

Castalia s.r.l. - all rights reserved - 1



#### **Keywords:**

steel connections, steel joints, welds, bolts, software, checks, verification, base plate, bending, compression, no tension, bearing, steel, yield, stress, strain, bolted connections, welded connections, anchors, slip resistant, plates, cleats, constraint, clamp, column, stiffener, fem analysis, fem models, fea, plate element, thickness, stress map, CSE, Castalia srl, steelchecks.com, castaliaweb.com, C.S.E.

#### Parole chiave:

connessioni acciaio, collegamenti acciaio, saldature, bullonature, bulloni, software, verifiche, piastra di base, flessione, compressione, no-tension, contrasto, supporto, acciaio, snervamento, sforzo, deformazione, connessioni bullonate, connessioni saldate, ancoraggi, unioni ad attrito, piastre, vincoli, incastro, colonna, irrigidimento, analisi fem, modelli fem, elemento piastra, spessore, mappa di sforzo, CSE, Castalia srl, steelchecks.com, castaliaweb.com, C.S.E.



# **1 INTRODUCTION**

This tutorial is a tool to start the understanding of how CSE works. No special efforts to have realistic force values have been made, however this will explain several features of the program. By following this tutorial you will be able to:

- Create a dummy fem model to study a constraint joint
- Assign the materials and cross-section to the fem elements
- Search members
- Search jnodes
- Add a generic polygon plate
- Add a bolt layout with free bolts positioning
- Add a weld layout
- Set the checks to be performed
- Have a general look at the results
- Understand bearing surface results

The connection here studied was designed not to be realistic, but in order to give a simple and clear example of a connection which needs an irregular polygon plate and a "free" disposition of the bolts.

This tutorial is some like 60 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

*N.B.* this tutorial refers to CSE version reported on the first page of this document. If you are using a newer version, keep in mind that some dialog or commands may be different, although the logic of the program has remained the same. If you find some differences, see the up-to-date PDF guide or the context sensitive help for information.



# **2 CONSTRAINT**

Initialization FULL version	x
-FULL MODE-	
FULL	
Using FULL version you can create freely new RENODES, and besides yu can record new parametric ones All LIGHT version commands are available, but the interface is not simplified	
LIGHT version uses a simplified interface but only standard nodes can be created, belonging to the database Proper values for parameters can be set. No new parametric RENODE can be recorded	

In the initial dialog box, choose the full mode (complete, with no limitations).

	iE - (CSE1)		
Se.	Ble Modify Display Draw Enquin	Eem (Nodes Benode Checks ID Model /Classes Joints Window 2	- 8 X
		SHIII≓⊨⊠à≌≶µA₽⊨♦⊅⊅₽₽∞≢≣⊠⊍∙=≈x>⊅Z& ±±₽ ≜≢₽ ₽	
_			
21	Variables and conditions	2	2 2 7
10	Renode Selected components	₹.	- W
120			8
21			*
11			
81			
N.			
No.			
Mars			
82			
验			
ints]			
×+			
>+			
511			
3X			
환교 國法主 러퍼킹 비행 전화 한 치지하여 메르아 티버베르에 空間 이			
E			
C			
2			
100			
-			
-			
<u> 1</u>			
<b>F</b>			
101			
10			
			-
		1× u	G=1
Tere	t info, press F1		mm N s 'C
.00	and been a		

Initial window content: blank.

## 2.1 STEP 1: GETTING THE FEM MODEL

### **2.1.1** Getting the fem elements

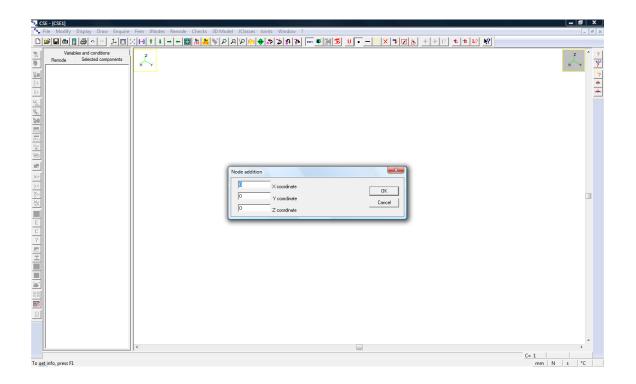
Activate right window by clicking left inside it, then execute the command **FEM-Elements-Add**: the following dialog box will appear.



Beam and truss addition		X
Element Trues	Section number	Mode
	?	Cancel vs and CR

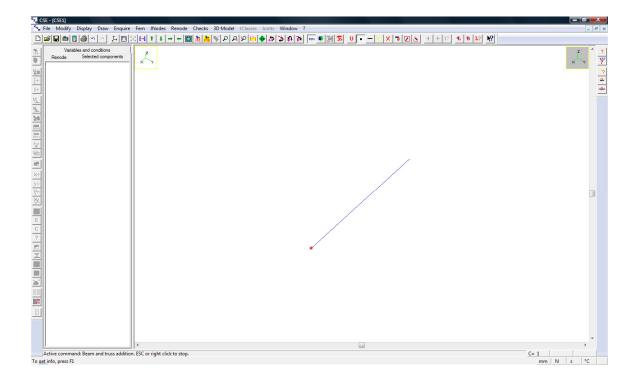
Select "Beam" and "Single" (they become red); leave third node default value (1 = to the infinite in +Y direction). Press OK.

The first node of the element must be added now. You can click an existing node o define a new one. Since there are not nodes yet, click the SPACE BAR on your keypad: the following dialog box will appear. In this dialog box you can set first node coordinate: leave (0, 0, 0) and press OK.

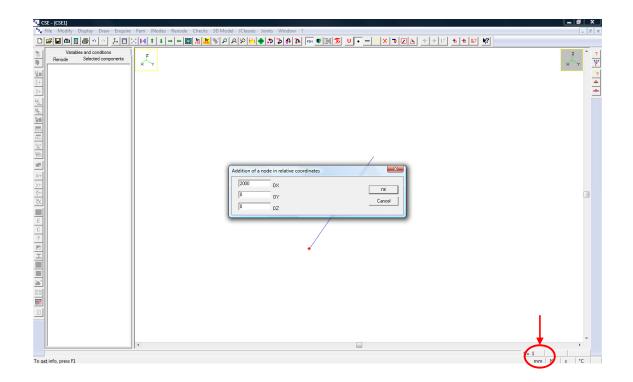


The first node has been added. If you move the mouse, a line starting from the node will join mouse pointer and the node. If there were other nodes you could click them, but since there is only one node, press the SPACE BAR again in order to add the second node.



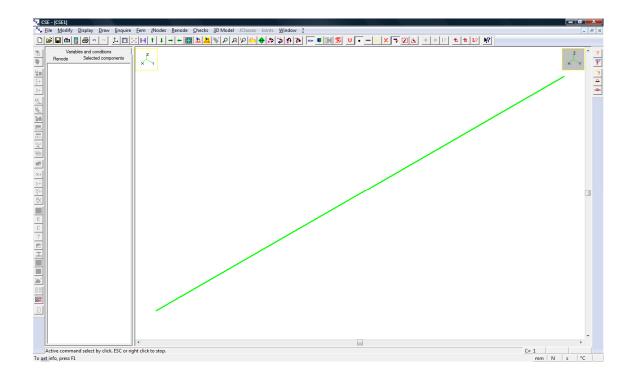


A dialog box appears: it defines second node coordinates using a vector starting from the first node.



Type 2000 in DX box to define a vector in +X direction with a length equal to 2000mm (current units are displayed in bottom right part of CSE window; use **Modify – Units**  $\bigcup$  to change units).





Beam element has been added. Press ESC to end elements addition command.

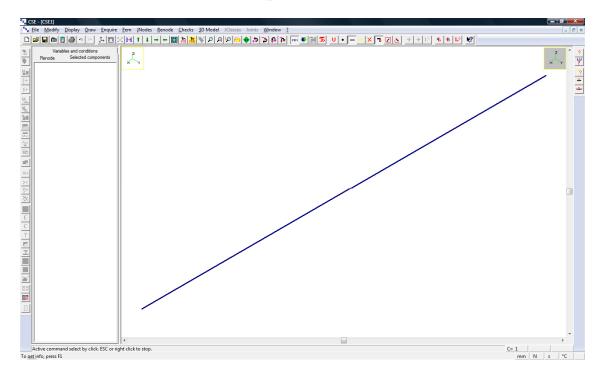
#### 2.1.2 Assigning cross-section

To assign a cross-section to finite elements, chosen elements must be selected. First of all click **Modify – Select – 1D** to enable finite elements selection. You can also use the button in the toobar.

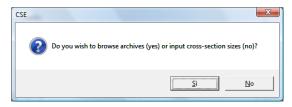


Now select the element clicking on it with mouse left click. Selected element is blue.



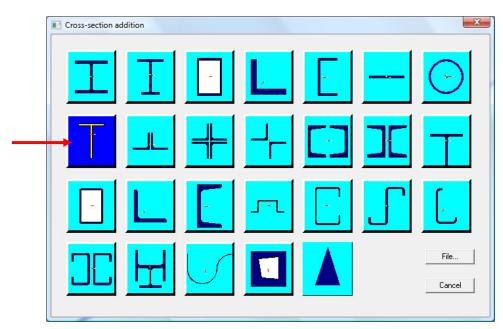


Use **FEM-Elements-Cross-sections** command to assign a shape. If you are using the demo this command just applies the cross-section HEB200 to the elements, otherwise a dialog box asks if you want to choose a shape from CSE archive (Si=Yes) or add a new shape (No):

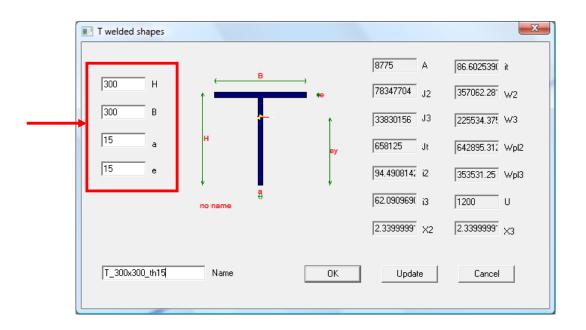


Click "No" to add a new shape and you will be prompted to choose a cross-section kind, with the following dialog box.





