NMS EXERCISE 1

Cantilever beam model

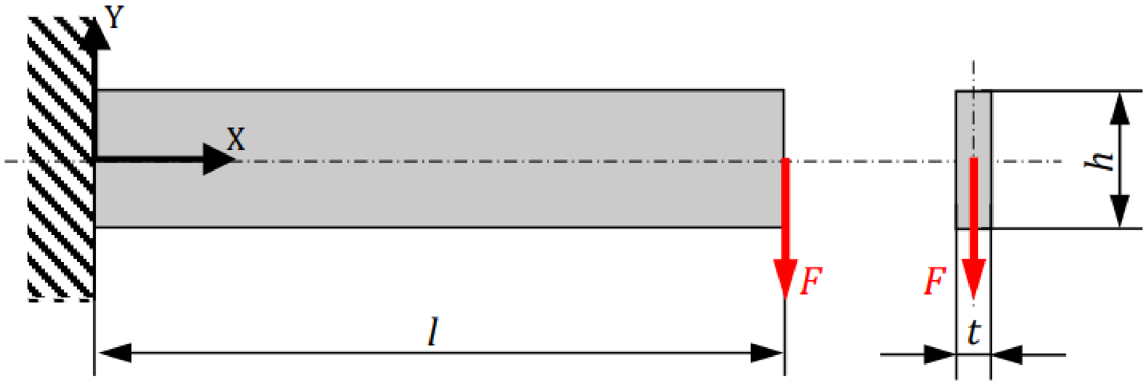
Students:

COLUCCI Carlo Vittorio	329703
COVETTI Alessio	329876
PLACIDA Pierpaolo	323197

Politecnico di Torino
1859

Academic year 2023/24

Project goals



The proposed analysis aims to investigate the behaviour of a **cantilever beam**, exploiting the finite element method (FEM).

The main exercise goals will be:

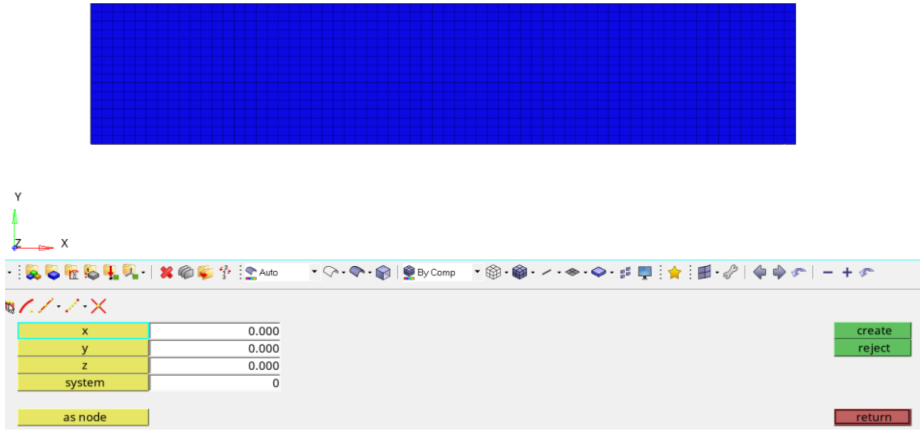
- Assure the convergence between numerical and expected analytical results.
- Compare the features of two distinct element types used for the simulations.
- Analyse the influence of different contraints and loads conditions on stress and strain distributions.

Model creation

Material

Name	Value
Solver Keyword:	MAT1
Name:	Steel
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comments:	Hide In Menu/Export
E:	210000.0
G:	80769.2
NU:	0.3
RHO:	7.85e-09

Geometry



Property

Name	Value
Solver Keyword:	PSHELL
Name:	Property 1
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	PSHELL
Material:	(1) Steel
User Comments:	Hide In Menu/Export
T:	5.0

Set up of the cantilever beam model **material properties** and **geometrical dimensions**.

A property for the beam is defined in order to assign thickness value and **“PSHELL”** card image.

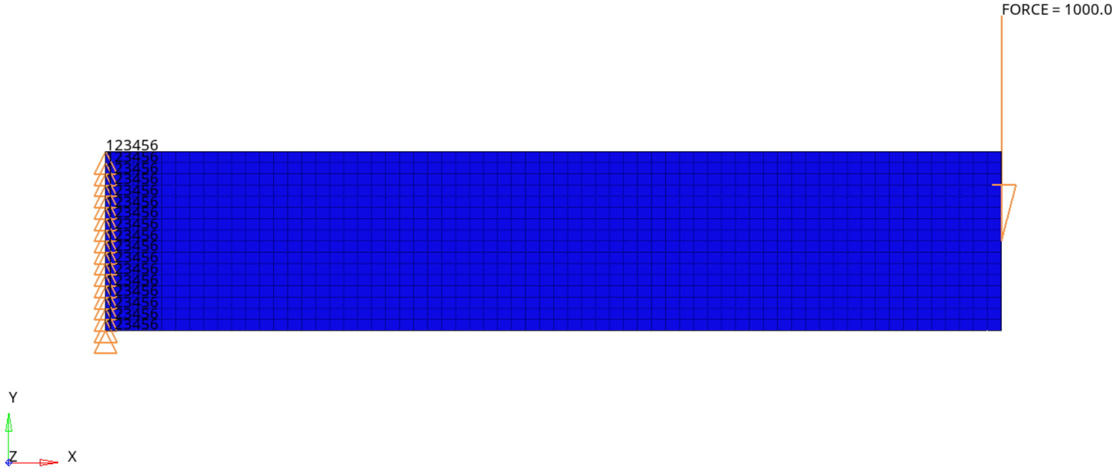
Model creation

<u>Boundary conditions:</u>

Constraints are applied on the left side of the beam and can be simulated in two ways:

- constrain the horizontal displacement of all the nodes to zero and the vertical displacement only of the middle node to zero.
- constrain the horizontal and vertical displacement of all nodes to zero.

A vertical load of **1000 N** is set on the structure right edge. The force can be set concentrated on the middle node or eventually distributed on all edge nodes.



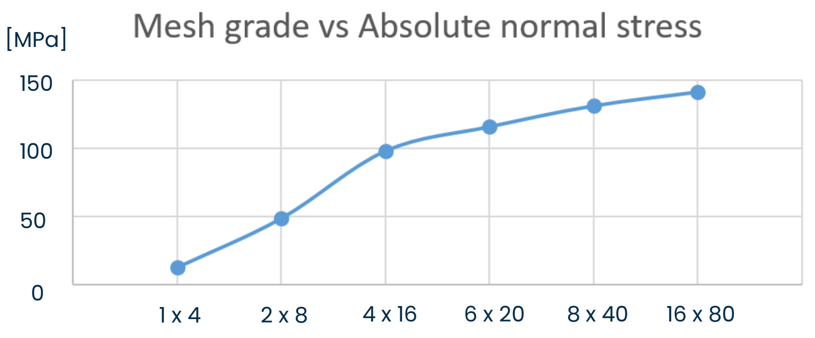
To make a comparison between 3 and 4 nodes element, the depicted configuration is employed, having the load concentrated, and both vertical and horizontal displacement constrained to zero on all nodes.

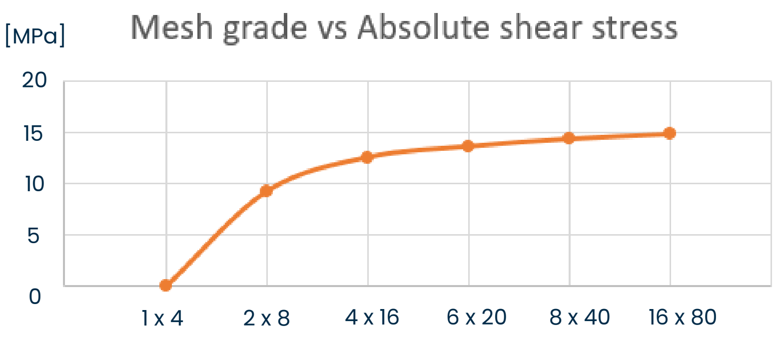
3 nodes element analysis

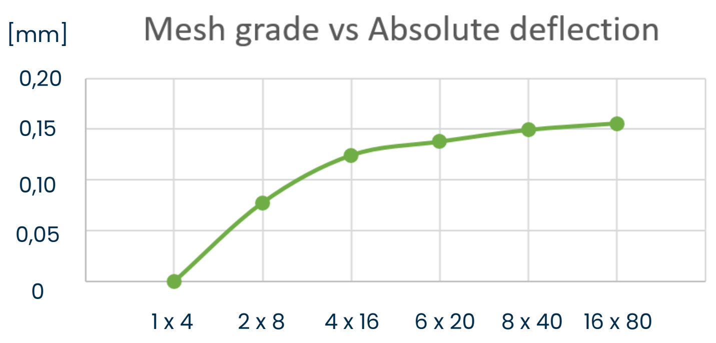
Simulations results collection at increasing mesh grade:

	Number of elements	| Normal stress s |	| Shear stress t |	| Deflection |
		Mpa	Mpa	mm
Theoretical values		150	15	0,1517
	1 x 4	12,86	-	-
	2 x 8	48,84	9,271	0,07765
	4 x 16	98,29	12,58	0,1242
	6 x 20	116,1	13,63	0,1377
	8 x 40	131,4	14,36	0,1492
	16 x 80	141,6	14,83	0,1556

The following plots highlight how the registered stresses and deflection values approach the analytical ones by increasing the number of degrees of freedom:





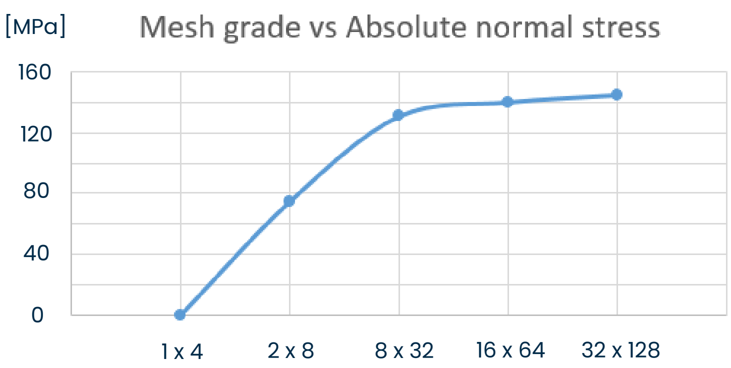


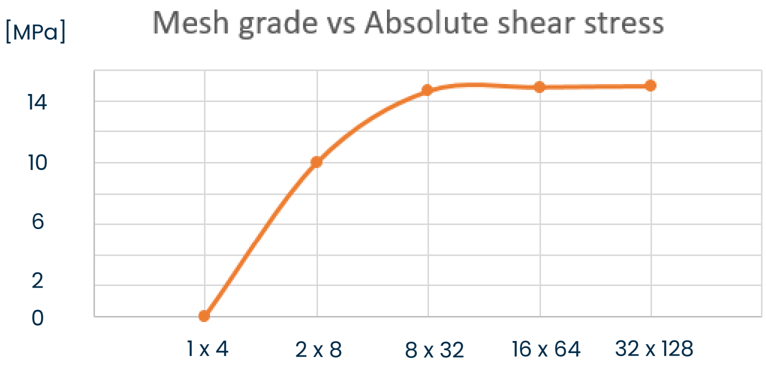
4 nodes element analysis

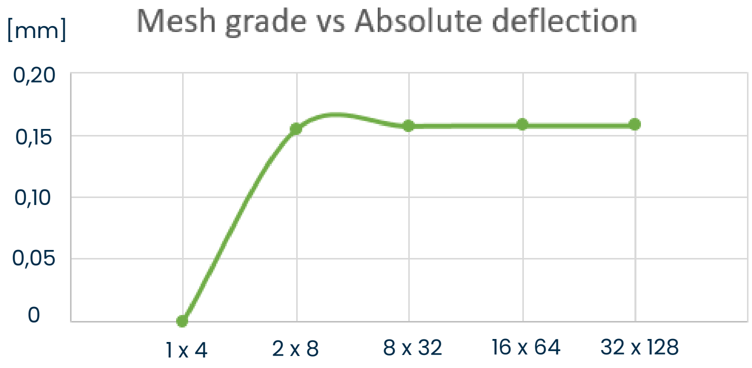
Simulations results collection at increasing mesh grade:

	Number of elements	| Normal stress s |	| Shear stress t |	| Deflection |
		Mpa	Mpa	mm
Theoretical values		150	15	0,1517
	1 x 4	4,987E-13	-	-
	2 x 8	75	10	0,1547
	8 x 32	131,2	14,65	0,1576
	16 x 64	140,6	14,91	0,1579
	32 x 128	145,3	14,98	0,158

As in the 3 nodes case, the registered stresses and deflection values approach way more the analytical ones when increasing the mesh size:







Convergence of the solutions and comparison

The choice of the most significant mesh size is based on the **relative error** of the analyses values with respect to the theoretical ones.

