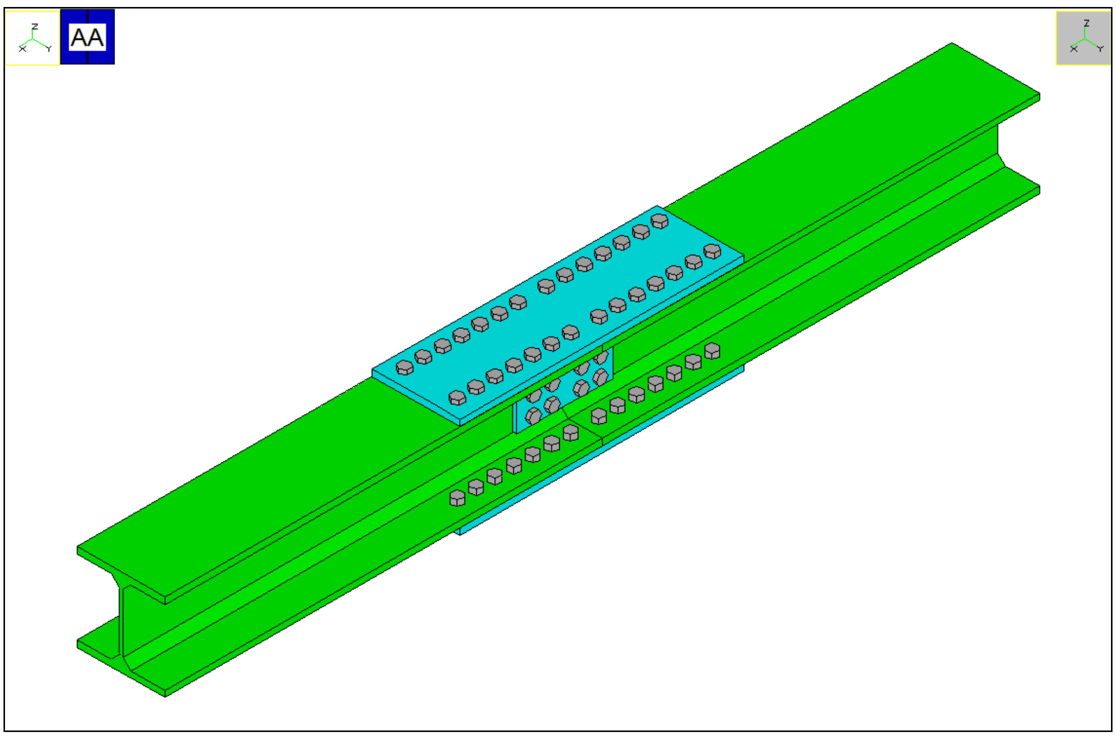
Paolo Rugarli

C.S.E.

Connection Study Environment

Tutorial 5: Splice joint (bolted connection)



CASTALIA

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

Referring to CSE Version 4.40
Rev.3 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

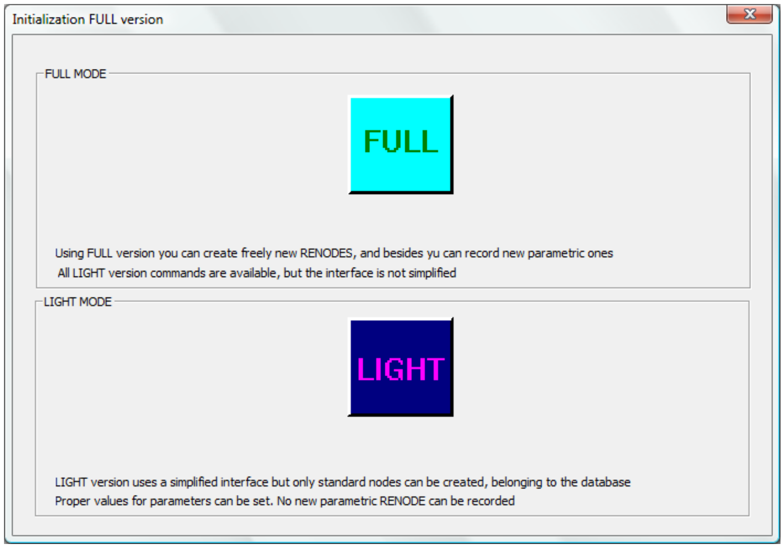
This tutorial is a tool to start the understanding of how CSE works. No special efforts to have realistic force values have been made, however this will explain several features of the program. By following this tutorial you will be able to:

- Create a dummy fem model to study a splice joint
- Assign the materials and cross-section to the fem elements
- Search members
- Search jnodes
- Add plates and bolt layouts
- Set the checks to be performed
- Have a look at the results
- Comprehend block tearing results
- Comprehend members net sections results
- Comprehend how to use flexibility index for shear-only bolts layout

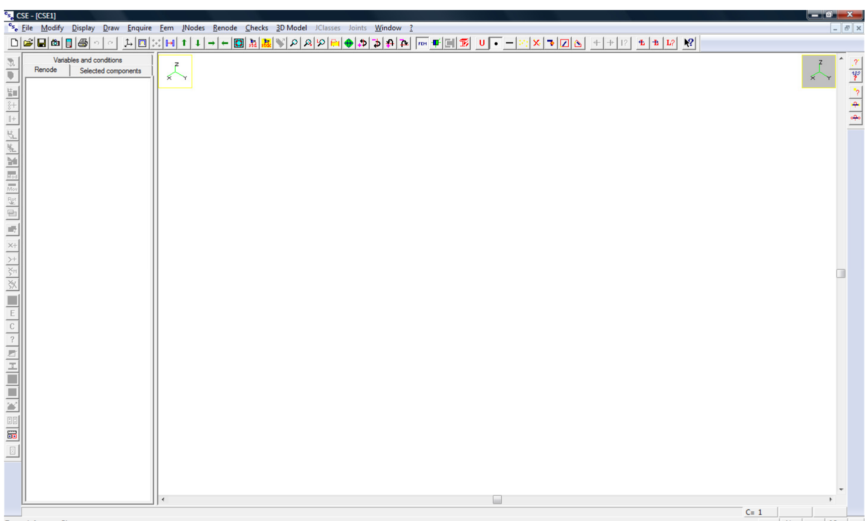
This tutorial is some like 59 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

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

2 SPLICE JOINT (BOLTED CONNECTION)



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



Initial window content: blank.

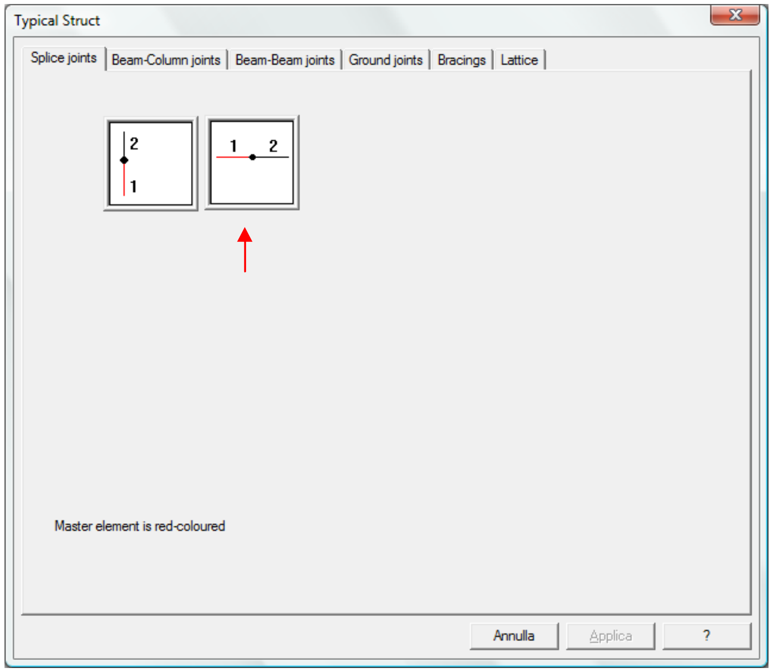
2.1 STEP 1: GETTING THE FEM MODEL

2.1.1 Getting the fem elements

Activate right window by clicking left inside it.

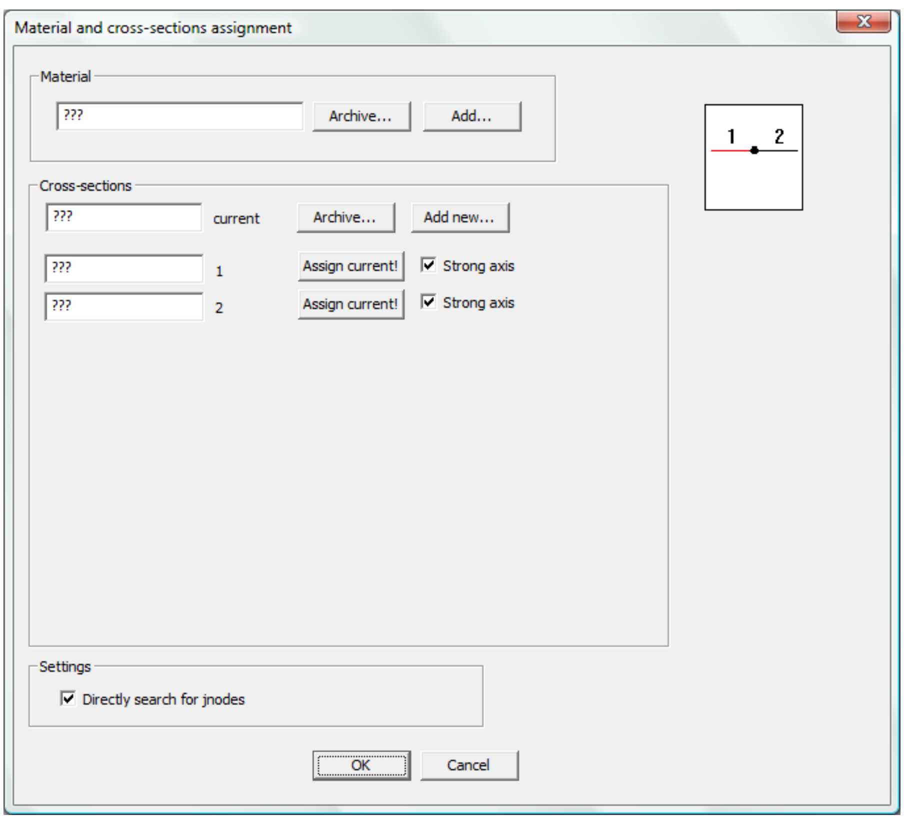
Execute the command

FEM-Elements-Typical Structures.



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

You will get the following dialog box:

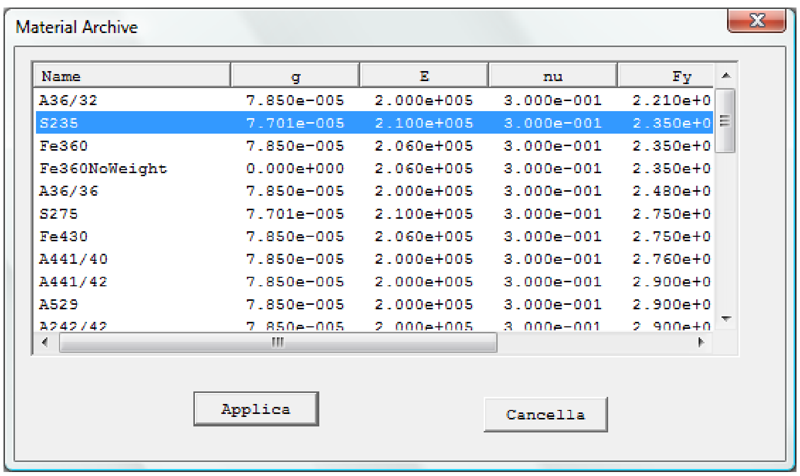


One of the beam elements has a “connection code”: it is necessary to define jnode hierarchy. The element without connection code is the master (in red), the other one is the slave: it is a **hierarchical** jnode.

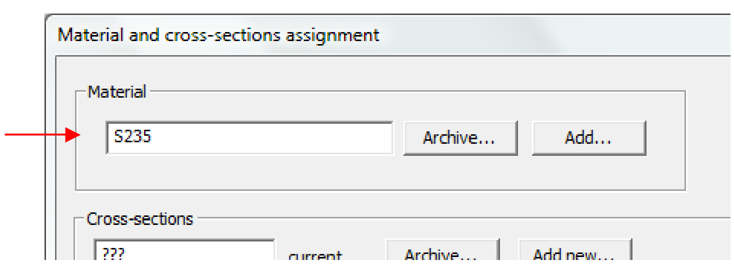
2.1.2 Assigning material

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.

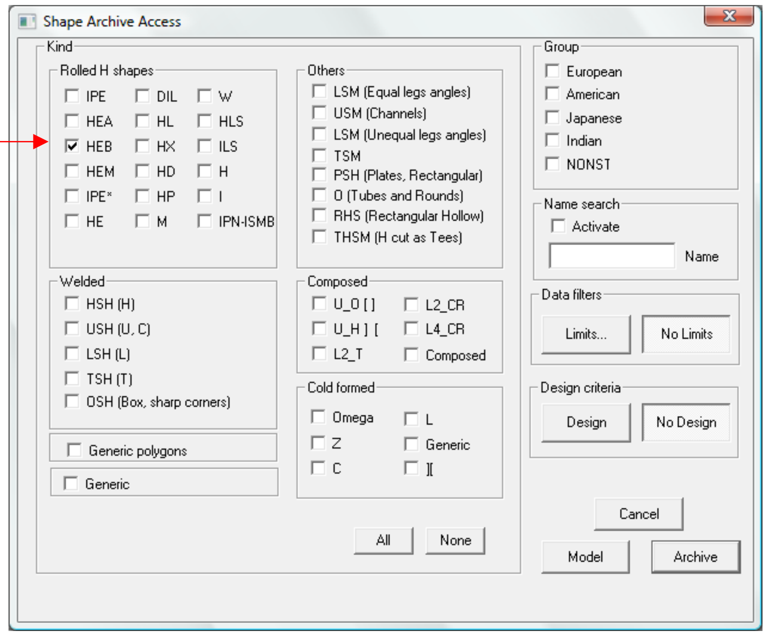


If you are using the demo the button “Archive” automatically applies the material S235.

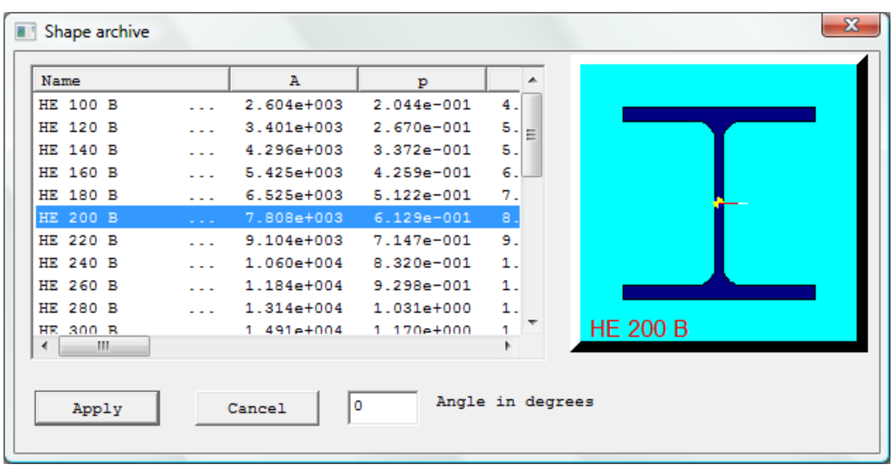
With the “Typical node” tool all the members have the same material. It is possible to define different materials with the standard fem commands.

2.1.3 Assigning cross-section

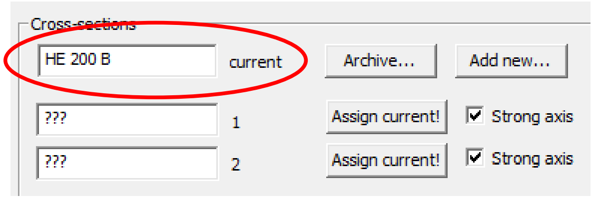
As for the material, it is possible to add new cross-sections or choose them from the archive. Press "Archive" in the "Cross-sections" box to browse the archive.



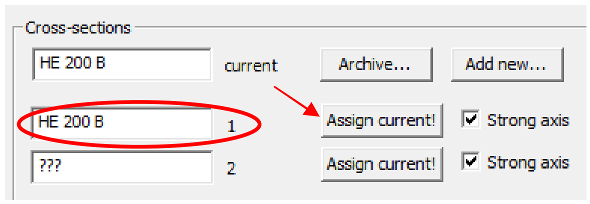
Select HEB check box and then "Archive" to filter HEB kind among all the available cross-sections. Then choose the section HEB200 by selecting the appropriate row, and press the "Apply" button.

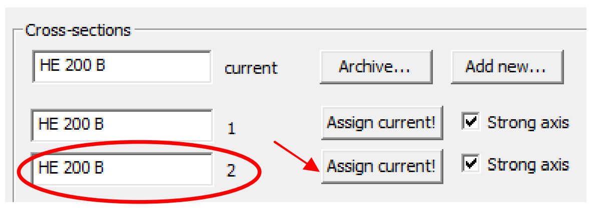


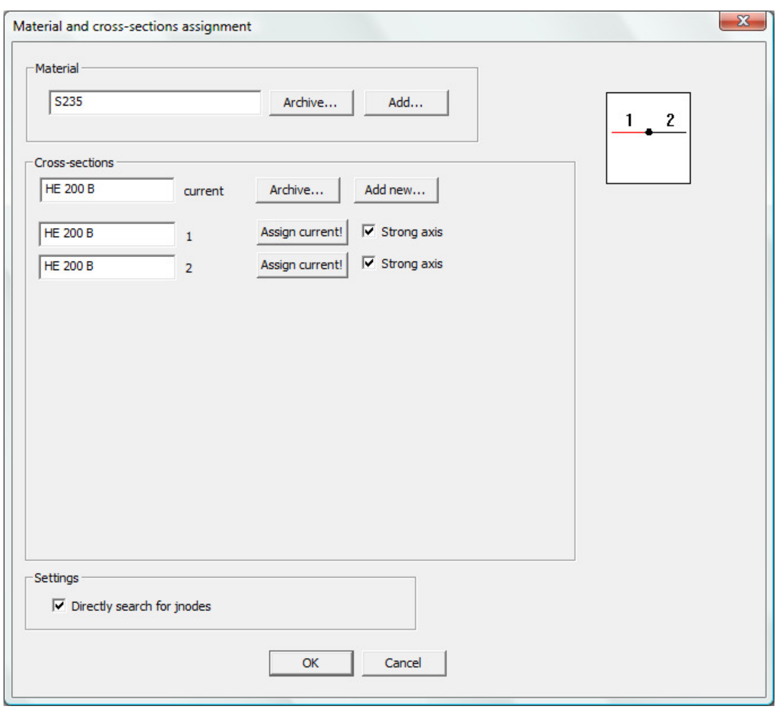
The chosen shape will appear in the box called "current".



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





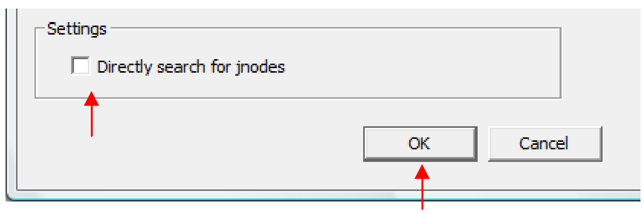


N.B. It is possible to change the current cross-section before assign it to member 2 in order to define different shapes for the members.

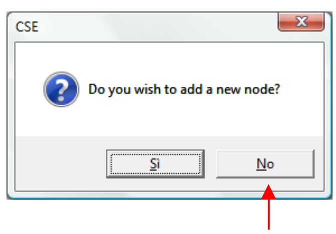
It is possible to apply a rotation of 90° to the members removing the tick from “Strong axis” boxes. In some nodes, not in this case, it is possible to define hinges for the slaves with proper tick boxes.

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.

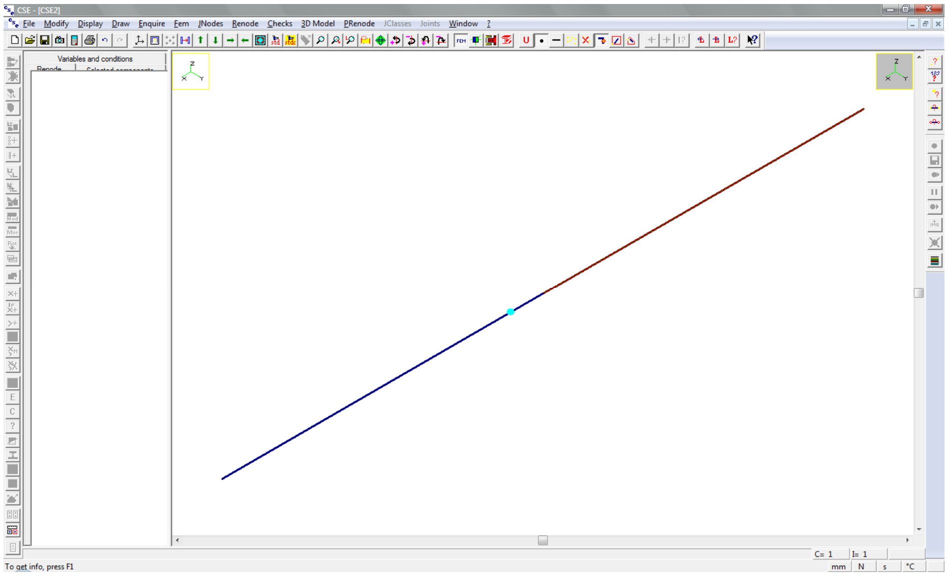
Remove that tick to see and understand step by step what could be done automatically (steps 2, 3 and 4 of this tutorial).



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!

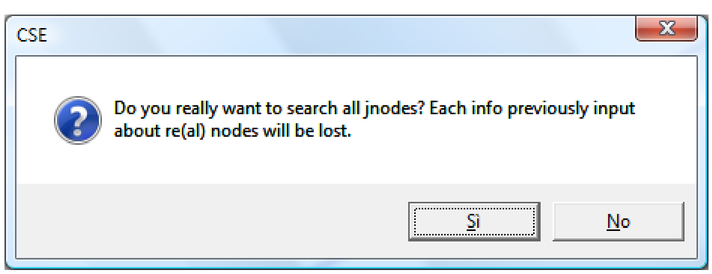
2.3 STEP 3: SEARCHING JNODES

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:

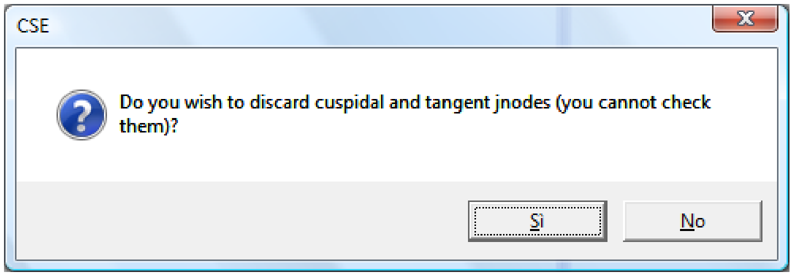
JNODE-Search jnodes!

and answer "yes" to the following question:



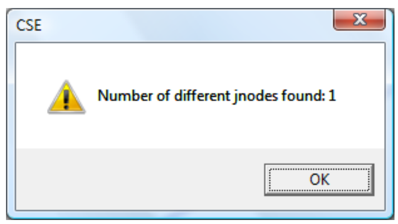
As no real nodes info has been defined you will not lose anything. Jnode search is usually done once for all in a model. Before beginning to add RENODES, you will check that the JNODES found are correct. This depends also on how the fem model has been prepared.

