

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. This tutorial is aimed at teaching base commands relative to renode construction. The start point is the end of tutorial 003a (part 1 of tutorial 3).

By following this tutorial you will be able to:

- Build up a complex renode adding plates, angles, bolts and welds
- Apply cuts, bevels, shorten and lengthen members
- Understand computing options of bolt layouts and of members and components
- Understand how to use the left side panes
- Understand how to add new variables
- Understand how to add user-defined checks
- Export a DXF to be processed by other CAD programs

Running the checks and discussing results will be done in another tutorial.

This tutorial is some like 67 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

This is the second part of a set of 4 parts. In this part we will join the Y beams to the column.

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 HOW TO BUILD A MULTIPLE MEMBERS RENODE



Initial windows content: renode view at the end of tutorial 003a.

2.1 STEP 4: JOINING THE Y BEAMS TO THE COLUMN

2.1.1 Shorten the members

Select the +Y beam. It gets blue. Then apply the **Renode-Members-Trim-extend** command. Input -20 to shorten the member by 20mm. Then un select the +Y beam and select the -Y beam, repeating the same operation. You get here.





2.1.2 Add end plate

Now select +Y beam only and execute the command **Renode-Components-Add through** (**button** in the left bar). Choose **Rectangular Plate**:



You are now in this dialog, choose the parameters as shown:



×
360 🕂 Height (DY)
200 📩 Length (DX)
20 Thickness (DZ)
P1 Name
S 235 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





Press once the right arrow in **Rx**, and then press **Ins. Point** and choose like this the insertion point:



Component	
$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	

Now press OK and choose the +Y beam end-face center, like this:



If you have problems in understanding which point to click in the scene, you can:

1) Choose a simpler view to get the centre (maybe a lower viewpoint may help);

2) Execute the **Display-Scene points** command and remove the tick from mid side points, and/or to actually see all significant scene points.

3) Zoom In using the mouse wheel.



4) Eventually extract only the +Y beam and then execute the addition command, so that only its points are in the sub-scene.

For instance this is how it would appear the scene with this choices for the scene points

Scene points display mode and creatio	n 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
ОК	Cancel

and a lower viewpoint:



Notice that black points are segment extremities, while blue points are face centres. The face centre is now pointed by the row and is clearly visible. With also the mid segment points we would get the following:





And here you notice that mid side points are green. So we must look for a blue point with two green points like a sandwich, as pointed by the arrow.

Once you've correctly clicked the face centre, the plate is added, like this (no scene points are shown):





2.1.3 Join the beam to the end plate

Now we will add a fillet weld layout to the beam, in order to join the plate to the beam.

Before executing the command we have open the left pane **Selected Components** tree to see HEB200 sizes for the selected member.

Execute the **Renode-Components-Add weld layout** command (1+ button in the left bar). You can now select a face to be welded. Choose the end face of the +Y beam, like this:

Click left. You get into the weld layout addition dialog:



Do NOT tick "penetration welding layout" in order to use fillet welds.

To initialise, choose a 10mm thickness for the fillet and press the **Apply to all sides** button, leaving the 50 and 5 mm defaults: they decide which sides are too short to receive a fillet weld, and if the side must be entirely covered or not; here we leave 5mm blank at extremities. You now get here:



Veld layout input	
General data Penetration welding layout Shop 10 Weld thickness Initialization Apply to all sides having length higher than 50 Modify single welds 1 • 4 Weld 7 Modify single welds 1 • 10 • 10 • 10 • 10 • 10 • 10 • • Length 10 • • Position 90 Angle between active faces 7.1 Throat Face sides - 1 • • Side Add weld • - -	
6349.8188950552 Area (* Use Jp 43720873.577678 Ju (* Use Jp 16083974.435429 Jv Jp: polar inertia 59804848.013108 Jp Jr: sum (1/3)Lt^3	ZOOM Fill PANX 1: 1.25 Print Copy

Now we will decrease to 6mm the thickness of the web fillets. To do that, use the arrow keys of the Weld control, until one of the web fillets gets green (selected). Notice that each fillet shows the thickness and the - smaller - throat thickness. Once the fillet is selected change its thickness to 6. Like this:



Weld layout input	×
General data Penetration welding layout 10 Weld thickness Penetration Apply to all sides having length higher than 50 Modify single welds 2 - Weld Remove! 124 Length 6 - • Position 90 Angle between active faces 4.2 Throat	
5999.0939315866 Area Image: Constraint of the system 43271477.991154 Ju Image: Constraint of the system 16048308.221726 Jv Jp: polar inertia 59319786.212880 Jp Jr: sum (1/3)Lt^3 OK Cancel	ZOOM 1: 1.25 Fill PANX Fill PANX Font size

Now select the other fillet and do the same, like this:



Notice that the moment of inertia of the weld layout (Ju, Jv, Jp, where u and v are the principal axes) are printed in the read only edit boxes at the bottom left of the dialog. Notice that the program Castalia s.r.l. - all rights reserved - 12



understood the angle between the faces in contact (**angle between active faces**). Also notice that the program, for each selected fillet weld, prints the **Throat** value. Also notice that you can remove or add single welds, by selecting a side with the **Side** arrow keys, and then using the **Add weld** (to the current side) and **Remove!** (the selected fillet) buttons. So you can describe the weld layout as you want. There are no limitations. The program will accept any set of welds (even *strange*, i.e. not symmetric or completely special, for instance segmented welds). They all will belong to a weld layout. Here we leave the most common choice.

Press OK and you will see the weld added, joining the beam and the plate:



2.1.4 Join the end plate to the column

Now unselect all (\times), and select the plate added, the weld and the +Y beam. This will help us to decide bolt position as a sub-scene will be created using also the selected components. Then execute the **Renode-Components-Add bolt layout** command (\$+ button in the left bar). Choose the outer face like this:





Click left. You will get this message:



don't worry: the bolt layout has been initialized with a one bolt bolt-layout which is ill positioned. If you are still working in the same session of the previous tutorial part, you will see a different bolt layout initialisation, basically with the same data of the already defined B1, B2, B3 and B4. You get here:

Press the Change button and choose M20-8.8 between EURO-HEX bolts, like this:

C.S.E.					
t choice					
Bolt C EUROHEX C INDIANHEX C AMERICANHEX C EUROHEXHSFG C INDIANHEXHSFG C AMERICANHEXHS 8.8 Class M20 Diameter	M20-8.8F				
Resistance area					
Hole	HEX: hexagon; HSFG high strength friction grip; HS high strength				
C Precision	OK Cancel				

Press OK, you are back in the bolt layout addition dialog. Place a tick in the **Do not print messages** check box to avoid possible messages breaking down your actions.

Select 5 rows and 2 columns. Specify 55 mm as Row distance and 120mm as column distances. You are here:

Bolt layout input		. 🗆 🗙
Bolts		
M20-8.8F © Regular © Circular	120.0	
C Staggered C Free		
Change Empty inside		
Regular, staggered or circular arrangement		
Quantity Distances		
5 Rows 55 Rows		
2 Columns 120 Columns		
Free arrangement		
Add Sel/Unsel All + + ×	<u> </u>	
Remove + Bolt None! + Y		
Center offset Base information	<u> </u>	
Dx 35 Net length		
0 June 15 Minimum thickness		
0 Angle ! 1 Multiplicity		
✓ Shear only bolts I Flexibility index		
Compressed bolts	6	
Slip resistant	ö	
Is an anchor Anchor data		
Use bearing surface Bearing data		
Use bolt net-area for bearing calculation		
✓ Add inertia of bolts in bearing calculation	Face >bearing	
Check block tear Block Tear	ZOOM PANX	
- Limit values of elementary actions	1: 1.1111111	
1411200 N N	Print Copy Cill Hay	vagons
1206371.57 N V Plastic	Font size	
69518928.98 N mm Mz	V Do not print messages	
OK Cancel Set		



The web bolts are ok but the first and last column will have to be shifted. Let's switch to a free Arrangement, by clicking **Free**. The dialog changes like this:



Now the red circle bolt hole is the current bolt. Press the Sel/Unsel button, it gets yellow, like this:





Now use the Bolt arrow keys to get red the right bolt in the same row, and then press Sel/Unsel, you get here:





Now using the Left arrow in the Y control, shift the two selected nodes downward, until the original 70mm distance from the lower border of the plate gets equal to 40mm. Like this:



Bolt layout input	
Bolts	
M20-8.8F C Regular C Circular	<u> </u>
C Staggered (* Free Change	
Begular_stangered or circular arrangement	
Quantity	
5 Rows 55 Rows	
2 Columns 120 Columns	
Free arrangement	
Add Sel/Unsel All!	
Remove Bolt None! Y	
Center offset Base information	
0 Dx 35 Net length	
0 A Dy 15 Minimum thickness -	
Angle ! 1 Multiplicity	
✓ Shear only bolts I Flexibility index	
Compressed bolts	
Slip resistant	
🗆 Is an anchor 🛛 👘 🕹 👘 👘 👘	
Use bearing surface Bearing data	
Use bolt net-area for bearing calculation	
Add inertia of bolts in bearing calculation	Face >bearing
Check block tear Block Tear	Z00M PANX
Limit values of elementary actions	
1411200 N N	Entration Print Conu Fill Hevagons
1206371.57 N V Plastic	TOR Size The size
91320315.24 N mm Mz	✓ Do not print messages
OK Cancel Set	

