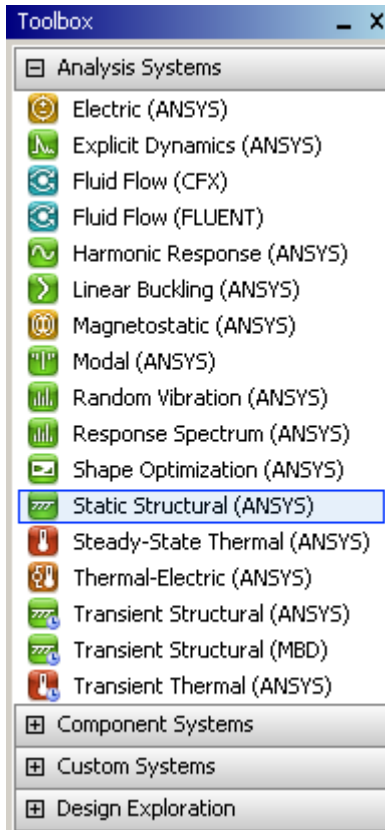


Corso di
Metodi Computazionali per l'Analisi Strutturale (MCAS)
CLM Ing. Veicoli/Meccanica

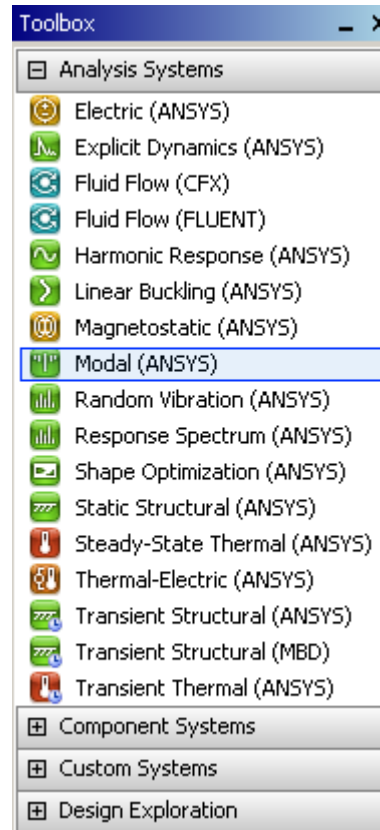
Esercitazioni guidate di
Ansys Workbench

1. Modellazione di parti singole
2. Connessioni fra più parti (Connections, Contact&Target)
3. Semplificazioni dei contatti
4. Riduzione del modello sfruttando le simmetrie
5. Carico di forza remota
6. Vincolo di spostamento remoto
7. Utilizzo del sottomodello

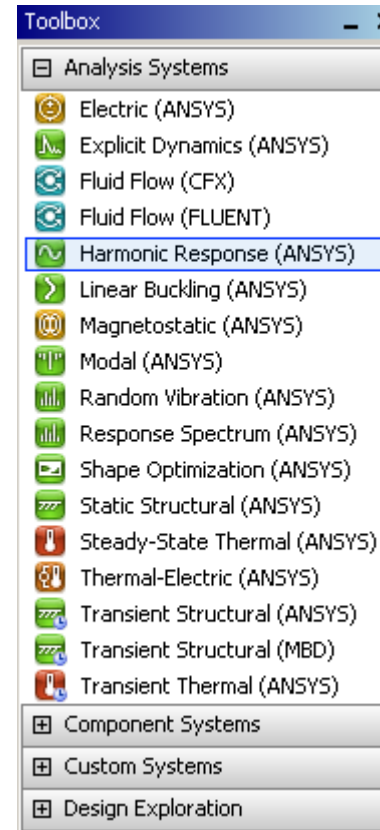
Diverse tipologie di analisi con Ansys Workbench



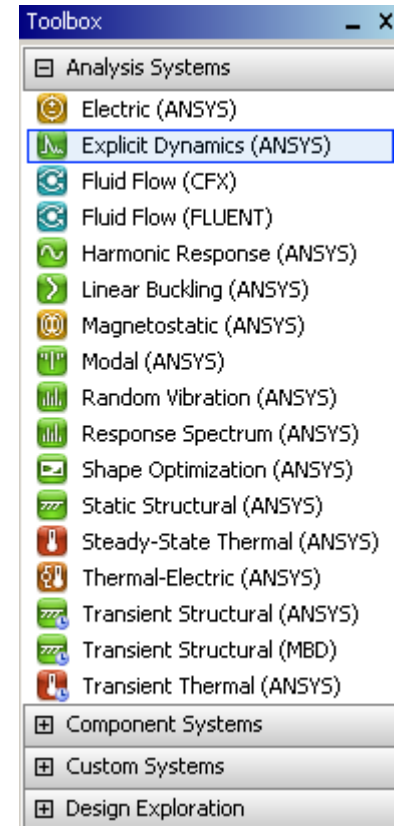
Statica



Modale



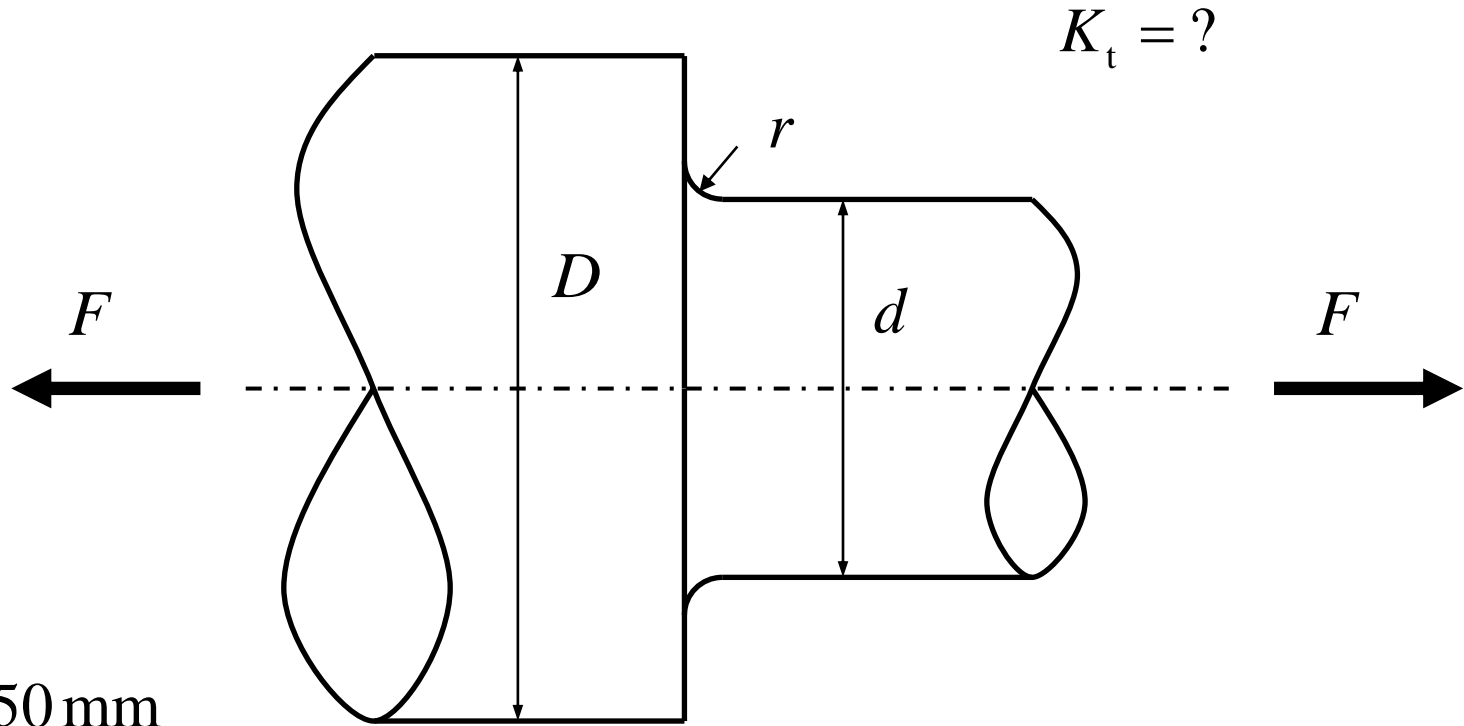
Risposta
armonica



Transitorio,
Analisi
termiche etc.

Lez. 1. Modellazione di parti singole in Ansys Workbench

Albero con variazione di diametro e raggio di raccordo

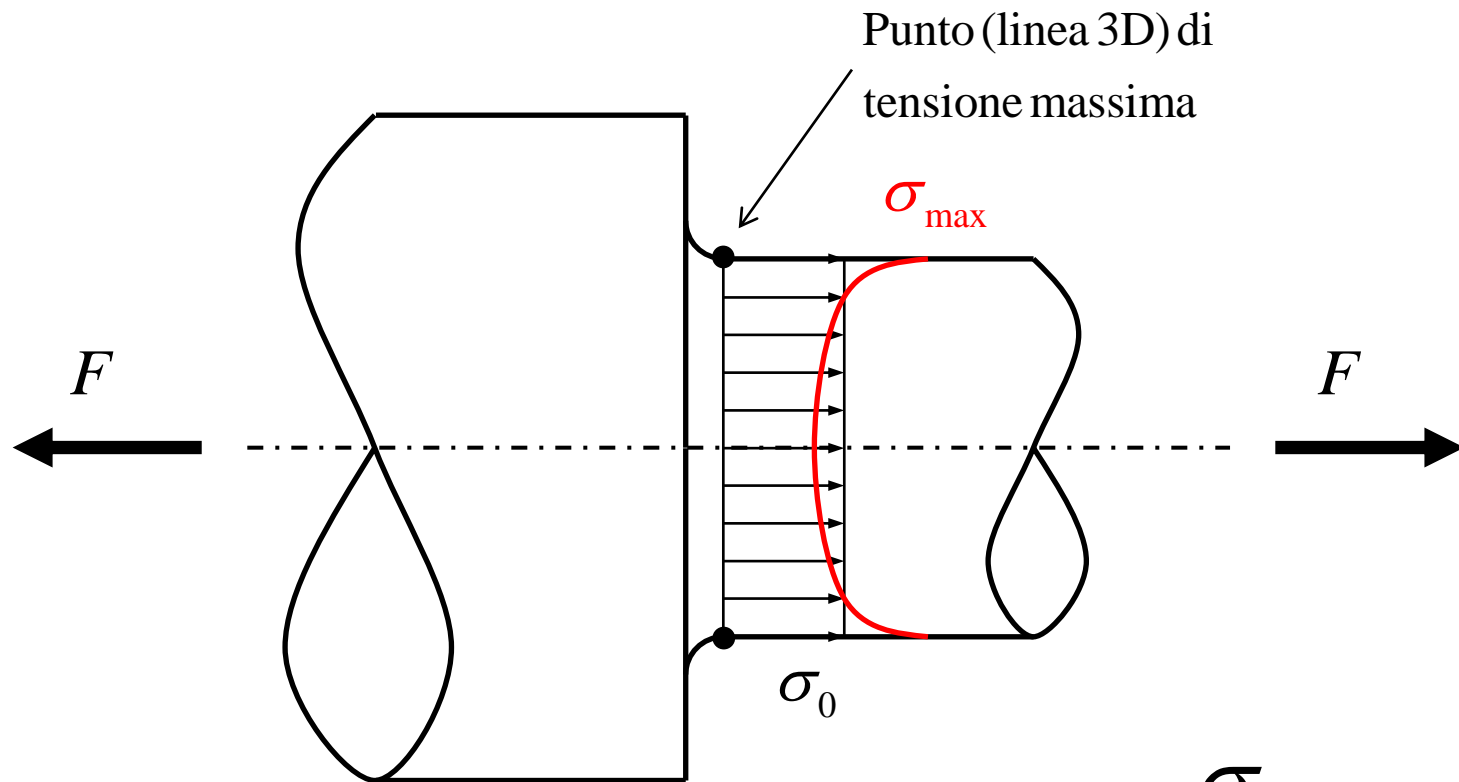


$$D = 150 \text{ mm}$$

$$d = 100 \text{ mm}$$

$$r = 10 \text{ mm}$$

Albero con variazione di diametro e raggio di raccordo



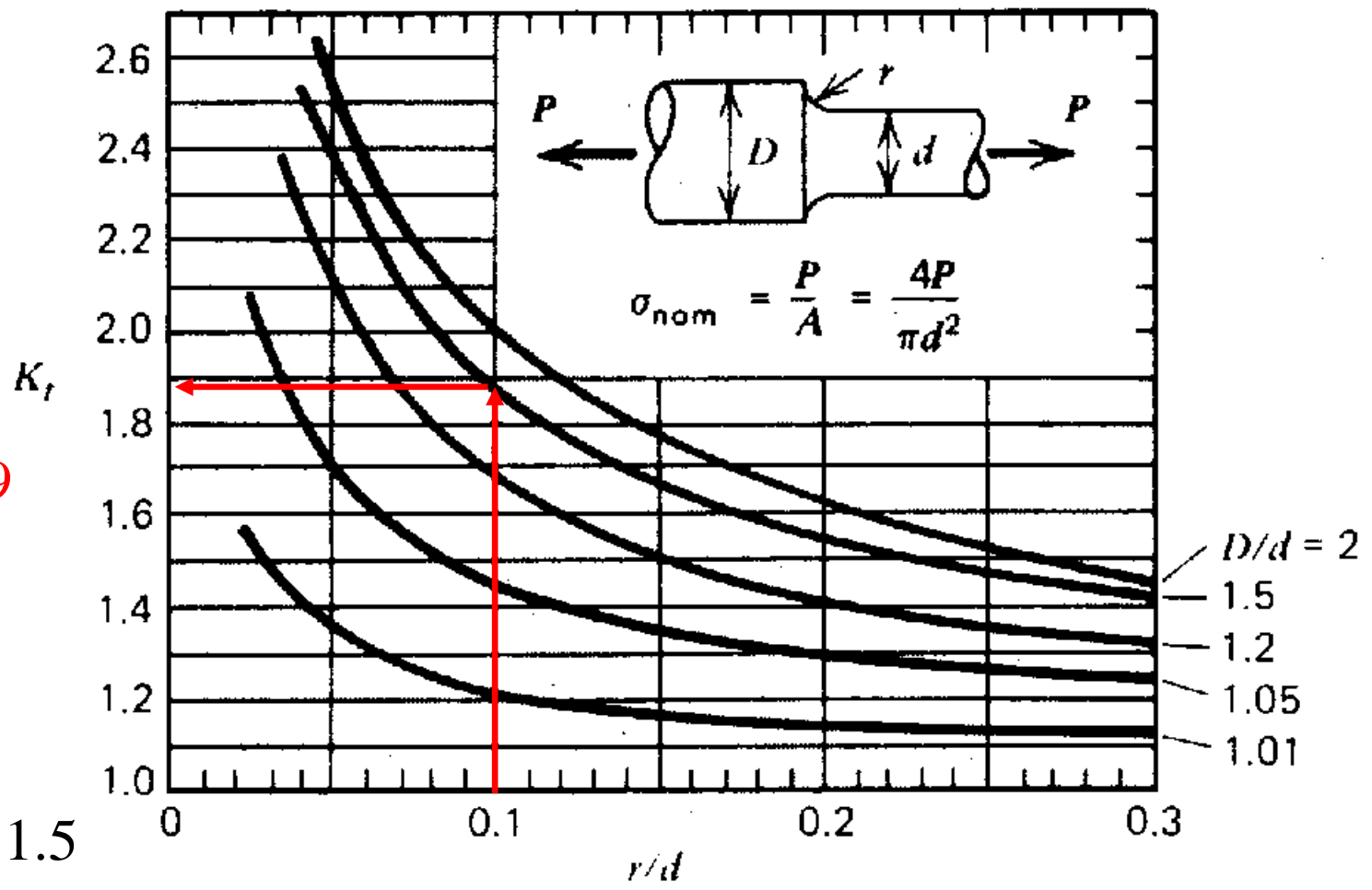
$$K_t = \frac{\sigma_{\max}}{\sigma_0}$$

Analisi lineare:

K_t non dipende dal valore della tensione nominale

Albero con variazione di diametro e raggio di raccordo

R.C. Juvinall, K.M. Marshek – Fundamentals of machine component design – Wiley

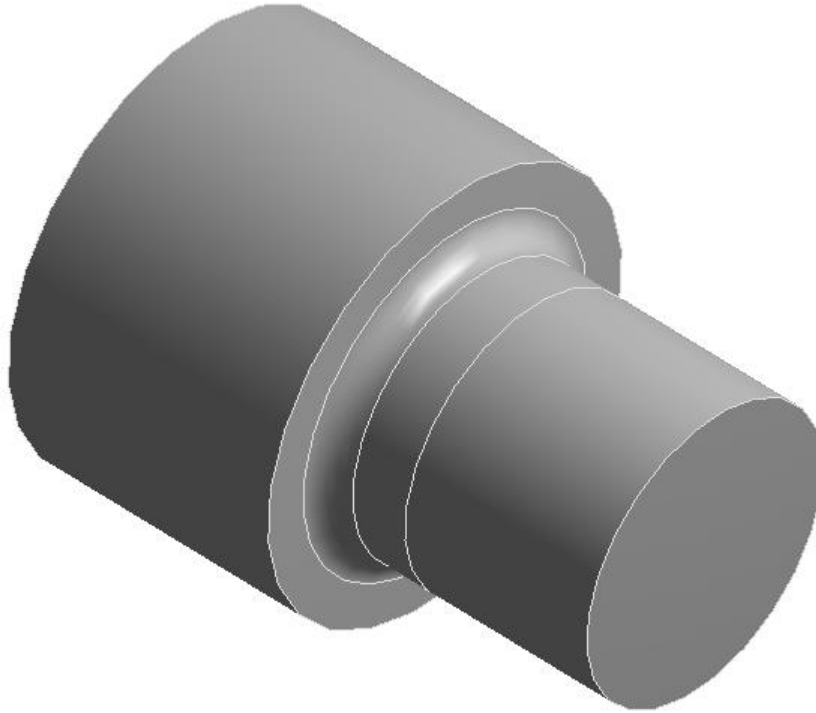


$K_t \approx 1.9$

$D/d = 1.5$

$r/d = 0.1$

Albero con variazione di diametro e raggio di raccordo



Formati di importazione:

Catia

Pro/Engineer

Solidworks

Unigraphics NX

...

Parasolid (.x_t, .x_b)

IGES (.igs, .iges)

STEP (.stp, .step)

...

Geometria da CAD: Albero.x_t (parasolid) →
Importazione su Ansys Wb

Albero con variazione di diametro e raggio di raccordo

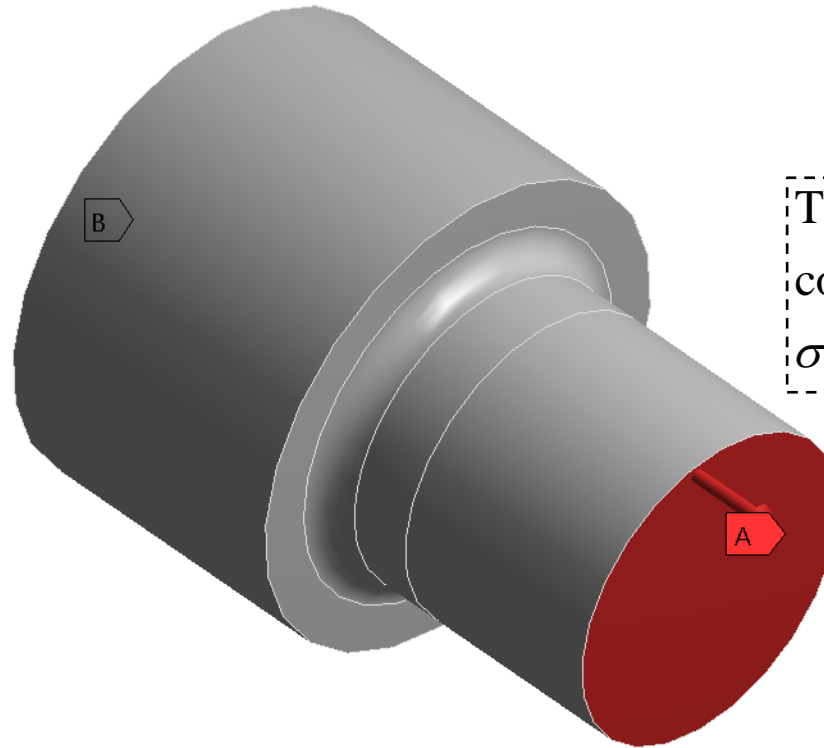
A: Static Structural

Static Structural

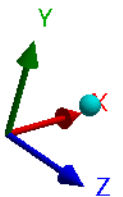
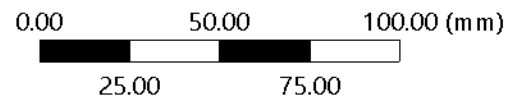
Time: 1. s

A Pressure: -100. MPa

B Fixed Support

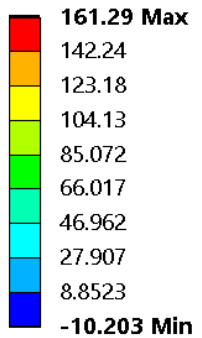


Tensione nominale introdotta
come pressione remota:
 $\sigma_0 = 100 \text{ MPa}$

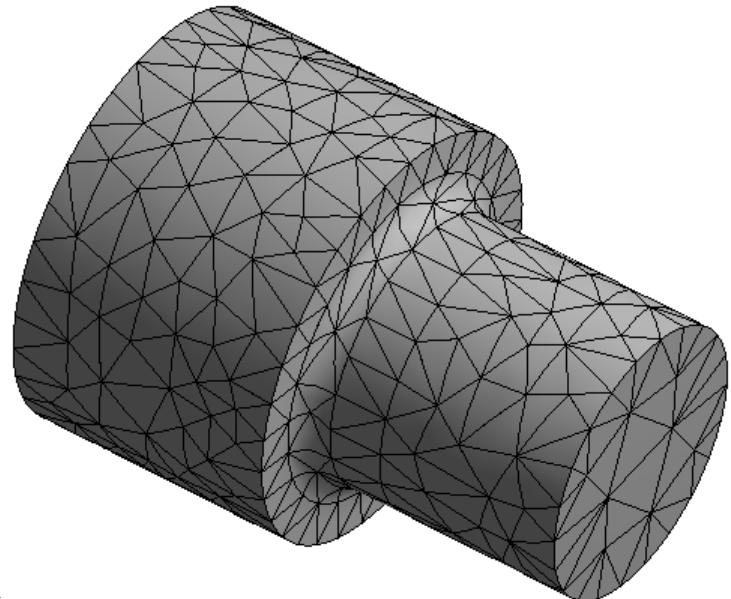
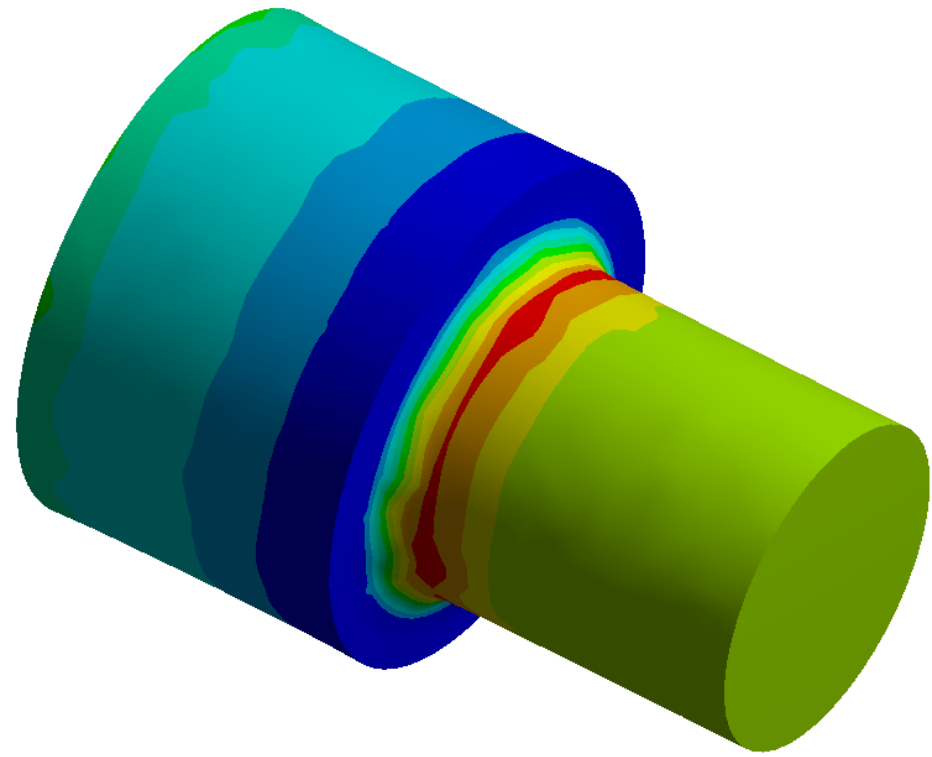


Albero con variazione di diametro e raggio di raccordo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



Mesh 1
(default)



$$\sigma_{\max} = 161 \text{ MPa}, \quad \sigma_0 = 100 \text{ MPa}$$

$$K_t(\text{ANSYS}) = \frac{\sigma_{\max}}{\sigma_0} = 1.61 < K_t(\text{JUV})$$

Albero con variazione di diametro e raggio di raccordo

A: Static Structural

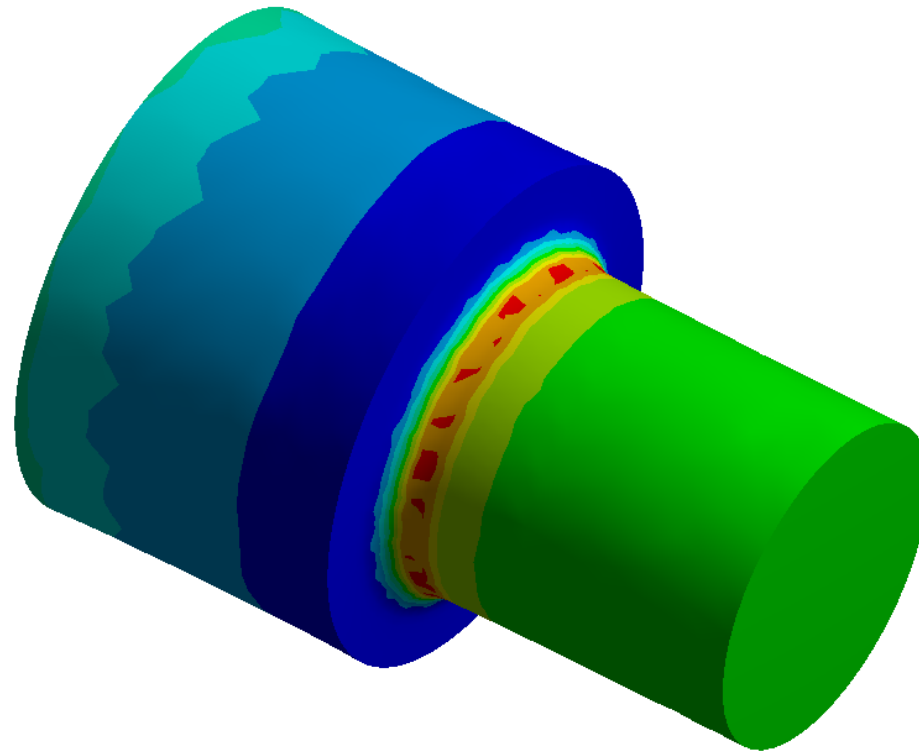
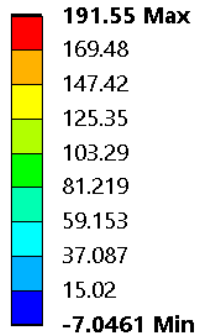
Normal Stress

Type: Normal Stress(Z Axis)

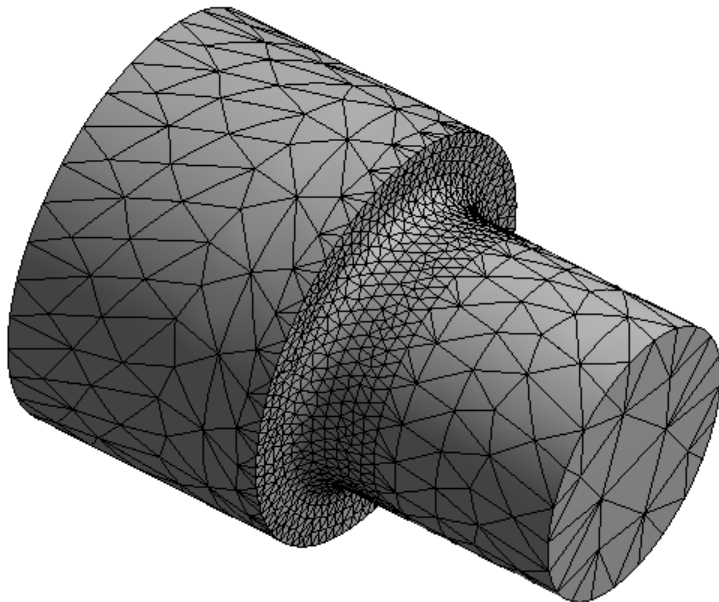
Unit: MPa

Global Coordinate System

Time: 1



Mesh 2

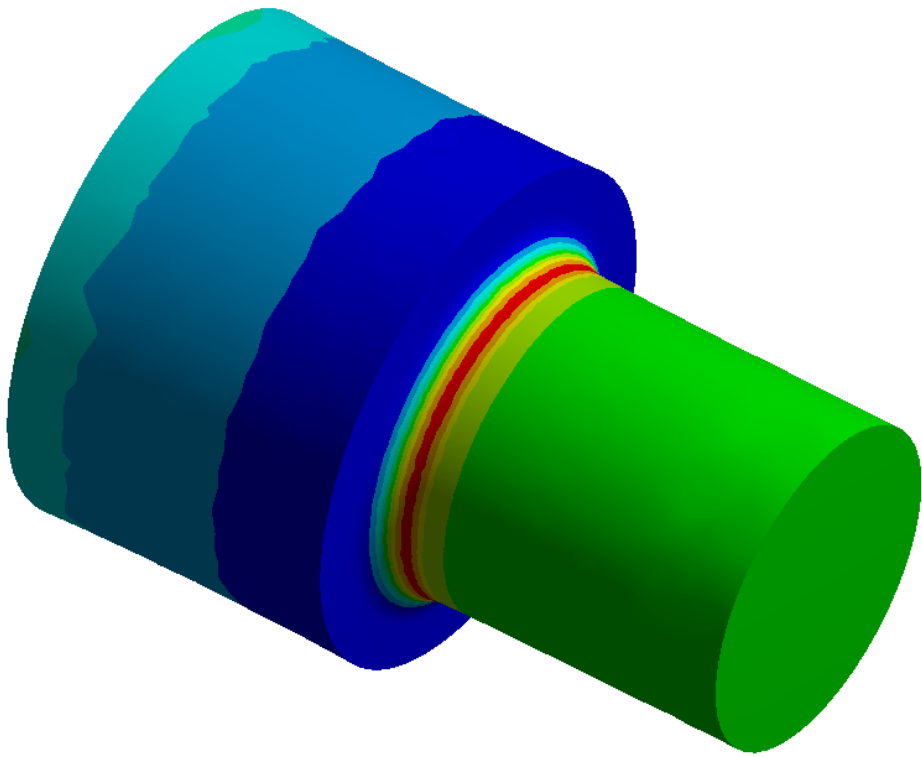
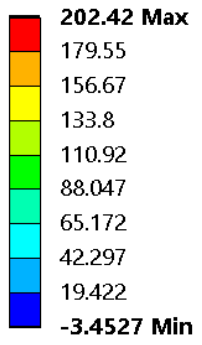


$$\sigma_{\max} = 192 \text{ MPa}, \quad \sigma_0 = 100 \text{ MPa}$$

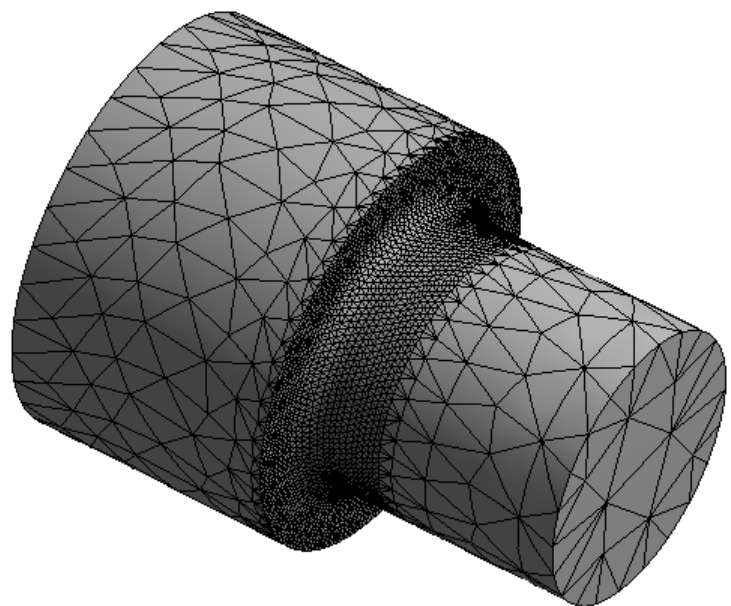
$$K_t(\text{ANSYS}) = \frac{\sigma_{\max}}{\sigma_0} = 1.92 = K_t(\text{JUV})$$

Albero con variazione di diametro e raggio di raccordo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



Mesh 3

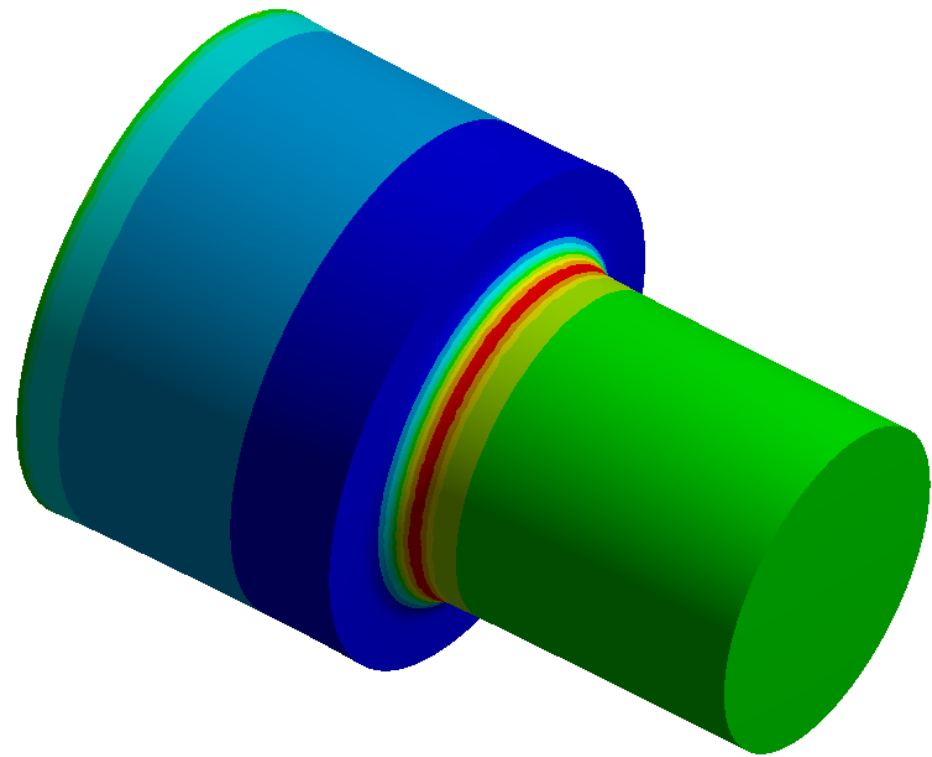
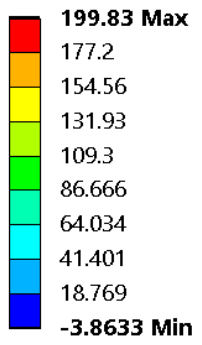


$$\sigma_{\max} = 202 \text{ MPa}, \quad \sigma_0 = 100 \text{ MPa}$$

$$K_t(\text{ANSYS}) = \frac{\sigma_{\max}}{\sigma_0} = 2.02 > (\approx) K_t(\text{JUV})$$

Albero con variazione di diametro e raggio di raccordo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



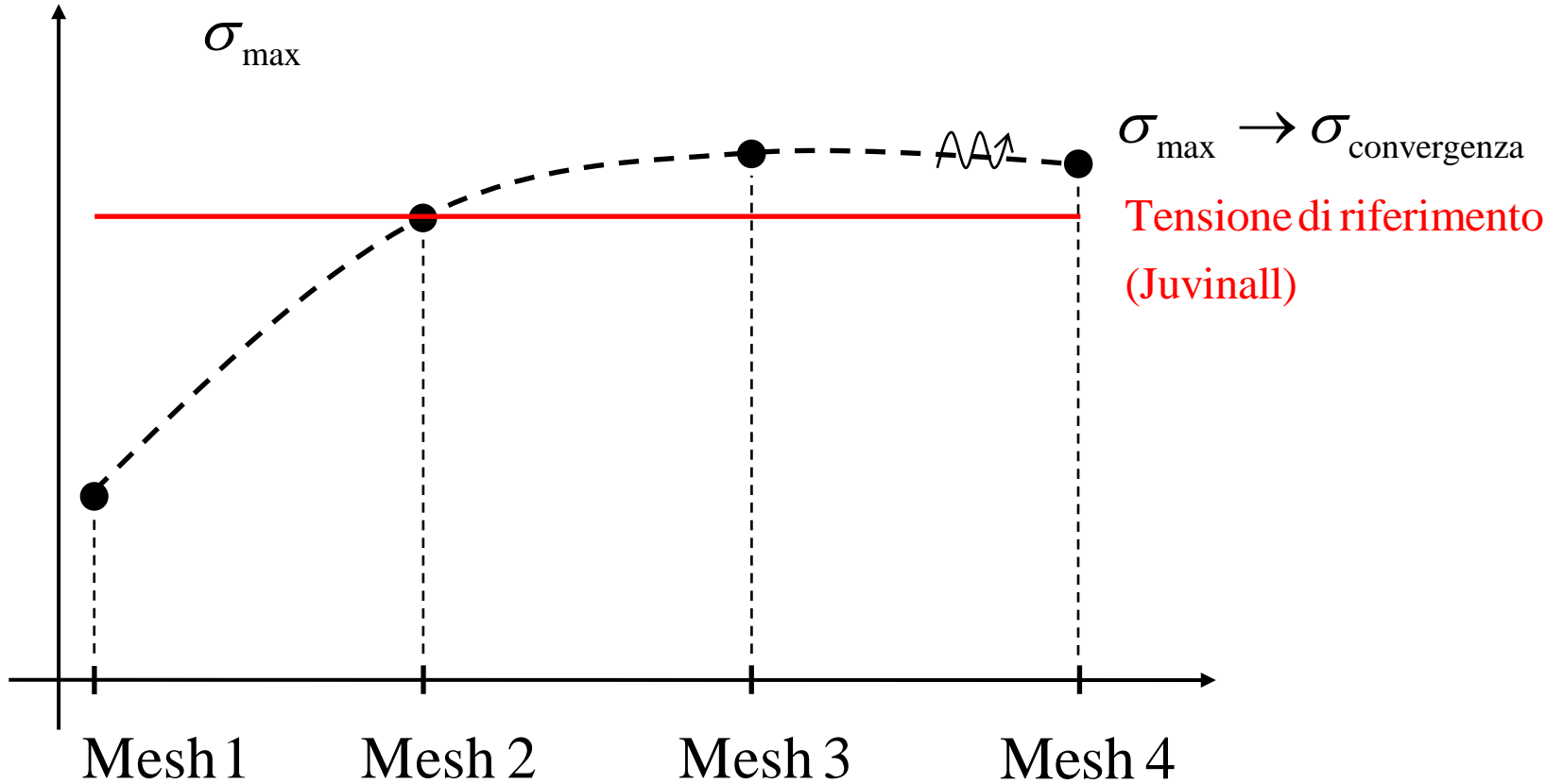
Mesh 4



$$\sigma_{\max} = 200 \text{ MPa}, \quad \sigma_0 = 100 \text{ MPa}$$

$$K_t(\text{ANSYS}) = \frac{\sigma_{\max}}{\sigma_0} = 2.00 > (\approx) K_t(\text{JUV})$$

