

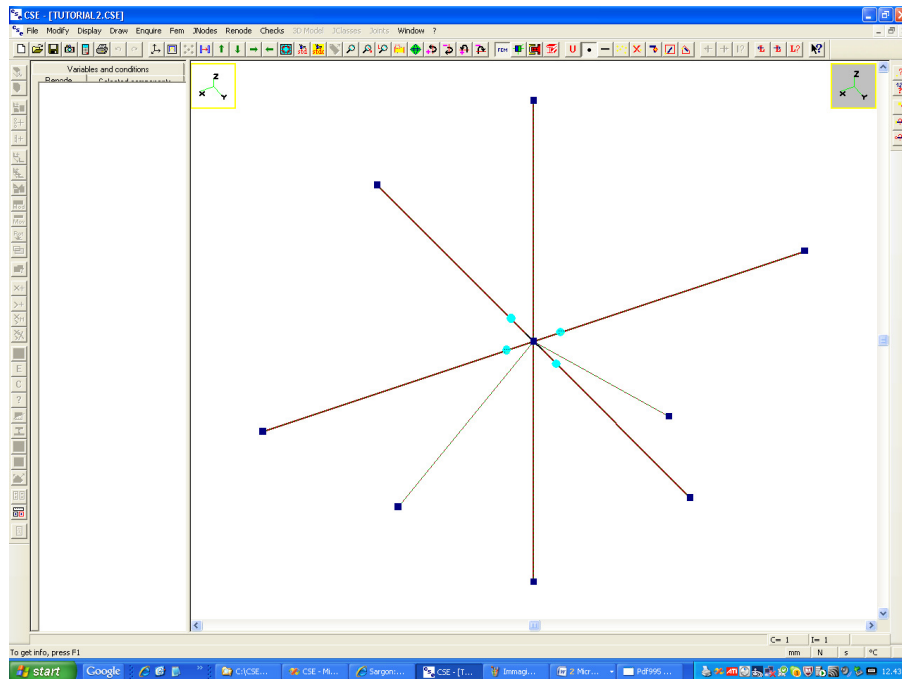


Paolo Rugarli



## Connection Study Environment

Tutorial 2: building a simple structure using fem commands



<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.6 November, 23, 2011

**Keywords:**

steel connections, steel joints, welds, bolts, software, checks, verification, base plate, bending, compression, no tension, bearing, steel, yield, stress, strain, bolted connections, welded connections, anchors, slip resistant, plates, cleats, constraint, clamp, column, stiffener, fem analysis, fem models, fea, plate element, thickness, stress map, CSE, Castalia srl, steelchecks.com, castaliaweb.com, C.S.E.

**Parole chiave:**

connessioni acciaio, collegamenti acciaio, saldature, bullonature, bulloni, software, verifiche, piastra di base, flessione, compressione, no-tension, contrasto, supporto, acciaio, snervamento, sforzo, deformazione, connessioni bullonate, connessioni saldate, ancoraggi, unioni ad attrito, piastre, vincoli, incastro, colonna, irrigidimento, analisi fem, modelli fem, elemento piastra, spessore, mappa di sforzo, CSE, Castalia srl, steelchecks.com, castaliaweb.com, C.S.E.



# 1 INTRODUCTION

This tutorial is a tool to start the understanding of how CSE works. This tutorial is aimed at teaching base commands relative to fem model management. You can bypass these commands for two reasons: a) You import an already prepared fem model; b) You use the command **Fem-Elements-Typical structures** and so the fem model is partly done automatically. It may happen, however, that the typical structures are not enough for your needs or that you don't have an already prepared fem model, so you could need these commands.

If you are using CSE (not the demo version as the demo version cannot import fem models) and do not have a copy of any FEM program interfaced with CSE, *keep in mind that you can freely download a demo version of Sargon limited to 50 nodes ([www.steelchecks.com/PRO/SR/demo.asp](http://www.steelchecks.com/PRO/SR/demo.asp))*, that will enable you to create small size fem models just in order to describe the connection(s). This is somewhat similar to what is explained here with an important difference: if you prepare the fem model in Sargon DEMO and run it you will be able to import it in CSE (not if using CSE demo version) *with all the relevant internal actions computed via fem analysis*. Here the fem model will not actually be run (not the forces will be assigned), and we will use internal actions values input by hand. Here the fem model is basically needed for geometrical reasons.

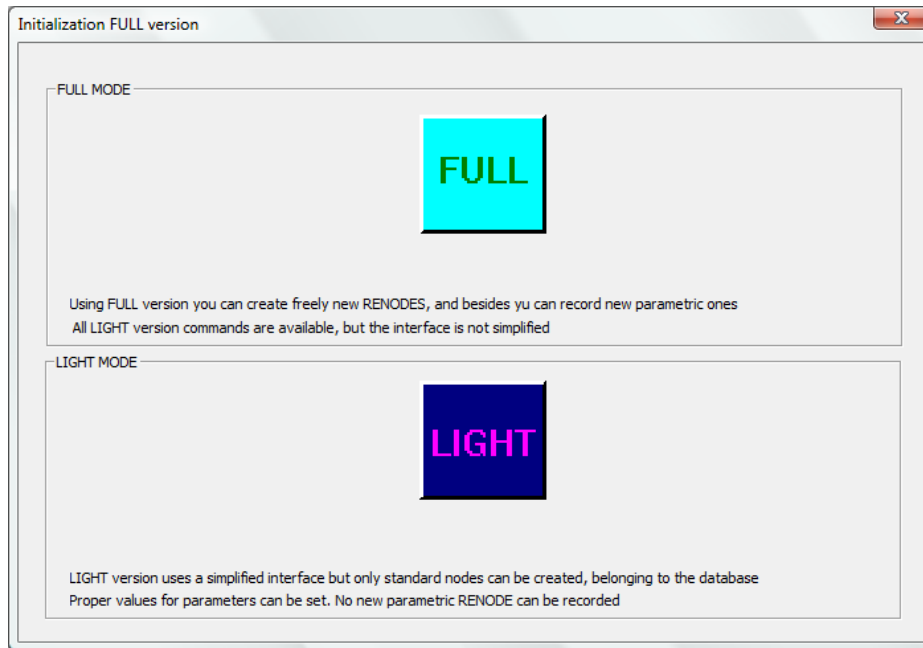
By following this tutorial you will be able to:

- Create a dummy fem model to study a given connection
- Assign the materials and cross-sections to the fem elements
- Understand end releases application and connection codes
- Understand offset assignment
- Search members
- Search jnodes
- Select and study jnodes
- Switch to renode view so as to begin to work over connection

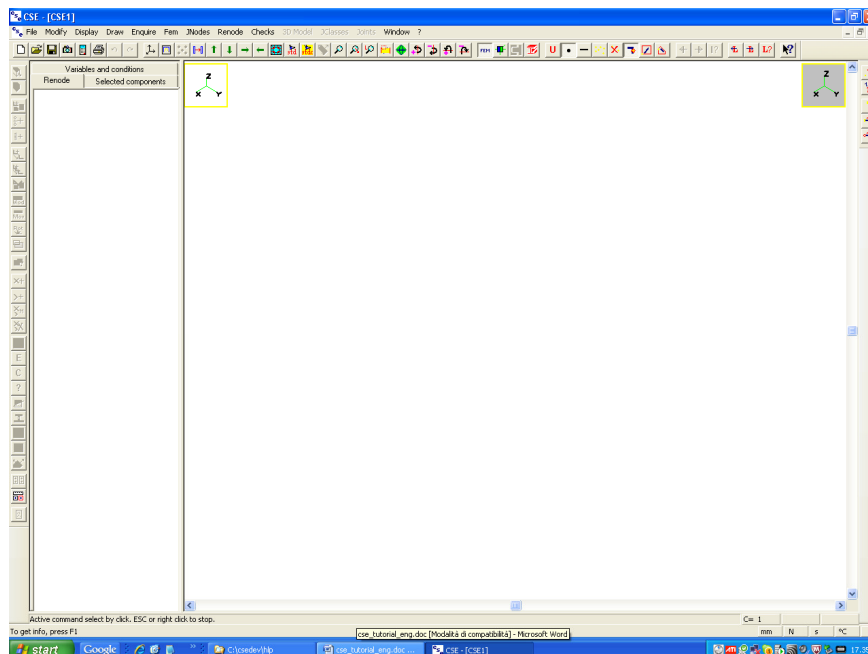
This tutorial is some like 36 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

*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 HOW TO BUILD A SIMPLE DUMMY STRUCTURE USING FEM COMMANDS



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



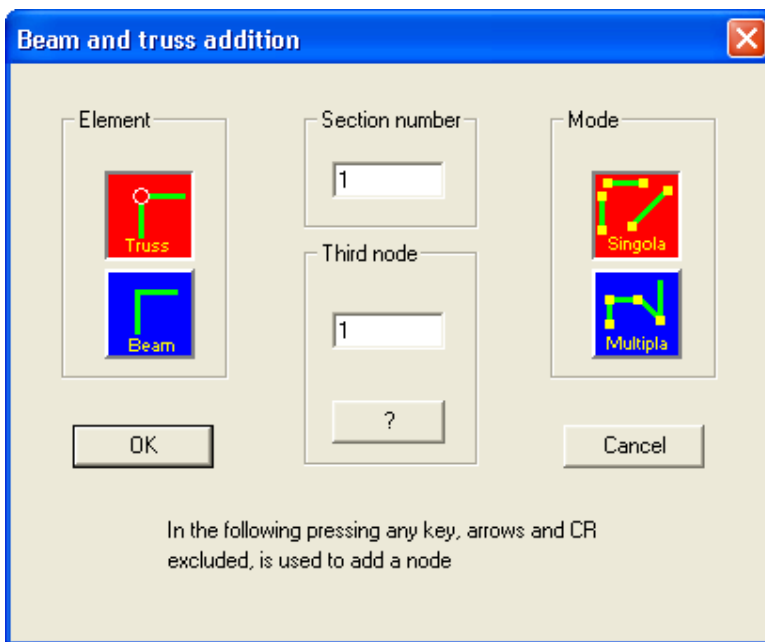
Initial window content: blank.

## 2.1 STEP 1: GETTING THE FEM ELEMENTS

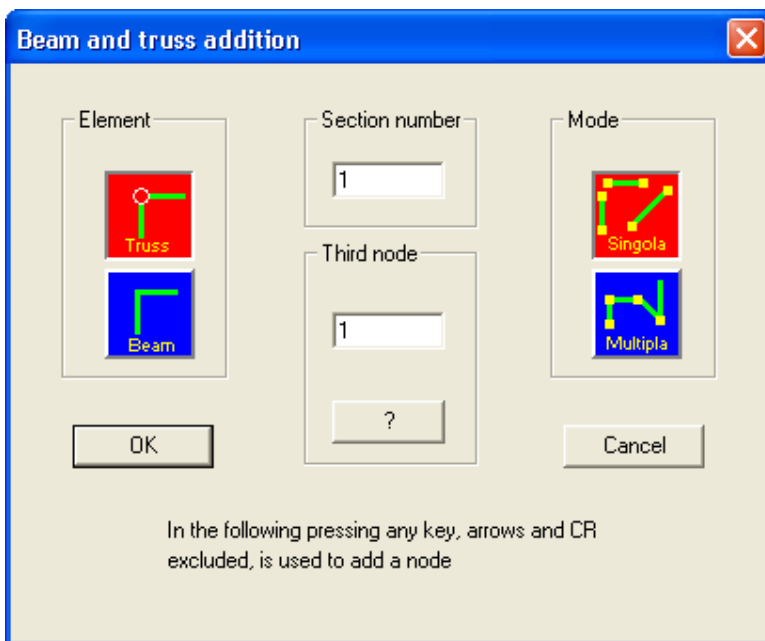
Activate right window by clicking left inside it.

Execute the command

### FEM-Elements-Add

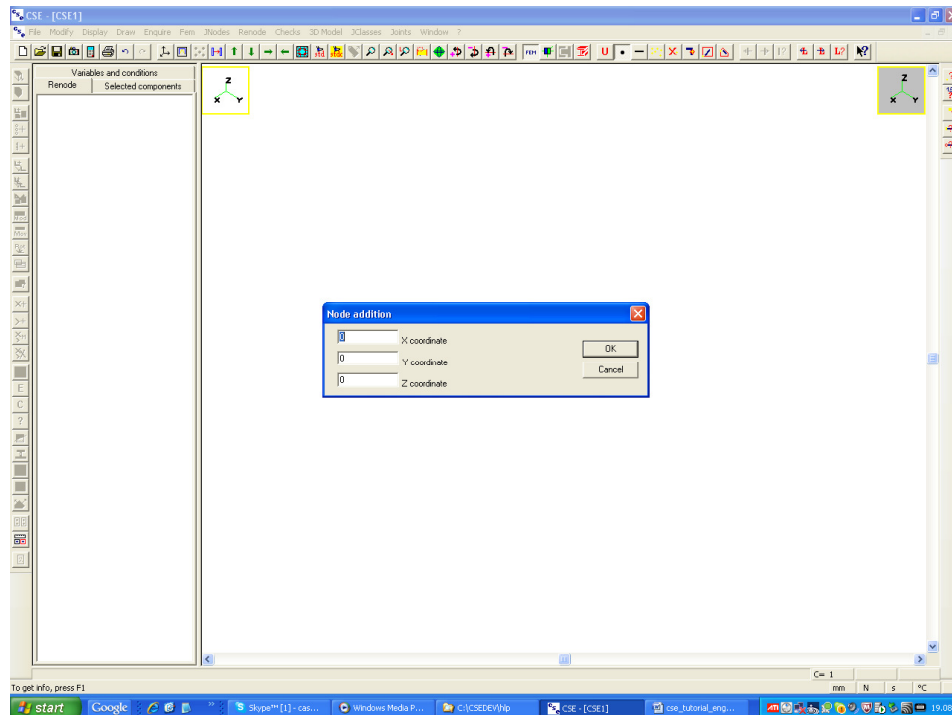


Choose to add beams clicking over the **Beam** button: it gets red. Now choose the "**Multiple**" button, it gets red. This means that the elements will be added so that from the second element onward first node of element "i" is the same node of the second node of element "i-1". We are in for adding a two elements column (Z direction).

