



Tutorial

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

Updated on: May 14th, 2024

Tested with: SDC Verifier 2023 R2

Simcenter3D 2306

- 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;
- 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/optimization-tool-help-sdc-for-simcenter-3d)

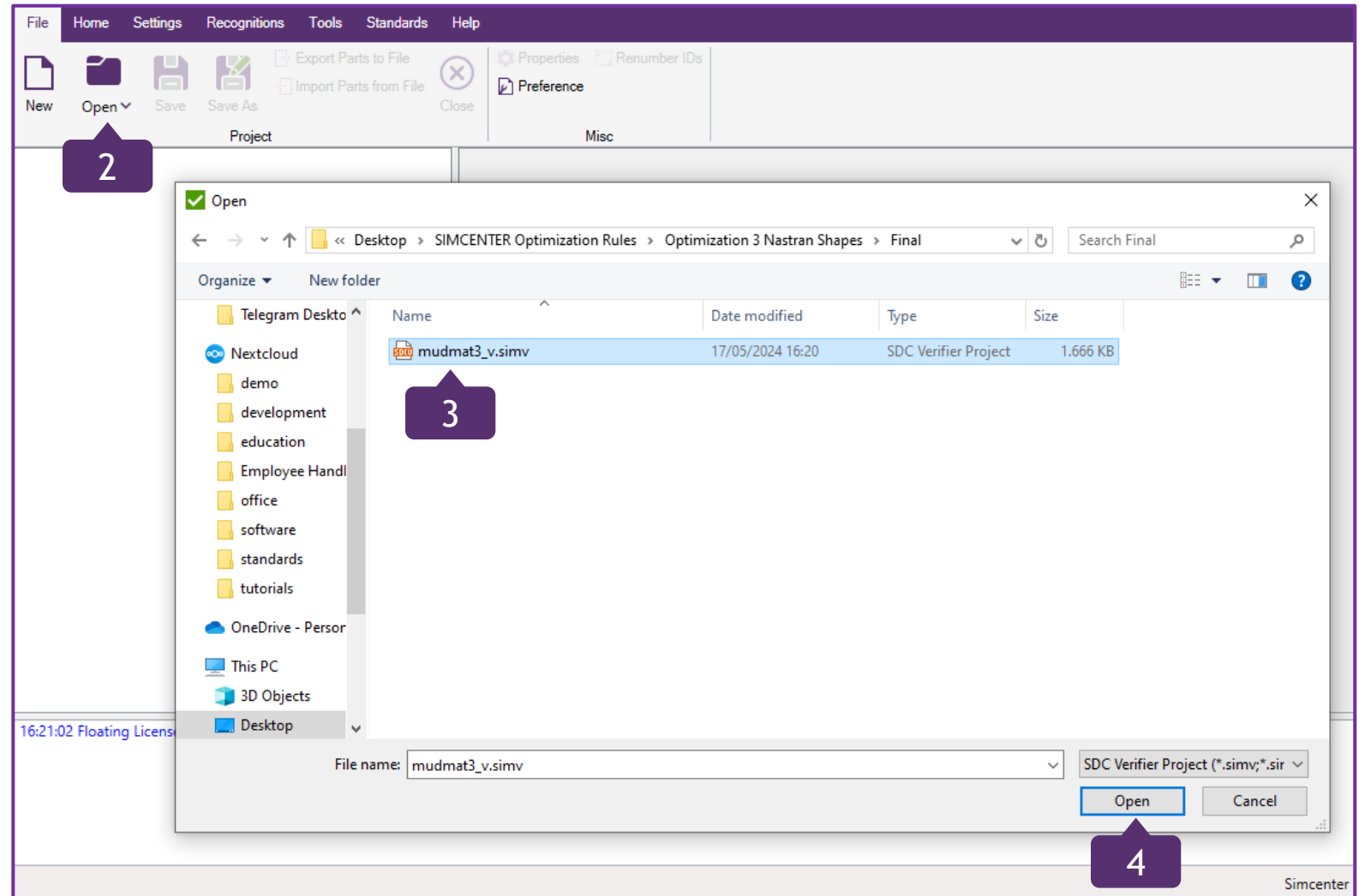
Open the Starter Model

1 Launch SDC Verifier for Simcenter 3D

2 In *File* section, press *Open*

3 Select a project *mudmat3_v.simv*

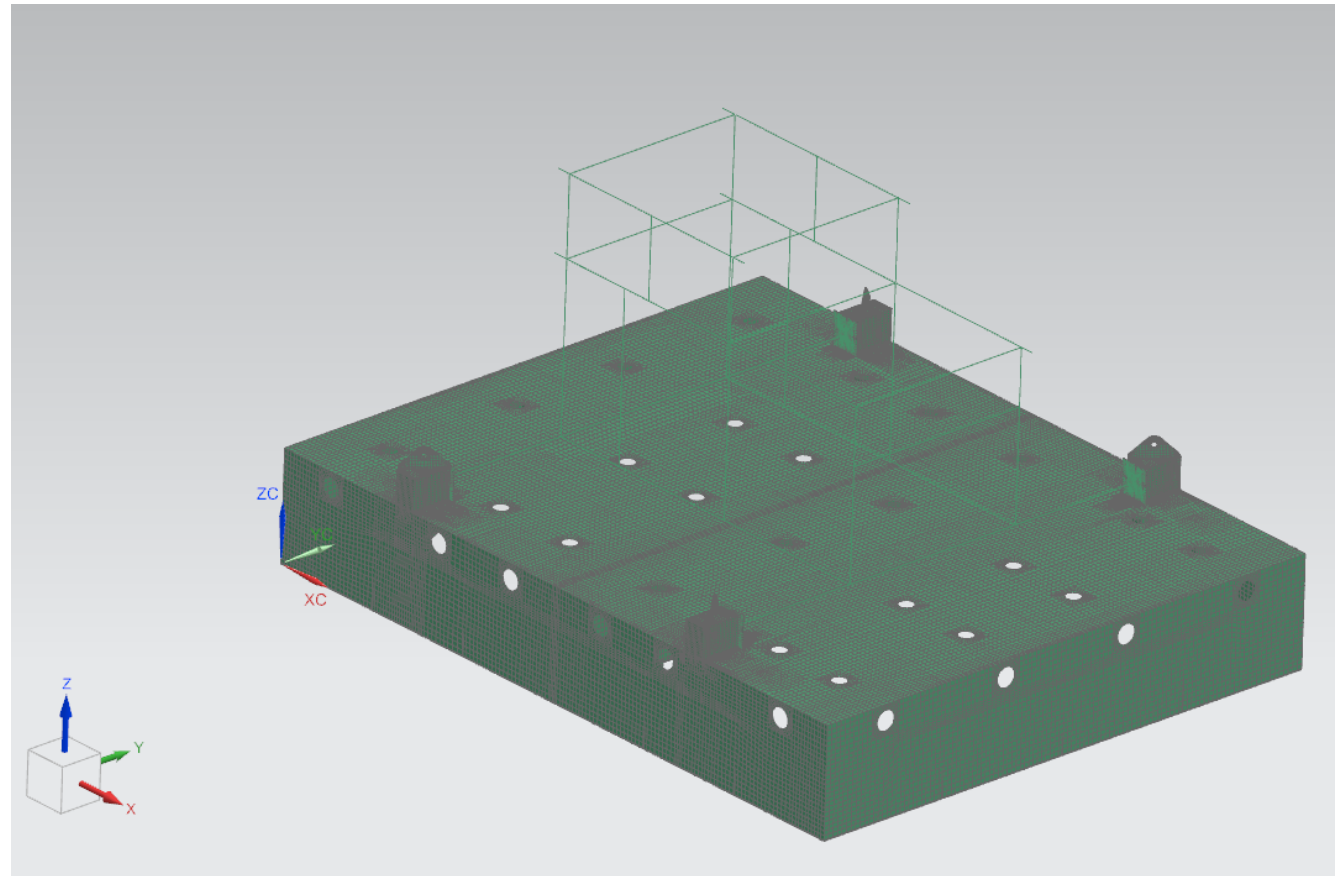
4 Press *Open*



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

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)
 - ▶ 1..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/>

