



## **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:

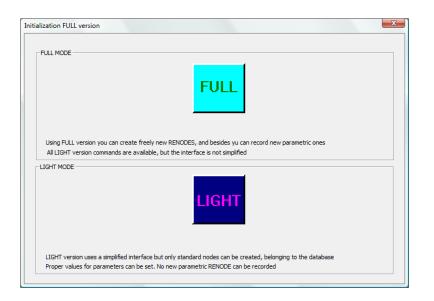
- Create a dummy fem model to study a splice joint
- Assign the materials and cross-section to the fem elements
- Search members
- Search jnodes and get information about them
- Add plates and bolts
- Add penetration welds
- Set the checks to be performed
- Have a look at the results

This tutorial is some like 39 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** SPLICE JOINT WITH PENETRATION WELDS



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

	SE-[CSE1]	- 8	
	Eine Modify Danky Dawn Enguine Een Modes Benode Onecks 20 Model XCases Joints Window I #BADDEAOO J.D.X.H.1.1.4.4.4.B.D.P.P.A.P.int.4.5.5.4.D.m.4.3.5.0.0.4.4.2.2.5.4.4.1.5.5.0.0.4.	-	. [#]×]
<ul> <li>第     <li>第     <li>第     <li>第     <li>第     <li>第     <li>1+     </li> </li></li></li></li></li></li></ul>	Veralitie and construm	*	^ <u>?</u> * *
北京諸国家の			
에레 빨 첫 치조			
× = c ? =			
▶ 쿄쓰복 귕평컵립달경리 팀 치치하였 ■비어수퇴번■■계 88 118 : 0			
		,	-
Tog	t info, pres FI	C= 1 mm N s *C	

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.

Execute the command

## **FEM-Elements-Typical Structures**.

Castalia s.r.l. - all rights reserved - 4



	Bracings Lattice	
Splice joints Beam-Column joints Beam-Beam joints Ground joints		
1		
Master element is red-coloured		

In the pane "splice joints" click inside the second image (from the left).

You will get the following dialog box:

laterial			
???		Archive Add	1 2
ross-sections			
???	current	Archive Add new	
???	1	Assign current! 🔽 Strong axis	
???	2	Assign current! 🔽 Strong axis	
ettings			
ettings ☞ Directly sear	ch for jnodes		



One of the beam elements has a "connection code": it is necessary to define jnode hierarchy. The element without connection code is the master (in red), the other one is the slave: it is a **hierarchical** jnode.

## 2.1.2 Assigning material

Use the button "Archive" in the box called "Material" in order to assign a material to the member, choosing it from the archive. It is also possible to add a new material with the button "Add".

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

Name	g	E	nu	Fy 4
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
5275	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
•				•

Chosen material has been applied.

Material	
S235	Archive Add
,	

### If you are using the demo the button "Archive" automatically applies the material S235.

With the "Typical node" tool all the members have the same material. It is possible to define different materials with the standard fem commands.



## 2.1.3 Assigning cross-section

As for the material, it is possible to add new cross-sections or choose them from the archive. Press "Archive" in the "Cross-sections" box to browse the archive.

Image: Construction     Image: Construct	Kind         Rolled H shapes         IPE       DIL         HEA       HL         HEB       HX         ILS         HEM       HD         IPE*       HP         HE       M         IPE*       HP         Welded       HSH (H)         USH (U, C)       LSH (L)         TSH (T)       0SH (Box, sharp corners)	□U_H][ □L	gs angles) Japanese Indian NONST unds) ar Hollow) 2_CR Data filters	Name No Limits
	Generic polygons		aeneric	No Design

Select HEB check box and then "Archive" to filter HEB kind among all the available cross-sections. Then choose the section HEB200 by selecting the appropriate row, and press the "Apply" button.

Name	A	p		
HE 100 B	 2.604e+003	2.044e-001	4.	
HE 120 B	 3.401e+003	2.670e-001	5.	
HE 140 B	 4.296e+003	3.372e-001	5.	
HE 160 B	 5.425e+003	4.259e-001	6.	
HE 180 B	 6.525e+003	5.122e-001	7.	
HE 200 B	7.808e+003	6.129e-001	8.	
HE 220 B	 9.104e+003	7.147e-001	9.	
HE 240 B	 1.060e+004	8.320e-001	1.	
HE 260 B	 1.184e+004	9.298e-001	1.	
HE 280 B	 1.314e+004	1.031e+000	1.	
HR 300 B	1 491e+004	1 170e+000	1	HE 200 B
< III			•	

The chosen shape will appear in the box called "current".



-	Cross-sections			
$\langle$	HE 200 B	current	Archive	Add new
	???	1	Assign current!	Strong axis
	???	2	Assign current!	✓ Strong axis

Current shape can now be applied to the members with the "Assign current!" button on the left of member 1 and member 2 boxes.

Cross-sections			
HE 200 B	current	Archive	Add new
	$\sim$	<b>*</b>	E
HE 200 B	1	Assign current!	Strong axis
???	2	Assign current!	Strong axis
<b>6</b>			
Cross-sections			
Cross-sections	current	Archive	Add new
HE 200 B	_		
	current	Archive	Add new
HE 200 B	_		
HE 200 B	1	Assign current!	Strong axis
HE 200 B	1	Assign current!	Strong axis

laterial		_	1	
S235		Archive	Add	1 2
ross-sections				
HE 200 B	current	Archive	Add new	
HE 200 B	1	Assign current!	Strong axis	
HE 200 B	2	Assign current!	Strong axis	



*N.B.* It is possible to change the current cross-section before assign it to member 2 in order to define different shapes for the members.

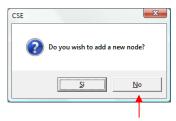
It is possible to apply a rotation of  $90^{\circ}$  to the members removing the tick from "Strong axis" boxes. In some nodes, not in this case, it is possible to define hinges for the slaves with proper tick boxes.

Pressing OK with the tick on "Directly search for jnodes", members and jnodes would be automatically searched and the resulting 3D renode would be automatically shown.

Remove that tick to see and understand step by step what could be done automatically (steps 2, 3 and 4 of this tutorial).