The following dialog appears:



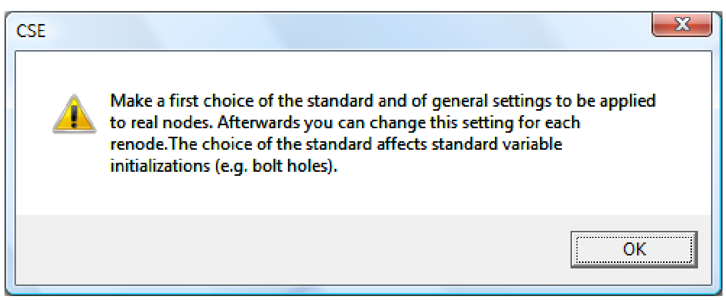
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:

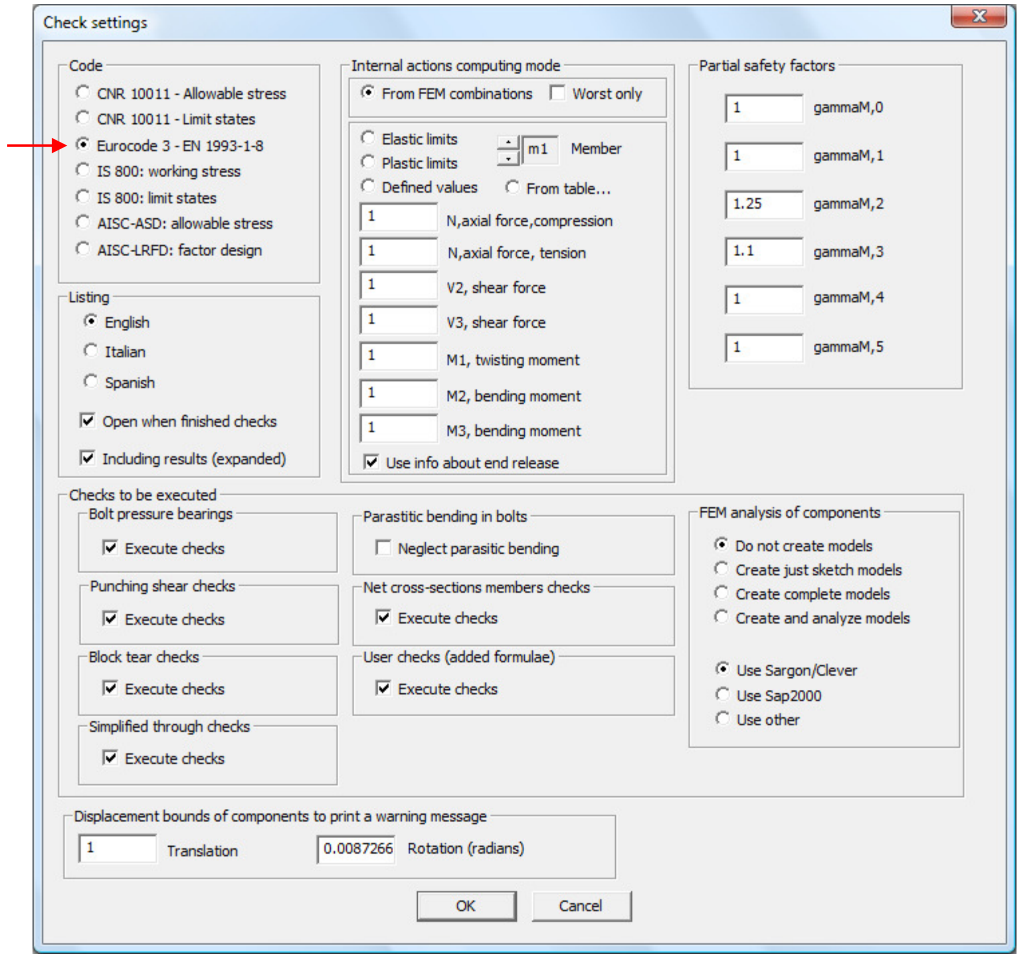


This means that in this model there is just one jnode, the splice.

Now the following message appears:

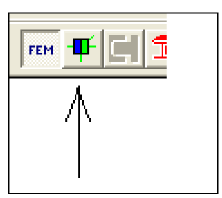


asking to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

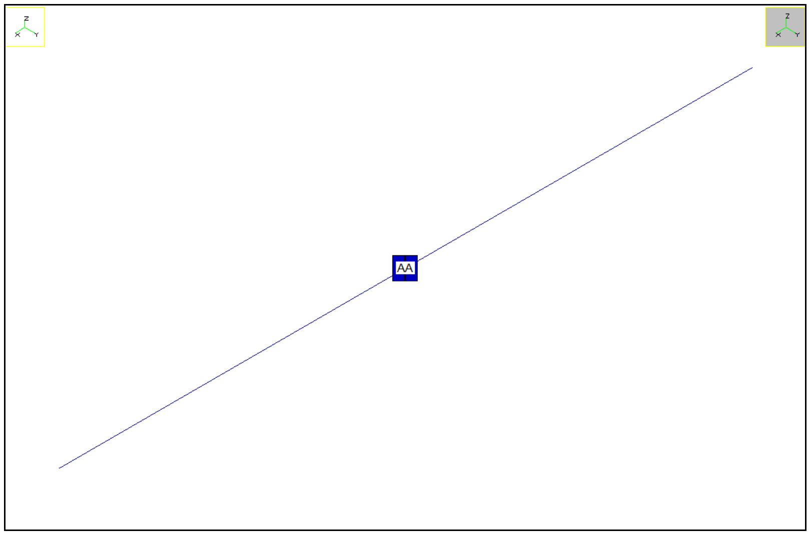


Choose Eurocode 3 and leave all other defaults, we will change them later..

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



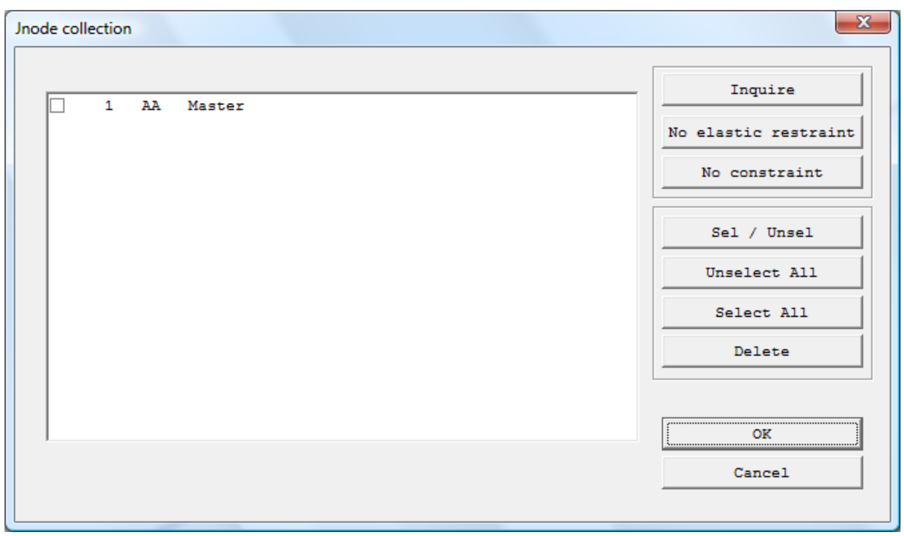
In jnode view you get the following:



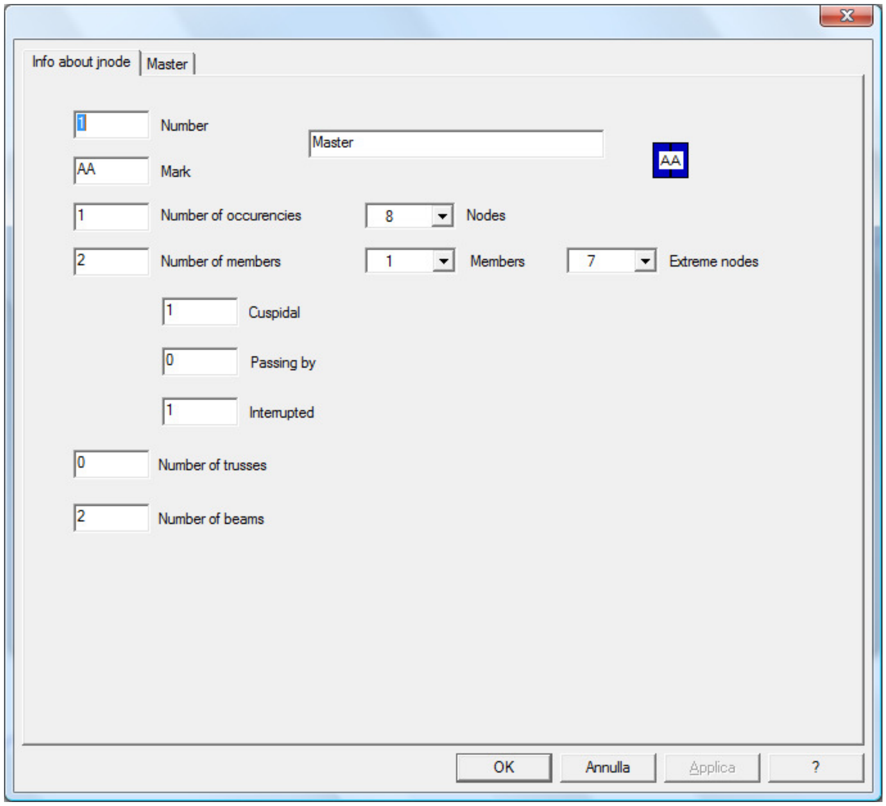
You can now get some info about the jnode found. Execute the command

JNODE-Edit...

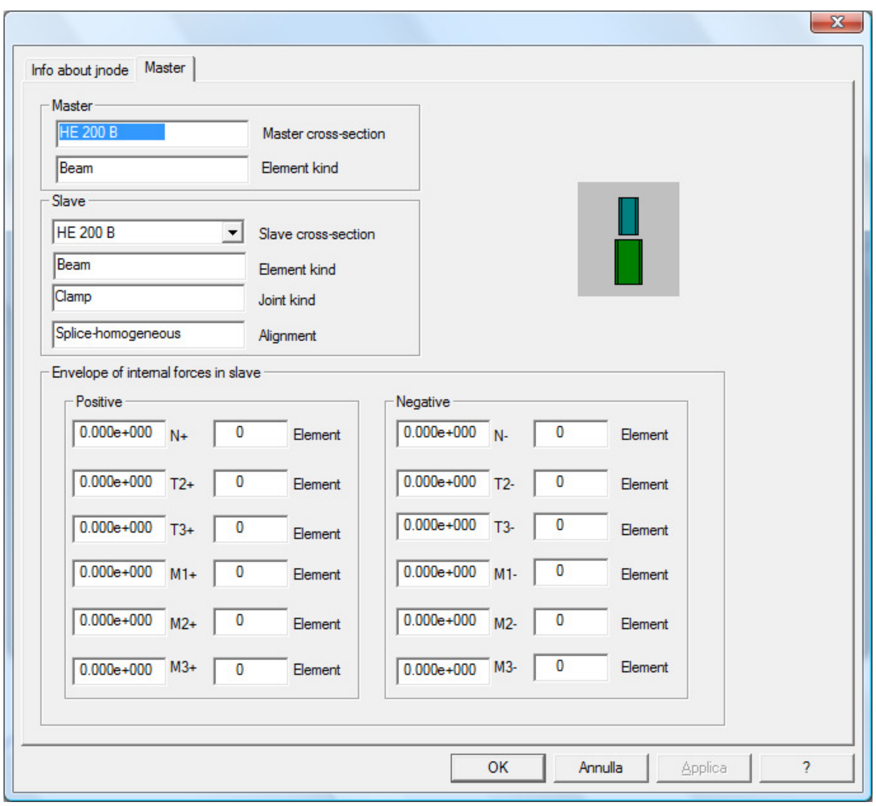
and get the following dialog:



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 TUTORIAL5.CSE.

Execute the command

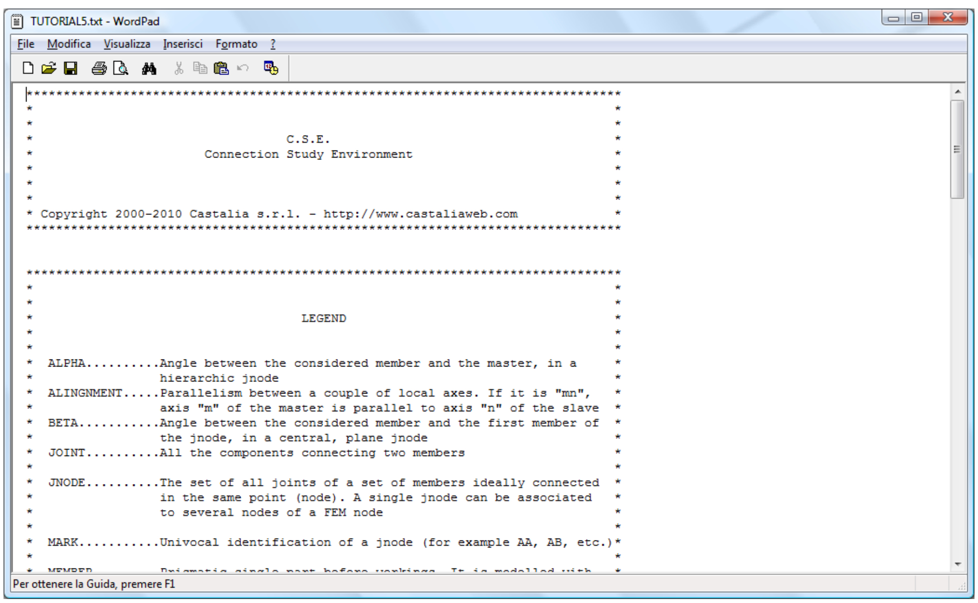
JNODES-Create listing!

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

Now execute the command

JNODES-Open listing!

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



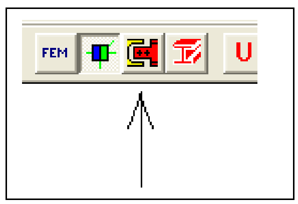
In particular you will get the following info:

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

GLOBAL TYPOLOGY OF THE JNODE: HIERARCHIC JNODE

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

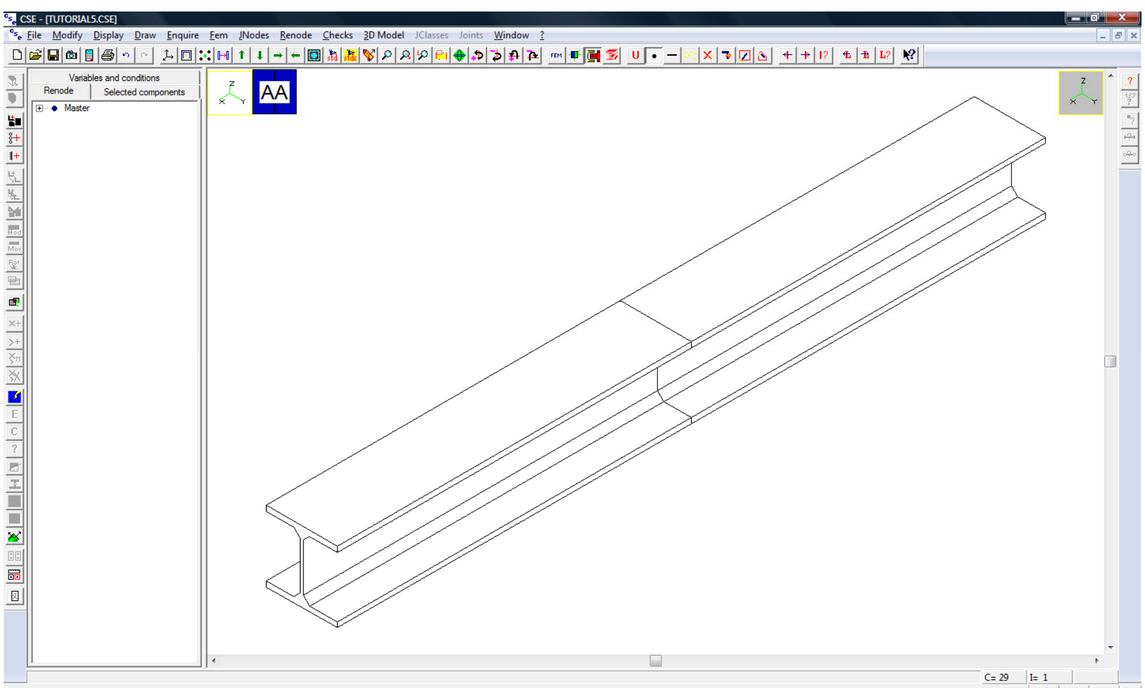
```
MEMBERS IN THE JNODE: TOTAL 2
    1     2

   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:

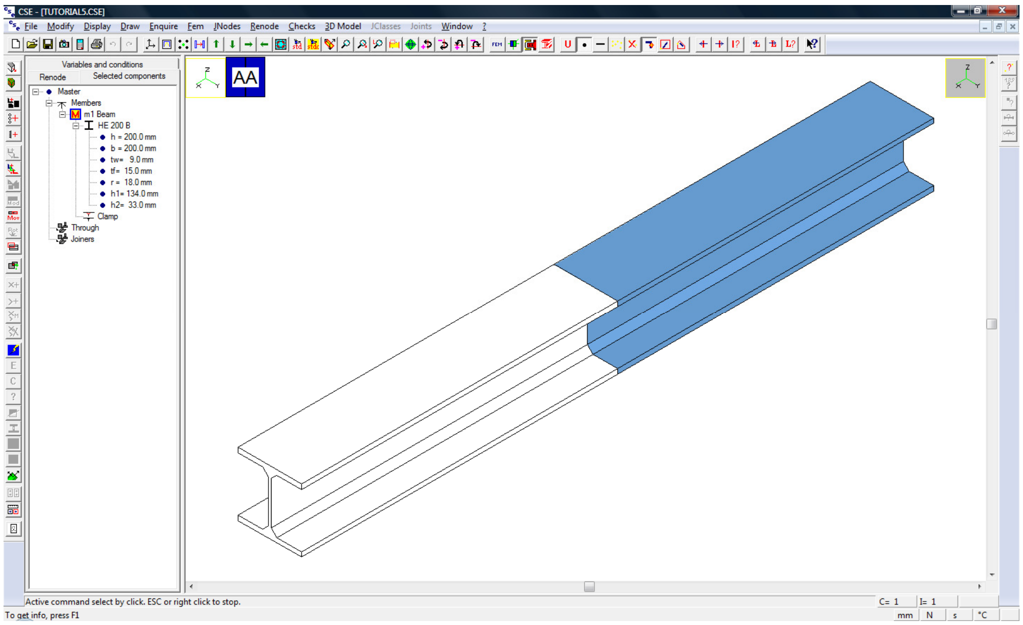


By pressing it you will get into the 3D environment where RENODES are built up, checked, and studied. You will see what follows:



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.

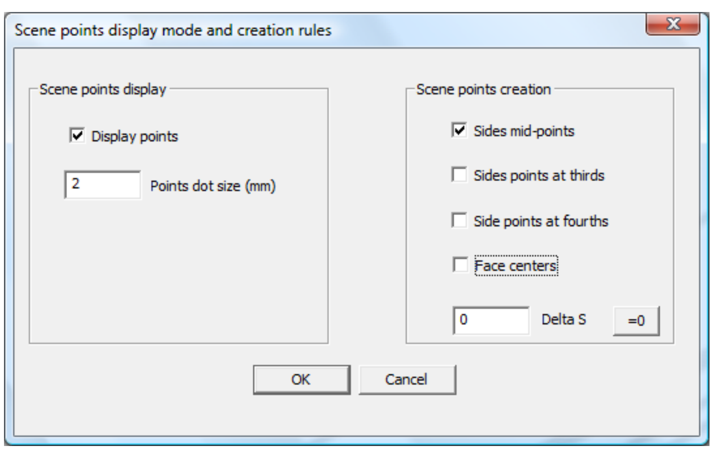
Note that clicking a member it gets selected (blue) and that the "Selected components" sub-pane in the left window is upgraded. Clicking and re-clicking you select and unselect.



2.5 STEP 5: CONSTRUCTION OF THE REAL NODE

2.5.1 Addition of rectangular plates

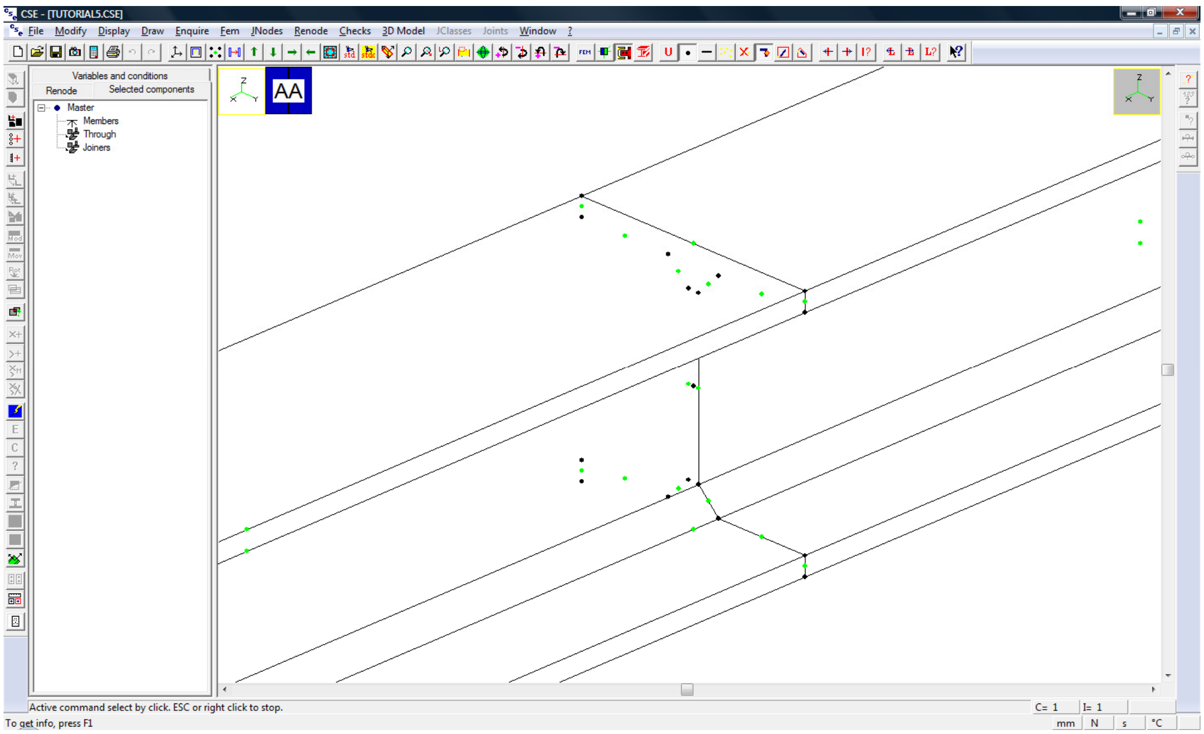
Before adding plates, use **Display – Scene points** and set these parameters in the following dialog box:



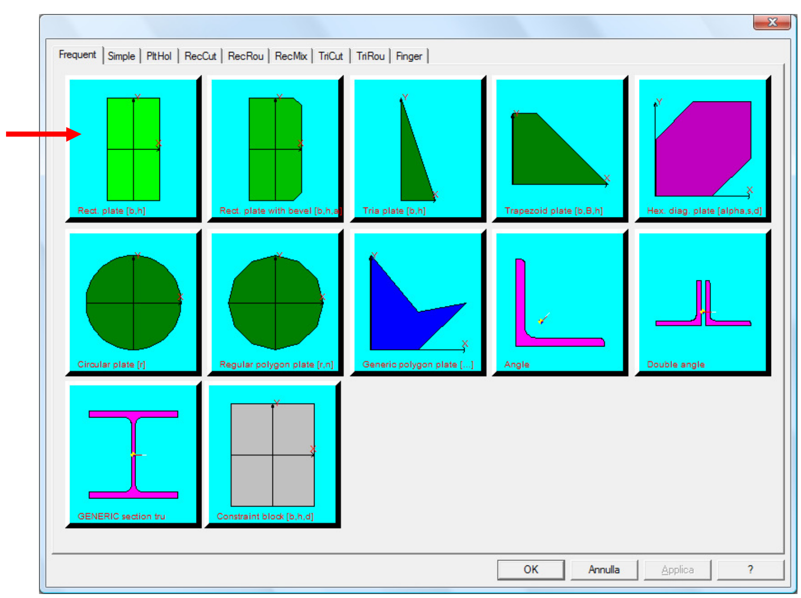
- Tick Display points
- Type *'2'* as dots size

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

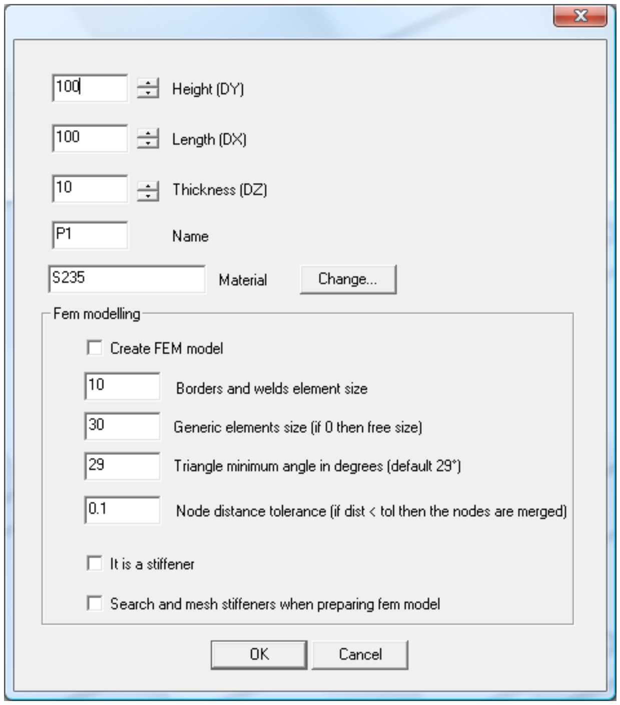
Use Draw – Rotate down  twice to have a clear view of the webs in the connection zone, and scroll mouse wheel to have a closer view.



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

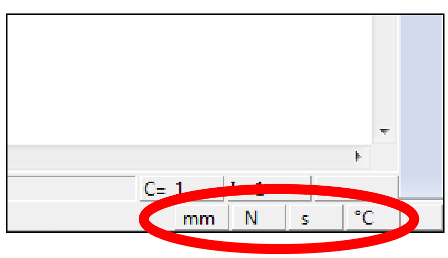


The dialog box that appears is used to set rectangular plate sizes and parameters.

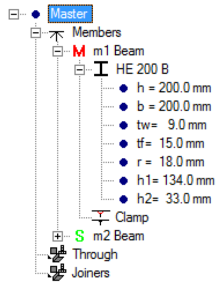


We are going to add on of the plates of the web.

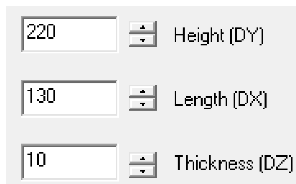
*Note: we are using millimetres as current units. Use **Modify – Units***  *to define current units, or click in the bottom right section on the unit you want to change. You can not change units while a dialog box is opened.*



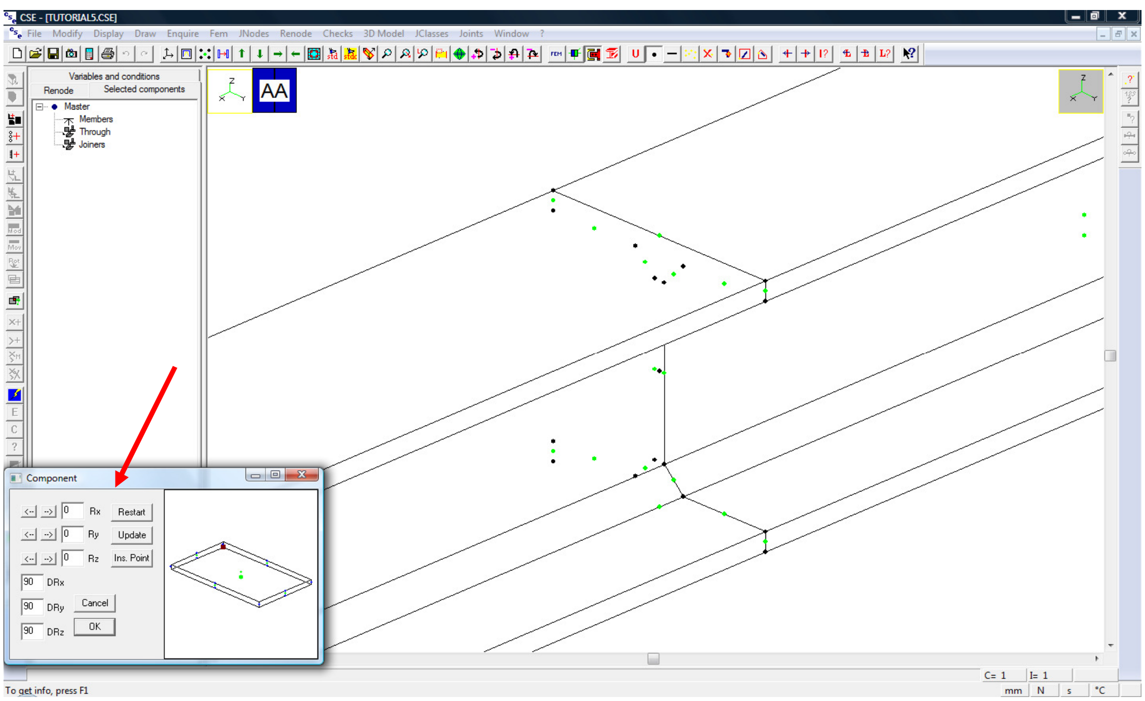
Note: open member data in the Renode pane to see cross-section sizes, or alternatively (recommended) activate Selected components pane and select one of the members. Remember to activate graphic view before proceeding, clicking in it with the mouse.



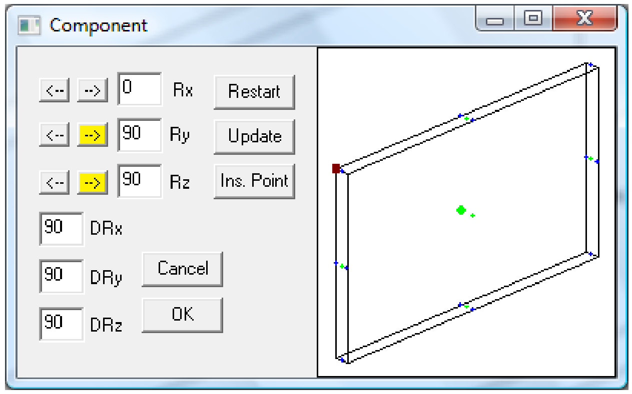
Define plate sizes as follow and leave all the other default data. Now we are not going to use automatic fem model creation.



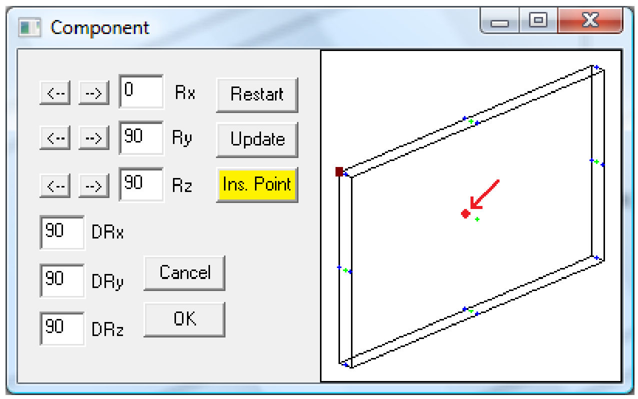
Click OK in the rectangular plate dialog box, and a new dialog will appear: here you can rotate the component you are going to insert, and choose the insertion point.



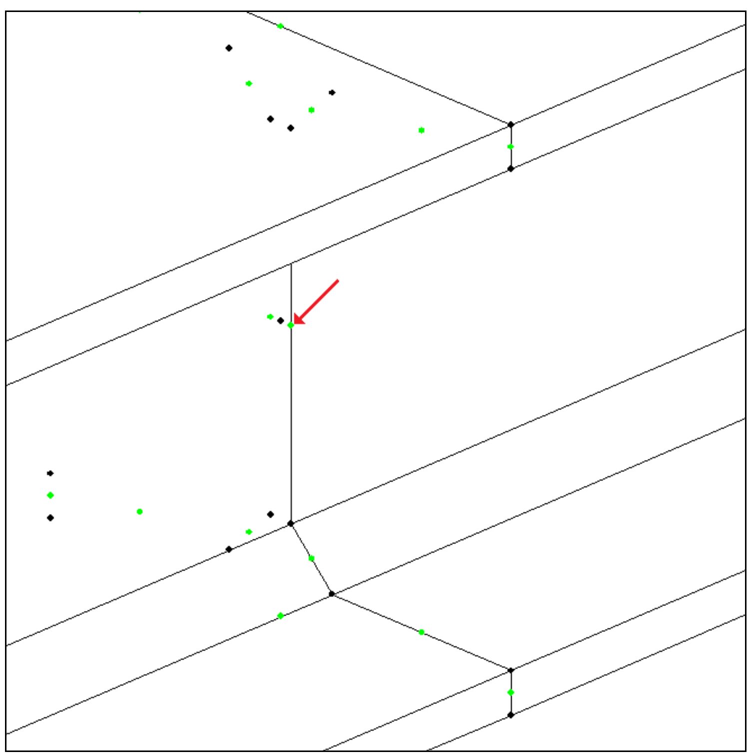
Click → in Ry row, and then → in Rz row to place the plate in the correct position



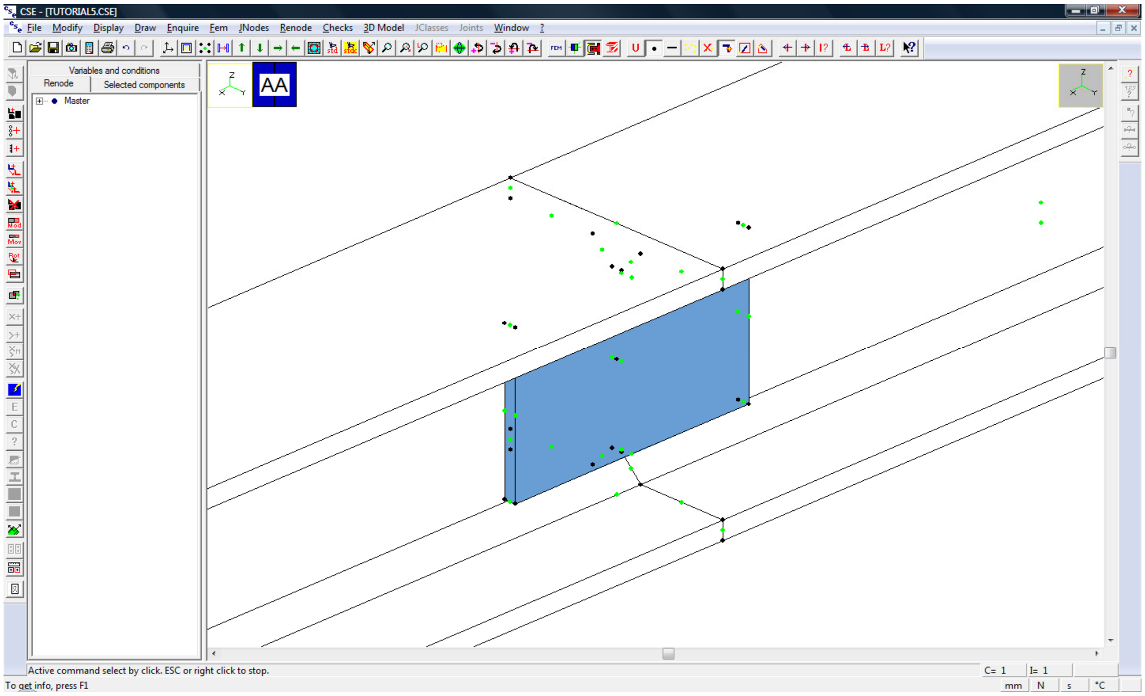
Now click “Ins. Point” and select the centre of the hidden face, then press OK.



In the scene, select the point to which previously chosen point will correspond.

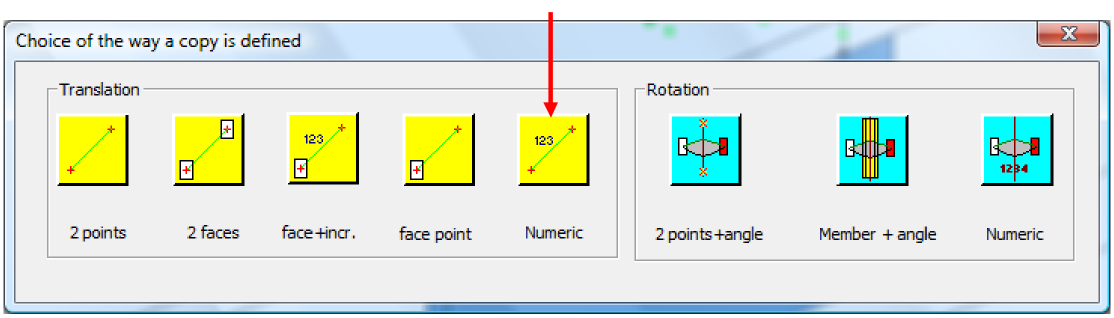


The plate will be added immediately in the scene.

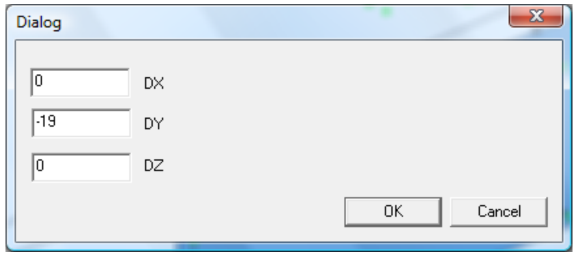


Now use **Renode – Components – Copy**  to get a copy of the plate on the other side of members webs.

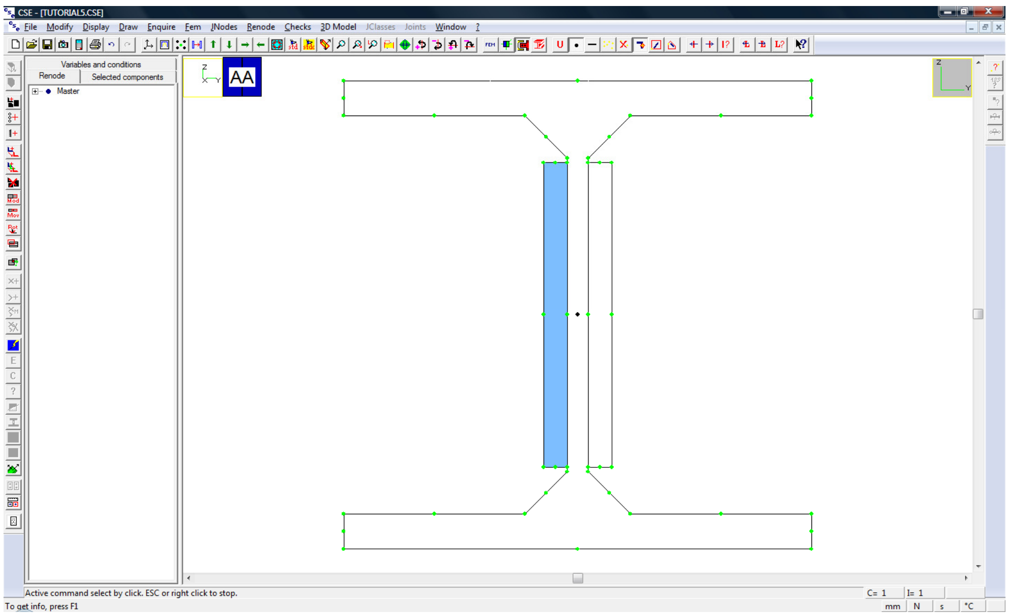
There are different ways to define copy vector; choose Translation – Numeric.



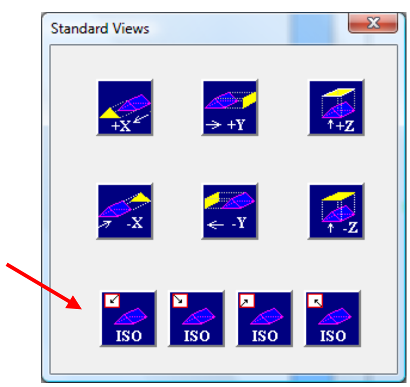
We need a 19mm translation in –Y direction: -19 is the sum of plate thickness (10mm) and web thickness (9mm). Tupe –19 in DY box and press OK.



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

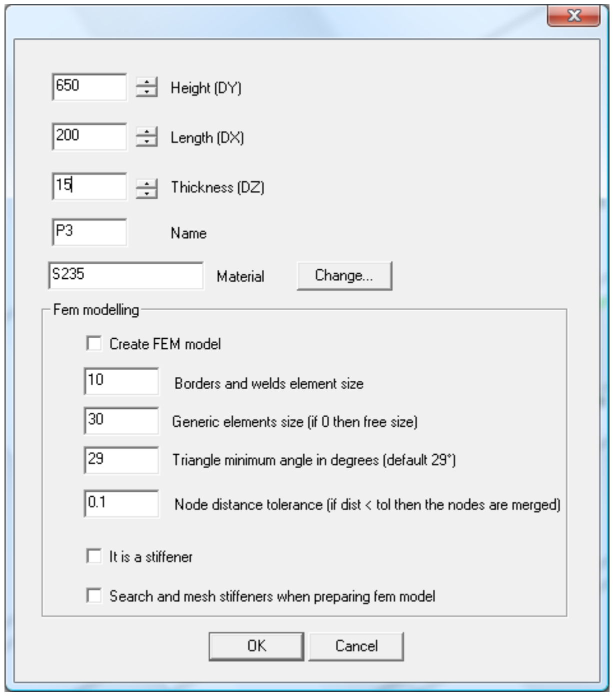


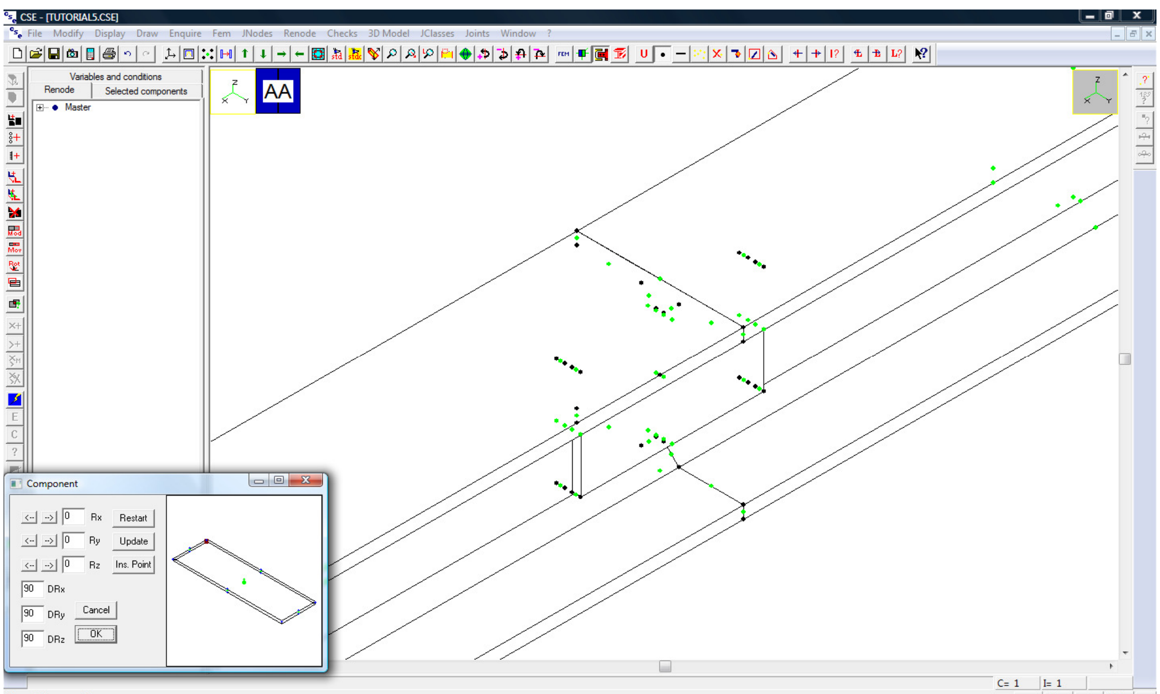
Switch back to the first isometric view from the left .

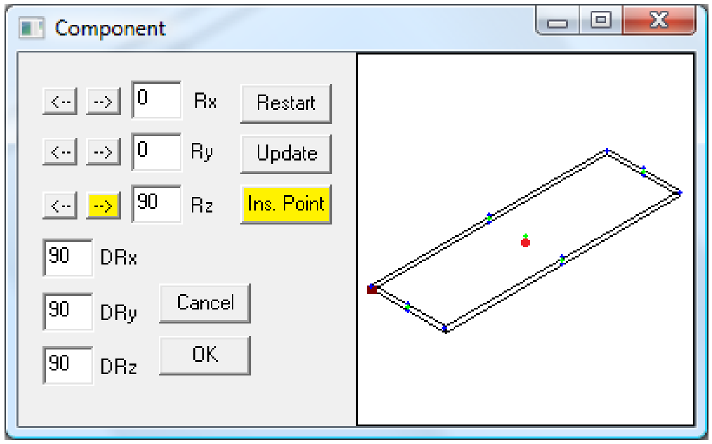


To add the plates on the flanges, add a new component as done before: **Add through** , then choose the rectangular plate.

