

SCS
STEEL
CONNECTION
STUDIO

Manual

Version 19B

“SCS Steel Connection Studio” is a **Registered Trademark** in the European Community (010010221), the International Bureau of WIPO (1155473), USA (4,493,056), Australia (1553799) and other countries.
US **Patent** pending

Table of contents

1	Introduction	8
1.1	Eula	8
1.2	General remarks	8
1.3	Demo version	9
1.4	Language	9
2	Installation	10
3	Licensing procedures	11
3.1	General information	11
3.1.1	Online check	12
3.2	Internet Floating license	13
3.2.1	Basics	13
3.2.2	Download license option	13
3.2.3	INSTALLATION ON MULTIPLE DEVICES, read carefully thanks	14
4	Initial window (form)	16
4.1	Basics	16
4.2	Buttons to open single modules	18
4.3	Other buttons	19
5	General layout in modules	21
5.1	Data tab	22
5.2	Message window	22
5.3	'Your comment' window	22
5.4	Graphic window	22
5.5	'Update calcs' button	23
5.6	Drop- down menu for graphic options	23
5.7	'Report 'button and related options	23
5.8	'Create DXF' button	23
5.9	Result tab	23
6	Menu and button bar	24
6.1	File menu	24
6.1.1	New	24
6.1.2	Open	24
6.1.3	Save	24
6.1.4	Save as	24
6.1.5	Print	25
6.1.6	File list	25
6.2	Edit menu	26
6.2.1	Undo	26
6.2.2	Redo	27
6.2.3	Edit Database	27
6.2.4	Cut	27
6.2.5	Copy	27
6.2.6	Paste	27
6.2.7	Increase drawing text size	27
6.2.8	Decrease drawing text size	28
6.3	Tools menu	28
6.3.1	General options	29
6.3.2	Member filter	31
6.3.3	Units (General) and Units (Form)	33

6.4	Drawing view menu	34
6.4.1	Fit all	34
6.4.2	Zoom window	34
6.4.3	Pan	34
6.4.4	Magnify	34
6.4.5	Shrink	34
6.4.6	Previous view	34
6.4.7	Reset dimension location	35
6.5	Import menu	35
6.5.1	Sap2000	35
6.5.2	Staad	39
6.5.3	Tekla Structures	39
6.6	Examples menu	40
6.7	Help menu	40
6.7.1	Contents	40
6.7.2	About	41
6.7.3	Send request information	41
7	Data tab	43
7.1	General	43
7.1.1	Initial description	43
7.1.2	Standard	43
7.1.3	Language	43
7.1.4	Calculation update	43
7.1.5	Eccentricity calculation method	44
7.1.6	Connection axis	44
7.1.7	Connection type	45
7.1.8	Category for shear	46
7.1.9	Moment calculation method	46
7.1.10	Double sided connection	47
7.1.11	Seismicity: joint is dissipative	47
7.2	Loads	48
7.2.1	Cases	48
7.2.2	γM	49
7.2.3	Optimize Bolt Tension + Shear combo	50
7.3	Members	50
7.3.1	Select member	50
7.3.2	Section properties button	51
7.3.3	Select material	52
7.3.4	Other options	52
7.4	Plate & bolts	53
7.4.1	Bolts	53
7.4.2	Plate material	54
7.5	Geometry	54
7.5.1	Bolt options	54
7.5.2	Plate options	55
7.5.3	Beam clearance	55
7.5.4	Eccentricity of bolt group, plate and secondary member	55
7.5.5	Plate buckling length	56
7.5.6	Block shear $k_{unif}=1$	56
7.6	Weld, notch & stiffeners	57
7.6.1	Weld	57
7.6.2	Weld eccentricity	57

7.6.3	Do not consider compression for welds	57
7.6.4	Stiffeners	58
7.6.5	Notch	58
7.6.6	False flanges	59
8	Result tab	60
8.1.1	Formulas	60
8.1.2	Case selection	60
8.2	Sum up	61
8.3	Bolts	61
8.3.1	T-stub button	64
8.4	Plate bearing and/or beam bearing	64
8.5	Plate block shear	65
8.6	Section block shear	66
8.6.1	Splice	67
8.7	Plate/beam strength	67
8.8	More	67
8.9	Welding	67
8.9.1	American codes	68
8.10	Stiffness	68
9	Fin-plate column-beam (FP1)	69
9.1	About FP1	69
9.2	Connection type	69
9.3	Plate welding to primary	70
9.3.1	Fin plate welded only on primary member web	70
9.3.2	Fin plate welded on primary member web and top stiffener	72
9.3.3	Fin plate welded on primary member web and both stiffeners	73
9.3.4	K N web and eccentricity	74
10	Fin-plate beam-beam (FP2)	75
10.1	About FP2	75
10.2	Connection type	75
10.3	Plate welding to primary	75
10.3.1	Fin plate welded on primary member web	76
10.3.2	Fin plate welded on primary member web and top flange	76
10.3.3	Fin plate welded on primary member web and both flanges	77
11	Double-fin plate (DS1)	79
11.1	About DS1	79
11.2	Connection type	79
12	Splice (SP4)	80
12.1	About SP4	80
12.2	Connection type	80
13	End plate column-beam (EM3)	82
13.1	About EM3	82
13.2	Top/bottom configuration	82
13.3	Loads	83
13.3.1	k_{wc}	83
13.3.2	Axial force modifies M_{maj}	83
13.3.3	Inelastic deformations are accepted	84
13.4	Members	84
13.4.1	Supplementary web plates	84

13.4.2	Continuous column in moment connection	85
13.5	Weld, Notch & Stiffeners	85
13.5.1	Continuity stiffeners on column	85
13.5.2	Diagonal stiffeners	86
13.5.3	Intermediate stiffeners on beam	86
13.5.4	Backing plates	87
14	Apex end plate (AP7)	88
14.1	About AP7	89
14.2	CHS connections	89
15	Welded end plate (WE1)	90
15.1	About WE1	90
16	Bolted angles column-beam (QS5)	91
16.1	About QS5	91
16.2	Connection type	92
17	Bolted angles beam-beam (QS6)	93
17.1	About QS6	93
18	Flexible end plate column-beam (XL1)	94
18.1	About XL1	94
18.2	Plate welded to secondary	94
19	Flexible end plate beam-beam (XL2)	95
19.1	About XL2	95
19.2	Secondary plate bolted to	96
20	Brace bolted to a plate welded to support (BR9)	97
20.1	About BR9	97
20.2	Plate shape	97
20.3	Plate welding to primary	98
20.4	Lug angle	98
20.5	Force distribution method: Uniform force method (UFM), KISS method	98
20.6	Members	99
20.6.1	Secondary member	99
21	Column base plate and anchors (CB8)	103
21.1	About CB8	103
21.2	Shear force	103
21.3	Base plate calculation method	104
21.4	Top configuration	104
21.5	Bottom configuration	104
21.6	T-stub prying force	104
21.7	Plate & Bolts	104
21.8	Geometry	105
21.9	Circular hollow section connections	106
22	General check (ME0)	108
22.1	About ME0	108
22.2	General	108
22.3	Loads	108
22.4	Graphics	108
22.5	Members	108
22.6	Bolts	109
22.7	Welds	110

23	Database editing	111
23.1	Input for bolt size db: Agr and Ares	111
23.2	How to control visibility	112
23.3	Automatic import	113
24	Reports	114
24.1	Calculation report	114
24.2	T-stub report	114
25	Request support	115
26	Validation	116
26.1	General	116
26.2	Flexible end plate with EC3	116
26.2.1	Geometrical and mechanical data	117
26.2.2	Loads	117
26.2.3	Joint shear resistance	117
26.2.4	Joint tying resistance	119
26.2.5	References	120
26.3	Fin plate connection with EC3	120
26.3.1	Geometrical and mechanical data	121
26.3.2	Loads	121
26.3.3	Joint shear resistance	121
26.3.3.5	Fin plate in bending and buckling of the fin plate	122
26.3.3.6	Beam web in bearing	122
26.3.3.7	Beam web in shear	122
26.3.3.8	Block shear of the beam web	122
26.3.4	Joint tying resistance	122
26.3.4.1	Bolts in shear	122
26.3.4.2	Fin plate in bearing	123
26.3.4.3	Fin plate in tension	123
26.3.4.4	Beam web in bearing	123
26.3.4.5	Beam web in tension	123
26.3.5	References	123
26.4	AS4100 code validation examples	124
26.4.1	Worked Example 9.9.1	124
26.4.2	Worked Example 9.9.2	126
26.4.3	Worked Example 9.9.3	127
26.4.4	Worked Example 9.9.5	128
26.4.5	Worked Example 9.9.6	129
26.4.6	Worked Example 9.9.7	130
26.4.7	Worked Example 9.9.8	132
26.4.8	Worked Example 9.9.9	133
26.5	End plate moment connection with EC3	135
26.6	Section property validation	137
26.7	SNiP validation	138
27	Technical issues	141
27.1	Moment connections: formula to find Y_c in column when only 2 bolts are present without transverse stiffener	141
27.2	Instantaneous center of rotation method with no vertical shear	141
27.3	SNiP notes	141
27.3.1	Block shear	141
27.3.2	Compression and buckling	142

27.3.3	Bolt bearing and minimum distances	142
27.3.4	Gamma coefficients	142
27.3.5	Weld check	142
27.4	AS 4100 notes	143
27.4.1	Block shear and long joints	143
27.5	Double sided connections	143
27.5.1	Moment end plates (em3 and we1)	143
27.6	Base plate calculations according to AISC Design Guide 1	143
27.7	Properties of some profiles according to the AISC manual	143
27.8	Japanese standard notes	144
27.9	Prying action check in shear connections (modules XL1, XL2, QS5, QS6)	144
28	API	146
29	Automatic design by using Sap2000, Etabs, Staad	147
29.1	General concepts	147
29.1.1	Meaning of file statuses	152
29.2	Project file save	153
29.3	Other notes	154
30	Automatic design by Tekla Structures	155
30.1	General concepts	155
30.2	Change Tekla Structure version	158
30.3	Other notes	160
31	Resistance Domains	161
31.1	General concepts	161
31.2	3D Domains	162
31.3	Export	163
31.4	Other notes	163
32	Standards and bibliography	164
33	EULA - End User License Agreement	166

1 Introduction

1.1 Eula

It's very important the E.U.L.A. (End User License Agreement) is read, understood and fully accepted by the user.

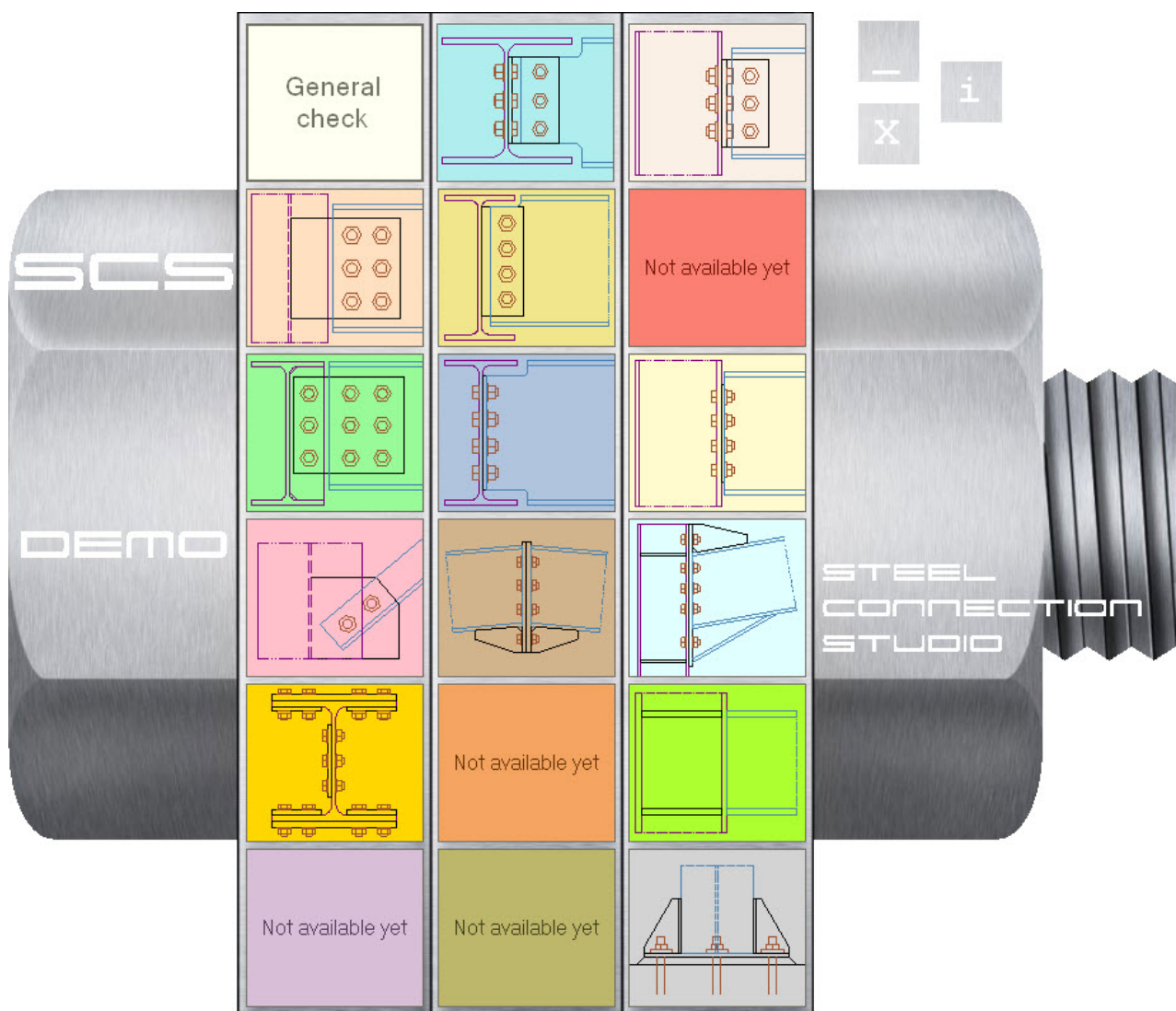
Please read it very carefully at par. 33 *EULA - End User License Agreement*.

Full acceptance is essential in order to use SCS.

1.2 General remarks

SCS is a software tool to help engineers designing bolted and welded connection.

It is divided into modules, each specific to a particular type of connection.



1.3 Demo version

With a demo license (see previous image, the word "Demo" is visible on the left hand side of the 'bolt') you cannot:

- use the program for commercial purposes
- save data (that is, files)
- the members you choose can't be deeper than 14.21" = 361mm or wider than 10" = 254mm. If width or depth are exceeded, exceeding value is changed to the maximum allowed value
- you cannot generate more than 25 reports.

Besides these limitations, the demo version has all the same features of the licensed version.

1.4 Language

SCS has only one language (English) for all communications with the user. This means that the 'Help' system and all menus and messages for the user are in English.

Support too is provided in English.

The option for other languages, if licensed, is available only for all data related to the report, that is for communications to 'end customer', because local regulations might require reports to be in the local language when engineering connection calculations are submitted to National authorities. Therefore, when the option for other languages is licensed, the user can choose a different language and the translations will appear only in the 'data tab' and 'results tab' because those are used to actually write the report.

In other words, a good knowledge of English is required to use SCS, as specified also in the EULA.

2 Installation

SCS installs through an .exe as standard Windows programs.

Please contact support@scs.pe or support@steelconnectionstudio.com if you experience any problem.

3 Licensing procedures

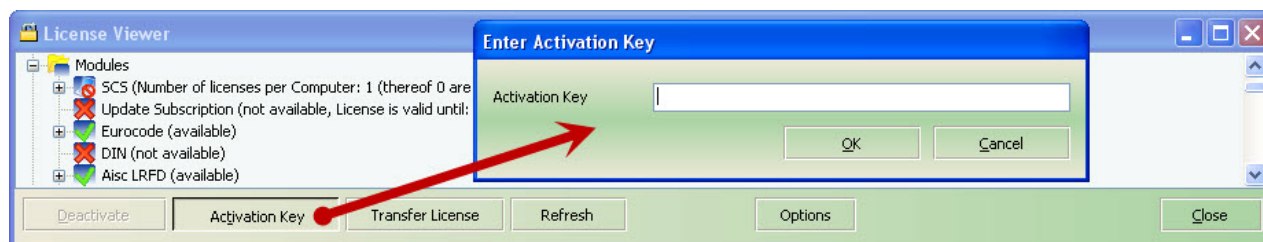
3.1 General information

Initially SCS will ask for the 'Serial Number', which is stated in the e-mail you have received as first. We suggest you use the 'copy' and 'paste' feature of Windows to avoid mistakes.

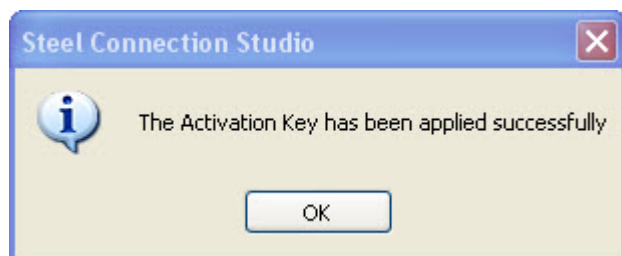
Then, all activation keys received in the mail after your purchase must be copied and pasted into the 'Activation key' box.

If you copy the keys during your first activation (serial number is input), the activation key box will appear automatically to input all the keys.

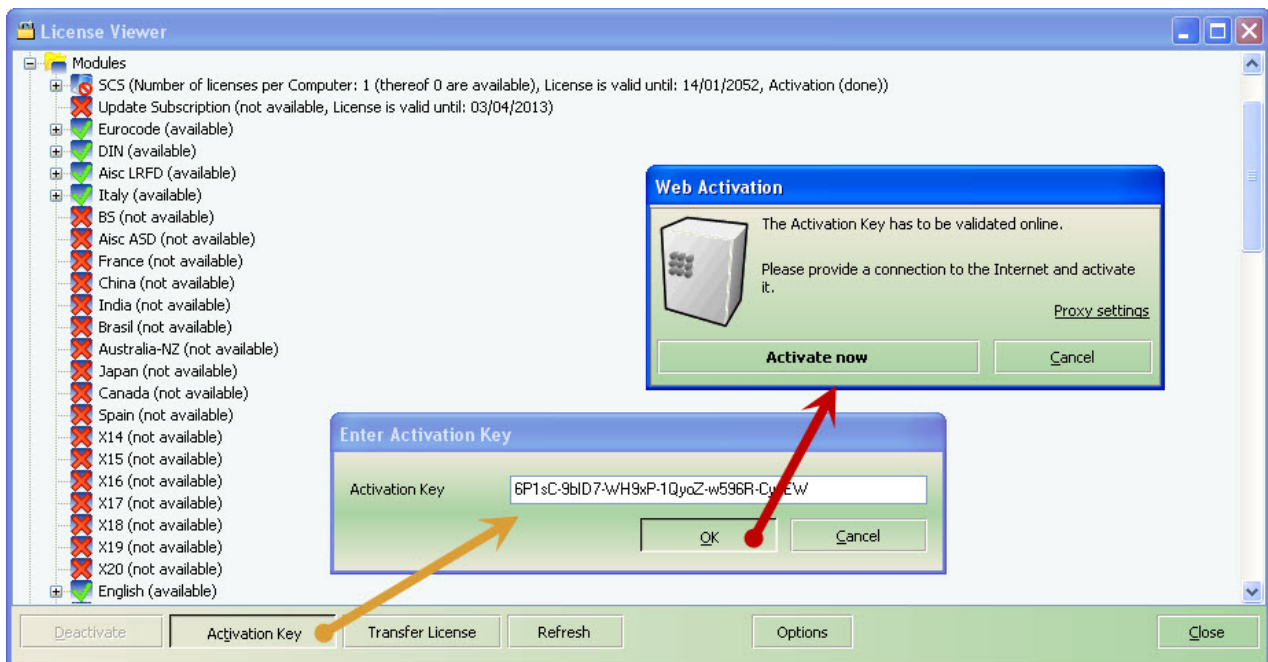
If you are buying keys at a later time, to be able to input them please click on “i” button on the initial window (form), then on “License information” and “Activation Key”.



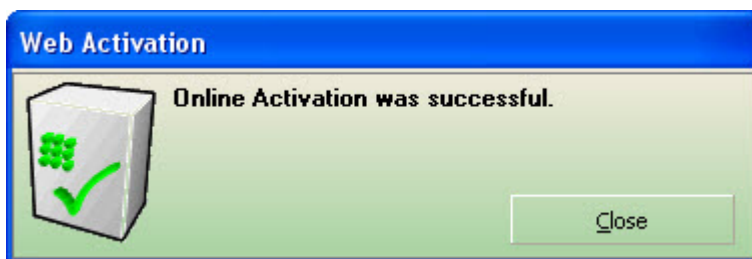
You can digit the license or you can copy the license key and past it into the Activation key box, then click the OK button. If everything is ok, SCS shows you this message:



If online activation is requested, then when you click OK it appears another window that asks you if you want to activate the license now or in a second time.

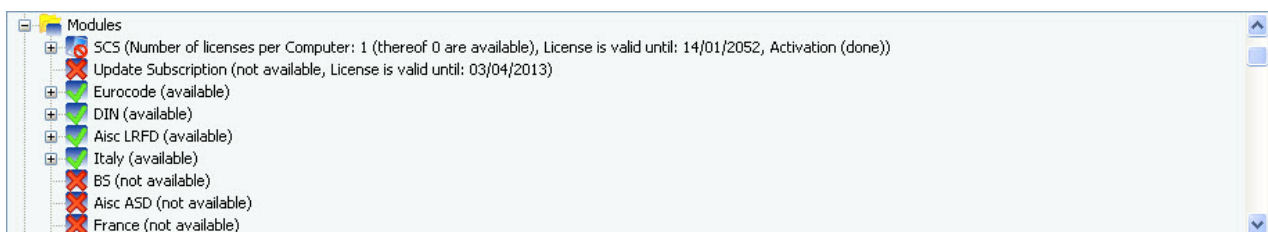


If you have an active internet connection, you can click “Activate now”, and if there are n issues, SCS shows you this message:



If more keys are to be registered, the procedure must be repeated as necessary.

After the activation, the new license will show as activated on the computer.



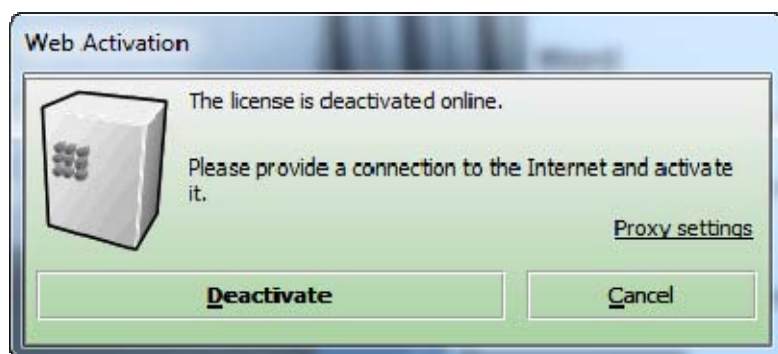
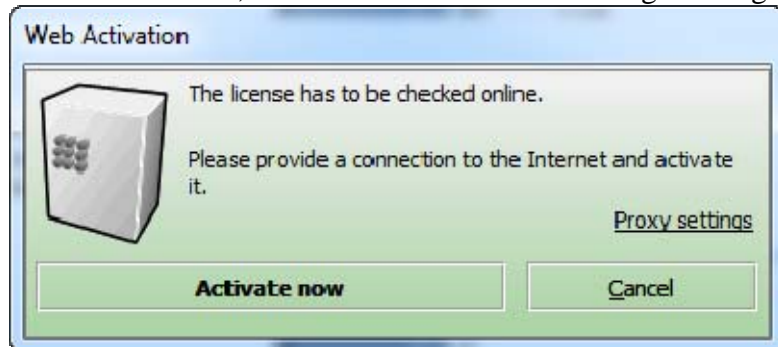
3.1.1 Online check

The license needs to be checked online every 3 months approximately. When the user is required to do so, it's necessary to make the check otherwise the license will stop working after a short 'grace' period.

3.2 Internet Floating license

3.2.1 Basics

The internet floating license allows customers to use the same license over multiple devices, though not simultaneously. The license is, through a internet connection, checked out and, at the end of the session, checked back in. The following messages will appear at start and end.



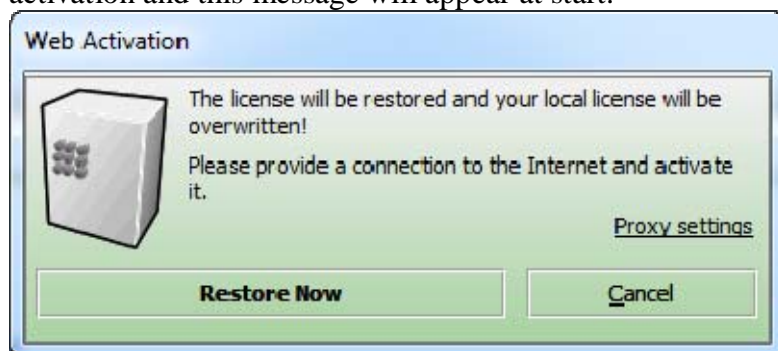
If the user wants to temporarily lock the license to one device because he/she knows he/she will not have an internet connection available (example: a long flight) or simply because for some time the license will be used only on one device, this can be done clicking, in the general options, the following option.

☒ Turn off Internet floating license
(only takes effect if floating license is owned)

Unchecking the option will reactivate the floating logic.

3.2.2 Download license option

Some customers with a high number of licenses download the license from the server at every activation and this message will appear at start:



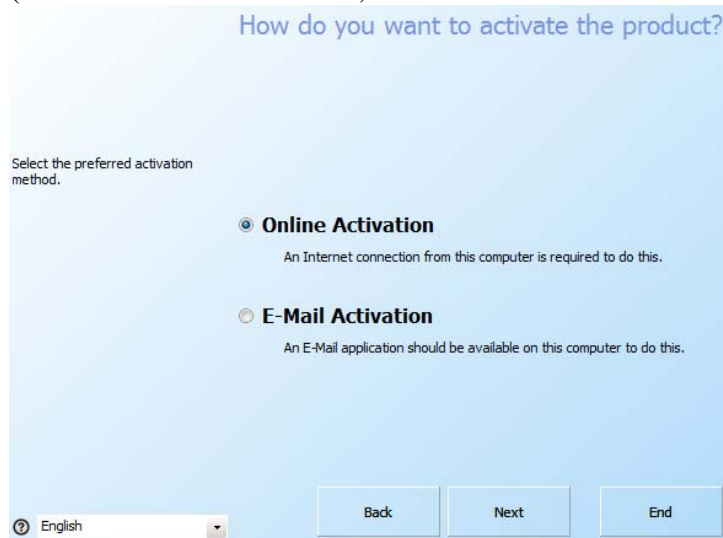
The advantage of this system is that activation is faster (only serial number and one activation code are needed) and that, if there are changes to the license (that is, the customer buys additional standards or options), it is not necessary to update all licenses on every device (the updated license will be downloaded automatically).

3.2.3 *INSTALLATION ON MULTIPLE DEVICES, read carefully thanks*

If a customer buys 2 floating licenses, he/she will receive:

- One serial number
- 2 sets of activation codes

The first 2 activations on the first 2 devices should be done with the standard procedure, that is choosing 'Online activation', then inputting the SERIAL NUMBER first then the activation codes (first set for the first device, second set for the second device).



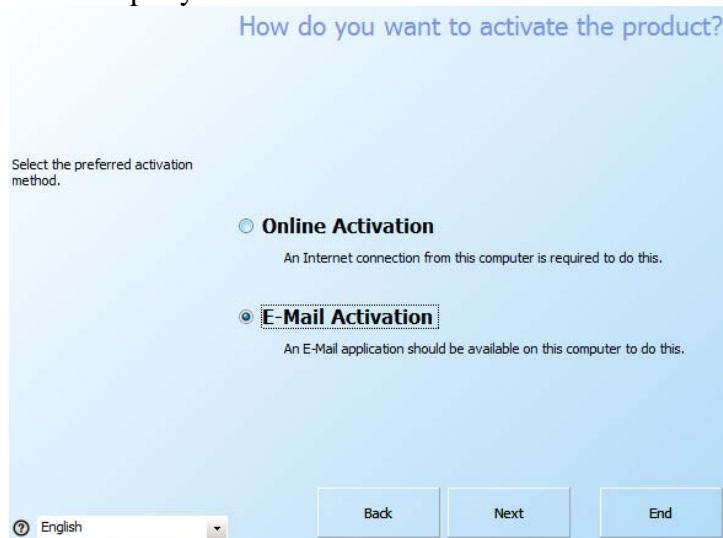
How to install on additional machines/devices?

Remember that activation codes can be used only once.

This means that for activating the software on the third, fourth, fifth (and so on....) device the same activation codes CANNOT be used.

Therefore for activation on additional devices you must choose:

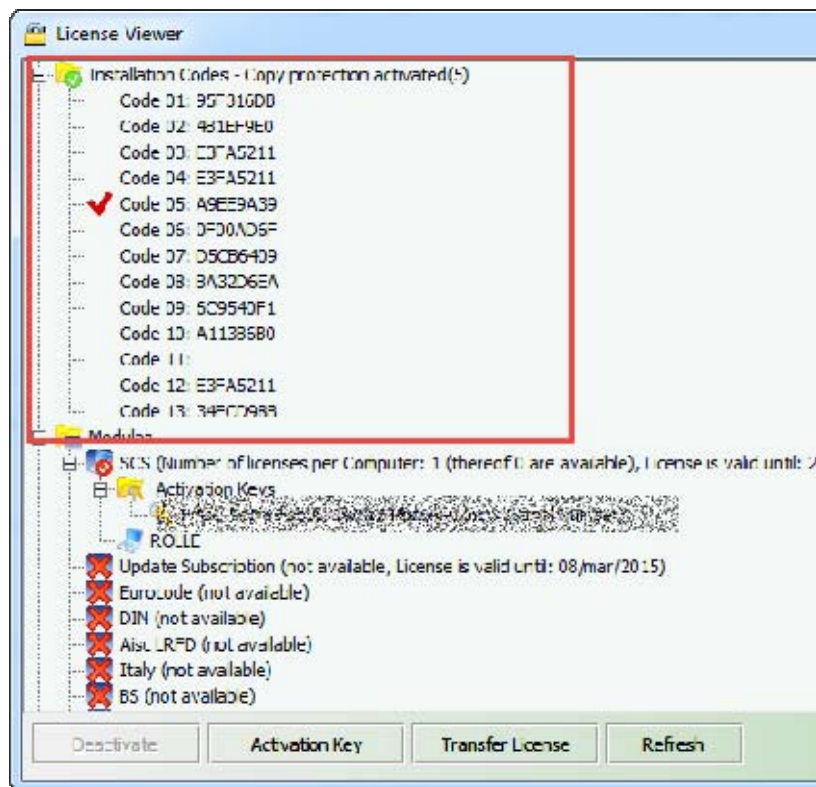
- **E-Mail activation (and not Online activation)**
- Input your serial number



Then, please send email to us and you'll shortly receive the 'unlock code' and a set of activation codes to be used on that specific device.

If you can't send automatically the email, copy and paste codes to an email you'll send to register@steelconnectionstudio.com

If, by mistake, online activation is chosen, the license can be validated (unless the maximum number of users are currently using it) but SCS still can't be operated (you will receive message "Insert all activation codes you bought, standard and/or language code missing") because standard and language activation codes were not given and inserted. In this case, you will have to open SCS license viewer and send a snapshot with the list of 'Installation codes' (see image below, you might have to scroll up first). After we receive this, we can send the activation set of language and standard codes.



4 Initial window (form)

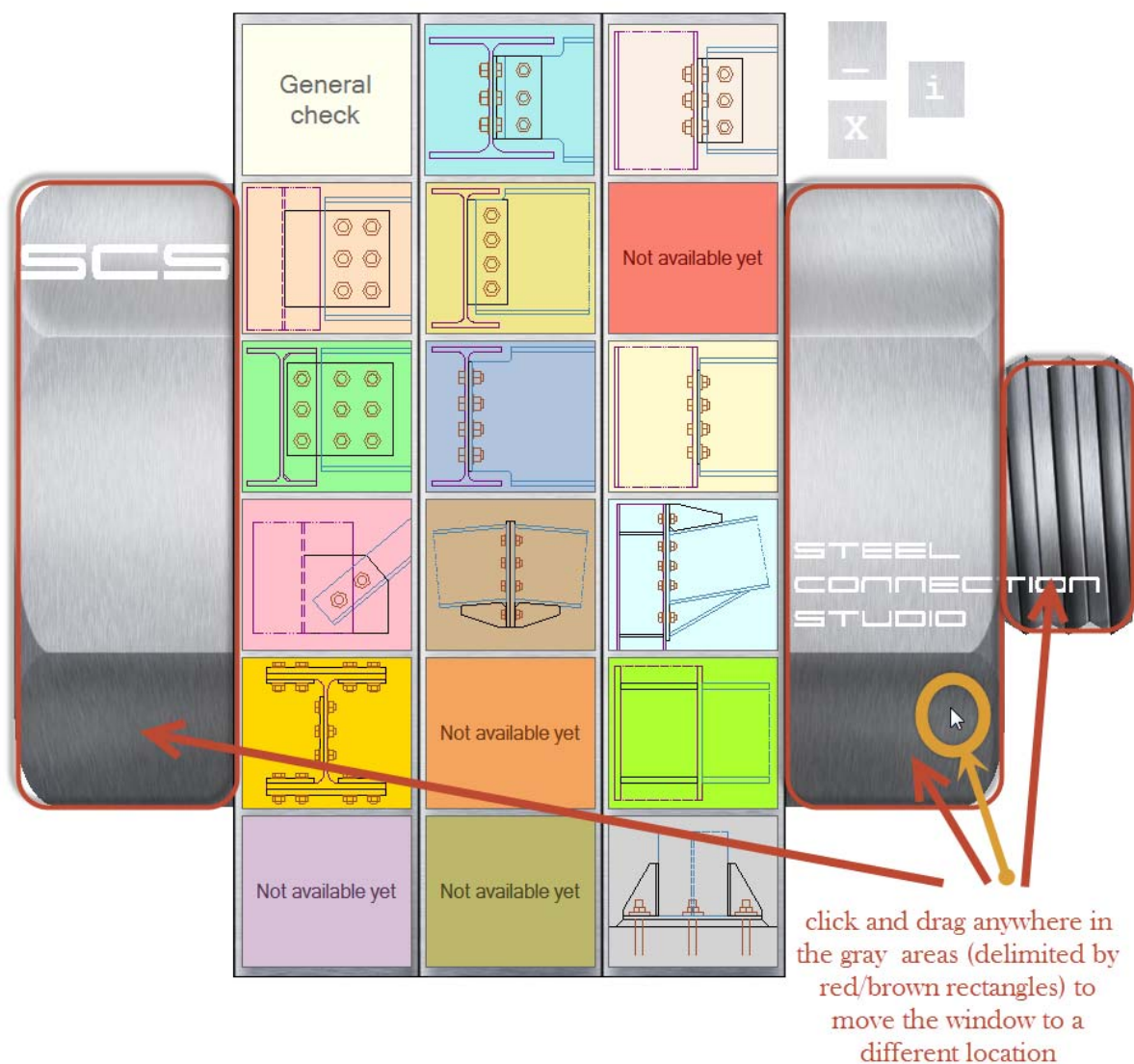
4.1 Basics

The bolted shape window (form) that initially opens when SCS is started is the control panel where the user can open the single modules to make connection design.

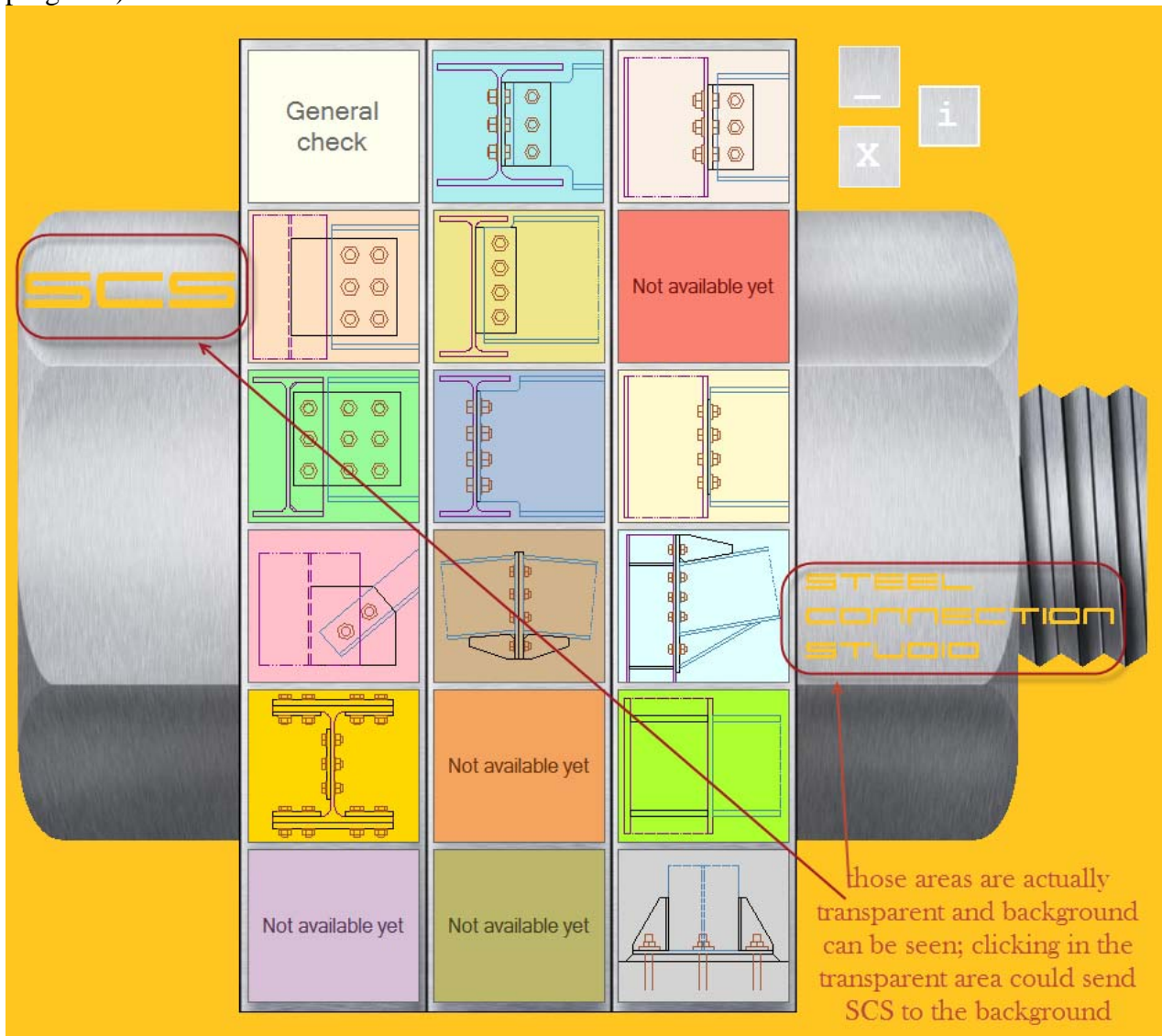
More modules, also of the same type, can be opened simultaneously, allowing the analysis and comparison of different files at the same time.

From this window the user can also execute actions like checking for updates, reviewing licensing information, open the Eula or access the help system.

To reposition the window in the desktop, the user can click anywhere in the gray area and drag the window to the desired location (see example below).



It should be noted that the words 'SCS' and 'STEEL CONNECTION STUDIO' are actually transparent and clicking on them would make active any other program that is actually 'below' the SCS window in the desktop (therefore causing SCS to go behind other programs active on the desktop; if this happens, SCS can be found from the task bar or minimizing those other active programs).

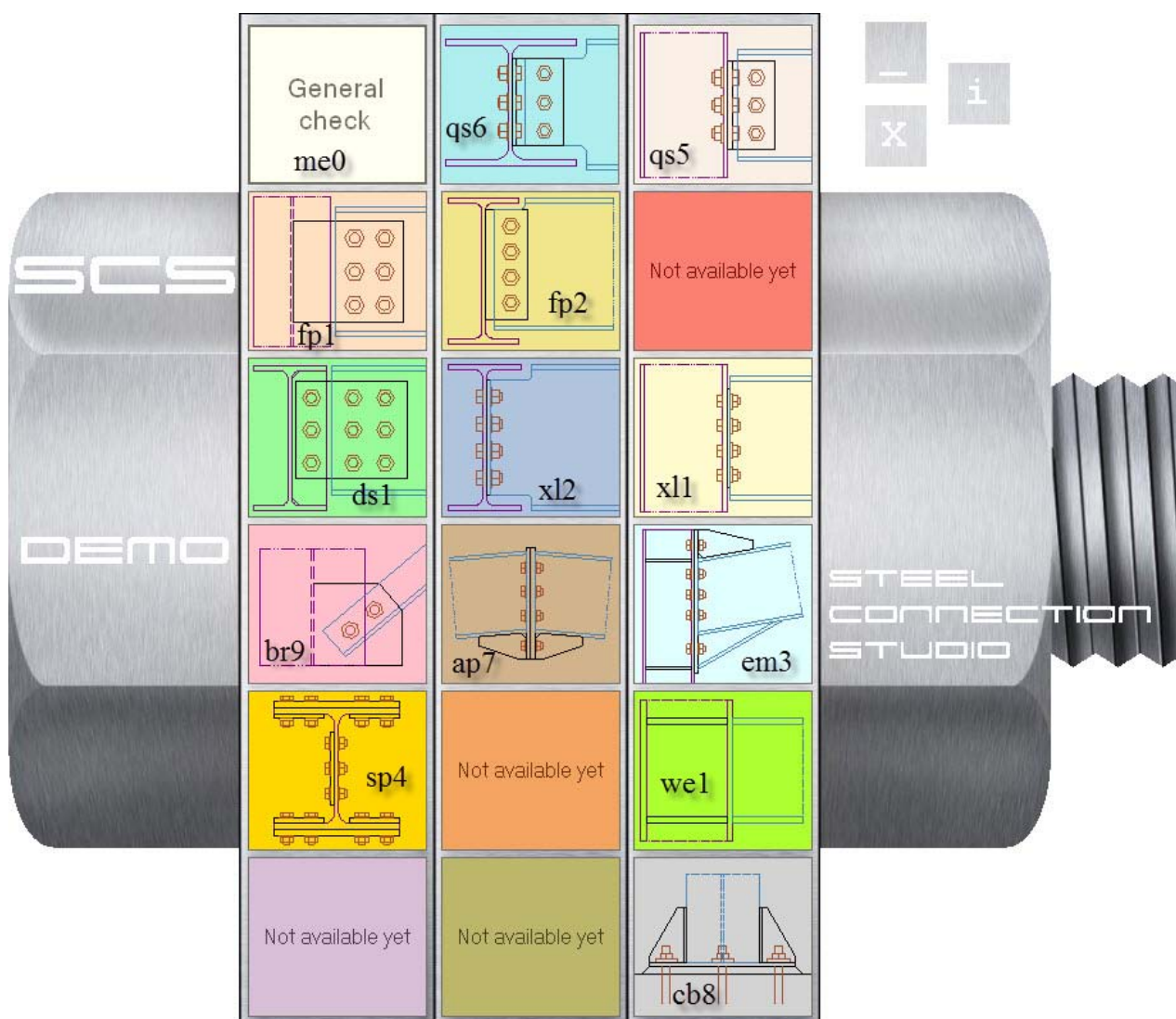


4.2 Buttons to open single modules

The available modules are (see following picture for clear reference):

- FP1 – fin plate design between column and beam – pink button, 2nd row, 1st column;
- FP2 – fin plate design between two beams – khaki button, 2nd row, 2nd column;
- EM3 – moment end plate, column-beam design – light cyan button, 3rd row, 3rd column;
- SP4 – splice design (beam-beam or column-column) – gold button, 4th row, 1st column;
- QS5 – bolted angles, beam-column design – linen button, 1st row, 3rd column;
- QS6 – bolted angles, beam-beam design – pale turquoise button, 1st row, 2nd column;
- AP7 – moment end plate, beam-beam design – tan button, 4th row, 2nd column;
- CB8 – column base plate and anchor bolts design – light gray button, 6th row, 3rd column;
- BR9 – brace design – pink button, 4th row, 1st column;
- ME0 – general check – ivory button, 1st row, 1st column;
- XL1 – flexible end-plate, beam-column design – lemon chiffon button, 3rd row, 1st column;
- XL2 – flexible end-plate, beam-beam design – light steel blue button, 3rd row, 2nd column;
- DS1 – double simple plate design between two beams – pale green, 3rd row, 1st column;
- WE1 – welded beam-column design – green yellow button, 5th row, 3rd column.

The button with the written “Not available yet” are modules that aren’t available yet.





In *License Information* you can check your license situation or execute tasks like activate a new license with *Activation key* or transfer a license to another pc. See chapter 3 *Licensing procedures*.

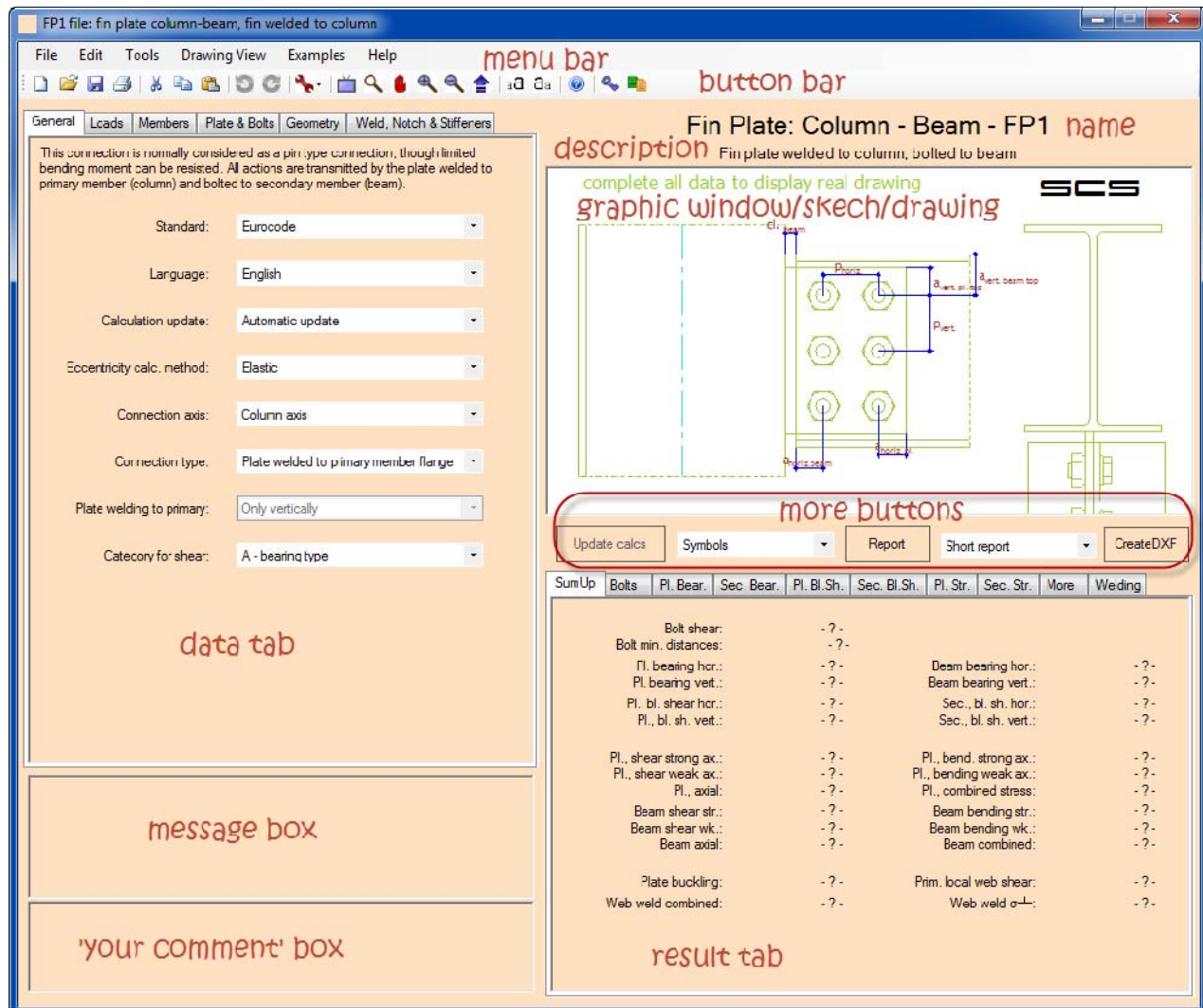
With *Help* button you can access the help manual.

The program automatically performs the search for software updates every two weeks. However, clicking the *Check for updates* button you can launch the search for software updates whenever you want (an internet connection is required).

The *Eula* button will then open the End User License Agreement, see par. 30

5 General layout in modules

SCS modules have their working window (form) built as below:



- the menu bar to manage file functions, units and database of profiles, bolts and materials, to manage the display of the graphic window, access help, examples and all other tools and options of SCS;
- the button bar that recalls frequently used commands from the menu bar;
- the data tab, where the user can insert the data of the connection (standard, loads, geometry and materials);
- the message window with warnings and /or error message that are displayed automatically during the design;
- the name of the module being used, with the code being used to save its files and with a brief description
- 'your comment' box where the user can write your own notes
- the graphic window (called also sketch or drawing), that usually represents the connection in two different views, front and top view, used to display in real-time the geometrical choices made by the user;
- the result tab, where the user can find all the checks of the designed connection;

- other buttons between the graphic window and the result tab, to manually update the calculations, to manage the graphic window information, to generate report, or to export the graphical window data into .dxf files.

5.1 Data tab





Here the user can insert the data for the design connection (standard, loads, profiles, bolts, welding information, stiffeners, notches...).

See specific paragraph 7 Data tab (which has a general approach) and the following paragraphs for single modules where different behaviors are eventually described and discussed.

5.2 Message window

Here there are messages displayed automatically during the design.

Different types of messages can be shown:

- suggestions, symbol 
- information messages, with symbol 
- warnings, symbol . This kind of message warns the designer about some potential problem. It takes a decision by the user (engineer) to decide if the warning is acceptable (that is, he/she considers the connection as ok) or not (design needs to be changed)
- ERROR messages, symbol . This kind of message is a SERIOUS ERROR, invalidating calculation hypotheses therefore it should not be overlooked and any connections showing this kind of message should not be considered as ok.

5.3 'Your comment' window

The user can write comments in this area. Data will be saved with the file and re-displayed when loading a file. The color of the text can be customized through the 'General option' button/menu. Your comments will also be written in the report.

5.4 Graphic window

Graphic window show the connection in two different views, front and top view (or lateral view, according to the module), and reports the main geometric data. To display the drawing the user have to complete all data in *Members*, *Plate & Bolts* and *Geometry*.

With the *Drop-down* menu below the window, the user can choose some different types of information to be displayed on the graph: the symbols used when entering data of the geometry, the dimensions, the governing limit state, that is most cumbersome limit state for the connection (the one in facts governing the design)

If the user changes any geometrical data of the connection, the changes show up in the graphic window in real-time.

The dimensions can be changed by clicking directly on them.

The location of the dimensions can be also changed by clicking and dragging them. This also applies to angles, welds, text. If **CTRL** is clicked while clicking and dragging, only the position of the text of the dimension moves. See the website (www.scs.pe) for some example videos.

When geometrical data entry is not complete, the display shows a 'dummy' connection with hypothetical data to help the user input values and understand geometry even though data are not completely entered. 'Fake' values are drawn in green and the warning 'complete all data to display real geometry' is visible on top instead of the sum up of the entered data.

5.5 '*Update calcs*' button

If the user choose Manual update of calculations (General/Calculation update/Manual update), the *Update calcs* button needs to be clicked to start a new calculation of the connection and get updated results.

If the user choose *Automatic update*, the *Update calcs* button is disabled, and the program will update calculation and result every time any data is changed in the model.

The automatic is turned off and option is turned to manual if rows of bolts are more than 10. The reason is that update of calcs is slow if the eccentricity calculation method is 'Instantaneous center of rotation'. The automatic turning of the option to manual doesn't apply to modules em3, cb8, ap7.

5.6 *Drop- down menu for graphic options*

With the *Drop-down* menu below the graphic window, the user can choose some different types of information to be displayed on the graph: the dimensions with symbols used when entering data on the geometry, the dimensions, the limit state which is more burdensome for the connection or nothing.

If the user select *Gover. Limit state*, the graphic window shows the item who referred the limit state which is more burdensome for the connection, and a text message it shows the percentage of the corresponding test.

5.7 '*Report*' button and related options

With this button the user can export the results data into a RTF file. The user can choose among four different kinds of reports: short, standard, full with selected cases and full with all cases).

The short report contains the general information, that is what shown in the 'general' page along with loading data. Results are shown only in the 'sum up' form.

The standard report contains the same results of the short report but it gives a full description of the input data.

The 'full (selected cases)' report adds to the standard report also the detailed analysis of the checks. What's reported though is only relative to the selected load cases (letters from "a" to "e") in the various tabs. If any tab has a selected case with no loading, report of that specific tab is not reported.

The 'full (all cases)' report is as above with all the detailed checks, case per case. Cases with no loading are not reported though.

5.8 '*Create DXF*' button

With this button the user can export the contents of the graphic window into a DXF file.

5.9 *Result tab*

The results of connection design are here thoroughly reported: the 'sum up tab' gives a brief outlook while single tabs analyze in detail results of single limit states, load combination by load combination and bolt group by bolt group if more than one bolt group is present (modules DS1, QS5, QS6, SP4, BR9 if lug angles are present).

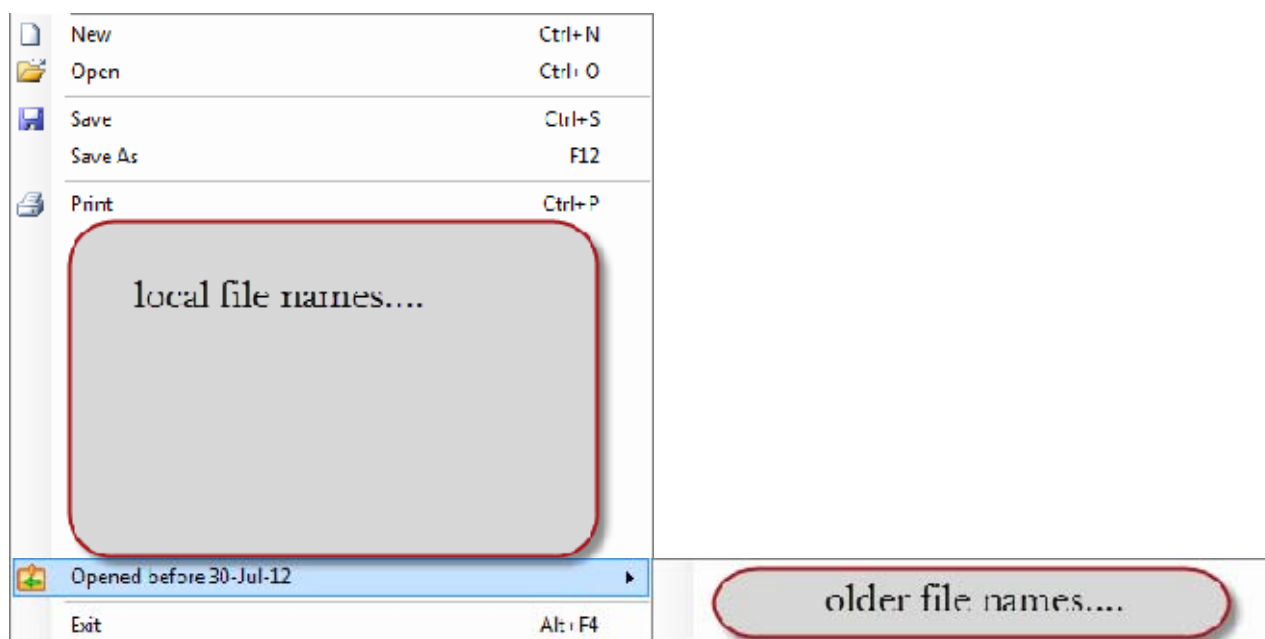
See specific paragraph 8 Result tab (which has a general approach) and the following paragraphs for single modules where different behaviors are eventually described and discussed.