Analisi di convergenza



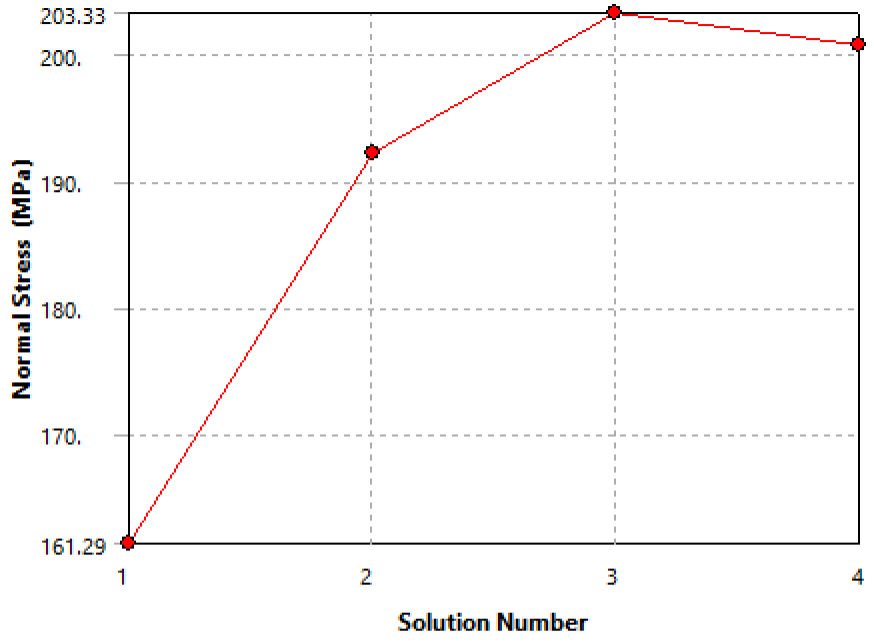
Analisi di convergenza (*h*-convergence)

Details of "Solution (A6)"

Adaptive Mesh Refinement	
Max Refinement Loops	5.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	21. s
MAPDL Memory Used	1.0596 GB
MAPDL Result File Size	95.875 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Details of "Convergence"

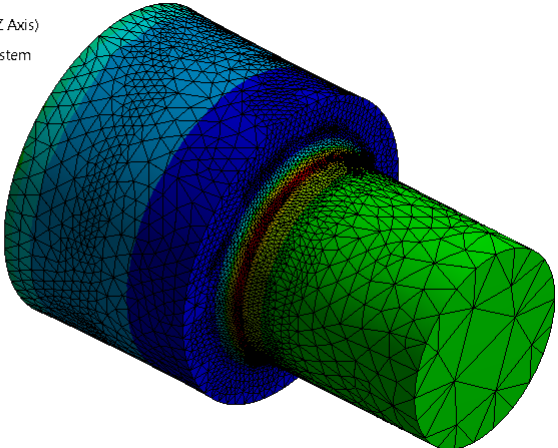
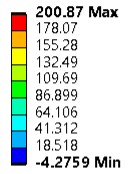
Definition	
Type	Maximum
Allowable Change	2. %
Results	
Last Change	-1.2179 %
Converged	Yes



	Normal Stress (MPa)	Change (%)	Nodes	Elements
1	161.29		3176	1695
2	192.15	17.461	19201	12230
3	203.33	5.6541	71786	49675
4	200.87	-1.2179	222780	157747

Convergenza Ok

A: Static Structural
 Normal Stress
 Type: Normal Stress(Z Axis)
 Unit: MPa
 Global Coordinate System
 Time: 1

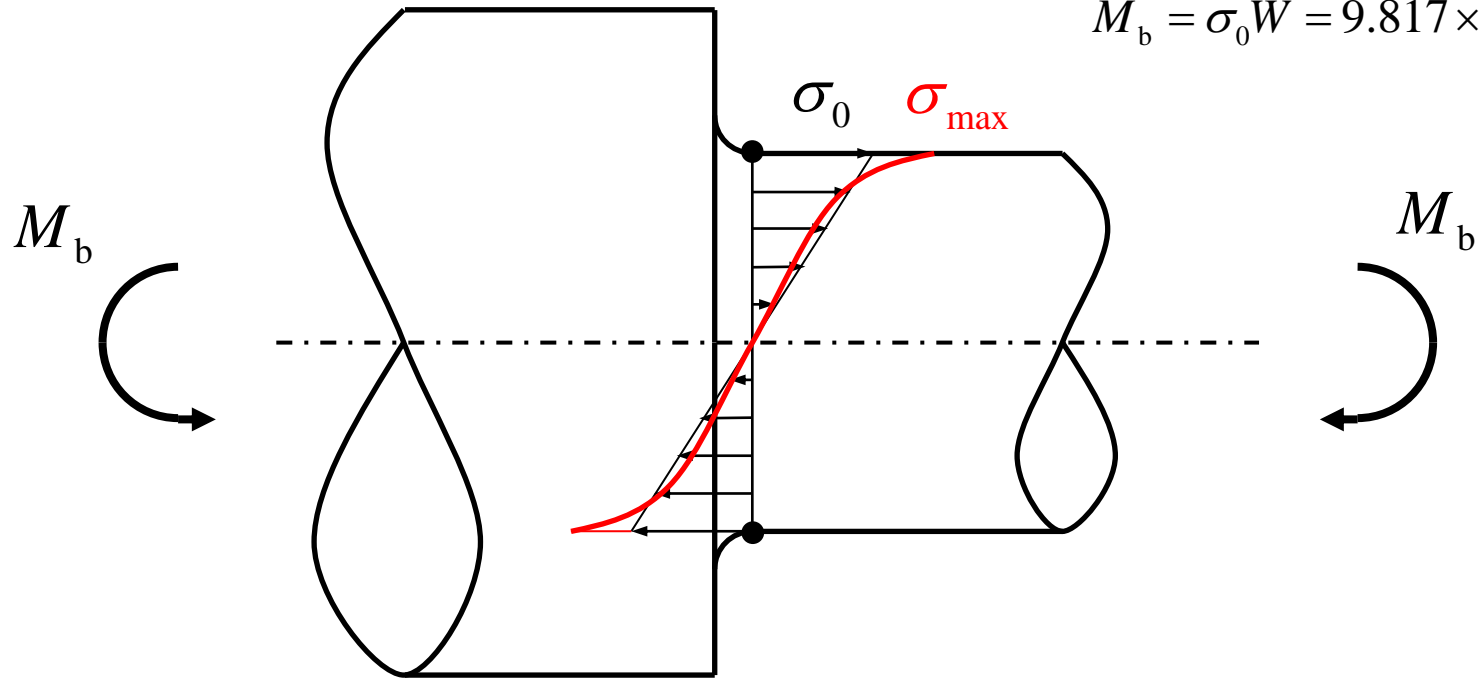


Esercizio da svolgere

$$\sigma_0 = 100 \text{ MPa}$$

$$W = \frac{\pi}{32} d^3 = 9.817 \times 10^4 \text{ mm}^3$$

$$M_b = \sigma_0 W = 9.817 \times 10^6 \text{ Nmm}$$



$$D = 150 \text{ mm}$$

$$d = 100 \text{ mm}$$

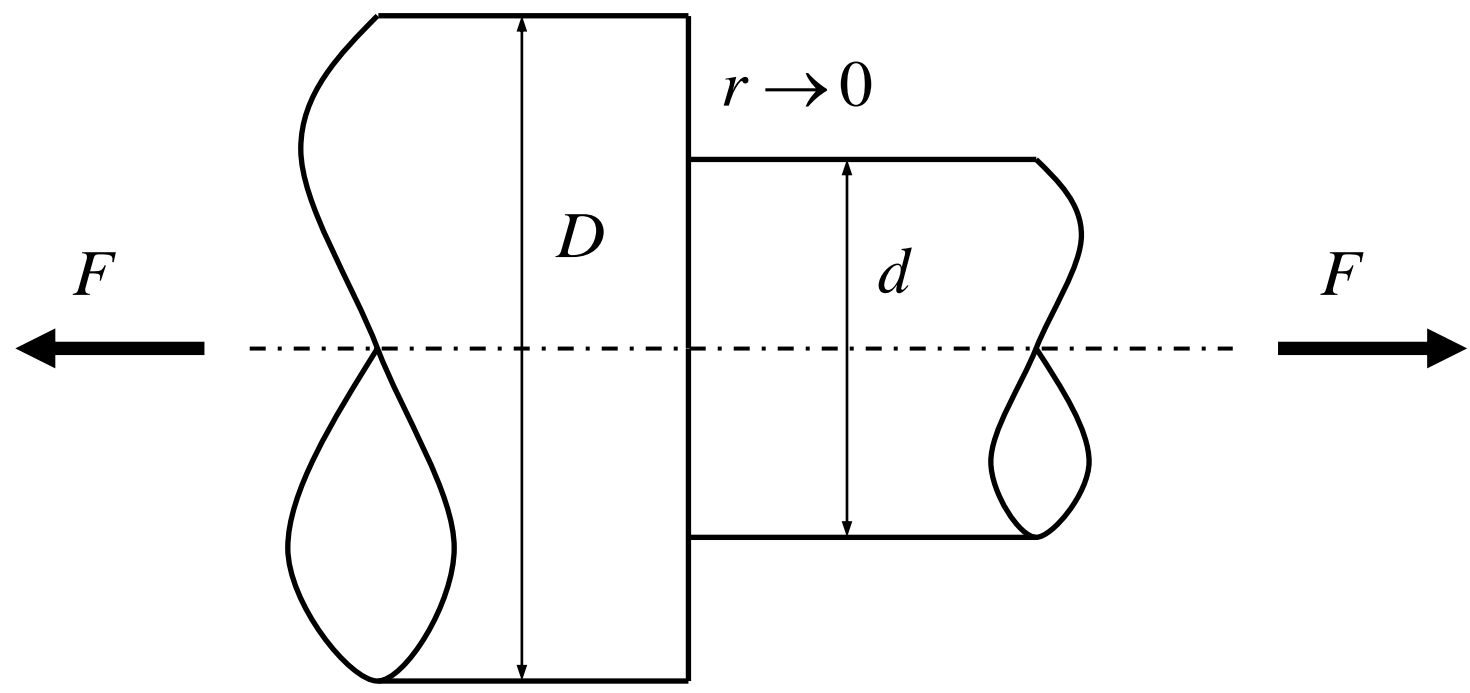
$$r = 10 \text{ mm}$$

$$K_t = \frac{\sigma_{\max}}{\sigma_0} = ?$$

Albero con variazione di diametro, raggio di raccordo nullo

$$K_t = \infty$$

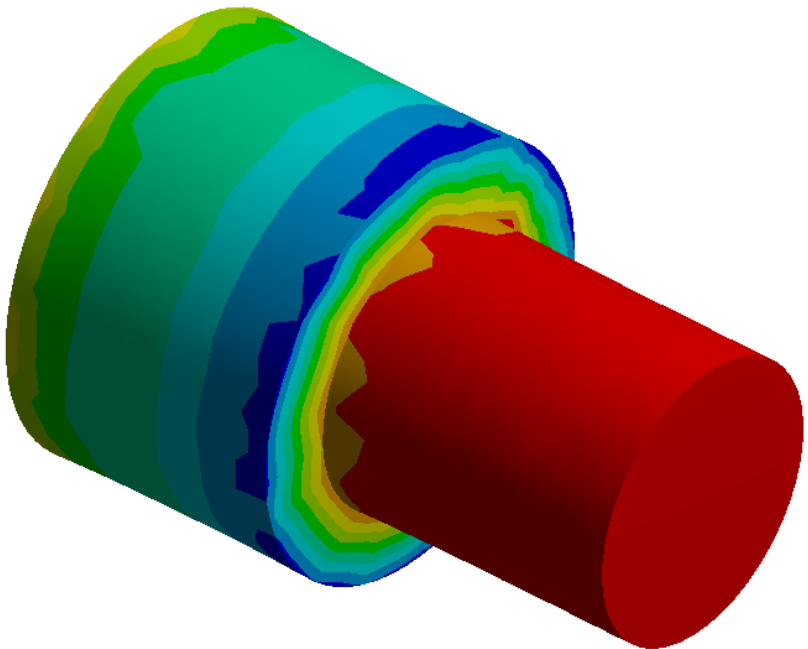
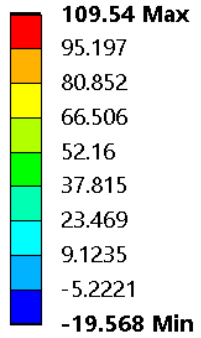
$$K_t(\text{ANSYS}) = ?$$



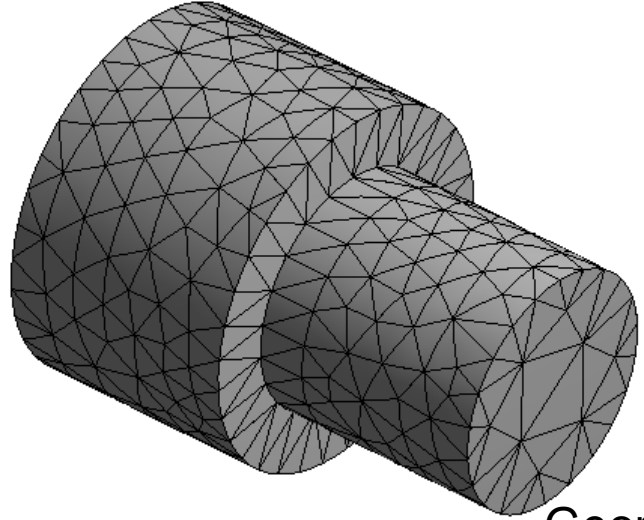
σ_{\max} valore finito, funzione crescente dell'infittimento locale della mesh

Albero con variazione di diametro, raggio di raccordo nullo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



Mesh 1

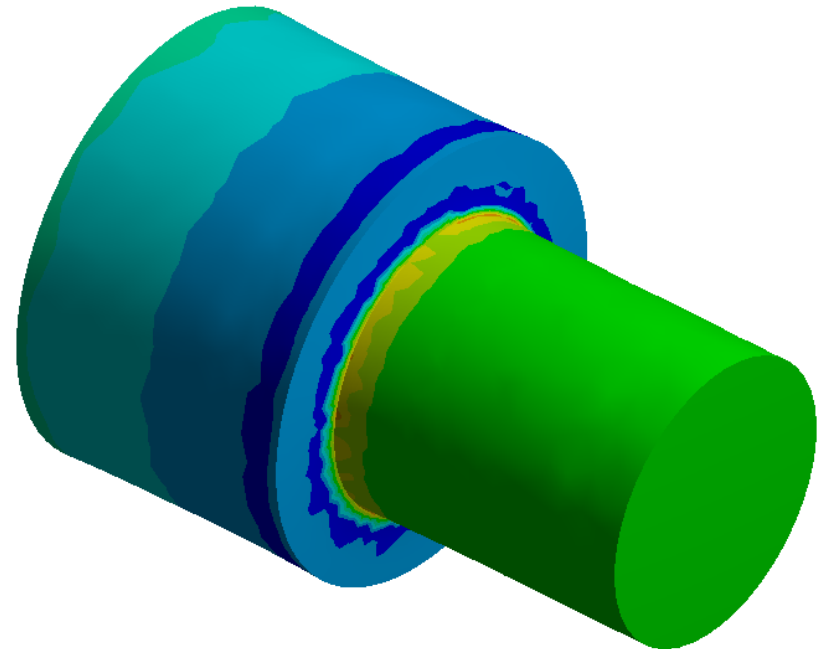
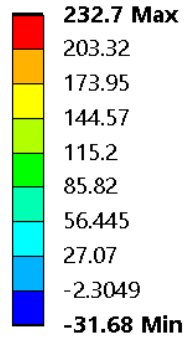


$$\sigma_{\max} = 110 \text{ MPa}$$

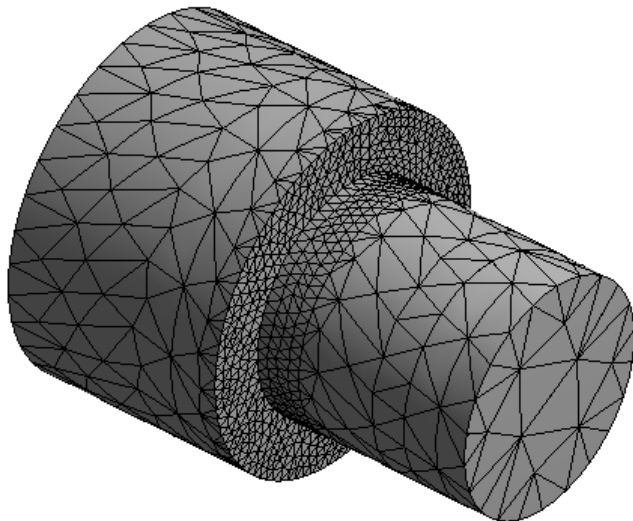
Geometria da CAD: Albero_NoRacc.x_t

Albero con variazione di diametro, raggio di raccordo nullo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



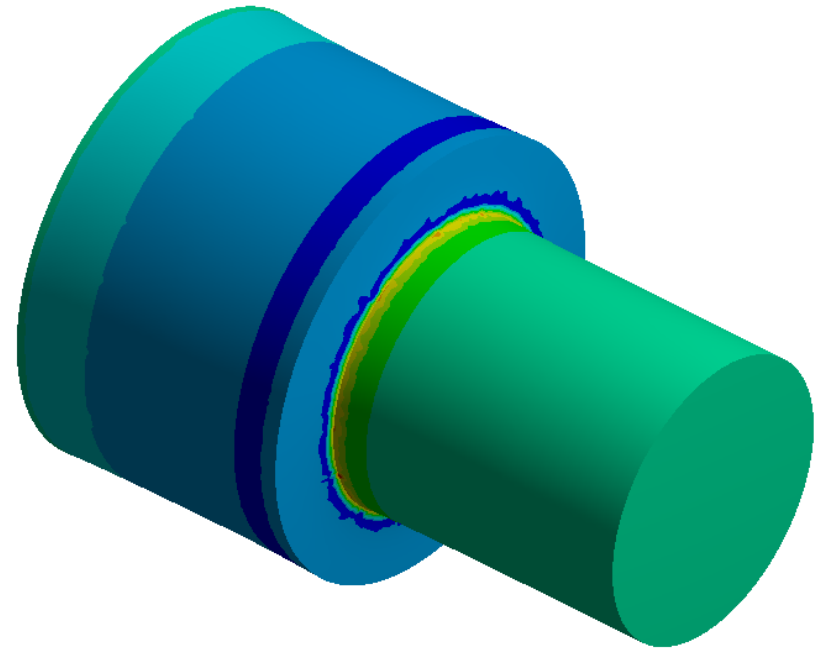
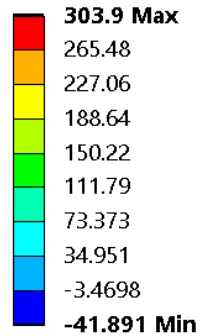
Mesh 2



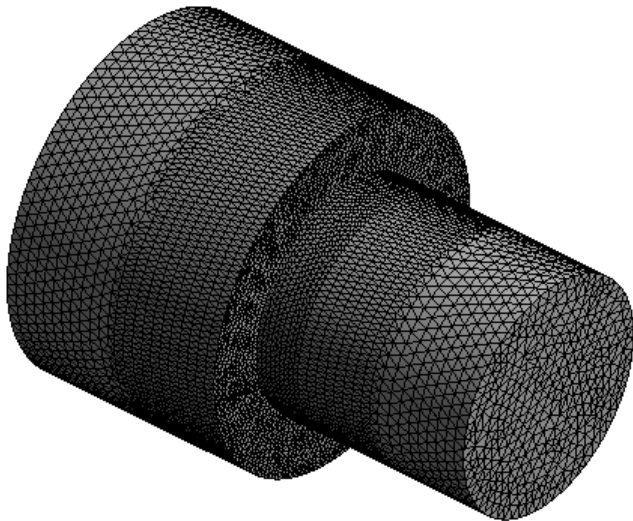
$$\sigma_{\max} = 233 \text{ MPa}$$

Albero con variazione di diametro, raggio di raccordo nullo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1



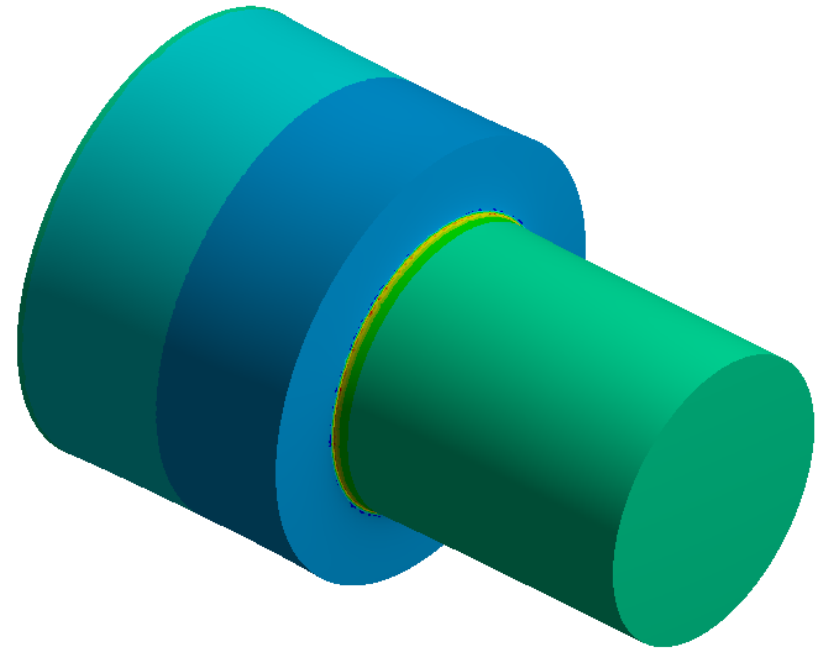
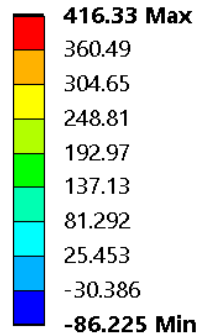
Mesh 3



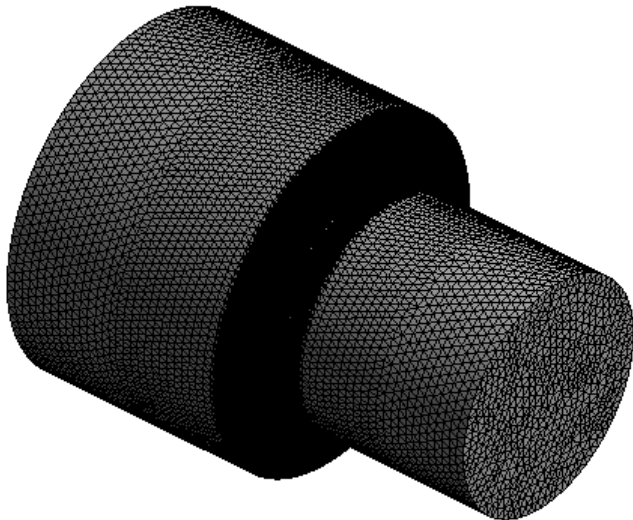
$$\sigma_{\max} = 304 \text{ MPa}$$

Albero con variazione di diametro, raggio di raccordo nullo

A: Static Structural
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1

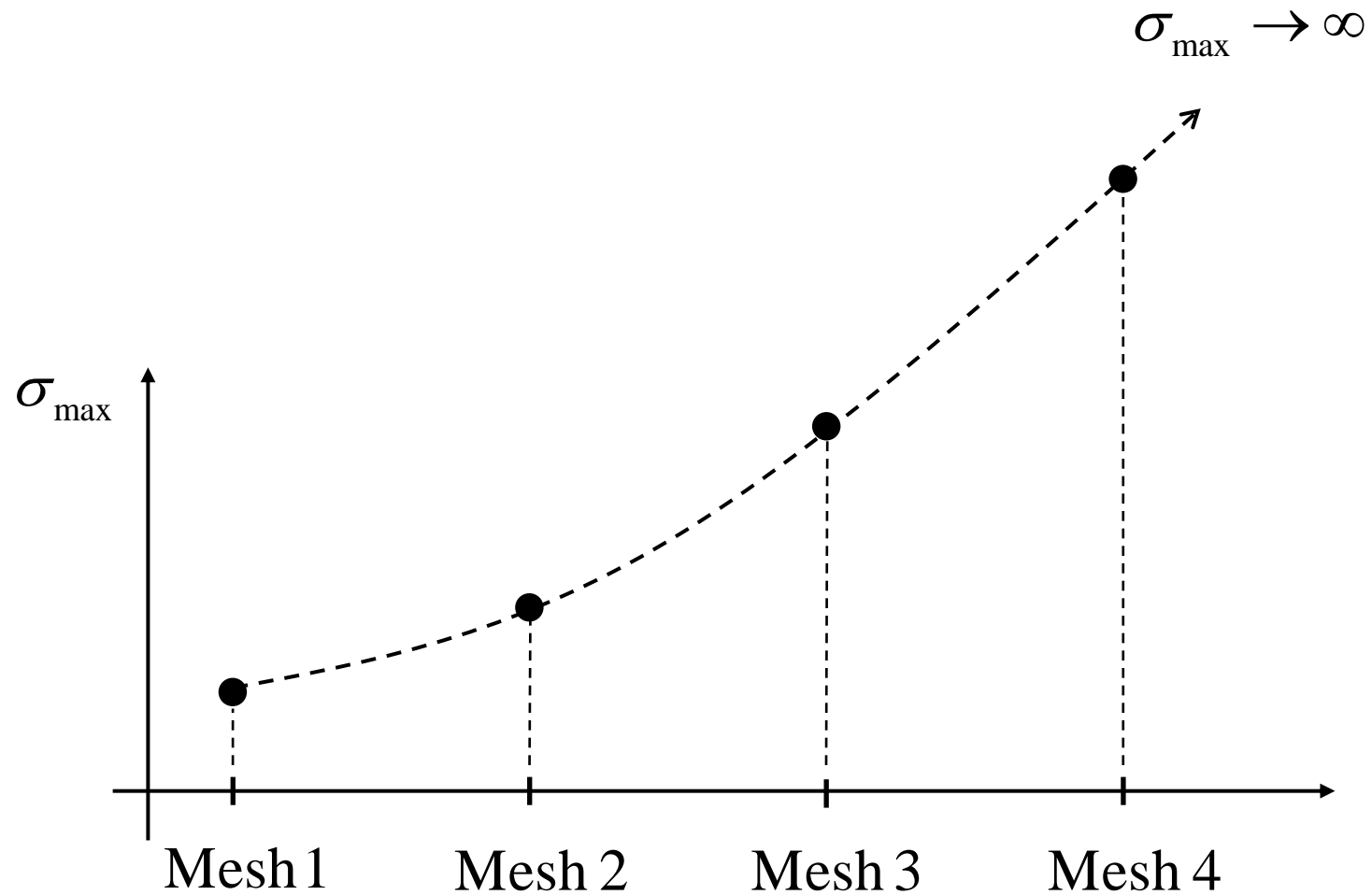


Mesh 4



$$\sigma_{\max} = 416 \text{ MPa}$$

Raggio di raccordo nullo, NO convergenza:
Tensione sempre più alta all'aumentare dell'infittimento della mesh

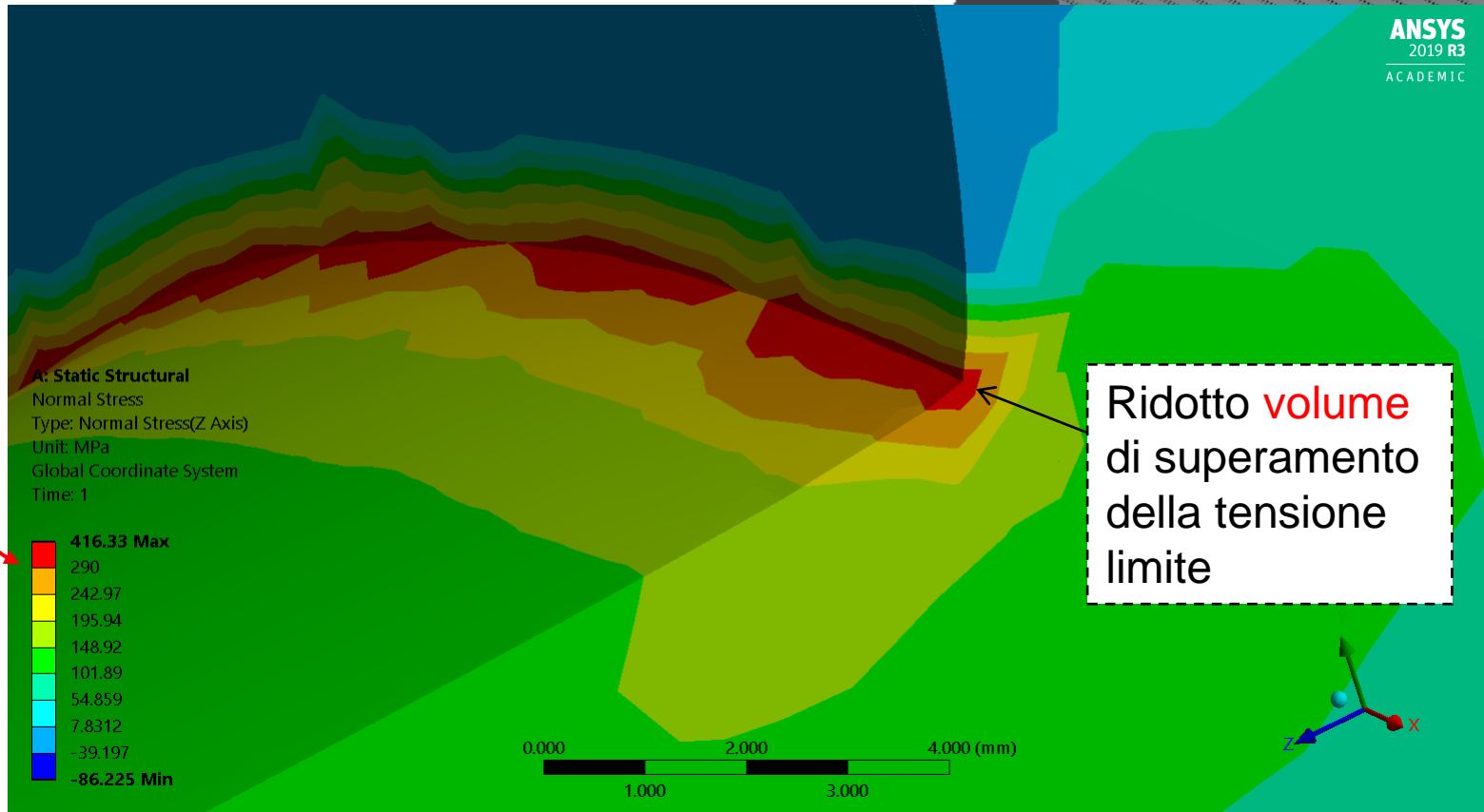
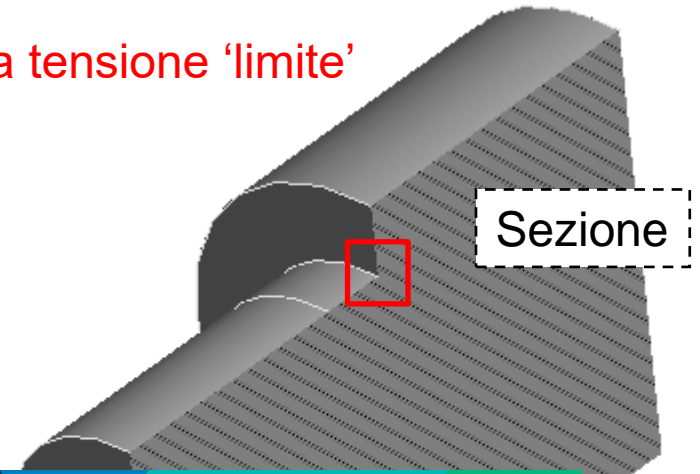


Criterio di resistenza?

Dimensione del volume di materiale che ha superato la tensione 'limite'

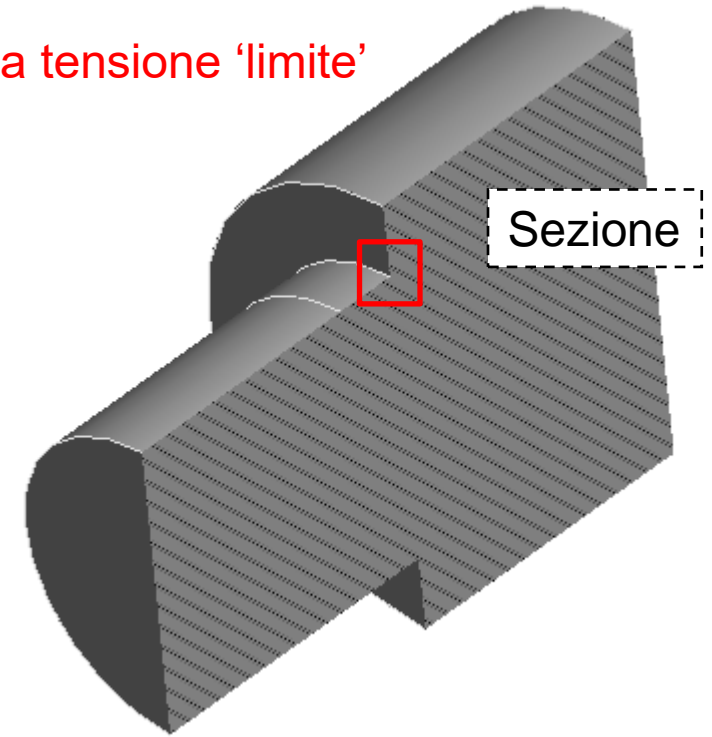
Tensione equivalente
von Mises

Es. $S_Y = 290 \text{ MPa}$

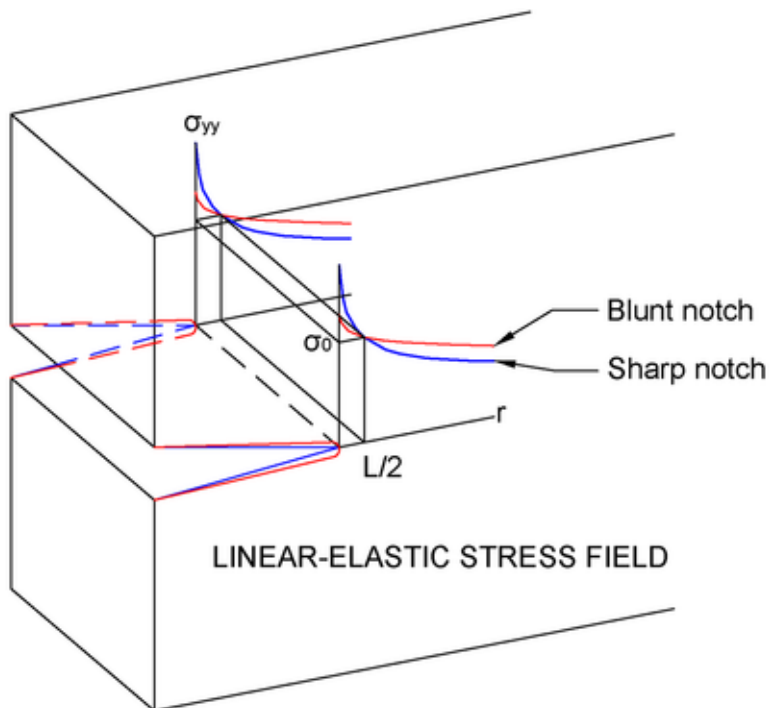


Criterio di resistenza?