The selected mesh, 4 nodes element (16x64 configuration), has a relative error lower than 8% for all the evaluated quantities. The choice aims to use a model with **high accuracy** and **limited computational cost**.

3 nodes element

Number of elements	| Normal stress s |	| Shear stress t |	| Deflection |
	Error wrt Theor val (%)	Error wrt Theor val (%)	Error wrt Theor val (%)
1 x 4	91,4		
2 x 8	67,4	38,2	48,8
4 x 16	34,5	16,1	18,1
6 x 20	22,6	9,1	9,2
8 x 40	12,4	4,3	1,6
16 x 80	5,6	1,1	-2,6

4 nodes element

Number of elements	| Normal stress s |	| Shear stress t |	| Deflection |
	Error wrt Theor val (%)	Error wrt Theor val (%)	Error wrt Theor val (%)
1 x 4	100,0		
2 x 8	50,0	33,3	-2,0
8 x 32	12,5	2,3	-3,9
16 x 64	6,3	0,6	-4,1
32 x 128	3,1	0,1	-4,2

Influence of different boundary conditions on stress and strain field

The results show the difference between applying the same force value first with a concentrated and then with a distributed load. The mesh size is the one previously chosen, the 4 nodes element 16 x 64.

Constraints: horizontal displacement of all the nodes to zero and the vertical displacement only of the middle node to zero.

Concentrated load

| Normal stress s |	| Shear stress t |	| Deflection |
at half beam lenght	at half beam lenght	at half beam lenght
Mpa	Mpa	mm
140,6	14,91	0,1578

Distributed load

| Normal stress s |	| Shear stress t |	| Deflection |
at half beam lenght	at half beam lenght	at half beam lenght
Mpa	Mpa	mm
140,6	14,91	0,1578

Influence of different boundary conditions on stress and strain field

As before, the difference between concentrated and distributed load is shown. The mesh size is the one previously chosen, the 4 nodes element 16 x 64.

Constraints: horizontal and vertical displacement of all nodes set to zero.

Concentrated load

| Normal stress s |	| Shear stress t |	| Deflection |
at half beam lenght	at half beam lenght	at half beam lenght
Mpa	Mpa	mm
140,6	14,91	0,1549

Distributed load

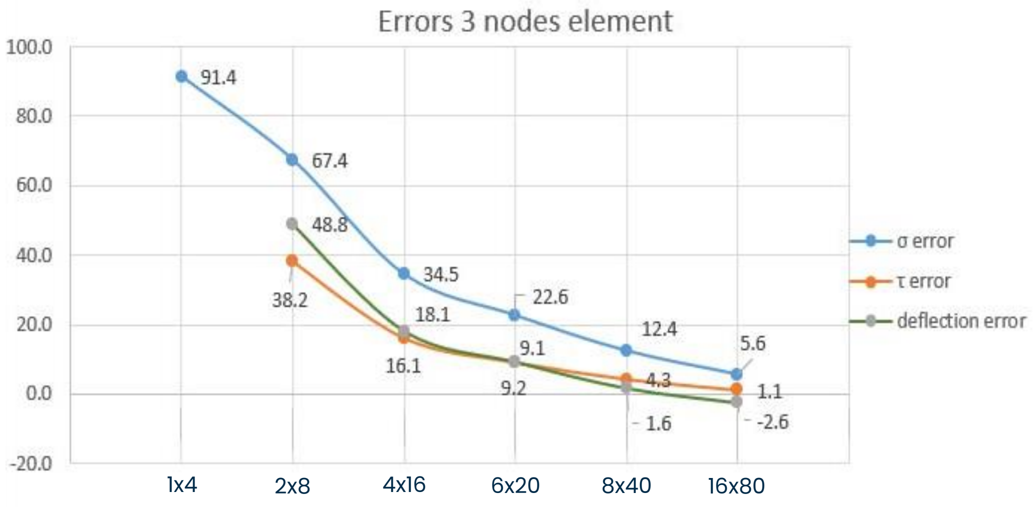
| Normal stress s |	| Shear stress t |	| Deflection |
at half beam lenght	at half beam lenght	at half beam lenght
Mpa	Mpa	mm
140,6	14,91	0,1579

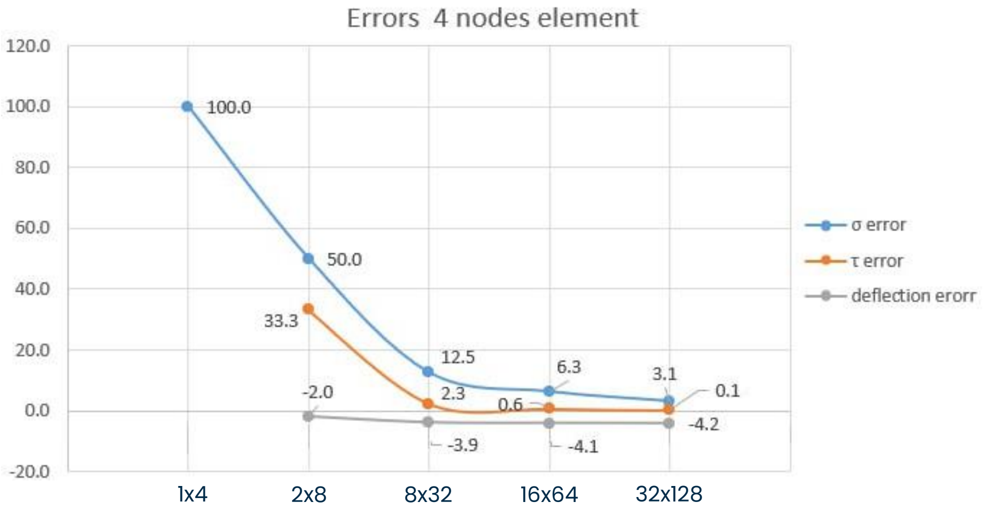
Conclusions

The results of the four different load cases are almost the same, proving that the most significant mesh was **properly chosen**.

Moreover, the importance of the mesh size can be highlighted by looking at the results of the previous analyses with few elements, which are far from the theoretical values due to **low accuracy**.

It is also notable from the charts of the relative errors that the 4 nodes elements predict better the results with respect to the 3 nodes elements, requiring **fewer elements** to achieve the same results.





NMS EXERCISE 2

Rod engine verification

Students:		
	COLUCCI Carlo Vittorio	329703
	COVETTI Alessio	329876
	PLACIDA Pierpaolo	323197

Academic year 2023/24

Politecnico di Torino

1859

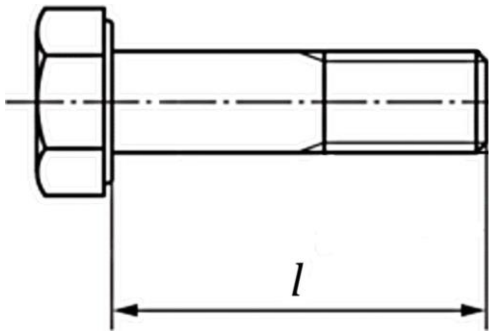
Project overview

Subject of the study:
Rod of an internal combustion engine.

Goals of the study:

- Characterization of the different load cases involved in the component working life;
- Verification of the component reliability under heavy static load conditions.

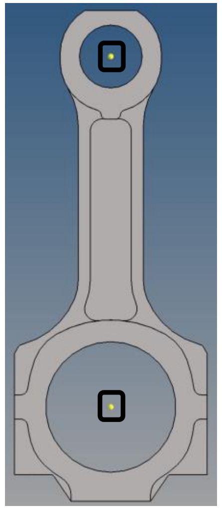
Clamping condition:
Big eye bottom part joint by means of two bolts M8x1.25: $l = 35mm$ (standardised length).

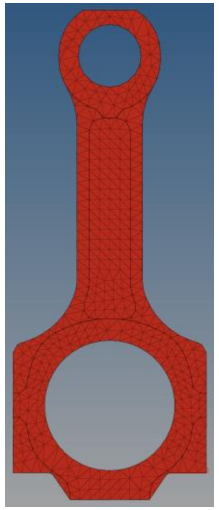


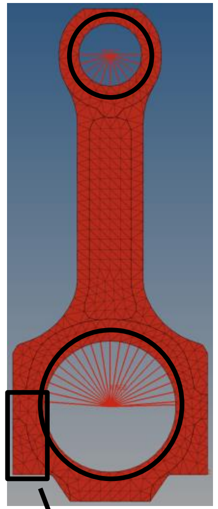
Model generation

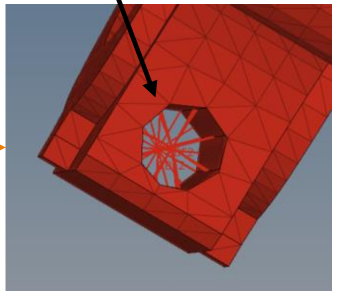
Material and property definition → Nodes positioning → 3D mesh → RBE2 (rigid elements) positioning → Constraints and loads application → Load step definition

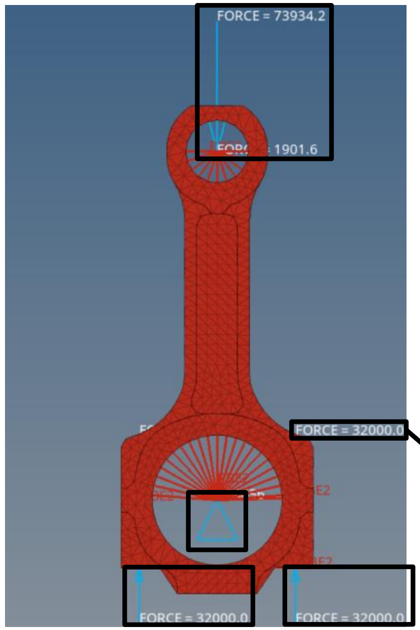
Name:	Cast iron
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comments:	Hide In Menu/Export
E:	165000.0
G:	
NU:	0.28
RHO:	7.2e-09