6 Menu and button bar


Most of the functions are quite common and similar to many other programs, so self-explanatory.

6.1 File menu

In this menu the user can create a new file, open an existing file, save changes to the file, save your work as a different file, print the contents of the graphic window, or exit to SCS program. Short-cuts are available too and they are shown in the right hand side of the menu.




6.1.1 New

Also available from the button bar clicking 

This button creates a new SCS file.

Short cut key: ctrl + N


6.1.2 Open

Also available from the button bar clicking 

This button opens an existing SCS file.

Short cut key: ctrl + O

6.1.3 Save

Also available from the button bar clicking 

This button is to save the file. If no file name has been defined yet, a file name will be required, similarly to 'Save as'.


Short cut key: ctrl + S

6.1.4 Save as

This button is to save the current file with a different name and/or in a different location..

Short cut key: F12

6.1.5 Print

Also available from the button bar clicking 

This button is to print current drawing in the *Graphic window*.

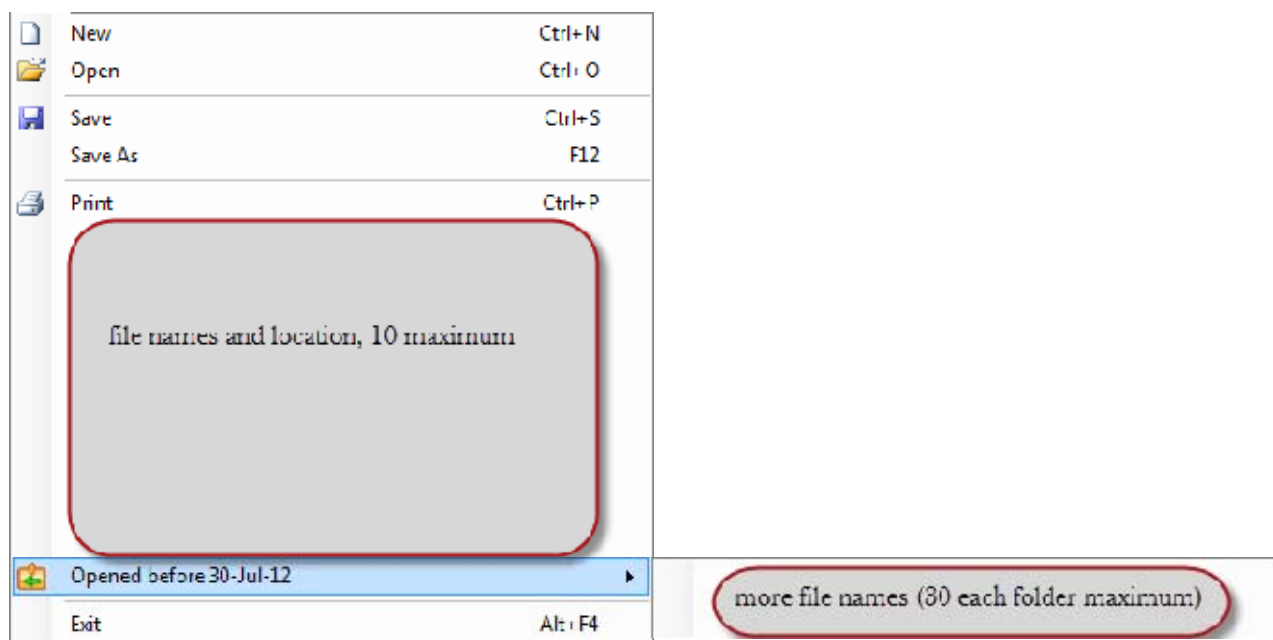
A window to set up the printer will pop up before 'real' printing.

Short cut key: ctrl + P

6.1.6 File list

To help the user to find quickly previously saved files, SCS stores location and name of those files.

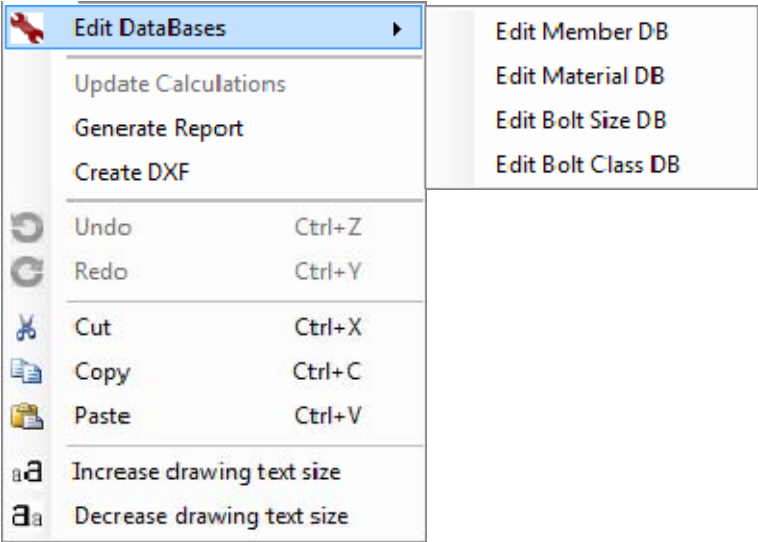
If more than 10 files were saved/opened, SCS creates folders, each containing maximum 30 files, with information about old files. When 30 files are reached, a new folder is created and the date of the last time it was opened the first file in the folder list is shown.




If any file cannot be reopened because it was deleted or location was changed, a message will be displayed and the path of this file will be **grayed** but still visible.

6.2 Edit menu

In this menu the user can edit data base of member, material, bolt size and bolt class used in the calculation, she/he can generate a report or create a DXF file. The user can also undo the previous change, return the last undo, cut/copy/paste data from to clipboard and he/she can change the font size of the drawing/sketch in the graphical window.



6.2.1 Undo

Also available from the button bar clicking 

This button is to go back step by step and cancel any input.

It's not possible to go back more than 25 'input'. Data like general options, units are also not included in the memorized input that can be restored.

Short cut key: ctrl + Z

The undo button might not change back all data if any input causes change in other related fields.

As an example, let's consider that in the following situation

The image shows a form titled 'Plate standard bolt spacing'. At the top, there is a checkbox labeled 'Plate standard bolt spacing' which is currently unchecked. Below the checkbox, there are five input fields with labels and units: 'P_{horiz.} =', 'P_{vert.} =', 'a_{horiz. pl.} =', 'a_{horiz. beam} =', and 'depth_{pl.} =', all followed by 'mm'. The input fields are empty.

the option 'Plate standard bolt spacing' is checked.


The image shows the same form as above, but now the checkbox 'Plate standard bolt spacing' is checked. The input fields are now populated with values: 'P_{horiz.} = 54 mm', 'P_{vert.} = 54 mm', 'a_{horiz. pl.} = 36 mm', 'a_{horiz. beam} = 36 mm', and 'depth_{pl.} = 162 mm'.

Checking this option will make the data below automatically complete. If the *Undo* button is now clicked, the 'Plate standard bolt spacing' is unchecked and the data below made editable but, see below, the values are not set back to empty values are they were before.

☐ Plate 'standard' bolt spacing

$p_{\text{horiz.}}$ =	54	mm
$p_{\text{vert.}}$ =	54	mm
$a_{\text{horiz. pl.}}$ =	36	mm
$a_{\text{horiz. beam}}$ =	36	mm
depth _{pl.} =	162	mm


6.2.2 Redo

Also available from the button bar clicking 

If *Undo*. button has been used, prior to make further changes, it's possible to move forward again and re-input data.

Short cut key: ctrl + Y

6.2.3 Edit Database


Also available from the button bar clicking 

This button is to edit database of SCS. Here, the user could see/modify the existing data or add new data in database of members, materials, bolt sizes and bolt classes used in the calculations.

Edit Member DB Edit Material DB Edit Bolt Size DB Edit Bolt Class DB

See also par. 23 *Database editing*.


6.2.4 Cut

Also available from the button bar clicking 

This button is to cut data into clipboard.

Short cut key: ctrl + X

6.2.5 Copy

Also available from the button bar clicking 

This button is to copy data to clipboard.

Short cut key: ctrl + C

6.2.6 Paste


Also available from the button bar clicking 

This button is to paste data from clipboard.

Short cut key: ctrl + V


It can be noted that pasting into multiple cells like in the load data table is possible, for example copying data from an Excel file (or similar).

6.2.7 Increase drawing text size

Also available from the button bar clicking 

This button is to increase the text size of the text in the drawing/sketch (graphic window).

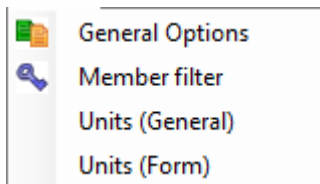
6.2.8 *Decrease drawing text size*

Also available from the button bar clicking .

This button is to decrease the text size of the text in the drawing/sketch (graphic window).

6.3 *Tools menu*

The *Tools* menu allows to access the following commands:



6.3.1 General options

Also available from the button bar clicking 

The submenu *Options* let the user manage a broad range of settings, as for example the number of washers for each bolt, or the shape of the stiffeners in the moment connections and so on.

It should be noted that those options are common to all modules and, if changed into one module, they apply to all the others too.

Also note that some options like the 'Default standard' or 'Default language' (when multiple standards and languages are licensed) only apply when a new file will be opened. That is, if language or standard of the module being used needs to be changed, it should not be changed from the 'General options' but it should be changed in the 'General' tab of the form.

All options are supposed to be self explanatory but a few comments are probably needed:

1) E (Young Elastic modulus) for calculations is taken as the one defined for material (see image below). The user could choose to overwrite E value of material using what's defined in the standard, regardless of what is defined in the database of the material.

Default values in standards:

Eurocode, NTC 2008 (Italy), Din18800: 210000 MPa = 30458 ksi

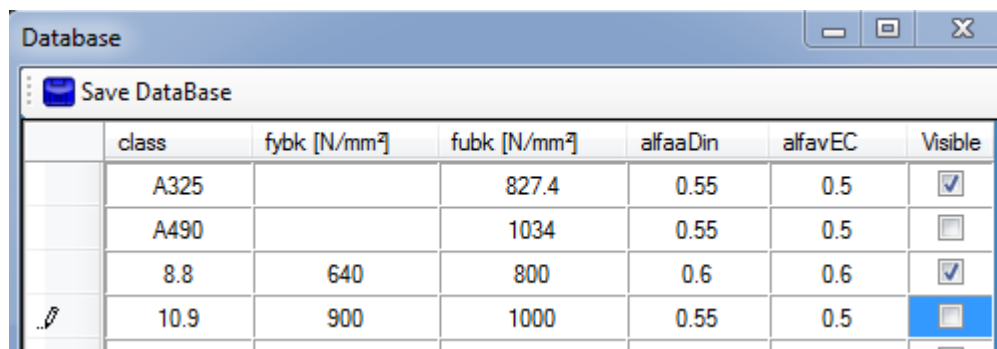
Aisc LRFD: 29000 ksi = 199948 MPa

2) the *Reset* button applies only to what encircled

3) the *Reset* button applies only to what encircled

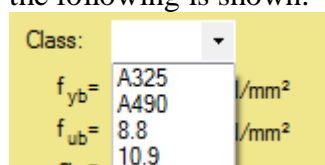
4) if this option is not checked, the 'visible' filter related to Material, Class and Bolt databases is actually not applied in the selection boxes, that is everything is visible

Example with the 'Class' database: let's consider a database with the following 'visible' data:



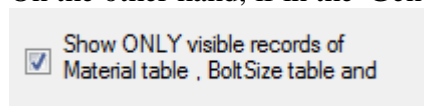
class	fybk [N/mm²]	fubk [N/mm²]	alfaaDin	alfavEC	Visible
A325		827.4	0.55	0.5	<input checked="" type="checkbox"/>
A490		1034	0.55	0.5	<input type="checkbox"/>
8.8	640	800	0.6	0.6	<input checked="" type="checkbox"/>
10.9	900	1000	0.55	0.5	<input type="checkbox"/>

If the 'General Option' checkbox is unchecked, when the class bolt selection is opened, the following is shown:

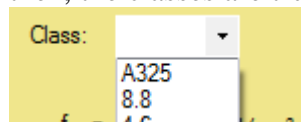


Therefore, class A490 and class 10.9 are actually visible.

On the other hand, if in the 'General option' the choice is




then, the classes are truly not visible when the class bolt selection is opened.



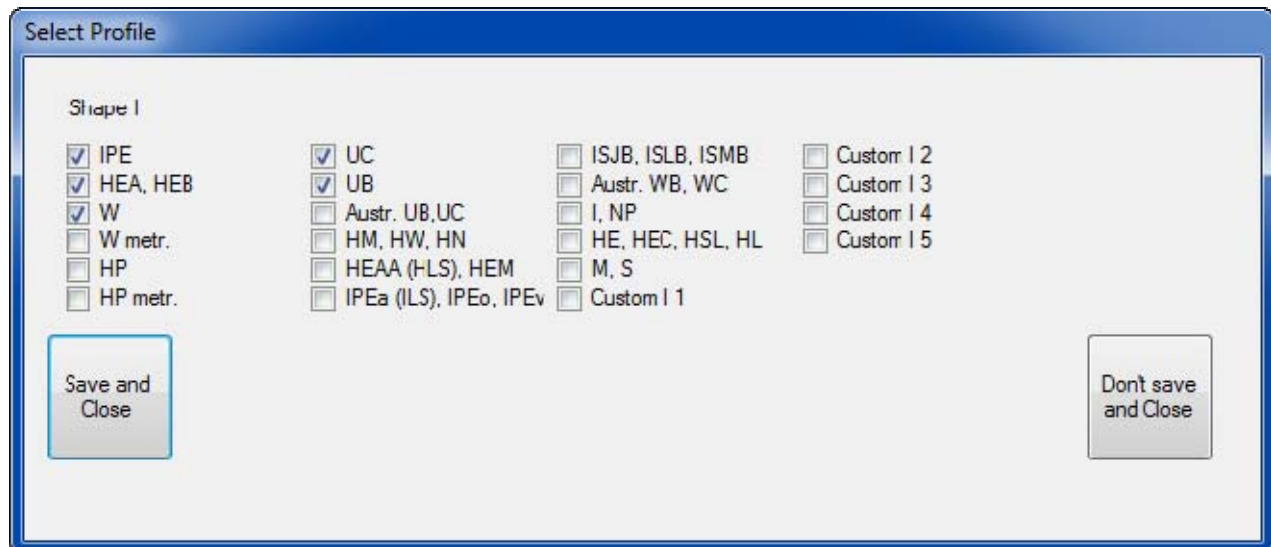
5) Increasing this value might be useful when importing from Staad or Sap2000 but it could slow consistently SCS if the number becomes too big (maximum number of combinations is set to 499). If a file with say, 15 load combinations is then opened with the option set to a lower number (say, 12), a warning will be displayed in the opening computer and only the first 12 combinations will be imported, the others will be lost.

6) When a company has the internet floating license and it wants to release one license from this option, locking it to a computer (example: one user will travel by plane and he/she will not have access to the internet to open the license), this option can be turned on.

6.3.2 Member filter

Also available from the button bar clicking 

In the submenu *Member filter* the user can apply a filter to the profile database that makes available for the calculations only the types of selected profile.



The user cannot clear a member group if that group is already selected in any active form/module.

In addition, **if a file is opened with members of a 'hidden' group, that group is made visible** (and the option saved so that group will be set visible in the filter).

HINT: If a group contains members that are not commonly used (say, HEB profiles deeper than 600mm), the user can change the group of those members to another group (say, *Custom I 5*) that is not commonly used and set as not visible.

If a module (for example BR9) allow other shapes (non I), groups of those other shapes available in the module can be edited.

Select Profile

Shape I

<input checked="" type="checkbox"/> IPE	<input checked="" type="checkbox"/> UC	<input type="checkbox"/> ISJB, ISLB, ISMB	<input type="checkbox"/> Custom I 2
<input checked="" type="checkbox"/> HEA, HEB	<input checked="" type="checkbox"/> UB	<input type="checkbox"/> Austr. WB, WC	<input type="checkbox"/> Custom I 3
<input checked="" type="checkbox"/> W	<input type="checkbox"/> Austr. UB, UC	<input type="checkbox"/> I, NP	<input type="checkbox"/> Custom I 4
<input type="checkbox"/> W metr.	<input type="checkbox"/> HM, HW, HN	<input type="checkbox"/> HE, HEC, HSL, HL	<input type="checkbox"/> Custom I 5
<input type="checkbox"/> HP	<input type="checkbox"/> HEAA (HLS), HEM	<input type="checkbox"/> M, S	
<input type="checkbox"/> HP metr.	<input type="checkbox"/> IPEa (ILS), IPEo, IPEv	<input type="checkbox"/> Custom I 1	

Shape L

<input type="checkbox"/> L (AISC)
<input checked="" type="checkbox"/> L equal metr.
<input checked="" type="checkbox"/> L unequal metr.
<input type="checkbox"/> Custom L 1
<input type="checkbox"/> Custom L 2
<input type="checkbox"/> Custom L 3

Shape U

<input checked="" type="checkbox"/> U (UPN)	<input type="checkbox"/> ISJC, ISLC, ISMC
<input checked="" type="checkbox"/> UAP, UPE	<input type="checkbox"/> Custom U 1
<input type="checkbox"/> C, MC	<input type="checkbox"/> Custom U 2
<input type="checkbox"/> JIS, GB-C, YR-C	<input type="checkbox"/> Custom U 3
<input type="checkbox"/> PFC	
<input checked="" type="checkbox"/> UAP, UPE	

Shape PI

<input type="checkbox"/> Imperial bars/PI.
<input checked="" type="checkbox"/> PI.
<input type="checkbox"/> Custom PI. 1
<input type="checkbox"/> Custom PI. 2
<input type="checkbox"/> Custom PI. 3

Save and Close

Don't save and Close

6.3.3 Units (General) and Units (Form)

In the submenus *Units (General)* and *Units (Form)* the user can manage units, format and decimal digits used for input data and read calculation result. The difference between the 2 menus is that *Units (General)* will load the General set of units, while *Units (Form)* will load the units actually used in the form that's being edit. Therefore is only a matter of the initial data that are shown as default; then, saving options are the same for the 2 commands.

The window that opens allows to set up, field by field (see images), the units to use, format automatic, fixed decimal or scientific) and digits.

The 'Metric' and 'Imperial' check buttons can be used to quickly change units.

	Units	Format	Decimal digits
Length:	mm	Automatic	3
Force:	kN	Automatic	4
Area:	mm ²	Automatic	4
Stress:	N/mm ²	Automatic	4
Moment:	kN*m	Automatic	4
Section Mod.:	cm ³	Scientific	2
Mmt Inertia:	cm ⁴	Automatic	4
J omega:	mm ⁶	Automatic	4

Set 'standard' units: ☐ Metric ☐ Imperial

Save and Close Apply to: this form and general options Don't save and Close

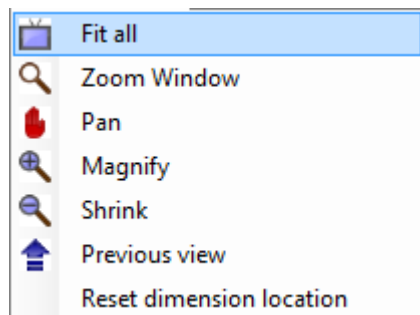
If new settings are saved, the choices in 'Apply to' set what will be changed (options are self explanatory).

It should be noted that **the user can have different units in each file** (and different from the general units which apply when a module is opened from scratch).


6.4 Drawing view menu

This menu is used to manage the display of the drawing in the graphic window.

The submenu *Fit all* is to fit all the drawing to the graphic window. *Zoom Window* is to select a new view of the drawing. *Pan* is to move drawing into the graphic window. *Magnify* is to enlarge drawing view. *Shrink* is to restrict drawing view. *Previous view* is to see previous view.




6.4.1 Fit all

Also available from the button bar clicking 


This zoom button is to fit all the drawing to the graphic window.

6.4.2 Zoom window

Also available from the button bar clicking 


This zoom button is to select a new view. First click on the screen with the left button of the mouse to select the top of the left point of the zoom window, then hold down the left mouse button, drag the mouse to the second point and release the left button of the mouse to complete the selection of the zoom window. A red rectangle will appear after the first click.

6.4.3 Pan

Also available from the button bar clicking 

This button is to pan the view in the graphic window.


6.4.4 Magnify

Also available from the button bar clicking 

This zoom button is to magnify drawing. The drawing is increased by a 1/0.85 factor.

If you have a mouse with scrolling wheel, click on the graphic window directly then scroll to apply *magnify/shrink*. When applying *magnify* with the mouse, the cursor location is kept still and the *magnify* factor applied relative to this position.


6.4.5 Shrink

Also available from the button bar clicking 

This zoom button is to shrink the drawing. A 0.85 factor is applied.

If you have a mouse with scrolling wheel, click on the graphic window directly then scroll to apply *magnify/shrink*. When applying *shrink* with the mouse, the cursor location is kept still and the *shrink* factor applied relative to this position.

6.4.6 Previous view


Also available from the button bar clicking 

This button is to go back to the previous view.

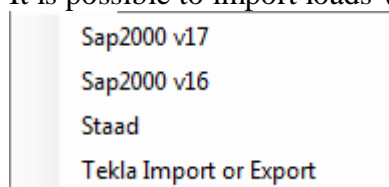
6.4.7 Reset dimension location

If the dimensions in the sketch have been moved around (by click and drag) they can be reset to the original position by clicking here.

6.5 Import menu

Also available from the button bar clicking 

It is possible to import loads visually selecting joints from the following programs:



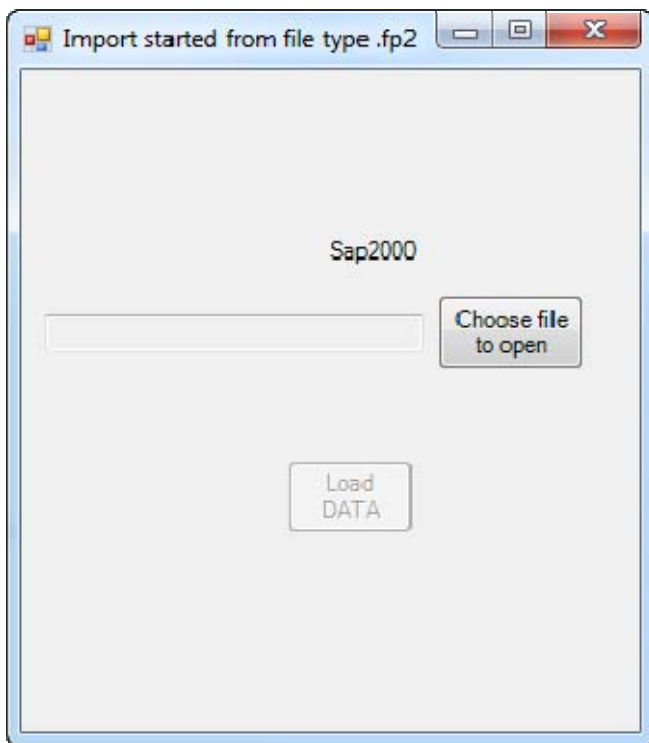
From Sap2000 and Staad the following data will be imported:

- Load combination values for the available load fields. Warning: if, say, weak axis moment is not available in the SCS module (example: FP2), no values will be imported even though calculation software results are different from zero.
- The primary and secondary member sizes; if the name in the Sap2000 or Staad is not available, a warning will be given. The import only checks the name, not the properties, therefore if Sap2000 or Staad properties are different from SCS properties, no message will be given. The user is supposed to check this.

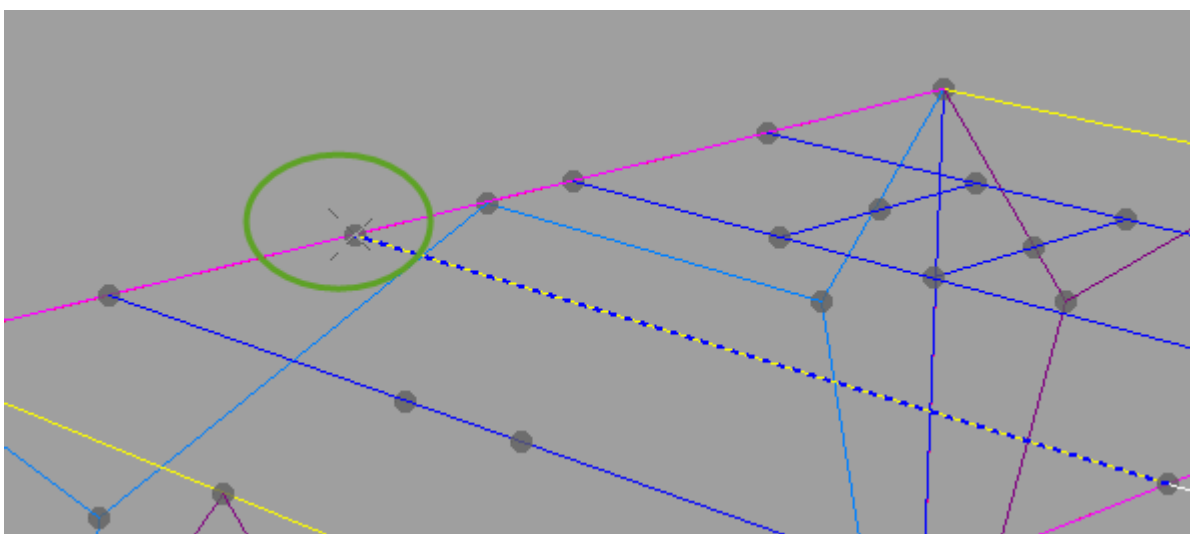
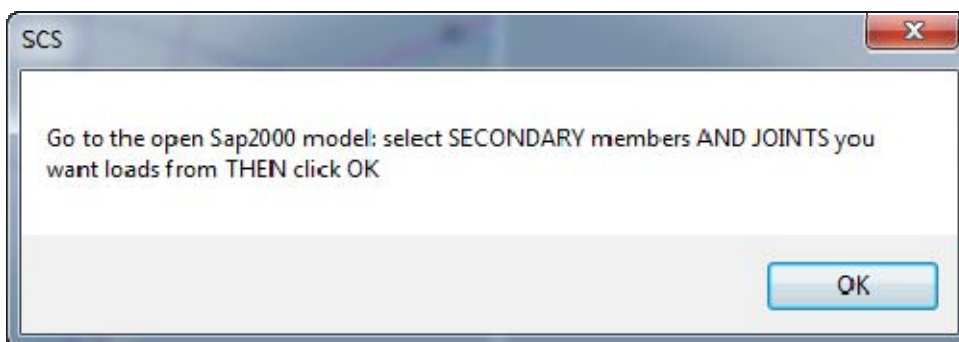
WARNING: The user must carefully check that the import is correct (units, loads, combinations, etc) because, due to some notes here reported and different versions of software (with maybe some particular settings unidentified when preparing the macro) and the fact two different entities are communicating through APIs, many things could possibly go wrong or in a way not fully understood by the user.

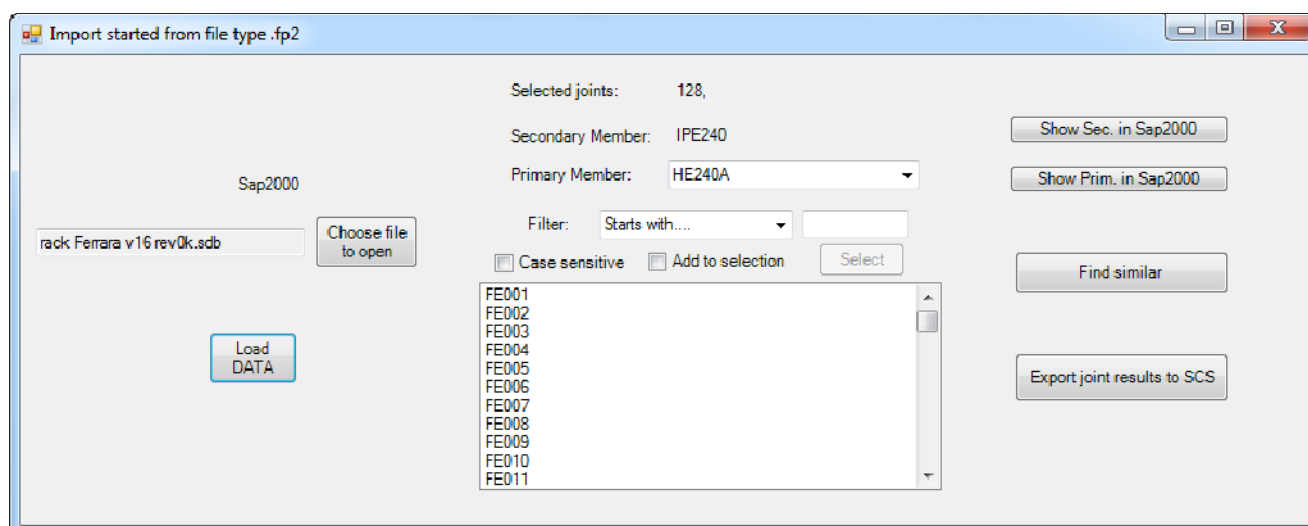
6.5.1 Sap2000

It starts with the following window



which allows to open up a file. Note: if another import is contemporarily opened in another module of SCS, the same file is referenced and the button 'Choose file to open' will be grayed out. The user is then prompted to choose at least one joint and the secondary member in this joint.



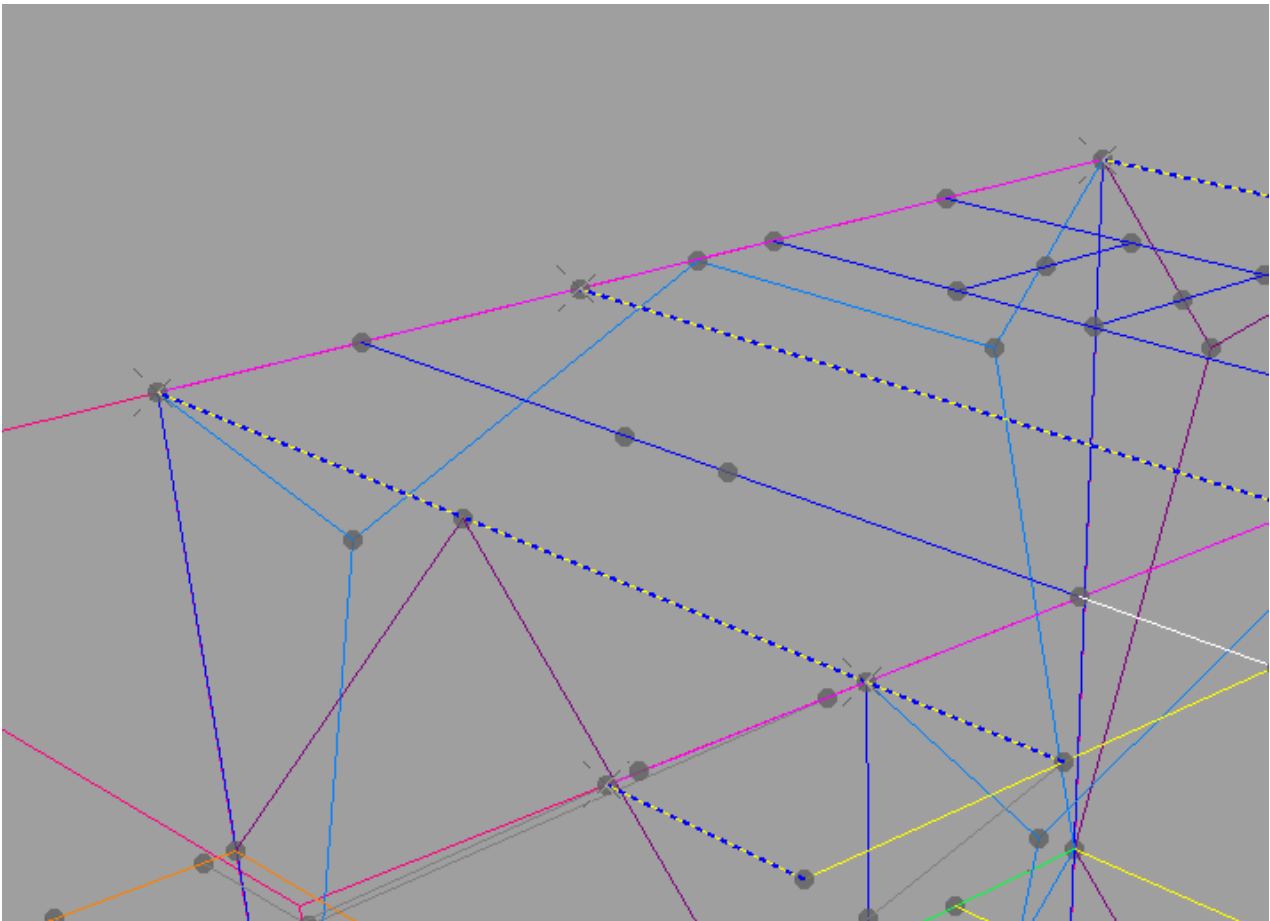


Once selection is made, SCS loads the secondary member property name (in a label since it's fixed), it guesses the primary member (according to the module where the import was started from; example: if it's a beam to beam connection, a beam will be looked for, if it's a column to beam connection, a column will be selected and so on). The primary member is in a combo box because SCS might not be correct in the selection and the user can change the selection.

It might then come useful hitting the button 'Find similar': SCS will look for other joints with the same secondary and primary member that are consistent with the module where the user is working with (as above, beam or column, or, say, the angle if it's a brace or consecutive members if it's a splice or apex).

SCS will also look for member orientation when making the selection in modules where the orientation is fixed (say, the moment connection) while it will not make any difference if the orientation of the column can be set by the user (example: modules fp1 or xl1). This also means that, in the latter case, the 'Find similar' button will choose joints independently on the fact they frame to the column on the weak or strong axis. The user can correct any behavior he/she doesn't like, manually modifying the selection (either unselecting some joints/members or, the opposite, adding some others).

After hitting the button 'Find similar', to effectively load the joint is necessary to re-click the button 'Load data'.



Import started from file type .fp2

Selected joints: 4, 6, 11, 12, 60, 127, 128.

Secondary Member: IPE240

Primary Member: HE240A

Sap2000

rack Ferrara v16 rev0k.sdb

Choose file to open

Load DATA

Filter: Starts with....

☐ Case sensitive ☐ Add to selection

Select

FE001
FE002
FE003
FE004
FE005
FE006
FE007
FE008
FE009
FE010
FE011

Show Sec. in Sap2000

Show Prim. in Sap2000

Find similar

Always visually check that selection is ok and complete

Export joint results to SCS

To finally export to SCS, some load cases must be selected (a special filter is provided to help with the task).

The import macro will automatically take care of the units, eventually changing them in SCS.

The user can then close the import form and finish up the design in SCS.

Please also note:

- if it's a shear connection and if the eccentricity is negative (which is likely the case because of the way eccentricity is considered in SCS), a negative moment will increase a positive shear, therefore the macro will automatically input in SCS the maximum moment in absolute value as negative, in order to be conservative;
- when an envelope combination or a response combination is chosen, the maximum values and the minimum values will be reported with two combinations (the only

exception being the moment, having the sign opposite with the shear as explained above); this might not be the worst possible combination since, say just as an example, the compression might combine with positive shear in a different way than tension, possibly in a ‘worse’ way; the engineer should take care of those situation, knowing that, when choosing an envelope, he/she is trading speed with precision (choosing single combinations instead of envelope will give back the precision, when needed)

- if more joints are selected, each value is taken with the maximum absolute value; as above, choosing more joints means trading speed with precision

As repeated already, every case could need special attention and the engineer must verify under his/her responsibility, liability and engineering judgement that imported loads are ok for design and correctly consider the possible worst situations and, on the opposite side, do not overdesign the structure.

6.5.2 *Staad*

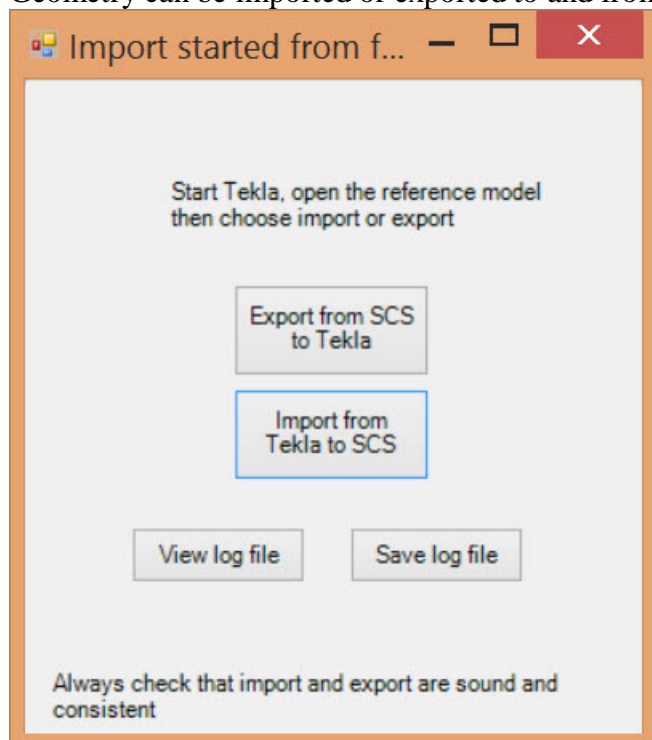
Very similar to Sap2000 above, please read all the considerations.

One difference is that the Staad model already open in the background will automatically be selected, it's not necessary to open the file from inside SCS.

See also the video available on www.steelconnectionstudio.com

6.5.3 *Tekla Structures*

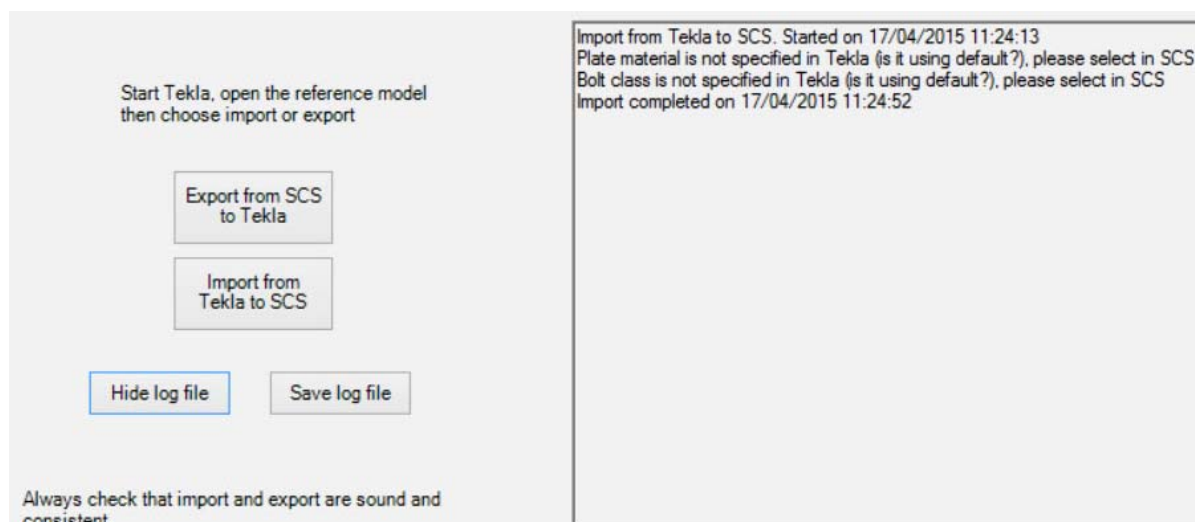
Geometry can be imported or exported to and from Tekla Structures.



For exporting, SCS requires to select in Tekla the primary and secondary member.

For importing, SCS requires the user to select a connection in Tekla.

All the warnings and messages are displayed in the dedicated window. The file can be saved as .txt.



Currently the following modules operate with Tekla Structures: **fp1, fp2, xl1, xl2, qs5, qs6, sp4, em3, we1, cb8**.

SCS can export connections to the following Tekla joints:

J131, J146, J149, J185, J29, J27, J101, J77, J141, J31, J14 and J1047.

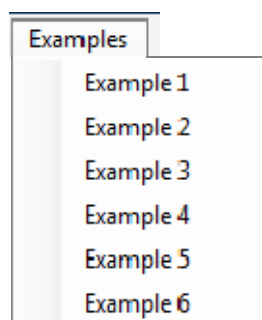
SCS can on the other hand import connections into SCS from the following Tekla joints:

J103, J131, J186, J187, J188, J146, J149, J185, J29, J144, J27, J101, J115, J310000032, J141, J77, J132, J42, J40, J102, J31, J1004, J1014, J1016, J1042, J1047 and J14.


To change Tekla Structure version, refers to the section 30.2

6.6 Examples menu

Each module has a different number of examples (usually around 6) that can be readily opened from this menu.




It's suggested to closely inspect all examples, module by module, because examples can show all the features that can be input and calculated by SCS.

Only in the demo version, examples can be accessed by the button .

6.7 Help menu

6.7.1 Contents

Also available from the button bar clicking .

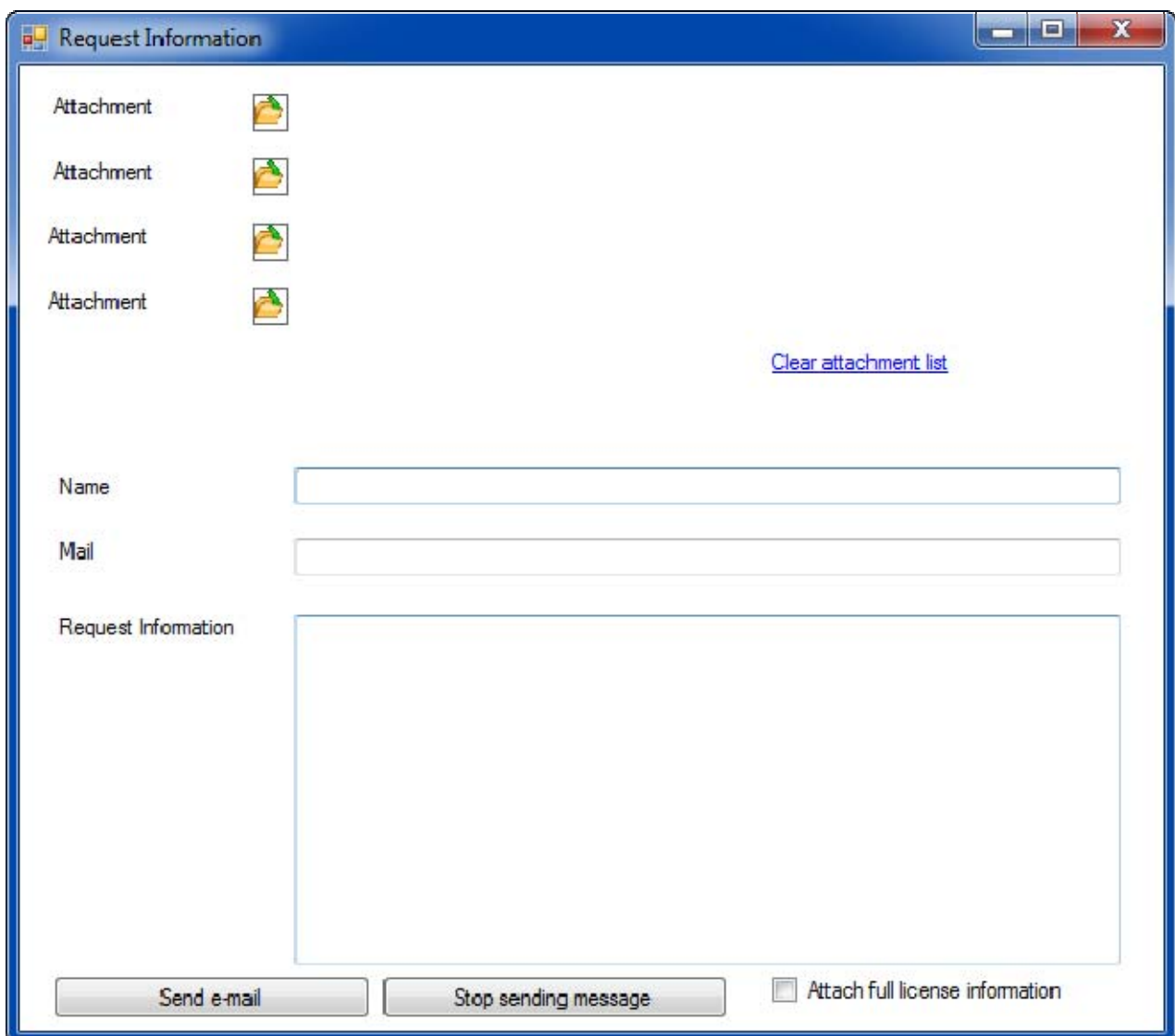
This button will connect the user to the 'Help manual'
Short cut key: ctrl + F1

6.7.2 About

SCS version is displayed when clicking this button.



6.7.3 Send request information

A screenshot of a "Request Information" dialog box. The dialog has a blue title bar with standard window controls. The main area contains a list of four "Attachment" items, each with a small icon of a document with a green leaf. To the right of this list is a link that says "Clear attachment list". Below the attachments, there are two text input fields labeled "Name" and "Mail". At the bottom, there is a large text area labeled "Request Information". At the very bottom of the dialog, there are two buttons: "Send e-mail" and "Stop sending message", and a checkbox labeled "Attach full license information".

[Click here](#) to send technical requests to our support team. The form is self-explanatory.

7 Data tab

7.1 General

7.1.1 Initial description

This connection is considered as a pin type connection, though limited bending moment can be resisted. All actions are transmitted by the angles bolted both to primary member and secondary member. Connection has two bolt groups: the first one is at the secondary member, number 2 is at the main member.

A quick general description is written for each module, with emphasis on how to consider connection (pinned or rigid), hence loading to be applied.

In order to give the user maximum flexibility and broad range of activity, even connections that are supposed to resist only as pins are calculated for applied bending moments. As per the EULA **the user should be enough experienced and qualified to understand limitations and apply loading as per his/her engineering sensibility**. The same considerations apply for weak axis loads (or torsion) that are better kept to zero in SCS and in the global analysis model (they should be resisted by other means like bracings, supports, diaphragms and so on).

7.1.2 Standard



You can choose among different international standards for designing connections: Eurocode, DIN 18800, AISC-LRFD, etc.

Each Standard requires specific license (buy at www.steelconnectionstudio.com)

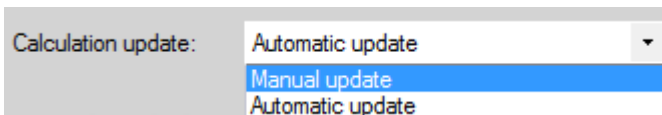
7.1.3 Language

You could choose among different languages for the name of the labels, the message text and the report: English, Italian, French, etc.

Each Language requires specific license (buy at www.steelconnectionstudio.com)



7.1.4 Calculation update

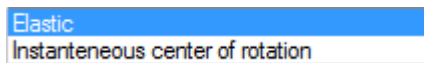


The user can choose between two type of calculation update: automatic or manual. If the user chooses automatic (it's default and it's suggested), the program will update calculation and results every time data are changed in the model. This can be greatly helpful in understanding the connection in real time and apply the specific corrections. Otherwise, if the manual update option is on, the user must click the *Update calcs* button to have a result update.

If the number of columns or rows of bolts in the connection is more than 10, the program automatically switches from automatic calculation to manual calculation, because the eccentricity

calculation could be burdensome (therefore the update could take some seconds) if the chosen option is *Instantaneous center of rotation*.

7.1.5 Eccentricity calculation method

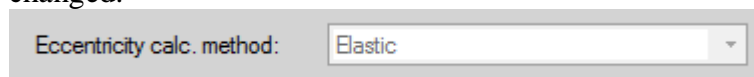


This eccentricity refers to a force applied in the plane of the bolts but not acting through the bolt group center of gravity.

Two eccentricity calculating methods can be chosen (for a complete description see chapter 7 of [26] in chapter *Standards and bibliography*) in most modules.

According to [26], "the Instantaneous center of rotation method is more accurate". It requires though an iterative procedure and its calculation can be cumbersome for the computer if there are many bolts (over 10 rows) and in special configurations (very high eccentricity).

Some modules (EM3, AP7, CB8 but also XL1, XL2) where emphasis is on bending moment (eccentricity is outside the bolt plane) apply automatically the *Elastic* option and this can't be changed.



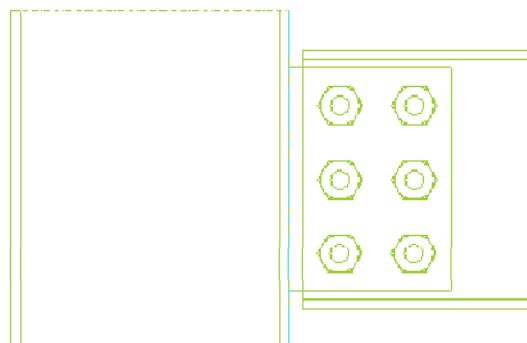
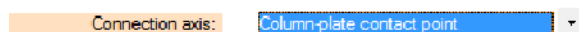
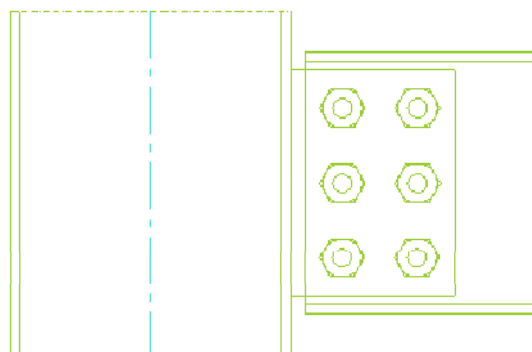
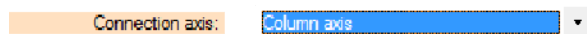
7.1.6 Connection axis

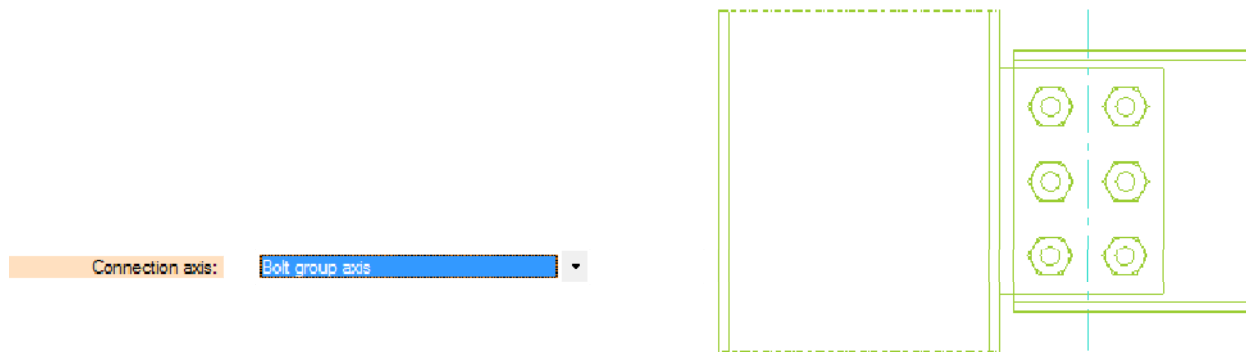
The position of the connection axis is an advanced feature of SCS to calculate different eccentricities for the parts composing the connection.

Positioning the vertical axis of the connection differently changes eccentricities of the secondary member, the primary member (this is actually not dealt with in SCS right now, since there's no global verification of the primary member, only local checks), bolt groups, welds, plates.

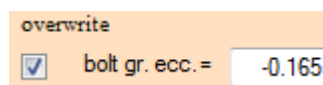
The connection axis is showed in the sketch by means of a dash-dot cyan line.

See some example positions below (from FP1):



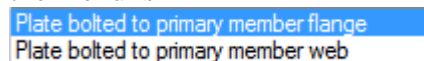


In addition to the 'standard' positions offered by the selection box (they might change according to modules), the position of the blue axis can be overwritten using the dedicated command (overwrite check box)

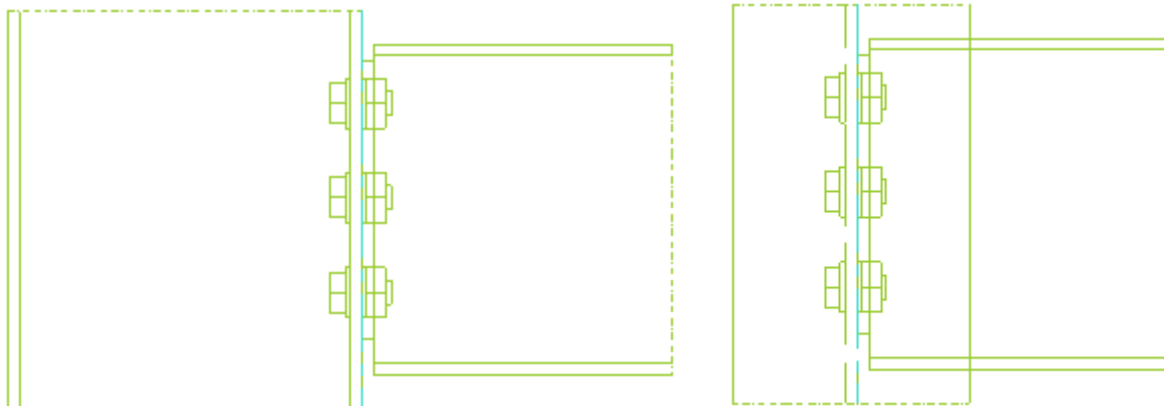


7.1.7 Connection type

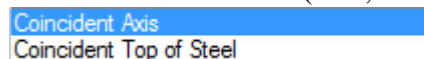
The modules with a column as primary member (essentially FP1, QS5, XL1, BR9), let the user choose between a connection to the column flange or column web. If XL1 is taken as a reference, the menu is



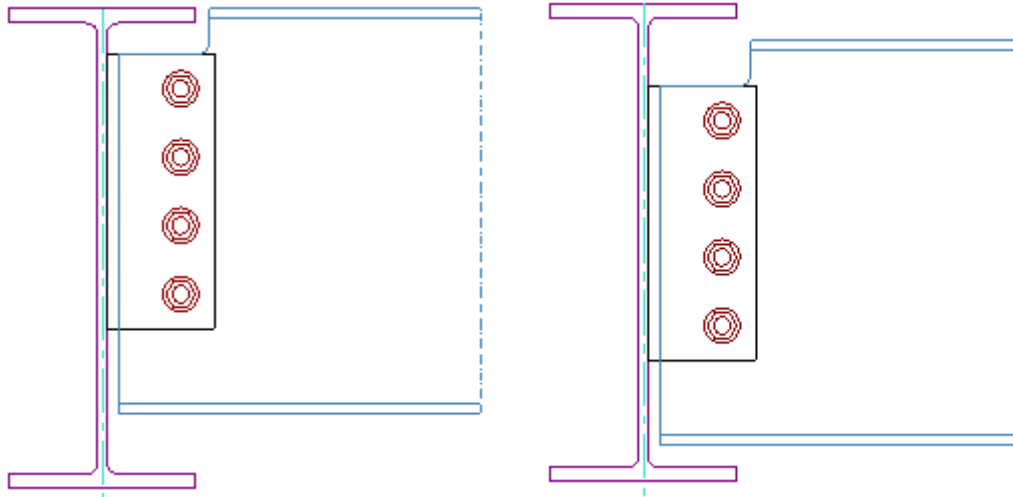
And the options respectively turn into those schemes



Modules beam - beam (FP2, DS1, QS6, XL2) have the following options available



If FP2 is the module, the connection becomes



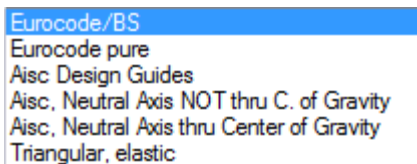
7.1.8 Category for shear

Through this choice the connections can be set to have bolts work in friction:

- 1) bearing type (shear by contact);
- 2) slip-resistant at serviceability limit state;
- 3) slip-resistant at ultimate limit state .

