



Tutorial

Eurocode3 Members (EN1993-1-1, 2005) Optimization. Beam Rule

Updated on: April 19th, 2024

Tested with: SDC Verifier for Simcenter 3D
2024 R1.1

Simcenter3D 2306

- This step-by-step tutorial demonstrates the functionality of SDC Verifier Optimization Tool, incorporating Beam Rule;
- Model members are Optimized, based on Criteria Plot results;
- Shape Library Overview;
- Optimization Rules Overview;
- Optimization results in Tables and Plots;
- Results Comparison;
- Automatic Beam Cross-section Change
- The model change by adding Beam Properties
- Complete information on Optimization Tool may be found on our website via this link: [Optimization Tool | Help | SDC for Simcenter 3D \(sdcverifier.com\)](https://sdcverifier.com)

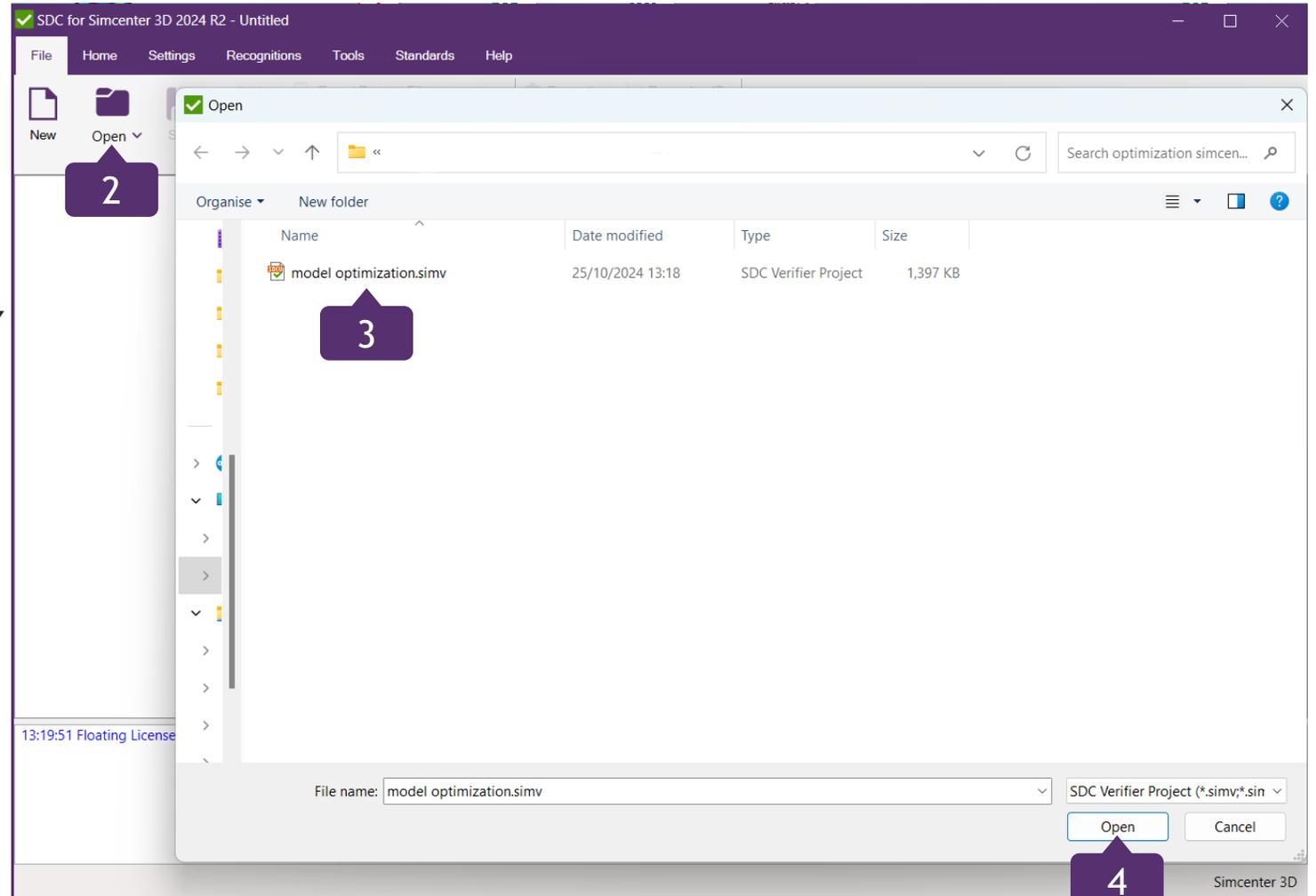
Open the Starter Model

1 Launch SDC Verifier for Simcenter 3D

2 In *File* section, press *Open*

3 Select a project
Optimization1_Beam_Rule_NShapes.simv

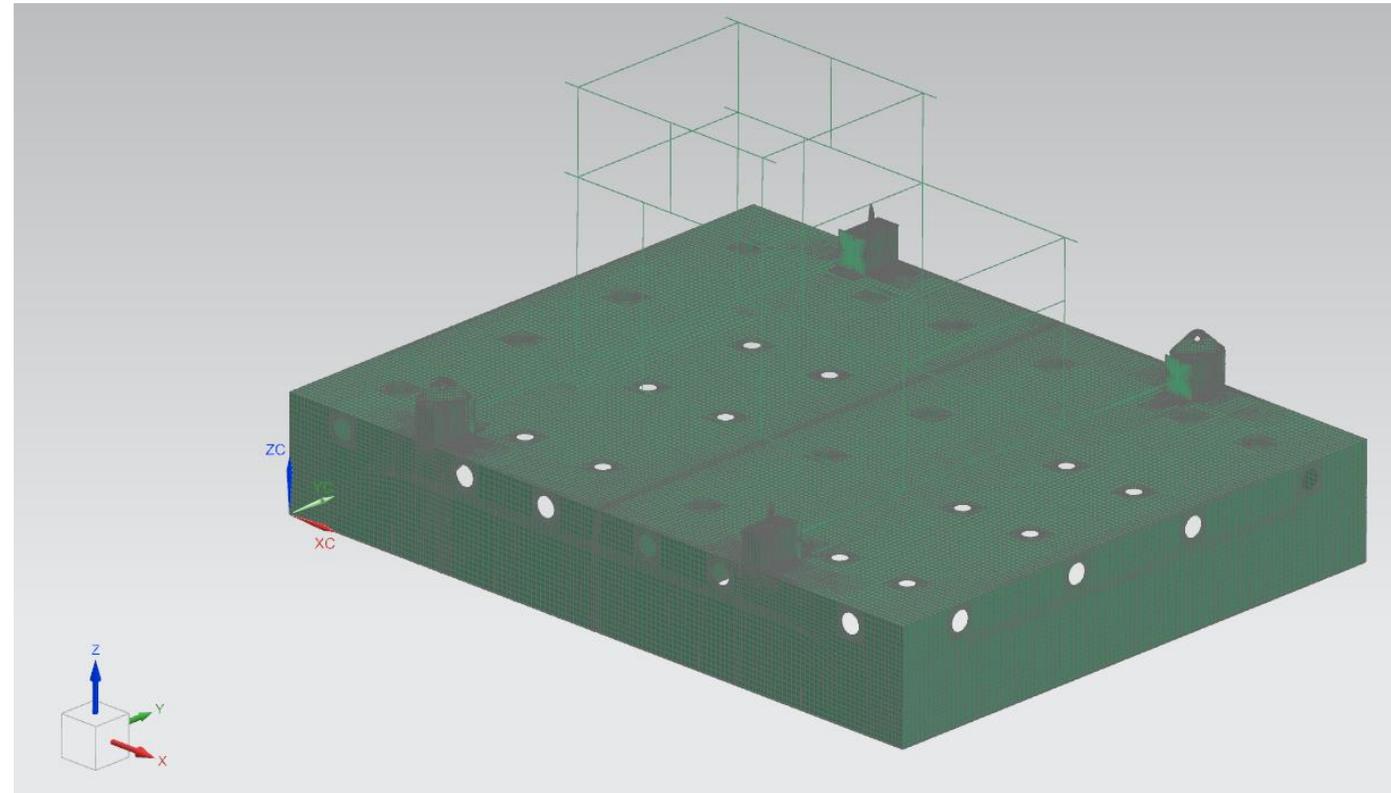
4 Press *Open*



This tutorial uses a Project with predefined Individual Loads, Load Sets and Load Groups.

The model contains Plate, Beam elements and Welds. Also, a relevant Standard has been previously added.