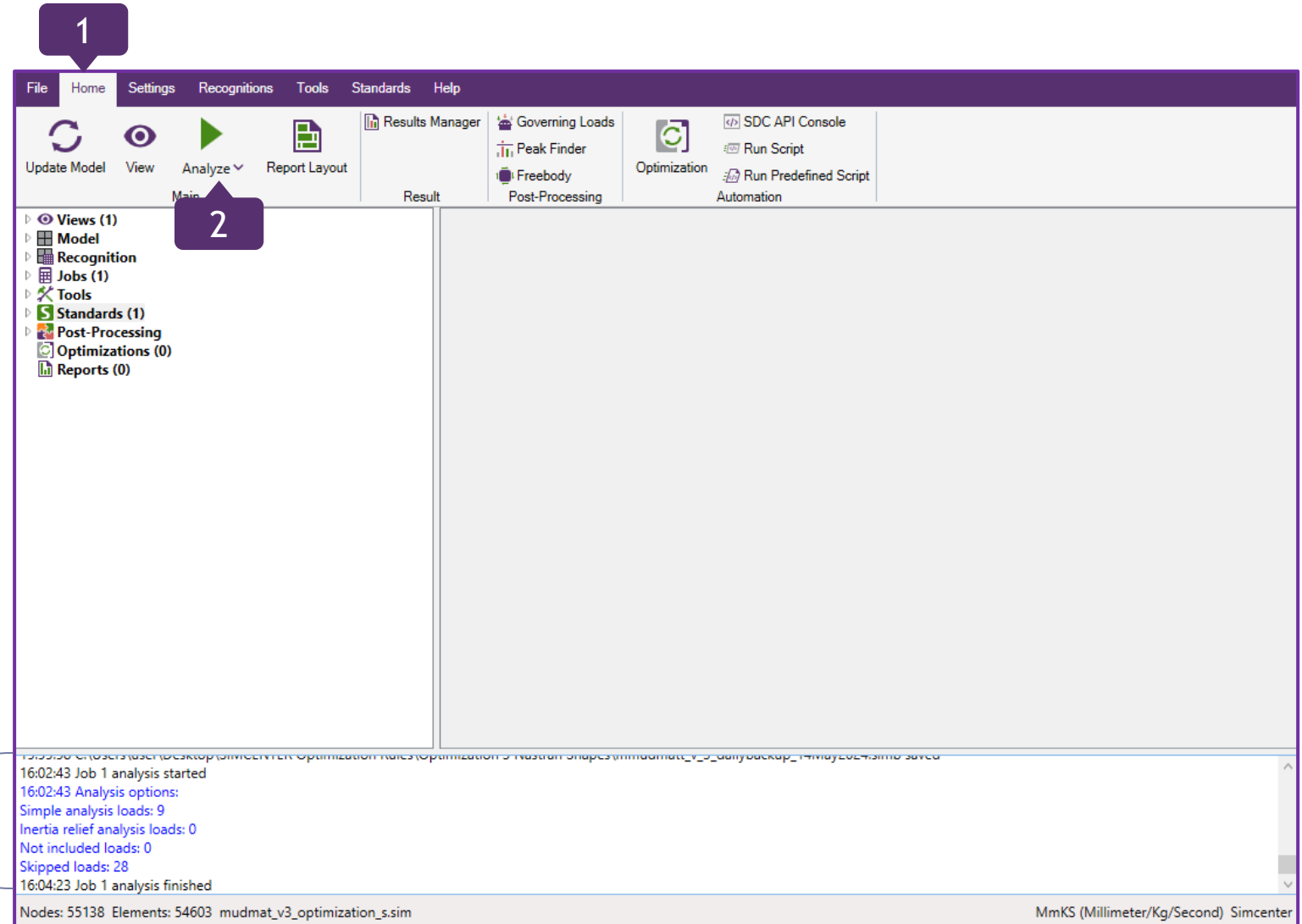
Run Analysis

1

Go to *Home* section on the Ribbon

2

Press  on the toolbar to analyze Job



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

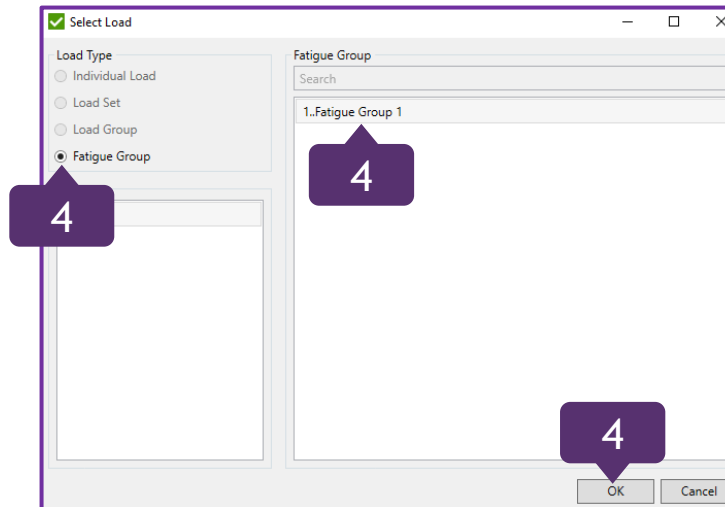
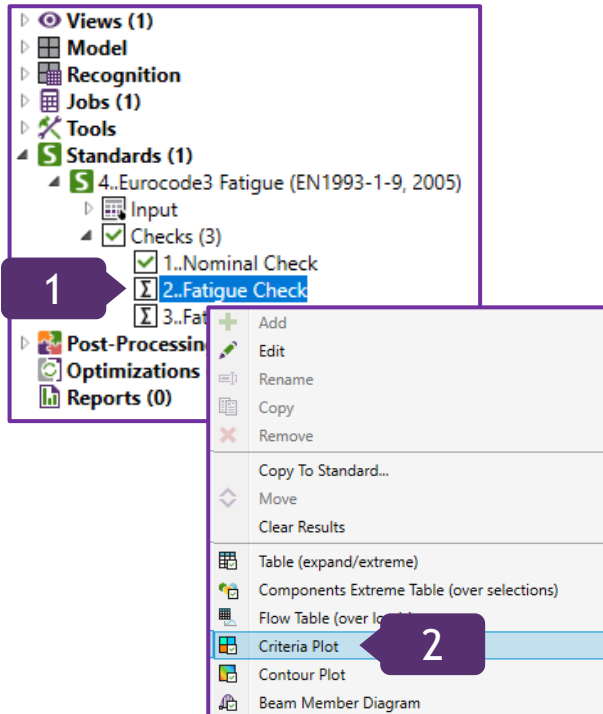
Parameter: *Summed Damage*; Direction: *Overall*

