

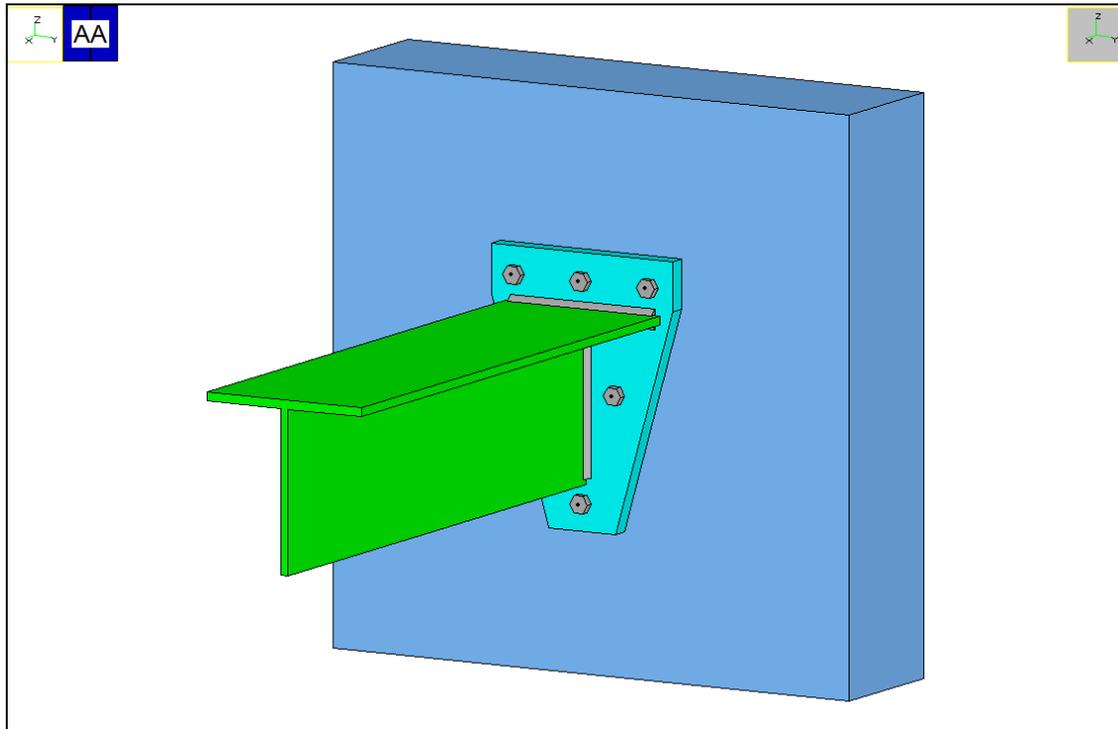


Paolo Rugarli



Connection Study Environment

Tutorial 6: Constraint with polygon plate and free position bolt layout



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

Via Pinturicchio, 24

20133 Milan - Italy

staff@castaliaweb.com

Copyright © 2000-2011 – Castalia srl

Referring to CSE Version 4.40

Rev.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:

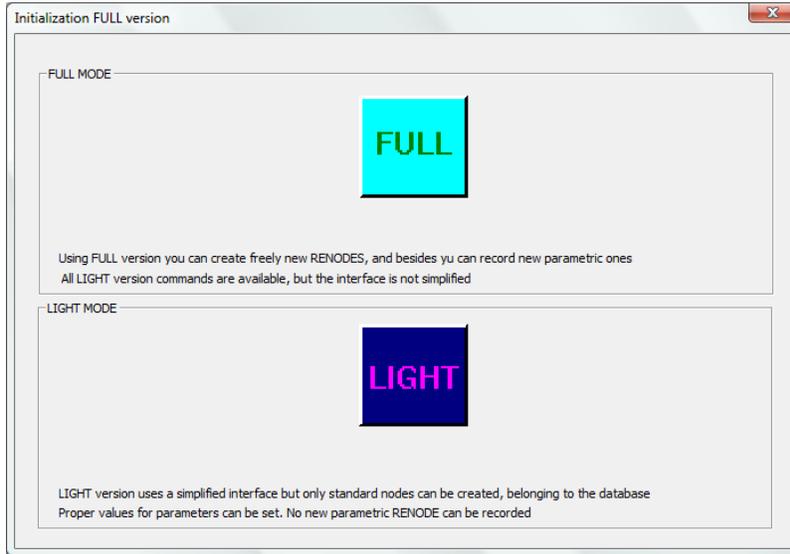
- Create a dummy fem model to study a constraint joint
- Assign the materials and cross-section to the fem elements
- Search members
- Search jnodes
- Add a generic polygon plate
- Add a bolt layout with free bolts positioning
- Add a weld layout
- Set the checks to be performed
- Have a general look at the results
- Understand bearing surface results

The connection here studied was designed not to be realistic, but in order to give a simple and clear example of a connection which needs an irregular polygon plate and a “free” disposition of the bolts.

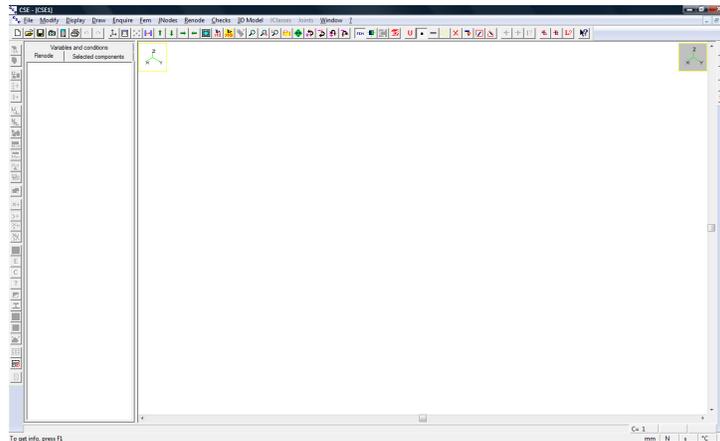
This tutorial is some like 60 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

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 CONSTRAINT



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

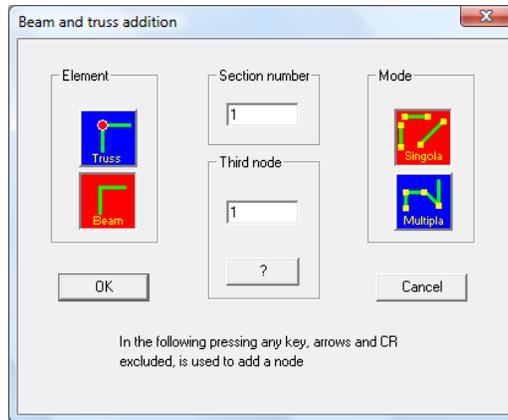


Initial window content: blank.

2.1 STEP 1: GETTING THE FEM MODEL

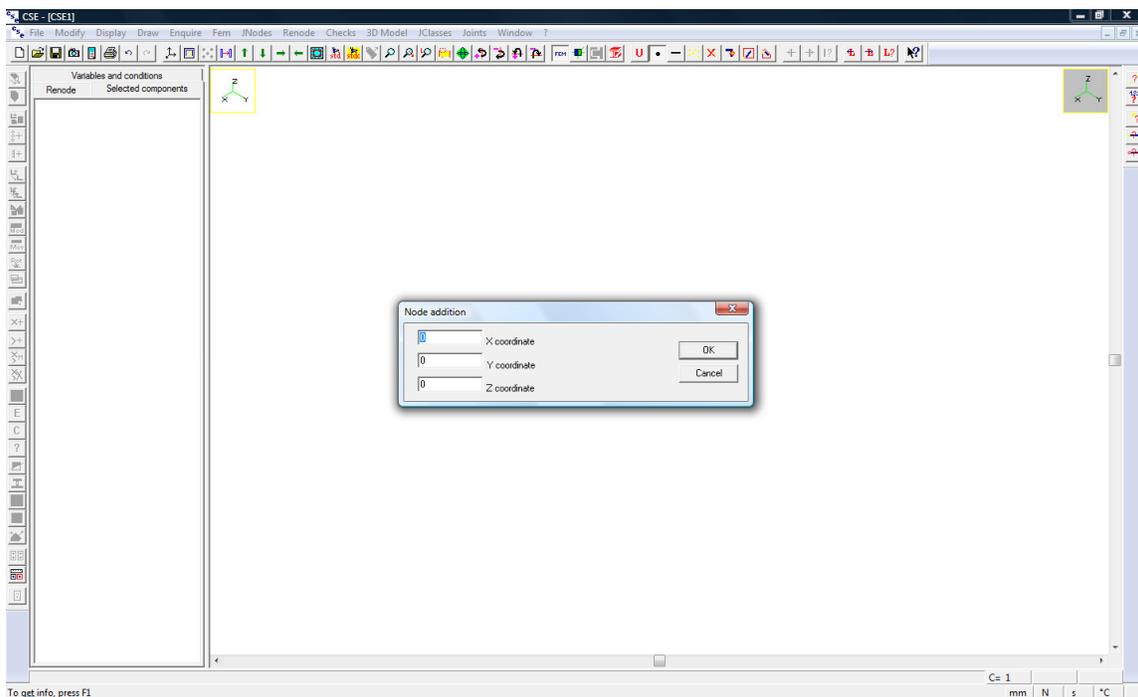
2.1.1 Getting the fem elements

Activate right window by clicking left inside it, then execute the command **FEM-Elements-Add**:
the following dialog box will appear.

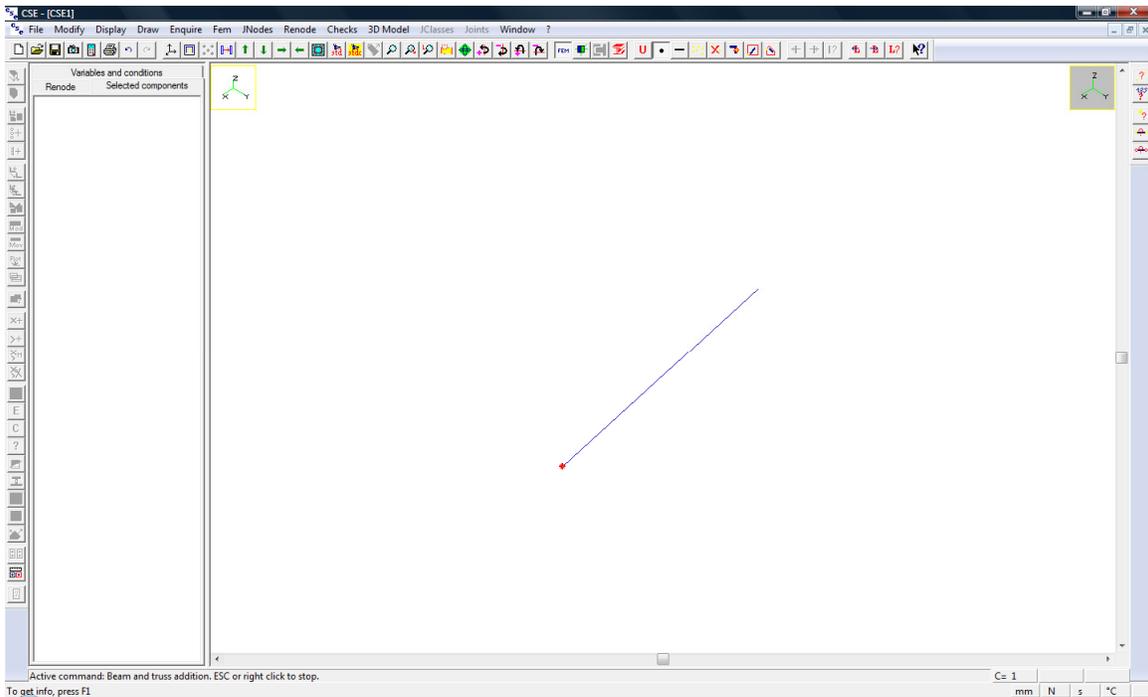


Select “Beam” and “Single” (they become red); leave third node default value (1 = to the infinite in +Y direction). Press OK.

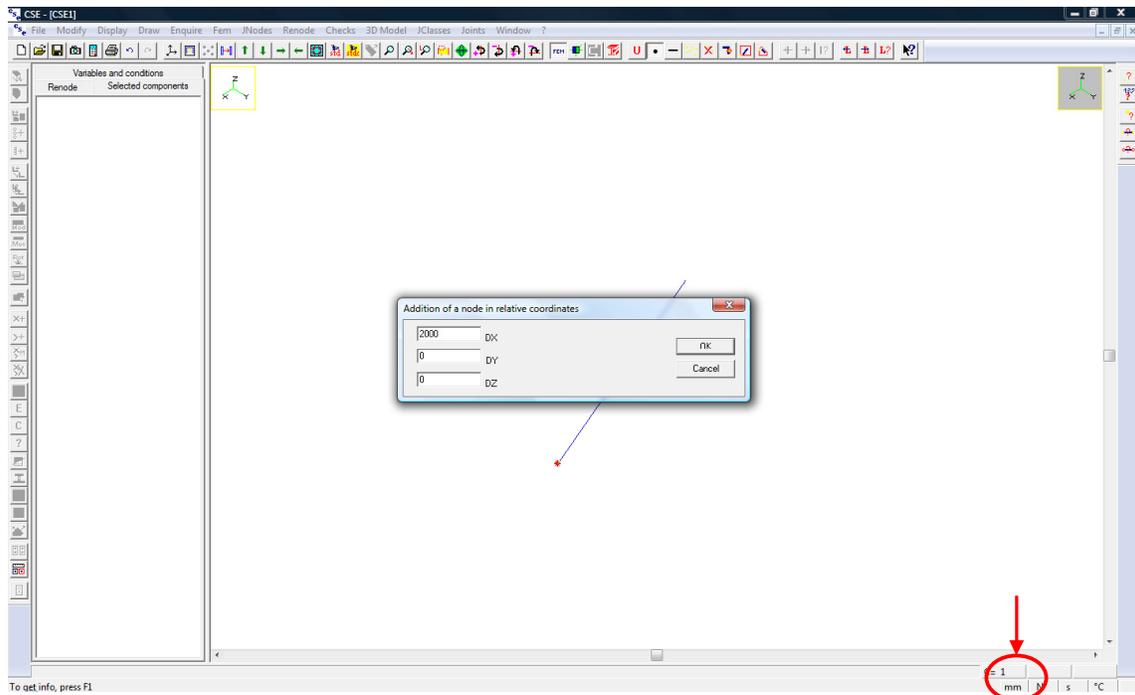
The first node of the element must be added now. You can click an existing node or define a new one. Since there are not nodes yet, click the SPACE BAR on your keypad: the following dialog box will appear. In this dialog box you can set first node coordinate: leave (0, 0, 0) and press OK.



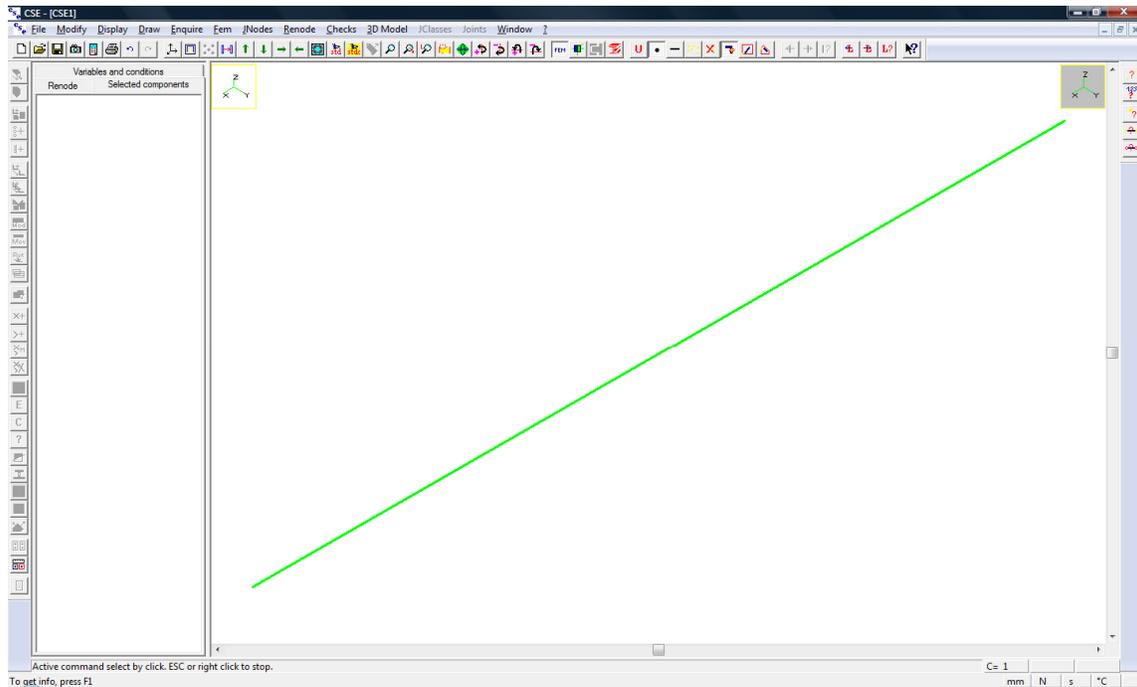
The first node has been added. If you move the mouse, a line starting from the node will join mouse pointer and the node. If there were other nodes you could click them, but since there is only one node, press the SPACE BAR again in order to add the second node.



A dialog box appears: it defines second node coordinates using a vector starting from the first node.



Type 2000 in DX box to define a vector in +X direction with a length equal to 2000mm (current units are displayed in bottom right part of CSE window; use **Modify – Units**  to change units).



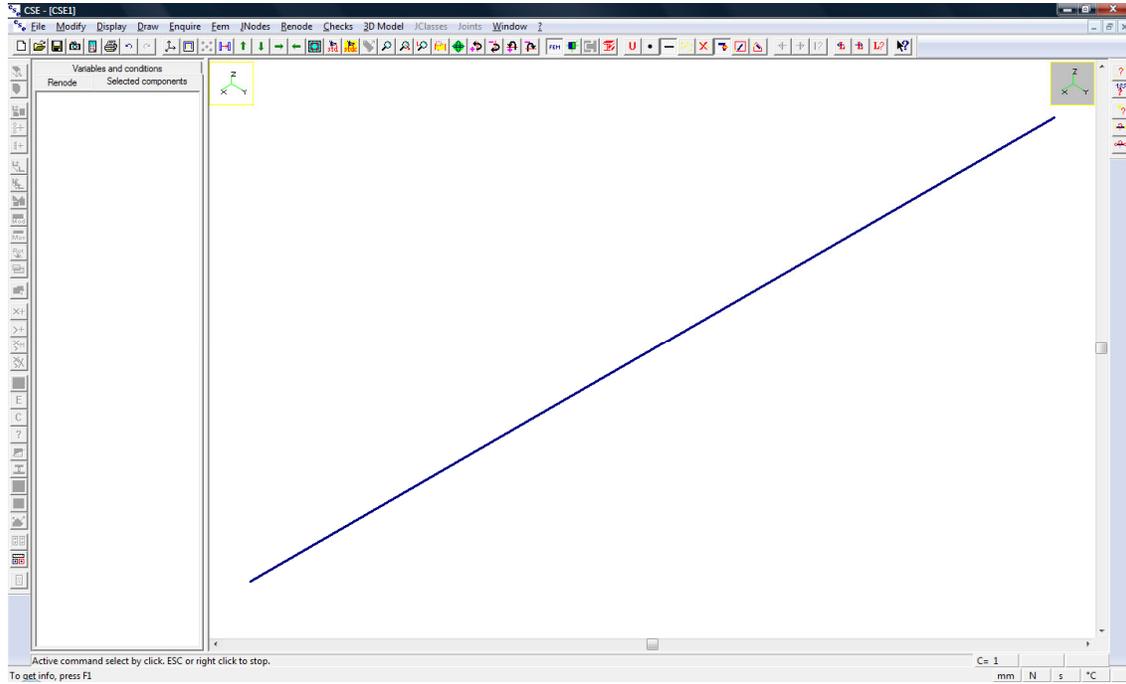
Beam element has been added. Press ESC to end elements addition command.

2.1.2 Assigning cross-section

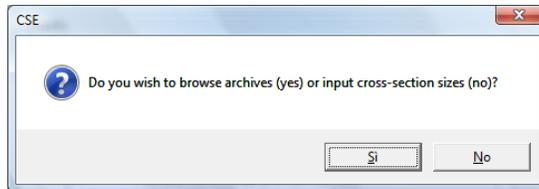
To assign a cross-section to finite elements, chosen elements must be selected. First of all click **Modify – Select – 1D** to enable finite elements selection. You can also use the button in the toolbar.



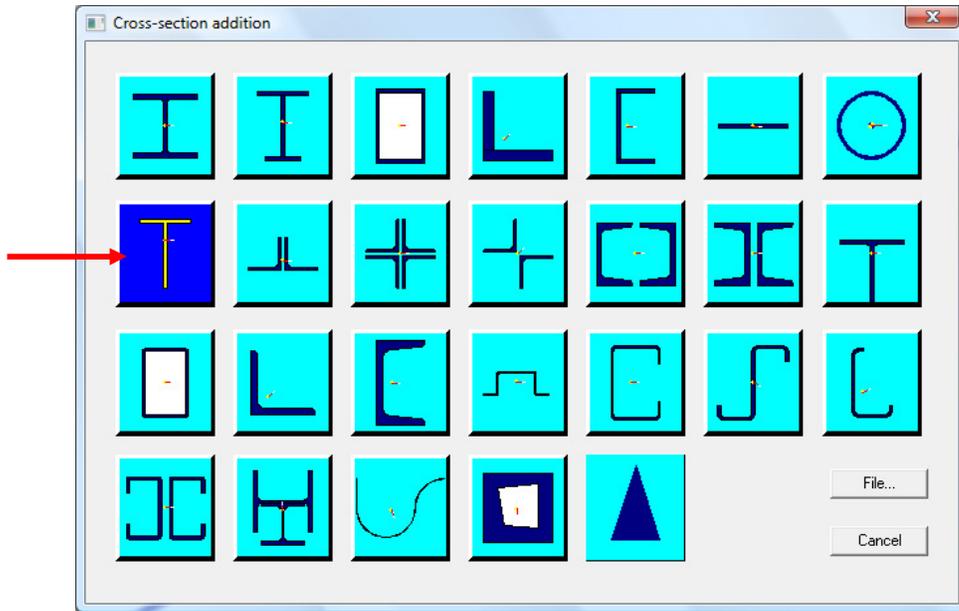
Now select the element clicking on it with mouse left click. Selected element is blue.



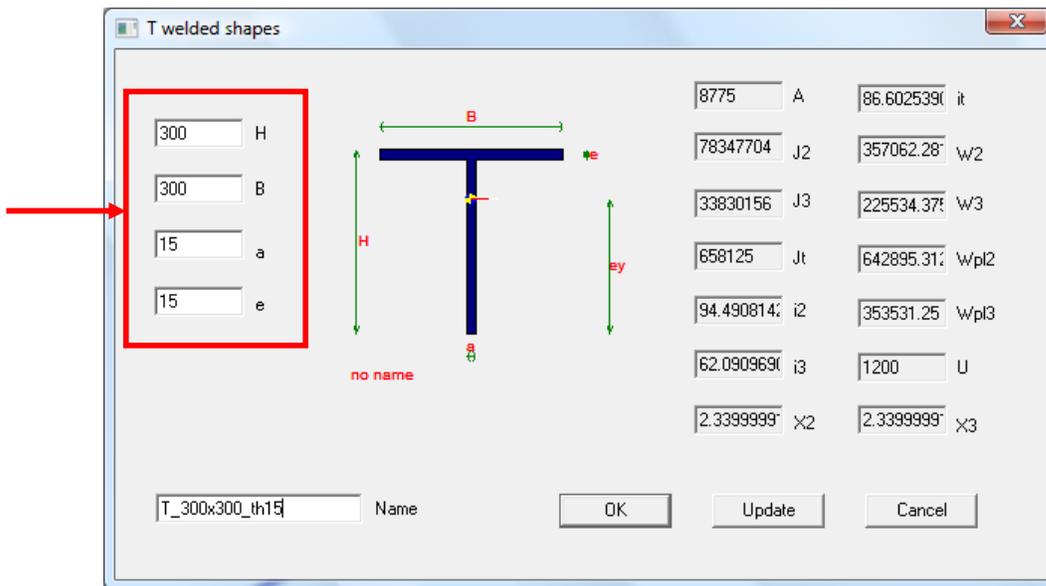
Use **FEM-Elements-Cross-sections** command to assign a shape. If you are using the demo this command just applies the cross-section HEB200 to the elements, otherwise a dialog box asks if you want to choose a shape from CSE archive (Si=Yes) or add a new shape (No):



Click “No” to add a new shape and you will be prompted to choose a cross-section kind, with the following dialog box.