Now press the Sel/Unsel button and unselect the right bolt in the last row. Go back to the left bolt in the same row and unselect it. Then select the two bolts of the topmost row, like this:

Bolt layout input	
Bolts Arrangement	· · · · · · · · · · · · · · · · · · ·
M20-8.8F C Regular C Circular	120.0
C Staggered @ Free	
Change Empty inside	
Regular, staggered or circular arrangement	
Quantity Distances	
5 Rows 55 Rows	
2 Columns	
Free arrangement	
Add Sel/Unsel All!	
Remove Bolt Nonel Y	
Center offset Base information	
0 Dx 35 Net length	
0 A Dy 15 Minimum thickness	
Angle I 1 Multiplicity	
Shear only bolts 1 Flexibility index	
Compressed bolts	
Slip resistant Friction data	
🗆 Is an anchor 🛛 👘 👘 👘 👘 👘	
Use bearing surface Bearing data	
Use bolt net-area for bearing calculation	
Add inertia of bolts in bearing calculation	Face _>bearing
Check block tear Block Tear	200M PANX 1: 1.1111111
Limit values of elementary actions	
1411200 N N	Font size Print Copy Fill Hexagons
1206371.57 N V Plastic	
91320315.24 N mm Mz	✓ Do not print messages
OK Cancel Set	

Castalia s.r.l. - all rights reserved - 19



Now using the right arrow Y shift the two selected bolts upward, until 70mm gets 40, like this:



We have a doubt about hexagon interference with the welds. Press the **Hexagons** button and you see what follows:



The hexagons do not touch the fillets. It's ok. This image was pasted after having pressed the **Copy** button in the dialog. You can do as well for your docs.



So it's now clear that you can *study* (because you see all interferences) any possible bolt disposition. Now we must decide what kind of bolt layout this will be. As the beam is bent we just cannot use shear only bolts: it would be meaningless. Clearly the bolts will react with a traction / compression to the bending. Now, we must only decide if these bolts will work using a bearing surface or not. If yes, the plate will find a bearing in the column flange. We can assume this is true but we will have to stiffen the column. So:

- remove the tick from Shear only bolts;
- place a tick in the "Use bearing surface" check box;
- press the "bearing data" button.

You	get	here:
-----	-----	-------

Bearing compo	onent mat	erial data						6
Unlimited elastic law with limit compressive stress C Unlimited elastic analysis - No tension					Es/m	Legenda y: yield; u: ultimate; Es: bolt elastic modulus m: homogeneization factor sigma,max: maximum compression stress E: modulus of elasticity gammaM: material partial safety factor		
Nonlinear no-te	ension constil r analysis - Ni	tutive law o tension						
210000	-235	-0.005595	0	0	0	1.05	eu F+	1
E	sy	eu				gammaM	sy	Elastic perfectly plastic
s1	e1	eu				gammaM	eu e1, s1	C Parabola-rectangle
eu	su	e2	s2	e3	\$3	gammaM	eu e3 e2 s3 s2 su	C Trilinear
				[ОК)	Cancel		

Here we must decide how the bearing surface will react. Clearly this is complex. however, if the stiffeners are placed in correct position we can assume that the plate finds a sufficiently stiff bearing. So we will leave an elastic, no-tension constitutive law. The m value is the reduction in the elasticity modulus (from that of the pulled bolts). We can assume a high value, meaning that the bearing will not be very stiff. Place a 100 for m (E bearing = 1/100 E bolts, 206,000N/mm²/100=2060N/mm²). Now we must decide a limit stress for the bearing. Which is the weakest part in this chain? Clearly it is the column flange, which can bend as plate loaded normally to its plane. We can establish a limit value here for the stress, *i.e. a limit value for the normal pressure to be applied over the flange*. If we compute this number we can be sure that, if the bearing object is checked, this means that the normal pressure has not become higher than a limit value, and so the bending in the flange was not dangerous.

Assuming a cantilever simplified scheme, for the column flange, we get for a 1mm width:

 $M = pBL^2/2$

Castalia s.r.l. - all rights reserved - 21



and so

 $p < 235 t^2/(3L^2)$

where:

t is the flange thickness = 15mm

L is the span = $0.5xB - R - 0.5t_w = 100-18-4.5 = 77.5mm$

(R fillet radius for HEB200, tw web thickness)

so

p < 235 15²/(3 77.5²)=2.93 N/mm²

This is a simplified computation, but gives order of magnitude. Input 3 (N/mm²) as limit stress for the bearing object (**Sigma max**).

