



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
- Add plates and bolt layouts
- Set the checks to be performed
- Have a look at the results
- Comprehend block tearing results
- Comprehend members net sections results
- Comprehend how to use flexibility index for shear-only bolts layout

This tutorial is some like 59 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 (BOLTED CONNECTION)

FULL MODE		
	FULL	
Using FULL version you can create All LIGHT version commands are a	e freely new RENODES, and besides yu can record new parametric ones available, but the interface is not simplified	
	LIGHT	
LIGHT version uses a simplified int	terface but only standard nodes can be created, belonging to the database	

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

5. C	SE - [CSE1]		- 8 - ×-
5.	File Modify Display Draw Enquire	Fem [Nodes Benode Checks 3D Model /Classes Joints Window 2	- Ø ×
2	Renode Selected components	2	1 1
		×Y	× ~ <u>1</u>
<u>111</u>			
1+			-
5			
先			
20			
<u></u>			
Bet			
107			
×+			
>+ >+			
3			
E			
C			
I			
- E E E			
<u> </u>			
			· · ·
		(= 1	
To g	t info, press F1		IN SC

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	
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			
Directly sea	arch for inodes		
I. Direcuy sec	aren for jilodes		
		Concerned and Concerned	



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	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
3275	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
4529	7.850e-005	2.000e+005	3.000e-001	2.900e+0
4242/42	7 850e-005 III	2 000e+005	3 000e-001	2 900e+0 *
	Applica	Cancella		

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.

I Ush (eox, sharp comers) I Dmega L I Generic polygons I Z Generic I Generic I C I I	Kind Rolled H shapes IPE DIL W HEA HL HLS HEB HX ILS HEM HD H IPE* HP I HE M IPN+ISMB	Others LSM (Equal legs USM (Channels) LSM (Unequal leg TSM PSH (Plates, Rec 0 (Tubes and Rc RHS (Rectangula THSM (H cut as Composed U_0 [] L2_T Cold formed	angles) European angles) American js angles) Indian stangular) Name search unds) Activate 2_CR Limits 4_CR Design criteria	Name No Limits
Lance	Generic Generic	C Omega C L Z C C C C	Design	lo 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	р	^	
IE 100 B	 2.604e+003	2.044e-001	4.	
IE 120 B	 3.401e+003	2.670e-001	5. =	
IE 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.	
	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.	
TR 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	*	E
HE 200 B	1	Assign current!	I✓ Strong axis
???	2	Assign current!	Strong axis
6			
Cross-sections			
Cross-sections	current	Archive	Add new
Cross-sections	current	Archive	Add new
Cross-sections HE 200 B HE 200 B	current	Archive	Add new
Cross-sections HE 200 B HE 200 B HE 200 B	current	Archive Assign current!	Add new Strong axis Strong axis
Cross-sections HE 200 B HE 200 B HE 200 B	current	Archive Assign current!	Add new Strong axis Strong axis
Cross-sections HE 200 B HE 200 B HE 200 B	current	Archive Assign current!	Add new

		_	1	
5235		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° 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:

ode	Internal actions computing mode	Partial safety factors
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 sting English Italian Spanish Open when finished checks	C Elastic limits Im1 Member C Plastic limits Im1 Member C Defined values C From table 1 N,axial force, compression 1 N,axial force, tension 1 V2, shear force 1 V3, shear force 1 M1, twisting moment 1 M2, bending moment 1 M3, bending moment	1 gammaM,1 1.25 gammaM,2 1.1 gammaM,3 1 gammaM,4 1 gammaM,5
Including results (expanded)	✓ Use info about end release	
Punching shear checks ■ Execute checks ■ Execute checks ■ Execute checks ■ Execute checks ■ Execute checks ■ Block tear checks	Parastitic bending in bolts Reglect parasitic bending Net cross-sections members checks Execute checks User checks (added formulae)	FEM analysis of components © Do not create models © Create just sketch models © Create complete models © Create and analyze models © Use Sargon/Clever
Execute checks Simplified through checks Execute checks	Execute checks	C Use Sap2000 C Use other
isplacement 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
	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-sectior	1				
Beam		B	ement kind					
Clamp		Jo	oint kind					
Splice-homogene	eous	A	ignment					
nvelope of inten	nal forces	in slave -						
Positive				Negative				
Positive 0.000e+000	N+	0	Element	Negative	N-	0	Element	
Positive 0.000e+000 0.000e+000	N+ 	0	Element Element	Negative 0.000e+000 0.000e+000	N- T2-	0	Element	
Positive 0.000e+000 0.000e+000 0.000e+000	N+ T2+ T3+	0	Eement Eement Eement	Negative 0.000e+000 0.000e+000 0.000e+000 0.000e+000	N- T2- T3-	0	Element Element Element	
Positive 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000	N+ T2+ T3+ M1+	0	Element Element Element Element	Negative 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000	N- T2- T3- M1-	0	Element Element Element Element	
Positive 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000	N+ T2+ T3+ M1+ M2+		Bement Bement Bement Bement Bement	Negative 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000 0.000e+000	N- T2- T3- M1- M2-		Eement Bement Bement Bement Bement	

