

Università degli Studi di Padova – Dipartimento di Ingegneria Industriale
Corso di Laurea in Ingegneria Meccanica

Relazione per la prova finale
Analisi FEM di un riduttore bistadio mediante Solidworks
Simulation

Tutor universitario: Prof. Alberto Campagnolo

Studente: *Salvagnin Francesco*

Padova, 22/03/2024

Gli obiettivi principali del lavoro sono quelli di usare i due casi proposti di analisi FEM, ovvero l'analisi del *telaio piano* e della *piastra forata*, come addestramento all'utilizzo di *Solidworks Simulation*, per poter svolgere poi le *verifiche a fatica flessionale* (calcolo K_t) e la *verifica a deformabilità* dell'albero intermedio del riduttore.

In conclusione si andranno a confrontare i risultati ottenuti dalle *analisi FEM* con i risultati calcolati in modo analitico durante il dimensionamento dell'albero, in modo da poter ragionare sulle *potenzialità* e sui *limiti* del *metodo degli elementi finiti*.

Introduzione sul
metodo degli
elementi finiti

- Nodo
- Elemento finito
- Mesh

Addestramento a
Solidworks Simulation

- Telaio piano: verifica strutturale
- Piastra forata: K_{tg}

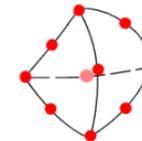
Albero intermedio
riduttore

- Verifica a fatica flessionale nelle sezioni più critiche: K_t
- Verifica di deformabilità: f_{max} e φ_{max}

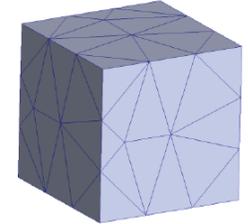
• nodo



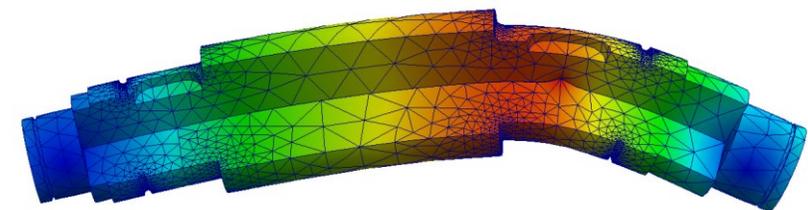
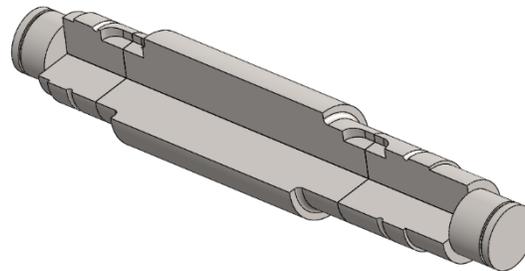
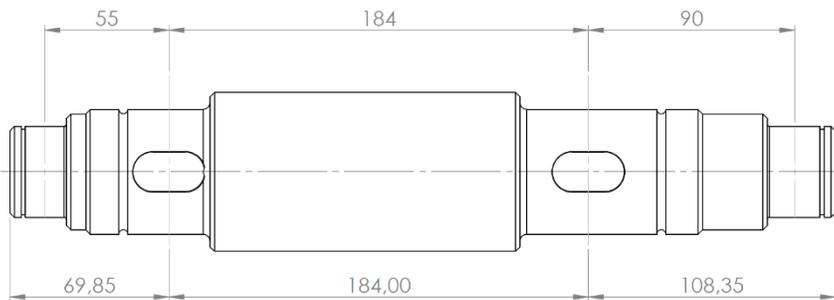
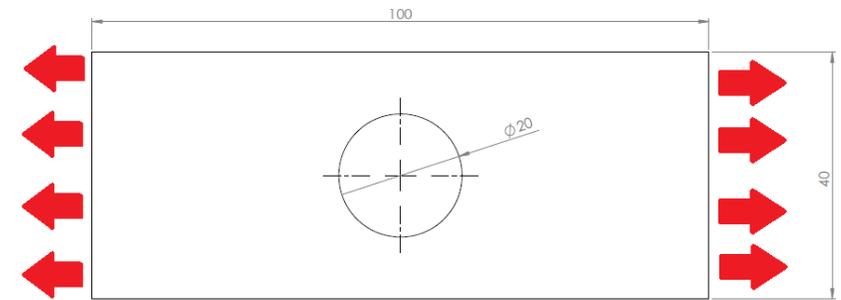
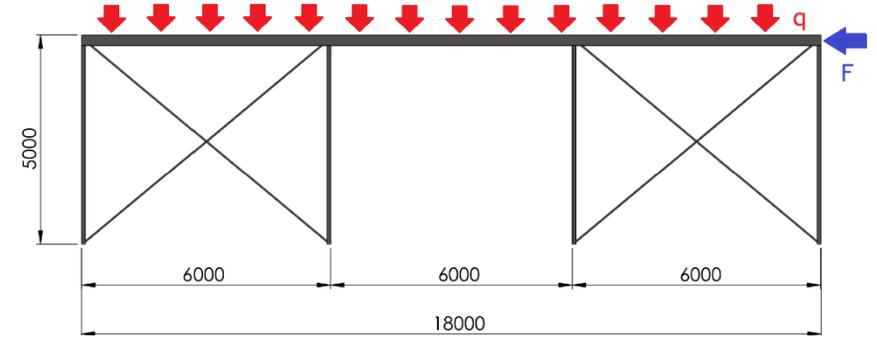
Gradi di libertà, DOF



Elemento finito



Mesh



Concetti *chiave* per l'analisi *FEM*:

