



Tutorial

# Structural Checks according to Standards

Updated on: December 13th, 2023

Tested with: SDC Verifier 2023 R2.1

**SDC Verifier** is a powerful all-in-one software solution for structural design, FEA analysis, and verification according to standards.

This step-by-step tutorial is designed to **get** you **started** with the main SDC Verifier features.  
You will learn how to:

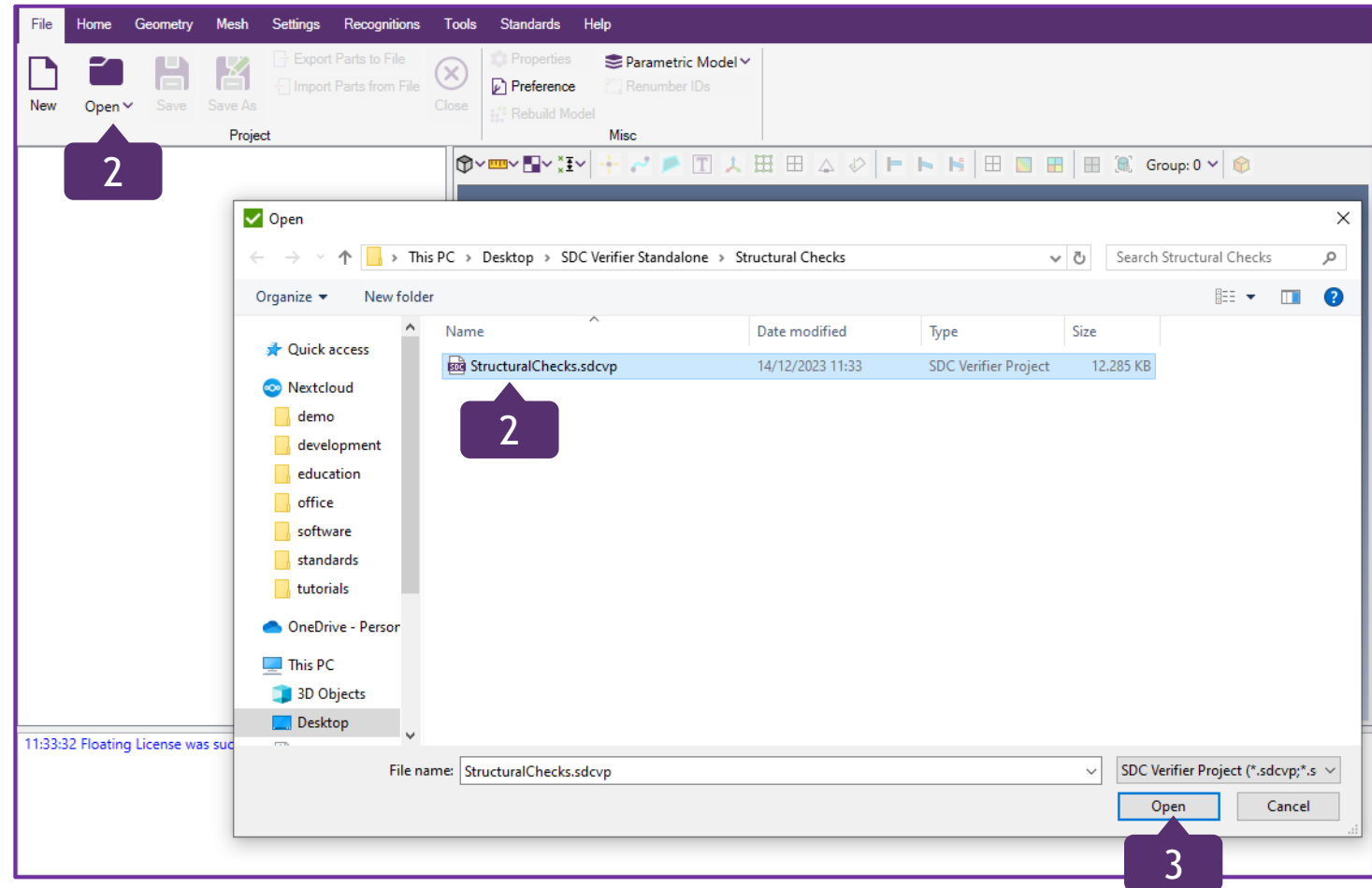
- Launch SDC Verifier;
- Recognize Beam Members;
- Analyse the model with Nastran solver
- Add two Standards: AISC 360-22 Members (2022) and Eurocode3 Fatigue (EN 1993-1-9, 2005);
- Run Beam Member Check;
- Recognise and preview Welds;
- Run Fatigue Check;
- Create Criteria Plot for fatigue Check

# Open the Starter Model

1 Launch SDC Verifier 2023 R2.1

2 Press Open and select project *StructuralChecks*

3 Press *Open*



# Beam Members Recognition

1

In *Recognitions* tab, press on *Beam Member*

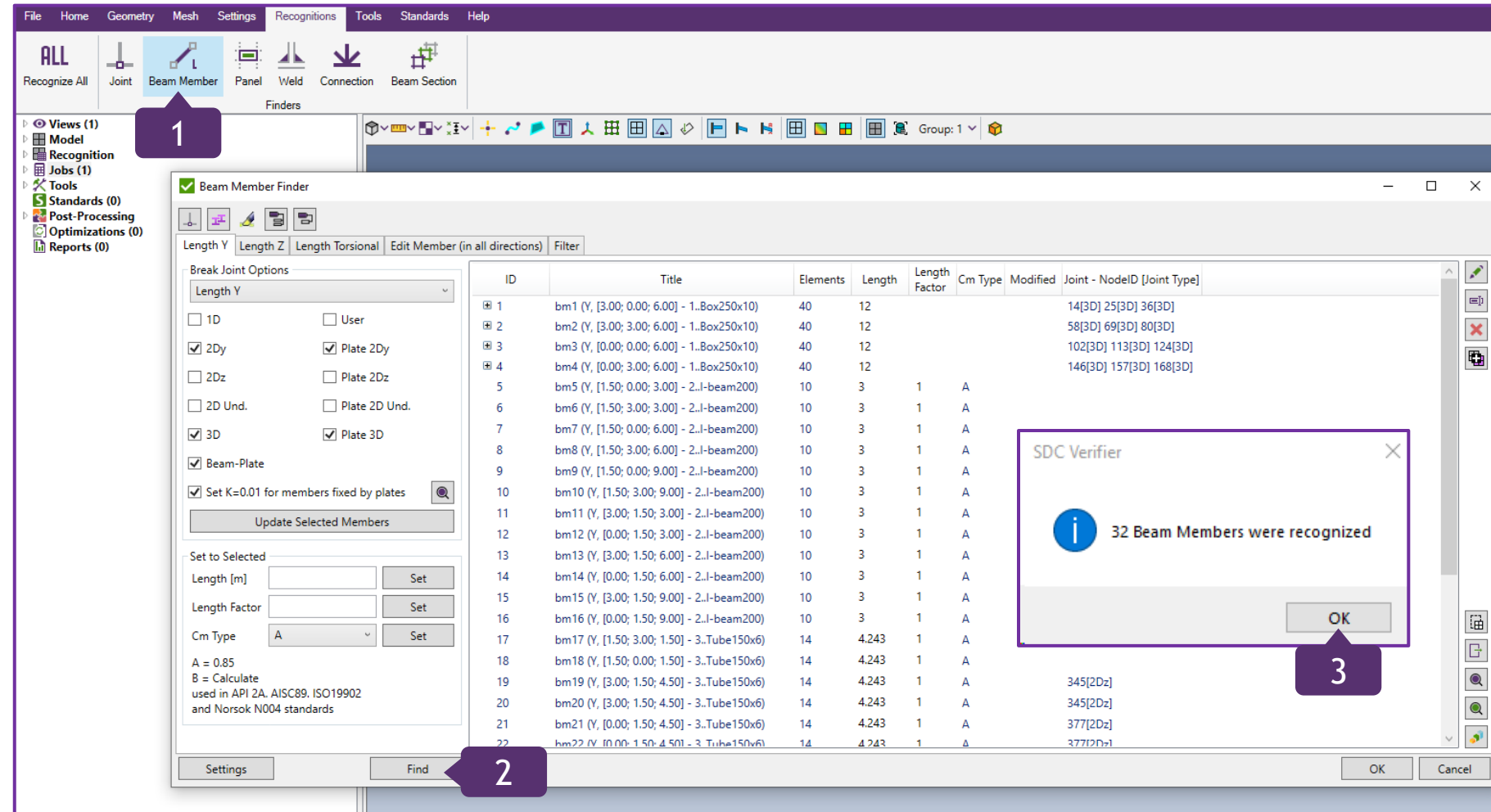
2

Press *Find*

3

Press *OK*

The Recognitions tab provides users with a variety of Recognition Tools to define such structural members as Joints, Beam Members, Panels, Welds, Connections and Beam Sections. Recognize All command allows to define all structural members at once and for a short period of time.



The Beam Member Finder Tool detects the beams on the model in all directions: Length Y, Length Z and Length Torsional.

# Plot Beam Members

1

In Length Y section, with Ctrl+A combination, select all Beam Members

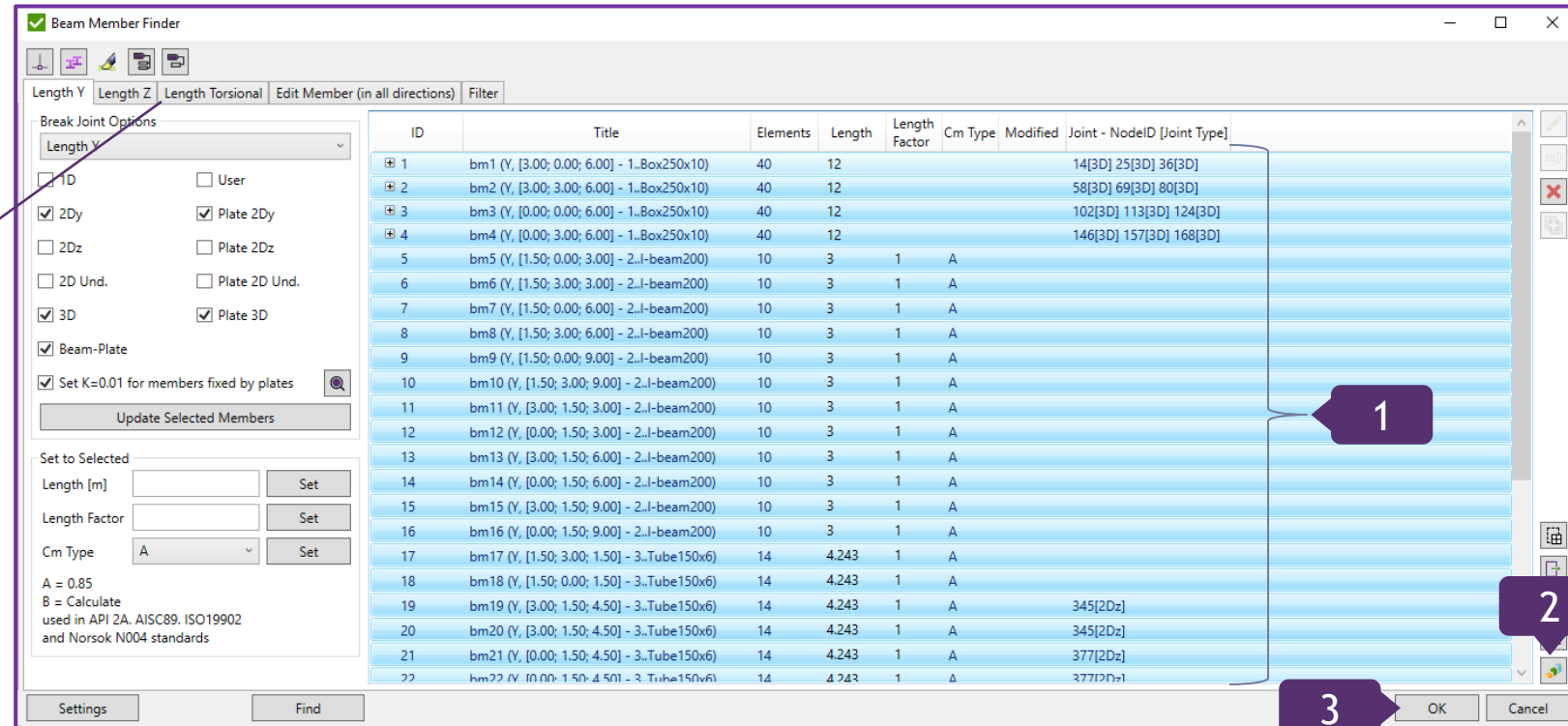
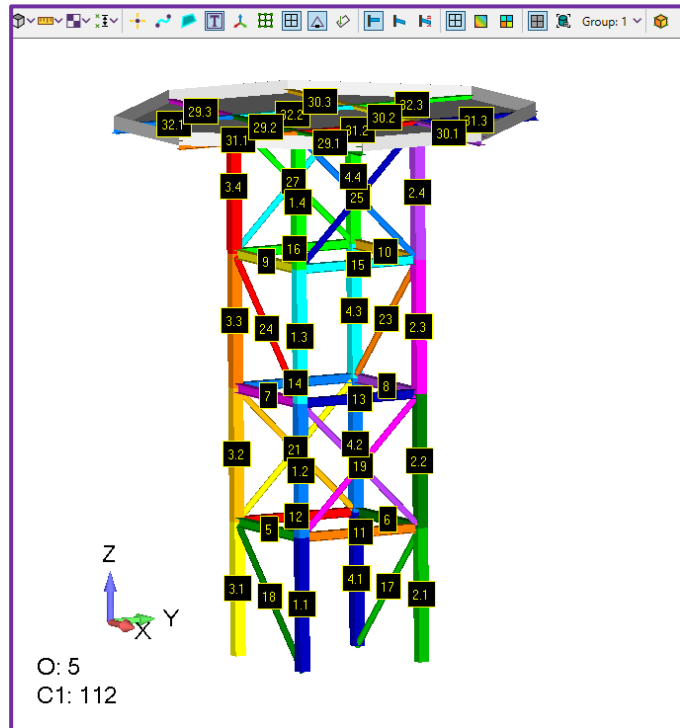
2

