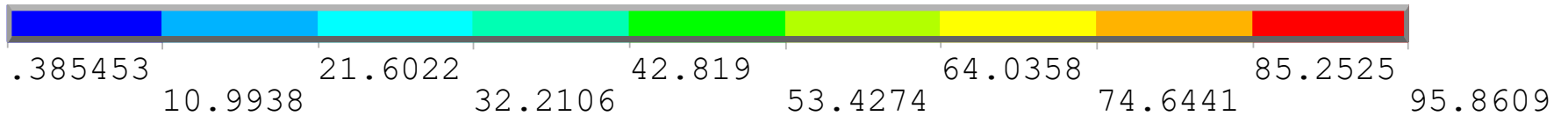
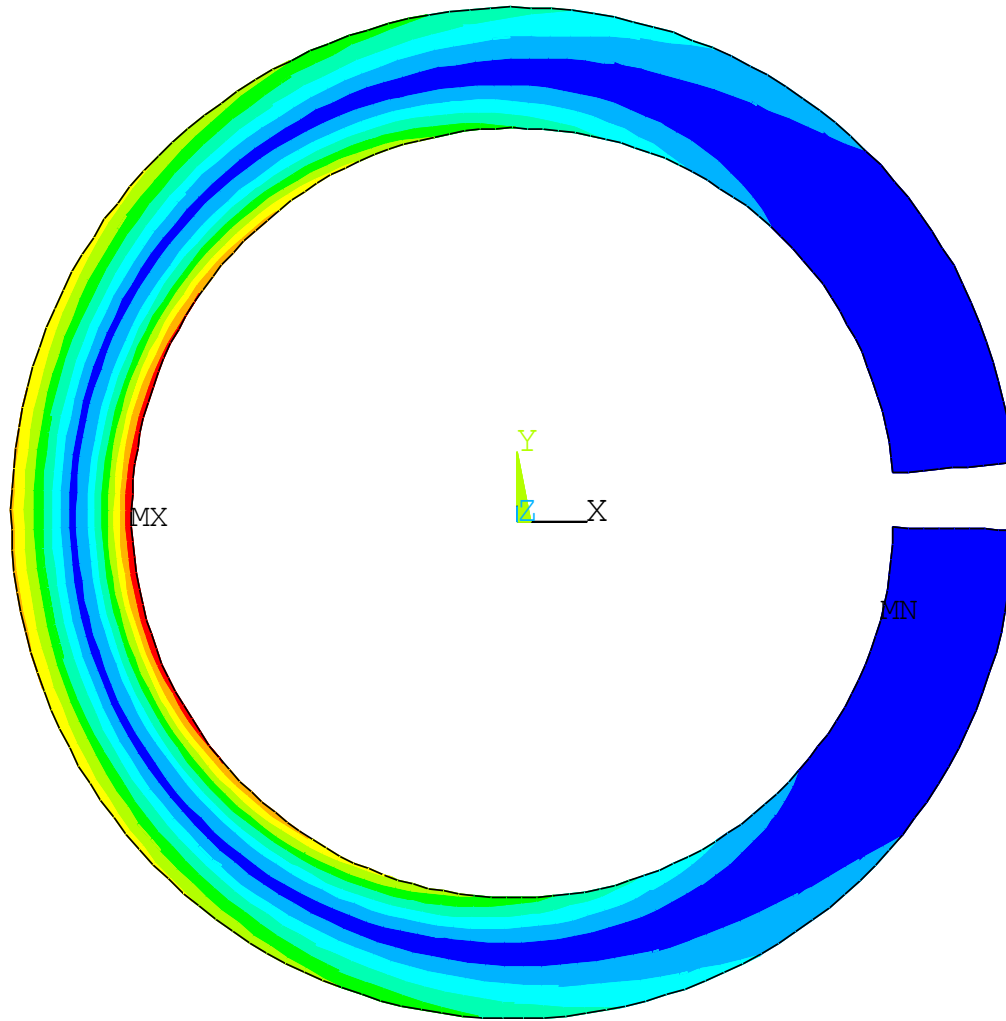
# Postprocessing: Plot results - Nodal Solution

## Tensione eq. von Mises

- [-] Preferences
- [+] Preprocessor
- [+] Solution
- [-] General Postproc
  - [-] Data & File Opts
  - [-] Results Summary
  - [+] Read Results
  - [+] Failure Criteria
  - [-] Plot Results
    - [-] Deformed Shape
      - [-] Contour Plot
        - [+] Nodal Solu
        - [+] Element Solu
        - [+] Elem Table
        - [+] Line Elem Res
    - [+] Vector Plot
    - [+] Plot Path Item
    - [+] Concrete Plot
    - [+] ThinFilm
  - [+] List Results



# Postprocessing: Tensione eq. von Mises

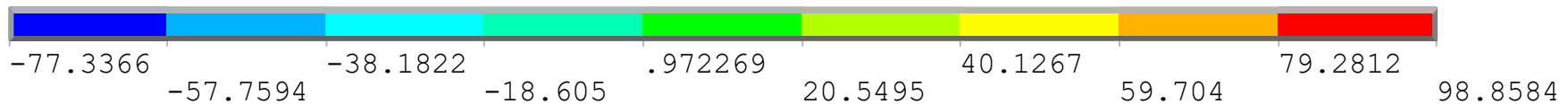
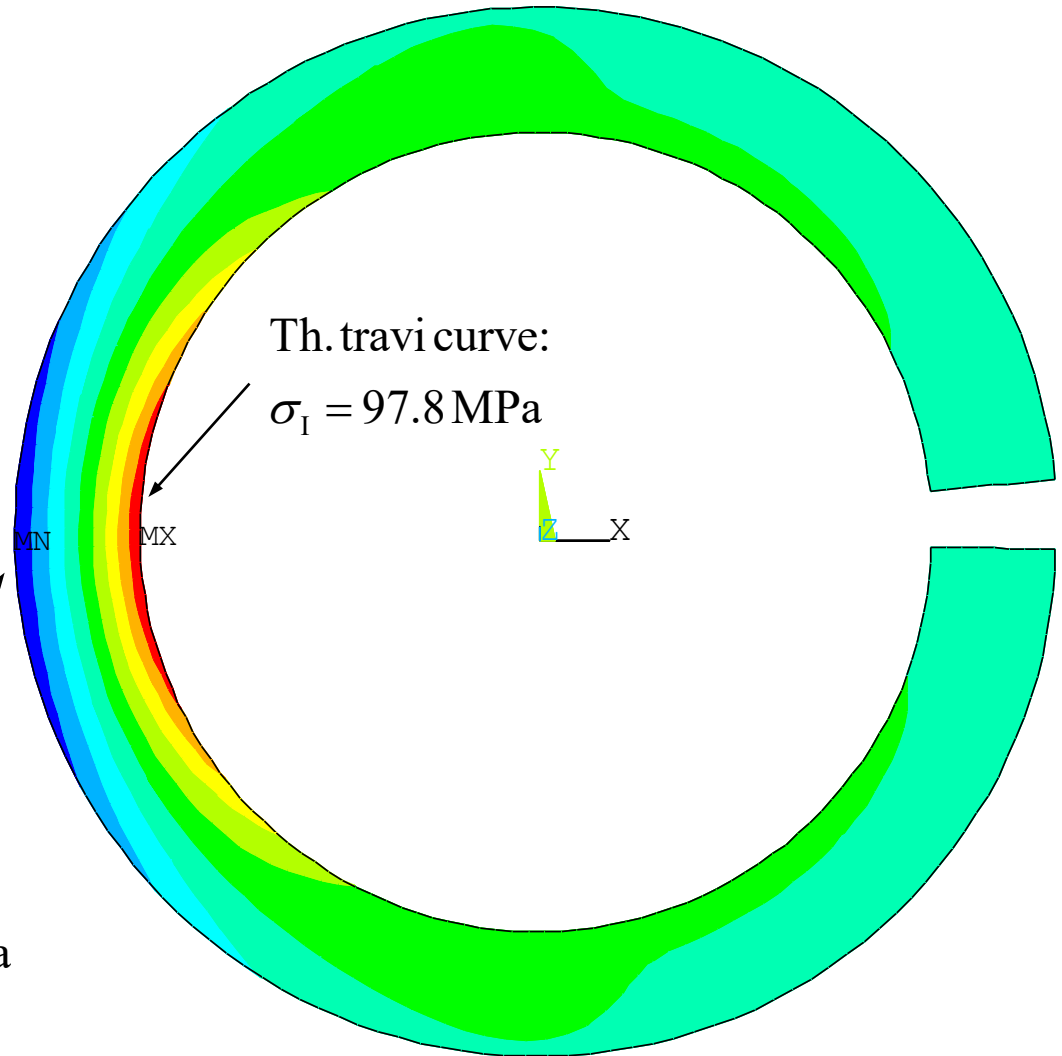


# Postprocessing: sigma\_Y

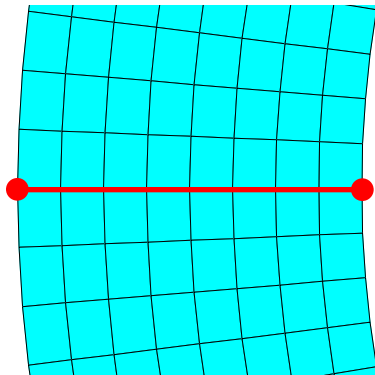
## Confronto con modello trave curva

Calcolo flessione, trave a forte curvatura (anello Seeger)