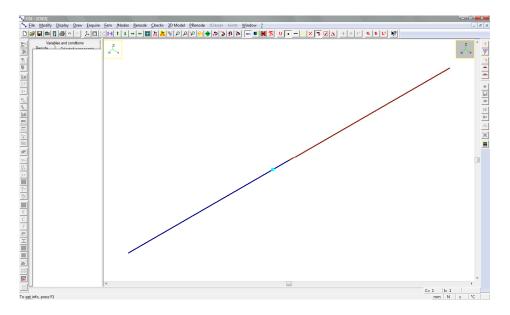
Settings		
Directly search for jnodes		
└──_♠		
	ОК	Cancel

Choose "No" in the following dialog box. If you choose "Si" (Yes), will be opened the "Typical nodes" dialog box and it will be possible to select another kind of node, define its properties, and so on.



A fem model of the defined node has been automatically created.





# 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!**

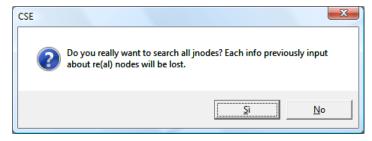
# 2.3 STEP 3: SEARCHING JNODES

Now that members have been searched, you need to find different "jnodes" that is what will next get 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!**

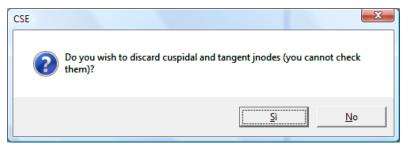
and answer "yes" to the following question:





As no real nodes info has been defined you will not lose anything. Jnode 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 appears:



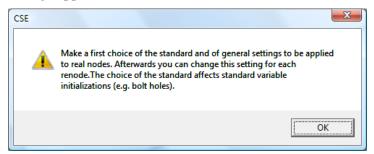
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:



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

Now the following message appears:



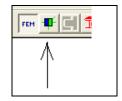
asking to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

Code	Internal actions computing mode	Partial safety factors
C CNR 10011 - Allowable stress		1 gammaM,0
Eurocode 3 - EN 1993-1-8     IS 800: working stress     IS 800: limit states     AISC-ASD: allowable stress     AISC-LRFD: factor design	C Elastic limits Im1 Member C Plastic limits T Plastic limits T Defined values C From table N,axial force, compression N,axial force, tension	1         gammaM, 1           1.25         gammaM, 2           1.1         gammaM, 3
isting	1 V2, shear force	gammaM,4
English	1 V3, shear force	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
C Italian C Spanish	1 M1, twisting moment	1 gammaM,5
$\overline{ullet}$ Open when finished checks	1     M2, bending moment       1     M3, bending moment	
<ul> <li>Including results (expanded)</li> </ul>	✓ Use info about end release	
Bolt pressure bearings	Parastitic bending in bolts	• Do not create models
Punching shear checks	Net cross-sections members checks	C Create just sketch models
Execute checks		C Create complete models C Create and analyze models
Block tear checks	User checks (added formulae)	Use Sargon/Clever
Execute checks	Execute checks	C Use Sap2000
Simplified through checks		C Use other
Displacement bounds of components t	o print a warning message	
1 Translation	0.0087266 Rotation (radians)	

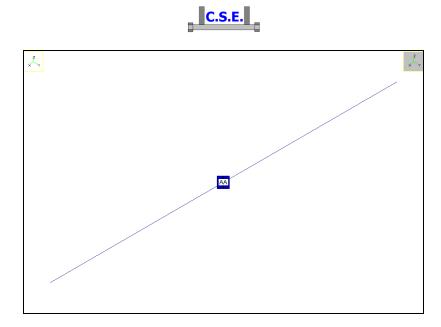
C.S.E.

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.



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 collection	x
1 AA Master	Inquire
	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



	X
Info about jnode	Master
1 AA	Number Master
1	Number of occurencies 8  Nodes
2	Number of members 1  Vembers 7 Extreme nodes
	1     Cuspidal       0     Passing by       1     Interrupted
0	Number of trusses
2	Number of beams
	OK Annulla Applica ?

and	clicking	to	"Master"	the	follo	wing:

		N	laster cross-secti	on				
Beam		E	lement kind					
lave								
HE 200 B		▼ SI	ave cross-section	1				
Beam		В	ement kind					
Clamp		Jo	int kind					
Splice-homogene	eous	A	ignment					
nvelope of inten	nal forces	in slave -						
D								
Positive				Negative				
0.000e+000	N+	0	Element		000 <sub>N</sub> .	0	Element	
		0	Element	0.000e+		0	Element Element	
0.000e+000	T2+			0.000e+	000 N-		_	
0.000e+000	T2+	0	Element	0.000e+	000 N. 000 T2- 000 T3-	0	Element	
0.000e+000	T2+ T3+ M1+		Element	0.000e+	000 N- 000 T2- 000 T3- 000 M1-	0	Element Element	

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 connections info and



internal 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 TUTORIAL4.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.

TUTORIAL4.txt - WordPad	
<u>File M</u> odifica <u>V</u> isualizza Inserisci F <u>o</u> rmato <u>?</u>	
🗅 😅 🖬 🍓 📐 🛤 🐰 🖻 🎕 い 👒	
*******	***
*	*
*	*
* 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 inode, in a central, plane inode	
* JOINTAll the components connecting two members	
* vointitute components connecting two members	*
* JNODEThe set of all joints of a set of members ideally connected	*
* in the same point (node). A single jnode can be associated	*
* to several nodes of a FEM node	*
*	*
* MARKUnivocal identification of a jnode (for example AA, AB, etc.	.)*
* * MEMPED Driamatic single next before workings. It is modelled with	*
Per ottenere la Guida, premere F1	

In particular you will get the following info:

\* JNODE 1 MARK AA \* GLOBAL TYPOLOGY OF THE JNODE: HIERARCHIC JNODE
NODES ASSOCIATED TO THIS JNODE: TOTAL 1
8

Castalia s.r.l. - all rights reserved - 15

```
MEMBERS IN THE JNODE: TOTAL 2

1 2

JOINT 1 (Beam - Beam) MASTER= HE 200 B SLAVE= HE 200 B

Beam Beam

End joint - Splice-homogeneous - Clamp -

ALPHA= 0.000000 ( 0.00°) COS= 1.000000 SIN= 0.000000 TAN= 0.00000e+000
```

# 2.4 STEP 4: SELECTING THE PROPER JNODE

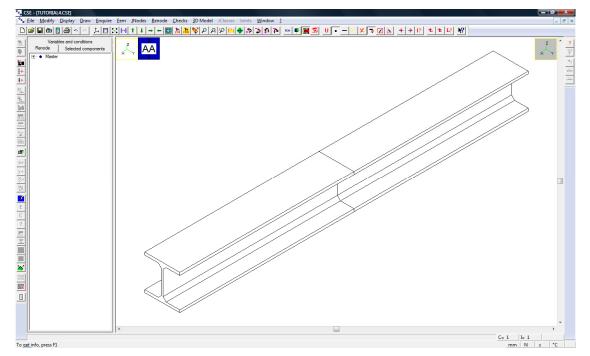
C.S.E.

We are sure that jnodes are correct, we can move forward to select one of them 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 you have selected a jnode (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 possible. The following button in the main toolbar is active now:

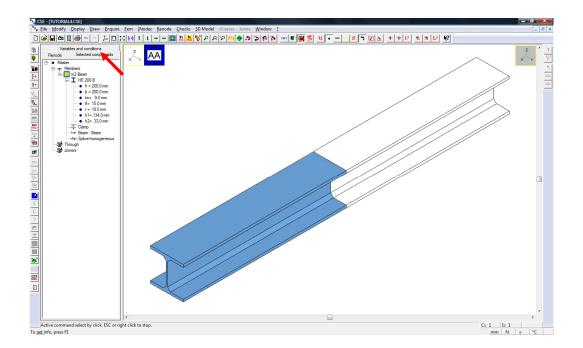


By pressing it you will get into the 3D environment where RENODES are built up, checked, and studied. You will see what follows:





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.



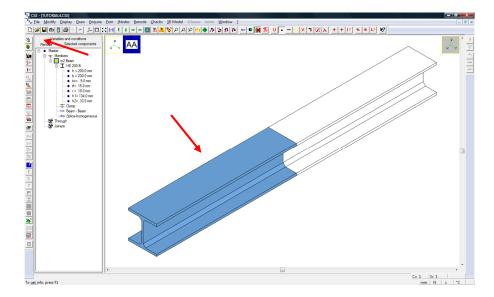
# 2.5 STEP 5: CONSTRUCTION OF THE REAL NODE

*N.B.* in this tutorial the renode will be manually build step by step, but it is also possible to apply one of the available parametric renodes of the archive that apply to the current renode. The command is Renode - Assign Prenode **b**.

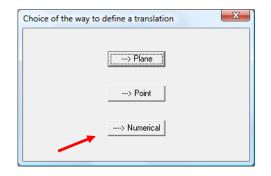
## 2.5.1 Addition of plates

We need to trim members in order to get the space where plates can be inserted. Select a member, then use **Renode – Members - Trim-extend** command ().





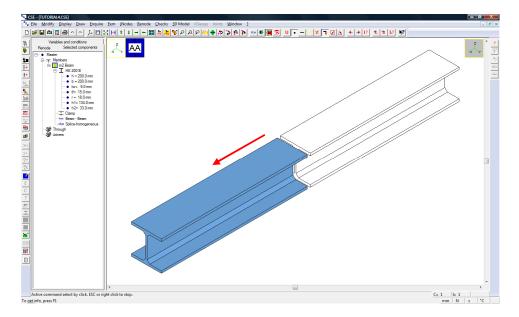
The following dialog box appears; choose the numerical mode.



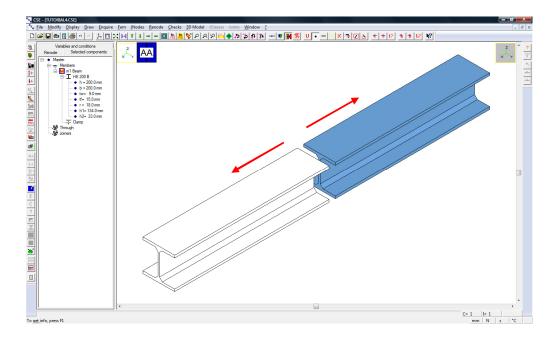
In the next dialog box type '-20' and press OK the define a trimming equal to 20mm.

Extension or shortening					
-20 Extension (+) or shortening (-)	ОК	Cancel			





Now unselect the first member, select the second one and repeat the previous operations in order to trim the other member.



Use **Display – Scene points** to show some relevant points useful to easily add the plates. In the dialog box, tick "Display points" on the left and keep only "Face centres" tick on the right. This will create and show the centre of every face in the scene.

ne points display mode and creation rules	;
Scene points display	Scene points creation
<ul> <li>Display points</li> </ul>	Sides mid-points
1 Points dot size (mm)	☐ Sides points at thirds
	☐ Side points at fourths
	Face centers
	0 Delta S =0

N.B. you can modify points size in the scene with the proper parameter in the dialog box

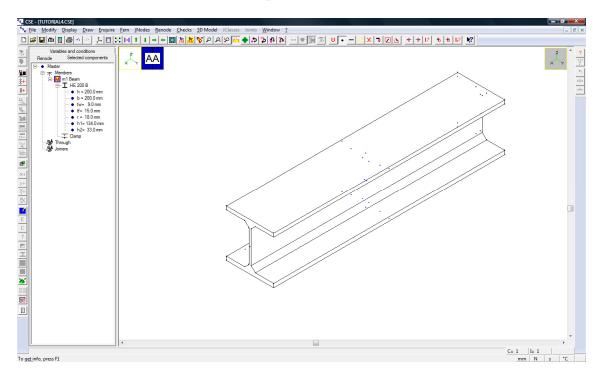
Now you can see centres of the faces (in addition to points on the corners, always displayed if the box on the left in previous dialog box is ticked). You can also create points without showing them in the scene.