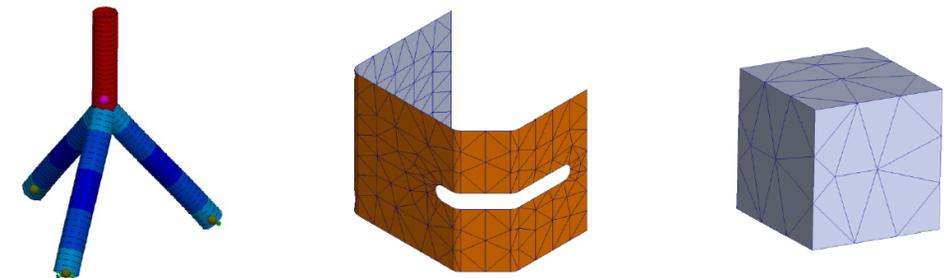
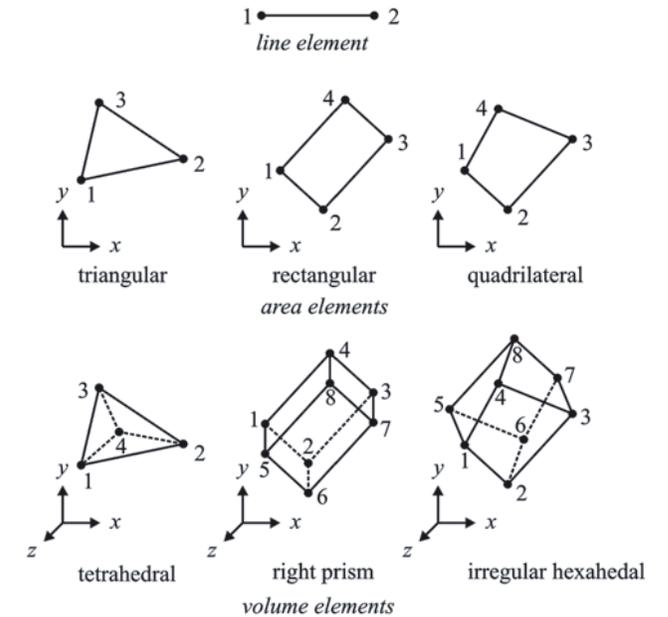
- i. Nodo
- ii. Elemento finito
 - a. Varie tipologie di elemento finito (**mono, bi, tri-dimensionale**)
 - b. Elementi del *primo/secondo ordine*
- iii. Mesh

Step dell'analisi *FEM* per casi *statici lineari*:

(Condizioni note: *geometria, materiale, vincoli, carichi*)

- i. Spostamento Δs di ogni nodo
- ii. Deformazioni corrispondenti $\varepsilon = \Delta s/s$
- iii. Tensioni agenti $\sigma = \sigma(\varepsilon, E, \nu)$

Ipotesi per il caso *statico lineare*: **carichi costanti nel tempo, risposta lineare del materiale, piccoli spostamenti.**



Dati:

$F = 60 \text{ kN}$

$q = 20 \text{ kN/m}$

$L_1 = 6000 \text{ mm}$

$L_2 = 5000 \text{ mm}$

Acciaio Fe430

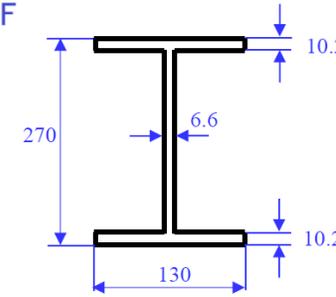
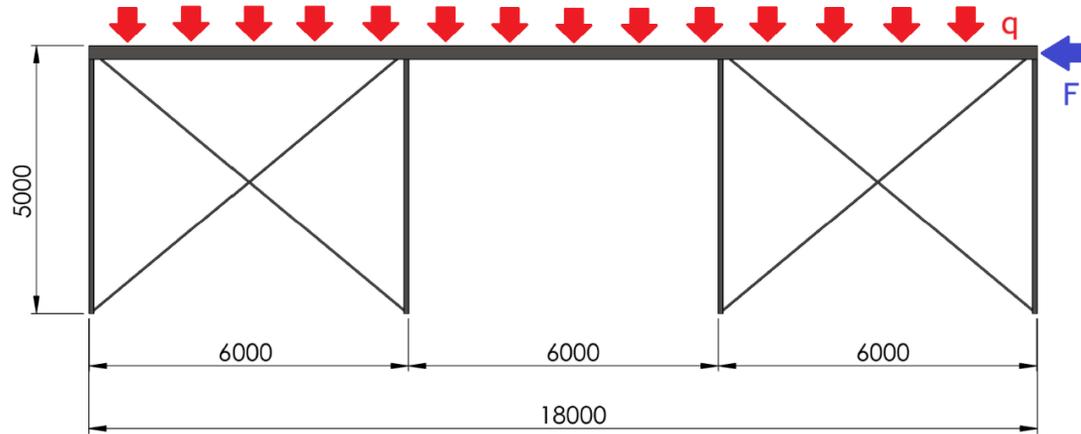
$\sigma_{adm} = 190 \text{ MPa}$

$E = 206000 \text{ MPa}$

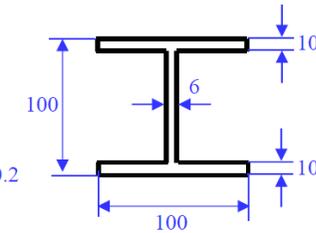
$\nu = 0,3$

Richieste:

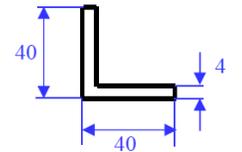
- Deformata (spostamenti dei nodi)
- Reazioni vincolari
- Verifica strutturale



