



Tutorial

# **Eurocode3 Fatigue (EN1993-1-9, 2005) Optimization. Plate Element Rule**

Updated on: April 2nd, 2024

Tested with: SDC Verifier 2023 R2

Femap version 2023.2

- This step-by-step tutorial demonstrates the functionality of SDC Verifier Optimization Tool, incorporating Plate Element 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 Femap \(sdcverifier.com\)](https://sdcverifier.com)

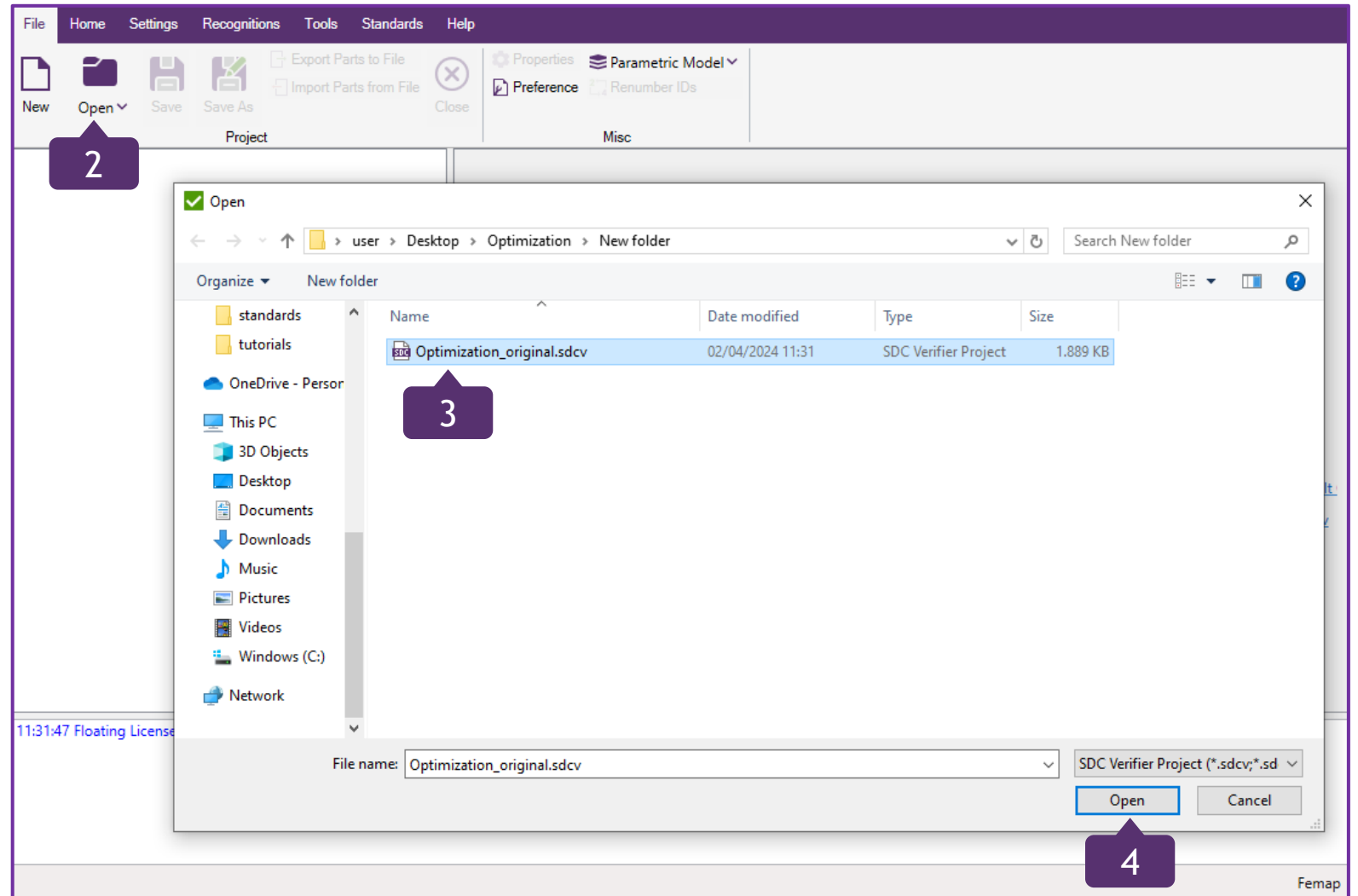
# Open the Starter Model

1 Launch SDC Verifier for Femap

2 In *File* section, press *Open*

3 Select a project  
*Optimization\_original.sdcv*

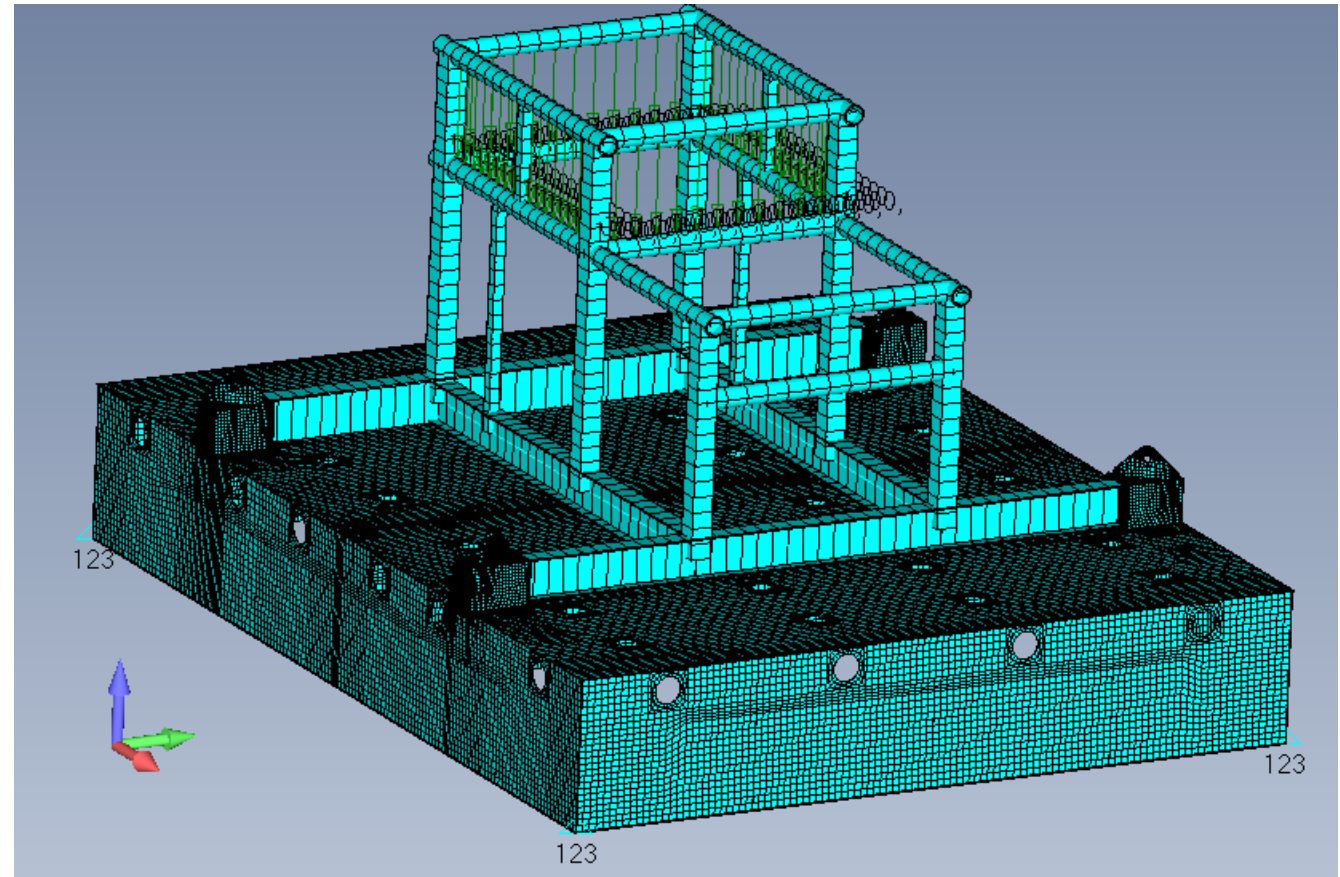
4 Press *Open*



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)
- ▷ Model
- ▷ Recognition
- ▶ Jobs (1)
  - ▶ 1..Job 1
    - ▷ Individual Loads (9)
    - ▷ Predefined Load Cases (0)
    - ▷ Load Sets (28)
    - ▷ Load Groups (5)
    - ▷ FG Fatigue Groups (1)
    - ▷ Tables (0)
    - ▷ Plots (0)
- ▷ Tools
- ▶ Standards (1)
  - ▷ 4..Eurocode3 Fatigue (EN1993-1-9, 2005)
- ▷ 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/eurocode3-fatigue/>