6

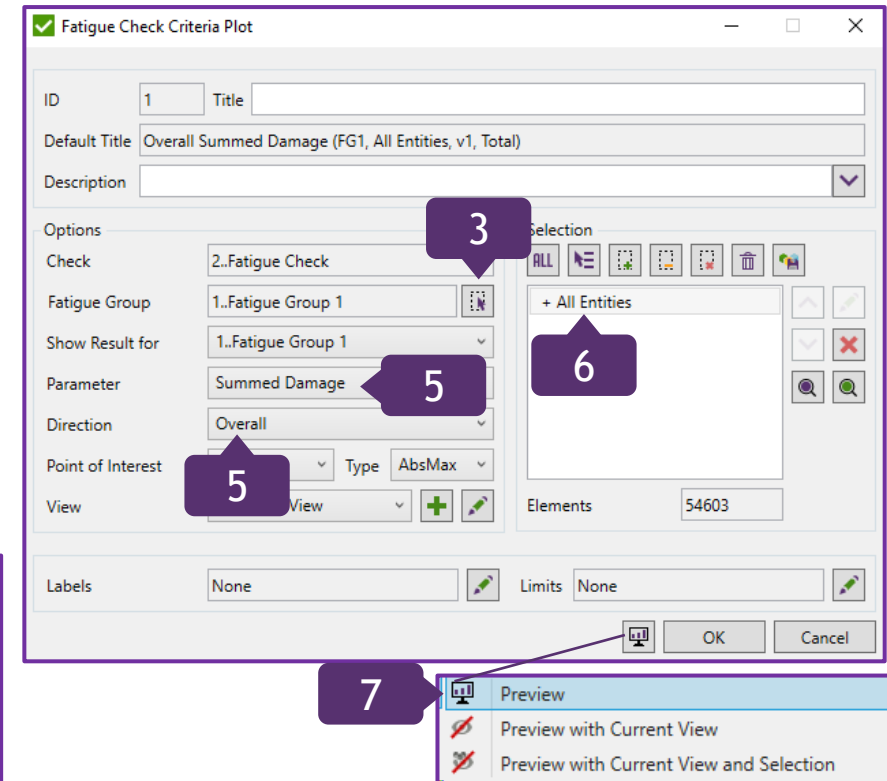
Selection: *All Entities*

7

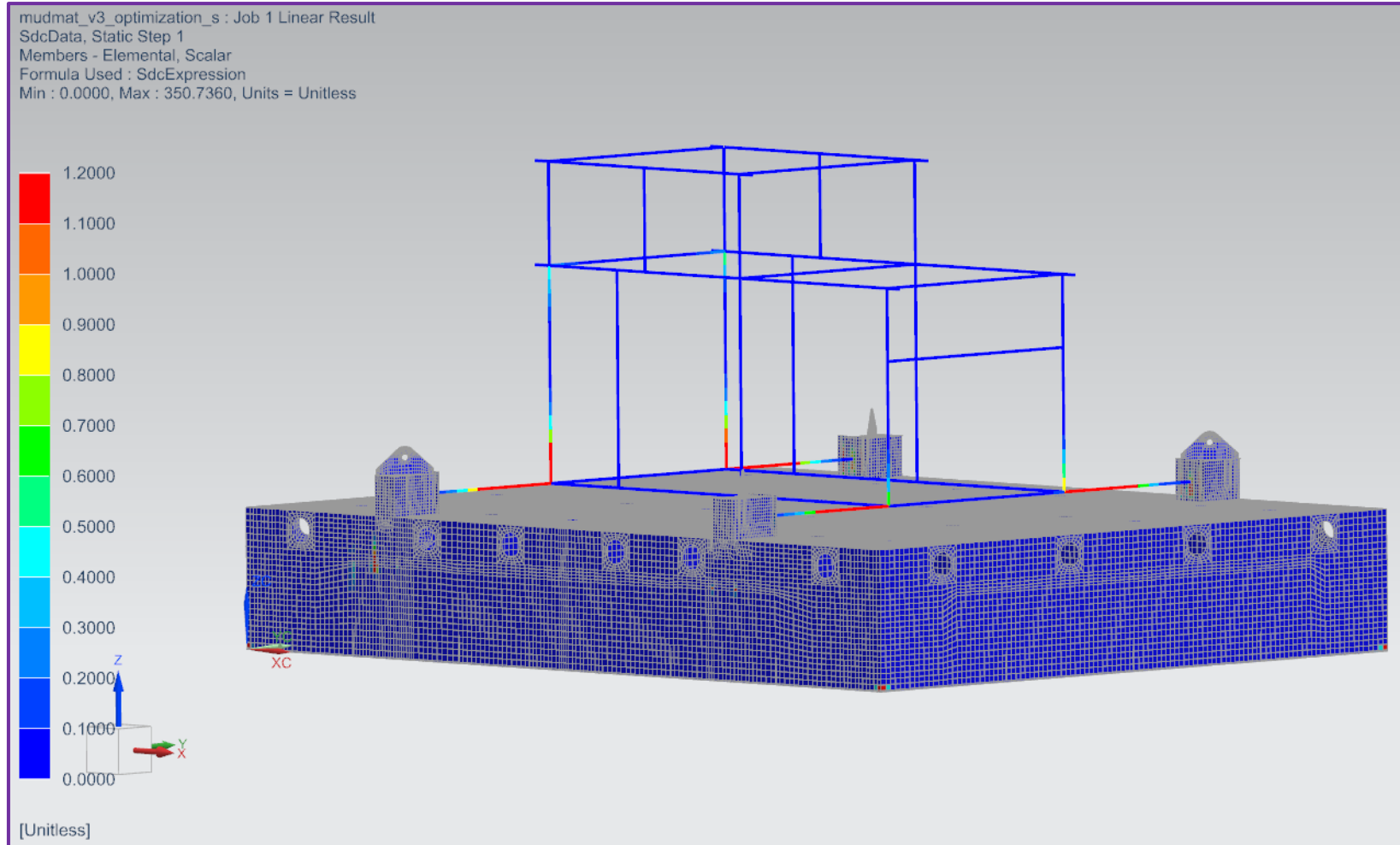
Press  and then *Preview*



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 Simcenter 3D 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



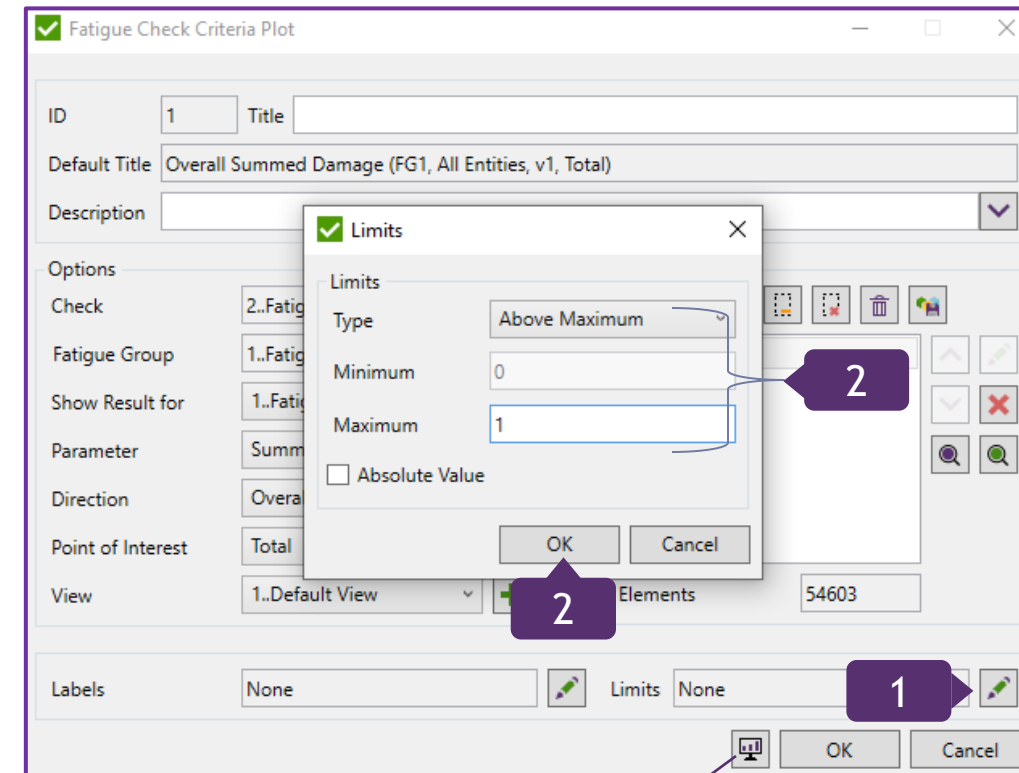
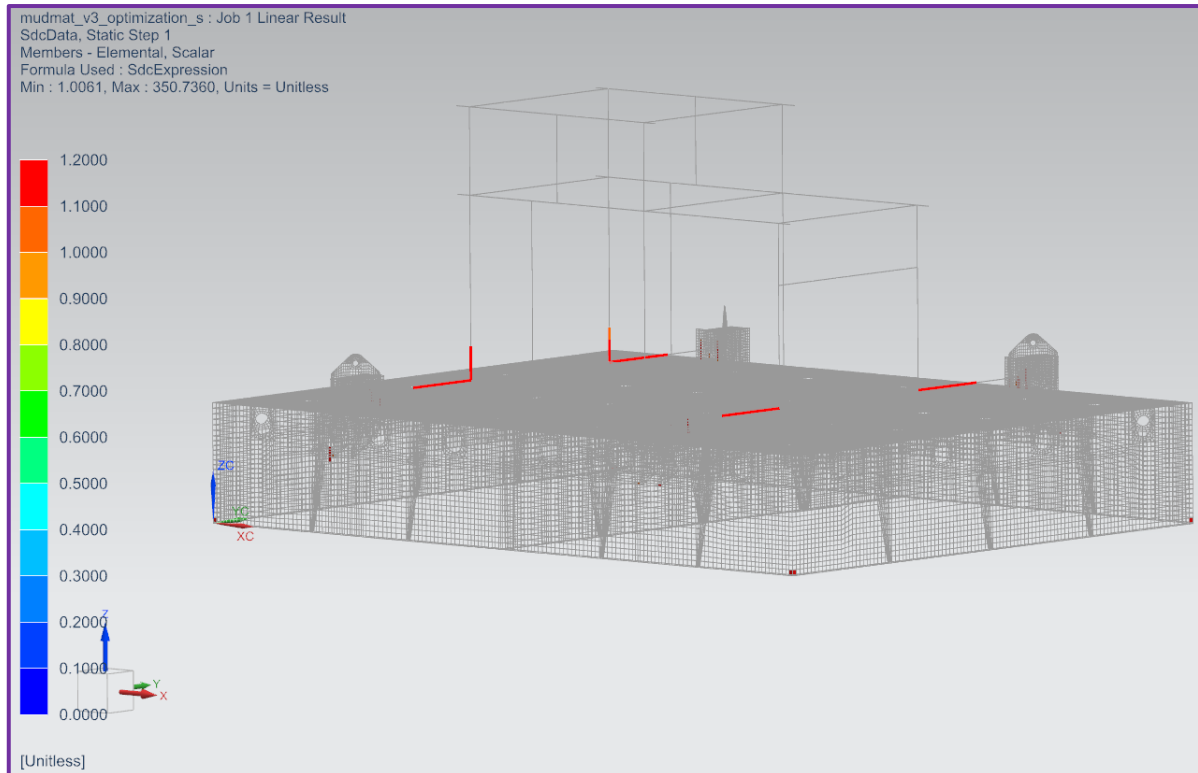
2