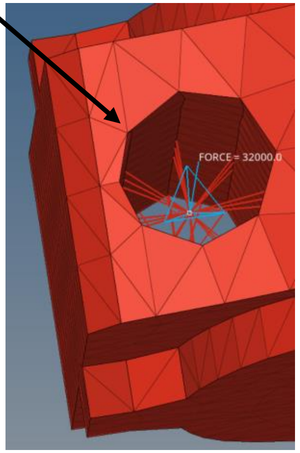






Load Steps (1)
Constraint and loads 1

SPC:	(1) Constraints & loads
LOAD:	(1) Constraints & loads



Inside the screw hole, to take into account the 90% of the load distribution in the threads: $l_{RBE2} = 6 \cdot pitch_{M8} = 6 \cdot 1.25mm.$

Load cases

Load case at engine rotational speed of 1900rpm:

- Force related to the maximum pressure on the piston, enacted by the fuel combustion:

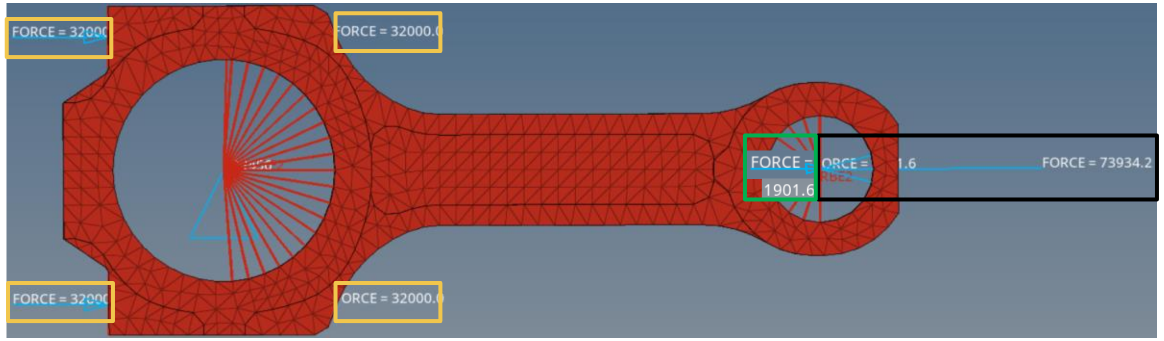
$$F_{p_{max}} = S_{piston} \cdot p_{max} = \underline{-73934.2N}$$

- Inertia force of the piston:

$$F_i = m_{piston} \cdot a_{TDC,\,1900} = \underline{+1901.6N}$$

- Clamping loads:

$$F_c = \underline{\pm 32000N}$$



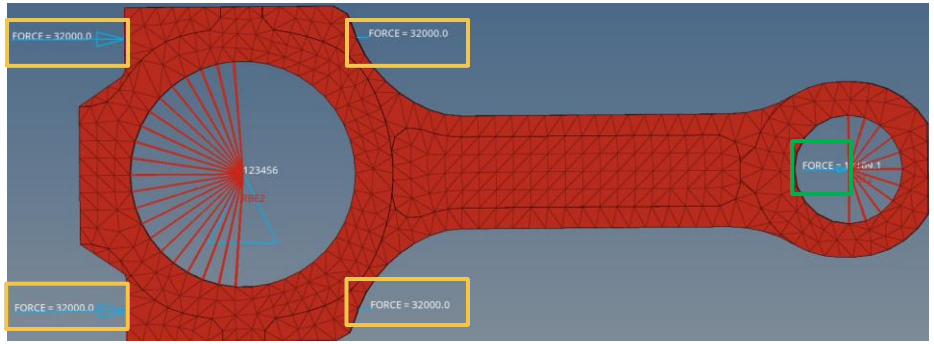
Load case at engine rotational speed of 5000rpm:

- Inertia force of the piston:

$$F_i = m_{piston} \cdot a_{TDC,\,5000} = \underline{+13169.1N}$$

- Clamping loads:

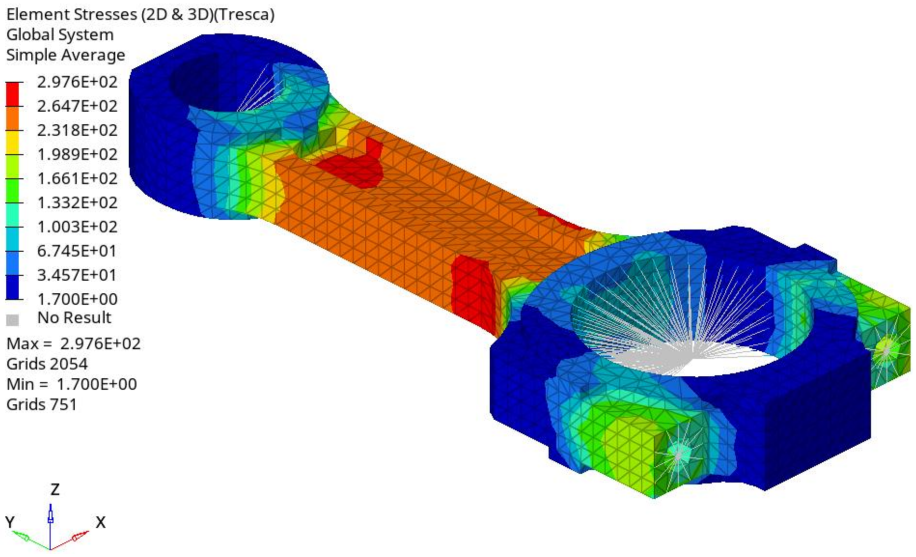
$$F_c = \underline{\pm 32000N}$$



Postprocessing analysis

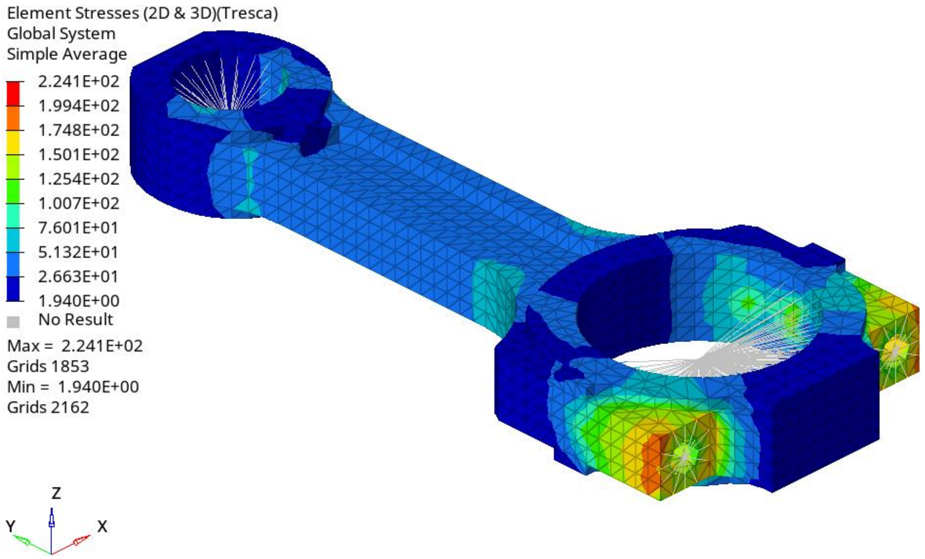
Initial mesh adopted to carry out the simulations: 4mm, R-trias 1° order elements.

Stress evaluation criteria: Tresca (conservative evaluation).



1° load case

Stress concentration along the rod stem, that is the link where the load is transferred from the small eye to the big constrained one.



2° load case

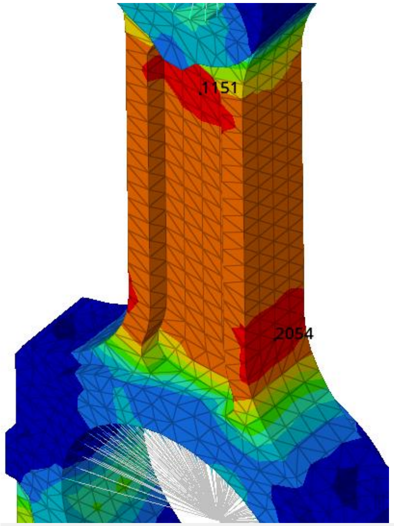
Stress concentration in the clamped zone much higher than the distribution on the rod stem, loaded only by the piston inertia force.

Component verification

Cast iron average mechanical parameters:

- Tensile and compressive yield strength: $\sigma_{T,y} = 428MPa$, $\sigma_{C,y} = 991MPa$.

1° load case



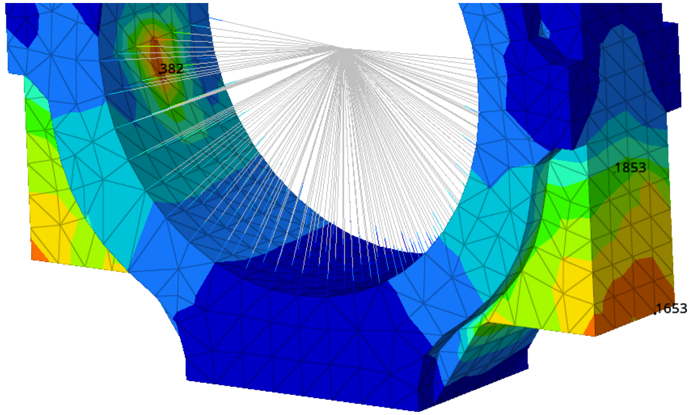
Stress intensification in the geometry variation zones.

$$SF = \frac{\sigma_{C,y}}{\sigma_{Tresca,\,MAX}} = \frac{991}{297.6} = 3.3$$

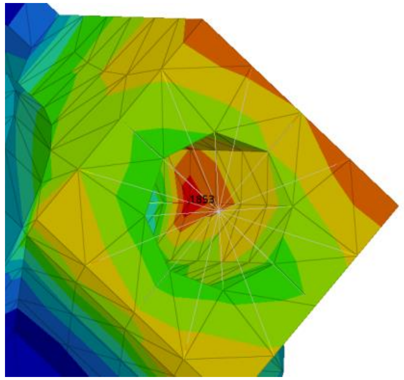
Node ID	Node Coordinates	Contour(Element Stresses (2D & 3D))
2054	-13.5041 52.3716 10.9999	2.976E+02
1151	-0.000462959 106.471 7.49857	2.709E+02

The safety factor is computed considering the compressive yield strength, because the rod, in both cases, is loaded at the most in compression.

2° load case



Stress intensification nearby the clamping region and in sharp zones.



$$SF = \frac{\sigma_{C,y}}{\sigma_{Tresca,\,MAX}} = \frac{991}{224.1} = 4.4$$

Node ID	Node Coordinates	Contour(Element Stresses (2D & 3D))
1853	35.7025 -11.4671 7.83785	2.241E+02
382	-26.3227 -6.00944 11	2.001E+02
1653	40.004 -27.9833 7.00016	1.913E+02

Results discussion

The 2° load case is the most relevant because, by the postprocessing analysis, a non-uniform stress distribution is highligthed on the rod along:

- The big eye internal face, where the constraint unloads the clamping forces;
- The inner part of the screw holes, where the load is applied by means of the threads;
- The sharp edges at the big eye bottom surface.

Three analyses are undertaken, with the goal to get closer to the real phenomenon and to smoothen the stress distribution:

- Different constraint positioning; } Boundary condition
- Downsizing of the mesh: $4mm \rightarrow 2mm$;
- Increase of the element order: 1° order → 2° order.