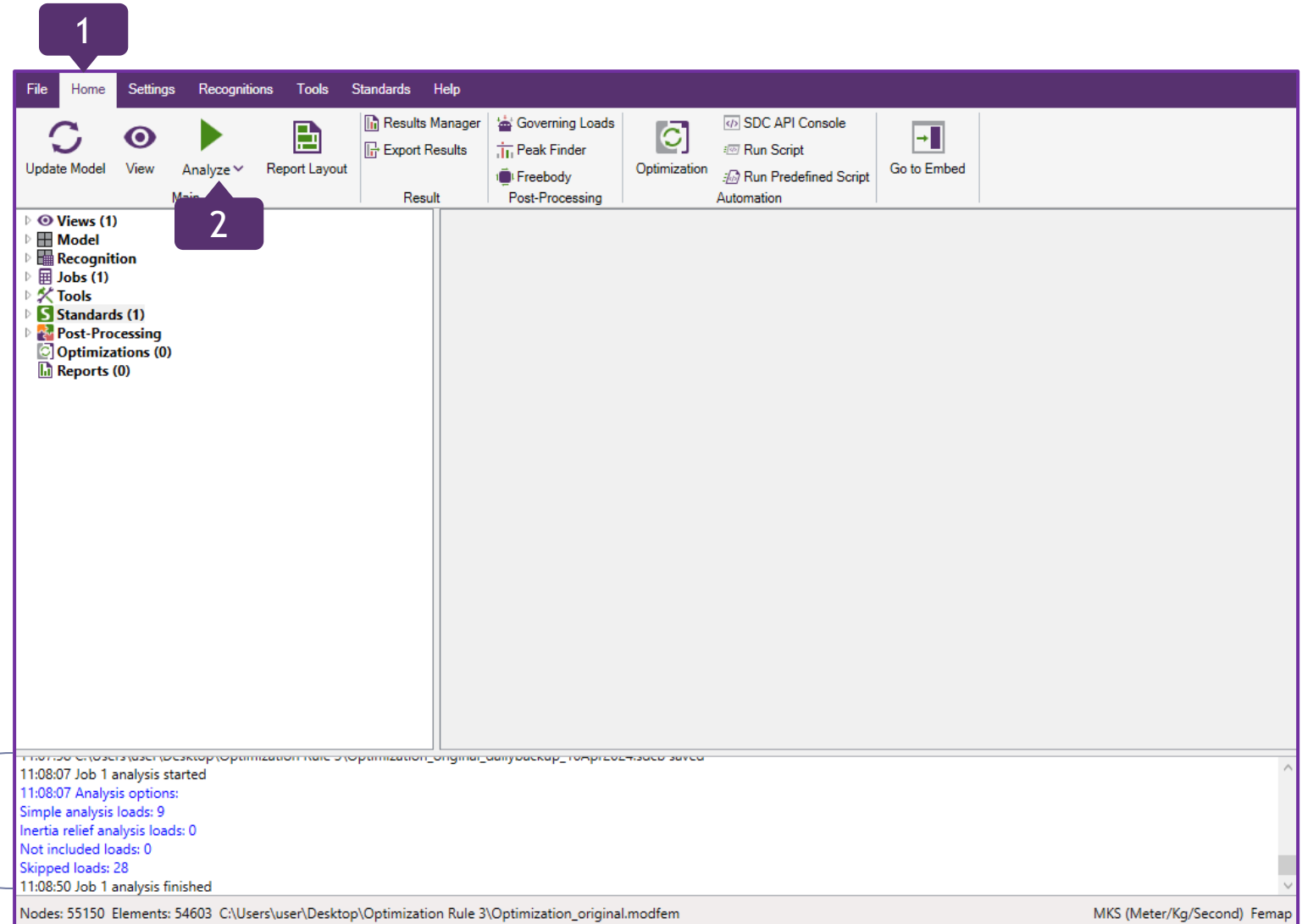
# Run Analysis

1

Go to *Home* section on the Ribbon

2

Press  on the toolbar to analyze Job



Job 1 analysis started and finished.

# Eurocode3 Fatigue (EN1993-1-9,2005) Criteria Plot

1

Expand Standards => 3..Eurocode3 Fatigue (EN1993-1-9, 2005) => Checks (22) and select 2..*Fatigue Check*

2

Execute right click on 2..*Fatigue Check* and select *Criteria Plot*

3

Press  to select Load Group

4

In Fatigue Group, select Fatigue Group, and then 1..*Fatigue Group 1*; Press *OK*

5

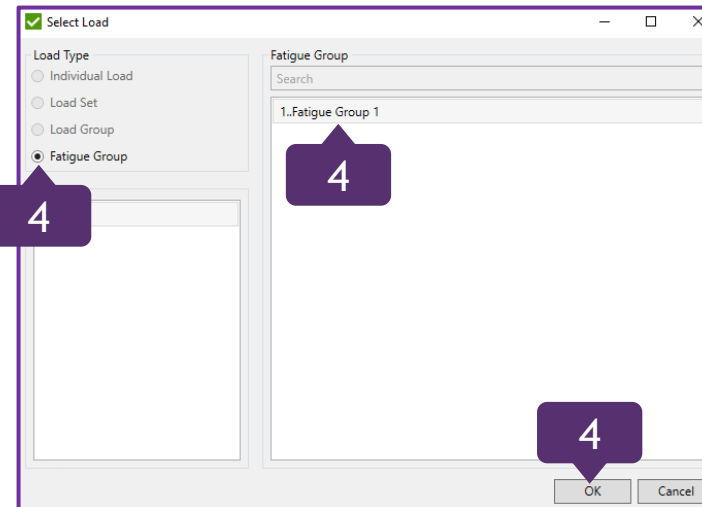
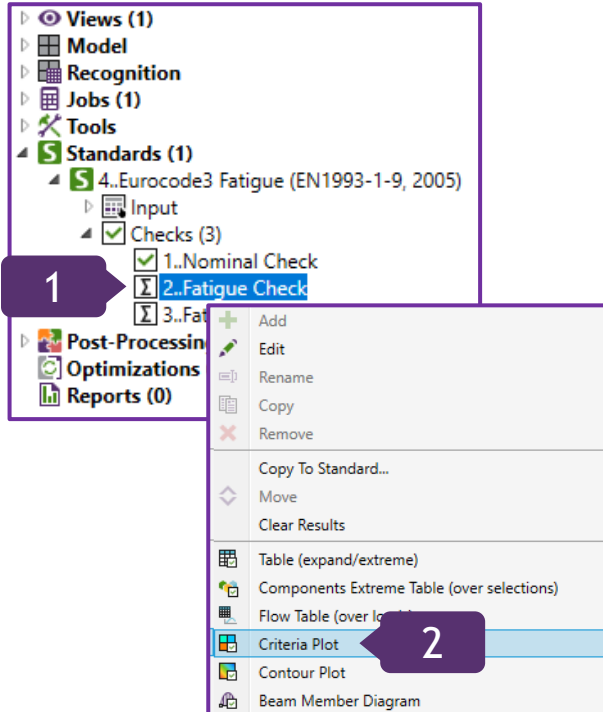
Parameter: *Summed Damage*; Direction: *Overall*