Choose welded T shape, and set the following parameters in next dialog box.



Click "Update" button to update cross-section image and properties (area, moments of inertia, ecc.); Type a name for the section, then press OK to assign this cross-section to selected beam element.

## 2.1.3 Assigning material

Execute the command **FEM-Elements-Materials**. If you are using the demo this automatically applies the material S235 to selected element. Otherwise choose "Archive" in the following dialog:

Castalia s.r.l. - all rights reserved - 9



Access to Material Archi	ve	x
Kind	- Fy	Ft
	Activate	Activate
	0 Min	0 Min
🗖 NONST	0 Max	0 Max
Model	Archive	Cancel

and then choose the desired material by selecting the appropriate row and pressing the button "Applica (Apply)" in the following dialog:

Name	g	E	nu	ЕУ ▲
A36/32	7.850e-005	2.000e+005	3.000e-001	2.210e+0
5235	7.701e-005	2.100e+005	3.000e-001	2.350e+0 🗏
Fe360	7.850e-005	2.060e+005	3.000e-001	2.350e+0
Fe360NoWeight	0.000e+000	2.060e+005	3.000e-001	2.350e+0
A36/36	7.850e-005	2.000e+005	3.000e-001	2.480e+0
S275	7.701e-005	2.100e+005	3.000e-001	2.750e+0
Fe430	7.850e-005	2.060e+005	3.000e-001	2.750e+0
A441/40	7.850e-005	2.000e+005	3.000e-001	2.760e+0
A441/42	7.850e-005	2.000e+005	3.000e-001	2.900e+0
A529	7.850e-005	2.000e+005	3.000e-001	2.900e+0
A242/42	7 850e-005	2 000e+005	3 000e-001	2 900e+0 T
<				•

S235 has been assigned to selected beam element.

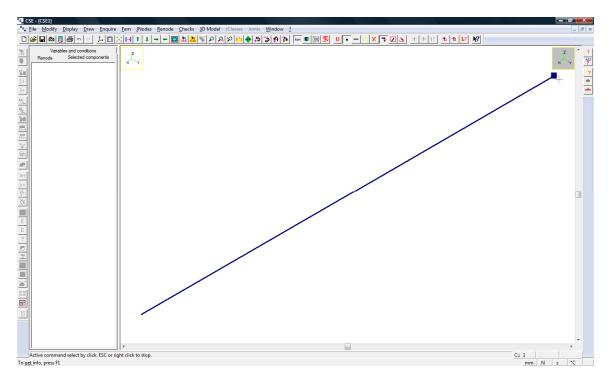
#### 2.1.4 Constraint

To assign a constraint to a node (or to more than one node), chosen node(s) must be selected. First of all click **Modify – Select – 0D** to enable nodes selection. You can also use the button in the toolbar.

1					
• -	$\mathbb{M}_{i}^{*}$	X	7	$\square$	$\bigcirc$

Now select top-right node clicking it with mouse left button. A blue square highlights selected node.



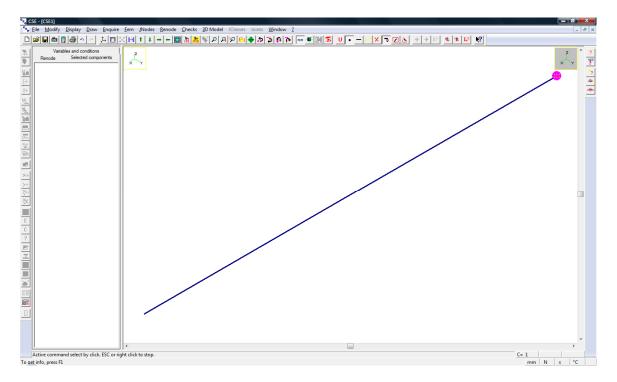


Use **FEM – Assign constraint** command. In the dialog box click "Clamped" to define a constraint in which all displacements and rotations are not allowed: when you click this button, indeed, all  $T_i$ and  $R_i$  boxes are automatically ticked ("Free" removes all the ticks, and manually ticking proper boxes you can define hinges or other constraint kinds).

Selected nodes constraints	X
Components	
I Tx	Clamped
Г Ту	Free
I Tz	
Rx Rx	
🔽 Ry	OK
Rz Rz	Cancel

Press OK to assign constraint; in the scene a circle is displayed on constrained node, and 6 pixels in it mean that all translations and rotations are not allowed.





The fem model is now completed. You do not need to add actions because, since the fem model has been prepared inside CSE, CSE will use fractions of elastic or plastic limits or defined values as design generalized stresses (internal forces, i.e. axial forces, shears, bending moments and torque).

## 2.2 STEP 2: SEARCHING MEMBERS

To move to connection design you now have to detect which members are present in the fem model.

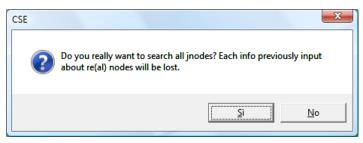
To do that just execute the command: FEM-Search members!

## 2.3 STEP 3: SEARCHING JNODES

Now that members have been searched, you need to find different "jnodes", that is what will be next a true, real node (renode). The program scans the member model and finds how many equal and how many different jnodes there are in the model. Then each jnode will be marked and you will be able to select it in order to work on it.



To search jnodes just execute the command **JNODE-Search jnodes!** and answer "Si" (Yes) to the following question:



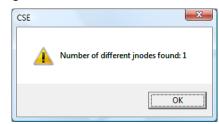
As no real nodes info has been defined you will not lose anything. Jnodes search is usually done once for all in a model. Before beginning to add RENODES, you will check that the JNODES found are correct. This depends also on how the fem model has been prepared.

The following dialog box appears:

CSE		×
?	Do you wish to discard cuspidal and tangent jnodes (you them)?	ı cannot check
	<u>S</u>	No

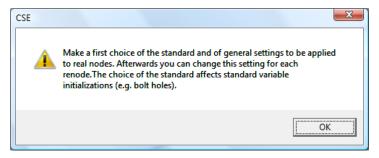
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:



This means that in this model there is just one jnode, the constraint.

Now the following message appears:





It asks to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

ode	Internal actions computing mode	Partial safety factors
CNR 10011 - Allowable stress	From FEM combinations □ Worst only	1 gammaM,0
Eurocode 3 - EN 1993-1-8     IS 800: working stress     IS 800: limit states	C Elastic limits Im1 Member C Plastic limits C Defined values C From table	1 gammaM,1
AISC-ASD: allowable stress	1 N,axial force,compression	
O AISC-LRFD: factor design	1 N,axial force, tension	1.1 gammaM,3
ing English	1     V2, shear force       1     V3, shear force	1 gammaM,4
C Italian	1 M1, twisting moment	1 gammaM,5
<ul> <li>Open when finished checks</li> </ul>	1 M2, bending moment 1 M3, bending moment	
<ul> <li>Including results (expanded)</li> </ul>	✓ Use info about end release	
necks to be executed	L	
Bolt pressure bearings	Parastitic bending in bolts	FEM analysis of components
Execute checks	Neglect parasitic bending	Do not create models     Create just sketch models
Punching shear checks	-Net cross-sections members checks	C Create complete models
✓ Execute checks	₩ Execute checks	C Create and analyze models
Block tear checks	User checks (added formulae)	G 11 0 10
✓ Execute checks	✓ Execute checks	Use Sargon/Clever     Use Sap2000
Simplified through checks		C Use other
Displacement bounds of components to	print a warning message	
· · · · · ·	0.0087266 Rotation (radians)	

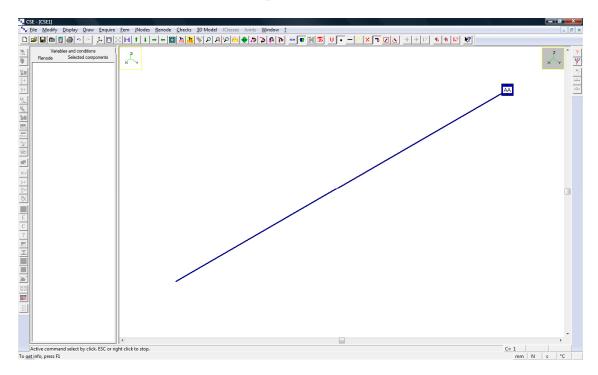
Choose Eurocode 3 and leave all other defaults, we will change them later.

Now you can switch to jnode view. This is done by pressing this button in the main toolbar.

FEM	₩ C 1
	$\overline{\uparrow}$

In jnode view you get the following:





You can now get some info about the jnode found. Execute the command

#### JNODE-Edit...

and get the following dialog:

Jnode collectio	n			X
				Inquire
	AA	Simple	Rigid constraint	No elastic restraint
				No constraint
				Sel / Unsel
				Unselect All
				Select All
				Delete
				OK
				Cancel
	_			

Select the appropriate row and then "Inquire", you get the following property sheet:



Info about jnode	Constraint
1	Number Simple Rigid constraint
AA	Mark
1	Number of occurencies 7   Nodes
1	Number of members 1  Members 7 Extreme nodes
	1 Cuspidal
	0 Passing by
	0 Interrupted
0	Number of trusses
1	Number of beams
	OK Annulla Applica ?

and clicking to "Constraint" the following:

Rigid co	instraint						
velope of c ⊢Positive -	onstraint rea	actions for	all jnode instanc	es and load combi	nations -		
-0	Rx	0	Node	-0	Rx	0	Node
-0	Ry	0	Node	-0	Ry	0	Node
-0	Rz	0	Node	-0	Rz	0	Node
-0	Mx	0	Node	-0	Mx	0	Node
-0	My	0	Node	-0	My	0	Node
-0	Mz	0	Node	-0	Mz	0	Node

The first dialog tells how many jnodes "AA" there are in the model, and which fem model nodes, members, and elements the jnode is using. The second dialog presents connection info and constraint forces envelope; this part is blank as this fem model has been prepared inside CSE and is not coming from a true fem analysis.

