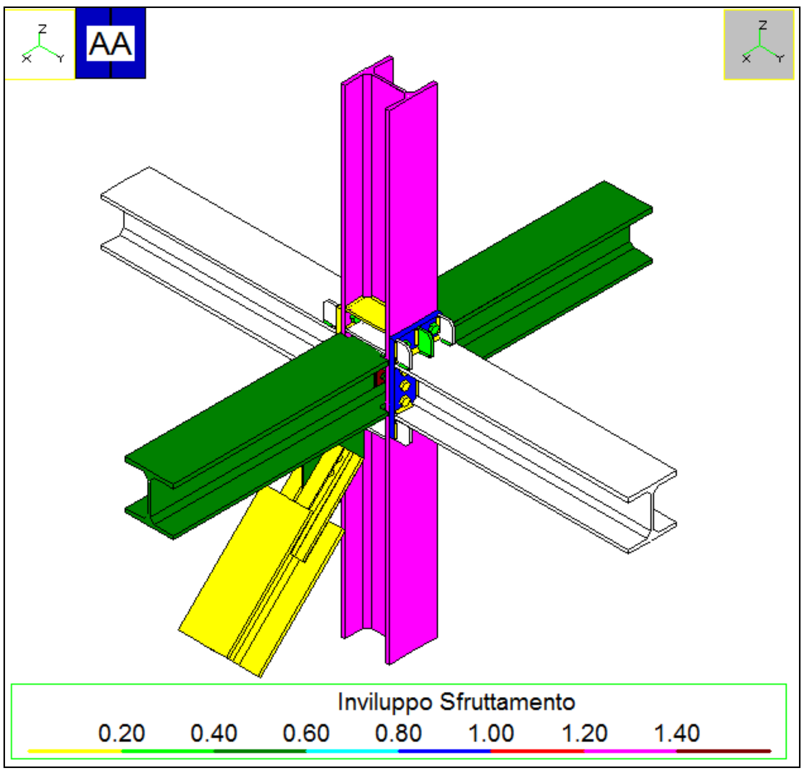
Paolo Rugarli

Connection Study Environment

Tutorial 3: building a multiple-members renode. Part 4



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

Referring to CSE Version 4.40
Rev.5 November, 04, 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 (part 4 of tutorial 3) is aimed at understanding basic post processing options in order to control check results, and to discuss them. The start point is the end of tutorial 003c (part 3 of tutorial 3).
By following this tutorial you will be able to:

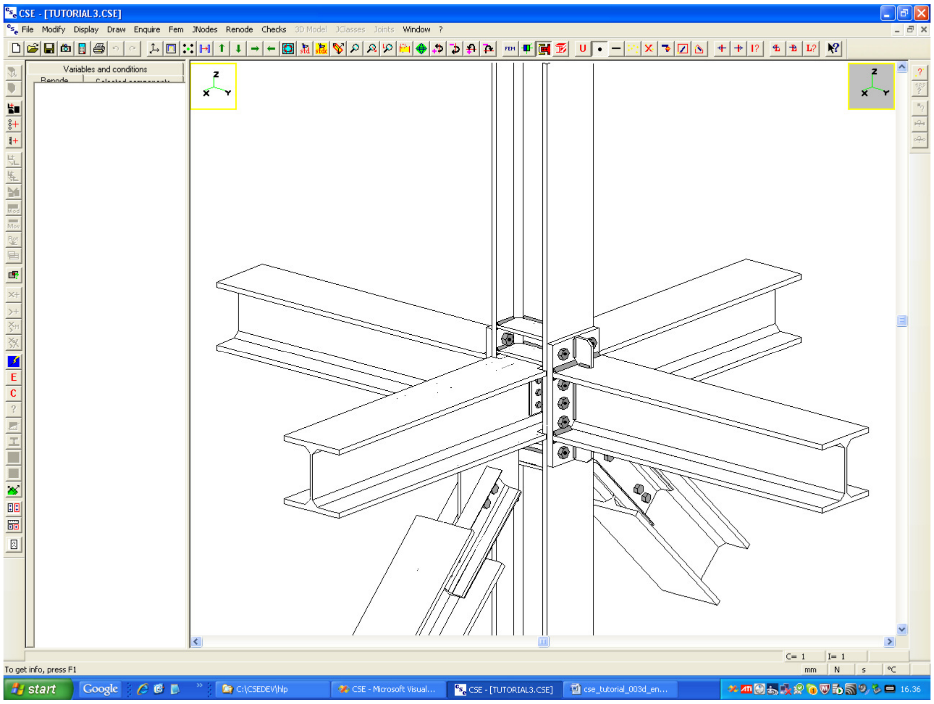
- Build up a complex renode adding plates, angles, bolts and welds
- Apply cuts, bevels, shorten and lengthen members
- Understand computing options of bolt layouts and of members and components
- Understand how to use the left side panes
- Understand how to add new variables
- Understand how to add user-defined checks
- Export a DXF to be processed by other CAD programs

This tutorial is some like 58 pages long because we have explained step by step anything with images, however it takes very few minutes to actually do these things.

This is the 4th part of a set of 4 parts. In this part we will execute the checks and discuss results.

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

2 HOW TO BUILD A MULTIPLE MEMBERS RENODE



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

2.1 FOREWORD

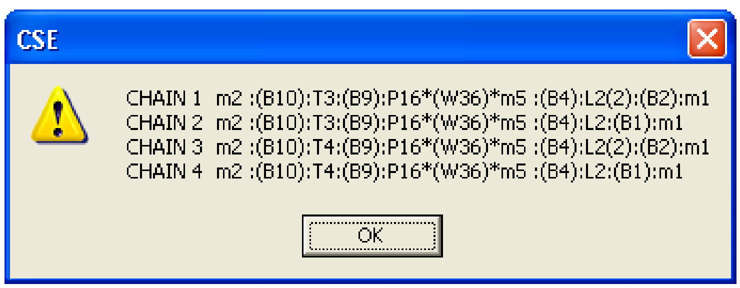
Up to now we have essentially built up the *scene*. We have added components to create connections, and modified members to match our idea of how connections should be done. We wish to point out the following:

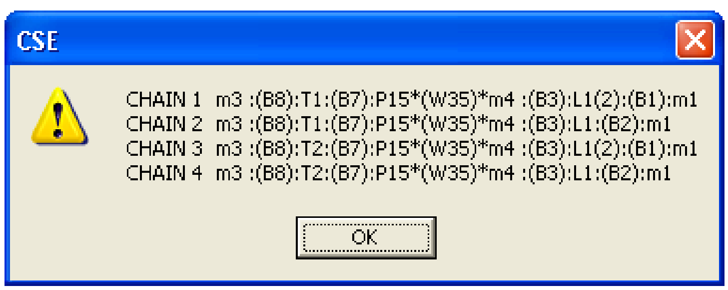
1. In doing this we have been free to put components where we wished, and to join them with a quite general set of tools. Bolts can be put where we want as well as fillet welds, plates, trunks, and so on.
2. We have executed a set of commands and got what we had in mind, but not necessarily we have always followed the shortest way, as our aim was also to see how commands work and how you can *study* a connection from within CSE.
3. The connection itself is clearly not optimal, this has been done also in order to show that CSE is able to detect these possible problems.
4. The commands we have used are the more general ones, no macro or automatic building up of typical joints has been used. This is because this program is born with the aim of solving the most general problem possible. However this proves that the program is able to deal with

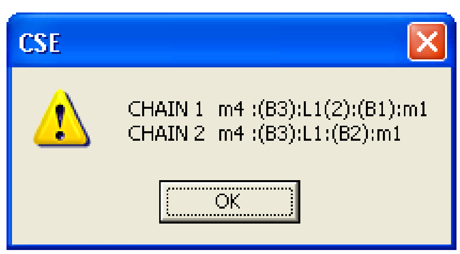
complex, un-programmed scenarios, that is, basically, that is able to manage the configuration you need. Macro commands, and typical joints management are feature that will be added in a later development stage, now that the general tools have been set up. These macro commands will be got by adding higher level commands that will make automatic what now is done step by step (for instance adding a plate welded to the end face of a member will get one only command). At the present stage of development CSE has been developed to solve the most general problems. However, no matter the pages used to carefully explain each thing, the construction of a complex renode is usually done in 10-20 minutes. Then is the program to check for hundreds combinations.

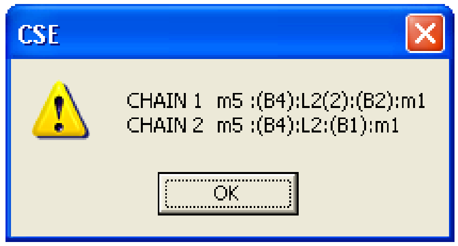
2.2 STEP 6: CHECKING RENODE COHERENCE AND OVERLAPS

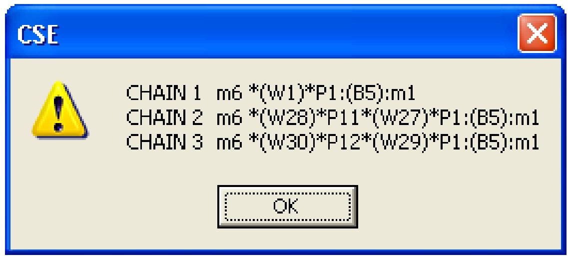
Before we move forward with the checks we first would like to be sure that there is no problem in the connections: components not properly joined or completely free, for instance. To do that we execute the command **Renode-Check coherence**. As soon it is executed the following messages do appear:

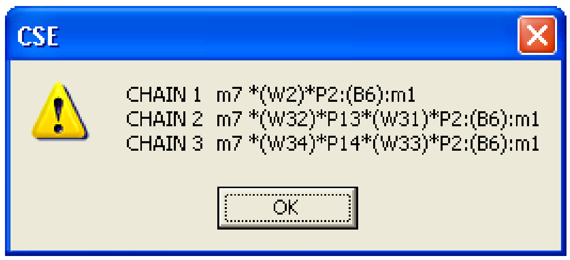




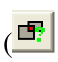








Each message list the chains connecting member "i" to the master (i.e. member 1, the column). As no error message has been issued, we are sure that all components are properly connected (the command is also used to re-check proper connections of joiners, so if a joiner is misplaced, here is where you can find it).

Now let's see if there are overlaps. Execute the command **Renode-Check overlaps** ( button in the left bar).

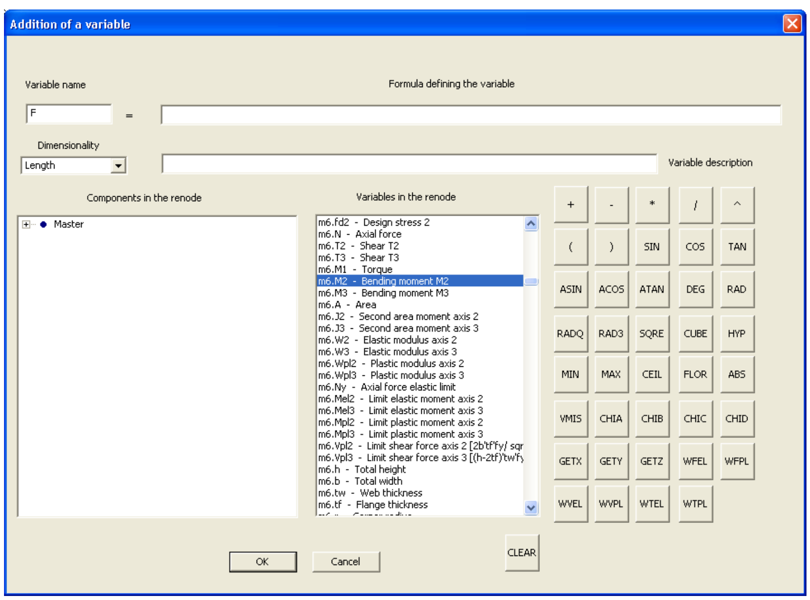
This message assures that there are no overlaps.

2.3 STEP 7: ADDING USER CHECKS

The program executes quite a large number of checks. However the user can add his/her checks in order to be sure that some possible failure modes are avoided. User checks may help to avoid unnecessary fem analyses, or to add specific tests to what is done by the program. This tool has been conceived to be general and flexible. We will use already defined standard variables to possibly define new variables. We then will use both standard and added variables to execute specific checks. These can be pre-conditions that the connections must satisfy, or true checks. In any case, when checks will be executed, these "user checks" will also be executed, and the results added to those got by standard program work.

We are concerned about a possible failure mode, that we wish to check in addition to standard checks. This is the buckling check of the stiffeners used to bear the Y beam bending. This check is not among those performed automatically by the program as the stresses in the internal stiffeners (those only connected to one component different parts) can be computed only via fem analysis of their owner (the column). To perform this check, as we would do in by hand calculation, we will make some simplifying assumption.

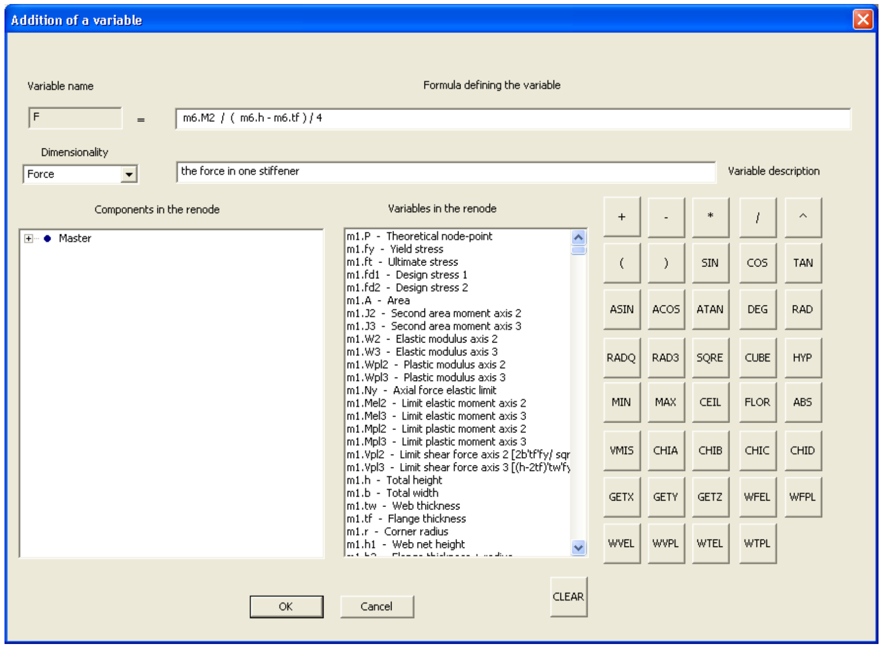
Click left in the left pane to make it active. Execute the command **Renode-Variables and conditions-Add variable** (button in the left bar). The following dialog box appears:



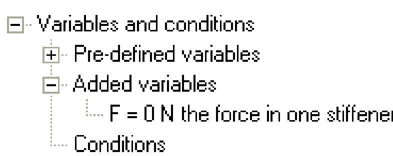
Input "F" as variable name. Choose among the pre-defined variables the variable m6.M2, and double click. It is written in the formula edit box. This, m6.M2, is the strong axis bending of member m6, which is one of the two Y beams, the positive side one. Its value is computed runtime, during the checks. Presently its value is 0, so F will be 0. However, runtime, the true m6.M2 value will be used and so F will change. Using the predefined variables and the operator buttons, or standard keyboard keys, enter the following formula for F

m6.M2 / (m6.h - m6.tf) / 4

and the following description for the new variable: "the force in one stiffener". Then choose the correct dimensionality for the variable, here is a Force, like this:

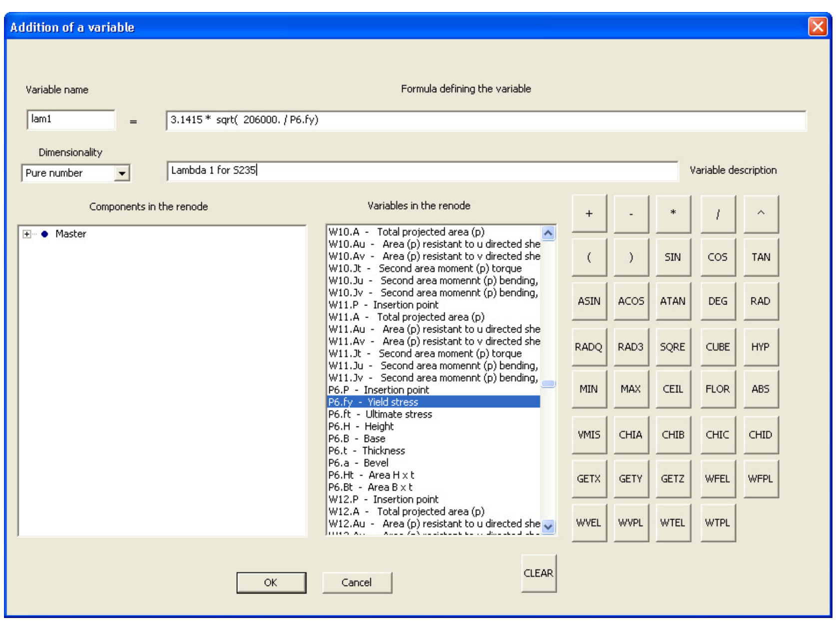


Press **Ok** and have a look at the **Variables and Conditions** pane, on the left, section **Added Variables**. You will find F:



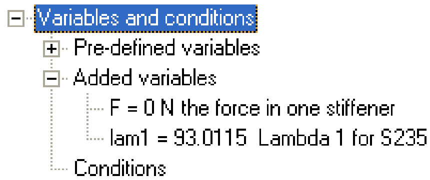
The value is 0 for the reason explained.

Now add one more variable, re-executing the command, and name it *lam1*, like this:

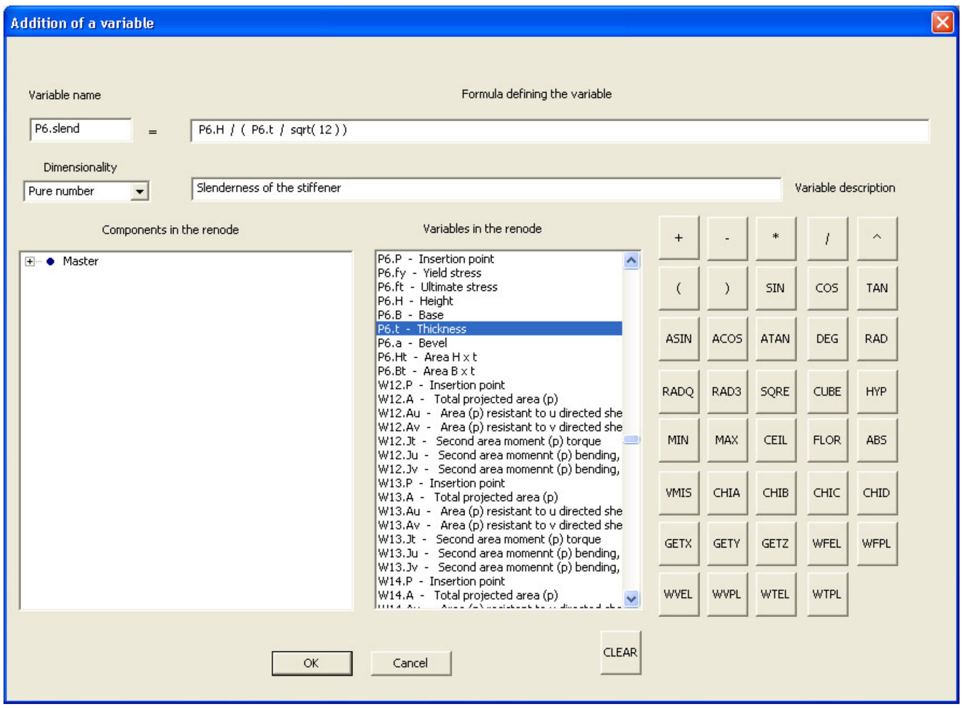


We have used P6.fy as P6 is one of the stiffeners involved in the check.

Notice that the value has been computed:

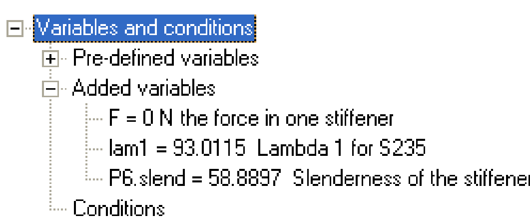


Now add a new variable and call it P6.slend:

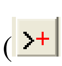


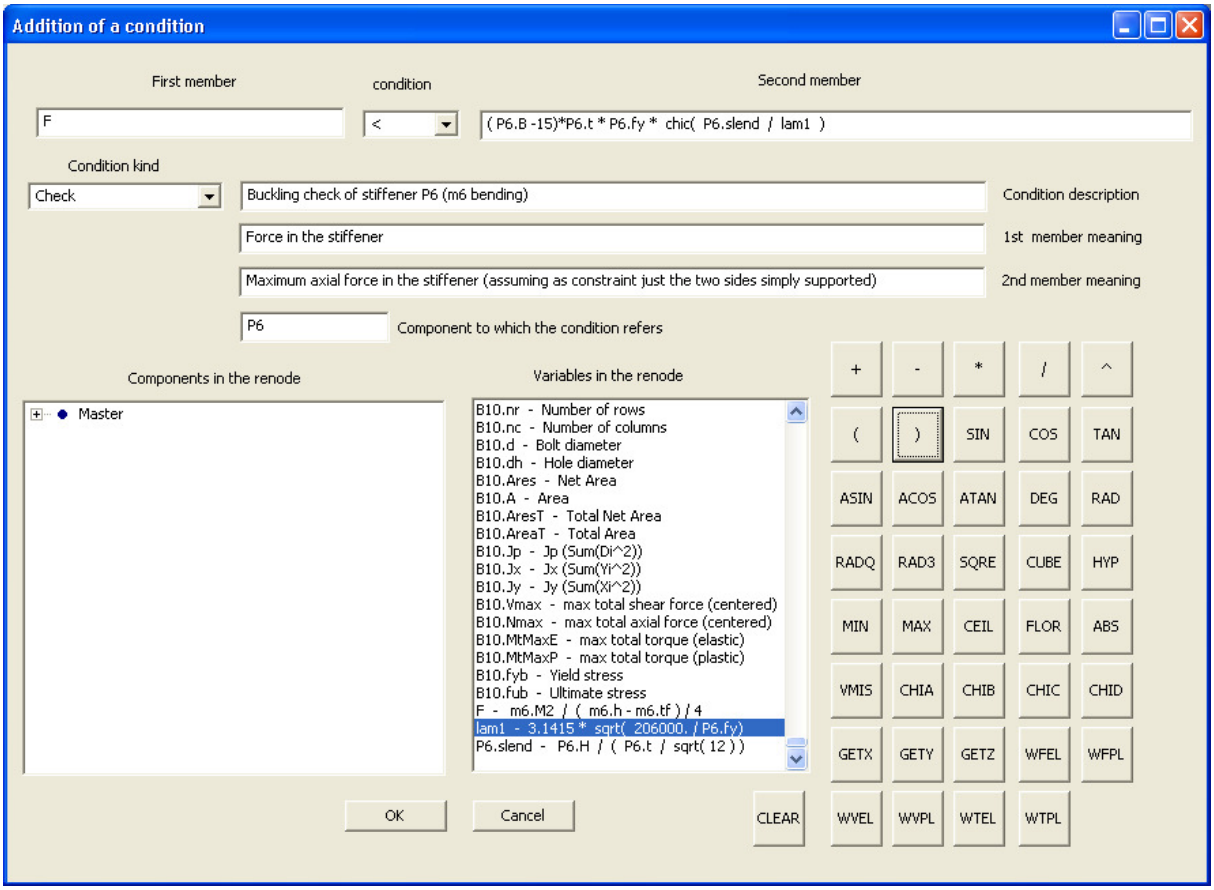
This is the slenderness of the stiffener.

Notice its value:



We have all the helpers we needed. We can now add a new condition (we could also add the condition without the need of defining helper variables, but adding them formulas are shorter).

Execute the command **Renode-Variables and conditions-Add condition** ( button in the left bar), and complete the dialog as explained:



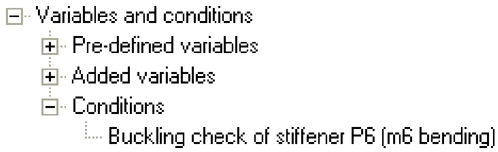
Notice that in the list of the available variables, added variables have been added at the bottom. Notice that we have described a check and linked this check to component P6 (**component to which the condition refers**). The formula is soon explained:

F < (P6.B -15)*P6.t * P6.fy * chic(P6.slend / lam1)

- F: axial force in the stiffener
- (P6.B -15) *P6.t: it is the stiffener axial area neglecting bevel.
- chic() is $\chi(\lambda/\lambda_1)$ of the slenderness curve c according to Eurocode. The yield stress is reduced to keep into account the slenderness. The buttons "CHIA", "CHIB", "CHIC", "CHID" implement buckling curves "a", "b", "c", "d", reduction factors according to Eurocode 3.