(Downsizing of the mesh and increase of the element order: } Mesh quality)

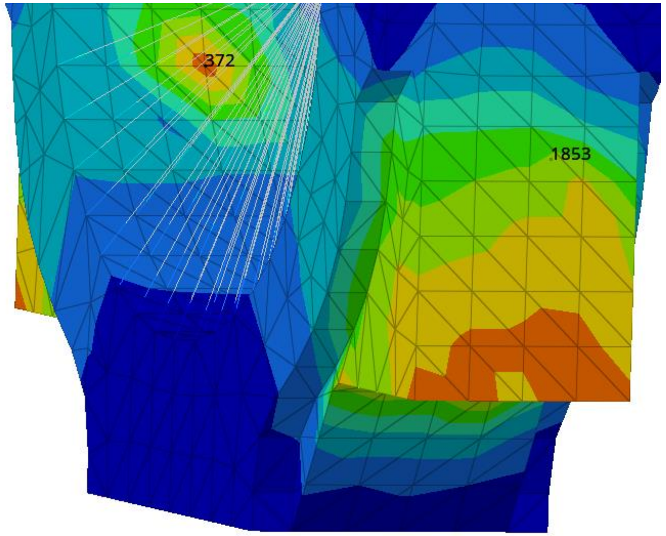
The same values for the captions are selected, to compare the differences.

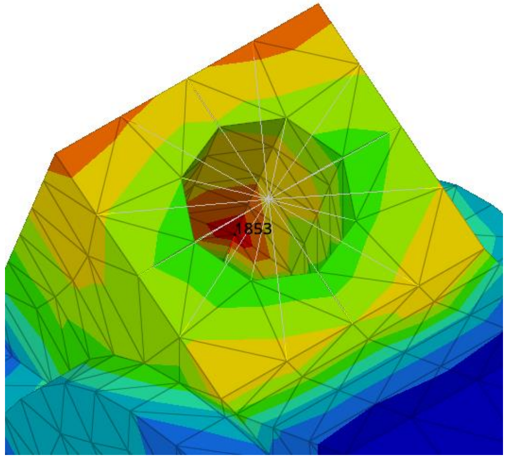
Different constraint positioning

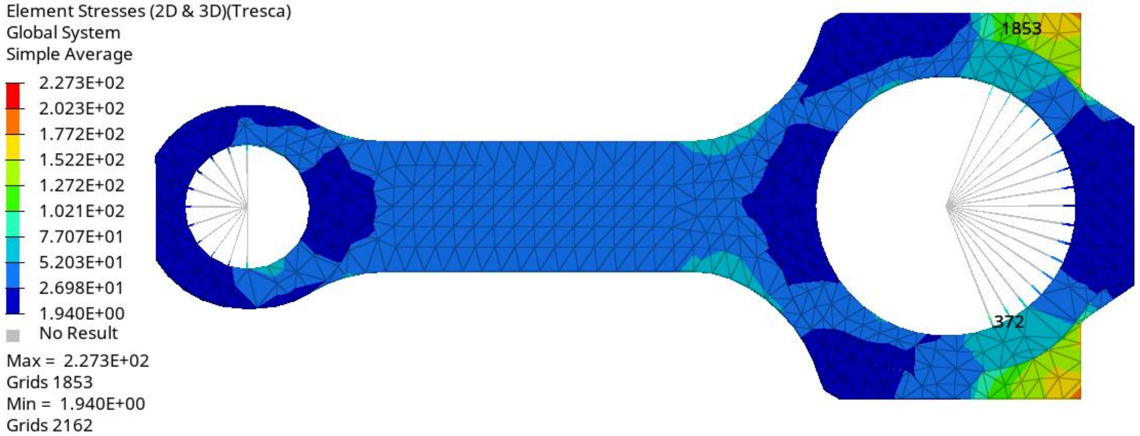
Expectation: lack of stress intensification inside the big eye.

Results:

- Little stress distribution improvement in the internal big eye surface, attending a lower peak stress too: $200.1MPa \rightarrow 196MPa$;
- Worse stress level in the screw holes, where the load is applied: $224.1MPa \rightarrow 227.3MPa$. The stress around the threads unloads in a narrower space in this case, thus featuring this higher stress concentration.







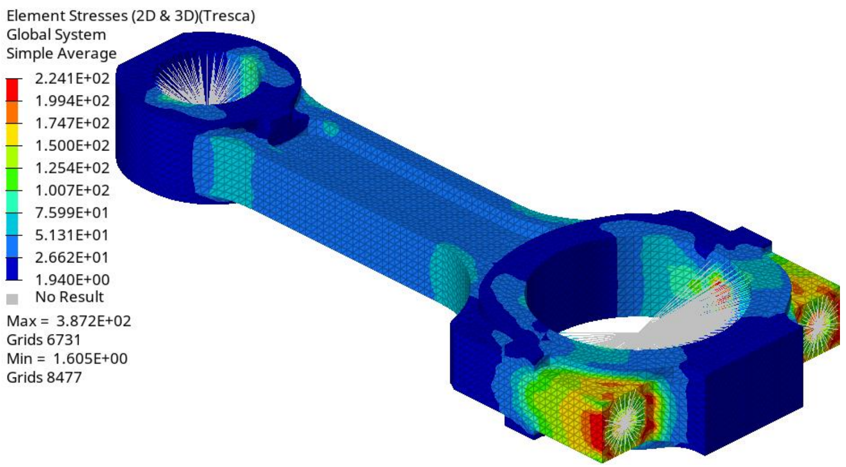
Node ID	Node Coordinates	Contour(Element Stresses (2D & 3D))
372	-25.1272 -9.88046 11	1.960E+02
1853	35.7028 -11.4682 7.8377	2.273E+02

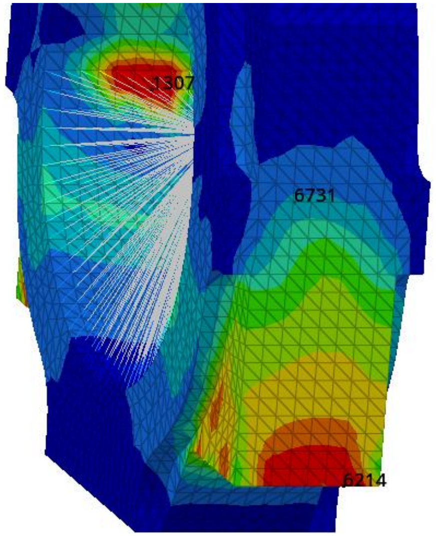
Mesh downsizing

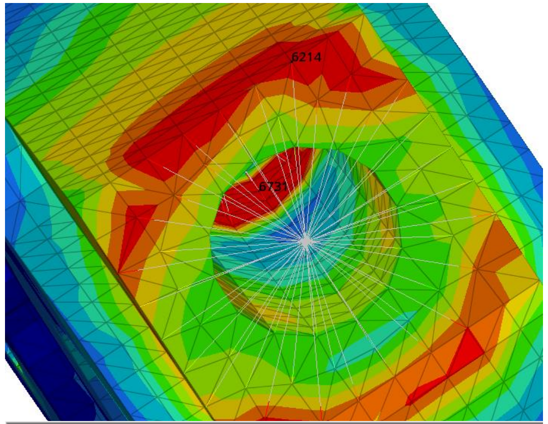
Expectation: better discretization → accuracy ↑.

Results:

- Higher overall maximum stress: $224.1MPa \rightarrow 387.2MPa$;
- Stress intensification along the sharp edges and the geometry variations: $191.3MPa \rightarrow 224.8MPa$;
- Better stress distribution along the internal surface of the big eye, complying with the expected simmetry with respect to the Y-midline, but with higher stresses: $200.1MPa \rightarrow 307.2MPa$;
- Higher stress values inside the screw holes: $224.1MPa \rightarrow 387.2MPa$.







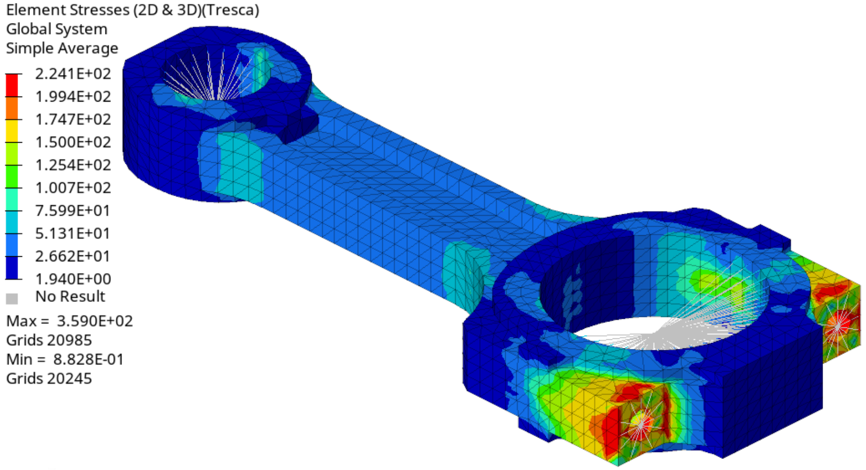
Node ID	Node Coordinates	Contour(Element Stresses (2D & 3D))
6731	27.9997 -0.191351 11	3.872E+02
6214	40.0059 -27.9754 7.00009	2.248E+02
1307	-26.9246 -2.01578 9.99999	3.072E+02

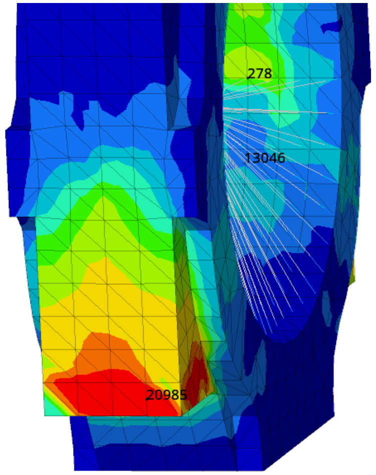
Increase of the element order

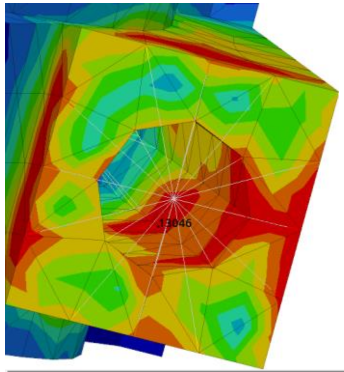
Expectation: DOFs per element ↑ (better estimation of the displacements field) → accuracy ↑.

Results:

- Higher overall maximum stress: $224.1 MPa \rightarrow 359 MPa$;
- Severe stress intensification along the sharp edges and the geometry variations: $191.3 MPa \rightarrow 359 MPa$;
- Better stress distribution along the internal surface of the big eye, complying, in this study case too, with the expected simmetry with respect to the Y-midline. Difference in the peak stress attending a lower value: $200.1 MPa \rightarrow 125.2 MPa$;
- Higher stress values inside the screw holes: $224.1 MPa \rightarrow 311.5 MPa$.







Node ID	Node Coordinates	Contour(Element Stresses (2D & 3D))
278	26.9252 -2.0087 11	1.252E+02
20985	-40.0049 -26.0644 15.001	3.590E+02
13046	35.703 -13.5298 7.83676	3.115E+02

Conclusions

Study zones	2° load case	Different constraint positioning	Mesh downsizing	2° order elements
Maximum value	224.1 MPa	227.3 MPa	387.2 MPa	359 MPa
Sharp edges and the geometry variations	191.3 MPa	-	224.8 MPa	359 MPa
Internal big eye surface	200.1 MPa	196 MPa	307.2 MPa	125.2 MPa
Screw holes	224.1 MPa	227.3 MPa	387.2 MPa	311.5 MPa

The mesh downsizing and the 2° order simulations show smoother stresses distributions and results quite similar among them, standing for a convergence of the simulation to the real phenomenon. Thus, the new retrieved values can be considered as reference ones.