Press OK and then Cancel to exit from both dialogs, including "Jnode Collection" dialog.



You can now wish to examine a listing for all jnodes found. This helps to understand if the fem model has been prepared correctly, and is an important tool to pre-study future RENODES. Now save the model executing the command **File-Save**, and specify a name, for instance TUTORIAL6.CSE.

Execute the command **JNODES-Create listing!**: you will get a message telling you that the stress file is missing: it's ok as we are not coming from a true fem analysis.

Now execute the command **JNODES-Open listing!** You will get a file with useful preliminary info about JNODES topology and categorization.

TUTORIAL6.txt - WordPad		х
<u>File M</u> odifica <u>V</u> isualizza Inserisci F <u>o</u> rmato <u>?</u>		
□ ☞ 🖬 🖨 🕼 👭 🕺 🖻 🛍 ∽ 💁		
*********	***	^
*	*	
* C.S.E.	*	
* Connection Study Environment	*	=
*	*	
· *	*	
*	*	
* Copyright 2000-2010 Castalia s.r.l http://www.castaliaweb.com	*	
*****	***	
	*	
* LEGEND	-	
*	*	
*	*	
* ALPHAAngle between the considered member and the master, in a	*	
* hierarchic jnode	*	
* ALINGNMENTParallelism between a couple of local axes. If it is "mn", * axis "m" of the master is parallel to axis "n" of the slave		
* BETAAngle between the considered member and the first member of		
* the jnode, in a central, plane jnode	*	
* JOINTAll the components connecting two members	*	
* * JNODEThe set of all joints of a set of members ideally connected	*	
* JNUDEine set of all joints of a set of members ideally connected * in the same point (node). A single inode can be associated	-	
* to several nodes of a FEM node	*	
*	*	
* MARKUnivocal identification of a jnode (for example AA, AB, etc	.)*	
	*	-
Per ottenere la Guida, premere F1		

In particular you will get the following info:



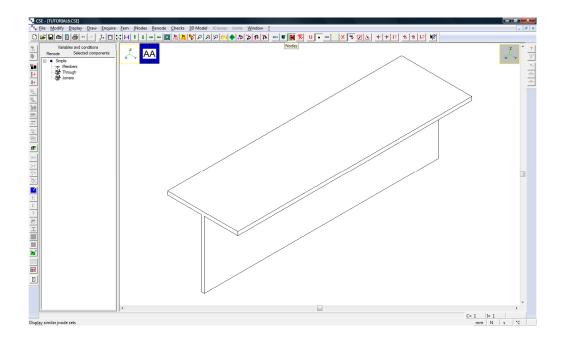
# 2.4 STEP 4: SELECTING THE PROPER JNODE

We are sure that jnode is correct; we can move forward to select it to transform it into a **RE**al **NODE**. Left click with mouse over the square "AA" in the graphic pane: it gets yellow. This means that jnode is selected (all *instances* of that jnode will be selected, here there is just one *instance*; in a true fem model there can be tens of different instances of the same jnode).

Since there is just one jnode selected the switch to real node is allowed. The following button in the main toolbar is active now:

FEM 🕂 🛃 🚺 U

By pressing it you will get into the 3D environment where RENODES are built up, checked, and studied. You will see what follows:



Note that the left window is not empty anymore: it gives you info about the components selected, the components present in the renode, and the variables and conditions present in the model. Note that clicking a member it gets selected (blue) and that the "Selected components" sub-pane in the left window is upgraded. Clicking and re-clicking you select and unselect.



# 2.5 STEP 5: CONSTRUCTION OF THE REAL NODE

## 2.5.1 Addition of generic polygon plate

Before adding plates, use **Display – Scene points** in the following dialog box:

Scene points display mode and creation rule	s				
Scene points display	Scene points creation				
✓ Display points	☑ Sides mid-points				
2 Points dot size (mm)	Sides points at thirds				
	☐ Side points at fourths				
	Face centers				
	0 Delta S =0				
Cancel					

- Tick "Display points"
- Type '2' as dots size
- Remove all the ticks in "Scene points creation" section, but keep "Sides mid-points" ticked.

Use **Renode – Components – Add through** and click the generic polygon plate in the following property sheet:

				×
Frequent Simple PitHol Rec	Cut   RecRou   RecMix   TriCut	TriRou   Finger		
Rect, plata (2, h)	Ret: plate with bavel (p.n.a	Trie plate (b,h)	Trapezoid plate (b.B.n)	Hex dieg plate [alpha.d]
Circular state [t]	Regular polygon plate (r.n.)	Generic polygon gete []	Angis	Double angle
GENERIC section fru	Constraint block (b,h,d)			
			OK Annulla	Applica ?



The dialog box that appears is used to irregular polygon plate shape and parameters.

Irregular polygon plate				
Polygon				
0 Thickness				
P1 Name				
S235 Material Change				
Fem modelling         Create FEM model         10       Borders and welds element size         30       Generic elements size (if 0 then free size)				
29 Triangle minimum angle in degrees (default 29°)				
0.1 Node distance tolerance (if dist < tol then the nodes are merged)				
It is a stiffener  Search and mesh stiffeners when preparing fem model  OK Cancel				
OK Cancel				

To define plate shape press "Polygon" button: the following dialog box will appear.

Closed polygon input	X
	New side
	First point       Modify points       <<
	0 x Modify point 0 y Remove
	Adding of preset polygons
Cancel Green point is that currently select Red point is the last added OK Polygon is drawn closed dependi	

In the left part of the dialog box there is plate image (currently it is empty); on the right there are buttons to define plate shape.



Click "First point" to define first point coordinates. The following dialog box appears: press OK to define point (0, 0) as origin.

Input X and Y		-X	
	0 Y	Cancel OK	]

Now the first point is displayed.

Closed polygon input	×
	New side
	First point.
	Modify points
	0 x Modify point
	0 y Remove
	Adding of preset polygons
	Special polygons
	nt is that currently selected
<u> </u>	is the last added : drawn closed depending on last input point.

Use this button to add a side of the polygon:

The following dialog appears; type '130' in DX box to define relative coordinates between second point and first point (millimetres are current units):

Input DX and DY				X
130 DX	0	DY	Cancel	ОК

N.B. Reference axes for plate definition are the following:





The first side has been added (last added point is red, currently selected point is green).

Closed polygon input	X
	New side
· · · · · ·	First point           Modify points           0         x         Modify point           0         y         Remove
	0         y         Remove           Adding of preset polygons
Cancel Green point is that currently s Red point is the last added DK Polygon is drawn closed dep	

Add a new side::

Press dy th	en type '110' in DX box and '390' in DY b	ox and press OK.
	Input DX and DY	

Input DX and DY		_		×
110 DX	390	DY	Cancel	ОК

The second side has been added (see red arrow in the following figure). A third side closed the polygon connecting last added point to the first point (blue arrow)

Closed polygon input	X
	New side
Cancel Green point is that currently select Red point is the last added OK Polygon is drawn closed depend	

Castalia s.r.l. - all rights reserved - 22



Add the following sides ad previously done. The figure is updated when a new side is added.

	Input DX and DY			x	
Press, then type	DX	85 DY	Cancel	ОК	→ок
	Input DX and DY			x	1
Press, then type	-350 DX	0 DY	Cancel	ОК	→ок
	Input DX and DY	70		x	1
Press, then type	DX	-85 DY	Cancel	ОК	→ок

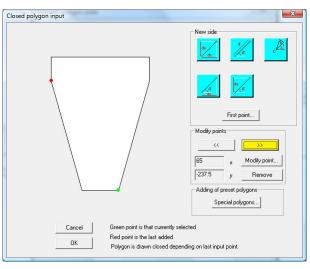
Plate shape has been defined.

Closed polygon input	×
	New side
Cancel Green point is that currently sele Red point is the last added OK Polygon is drawn closed depen	

Castalia s.r.l. - all rights reserved - 23



N.B.: the polygon can be modified adding you can new sides (here we used only a method, but straight sides can be defined in other ways, and circular sides can be added too – see the guide for more information); previously defined point can be removed or modified. To do that, use the arrows to select desired point (highlight it in green), then use "Remove button" to delete it or "Modify point" to change its coordinates.



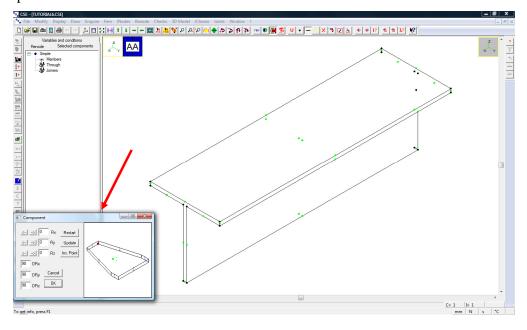
N.B. From the version 4.30 on, a predefined plate with the shape here defined is available.

Press OK to end plate shape definition; you get back in previous dialog box. Define plate thickness (30mm), keep all the other parameters and press OK to add the plate.

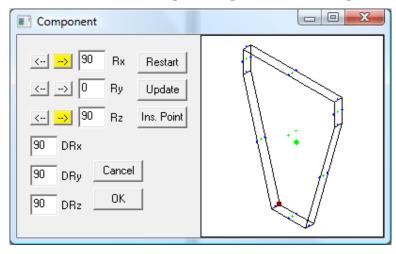
Irreg	ular polygon plate
	Polygon
ſ	30 Thickness
→ [ [	P1 Name
Г	S235 Material Change
	Fem modelling
	Create FEM model
	10 Borders and welds element size
	30 Generic elements size (if 0 then free size)
	29 Triangle minimum angle in degrees (default 29°)
	0.1 Node distance tolerance (if dist < tol then the nodes are merged)
	Tt is a stiffener
	Search and mesh stiffeners when preparing fem model
	OK Cancel



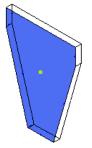
A new dialog will appear: here you can rotate the component you are going to insert, and choose the insertion point.