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

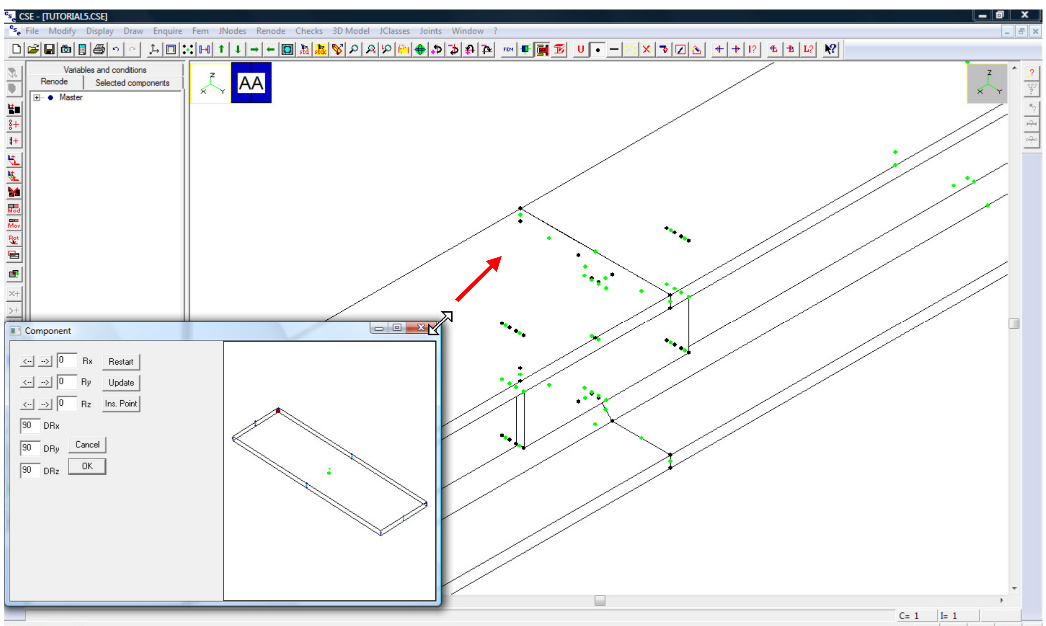




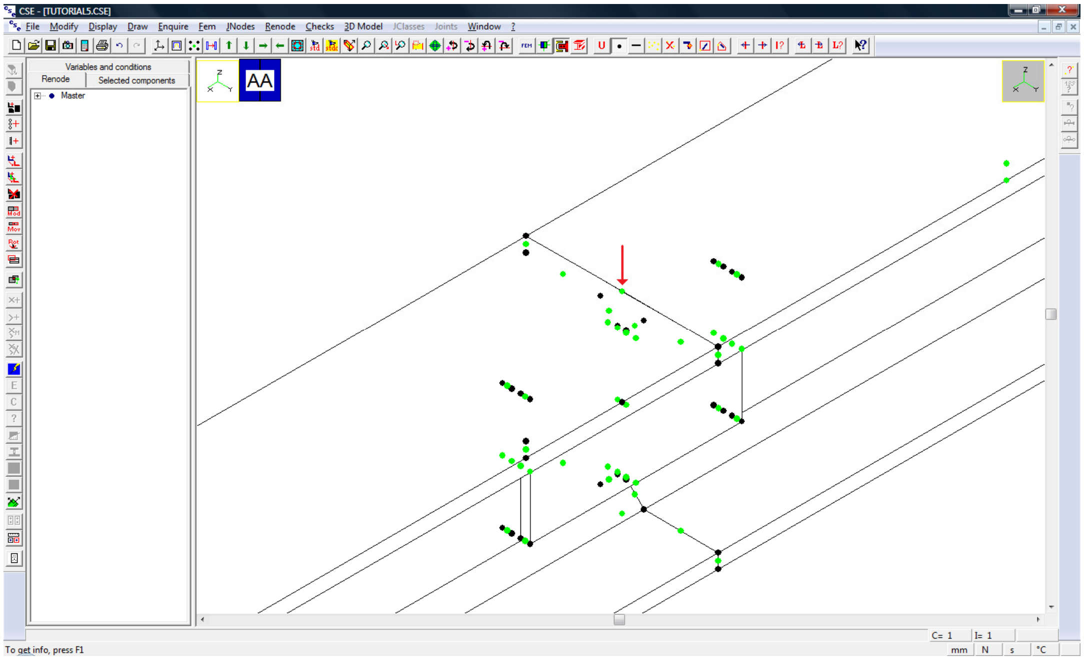


- Click → in Rz row
- Click “Ins.Point”
- Click centre point of the lower face
- Press OK

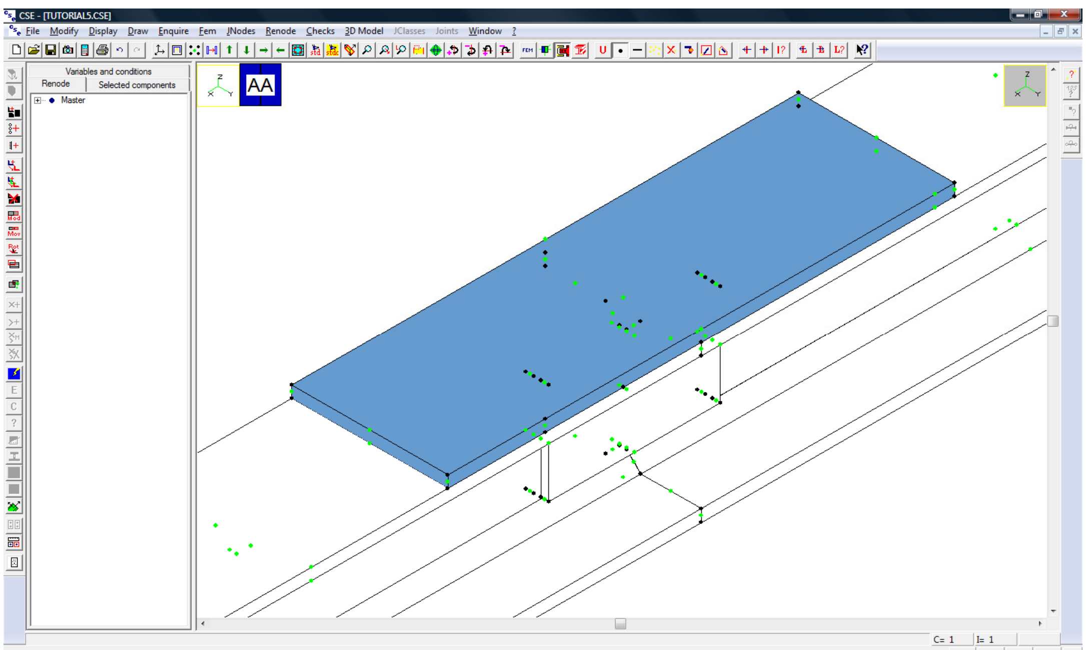
Note: you can enlarge dialog box with the mouse.



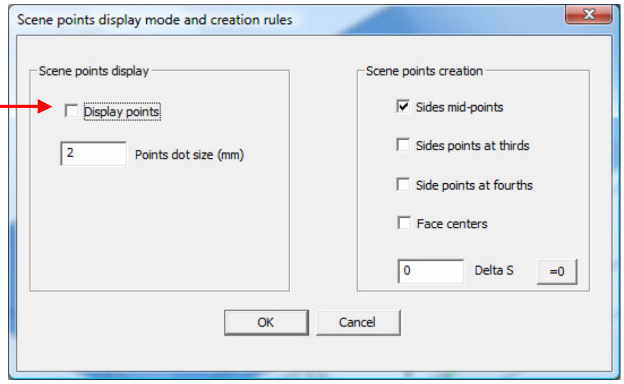
In the scene, click the point to which must correspond previously chosen point: it is the midpoint of flange upper side (zoom with mouse wheel before clicking in order to have a closer view).



The component has bee added.

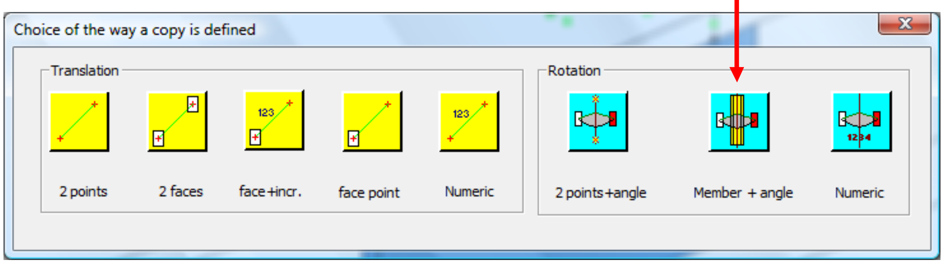


Hide **Scene points** because they are not necessary.

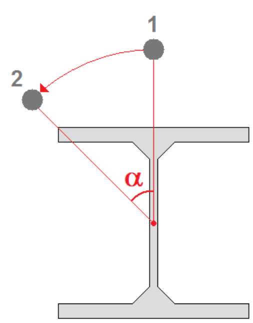


Enclose the renode in the view (**Draw – Enclose**).

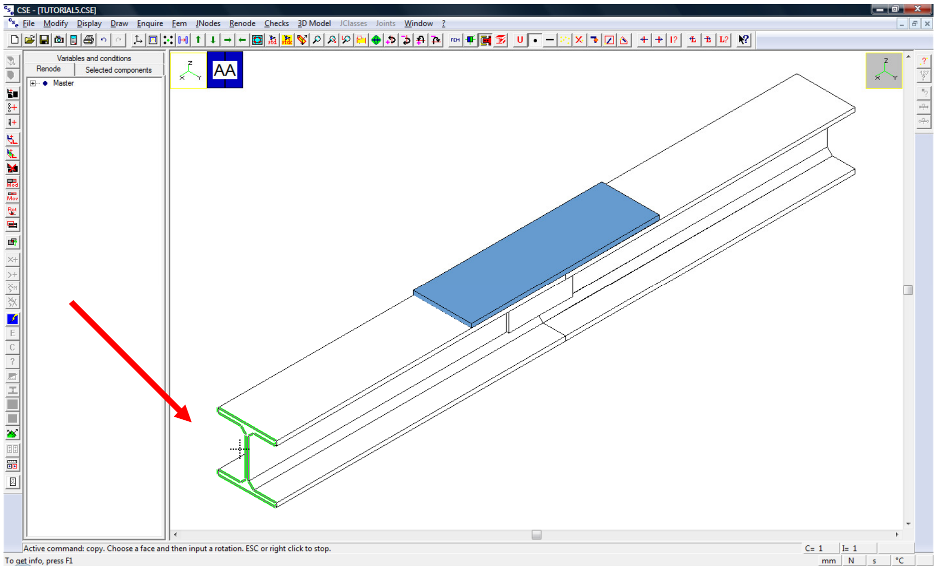
Upper plate is currently selected. We are going to create a copy on the lower flanges: **Copy**, then choose Rotation – Member+Angle.



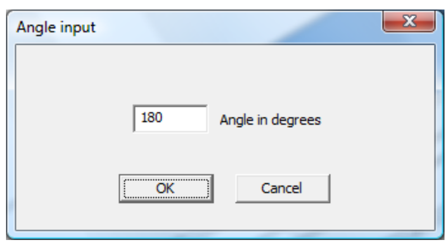
Using this mode, you must firstly click a face of the member whose axis must be taken as rotation axis, then an angle must be typed: a copy (2) of the original object (1) will be created.



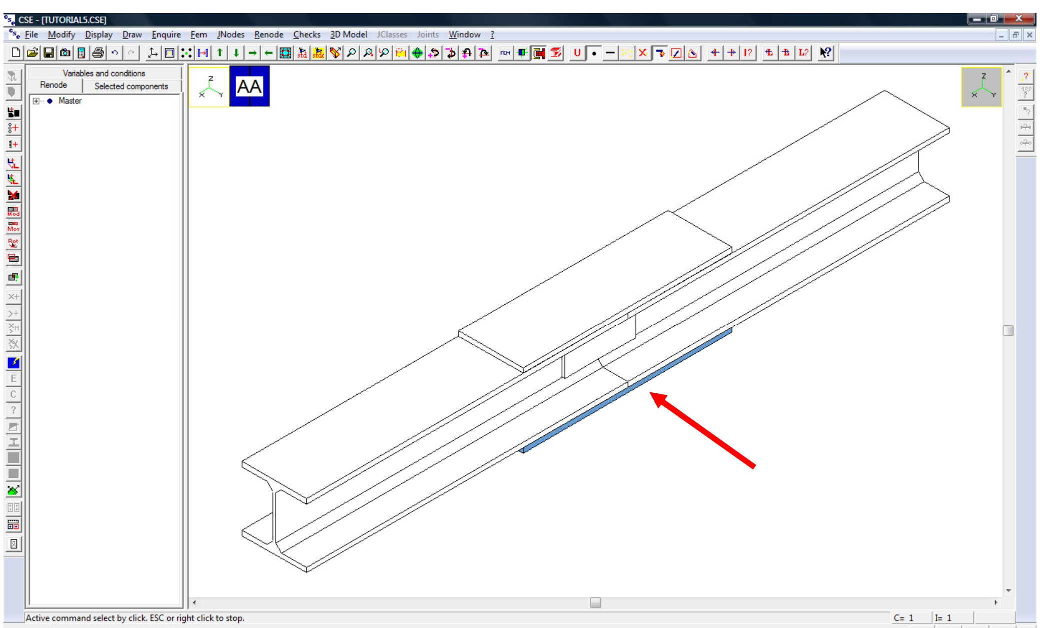
Click on any of the face of any of the members, since they have the same axis (click, for example, an end face).



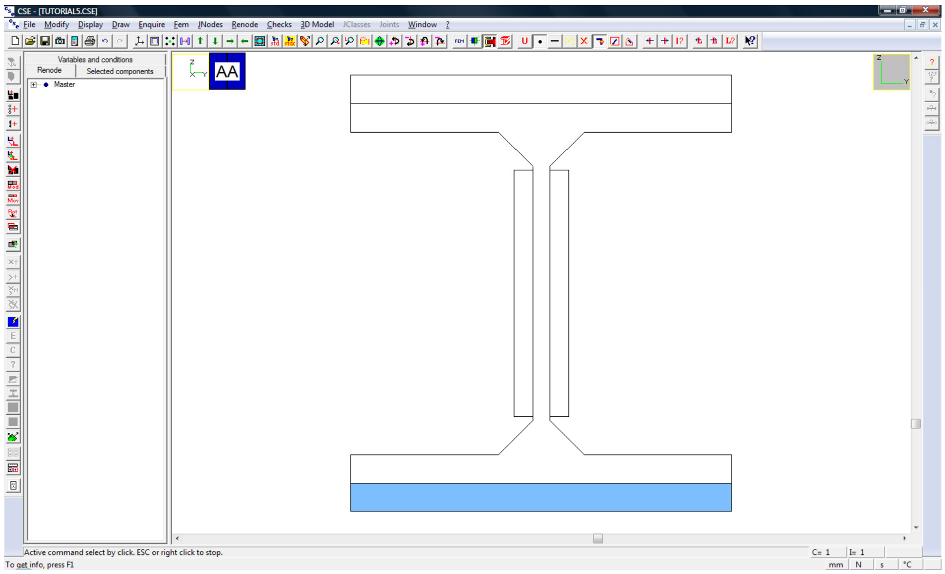
Press OK in the following dialog box to have a rotation of 180 degrees.



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

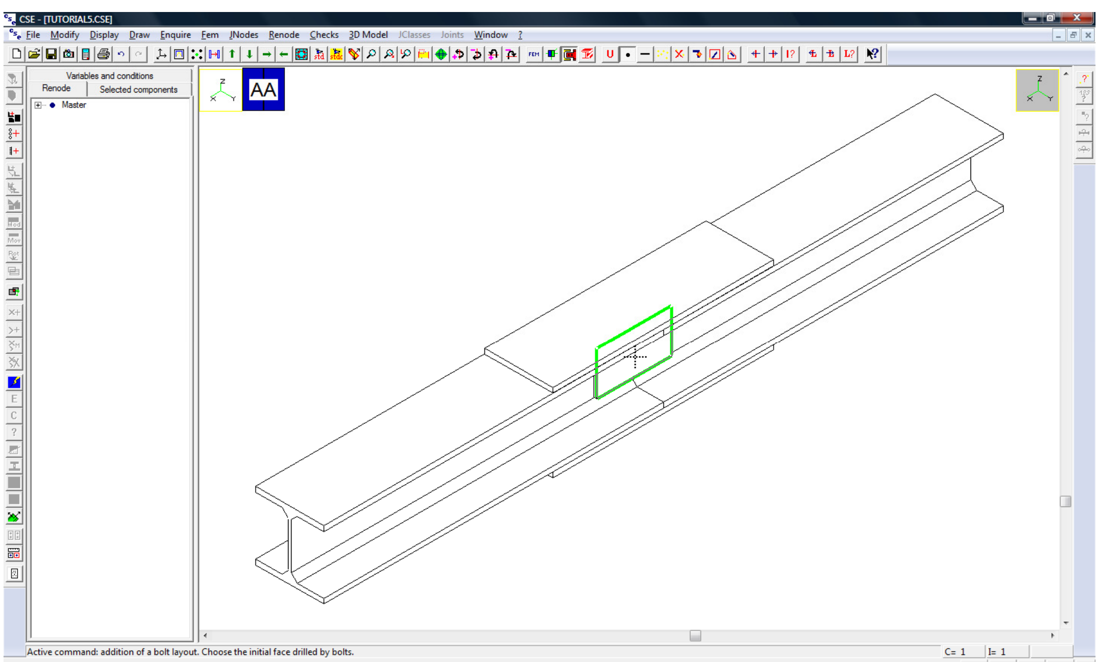


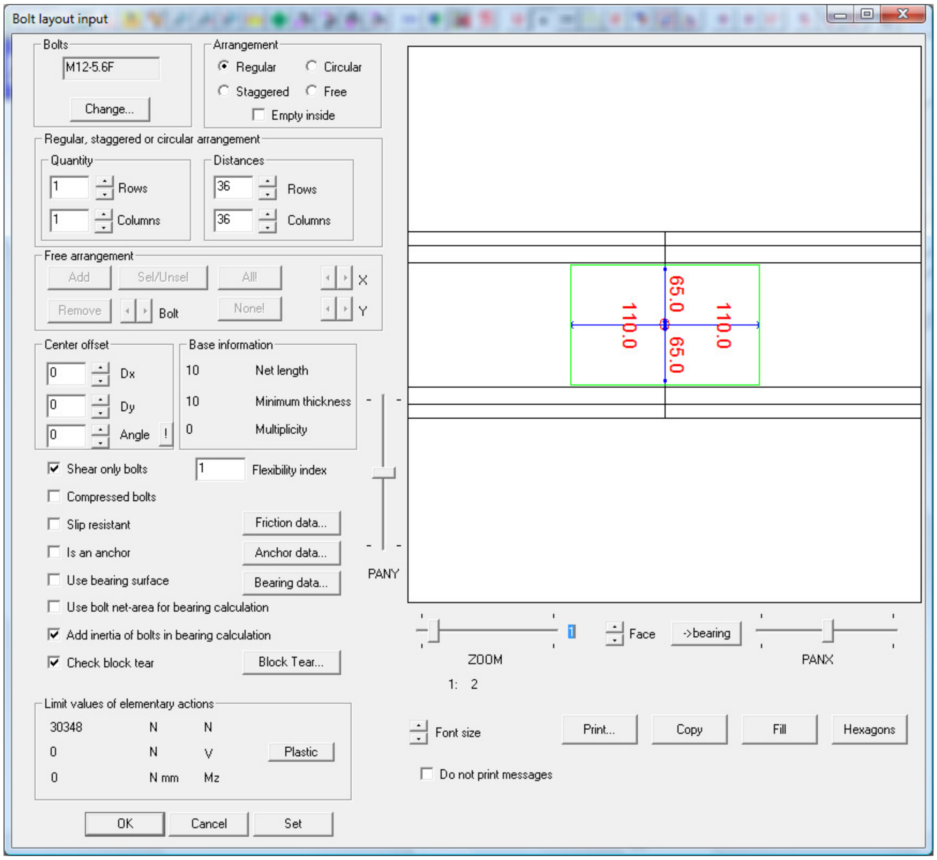
As done before, change view to see the renode frontally, then switch back to the first isometric view.



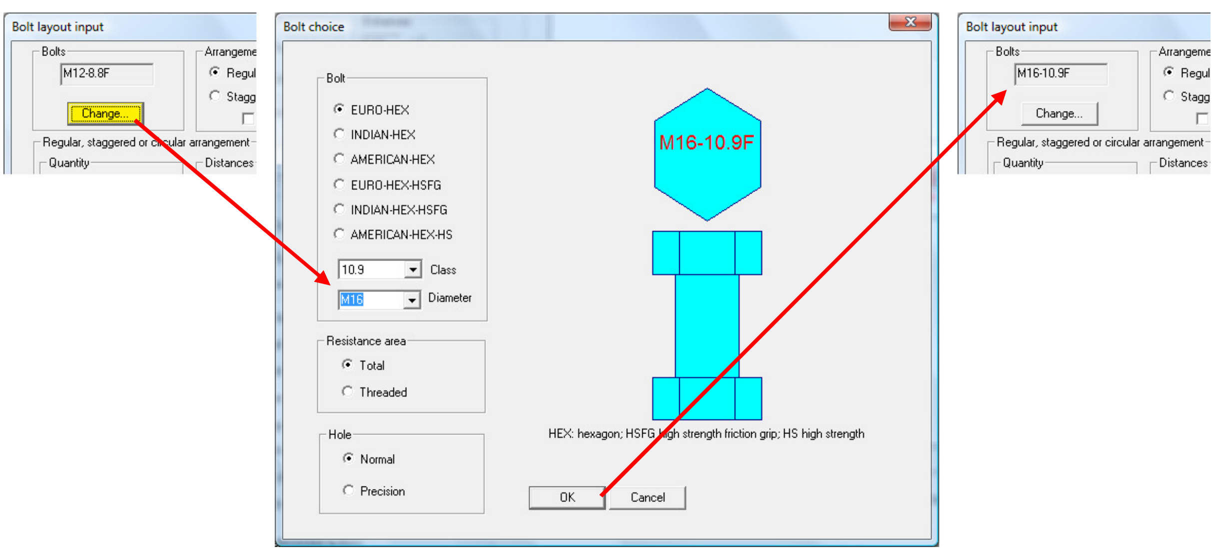
2.5.2 Addition of bolts

Unselect all the components, then use **Renode – Components – Add bolt layout** and click in the scene the external face of the first added plate.

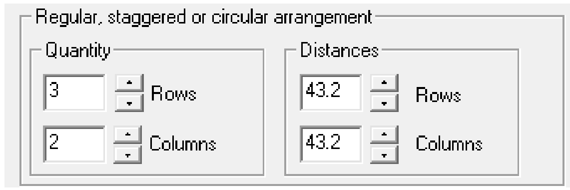




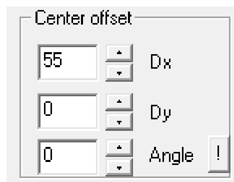
Click the "Change" button in top left part of the dialog box, then choose 10.9 class and M16 diameter in the new dialog box. Press OK to return on previous dialog.



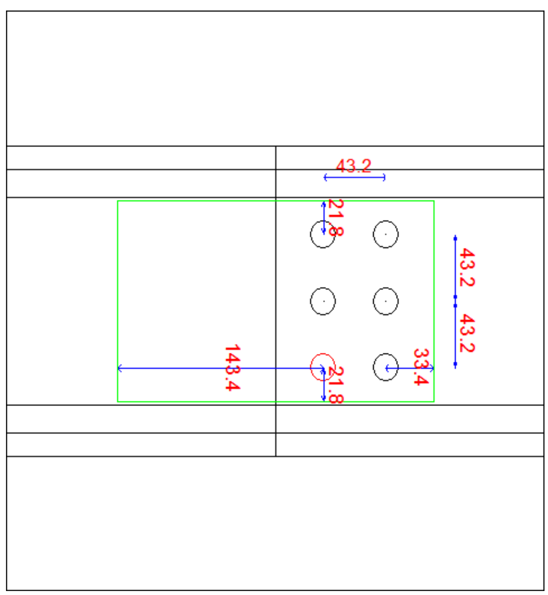
Keeping the regular arrangement, define 3 rows and 2 columns using the proper arrows. Since we have chosen a diameter greater than the previous one, distances have been automatically settled to minimum distances between M16 according to EN1993-1-8.



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



The result is shown in the following image. Use zoom control to have a closer view; you can **Copy** the figure with the proper button, show/hide bolts **hexagons** and **fill** faces with different colours.

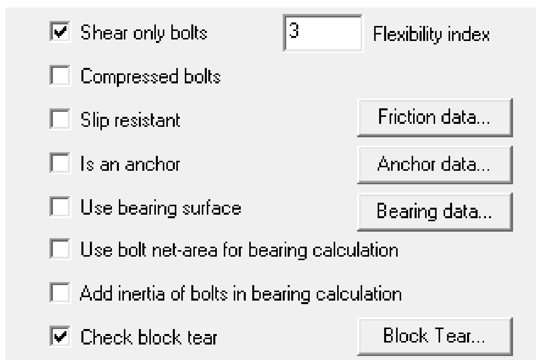


Distances between bolts and between bolts and current face sides are displayed. Use Face buttons to change current face (the green one).

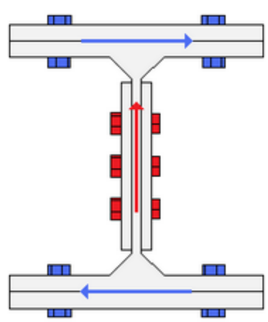
Note that if bolts distances or distances from sides are too small for chosen standard, a warning message will appear then you try to insert the layout in the scene.

Now define the following options:

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

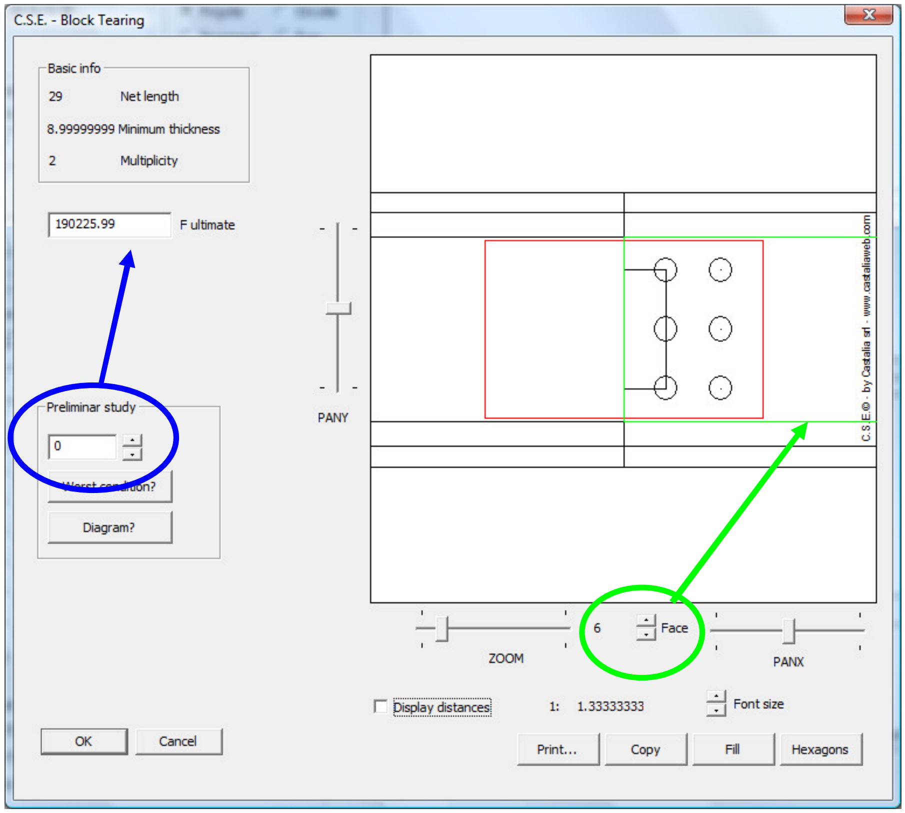


Using shear only bolts in the renode we are building, web bolts contribution in carrying a shear parallel to the flanges will be smaller, as flanges bolts contribution in carrying a shear parallel to the web.

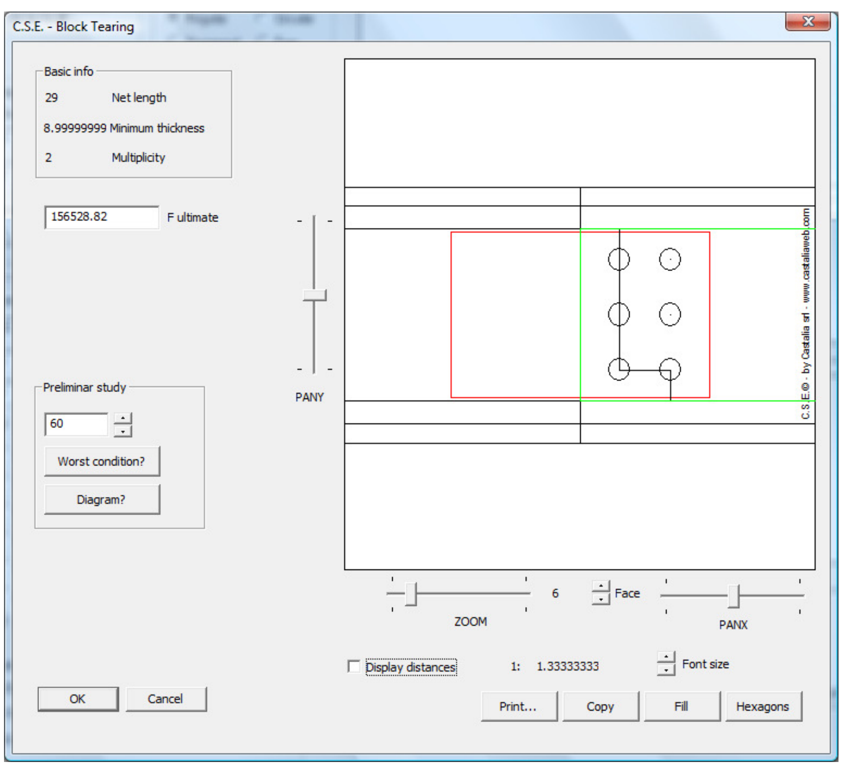


Flexibility index is use to modify bolt layout stiffness; using an index greater than 1, bolt layout will be more flexible and forces will be carried by other components. This aspect will be explained in the appendix.

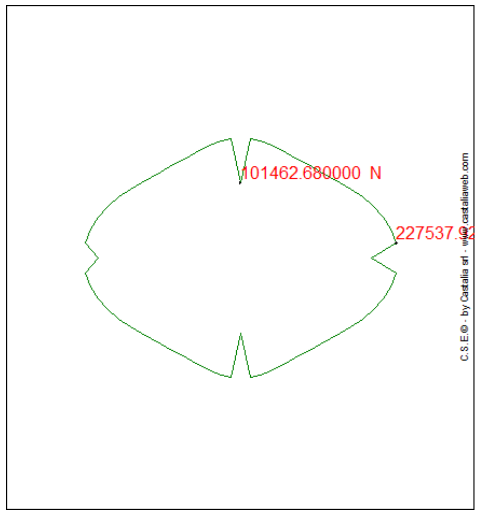