Use **Draw – Extract** is to extract selected member. Tick "Selected objects" in the following dialog box and press OK.

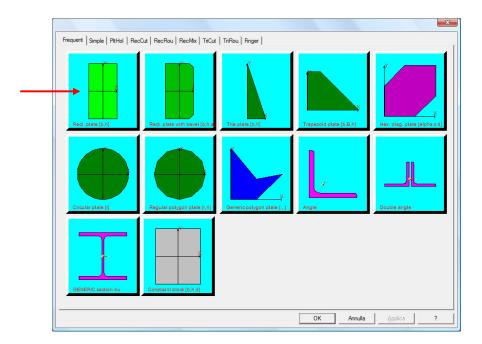
Extraction	X
1 · Description undefined.     2 · Description undefined.     3 · Description undefined.     4 · Description undefined.     5 · Description undefined.     6 · Description undefined.     8 · Description undefined.     9 · Description undefined.     10 · Description undefined.     10 · Description undefined.	What extract Selected groups Selected objects
	Select all!
	Unselect all
	Negate current selection
	Cancel

You will get the following. Zoom to have a closer view (use mouse wheel or commands in *Draw* menu)





To add the plate, use **Renode – Components – Add through** (<sup>to</sup> button in left toolbar), then choose the rectangular plate in the following property sheet.

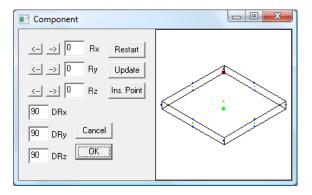


A new dialog box appears. Define plate size as follows: height and length equal to 200(mm, the current units), thickness equal to 20mm. You can see member sizes in the alphanumeric view on the left.



🤟 CSE - [TUTORIAL4.CSE]	x
😪 File Modify Display Draw Enquire Ferr JNodes Renode Checks 3D Model JClasses Joints Window ?	- 8 ×
Verdels and condom         Texce       Solidat condom         Te	
C= 1  = 1	
To get info, pres F1 N s °C	C

Press OK; in the following dialog box choose the orientation of the plate and the insertion point to add the component in the scene.



Press right Ry button once to rotate the plate in the position needed.



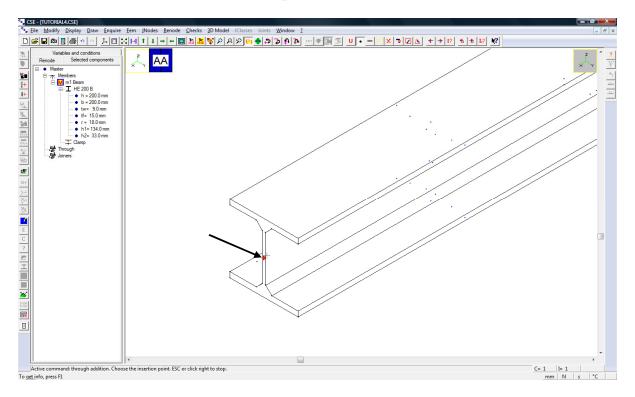
Component	
$ \begin{array}{cccccccccccccccccccccccccccccccccccc$	

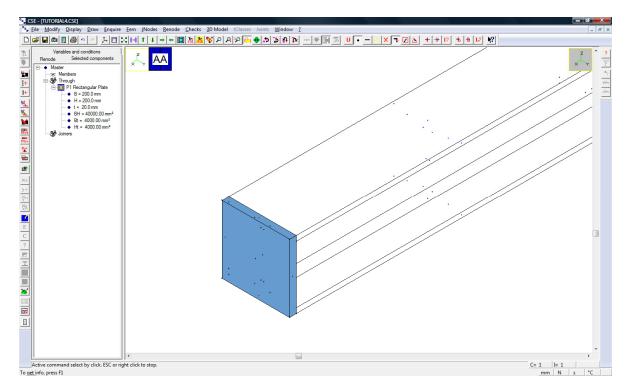
Press "Ins(ert) Point" and click the centre of the second face, then click OK.

Component	
<	

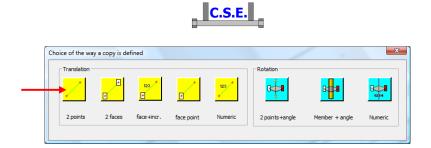
Now click in the scene the point corresponding to the insertion point of previous dialog: it is the centre of member end face.



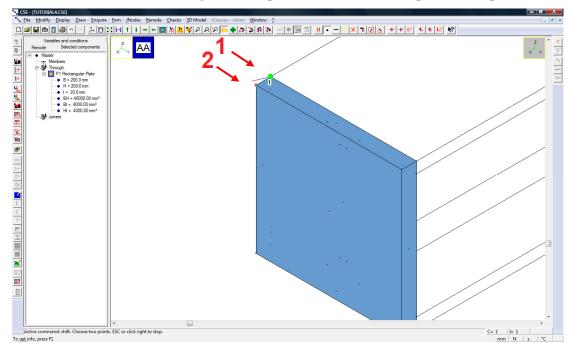


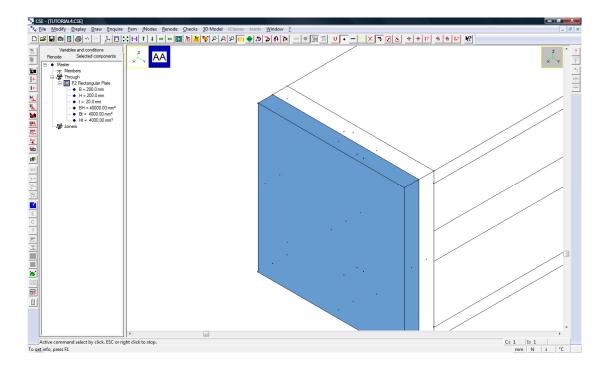


The first plate has been added. Now copy it to get the second one. Use **Renode – Components – Copy** and select a "translation" using "2 points".



In the scene, define a vector clicking, for example, the corners on the top left of the plate.





