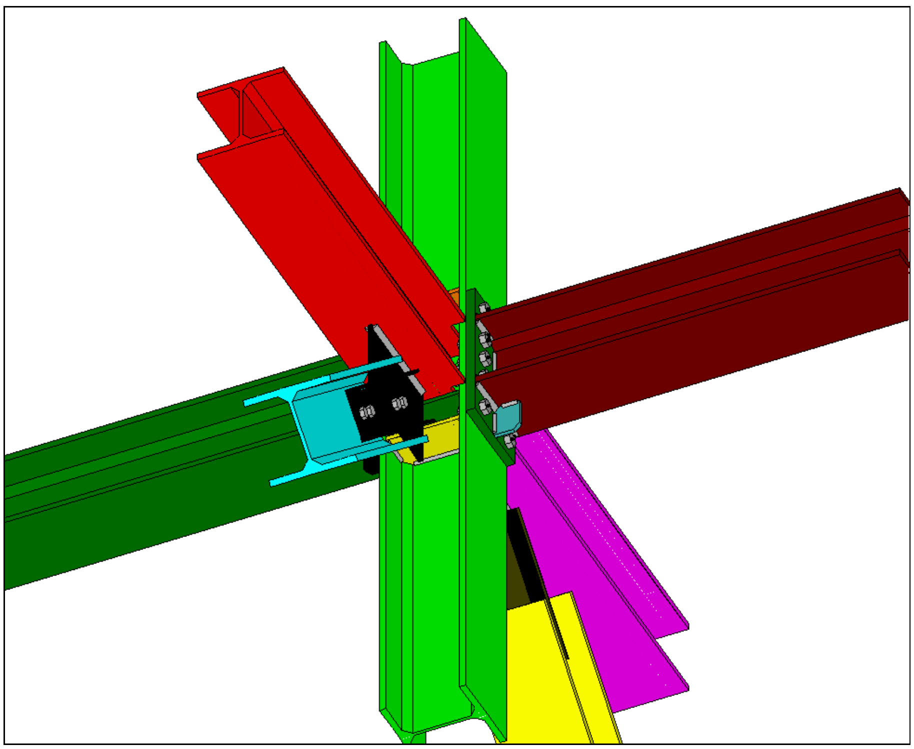
Paolo Rugarli

Connection Study Environment

Tutorial 3: building a multiple-members renode. Part 2



http://www.castaliaweb.com - http://www.steelchecks.com
Via Pinturicchio, 24
20133 Milan - Italy
staff@castaliaweb.com

Referring to CSE Version 4.40
Rev.4 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 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

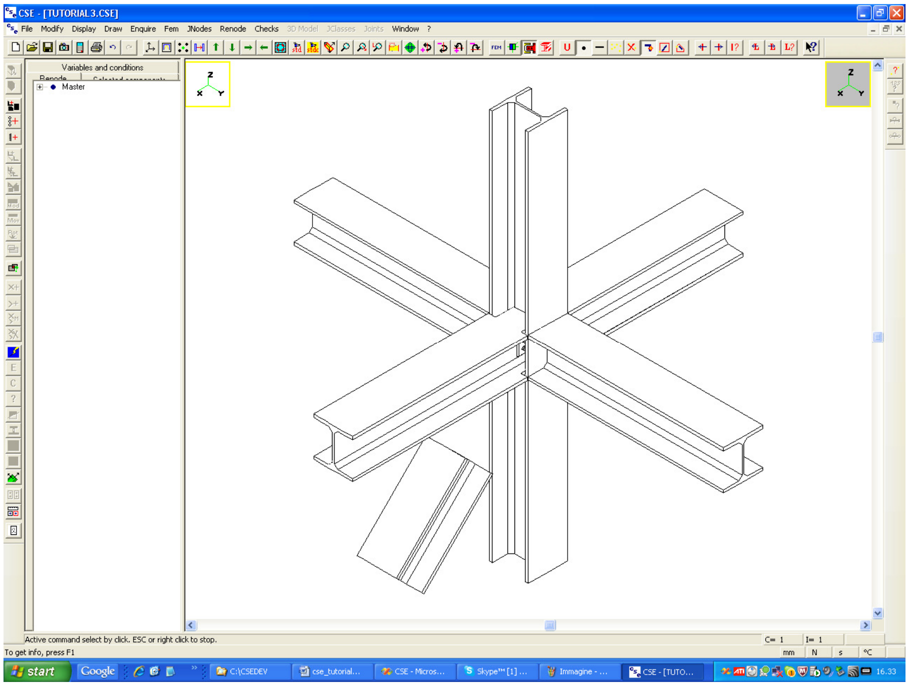
Running the checks and discussing results will be done in another tutorial.

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

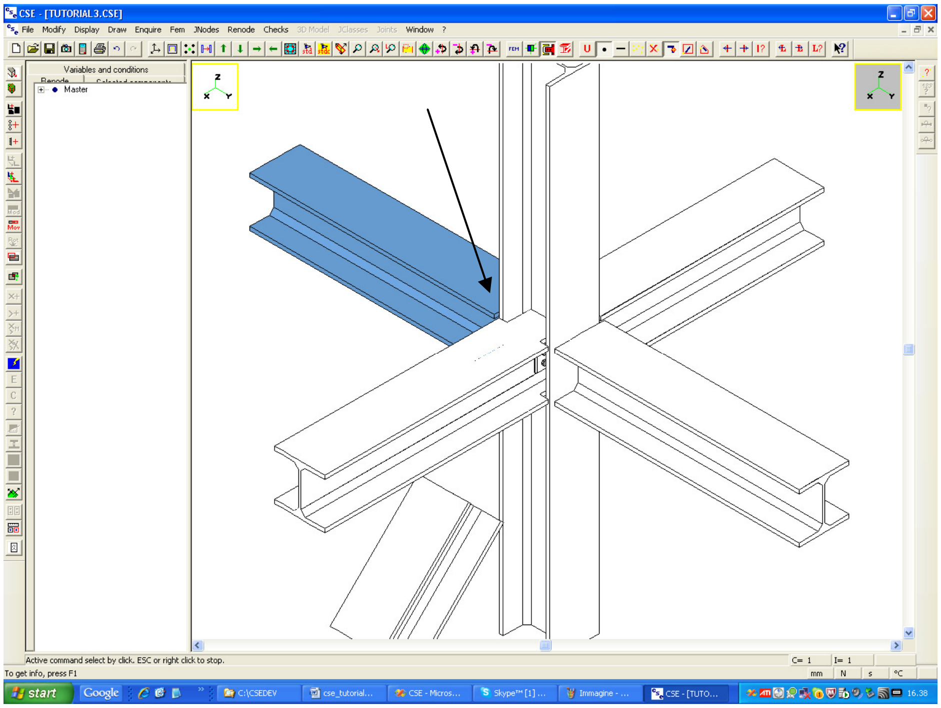


Initial windows content: renode view at the end of tutorial 003a.

2.1 STEP 4: JOINING THE Y BEAMS TO THE COLUMN

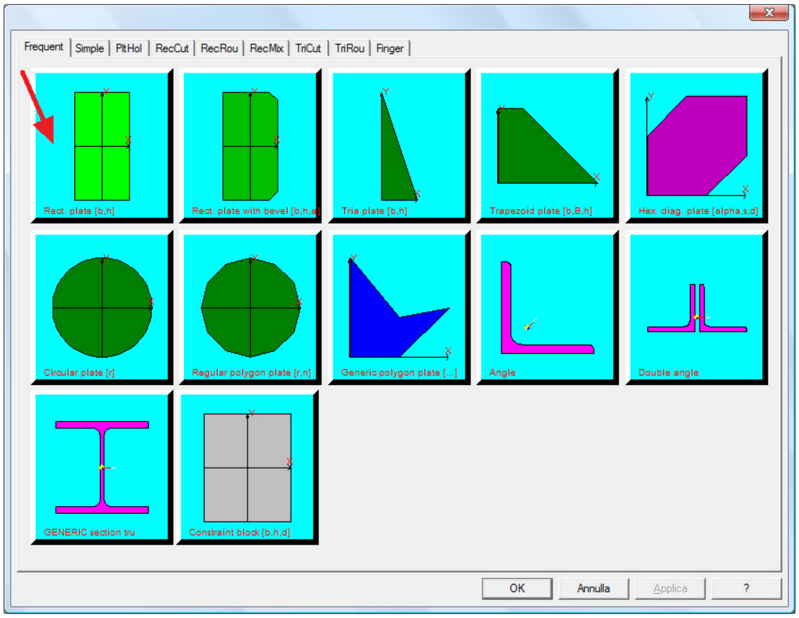
2.1.1 Shorten the members

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.

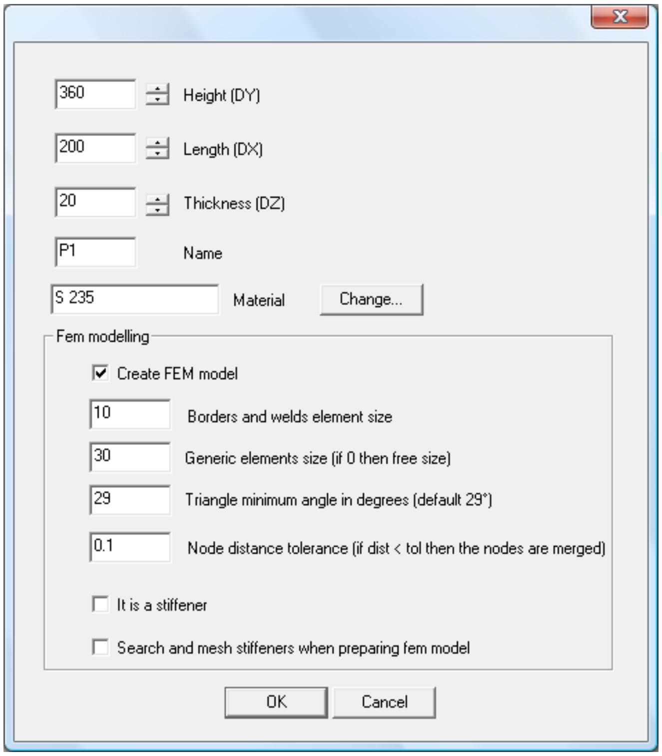


2.1.2 Add end plate

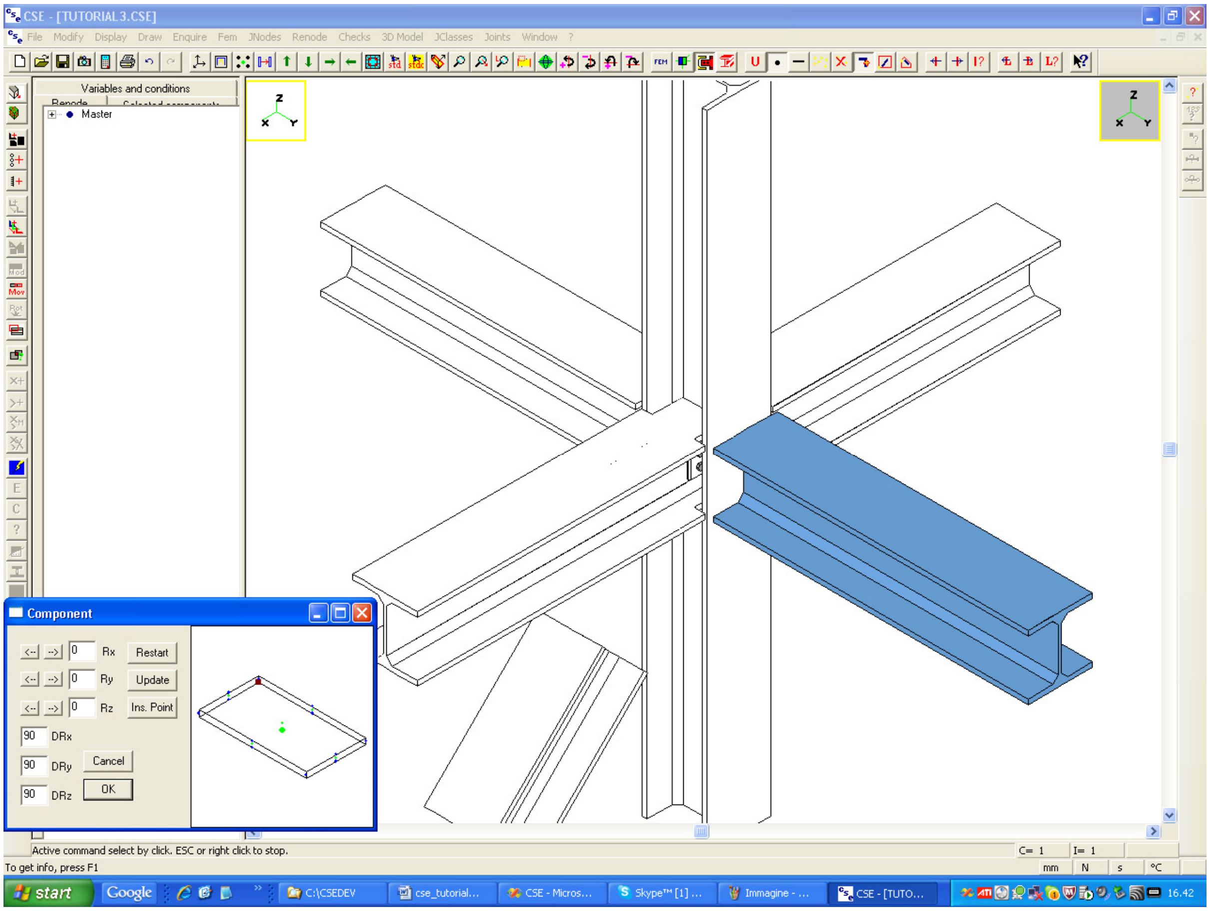
Now select +Y beam only and execute the command **Renode-Components-Add through** (button in the left bar). Choose **Rectangular Plate**:



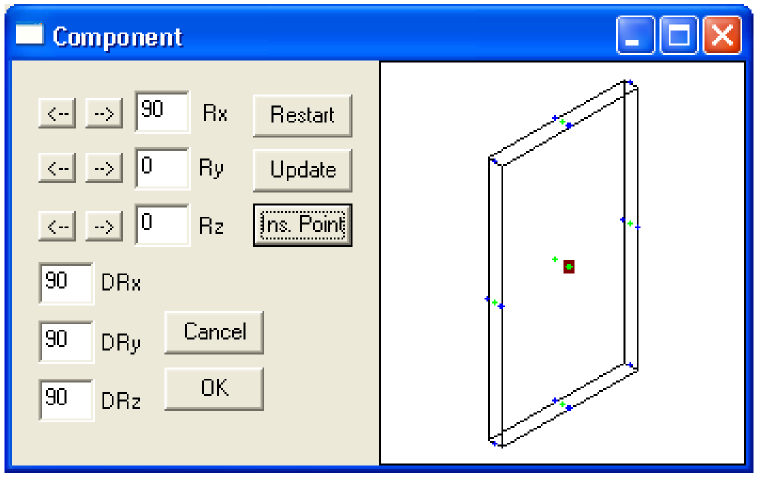
You are now in this dialog, choose the parameters as shown:



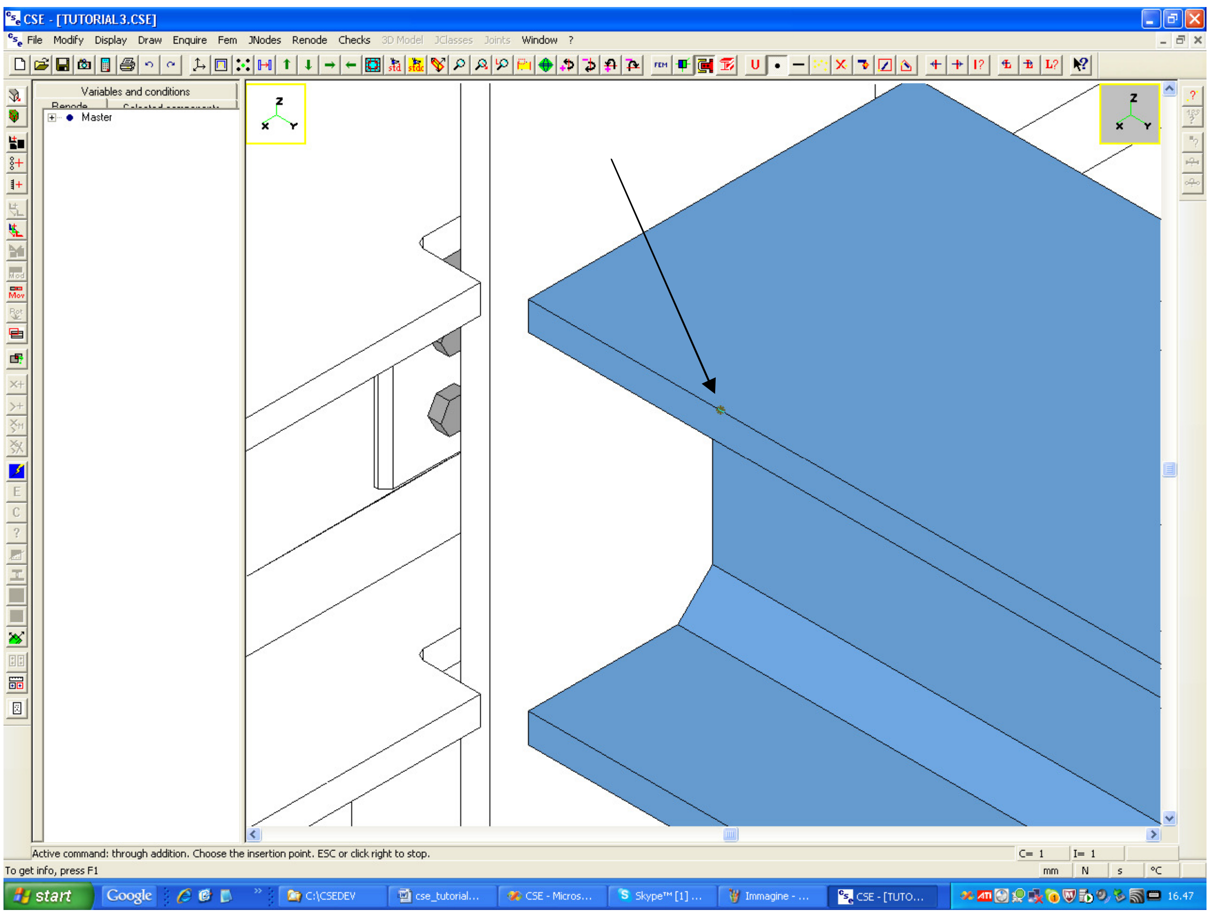
Press **OK** you are here:



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



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:

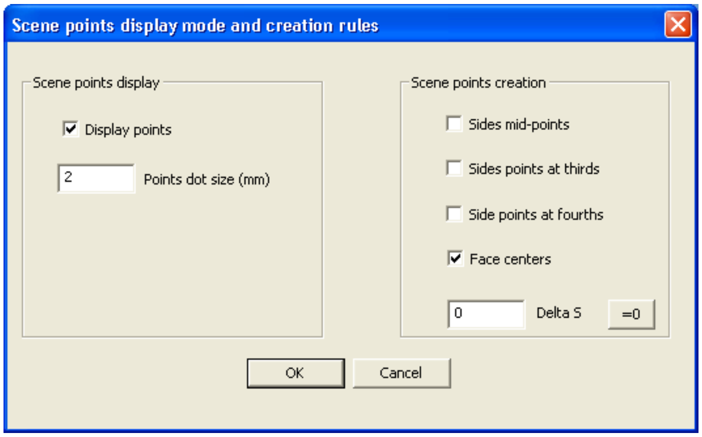
1) Choose a simpler view to get the centre (maybe a lower viewpoint may help);

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

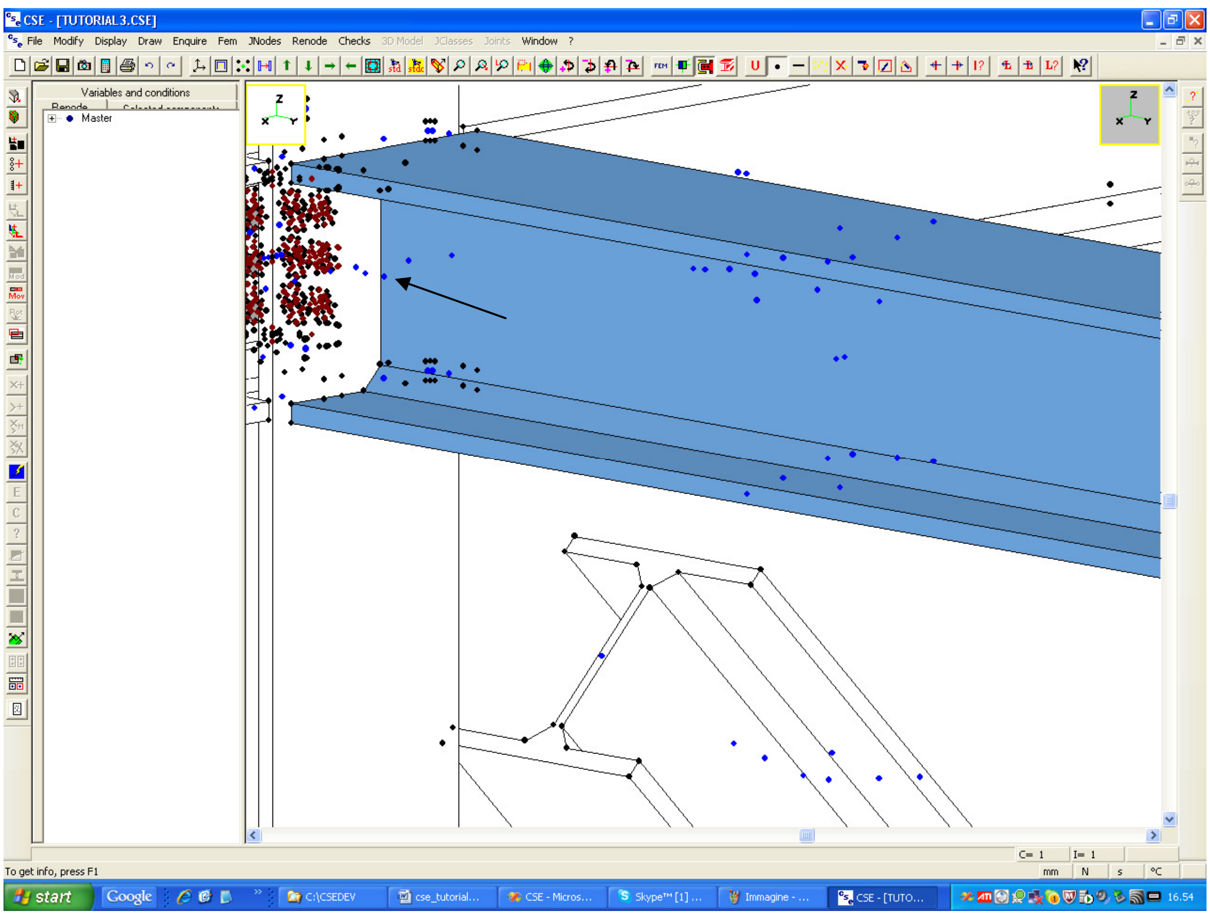
3) Zoom In using the mouse wheel.

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

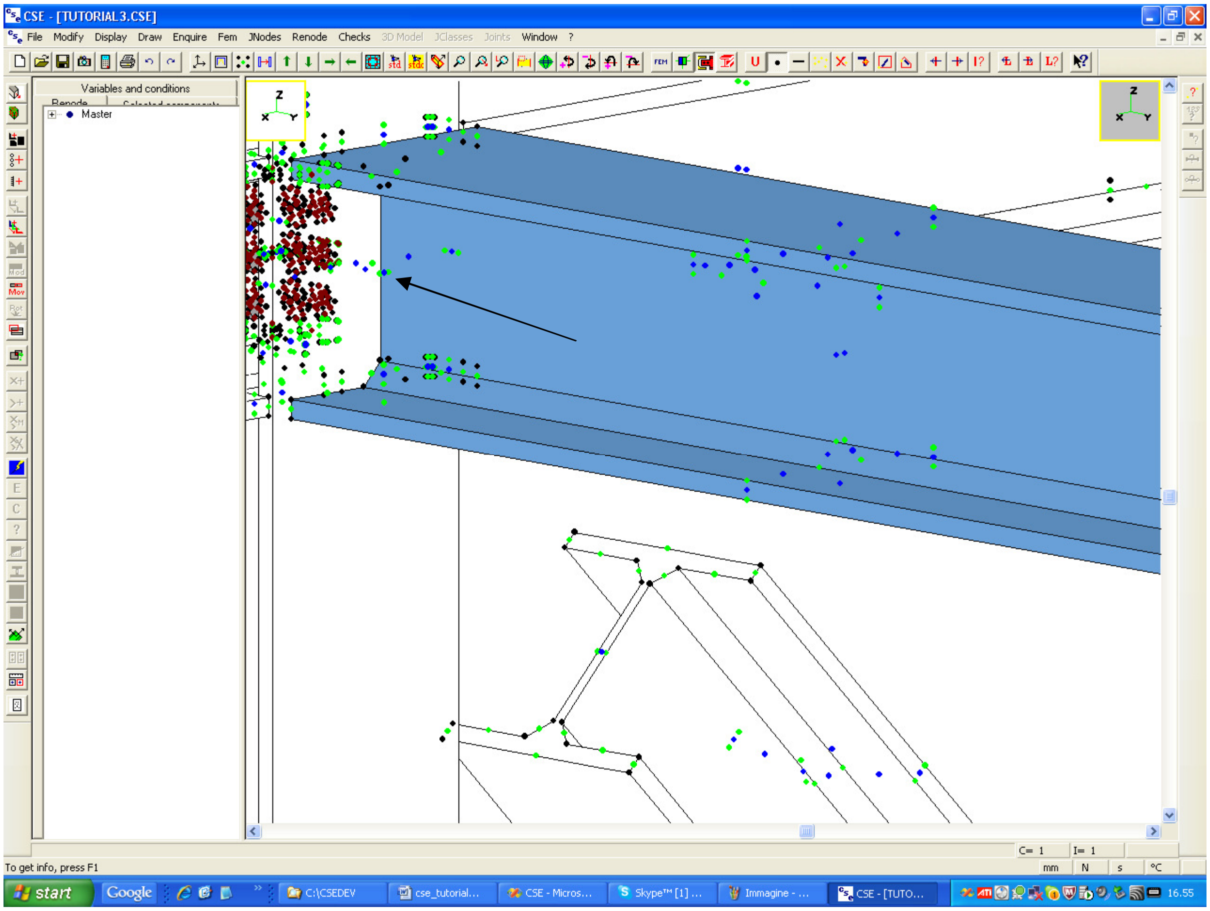
For instance this is how it would appear the scene with this choices for the scene points



and a lower viewpoint:

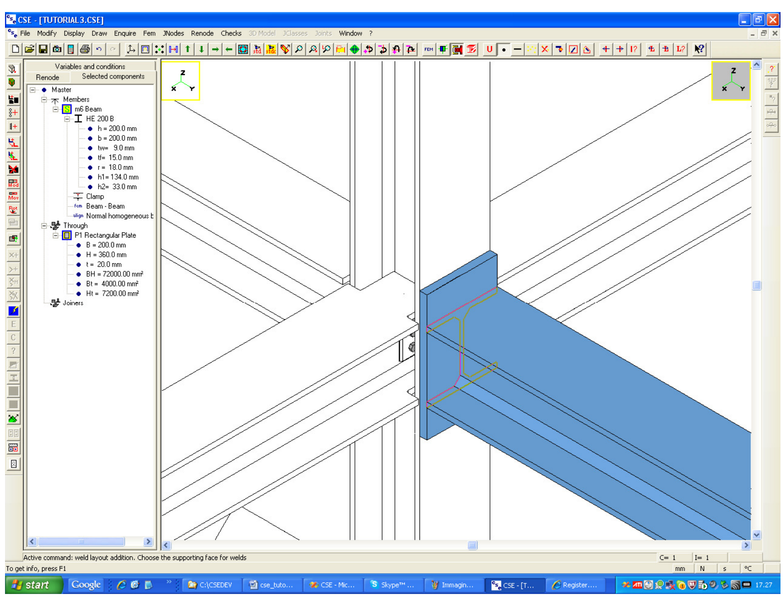


Notice that black points are segment extremities, while blue points are face centres. The face centre is now pointed by the row and is clearly visible. With also the mid segment points we would get the following:



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):



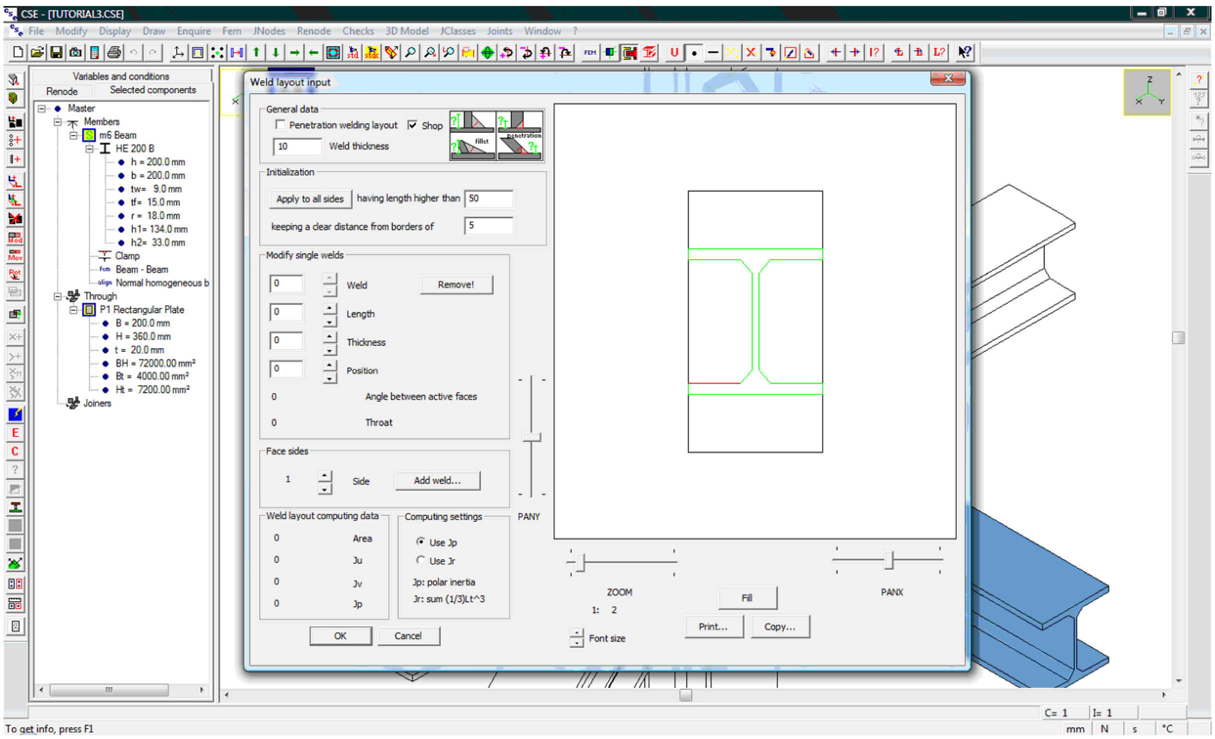
2.1.3 Join the beam to the end plate

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 (button in the left bar). You can now select a face to be welded. Choose the end face of the +Y beam, like this:

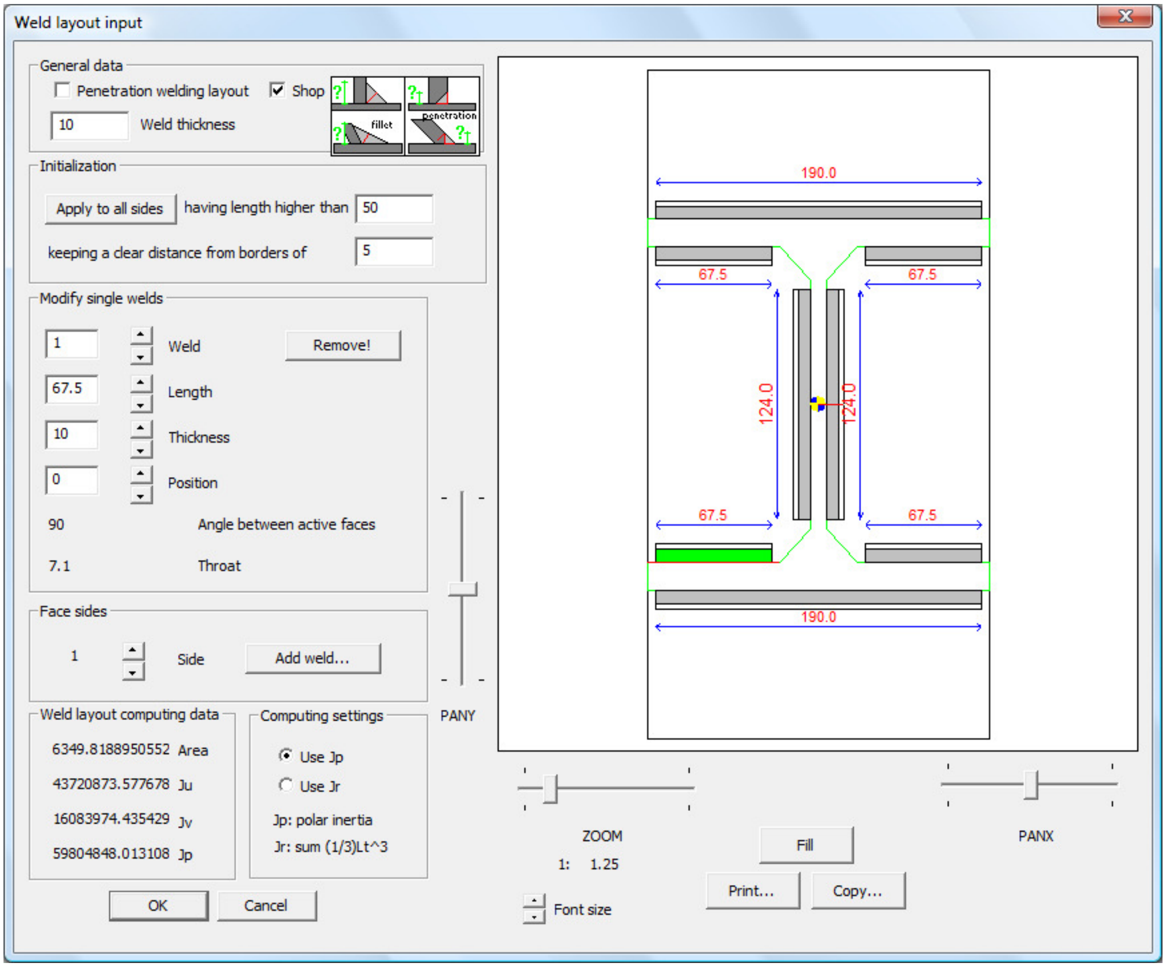
Click left. You get into the weld layout addition dialog:



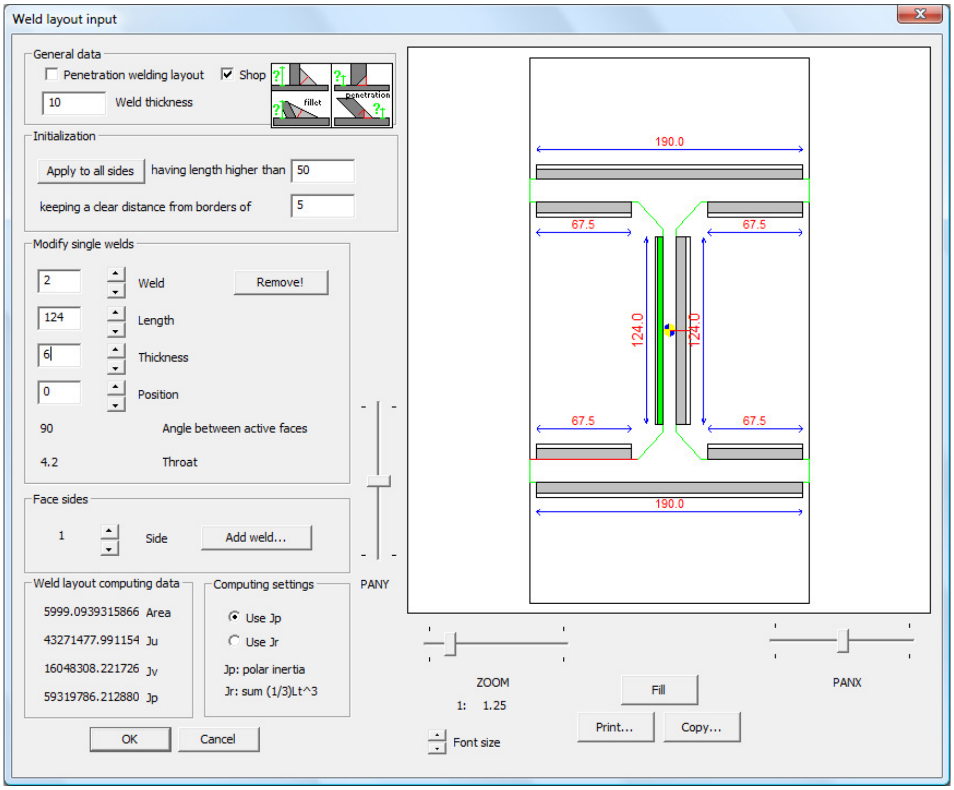
Do NOT tick “penetration welding layout” in order to use fillet welds.