Now we still have to define the surface where the bearing will react, and which object is the bearing. To do that press the -> bearing button. You get here:

Bearing surface polygon	
Bearing surface polygon Bearing surface definition bearing surface U bearing surface U bearing surface Sub current face bearing surface Sub current face U: union; Int: intersection; Sub subtraction Object to be checked as bearing = current face owner Definition of c 20 Span c 20 Span c 20 plate thickness 20 plate thickness	t by c ad by c
235 fy (yield stress) 3 fjd (bearing surface design stress)	
Bearing data 0 Area 0 Compressive force N	200M PANX 1: 1.1764705(Font size red: clicked face; green: current face; yellow: bearing object chosen face Print Copy Fill Hexagons

First of all using the **Face** arrow keys select (gets green) one face of the column, and then press the **= current face owner** button, in the **Object to be checked as bearing** box.

Then go back and select the plate face, and notice that the column face previously chosen is yellow, meaning that the bearing object has been chosen.



Bearing surface polyg	gon				X
	ition				
bearing surface	= current face			120.0	
bearing surface	= current face border	d by c	←	120.0	
bearing surface	U current face borden	red by c	4		
bearing surface	Int current face		· · · · · · · · · · · · · · · · · · ·		t l
bearing surface	Sub current face				
U: union; Int: int	ersection; Sub subtraction		L	ᠵᠠ᠘/ᡆ᠋	⊐ ^ĕ
		- [-	\odot	ldhl ⊙	
Object to be checked	as bearing				22.0
	re owner		\odot		Lt I
	Le Owner				222
Definition of c			•	<u> </u> ()	1
				⊐/ \ 	⊐₽
20 Span o	c Pierre	- -			⊐8
		PANY	<u>−</u> <u></u>		
Compute c usir	ng data below!		U .0		
20 plate t	thickness				
225 6.6.1	ld store and				
233 Fy (yie	ad stress)	<u> </u>	- 2	E Face	<u>'</u>
3 fjd (be	aring surface design stress)	<u> </u>	1		
- Pearing data			ZOOM	E	PANX
		1:	1.1764705	Font size	
Aica		red: click	ed face; green: current	t face; yellow: beari	ng object chosen face
0 Compre	essive force N				
	Cancel	Print	Conv	Fill	Hexagons
					ionagono

Now press the "bearing surface = current face" button, and you see that the bearing surface gets gray:



So we have defined all. Press OK, go back to the main bolt layout addition dialog, and press OK there. You may get this message:



Warning! If the bearing surface has points outside the bolted faces, these points press in the void and the model is not coherent as resistance is over estimated. Besides the bearing surface area outside the faces in contact will generate forces outside the fem model.

OK

Here this message is just a warning: no point of the bearing surface is really out of the surfaces in contact. Next message warns us that there are points of the bearing surface outside the faces in contact (in fact some of the faces in contact, those of the column, have points outside the bearing surface, and vice versa, in this case this is not a problem).

CSE	
2	At least one point of bearing polygon has been found which is outside the faces in contact. Continue anyhow?
	<u> </u>

press yes (sì) and go forward.

CSE

!

You see now the bolts applied:



Notice that since these are not "shear only" bolt layouts (the bolts carry axial force), a dot is drawn at the hexagon center.



2.1.5 Copy the +Y plate, weld and bolt layout to the -Y beam

Now we would like to copy all added components (the plate, the weld and the bolt layout) to the other (-Y) beam. Select the objects you wish to copy:



Execute the **Renode-Components-Copy** command (**S** button in the left bar). Press the second button from the right.



Now select a face of the column:





left click

Angle input		×
	180 Angle in degrees	
[Cancel	

leave 180 degrees and press **OK**. You hear the "clack" meaning the copy waas done without overlaps.

Now you see that the -Y beam is joined to the column:



Castalia s.r.l. - all rights reserved - 26



2.1.6 Adding the stiffeners

Now we must add the stiffeners to the column, to avoid an excessive bending of the column flanges. This will also show once more how CSE can be used as a tool to study connection, that is a tool to understand how things must be done.

The first thing is to understand how much room there is to place these stiffeners. We will first add a stiffener and place its welds. Then the stiffener and the welds will be shifted and copied to be put in place correctly.

We will initially work in the +Y part of the column and then copy using rotation around the column axis, as already done.

Set a +X view (**Draw-Std Views** - \mathbb{H} in the main bar -, and choose +X), then zoom in using the mouse wheel. Then execute the command Enquire-Geometry (\mathbb{P} in the right bar) and look for the free space to put a first stiffener, like this



The "1" is over the first point you have clicked. The mouse is over the second point. You see that there is about 23mm free. So a stiffener 10mm thick with a weld 6mm thick at the top and a weld 6 mm thick at the bottom is 10+6+6=22mm: it can be placed. Click right to exit from **Enquire-Geometry** command. Now choose an Isometric view (once more \mathbb{A}), unselect all (\times) and then



select the column. we are going to add a stiffener and by selecting the column its sizes will be correctly initialised. You see now what follows:



Execute the command **Renode-Components-Add through** (**b** in the left bar):



Choose "rectangular plate with bevels". You are now here:



		Type						
170	h	h <mark>≠t</mark> ia	h ≠	x] _				
95.5	ь							
10	t							
18	a	S 235	Material	Choose				
P3	Name							
Fem mo	delling Create FEM mode	el						
10	Border	rs and welds element size	(
30	Generi	c elements size (if 0 then	free size)					
29	Triang	le minimum angle in degre	es (default 29°)					
0.1	Node	distance tolerance (if dist	< tol then the node	es are				
	✓ It is a stiffener ✓ Search and mesh stiffeners when preparing fem model							

Leave all sizes as they are already correct (due to the column selection you've done before the command). The 10mm thickness default is ok for us. Press **OK**. You are here:



Click twice the right arrow in the Rz row, press the Ins Point button and choose the following point:





Now press OK. Zoom over the top of the column using the muse wheel. Click the correct point at mid of column web, like this:



You will immediately see what follows:



SE - [TUTORIAL3.CSF]	BX
⁵ File Modify Digglay Draw Enquire Fem 3Nodes Rende Checks 30 Model 3Classes Sonis Window ?	- 8 ×
L M C L L C M C L L C M C L L C M C M C	
To get hfo, press F1 mm N s *	c
🐮 start 🔰 Google 👔 🖉 🗈 🚬 🐂 ChickEde Mpeloog 🖉 ose_lutoral_0005_cm 🙀 CSE - [11/1014/4.5.CSE] 🔮 Immegne - Part 🛲 🕮 🖏 🖒 🗑 🔊 🗞 着	11.20

Choice of the v	way a copy i	is defined					X
Translation –					Rotation		
*	÷ t	123 * +	* *	123 * +	B ∳∎	Banda and a state of the state	
2 points	2 faces	face+incr.	face point	Numeric	2 points+angle	Member + angle	Numeric

Choose the last yellow button (Numeric). You get here:

Dialog		
0	DX	
0	DY	
0	DZ	
1		Cancel
		Lancel

Input -100 in the DZ cell. Press OK. You now see this:





Now we will add welds. Execute the command **Renode-Components-Add weld layout** (¹⁺ button in the left bar). Then choose a first contact face like this, clicking left:



You are now here:



General data	_			
Penetration welding layout Shop ?				
10 Weld thickness ? fillet	ion T			
Initialization				
Apply to all sides having length higher than 50				
keeping a clear distance from borders of 5				
Modify single welds				
0 Weld Remove!				
0 Thickness				
Desition				
	-			
0 Angle between active faces				
0 Throat				
Face sides				
1 Side Add weld				
Weld layout computing data Computing settings PA	NY			
0 Area (Use Jp				
0 Ju C Use Jr				-
0 Jv Jp: polar inertia	, -	1	1	
0 In Jr: sum (1/3)Lt^3	ZOOM	Fill		PANX
	1: 2			
OK Cancel	- Eont size	Print Copy		

Input 6 in the fillet weld thickness field; press Apply to all sides button. You will see this:



Press OK. You see this:





Unselect all (X) button.

Now re-execute the weld layout addition command (¹⁺ left bar) and choose the next face, like this:



Repeat the same operations in the weld layout addition dialog:6mm as weld thickness, **Apply to all sides**, **OK**. You will be here:





Unselect all and repeat the command for the third time, in the same way choosing this face:



We have added 3 weld layouts and the stiffener is completely welded to the column. Now choose a +X view and zoon near the stiffener. You will see this:





Select the stiffener and the three welds (you'll click over the stiffener and two welds), like this:



We are now ready to shift these components in place. However we have to understand the amount of the shift.

Execute the **Enquire-Geometry** command (?) button in the right bar), and click first over the lower left point of the object selected, like this



SE - (TUTORIAL3.CSE) SE Modfy Display Draw Enquire Fem D	JNodes Renode Checks 30 Model 3Classes	35mts Window ? 유명은 태남 중 중 중 관 · 편 문 명 3	5 <u>U</u>	- 5 ×
Variables and conditions Banoda Calculated associated Banoda Calculated associated Banoda Calculated associated Banoda Calculated associated Banoda Calculated associated	x y			
	1			
E C ?				
Inquire geometry	×	a		
Number 1 966 ×1 95 Number 2 978 ×2 4.49393939	Y1 95 Z1 2884 Y2 62 Z2 2884			
US 33,376924 DX -90,5 Active command: inquire geometry. ESC o tes To get info, press F1	DY 23 DZ 0			C= 1 I= 1 mm N s °C
🛃 start 🛛 Google 🛛 🖉 🗈	» 🎦 C:\CSEDEV Debug 🔄 🖄 cse	tutorial_003b_en Se - [TUTORIAL3.CSE]	🍟 Immagine - Paint	

Now while still in the command execute the command **Draw-Enclose** (button in the main bar), and then zoon near the mid column point, like this (you are still in the **Enquire-Geometry** command).