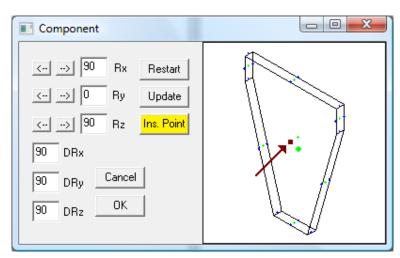
Click  $\rightarrow$  in Rx row, and then  $\rightarrow$  in Rz row to place the plate in the correct position



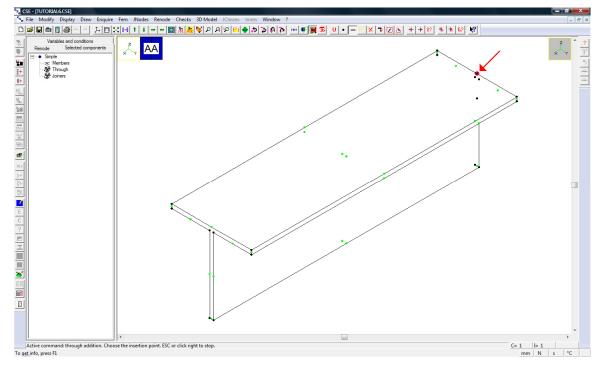
Now click "Ins. Point" and select the centre of the face shown hereunder, then press OK.







In the scene, select the point to which previously chosen point will correspond.



The plate will be added immediately in the scene.

Use **Draw – Enclose [1]** to enclose all the objects in the view.



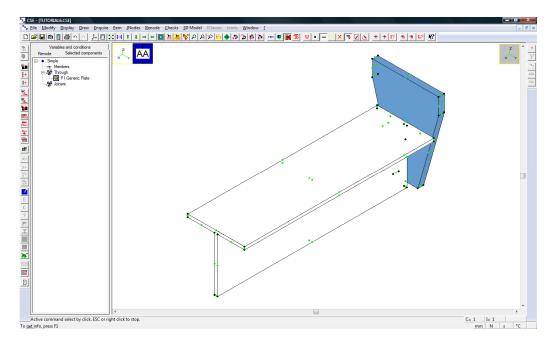
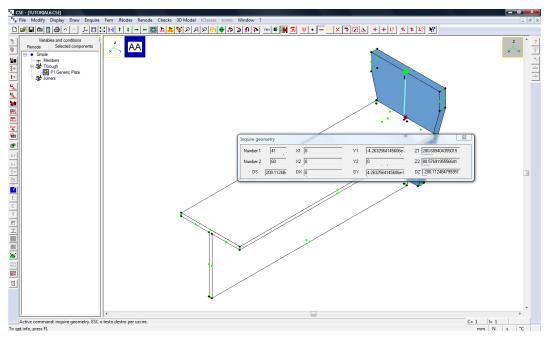


Plate position is not the correct one yet. We need to move it.



Press ESC or mouse right button to end enquire command.



Plate is the only currently select object (if it is not, because you changed selection status of the objects, select the plate with a click and/or unselect the member).

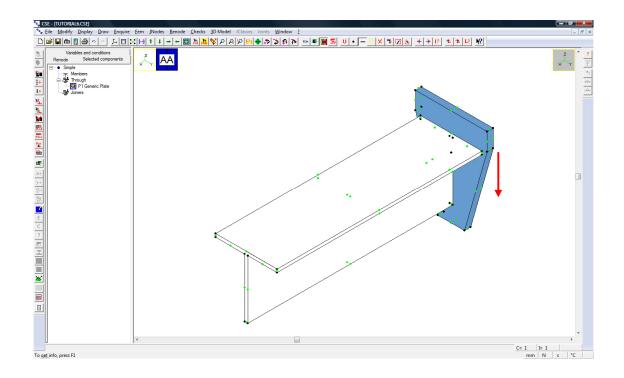
Use **Renode – Shift** and click "Translation – Numeric" in the following dialog box.

Translation –					Rotation		
<b>*</b>	<mark>.</mark> ₽	123 *		123 * +	B	B	1234
2 points	2 faces	face+incr.	face point	Numeric	2 points +angle	Member + angle	Numeric

Type '-105' in DZ box to define a 105mm translation in -Z direction, then press OK.

Dialog			<u> </u>
0	DX		
0	DY		
-105	DZ		
		<u></u>	Cancel

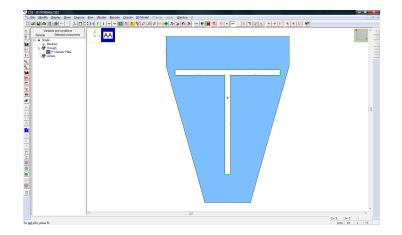
The plate has been shifted downward:



Castalia s.r.l. - all rights reserved - 28



Use **Draw – Std view** and choose a +X view to display the following:



Use the command again and choose the first isometric view.

Use Scene points 📰 command: tick "Face centres" and un-tick "Sides mid-points".

Scene points display mode and creation rules				
Scene points display	Scene points creation			
✓ Display points	Sides mid-points			
2 Points dot size (mm)	☐ Sides points at thirds			
	☐ Side points at fourths			
	Face centers			
	0 Delta S =0			
С. ОК	Cancel			

Since this renode is a constraint, we need a constraint block (it can be the foundation, a wall, etc.).

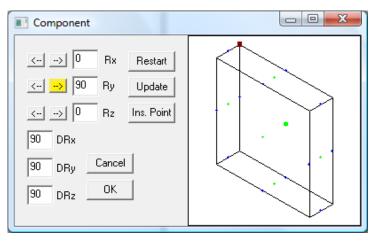
To add the constraint block use **Renode – Components - Add through**, then choose constraint block (last button).

Type Height=1000, Length=1000 and Thickness =250, then press OK.



1000 Height (DY)	
1000 🕂 Length (DX)	
250 Thickness (DZ)	
II Name	
OK Cancel	

In the following dialog press once the  $\rightarrow$  button in Ry row to place the constraint block in its correct position.



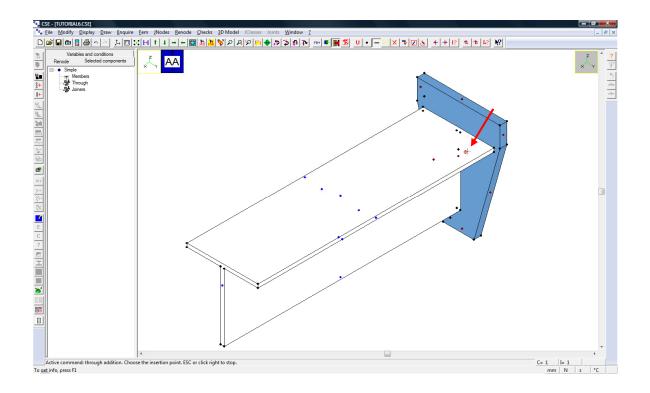
Press "Ins. Point" button and click the point indicated by the arrow in next figure, then press OK.

Component	
$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	

Castalia s.r.l. - all rights reserved - 30



In the scene, click the point to which must correspond previously chosen point.

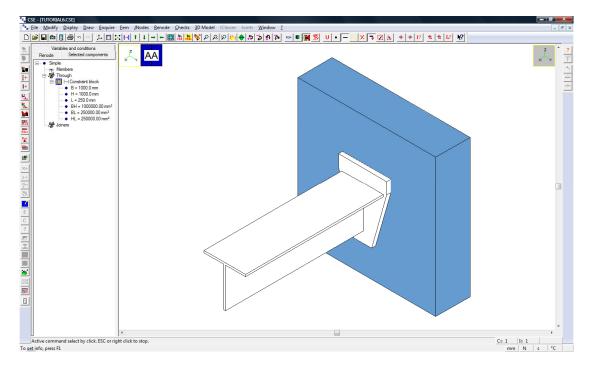


Constraint block has been added.

Enclose the structure 🖾 and hide Scene points 🐱 because they are not necessary.

Scene points display	Scene points creation
Display points	✓ Sides mid-points
2 Points dot size (mm)	☐ Sides points at thirds
	Side points at fourths
	Face centers
	0 Delta S =0
ОК	Cancel

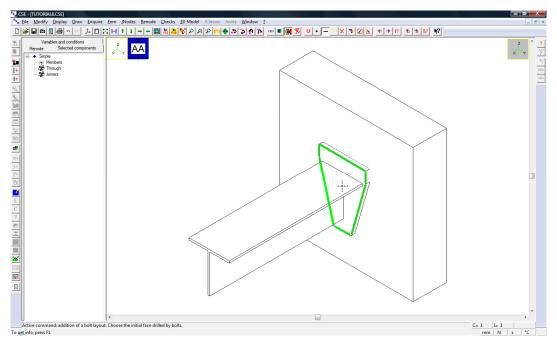




Unselect all the components  $\mathbf{x}$ .

#### 2.5.2 Addition of bolt layout

Use **Renode – Components – Add bolt layout** <sup>\*+</sup> and click in the scene the front face of the plate.