D_I, mm	D_E, mm	Spessore radiale, mm	
25	33	4	
D_m, mm	p, MPa	Spessore assiale	
29	2	2	
	F, N		
M_f, N mm	16		
464			
W, mm^3	A, mm^2	r_I, mm	r_E, mm
5.333	8	12.5	16.5
	r_G, mm	r_N	e, mm
sigma_0, MPa	14.500	14.408	0.092
87	c_I, mm	c_E, mm	
	1.908	2.092	
sigma_I,B, MPa	sigma_E,B MPa	sigma_T, MPa	
95.8	-79.6	2.0	
sigma_I, MPa	sigma_E, MPa		
97.8	-77.6		

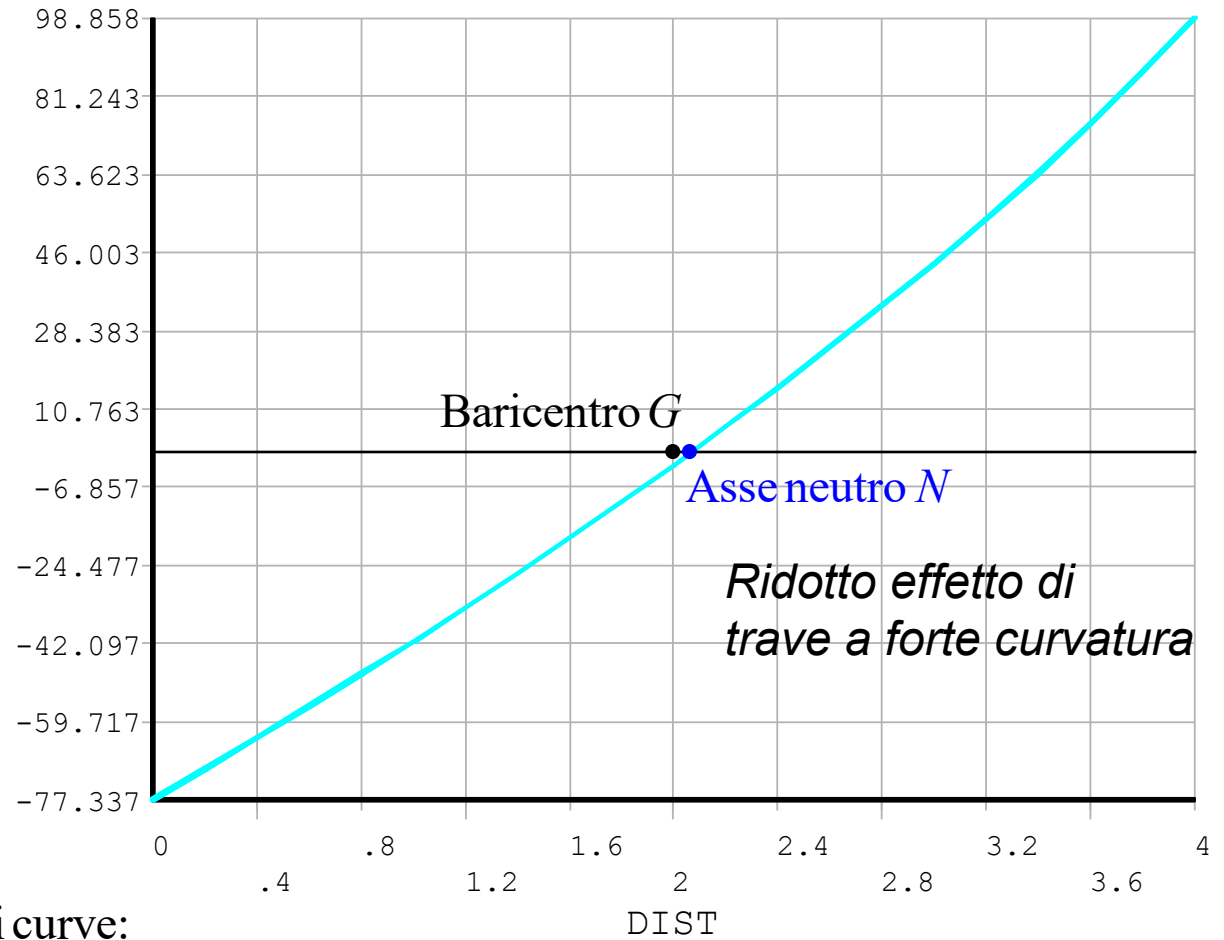


# Postprocessing: sigma\_Y, utilizzo del "path"



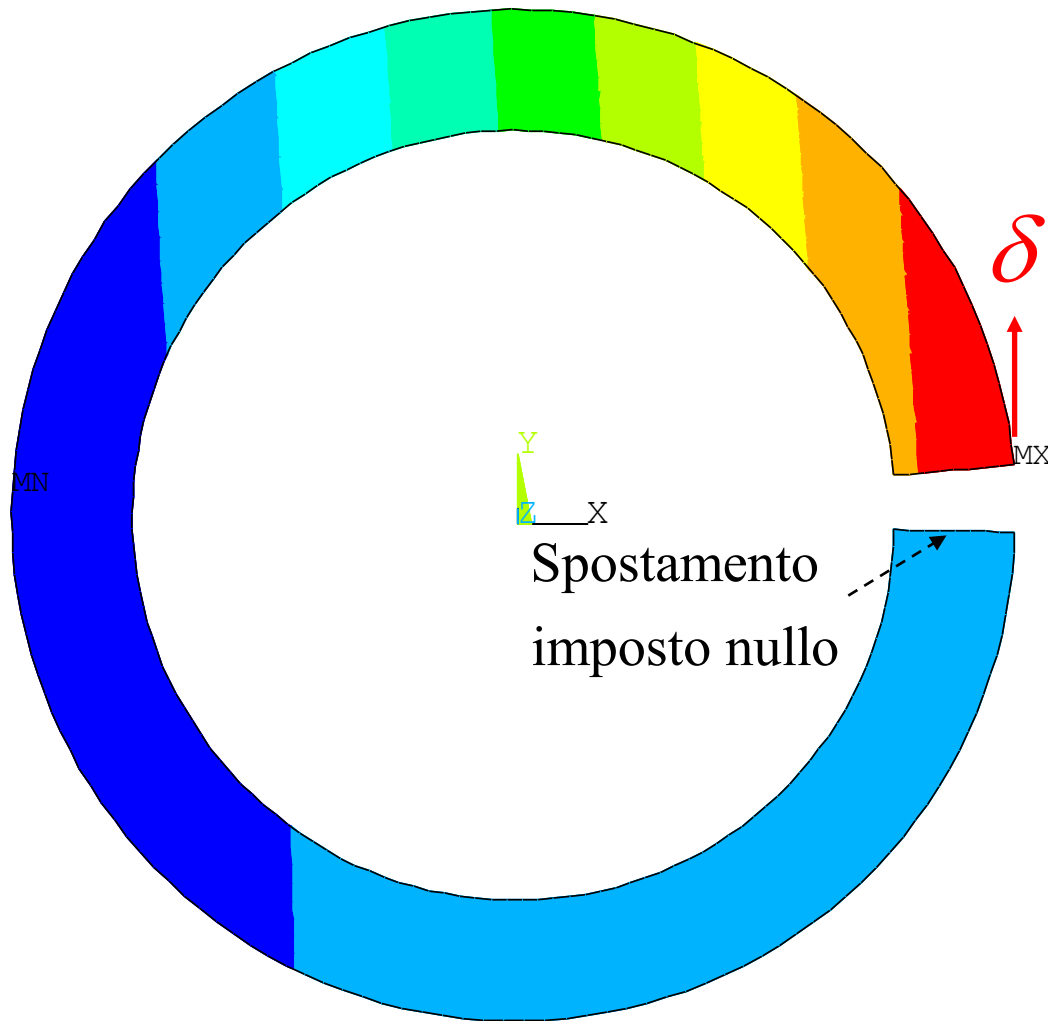
Th. travi curve:  
 $\sigma_E = -77.6 \text{ MPa}$

Th. travi curve:  
 $\sigma_I = 97.8 \text{ MPa}$



*Ridotto effetto di  
trave a forte curvatura*

# Postprocessing: Spostamento secondo Y



Calcolo di rigidezza:

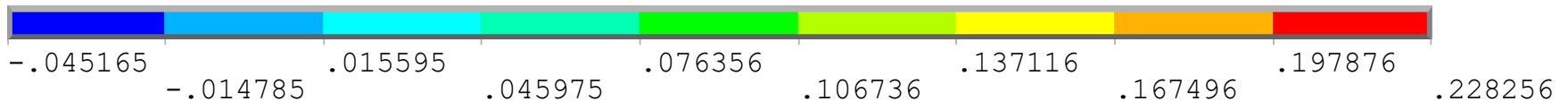
$$\text{Es.: } s = 1.5 \text{ mm}$$

$$F = Ps = 12 \text{ N}$$

$$\delta = 0.228 \text{ mm}$$

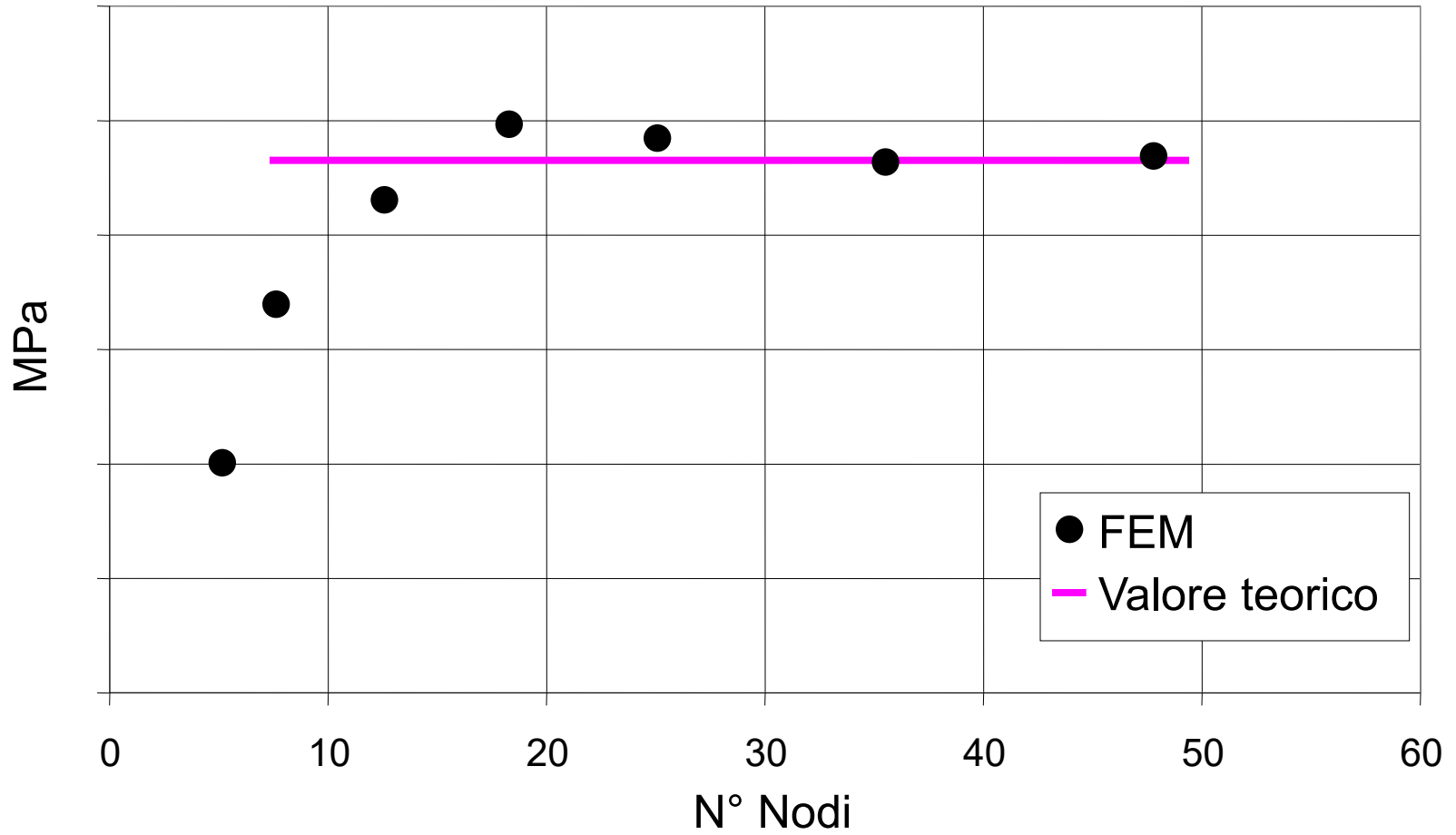
$$\rightarrow K = \frac{F}{\delta} = 52.6 \text{ N/mm}$$

N.B.: si potrebbe calcolare la rigidezza anche per spessori diversi, tuttavia piccoli altrimenti no Plane Stress



# Analisi di convergenza

## Tensione Max



# Ansys Workbench

The screenshot displays the ANSYS Workbench interface for a static structural analysis. The main window shows a 3D model of a mechanical assembly consisting of a cylindrical shaft and a flange-like component. The shaft is labeled "Corpi diversi" and has a fixed support (A) at its base. A force (B) of 100.5 N is applied to the top of the shaft. The flange component has a red dashed circle highlighting the contact surface, labeled "Superficie di interfaccia". The interface includes a project tree on the left, a simulation setup panel on the right, and a details panel at the bottom left.

**Statico strutturale**  
Tempo: 1. s  
12/02/2008 10.51

**A** Vincolo fisso  
**B** Forza: 100.5 N

**Progetto**

- Modello
  - Geometria
  - Connessioni**
  - Mesh
- Statico strutturale
  - Impostazioni analisi
  - Vincolo fisso
  - Forza
- Soluzione
  - Informazioni sulla soluzione
  - Sollecitazione equivalente

**Definizione**

Tipo di fisica	Strutturale
Tipo di analisi	Statico strutturale

**Opzioni**

Temperatura di rif.	22. °C
---------------------	--------

0.00 22.50 45.00 67.50 90.00 (mm)

ANSYS

- Possibilità di **importare** modelli 3D dai principali **Software CAD**:

Pro/E, OneSpace, **SolidWorks**, CATIA, Unigraphics

- Identificazione automatica delle interfacce di contatto – **Connessioni (Connections)**

- Utilizzo semplice ed intuitivo / alcune limitazioni nell'utilizzo di funzioni avanzate

# Introduzione al Metodo agli Elementi Finiti

**Esempio di utilizzo di Ansys Workbench**

Ing. Ciro Santus

<http://people.unipi.it/static/ciro.santus/>

# Ansys Workbench

Unsaved Project - Workbench

File View Tools Units Extensions Jobs Help

Project

Import... Reconnect Refresh Project Update Project ACT Start Page

**Toolbox**

- Analysis Systems
  - Design Assessment
  - Eigenvalue Buckling
  - Electric
  - Explicit Dynamics
  - Harmonic Acoustics
  - Harmonic Response
  - Magnetostatic
  - Modal
  - Modal Acoustics
  - Random Vibration
  - Response Spectrum
  - Rigid Dynamics
  - Static Structural
  - Steady-State Thermal
  - Thermal-Electric
  - Topology Optimization
  - Transient Structural
  - Transient Thermal
- Component Systems
- Custom Systems
- Design Exploration

**Project Schematic**

A

- Static Structural
- Engineering Data
- Geometry
- Model
- Setup
- Solution
- Results

Static Structural

**Properties of Project Schemer**

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Solution Process	
5	Update Option	R. ▾

**Messages**

	A	B	C	D
1	Type	Text	Association	Date/Time

View All / Customize...

Ready

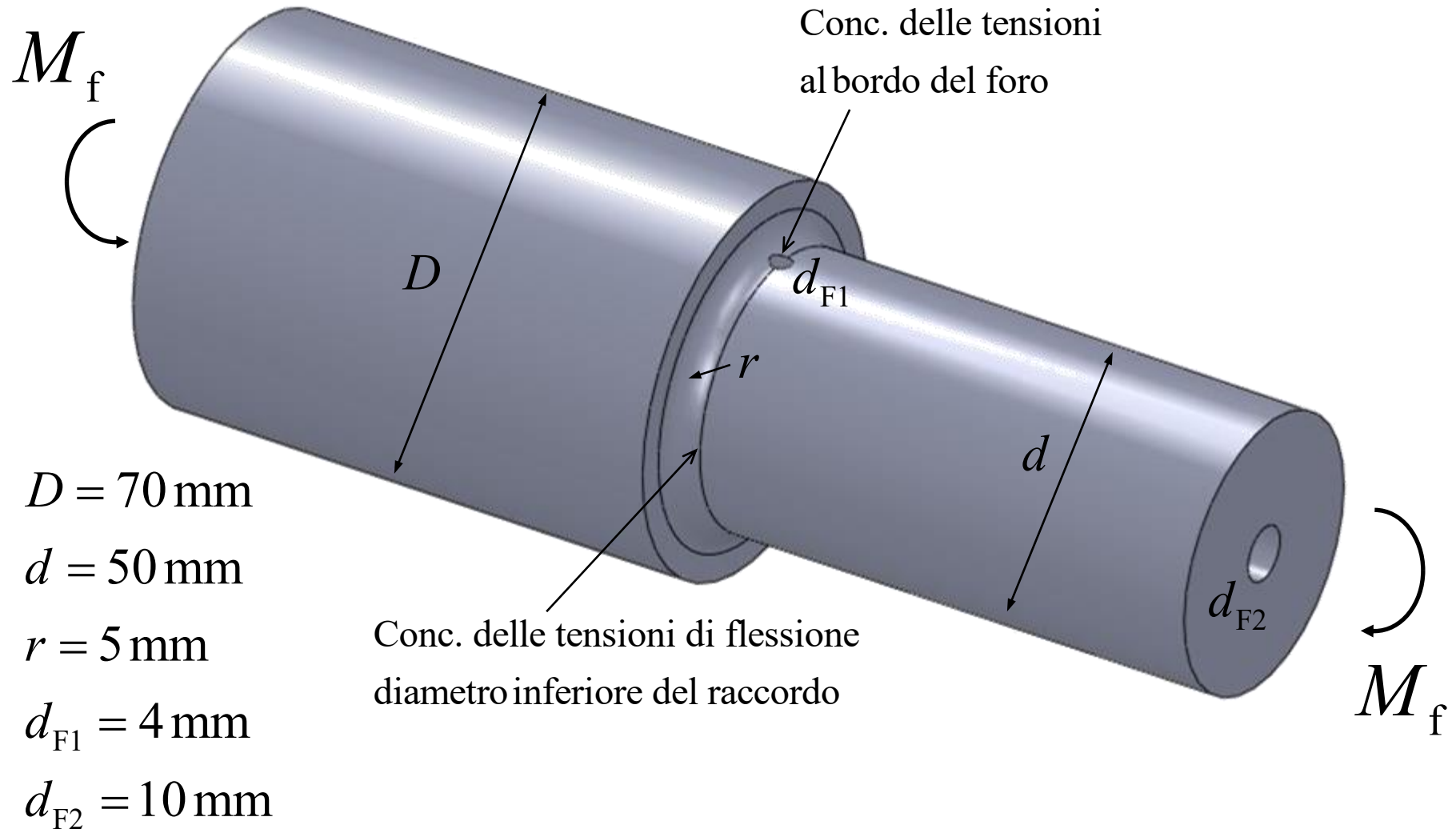
Job Monitor... Show Progress Hide 0 Messages