6

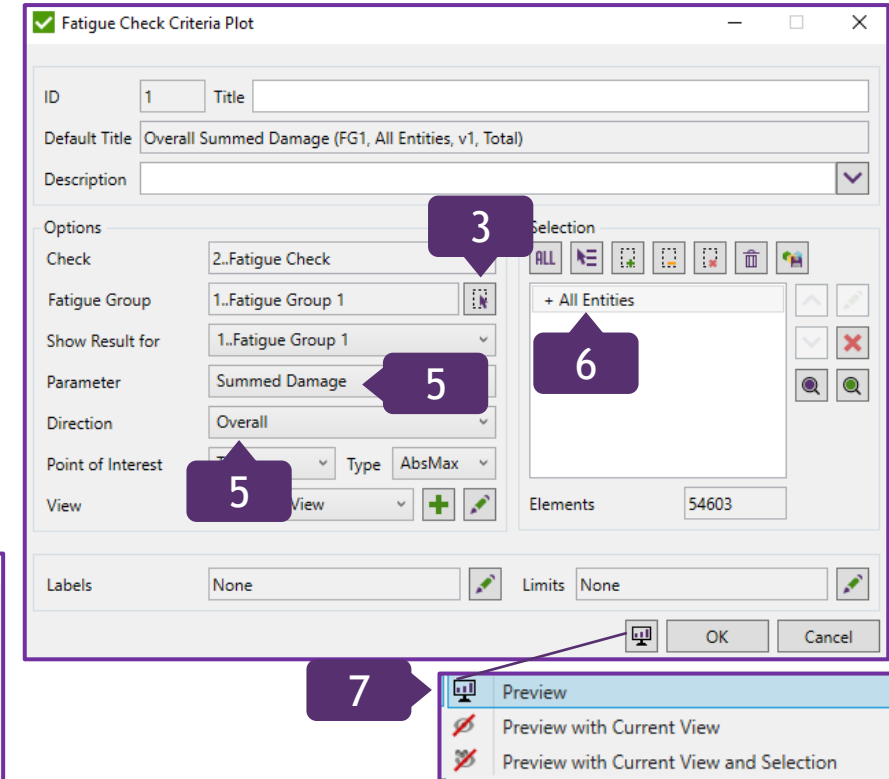
Selection: *All Entities*

7

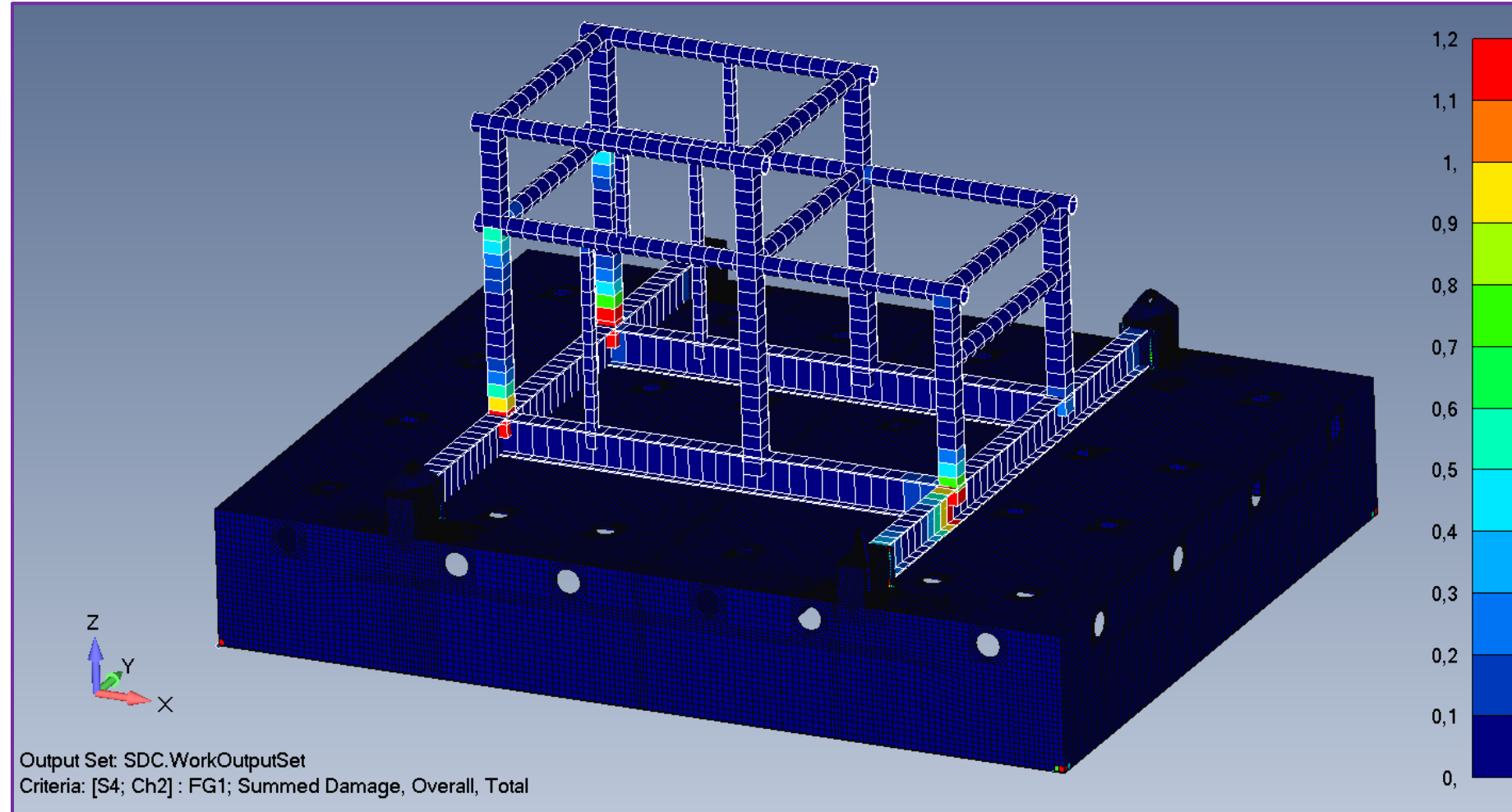
Press  and then *Preview*



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



The Plot has been created to see the Overall Summed Damage on full model.  
It is displayed in Femap window.





# Set Limits for Criteria Plot (Additional Functionality)

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.

1

In Limits, press



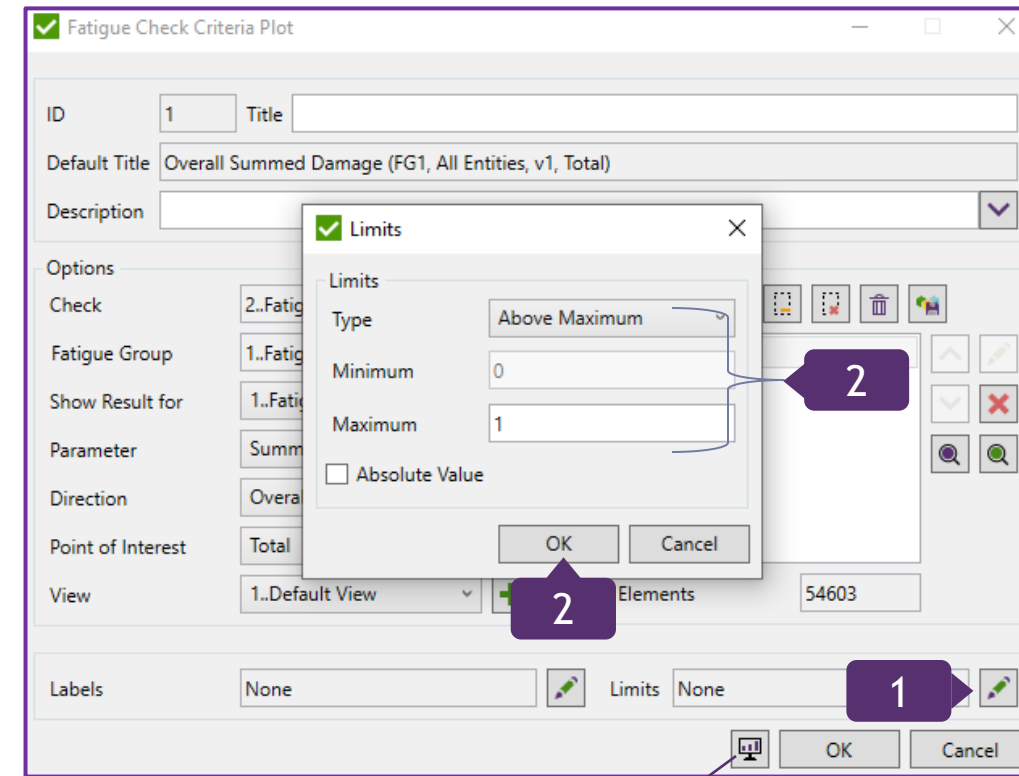
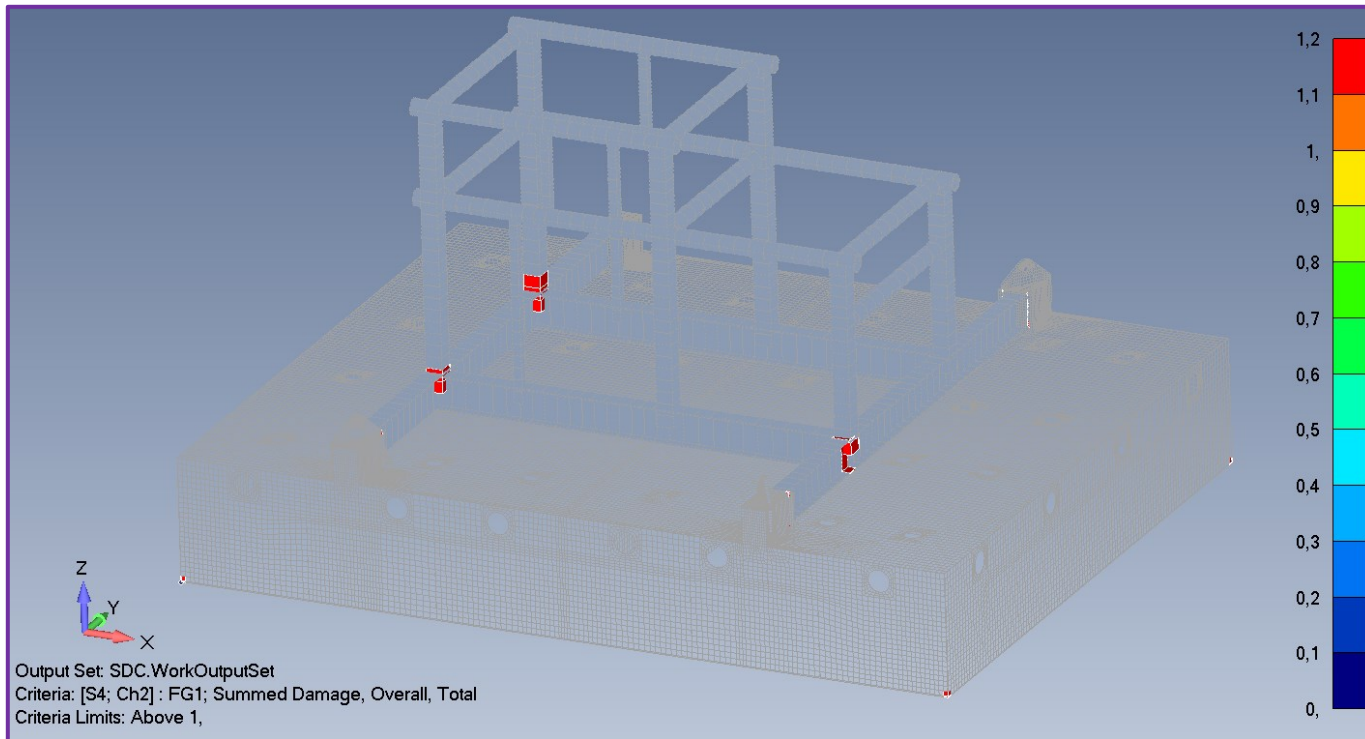
2