If the second option is chosen, the load table displays a checkbox where to let SCS know which cases should be consider as serviceability limit states.

7.1.9 Moment calculation method

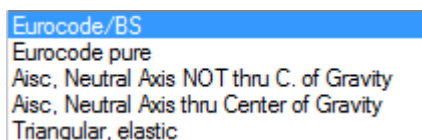


If the module has the bolt group resisting a bending moment (primarily EM3, AP7 but also XL1, XL2, QS5 and QS6 might have some moment on bolts, see further below for remarks) through tension in bolts (eccentricity out of bolt group plane), this option let the user choose the method to calculate forces on bolts (and plates).

- 1) Eurocode/BS: see [42] in bibliography
- 2) Eurocode pure: see [00]
- 3) AISC Design Guides: see [23], [24], [25]
- 4) AISC neutral axis not through center of gravity and through c. of g.: see chapter 7 of [26]
- 5) Triangular elastic: this method takes the center of compression at the compressed flange then calculates bolt tension linearly, creating a triangular distribution where maximum tension action is in the bolt row more distant from compression center. Pure tension action (coming from positive N, not from bending moment) is distributed among all bolts (not only to bolts in tension because of bending moment). Pure compression is not distributed to bolts.

The most sophisticated and advanced methods (therefore suggested to be used) are supposed to be the first three (EC and AISC D.G.).

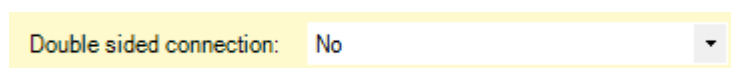
The menu doesn't include the third option (AISC Design Guides, see figure below) in modules like XL1, XL2, QS5 and QS6 because this method is not applicable in those modules.



The program doesn't block the user use to use, for example, Eurocode as standard but a method by AISC to calculate moment forces even though this is probably not suggested. As per the EULA, the user is supposed to take all kinds of engineering decisions.

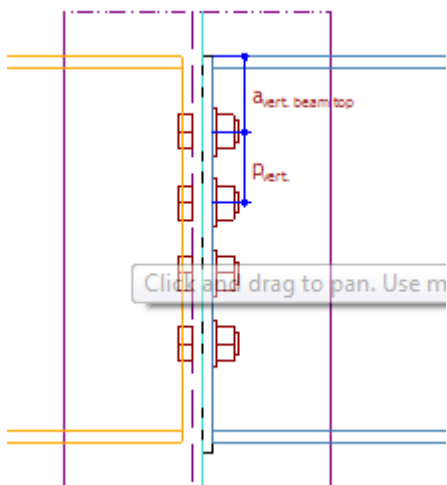
7.1.10 Double sided connection

Depending on the joint configuration, the following option might display

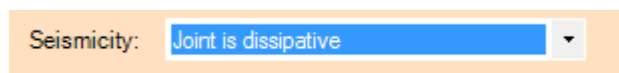


If this option is set to yes, some local limit states check might vary. For example, the primary local web shear might double, while, on the other hand, the primary web tie force might go to zero in a fin plate because the axial force is supposed to go through the web to the other fin plate on the opposite side.

The double sided configuration is shown in the sketch with a mirrored yellow secondary beam.

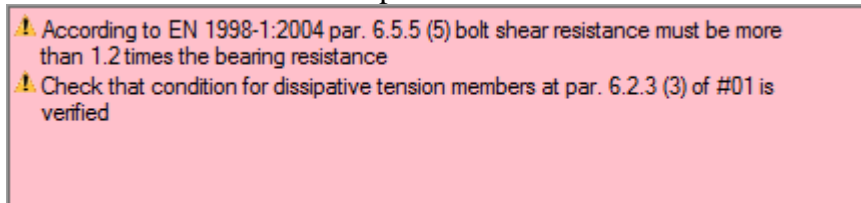


7.1.11 Seismicity: joint is dissipative



Some standards (currently Eurocode and Italian NTC) have this option.

When it's set to 'dissipative', SCS will make two further checks (see picture below) and, if they are not satisfied, a warning will display. If they are ok, no message will be shown. The reference of the two checks is below reported:



7.2 Loads

7.2.1 Cases

Case	Pu	Vu maj	Vu min	Mu maj	Mu min	Tu	Serv.
a							<input type="checkbox"/>
b							<input type="checkbox"/>
c							<input type="checkbox"/>
d							<input type="checkbox"/>
e							<input type="checkbox"/>

AISC LRFD
symbols

Unit: kN, kN*m

Case	N Ed	Vmaj Ed	Vmin Ed	Mmaj Ed	Mmin Ed	T Ed	SLS
a							<input type="checkbox"/>
b							<input type="checkbox"/>
c							<input type="checkbox"/>
d							<input type="checkbox"/>
e							<input type="checkbox"/>

Eurocode symbols

Up to five load combinations (identified by SCS with a letter from 'a' to 'e', see *Case* column) can be inserted.

The figure above (from EM3) has a full range of forces/moments acting on the connection but a reduced number of forces/moments is usually present in the various modules. Symbols of forces/moments:

- first column (N in Eurocode, P in AISC) is for axial forces. Positive force means tension, negative means compression
- V is shear; 'maj' stands for shear through major axis of the secondary member, 'min' through minor axis
- M is as above but for bending moment
- T is torsion

Design loads (factored if required by the chosen standard) should be inserted.

The units are recalled above the table and the user can change these units through the *Units buttons* in the menu *Tools* of the toolbar.

7.2.1.1 Formulas

The load in each cell can be inserted as a formula. Example, instead of 640, the user can input: $=320*2+4-10+6$

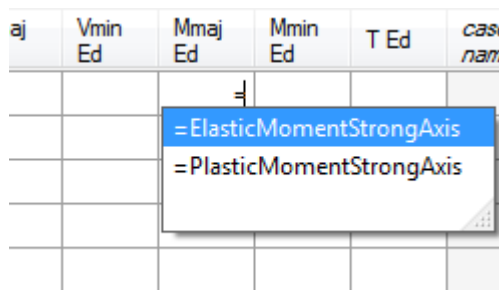
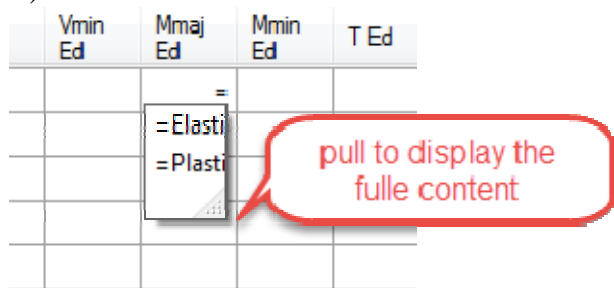
Only the result will be displayed when the user exits the cell but the formula will be kept in memory and saved with the file.

7.2.1.2 Secondary member strength reference

After the user enters in a cell the '=' symbol, a reference to the strength will be displayed. Example, when in the strong axis moment, the elastic and plastic strength reference option will be given.

The user can choose the reference and then multiply it by the necessary factor.

Please note that the strength value will be updated if the sec. member is changed (or the material is).



This can be useful in the following two situations:

- Design must be done for a fixed percentage of the capacity. Say for example 60% of shear. It's enough to choose the strength and then add '*0.6'
- Design must be done for seismic reason for full capacity eventually amplified. It's enough, as above, to multiply by the desired overstrength factor, for example: '=ElasticMomentStrongAxis*1.1*1.1'

7.2.2 γ_M

	overwrite	
γ_{M0}	1.1	<input type="checkbox"/>
γ_{M1}	1.1	<input type="checkbox"/>
γ_{M2}	1.25	<input type="checkbox"/>
$\gamma_{M3,ser}$	1.1	<input type="checkbox"/>
γ_{M7}	1.1	<input type="checkbox"/>

If Eurocode (or similar, NTC Italy for example) is the selected standard, partial safety factors γ_M are here shown. It is possible to adjust values overwriting suggested EC values.

Some factors only appear if the bolts resist by friction.

Din value is only one (below)

γ_M	1.1	<input type="checkbox"/>
------------	-----	--------------------------

7.2.3 Optimize Bolt Tension + Shear combo

☒ Optimize Bolt Tension+Shear combo

This option is only active in EM3 when the *eccentricity calculation method* is active and one of the first 2 options (Eurocode) is chosen.

Choosing the option will distribute Shear not evenly to all bolts (which happens when the checkbox is unchecked) but will give more shear to bolts working with lower tension (or no tension at all).

7.3 Members

7.3.1 Select member

Primary Member: IPE300

depth	h =	300 mm
width	b =	150 mm
flange thickness	t_f =	10.7 mm
web thickness	t_w =	7.1 mm
toe (fillet) radius	r =	15 mm

You can choose the members to insert as primary or secondary members from the profile database.

To filter groups and reduce the list of members, see 6.3.2 *Member filter*.

To edit members, see 6.2.3 *Edit*

The geometrical data of the selected profile are shown in the labels below the selection box.

The symbols are consistent with the selected standard.

Some members (usually columns) have additional properties, depending on the module.

Some notes could be visible. For example 'No strength check is made for members' means that no general check is made for member (usually column) strength. This doesn't mean that applicable local checks requested by the connection (just as a general example, web resistance checks) are not addressed. If a strength check is needed, the user could use module ME0.

7.3.1.1 Web doubler plate

Primary Member: W16X31 Material: A992

d =	16 in	F_y =	50 ksi
b_f =	5.5 in	F_u =	65 ksi
t_f =	0.44 in		
t_w =	0.28 in		
$k - b_f$ =	0.4 in		
b_1 =	4 in		

☒ Both sides

Reinf. web pl. t_s = 0 in

No strength check is made for members.

A (welded) reinforcing web plate can be specified if available.

The check box Both sides means that the web plate is welded to both sides.

7.3.1.2 Column length over connection

Primary Member: W16X31 Material: A992

d = 16 in $F_y = 50$ ksi

$b_f = 5.5$ in $F_u = 65$ ksi

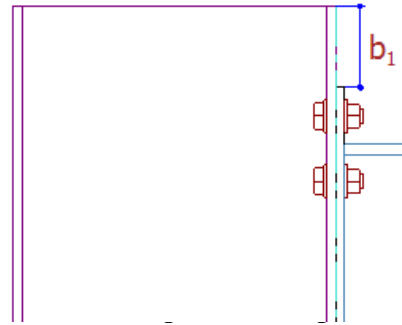
$t_f = 0.44$ in $thk\ c = 40$ in

$t_w = 0.28$ in ☐ Both sides

k - b = 0.4 in Reinf. web of t = 0 in

$b_1 = 4$ in ☐ Continuous prim.

No strength check is made for moment.



This option is related to the length of the column over the connection. This option is available when this length becomes important to evaluate some limit states. Checking the *Continuous prim.* option will give an infinite value to b_1 .

7.3.2 Section properties button

Primary Member: UB762x267x197

h = 769 mm

b = 268 mm

$t_f = 25$ mm

$t_w = 16$ mm

r = 16 mm

If the small button highlighted above is clicked, a window with the properties of the member is displayed.

Properties

UB762x267x197

A gross =	2.512E+04	mm ²
A gross shear =	1.292E+04	mm ²
I strong =	2.378E+09	mm ⁴
I weak =	8.048E+07	mm ⁴
r strong =	307.7	mm
r weak =	56.6	mm
Wel S strong =	6.186E+06	mm ³
Wel S weak =	6.006E+05	mm ³
Wpl Z strong =	7.053E+06	mm ³
Wpl Z weak =	9.438E+05	mm ³
Jt =	3.807E+06	mm ⁴
Jw =	1.11E+13	mm ⁶
Sw =	8.35E+07	mm ⁴

Close

The properties include the presence of haunches or notches.

Plastic modulus is calculated without considering toe radius.

When haunches are present, Inertia, Plastic modulus and Elastic modulus are calculated without considering the toe radius.

The window can stay open and the user can keep working. Many windows with the properties can be kept open simultaneously.

7.3.2.1 Section properties validation

See par. 26.7 SNIp validation.

7.3.3 Select material

Similarly to *Select member* option, the material can be selected for the members.

To edit members, see 6.2.3 *Edit*

To filter materials and reduce its list, use the Visible checkbox in 6.2.3 *Edit* the but also activate the real usage of this filter, see 6.3.1 *General options*.

The labels below the material show yield strength and ultimate strength used for calculations according to member maximum thickness. For example, the member in the figure above has a maximum thickness of 24mm, therefore yield strength applied is the one in the range $>16\text{mm}$ and $\leq 40\text{mm}$ as the database show (image below). The third label below the material (thk \leq) shows this higher applicable limit.

Material	E [N/mm ²]	f _{y1} [N/mm ²]	f _{u1} [N/mm ²]	lim1 [mm]	f _{y2} [N/mm ²]	f _{u2} [N/mm ²]	lim2 [mm]
S235	2.1E+05	235	360	16	225	360	40
S275	2.1E+05	275	410	16	265	410	40

7.3.4 Other options

If the module includes a verification of the secondary member (usually included among limit states when the member could be weakened by notches) and the selected standard is Eurocode (or similar), the user have tools to define exactly the class of the member (below). Class can also be overwritten.

7.4 Plate & bolts

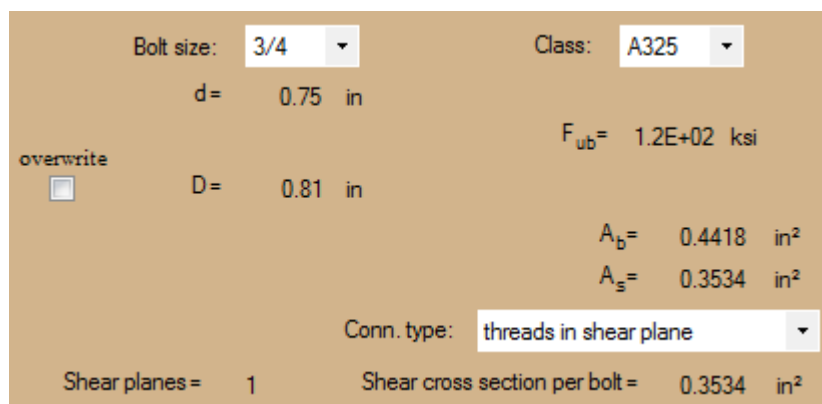
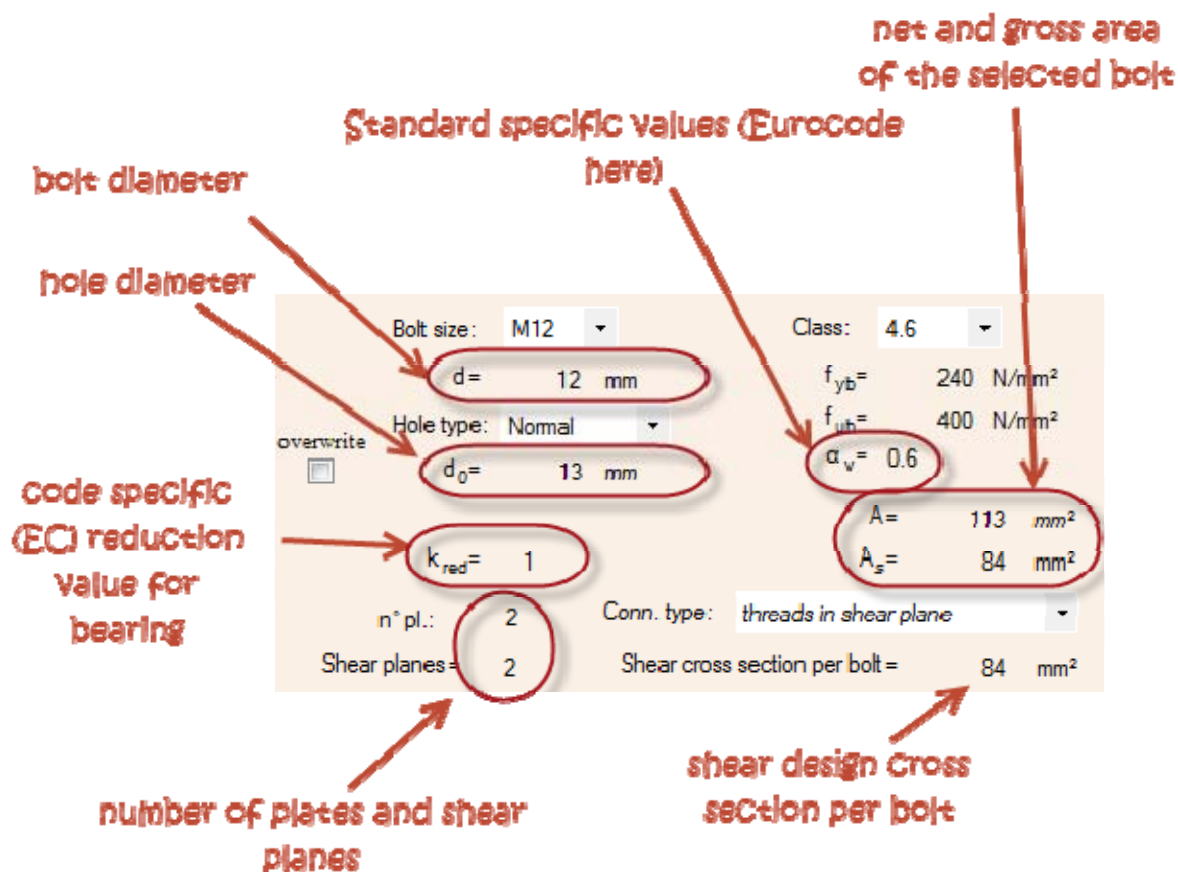
7.4.1 Bolts

Bolt size and bolt class can be chosen from the database.

To edit the database, see 6.2.3 *Edit*

To filter bolt sizes and classes to reduce the list of available sizes/classes, use the Visible checkbox in 6.2.3 *Edit* the but also activate the real usage of this filter, see 6.3.1 *General options*.

Images below (Eurocode in QS5 and AISC LRFD in AP7) should be self explanatory.



The ones below are the default values used for bolts, washers and nuts used in SCS drawing:

For bolts: head depth = $0.65 \cdot D_{\text{nom.bolt}}$

$$s = 1.60 * D_{\text{nom.bolt}}$$

bolt shank length = +5mm (=0.2") over the nut

For nuts: depth = $0.80 * D_{\text{nom.bolt}}$

$$s = 1.60 * D_{\text{nom.bolt}}$$

number of nuts each bolt = 1

For washers: depth = $0.20 * D_{\text{nom.bolt}}$

$$D_{\text{est}} = 2.1 * D_{\text{nom.bolt}}$$

n° of washer for each bolt = 1

The options in blue can be edited, see 6.3.1 *General options*.

See 4.1 *Basics*.



7.4.2 Plate material

Plate material:	St37-2	
$f_{yp} =$	235	N/mm ²
$f_{up} =$	360	N/mm ²
thk ≤:	40	mm

See 7.3.3 *Select material*. Same considerations apply.

7.5 Geometry

7.5.1 Bolt options

bolt columns =	1	bolt rows =	2	total bolts =	2
plate thk =	6	mm	cl. beam =	10	mm
<input checked="" type="checkbox"/> Plate 'standard' bolt spacing			beam clear. from flanges =	10	mm
$P_{\text{horiz.}} =$	54	mm	<input checked="" type="checkbox"/> Bolts are vertic. sym. in second.		
$P_{\text{vert.}} =$	54	mm	$a_{\text{vert. beam top}} =$	49	mm
$a_{\text{horiz. pl.}} =$	36	mm	$a_{\text{vert. beam bottom}} =$	49	mm
$a_{\text{horiz. beam}} =$	36	mm			
depth _{pl.} =	108	mm	<input checked="" type="checkbox"/> Bolts are vert. sym. in plate		
width _{pl.} =	82	mm	$a_{\text{vert. pl. top}} =$	27	mm
			$a_{\text{vert. pl. bottom}} =$	27	mm

Here you can set up the geometry data in the connection. You can choose the number of column and rows of the bolts.

If the user chooses to apply *Plate "standard" bolt spacing*, the software set bolt spacing, and updates them automatically if the user changes the bolts diameter. Unless differently specified in the modules, the 'standard' bolt spacing is:

- 3 times the bolt hole diameter for horizontal and vertical spacing between bolts
- 2 times the bolt hole diameter for horizontal spacing from edges (usually plate on one side, beam on the other)
- 1.5 times the bolt hole diameter for vertical spacing from applicable borders.

If the option is unchecked, every single value is defined by the user.

Note that if bolt columns are 1 (same thing would happen if rows is 1) as in the figure above, the horizontal spacing is grayed because it's actually unnecessary (there are not consecutive bolts so there's no distance to be defined).

With *Bolts are vertical sym. in plate* unchecked, the user can set the vertical position of bolts from the top and bottom edges of the plate. If it's checked, SCS will automatically take care of distances considering the connection symmetric.

Same considerations apply to *Bolts are vertical sym. in secondary*.

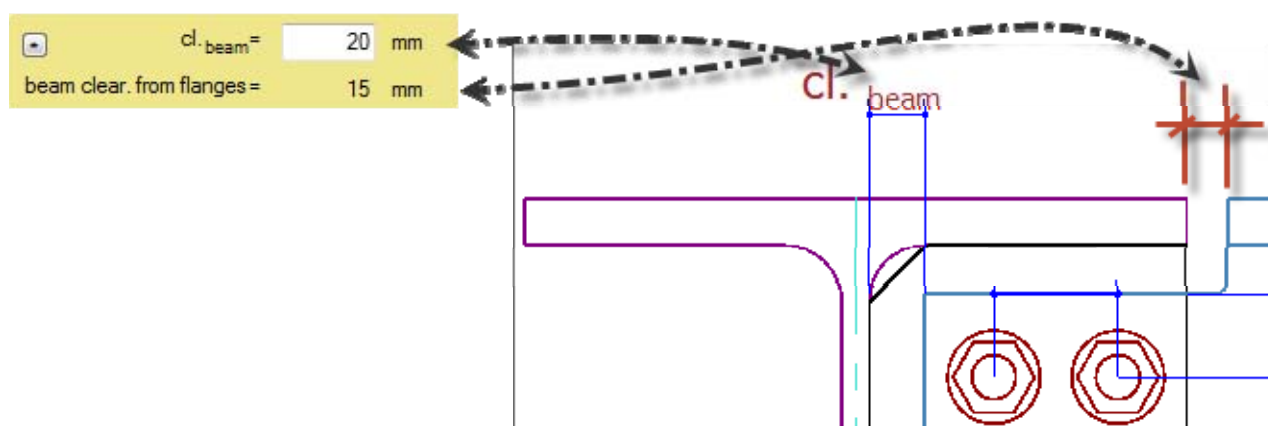
You can see the graphical representation of the input data in the *Graphic window* if you select *symbols* in the *drop-down menu* above the *Results space*.

7.5.2 Plate options

The user can define the thickness, depth and width of the plate.

Through the position of the bolts seen in previous paragraph, the user can set the vertical position of plate from the secondary member top of steel.

7.5.3 Beam clearance



The distance between the primary member and secondary member of the connection can be specified through *cl.beam*. See figure for graphical representation of *beam clearance from flanges*.

7.5.4 Eccentricity of bolt group, plate and secondary member

overwrite		
<input type="checkbox"/>	bolt gr. ecc. =	-1.8 in
<input type="checkbox"/>	pl. ecc. =	-1.8 in
<input type="checkbox"/>	sec. ecc. =	-3.2 in

SCS assigns eccentricity to bolt group(s), plate(s), weld(s) and secondary member according to connection axis location (see par. 7.1.5 *Eccentricity calculation method*)

Sign convention: if connection axis is on the left of the critical section, the eccentricity is negative, that is a positive (downward) shear will increase negative moment and decrease positive moment.

It's the opposite situation if connection axis is on the right of the critical section.

For the plate eccentricity, two critical sections are considered:

- 1) the section near the primary member (usually where plate is welded to primary member)
- 2) the section at the centerline of bolts.

The maximum eccentricity in absolute value is the one chosen.

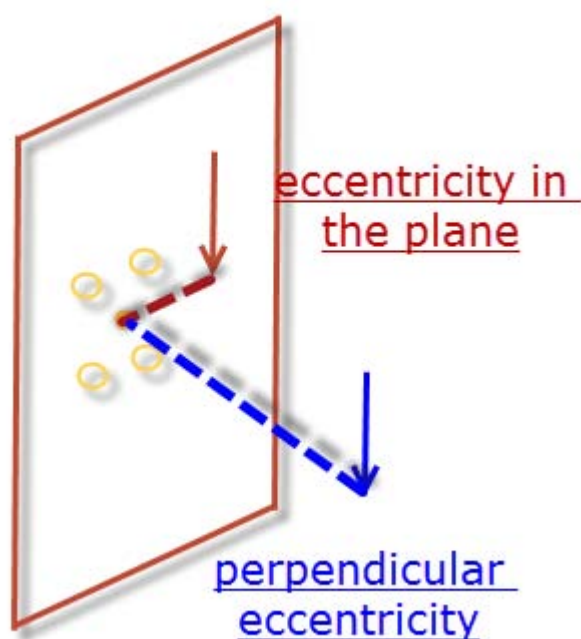
With the checkboxes on the left, it's possible to overwrite any value to calculate the connection as per user's engineering judgment.

7.5.4.1 Bolt group eccentricity

In modules like em3, ap7, xl1 and xl2, bolt group eccentricity refers to perpendicular eccentricity and this is shown by symbol ' \perp '

bolt gr. \perp ecc.

In other word, the \perp (perpendicular) eccentricity is the distance from the axis to the plane where bolts are while normal eccentricity as used in FP1 and other modules is the distance, in the plane of bolts, between axis and bolt group center of gravity.



In module cb8 (base plate) the bolt group eccentricity is referred to the eccentricity to the single anchor bolts that will generate, with shear, a local bending moment on the anchors.

7.5.5 Plate buckling length

☐ $L_{cr} = 1.8$ in

☒ $L_{cr} = 2$ in

This value is used as buckling length of the plate. SCS calculates it automatically (see formula) as minimum of distance between bolts and some other data specific to modules. The user can overwrite the value with anything else. If it's input zero, no buckling is considered for the plate.

7.5.6 Block shear $k_{unif}=1$

☒ block shear $k_{unif}=1$

Checking this option will force k_{unif} value in Eurocode or U_{bs} in AISC to be taken as 1 (all cases).

When there are eccentricities, SCS correctly takes this coefficient as 0.5. In some cases though, those eccentricities are negligible and, using this option, the engineer can decide not to consider eccentricity.

If the option is not checked, SCS will calculate automatically which factor should be applied, case by case.

7.6 Weld, notch & stiffeners

7.6.1 Weld

Eurocode example

code specific data

$f_{u,weld} = 360 \text{ N/mm}^2$

Web weld: double fillet

fillet number: 2

$\beta_w = 0.8$

$a_{web} = 5 \text{ mm}$

$l_{web} = 100 \text{ mm}$

throat thickness

weld length

Weld design data can be set up in this area. Options are quite self explanatory.

Weld set up might be required for more welds.

When the cursor is inside the weld throat textbox, a suggested value pops up in the message window, giving a "Weld possible suggested try value". This is given as per [10] par. 8.4.1.4 (833) multiplying thickness of plate by the multiplier below (double fillet weld assumption), according to material yield:

f_y	Multiple of thickness
≤ 235	0.5
≤ 275	0.6
≤ 355	0.7
≤ 450	0.8
> 450	0.9

7.6.2 Weld eccentricity

overwrite ☐

$e = 0 \text{ mm}$

Same considerations as in par. 7.5.4 *Eccentricity of bolt group, plate and secondary member* apply here.

7.6.3 Do not consider compression for welds

☐ Do not consider compression for welds

Checking this option will make compression transferred by contact, not by the weld. If it's unchecked (default), the weld will also work in compression, neglecting contact of parts.

7.6.4 Stiffeners

Stiffeners on primary member:
No calculation is made on the stiffeners.
'Single stiffener on the back side' option is only available when connection is on column web.

☐ Single stiffener on the back side

Stiffeners on primary member:
No calculation is made on the stiffeners, they will only be added to the sketch.

☒ Single stiffener on the back side $t_{stiff.} =$ mm

In some type of connections, it is possible to add stiffeners on primary member. No calculations is made on the stiffener, they are only added to the sketch.

7.6.5 Notch

Notch configuration:

Notch: Top and bottom notch

☐ Different depth for bottom notch:

$l_n =$ mm
 $h_n =$ mm
 $r_n =$ mm

notch length
notch height (depth)
radius

Flange notch: No flange notch

No flange notch
One side notch
Two side fl. notch

The user can select notch configuration for secondary member. Notch can be applied to the member (full flange and part of web) or to flanges only (one or two sides can be notched). A combination of the different kinds of notches is not possible. If a double notch is chosen (top and bottom), a different depth is possible to be applied to the notches. If this option is taken, a new box where to input bottom depth will be visible.

Notch: Top and bottom notch

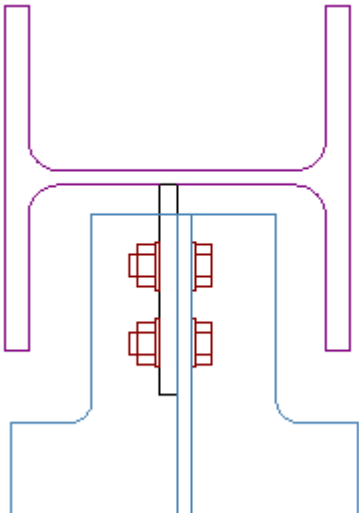
☒ Different depth for bottom notch: in

$l_n =$ in
 $h_n =$ in
 $r_n =$ in

A two side flange notch example is shown below.

Flange notch: Two side fl. notch

$l_{f,n} =$ mm
 $h_{f,n} =$ mm
 $r_{f,n} =$ mm



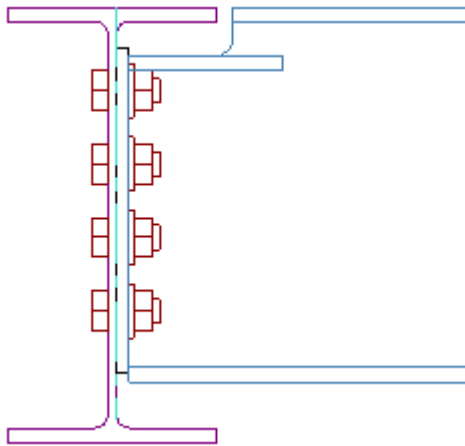
7.6.6 *False flanges*

If the beam is notched, the option of adding false flanges turns on.

☒ Add false flanges

False flanges are added where notches are (therefore, if both notches are present), false flanges add on top and on bottom.

To avoid the input of many unwanted data, the false flange is considered by default as wide as the beam and with the same thickness of flanges.



8 Result tab

8.1.1 Formulas

To the right of the numeric result and its unit of measurement there is the formula used to obtain it, which also contains the reference from which it was taken. The reference number into the brackets is referred to the position of the formula in the standard, and the number after the symbol # is the own reference of SCS to the standard as shown in chapter *Standards and bibliography*. If you move the mouse over the word *formula*, the equation appears.

$M_{ecc,Ed,bolt\ gr.}$	=	2	kN*m	<i>formula</i>
β_{Lf}	=	0.971		$1 - \frac{L_j - 1.5d}{200d}$; $0,75 \leq \beta_{Lf} \leq 1,0$ (3.5) #00
$F_{v,Rd,1\ bolt}$	=	94.08	kN	<i>formula</i>
$F_{v,Rd,bolt\ gr.}$	=	913.8	kN	<i>formula</i>

Above you can show the formula of β_{Lf} , and below you can see the formula for the next data and the word *formula* on the right of β_{Lf} .

$M_{ecc,Ed,bolt\ gr.}$	=	2	kN*m	<i>formula</i>
β_{Lf}	=	0.971		<i>formula</i>
$F_{v,Rd,1\ bolt}$	=	94.08	kN	$\frac{\alpha_v f_{ub} A_{ns h.pl.}}{\gamma_{M2}}$ (ta. 3.4) #00
$F_{v,Rd,bolt\ gr.}$	=	913.8	kN	<i>formula</i>

It is possible to see only one formula at a time.

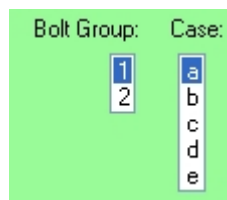
In SCS there are some references to formulas with an asterisk (example. 31*), this meaning that the formula is not in standard in that form but it is derived by modifying in some way the original formula present in the standard (31).

8.1.2 Case selection

Where the bolt groups are more than one, with *bolt group button* you can change the bolt group and visualize the numerical values of the checks related to the selected bolt group.

For example, in the splice connection the bolt group 1 refer to the bolts on flanges, while bolt group 2 refers to the bolts on the web.

With *case button* you can change the load case and visualize the numerical values of the tests related to the selected load case.



Shear:				Case
$F_{v,Ed,hor,bolt\ gr.}$	=	9	kN	<i>formula</i>
$F_{v,Ed,vert,bolt\ gr.}$	=	100	kN	<i>formula</i>
$M_{Ed,bolt\ gr.}$	=	1	kN*m	<i>formula</i>
$M_{ecc,Ed,bolt\ gr.}$	=	1	kN*m	<i>formula</i>
β_{1f}	=	0.998		<i>formula</i>

8.2 Sum up

Here are displayed the results of checks performed by SCS. Check by check, the maximum exploitation rate is shown. In the parentheses following the percentage, the case giving the reported exploitation (again, the worst one) is indicated.

Bolt shear:	ok 66% (d)		
Bolt min. distances:	ok		
Pl. bearing hor.:	ok 43% (d)	Beam bearing hor.:	ok 78% (d)
Pl. bearing vert.:	ok 28% (d)	Beam bearing vert.:	ok 34% (d)
Pl., bl. shear hor.:	ok 9% (b)	Sec., bl. sh. hor.:	ok 12% (b)
Pl., bl. sh. vert.:	ok 15% (a)	Sec., bl. sh. vert.:	ok 12% (a)
Pl., shear strong ax.:	ok 22% (a)	Pl., bend. strong ax.:	ERR 1.02 (d)
Pl., shear weak ax.:	ok 1% (a)	Pl., bending weak ax.:	ok 22% (a)
Pl., axial:	ok 6% (b)	Pl., combined stress:	ERR 1.03 (d)
Beam shear str.:	ok 19% (a)	Beam bending str.:	ok 17% (d)
Beam shear wk.:	ok 0% (a)	Beam bending wk.:	ok 2% (a)
Beam axial:	ok 3% (b)	Beam combined:	ok 18% (d)
Plate buckling:	ERR 1.02 (d)	Prim. local web shear:	ok 0% (a)
Web weld combined:	ok 94% (d)	Web weld σ_{\perp} :	ok 62% (d)

If the check is not satisfied, instead of the 'ok', a bold warning 'ERR' is displayed. Other errors might appear as below

Bolt group 1:			
Bolt shear:	ok 87% (b)		
Bolt min. distances:	ERR: check distances		
Pl. bearing hor.:	ok 48% (a)	Beam bearing hor.:	ok 60% (a)
Pl. bearing vert.:	ok 83% (b)	Beam bearing vert.:	ok 92% (b)
Pl., bl. shear hor.:	ok 34% (a)	Sec., bl. sh. hor.:	ok 64% (a)
Pl., bl. sh. vert.:	ok 3% (b)	Sec., bl. sh. vert.:	ok 5% (b)
Pl., shear strong ax.:	ok 4% (b)	Pl., bend. strong ax.:	ok 33% (b)
Pl., shear weak ax.:	ok 0% (a)	Pl., bending weak ax.:	ok 0% (a)
Pl., axial:	ok 73% (a)	Pl., combined stress:	ok 93% (b)
Beam shear str.:	ok 7% (b)	Beam bending str.:	ok 28% (b)
Beam shear wk.:	ok 0% (a)	Beam bending wk.:	ok 0% (a)
Beam axial:	ok 33% (a)	Beam combined:	ok 56% (b)
Plate buckling:	ok 33% (b)	Prim. local web shear:	ok 0% (a)
Web weld combined:	ok 66% (a)	Web weld σ_{\perp} :	ok 45% (a)

It is important to understand that this is just a quick sum up that could be useful but all the checks here reported have details in the results tab

8.3 Bolts

SCS gives in this tab the details of bolt shear, bolt tension and their combination for every bolt group of the connection.

For each load case and bolt group, SCS shows values and formulas used in calculations.

At the bottom of the page, the forces for each bolts in the selected load case are shown according to 3 dimensional components Rx, Ry and Rz.

Shear:				type of check	Case		
$F_{v,Ed,hor,bolt\ gr.}$	=	0.2248	kips formula	data and formulas used for the check	a		
$F_{v,Ed,vert,bolt\ gr.}$	=	22.48	kips formula		b		
$M_{Ed,bolt\ gr.}$	=	0	kips*ft formula		c		
$M_{ecc,Ed,bolt\ gr.}$	=	0	kips*ft formula		d		
β_L	=	1	formula		e		
$F_{v,Rd,1\ bolt}$	=	21.15	kips formula				
$F_{v,Rd,bolt\ gr.}$	=	126.9	kips formula				
$F_{v,Ed,bolt\ gr.}$	=	22.48	kips formula				
Bolt shear:				ok 18%	formula	check shows into the sum up	
Tension:				type of check			
$N_{Ed,bolt\ gr.}$	=	2.248	kips formula	<input type="button" value="Draw in-plane forces"/> <input type="button" value="Draw Sketch"/> <input type="button" value="T-Stub notes"/>			
$M_{\perp Ed,bolt\ gr.}$	=	2.213	kips*ft formula				
$N_{Rd,1\ bolt}$	=	31.73	kips formula				
$N_{Rd,bolt\ gr.}$	=	45.51	kips formula				
$N_{totEd,bolt\ gr.}$	=	16.81	kips formula				
Bolt tension:				ok 37%	formula	check shows into the sum up	
Bolt combined:				ok 29%	Bolt 2		
$M_{\perp Rd,bolt\ gr.}$				=	6.687	kips*ft formula	
Moment cap.:				ok 33%	formula	check shows into the sum up	
forces per bolt							
kips	Bolt:	Rx:	Ry:	Rz:			
	1	-0.03748	-3.747	5.192			
	2	-0.03748	-3.747	5.192			
	3	-0.03748	-3.747	2.838			
	4	-0.03748	-3.747	2.838			
	5	-0.03748	-3.747	0.3747			
	6	-0.03748	-3.747	0.3747			
	Σ	-0.2248	-22.48	16.81			

Shear:

$F_{v,Ed,hor,bolt\ gr.}$

=

201.2

kN

formula

$F_{v,Ed,vert,bolt\ gr.}$

=

100

kN

formula

$M_{Ed,bolt\ gr.}$

=

0

kN*m

formula

$M_{ecc,Ed,bolt\ gr.}$

=

7

kN*m

formula

β_L

=

1

formula

$F_{v,Rd,1\ bolt}$

=

120.6

kN

formula

 $F_{v,Rd,bolt\ gr.}$

=

442.9

kN

formula

 $F_{v,Ed,bolt\ gr.}$

=

224.7

kN

formula

type of check

Bolt Group:

Case

1

2

a

b

c

d

e

data and formulas used for the check

Bolt shear:

ok 51%

formula

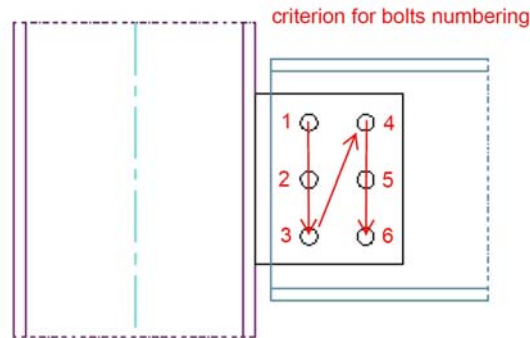
check shows into the sum up

Draw in-plane forces

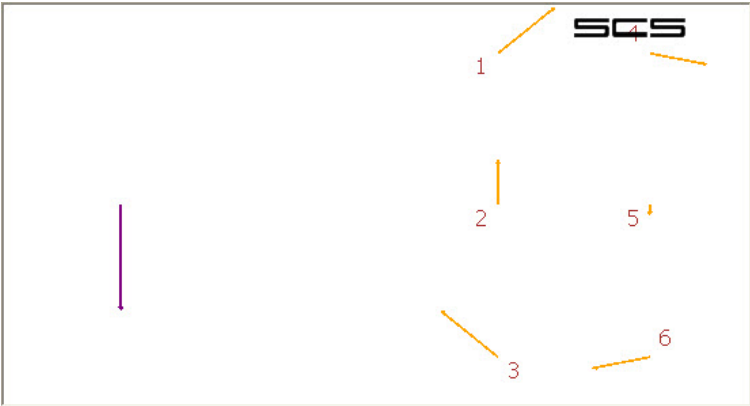
Draw Sketch

forces per bolt

The criterion for bolts numbering is shown in the picture below: bolts are numbered vertically first then horizontally.



The button *draw-in-plane forces* shows the forces in the plane (bolt shear) for each bolt in the graphic window.



To return on the image of sketch, the user can move the mouse into the drawing window or could click the *Draw sketch* button.

8.3.1 T-stub button

This button is available only when bolts are in tension and the selected calculation method and/or standard uses the T-stub method.

See paragraph 24.2 *T-stub report*.

T-Stub notes

8.4 Plate bearing and/or beam bearing

This tab deals with plate bearing, in relation to both horizontal and vertical forces.

The total reference thickness refers to the sum of the parts of the connection involved in the check. For example, if it's the plate tab and there are two shear planes, this cell gives the sum of the thickness of the plates. If (FP1 for example), the web of the beam is reinforced by a plate, the cell (beam bearing) gives the sum of the web and the reinf. plate.

This tab also gives advice about minimum spacings/gauges and their optimum values.

Tot. ref. thk=	16	$P_{1,min.} = 48.4 \text{ mm}$	$P_{1,opt.} = 77 \text{ mm}$	<div>1 a</div>
		$e_{1,min.} = 26.4 \text{ mm}$	$e_{1,opt.} = 66 \text{ mm}$	<div>2 b</div>
		$P_{2,min.} = 52.8 \text{ mm}$	$P_{2,opt.} = 66 \text{ mm}$	<div>c</div>
		$e_{2,min.} = 26.4 \text{ mm}$	$e_{2,opt.} = 33 \text{ mm}$	<div>d</div>
				<div>e</div>

Tot. ref. thk=	0.394	$s_{min.} = 1.89 \text{ in}$	$s_{opt.} = 2.13 \text{ in}$	<div>a</div>
		$e_{1,min.} = 0.866 \text{ in}$	$e_{1,opt.} = 2.13 \text{ in}$	<div>b</div>
		$g_{min.} = 1.89 \text{ in}$	$g_{opt.} = 2.13 \text{ in}$	<div>c</div>
		$e_{2,min.} = 0.866 \text{ in}$	$e_{2,opt.} = 1.06 \text{ in}$	<div>d</div>
				<div>e</div>

Bolt distances from the edge of plate and from the other bolts are checked. If the check fails, SCS returns an error message.

Tot. ref. thk=	16	$P_{1,min.} = 48.4 \text{ mm}$	$P_{1,opt.} = 77 \text{ mm}$	<div>1 a</div>
		$e_{1,min.} = 26.4 \text{ mm}$	$e_{1,opt.} = 66 \text{ mm}$	<div>2 b</div>
		$P_{2,min.} = 52.8 \text{ mm}$	$P_{2,opt.} = 66 \text{ mm}$	<div>c</div>
		$e_{2,min.} = 26.4 \text{ mm}$	$e_{2,opt.} = 33 \text{ mm}$	<div>d</div>
Horizontal forces		Vertical forces		<div>e</div>
$P_{1,hpl} =$	ok	$P_{1,vpl} =$	ok	
$e_{1,hpl} =$	ERR min 26.4 mm	$e_{1,vpl} =$	ok	
$P_{2,hpl} =$	ok	$P_{2,vpl} =$	ok	
$e_{2,hpl} =$	ok	$e_{2,vpl} =$	ERR min 26.4 mm	

If Eurocode (#00) is selected, SCS shows here the bearing resistance coefficients used in calculations (see Table 3.4 #00).

$\alpha_{b,hpl} = 0.379$ $\alpha_{d,hpl} = 0.379$ $k_{1,hpl} = 2.5$ $\alpha_{b,vpl} = 0.515$ $\alpha_{d,vpl} = 0.515$ $k_{1,vpl} = 1.48$

If the standard is Din (#10) SCS, similar considerations apply.

$\alpha_{b,hpl} = 2.14$ $\alpha_{b,vpl} = 1.35$

The check is made in relation to horizontal and vertical forces acting on plate and beam.

$F_{b,Rd,h,pl}=$	87.27	kN	<i>formula</i>
$F_{b,Ed,h,pl}=$	19.11	kN	<i>Bolt 3</i>
$F_{b,Rd,v,pl}=$	70.35	kN	<i>formula</i>
$F_{b,Ed,v,pl}=$	6.667	kN	<i>Bolt 3</i>
Plate bear. h.:	ok 22%	Plate bear. v.:	ok 9%

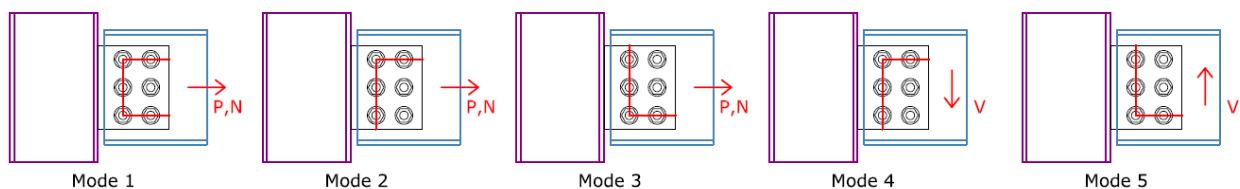
$F_{b,Rd,h,bw} =$	131.2	kN	<i>formula</i>
$F_{b,Ed,h,bw} =$	1	kN	<i>Bolt 6</i>
$F_{b,Rd,v,bw} =$	113	kN	<i>formula</i>
$F_{b,Ed,v,bw} =$	8.333	kN	<i>Bolt 6</i>
Beam web bear h.:	ok 1%	Beam web bear v.:	ok 7%

$\Phi R_{n,h,pl}=$	24.6	kips	<i>formula</i>
$P_{h,pl}=$	0.3107	kips	<i>Bolt 5</i>
$\Phi R_{n,v,pl}=$	16.4	kips	<i>formula</i>
$P_{v,pl}=$	1.462	kips	<i>Bolt 4</i>
Plate bear. h.:	ok 1%	Plate bear. v.:	ok 9%

$\Phi R_{n,h,bw} =$	19.6	kips	<i>formula</i>
$P_{h,bw} =$	10	kips	<i>Bolt 2</i>
$\Phi R_{n,v,bw} =$	23.23	kips	<i>formula</i>
$P_{v,bw} =$	3.8	kips	<i>Bolt 1</i>
Beam web bear h.:	ok 51%	Beam web bear v.:	ok 16%

8.5 Plate block shear

When checking plate block shear (also known as block tearing) SCS takes in account three different modes of block shear for axial loads and two modes of block shear for shear loads for each bolt group on the plate. The design block tearing resistance for mode 1, 2 and 3 considers the bolt group subject only to the axial load, while the design block tearing resistance for mode 4 and 5 considers the bolt group subject only to the shear load..



Block shear (block tearing) - Plate - N = V _{hor,sd} ,bolt gr.						k _{unif} =	0.5
		V _{eff,Rd} =	formula				
Mode 1:	Ant=	1056 mm²	Anv=	1848 mm²	V _{eff,Rd1} =	439.9 kN	
Mode 2:	Ant=	1320 mm²	Anv=	924 mm²	V _{eff,Rd2} =	349.8 kN	
Mode 3:	Ant=	1320 mm²	Anv=	924 mm²	V _{eff,Rd3} =	349.8 kN	
V _{eff,Rd N Pl} =		349.8 kN	formula				
V _{eff,Ed N Pl} =		0 kN	formula				
Plate block sh. - N:		ok 0%					

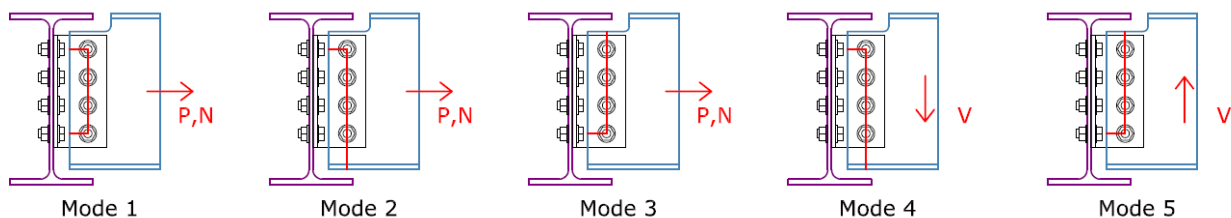
e

Block shear (block tearing) - Plate - $V = V_{vert, sd, bolt\ gr.}$					
Mode 4:	Ant=	924 mm ²	Anv=	1320 mm ²	$V_{eff, Rd4} = 342.1 \text{ kN}$
Mode 5:	Ant=	924 mm ²	Anv=	1320 mm ²	$V_{eff, Rd5} = 342.1 \text{ kN}$
$V_{eff, Rd\ V\ Pl} =$		342.1 kN	<i>formula</i>		
$V_{eff, Ed\ V\ Pl} =$		50 kN	<i>formula</i>		
Plate block sh. - V:		ok 15%			

Ant is the net area subjected to tension, while Anv is the net area subjected to shear. The areas Ant and Anv for the mode 2&3 are reversed compared to the areas of mode 4&5.
About kunif value, see par. 7.5.6 *Block shear kunif*=1.

8.6 Section block shear

Same considerations of previous paragraph relative to plate apply. The 5 modes are shown in the figure below.

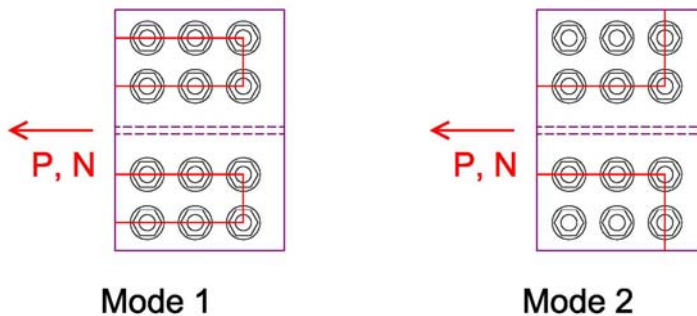


Block shear (block tearing) - Beam - N = V _{hor,sd} ,bolt gr.						k _{unif} =	0.5
		V _{eff,Rd} =	formula				
Mode 1:	Ant =	766.8 mm²	Anv =	383.4 mm²	V _{eff,Rd 1} =	181.1 kN	
Mode 2:	Ant =	2818 mm²	Anv =	191.7 mm²	V _{eff,Rd 2} =	489.9 kN	
Mode 3:	Ant =	944.3 mm²	Anv =	191.7 mm²	V _{eff,Rd 3} =	182.5 kN	
V _{eff,Rd N beam} =		181.1 kN	formula				
V _{eff,Ed N beam} =		0 kN	formula				
Beam block sh. - N:		ok 0%					

Block shear (block tearing) - Beam - $V = V_{\text{vert},sd,bolt\ gr.}$					
Mode 4:	Ant=	191.7	mm ²	Anv=	2818
Mode 5:	Ant=	191.7	mm ²	Anv=	944.3
$V_{\text{eff},Ed\ V\text{-beam}}$	=	167.7	kN	<i>formula</i>	
$V_{\text{eff},Ed\ V\text{+beam}}$	=	0	kN	<i>formula</i>	
Beam block sh. - V:		ok	0%		

8.6.1 Splice

In the splice connection (.sp4 files), only the 2 modes below are considered for bolt group 1 (the one for flanges) when checking block shear on the member.



8.7 Plate/beam strength

Plate and/or beam strength is checked if required by selected standard. Resistance to axial load, bending (strong and weak), shear (strong and weak) are verified. Even torsion and combination of stresses are calculated when relevant.

8.8 More

More limit states are checked, depending on the connections. They are grouped in the 'More' tab.

<i>Lat. tors. plate buckling</i>					
\bar{n}_{Lr}	=	20.6	<i>formula</i>		
p_b	=	275	N/mm ²	<i>formula</i>	
m_{Lr}	=	1	<i>formula</i>		
$M_{Rd,buckl}$	=	19.6	kN*m	<i>formula</i>	
$M_{sd,str}$	=	9.85	kN*m	<i>formula</i>	
Pl. buckling:		ok	50%	<i>formula</i>	

lateral torsional plate buckling with the standard Eurocode/BS

8.9 Welding

Design of welds can be performed checking results in this tab. Usually (depending on the module and the choices made by the user) two parts are visible: one about the web weld and another one about the weld to the flange. Depending on the module, the weld could be between plate and primary beam (e.g.: FP1) or between plate and secondary beam (e.g.: EM3).

<i>Web weld:</i>				
A_{w-web}	=	1.579	in ²	<i>formula</i>
$N_{E,w}$	=	2.698	kips	
$V_{E,w, str}$	=	11.24	kips	
$V_{E,w, wk}$	=	1.349	kips	
$M_{E,w, str}$	=	2.581	kips*ft	<i>formula</i>
$M_{E,w, wk}$	=	-0.04425	kips*ft	<i>formula</i>
$T_{E,w}$	=	0	kips*ft	<i>formula</i>
$W_{R,w, str}$	=	1.865	in ³	<i>formula</i>
$W_{R,w, wk}$	=	0.0293	in ³	<i>formula</i>
$W_{Rw,T}$	=	0.5611	in ³	<i>formula</i>
σ_{\perp}	=	15.57	ksi	<i>formula</i> normal stress perpend. to the throat
τ_{\perp}	=	-15.57	ksi	<i>formula</i> shear stress perp. to the weld axis
τ_{\parallel}	=	5.035	ksi	<i>formula</i> shear stress parallel to the weld axis
$\sigma_{w,v}$	=	32.34	ksi	<i>formula</i> global stress on the weld
$\sigma_{w,Rd}$	=	52.21	ksi	<i>formula</i> design resistance of the weld
Weld comb.:		ok 62%		<i>formula</i>
Weld σ_{\perp} :		ok 0%		<i>formula</i>

8.9.1 American codes

When the standard is set to any of the American codes (AISC or CSA), the welds are calculated according to the Elastic method (#20). The instantaneous centre of rotation method for the welds has not been inserted in SCS yet.