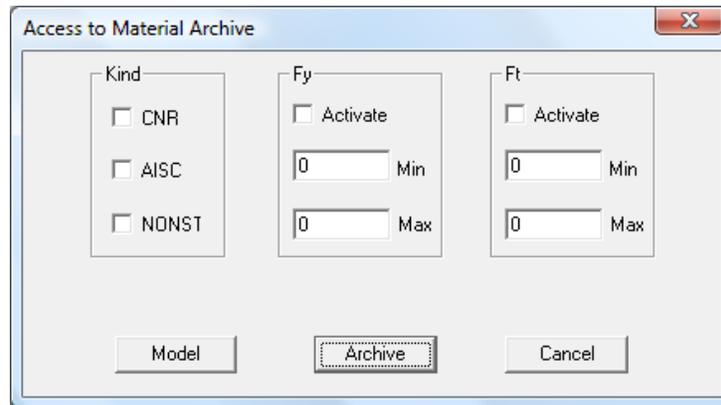
Choose welded T shape, and set the following parameters in next dialog box.



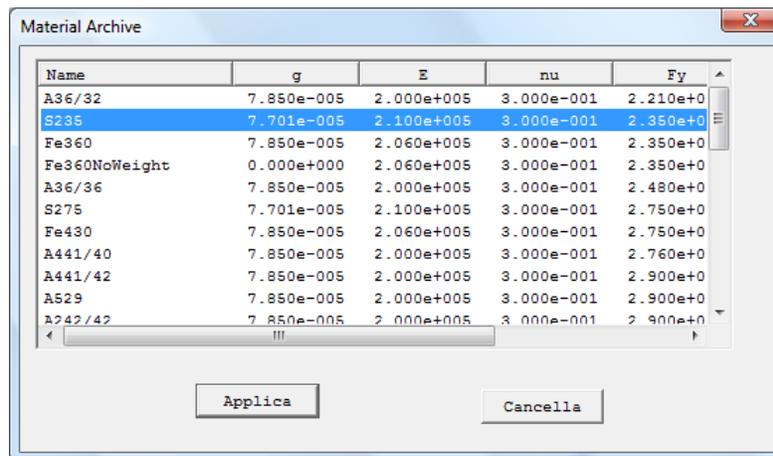
Click “Update” button to update cross-section image and properties (area, moments of inertia, ecc.); Type a name for the section, then press OK to assign this cross-section to selected beam element.

2.1.3 Assigning material

Execute the command **FEM-Elements-Materials**. If you are using the demo this automatically applies the material S235 to selected element. Otherwise choose “Archive” in the following dialog:



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



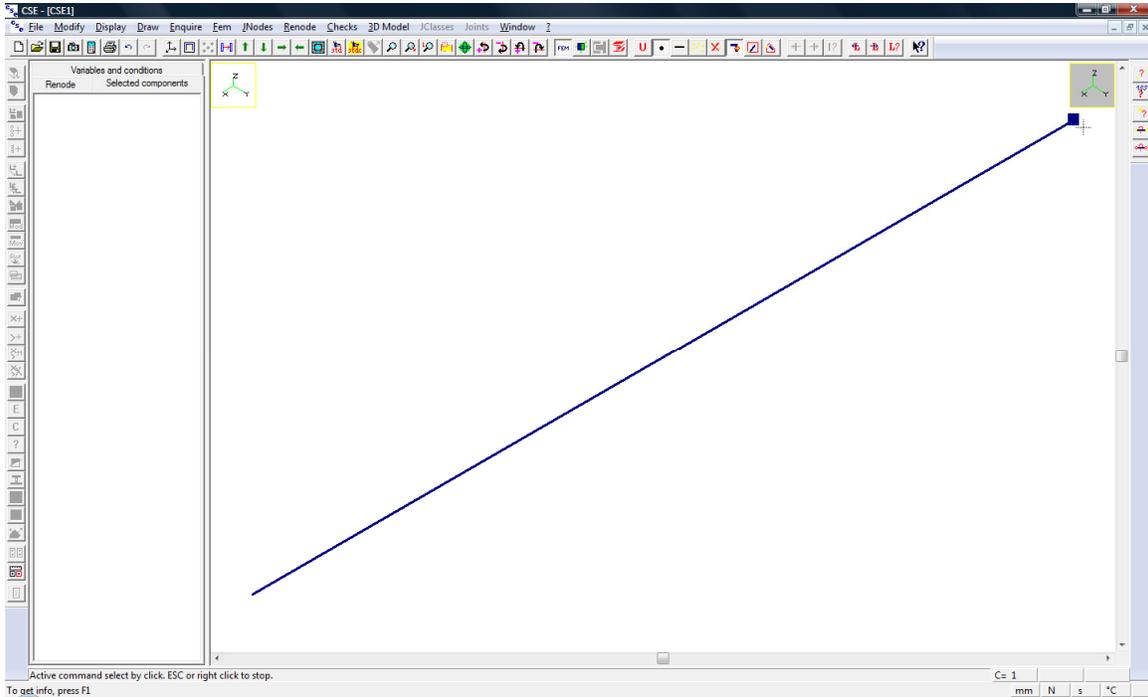
S235 has been assigned to selected beam element.

2.1.4 Constraint

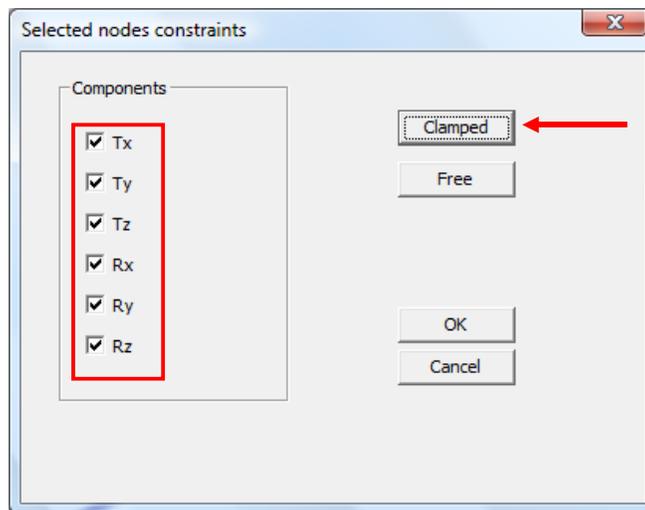
To assign a constraint to a node (or to more than one node), chosen node(s) must be selected. First of all click **Modify – Select – 0D** to enable nodes selection. You can also use the button in the toolbar.



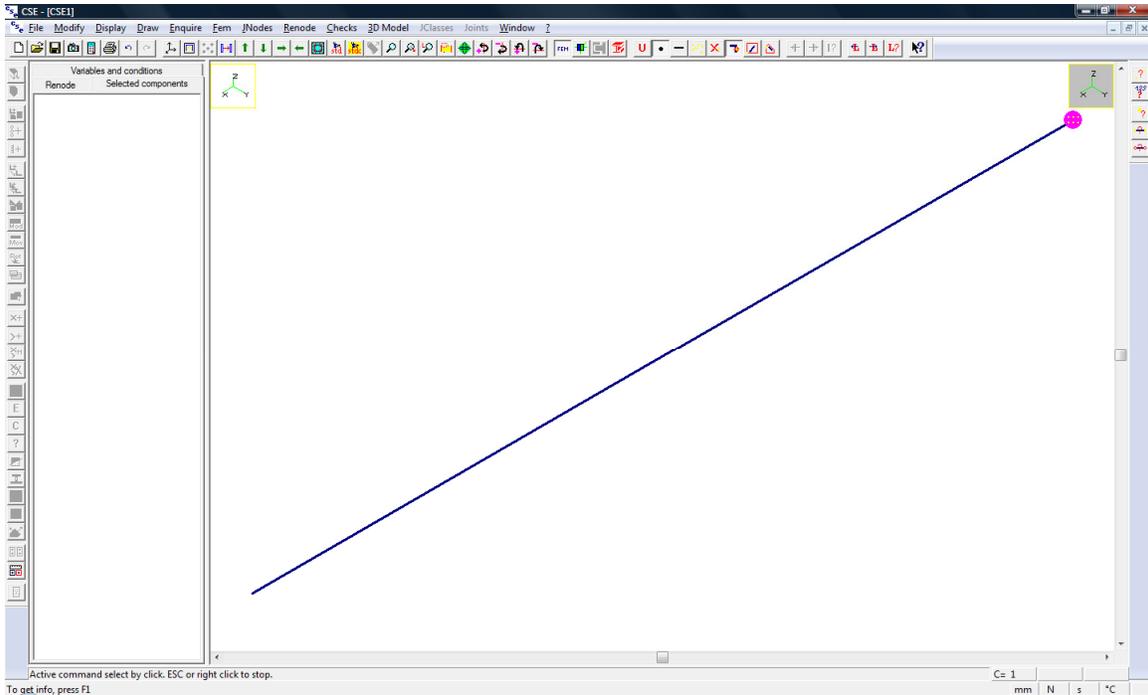
Now select top-right node clicking it with mouse left button. A blue square highlights selected node.



Use **FEM – Assign constraint** command. In the dialog box click “Clamped” to define a constraint in which all displacements and rotations are not allowed: when you click this button, indeed, all T_i and R_i boxes are automatically ticked (“Free” removes all the ticks, and manually ticking proper boxes you can define hinges or other constraint kinds).



Press OK to assign constraint; in the scene a circle is displayed on constrained node, and 6 pixels in it mean that all translations and rotations are not allowed.



The fem model is now completed. You do not need to add actions because, since the fem model has been prepared inside CSE, CSE will use fractions of elastic or plastic limits or defined values as design generalized stresses (internal forces, i.e. axial forces, shears, bending moments and torque).

2.2 STEP 2: SEARCHING MEMBERS

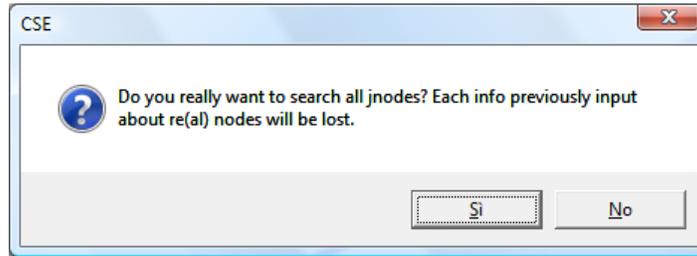
To move to connection design you now have to detect which members are present in the fem model.

To do that just execute the command: **FEM-Search members!**

2.3 STEP 3: SEARCHING JNODES

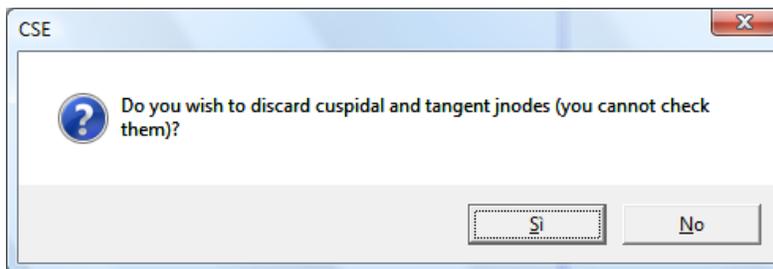
Now that members have been searched, you need to find different "jnodes", that is what will be next a true, real node (renode). The program scans the member model and finds how many equal and how many different jnodes there are in the model. Then each jnode will be marked and you will be able to select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

To search jnodes just execute the command **JNODE-Search jnodes!** and answer "Si" (Yes) to the following question:



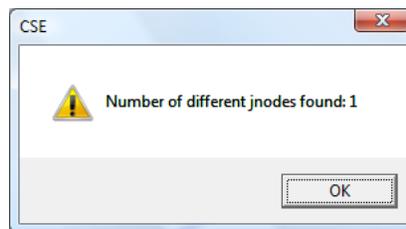
As no real nodes info has been defined you will not lose anything. Jnodes search is usually done once for all in a model. Before beginning to add RENODES, you will check that the JNODES found are correct. This depends also on how the fem model has been prepared.

The following dialog box appears:



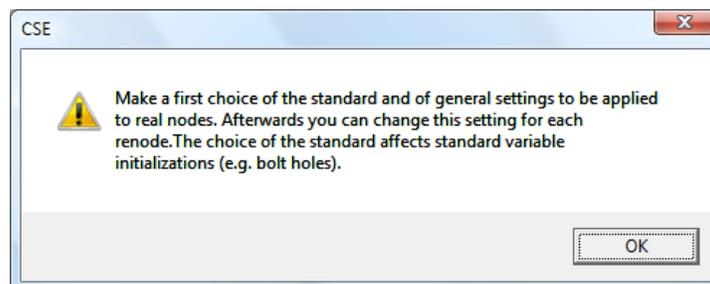
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:

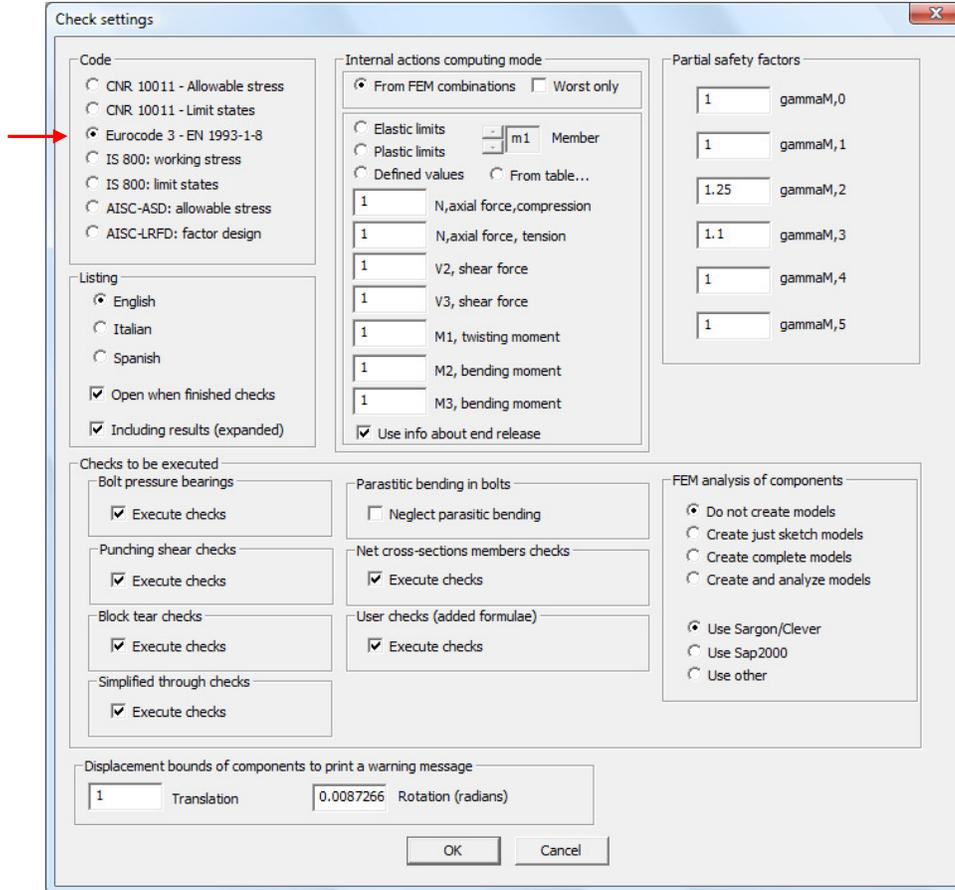


This means that in this model there is just one jnode, the constraint.

Now the following message appears:

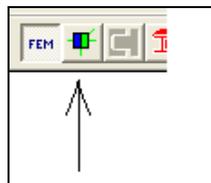


It asks to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

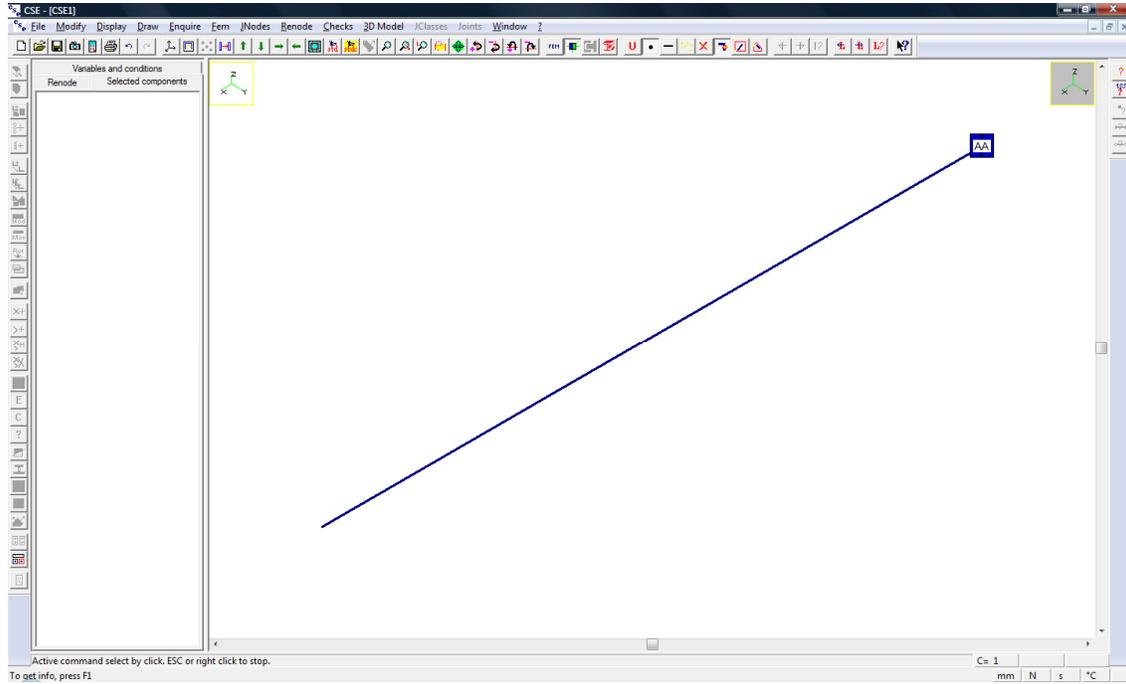


Choose Eurocode 3 and leave all other defaults, we will change them later.

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



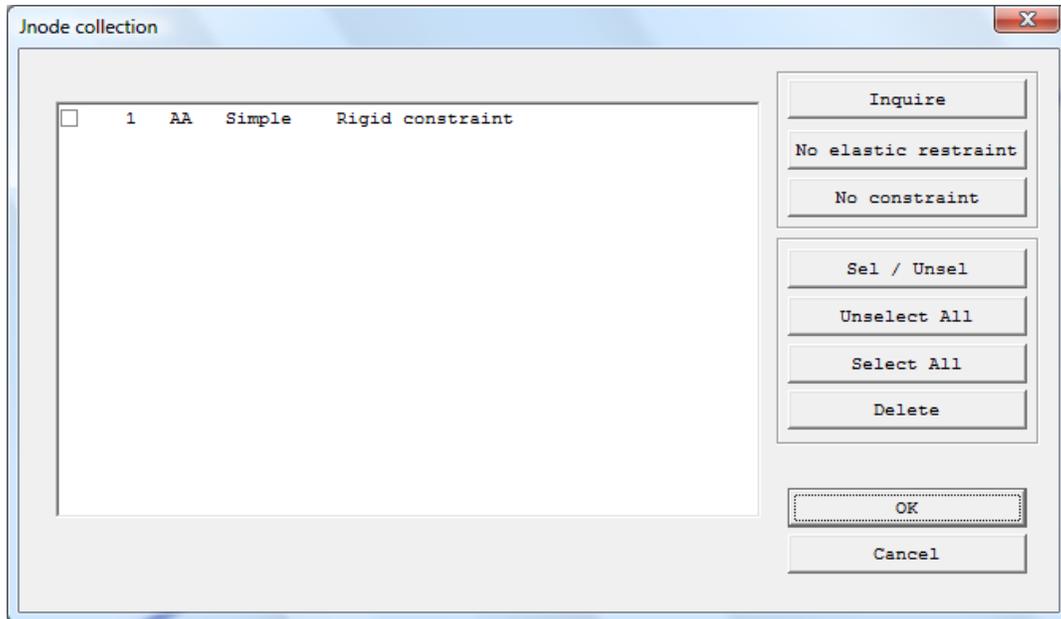
In jnode view you get the following:



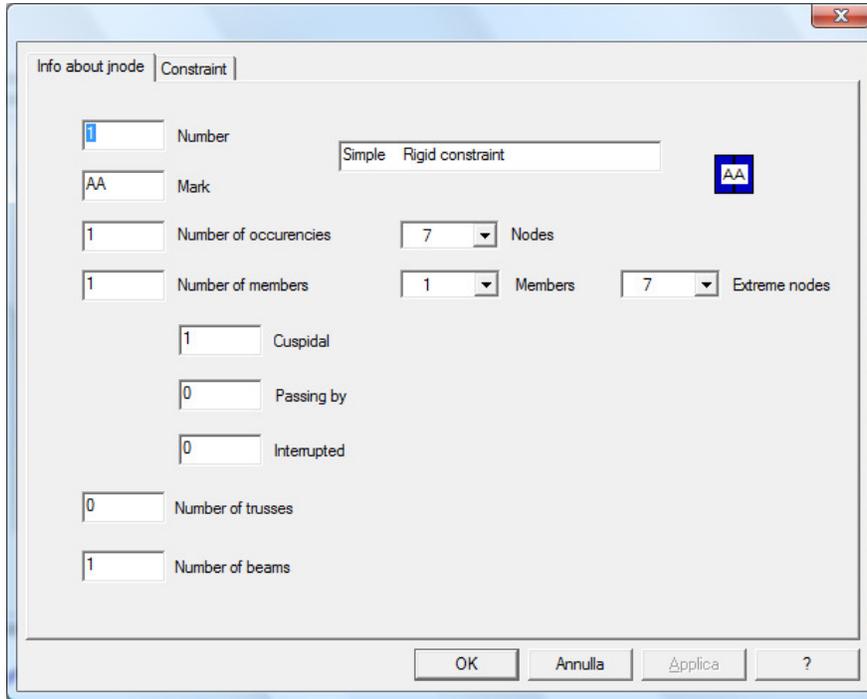
You can now get some info about the jnode found. Execute the command

JNODE-Edit...