Press  and select *Plot Members ID Labels*;

3

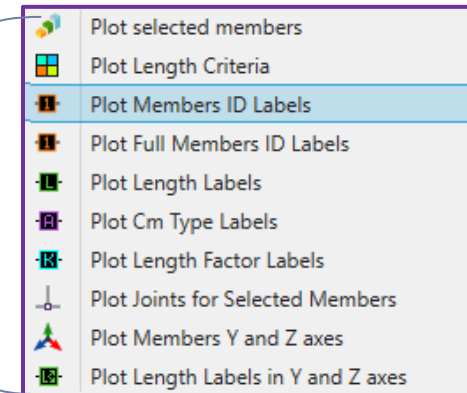
Press OK

Plotting can be done in Length Z and Length Torsional sections as well.



The members have been recognized and the forces will be summarized.

In order to make sure that the recognition has been carried out properly, other Plotting options can be selected .



1

Press **Edit Member (in all directions)** section to modify the configurations

In this case, the big member is split into four sub-members in Joint Node ID 14, 25 and 36

The specific Members can be edited by nodes.

If necessary, the connections can be added or removed in the relevant points to split them manually.

1..bm1 ([3.00; 0.00; 6.00] - 1..Box250x10)  
2..bm2 ([3.00; 3.00; 6.00] - 1..Box250x10)  
3..bm3 ([0.00; 0.00; 6.00] - 1..Box250x10)  
4..bm4 ([0.00; 3.00; 6.00] - 1..Box250x10)  
19..bm19 ([3.00; 1.50; 4.50] - 3..Tube150x6)  
20..bm20 ([3.00; 1.50; 4.50] - 3..Tube150x6)  
21..bm21 ([0.00; 1.50; 4.50] - 3..Tube150x6)  
22..bm22 ([0.00; 1.50; 4.50] - 3..Tube150x6)  
25..bm25 ([3.00; 1.50; 10.50] - 3..Tube150x6)  
26..bm26 ([3.00; 1.50; 10.50] - 3..Tube150x6)  
27..bm27 ([0.00; 1.50; 10.50] - 3..Tube150x6)  
28..bm28 ([0.00; 1.50; 10.50] - 3..Tube150x6)  
29..bm29 ([1.50; 0.00; 12.00] - 2..I-beam200)  
30..bm30 ([1.50; 3.00; 12.00] - 2..I-beam200)  
31..bm31 ([3.00; 1.50; 12.00] - 2..I-beam200)  
32..bm32 ([0.00; 1.50; 12.00] - 2..I-beam200)

Beam Member Finder

1

Length Y Length Z Length Torsional Edit Member (in all directions) Filter

Beam Member 1..bm1 ([3.00; 0.00; 6.00] - 1..Box250x10)

Joint Node ID	Y	Z	Torsional	Joints
14	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	3D
25	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	3D
36	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	3D

Ly 3 3 3 3

Lz 3 3 3 3

Ltor 3 3 3 3

12

Settings Find OK Cancel

# Beam Member Finder Filter Functionality

This slide is devoted to demonstrative purposes of Filter section functionality.

1 Press **Filter** section

2 **By Length [m]** is ON;  
Select **>=** and **5**

3 Press **Filter**

Filter section allows to single out the groups of beams with the similar parameters.

Beam Member Finder

Length Y | Length Z | Length Torsional | Edit Member (in all directions) | **Filter**

Beam Member: Length Y

Criteria

- ☐ By selection (full member in selection)
- ☐ By elements (one element of member fits)
- ☒ **By Length [m]** **>=** **5**
- ☐ Members with Different Elements Orientation

Apply To Selected

Length [m]  Set

Length Factor (K)  Set

Cm Type **A** Set

**Filter**

Current Filter: Beam Member Length Y; By selection All Entities;

Title	Length [m]	Length Factor (K)	Cm Type
1.1..bm1.1 (Y, [3.00; 0.00; 1.50] - 1..Box250x10)	3	1	A
1.2..bm1.2 (Y, [3.00; 0.00; 4.50] - 1..Box250x10)	3	1	A
1.3..bm1.3 (Y, [3.00; 0.00; 7.50] - 1..Box250x10)	3	1	A
1.4..bm1.4 (Y, [3.00; 0.00; 10.50] - 1..Box250x10)	3	1	A
2.1..bm2.1 (Y, [3.00; 3.00; 1.50] - 1..Box250x10)	3	1	A
2.2..bm2.2 (Y, [3.00; 3.00; 4.50] - 1..Box250x10)	3	1	A
2.3..bm2.3 (Y, [3.00; 3.00; 7.50] - 1..Box250x10)	3	1	A
2.4..bm2.4 (Y, [3.00; 3.00; 10.50] - 1..Box250x10)	3	1	A
3.1..bm3.1 (Y, [0.00; 0.00; 1.50] - 1..Box250x10)	3	1	A
3.2..bm3.2 (Y, [0.00; 0.00; 4.50] - 1..Box250x10)	3	1	A
3.3..bm3.3 (Y, [0.00; 0.00; 7.50] - 1..Box250x10)	3	1	A
3.4..bm3.4 (Y, [0.00; 0.00; 10.50] - 1..Box250x10)	3	1	A
4.1..bm4.1 (Y, [0.00; 3.00; 1.50] - 1..Box250x10)	3	1	A
4.2..bm4.2 (Y, [0.00; 3.00; 4.50] - 1..Box250x10)	3	1	A
4.3..bm4.3 (Y, [0.00; 3.00; 7.50] - 1..Box250x10)	3	1	A
4.4..bm4.4 (Y, [0.00; 3.00; 10.50] - 1..Box250x10)	3	1	A
5..bm5 (Y, [1.50; 0.00; 3.00] - 2..I-beam200)	3	1	A
6..bm6 (Y, [1.50; 3.00; 3.00] - 2..I-beam200)	3	1	A
7..bm7 (Y, [1.50; 0.00; 6.00] - 2..I-beam200)	3	1	A
8..bm8 (Y, [1.50; 3.00; 6.00] - 2..I-beam200)	3	1	A
9..bm9 (Y, [1.50; 0.00; 9.00] - 2..I-beam200)	3	1	A

Settings Find OK Cancel

The filtered members by Length

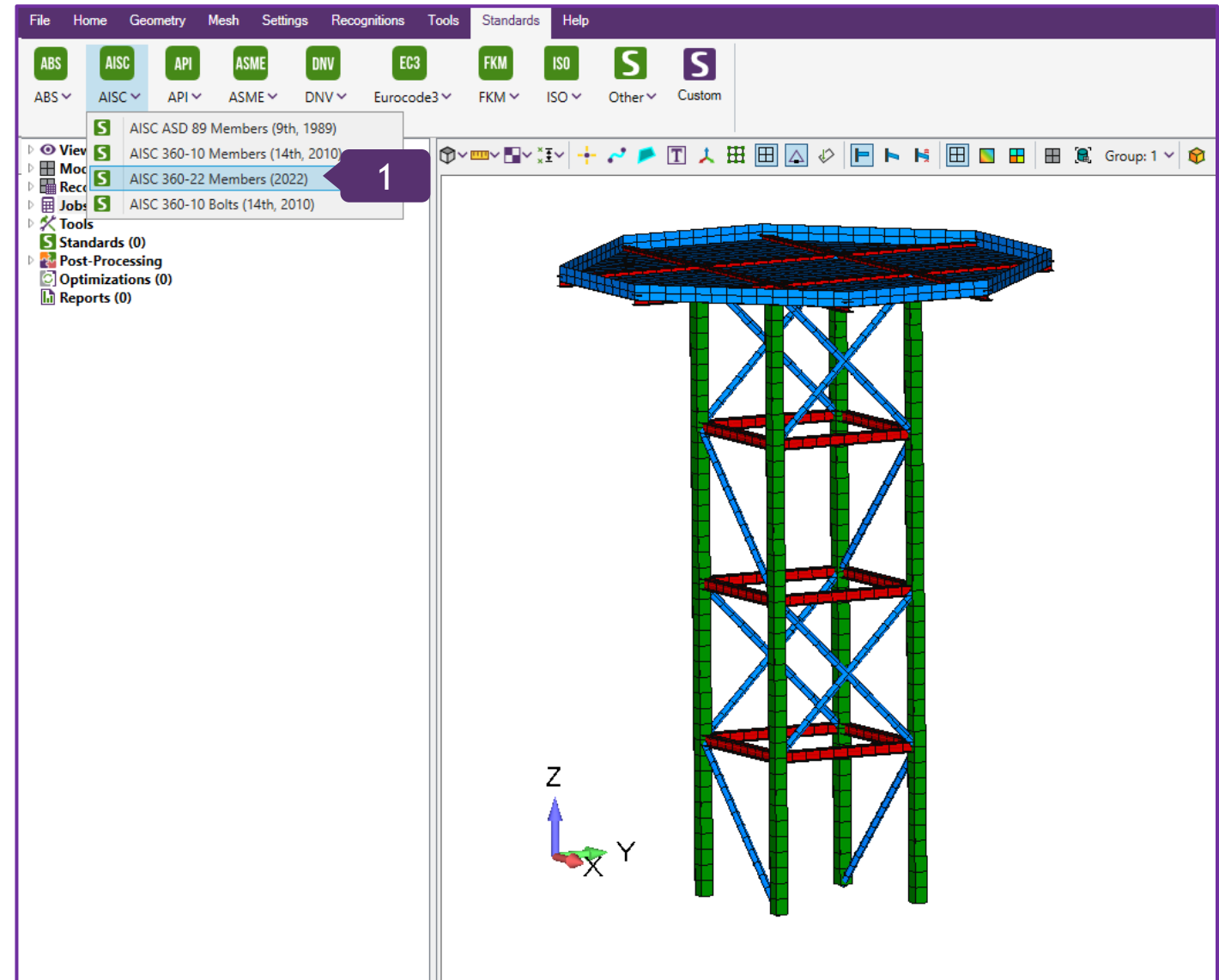
Current Filter: Beam Member Length Y; By Length >= 5			
Title	Length [m]	Length Factor (K)	Cm Type
1..bm1 (Y, [3.00; 0.00; 6.00] - 1..Box250x10)	12	1	A
2..bm2 (Y, [3.00; 3.00; 6.00] - 1..Box250x10)	12	1	A
3..bm3 (Y, [0.00; 0.00; 6.00] - 1..Box250x10)	12	1	A
4..bm4 (Y, [0.00; 3.00; 6.00] - 1..Box250x10)	12	1	A
29..bm29 (Y, [1.50; 0.00; 12.00] - 2..I-beam200)	9	1	A
30..bm30 (Y, [1.50; 3.00; 12.00] - 2..I-beam200)	9	1	A
31..bm31 (Y, [3.00; 1.50; 12.00] - 2..I-beam200)	9	1	A
32..bm32 (Y, [0.00; 1.50; 12.00] - 2..I-beam200)	9	1	A

# Add AISC 360-22 Members (2022) Standard

1

In *Standards* section of the *Ribbon*, press on *AISC* and select *AISC 360-22 Members (2022)*

Standards can be added either through the Model Tree, or in the Ribbon - Standards section.