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




3

Press  and then *Preview*

The Result



3

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



# Eurocode3 Fatigue (EN1993-1-9,2005) Criteria Plot for One Property

1

Select + *All Entities* and press  to remove them

2

Press  to add Condition;  
Select *Components*

3

Select *11..Optimization Elements*;  
Press *OK*

4

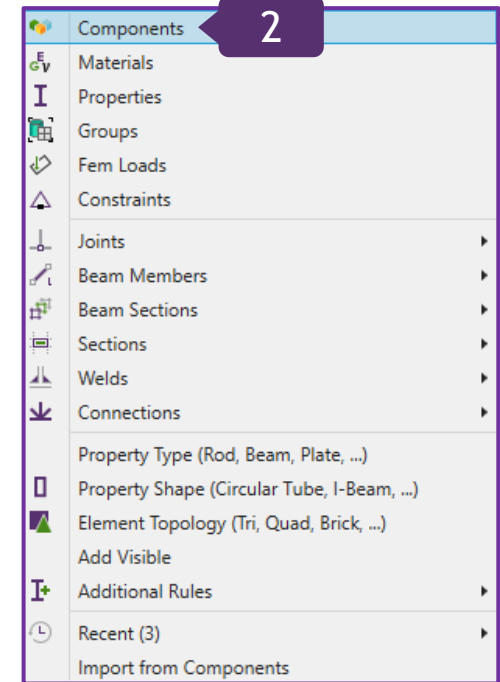
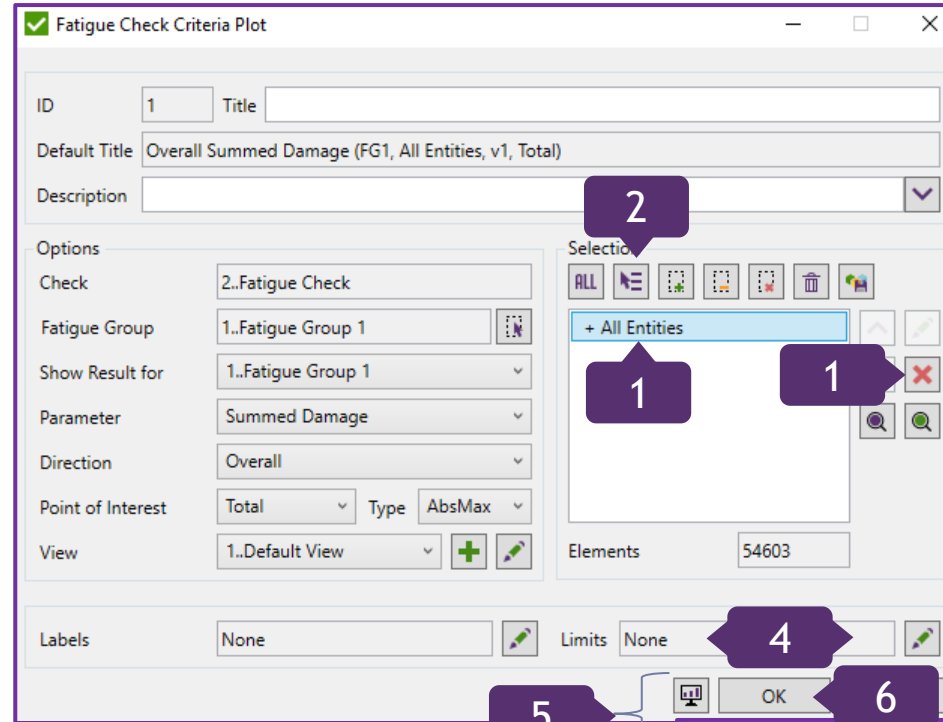
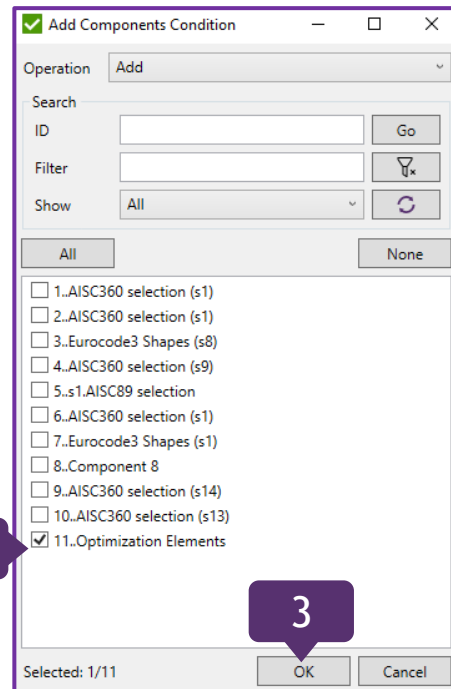
Limits: *None*

5

Press  and  
then *Preview*

6

Press *OK*

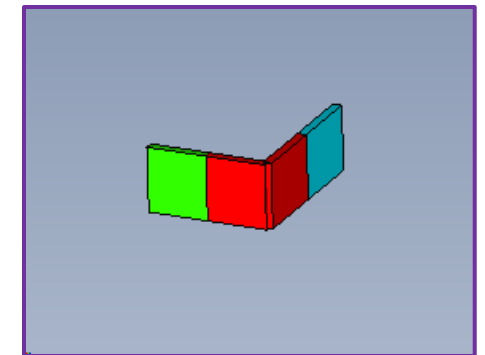


The members of Components have UF Overall value above 1. An Optimization Rule for these members will be created.