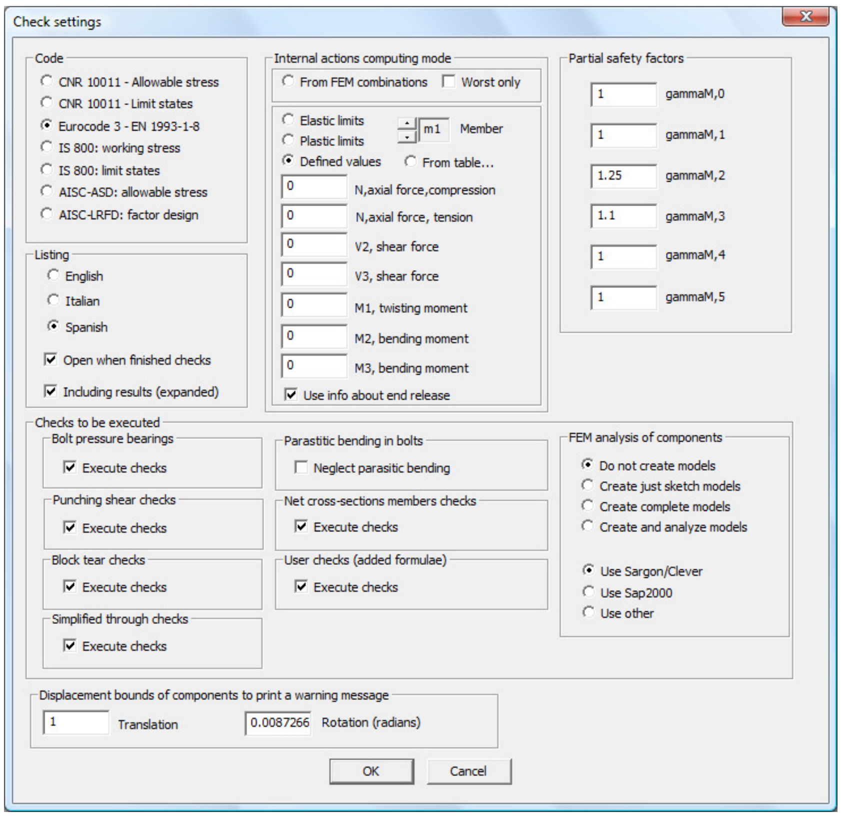
Note that this formula is pretty much on the safe side, as clearly the force will be lower, the stiffener is probably clamped and it is joined in three sides, not just two. However, if this check will be satisfied (for all combinations), then clearly the stiffener buckling will not be a problem.

Now in the "Conditions" section of the Variables and Conditions pane, has appeared our new condition:



2.4 STEP 8: SETTING CHECK OPTIONS

We are now ready to execute the command Checks-Set in order to properly choose what checks will be done, and how. Click in the right part of the program window to activate it. Now execute the command Checks-Set and set data in this way:



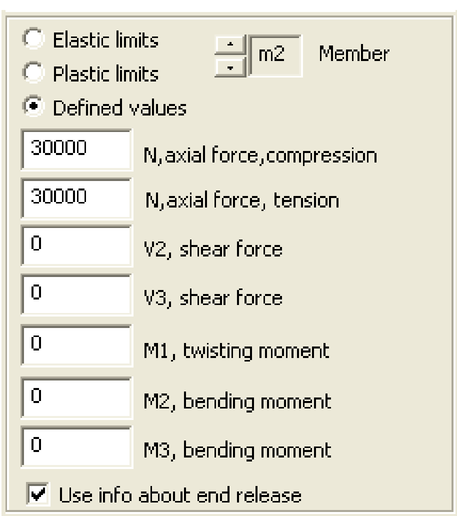
1. We will use Eurocode 3
2. The listing will be in English
3. The listing will be opened at the end of the checks and it will include the results
4. The connections will be checked using **defined values** for the actions, i.e. we will set each member actions. We could also choose fractions of the **elastic or plastic limits** or import combinations from a table. As this model has not been imported by an outside fem model, we cannot choose **As in load combinations (from FEM)**, which is the other possible choice. Since this is a Hierarchical renode, the slave connections to the master will be checked so the master actions are not meaningful (all 7 data for the master - m1 - can be set

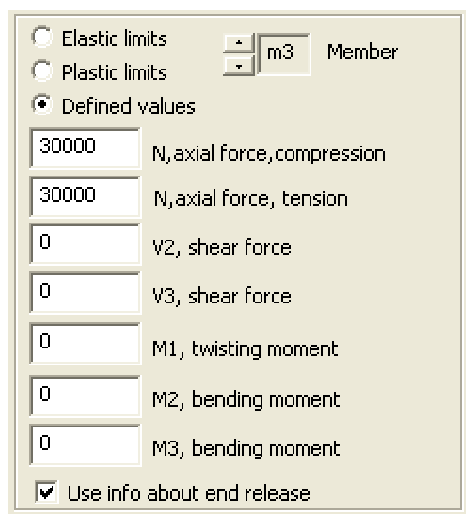
equal to 0: from **N axial force** to **M3 bending moment**). Later, using the arrow keys to change the member, we will set the internal actions in the members, to be checked.

5. The γ_M for the material are set as default values as the standard suggests, but you can change them (**partial safety factors**).
6. Among the "**Checks to be executed**" we choose all checks, but we leave "**Do not create models**" in the FEM analysis of components box. This is because this part will be done later. FEM checks are to deepen stress knowledge of components and are a tool to use selectively in order to avoid an excess of number crunching. Basically they should be used when needed. Later in this section we will discuss the checks made by the program.
7. The "**displacement bounds of components to print a warning message**" is a useful field in order to check displacements. A connection can be ill designed not only if its resistance checks are not passed, but also if the displacements are too high. Typically if some "shear only" bolt layout is in tension or compression, or flexure, and no other joiner sufficiently stiff is capable of taking the actions, then the displacements will be high. This is due to the very low axial stiffness of bolts in "shear only" bolt layouts.

Now using the arrow keys set as active member the member m2. This is one of the two diagonals.

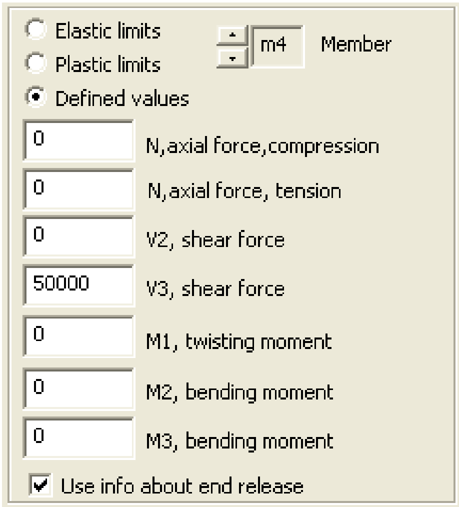
Assume that a 30000N force (30kN) acts in the diagonal. So set the values in this way (left):

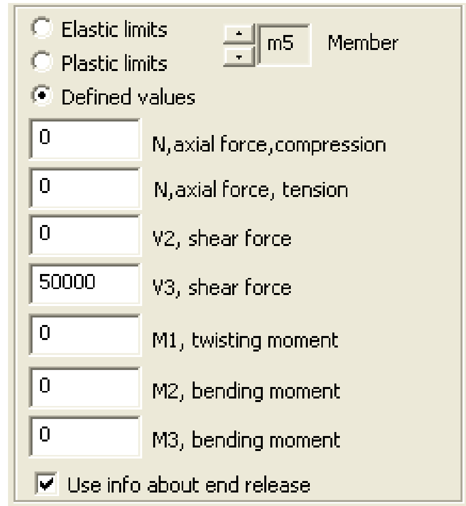




Repeat the same input for member m3 (above, right), which is another diagonal. They both are trusses so just the axial force will be used. Notice that the compression force must be input positive. Also note that as we have chosen "**Defined values**" these are forces and moments in the current active unit.

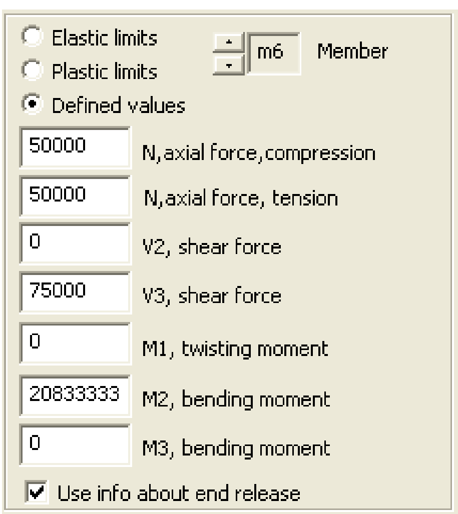
Now for member 4 and 5 (X axis beams) enter the following data:

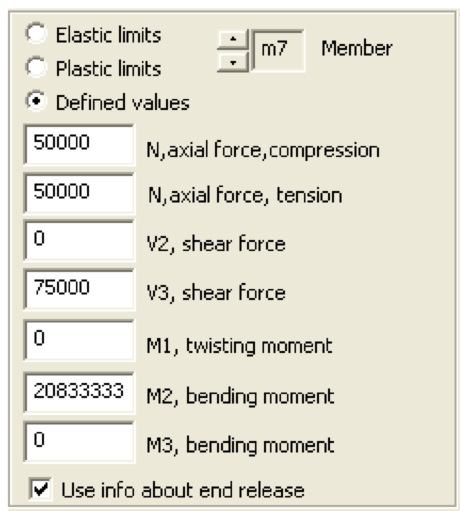




This is shear force V3. Imagine a 3.3333kN/m^2 distributed load. A beam with a span L=5m. A distance between beams i=4m. We have: V3= 1,5 x 3.3333 x 4 x 5 x 0.5 = 50kN, where we used γ_q=1,5 load factor (i.e. *the values input are factored loads*).

Now for members m6 and m7 (the clamped Y beams) enter for instance the following numbers:



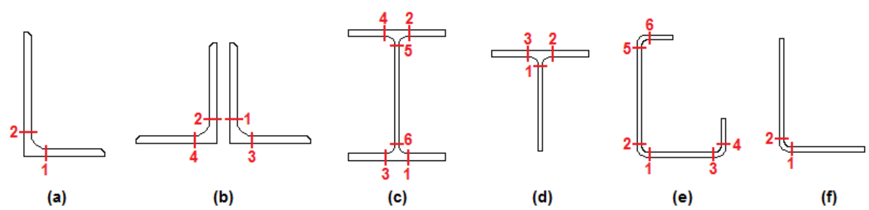


(50kN as axial force, 75 kN as shear force, 20.833kNm as bending). However you basically can enter what you like, it's up to you.

The following checks will be done for all combinations:

1. Resistance checks of bolt layouts (bolts under N, M, V). This is always done. If there are anchors or slip resistant bolts, the slip and the pull out will also be checked.
2. Resistance check of weld layouts (fillet welds). This is always done.
3. Bearing bolt pressure checks for all bolted components: we asked it (**bolt pressure bearings** check box).

4. Punching shear checks for bolted objects.
5. Net cross-section member check. We asked it (**Net cross-sections members check**); members will be scanned by planes normal to their axis getting true sections (due to bolt holes and cuts). These true cross sections will be checked for axial force + bending due to single bolts and single fillet welds (or part of fillet welds) actions. No shear check is performed (usually this is done by block tearing checks, but we can add user check if we need them).
6. Block tear checks. This will be done for all bolted components (plates, angles, members) using component bolted face and very general rules. We asked for these checks (**block tear checks**).
7. Simplified checks for through. In this model applies to angles and to UPN trunks. Flat plates are not checked by this option. Cross-section trunks are checked assuming section cuts as relevant cross-section (see picture below, from the guide). Shear is included in the checks as the sections checked are rectangular. Here one can define to keep into account torsion or not: this was one of the flag we decided for each component, when adding them. These flags can of course be modified when needed. The effects of single bolts and single welds (or part of welds) over the highlighted cross-sections will be taken into account. One more check is done at mid of trunk length using the cross section extruded. We asked for these checks: **simplified through checks**.



8. User checks. These are the checks defined by the user (**user checks added formulae**): we asked for them.
9. Bearing component checks in bolt layouts using bearing surface. This applies to the column, in this example, as we asked that the pressure acting over the column flange do not exceed $3N/mm^2$. We shall better see in the next sections.
10. Finite element checks: these are not presently checked.

The checks done automatically by the program are quite many. However it's clear that some component behaviour will not checked unless using fem analysis or specific user checks. For instance here the true local bending of the column flange and the true bending of the plate joining the Y beam is clearly one interesting point that we will deepen using fem analysis (if user checks have not be put in place, to avoid this need).

2.5 UNDERSTANDING THE COMBINATIONS CREATED

As we know, this model was not imported by a fem analysis. No true analysis of this connection is available as there is no fem model. Nonetheless, the program is capable of executing checks by generating dummy load combinations in order to test the connections. Clearly this is not as when using a fem model, however most connections can be designed.

The user inputs the fraction of elastic or plastic limits to be used for each member and for each internal force component, or, the user inputs directly the forces and moments to be used for each member. In hierarchical renodes master internal forces are not important: the program will check each slave connection to the master in 24 different combinations of internal forces at the extreme of that particular slave. So, if we have - as here - 7 members in the renode there will be 24x7=168 load combinations. They are organized in this way:

1-24	master or first member combinations;
25-48	member 2 load combinations
49-72	member 3 load combinations
73-96	member 4 load combinations
97-120	member 5 load combinations
121-144	member 6 load combinations
145-168	member 7 load combinations.

Within the 24 combinations of each member actions are organized in this way:

Combi 1	positive axial force N
Combi 2	positive shear force V2
Combi 3	positive shear force V3
Combi 4	positive torque M1
Combi 5	positive bending M2 (strong axis)
Combi 6	positive bending M3 (weak axis)

From 7 to 12 the signs are negative:

Combi 7	negative axial force N
Combi 8	negative shear force V2

Combi 9 negative shear force V3

Combi 10 negative torque M1

Combi 11 negative bending M2 (strong axis)

Combi 12 negative bending M3 (weak axis)

From combination 13 to 24 each combination embeds a mix of N, M2, M3. Like this:

Combi 13 $0.5N_p + 0.5\ M_2$

Combi 14 $0.5N_p -0.5\ M_2$

Combi 15 $0.5N_p+0.5M_3$

Combi 16 $0.5N_p-0.5M_3$

Combi 17 $-0.5N_m + 0.5\ M_2$

Combi 18 $-0.5N_m -0.5\ M_2$

Combi 19 $-0.5N_m+0.5M_3$

Combi 20 $-0.5N_m-0.5M_3$

Combi 21 $0.5M_2+0.5M_3$

Combi 22 $0.5M_2-0.5M_3$

Combi 23 $-0.5M_2+0.5M_3$

Combi 24 $-0.5M_2-0.5M_3$

So if we wish to see what happens due to a compression in member 6 we must look at combination:

$$24x5+7=127$$

This method is kept also if the combinations are blank (as some internal action is considered 0), so that the rule is always the same.

This working mode bypasses the problem due to the lack of a finite element model and allows to study connections independently from a fem model. Clearly is better to have a finite element model as there can be interaction effects between the members which are not kept into account by this kind of analysis, which tests the member connections one member at a time.

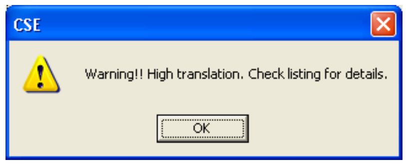
If we were using a fem model, than the combinations would have been those of the fem model. In standard CSE installations (no the CSE demo) you can use Sargon Demo to generate a fem model to be imported into CSE.

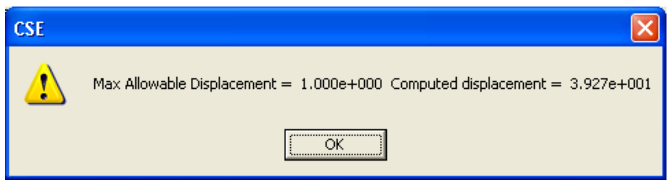
One more explanation.

If there is more than one "instance" (i.e. repetition) of the same renode in different part of the structure, the combinations to be checked are to be multiplied times the number of instances of the renode in the structure. The first set of n combinations will refer to the first instance, the second set to the second instance, and so on. So n+1 will be the first combination of instance 2. Here we have 1 instance only.

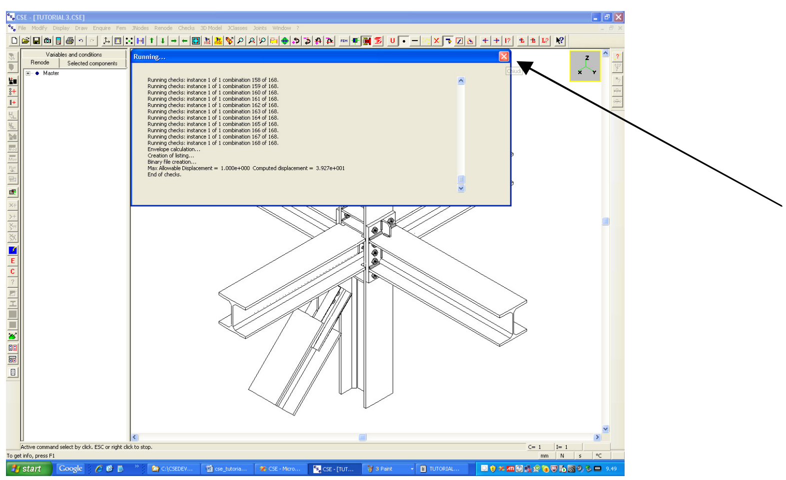
2.6 STEP 9: EXECUTING FIRST LEVEL CHECKS

Press OK and leave the check settings dialog box. Now execute the command **Checks-Check renode!** (the button in the left bar). A window (log window) opens and scrolls. During the execution you get these messages:





The program warns us that there is an high displacement, higher than the bound specified. Note that this displacement is an estimate, what is important is the order of magnitude. We shall later understand why. At the end of the job you see:



When the execution of the checks is ended a new WORDPAD window opens (as we asked it) with the listing. Close the log window clicking over the white cross over red background, at top right corner, as shown in the picture above.

Here is the log window content:

```
Beginning of checks...
Connection analysis...
CHAIN 1 m2  :(B10):T3:(B9):P16*(W36)*m5  :(B4):L2(2):(B2):m1
```

```
CHAIN 2 m2 :(B10):T3:(B9):P16*(W36)*m5 :(B4):L2:(B1):m1
CHAIN 3 m2 :(B10):T4:(B9):P16*(W36)*m5 :(B4):L2(2):(B2):m1
CHAIN 4 m2 :(B10):T4:(B9):P16*(W36)*m5 :(B4):L2:(B1):m1

CHAIN 1 m3 :(B8):T1:(B7):P15*(W35)*m4 :(B3):L1(2):(B1):m1
CHAIN 2 m3 :(B8):T1:(B7):P15*(W35)*m4 :(B3):L1:(B2):m1
CHAIN 3 m3 :(B8):T2:(B7):P15*(W35)*m4 :(B3):L1(2):(B1):m1
CHAIN 4 m3 :(B8):T2:(B7):P15*(W35)*m4 :(B3):L1:(B2):m1

CHAIN 1 m4 :(B3):L1(2):(B1):m1
CHAIN 2 m4 :(B3):L1:(B2):m1

CHAIN 1 m5 :(B4):L2(2):(B2):m1
CHAIN 2 m5 :(B4):L2:(B1):m1

CHAIN 1 m6 *(W1)*P1:(B5):m1
CHAIN 2 m6 *(W28)*P11*(W27)*P1:(B5):m1
CHAIN 3 m6 *(W30)*P12*(W29)*P1:(B5):m1

CHAIN 1 m7 *(W2)*P2:(B6):m1
CHAIN 2 m7 *(W32)*P13*(W31)*P2:(B6):m1
CHAIN 3 m7 *(W34)*P14*(W33)*P2:(B6):m1

Connection analysis ended.
Computing-forces loading...
Computing-forces loading ended.
Creation of info about renode...
Creation of info about renode ended.
Creation of the internal fem-model...
Creation of the internal fem-model ended.
Solving of the internal fem-model...
Solving of the internal fem-model ended.
Beginning of automatic checks.
Printing of initial info in the listing.
Beginning of automatic checks...
Running checks: instance 1 of 1 combination 1 of 168.
Running checks: instance 1 of 1 combination 2 of 168.
Running checks: instance 1 of 1 combination 3 of 168.
Running checks: instance 1 of 1 combination 4 of 168.
Running checks: instance 1 of 1 combination 5 of 168.
[...............................................................]
Running checks: instance 1 of 1 combination 167 of 168.
Running checks: instance 1 of 1 combination 168 of 168.
Envelope calculation...
Creation of listing...
Binary file creation...
Max Allowable Displacement =  1.000e+000  Computed displacement =  3.927e+001
End of checks.
```

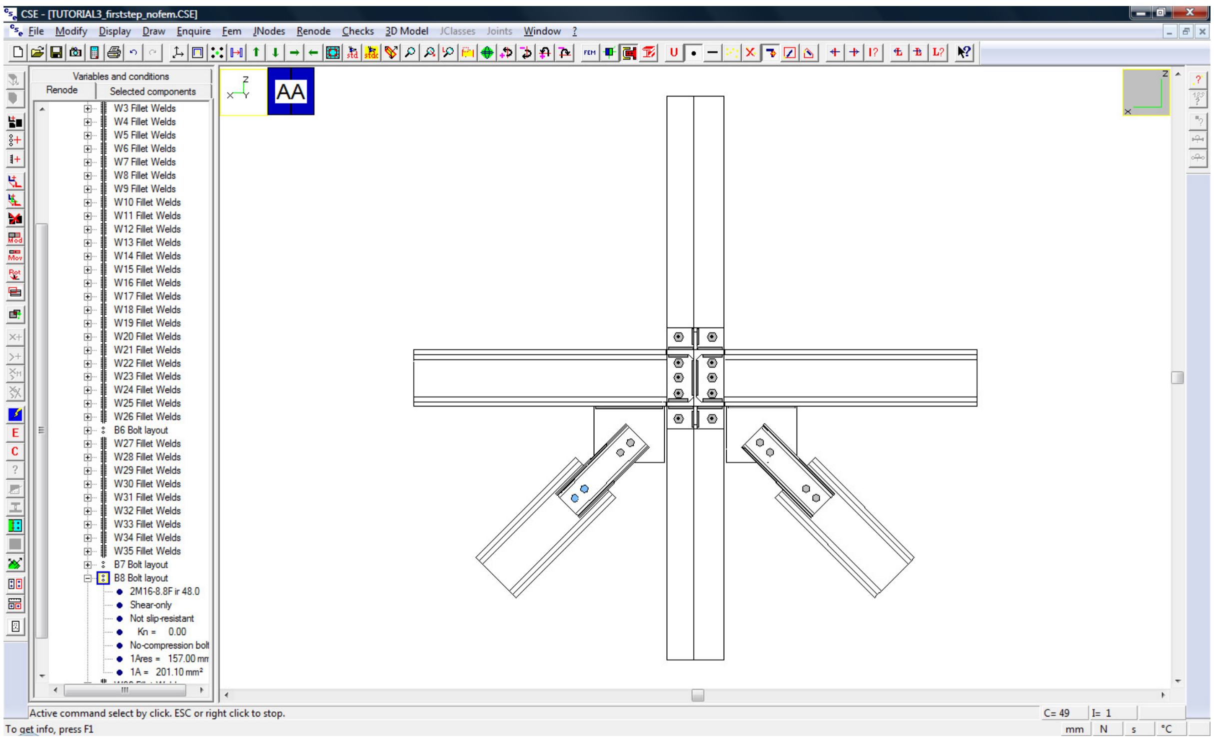
2.7 STEP 10: LOOKING AT RESULTS

2.7.1 Listing

As the listing is already open we can look at it. In particular as a high displacement has been detected, at the end of the listing we will find more info. We find:

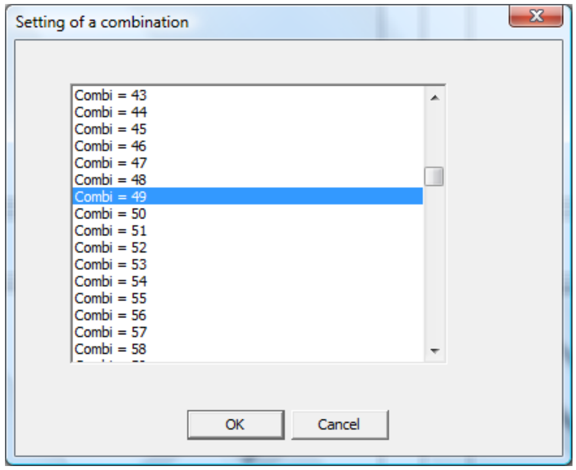
```
---------------------------
Notional Displacement info
---------------------------

Maximum translation        Instance    Combination    Component

    3.927e+001                 1            49            B8


Maximum rotation           Instance    Combination    Component

    4.778e-003                 1            25            B4
```

Combination 49 is a traction (axial force) in member 3, i.e. the diagonal. B8 is a bolt layout joining a plate to a diagonal:

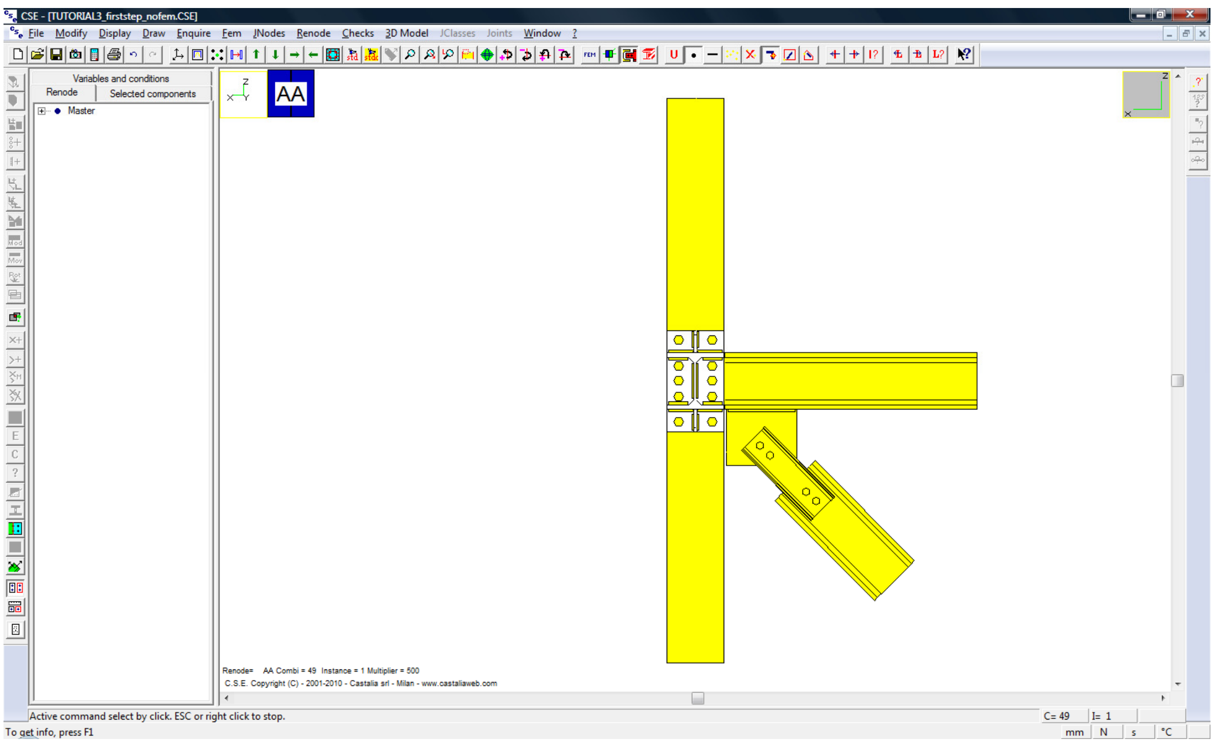


To understand which is B8 double click in the left pane over the B8 row: it gets yellow, selected. You can then have a look in the scene at what component is blue (selected in right view).

