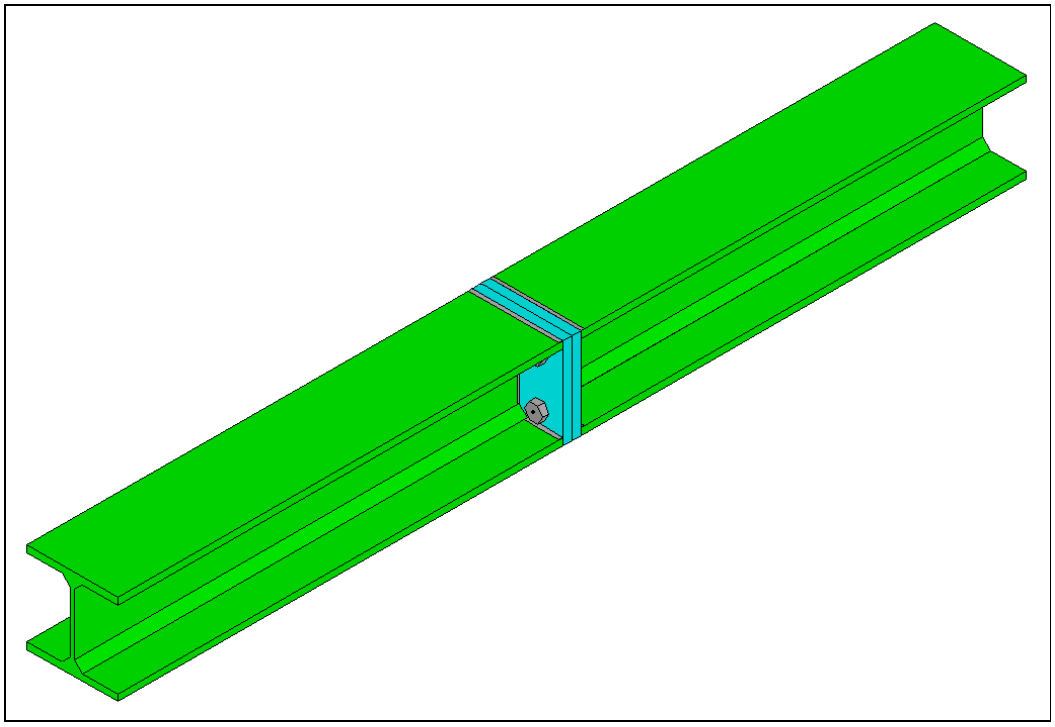
Paolo Rugarli

Connection Study Environment

Tutorial 4: Splice joint (with penetration welds)



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

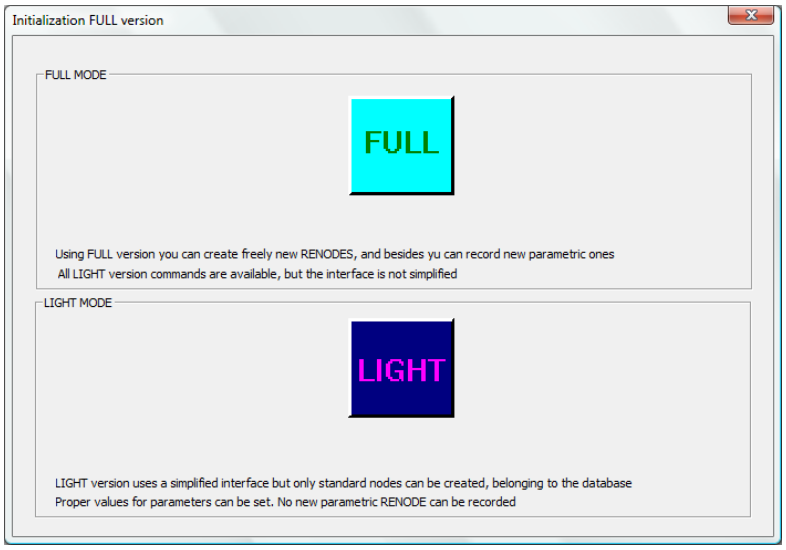
This tutorial is a tool to start the understanding of how CSE works. No special efforts to have realistic force values have been made, however this will explain several features of the program. By following this tutorial you will be able to:

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

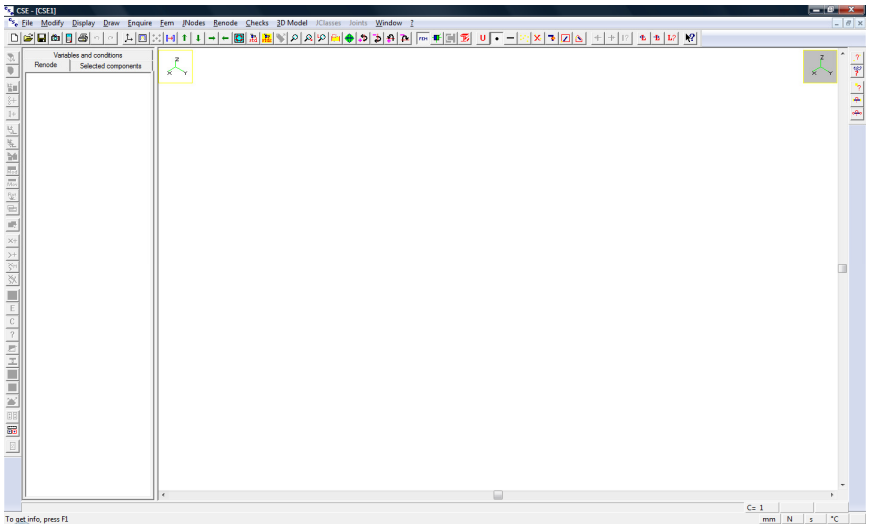
This tutorial is some like 39 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

N.B. this tutorial refers to CSE version reported on the first page of this document. If you are using a newer version, keep in mind that some dialog or commands may be different, although the logic of the program has remained the same. If you find some differences, see the up-to-date PDF guide or the context sensitive help for information.

2 SPLICE JOINT WITH PENETRATION WELDS



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



Initial window content: blank.

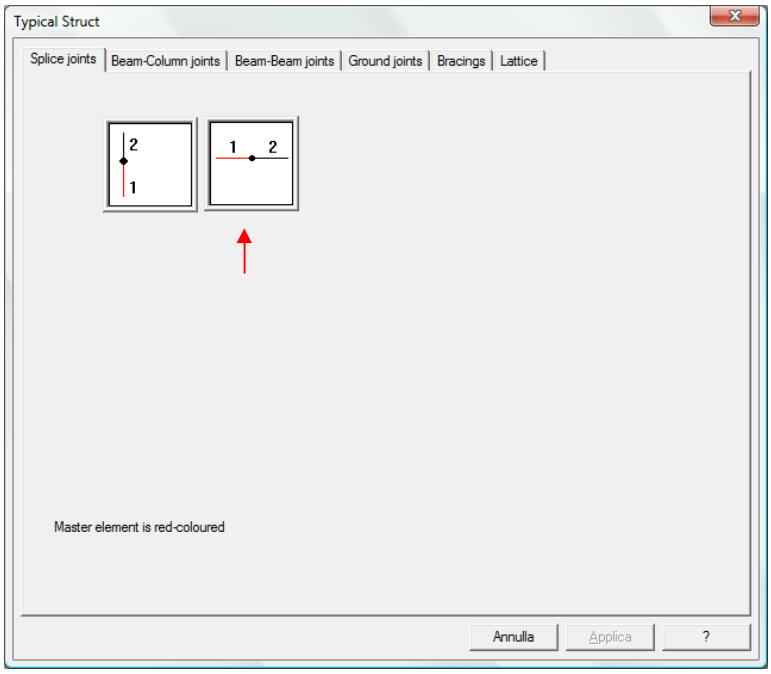
2.1 STEP 1: GETTING THE FEM MODEL

2.1.1 Getting the fem elements

Activate right window by clicking left inside it.

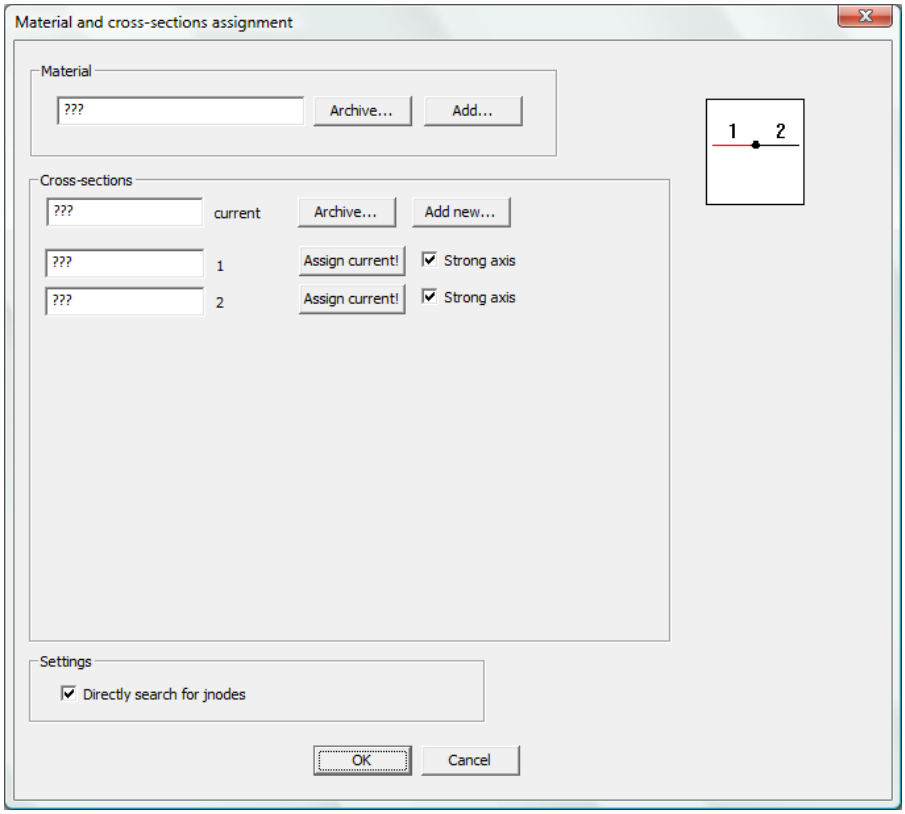
Execute the command

FEM-Elements-Typical Structures.



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

You will get the following dialog box:

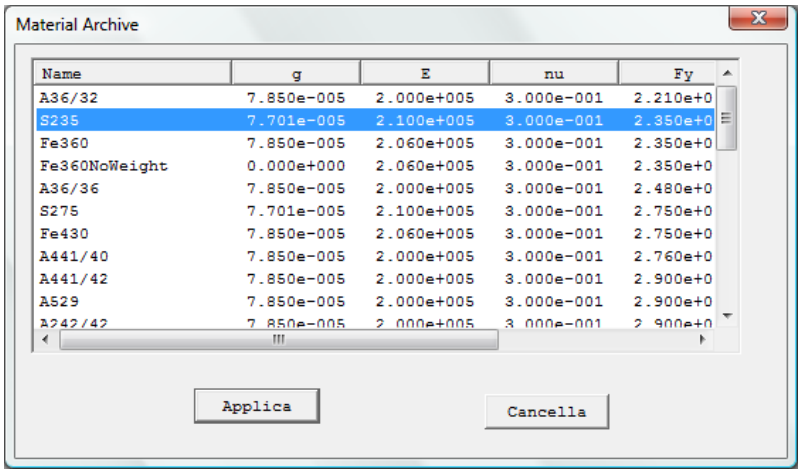


One of the beam elements has a “connection code”: it is necessary to define jnode hierarchy. The element without connection code is the master (in red), the other one is the slave: it is a **hierarchical** jnode.

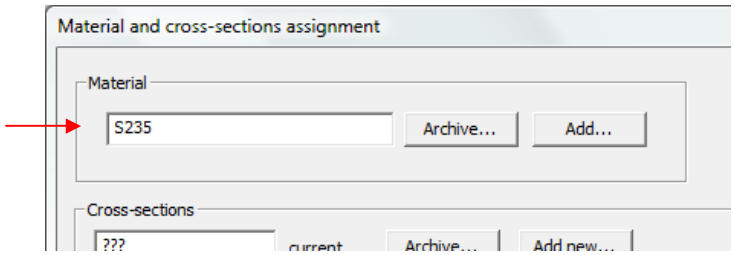
2.1.2 Assigning material

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

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



Chosen material has been applied.

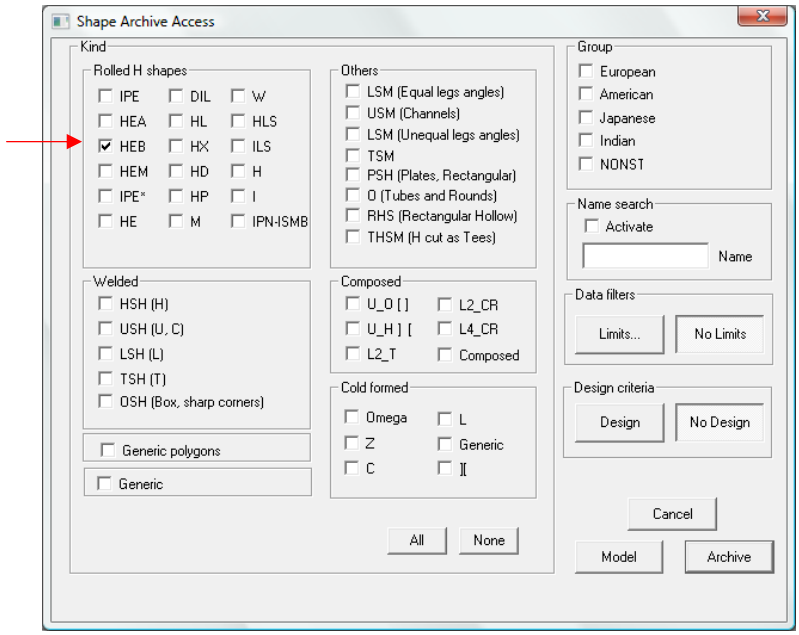


If you are using the demo the button “Archive” automatically applies the material S235.

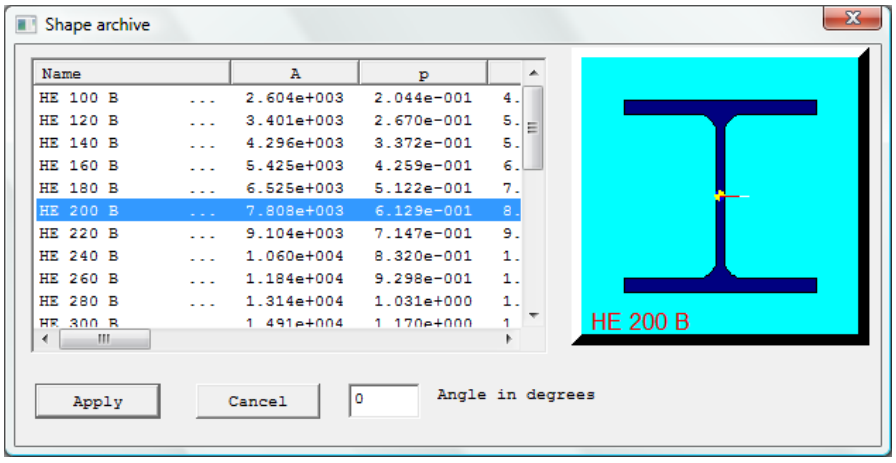
With the “Typical node” tool all the members have the same material. It is possible to define different materials with the standard fem commands.

2.1.3 Assigning cross-section

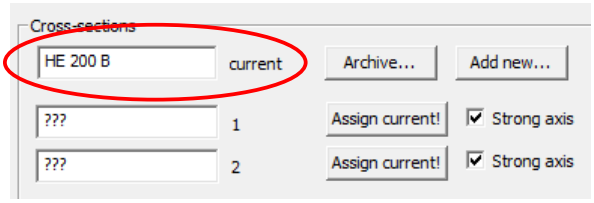
As for the material, it is possible to add new cross-sections or choose them from the archive. Press “Archive” in the “Cross-sections” box to browse the archive.



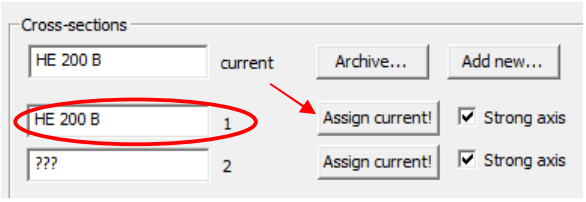
Select HEB check box and then "Archive" to filter HEB kind among all the available cross-sections. Then choose the section HEB200 by selecting the appropriate row, and press the “Apply” button.