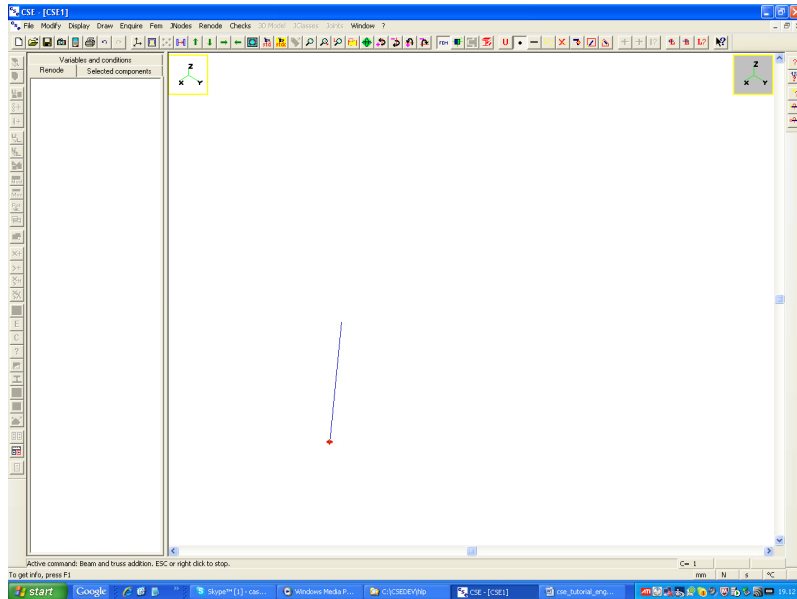


Leave as **Third node** the number 1: it means that strong axis (i.e. the axis normal to web in H cross-section elements) will be directed in +Y direction. Now press **OK**. You will apparently see no change in the screen: the program is waiting for your decisions. Notice that in the status bar appears the hint "**Active command: beam and truss addition. ESC or right click to stop**". This means you've actually entered into the command and that to stop it you can press ESC or right click mouse button.

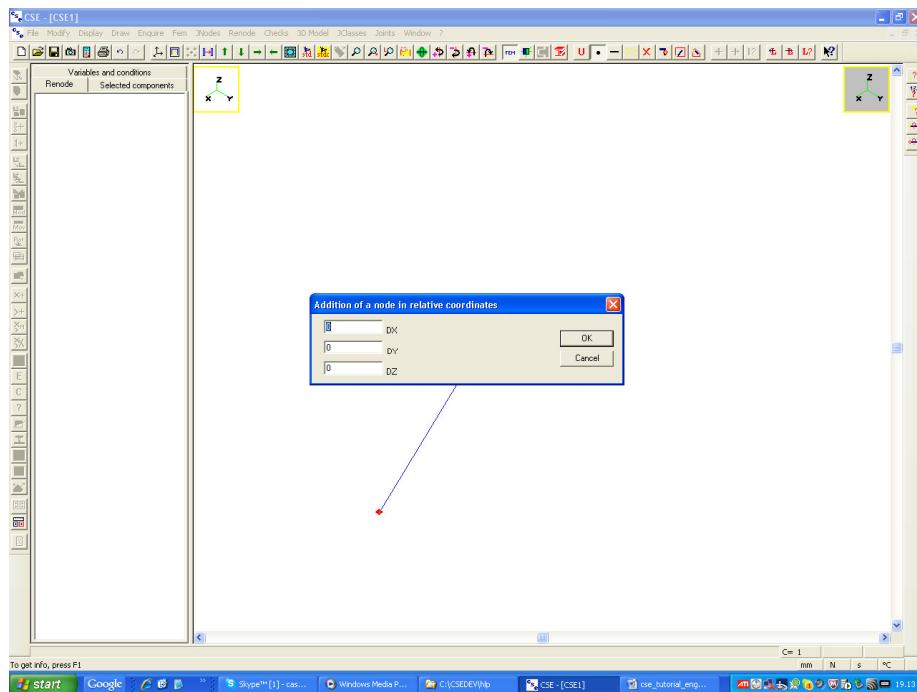
Since there are no nodes already added, *press the space bar* to add a node giving coordinates. You will see what follows:



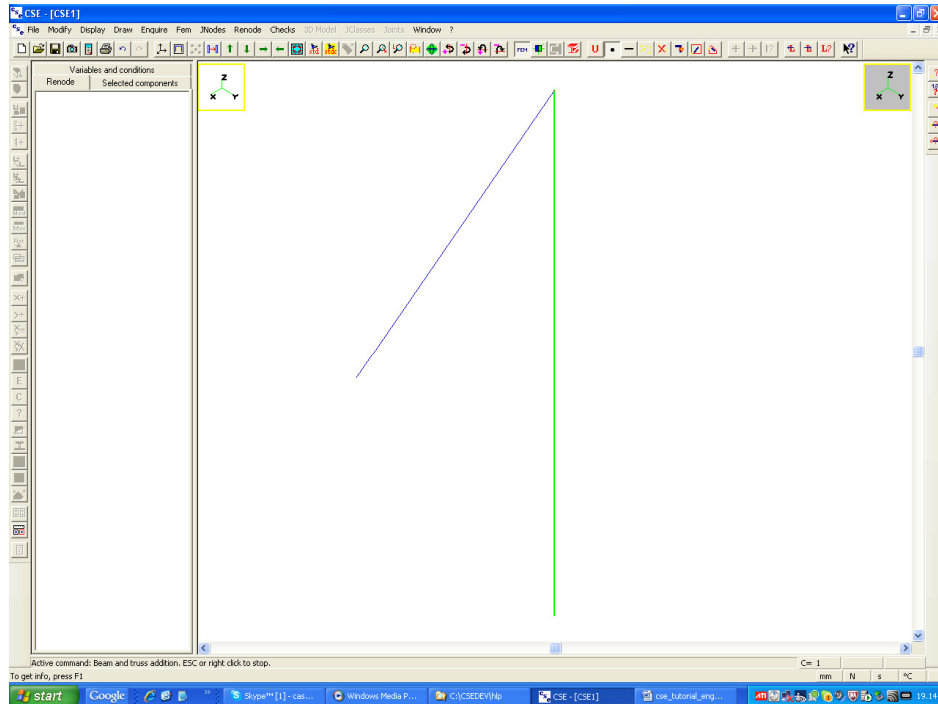
As we are not interested in the absolute position of the fem dummy structure you can leave all 0. Notice that we are using mm which is the default unit. So press **OK**.




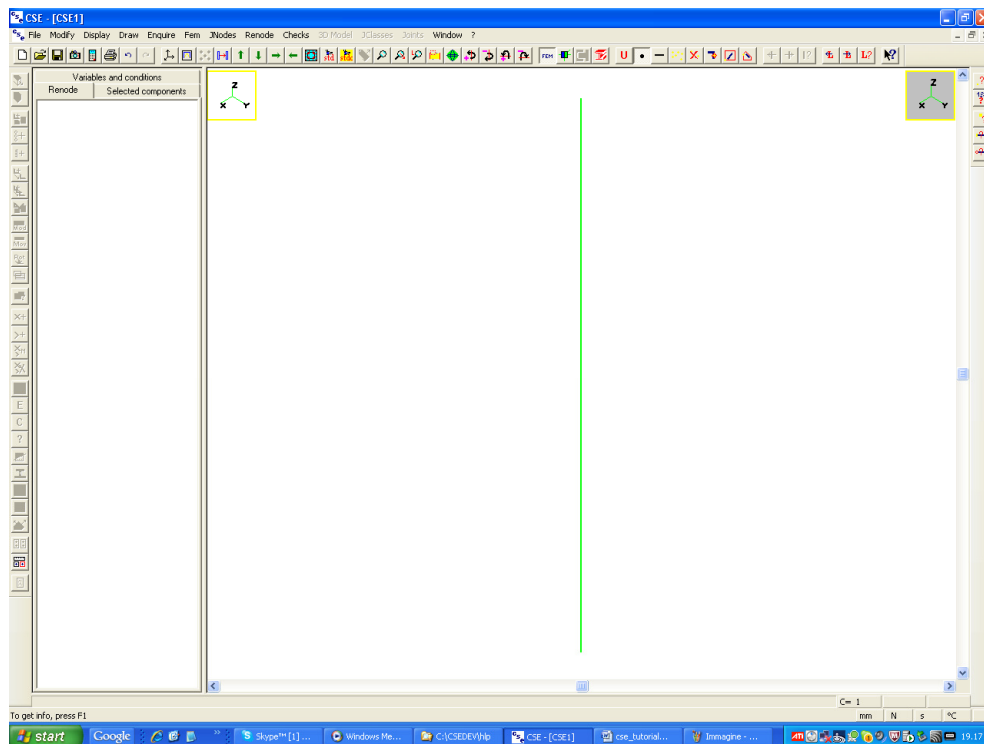
You now see that a node has been added and that the mouse is "attached" to it. Now we can add the second node of the element. Once more there is no suitable node already added, so we *press once more the space bar*, getting what follows:




Notice that now we are asked for "DX" "DY" and "DZ", not anymore "X", "Y", "Z". Enter +2000 for DZ and press OK. You see now what follows:

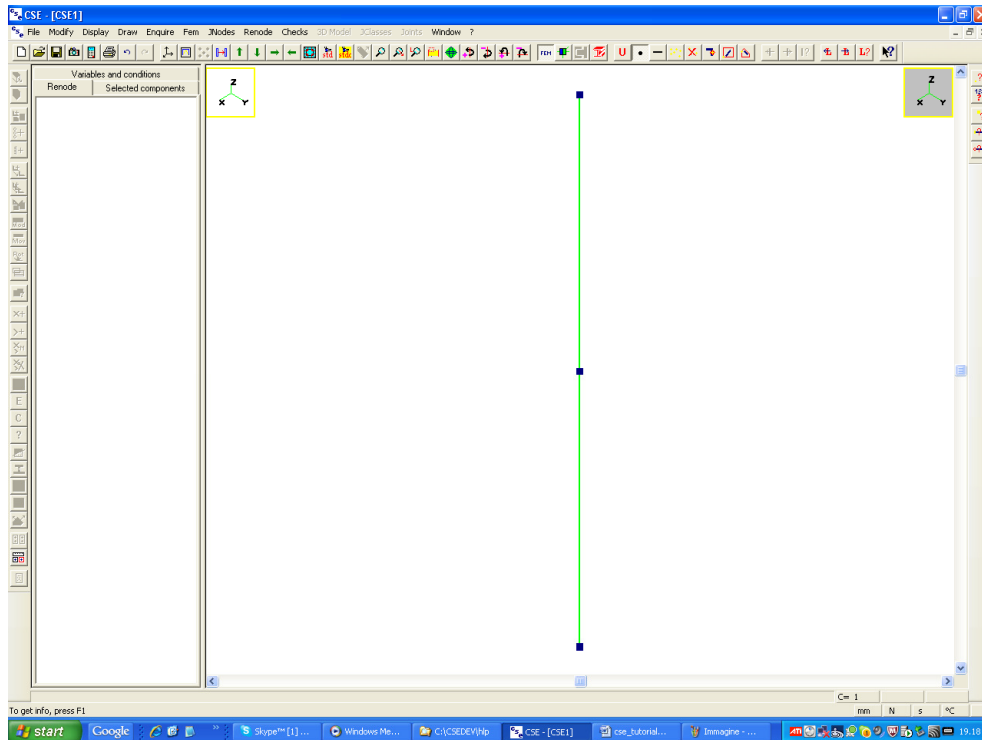



An element (in green) has been added. Press once more the space bar and enter once more  $DZ=2000$ . You'll get now the second (column member) finite element. Right click the mouse to stop the element addition command and enclose the structure into the screen by pressing the  button in the main bar (**Draw-Enclose** command). You see now this:

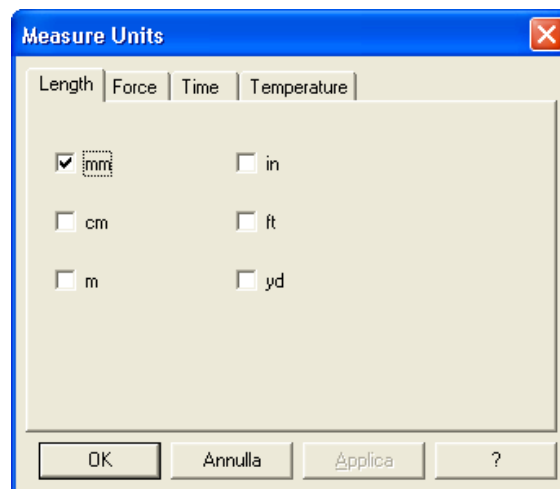




To see the nodes, select them all by pressing the  button (**Modify-Select All** command) in the main bar (by default you are selecting nodes). You now will see what follows:



Now we are going to add two beam elements in X direction (always having strong axis in +Y direction), but now we will use a different length unit. So, to change the length unit press the  button in the main bar (**Modify-Units** command). You'll see what follows:

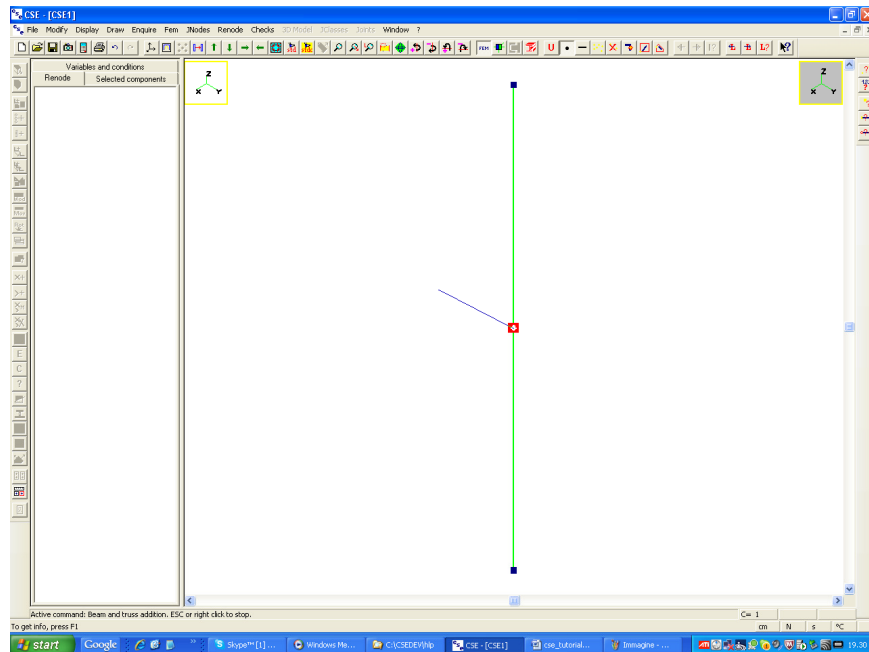


Choose "cm" and press **OK**. From now onward we will "talk" to the program using cm (i.e. centimetres) as length unit. Notice that in the status bar the length unit has changed from "mm" to "cm". You can change more times to completely understand how it works and to get familiar with the status bar info about measure units.