Dimensione del volume di materiale che ha superato la tensione 'limite'

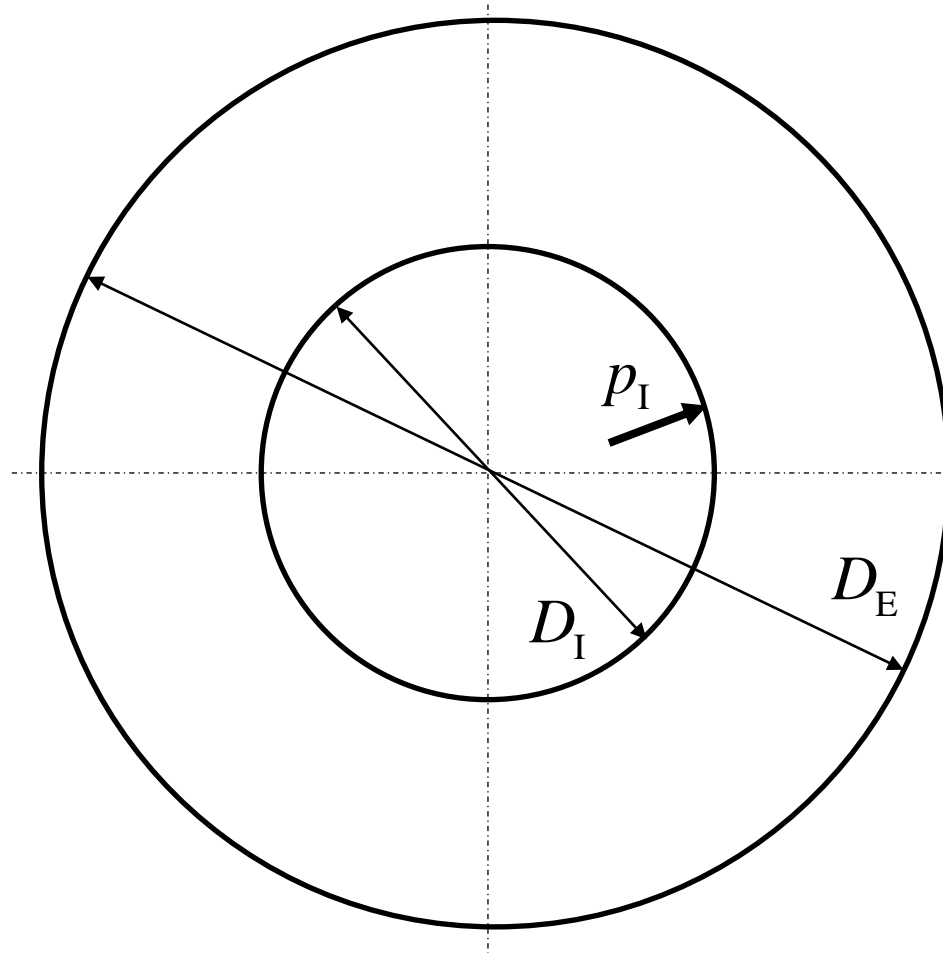


Teoria della distanza critica (Theory of Critical Distance, TDC)



Invece di analizzare la tensione nel punto di fondo intaglio, secondo la TCD, si valuta la tensione all'interno, in corrispondenza di un **punto** o mediando la tensione **lungo una linea**. La tensione ottenuta è un buon indicatore della frattura, anche nel caso di tensione a forte gradiente o addirittura singolare (raggio nullo)

Cilindro ad elevato spessore radiale

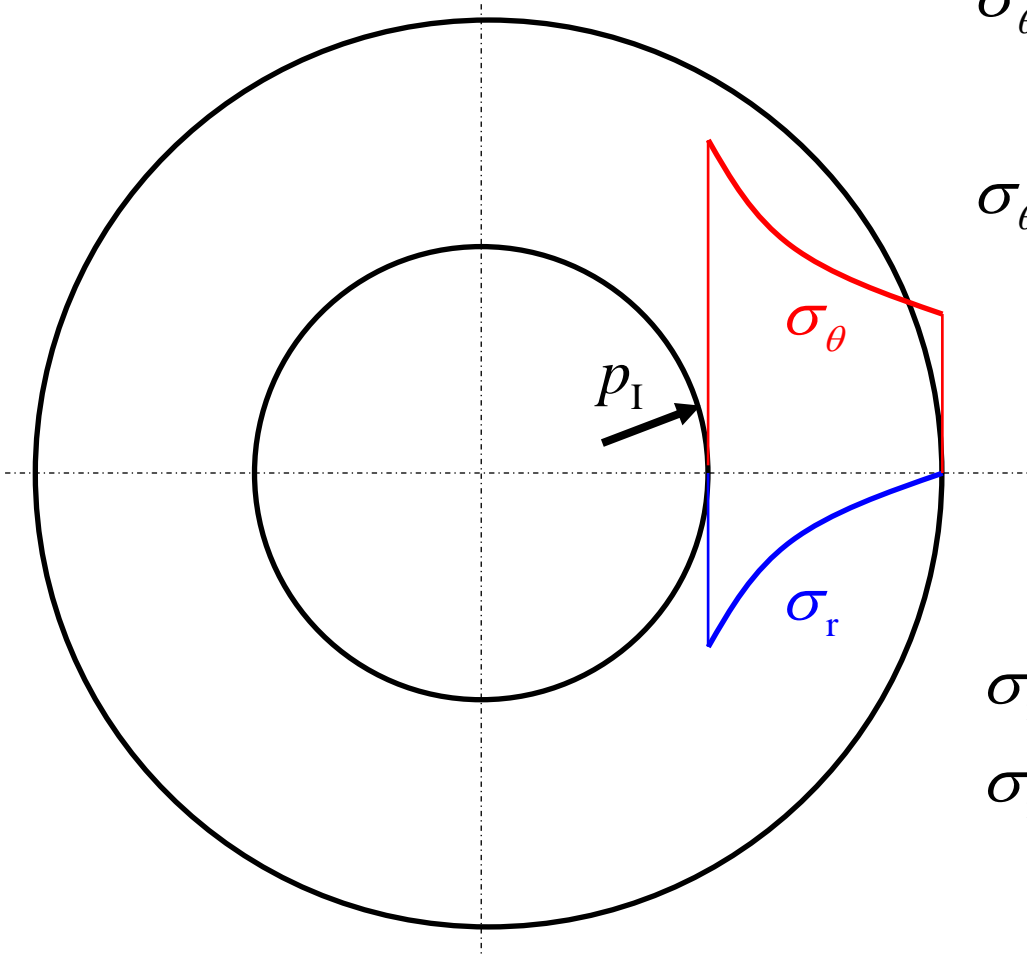


$$D_E = 1000 \text{ mm}$$

$$D_I = 400 \text{ mm}$$

$$p_I = 100 \text{ bar} = 10 \text{ MPa}$$

Cilindro ad elevato spessore radiale – pressione interna



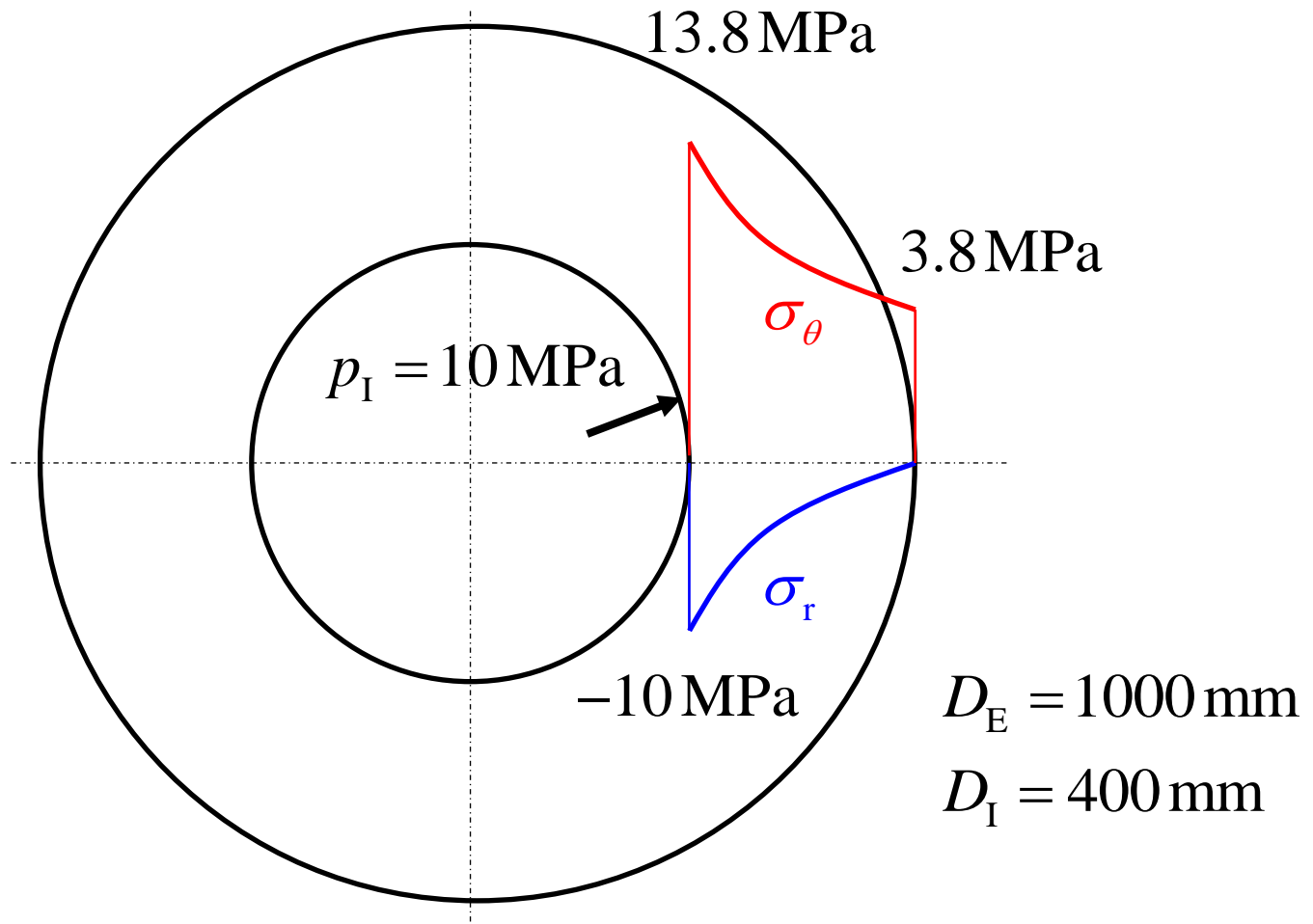
$$\sigma_{\theta}(r = D_I / 2) = p_I \frac{D_E^2 + D_I^2}{D_E^2 - D_I^2}$$

$$\sigma_{\theta}(r = D_E / 2) = 2p_I \frac{D_I^2}{D_E^2 - D_I^2}$$

$$\sigma_r(r = D_I / 2) = -p_I$$

$$\sigma_r(r = D_E / 2) = 0$$

Cilindro ad elevato spessore radiale – pressione interna



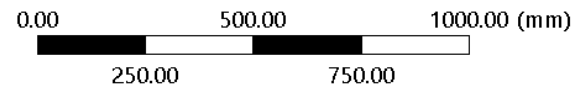
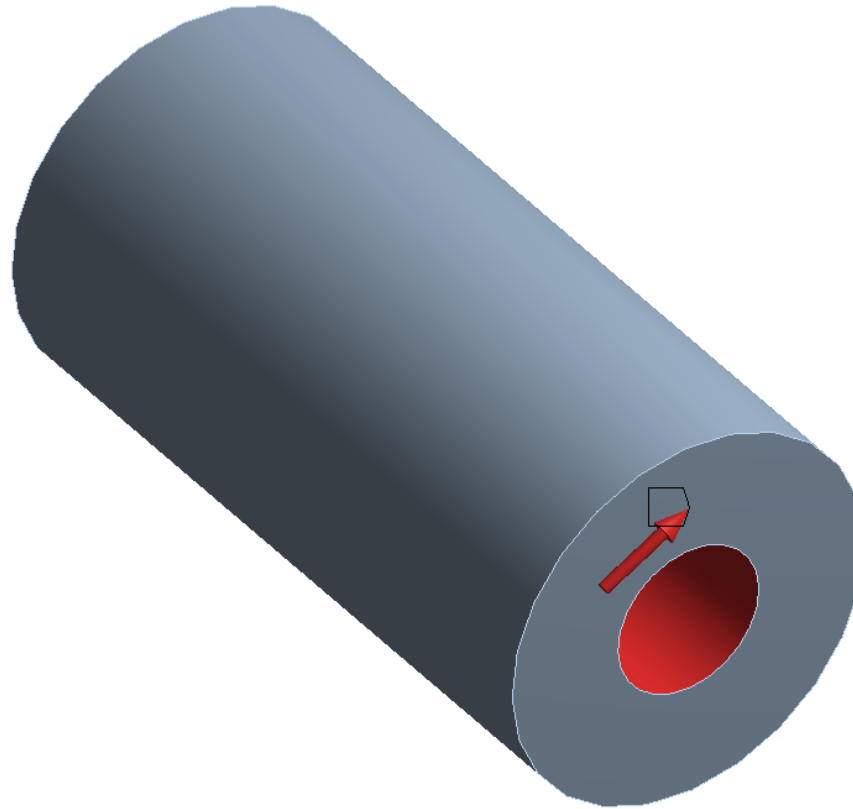
Cilindro ad elevato spessore radiale – pressione interna

A: Static Structural (ANSYS)

Static Structural

Time: 1. s

 Pressure: 10. MPa



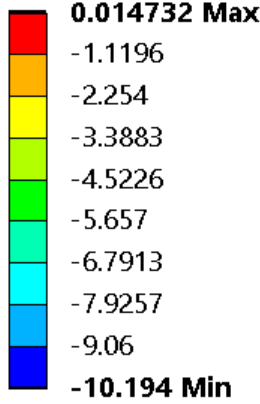
TubeSpesso_ElevataLunghezza.x_t

Cilindro ad elevato spessore radiale – pressione interna

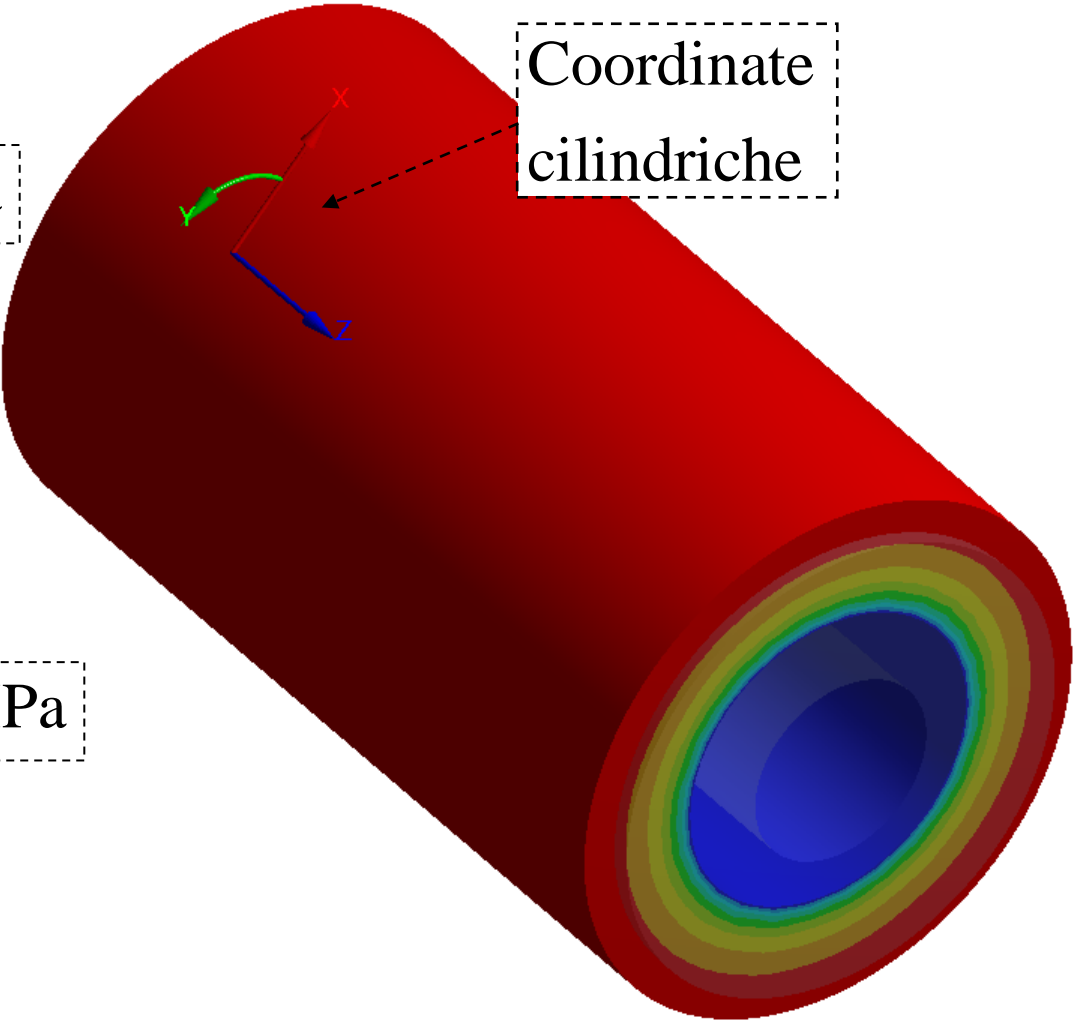
Tensione normale radiale

A: Static Structural (ANSYS)
Normal Stress - r
Type: Normal Stress(X Axis)
Unit: MPa

$$\sigma_r (R. esterno) = 0 \text{ MPa}$$



$$\sigma_r (R. interno) = -10 \text{ MPa}$$

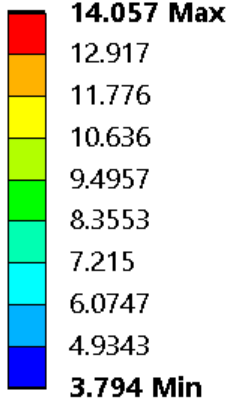


Cilindro ad elevato spessore radiale – pressione interna

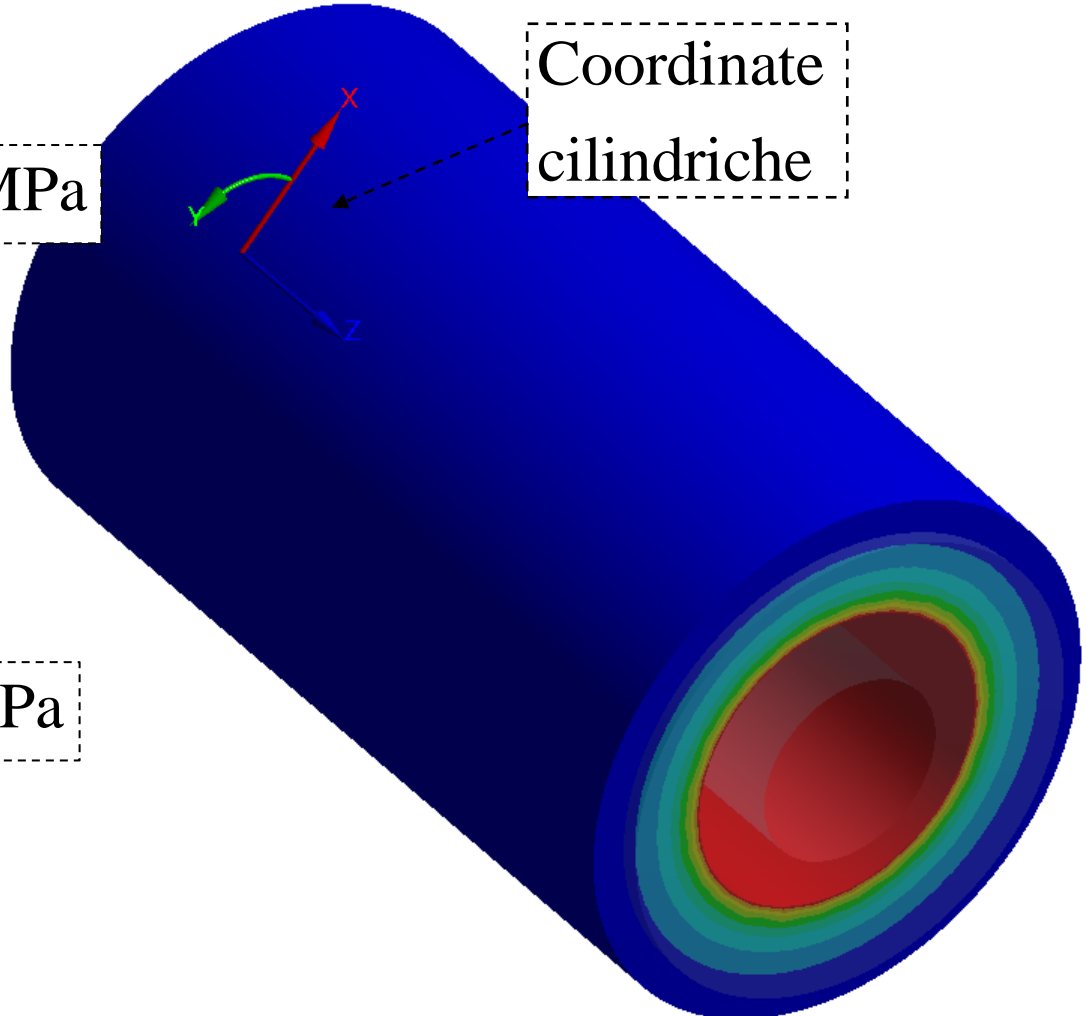
Tensione normale circonferenziale

A: Static Structural (ANSYS)
Normal Stress - t
Type: Normal Stress(Y Axis)
Unit: MPa

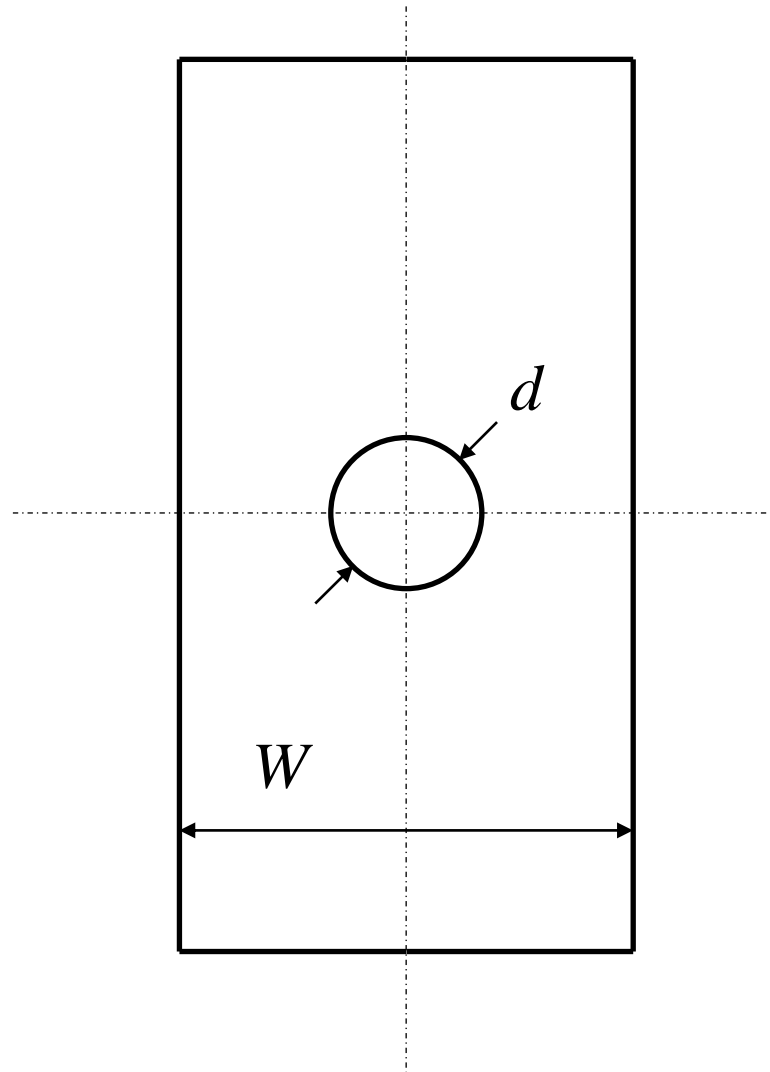
σ_{θ} (R.interno) = 13.8 MPa



σ_{θ} (R.esterno) = 3.8 MPa



Lastra forata di piccolo spessore



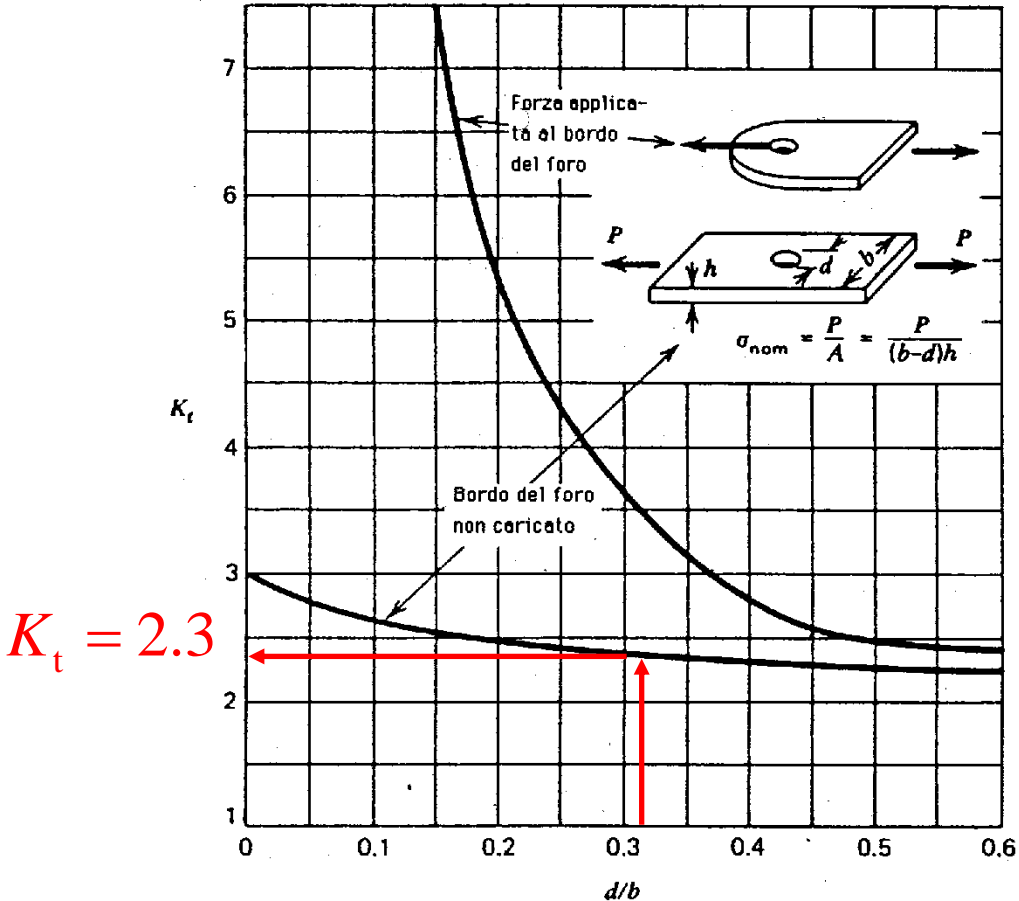
$$K_t = ?$$

$$W = 30 \text{ mm}$$

$$d = 10 \text{ mm}$$

Lastra forata di piccolo spessore

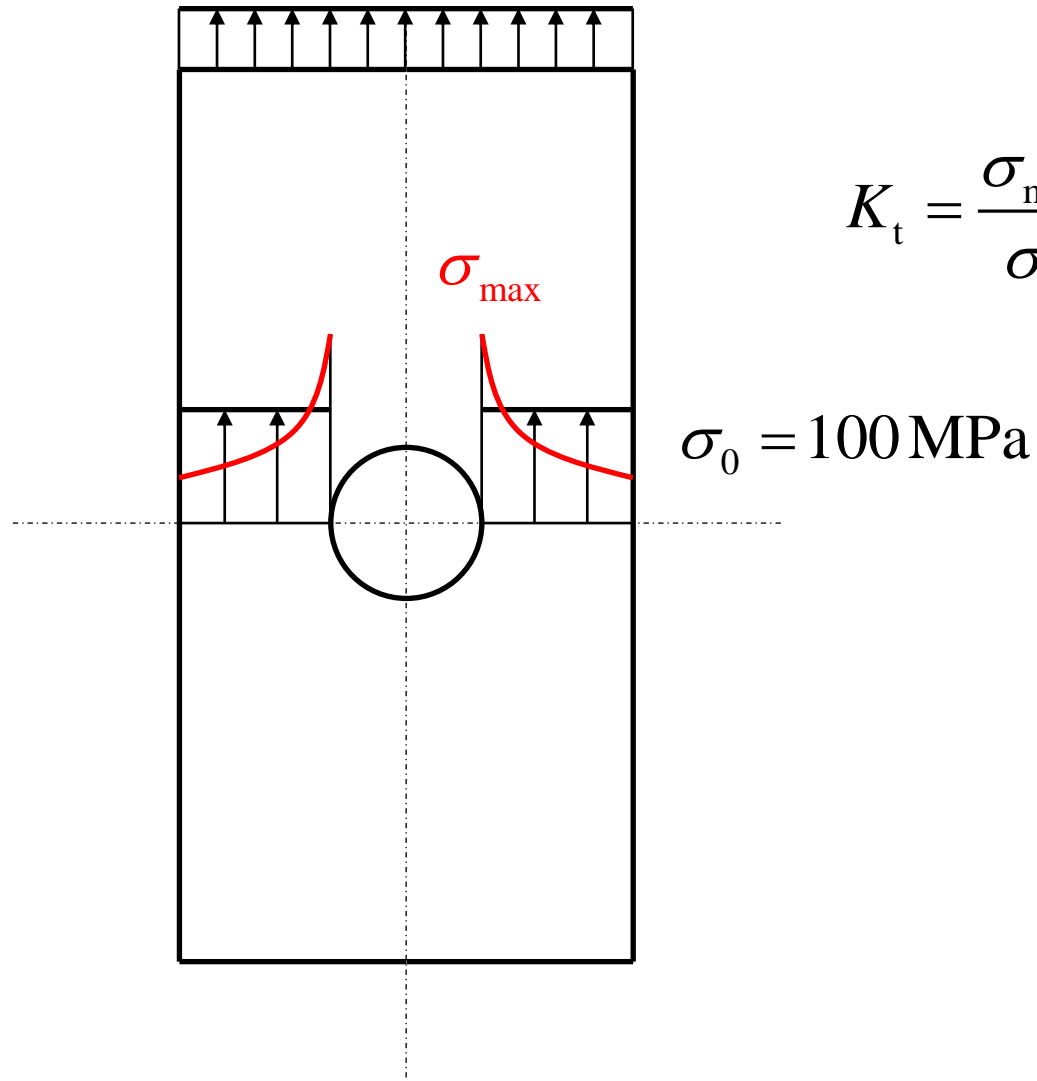
R.C. Juvinall, K.M. Marshek –
Fundamentals of machine
component design – Wiley



$$d / W = 0.333$$

Lastra forata di piccolo spessore

$$\sigma = \sigma_0 \frac{W - d}{W} = 66.7 \text{ MPa}$$



Lastra forata di piccolo spessore

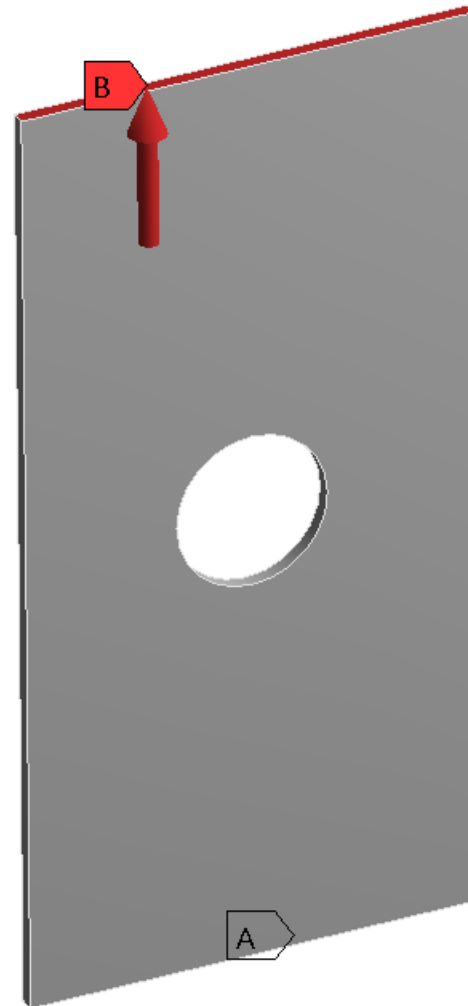
A: Static Structural (ANSYS)

Static Structural

Time: 1. s

A Displacement

B Pressure: -66.7 MPa



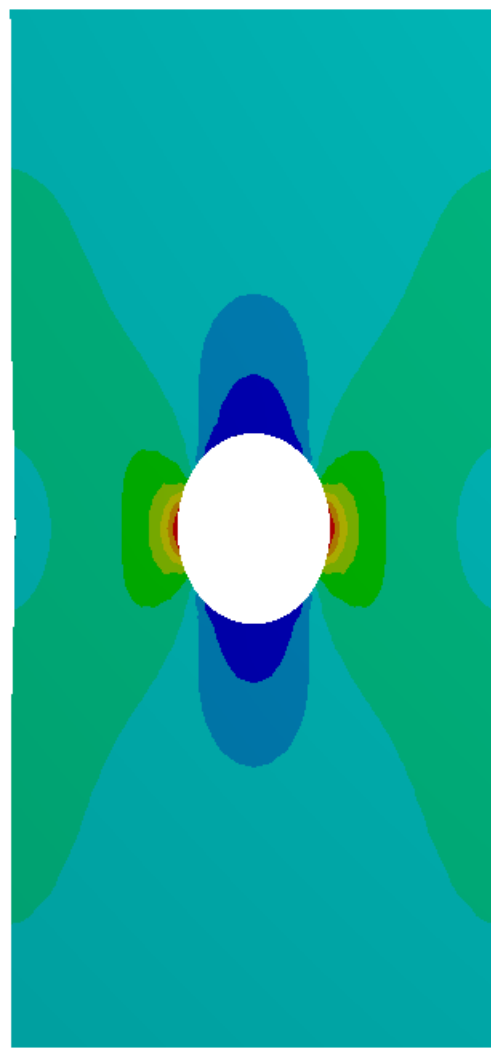
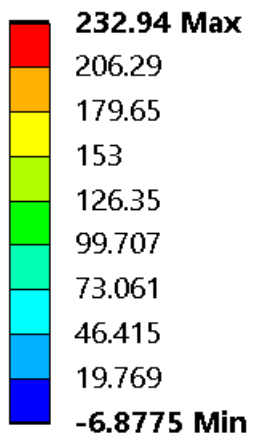
Lastra_Foro_PiccoloSpessore.x_t

Lastra forata di piccolo spessore

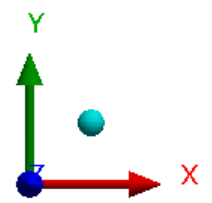
A: Static Structural (ANSYS)

Normal Stress - Y
Type: Normal Stress(Y Axis)
Unit: MPa
Global Coordinate System
Time: 1

Tensione normale
direzione verticale



$$K_t = 2.33$$

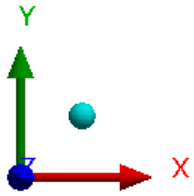
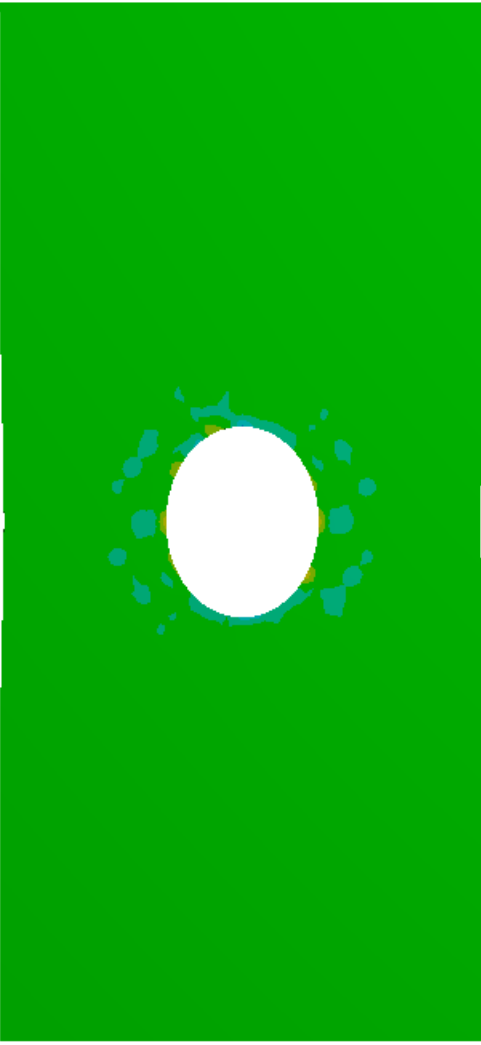
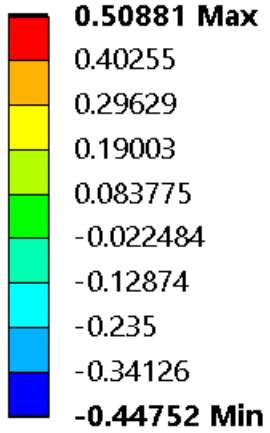


Lastra forata di piccolo spessore – *Plane Stress*

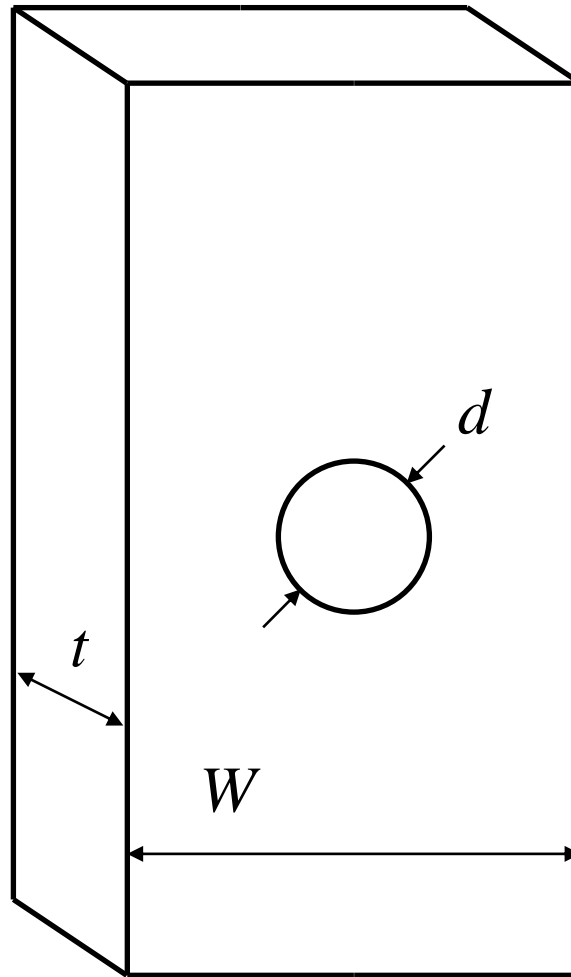
A: Static Structural (ANSYS)

Normal Stress - Z
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1

Tensione normale
allo spessore
NULLA



Lastra forata di elevato spessore



t, d confrontabili

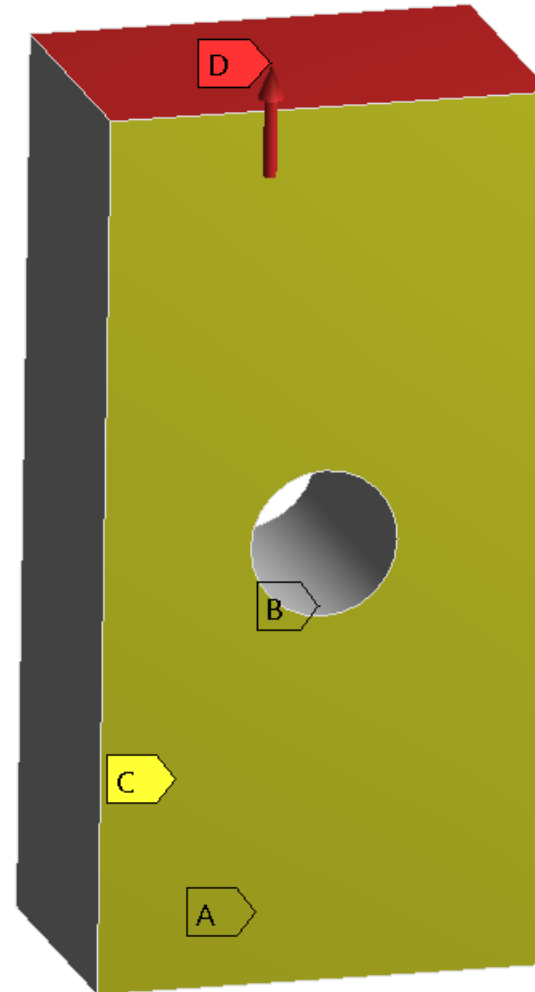
Lastra forata di elevato spessore – *Plane Strain*