Now what can have happened? To understand select combination 49 by executing the command **Checks-Combi?** (the  button in the main bar), choose combination 49 in the following dialog:

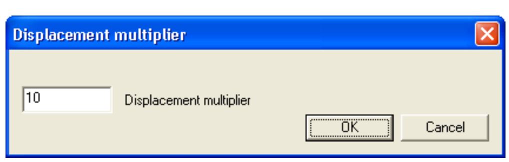


Now let's have a look at the displaced. Execute the **Checks-Displaced** command (button in the left bar), you see (using a +Y view):

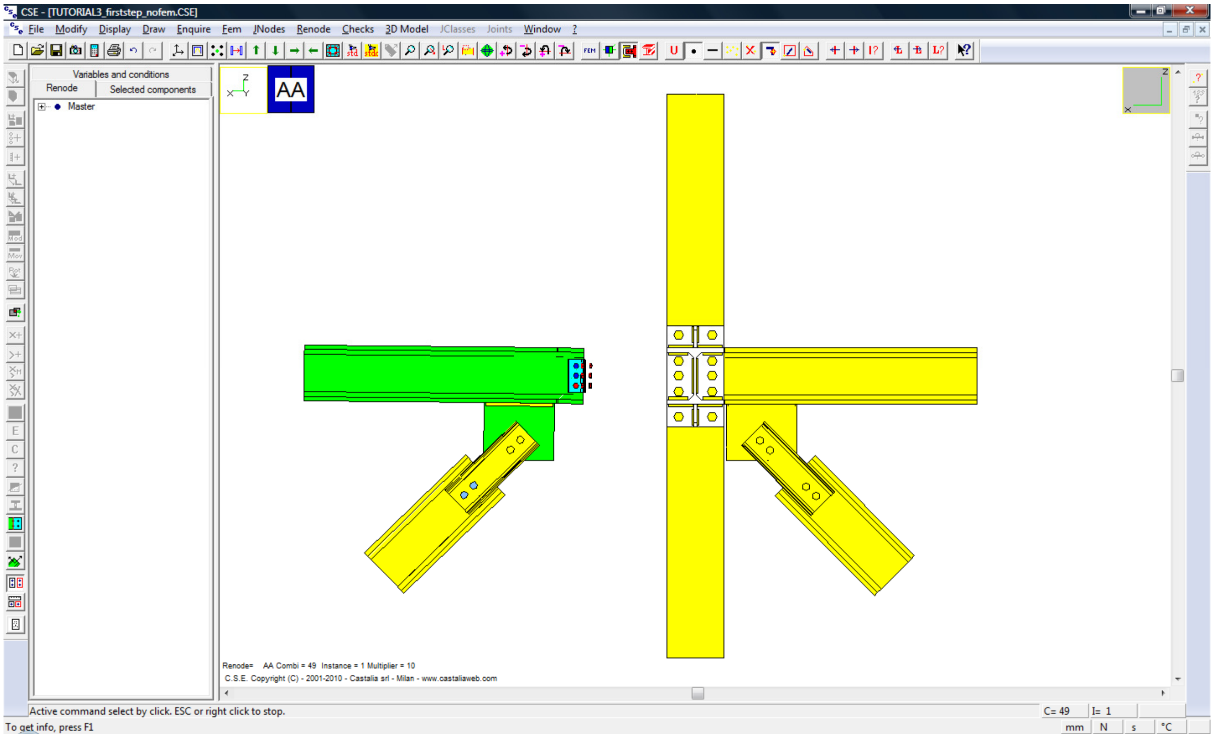


This picture was pasted by CSE after using the **File-Photograph** command (the button in the main bar).

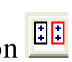
Clearly the displaced scale is too high. Execute the command **Checks-Displaced scale** (the button in the left bar) and set 10 as displacement multiplier:



You now see this:



It is now clear what has happened. When defining the bolts joining the X beam to the column we did not care about the traction of the X beam (as we were interested just in the shear). However when we joined the diagonals we used a connection which transferred axial force to the joint between the X beam and the column. Now the analysis has correctly detected this problem: forces have been transferred by the shear-only bolt layouts (as there is no other possible mean to transfer those forces), but as their stiffness is very low a high displacement occurred. So we will have to change those bolts removing the "shear only" flag. This explains the high displacements and also explains to us how our connection is working: the diagonals do transfer forces in the X dir to the column. So if ever we were not aware of this fact, the displaced is a clear recall.

Exit from the displaced command re-pressing the button  in the left bar.

Let's go back to have a look at the listing.

This is quite comprehensive, full of details, in input and output. All components and joiners are listed. We would like to have a look at a couple of things.

This is a recall of the action used:

```
Member m 1 Actions: Np-> 0.000e+000 Nm-> 0.000e+000 V2-> 0.000e+000 V3-> 0.000e+000 M1-> 0.000e+000 M2-> 0.000e+000 M3-> 0.000e+000
Member m 2 Actions: Np-> 3.000e+004 Nm-> 3.000e+004 V2-> 0.000e+000 V3-> 0.000e+000 M1-> 0.000e+000 M2-> 0.000e+000 M3-> 0.000e+000
Member m 3 Actions: Np-> 3.000e+004 Nm-> 3.000e+004 V2-> 0.000e+000 V3-> 0.000e+000 M1-> 0.000e+000 M2-> 0.000e+000 M3-> 0.000e+000
Member m 4 Actions: Np-> 0.000e+000 Nm-> 0.000e+000 V2-> 0.000e+000 V3-> 5.000e+004 M1-> 0.000e+000 M2-> 0.000e+000 M3-> 0.000e+000
Member m 5 Actions: Np-> 0.000e+000 Nm-> 0.000e+000 V2-> 0.000e+000 V3-> 5.000e+004 M1-> 0.000e+000 M2-> 0.000e+000 M3-> 0.000e+000
Member m 6 Actions: Np-> 5.000e+004 Nm-> 5.000e+004 V2-> 0.000e+000 V3-> 7.500e+004 M1-> 0.000e+000 M2-> 2.083e+007 M3-> 0.000e+000
Member m 7 Actions: Np-> 5.000e+004 Nm-> 5.000e+004 V2-> 0.000e+000 V3-> 7.500e+004 M1-> 0.000e+000 M2-> 2.083e+007 M3-> 0.000e+000
```

Notice the user checks section of the listing:

```
--------------------------
Users's defined variables
--------------------------

F = m6.M2  /  (  m6.h - m6.tf ) / 4
       the force in one stiffener

lam1 = 3.1415 *  sqrt(  206000. / P6.fy)
       Lambda 1 for S235

P6.slend =  P6.H  /  (  P6.t  /  sqrt( 12 ) )
       Slenderness of the stiffener



------------------------
User checks description
------------------------

---1---Check of component P6

              F < ( P6.B -15)*P6.t * P6.fy *  chic(  P6.slend  /  lam1  )
              Force in the stiffener < Maximum axial force in the stiffener (assuming as constraint
              just the two sides simply supported)
              Buckling check of stiffener P6 (m6 bending)
```

and the results for the relevant combinations, always using m6 bending M_2:

```
---------------------------------------------------------
Throughs whose worst exploitation is due to user's checks
---------------------------------------------------------
```

Inst	Combi	Name	Check	Description	vL	vR	Expl
1	125	P6	1	Buckling check of stiffener P6 (m6 bending)	2.815e+004	1.448e+005	0.194
1	131	P6	1	Buckling check of stiffener P6 (m6 bending)	-2.815e+004	1.448e+005	0.194
1	133	P6	1	Buckling check of stiffener P6 (m6 bending)	1.408e+004	1.448e+005	0.097
1	134	P6	1	Buckling check of stiffener P6 (m6 bending)	-1.408e+004	1.448e+005	0.097
1	137	P6	1	Buckling check of stiffener P6 (m6 bending)	1.408e+004	1.448e+005	0.097
1	138	P6	1	Buckling check of stiffener P6 (m6 bending)	-1.408e+004	1.448e+005	0.097
1	141	P6	1	Buckling check of stiffener P6 (m6 bending)	1.408e+004	1.448e+005	0.097
1	142	P6	1	Buckling check of stiffener P6 (m6 bending)	1.408e+004	1.448e+005	0.097
1	143	P6	1	Buckling check of stiffener P6 (m6 bending)	-1.408e+004	1.448e+005	0.097
1	144	P6	1	Buckling check of stiffener P6 (m6 bending)	-1.408e+004	1.448e+005	0.097

Here as explained in the legenda at the beginning of the file

```
| vL ......................... Left hand value of user's check                 |
| vR ......................... Right hand value of user's check                |
```

The listing file is named:

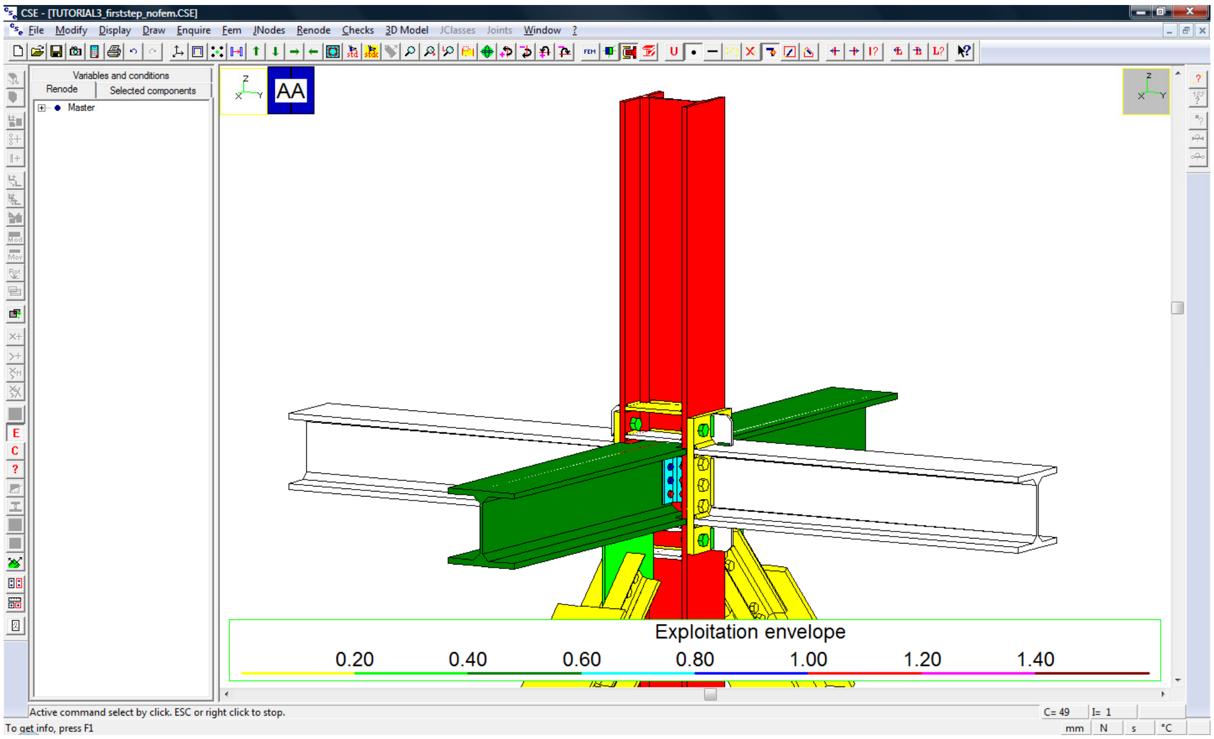
TUTORIAL3.CSE.AA.EURO3.OUT

which is

MODEL_NAME.CSE.RENODE_IDENTIFIER.NORM.OUT

2.7.2 Envelope results

Now let's have a look at the resistance checks. Execute the command **Checks-Envelope** (E button in the left bar), choose a proper view and you see (keeping into account symmetry) all components results.



As there are red components, checks are NOT satisfied. In particular:

1. The column is red;
2. The bolts joining X beams to the column are red;
3. There are elements white: they have not been checked (we will now explain why)
4. Bolts and welds (excluding those connecting X beams to column) are checked.

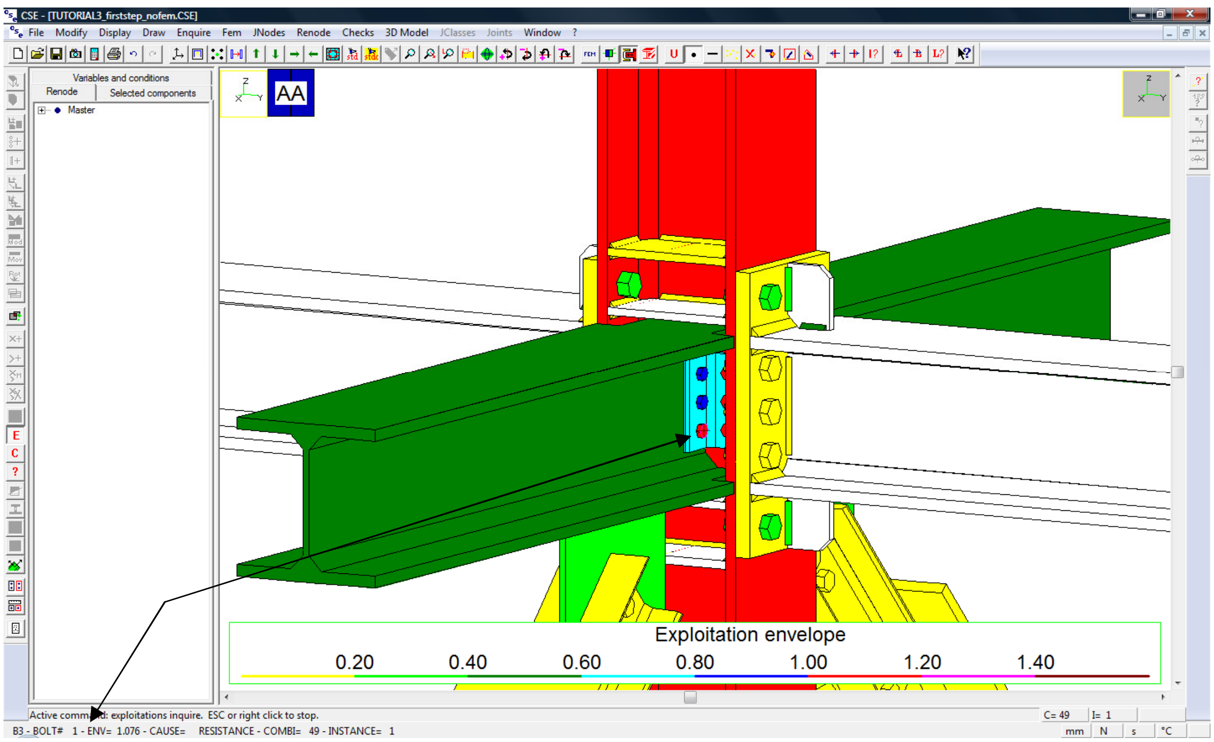
As mentioned there are white components:

1. The Y beams, as they are not cut, there are no holes over them no section reduction. Member checking of unmodified members is not done in CSE, it should have been done previously. However if we are interested we can create fem model for them later.
2. All the column stiffeners but P6 (which is yellow). The stiffeners are inside the column. So they are not loaded in this first run of analysis. To get their stress state we should look at the column fem model, where they will appear and be loaded. However, as we have added a user check linked to P6, P6 has actually been checked for user's checks. As it is yellow, checks are ok.
3. The stiffeners joining the Y beams to the plates. They are loaded (as they are not internal to one only component, they join beams to plates) however simple plates have only bolt

pressure bearing checks and block tear checks, and here there is no bolt. If interested in the stress state of these stiffeners you can create a fem model for them. On the contrary the other plates in the scene are not white as they have been checked for the bolt bearing pressure, and for block tearing, as they are bolted.

There are part of the checks that we will better do using fem models. However checking without fem models is faster and so it is the first step always suggested.

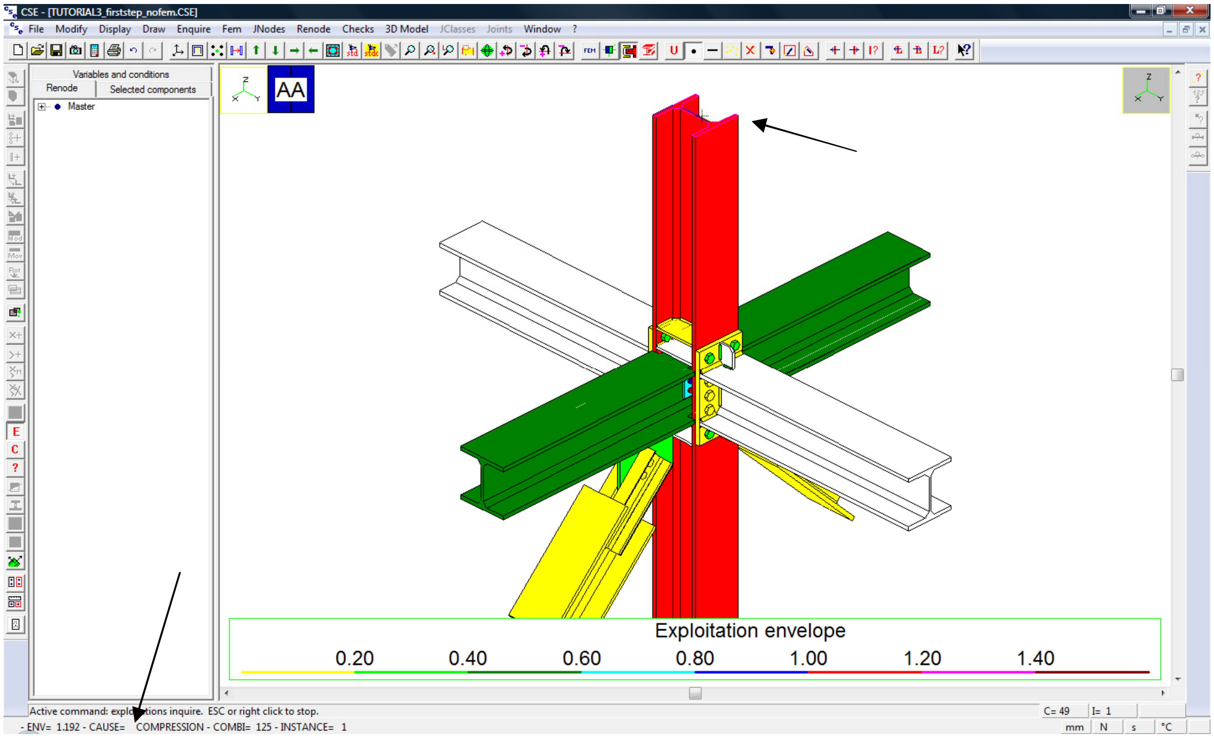
Let's have a look at the exploitations. Execute the command **Checks-Enquire** (the ? button in the left bar, it is active only if **E**nvelopes are active - E - of exploitation map of a single **C**ombination - C -). Notice that you can choose the faces of components and that in the status bar results are listed:



Here we are looking at results for a red bolt, one of its faces is highlighted.

Exploitation is 1.076 and combination is 49. 49 means positive axial force in the m3 member (24+24+1), i.e. the diagonal. As the value is slightly over 1 we can presume that changing bolt diameter from 10mm to 12mm will solve the problem.

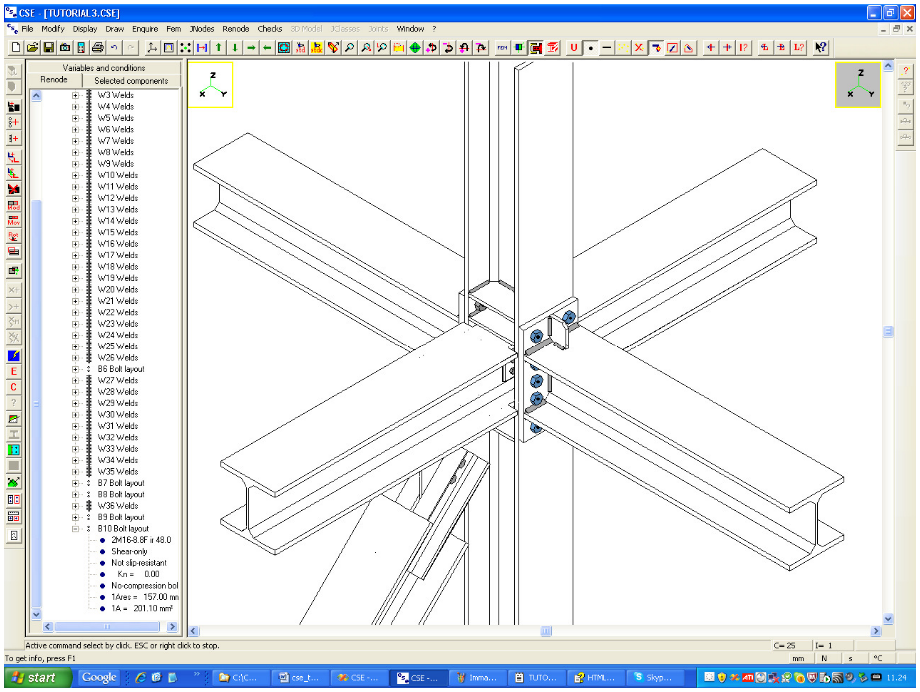
Now we look at the column:



The exploitation is 1.192 and the cause of the exploitation is "COMPRESSSION". This means that the column, considered as bearing object for the bearing surface bolt layouts, has received a pressure higher than 3N/mm^2. The combination is 125, that is a bending moment M_2 in member m6 (24x5+5=125). This is exactly the condition we were interested in. Let's have a better look at this bolts with bearing surface. We will select the +Y bolts. Exit from the Envelope command by un-pressing the button in the left bar.