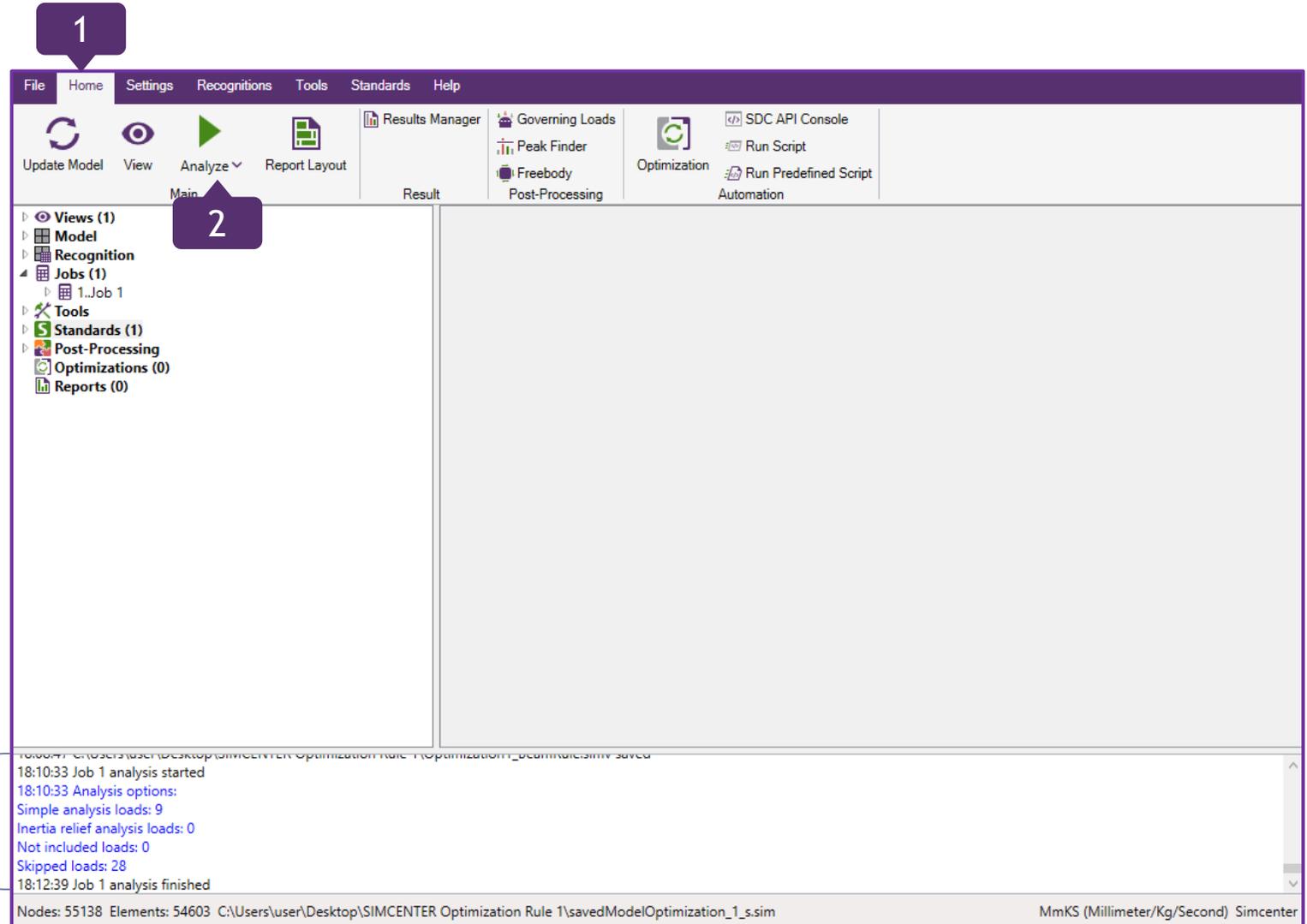
- ▲ Views (1)
 - 1..Default View
- ▶ Model
- ▶ Recognition
- ▲ Jobs (1)
 - ▲ 1..Job 1
 - ▶ Individual Loads (9)
 - ▶ Predefined Load Cases (0)
 - ▶ Load Sets (28)
 - ▶ Load Groups (5)
 - FG Fatigue Groups (0)
 - Tables (0)
 - Plots (0)
- ▶ Tools
- ▲ Standards (1)
 - ▲ 1..Eurocode3 Members (EN1993-1-1, 2005)
 - ▶ Input
 - ▶ Checks (22)
- ▶ Post-Processing
- ▶ Optimizations (0)
- ▶ Reports (0)



A separate Tutorial with detailed instructions on how to add, define and edit the Standard can be found via this link:
<https://sdcverifier.com/tutorials/aisc-360-10/>

1 Go to *Home* section on the Ribbon

2 Press  on the toolbar to analyze Job



Job 1 analysis started and finished.

```
18:10:33 Job 1 analysis started
18:10:33 Analysis options:
Simple analysis loads: 9
Inertia relief analysis loads: 0
Not included loads: 0
Skipped loads: 28
18:12:39 Job 1 analysis finished
```

Nodes: 55138 Elements: 54603 C:\Users\user\Desktop\SIMCENTER Optimization Rule 1\savedModelOptimization_1_s.sim

MmKS (Millimeter/Kg/Second) Simcenter

Eurocode3 Members Criteria Plot

1

Expand Standards => 3..Eurocode3 Members (EN1993-1-1, 2005) => Checks (22) and select 22. *Buckling and Overall*

2

Execute right click on 22. *Buckling and Overall* and select *Criteria Plot*

3

Press  to select Load Group

4

In Load Type, select Load Group, and then 5..*Operations All*; Press *OK*

5

Parameter: *Uf Overall*

6

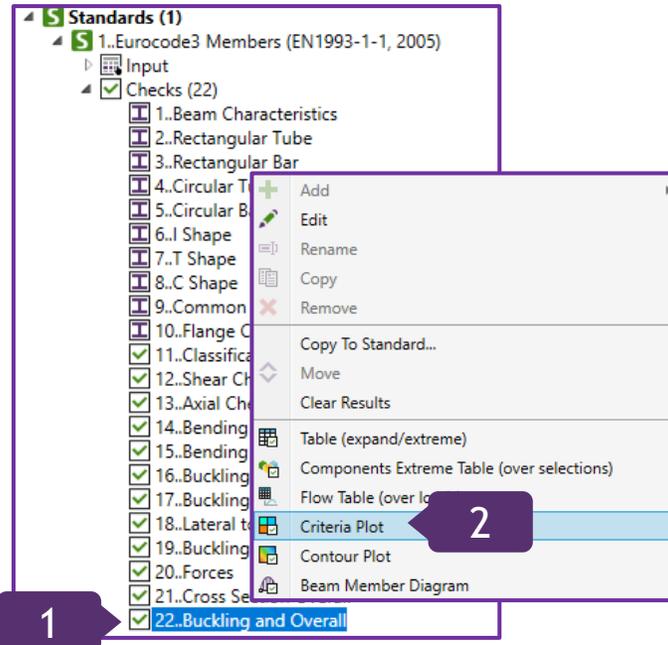
LG Parameter: *Absolute*

7

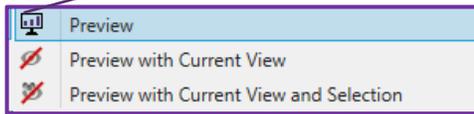
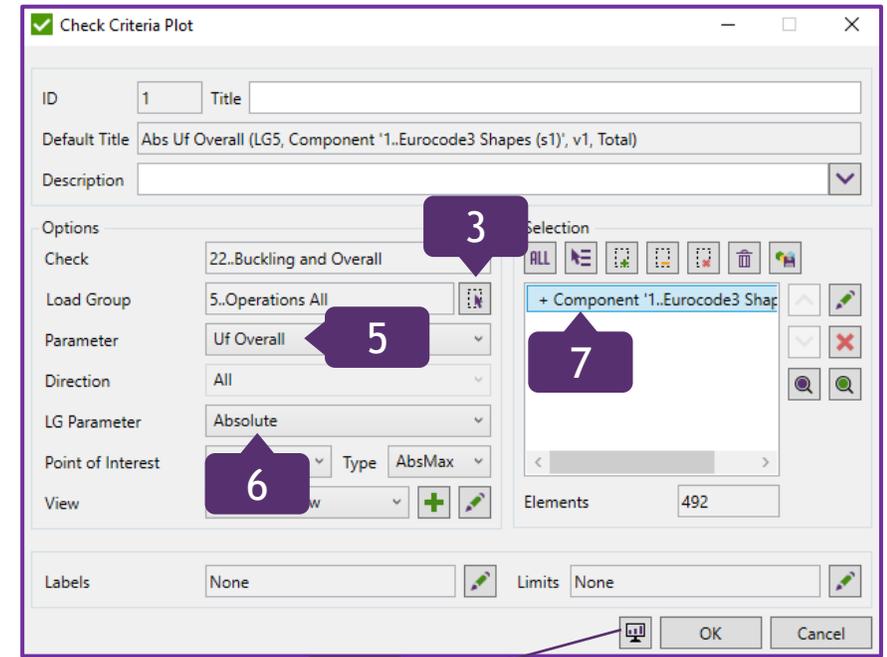
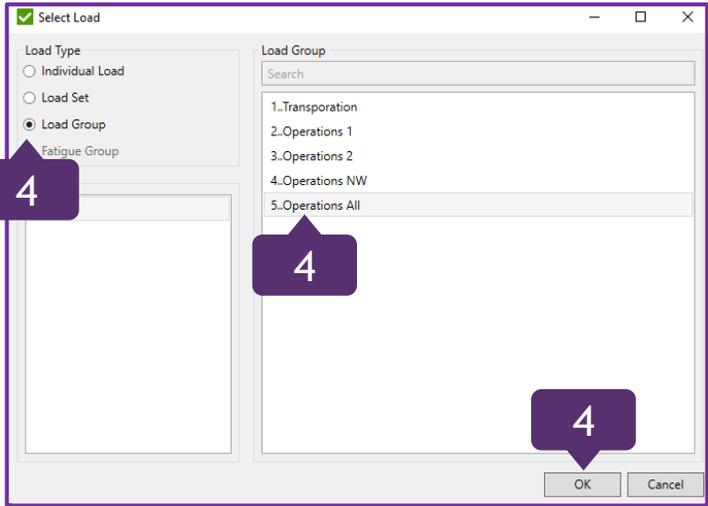
Selection: + *Component '1..Eurocode3 Shapes (s1)'*

