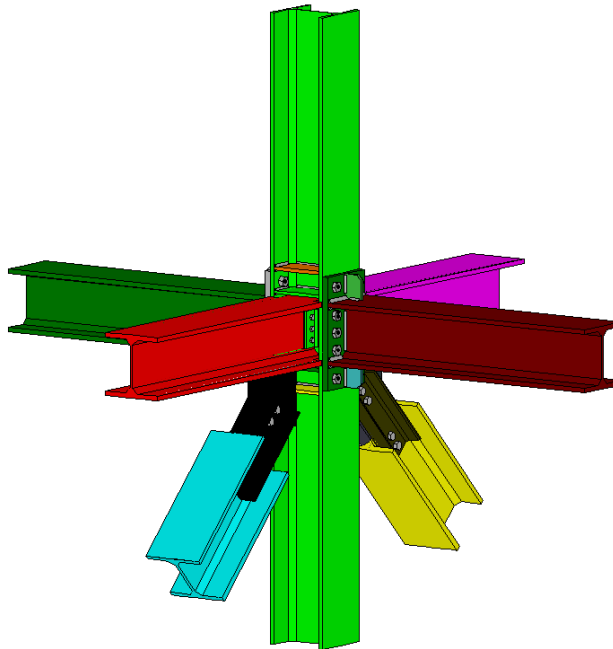


Paolo Rugarli



Connection Study Environment

Tutorial 3: building a multiple-members renode. Part 1



<http://www.castaliaweb.com> - <http://www.steelchecks.com>

Via Pinturicchio, 24

20133 Milan - Italy

staff@castaliaweb.com

Copyright © 2000-2011 – Castalia srl

Referring to CSE Version 4.40

Rev.5 November, 23, 2011

**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 002.

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

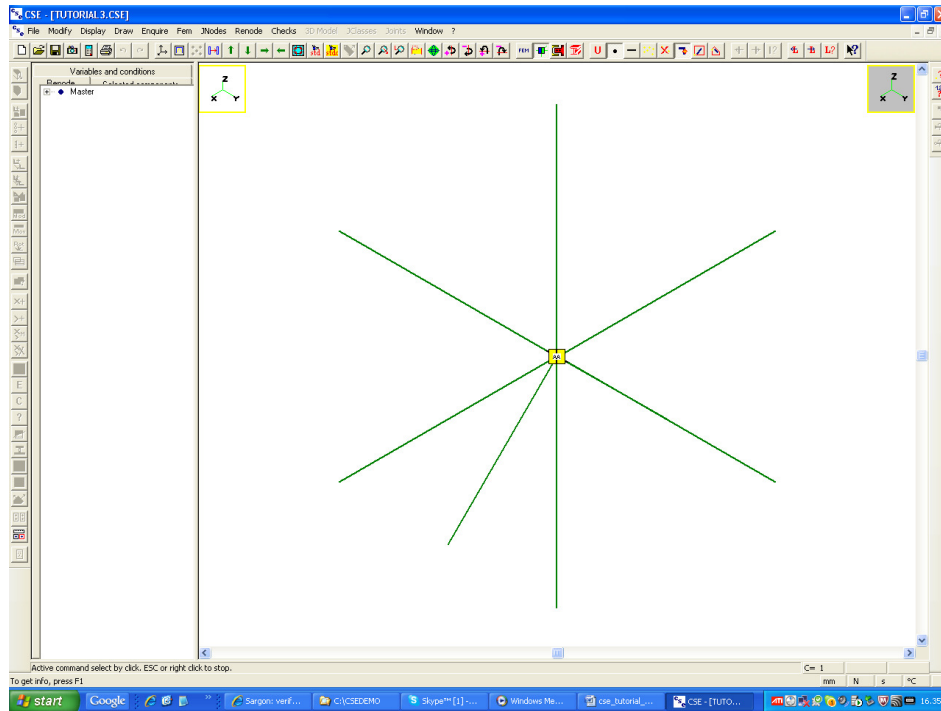
Running the checks and discussing results will be done in another tutorial.

This tutorial is some like 47 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 first part of a set of 4 parts. In this part we will join the X 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.

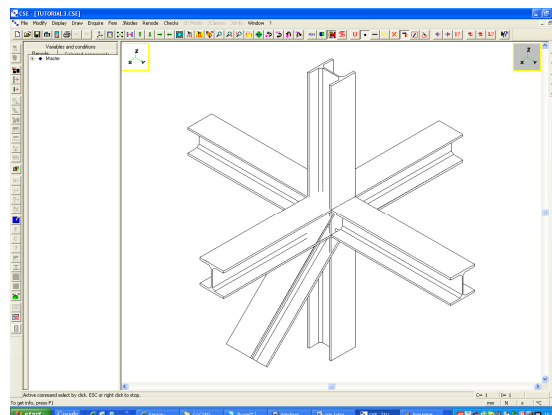
2 HOW TO BUILD A MULTIPLE MEMBERS RENODE



Initial window content: jnode view at the end of tutorial 002.

2.1 STEP 0: SELECTING A JNODE AND PASSING TO RENODE VIEW

The only jnode available is selected (yellow). So we can switch to renode view of that jnode, in order to actually build it. We can do that by executing the command **Display-Renode** ( button in the main bar), which is active if and only if one only jnode is selected. Once we have pressed the button (or executed the command from the menu) we switch to the renode view of that jnode. Nothing has been defined yet.




Notice that, by default, the member length from the ideal node outward is 1m (1000mm). So these are not the whole members, but just a segment of 1m. Among the general settings is available the piece length.

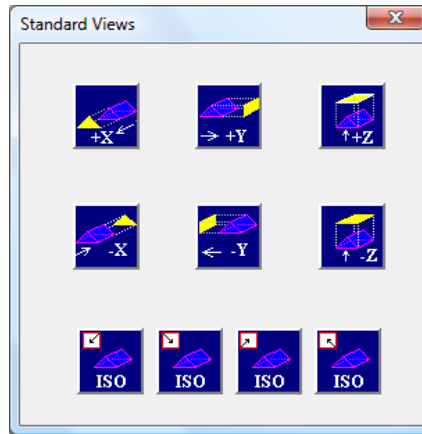
2.2 STEP 2: CHOOSE THE STANDARD

Although in this tutorial we are not going to actually check the connection, we will set the standard we are going to use as many information, during the work, will depend on the selected standard. So now that we are in the Renode view, execute the command **Checks-Set**, the following dialog appears. Choose Eurocode 3 and press **OK**. We will later be back in this dialog to understand how to use it, before the checks.

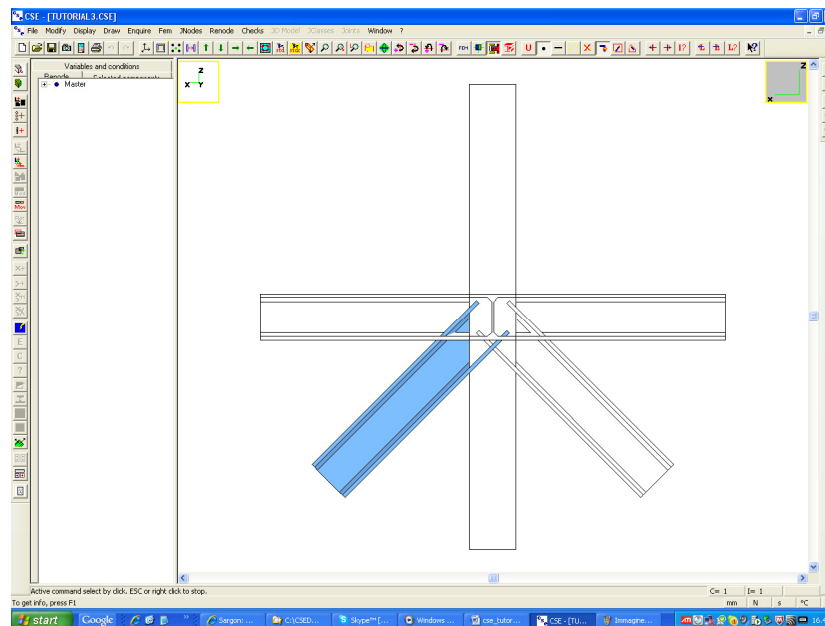
2.3 STEP 3: THE HINGED BEAMS (X BEAMS) CONNECTIONS


2.3.1 Shorten the diagonals

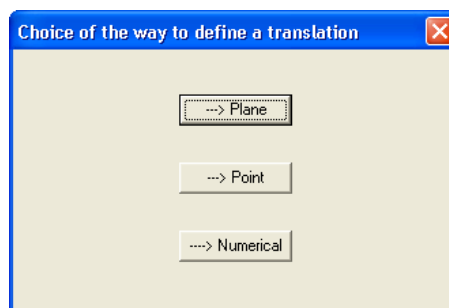
First of all let's shorten the two diagonals to avoid overlaps: we will later care about them, now we just want them not interfering with our work. So choose a +Y view by executing the command **Draw-Std view** ( button in the main bar), we get this dialog:



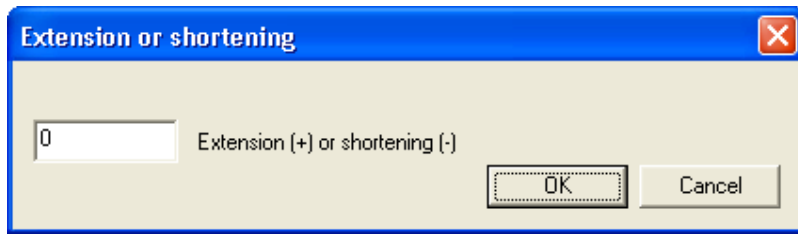
press the +Y button and you see a new view of the renode. Now click over the left diagonal so that it gets blue (selected). We are going to shorten it. As we are not interested in a special position, but just to shorten it of a bit, we will use a numerical way to assign shortening.



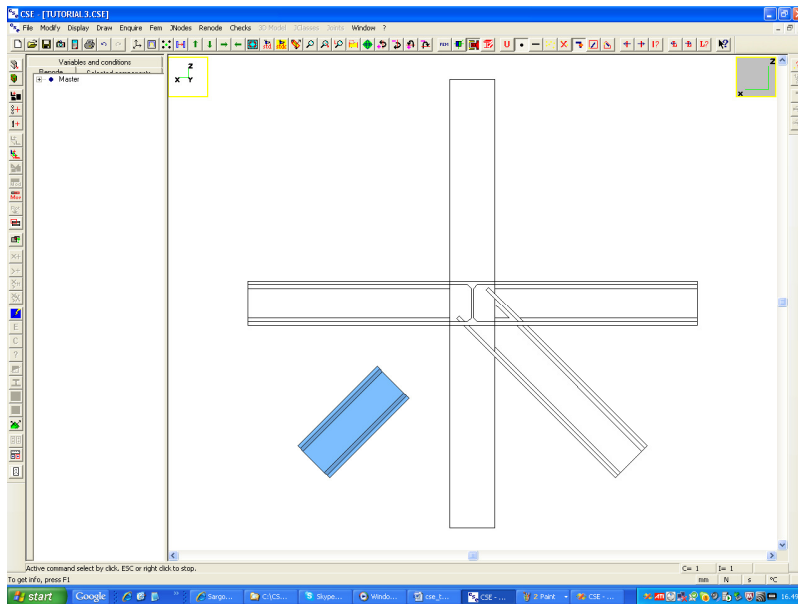
Since there is only one member selected we can apply a shortening. To do that execute the command **Renode-Members-Trim-extend** ( button in the left toolbar). The following dialog appears



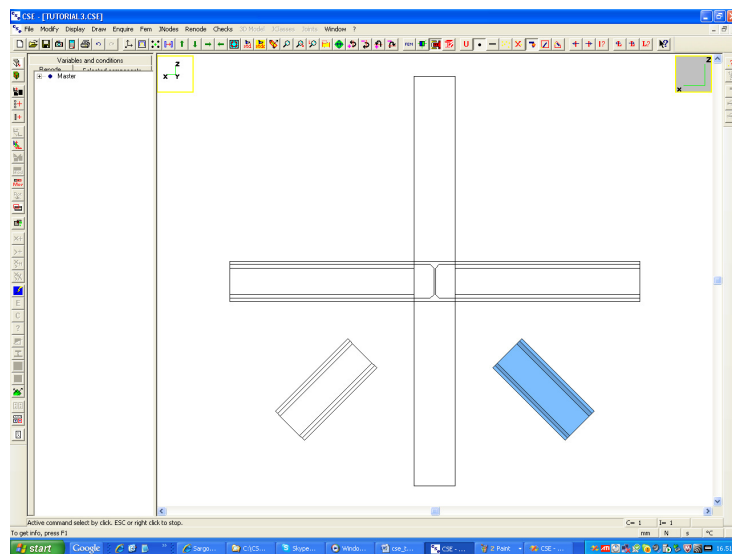
choose **Numerical** and you get here




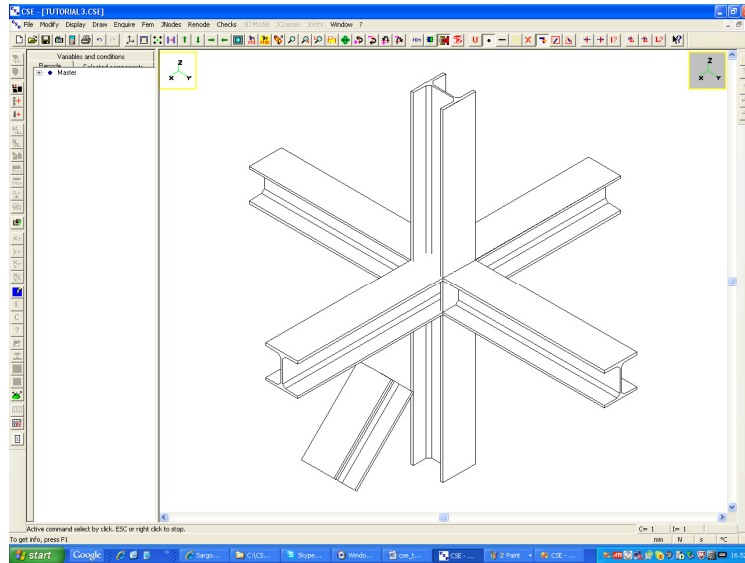
now just input -500 (mm as we are using mm which is the default unit). Press **OK** and you see what follows:



Now click over the shortened diagonal to unselect it, and then to the still un-shortened diagonal to select it. Reapply **Renode-Members-Trim extend** with the same value. You will get what follows:

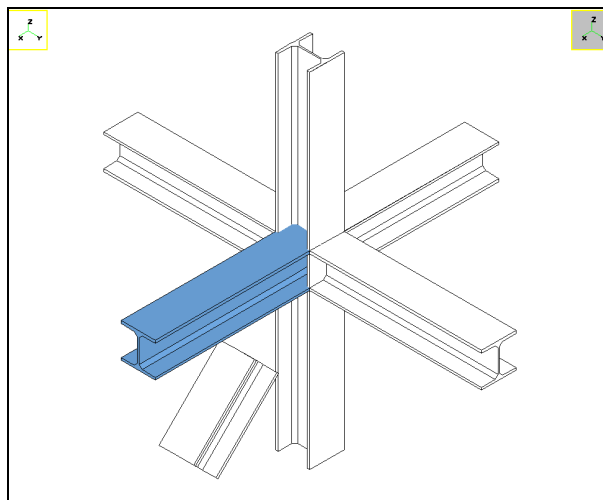


Unselect the diagonal clicking over it and switch to a Isometric view (**Display-Std view**, -  in the main bar - and then Iso):

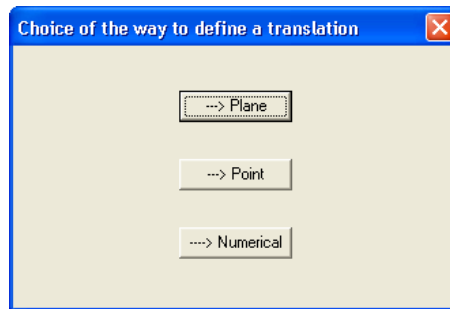



2.3.2 Shorten the X beams

We now would like to care about the two beams hinged, i.e. the beams directed like X axis and whose wireframe ending point is inside the column web (remember that the Y beams have an offset, so they end at the flanges outer planes). First of all we will shorten them so that their ending face will be coplanar to web faces (one face for each member). Basically we want to shorten them by half the web thickness. Now we have already seen how to do a shortening introducing the shortening value directly. We will use another method. Select the +X member, it gets blue.

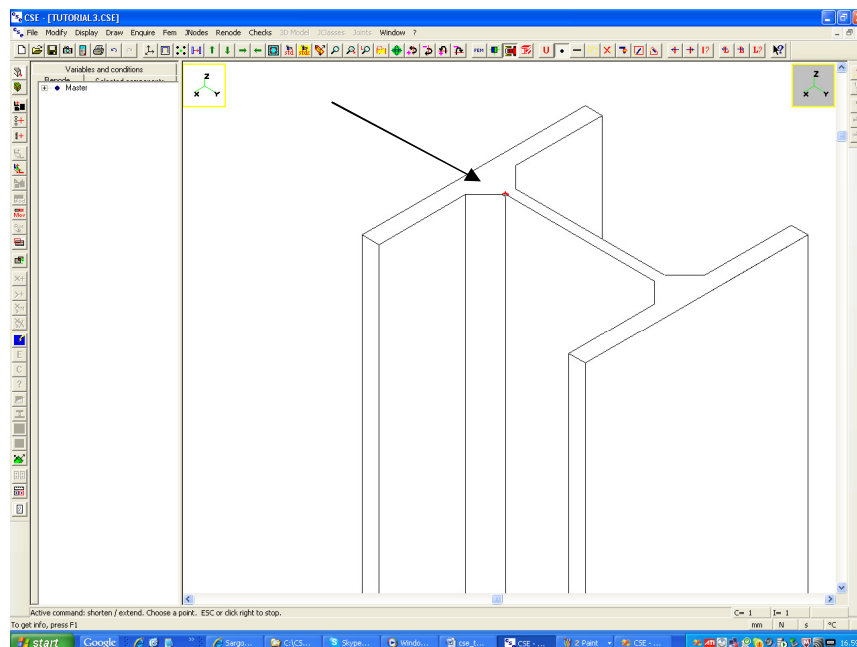



Re-execute **Renode-Members-Trim extend** but now choose "**Point**".

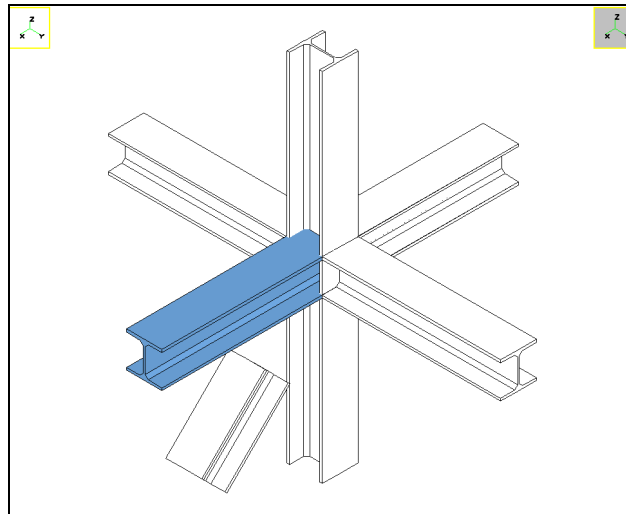


Notice that moving the mouse a red circle appears, moving as well. It is marking "significant points" present in the scene, we will later better understand what are those points. If you see two circles or none, just redraw (**Display-Redraw**,  button in the main bar)

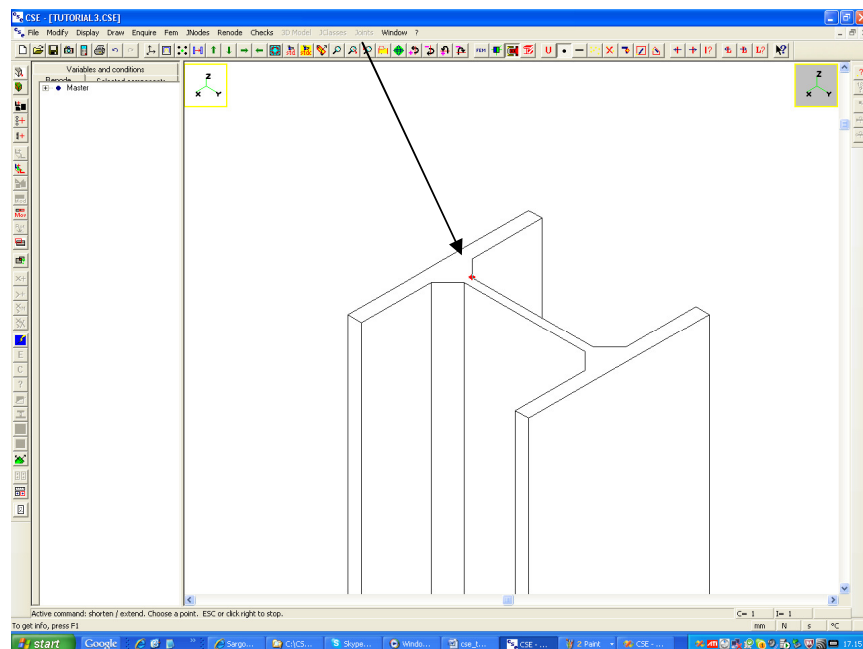
We have to choose a point in the scene so that the plane of the end of the member will pass from that point. In other words the shortening is defined by defining a point where the new end-plane has to pass. Using the mouse wheel, zoom over the top of the column and click over the web extreme point marked in the picture below. The point chosen will belong to the selected member end face plane, and so the selected member will be shortened by half the web thickness. Once you click left over the point, you have finished the command and the member is trimmed. If you have enabled sounds, you will hear a sound meaning that there are still overlaps: actually the member, although trimmed, still is in conflict with the column, as we will better see.