# Define AISC 360-22 Members (2022) Standard

1

Resistance Factors: *LRFD*

2

In *Section Build Type*, press



3

In *Properties*, *Build Type: Rolled* and press *To All*

4

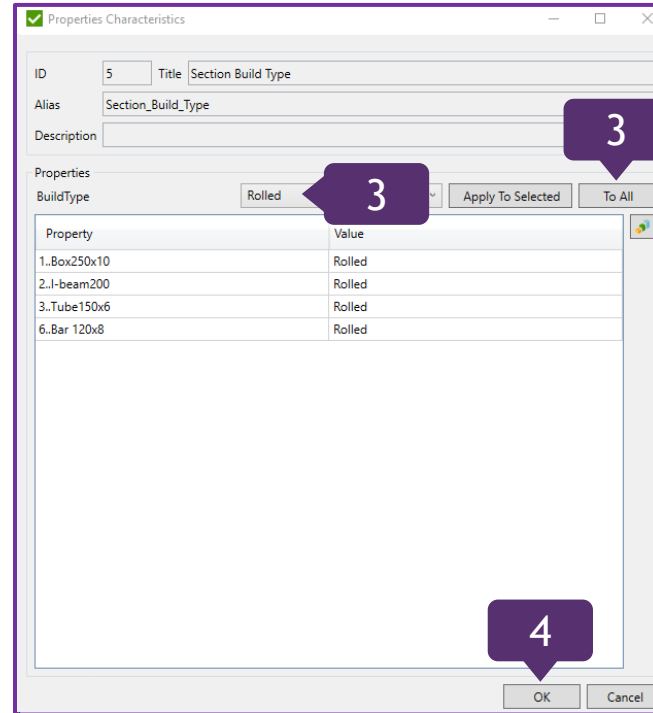
Press *OK*

5

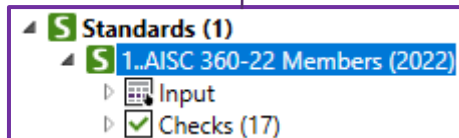
Press  to check the number of supported shapes; Close the window

6

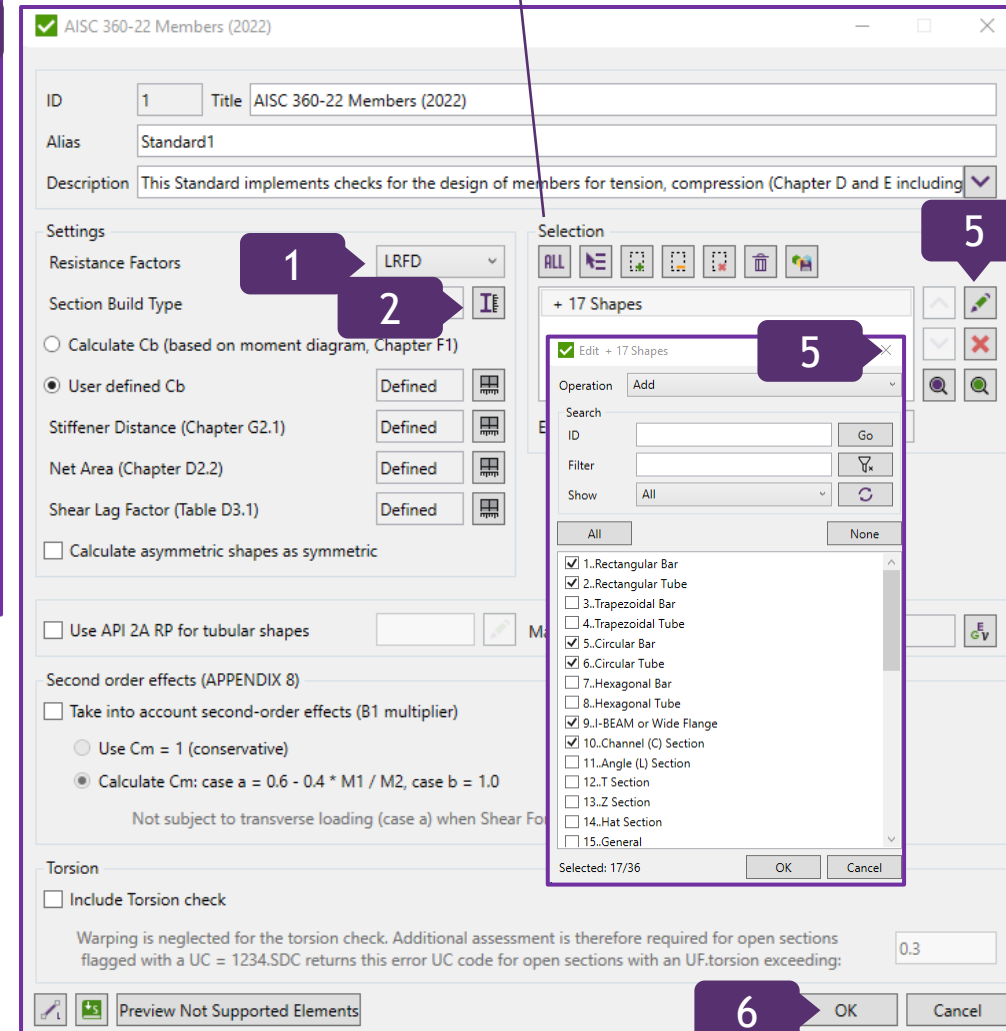
Press *OK*



The Standard has been added.



A selection section gives a possibility to define a certain part of the model to be verified.



When defining the Standard, SDC Verifier informative Help page might be helpful. Press F1 to open it.

# The Access to Formulas in Custom Check

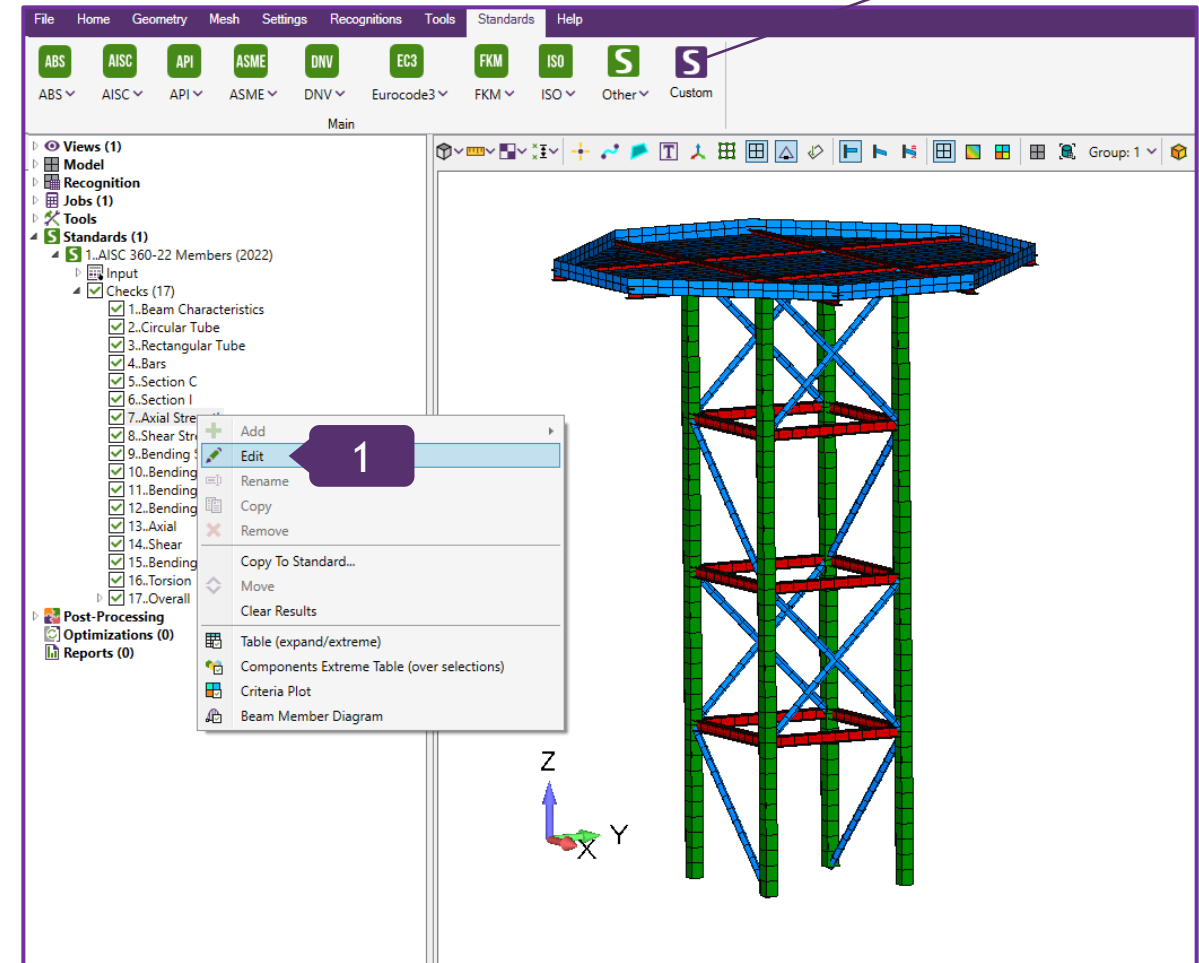
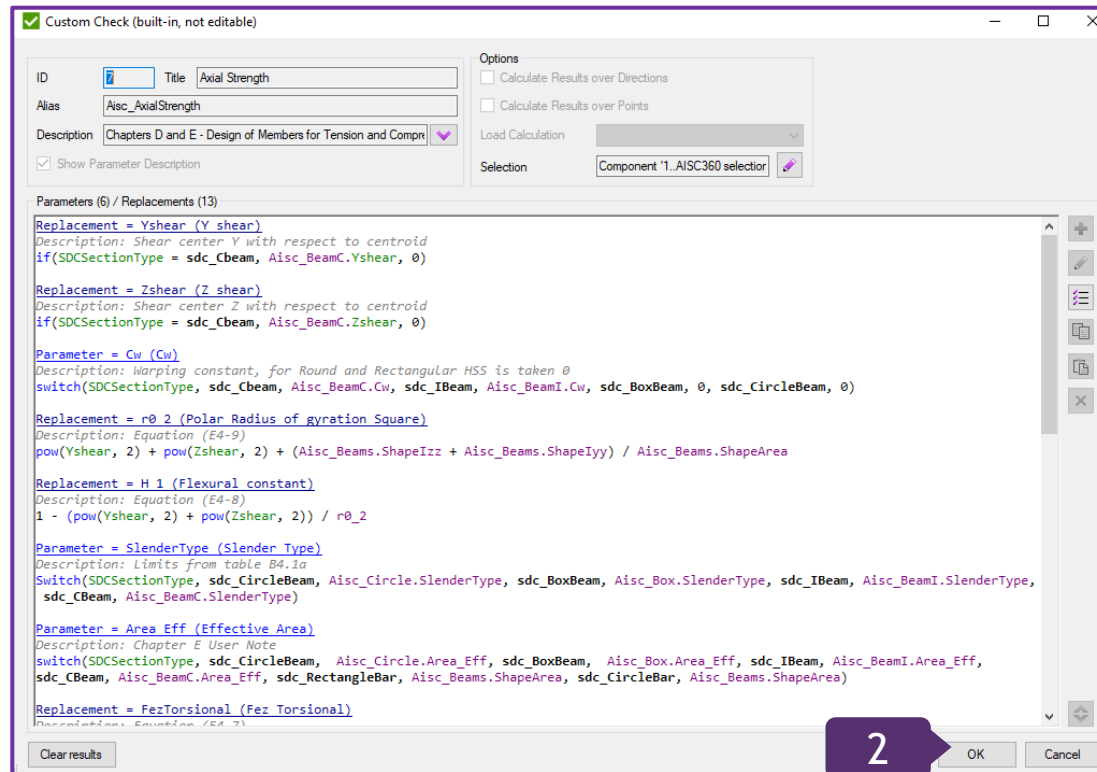
1

Execute right click on **7..Axial Strength** and select **Edit**

2

Press **OK**

The formulas and the calculation procedure of the Checks are open. Every Parameter has a description, in which the source Standard the formula has been taken from is listed. Also, the formulas can be edited and modified, but they need to be converted to Custom Check. To conclude, the formulas can be written from scratch by selecting Custom feature.