8

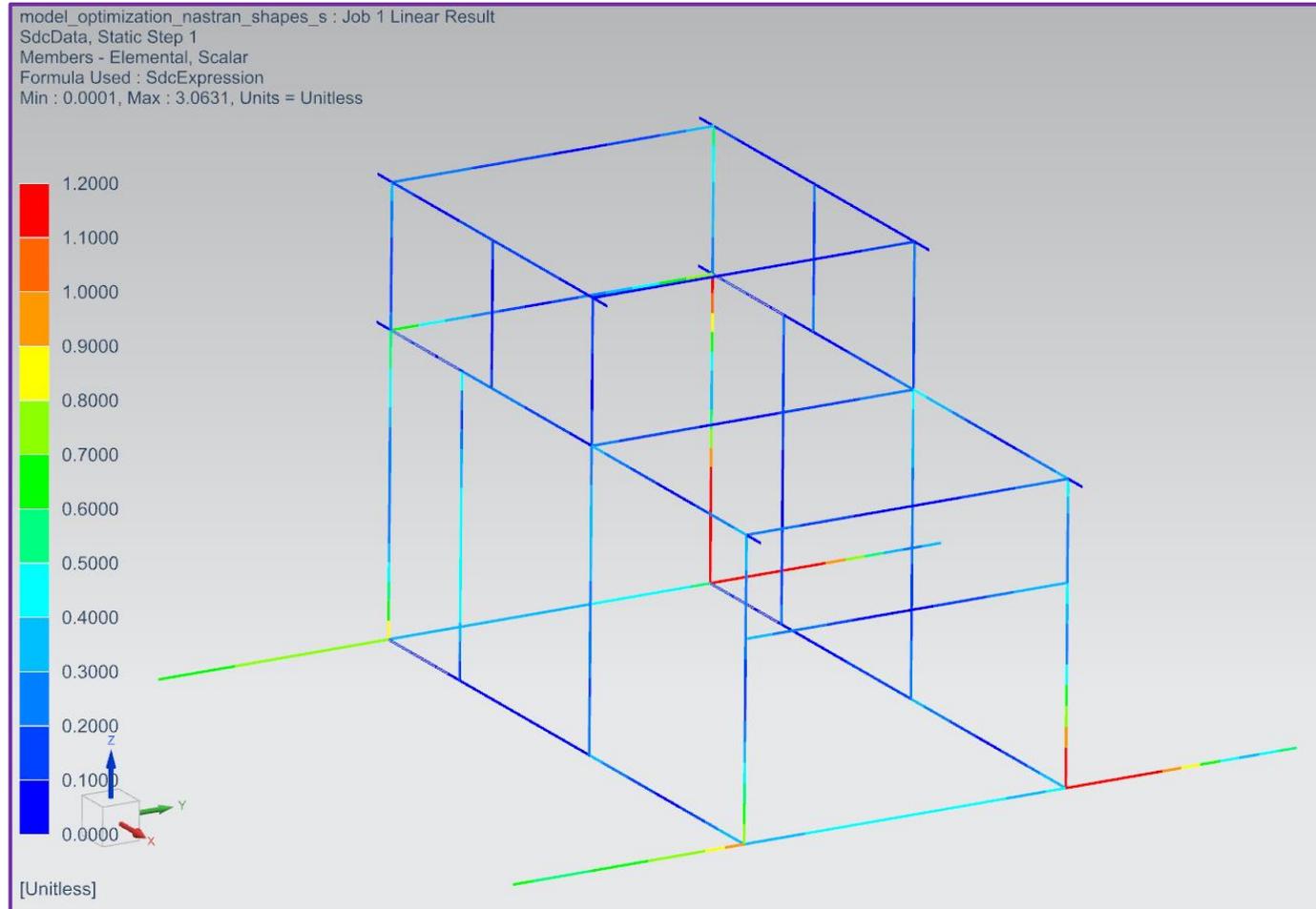
Press  and then *Preview*



The purpose of creating Criteria Plot is to preview the results of Eurocode3 Members Check and pick the members for Optimization.



The Plot has been created to see the UF Overall on Beam members. It is displayed in Simcenter 3D window.



Set Limits for Criteria Plot (Additional Functionality)

1

In Limits, press

2

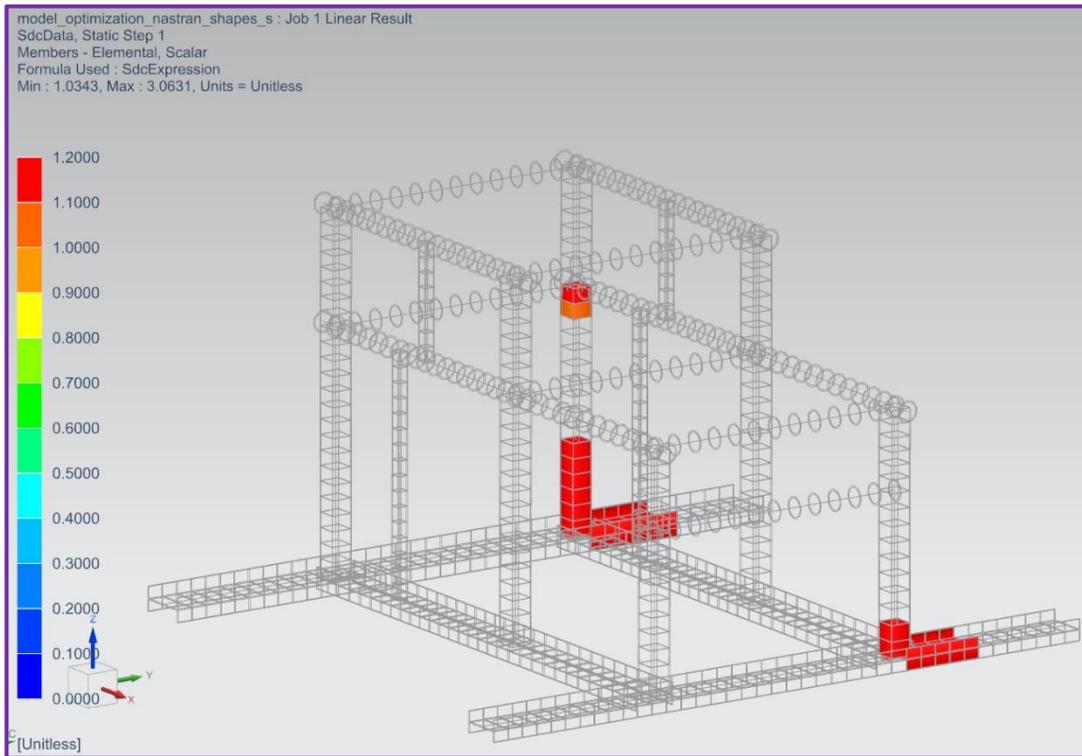
Type: *Above Maximum*;
Maximum: *1*;
Press *OK*

3

Press and then *Preview*

In order to single out the segments with high Utilization Factor, excluding the rest of the elements, Limits function serves for that. The settings of Check Criteria Plot from Slide 6 should remain the same.

The Result



- 3 Preview
- Preview with Current View
- Preview with Current View and Selection

Eurocode3 Members Criteria Plot for One Property

1

Select + Component '1..Eurocode3 Shapes (s1)' and press to remove it

2

Press to add Condition;
Select *Properties*

3

Select *17..Beam Square tube d=150 mm_Steel*;
Press *OK*

4

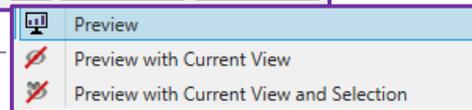
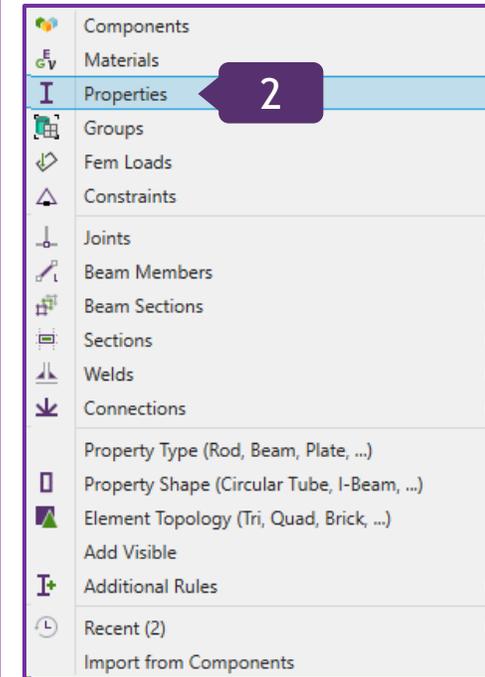
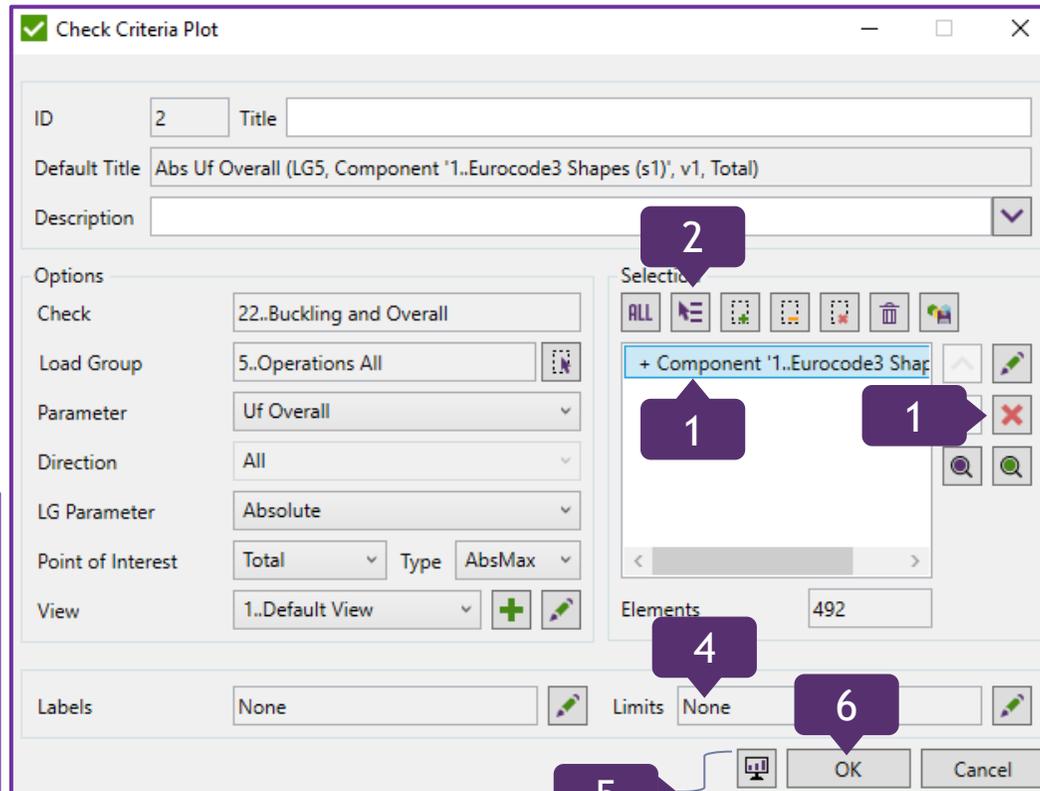
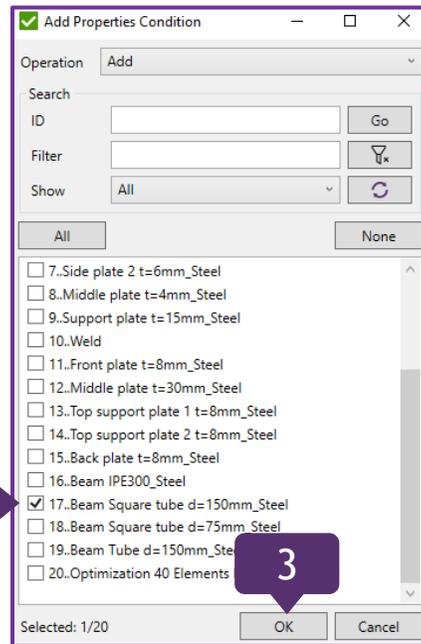
Limits: *None*