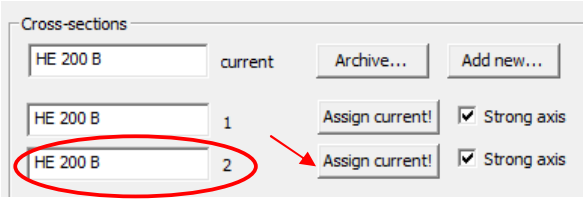


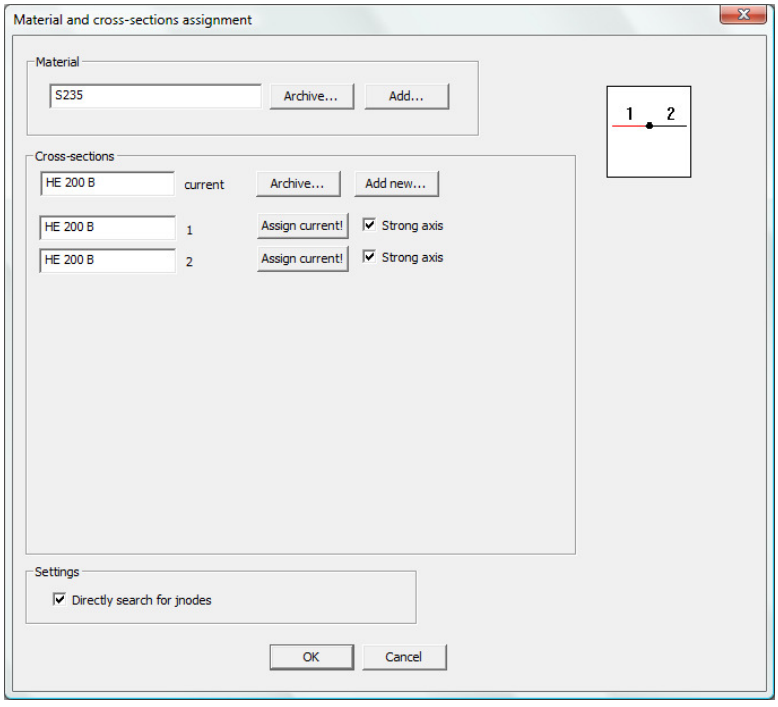
The chosen shape will appear in the box called “current”.



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





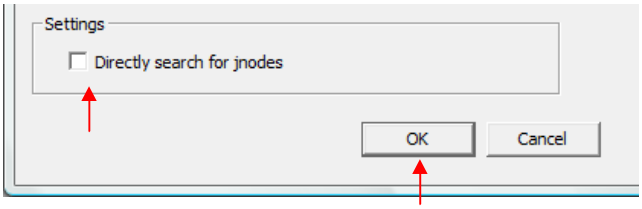


N.B. It is possible to change the current cross-section before assign it to member 2 in order to define different shapes for the members.

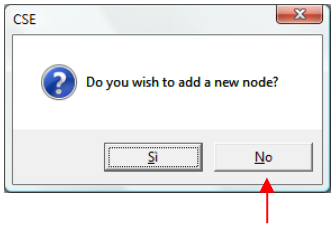
It is possible to apply a rotation of 90° to the members removing the tick from "Strong axis" boxes. In some nodes, not in this case, it is possible to define hinges for the slaves with proper tick boxes.

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

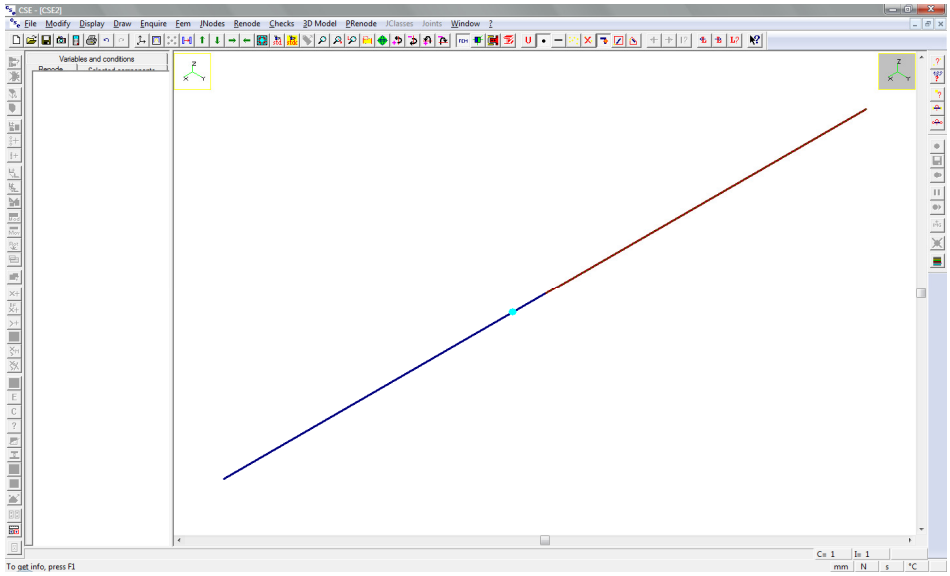
Remove that tick to see and understand step by step what could be done automatically (steps 2, 3 and 4 of this tutorial).



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



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



2.2 STEP 2: SEARCHING MEMBERS

To move to connection design you now have to detect which members are present in the fem model. To do that just execute the command:

FEM-Search members!

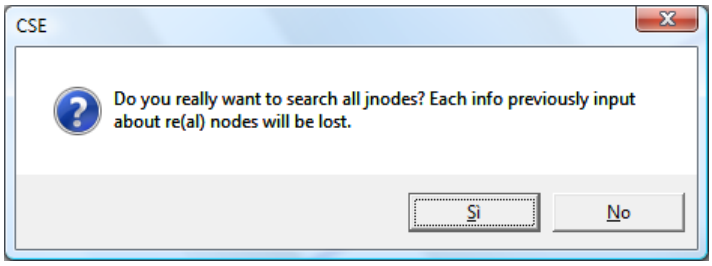
2.3 STEP 3: SEARCHING JNODES

Now that members have been searched, you need to find different "jnodes" that is what will next get a true, real node (renode). The program scans the member model and finds how many equal and how many different jnodes there are in the model. Then each jnode will be marked and you will be able to select it in order to work on it.

To search jnodes just execute the command:

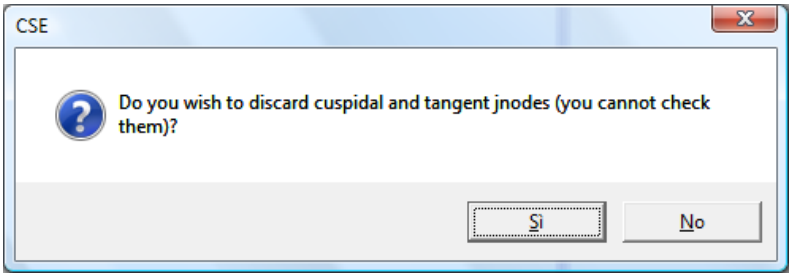
JNODE-Search jnodes!

and answer "yes" to the following question:



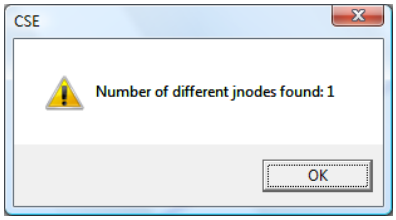
As no real nodes info has been defined you will not lose anything. Jnode search is usually done once for all in a model. Before beginning to add RENODES, you will check that the JNODES found are correct. This depends also on how the fem model has been prepared.

The following dialog appears:



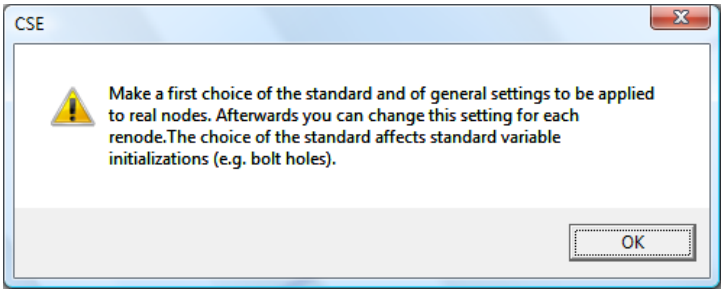
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:

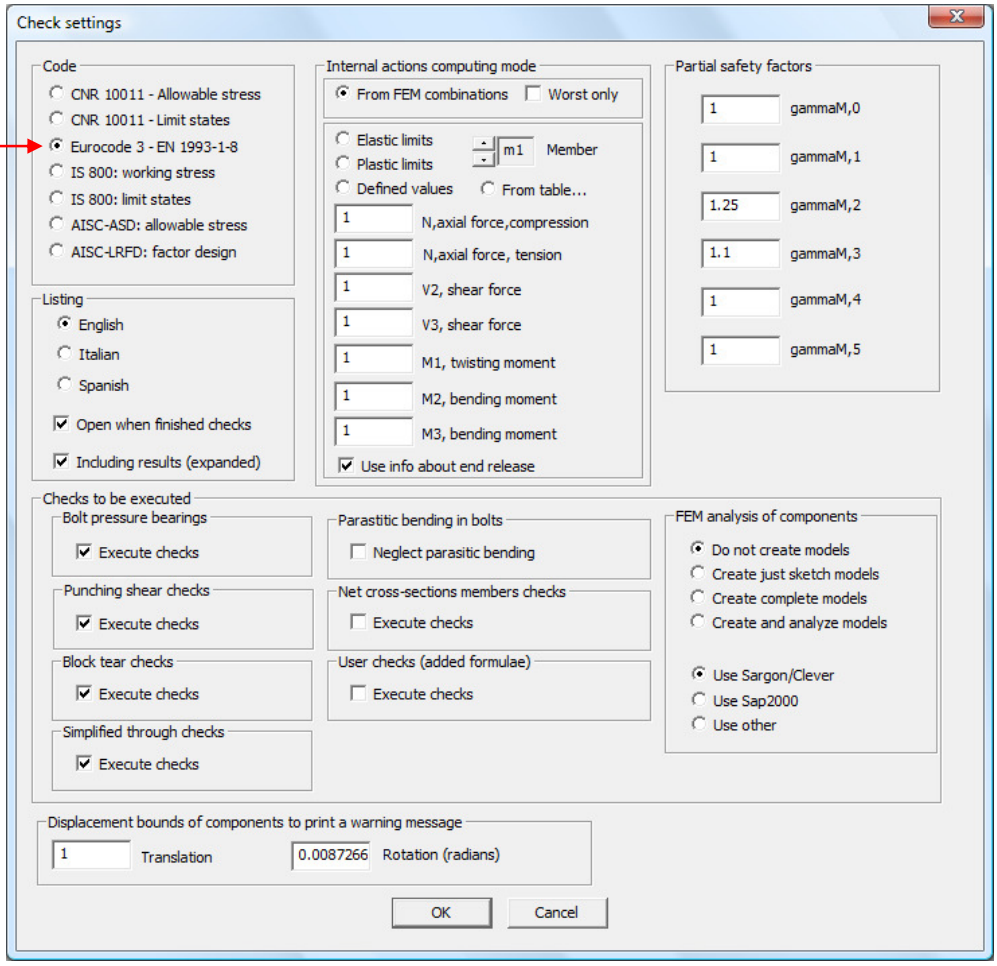


This means that in this model there is just one jnode, the splice.

Now the following message appears:

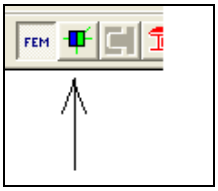


asking to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

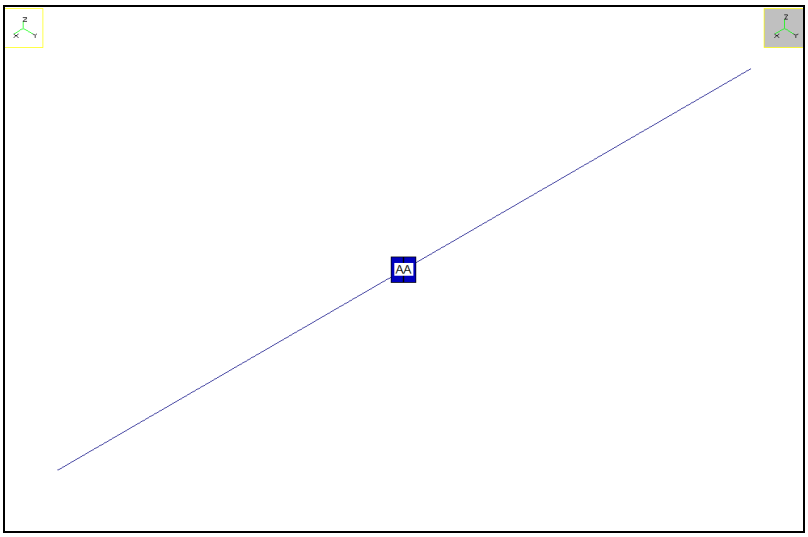


Choose Eurocode 3 and leave all other defaults, we will change them later..

Now you can switch to jnode view. This is done by pressing this button in the main toolbar.



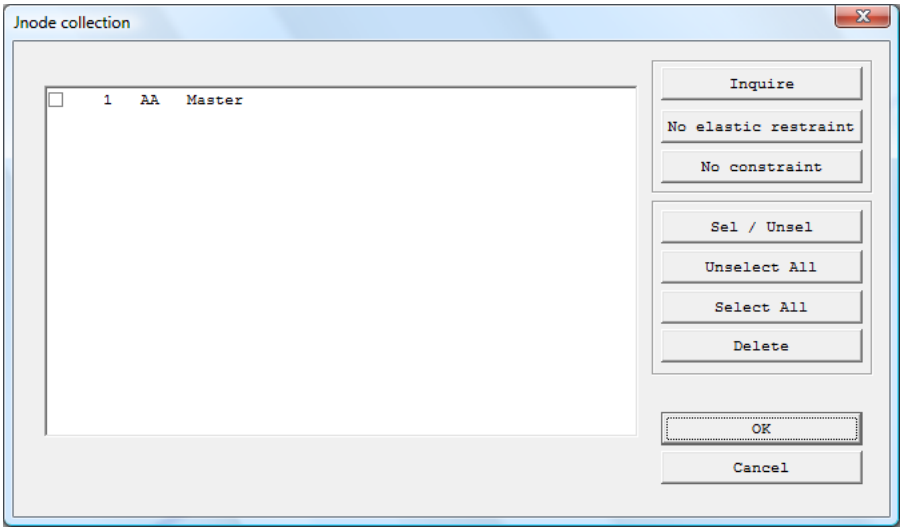
In jnode view you get the following:



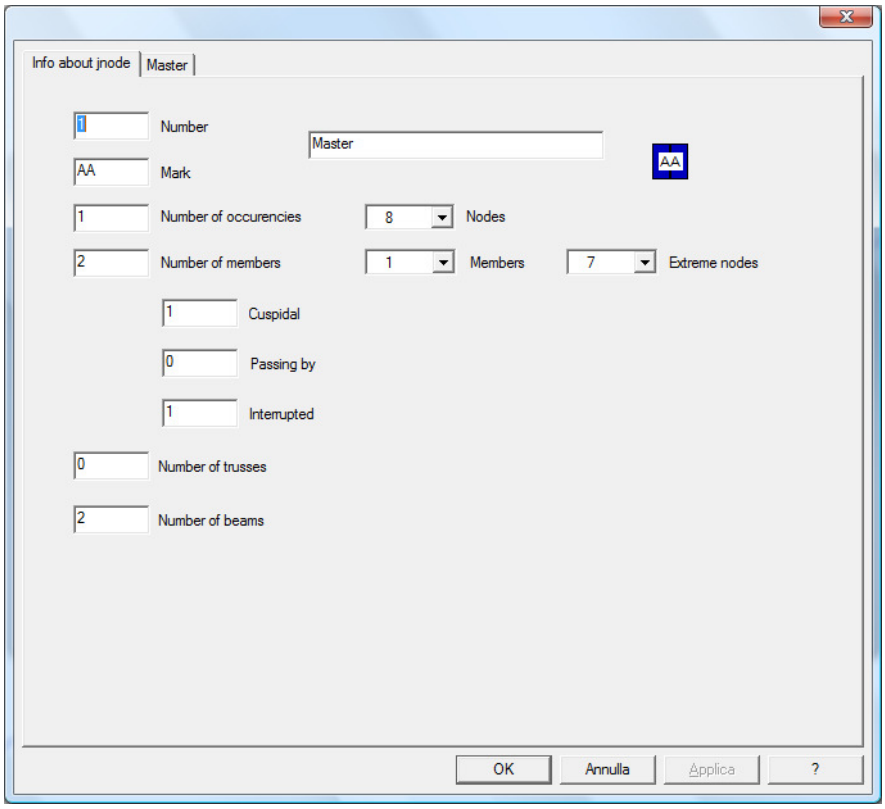
You can now get some info about the jnode found. Execute the command

JNODE-Edit...