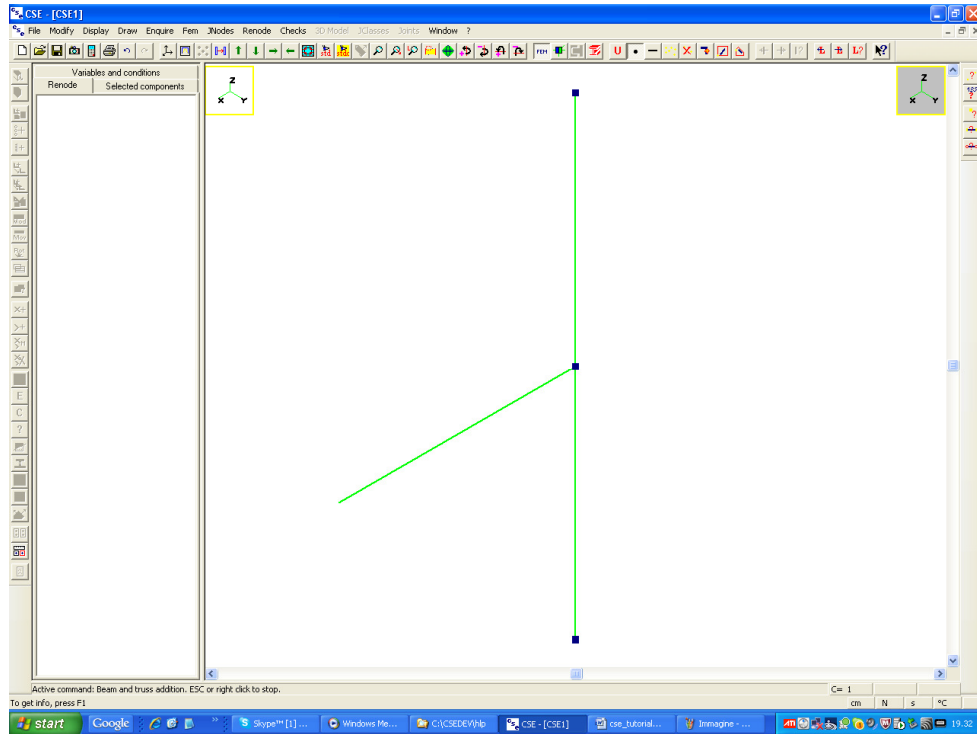
Execute once more the **Fem-Elements-Add** command. Now choose "**Beam**" and "**Single**". Leave 1 as third node. Now, once pressed **OK** notice that moving the mouse you will see a moving circle going from one of the available nodes to the others. If you don't see the moving circle just press this


button (**Draw-Redraw** command) in the main bar: 

Choose the mid height node and press the left mouse button. This will be the first new element node. You see now what follows (we are attached to the chosen node):

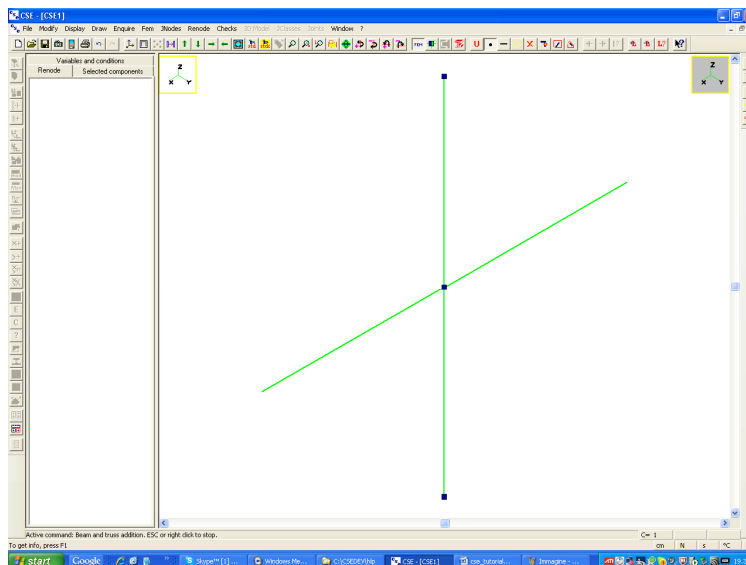


Now we need a node which does not exist. Just press the space bar and enter +200 for the DX value. So we are moving 200cm in +X direction to add the second element node. You now see what follows:




Notice that since we have chosen the **Single** way to add elements we can start a new element being completely free. Choose once more the mid height node of the column, by pressing the left mouse button near it. If you have problems in seeing the moving circle just redraw (  button in the main bar).

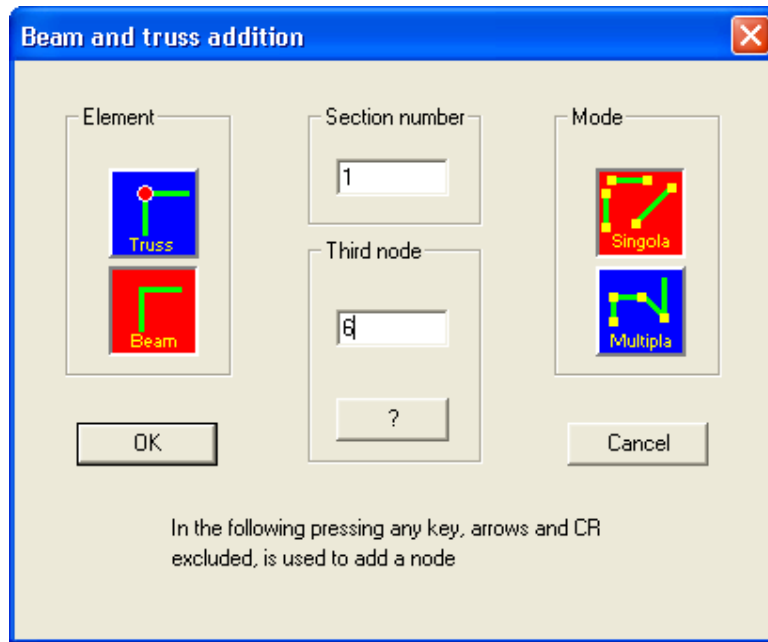
Now press the space bar and enter -200 for the DX value. You will see what follows:



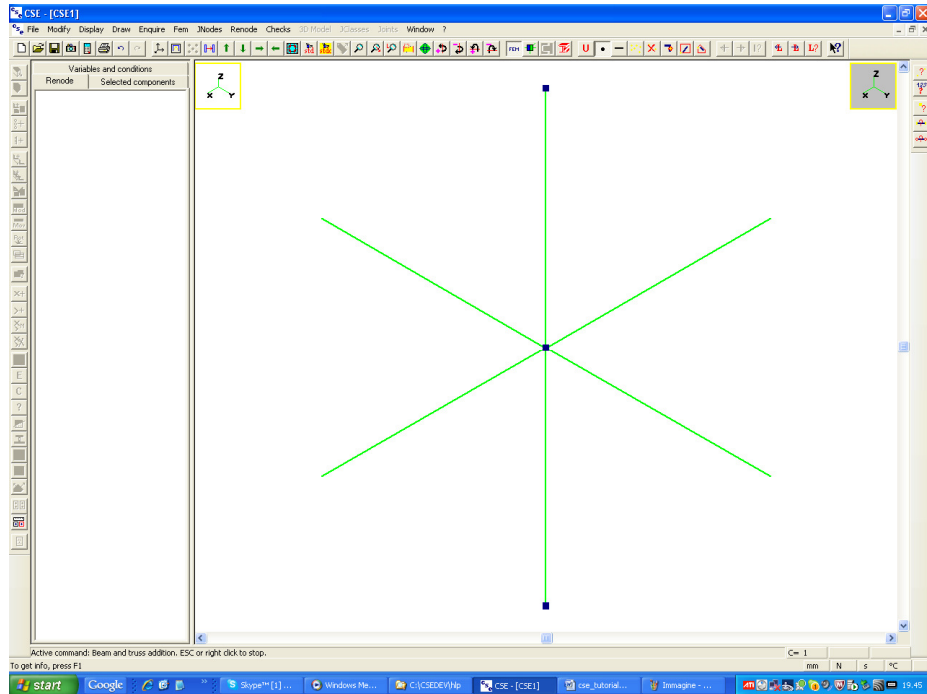
Now exit from the command by pressing the right mouse button: the status bar hint disappears.

Now switch back to mm executing the **Modify-Units** command (  button in the main bar), and choose "mm" as length unit. Notice that in the status bar the current length unit is updated.

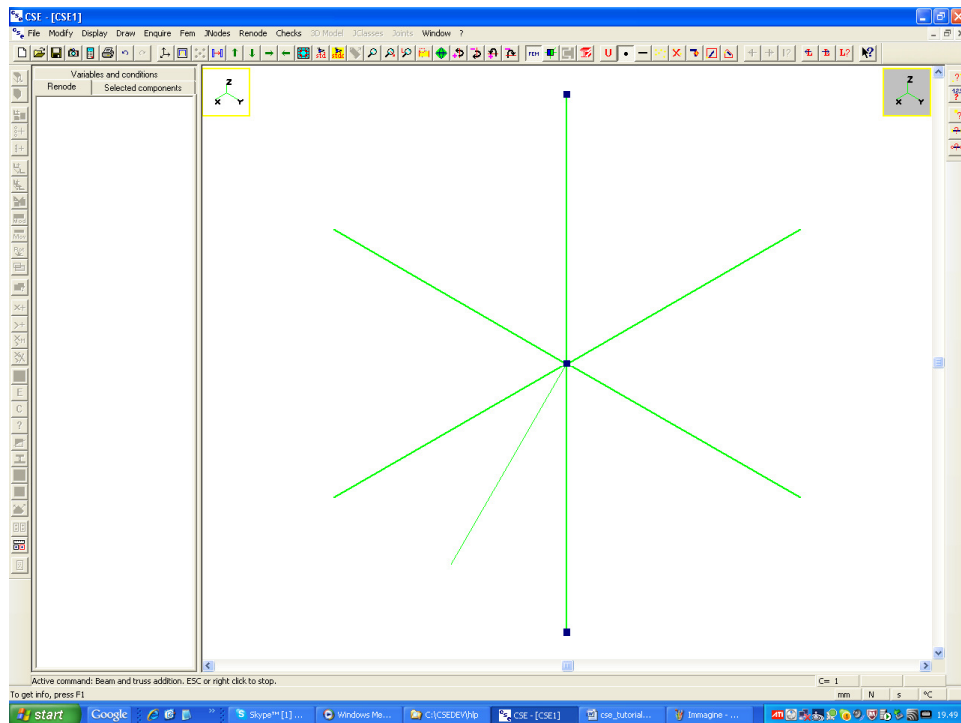
We are now ready to add two more elements in Y direction. However these elements cannot have the same third node of the previous or otherwise the three element nodes would be aligned. That's why we stopped the previous addition command. Execute once more the **Fem-Elements-Add** command and specify "**Beam**", "**Single**", and as third node the number 6: this means a third node in -X infinite direction (1,2,3,4,5,6=+Y,+Z,+X,-Y,-Z,-X). However this is just one way to specify orientation: we will later see a simpler way. Now you are here:



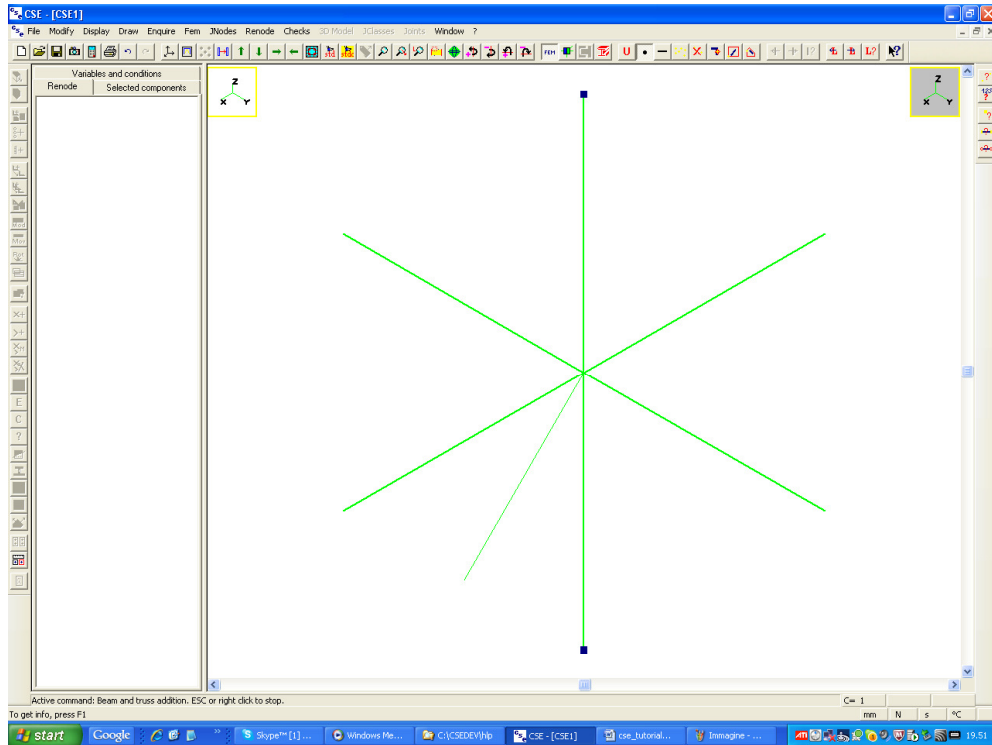
Press **OK**. You will basically do what you've already done but you will enter +2000 and -2000 as DY. Right click the mouse to exit from the addition command. At the end you will have added two more elements and get here:




Now we will add two truss elements in plane xz. Execute once more the **Fem-Elements-Add** command and specify "**Truss**", "**Single**", and third node +1. First click the central node and then specify DX=+1000 and DZ=-1000. This will get you here:



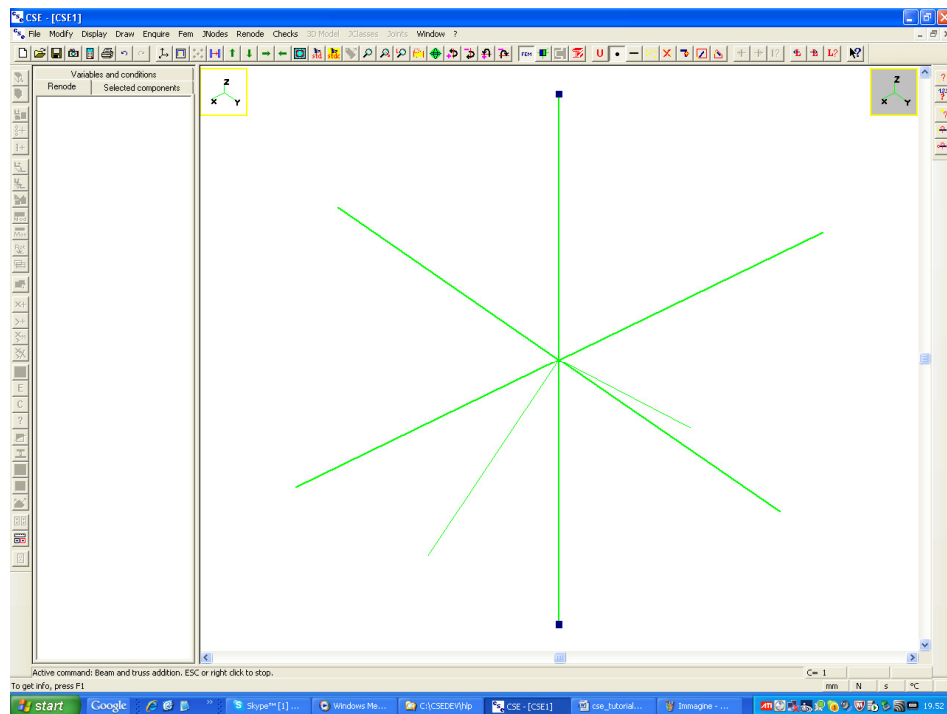
Now add a new element always connecting to the central node and specify DX=-1000 DZ=-1000. You will get here:



Apparently nothing has happened (no element added) but this is only due to your point of view.

Slightly change it by pressing the  button in the main bar (**Draw-Rotate clockwise** command).

You will see






Right click to stop the element addition command.

We have added some elements. The structure is purely dummy: it must not necessarily be constrained as we are not going to compute it using FEM tools. We are just going to compute the connection using elastic or plastic limits as input forces to check the connection.

Now we assign cross-sections and materials.

## 2.2 STEP 2: ASSIGNING CROSS-SECTIONS AND MATERIALS.

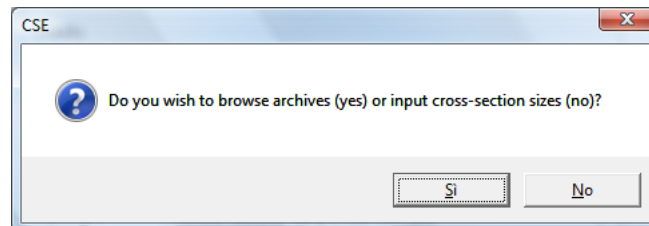
In this tutorial we are using features that can be applied both to standard version and to DEMO version. So, since in the DEMO version you can just apply HEB200 to members, we will also apply this kind of cross-section for standard version users.

To apply a cross-section you first have to select the finite elements to which the cross-section will be applied. First you must tell to the program that you want to select elements, and not nodes. To do that you press the  button in the main bar (command **Modify-Select-1D**). Now to select all elements you can press the  button in the main bar (**Modify-Select-All** command). Otherwise you can click one by one by first pressing the  button in the main bar (**Modify-Select-Click** command), and then exit from the command "select by click" by clicking the right button. The elements selected get blue.

Now execute the following command

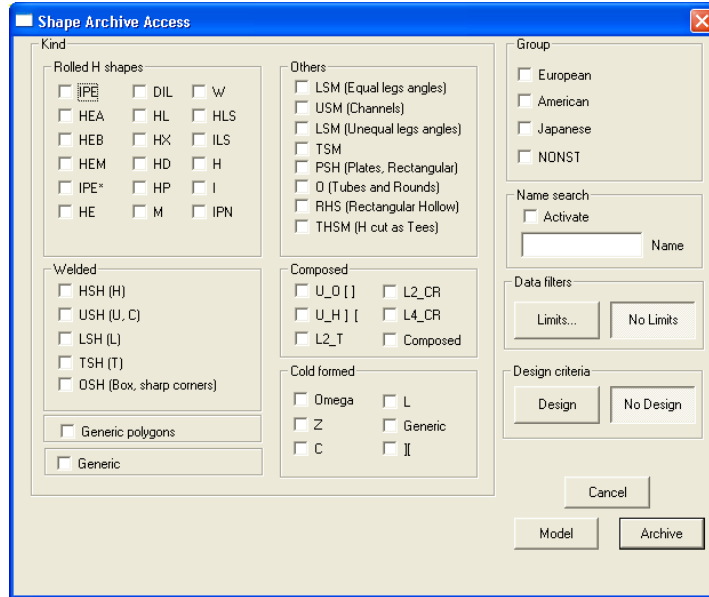
### FEM-Elements-Cross-sections.

If you are using the demo this just applies the cross-section HEB200 to the column, otherwise you will be prompted to choose a cross-section kind, with the following dialog:

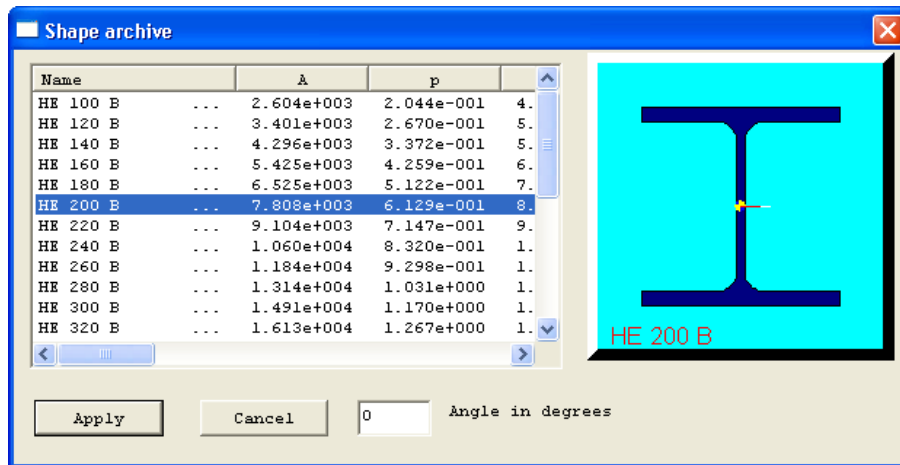


Click "Si" (Yes) to choose a shape from CSE archive (new shapes can be added by clicking "No").

The following dialog appears:



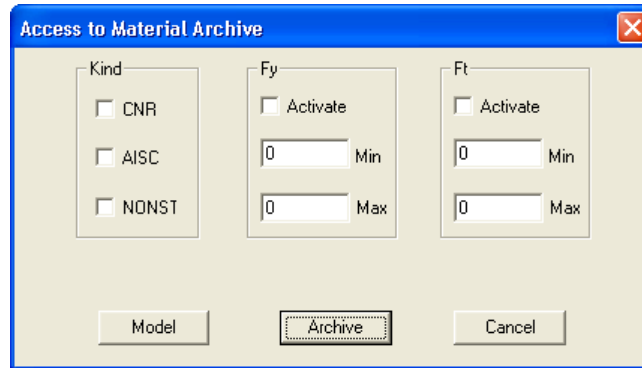
Select HEB check box and then "**Archive**" to browse among the cross-section archive. Then choose the section HEB200 by selecting the appropriate row, and press the **Apply** button:



Now to apply materials execute the command

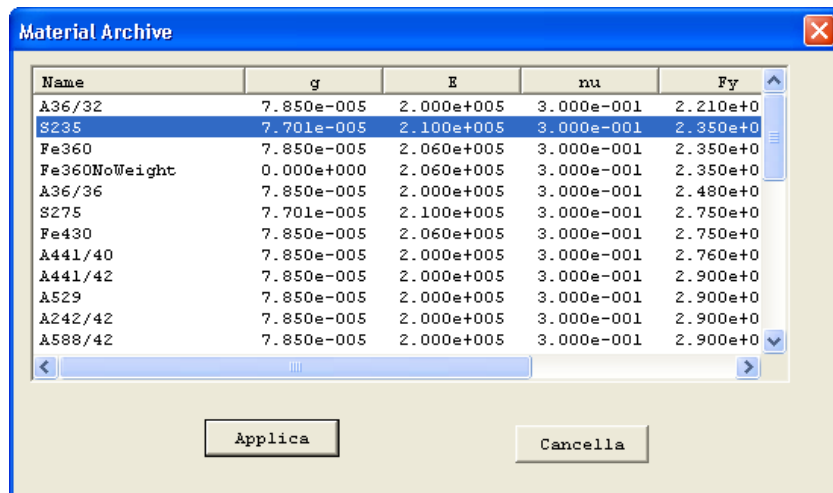
### FEM-Elements-Materials

If you are using the demo this automatically applies the material S235 to all selected elements (just one in this case). Otherwise choose **Archive** in the following dialog:



The "Access to Material Archive" dialog box contains three sections. The first section, labeled "Kind", has three checkboxes: "CNR", "AISC", and "NONST". The second section, labeled "Fy", has an "Activate" checkbox and two input fields for "Min" and "Max", both containing the value "0". The third section, labeled "Ft", also has an "Activate" checkbox and two input fields for "Min" and "Max", both containing the value "0". At the bottom, there are three buttons: "Model", "Archive" (which is highlighted with a dashed border), and "Cancel".

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

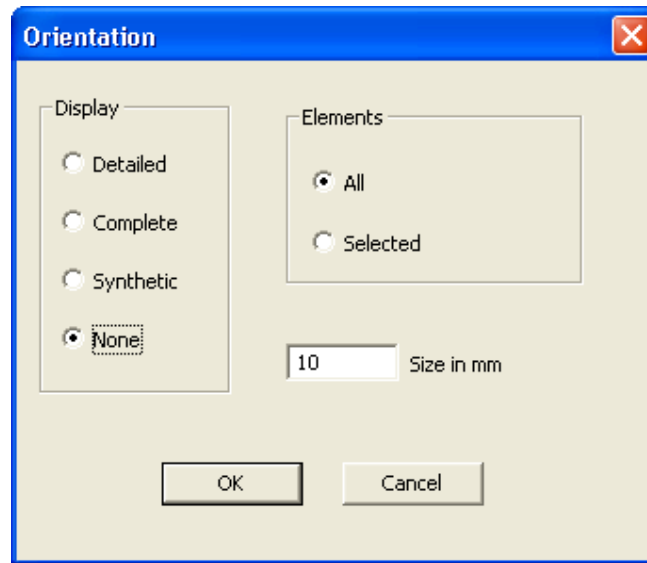


The "Material Archive" dialog box displays a table of materials. The table has five columns: "Name", "g", "E", "nu", and "Fy". The "S235" row is selected. Below the table are two buttons: "Applica" and "Cancella".

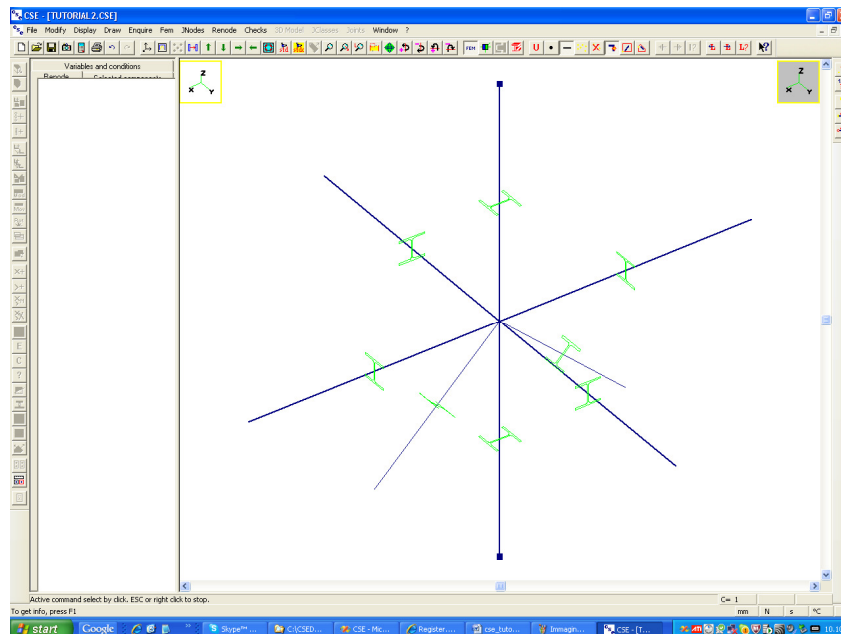
Name	g	E	nu	Fy
A36/32	7.850e-005	2.000e+005	3.000e-001	2.210e+0
S235	7.701e-005	2.100e+005	3.000e-001	2.350e+0
Fe360	7.850e-005	2.060e+005	3.000e-001	2.350e+0
Fe360NoWeight	0.000e+000	2.060e+005	3.000e-001	2.350e+0
A36/36	7.850e-005	2.000e+005	3.000e-001	2.480e+0
S275	7.701e-005	2.100e+005	3.000e-001	2.750e+0
Fe430	7.850e-005	2.060e+005	3.000e-001	2.750e+0
A441/40	7.850e-005	2.000e+005	3.000e-001	2.760e+0
A441/42	7.850e-005	2.000e+005	3.000e-001	2.900e+0
A529	7.850e-005	2.000e+005	3.000e-001	2.900e+0
A242/42	7.850e-005	2.000e+005	3.000e-001	2.900e+0
A588/42	7.850e-005	2.000e+005	3.000e-001	2.900e+0

## 2.3 STEP 3 REVIEWING ELEMENT ORIENTATION

To have a look at how local axes have been assigned (and cross-sections) we will use the command **Display-Orientation**, which opens this dialog box:

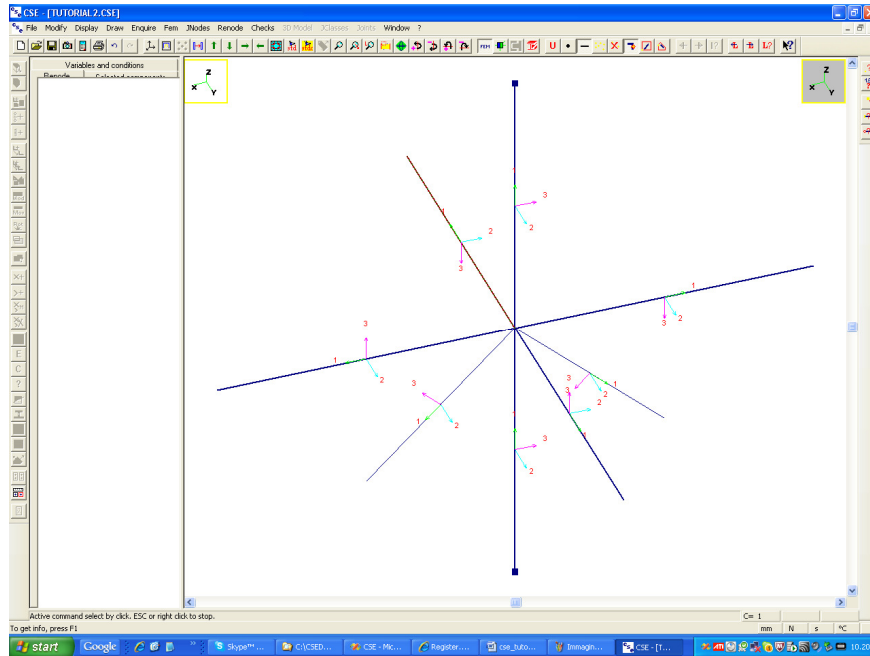



First choose **Detailed** and leave **All** (this means we will see orientation for all finite elements). Press **OK**, you will see what follows.