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



Notice that the distance to shift down is -784 mm to get a contact. if we shift down -783.5 we will not be in contact. Click right to exit from the command **Enquire-Geometry**. Execute the command **Renode-Shift** (More button in the left bar), you see this:

Choice of the way a copy is defined 🗧 🔰											
Translation -					Rotation						
*	÷	123 * F	*	123 * +	B∳ ∎	B	1234				
2 points	2 faces	face+incr.	face point	Numeric	2 points+angle	Member + angle	Numeric				
					-						

Choose the yellow Numeric. Enter DZ=-783.5. You now see this:



Now we will copy these components down. Using the Enquire-Geometry command you will understand that we have to copy downward of -223mm. Execute the command **Renode-Components-Copy** (button in the left bar) choose the Numeric input mode as before (but now is a copy). Enter DZ=-223. You see this:





Now we will copy once more down, to have two stiffeners near the beam flanges. Using the Enquire-Geometry command we understand that a good copy vector would be DZ=-79.5mm. Then re-execute the copy command (we are always copying the selected, blue components) and enter DZ=-79.5 in the numeric mode. You get this:





Now unselect all and select the upper stiffener with its welds, like this:



Re-execute the copy command and enter +79.5mm as DZ. You are now here:



°se CS	E - [TUTORIAL3.C	iE]											
Se File	e Modify Display I S 🕞 🕞 🚳	olel (나이다)	JNodes Renoo	de Checks 3D M → ← 🔯 🦣	odel JClasses Joints	Window	? ▶ <mark>]]}}]]}]]]]]</mark>]]]]]]]]]]]]]]]]]]]]]]]]]	🗃 🌮 V 🗖	• - :	: × 🕞 🔽	16 + + 12	€ <u>-</u> E L? K	_ = ×
	e Modify Display I	iraw Enquire Fem	JNodes Renor Image: Image of the second secon	de Checks 30 M → ← 🖾 №									
			<										×
To get in	nfo, press F1											<u>C= 1</u> I= mm	= 1 N s °C
-	start Goog	le 🛛 🖉 Ď	- ** 🛛 😂 Ci	CSEDEV\Debug	📓 cse_tutorial_	003b_en	SE - [TUTORIALS	B.CSE] 🦉 Imm	agine - Pa	int		M () () () () () () () () () () () () ()	👽 🧐 😓 🚮 🗖 11.58

Now select all the stiffeners and their welds, like this (do not forget any):



Castalia s.r.l. - all rights reserved - 41



We will now copy them to the other side of the column. Choose the **Renode-Components-Copy** command once more, but now select in this dialog:

C	Choice of the way a copy is defined 🛛 🔀											
	Translation -					Rotation						
	+	÷ t	123 * +	*	123 * +	₿₽₽₽₽	B	1234				
	2 points	2 faces	face+incr.	face point	Numeric	2 points+angle	Member + angle	Numeric				

the "member+angle" button. Then select a face of the column (if you have problems in selecting a face of the column zoom out and choose the face at the very top of the column: you can also change view and zoom out) and leave 180 in this dialog:

Angle input			
1	180 Ar	ngle in degrees	
[OK	Cancel	
<u></u>			

You have copied all components to the other side. To be sure, choose a **-X** (minus, not plus) Std view (**Draw-Std views**), and you will see the following:



Also from a +Z view you will see what follows:





In an Isometric view you will see:



Please keep in mind that you could have placed as many stiffeners you wish, where you wish: this is just a demo. Maybe that 2 stiffeners per side instead of 4 could be enough. If you want you can easily delete some of the components, just select them and delete (**Renode-Components-Delete**).



We are studying how to do the connection. So we can change and modify, as well. For instance, if we want to have more room for the stiffener we could displace the top and bottom row of the bolts, and so get more room. We did not do that because if we displace the bolts the lever acting over that plate in bending (a cantilever from the bolts to the welds joining the beam flanges to the plate) will increase. As an exercise we can decide to displace those bolts.

Unselect all and select just one bolt layout, that in the +Y side.



Then execute the command **Renode-Components-Modify** (Be button in the left bar). This command is active when just one component is selected.

You once more get into the bolt layout main dialog. Like this:





Notice that the two bolts of the top row are already selected. Using the right arrow in the Y control, decrease the 40mm distance from the top of the plate to 35. Like this:





Now unselect the two bolts as already explained (**Sel/Unsel** and **Bolt** arrows). Select the bolts of the bottom row, like this:





Using the left arrow change the 40mm distance to 35, like this:





Press OK.