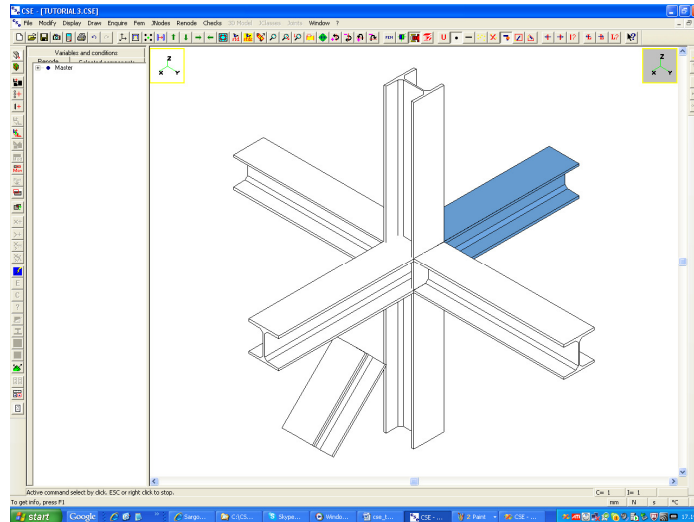
Now execute **Display-Enclose** command or better press the  button in the main bar: you see what follows:



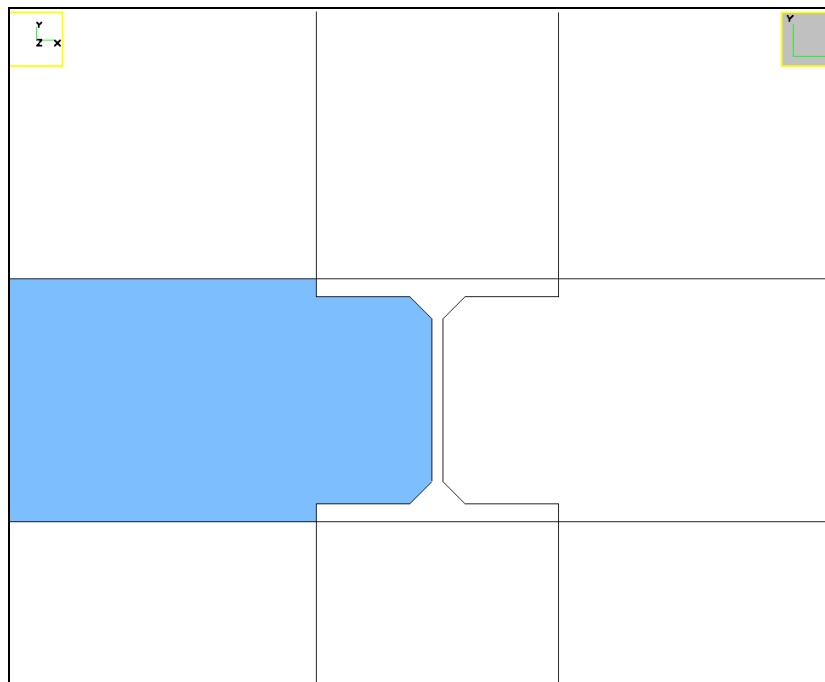
Now click over the blue member to unselect it, and click the other X member to select it. Execute once more the **Renode-Members-Trim extend** command, and choose once more "Point". Zoom over the beam top and choose the point as shown in the next picture, this will trim also this member to web plane (the other one).



Execute once more **Display-Enclose** ( in the main bar) to see the isometric view, you now see what follows:

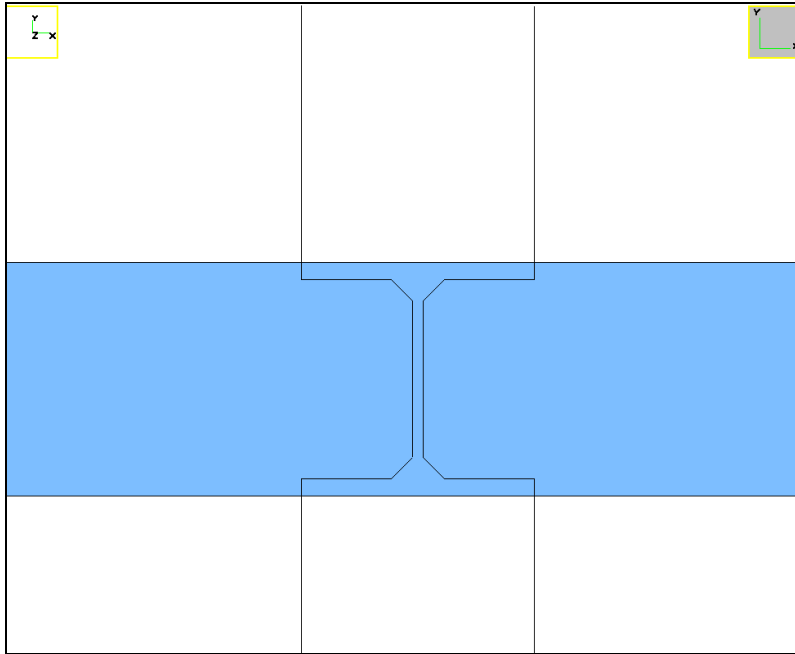



We have just trimmed the two beams to web faces. Now we have to cut them in order to avoid overlaps with the column. Choose a +Z view by executing Display-Std Views and zoom near the column, you will see what follows.

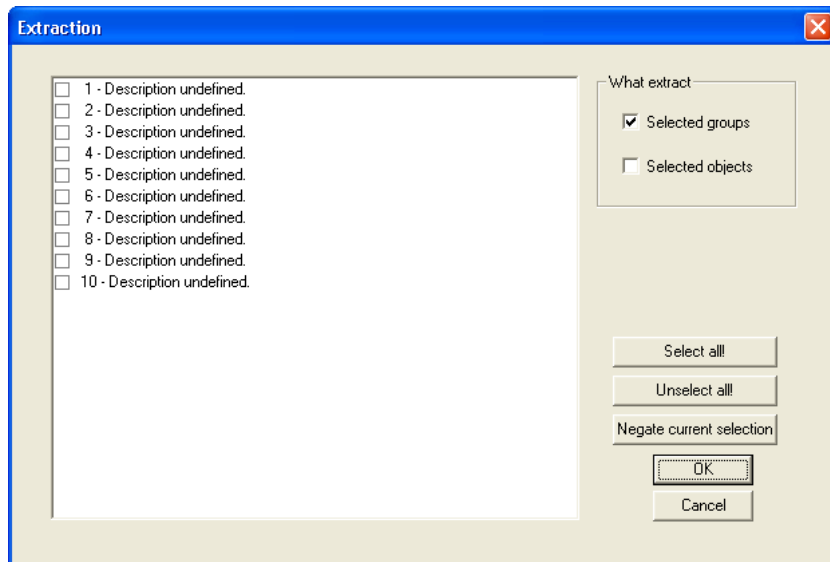


2.3.3 Solve the X beams-column overlaps

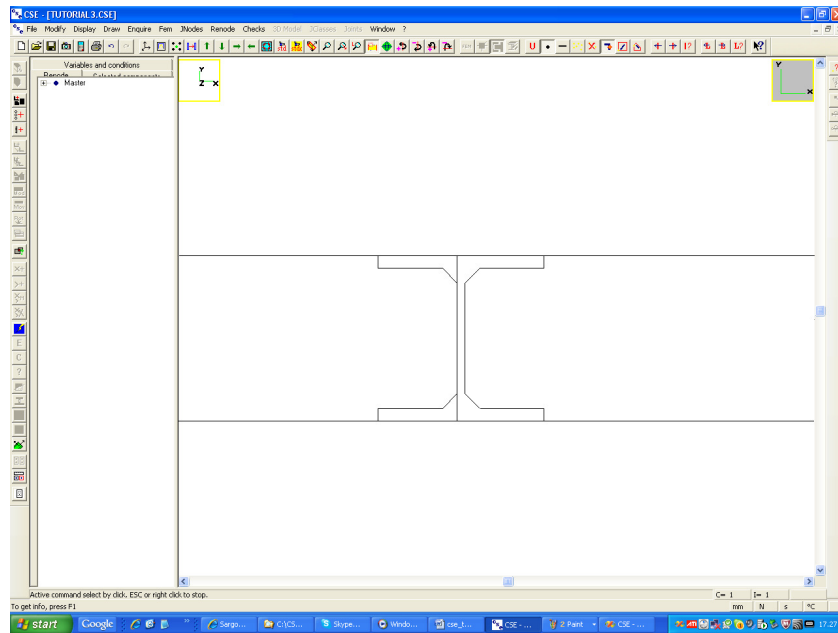
The overlaps between the X beams and the columns are clear. We will cut the selected element first, then we will cut the other. However to work better it would be fine to see only the members actually involved in the operation we are going to do. So select the column and the two X beams, like this:




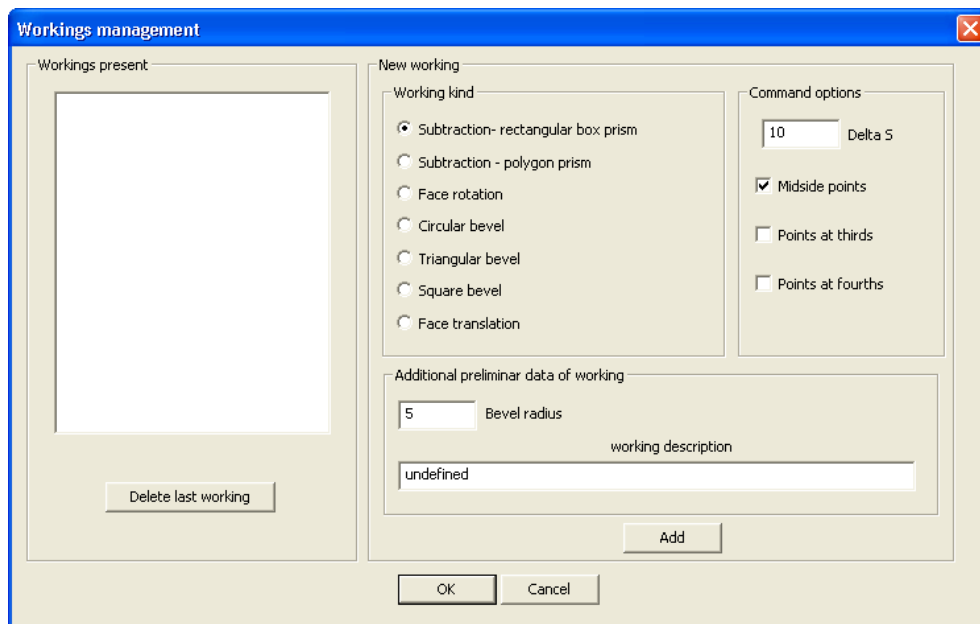
then execute the command Draw-Extract (the  button in the main bar). The following dialog appears.



Remove the tick from "**Selected groups**" and place a tick in the "**Selected objects**" check box. Then press **OK**. We are going to "extract" only the selected members. You now see what follows.

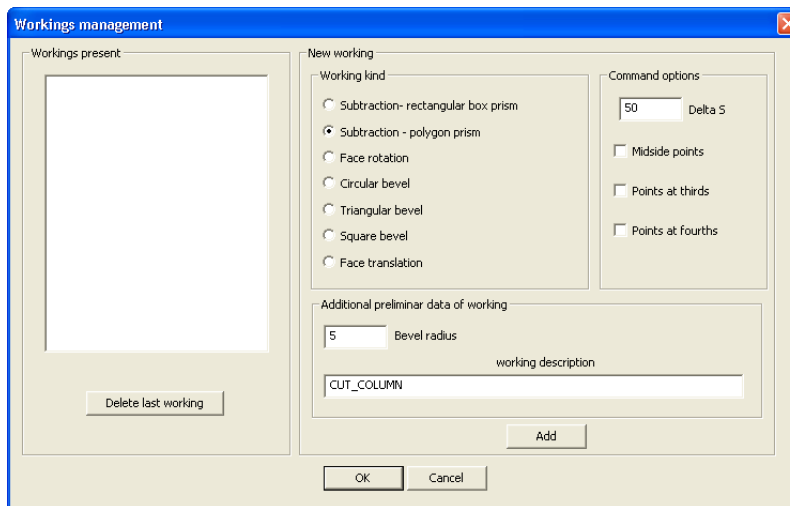


The Y beams and the diagonals (covered) have disappeared. The selection is reset. Click the left beam, it gets blue. We are going to cut a piece of this beam away. As one only component is selected the command **Renode-Workings** is available. Execute it (you can use the  button in the left toolbar). The following dialog appears. It is used to choose which action must be applied to the selected object. Choose **Subtraction- polygon prism**. Then let's give a name to the action: write CUT_COLUMN in the "working description" field. Now to define a cut polygon we will use significant point. We can choose how many they are and where, by setting what is in The Command options box.

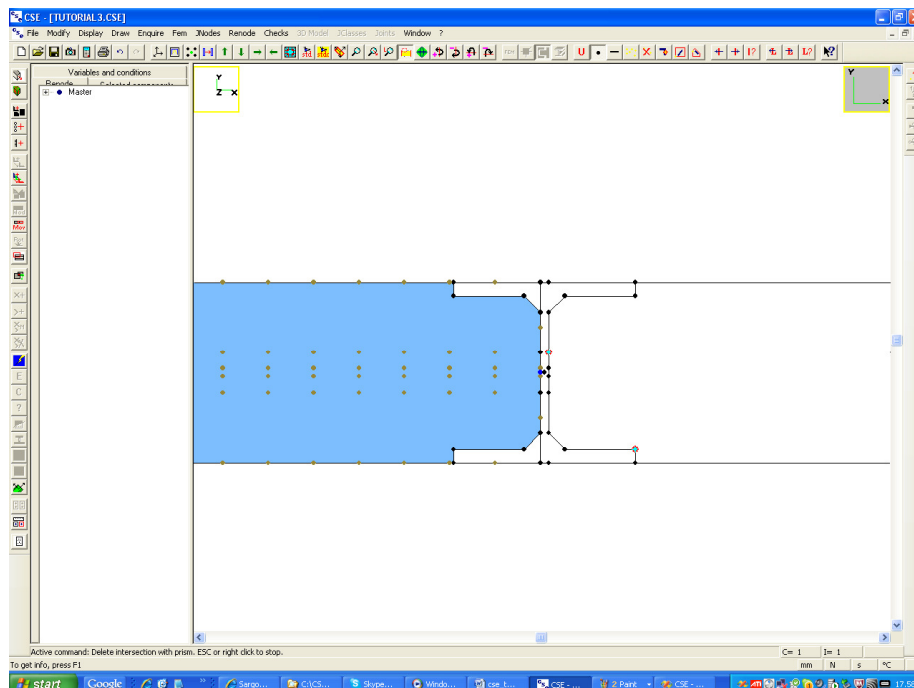


Consider the side of each face. It has some significant points which we can decide to be clickable, or not: the mid side point, points at third, at fourths, or point equally spaced by a given distance, which is here named "**Delta S**". Having more points to be clickable will help us to better define the polygon, but too many points will be boring from a graphical point of view. So choose 50mm as **Delta S**, and remove the tick from the **Midside points** check box. Finally we would not accept sharp corners so specify a **bevel radius** for the cuts we are going to apply, leave 5mm. Finally you will have the following:


press the **Add** button (*not OK*, or it will just end the command, which is also used to remove existing actions).

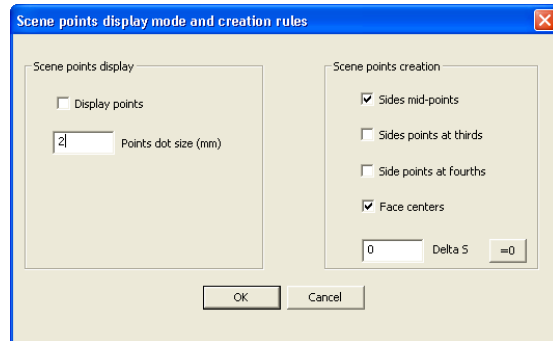


You see now something like this:



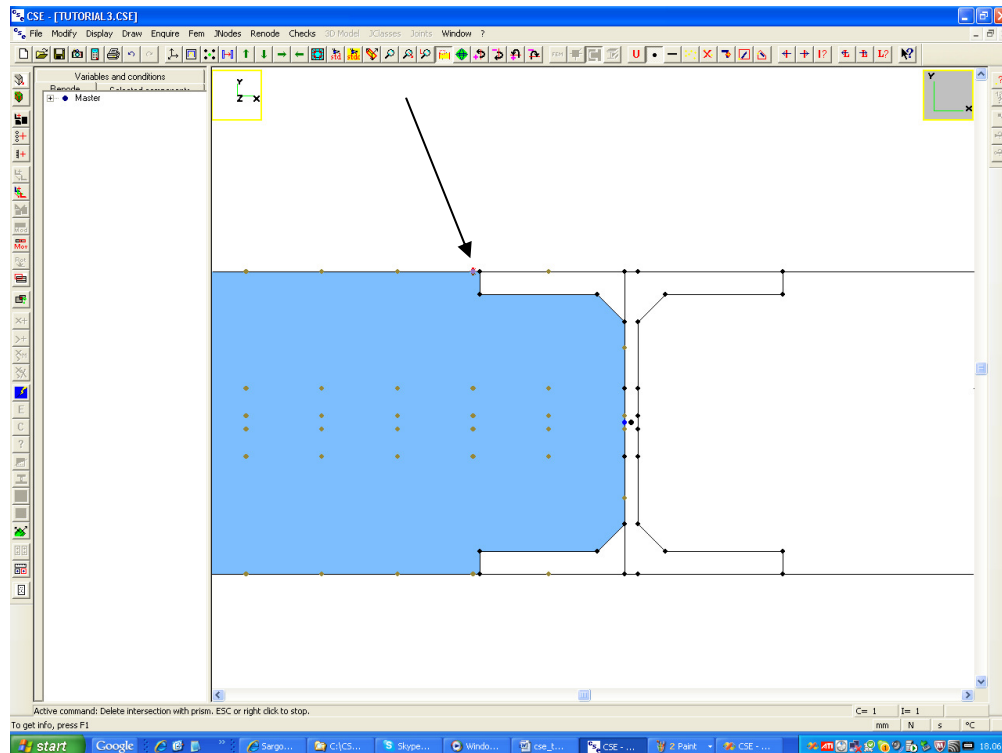
Notice that the sides of the selected member are divided in spans of 50 mm, and that the significant points are shown with a dot.