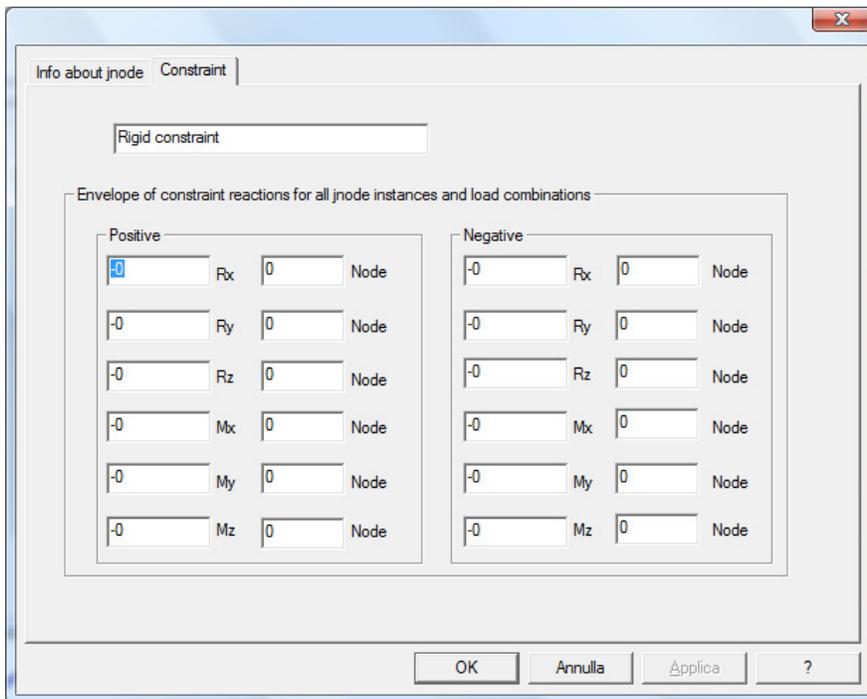
and get the following dialog:



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



and clicking to "Constraint" 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 connection info and constraint forces envelope; this part is blank as this fem model has been prepared inside CSE and is not coming from a true fem analysis.

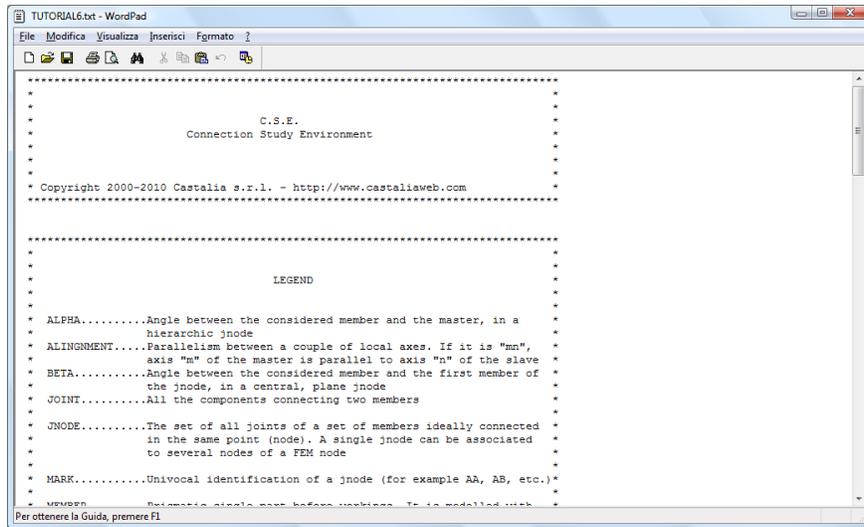
Press OK and then Cancel to exit from both dialogs, including "Jnode Collection" dialog.

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 TUTORIAL6.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: SIMPLE CONSTRAINT