Corrente superiore: IPE 270

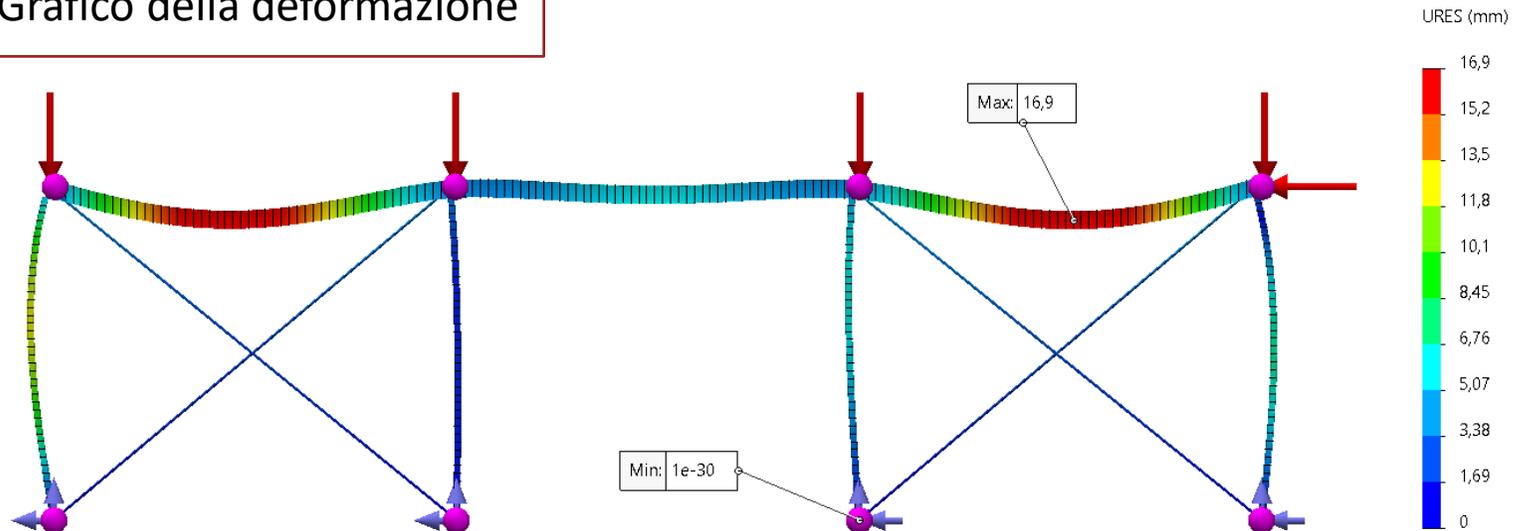


Colonne: HE 100 B

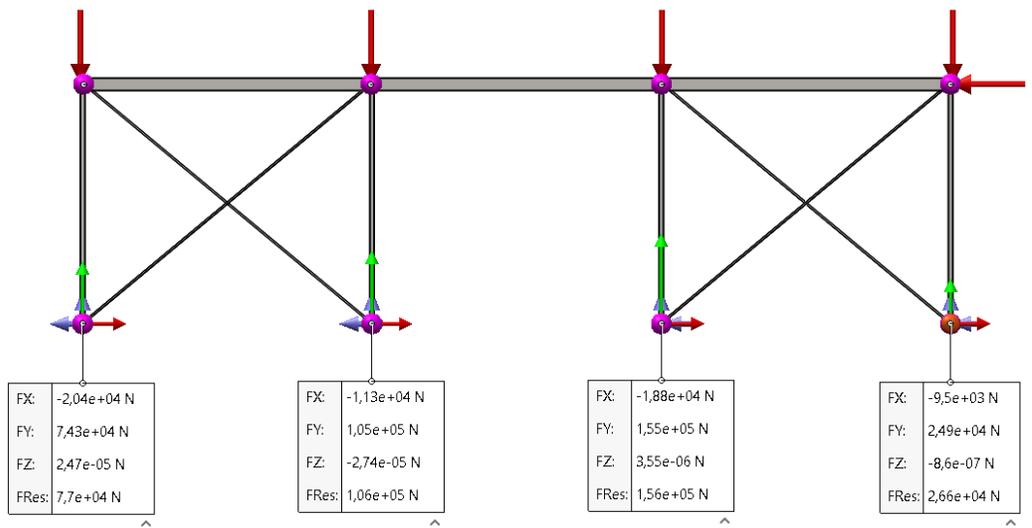


Controventi: L 40x4

Grafico della deformazione



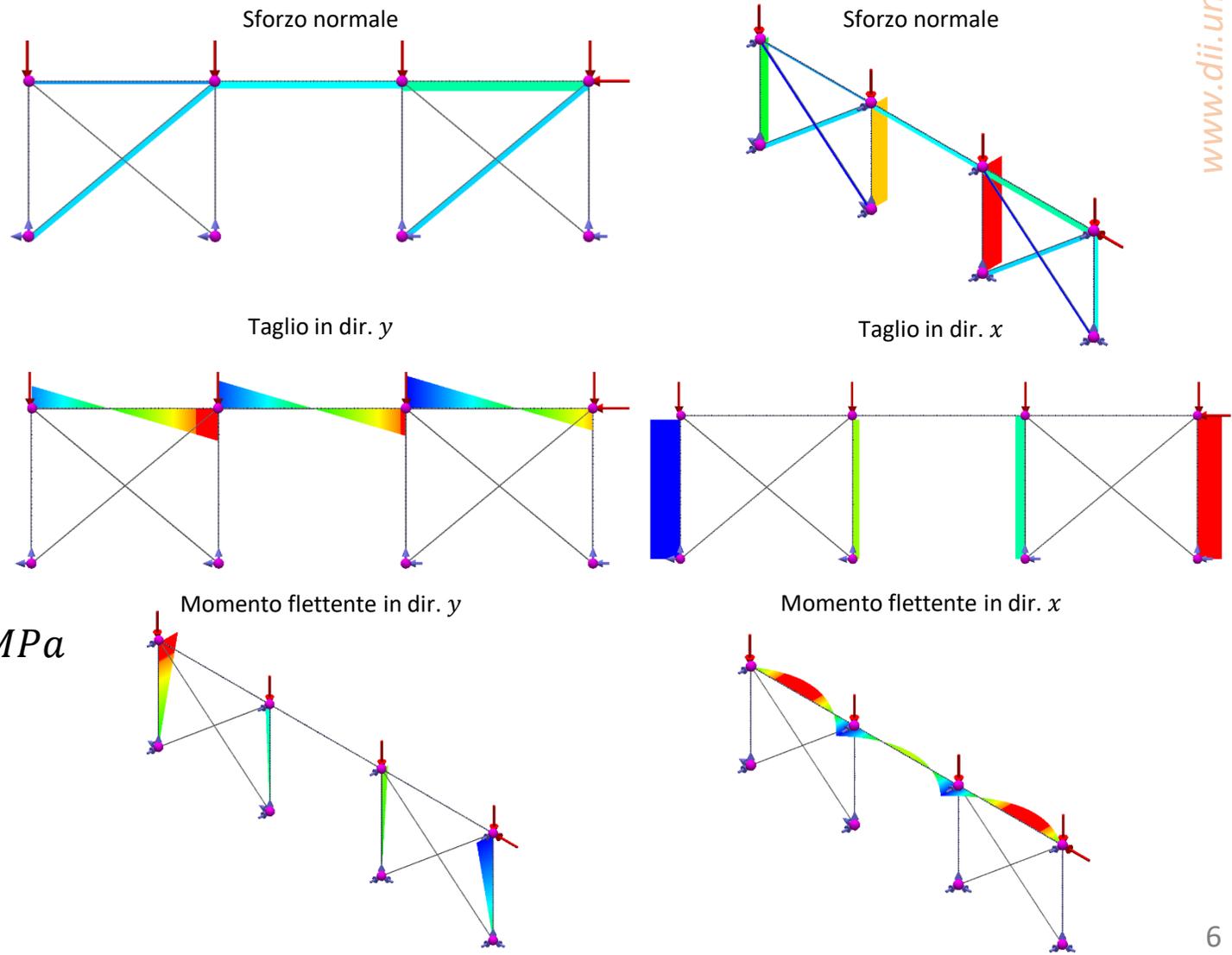
Reazioni vincolari



Verifica strutturale

Corrente superiore $\sigma_{id} = 138 \text{ MPa} < \sigma_{adm} = 190 \text{ MPa}$
 Colonne $\sigma_{id} < \sigma_{adm}$
 Controventi $\sigma_{id} < \sigma_{adm}$