When a weld is loaded by eccentricity, the resisting value of the weld is compared with the shear that the weld must withstand to resist the moment caused by the eccentricity.

8.10 Stiffness

Where relevant (EC most of all) the stiffness of the connection is shown. It should be noted that stiffness could change depending on sign of bending moment.

k_{13}	=	+Infinito	mm	<i>formula</i>
e_k	=	Non un	mm	<i>formula</i>
$k_{stiff.}$	=	1.922	mm	<i>formula</i>
z_{eq}	=	323.5	mm	<i>formula</i>
$S_{j,ini,Mpos}$	=	4.223E+04	kN*m	<i>formula</i>
$S_{j,ini,Mneg}$	=	4.223E+04	kN*m	<i>formula</i>
μ	=	+Infinito		<i>formula</i>
S_j	=	Non un	kN*m	<i>formula</i>

9 Fin-plate column-beam (FP1)

9.1 About FP1

This module is about fin plate connections between a column and a beam. The connection is also known as shear plate, shear tab or other.

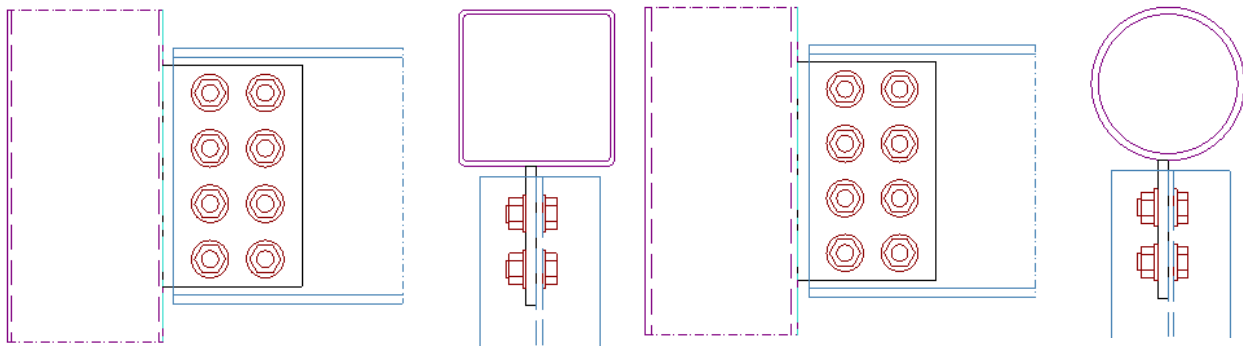
This connection is considered as a pin type connection but a limited bending moment (usually generated by eccentricity of shear). Shear and tension are both transmitted by the plate welded to primary member (column) and bolted to secondary member (beam).

9.2 Connection type

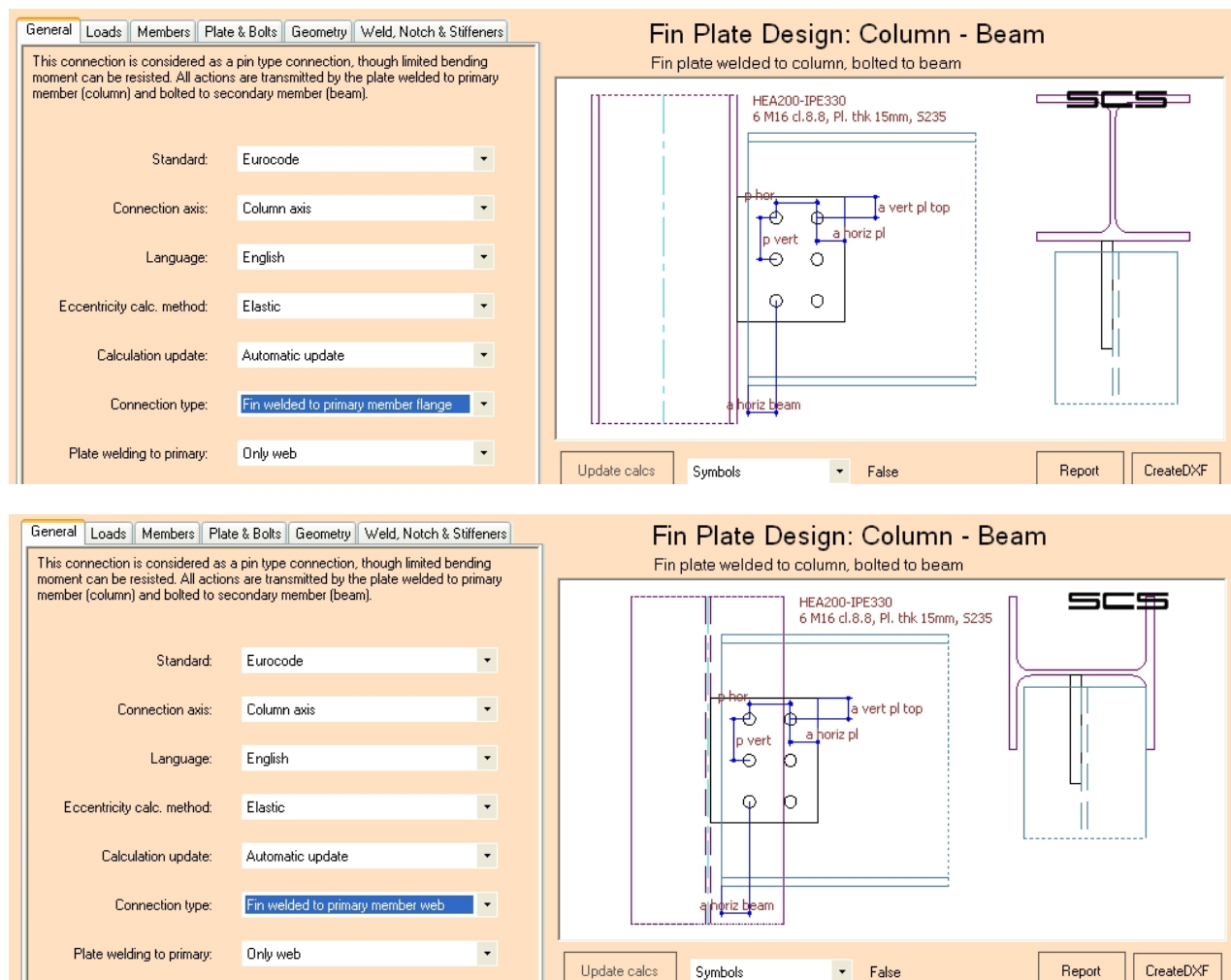
The column shape can be a I/H shaped member (I in the selection box) or a round/rectangular hollow section (Ro. o Re. in the selection).

Primary Member: ▼

h =	mm	<input type="text"/> ▼
b =	mm	<input type="text"/> ▼
t_f =	mm	<input type="text"/> ▼
t_w =	mm	<input type="text"/> ▼
r =	mm	<input type="text"/> ▼



You can choose between two connection types for I/H shaped primary members: fin plate welded to primary member web or fin plate welded to primary member flange.



9.3 Plate welding to primary

If the connection type is fin plate welded to primary member web, then you could choose among three different type of weld: fin plate welded only on primary member web; fin plate welded on primary member web and top stiffener; fin plate welded on primary member web and top and bottom stiffeners.

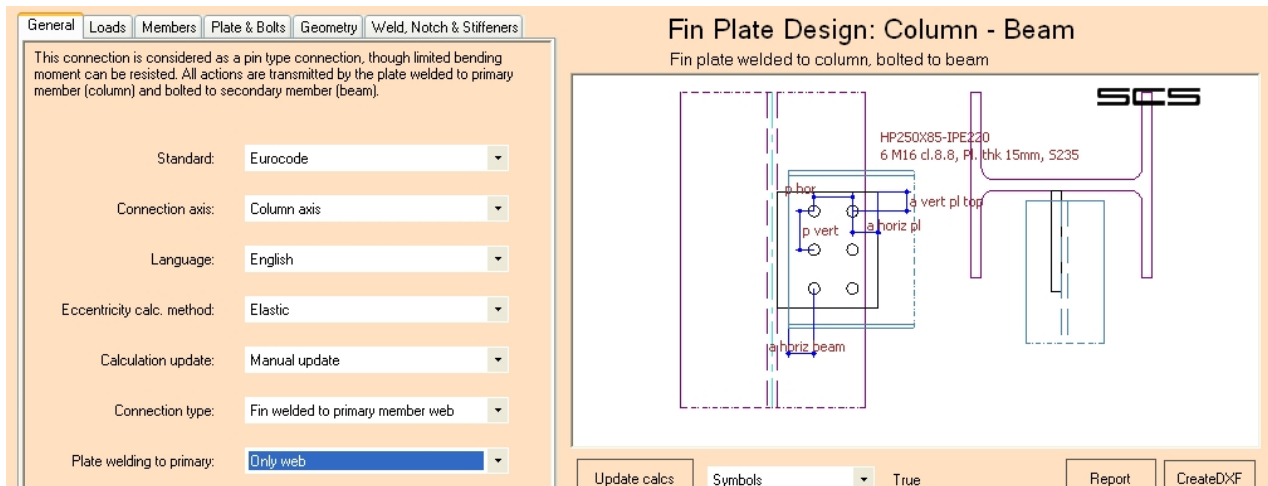
If the connection type is fin plate welded to primary member flange, only fin plate welded only on primary member web (flange in this case) is allow.

9.3.1 Fin plate welded only on primary member web

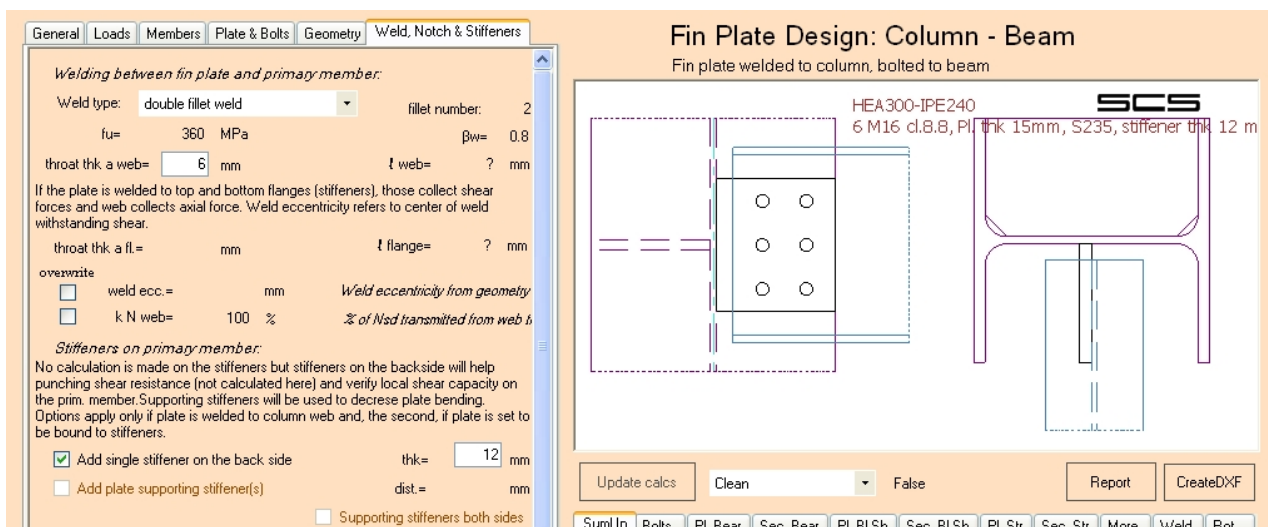
In this configuration, in *Weld, Notch & Stiffeners* you can add a stiffener on the back side of the column and/or notch the beam.

You can choose to add a stiffener on the back side of the column and select the thickness of the plate.

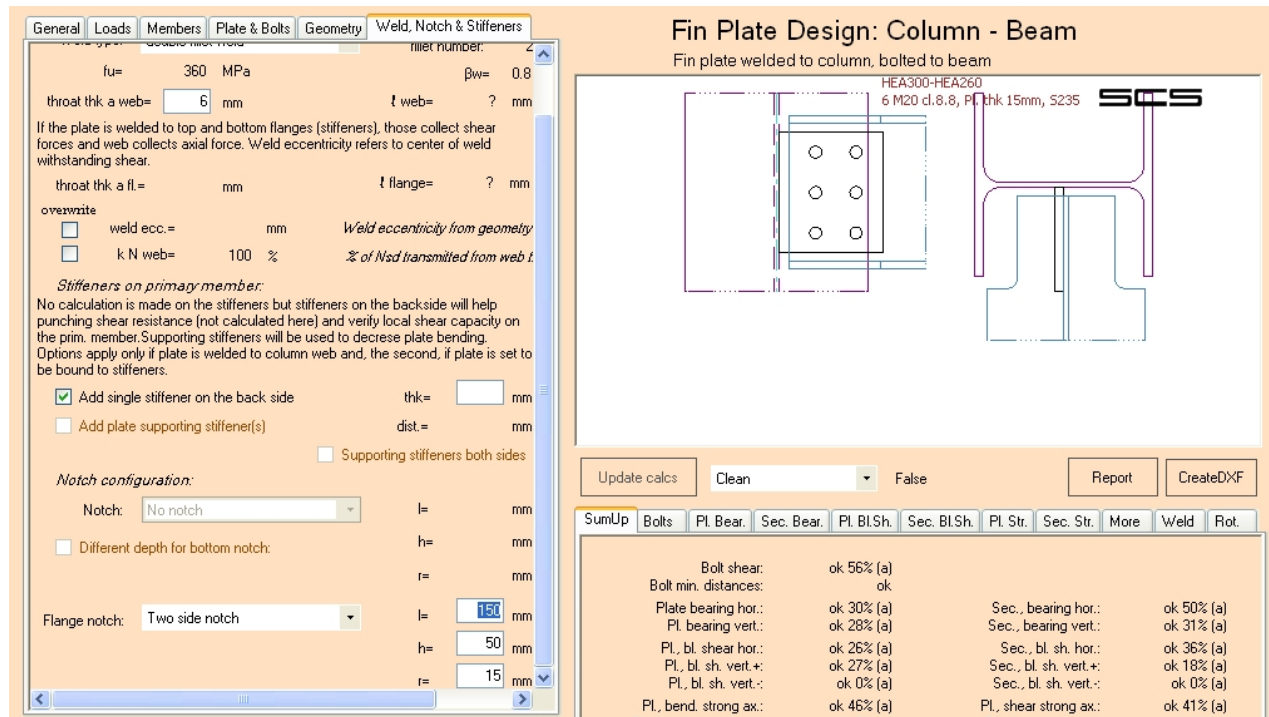
You can also make a notching on the beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



Fin plate welded on primary web.



Fin plate welded on primary web with single stiffener on the back side.



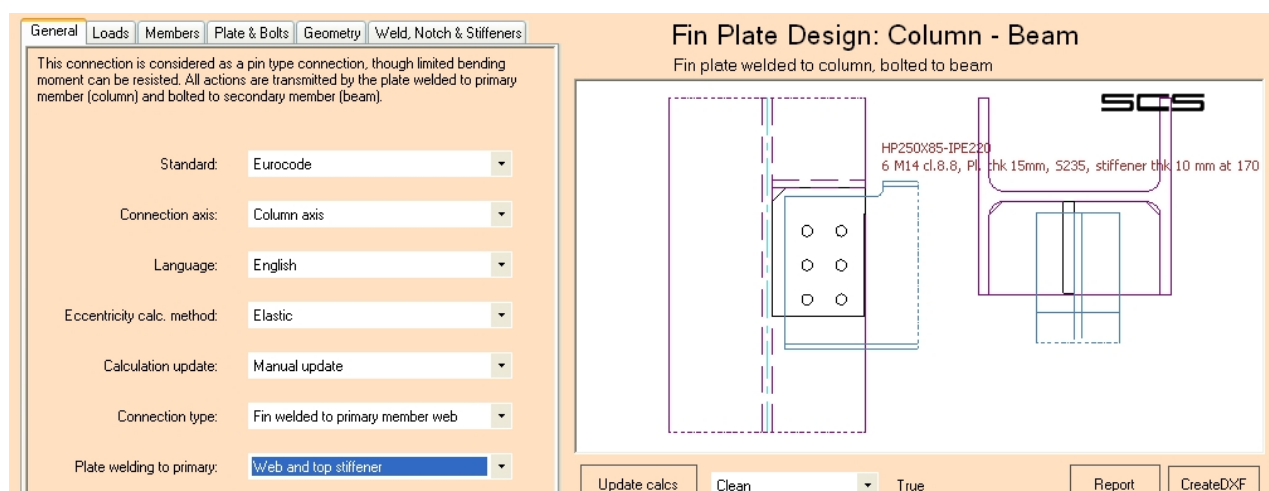
Fin plate welded on primary web with stiffener on column and notch on beam flanges.

9.3.2 Fin plate welded on primary member web and top stiffener

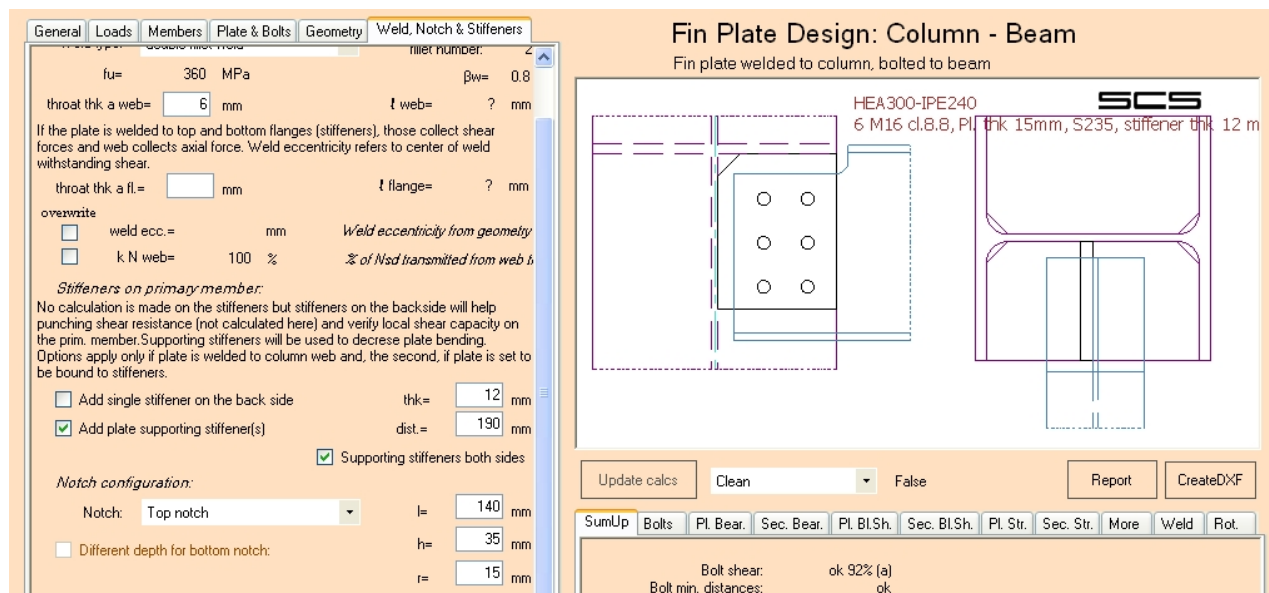
In this configuration, in *Weld, Notch & Stiffeners* you have to complete the geometric data of the plate supporting stiffener, specifying the thickness and the distance from the bottom edge of the plate.

You can also choose to add stiffener on the opposite side of the column web, and/or to add a stiffener on the back side of the column in the middle of the joint (it have the same thickness of the plate supporting stiffener).

You can also make a notching on the beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



Fin plate welded on primary web and stiffener with top notch on secondary member.



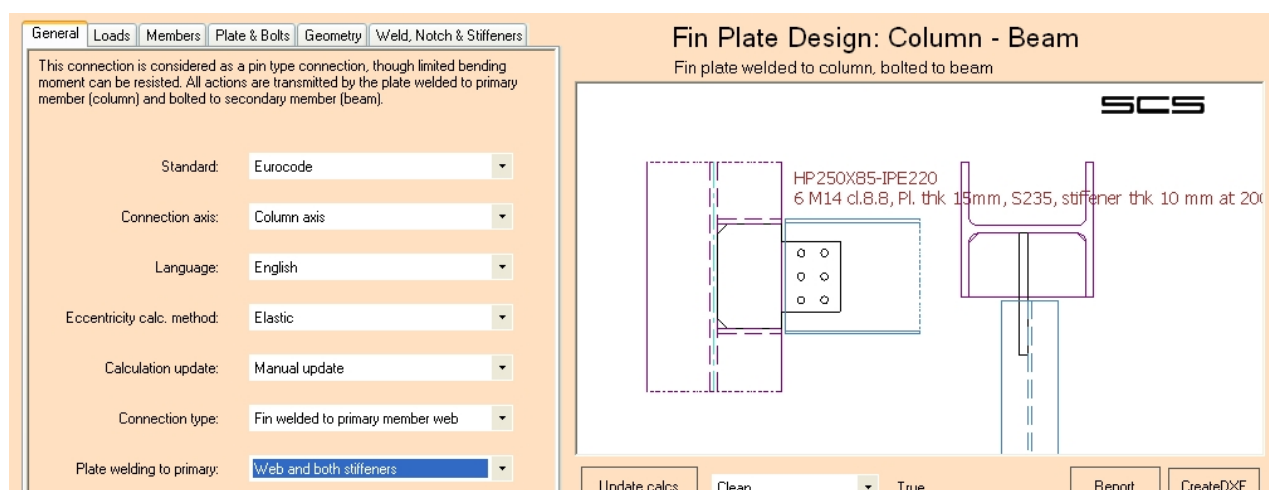
Fin plate welded on primary web and stiffener with top notch on secondary member.

9.3.3 Fin plate welded on primary member web and both stiffeners

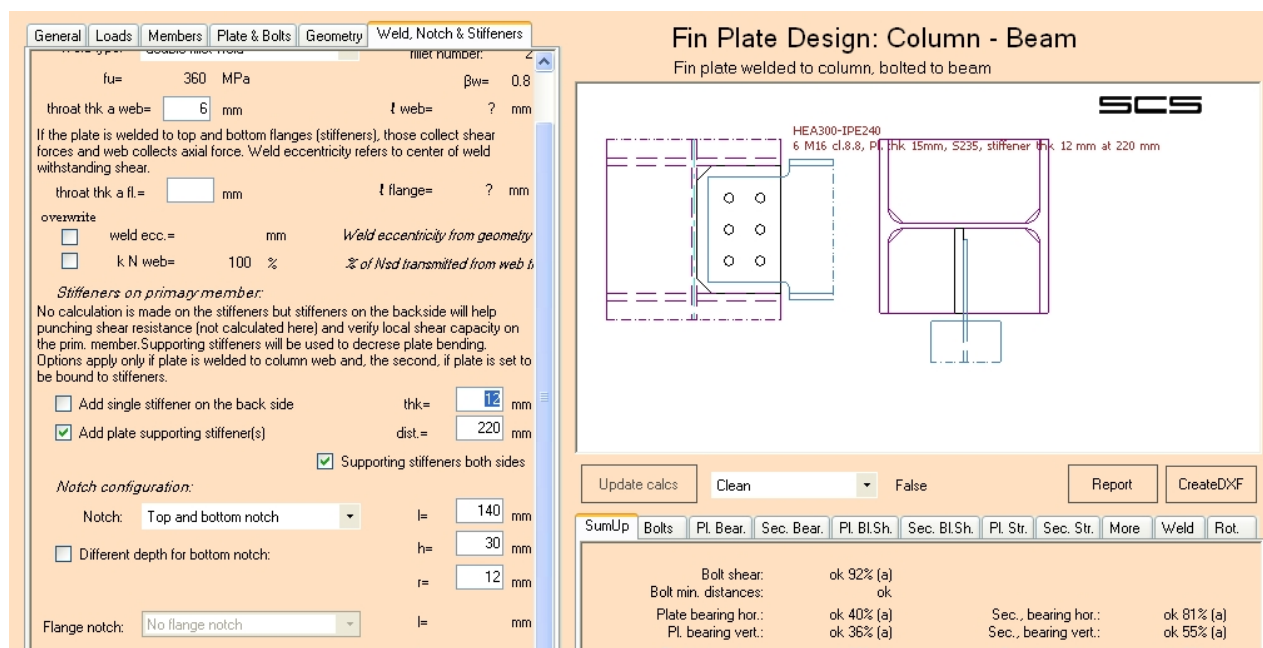
In this configuration, in *Weld, Notch & Stiffeners* you have to complete the geometric data of the plate supporting stiffeners, specifying the thickness and the distances from the center of the fin plate.

You can also choose to add stiffeners on the opposite side of the column web, and/or to add a stiffener on the back side of the column in the middle of the joint (it have the same thickness of the plate supporting stiffeners).

You can also make a notching on the beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



Fin plate welded on primary web and both stiffeners with no notch on secondary member.



Fin plate welded on primary web and both stiffeners with notches on secondary member.

NOTE: See also par. *Weld, Notch & Stiffeners* to complete the geometry of the connection.

If the plate is welded not only to the web but also to at least one flange, the eccentricity is taken as zero unless the bolt group center lies beyond the weld.

9.3.4 *K N web and eccentricity*

The web weld is appointed to take all the N action. This can be changed overwriting the default 100% value.

If the flange weld is present, the part not withstand by the weld is withstood by the weld flange (in shear).

If the plate is welded to the web only, all forces go into the web weld.

If the plate is welded also to one or two flanges, forces are divided between the web and the flange weld(s).

The percentage is shown in the dedicated textbox, that automatically divide forces 50% on web and the rest on flanges. The engineer can choose, through the 'overwrite' button, to change this percentage.

Consider the following assumptions by SCS:

- the eccentricity is relative to the web weld only; the flange weld(s) have no eccentricity
- the eccentricity increases algebraically the moment applied to the weld; therefore, a positive shear with a negative eccentricity (weld axis on the right hand side of the connection axis) will actually decrease a positive moment
- if the plate is welded to both flanges, the total moment (including eccentricity) is divided between the distance between flange welds (lever arm) into opposite shears acting on flange welds. The moment on the web weld is considered as zero in this situation.

10 Fin-plate beam-beam (FP2)

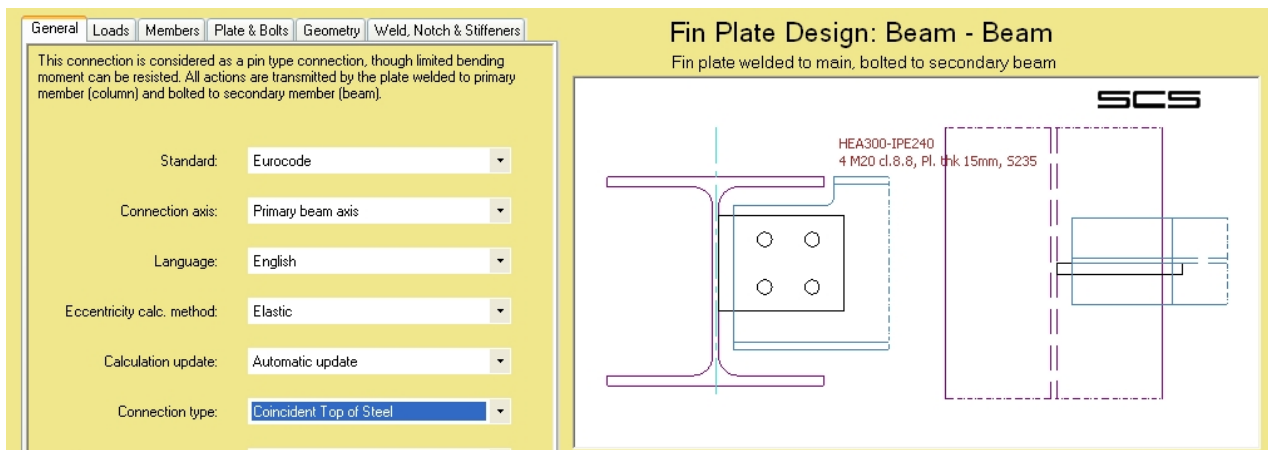
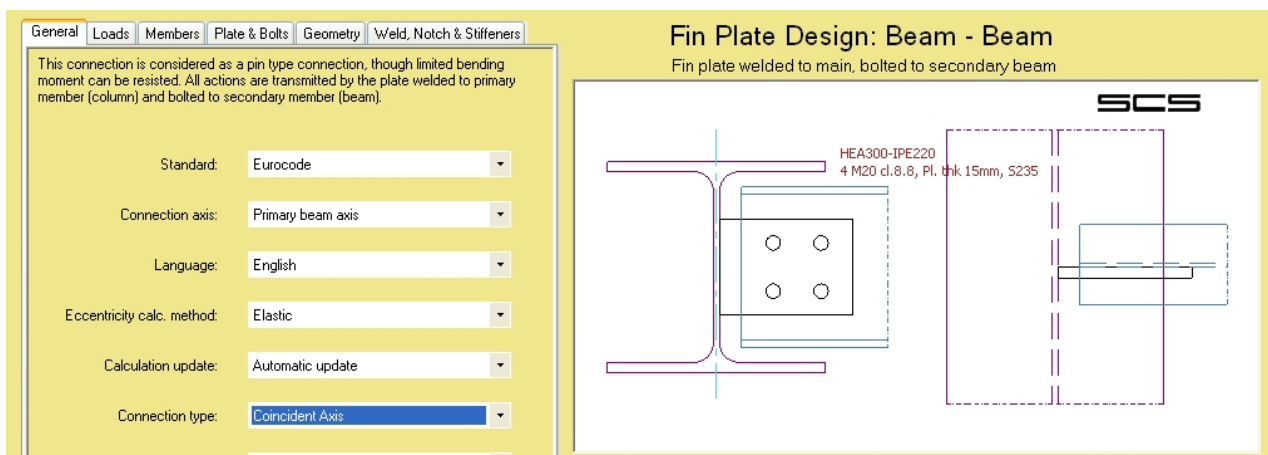
10.1 About FP2

The connection is the same as FP1 but it's between beams (secondary and primary)..

10.2 Connection type

You can choose between two connections type:

- coincident axis of the two beams;
- coincident top of steel of the two beams.



10.3 Plate welding to primary

You can choose among three different type of weld for the plate to primary member:

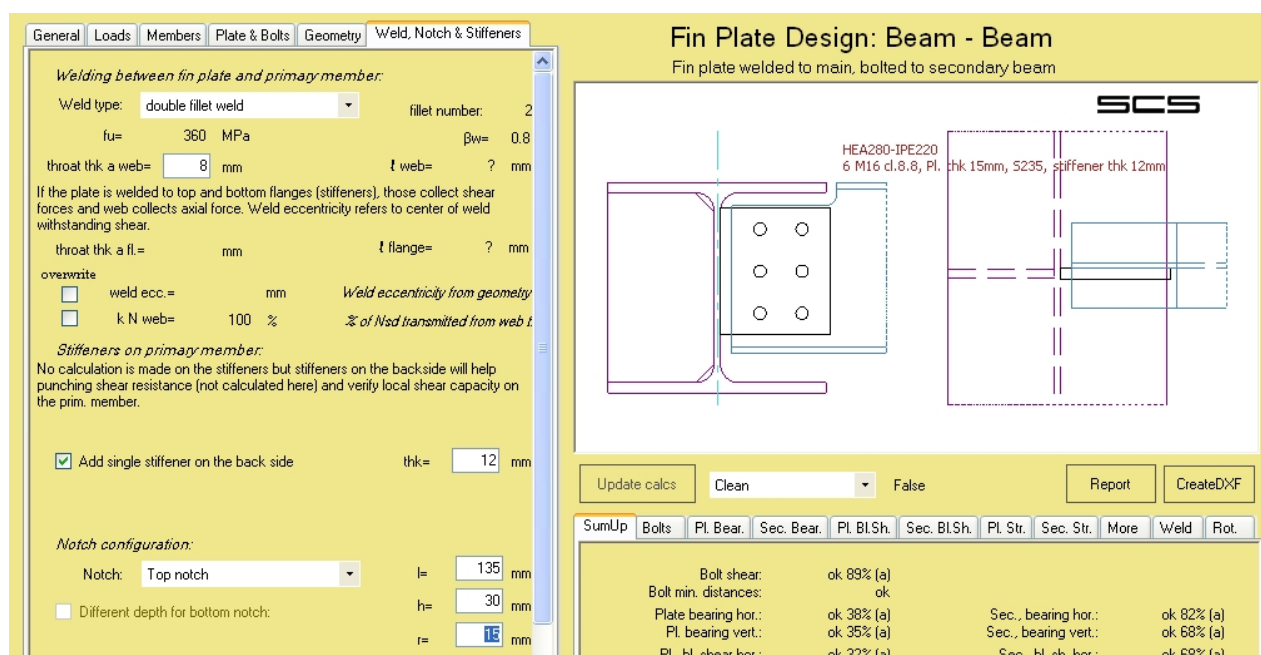
- fin plate welded only on primary web;
- fin plate welded on primary member web and top flange;
- fin plate welded on primary member web and top and bottom flanges.

10.3.1 Fin plate welded on primary member web

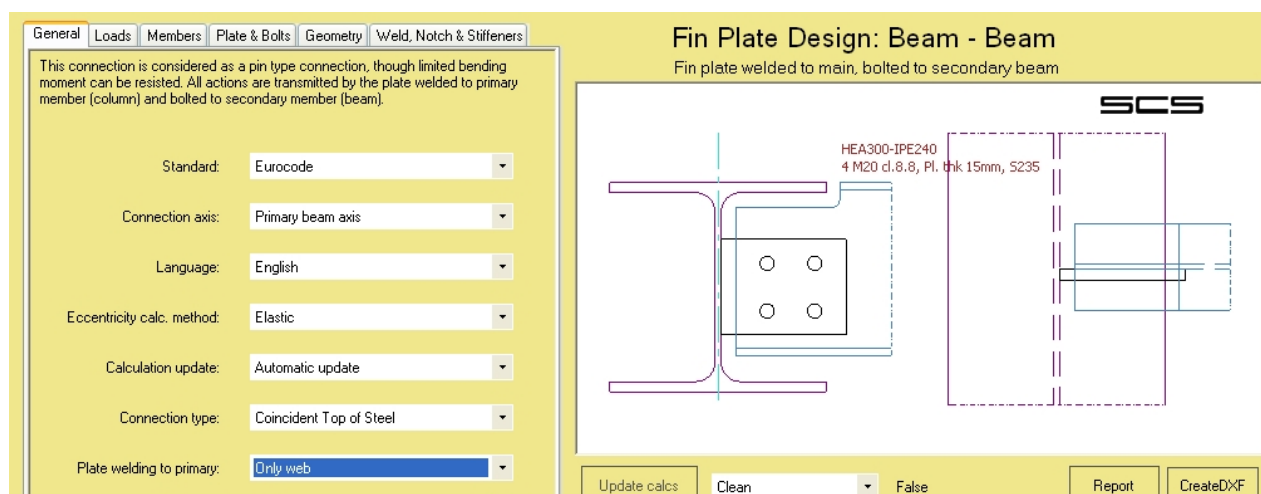
In this configuration, in *Weld, Notch & Stiffeners* you can add a stiffener on the back side of the primary beam and/or notch the secondary beam.

You can choose to add a stiffener on the back side of the primary beam and specify the thickness of the plate.

You can make a notching on the secondary beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the secondary beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



Fin plate welded only on primary web with stiffener on the back of primary beam and top notch on secondary beam.



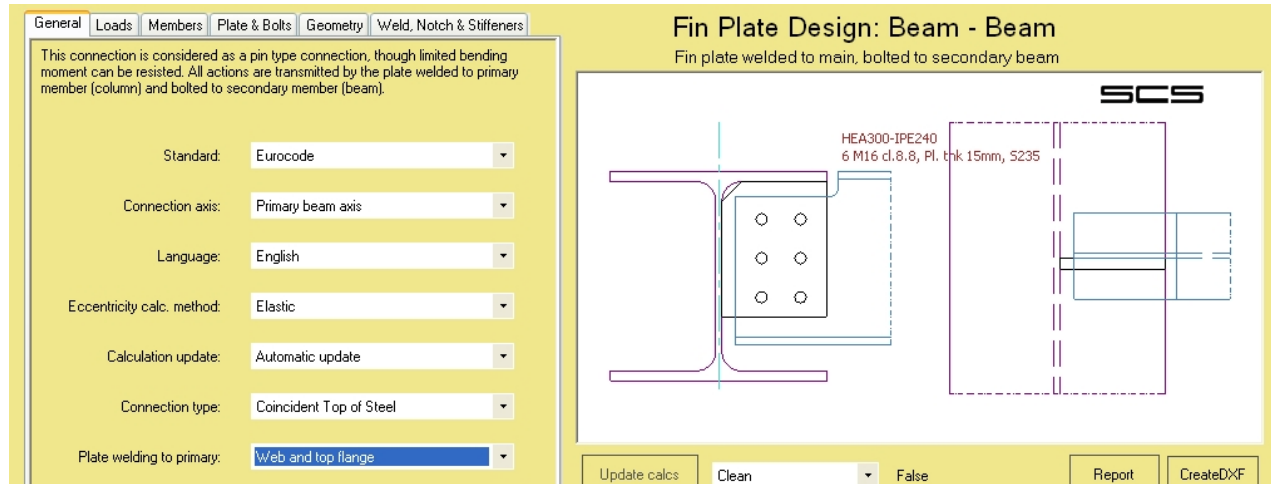
Fin plate welded only on primary web with bolts group vertical symmetric with primary beam.

10.3.2 Fin plate welded on primary member web and top flange

In this configuration, in *Weld, Notch & Stiffeners* you can add a stiffener on the back side of the primary beam and/or notch the secondary beam.

You can choose to add a stiffener on the back side of the primary beam and specify the thickness of the plate.

You can make a notching on the secondary beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the secondary beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



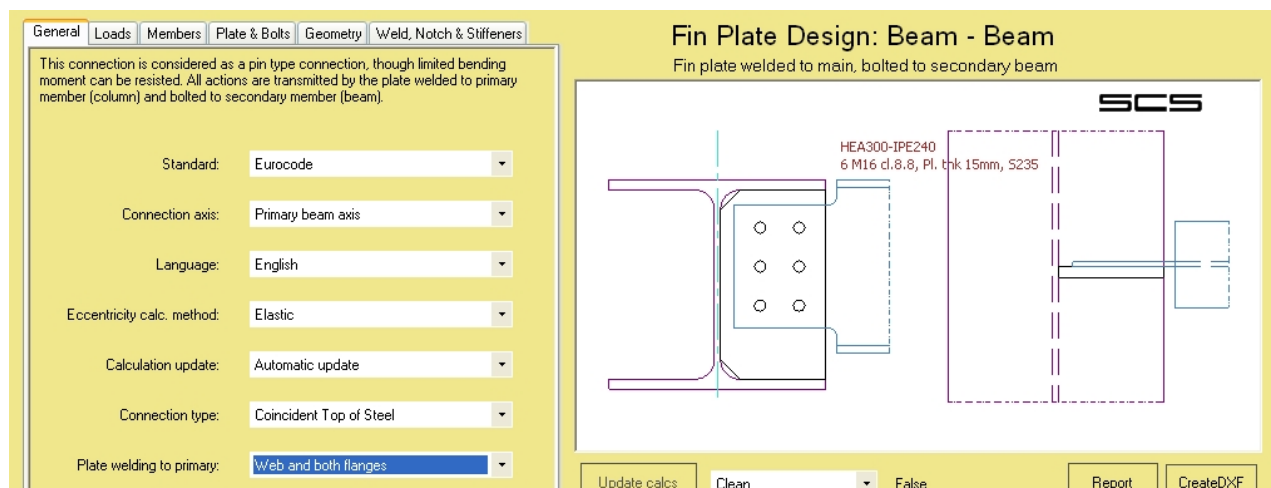
Fin plate welded on primary web and top flange with top notch on secondary beam and bolts group vertical symmetric with secondary beam.

10.3.3 Fin plate welded on primary member web and both flanges

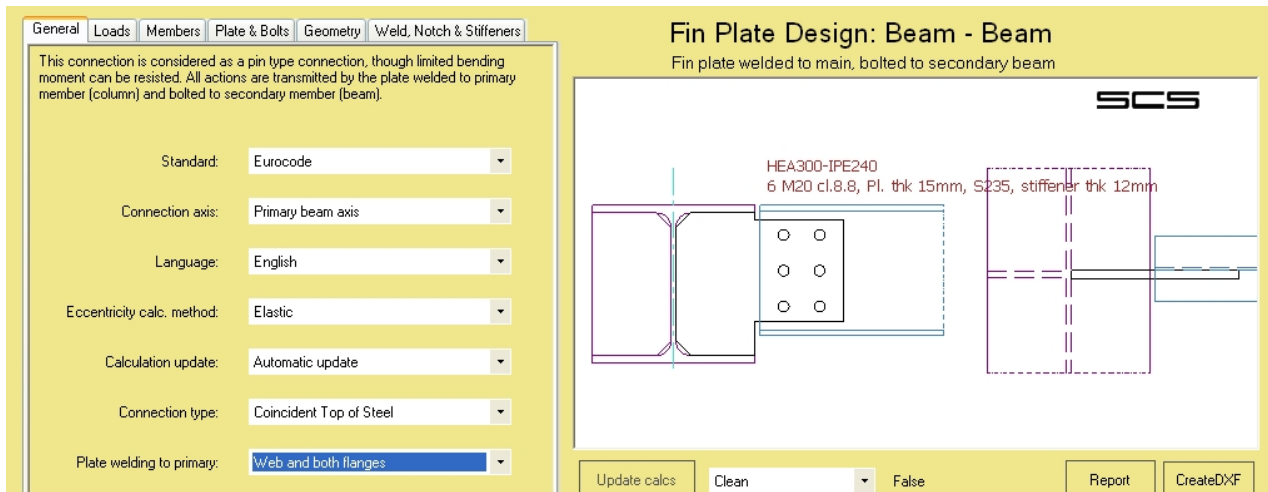
In this configuration, in *Weld, Notch & Stiffeners* you can add a stiffener on the back side of the primary beam and/or notch the secondary beam.

You can choose to add a stiffener on the back side of the primary beam and specify the thickness of the plate.

You can make a notching on the secondary beam flanges and web (only on top, only on bottom or both notch) or make a notching only on the secondary beam flanges (on the right, on the left or both notch) and select length, height and radius of the notches.



Fin plate welded on primary web and flanges with double notch on secondary beam.



Fin plate welded on primary web and flanges with stiffener on the back side and no notch on secondary beam.

NOTE: See also par. Weld, Notch & Stiffeners to complete the geometry of the connection.

11 Double-fin plate (DS1)

11.1 About DS1

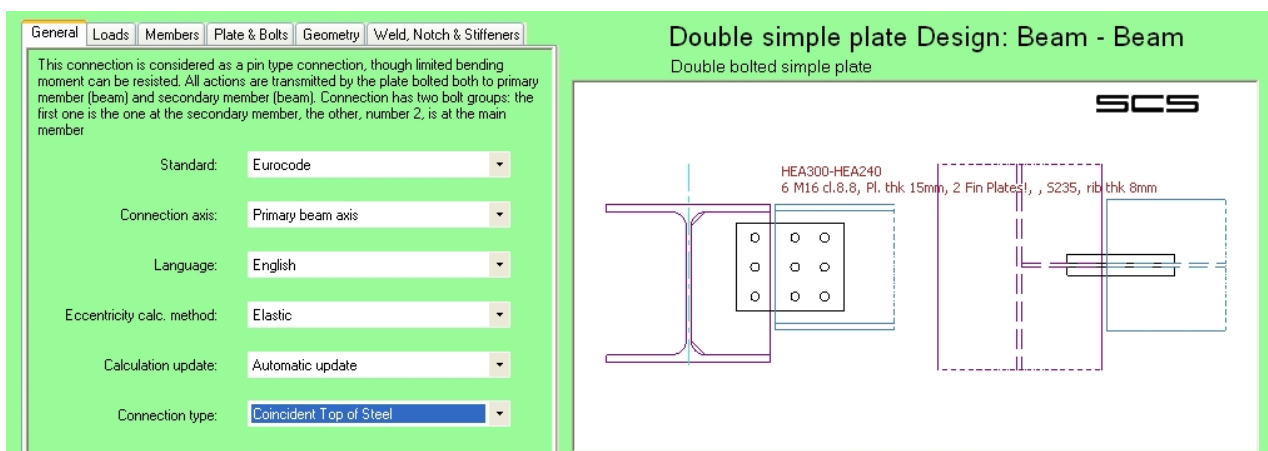
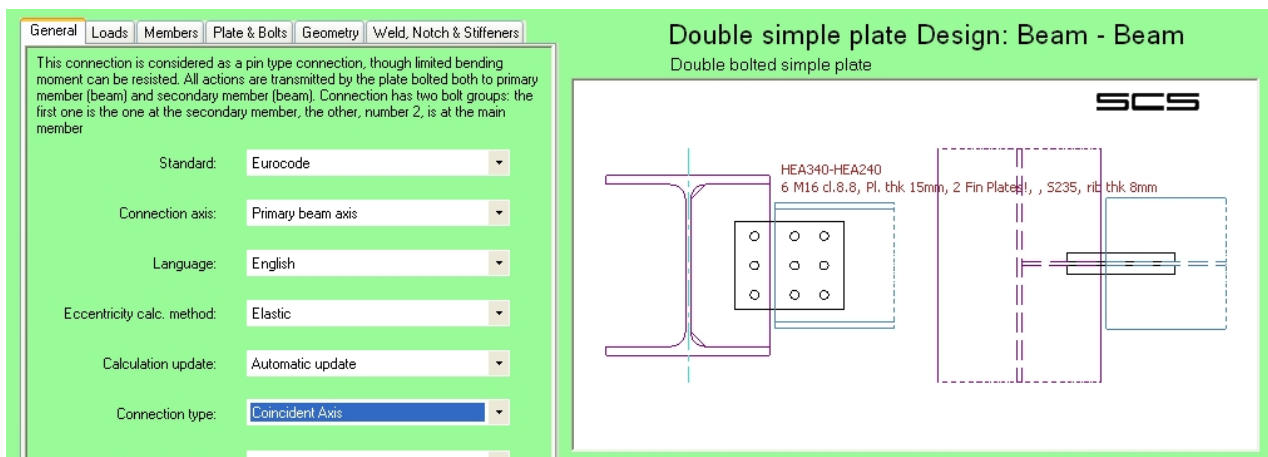
This module allows to calculate a special connection realized bolting 2 plates both to primary beam and secondary.

This connection is considered as a pin type connection, though limited bending moment can be resisted. All actions are transmitted by the plate bolted both to primary member (beam) and secondary member (beam). The connection has two bolt groups: the first one is the one at the secondary member, the other, number 2, is at the main member.

11.2 Connection type

You can choose between two connection types:

- coincident axis of the two beams;
- coincident top of steel of the two beams.



12 Splice (SP4)

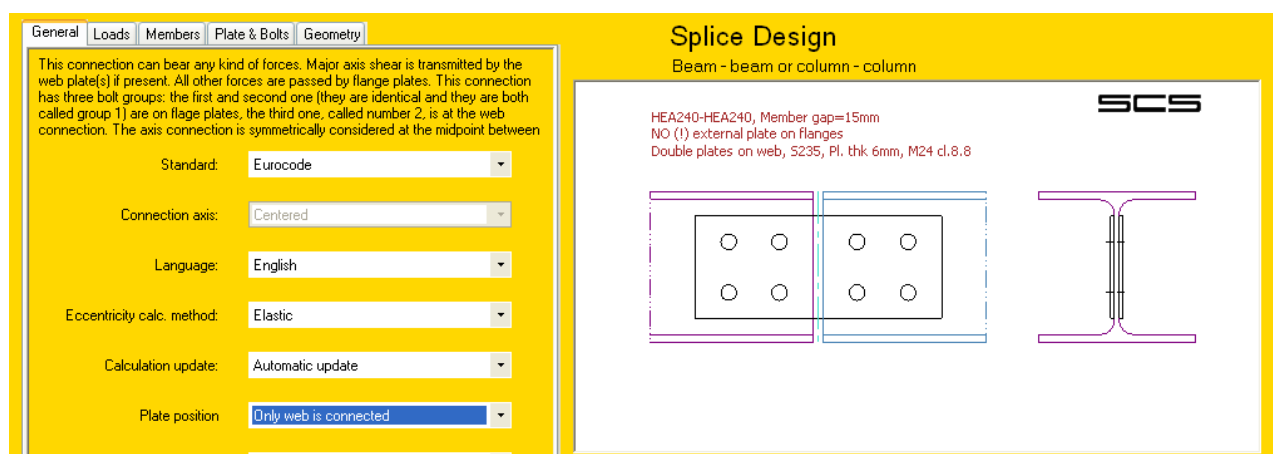
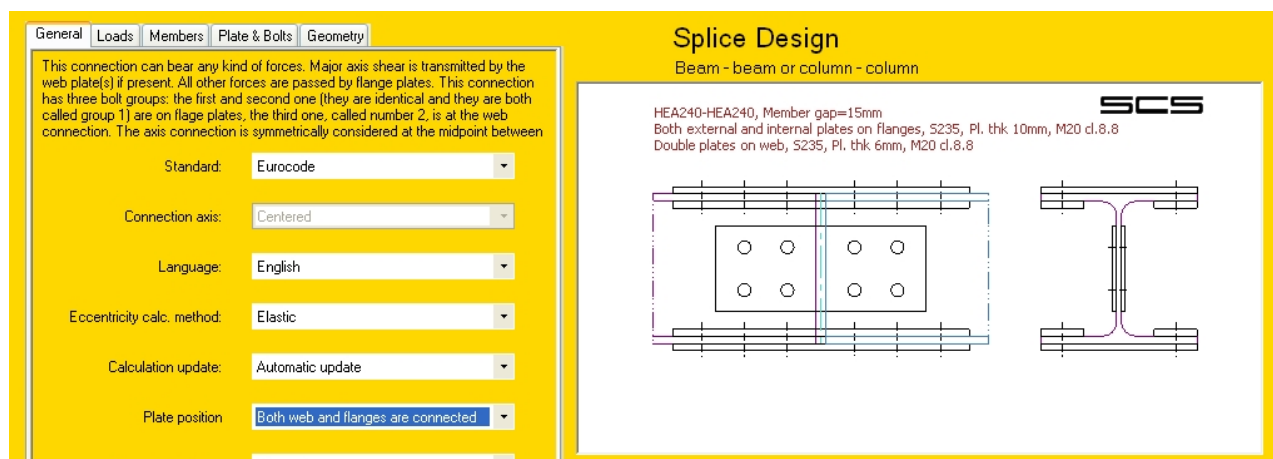
12.1 About SP4

This connection can bear any kind of forces and it's the state of the art when column-beam connection on main column axis must transmit consistent bending moment.

12.2 Connection type

You can choose among three different configurations for the splice:

- plates on web and flanges;
- plates only on web;
- plates only on flanges.



General

Loads

Members

Plate & Bolts

Geometry

This connection can bear any kind of forces. Major axis shear is transmitted by the web plate(s) if present. All other forces are passed by flange plates. This connection has three bolt groups: the first and second one (they are identical and they are both called group 1) are on flange plates; the third one, called number 2, is at the web connection. The axis connection is symmetrically considered at the midpoint between

Standard:

Eurocode

Connection axis:

Centered

Language:

English

Eccentricity calc. method:

Elastic

Calculation update:

Automatic update

Plate position

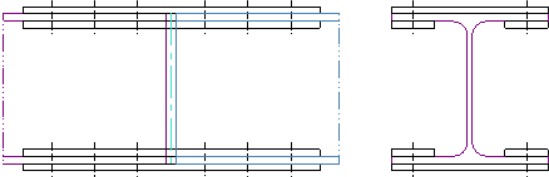
Only flanges are connected

Splice Design

Beam - beam or column - column

HEA240-HEA240, Member gap=15mm
Both external and internal plates on flanges, S235, Pl. thk 10mm, M20 cl.8.8
NO (!) plate on web

SCS



13 End plate column-beam (EM3)

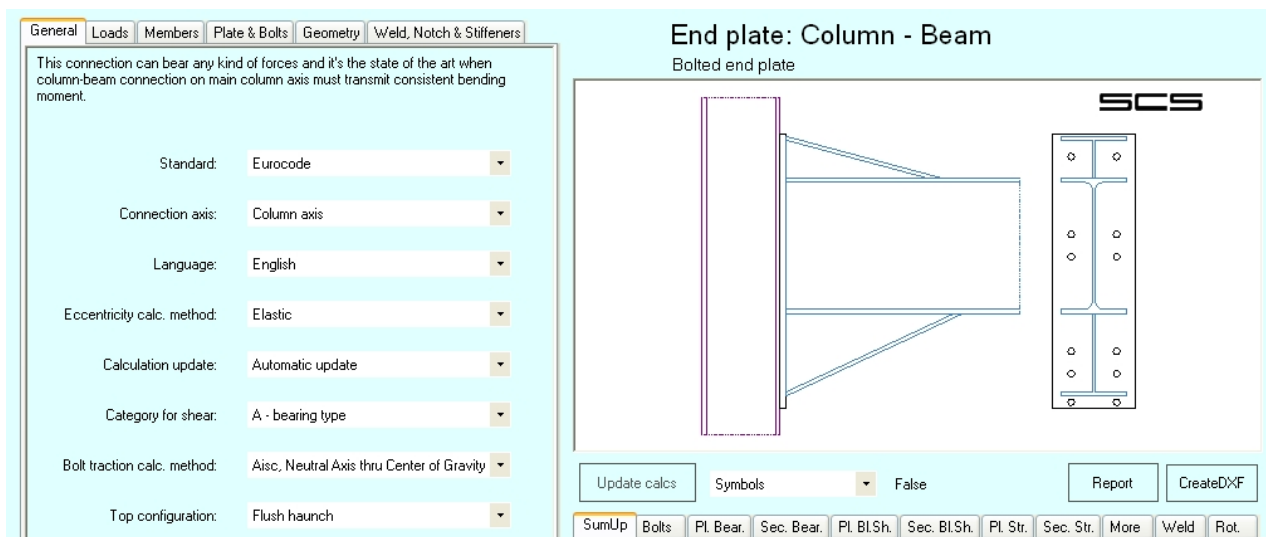
13.1 About EM3

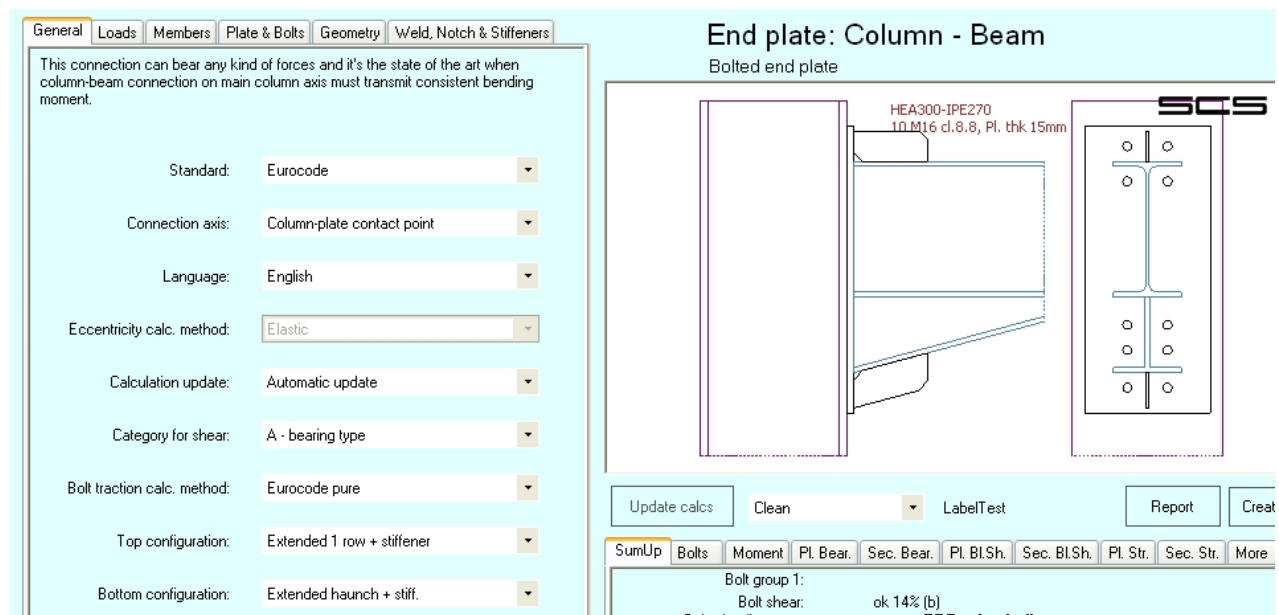
This connection can bear any kind of forces and it's the state of the art when column-beam connection on main column axis must transmit consistent bending moment.