The cross-sections applied are shown in the view with their actual orientation. So you can decide whether they are ok, or not.

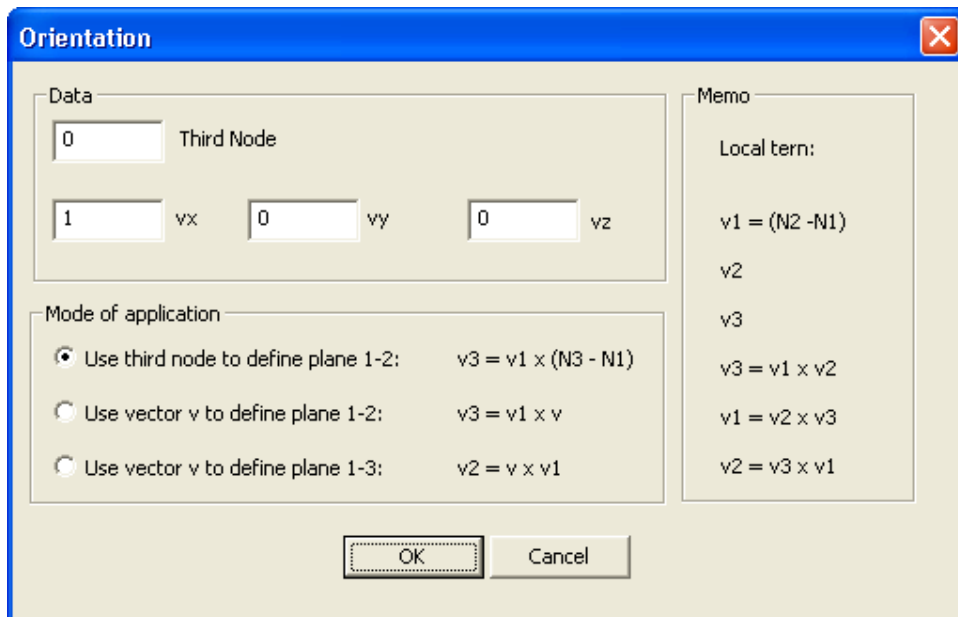
Now re-execute the command **Display-Orientation** and choose "**Complete**", in the dialog box. You will see what follows, that is the local terms of the elements:



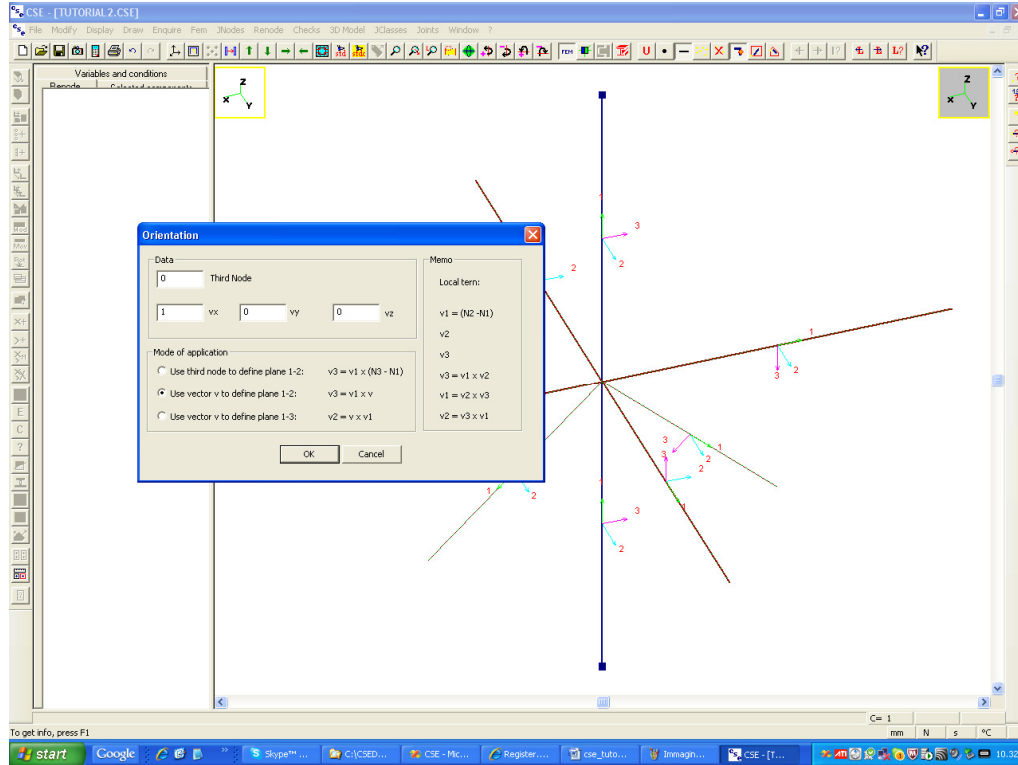
We will - as an exercise - change the orientation of some elements. So first of all unselect all elements by pressing the  button in the main bar (**Modify-Select-None** command). Then select the column elements by clicking over them (clicking left in the right pane automatically starts the select by click command). You will see the vertical elements blue.

If we want to change the orientation, we will have to apply the command **Fem-Elements-Orientation** to the selected elements.

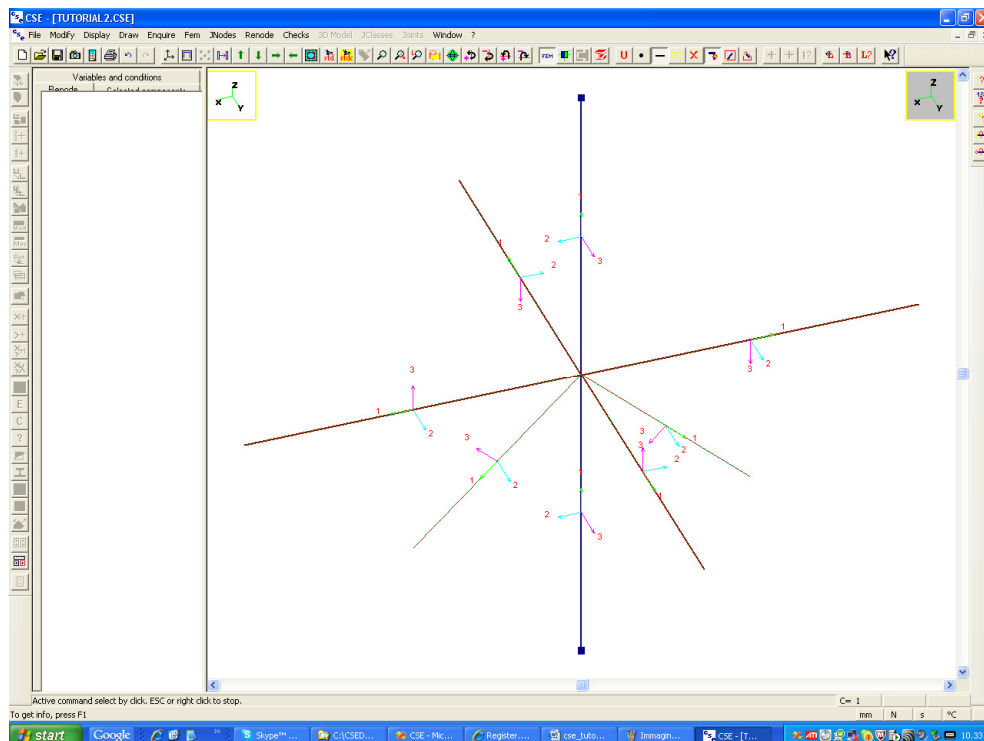
When executing this command the following dialog box appears



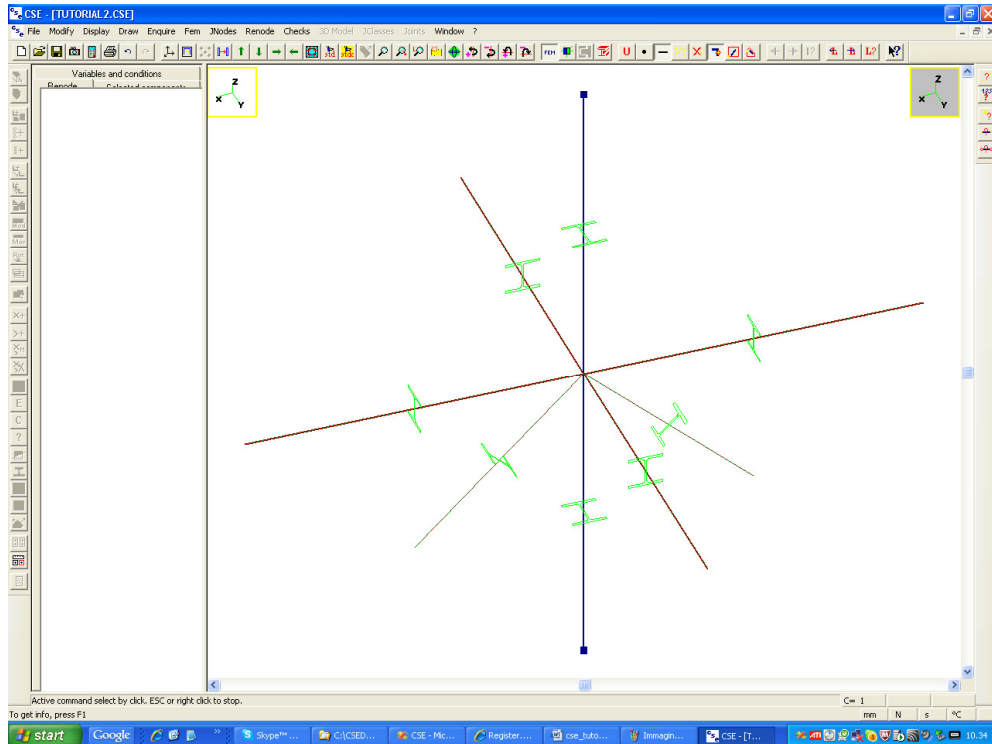
which lets you decide which method to use in order to specify selected elements orientation. You can use the third node method (as we have done until now) or you can use a vector to define plane 1-2 or plane 1-3. Select the **"Use vector v to define plane 1-2"** option and enter the vector (1,0,0) like this:



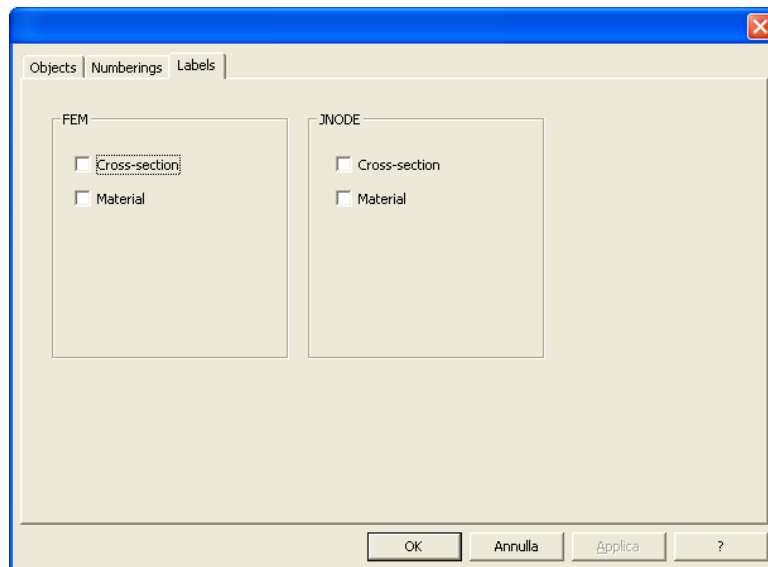
press **OK** and notice that the axis 2 has changed orientation (it now points in X direction):



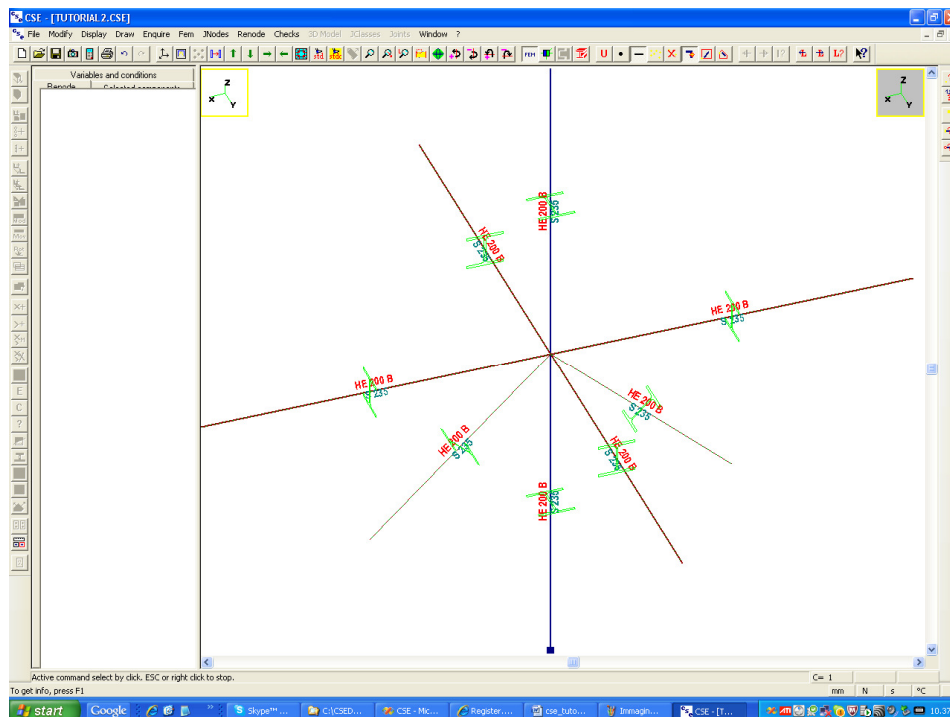
If you ask for **Detailed** orientation (**Display-Orientation** and "**Detailed**" in the dialog box) you will see that the cross-section has been turned:



We can now ask to the program to display the cross-section and material labels. To do that execute the command **Display-Objects** and choose the pane "**Labels**" in the property sheet:



Place a tick in the **Cross-section** and **Material** check boxes. You will now see what follows:



If you wish to remove all these labels and the cross-section drawing from the view, Execute the command **Display-Objects**, choose "**Labels**" and remove the tick from the check boxes. Then execute **Display-Orientation** and choose "**None**" in the dialog box.