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.

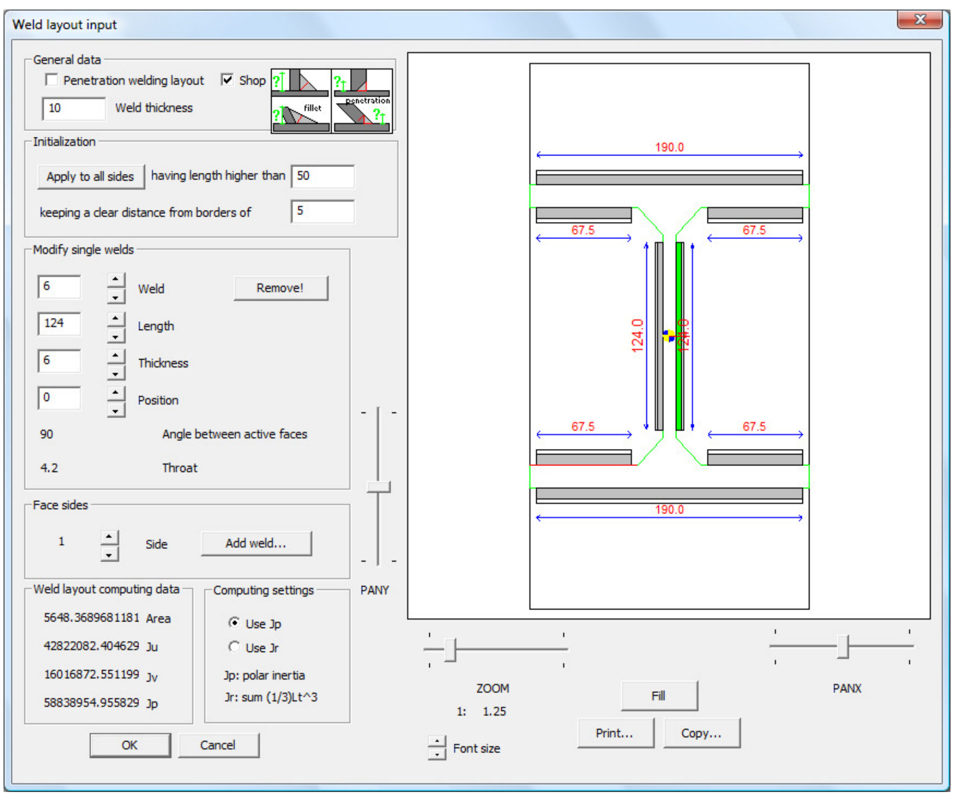
You now get here:



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:

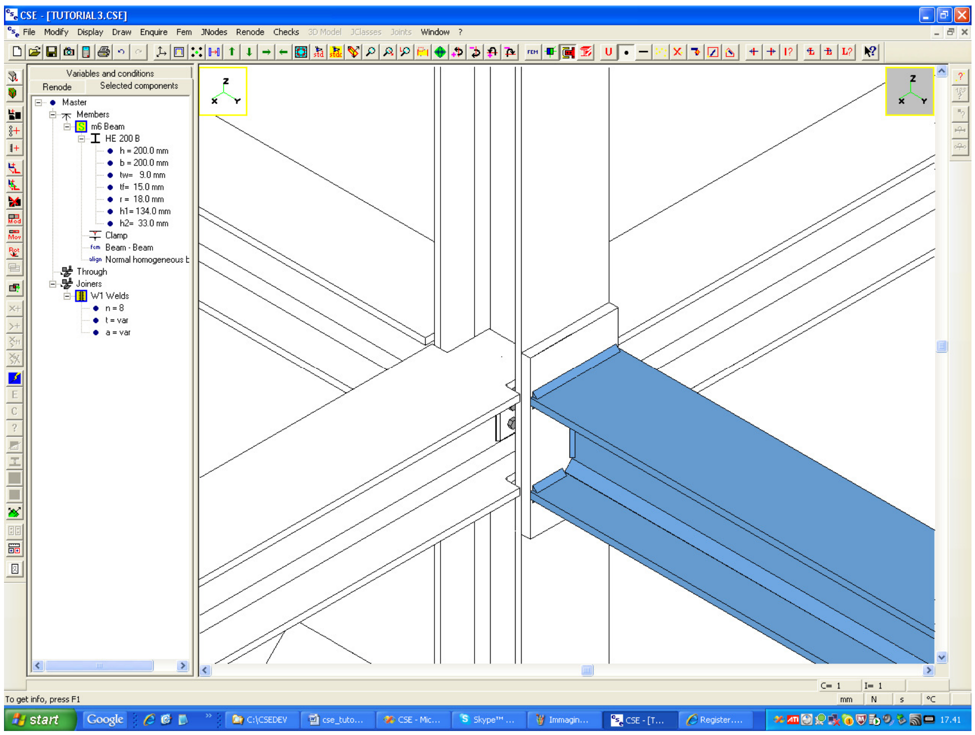


Now select the other fillet and do the same, like this:

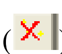


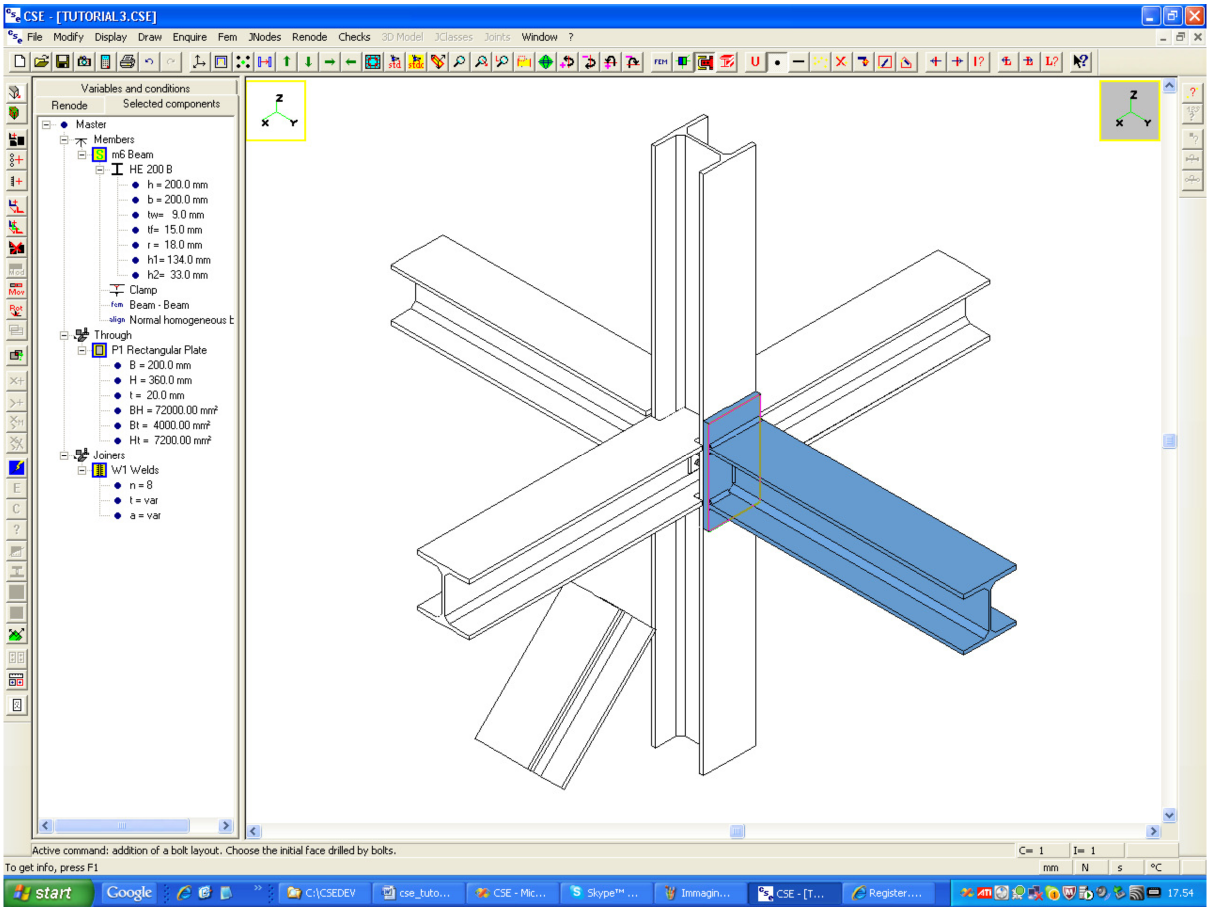
Notice that the moment of inertia of the weld layout (Ju, Jv, Jp, where u and v are the principal axes) are printed in the read only edit boxes at the bottom left of the dialog. Notice that the program understood the angle between the faces in contact (**angle between active faces**). Also notice that 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:

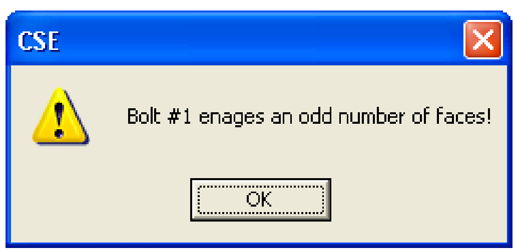


2.1.4 Join the end plate to the column

Now unselect all (), 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:

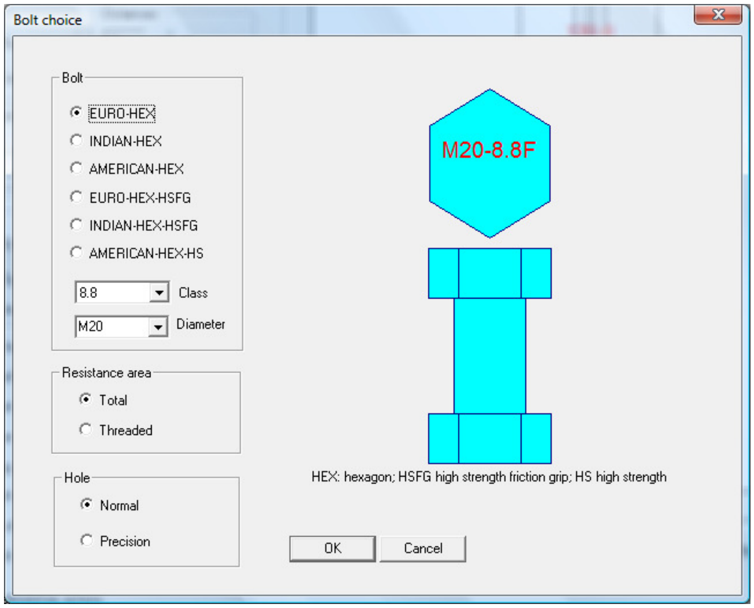


Click left. You will get this message:



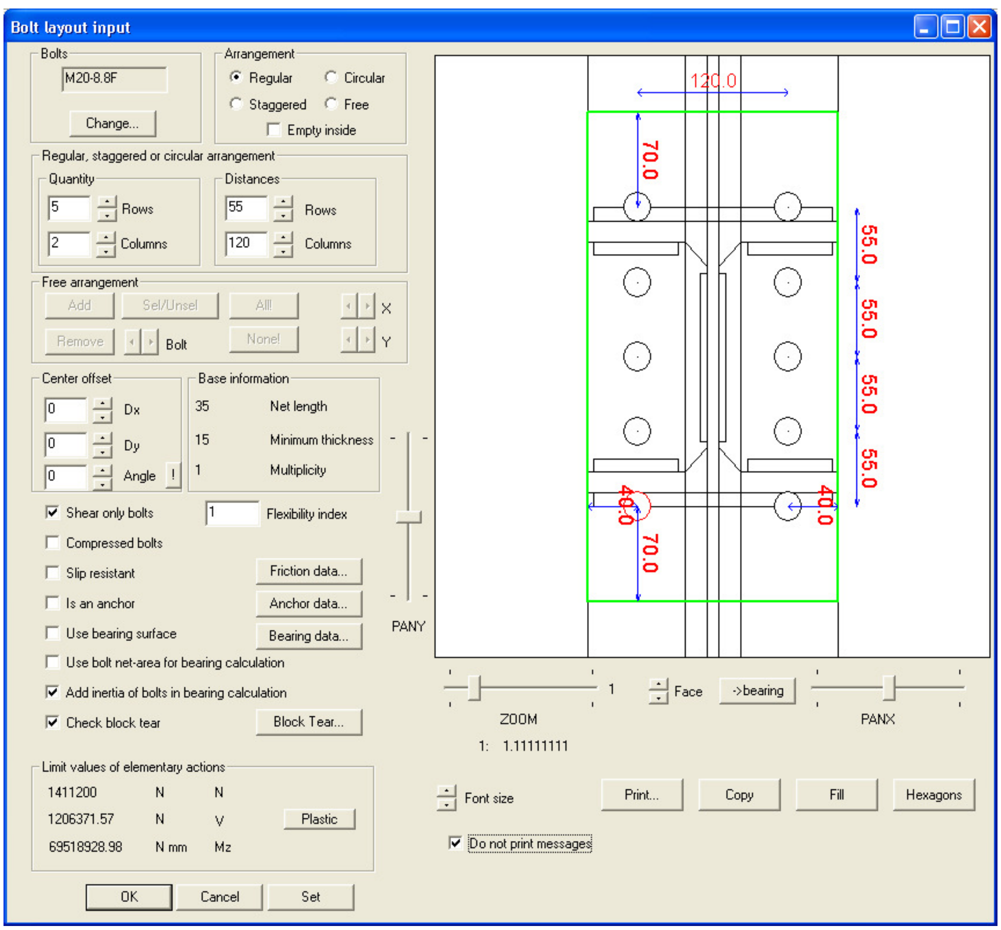
don't worry: the bolt layout has been initialized with a one bolt bolt-layout which is ill positioned. If you are still working in the same session of the previous tutorial part, you will see a different bolt layout initialisation, basically with the same data of the already defined B1, B2, B3 and B4.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

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

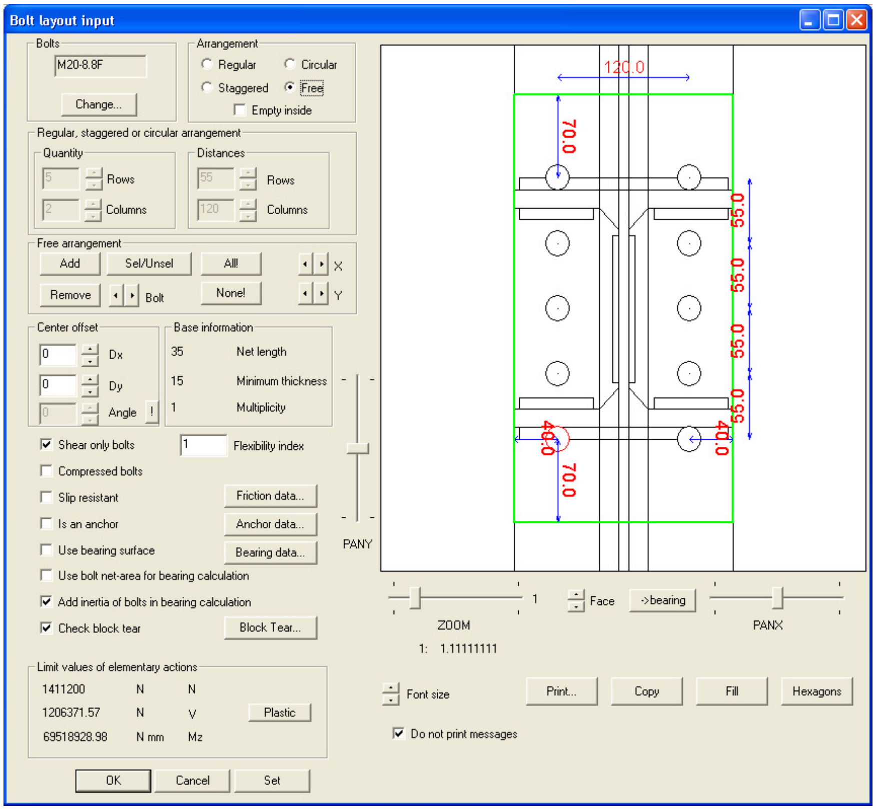


Press OK, you are back in the bolt layout addition dialog. Place a tick in the **Do not print messages** check box to avoid possible messages breaking down your actions.

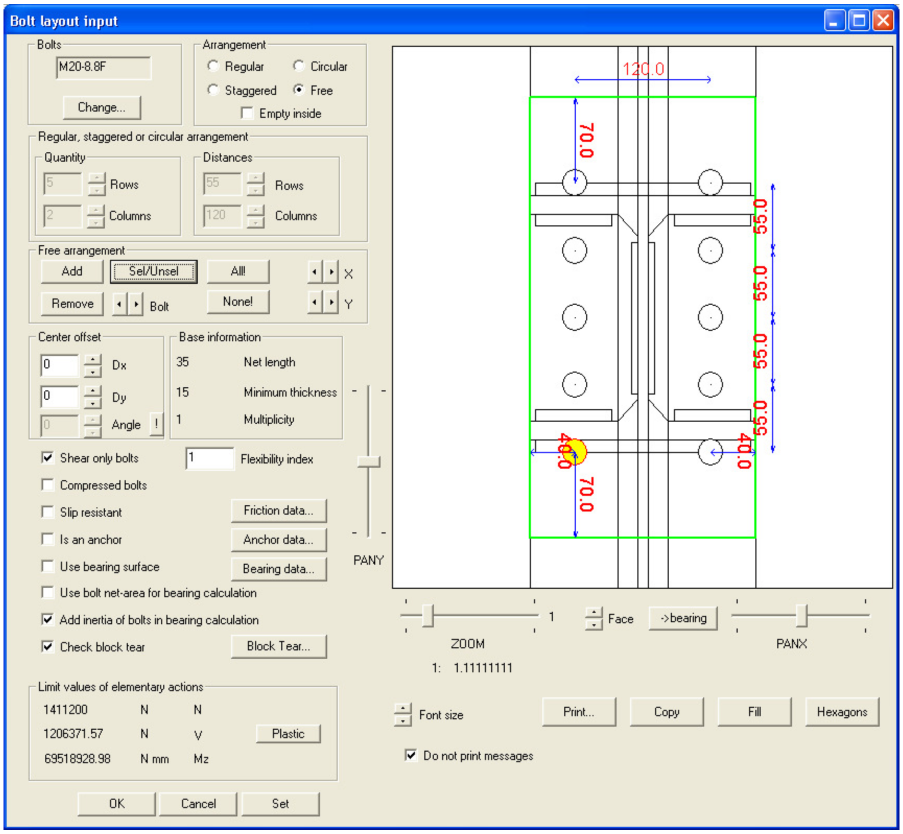
Select 5 rows and 2 columns. Specify 55 mm as Row distance and 120mm as column distances. You are here:



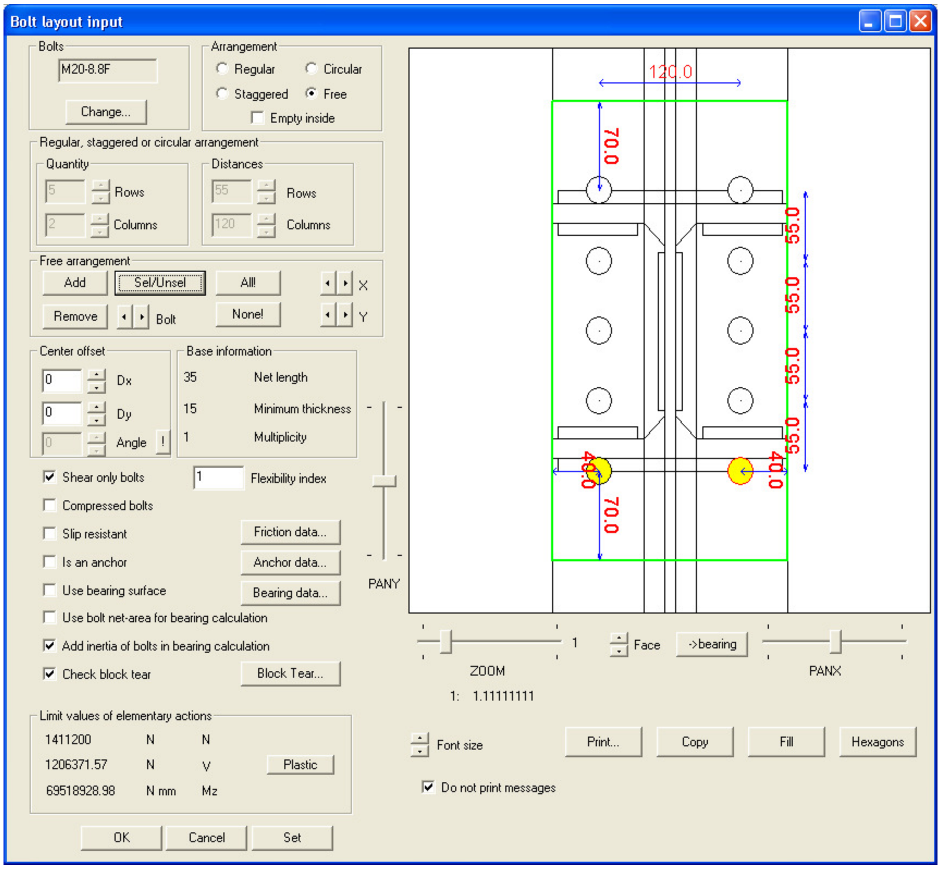
The web bolts are ok but the first and last column will have to be shifted. Let's switch to a free Arrangement, by clicking **Free**. The dialog changes like this:



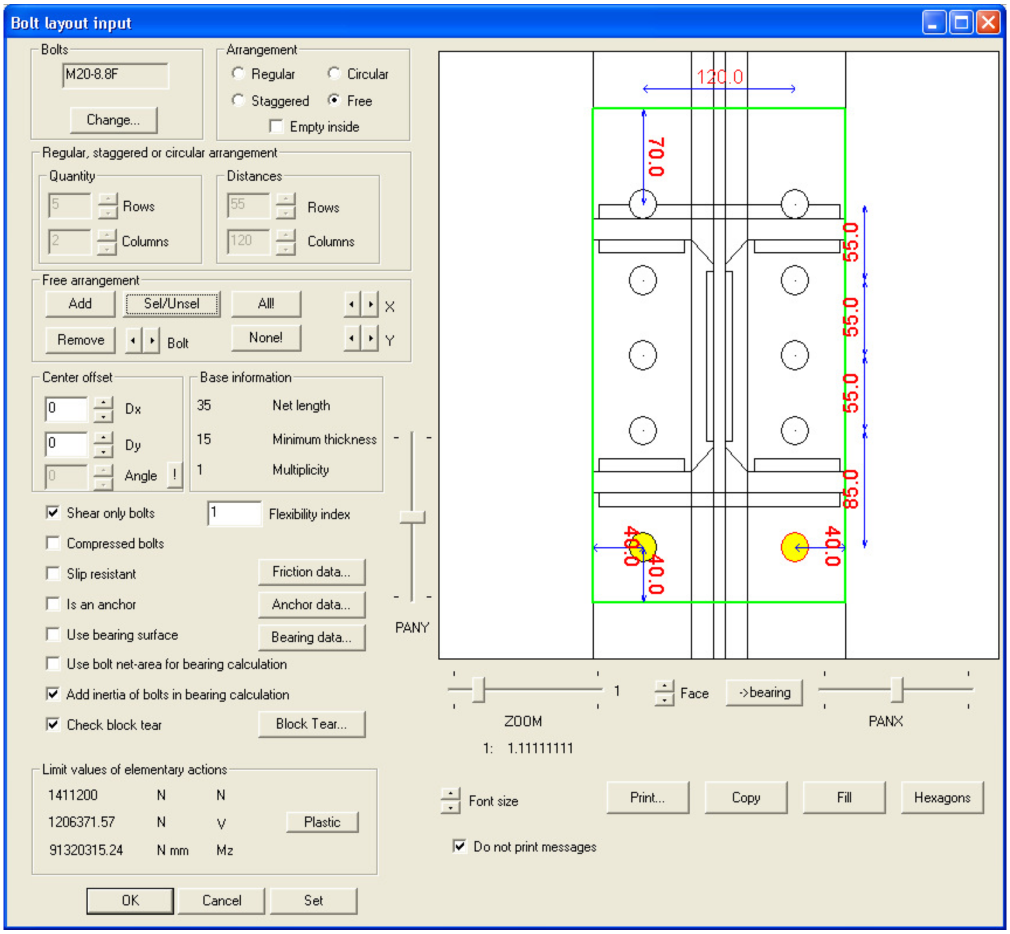
Now the red circle bolt hole is the current bolt. Press the Sel/Unsel button, it gets yellow, like this:



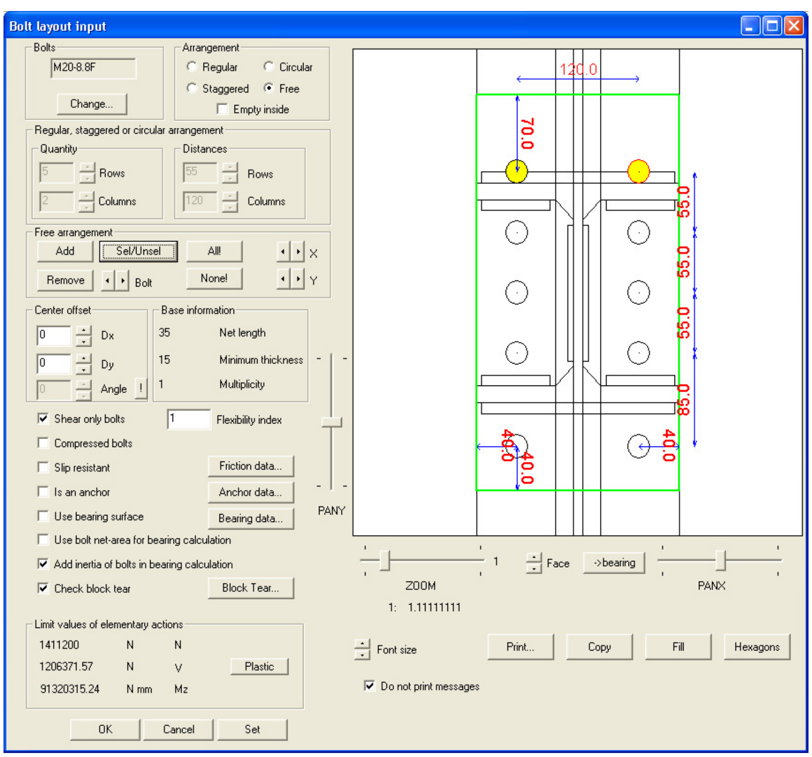
Now use the Bolt arrow keys to get red the right bolt in the same row, and then press Sel/Unsel, you get here:



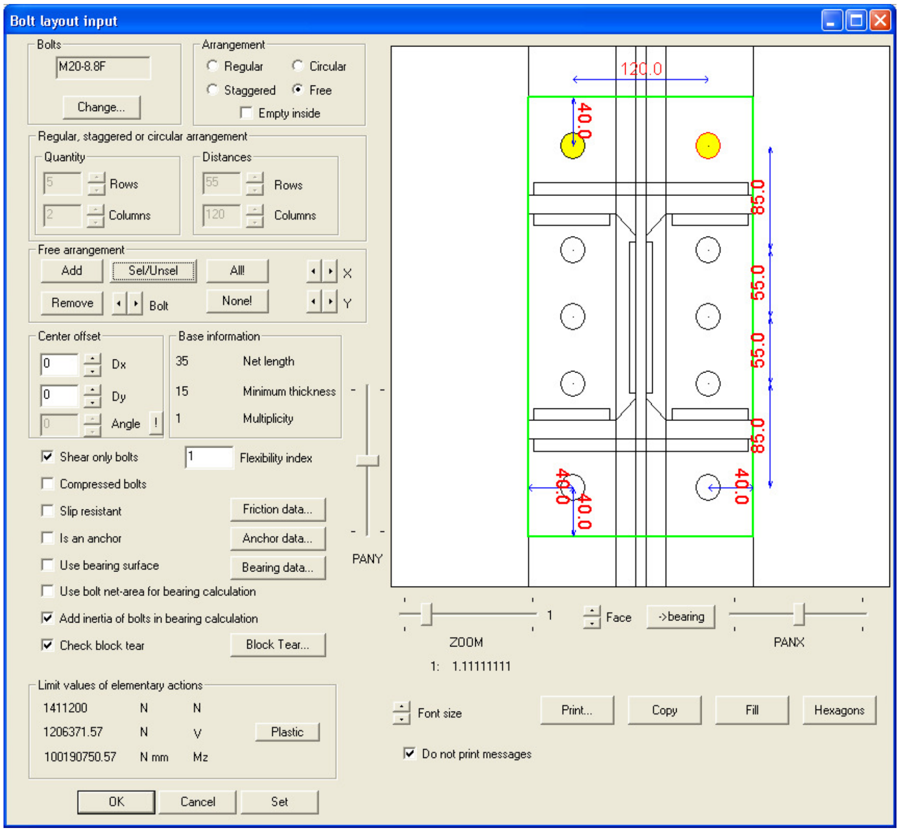
Now using the Left arrow in the Y control, shift the two selected nodes downward, until the original 70mm distance from the lower border of the plate gets equal to 40mm. Like this:



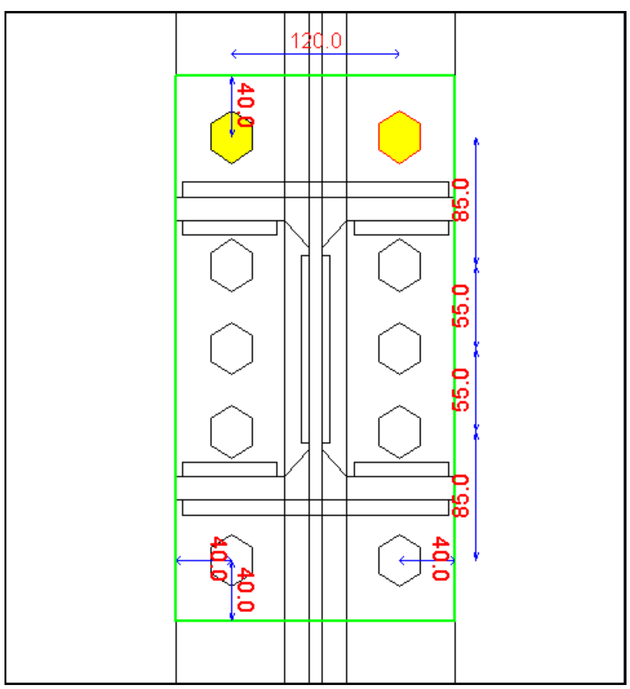
Now press the Sel/Unsel button and unselect the right bolt in the last row. Go back to the left bolt in the same row and unselect it. Then select the two bolts of the topmost row, like this:



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:

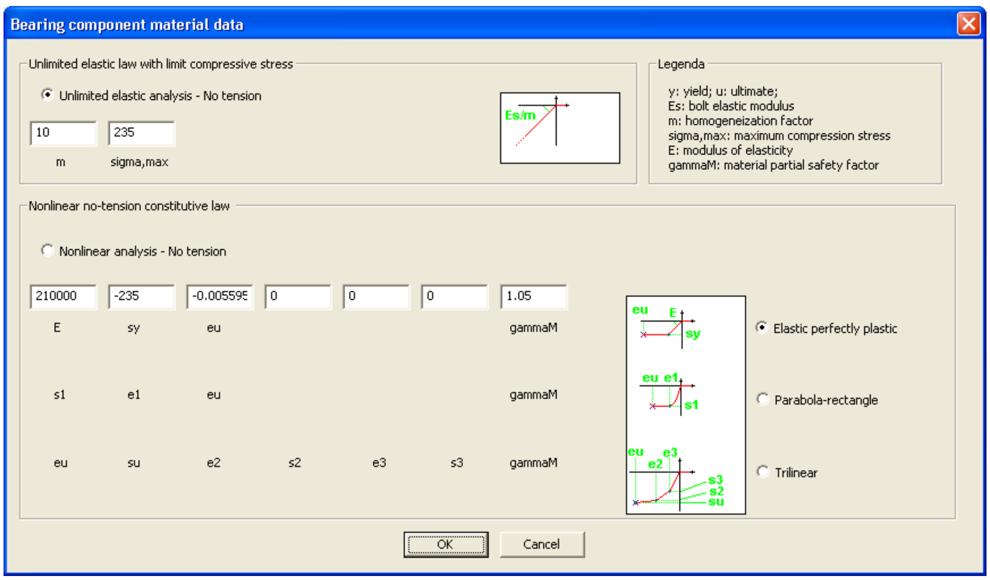


The hexagons do not touch the fillets. It's ok. This image was pasted after having pressed the **Copy** button in the dialog. You can do as well for your docs.