Diagrammi delle sollecitazioni



Dati:

$$\sigma = 1 \text{ MPa}$$

$$w = 1 \text{ mm}$$

Acciaio Fe360

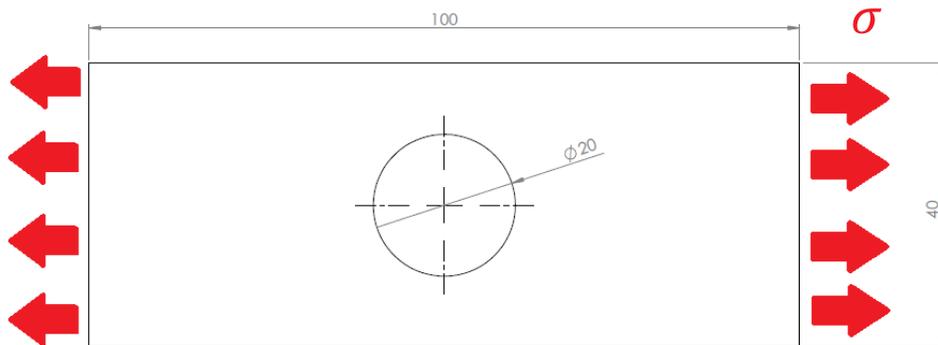
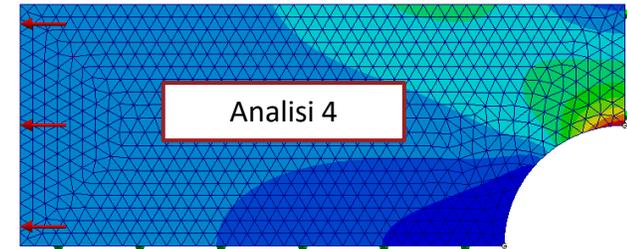
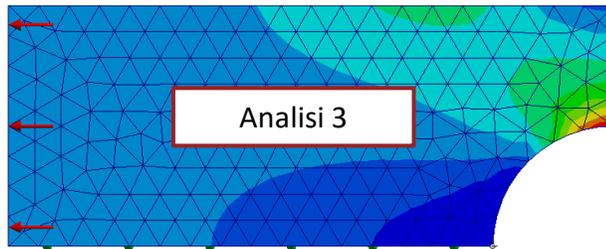
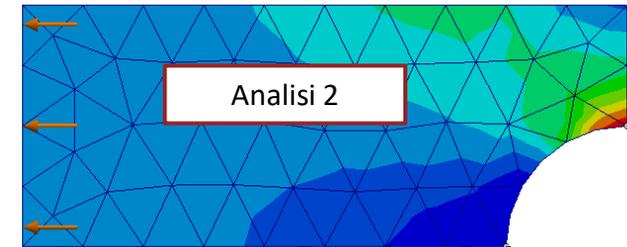
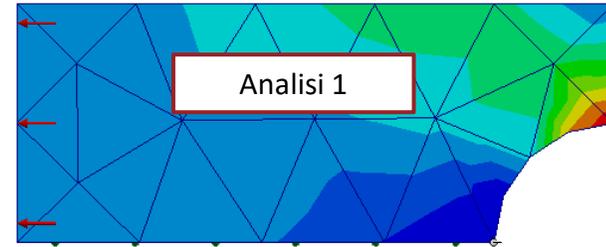
$$\sigma_r = 360 \text{ MPa}$$

$$E = 206000 \text{ MPa}$$

$$\nu = 0,3$$

Richieste:

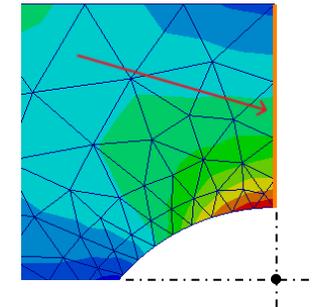
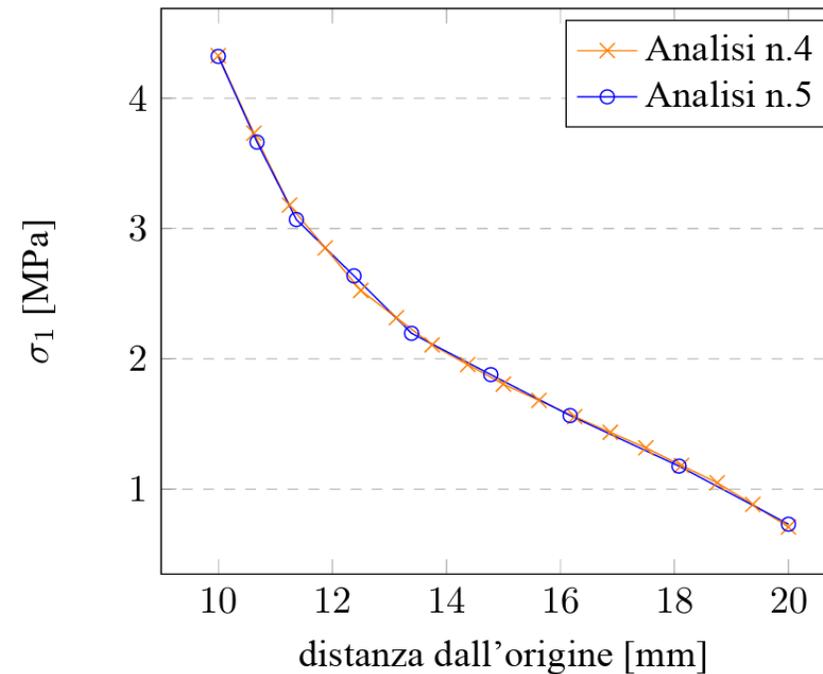
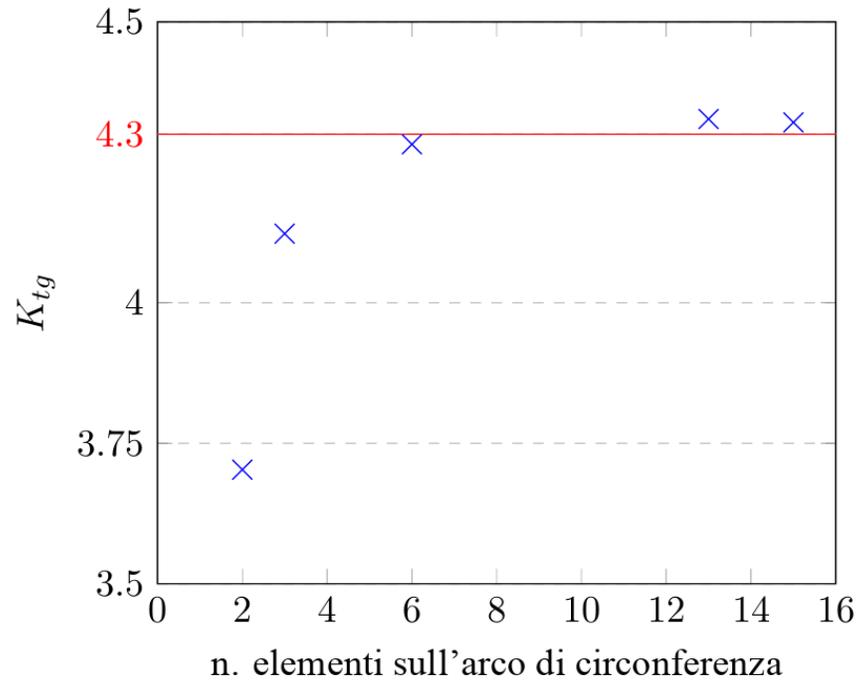
- Analisi sensibilità della mesh
- Calcolo $K_{t,g}$
- Confronto con valore Peterson