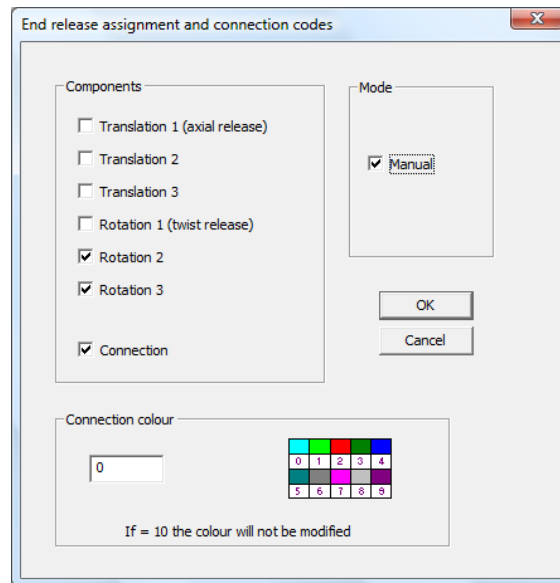
Next step will be to care about end releases and connection codes.

## 2.4 STEP 4: ASSIGNING END RELEASES AND CONNECTION CODES

Until now we did not care about end releases and connection codes. In this model we will consider the column as passing element, and the other elements as interrupted. Truss elements are interrupted at their ends by definition. If two beam elements are aligned, connected to the same node, they are unreleased, share the same cross-section and material, and have equal orientations, then they will be considered as a unique member. Here this conditions will be met both by X beams and by Y beams and by the column elements, which form a set of three members, passing in the mid-height node. To explain that some of the members should be divided, we must add or an end release or a connection code. An end release automatically implies a connection code, but you can place a connection code without assigning any end-release. This happens when a member is clamped in a node, but should be considered interrupted, as otherwise the program will consider it as part of a passing member.

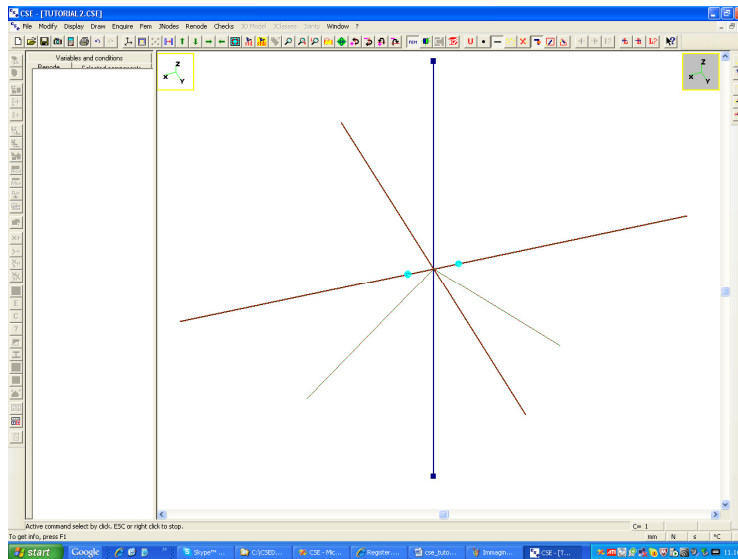
We consider that X elements are hinged at their extremes, while Y elements are clamped.

To assign end releases to X beams, we execute the command **Fem-Assign end release**, and get the following Dialog, where we place ticks in the "**Rotation 2**", "**Rotation 3**" and "**Manual**" check boxes. The first two ticks mean that the element will not transfer bending moment around the axis 2 and axis 3. Notice that as soon as you place a tick in one of the 6 possible check boxes associated to local degrees of freedom for the element extreme, the program automatically adds a tick in the "**Connection**" check box, to assign a connection code, as well. A released extreme cannot be continuous, it must end the member. The "**manual**" check box means that we will place the end releases by clicking near the element extremes and not using selected elements and nodes info.



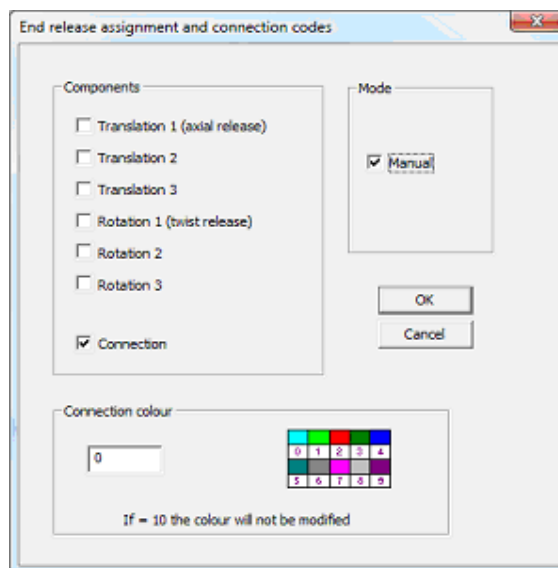
*Note: the connection colour is used to univocally define the orientation of symmetrical jnodes that occur more than once in a structure; for the same reason, coloured connection codes can be applied also to truss elements. Here we have one jnode only, so just leave the default colour.*

Now left click near the column-connected extremes of the X beams. Then exit from the command by clicking right. You will get what follows.

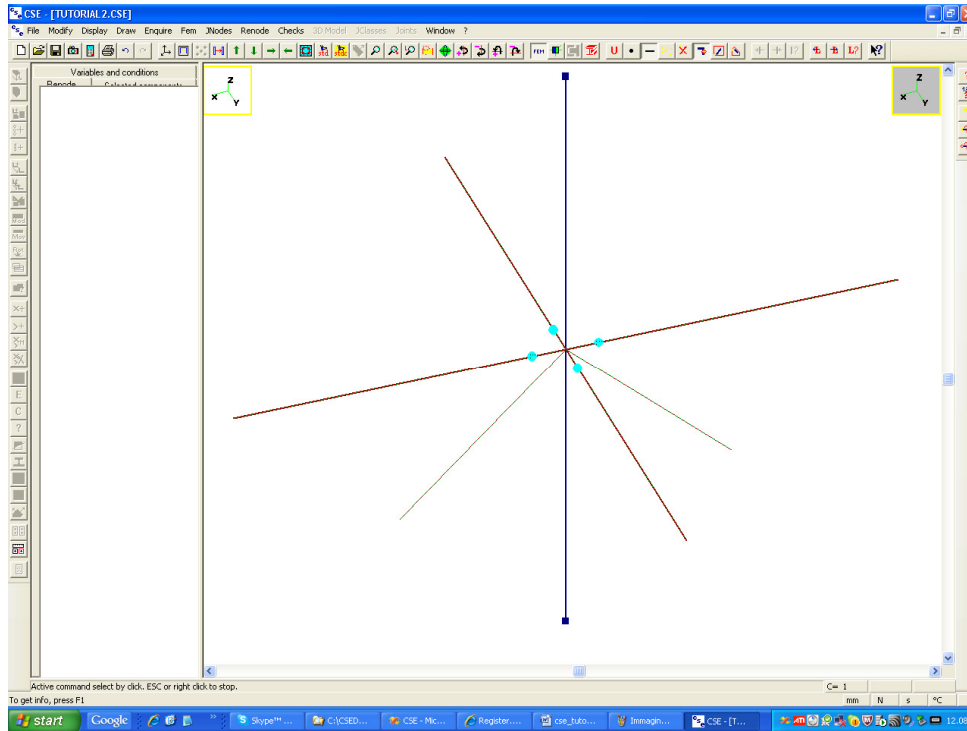


The cyan circles mean that there is an end release. If you see well you will also see internal lighted pixels associated to the dofs released (end release symbol size may be increased by using **Display-Sizes** command).

Now we must consider Y elements. They are not released but we must teach the program that these elements cannot be considered as one member (although the fem elements are equal, aligned, and could actually form a member). So we must place a "connection code" which is a null release useful only to teach to the program that the only un-interrupted member will be the column (where once more we have aligned, equally oriented elements), and not the Y element beams. So re-execute the command **Fem-Assign end release**, and place a tick just in the "**Connection**" check box and in the "**Manual**" check box.



Press **OK** and then click near the extremes of the Y element connected to the column. Click right to exit from the command. You will now see what follows:



Notice that the cyan circles applied to Y elements do not have pixel lighted inside. This means that there are not, actually, end releases, but just a connection code. Now the program will understand that the two elements form two distinct members. So the only member made up by two finite elements will be the column.

## 2.5 STEP 5: ASSIGNING OFFSETS

Offsets can be useful in defining fem behaviour to correctly keep into account distances which are actually present in the model. They can also be used to prepare the correct shortening of members when the cross-section of the other is not going to change.


Here we will apply two axial offsets to the Y beams, in order to let the members end just over the outer plane of the column flanges.

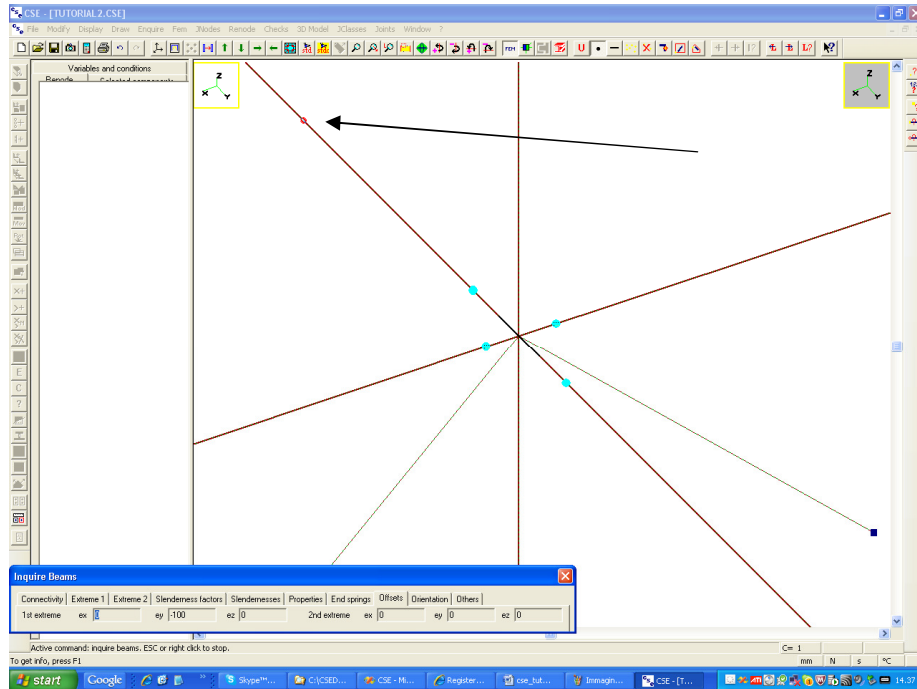
Execute the command **Fem-Elements-Offsets**. You get the following dialog box:

**Offsets**

Offsets go from node to element-extreme in global or local coordinate system

Choose **Manual** and **Local System**. The first choice means that we will have to click near the element extreme where we want to apply the release. The second choice means that we are going to define the offsets in the **Local system** of the element. So, since it is an axial shortening, we input 100 as DX. Ideal node will be shifted in + axis 1 direction by 100mm to find the true, offset extreme. Now keep in mind that, by the way elements have been added, axis 1 always points from column node outward, so a positive offset will always be a shortening for the elements. Once clicked near the extreme you will notice that the offset segment is black, i.e. is colored differently by the element axis. To exit from the command, as always, click right.

You can inquire the element offset by using the **Enquire-Beams** command (use the  button in the right Enquire commands toolbar). Moving the mouse you see that a moving circle is over the nearest element centre (see picture below). Selecting the **Offset** pane (see picture) you will real time get info about the offsets applied. Keep in mind that the offset are output in the global axes system, not in the local one. So what before was input as +100 can get -100 for one of the two offset elements.