Diversi moduli / tipi di analisi

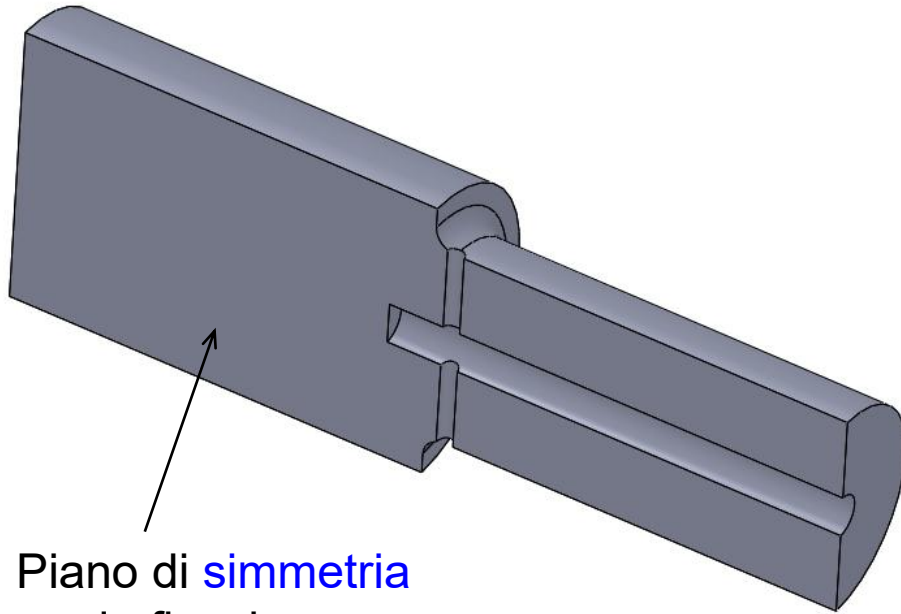
Es.:

- Statico strutturale
- Modale
- Transitorio dinamico
- Analisi termica
- Etc.

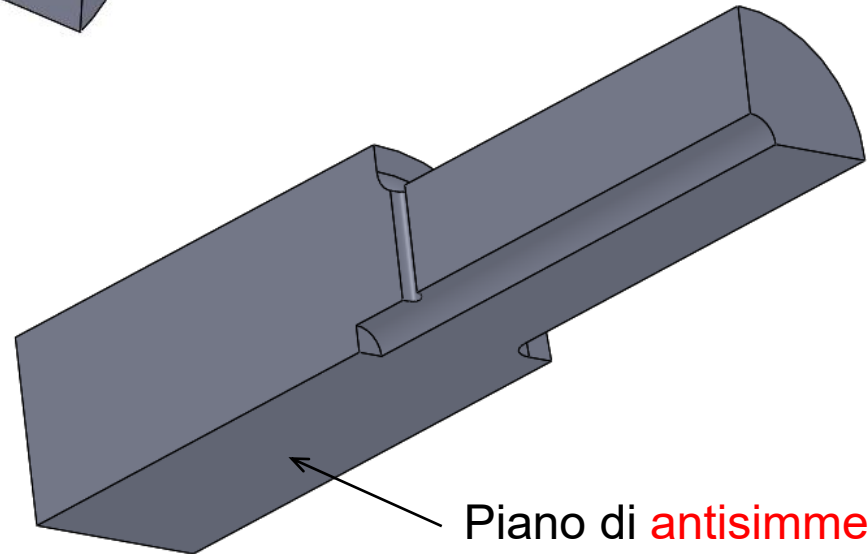
# Esempio 1: Albero con variazione di diametro e raggio di raccordo, determinazione del $K_t$