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 TUTORIAL5.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.

TUTORIAL5.txt - WordPad	
<u>File M</u> odifica <u>V</u> isualizza <u>I</u> nserisci F <u>o</u> rmato <u>?</u>	
□☞■●Δ.#4 ※◎竈ゃ 喝	
***************************************	***
* ·	*
*	*
* C.S.E.	
Connection Study Environment	
	2 •
* Copyright 2000-2010 Castalia s.r.l http://www.castaliaweb.com	
******	***
***************************************	***
	*
* TECENID	
* DEGEND	÷
*	*
* 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	
* In the same point (node). A single jnode can be associated to several nodes of a FEW node	
* CO Several nodes of a FEM node	*
* MARK)*
*	*
* MEMDED Drigmatic single part before vertings. It is modelled with	
Per ottenere la Guida, premere F1	

In particular you will get the following info:

```
MEMBERS IN THE JNODE: TOTAL 2

1 2

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

Beam 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:



Castalia s.r.l. - all rights reserved - 16



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 rectangular plates

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

Scene points display mode and creation rules	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			
OK Cancel				

- Tick Display points
- Type '2' as dots size



• Remove all the ticks in "Scene points creation" section, but keep "Sides mid-point" ticked.

Use Draw – Rotate down it twice to have a clear view of the webs in the connection zone, and scroll mouse wheel to have a closer view.



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



Castalia s.r.l. - all rights reserved - 18



The dialog box that appears is used to set rectangular plate sizes and parameters.

×
100 ÷ Height (DY)
100 ÷ Length (DX)
10 Thickness (DZ)
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
\square Search and mesh stiffeners when preparing fem model
OK Cancel

We are going to add on of the plates of the web.



Note: open member data in the Renode pane to see cross-section sizes, or alternatively (recommended) activate Selected components pane and select one of the members. Remember to activate graphic view before proceeding, clicking in it with the mouse.



Define plate sizes as follow and leave all the other default data. Now we are not going to use automatic fem model creation.

220	÷	Height (DY)
130	÷	Length (DX)
10	÷	Thickness (DZ)

Click OK in the rectangular plate dialog box, and a new dialog will appear: here you can rotate the component you are going to insert, and choose the insertion point.



Click \rightarrow in Ry 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 hidden face, 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.



Now use **Renode – Components – Copy** to get a copy of the plate on the other side of members webs.

There are different ways to define copy vector; choose Translation - Numeric.



We need a 19mm translation in -Y direction: -19 is the sum of plate thickness (10mm) and web thickness (9mm). Tupe -19 in DY box and press OK.



Dialog			×
0	DX		
-19	DY		
0	DZ		
		OK	Cancel

A second plate has been added: use **Draw – Standard views** and choose +X view to see both plates.

SE - [TUTORIAL5.CSE]		
°5₀ Eile Modify Display Draw Enqui	ire <u>E</u> em [Nodes <u>R</u>enode <u>C</u>hecks <u>3</u>D Model JClasses Joints <u>W</u>indow <u>?</u>	_ 6 ×
	▯៲∷ฅๅๅๅ→⊢◙≱₽%₽₽₽₽₩₩₽₽₽₽₩™₽₿₫ ฃ・−≥>	5 3 2 6 4 1 1 7 6 1 1 2 1 2
Varables and conditions Rerode Selected components :: : :: : :: : :: :		
Active command select by click. ESC or To get info, pres F1	right click to utsp.	Cr 1 is 1 , , , , , , , , , , , , , , , , , ,

Switch back to the first isometric view from the left .

Standard Views		×
+X 4	<u>→ +</u> Y	↓ +Z
7 -X	✓ -Y	↑-Z
	SO ISO	ISO

To add the plates on the flanges, add a new component as done before: Add through **b**, then choose the rectangular plate.



Type Height=650, Length=200 and Thickness =15, then press OK.

	×			
650 🕂	Height (DY)			
200 🕂	Length (DX)			
15	Thickness (DZ)			
P3	Name			
S235	Material Change			
Fem modelling				
🗖 Create Fl	EM model			
10	10 Borders and welds element size			
30	Generic elements size (if 0 then free size)			
29	29 Triangle minimum angle in degrees (default 29*)			
0.1 Node distance tolerance (if dist < tol then the nodes are merged)				