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



Zoomed-in elements before  
Optimization



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;

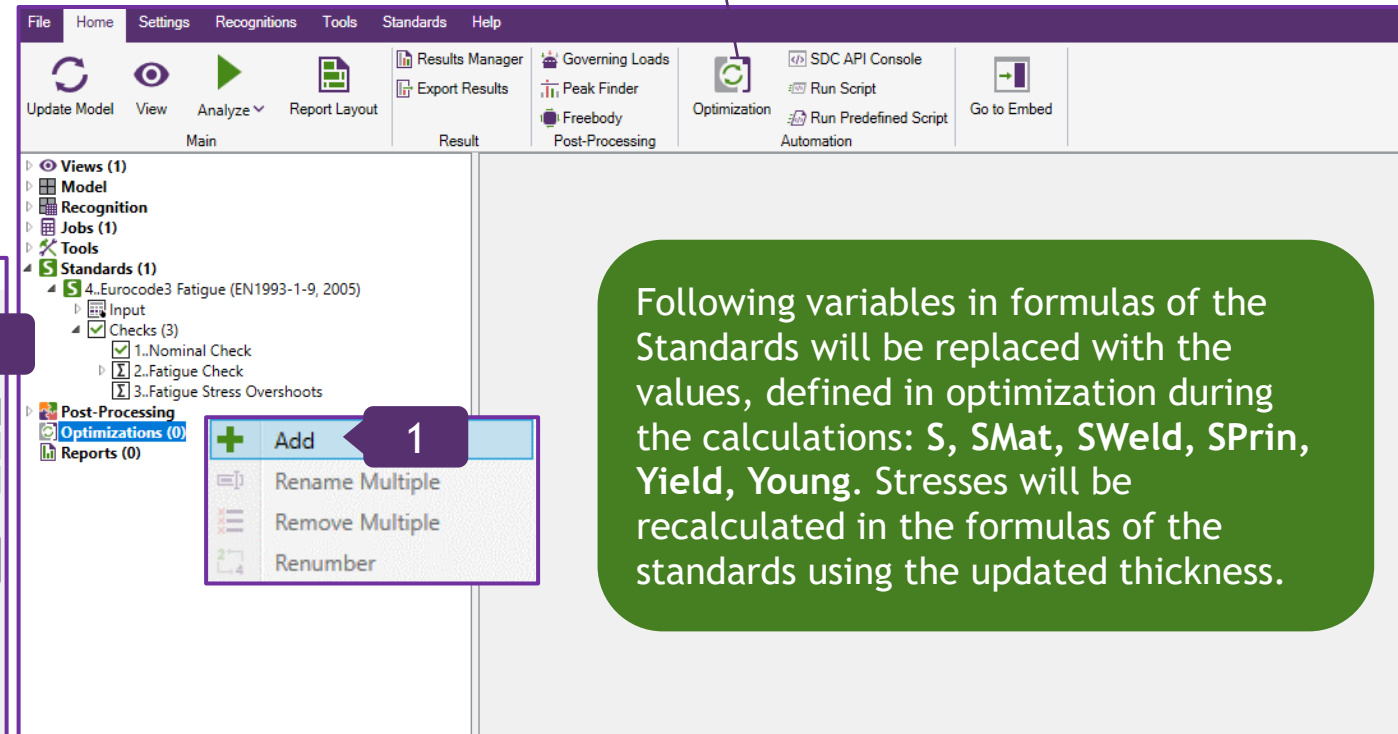
*Weld Strength Rule* - weld type and dimensions (leg sizes, throat thickness etc.) can be optimized.

**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 *Optimizations* and select *Add*
- 2 Title: *Eurocode3 Fatigue (EN1993-1-9, 2005)*
- 3 Press  to create first *Optimization Rule*; Select *Add Plate Element Rule*

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



Eurocode3 Fatigue (EN1993-1-9, 2005) Standard, along with its Checks and Parameters, can be optimized by Plate Element Rule.

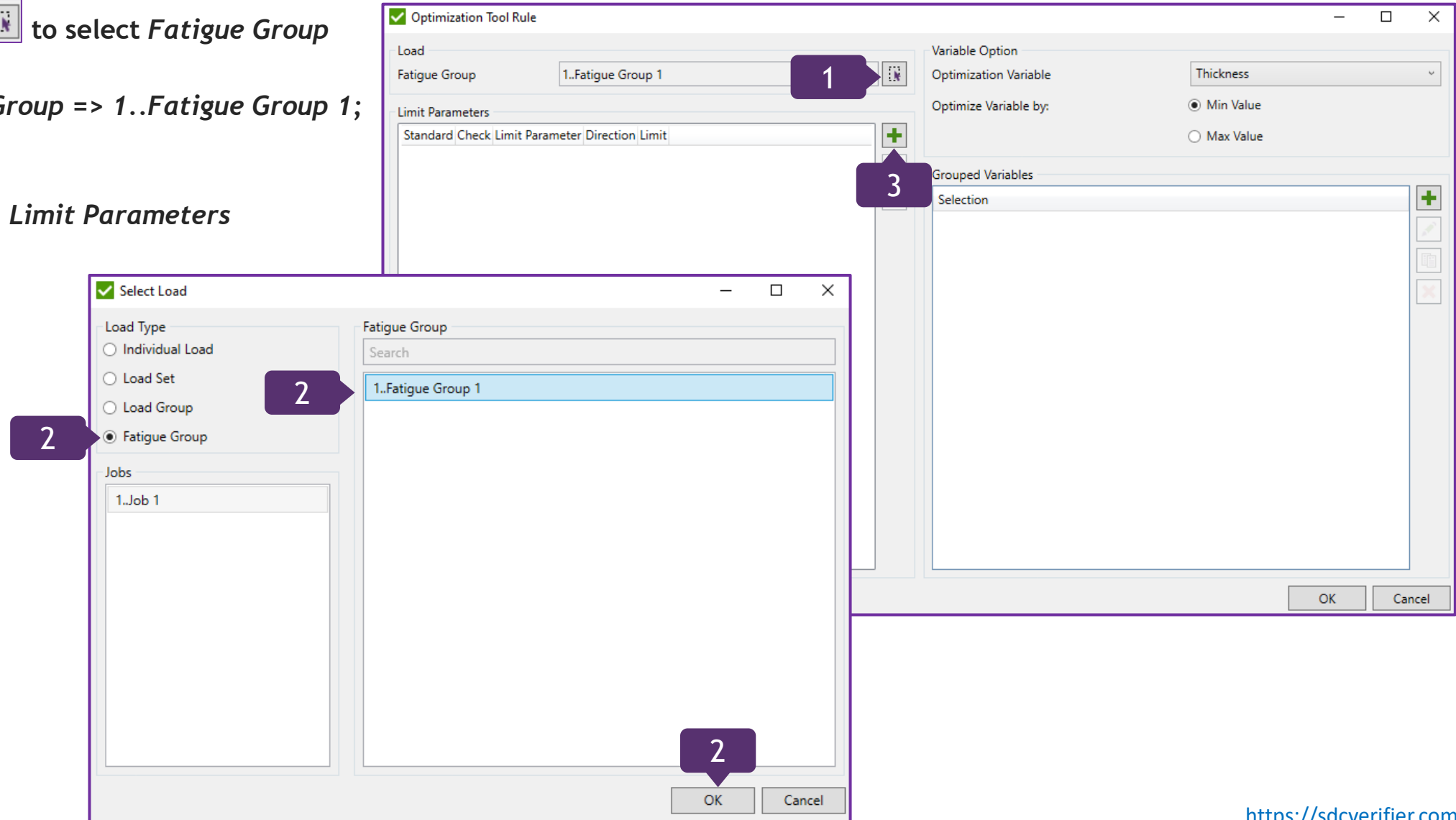
Plate Element Rule is used to optimize plate/shell element thickness, yield stress and/or young modulus. It is typically used for Fatigue Standards. Common options are described in Optimization Rule.

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

1 In Load, press  to select *Fatigue Group*

2 Select *Fatigue Group* => 1..Fatigue Group 1;  
Press *OK*

3 Press  to add *Limit Parameters*



# Optimization Tool Rule. DNV Buckling Strength (Continuation)

4

Standard: *Eurocode3 Fatigue (EN1993-1-9, 2005)*;  
Check: *2..Fatigue Check*;  
Parameter: *6..Summed Damage*