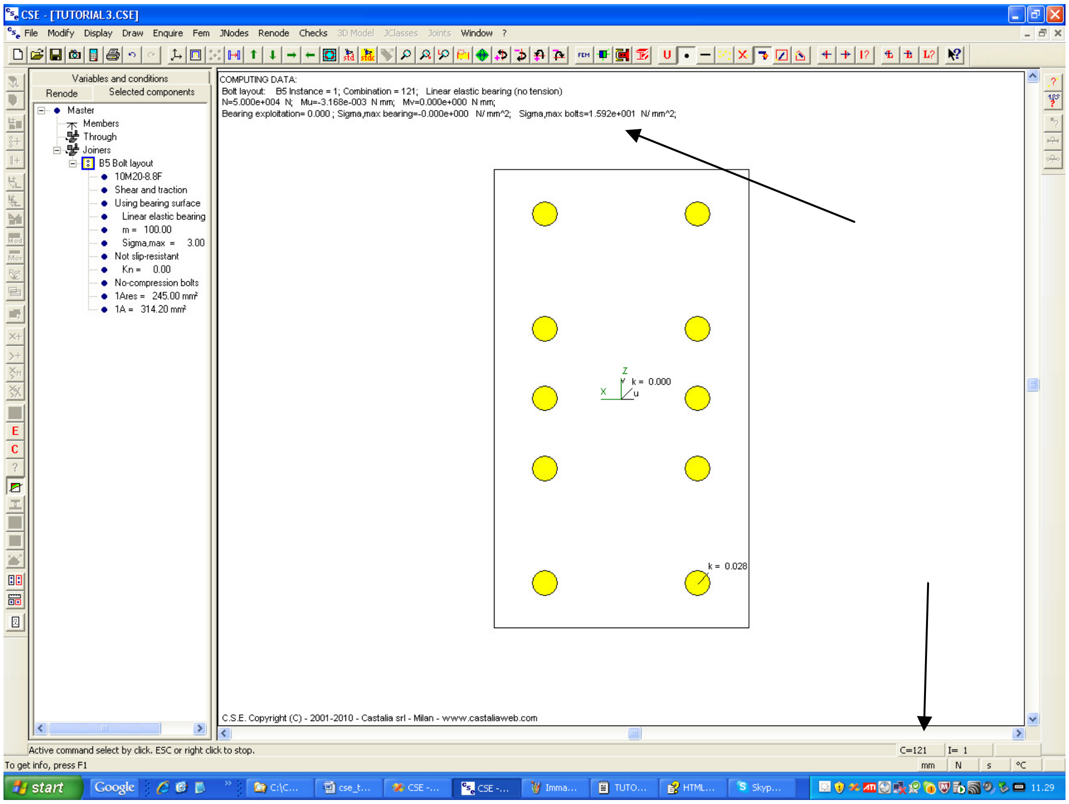
2.7.3 Bolt layout with bearing surface

Select the bolt layout involved in the bearing surface problem.



Choose the combination 121 (i.e. axial force in the Y beam m6) by using the **Checks-Combi?** command (L? button in the main bar). Then execute the command **Checks-Display bearing surface results** (the button in the left bar). This command is available as checks have been executed and as one only bolt layout is selected.

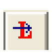
You see what happens when the y beam is pulled by an axial force N:

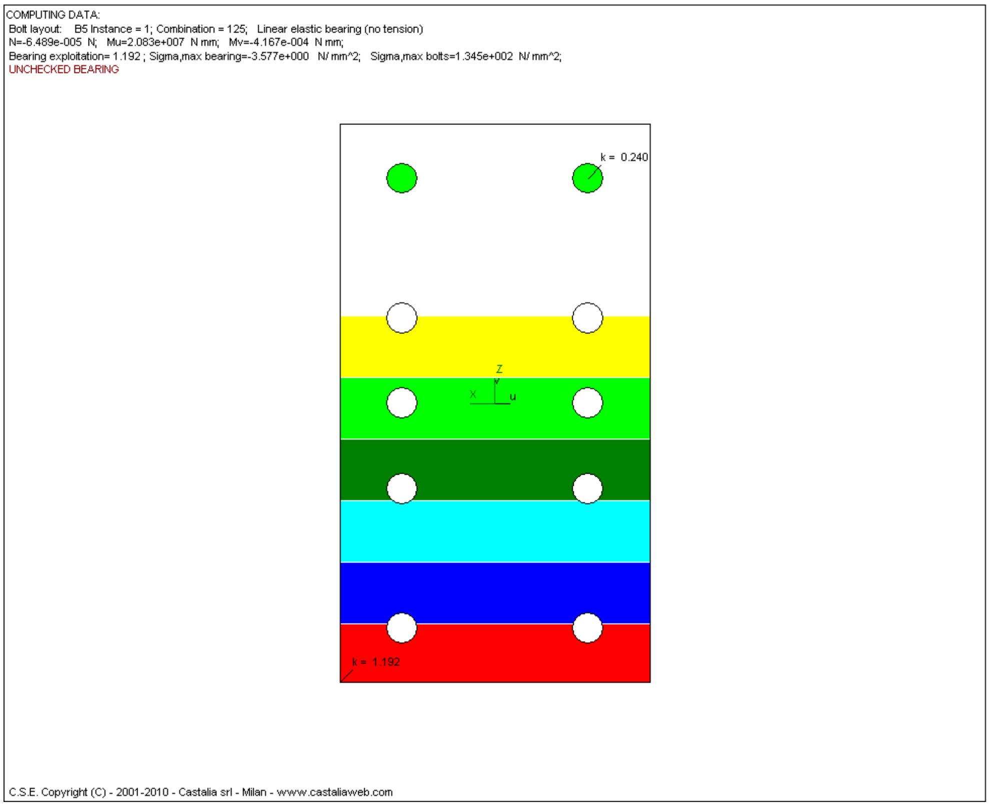


All bolts are pulled. Stress in bolts is 15.92N/mm^2. Now it turns out that:

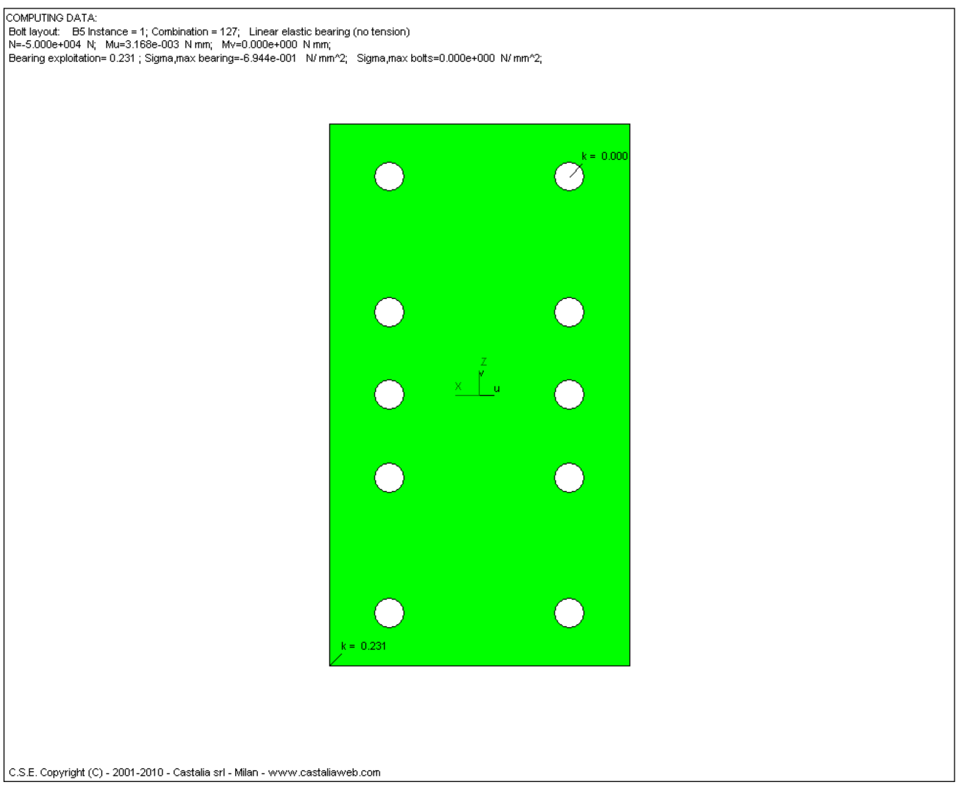
$$15.92 \times 10 \times 20^2 \times \pi/4 = 50000\text{N}$$

This tension force in the bolts will bend the column flange. FEM model of the column will show how, however we could have also added specific user checks to check this condition.

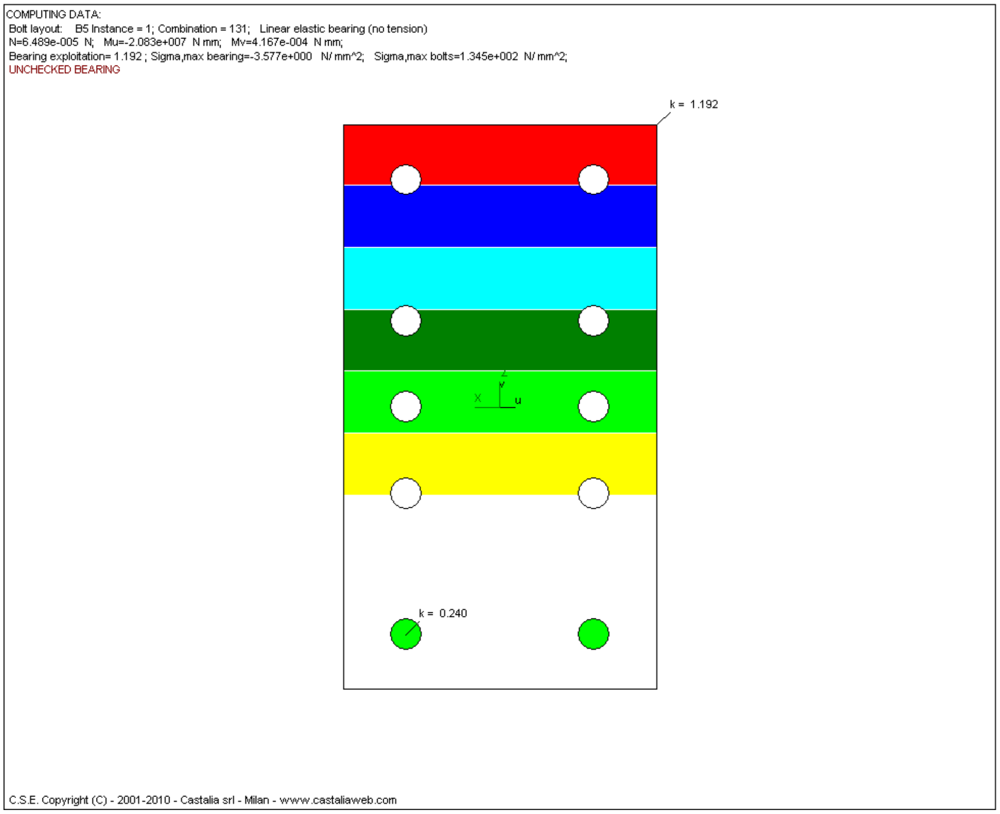
Now using the  button in the main bar switch to combinations 125, 127, 131 (positive M_2 bending in m6, compression in m6, negative M_2 bending in m6). You will see these stress maps:



M_2 positive



N negative (compression)



M_2 negative

The stress map is linear as we have asked for a linear no-tension constitutive law for the bearing (see part 2 of this tutorial).

The maximum compression in the bearing is got in the bending moment combinations and is -3.577N/mm^2, so since the maximum assumed for the bearing was 3N/mm^2, we have the exploitation

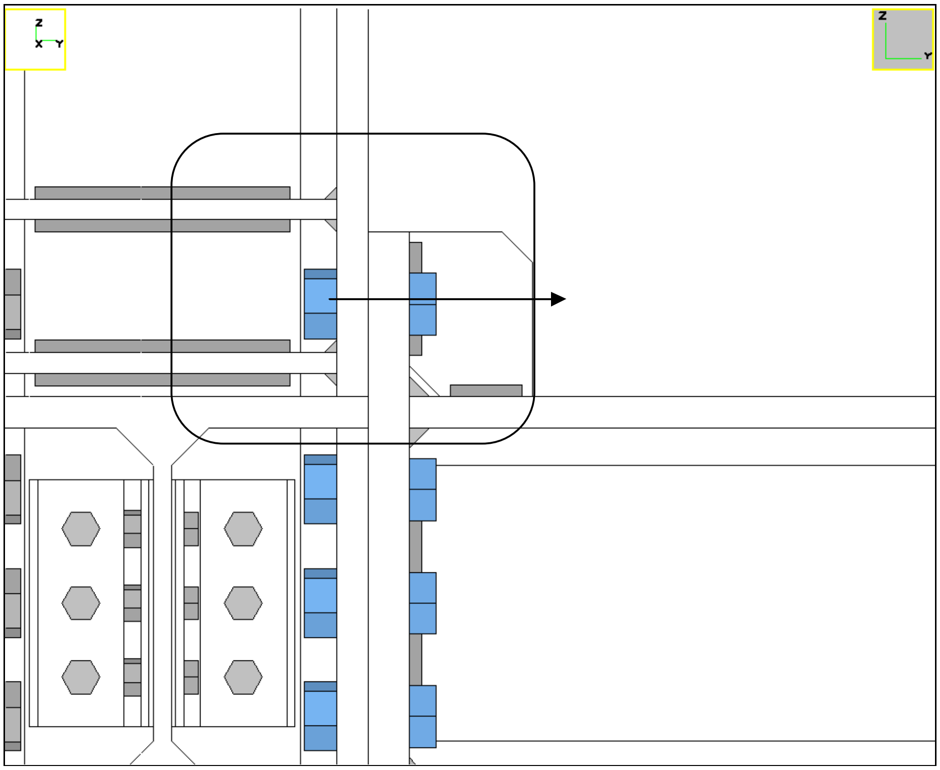
$$3.577 \ / \ 3 = 1.192$$

which is why the column is red. So the pressures computed do not match our request to be lower than 3N/mm^2. However as this request was due to a simplified static scheme, so it will be better to have a look at the fem model.

The bolts are pulled with a stress equal to 134.5N/mm^2, which is not high for the bolts but could be high for the bending of the column flange.

Un-press the button to get out of this command.

Now let's look at the connection from a +X view. Zoom over the plate-to-column connection.



As the bolt pulls with such a stress will the column flange be able to bear the load? The fem model will help, but we can now also understand it by a simple check.

Considering a clamped-clamped beam with a concentrated force at mid span (the force is $134.5\pi 20^2/4 \cong 42300$N), the span being the distance from the stiffeners 74.5mm, having a rectangular cross section H=15mm, B≅80mm, we get in the elastic range ($M=PL/8$, $W=BH^2/6$)):

$$\sigma = (42300*74.5/8) / (80 \times 15^2/6) = 131 \text{ N/mm}^2 < 235$$

2.8 STEP 11: ADDING CHECKS USING FEM ANALYSIS OF COMPONENTS

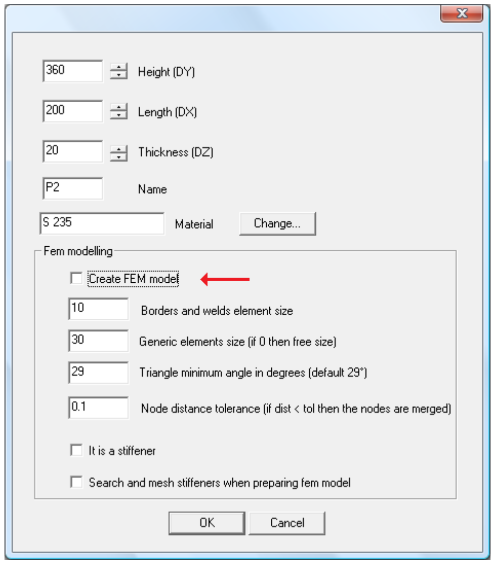
We have understood quite many things however it's now time to have a look at the fem models. This will help us to understand what happens to mainly two components we are interested in:

1. The column with its stiffeners
2. One of the two plates connecting the Y beam to the column

We are also interested in the stiffener joining the Y beam to the plate and in the plate connecting the diagonal. We will have a look at all.

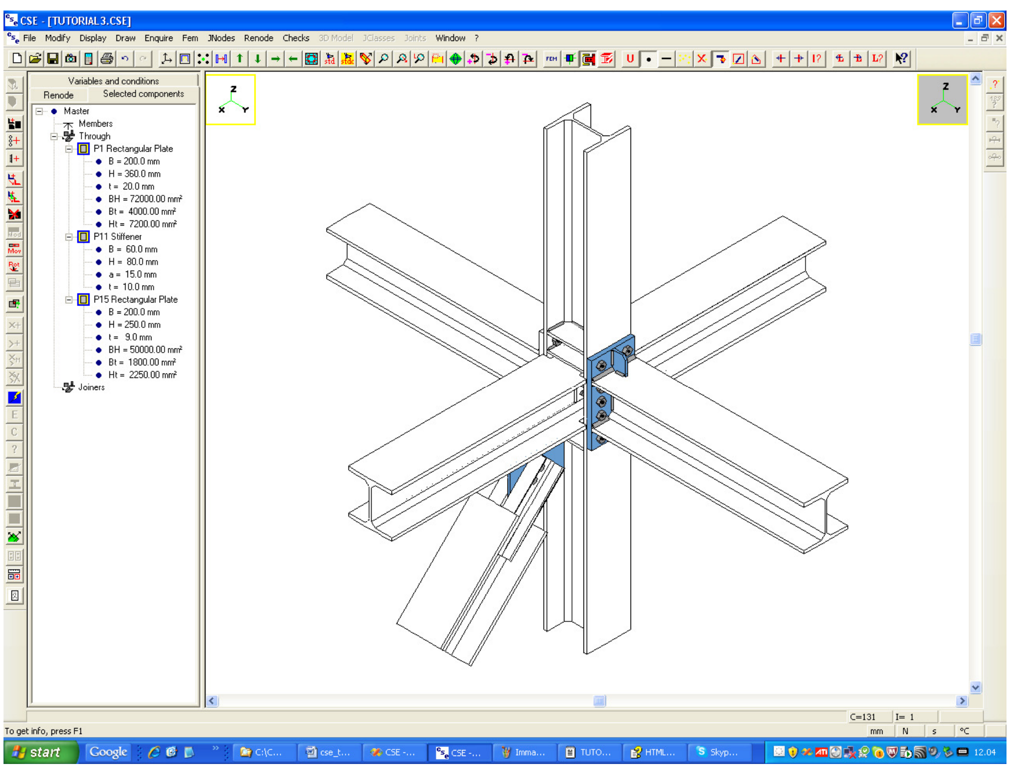
The first thing we are going to do is to switch on the **Create FEM model** flag to these components, un switching it from their twins.

We select one by one the components and switch/un-switch the flag according to the need. Here we un switch the flag:

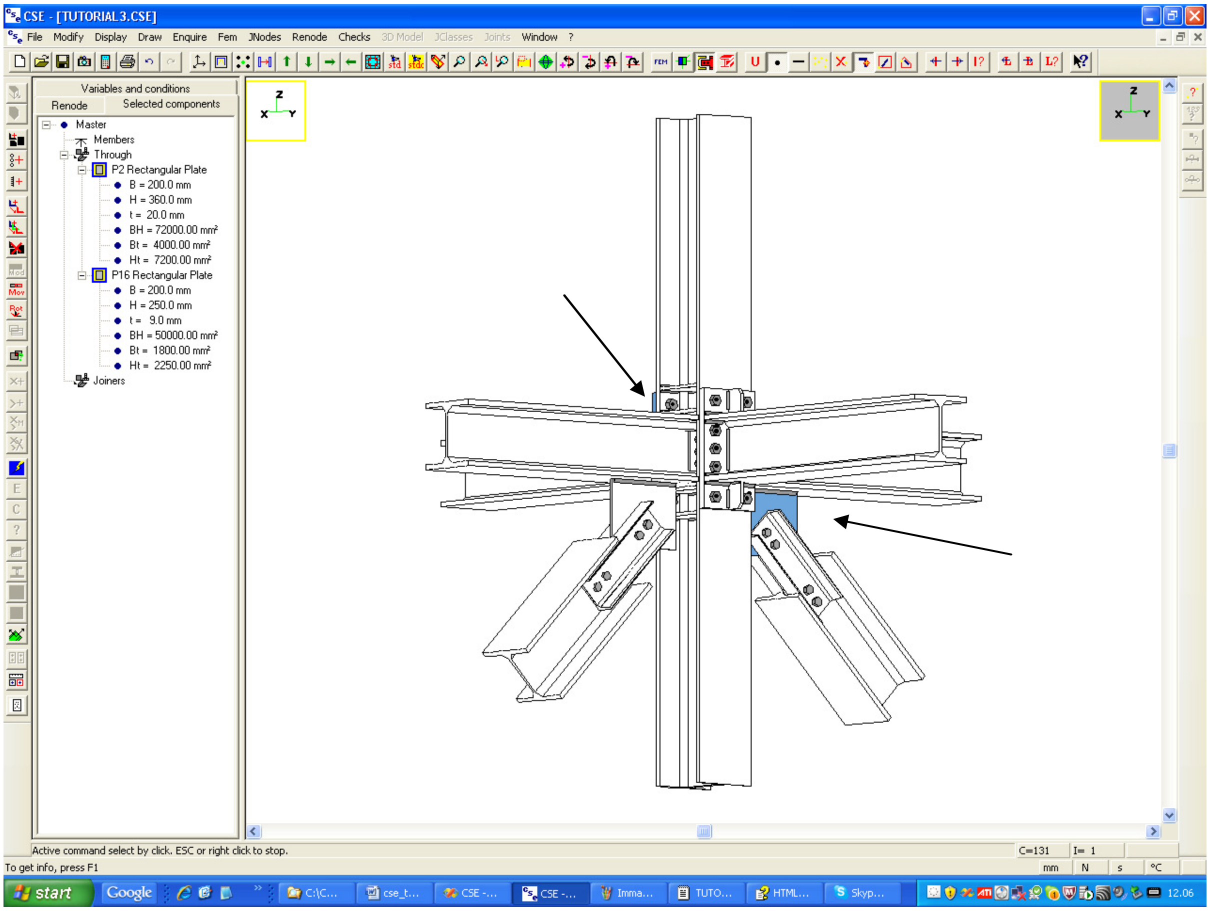


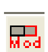
Keep the flag for components:

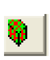
P1, P11, P15, the elements shown in the following picture:

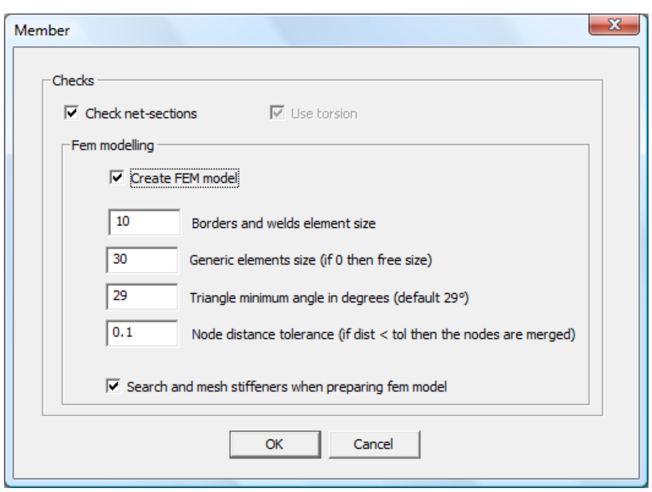


Remove the flag from other components: notice that the flag was only set and must be removed, from the following components (P2 and P16):



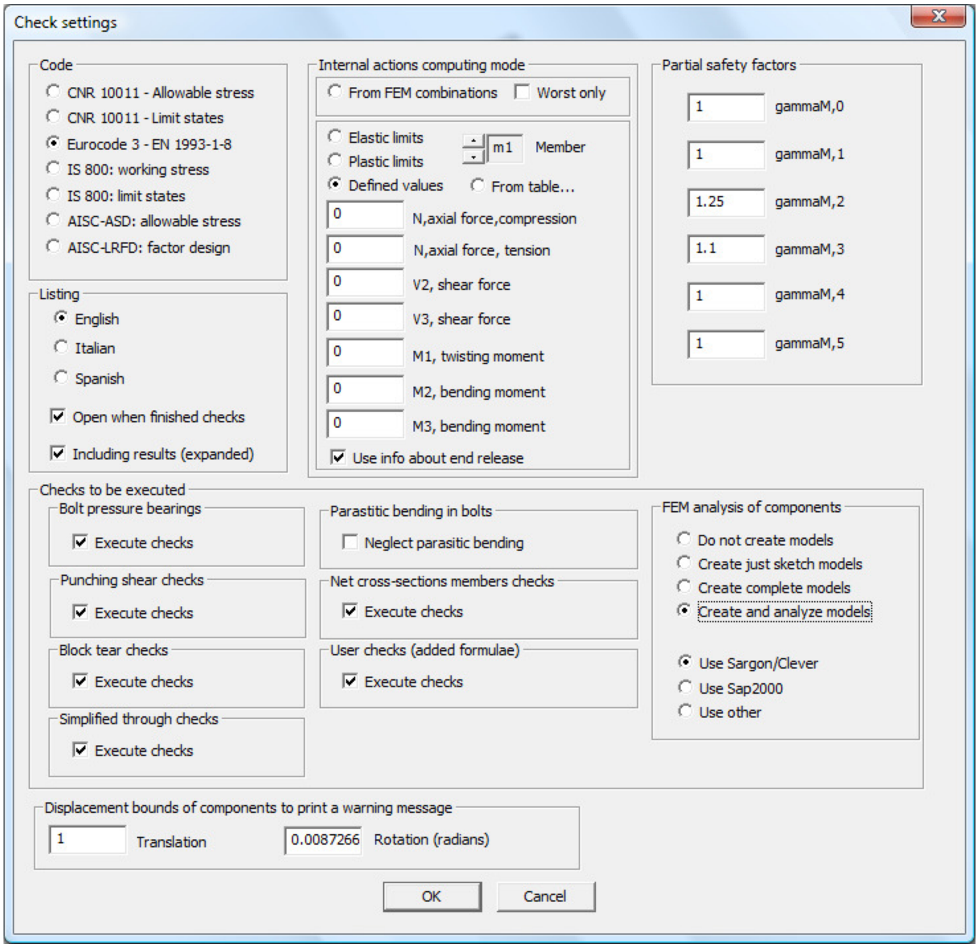
To modify the flag select the component one by one. Then execute the command **Renode-Components-Modify** (the  button in the left bar).