🗖 lt is a stif	☐ It is a stiffener			
🗖 Search a	nd mesh stiffeners when preparing fem model			
	OK Cancel			





Component	
⟨ > 0 Rx Restart ⟨ > 0 Ry Update ⟨ > 90 Rz Ins. Point 90 DRx 90 DRy Cancel 90 DRz	

- Click \rightarrow in Rz row
- Click "Ins.Point"
- Click centre point of the lower face
- Press OK

Note: you can enlarge dialog box with the mouse.



In the scene, click the point to which must correspond previously chosen point: it is the midpoint of flange upper side (zoom with mouse wheel before clicking in order to have a closer view).





The component has bee added.



Hide Scene points 🐱 because they are not necessary.



Scene points display	Scene points creation Sides mid-points Sides points at thirds Side points at fourths Face centers
	0 Deita S _=0

Enclose the renode in the view 🛄 (**Draw – Enclose**).

Upper plate is currently selected. We are going to create a copy on the lower flanges: **Copy** then choose Rotation – Member+Angle.

Choice of the way	/ a copy is de	fined			-	20	×
Translation -					Rotation		
*	e E	123 * +	•	123 * +	Br∰∎	B	
2 points	2 faces	face+incr.	face point	Numeric	2 points+angle	Member + angle	Numeric

Using this mode, you must firstly click a face of the member whose axis must be taken as rotation axis, then an angle must be typed: a copy (2) of the original object (1) will be created.



Click on any of the face of any of the members, since they have the same axis (click, for example, an end face).





Press OK in the following dialog box to have a rotation of 180 degrees.

Angle input		x
	180 Angle in degrees	
	Cancel	

A copy of the plate has been added on the lower flanges.



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



As done before, change view to see the renode frontally, then switch back to the first isometric view.



2.5.2 Addition of bolts

Unselect all the components — Add bolt layout $\stackrel{\text{(i)}}{\longrightarrow}$ and click in the scene the external face of the first added plate.



Castalia s.r.l. - all rights reserved - 29



Click the "Change" button in top left part of the dialog box, then choose 10.9 class and M16 diameter in the new dialog box. Press OK to return on previous dialog.





Keeping the regular arrangement, define 3 rows and 2 columns using the proper arrows. Since we have chosen a diameter greater than the previous one, distances have been automatically settled to minimum distances between M16 according to EN1993-1-8.

Regular, staggered or circular arrangement			
QuantityDistances			
3 • Rows	43.2 • Rows		
2 📩 Columns	43.2 Columns		

The image is updated in real time; type '55' in Dx box of "Center offset" section to move bolt layout 55m rightwards.

Center o	offset-		
55	•	Dx	
0	•	Dy	
0	\vdots	Angle	!

The result is shown in the following image. Use zoom control to have a closer view; 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 \therefore 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:

- tick "Shear only bolts": bolts will have no resistance under axial forces
- tick "Check block tear" to include block tear in the checks
- remove all the other ticks
- refine a flexibility index equal to 3:

✓ Shear only bolts 3	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				
Add inertia of bolts in bearing calculation				
Check block tear	Block Tear			

Using shear only bolts in the renode we are building, web bolts contribution in carrying a shear parallel to the flanges will be smaller, as flanges bolts contribution in carrying a shear parallel to the web.



Flexibility index is use to modify bolt layout stiffness; using an index greater than 1, bolt layout will be more flexible and forces will be carried by other components. This aspect will be explained in the appendix.

Click "Block tear" button to see a dialog box with information about block tear failure paths of the faces involved by bolts. Use the arrows to select a face (green): in function of settled force direction



(blue circle, 0 = horizontal force), failure path of current face is displayed in the image and ultimate force of the plate is given.



Use arrows to change force direction: image and F ultimate will be updated in real time.



Castalia s.r.l. - all rights reserved - 33