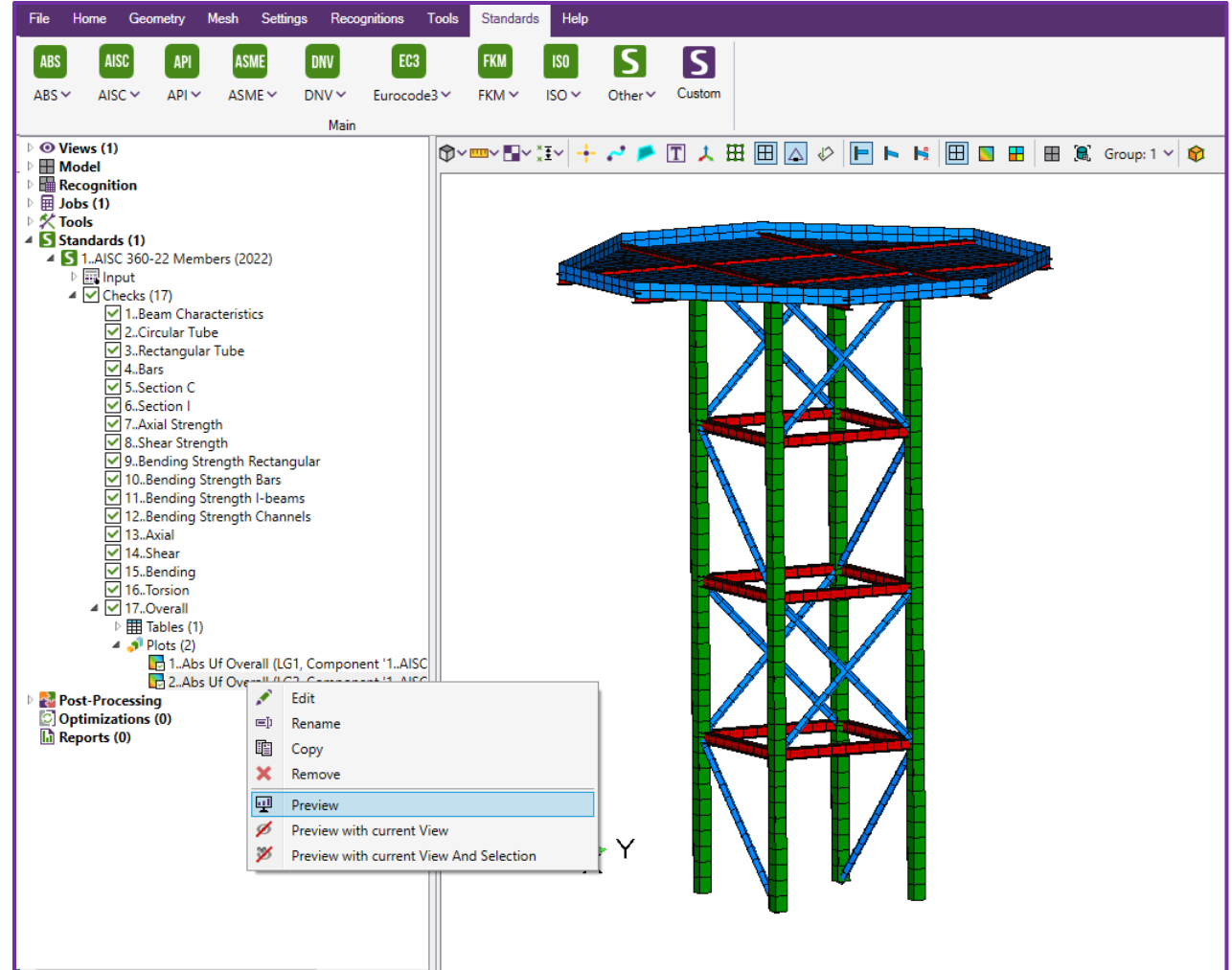
# Preview the Results of the Checks

1

Expand 17..Overall => Plots, and execute right click on 2..Abs Uf Overall

2

Select *Preview*



# Create a Table

1

Expand **Tables**, and execute right click on **1..Over Load Groups**;  
Select **Edit**

2

Loads Count: **Selected loads: 2**;  
Direction/Parameter: **All**;  
Extreme Options: **Absolute**

3

Press **Fill Table**

4

Press **OK**

To obtain more details,  
additional Loads can be applied  
by incorporating this option.

The screenshot shows the SDC Verifier software interface. The top menu bar includes File, Home, Geometry, Mesh, Settings, Recognitions, Tools, Standards, and Help. The Standards menu is open, showing various standards like ABS, AISC, API, ASME, DNV, Eurocode3, FKM, ISO, Other, and Custom. The AISC standard is selected. The left sidebar shows a tree view of the model, including Views (1), Model, Recognition, Jobs (1), Tools, and Standards (1). The Standards (1) folder is expanded, showing 1..AISC 360-22 Members (2022). The 1..AISC 360-22 Members (2022) folder is expanded, showing Input, Checks (17), Tables (1), Plots (2), Post-Processing, Optimizations (0), and Reports (0). The Checks (17) folder is expanded, showing 1..Beam Characteristics, 2..Circular Tube, 3..Rectangular Tube, 4..Bars, 5..Section C, 6..Section I, 7..Axial Strength, 8..Shear Strength, 9..Bending Strength Rectangular, 10..Bending Strength Bars, 11..Bending Strength I-beams, 12..Bending Strength Channels, 13..Axial, 14..Shear, 15..Bending, 16..Torsion, and 17..Overall. The 17..Overall check is selected. The Tables (1) folder is expanded, showing 1..Over Load Groups. The 1..Over Load Groups table is right-clicked, and the context menu is open, showing Edit, Rename, Copy, and Remove. The Edit option is selected. The main window displays a 3D model of a tower structure. The output set is SDC.WorkOutputSet. The elemental contour is End A [S1; Ch17] : LG2; Uf Overall / End B [S1; Ch17] : LG2; The second contour is / Unknown. The third contour is / Unknown.

Load	Uf Axial	Uf Bending	Uf Bending	Uf Shear	Uf Axial ar	Uf Overall
Load Group '1..Envelope (IL)'	0.28	0.45	0.21	0.17	0.76	0.76
Load Group '2..Envelope (LS)'	0.36	0.56	0.27	0.21	0.95	0.95

# Create Additional Load Groups

Vertical Only and Side Only loading cycles will be included in the Fatigue Analysis.

1

Select *Load Groups*;  
Execute right click and select *Add*

2

Title: *Vertical Only*

3

Select *1..Gravity* and *2..Vertical force*;  
Press   
Press *OK*


4

Select *Load Groups* one more time;  
Execute right click and select *Add*

5

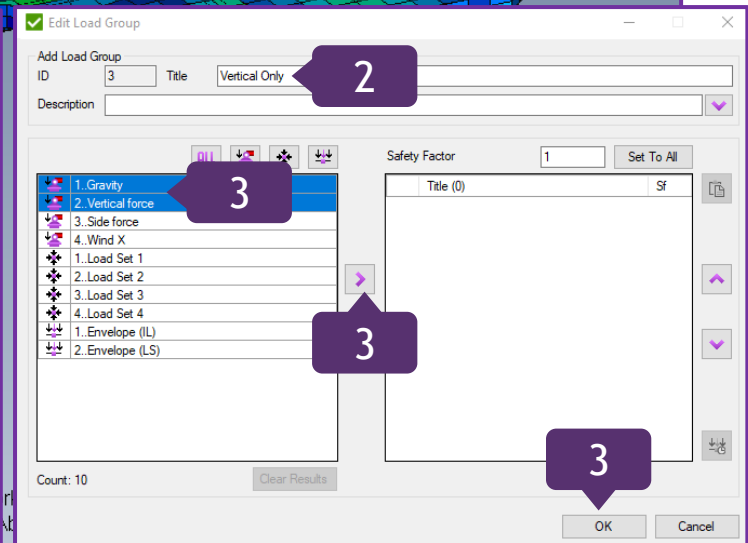
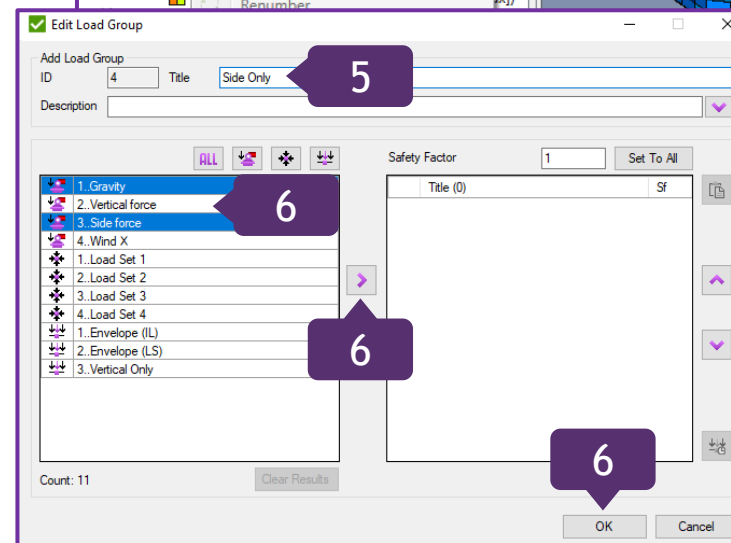
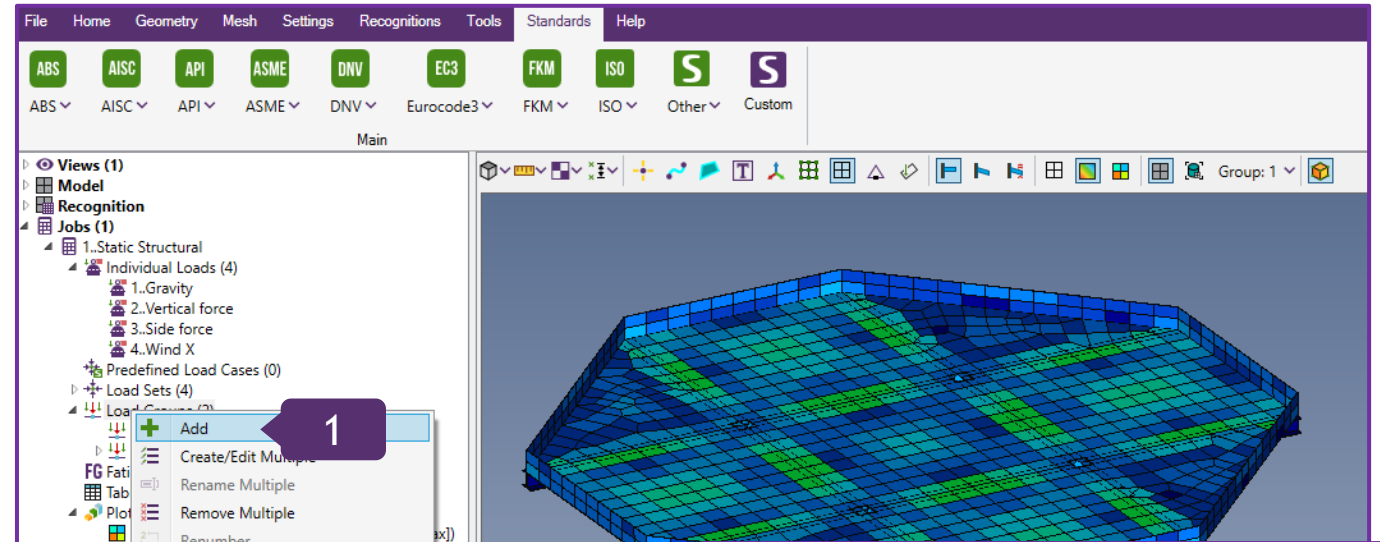
Title: *Side Only*

6

Select *1..Gravity* and *3..Side force*;  
Press   
Press *OK*

Created Load Groups

- Load Groups (4)
  - 1..Envelope (IL)
  - 2..Envelope (LS)
  - 3..Vertical Only
  - 4..Side Only



# Create Fatigue Group

1

Select *Fatigue Groups*;  
Execute right click and select *Add*

2

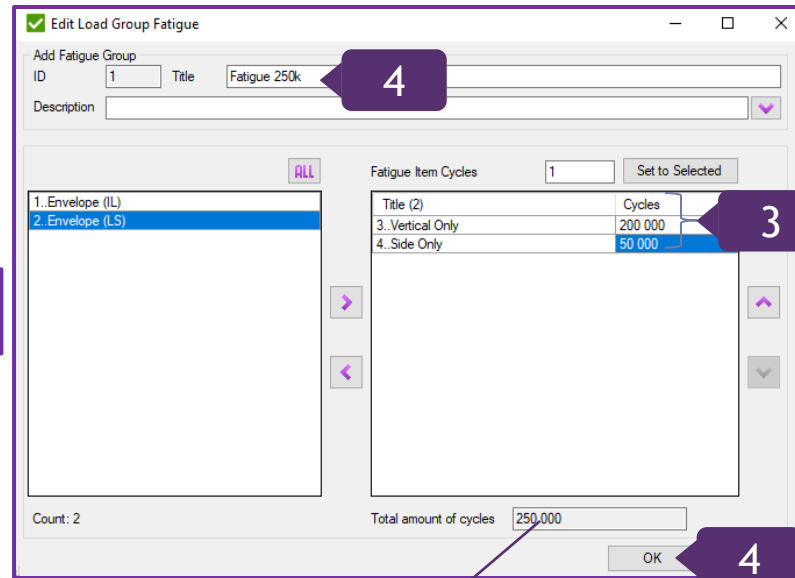
Select *3..Vertical Only* and *4..Side Only*;  
Press 