# Utilizzo delle simmetrie – Simmetria rispetto ad o più un piani

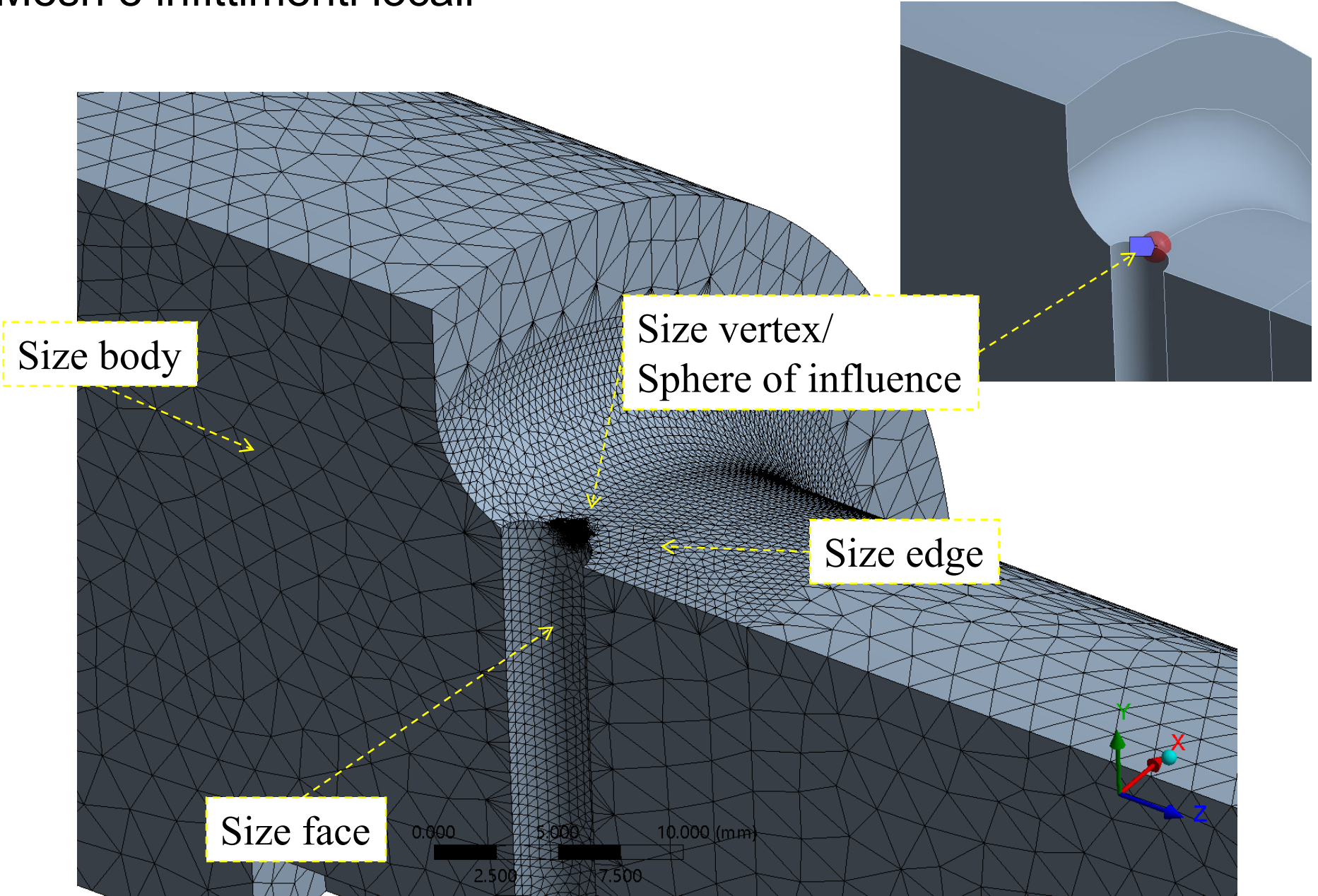


Piano di **simmetria**  
per la flessione

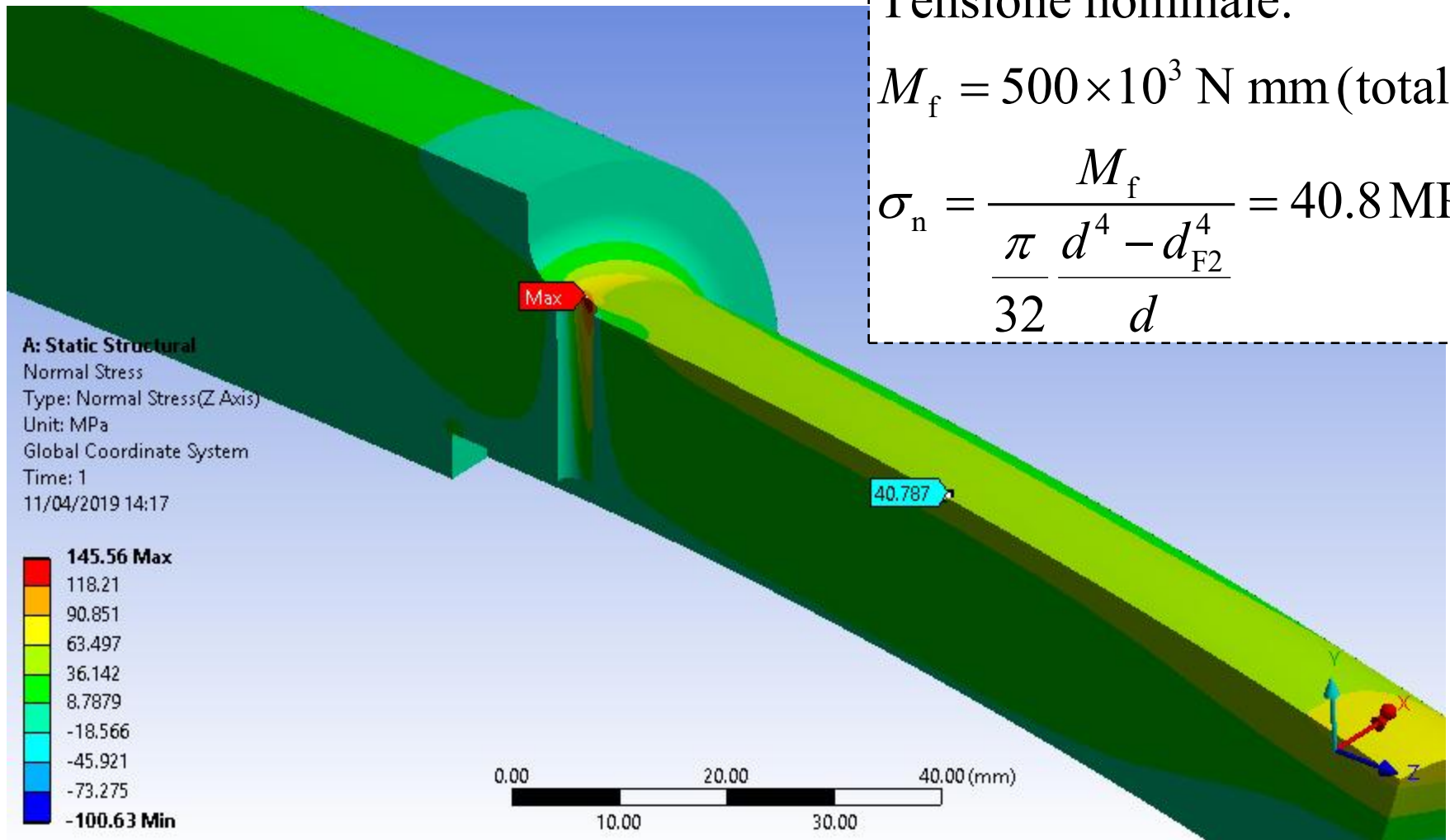


Piano di **antisimmetria**  
per la flessione

# Mesh e infittimenti locali



# Soluzione, tensione nominale

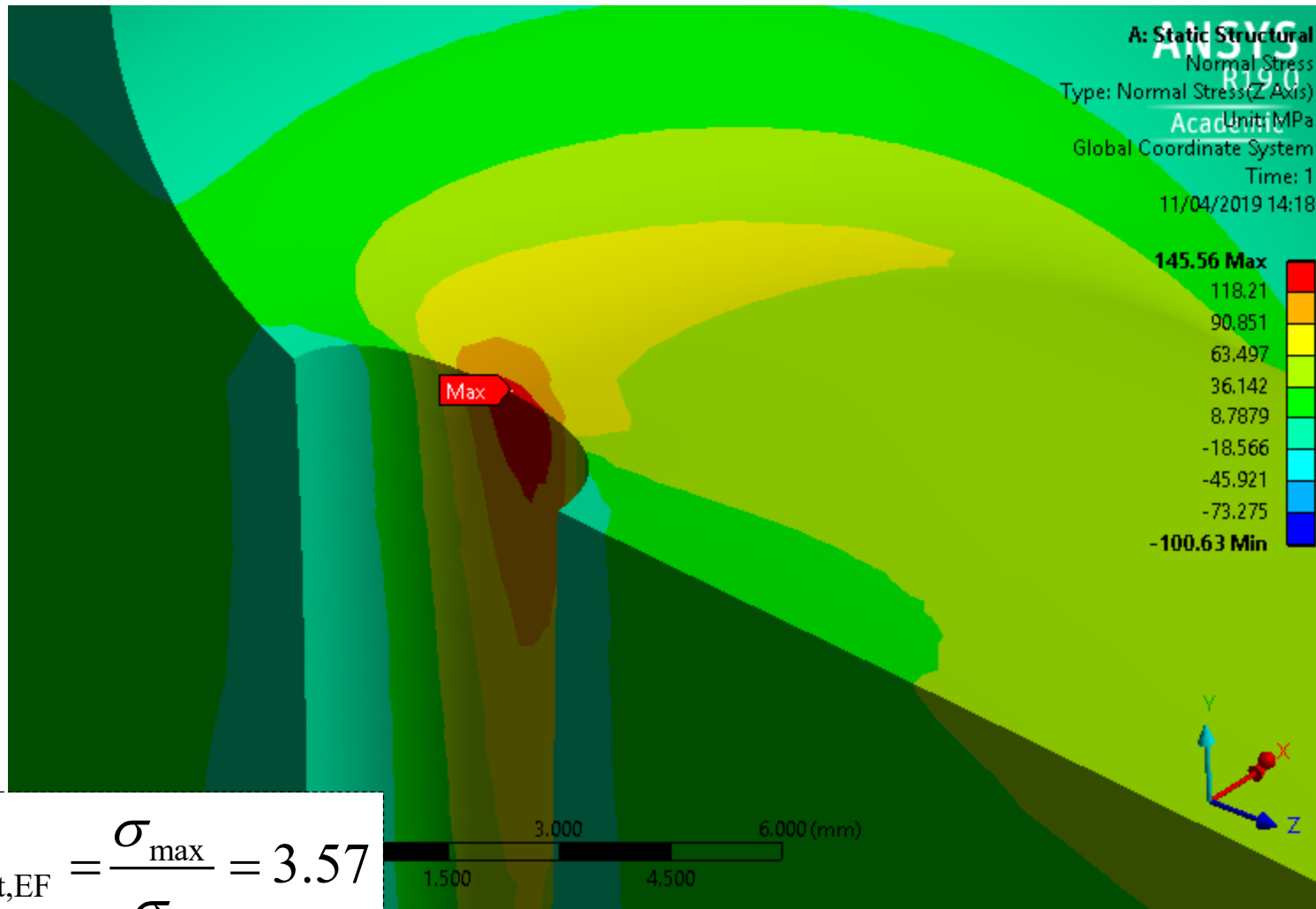


Tensione nominale:

$$M_f = 500 \times 10^3 \text{ N mm (totale)}$$

$$\sigma_n = \frac{M_f}{\frac{\pi}{32} \frac{d^4 - d_{F2}^4}{d}} = 40.8 \text{ MPa}$$

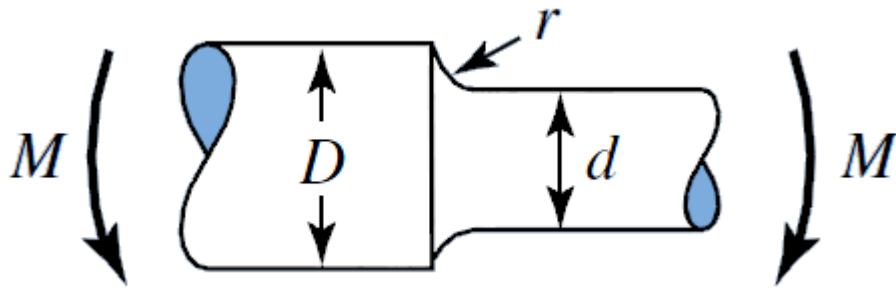
# Soluzione, concentrazione delle tensioni



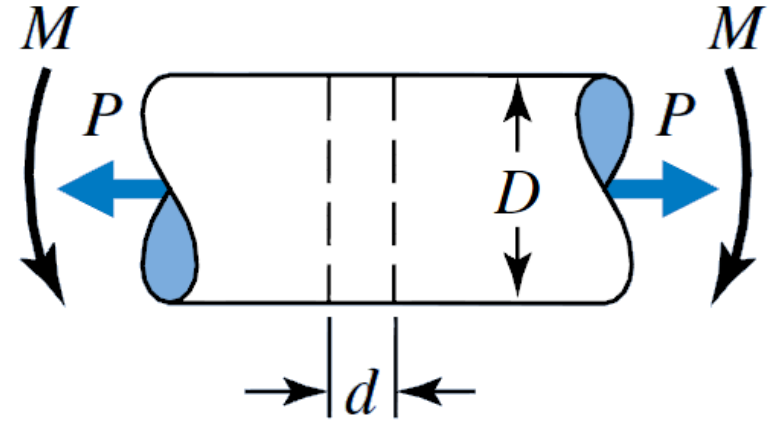
$$K_{t,EF} = \frac{\sigma_{\max}}{\sigma_n} = 3.57$$