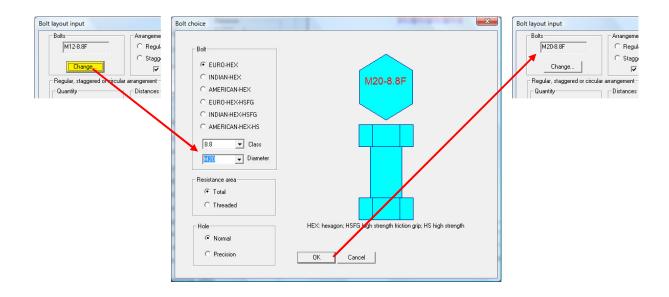


Castalia s.r.l. - all rights reserved - 32



layout input		
Bolts	Arrangement	
M12-5.6F	Regular C Circular	
	C Staggered C Free	
Change	Empty inside	
<ul> <li>Regular, staggered or circular a</li> </ul>	rrangement	
Quantity	Distances	
1 · Rows	36 · Rows	
Columns	36 Columns	8
Free arrangement		
Add Sel/Unsel	All ↔ X	
Remove + Bolt	None! Y	13 13
Center offset Ba	se information	iv iv
0 - Dx 280	) Net length	27.
0 - Dy 30	Minimum thickness -	- \ 0
0 <u>·</u> Angle ! 1	Multiplicity	
✓ Shear only bolts	Flexibility index	
Compressed bolts		
🔲 Slip resistant	Friction data	
🔲 Is an anchor	Anchor data	-
🗌 Use bearing surface	Bearing data	
🔲 Use bolt net-area for bearin	ig calculation	
🔽 Add inertia of bolts in bearing	ng calculation	- Face >bearing -
Check block tear	Block Tear	ZOOM PANX
		1: 2
- Limit values of elementary actio	ns	
30348 N	N	Font size Print Copy Fill Hexagons
27143.36 N	V Plastic	
0 N mm	Mz	Do not print messages
OK Ca	ancel Set	

Click the "Change" button in top left part of the dialog box, then choose 8.8 class and M20 diameter in the new dialog box. Press OK to return in previous dialog.



Keep the regular arrangement, define 3 rows and 3 columns using the proper arrows. Type '190' as rows distance and '130' as columns distance. The image is updated in real time.

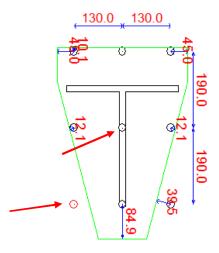


Regular, staggered or circular arrangement	
Quantity	Distances
3 · Rows	190 · Rows
3 Columns	130 Columns

When you type 130 as distance between the columns, the following message appears:



Click OK, and you will see that there are bolts out of plate borders. It is not a problem because we are going to remove those bolts (tick "Do not print messages" box to inhibit warning messages).



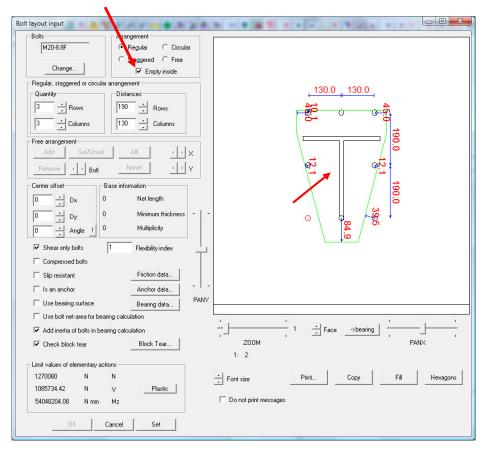
Previous image was obtained pressing "Copy" button in the dialog box.

Note that it is not possible to insert bolt layout now, because of the incorrect bolts (there is also a bolt in correspondence of member web).

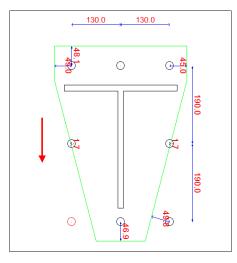


Tick "Empty inside" to remove central bolt.





Type '-38' in Dy box of "Center offset" section to move bolt layout 38m downward; if you have problems in typing '-', type 38 then add '-'.



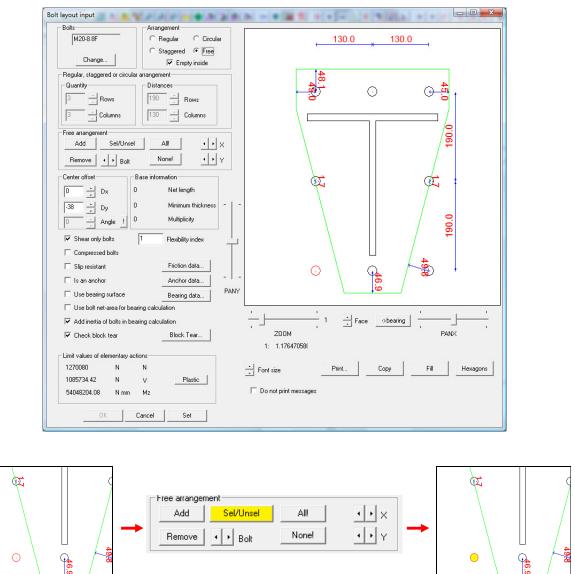
Castalia s.r.l. - all rights reserved - 35



Select *Free* in "Arrangement" section to freely move-add-remove single bolts. Initially, previous bolts disposition is kept.

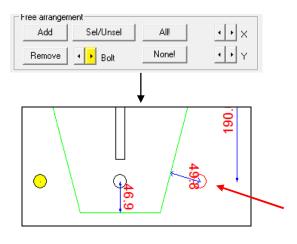


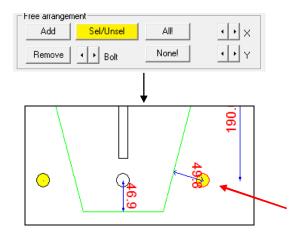
We need to remove bolts out of the plate. Current bolt has red boundary: click *Sel/Unsel* to select it: it becomes yellow.



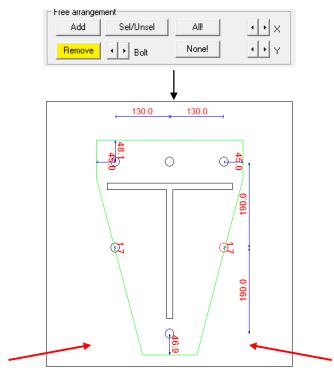
Press Bolt right arrow twice to make as current bolt the other bolt out of the plate, then select it.







Click *Remove* button to remove selected bolts.



Castalia s.r.l. - all rights reserved - 37



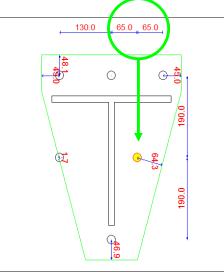
olts	Arrangement	
M20-8.8F	C Regular C Circular	130.0 130.0
	C Staggered @ Free	
Change	Empty inside	
legular, staggered or circular a	arrangement	
Quantity	Distances	
3 Rows	190 Rows	
3 Columns	130 Columns	
ree arrangement		
Add Sel/Unsel	All + X	· · · · · · · · · · · · · · · · · · ·
Remove · Bolt	Nonel · · Y	
enter offset Ba	ase information	
0 - Dx 28	D Net length	Q
-38 - Dy 30	Minimum thickness	
0 Angle 1	Multiplicity	
✔ Shear only bolts	1 Flexibility index	· · · · · · · · · · · · · · · · · · ·
Compressed bolts		
Slip resistant	Friction data	
Is an anchor	Anchor data	
Use bearing surface	Bearing data	ο
Use bolt net-area for bearing		
Add inertia of bolts in beari	ng calculation	Face ->bearing
<ul> <li>Check block tear</li> </ul>	Block Tear	ZOOM PANX
		1: 1.17647058
imit values of elementary actic		
	N	Font size Print Copy Fill Hexagons
	V Plastic	
89493723.18 N mm	Mz	Do not print messages

After bolts removal, current bolt is the one indicated in next figure: select it using *Sel/Unsel* button.

Keep *X left arrow* pressed to move selected button leftward.

Free arranger	nent		
Add	Sel/Unsel	All!	• • ×
Remove	<ul> <li>Image: Bolt</li> </ul>	None!	< ↓ ↓ Y

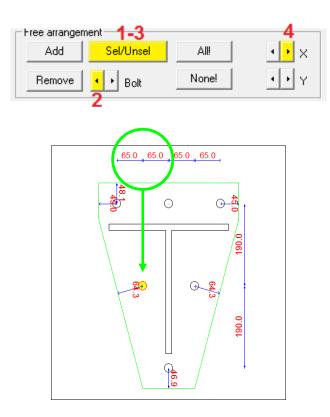
Keep it pressed until bolt is in the middle between two upper bolts (distance =65mm from both bolts).



N.B.: use zoom and pan commands to have a closer/full view of the figure.



- 1. Press *Sel/Unsel* to unselect current bolt (alternatively you can use *None!* to unselect all bolts)
- 2. Press Bolt left arrow to make as current bolt the other bolt we have to shift
- 3. Pres Sel/Unsel to select it
- 4. Keep X right arrow pressed to move it (bolt layout must be symmetrical)



Press None! button to unselect all bolts (it is not closely necessary).

Bolts positioning is now complete; you can **Copy** the figure with the proper button, show/hide bolts **hexagons** and **fill** faces with different colours.

Distances between bolts and between bolts and current face sides are displayed. Use Face buttons to change current face (the green one).

Note that if bolts distances or distances from sides are too small for chosen standard, a warning message will appear then you try to insert the layout in the scene.

Now define the following options:



- remove tick from "Shear only bolts" because they must work also under axial forces ٠
- do not tick "Compressed bolts" because compression will be carried by bearing surface •
- do not tick "Slip resistant" because this is not a no-slip bolting ٠
- do not tick "It is an anchor" •
- tick "Use bearing surface" •
  - tick "Use bolt net-area for bearing calculation"
  - tick "Add inertia of bolts in bearing calculation" ٠
- tick "Check block tear" to include block tear in the checks

□ Shear only bolts 1	Flexibility index
Compressed bolts	
🔲 Slip resistant	Friction data
🗌 Is an anchor	Anchor data
🔽 Use bearing surface	Bearing data
🔽 Use bolt net-area for bearing (	calculation
$\checkmark$ Add inertia of bolts in bearing	calculation
Check block tear	Block Tear

Press Bearing data button do set parameters to define parameters of compressed object material. Select "Non-linear analysis - No tension" and "Parabola-rectangle".