13.2 Top/bottom configuration

You can choose among many different configurations for the top of the connection: flush;

- extended one row (no stiffener);
- extended one row with stiffener;
- extended two rows with stiffener;
- flush haunch;
- extended haunch (no stiffener);
- extended haunch with stiffener;



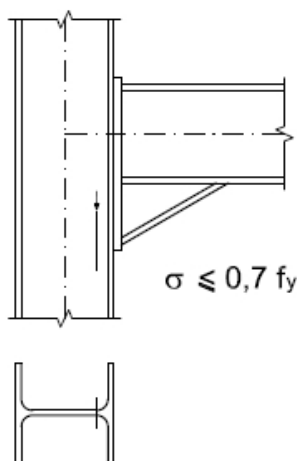


13.3 Loads

Usual considerations apply but if calculation method is AISC Design Guides, a further column appears to define further axial load over the column. This load will add up to the shear transmitted by the beam.

13.3.1 k_{wc}

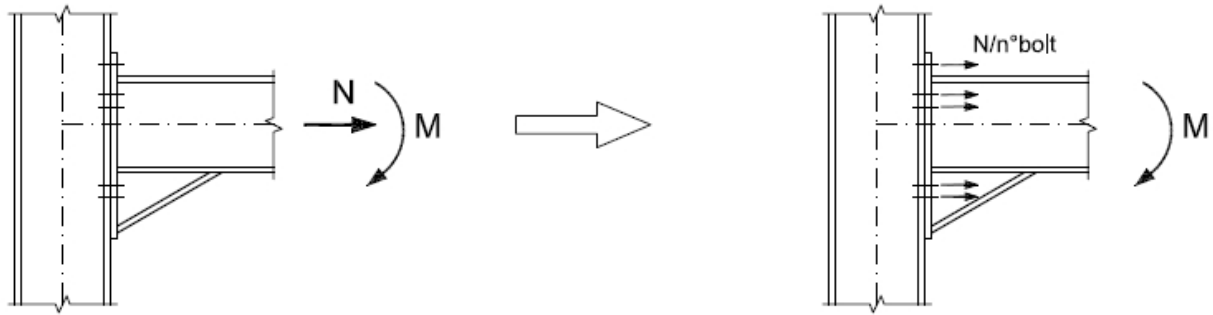
If you chose the calculation according to Eurocode, the program performs the calculations with the fixed value of k_{wc} (set equal to 1 by default). However, SCS gives the user the ability to overwrite this value if appropriate. In this case, the program will perform the calculations with the new value of k_{wc} set by the user (for further details see par. 6.2.6.2.2 from #00 on bibl.).



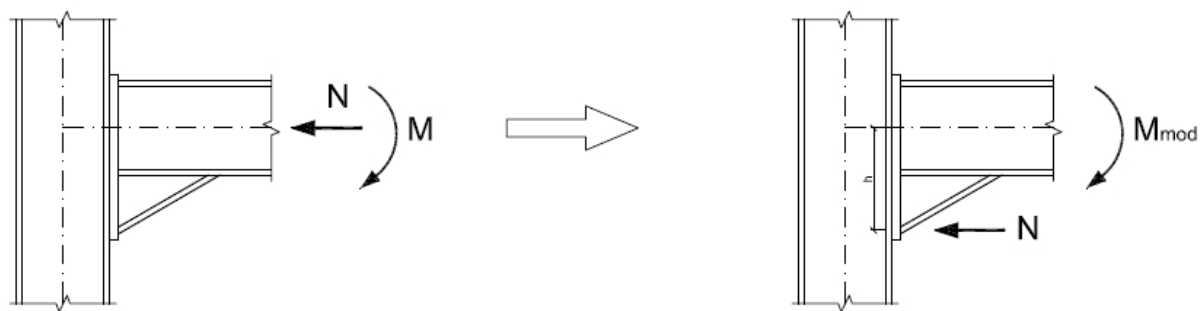
13.3.2 Axial force modifies M_{maj}

☒ Axial force modifies M_{maj}

If there are a moment and a tension force on the beam, for the calculation it is chosen to divide the tension force equally among all the bolts on the end-plate.



If there are a moment and a compression force on the beam, for the calculation it is chosen to move the force to the extreme flange of the connection (haunch flange or beam flange) and modify the moment as consequence $M_{mod} = M - N \cdot h$. (#5 on bibl.).



13.3.3 Inelastic deformations are accepted

☒ Inelastic deformations are accepted

This is active when the *Moment calculation method* is 'Aisc Design Guides'. See [24] for details.

13.4 Members

13.4.1 Supplementary web plates

The user can choose between two different configurations about the supplementary web plate for the column web. It is possible to add one or two web plates, and there are two different types of welds: full penetration butt weld or fillet weld.

To add one supplementary web plate, just put a non-zero width into the specific cell. To add two equal supplementary web plates at the column web, tick the square “Both sides”.

The supplementary web plate on the column has a width equal to the distance between the two toe radii of the column (in accordance with #00 – par. 6.2.6).

The length of the supplementary web plate is taken as the length of secondary member connection (with haunches where present) plus the maximum length between the effective length in tension and the effective length in compression.

For the calculation of the effective lengths in tension and in compression for Eurocode, British Standard and Italian standard (NTC), SCS follows the British Standard (for further information see #00 page 72; #42 page. 37).

N.B.: the drawing of the supplementary web plate is just a general instruction and the engineer will take care of defining details (welding included) properly.

13.4.2 Continuous column in moment connection

You can choose the length of the column over the connection (an infinitive value of b_1 means that the column is not interrupted near the joint).

General		Loads		Members		Plate & Bolts		Geometry		Weld, Notch & Stiffeners	
Primary Member:		HEA160		Material:		S275					
h=	152	mm	f_y =	275	MPa						
b=	160	mm	f_u =	430	MPa						
t_f =	9	mm	thk \leq :	40	mm						
t_w =	6	mm	<input type="checkbox"/> Both sides								
r=	15	mm	Reinf. web pl. t_s =			mm					
b_1 =	∞	mm	<input checked="" type="checkbox"/> Continuous prim.								

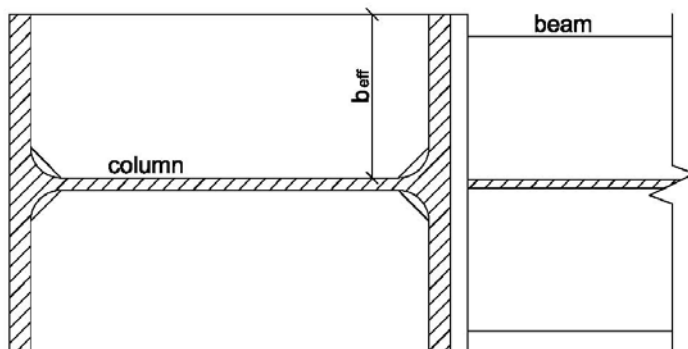
General		Loads		Members		Plate & Bolts		Geometry		Weld, Notch & Stiffeners	
Primary Member:		HEA160		Material:		S275					
h=	152	mm	f_y =	275	MPa						
b=	160	mm	f_u =	430	MPa						
t_f =	9	mm	thk \leq :	40	mm						
t_w =	6	mm	<input type="checkbox"/> Both sides								
r=	15	mm	Reinf. web pl. t_s =			mm					
b_1 =	50	mm	<input type="checkbox"/> Continuous prim.								

13.5 Weld, Notch & Stiffeners

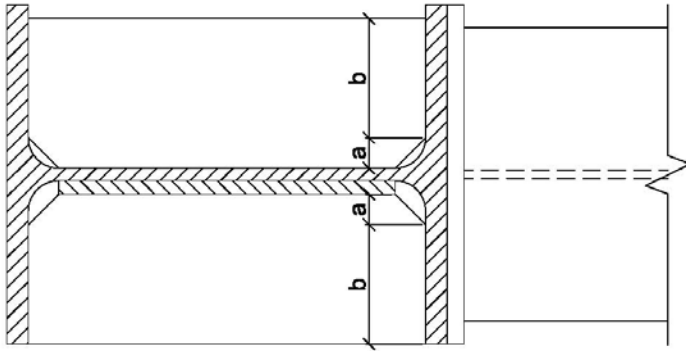
13.5.1 Continuity stiffeners on column

The stiffener on the column have a depth equal to the distance between the edge of the flange and the web of the column, and a scallop equal to the radius of the profile. If the depth is more than $13 \cdot \text{thickness of the plate}$, for the calculation SCS take $b_{\text{eff}} = 13 \cdot \text{plate thk}$.

N.B.: stiffeners dimensions are not defined in detail by SCS; the engineer is responsible of giving the correct instructions to detailers.

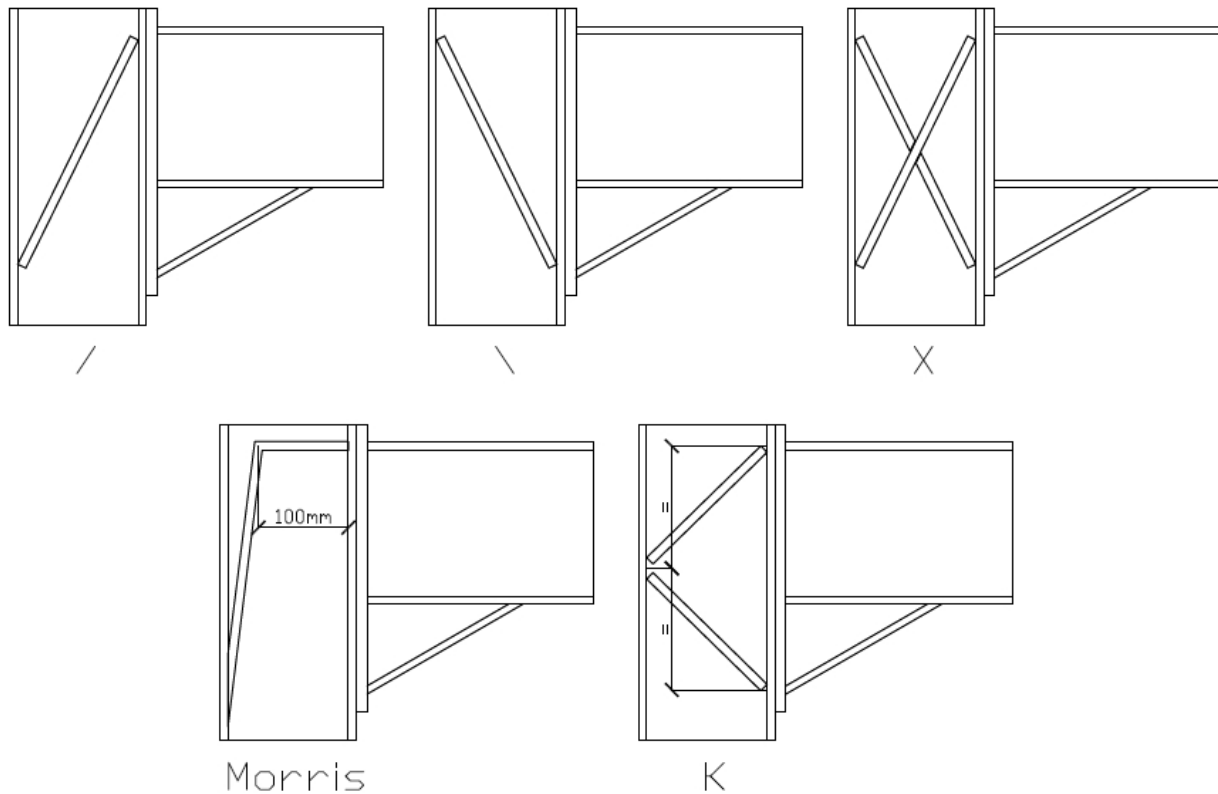


In case of stiffeners and web plate both present on the column, the relevant stiffener is shortened in width where there is the web plate.



13.5.2 Diagonal stiffeners

You can choose among some different types of diagonal stiffener on the column:



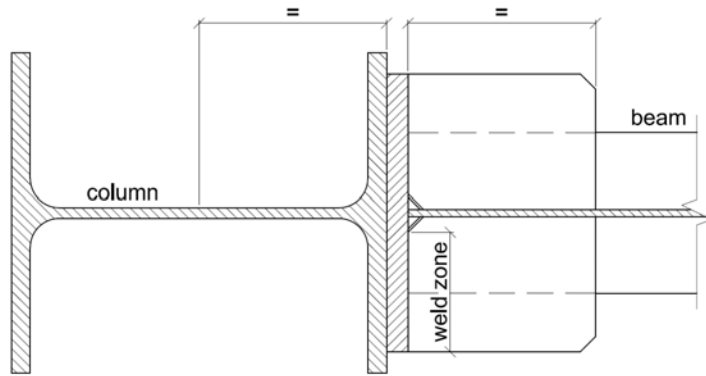
In the calculation of the diagonal stiffener the welds are considered as specified in #42:

“Welds connecting diagonal stiffeners to the column flange should be ‘fill-in’ welds with a sealing run providing a combined throat thickness equal to the thickness of the stiffener...”.

N.B.: as previously seen, it's task of the user to define all the correct detailing of stiffeners.

13.5.3 Intermediate stiffeners on beam

The intermediate stiffener on the beam have a length equal to half the width of the column, and a depth equal to the distance between the edge of the plate and the web of the beam.



13.5.4 Backing plates

Backing plates can be applied to column, It's possible to decide how many bolts to support by the backing plate. Top and bottom backing plates can be different.

14 Apex end plate (AP7)

FileEditToolsDrawing ViewExamplesHelp

General

Loads

Members

Plate & Bolts

Geometry

Weld, Notch & Stiffeners

Unit: kN kN*m

Case	N Ed	Vmaj Ed	Vmin Ed	Mmaj Ed	Mmin Ed	T Ed
a	35			350		
b						
c						
d						
e						

☐ Axial force modifies Mmaj

☐ Optimize Bot Tension+Shear combo

YM0= 1.1

YM1= 1.1

YM2= 1.25

overwrite

Reinforcement zone

Apex end plate: Beam - Beam - AP7

Bolted end plate

HEB340 - HEB340 S275

8 M27 (Ø30mm) cl.8.3, End Plate 500x270x3mm S275

Update values

Dimensions

Report

Short report

Create DXF

SumUp

Bolts

Moment

Pl. Bear.

Welding

Stiff.

Bolt shear: ok 0% (a)

Bolt min. distances: ok

Pl. bearing hor.: ok 0% (a)

Pl. bearing vert.: ok 0% (a)

Bolt tension: ok 78% (a)

Moment cap.: **ERR 1.30 (a)**

Beam f.+web compr.: ok 67% (a)

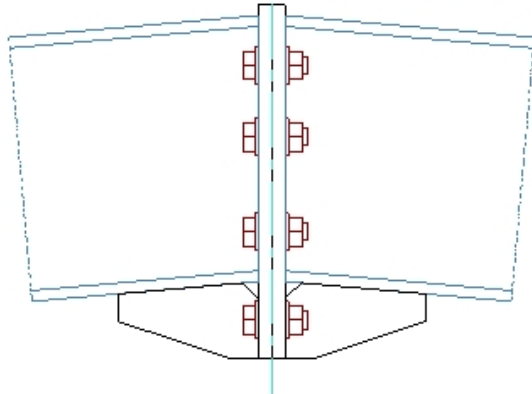
Stiffener min. thk: ok

Bolt combined: ok 37% (e)

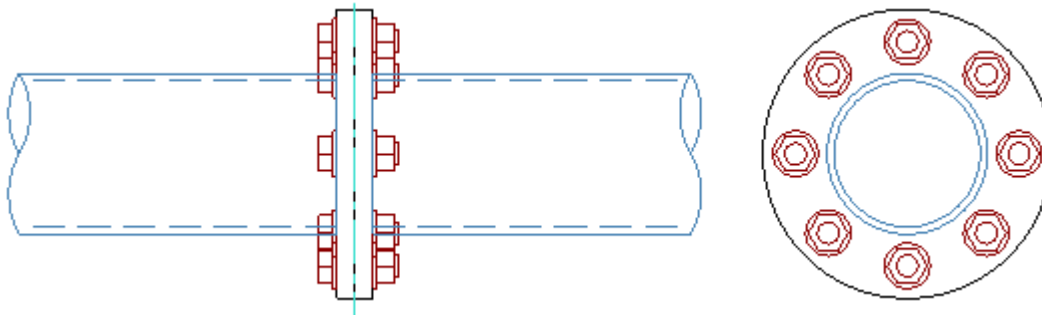
88/166

14.1 About AP7

The module is similar to EM3 but is' about an apex connection between beams of the same size. This connection can bear any kind of forces and it's the state of the art when beam-beam connection on main beam axis must transmit consistent bending moment.



14.2 CHS connections



Module AP7 allows the use of round sections: in this case the connection can be seen as beam to beam or column to column connection.

Connections between CHS elements can be checked according to T-Stub concept or through the method proposed in [41]. The first method is based on a generalization of Eurocode T-Stub concept, both for stiffened or unstiffened configurations. Method as per [41] does not allow the use of stiffeners.

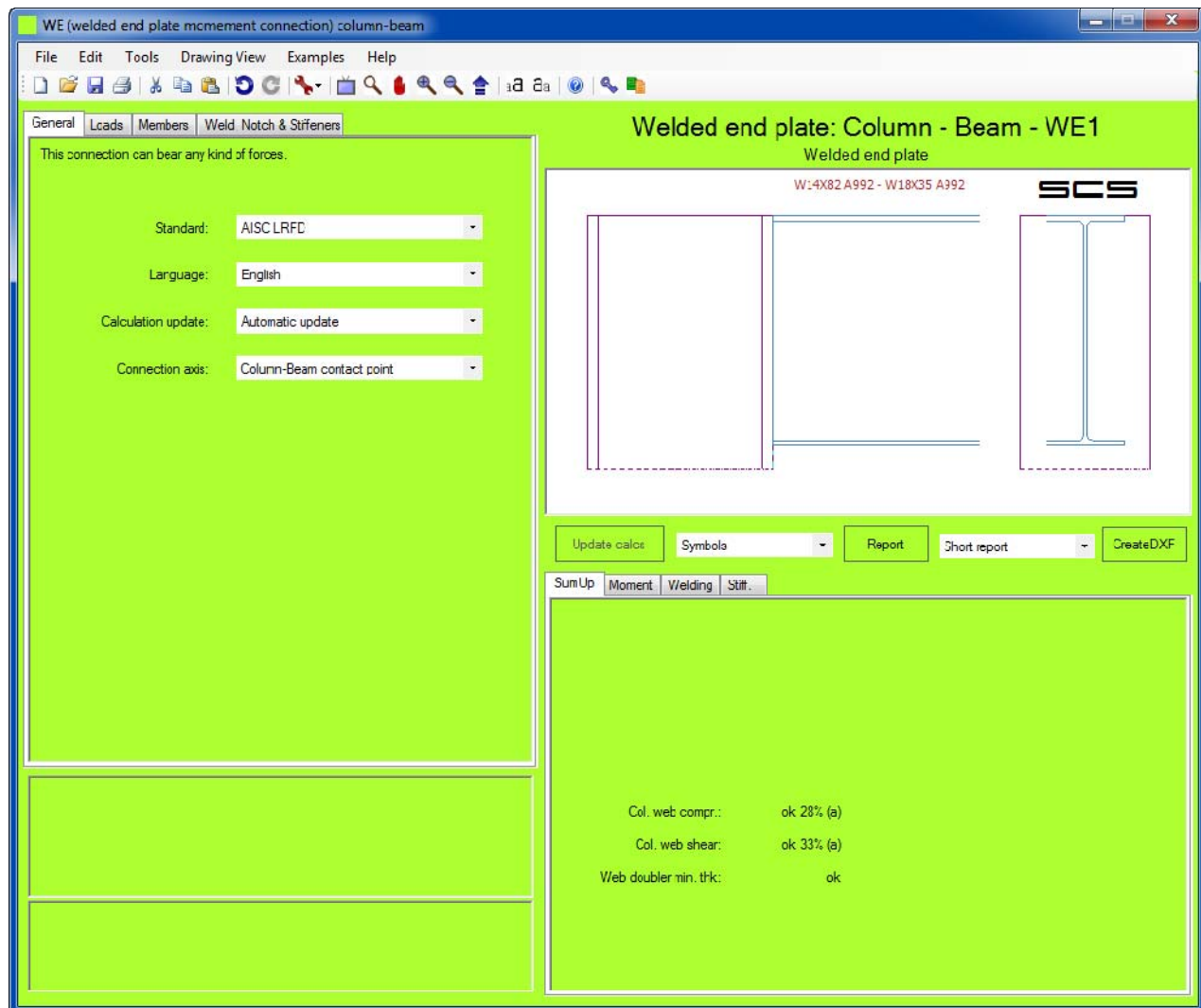
Moment distribution in the bolts is according to the AISC method “neutral axis through center of gravity” (chapter 7 of [26]). Due to double symmetry, strong and weak axis moments are directly combined as vector by SCS and applied to the bolt configuration, defining the corresponding inclined neutral axis.

The effect of compression can be taken into account with the dedicated check box.

15 Welded end plate (WE1)

15.1 About WE1

This module is like EM3 but the beam is directly welded over the column flange.



16 Bolted angles column-beam (QS5)

Bolted angles: Column - Beam - QS5
All bolted double angles (web cleats)

HEA160 S275 - IPE200 S275
3 M16 (w/c 0.17m) cl.6.82 L 0.078x0.07x0.007m S235

Plate & Bolts

bol. columns = 1 bolt rows = 3 total bolts = 3
plate thk = 0.007 m cl. beam = 0.01 m
☒ Plate 'standard' bolt spacing beam clear. from flanges = 0.01 m
P_{horiz.} = 0.051 m ☒ Bolts are vert. sym. in second
P_{vert.} = 0.051 m a_{vert. beam top} = 0.049 m
a_{horiz. pl.} = 0.034 m a_{vert. beam bottom} = 0.049 m
a_{horiz. beam} = 0.034 m ☒ Bolts are vert. sym. in plate
depth_{pl.} = 0.153 m a_{vert. pl. top} = 0.0255 m
width_{angle leg} = 0.078 m a_{vert. pl. bottom} = 0.0255 m
override
☐ bolt gr. ecc. = -0.12 m BoltGroupCenter distance from Conn. Axis
☐ pl. ecc. = -0.12 m Section at bolts
☐ sec. ecc. = -0.154 m Section at end of plate
☐ L_{or} = 0.051 m formula
☐ block shear k_{unif} = 1

Bolt group 2 is the bolt group near the primary member.
bolt columns2 = 1 bolt rows 2 = 3 total bolts 2 = 6
width_{angle leg} = 0.07 m P_{vert. 2} = 51 m
a_{hor. plate} = 0.03 m a_{hor. prim.} = 0.0372 m
☐ bolt gr. ecc. = -0.076 m BoltGroupCenter distance from Conn. Axis

Weld, Notch & Stiffeners

Update calcs Symbols Report Short report CreateDXF

Summary Bolts Pl. Bear. Sec. Bear. Pl. Bl. Sh. Sec. Bl. Sh. Pl. Str. Sec. Str. More

Bolt group 1:

Bolt shear:	ok 13% (a)		
Bolt min. distances:	ok		
Pl. bearing hor.:	ok 10% (b)	Beam bearing hor.:	ok 23% (b)
Pl. bearing vert.:	ok 0% (a)	Beam bearing vert.:	ok 0% (c)
Pl. bl. shear hor.:	ok 14% (b)	Sec., bl. sh. hor.:	ok 28% (b)
Pl., bl. sh. vert.:	ok 0% (a)	Sec., bl. sh. vert.:	ok 0% (e)
Pl., shear strong ax.:	ok 0% (a)	Pl., bend. strong ax.:	ok 0% (a)
Pl., shear weak ax.:	ok 0% (a)	Pl., bending weak ax.:	ok 0% (a)
Pl., axial:	ok 16% (a)	Pl., combined stress:	ok 16% (a)
Beam shear str.:	ok 0% (a)	Beam bending str.:	ok 0% (e)
Beam shear wk.:	ok 0% (a)	Beam bending wk.:	ok 0% (e)
Beam axial:	ok 5% (a)	Beam combined:	ok 5% (e)
Plate buckling:	ok 0% (a)	Prim. local web shear:	ok 0% (a)

Bolt group 2:

Bolt shear:	ok 0% (a)
Bolt min. distances:	ok

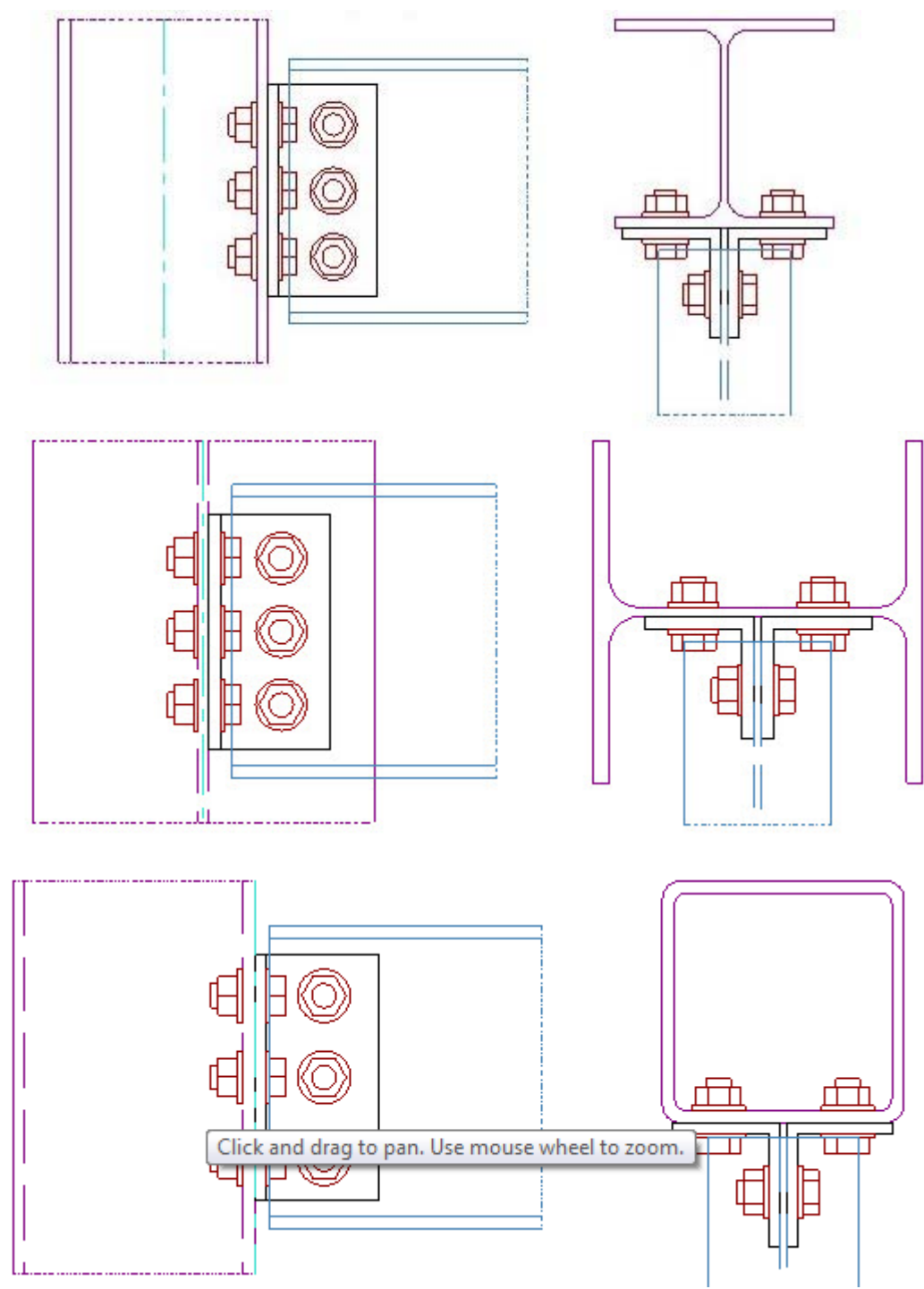
16.1 About QS5

This connection is also known as cleats.

These connections can bear axial and shear forces between beam and column. Shear and axial are both transmitted by the angles bolted to primary member (column) and bolted to secondary member web (beam).

It's not possible yet to design the connection with only one angle (on one side).

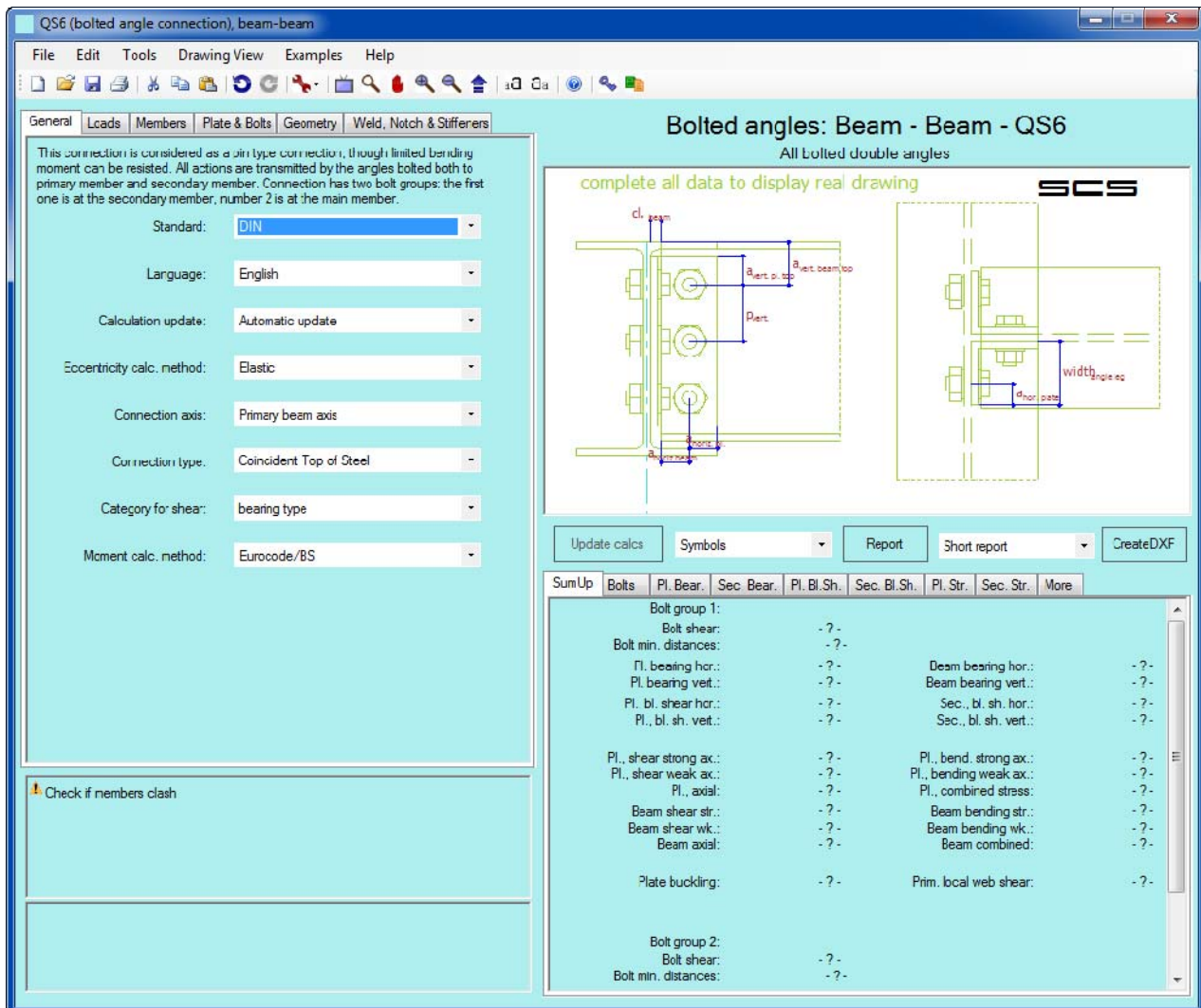
16.2 Connection type



17 Bolted angles beam-beam (QS6)

17.1 About QS6

The connection is same as QS5 but it's about the connection between beams (primary and secondary).



18 Flexible end plate column-beam (XL1)

Flexible End Plate Design: Column - Beam - XL1
Flexible end plate welded to secondary beam, bolted to primary

complete all data to display real drawing

SCS

Update calcs Symbols Report Short report CreateDXF

SumUp	Bolts	Pl. Bear.	Prim. Bear.	Pl. Bl. Sh.	Pl. Str.	Sec. Str.	More	Welding
Bolt shear:	-? -							
Bolt min. distances:	-? -							
Pl. bearing hor.:	-? -					Beam bearing hor.:	-? -	
Pl. bearing vert.:	-? -					Beam bearing vert.:	-? -	
Pl., bl. sh. vert.:	-? -							
Pl., shear strong ax.:	-? -							
Pl., shear weak ax.:	-? -							
Beam shear str.:	-? -					Beam bending str.:	-? -	
Beam shear wk.:	-? -					Beam bending wk.:	-? -	
Beam axial:	-? -					Beam combined:	-? -	
Web weld combined:	-? -					Prim. local web shear:	-? -	
						Web weld σ:	-? -	
Bolt tension:	-? -					Bolt combined:	-? -	
Moment cap.:	-? -							

18.1 About XL1

These connections can bear axial and shear forces between beam and column. Shear and axial are both transmitted by the flexible plate bolted to primary member (column) and welded to secondary member (beam).

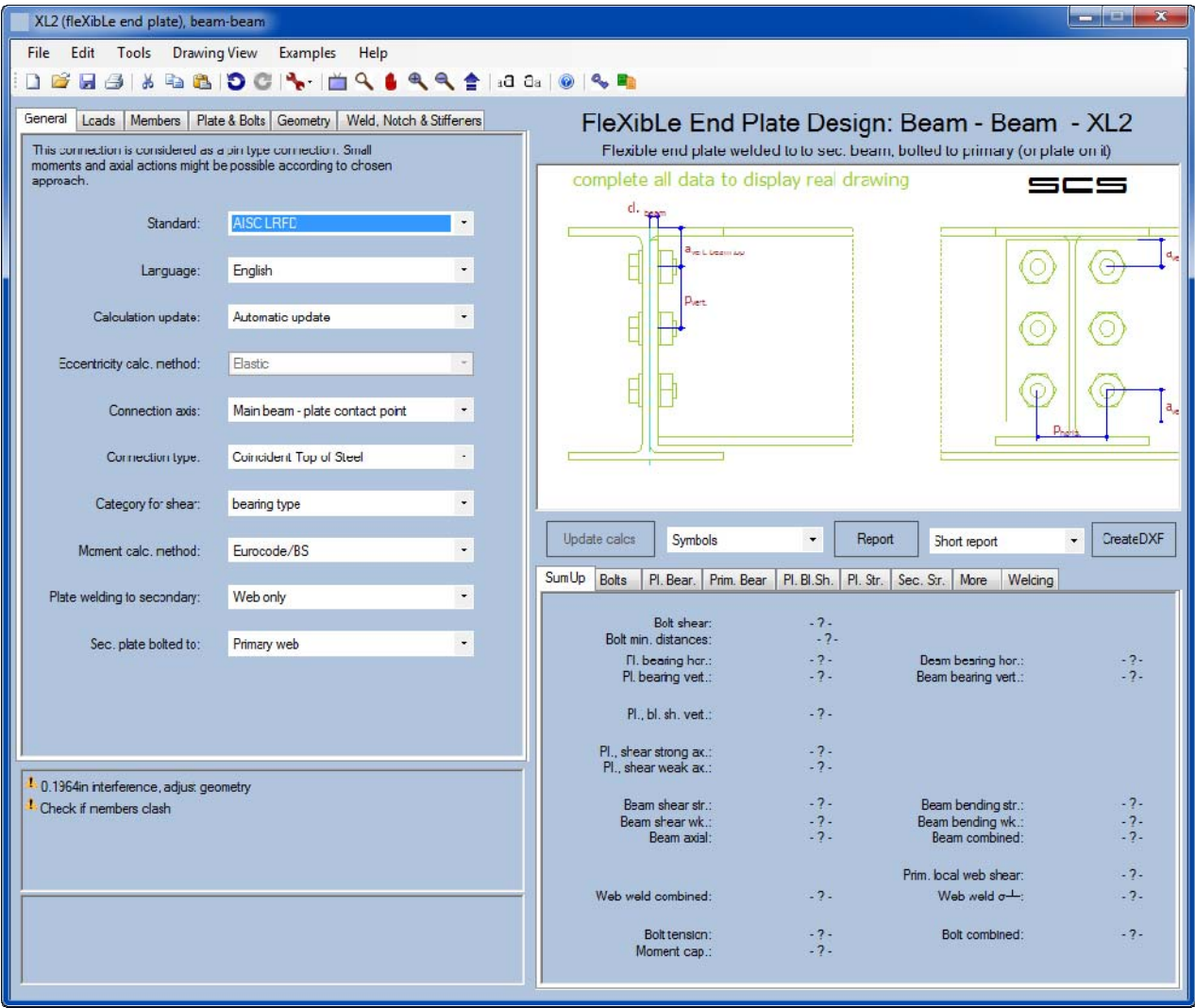
18.2 Plate welded to secondary

You can choose between two different configurations:

- Flexible end plate welded only on beam web;
- Flexible end plate welded on beam web and beam flanges.

It should be noted that welding to flanges might increase resistance but might also increase rigidity, invalidating pin assumption according to some literature.

19 Flexible end plate beam-beam (XL2)



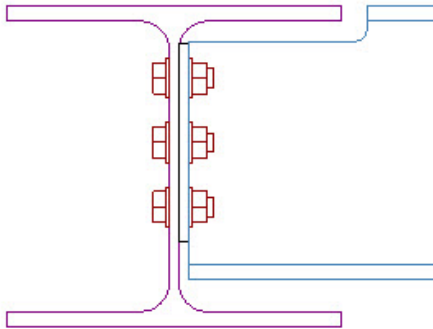
19.1 About XL2

This module is similar to XL1 but it deals with a beam-beam connection.

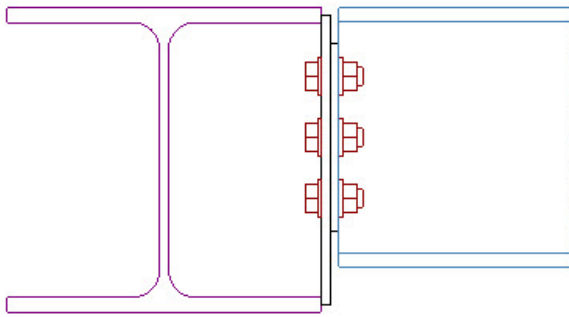
19.2 Secondary plate bolted to

You can choose between two different configurations:

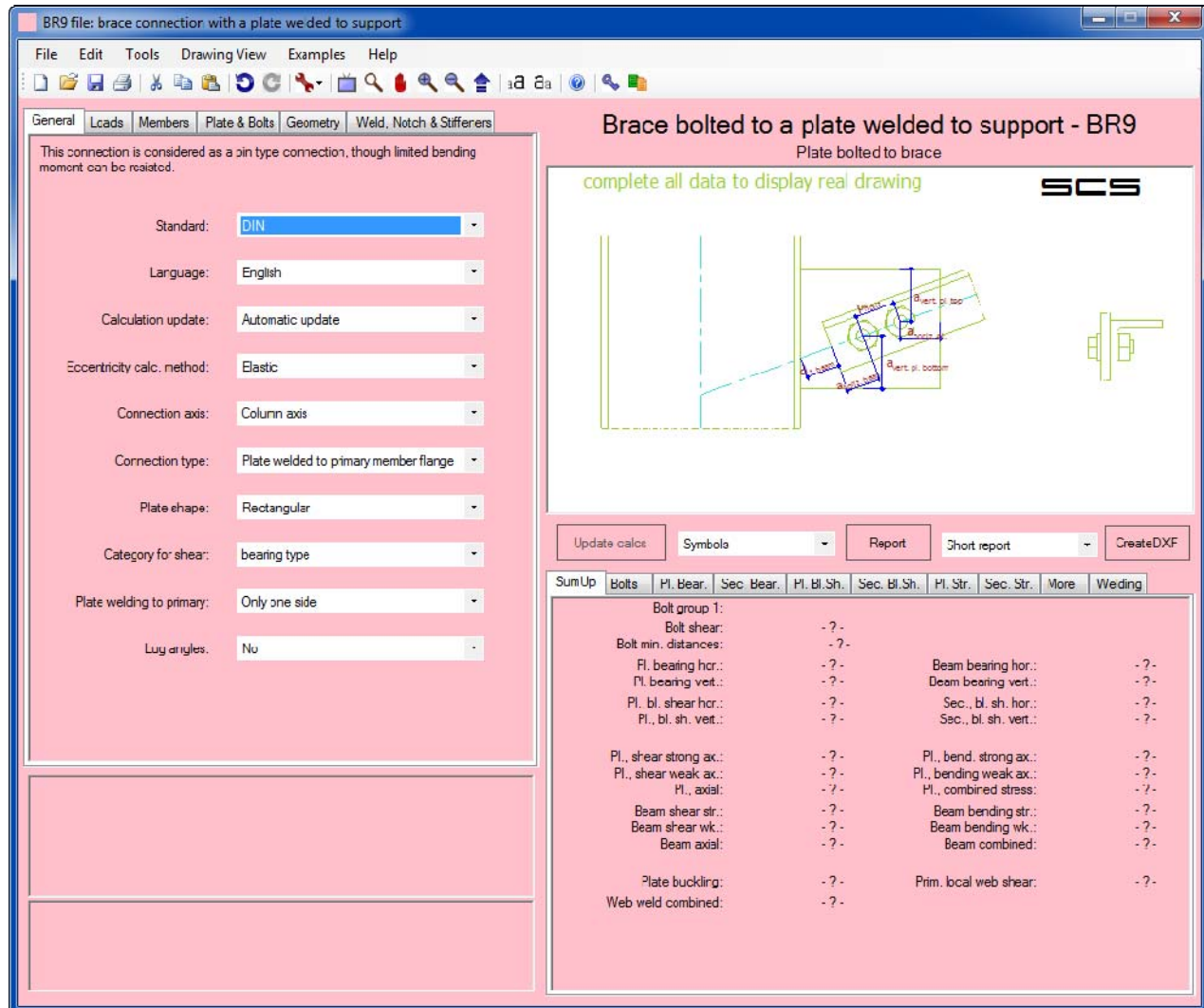
- Flexible end plate bolted to primary member web;



Flexible end plate bolted to a plate welded to the primary member flanges.



20 Brace bolted to a plate welded to support (BR9)



20.1 About BR9

This connection can bear mainly axial forces and it's the state of the art for the bracing connection.

20.2 Plate shape

You can choose among four different plate shapes for the connection:

- rectangular;
- rectangular with front cuts;
- rectangular with side cuts;
- rectangular inclined.

After these plate shape selection, the User can set the geometry of the plate in the tab *Geometry*.

20.3 Plate welding to primary

Plate welding to primary:

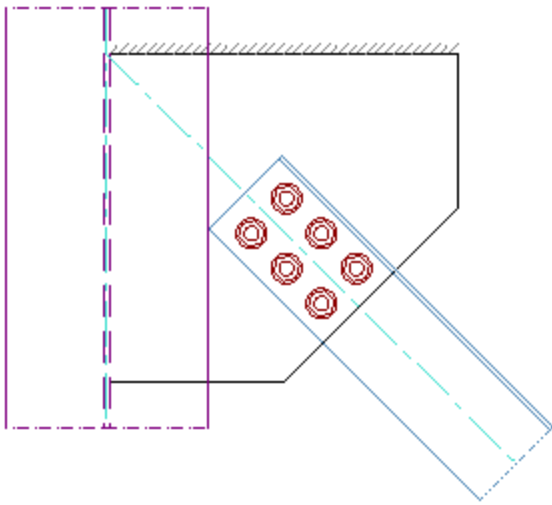
Two sides

You can choose between two different configurations for the connection:

- only one side;
- two sides.

In the first case, the plate is welded only to the primary member (web or flange depending by the connection type choice).

In the second case, the plate is also welded to a support. The support is represented by a line as in figure below.



20.4 Lug angle

If the brace shape is L, UPN, the user can connect the brace adding also a lug angle.

20.5 Force distribution method: Uniform force method (UFM), KISS method

When the connection is on two sides, the 'force distribution method' option becomes visible.

Plate welding to primary:

Two sides

Force distribution method:

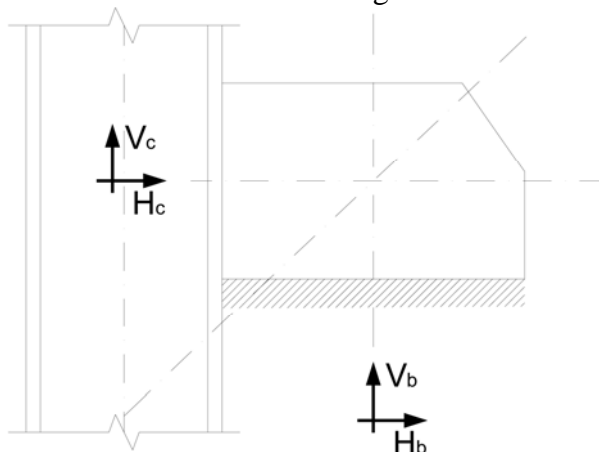
Uniform Force Method (share moment)

This option allows the user to choose with method will be adopted by SCS to calculate the forces acting on the vertical and horizontal (considered with $\alpha=0^\circ$)

Uniform Force Method (share moment)
KISS Method
Only force components
UFM, moment to vertical weld
UFM, moment to horizontal weld

The methods follow instructions in #26 (AISC Design Manual), please refer to it (chapter 13) for details. Broadly speaking:

- UFM (first option) assigns to vertical and horizontal welds only axial and shear actions if the geometry is as requested by UFM method, that is axis from welds meet on bracing axis; if this doesn't happen, moment is shared between horizontal and vertical weld; 4th and 5th options follow the same method but gives moment to vertical or horizontal weld respectively.



- KISS method (keep it simple, stupid), gives vertical component of brace force to vertical weld and horizontal component to horizontal weld but it adds a moment to the weld given by multiplying the force on the weld for the eccentricity distance.
 - the third option, only gives vertical component of brace force to vertical weld and horizontal component to horizontal weld; it can be considered like UFM Special Case 1 in #26.

The forces calculated by UFM can be found in the 'Geometry' tab:

Case	Hc	Vc	Hb	Vb
<i>a</i>	70.18	70.88	107.38	31.58
<i>b</i>				
<i>c</i>				
<i>d</i>				
<i>e</i>				

If one of the two connection is not a weld, the user can find the force here and set up another design check in a different module (fin plate, flexible end plate, bolted cleats....).

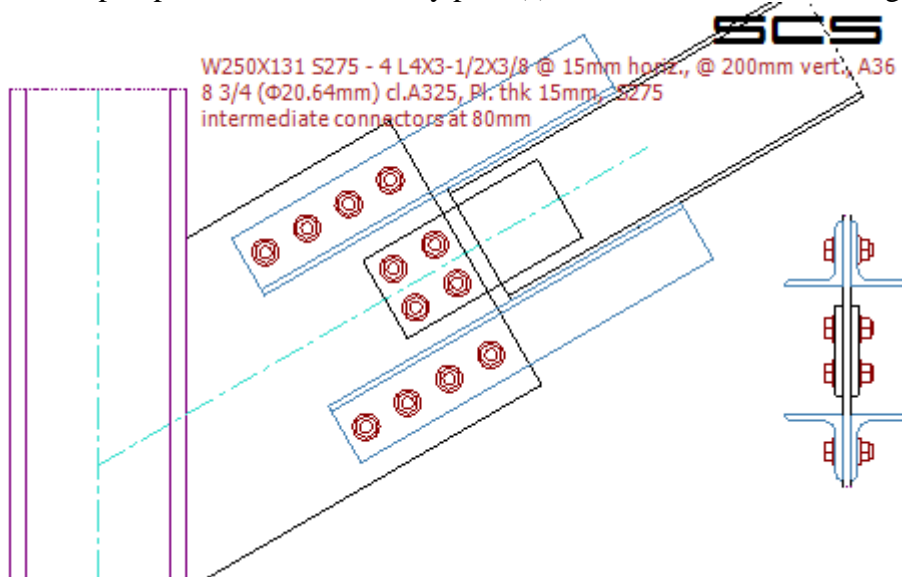
20.6 Members

20.6.1 Secondary member

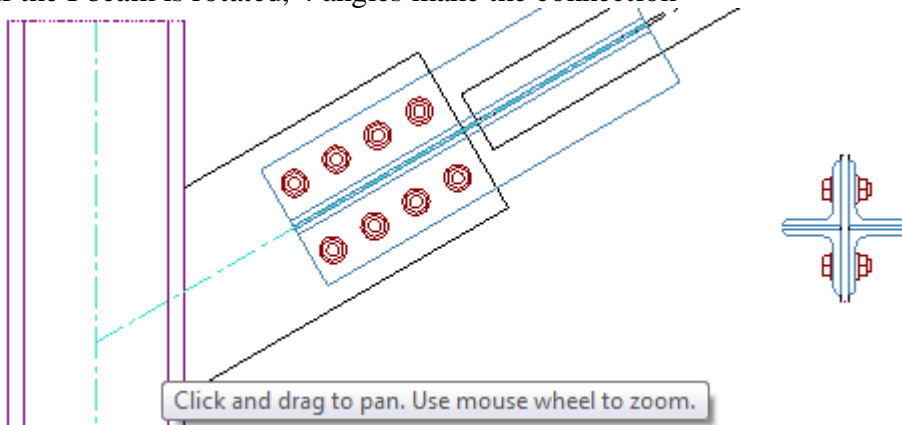
Here you can choose the profile shape for the brace, that it could be a L, a channel, an I, an RHS, a tube or a plate.

If you choose a L, a channel or a plate for the shape of the brace, you can also choose whether the selected profile is double or single.

An I shaped profile is connected by plate(s) on the web and double angles on the flanges.

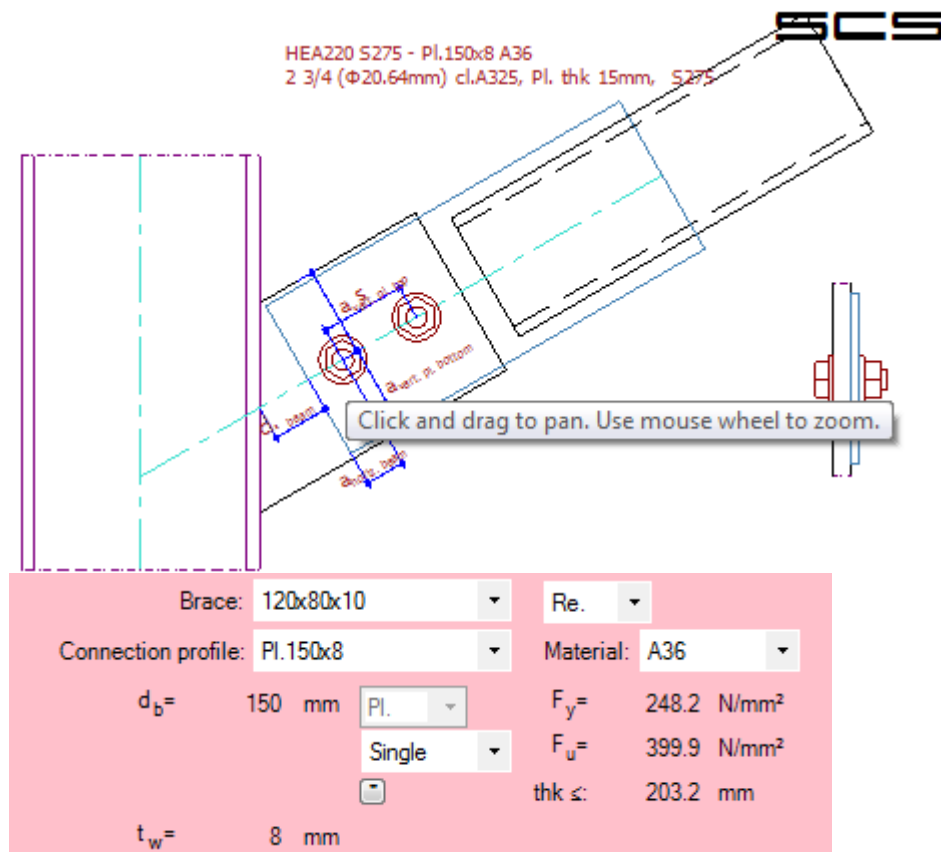


If the I beam is rotated, 4 angles make the connection



Brace:	IPE240	I	<input type="checkbox"/> Rotate 90°
Connection profile:	L4X3-1/2X3/8	Material:	A36
$d_b =$	101.6 mm	$F_y =$	248.2 N/mm ²
$b_f =$	88.9 mm	$F_u =$	399.9 N/mm ²
	<input type="button" value="L"/> <input type="button" value="Quadruple"/>	thk ≤:	203.2 mm
$t_w =$	9.53 mm	<input type="checkbox"/> Switch legs	
$k-b_f =$	19.05 mm	Reinf. web pl. $t_s =$	0 mm

SCS currently does not take care of verifying the connection between angles and the I brace, therefore the user must take care of that.



Tubulars are connected by plates. SCS currently does not take care of designing the weld between the plate and the pipe/RHS, therefore the engineer must take care of that.

In *Tools\Member filter* from the toolbar, you can filter the available elements for the brace.