NODES ASSOCIATED TO THIS JNODE: TOTAL 1
7

MEMBERS IN THE JNODE: TOTAL 1
1

      CONSTRAINT - KIND= FIXED      SHAPE=T_300x300_th15

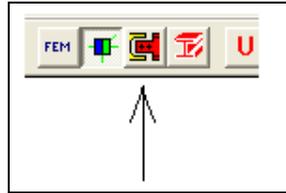
RIGID CONSTRAINT

```

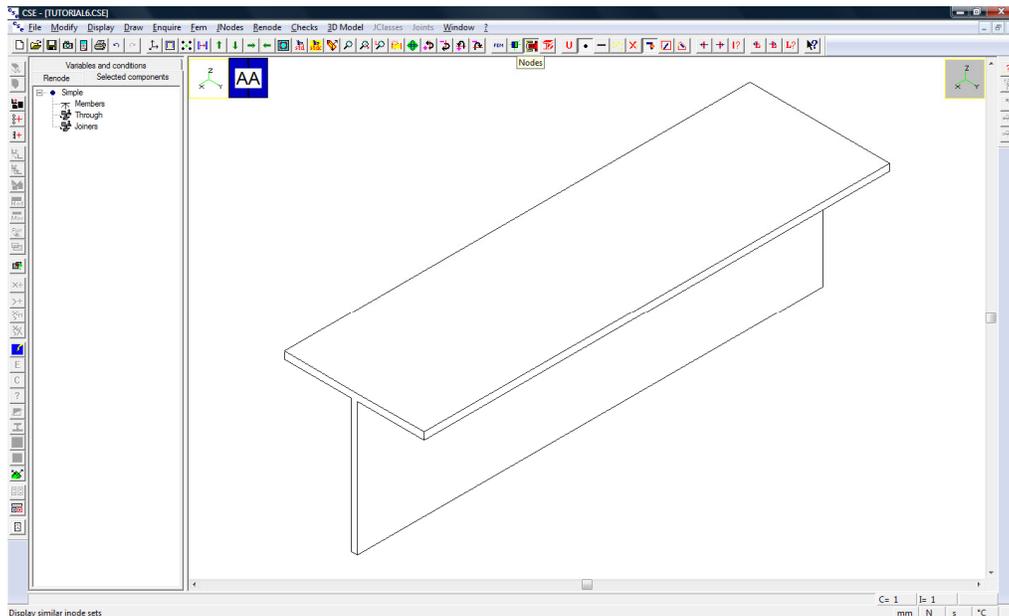
2.4 STEP 4: SELECTING THE PROPER JNODE

We are sure that jnode is correct; we can move forward to select it to transform it into a **Real NODE**. Left click with mouse over the square "AA" in the graphic pane: it gets yellow. This means that jnode is selected (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 allowed. 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:



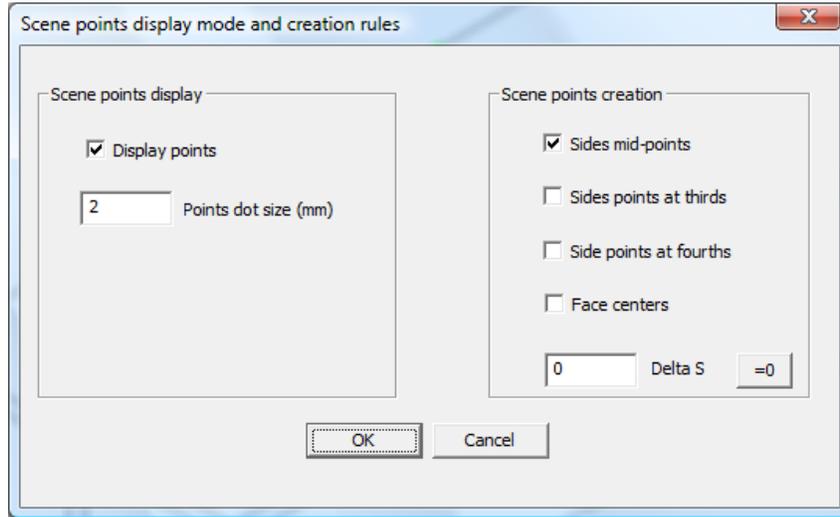
Note that the left window is not empty anymore: it gives you info about the components selected, the components present in the renode, and the variables and conditions present in the model.

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

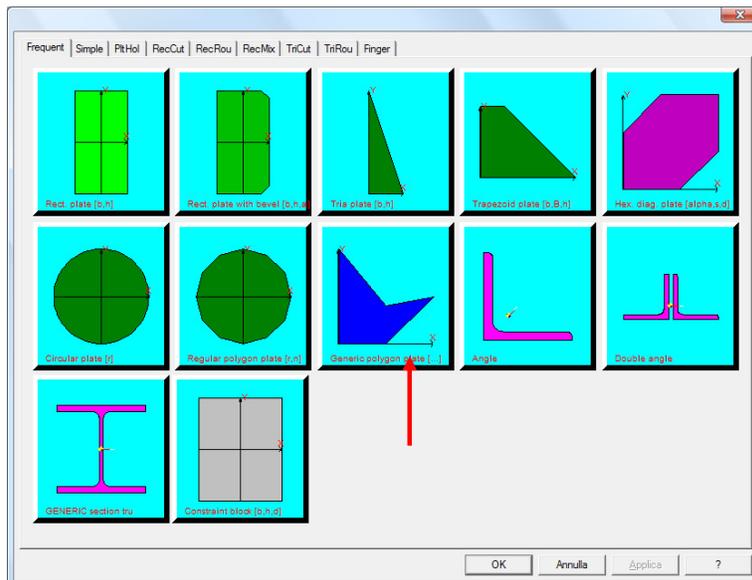
2.5.1 Addition of generic polygon plate

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

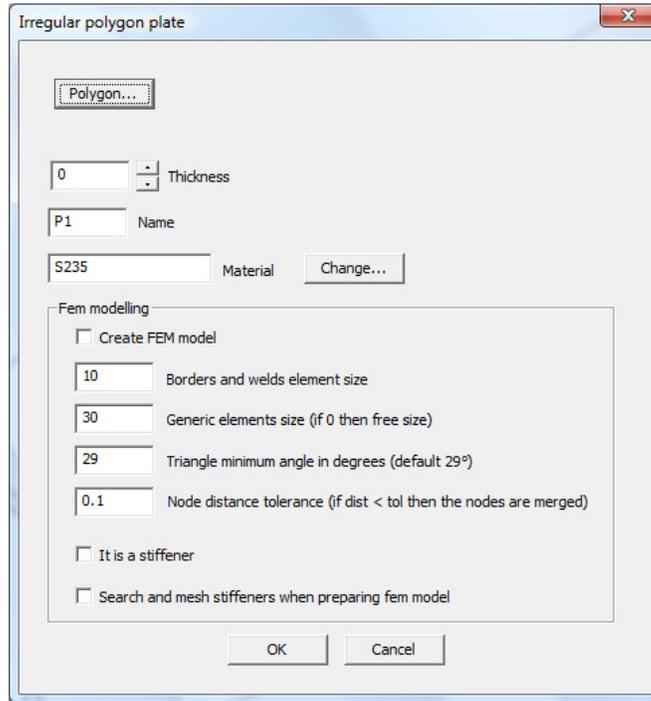


- Tick “Display points”
- Type ‘2’ as dots size
- Remove all the ticks in “Scene points creation” section, but keep “Sides mid-points” ticked.

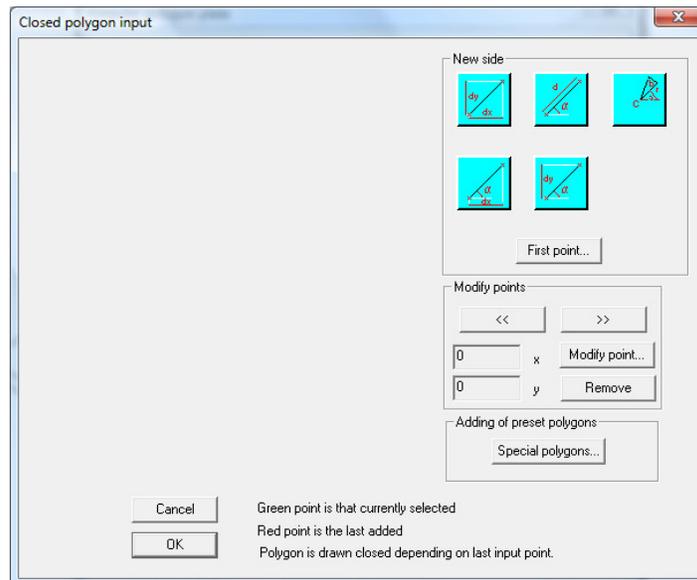
Use **Renode – Components – Add through**  and click the generic polygon plate in the following property sheet:



The dialog box that appears is used to irregular polygon plate shape and parameters.

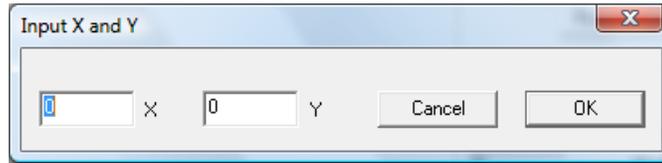


To define plate shape press “Polygon” button: the following dialog box will appear.

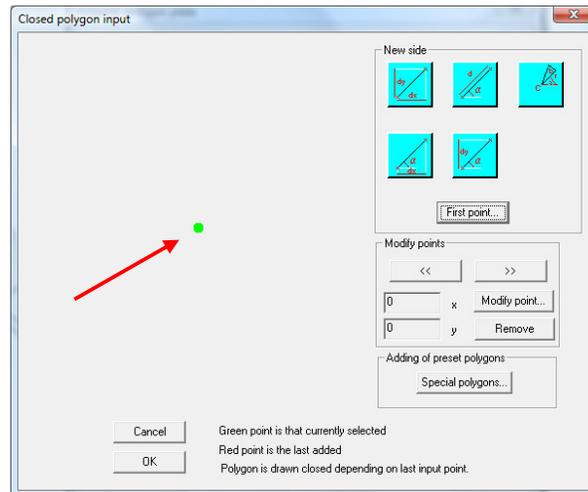


In the left part of the dialog box there is plate image (currently it is empty); on the right there are buttons to define plate shape.

Click “First point” to define first point coordinates. The following dialog box appears: press OK to define point (0, 0) as origin.

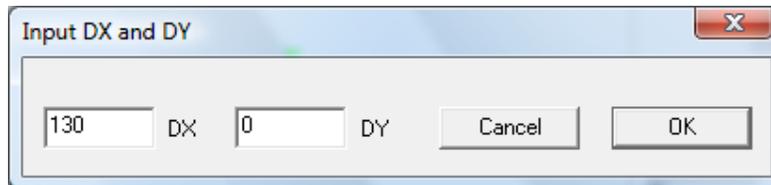


Now the first point is displayed.

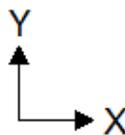


Use this button to add a side of the polygon:

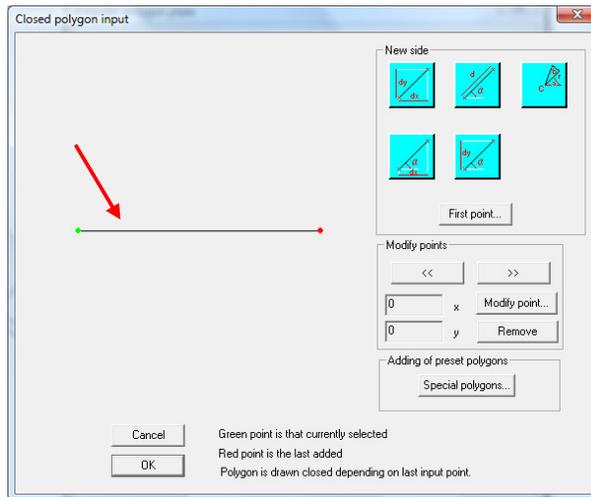
The following dialog appears; type ‘130’ in DX box to define relative coordinates between second point and first point (millimetres are current units):



N.B. Reference axes for plate definition are the following:



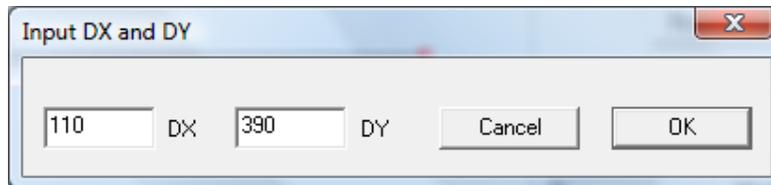
The first side has been added (last added point is red, currently selected point is green).



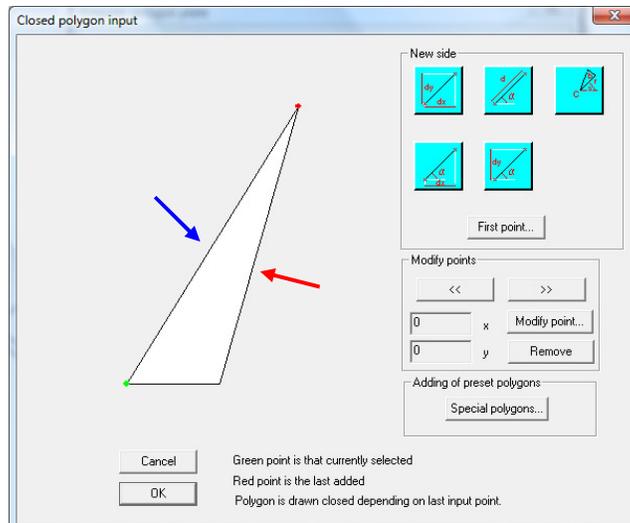
Add a new side::



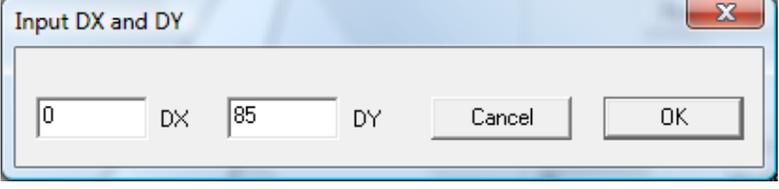
Press  then type '110' in DX box and '390' in DY box and press OK.

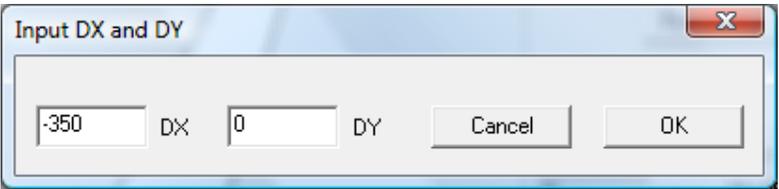


The second side has been added (see red arrow in the following figure). A third side closed the polygon connecting last added point to the first point (blue arrow)



Add the following sides as previously done. The figure is updated when a new side is added.

Press , then type  → OK

Press , then type  → OK

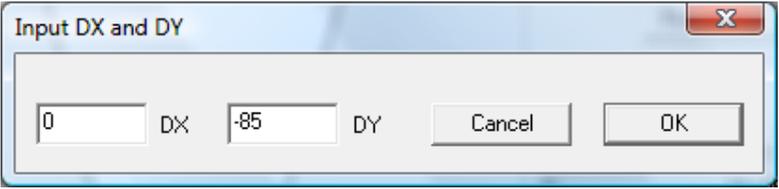
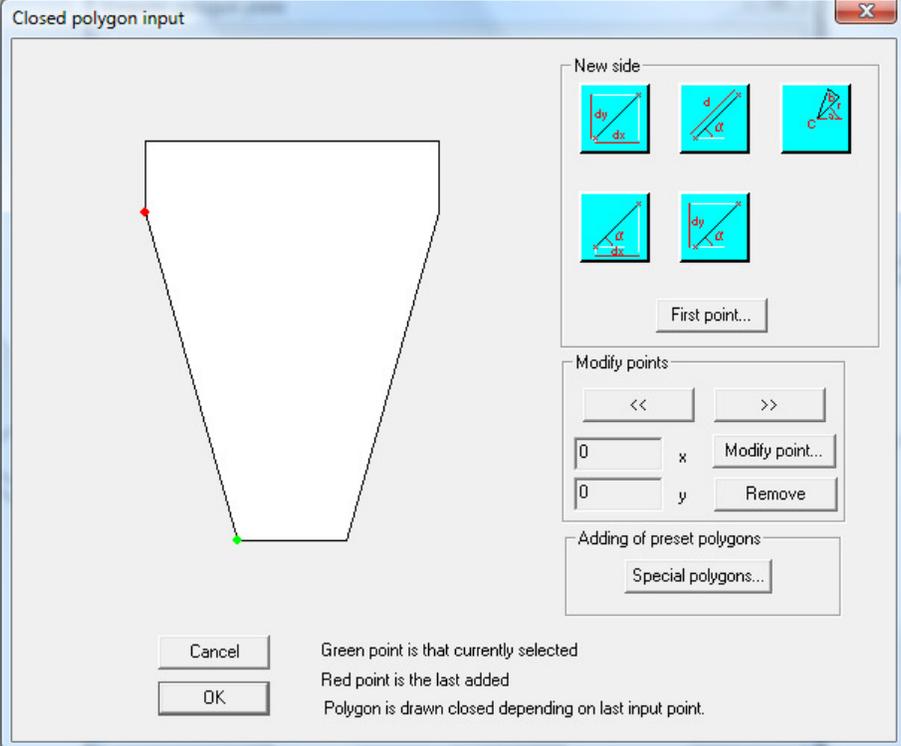
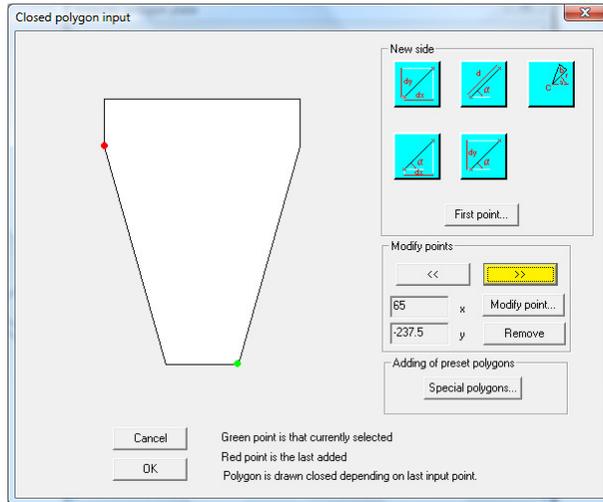
Press , then type  → OK

Plate shape has been defined.



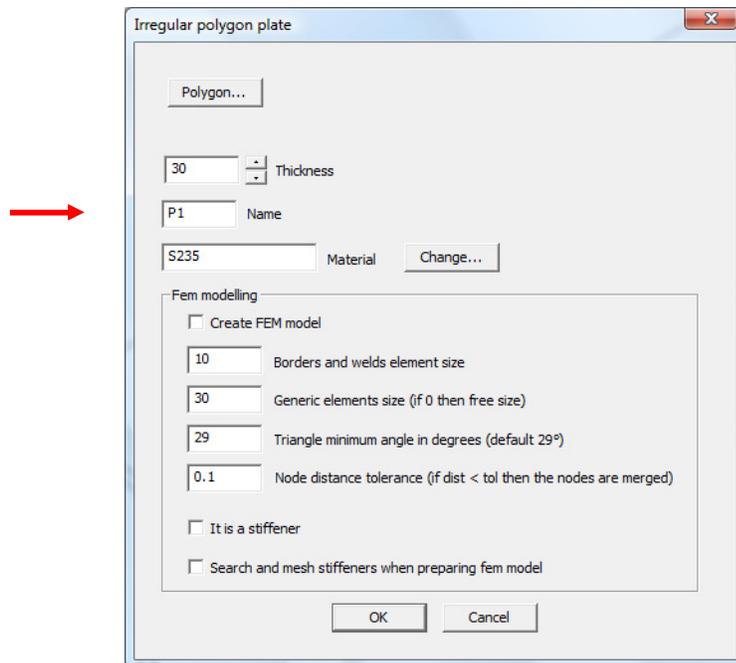
Green point is that currently selected
 Red point is the last added
 Polygon is drawn closed depending on last input point.

N.B.: the polygon can be modified adding you can new sides (here we used only a method, but straight sides can be defined in other ways, and circular sides can be added too – see the guide for more information); previously defined point can be removed or modified. To do that, use the arrows to select desired point (highlight it in green), then use “Remove button” to delete it or “Modify point” to change its coordinates.

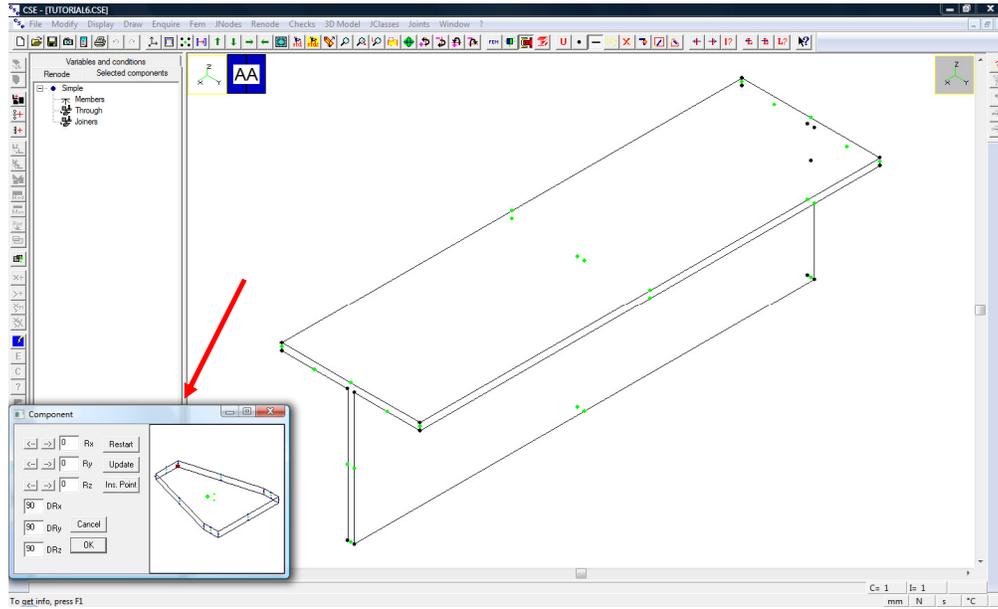


N.B. From the version 4.30 on, a predefined plate with the shape here defined is available.

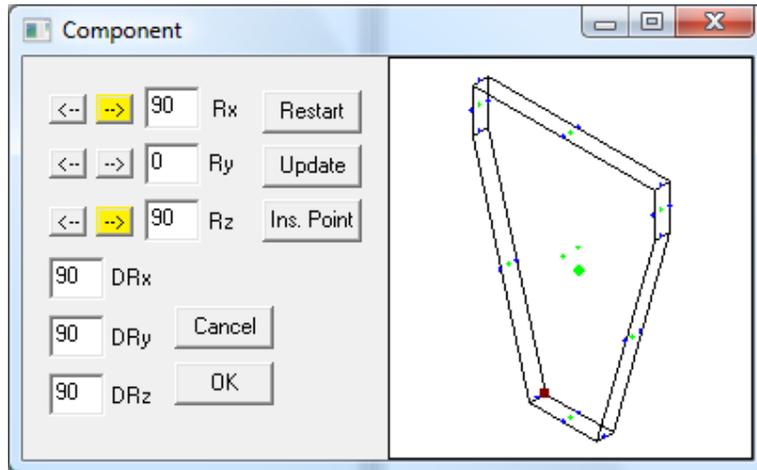
Press OK to end plate shape definition; you get back in previous dialog box. Define plate thickness (30mm), keep all the other parameters and press OK to add the plate.



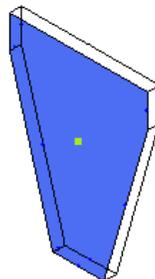
A new dialog will appear: here you can rotate the component you are going to insert, and choose the insertion point.

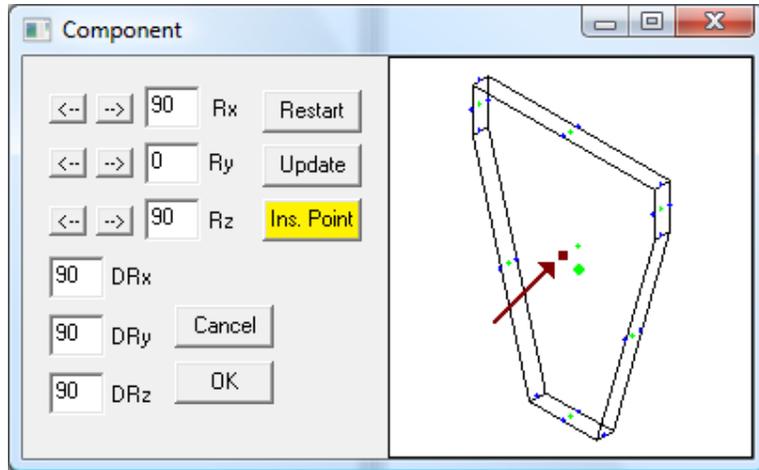


Click → in Rx row, and then → in Rz row to place the plate in the correct position

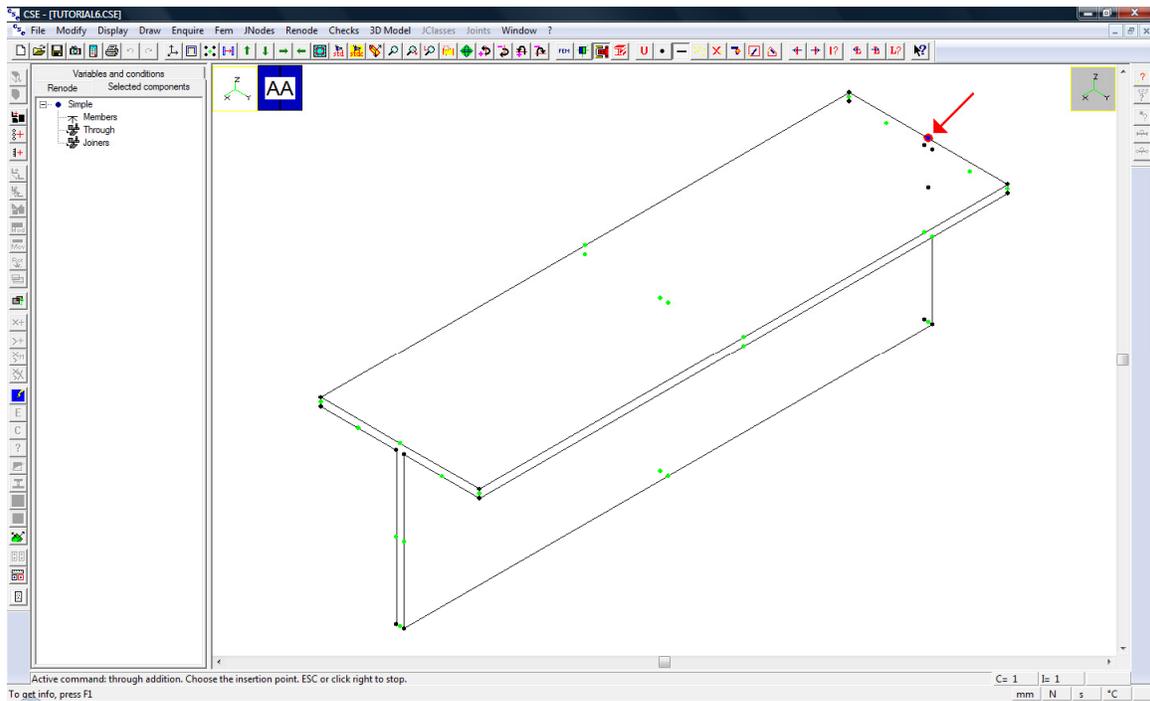


Now click “Ins. Point” and select the centre of the face shown hereunder, then press OK.





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



The plate will be added immediately in the scene.

Use **Draw – Enclose**  to enclose all the objects in the view.

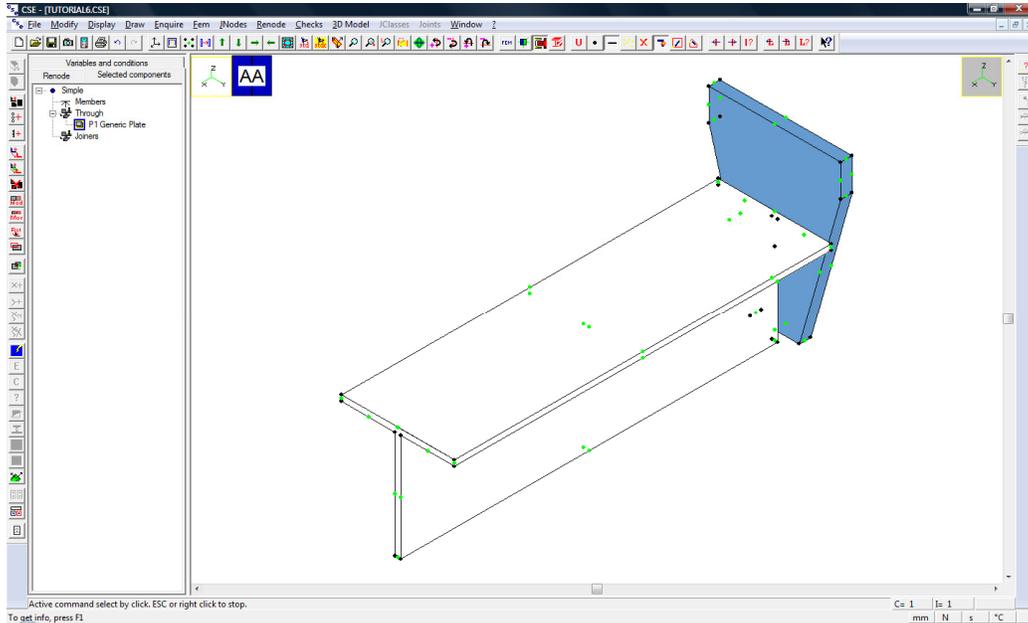
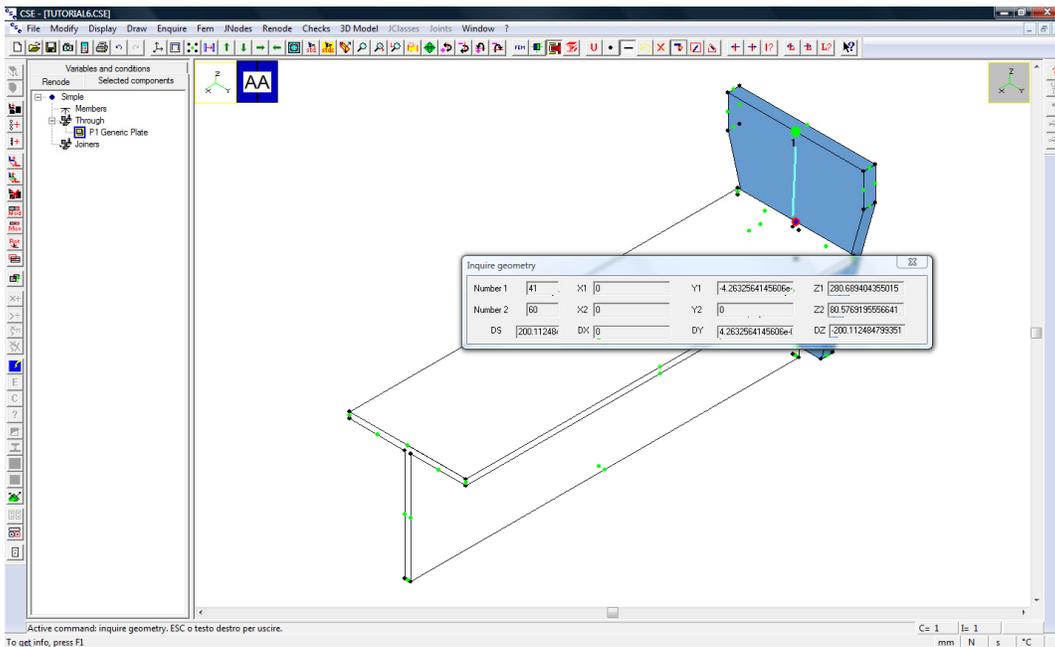


Plate position is not the correct one yet. We need to move it.