Castalia s.r.l. - all rights reserved - 25



A copy of the first plate has been created. Hide displayed points removing the tick in the dialog box and exit from the extraction mode (re-execute the command, and if it is not currently available, just end all the other commands pressing ESC or mouse right button).

 • Cet HuldCANCEL
 • Cet Books Docks Doc

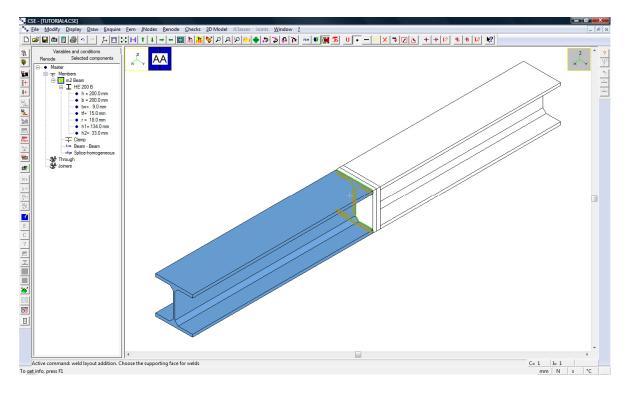
Enclose all the components in the view.

## 2.5.2 Addition of joiners

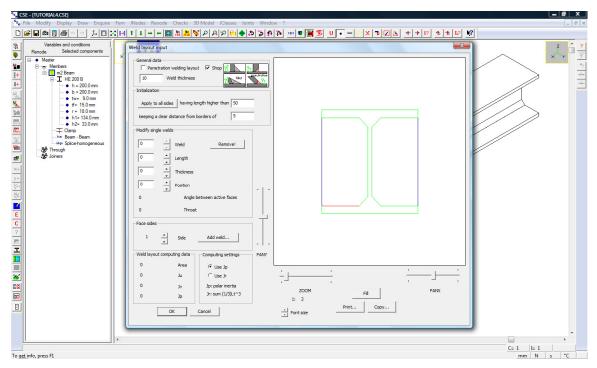
Note: keep only m2 selected, in order to limit the choice of the face to be welded to this component faces. Keep also "Selected components" pane active in order to have HEB200 data in alphanumeric view (alternatively, use "Renode" pane and click on  $\pm$  to open cross-section data; in this case, member selection is not needed). If you click on alphanumeric view, this one becomes the active view: click in graphic view to activate it again.

Now press  $\ddagger$  button (**Renode – Components – Add weld layout**), then move mouse pointer towards member end face in the connection zone.





When end face is highlighted, click mouse left button to select it: the following dialog will appear.

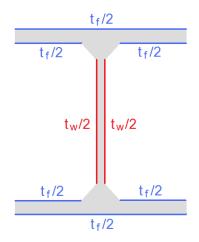


Note that behind the dialog you can see HEB200 data in alphanumeric view on the left.

CSE automatically recognizes the second face to be welded, and in the dialog box figure you can see member section and plate borders.



We are going to define a penetration welds layout as shown in the following figure, with welds having a thickness equal to the half of corresponding plate thickness.

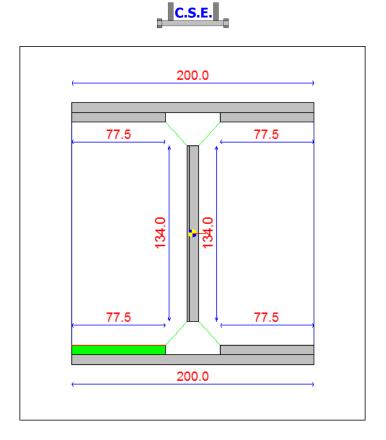


As shown in "Selected components" pane,  $t_f$  is 15mm and  $t_w$  is 9mm, so  $t_f/2$  is 7.5mm and  $t_w/2$  is 4.5mm.

In the top left part of the dialog, tick "Penetration welding layout", then type '7.5' as weld thickness  $(t_f/2)$  and '0' in "keeping a clear distance from borders". Use "Shop" welds.

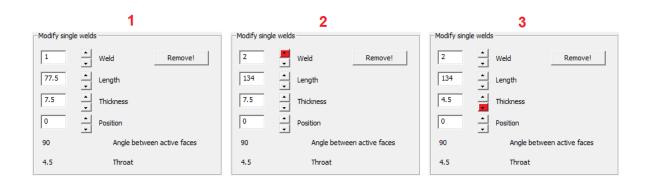
Apply to all sides having length higher than 50	General data Penetration welding layout 7.5 Weld thickness	Shop ?
	- Initialization	
keeping a clear distance from borders of 5		

Click "Apply to all sides" and the following welds will be automatically added:



The welds on the web currently overlap, because they should have a thickness equal to 4.5mm instead of 7.5. Now we are going to modify these parameters.

In the "Modify single welds" section, select weld number 2 with the upper "Weld" arrow (step 2 in the following image), then decrease its thickness with the lower "Thickness" arrow (step 3).



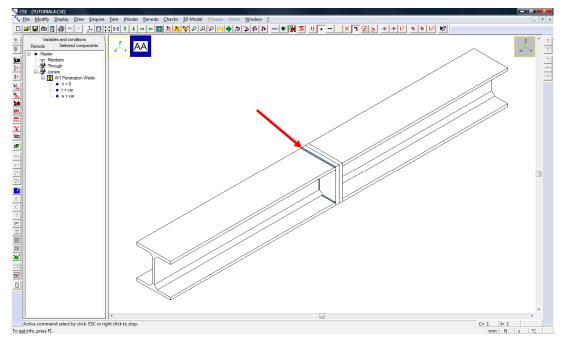
Do the same with weld number 6.



Weld layout input	x
General data         Image: Penetration welding layout       Shop         7.5       Weld thickness         1       Apply to all sides         having length higher than       50         keeping a clear distance from borders of       5         Modify single welds       6         6       Weld         134       Length         4.5       Thickness         0       Position         90       Angle between active faces         4.5       Throat	200.0 77.5 77.5 77.5 77.5 77.5 77.5 77.5
Weld layout computing data         Computing settings         PANY           6531         Area         (° Use Jp         (° Use Jr           19892199.125         Jv         (° Use Jr         Jp: polar inertia           67801972.4375         Jp         OK         Cancel	ZOOM 1: 2 Fill PANX Font size

Now there is no overlap between welds. Leave "Use Jp" for torsion computation and press OK to insert the layout.