To modify the flag of the column select it and execute the command **Renode-Members-Modify** (the  button in the left bar), getting to this dialog:

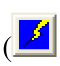


Leave the mesh size defaults. Also leave set the **Search and mesh stiffeners when preparing fem model** flag on.

Now re-execute the command Checks-Set and ask for fem model creation and analysis:



We will use the Clever solver, bundled with CSE.

Execute the command **Checks-Check renode!** ( button in the left bar).

The execution clearly takes more time. Four fem models are created and solved (P1, P11, P15, m1).

Total execution time (renode checks and fem models solving both for 168 load combinations) is 5min, 40 sec on a XP64 in an Intel i7 CPU, 2.67Ghz clock machine. The most part of this time is spent to solve the m1 fem model, which has some like 50,000 degrees of freedom.

P1	3354 degrees of freedom
P11	528 degrees of freedom
P15	1170 degrees of freedom
m1	64254 degrees of freedom (this model includes stiffeners)

If you look at the log window, the following content was added, referring to fem model creation:

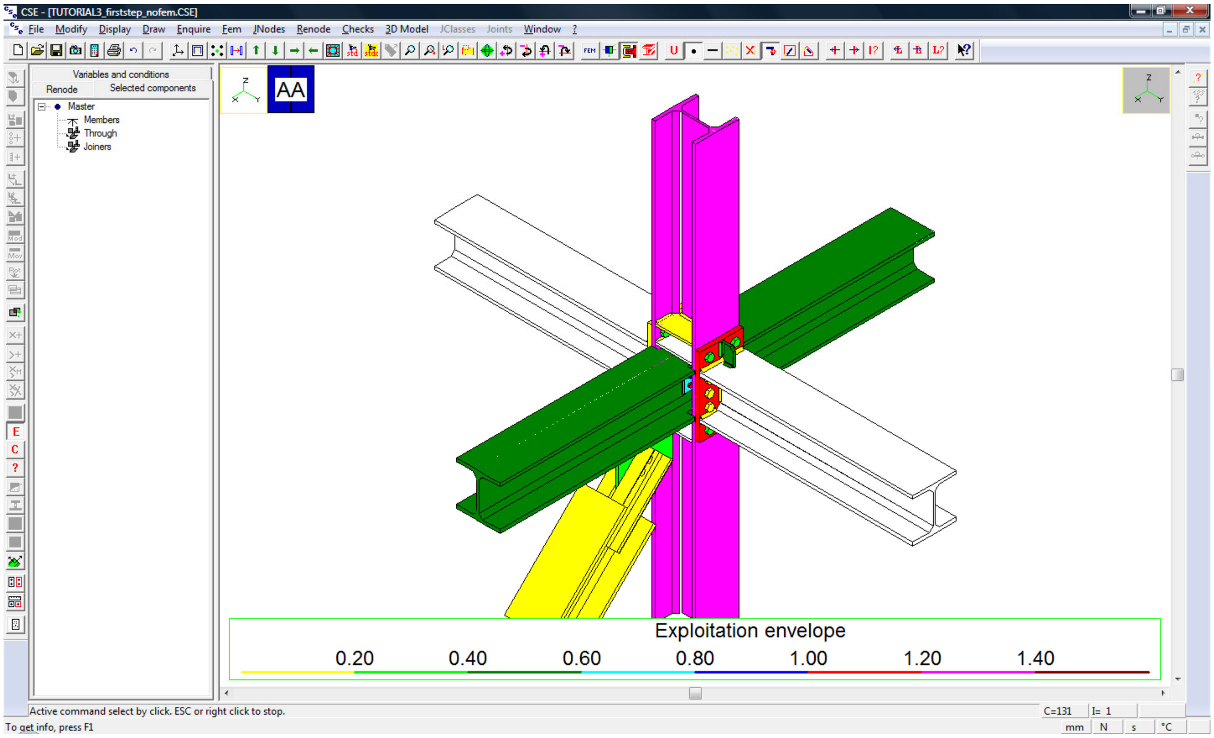
```
Creation of the fem model of component P1  sub-component 1 ... (bolt loads)
Creation of the fem model of component P1  sub-component 1 ... (weld loads)
Creation of the fem model of component P1 ... (bearing pressures)
Maximum resultant of applied forces (should be 0): 7.138222e-003   N. Combination 129 .
Analysis of fem model of component P1 ...
Creation of fem model of component P11 ...
Creation of fem model of component P11 ...
Creation of the fem model of component P11 ... (boundary definition)
Creation of the fem model of component P11  sub-component 1 ... (bolt nodes determination)
Creation of the fem model of component P11  sub-component 1 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.937036e+001 ; Average min angle = 4.205353e+001

Creation of the fem model of component P11  sub-component 1 ... (bolt loads)
Creation of the fem model of component P11  sub-component 1 ... (weld loads)
Maximum resultant of applied forces (should be 0): 4.240742e-005   N. Combination 125 .
Analysis of fem model of component P11 ...
Creation of fem model of component P15 ...
Creation of fem model of component P15 ...
Creation of the fem model of component P15 ... (boundary definition)
Creation of the fem model of component P15  sub-component 1 ... (bolt nodes determination)
Creation of the fem model of component P15  sub-component 1 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.906870e+001 ; Average min angle = 4.709734e+001

Creation of the fem model of component P15  sub-component 1 ... (bolt loads)
Creation of the fem model of component P15  sub-component 1 ... (weld loads)
Maximum resultant of applied forces (should be 0): 1.953599e-002   N. Combination 49 .
Analysis of fem model of component P15 ...
Creation of fem model of component  ...
Creation of fem model of component m1 ...
Creation of the fem model of component m1 ... (boundary definition)
Creation of the fem model of component m1  sub-component 1 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 1 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.942887e+001 ; Average min angle = 4.702239e+001

Creation of the fem model of component m1  sub-component 2 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 2 ... (weld nodes determination)
Beginning of Delaunay Triangulation
```

```
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.903861e+001 ; Average min angle = 4.636241e+001

Creation of the fem model of component m1  sub-component 3 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 3 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.930719e+001 ; Average min angle = 4.747986e+001

Creation of the fem model of component m1  sub-component 4 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 4 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.914074e+001 ; Average min angle = 4.632888e+001

Creation of the fem model of component m1  sub-component 5 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 5 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.901203e+001 ; Average min angle = 4.626644e+001

Creation of the fem model of component m1  sub-component 6 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 6 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 7 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 7 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 8 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 8 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001
```

```
Creation of the fem model of component m1  sub-component 9 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 9 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 10 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 10 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 11 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 11 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 12 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 12 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001

Creation of the fem model of component m1  sub-component 13 ... (bolt nodes determination)
Creation of the fem model of component m1  sub-component 13 ... (weld nodes determination)
Beginning of Delaunay Triangulation
End of Delaunay Triangulation
Beginning of Ruppert optimization
End of Ruppert optimization
Triangle quality measures: Min angle = 2.961768e+001 ; Average min angle = 4.530202e+001
```

As you see the fem models for component P1, P11, P15 and m1 were created, and analyses performed.

An immediate look at the exploitation shows that the map has changed:



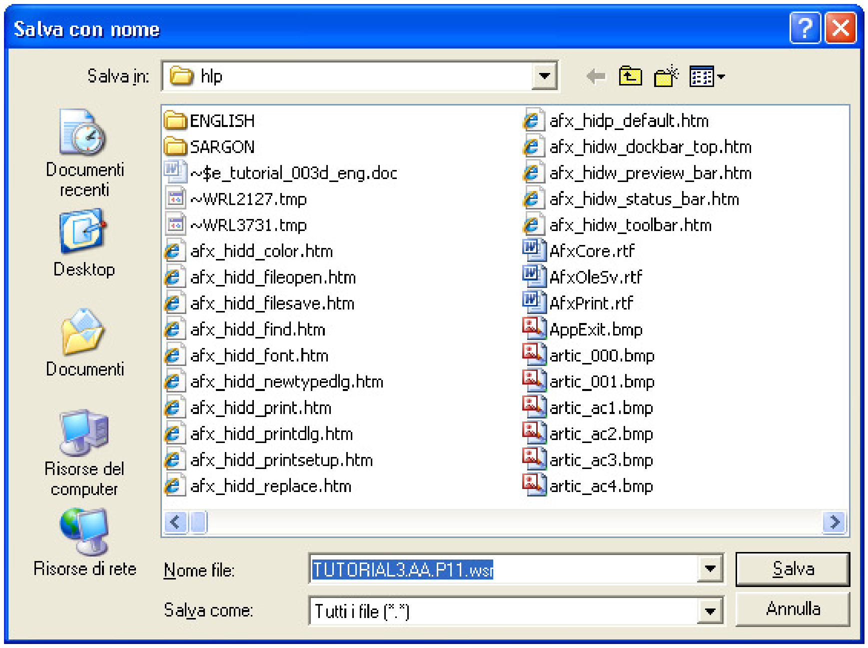
Now:

1. The plate joining +Y beam to the column is light red, not yellow anymore;
2. The stiffener joining +Y beam to plate is green (and not anymore white: it has been checked by fem analysis)
3. The column is magenta, which means fem checks were more severe than our simplified check on bearing pressure < $3 N/mm^2$.
4. The plate joining the diagonal is dark green and not anymore light green.
5. All other exploitations are the same, particularly those of the twins which were not analysed via fem analysis.

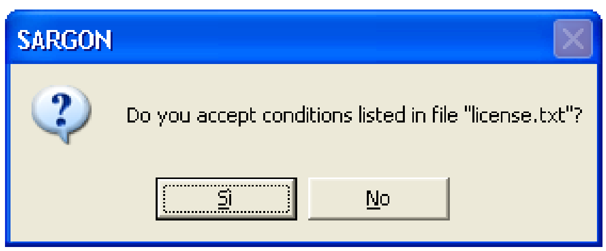
2.8.1 Looking at fem model results, stiffener joining plate and +Y beam

Select just this component (P11). Press the  button in the left bar (command **Checks-Display component fem results**): this command will run **Sargon-Reader** opening the already run fem model of component P11. Sargon-Reader (as the name suggests) is used to see a model, and to study results, you cannot modify a model: it is free and is in bundle with CSE. It can also be downloaded from Castalia web site, but as mentioned. is installed by default with CSE as standard tool to see fem model results. Notice that if the models were run using Sap2000, Sap2000 would open instead.

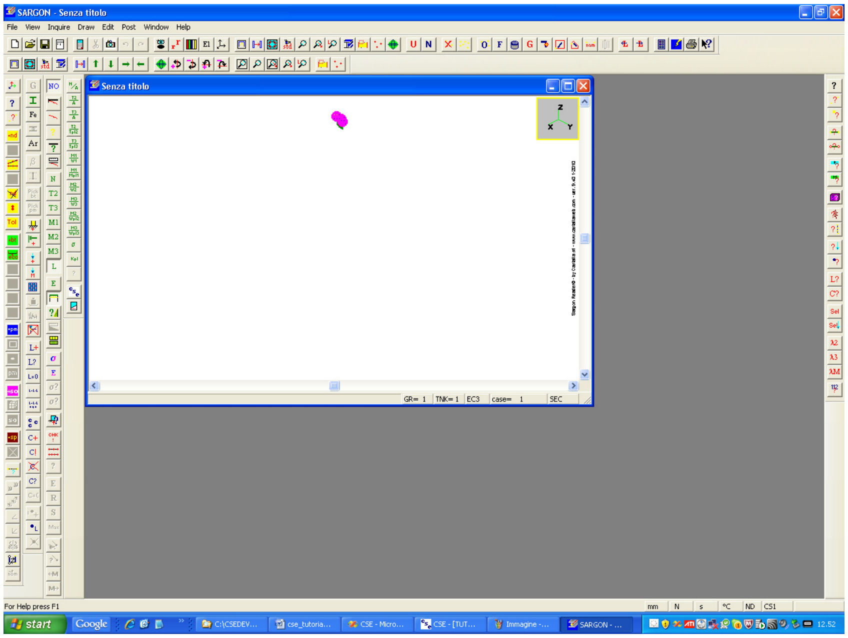
One the command is executed you get here:



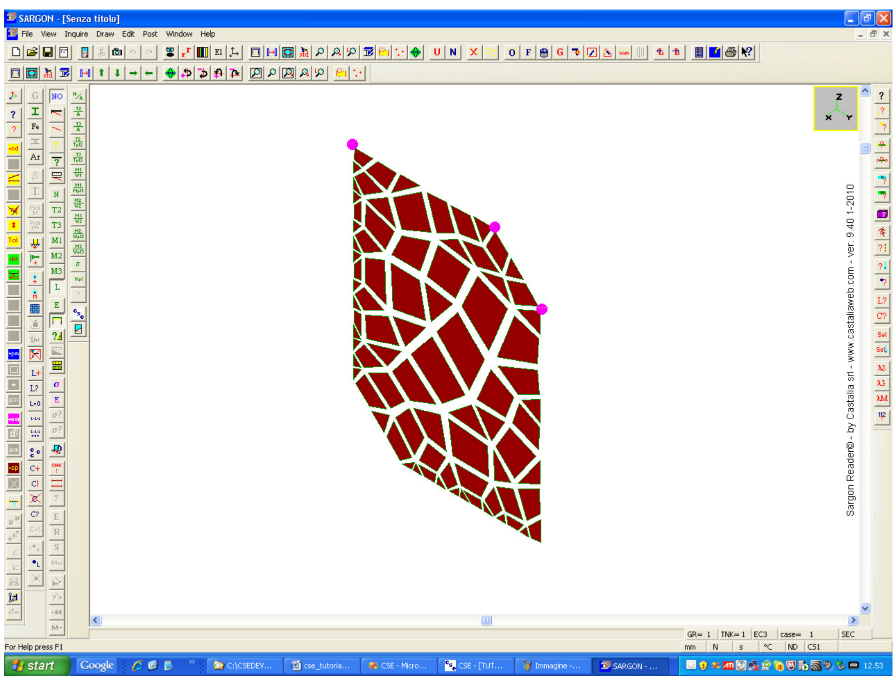
basically you are asked to save the model in the native Sargon format (.WSR). The model name is *Model_name.Renode_identifier.Component_identifier.WSR*. Save the model in the correct format. Then the following message appears:



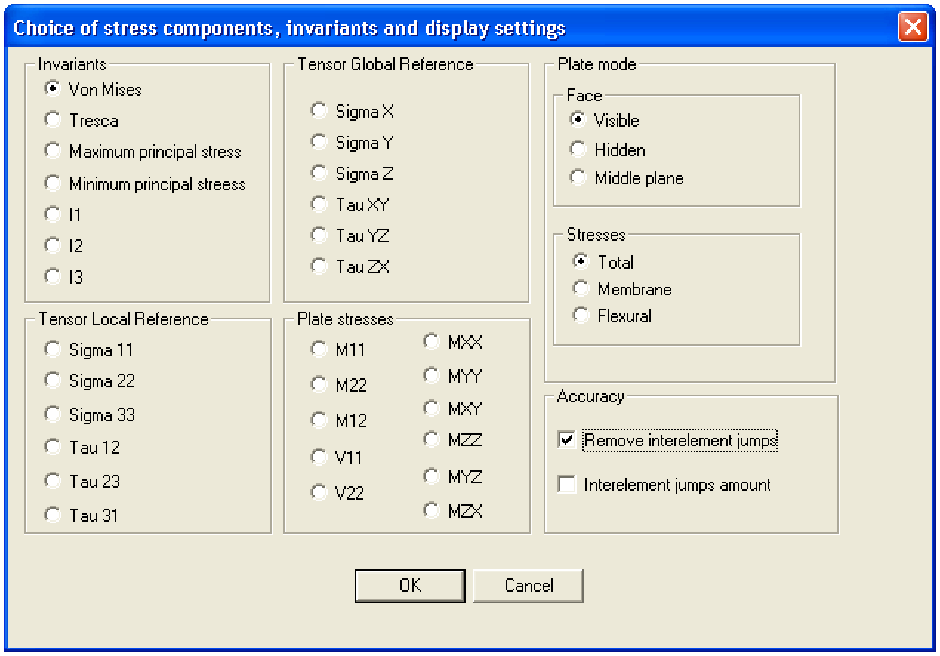
Answer yes (you can read the license file in the Sargon Reader folder). You now are here:

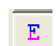


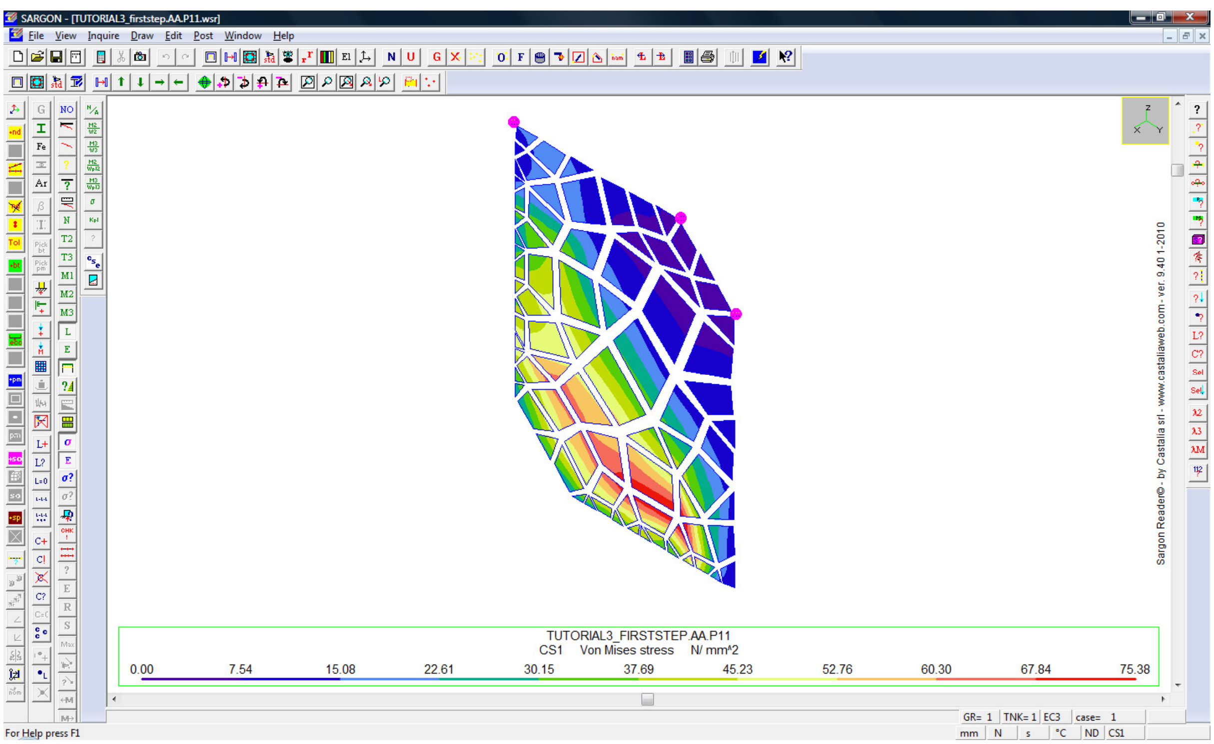
Maximize the window and using the mouse wheel zoom in the model. You see what has been created:



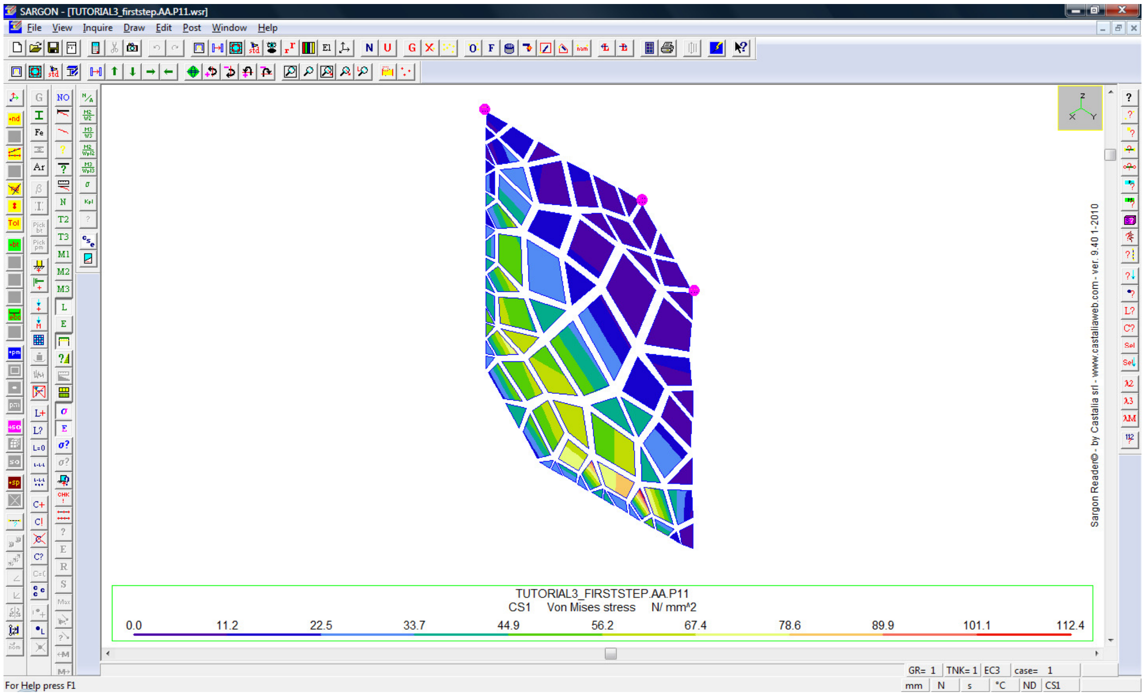
This is a coarse model as the stiffener is small and the mesh size we have left was default value. Let us see Von Mises envelope stress map : press first this button  in the third left bar , you get the following dialog:



Place a tick in the **Remove inter element jumps** check box. Press OK. You now see stress map for first load case which is blank (it is m1 axial force +N). Now press the  button in the same bar, this means Envelope of all load combinations. You get this stress map:



As the peak Von Mises stress is 75.38MPa, the component is clearly checked. Please note that the maximum Von Mises stress ***not removing inter element jumps*** is used by CSE to compute the exploitation in the component. Here if you plot the map **not removing inter element jumps** you get:

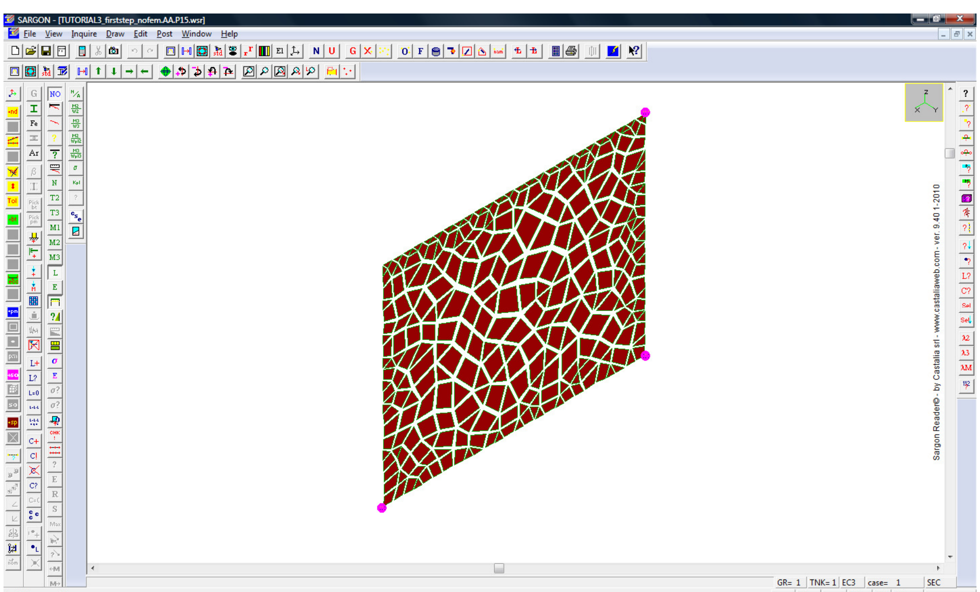


a peak of 112.4MPa. Now 112.4/235 = 0.478 which is in the 0.4-0.6 range (dark green).