A new safety factor is computed:

$$SF = \frac{\sigma_{C,y}}{\sigma'_{Tresca,\,MAX}} = \frac{991}{387.2} = 2.56$$

NMS EXERCISE 3

Study of a commercial vehicle frame

Students: COLUCCI Carlo Vittorio 329703
COVETTI Alessio 329876
PLACIDA Pierpaolo 323197

Academic year 2023/24

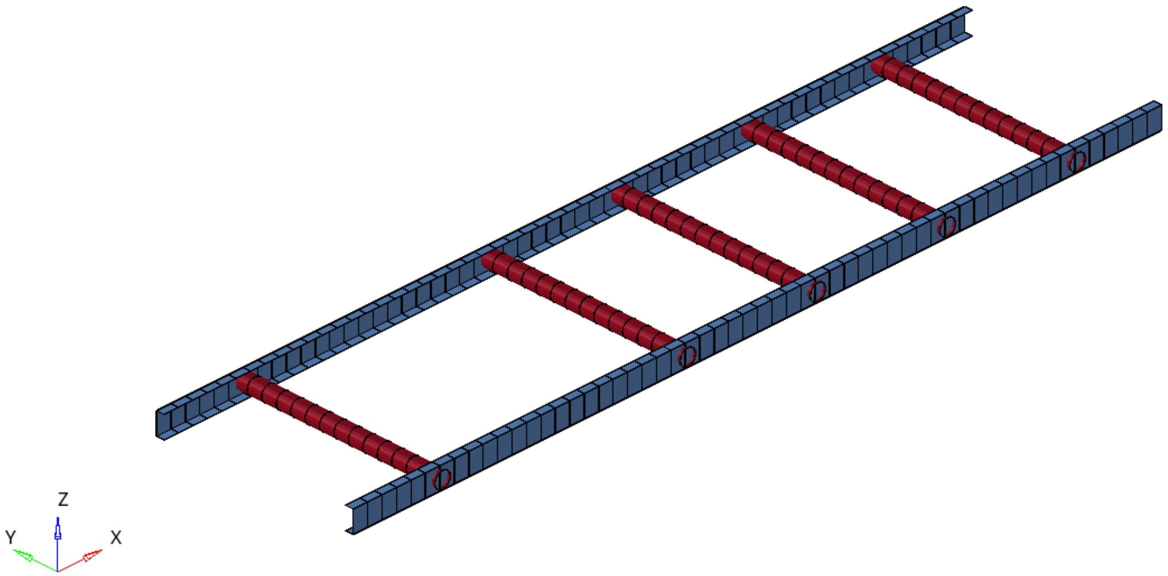
Politecnico di Torino
1859

Targets

This project aims to evaluate the **bending** and **torsional** stiffnesses of a commercial vehicle frame.

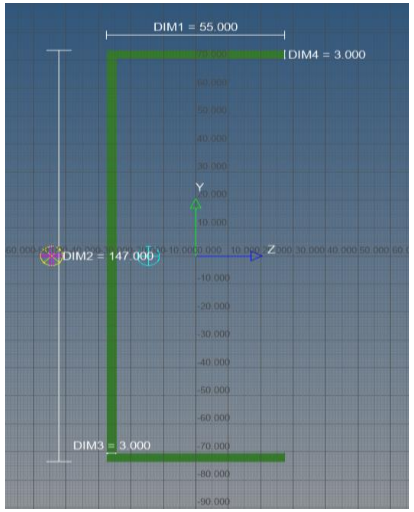
In order to investigate the structure properties, a **1-D elements** approach is exploited.

Moreover, to understand the relation between the main model parameters and stiffness, some **variations** in the frame properties will be analyzed.



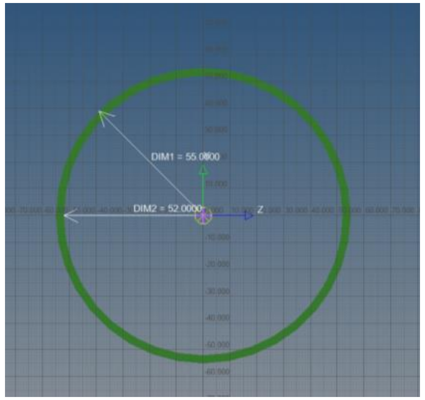
Model creation

Cross-sections definition → Material and properties definition → Line mesh 1-D elements



"Channel section" for the longitudinal beams

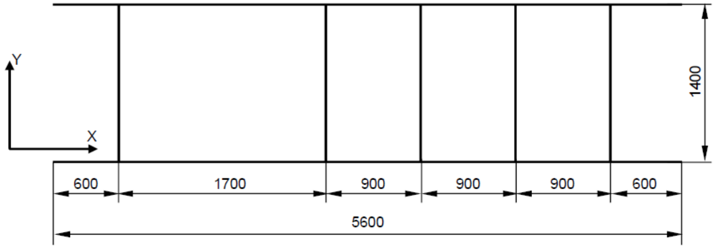
"Tube section" for the cross beams

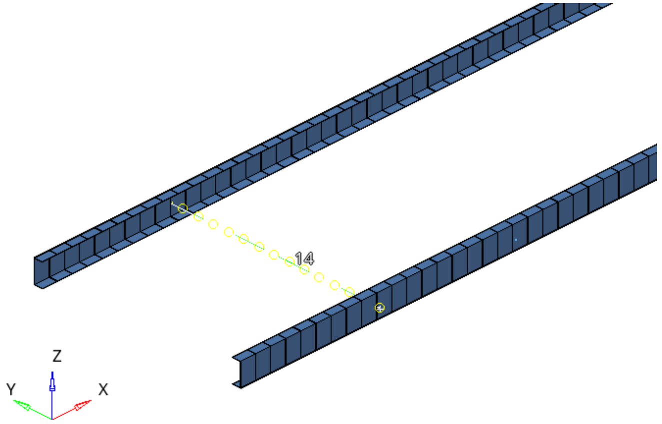


Name:	steel
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comments:	Hide In Menu/Export
E:	206000.0
G:	
NU:	0.3
RHO:	7.85e-09

ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	PBEAML
Material:	(1) steel
User Comments:	Hide In Menu/Export
Beam Section:	(1) channel_section.1
pbeamlintStationslen =:	0
SECTION_FOR_VABS:	☐
TYPE:	CHAN
DIM1A:	55.0
DIM2A:	147.0
DIM3A:	3.0
DIM4A:	3.0

ID:	2
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	PBEAML
Material:	(1) steel
User Comments:	Hide In Menu/Export
Beam Section:	(2) tube_section.2
pbeamlintStationslen =:	0
SECTION_FOR_VABS:	☐
TYPE:	TUBE
DIM1A:	55.0
DIM2A:	52.0



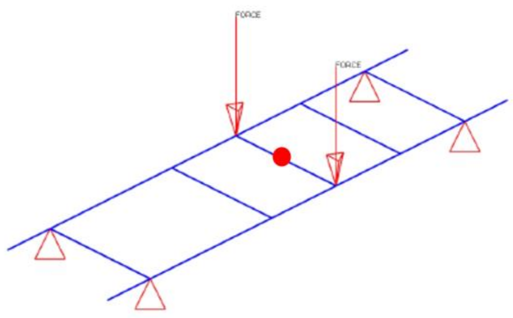


Element size 100 mm

Description of applied loads and constraints

Given the load conditions, it is supposed a **Center of Gravity** at half length of the third cross beam. Being an **industrial vehicle**, it is assumed a full load case of **4000 kg** total mass.

Bending load case

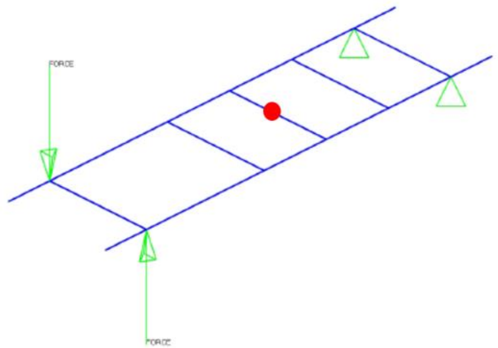


Constraints: At **all** suspension attachments. All degrees of freedom set to zero, except the rotation **around y axis**.

Loads: Two forces applied as shown, representing **vehicle weight**.

$$F_b = \frac{m \cdot g}{2} = 19620\ N$$

Torsion load case



Constraints: Only at **rear** suspension attachments. All degrees of freedom set to zero, except the rotation **around x axis**.

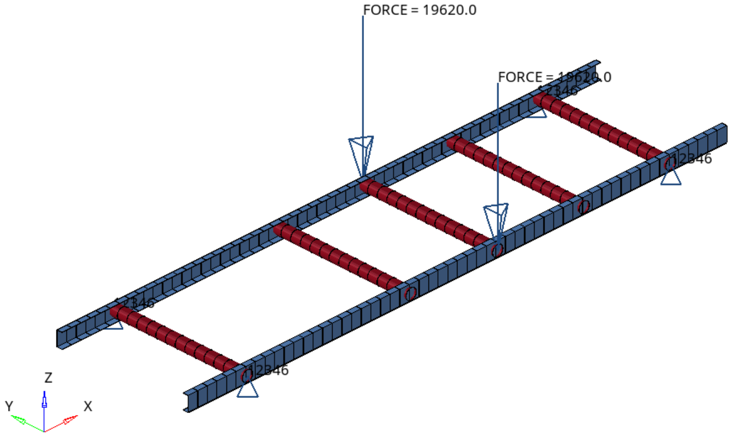
Loads: Two forces applied as shown. Values come from **load transfer** considerations.

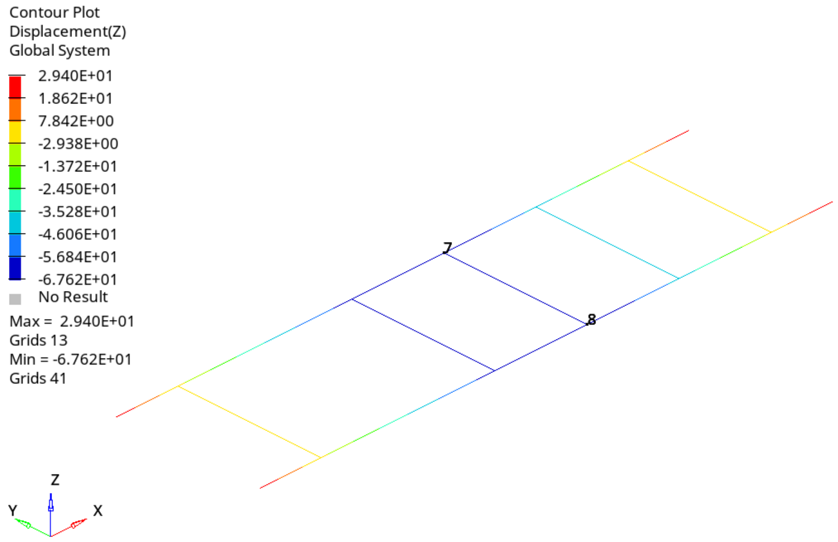
$$\frac{F_t}{2} = \frac{m \cdot g \cdot b}{2 \cdot W} = 8000\ N$$

Bending stiffness evaluation

Bending stiffness is defined as the ratio of **total applied load** on vehicle frame and the relative **displacement** in the plane of application of the force:

$$K_b = \frac{2 \cdot F_b}{|\delta_z|} = \frac{2 \cdot 19620\ N}{66{,}15\ mm} = 593{,}2\ N/mm$$