A: Static Structural

Static Structural

Time: 1. s

- A** Displacement
- B** P1Strain-1
- C** P1Strain-2
- D** Pressure: -66.7 MPa

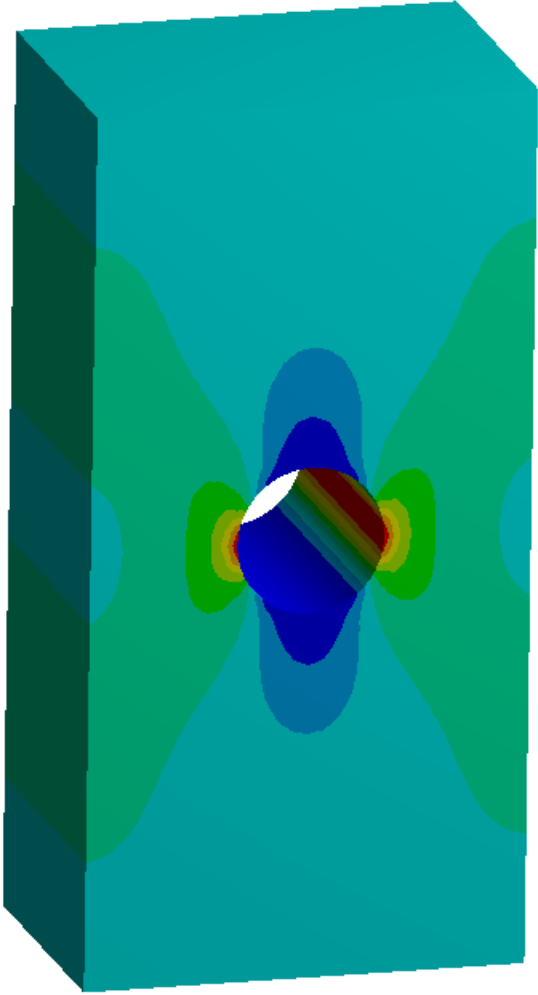
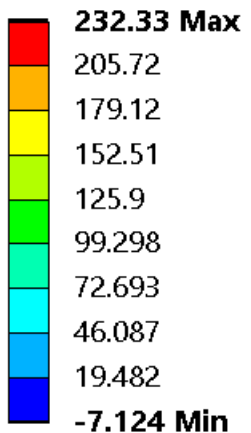


*Completa impedita contrazione laterale
(indipendentemente dallo spessore)*

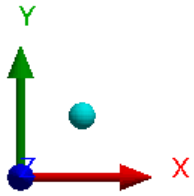
Lastra_Foro_ElevatoSpessore.x_t

Lastra forata di elevato spessore – *Plane Strain*

A: Static Structural
Normal Stress - Y
Type: Normal Stress(Y Axis)
Unit: MPa
Global Coordinate System
Time: 1

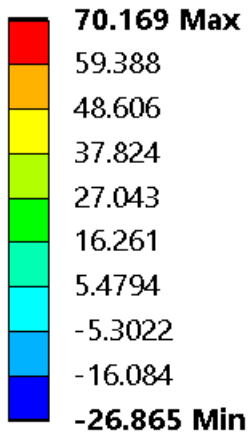


Tensione normale direzione verticale
Stessa soluzione nel piano:
plane strain / plane stress



Lastra forata di elevato spessore – Plane Strain

A: Static Structural
 Normal Stress - Z
 Type: Normal Stress(Z Axis)
 Unit: MPa
 Global Coordinate System
 Time: 1

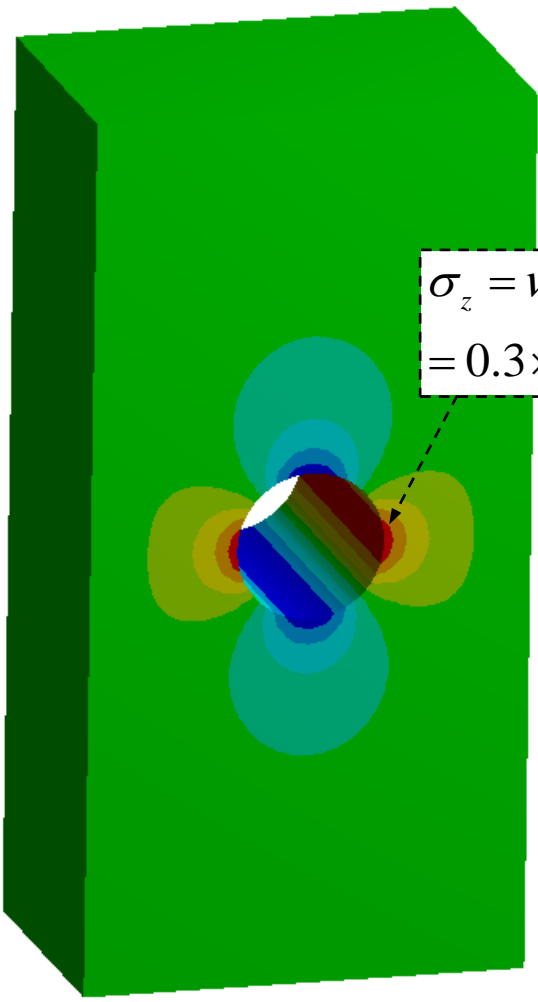


Tensione normale dir. trasversale

Plane strain:

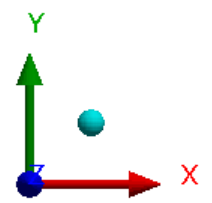
$$\sigma_z = \nu(\sigma_x + \sigma_y)$$

norm. allo spessore nel piano



$$\sigma_z = \nu(\sigma_x + \sigma_y)$$

$$= 0.3 \times (0 + 232) = 70 \text{ MPa}$$



Lastra forata di elevato spessore – *Problema 3D generico*

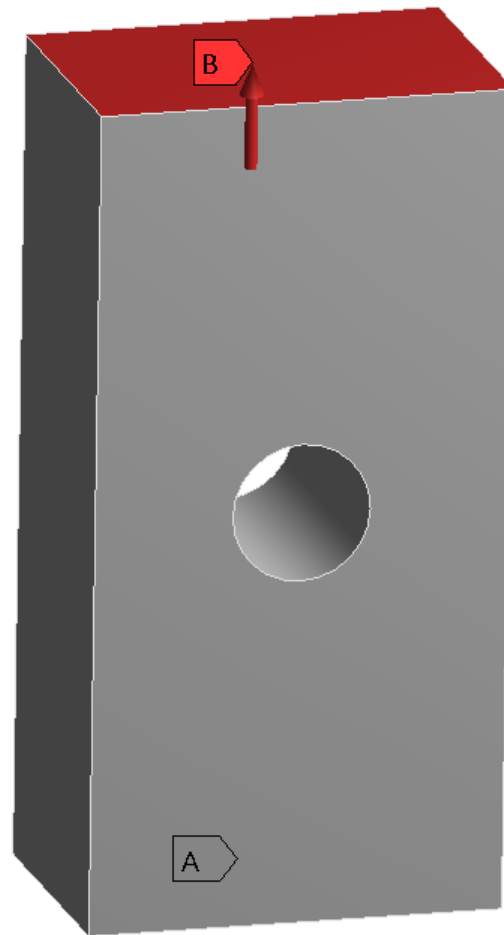
A: Static Structural

Static Structural

Time: 1. s

A Displacement

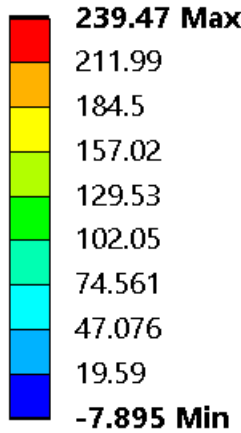
B Pressure: -66.7 MPa



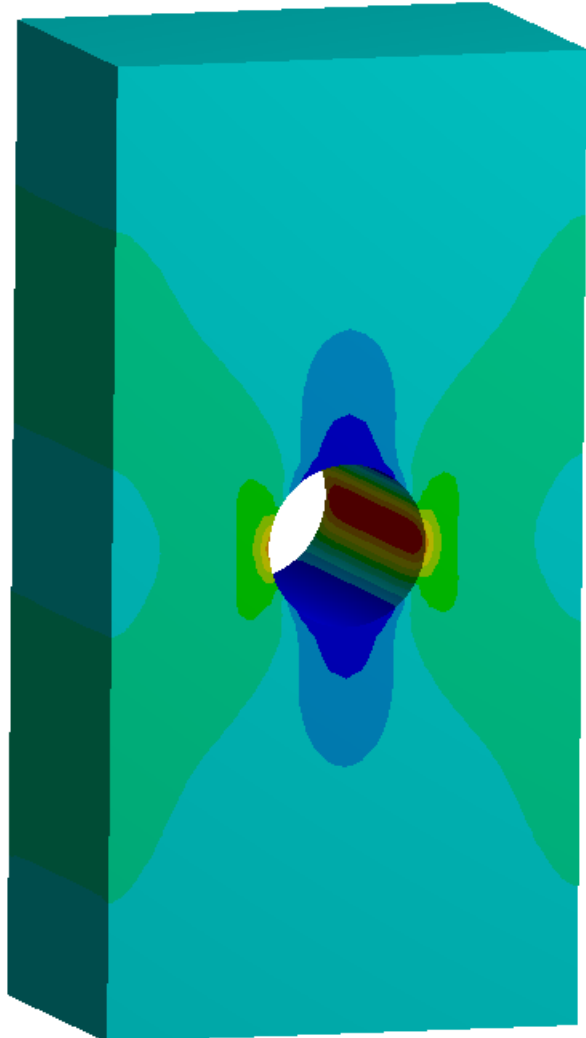
Nessun vincolo sulle facce laterali

Lastra forata di elevato spessore – *Problema 3D generico*

A: Static Structural
Normal Stress - Y
Type: Normal Stress(Y Axis)
Unit: MPa
Global Coordinate System
Time: 1

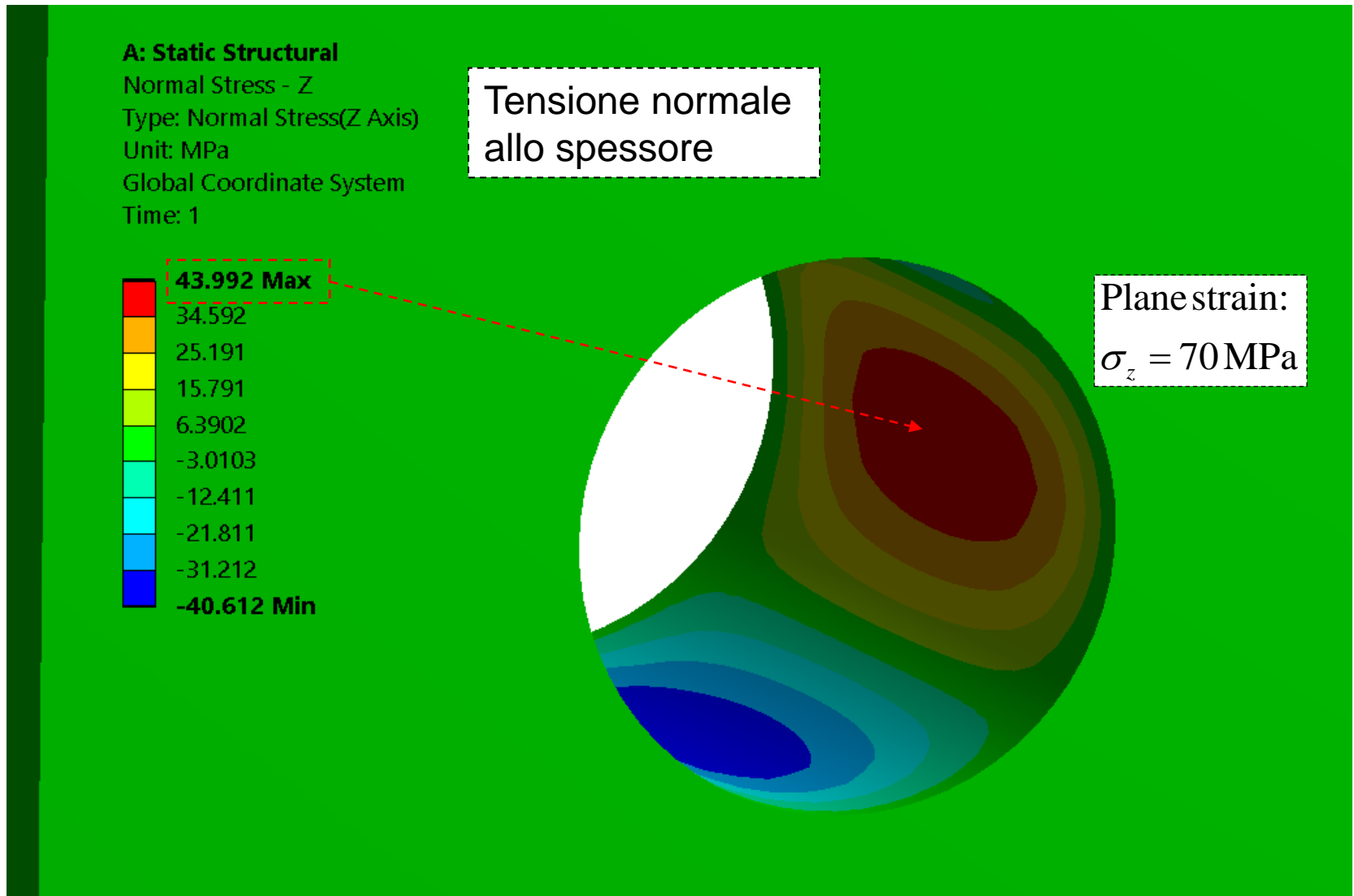


Tensione normale
direzione verticale



Tensioni nel piano molto simile a plane stress (piccolo spessore).
Stessa soluzione nel piano: plane stress / plane strain / plane strain generalizzato.

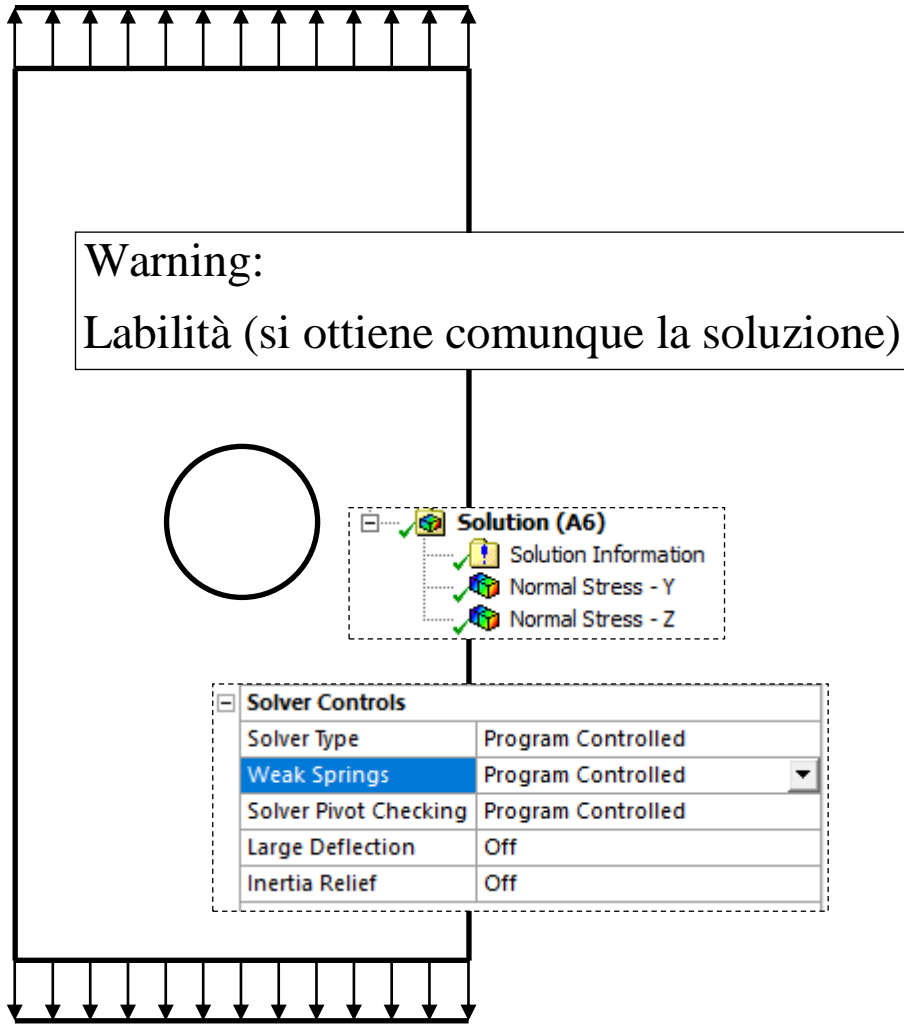
Lastra forata di elevato spessore – *Problema 3D generico*



Tensione non nulla, a causa della parziale impedita contrazione laterale.
Tensione massima circa uguale a plane strain (inferiore).

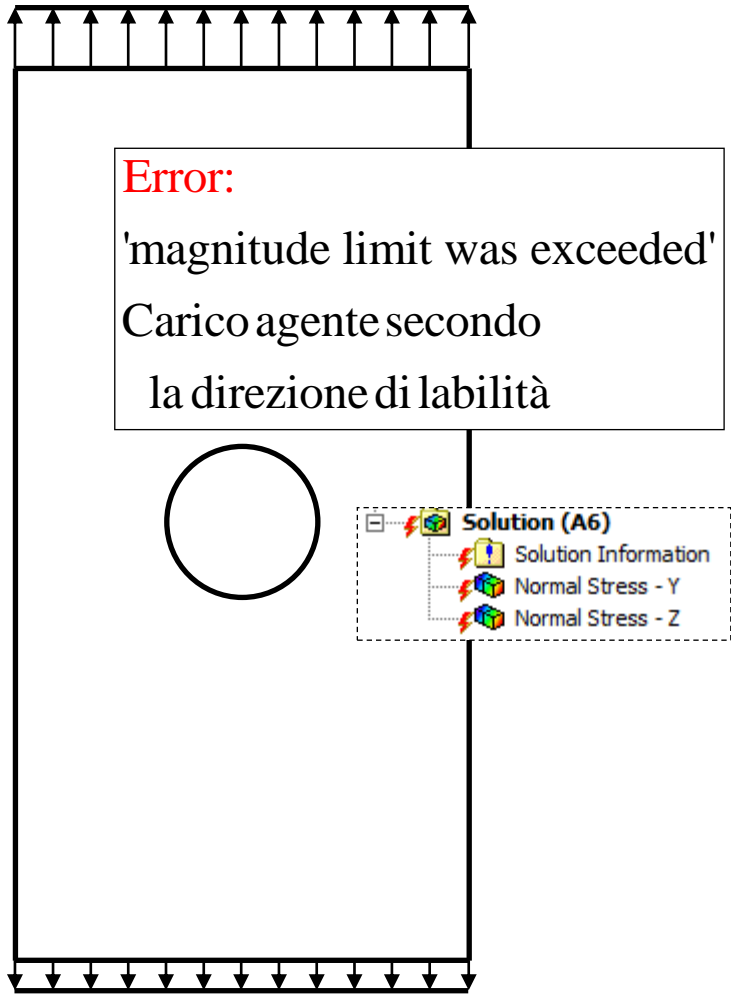
Modello isostatico / labile

$\sigma = 66.7 \text{ MPa}$



$\sigma = 66.7 \text{ MPa}$

$\sigma = 66.7 \text{ MPa}$



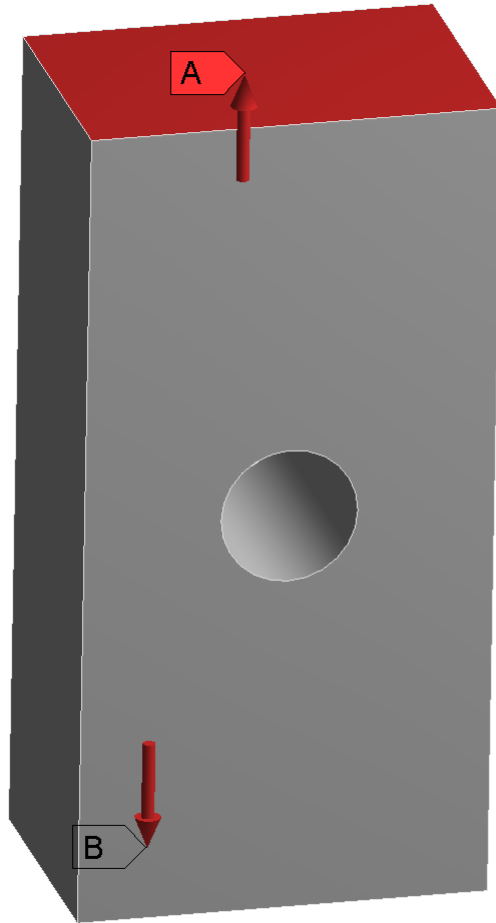
$\sigma = 50 \text{ MPa}$

Sommario Lez. 1

1. Comandi di base: vincolo, carico, visualizzazione tensioni, etc.
2. Valore di riferimento risultato cercato:
 - utile per **validare** il modello (ordine di grandezza),
 - possibili differenze con modello FE.
3. Convergenza tensione massima:
 - **OK** in presenza di raggio di raccordo,
 - **NO** senza raggio di raccordo “sharp notch”,
soluzione FE finita ma crescente con l’infittimento della mesh.
4. Elevato **onere computazionale** anche con modelli molto semplici.
5. Soluzione Plane Stress - Plane Strain e 3D.

Esercitazione da svolgere 1:

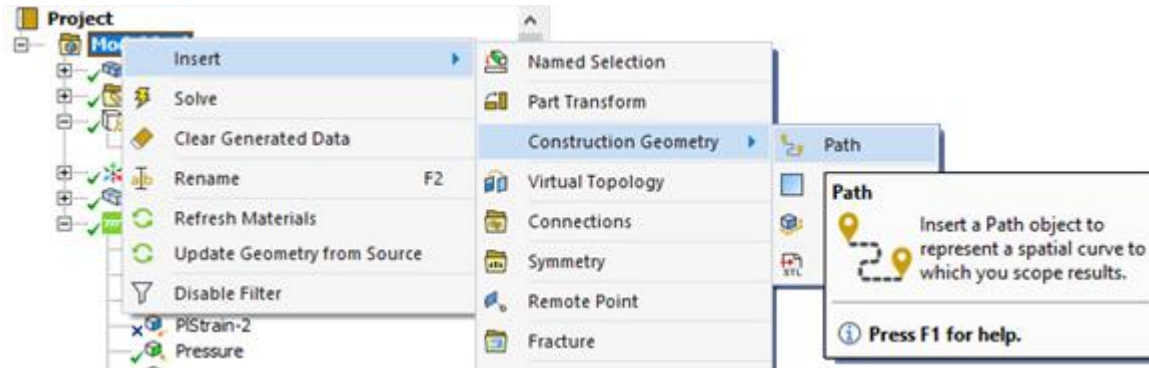
Definire una corretta distribuzione dei vincoli per eliminare la labilità nel caso in cui siano applicate azioni distribuite di trazione (pressione negativa) sulle due facce opposte



Come ottenere una condizione di vincolo isostatica, senza perturbare la distribuzione di deformazione delle superfici?

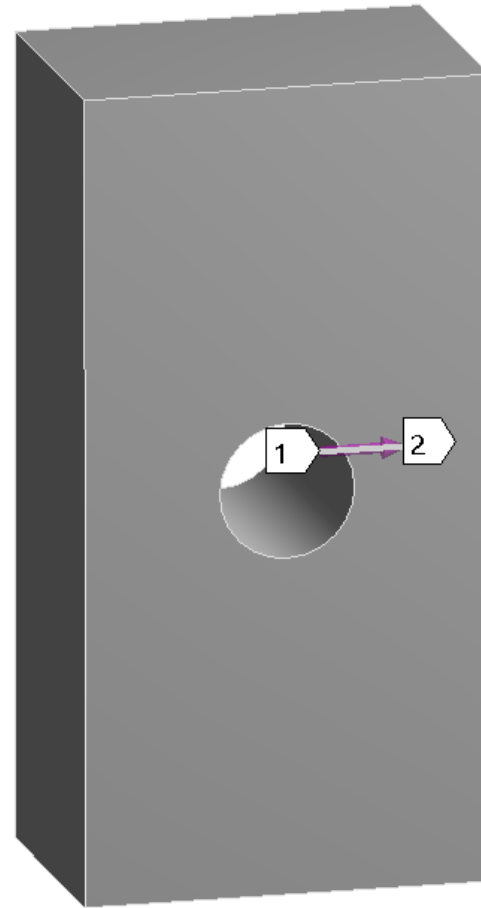
Esercitazione da svolgere 2:

Confrontare la tensione Z (perpendicolare al piano della lastra) fra Pl. Strain e il caso tridimensionale generico, utilizzando il Path di Workbench



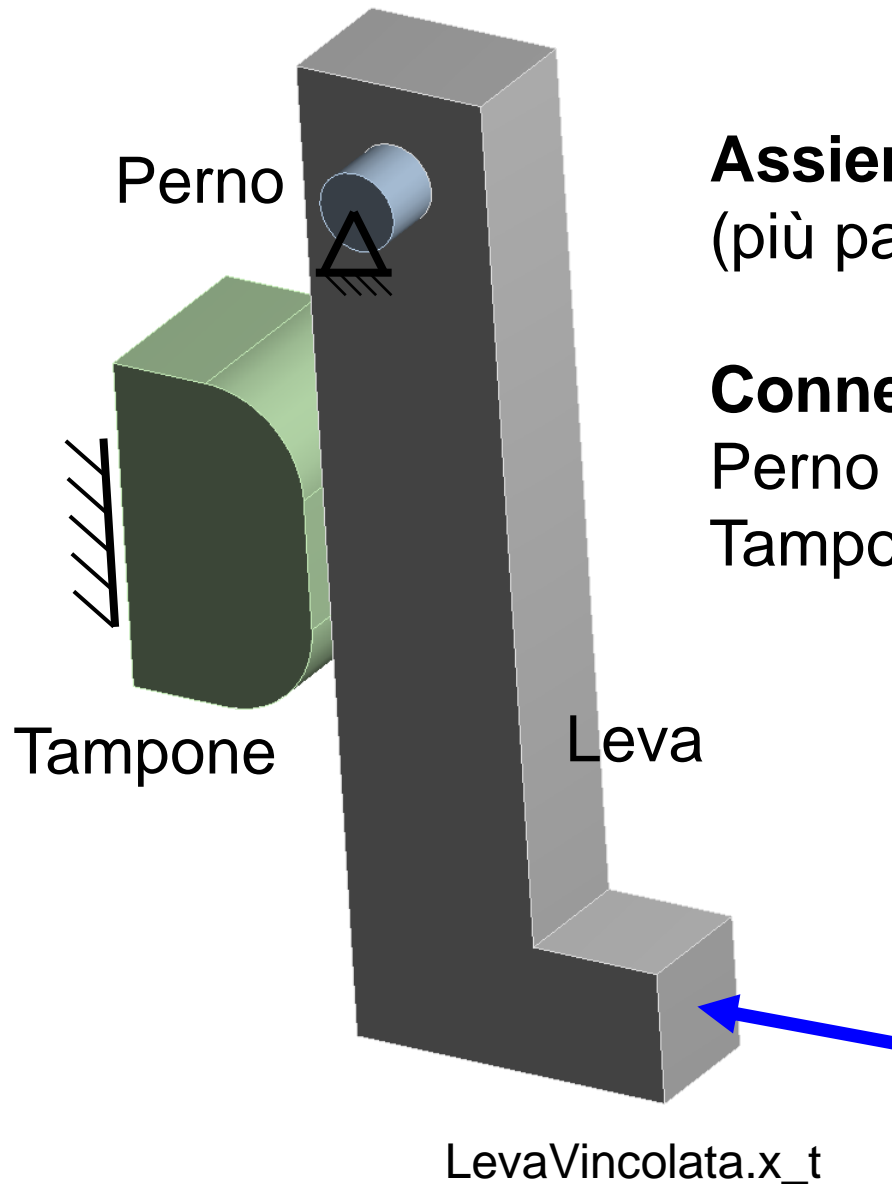
Details of "Path"

Definition	
Path Type	Two Points
Path Coordinate System	Global Coordinate System
Number of Sampling Points	99.
Suppressed	No
Start	
Coordinate System	Coordinate System - Path
Start X Coordinate	-5. mm
Start Y Coordinate	0. mm
Start Z Coordinate	0. mm
Location	Click to Change
End	
Coordinate System	Coordinate System - Path
End X Coordinate	-15. mm
End Y Coordinate	0. mm
End Z Coordinate	0. mm
Location	Click to Change



Definizione dei due punti estremi del Path

Lez. 2. Modellazione di più parti in Ansys Workbench, utilizzo delle connessioni



Assieme

(più parti assemblate in CAD)

Connessioni:

Perno – Leva

Tampone – Leva

LevaVincolata.x_t

Engineering data: introduzione, modifica dei materiali

Default: *Acciaio strutturale (Structural Steel)*

Possibilità di introdurre/duplicare/importare altri materiali da libreria ed eventualmente modificarne le proprietà

Outline of Schematic A2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data				Source
2	Material				Description
3	Aluminum Alloy				Gen General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
4	Polyethylene				Gen
5	Structural Steel				C:V Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
6	Structural Steel 2				C:V Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 5: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's Modulus and Poisson...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Alternating Stress Mean Stress	Tabular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

Assegnazione dei materiali

The screenshot shows the ANSYS Mechanical interface. The 'Outline' tree on the left lists the model components. The 'Details of "Leva"' panel is open, showing the material assignment. A callout box points to the 'Leva' part in the tree, and another callout box highlights the 'Aluminum Alloy' material assignment in the details panel.

Outline Tree:

- Project
 - Model (A4)
 - Geometry
 - Leva
 - Perno
 - Tampone
 - Materials
 - Coordinate Systems
 - Connections
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Fixed Support 2
 - Force

Details of "Leva" Panel:

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Material	
Assignment	Aluminum Alloy
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	

3D Model: A 3D model of a mechanical part (a lever) is shown in a blue background. The part is colored green. A scale bar at the bottom indicates dimensions in millimeters (0.00 to 70.00 mm). A coordinate system (X, Y, Z) is visible at the bottom right.

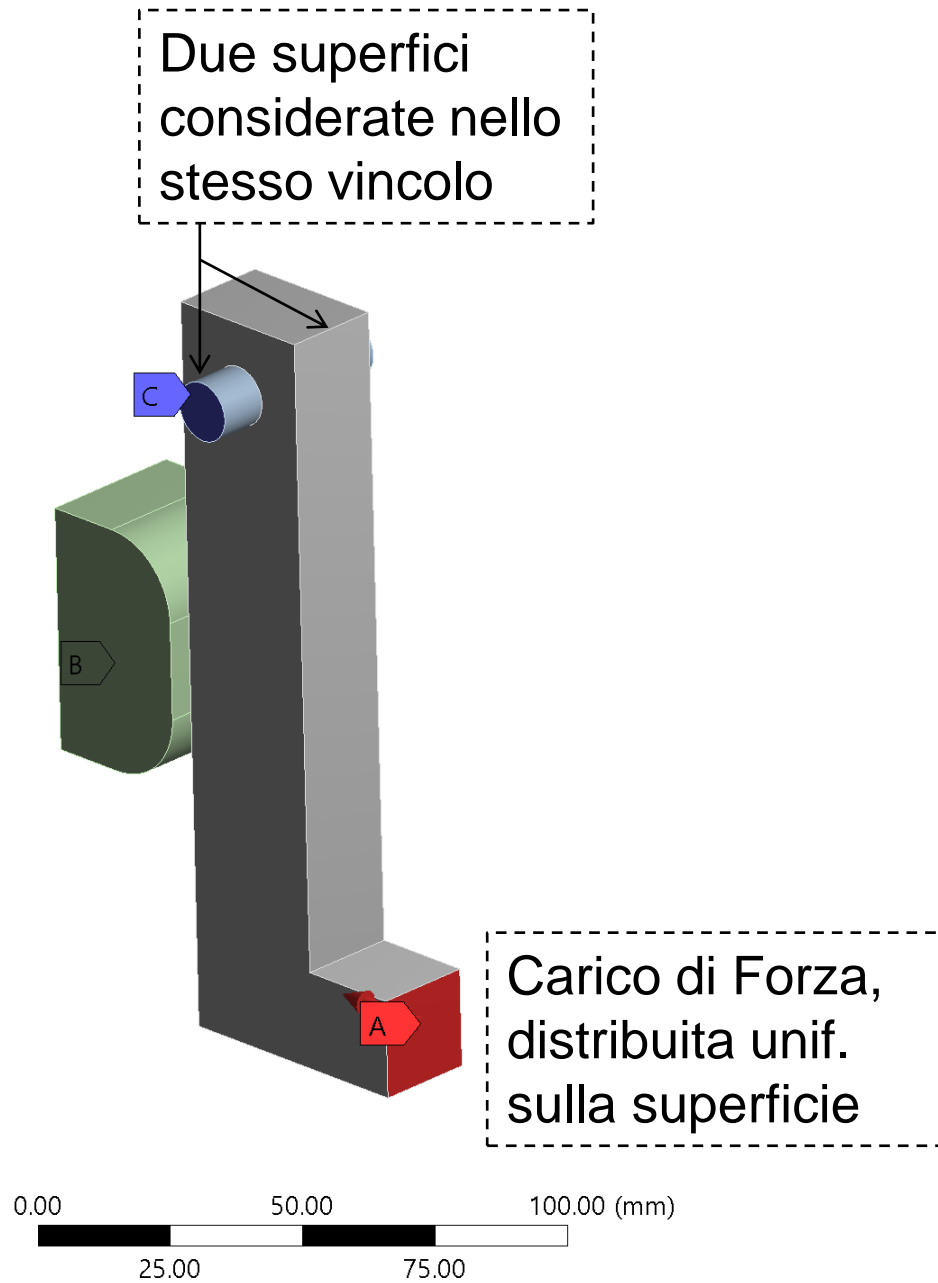
Vincoli e carichi

A: Static Structural (ANSYS)

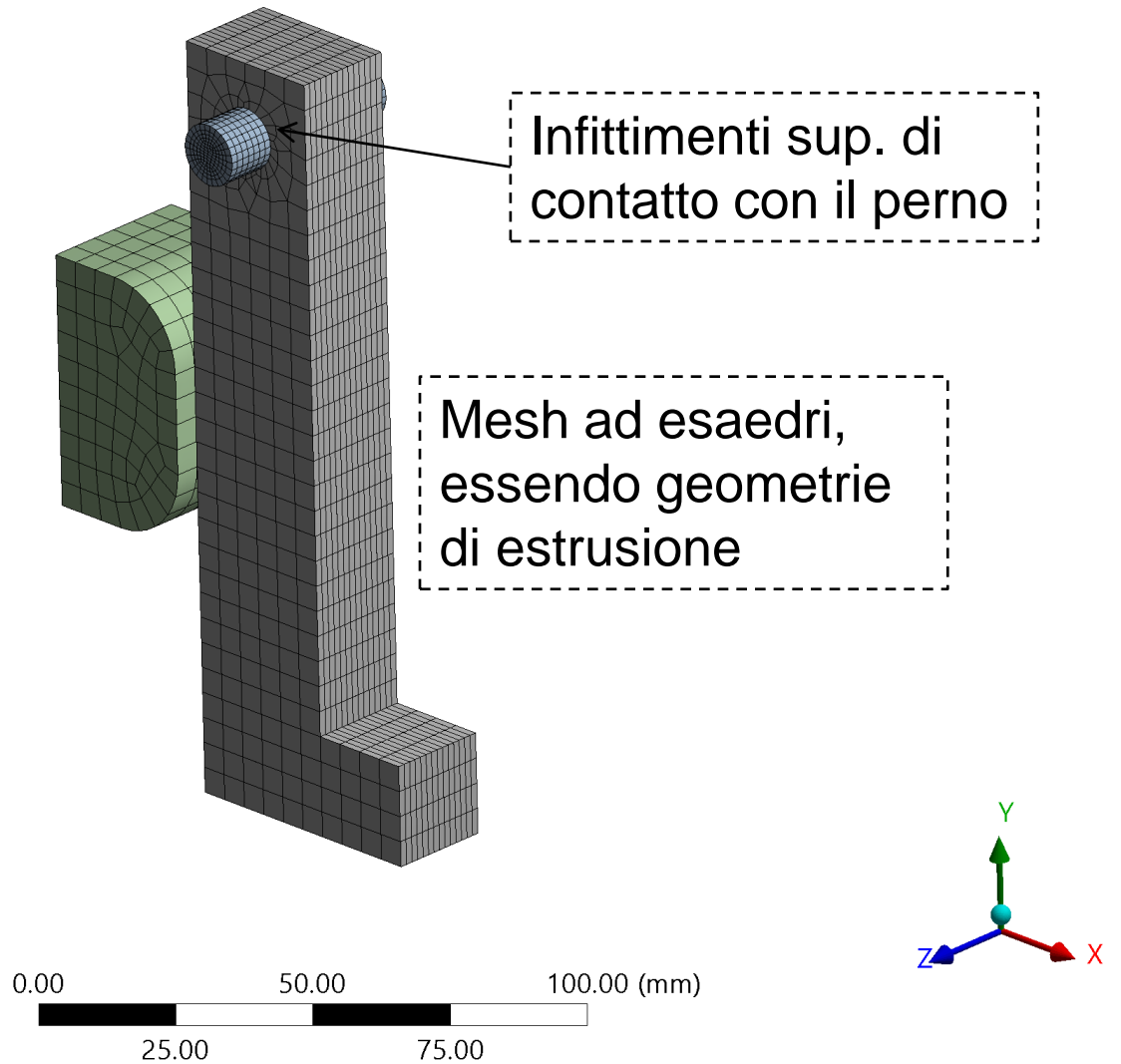
Static Structural

Time: 1. s

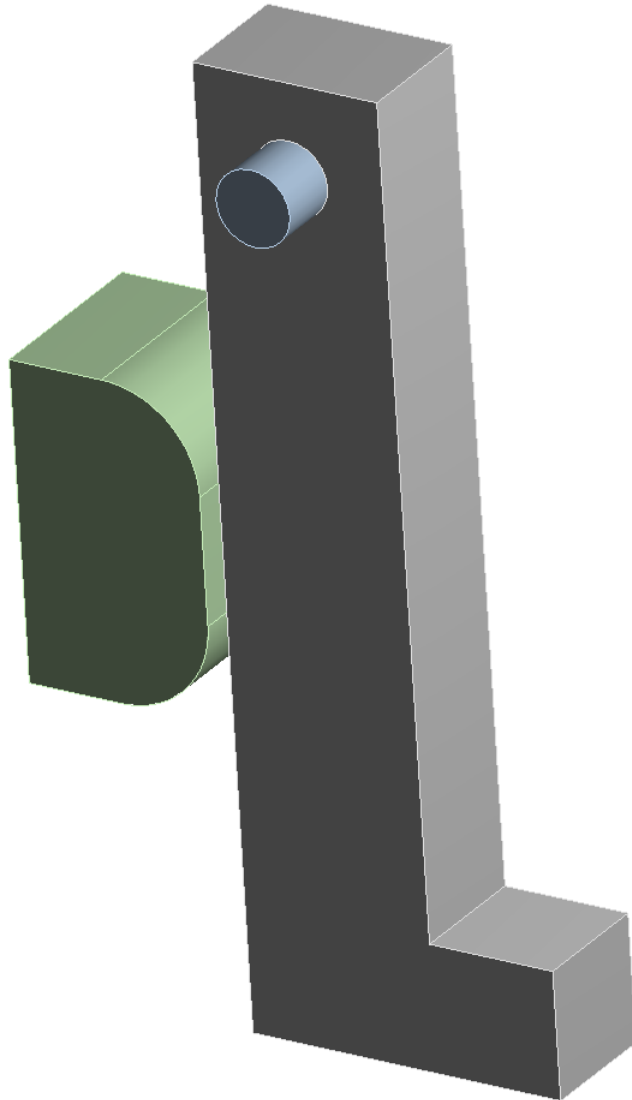
- A** Force: 1000. N
- B** Fixed Support
- C** Fixed Support 2



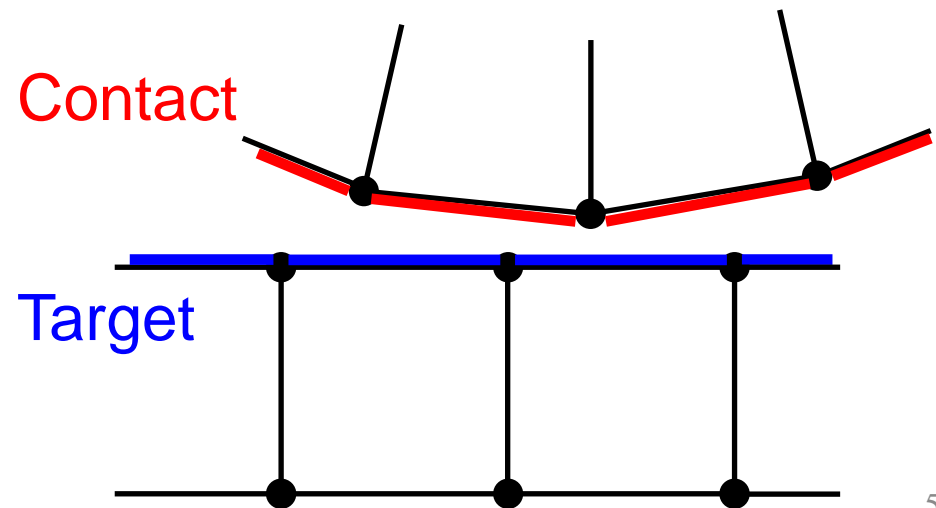
Mesh, infittimento



Introduzione della geometria



Riconoscimento automatico delle **connessioni** fra le parti aventi superfici adiacenti



Ansys Workbench, connessioni

The screenshot displays the Ansys Workbench interface for a static structural analysis. The main window shows a 3D model of a mechanical assembly with a contact defined between two parts: 'Leva' (red) and 'Tampone' (blue). The contact is named 'No Separation - Leva To Tampone'. The interface includes a menu bar, a toolbar, an Outline tree on the left, and a Details panel at the bottom left. The central workspace shows the 3D model with a coordinate system and a scale bar. The right side of the interface features a 'Contact Box' and a 'Target Body' view, both showing the contact geometry in a simplified 2D/3D perspective.