3

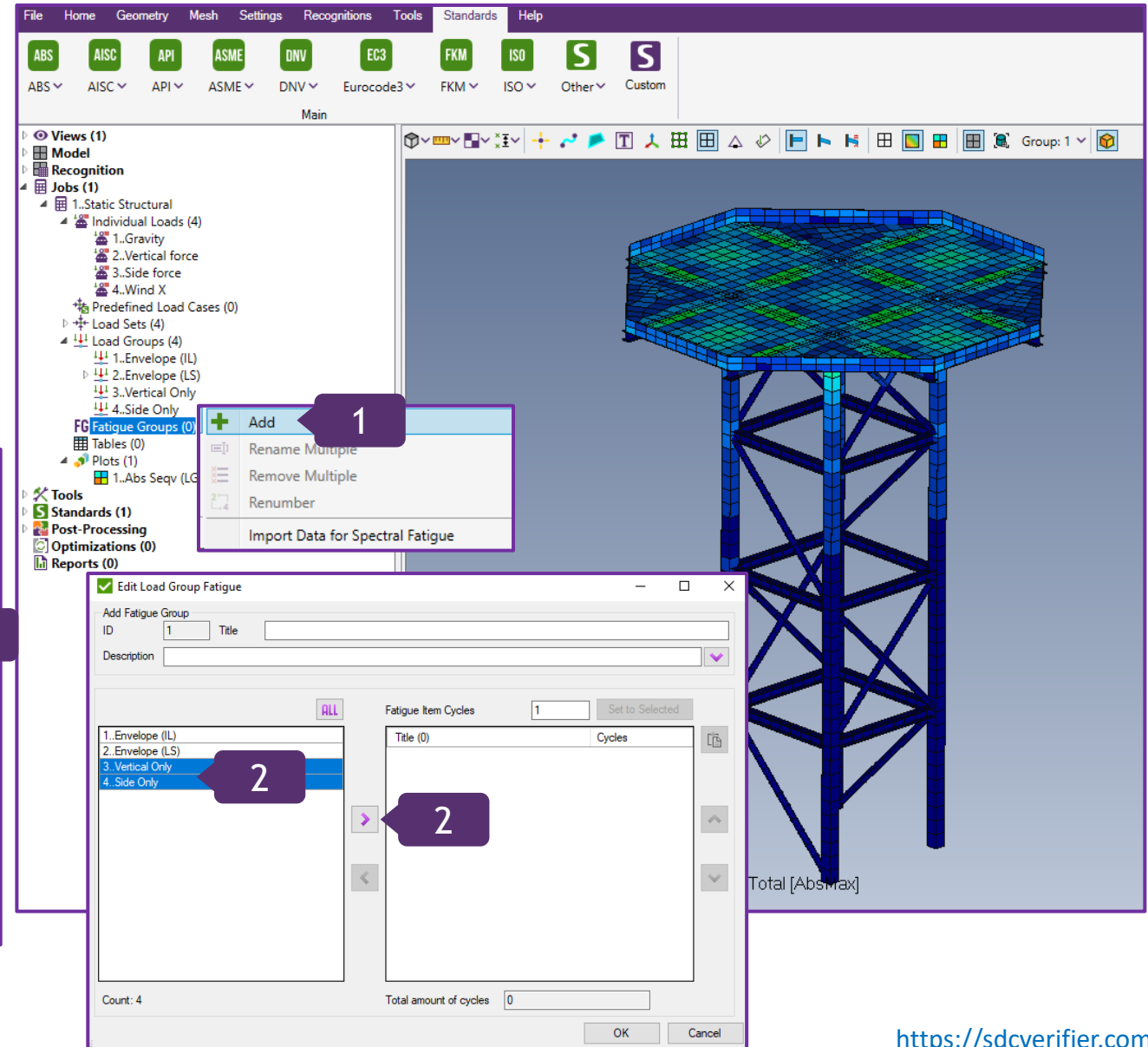
*3..Vertical Only*: 200 000 cycles;  
*4..Side Only*: 50 000 cycles

4

Title: *Fatigue 250k*;  
Press *OK*



Total amount of cycles has been calculated automatically.





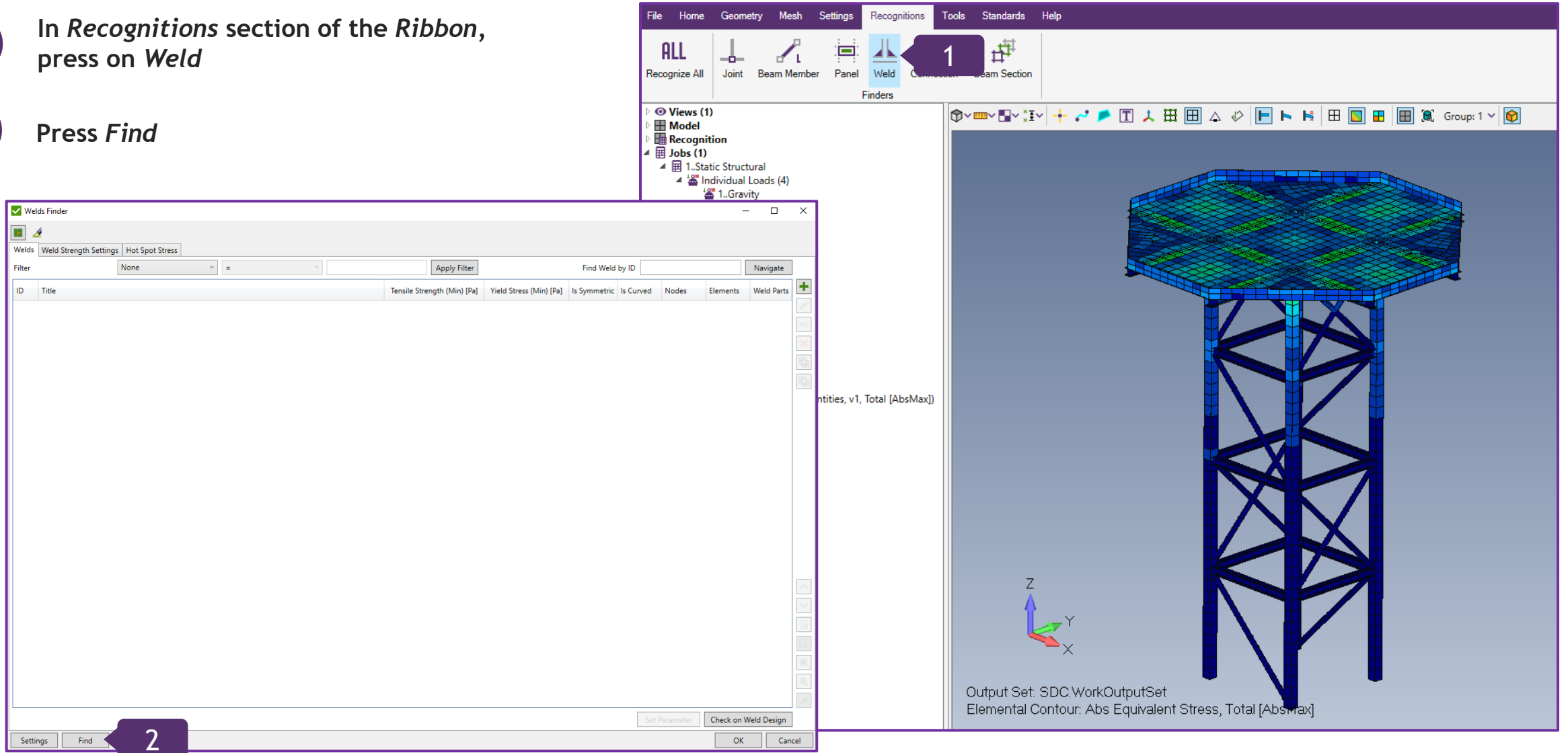
# Recognize Welds

1

In *Recognitions* section of the *Ribbon*, press on *Weld*

2

Press *Find*



# Preview Welds

1

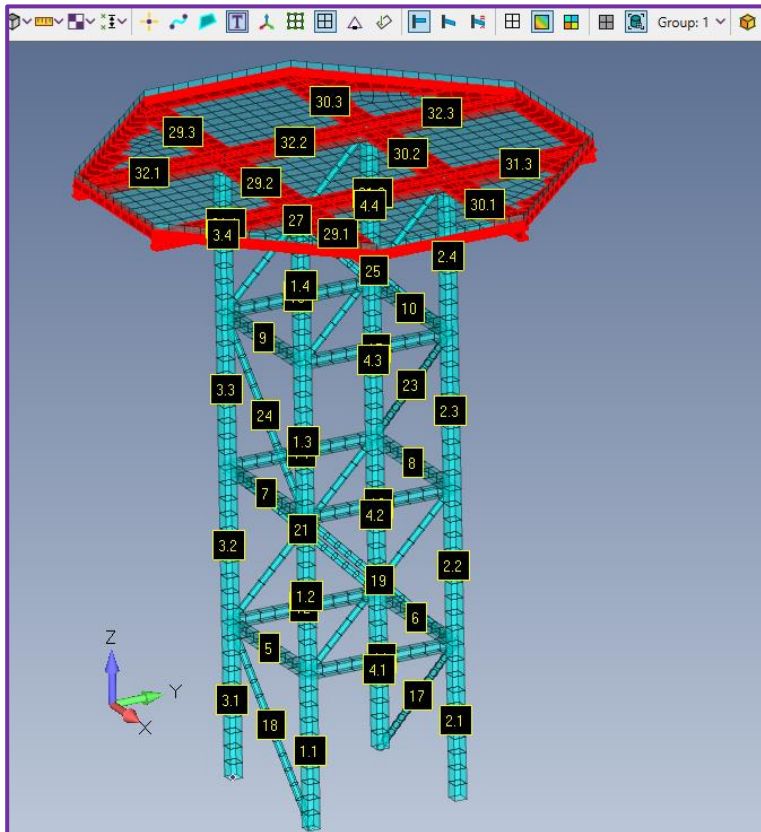
In *Welds* section, select all *Welds*

2

To preview Welds, click on 

3

Press *OK*



Welds Finder

Welds | Weld Strength Settings | Hot Spot Stress

Filter: None = Apply Filter Find Weld by ID: Navigate

ID	Title	Tensile Strength (Min) [Pa]	Yield Stress (Min) [Pa]	Is Symmetric	Is Curved	Nodes	Elements	Weld Parts
1	Weld 1 [-1.5; -1.5; 12]	355000000	240000000	Yes	No	15	31	2
2	Weld 2 [-3; 1.5; 12]	355000000	240000000	Yes	No	11	20	2
3	Weld 3 [-1.5; 4.5; 12]	355000000	240000000	Yes	No	15	31	2
4	Weld 4 [1.5; 6; 12]	355000000	240000000	Yes	No	11	20	2
5	Weld 5 [4.5; 4.5; 12]	355000000	240000000	Yes	No	15	31	2
6	Weld 6 [6; 1.5; 12]	355000000	240000000	Yes	No	11	20	2
7	Weld 7 [1.5; -3; 12]	355000000	240000000	Yes	No	11	20	2
8	Weld 8 [4.5; -1.5; 12]	355000000	240000000	Yes	No	15	31	2
9	Weld 9 [4.5; 0; 12]	355000000	240000000	Yes	No	11	31	3
10	Weld 10 [1.5; 0; 12]	355000000	240000000	Yes	No	11	30	3
11	Weld 11 [-1.5; 0; 12]	355000000	240000000	Yes	No	11	31	3
12	Weld 12 [4.5; 3; 12]	355000000	240000000	Yes	No	11	31	3
13	Weld 13 [1.5; 3; 12]	355000000	240000000	Yes	No	11	30	3
14	Weld 14 [-1.5; 3; 12]	355000000	240000000	Yes	No	11	31	3
15	Weld 15 [3; -1.5; 12]	355000000	240000000	Yes	No	11	30	3
16	Weld 16 [3; 1.5; 12]	355000000	240000000	Yes	No	11	30	3
17	Weld 17 [3; 4.5; 12]	355000000	240000000	Yes	No	11	31	3
18	Weld 18 [0; -1.5; 12]	355000000	240000000	Yes	No	11	30	3
19	Weld 19 [0; 1.5; 12]	355000000	240000000	Yes	No	11	30	3
20	Weld 20 [0; 4.5; 12]	355000000	240000000	Yes	No	11	31	3