Select Profile

Shape I

<input checked="" type="checkbox"/> IPE	<input checked="" type="checkbox"/> UC	<input checked="" type="checkbox"/> ISJB, ISLB, ISMB	<input checked="" type="checkbox"/> Custom I 2
<input checked="" type="checkbox"/> HEA, HEB	<input checked="" type="checkbox"/> UB	<input type="checkbox"/> Austr. WB, WC	<input type="checkbox"/> Custom I 3
<input checked="" type="checkbox"/> W	<input type="checkbox"/> Austr. UB, UC	<input type="checkbox"/> I, NP	<input type="checkbox"/> Custom I 4
<input checked="" type="checkbox"/> W metr.	<input checked="" type="checkbox"/> HM, HW, HN	<input type="checkbox"/> HE, HEC, HSL, HL	<input type="checkbox"/> Custom I 5
<input checked="" type="checkbox"/> HP	<input checked="" type="checkbox"/> HEAA (HLS), HEM	<input type="checkbox"/> M, S	<input type="checkbox"/> WWF
<input type="checkbox"/> HP metr.	<input type="checkbox"/> IPEa (ILS), IPEo, IPEv	<input checked="" type="checkbox"/> Custom I 1	

Shape L

☒ L (AISC)

☒ L equal metr.

☒ L unequal metr.

☒ Custom L 1

☐ Custom L 2

☐ Custom L 3

Shape U

<input checked="" type="checkbox"/> U (UPN)	<input type="checkbox"/> ISJC, ISLC, ISMK
<input checked="" type="checkbox"/> UAP, UPE	<input checked="" type="checkbox"/> Custom U 1
<input checked="" type="checkbox"/> C, MC	<input type="checkbox"/> Custom U 2
<input type="checkbox"/> JIS, GB-C, YB-C	<input type="checkbox"/> Custom U 3
<input type="checkbox"/> PFC	
<input checked="" type="checkbox"/> UAP, UPE	

Shape Re

☒ HSS (RHS, SHS)

☒ RHS, SHS metric

☒ Custom RHS/SH

☒ Custom RHS/SH

☒ Custom RHS/SH

Shape PI

☐ Imperial bars/PI.

☒ PI.

☒ Custom PI. 1

☐ Custom PI. 2

☐ Custom PI. 3

Shape Ro

☒ HSS (Round), PI

☒ Pipe metric

☒ Custom Pipe 1

☐ Custom Pipe 2

☐ Custom Pipe 3

Save and Close

Don't save and Close

21 Column base plate and anchors (CB8)

Column base plate and anchors - CB8
Base plate welded to primary, anchor bolted to concrete

HEA400 S275 - Concrete $f_{ck}=21\text{N/mm}^2$
6 M27 ($\Phi 30\text{mm}$) cl. 6.8 L=700mm, Base Plate 630x520x30mm S275

General | **Loads** | **Members** | **Plate & Bolts** | **Geometry** | **Weld, Notch & Stiffeners**

Bolt size: M27 **Class:** 6.8

$d = 27\text{ mm}$ $f_{yb} = 480\text{ N/mm}^2$
 $l_{\text{anchor}} = 700\text{ mm}$ $f_{ub} = 600\text{ N/mm}^2$
 $d_0 = 30\text{ mm}$ $\alpha_v = 0.5$

Plate material: S275
 $f_{yp} = 265\text{ N/mm}^2$
 $f_{up} = 410\text{ N/mm}^2$
 $\text{thk}_p = 40\text{ mm}$

Conn. type: threads in shear plane
 Shear planes = 1 Shear cross section per bolt = 459 mm²

Conc. base: Conservative: same as base pl.
 $w_f = \text{mm}$
 $h_f = \text{mm}$
 $s_f = \text{mm}$

Overwrite:
 $\beta_j = 0.667$
 $f_{ok} = 21\text{ N/mm}^2$ $\text{thk}_{\text{grout}} = 50\text{ mm}$
 $E_{cm} = 2.058\text{E}+04\text{ N/mm}^2$ $C_{fd} = 0.2$
 $\alpha_{oc} = .85$

⚠ According to EC, external distance between bolts should not be larger than beam width

Results Summary:

Category	Value	Status
Bolt shear:	ok 65% (a)	ok
Bolt min. distances:	ok	ok
Pl. bearing hor.:	ok 18% (a)	ok
Pl. bearing vert.:	ok 5% (a)	ok
Bolt tension:	ERR 1.23 (b)	ERR
Moment cap.:	ERR 1.23 (a)	ERR
Beam fl.+web compr.:	ok 34% (a)	ok
Pl., shear strong ax.:	ok 6% (a)	ok
Pl., shear weak ax.:	ok 24% (a)	ok
Bolt combined:	ERR 1.02 (b)	ERR
Friction:	ok	ok
Conc. pressure:	ERR 1.03 (a)	ERR

21.1 About CB8

This connection can bear any kind of forces and it's the state of the art when column is based on a concrete foundation.

21.2 Shear force

You can choose the elements that resist shear force among these choices:

- Friction and anchor bolts;
- Shear key and frictions;
- Friction only;
- Shear key only;
- Anchor bolts only.

21.3 Base plate calculation method

You can choose between two different configurations for connection:

- Eurocode #01;
- British Code #42;
- AISC design guides #22.

21.4 Top configuration

Here you can choose the position of the anchors on the “top” of the base plate compared to the “top flange” of the column:

- External anchors;
- Internal anchors;
- Both external and internal anchors.

21.5 Bottom configuration

Here you can choose the position of the anchors on the “bottom” of the base plate compared to the “bottom flange” of the column:

- External anchors;
- Internal anchors;
- Both external and internal anchors.

21.6 T-stub prying force

The user can decide how to consider T-stub prying force if relevant with the selected standard:

- always;
- never;
- worse condition;
- according to bolt length.

For base plate connection, it's suggested to choose “worse” or, in case, “according to bolt length” or “never”. It's not suggested to choose “always”.

21.7 Plate & Bolts

In this tab the user can choose the characteristics of the concrete foundation and also set the geometrical data of the concrete foundation:

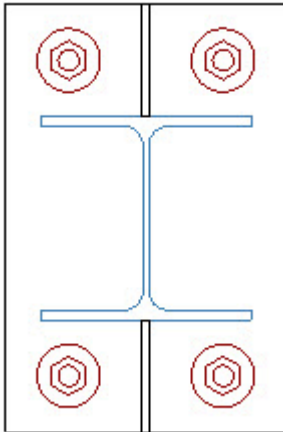
- input detail of concrete base;
- conservative: same dimension as base plate ($\sqrt{A_{\text{conc}}/A_{\text{base plate}}}= 1$);
- no limits: use optimum values ($\sqrt{A_{\text{conc}}/A_{\text{base plate}}}= 3$);
- intermediate value between cases “Conservative” and “No limits” ($\sqrt{A_{\text{conc}}/A_{\text{base plate}}}= 2$).

Conc. base: <input type="text" value="Input details of concrete base"/>		$w_f =$ <input type="text" value="600"/> mm
$\sqrt{(A_{conc}/A_{plate})} =$ <input type="text" value="1.06"/>		$h_f =$ <input type="text" value="600"/> mm
overwrite <input type="checkbox"/>	$\beta_j =$ <input type="text" value="0.667"/>	$s_f =$ <input type="text" value="500"/> mm
	$f_{ck} =$ <input type="text" value="30"/> MPa	$thk_{grout} =$ <input type="text" value="30"/> mm
<input type="checkbox"/>	$E_{cm} =$ <input type="text" value="3.284E+04"/> MPa	$C_{fd} =$ <input type="text" value="0.2"/>
<input checked="" type="checkbox"/>	$\alpha_{cc} =$ <input type="text" value="0.85"/>	

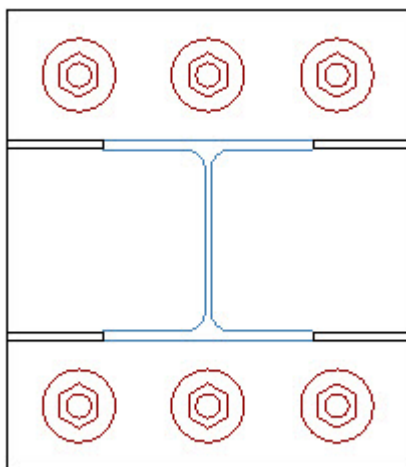
21.8 Geometry

Here you can choose the stiffeners of the base plate, for the strong and weak axis:

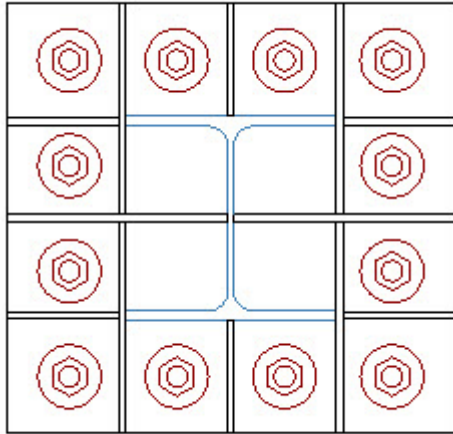
- none;
- central stiffeners;
- external stiffeners;
- both central and external stiffeners.



Example of central stiffeners on the strong side.

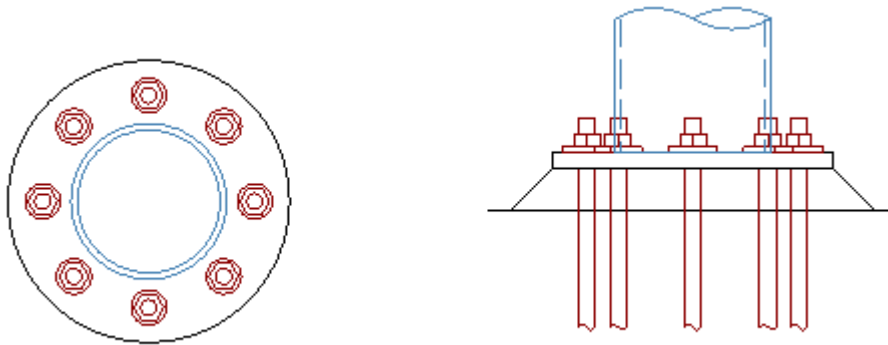


Example of external stiffeners on the weak side.



Example with both stiffeners on the strong and weak axes.

21.9 Circular hollow section connections

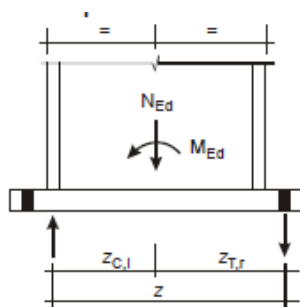


Module CB8 allows the use of round sections from version 17.4 on.

Capacity of base plate side in tension can be checked according to T-Stub concept or through the method proposed in [41]. The first method is based on a generalization of Eurocode T-Stub concept, both for stiffened or unstiffened configurations. Method as per [41] does not allow the use of stiffeners.

Capacity of base plate side in compression is based on Eurocode approach of T-Stub in compression (chapter 6.2.5 of [00]), so the compressed area of concrete depends on diffusion in the plate thickness. When the stiffeners are present, the user can choose to increase the concrete compressed area. Maximum compressed area is equal to steel plate area. When the user considers the diffusion in concrete foundation depth, a vertical conical diffusion is considered, which rules are compliant with the chosen Standard.

As per Eurocode approach (chapter 6.2.8 of [00]), acting moment is distributed in both compression and tensile sides, considering arms (z_T and z_C) from the neutral axis (that always intersects the center of the column) to the centers of gravity of each side.



Due to double symmetry, strong and weak axis moments are directly combined as vector by SCS and applied to the bolt configuration, defining the corresponding inclined neutral axis.

22 General check (ME0)

22.1 About ME0

This module can verify members, bolts and welds generally.

22.2 General

Here the user can choose which kind of elements to verify: any combination of members, bolts and welds is allowed.

☒ Member ☒ Bolts ☒ Welds

22.3 Loads

V_{\max} (2nd column of the load table) is the shear force applied along the Y axis;

M_{\max} (4th column) is the bending moment applied around the X axis;

V_{\min} (3rd column) is the shear force applied along the X axis;

M_{\min} (5th column) is the bending moment applied around the Y axis;

Axial force (1st column) positive is tension.

The last column represents the torsion in the plane XY

22.4 Graphics

x' and y' represented in the graphic window are the main inertia axes of configuration of bolts, weld and non-standard members built by the user.

22.5 Members

You can choose between two different configurations for connection:

- member from database;
- non standard member (elastic analysis).

The database includes a large number of profiles and shapes and the possibility to add profiles by the customer (in the toolbar *Edit Database\Edit Member DB*).

For a *Non standard member*, only strength limit state are considered for member checks (for example, lateral/torsional buckling aren't considered). The user has to insert the geometrical data of the member in the table data as a sum of rectangles, where for each rectangle you have to fill the row of the table data with:

- x and y coordinates of start point;
- size (width) of rectangle;
- length of rectangle;
- angle of rectangle in the xy plane;

General

Loads

Members

Plate & Bolts

Welds

Non-standard member (elastic analysis)

Only strength limit states are considered for member checks (e.g.: lateral/torsional buckling not considered).

Material: S235

$f_y =$

?

N/mm²

$f_u =$

?

N/mm²

Unit: mm

thk \leq

40

mm

	Pos. X	Pos. Y	Size	Length	Angle
*					

22.6 Bolts

Here you can choose the bolt geometry and the material, and for each bolt the coordinates of the centre into the data tab.

General

Loads

Members

Plate & Bolts

Welds

Bolts are considered as working in compression too.

Bolt size:

Class:

d =

?

mm

$f_{yb} =$

?

N/mm²

$f_{ub} =$

?

N/mm²

$\alpha_v =$

?

A =

?

mm²

A_s =

?

mm²

n° pl.:

1

Conn. type: threads in shear plane

Shear planes = 1

Shear cross section per bolt = ? mm²

Unit: mm

Bolt setup

With the button 'Bolt setup' the coordinates of a regular pattern of bolts can be set up automatically. This feature can be used as a starting point, then coordinates can be changed to arrange an irregular pattern.

SetUp form

I strong=

bolt columns =

3

bolt rows =

3

P horiz. =

50

mm

P vert. =

60

mm

0

°

Update and Close

Don't update and Close

22.7 Welds

The user can insert data welds as rectangles filling in the table (1 row, 1 rectangle):

- x and y coordinates of start point;
- size (width) of rectangle;
- length of rectangle;
- angle of rectangle in the xy plane;

General

Loads

Members

Plate & Bolts

Welds

Welding between fin plate and primary member:

f_{u,weld} =

N/mm²

overwrite

β_w =

0.8

Unit: mm

☐

α_{throat} =

45 °

0 mm

	Pos. X	Pos. Y	Size (throat)	Length	Angle
*					

23 Database editing

The database can be edited through the forms opened by clicking



and choosing which database to edit.

The database is stored in the db.mdb (Microsoft Access) file inside [installation folder]/DB but the user MUST NOT edit the access file. A copy of the DB given by Steel Studio is stored under [installation folder]/DB/dbcopy.

23.1 Input for bolt size db: Agr and Ares

Please consider the following information when you input new bolt sizes:

	Bolt	Visible	d [in]	Agr [in²]	Ares [in²]
▶	M8	<input checked="" type="checkbox"/>	0.315	1.0775	2.0558

1) the Agr field is to input the gross area of the bolt. If no value is input, SCS will automatically calculate it according to the formula

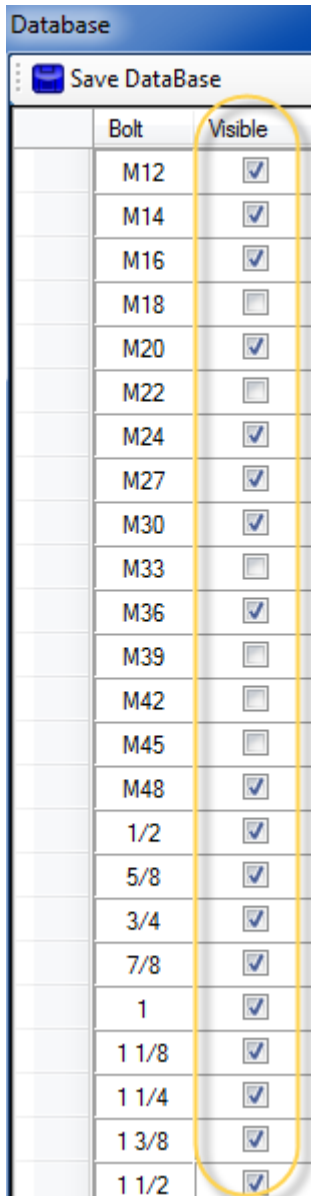
$$A_{gr} = \pi d^2 / 4$$

2) Ares field is to input the net SHEAR area of the bolt. This value must always be input because some codes require it as a specific bolt property. Some other code (AISC for example) will automatically calculate it (as Agr/1.25) but, again, the db requires one load to be input here.

23.2 *How to control visibility*

SCS has 4 different databases, actually stored in one .mdb file.

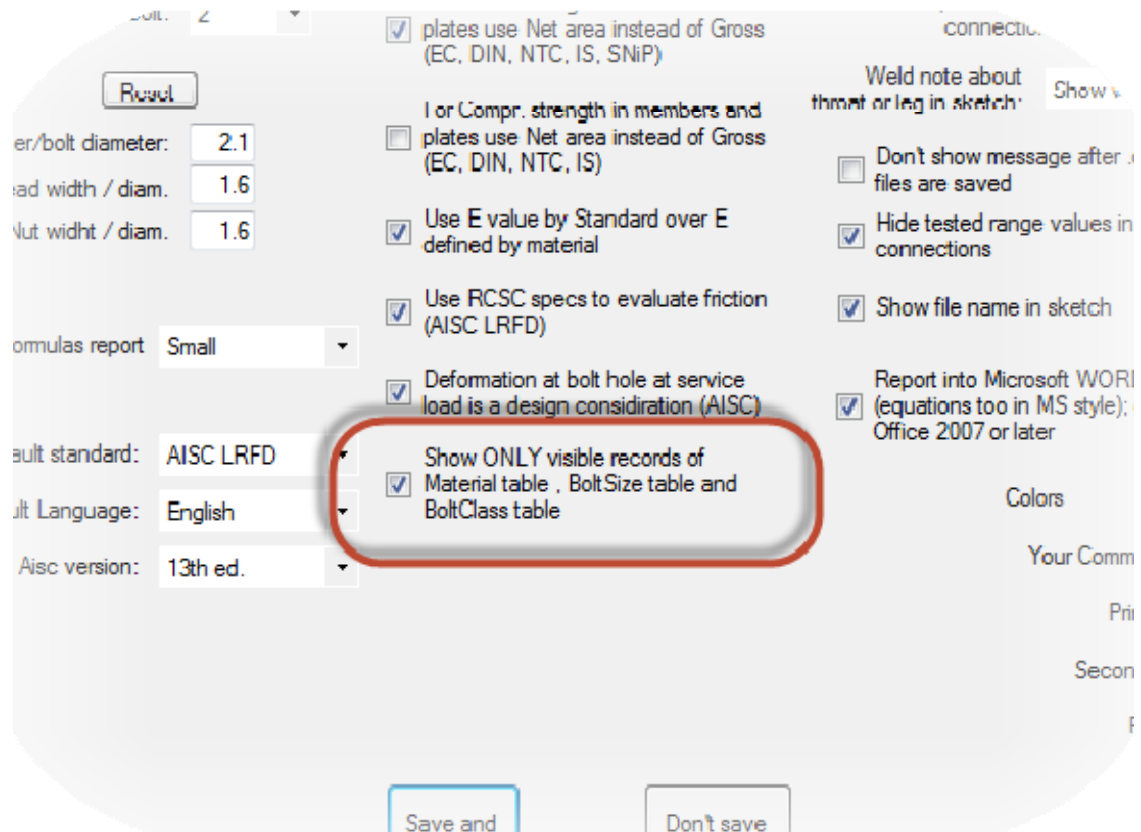
While visibility of the 'Member' db can be controlled through the 'Member filter' option, the visibility of records in the other databases can be controlled through the 'visibility' checkbox.



The screenshot shows a window titled 'Database' with a 'Save DataBase' button. Below is a table with two columns: 'Bolt' and 'Visible'. The 'Visible' column contains checkboxes for each row. A yellow oval highlights the 'Visible' column.

Bolt	Visible
M12	<input checked="" type="checkbox"/>
M14	<input checked="" type="checkbox"/>
M16	<input checked="" type="checkbox"/>
M18	<input type="checkbox"/>
M20	<input checked="" type="checkbox"/>
M22	<input type="checkbox"/>
M24	<input checked="" type="checkbox"/>
M27	<input checked="" type="checkbox"/>
M30	<input checked="" type="checkbox"/>
M33	<input type="checkbox"/>
M36	<input checked="" type="checkbox"/>
M39	<input type="checkbox"/>
M42	<input type="checkbox"/>
M45	<input type="checkbox"/>
M48	<input checked="" type="checkbox"/>
1/2	<input checked="" type="checkbox"/>
5/8	<input checked="" type="checkbox"/>
3/4	<input checked="" type="checkbox"/>
7/8	<input checked="" type="checkbox"/>
1	<input checked="" type="checkbox"/>
1 1/8	<input checked="" type="checkbox"/>
1 1/4	<input checked="" type="checkbox"/>
1 3/8	<input checked="" type="checkbox"/>
1 1/2	<input checked="" type="checkbox"/>

It must be noted that the 'visibility' filter only applies if the following checkbox is ticked in 'General options':



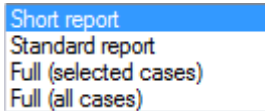
23.3 Automatic import

Please notice that if you load a file that has a custom database section that is not inside your database (for example a file saved by another user that created a custom section), SCS will automatically import the new property inside the database.

24 Reports

24.1 Calculation report

You can generate reports according to the following options:



- Short report: only sketch, 'general', 'loads', 'sumup' data and error 'comments' will be reported;
- Standard report: complete information on input data plus the 'sumup' tab results and comments;
- Full (selected cases): report will display all data and results of each tab; the results will be displayed only for the load case ('a', 'b', 'c', 'd' or 'e') selected in each tab.
- Full (all cases): report will display data and results of each tab; the results will be displayed for all load cases ('a' to 'e')

The reports can later be saved in .txt or .rtf (Word compatible) and modified by the user to allow maximum flexibility.

Two buttons are shown side-by-side: 'Save RTF' and 'Save TXT'. Both buttons have a light gray background and dark text.

Please note that the sketch in the report is the same displayed in your sketch window, therefore you can customize what you will see on report.

24.2 T-stub report

This report is available only when the Eurocode or Eurocode/BS methods are used for 'Moment calc. method' and displays pertinent data.

25 Request support

Please use menu under Help - Send support request or e-mail support@scs.pe or support@steelconnectionstudio.com.

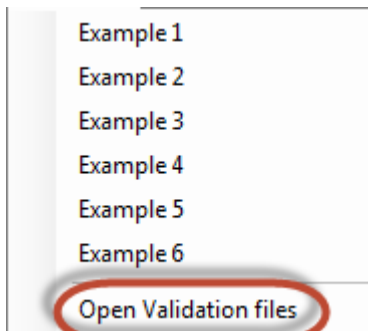
Installation support is included during Demo version evaluation and for the first 30 days after purchase. For longer support, you must have purchased 'Update subscription'.

26 Validation

26.1 General

Several validation examples are in the folder 'Validation' under [Installation directory]/DB/Validation.

This folder can be opened directly clicking



Some of the validation examples are reported and commented in the following pages.

26.2 Flexible end plate with EC3

This example is taken from ECCS Technical Committee 10, Structural Connections No. 126, 2009, on page 67 (#1e).

A flexible end plate welded to secondary beam, bolted to primary is analyzed for a shear on strong axis loading. This connection is considered as a pin type connection. Small moments and axial actions might be input. Here below the calculation hypothesis adopted in SCS:

Standard: Eurocode;

Bolt eccentricity calculation method: Elastic ;

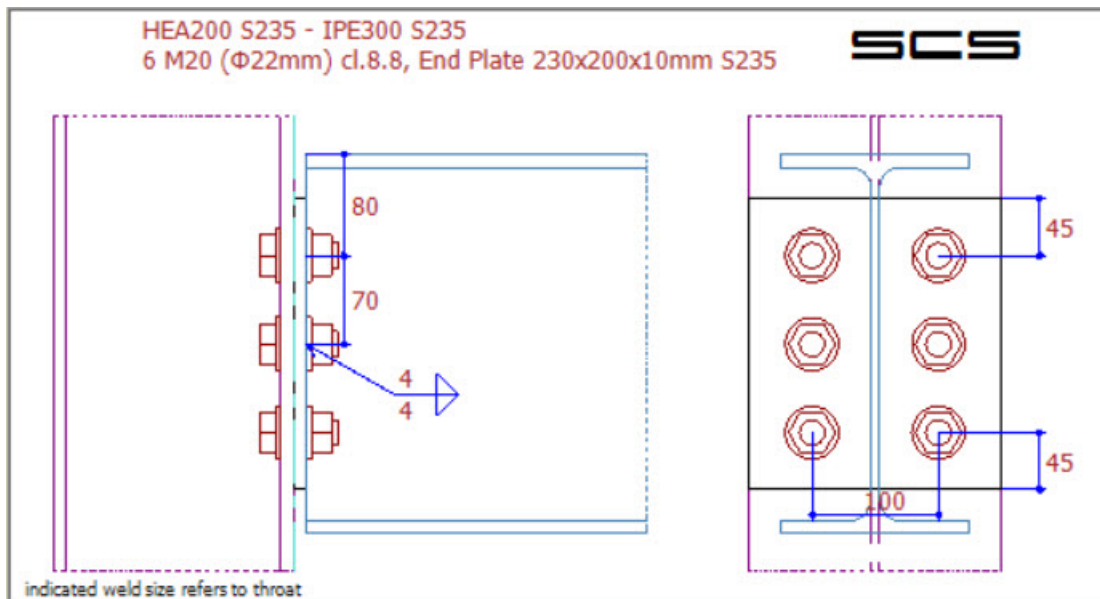
Connection axis: Column-Plate contact point;

Category for shear: Bearing type;

Moment calc. method: Eurocode/BS.

Main limit state for the connection are below explained and compared with published results in #1e.

26.2.1 Geometrical and mechanical data



26.2.2 Loads

A shear load on strong axis of 200kN is considered.

26.2.3 Joint shear resistance

26.2.3.1 Bolts in shear

$F_{V,Rd,1 \text{ bolt}} = \alpha_v A f_{ub} / \gamma_{M2} = 0.6 \cdot 245 \text{ mm}^2 \cdot 800 \text{ N/mm}^2 / 1.25 = 94.08 \text{ kN}$ (from table 3.4 EC3-1-8), which is the same value on #1e.

$F_{V,Rd,bolt \text{ group}} = n F_{V,Rd} = 6 \cdot 94.08 \text{ kN} = 564.5 \text{ kN}$

$F_{V,Rd,1 \text{ bolt}} =$	94,08	kN	formula
$F_{V,Rd,bolt \text{ gr.}} =$	564,5	kN	formula
$F_{V,Ed,bolt \text{ gr.}} =$	200	kN	formula
Bolt shear:	ok 35%		formula

26.2.3.2 Header plate in bearing

$F_{b,Rd,v,plate} = k_1 \alpha_b d t_p f_{up} / \gamma_{M2} = 2.5 \cdot 0.68 \cdot 20 \text{ mm} \cdot 10 \text{ mm} \cdot 360 \text{ N/mm}^2 / 1.25 = 98.18 \text{ kN}$ (from table 3.4 EC3-1-8)

$F_{b,Rd,v,pl} =$	98,18	kN	$\frac{k_{s,vpl} \alpha_{b,vpl} k_{red,vert.} f_u d t}{\gamma_{M2}}$ (ta. 3.4) #00
-------------------	-------	----	--

$F_{b,Rd,vertical} = n F_{b,Rd} = 6 \cdot 98.18 \text{ kN} = 589.1 \text{ kN}$

which are the same values in #1e.

26.2.3.3 Column flange in bearing

$F_{b,Rd,v,bw} = k_1 \alpha_b d t_p f_{up} / \gamma_{M2} = 116.73 \text{ kN}$ (from table 3.4 EC3-1-8)

$F_{b,Rd,v,bw} =$	116,7	kN	$\frac{k_{s,vbw} \alpha_{b,vbw} k_{red,vert.} f_u d t}{\gamma_{M2}}$ (ta. 3.4) #00
-------------------	-------	----	--

$F_{b,Rd,v} = n F_{b,Rd} = 6 \cdot 98.18 \text{ kN} = 700.36 \text{ kN}$

which are the same values in #1e.

26.2.3.4 Gross section of the header plate in shear

SCS let you decide between the net and the gross area for the value of A_v . Using gross area, it becomes:

$V_{Rd, str, pl} = A_{v, gross} f_y / (\sqrt{3} \gamma_{M0}) = 2'300 * 235 / (\sqrt{3} * 1.0) = 312.1 \text{ kN}$ (from formula 6.18 EC3-1-8)
this value is the half plate shear resistance, so it's compared with half shear load: 100kN.

$V_{Rd, str, pl} =$	312,1 kN	$= \frac{2300 \text{ (mm}^2\text{)} * 235 \text{ (N/mm}^2\text{)}}{\sqrt{3} * 1}$
$V_{Ed, str, pl} =$	100 kN	formula
Shear str:	ok 32%	formula

26.2.3.5 Net section of the header plate in shear

SCS let you decide between the net and the gross area for the value of A_v . Using net area, it becomes:

$V_{Rd, str, pl} = A_{v, net} f_y / (\sqrt{3} \gamma_{M0}) = 1'640 * 235 / (\sqrt{3} * 1.0) = 222.5 \text{ kN}$ (from formula 6.18 EC3-1-8)
this value is the half plate shear resistance, so it's compared with half shear load: 100kN.

$V_{Rd, str, pl} =$	222,5 kN	$= \frac{1640 \text{ (mm}^2\text{)} * 235 \text{ (N/mm}^2\text{)}}{\sqrt{3} * 1}$
$V_{Ed, str, pl} =$	100 kN	formula
Shear str:	ok 45%	formula

26.2.3.6 Shear block of the header plate

$V_{eff, Rd} = V_{eff, Rd, 5} = f_{up} A_{nt} / \gamma_{M2} + f_{yp} A_{nv} / (\sqrt{3} \gamma_{M0}) = 360 * 780 / 1.25 + 235 * 2'600 / (\sqrt{3} * 1.0) = 577.4 \text{ kN}$
(from formula 3.9 EC3-1-8)

which is the same value in #1e.

Plate block shear (block tearing) - $V = V_{vert, sd, bolt}$ gr.					
Mode 4:	$A_{nt} =$	780 mm ²	$A_{nv} =$	2600 mm ²	$V_{eff, Rd4} =$ 577,4 kN
Mode 5:	$A_{nt} =$	780 mm ²	$A_{nv} =$	2600 mm ²	$V_{eff, Rd5} =$ 577,4 kN
	$V_{eff, Rd V Pl} =$	577,4 kN	formula		

26.2.3.7 Beam web in shear

$V_{Rd, str, sec} = A_v f_y / (\sqrt{3} \gamma_{M0}) = 1'633 \text{ mm}^2 * 235 \text{ N/mm}^2 / (\sqrt{3} * 1.0) = 221.6 \text{ kN}$ (from formula 6.18 EC3-1-8)

which is the same value in #1e.

$V_{Rd, str, sec} =$	221,6 kN	$\frac{A_v f_y}{\sqrt{3} \gamma_{M0}} \text{ (6.18) \#01}$
$V_{Ed, str, sec} =$	200 kN	formula
Shear str:	ok 90%	formula

26.2.3.8 Column in shear

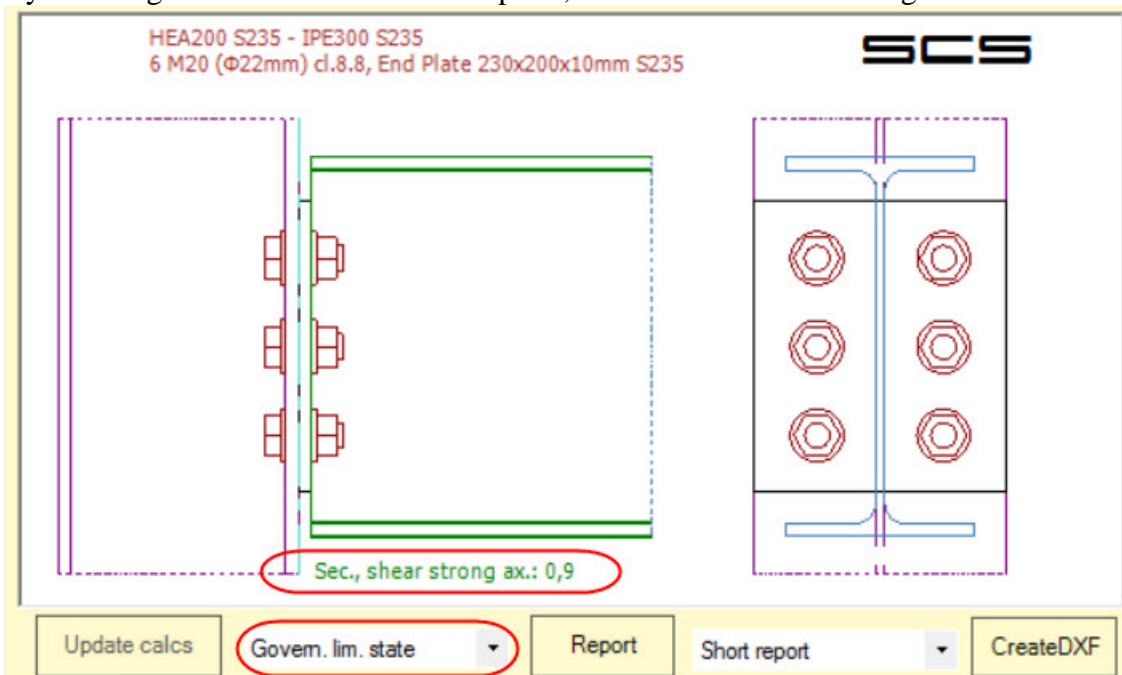
$V_{Rd, loc} = A_{v, net} f_y / (\sqrt{3} \gamma_{M0}) = 2'900 * 235 / (\sqrt{3} * 1.0) = 393.5 \text{ kN}$ (from check 6.5 #41)

this value is the half plate shear resistance, so it's compared with half shear load: 100kN.

Prim. local shear:			
$A_v =$	2900 mm ²	formula	
$V_{Rd, loc} =$	393,5 kN	$\frac{A_v f_y}{\sqrt{3} \gamma_{M0}} \text{ (6.5 check 10°) \#41}$	
$V_{Ed, loc} =$	100 kN	formula	
Prim. local shear:	ok 25%	formula	

26.2.3.9 Joint shear resistance

By selecting the "Govern. lim. state" option, SCS shows in the drawing the worst limit state:



which is the same failure mode reported in the exercise #1e: beam web in shear (200kN / 221.56kN = 0.9).

26.2.4 Joint tying resistance

26.2.4.1 Bolt in tension

$N_{Rd,1bolt} = 0.9 f_{ub} A_s / \gamma_{M2} = 0.9 \cdot 800 \cdot 245 \text{ N/mm}^2 / 1.25 = 141.1 \text{ kN}$ (from table 3.4 EC3-1-8)

$N_{Rd,1bolt} =$	141,1 kN	$= \frac{0,9 \cdot 800 \text{ (N/mm}^2\text{)} \cdot 245 \text{ (mm}^2\text{)}}{1,25}$
$N_{Rd,bolt gr.} =$	129 kN	formula

26.2.4.2 Header plate in bending

This limit state in SCS is included into the resistance of bolt in tension. In fact, $N_{Rd,bolt \text{ group}}$ is calculated considering the T-stub behavior for flexible end plate and column flange.

26.2.4.3 Beam web in tension

$N_{Rd,sec} = A f_y / \gamma_{M0} = 1'633 \text{ mm}^2 \cdot 235 \text{ N/mm}^2 / 1.0 = 383.8 \text{ kN}$ (from formulas 6.6 and 6.7 EC3-1-8)

$N_{Rd,sec} =$	383,8 kN	$\min \left(\frac{A f_y}{\gamma_{M0}} \text{ (6.6) } \# 0.1, \frac{0.9 A_{net} f_u}{\gamma_{M2}} \text{ (6.7) } \# 0.1 \right)$
$N_{Ed,sec} =$	0 kN	formula
Axial:	ok 0%	formula

26.2.4.4 Joint tying resistance

The worked example reports:

Tying resistance of the joint = 534.44kN; Failure mode: beam web in tension.

SCS, instead, calculate the tensile strength of the bolt group using the T-stub method

Tension:			
$N_{Ed, \text{bolt gr.}}$	1 kN	formula	<input type="button" value="Draw in-plane forces"/> <input type="button" value="Draw Sketch"/> <input type="button" value="T-stub notes"/>
$M_{\perp Ed, \text{bolt gr.}}$	0 kN*m	formula	
$N_{Rd, 1 \text{ bolt}}$	141,1 kN	formula	
$N_{Rd, \text{bolt gr.}}$	129 kN	formula	

Beam side:

Individual: Mode1: $(4 * (\text{Min}(2\pi * 41,9, 4 * 41,9 + 1.25 * 50))) * 0.25 * 10^2 * 235 / 1) / 41,9 = 129 \text{ kN}$;
 Mode2: $(2 * (\text{Min}(2\pi * 41,9, 4 * 41,9 + 1.25 * 50))) * 0.25 * 10^2 * 235 / 1 + 50 * 141,1 * 2 * 1E3) / (41,9 + 50) = 182,9 \text{ kN}$;
 Mode3: $141,1 * 2 = 282,2 \text{ kN}$;
 Web tension: 289 kN

Group*: Rows 1 to 2: Mode1: $(4 * (\text{Min}(230,1, 230,1))) * 0.25 * 10^2 * 235 / 1) / 41,9 = 129 \text{ kN}$;
 Mode2: $(2 * (230,1) * 0.25 * 10^2 * 235 / 1 + 50 * 141,1 * 4 * 1E3) / (41,9 + 50) = 336,4 \text{ kN}$;
 Mode3: $141,1 * 4 = 564,5 \text{ kN}$;
 Web tension: 405,8 kN

And the failure mode is mode 1.

26.2.5 References

- #00: EN 1993-1-8: 2005, Design of steel structures, Design of joints, Cen.
- #01: EN 1993-1-1: 2001, Design of steel structures, General structural rules, Cen.
- #41: Joints in steel constructions, Simple Connections, SCI and BCSA, 2002.
- #1e: Technical Committee 10, Structural Connections No. 126, 2009.

26.3 Fin plate connection with EC3

This example is taken from ECCS Technical Committee 10, Structural Connections No. 126, 2009, on page 74 (#1e).

A fin plate welded to column, bolted to beam is analyzed for a shear on strong axis loading. This connection is normally considered as a pin type connection, though limited bending moment can be resisted.

Here below the calculation hypothesis adopted in SCS:

Standard: Eurocode

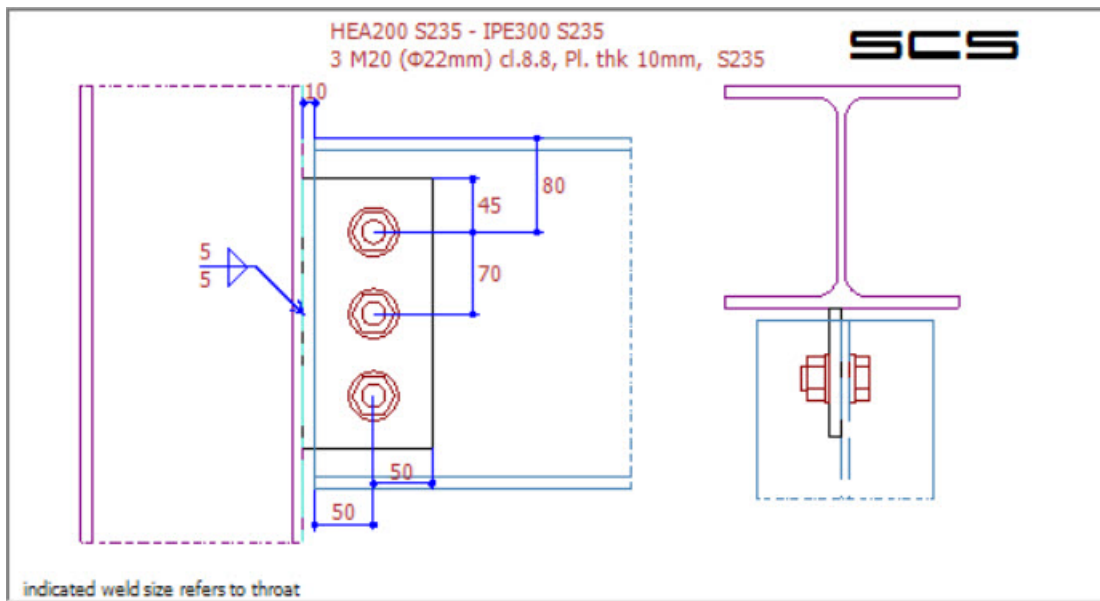
Bolt eccentricity calculation method: Elastic

Connection axis: Column axis

Category for shear: Bearing type

Main limit state for the connection are below explained and compared with published results in #1e.

26.3.1 Geometrical and mechanical data



26.3.2 Loads

A shear load on strong axis of 100kN is considered.

26.3.3 Joint shear resistance

26.3.3.1 Bolts in shear

$$F_{V,Rd,1\text{ bolt}} = \alpha_v A f_{ub} / \gamma_{M2} = 0,6 * 245 \text{ mm}^2 * 800 \text{ N/mm}^2 / 1,25 = 94,08 \text{ kN (from table 3.4 EC3-1-8)}$$

$$V_{Rd,bolt\text{ group}} = n F_{V,Rd} / \sqrt{1 + (6 z / ((n+1)p))^2} = 3 * 94,08 \text{ kN} / \sqrt{1 + (6 * 60 \text{ mm} / ((3+1) * 70 \text{ mm}))^2} = 173,3 \text{ kN}$$

PLF		formula
$F_{V,Rd,1\text{ bolt}}$	94,08 kN	$\frac{\alpha_v f_{ub} A n_{sh.pl.}}{\gamma_{M2}} \text{ (ta. 3.4) \#00}$
$F_{V,Rd,bolt\text{ gr.}}$	173,3 kN	formula
$F_{V,Rd}$	100 kN	formula

which are the same values in #1e.

26.3.3.2 Fin plate in bearing

$$F_{b,Rd,vert} = k_1 \alpha_b d t_p f_{up} / \gamma_{M2} = 2,5 * 0,68 * 20 \text{ mm} * 10 \text{ mm} * 360 \text{ N/mm}^2 / 1,25 = 98,18 \text{ kN (from table 3.4 EC3-1-8)}$$

which is the same values in #1e.

$F_{b,Rd,v.pl}$	98,18 kN	$= 1,705 * 1 * 360 \text{ (N/mm}^2\text{)} * 20 \text{ (mm)} * 10 \text{ (mm)} / 1,25$
$F_{b,Ed,v.pl}$	33,33 kN	Bolt 3

26.3.3.3 Fin plate in shear

SCS let you decide between the net and the gross area for the value of A_v . Using gross area, it becomes:

$$V_{Rd,str.pl} = A_{v,gross} f_y / (\sqrt{3} \gamma_{M0}) = 2'300 * 235 / (\sqrt{3} * 1,0) = 312,1 \text{ kN (from formula 6.18 EC3-1-8).}$$

Using net area, it becomes:

$$V_{Rd,str.pl} = A_{v,net} f_y / (\sqrt{3} \gamma_{M0}) = 1'640 * 235 / (\sqrt{3} * 1,0) = 222,5 \text{ kN (from formula 6.18 EC3-1-8).}$$

$V_{Rd,str.pl}$	222,5 kN	$\frac{A_v f_y}{\sqrt{3} \gamma_{M0}} \text{ (6.18) \#01}$
-----------------	----------	--

26.3.3.4 Block shear of the fin plate

SCS calculate the block shear for five modes of rupture, and obtain:

$V_{Rd5} = 0,5 f_{up} A_{nt} / \gamma_{M2} + f_{yp} A_{nv} / (\sqrt{3} \gamma_{M0}) = 0,5 \cdot 360 \cdot 390 / 1,25 + 235 \cdot 1'300 / (\sqrt{3} \cdot 1,0) = 232,54 \text{ kN}$
which is the same value in #1e.

Plate block shear (block tearing) - V = Vvert, sd, bolt gr.					
Mode 4:	$A_{nt} =$	390 mm ²	$A_{nv} =$	1300 mm ²	$V_{eff, Rd4} =$ 232,5 kN
Mode 5:	$A_{nt} =$	390 mm ²	$A_{nv} =$	1300 mm ²	$V_{eff, Rd5} =$ 232,5 kN
$V_{eff, Rd VPl} =$		232,5 kN	min ($V_{eff, Rd4}$; $V_{eff, Rd5}$)		

26.3.3.5 Fin plate in bending and buckling of the fin plate

SCS calculate the resistance of fin plate using the formulas 6.14 (bending) and 6.55 (buckling) of #01:

$M_{Rd, str, pl} =$	17,65 kN*m	$w_{el, min} f_y \min \left(\frac{1}{\gamma_{M0}}, \frac{\chi_{LT}}{\gamma_{M1}} \right)$	(6.14, 6.55) #01
$M_{Ed, str, pl} =$	6 kN*m	formula	
Bending str:	ok 34%	formula	

26.3.3.6 Beam web in bearing

$F_{b, Rd, v, bw} = 2,027 \cdot 1 \cdot 360 \cdot 20 \cdot 7,1 / 1,25 = 82,88 \text{ kN}$ (from table 3.4 EC3-1-8).
which is the same value in #1e.

$F_{b, Rd, v, bw} =$	82,88 kN	$\frac{k_{1, vbw} \alpha_{b, vbw} k_{red, vert.} f_u dt}{\gamma_{M2}}$	(ta. 3.4) #00
$F_{b, Ed, v, bw} =$	33,33 kN	Bolt 3	

26.3.3.7 Beam web in shear

SCS let you decide between the net area and the gross area for the value of A_v .

Using the gross area of beam web:

$V_{Rd, str, sec} = A_{v, gross} f_y / (\sqrt{3} \gamma_{M0}) = 2'568 \text{ mm}^2 \cdot 235 \text{ N/mm}^2 / (\sqrt{3} \cdot 1,0) = 348,4 \text{ kN}$ (from formula 6.18 EC3-1-8).

which is the same value in #1e.

Using the net area of beam web:

$V_{Rd, str, sec} = A_{v, net} f_{ubw} / (\sqrt{3} \gamma_{M2}) = 2'100 \text{ mm}^2 \cdot 235 \text{ N/mm}^2 / (\sqrt{3} \cdot 1,25) = 230,9 \text{ kN}$ (from formula 6.18 EC3-1-8).

26.3.3.8 Block shear of the beam web

SCS calculates the block shear for five modes of rupture, and obtains a different value from the example because it considers also the flange on the vertical rupture for block shear.

Beam block shear (block tearing) - V = Vvert, sd, bolt gr.					
Mode 4:	$A_{nt} =$	276,9 mm ²	$A_{nv} =$	2797 mm ²	$V_{eff, Rd4} =$ 419,4 kN
Mode 5:	$A_{nt} =$	276,9 mm ²	$A_{nv} =$	2797 mm ²	$V_{eff, Rd5} =$ 419,4 kN
$V_{eff, Ed V-beam} =$		419,4 kN	formula		

26.3.4 Joint tying resistance

26.3.4.1 Bolts in shear

$F_{v, Rd, 1bolt} = 0,6 \cdot 800 \text{ N/mm}^2 \cdot 245 \text{ mm}^2 / 1,25 = 94,08 \text{ kN}$ (from table 3.4 EC3-1-8).

$F_{v, Rd, bolt group} = 3 \cdot 94,08 \text{ kN} =$

282,2kN

$F_{v,Rd,1 \text{ bolt}} =$	94,08 kN	$\frac{\alpha_v f_{ub} A_{nzh.pl.}}{\gamma_{M2}} \quad (\text{ta. 3.4}) \#00$
$F_{v,Rd,bolt gr.} =$	282,2 kN	formula

that is 12% less than the value from exercise, due to the different reducing coefficient used (1,25 or 1,10).

26.3.4.2 Fin plate in bearing

$F_{b,Rd,hor} = k_1 \alpha_b d t_p f_{up} / \gamma_{M2} = 2,5 * 0,75 * 20\text{mm} * 10\text{mm} * 360\text{N/mm}^2 / 1,25 = 108\text{kN}$ (from table 3.4 EC3-1-8)

here there's a difference from example on horizontal direction, because SCS limits the horizontal bearing resistance in case with one bolt column as per 3.2 in #00 (single lap joints with only one bolt row):

$F_{b,Rd} < 1,5 f_u d t / \gamma_{M2}$ (from formula 3.2 EC3-1-8)

$F_{b,Rd,h.pl.} =$	86,4 kN	$= 1,5 * 1 * 360 (\text{N/mm}^2) * 20 (\text{mm}) * 10 (\text{mm}) / 1,25$
$F_{b,Ed,h.pl.} =$	42,86 kN	Bolt 3

26.3.4.3 Fin plate in tension

SCS calculates the area using Withmore theory. In this case with only one bolt row, it becomes:

$N_{Rd,pl} = 1'400\text{mm}^2 \times 235\text{N/mm}^2 / 1,00 = 329\text{kN}$ (from formula 6.6 EC3-1-8)

$N_{Rd,pl} =$	329 kN	$\min \left(\frac{A f_y}{\gamma_{M0}} \quad (6.6) \#01, \frac{0,9 A_{net} f_u}{\gamma_{M2}} \quad (6.7) \#01 \right)$
$N_{Ed,pl} =$	0 kN	formula
Axial:	ok 0%	formula

26.3.4.4 Beam web in bearing

$F_{b,Rd,v,bw} = 1,5 * 1 * 360 * 20 * 7,1 / 1,25 = 61,34\text{kN}$ (from table 3.4 EC3-1-8).

here there's a difference from example on horizontal direction, because SCS limits the horizontal bearing resistance in case with one bolt column as per 3.2 in #00 (single lap joints with only one bolt row):

$F_{b,Rd} < 1,5 f_u d t / \gamma_{M2}$ (from formula 3.2 EC3-1-8).

$F_{b,Rd,h,bw} =$	61,34 kN	$\frac{\min \{ 1,5 k_1 h_{bw} \alpha_b h_{bw} \} k_{red,hor.} f_u d t}{\gamma_{M2}} \quad (\text{ta. 3.4; 3.2}) \#00$
-------------------	----------	---

26.3.4.5 Beam web in tension

SCS calculates the area using Withmore theory. In this case with only one bolt row, it becomes:

$N_{Rd,sec} = 5'381\text{mm}^2 \times 235\text{N/mm}^2 / 1,00 = 1'265\text{kN}$ (from formula 6.6 EC3-1-8)

$N_{Rd,sec} =$	1265 kN	$\min \left(\frac{A f_y}{\gamma_{M0}} \quad (6.6) \#01, \frac{0,9 A_{net} f_u}{\gamma_{M2}} \quad (6.7) \#01 \right)$
$N_{Ed,sec} =$	0 kN	formula

26.3.5 References

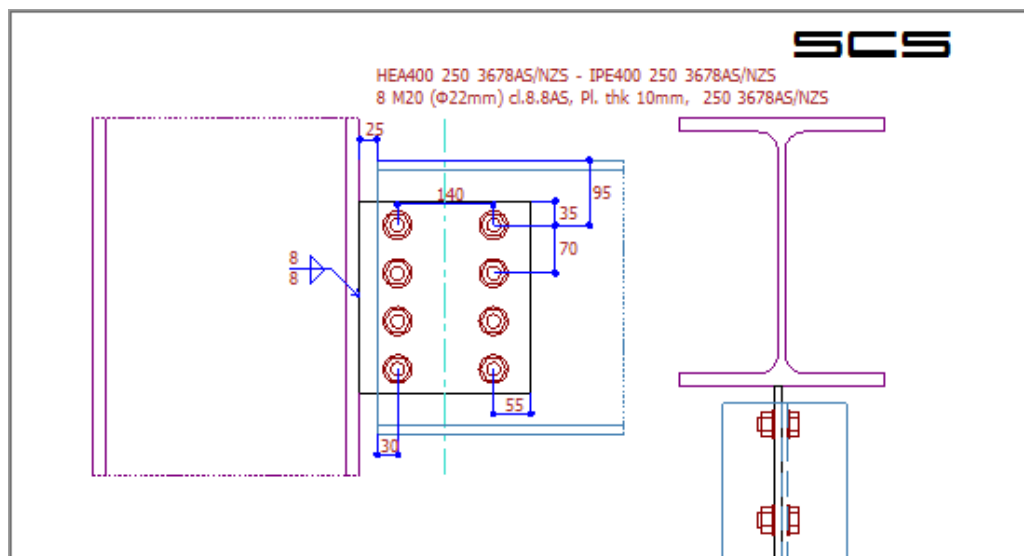
- #00: EN 1993-1-8: 2005, Design of steel structures, Design of joints, Cen.
- #01: EN 1993-1-1: 2001, Design of steel structures, General structural rules, Cen.
- #41: Joints in steel constructions, Simple Connections, SCI and BCSA, 2002.
- #1e: Technical Committee 10, Structural Connections No. 126, 2009.

26.4 AS4100 code validation examples

Following considerations about validation of SCS and support for the user take into account worked examples proposed in “The Behavior and Design of Steel Structures to AS 4100, 3rd edition – N.S. Trahair and M.A. Bradford”.

Referring to chapter 9.9, we start from example 9.9.1 and go on with the other ones.

26.4.1 Worked Example 9.9.1



The example requires to evaluate the maximum bolt shear force for a connection with a fin plate (module FP1) with a shear equal to Q^* kN and a moment equal to $0.2Q^*$ kNm*. To make a comparison, here we start considering the vertical action $Q^* = 100$ kN, so the moment is 20 kNm. As the design actions include a moment, in SCS there is no need to add any eccentricity of the vertical action to check the bolts. The software gives the possibility to choose the position of axis of the connection, where load are applied, as shown in the following figure. In this case the correct choice is “bolt group axis”.

General Loads Members Plate & Bolts Geometry Weld, Notch & Stiffeners

This connection is normally considered as a pin type connection, though limited bending moment can be resisted. All actions are transmitted by the plate welded to primary member (column) and bolted to secondary member (beam).

Standard: AS 4100

Language: English

Calculation update: Automatic update

Bolt eccentric. calc. method: Elastic

Connection axis: Bolt group axis

Connection type: Bolt group axis

Plate welding to primary: Only vertically

Category for shear: Bearing type

Double sided connection: No

Choice of connection axis

In this way vertical action has an eccentricity from the column flange equal to $(25+30+140/2)=125$ mm.

General Loads Members Plate & Bolts Geometry Weld, Notch & Stiffeners

bolt columns = 2 bolt rows = 4 total bolts = 8

plate thk = 10 mm cl. beam = 25 mm

☐ Plate 'standard' bolt spacing beam clear. from flanges = -119.5 mm

s_g = 140 mm ☐ Bolts are vertic. sym. in second.

s_p = 70 mm $a_{vert. beam top}$ = 95 mm

$a_{horiz. pl.}$ = 55 mm $a_{vert. beam bottom}$ = 95 mm

$a_{horiz. beam}$ = 30 mm

depth pl. = 280 mm ☒ Bolts are vert. sym. in plate

width pl. = 250 mm $a_{vert. pl. top}$ = 35 mm

overwrite $a_{vert. pl. bottom}$ = 35 mm

☐ bolt gr. ecc. = 0 mm *BoltGroupCenter distance from Conn. Axis*

☐ pl. ecc. = 125 mm *Section at primary member*

☐ sec. ecc. = 100 mm *Section near prim.*

☐ $L_{or. pl.}$ = 140 mm *formula*

Eccentricity of the action from the column flange

Solution of example 1 comes from the assumption of a rigid connection rotating around the axis outgoing from the plane of the plate. The same approach can be used in SCS choosing as “bolt eccentricity calculation method” the “elastic” one:

General Loads Members Plate & Bolts Geometry Weld, Notch & Stiffeners

This connection is normally considered as a pin type connection, though limited bending moment can be resisted. All actions are transmitted by the plate welded to primary member (column) and bolted to secondary member (beam).