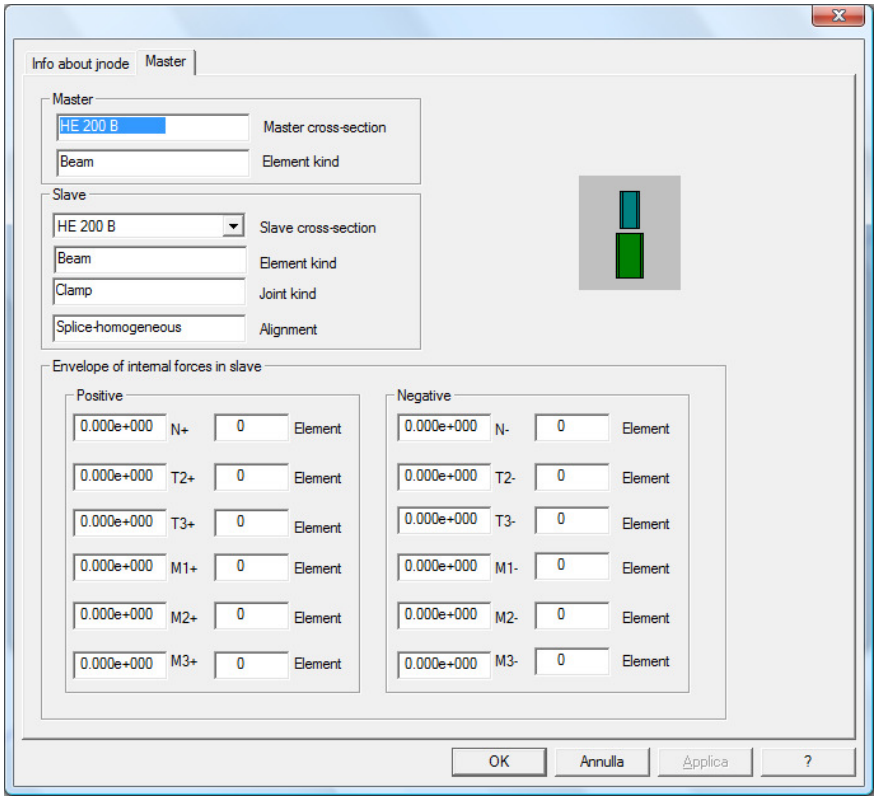
and get the following dialog:



Select the appropriate row and then "Inquire", you get the following



and clicking to "Master" the following:



The first dialog tells how many jnodes "AA" there are in the model, and which fem model nodes, members, and elements the jnode is using. The second dialog presents connections info and

internal forces envelope; this part is blank as this fem model has been prepared inside CSE and is not coming from a true fem analysis.

Press OK and then Cancel to exit from both dialogs, including "Jnode Collection" dialog.

You can now wish to examine a listing for all jnodes found. This helps to understand if the fem model has been prepared correctly, and is an important tool to pre-study future RENODES.

Now save the model executing the command **File-Save**, and specify a name, for instance TUTORIAL4.CSE.

Execute the command

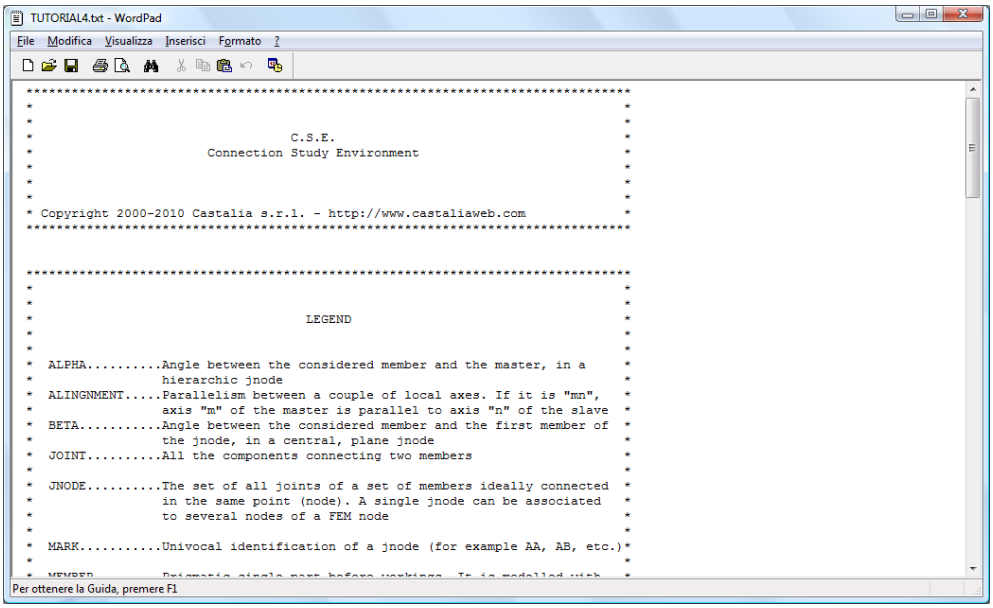
JNODES-Create listing!

you will get a message telling you that the stress file is missing: it's ok as we are not coming from a true fem analysis.

Now execute the command

JNODES-Open listing!

You will get a file with useful preliminary info about JNODES topology and categorization.



In particular you will get the following info:

```
*******************************************************************************
*                                                                             *
*                           JNODE     1       MARK    AA                      *
*                                                                             *
*******************************************************************************

GLOBAL TYPOLOGY OF THE JNODE: HIERARCHIC JNODE

NODES ASSOCIATED TO THIS JNODE: TOTAL 1
    8
```

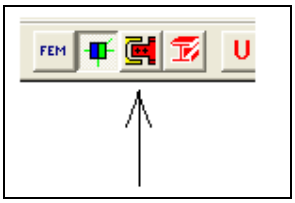
```
MEMBERS IN THE JNODE: TOTAL 2
     1      2

   JOINT    1 (Beam - Beam) MASTER= HE 200 B              SLAVE= HE 200 B
                              Beam                            Beam
      End joint - Splice-homogeneous - Clamp -
      ALPHA= 0.000000 (    0.00°) COS=  1.000000 SIN=  0.000000 TAN= 0.00000e+000
```

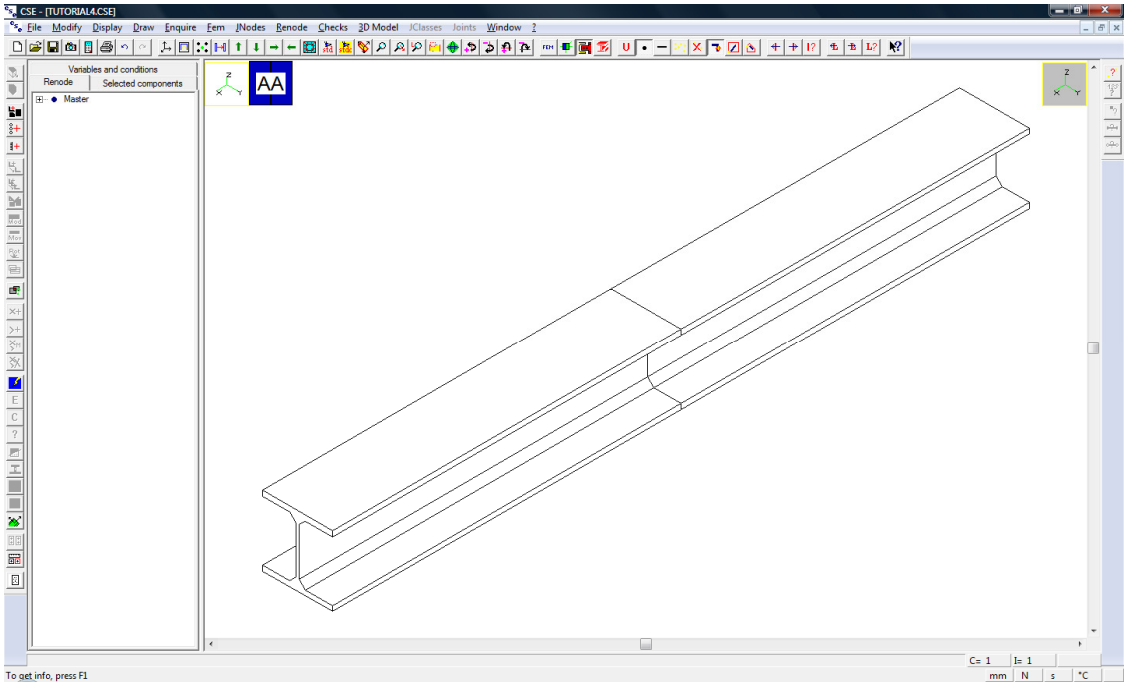
2.4 STEP 4: SELECTING THE PROPER JNODE

We are sure that jnodes are correct, we can move forward to select one of them to transform it into a **RE**al **NODE**. Left click with mouse over the square "AA" in the graphic pane: it gets yellow. This means you have selected a jnode (all *instances* of that jnode will be selected, here there is just one *instance*. In a true fem model there can be tens of different instances of the same jnode).

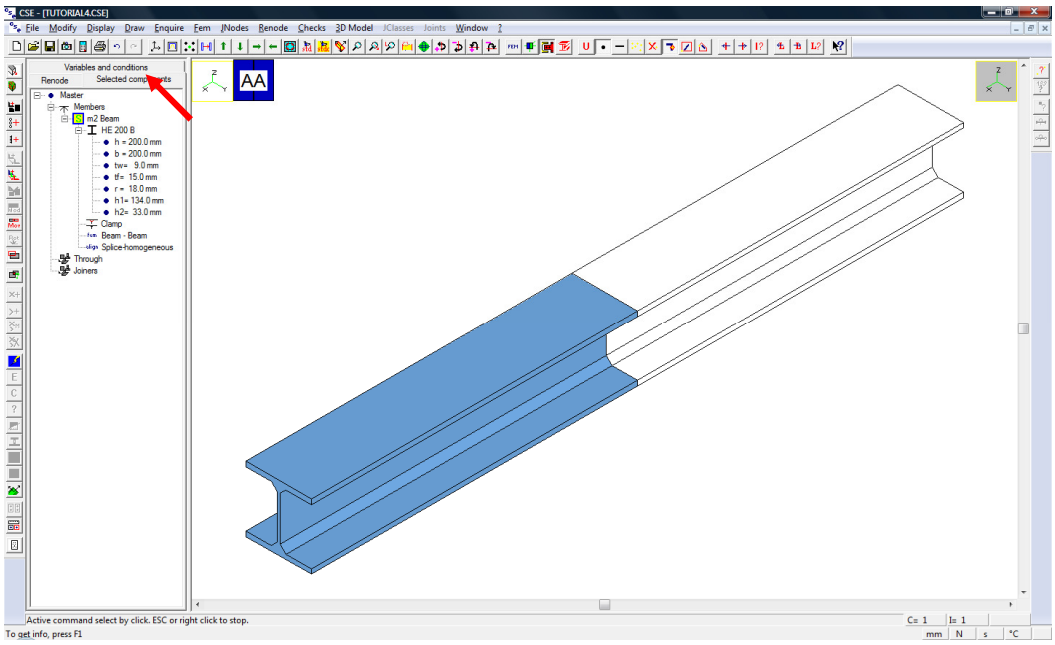
Since there is just one jnode selected the switch to real node is possible. The following button in the main toolbar is active now:



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



Note that the left window is not empty anymore: it gives you info about the components selected, the components present in the renode, and the variables and conditions present in the model.
Note that clicking a member it gets selected (blue) and that the "Selected components" sub-pane in the left window is upgraded. Clicking and re-clicking you select and unselect.

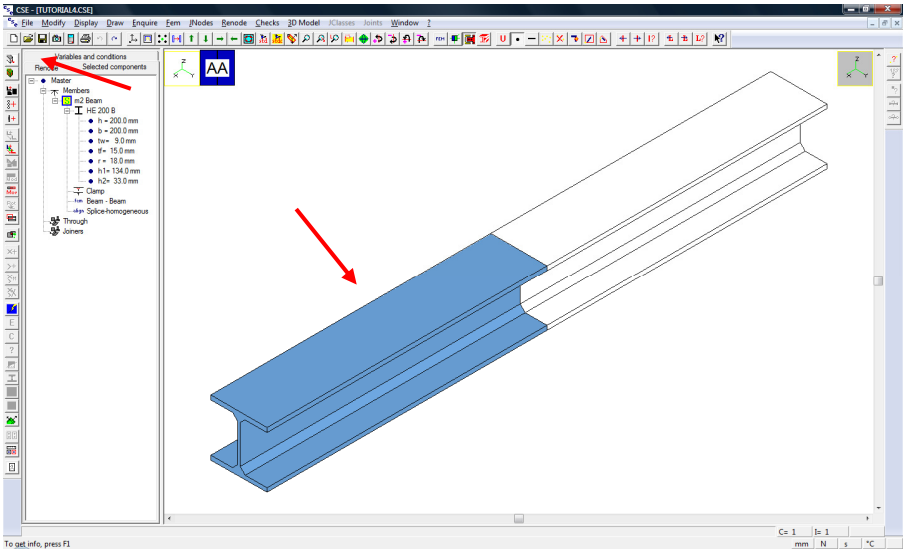


2.5 STEP 5: CONSTRUCTION OF THE REAL NODE

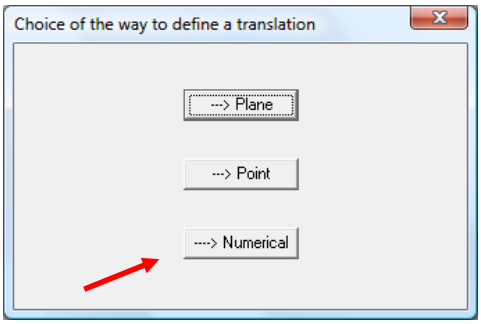
N.B. in this tutorial the renode will be manually build step by step, but it is also possible to apply one of the available parametric renodes of the archive that apply to the current renode. The command is Renode - Assign Prenode  .

2.5.1 Addition of plates

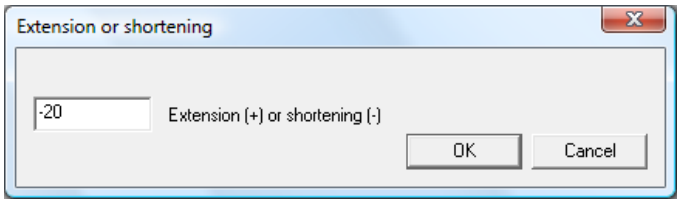
We need to trim members in order to get the space where plates can be inserted. Select a member, then use **Renode – Members - Trim-extend** command ().

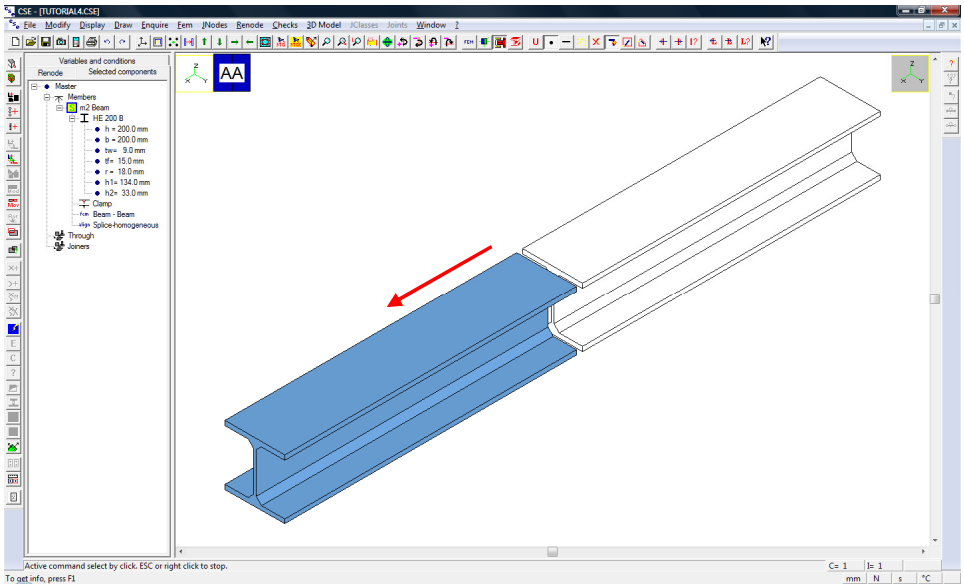


The following dialog box appears; choose the numerical mode.

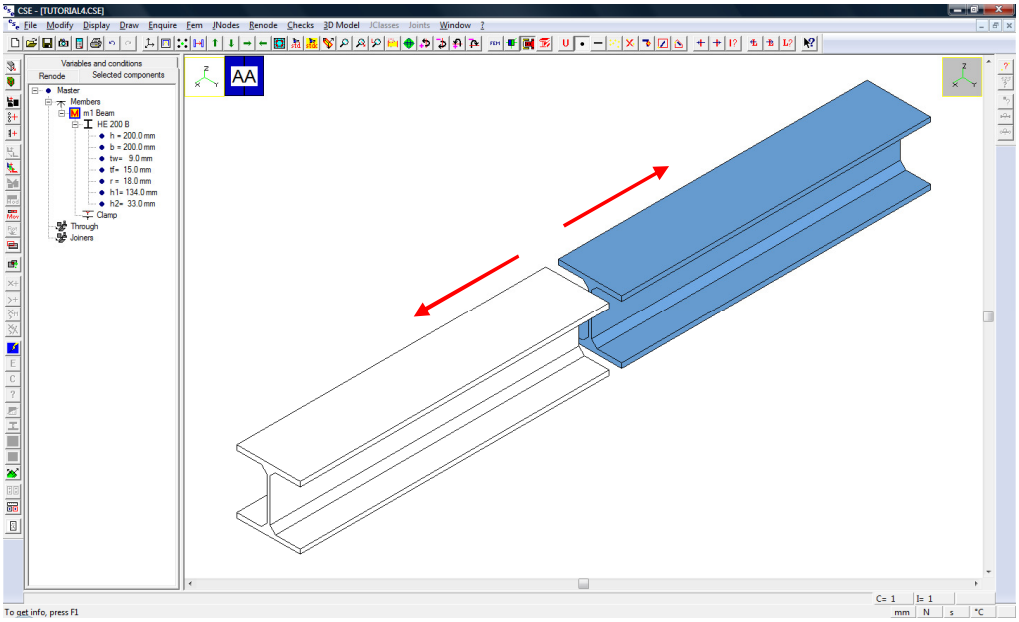


In the next dialog box type '*-20*' and press OK the define a trimming equal to 20mm.