Click "Block tear" button to see a dialog box with information about block tear failure paths of the faces involved by bolts. Use the arrows to select a face (green): in function of settled force direction (blue circle, 0 = horizontal force), failure path of current face is displayed in the image and ultimate force of the plate is given.



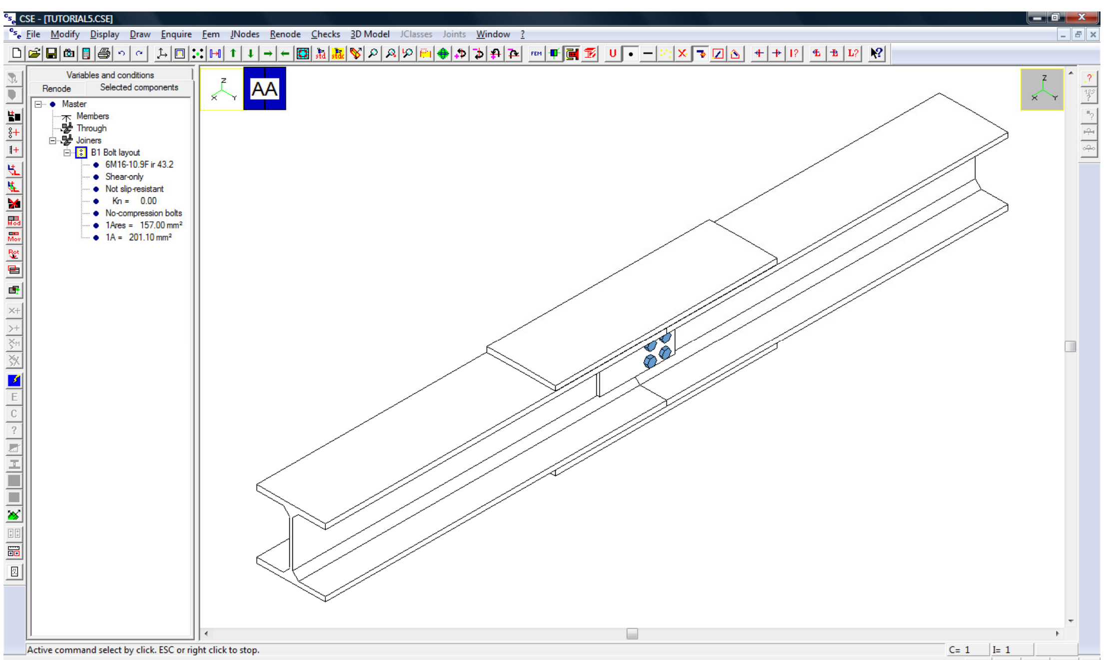
Use arrows to change force direction: image and F ultimate will be updated in real time.



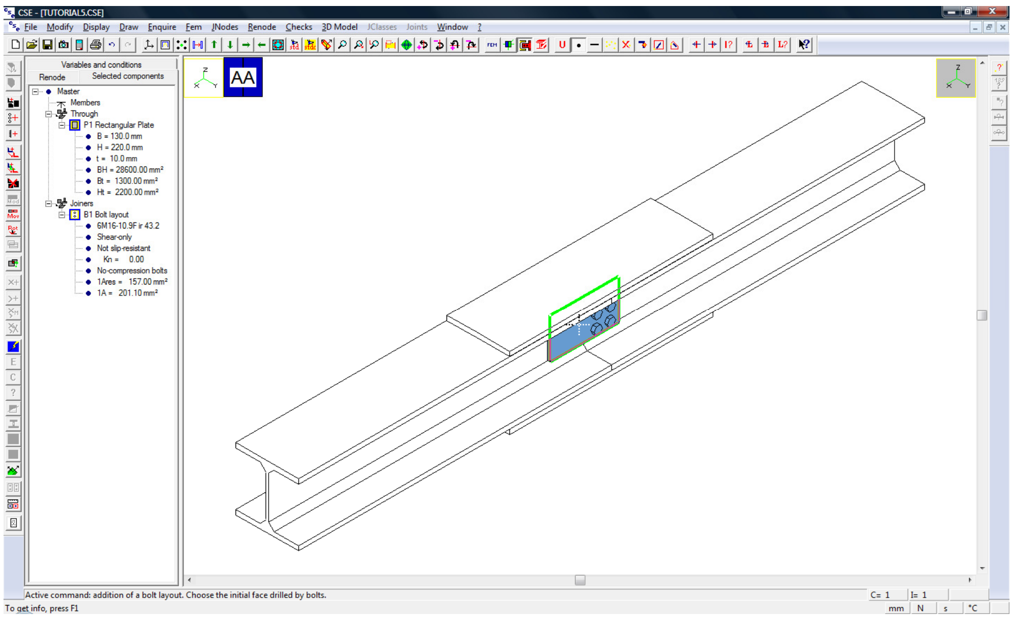
Assuming that all the bolts will carry the same load, the program predicts a maximum shear force equal to 156528,8N for a force with 60 degrees inclination. If you press the Diagram button you get what follows, i.e. a diagram plotting the maximum force vs the angle of the resultant. If you ideally draw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



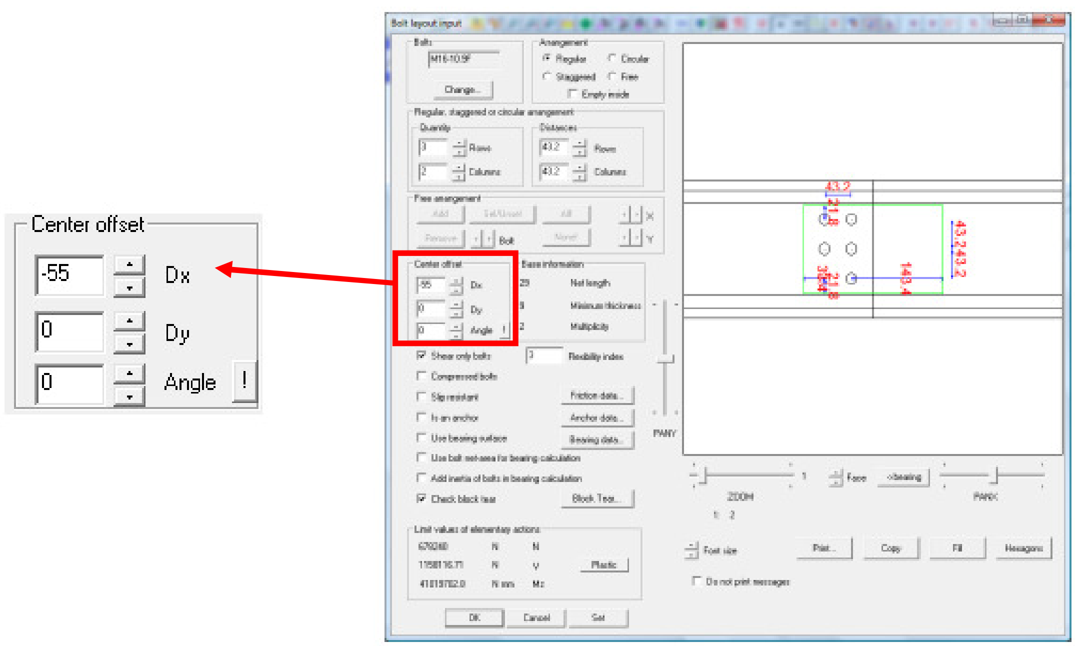
Use **Copy** button to copy the image. Press OK to exit from block tear dialog box, then press OK in main dialog box to insert the layout in the scene.



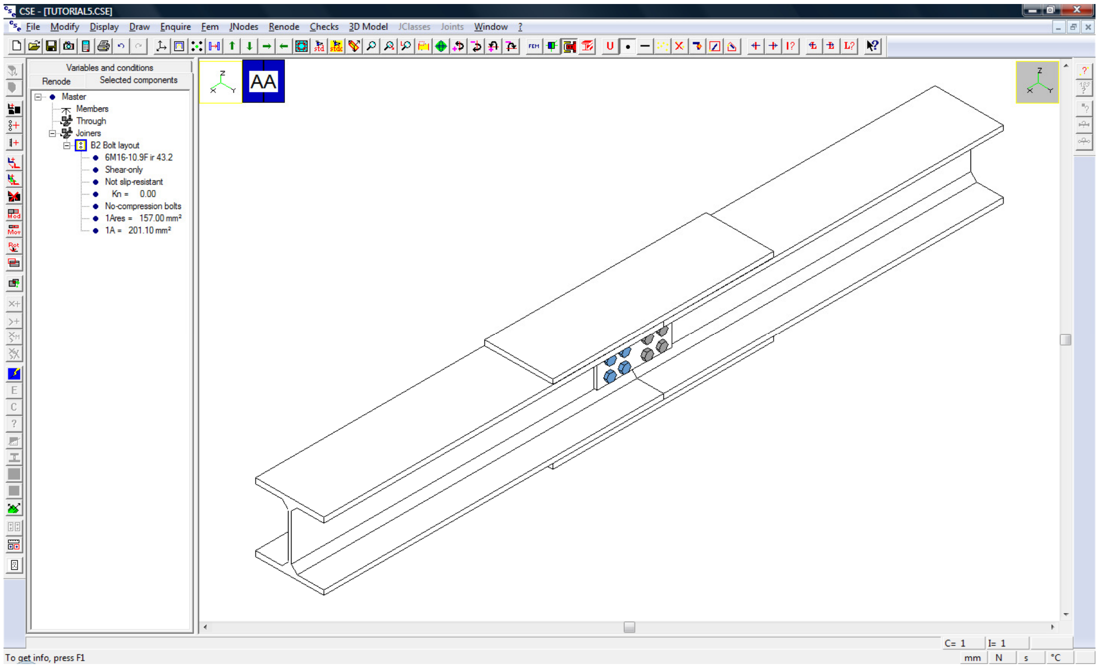
To add the other layout on the web, we muse unselect first bolt layout or select the plate, because in bolt layout addition, if there are selected components only their faces can be clicked. Select the plate, use **Add bolt layout** and click plate external face again.



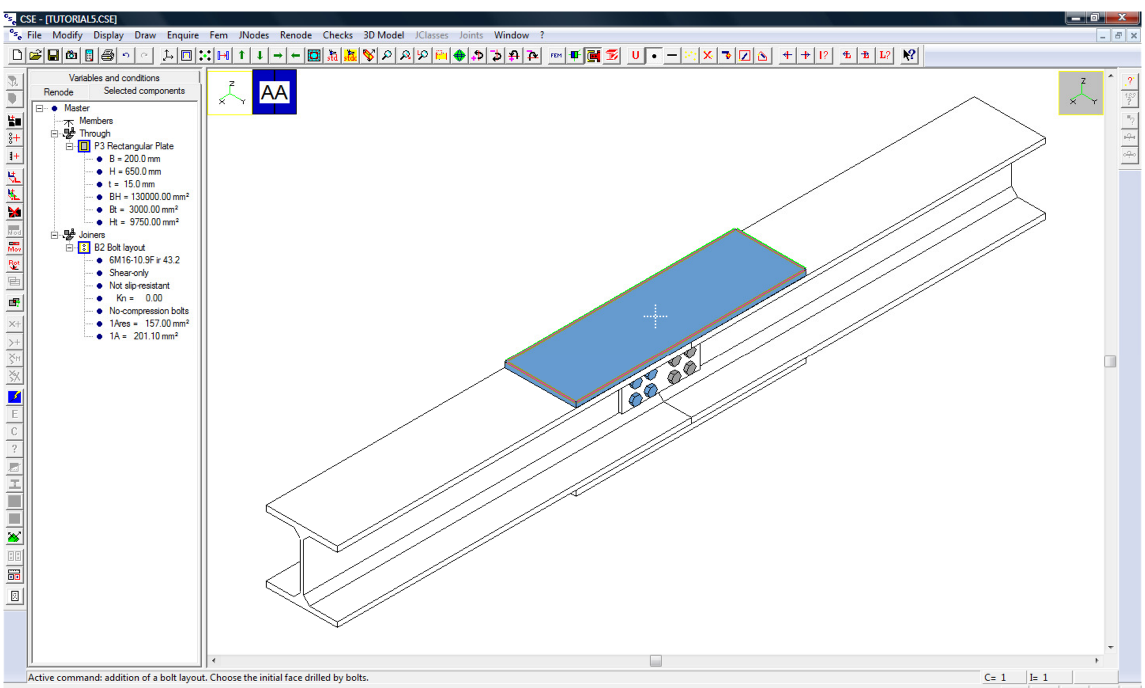
Type a '-' before 55 in Dx box, keep all other parameters and press OK.



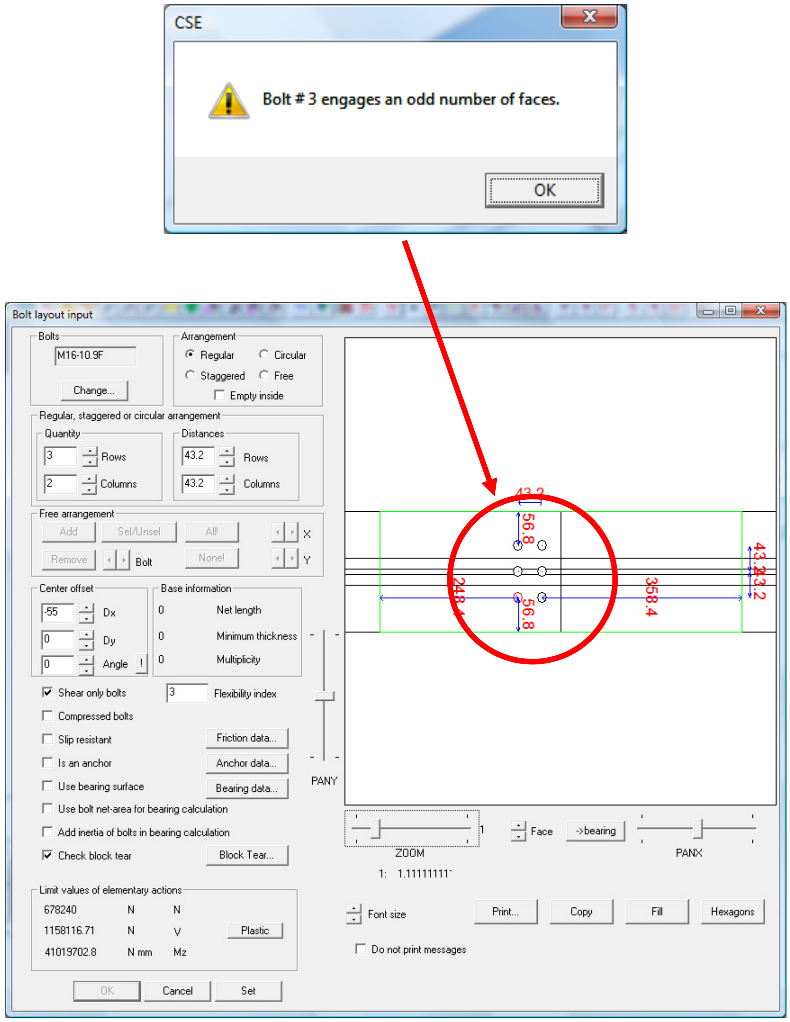
Second bolt layout has been added.



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

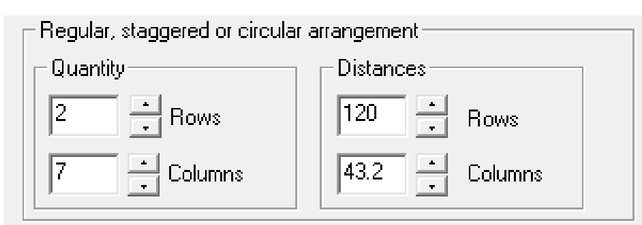


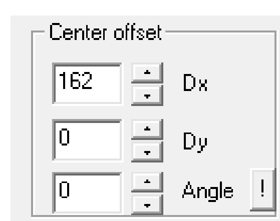
The following message will appear, but it is not a problem because initial layout settings are those of last defined layout, that can not apply to the layout we are going to insert. Press OK.



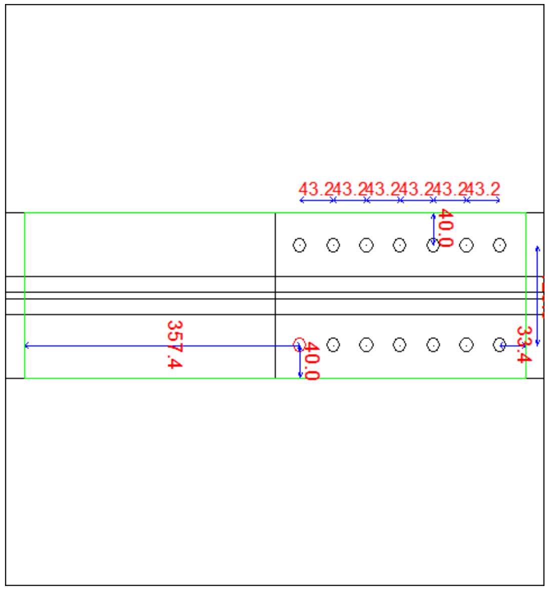
Two bolts are in correspondence of member web, but it is obvious that previous layout parameters must be changed.

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

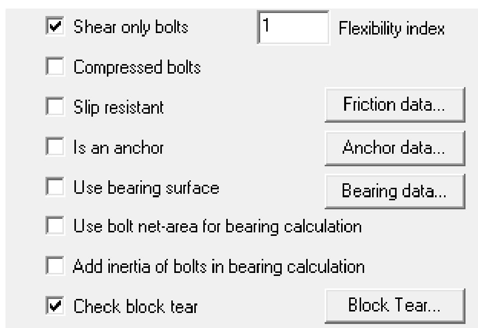




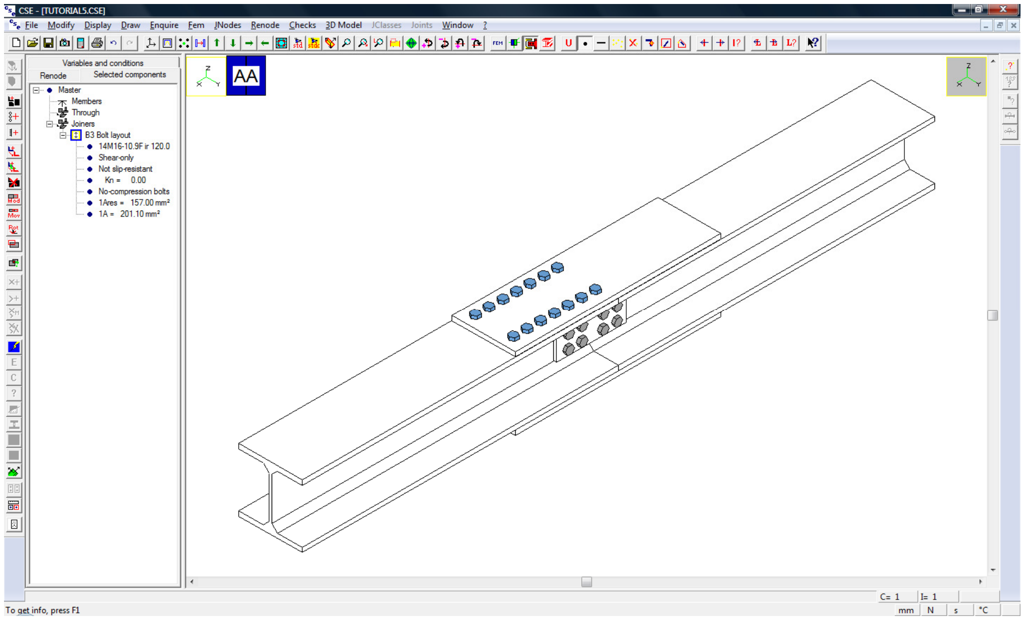
Layout will appear as follows:



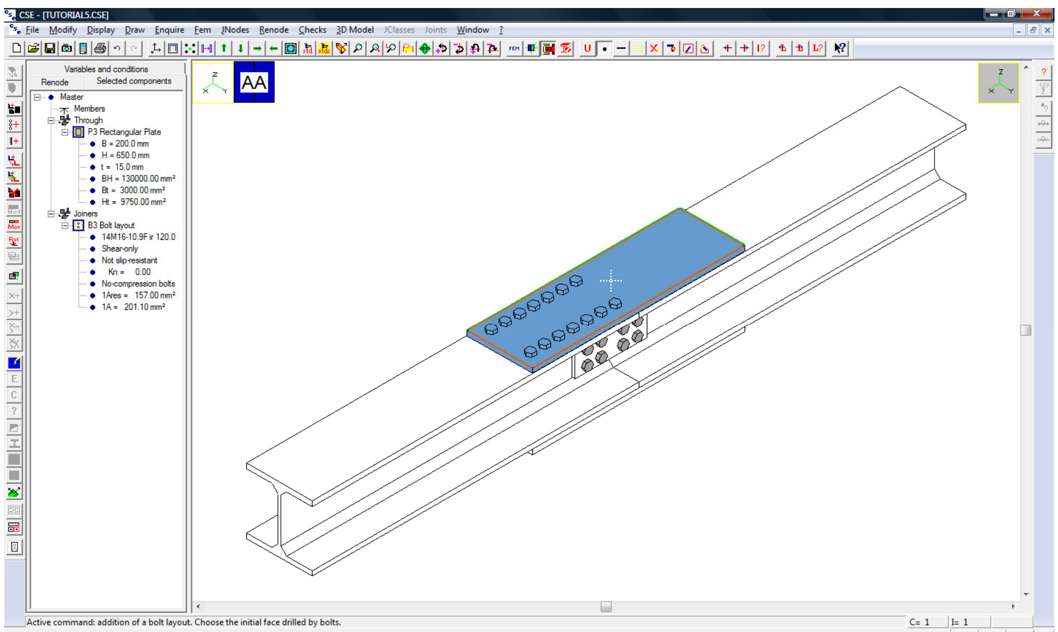
Keep the same ticks of previous layout, but type 1 as flexibility index:



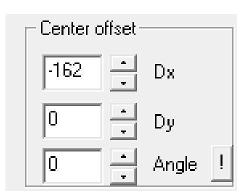
Press OK to insert the layout.

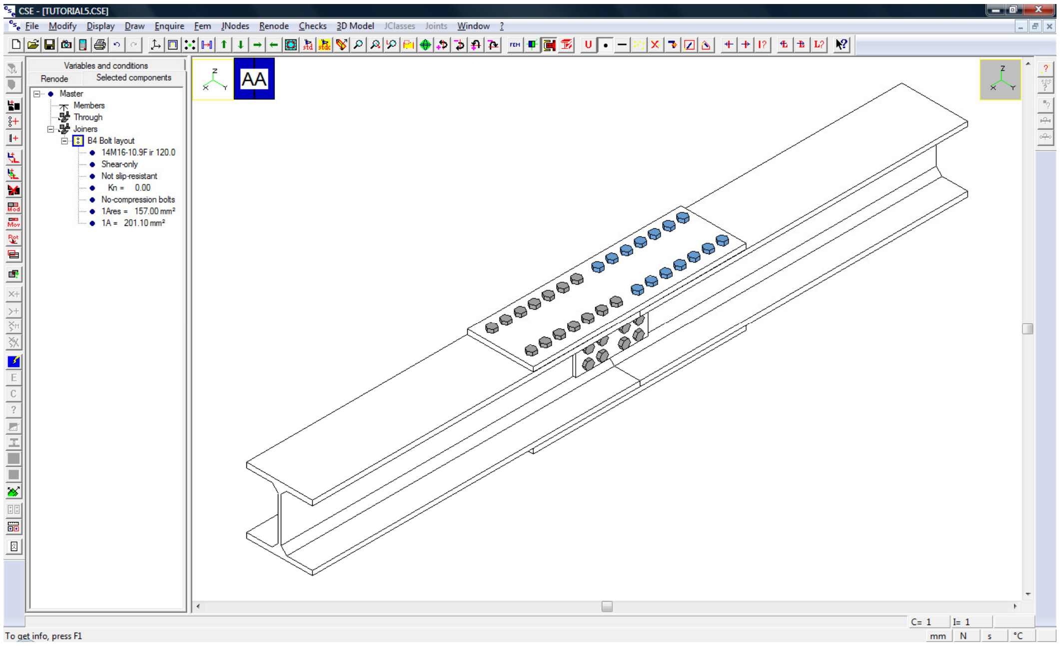


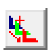
Select upper plate, then add a bolt layout  and click the same face as before.

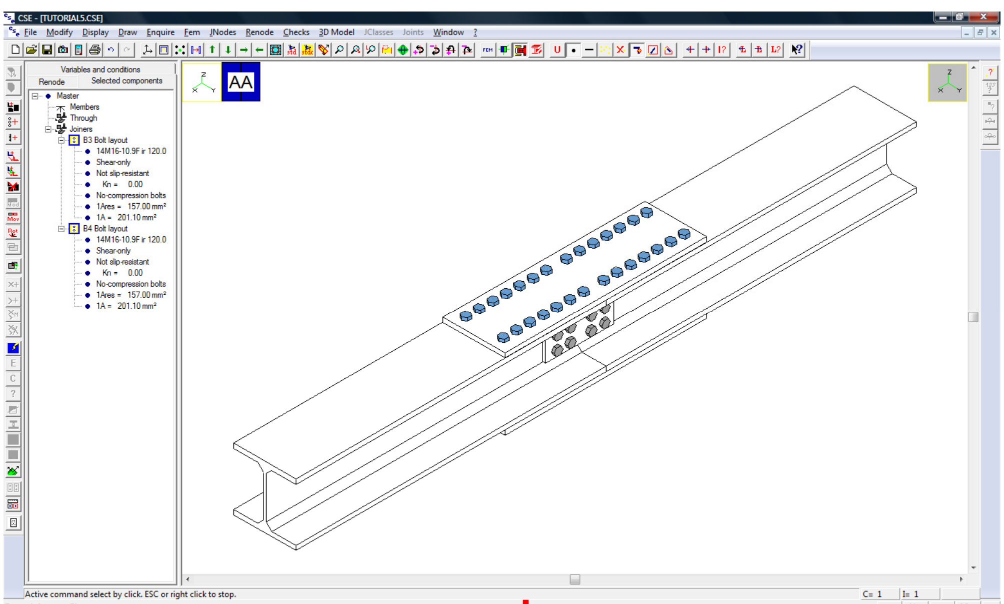


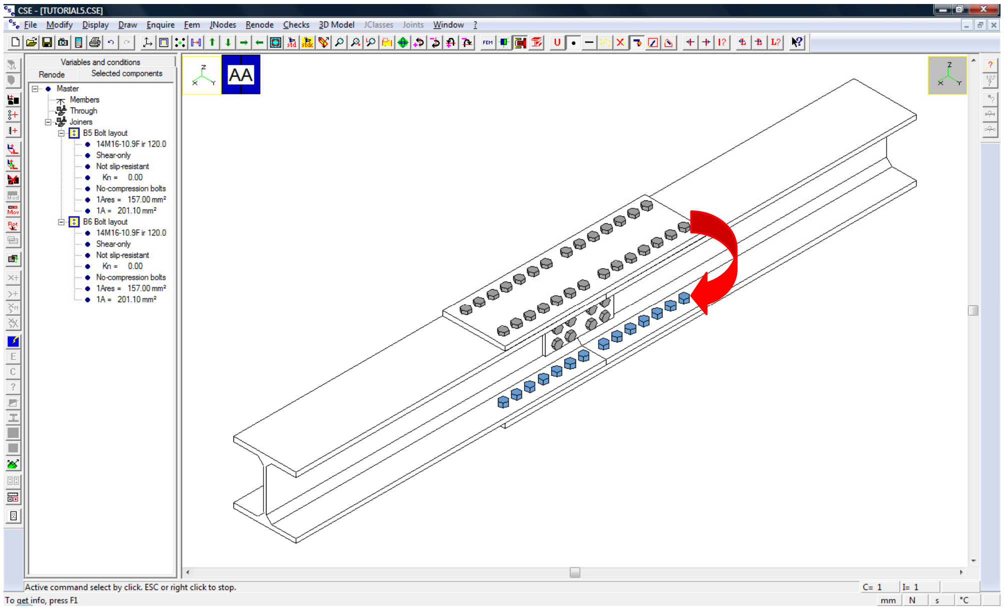
Type a ‘-’ in the dialog, keep all the other parameters and press OK.





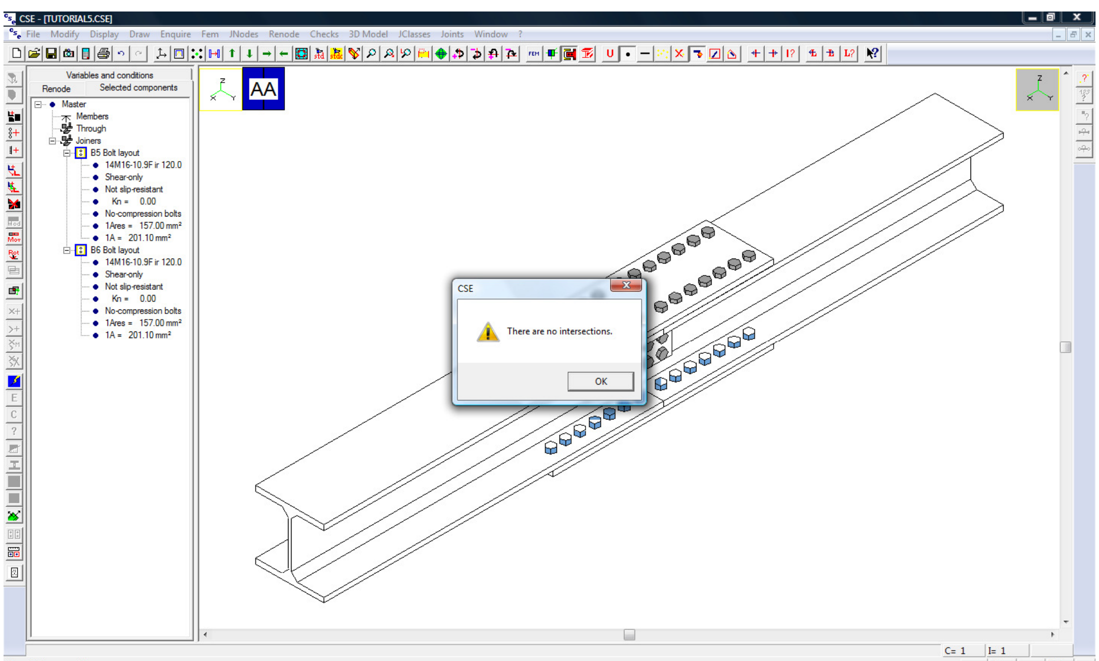
Two get lower plate and bolt layouts, select upper components, then use **Renode – Components – Recopy**  command. Last defined copy vector was the one used to copy upper plate under lower flanges, and now we need the same vector: Recopy command uses last defined vector.



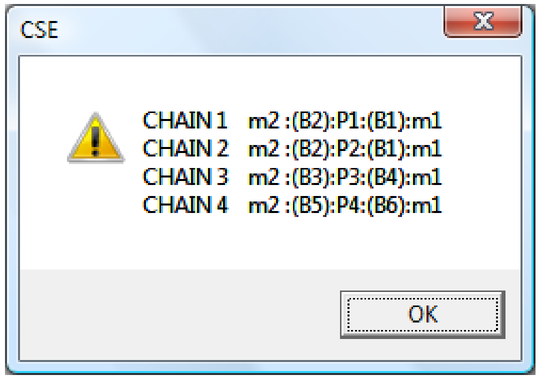


2.5.3 Overlaps and coherence controls

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

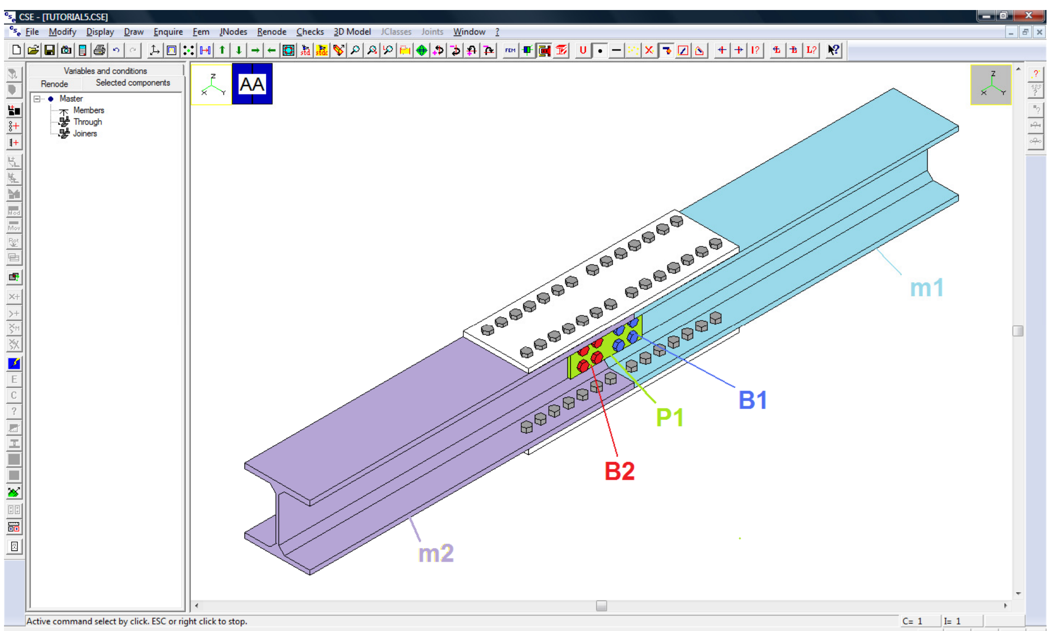


Use Renode – **Check coherence** to check the correct connection between the components.



Chains are listed: it means that all the components are properly connected. In this case we have 4 different paths from m2 member (the slave) to m1 (the master). For example, forces can follow this path (chain 1):

member m2 → bolt layout B2 → plate P1 → bolt layout B1 → member m2



2.6 STEP 6: SETTING CHECK SETTINGS

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