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

1 Select + *All Entities* and press  to remove them

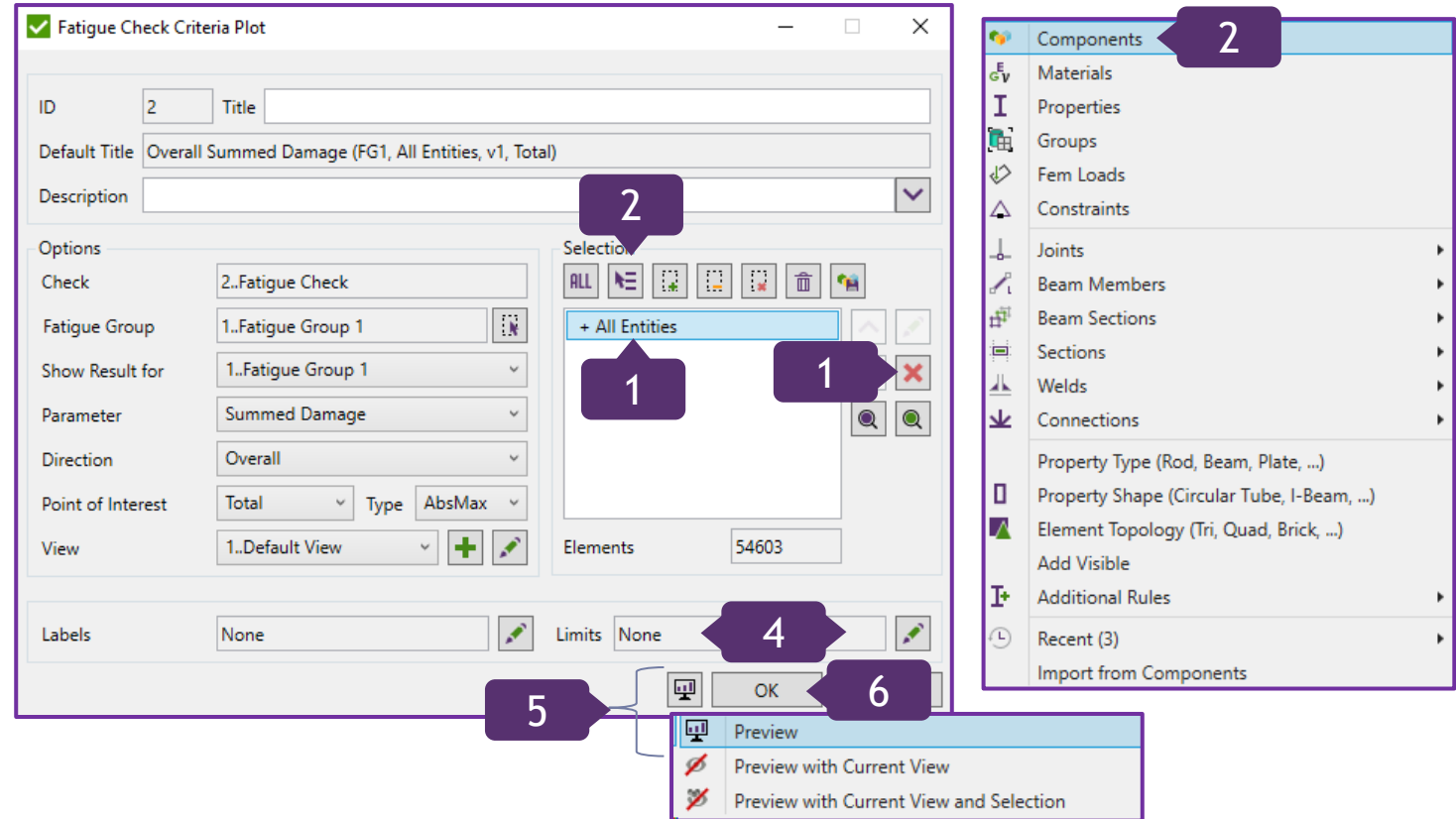
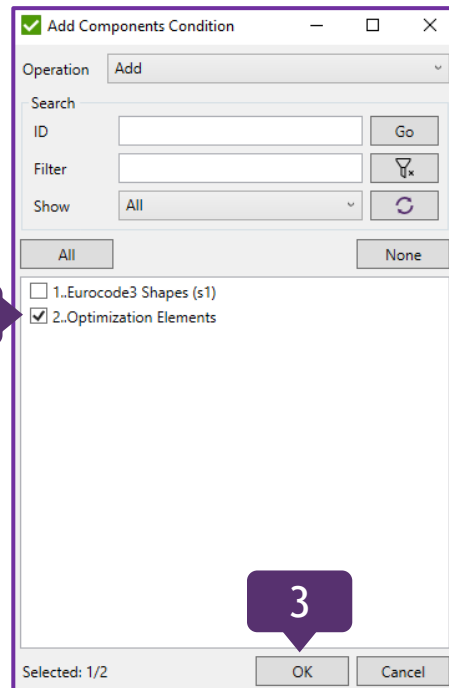
2 Press  to add Condition;
Select *Components*