# Soluzione di riferimento

Combinazione di concentrazione delle tensioni – prodotto di  $K_t$



$$K_{t,1} = 1.57$$



$$K_{t,2} = 2.69$$

$$K_t \approx K_{t,1} \times K_{t,2} = 4.22$$

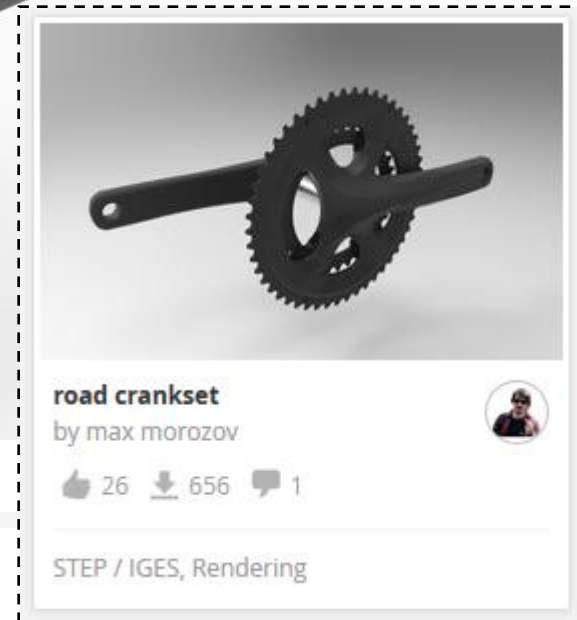
$$(K_{t,EF} = 3.57)$$

# Esempio 2: Complessivo pedale e corona per catena bicicletta

Possibilità di importare il modello CAD 3D, tuttavia è suggerito il “defeaturing”: eliminazione dei dettagli geometrici non utili ai fini dell’analisi



Fonte: GrabCAD - <https://grabcad.com/>



# Importazione del modello 3D

**A : Static Structural - Mechanical [ANSYS Academic Research Mechanical and CFD]**

File Edit View Units Tools Help

Solve New Analysis Show Errors Worksheet

Show Vertices Close Vertices 0.45 (Auto Scale) Wireframe Show Mesh Random Preferences

Size Location Convert Miscellaneous Tolerances Clipboard [ Empty ]

Reset Explode Factor: Assembly Center Edge Coloring Thicken

Solution Deformation Strain Stress Energy Damage Linearized Stress Probe Tools User Defined Result Campbell Diagram

**Outline**

Filter: Name

**Project**

- Model (A4)
  - Geometry
  - Coordinate Systems
  - Connections
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Solution (A6)
    - Solution Information

**Details of "Solution (A6)"**

- Adaptive Mesh Refinement**
  - Max Refinement Loops: 1.
  - Refinement Depth: 2.
- Information**

Status	Solve Required
MAPDL Elapsed Time	18. s
MAPDL Memory Used	1.6172 GB
MAPDL Result File Size	49.875 MB
- Post Processing**

Beam Section Results	No
On Demand Stress/Strain	No

**A: Static Structural**  
Solution  
Time: 1. s  
18/10/2018 15:40

**ANSYS R19.0 Academic**

0.00 100.00 (mm)  
50.00

**Geometry** / Print Preview / Report Preview /

**Graph**

0. 0.125 0.25 0.375 0.5 0.625 0.75 0.875 1.0  
[s]

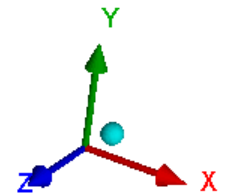
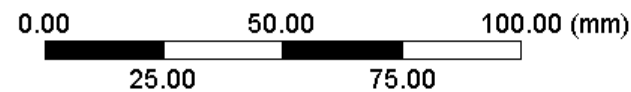
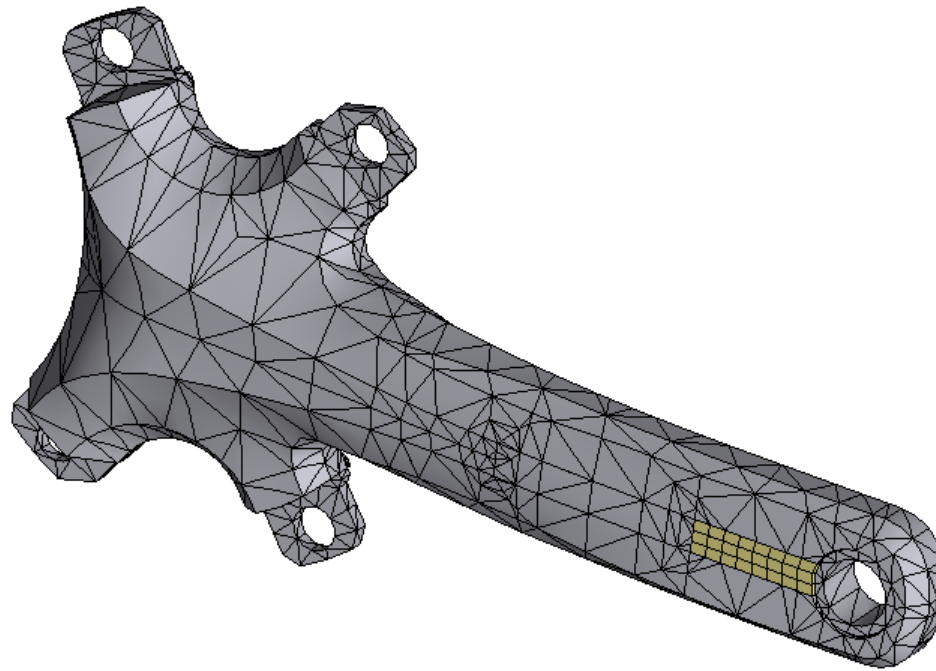
**Tabular Data**

Time [s]	Step	Substep
1.	1.	1.

Graphics Annotations Messages **Graph**

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees ra

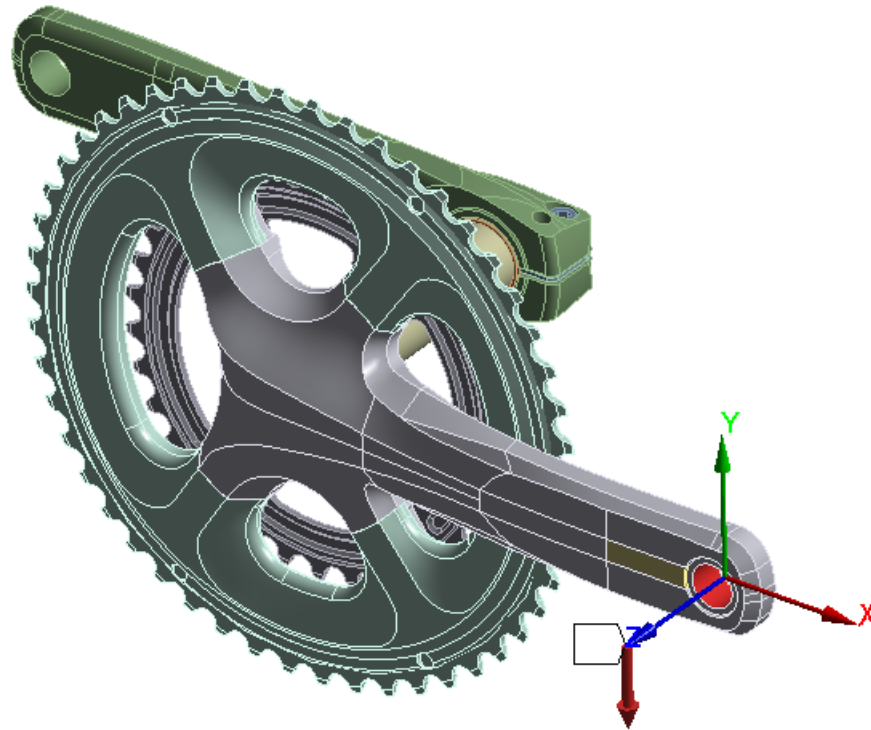
# Obiettivo dell'analisi



Elemento di interesse dell'analisi:  
Pedivella (destra) di pezzo con la flangia di collegamento alla corona