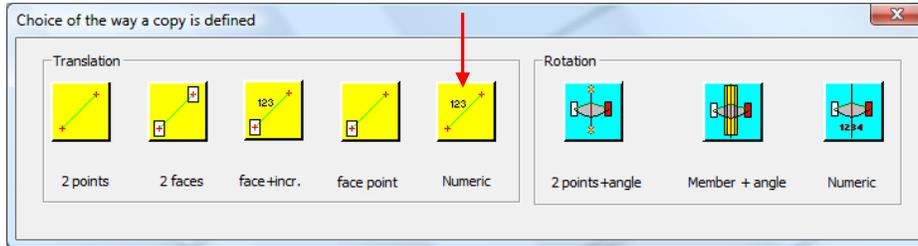
To know distances between points use **Enquire – Geometry** . For example, enquire the distance in Z direction between plate upper side and member upper side. The distance is approximately 200mm.



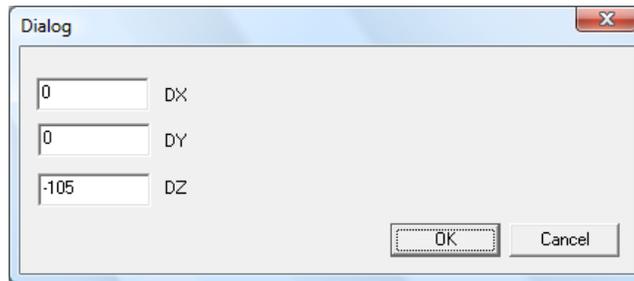
Press ESC or mouse right button to end enquire command.

Plate is the only currently select object (if it is not, because you changed selection status of the objects, select the plate with a click and/or unselect the member).

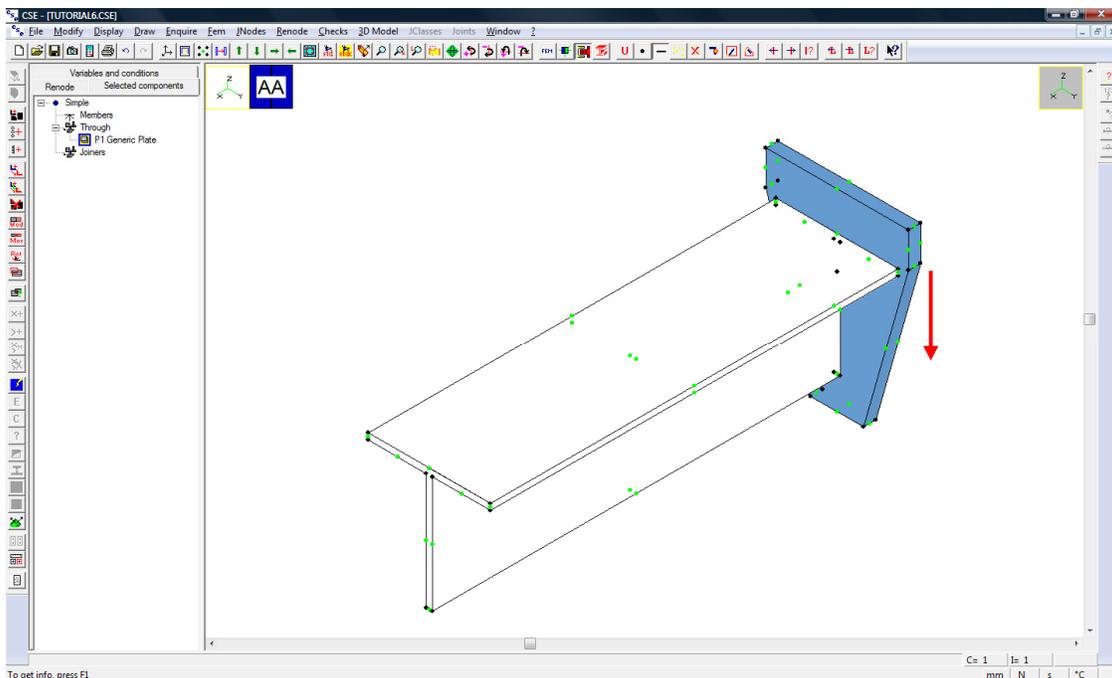
Use **Renode – Shift Mov** and click “Translation – Numeric” in the following dialog box.



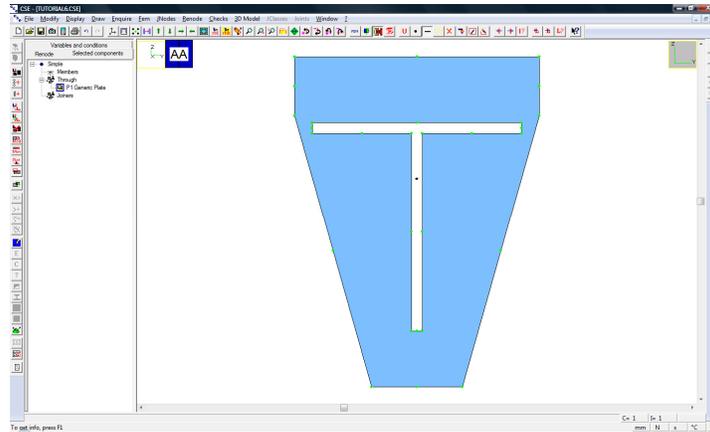
Type '-105' in DZ box to define a 105mm translation in -Z direction, then press OK.



The plate has been shifted downward:

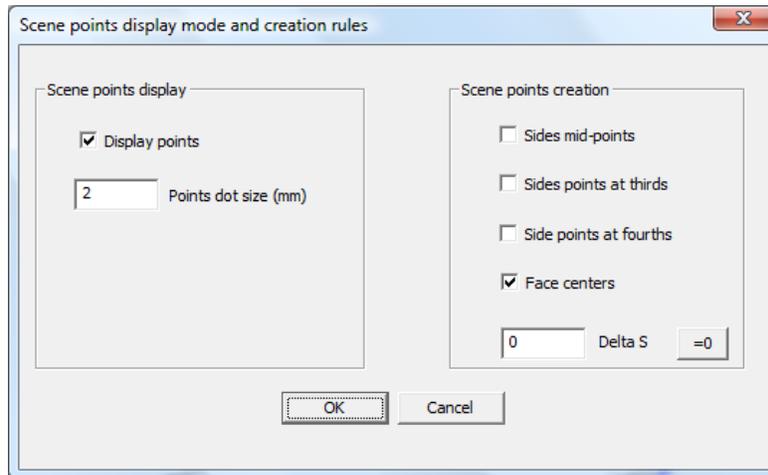


Use **Draw – Std view**  and choose a +X view to display the following:



Use the command again and choose the first isometric view.

Use **Scene points**  command: tick “Face centres” and un-tick “Sides mid-points”.



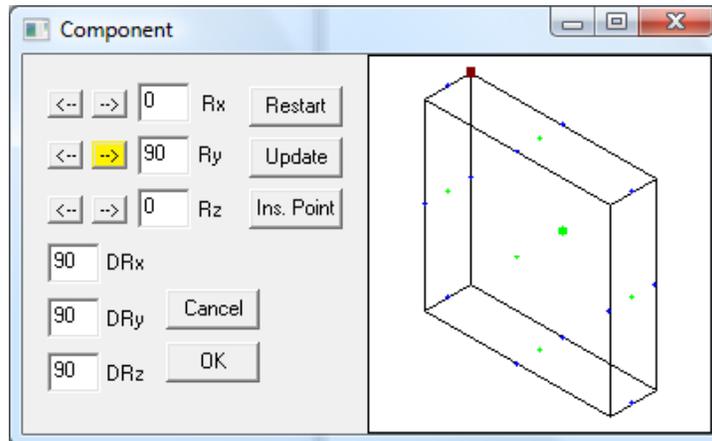
Since this renode is a constraint, we need a constraint block (it can be the foundation, a wall, etc.).

To add the constraint block use **Renode – Components - Add through** , then choose constraint block (last button).

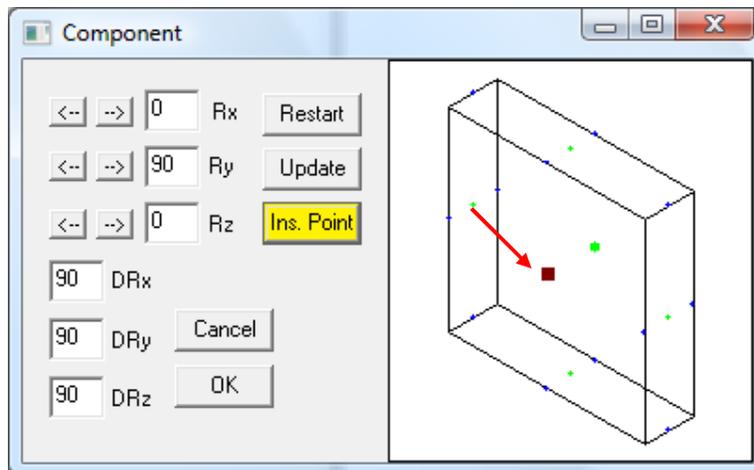
Type Height=1000, Length=1000 and Thickness =250, then press OK.



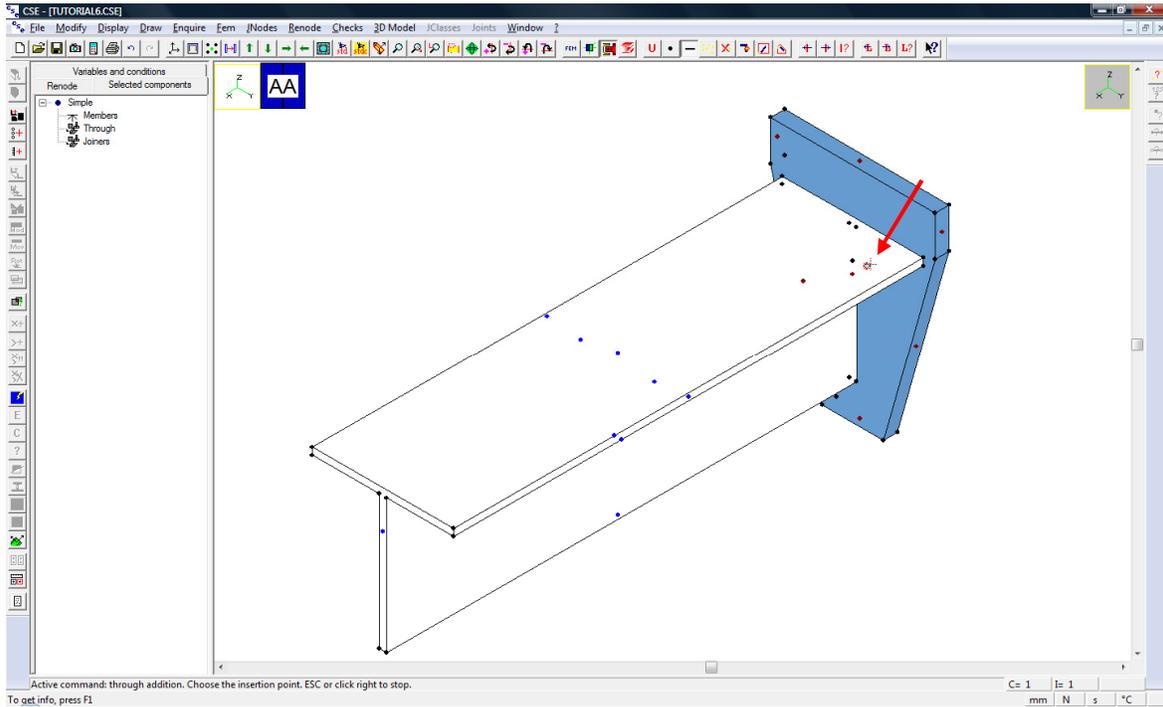
In the following dialog press once the → button in Ry row to place the constraint block in its correct position.



Press “Ins. Point” button and click the point indicated by the arrow in next figure, then press OK.

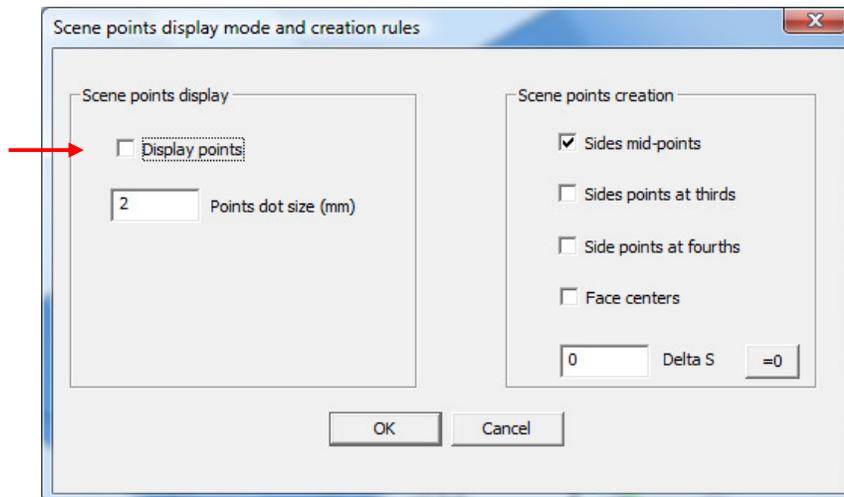


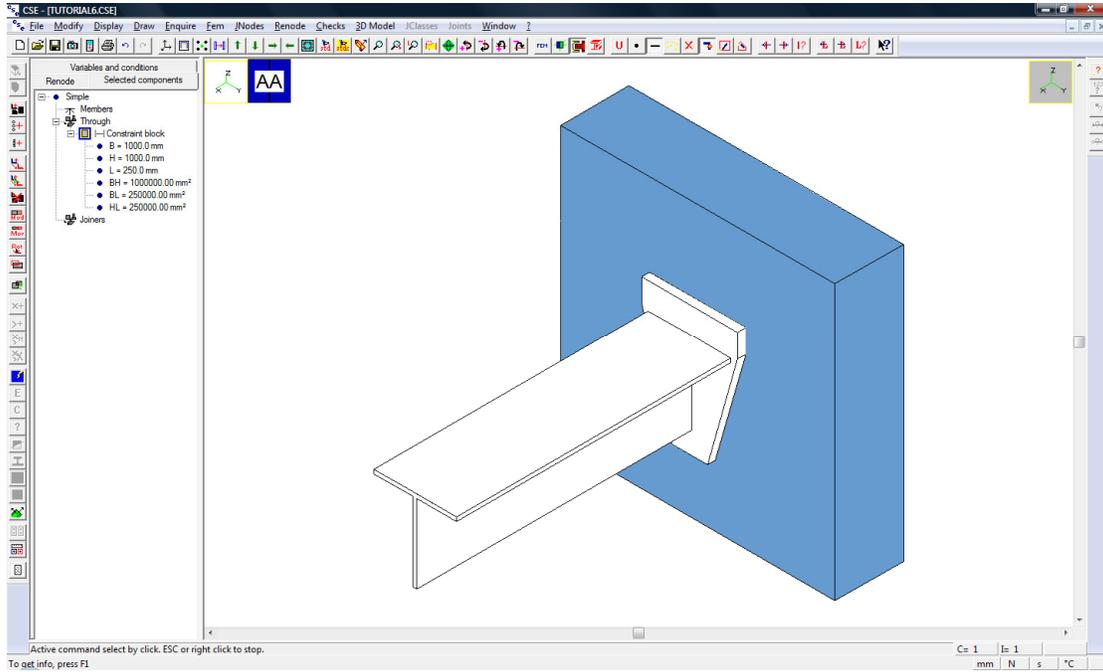
In the scene, click the point to which must correspond previously chosen point.



Constraint block has been added.

Enclose the structure  and hide **Scene points**  because they are not necessary.

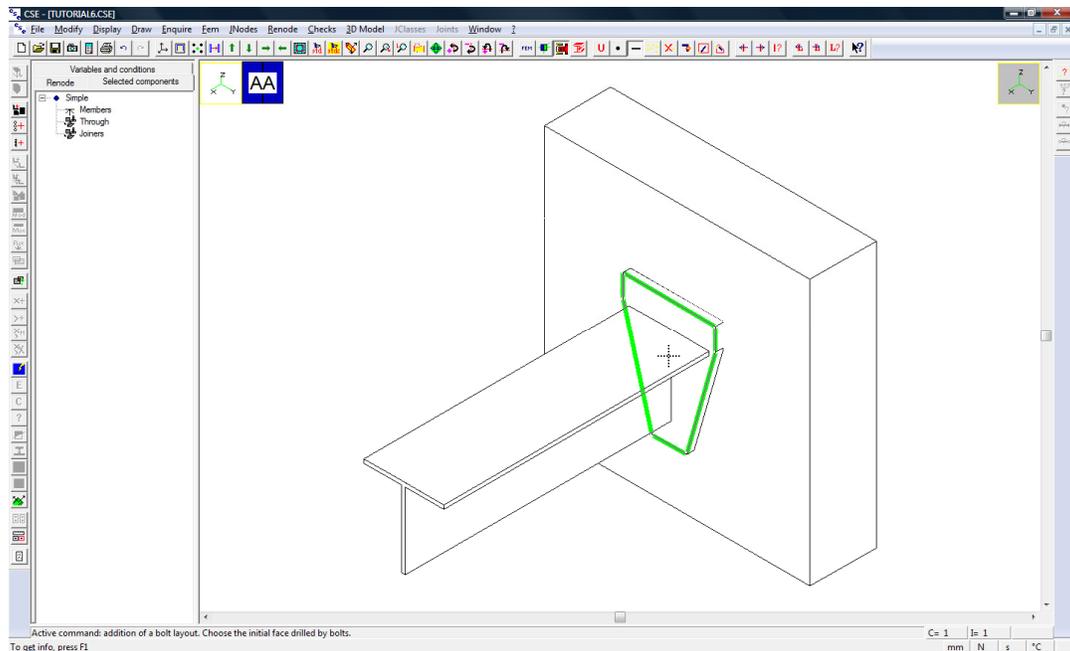


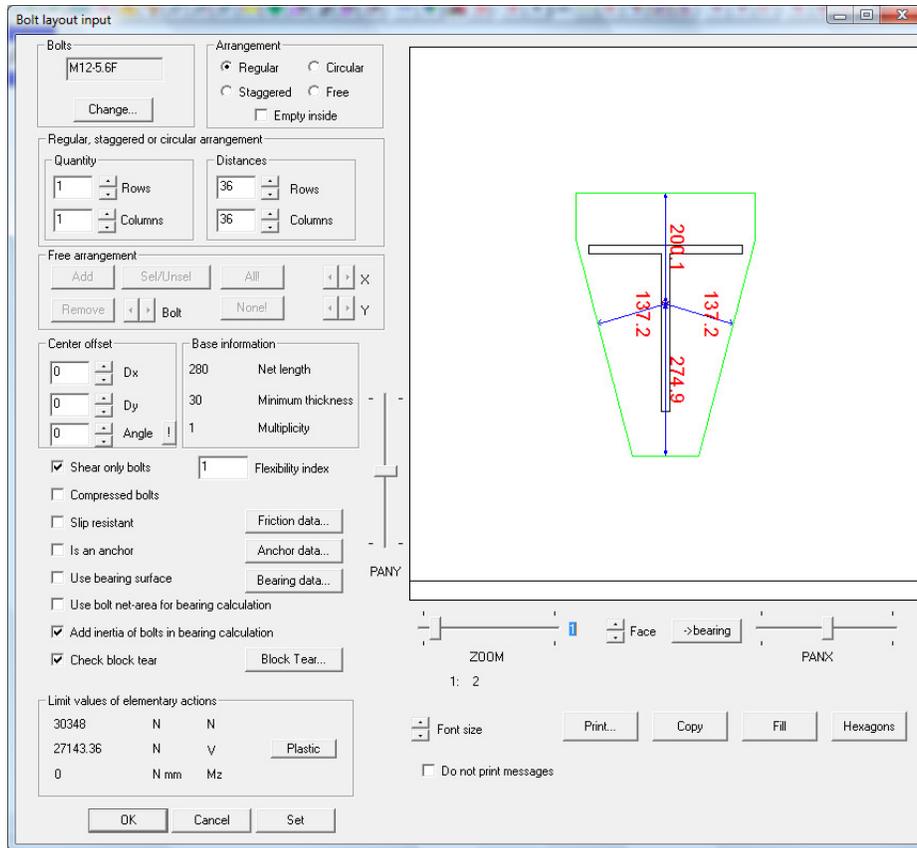


Unselect all the components .

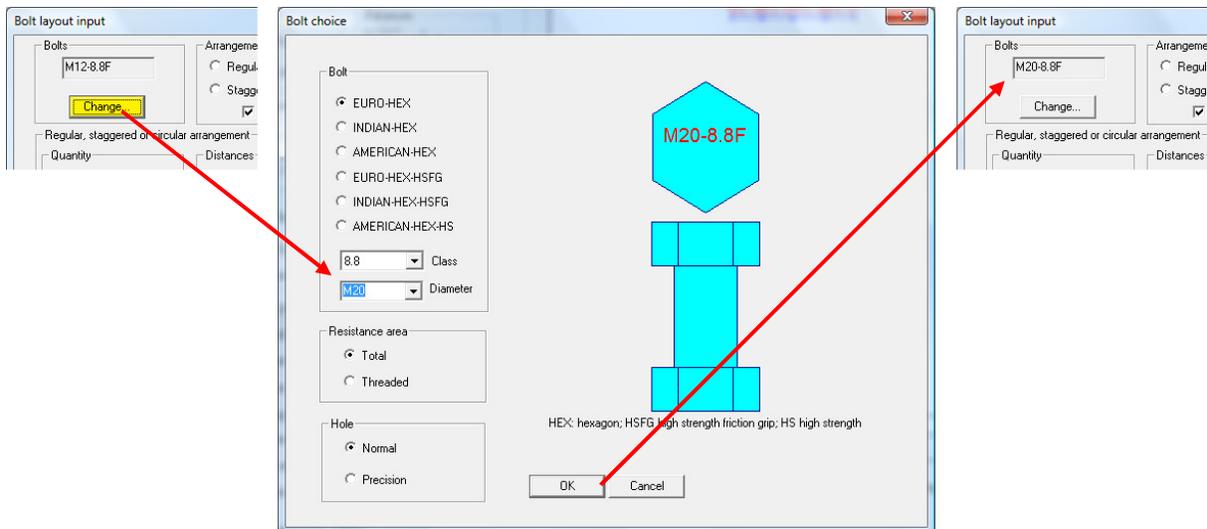
2.5.2 Addition of bolt layout

Use **Renode – Components – Add bolt layout**  and click in the scene the front face of the plate.





Click the “Change” button in top left part of the dialog box, then choose 8.8 class and M20 diameter in the new dialog box. Press OK to return in previous dialog.



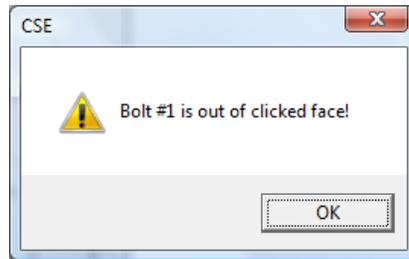
Keep the regular arrangement, define 3 rows and 3 columns using the proper arrows. Type ‘190’ as rows distance and ‘130’ as columns distance. The image is updated in real time.



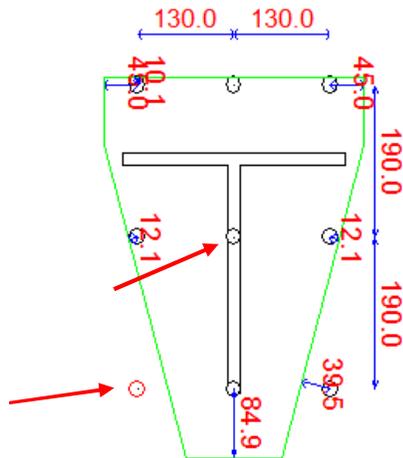
Regular, staggered or circular arrangement

Quantity	Distances
3 Rows	190 Rows
3 Columns	130 Columns

When you type 130 as distance between the columns, the following message appears:



Click OK, and you will see that there are bolts out of plate borders. It is not a problem because we are going to remove those bolts (tick “Do not print messages” box to inhibit warning messages).

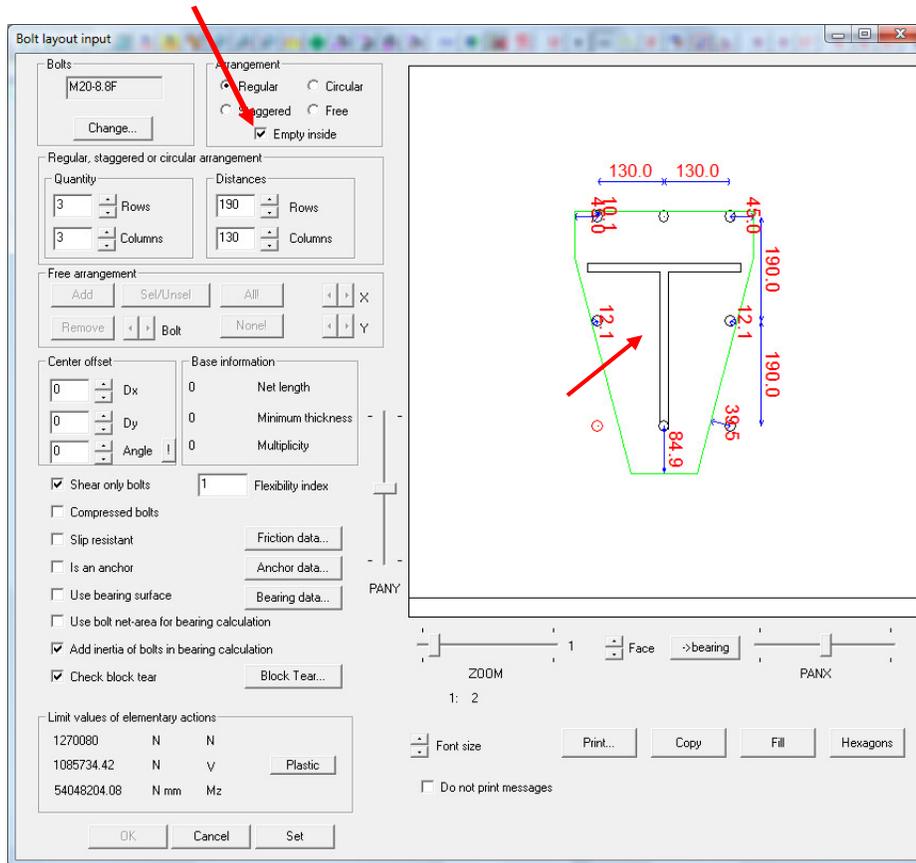


Previous image was obtained pressing “Copy” button in the dialog box.

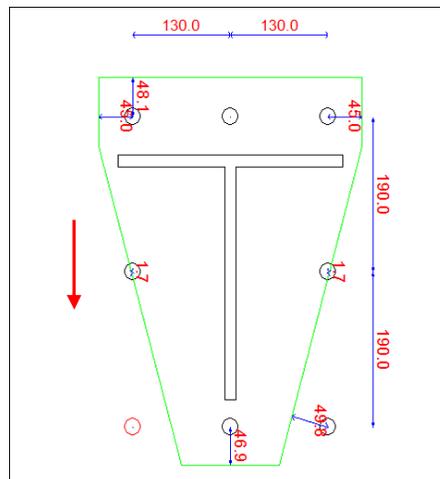
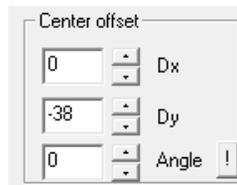
Note that it is not possible to insert bolt layout now, because of the incorrect bolts (there is also a bolt in correspondence of member web).



Tick “Empty inside” to remove central bolt.



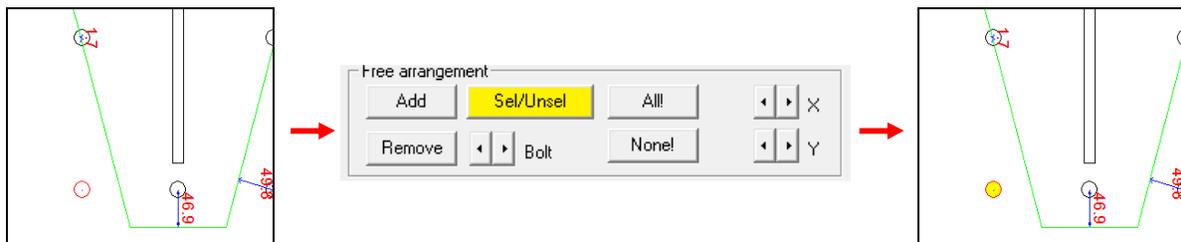
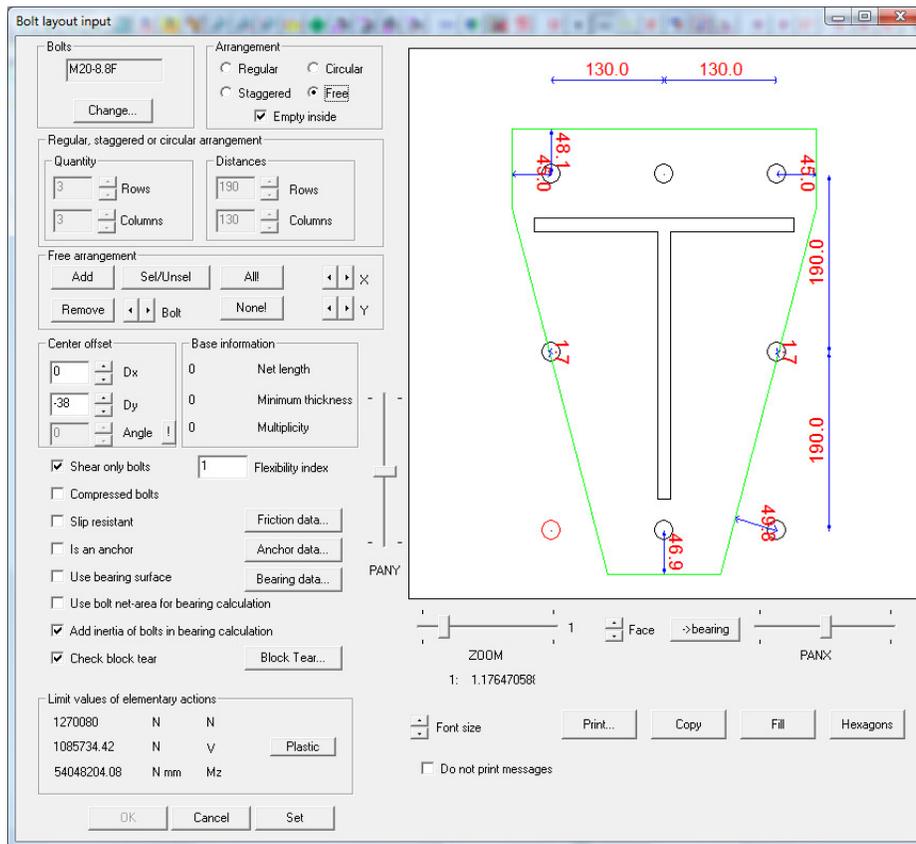
Type '-38' in Dy box of "Center offset" section to move bolt layout 38m downward; if you have problems in typing '-', type 38 then add '-'.



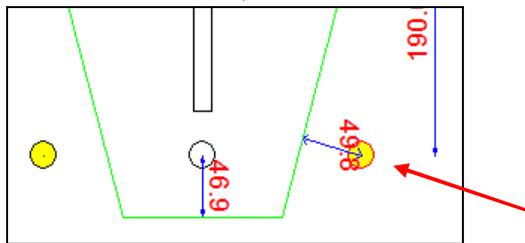
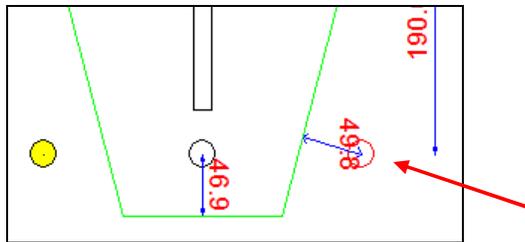
Select *Free* in “Arrangement” section to freely move-add-remove single bolts. Initially, previous bolts disposition is kept.



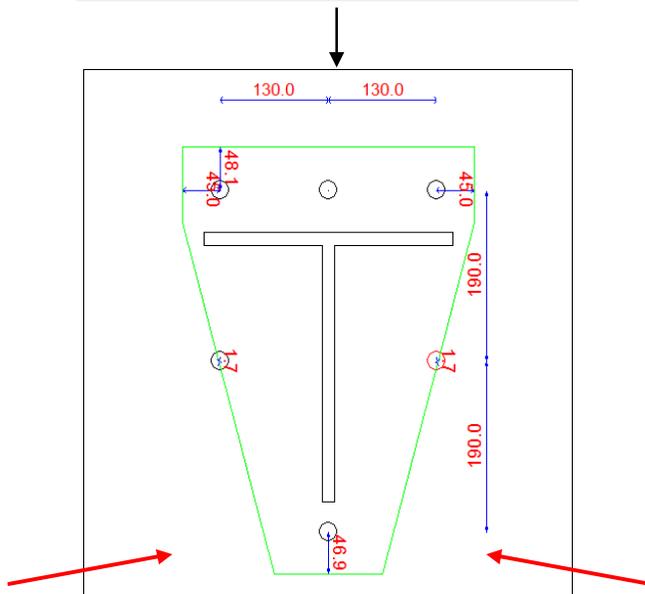
We need to remove bolts out of the plate. Current bolt has red boundary: click *Sel/Unsel* to select it: it becomes yellow.



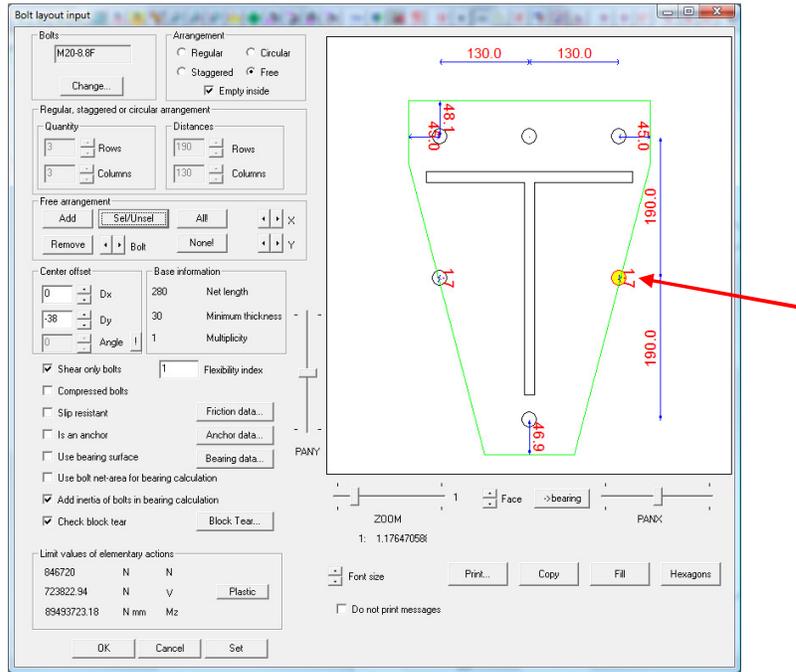
Press *Bolt right arrow* twice to make as current bolt the other bolt out of the plate, then select it.



Click *Remove* button to remove selected bolts.



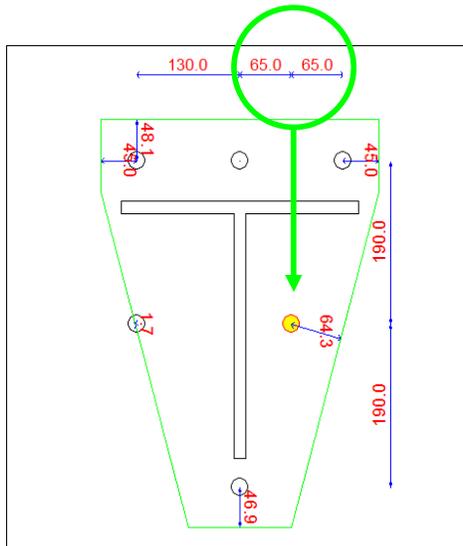
After bolts removal, current bolt is the one indicated in next figure: select it using *Sel/Uysel* button.



Keep *X* left arrow pressed to move selected button leftward.

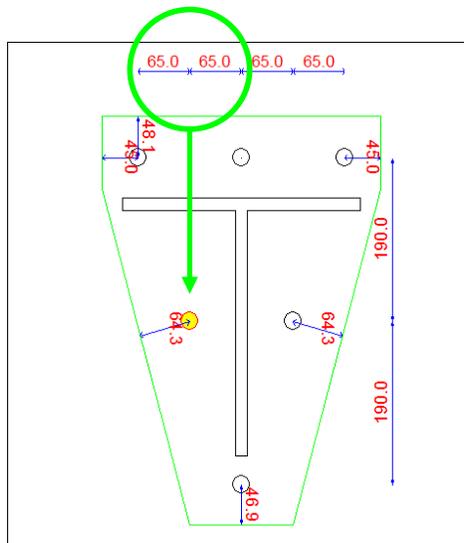


Keep it pressed until bolt is in the middle between two upper bolts (distance = 65mm from both bolts).



N.B.: use *zoom* and *pan* commands to have a closer/full view of the figure.

1. Press *Sel/Unsel* to unselect current bolt (alternatively you can use *None!* to unselect all bolts)