Torsional stiffness evaluation

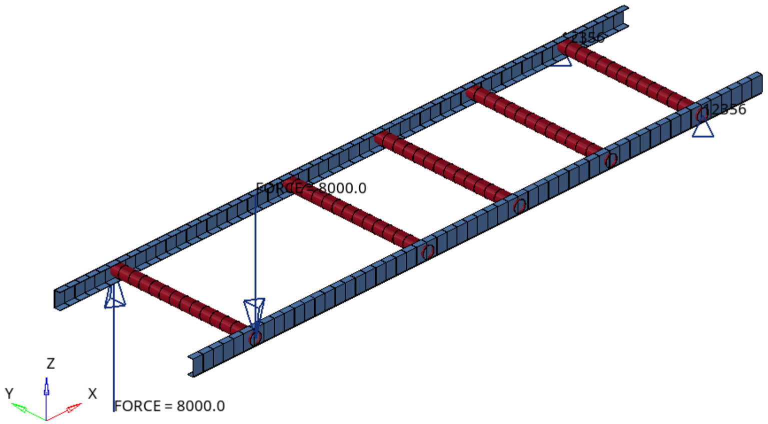
Torsional stiffness is defined as the ratio of **total applied torque** on the front suspensions plane and the relative **twist angle** of the frame in the plane of application of the force:

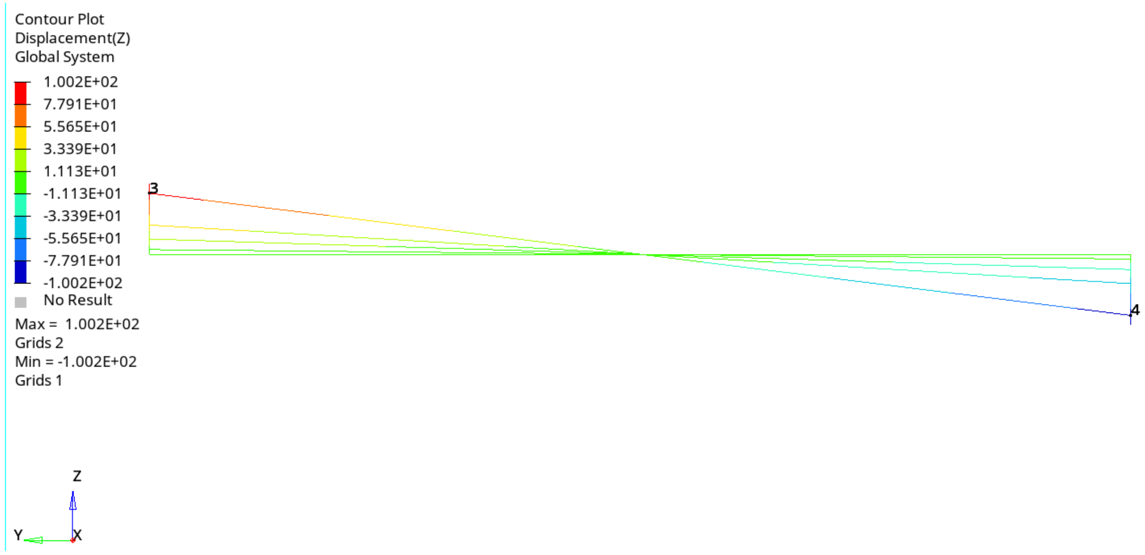
$$K_t = \frac{F_t \cdot t}{\vartheta_t} = \frac{8000\,N \cdot 1400\,mm}{0.1237\,rad}$$

$$= 90535{,}8\,Nm/rad$$

where:

$$\vartheta = \tan^{-1}\frac{|\delta|}{t/2} = \tan^{-1}\frac{87{,}04\,mm}{\frac{1400}{2}\,mm} = 7{,}1\,°$$





Mesh and material variation

Same results previously obtained have been confirmed **decreasing** the element size from 100 to 10 mm.

New study: Material change to **Aluminum**.

Name:	steel
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comments:	Hide In Menu/Export
E:	206000.0
G:	
NU:	0.3
RHO:	7.85e-09

→

Name:	Aluminum
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comments:	Hide In Menu/Export
E:	70000.0
G:	
NU:	0.33
RHO:	2.7e-09

Expectations: Lower bending and torsional stiffness, due to the lower **Young Modulus**.

	Bending deflection [mm]	Bending stiffness [N/mm]	Torsion deflection [mm]	Twist angle [°]	Torsional stiffness [Nm/rad]
Steel	-66,15	593,2	87,04	7,1	90535,8
Aluminum	-194,7	201,5	260,4	20,4	31448,5

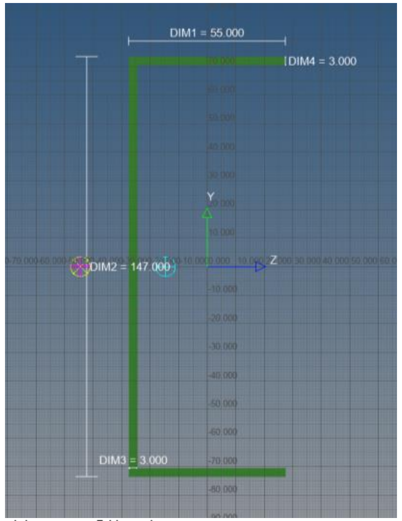
Analysis with a cross section variation

Goal: Define a **more rigid** structure.

Boundary conditions: Same **external dimensions** and **thickness** of previous analyses.

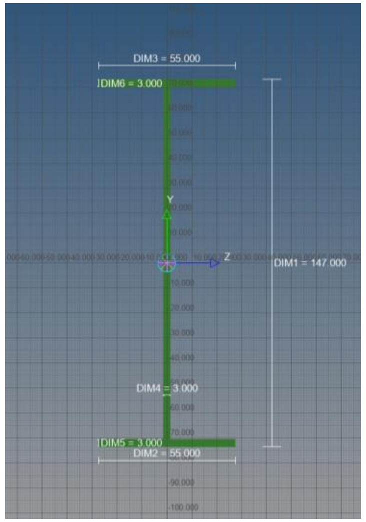
Method: **Cross section** variation analysis, seeking for highest relative **moments of inertia**.

Longitudinal beam hypothesis



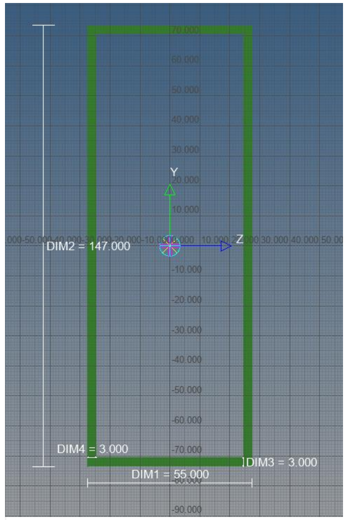
Moments Of Inertia :

Local		
IY	=	369452.7500
IZ	=	2411772.7500
IYZ	=	0.0000



Moments Of Inertia :

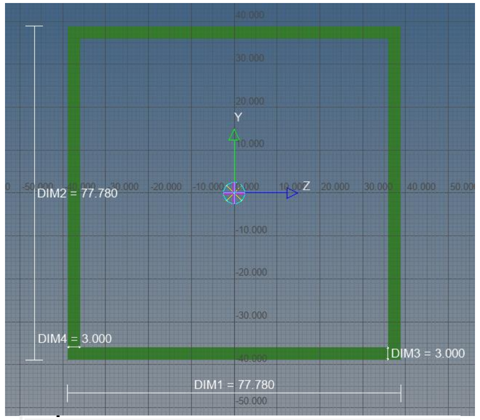
Local		
IY	=	83504.7500
IZ	=	2411772.7500
IYZ	=	0.0000



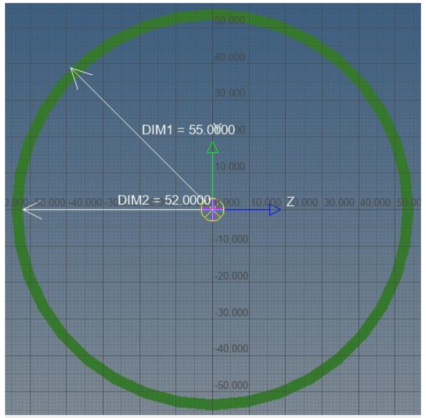
Moments Of Inertia :

Local		
IY	=	655718.0000
IZ	=	3112578.0000
IYZ	=	0.0000

Cross beam hypothesis

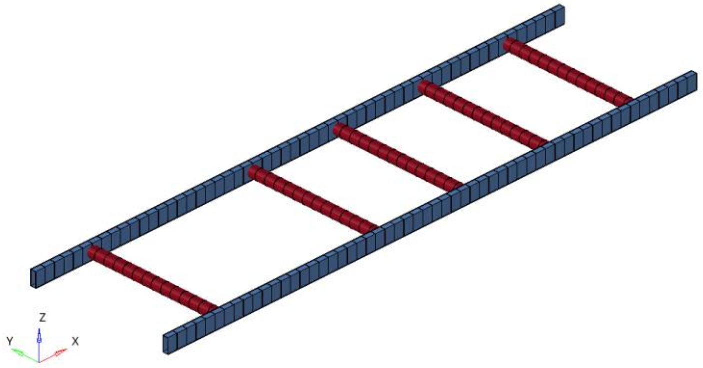


Principal		
Iv	=	837692.7987
Iw	=	837692.7987
Angle	=	0.0000
Polar	=	1675385.5974
Radius of Gyration	=	30.5534
Torsional Constant	=	1254520.1381



Principal		
Iv	=	1444354.2911
Iw	=	1444354.2911
Angle	=	0.0000
Polar	=	2888708.5821
Radius of Gyration	=	37.8451
Torsional Constant	=	2888708.5821

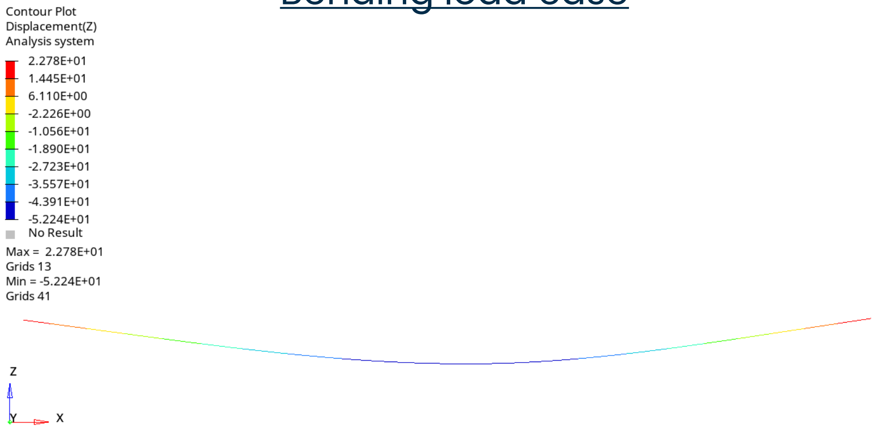
Results discussion



Longitudinal beams → Box cross section

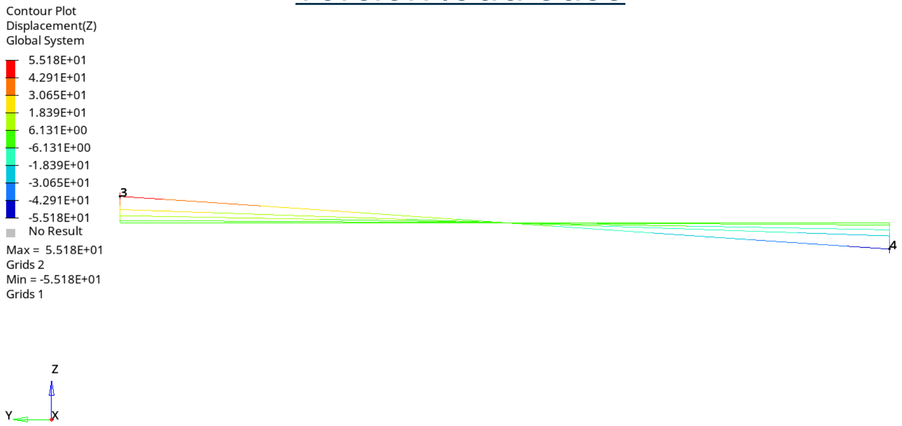
Cross beams → Tube cross section

Bending load case



$$K_b' = \frac{2 \cdot F_b}{|\delta_z'|} = \frac{2 \cdot 19620\ N}{51{,}08\ mm} = 768{,}2\ N/mm$$