[If dots are too small, just exit from the command clicking right, press the  button in the main bar (i.e. command **Display-Scene points**) and set 2mm or more for the dot size, as explained here:

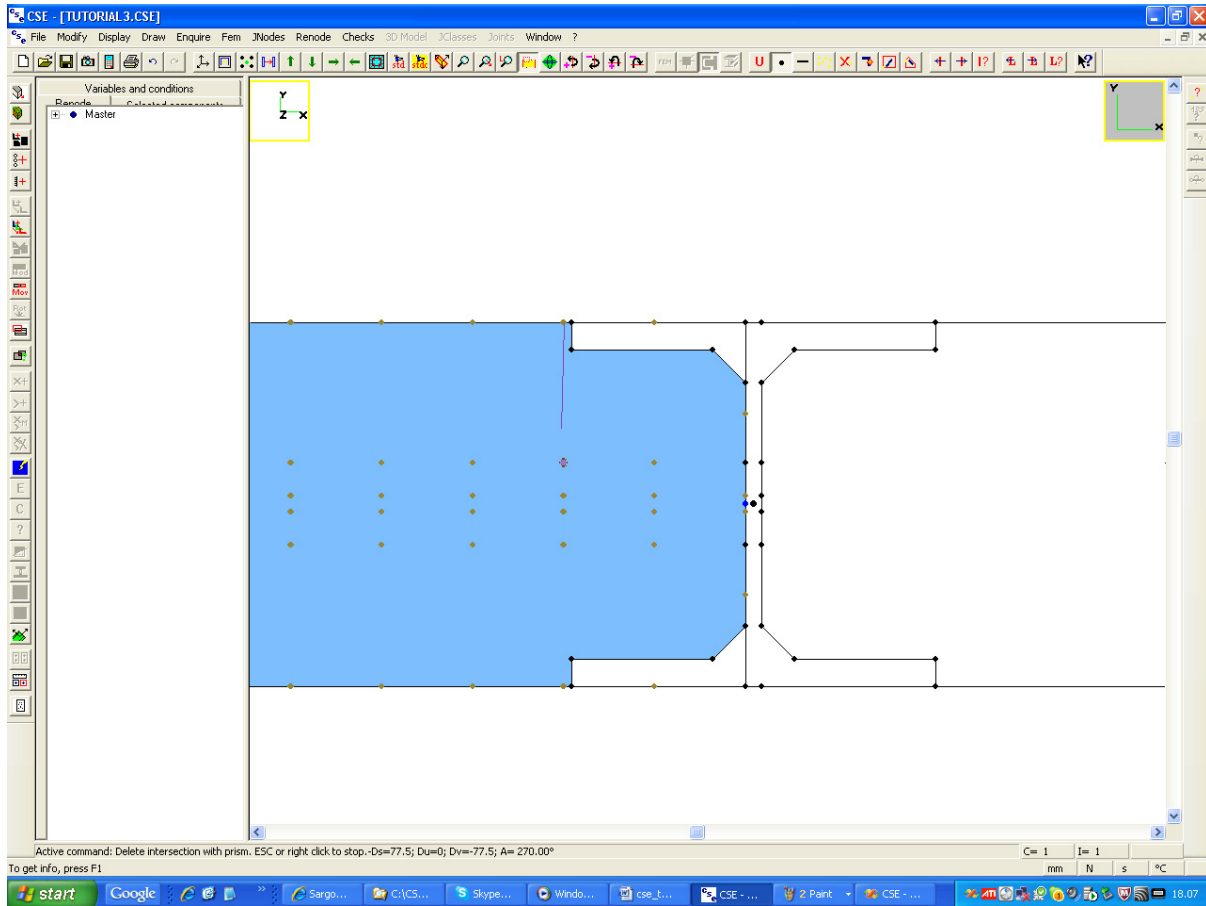


Then go back to the command you'll see larger dots. This depends on your display card]

Now if you move the mouse the red circle tells you which dotted point will be chosen if you click left. Using the mouse wheel zoom so that the points are clear. Click over the following point:

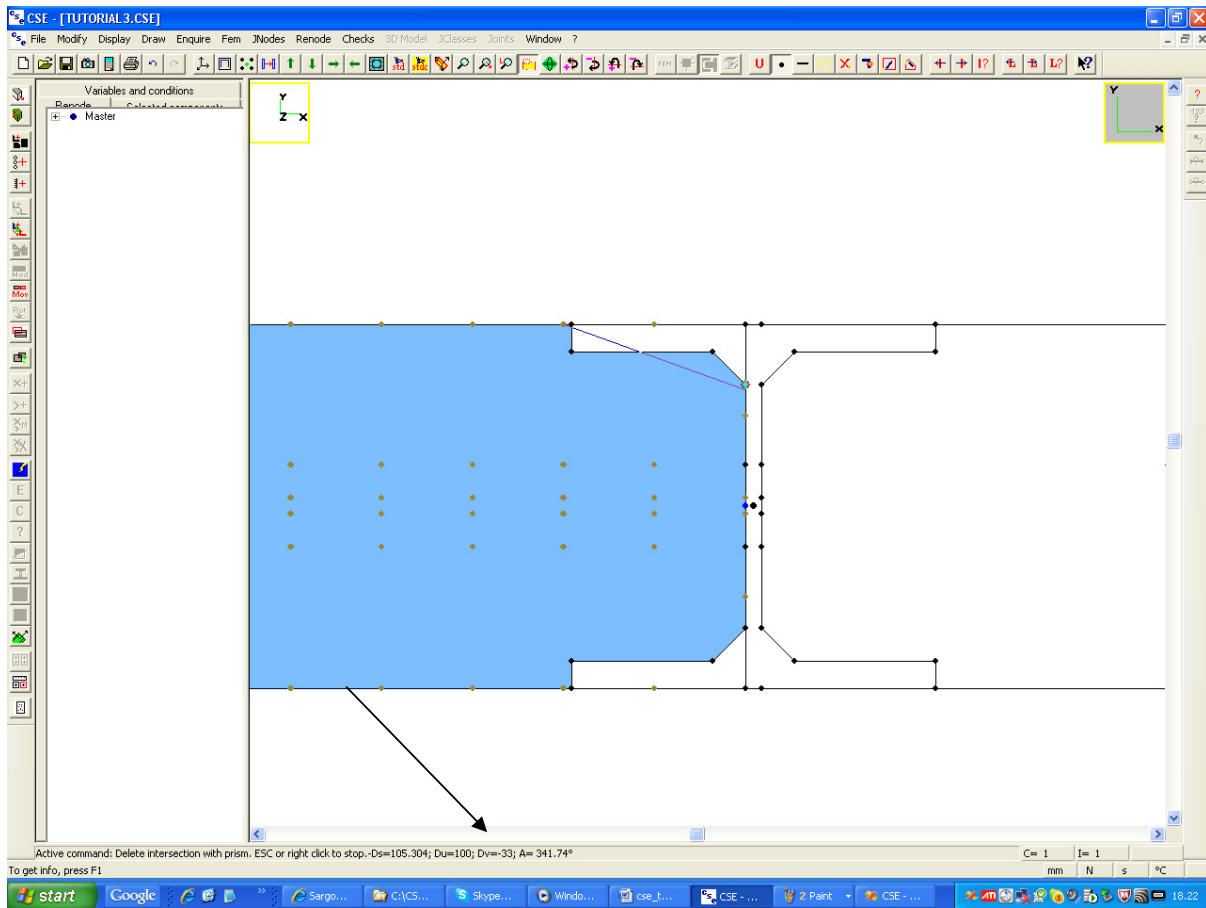


it is a brown point near the column corner, but out of it. The mouse will be attached to this point.



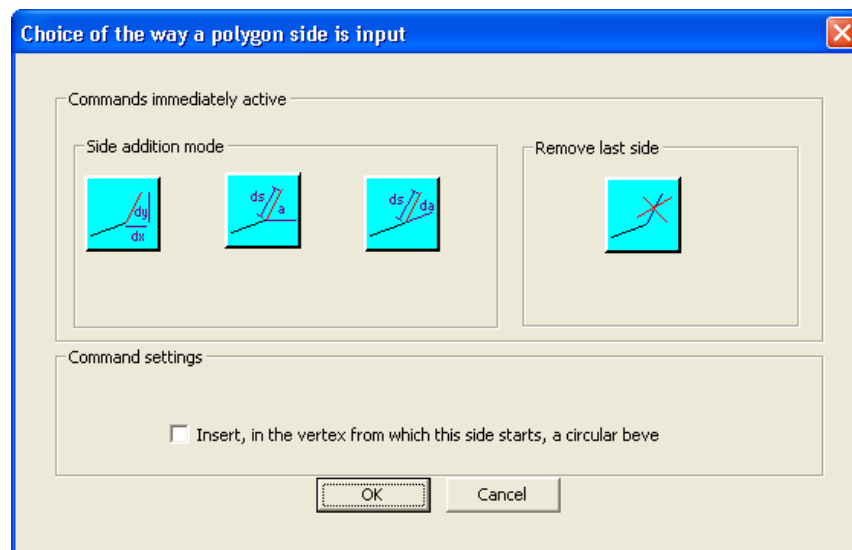
Now we would like to get down of a distance, but don't know how much. We would get down to the end of the column web point, but there is no point along the vertical at the correct height. If you place the mouse near the end-web point (like shown in the next picture) you will read in the status bar the DU (horizontal) DV (vertical) distance in the projected plane of the two points (the last clicked or defined, the mouse is attached to it, and the one you are over).

The DV you read is -33mm.

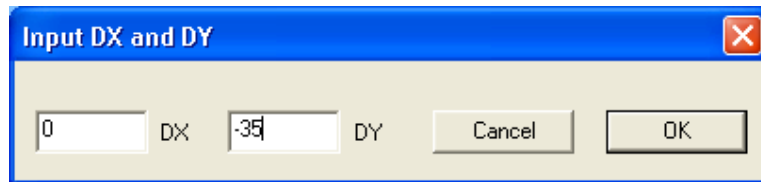



We now can decide to move down by 35mm, to stay inside the web length.

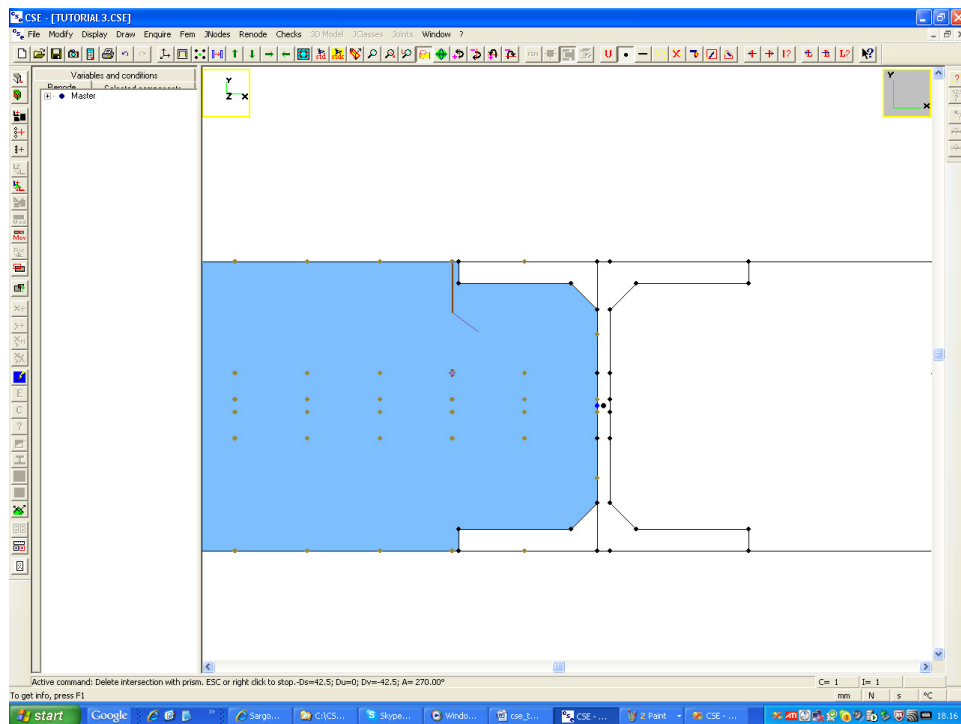
Now press the space bar as there is no point already in the scene to attach to. The following dialog appears:



Press the leftmost button. You get here:

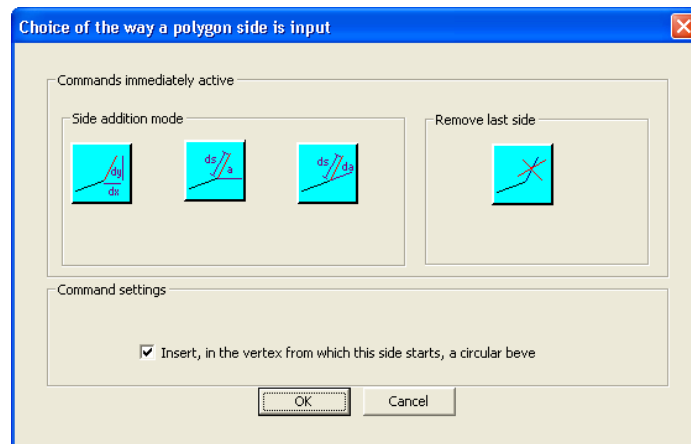


input -35mm as DY. You see now what follows (maybe you will need a redraw,  button in the main bar):



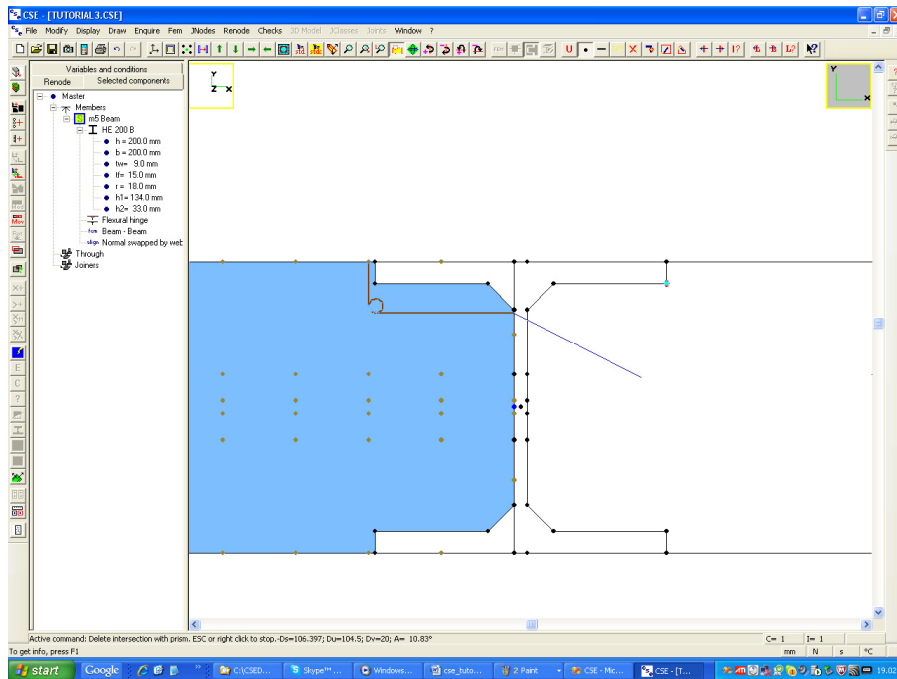
We are cutting the steel. Now we must move in +DX. The DU from the last point to the web is 100 (a multiple of 50)


Press once more the space bar place a tick in the **"Insert, in the vertex from which this side starts, a circular bevel"** check box to mean that in this corner there will be applied a circular bevel:

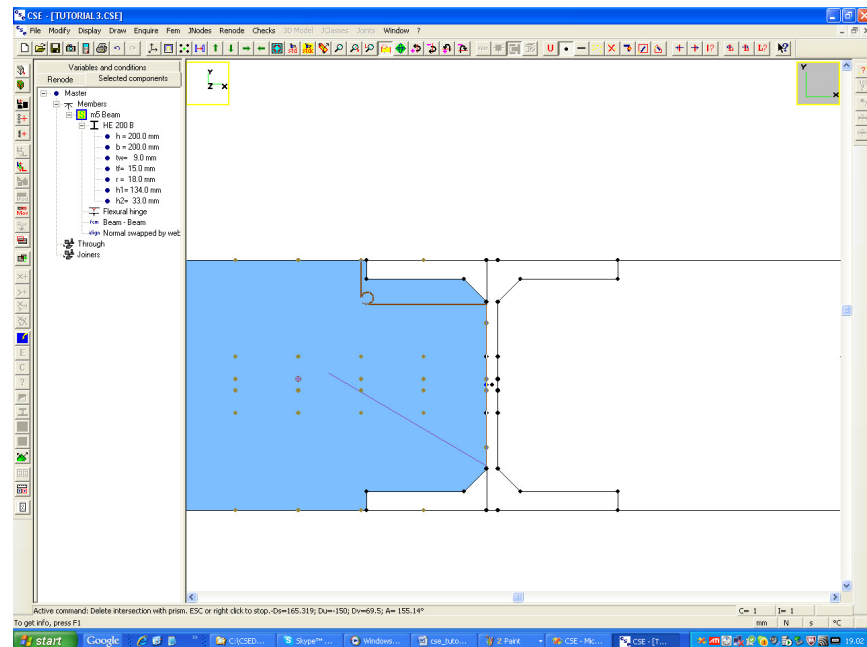


Choose once more the leftmost button and enter +100 as DX.

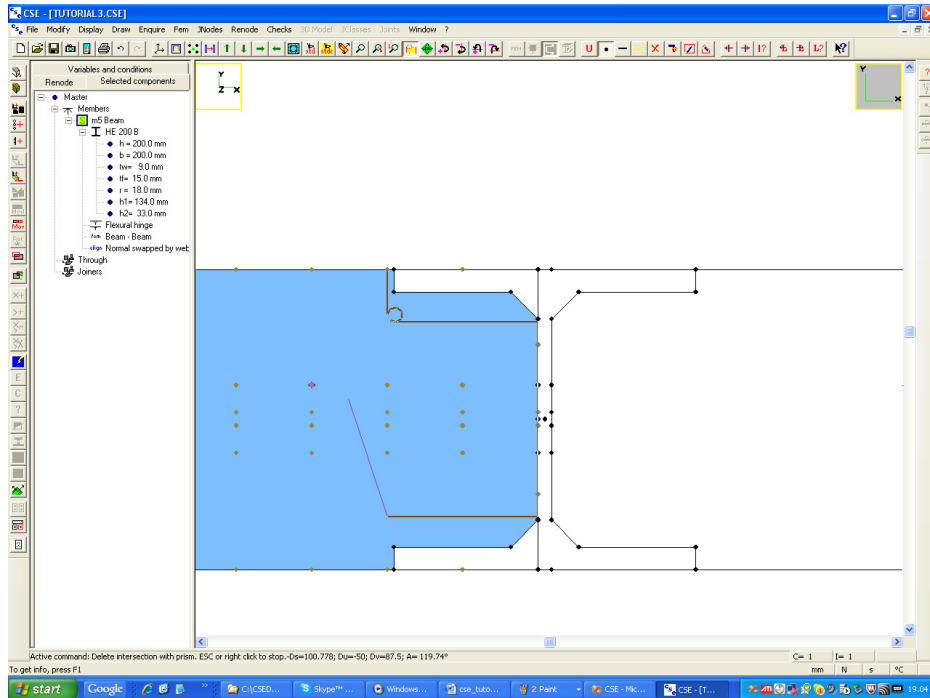
You'll get here:



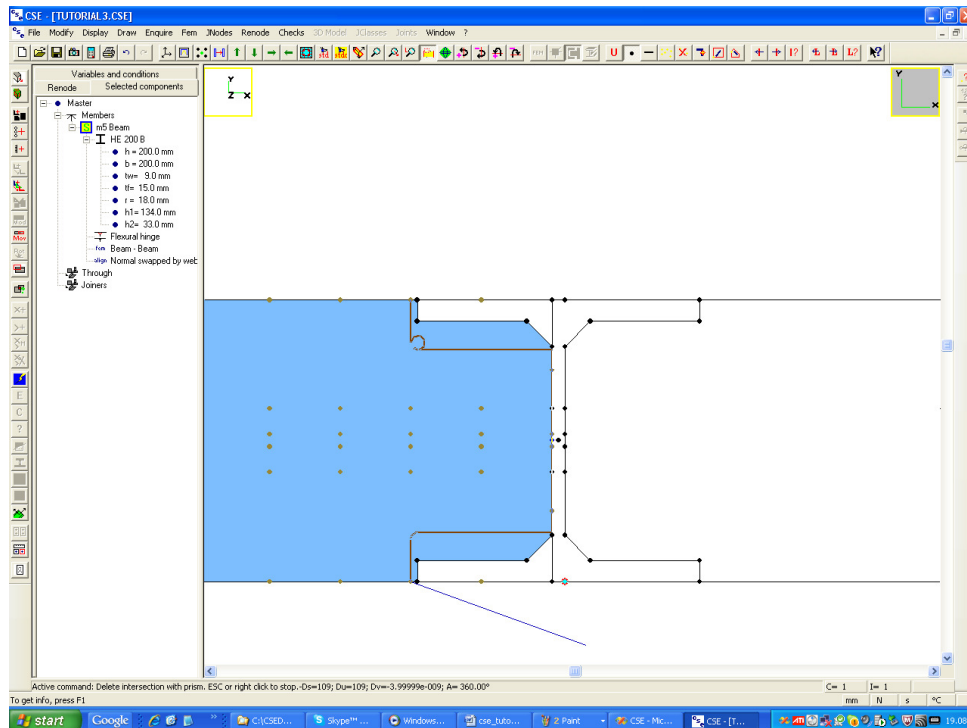
Press the space bar once more *remove the tick from the "Insert, in the vertex from which this side starts, a circular bevel "* check box and specify -130 as DY (we understood that -130 is the right distance examining the distances as already explained). You get here (always remember you can redraw  to refresh the screen):