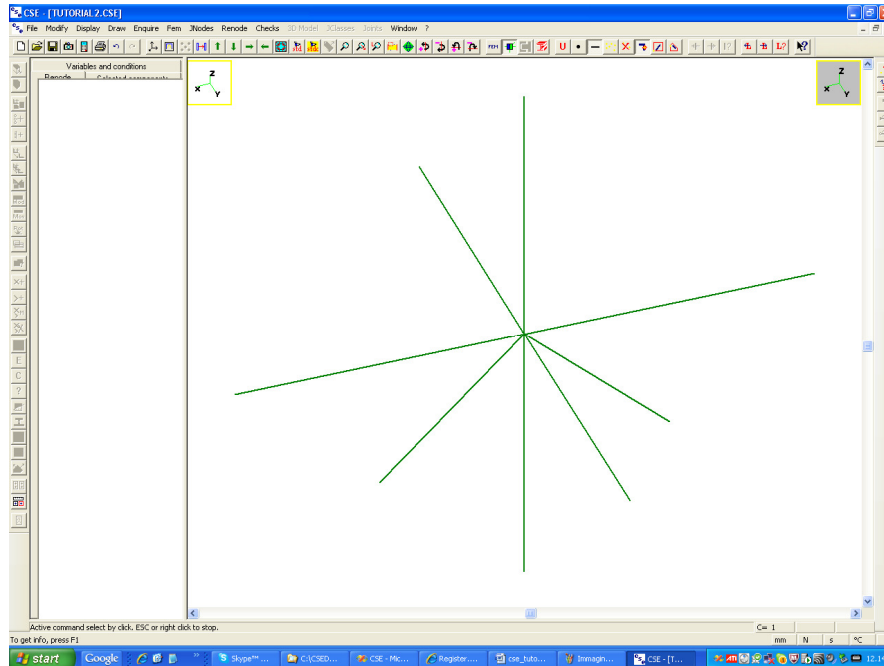
You can also use this command to get info about elements.

## 2.6 STEP 6: 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!**

Once you've done this, the program knows which are the members. You can have a look at them if you switch to the Jnode view. To do that press the  button in the main bar (command **Display-Jnode**). The jnode view is wireframe:



You are still selecting elements, not nodes (i.e. 1D, not 0D). When the jnode view is active the "elements" are actually members, while in the fem view (i.e. the previous view) the "elements" are the finite elements (beams and trusses) properly said.

So click over the column: you will notice that it all gets blue, meaning that the program considers the two finite elements as a unique member. If you click over the Y elements you will instead notice that the program has kept them distinct. This is due to connection codes. If the elements were different (i.e. they had different cross-section, or orientation, or material, or if they were not aligned) then no connection code would have been necessary as they could not have been considered as a unique member.

Now that we are sure that members have been correctly detected we can search jnodes. Here clearly just one jnode will be found.

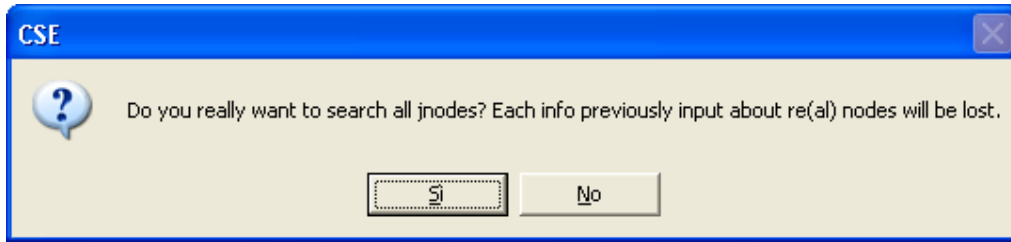
## 2.7 STEP 7: SEARCHING JNODES

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:

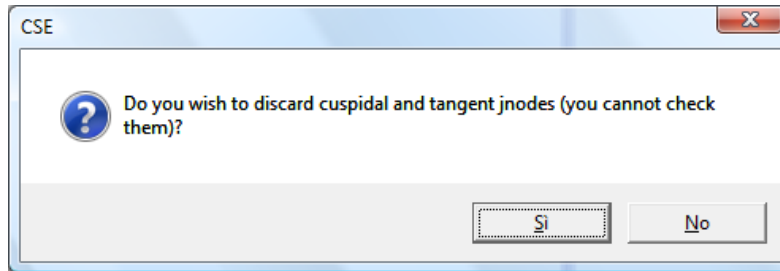
**JNODE-Search jnodes!**

and answer "yes" to the following question:



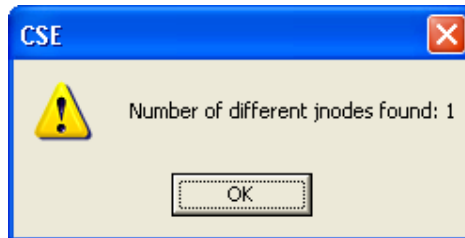
As no real nodes info has been defined you won't lose anything. Jnode search is usually done once for all in a model. Before beginning to add RENODES, you will check that the JNODES found are correct. This depends also on how the fem model has been prepared.

The following dialog appears:



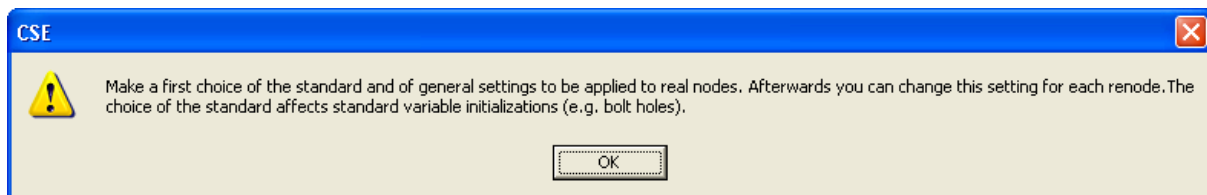
It is possible to discard cuspidal and tangent jnodes from the search (these jnodes cannot be computed).

You will get the following message after command execution:

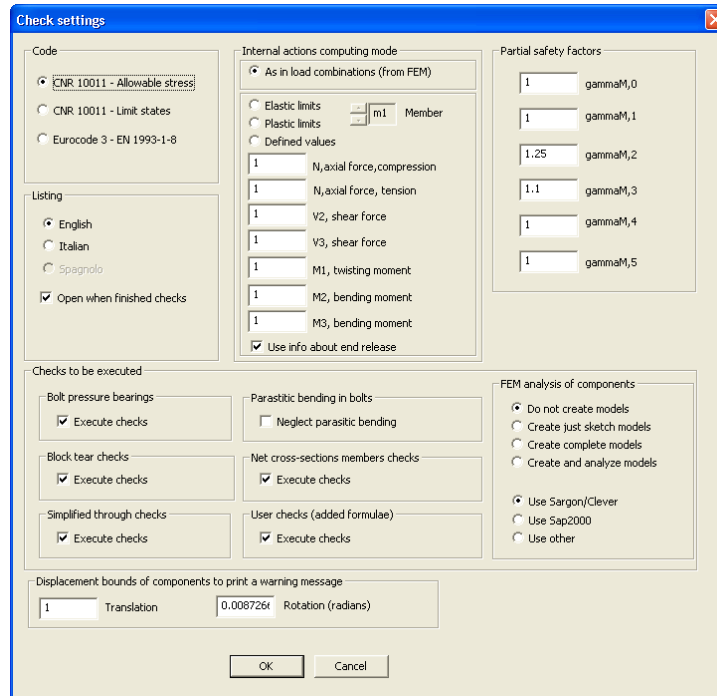


This means that in this model there is just one jnode, (master jnode, i.e. members all attaching to one un-interrupted member, the master, here the column).

Now the following message appears:

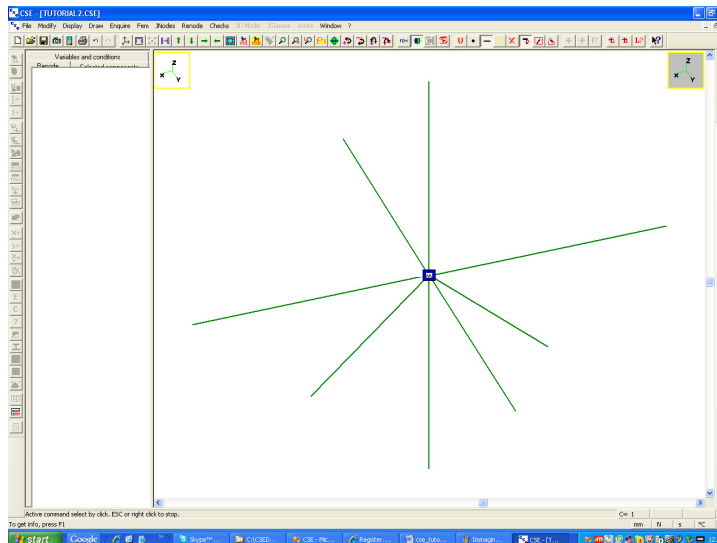



asking to make a first choice about the standard you are going to use. This settings will be applied to all renodes (here just one) as initialisation. Afterwards you will be able to assign different settings (e.g. about the checks to be done) to each different renode. So the following dialog appears:

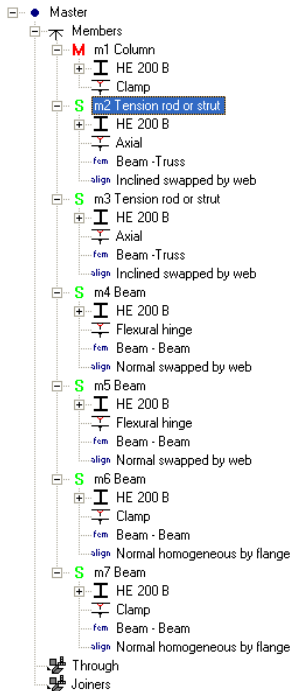


Choose Eurocode 3 and leave all other defaults, we will change them later.

We are in jnode view and see the following:



Since jnodes have been searched the program is able to assign labels to them. Here we have just one jnode labelled "AA". If you switch to the 0D selection by pressing the  button in the main bar (**Modify-Select-0D** command), and left click over the blue "AA" square in jnode view, you will select a jnode. Notice that the square gets yellow. This means that the jnode is selected. Notice that in the left pane, as soon as you select the jnode, the Renode pane is filled. You can open the + ticks and get:

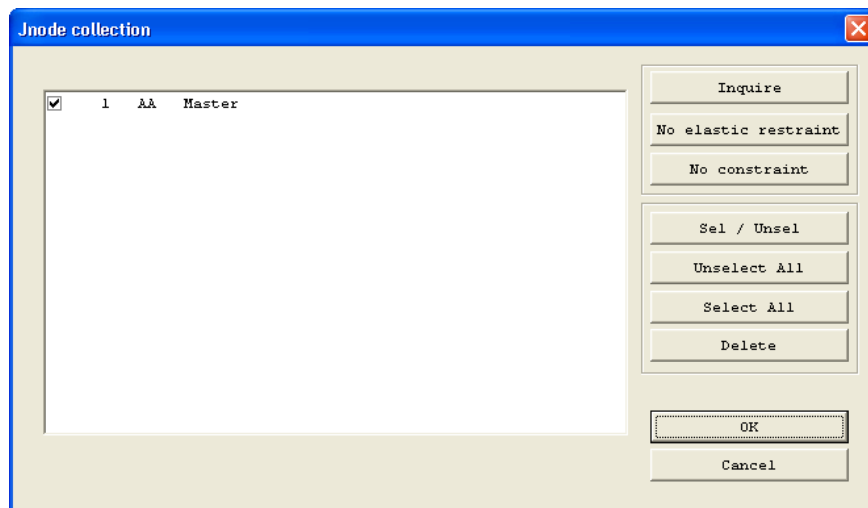


You get info about a Renode because you have selected one, and only one jnode. Here there is just one jnode, so you cannot select more than one. The renode is the 3D real world counterpart of a jnode. We have not built it yet, but here we get some info about the members. You will notice that each member is listed with its proper releases and cross-section, and material, and more. Notice, in particular, that the truss elements have been recognized as "tension rod or strut", and that the connection is "Axial". For hinged beams you find "Flexural hinge", for unreleased beams "Clamp".

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

Info about node Master

Number: 1 Mark: AA

Number of occurrences: 1 Nodes: 8

Number of members: 7 Members: 3 Extreme nodes: 7

Cuspidal: 0

Passing by: 1

Interrupted: 6

Number of trusses: 2

Number of beams: 6

OK Annulla Applica ?

and clicking to **"Master"** the following:

Info about node Master

Master

Master cross-section: HE 200 B

Element kind: Column

Slave

Slave cross-section: HE 200 B

Element kind: Tension rod or strut

Joint kind: Axial

Alignment: Inclined swapped by web

Envelope of internal forces in slave

Positive			Negative		
0.000e+000	N+	0	0.000e+000	N-	0
0.000e+000	T2+	0	0.000e+000	T2-	0
0.000e+000	T3+	0	0.000e+000	T3-	0
0.000e+000	M1+	0	0.000e+000	M1-	0
0.000e+000	M2+	0	0.000e+000	M2-	0
0.000e+000	M3+	0	0.000e+000	M3-	0

OK Annulla Applica ?

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 gives info about the master and the slave, as the current jnode has been recognized as of the "Master-Slave" kind.