5

Press and then
Preview

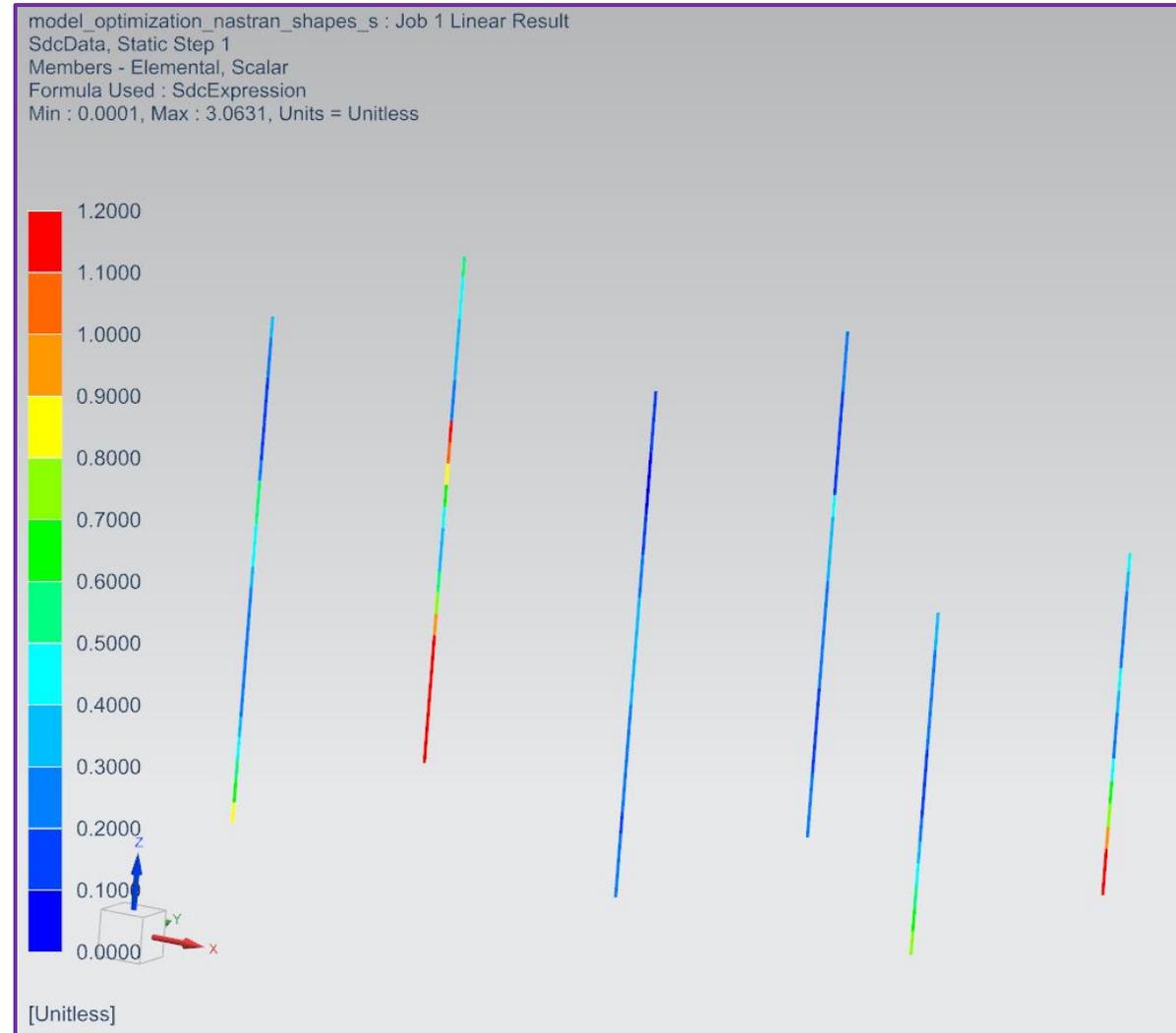
6

Press *OK*



Some members of 17..Beam Square tube d=150mm_Steel Property have UF Overall value above 1.
An Optimization Rule for these members will be created.

The Plot has been created to see the UF Overall for one Property.



Optimization Tool allows to take the best design decision for the structure by calculating different combinations of design input.

Optimization Rule consists of a set of conditions that represent the part of the model, the type of optimization and the referring parameters (usually, Utilization Factors) of design standards to be optimized.

The following types of the rules can be created:

Beam Rule - beam/bar element cross section, yield stress and young modulus can be optimized;

Plate Element Rule - plate/shell element thickness, yield stress and young modulus can be optimized;

Plate Buckling Rule - plate buckling plate thickness, yield stress and young modulus can be optimized;

Weld Strength Rule - weld type and dimensions (leg sizes, throat thickness etc.) can be optimized.

Note: Only one Rule of each type can be created within one optimization tool. Plate element and Plate buckling Rules cannot be created within one optimization tool.

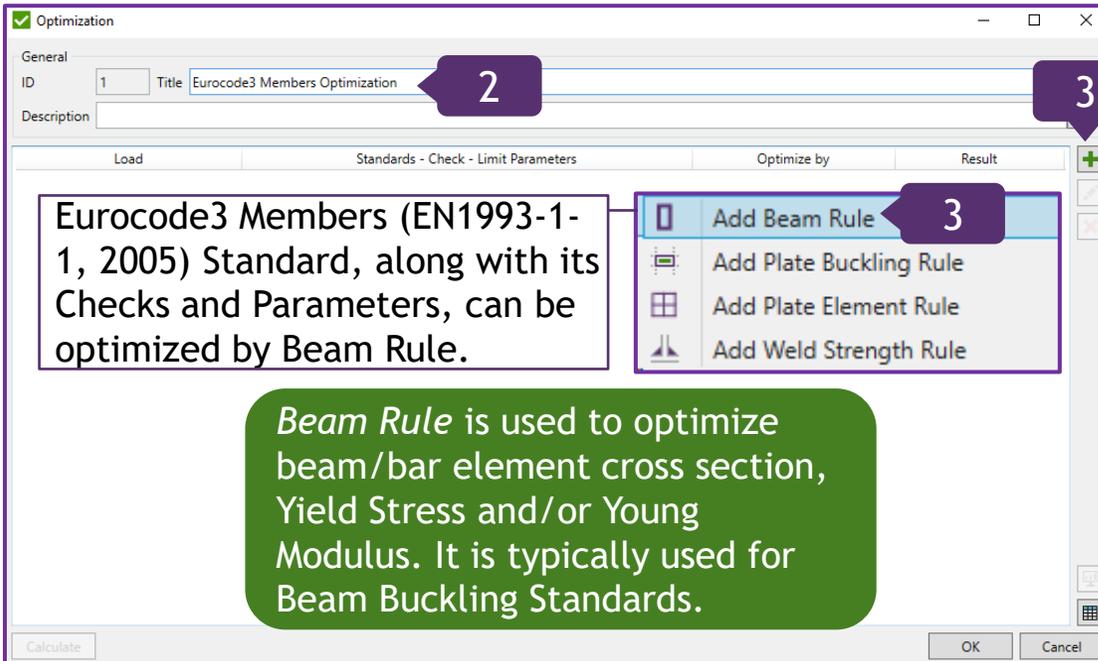
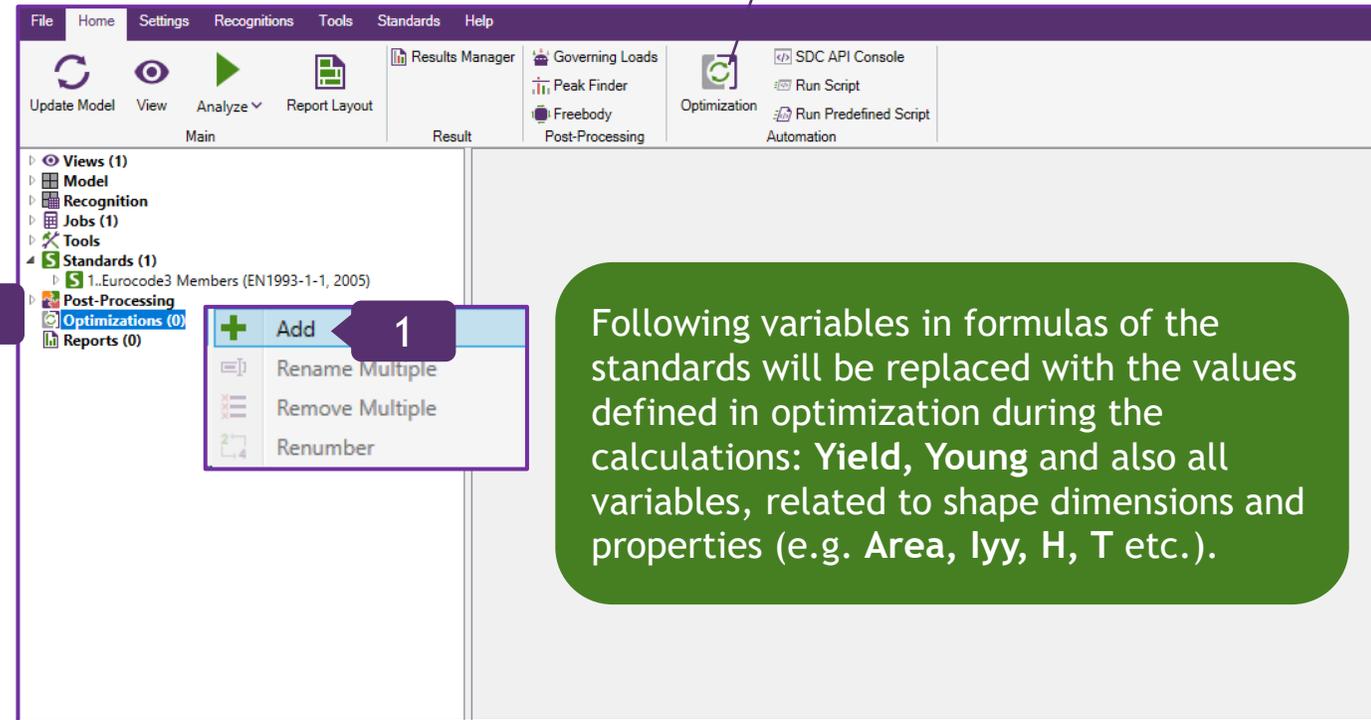
Create Optimization Rule