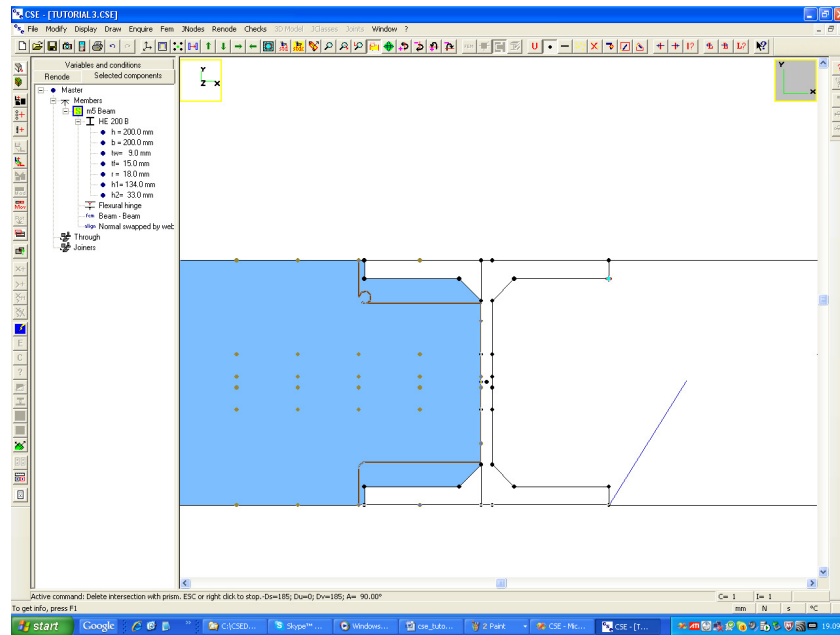
Now specify -100 as DX, always in the same way, you get here:



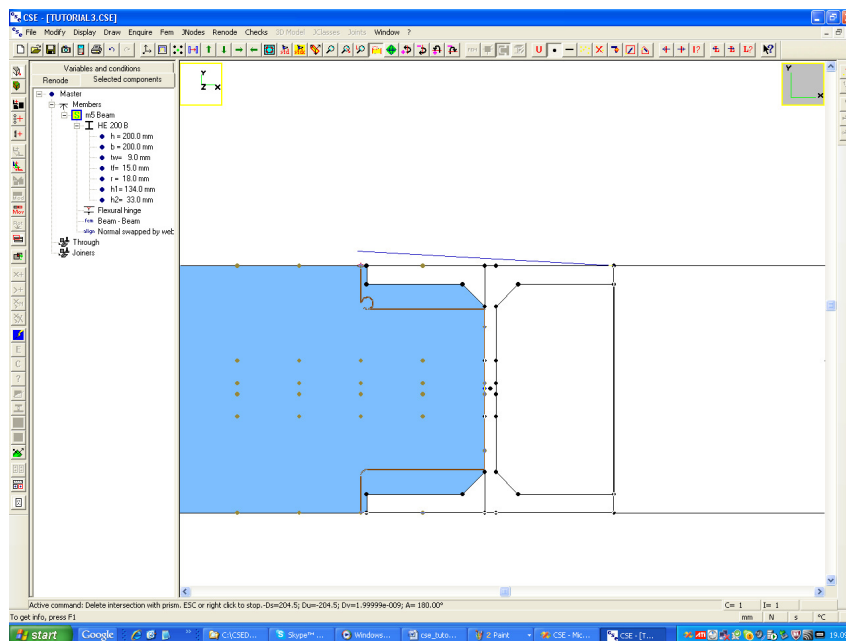
Now press once more the space bar, *place a tick in the " Insert, in the vertex from which this side starts, a circular bevel " check box*, specify -35 as DY you get here:



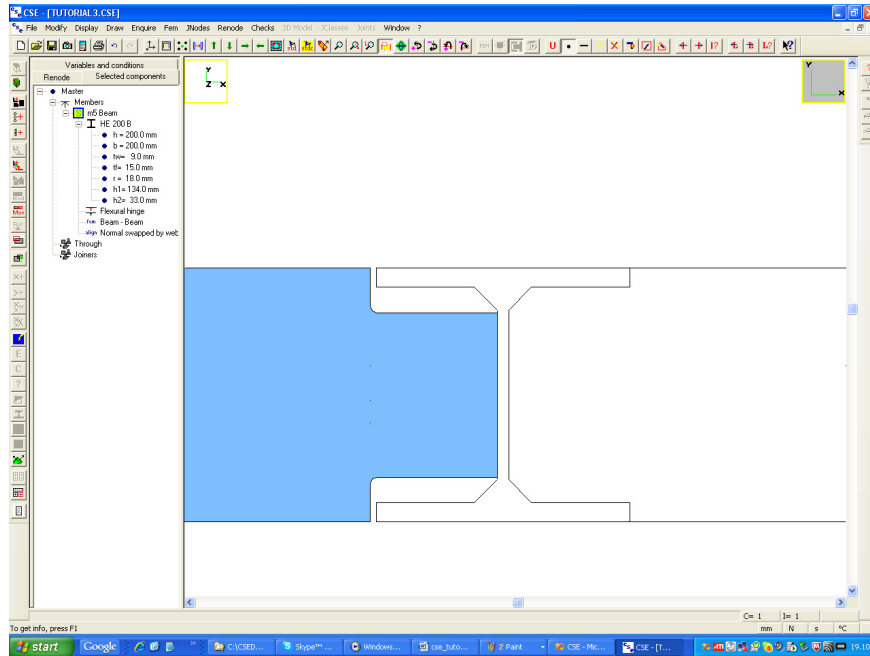
We can now close the polygon choosing existing points. The first, moving horizontally (remember to remove the tick from the circular bevel check box from now onward):



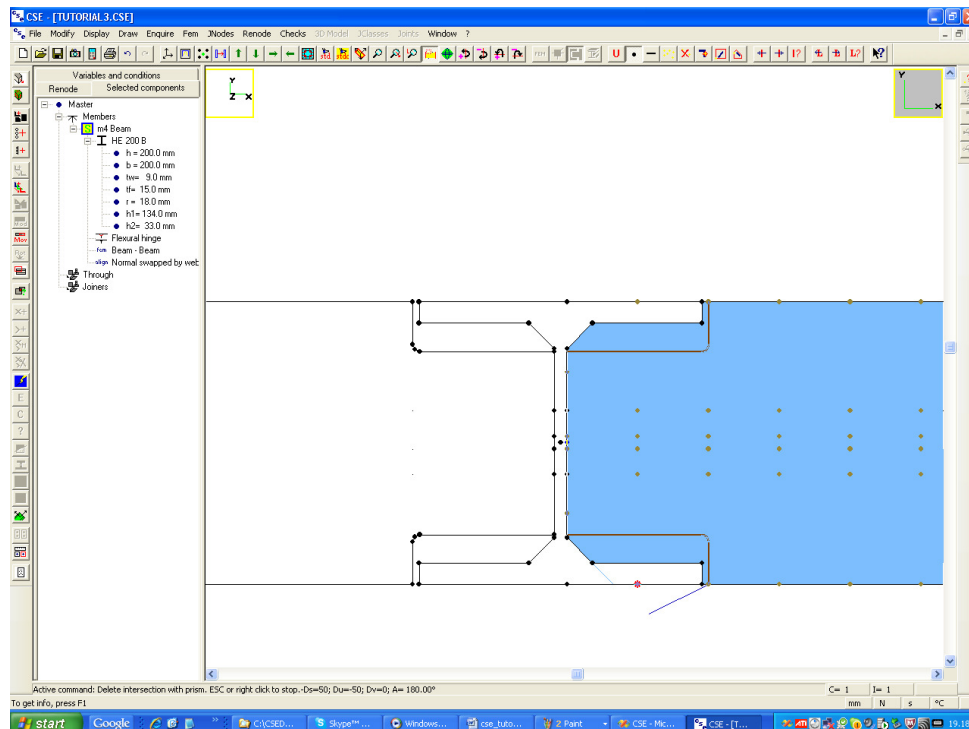
The second moving vertically



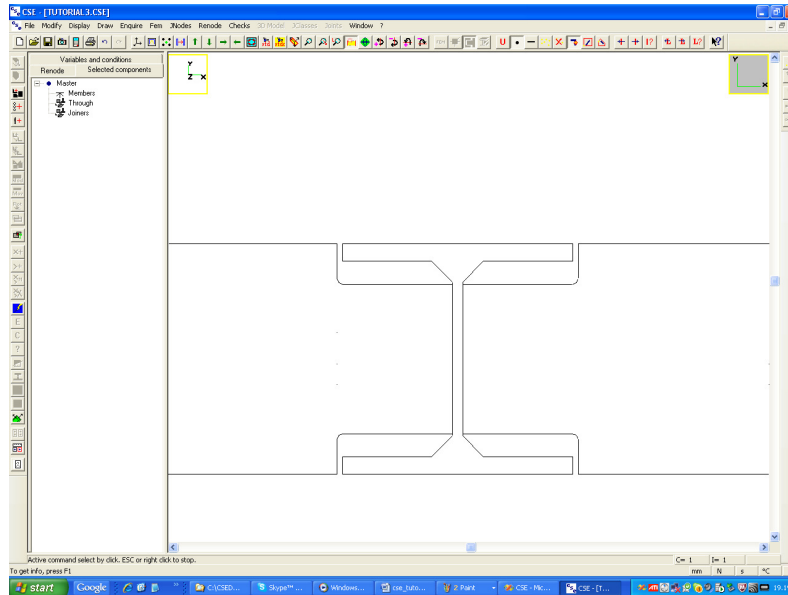
And the last point, back in horizontal direction, over the same very first point we have clicked, getting to an immediate end of the command (because you've re clicked the start point), here:



Notice that the bevels have been applied. There is no more overlap. Now unselect the left member, select the right one, and re-apply the command Renode-Workings. Do basically the same things in the symmetric mode. Here is a sketch of an intermediate step.

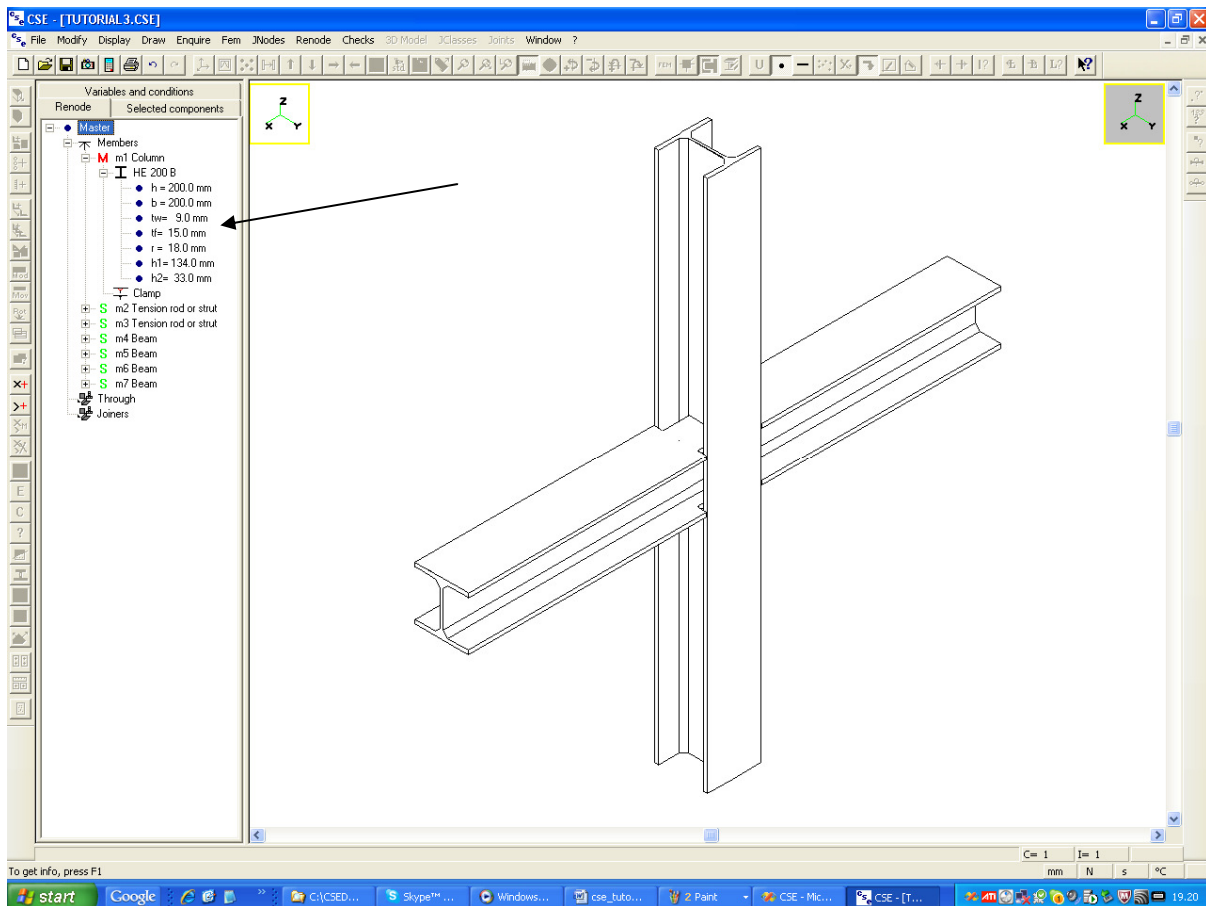


You will finally get here:




2.3.4 Apply double angles

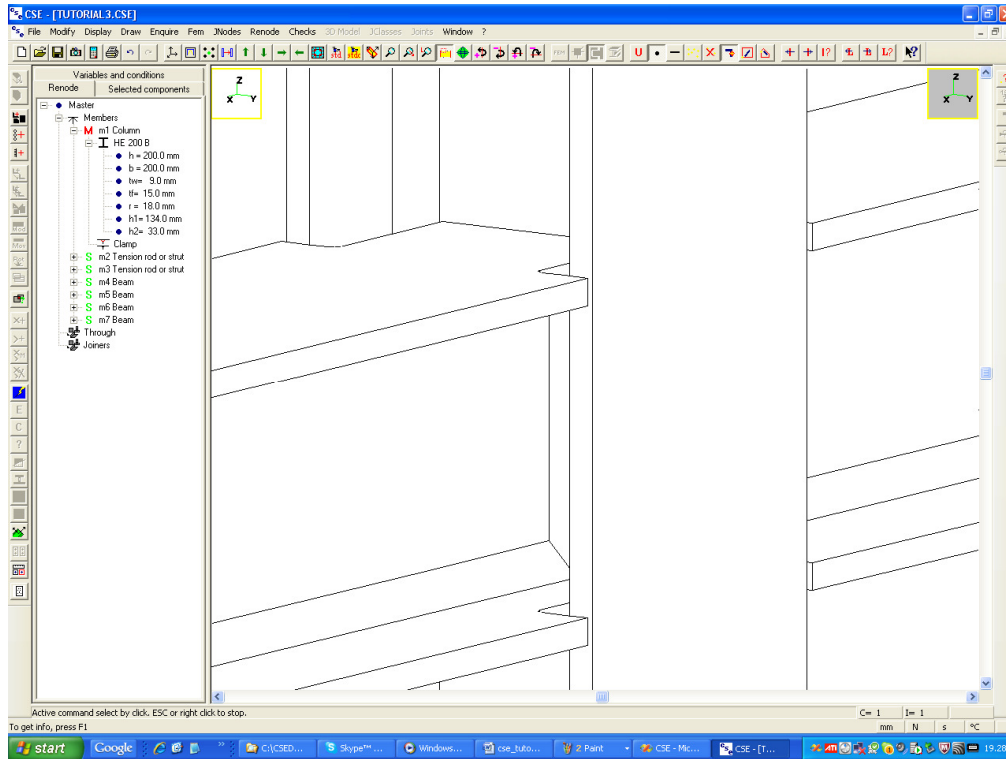
Now we will apply double angles. Switch to a Isometric view and have a look at the left pane "Renode".




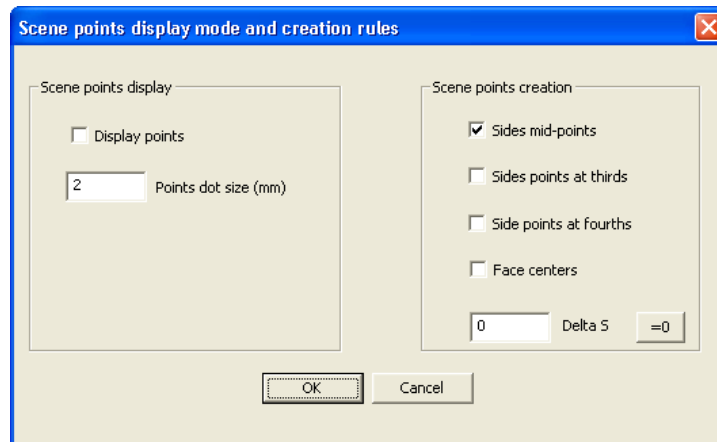


You see all the sizes you need. In particular you see that $h1 = 134\text{mm}$, so the clear web height is 134mm , and the angles will be say 120mm high. Also, as the web thickness is 9mm , the double angles will have to be spaced 9mm . Notice that we are still in extraction mode: as all the other members do not interact with what we are going to do we can stay here in extraction mode.

Slightly change the view so that the inner contact between the front beam and the column gets visible. You can do that by using these buttons in the main bar: . Get to see like this the inner part (a zoom has been also applied using the mouse wheel):

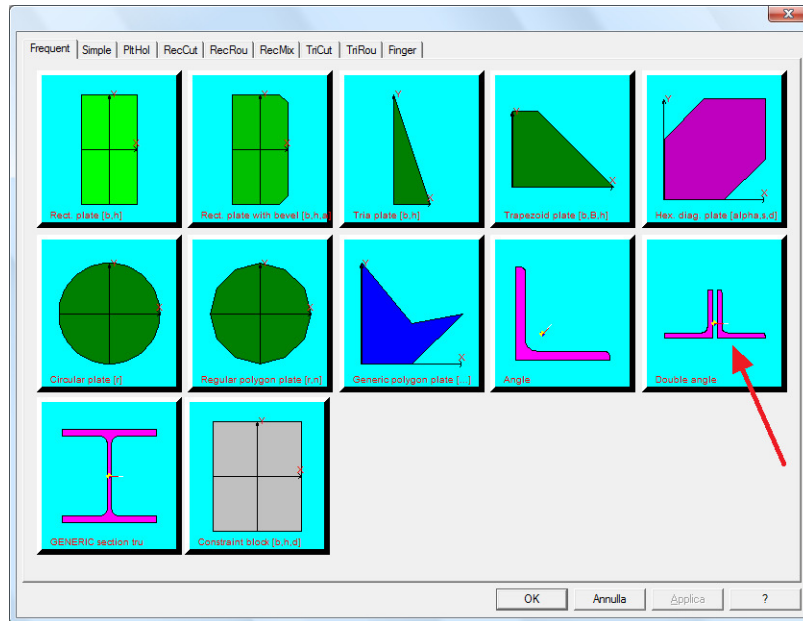


Now, before moving forward - for the reason that will be clear in a while - execute the **Display-Scene points** command ( button in the main bar), and set the significant scene points as here explained:

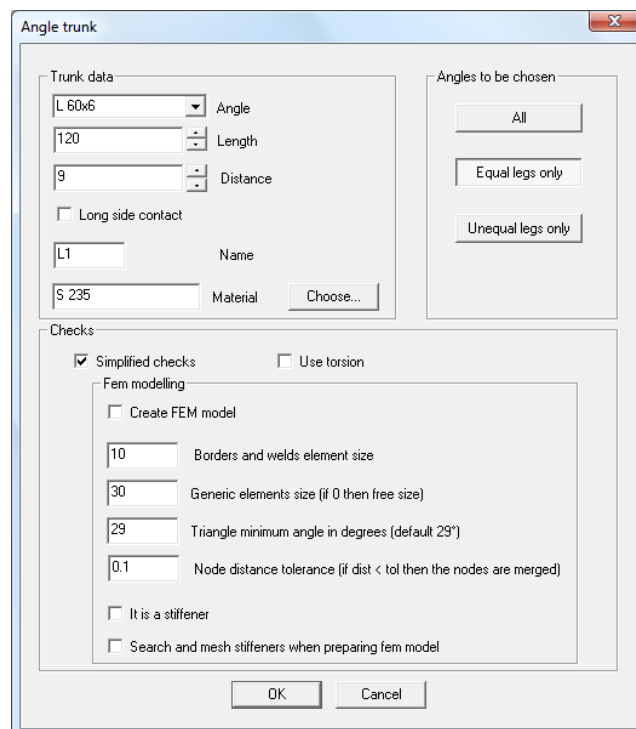


We avoided to use as significant point the face centers to avoid a possible confusion between the beam web end-sides mid-points, and the beam end-face center, as these points are very close. However this is not really necessary as we would detect 3 aligned points instead of 2, if leaving the face centers, as well.