3 Select 2..*Optimization Elements*;
Press *OK*

4 Limits: *None*

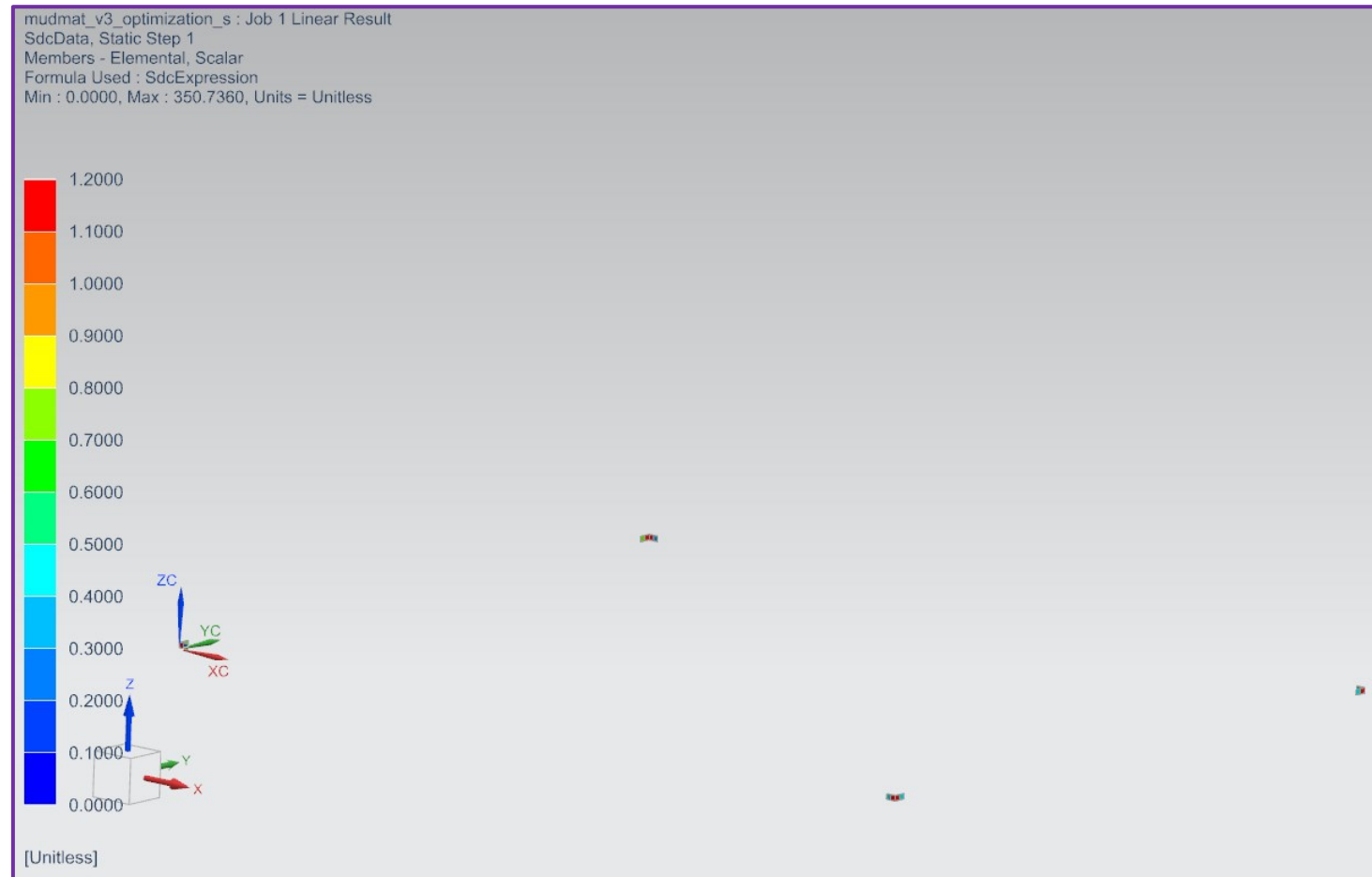
5 Press  and
then *Preview*

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;

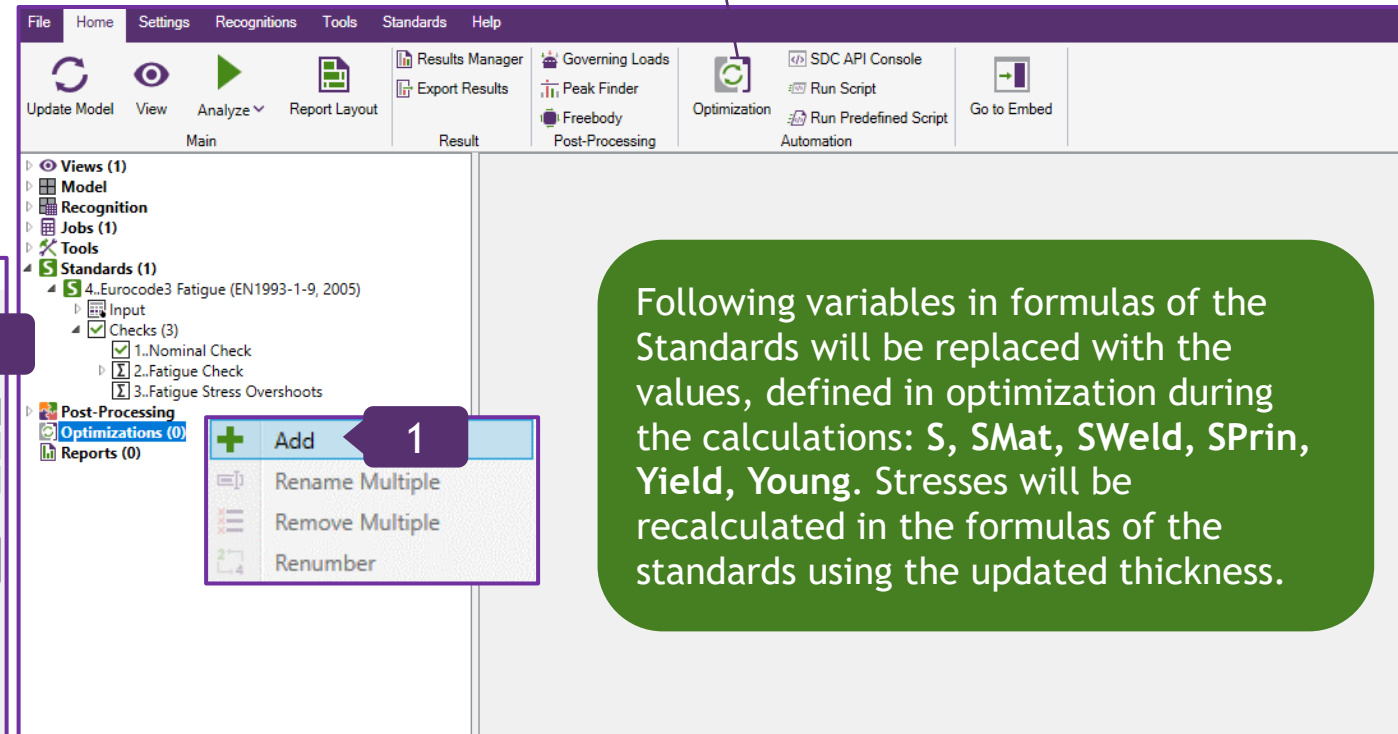
Weld Strength Rule - weld type and dimensions (leg sizes, throat thickness etc.) can be optimized.

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.



Following variables in formulas of the Standards will be replaced with the values, defined in optimization during the calculations: S, SMat, SWeld, SPrin, Yield, Young. Stresses will be recalculated in the formulas of the standards using the updated thickness.

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.