Internal envelope actions are blank as this fem model has been prepared inside CSE and is not coming from a true fem analysis.





```
*
*
* ALPHA.....Angle between the considered member and the master, in a
*             hierarchic jnode
* ALIGNMENT....Parallelism between a couple of local axes. If it is "mn",
*             axis "m" of the master is parallel to axis "n" of the slave
* BETA.....Angle between the considered member and the first member of
*             the jnode, in a central, plane jnode
* JOINT.....All the components connecting two members
*
* JNODE.....The set of all joints of a set of members ideally connected
*             in the same point (node). A single jnode can be associated
*             to several nodes of a FEM node
*
* MARK.....Univocal identification of a jnode (for example AA, AB, etc.)*
*
* MEMBER.....Prismatic single part before workings. It is modelled with
*             one or more aligned finite elements (beams or trusses)
*   CONNECTED....A member having the jnode at hand as one of its extremes,
*             with a connection code in that extreme
*   PASSING.....A member having the jnode at hand as an internal node
*
* N.....progressive number of the jnode
*
* NM.....Number of members connected to the jnode
*
* NO.....Number of instances of the jnode in the structure
*
* NODE.....Numbered point in the space, in which one or more finite
*             elements arrive. It is used in a F.E.M. model
*   EXTREME.....Terminal node of a member
*
*   INTERNAL.....Node of a member that is not one of its extremes
*
* TYPOLOGY.....Overview description of a jnode
```

#### JNODES CLASSIFICATION

```
* Simple:          a single member is connected to a support
* Hierarchic:      there is only one master member in the node
* Central:         all the members are connected in the node
* Tangent:         there are two or more passing members
* Cuspidal:        there are two or more non-interrupted members
```

#### HIERARCHIC JNODES CLASSIFICATION

```
* Splice joint - homogeneous:      alignments 11-22-33
* Splice joint - exchanged:        alignments 11-23-32
* Splice joint:                    alignment 11
* Normal, homogeneous by flange:   alignments 22-13-31
* Normal, homogeneous by web:      alignments 33-12-21
* Inclined, homogeneous by flange: alignment 22
* Inclined, homogeneous by web:    alignment 33
* Normal, exchanged by flange:     alignments 23-12-31
* Normal, exchanged by web:        alignments 32-13-21
* Inclined, exchanged by flange:   alignment 23
* Inclined, exchanged by web:      alignment 32
* Centric by flange:               alignment 12
* Centric by web:                  alignment 13
* Plumbed by flange:              alignment 21
* Plumbed by web:                  alignment 31
```

```
*****
*
* UNITS:      mm      N
*
*****
```

```
*****
*
*
```



\* MEMBERS \*

NM	NODE1	NODE2	SECTION	LENGTH
1	8	14	HE 200 B	1.414e+003
2	8	15	HE 200 B	1.414e+003
3	7	9	HE 200 B	4.000e+003
4	8	10	HE 200 B	2.000e+003
5	8	11	HE 200 B	2.000e+003
6	8	12	HE 200 B	2.000e+003
7	8	13	HE 200 B	2.000e+003

SHAPE HE 200 B

Pieces of	1.4e+003 mm	Number	2	Weight	1.734e+003	N
Pieces of	4.0e+003 mm	Number	1	Weight	2.452e+003	N
Pieces of	2.0e+003 mm	Number	4	Weight	4.904e+003	N

Total length 1.483e+004 mm Total weight 9.089e+003 N

\*\*\*\*\*  
\* LIST OF JNODES FOUND \*  
\*\*\*\*\*

N	MARK	NO	NM	TPOLOGY
1	AA	1	7	Master

Internal actions not available

\*\*\*\*\*  
\* JNODE 1 MARK AA \*  
\*\*\*\*\*

GLOBAL TYPOLOGY OF THE JNODE: HIERARCHIC JNODE

NODES ASSOCIATED TO THIS JNODE: TOTAL 1  
8

MEMBERS IN THE JNODE: TOTAL 7

3 2 1 4 5 6 7

JOINT 1 (Beam -Truss) MASTER= HE 200 B SLAVE= HE 200 B  
Column Tension rod or strut  
Intermediate joint - Inclined swapped by web - Axial -  
ALPHA= 2.356194 ( 135.00°) COS= -0.707107 SIN= 0.707107 TAN= -1.00000e+000

JOINT 2 (Beam -Truss) MASTER= HE 200 B SLAVE= HE 200 B  
Column Tension rod or strut  
Intermediate joint - Inclined swapped by web - Axial -  
ALPHA= 2.356194 ( 135.00°) COS= -0.707107 SIN= 0.707107 TAN= -1.00000e+000

JOINT 3 (Beam - Beam) MASTER= HE 200 B SLAVE= HE 200 B  
Column Beam  
Intermediate joint - Normal swapped by web - Flexural hinge -  
ALPHA= 1.570796 ( 90.00°) COS= 0.000000 SIN= 1.000000 TAN= 1.63312e+016


JOINT 4 (Beam - Beam) MASTER= HE 200 B SLAVE= HE 200 B  
Column Beam  
Intermediate joint - Normal swapped by web - Flexural hinge -  
ALPHA= 1.570796 ( 90.00°) COS= 0.000000 SIN= 1.000000 TAN= 1.63312e+016

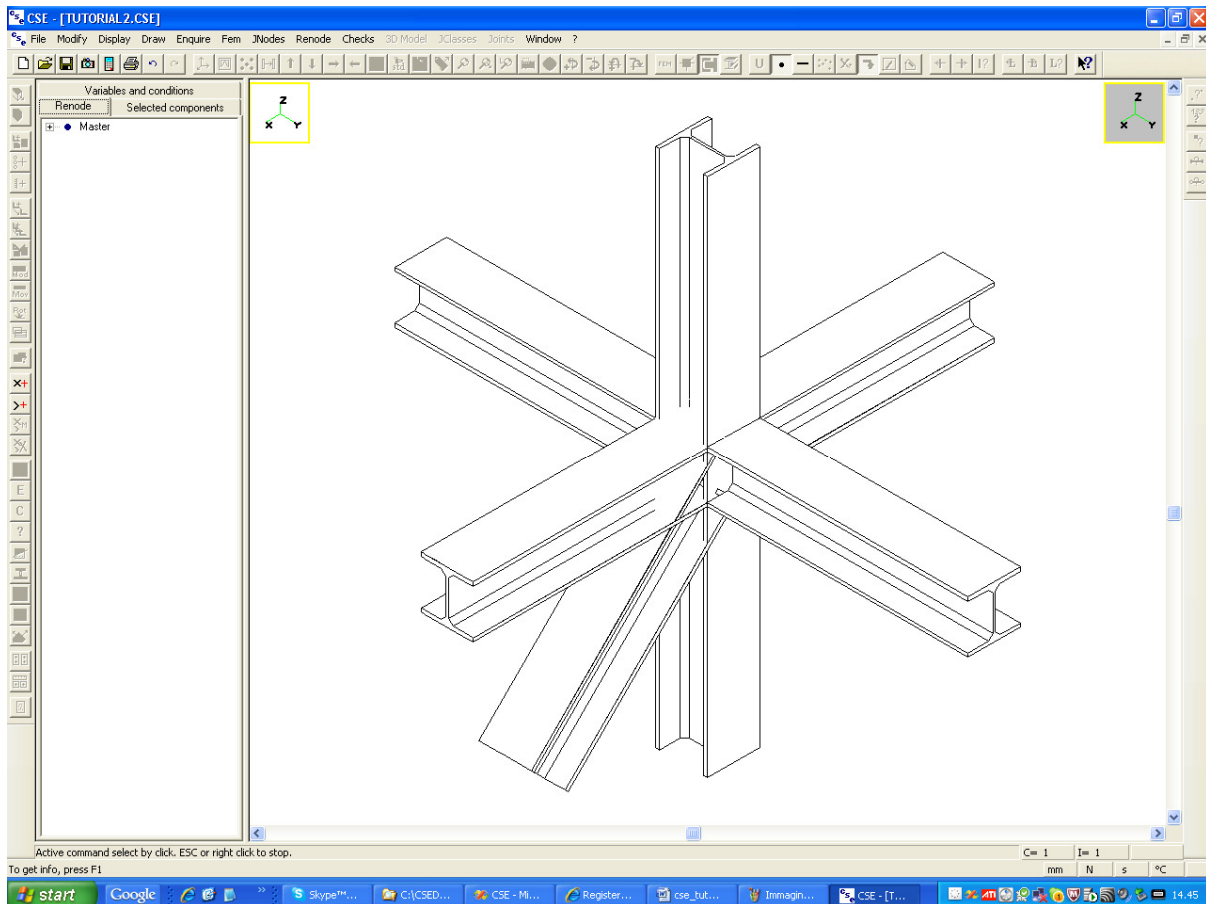
JOINT 5 (Beam - Beam) MASTER= HE 200 B SLAVE= HE 200 B  
Column Beam  
Intermediate joint - Normal homogeneous by flange - Clamp -  
ALPHA= 1.570796 ( 90.00°) COS= 0.000000 SIN= 1.000000 TAN= 1.63312e+016



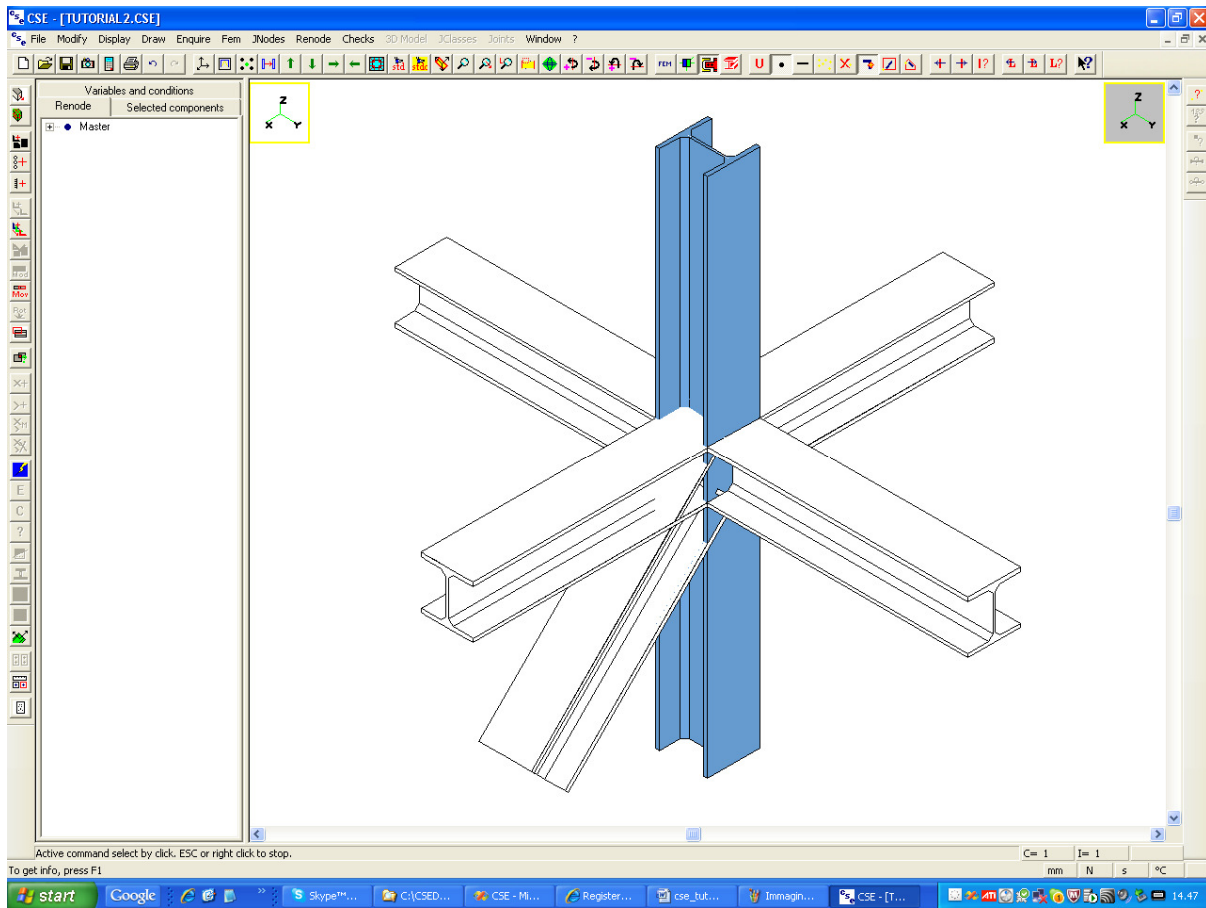
JOINT 6 (Beam - Beam) MASTER= HE 200 B SLAVE= HE 200 B  
Column Beam  
Intermediate joint - Normal homogeneous by flange - Clamp -  
ALPHA= 1.570796 ( 90.00°) COS= 0.000000 SIN= 1.000000 TAN= 1.63312e+016

## 2.8 STEP 8: SWITCH TO TRUE 3D VIEW TO BUILD RENODE (RENODE VIEW)

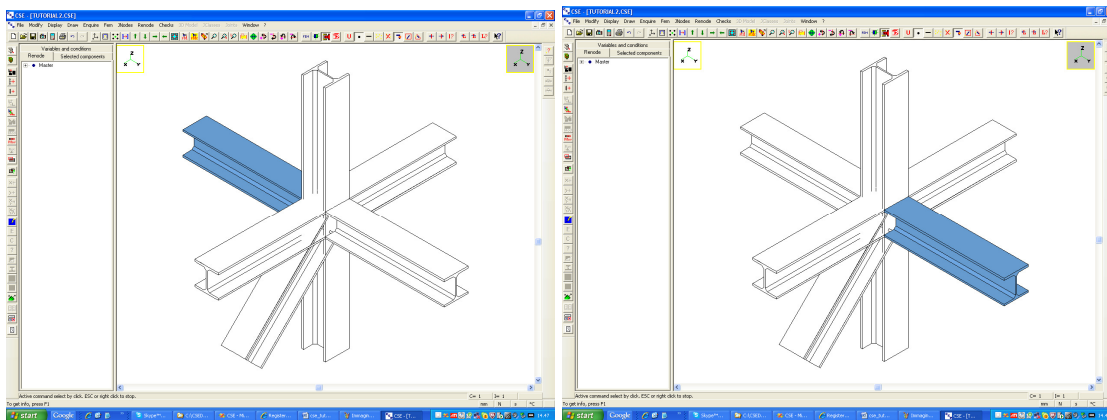
The tutorial is now finished. In a next one we will start from here to actually build the connection. However, as last step, let's switch to the renode view, i.e. to the environment where we will actually build the renode. If one and only one jnode is selected we can switch to renode view by pressing this button in the main bar (**Display-Renode**): . We will see what follows, i.e. the real node still not prepared.



However we can see that Y member are already correctly positioned facing the column flange, as requested by the offset we have applied. Also, clicking over the members we can see that the column is one member only



while the Y beam is made up by two members:



We are now ready to actually build up the connection. This will be explained in another tutorial.



<b>1</b>	<b><u>INTRODUCTION.....</u></b>	<b>3</b>
<b>2</b>	<b><u>HOW TO BUILD A SIMPLE DUMMY STRUCTURE USING FEM COMMANDS .....</u></b>	<b>4</b>
<b>2.1</b>	<b>STEP 1: GETTING THE FEM ELEMENTS.....</b>	<b>5</b>
<b>2.2</b>	<b>STEP 2: ASSIGNING CROSS-SECTIONS AND MATERIALS. ....</b>	<b>15</b>
<b>2.3</b>	<b>STEP 3 REVIEWING ELEMENT ORIENTATION .....</b>	<b>17</b>
<b>2.4</b>	<b>STEP 4: ASSIGNING END RELEASES AND CONNECTION CODES.....</b>	<b>22</b>
<b>2.5</b>	<b>STEP 5: ASSIGNING OFFSETS.....</b>	<b>25</b>
<b>2.6</b>	<b>STEP 6: SEARCHING MEMBERS.....</b>	<b>27</b>
<b>2.7</b>	<b>STEP 7: SEARCHING JNODES.....</b>	<b>28</b>
<b>2.8</b>	<b>STEP 8: SWITCH TO TRUE 3D VIEW TO BUILD RENODE (RENODE VIEW) .....</b>	<b>36</b>