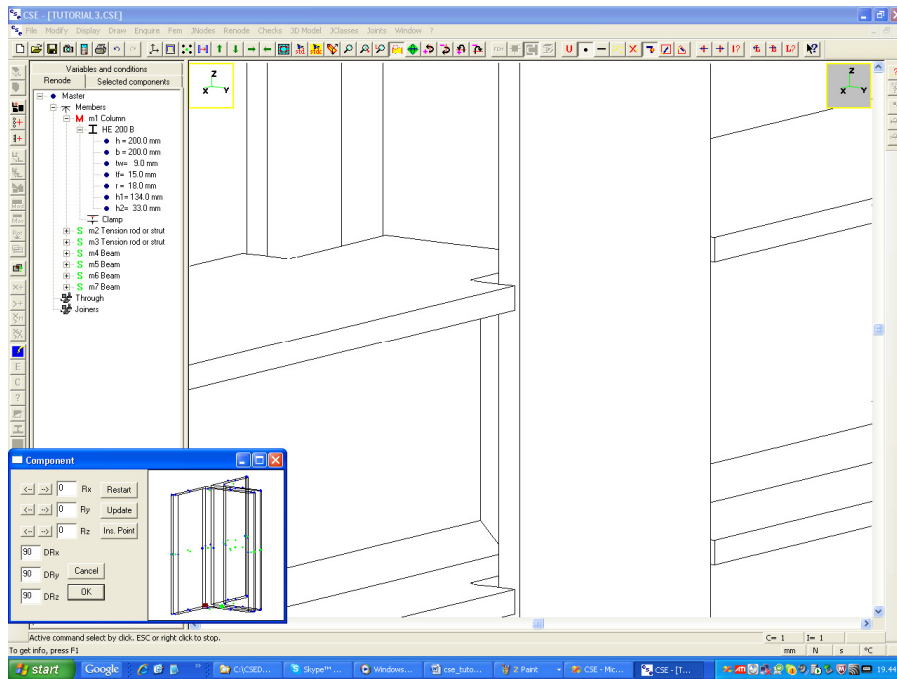
Now execute the **Renode-Components-Add through** command ( button in the left bar) you get the following property sheet:




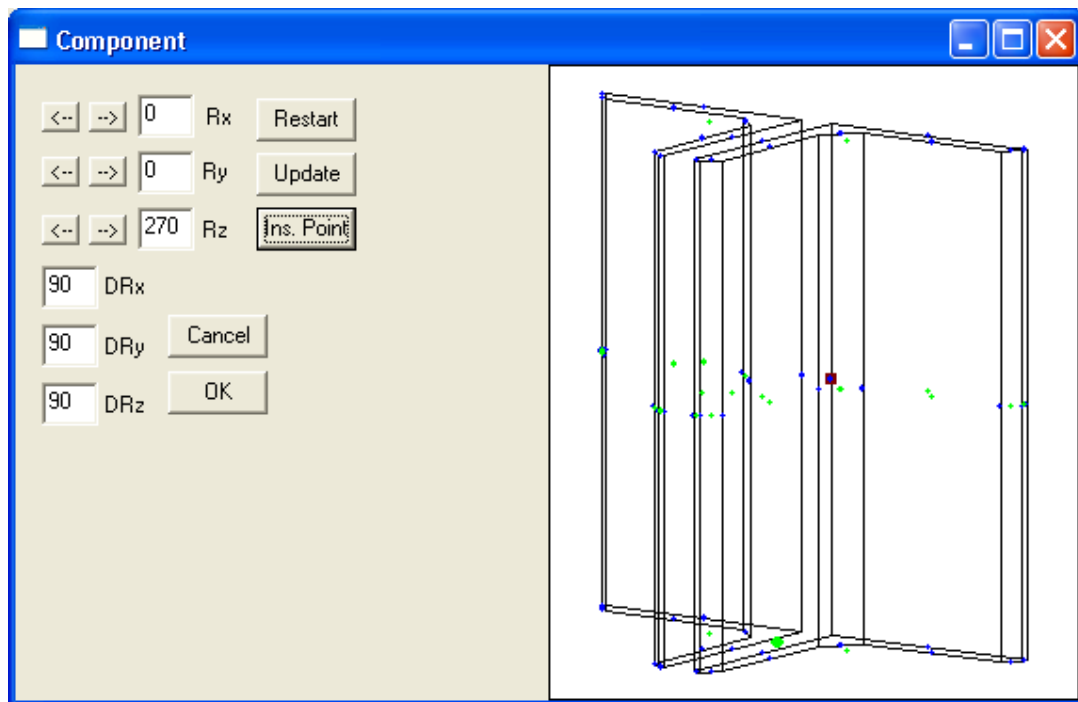
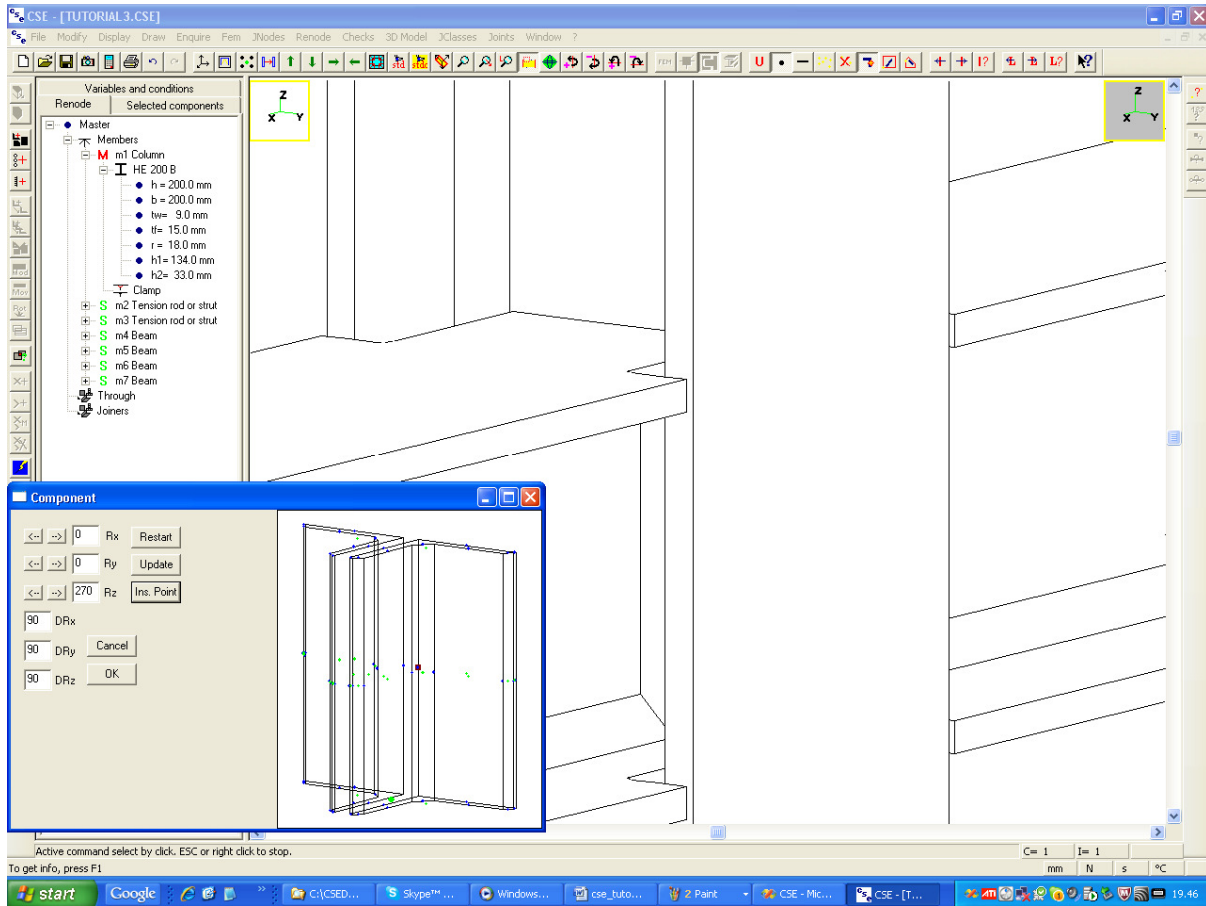
Choose double angles. You get here.



As shown specify L 60x6 (after choosing **equal legs only**), and 120 as length (part of the beam web height), 9 mm as internal distance between the sides. Notice that a new component named "L1" will be added. The material is that of the member by default S235, but you can change it. Choose to execute "**Simplified checks**" for this component, remove the tick from the "**Use torsion**" check box, and leave all default values. You are asking not to prepare the fem model of this component. To check it in a simplified way and not to consider the torsion applied to the legs cross-sections (a generalized force which usually is balanced by contact pressures of the leg faces). Press OK you will get here.

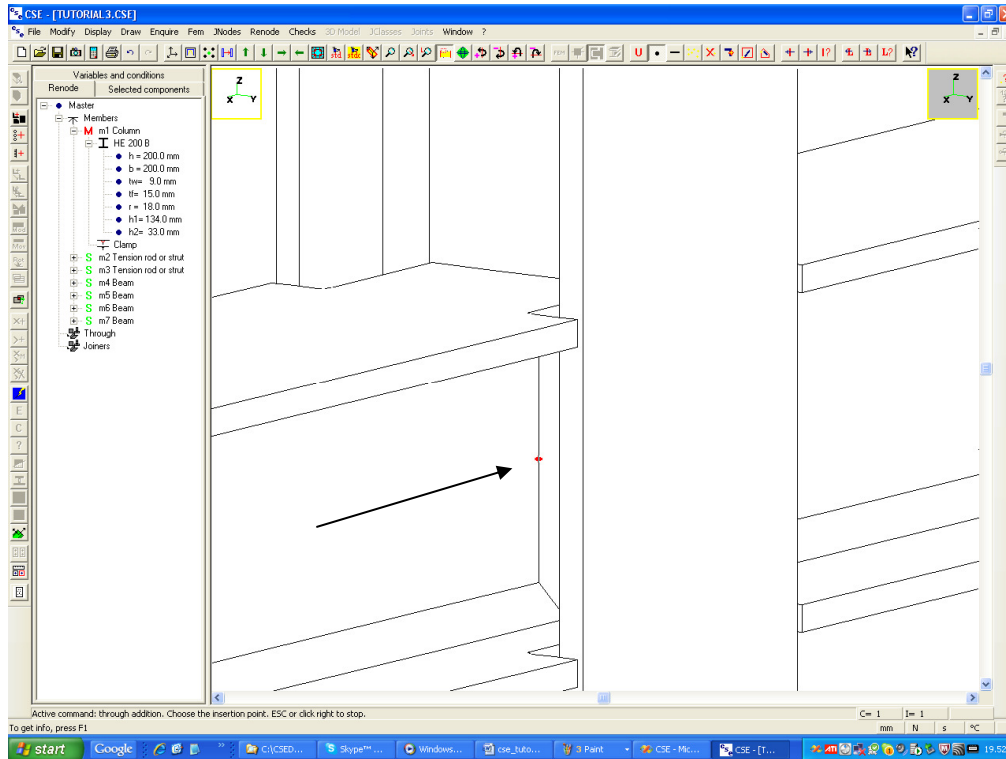


Enlarge the small window resizing it, so that you will better see the significant points. Then press once the  button in the Rz row (i.e. apply a -90 angle variation around axis Z). The object rotates *clockwise* around the global Z axis. You get what is shown in the next picture. Now press the Ins Point button, and choose the point shown in the next picture.

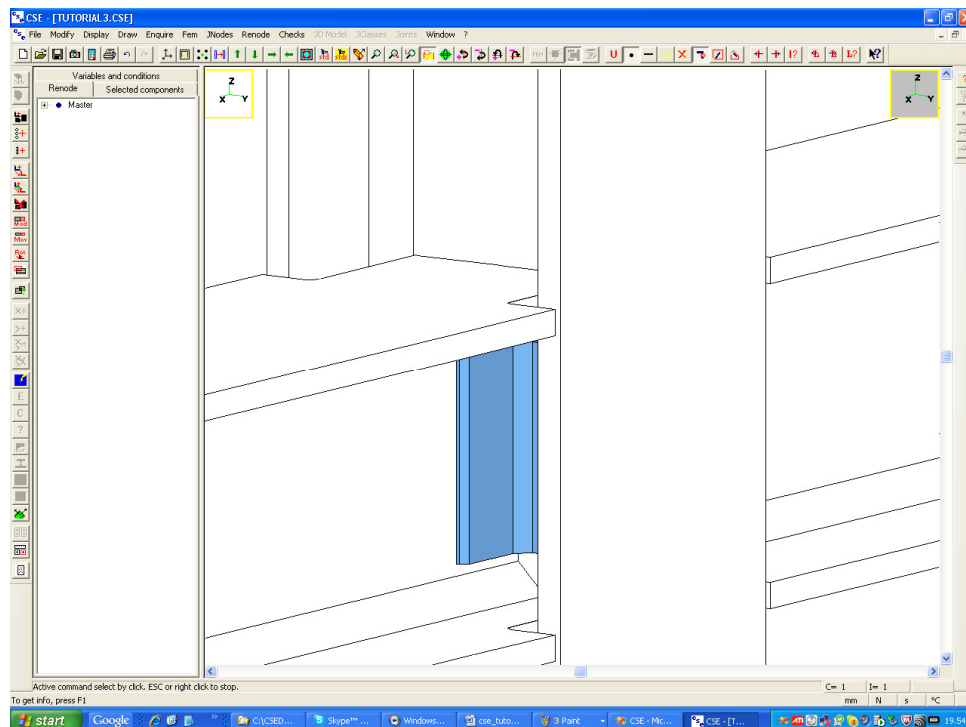


This point is the mid-side point of the right corner side, it is at mid height. This point will be placed in the scene by the next operation, so that the point chosen here will coincide with the point chosen

in the scene. Press **OK** and you are back in the global scene. Notice that moving the mouse the red circle tells you which significant points are available.

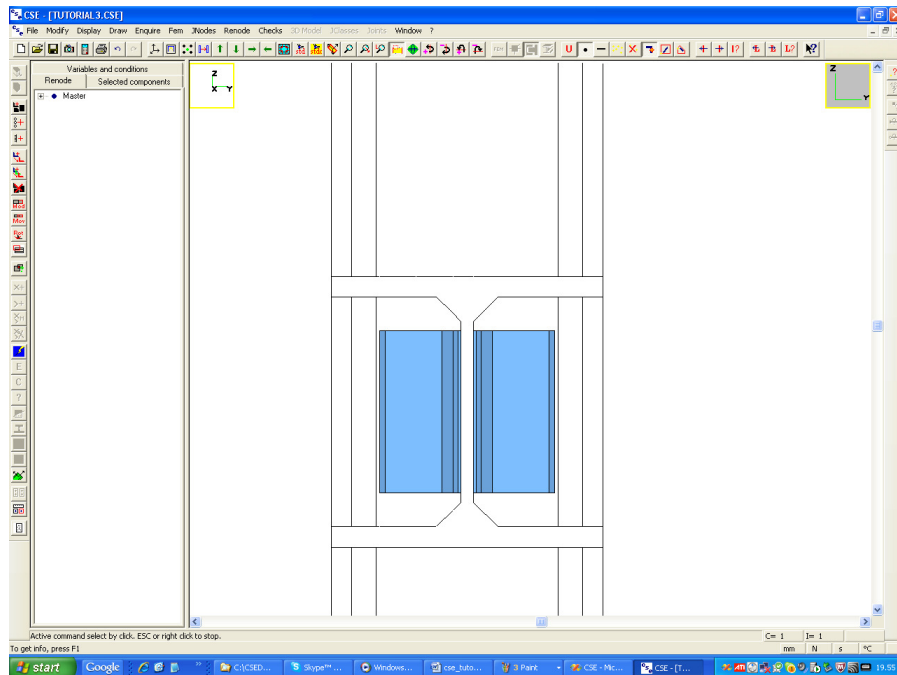



Choose the visible web side mid point, as shown in the picture above. You will get what follows:

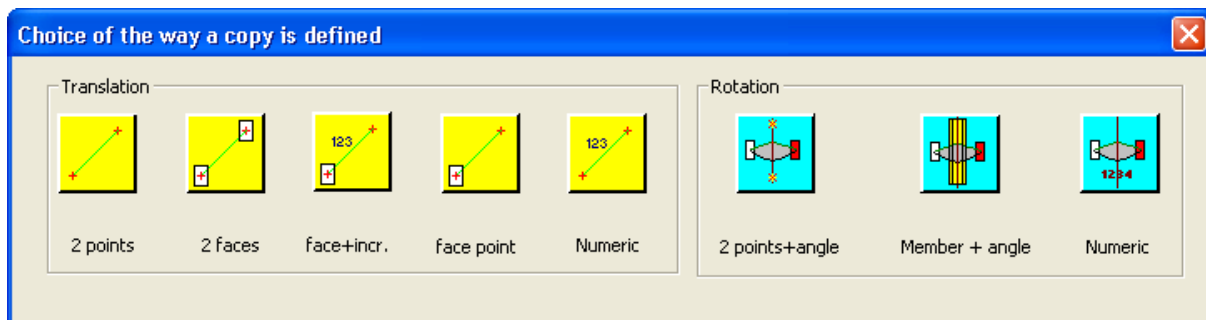


If you use the sounds a click tells you that no overlap has occurred.

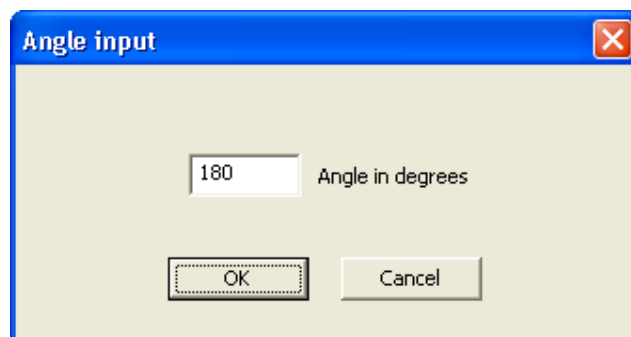
Change views to be sure the component has been inserted properly, for instance use a +X view:



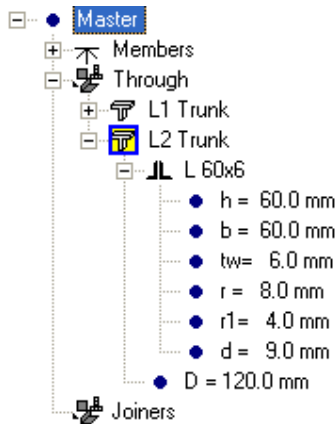
Now we can copy this component to the other side of the column. Choose the command **Renode-Components-Copy** (the  button in the left tool bar), you get here:



Choose the **Rotation** method, **Member + angle**, i.e. the 2nd button from the right. You are back in the scene. Notice that moving the mouse faces get highlighted. Choose a face of the column, that is, tell the program you wish to use the column as member, whose axis is the rotation axis. As all the column faces have their centers in the mid height search around the mid height. Using the mouse wheel you can zoom out. Once the column face is "under the mouse" i.e. it's green, left click. It will open the next dialog. Leave 180 and press **OK**.





We are copying component L1 around the column axis to create a new component, which will be L2. Once you've pressed the **OK** button, you will hear the "click" meaning that the copy was ok. In the left pane, **Renode**, a new component appears, L2, and is selected.



We are still in the +X view. Let's stay there as is comfortable for the next operation which will be to add the bolts joining L1 and L2 to the column.