1 In the Model Tree, execute right click on *Optimization* and select *Add*

2 Title: *Eurocode3 Members Optimization*

3 Press  to create first *Optimization Rule*; Select *Add Beam Rule*

An alternative method of using Optimization Tool is placed in Home section of the Ribbon.



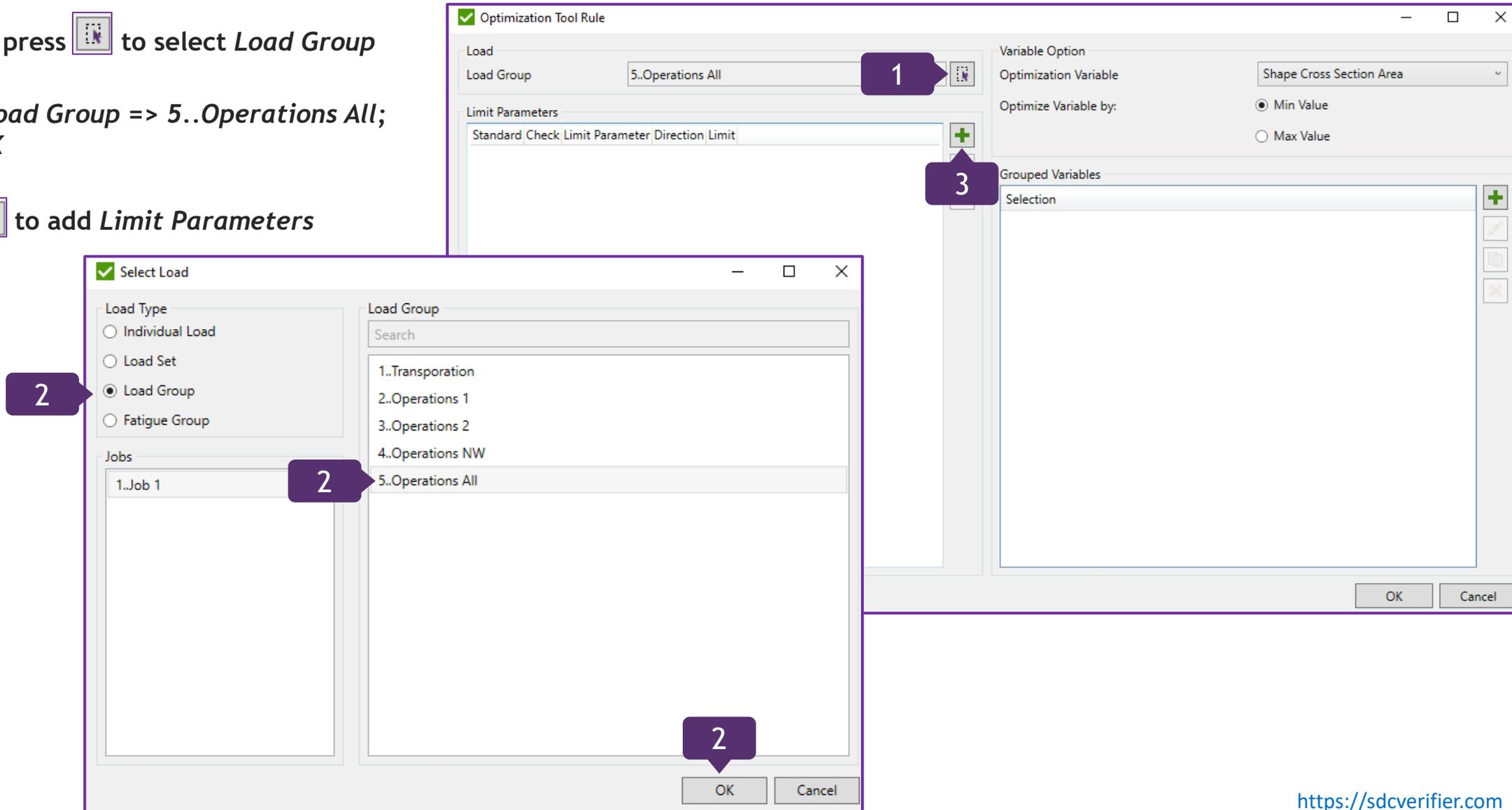
Note: If there are multiple Standards, calculated in the SDC Verifier Project, all of them will be listed in Select Limit Parameters.

Optimization Tool Rule. Eurocode3 Members

1 In Load, press  to select *Load Group*

2 Select *Load Group* => *5..Operations All*;
Press *OK*

3 Press  to add *Limit Parameters*



The screenshot shows two overlapping dialog boxes. The background dialog is titled "Optimization Tool Rule" and has a "Load" section with a dropdown menu set to "5..Operations All" (annotated with a '1'). Below it is a "Limit Parameters" table with columns "Standard", "Check", "Limit Parameter", "Direction", and "Limit", and a "+" button (annotated with a '3'). The foreground dialog is titled "Select Load" and has a "Load Type" section with radio buttons for "Individual Load", "Load Set", "Load Group" (selected), and "Fatigue Group" (annotated with a '2'). It also has a "Jobs" section with a list containing "1..Job 1" and "5..Operations All" (annotated with a '2'). The "Load Group" section has a search box and a list of options: "1..Transporation", "2..Operations 1", "3..Operations 2", "4..Operations NW", and "5..Operations All" (annotated with a '2'). At the bottom of the "Select Load" dialog are "OK" and "Cancel" buttons (annotated with a '2'). The "Optimization Tool Rule" dialog also has a "Variable Option" section with a dropdown set to "Shape Cross Section Area" and radio buttons for "Min Value" (selected) and "Max Value". At the bottom of the "Optimization Tool Rule" dialog are "OK" and "Cancel" buttons.

4

Standard: 3..Eurocode3 Members (EN1993)-1-1, 2005);
Check: 22..Buckling and Overall;
Parameter: 7..Uf Overall

5

In Limit, press

6

Type: *Between*;
Minimum: 0 and Maximum: 1;