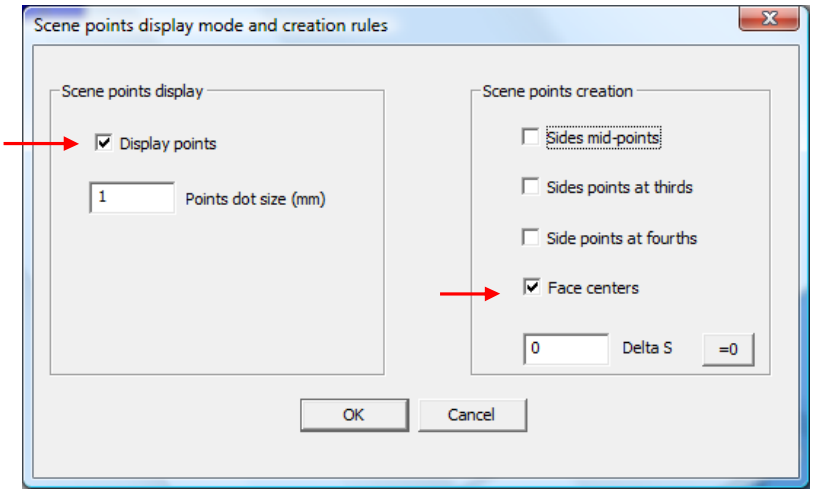




Now unselect the first member, select the second one and repeat the previous operations in order to trim the other member.



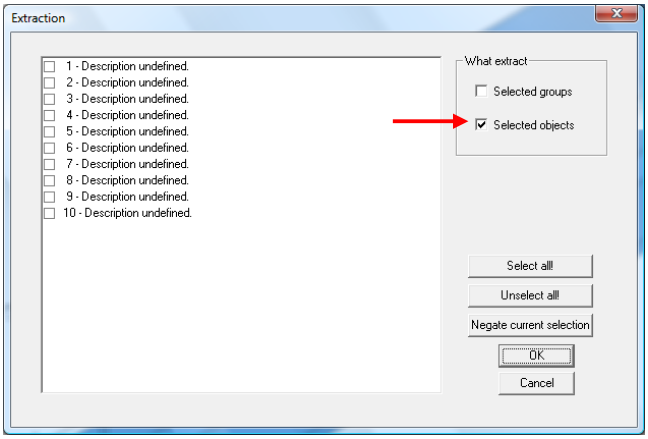
Use **Display – Scene points**  to show some relevant points useful to easily add the plates. In the dialog box, tick “Display points” on the left and keep only “Face centres” tick on the right. This will create and show the centre of every face in the scene.



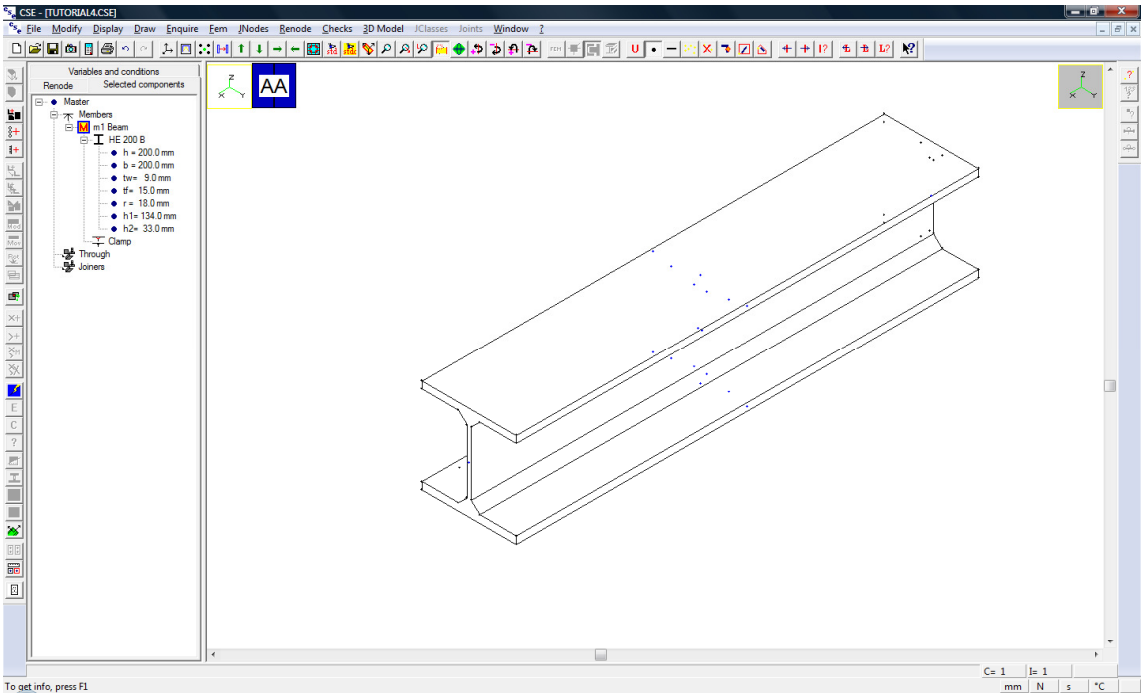
N.B. you can modify points size in the scene with the proper parameter in the dialog box

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

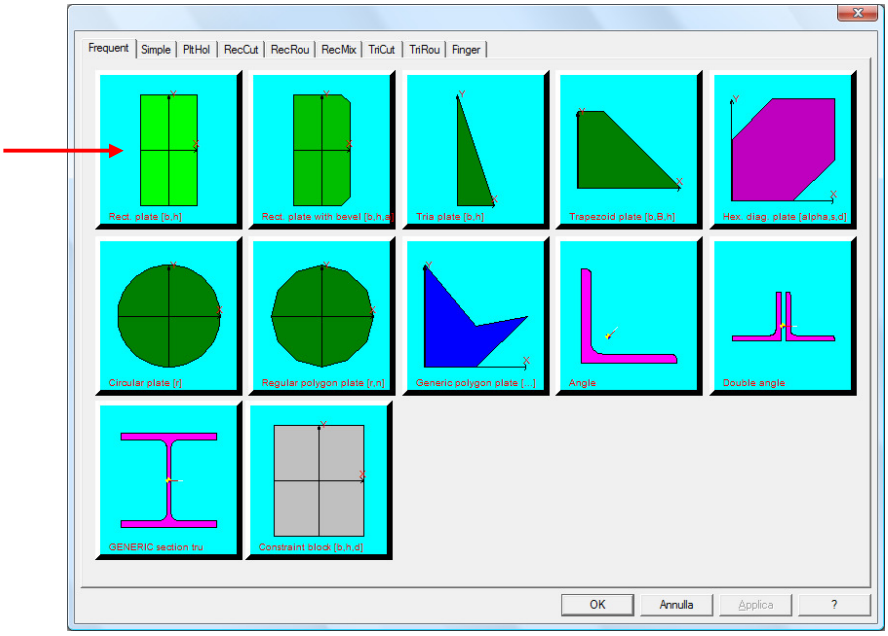
Use **Draw – Extract**  to extract selected member. Tick "Selected objects" in the following dialog box and press OK.



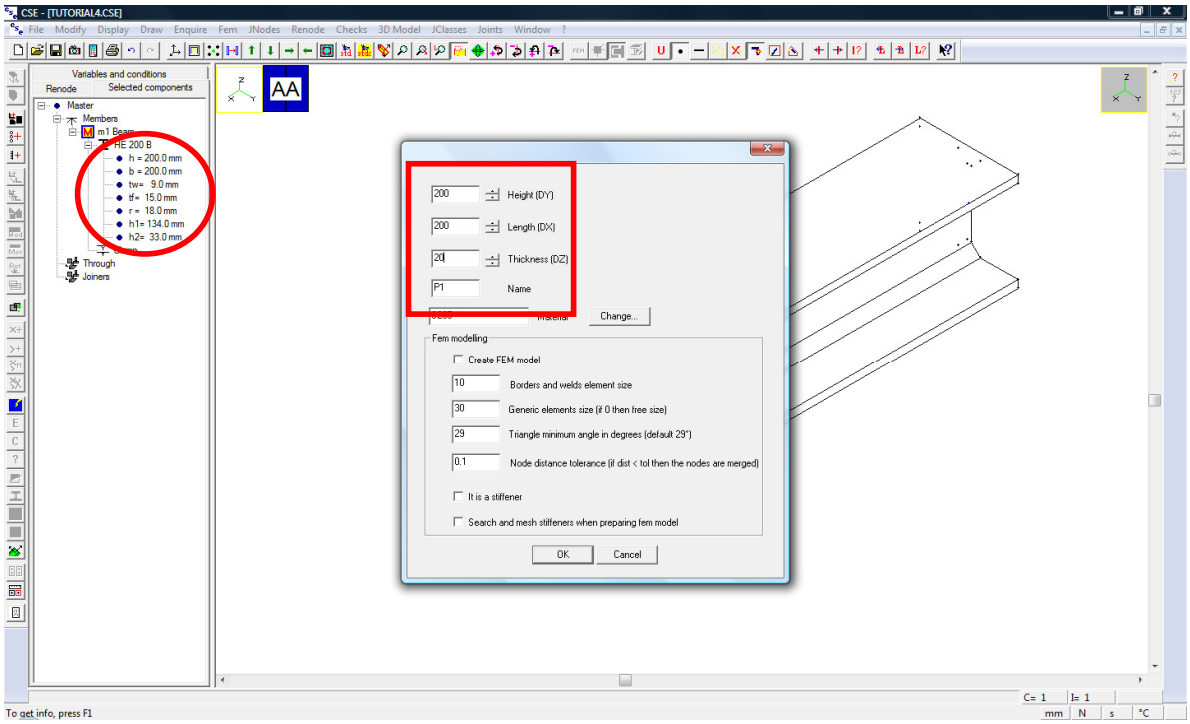
You will get the following. Zoom to have a closer view (use mouse wheel or commands in *Draw* menu)



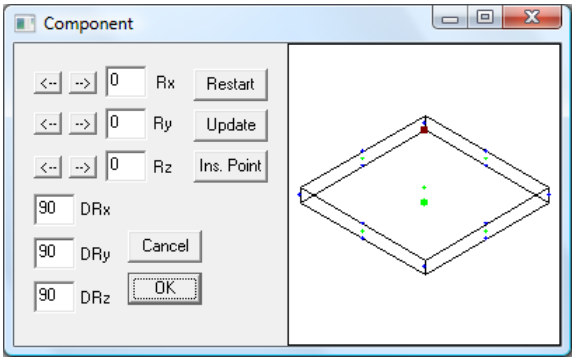
To add the plate, use **Renode – Components – Add through** ( button in left toolbar), then choose the rectangular plate in the following property sheet.



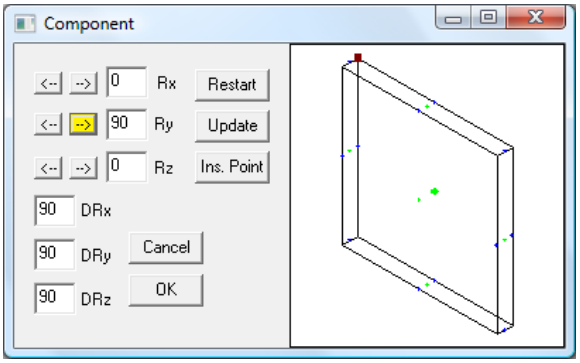
A new dialog box appears. Define plate size as follows: height and length equal to 200(mm, the current units), thickness equal to 20mm. You can see member sizes in the alphanumeric view on the left.



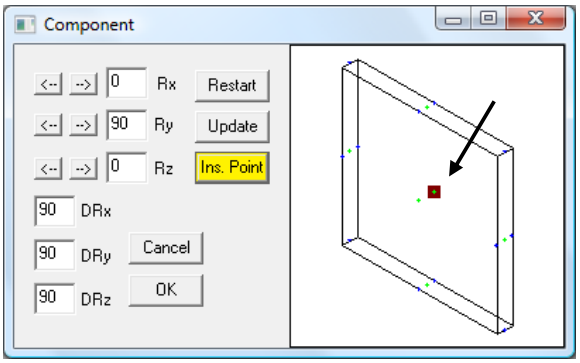
Press OK; in the following dialog box choose the orientation of the plate and the insertion point to add the component in the scene.



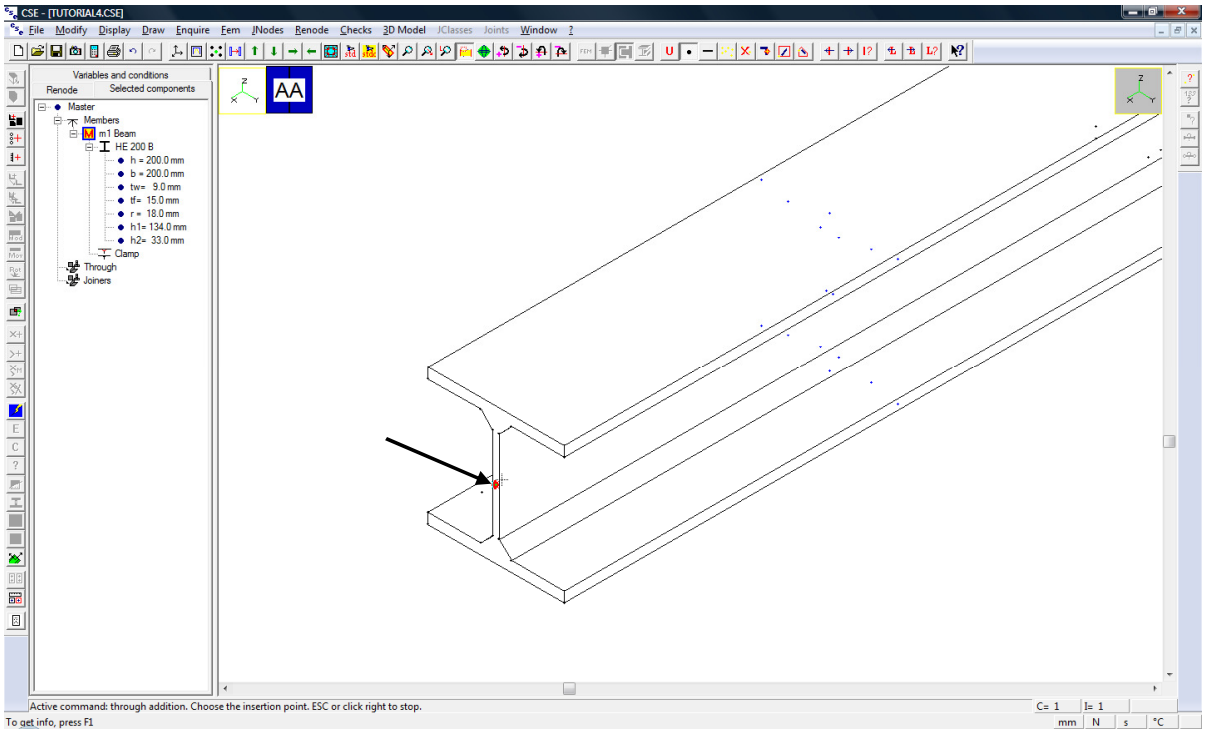
Press right Ry button once to rotate the plate in the position needed.

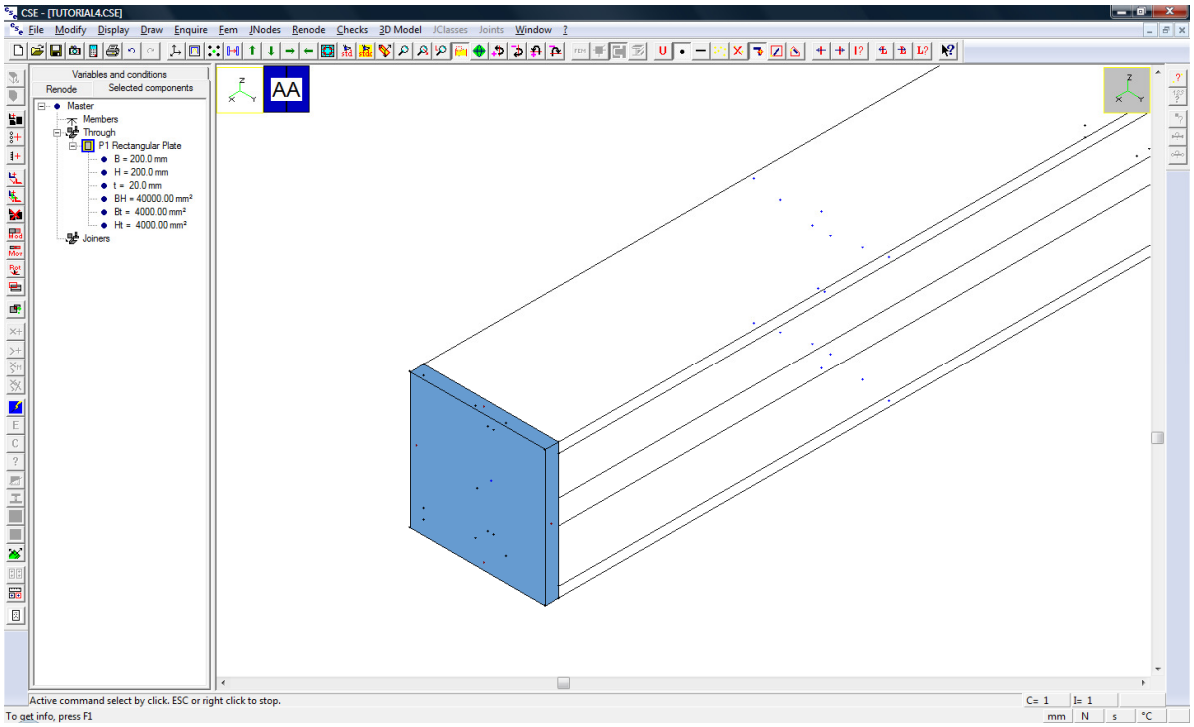


Press “Ins(ert) Point” and click the centre of the second face, then click OK.