C Unlim	lastic law with li ited elastic anal 235					Es/m		tic modulus eization factor maximum compression stress
m	sigma,max							aterial partial safety factor
Nonlinear	no-tension cons	titutive law —						
• Nonlir	near analysis - I	No tension						
-21.25	-0.002	-0.0035	0	0	0	1.5	eu F+	7
E	sy	eu				gammaM	sv sv	C Elastic perfectly plastic
1							1000 C 100 C 100 C	
-	27						eu e1.	
s1	e1	eu				gammaM	eu e1,	Parabola-rectangle
		eu				gammaM	s1	
		eu e2	s2	e3	s3	gammaM gammaM	eu e1, s1 eu e3 e2 s3	© Parabola-rectangle

Type the following values:

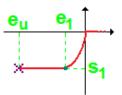
-21.25 in s<sub>1</sub> box

-0.002 in e<sub>1</sub> box



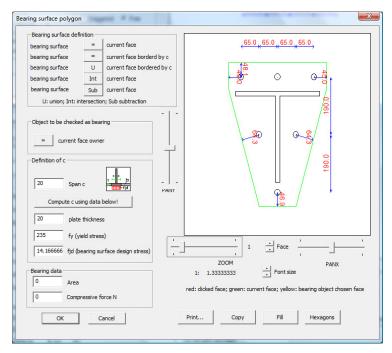


Previous settings define the following constitutive law (units are N and mm):



This will be the constitutive law of the constrain block (which simulates a concrete wall) in bearing computation.

To define bearing surface, press  $\rightarrow$  bearing button under the figure; you will get the following dialog box:

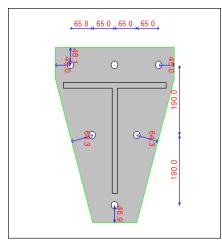


On the left there are the controls to define the surface (or the surfaces). On the right, in the image, plate boundary is currently selected face. If plate is stiff enough to assume the it is rigid, press the following button:



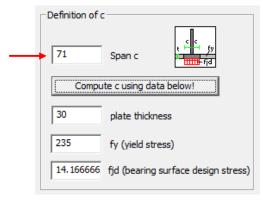
Bearing surface definition							
bearing surface	=	current face					
bearing surface	=	current face borderd by c					
bearing surface	U	current face bordered by c					
bearing surface	Int	current face					
bearing surface	current face						
U: union; Int: intersection; Sub subtraction							

Current surface is assumed as bearing surface.



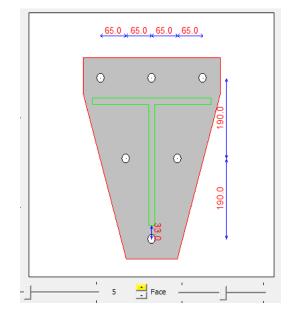
If plate is not stiff enough, you need to define an area near to cross-section imprint in which stress is diffused. To do that, first of all you need to compute the "thickness" that must be added to the cross-section.

Type 30 in "plate thickness" box;  $f_y$  is the value of plate yield stress, so keep it;  $f_{jd}$  (bearing surface design stress) is automatically computed on the base of previously define values in "bearing component material data dialog box ( $s_1$  and  $\gamma_M$ ). Press *Compute c using data below*:



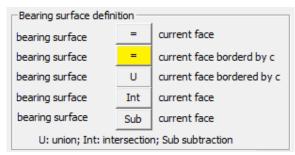
CSE automatically computes the thickness that must be added to cross-section.



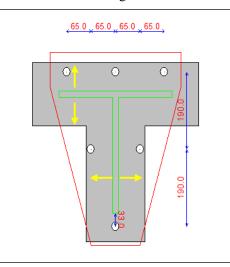


Keep Face upper arrow pressed until cross-section becomes current face.

Click the following button to get current face as bearing surface, with the addition of a border with a thickness equal to c=71mm, that is previously computed parameter.

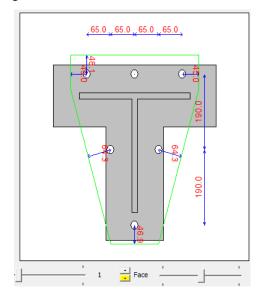


This is what you get; yellow arrows added to the figure show the addition to shape borders (c):





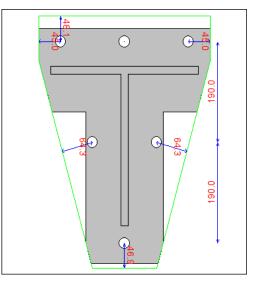
Bearing surface is out of plate boundary: select plate face by keeping Face lower arrow pressed until plate boundary becomes green.



Press the following button to get as bearing surface the portion of current bearing surface included in plate boundary.

Bearing surface definition							
bearing surface	=	current face					
bearing surface	=	current face borderd by c					
bearing surface	U	current face bordered by c					
bearing surface	Int	current face					
bearing surface	Sub	current face					
U: union; Int: intersection; Sub subtraction							

This is the bearing surface we needed:



Castalia s.r.l. - all rights reserved - 44



Finally, we have to define the component on which compression checks must be done. Select face 3 or 4 and zoom out.

Bearing surface	definition —	4							_
bearing surface	=	current face		-					1
bearing surface	=	current face border	d by c						
bearing surface	U	current face border	ed by c			348			
bearing surface	Int	current face				057.065.065.065	0		
bearing surface	Sub	current face				VJ. WJ. WJ. WJ.	÷Y		
U: union; In	it: intersectio	n; Sub subtraction		←	370.0	<b>0</b>	a .	370.0	
Object to be che	ecked as bear nt face owne	-					90.0 90		
-	Span c c using data	below!	-   - PANY			272.0	<u></u>		
30 p	late thicknes	s							
235 f	y (yield stres	s)			•	•	Ť	1.71	_
14.166666 fj	d (bearing su	rface design stress)	[-]		3	Face	1		
Bearing data —				1: 2.5	20011	Font size	e	PANA	
96661 A	rea								
1369365 C	ompressive f	orce N		rea: clicked	race; green: c	urrent face; yello	v: bearing	) object chosen	та
ОК	1	ancel		Print	Сору	Fill	He	xagons	

Click the following button.



Now, if select any face but face 3 or 4, you will see constraint block boundary in yellow, because it is the object to be checked.

Press OK to keep this bearing surface and get back in main bolt layout addition dialog box.

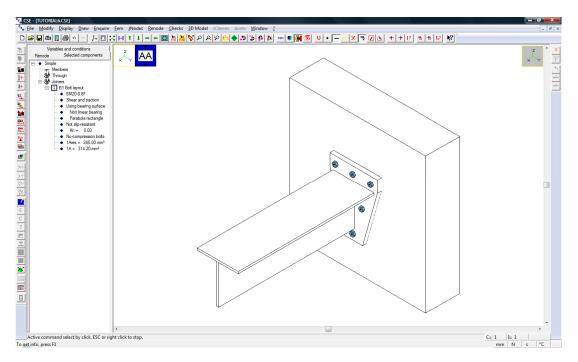
Bearing surface def	inition —		
bearing surface	=	current face	
bearing surface	=	current face borderd by c	
bearing surface	U	current face bordered by c	
bearing surface	Int	current face	65. <b>6</b> 5. <b>6</b> 5. <b>6</b> 5.0
bearing surface	Sub	current face	43.83.83.9
U: union; Int: ir	tersectio	n; Sub subtraction	
	sing data thicknes	55	
	ield stres earing su	urface design stress)	1 Face
Bearing data 96661 Area 1369365 Comp	ressive f	iorce N	ZOOM     PANX       1:     2.5     Font size   red: dicked face; green: current face; yellow: bearing object chosen face

C.S.E.

Bearing surface is now displayed in main dialog box too; press OK to add bolt layout.

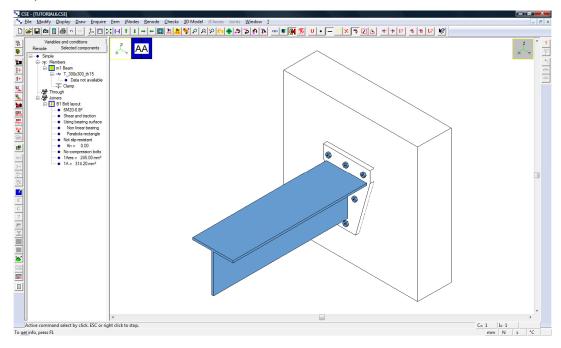
Bolt layout input	
C Staggered @	
Regular, staggered or circular anangement         Quantity         3       Rows         130       Rows         130       Columns         Free anangement         Add       Sel/Unsel         All         Remove       +         Base information         0       +         -       Dx         33       -         0       +         -       Dx         280       Net length         30       -         0       -         -       Angle 1         1       Multiplicity         Compressed botts       1         Stip resistant       Friction data         It is an anchor       Anchor data         If use bearing surface       Bearing data	
Use bolt net-area for bearing calculation     Add inertia of bolts in bearing calculation     Check block tear     Block Tear.	1Face>bearing ZOOM PANX 1: 1.176470588
Linit values of elementary actions           846720         N         N           723822.94         N         V         Plastic           77422070.74         N mm         Mz           DK         Cancel         Set	Fill Hexagons





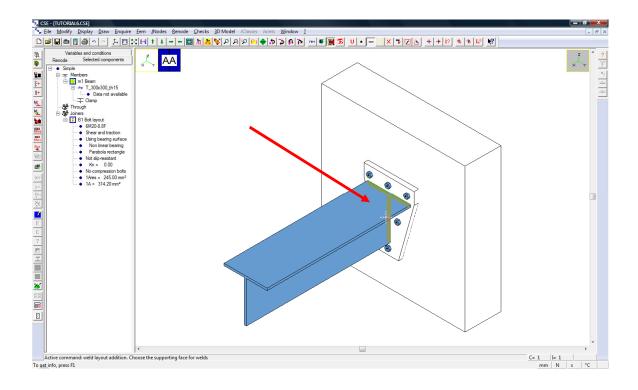
### 2.5.3 Addition of weld layout

Before adding weld layout, select member in order to limit the selection of the face to be welded to selected components only (alternatively you could unselect bolt layout and all the faces will be available).