Standard: AS 4100

Language: English

Calculation update: Automatic update

Bolt eccentric. calc. method: Elastic

Connection axis: Instantaneous center of rotation

Connection type: Plate welded to primary member web

Plate welding to primary: Only vertically

Category for shear: Bearing type

Double sided connection: No

Bolt eccentricity calculation method

Direction of forces can be displayed in SCS choosing the option “draw in-plane forces”.

Update calcs Dimensions Report Full (all cases) CreateDXF

SumUp Bolts Pl. Bear. Sec. Bear. Pl. Bl. Sh. Sec. Bl. Sh. Pl. Str. Sec. Str. More Welding

Shear:

$V_{sf,hor,bolt\ gr.}^* = 0$ kN formula

$V_{sf,vert,bolt\ gr.}^* = 100$ kN formula

$M_{f,bolt\ gr.}^* = 20$ kN*m formula

$M_{f,ecor}^* = 20$ kN*m formula

$\beta_{Lf} = 1$ formula

$V_{sf,1\ bolt} = 92.63$ kN formula

$V_{sf,bolt\ gr.}^* = 250.1$ kN formula

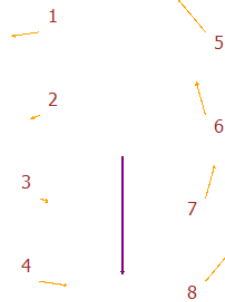
$V_{sf,bolt\ gr.}^* = 100$ kN formula

Bolt shear: ok 40% formula

kN	Bolt:	Rx:	Ry:	Rz:
	1	23.81	3.373	0
	2	7.937	3.373	0
	3	-7.937	3.373	0

Draw in-plane forces

Draw Sketch



Plane forces (vertical action + moment)

Values of forces for each bolt are displayed along x and y direction as shown in the following figure.

$V_{sf,1\ bolt}^* = 92.63$	kN	formula
$V_{sf,bolt\ gr.}^* = 250.1$	kN	formula
$V_{sf,bolt\ gr.}^* = 100$	kN	formula
Bolt shear:	ok 40%	formula

kN	Bolt:	Rx:	Ry:	Rz:
	1	23.81	3.373	0
	2	7.937	3.373	0
	3	-7.937	3.373	0
	4	-23.81	3.373	0
	5	23.81	-28.37	0
	6	7.937	-28.37	0
	7	-7.937	-28.37	0
	8	-23.81	-28.37	0
Σ		0	-100	0

Draw in-plane forces

Draw Sketch

Bolt forces

Worst condition is reached by bolts 5 and 8: , value that confirms the

$$\sqrt{(23.81)^2 + (28.37)^2} = 37 \text{ kN}$$

solution of worked example.

26.4.2 Worked Example 9.9.2

This example refers to the same connection of example 9.9.1 and the aim is to evaluate the bolt shear capacity and the maximum value of Q^* that can be supported considering bolts shear capacity. First of all, we can see that the resistance of one bolt (92.6 kN) is confirmed in SCS:

SumUp	Bolts	Pl. Bear.	Sec. Bear.	Pl. Bl.Sh.	Sec. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
$M_{f, ecc}^* = 20 \text{ kN}\cdot\text{m}$ formula $\beta_{Lf} = 1$ formula $V_{sf, 1 \text{ bolt}} = 92.63 \text{ kN}$ formula $V_{sf, bolt \text{ gr.}} = 250.1 \text{ kN}$ formula $V_{sf, bolt \text{ gr.}}^* = 100 \text{ kN}$ formula Bolt shear: ok 40% formula									
kN	Bolt:	Rx:	Ry:	Rz:					
	1	23.81	3.373	0	Draw in-plane forces				
	2	7.937	3.373	0	Draw Sketch				
	3	-7.937	3.373	0					
	4	-23.81	3.373	0					
	5	23.81	-28.37	0					
	6	7.937	-28.37	0					
	7	-7.937	-28.37	0					
	8	-23.81	-28.37	0					
	Σ	0	-100	0					

Shear resistance of one bolt

With the actions of the previous example ($Q^* = 100 \text{ kN}$ and $0.2Q^* = 20 \text{ kNm}$), the bolt with the maximum shear works at 40% (in fact $37 \text{ kN} / 92.6 \text{ kN} = 0.4$).

If $Q^* = 250 \text{ kN}$ (and $0.2Q^* = 50 \text{ kNm}$), as for the example 2 of the book, connection work at 100%:

Units: kN, kN*m

Case	N*	V*x	V*y	M*x	case name
1		250		50	
2					
3					
4					
5					

SumUp	Bolts	Pl. Bear.	Sec. Bear.	Pl. Bl.Sh.	Sec. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
$M_{f, ecc}^* = 50 \text{ kN}\cdot\text{m}$ formula $\beta_{Lf} = 1$ formula $V_{sf, 1 \text{ bolt}} = 92.63 \text{ kN}$ formula $V_{sf, bolt \text{ gr.}} = 250.1 \text{ kN}$ formula $V_{sf, bolt \text{ gr.}}^* = 250 \text{ kN}$ formula Bolt shear: ok 100% formula									
kN	Bolt:	Rx:	Ry:	Rz:					
	1	59.52	8.433	0	Draw in-plane forces				
	2	19.84	8.433	0	Draw Sketch				
	3	-19.84	8.433	0					
	4	-59.52	8.433	0					
	5	59.52	-70.93	0					
	6	19.84	-70.93	0					
	7	-19.84	-70.93	0					
	8	-59.52	-70.93	0					
	Σ	0	-250	0					

Bolt shear capacity equal to 100%

26.4.3 Worked Example 9.9.3

This example refers to the same connection of example 9.9.1 and the aim is to evaluate bearing capacity of the plate.

There are some differences between the worked example and SCS. As shown in following figure, adopted formulation in SCS for bearing capacity considers also the factor a_e (that depends on distance from the edges and distance between holes, as specified in AS4100).

SumUp	Bolts	Pl. Bear.	Sec. Bear.	Pl. Bl.Sh.	Sec. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
Tot. ref. thk = 10 mm		$s_{p,min.} = 50$ mm	$s_{p,opt.} = 60$ mm						
		$e_{1,min.} = 30$ mm	$e_{1,opt.} = 60$ mm						
		$s_{g,min.} = 50$ mm	$s_{g,opt.} = 60$ mm						
		$e_{2,min.} = 30$ mm	$e_{2,opt.} = 40$ mm						
Horizontal forces					Vertical forces				
$s_{p,h,pl} = 140$ mm	ok	$s_{p,v,pl} = 70$ mm	ok						
$e_{1,h,pl} = 55$ mm	ok	$e_{1,v,pl} = 35$ mm	ok						
$s_{g,h,pl} = 70$ mm	ok	$s_{g,v,pl} = 140$ mm	ok						
$e_{2,h,pl} = 35$ mm	ok	$e_{2,v,pl} = 55$ mm	ok						
$\Phi V_{b,h,pl} = 199.3$ kN		$\Phi (= 0.9) * \min(3.2 d_f, a_e) t_p f_{up} (9.3.2.4) \# 90$							
$V^*_{b,h,pl} = 59.52$ kN		Bolt 8							
$\Phi V_{b,v,pl} = 125.5$ kN		formula							
$V^*_{b,v,pl} = 70.93$ kN		Bolt 8							
Plate bear. h.: ok 30%		Plate bear. v.: ok 57%							

The lowest resistance reached in SCS is 125.5 kN: in the following figure is shown the numeric formula that gives this result (in “general options” can be chosen the options “literal formula” or “numeric formula”).

SumUp	Bolts	Pl. Bear.	Sec. Bear.	Pl. Bl.Sh.	Sec. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
Tot. ref. thk = 10 mm		$s_{p,min.} = 50$ mm	$s_{p,opt.} = 60$ mm						
		$e_{1,min.} = 30$ mm	$e_{1,opt.} = 60$ mm						
		$s_{g,min.} = 50$ mm	$s_{g,opt.} = 60$ mm						
		$e_{2,min.} = 30$ mm	$e_{2,opt.} = 40$ mm						
Horizontal forces					Vertical forces				
$s_{p,h,pl} = 140$ mm	ok	$s_{p,v,pl} = 70$ mm	ok						
$e_{1,h,pl} = 55$ mm	ok	$e_{1,v,pl} = 35$ mm	ok						
$s_{g,h,pl} = 70$ mm	ok	$s_{g,v,pl} = 140$ mm	ok						
$e_{2,h,pl} = 35$ mm	ok	$e_{2,v,pl} = 55$ mm	ok						
$\Phi V_{b,h,pl} = 199.3$ kN		formula							
$V^*_{b,h,pl} = 59.52$ kN		Bolt 8							
$\Phi V_{b,v,pl} = 125.5$ kN		$= 0.9 * 1 * \min(3.2 * 20 \text{ (mm)}, 34 \text{ (mm)}) * 410 \text{ (N/mm}^2) * 10 \text{ (mm)}$							
$V^*_{b,v,pl} = 70.93$ kN		Bolt 8							
Plate bear. h.: ok 30%		Plate bear. v.: ok 57%							

26.4.4 Worked Example 9.9.5

This example refers to the same connection of example 9.9.1 and the aim is to evaluate the fillet weld capacity.

As in the example 5 the weld works at 100% with $Q^* = 103.7$ kN, here we consider the same action.

Case	N*	V*x	V*y	M*x	case name
1		103.7		20.74	
2					
3					
4					
5					

SumUp	Bolts	Pl. Bear.	Sec. Bear.	Pl. Bl.Sh.	Sec. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
Web weld									
$A_{throat} =$	3168	mm ²	formula						
$N_{w,web}^* =$	0	kN	$V_{x,w,web}^* =$	103.7	kN	$V_{y,w,web}^* =$	0	kN	
$M_{Ed,str} =$	33.7	kN*m	formula						
$M_{Ed,wk} =$	0	kN*m	formula						
$T_{Ed} =$	0.9644	kN*m	formula						
$W_{R,str} =$	1.478E+05	mm ³	formula						
$W_{R,wk} =$	2987	mm ³	formula						
$W_{R,T} =$	4.96E+04	mm ³	formula						
$\sigma_{\perp} =$	161.2	N/mm ²	formula						
$\tau_{\perp} =$	-161.2	N/mm ²	formula						
$\tau_{\parallel} =$	52.18	N/mm ²	$\frac{ V_{Ed,str} }{A_{throat}} + \frac{ T_{Ed} }{W_{R,T}}$						
$\sigma_v =$	233.9	N/mm ²	formula						
$\sigma_{Rd} =$	230.4	N/mm ²	formula						
Weld comb.:	ERR 1.02		formula						

Scs gives a little different value: 102%. This happens because the software takes into account the effects of the torsion (T_{ed}), due to the eccentricity of weld from the axis of fin plate.

If we consider $T_{ed} = 0$, τ_{\parallel} becomes $103.7 \text{ kN} / 3168 \text{ mm}^2 = 32.73 \text{ N/mm}^2$.

So $\sigma_v =$, that compared with σ_{Rd} , gives 100%.

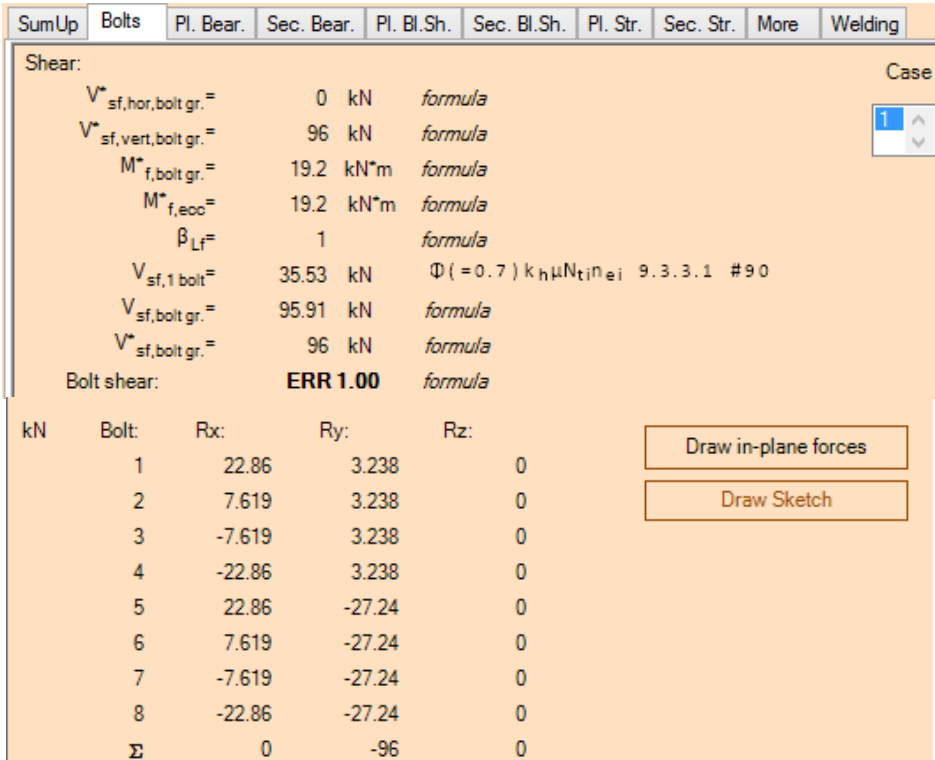
$$\sqrt{(161.2)^2 + (-161.2)^2 + (32.73)^2} = 230.3 \text{ kN}$$

26.4.5 Worked Example 9.9.6

This example refers to the same connection of example 9.9.1 and the aim is to evaluate the value of Q^* at which the first bolt slip occurs.

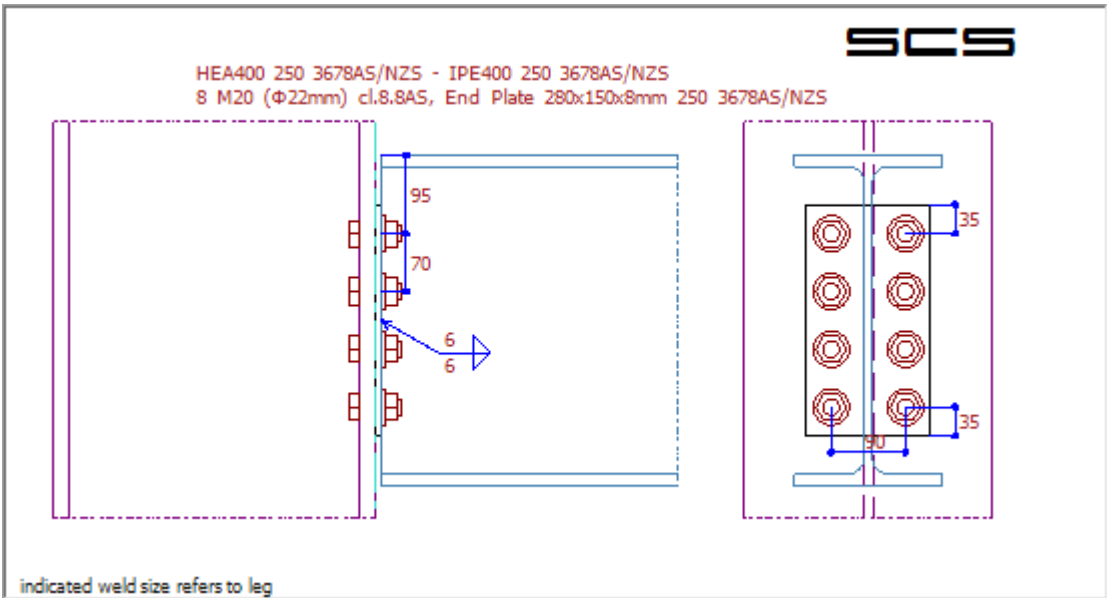
As in example 6 slip occurs with $Q^* = 96 \text{ kN}$ (assuming friction $\mu = 0.35$), here we consider the same conditions.

Case	N*	V*x	V*y	M*x	case name
1		96		19.2	
2					
3					
4					
5					

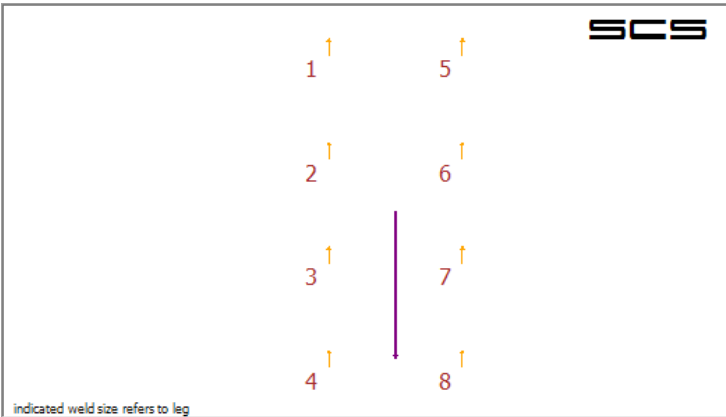


26.4.6 Worked Example 9.9.7

The example requires to evaluate the maximum bolt forces for a connection with a flexible end plate (module XL1), considering as external actions shear and moment equal to Q^* kN and $0.05Q^*$ kNm. To make a comparison, here we consider the vertical action $Q^* = 100$ kN, so the moment is 5 kNm.



As shown in the following figures, maximum shear is 12.5 kN, that confirms example result, maximum tension is 8.929 kN, that is less than the example result ($0.107Q^* = 10.7$ kN).



kN	Bolt:	Rx:	Ry:	Rz:
	1	0	-12.5	0
	2	0	-12.5	0
	3	0	-12.5	8.929
	4	0	-12.5	8.929
	5	0	-12.5	0
	6	0	-12.5	0
	7	0	-12.5	8.929
	8	0	-12.5	8.929
	Σ	0	-100	35.71

Calculation in SCS is done with the AISC method of neutral axis thru the center of gravity, that is very conservative. The book considers a more conservative method, with neutral axis thru the center of gravity and a triangular distribution of forces. In this way the farthest row of bolts is the most loaded.

This connection is considered as a pin type connection. Small moments and axial actions might be input.

Standard: AS 4100

Language: English

Calculation update: Automatic update

Bolt eccentric. calc. method: Elastic

Connection axis: Column-Plate contact point

Connection type: Plate bolted to primary member flange

Category for shear: Bearing type

Moment calc. method: Aisc, Neutral Axis thru Center of Gravity

Plate welding to secondary:

Eurocode/BS #42

Eurocode #00

Aisc, Neutral Axis NOT thru C. of Gravity (prying check included) #26

Aisc, Neutral Axis thru Center of Gravity (prying check included) #26

Triangular, elastic (prying check included)

indicated weld size

Update calcs

SumUp Bolts

M_r

M

V_s

V

Moment calculation method

It is important to observe that in this case "column axis" is at "column-plate contact point".

Both shear and tension resistances of bolts are confirmed by SCS.

SumUp	Bolts	Pl. Bear.	Prim. Bear.	Pl. Bl. Sh.	Pl. Str.	Sec. Str.	More	Welding
$V_{sf,vert,bolt\ gr.}^*$	100	kN	formula					
$M_{f,bolt\ gr.}^*$	0	kN*m	formula					
$M_{f,ecc}^*$	0	kN*m	formula					
β_{Lf}	1		formula					
$V_{sf,1\ bolt}$	92.63	kN	formula					
$V_{sf,bolt\ gr.}$	741	kN	formula					
$V_{sf,bolt\ gr.}^*$	100	kN	formula					
Bolt shear:	ok 13%		formula					
Tension:								
$N_{sf,bolt\ gr.}^*$	0	kN	formula					Draw in-plane forces
$M_{fL,bolt\ gr.}^*$	5	kN*m	formula					Draw Sketch
$N_{sf,1\ bolt}$	162.7	kN	formula					
$N_{sf,bolt\ gr.}$	90.77	kN	formula					
$N_{sf,tot,bolt\ gr.}^*$	35.71	kN	formula					

Shear and tension capacity of one bolt

26.4.7 Worked Example 9.9.8

This example refers to the same connection of example 9.9.7 and the aim is to evaluate the bearing load capacity. As noted in example 3, SCS considers also the factor a_e (that depends on distance from the edges and distance between holes, as specified in AS4100). So the software output is more conservative for the end plate, as shown in the following figures.

Tot. ref. thk = 8	$s_{p,min.}$ = 50 mm	$s_{p,opt.}$ = 60 mm	
	$e_{1,min.}$ = 30 mm	$e_{1,opt.}$ = 60 mm	
	$s_{g,min.}$ = 50 mm	$s_{g,opt.}$ = 60 mm	
	$e_{2,min.}$ = 30 mm	$e_{2,opt.}$ = 40 mm	
Horizontal forces		Vertical forces	
$s_{p,h,pl}$ = 90 mm	ok	$s_{p,v,pl}$ = 70 mm	ok
$e_{1,h,pl}$ = 30 mm	ok	$e_{1,v,pl}$ = 35 mm	ok
$s_{g,h,pl}$ = 70 mm	ok	$s_{g,v,pl}$ = 90 mm	ok
$e_{2,h,pl}$ = 35 mm	ok	$e_{2,v,pl}$ = 30 mm	ok
$\Phi V_{b,h,pl}$ = 85.61 kN	formula		
$V_{b,h,pl}^*$ = 0 kN	Bolt 8		
$\Phi V_{b,v,pl}$ = 100.4 kN	$\Phi (=0.9) * \min(3.2 d_f, a_e) t_{p fup} (9.3.2.4) \# 90$		
$V_{b,v,pl}^*$ = 12.5 kN	Bolt 8		
Plate bear. h.: ok 0%		Plate bear. v.: ok 12%	

Tot. ref. thk = 8	$s_{p,min.} = 50$ mm	$s_{p,opt.} = 60$ mm	
	$e_{1,min.} = 30$ mm	$e_{1,opt.} = 60$ mm	
	$s_{g,min.} = 50$ mm	$s_{g,opt.} = 60$ mm	
	$e_{2,min.} = 30$ mm	$e_{2,opt.} = 40$ mm	

Horizontal forces		Vertical forces	
$s_{p,h,pl} = 90$ mm	ok	$s_{p,v,pl} = 70$ mm	ok
$e_{1,h,pl} = 30$ mm	ok	$e_{1,v,pl} = 35$ mm	ok
$s_{g,h,pl} = 70$ mm	ok	$s_{g,v,pl} = 90$ mm	ok
$e_{2,h,pl} = 35$ mm	ok	$e_{2,v,pl} = 30$ mm	ok

$\Phi V_{b,h,pl} = 85.61$ kN	formula
$V_{b,h,pl}^* = 0$ kN	Bolt 8
$\Phi V_{b,v,pl} = 125$ kN	Bolt 8
$V_{b,v,pl}^* = 12.5$ kN	Bolt 8

Plate bear. h.: ok 0%	Plate bear. v.: ok 12%
-----------------------	------------------------

Vertical bearing capacity in SCS is 100.4 kN.

26.4.8 Worked Example 9.9.9

This example refers to the same connection of example 9.9.7 and the aim is to evaluate the fillet weld capacity.

As in example 9 the weld works at 100% with $Q^* = 373.4$ kN, here we consider the same action.

Case	N*	V*x	V*y	M*x	case name
1		373.4		-18.67	
2					
3					
4					
5					

"Column axis" is at "column-plate contact point" and SCS considers the eccentricity of the weld, that is equal to the thickness of the end plate (8 mm).

This connection is considered as a pin type connection. Small moments and axial actions might be input.

Standard: AS 4100

Language: English

Calculation update: Automatic update

Bolt eccentric. calc. method: Elastic

Connection axis: Column-Plate contact point

Connection type: Plate bolted to primary member flange

Category for shear: Bearing type

Moment calc. method: Aisc, Neutral Axis thru Center of Gravity

Plate welding to secondary: Web only

Connection axis position

General | Loads | Members | Plate & Bolts | Geometry | Weld, Notch & Stiffeners

Welding between fin plate and primary member:
 f_{uw} = 480 N/mm² categ.: SP
 Web weld: double fillet fillet number: 2
 $t_{t, web}$ = 6 mm l_{web} = 280 mm
 Welding the plate to flanges might increase capacity but it will also increase stiffness (not suggested by some literature)
☐ Weld plate to beam flanges
 Flange weld: single fillet
☐ Do not consider compression for welds
 overwrite
☐ **ecc. web weld = -8 mm Weld distance from Conn. Axis**
Stiffeners on primary member:
 No calculation is made on the stiffener thickness.
 'Single stiffener on the back side' option is only available when connection is on column web.
☐ Single stiffener on the back side
Notch configuration:
 Notch: No notch
☐ Different depth for bottom notch:
☐ Add false flanges
 Flange notch: No flange notch

Eccentricity of weld

As the example doesn't consider this eccentricity, to compare the result it is imposed equal to 0:

General | Loads | Members | Plate & Bolts | Geometry | Weld, Notch & Stiffeners

Welding between fin plate and primary member:
 f_{uw} = 480 N/mm² categ.: SP
 Web weld: double fillet fillet number: 2
 $t_{t, web}$ = 6 mm l_{web} = 280 mm
 Welding the plate to flanges might increase capacity but it will also increase stiffness (not suggested by some literature)
☐ Weld plate to beam flanges
 Flange weld: single fillet
☐ Do not consider compression for welds
 overwrite
☒ **ecc. web weld = 0 mm User defined**
Stiffeners on primary member:
 No calculation is made on the stiffener thickness.
 'Single stiffener on the back side' option is only available when connection is on column web.
☐ Single stiffener on the back side
Notch configuration:
 Notch: No notch
☐ Different depth for bottom notch:
☐ Add false flanges
 Flange notch: No flange notch

Eccentricity of weld imposed equal to 0

In this way SCS confirms the result of the example.

SumUp	Bolts	Pl. Bear.	Prim. Bear.	Pl. Bl.Sh.	Pl. Str.	Sec. Str.	More	Welding
Web weld								
$A_{throat} =$	2376	mm ²	formula					
$N_{w,web}^* =$	0	kN	$V_{x,w,web}^* =$	373.4	kN	$V_{y,w,web}^* =$	0	kN
$M_{Ed,str} =$	-18.67	kN*m	formula					
$M_{Ed,wk} =$	0	kN*m	formula					
$T_{Ed} =$	0	kN*m	formula					
$W_{R,str} =$	1.109E+05	mm ³	formula					
$W_{R,wk} =$	1680	mm ³	formula					
$W_{R,T} =$	3.051E+04	mm ³	formula					
$\sigma_{\perp} =$	119.1	N/mm ²	formula					
$\tau_{\perp} =$	-119.1	N/mm ²	formula					
$\tau_{\parallel} =$	157.2	N/mm ²	formula					
$\sigma_v =$	230.3	N/mm ²	formula					
$\sigma_{Rd} =$	230.4	N/mm ²	formula					
Weld comb.:	ok 100%		formula					

26.5 End plate moment connection with EC3

Following considerations have the aim to validate calculations of a moment connection, including T-Stub effects as per “UNI EN 1993-1-8: 2005, Design of steel structures, Design of joints, Cen (#00)” and “Joints in steel constructions, Moment Connections, SCI and BCSA, 1995 (#42)”. As support for the user, here we take into account the worked example 6.6.3 proposed in “Design of Joints in Steel and Composite Structures – J.P. Jaspart and K. Weynard”.

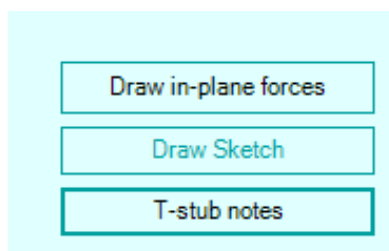
In order to make a better comparison between SCS and the worked example, the chosen moment calculation method is “Eurocode/BS #42”.

Column web panel in shear is confirmed: 548.9 kN

Column web in compression confirmed: 460.9 kN

Flange and web of the beam in compression confirmed: 1065 kN

T-stub details can be extracted from table “Bolts” and button “T-Stub Details”.



Positive Moment

Row 1: FT_Rd=244.5kN; Beam side; Individual; Mode 1; l_{eff}=120mm
 Row 2: FT_Rd=332.2kN; Beam side; Individual; Mode 1; l_{eff}=309.4mm
 Row 3: FT_Rd=85.9kN; Beam side; Group*: Rows 2 to 3: Mode 1; l_{eff}=389.4mm
 Row 4: FT_Rd=312.1kN; Beam side; Individual; Mode 1; l_{eff}=290.7mm

Negative Moment

Row 4: FT_Rd=312.1kN; Beam side; Individual; Mode 1; l_{eff}=290.7mm
 Row 3: FT_Rd=292kN; Beam side; Individual; Mode 1; l_{eff}=272mm
 Row 2: FT_Rd=126.1kN; Beam side; Group*: Rows 3 to 2: Mode 1; l_{eff}=389.4mm
 Row 1: no tension action

The initial of the output of T-stub details

The output shows in the initial of document the final resistances of each row of bolts, that are the result of comparison of the following limit states:

- Column flange in bending (T-stub);
- End plate in bending (T-Stub);
- Column web tension;
- Beam web tension.

Referring to details for positive moment, the three modes of T-Stub for column flange (column side) and end plate (beam side) are shown with numerical values and various effective lengths (L_{eff}) are indicated. Individual or group behavior of rows is present.

Web tension is calculated for both individual and group behaviors. When the value is “N.A.”, SCS considers the inapplicability of this limit state (for example when a bolts are near to flange and the web doesn’t work alone).

The words “Triang. limit not to be applied” refer to Step 1C of #42 (plastic distribution limit).

```

Row 2
Column side:
Individual: Mode1: (4*(Min(2π*33.6, 4*33.6+1.25*90))*0.25*16.5^2*235/1)/33.6= 402kN;
          Mode2: (2*(Min(2π*33.6, 4*33.6+1.25*90))*0.25*16.5^2*235/1+42.1*254.2*2*1E3)/(33.6+42.1)= 371.7kN;
          Mode3: 254.2*2= 508.3kN;
          Web tension: 464kN
Group*: Rows 1 to 2: Mode1: (4*(Min(332.1, 332.1))*0.25*16.5^2*235/1)/33.6= 631.4kN;
          Mode2: (2*(332.1)*0.25*16.5^2*235/1+42.1*254.2*4*1E3)/(33.6+42.1)= 705.1kN;
          Mode3: 254.2*4= 1016.6kN;
          Web tension: 653.8kN
Beam side:
Individual: Mode1: (4*(Min(2π*49.2, Max(4*49.2+1.25*60, α(24.9)(=6.3)*49.2)))*0.25*15^2*235/1)/49.2= 332.2kN;
          Mode2: (2*(Min(2π*49.2, Max(4*49.2+1.25*60, α(24.9)(=6.3)*49.2)))*0.25*15^2*235/1+60*254.2*2*1E3)/(49.2+60)= 354.1kN;
          Mode3: 254.2*2= 508.3kN;
          Web tension: N.A.
Triang.limit not to be applied
  
```

T-stub details: group/individual behavior and L_{eff} calculations

SCS output and the worked example match very well and some differences is due to a different approaches:

- 1) Web tension capacity in SCS is calculated as per step 1B of #42 (diffusion of load from bolts to web with an angle equal to 60°). Worked example follows 6.2.6.3 of #00. This approach for T-stub calculations in SCS is used if the “Eurocode #00” calculation method is chosen.
- 2) Effective length L_{eff2} (for mode 2) in SCS the minimum between circular yielding and side yielding (table 2.4 #42). The worked example does not consider circular patterns (as per table 6.4 #00). As above, this approach is used in SCS if the “Eurocode #00” calculation method is chosen.

Steps in calculating the distribution of forces (reduction of individual row capacity if group capacity is smaller) from the top to the bottom gives final resistances mentioned above.

These resistances are compared by SCS with web compression capacity (460.9 kN, see above). So at the end the software checks the horizontal equilibrium:

$$\text{Row1} + \text{Row2} + \text{Row3} = 244.5 \text{ kN} + 332.2 \text{ kN} + 85.9 \text{ kN} = 662.6 \text{ kN} > 460.9 \text{ kN}$$

Considering Row3 = 0, then Row2 = 460.9 kN – 244.5 kN = 216.4 kN

Moment capacity is: 244.5 kN x 0.527 m + 216.4 kN x 0.442 m = 224.5 kN

$$M_{\perp \text{ Rd, bolt gr.}} = 224.5 \text{ kN}\cdot\text{m} \quad \text{formula}$$

As worked example gives 224 kNm, (difference is less the 0.25%).

Stiffness coefficient have some differences due to:

- SCS considers $z = z_{eq} = 447.9 \text{ mm}$ (6.31 #00) in calculation of k_1 ;
- SCS considers $d_c = \text{depth} - 2 \times \text{flange thickness}$ (the book subtracts also 2xradius).

Final value of stiffness $S_j = 72,6 \text{ MN}$. The book provides $S_j = 75.21 \text{ MN}$.

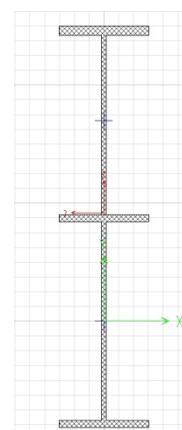
26.6 Section property validation

Calculated values are compared to values by Sap2000 (CSI Berkeley).

SCS geometry input:

Secondary Member:	IPE450	Depth=	400.0 mm
h=	450.0 mm	Angle (≤ 45)=	13 °
b=	190.0 mm	<input type="checkbox"/> Fl.-web same as sec.	
t _f =	14.6 mm	t _{f haunch} =	20.0 mm
t _w =	9.4 mm	t _{w haunch} =	10.0 mm
r=	0.0 mm		

Sap geometry input:



Comparison of results:



IPE450 + Haunch: depth 400 mm, web thk 10, flange 190 x 20		
A gross=	17103.5	mm ²
A gross shear=	7892.8	mm ²
I strong=	156328098	mm ⁴
I weak=	28182692.5	mm ⁴
r strong=	302.3	mm
r weak=	40.6	mm
Wel S strong=	3444550.6	mm ³
Wel S weak=	296659.9	mm ³
Wpl Z strong=	4337524.4	mm ³
Wpl Z weak=	462825.5	mm ³

Cross-section (axial) area	17103.52
Moment of Inertia about 3 axis	1.563E+09
Moment of Inertia about 2 axis	28182692
Radius of Gyration about 3 axis	302.3262
Radius of Gyration about 2 axis	40.5928
Section modulus about 3 axis	3444551.
Section modulus about 2 axis	296659.92
Plastic modulus about 3 axis	4337524.
Plastic modulus about 2 axis	462825.5

Difference on results is < 0.1%.

26.7 SNiP validation

SCS was approved as per the following certificates:

СИСТЕМА СЕРТИФИКАЦИИ ГОСТ Р ФЕДЕРАЛЬНОЕ АГЕНТСТВО ПО ТЕХНИЧЕСКОМУ РЕГУЛИРОВАНИЮ И МЕТРОЛОГИИ	
	СЕРТИФИКАТ СООТВЕТСТВИЯ
№ RA.US.AB86.H01159 Срок действия с 05.04.2019 по 04.04.2021 № 0351059	
ОРГАН ПО СЕРТИФИКАЦИИ RA.RU.11AB86	
ООО ЦСПС. Орган по сертификации программной продукции в строительстве 125057 г. Москва, Ленинградский проспект, дом 63, тел. (499) 157-1990	
ПРОДУКЦИЯ Программа SCS – Steel Connection Studio для проектирования соединений стальных конструкций различного назначения	код ОК 58.29.29.000
<i>обеспечение программное прикладное прочее на электронном носителе, серийный выпуск</i>	
СООТВЕТСТВУЕТ ТРЕБОВАНИЯМ НОРМАТИВНЫХ ДОКУМЕНТОВ	
ГОСТ Р ИСО 9127-94, разд.6, пп.6.1.1, 6.3.1, 6.5.1, 6.5.2; ГОСТ Р ИСО/МЭК 12119-2000, разд.3, пп.3.1.1, 3.2.1 – 3.2.5; ГОСТ Р ИСО 9126-93, разд.4, пп.4.1 - 4.4; ГОСТ 28195-89, разд.2, п.2.1 (пп.1.1, 1.2, 2.1 - 2.3, 3.1 - 3.3, 6.1, 6.2); ГОСТ 28806-90, разд.2, пп.13 – 16	код ТН ВЭД
ИЗГОТОВИТЕЛЬ Steel Studio Inc. 775 Pleasant St. – Suite 5, East Weymouth, MA, 021189, USA, tel.: +1 617 418 3154	
СЕРТИФИКАТ ВЫДАН Steel Studio Inc. 775 Pleasant St. – Suite 5, East Weymouth, MA, 021189, USA tel.: +1 617 418 3154	
НА ОСНОВАНИИ	
Заключение ООО ЦСПС № 01-09-19 от 02 апреля 2019 г. на 2-х страницах	
ДОПОЛНИТЕЛЬНАЯ ИНФОРМАЦИЯ	Схема сертификации 3
М.П.	Руководитель органа  подпись
Эксперт	С.Д.Ратнер инициалы, фамилия Т.Н.Бубнова инициалы, фамилия
Сертификат не применяется при обязательной сертификации	

ЗАКЛЮЧЕНИЕ № 01-09-19

ОРГАНА ПО СЕРТИФИКАЦИИ ПРОГРАММНОЙ ПРОДУКЦИИ В СТРОИТЕЛЬСТВЕ
на базе ООО «Центр сертификации программной продукции в строительстве» (ООО ЦСПС)

о соответствии разделам и пунктам нормативных документов
программы SCS – Steel Connection Studio от 02.04.2019г.

(к сертификату соответствия № RA.US.AB86.H01159,
срок действия 05.04.2019 – 04.04.2021)

1. Обозначение программной продукции

Программа SCS – Steel Connection Studio.

2. Название программной продукции

Программа проектирования соединений стальных конструкций различного назначения SCS – Steel Connection Studio.

3. Версия – 19А

4. Назначение программной продукции

Проектирование и расчет соединений стальных конструкций различного назначения с интерактивным графическим сопровождением в виде двумерной модели.

5. Решаемые задачи:

- расчет сварных соединений
- расчет болтовых соединений

6. Соответствует требованиям пунктов нормативных документов по состоянию на 02 апреля 2019г.

ГОСТ Р ИСО 9127-94 "Документация пользователя и информация на упаковке потребительских программных пакетов":

- раздел 6. Справочная документация (ОБ). Подраздел 6.1. Обозначение пакета (ОБ), п.6.1.1. Подраздел 6.3. Функциональное описание программного средства (ОБ), п.6.3.1. Подраздел 6.5. Использование программного средства (ОБ), пп.6.5.1, 6.5.2.

ГОСТ Р ИСО/МЭК 12119-2000 "Информационная технология. Пакеты программ. Требования к качеству и тестирование":

- раздел 3. Требования к качеству. Подраздел 3.1. Описание продукта, п.3.1.1. Подраздел 3.2. Документация пользователя, пп.3.2.1 – 3.2.5.

ГОСТ Р ИСО/МЭК 9126-93 "Информационная технология. Оценка программной продукции. Характеристики качества и руководства по их применению":

- раздел 4 Характеристики качества программного обеспечения, пп.4.1 – 4.4.

ГОСТ 28195-89 "Оценка качества программных средств. Общие положения"

- раздел 2 Номенклатура показателей качества программных средств, п.2.1 (пп.1.1, 1.2, 2.1 - 2.3, 3.1 - 3.3, 6.1, 6.2).

ГОСТ 28806-90 "Качество программных средств. Термины и определения":

- раздел 2 Общие характеристики качества программного средства, пп.13 – 16.

7. Адекватность и эффективность в части прикладных характеристик подтверждена соответствием требованиям пунктов нормативных документов по состоянию на 02 апреля 2019г.

Заместитель генерального
директора ООО ЦСПС

Эксперт

Д.Ю.Бубнов

Т.Н.Бубнова

стр. 2 Заключения к сертификату
соответствия № RA.US.AB86.H01159
(срок действия 05.04.2019 – 04.04.2021)

СП 16.13330.2017 - Актуализированная редакция СНиП II-23-81* "Стальные конструкции":

- раздел 4. Общие положения. Подраздел 4.2. Основные расчетные требования, пп.4.2.4 - 4.2.7.
- Подраздел 4.3. Учет назначения и условий работы конструкций, пп.4.3.1, 4.3.2;
- раздел 6. Расчетные характеристики материалов и соединений, пп.6.1 - 6.9.

8. Программная документация

SCS – Steel Connection Studio Manual, 163 с.

**Заместитель генерального
директора ООО ЦСПС**

Эксперт

Д.Ю.Бубнов

Т.Н.Бубнова

27 Technical issues

Different technical matters are here discussed

27.1 Moment connections: formula to find Y_c in column when only 2 bolts are present without transverse stiffener

Formulas for the above situations are given in Aisc Design Guide (DG) 16 page 18 for the beam but, if the column has no transverse stiffener (in which case the applied formula is the same as for the beam), Design Guides give no specific guidance.

SCS uses the following formula in this case (symbols are the same as in DG4):

$$Y_c = \frac{b_{fc}}{2} \left[h_0 \frac{1}{s} + h_0 \frac{1}{s} \right] + \frac{2}{g} [h_0 \cdot s] = \frac{b_{fc} \cdot h_0}{2s} + \frac{2sh_0}{g}$$

The formula is supposed to be conservative and it's calculated comparing similar situations in DG4 (see page28) and DG16 (as above). If the engineer/user is not comfortable with the above formula, it's suggested to insert a transverse stiffener to go back into the case of DG16.

27.2 Instantaneous center of rotation method with no vertical shear

If no vertical shear is input but an eccentricity moment (or axial action) is applied to the bolt group, the elastic method should be used. If the inst. center of rotation method is chosen, a vertical shear of 0.45kN=0.1kips per bolt is considered in the solution to help convergence.

27.3 SNIp notes

SNIp are an old standard though still used in some ex USSR countries. The updated and more recent code (2011 and 2017) is also available. Some difficulties have been met in programming SCS to SNIp calcs due to the fact not much material (commentary, explanations and examples for connections) is available. The following notes try to explain the approach of our technical team.

27.3.1 Block shear

SNIp gives no instruction on how to calculate block shear in connections, therefore the Eurocode approach and calculations are followed. This seems the most conservative approach and Eurocode rules have been chosen over other standards for geographical reasons. If the user wants to follow literally the SNIp and doesn't want to consider block shear, he/she can avoid considering block shear results.

By the way, block shear is calculated using $\gamma_{M2}=1.25$ as per standard Eurocode value. The operation coefficient γ_c is applied to formulas.

In other cases the Eurocode has been followed where there are no instructions in standard #70, #71 or #72.

27.3.2 Compression and buckling

Compression/buckling checks are performed according to the updated code of SNiP (SP16.13330 2011 and SP16.13330 2017).

27.3.3 Bolt bearing and minimum distances

Exact minimum distances required by SNiP are in table 39 of #70. SCS simplified approach is to consider 1.3 times d_0 (bolt hole) as in case $2*d_0$ (high strength bolts) of the aforementioned table. Bearing coefficients is taken 0.8 or 0.75 as per part 2 of table 35 if the distance of the bolts is $1.5*d_0$ from edge (or $2*d_0$ between bolts) but this value is adjusted linearly if the distance is more or less, considering the linear interpolation passing through the coordinate zero and having as maximum the value for edge distance equal to $4*d_0$.

Example:

Say that a group of M20 ($d_0=22$) bolts is at $2*d_0=44$ from edge and has $3*d_0=66$ as bolt to bolt distance in both directions. If material is S275, the bearing coefficient base value is 0.8 and it's adjusted to $\gamma_{bolt}=0.8/1.5*2=1.067$ since edge distance is more than $1.5 d_0$ (bolt to bolt distance would give a higher value, so it's not governing).

For SP16.13330 2011 bolt bearing and minimum distances are as per tables 40 and 41 of #71 and for SP16.13330 2017 are as per tables 40 and 41 of #72.

27.3.4 Gamma coefficients

γ_m is the material coefficient (table 2 of #70, table 3 of #71 and of #72). 1.1 is the default value given by SCS.

γ_c is the operation condition coefficient (table 6 of #70, table 1 of #71 and of #72). This is used for member and weld check formulas. 1 is the default value given by SCS.

$\gamma_{c \text{ bolt}}$ is the operation condition coefficient (table 35 of #70, upper part, table 1 of #71 and of #72) for bolts. The coefficient ranges from 0.9 to 1. 0.9 is the default value given by SCS.

γ_b is the operation condition coefficient in formula (131) of #70, 192 of #71 and #72 depending on the number of bolts.

γ_h is the reliability coefficient (table 36 of #70, table 42 of #71 and #72) used for slip resistant connections. 1.3 is the default value given by SCS.

$\gamma_{concrete}$ is the concrete partial safety factor used for base plate calculations as per Eurocode.

27.3.5 Weld check

Each weld should be checked at 2 locations for SNiP, as per fig. 20 of #70, fig.20 of #71 and fig.20 of #72, that is at border of fusion (subscript z) and at joint (subscript y). Each check has a different β (table 34 of #70, table 39 of #71 and of #72), a different γ_w (actually usually 1) and different R_w (see table 3 and 4 of #70 or table 4 of #71 and of #72 to calculate them).

SCS suggest an R_w value (considering $\gamma_w = 1$) that, combined with a $\beta=0.7$ should provide a quite conservative check. If this is not accurate enough or it doesn't verify the joint, the user should check welds applying values as per SNiP tables just mentioned.

27.4 AS 4100 notes

27.4.1 Block shear and long joints

The same considerations under the homonym paragraph in the “Snip Notes” apply.

27.5 Double sided connections

27.5.1 Moment end plates (em3 and we1)

SCS doesn't allow to choose a double sided connection in modules em3 and we1 because it would create some unnecessary complexity in calcs.

In facts, a double sided moment connection with the beams from opposite sides having tension on the same, say, top flange and compression on the bottom flange it's beneficial for the column web panel shear (and it doesn't influence other limit states), therefore it's conservative to design for one sided connection only. It would be unconservative (it can happen for wind moment frames) for the web panel shear to have the beam on one side with tension on top flange and the one on the other side with compression on the top flange because this would add up to web panel shear. If moment have the same value on the opposite sides, the web panel shear force would be double the value of the one sided connection.

Therefore, in the situation of moments acting as described above (lateral loads prevailing over gravity loads), the user would have to consider an increased web panel shear (double the reported utility ratio for the single sided if moments have same value).

In all the other situations, it is suggested to design the double sided connection as single sided (note that compression buckling is always checked even though #24 suggests it is actually present only if a double sided connection is in place, see par. 2.2).

For more information see #42 and #24

27.6 Base plate calculations according to AISC Design Guide 1

SCS can provide solution according to #22 to problems where more rows of bolts work to balance bending moment.

Formula 3.4.3 in #22 works if only one row of bolts is in tension.

SCS finds a solution also in the case more rows of bolts are in tension, starting from equations 3.4.2 in #22.

In the case where more solutions can be possible (say one with only one row of bolts in tension and another one with 2 rows of bolts), SCS will choose the one with more rows in order to reduce tension per bolt. This might not be beneficial for minimum required plate thickness though, that could increase if more rows of bolts are input.

27.7 Properties of some profiles according to the AISC manual

It has been noted that some properties provided by SCS following structural analysis rules slightly differ from the properties provided in the AISC manual for the same profiles. We have inquired the AISC steel solution center with the following question:

“We would like to understand how the properties of Angles and C-Shapes in the AISC Manual were calculated. Our calculations (checked also with other commercial tools) show that the

properties listed do not match "exact" values. It seems for example that, for Angles, the radii in the shapes were not taken into consideration when calculating the Area. But the other values (I, S, r, Z) still do not match with our calcs..... so, what are the exact hypothesis considered when calculating shape properties for Angles and C-Shapes?"

We have received this answer:

"The area will be similar, but the heel and toe radii are accounted for. When calculating the angle just based on the leg widths and thicknesses, the heel fillet radius would add to this area whereas the radii at the toes of the angle would reduce this area so the resulting area would be similar to one calculated just based on the leg widths and thicknesses. We do not provide the equations used to generate the section properties for the different structural shapes provided in the AISC manual. While the values you calculate may not be exactly the same as those shown in the manual tables, they should be close enough that any differences for the purpose of design would be considered negligible. For example, just calculating the area of an L8x8x1 1/8 angle using the width and thickness of the angle only and ignoring the contribution of the fillet radii at the heel and toes, I get an area equal to 16.73 in² vs. the 16.8 in² in the manual. This works out to a difference of about 0.3%. The permitted ASTM A6 area tolerance for an angle as shown in Table 1-25 is - 2.5%."

27.8 Japanese standard notes

A few comments about standard #110:

- 1) The choice of the category shear "slip resistant at serviceability" is available, but this option is actually not provided in the Japanese Code. Checks as per C11.6.1/C11.6.9 of #110 are included in the option "slip resistant at ultimate".
- 2) Weld checks are as per 11.5.2 of #110. SCS calculates the weld stresses as per Eurocode approach ("directional method", 4.5.3.2 #00). Final percentage is calculated as a ratio between stresses and not between the actions and the respective capacities. Check for combined actions (axial force, bending moment, shear force) as per C11.5.3 is calculated more conservatively, without quadratic elevation.
- 3) Check as per C11.6.3 is calculated more conservatively, without quadratic elevation.
- 4) Combined shear and tension in bolts (bearing connections) follows Eurocode approach (table 3.4 #00) since no exact guidance is provided in #110.

27.9 Prying action check in shear connections (modules XL1, XL2, QS5, QS6)

Prying action check, as per moment connections, is included in T-stub formulas when Eurocode methods are chosen.

When connection is on the column flange, analytical solutions are provided both for beam side (end plate or double angles) and column side (flange). When connection is on column web, prying action is checked only for beam side (end plate or double angles) and column web is checked for tying capacity.

When connection is beam to beam (XL2, QS6), prying action is checked only for secondary beam side (end plate or double angles). Primary beam web is checked for tying capacity. As literature does not show how to calculate tying capacity for beam web, SCS considers the analytical solutions for column web of [41] and [28] adapting the theory to a beam web.

Stiffeners on the back side of primary member web can be inserted to control the tie force.

28 API

API (Application Programming Interface) is now available in SCS for building macros with VB.NET or C# and make SCS work with other software.

Check website www.scs.pe for updated examples.

SCS targets 4.0 version of .NET framework.

It's possible to build macros using Visual Studio Express 2010 (free distribution, check Microsoft website) or later versions.

The namespace of SCS APIs is mySCS. Add as a reference SCS.exe in the installation folder

Note:

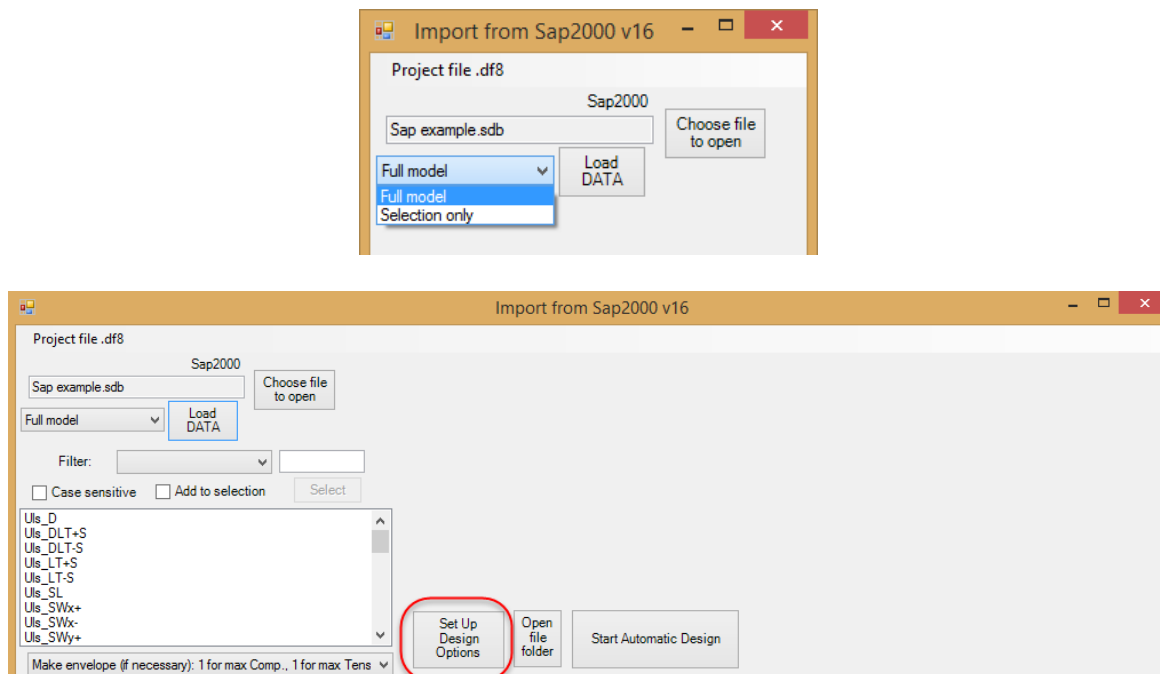
It might be necessary to set 'x86' as target architecture in Visual Studio to make SCS API work.

29 Automatic design by using Sap2000, Etabs, Staad

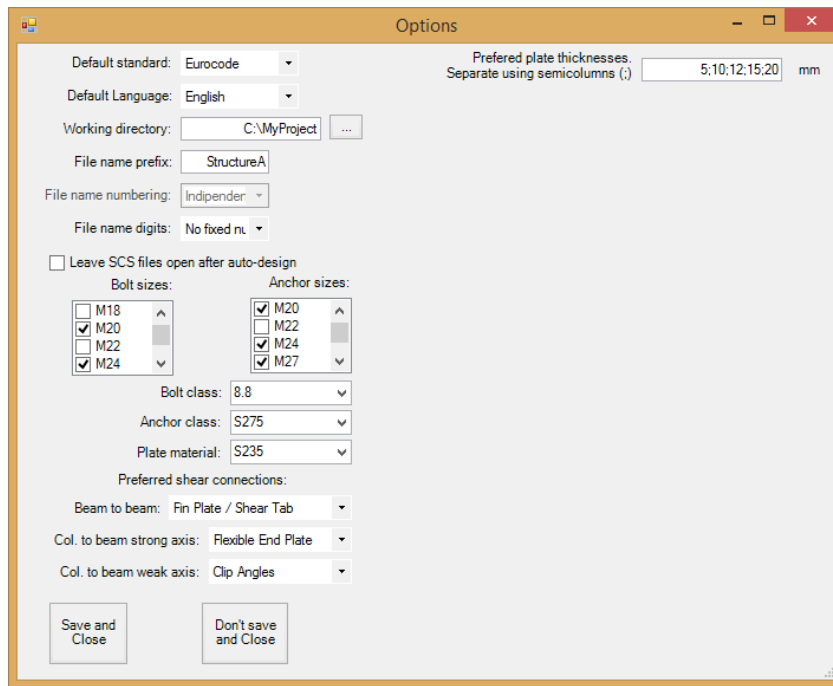
This application permits to the user to design automatically joints of structures modeled in **SAP2000, Etabs and Staad** according to load combinations provided in the model and to technical choices of the designer.