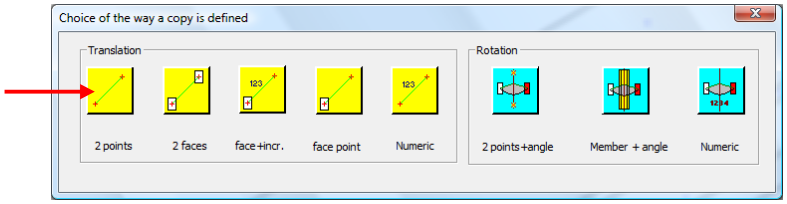


Now click in the scene the point corresponding to the insertion point of previous dialog: it is the centre of member end face.

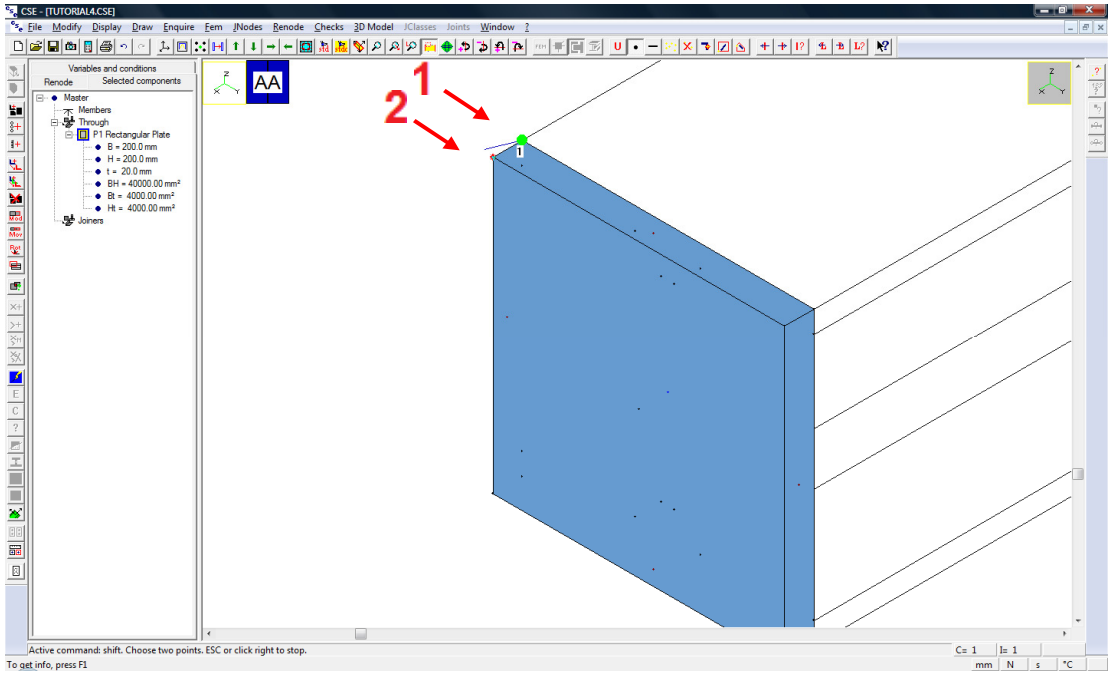


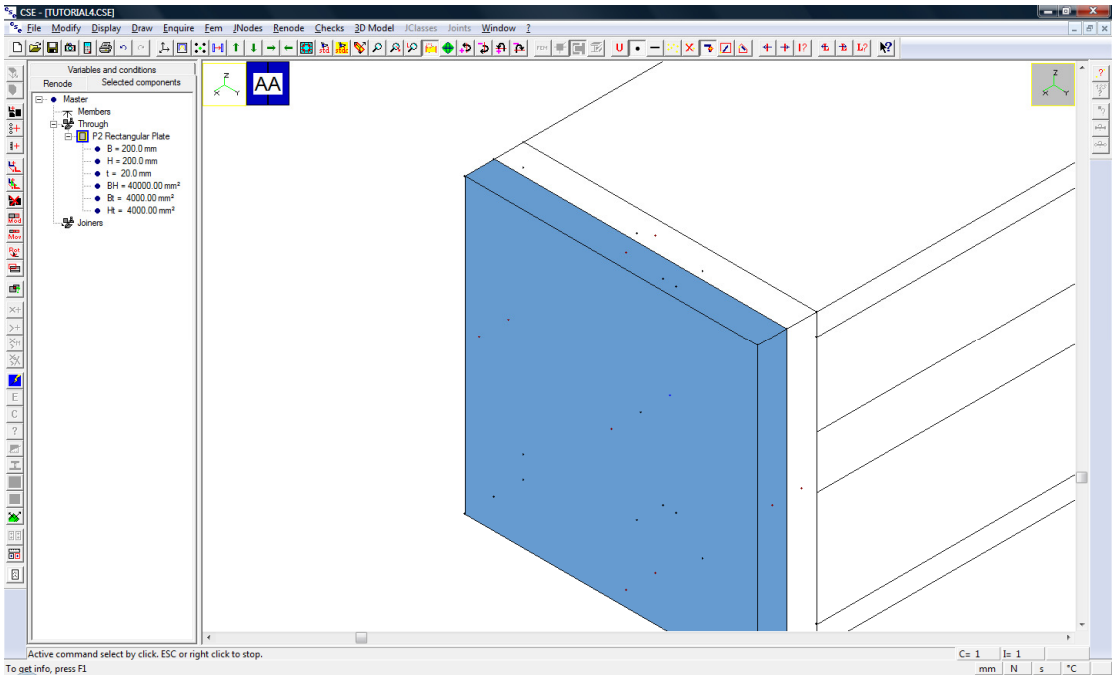


The first plate has been added. Now copy it to get the second one. Use **Renode – Components – Copy**  and select a “translation” using “2 points”.



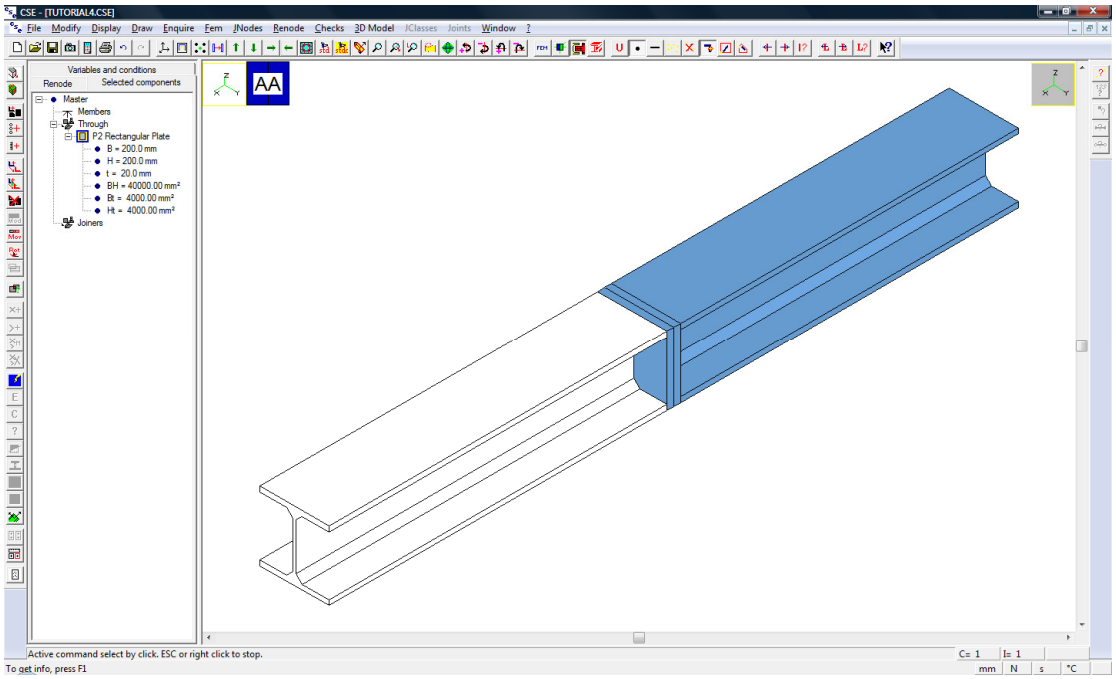
In the scene, define a vector clicking, for example, the corners on the top left of the plate.





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

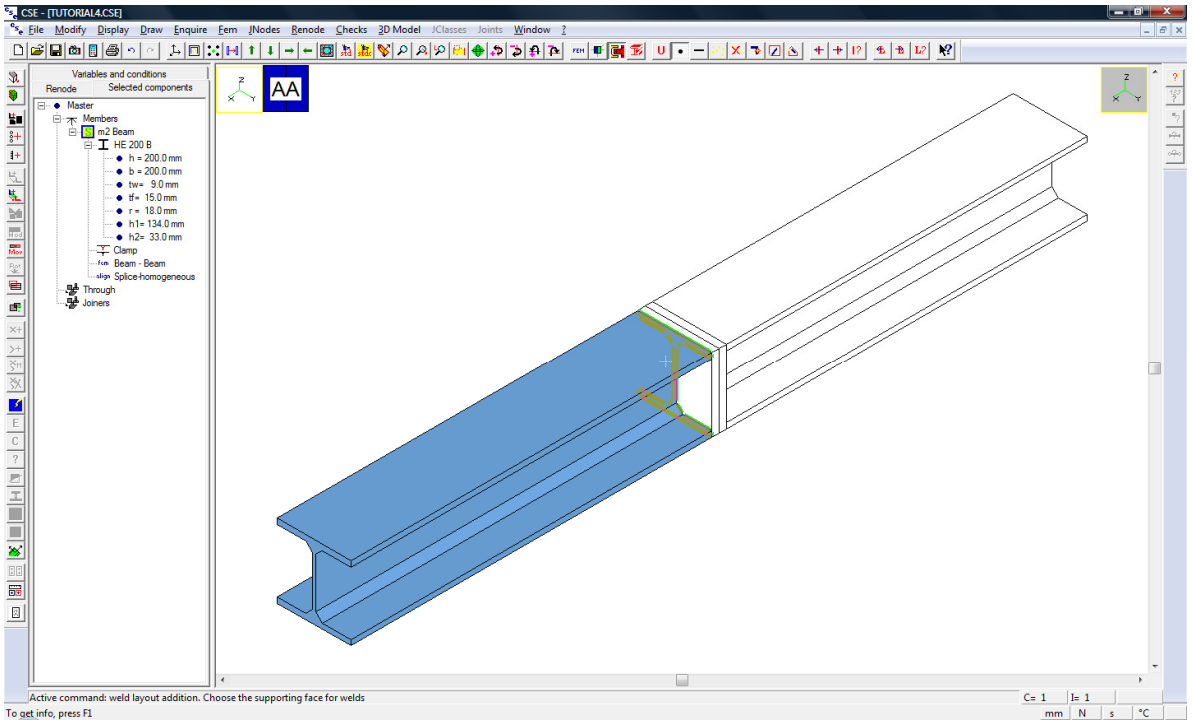
Enclose all the components in the view.



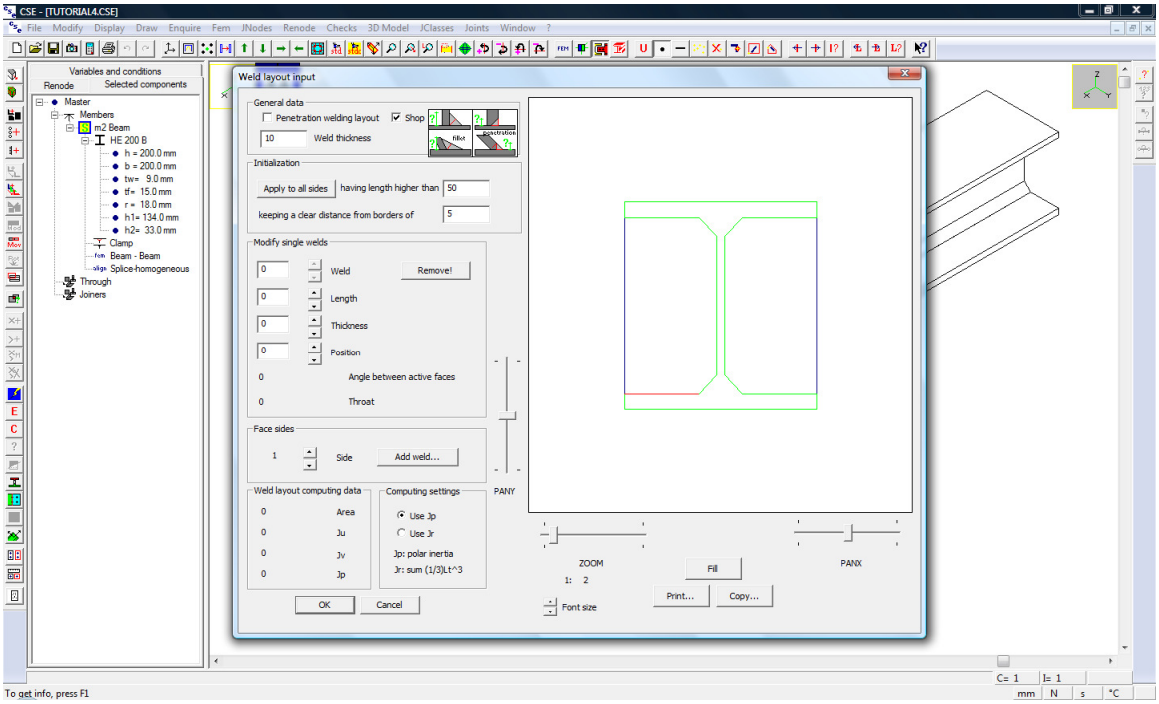
2.5.2 Addition of joiners

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

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



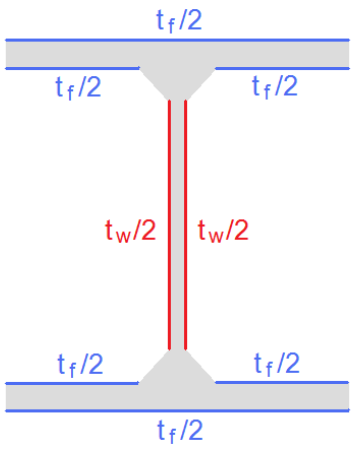
When end face is highlighted, click mouse left button to select it: the following dialog will appear.



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

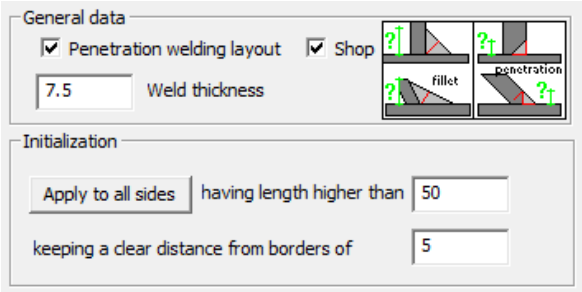
CSE automatically recognizes the second face to be welded, and in the dialog box figure you can see member section and plate borders.

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

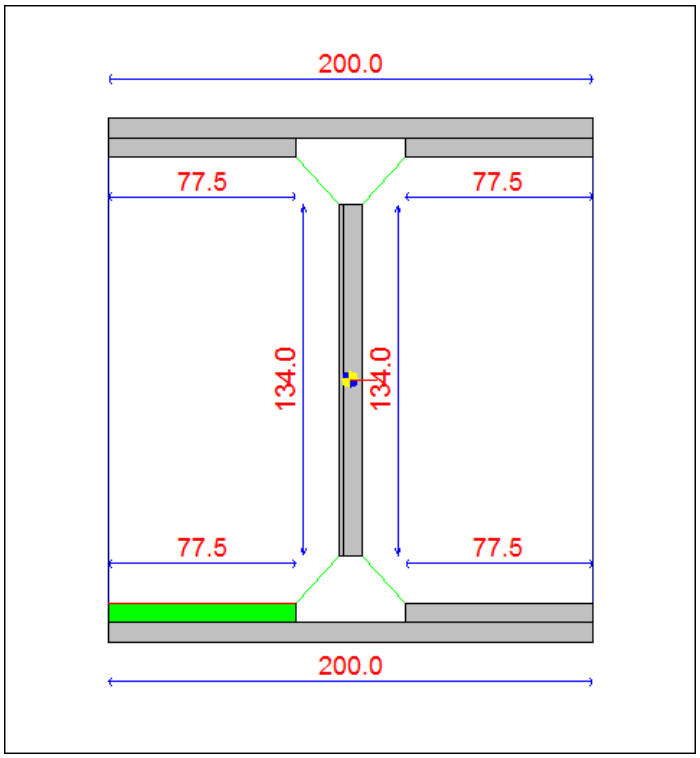


As shown in "Selected components" pane, t_f is 15mm and t_w is 9mm, so $t_f/2$ is 7.5mm and $t_w/2$ is 4.5mm.