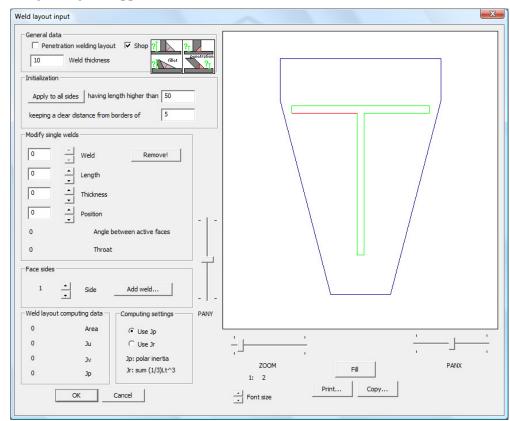


Use **Renode – Components – Add weld layout** +, then click in the scene member end face.



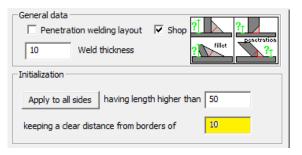


#### The following dialog box appears:

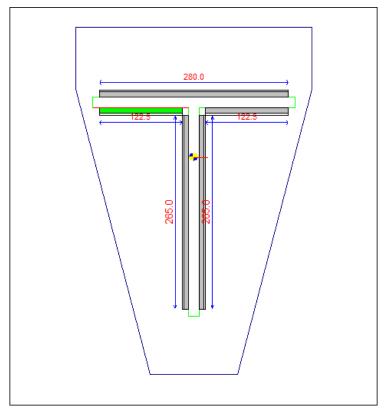




Do NOT tick "Penetration welding layout" because we are going to add a fillet welds layout. Type 10(mm) in "keeping a clear distance from borders of" box; keep other default parameters.



Press *Apply to all sides* button to add a fillet weld on all the sides having a length higher than 50(mm). Welds thickness will be equal to 10(mm). You will get the following.

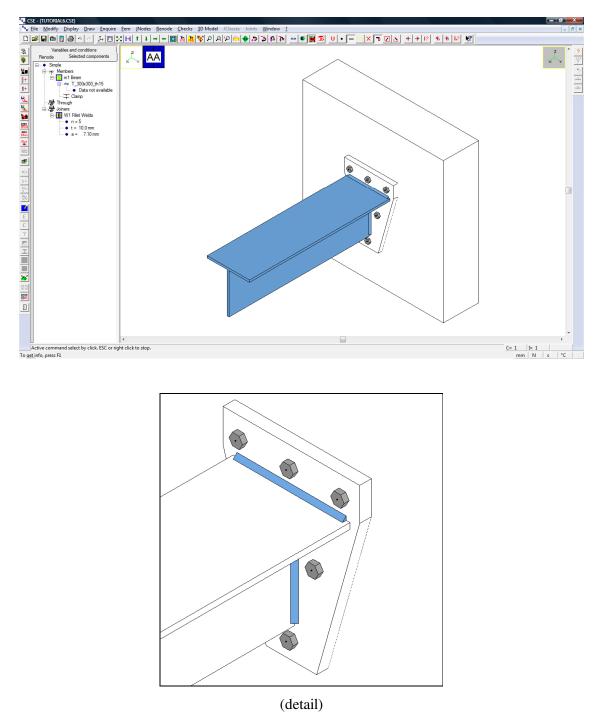


*N.B.:* Modify single welds section let you modify welds position, length and thickness; Add weld button let you add single welds on current side; see the guide for more information.

Keep default parameter for torsion computation (Use Jp).

Press OK to add the layout.





Use commands in Draw menu to see the structure from other points of view, to zoom, etc.

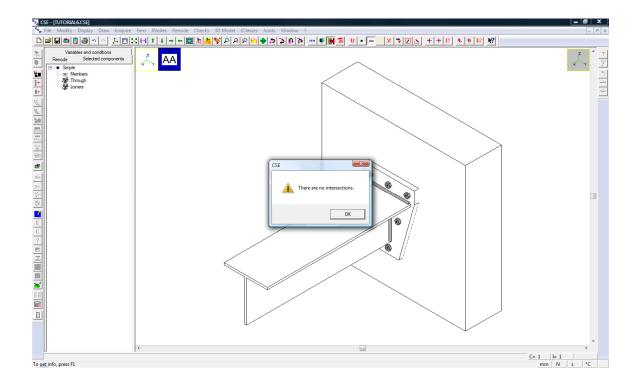


Use standard view in and select first isometric view to get "usual" view.

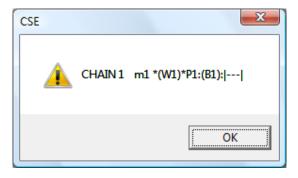


### **2.5.4** Overlaps and coherence controls

The renode has been completed. Use **Renode-Check Overlaps** to be sure that components do not overlap. In this case there are not overlaps.



Use Renode – Check coherence to check the correct connection between the components.



Chains are listed: it means that all the components are properly connected. In this case we have a single path (chain) only from member to constraint block. This path is:

member m1  $\rightarrow$  weld layout W2  $\rightarrow$  plate P1  $\rightarrow$  bolt layout B1  $\rightarrow$  constraint block |---|

# C.S.E.

## 2.6 STEP 6: SETTING CHECK SETTINGS

Before executing the checks, let us specify how we want them to be done (we now choose the settings that initially have left as default values). Execute the command **Checks-Set**. You get a dialog like that following.

The standard has already been chosen, Eurocode 3. This is a limit state standard.

In the "Listing" section choose the language, then tick the proper boxes if you want the listing to be automatically opened after the checks and if you desire a listing that includes checks results.

Since we have not a true fem model we must choose as "Internal actions computing mode", "Elastic limits", "Plastic limits" or "Defined Values". Choose elastic limits, for instance. Then specify the multiplier of internal actions for each component. You can specify the number you want. This will generate 24 notional load combinations. 6 for positive internal forces, one by one. Six more for negative internal forces. Next 12 will be a mix of N, M<sub>2</sub>, M<sub>3</sub>, i.e. axial force and bending moments. Note that the first 24 combination, if using forces NOT coming from fem, refer to the master and are empty.

For example, define a multiplier equal to 0.3 for all the limit forces and moments.

Keep default "Partial safety factors".

In "Checks to be executed" section tick the following options

- Bolt pressure bearing
- Punching shear checks
- Block tear checks
- Net cross-sections members checks
- Do not create models

Keep default "Displacements bounds" for displacement checks.

le	Internal actions computing mode	Partial safety factors
CNR 10011 - Allowable stress CNR 10011 - Limit states	C From FEM combinations 🗌 Worst only	1 gammaM,0
Eurocode 3 - EN 1993-1-8 IS 800: working stress	C Elastic limits	1 gammaM,1
IS 800: limit states AISC-ASD: allowable stress	C Defined values From table 0.3 N,axial force,compression	1.25 gammaM,2
AISC-LRFD: factor design	0.3 N,axial force, tension	1.1 gammaM,3
ing	0.3 V2, shear force	1 gammaM,4
English	0.3 V3, shear force	
ີ Italian	0.3 M1, twisting moment	1 gammaM,5
) Spanish	0.3 M2, bending moment	
Open when finished checks	0.3 M3, bending moment	
Including results (expanded)	↓ Use info about end release	
cks to be executed		
Bolt pressure bearings	Parastitic bending in bolts	FEM analysis of components
Execute checks	Neglect parasitic bending	O not create models Create just sketch models
Punching shear checks	Net cross-sections members checks	C Create complete models
Execute checks	Execute checks	C Create and analyze models
Block tear checks	User checks (added formulae)	Use Sargon/Clever
Execute checks	Execute checks	O Use Sap2000
Simplified through checks	]	C Use other
placement bounds of components to Translation	o print a warning message 0.0087266 Rotation (radians)	

C.S.E.

Press **OK** to save settings.

Notes: "simplified through checks" do not cover plates: they can be checked via fem models, but we are not going to use them now; "user checks" have not been defined here.

## 2.7 STEP 7: EXECUTING CHECKS

۶

To execute the checks press the following button in the left toolbar (Checks – Check renode):



Running checks: instance 1 of 1 com	bination 18 of 24.	
Running checks: instance 1 of 1 com		
Running checks: instance 1 of 1 com		
Running checks: instance 1 of 1 com		
Running checks: instance 1 of 1 com		
Running checks: instance 1 of 1 com		
Running checks: instance 1 of 1 com		
Envelope calculation		
Creation of listing		
Binary file creation	step 1 of 6	
Binary file creation	step 2 of 6	
Binary file creation	step 3 of 6	
Binary file creation	step 4 of 6	
Binary file creation	step 5 of 6	
Binary file creation	step 6 of 6	E
End of checks.		

The window above will appear, it is a log window explaining what happens. At the end just close it by clicking over the red-background cross.

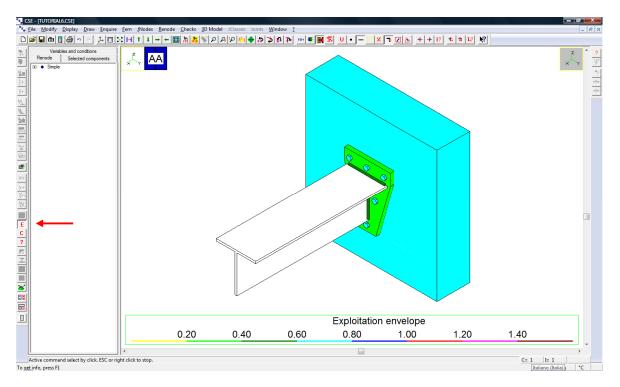
The output file has been automatically opened. Have a look at the file if you wish, then minimize or close the output file window.