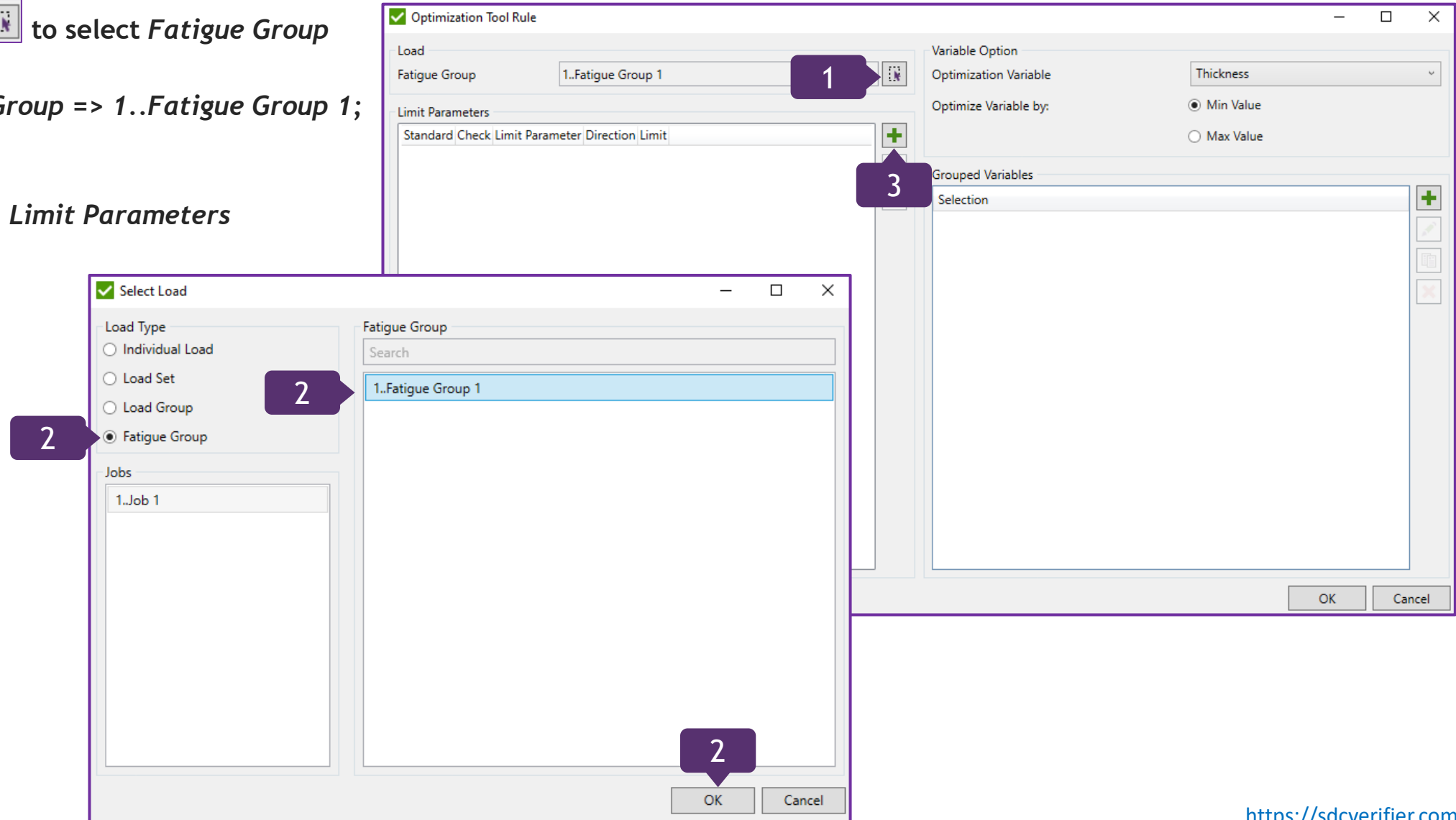
Note: If there are multiple Standards, calculated in the SDC Verifier Project, all of them will be listed in Select Limit Parameters.

Optimization Tool Rule. DNV Buckling Strength of Plated Structures

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

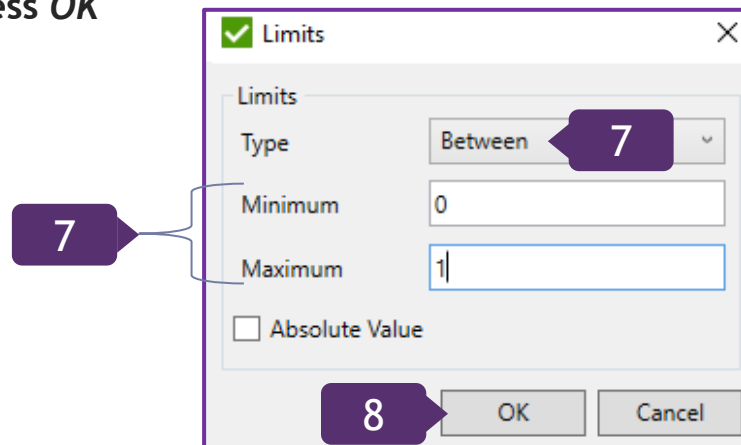
Type: *Between*;
Minimum: *0* and Maximum: *1*;