Settings Find Set Parameter Check on Weld Design OK Cancel

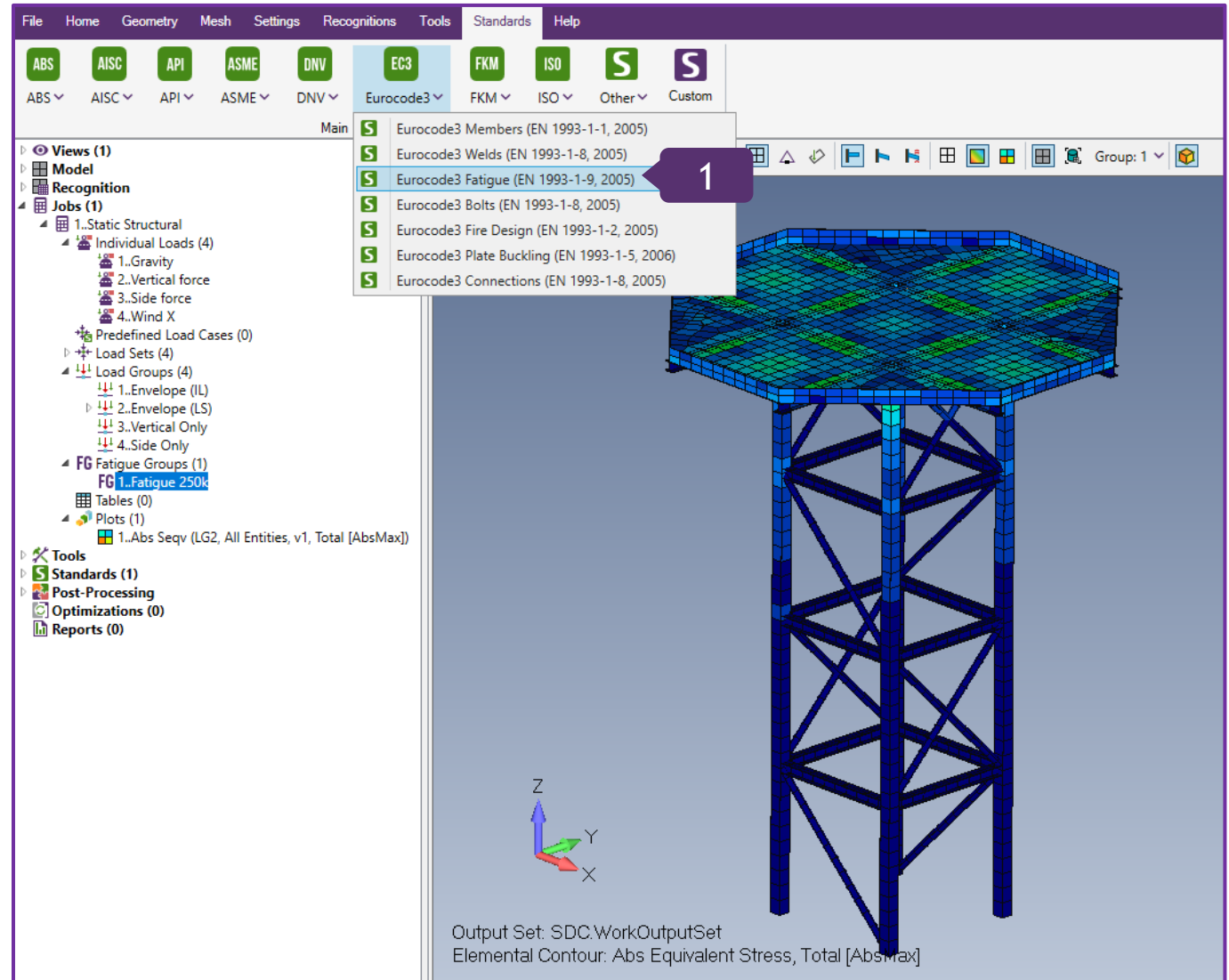
All Property, Material and Thickness changes have been recognized, along with the connection of a certain angle as welded elements.



# Add Eurocode3 Fatigue (EN 1993-1-9, 2005) Standard

1

In *Standards* section of the *Ribbon*, press on *Eurocode3* and select *Eurocode3 Fatigue (EN 1993-1-9, 2005)*



# Define FAT Class

1

In *FAT Class*, press 

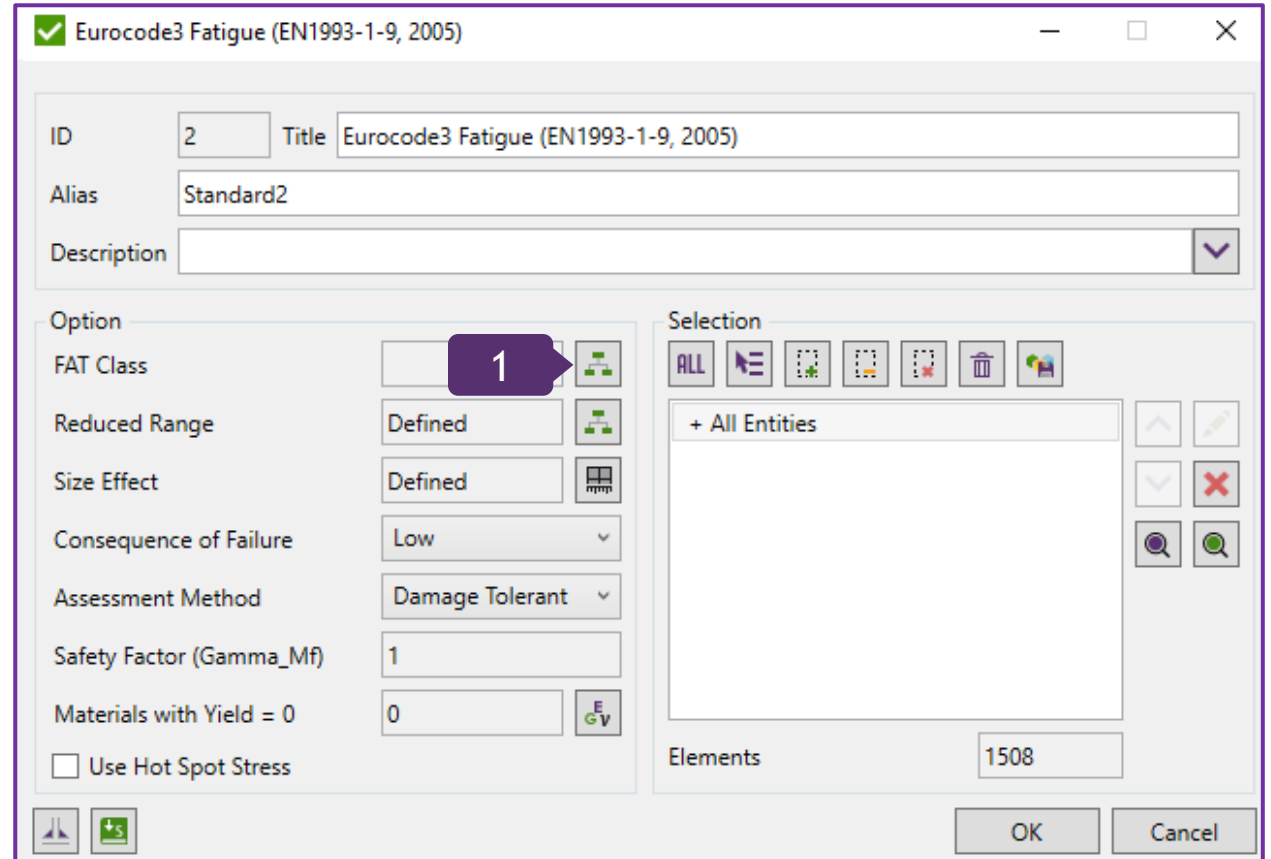
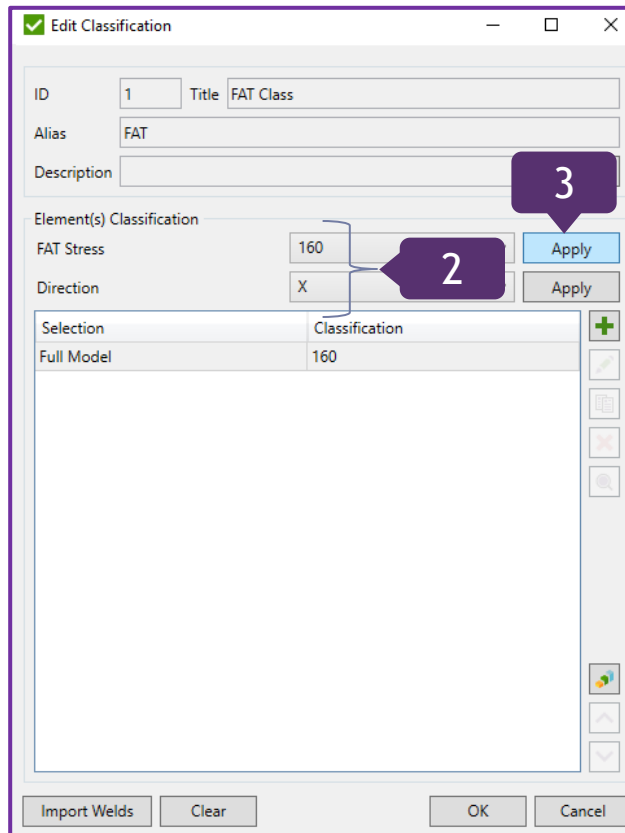
2

FAT Stress: 160;  
Direction: X

3

Press *Apply*

FAT Class is the fatigue strength for nominal stress ranges. It is represented by a series of  $(\log \Delta \sigma_R) - (\log N)$  curves and  $(\log \Delta \tau_R) - (\log N)$  curves (S-N-curves), which correspond to typical detail categories. Each detail category is designated by a number, which represents in N/mm<sup>2</sup> the reference value  $\Delta \sigma_c$  and  $\Delta \tau_c$  for the fatigue strength at 2 million cycles.



# Define Welds for X Direction

1

Press  to define new condition

2

Press *Add all Welds*

3

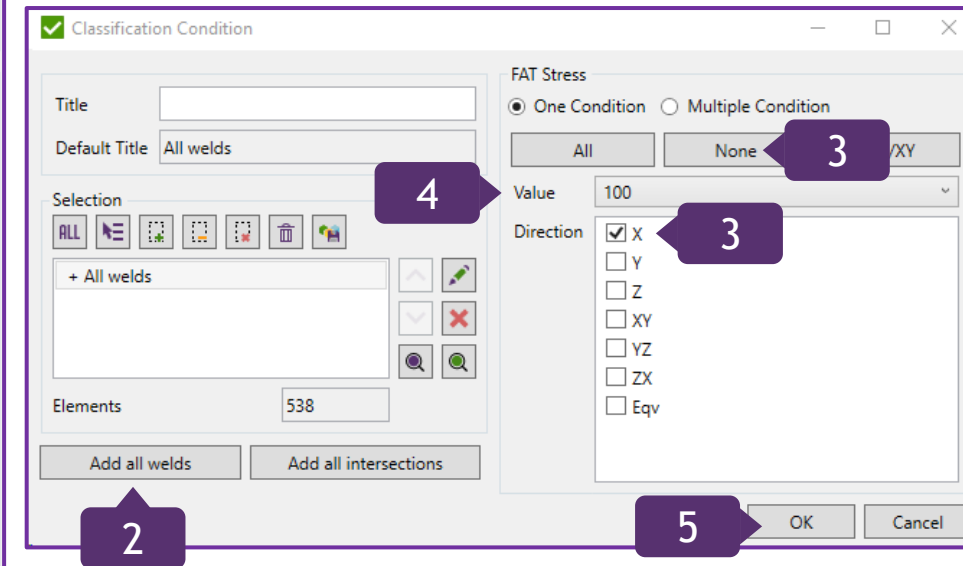
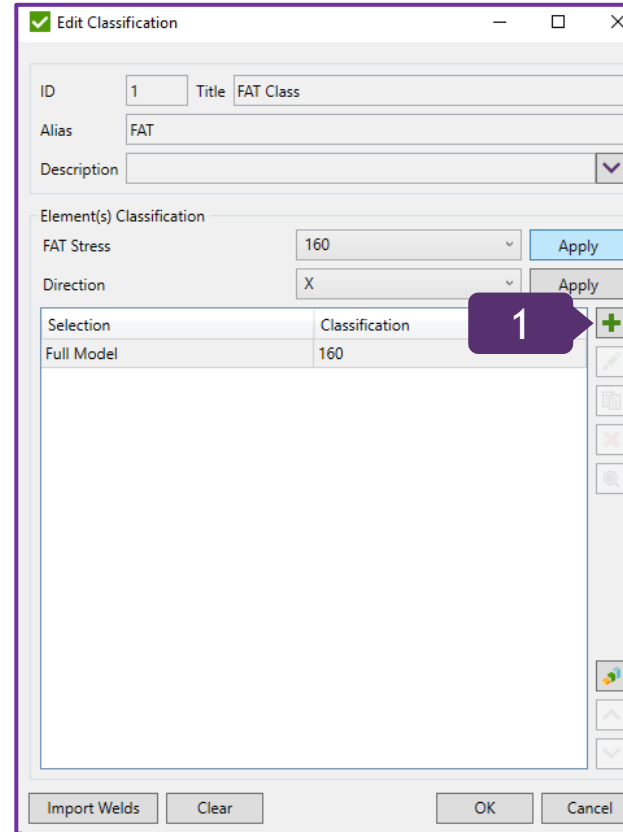
Press *None*, and then select *Direction: X*