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:



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/mm^2/100=2060N/mm^2). 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:

$$M= pBL^2/2$$

$$W = Bt^2/6$$

$$M/W < 235 \text{ N/mm}^2.$$

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

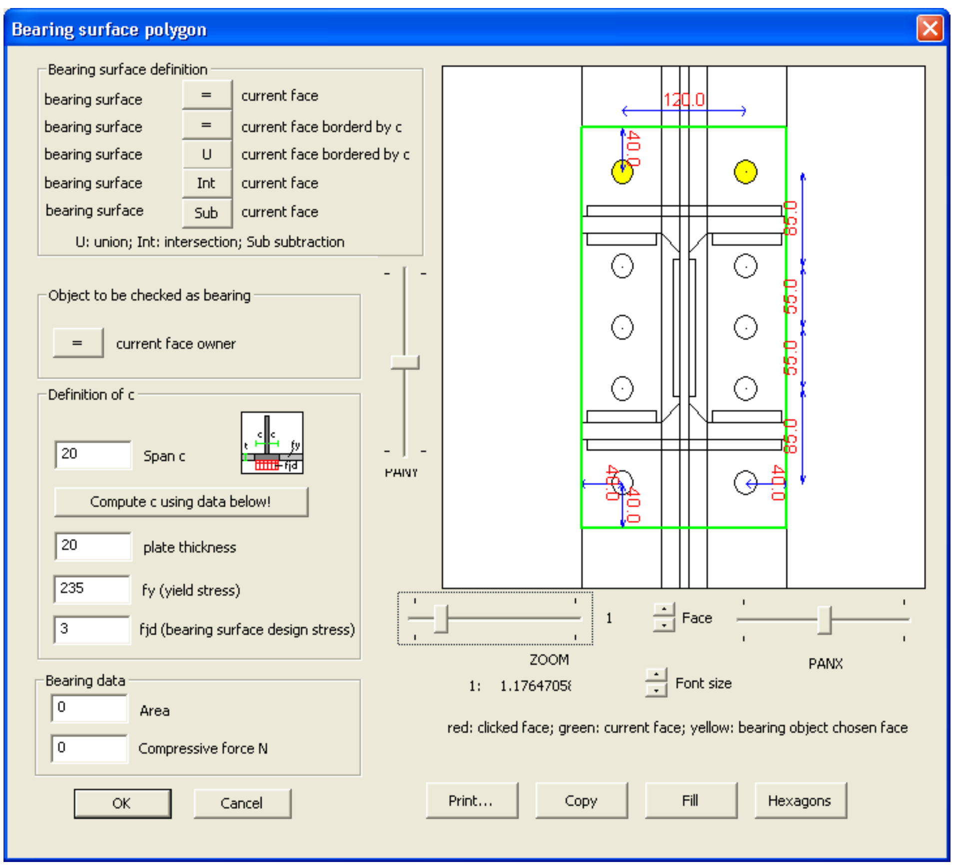
(R fillet radius for HEB200, t_w web thickness)

so

$$p < 235\ 15^2/(3\ 77.5^2)=2.93 \text{ N/mm}^2$$

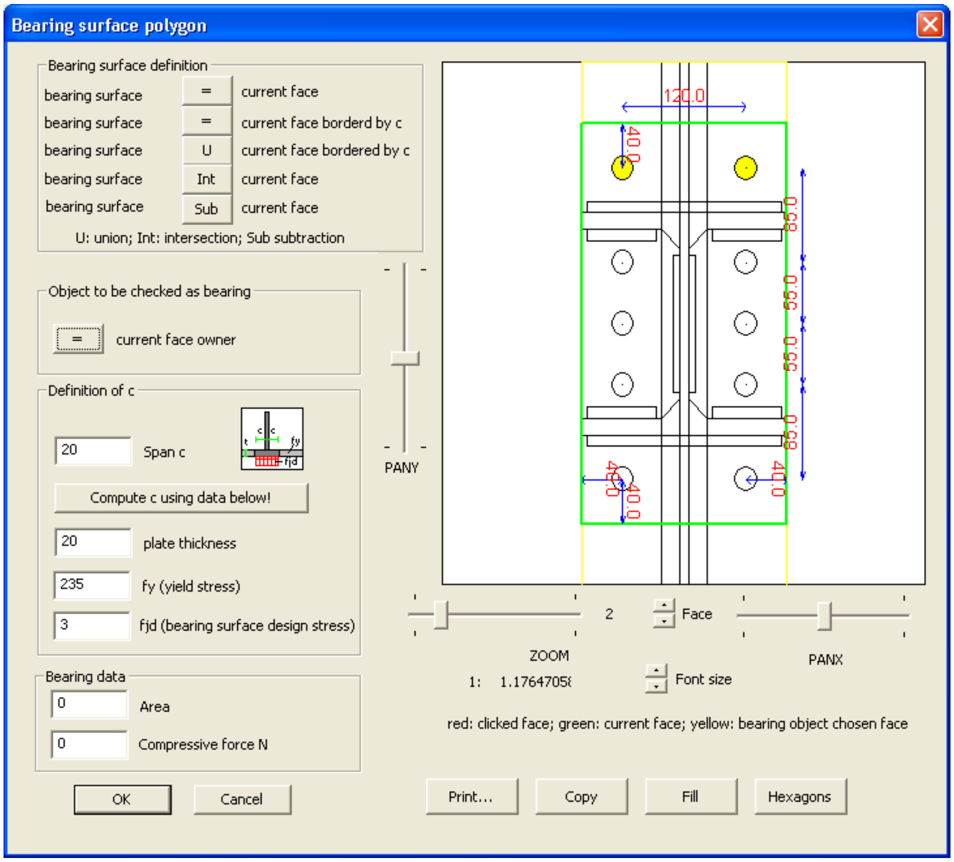
This is a simplified computation, but gives order of magnitude. Input 3 (N/mm^2) as limit stress for the bearing object (**Sigma max**).

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:

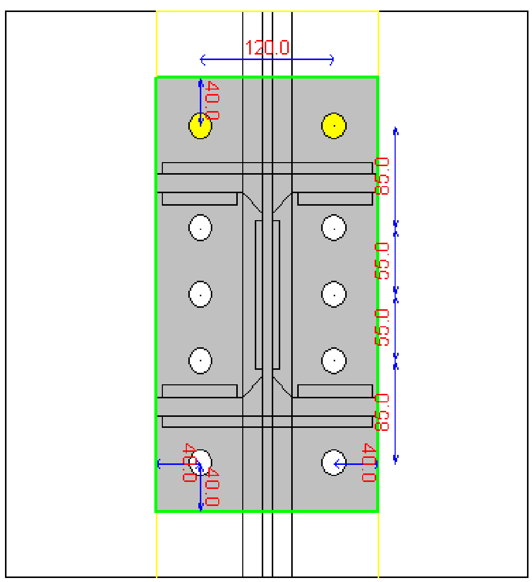


First of all using the **Face** arrow keys select (gets green) one face of the column, and then press the **= current face owner** button, in the **Object to be checked as bearing** box.

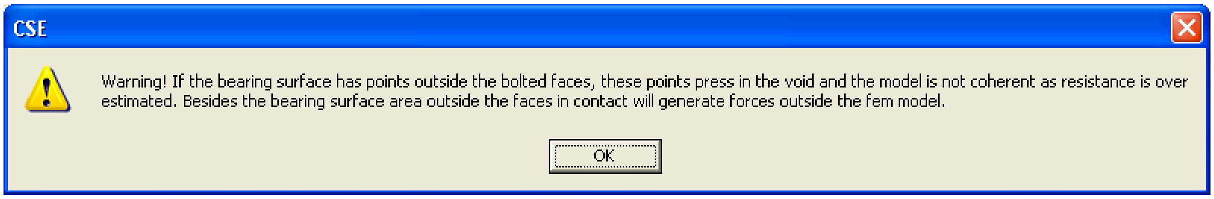
Then go back and select the plate face, and notice that the column face previously chosen is yellow, meaning that the bearing object has been chosen.



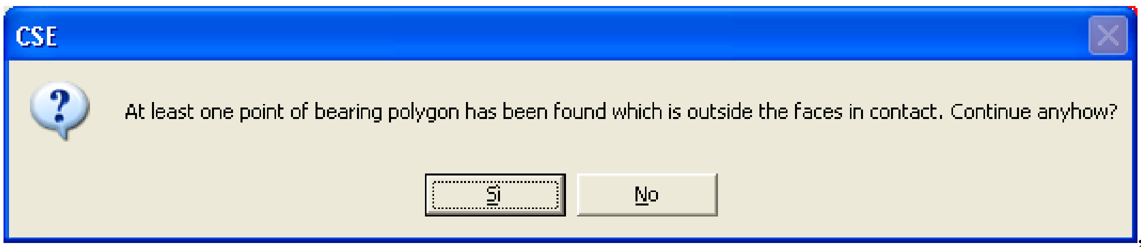
Now press the "bearing surface = current face" button, and you see that the bearing surface gets gray:



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:



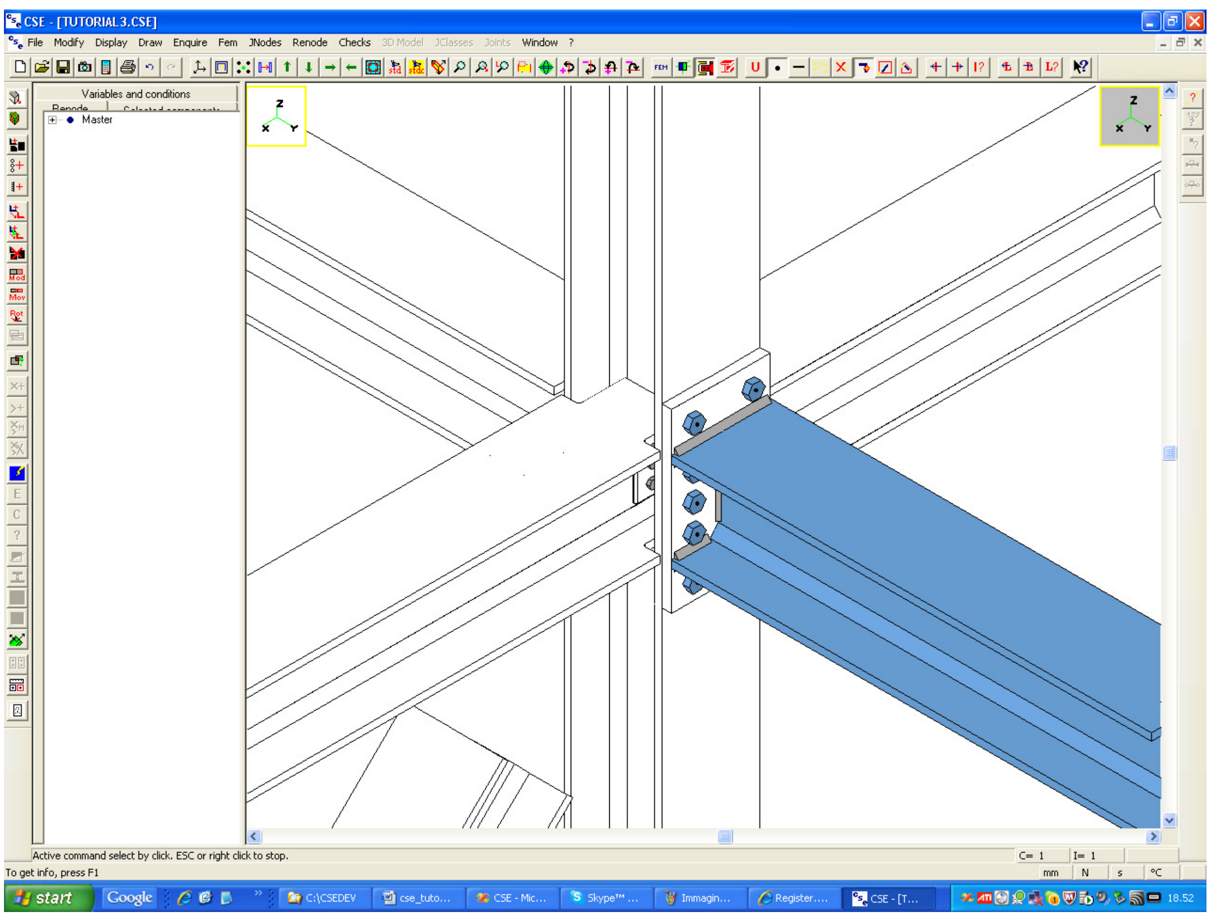
Here this message is just a warning: no point of the bearing surface is really out of the surfaces in contact. Next message warns us that there are points of the bearing surface outside the faces in contact (in fact some of the faces in contact, those of the column, have points outside the bearing surface,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



,

press yes (sì) and go forward.

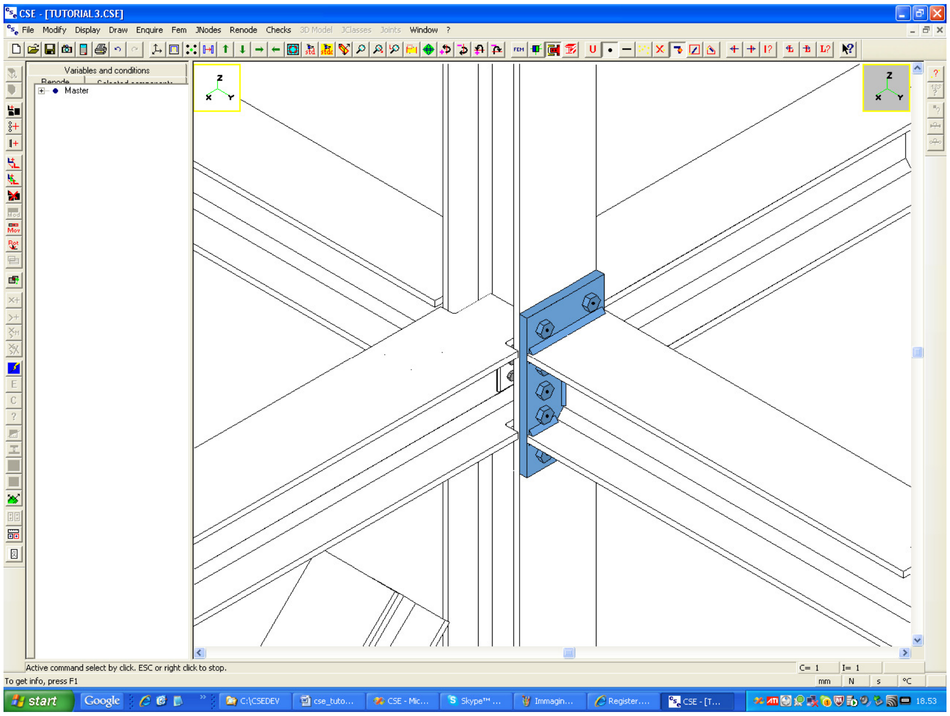
You see now the bolts applied:

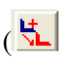


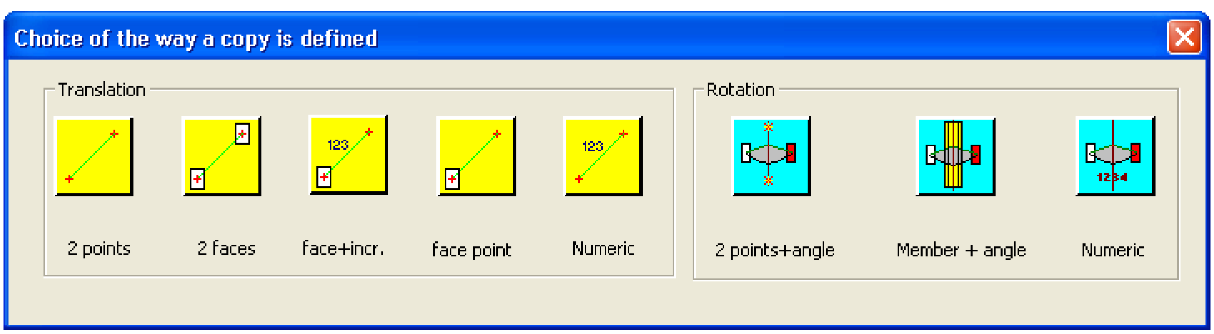
Notice that since these are not "shear only" bolt layouts (the bolts carry axial force), a dot is drawn at the hexagon center.

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

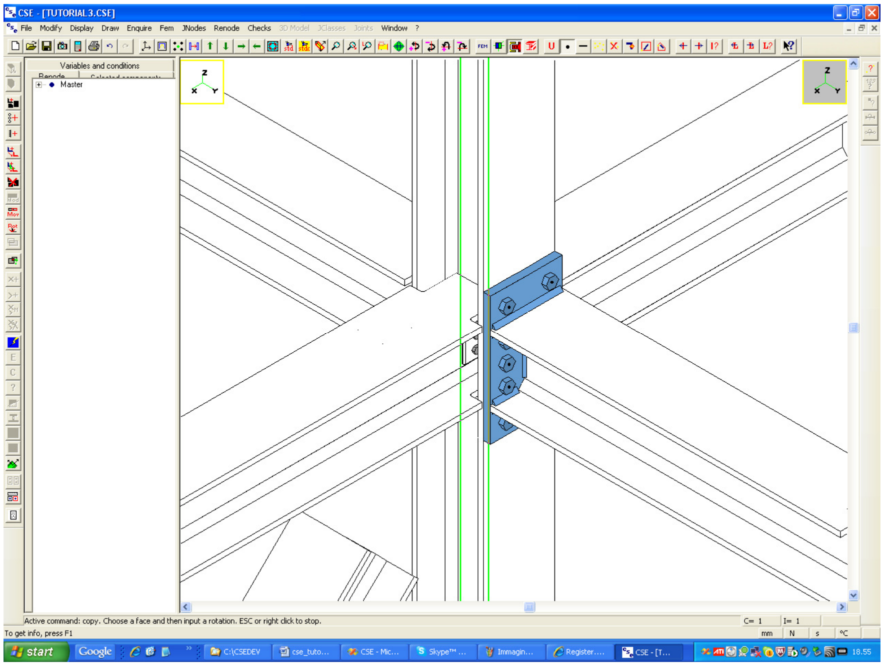
Now we would like to copy all added components (the plate, the weld and the bolt layout) to the other (-Y) beam. Select the objects you wish to copy:



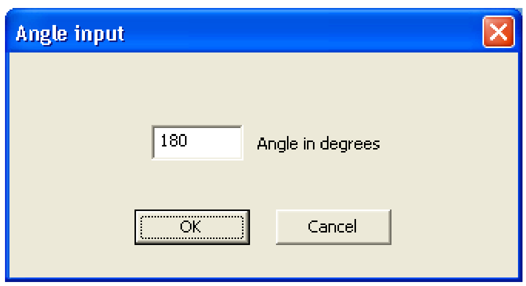
Execute the **Renode-Components-Copy** command ( button in the left bar). Press the second button from the right.



Now select a face of the column:

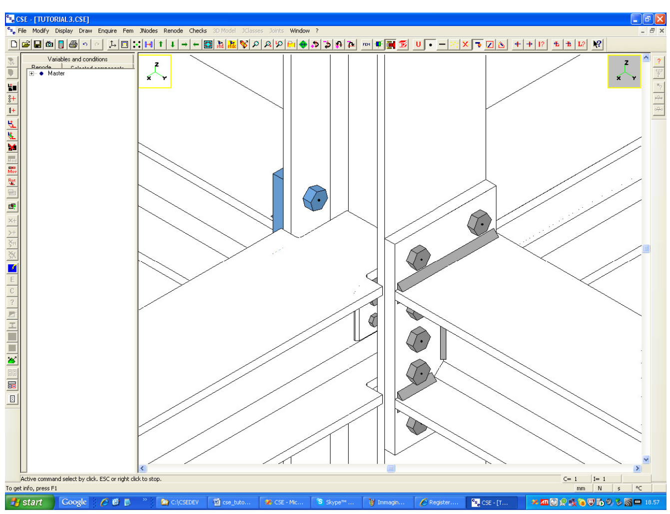


left click



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:



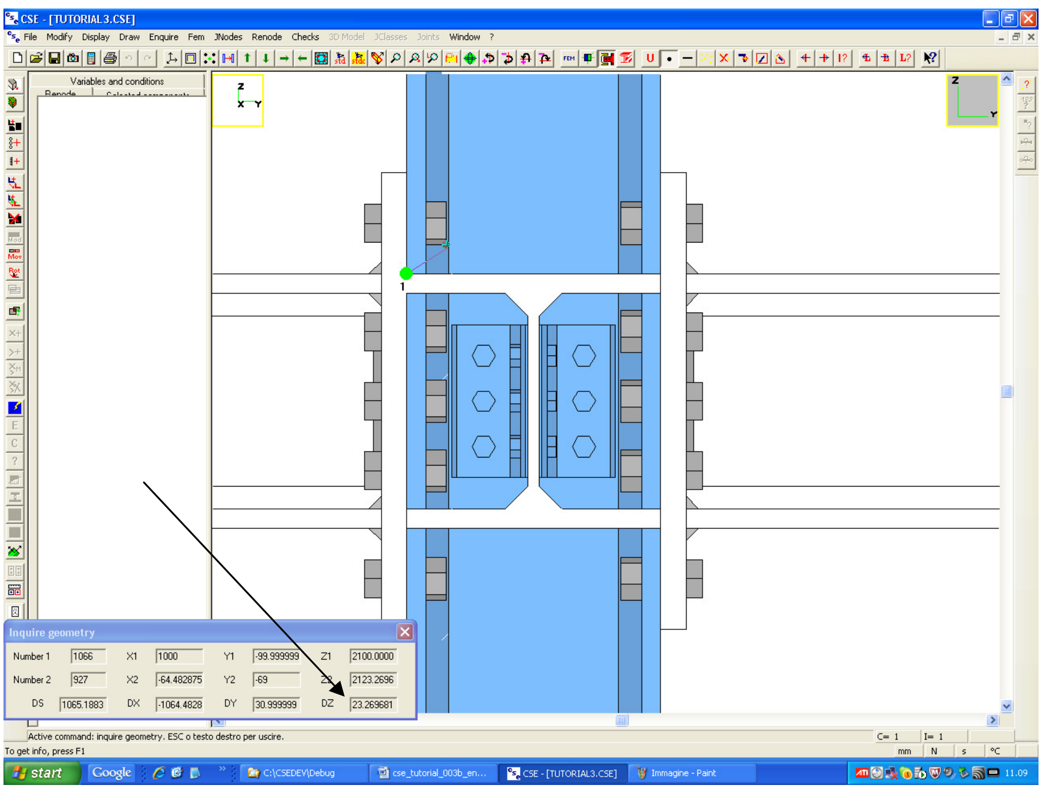
2.1.6 Adding the stiffeners

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.

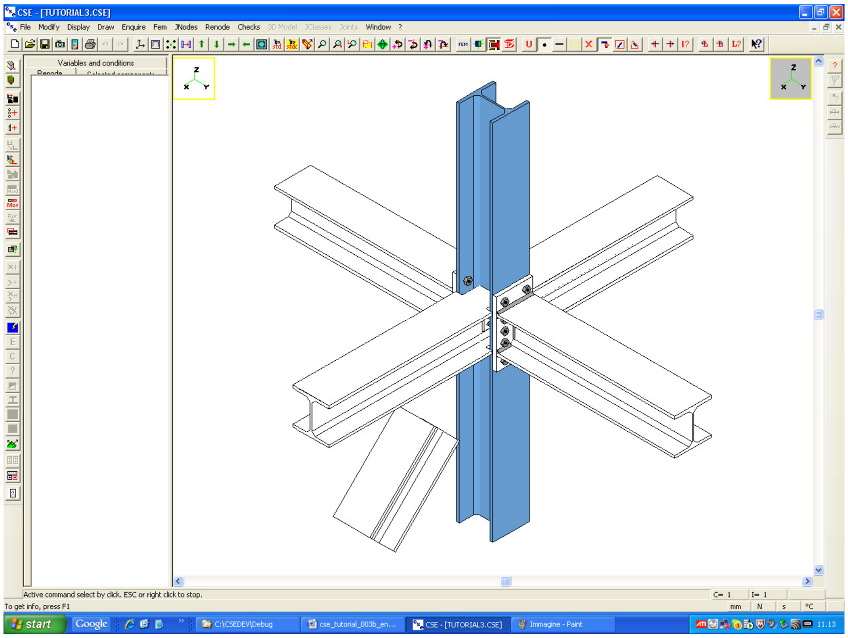
The first thing is to understand how much room there is to place these stiffeners. We will first add a stiffener and place its welds. Then the stiffener and the welds will be shifted and copied to be put in place correctly.

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

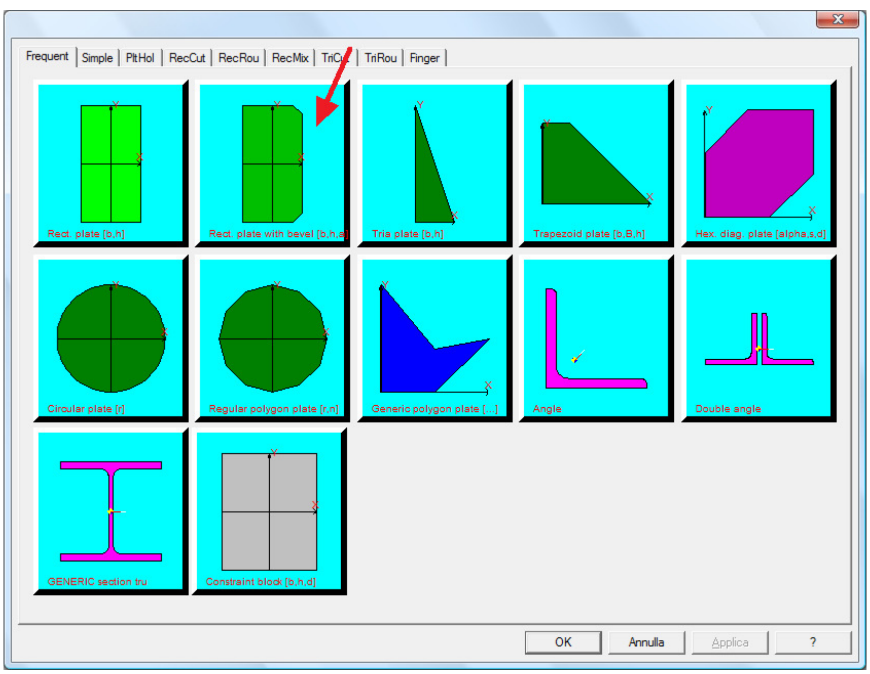
Set a +X view (**Draw-Std Views** - in the main bar -, and choose +X), then zoom in using the mouse wheel. Then execute the command Enquire-Geometry (in the right bar) and look for the free space to put a first stiffener, like this



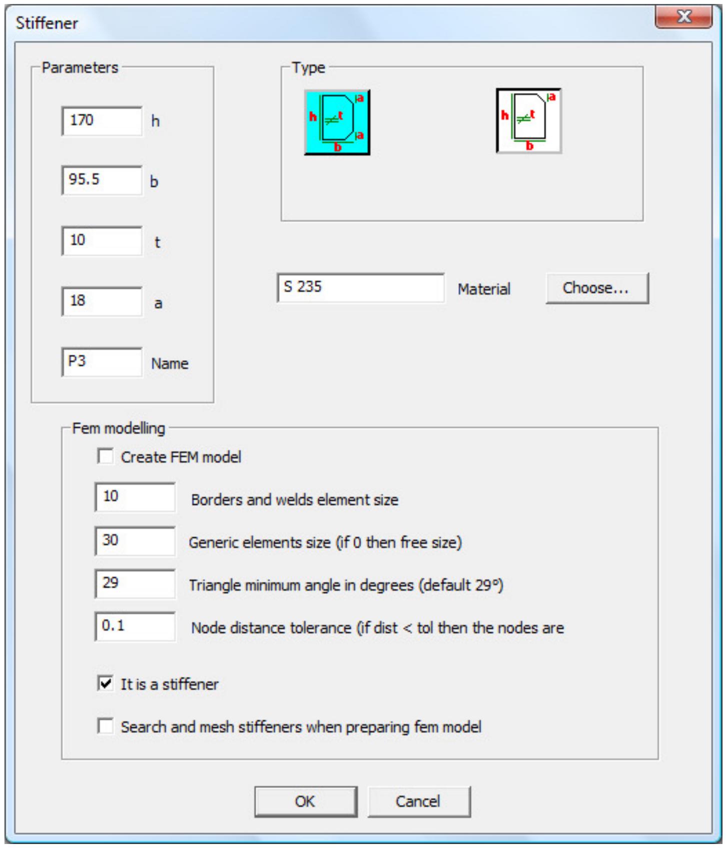
The "1" is over the first point you have clicked. The mouse is over the second point. You see that there is about 23mm free. So a stiffener 10mm thick with a weld 6mm thick at the top and a weld 6 mm thick at the bottom is 10+6+6=22mm: it can be placed. Click right to exit from **Enquire-Geometry** command.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



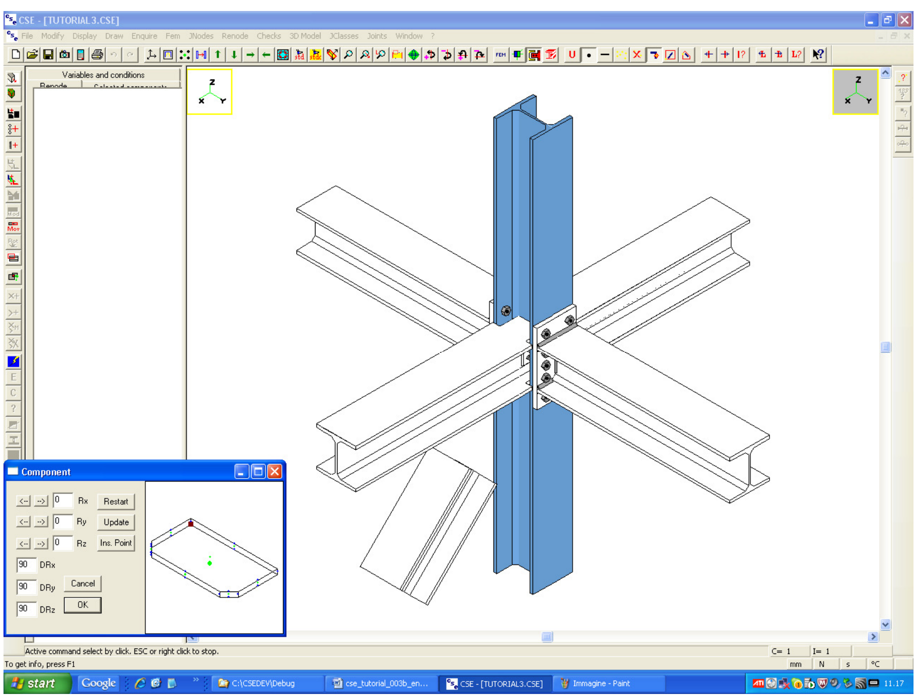
Execute the command **Renode-Components-Add through** ( in the left bar):