Torsion load case



$$K_t' = \frac{F_t \cdot t}{\vartheta_t'} = \frac{8000\ N \cdot 1400\ mm}{0.0684\ rad} = 163793{,}2\ Nm/rad$$

Conclusions

Comparison:

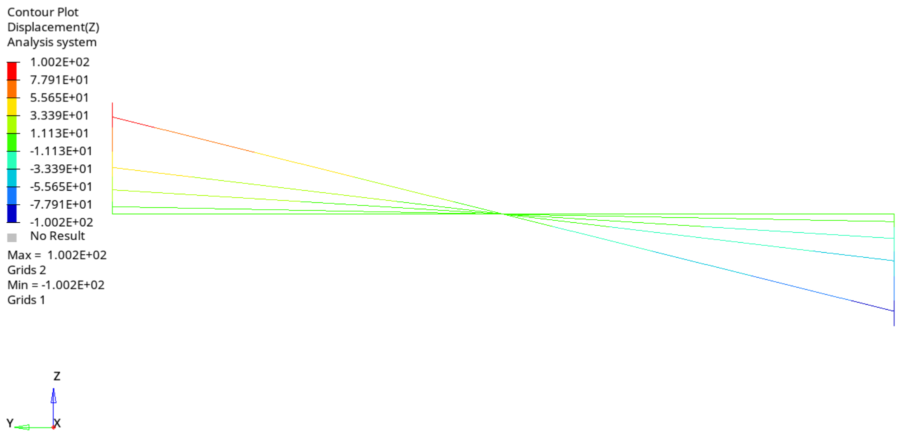
	Bending stiffness [N/mm]	Torsional stiffness [Nm/rad]
STEEL (100 mm element size)	593,2	90535,8
ALUMINUM (100 mm element size)	201,5	31448,5
STEEL (New cross section)	768,2	163793,2

- By having lower values of E and G, the relative stiffnesses of Aluminum structure are lower.
- Bending and torsional stiffness vary accordingly to hypothesis concerning the moments of inertia related to the specific load case.

Observation:

Considering the torsional loading case, the frame rotates around the x-axis, but its longitudinal beams only bend in their plane (XZ) registering a null y-axis displacement.

This issue may result by the elementary type applied mesh: 1D line mesh.



NMS EXERCISE 4

Car body T-joint stiffness evaluation

Students: COLUCCI Carlo Vittorio 329703

COVETTI Alessio 329876

PLACIDA Pierpaolo 323197

Academic year 2023/24

Politecnico di Torino

1859

Project overview

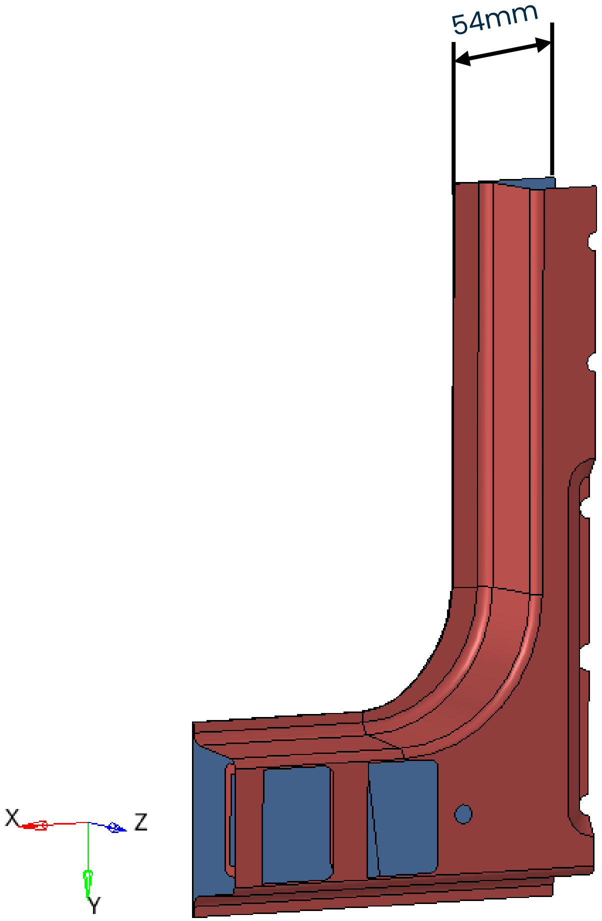
Subject of the study:
Car body T-joint.

Clamping condition:

- Top sheet joint to the bottom one by means of spot welds;
- T-joint lower part constrained to emulate the rigid body behaviour of the vehicle sills.

Goals of the study:

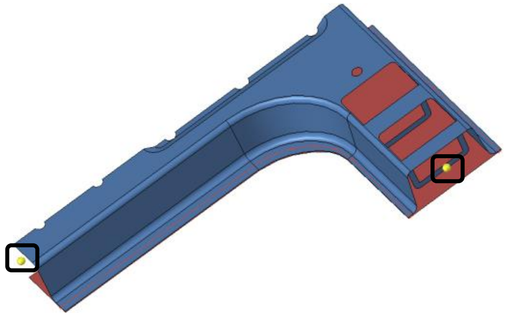
- Evaluation of the T-joint bending stiffness around the X and Z axes (model FOR) and of its torsional stiffness around the Y axis;
- Evaluation of the fluctuation of the different stiffness values, when changing the number of spot welds.

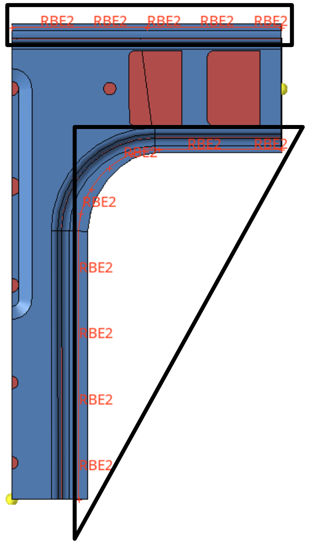


Model generation

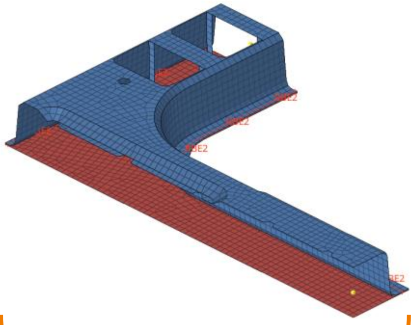
Material and property definition → Nodes and reference lines generation → Spot welds positioning → 2D mesh (5mm elements) → Model reflection →

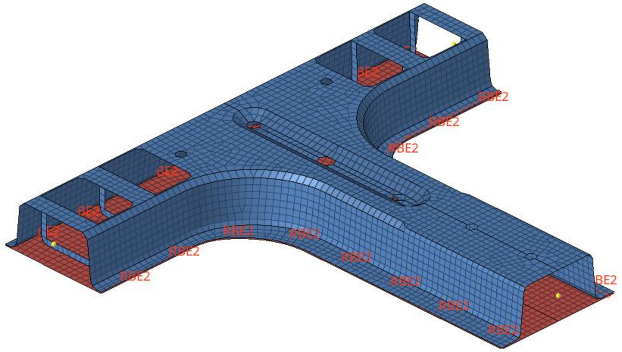
Name:	Steel
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	MAT1
User Comme...	Hide In Menu/
E:	206000.0
G:	
NU:	0.3
RHO:	7.85e-09
Name:	Steel property
ID:	1
Color:	
Include:	[Main Model]
Defined:	☑
Card Image:	PSHELL
Material:	(1) Steel
User Comme...	Hide In Menu/
T:	1.0





The spot welds are simulated as rigid elements.

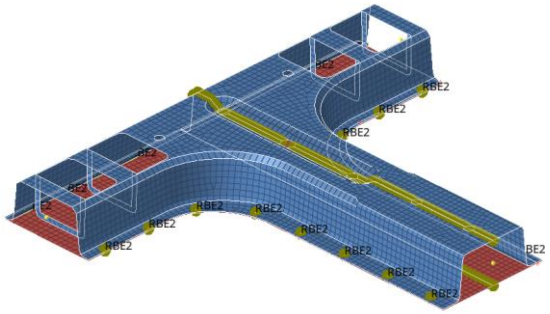


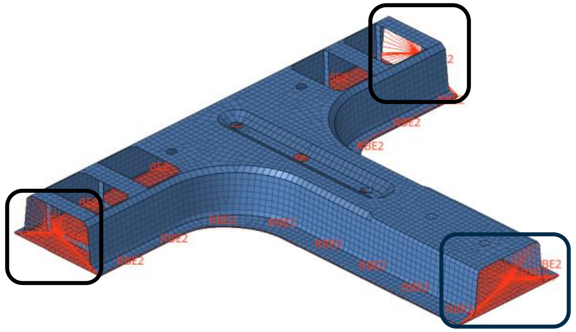


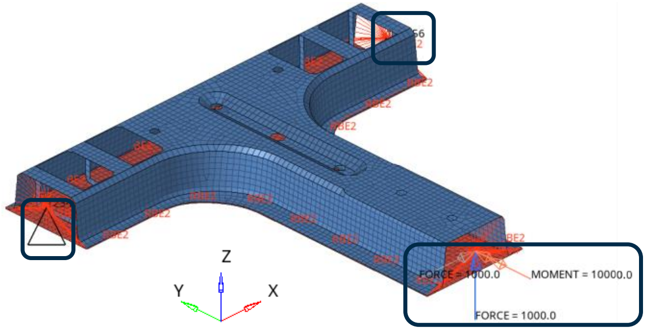
The starting model is just half component.

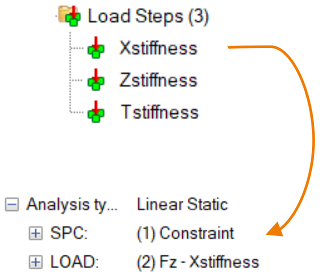
Model generation

→ Edge equivalence → RBE2 (rigid elements) positioning → Constraints and loads application → Load step definition









- Continuity of the 2D mesh must be ensured at the simmetry plane;
- The spot welds rigids must be attached to the mesh.

Load cases

Three different loads to evaluate the T-joint stiffness:

- Load along the Z-direction → bending stiffness around the X axis:

$$F_Z = 1000N$$

- Load along the X-direction → bending stiffness around the Z axis:

$$F_X = 1000N$$

- Moment around the Y-direction → torsional stiffness:

$$M_Y = 10000N \cdot mm$$

Three different spot welds spacings at constant end offset (20mm):

- 30mm;
- 50mm (reference value);
- 70mm.

Three simulations with these spacing values are undertaken and the postprocessing results are examined.