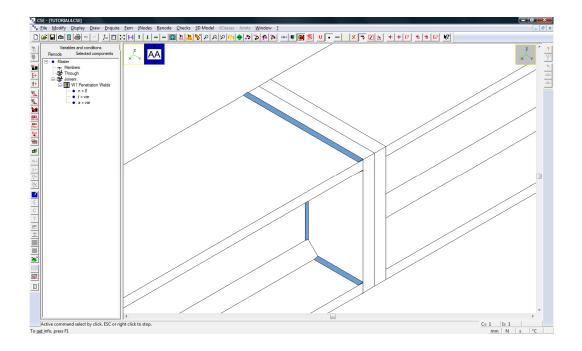
Unselect the member, keeping weld layout selected.



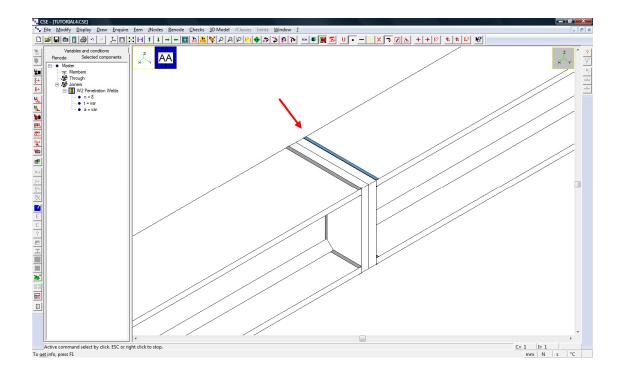
Castalia s.r.l. - all rights reserved - 30



Unselect the weld layout and zoom to get a closed view of the welds (use mouse wheel or **Draw – Zoom in** and **Zoom in Click** commands).



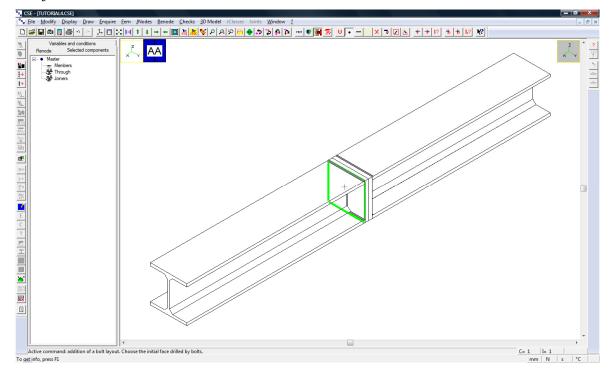
Now select m2 and repeat the previous operations in order to add a new weld layout connecting m1 to its plate (the new layout could be obtained also with copy command).



Castalia s.r.l. - all rights reserved - 31



Finally, we need to add a bolt layout. Unselect all the objects ( $\times$ ). Use **Renode – Components – Add bolt layout**  $\stackrel{\text{(s+)}}{=}$ , then click an external face of a plate: CSE will automatically recognizes all the objects that must be drilled.



You get the following dialog box.

Bolt layout input	ALADPIN	
Bolts	Arrangement	
M12-5.6F	Regular C Circular	
	C Staggered C Free	
Change	Empty inside	
Regular, staggered or circular a		
Quantity	Distances	
1 Rows	36 · Rows	
1 Columns	36 Columns	
Free arrangement		
Add Sel/Unsel		Þ
Remove + Bolt	None! • Y	
Center offset Ba	se information	i i i
0 <u>*</u> Dx 40	Net length	
0 - Dy 20	Minimum thickness -	[-] [E
0 Angle ! 1	Multiplicity	
Shear only bolts	Flexibility index	
Compressed bolts		
🔲 Slip resistant	Friction data	
🔲 Is an anchor	Anchor data	1 -
Use bearing surface	Bearing data	NY
🔲 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 action		
	N	Font size     Print     Copy     Fill     Hexagons
	V Plastic	
0 N mm	Mz	Do not print messages
OK Ca	ancel Set	

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



To define bolts class and diameters, press *Change* button in dialog box top left part, set 10.9 as class and M22 (EURO-HEX bolts) as diameter in the new dialog, then press OK.

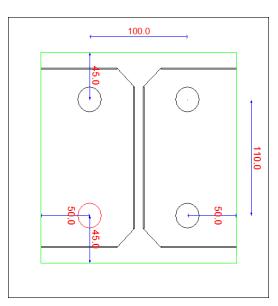
Bolt layout input	Bolt choice		Bolt layout input
Bolts M128.8F Change Regular, staggered or oxylar arrangemen Quantity Distanc	Bok © EURO-HEX C INDIAN-HEX C AMERICAN-HEX C EURO-HEX-HSFG C INDIAN-HEX-HSFG C AMERICAN-HEX-HS 10.9 Class 10.9 Class 10.9 Diameter Resistance area © Total C Threaded Hole	HEX: hexagor: HSEPI high strength friction grip: HS high strength	Bolts M22-10.9F Change Regular, staggered or circular arrangemen Quantity Distance
	© Normal	OK Cancel	

Keeping the regular disposition of bolts, define 2 rows and 2 columns, with a distance equal to 110mm between the rows and equal to 100mm between the columns.

Regular, staggered or circular arrangement				
Quantity	Distances			
2 · Rows	110 · Rows			
2 Columns	100 Columns			

The image is updated in real time.

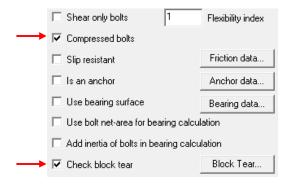




Note that distances between bolts and to current face borders are displayed. Use the buttons under the figure to change the current face. If minimum distance between bolts or from the borders is smaller than current standard limit values, a warning message will appear when we will try to add the layout.