4

*Value: 100*

5

Press *OK*



# Define Welds for Y Direction

1

Press  to define new condition

2

Press *Add all Welds*

3

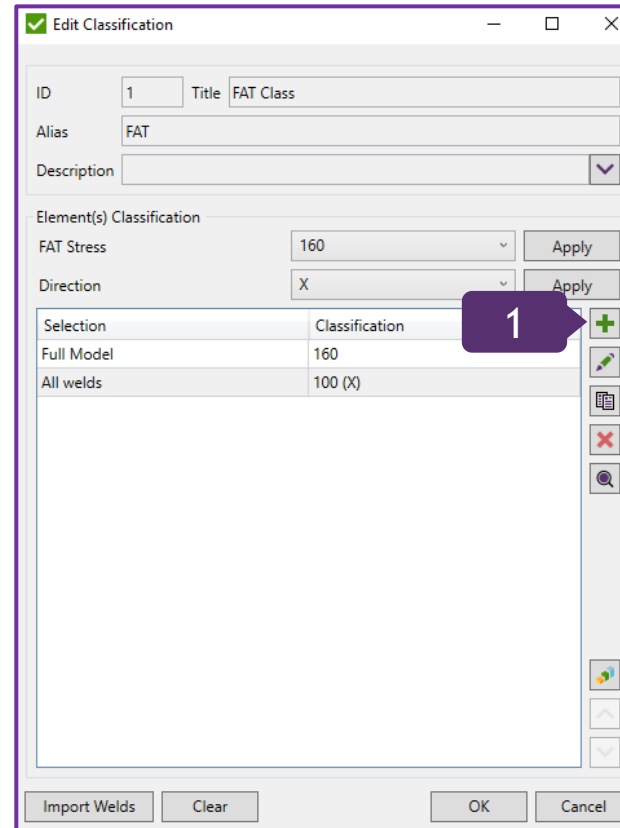
Press *None*, and then select *Direction: Y*

4

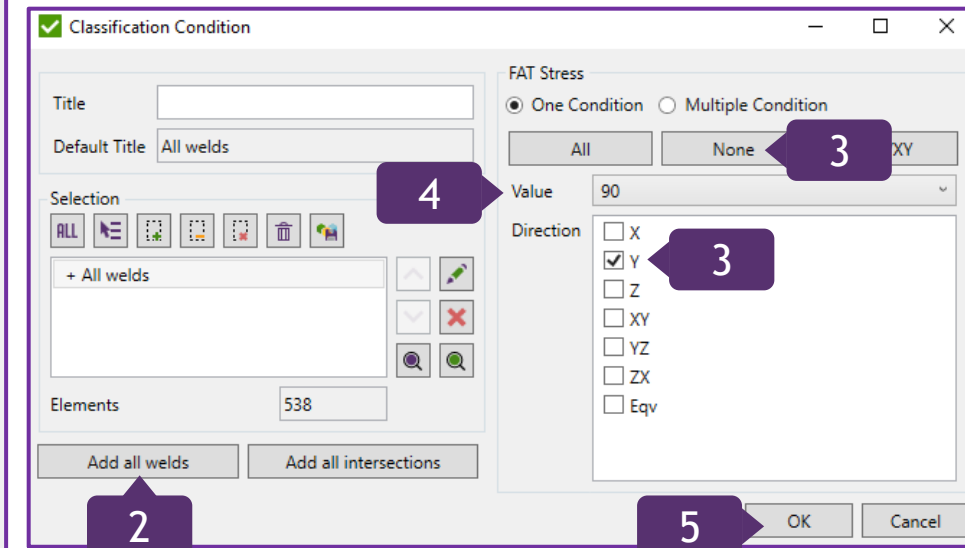
Value: 90

5

Press *OK*



The 'Edit Classification' dialog box is shown. It has fields for ID (1), Title (FAT Class), Alias (FAT), and Description. Below these are 'Element(s) Classification' settings: FAT Stress (160) and Direction (X), each with an 'Apply' button. A table lists 'Selection' (Full Model, All welds) and 'Classification' (160, 100 (X)). A green plus icon is highlighted with a purple circle and arrow labeled '1'. At the bottom are 'Import Welds', 'Clear', 'OK', and 'Cancel' buttons.



The 'Classification Condition' dialog box is shown. It has a 'Title' field and a 'Default Title' (All welds). Below are 'Selection' icons and a list box containing '+ All welds'. A purple circle and arrow labeled '4' points to the selection area. Below the list box is an 'Elements' field showing '538'. At the bottom are 'Add all welds' (labeled with a purple circle and arrow '2') and 'Add all intersections' buttons. On the right, 'FAT Stress' settings are shown: 'One Condition' is selected, 'All' and 'None' buttons (with 'None' highlighted by a purple circle and arrow '3'), and a 'Value' field set to '90'. Below this is a 'Direction' section with checkboxes for X, Y (checked, highlighted by a purple circle and arrow '3'), Z, XY, YZ, ZX, and Eqv. At the bottom right are 'OK' (labeled with a purple circle and arrow '5') and 'Cancel' buttons.

# Define Welds for XY Shear Direction

1

Press  to define new condition

2

Press *Add all Welds*

3

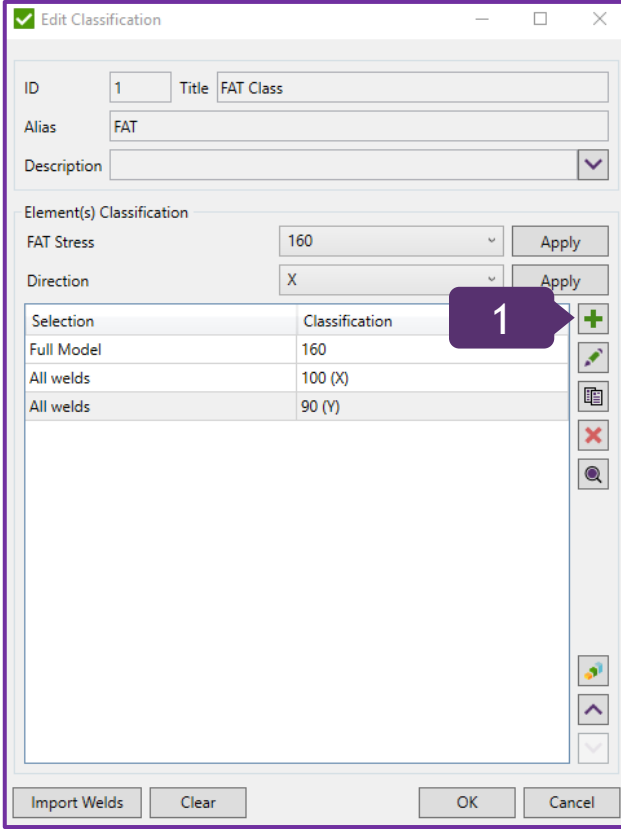
Press *None*, and then select *Direction: XY*

4

Value: 71

5

Press *OK*



**Edit Classification**

ID: 1 Title: FAT Class

Alias: FAT

Description:

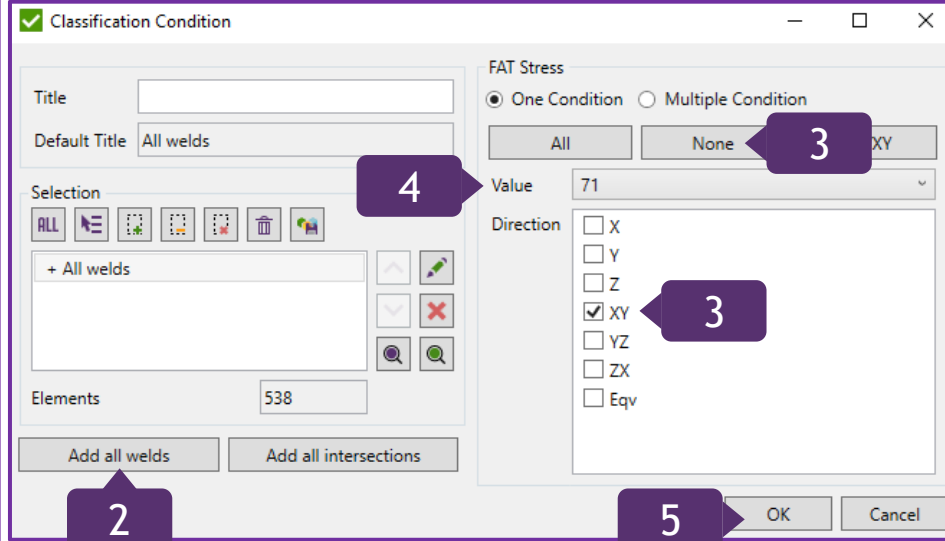
Element(s) Classification

FAT Stress: 160 Apply

Direction: X Apply

Selection	Classification
Full Model	160
All welds	100 (X)
All welds	90 (Y)

Buttons: Import Welds, Clear, OK, Cancel



**Classification Condition**

Title:

Default Title: All welds

Selection: ALL, [Icons]

Elements: 538

Buttons: Add all welds, Add all intersections

FAT Stress: ☒ One Condition ☐ Multiple Condition

Direction: All, None, XY

Value: 71

Direction: ☐ X, ☐ Y, ☐ Z, ☒ XY, ☐ YZ, ☐ ZX, ☐ Eqv

Buttons: OK, Cancel

# Define All Welds Intersections

1 Press  to define new condition

2 Press *Add all intersections*

3 Direction: *All*

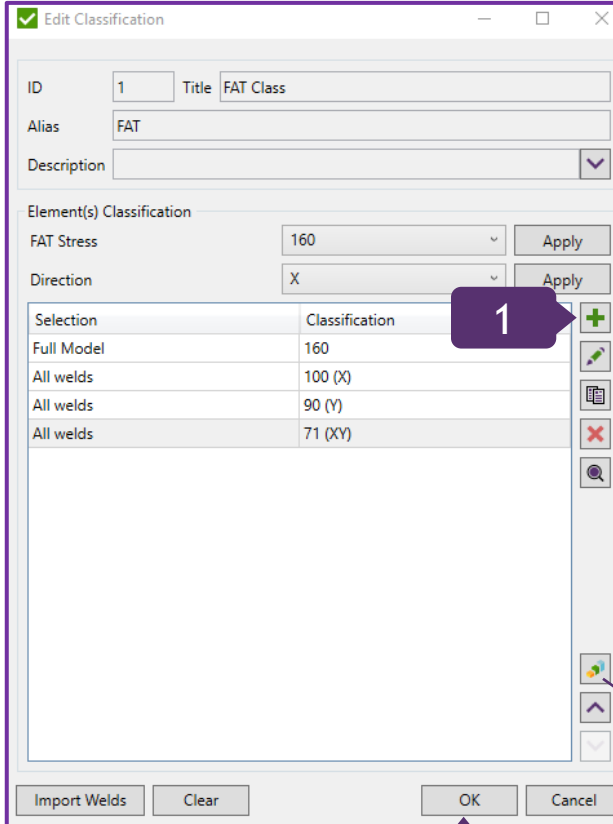
4 Value: *71*

5 Press *OK*

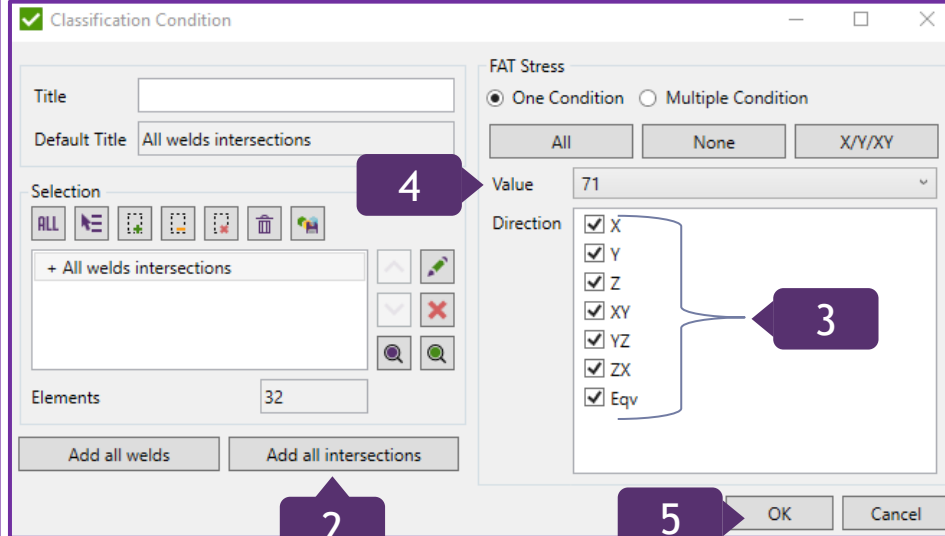
Classification has been defined.

Note: Every successive rule overwrites the previous one.