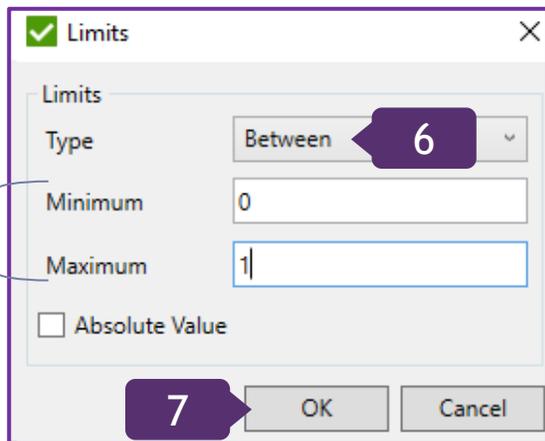
7

Press OK

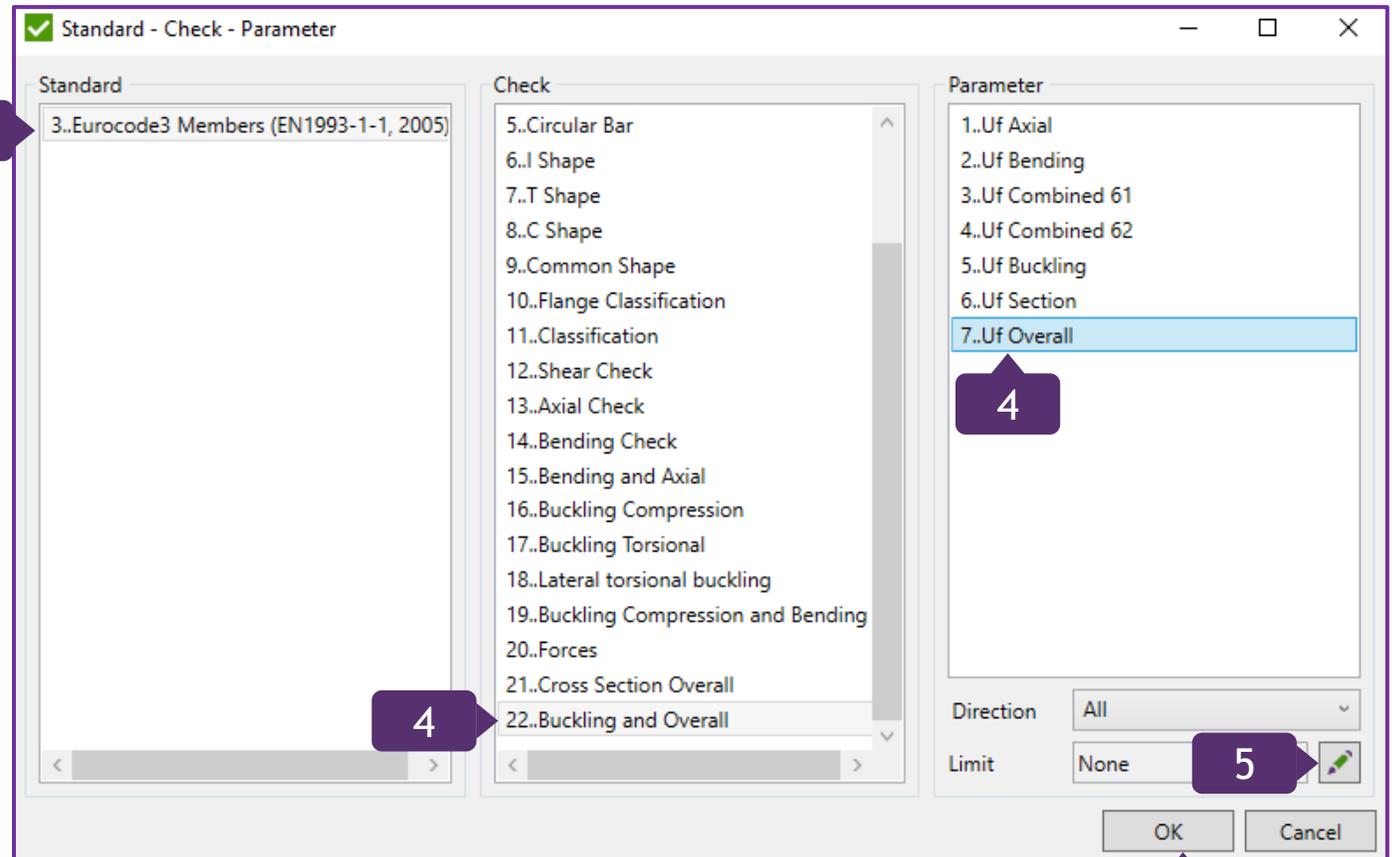
8

Press OK

Limits are used to filter results and find the best matching optimal result. A result that passes the limits and matches the variable type (e.g. Min Cross Section Area) will be chosen as an optimal result.



Note: Only checks that fit the type of the rule (beam, plate buckling etc.) will be displayed in the list.



Note: If the parameter had already been added it will not be shown in the list when adding another parameters.

Add Properties from List

1 Optimization Variable: *Shape Cross Section Area*;
Optimize Variable by: *Min Value* is ON

2 Press  to add multiple Grouped Variables

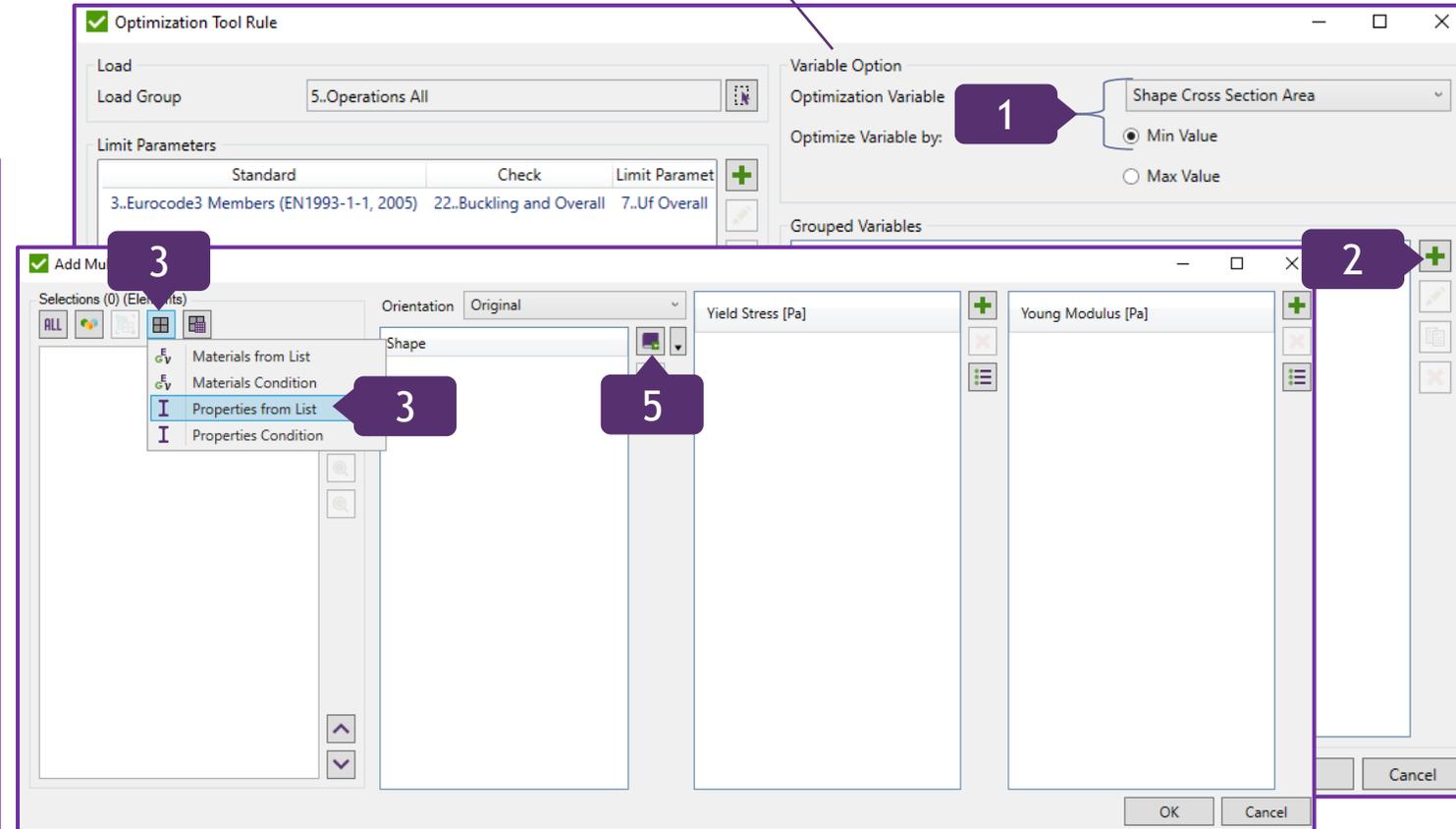
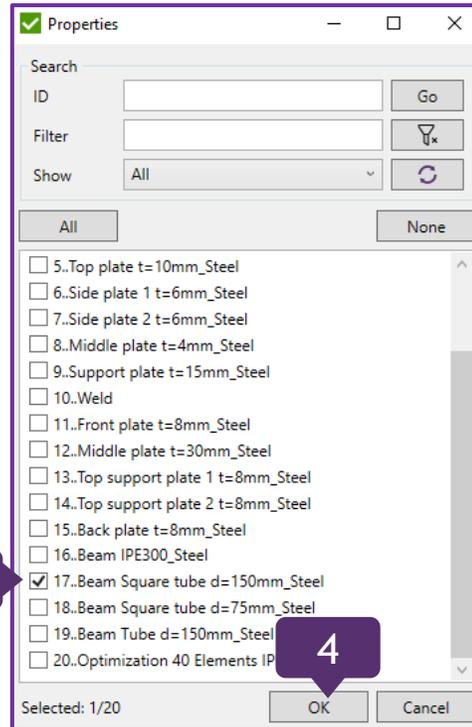
3 Press  and select Properties from List

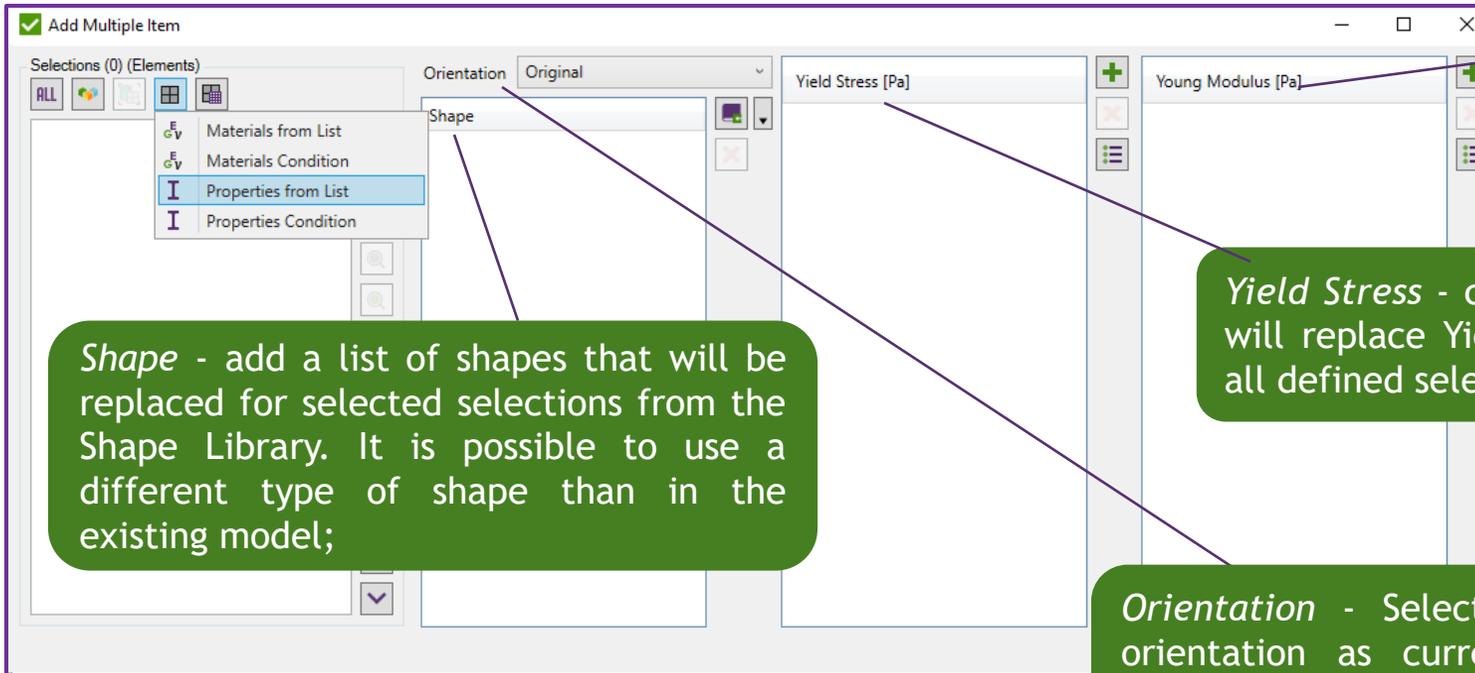
Select 17.. *Beam Square tube*
d=150mm_Steel;
Press OK