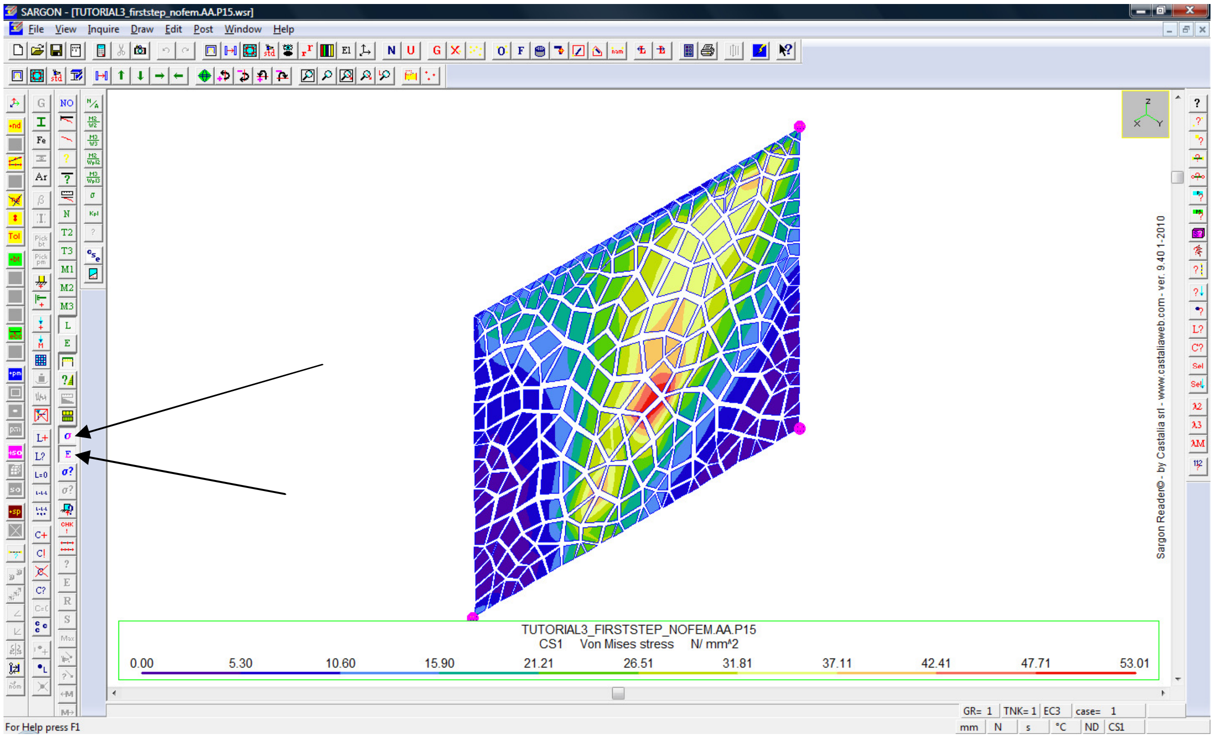
Exit from Sargon Reader (close its window) and activate (if necessary) CSE. You are back in CSE now.

2.8.2 Looking at fem model results, plate joining diagonal to +X beam

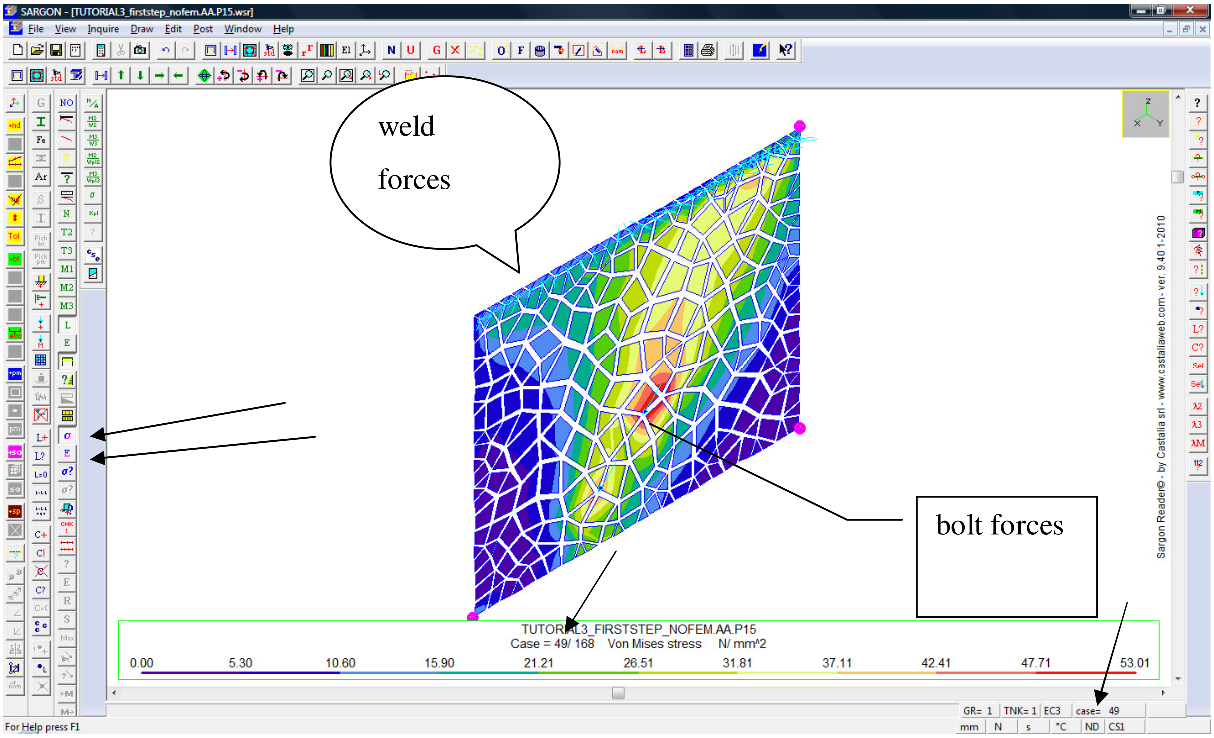
Now select only the plate P15. Press the  button in the left bar (command **Checks-Display component fem results**) you will have to save this model, as well. You are back into Sargon Reader. The model is now this one:



Notice that the constraints are dummy. The component must be self-balanced, i.e. loaded by self balanced loads. Constraint reactions should be negligible. Here the only combinations interesting are the +/- N force in the diagonal m3. This means combination number 49 and #54. Choose once more the Envelope of the Von Mises stress (remove interelement jumps). This is the map you get:

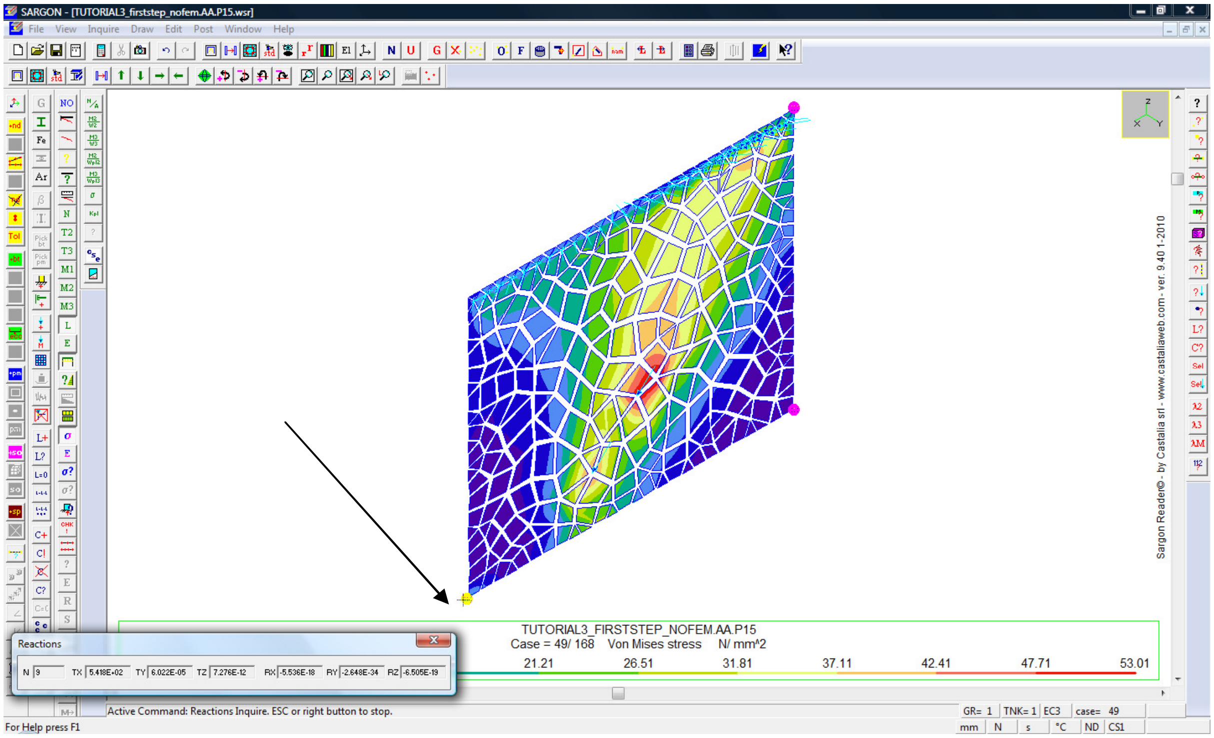


A diagonal stress band is clearly visible. The stress values are quite low (53MPa). CS1 in the stress legend stands for **C**ombination **S**et **1**. This is an envelope. Now let us learn how to see the stresses in a particular load case. Un press the Envelope button. Using the  button in the right bar switch to load case 49 (you see the load case number in the status bar), you are here:

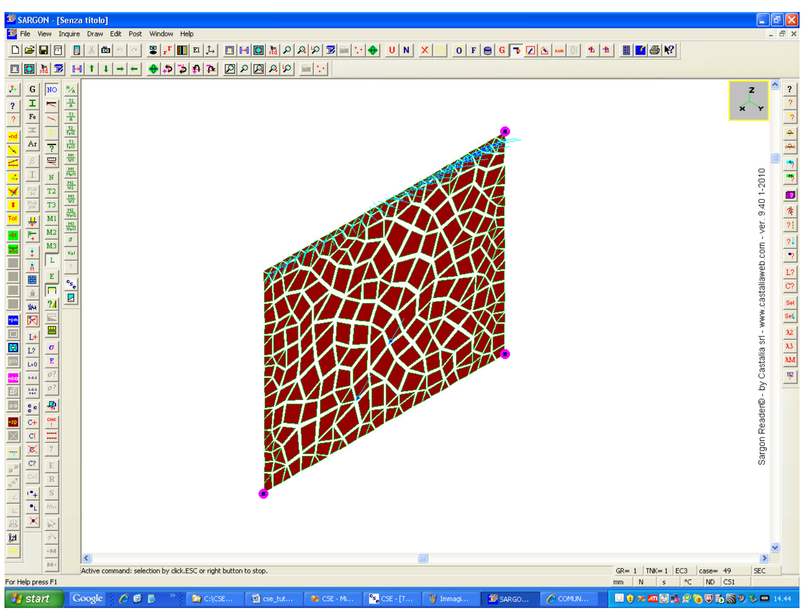


The forces transferred by the welds and by the bolts are clearly visible. We are interested in the constraint reactions as they must tend to be negligible. *Each non member component fem model must be loaded by self balanced loads* (the member models, instead, are clamped at non-renode extremities). As the model needs dummy constraints to run, 6 constraints are applied to the model in order to have rigid body motions prevented. If the models were really self-balanced, the constraint reactions of these constraints would have been strictly null. However, small un balanced loads are always present due to rounding errors and due to intrinsic problems related to discretization. So we would check the amount of constraint reactions and compare them to the applied loads. We could check envelope values of reactions or we can check the reactions of a single load case. Let's begin with the latter.

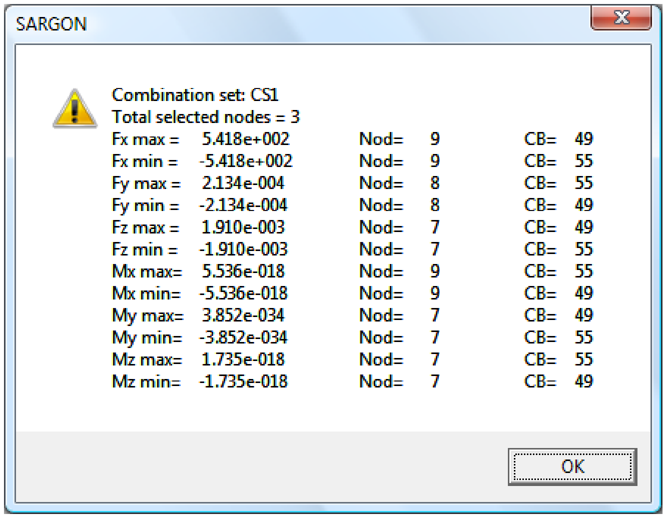
Press the  button in the left bar, and point the mouse near the constrained nodes (magenta circle over them is shown).



You get the reactions. Click right to exit from the command. To have the envelopes (i.e. maxima for all combinations) select all constrained nodes by clicking left over them, so that they get selected (blue) like this:



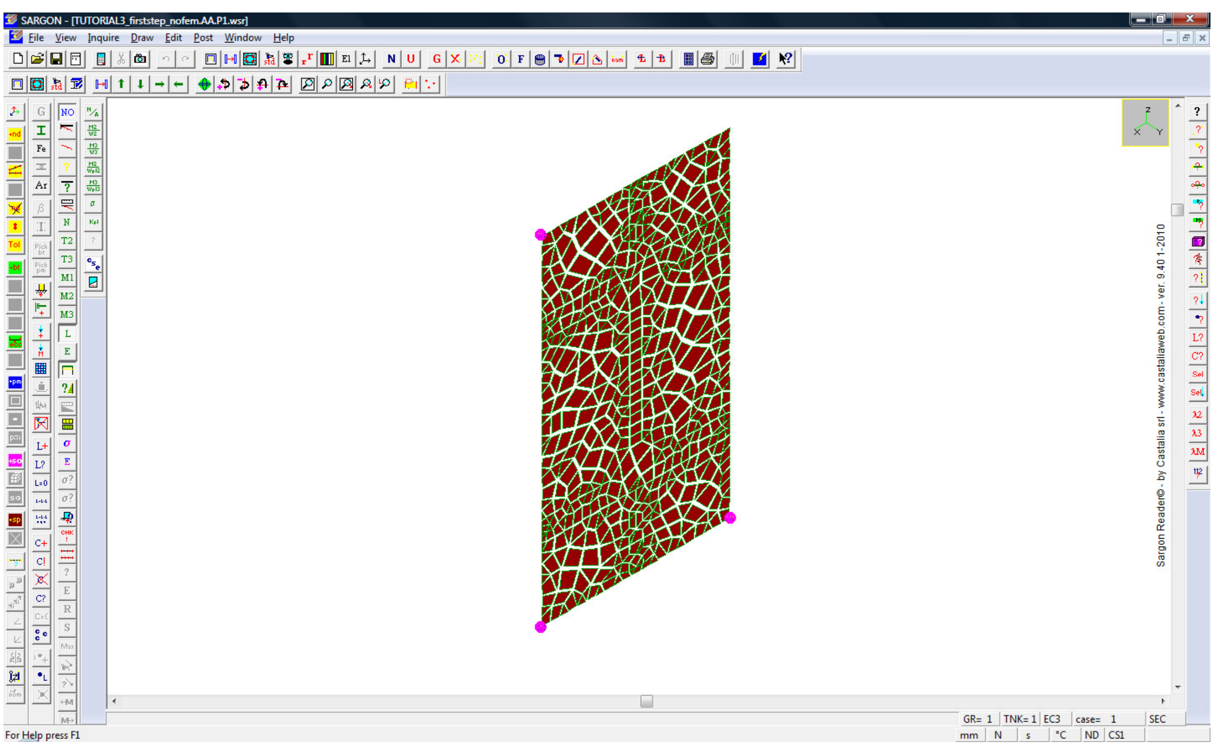
All selected nodes get with a small blue square. Now execute the command Post-**Reactions-Envelope of envelope** (envelope for all combinations and all selected nodes), you get the following dialog with the values:



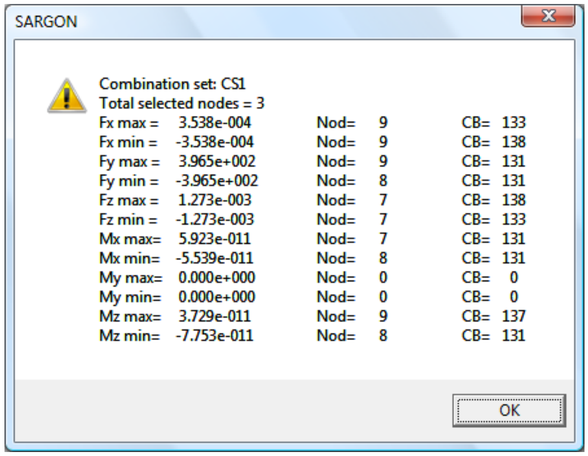
As you see the max reaction is 541,8N which must be compared with a 30000N force. The ratio is 1.8%. We can consider slight this reactions. They could be lowered decreasing the mesh size. Component P15 is checked. Now let's see component P1. Exit from Sargon Reader.

2.8.3 Looking at fem model results, plate joining +Y beam to column

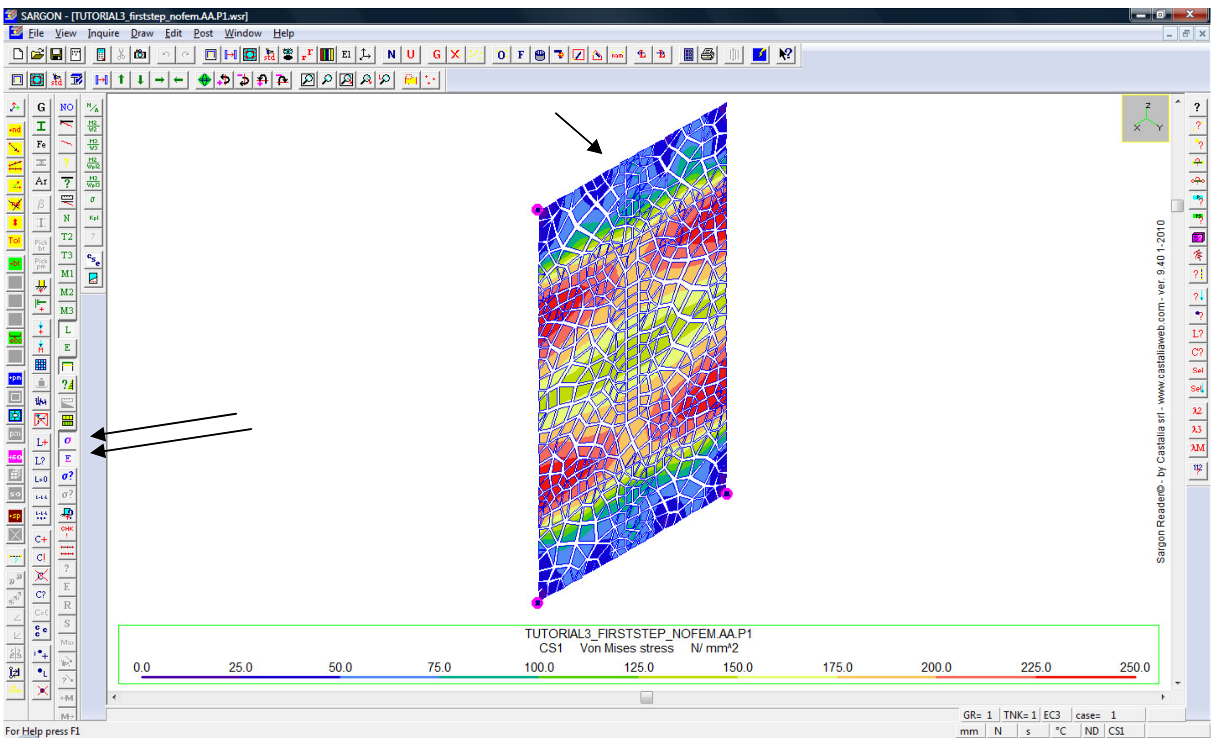
Select component P1. Press the button in the left bar (command **Checks-Display component fem results**) you will have to save this model, as well. You are back into Sargon Reader. The model is now this one:

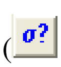


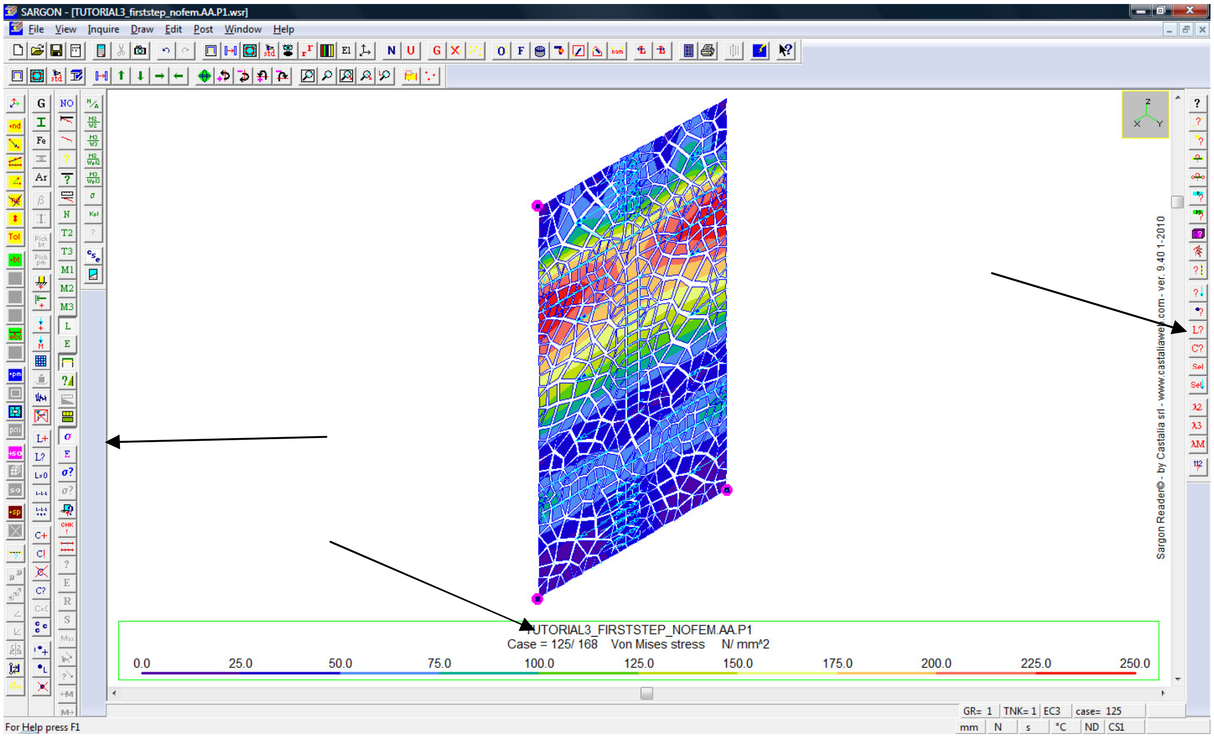
First of all select constrained nodes and check for constraint reaction envelope, as explained in the previous section (click left over the constrained nodes, execute **Post-Reactions-Envelope of envelope**). You get this data:



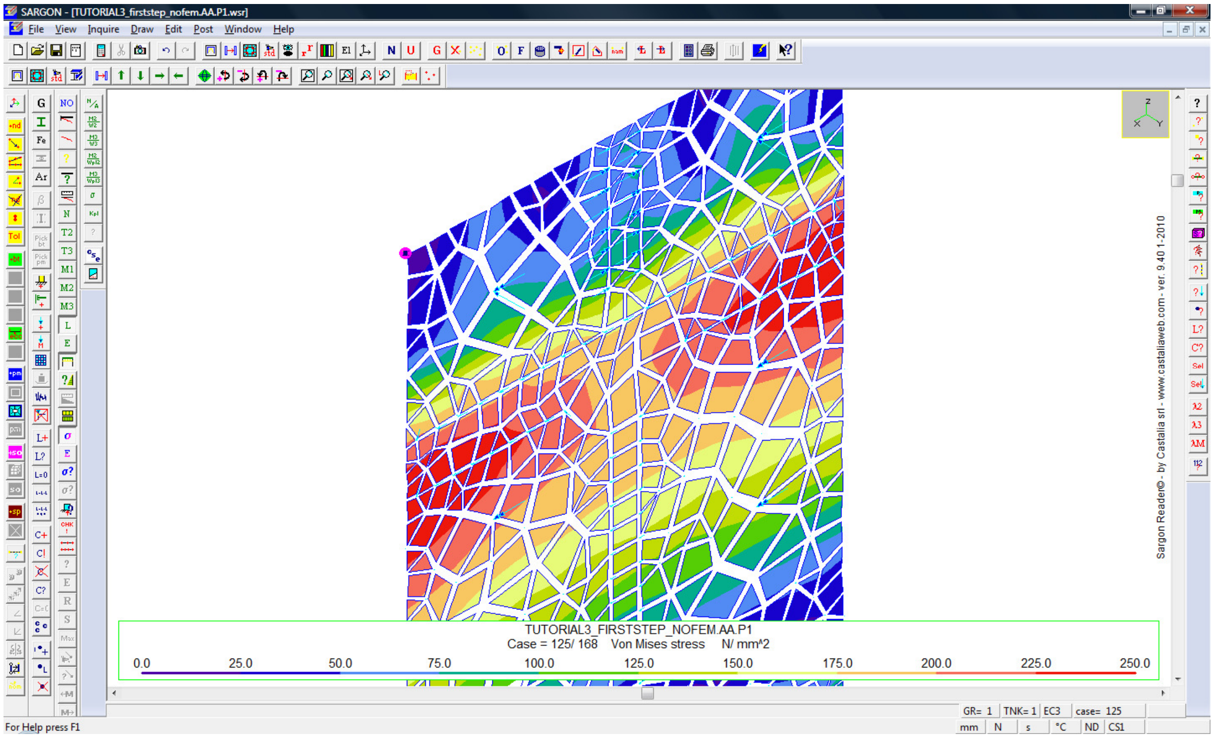
Forces are quite low (396N). We can consider affordable the analysis. Now look at the Von Mises stress envelope (inter element jumps removed). You get this image:



The maximum stress is 250 MPa, slightly higher than the yield stress. Notice the positive effect of the upper and lower mid-width stiffeners (topmost arrow for the upper). However there are regions where the yield stress is exceeded. Enquiring the values ( button in the left bar) we understand that the red areas are reached in two different combinations: 125 (i.e. member 6 positive M2) and 131 (i.e. member 6 negative M2). We know from images at pages 28-29 that the only pulled bolts are the topmost. They cause the high stresses. Adding 2 new stiffeners at plate sides (one per side) would surely improve the plate behaviour. Let's move to combination 125 (using the  button in the right bar) and see the NOT envelope stress map. It's this one, the red is just up:



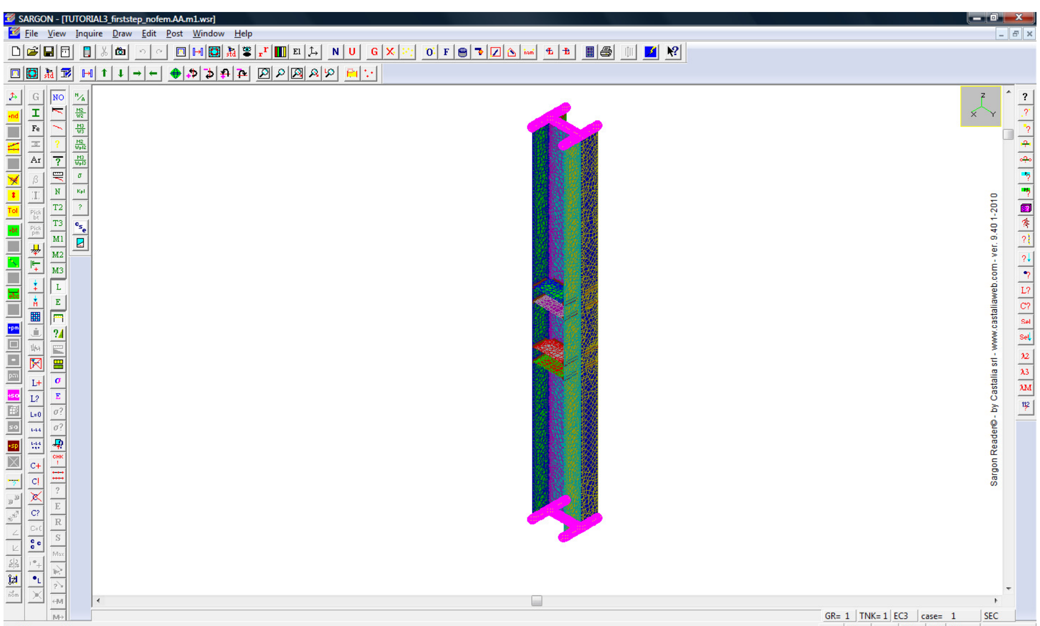
This is due to bolt traction. In fact the red area is over the weld line. In the next picture we see in detail that the 3 fillet welds joining the topmost flange of the +Y beam to this plate are modelled by 3 node-alignments, loaded with nodal forces which are equivalent to fillet weld reactions. The topmost stiffener, at mid width, dramatically decreases the stresses in the central area of the plate. This all is clearly noticed by the fem model automatically created by CSE.



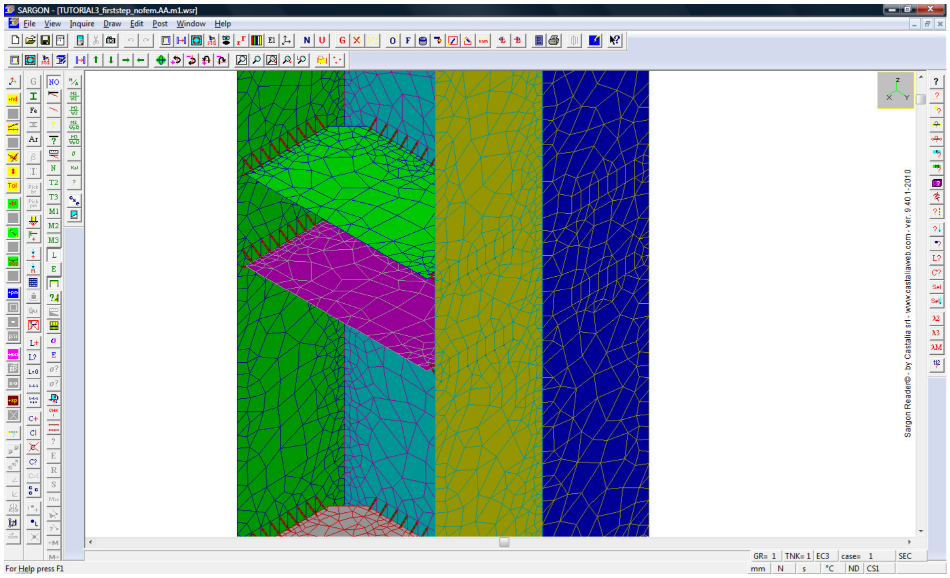
So we decide to add some kind of stiffeners to the plate.

2.8.4 Looking at fem model results, column

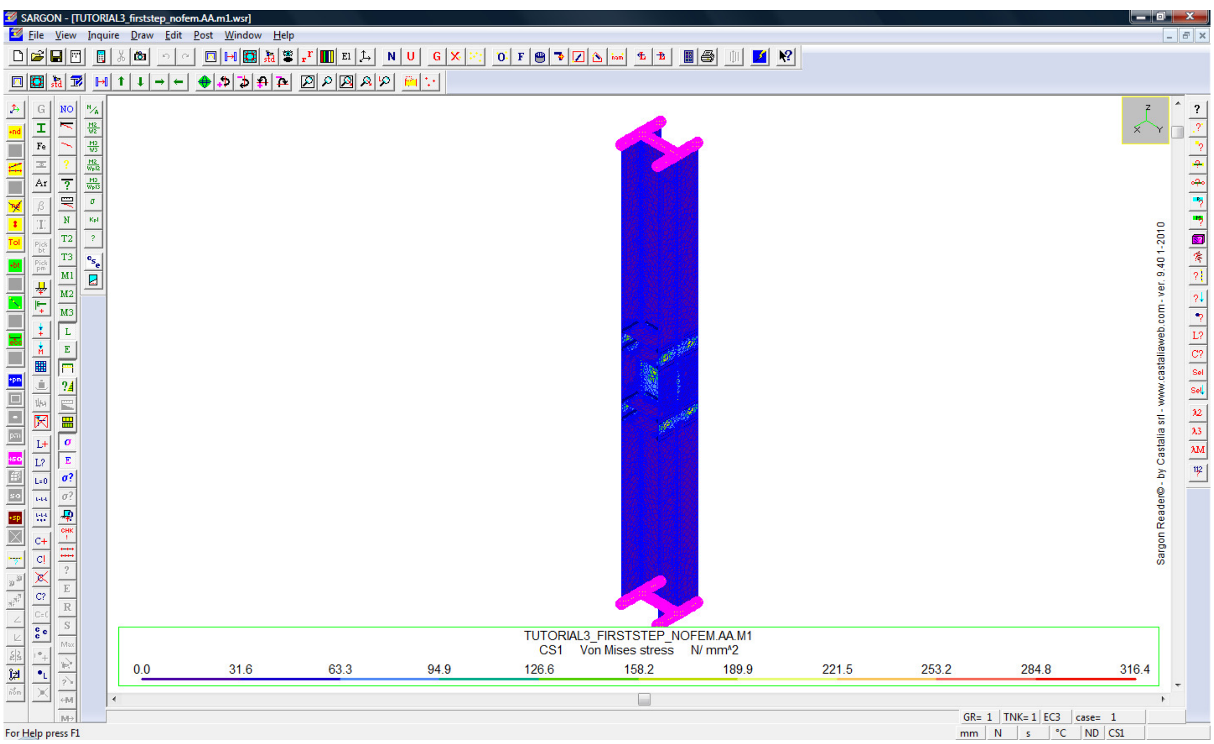
Exit from Sargon Reader, back in CSE, select the column alone. Press the  button in the left bar (command **Checks-Display component fem results**) you will have to save this model, as well. You are back into Sargon Reader. The model is now this one:



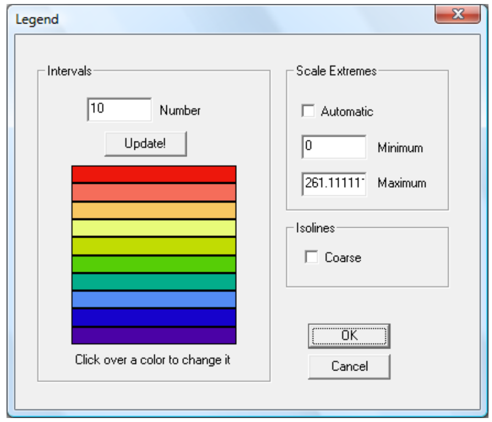
The member is clamped at not-renode extremities. All the stiffeners are inside the model, next is a detail (use the button in the main bar to set the shrink factor to 1):



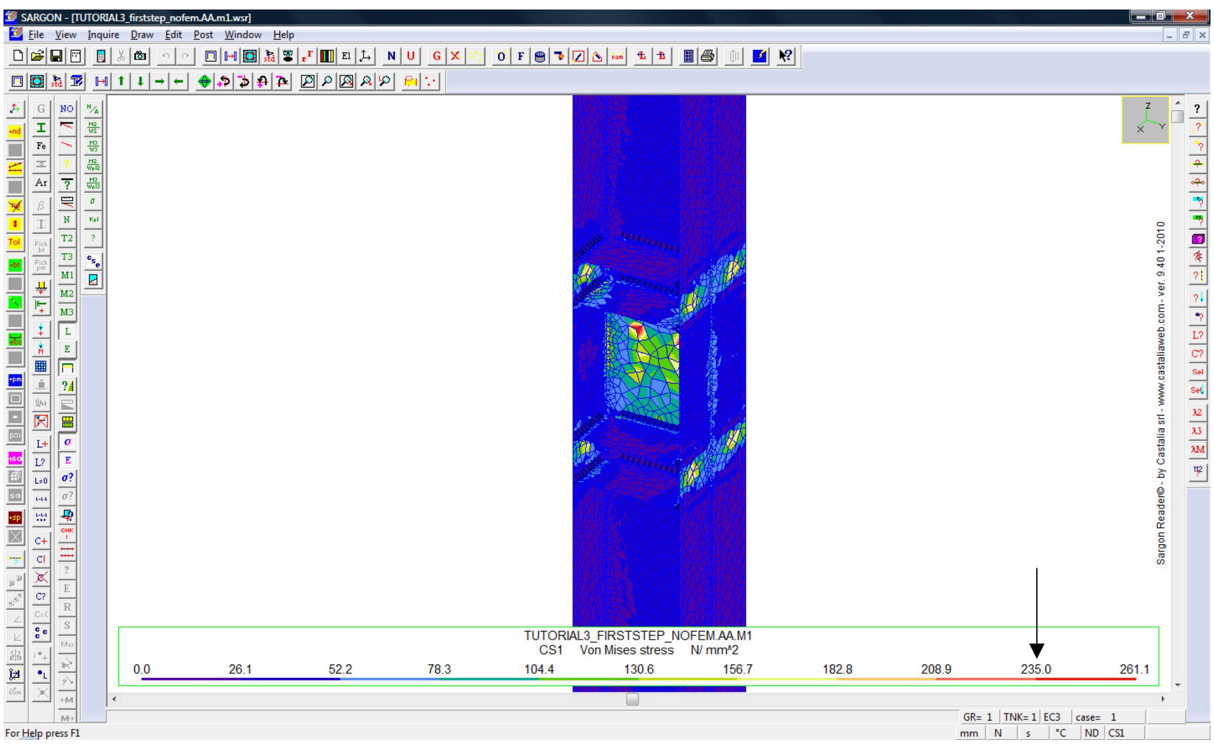
It is now not necessary to check for constraint reactions. Let's look at Von Mises envelopes (do not remove interelement jumps):



The top stress value is 316.4MPa, quite higher than yield stress. This is the value that gives magenta colour in renode view exploitation envelope. Note that if you set a scale base on yield stress, you can see that this values is exceeded in very limited zones. Use **Post-Legend**, remove the tick from “automatic” and type 261,11111 in “maximum” box, then press OK.



Now you can see in red the elements exceeding 235MPa: these few elements are in correspondence of bolts on the web.



Since FEM analysis accuracy is more reliable than bearing surface simplified check, we can assume the component as checked (even if few elements have σ_{VM}>235MPa) considering that we assumed parameters on the safe side (see tutorial 003 part B) and this kind of check is a simplified one.

2.9 STEP 12: CHANGES IN THE DESIGN

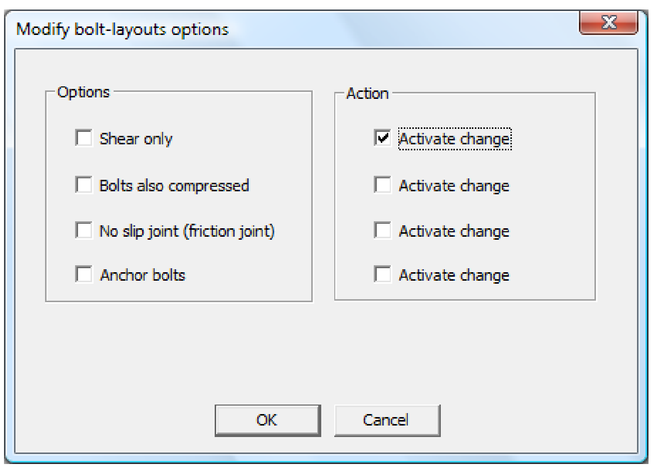
We have learned quite a lot from these checks. We are going to:

1. Remove the flag "shear only" from the bolts joining X beams to the column to avoid displacement problems;
2. Increase the bolt diameter to 12 to avoid resistance problems;
3. Add a set of stiffeners to the plate joining the Y beams to the column.

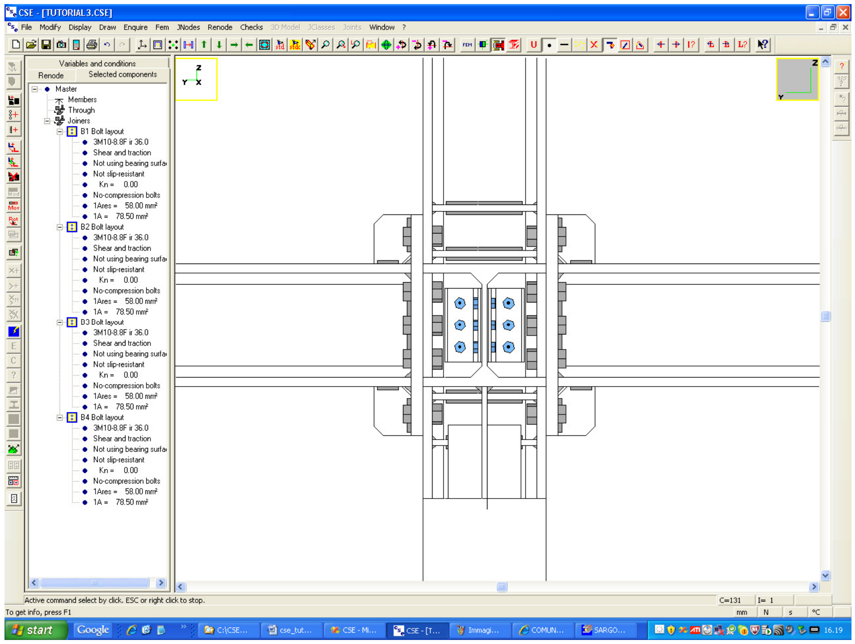
We can do that straightforwardly.

2.9.1 Remove the shear only flag

Select all the bolt layout connecting the X beams to the column (B1, B2, B3, B4); execute the command **Renode-Components-Modify bolt layout settings**. You get here:

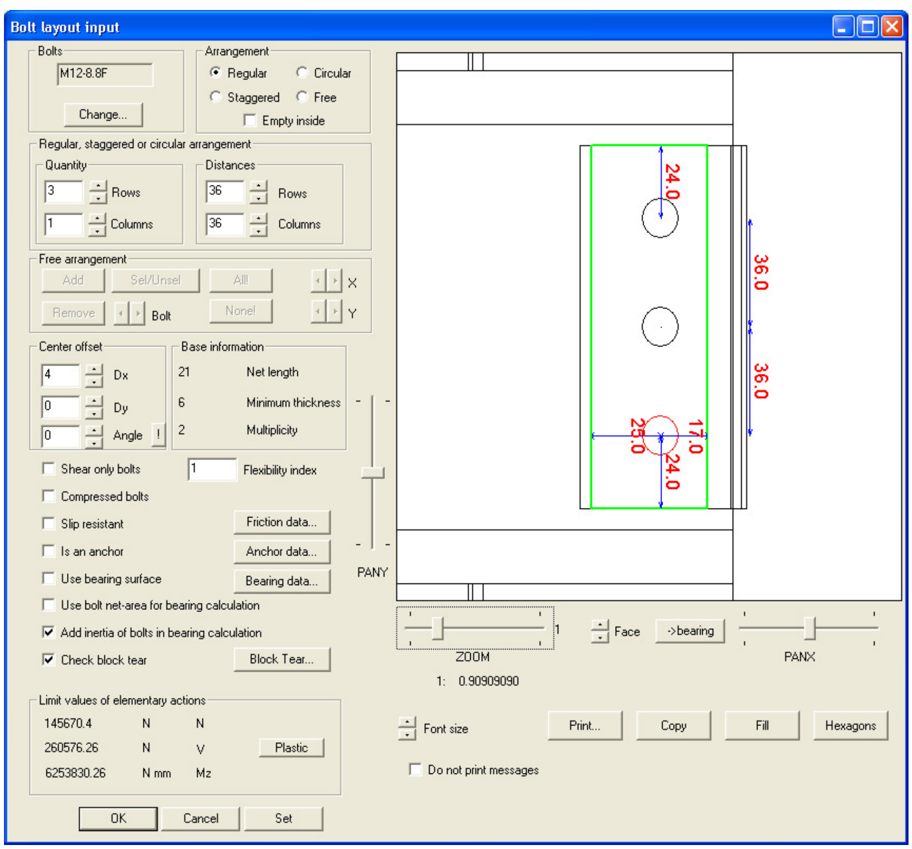


Remove the tick from shear only bolts, place a tick in the proper “activate change” check box. Press **OK**.



2.9.2 Increase the bolt size

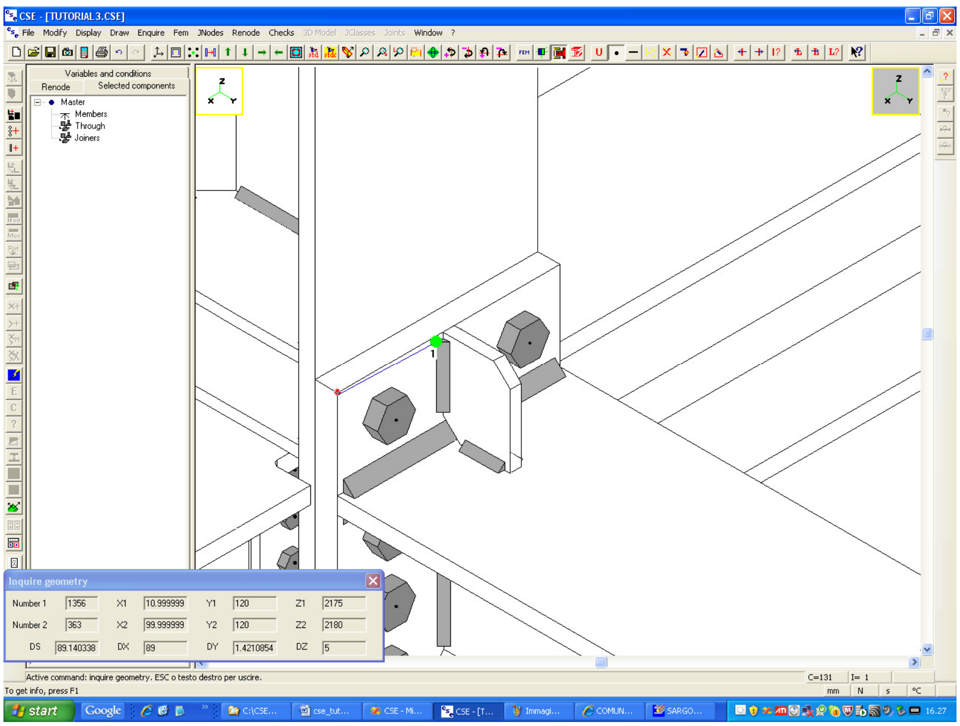
Select one by one the bolt layout joining the X beams to the column. Execute one by one the command **Renode-Components-Modify** (the Mod button in the left bar), get into the bolt layout addition/modification dialog. Change the bolt size to 12mm (M12).



When you will change the beam-web bolts you will receive a warning about distances. Decrease the 6mm DX offset to 4mm, as shown in the picture above.