Outline Tree:

- Project
 - Model (A4)
 - Geometry
 - Leva
 - Perno
 - Tampone
 - Materials
 - Coordinate Systems
 - Connections
 - Contacts
 - No Separation - Leva To Perno
 - No Separation - Leva To Tampone
 - Mesh
 - Static Structural (A5)
 - Analysis Settings

Details of "No Separation - Leva To Tampone":

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	3 Faces
Contact Bodies	Leva
Target Bodies	Tampone
Protected	No
Definition	
Type	No Separation
Scope Mode	Automatic
Behavior	Symmetric
Trim Contact	Program Controlled
Trim Tolerance	0.43661 mm
Suppressed	No
Advanced	
Formulation	Pure Penalty

Ansys Workbench, stato delle connessioni

- “Bonded (Vincolato)”: vincolo relativo sia normale (**bilaterale**) che tangenziale (default).
- “No separation (Senza separazione)”: vincolo relativo solo normale (**bilaterale**) ma non tangenziale.
- “Frictionless (Senza attrito)”: vincolo relativo solo normale (**unilaterale** di compressione) ma non tangenziale.
- “Rough (Rugoso)”: vincolo relativo sia normale (**unilaterale**) sia tangenziale, con intensità di quest’ultimo indipendente dell’azione normale, purché di compressione.
- “Frictional (Con attrito)”: vincolo relativo sia normale (**unilaterale**) sia tangenziale, fino al limite di attrito. Successivo comportamento di attrito dinamico.

Ansys Workbench, stato delle connessioni

- “Bonded”: (linearità)
- “No separation”: (linearità)
- “Frictionless”: (non-linearità, difficoltà di soluzione)
- “Rough”: (non-linearità, difficoltà di soluzione)
- “Frictional”: (non-linearità, difficoltà di soluzione)

Ansys Workbench, connessioni

The screenshot displays the Ansys Workbench interface for a Static Structural analysis. The main window shows a 3D model of a mechanical assembly with a contact defined between two parts. A callout box highlights the contact name and type.

Callout Box:

Entrambe le connessioni:
No Separation

Legend:

- Contact Bodies (Red)
- Target Bodies (Blue)

Outline:

- Project
 - Model (A4)
 - Geometry
 - Leva
 - Perno
 - Tampone
 - Materials
 - Coordinate Systems
 - Connections
 - No Separation - Leva To Perno
 - No Separation - Leva To Tampone
 - Mesh
 - Static Structural (A5)
 - Analysis Settings

Details of "No Separation - Leva To Tampone":

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	3 Faces
Contact Bodies	Leva
Target Bodies	Tampone
Protected	No
Definition	
Type	No Separation
Scope Mode	Automatic
Behavior	Symmetric
Trim Contact	Program Controlled
Trim Tolerance	0.43661 mm
Suppressed	No
Advanced	
Formulation	Pure Penalty

3D Model: The main view shows a 3D model of a mechanical assembly. A callout box highlights the contact between the 'Leva' (red) and 'Tampone' (blue) parts. A scale bar indicates 70.00 mm.

Contact Body View: A separate view shows the 'Leva' part (red) with a scale bar indicating 20.00 mm.

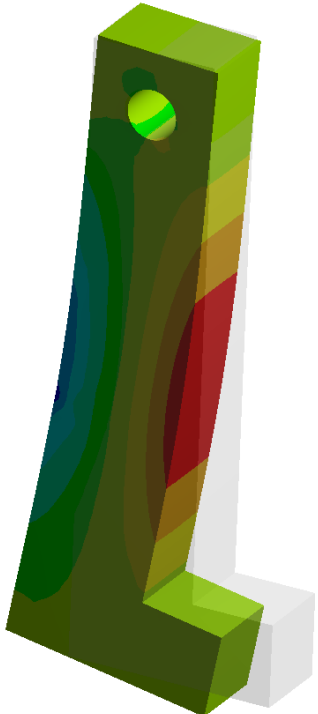
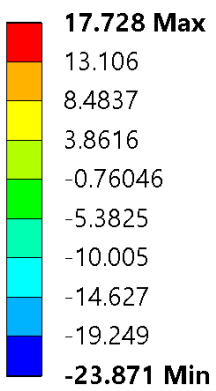
Target Body View: A separate view shows the 'Tampone' part (blue) with a scale bar indicating 10.00 mm.

Status Bar: 2 Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Soluzione, analisi dei risultati

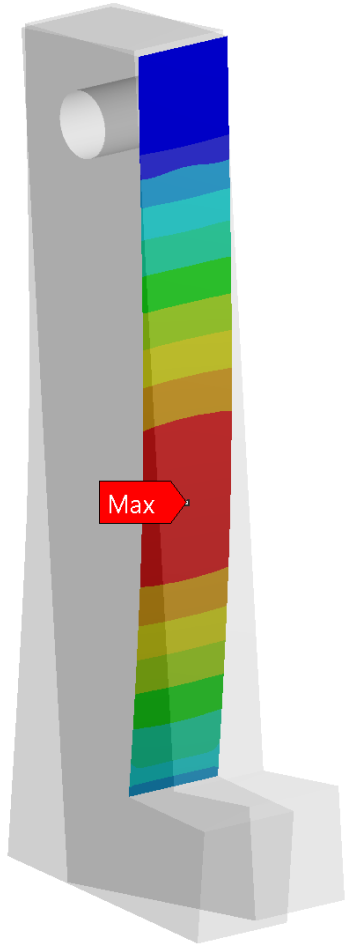
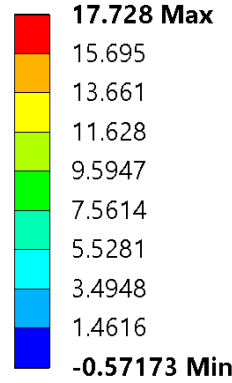
A: Static Structural (ANSYS)

Normal Stress
Type: Normal Stress(Y Axis)
Unit: MPa
Global Coordinate System
Time: 1



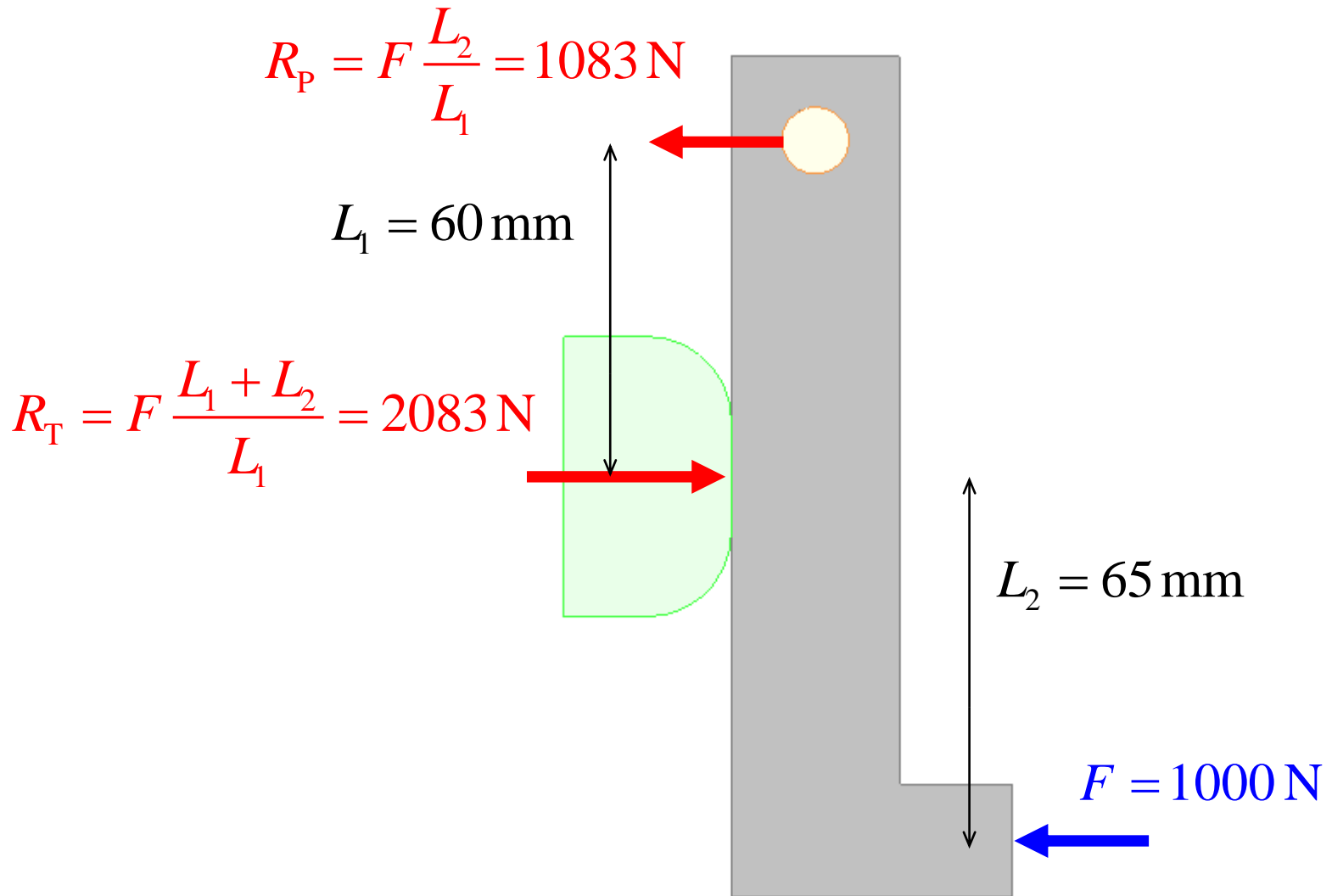
A: Static Structural (ANSYS)

Normal Stress 2
Type: Normal Stress(Y Axis)
Unit: MPa
Global Coordinate System
Time: 1



Possibilità di visualizzare i risultati solo su una parte, oppure superficie o linea

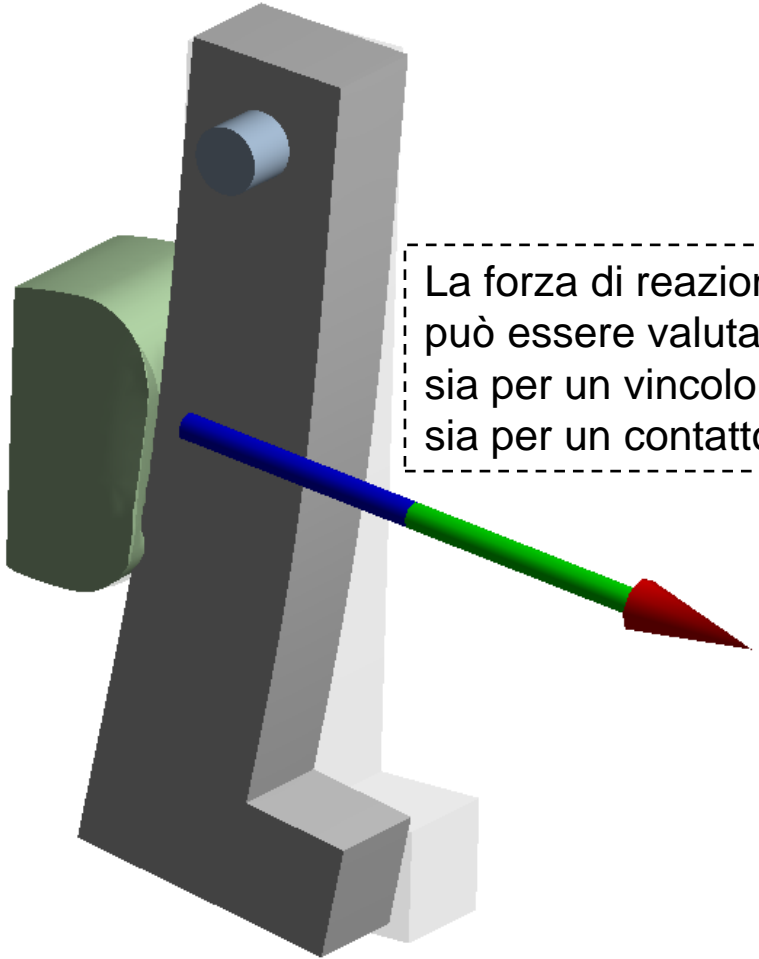
Forze e momenti di reazione



Risultati numerici, forza di reazione

The screenshot shows the ANSYS Workbench interface. In the Outline pane, the 'Force Reaction' result is highlighted with a red dashed box. The Details pane for 'Force Reaction' is expanded, showing the following data:

Definition	
Type	Force Reaction
Location Method	Contact Region
Contact Region	No Separation - Leva To Tampone
Orientation	Global Coordinate System
Extraction	Contact (Underlying Element)
Suppressed	No
Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time
Results	
<input type="checkbox"/> X Axis	1906.5 N
<input type="checkbox"/> Y Axis	-203.26 N
<input type="checkbox"/> Z Axis	9.3929e-007 N
<input type="checkbox"/> Total	1917.3 N



La forza di reazione può essere valutata sia per un vincolo sia per un contatto

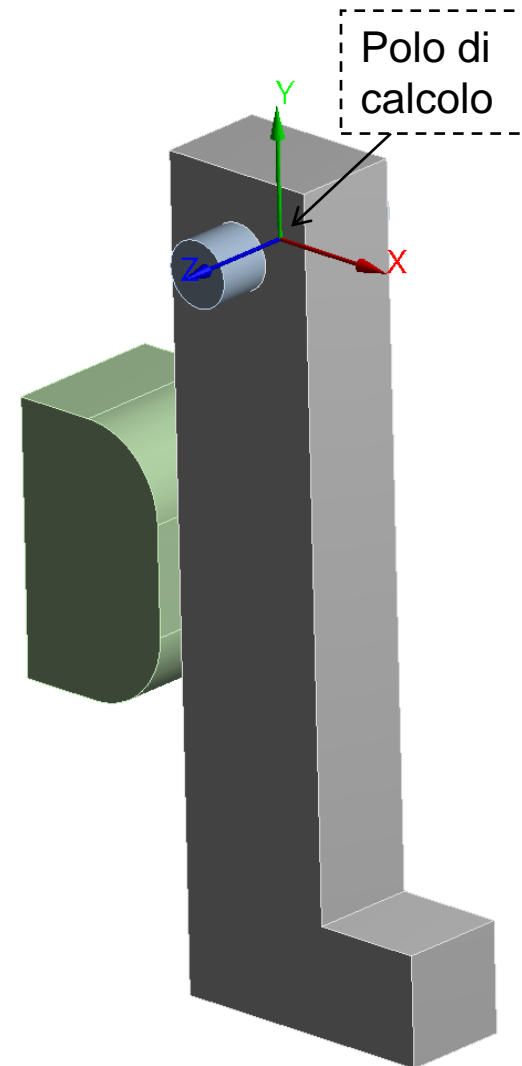
Valore atteso:
 $R_T = 2083 \text{ N}$
 Valore ottenuto:
 $R_T = 1917 \text{ N}$
 comp. orizzontale = 1907 N

Momento risultante

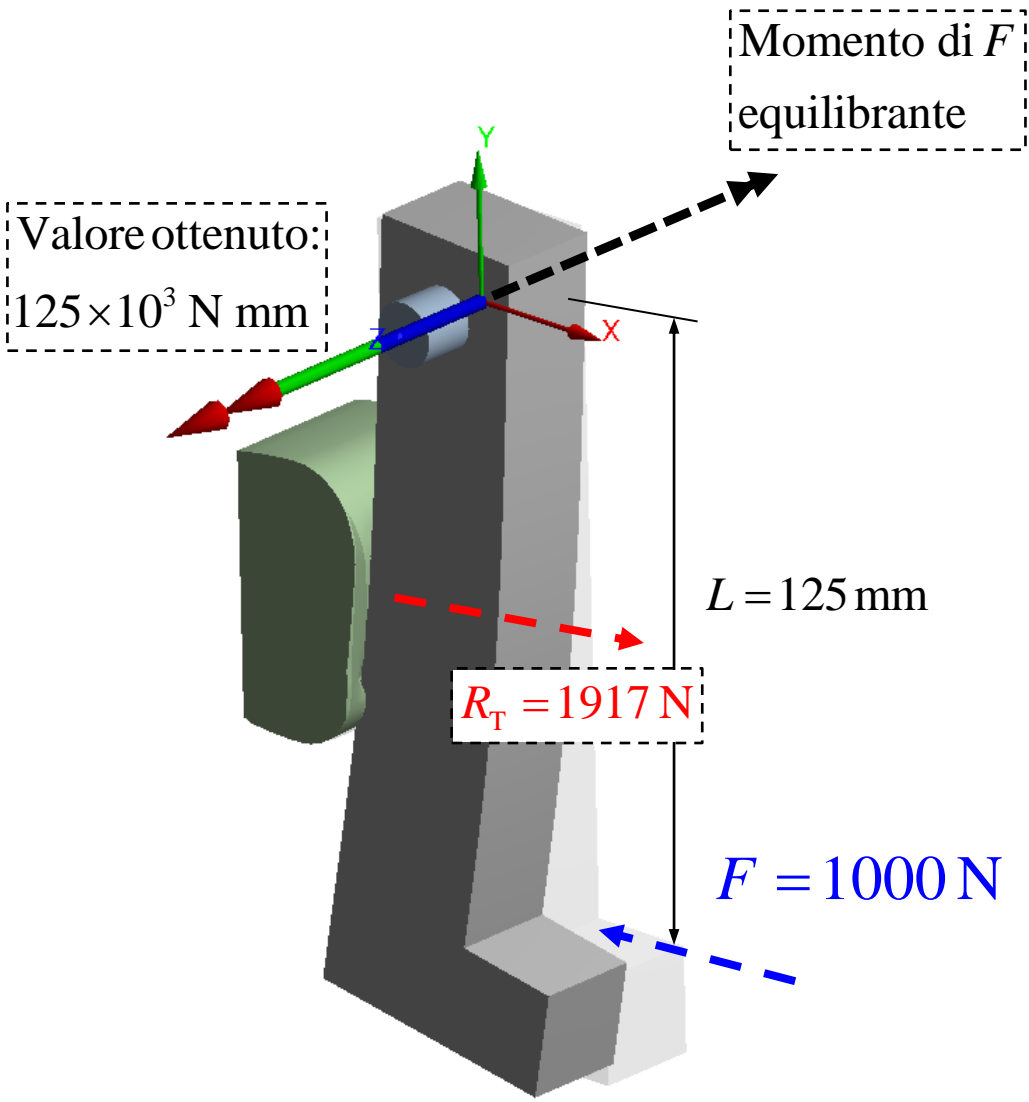
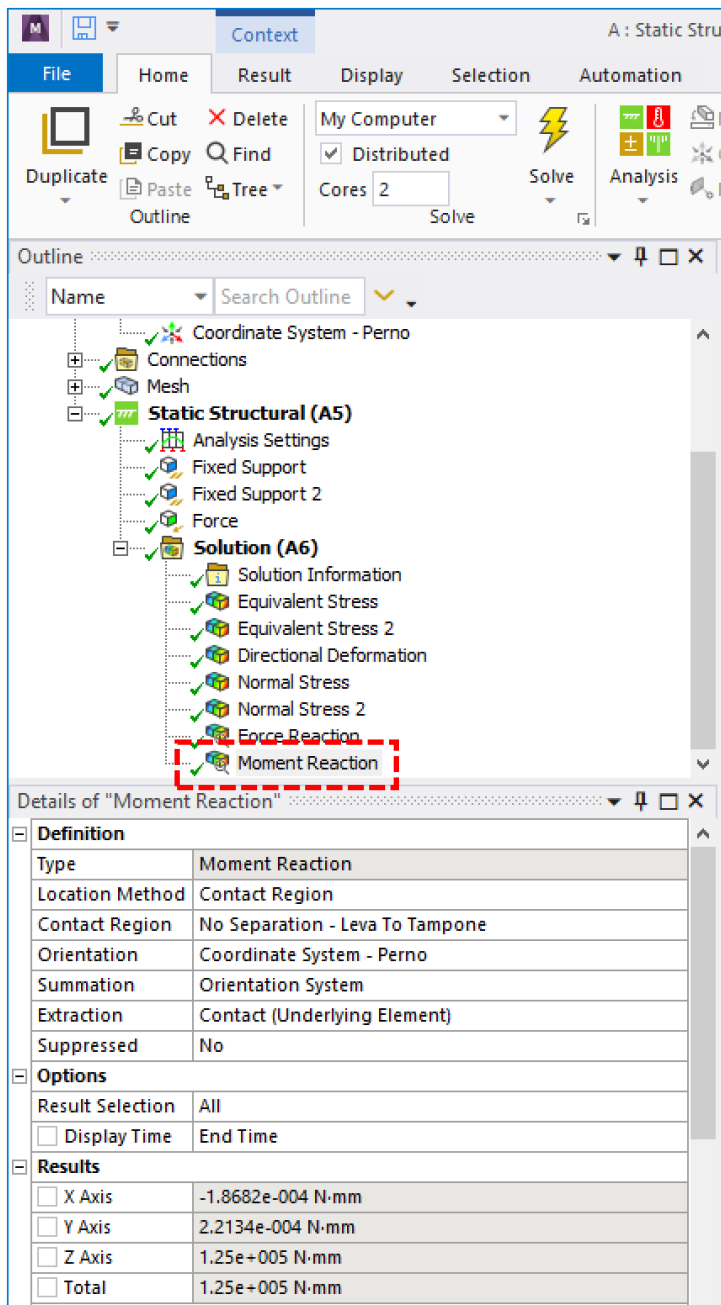
Definizione di un nuovo sistema di riferimento

The screenshot displays the ANSYS Workbench software interface. The 'Context' tab is active, showing the 'Coordinate Systems' section. The 'Outline' pane on the left shows the project hierarchy, with 'Coordinate System - Perno' highlighted in a red dashed box. The 'Details of "Coordinate System - Perno"' pane at the bottom shows the following configuration:

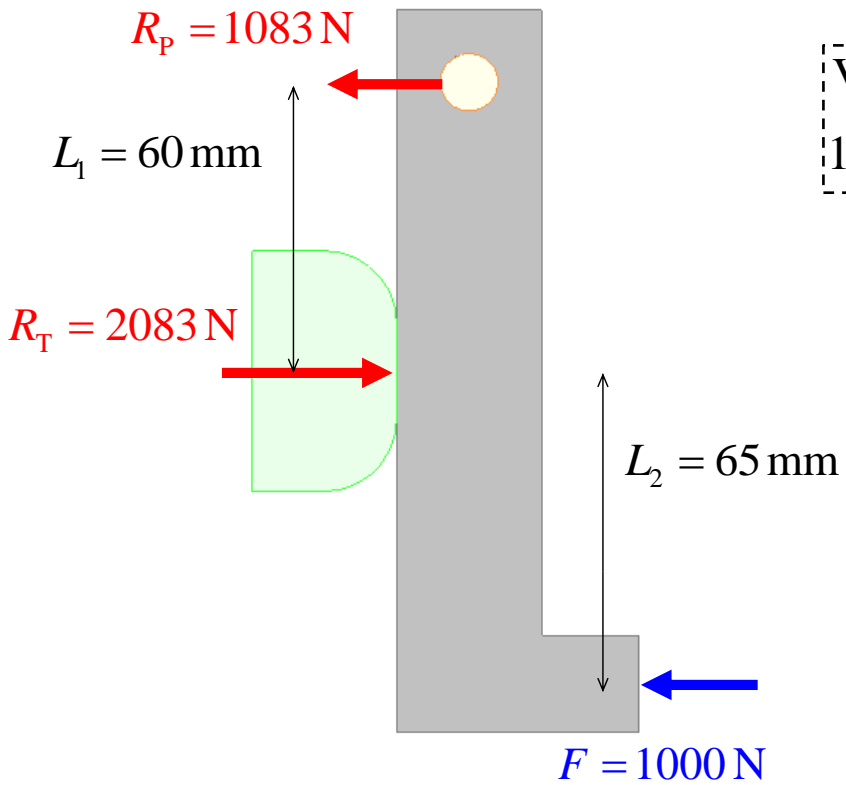
Definition	
Type	Cartesian
Coordinate System	Program Controlled
APDL Name	
Suppressed	No
Origin	
Define By	Geometry Selection
Geometry	Click to Change
Origin X	15. mm
Origin Y	135. mm
Origin Z	10. mm



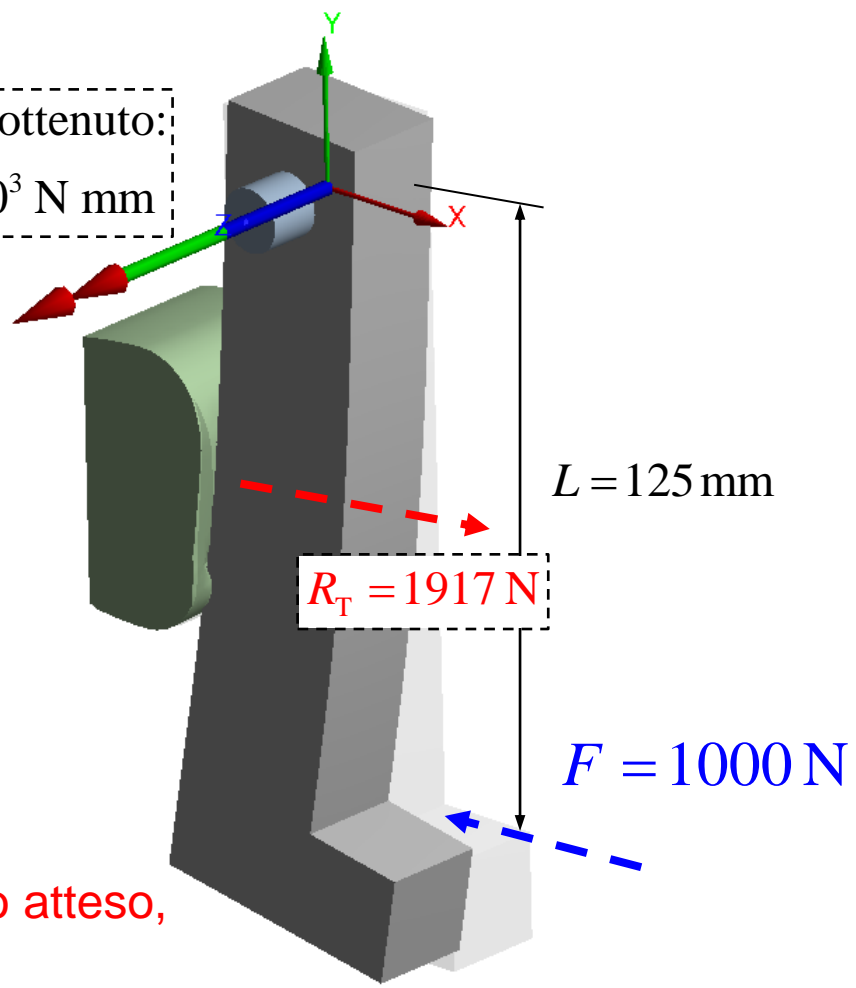
Momento della forza di contatto fra Tampone e Leva



Analisi critica dei risultati



Valore ottenuto:
 $125 \times 10^3 \text{ N mm}$



- Perché il valore di R_T non è uguale a quello atteso, nonostante il relativo momento sia esatto?
- Perché la forza di contatto R_T ha una componente verticale, seppur piccola?

Contacto senza attrito: Perno - Leva

The screenshot shows the ANSYS Workbench interface with the following details:

- Context:** A : Static Stru
- Outline:**
 - Materials
 - Coordinate Systems
 - Connections
 - Contacts
 - Frictionless - Leva To Perno (highlighted with a red dashed box)
 - Bonded - Leva To Tampone
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Fixed Support 2
 - Force
 - Solution (A6)
 - Solution Information
 - Equivalent Stress
 - Equivalent Stress 2
 - Directional Deformation
- Details of "Frictionless - Leva To Perno":**

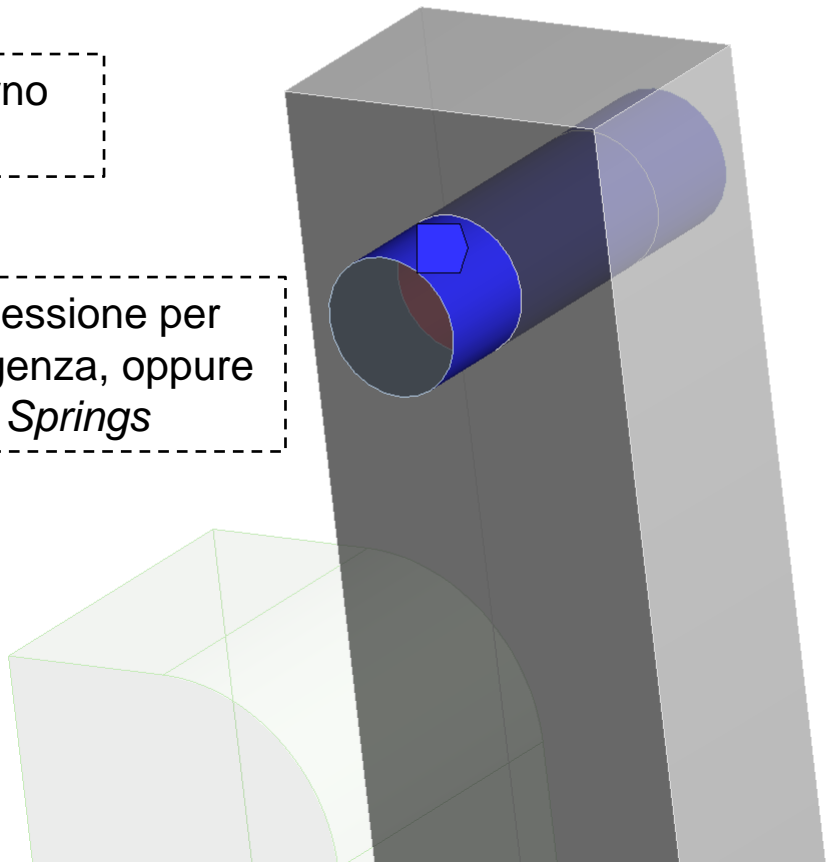
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Leva
Target Bodies	Perno
Protected	No
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Symmetric
Trim Contact	Program Controlle
Trim Tolerance	0.43661 mm
Suppressed	No

Frictionless - Leva To Perno

- Contact Bodies
- Target Bodies

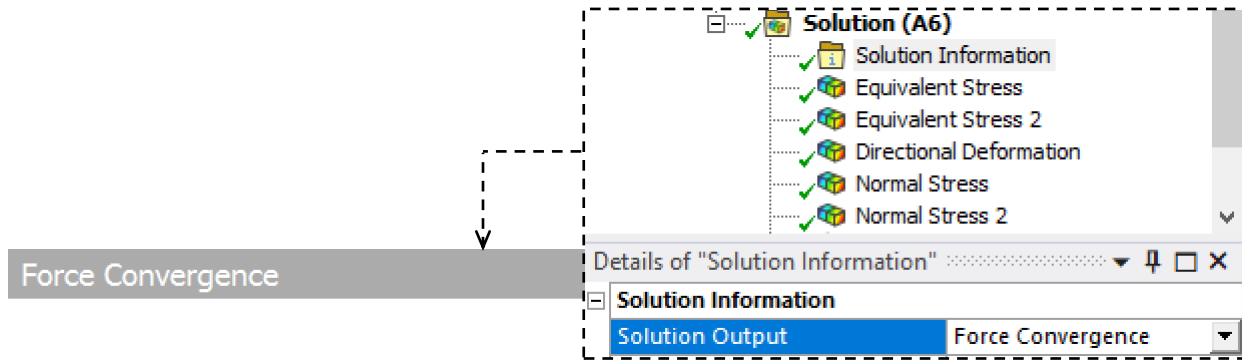
Contatto Leva-Perno
Frictionless

Bonded altra connessione per favorire la convergenza, oppure attivando le *Weak Springs*

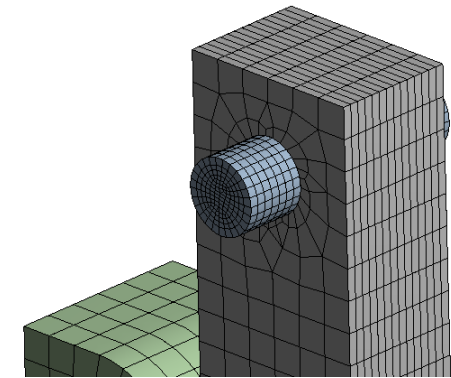
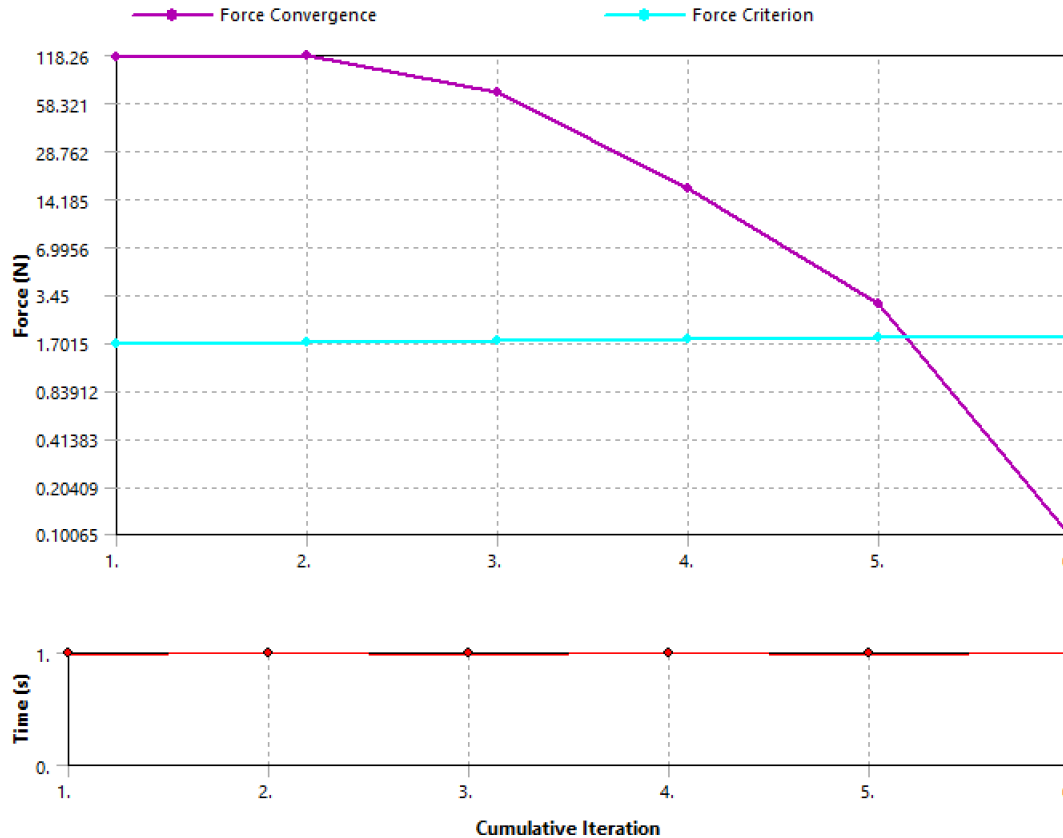


Analisi non lineare (contatto)

Iterazioni di convergenza



L'infittimento locale può migliorare la procedura di convergenza



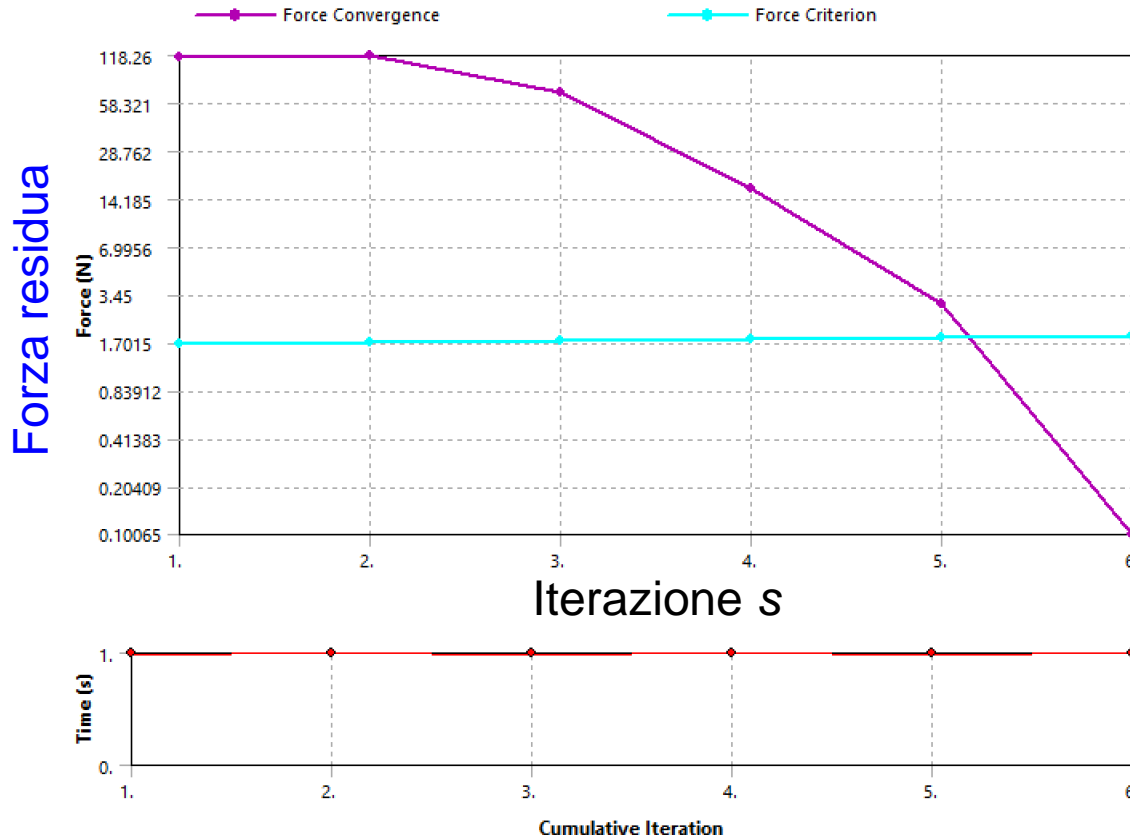
Hanno un ruolo importante anche le condizioni di vincolo delle parti coinvolte, sull'esito della convergenza del contatto

Analisi non lineare (contatto)

Iterazioni di convergenza

Forza residua: è una misura della variazione delle forze di contatto, ovvero la norma della differenza fra le forze all'iterazione precedente e quelle all'iterazione attuale

Force Convergence



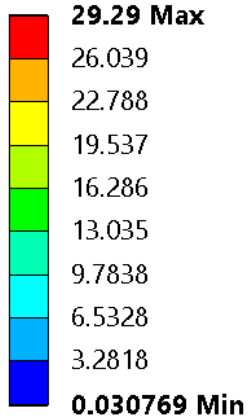
Sulla base della stima di quali contatti sono aperti/chiusi è valutata la forza alla generica iterazione s : $[K]_s\{U\}$, e confrontata con la forza allo step precedente. Il criterio è una frazione, es. 0.001 della forza complessiva allo step precedente e quindi anch'esso subisce delle (piccole) variazioni.