Now wee have to do the same for the other bolt layout. We can select the other bolt layout and do the same, or we can maybe do this, which is faster. Unselect all and select the other bolt layout. Like this:





Now delete it. Use the 🔰 button in the left bar. The bolt layout is deleted.



Now select once more the first (and unique) bolt layout like this:

Castalia s.r.l. - all rights reserved - 49





And execute the Copy command (Lin in the left bar), **Member+angle** mode, select a column face, leaving 180 degrees as we have done more times. You get here:



Now have a look from +X view:



°se - [TUTORIAL3.	.CSE]							
Se File Modify Display	Draw Enquire Fem JNo	des Renode Checks 3DMc	idel JClasses Joints Wir	ndow ? 🌰 🆛 🖘 🏷 🔤			مامرا ما ما ما	_ = ×
CSE [TUTORIAL3] * File Modfy Deplay D D D D D U D D D D D U D D D D D D U D <td>CSEJ Draw Enquire Fem We § O J. J. C. S. P. S. P</td> <td>des Renade Checks 30 Ma 4 1 1 4 + []]]]]</td> <td></td> <td></td> <td></td> <td></td> <td>+ + 12 5 1 12 K</td> <td></td>	CSEJ Draw Enquire Fem We § O J. J. C. S. P. S. P	des Renade Checks 30 Ma 4 1 1 4 + []]]]]					+ + 12 5 1 12 K	
	t builde ESC or data lide to	dan.					c. 1 1	>
Active command select To get info, press F1	t by click. ESC or right click to	stop.						I N s ℃
🚜 start 🔰 Go	ogle 🛛 🖉 🖪 👋	C:\CSEDEV\Debug	cse_tutorial_003b	Se - [TUTORIAL3	. 🦉 Immagine - Paint	🦉 Immagine - Paint	200 1 1 1 1 1 1 1 1 1 1	J 9, 🗞 🚮 🗖 12.27

There is more room now. We can shift the stiffeners a bit.

Unselect all. Now with a box we select the stiffener, its welds, and also the other side (hidden) stiffener with its welds. Execute the command **Modify-Select-Box** (button in the main bar). Choose a box like this:





At the second click the objects are selected.





To be sure you have correctly selected the other side components, have a look at the Selected components pane, in the left part of the screen. You see that there are - as expected - 6 weld layouts and 2 stiffeners selected. Now execute the command **Renode-Shift** (\bigcirc button in the left bar). Choose the numeric blue mode and input +5mm for DZ. You see now this:





Now unselect all and repeat the operation with the lower stiffeners, shifting them -5mm in Z direction. You will see this, finally:



So we have successfully displaced the stiffeners to increase the room between them and the X beams.



Now as the lever of the bolts has somehow increased, we would add stiffeners to the plate. We add stiffeners to help the plate in bearing bending stresses. This will end the addition of the stiffeners needed to improve connection stiffness. The stiffeners we are going to add will be placed at mid width of plate, joining the Y beams to the plate itself. We will first add one stiffener with its welds and then copy this stiffener with its welds one time to the bottom, and then the two +Y stiffeners with their welds another time to the other Y beam (the -Y one).

Choose an ISO view like this:



Execute the command **Renode-Components-Add through** (**b** button in the left bar). In this dialog



				X
Frequent Simple PltHol Re	cCut RecRou RecMix TriCu	TriRou Finger		
Rect. plate [0,h]	Rect. plate with bevel [b,h,a	Tria plate (b,h)	Trapezoid plate (b.B.h)	Hex. diag. plate [alpha.ad]
Circular plate [r]	Regular polygon plate [r,n]	Generic polygon plate []	Angle	Double angle
GENERIC section tru	Canstraint block [b,n.d]			
			OK Annulla	Applica ?

choose rectangular plate with bevels. You are now here:

Stiffener				X
Parameters	Туре			
100 h	h <mark>et</mark> ia b	h <mark>≠</mark> t		
100 b				
20 t				
10 a	S 235	Material	Choose]
P11 Nar	ne			
Fem modelling - Create f 10 30 29 0.1 V It is a st	TEM model Borders and welds element size Generic elements size (if 0 then free Triangle minimum angle in degrees (Node distance tolerance (if dist < to siffener and mesh stiffeners when preparing fe	e size) default 29°) ol then the nodes em model	are	
	OK Can	cel		

Press the right image-button to choose a one bevel plate (it gets light blue). Then choose the sizes in this way (we have understood these are correct sizes by using the **Enquire-Geometry** command):



Parameters —		Туре		
80	h	h <mark>et</mark> a	► <mark>≠</mark>	
60	ь			
10	t			
15	а	S 235	Material	Choose
P11	Name			
Fem mod ▼ Ci 10 30 29 0.1 ▼ It 56	elling reate FEM mo Bord Gene Trian Node is a stiffener earch and me	odel ders and welds element siz eric elements size (if 0 ther ngle minimum angle in degr e distance tolerance (if dis .sh stiffeners when prepar	e n free size) ees (default 29°) t < tol then the node ing fem model	:s are
100000000000000000000000000000000000000				