5 Press  to open
Shape Library

Optimization Variable - select the type of the variable to be optimized
(Shape Cross Section Area, Young Modulus, Yield Stress);

Optimize Variable by - find an optimal result based the on min/max value
of the variable type. For example, to optimize the structure according to
the lowest weight - select Min Value of Shape Cross Section Area.





Shape - add a list of shapes that will be replaced for selected selections from the Shape Library. It is possible to use a different type of shape than in the existing model;

Young Modulus - create a list of variables that will replace Young Modulus of the material in all defined selections;

Yield Stress - create a list of variables that will replace Yield Stress of the material in all defined selections;

Orientation - Select Left, Right, Up, Down or Original (the same orientation as current cross section in the model) cross section orientation that is applied for all selected shapes from the library. For example H cross section can be used as I with modified orientation.

Note: Optimization result will be calculated for all combinations of Selection + Shape + Yield Stress + Young Modulus.

1

Select the cross-sections, as shown on the screen

2

Press *OK*

3

Press *OK*

Shape Library contains a list of predefined or user-defined shapes that can be used in the Optimization. Shapes can be filtered, organized, added or modified.

The Shapes can be filtered by Name in this section.

The screenshot shows the 'Shape Library' dialog box with the 'Filter by Name' field set to 'HSS'. A list of shapes is displayed with columns for Name, Type, Width, Height, Area, I_{yy} , and I_{zz} . The shape HSS20X12X.625 is selected. To the right, a detailed view of the selected shape shows its properties: Area, Moment of Inertia, Torsional Constant, Y Shear Area, Z Shear Area, Nonstructural Mass/length, Warping Constant, and Perimeter. A diagram of the rectangular tube cross-section is shown with dimensions: width 304.8 mm, height 508 mm, and wall thickness 15.875 mm. The coordinate system (y, z) is also indicated.

1 The result

2

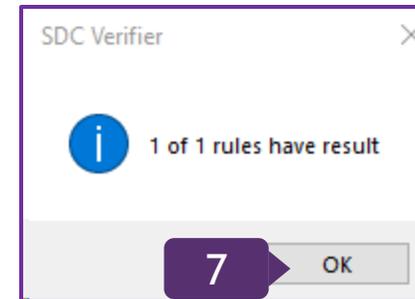
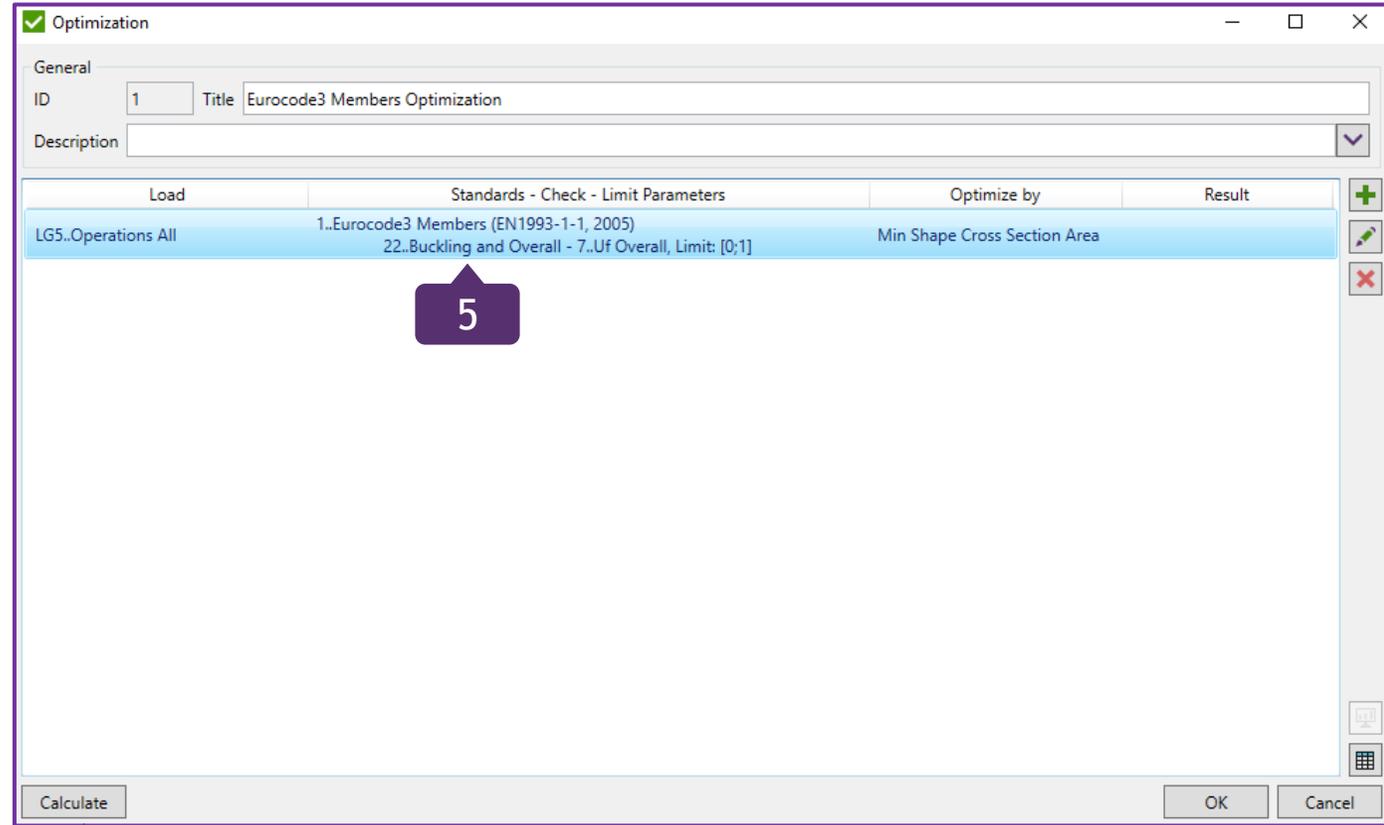
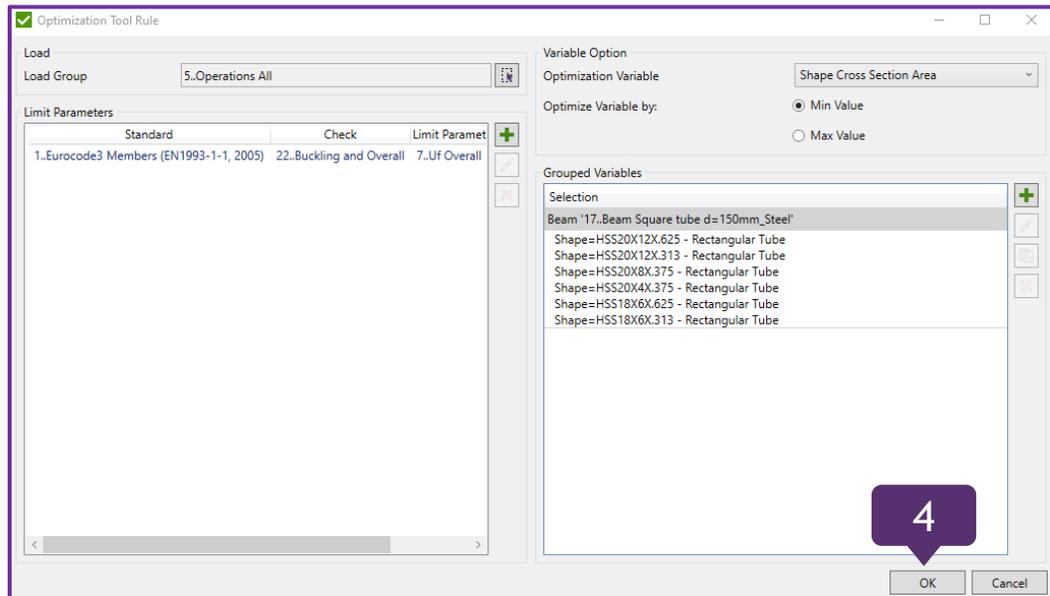
3

4 Press **OK**

5 Activate the section

6 Press *Calculate* to run *the Optimization*

7 Press **OK**



Optimization Results in a Table

1 Activate the section

2 Press and select *All Results*;
Press *Close*

3 Press and select *Optimal Result*;
Press *Close*

The screenshot shows the 'Optimization' window with a table of results. A 'Result Table' dialog is open, showing a list of beam shapes and their corresponding values. A callout box points to the 'All Result' button in the bottom right corner of the 'Result Table' dialog.