5

Direction: *Overall*

6

In Limit, press 

7

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

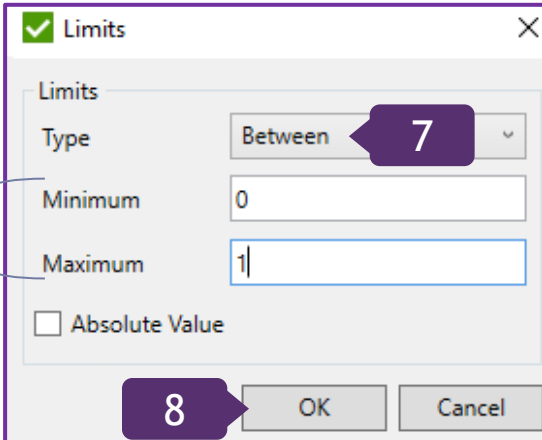
8

Press *OK*

9

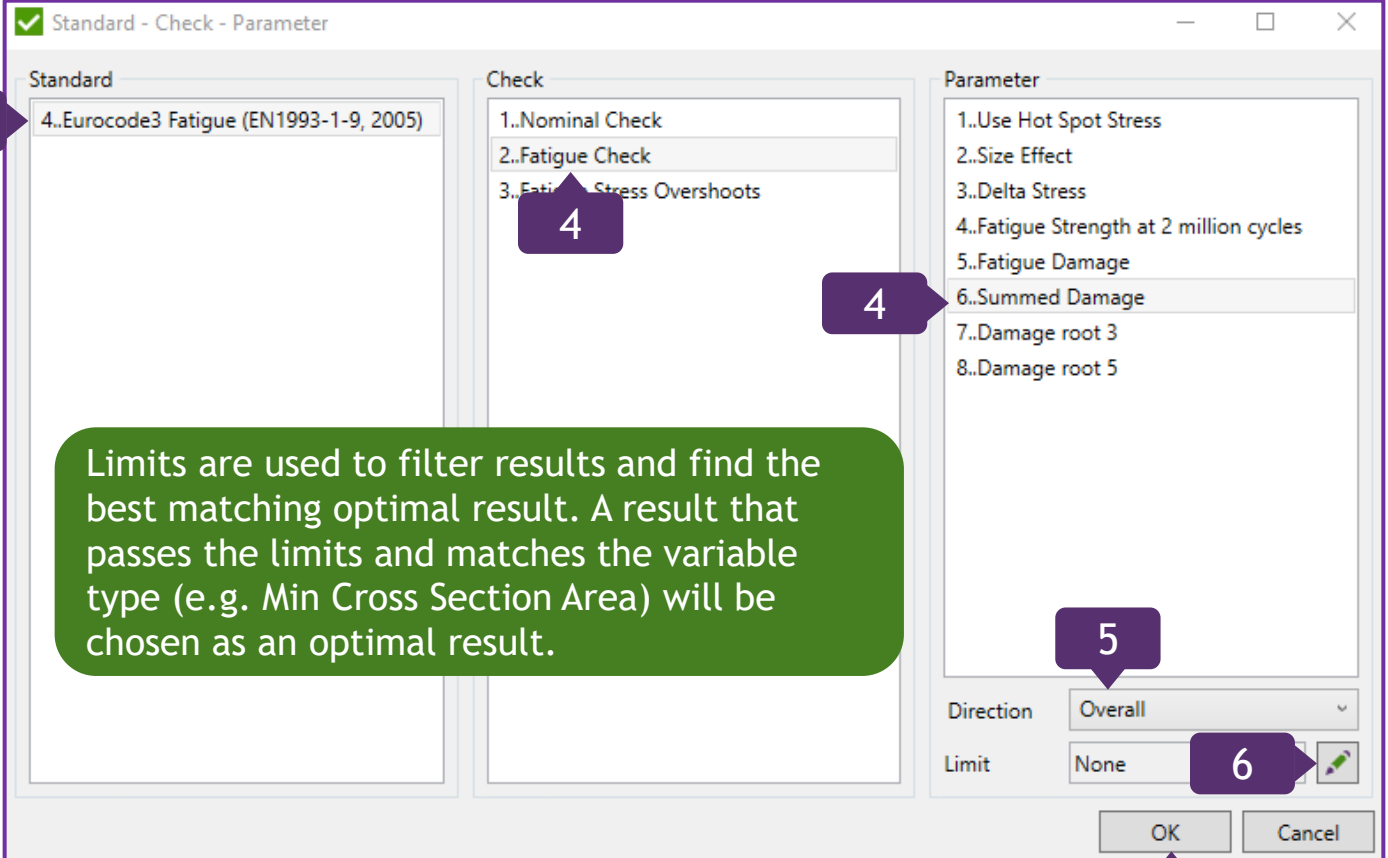
Press *OK*

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



The Limits dialog box is shown with the following settings:

- Limits: ☒
- Type: *Between* (indicated by callout 7)
- Minimum: *0*
- Maximum: *1*
- ☐ Absolute Value
- Buttons: *OK* (indicated by callout 8) and *Cancel*





The Standard - Check - Parameter dialog box is shown with the following settings:

- Standard: *4..Eurocode3 Fatigue (EN1993-1-9, 2005)* (indicated by callout 4)
- Check: *2..Fatigue Check* (indicated by callout 4)
- Parameter: *6..Summed Damage* (indicated by callout 4)
- Direction: *Overall* (indicated by callout 5)
- Limit: *None* (indicated by callout 6, with an edit icon next to it)
- Buttons: *OK* (indicated by callout 9) and *Cancel*

A green box contains the text: 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.

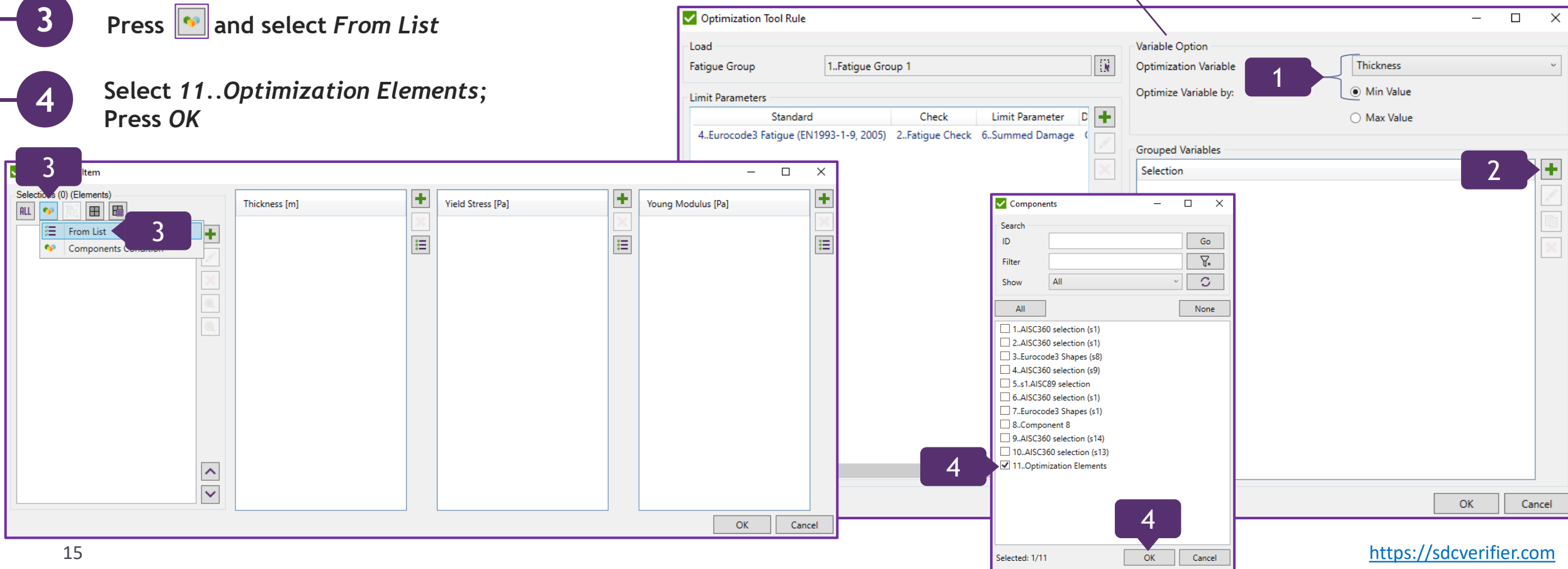
Note: If the parameter had already been added, it will not be shown in the list when adding another parameters.

# Add Multiple Item

- 1 Optimization Variable: Plate Thickness;  
Optimize Variable by: *Min Value* is ON
- 2 Press  to add multiple Grouped Variables
- 3 Press  and select *From List*
- 4 Select 11..*Optimization Elements*;  
Press OK

*Optimization Variable* - select the type of the variable to be optimized (Plate Thickness, 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 Plate Thickness.



The screenshot displays three overlapping windows from the SDC Verifier software:

- Optimization Tool Rule**: This window is in the background. It has a 'Load' section with 'Fatigue Group' set to '1..Fatigue Group 1'. The 'Limit Parameters' section contains a table with columns 'Standard', 'Check', and 'Limit Parameter'. The 'Variable Option' section shows 'Optimization Variable' set to 'Thickness' and 'Optimize Variable by' set to 'Min Value'. The 'Grouped Variables' section has a 'Selection' list with a '+' button next to it.
- Item**: This window is in the foreground. It has a 'Selections (0) (Elements)' list on the left with 'From List' and 'Components' buttons. The main area has three columns: 'Thickness [m]', 'Yield Stress [Pa]', and 'Young Modulus [Pa]'. Each column has a '+' button. The 'From List' button is highlighted with a callout '3'.
- Components**: This window is in the foreground. It has a 'Search' section with 'ID', 'Filter', and 'Show' fields. Below is a list of components with checkboxes. The component '11..Optimization Elements' is checked, and a callout '4' points to it. The 'Selected: 1/11' text is at the bottom.