The standard has already been chosen, Eurocode 3. This is a limit state standard.

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

Since we have not a true fem model we must choose as "Internal actions computing mode", "Elastic limits", "Plastic limits", “Defined Values” or combinations imported “From table”. Choose elastic for instance. Then specify the multiplier of internal actions for each component. You can specify the number you want. This will generate 24 notional load combinations. 6 for positive internal forces, one by one. Six more for negative internal forces. Next 12 will be a mix of N, 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.
Use the arrows to select m2, then type the desired multipliers. For example:

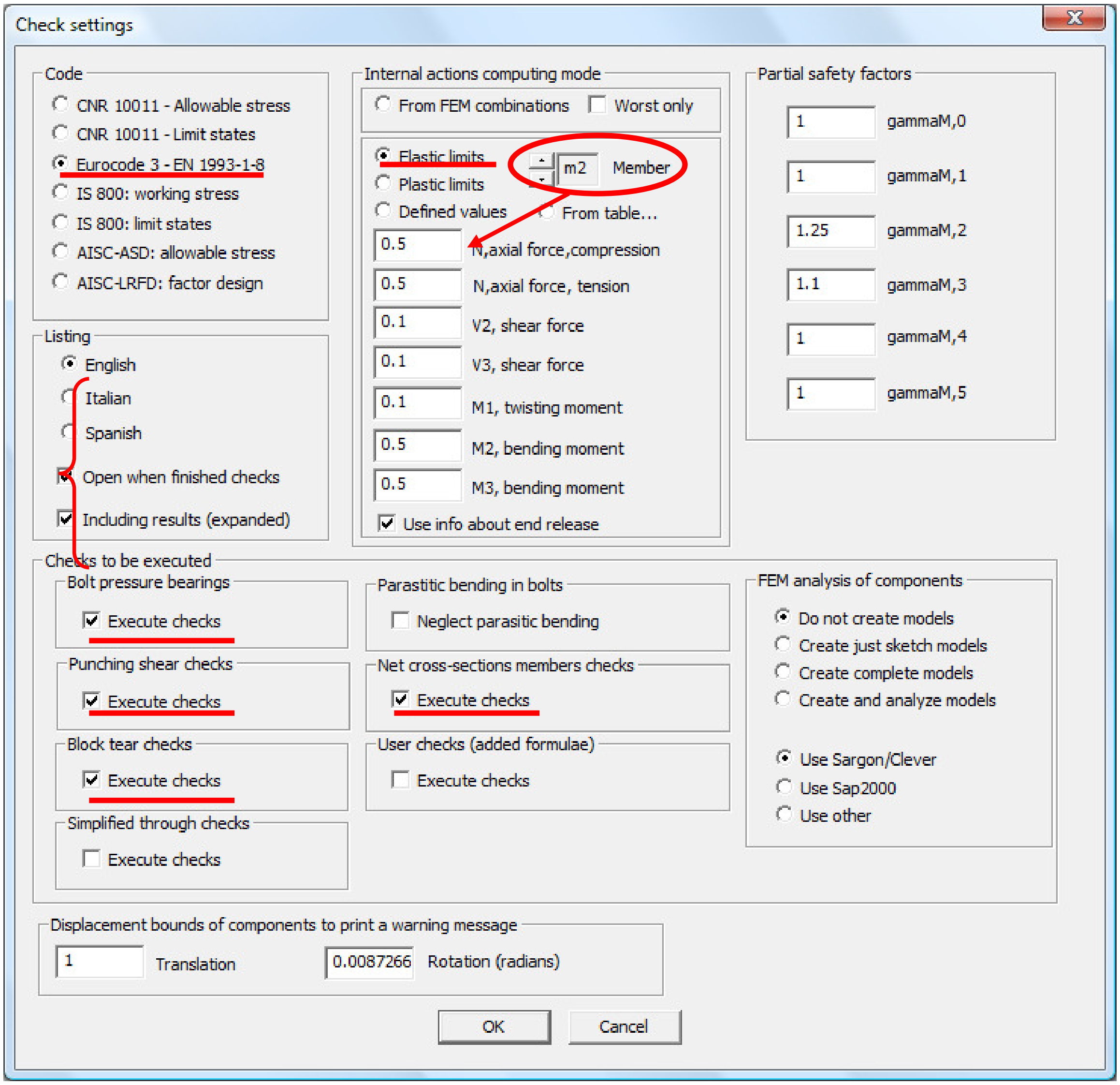
- 0,5 for compression, tension and bending moments
- 0,1 for shear 2, shear 3 and for torsion (we want to compute the renode mainly in bending and azial force)

Keep default “Partial safety factors”.

In “Checks to be executed” section tick the following options

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

Keep default “Displacements bounds” for displacement checks.

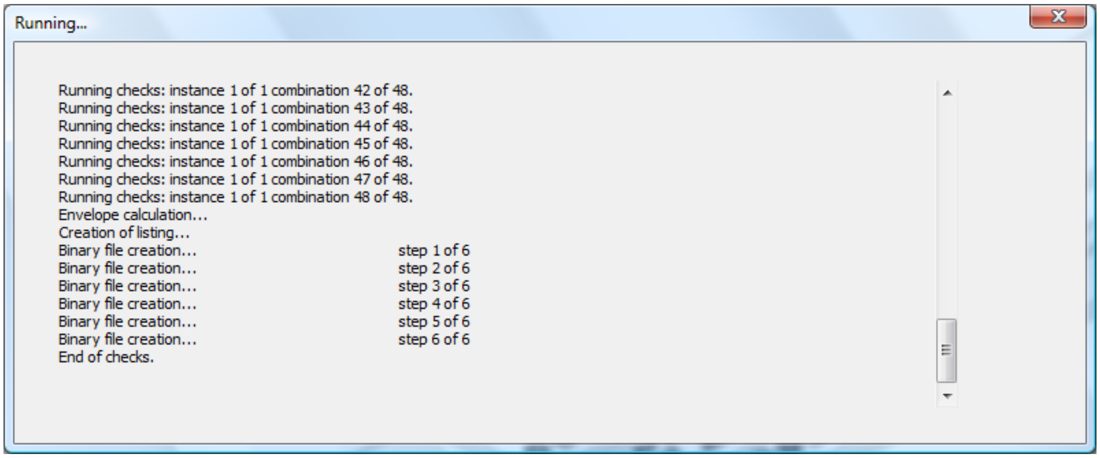


Press **OK** to save settings.

Notes: "simplified through checks" do not cover plates: they can be checked via fem models, but we are not going to use them now; "user checks" have not been defined here.

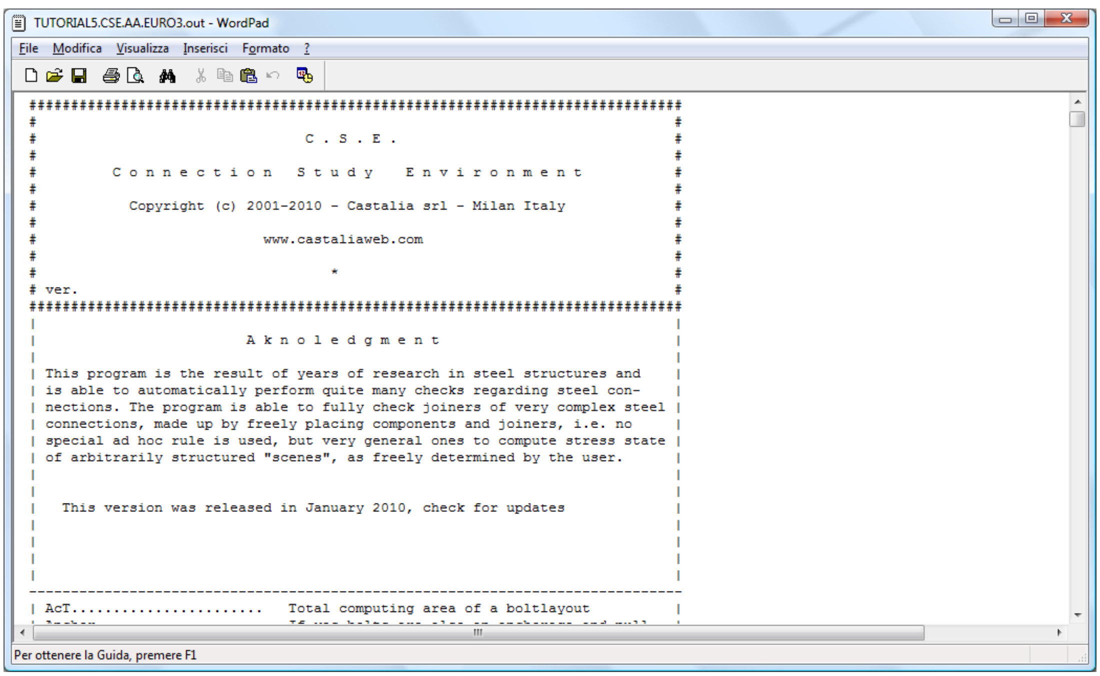
2.7 STEP 7: EXECUTING CHECKS

To execute the checks press the following button in the left toolbar (**Checks – Check renode**):



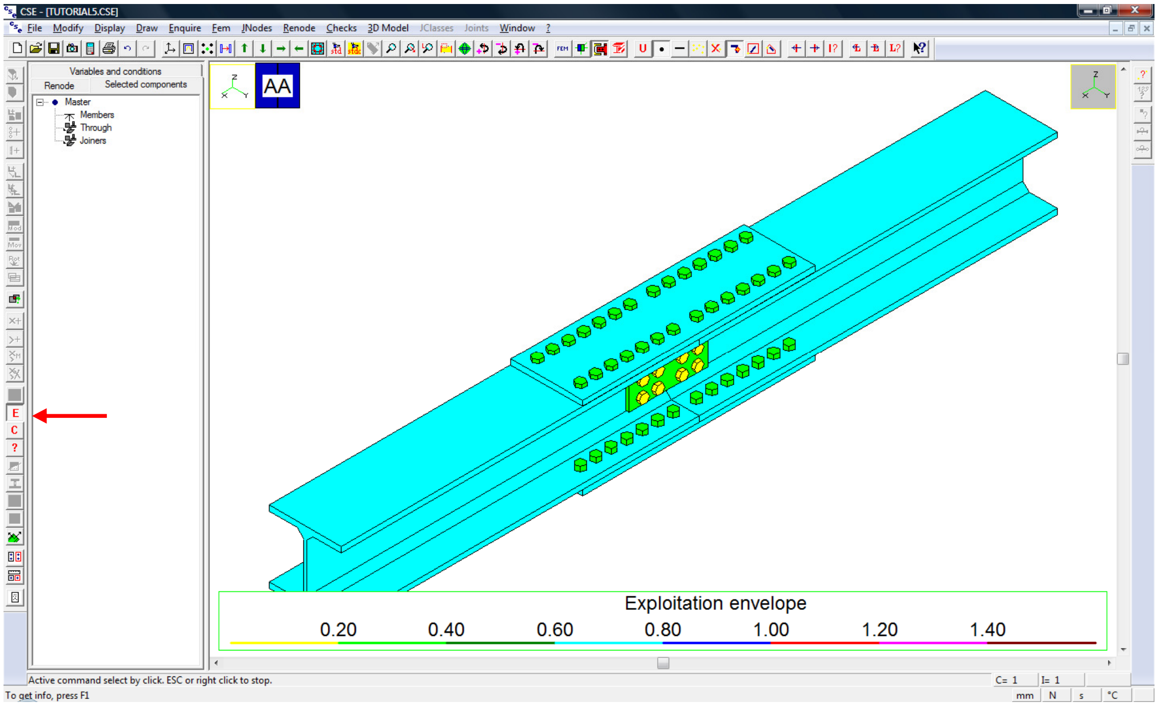
The window above will appear, it is a log window explaining what happens. At the end just close it by clicking over the red-background cross.

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

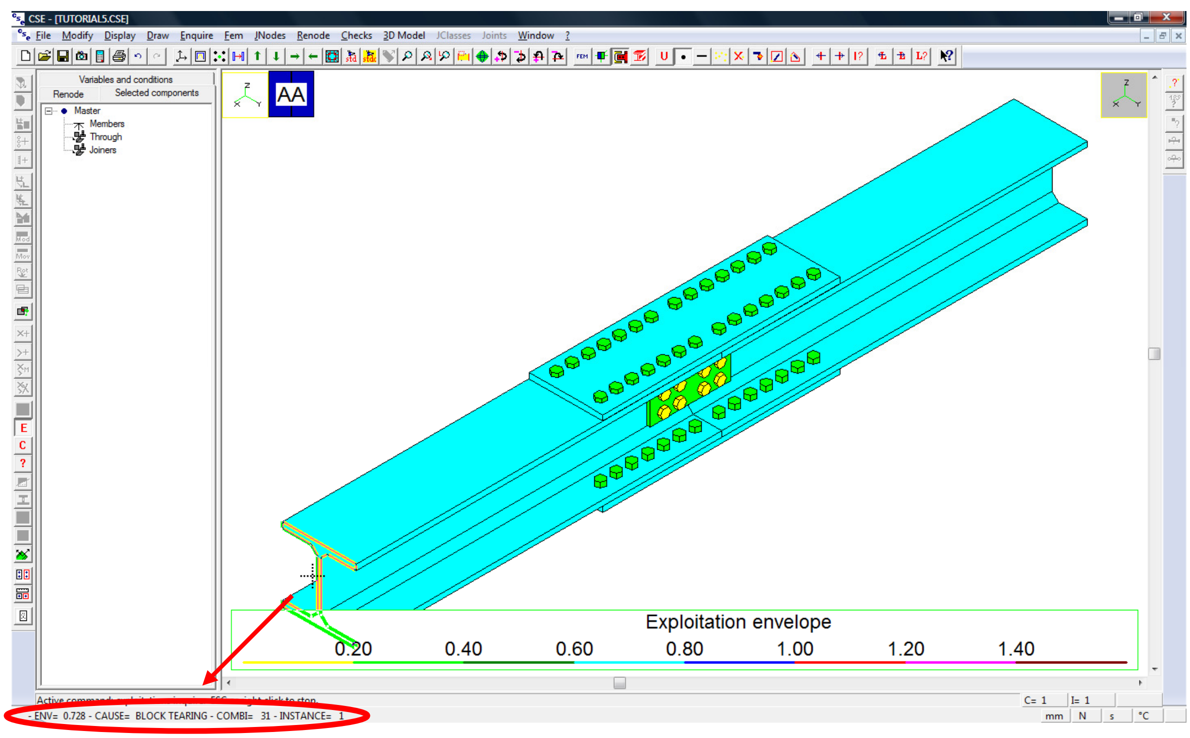


2.8 STEP 8: EXAMINING RESULTS

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

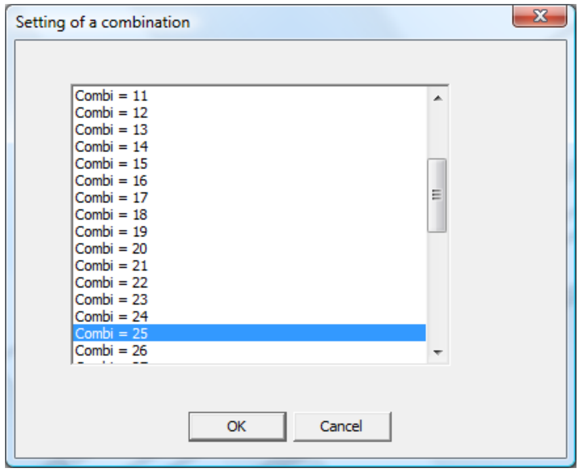


While the envelope is displayed, use **Checks – Enquire** ? to know components maximum exploitation, its cause and combination. Move mouse pointer near a face of the component you want to inquire and read information in left bottom part.



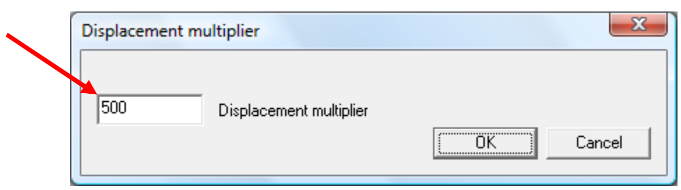
You can inquire bolts, plate or members maximum exploitation. If you use **Checks – Current results** , displayed and enquired exploitations refer to current combination.

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

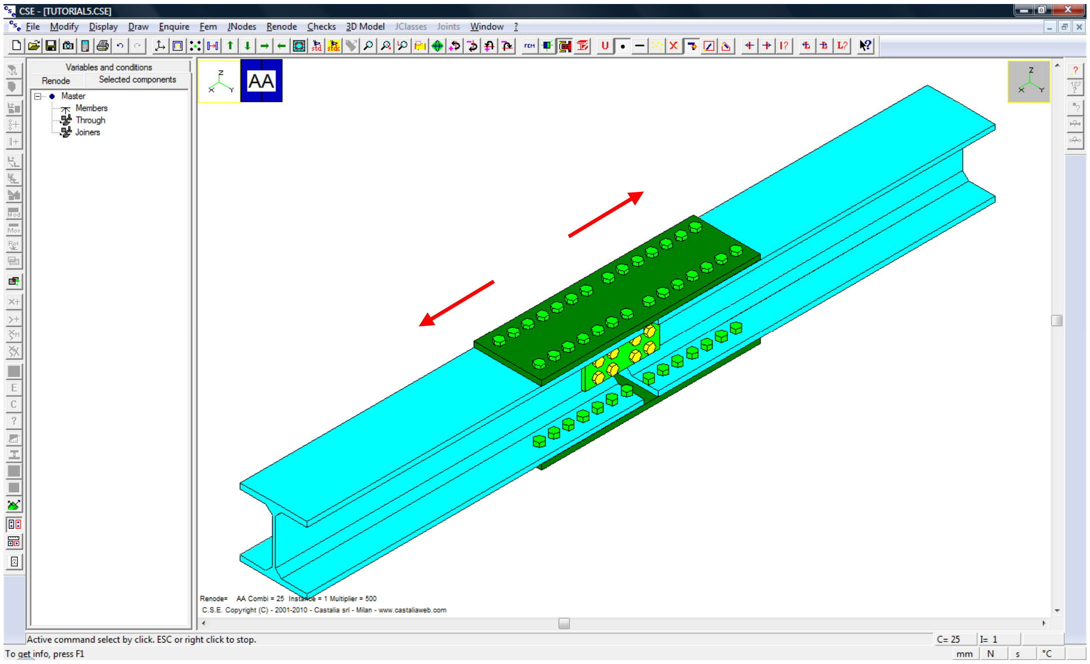


Use **Checks –Displaced** in left toolbar to display the displaced view of the renode in current combination.

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

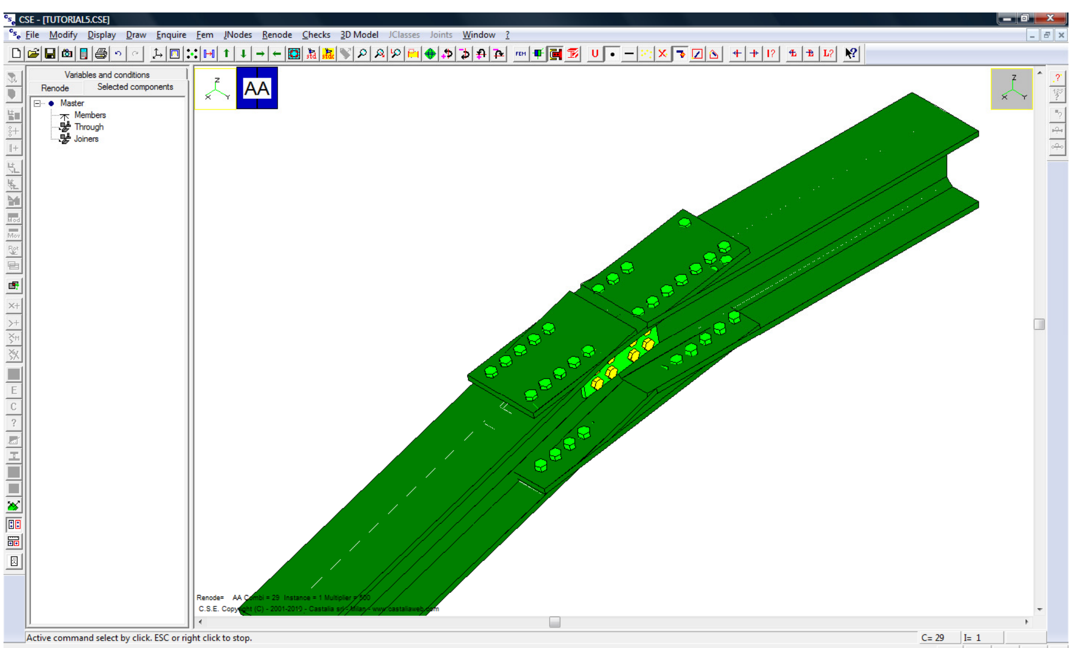


When the displaced view is shown, components are coloured with the range colour corresponding to their exploitation in current combination.



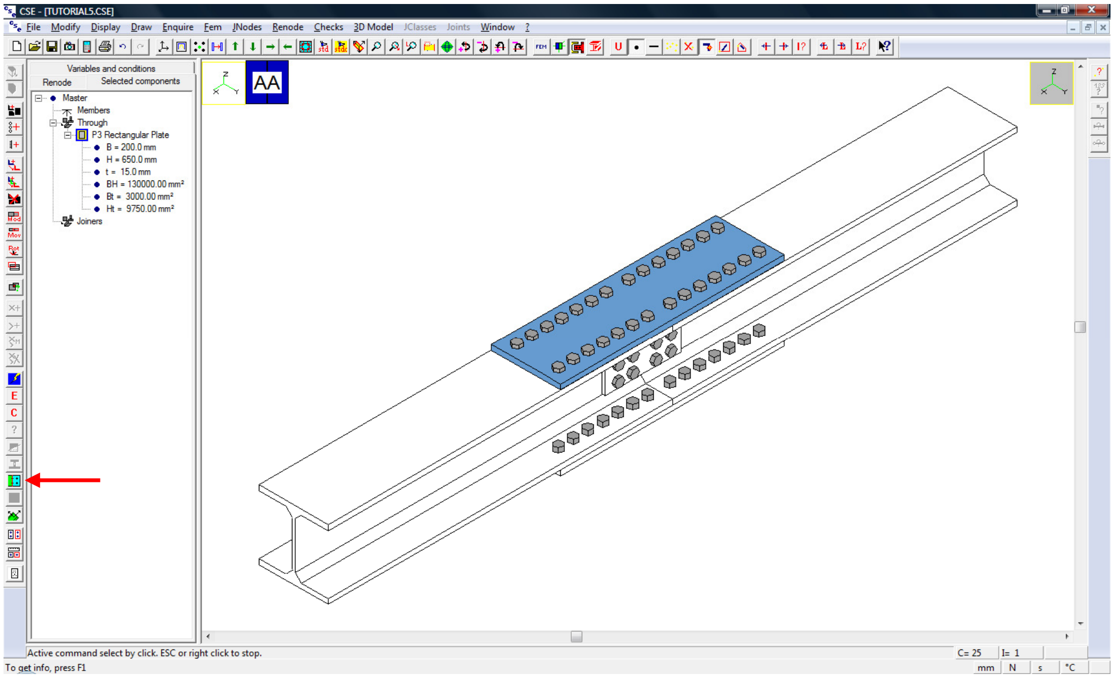
Switch to following combinations (**Checks - Combi next**) to see displaced in combination 26 (T_2^+), 27 (T_3^+), 28 (M_1^+) and so on.

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

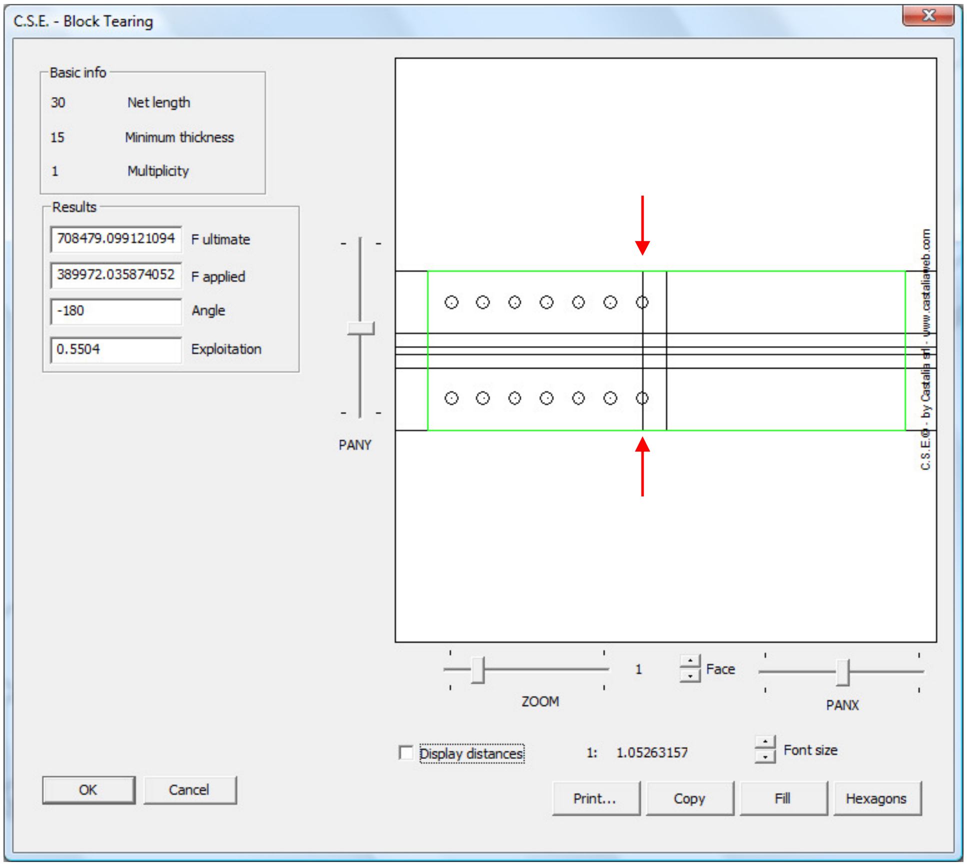


Un-press Displaced and switch back to combination 25 (tension) using or .

Now select P3 and click **Checks – Block tear results** .



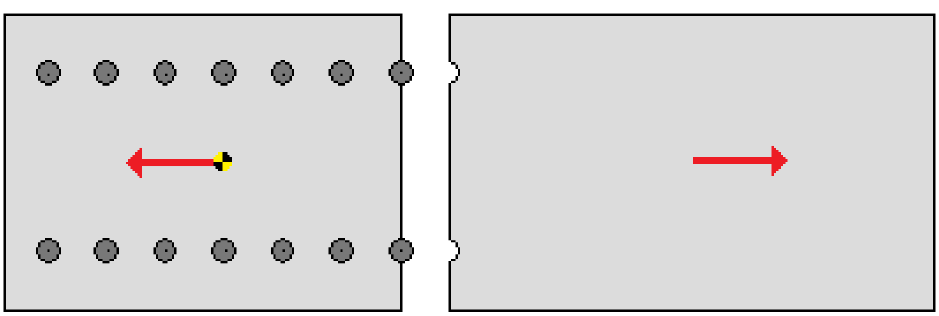
The following dialog box will appear:



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

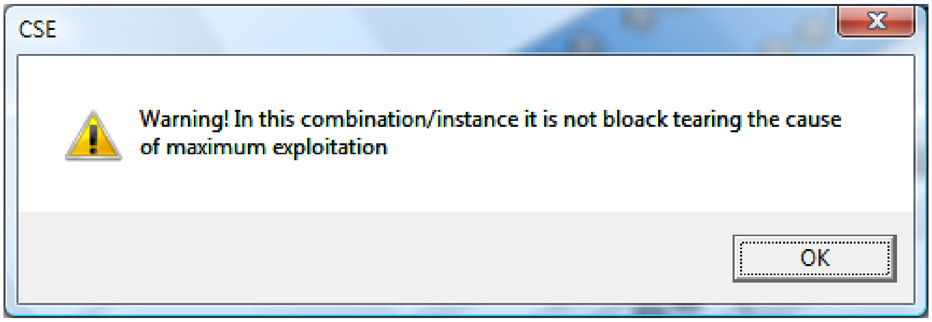
Remove the tick from “Display distances” to hide bolts distances.

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

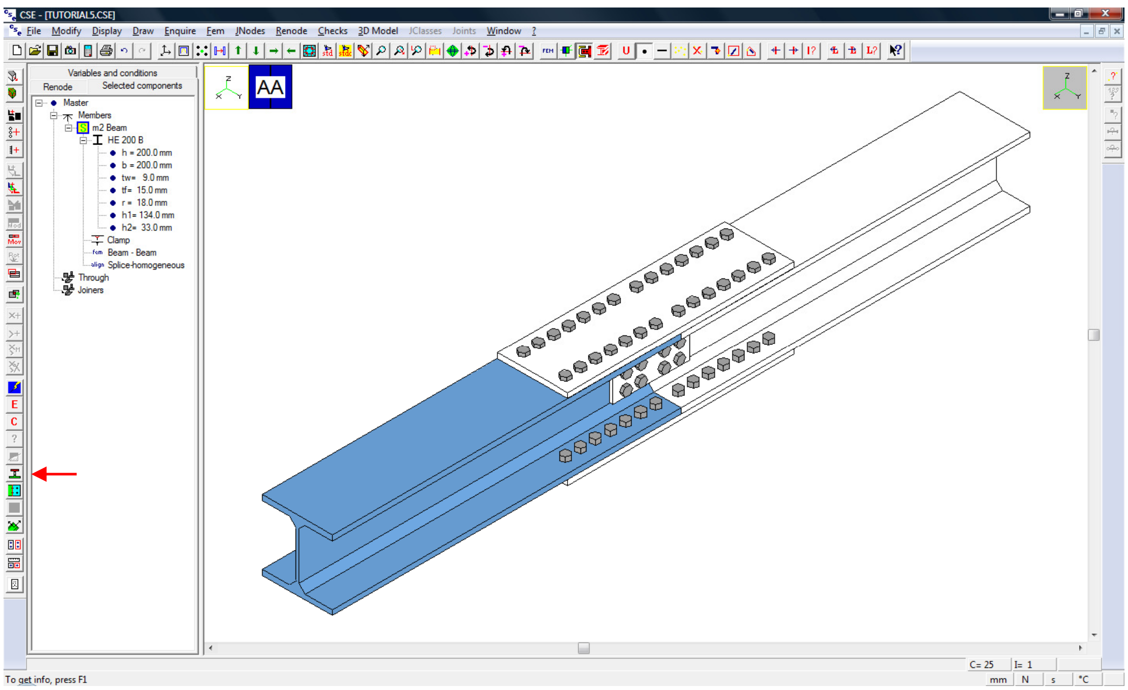


Exit from the dialog box clicking OK or cancel.

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

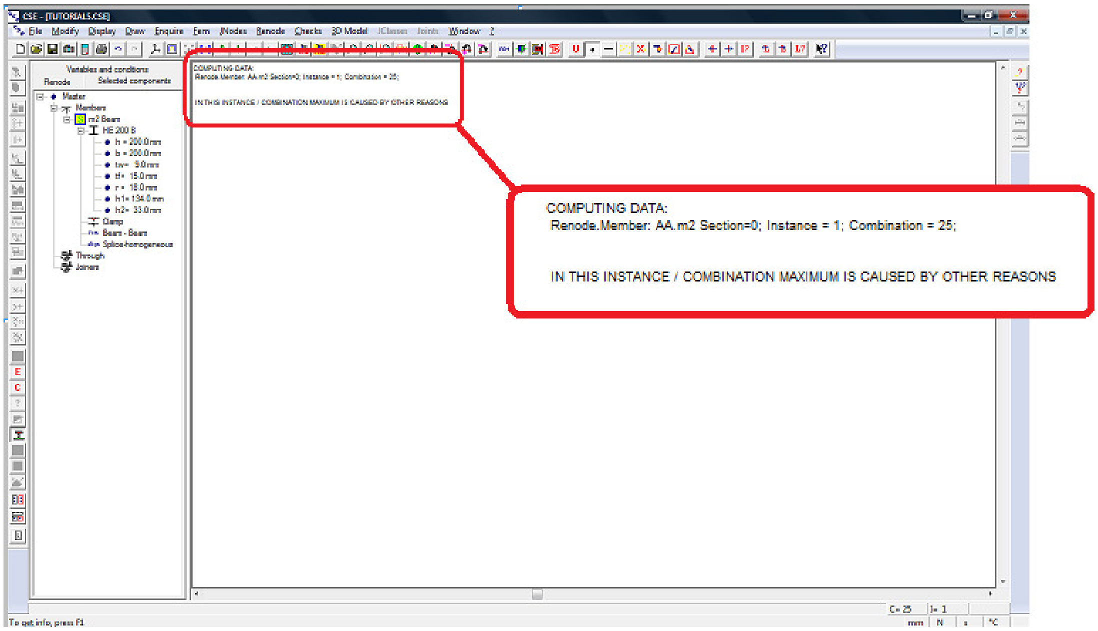