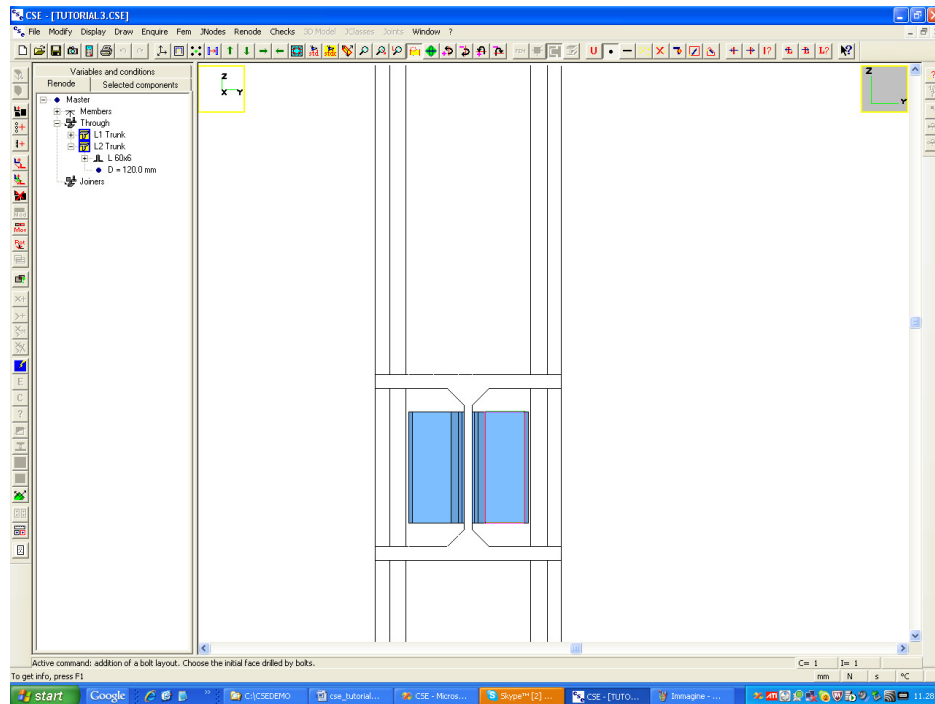
2.3.5 Join the angles to the column, have a look at bolt layout addition dialog

Unselect all (by the picture above you see that the component L2, hidden, is selected as it has just been added), by pressing the  button in the main bar (**Modify-Select-None** command). Now select the L1 trunk in front of you in the view by clicking over it. Notice that the left pane is upgraded, now it's yellow L1. This last selection is not strictly necessary but will help in the next command as we will have to select a face of the L1 trunk.

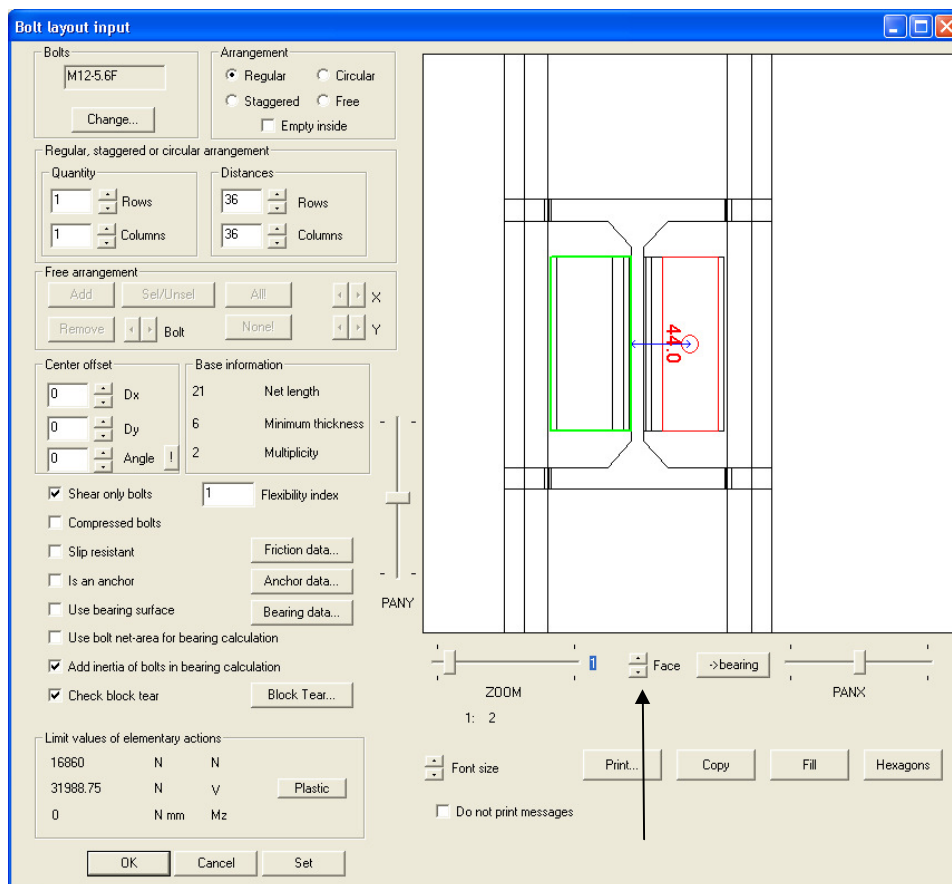
Now execute the bolt addition command (**Renode-Components-Add bolt layout**), by pressing the  button in the left toolbar. Notice that moving the mouse the face of the selected object (i.e. L1) gets highlighted. If no object was selected than all faces in the scene would have been selectable.

Now click the front face of L1 as shown in the next picture. Be sure to select the front face and not the rear face. The front face is smaller than the rear face. It is the first face bolts will drill.

Also note that unselecting the rear component L2 is needed to avoid a possible error in selecting the faces which would have equal projected images. This is specifically true due to the orthogonal view we are using.



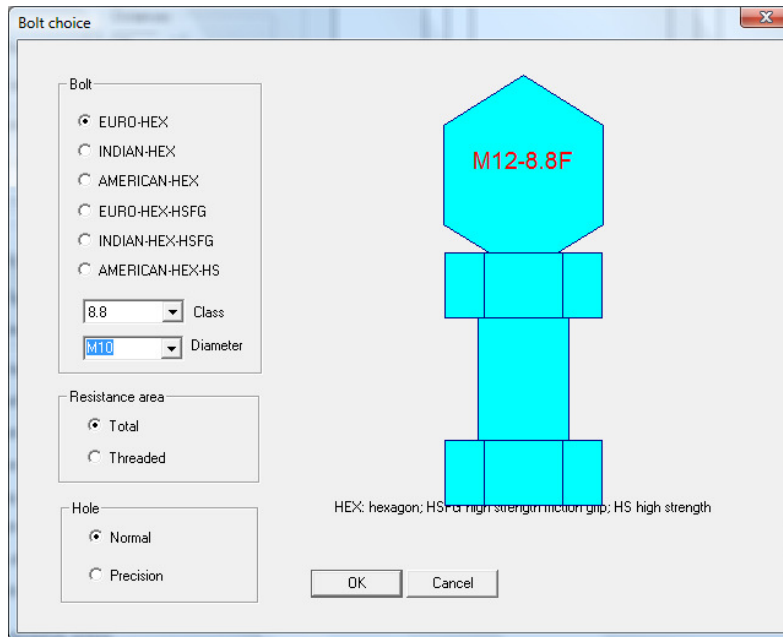
Now click left to select the face. You get into this very important dialog box, used to manage bolts.



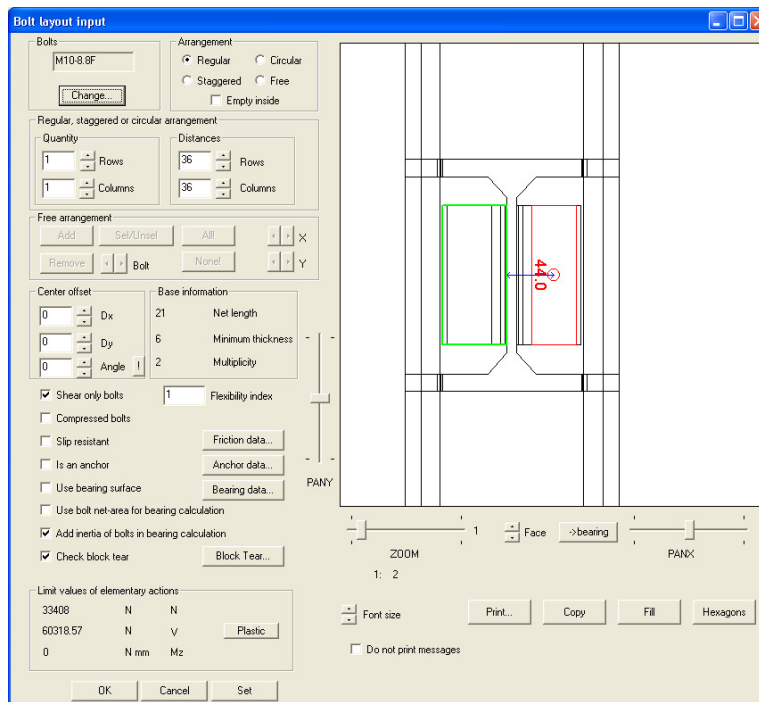
First of all notice that a one bolt initial bolt layout is available. The bolt is placed at the center of the clicked face (which is the red face). The green face is the currently selected face among those in the

scene. The distances are drawn using the green (selected) face. This distances are important to understand if bolts are properly placed. To change the selected face use the **Face** arrow keys pointed by arrow in the previous picture.

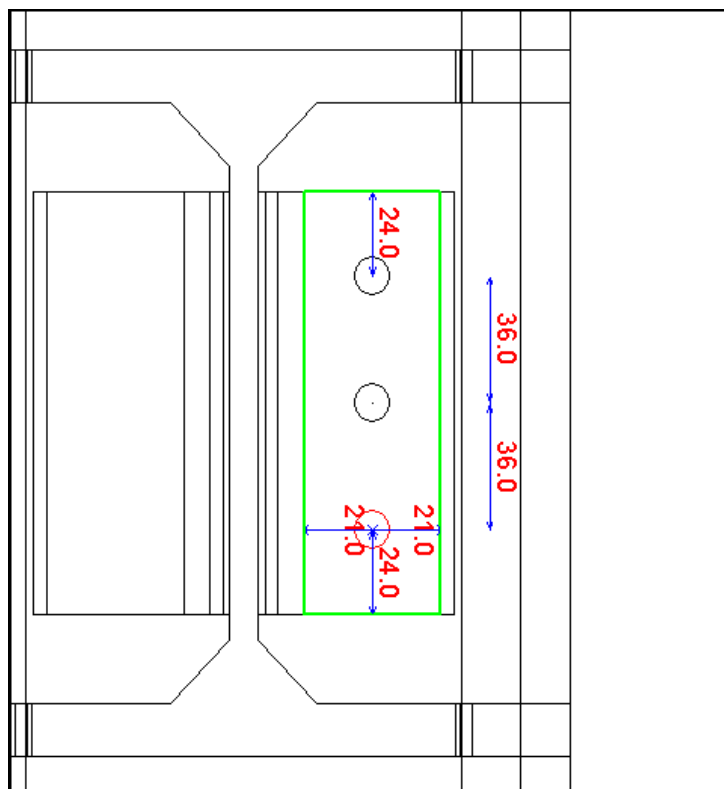
First of all we decide the bolt diameter and class. Press the **Change** button (it's up left). You will get here:



Choose class 8.8 and diameter M10 between EURO-HEX bolts. Leave **Total** area and **Normal** hole. Press **OK**. You are back in the main BL dialog, but some things have changed: the drawing and the bolt identification string.



Using the Rows arrow key change the row number to 3. Move to the clicked face using the Face arrow keys. The clicked face (where we are putting bolts) must get green. Then, using the Zoom slide control enlarge the drawing so that you can read the distances, like this:



The image you see here has been pasted from the dialog box. To do that you just press the **Copy** button, and then paste into the program you are using (e.g. Word). This is a very useful feature when preparing documentation referring to the connection studied with CSE. The distances from the border are ok (1.2d₀ from borders, 2.2d₀ between bolts).

We could basically consider finished the geometrical input phase. Let's have a closer look at the dialog.

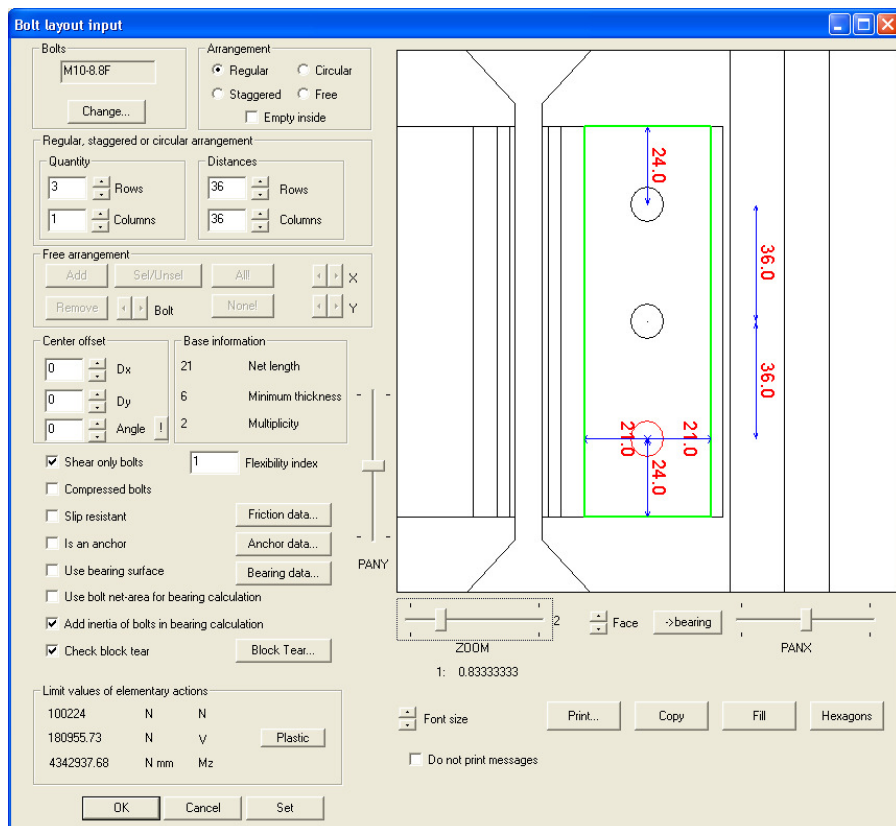
Notice that in the **Base Information** box, there are read only values that tells us that the net bolt length will be 21mm (6mm L1 thickness + 9mm column web thickness + 6mm L2 thickness = 21mm); that the minimum thickness of the drilled objects is 6mm; that the multiplicity of the bolts is 2, meaning that there are two sections in shear (as 3 objects are joined).

Notice that in the "**Limit values of elementary actions**" box, read only values appear, referring to ideal simple actions taken by the bolt layout: a shear (V) of 226,194.67N is applicable. This is merely the sum of the bolt shears according to the standard in use, and keeping into account multiplicity:

$$A = \pi \times 10^2 / 4 = 78.5398 \text{ mm}^2.$$

$$\gamma_{M2} = 1.25$$

$$3 \times 0.6 \times 800 \text{ N/mm}^2 \times 78.5398 \times 2 / 1.25 = 180,955 \text{ N}$$



Now let's choose the resistance settings of the bolt layout, i.e. how the bolt layout works, which kind of checks must be performed, and so on.

This is basically in the left middle bottom area of the dialog, from **Shear only bolts** to **Check block tear**.

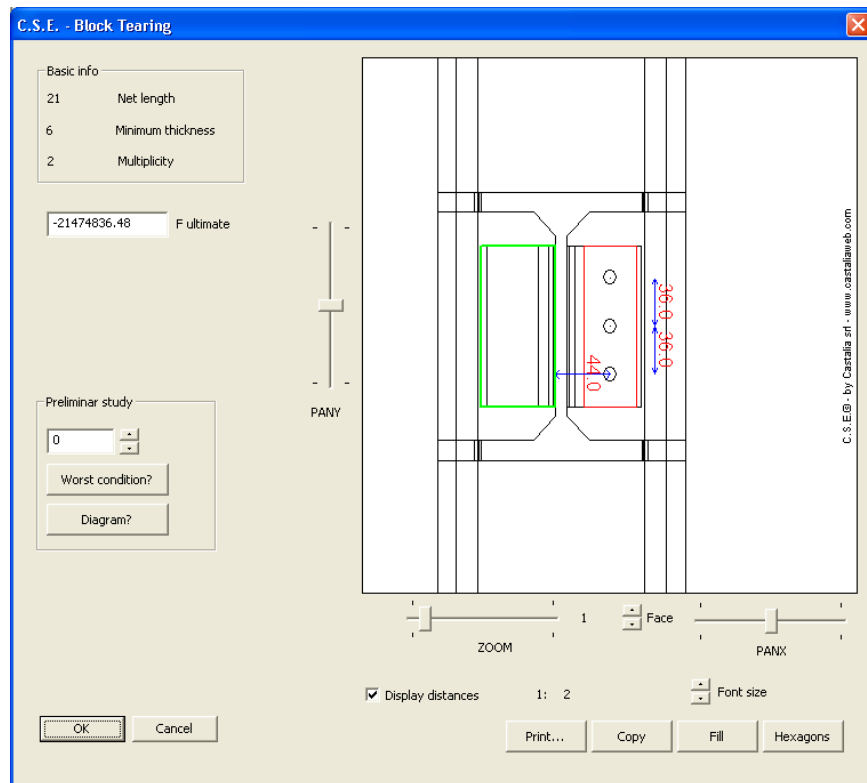
If we want a low axial stiffness for the bolts (no traction and no bending carried by the bolt layout) we can place a tick in the **Shear only bolts** check box. However here, if we just think at the X beam shear, there will be a small bending due to the lever between theoretical application point and the bolt plane, and no other joiner will be capable to carry that load. As the parasitic bending is low, we are not worried about possible slight tractions in the bolt, to carry that moment. *We leave the tick in the shear only check box: we will later see how this choice will affect results.* **Compressed bolts** is used where bending or compression is relevant, and we want to check bolt stem for compression: usually this is not required as the compression is taken by surfaces in contact. Here we are not expecting these action so leave the check box blank.

Slip resistant is a flag meaning that the bolts will be slip resistant. If the flag is on then we will have to specify **Friction data**, by pressing the button immediately at the right. You can have a look how it works, in this example we are not using friction bolts.

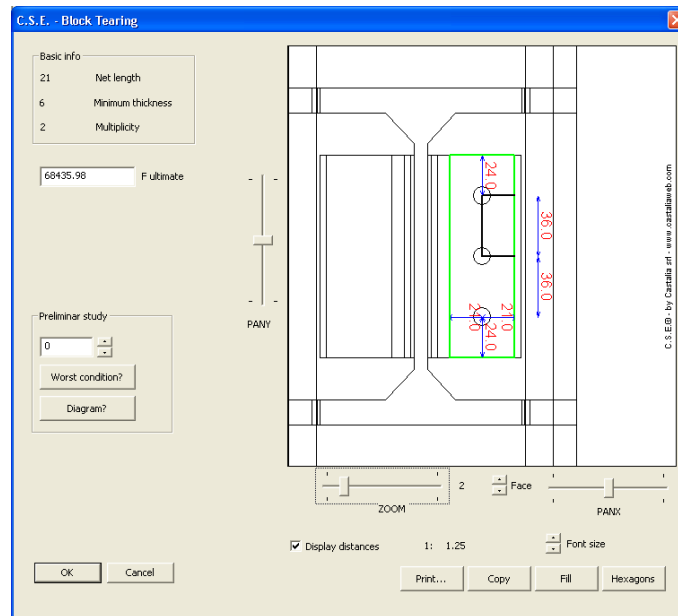
In an anchor is not applicable here, as this is not a steel-concrete connection.

Use **bearing surface** is a very important flag. When there is a bending (here the bending will be low, and equal to the shear force F times the small distance from theoretical application point to bolt resisting sections), this flag tells the program to use a compression-only surface-contact to carry bending and compression (and mixed actions). We are not using this feature now. If we would have used this, we would have also pressed the **Bearing data** button, to define a no-tension constitutive law, and the "**->bearing**" button, to define the surface to be used to help carrying the bending/compression. Also we could have used the flag **Use bolt net-area for bearing calculation** flag and the **Add inertia of bolts in bearing calculation** flag. As we are not using bearing surface, these flags are un-relevant.