analisi n.	n. elementi lungo il bordo	$\sigma_{p,el}$ [MPa]	$\sigma_{n,g}$ [MPa]	K_{tg}	$\Delta\%$ tra K_{tg} e $K_{tg,Pet}$
1	2	3,703	1	3,703	16,12%
2	3	4,123	1	4,123	4,29%
3	6	4,282	1	4,282	0,42%
4	13	4,327	1	4,327	0,62%
5	15	4,321	1	4,321	0,49%

$$K_{tg} = \frac{\sigma_{p,el}}{\sigma_{n,g}}$$

Valore Peterson $K_{tg} = 4,3$

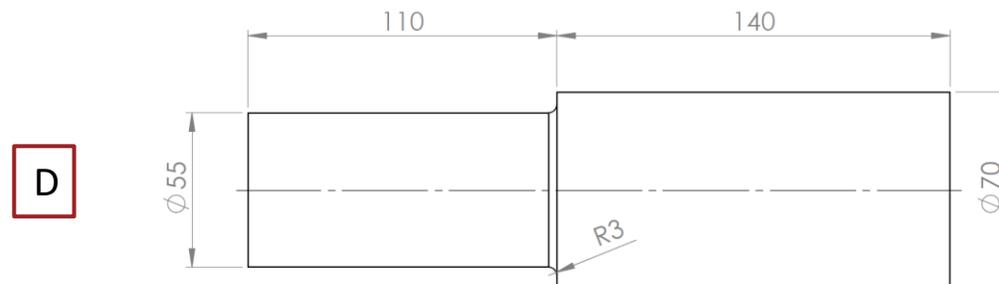
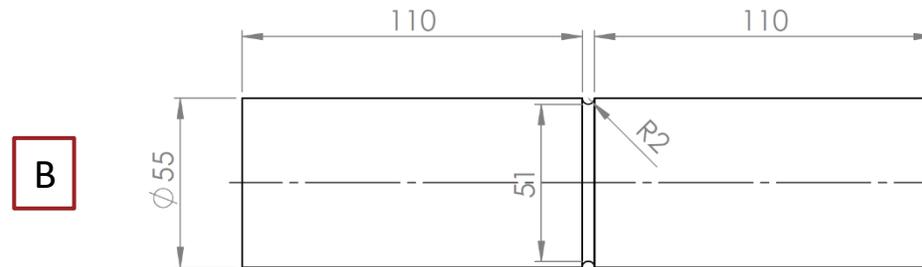
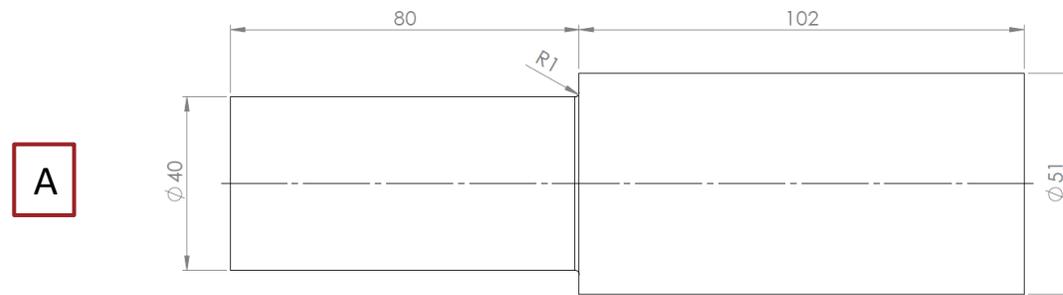
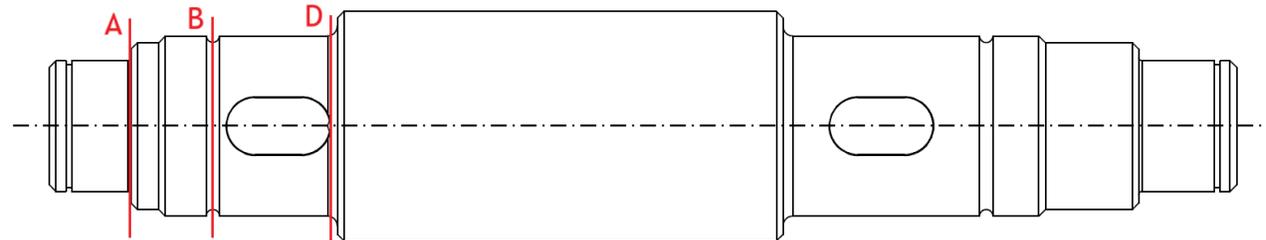


Dati:

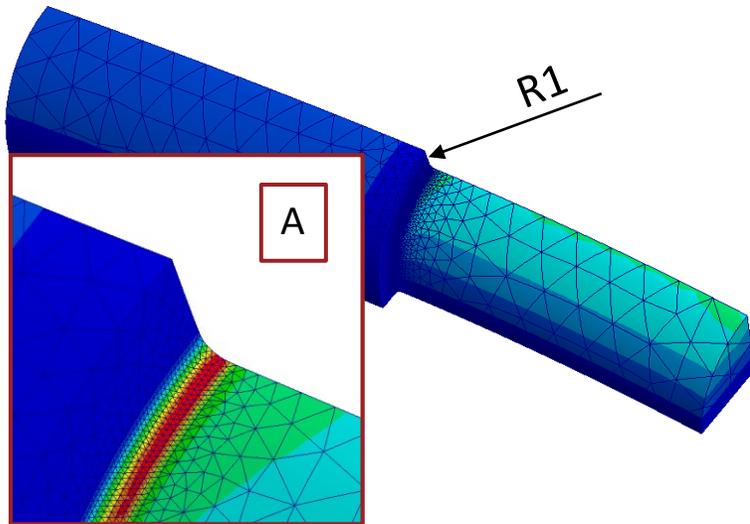
Materiale 39NiCrMo3
 $\sigma = 1 \text{ MPa}$

Richieste:

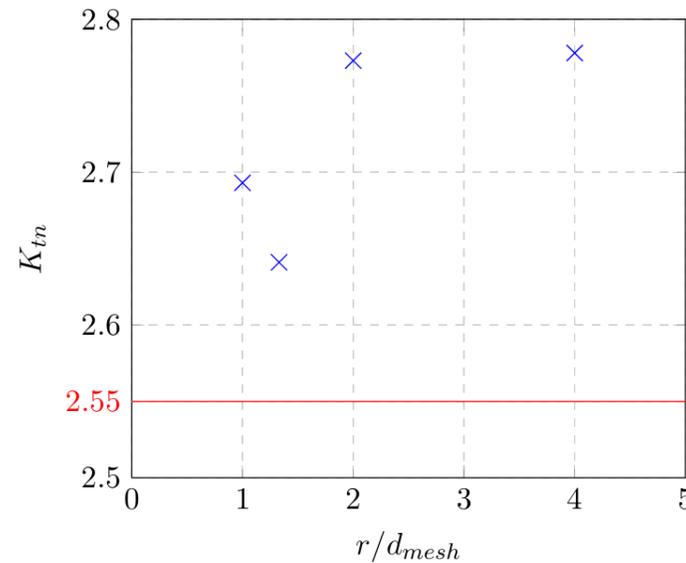
Determinare il fattore teorico di concentrazione delle tensioni K_t a flessione delle sezioni A, B, D



Sezione A-A: spallamento cuscinetto



Analisi n.	d_{mesh} [mm]	r/d_{mesh}	K_{tn}	$\Delta\%$ tra K_{tn} e $K_{tn,Pet}$
1	1,00	1	2,693	5,61%
2	0,75	1,33	2,641	3,57%
3	0,50	2	2,773	8,75%
4	0,25	4	2,778	8,94%

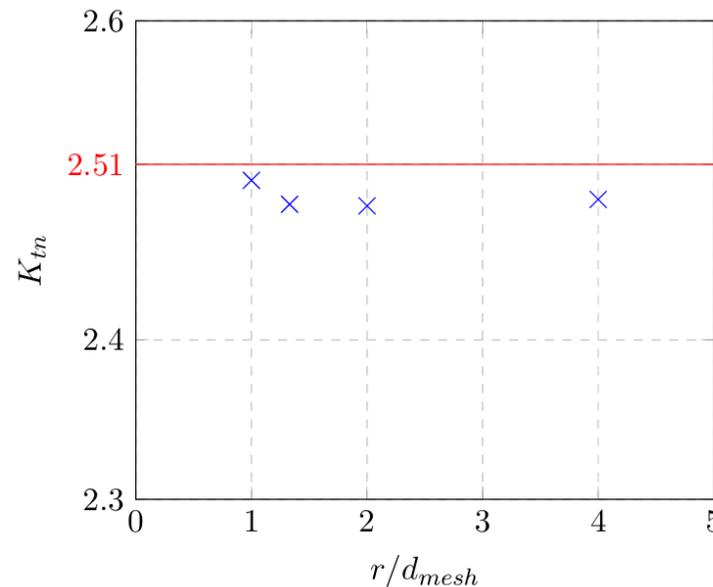
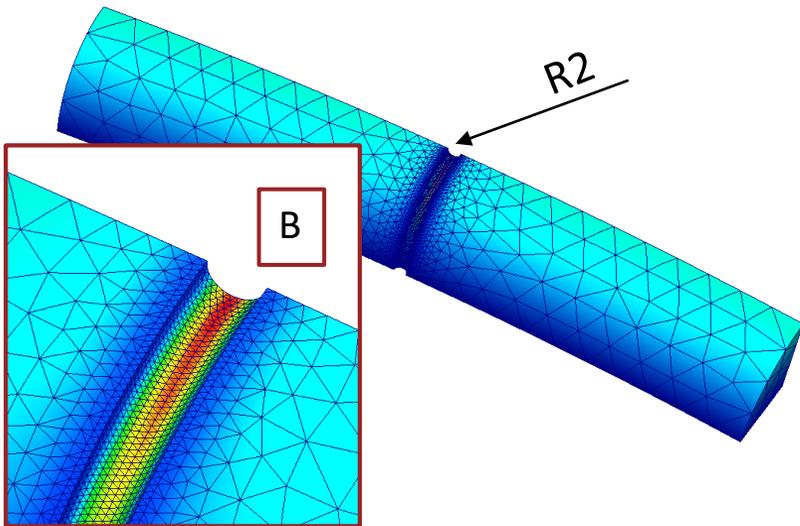


$$K_{tn} = \frac{\sigma_{p,el}}{\sigma_{nn}}$$

Valore Peterson $K_{tn} = 2,55$

Sezione B-B: gola di scarico filettatura

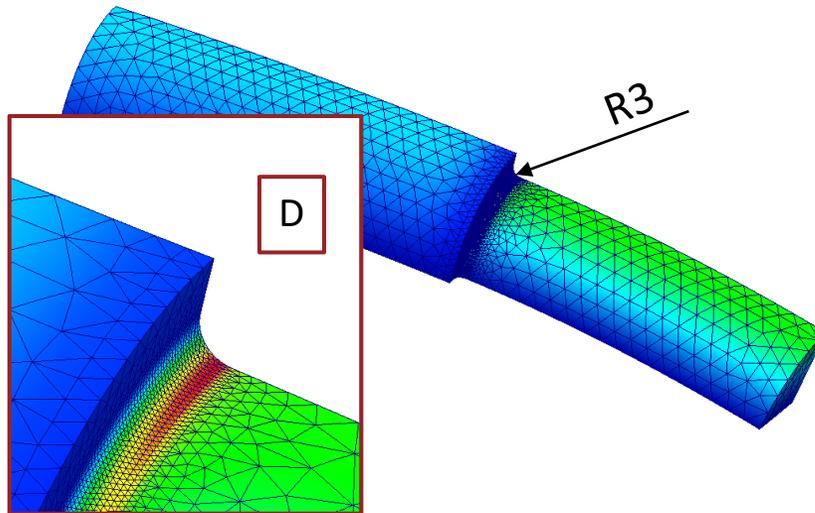
Analisi n.	d_{mesh} [mm]	r/d_{mesh}	K_{tg}	K_{tn}	$\Delta\%$ tra K_{tn} e $K_{tn,Pet}$
1	2,00	1	3,135	2,500	0,42%
2	1,50	1,33	3,117	2,485	0,99%
3	1,00	2	3,115	2,484	1,05%
4	0,50	4	3,121	2,488	0,86%