Group	Yield Stress [kPa]	Young Modulus [kPa]	Shape	1..Eurocode3 Members (EN1993-1-1, 2005) 22..Buckling and Overall 7..Uf Overall
Beam '17..Beam Square tube	Original Model (240000.00)	Original Model (210000000.0)	Original Model (17..Beam Square tube)	3.06
Beam '17..Beam Square tube			HSS20X12X.625 - Rectangular	0.12
Beam '17..Beam Square tube			HSS20X12X.313 - Rectangular	0.23
Beam '17..Beam Square tube			HSS20X8X.375 - Rectangular	0.31
Beam '17..Beam Square tube			HSS20X4X.375 - Rectangular	0.67
Beam '17..Beam Square tube			HSS18X6X.625 - Rectangular	0.30
Beam '17..Beam Square tube			HSS18X6X.313 - Rectangular	0.55

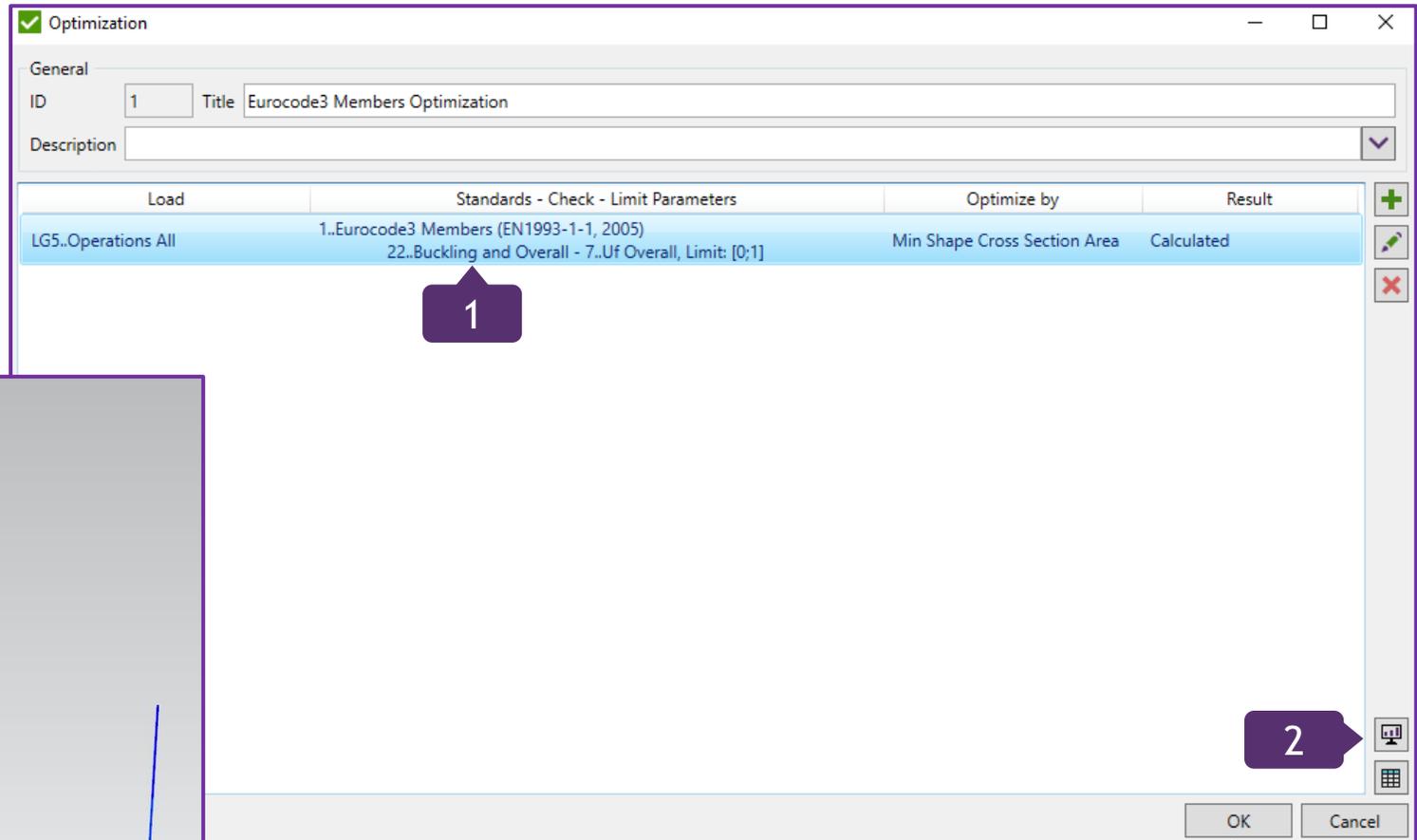
This is the Optimal Result, which will be used for changing the Beam shape.

1 Activate the section

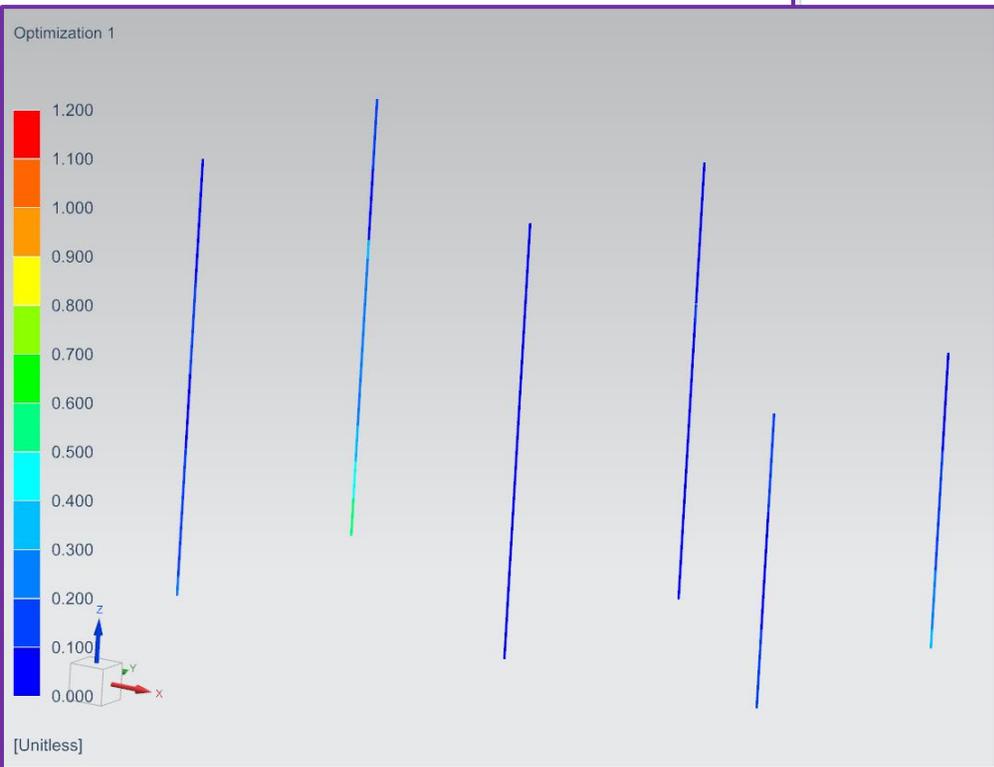
2 Press  to plot optimal result on the model

3 Press *OK*

The Result



Load	Standards - Check - Limit Parameters	Optimize by	Result
LG5..Operations All	1..Eurocode3 Members (EN1993-1-1, 2005) 22..Buckling and Overall - 7..Uf Overall, Limit: [0;1]	Min Shape Cross Section Area	Calculated

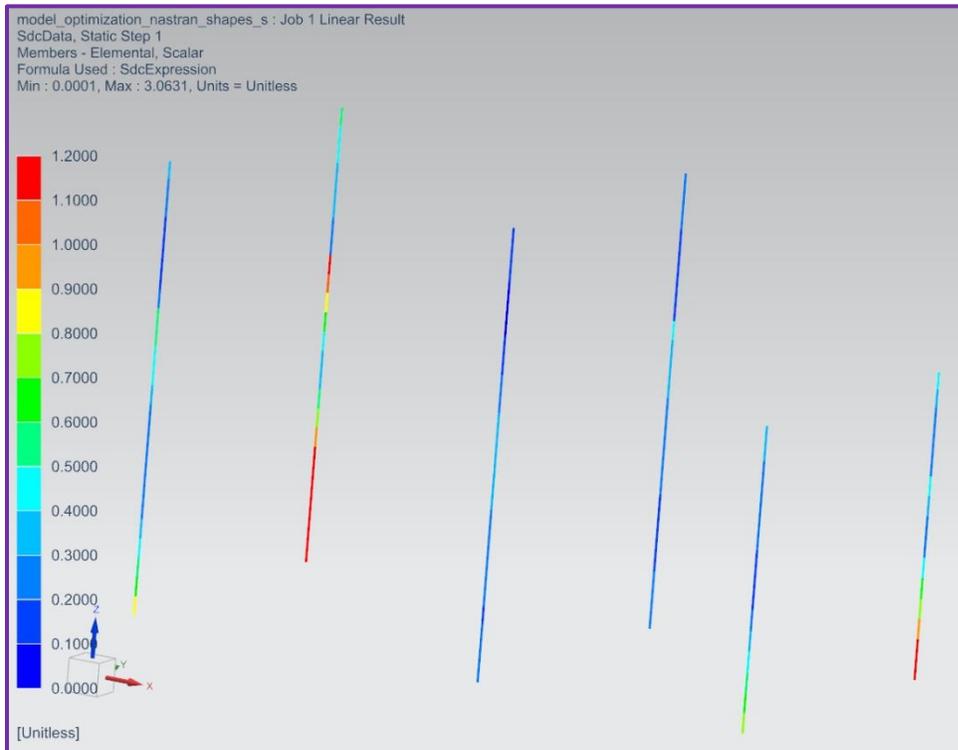


2

3

This Tutorial demonstrates only the workflow with the Optimization Tool. Optimization Rule can be set even more precisely. For example, using the Peak Finder you can group only the overshooting elements into a Component and run the Optimization on this Component. Multiple rules with different variables can be set.

The Result before the Optimization



The Result after the Optimization