Press one the left arrow in the Rx row, then once the left arrow in the Rz row. Press the Ins Point button and choose the insertion point. Like this:





Now press OK and choose in the scene the proper point which is at mid-width of the plate upper dise, like this:



Be careful as there are more points in the nearby. You will now see what follows:



Now execute the **Renode-Components-Addition of a weld layout** command (the *the button in the left bar)*, and choose the first face, like this:





When you click left you are now here:

Weld layout input		x
General data Penetration welding layout 10 Weld thickness Pintration Pintrati		
Modify single welds 0 0 0 0 0 0 0 0 0 0 0 Angle between active faces 0 Throat Face sides		
0 Area (° Use Jp 0 Ju (° Use Jr 0 Jv Jp: polar inertia 0 Jp Jr: sum (1/3)Lt^3	ZOOM 1: 2 Fill Fill Print	PANX

In the **Weld Thickness** edit box input 6. Then press the **Apply to all sides** button, you see what follows:



Weld layout input General data Penetration welding layout 6 Weld thickness Initialization Initialization Apply to all sides having length higher than 50 keeping a clear distance from borders of 5	
Modify single welds 1 • Weld Remove! 55 • Length 6 • Thickness 0 • Position 90 Angle between active faces 4.2 Throat Face sides 1 • Side Add weld	
Weil layout computing data Computing settings PANY 466.69047558312 Area (° Use Jp 117644.89071991 Ju C Use Jr 23667.369029702 Jv Jp: polar inertia 141312.25974961 Jp OK	ZOOM Fill PANX 1: 2 Print Copy

Press OK, you have placed the joiners between the stiffener and the plate:



Unselect all. Now re-execute the weld layout addition command and choose the other face:





Click left, input 6, then press the **Apply to all sides** button and note that no weld is applied: this is because the sides are shorter than 50mm (60-15=45). So input 40 in the "having length higher than" edit box, and re-press the Apply to all sides button: you now see the following:



Press OK, and the stiffener is joined:





As an exercise (it is not strictly necessary, let's apply a bevel to the stiffener). Unselect all (button in the main bar) and then select only the stiffener. Then execute the command **Renode-Workings** (button in the left bar), and choose **Triangular Bevel**, adding a name for this action, e.g. "bevel", and setting 15mm for the bevel size, like this:

Workings management		X
Workings present	New working Working kind C Subtraction - rectangular box prism C Subtraction - polygon prism C Face rotation C Circular bevel C Triangular bevel C Square bevel C Face translation Additional preliminar data of working IS Bevel radius working description bevel Add	Command options

Press the Add button. Now select the two points of the side to be cut, like this:





At the second click you see



We are now ready to copy. Select the welds of this stiffener, together with the stiffener itself, like this:





Now execute the **Renode-Components-Copy** command (button in the left bar), choose the **Member +angle** mode, and select a +Y beam face, like this:



Then leave 180 in this dialog

Angle input		X
	180 Angle in degrees	
[OK Cancel	







The generated objects are hidden but you can see them in the Selected Components pane, as shown in the picture above. Now add to the selection the original stiffener and its welds, like this (we have slightly changed the view):



Re-execute the **Renode-Components-Copy** command, use the same **Member+angle** option, *but now select a face of the column*. Leave 180 degrees and you have finished.

This is a +Z view:





This is a +X view:



And this a adjusted view to see in +Y area:





If we want to see exactly the opposite side, execute the command **Draw-World map** (button in the main bar), and press the space bar. This dialog appears:

Change View	
Actual View	New View
0.5646816	0.5646816
0.8064489	0.8064489
0.1781906	0.1781906
ОК	Cancel

In the "New view" column, put a minus sign before the X and Y view components, i.e. the first two edit boxes. Press OK. You now see from the opposite side:





We are now ready to join the diagonals to the column. This will be done in the third part of this tutorial.



<u>1</u>	INTRODUCTION	3
<u>2</u>	HOW TO BUILD A MULTIPLE MEMBERS RENODE	4
2.1	STEP 4: JOINING THE Y BEAMS TO THE COLUMN	4
2.1.	.1 Shorten the members	4
2.1.	.2 ADD END PLATE	5
2.1.	.3 JOIN THE BEAM TO THE END PLATE	
2.1.	.4 JOIN THE END PLATE TO THE COLUMN	
2.1.	.5 COPY THE +Y PLATE, WELD AND BOLT LAYOUT TO THE -Y BEAM	
2.1.	.6 ADDING THE STIFFENERS	