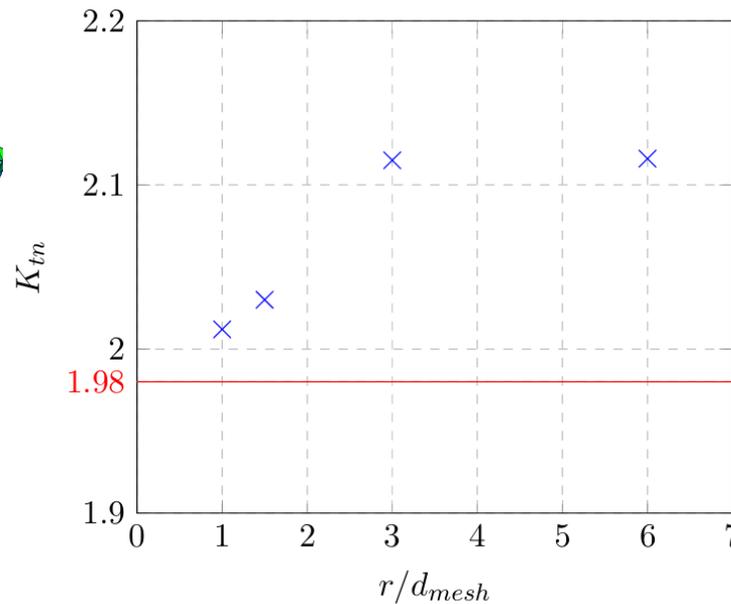
$$K_{tn} = K_{tg} \cdot \frac{d^3}{D^3}$$

Valore Peterson $K_{tn} = 2,51$

Sezione D-D: spallamento ruota dentata



Analisi n.	d_{mesh} [mm]	r/d_{mesh}	K_{tn}	$\Delta\%$ tra K_{tn} e $K_{tn,Pet}$
1	3,00	1	2,012	1,62%
2	2,00	1,5	2,030	2,53%
3	1,00	3	2,115	6,82%
4	0,50	6	2,116	6,87%

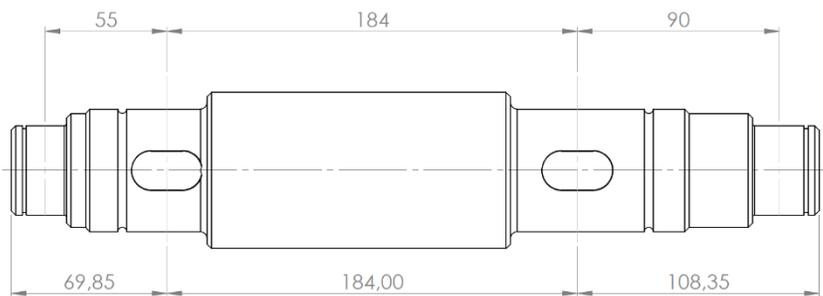


$$K_{tn} = \frac{\sigma_{p,el}}{\sigma_{nn}}$$

Valore Peterson $K_{tn} = 1,98$

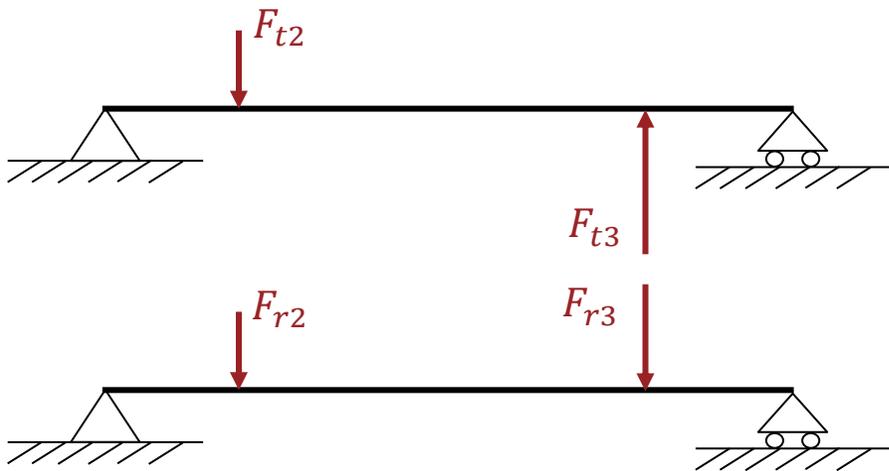
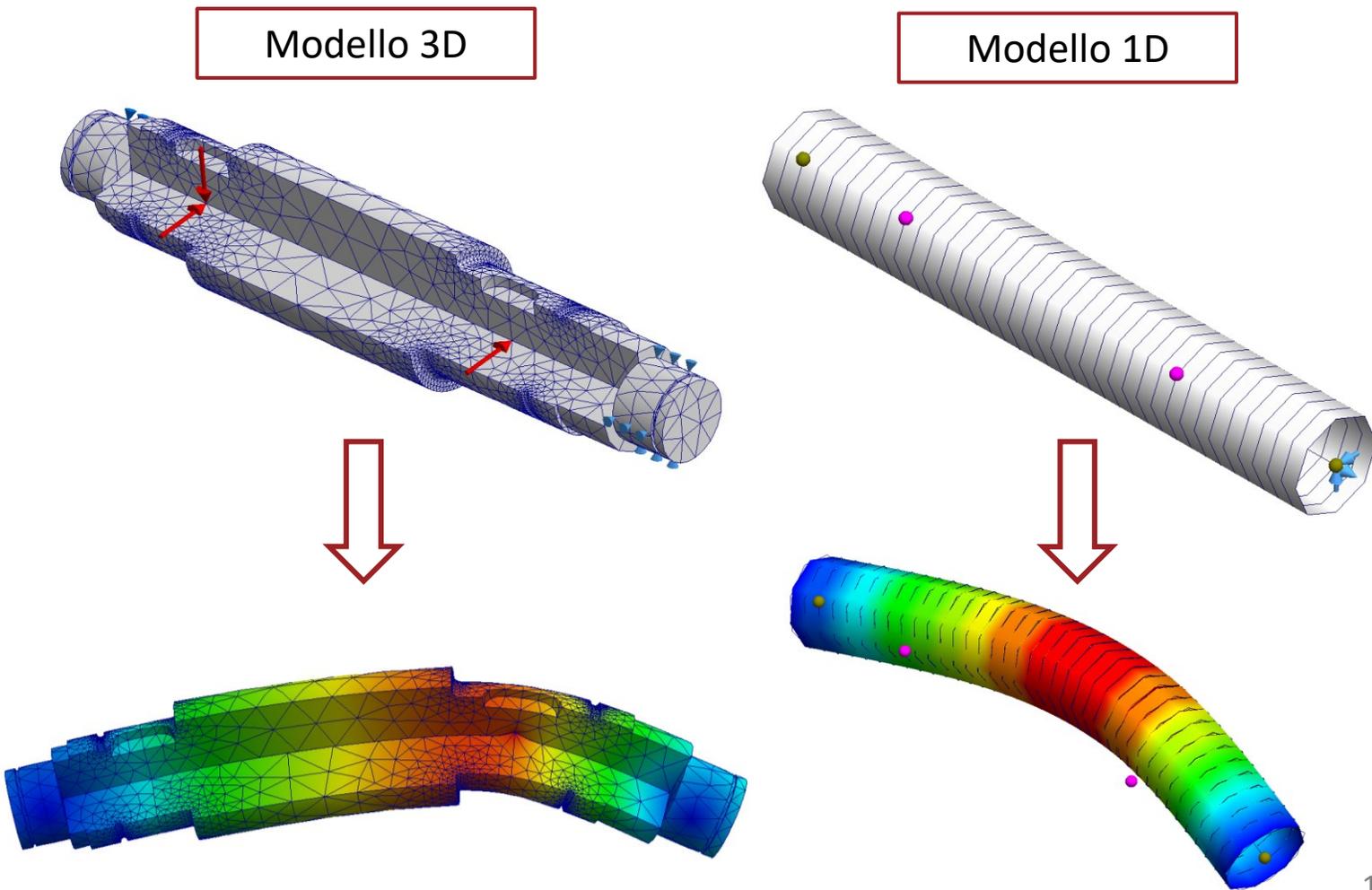
Richieste:

Determinare le rotazioni φ_s, φ_d e le frecce f_2, f_3 dovute alle forze F_t e F_r trasferite all'albero dalle ruote dentate



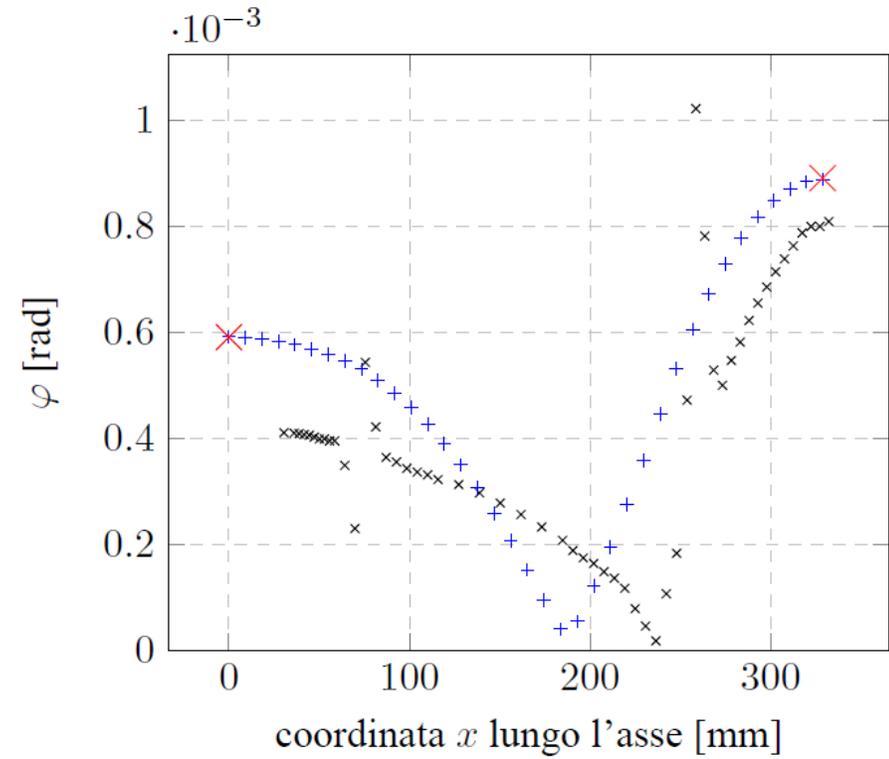
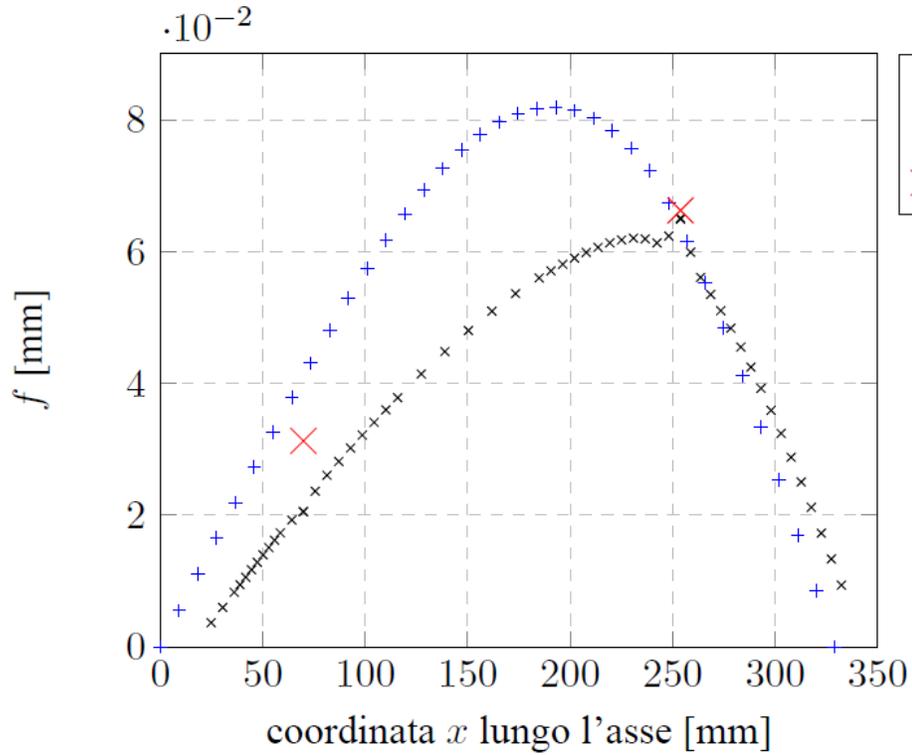
Modello 3D

Modello 1D



$$\begin{aligned}
 F_{r2} &= 1243 \text{ N} & F_{t2} &= 3416 \text{ N} \\
 F_{r3} &= 4974 \text{ N} & F_{t3} &= 13665 \text{ N}
 \end{aligned}$$

Confronto tra i modelli



Modello	f_max (mm)	phi_s (rad)	phi_d (rad)
3D	6,523E-02	4,10E-04	8,08E-04
1D	7,230E-02	5,906E-04	8,88E-04
Analitico	6,628E-02	0,00059	0,00089

$$f_{max} \leq f_{lim} = 0,127 \text{ mm da ASME}$$

$$\varphi_s \leq \varphi_{lim} = 10' = 0,00291 \text{ rad cuscinetto a sfere SKF (s)}$$

$$\varphi_d \leq \varphi_{lim} = 4' = 0,00116 \text{ rad cuscinetto a rulli SKF (d)}$$