Ok convergenza!

Eventuale suddivisione temporale del carico, tipicamente di un fattore $\frac{1}{2}$ (bisection), per favorire il percorso di convergenza ed arrivare più facilmente al pieno carico applicato, tuttavia non in questo caso

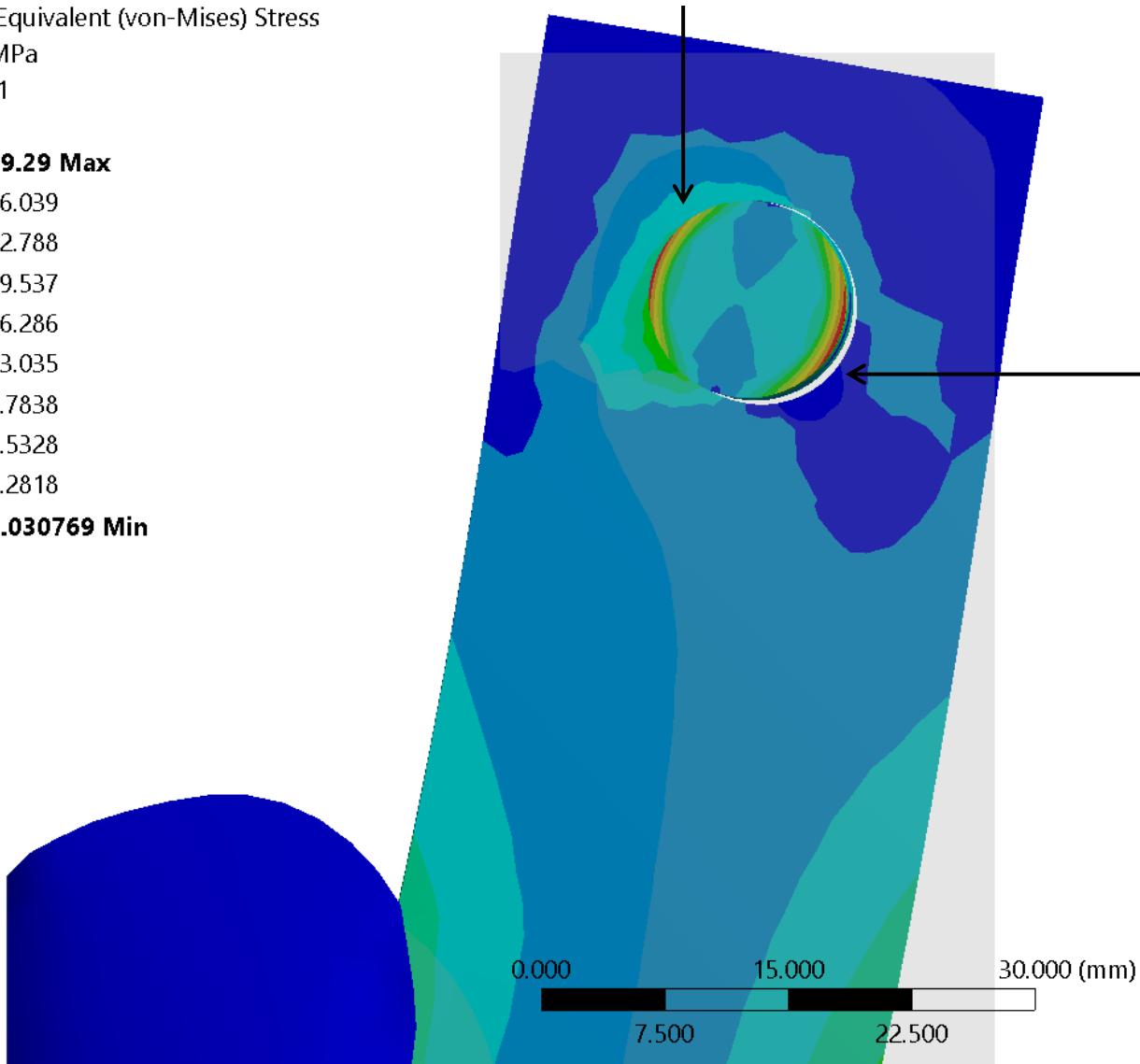
Connessione di contatto perno con sede

A: Static Structural (ANSYS)
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1



Compressione (contatto chiuso)

Contatto aperto



Sommario Lez. 2

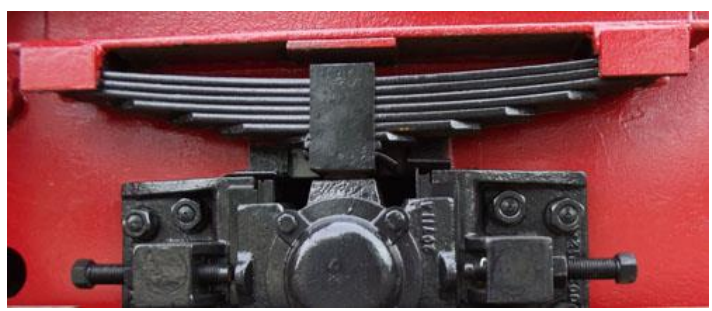
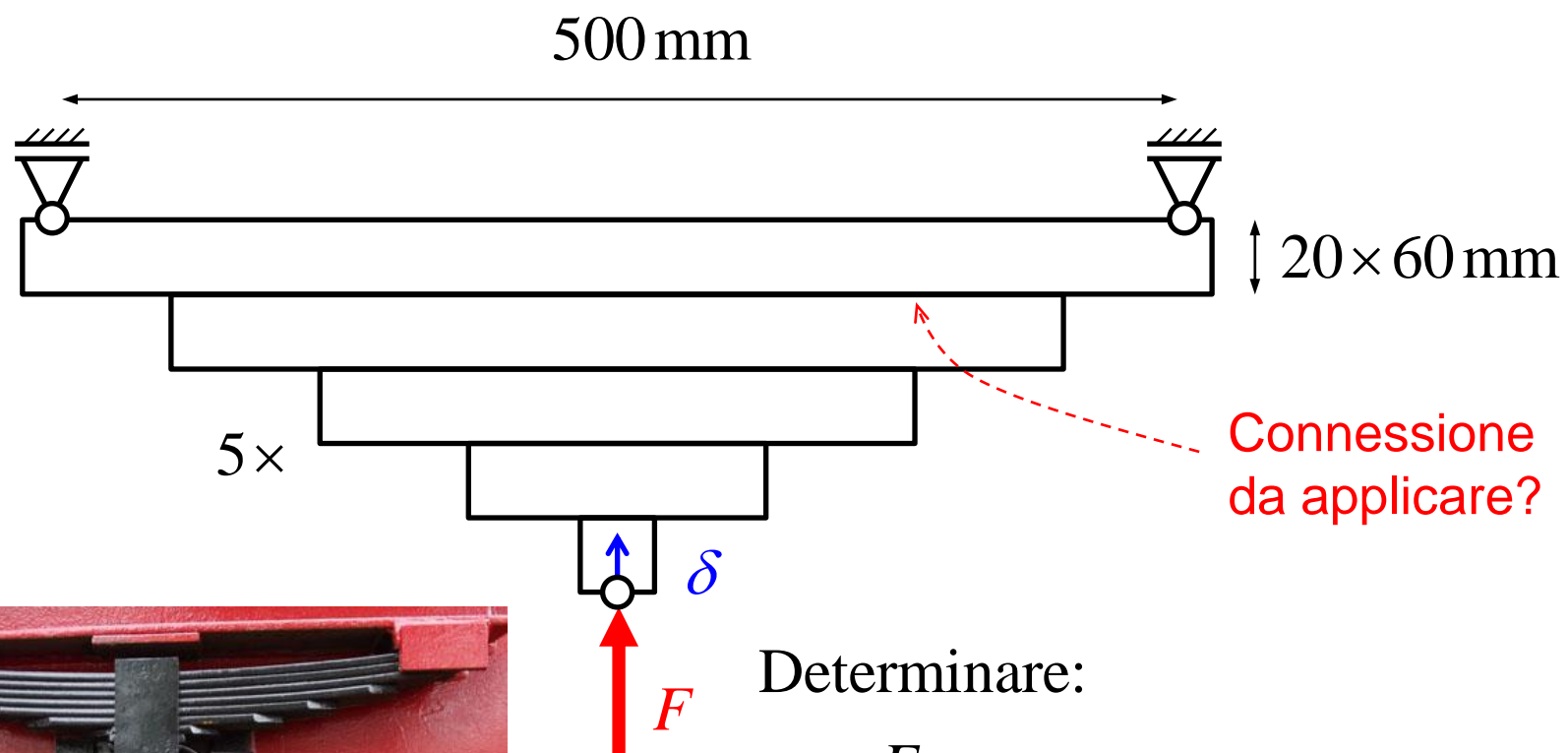
1. Connessioni:

- Vincolato Lineare
- Senza separazione Lineare
- Senza attrito Non lineare
- Rugoso (attrito infinito) Non lineare
- Con attrito Non lineare

2. Definizione proprietà elastiche materiale.

3. Reazione vincolare/connessione: forza, momento.

Esercitazione da svolgere:
Determinare la rigidezza di una molla a balestra



Determinare:

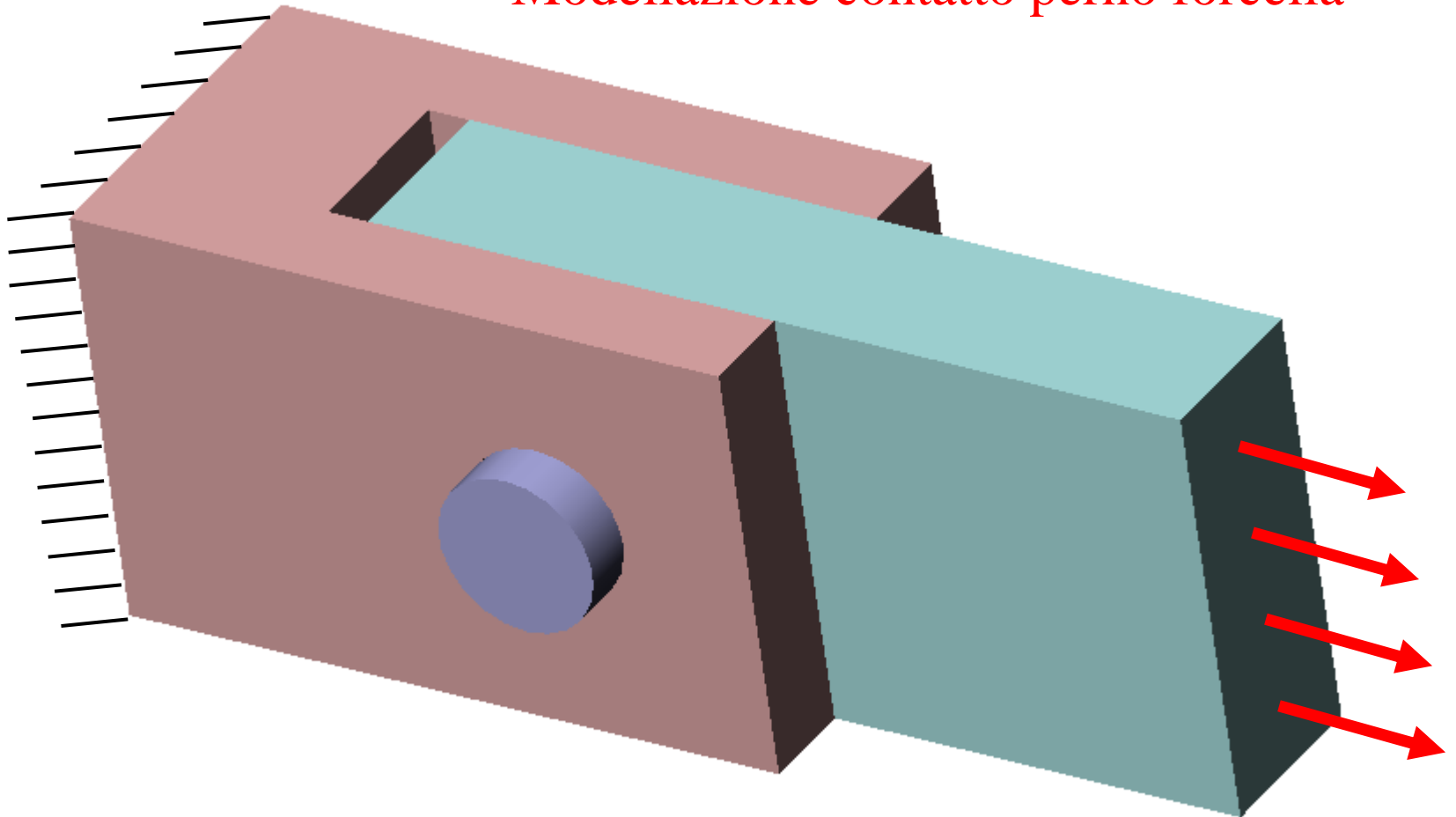
$$K = \frac{F}{\delta}$$

Modello trave uniforme resistenza:

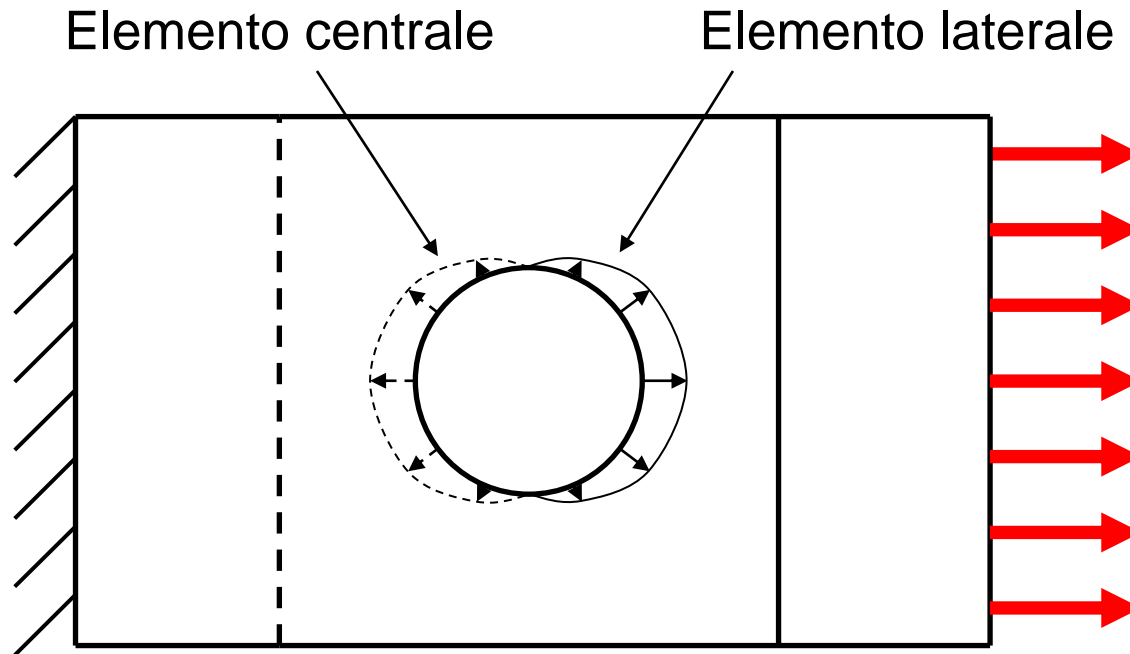
$$K = \frac{8}{3} \frac{E n b h^3}{L^3} = 10.2 \times 10^3 \text{ N/mm}$$

Modulo di Young considerato:
 $E = 200 \text{ GPa}$

Modellazione contatto perno forcella



Distribuzione di pressione sulla sup. laterale del perno

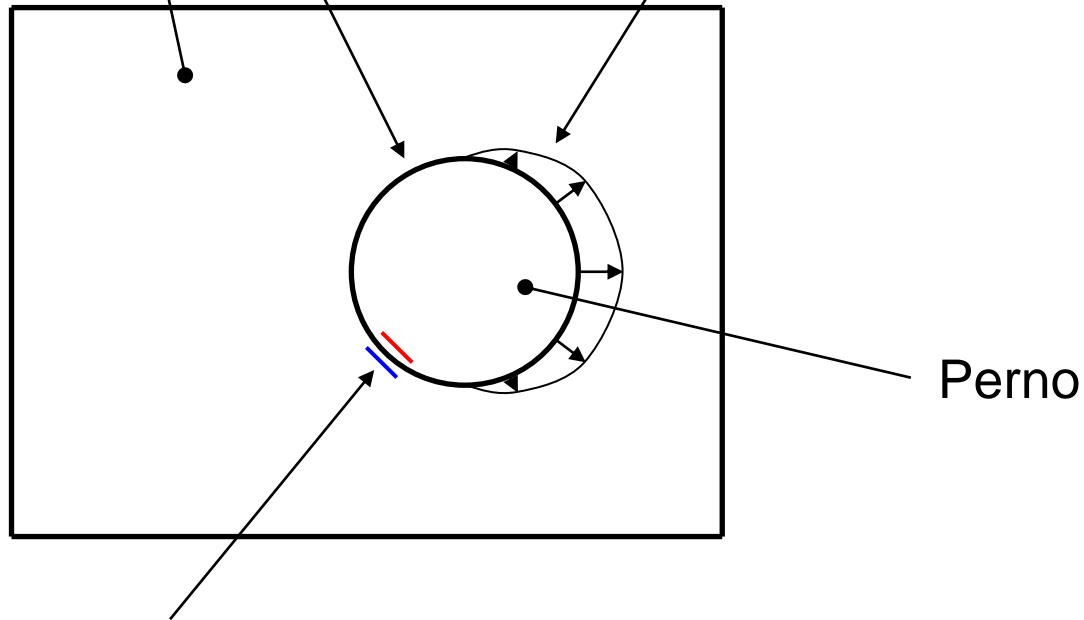


Distribuzione di pressione unilaterale

Elemento laterale

Zona scarica

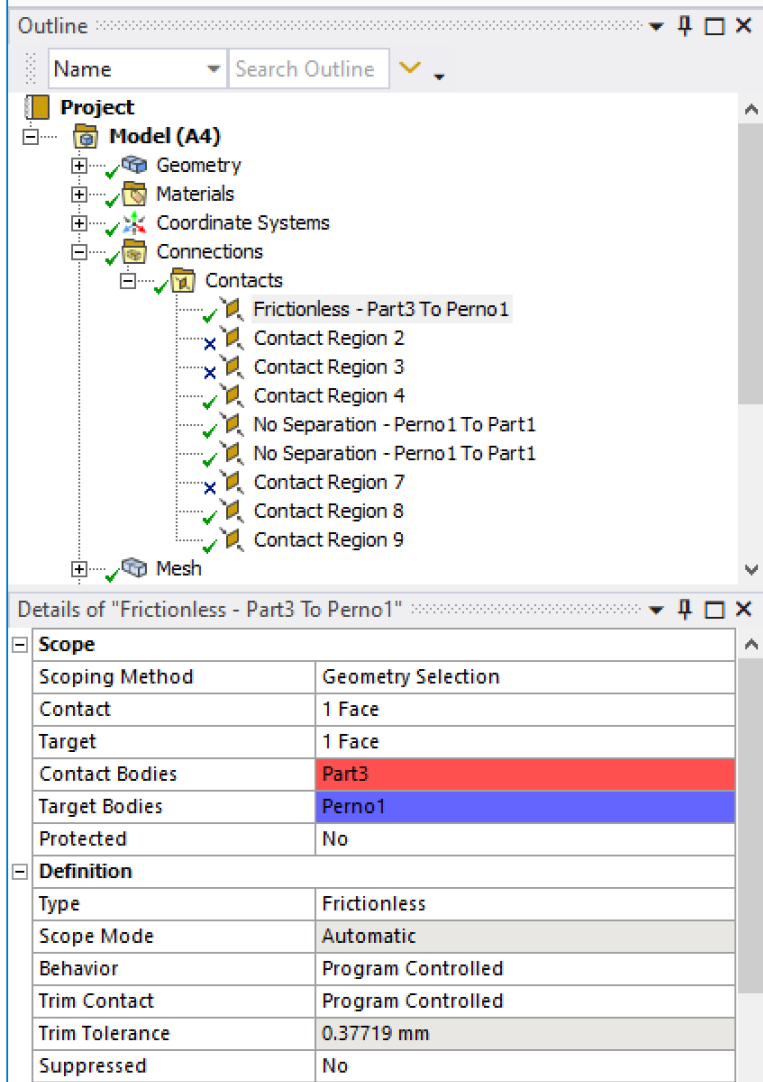
Zona caricata



Connessione, perno/forcella,
necessario impostare “Contatto senza attrito” o “Frictionless”

Connessioni



Prima soluzione: Impostare “Contatto senza attrito” (“Frictionless”) (*solo su una connessione*)

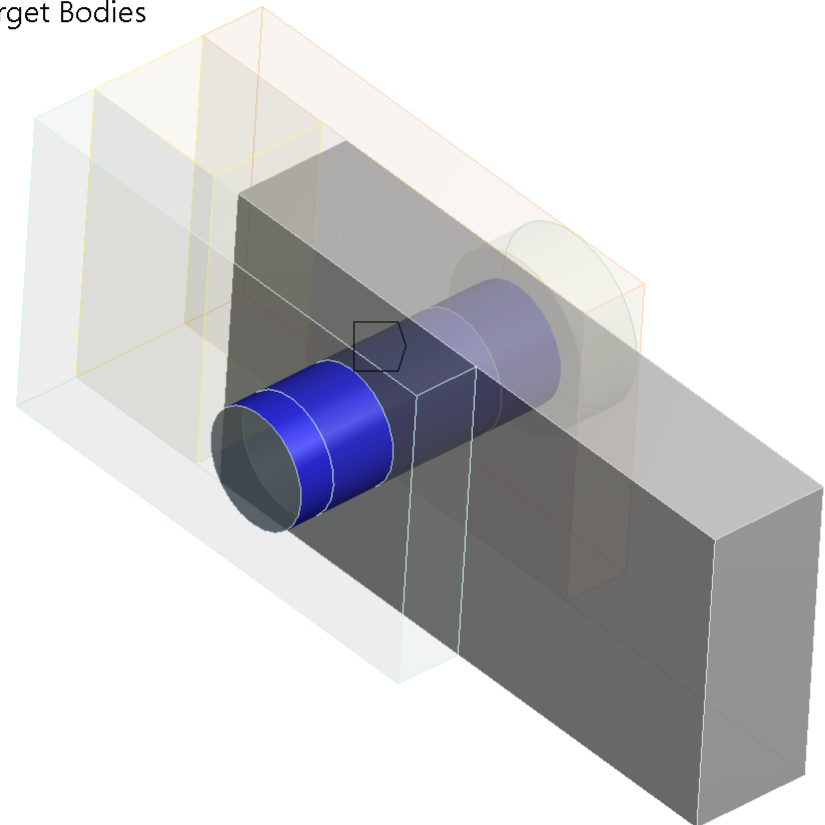


The screenshot shows the CAD software interface. The **Outline** panel on the left lists the project structure, including **Model (A4)** with sub-items like **Geometry**, **Materials**, **Coordinate Systems**, **Connections**, **Contacts**, and **Mesh**. Under **Contacts**, several contact regions are listed, with **Frictionless - Part3 To Perno1** selected. The **Details of "Frictionless - Part3 To Perno1"** panel on the right shows the following settings:

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Part3
Target Bodies	Perno1
Protected	No
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.37719 mm
Suppressed	No

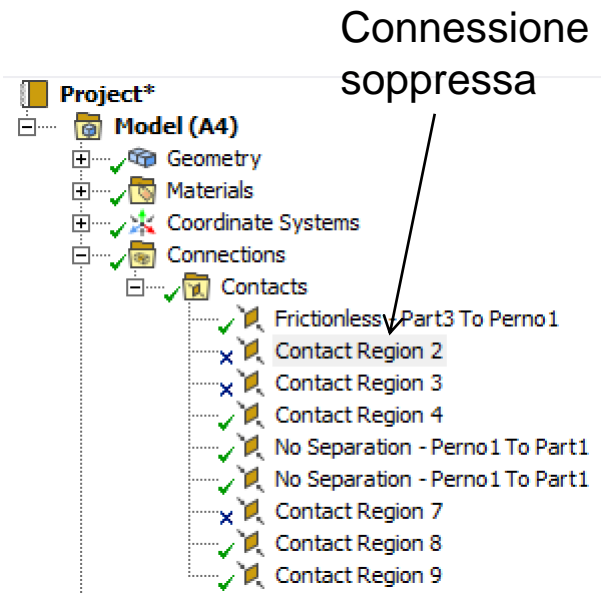
Frictionless - Part3 To Perno1

-  Contact Bodies
-  Target Bodies



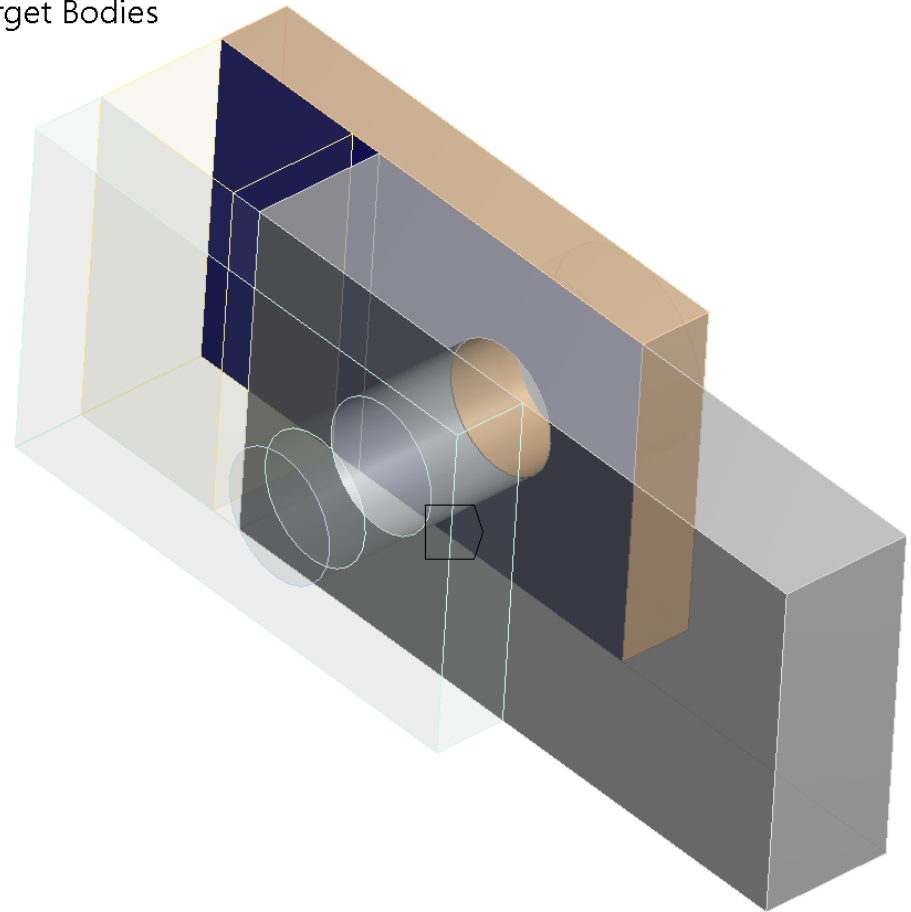
Connessioni

Esclusione delle connessioni laterali

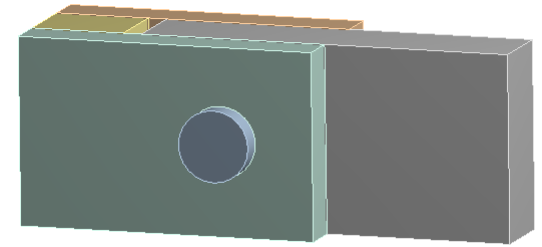
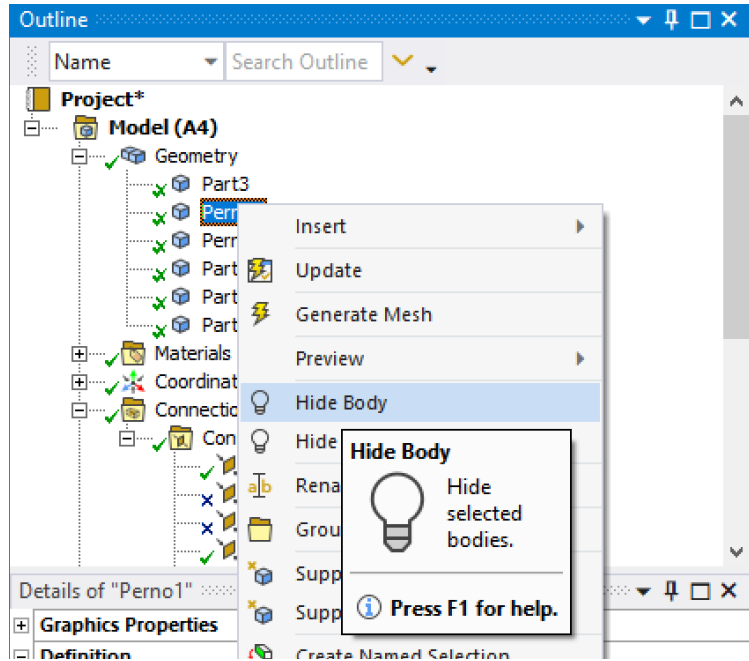


Contact Region 2

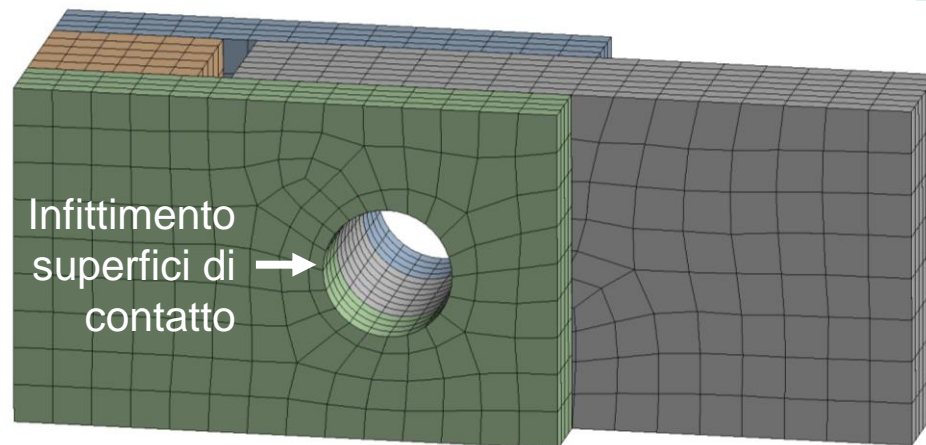
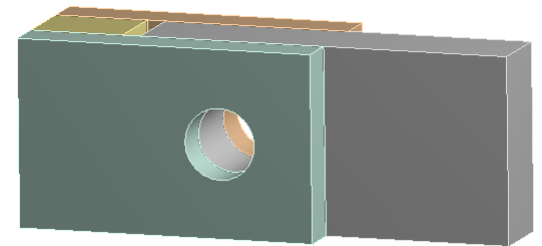
- Contact Bodies
- Target Bodies



Contatto, necessità di una mesh accurata



Temporanea esclusione (grafica) di un componente per esempio per facilitare una selezione



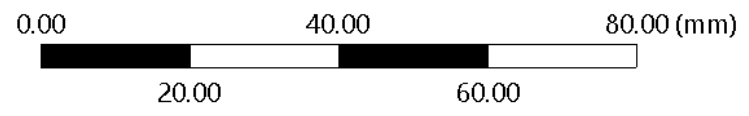
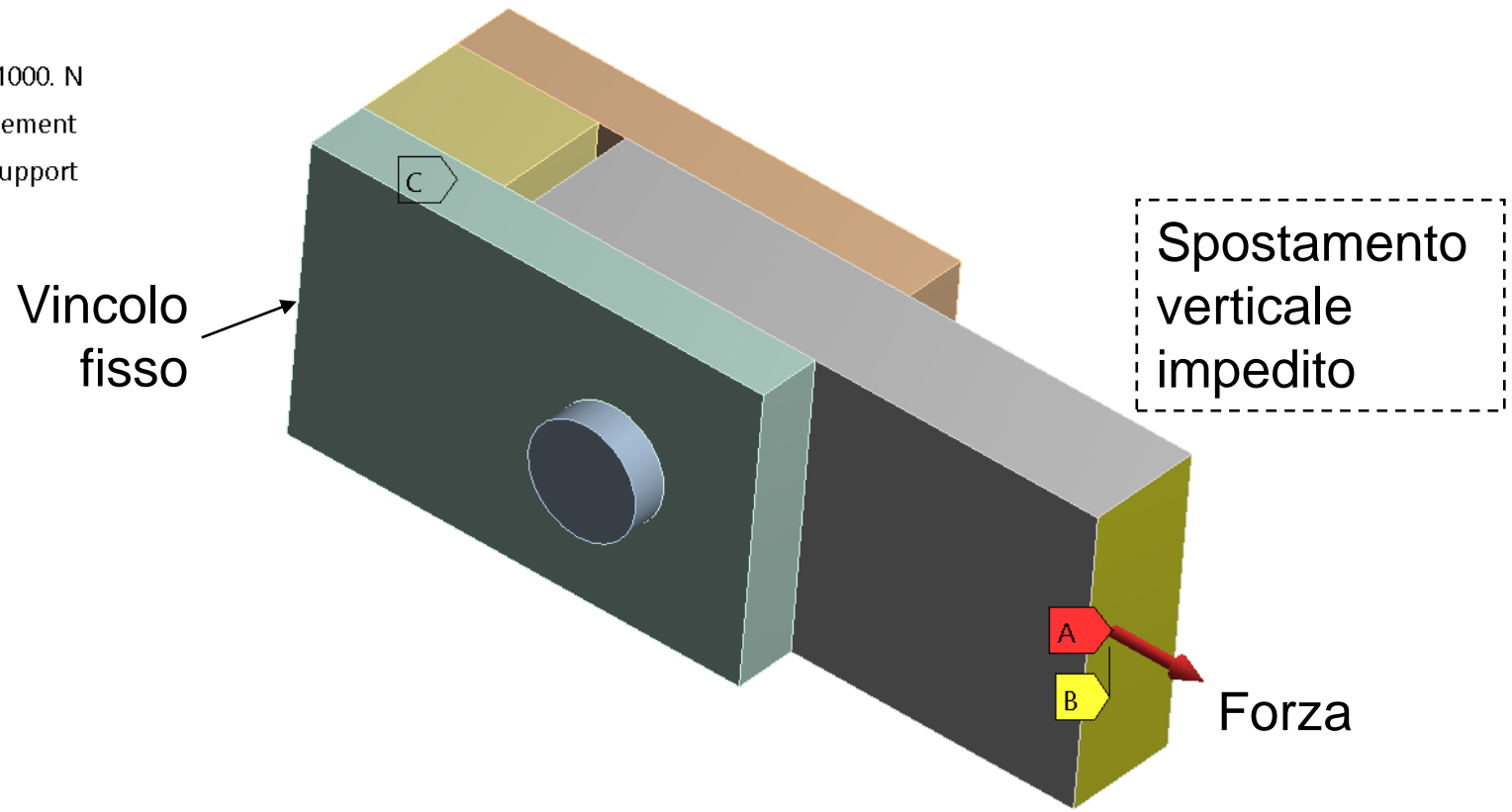
Vincoli e carichi

A: Static Structural

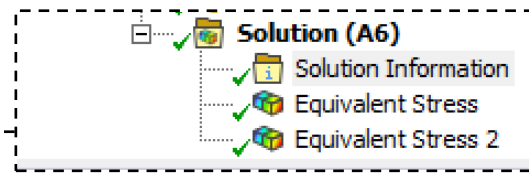
Static Structural

Time: 1. s

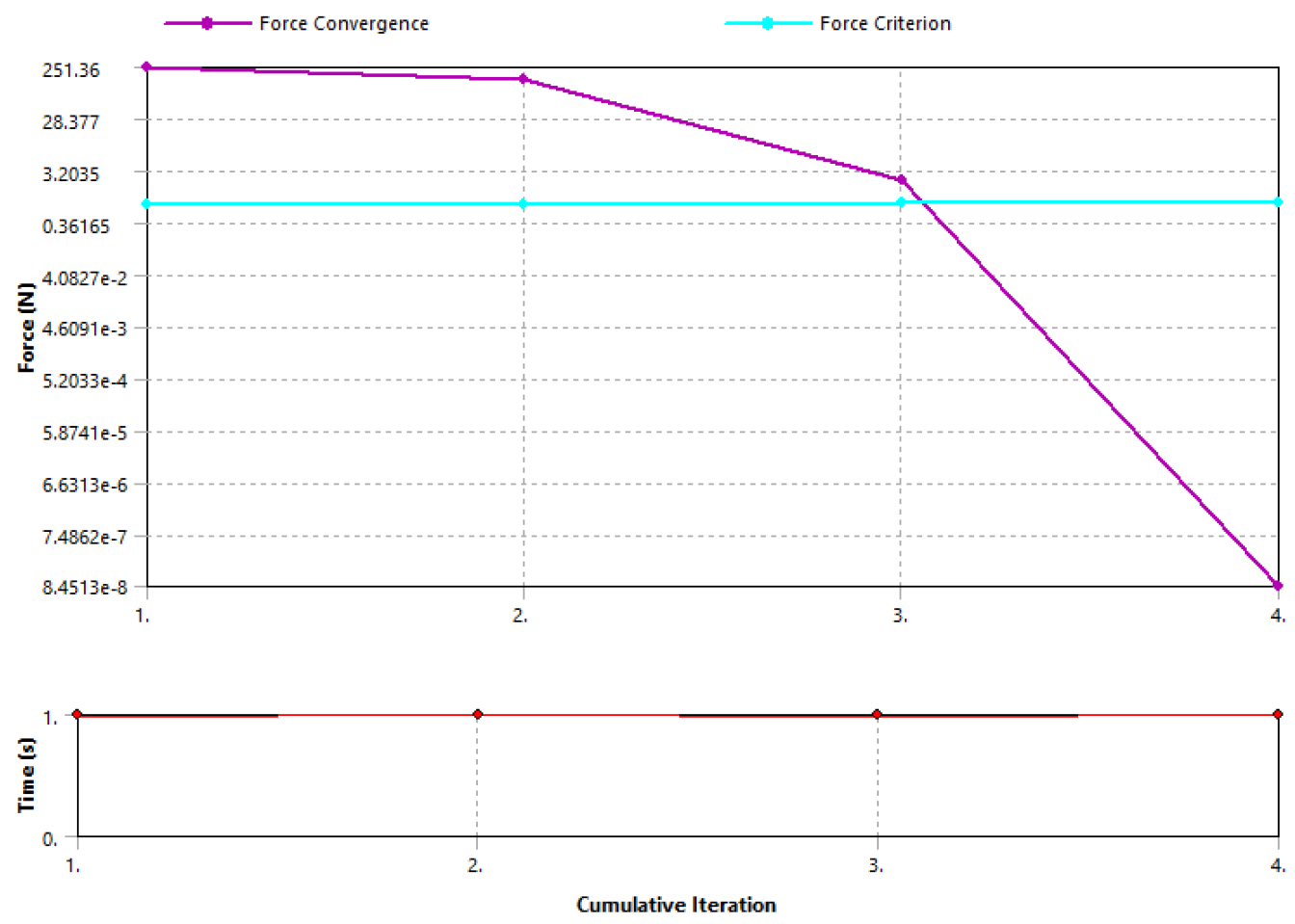
- A** Force: 1000. N
- B** Displacement
- C** Fixed Support