Suddivisione in elementi (Mesh) iniziale

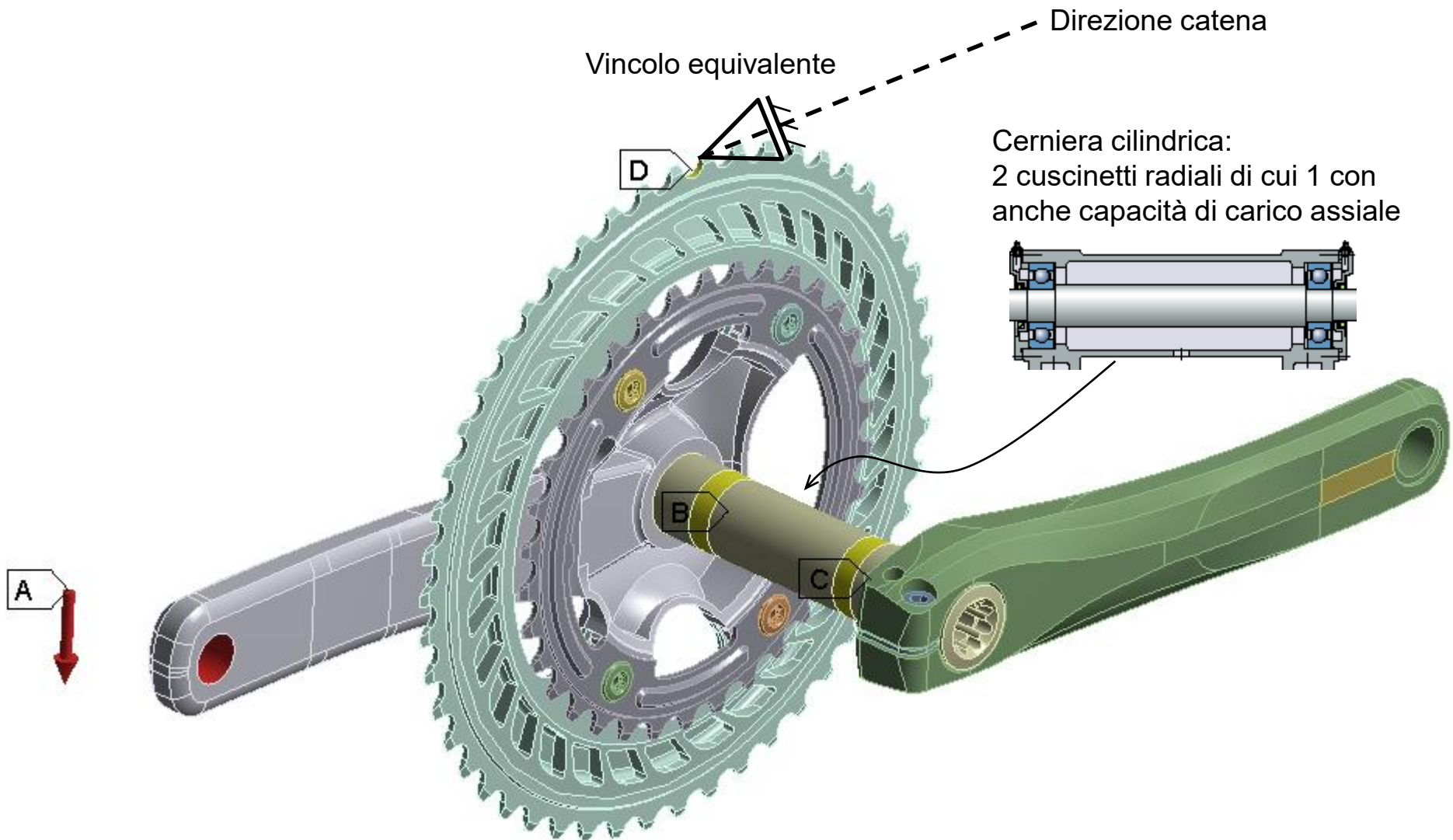
# Applicazione del carico



Possibilità di applicare una forza *Remota*, in corrispondenza del centro del pedale

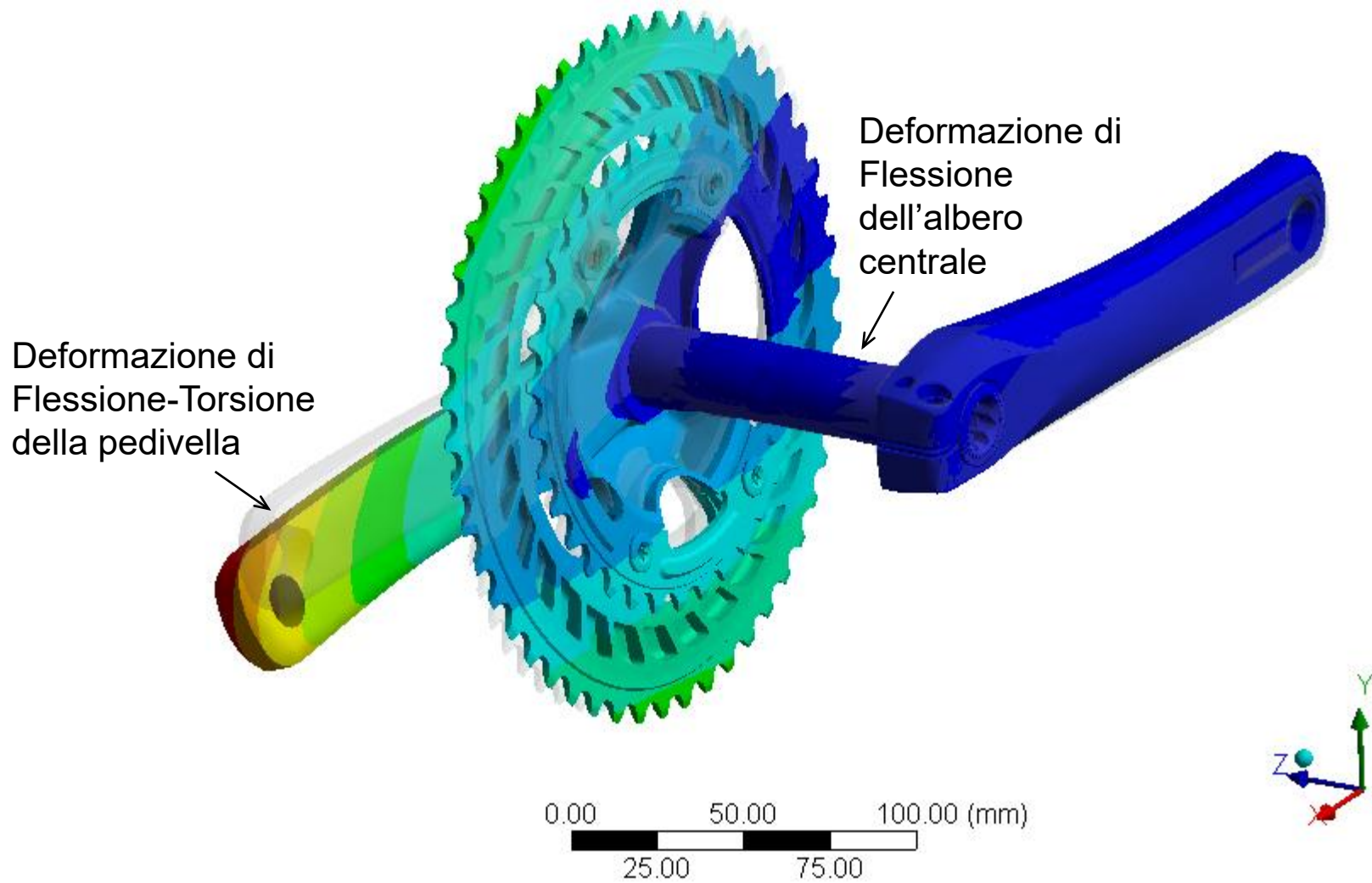
Sollecitazione di flessione e torsione agente sulla pedivella

# Applicazione dei vincoli

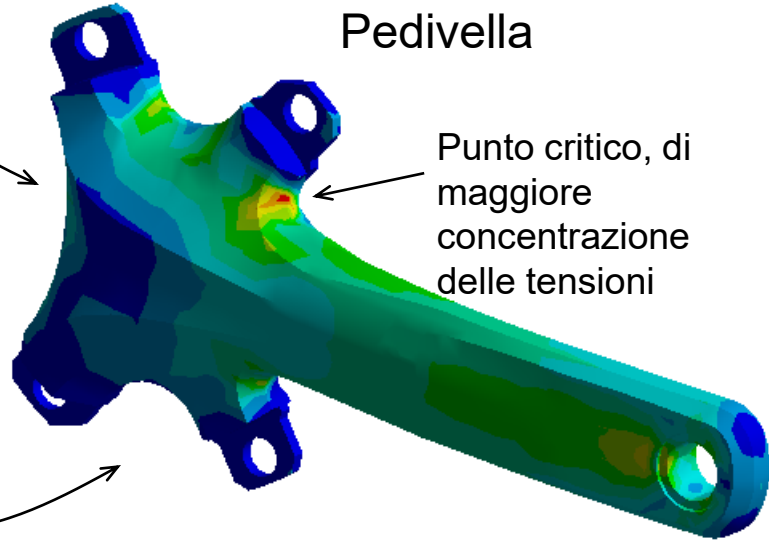
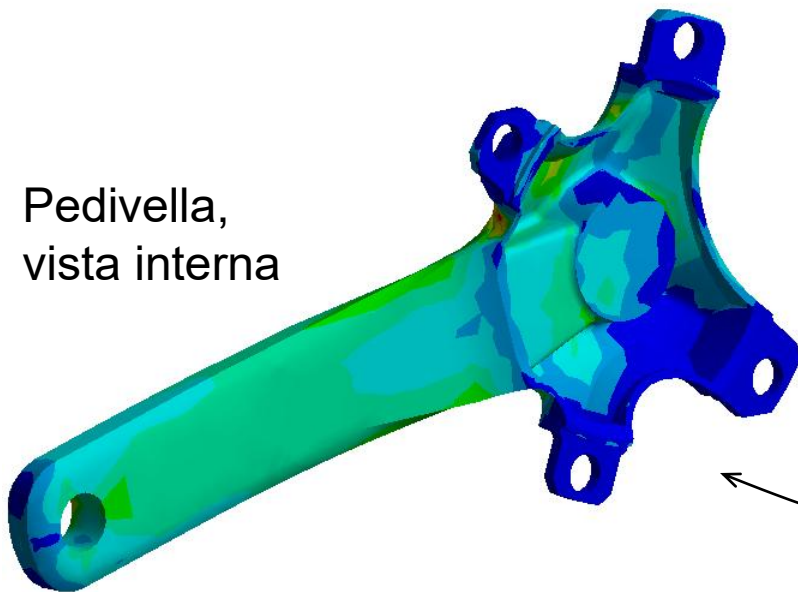
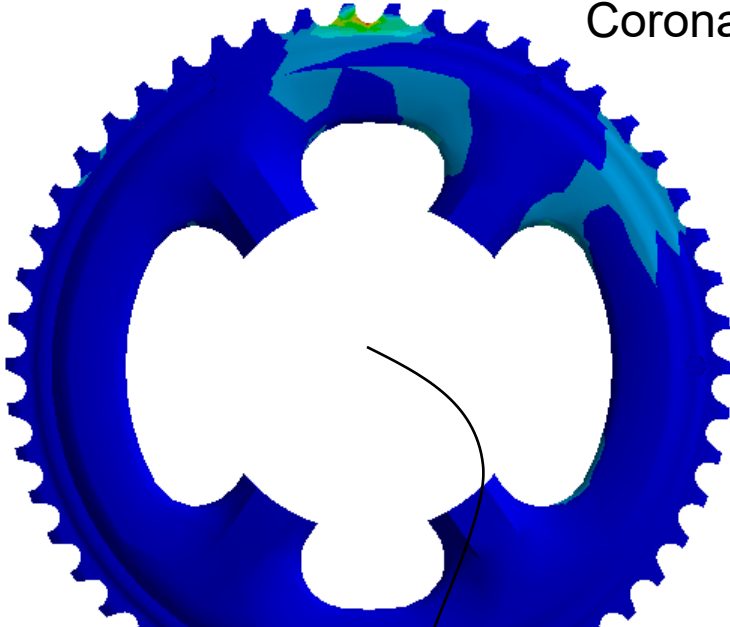