2.9.3 Add a set of stiffeners to the plate

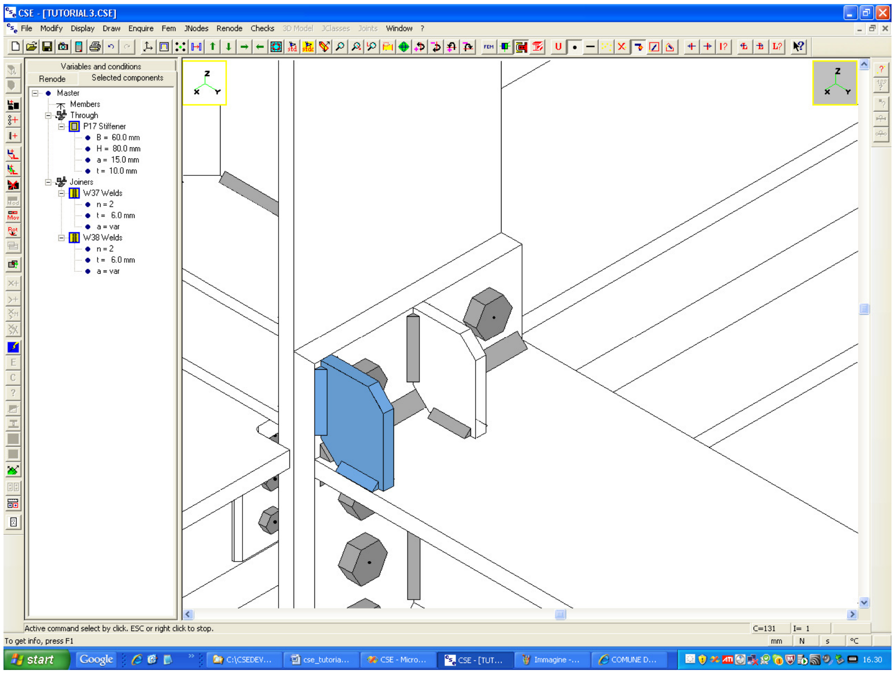
This can be done in several ways. One solution would be to have 3 stiffeners instead of one. Using the command **Enquire-Geometry** we can understand that there is room to copy the stiffener and its welds left and right, by 89mm as shown in the picture below (from the weld tip to the plate extremity).



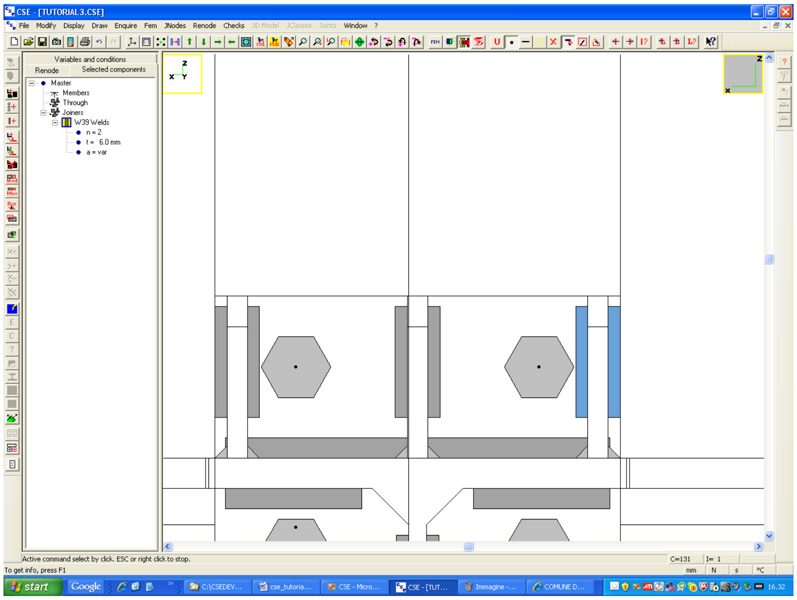
Select the stiffener alone before copying it.

Modify its **Create fem model** flag (command **Renode-Components-Modify,**  button in the left bar), to avoid fem model for this component (and its future copies).

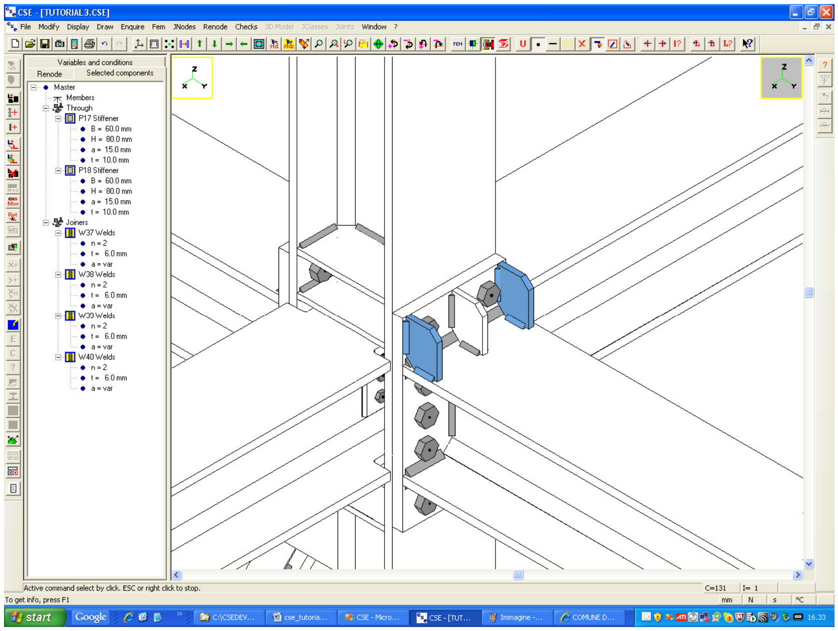
So now select the stiffener and its welds and copy DX=89 using the **Renode-Components-Copy** (yellow numerical input mode).



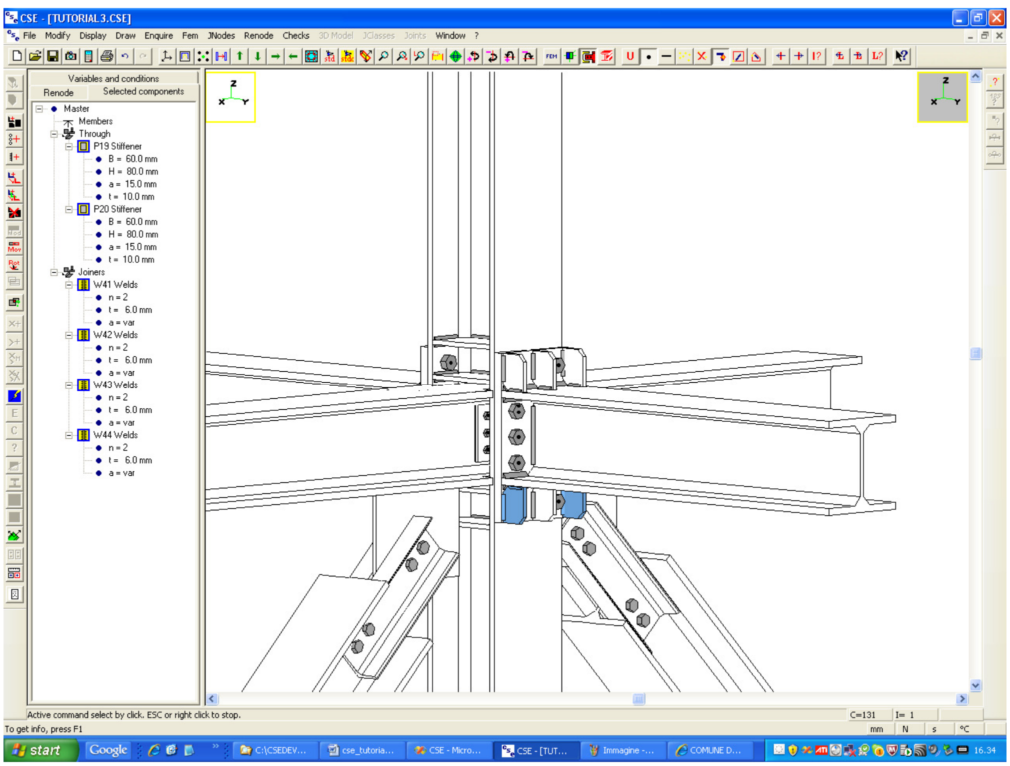
Now copy the selected once more in -DX, using DX=-178. A front view shows that there is no overlap:



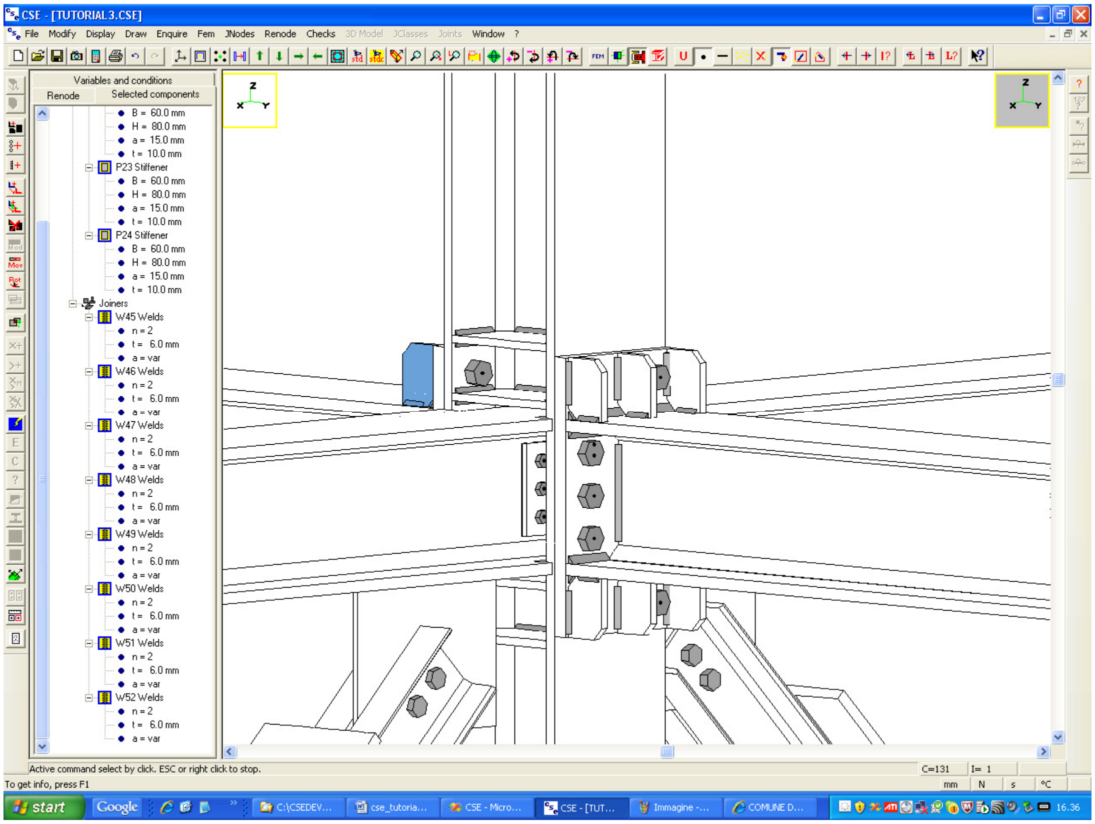
Now copy all the new stiffeners (2) and their welds, after selecting them:



You will use the copy command, member+angle, *selecting a +Y beam face and leaving 180°*. And you get:



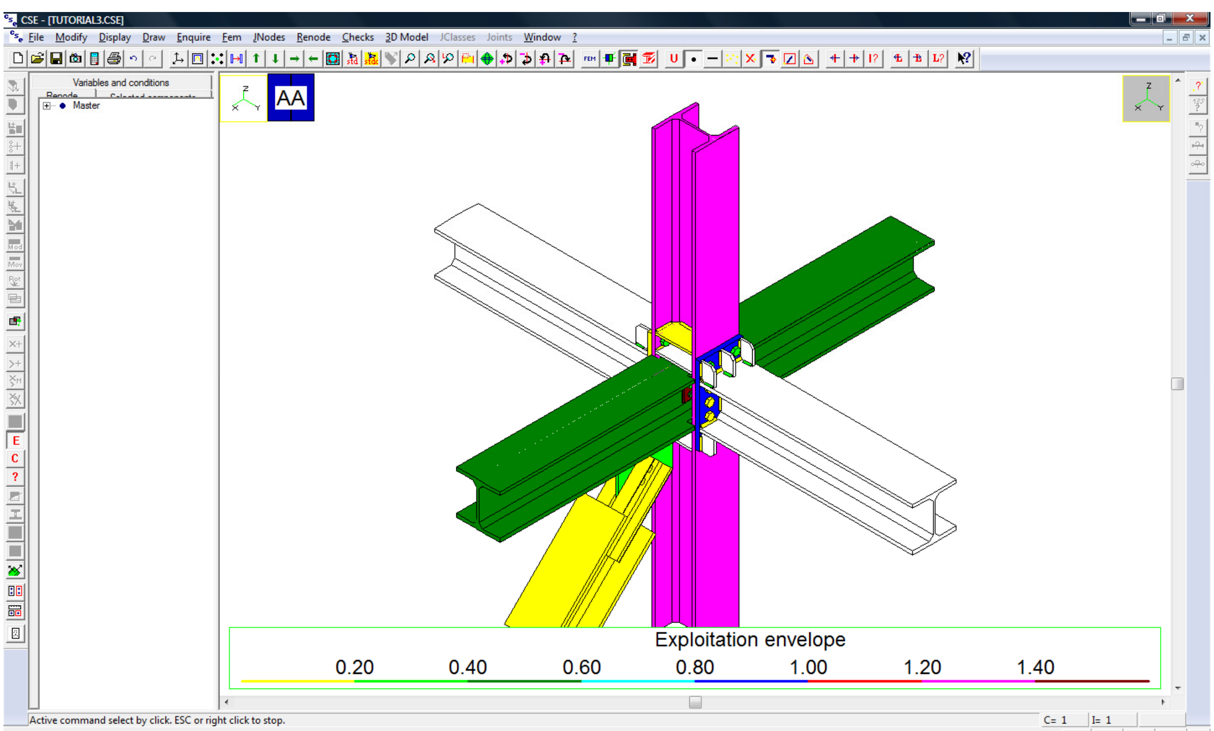
Now select the 4 new stiffeners, their welds and copy them with the member + angle mode, but now select a column face.



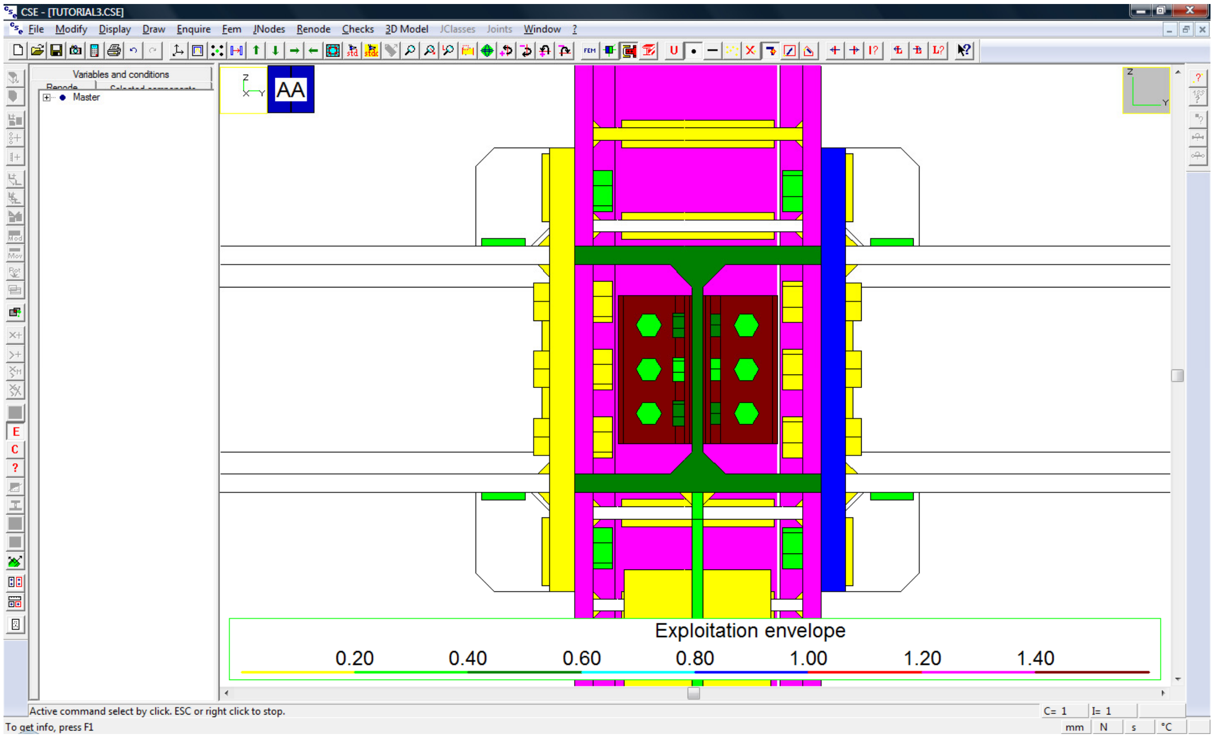
2.9.4 Re-run the checks and see what has changed.

Re-run the checks. No warning message about the displacement is issued.

This is the exploitation envelopes:



and the detail of the bolt connecting X beams to column:

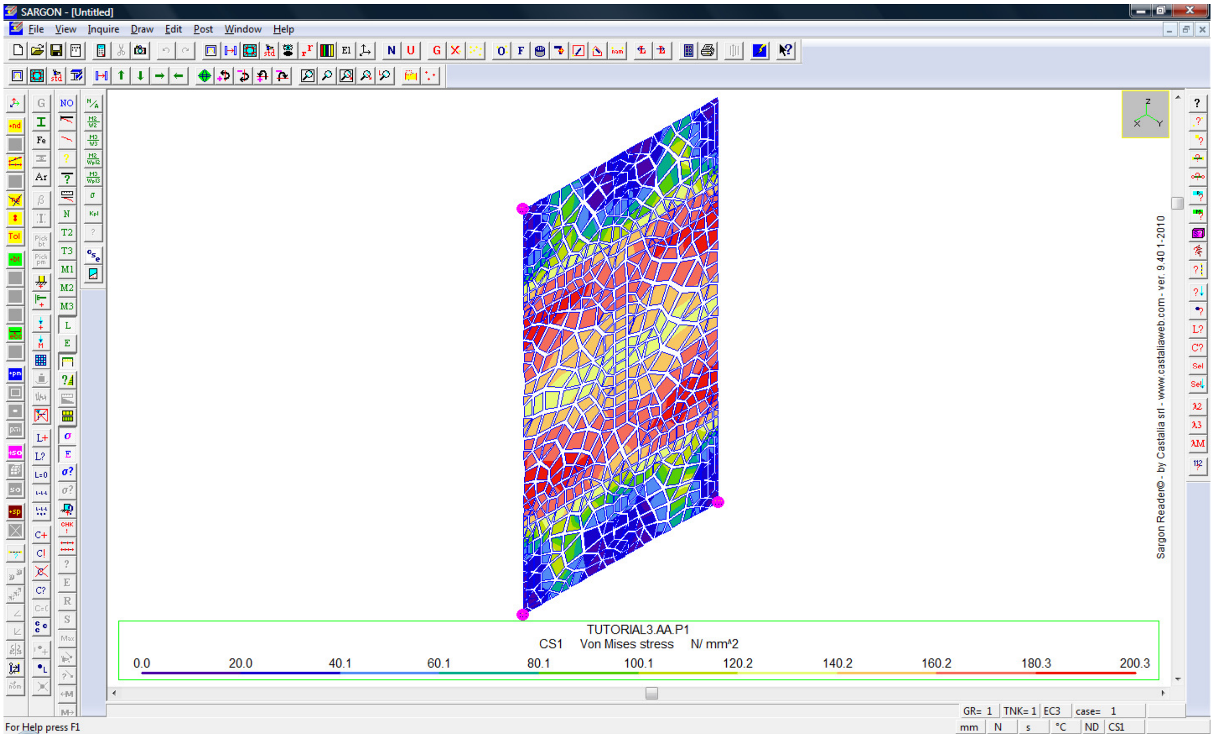


The following can be said:

1. The bolts are now checked.
2. The +Y plate is checked now (dark blue), the -Y is still yellow as no fem model was created for it.
3. The stiffeners are all white as they have not been checked by fem, anymore.
4. The column is still magenta.
5. The X beams are not anymore light blue but dark green. This is because the bolt diameter has increased, and so did the bearing surface for the bolt-stems bearing pressure checks.

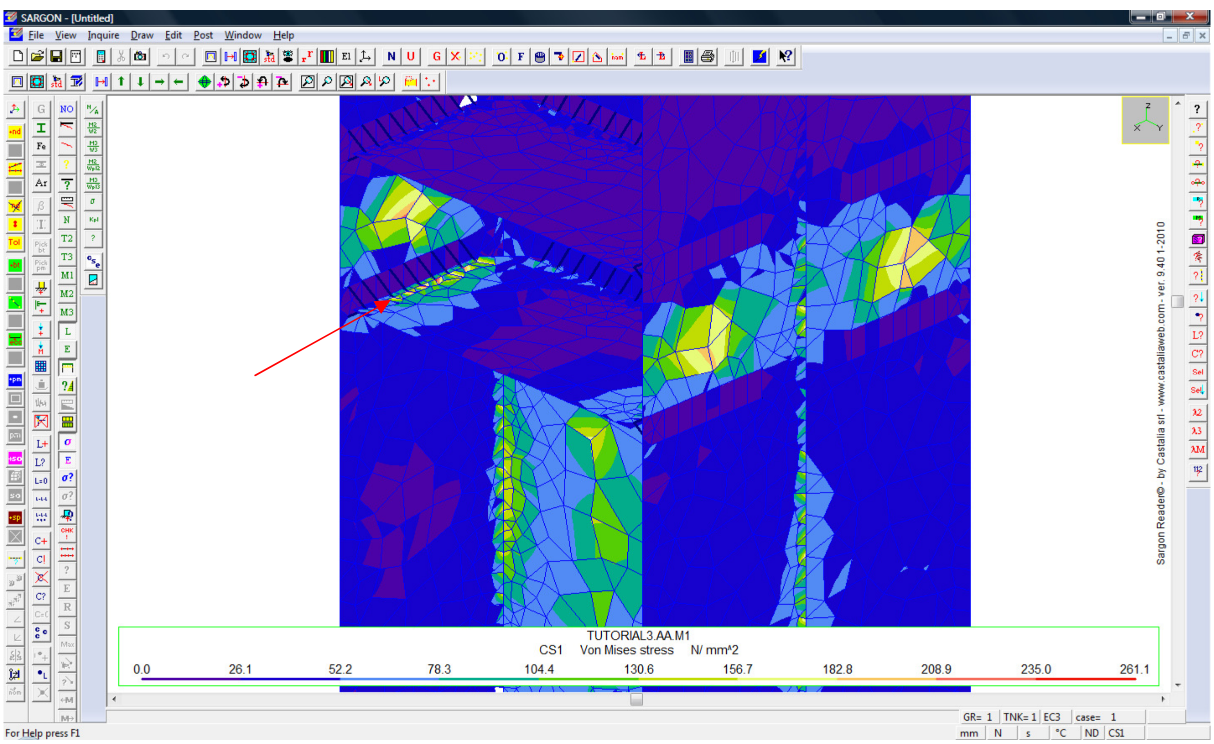
Let's have a look at the fem model results.

Here is the plate (P1) Von Mises stress envelope (interelement jumps NOT removed):



The maximum is 200.3 MPa. This values is lower than yield stress: the stiffeners we added help the plate to carry the loads.

Here is the column Von Mises stress envelope with the limit +235 fixed for the legend:



The situation is very much like the previous one. The component can be considered checked. If we wish to have it not-red in the exploitation envelope we can just remove the Create Fem model flag from this member (the column), increase the pressure limit from 3 to 10 in the bolt layout bearing material constitutive law definition, and re-run the checks. We will explain our fem analyses in the design documentation, to prove that the column can be accepted, or we will add specific user checks to keep into account in a simplified way the relevant checks.

1 INTRODUCTION 3

2 HOW TO BUILD A MULTIPLE MEMBERS RENODE 4

2.1 FOREWORD 4
2.2 STEP 6: CHECKING RENODE COHERENCE AND OVERLAPS 5
2.3 STEP 7: ADDING USER CHECKS 6
2.4 STEP 8: SETTING CHECK OPTIONS 11
2.5 UNDERSTANDING THE COMBINATIONS CREATED 15
2.6 STEP 9: EXECUTING FIRST LEVEL CHECKS 17
2.7 STEP 10: LOOKING AT RESULTS 18
2.7.1 LISTING 18
2.7.2 ENVELOPE RESULTS 22
2.7.3 BOLT LAYOUT WITH BEARING SURFACE 25
2.8 STEP 11: ADDING CHECKS USING FEM ANALYSIS OF COMPONENTS 30
2.8.1 LOOKING AT FEM MODEL RESULTS, STIFFENER JOINING PLATE AND +Y BEAM 37
2.8.2 LOOKING AT FEM MODEL RESULTS, PLATE JOINING DIAGONAL TO +X BEAM 41
2.8.3 LOOKING AT FEM MODEL RESULTS, PLATE JOINING +Y BEAM TO COLUMN 45
2.8.4 LOOKING AT FEM MODEL RESULTS, COLUMN 48
2.9 STEP 12: CHANGES IN THE DESIGN 51
2.9.1 REMOVE THE SHEAR ONLY FLAG 51
2.9.2 INCREASE THE BOLT SIZE 52
2.9.3 ADD A SET OF STIFFENERS TO THE PLATE 53
2.9.4 RE-RUN THE CHECKS AND SEE WHAT HAS CHANGED. 56