Assuming that all the bolts will carry the same load, the program predicts a maximum shear force equal to 156528,8N for a force with 60 degrees inclination. If you press the Diagram button you get what follows, i.e. a diagram plotting the maximum force vs the angle of the resultant. If you ideally draw a line from the centre at an angle a relative to the horizontal rightward semi-axis, you cut the green line in a point which gives you the maximum force that can be applied in that direction, without tearing the *face* of the object currently selected. Block tearing loops over the *faces* of each connected object.



Use **Copy** button to copy the image. Press OK to exit from block tear dialog box, then press OK in main dialog box to insert the layout in the scene.



Castalia s.r.l. - all rights reserved - 34



To add the other layout on the web, we muse unselect first bolt layout or select the plate, because in bolt layout addition, if there are selected components only their faces can be clicked. Select the plate, use Add bolt layout is and click plate external face again.



Type a '-' before 55 in Dx box, keep all other parameters and press OK.



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



Second bolt layout has been added.



Now select upper plate, use Add bolt layout 3+ and click upper face of upper plate.





The following message will appear, but it is not a problem because initial layout settings are those of last defined layout, that can not apply to the layout we are going to insert. Press OK.



Two bolts are in correspondence of member web, but it is obvious that previous layout parameters must be changed.

- Define 2 rows and 7 columns using the arrows, then type 120 as distance between rows.
- Type 162 as Dx offset.

Regular, staggered or circular arrangement			
Quantity	Distances		
2 • Rows	120 · Rows		
7 📩 Columns	43.2 Columns		

Center o	offset-		
162	• •	Dx	
0		Dy	
0		Angle	ļ



Layout will appear as follows:



Keep the same ticks of previous layout, but type 1 as flexibility index:

🔽 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				
Add inertia of bolts in bearing calculation				
Check block tear	Block Tear			

Press OK to insert the layout.





Select upper plate, then add a bolt layout 3+ and click the same face as before.



Type a '-' in the dialog, keep all the other parameters and press OK.

Center	offset-		
-162	•	Dx	
0	\vdots	Dy	
0		Angle	!





Two get lower plate and bolt layouts, select upper components, then use **Renode – Components – Recopy** command. Last defined copy vector was the one used to copy upper plate under lower flanges, and now we need the same vector: Recopy command uses last defined vector.



Castalia s.r.l. - all rights reserved - 40





2.5.3 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 4 different paths from m2 member (the slave) to m1 (the master). For example, forces can follow this path (chain 1):

member m2 \rightarrow bolt layout B2 \rightarrow plate P1 \rightarrow bolt layout B1 \rightarrow member m2



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", "Defined Values" or combinations imported "From table". Choose elastic 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, M₂, M₃, 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.

Use the arrows to select m2, then type the desired multipliers. For example:

- 0,5 for compression, tension and bending moments
- 0,1 for shear 2, shear 3 and for torsion (we want to compute the renode mainly in bending and azial force)

Keep default "Partial safety factors".

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

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

Keep default "Displacements bounds" for displacement checks.



de	Internal actions computing mode	Partial safety factors
CNR 10011 - Allowable stress	C From FEM combinations 🔲 Worst only	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	Elastic limits Plastic limits Plastic limits Defined values From table D.5 N,axial force, compression 0.5 N.axial force, tension	1 gammaM,1 1.25 gammaM,2 1.1 gammaM,3
	0.1 V2. shear force	
Fnglish	0.1 V3. shear force	1 gammaM,4
Italian	0.1 M1, twisting moment	1 gammaM,5
Spanish	0.5 M2, bending moment	
Open when finished checks	0.5 M3, bending moment	
 Including results (expanded) 	✓ Use info about end release	
esks to be executed		
Execute checks	Parastitic bending in bolts Neglect parasitic bending	Do not create models
Durahing abase abasela		C Create just sketch models