Applicazione di vincoli che riproducono i cuscinetti B – C  
e l'azione della catena, vincolo D

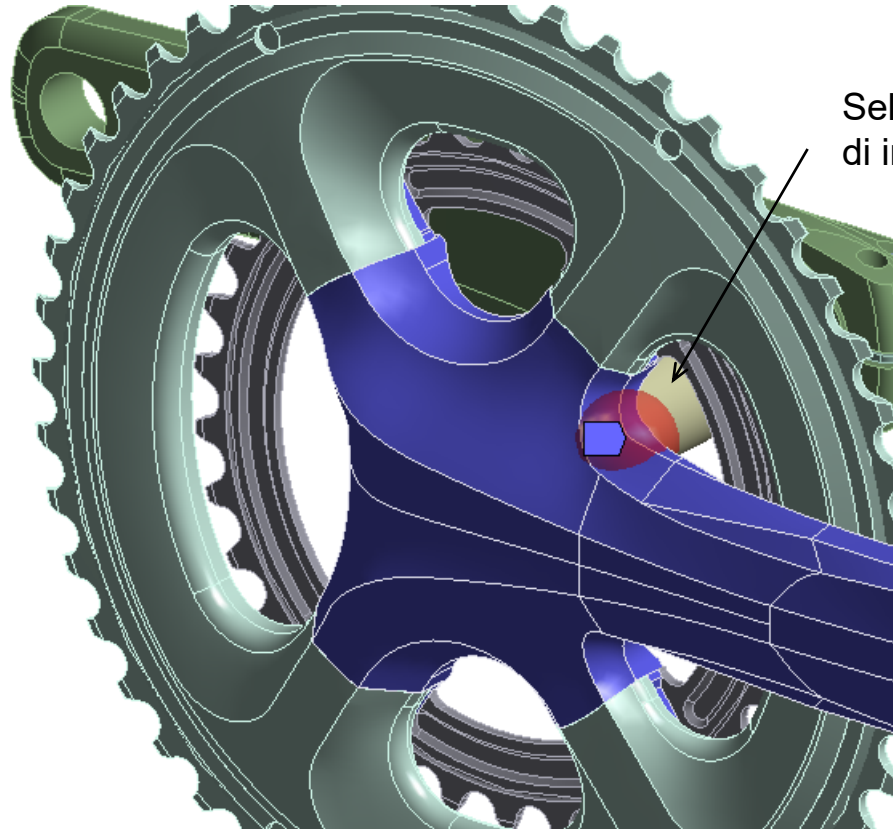
# Soluzione – campo di Spostamenti



# Soluzione – distribuzione della tensione equivalente (von Mises)

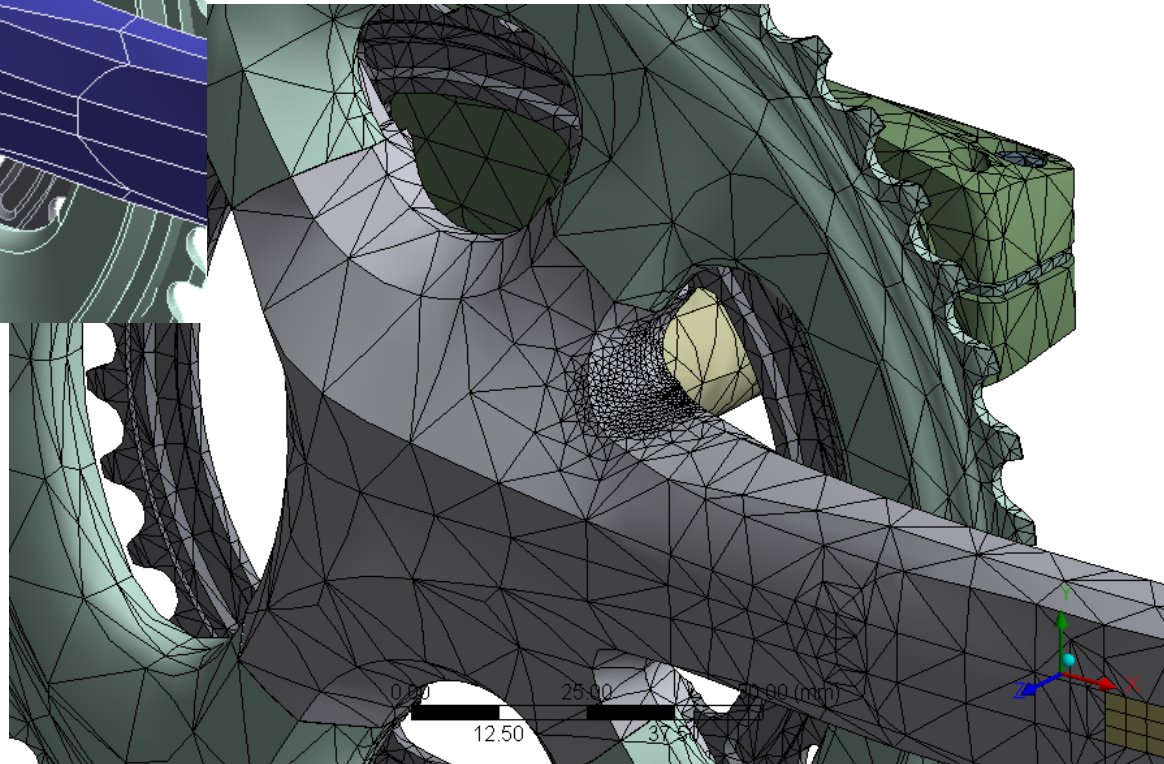


# Infittimento della Mesh nella zona di maggiore concentrazione



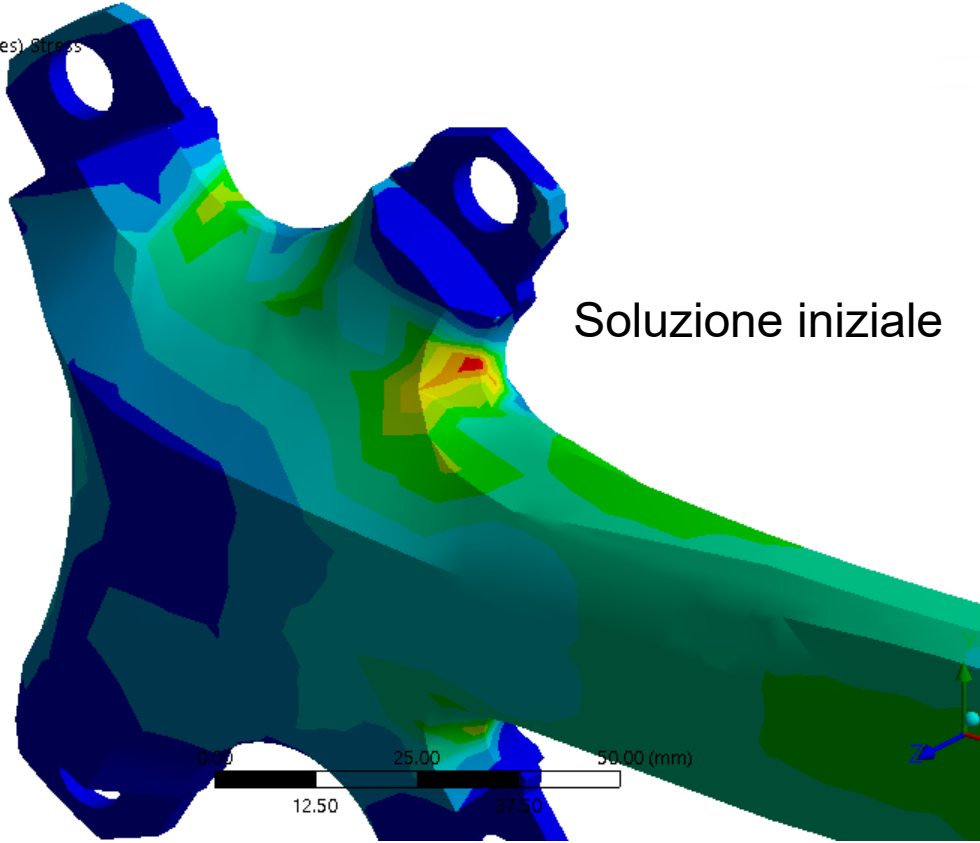
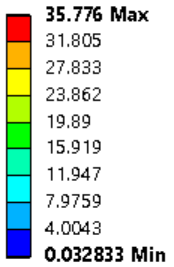
Selezione, a sfera, della zona di infittimento

Nuova Mesh con infittimento locale



# Soluzione con infittimento

**A: Static Structural**  
Equivalent Stress 2  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 1  
18/10/2018 16:17



**A: Static Structural**  
Equivalent Stress 2  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 1  
18/10/2018 16:16