2. Press *Bolt left arrow* to make as current bolt the other bolt we have to shift
3. Press *Sel/Unsel* to select it
4. Keep *X right arrow* pressed to move it (bolt layout must be symmetrical)



Press *None!* button to unselect all bolts (it is not closely necessary).

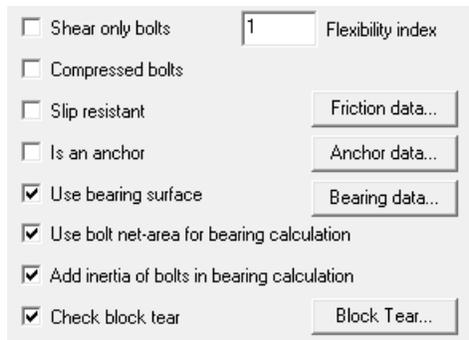
Bolts positioning is now complete; you can **Copy** the figure with the proper button, show/hide bolts **hexagons** and **fill** faces with different colours.

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

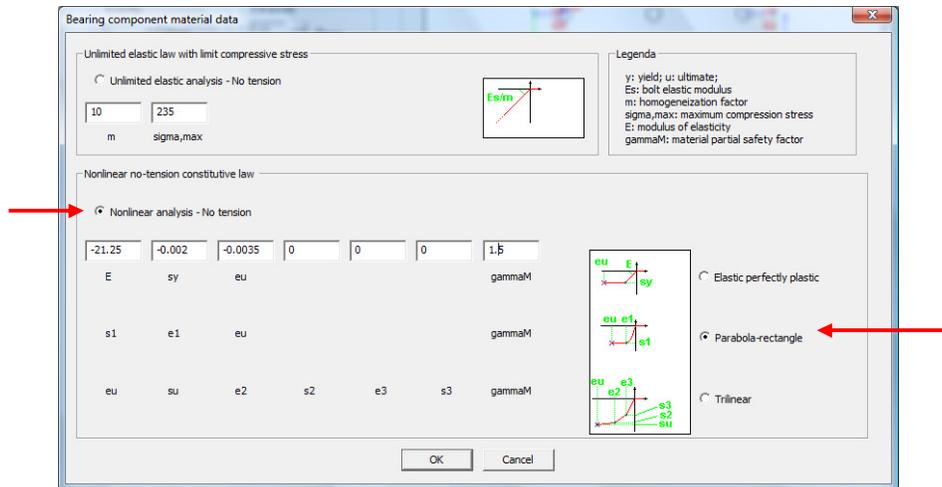
Note that if bolts distances or distances from sides are too small for chosen standard, a warning message will appear then you try to insert the layout in the scene.

Now define the following options:

- remove tick from “Shear only bolts” because they must work also under axial forces
- do not tick “Compressed bolts” because compression will be carried by bearing surface
- do not tick “Slip resistant” because this is not a no-slip bolting
- do not tick “It is an anchor”
- tick “Use bearing surface”
 - tick “Use bolt net-area for bearing calculation”
 - tick “Add inertia of bolts in bearing calculation”
- tick “Check block tear” to include block tear in the checks



Press *Bearing data* button do set parameters to define parameters of compressed object material. Select “Non-linear analysis – No tension” and “Parabola-rectangle”.

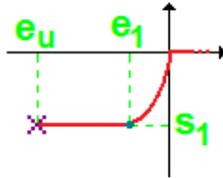


Type the following values:

-21.25 in s_1 box -0.002 in e_1 box -0.0035 in e_u box 1.5 in γ_M (gammaM) box

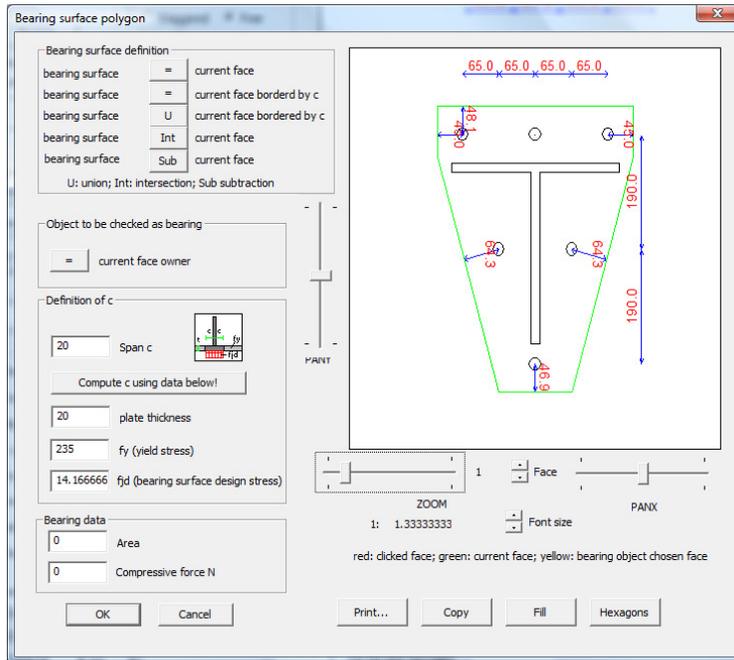


Previous settings define the following constitutive law (units are N and mm):



This will be the constitutive law of the constrain block (which simulates a concrete wall) in bearing computation.

To define bearing surface, press \rightarrow *bearing* button under the figure; you will get the following dialog box:



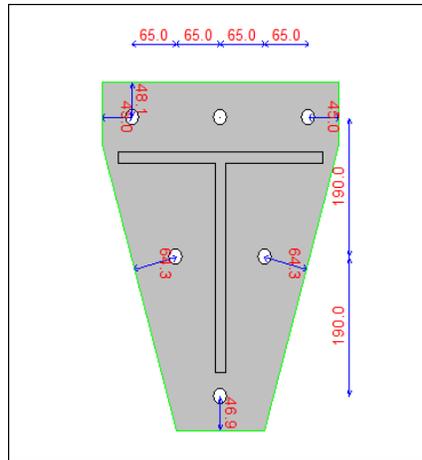
On the left there are the controls to define the surface (or the surfaces). On the right, in the image, plate boundary is currently selected face. If plate is stiff enough to assume the it is rigid, press the following button:

Bearing surface definition

bearing surface	=	current face
bearing surface	=	current face bordered by c
bearing surface	U	current face bordered by c
bearing surface	Int	current face
bearing surface	Sub	current face

U: union; Int: intersection; Sub subtraction

Current surface is assumed as bearing surface.



If plate is not stiff enough, you need to define an area near to cross-section imprint in which stress is diffused. To do that, first of all you need to compute the “thickness” that must be added to the cross-section.

Type 30 in “plate thickness” box; f_y is the value of plate yield stress, so keep it; f_{jd} (bearing surface design stress) is automatically computed on the base of previously define values in “bearing component material data dialog box (s_1 and γ_M). Press *Compute c using data below*:

Definition of c

Span c 

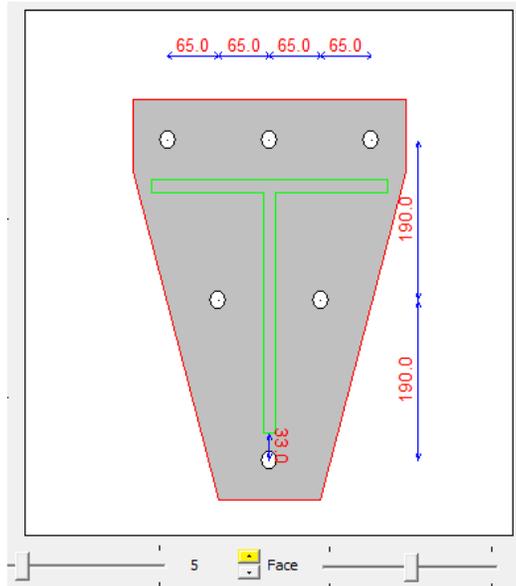
plate thickness

f_y (yield stress)

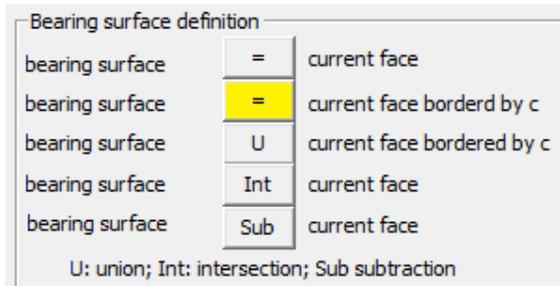
f_{jd} (bearing surface design stress)

CSE automatically computes the thickness that must be added to cross-section.

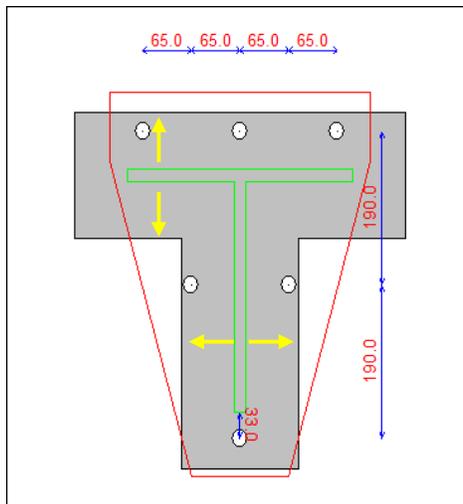
Keep *Face upper arrow* pressed until cross-section becomes current face.



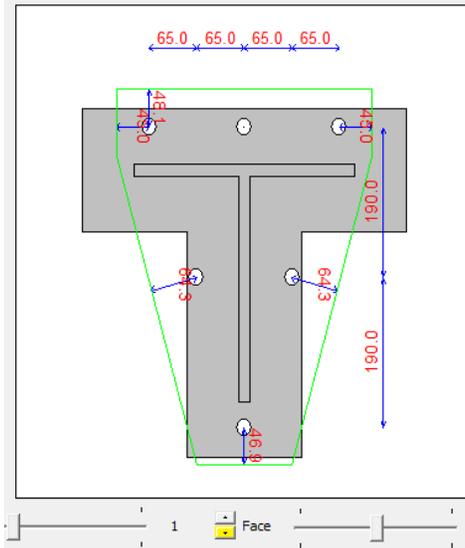
Click the following button to get current face as bearing surface, with the addition of a border with a thickness equal to $c=71$ mm, that is previously computed parameter.



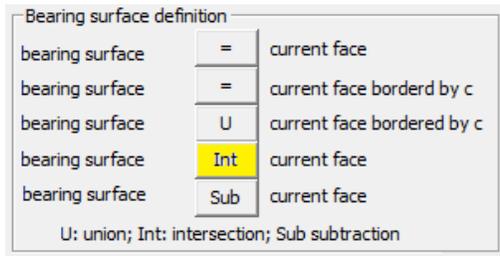
This is what you get; yellow arrows added to the figure show the addition to shape borders (c):



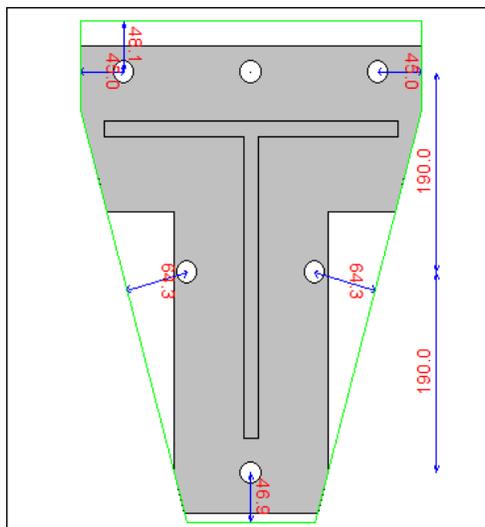
Bearing surface is out of plate boundary: select plate face by keeping Face lower arrow pressed until plate boundary becomes green.



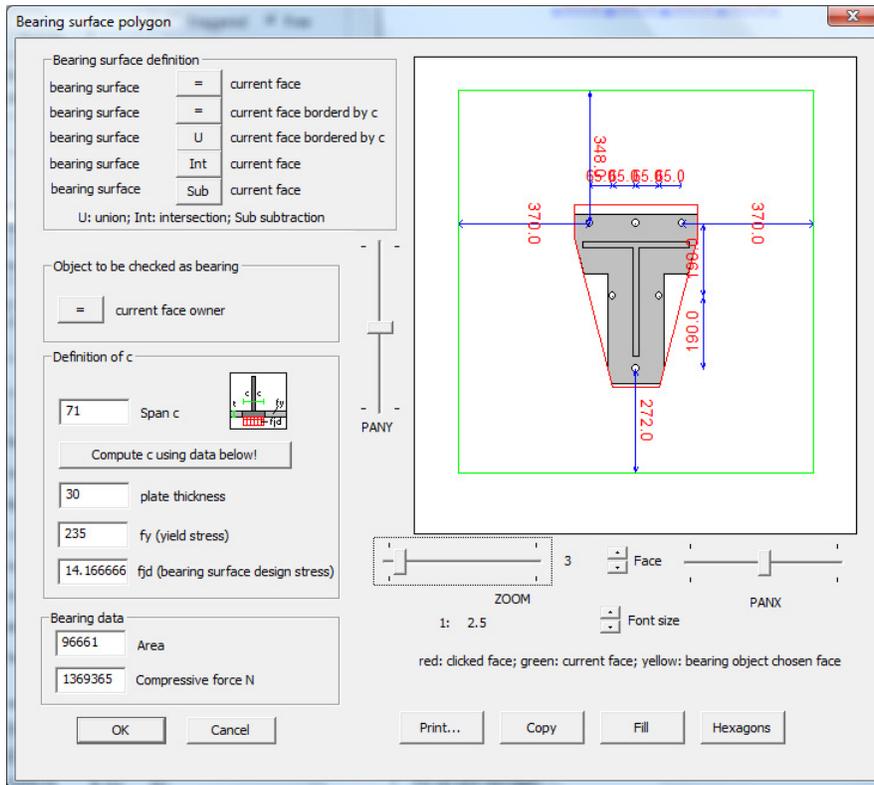
Press the following button to get as bearing surface the portion of current bearing surface included in plate boundary.



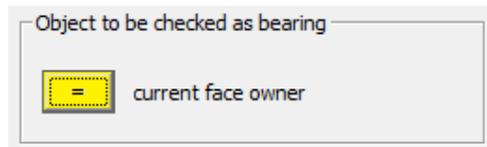
This is the bearing surface we needed:



Finally, we have to define the component on which compression checks must be done. Select face 3 or 4 and zoom out.

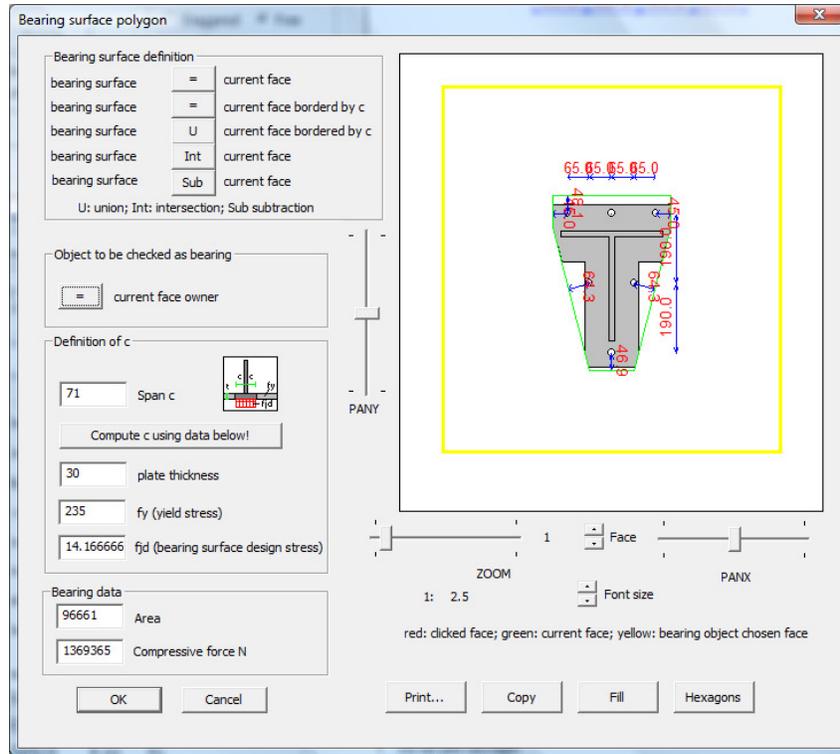


Click the following button.

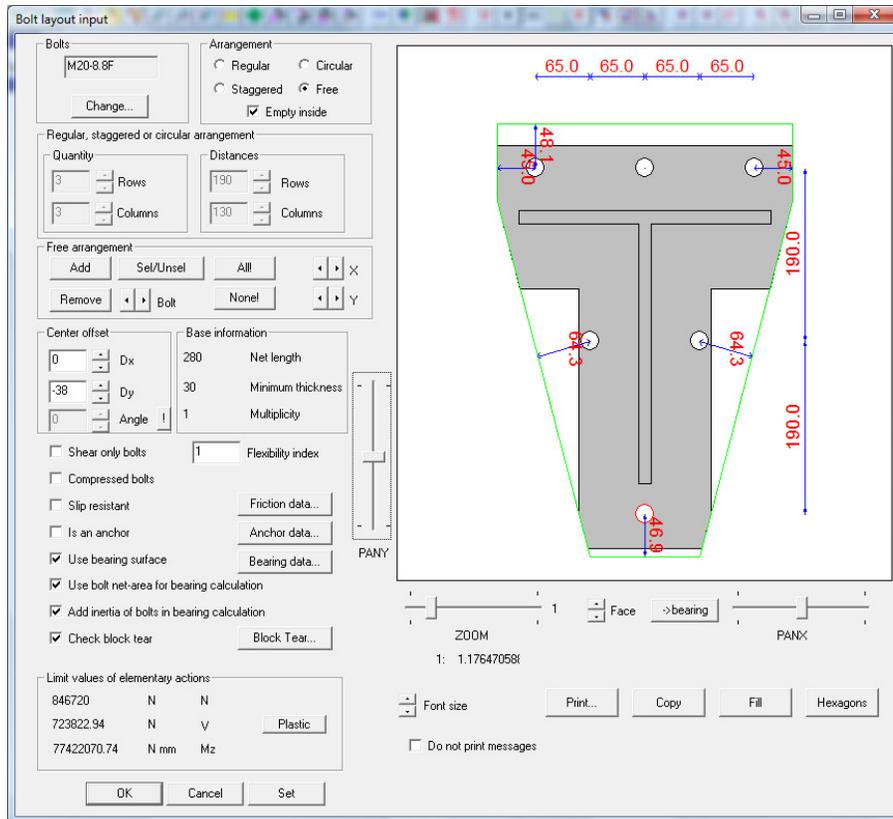


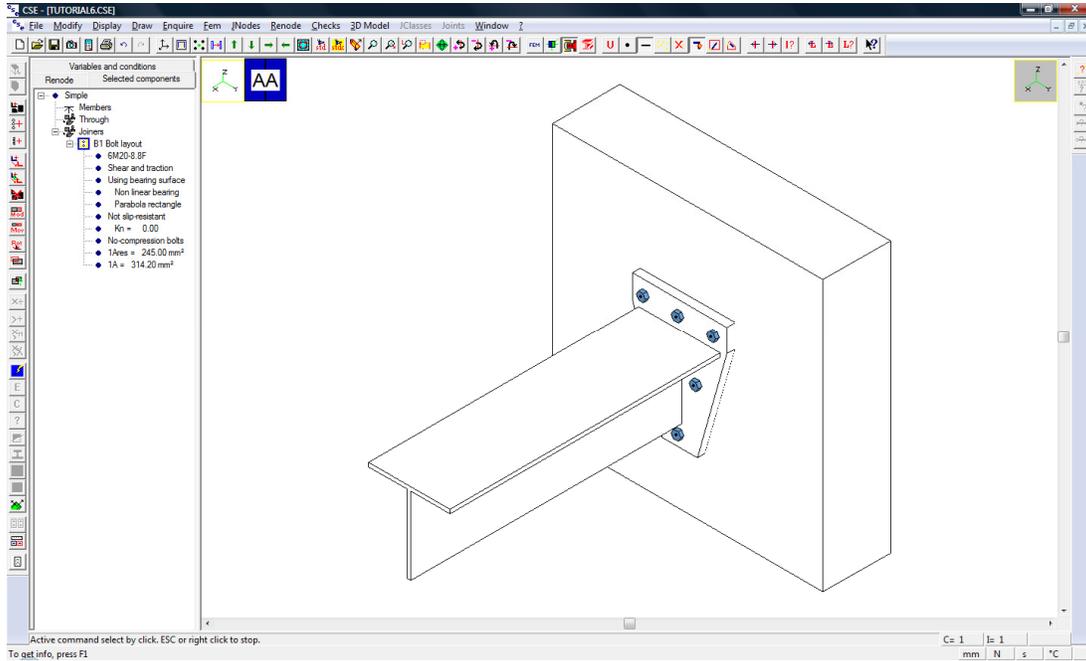
Now, if select any face but face 3 or 4, you will see constraint block boundary in yellow, because it is the object to be checked.

Press OK to keep this bearing surface and get back in main bolt layout addition dialog box.



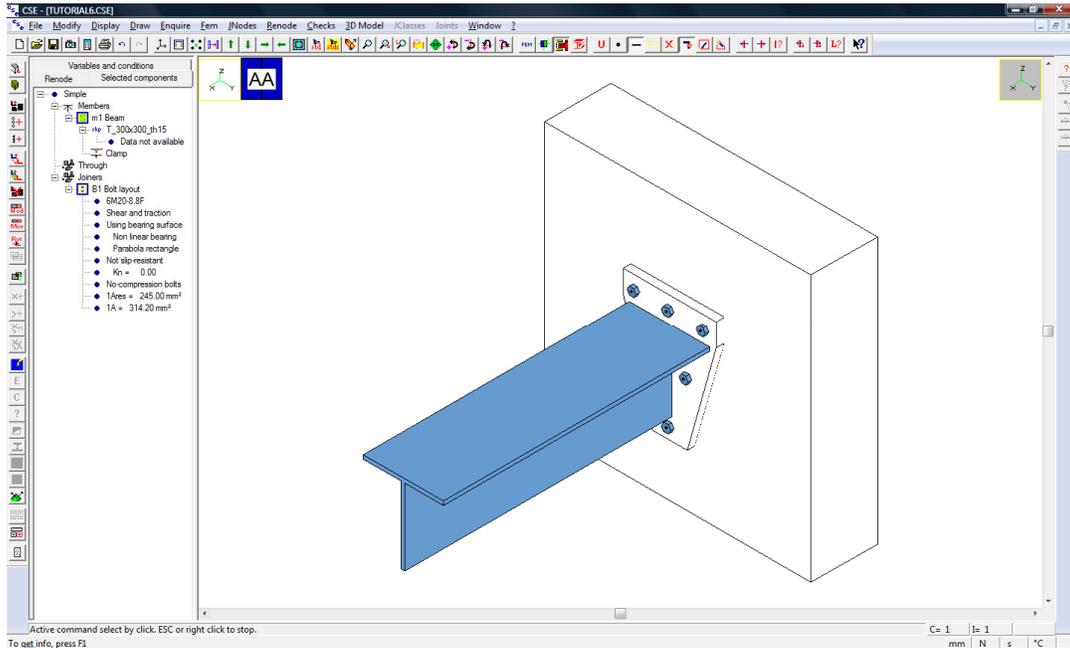
Bearing surface is now displayed in main dialog box too; press OK to add bolt layout.



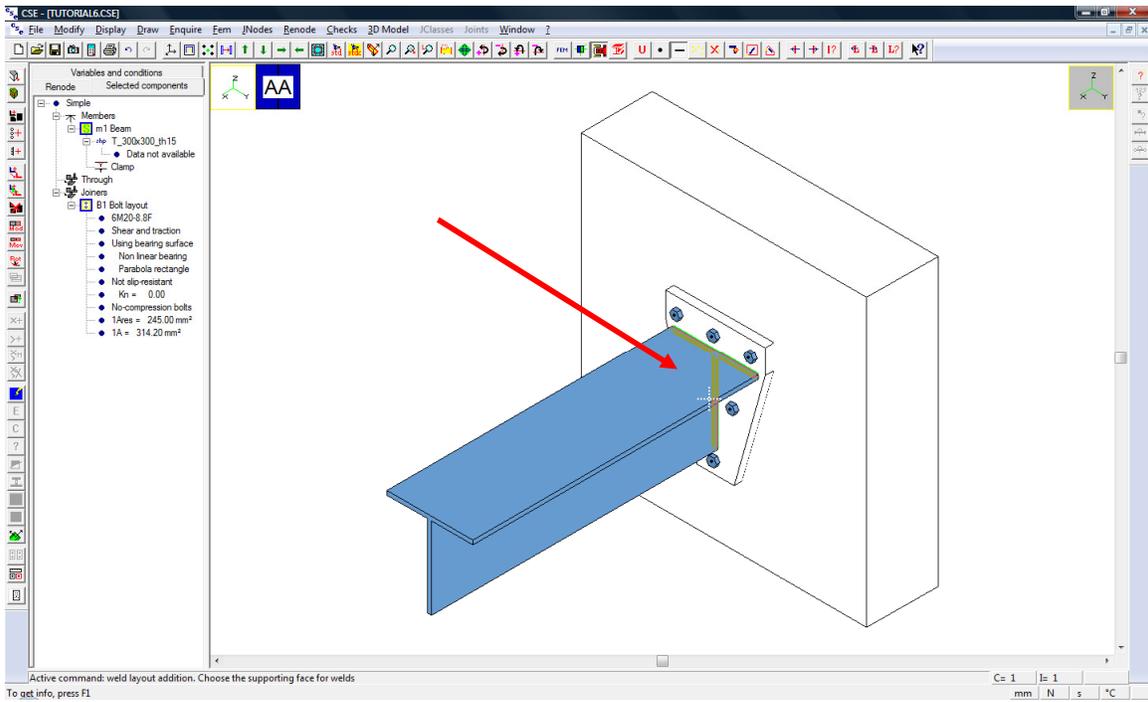