Numbered callouts indicate the steps:

- 1: Points to the 'Optimization Variable' dropdown in the 'Optimization Tool Rule' window.
- 2: Points to the '+' button in the 'Grouped Variables' section of the 'Optimization Tool Rule' window.
- 3: Points to the 'From List' button in the 'Item' window.
- 4: Points to the '11..Optimization Elements' checkbox in the 'Components' window.



# Add Multiple Item (Continuation)

5

Press  to define Thickness [m]:  
0.008; 0.010; 0.015


6

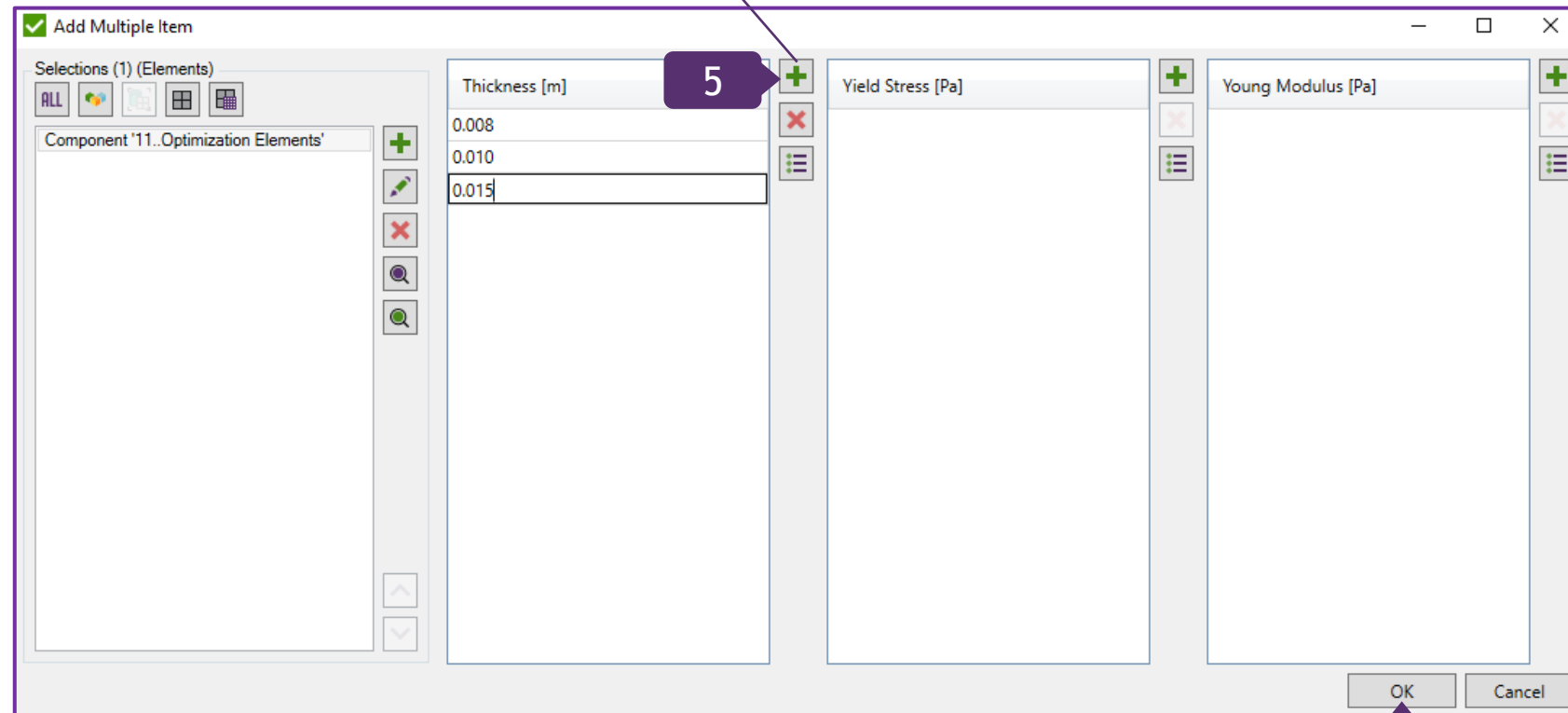
Press **OK**

*Thickness* - create a list of variables that will replace Thickness of the plate/shell elements in all defined selections;

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

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

Note: When defining Thickness [m], that refers to Step 5, it is required to press  each time the parameter is incorporated.



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

# Selection of Grouped Variables; Calculation

1

Press OK

2

Activate the section

3

Press *Calculate* to run the Optimization

4

Press OK

The image shows two windows from the SDC Verifier software. The 'Optimization Tool Rule' window is in the background, and the 'Optimization' window is in the foreground. Both windows have numbered callouts (1-4) indicating the steps for running an optimization.

**Optimization Tool Rule Window:**

- Load:** Fatigue Group 1..Fatigue Group 1
- Limit Parameters:** A table with columns: Standard, Check, Limit Parameter, and a delete icon (+). The row '4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check 6..Summed Damage' is selected.
- Variable Option:** Optimization Variable: Thickness. Optimize Variable by: ☒ Min Value, ☐ Max Value.
- Grouped Variables:** A list box showing 'Component '11..Optimization Elements'' with a '+' icon. Below it, three variables are listed: Thickness=0.008, Thickness=0.010, and Thickness=0.015.
- Buttons:** OK and Cancel.

**Optimization Window:**

- General:** ID: 1, Title: Eurocode3 Fatigue (EN1993-1-9, 2005), Description: (empty).
- Table:** A table with columns: Load, Standards - Check - Limit Parameters, Optimize by, and Result. The row 'FG1..Fatigue Group 1 4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check - 6..Summed Damage.Overall, Limit: [0;1] Min Thickness' is selected.
- Buttons:** Calculate, Change Model, OK, Cancel.

**SDC Verifier Message Box:**

- Text:** 1 of 1 rules have result.
- Buttons:** OK.

**Callouts:**

- 1:** Points to the OK button in the 'Optimization Tool Rule' window.
- 2:** Points to the '4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check - 6..Summed Damage' row in the 'Optimization' window table.
- 3:** Points to the 'Calculate' button in the 'Optimization' window.
- 4:** Points to the OK button in the 'SDC Verifier' message box.

**Annotation:** A callout box points to the 'Grouped Variables' list in the 'Optimization Tool Rule' window, containing the text: 'Grouped Variables that are calculated.'

# 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*

**Result Table**

Group	Thickness [m]	4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check 6..Summed Damage.Overall
Component '11..Optimization Elements'	0.010	0.72

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

3 Close

**Optimization**

General

ID 1 Title Eurocode3 Fatigue (EN1993-1-9, 2005)

Description

Load	Standards - Check - Limit Parameters	Optimize by	Result
FG1..Fatigue Group 1	4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check - 6..Summed Damage.Overall, Limit: [0;1]	Min Thickness	Calculated

1

**Result Table**

Group	Yield Stress [Pa]	Young Modulus [Pa]	Thickness [m]	4..Eurocode3 Fatigue (EN19 2..Fatigue Check 6..Summed Damage.Overall
Component '11..Optimization	Original Model (240.00e+6)	Original Model (210000.00e-	Original Model (0.006)	4.74
Component '11..Optimization			0.008	1.61
Component '11..Optimization			0.010	0.72
Component '11..Optimization			0.015	0.18

The results for all variables.