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

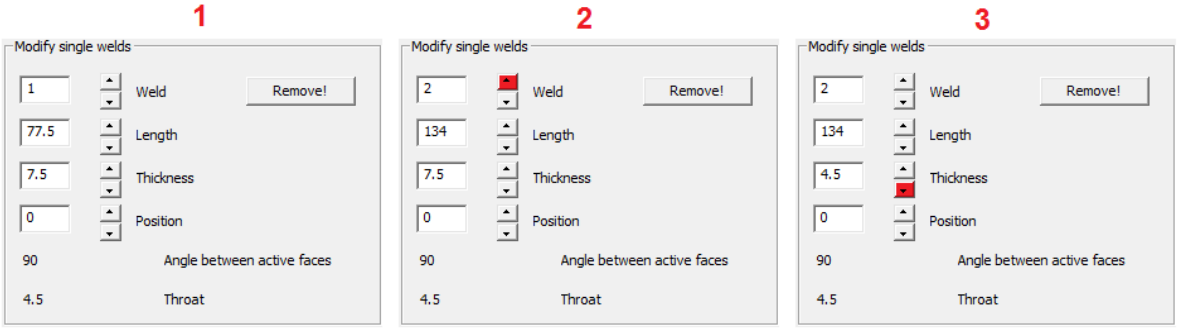


Click "Apply to all sides" and the following welds will be automatically added:

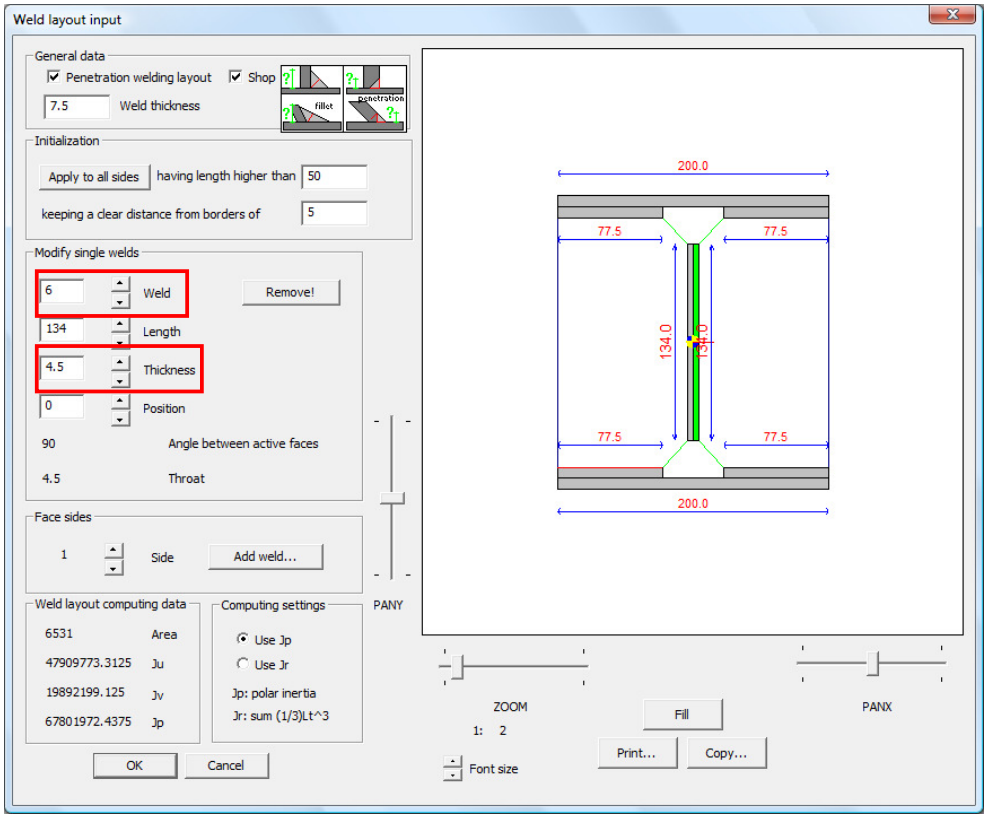


The welds on the web currently overlap, because they should have a thickness equal to 4.5mm instead of 7.5. Now we are going to modify these parameters.

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

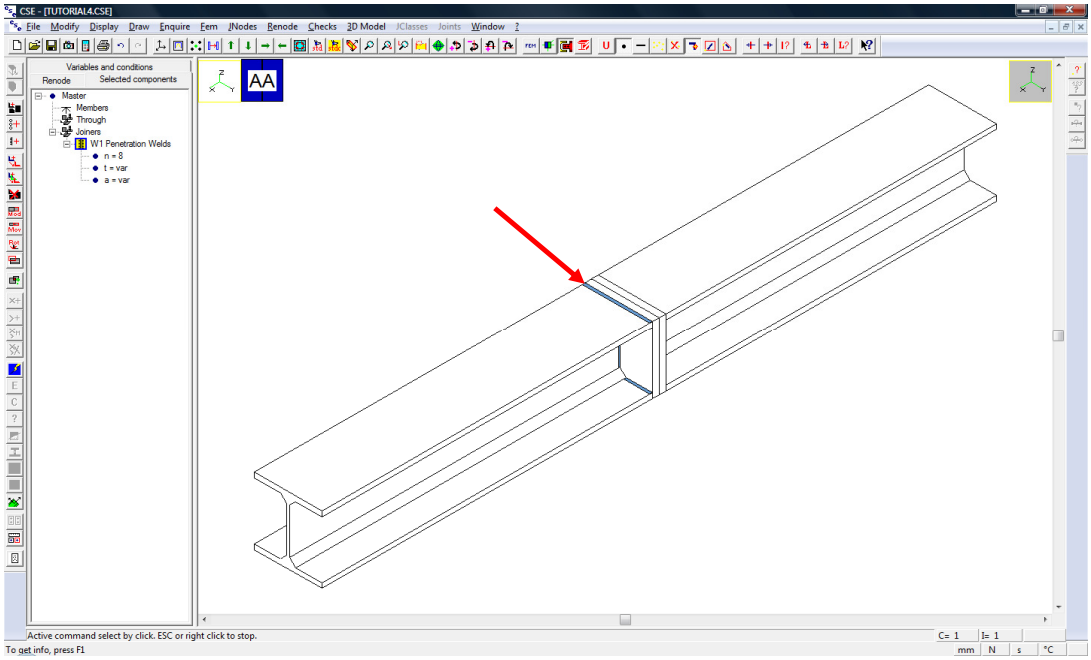


Do the same with weld number 6.

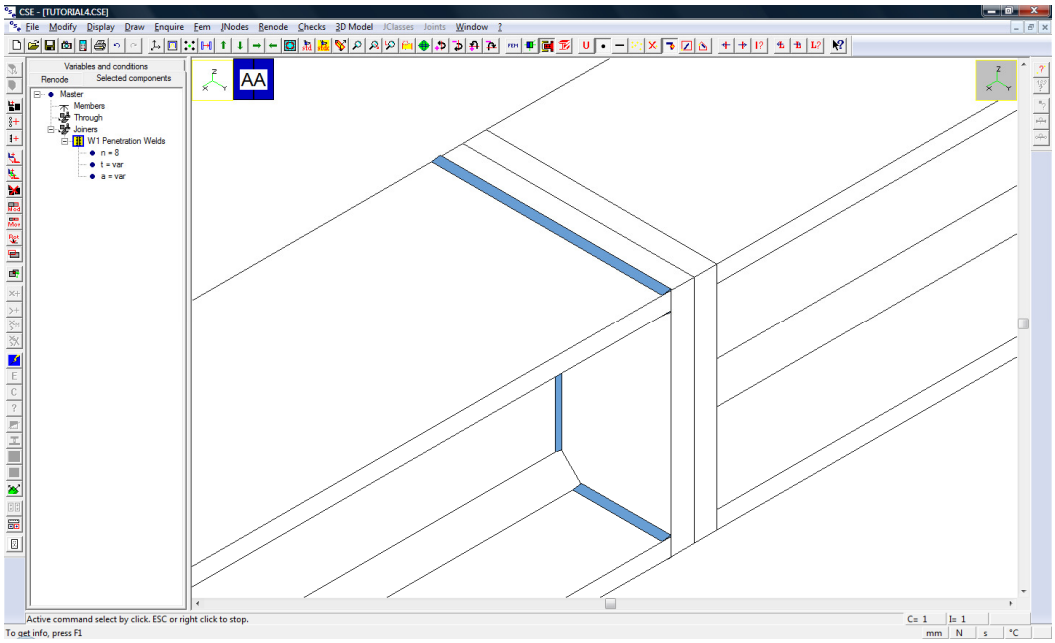


Now there is no overlap between welds. Leave "Use Jp" for torsion computation and press OK to insert the layout.

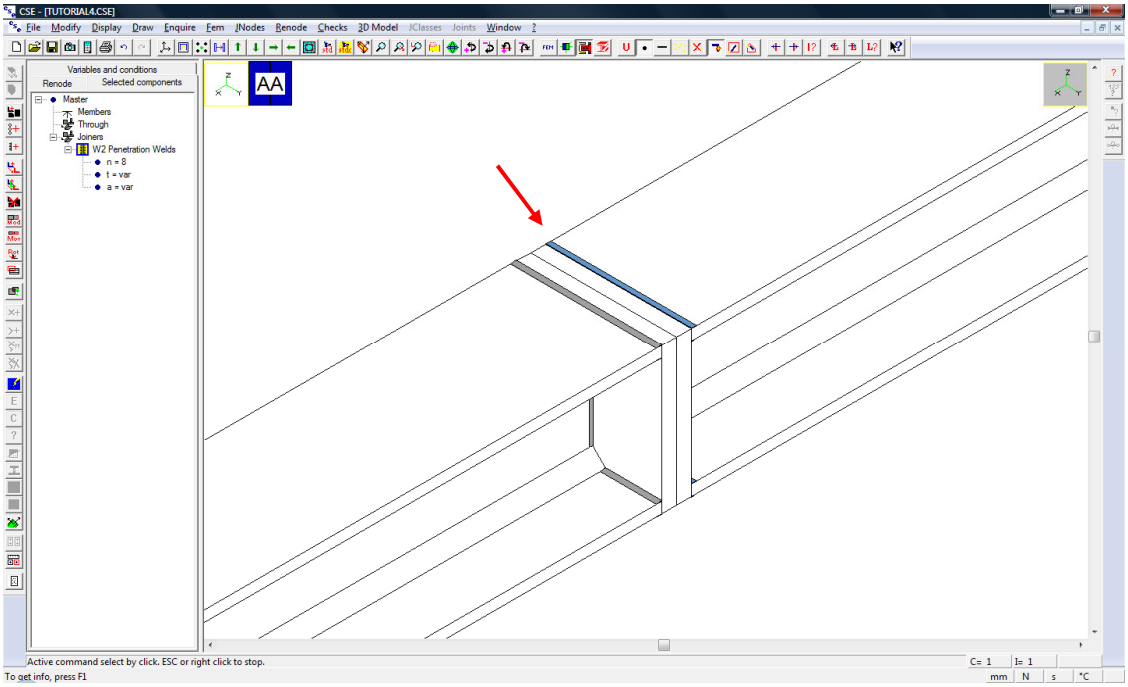
Unselect the member, keeping weld layout selected.



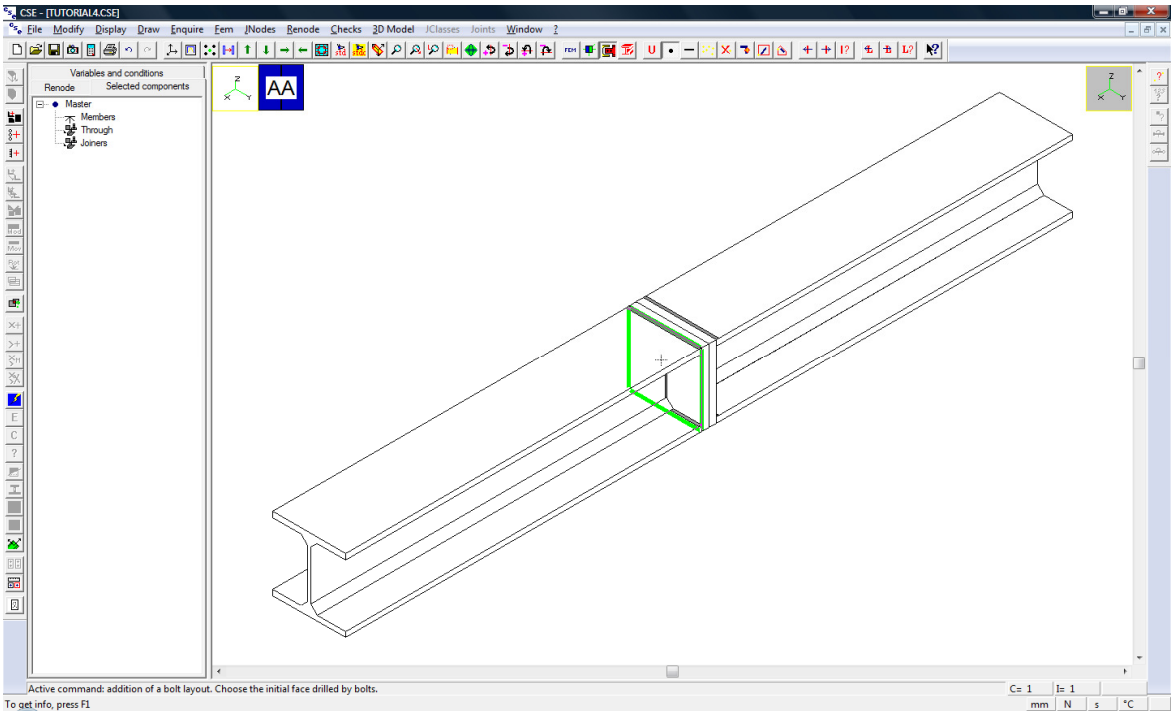
Unselect the weld layout and zoom to get a closed view of the welds (use mouse wheel or **Draw – Zoom in** and **Zoom in Click** commands).



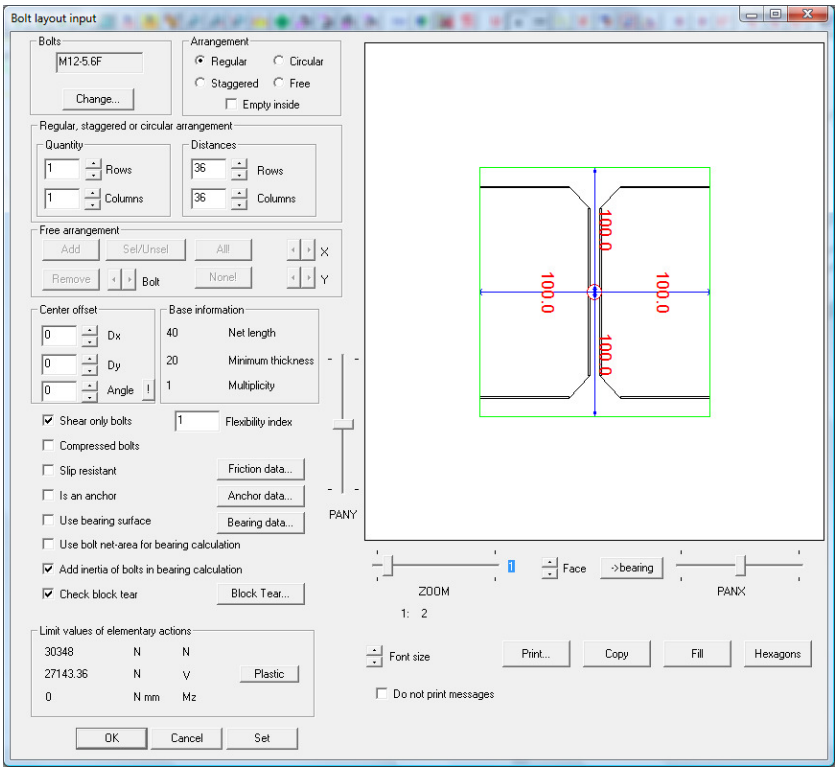
Now select m2 and repeat the previous operations in order to add a new weld layout connecting m1 to its plate (the new layout could be obtained also with copy command).