8

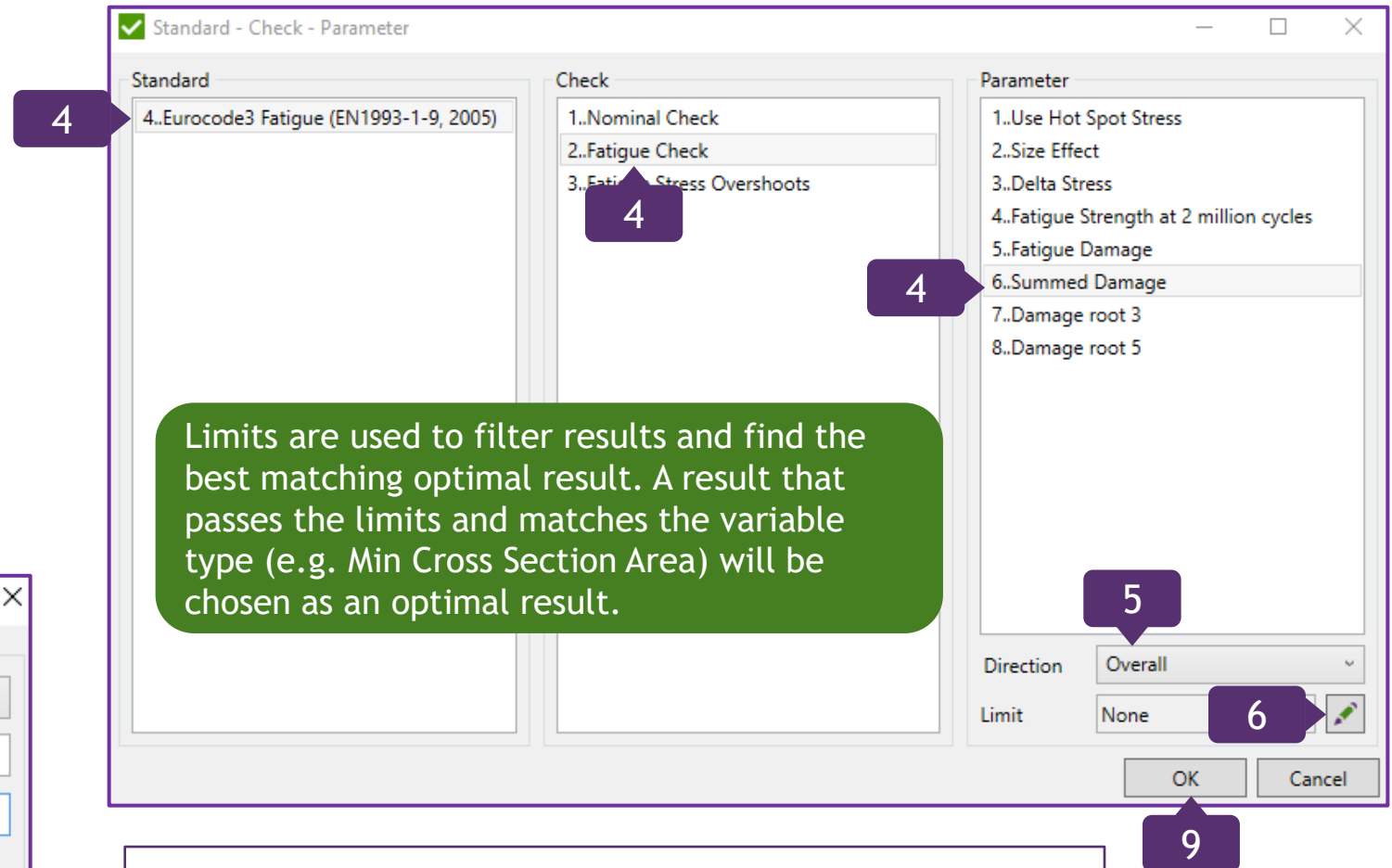
Press *OK*

9

Press *OK*



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



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

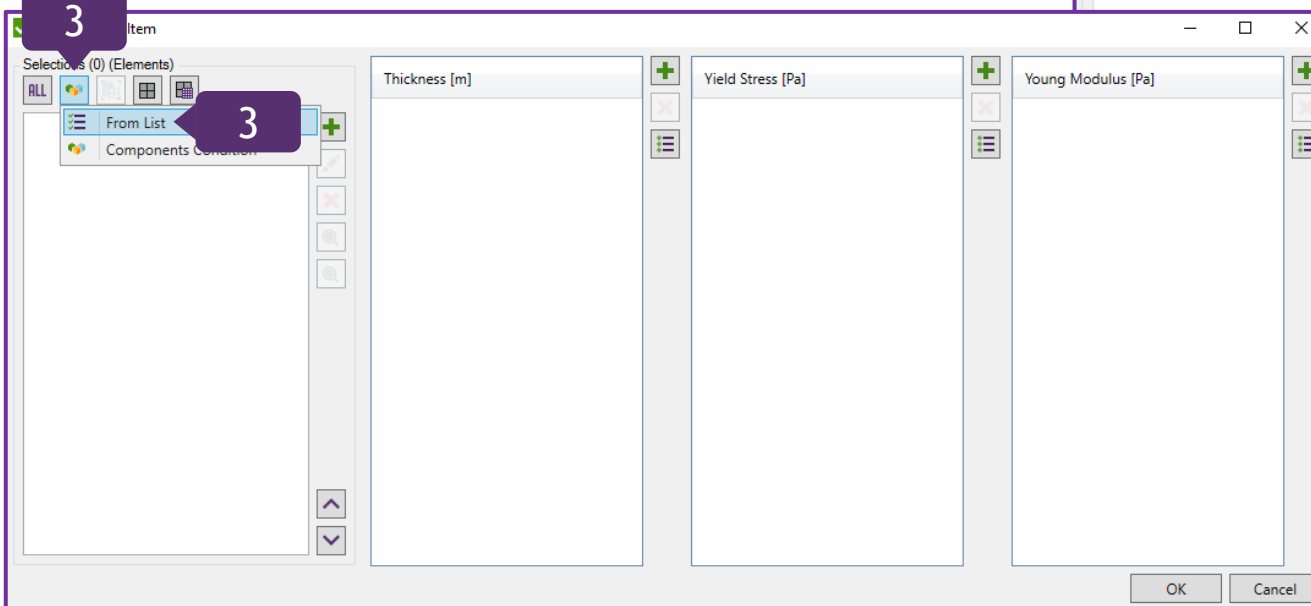
3

Press  and select *From List*

4

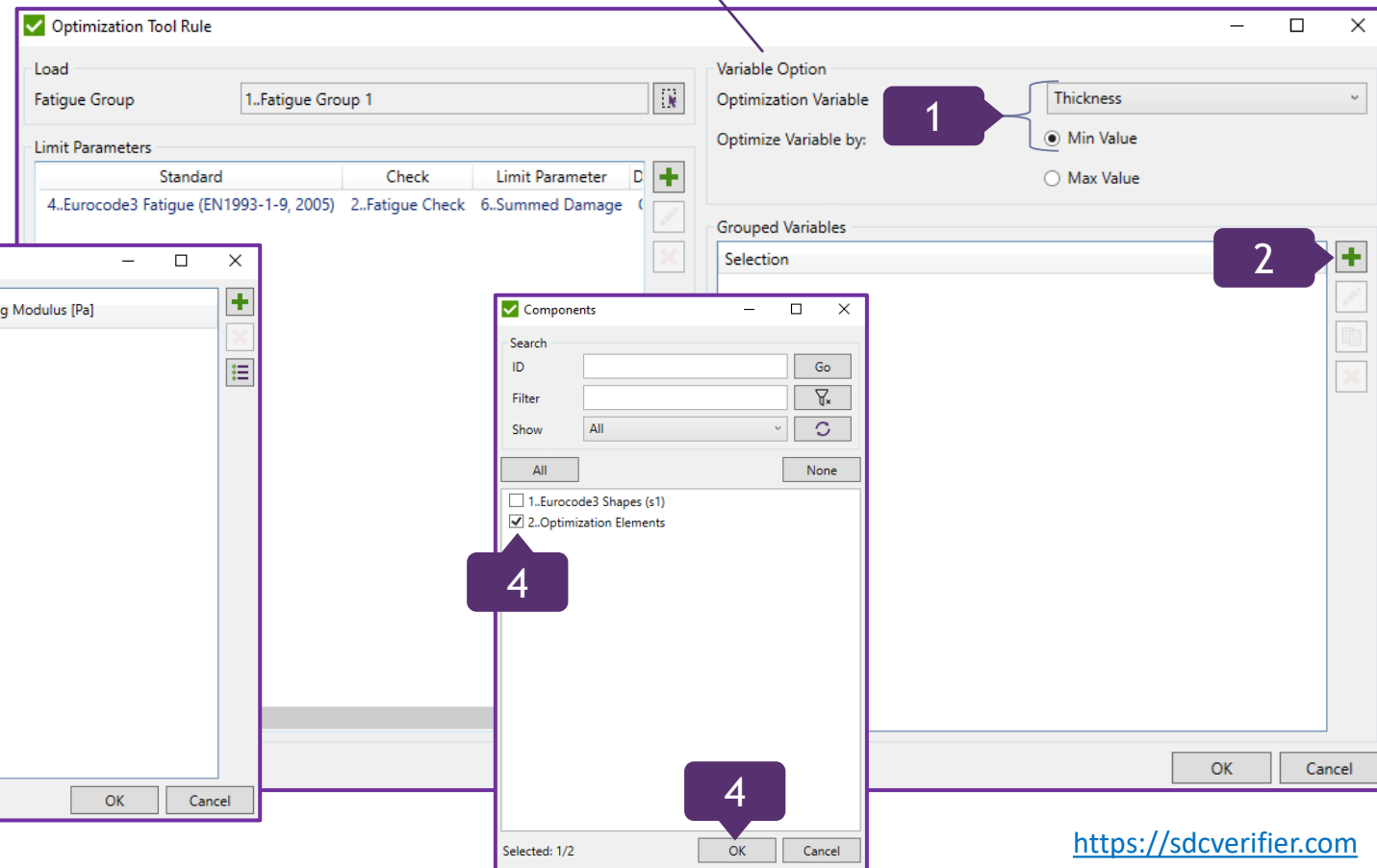
Select 2..Optimization Elements;
Press OK

3



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.



Add Multiple Item (Continuation)