2,3

Optimal Result

All Result

2 Close

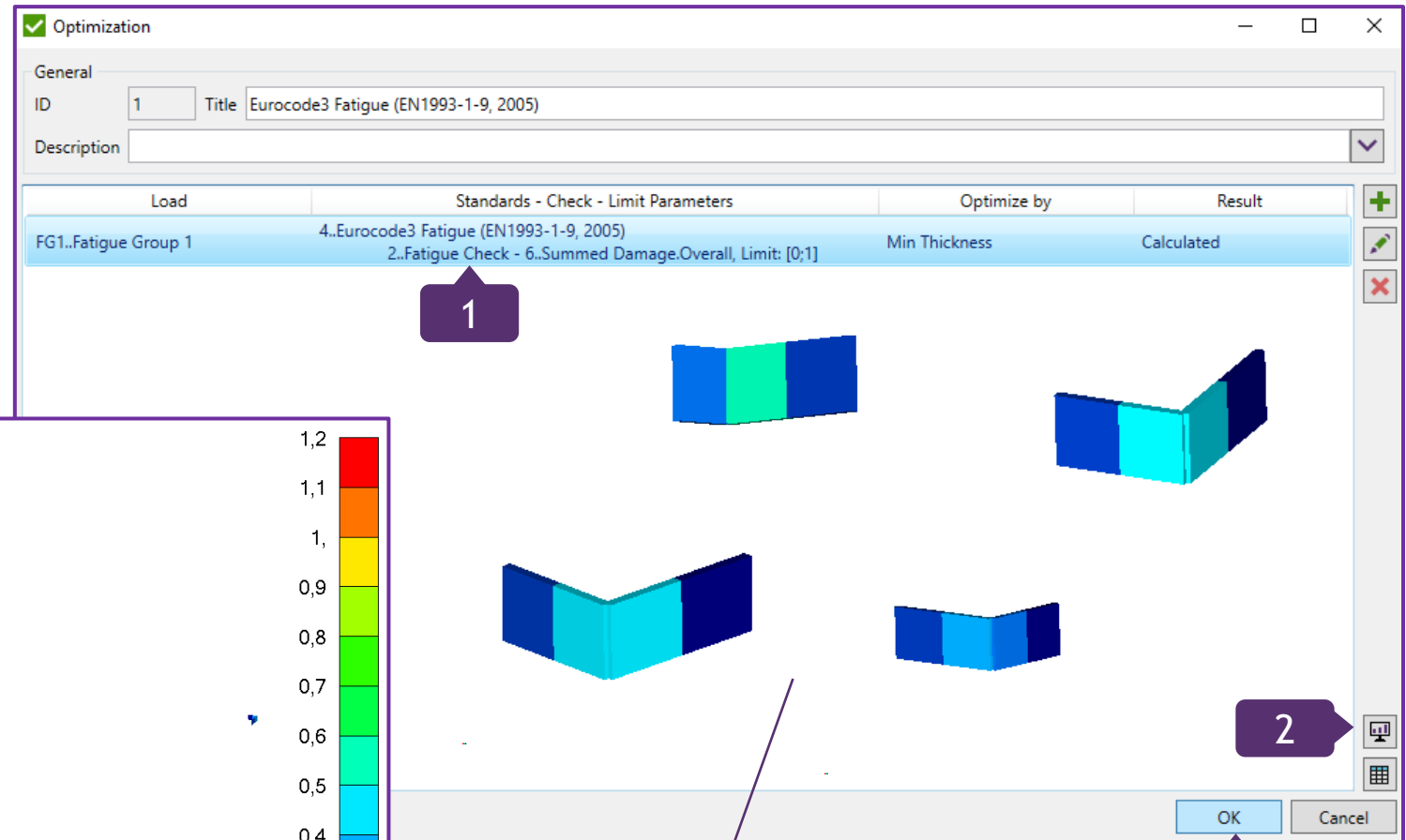
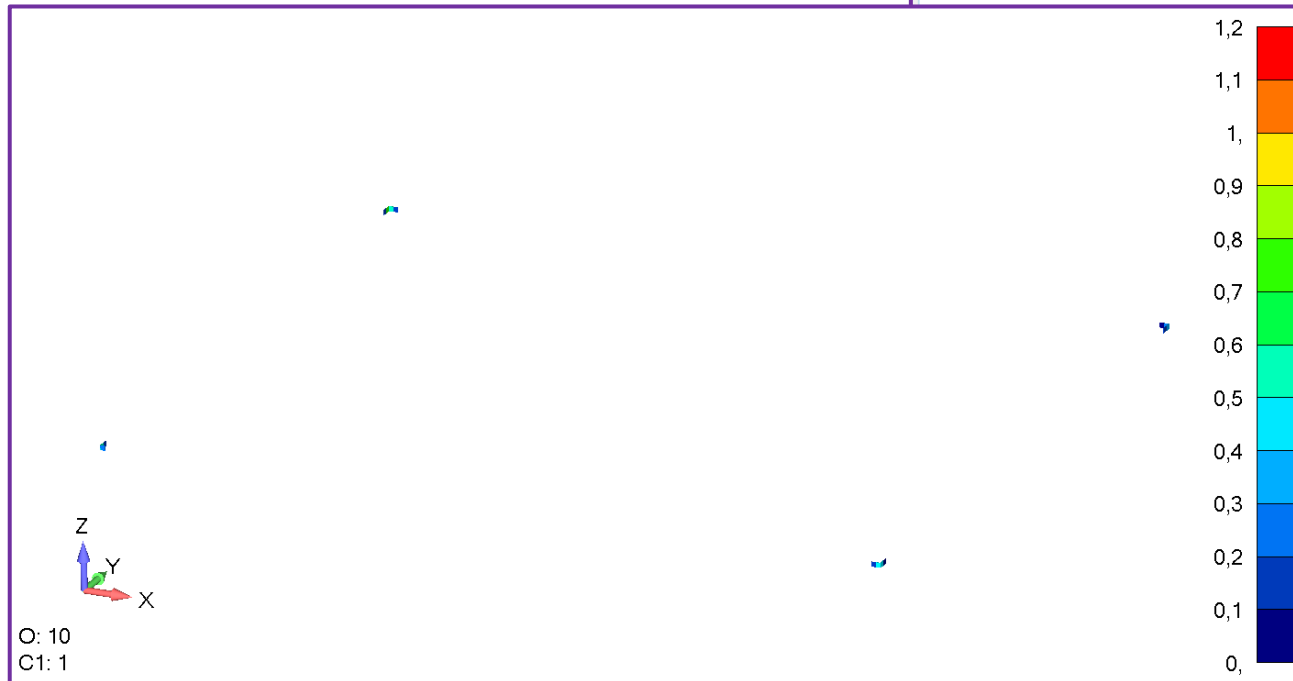
# Plot Optimization Results

1 Activate the section

2 Press  to plot optimal result on the model

3 Press *OK*

The Result



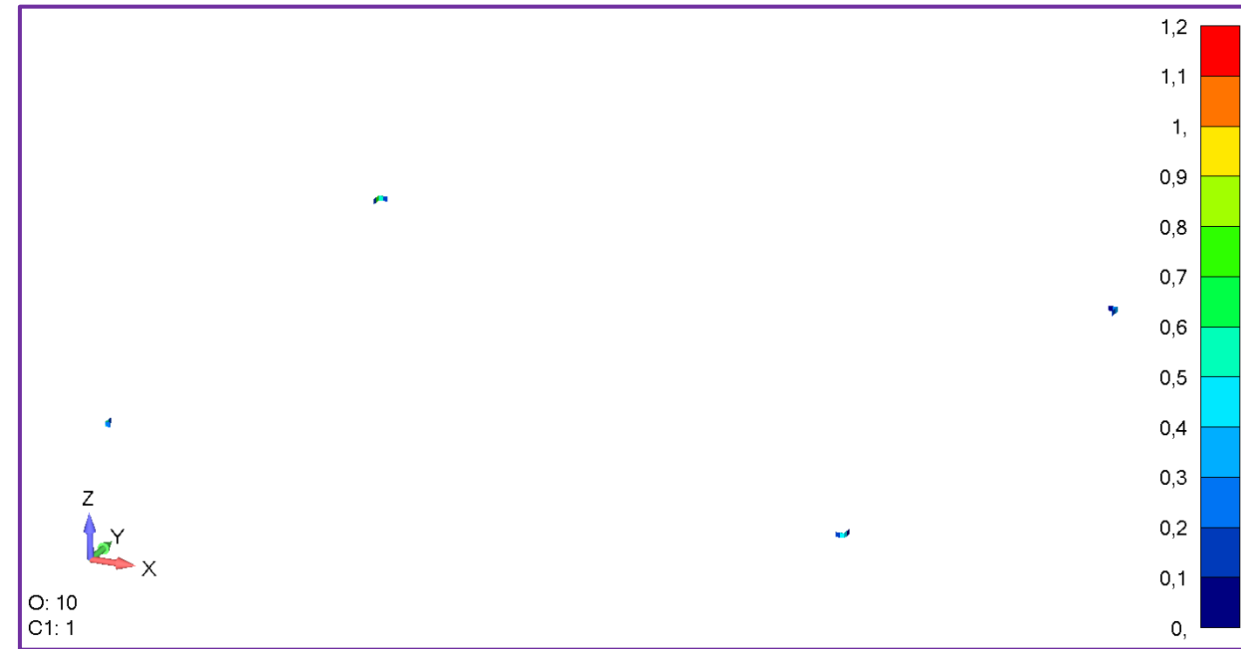
Zoomed-in elements after Optimization

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.



1 Activate the section

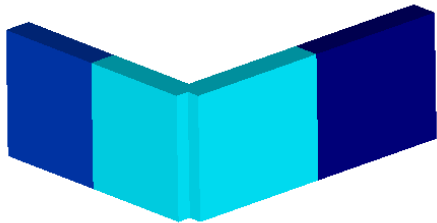
2 Press *Change Model*

3 Press *Yes*

4 Press *OK*

5 Press *OK*

A zoomed-in element



The Result