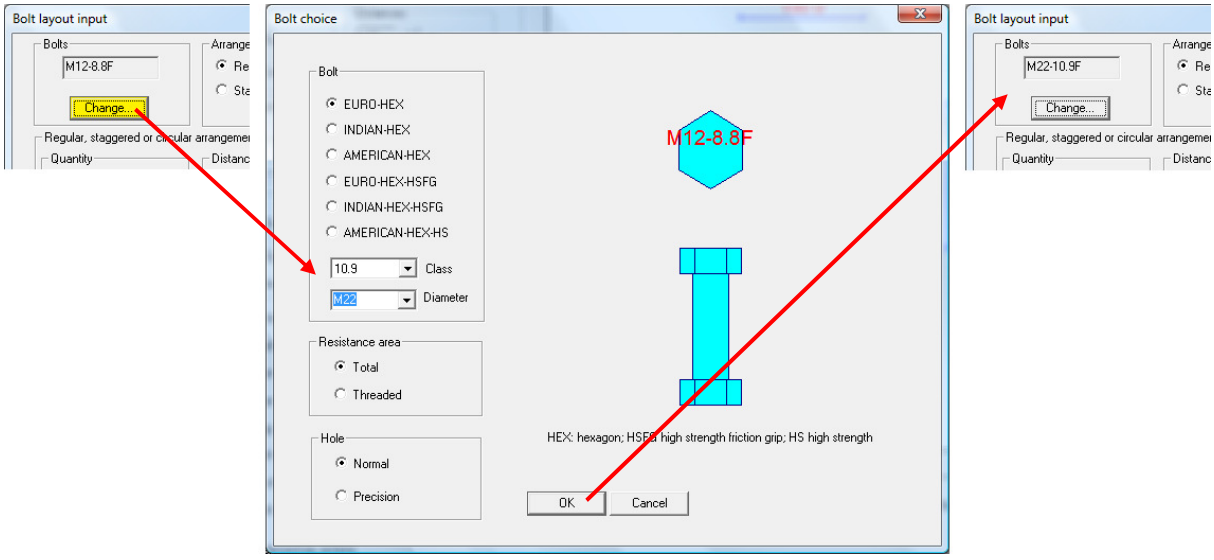
Finally, we need to add a bolt layout. Unselect all the objects ( ). Use **Renode – Components – Add bolt layout** , then click an external face of a plate: CSE will automatically recognizes all the objects that must be drilled.



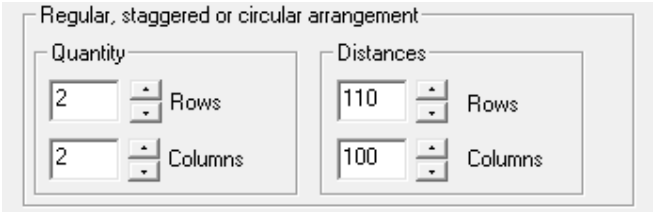
You get the following dialog box.



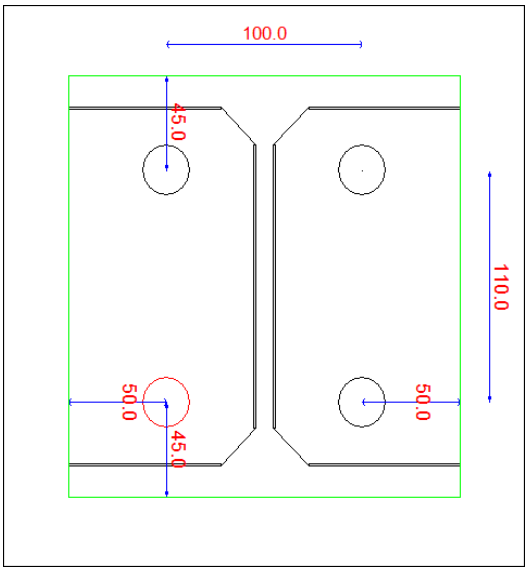
To define bolts class and diameters, press *Change* button in dialog box top left part, set 10.9 as class and M22 (EURO-HEX bolts) as diameter in the new dialog, then press OK.



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



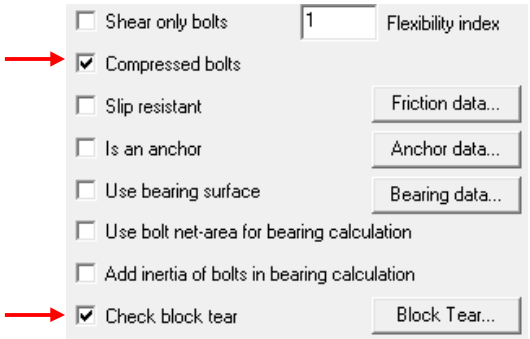
The image is updated in real time.



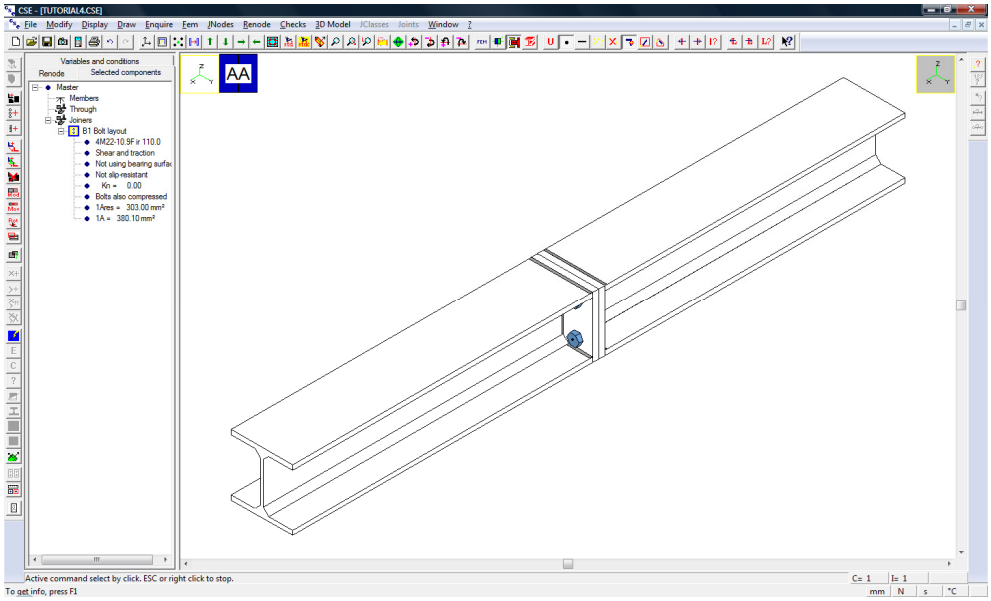
Note that distances between bolts and to current face borders are displayed. Use the buttons under the figure to change the current face. If minimum distance between bolts or from the borders is smaller than current standard limit values, a warning message will appear when we will try to add the layout.

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

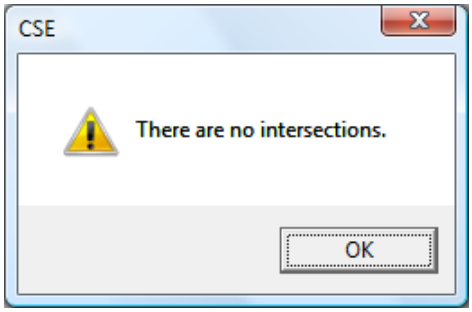
See the guide for more information about bolt layout computational hypotheses. Here just tick "Compressed bolts" to assign bending resistance to bolts only and "Check block tear" to include this failure mode in automatic checks.



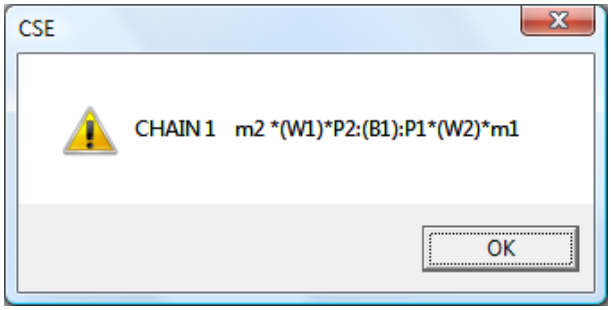
Press OK to insert the layout.



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



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



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

member m2 → weld layout W1 → plate P2 → bolt layout B1 → plate P1 → weld layout W2 → member m1

2.6 STEP 6: SETTING CHECK SETTINGS

Before executing the checks, let us specify how we want them to be done (we now choose the settings that initially have left as default values). Execute the command **Checks-Set**. You get a dialog with the following sections.

Code: the standard has already been chosen, Eurocode 3. This is a limit state standard.

In the **Listing** section choose the language, then tick the proper boxes if you want the listing to be automatically opened after the checks and if you desire a listing that includes checks results.

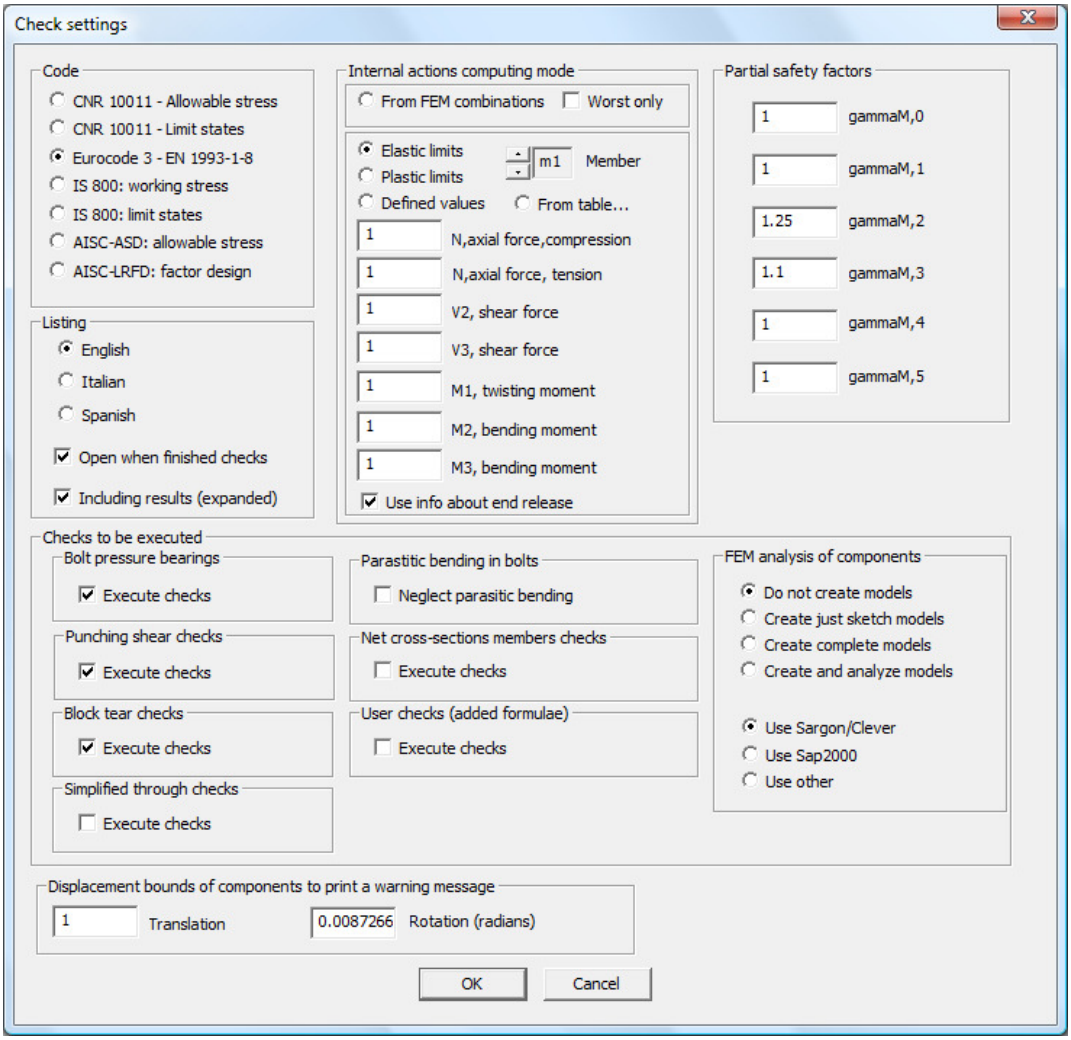
Since we have not a true fem model we must choose as **Internal actions computing mode**, "Elastic limits", "Plastic limits", "Defined Values" or combinations imported "From table". Choose elastic for instance. Then specify the multiplier of internal actions for each component. Here we assume 0.3 for all the components. You can specify the number you want.
This will generate 24 notional load combinations. 6 for positive internal forces, one by one. Six more for negative internal forces. Next 12 will be a mix of N, M_2, M_3, i.e. axial force and bending moments.
Note that the first 24 combination, if using forces NOT coming from fem, refer to the master and are empty, so you can leave default multipliers values for m1. Use the arrows to select m2, then type the desired multipliers (0.3 everywhere in this case).

Keep default **Partial safety factors**.

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

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

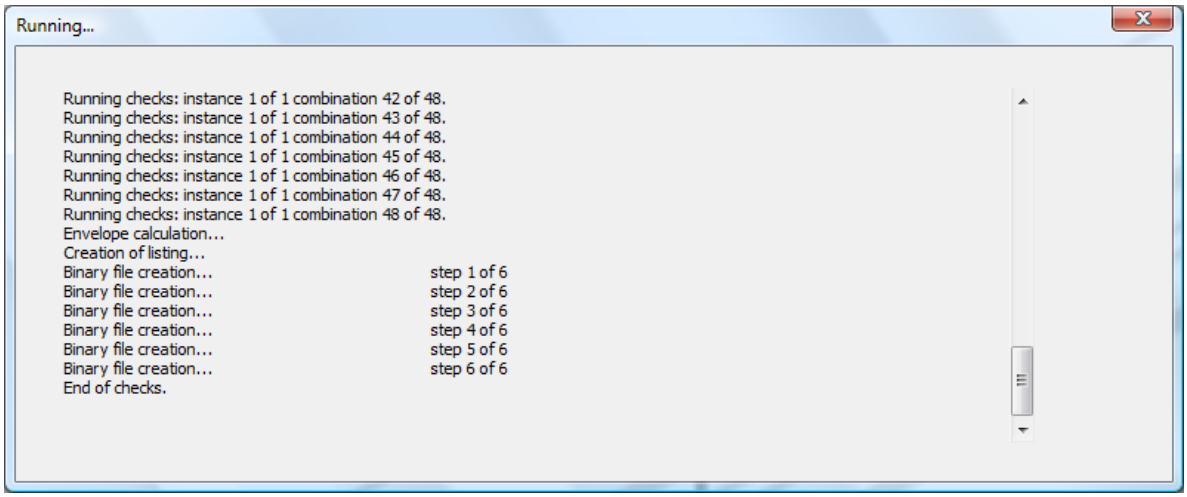
The following figure shows the dialog box with the settings we are using now.



Press **OK** to save settings.