	Freque checks	C Create complete models
Block tear checks	User checks (added formulae)	Use Sargon/Clever
I▼ Execute checks) Execute checks	C Use Sap2000
Simplified through checks		Use other
isplacement bounds of components to	print a warning message	
1 Translation	0.0087266 Rotation (radians)	

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		X
Running checks: instance 1 of 1 co Running checks: instance 1 of 1 co	nbination 42 of 48. nbination 43 of 48. nbination 44 of 48. nbination 45 of 48. nbination 46 of 48. nbination 47 of 48. nbination 48 of 48.	*
Enverope calculation Creation of listing Binary file creation Binary file creation Binary file creation Binary file creation Binary file creation Binary file creation End of checks.	step 1 of 6 step 2 of 6 step 3 of 6 step 4 of 6 step 5 of 6 step 6 of 6	E
		×

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.

UTUTORIAL5.CSE.AA.EURO3.out - WordPad	
<u>Eile M</u> odifica <u>V</u> isualizza Inserisci F <u>o</u> rmato <u>?</u>	
*****	<u> </u>
# # # C.S.E. #	
* * * * * * * * * * * * * * * * * * *	
* * www.castaliaweb.com *	
- * ÷	
A knoledgment	
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.

Castalia s.r.l. - all rights reserved - 45





While the envelope is displayed, use **Checks – Enquire** ? 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.



Castalia s.r.l. - all rights reserved - 46



You can inquire bolts, plate or members maximum exploitation. If you use **Checks – Current results C**, displayed and enquired exploitations refer to current combination.

Use **Checks – Combi** ^{L?}, select combination 25 in the dialog (m2 positive axial force) and press OK.

Setting of a combination	on	×
		_
Combi = 11		A
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		T
1		
	OK Cancel	

Use **Checks –Displaced** in left toolbar 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 in the dialog box.



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





Switch to following combinations P (Checks - Combi next) to see displaced in combination 26 (T_2^+) , 27 (T_3^+) , 28 (M_1^+) and so on.

The following image shows displaced view in combination 29 (positive M₂).



Un-press Displaced \blacksquare and switch back to combination 25 (tension) using \blacksquare or \blacksquare ?



Now select P3 and click Checks – Block tear results .



The following dialog box will appear:





In the figure is shown plate failure path in combination 25 (red arrows). F ultimate is the resistance of that failure path, the exploitation is the ration between applied force and ultimate force. Applied force is horizontal (-180degrees).

Remove the tick from "Display distances" to hide bolts distances.

For an applied force equal to the ultimate one, the following failure will happen.



Exit from the dialog box clicking OK or cancel.



Note: block tear results can be displayed only if block tear causes maximum exploitation for selected components in current combination, otherwise the following message is displayed:



Now un-select P3 and select m2, then use Checks – Display net sections results



When you execute the command, renode is hidden in the graphic view and net section results of selected member in current combination are displayed.

In this case, maximum exploitation for m2 in combination 25 is not caused by net sections checks, so the following message is displayed.





Switch to combination 26 (). Here we have net sections check as maximum exploitation, so results are displayed.





In the "Computing data" section the following are reported:

- Renode mark and member name
- Net section with maximum exploitation
- (Instance) and combination (here we have one instance only)
- Applied axial force and bending moments
- Exploitation
- Applied and allowable normal stress

Note that V2 multiplied for net section distance from theoretical extreme causes an high Mv moment. Applied V2 is the 10% of gross cross-section limit.

net section 9	
V I V	2
<u></u>	
Mvn () · · · · · · · · · · · · · · · · · ·	
distance	1

Follow next steps to know that distance.

Exit from net section results **I**, then use **Enquire – Net sections**. The following dialog box appears. It gives information about members with gross cross-section reductions due to bolt holes, cuts, bevels, etc.



Check net-sections of the selected member