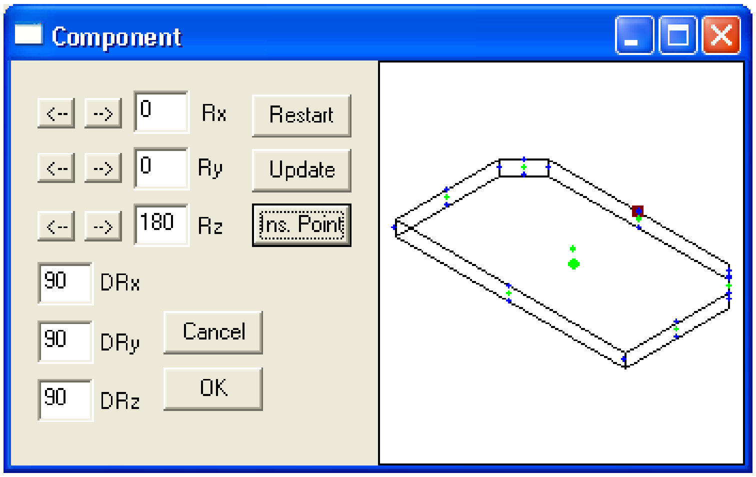
Choose "**rectangular plate with bevels**". You are now here:



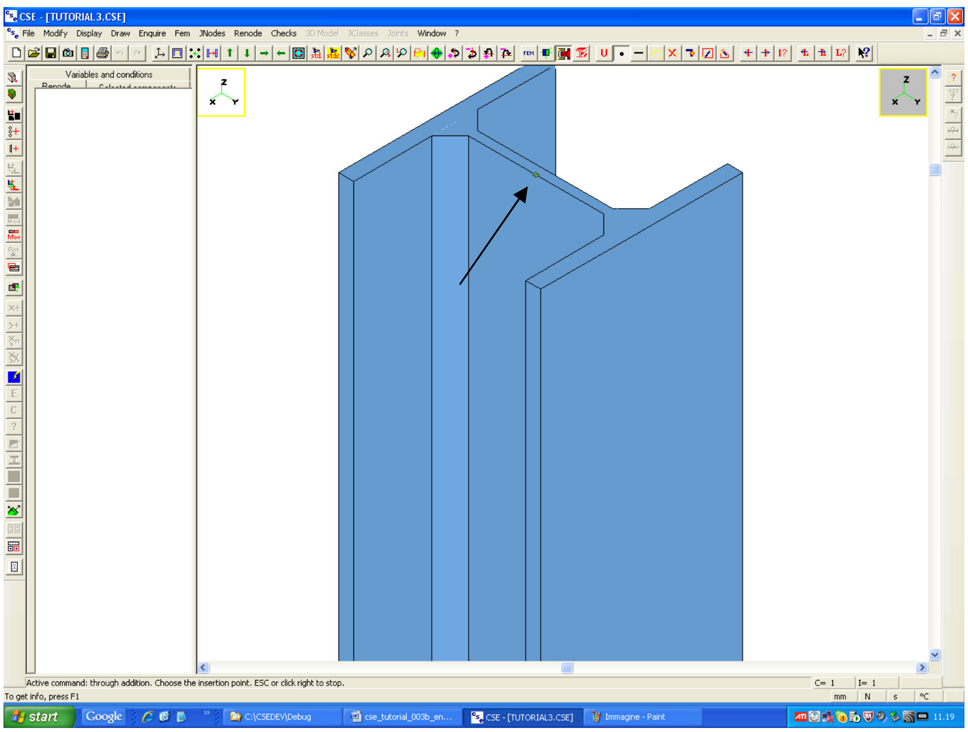
Leave all sizes as they are already correct (due to the column selection you've done before the command). The 10mm thickness default is ok for us. Press **OK**. You are here:



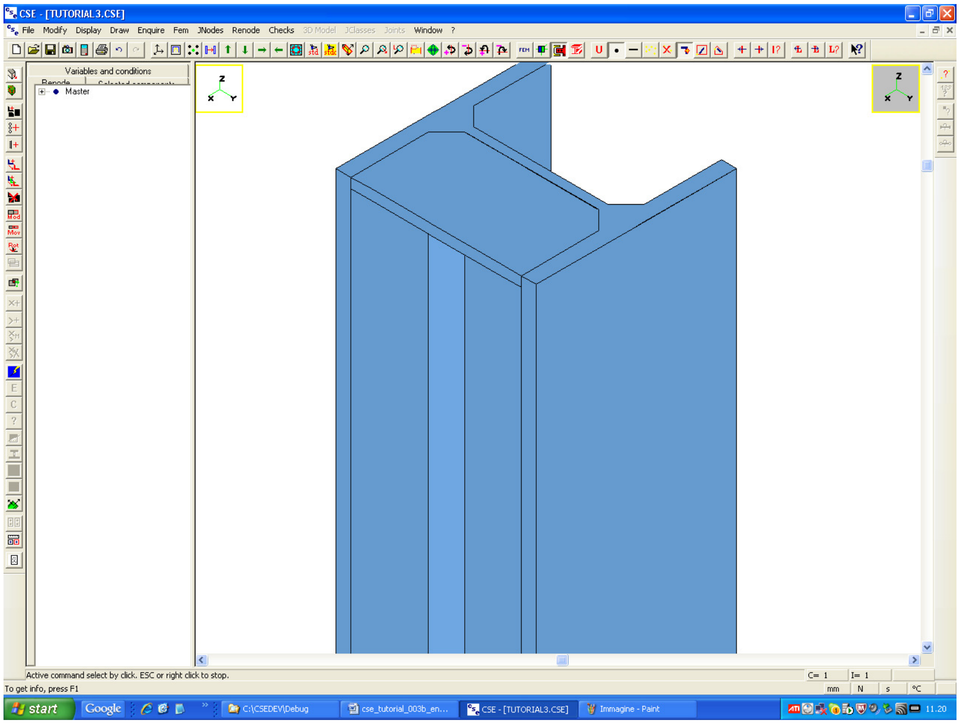
Click twice the right arrow in the Rz row, press the Ins Point button and choose the following point:



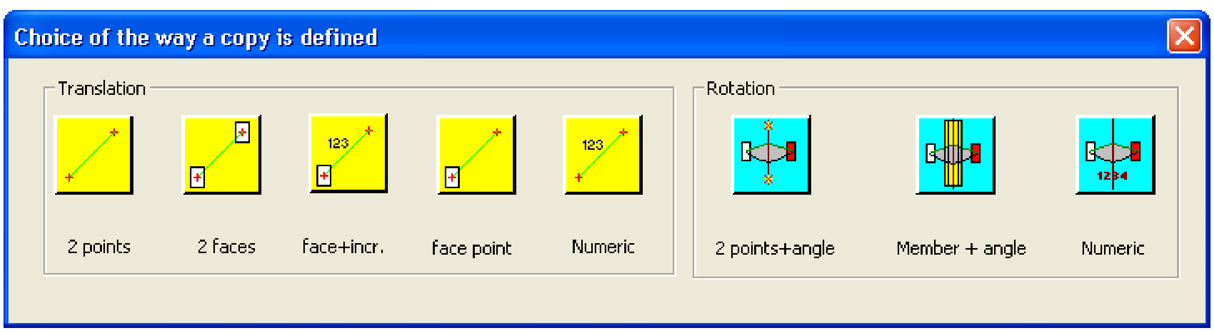
Now press OK. Zoom over the top of the column using the muse wheel. Click the correct point at mid of column web, like this:



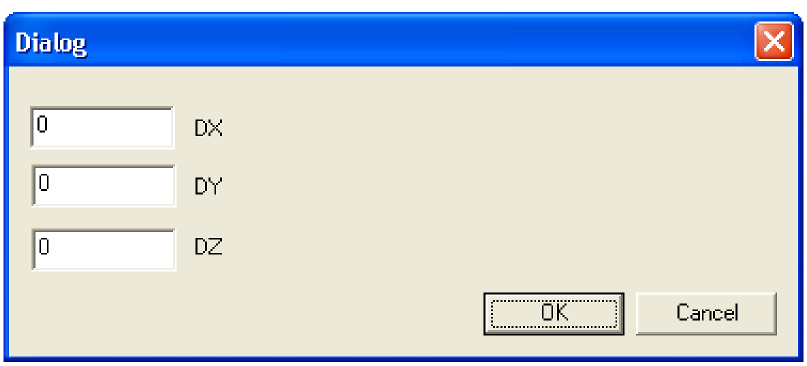
You will immediately see what follows:



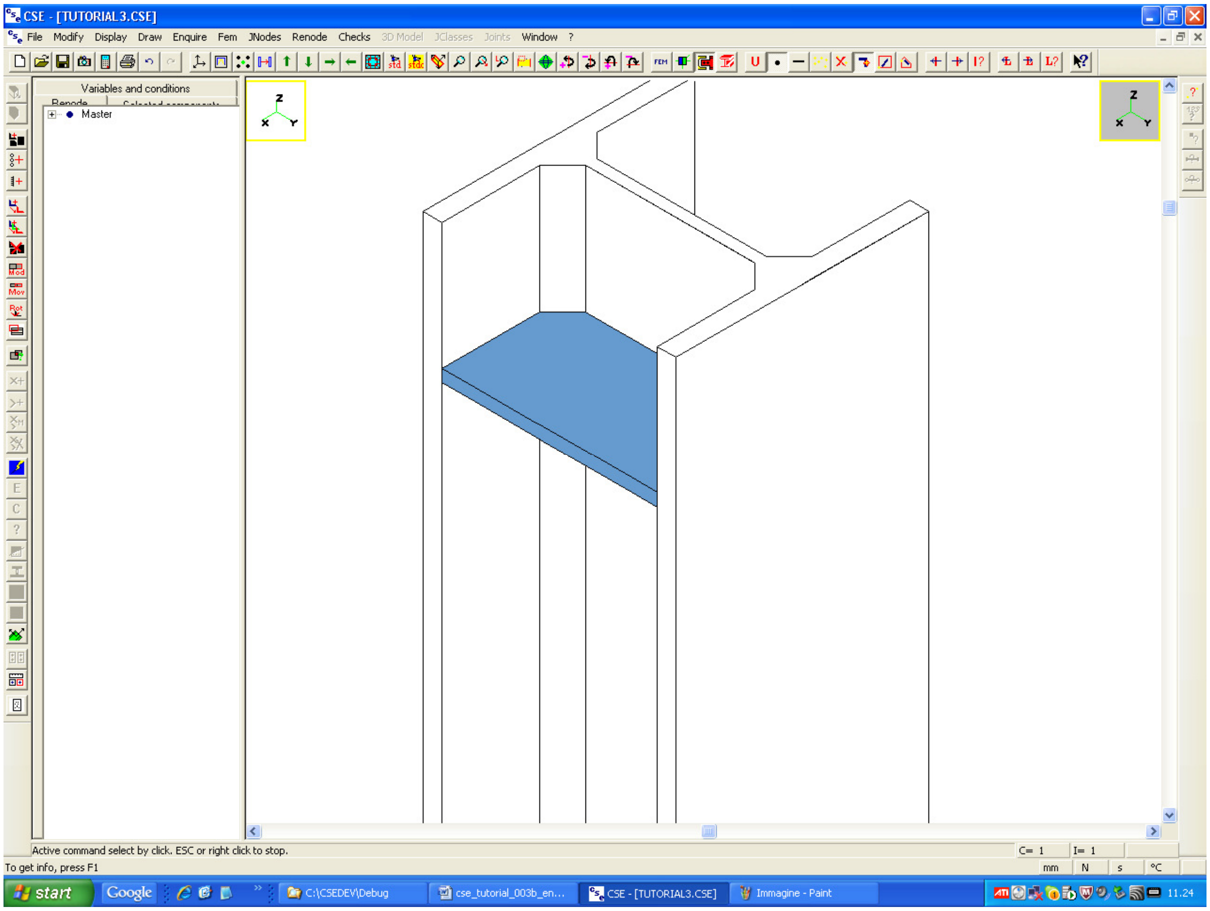
Now we are going to shift the stiffener a bit downward to have the room to add welds. Those all will be later shifted in position. Unselect the column so that the stiffener is the only component selected. Execute the command **Renode-Shift** (the  button in the left bar). The following dialog appears:



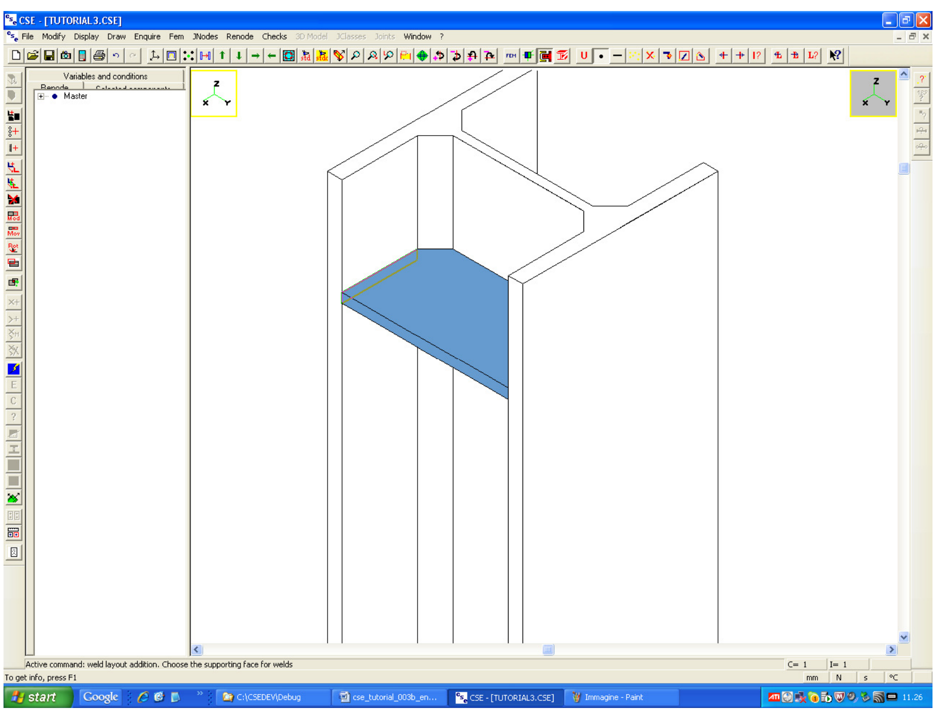
Choose the last yellow button (Numeric). You get here:



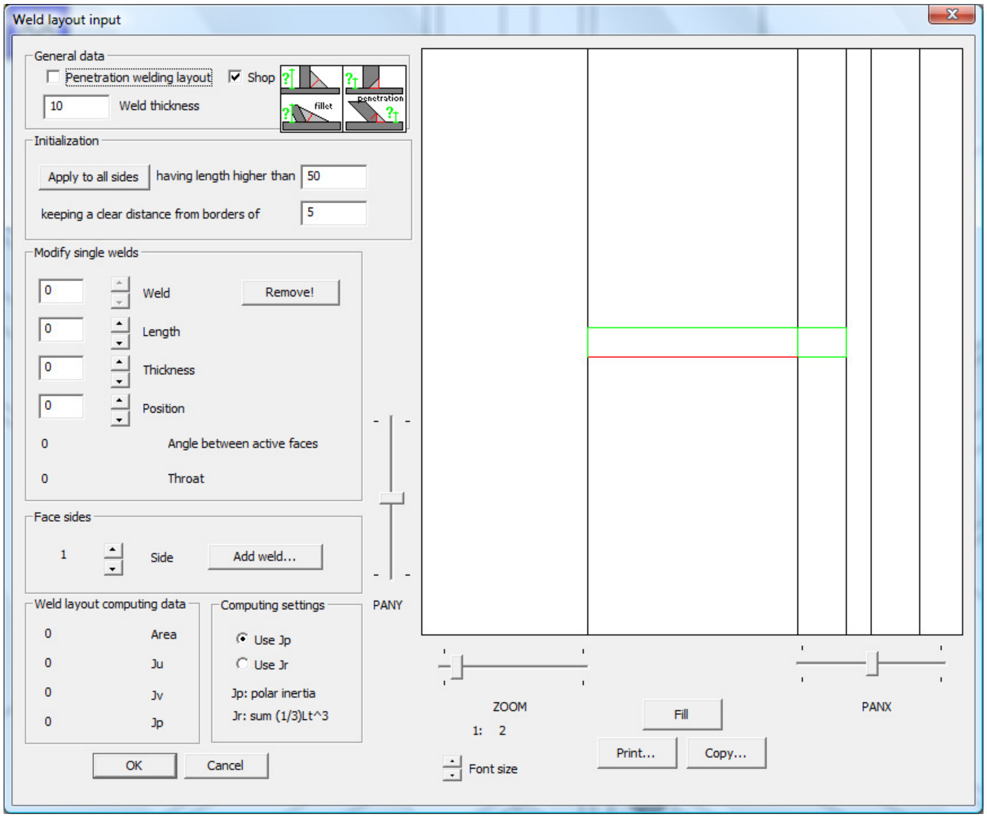
Input -100 in the DZ cell. Press **OK**. You now see this:



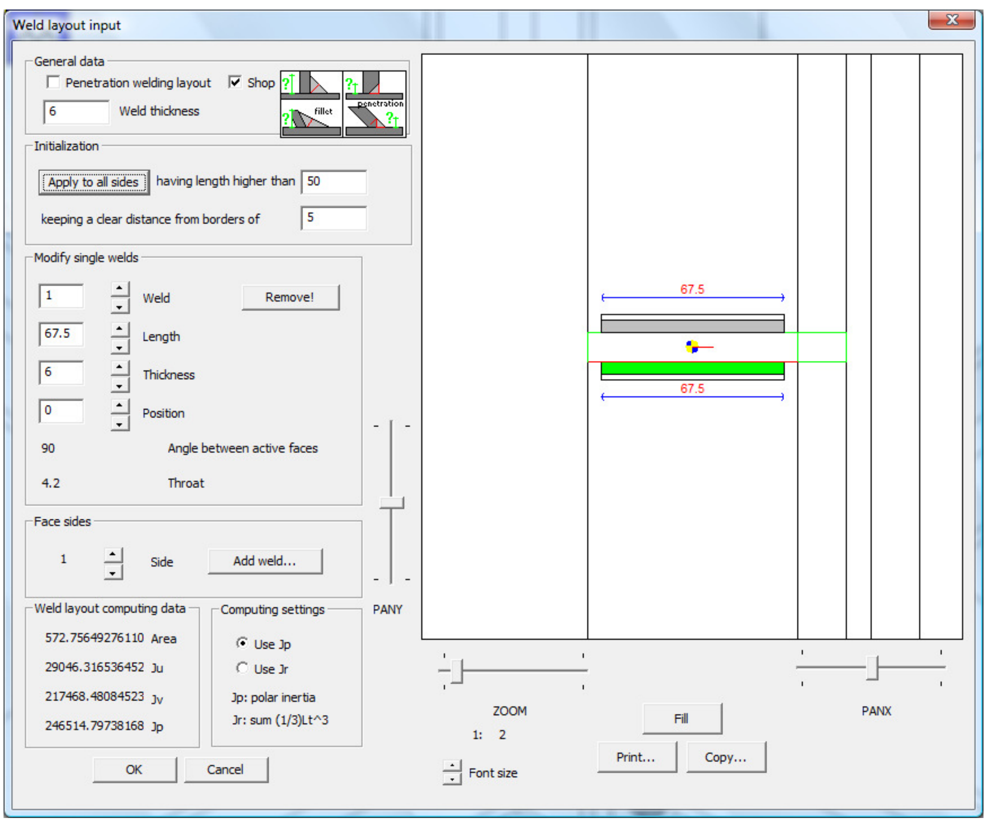
Now we will add welds. Execute the command **Renode-Components-Add weld layout** (button in the left bar). Then choose a first contact face like this, clicking left:



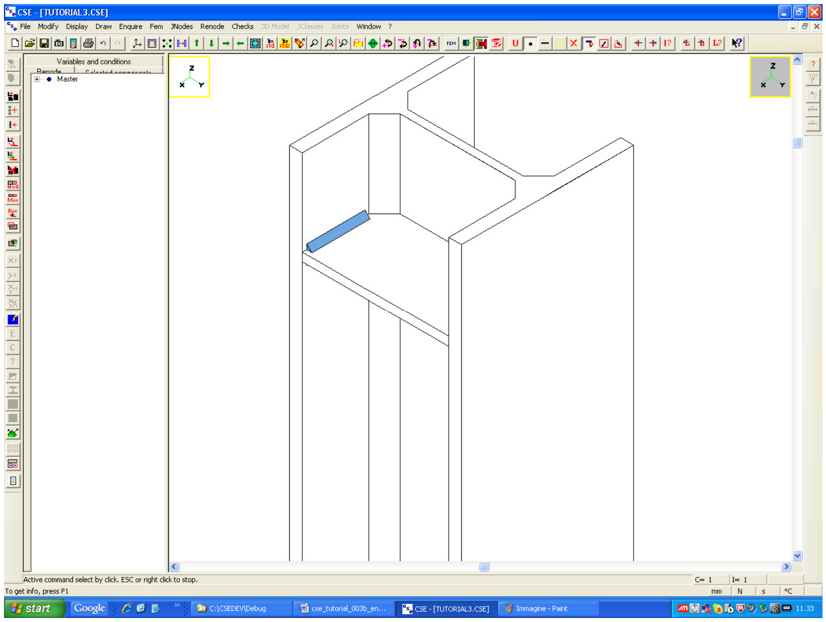
You are now here:



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

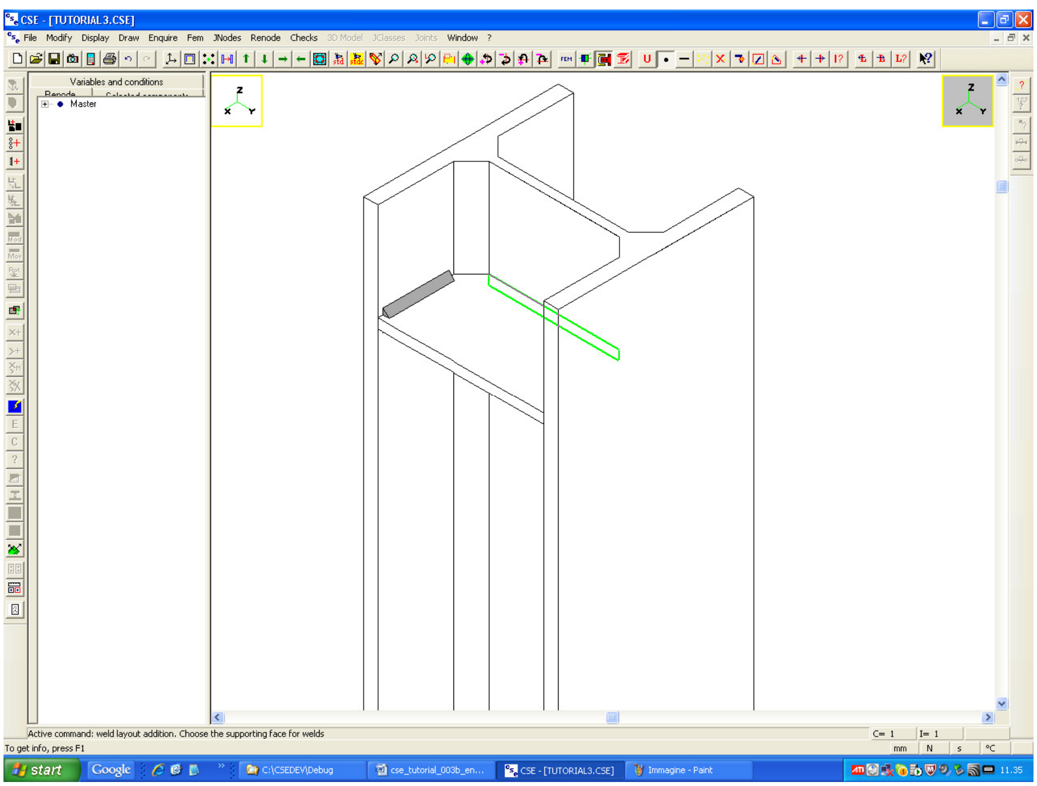


Press **OK**. You see this:

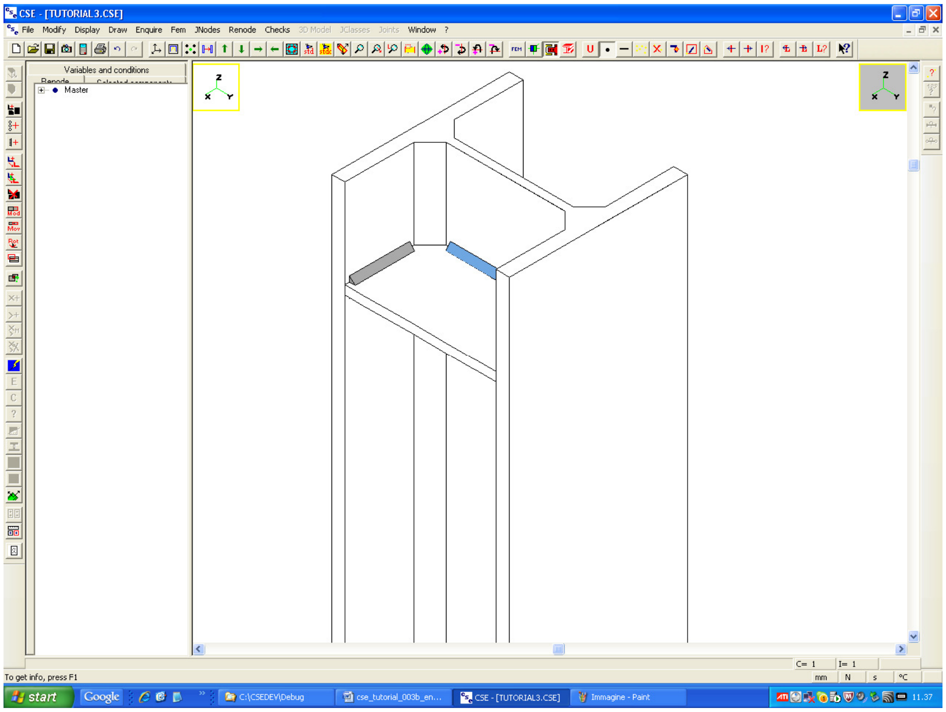


Unselect all ( ) button.

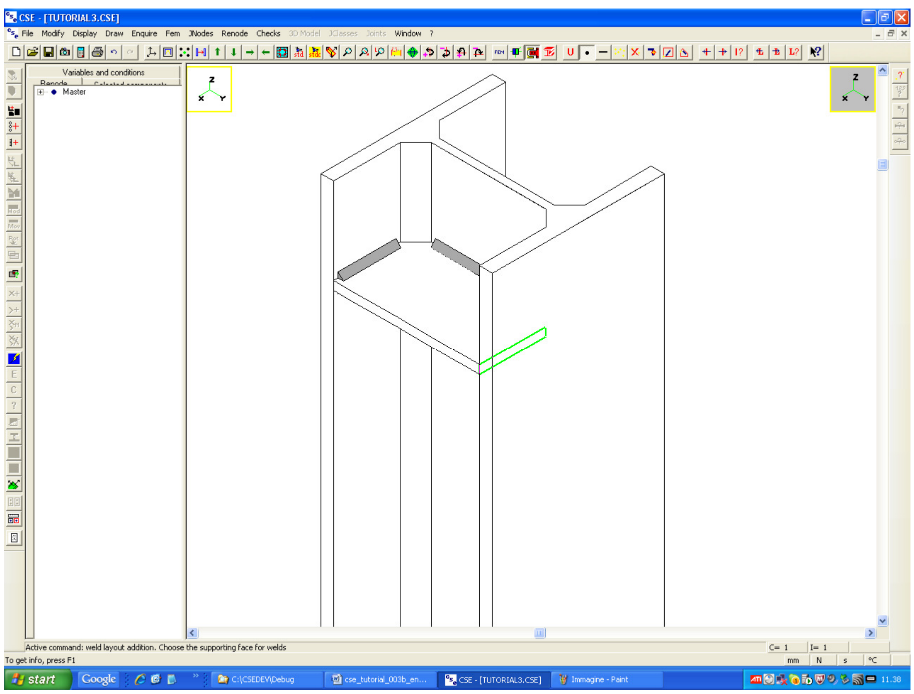
Now re-execute the weld layout addition command (left bar) and choose the next face, like this:



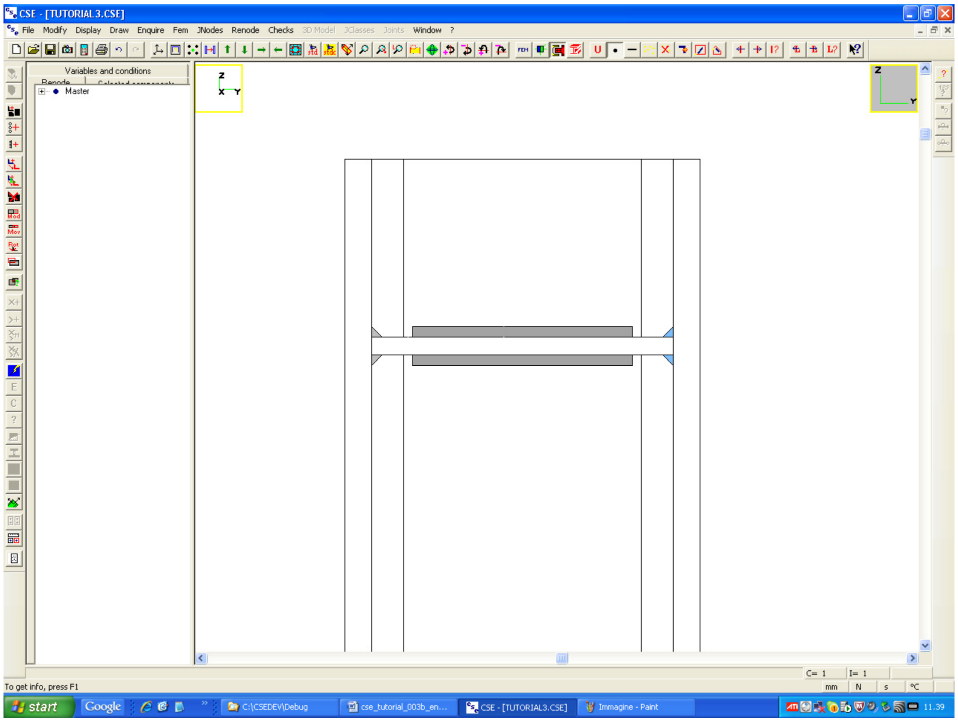
Repeat the same operations in the weld layout addition dialog:6mm as weld thickness , **Apply to all sides**, **OK**. You will be here:



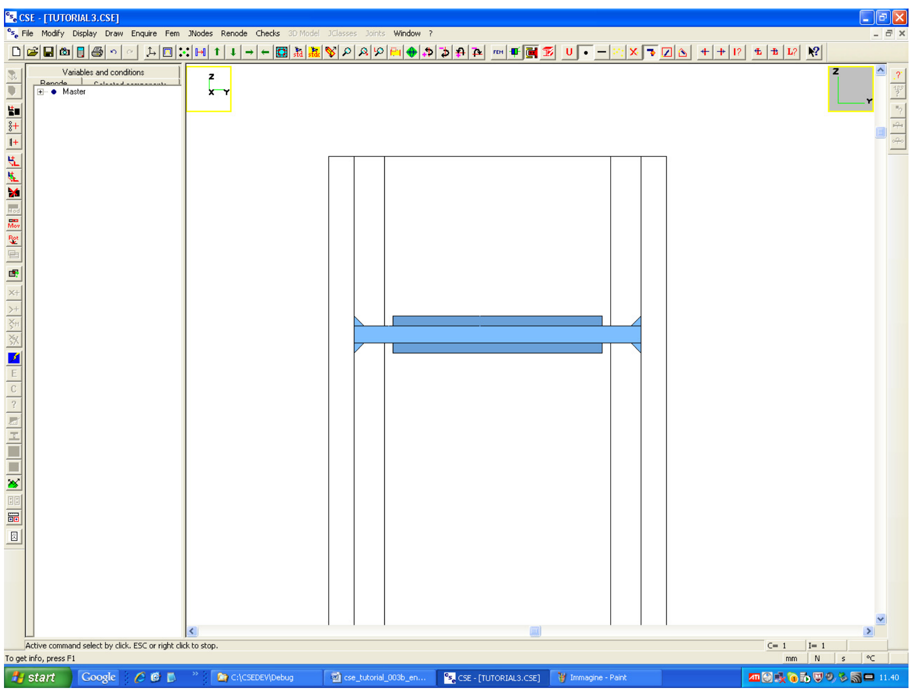
Unselect all and repeat the command for the third time, in the same way choosing this face:



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:

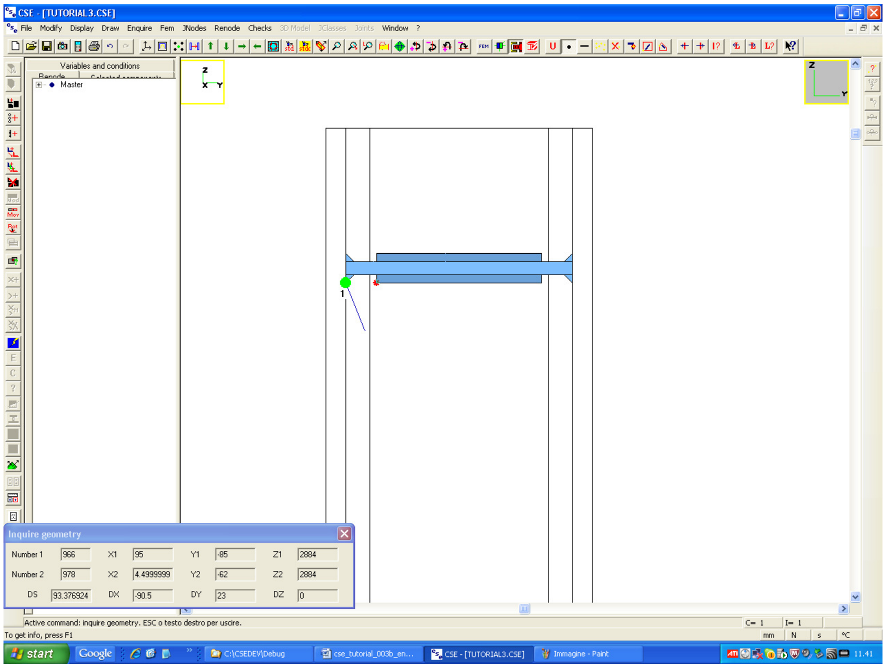


Select the stiffener and the three welds (you'll click over the stiffener and two welds), like this:

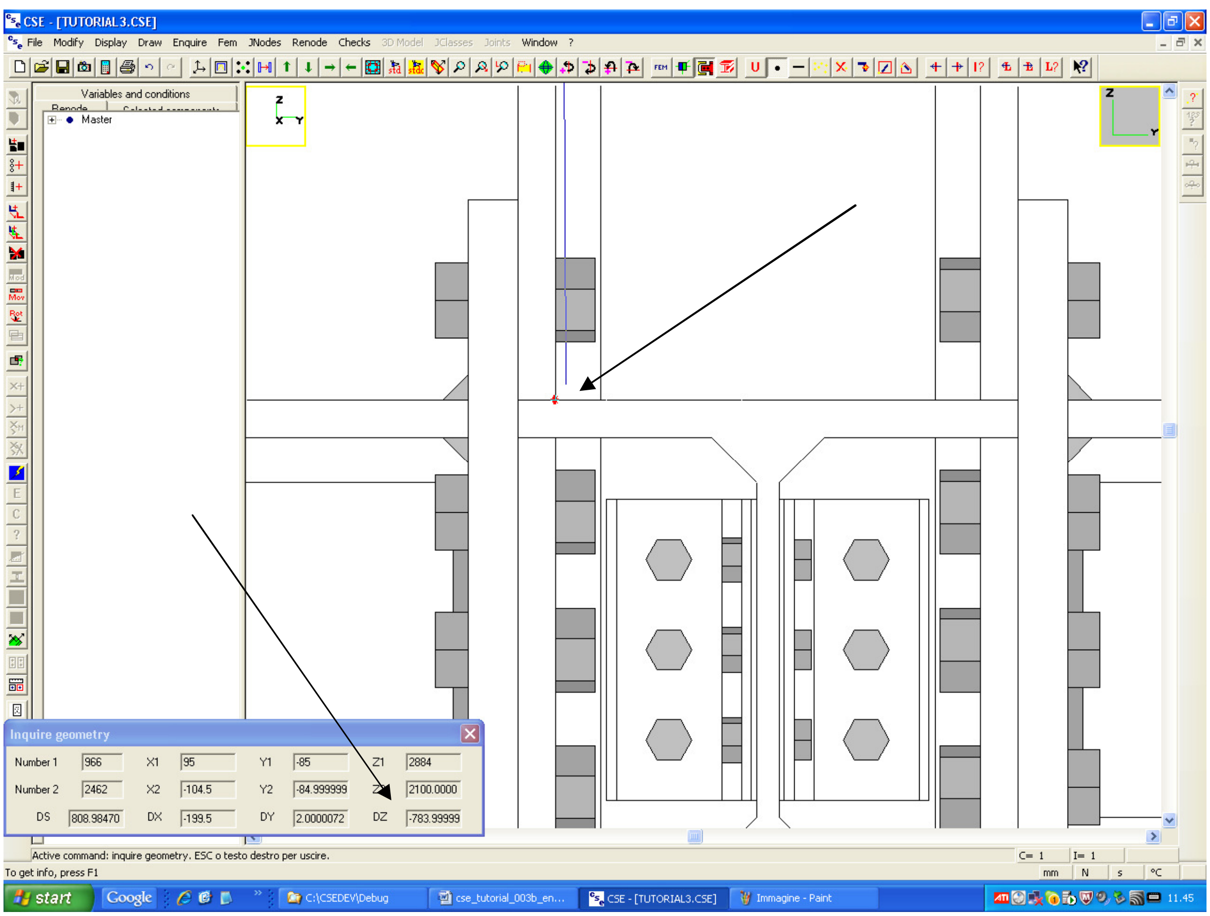


We are now ready to shift these components in place. However we have to understand the amount of the shift.

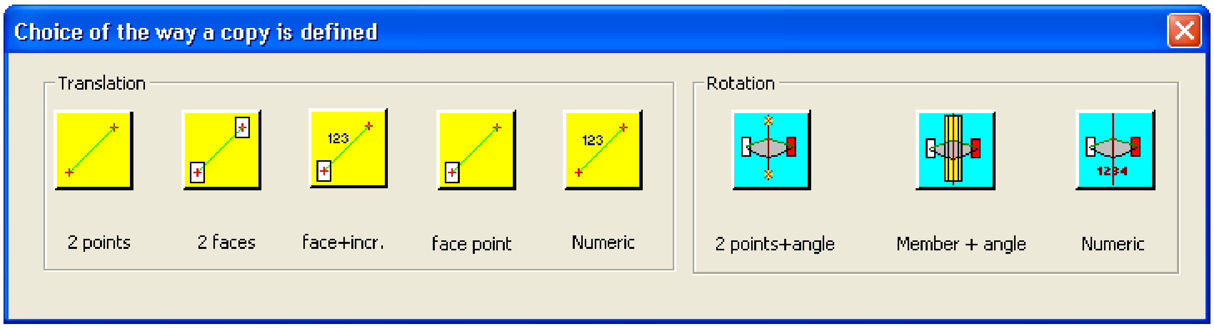
Execute the **Enquire-Geometry** command ( button in the right bar), and click first over the lower left point of the object selected, like this



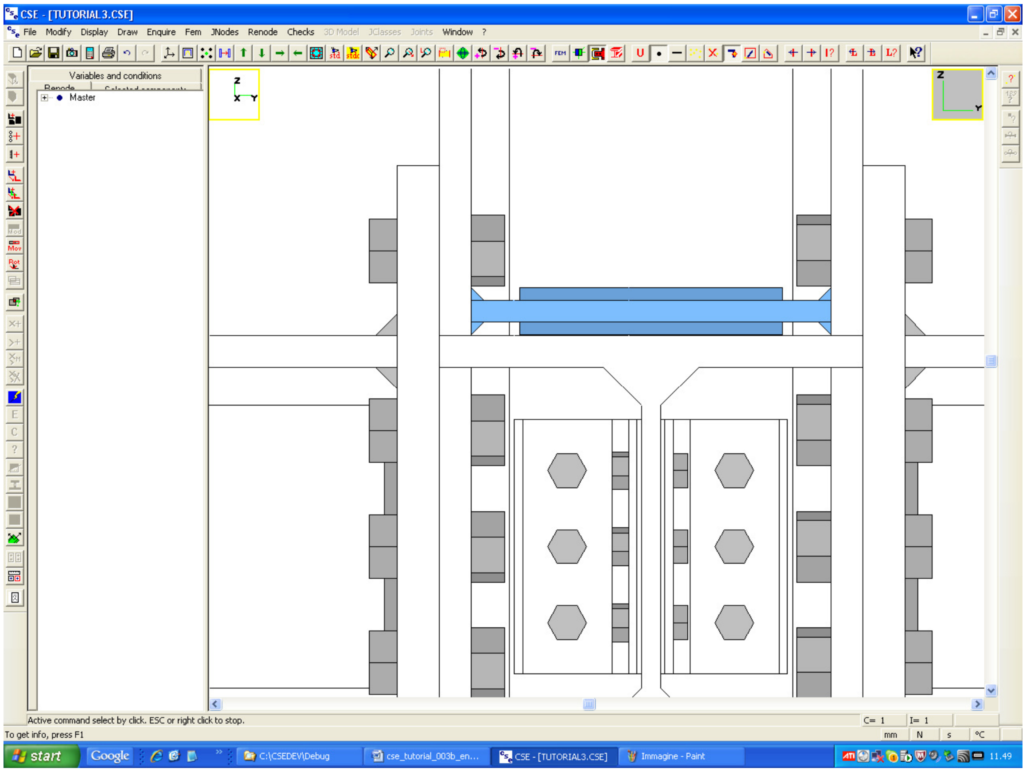
Now while still in the command execute the command **Draw-Enclose** (button in the main bar), and then zoon near the mid column point, like this (you are still in the **Enquire-Geometry** command).

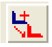


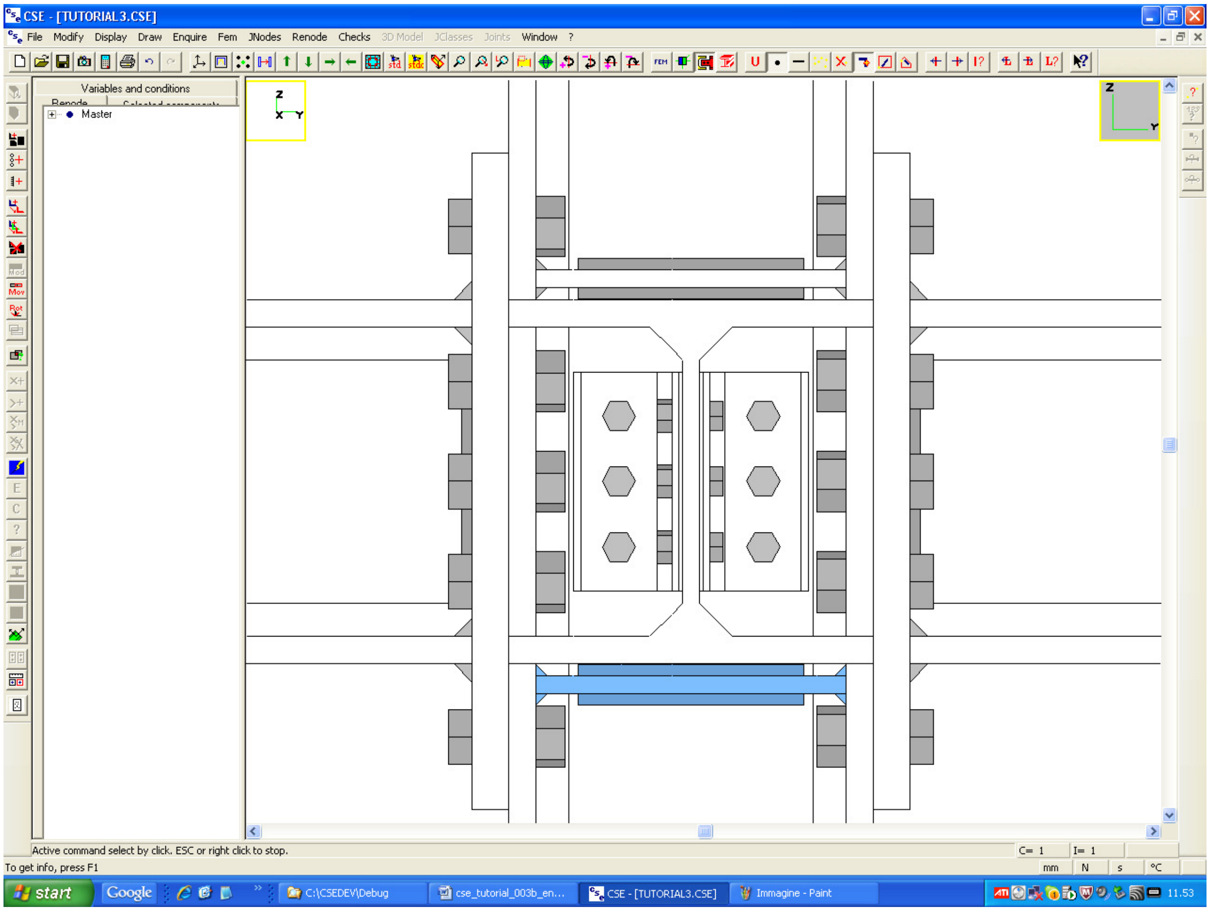
Notice that the distance to shift down is -784 mm to get a contact. if we shift down -783.5 we will not be in contact. Click right to exit from the command **Enquire-Geometry**. Execute the command **Renode-Shift** ( button in the left bar), you see this:



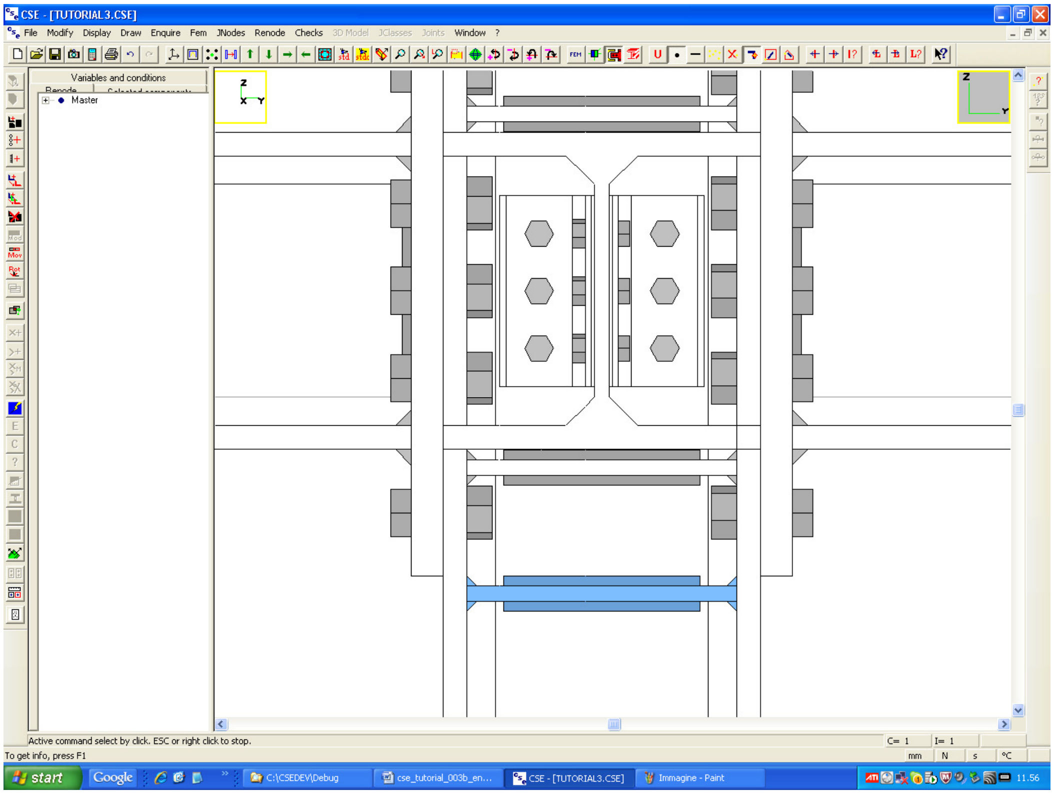
Choose the yellow Numeric. Enter DZ=-783.5. You now see this:



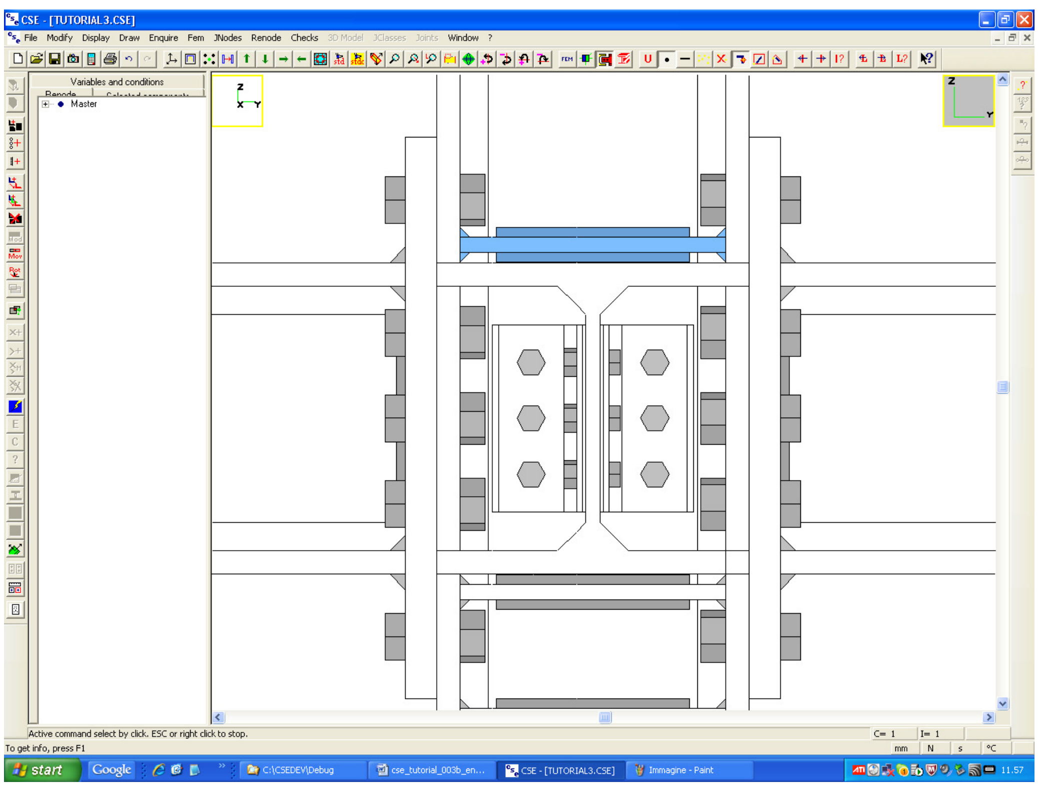
Now we will copy these components down. Using the Enquire-Geometry command you will understand that we have to copy downward of -223mm. Execute the command **Renode-Components-Copy** ( button in the left bar) choose the Numeric input mode as before (but now is a copy). Enter DZ=-223. You see this:



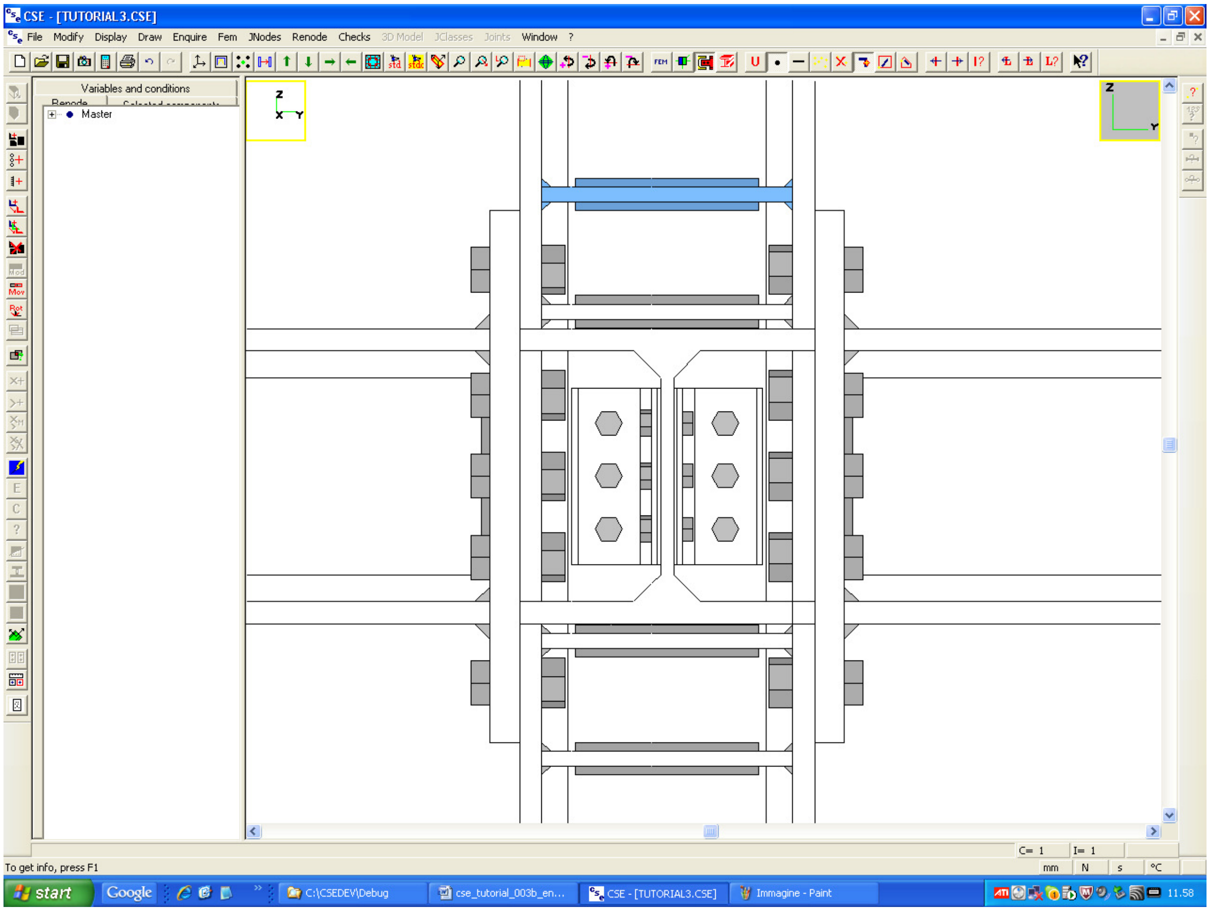
Now we will copy once more down, to have two stiffeners near the beam flanges. Using the Enquire-Geometry command we understand that a good copy vector would be DZ=-79.5mm. Then re-execute the copy command (we are always copying the selected, blue components) and enter DZ=-79.5 in the numeric mode. You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



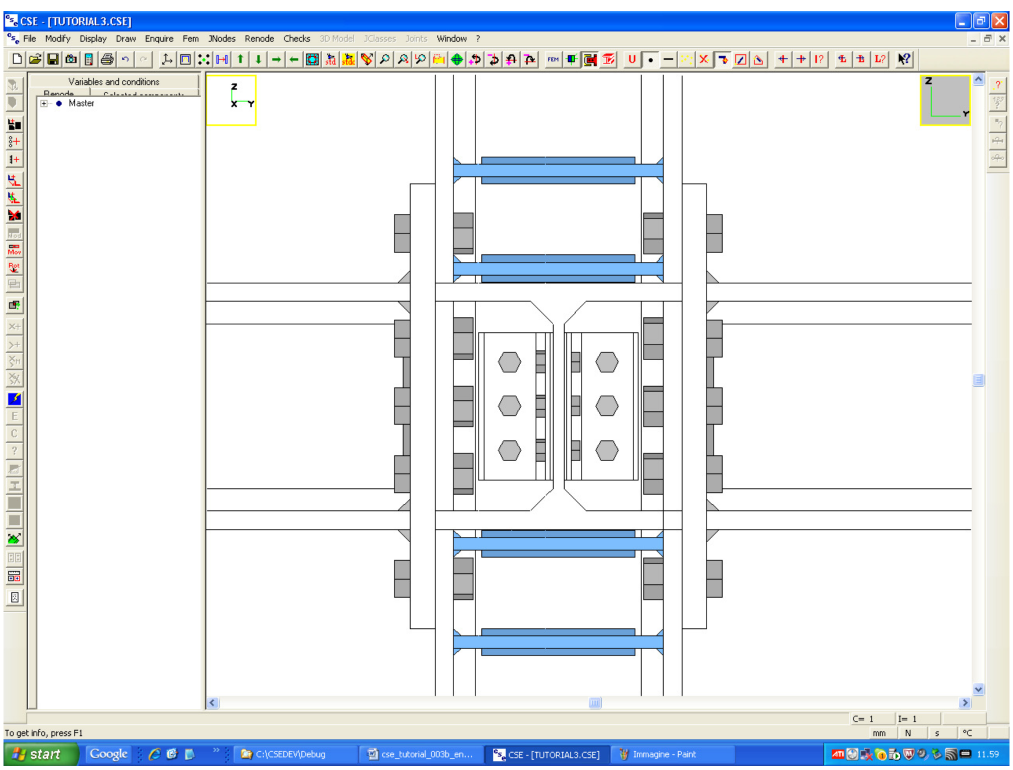
Now unselect all and select the upper stiffener with its welds, like this:



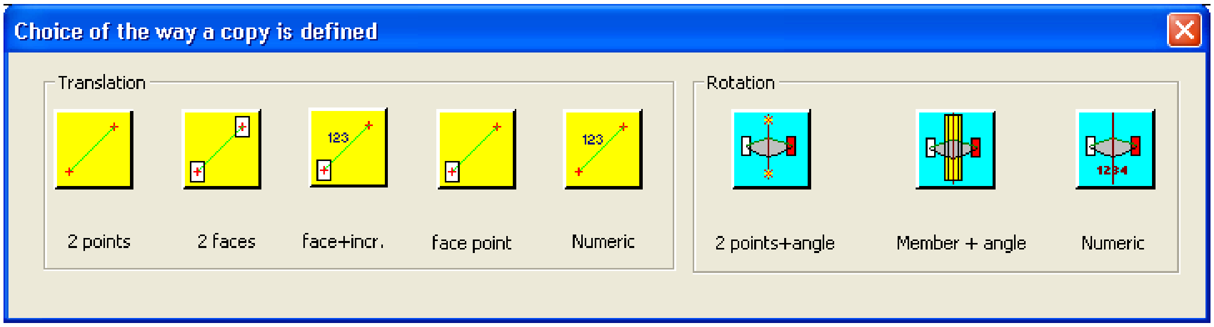
Re-execute the copy command and enter +79.5mm as DZ. You are now here:



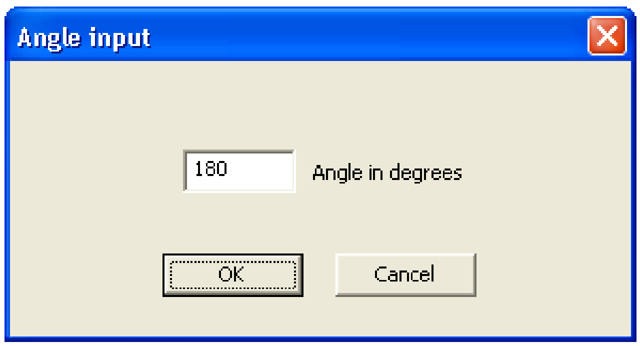
Now select all the stiffeners and their welds, like this (do not forget any):



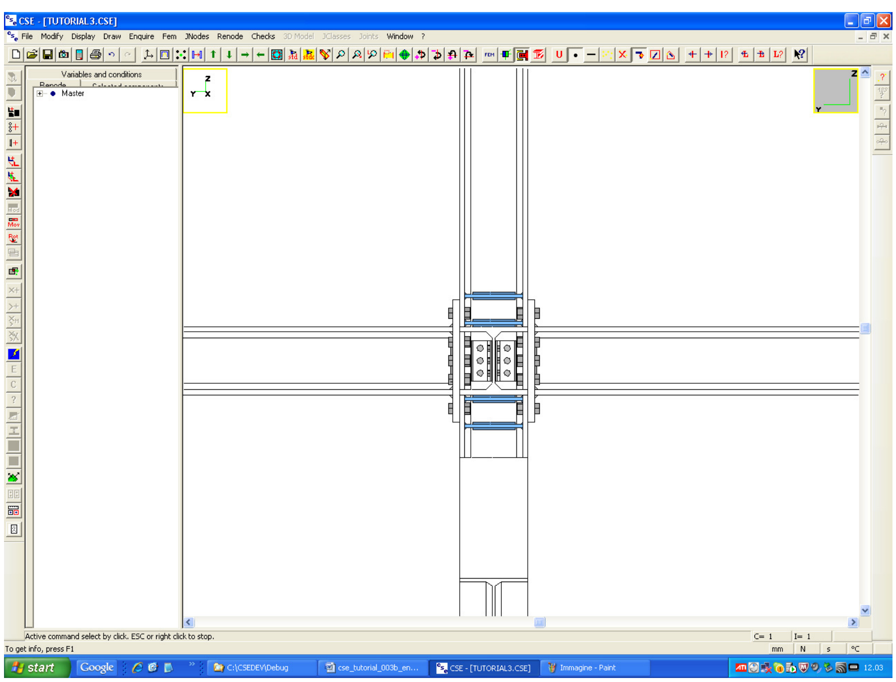
We will now copy them to the other side of the column. Choose the **Renode-Components-Copy** command once more, but now select in this dialog:



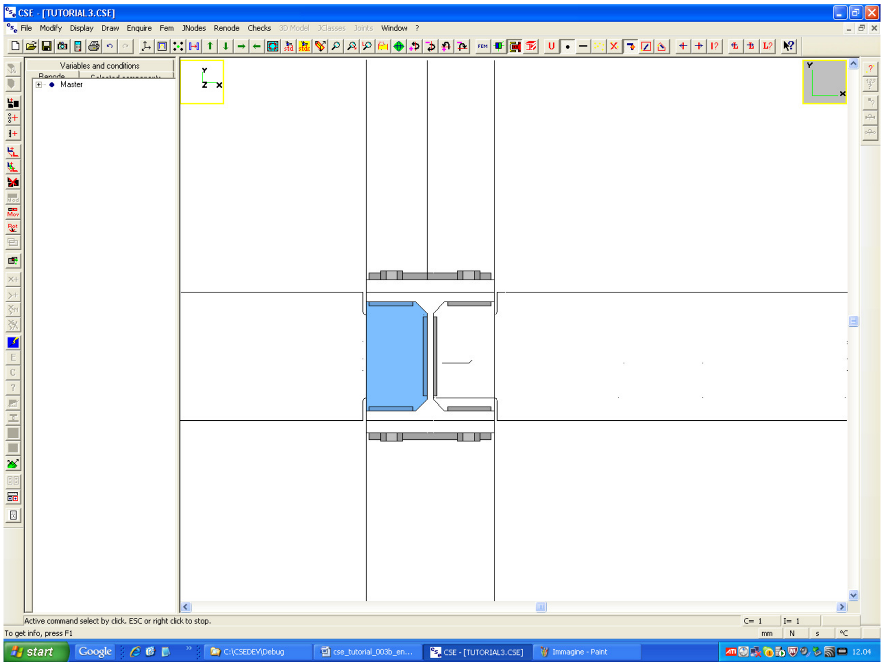
the "member+angle" button. Then select a face of the column (if you have problems in selecting a face of the column zoom out and choose the face at the very top of the column: you can also change view and zoom out) and leave 180 in this dialog:



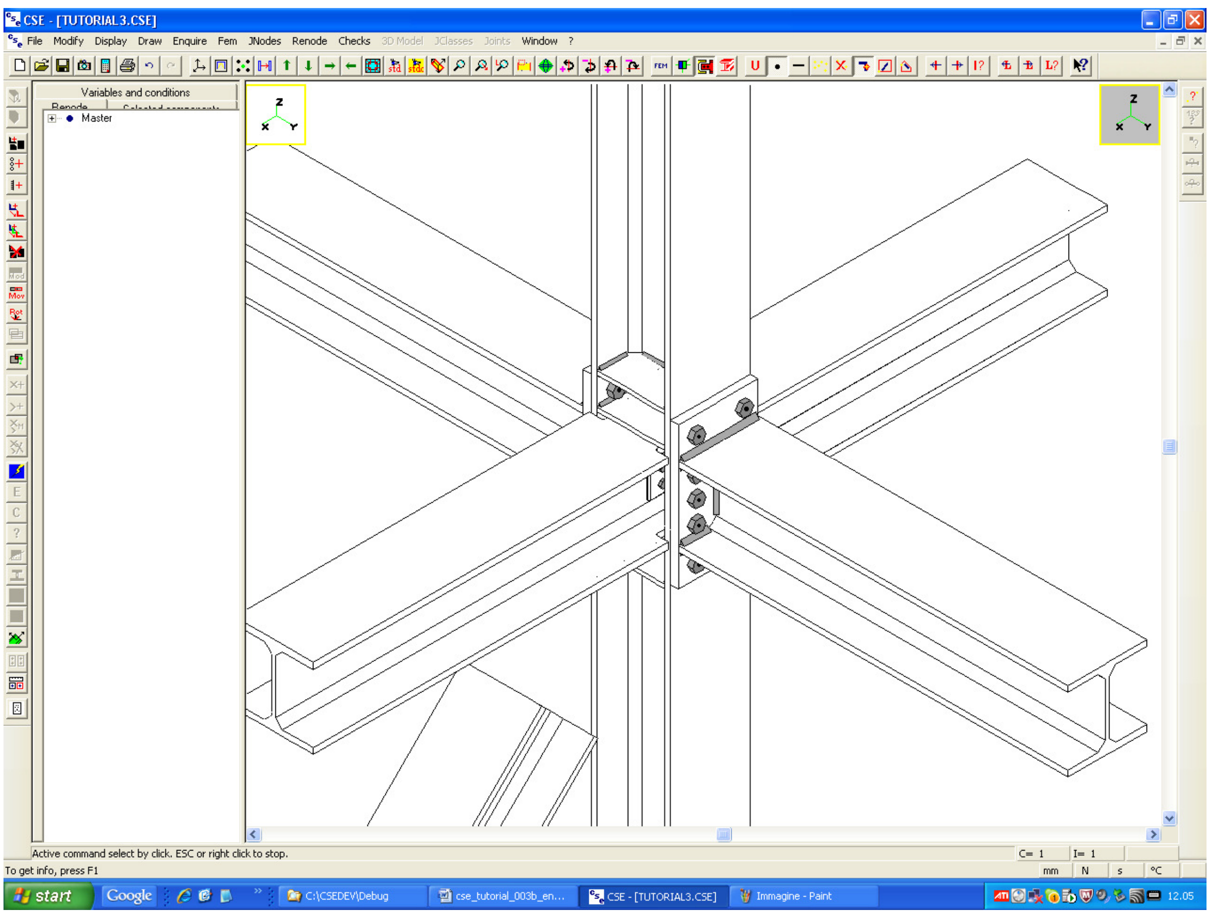
You have copied all components to the other side. To be sure, choose a **-X** (minus, not plus) Std view (**Draw-Std views**), and you will see the following:



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



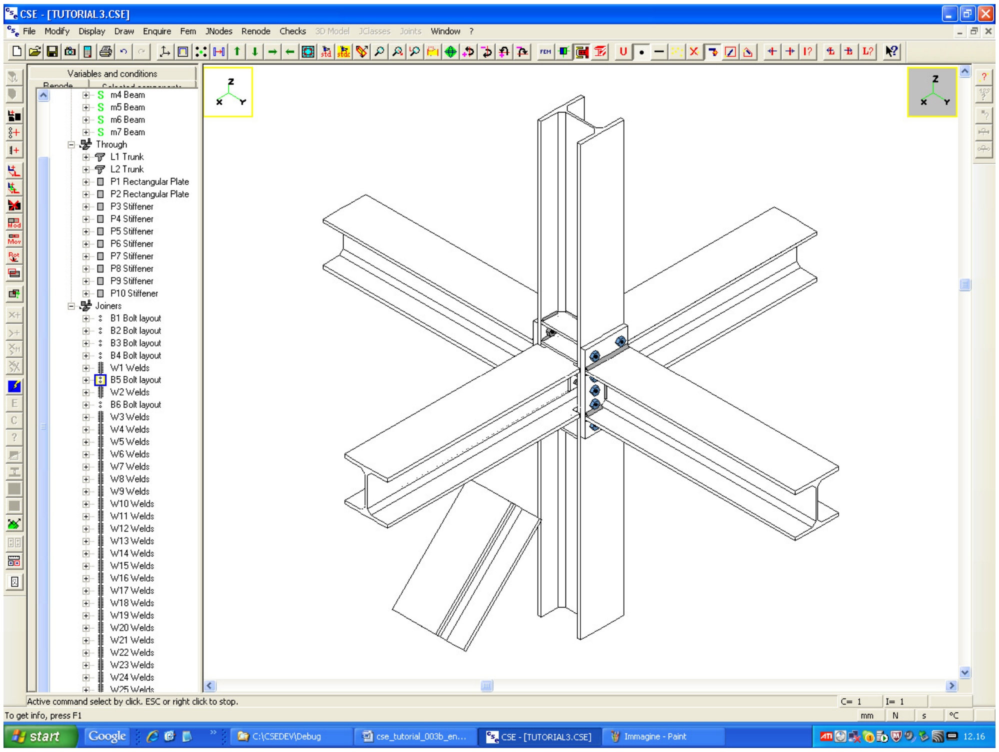
In an Isometric view you will see:

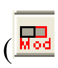


Please keep in mind that you could have placed as many stiffeners you wish, where you wish: this is just a demo. Maybe that 2 stiffeners per side instead of 4 could be enough. If you want you can easily delete some of the components, just select them and delete (**Renode-Components-Delete**).