50mm spot welds spacing

Bending stiffness direction: X axis;

Maximum Z displacement value: 24.36mm.

$$K_{X,bending} = \frac{F_Z}{f_Z} = 41.1N/mm$$

Bending stiffness direction: Z axis;

Maximum X displacement value: 0,2293mm.

$$K_{Z,bending} = \frac{F_X}{f_X} = 4361.1N/mm$$

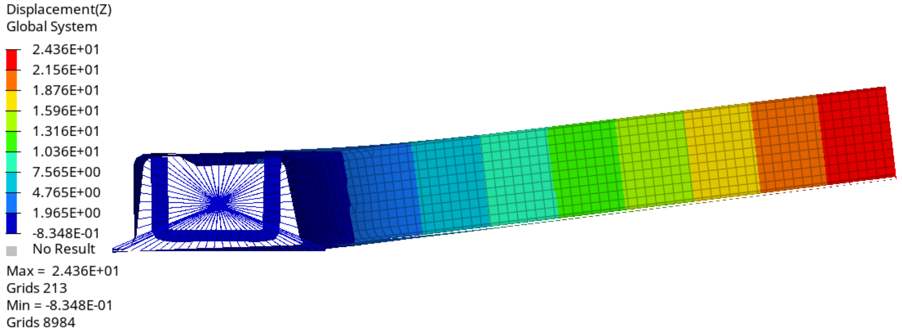
Torsion stiffness direction: Y axis;

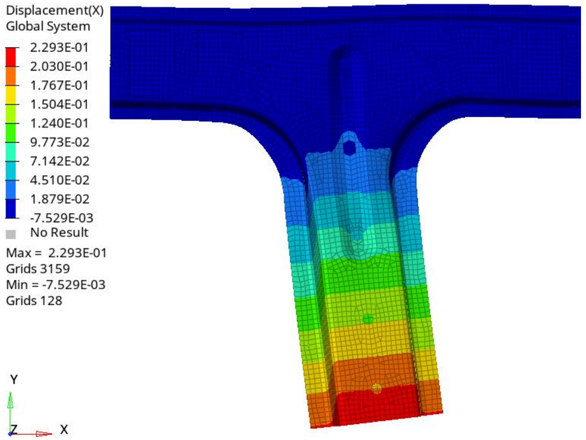
Maximum Z displacement value: 0,031mm.

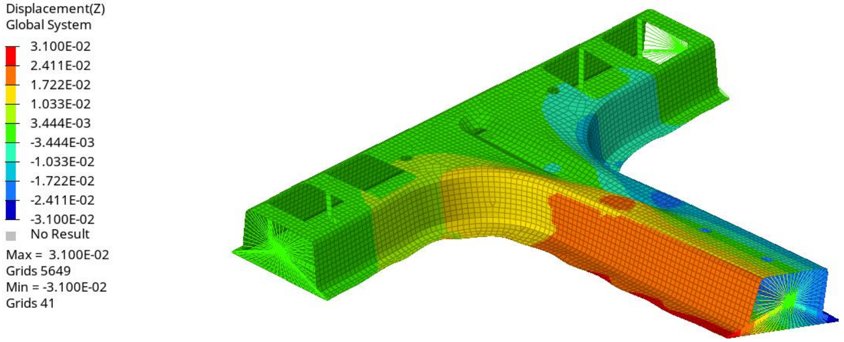
$$K_{Y,torsion} = \frac{M_Y}{\varphi_Y} = \frac{M_Y}{\sin^{-1}\left({}^{d_Z}/_{l}\right)} = \frac{10000N \cdot mm}{5.74 \cdot 10^{-4} rad}$$

$$= 1.74 \cdot 10^{7}\ N \cdot mm/rad$$

l =54mm







70mm spot welds spacing

Expectation: lower overall stiffness.

Bending stiffness direction: X axis;

Maximum Z displacement value: 30.44mm.

$$K_{X,bending} = \frac{F_Z}{f_Z} = 32.9N/mm$$

Bending stiffness direction: Z axis;

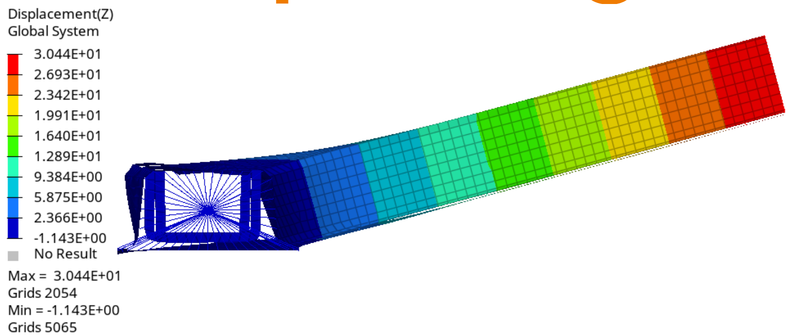
Maximum X displacement value: 0,2421mm.

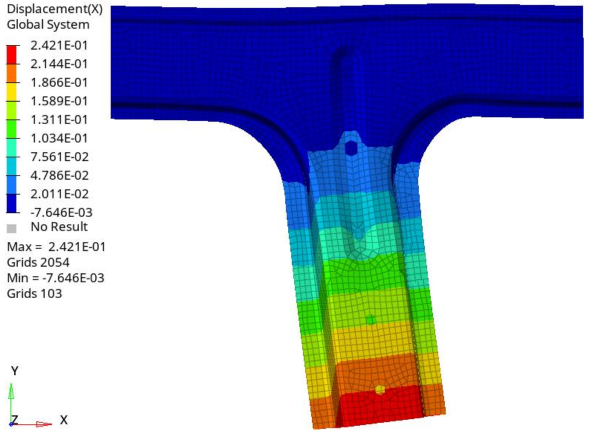
$$K_{Z,bending} = \frac{F_X}{f_X} = 4130.5N/mm$$

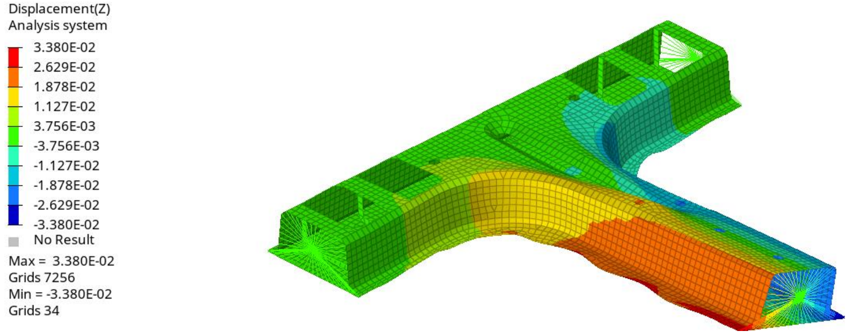
Torsion stiffness direction: Y axis;

Maximum Z displacement value: 0,0338mm.

$$K_{Y,torsion} = \frac{M_Y}{\varphi_Y} = \frac{M_Y}{\sin^{-1}\left({}^{d_Z}/_{l}\right)} = \frac{10000N \cdot mm}{6.26 \cdot 10^{-4} rad}$$
$$= 1.60 \cdot 10^{7}\ N \cdot mm/rad$$







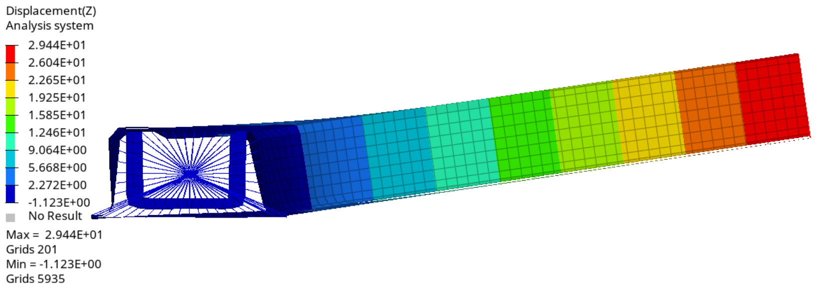
30mm spot welds spacing

Expectation: higher overall stiffness.

- **Bending stiffness direction:** X axis;

  Maximum Z displacement value: 29.44mm.

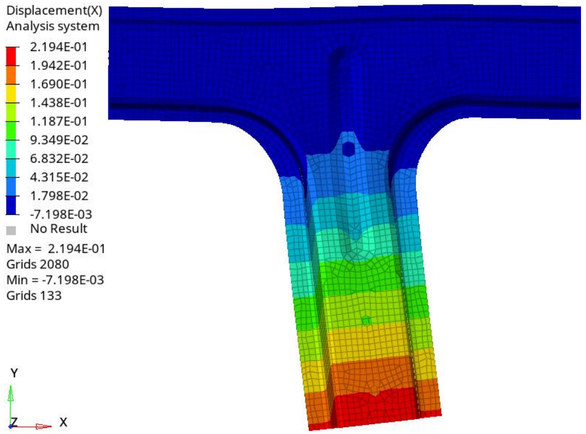
$$K_{X,bending} = \frac{F_Z}{f_Z} = 34N/mm$$



- **Bending stiffness direction:** Z axis;

  Maximum X displacement value: 0,2194mm.

$$K_{Z,bending} = \frac{F_X}{f_X} = 4557.9N/mm$$

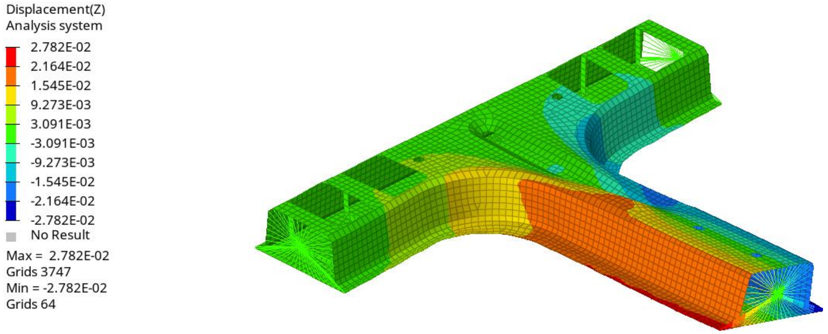


- **Torsion stiffness direction:** Y axis;

  Maximum Z displacement value: 0,02782mm.

$$K_{Y,torsion} = \frac{M_Y}{\varphi_Y} = \frac{M_Y}{\sin^{-1}\left({}^{d_Z}/_{l}\right)} = \frac{10000N \cdot mm}{5.15 \cdot 10^{-4} rad}$$

$$= 1.94 \cdot 10^7 \; N \cdot mm/rad$$



Conclusions

Spot weld spacing	Bending stiffness around the X axis	Bending stiffness around the Z axis	Torsional stiffness around the Y axis
30mm	$34 N/mm$	$4557.9 N/mm$	$1.94 \cdot 10^7\ N \cdot mm/rad$
50mm	$41.1 N/mm$	$4361.1 N/mm$	$1.74 \cdot 10^7\ N \cdot mm/rad$
70mm	$32.9 N/mm$	$4130.5 N/mm$	$1.60 \cdot 10^7\ N \cdot mm/rad$

Expected trend confirmed by the simulations:

Spot welds number ↑ → T-joint global stiffness properties ↑

Stiffness around the Z axis

Enough stiff to comply with the standards imposed for crash events

»

Stiffness around the X axis

Stiffness high enough to ensure the whole frame to behave such as a rigid box structure

Thanks for your attention!

For any question send an e-mail to the following contacts:
- Carlo Vittorio Colucci: s329703@studenti.polito.it;
- Alessio Covetti: s329876@studenti.polito.it;
- Pierpaolo Placida: s323197@studenti.polito.it.

Politecnico di Torino