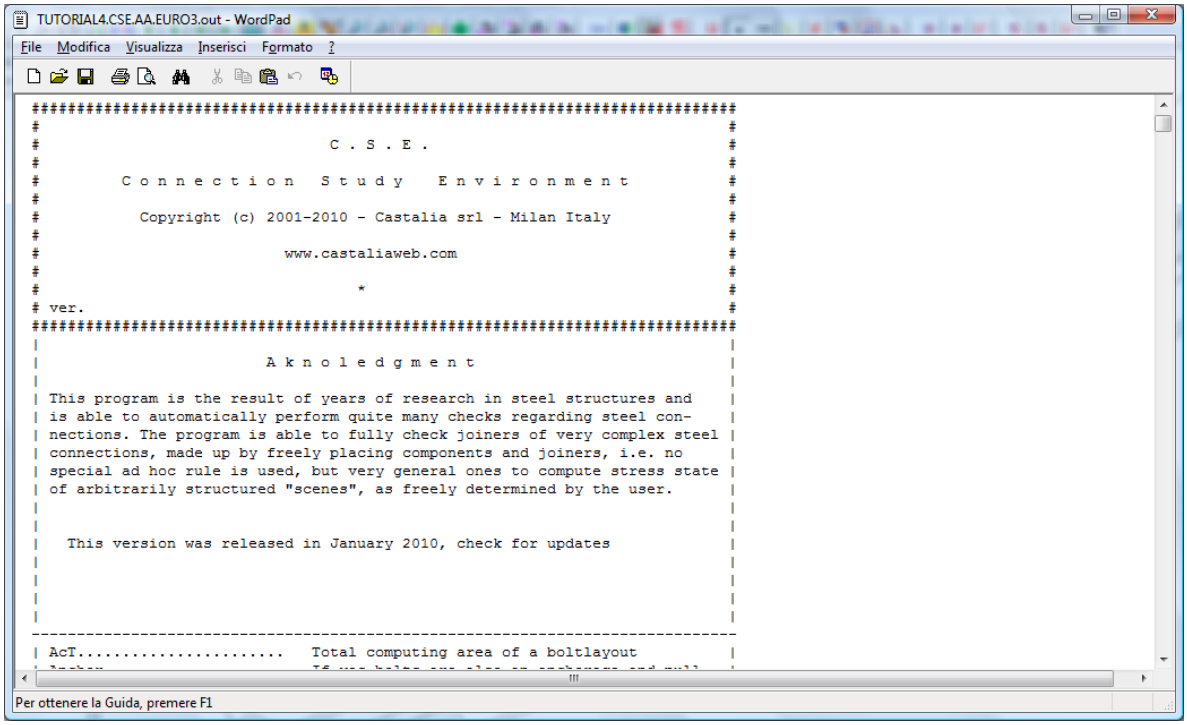
2.7 STEP 7: EXECUTING CHECKS

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



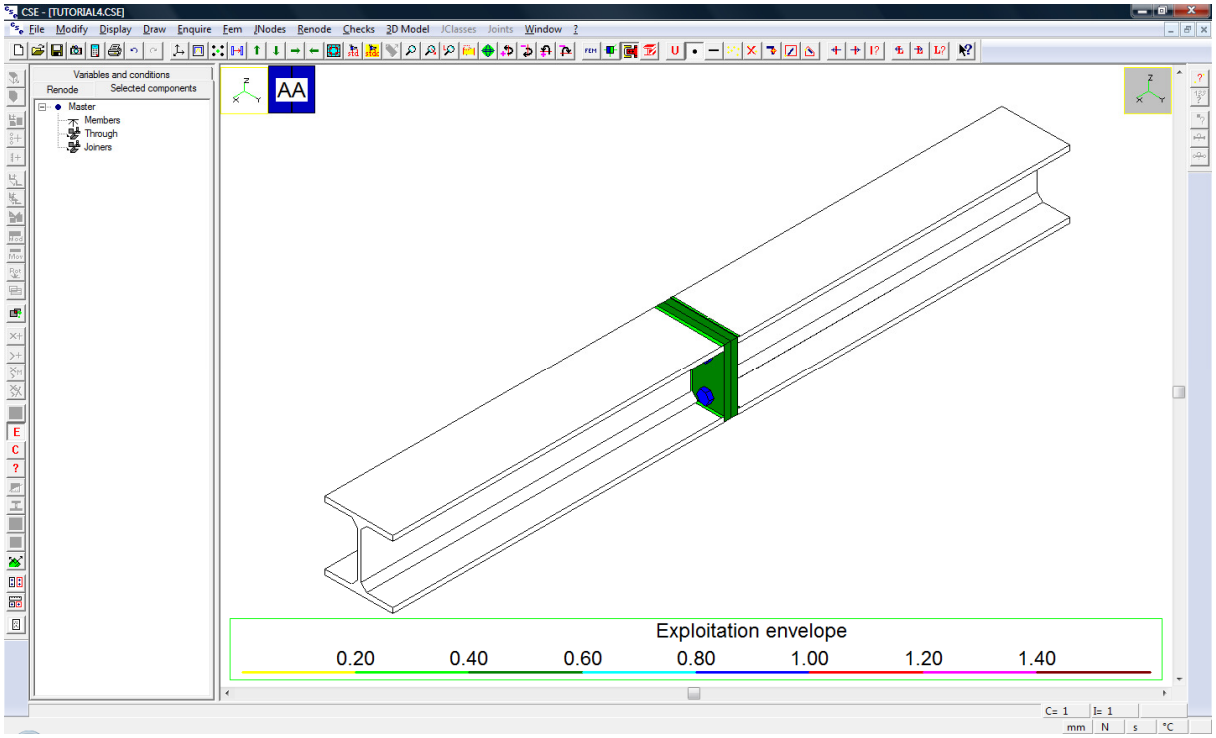
The window above will appear, it is a log window explaining what happens. At the end just close it by clicking over the red-background cross.

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



2.8 STEP 8: EXAMINING RESULTS

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

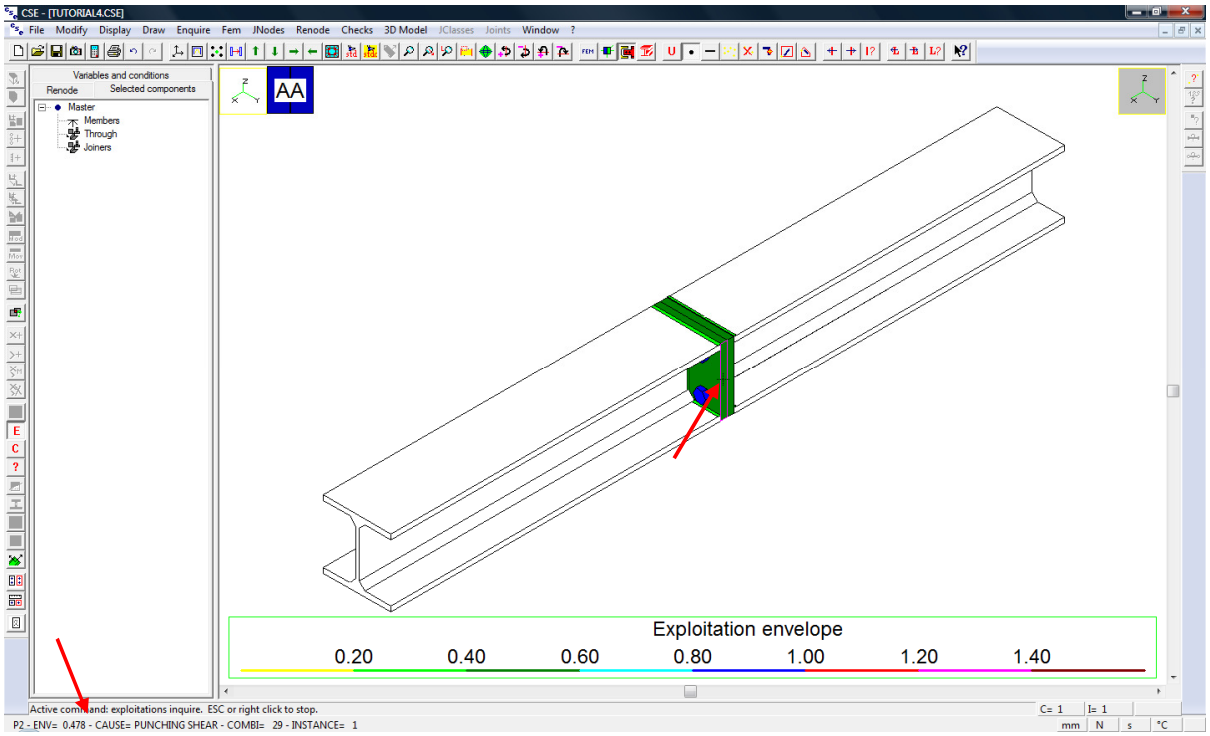


Only the following aspects have been checked, in this tutorial:

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

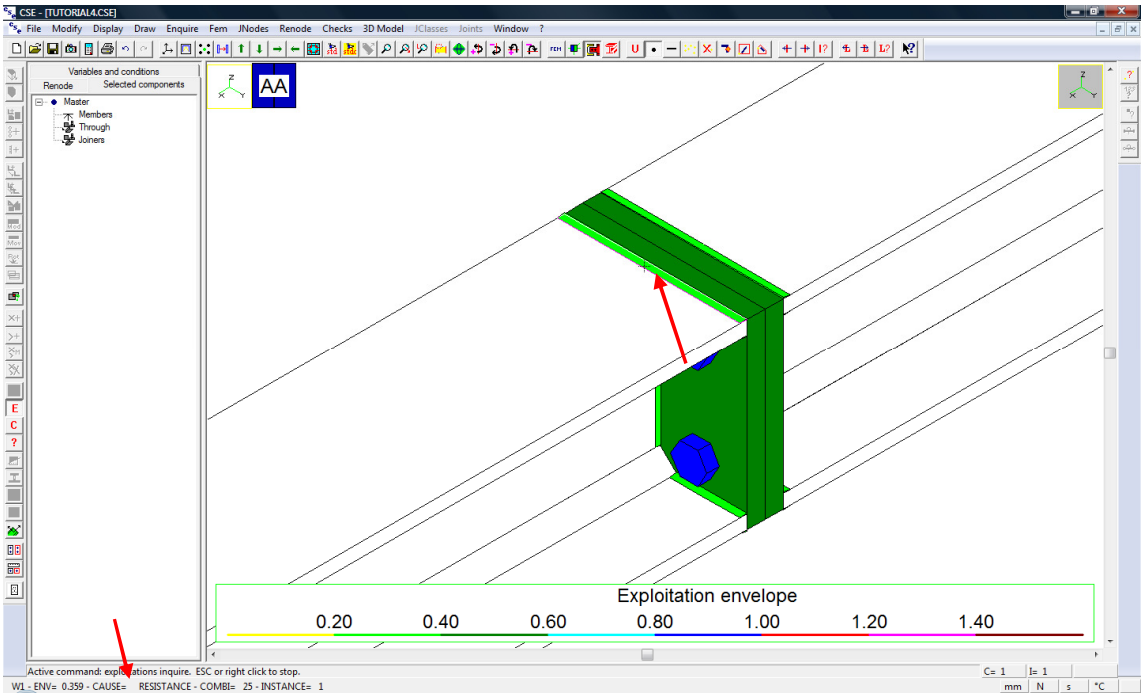
Members are white because they were not involved in any check. They do not have hole for bolts nor cuts. Members resistance check is covered by weld layout resistance check, being on the safe side (as explained later).

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

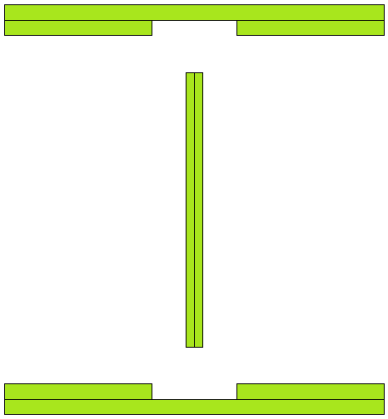


For example, maximum exploitation for P2 plate is 0,478 in combination 29 (m2 positive M2 bending moment); the cause is punching shear. Enquire the other components to know their maximum exploitation.

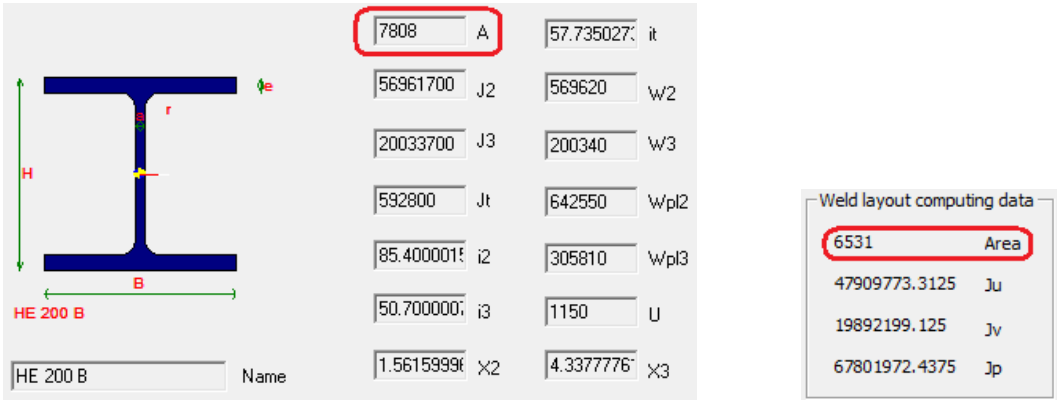
Welds maximum exploitation is 0,359 in combination 25 (m2 positive axial force). This is the maximum value for all the welds.



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



Weld layout has a smaller area than HEB200 shape.



Area of HEB200 = 7808mm^2

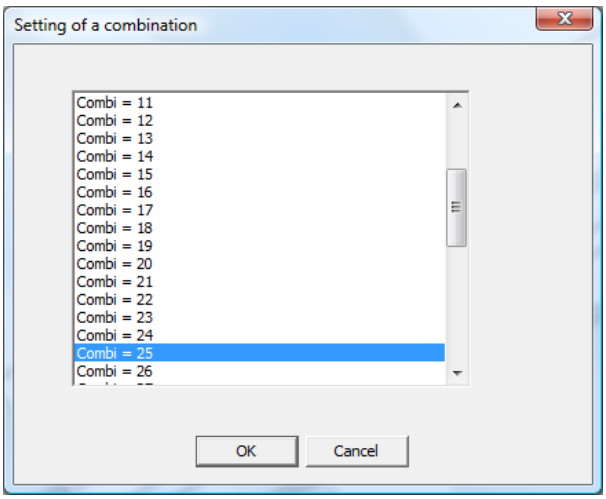
Weld layout total area = 6531mm^2

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

$$0,3 \times 7808mm^2 / 6531mm^2 = 0,359$$

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

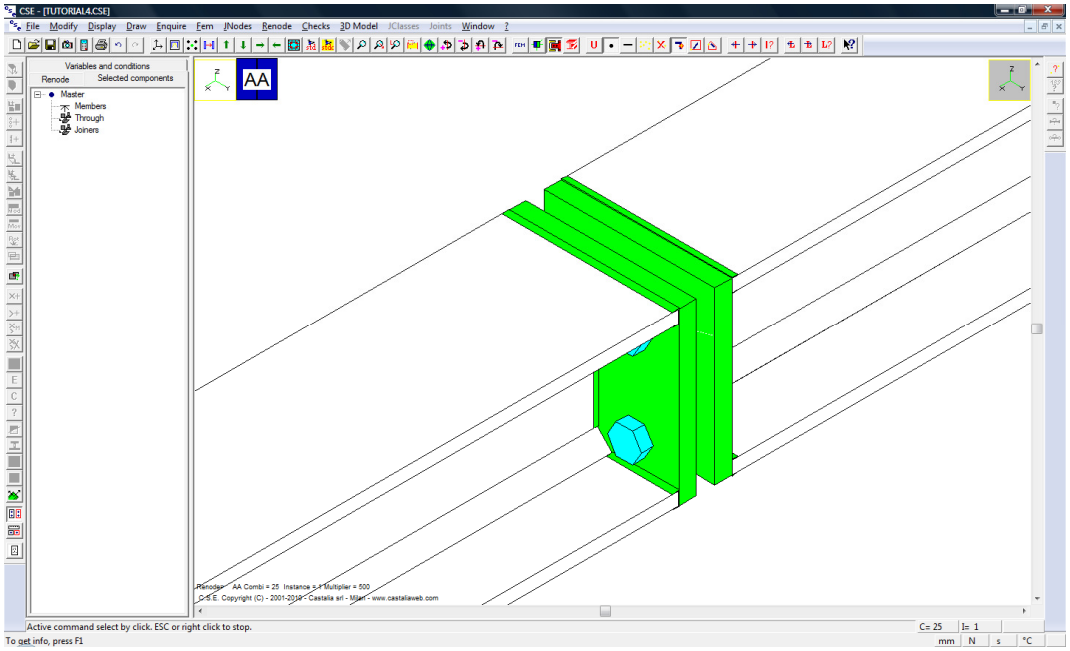
Use **Checks – Combi** L?, select combination 25 in the dialog (m2 positive axial force) and press OK.



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

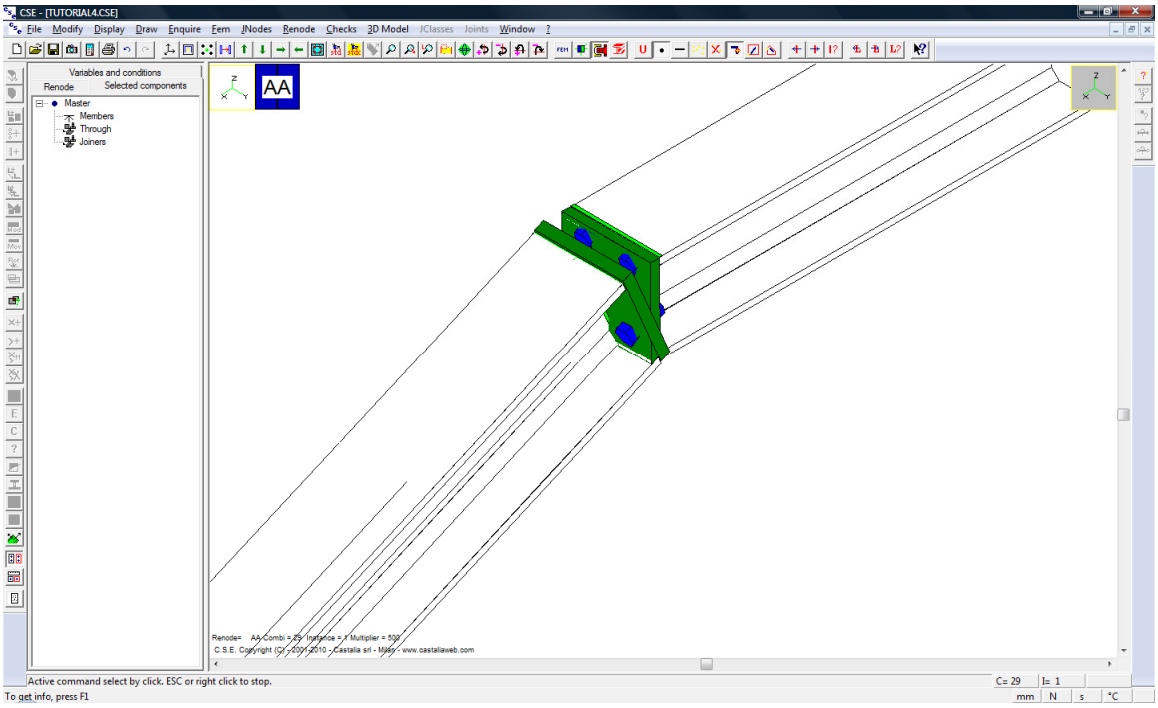
Note: if displacements are too small use **Checks – Displaced scale** and set a greater multiplier.

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



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

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


CSE - [TUTORIAL4.CSE]
File Modify Display Draw Enquire Fem JNodes Renode Checks 3D Model JClasses Joints Window ?
Variables and conditions
Renode
Selected components
Master
Members
Through
Joiners
AA
Active command select by click. ESC or right click to stop.
To get info, press F1
mm N s °C