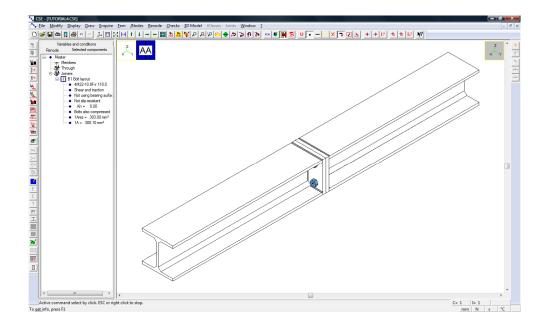
We have to decide bolt layout computational hypotheses (shear-only bolts, bolts in compression/tension, bolts in tension + bearing surface in compression, slip-resistant joints, anchors, etc.).

See the guide for more information about bolt layout computational hypotheses. Here just tick "Compressed bolts" to assign bending resistance to bolts only and "Check block tear" to include this failure mode in automatic checks.



Press OK to insert the layout.

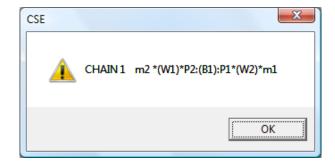




The renode has been completed. Use **Renode-Check Overlaps** to be sure that components do not overlap. In this case there are not overlaps.



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



The only chain of this renode is listed, so the connection is correct. The only connection path is:



## 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 with the following sections.

Code: 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", "Defined Values" or combinations imported "From table". Choose elastic for instance. Then specify the multiplier of internal actions for each component. Here we assume 0.3 for all the components. 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,  $M_2$ ,  $M_3$ , 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, so you can leave default multipliers values for m1. Use the arrows to select m2, then type the desired multipliers (0.3 everywhere in this case).

#### Keep default Partial safety factors.

In the section **Checks to be executed**, tick "Bolt pressure bearing checks", "Punching shear checks" and "Block tear checks". Since we no members net sections, user checks have not been defined and there are not cleats with the exception of the plates, we do not need any additional check (plates are not covered by simplified through resistance checks). We are not going to ask fem models automatic creation in this tutorial (see inherent tutorials for aspects not covered by the present one).



Keep default **Displacements bounds** for displacement checks.

TT1 C 11 ' C'	1 /1	1 1 1	• 1 1	•
The following figu	re shows the	dialog box	with the settings	we are lising now
The following figu		unulog box	with the settings	we use using now.

de	Internal actions computing mode	Partial safety factors
CNR 10011 - Allowable stress CNR 10011 - Limit states	C From FEM combinations C Worst only	1 gammaM,0
Eurocode 3 - EN 1993-1-8 IS 800: working stress	C Plastic limits	1 gammaM,1
IS 800: limit states	C Defined values C From table	1.25 gammaM,2
AISC-ASD: allowable stress AISC-LRFD: factor design	1         N,axial force, compression           1         N,axial force, tension	1.1 gammaM,3
Alse EN D. Inclui design	1 V2, shear force	ganinan,5
ing • Enalish	1 V3, shear force	1 gammaM,4
Ttalian	1 M1, twisting moment	1 gammaM,5
🖱 Spanish	1 M2, bending moment	,
Open when finished checks	1 M3, bending moment	
Including results (expanded)	✓ Use info about end release	
ecks to be executed		
Bolt pressure bearings	Parastitic bending in bolts	FEM analysis of components
Execute checks	Neglect parasitic bending	• Do not create models
Punching shear checks	Net cross-sections members checks	<ul> <li>Create just sketch models</li> <li>Create complete models</li> </ul>
Execute checks	Execute checks	C Create and analyze models
Block tear checks	User checks (added formulae)	C
Execute checks	Execute checks	<ul> <li>Use Sargon/Clever</li> <li>Use Sap2000</li> </ul>
Simplified through checks		C Use other
splacement bounds of components to	o print a warning message	
· · ·	0.0087266 Rotation (radians)	
T dibidion	•	

Press **OK** to save settings.

# 2.7 STEP 7: EXECUTING CHECKS

To execute the checks press the following button in the left toolbar.





nning		
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		
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	=
End of checks.		
		<b>T</b>

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.

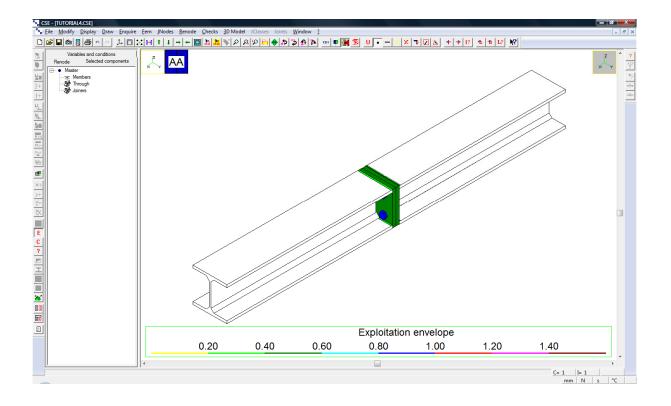
The output file has been automatically opened. Have a look at the file if you wish, then minimize or close the output file window.

📋 TUTORIAL4.CSE.AA.EURO3.out - WordPad	3
Eile Modifica Visualizza Inserisci Formato 2	
*****	*
÷ ÷	
# C.S.E. #	
≠	
* Connection Study Environment *	
# 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	
	-
Standard Citemen D	
Per ottenere la Guida, premere F1	. di



# 2.8 STEP 8: EXAMINING RESULTS

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



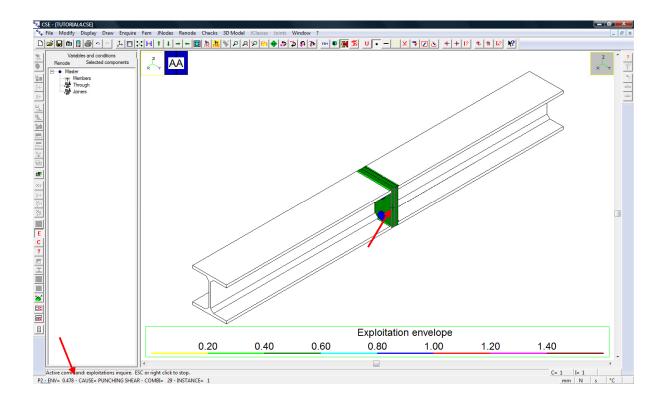
Only the following aspects have been checked, in this tutorial:

- bolts resistance
- welds resistance
- bolts bearing pressure on plates
- punching shear on plates
- block tear on plates

Members are white because they were not involved in any check. They do not have hole for bolts nor cuts. Members resistance check is covered by weld layout resistance check, being on the safe side (as explained later).