2.5.3 Addition of weld layout

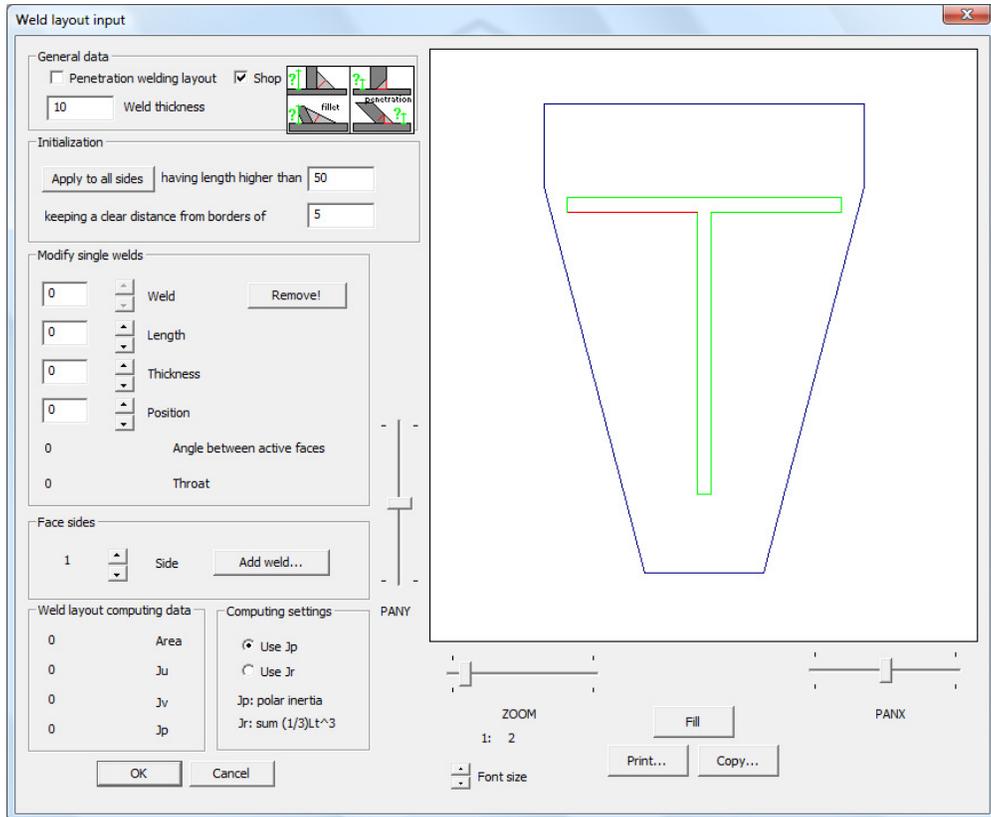
Before adding weld layout, select member in order to limit the selection of the face to be welded to selected components only (alternatively you could unselect bolt layout and all the faces will be available).



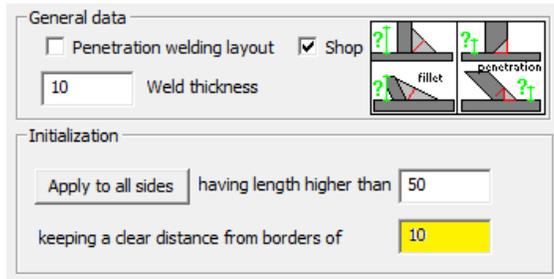
Use **Renode – Components – Add weld layout** , then click in the scene member end face.



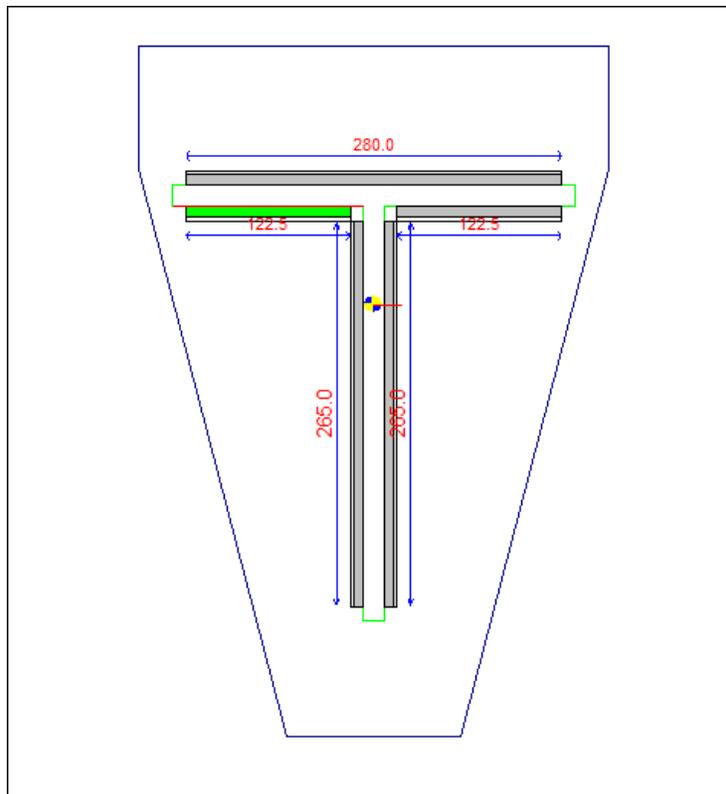
The following dialog box appears:



Do NOT tick “Penetration welding layout” because we are going to add a fillet welds layout.
 Type 10(mm) in “keeping a clear distance from borders of” box; keep other default parameters.



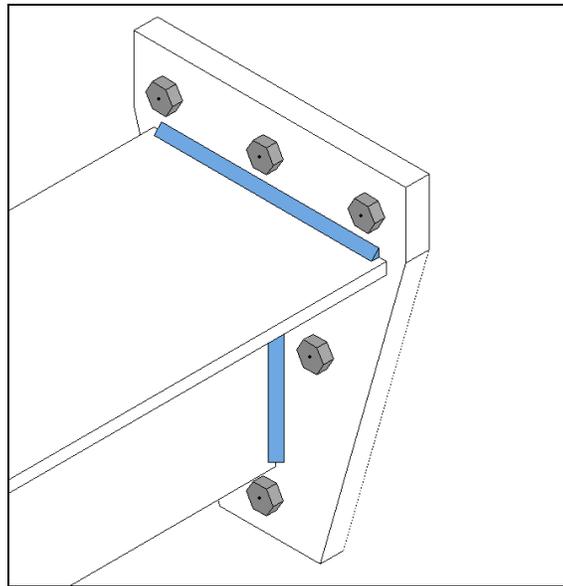
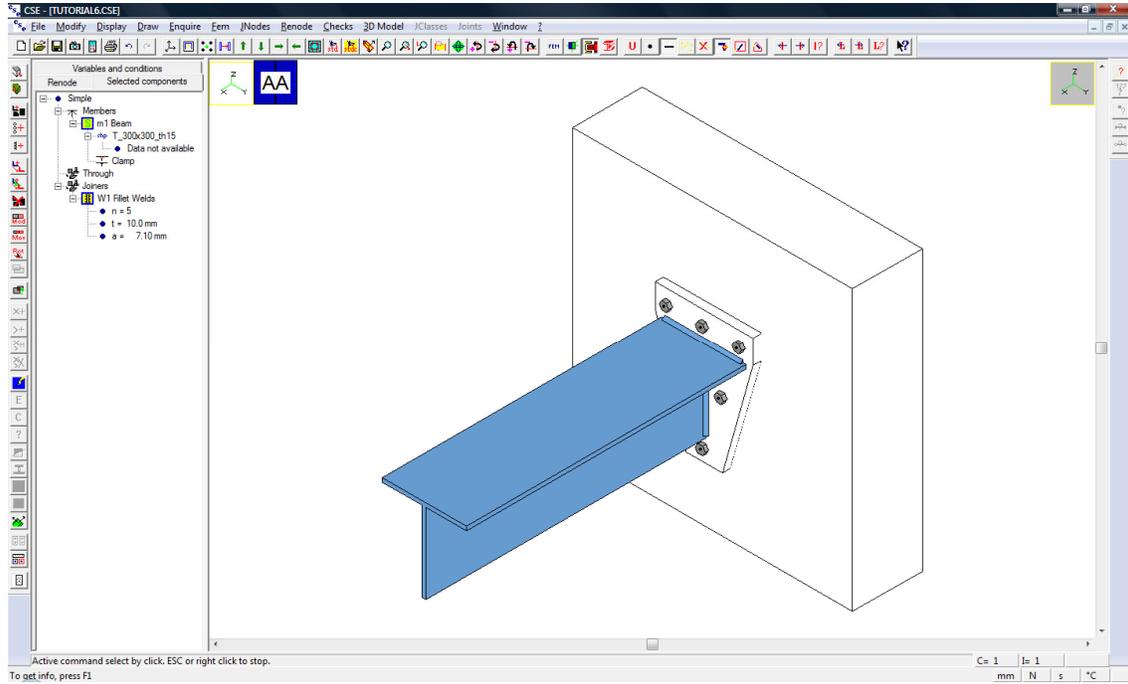
Press *Apply to all sides* button to add a fillet weld on all the sides having a length higher than 50(mm). Welds thickness will be equal to 10(mm). You will get the following.



N.B.: Modify single welds section let you modify welds position, length and thickness; Add weld button let you add single welds on current side; see the guide for more information.

Keep default parameter for torsion computation (*Use J_p*).

Press OK to add the layout.



(detail)

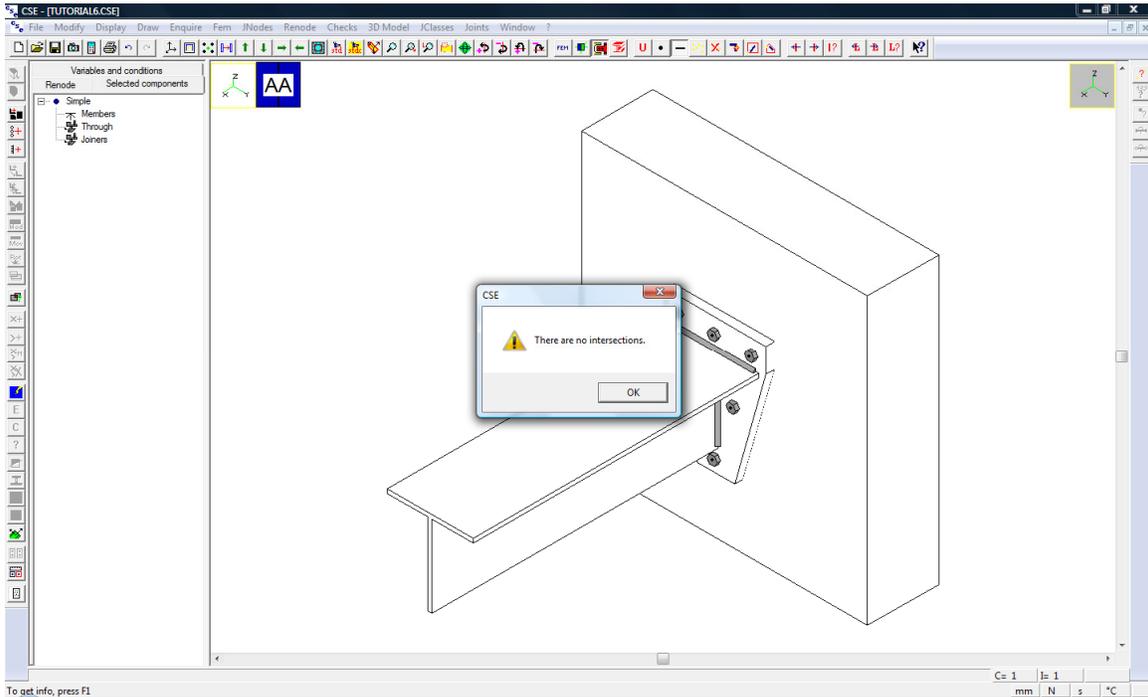
Use commands in **Draw** menu to see the structure from other points of view, to zoom, etc.



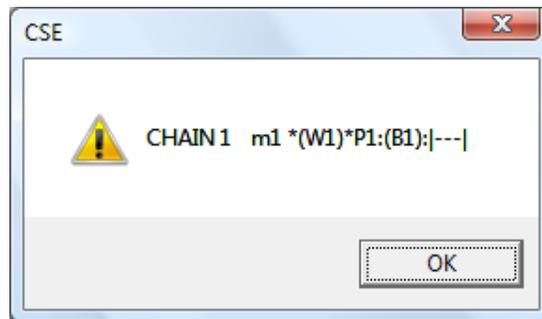
Use standard view  and select first isometric view to get “usual” view.

2.5.4 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 a single path (chain) only from member to constraint block. This path is:

member m1 → weld layout W2 → plate P1 → bolt layout B1 → constraint block |---|

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" or “Defined Values”. Choose elastic limits, 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.

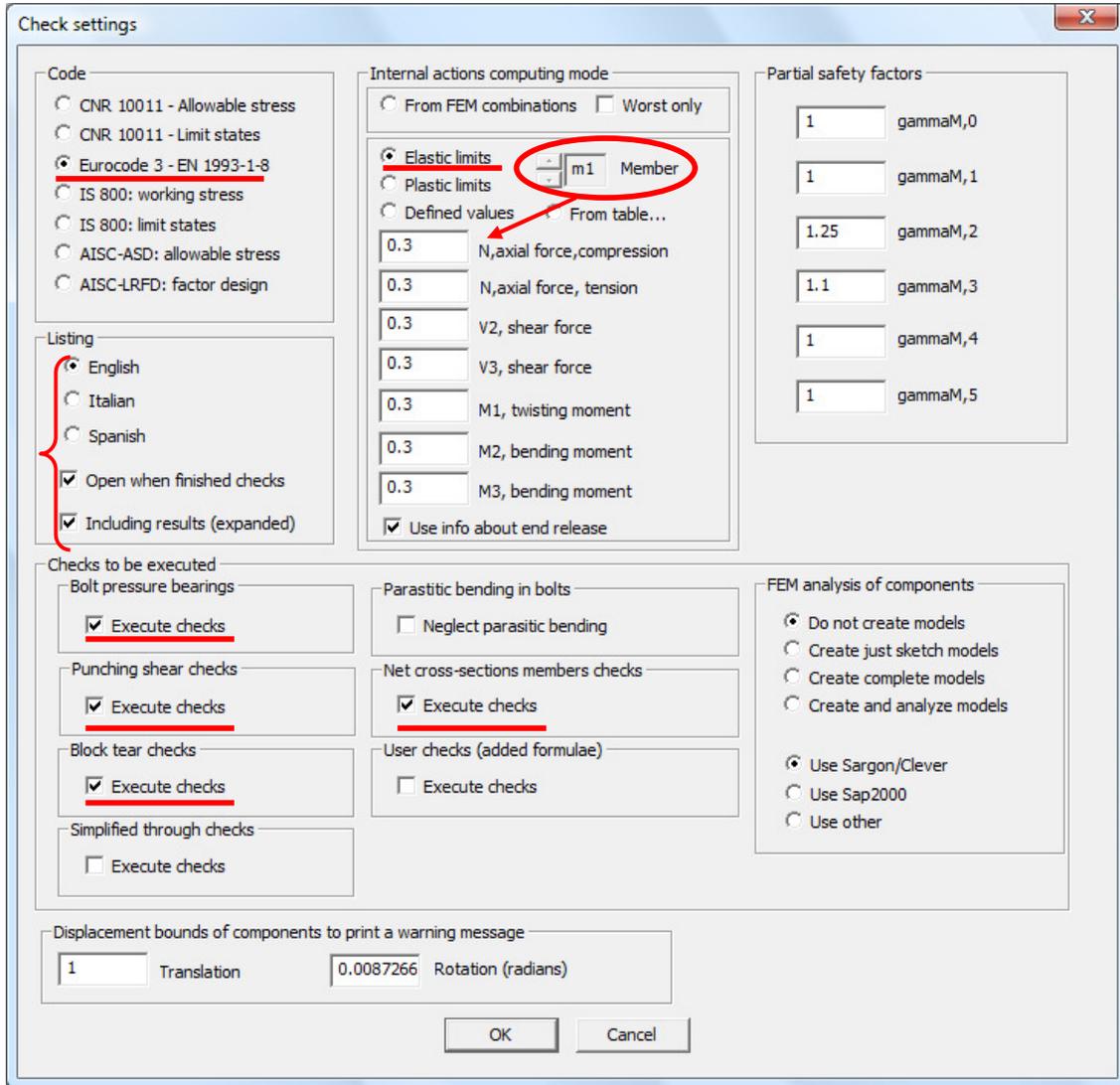
For example, define a multiplier equal to 0.3 for all the limit forces and moments.

Keep default “Partial safety factors”.

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

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

Keep default “Displacements bounds” for displacement checks.



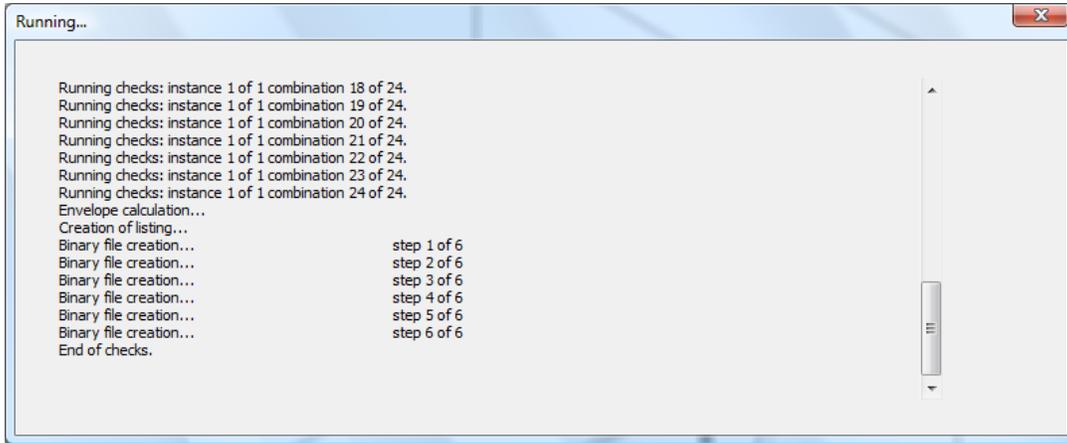
Press **OK** to save settings.

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

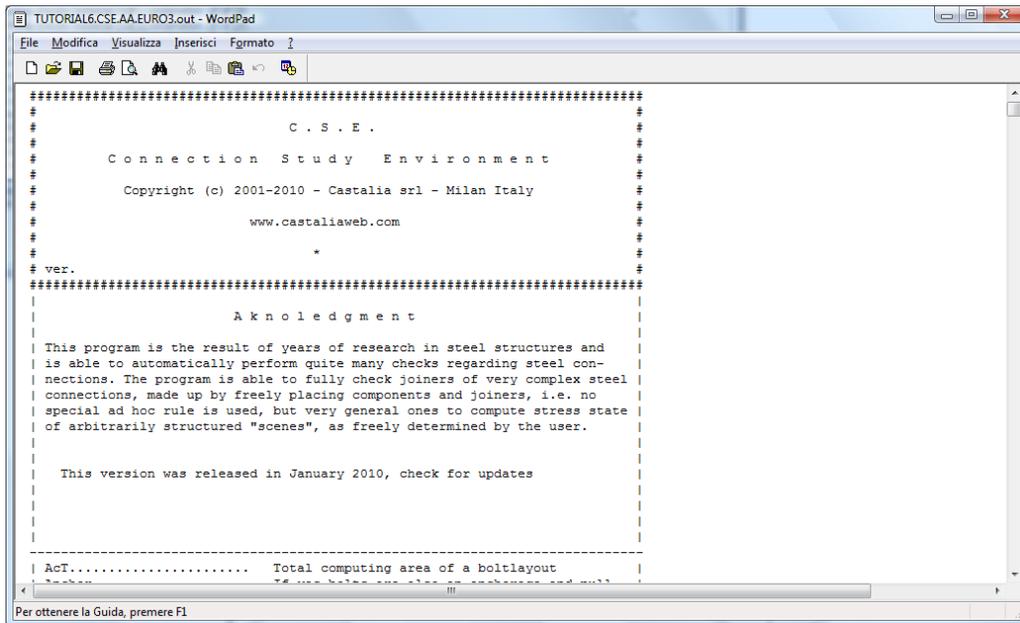
To execute the checks press the following button in the left toolbar (**Checks – Check renode**):





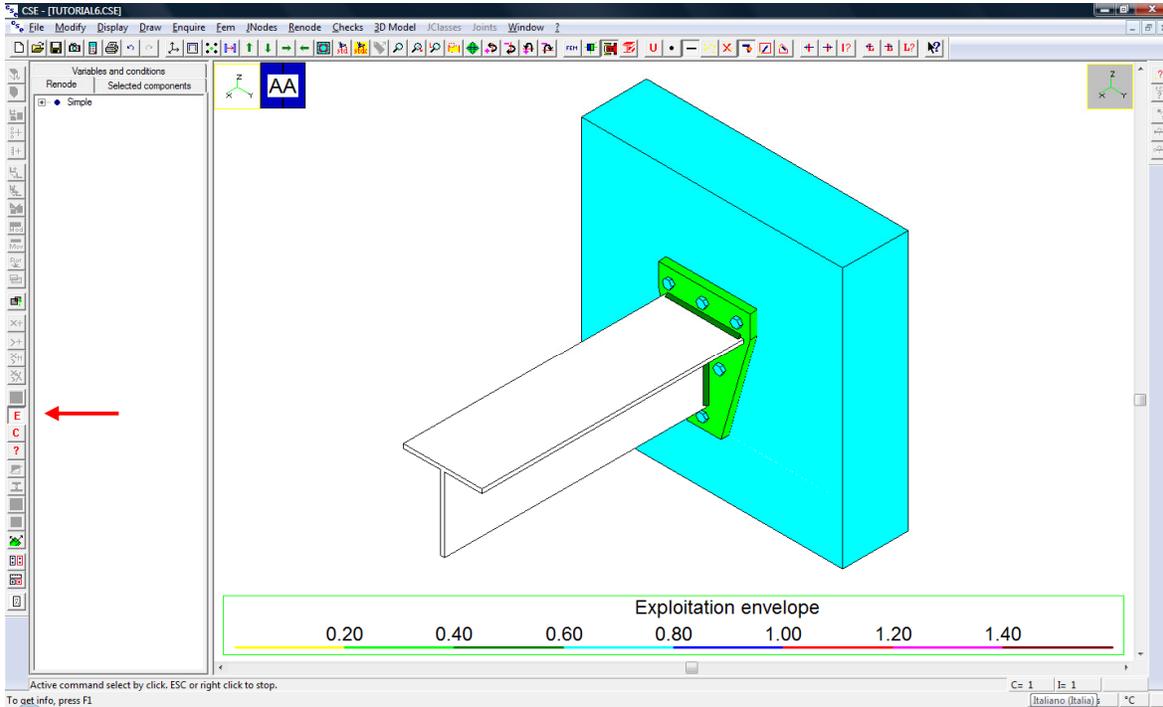
The window above will appear, it is a log window explaining what happens. At the end just close it by clicking over the red-background cross.

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

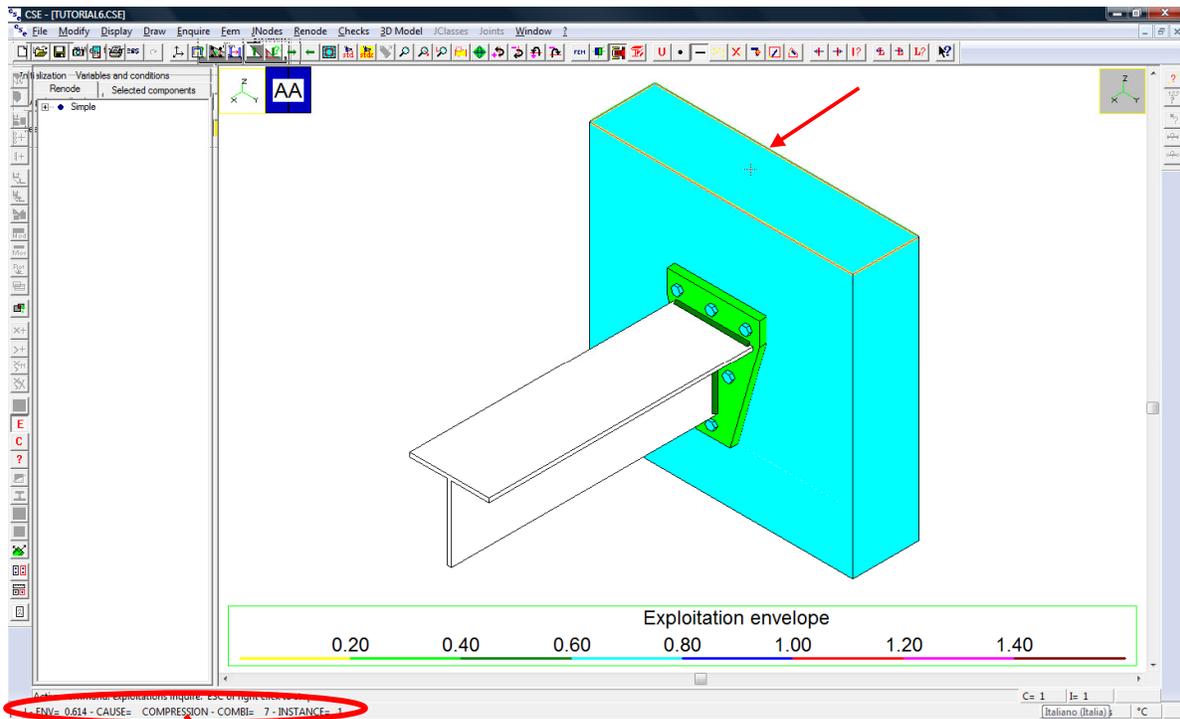


2.8 STEP 8: EXAMINING RESULTS

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.



|---| - ENV= 0.614 - CAUSE= COMPRESSION - COMBI= 7 - INSTANCE= 1 → DIFFERENZA TRA EXPL E RIS. CONTR.

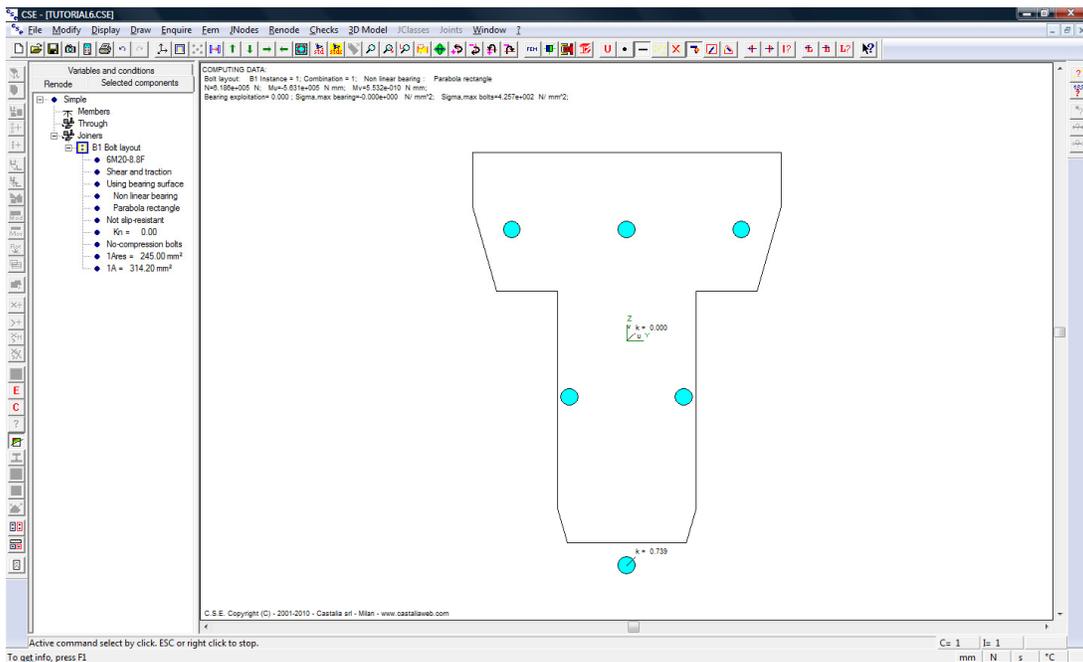
Maximum exploitation for constraint block is 0,614 in combination 7 (compression of the member) and it is due to bearing pressure checks (Cause = Compression).

You can inquire bolts (resistance checks), plate maximum exploitation (only bolt bearing pressure and block tearing checks where executed in this model for the plate; fem model was not required).

If you use **Checks – Current results** , displayed and enquired exploitations refer to current combination.

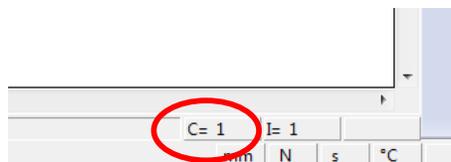
Note that member is white because it has no cuts, bevels or holes for bolts (no bolt bearing pressure checks, block tearing checks or net section checks), and no fem model was required.

Select bolt layout, then use **Checks – Display bearing surface results** . In the graphic view you will see the following.



Since member is in tension there is no compression on bearing surface (it is white). Bolts maximum exploitation is 0,739.

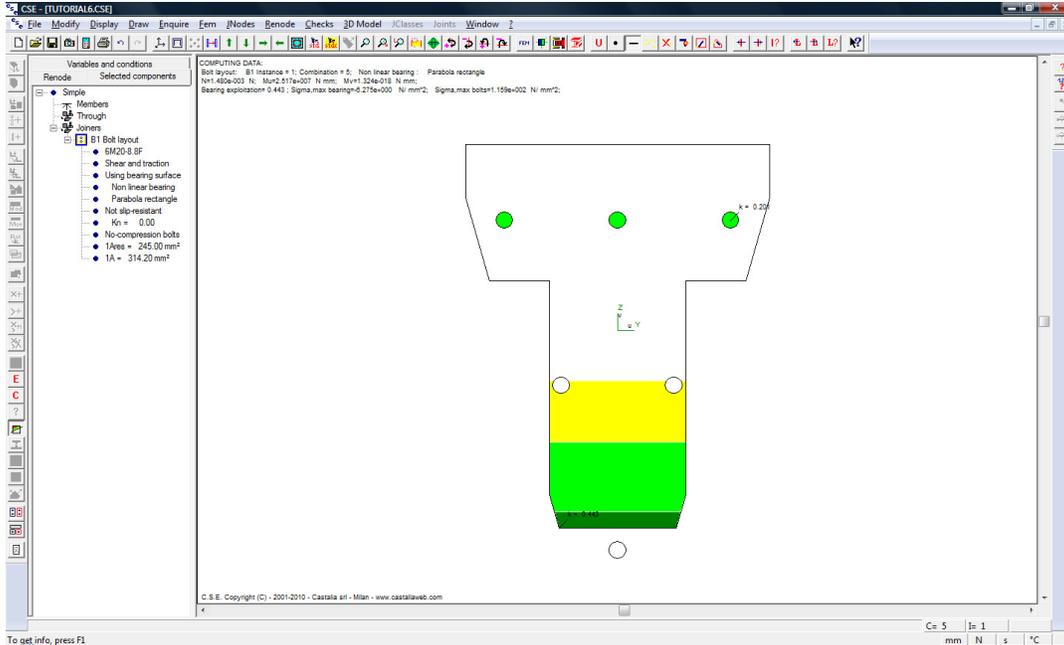
Switch through combinations using . Current combination is displayed in bottom-right part of CSE window.



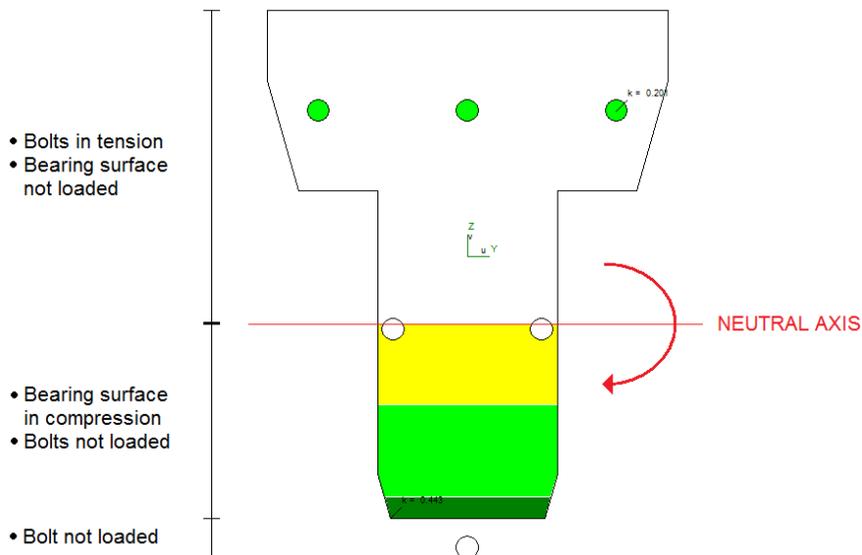
In combination 2 and 3 (shear V2 and V3) we have a small compression because there is an offset between member axis, bearing surface barycentre and bolt layout barycentre.

In combination 4 (twisting moment) there is no compression.

In combination 5 (bending moment M2) there is a significant compression.



Bolts are loaded over the neutral axis only, and bearing surface is loaded under neutral axis only.



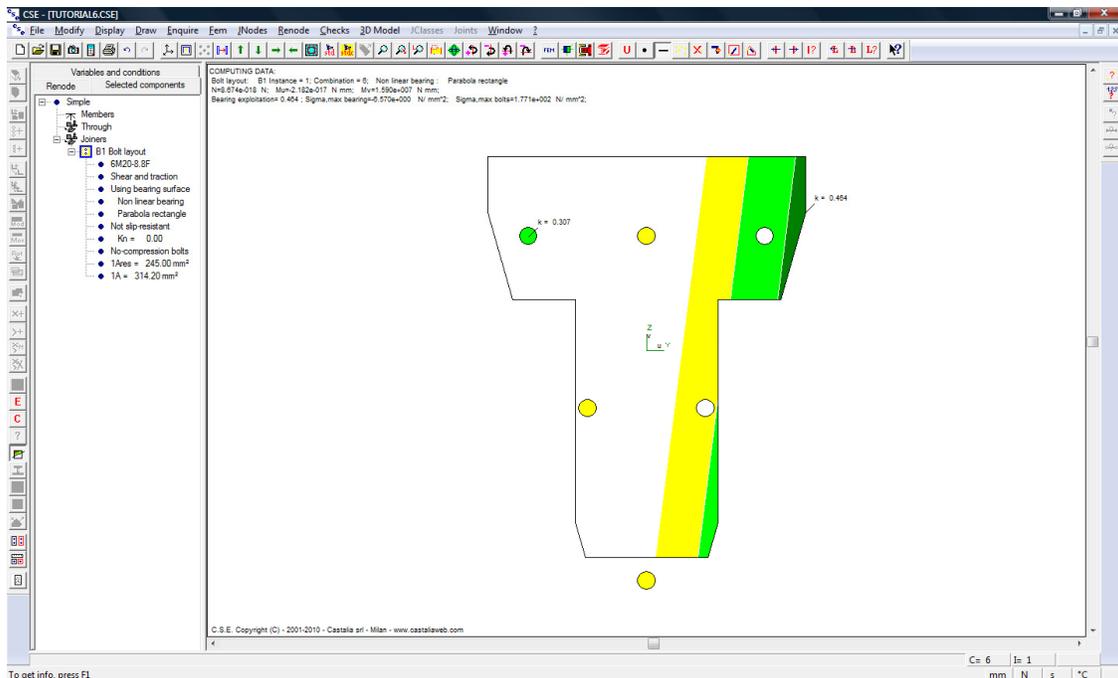
In the top-left part of graphic view you can find the following data:

- Bolt layout name; current instance and combination; kind of bearing surface constitutive law
- Axial force and bending moments
- Bearing surface exploitation and maximum stress; maximum stress in bolts