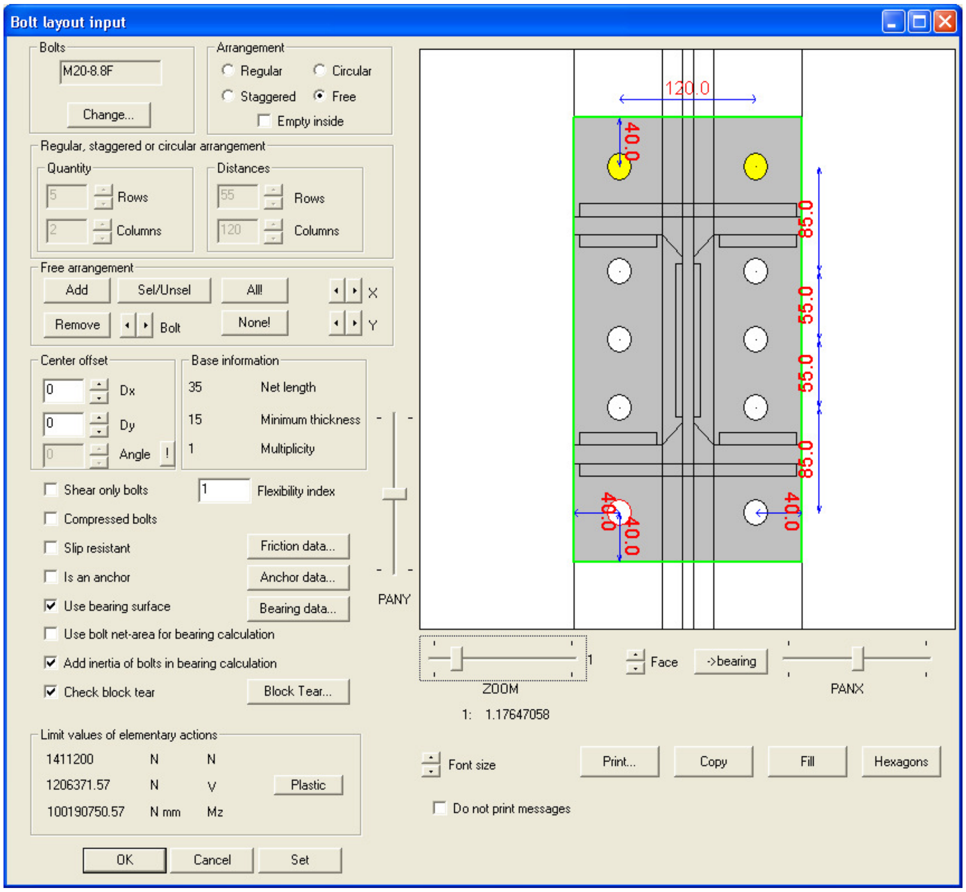
We are studying how to do the connection. So we can change and modify, as well. For instance, if we want to have more room for the stiffener we could displace the top and bottom row of the bolts, and so get more room. We did not do that because if we displace the bolts the lever acting over that plate 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.

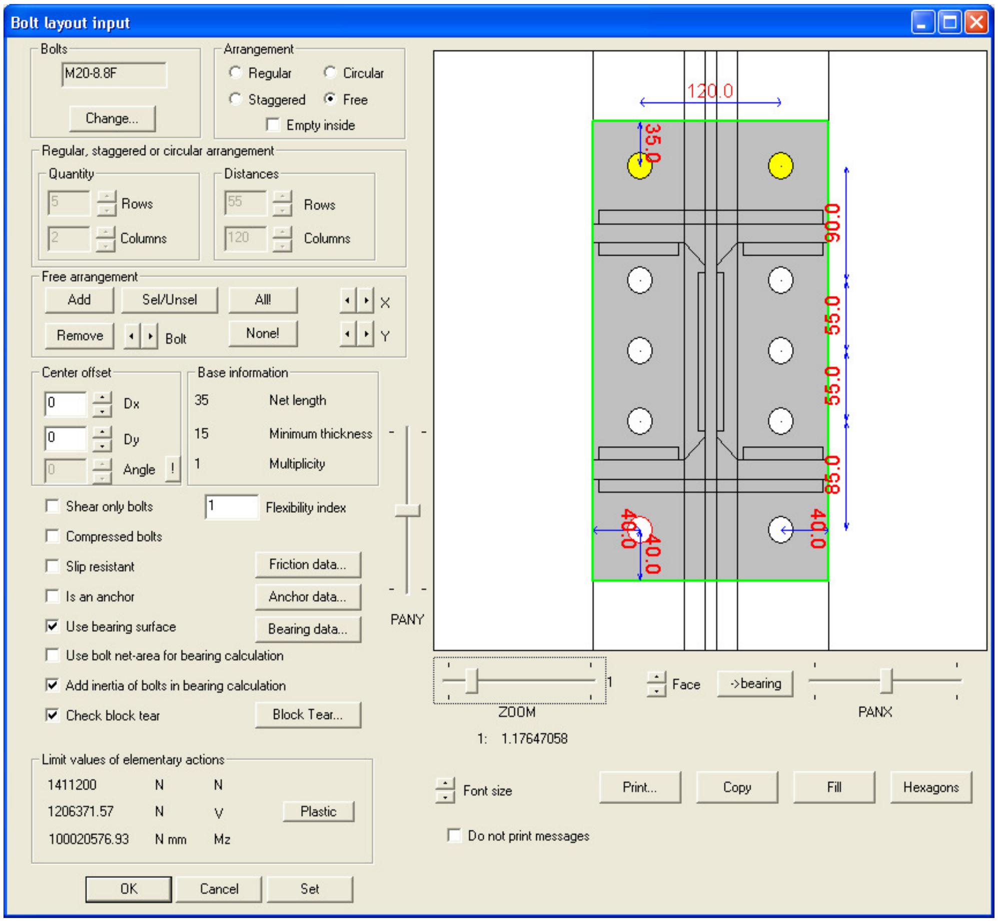


Then execute the command **Renode-Components-Modify** ( button in the left bar). This command is active when just one component is selected.

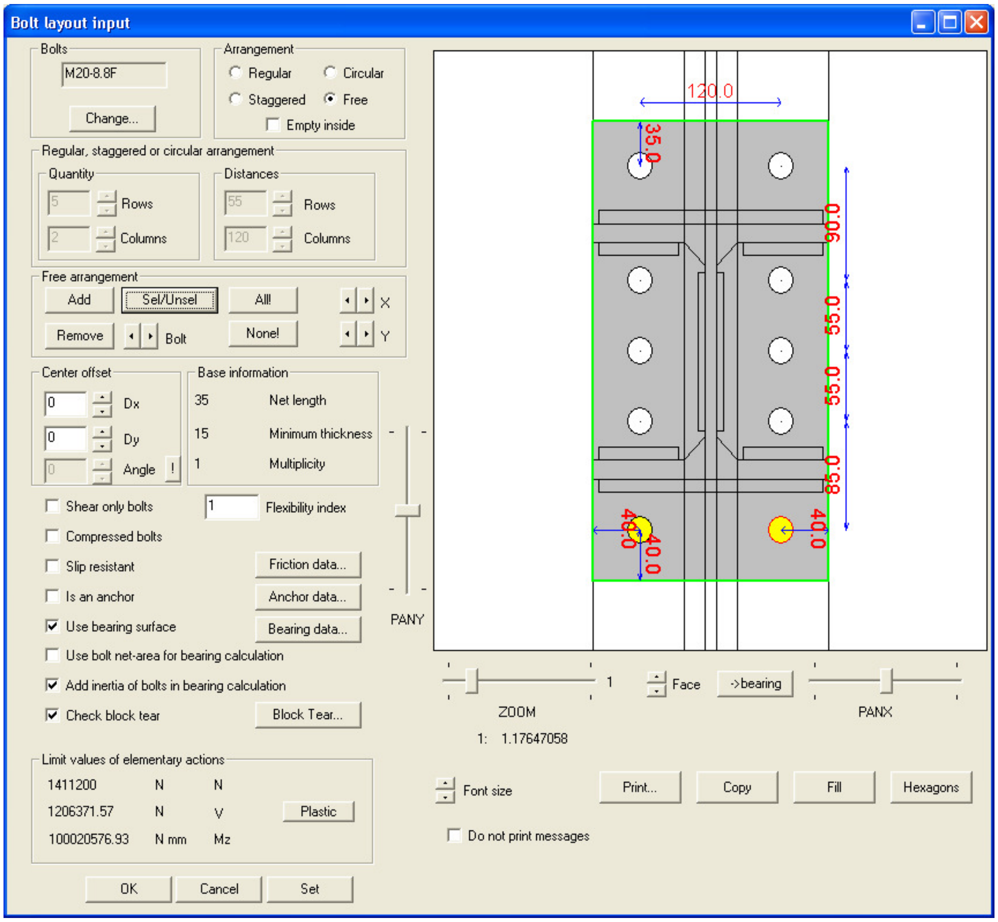
You once more get into the bolt layout main dialog. Like this:



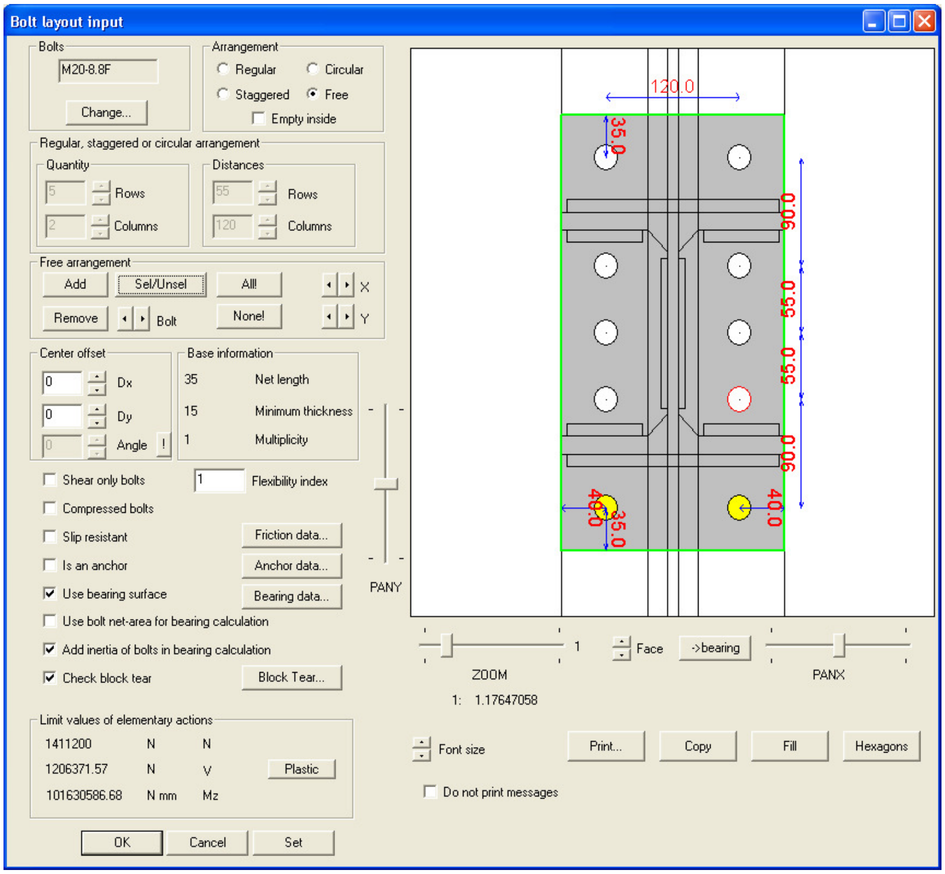
Notice that the two bolts of the top row are already selected. Using the right arrow in the Y control, decrease the 40mm distance from the top of the plate to 35. Like this:



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

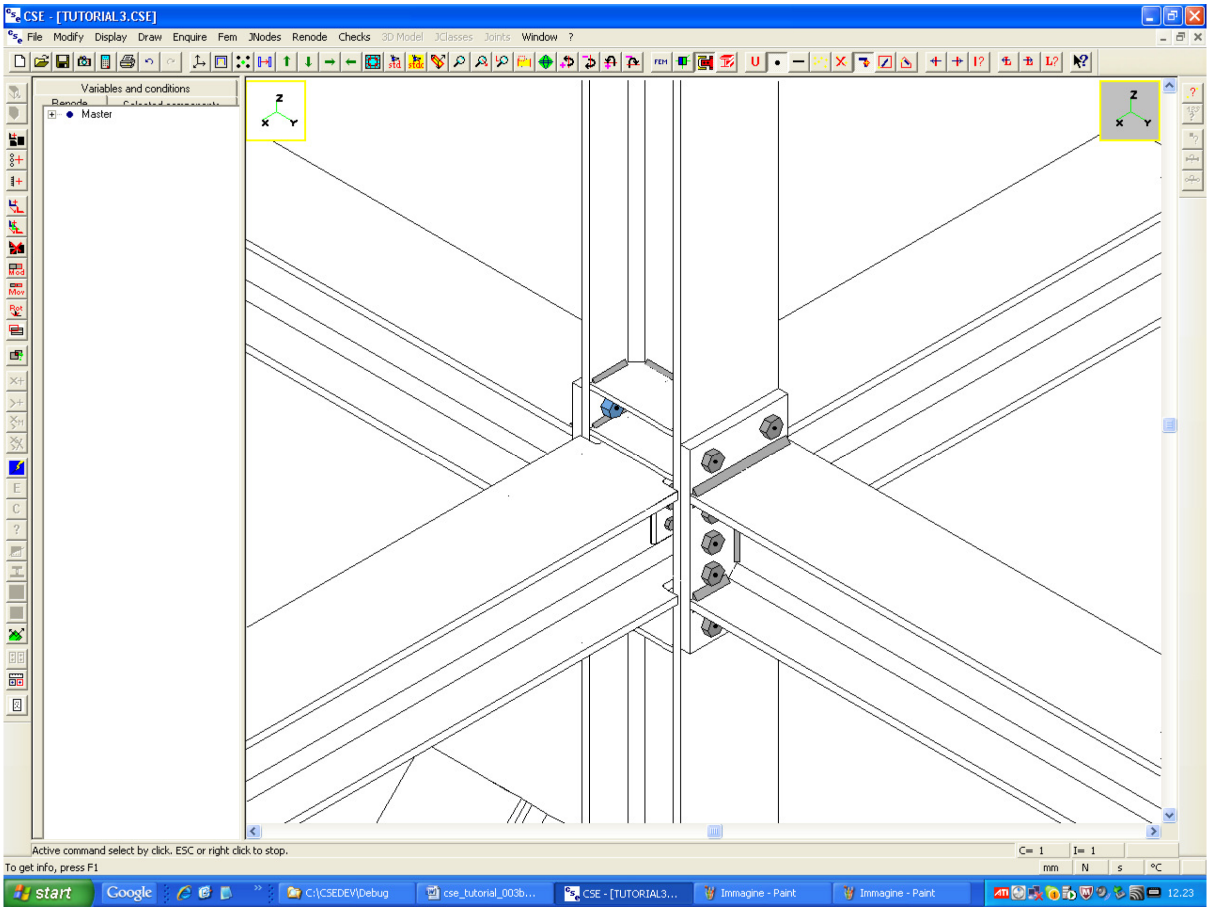


Using the left arrow change the 40mm distance to 35, like this:

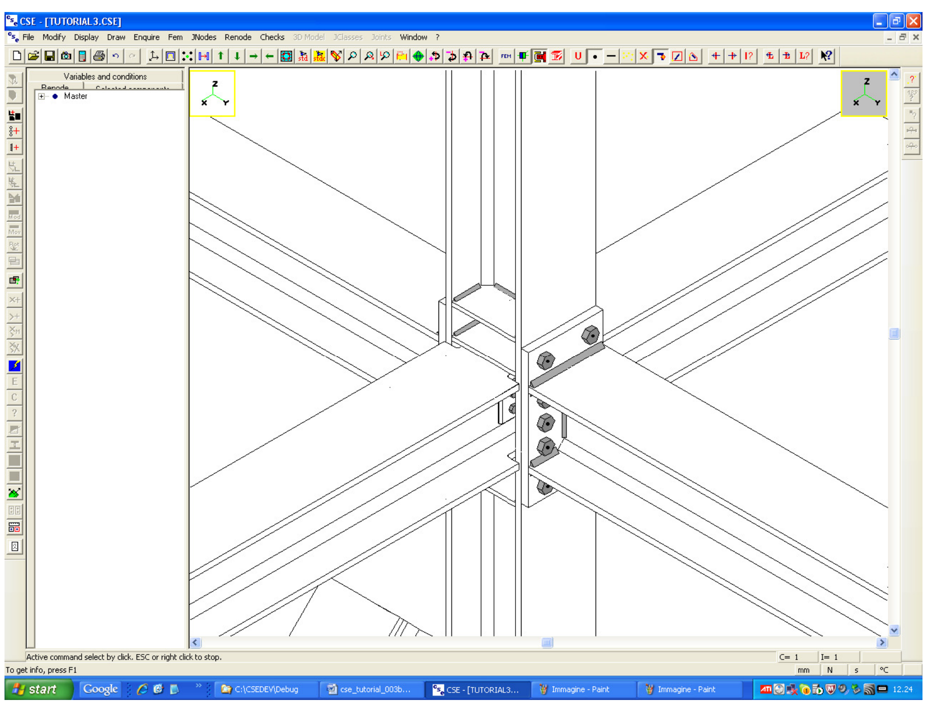


Press **OK**.

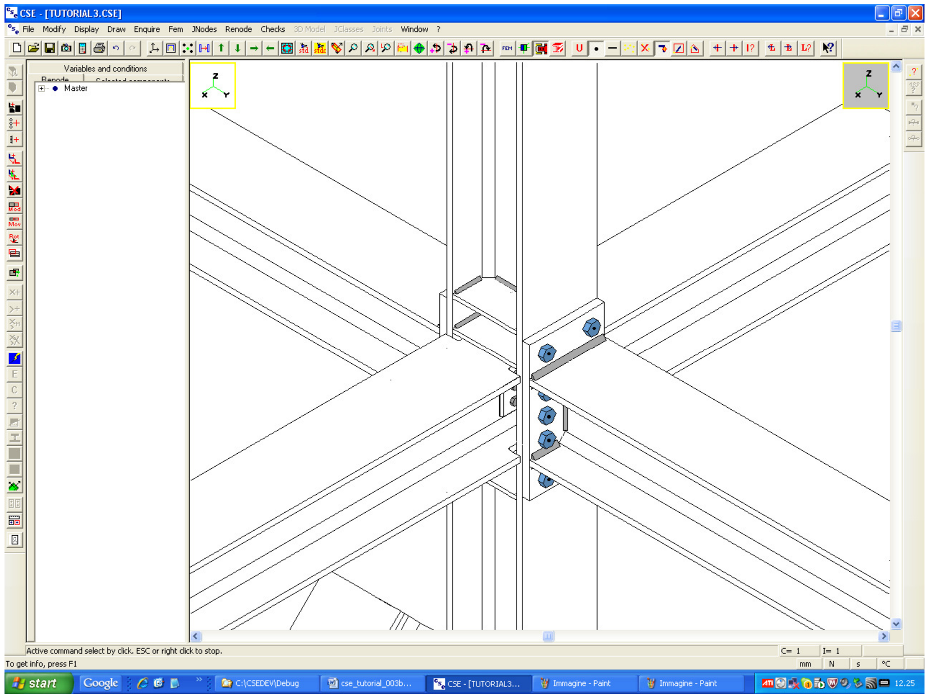
Now wee have to do the same for the other bolt layout. We can select the other bolt layout and do the same, or we can maybe do this, which is faster. Unselect all and select the other bolt layout. Like this:



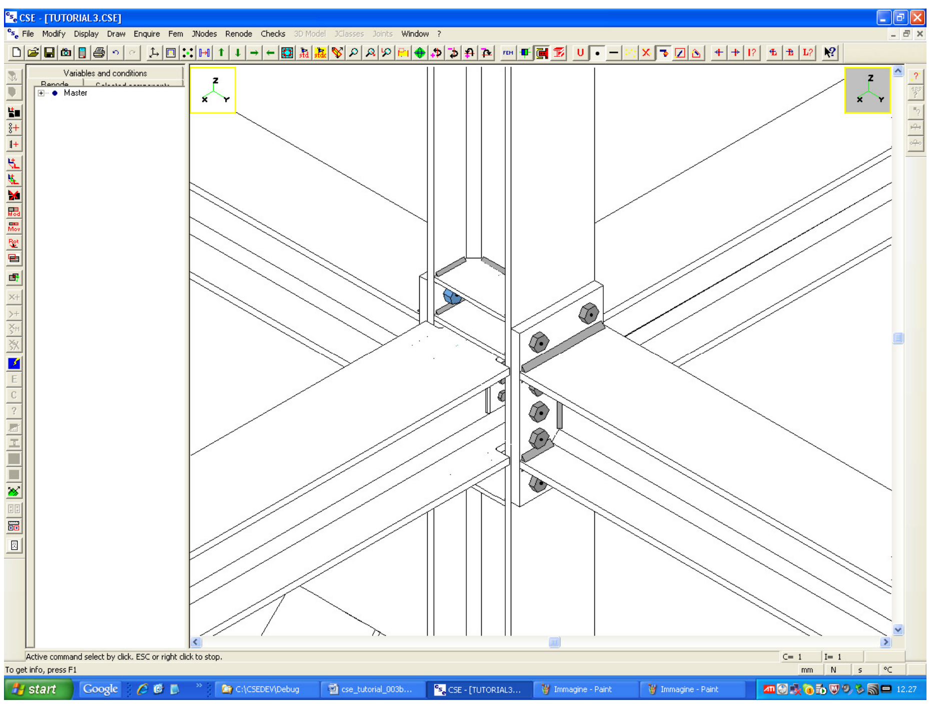
Now delete it. Use the  button in the left bar. The bolt layout is deleted.



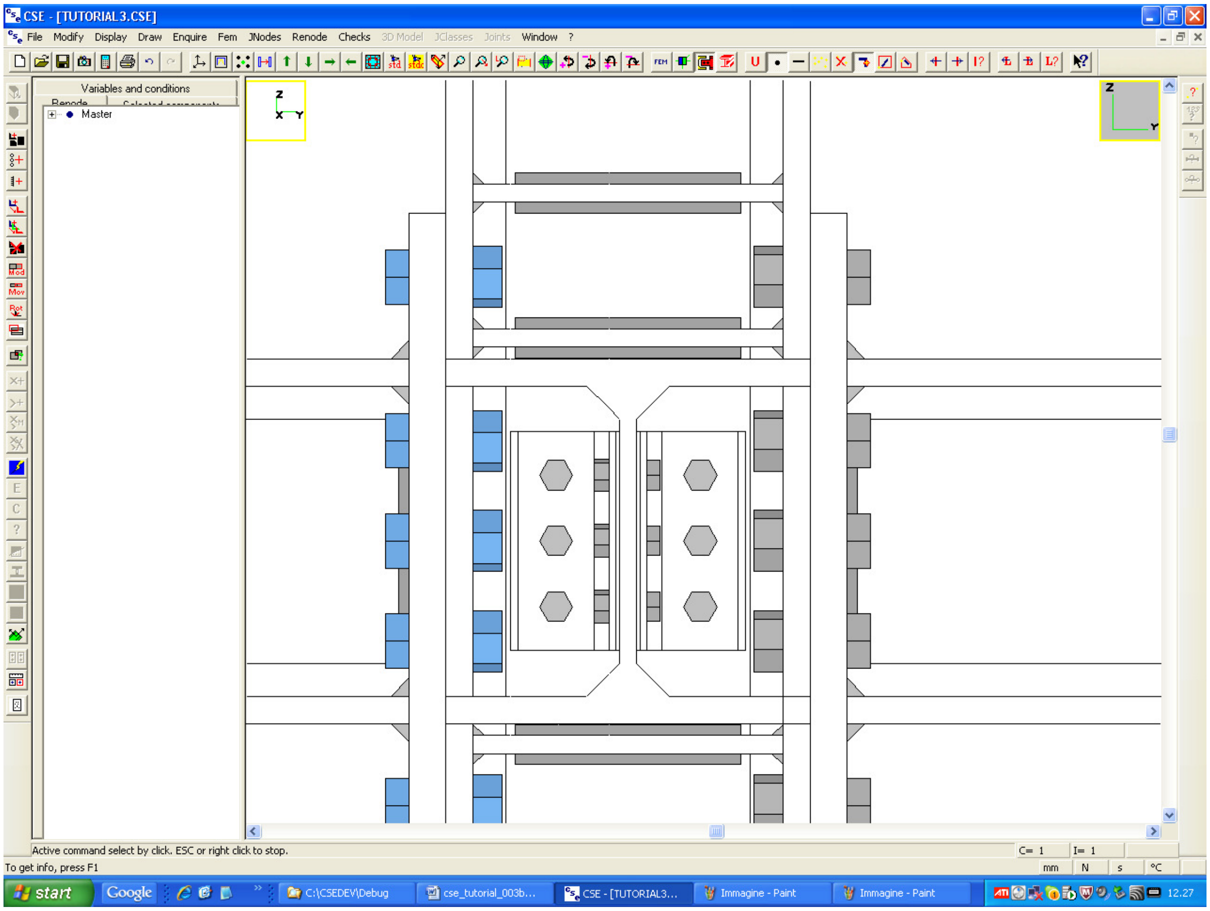
Now select once more the first (and unique) bolt layout like this:



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

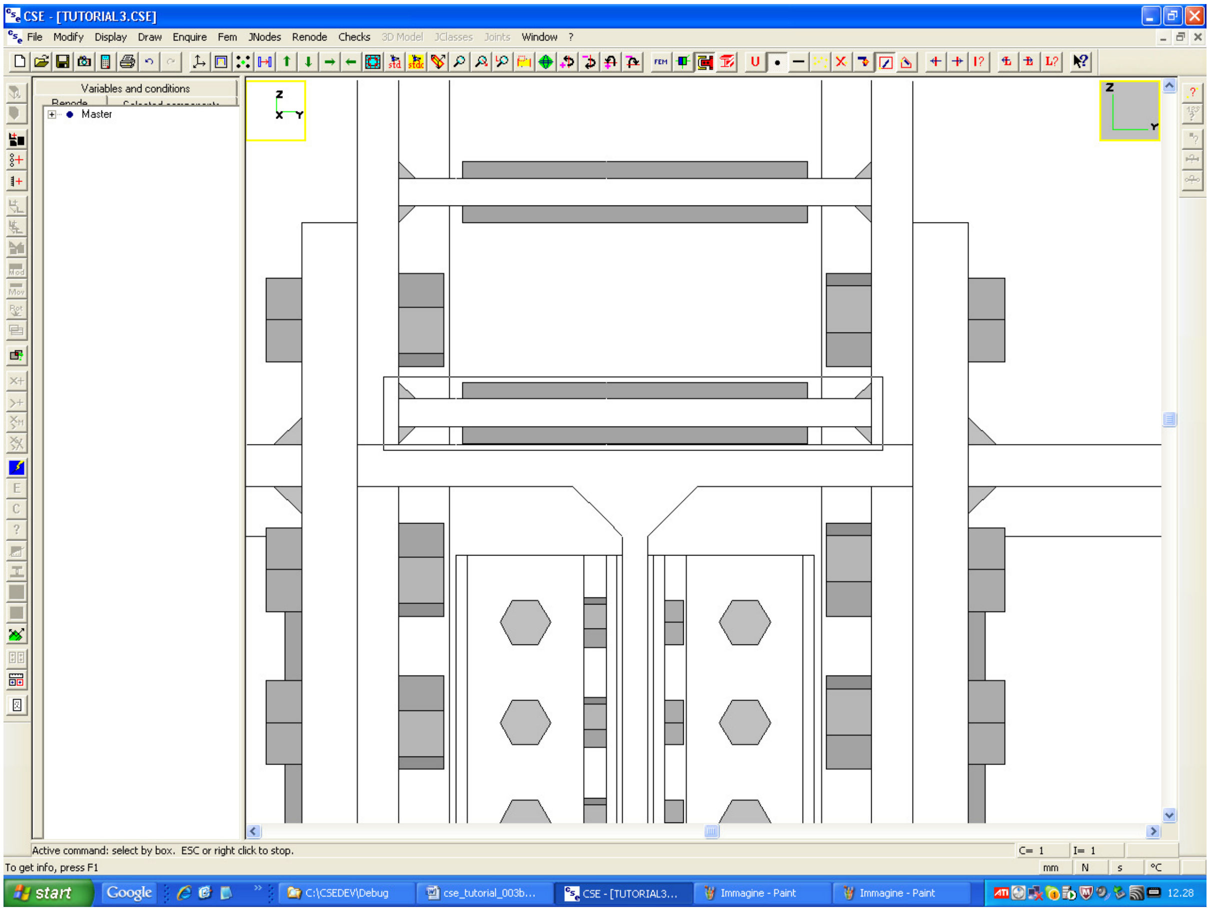


Now have a look from +X view:

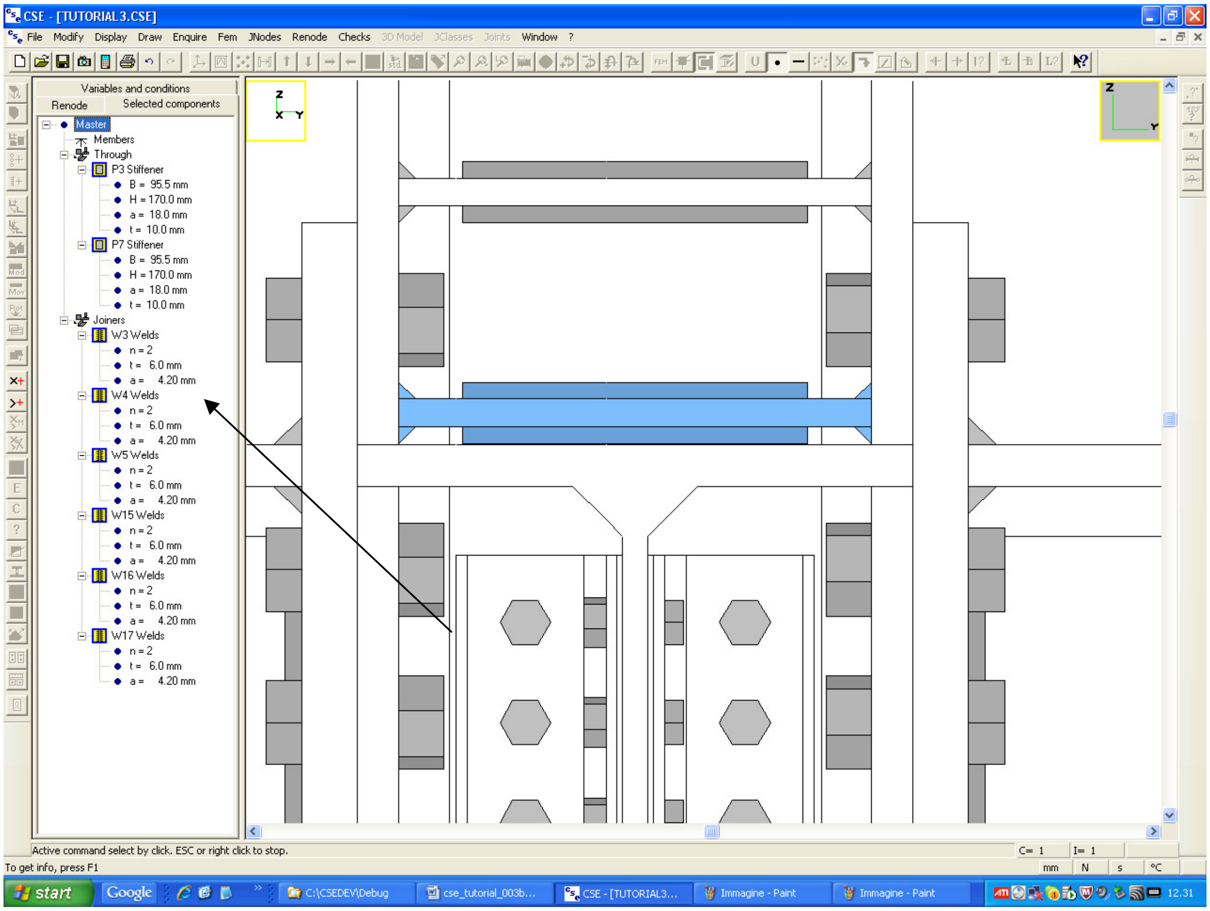


There is more room now. We can shift the stiffeners a bit.