Now un-select P3 and select m2, then use **Checks – Display net sections results** .

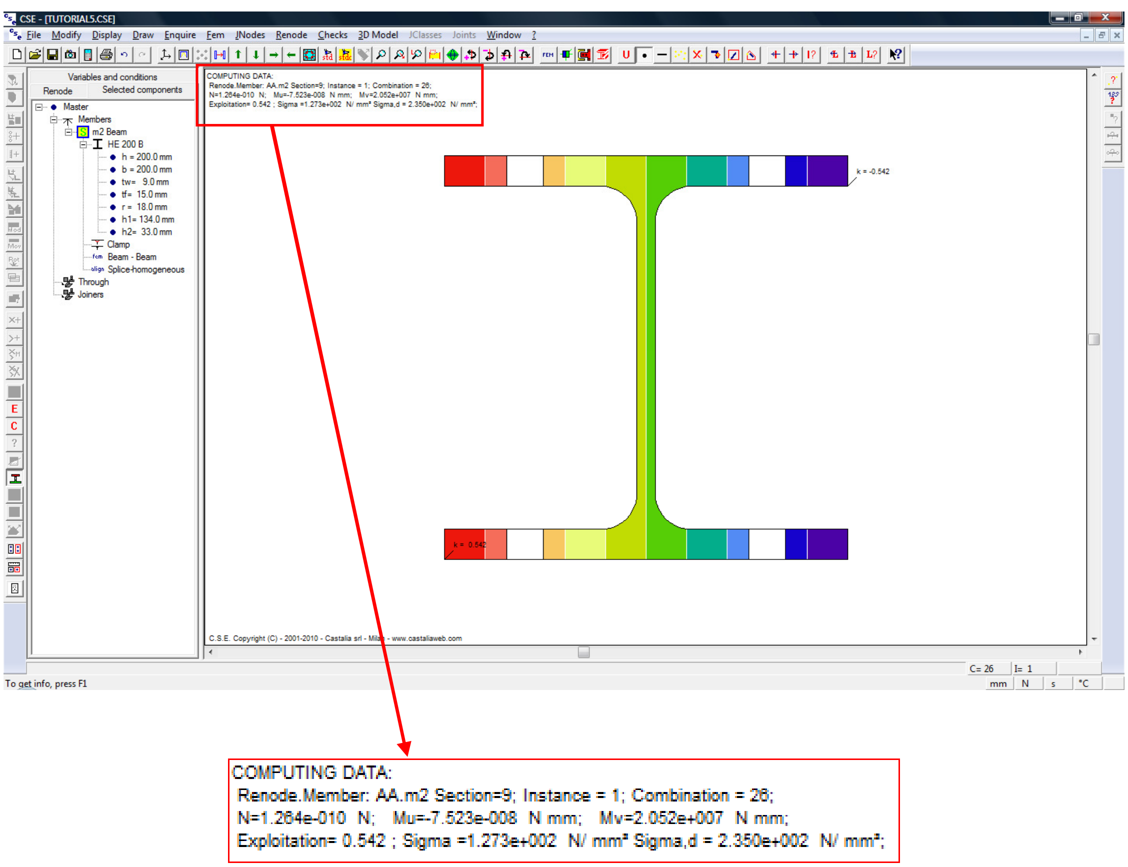


When you execute the command, renode is hidden in the graphic view and net section results of selected member in current combination are displayed.

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



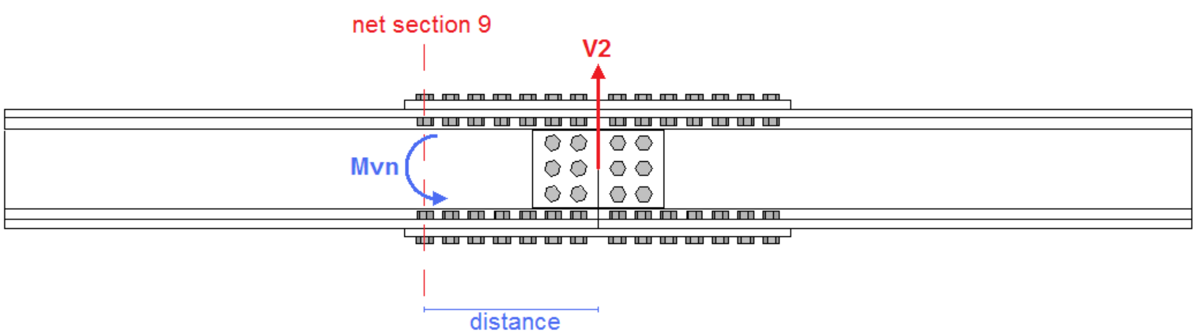
Switch to combination 26 (). Here we have net sections check as maximum exploitation, so results are displayed.



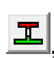
In the “Computing data” section the following are reported:

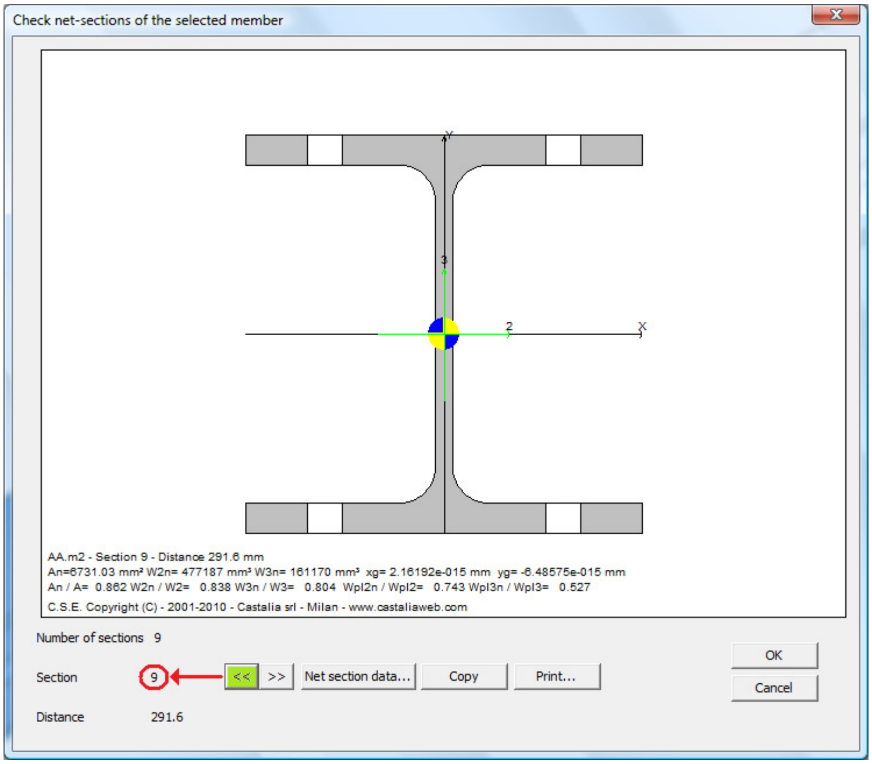
- Renode mark and member name
- Net section with maximum exploitation
- (Instance) and combination (here we have one instance only)
- Applied axial force and bending moments
- Exploitation
- Applied and allowable normal stress

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



Follow next steps to know that distance.

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



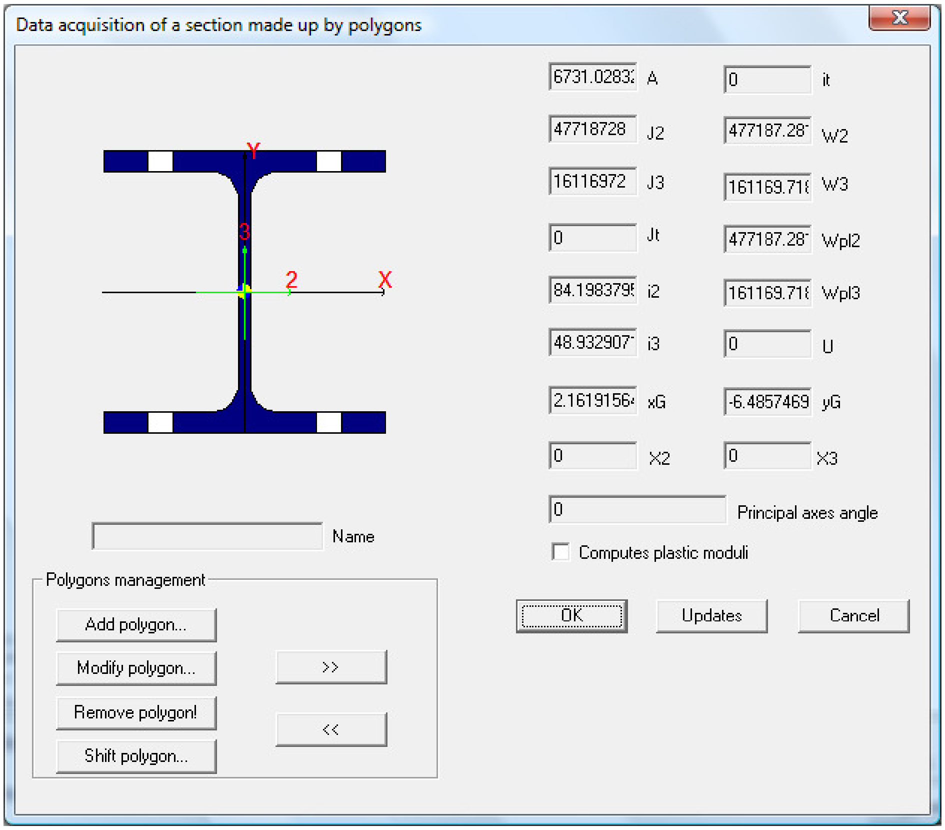
Use the arrows to select section 9, in which we have maximum exploitation in combination 25.

AA.m2 - Section 9 - Distance 291.6 mm
An=6731.03 mm² W2n= 477187 mm³ W3n= 161170 mm³ xg= 2.16192e-015 mm yg= -6.48575e-015 mm
An / A= 0.862 W2n / W2= 0.838 W3n / W3= 0.804 Wpl2n / Wpl2= 0.743 Wpl3n / Wpl3= 0.527

Given data are:

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

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


Data acquisition of a section made up by polygons
6731.0283: A
0 it
47718728 J2
477187.28 W2
16116972 J3
161169.71 W3
0 Jt
477187.28 Wpl2
84.198379 i2
161169.71 Wpl3
48.932907 i3
0 U
2.1619156 xG
-6.4857469 yG
0 X2
0 X3
0 Principal axes angle
Computes plastic moduli
Y
X
2
3
Name
Polygons management
Add polygon...
Modify polygon...
Remove polygon!
Shift polygon...
>>
<<
OK
Updates
Cancel

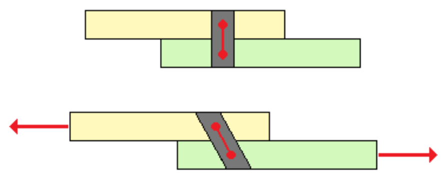
3 APPENDIX: FLEXIBILITY INDEX

3.1 WHAT FLEXIBILITY INDEX IS?

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

What we mean by… FLEXIBILITY INDEX OF A SHEAR-ONLY BOLT LAYOUT

Generally the translational or slipping stiffness (see next picture) of a bolt layout not using a bearing surface is directly proportional to the number *n* of bolts which make up the layout, the bolt radius *r* raised to the fourth power, and inversely proportional to the net length *l* of the bolt layout.



In CSE a parameter called the **flexibility index**, *f*, has been introduced in association with shear-only bolt layouts, for which the translational stiffness of the bolt layout is also inversely proportional to the cube of the flexibility index.

In summary, the translational stiffness of a bolt layout without a bearing surface is:

- directly proportional to n and r^4
- inversely proportional to l^3

If the bolt layout is shear-only, its translational stiffness is:

- directly proportional to n and r^4
- inversely proportional to l^3 and f^3

So if f is equal to 1, it does not affect the stiffness.

By using the flexibility index, the user can modify the stiffness of a shear-only bolt layout, in order to load some joiners more than others.

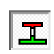
Let's take for instance a splice joint with bolted plates (web and flanges), under tension: in the plastic range, a plasticization of the web thickness will occur due to bolt pressure, so that the internal actions will be redistributed to the flange bolt layouts, which will continue to carry the load.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

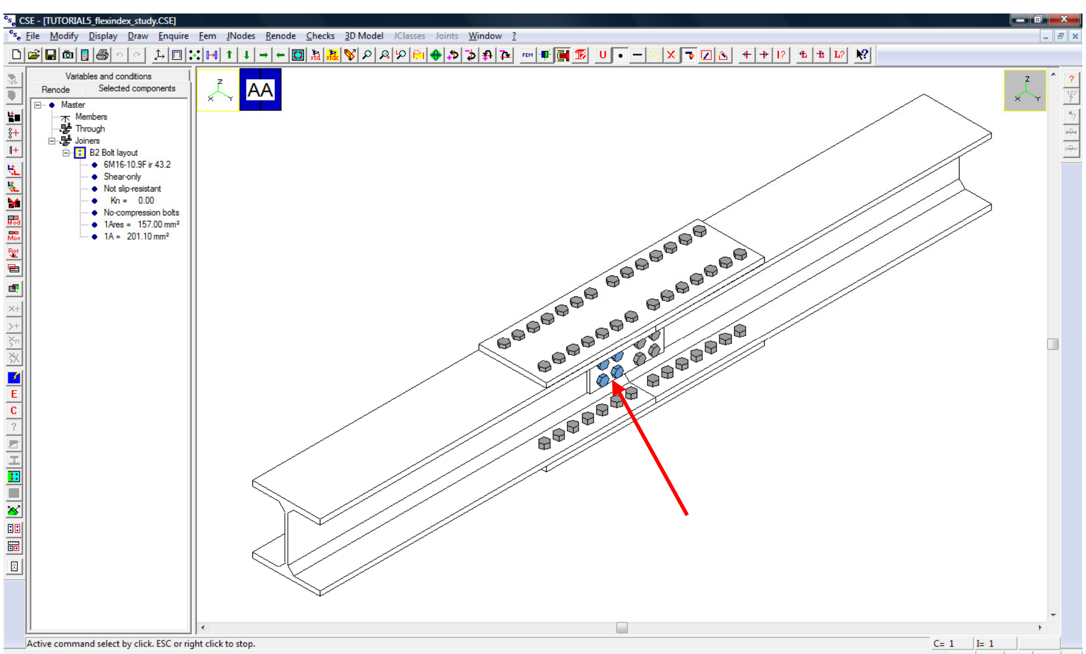
The definition of a flexibility index greater than 1 for web bolt layouts causes a reduction of their stiffness, and a greater amount of applied forces is carried by other bolt layouts (flanges bolt layouts has a flexibility index equal to 1).