TUTORIAL6.CSE.AA.EURO3.out - WordPad		
<u>File M</u> odifica <u>V</u> isualizza Inserisci F <u>o</u> rmato <u>?</u>		
		*
+ +		
# C.S.E. #		
# Copyright (c) 2001-2010 - Castalia srl - Milan Italy #		
# # # # www.castaliaweb.com # * *		
* * *		
# ver. #		
Aknoledgment		
This program is the result of years of research in steel structures and is able to automatically perform quite many checks regarding steel con- nections. The program is able to fully check joiners of very complex steel connections, made up by freely placing components and joiners, i.e. no special ad hoc rule is used, but very general ones to compute stress state of arbitrarily structured "scenes", as freely determined by the user.		
This version was released in January 2010, check for updates       		
AcT Total computing area of a boltlayout		
Per ottenere la Guida, premere F1		

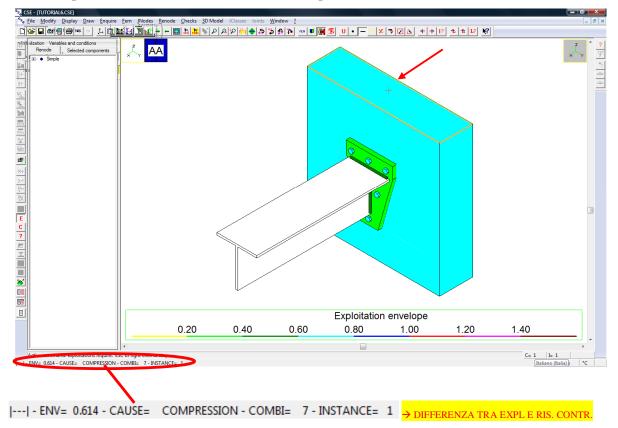
## 2.8 STEP 8: EXAMINING RESULTS

Use **Checks – Envelope** to display components exploitation envelope.





While the envelope is displayed, use **Checks – Enquire** <sup>?</sup> to know components maximum exploitation, its cause and combination. Move mouse pointer near a face of the component you want to inquire and read information in left bottom part.





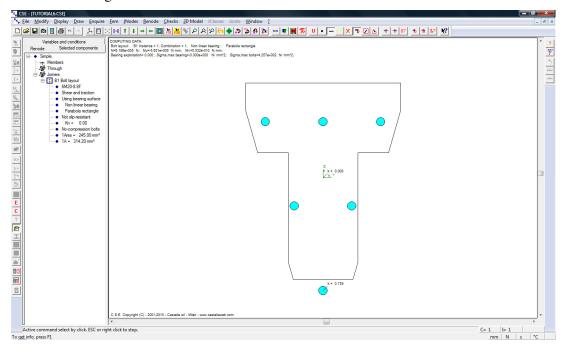
Maximum exploitation for constraint block is 0,614 in combination 7 (compression of the member) and it is due to bearing pressure checks (Cause = Compression).

You can inquire bolts (resistance checks), plate maximum exploitation (only bolt bearing pressure and block tearing checks where executed in this model for the plate; fem model was not required).

If you use **Checks – Current results** , displayed and enquired exploitations refer to current combination.

Note that member is white because it has no cuts, bevels or holes for bolts (no bolt bearing pressure checks, block tearing checks or net section checks), and no fem model was required.

Select bolt layout, then use **Checks – Display bearing surface results** [27]. In the graphic view you will see the following.



Since member is in tension there is no compression on bearing surface (it is white). Bolts maximum exploitation is 0,739.

Switch through combinations using <sup>1</sup>. Current combination is displayed in bottom-right part of CSE window.



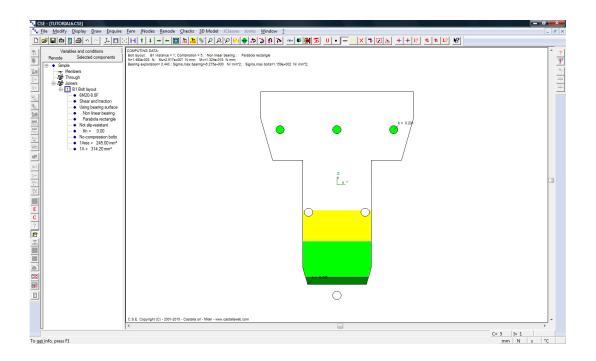
Castalia s.r.l. - all rights reserved - 56



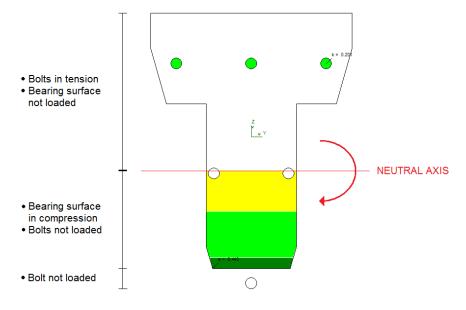
In combination 2 and 3 (shear V2 and V3) we have a small compression because there is an offset between member axis, bearing surface barycentre and bolt layout barycentre.

In combination 4 (twisting moment) there is no compression.

In combination 5 (bending moment M2) there is a significant compression.



Bolts are loaded over the neutral axis only, and bearing surface is loaded under neutral axis only.



Castalia s.r.l. - all rights reserved - 57



In the top-left part of graphic view you can find the following data:

- Bolt layout name; current instance and combination; kind of bearing surface constitutive law
- Axial force and bending moments
- Bearing surface exploitation and maximum stress; maximum stress in bolts



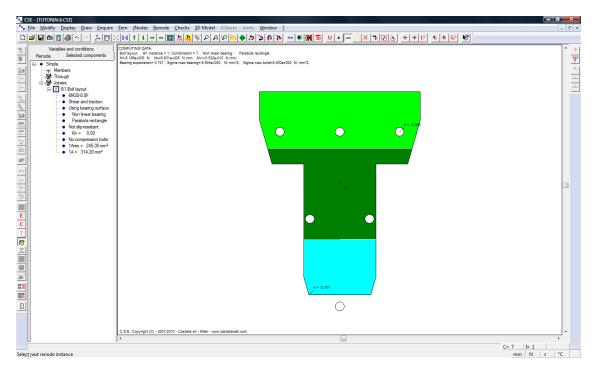
Switch to combination 6 (bending moment M3): since bolt layout is not symmetric with respect of axis 3, a M3 bending moment gives a neutral axis not symmetric to axis 3.



Switch to combination 7, in which we have maximum constraint block exploitation, due to bearing surface check.

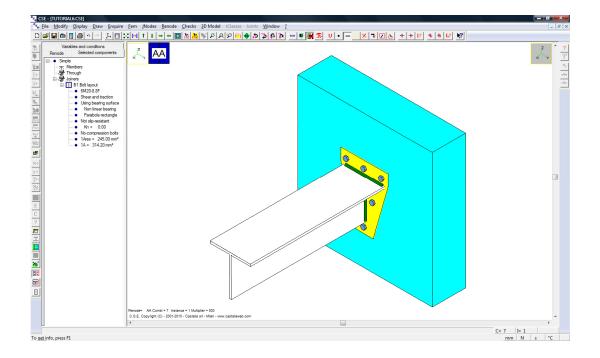
Since there is a offset between member axis, bearing surface barycentre and bolt layout barycentre, the lower part of bearing surface is more compressed than the upper part.





You can continue to switch between combinations and see bearing surface results.

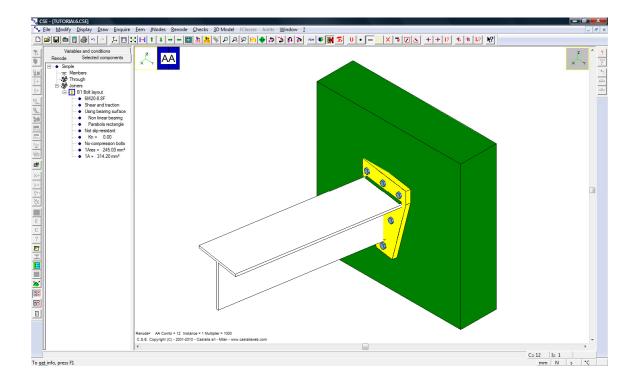
Use **Checks –Displaced** in left toolbar to display the displaced view of the renode in current combination.





Switch between combinations to see displaced structure in desired combination  $\underbrace{\textcircled{1}}_{\texttt{L}}$  and change displaced scale if you need it  $\underbrace{\textcircled{1}}_{\texttt{L}}$ .

See, for example, the displaced in combination 5 (amplification scale = 1000).





<u>1</u> I	NTRODUCTION	3
<u>2</u>	CONSTRAINT	4
2.1	STEP 1: GETTING THE FEM MODEL	4
2.1.1	GETTING THE FEM ELEMENTS	4
2.1.2	Assigning cross-section	7
2.1.3	ASSIGNING MATERIAL	9
2.1.4	CONSTRAINT	10
2.2	STEP 2: SEARCHING MEMBERS	12
2.3	STEP 3: SEARCHING JNODES	12
2.4	STEP 4: SELECTING THE PROPER JNODE	18
2.5	STEP 5: CONSTRUCTION OF THE REAL NODE	19
2.5.1	ADDITION OF GENERIC POLYGON PLATE	19
2.5.2	2 ADDITION OF BOLT LAYOUT	32
2.5.3	ADDITION OF WELD LAYOUT	47
2.5.4	OVERLAPS AND COHERENCE CONTROLS	51
2.6	STEP 6: SETTING CHECK SETTINGS	52
2.7	STEP 7: EXECUTING CHECKS	53
2.8	STEP 8: EXAMINING RESULTS	54