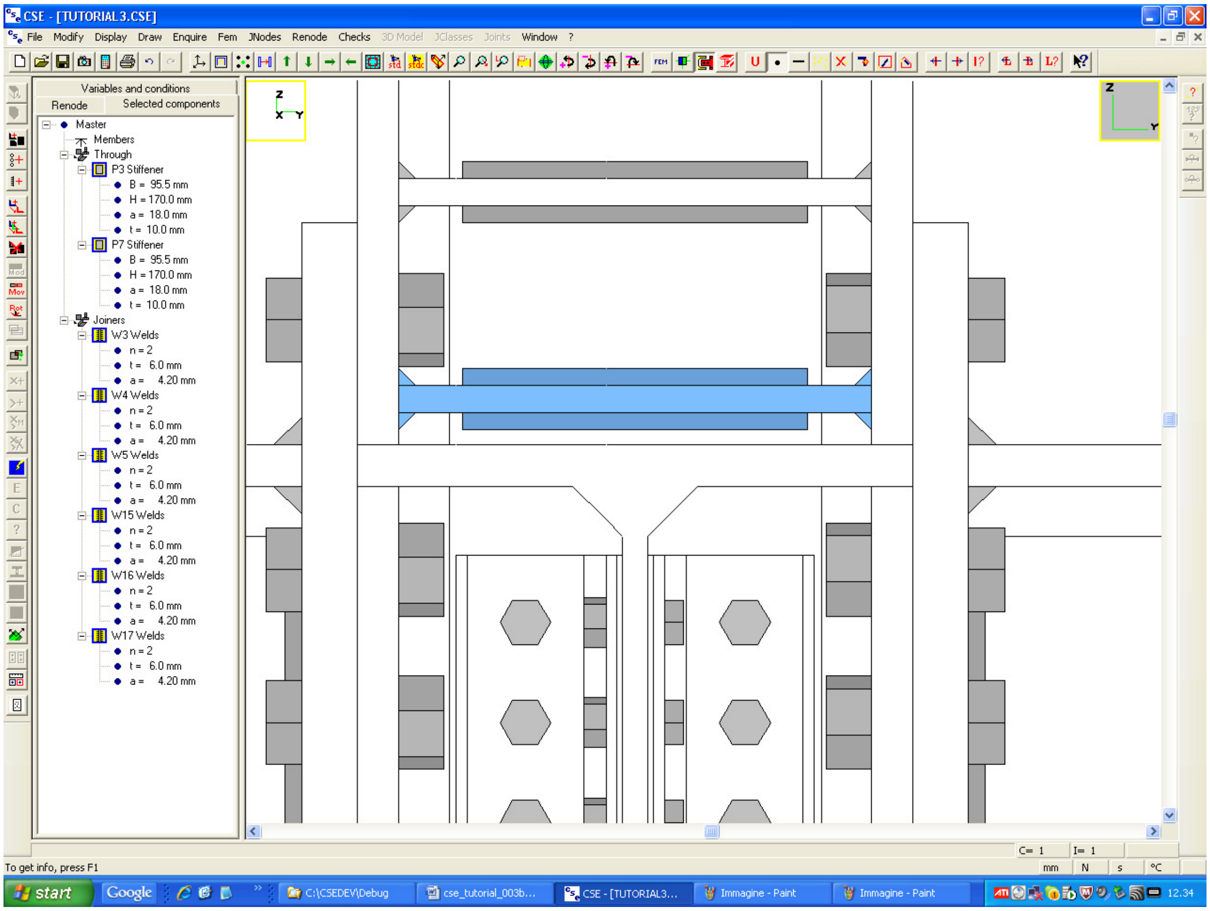
Unselect all. Now with a box we select the stiffener, its welds, and also the other side (hidden) stiffener with its welds. Execute the command **Modify-Select-Box** (button in the main bar). Choose a box like this:



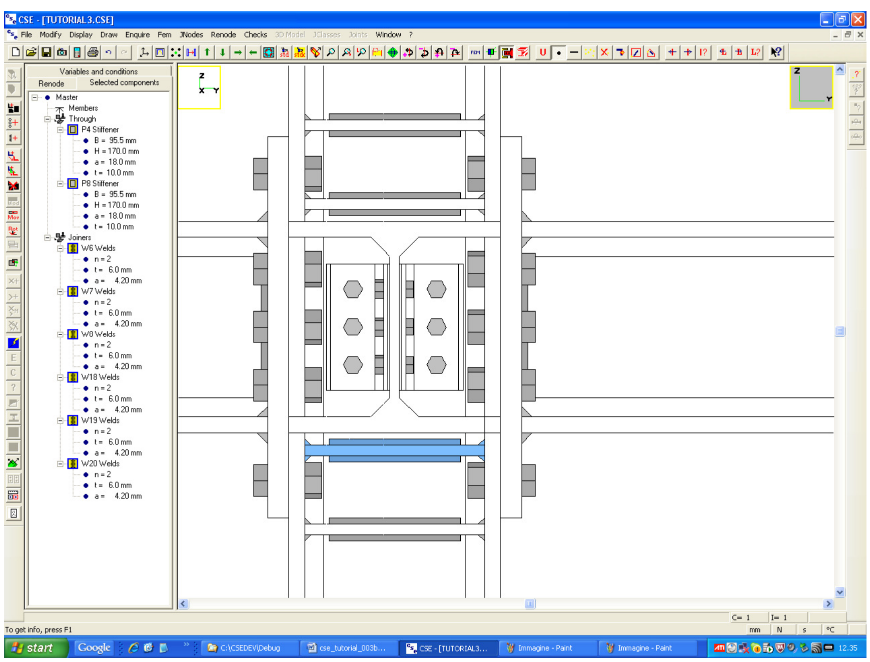
At the second click the objects are selected.



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** (button in the left bar). Choose the numeric blue mode and input +5mm for DZ. You see now this:



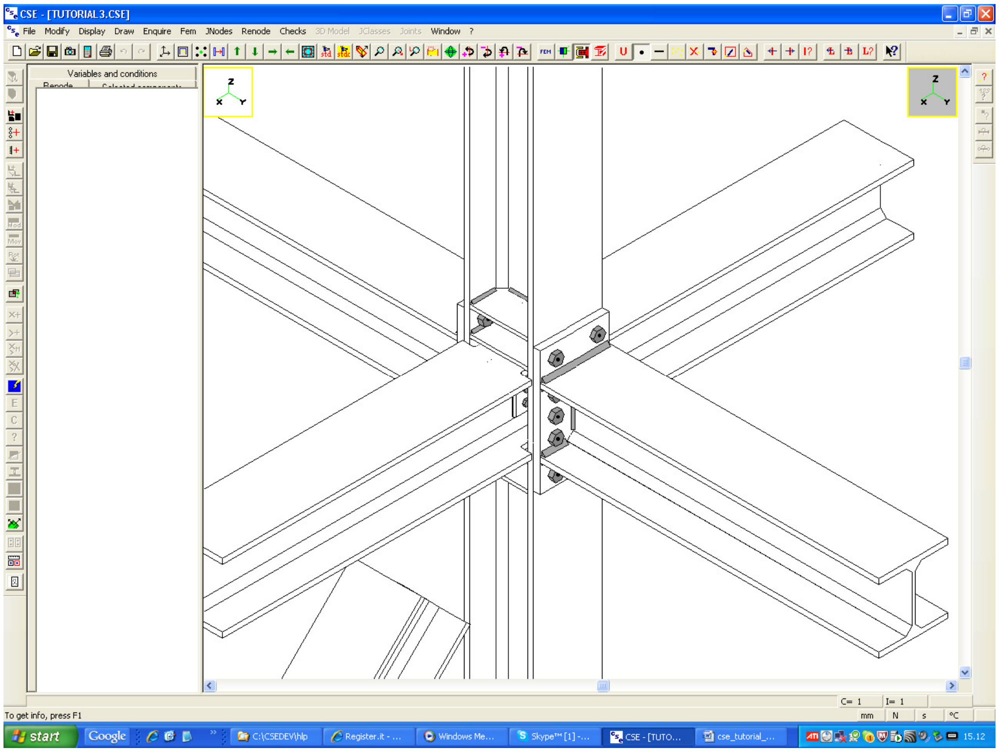
Now unselect all and repeat the operation with the lower stiffeners, shifting them -5mm in Z direction. You will see this, finally:



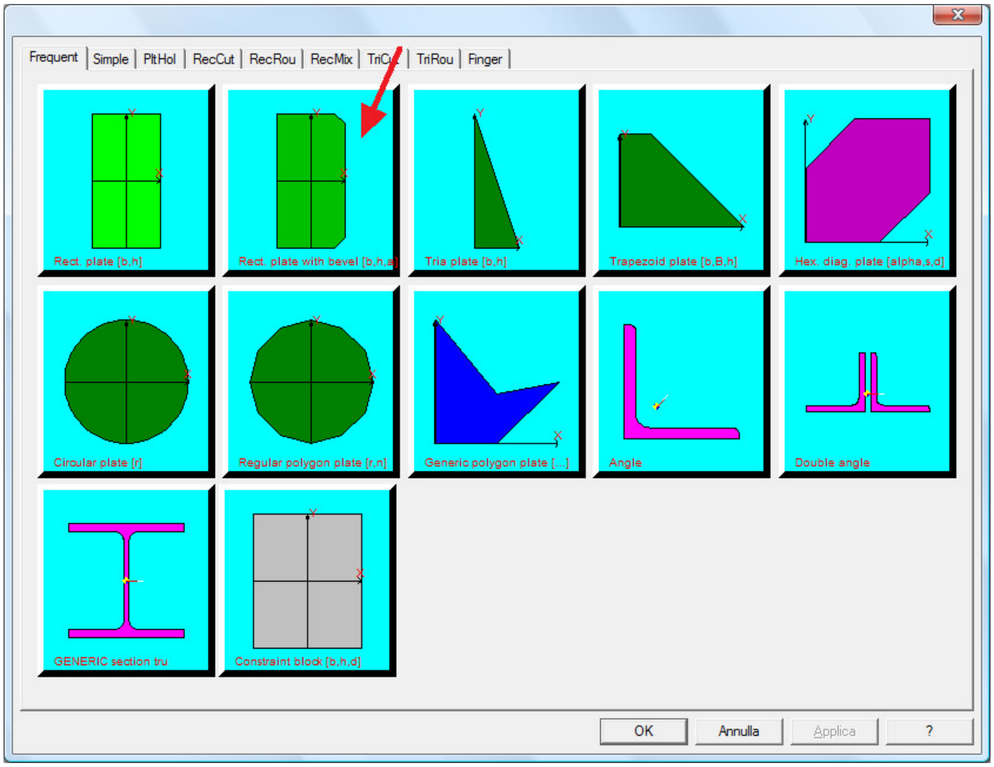
So we have successfully displaced the stiffeners to increase the room between them and the X beams.

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).

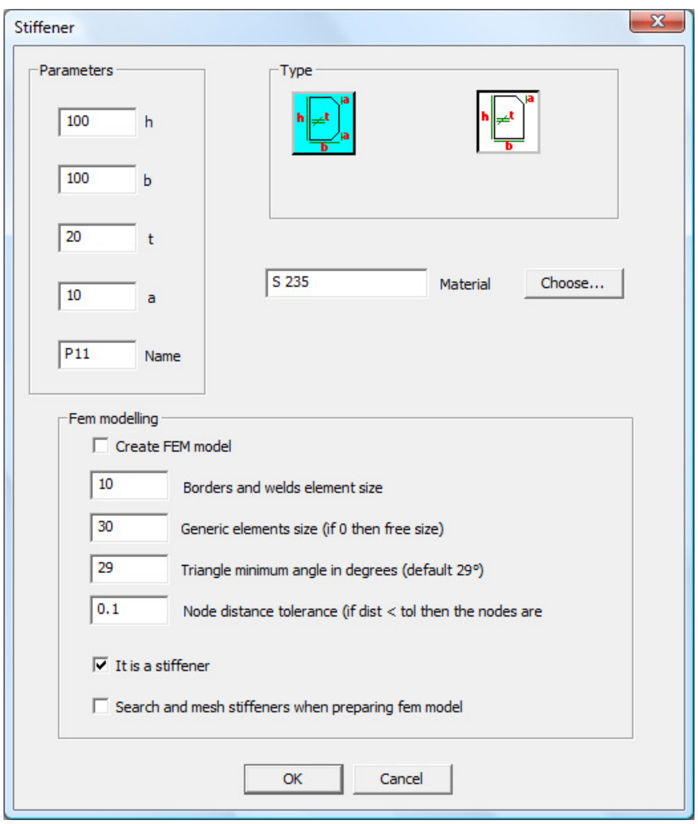
Choose an ISO view like this:



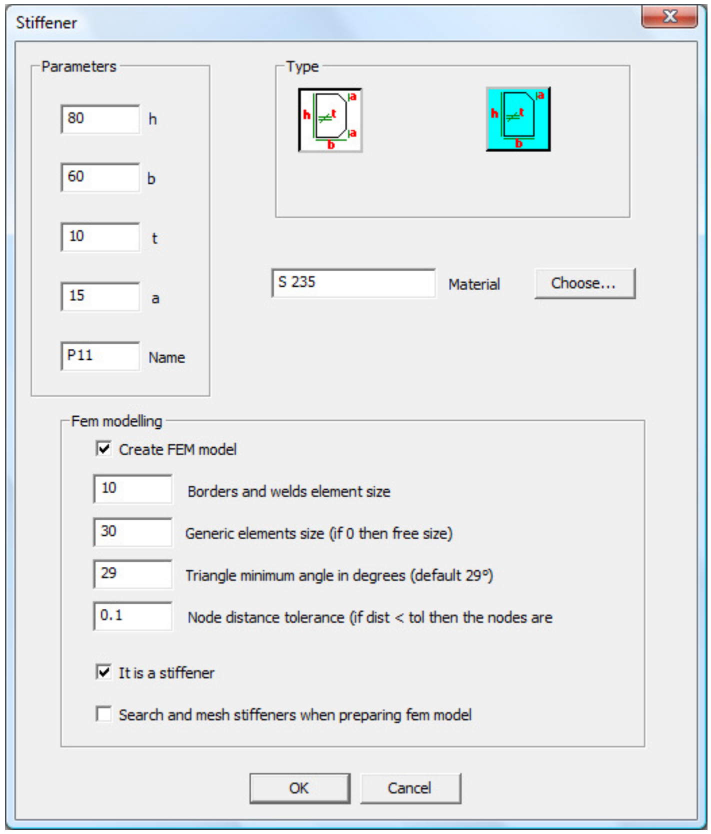
Execute the command **Renode-Components-Add through** ( button in the left bar). In this dialog



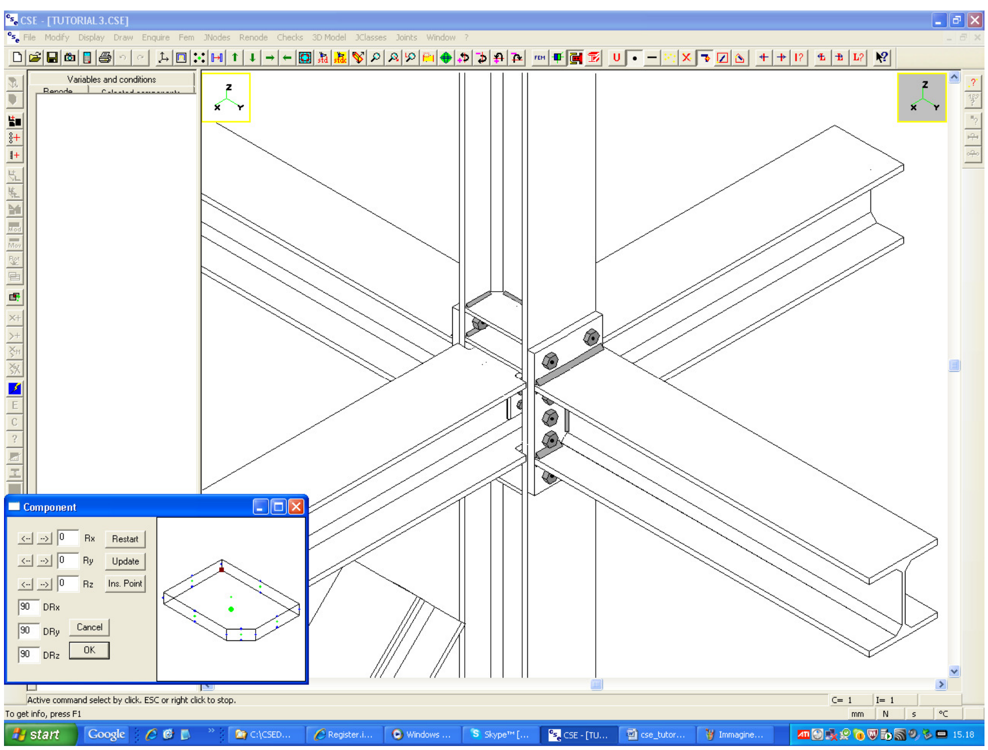
choose **rectangular plate with bevels**. You are now here:



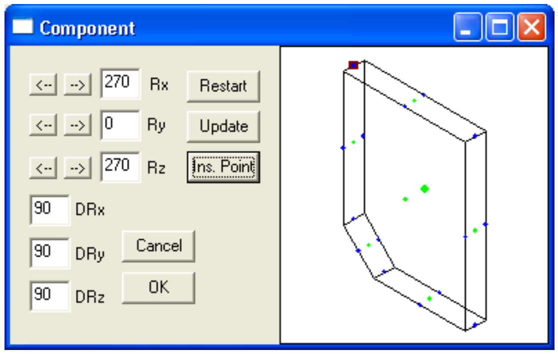
Press the right image-button to choose a one bevel plate (it gets light blue). Then choose the sizes in this way (we have understood these are correct sizes by using the **Enquire-Geometry** command):



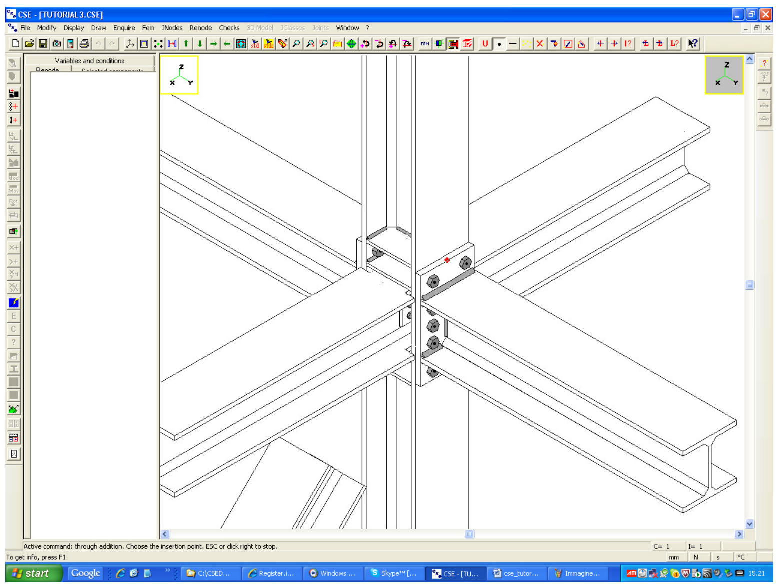
Now press OK and you are here:



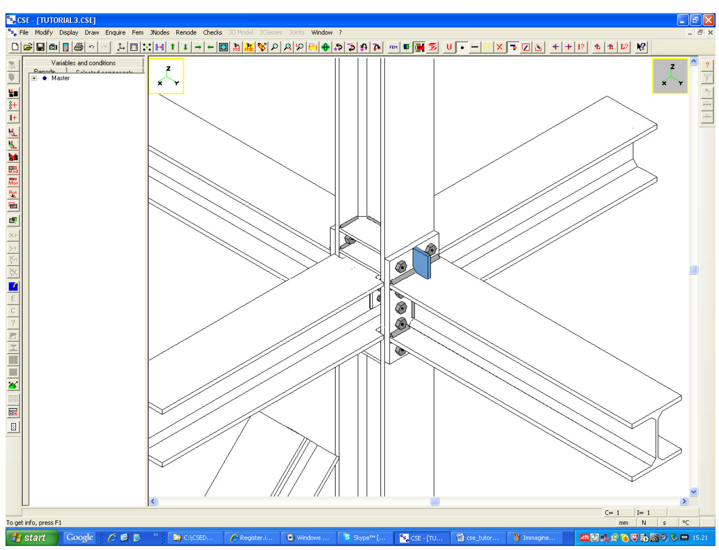
Press one the left arrow in the Rx row, then once the left arrow in the Rz row. Press the Ins Point button and choose the insertion point. Like this:



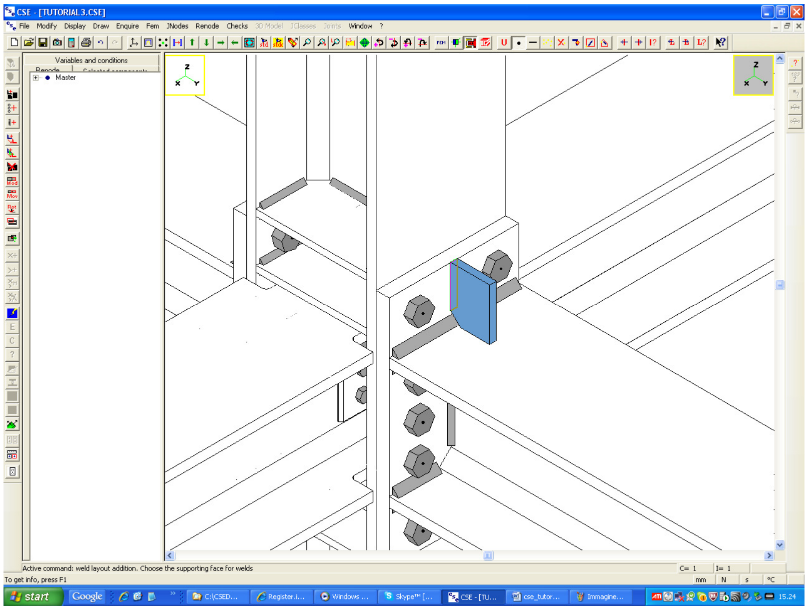
Now press OK and choose in the scene the proper point which is at mid-width of the plate upper dise, like this:



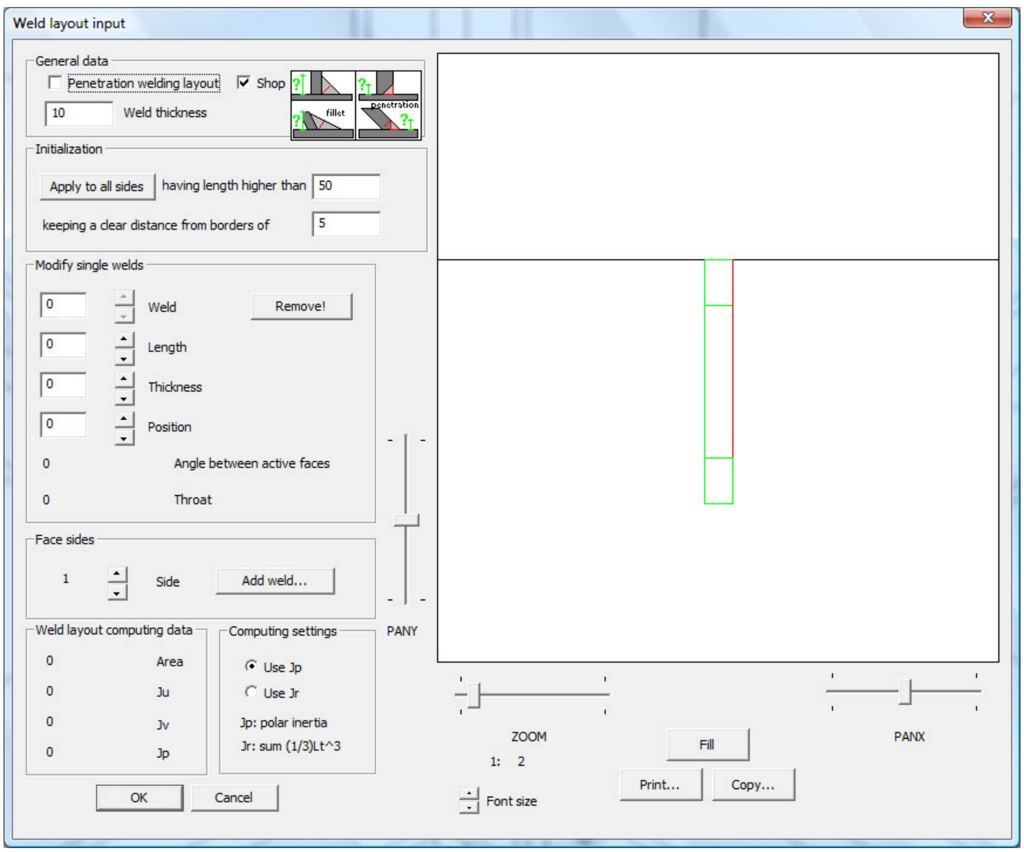
Be careful as there are more points in the nearby. You will now see what follows:



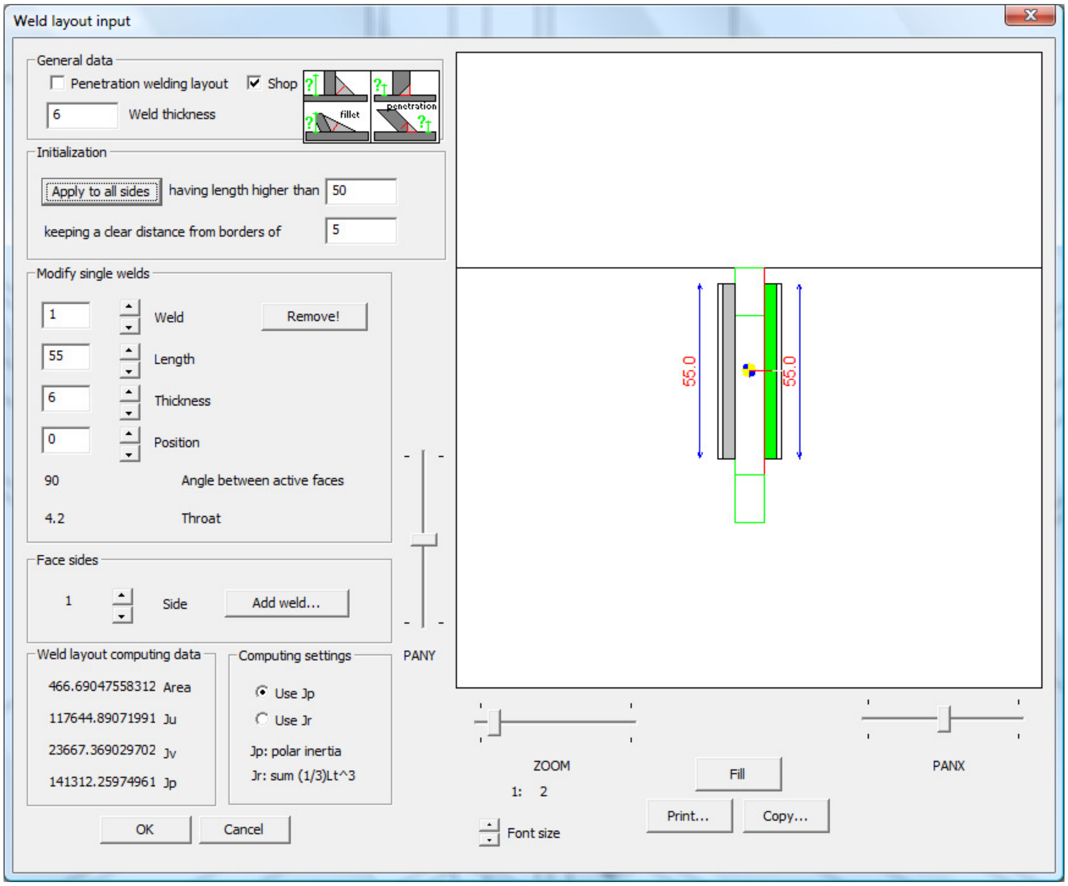
Now execute the **Renode-Components-Addition of a weld layout** command (the  button in the left bar), and choose the first face, like this:



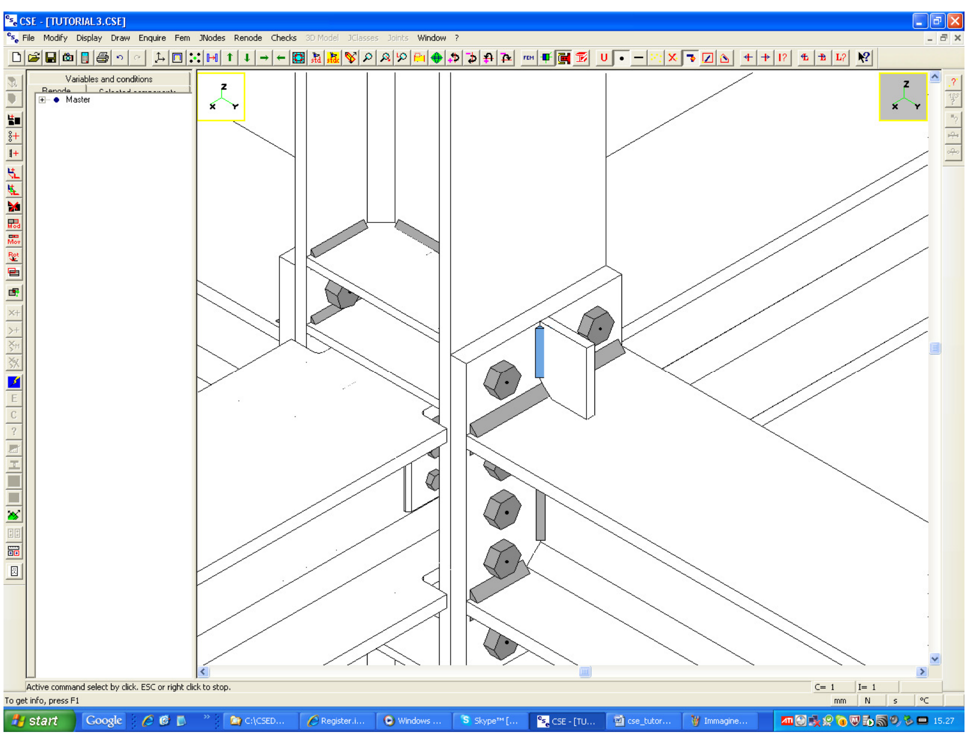
When you click left you are now here:



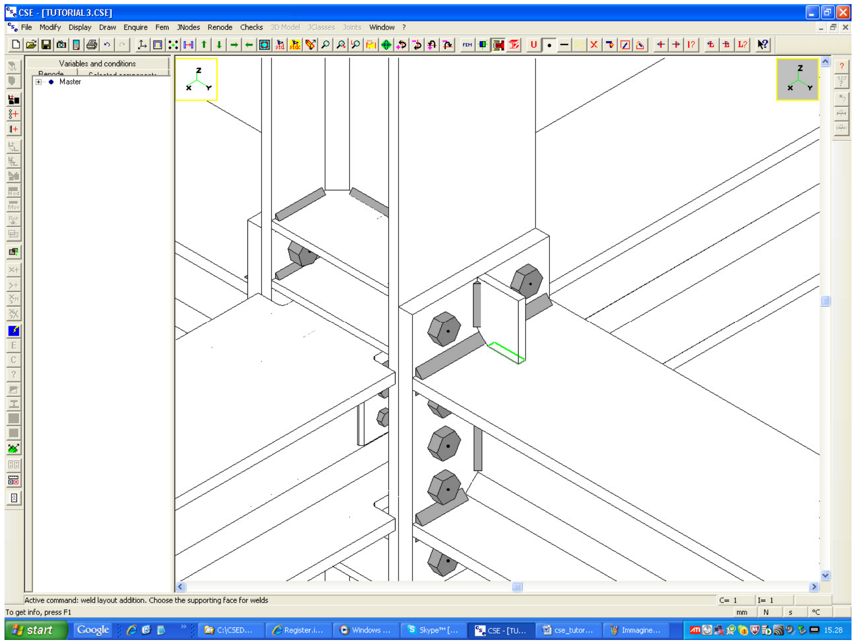
In the **Weld Thickness** edit box input 6. Then press the **Apply to all sides** button, you see what follows:



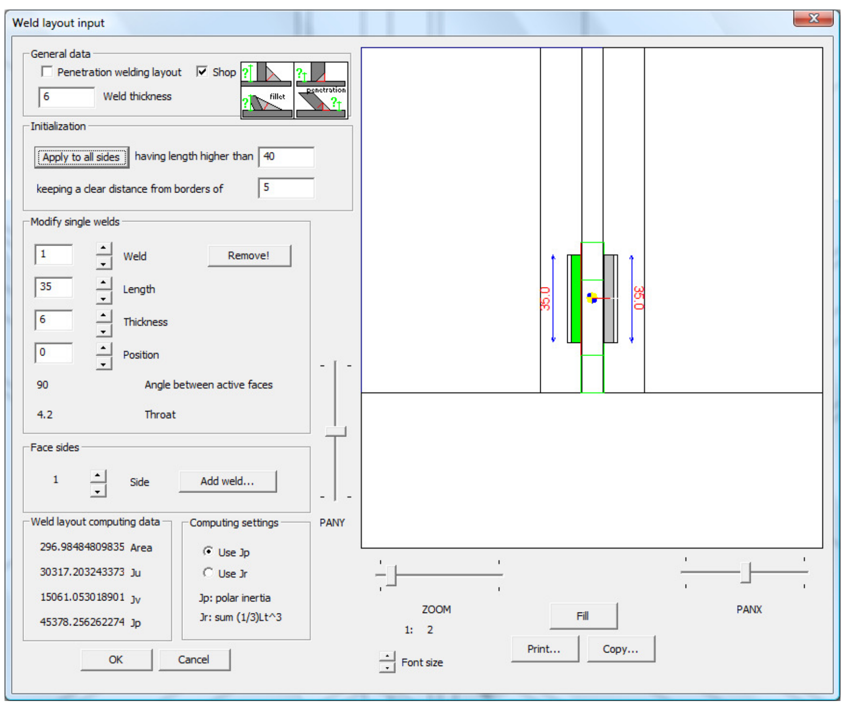
Press OK, you have placed the joiners between the stiffener and the plate:



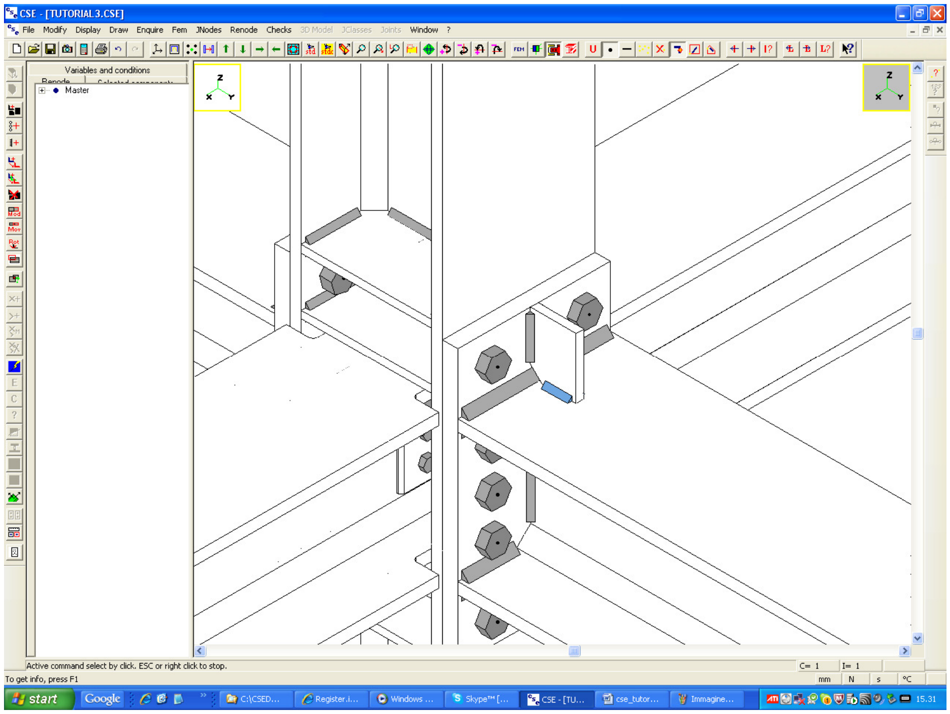
Unselect all. Now re-execute the weld layout addition command and choose the other face:



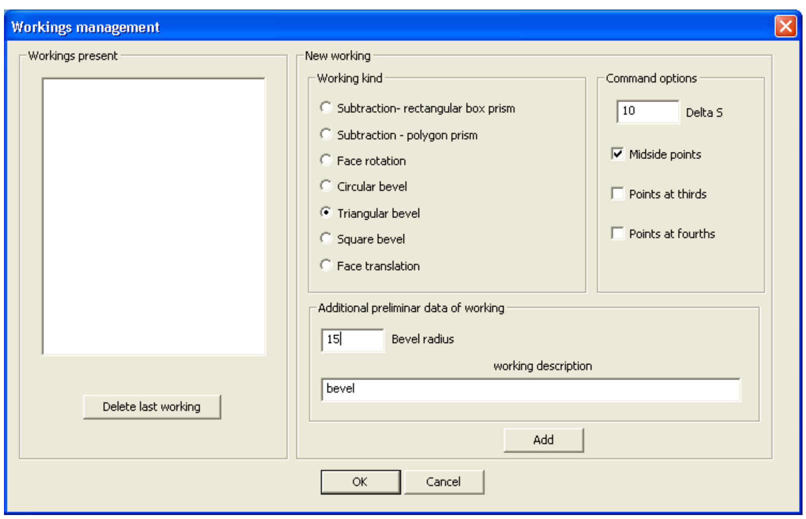
Click left, input 6, then press the **Apply to all sides** button and note that no weld is applied: this is because the sides are shorter than 50mm (60-15=45). So input 40 in the "having length higher than" edit box, and re-press the Apply to all sides button: you now see the following:



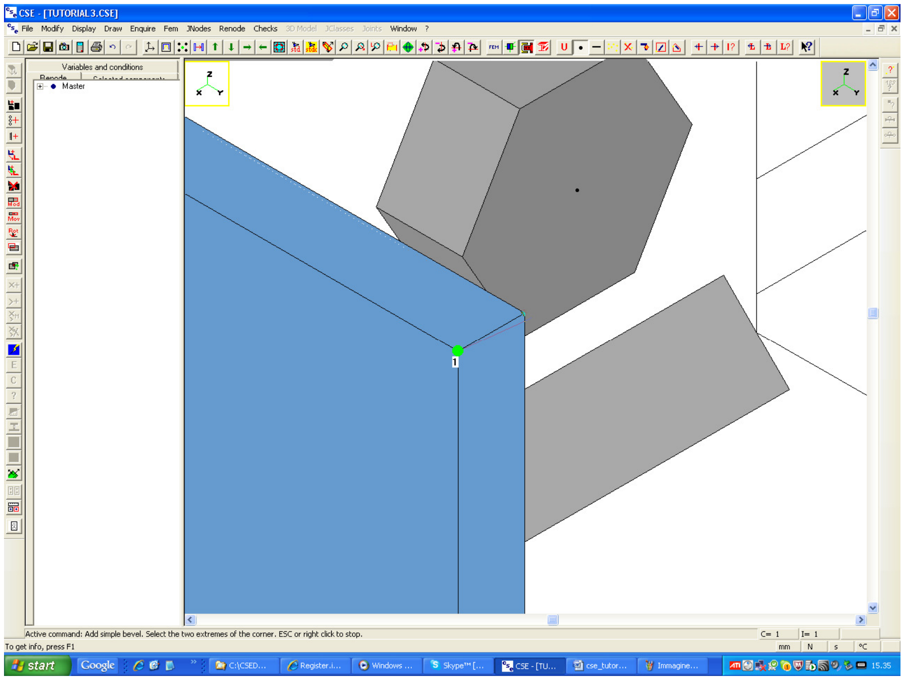
Press **OK**, and the stiffener is joined:



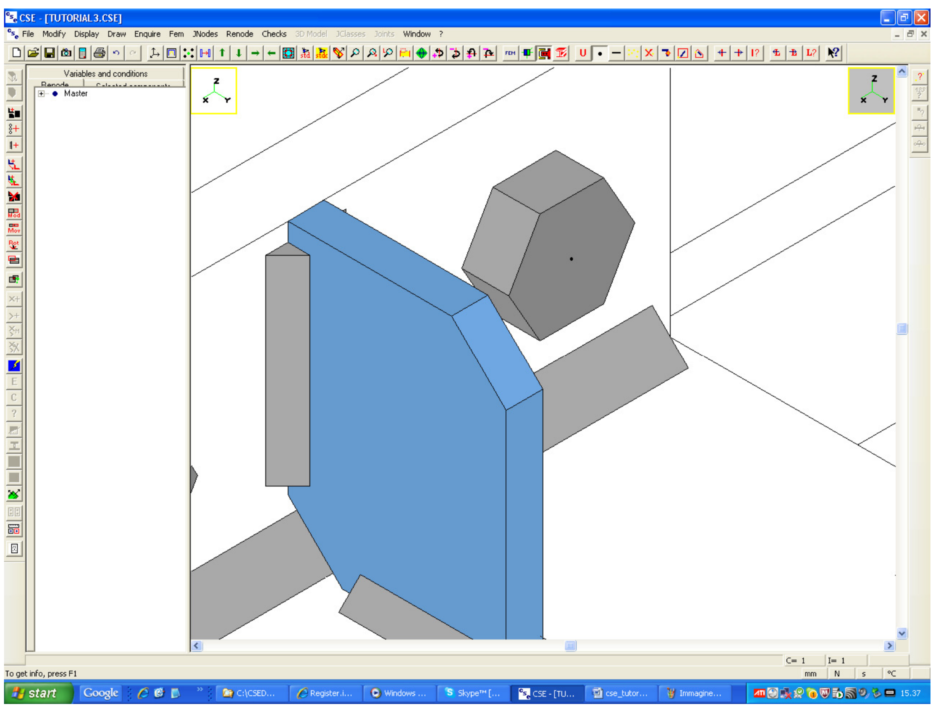
As an exercise (it is not strictly necessary, let's apply a bevel to the stiffener). Unselect all (button in the main bar) and then select only the stiffener. Then execute the command **Renode-Workings** (button in the left bar), and choose **Triangular Bevel**, adding a name for this action, e.g. "bevel",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



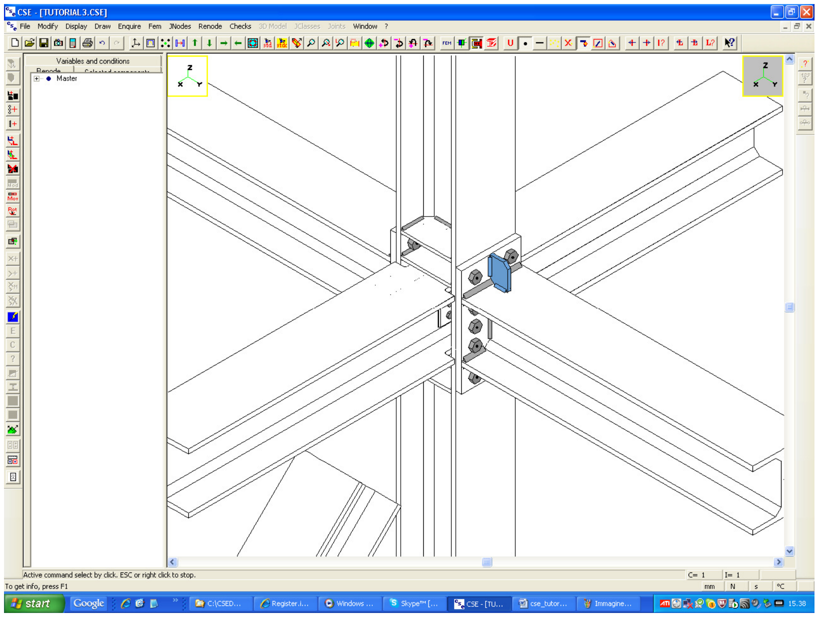
Press the **Add** button. Now select the two points of the side to be cut, like this:

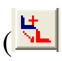


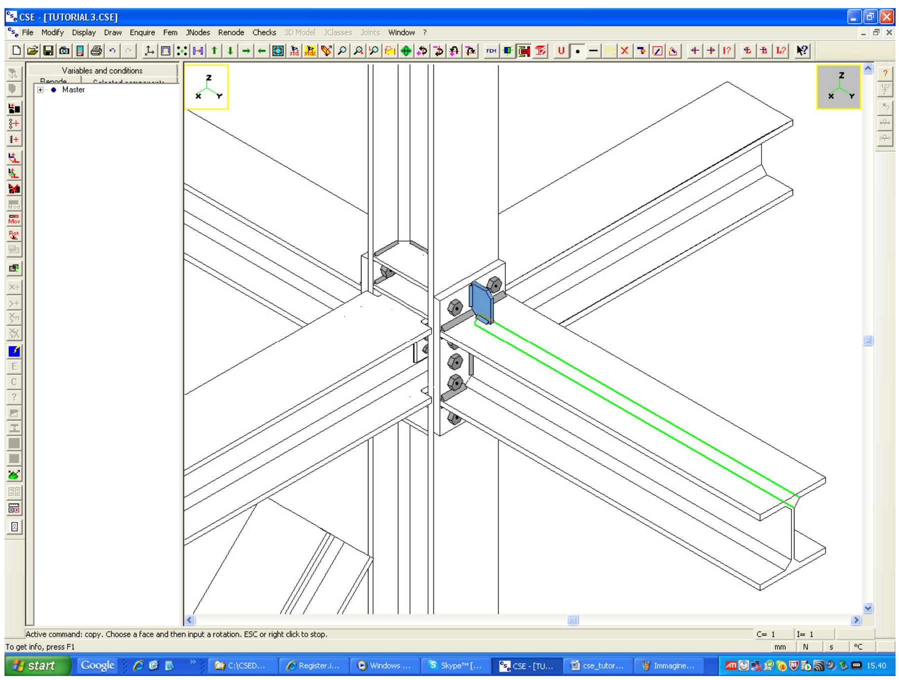
At the second click you see



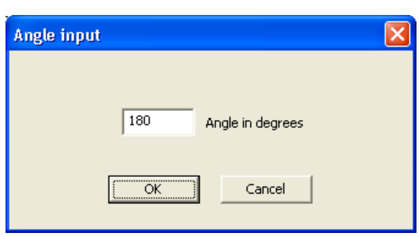
We are now ready to copy. Select the welds of this stiffener, together with the stiffener itself, like this:



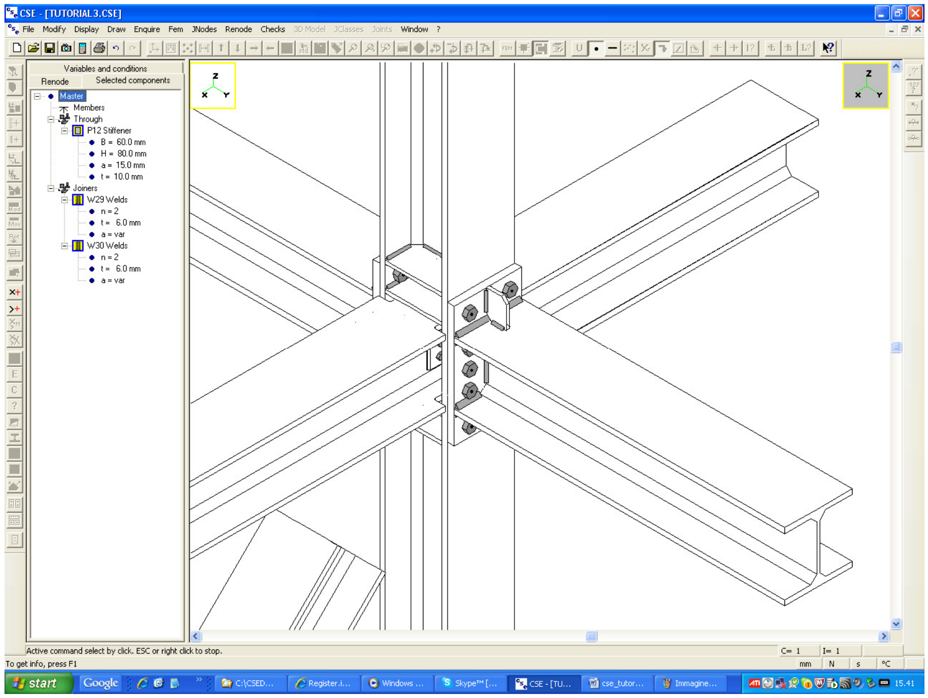
Now execute the **Renode-Components-Copy** command ( button in the left bar), choose the **Member +angle** mode, and select a +Y beam face, like this:



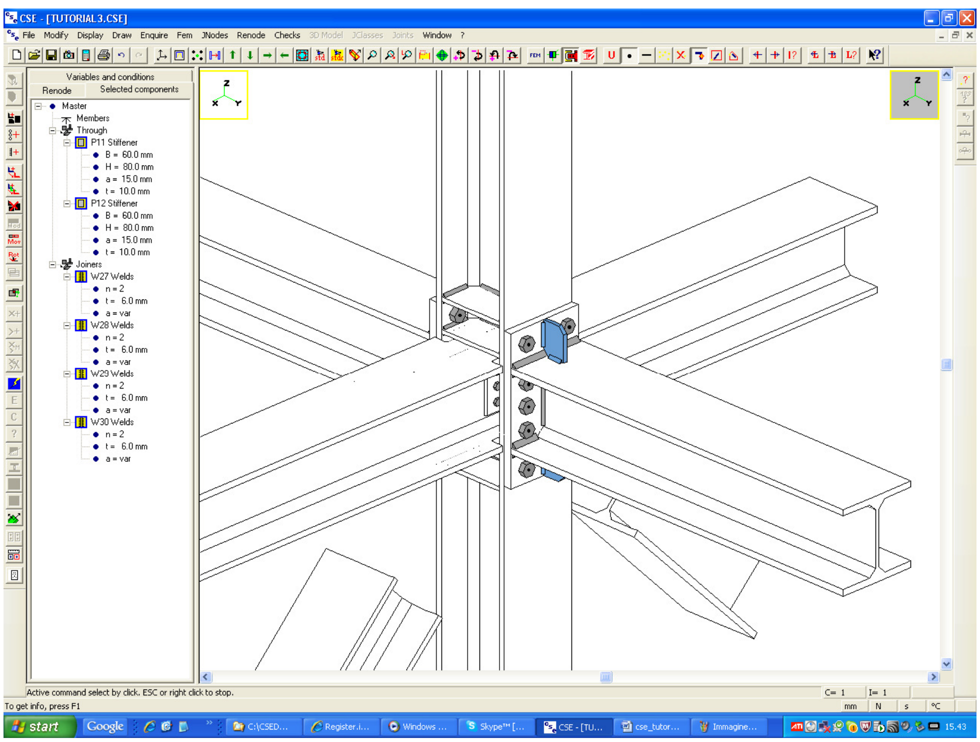
Then leave 180 in this dialog



and press **OK**. You get:

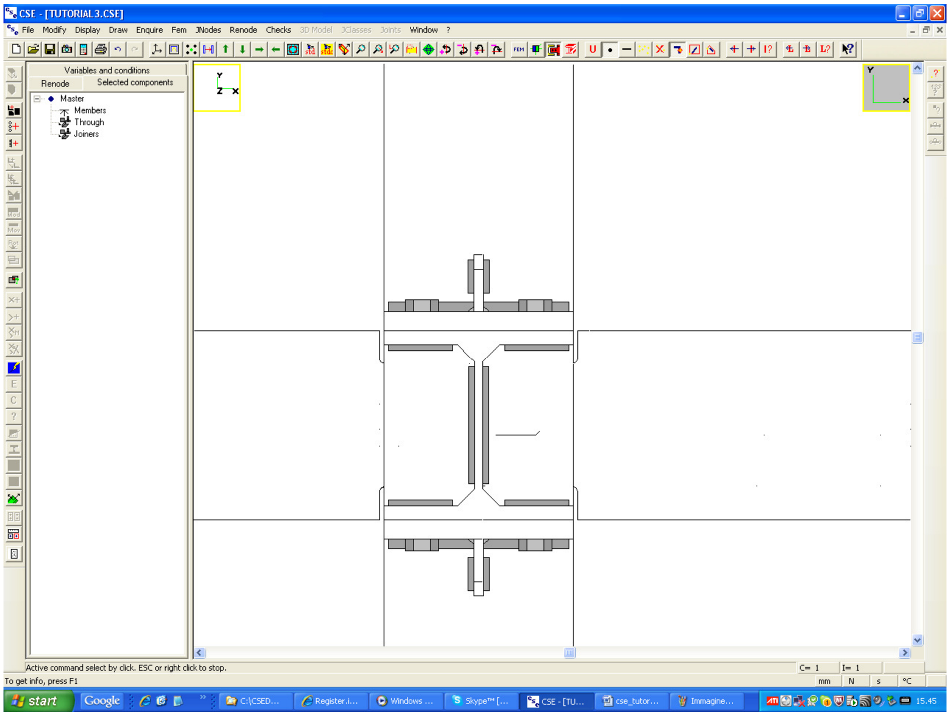


The generated objects are hidden but you can see them in the Selected Components pane, as shown in the picture above. Now add to the selection the original stiffener and its welds, like this (we have slightly changed the view):

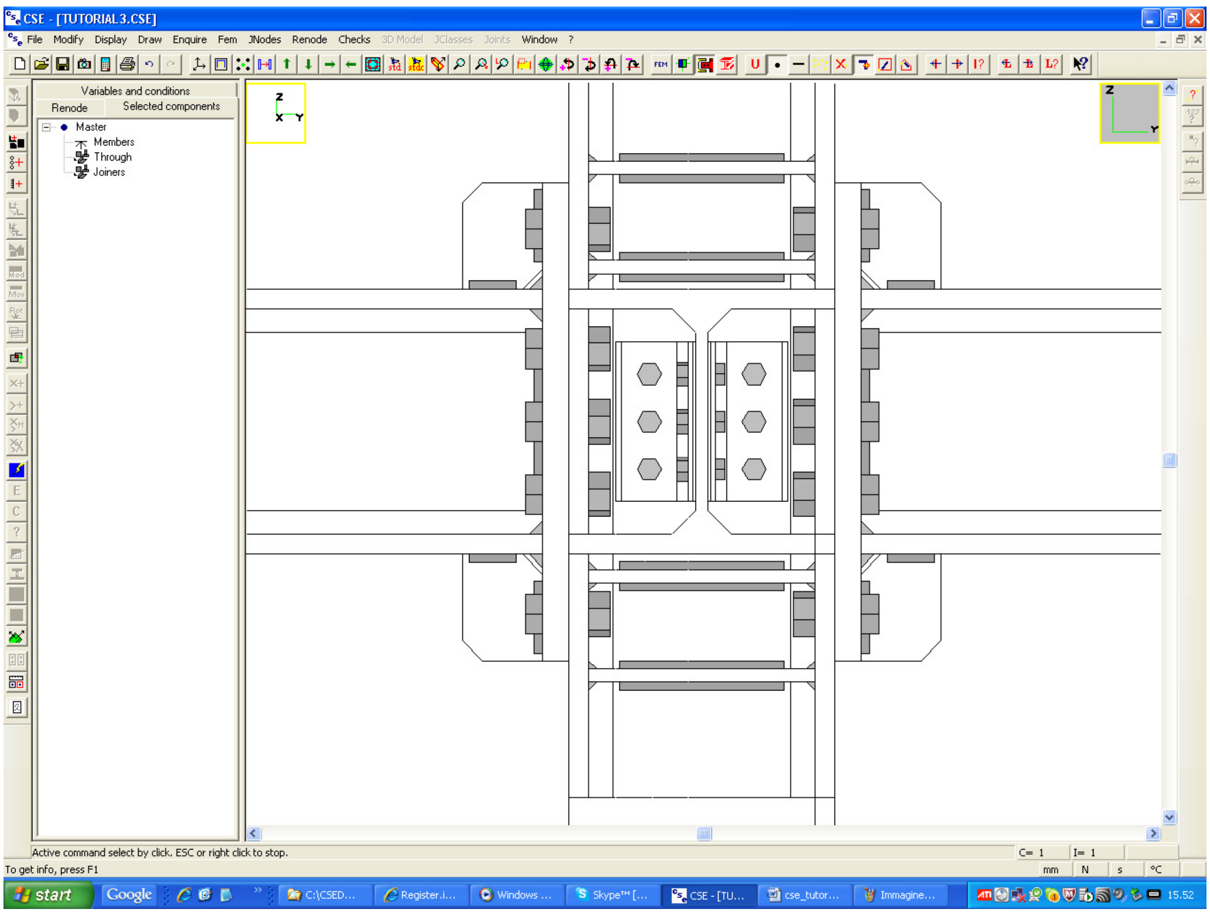


Re-execute the **Renode-Components-Copy** command, use the same **Member+angle** option, *but now select a face of the column*. Leave 180 degrees and you have finished.

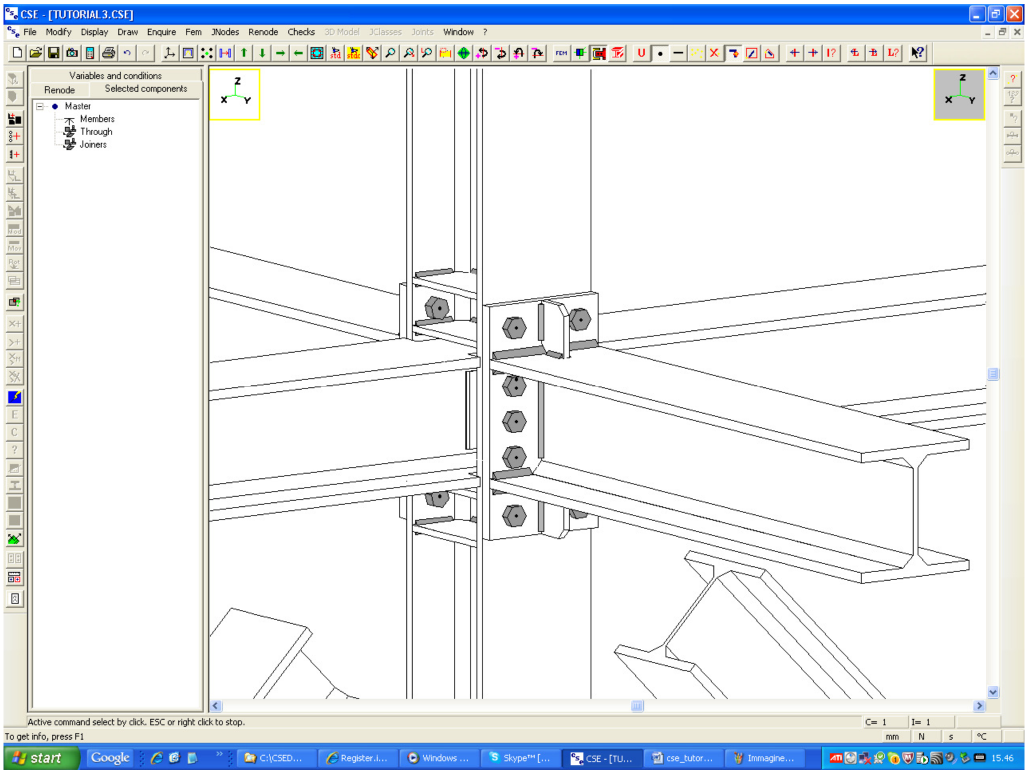
This is a +Z view:

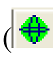


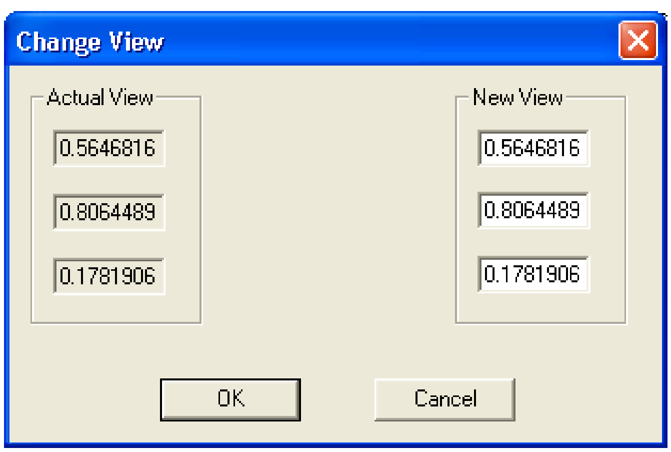
This is a +X view:



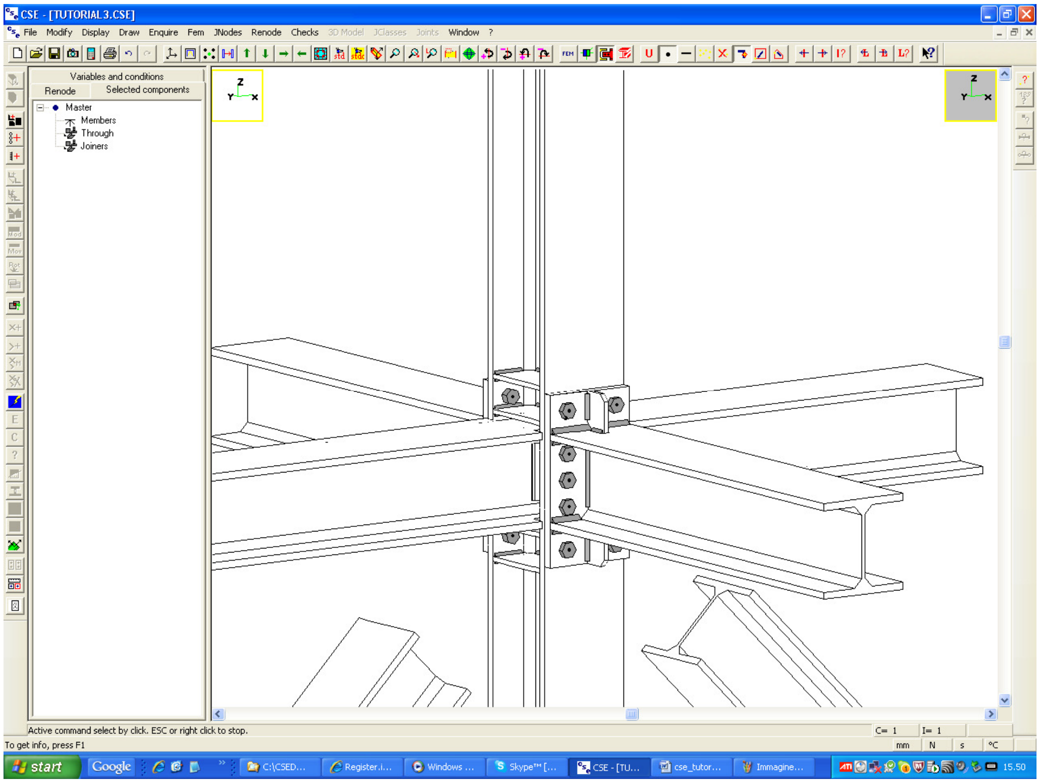
And this a adjusted view to see in +Y area:



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:



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.