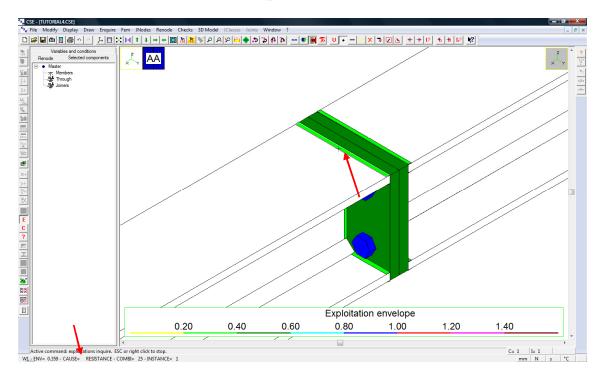
Now use the **Checks – Enquire** command ? and move mouse pointer towards a face of the welds to get info about its exploitation (note: use mouse wheel or zoom commands in order to get a closer view).



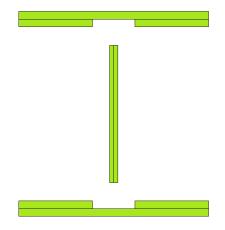
For example, maximum exploitation for P2 plate is 0,478 in combination 29 (m2 positive M2 bending moment); the cause is punching shear. Enquire the other components to know their maximum exploitation.

Welds maximum exploitation is 0,359 in combination 25 (m2 positive axial force). This is the maximum value for all the welds.





Axial force for computation is the elastic limit value of HEB200 shape multiplied by 0,3, but for the welds we have an exploitation greater than 0,3: the reason is that penetration welds layout we have defined does not cover the whole cross-section, as shown in the following figure:



Weld layout has a smaller area than HEB200 shape.



Area of HEB200 = 7808 mm<sup>2</sup> Weld layout total area = 6531 mm<sup>2</sup>

Welds exploitation is equal to the ratio between HEB200 area and weld layout area.

 $0.3 \times 7808 \text{mm}^2 / 6531 \text{mm}^2 = 0.359$ 

Note that if you use cross-section plastic limits to compute forces and moments for the checks, exploitations in welds will increase, because they're not computed using plastic limits.

Use **Checks – Combi** [12], select combination 25 in the dialog (m2 positive axial force) and press OK.

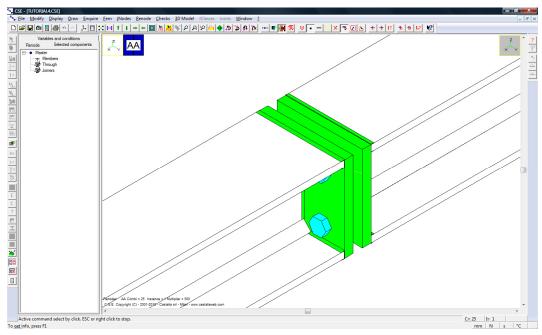
Setting of a combinat	ion			×
Combi = 11			*	
Combi = 12				
Combi = 13				
Combi = 14				
Combi = 15				
Combi = 16				
Combi = 17			=	
Combi = 18				
Combi = 19				
Combi = 20				
Combi = 21				
Combi = 22				
Combi = 23				
Combi = 24			_	
Combi = 25				
Combi = 26			*	
	OK	Cancel		

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



Note: if displacements are too small use **Checks – Displaced scale** and set a greater multiplier.

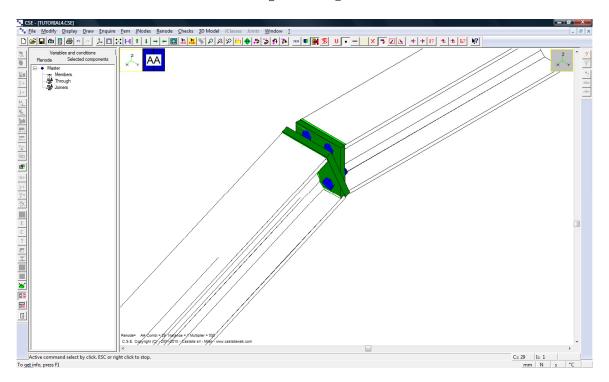
When the displaced view is shown, components are coloured with the range colour corresponding to their exploitation in current combination. In the following figure, displaced view in combination 25 is shown.



Switch to following combinations 1 to see displaced in combination 26 (T<sub>2</sub><sup>+</sup>), 27 (T<sub>3</sub><sup>+</sup>), 28 (M<sub>1</sub><sup>+</sup>) and so on.

The following image shows displaced view in combination 29 (positive  $M_2$ ).







1	INTRODUCTION	<u>3</u>
2	SPLICE JOINT WITH PENETRATION WELDS	4
2.1	STEP 1: GETTING THE FEM MODEL	4
2.1.	.1 GETTING THE FEM ELEMENTS	4
2.1.	2 Assigning material	6
2.1.	.3 Assigning cross-section	7
2.2	STEP 2: SEARCHING MEMBERS	
2.3	STEP 3: SEARCHING JNODES	
2.4	STEP 4: SELECTING THE PROPER JNODE	
2.5	STEP 5: CONSTRUCTION OF THE REAL NODE	
2.5.	.1 ADDITION OF PLATES	
2.5.	.2 ADDITION OF JOINERS	
2.6	STEP 6: SETTING CHECK SETTINGS	
2.7	STEP 7: EXECUTING CHECKS	
2.8	STEP 8: EXAMINING RESULTS	