```

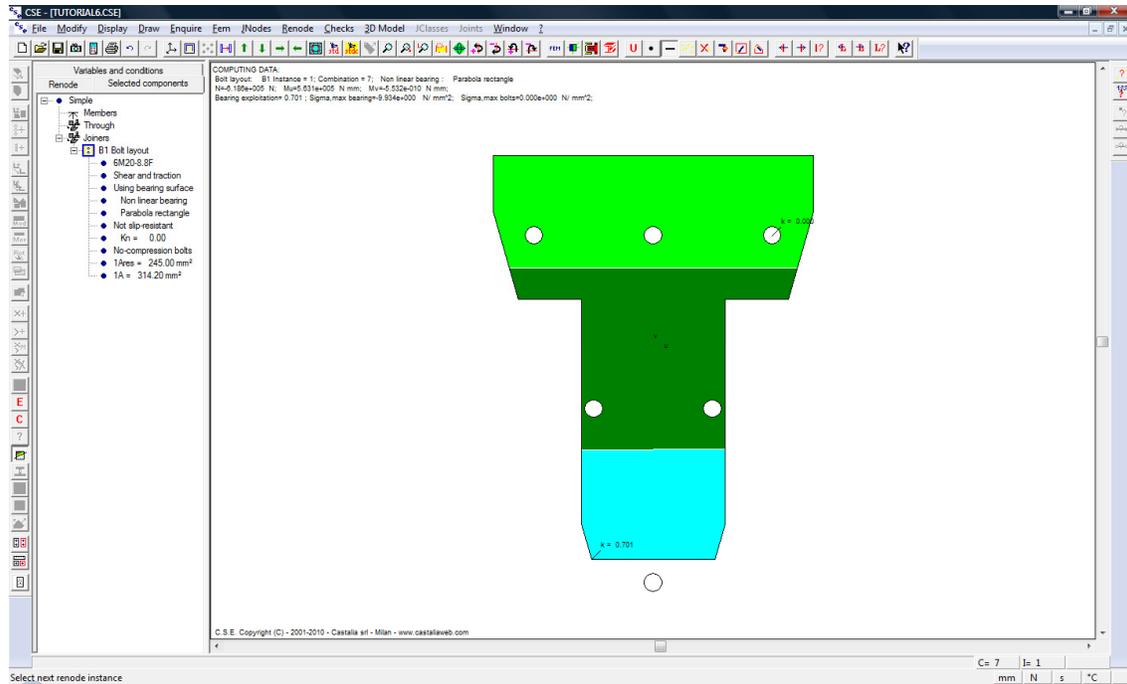
COMPUTING DATA:
Bolt layout: B1 Instance = 1; Combination = 5; Non linear bearing : Parabola rectangle
N=1.480e-003 N; Mu=2.517e+007 N mm; Mv=1.324e-018 N mm;
Bearing exploitation= 0.443 ; Sigma,max bearing=-6.275e+000 N/ mm^2; Sigma,max bolts=1.159e+002 N/ mm^2;
    
```

Switch to combination 6 (bending moment M3): since bolt layout is not symmetric with respect of axis 3, a M3 bending moment gives a neutral axis not symmetric to axis 3.



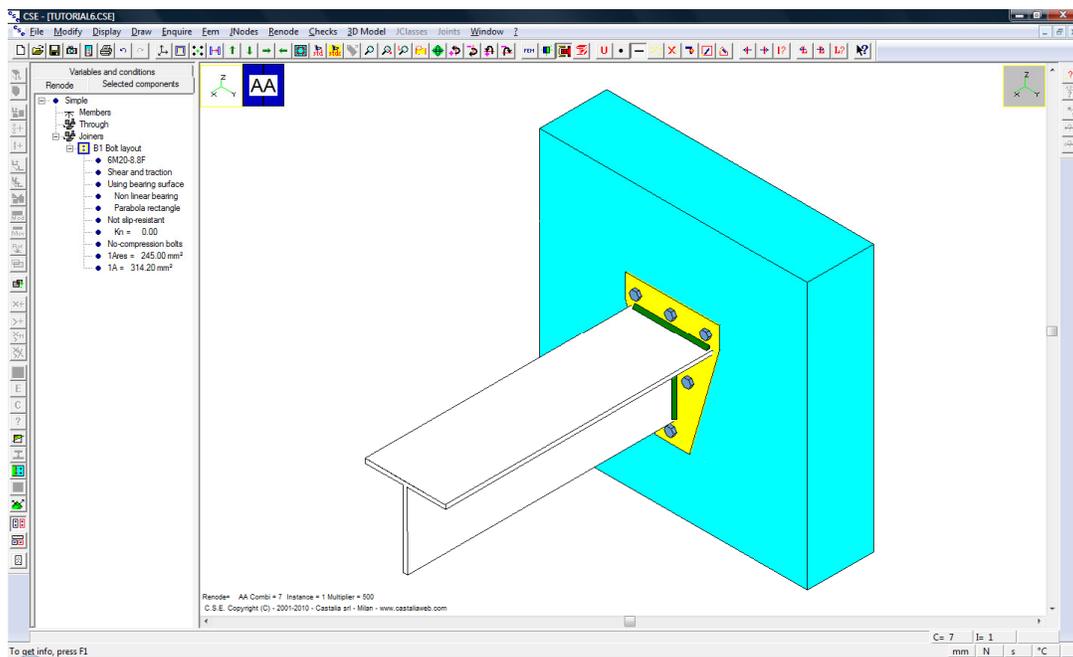
Switch to combination 7, in which we have maximum constraint block exploitation, due to bearing surface check.

Since there is a offset between member axis, bearing surface barycentre and bolt layout barycentre, the lower part of bearing surface is more compressed than the upper part.



You can continue to switch between combinations and see bearing surface results.

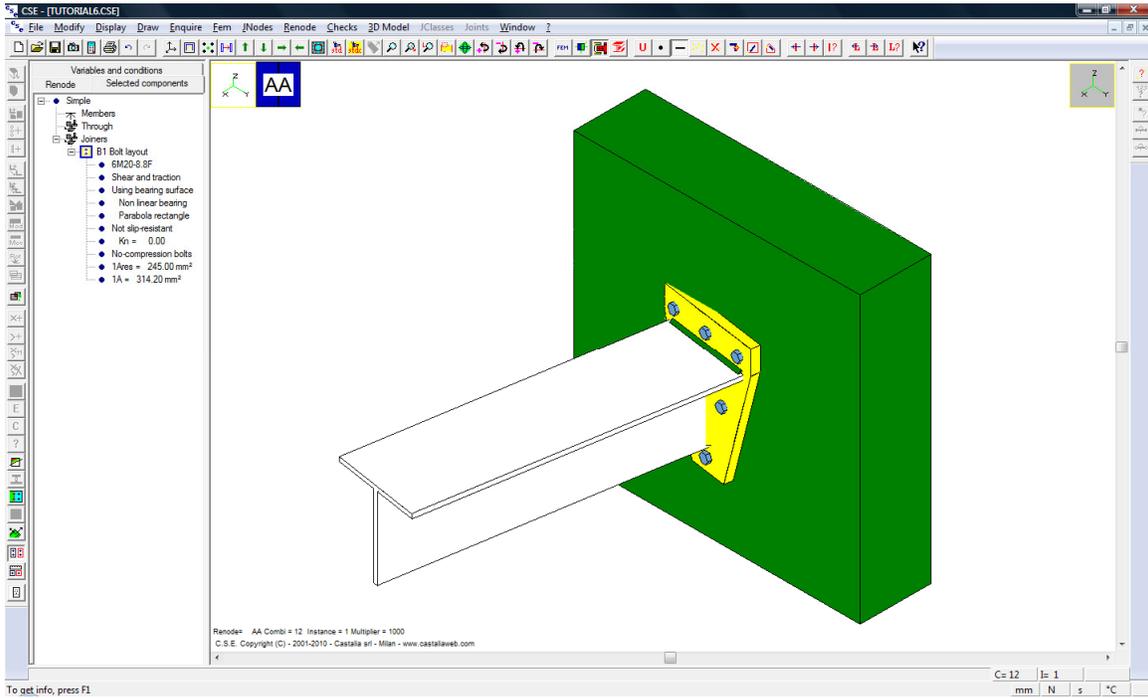
Use **Checks –Displaced**  in left toolbar to display the displaced view of the rende in current combination.





Switch between combinations to see displaced structure in desired combination  and change displaced scale if you need it .

See, for example, the displaced in combination 5 (amplification scale = 1000).



1	INTRODUCTION.....	3
2	CONSTRAINT.....	4
2.1	STEP 1: GETTING THE FEM MODEL.....	4
2.1.1	GETTING THE FEM ELEMENTS.....	4
2.1.2	ASSIGNING CROSS-SECTION	7
2.1.3	ASSIGNING MATERIAL	9
2.1.4	CONSTRAINT	10
2.2	STEP 2: SEARCHING MEMBERS.....	12
2.3	STEP 3: SEARCHING JNODES.....	12
2.4	STEP 4: SELECTING THE PROPER JNODE.....	18
2.5	STEP 5: CONSTRUCTION OF THE REAL NODE.....	19
2.5.1	ADDITION OF GENERIC POLYGON PLATE	19
2.5.2	ADDITION OF BOLT LAYOUT.....	32
2.5.3	ADDITION OF WELD LAYOUT.....	47
2.5.4	OVERLAPS AND COHERENCE CONTROLS	51
2.6	STEP 6: SETTING CHECK SETTINGS	52
2.7	STEP 7: EXECUTING CHECKS.....	53
2.8	STEP 8: EXAMINING RESULTS.....	54