Iterazioni di convergenza



Force Convergence



Contatto localizzato ad una sola connessione

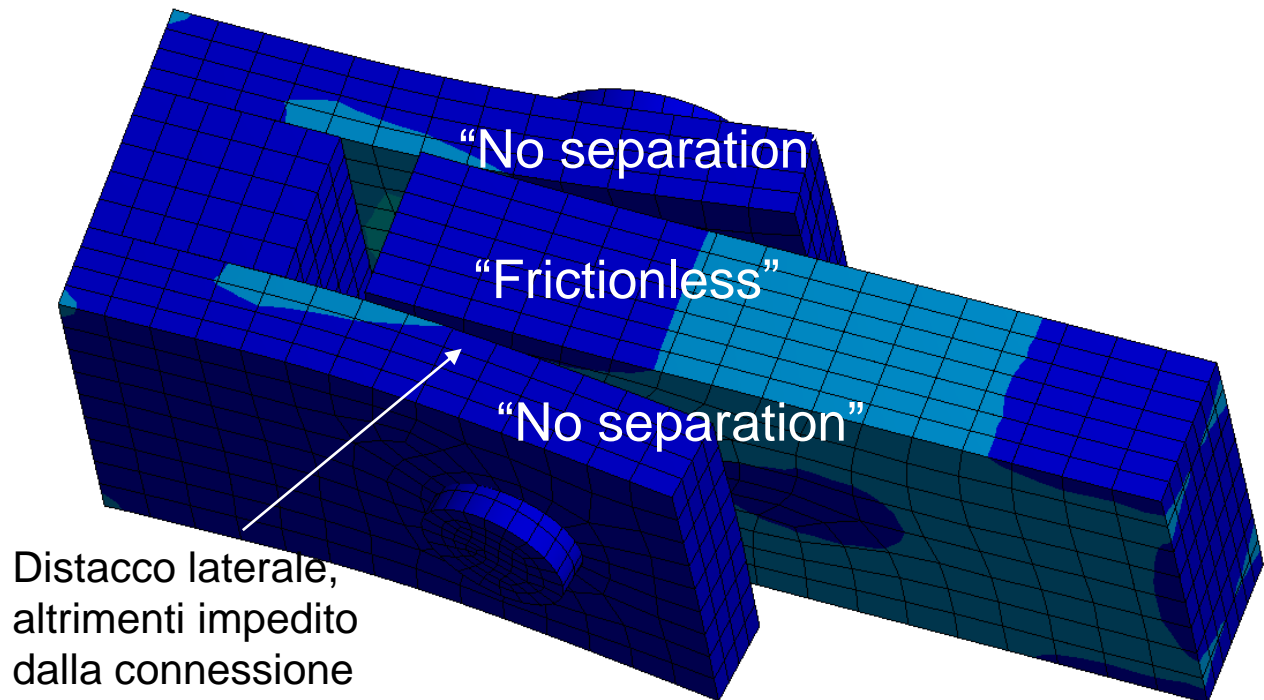
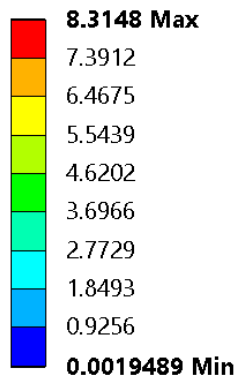
A: Static Structural

Equivalent Stress

Type: Equivalent (von-Mises) Stress

Unit: MPa

Time: 1



Contatto localizzato ad una sola connessione

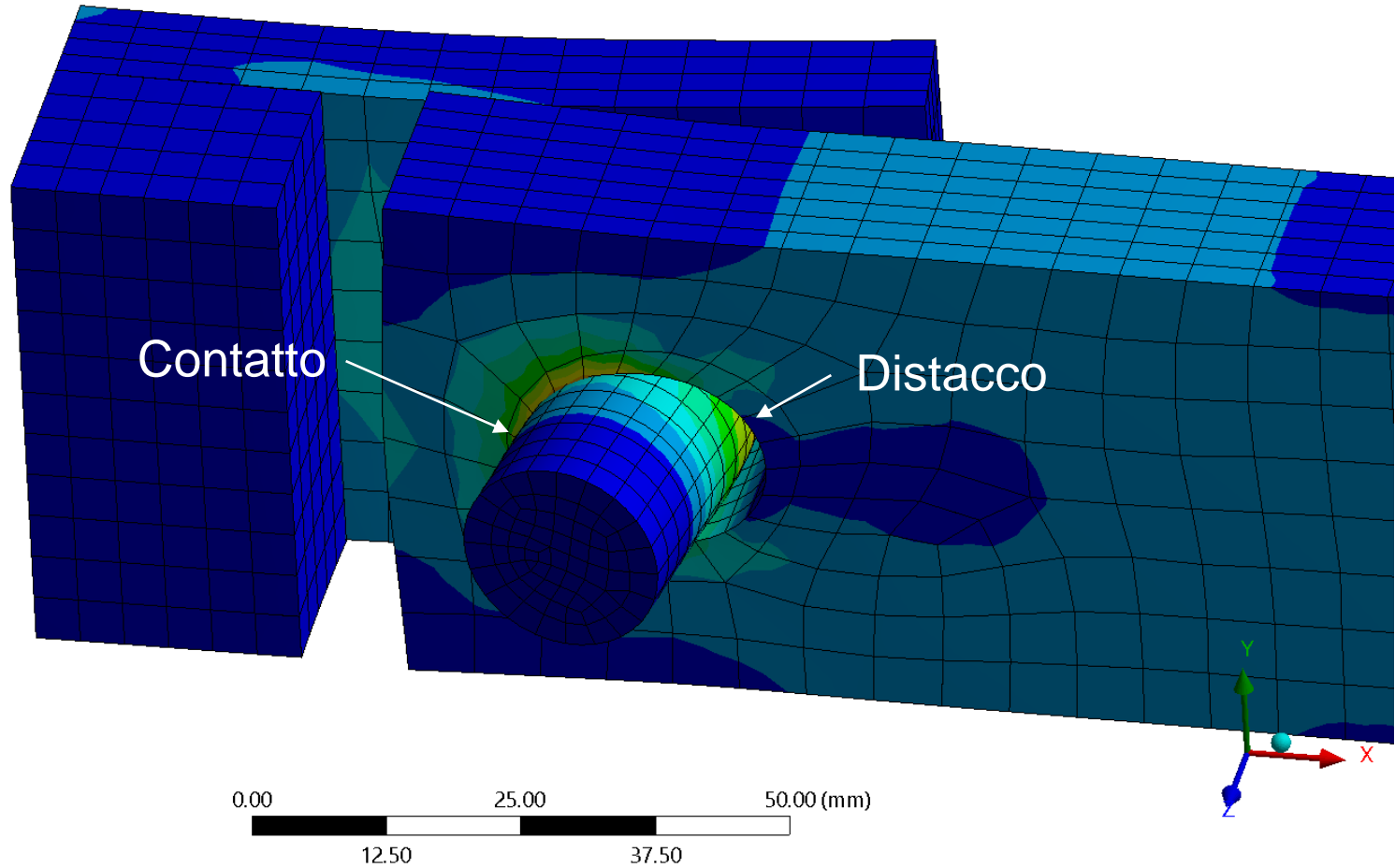
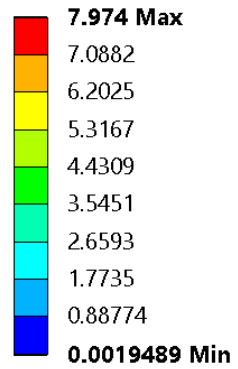
A: Static Structural

Equivalent Stress 2

Type: Equivalent (von-Mises) Stress

Unit: MPa

Time: 1



Possibilità di evitare non-linearità di contatto

Connessione con contatto (in questo caso senza attrito) implica **non linearità** e quindi si ha il processo iterativo di convergenza.

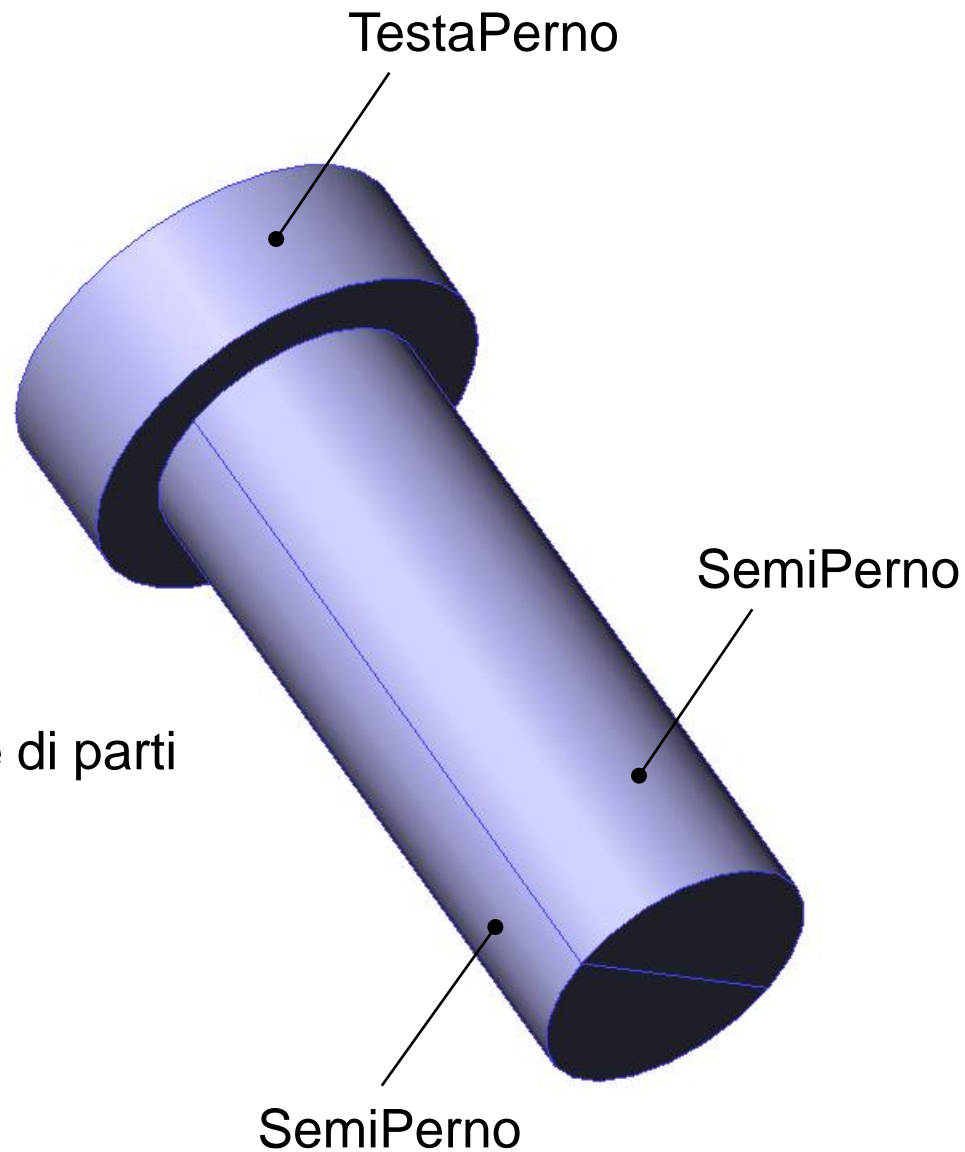
Essendo superfici conformi è possibile prevedere le zone di contatto ed escludere le porzioni scariche.

È possibile operare in due modi:

a) **Suddividere una o più parti** in sottoparti, in modo da “spezzare” la connessione di partenza

b) Oppure **suddividere la superficie** interessata al contatto, in più superfici, gestire le connessioni, e proseguire come nel caso precedente

a) Suddivisione del perno

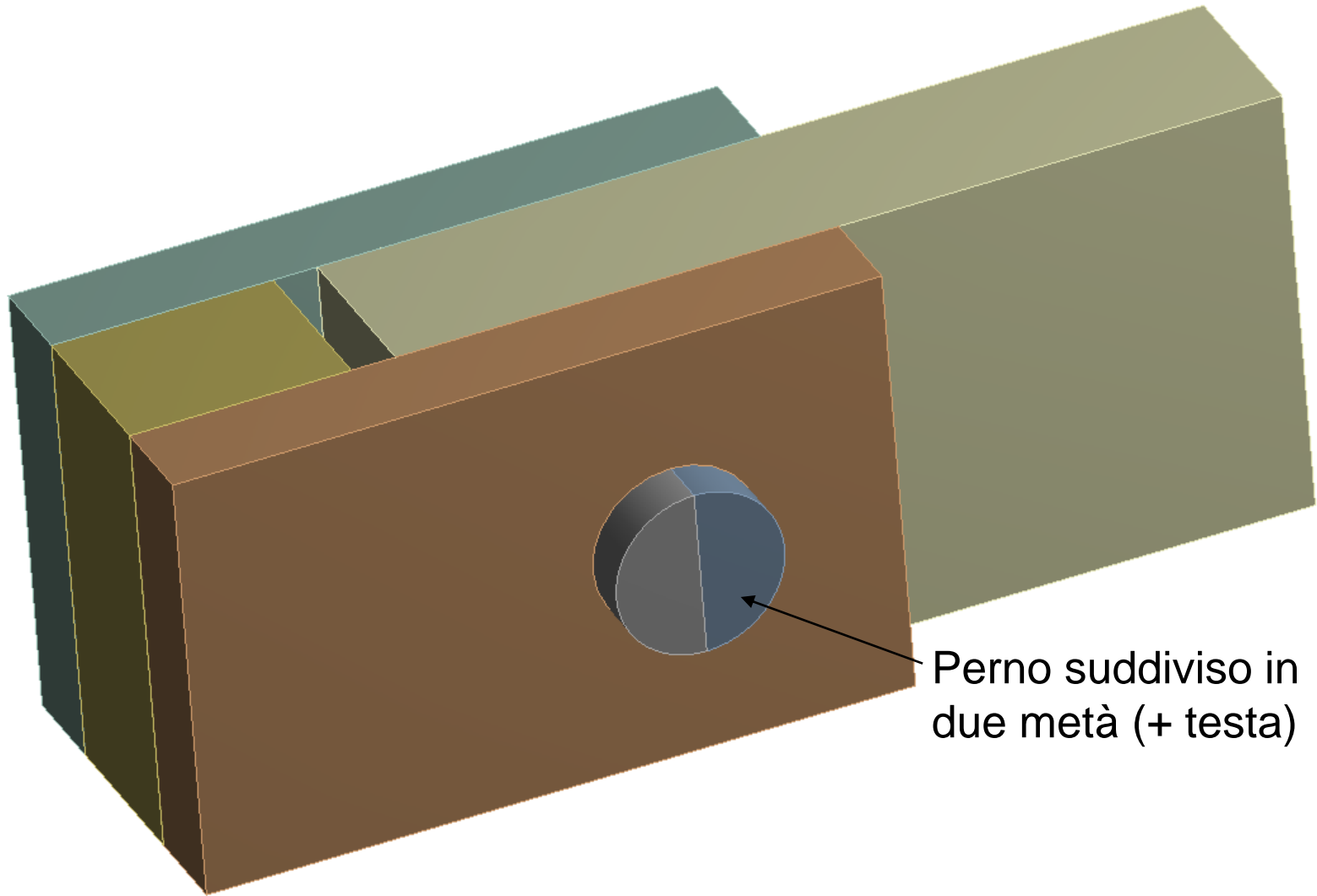


Perno come assieme di parti
(modellazione CAD):

- TestaPerno
- SemiPerno
- SemiPerno

Import in Ansys Wb

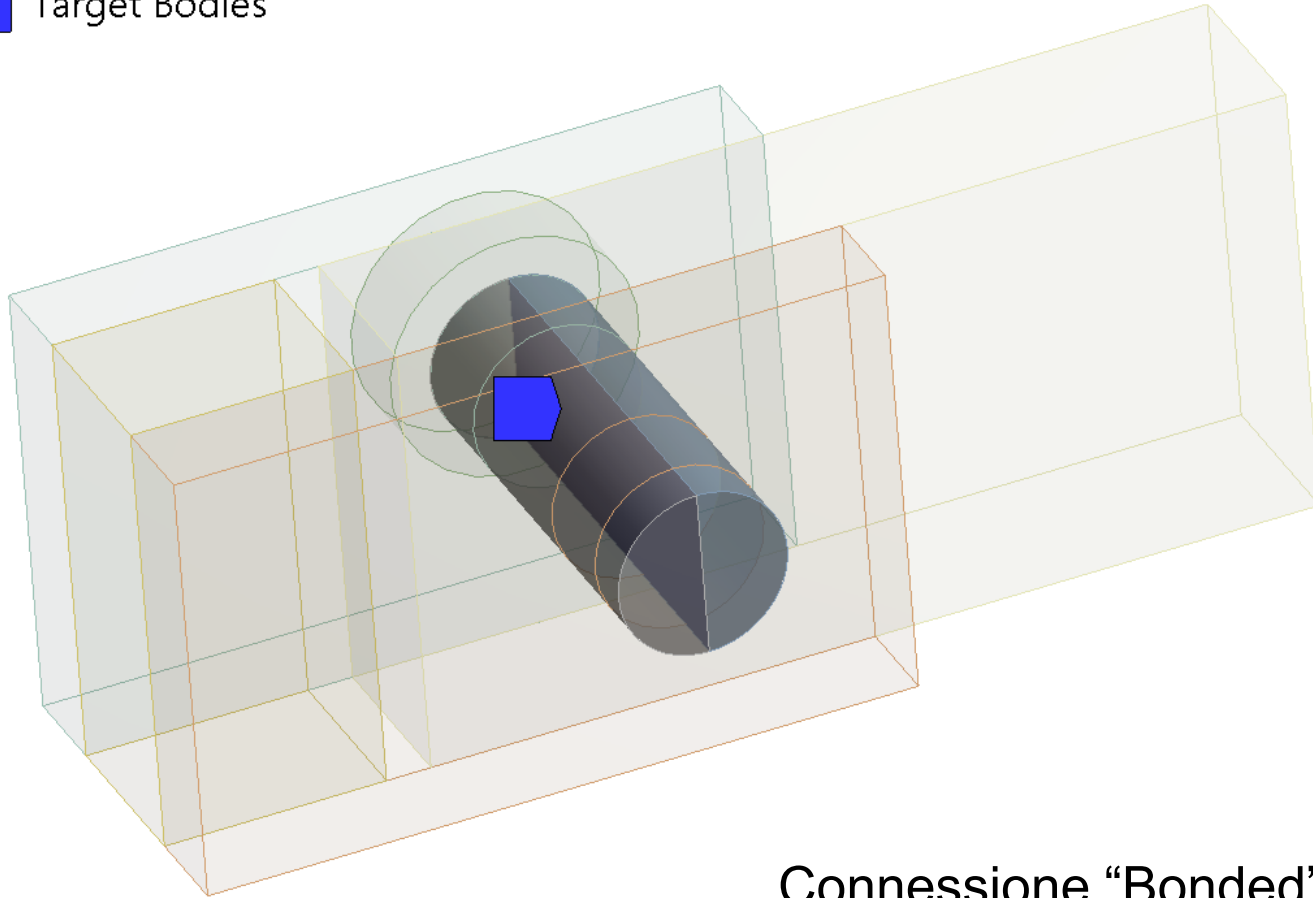
Perno diviso in più parti



Perno suddiviso in due metà (+ testa)

Contact Region 5

- Contact Bodies
- Target Bodies



Connessione "Bonded" fra le due metà (fittizie) dello stelo del perno

Connessioni

Outline

Name Search Outline

- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Contacts
 - No Separation - Part3 To SemiPerno
 - Contact Region 2
 - Contact Region 5
 - Contact Region 6
 - Contact Region 7
 - Contact Region 8
 - Contact Region 9
 - No Separation - SemiPerno To Part1
 - No Separation - SemiPerno To Part1
 - No Separation - TestaPerno To Part1
 - Contact Region 13
 - Contact Region 14

Details of "Contact Region 7"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	SemiPerno
Target Bodies	Part1
Protected	No

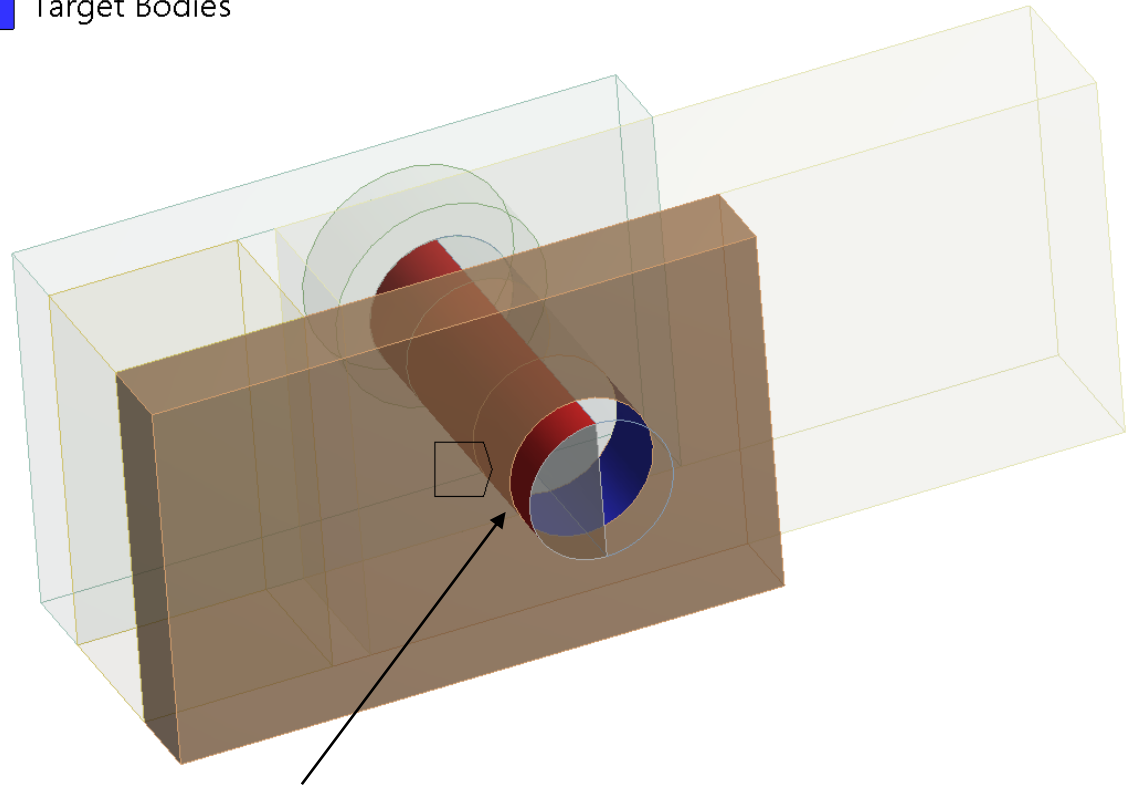
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.37719 mm
Suppressed	No

Advanced	
Formulation	Yes
Small Sliding	Off

Ready

Contact Region 7

- Contact Bodies
- Target Bodies

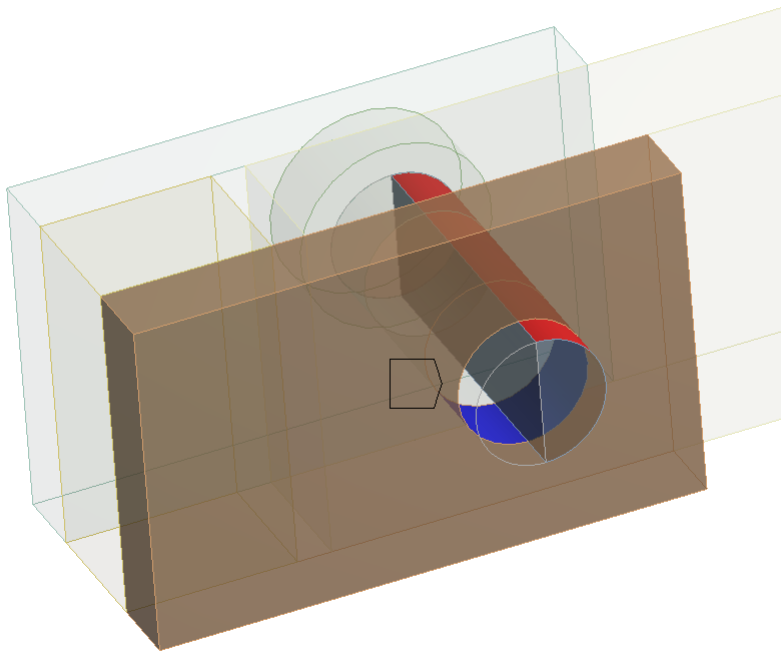


Connessione da escludere (suppressed),
il contatto si sviluppa dall'altra parte

Connessioni

No Separation - SemiPerno To Part1

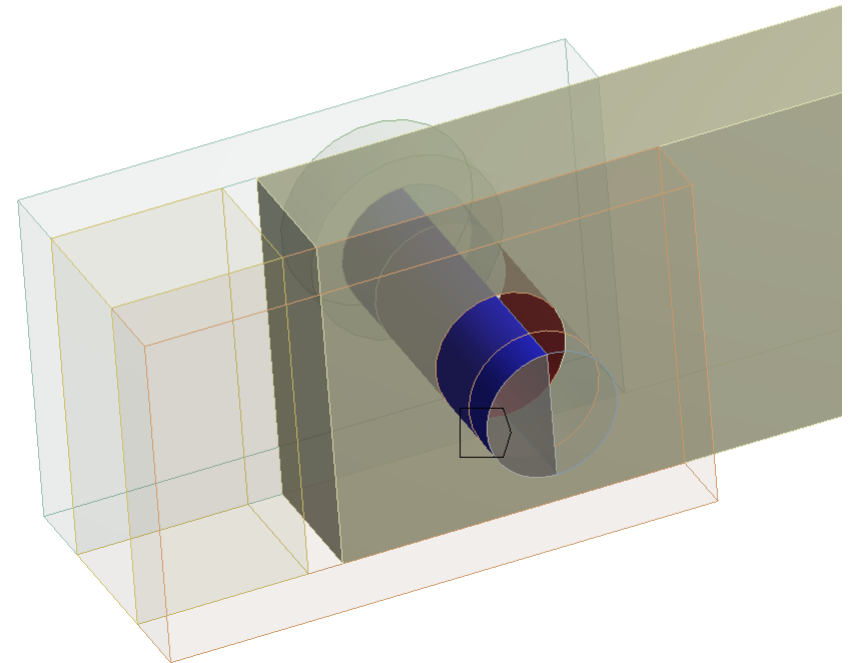
- Contact Bodies
- Target Bodies



Connessione da mantenere
("No separation")

No Separation - Part3 To SemiPerno

- Contact Bodies
- Target Bodies



Connessione da mantenere
("No separation")

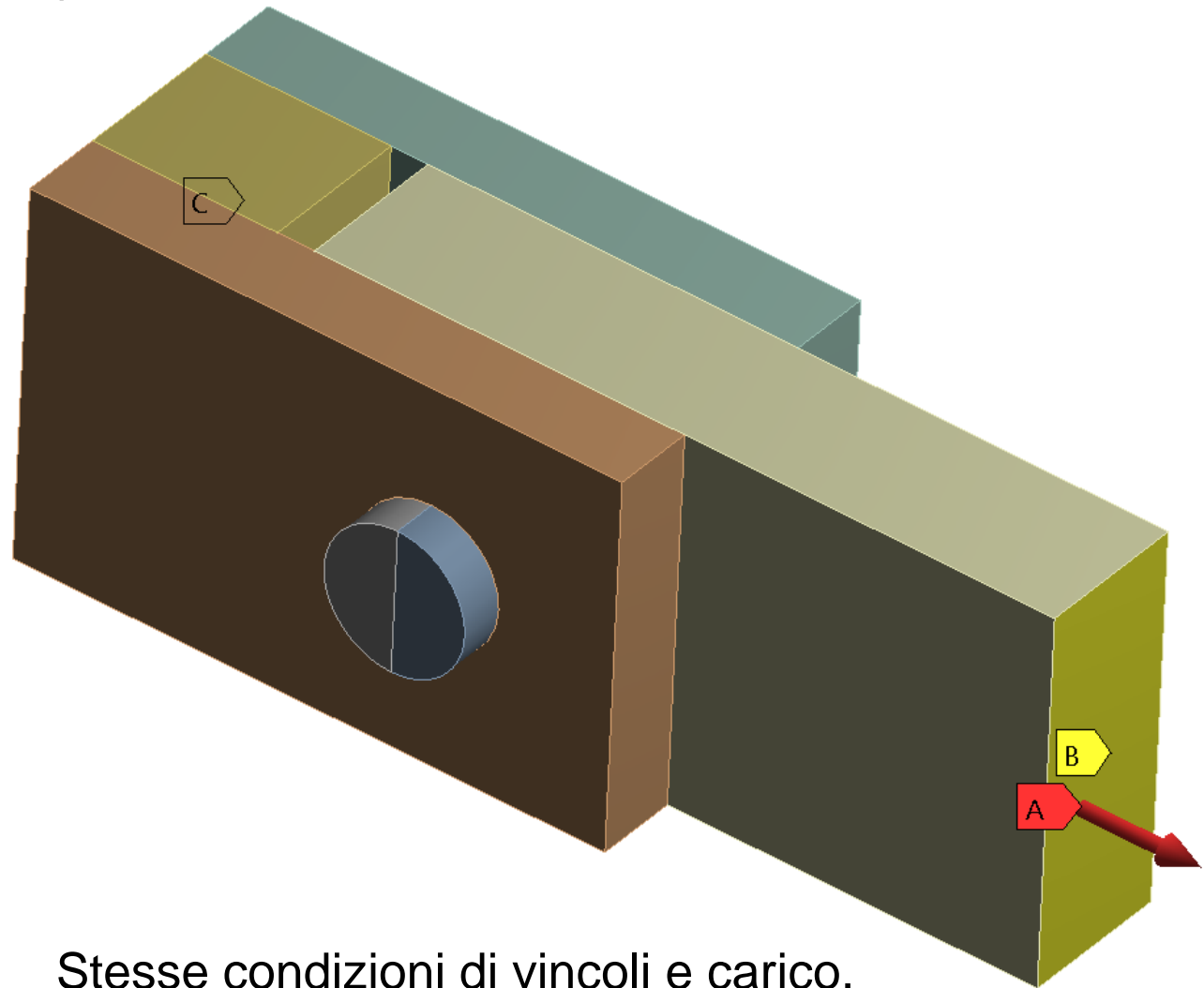
Vincoli e carichi

A: Static Structural (ANSYS)

Static Structural

Time: 1. s

- A** Force: 1000. N
- B** Displacement
- C** Fixed Support



Stesse condizioni di vincoli e carico,
vincolo direz. verticale, per eliminare
la labilità, oltre alle Weak Springs

Distacco sulle superfici con connessione soppressa

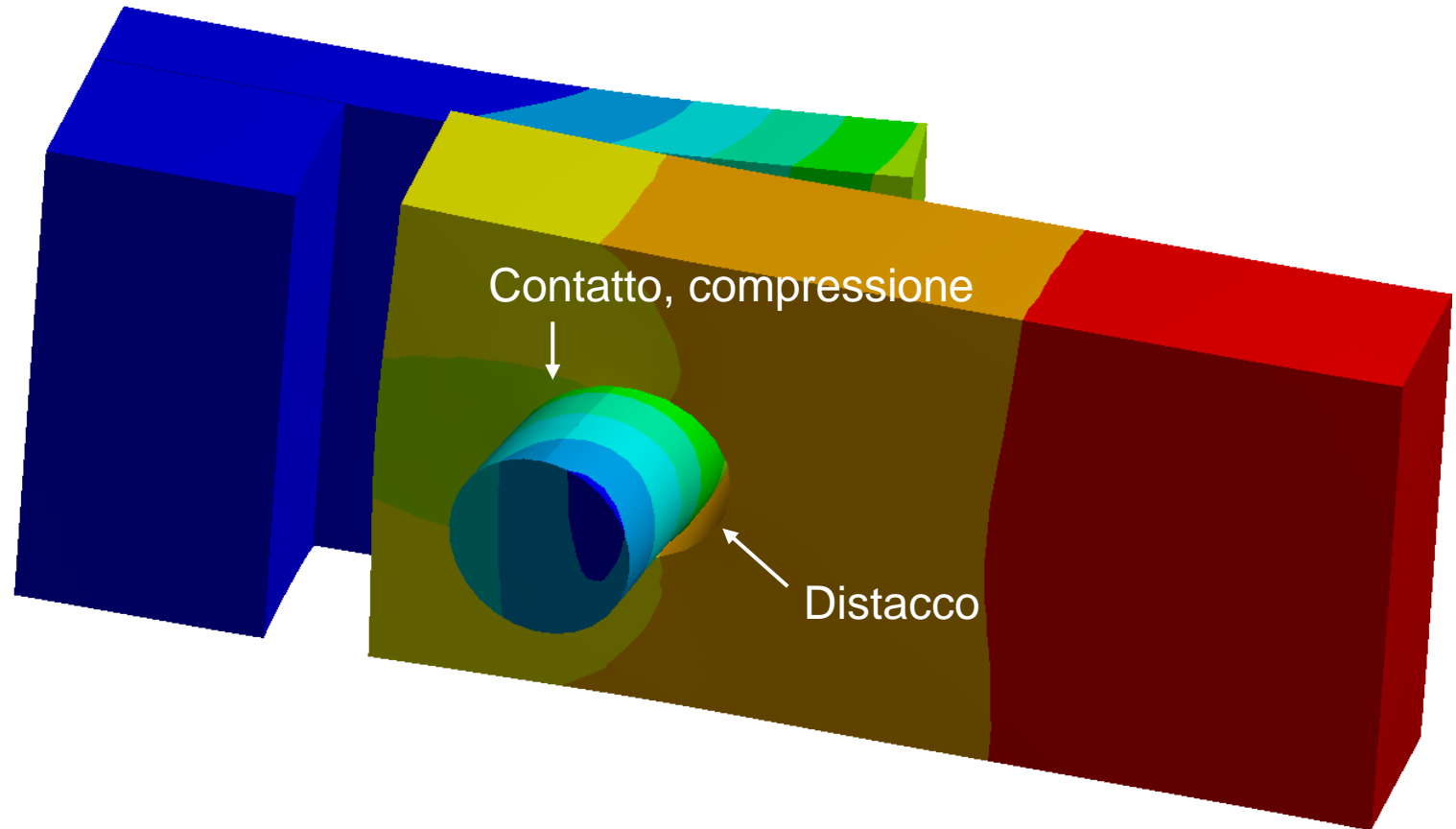
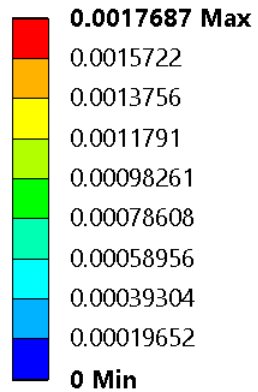
A: Static Structural (ANSYS)

Total Deformation

Type: Total Deformation

Unit: mm

Time: 1



Analisi dei risultati

Contatto caricato, tensione equivalente von Mises

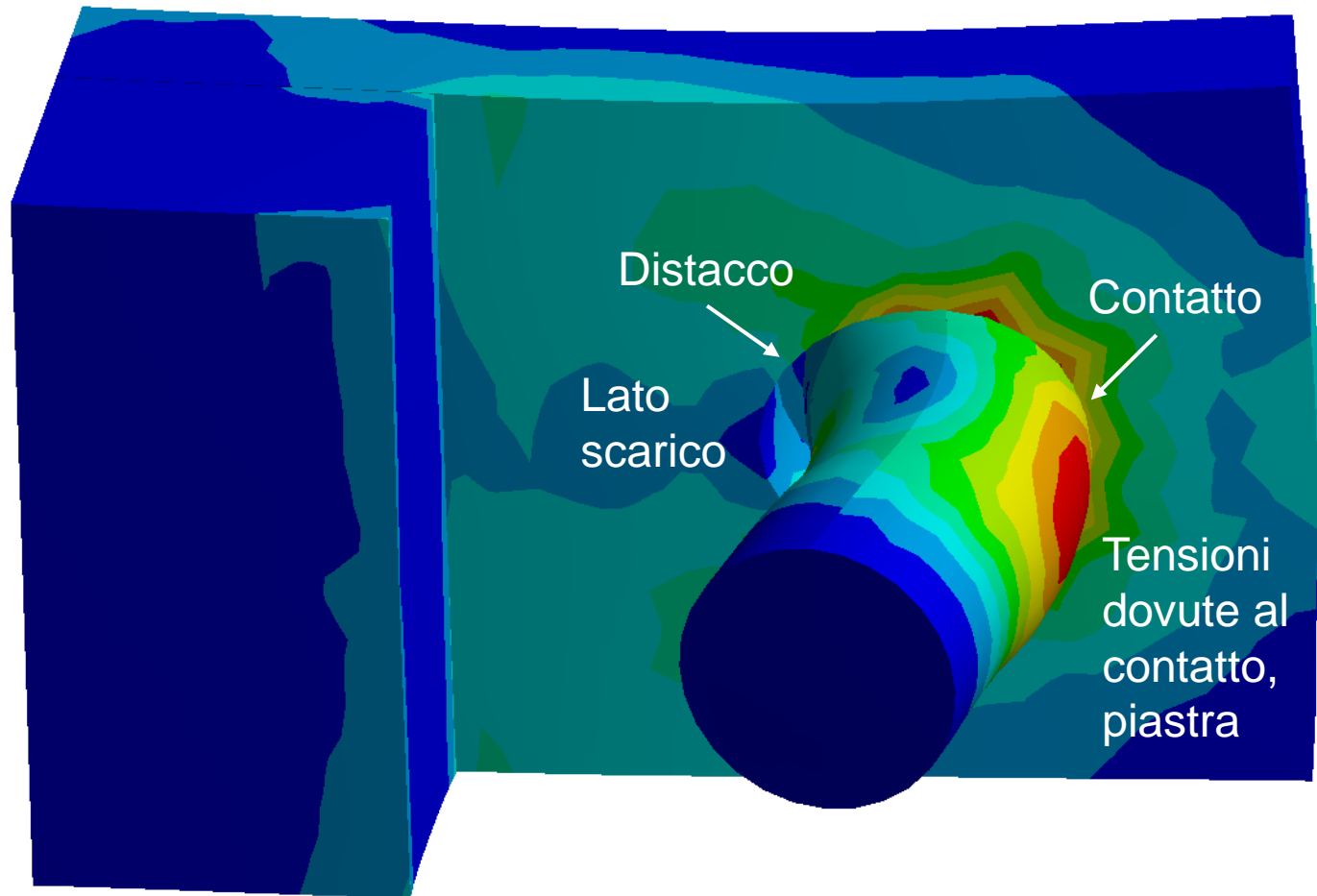
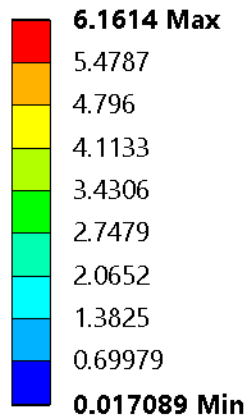
A: Static Structural (ANSYS)