Check net-sections of the selected member
AA.m2 - Section 9 - Distance 291.6 mm An=6731.03 mm ² W2n= 477187 mm ² W3n= 181170 mm ³ xg= 2.18192e-015 mm yg=-8.48676e-015 mm An / A= 0.862 W2n / W2= 0.838 W3n / W3= 0.804 Wpl2n / Wpl2= 0.743 Wpl3n / Wpl3= 0.527 C.S.E. Copyright (C) - 2001-2010 - Castalia srl - Milan - www.castaliaweb.com
Number of sections 9 Section 9 Vet section data Copy Print Cancel
Distance 291.6

Use the arrows to select section 9, in which we have maximum exploitation in combination 25.



Given data are:

- Renode mark and member name
- Net section number and its distance from theoretical extreme
- A, W2 and W3 of reduced section and ratios with correspondin values of gross cross-section
- Net section barycentre coordinates

Click "Net section data" button to have full net section information: the following dialog box will appear.



Data acquisition of a section made up by polygons		X
	6731.02832 A	0 it
X	47718728 J2	477187.28" W2
	16116972 J3	161169.718 W3
8	0 Jt	477187.28" Wpl2
X	84.1983795 i2	161169.718 Wpl3
	48.932907 ⁻ i3	0 U
	2.1619156 ² xG	-6.4857469 yG
	0 ×2	0 ×3
	0	Principal axes angle
Name Name	🔲 Computes plastic r	noduli
Add polygon	Upc	lates Cancel
Modify polygon		
Remove polygon!		
Shift polygon		

C.S.E.

3 APPENDIX: FLEXIBILITY INDEX

3.1 WHAT FLEXIBILITY INDEX IS?

In paragraph 2.5.2, a flexibility index equal to 3 was assumed for bolt layouts on members webs. Hereunder is flexibility index definition given in CSE guide.



Let's take for instance a splice joint with bolted plates (web and flanges), under tension: in the plastic range, a plasticization of the web thickness will occur due to bolt pressure, so that the internal actions will be redistributed to the flange bolt layouts, which will continue to carry the load. By setting a suitable flexibility index (greater than 1) in the web bolt layout, the user can model this effect and thus achieve the action-distribution desired.



The definition of a flexibility index greater than 1 for web bolt layouts causes a reduction of their stiffness, and a greater amount of applied forces is carried by other bolt layouts (flanges bolt layouts has a flexibility index equal to 1).

When a manual computation is done considering axial force, usually the load is divided between web and flanges considering the ratio between web area and flanges area: in this assumption, real stiffness of boltings is missed. Real distribution of forces depends from the ratio between web boltings and flanges boltings. CSE considers this aspect and, if we want to have a distribution similar to the one assumed in manual computation, we have to modify some bolt layouts stiffness (in this case, reducing flanges bolt layouts stiffness).

3.2 MODIFY WEB BOLT LAYOUTS

Let's see how flexibility index modifies forces distribution. First of all, create a copy of the previous model using **File – Save as**; name the new file "*TUTORIAL5_flexindex_study.CSE*", for example.

Unselect all components \checkmark ; if there is any active post-process command, end it (un-press corresponding button, for example switch from \checkmark to \checkmark in order to end net sections results display). Select one of web bolt layouts.



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



Use **Renode - Components - Modify** to enter in bolt layout modification dialog box. This dialog is the same used for bolt layout addition, but parameters are those of the layout we want to modify.

Define a flexibility index equal to 1, do not change any other value or parameter and click OK.



Now unselect currently selected bolt layout, select other web bolt layout and do the same as done before, in order to define a flexibility index equal to 1 for this layout too.

3.3 CHECK AND RESULTS

All bolt layouts have a flexibility index equal to 1, now. Execute checks

Display the envelope E: members exploitation has significantly increased. Enquire member exploitation ?.



Castalia s.r.l. - all rights reserved - 58



Maximum exploitation is in combination 31 (compression). Move to combination 31 \mathbf{L}^2 , show current combination exploitation \mathbf{c} and enquire member results $\mathbf{?}$.



Maximum exploitation for this member is caused by block tearing in correspondence of bolt layout B2, the one on the web. Applied loads are the same, but exploitation is greater than before (paragraph 2.8): this is due to flexibility index modification.