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

3.2 MODIFY WEB BOLT LAYOUTS

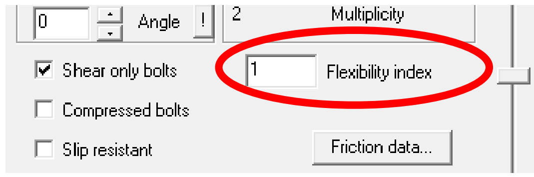
Let's see how flexibility index modifies forces distribution. First of all, create a copy of the previous model using **File – Save as**; name the new file "*TUTORIAL5_flexindex_study.CSE*", for example.

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



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

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

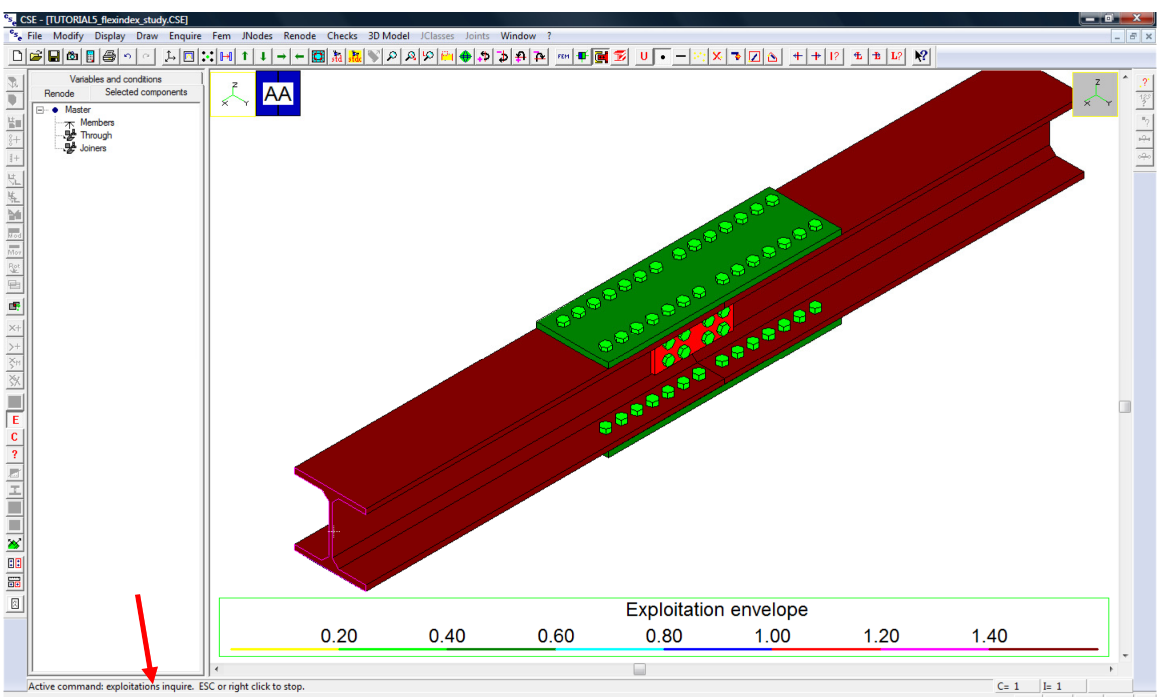


Now unselect currently selected bolt layout, select other web bolt layout and do the same as done before, in order to define a flexibility index equal to 1 for this layout too.

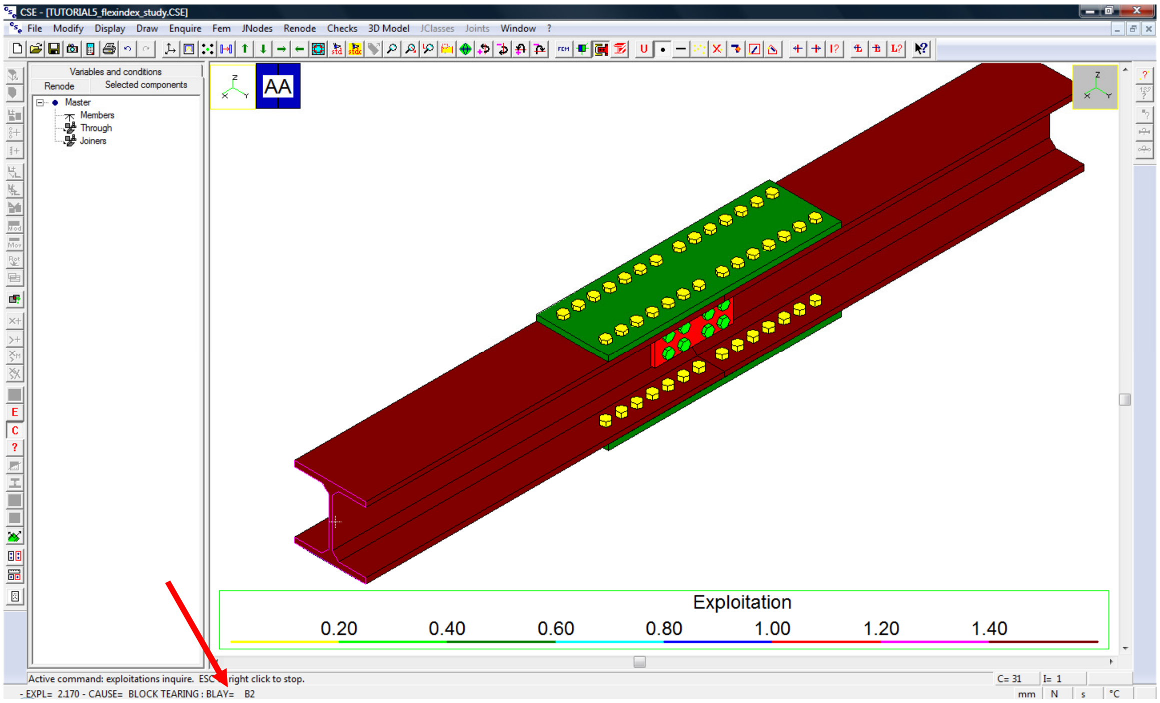
3.3 CHECK AND RESULTS

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

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



Maximum exploitation is in combination 31 (compression). Move to combination 31 , show current combination exploitation and enquire member results .

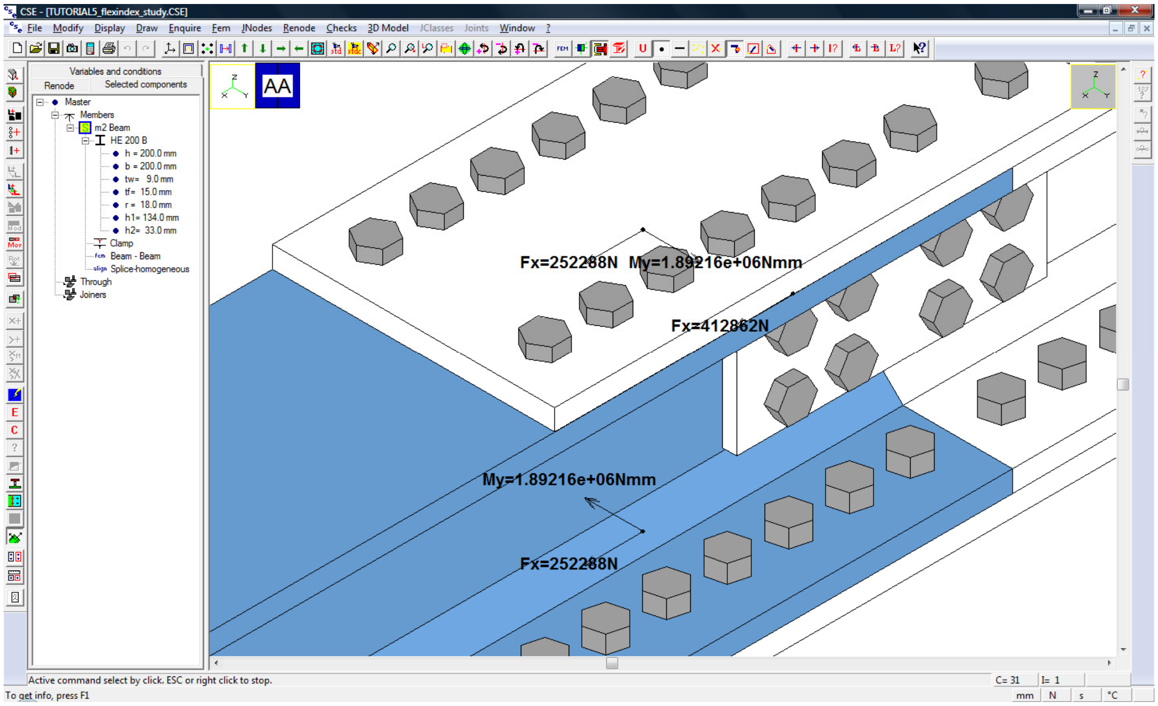


Maximum exploitation for this member is caused by block tearing in correspondence of bolt layout B2, the one on the web. Applied loads are the same, but exploitation is greater than before (paragraph 2.8): this is due to flexibility index modification.

End inquire command pressing ESC or mouse right button, then exit from current results display → .

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

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

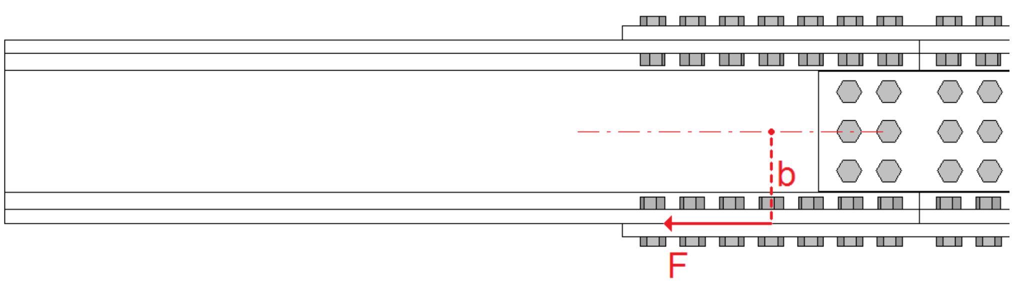


F_w is the force trensferred from the member to web bolt layout, F_f is the sum of the forces transferred from the member to upper flange and lower flange bolt layouts.

F_w = 412862N

F_f = 2 x 252288N= 504576N

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



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

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

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

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

h	200mm
b	200mm
t_w	9mm
t_f	15mm
r	18mm

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

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

$\%A_f$ is the ratio between flanges area and the sum of flanges area and web area:

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

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

3.4 FLEXIBILITY INDEX INCREASING

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

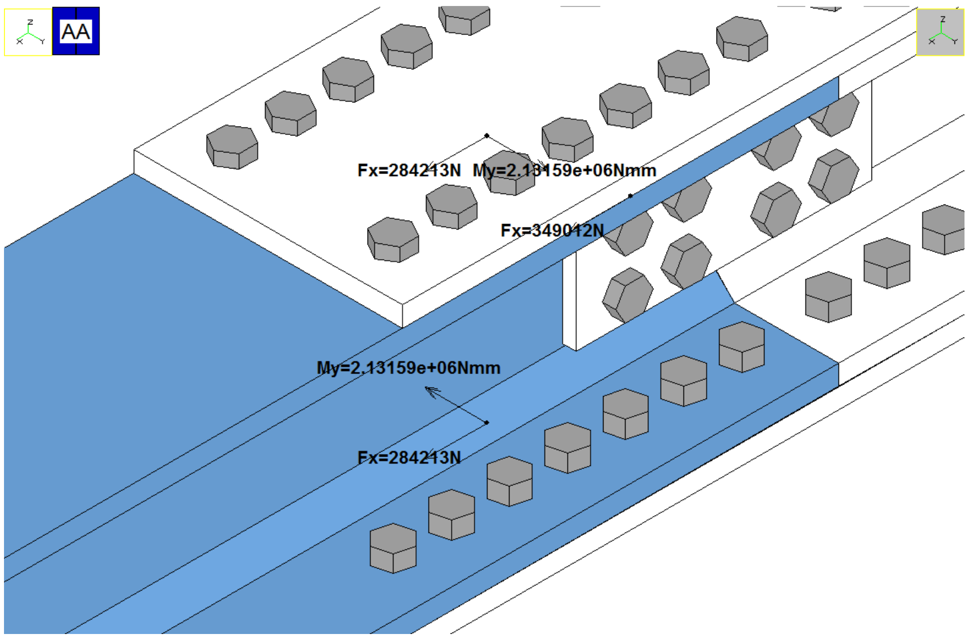
A. Exit from *Display forces*:  → 

B. As in 3.2, select a web bolt layout, this time set equal to 1.5 its flexibility index and press OK; do the same for other web bolt layout. *NB: if you have problems on your computer in typing '1.5', initially type '15', then place a dot between 1 and 5*

C. As in 3.3, re-execute checks, select m2 and display exchanged forces in combination 31

D. As in 3.3, compute $\%F_f$

You will see the following exchanged actions.



F_w = 349012N

F_f = 2 x 284213N= 568426N

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

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

f	1	1,5	2	2,5	3	3,5	4
$\%F_f(f_w)$	55	62					

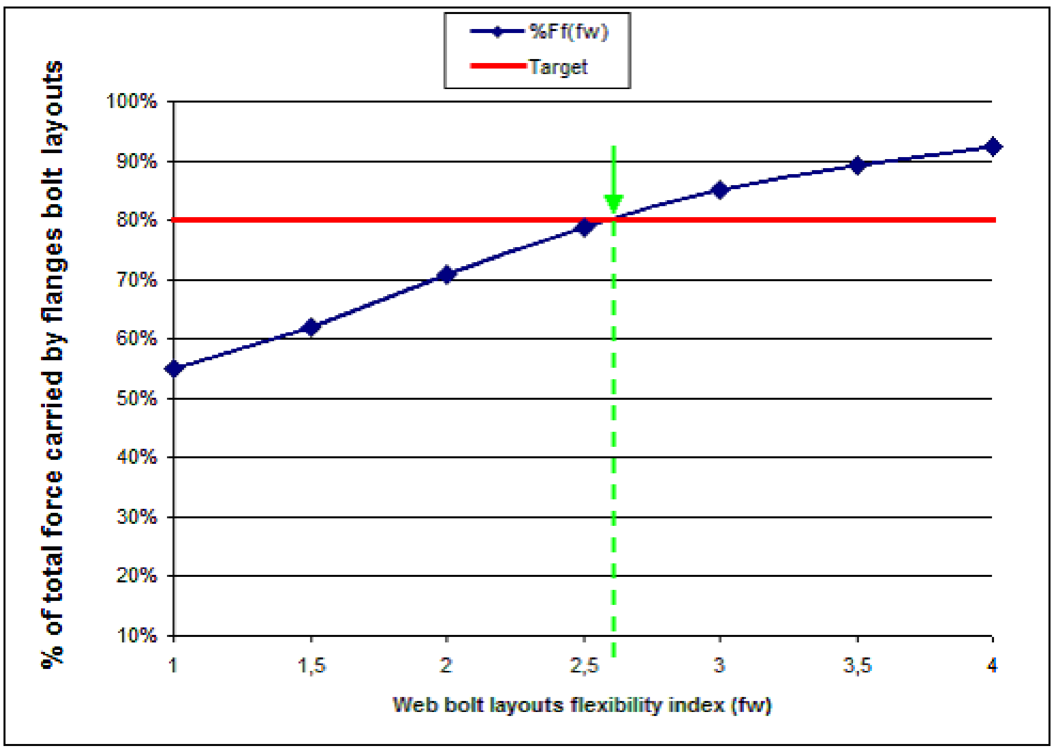
Repeat steps A, B, C and D, but this time set flexibility indices equal to 2, then execute checks and write down computed $\%F_f$.

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

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

3.5 CONCLUSIONS

Plot $\%F_f$ (f_w) versus f_w in a graph:



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

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