Equivalent Stress 3

Type: Equivalent (von-Mises) Stress

Unit: MPa

Time: 1



Tensione equivalente von Mises del perno

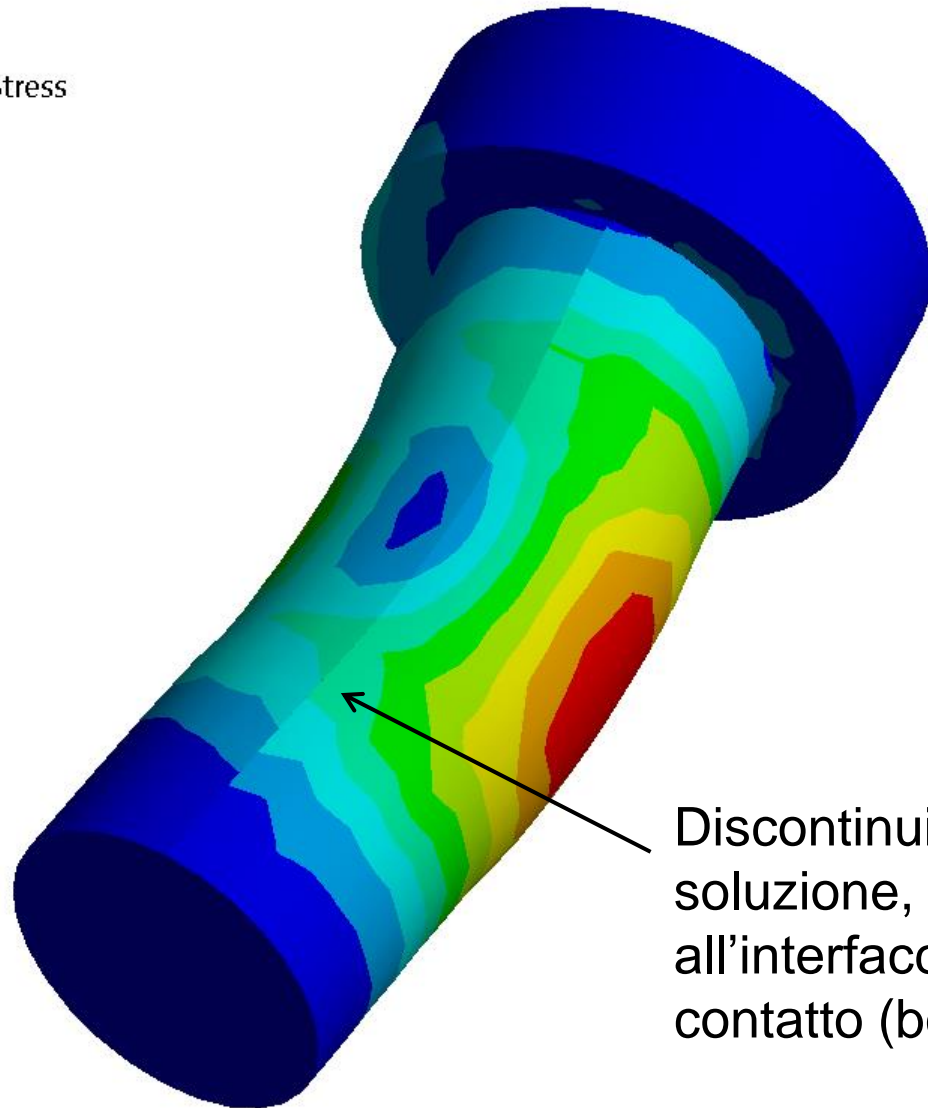
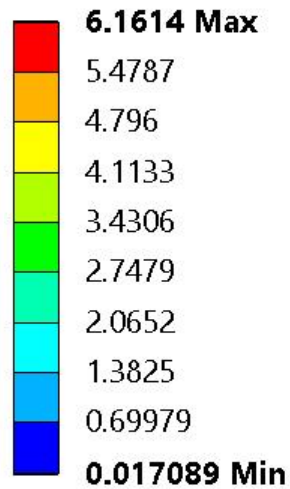
A: Static Structural (ANSYS)

Equivalent Stress 2

Type: Equivalent (von-Mises) Stress

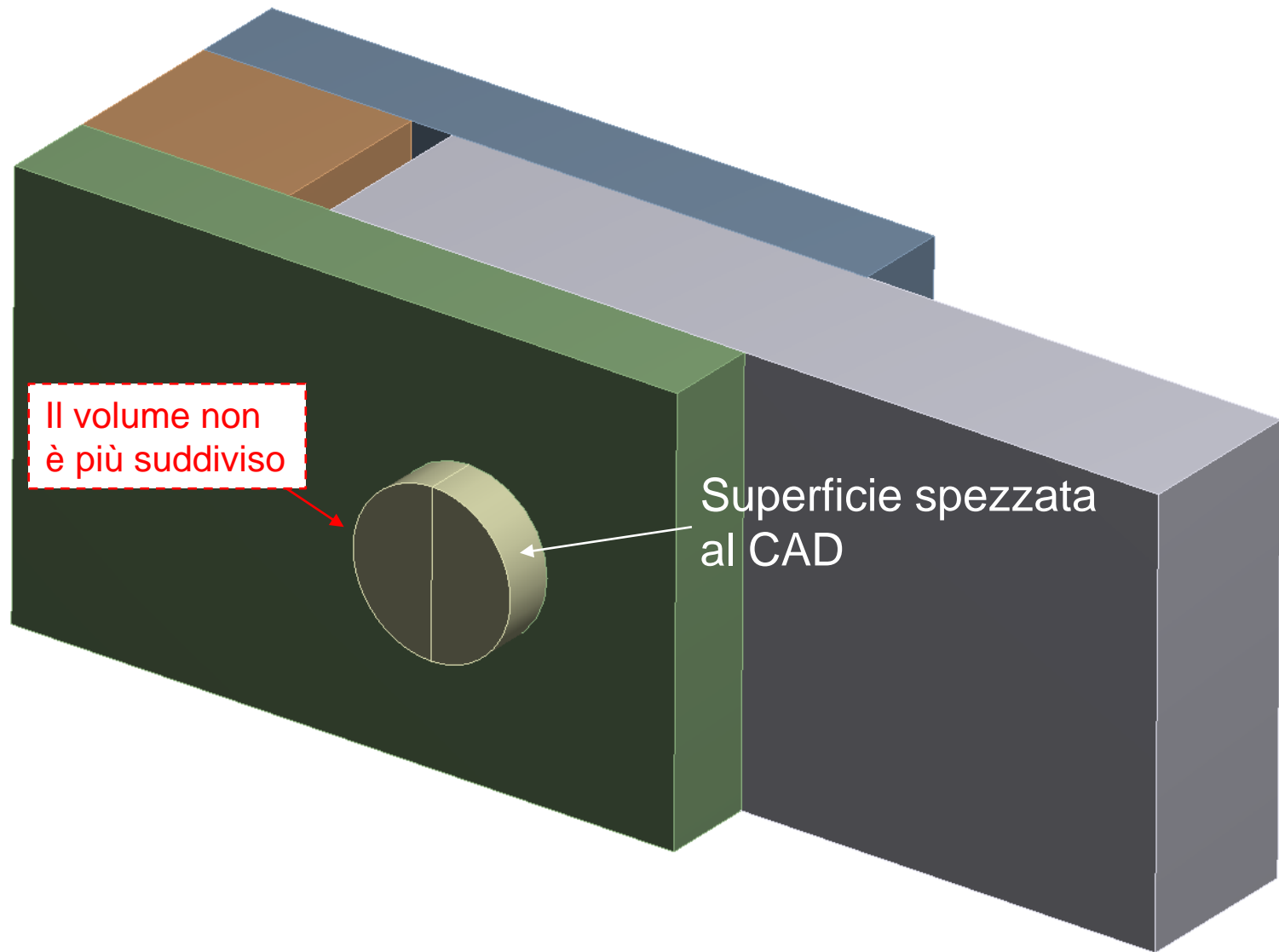
Unit: MPa

Time: 1



Discontinuità della
soluzione, dovuta
all'interfaccia di
contatto (bonded)

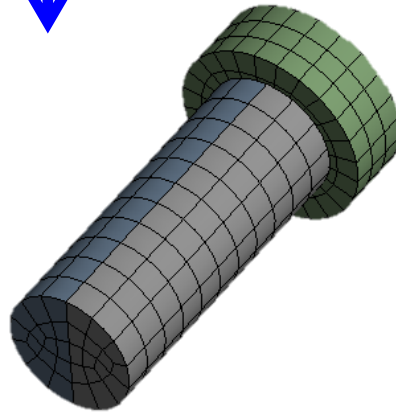
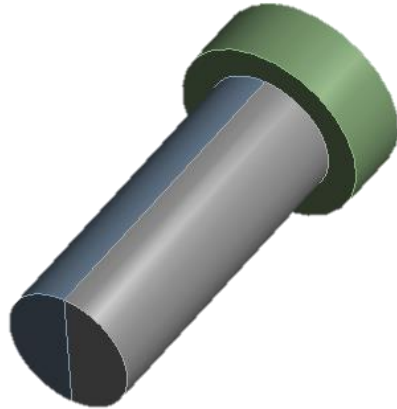
b) Suddivisione delle superfici del perno



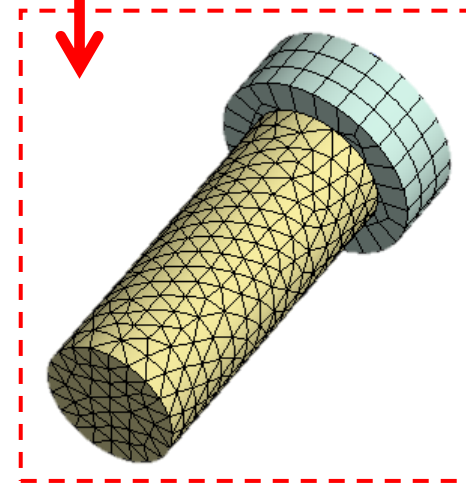
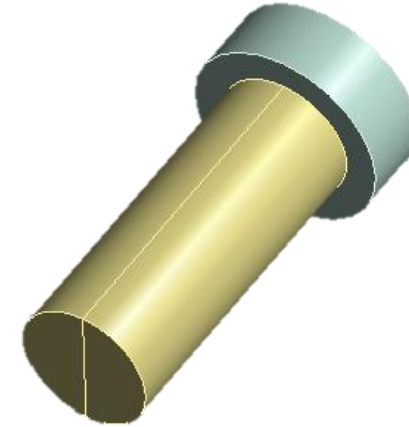
Mesh Perno a tetraedri

Geometria di estrusione:
"sweepable"
body (mesh a
esaedri, OK)

Volumi separati

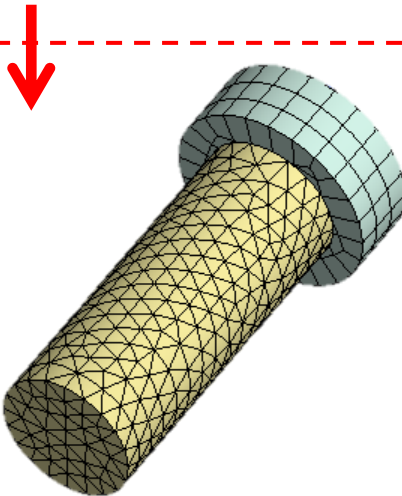
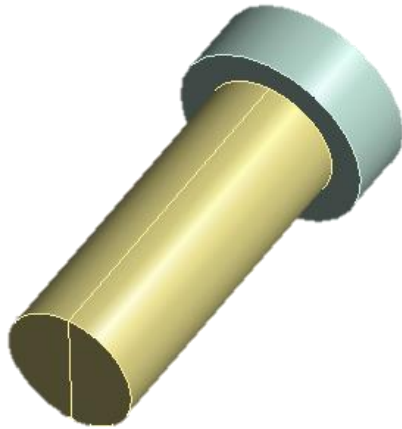


Area spezzata

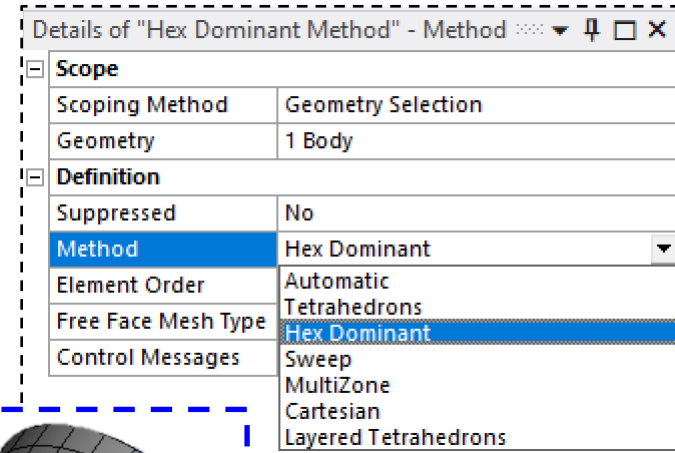
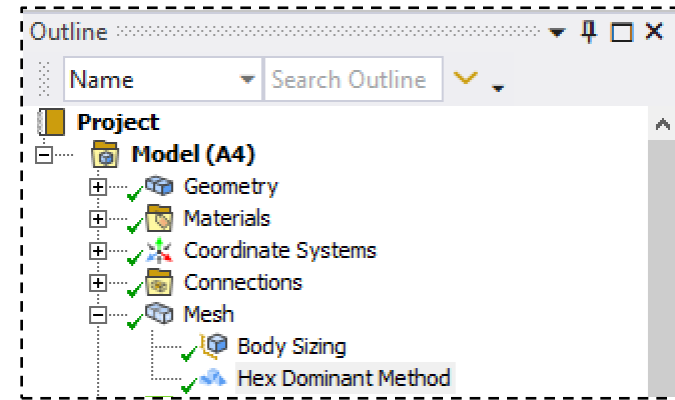
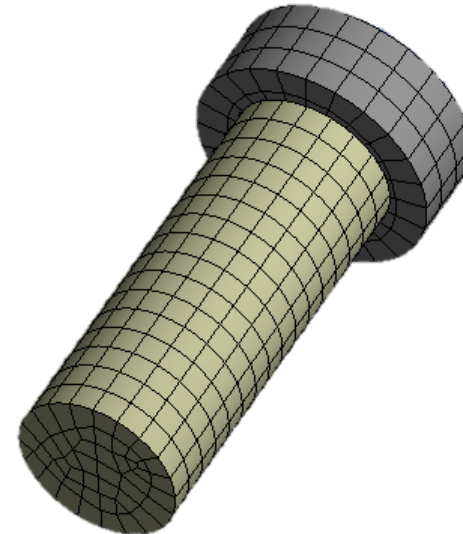


"Not sweepable" body (mesh a
tetraedri, minore accuratezza)

Mesh Perno a tetraedri



Nuova opzione
di Mesh:
Method →
Hex Dominant



Connessioni

Outline

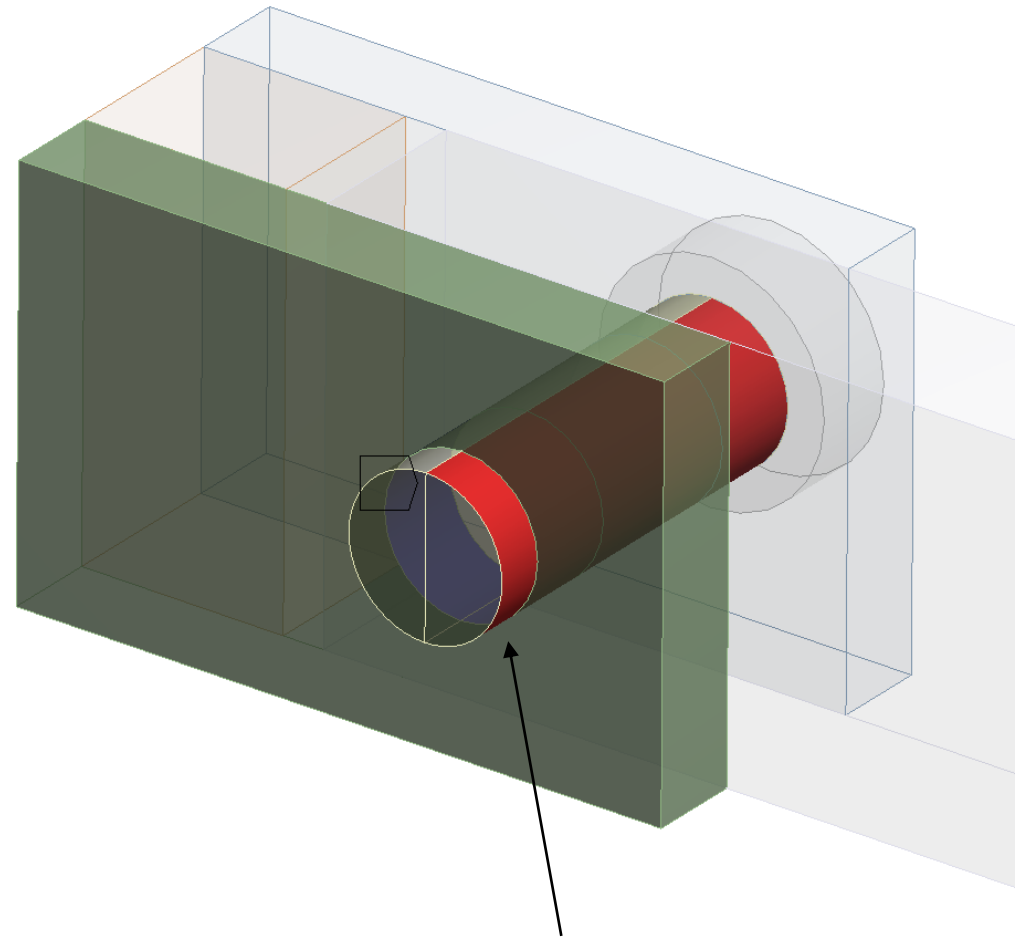
Name Search Outline

- Project*
 - Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Contacts
 - Bonded - Part3 To Perno1
 - Contact Region 2
 - Contact Region 3
 - Contact Region 4
 - No Separation - Perno1 To Part1
 - No Separation - Perno1 To Part1
 - No Separation - Perno2 To Part1
 - Contact Region 8
 - Contact Region 9
 - Mesh
 - Static Structural (A5)
 - Analysis Settings

Details of "No Separation - Perno1 To Part1"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Perno1
Target Bodies	Part1
Protected	No

Definition	
Type	No Separation
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Suppressed	No



Possibilità di escludere alcune superfici dalla connessione, in questo caso, delle due, rimane solo quella caricata

Vincoli e carichi come per il caso precedente

A: Static Structural (ANSYS)

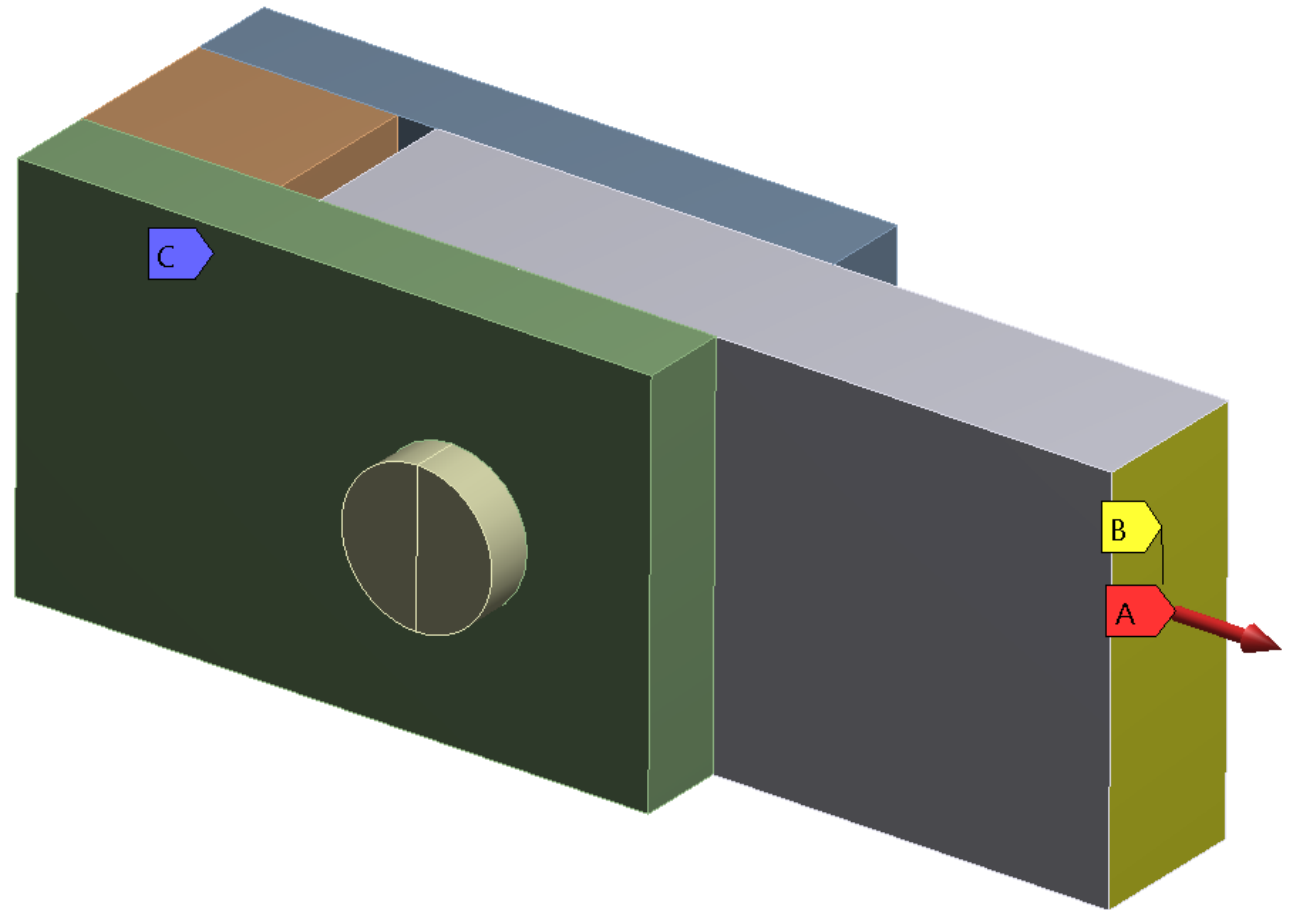
Static Structural

Time: 1. s

A Force: 1000. N

B Displacement

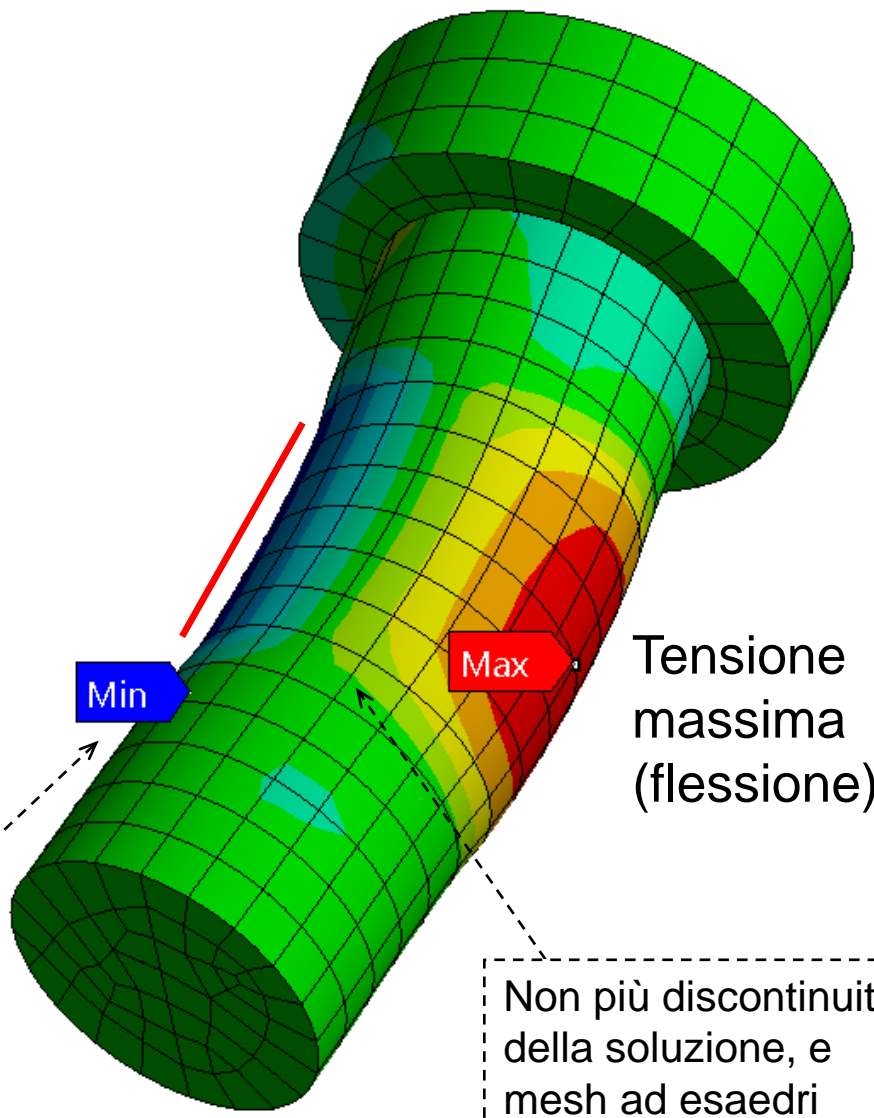
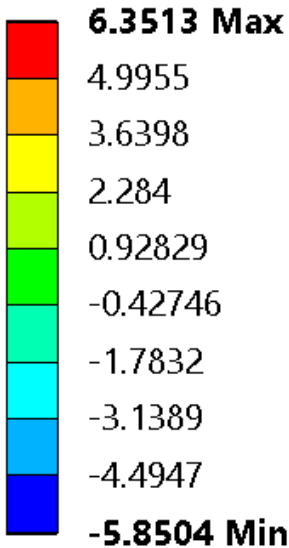
C Fixed Support



Stesse condizioni di vincoli e carico,
e Weak Springs

Stato di tensione del perno

A: Static Structural (ANSYS)
Normal Stress
Type: Normal Stress(Z Axis)
Unit: MPa
Global Coordinate System
Time: 1

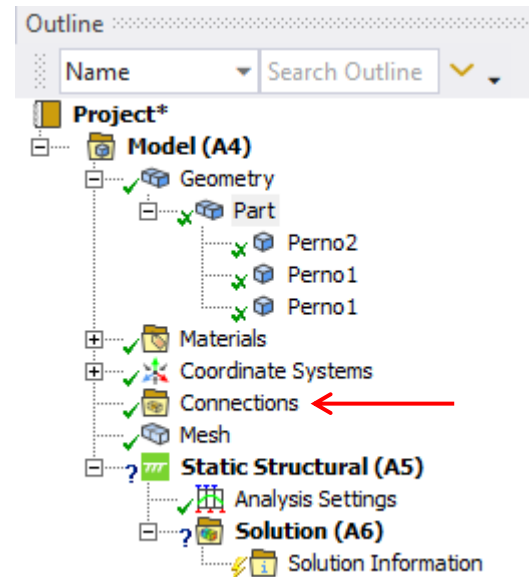
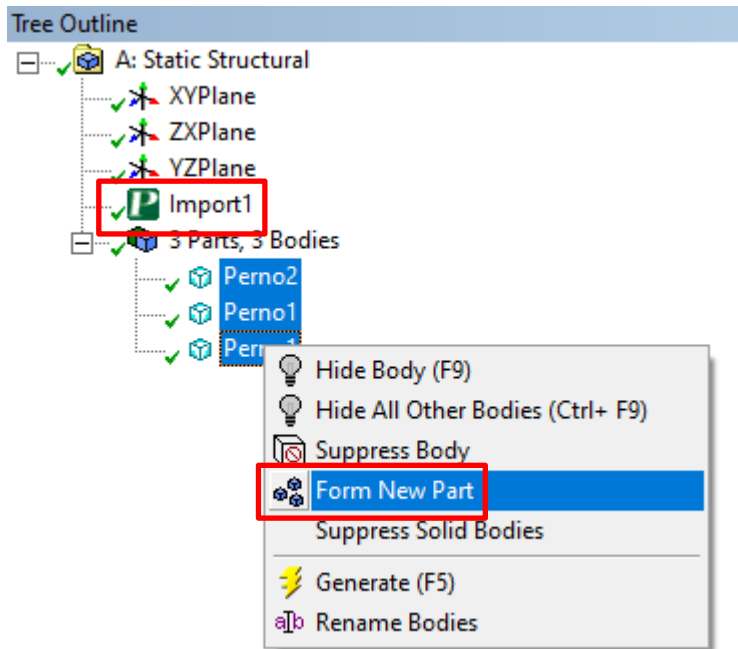


Tensione massima (flessione)

Connessione attiva, contatto concentrato ai bordi laterali

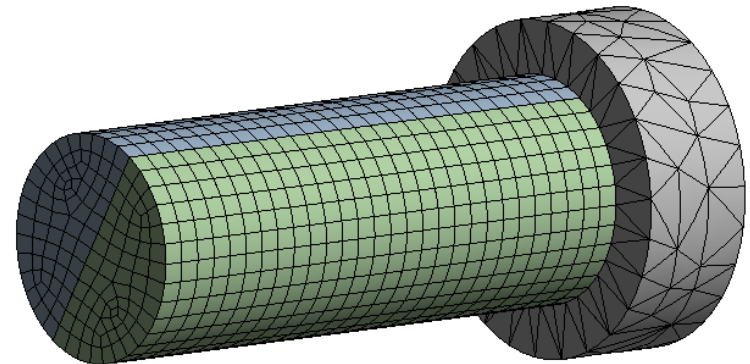
Non più discontinuità della soluzione, e mesh ad esaedri

Soluzione alternativa: creazione di un'unica parte (internamente divisa) mediante DesignModeler



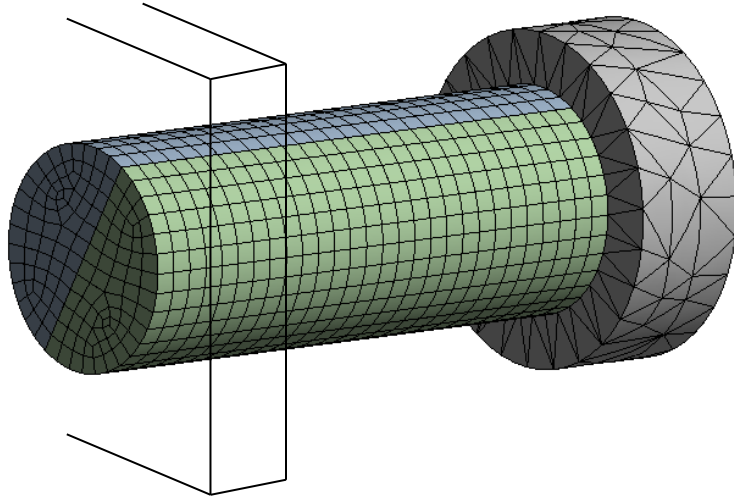
Nell'ambiente DesignModeler di Ansys Workbench si importa la geometria precedente (con i volumi divisi) e si crea una parte unica con «Form New Part»

N.B.: si tratta dell'equivalente dell'operazione di Glue (non di Add) in ambiente APDL.

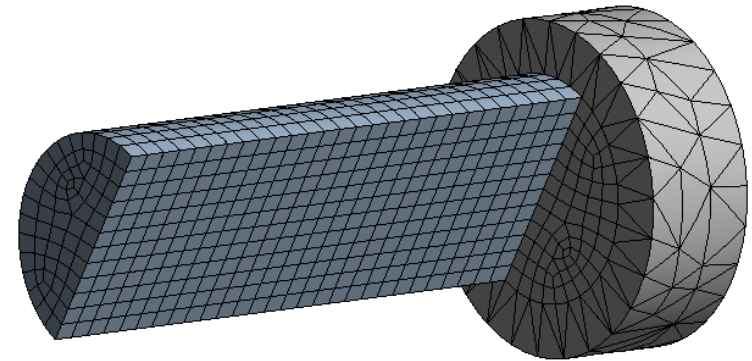


In questo modo non esistono più le connessioni interne fra i corpi che sono stati uniti

Modellazione in ambiente DesignModeler



Ovviamente saranno poi riconosciute le connessioni con le altre parti del modello (non mostrate), e il processo di selezione dei contatti può procedere allo stesso modo



La testa del perno era una parte successivamente unita nell'assieme in ambiente CAD, tuttavia in DM è stata ottenuta un'unica parte senza connessioni.

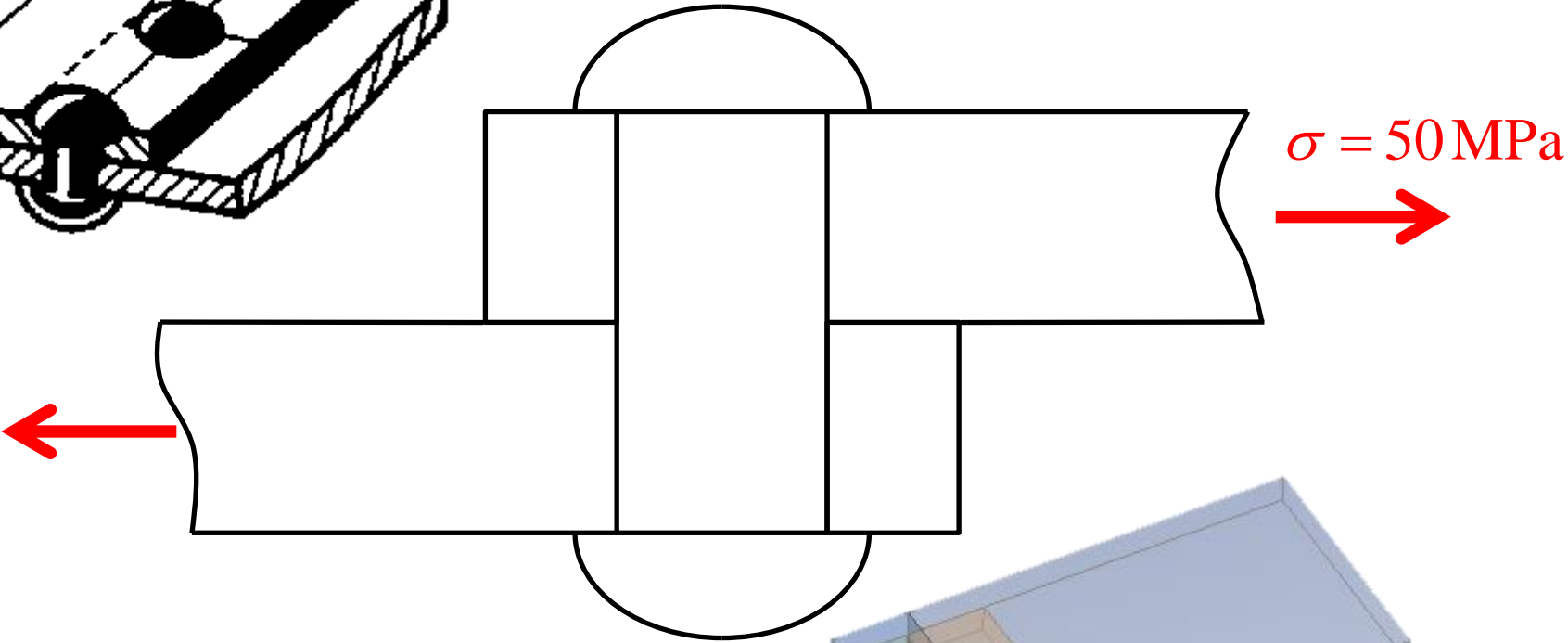
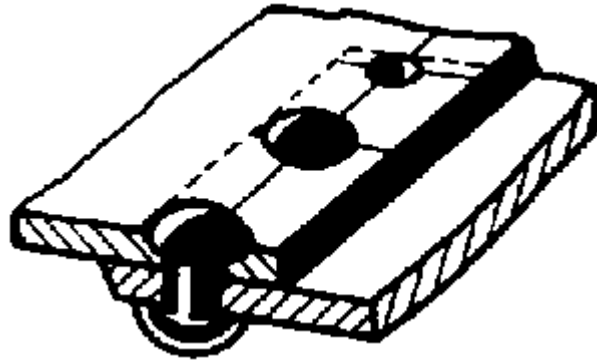
La mesh è coerente fra i sottovolumi di questa parte complessiva, e in particolare la mesh dei semiperni è ad esaedri

Sommario Lez. 3

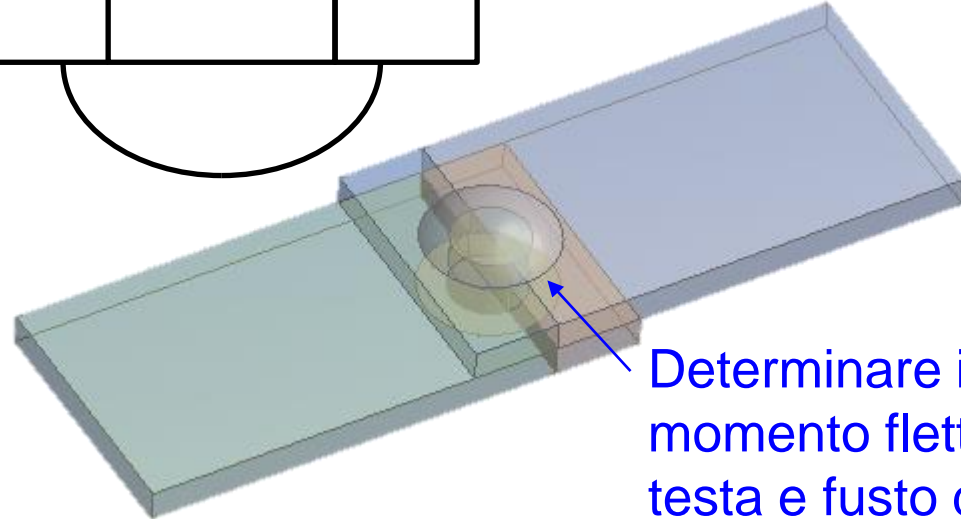
1. Ridurre o eliminare connessioni unilaterali.
2. Suddividere la connessione in più parti, e **deselezionare** le zone di contatto non caricate.
3. Suddivisione preliminare di un **volume** (integro) in più parti assemblate, per avere porzioni separate di superfici a contatto.
4. Alternativamente spezzare le **superfici** della parte (o delle parti) coinvolte nella connessione.

Esercitazione da svolgere: Giunto a sovrapposizione rivettato

N.B.: Considerare assente il pretensionamento del rivetto



Vincoli?
Connessioni?



Determinare il momento flettente fra testa e fusto del rivetto