End inquire command pressing ESC or mouse right button, then exit from current results display $\boxed{c} \rightarrow \boxed{c}$.

Select m2, then use **Checks – Display forces** Select m2, then use **Checks – Display forces** Select m2. Forces exchanged from member to connected bolt layouts in current combinatio (31) are now displayed (use **Font** and **Size** commands in **Display** menu if values and/or vectors are too small).

Zoom to have a closer view; you can also extract member to display member and forces only (E1, then tick "Selected objects).





 F_w is the force trensferred from the member to web bolt layout, F_f is the sum of the forces transferred from the member to upper flange and lower flange bolt layouts.

 $F_w = 412862N$ $F_f = 2 \times 252288N = 504576N$

Note that on the flanges there is also a My moment due to force multiplied by the offset from member axis.



 $\% F_f$ is the portion of total compression force carried by flanges bolt layouts:

$$\%F_{f} = \frac{F_{f}}{F_{f} + F_{w}} \times 100 = \frac{504576N}{504576N + 412862N} \times 100 = \frac{504576N}{917438N} \times 100 = 55 \qquad (f=1)$$

Castalia s.r.l. - all rights reserved - 60



55% of total compression force is carried by flanges bolts layout. This datum has to be compared to the ratio between web and flanges areas.

HEB200 sizes are the following. Let's compute web and flanges areas without considering curvature radii (it's a coarse assumption in order to simplify computation and explanation).

h	200mm
b	200mm
tw	9mm
t _f	15mm
r	18mm

 A_w is the area of the web, A_f is the area of the flanges.

 $A_w = t_w x (h - 2t_f) = 1530 mm^2$ $A_f = 2 x t_f x b = 6000 mm^2$

 $%A_{f}$ is the ratio between flanges area and the sum of flanges area and web area:

$$\%A_f = \frac{A_f}{A_f + A_w} \times 100 = \frac{6000N}{6000N + 1530N} \times 100 = 80$$

So, in our ideal –simplified- computation we will assign the 80% of total load to flanges bolt layouts. In a real case, with the assumed bolts configuration, this will not happen, but we want to compute joint on the base of this hypothesis anyway. To do that, we need to reduce web bolts layout stiffness.

3.4 FLEXIBILITY INDEX INCREASING

Modify web bolt layouts flexibility indices, re-execute checks and read forces as done previous paragraphs.

A. Exit from *Display forces*: $\searrow \rightarrow \bigotimes$



- B. As in 3.2, select a web bolt layout, this time set equal to 1.5 its flexibility index and press OK; do the same for other web bolt layout. *NB: if you have problems on your computer in typing* '1.5', *initially type* '15', *then place a dot between 1 and 5*
- C. As in 3.3, re-execute checks, select m2 and display exchanged forces in combination 31
- D. As in 3.3, compute $\%F_f$

You will see the following exchanged actions.



 $F_w = 349012N$

 $F_f = 2 \times 284213N = 568426N$

$$\%F_f = \frac{F_f}{F_f + F_w} \times 100 = \frac{568426N}{568426N + 349012N} \times 100 = \frac{504576N}{917438N} \times 100 = 62 \qquad (f=1,5)$$

Write down $\%F_f$ and its related web bolt layout flexibility index f_w :

f	1	1,5	2	2,5	3	3,5	4
%F _f (f _w)	55	62					

Repeat steps A, B, C and D, but this time set flexibility indices equal to 2, then execute checks and write down computed $\%F_{f}$.



Repeat steps A-D using flexibility indices equal to 2.5, 3, 3.5 and 4, computing $%F_f$ everytime. You will get:

f _w	1	1,5	2	2,5	3	3,5	4
%Ff	55	62	71	79	85	89	92

3.5 CONCLUSIONS

Plot $%F_{f}(f_{w})$ versus f_{w} in a graph:



The "ideal value" of $\% F_f = 80\%$ is reached with web bolt layouts flexibility indices between 2,5 and 3.

Flexibility index can be use by the designer to take into account web plasticization, with consequent forces re-distribution due to resistance reduction of a part of the member.



1	INTRODUCTION	
<u>2</u>	SPLICE JOINT (BOLTED CONNECTION)	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 RECTANGULAR PLATES	17
2.5.	2 ADDITION OF BOLTS	
2.5.	3 OVERLAPS AND COHERENCE CONTROLS	41
2.6	STEP 6: SETTING CHECK SETTINGS	
2.7	STEP 7: EXECUTING CHECKS	
2.8	STEP 8: EXAMINING RESULTS	
<u>3</u>	APPENDIX: FLEXIBILITY INDEX	
3.1	WHAT FLEXIBILITY INDEX IS?	
3.2	MODIFY WEB BOLT LAYOUTS	
3.3	CHECK AND RESULTS	
3.4	FLEXIBILITY INDEX INCREASING	
3.5	Conclusions	