29.1 General concepts

The full model or a selected part of it can be imported.

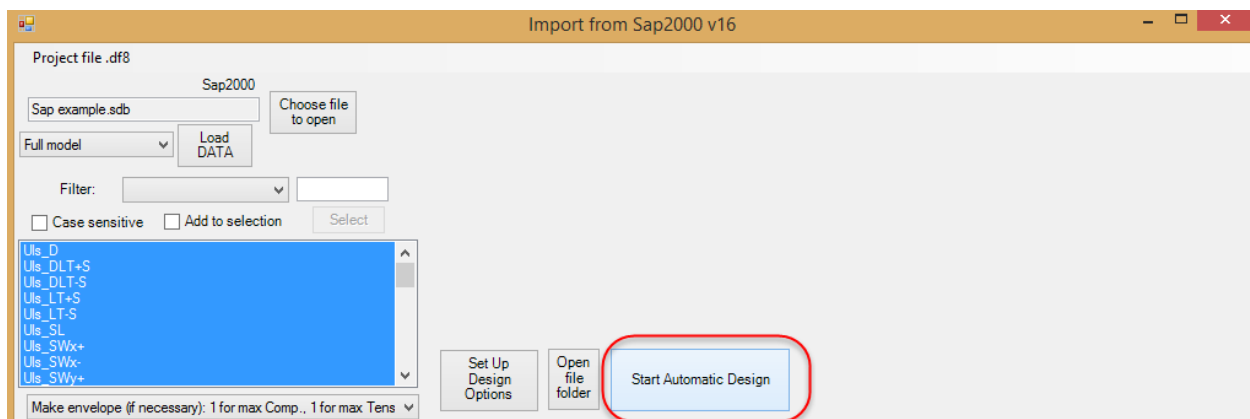


Once imported the model, the user, with the support of the button “Set Up Design Options”, can manage following items:

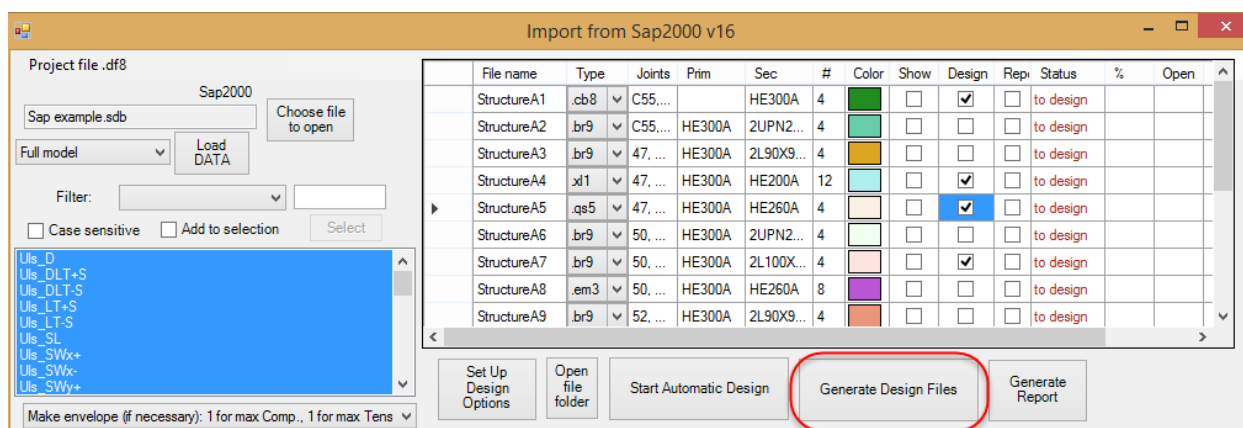


- The Default Standard to design between the several implemented codes;
- the working directory where designed joints will be located;
- personalization of file names, adding a prefix that can help the designer to distinguish designed files according to her/him will;
- preferred bolt and anchor bolts diameters and materials;
- preferred thicknesses and material for plates;
- preferred pin connections beam-beam or beam-column.

SCS automatically applies the choices of the designer when trying to autodesign the joints.



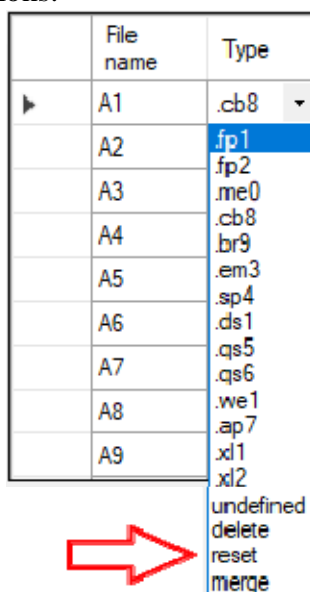
The first step of automatic design (for selected combinations) is the identification of joints in the model with the use of the button “Start automatic design”. SCS recognizes pin connections (to which applies choices of the designer done in “Set Up Design Options”), moment connections, brace connections, base plates, splices...



A list of joints is generated. Types of connection, associated joints (with the label adopted in the reference FEA software) and their number in the model (#), primary and secondary members are shown.

In this phase the user can choose to change some connection before starting the design of each joint. For example, if chosen preferred connections beam to columns are flexible end plate (XL1), but for a particular joint it is more suitable a fin plate (FP1), the designer can change the type for the singular case.

In the “Type” column of the connection table, user has also the possibility to make the following operations for each group of connections:



- “delete”: delete the group of connection from the “Automatic Design” process
- “reset”: return to the original situation cancelling the effects of the "delete" or "merge" operation
- “merge”: merge a group of connections in to another one. This option transfers all the joints of one group to another group selected by the user.

"Merge" option allows the user to merge connections between them (when considered similar by the user) and reduce the number of connections to be verified.

	File name ▲	Type	Joints	Prim	Sec	#	Color	Show
→	T2	und...	40, 45, 50, 55		L80X8	4		<input type="checkbox"/>
→	T3	.br9	41, 42, 46, 47, 51, 52, 56, 57	HE160A	L80X8	8		<input type="checkbox"/>
→	T4	.br9	41, 46, 51, 56	HE160A	L60X6	4		<input type="checkbox"/>
	T5	.em3	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	HE160A	16		<input type="checkbox"/>
	T6	.fp1	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	IPE140	16		<input type="checkbox"/>
	T7	.br9	42, 43, 47, 48, 52, 53, 57, 58	HE160A	L80X8	8		<input type="checkbox"/>
	T8	.br9	43, 48, 53, 58	HE160A	L80X8	4		<input type="checkbox"/>
	T9	.br9	44, 49, 54, 59	HE160A	L80X8	4		<input type="checkbox"/>
*								<input type="checkbox"/>

Set Up Design Options

Open file folder

Reload all data (click if you make any changes in your model)

Generate Design Files

Generate Report

Once the user has identified the two groups of connections that he/she wants to merge, he/she can click on the "merge" corresponding to the group of joints which he/she wants to make part of another connection.

	File name ▲	Type	Joints	Prim	Sec	#	Color	Show	Design	Rep	Status	%	Open	Del	Notes
	T2	und...	40, 45, 50, 55		L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T3	.br9	41, 42, 46, 47, 51, 52, 56, 57	HE160A	L80X8	8		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
▶	T4	.br9	41, 46, 51, 56	HE160A	L60X6	4		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T5	.fp1	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	HE160A	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T6	.me0	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	IPE140	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T7	.cb8	42, 43, 47, 48, 52, 53, 57, 58	HE160A	L80X8	8		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T8	.em3	43, 48, 53, 58	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T9	.sp4	44, 49, 54, 59	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
*		.ds1						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		.qs6						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		.we1						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		.ap7						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		.xl1						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		.xl2						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		undefined						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		delete						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		reset						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
		merge						<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					

Set Up Design Options

Reload all data (click if you make any changes in your model)

Generate Design Files

Generate Report

Then the user will choose the target connection.

	File name ▲	Type	Joints	Prim	Sec	#	Color	Show	Design	Rep	Status	%	Open	Del	Notes
	T2	und...	40, 45, 50, 55		L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T3	.br9	41, 42, 46, 47, 51, 52, 56, 57	HE160A	L80X8	8		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
▶	T4	merge	T1.cb8	HE160A	L60X6	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
	T5	.em3	T10.em3	HE160A	HE160A	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T6	.fp1	T11.br9	HE160A	IPE140	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T7	.br9	T12.sp4	HE160A	L80X8	8		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T8	.br9	T13.fp2	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T9	.br9	T14.fp2	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
*			T15.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T16.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T3.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T5.em3					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T6.fp1					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T7.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T8.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
			T9.br9					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					

Set Up Design Options

Open file folder

Reload all data (click if you make any changes in your model)

Generate Design Files

Generate Report

After this, as shown in the following picture, the transferred group of connections is “disabled” and the name is changed to the hosting group of the connection name. Also, the data of the hosting group of connections is updated (joints name, number of joints).

The load cases of the “merged” group of connection are also transferred to the hosting group, if the joint is later generated. If it is already existing, an update is needed (check the “design” cell and regenerate).

	File name	Type	Joints	Prim	Sec	#	Color	Show	Design	Rep	Status	%	Open	Del	Notes
	T2	und...	40, 45, 50, 55		L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T3	br9	41, 42, 46, 47, 51, 52, 56, 57, 41, 46, 51, 56	HE160A	L80X8	12		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T3.br9	merge	41, 46, 51, 56	HE160A	L60X6	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
	T5	em3	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	HE160A	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T6	fp1	41, 42, 43, 44, 46, 47, 48, 49, 51, 52, 53, 54, 56, 57, ...	HE160A	IPE140	16		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T7	br9	42, 43, 47, 48, 52, 53, 57, 58	HE160A	L80X8	8		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T8	br9	43, 48, 53, 58	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
	T9	br9	44, 49, 54, 59	HE160A	L80X8	4		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	to design				
*								<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					

Set Up Design Options

Open file folder

Reload all data (click if you make any changes in your model)

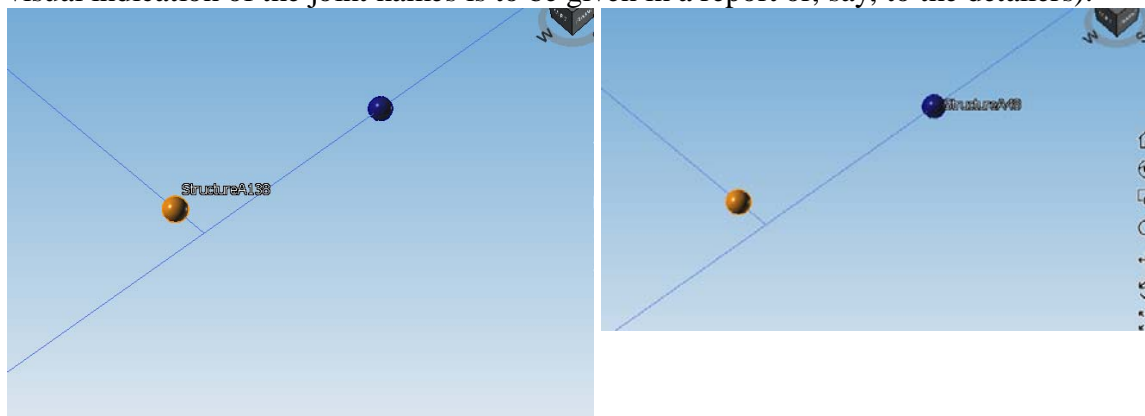
Generate Design Files

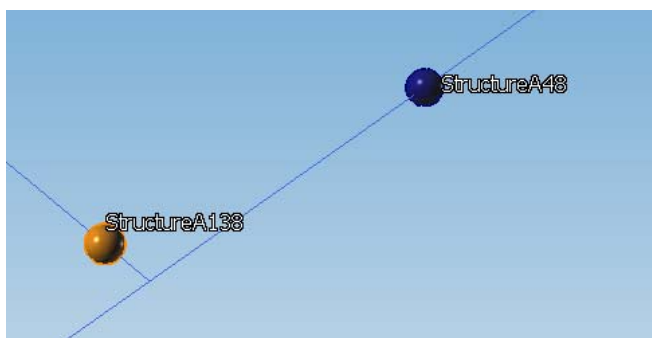
Generate Report

If the profile of the secondary member of the connection group to be merged is different from the hosting group of connections, the profile considered is that of hosting group.

A 3D reproduction of the FEA model is shown in the main window for a quick identification of joints with colored indicators. Colors are automatically associated to each file by SCS and shown in table list: the user can change them according to own will. Selecting the check box in column “show”, joint(s) appear into the 3D reproduction.

By clicking (standard left click) directly over the joints the name of the file name associated with the joint is displayed. By left clicking another joint, the previous one is deleted and the new one is shown. By clicking the second joint (and in case many more) with the mouse right click, the labels of the joints are added consecutively so many can be represented (which may come useful if a visual indication of the joint names is to be given in a report or, say, to the detailers).





The second step is the generation of files (“Generate Design Files”) for selected files in column “Design”.

SCS has an internal routine that tries to design the joints according to what set up in the “Design options previously discussed”. At the end of the routine, SCS gives indications about the outcome of automatic design. Various statuses are provided and shown in the list (see following paragraph). Also notes are shown at the last column of the table list. In this way the design has a clear idea about the general situation: verified joint, overstressed joints, joints with geometrical problems and so on.

It is important to underline that if a specific file is already existing/generated, clicking again the “Generate Design Files” for that selected file will not trigger the automatic design routine again because the software assumes that the user might have done personal changes to the design of the file. On the other hand SCS will:

1. update the load combinations (if the same combinations of the previous file generation are selected, the previous loads will be deleted and the new ones inserted, that is an update will be done)
2. update the materials and/or the member profiles if they changed.

Therefore this is an excellent tool to keep your SCS files updated with the primary design model if any changes are done in either Sap2000, Staad and so on.

29.1.1 Meaning of file statuses

TO DESIGN: this status appears when file is not designed yet.

OK: this status appears when no problems are found. Load conditions are acceptable, no errors occur, no particular troubles user are detected by SCS. Anyway some warnings can be visible in the “message box”, opening the file. The governing limit state exploitation ratio is shown on the right side of the status column.

OVERSTRESSED: this status appears when load conditions for designed joint are over 100%. As above the maximum ratio is shown.

TO CHECK: this status tells the user to manage particular situations. It appears for example in the following cases:

- when the “Start Automatic Design” button is used and SCS finds a file with the same name (prefix+number) but with both secondary and primary members different from the previous ones;

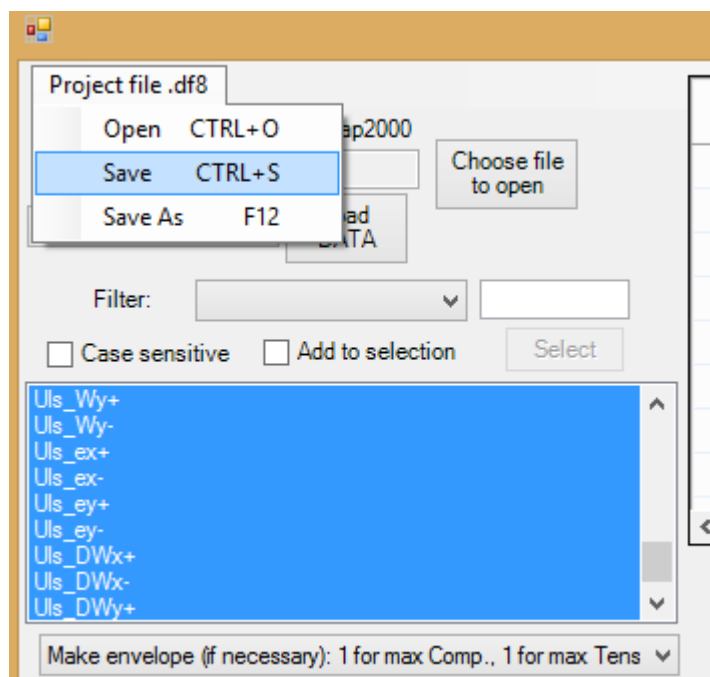
- when SCS finds all loads equal to zero in its load table;
- other situations that require the user to open the file and check the problems.

SOLVE PROBLEMS: this status warns the user about problems not due to an overstressed condition, but due to other circumstances as:

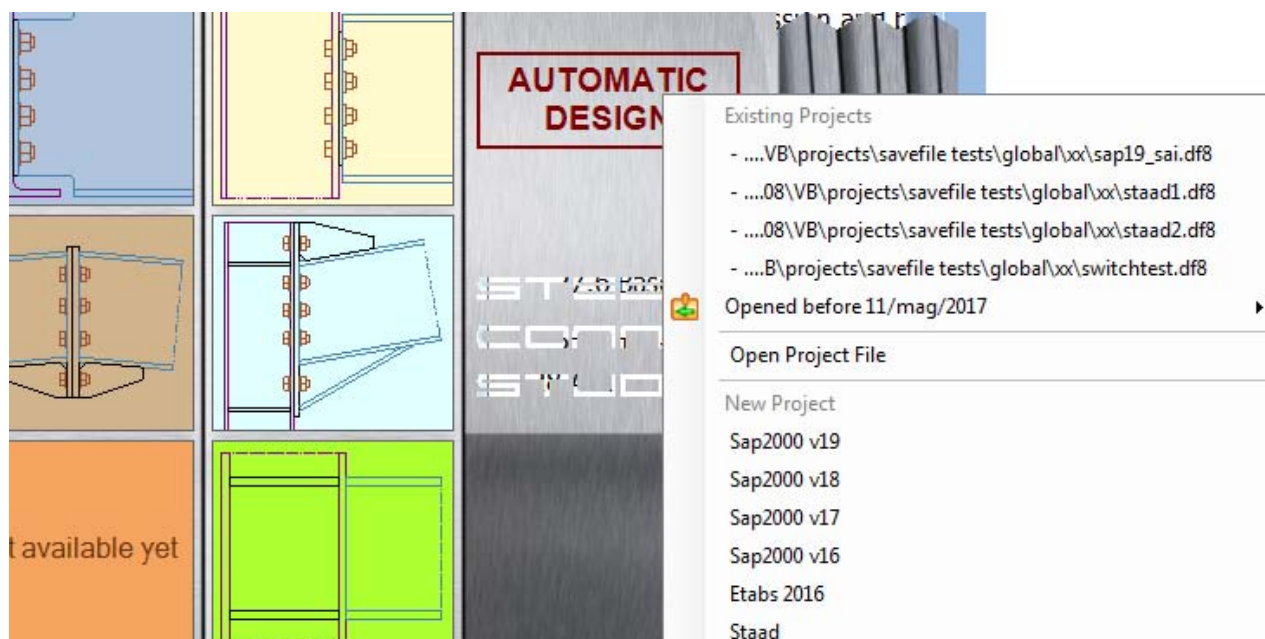
- when in automatic design some errors occurs: opening the file the error is shown in the “message box”;
- when SCS doesn't find some profiles or materials in its database;
- other situations that require the user's attention

29.2 Project file save

Once general options are set and files generated, the designer can save all the information in a project file.



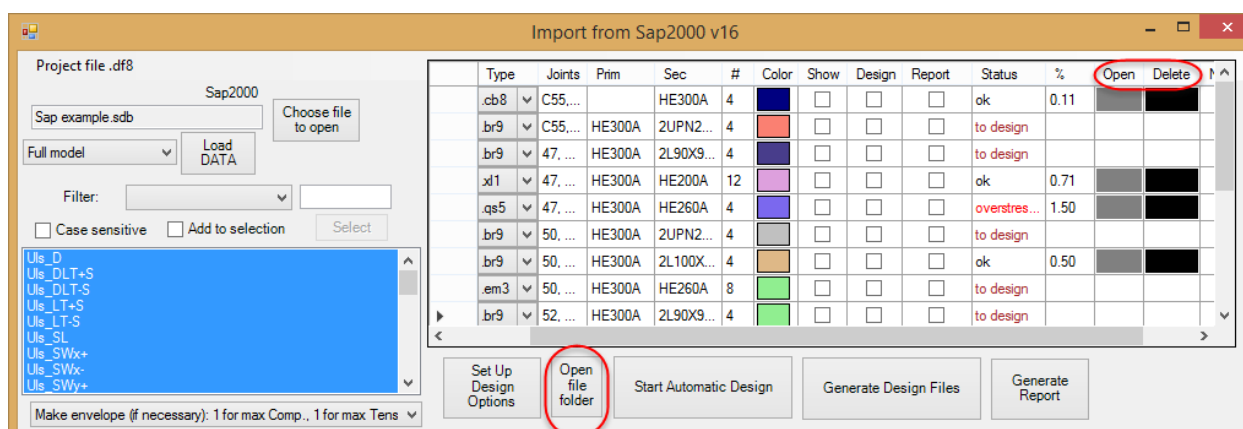
A file (extension .df8) is generated. The file will be visible at the next click of the ‘Automatic design button’ in the main (initial) window among the “Existing Projects”.



29.3 Other notes

When a project is opened and the reference sap model is open, SCS warns the user giving the possibility to choose to open the old file (the one associated to the saved project) or to keep the open file.

The two columns “Open” and “Delete” help the user to make files management faster, opening or deleting them directly from the main window without the need to work in the folder. Anyway the directory can be opened with the button “Open file folder”.



Selecting designed files in column “Report” it is possible to manage the generation of reports for each connection with the support

30 Automatic design by Tekla Structures

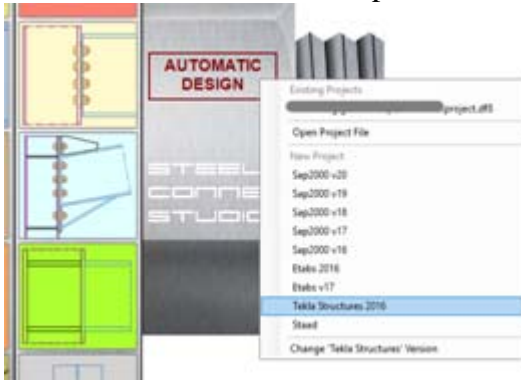
This application allows the user to automatically check joints of structures modeled in **Tekla Structures** software.

Many of the commands and options for this process are similar to those seen in chapter 29, and the following guidelines focus more on the parts that differ.

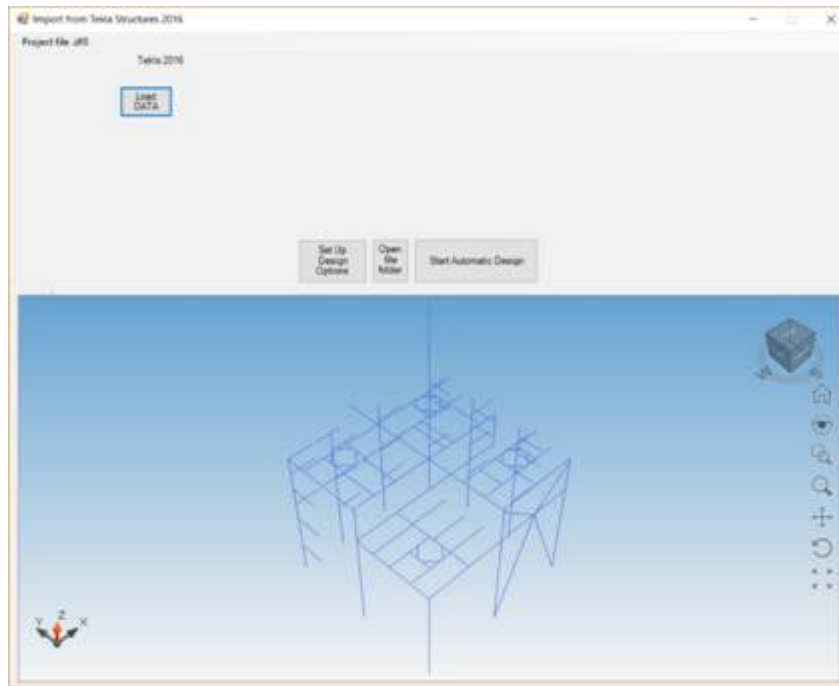
30.1 General concepts

This process permits to the user to import the detailing and load (if any) information of steel structure connections from Tekla and make the automatic design with SCS software.

To begin the Automatic Design process first start Tekla Structures and open the model. Once the model is open, by clicking on the “Automatic Design” button choose the “Tekla Structure” option. If the version of the Tekla Structure software in use differs from the one displayed in the "AUTOMATIC DESIGN" options window, change the version as indicated in section 30.2.



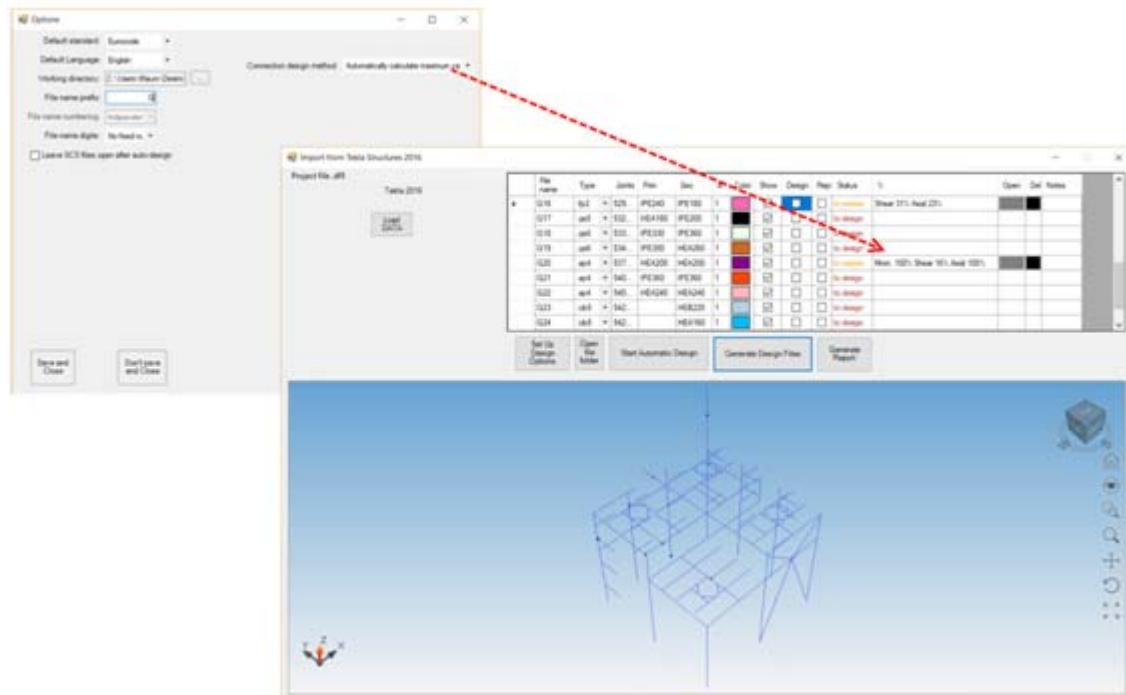
Click on “Load data” in order to import the model in SCS.



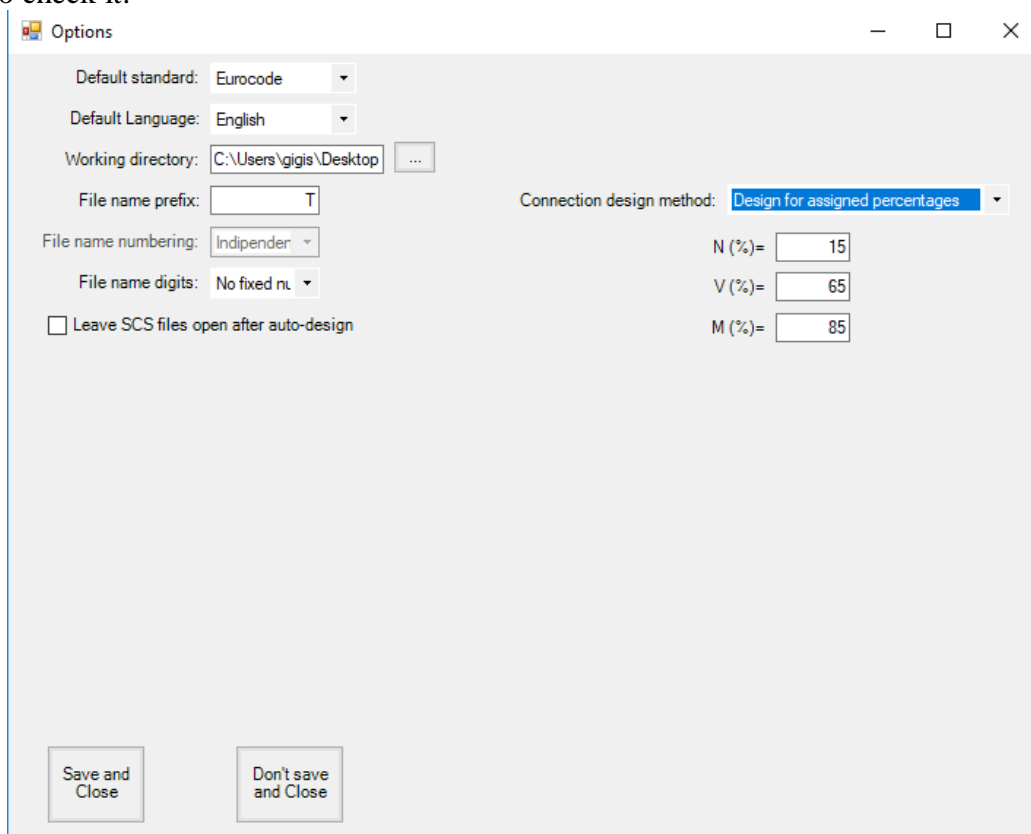
In "Set Up Design Options" there are options to set the desired standard to use for the design, set the entity of loads to be considered, set the names and saving path for the "Design process" files etc.

If the loads are present in the Tekla model on the connections, then that are taken into account for the design of the connections. Otherwise, if no loads are in the Tekla model as it usually happens, SCS offers 2 other possibilities to consider loads:

1. "Automatically calculate maximum value": the maximum forces resisted by that connection are automatically calculated by SCS (shear, axial load and moment value if possible), indicating a percentage of the maximum load that the beam framing into the connection can sustain; the engineer is then supposed to "assess" the connection, opening the SCS files and confirming the loads by either changing them or saving the file (after a save the 'Status' of the connection is changed into 'ok' if loads are acceptable).

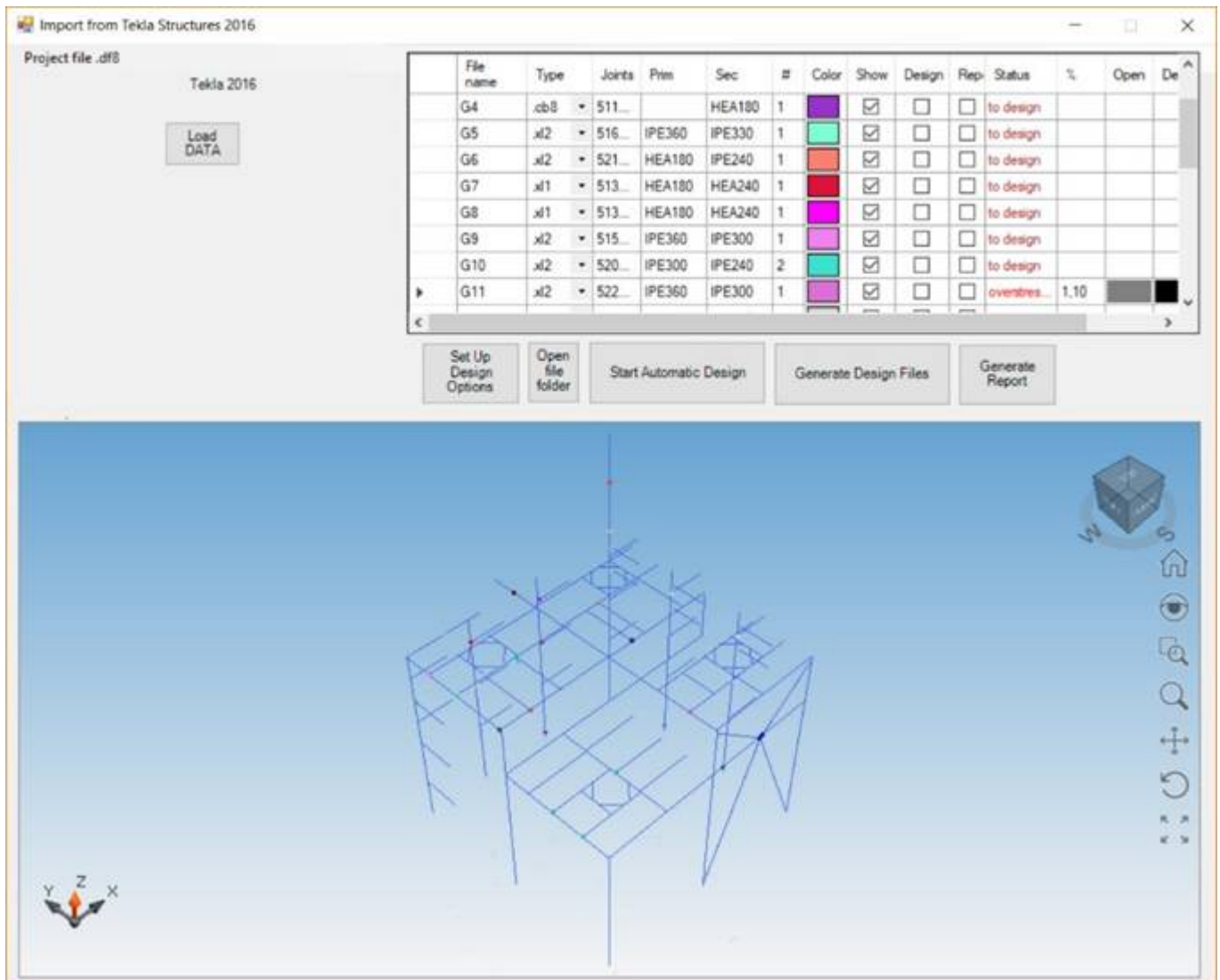


2. "Design for assigned percentages": the user specifies the percentage of the maximum forces (that the beam framing into the connection can sustain) to be input in the connection to check it.



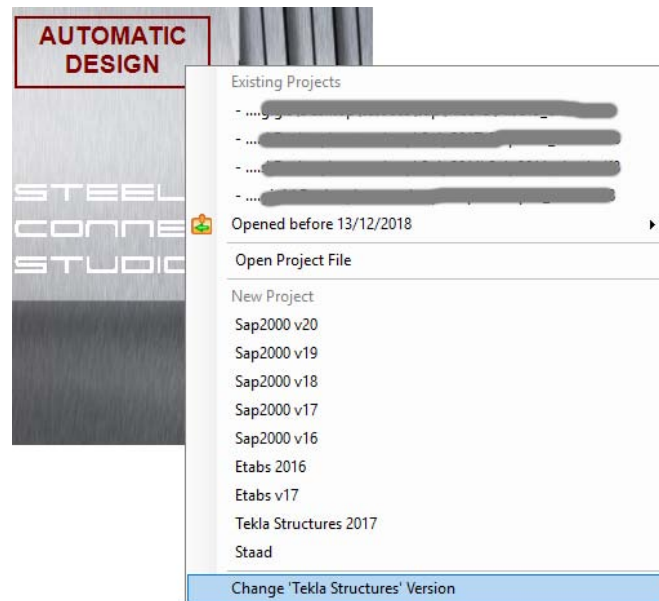
Once everything has been set, SCS is ready to "Start Automatic Design". This command imports the connections of the structure, grouping the ones that are similar.

Then, select the connections that need to be verified in the column "Design" and click the button "Generate design Files" to let SCS create the files as just explained.

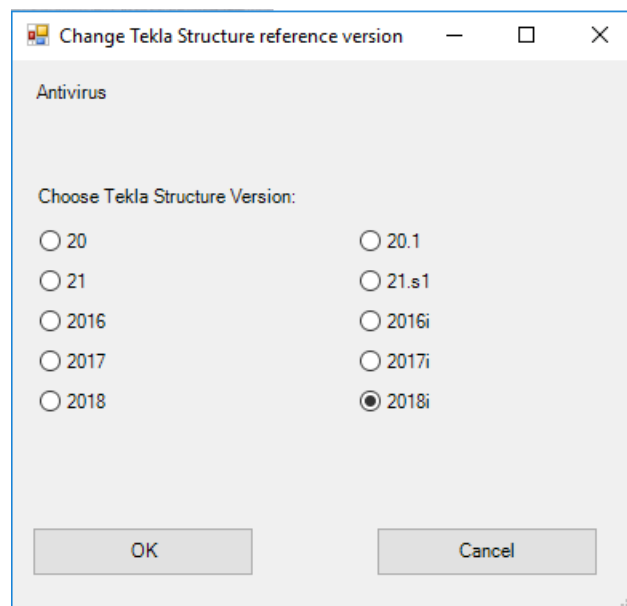


30.2 Change Tekla Structure version

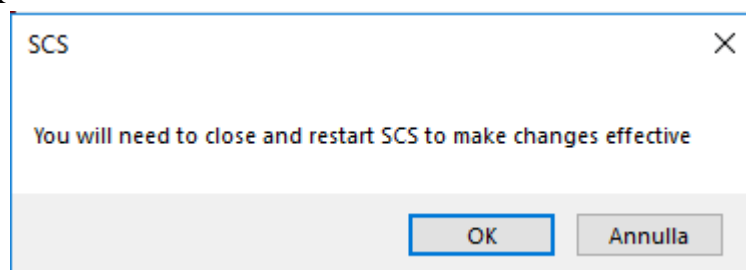
To change the version of the Tekla Structure software from which to import the model for Automatic Design, after clicking on “Automatic Design” button, click on “Change Tekla Structure Version”



Choose the desired Tekla Structure version

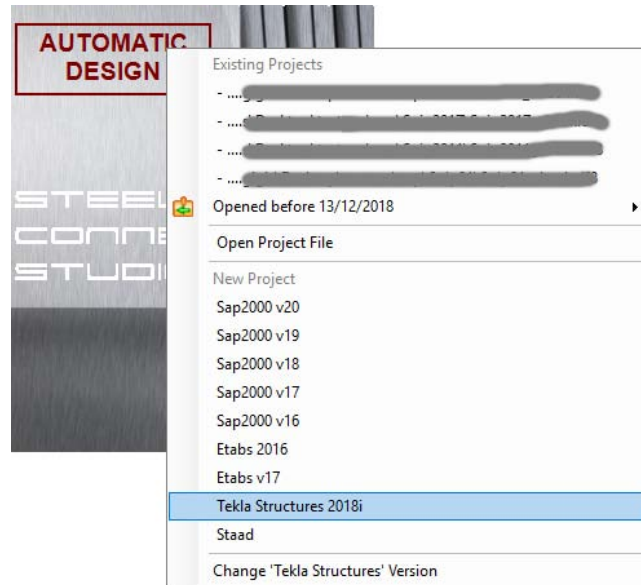


Click on button “ok”



Close and restart SCS.

In this way the version of Tekla Structure software will updates in the “Automatic Design” button window.



30.3 Other notes

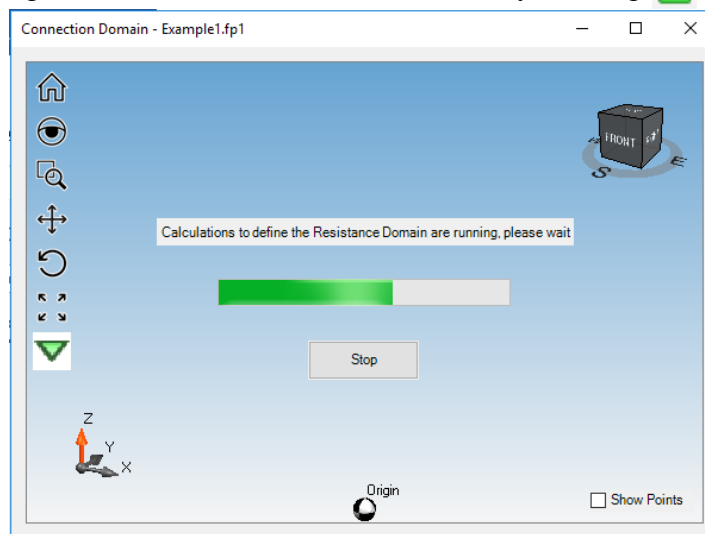
The connections that SCS can import/export with Tekla Structure software are specified in section 6.5.3.

31 Resistance Domains

The generation of resistance domains is created through points that, by connecting to each other's, define the boundaries of the domain. The calculation process used in SCS to figure out these points is iterative, considering each limit state of the selected configuration. Therefore, the shape of the domain is unique for each connection, and it depends on the geometric dimensions, the materials, the load cases, and the standard used to perform the various checks.

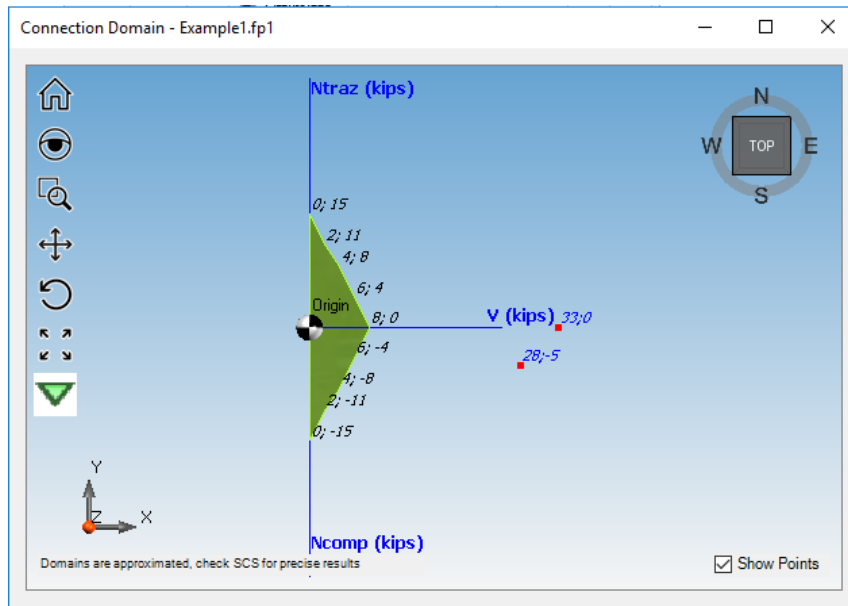
31.1 General concepts

The resistance domain generation starts from the button bar by clicking 



Once the loading bar has completed, the resistance domain is displayed in the same window. Axes and direction names used are as follows:

- “Ncomp”: compression force direction
- “Ntraz”: tension force direction
- “V”: shear force axis
- “Mmax”: positive moment direction (in 3D domains)
- “Mmin”: negative moment direction (in 3D domains)

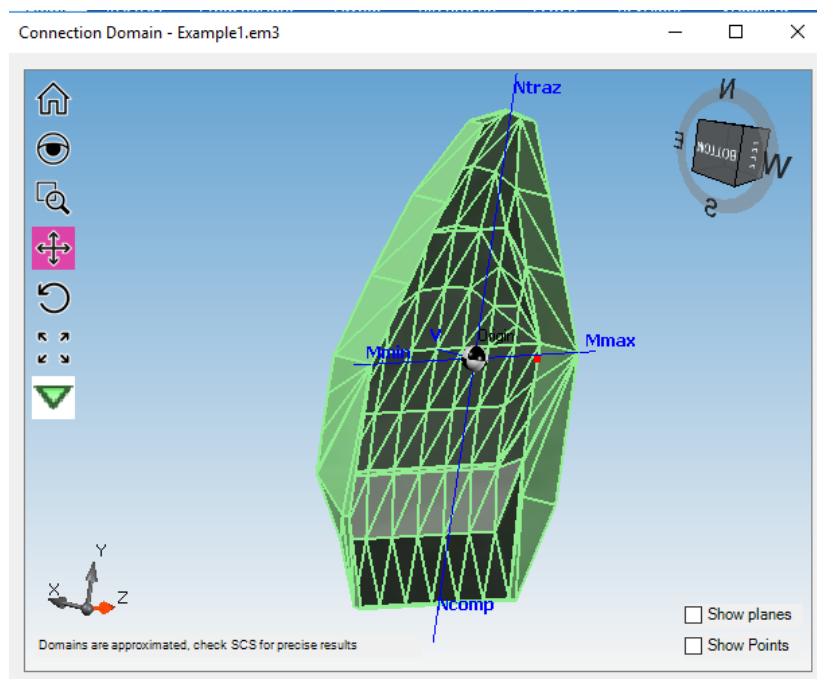


The points corresponding to the load cases are highlighted in red color.

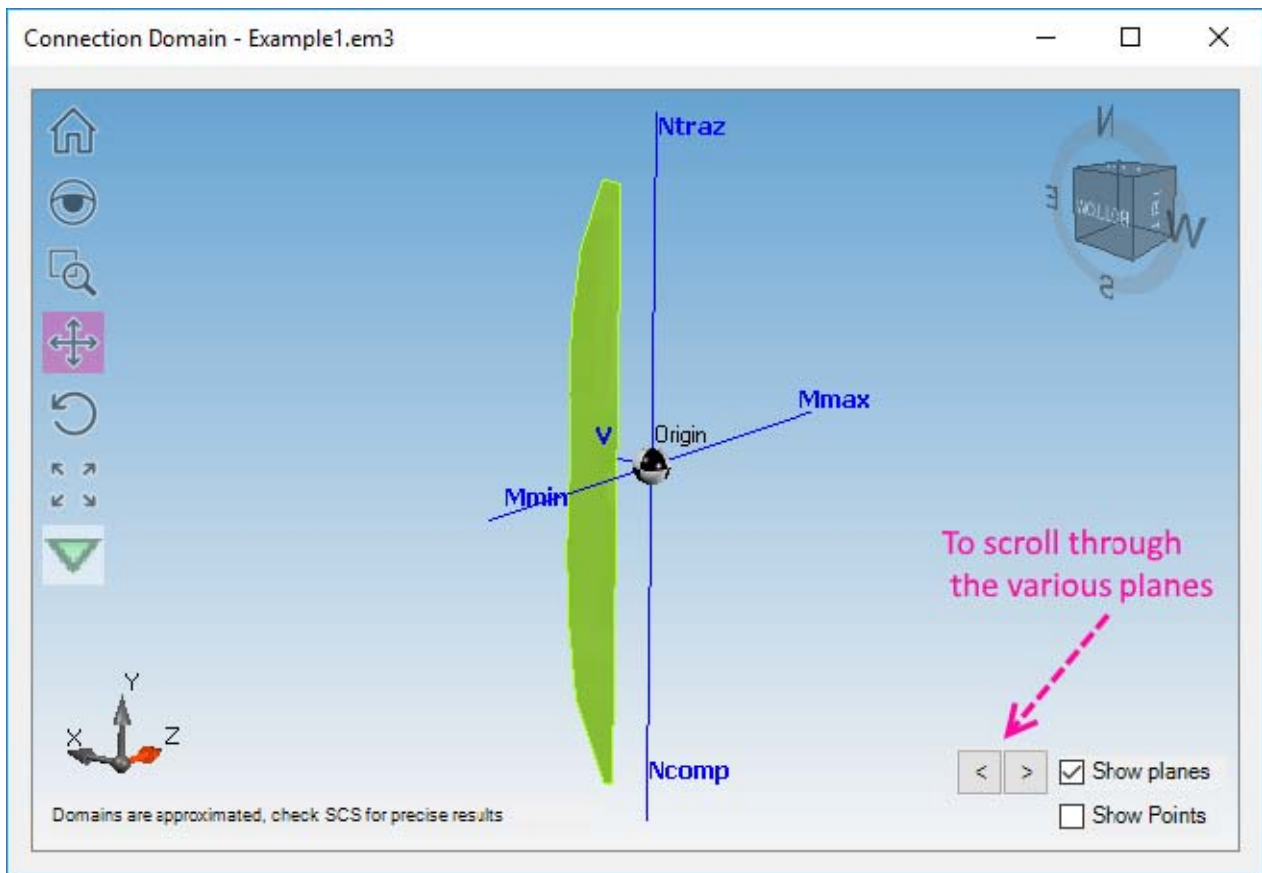
By checking the "Show Points" box, user can see the values of the points used to create the resistance domain (in black color) and those corresponding to the load cases (in blue color). The first component of the coordinates of each point is the value of the axial force, the second is the value of the shear force and in the 3D domains, the third component corresponds to the value of the bending moment.

31.2 3D Domains

For the moment resisting connections, the resistance domains are generated in three dimensions, because bending moment is also included.



In this type of domain, by checking the “Show planes” box, user can browse through the various plans that make the resistance domain.



The 3D resistance domain is available in these modules: AP7, CB8, EM3, SP4 and WE1.

31.3 Export

The resistance domains can be exported as an image in BMP format by clicking on the button



31.4 Other notes

The resistance domains generation is available in the following modules: AP7, CB8, DS1, EM3, FP1, FP2, QS5, QS6, SP4, WE1, XL1 and XL2.

32 Standards and bibliography

- #00) EN 1993-1-8: 2005, DESIGN OF STEEL STRUCTURES, DESIGN OF JOINTS, CEN
- #01) EN 1993-1-1: 2001, DESIGN OF STEEL STRUCTURES, GENERAL STRUCTURAL RULES, CEN
- #02) EN 1992-1-1: 2005, DESIGN OF CONCRETE STRUCTURES, GENERAL RULES AND RULES FOR BUILDING, CEN
- #03) EUROPEAN RECOMMENDATIONS FOR THE DESIGN OF SIMPLE JOINTS IN STEEL STRUCTURES, JASPART, J. P., DEMONCEAU, J. F., RENKIN, S., GUILLAUME, M. L., ECCS - TECHNICAL COMMITTEE 10 - STRUTURAL CONNECTIONS, ECCS GUIDE N° 126, PORTUGAL, 2009
- #04) COLUMN BASE IN STEEL BUILDING FRAMES, F. WALD AND OTHERS, WEYNAND, 1999
- #10) DIN 18800-1: 2008, STEEL STRUCTURES, DESIGN AND CONSTRUCTION.
- #11) DIN 18800-2: 2008, STABILITY, BUCKLING OF BARS AND SKELETAL STRUCTURES.
- #12) DIN 18800-7: 2008, STEEL STRUCTURES, EXECUTION AND CONSTRUCTOR'S QUALIFICATION.
- #20) AISC 360 (2005 / 2010 / 2016), SPECIFICATIONS FOR STRUCTURAL STEEL BUILDING
- #21) AISC DESIGN GUIDE 1 - 1ST ED., DEWOLF, J. T., BICKER, D. T., COLUMN BASE PLATES, USA, 2003
- #22) AISC DESIGN GUIDE 1 - 2ND ED., FISHER, J. M., KLOIBER, L. A., BASE PLATE AND ANCHOR ROD DESIGN, USA, 2010
- #23) AISC DESIGN GUIDE 4 - 2ND ED., MURRAY, T. M., SUMNER, E. A., EXTENDED END PLATE MOMENT CONNECTIONS - SEISMIC AND WIND APPLICATIONS, AMERICAN INSTITUTE OF STEEL CONSTRUCTION, USA, 2003
- #24) AISC DESIGN GUIDE 13, CARTER, C. J., STIFFENING OF WIDE FLANGE COLUMNS AT MOMENT CONNECTIONS, USA, 1999
- #25) AISC DESIGN GUIDE 16, MURRAY, T. M., SHOEMAKER, W. L., FLUSH AND EXTENDED MULTIPLE ROW MOMENT END PLATE CONNECTIONS, USA, 2002
- #26) AISC STEEL CONSTRUCTION MANUAL (13TH / 14TH / 15TH)
- #27) RCSC SPECIFICATION FOR STRUCTURAL JOINTS USING ASTM A325 OR A490 BOLTS, RCSC COMMITTEE A.1, RESEARCH COUNCIL ON STRUCTURAL CONNECTIONS, USA, 2009
- #28) AISC DESIGN GUIDE 29, LARRY S. MUIR, WILLIAM A. THORNTON, VERTICAL BRACING CONNECTION-ANALYSIS AND DESIGN, USA, 2014
- #30) NTC-2008, NORME TECNICHE PER LE COSTRUZIONI, D.M. 14 GENNAIO 2008
- #31) NTC-2018, AGGIORNAMENTO DELLE NORME TECNICHE PER LE COSTRUZIONI, D.M. 17 GENNAIO 2018
- #40) BRITISH STANDARD BS 5950-1: 2000, STRUCTURAL USE OF STEELWORK IN BUILDING – PART 1: CODE OF PRACTICE FOR DESIGN – ROLLED AND WELDED SECTIONS, BSI, 2000
- #41) JOINTS IN STEEL CONSTRUCTIONS, SIMPLE CONNECTIONS, SCI AND BCSA, 2002
- #42) JOINTS IN STEEL CONSTRUCTIONS, MOMENT CONNECTIONS, SCI AND BCSA, 1995
- #50) AISC 341 (2005 / 2010 / 2016), SEISMIC PROVISIONS FOR STRUCTURAL STEEL BUILDINGS
- #51) AISC 358 (2005 / 2010 / 2016), PREQUALIFIED CONNECTIONS FOR SPECIAL AND INTERMEDIATE STEEL MOMENT FRAMES FOR SEISMIC APPLICATIONS
- #52) AISC SEISMIC DESIGN MANUAL (13TH / 14TH / 15TH)
- #60) IS800:2007, GENERAL CONSTRUCTION IN STEEL - CODE OF PRACTICE, BUREAU OF INDIAN STANDARDS
- #70) SNiP II-23-81, CONSTRUCTION NORMS ADN RULES - DESIGN NORMS - STEEL STRUCTURES, STATE CONSTRUCTION COMMITTEE OF THE USSR - GOSSTROY OF THE USSR, MOSCOW 1981

- #71: CODE OF RULES SP16.13330.2011 - STEEL STRUCTURES. REVISED EDITION SNIP II-23-81, RUSSIAN FEDERATION, 2011
- #72: CODE OF RULES SP16.13330.2017 - STEEL STRUCTURES. REVISED EDITION SNIP II-23-81, RUSSIAN FEDERATION, 2017
- #80) CSA S16-09, DESIGN OF STEEL STRUCTURES, CANADIAN STANDARDS ASSOCIATION, CANADA, 2009
- #81) HANDBOOK OF STEEL CONSTRUCTION 10TH EDITION, CANADIAN INSTITUTE OF STEEL CONSTRUCTION, CANADA, 2010
- #82) CSA S16-14, DESIGN OF STEEL STRUCTURES, CANADIAN STANDARDS ASSOCIATION, CANADA, 2014
- #90) AUSTRALIAN BUILDING CODES BOARD, AS4100: 1998, AUSTRALIAN STANDARD - STEEL STRUCTURES, STANDARDS AUSTRALIA, NSW, 1998
- #100) DTR-BC-2.44: ALGERIAN STANDARD - RULES OF DESIGN AND CALCULATION OF STEEL STRUCTURES, MINISTER DE L'HABITAT, 1998
- #110) SSSCS: STANDARD SPECIFICATIONS FOR STEEL AND COMPOSITE STRUCTURES, JAPAN SOCIETY OF CIVIL ENGINEERS (JSCE), 2007
- #120) ABNT NBR 8800: BRAZILIAN STANDARD - DESIGN OF STEEL AND COMPOSITE STRUCTURES FOR BUILDINGS, 2008

33 EULA - End User License Agreement

It's very important the E.U.L.A. (End User License Agreement) is read, understood and fully accepted by the user.

Full acceptance is essential in order to use SCS.

To read EULA, click link in the main form after having clicked 'i' first