Selection	Classification
Full Model	160
All welds	100 (X)
All welds	90 (Y)
All welds	71 (XY)
All welds intersections	71 (All Directions)



The 'Edit Classification' dialog box shows the configuration for a new classification. It includes fields for ID (1), Title (FAT Class), Alias (FAT), and Description. Below these are 'Element(s) Classification' settings for FAT Stress (160) and Direction (X). A table lists the classification rules: Full Model (160), All welds (100 (X)), All welds (90 (Y)), and All welds (71 (XY)). A green arrow labeled '1' points to the '+' button in the table's right margin. At the bottom, there are 'Import Welds', 'Clear', 'OK', and 'Cancel' buttons. A green arrow labeled '5' points to the 'OK' button.



The 'Classification Condition' dialog box shows the configuration for a new condition. It includes fields for Title and Default Title (All welds intersections). Below these are 'Selection' icons, a list of elements (+ All welds intersections), and 'Elements' (32). The 'FAT Stress' section has radio buttons for 'One Condition' (selected) and 'Multiple Condition', and buttons for 'All', 'None', and 'X/Y/XY'. The 'Value' is set to 71. The 'Direction' section has checkboxes for X, Y, Z, XY, YZ, ZX, and Eqv, all of which are checked. A green arrow labeled '3' points to the 'Eqv' checkbox. At the bottom, there are 'Add all welds', 'Add all intersections', 'OK', and 'Cancel' buttons. A green arrow labeled '2' points to the 'Add all intersections' button, and a green arrow labeled '4' points to the 'Selection' icons.

Full Model and All Welds can be previewed and plotted by pressing 

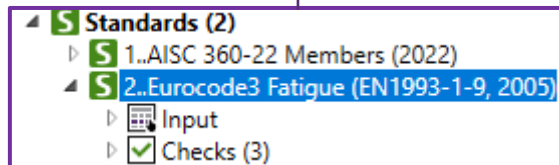
# Define Consequence of Failure and Assessment Method

1 Consequence of Failure: *High*

2 Assessment Method: *Safe Life*

3 Press *OK*

The Standard has been added.



Note: The value of Safety Factor (Gamma\_Mf) shifts in accordance to the configurations of Consequence of Failure and Assessment Method.

A screenshot of a configuration dialog box titled 'Eurocode3 Fatigue (EN1993-1-9, 2005)'. The dialog has several fields and options. The 'ID' field is set to '2' and the 'Title' field is set to 'Eurocode3 Fatigue (EN1993-1-9, 2005)'. The 'Alias' field is set to 'Standard2'. The 'Description' field is empty. The 'Option' section contains several settings: 'FAT Class' is 'Defined', 'Reduced Range' is 'Defined', 'Size Effect' is 'Defined', 'Consequence of Failure' is 'High' (with a callout '1'), 'Assessment Method' is 'Safe Life' (with a callout '2'), 'Safety Factor (Gamma\_Mf)' is '1.35', and 'Materials with Yield = 0' is '0'. There is a checkbox for 'Use Hot Spot Stress' which is unchecked. The 'Selection' section has a list of icons and a search bar. The 'Elements' field shows '1508'. At the bottom right, there are 'OK' and 'Cancel' buttons. A callout '3' points to the 'OK' button.

# Criteria Plot for Fatigue Check

1 Expand *Checks*, and execute right click on *2..Fatigue Check*

2 Select *Criteria Plot*

3 Press  to select *Fatigue Group*

4 Select *1.. Fatigue 250k* and press *OK*

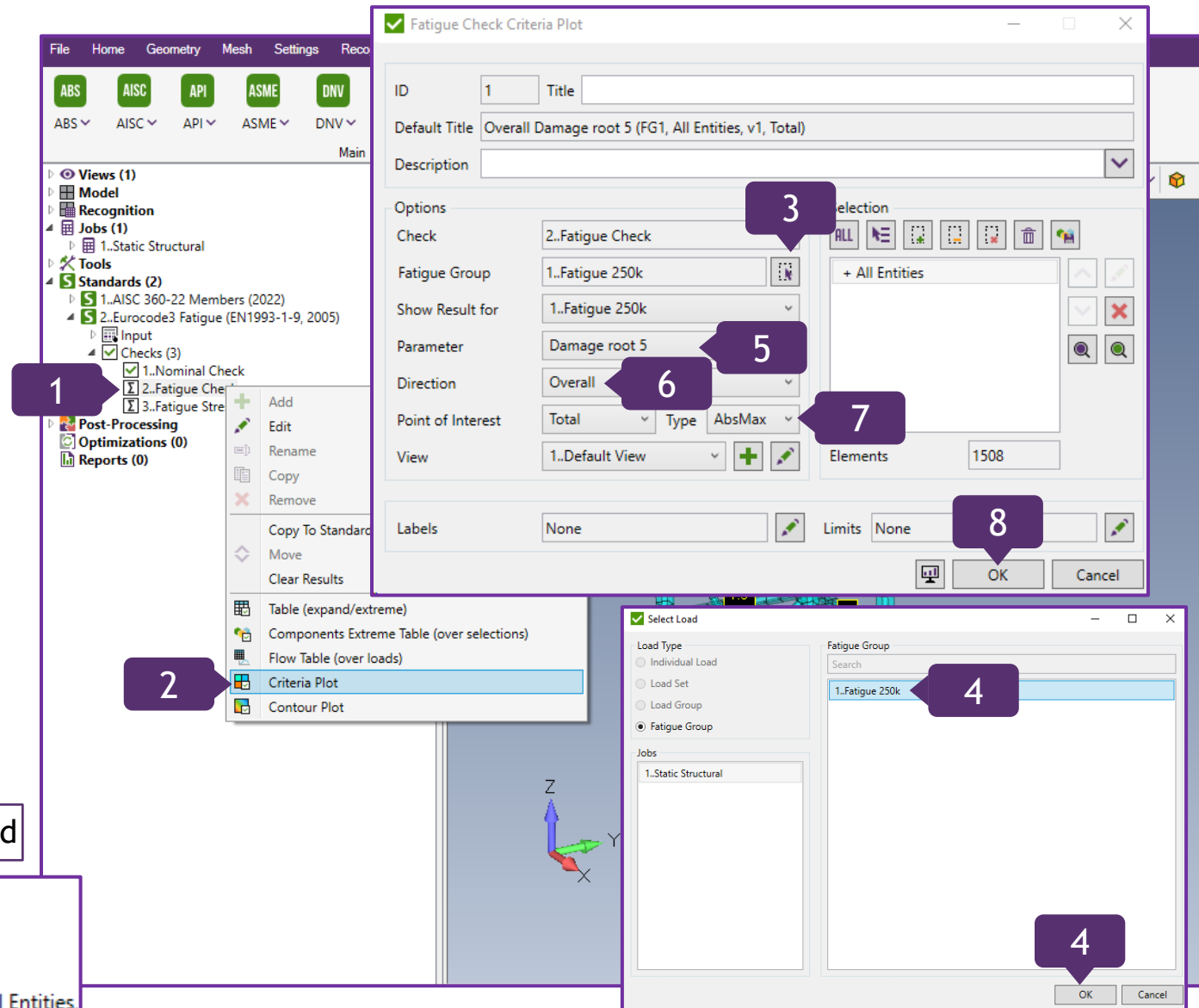
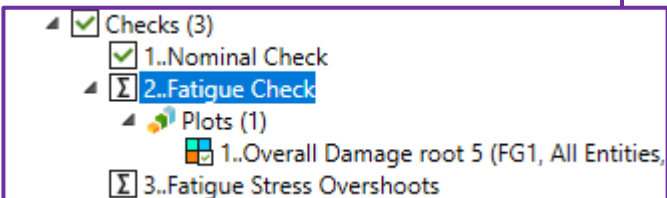
5 Parameter: *Damage root 5*

6 Direction: *Overall*

7 Point of Interest: *Total*;  
Type: *AbsMax*

8 Press *OK*

The Criteria Plot has been created

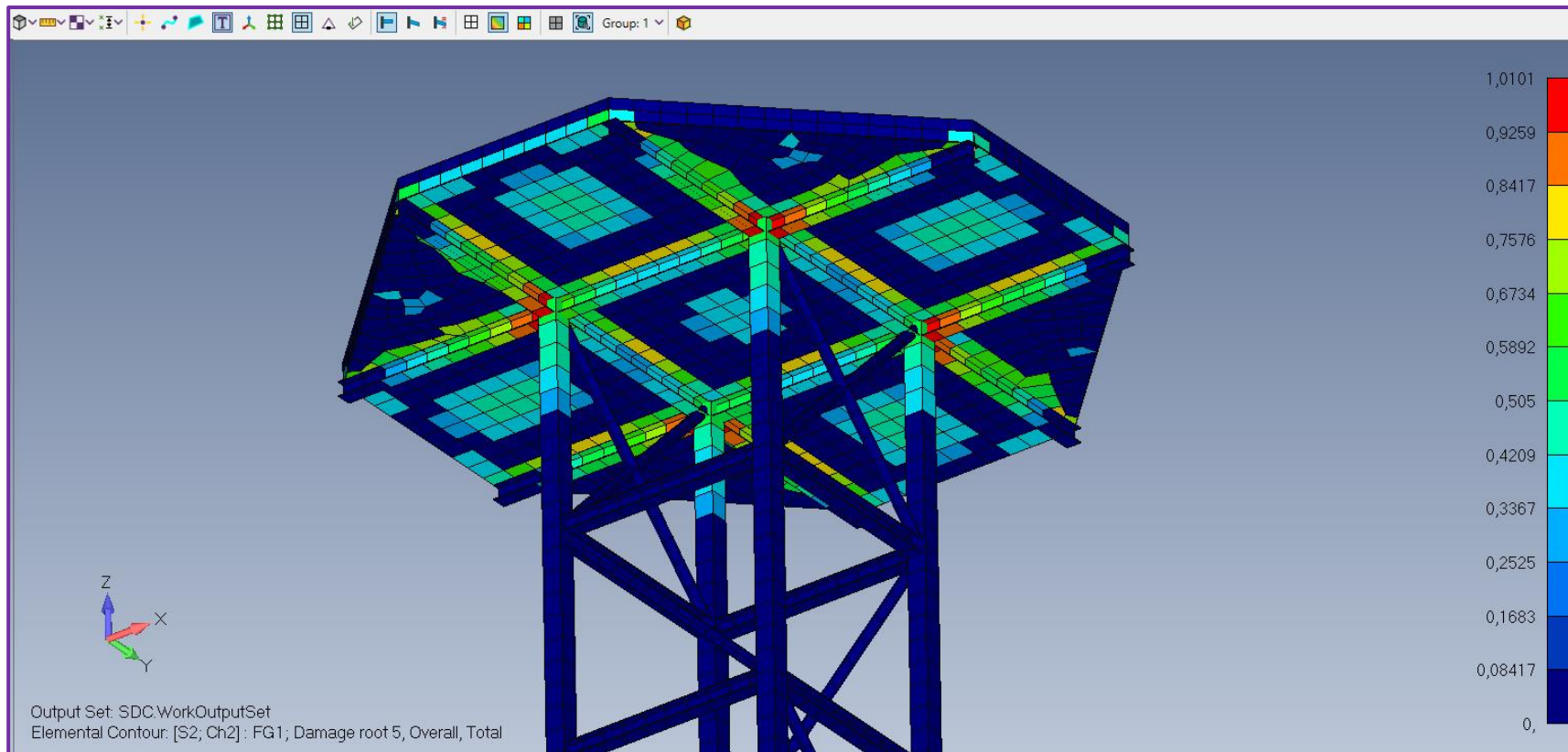




# Preview the Plot

1 Expand *Plots*, and execute right click on *1..Overall Damage root 5 (FG1, All Entities, v1, Total)*

2 Select *Preview*



Views (1)  
Model  
Recognition  
Jobs (1)  
1..Static Structural  
Tools  
Standards (2)  
1..AISC 360-22 Members (2022)  
2..Eurocode3 Fatigue (EN1993-1-9, 2005)  
Input  
Checks (3)  
1..Nominal Check  
2..Fatigue Check  
Plots (1)  
1..Overall Damage root 5 (FG1, All Entities, v1, Total)  
3..Fatigue Stress Overshoots

Post-Processing  
Optimizations (0)  
Reports (0)

Edit  
Rename  
Copy  
Remove  
Preview  
Preview with current view  
Preview with current View And Selection

As it might be seen on the plot, the structure is above the safe value for fatigue damage in its specific parts. Therefore, the modifications are required, and they can be carried out with incorporation of Optimizations Tool or manually.

1

In *Model* section, expand *Properties* and select *2..I-beam200*

2

Execute right click and select *Edit*

3

Press on *Shape*

4

Make the required modifications in *Cross Section Definition* window

5

Close the window

There is a separate tutorial “Modelling with SDC Verifier”, in which the creation and modification of Properties are described in details (slides 24-25; 29-31)