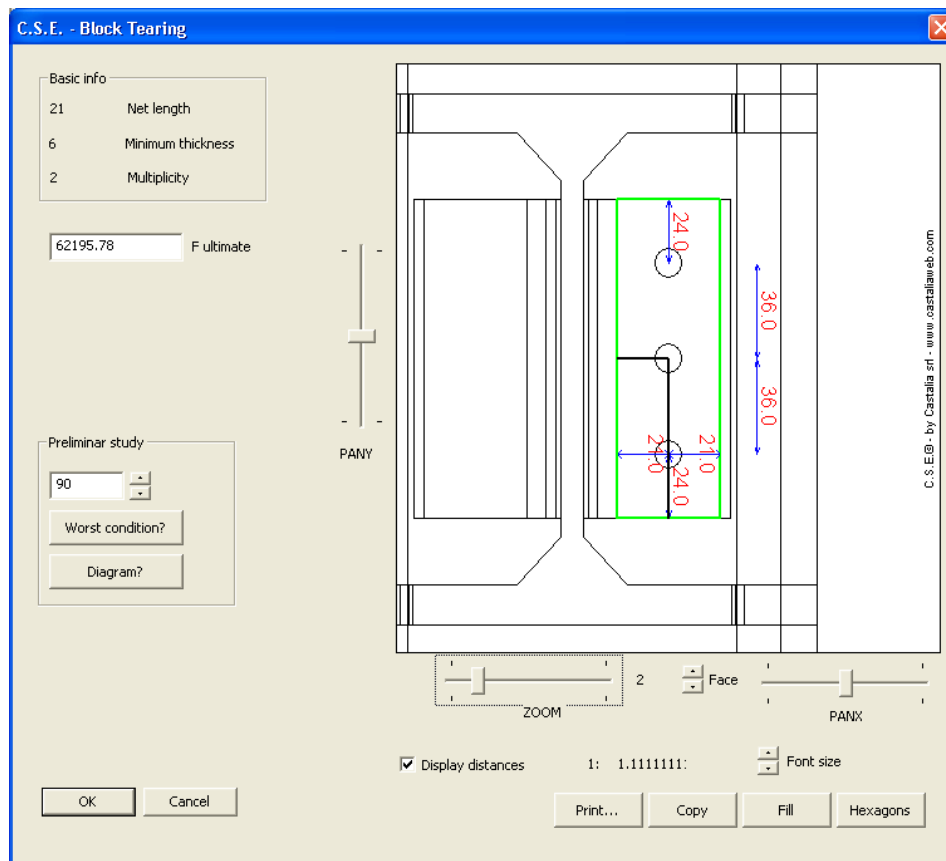
Finally we find the **Check block tear** flag, which is important and applicable here. If we leave the tick in the check box, then the Bolt Layout will be checked for block tear, i.e. all the components joined by this bolt layout will be checked for block tear. Block tearing is a quite complex matter, still not completely addressed by standards in our opinion. CSE performs very many checks using hypotheses on the safe side. Leave the tick. To have a preview (remember that CSE is a program to study connection, not just a program to check them) of the situation of this connection for block tearing, press the button **Block Tear**. You get this dialog.



First of all use the Face arrow keys to select the L1 (or L2) bolted smaller face. You get here:

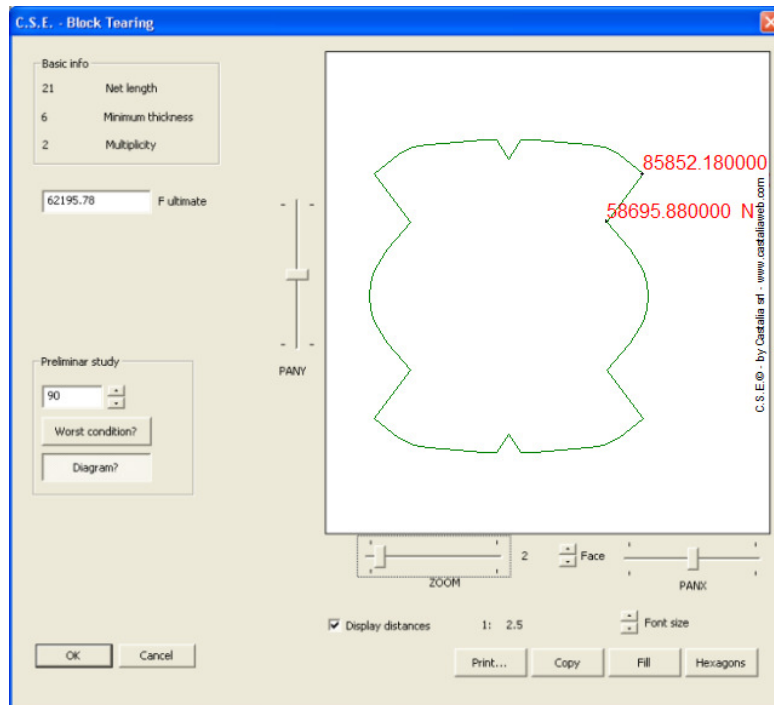


Now using the arrow keys in the "Preliminary study" box, browse to 90 degrees, which will be the direction of the force expected. You get here:



Assuming that all the bolts will carry the same load, the program predicts a maximum shear force equal to 62195N. Notice that the failure line is in thick black and use two bolts, not all three, as the program has computed this situation is worse. 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 α 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.



The minimum is 58695N. If you press the **Worst condition** button, after un pressing the **Diagram** one you see what it means, i.e. you see the predicted failure line for a force which turns out to be inclined 35° degrees. Notice that the failure line segments are not parallel to the force. This is in fact a possible mixed shear-normal stress condition over the failure line which is not clearly addressed by standards.

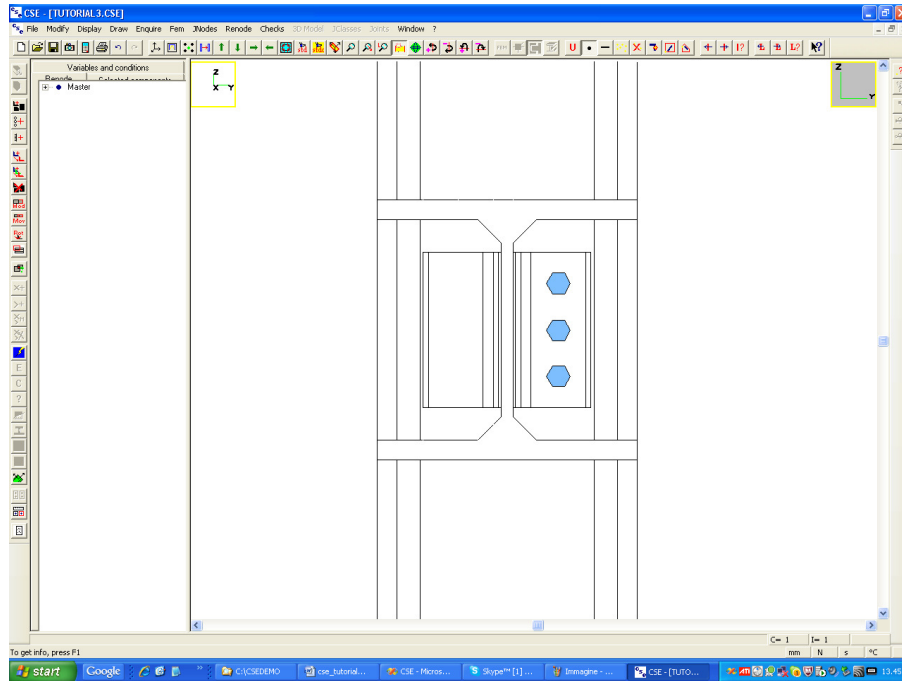
Block tearing check in a very general context like that of CSE (it works with any possible components and directions, load combination and instance of the renode) is currently a research matter. The program implements a very general tool, giving safe side result in most of the cases. Further studies are on scheduled on this matter.

Before the true checks, in this preview, the bolts are assumed to be equally loaded by a shear force equal to global resultant divided by the number of bolts. In the true checks, each bolt in each combination will be considered with its true shear force.


Exit from here. It seems that the maximum force will be about $60 + 60$ kN for each beam (as we are in for adding a new bolt layout). This means $120\text{kN} < 180\text{kN}$ predicted by the initial guess.

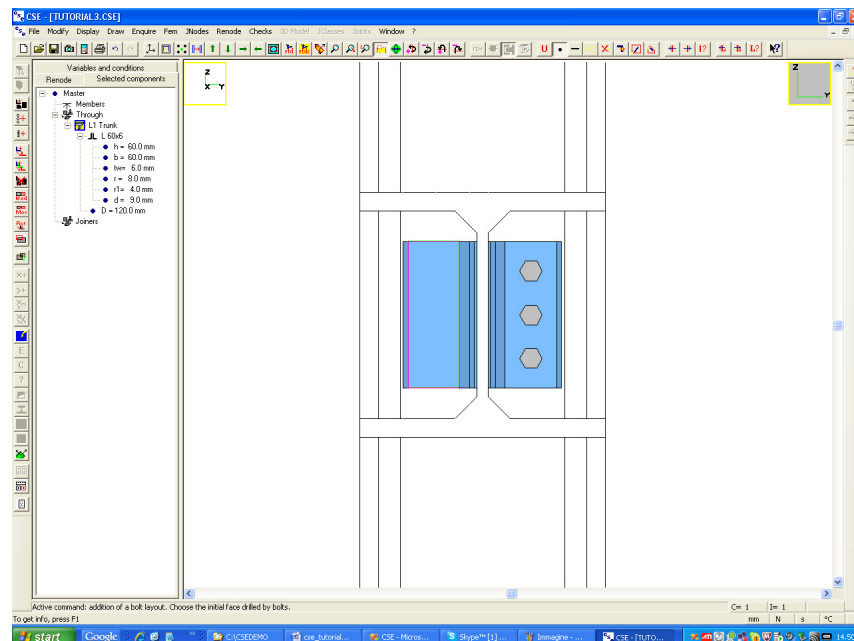
Press **OK** or **Cancel** and go back to main BL addition dialog box.

Press **OK** to finally apply the bolts. You will see:

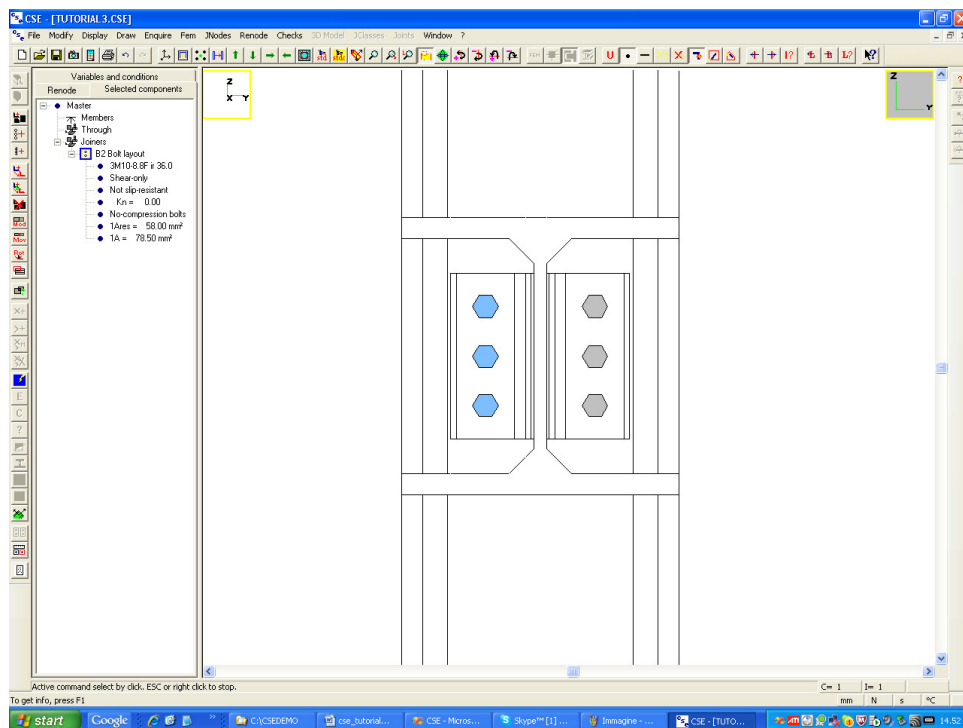


Now let's add a new bolt layout, equal to the previous, to the left side of the double angle. Notice that the L1 selection is lost, and that B1 (the newly added bolt layout is selected). By two clicks you will unselect the BL and select L1, once more.



Press once more the bolt layout addition button () , and now choose the symmetric face on the left. Like this:

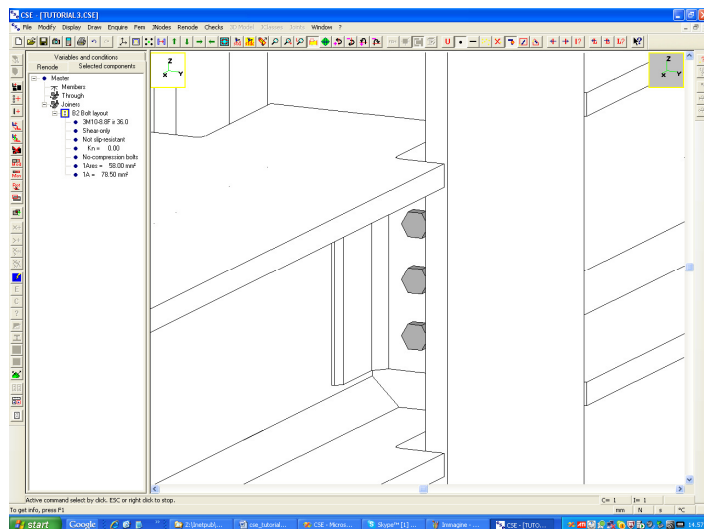




You get into the main BL addition dialog. Notice that all is already set as for the last added BL. Just press OK and you get here:

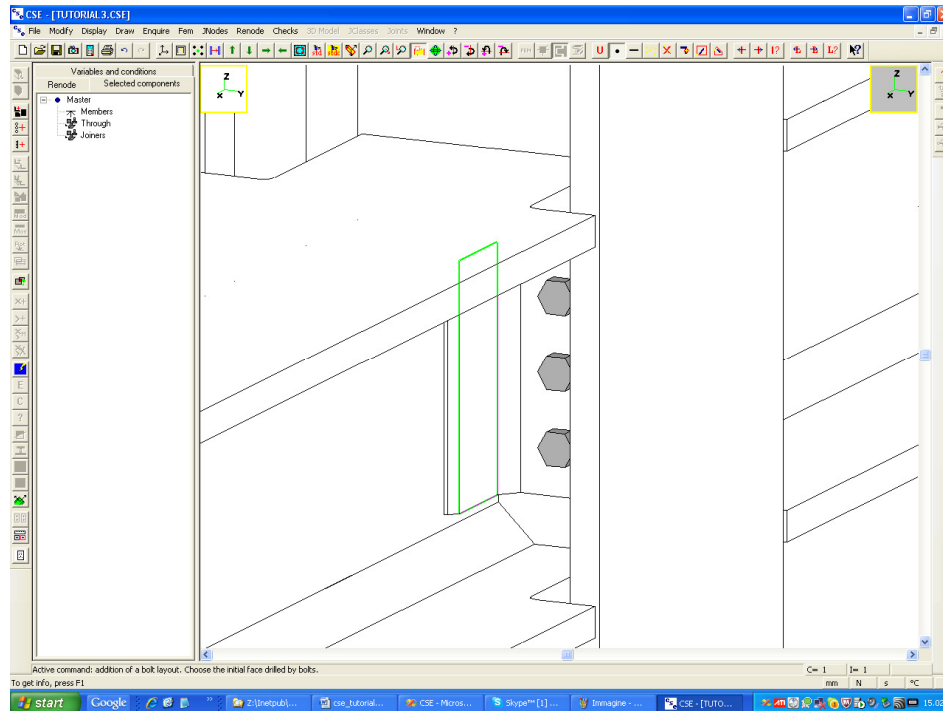


2.3.6 Join the angles to the X beams

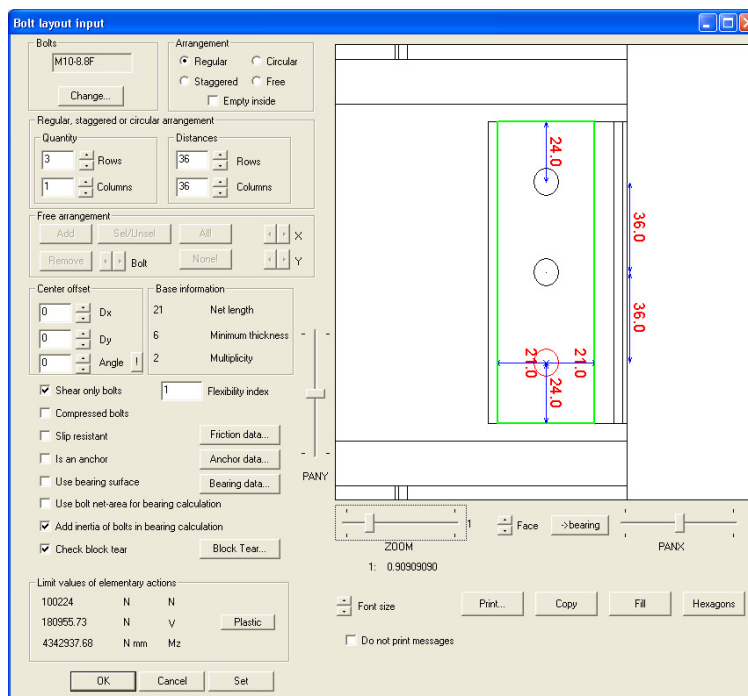
Now we must add a bolt layout joining L1 with the +X beam web. To do that we must change view. First execute **Draw-Std views** command ( button in the main bar), and press the **Isometric** button in the dialog. Then adjust the view by using the  buttons in the main bar, and zoom to get something like this:



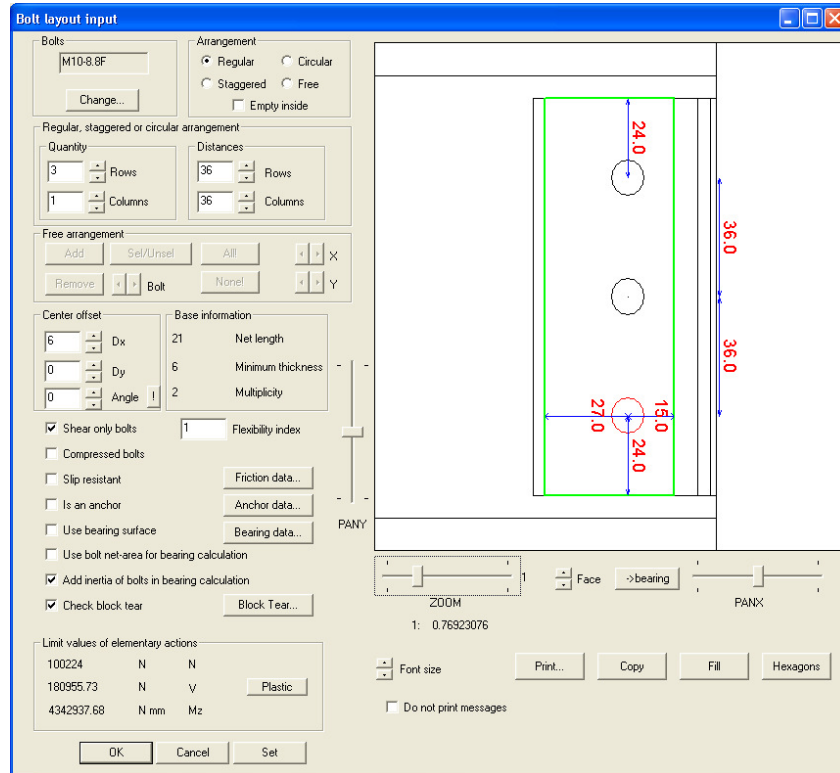
The aim is to see the outer face of the L1 leg connected to the beam web. This face will be drilled by the next bolt layout. Unselect all ( button in the main bar) so you now will browse among all scene faces. Press once more the Add bolt layout button in the left bar () and choose the proper face, like this:



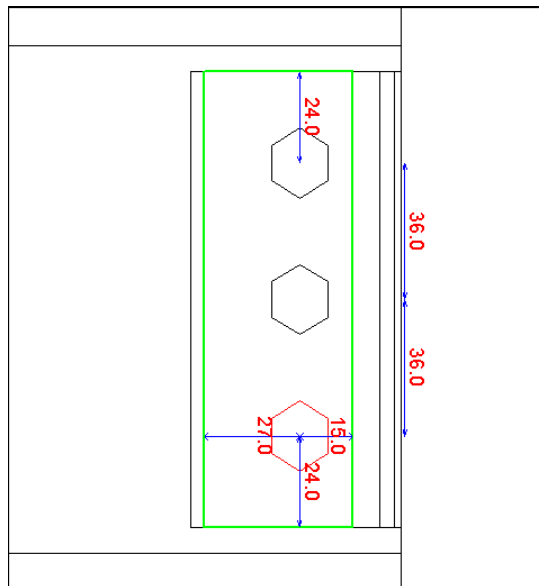
Click left. You are once more in the bolt layout addition dialog. Like this:



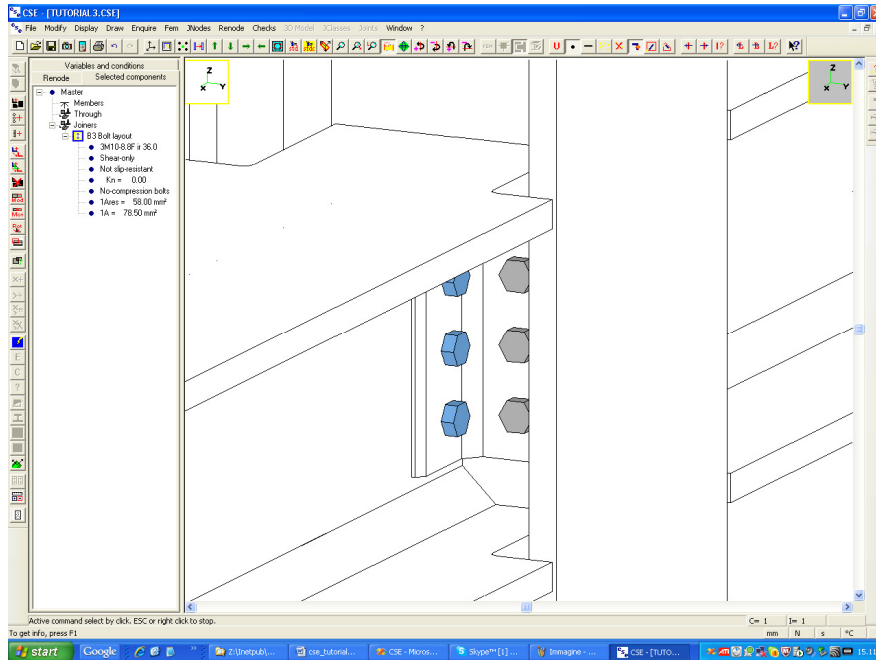
In order to reduce the lever of the shear force we will shift all bolts to the right. Press the Dx up arrow key to shift to the right until you get here:



Notice that the drawing is continuously updated. Press the Hexagon button to see if the hexagon will interfere with the fillet:

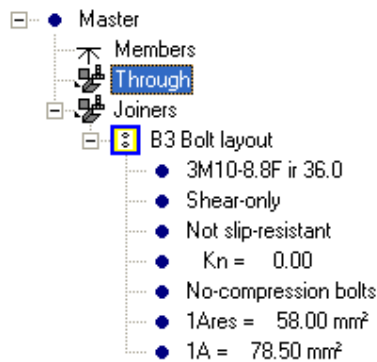



The distances are ok. Press OK and see what follows:

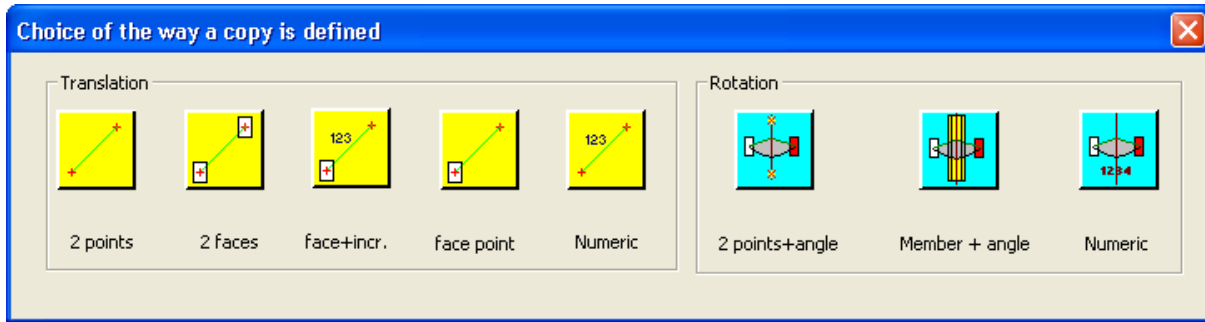


We still did not use special features which allow the positioning of each bolt or group of bolts just where you want. We are adding bolts in a standard manner. However, keep in mind that you can do what you want: the placement is free.

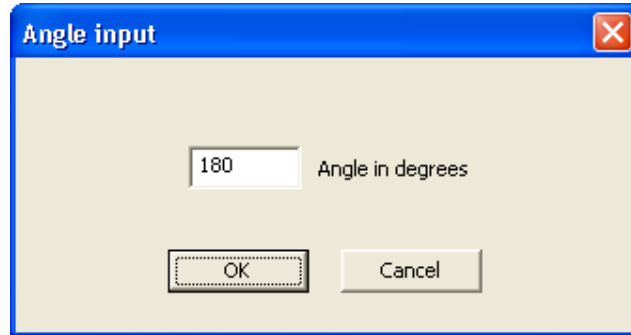
Now the connection is not finished yet: we must still add the bolt layout which joins L2 with -X beam web. We can do that copying the B3 just added. Notice that B3 is the only component selected:



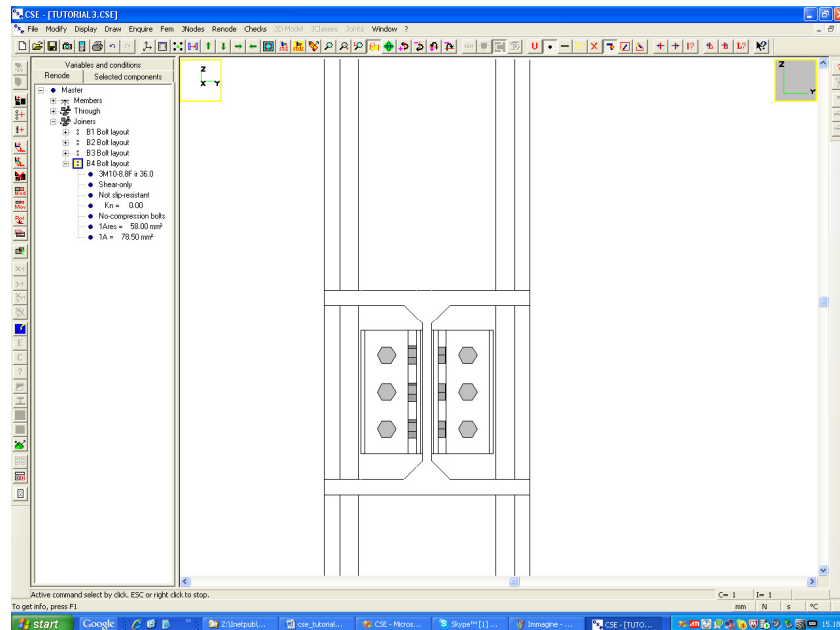
We can copy it around the column axis. Press the copy button ( button, command **Renode-Components-Copy**), press the second button from the right



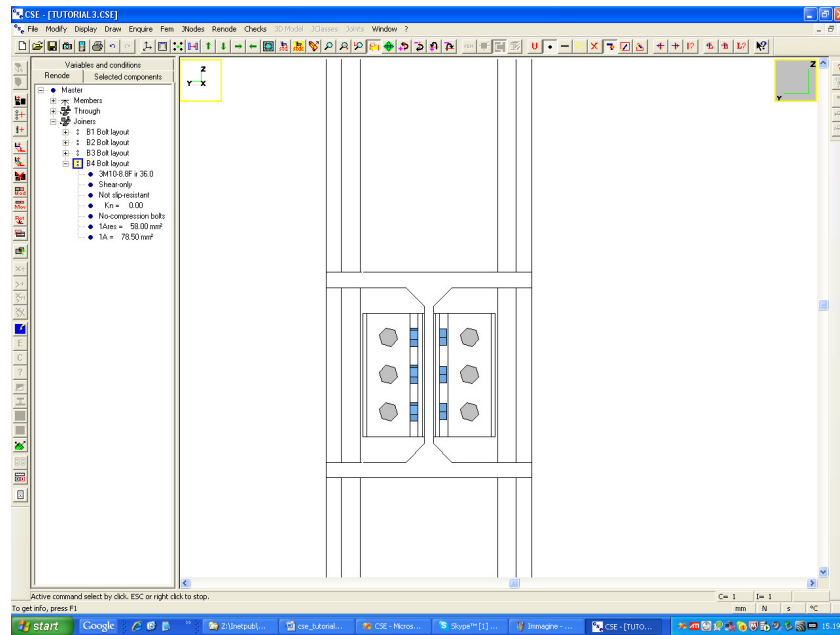
and choose a column face, then



leave 180 degrees. You hear a "click", and notice that in the left pane appears B4, selected. Now to see that everything is ok, execute command Draw-Std views, and choose +X, zooming (mouse wheel) you will see:



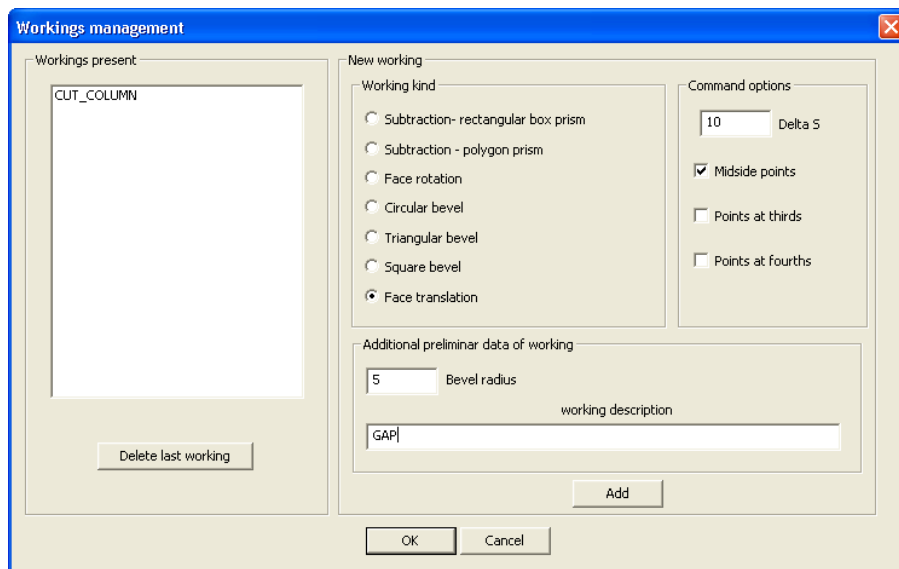
then once more **Draw-Std views** and choose -X, zooming (mouse wheel) you'll see:



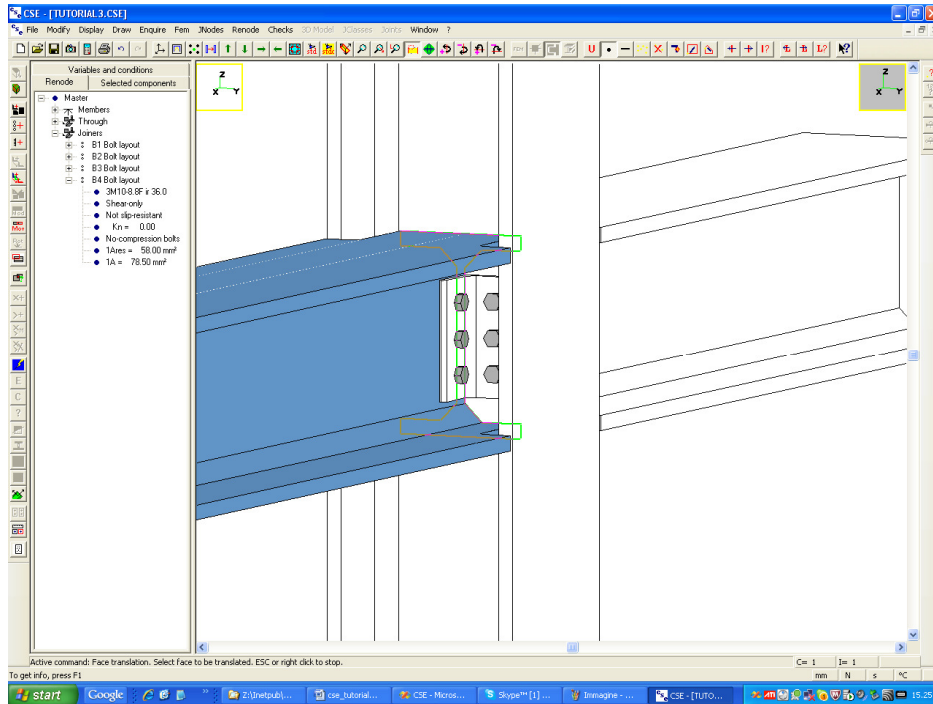
and now is clear what B4 is.

2.3.7 Adding a gap between beam end faces and the column web

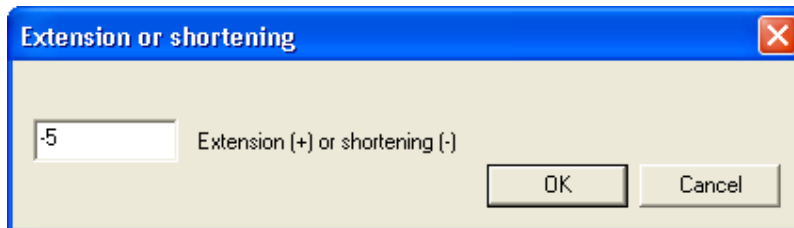
Basically we have joined the two X beams to the column. Before considering finished this part of the connection we will probably wish to shift the end faces of the beam from the column web, to avoid problems for rotation. A 5 mm gap will be added. To do that we cannot anymore apply a shortening to members, as the two beams are not prism anymore. We will shorten by shifting end faces. Unselect all and then select the +X beam. Execute the command Renode-Workings, you get here:



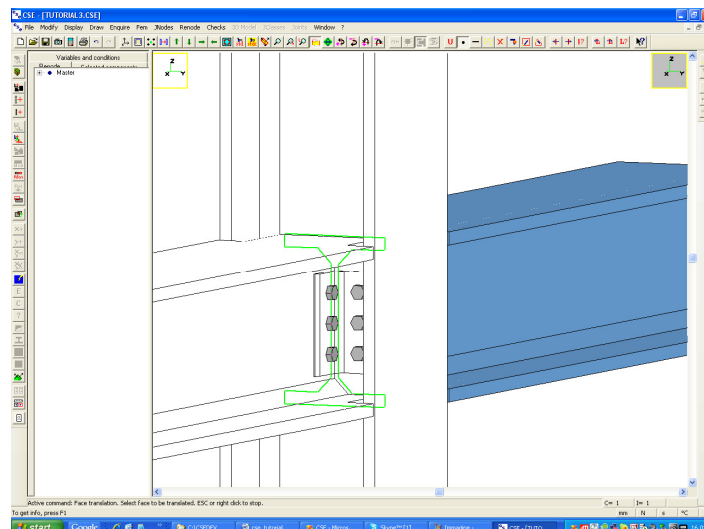
Write "GAP" as description, choose "Face translation", and then select the end face of the +X beam, the face in contact with the column web:



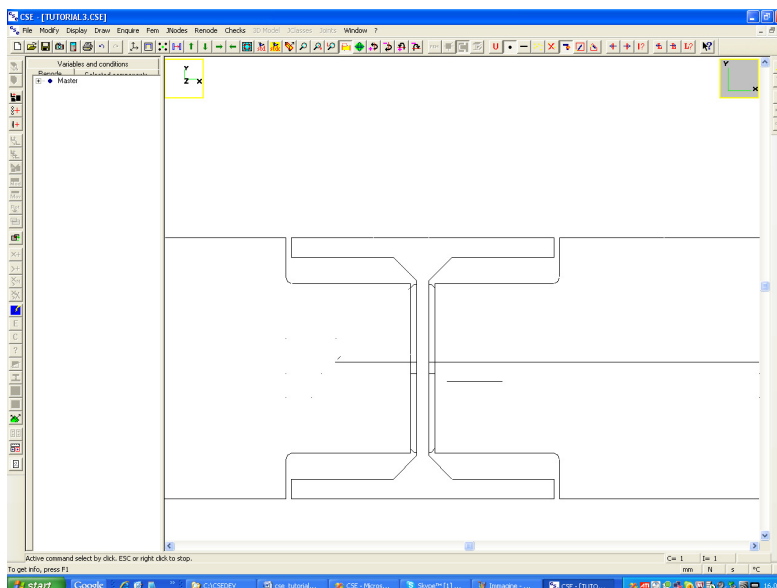
Click left. Then input -5 (mm):



You will hear a click meaning that the operation was successful. Now unselect the +X member and select the -X member. Repeat the command, and choose the end face like this:




Afterwards input -5 mm, as done previously. Now change the view to +Z and zoom, you will see what follows:




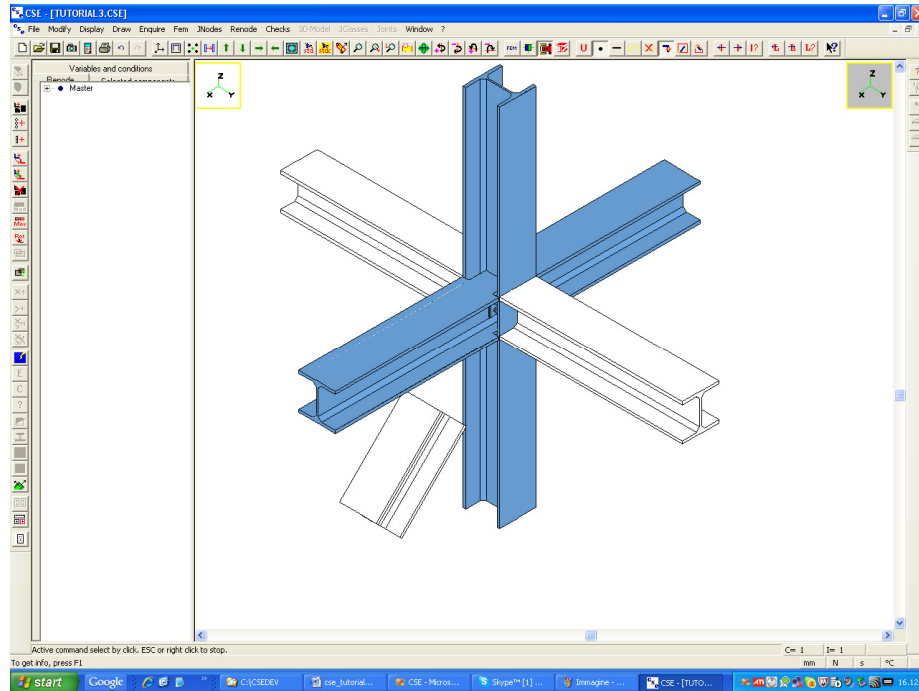
There is a gap as necessary between the end faces and the column web.

2.3.8 Check for overlaps

Now switch to Iso view and execute the command **Renode-Check overlaps!** (the  button in the left toolbar). As you are still in extraction mode, the check will only be done *only considering the extracted objects*. The following message informs you that anything is OK.



You can now exit from extraction mode, by un pressing the  button in the main bar (**Draw-Extract** command). You will see



We are now ready to joint Y beams. This will be done in part 2 of this tutorial.



1	<u>INTRODUCTION.....</u>	3
2	<u>HOW TO BUILD A MULTIPLE MEMBERS RENODE.....</u>	4
2.1	STEP 0: SELECTING A JNODE AND PASSING TO RENODE VIEW.....	4
2.2	STEP 2: CHOOSE THE STANDARD	5
2.3	STEP 3: THE HINGED BEAMS (X BEAMS) CONNECTIONS.....	5
2.3.1	SHORTEN THE DIAGONALS	5
2.3.2	SHORTEN THE X BEAMS	8
2.3.3	SOLVE THE X BEAMS-COLUMN OVERLAPS	11
2.3.4	APPLY DOUBLE ANGLES.....	23
2.3.5	JOIN THE ANGLES TO THE COLUMN, HAVE A LOOK AT BOLT LAYOUT ADDITION DIALOG	30
2.3.6	JOIN THE ANGLES TO THE X BEAMS	39
2.3.7	ADDING A GAP BETWEEN BEAM END FACES AND THE COLUMN WEB	44
2.3.8	CHECK FOR OVERLAPS.....	46