5

Press  to define Thickness [mm]:
8;10;15


6

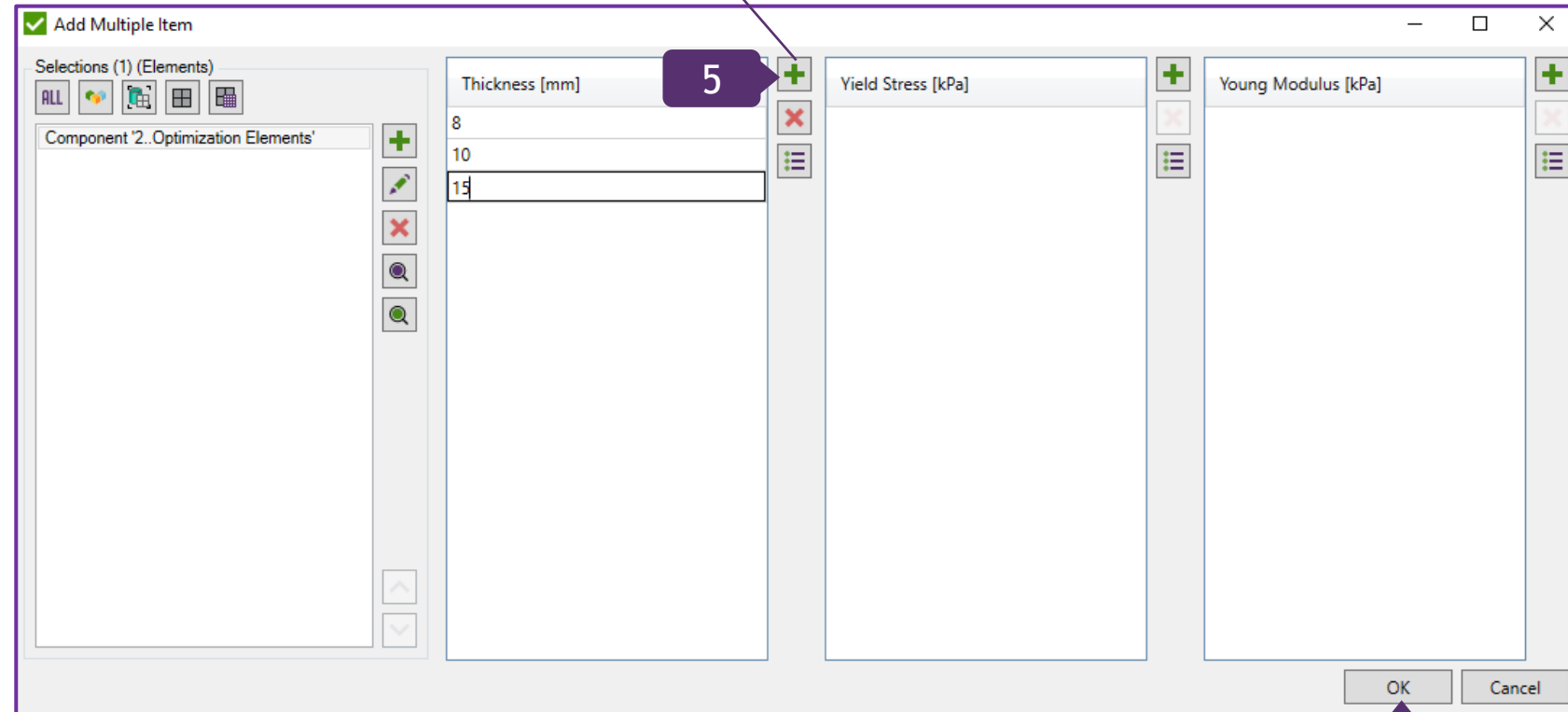
Press **OK**

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 [mm], 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.

6

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. The 'Optimization' window has a 'General' tab with ID 1 and Title 'Eurocode3 Fatigue (EN1993-1-9, 2005)'. It contains a table with columns: Load, Standards - Check - Limit Parameters, Optimize by, and Result. The table has one row: FG1..Fatigue Group 1, 4..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check - 6..Summed Damage.Overall, Limit: [0;1], Min Thickness. A callout '2' points to the 'Standards - Check - Limit Parameters' column. A callout '3' points to the 'Calculate' button at the bottom left. A callout '4' points to the 'OK' button in a small 'SDC Verifier' dialog box that says '1 of 1 rules have result'. The 'Optimization Tool Rule' window has a 'Load' section with 'Fatigue Group' set to '1..Fatigue Group 1'. It has a 'Limit Parameters' table with columns: Standard, Check, Limit Parameter. The table has one row: 1..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check 6..Summed Damage. A callout '1' points to the 'OK' button at the bottom right. A callout '1' also points to the 'Grouped Variables' section in the 'Optimization Tool Rule' window, which lists 'Component '2..Optimization Elements'' with 'Thickness=8.00', 'Thickness=10.00', and 'Thickness=15.00'. A text box at the bottom right says 'Grouped Variables that are calculated.'

Optimization Tool Rule

Load

Fatigue Group 1..Fatigue Group 1

Limit Parameters

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

Variable Option

Optimization Variable Thickness

Optimize Variable by: ☒ Min Value ☐ Max Value

Grouped Variables

Selection

Component '2..Optimization Elements'

Thickness=8.00
Thickness=10.00
Thickness=15.00

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	

SDC Verifier

1 of 1 rules have result

OK

Calculate Change Model OK Cancel

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 [mm]	1..Eurocode3 Fatigue (EN1993-1-9, 2005) 2..Fatigue Check 6..Summed Damage.Overall
Component '2..Optimization Elements'	10.00	0.73

This is the Optimal Result, which will be used for changing the Plate 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 [kPa]	Young Modulus [kPa]	Thickness [mm]	1..Eurocode3 Fatigue (EN19 2..Fatigue Check 6..Summed Damage.Overall
Component '2..Optimization	Original Model (240000.00)	Original Model (210000000.0)	Original Model (6.00)	4.62
Component '2..Optimization			8.00	1.61
Component '2..Optimization			10.00	0.73
Component '2..Optimization			15.00	0.19

The results for all variables.

2,3

Optimal Result

All Result

2

2 Close

1

Activate the section

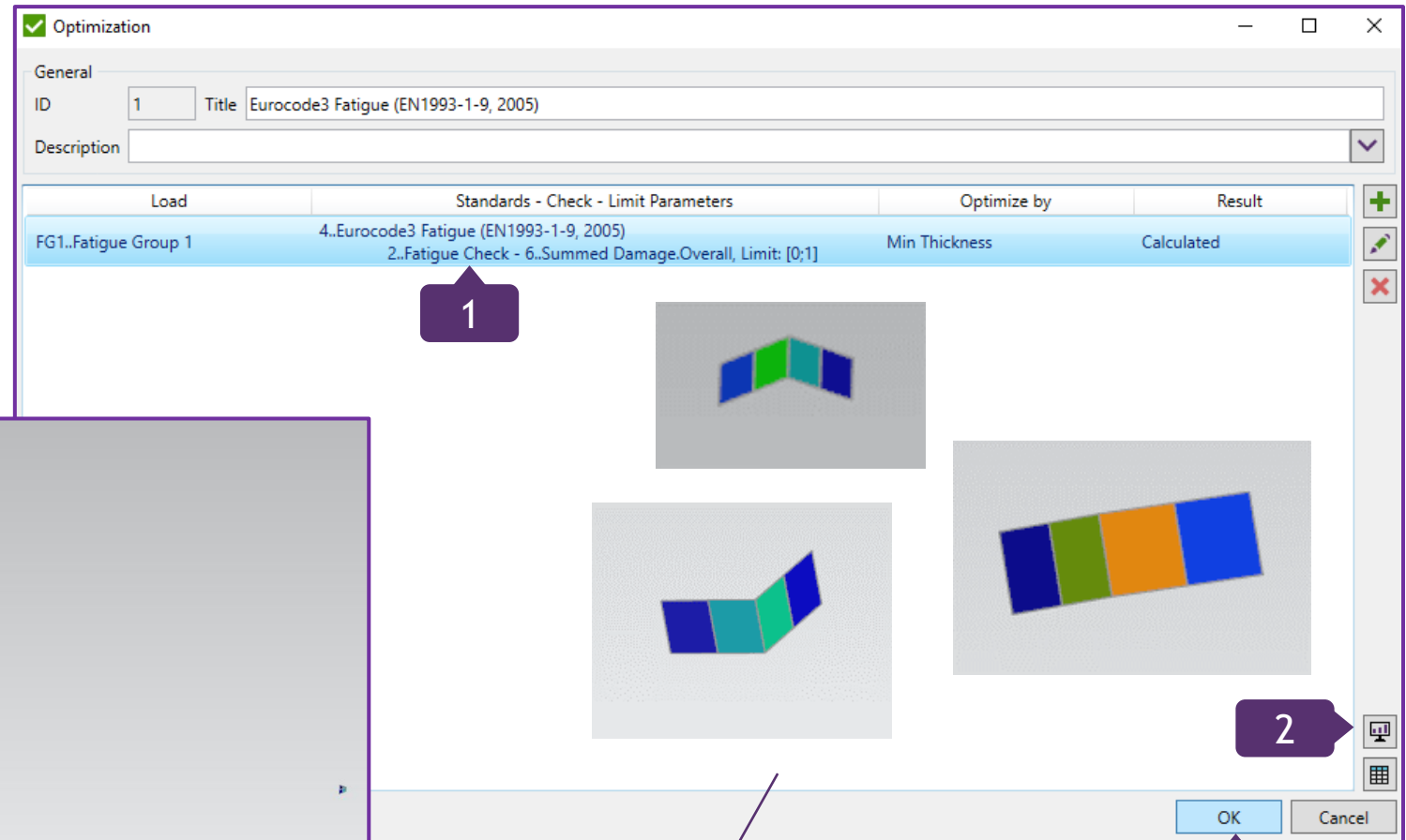
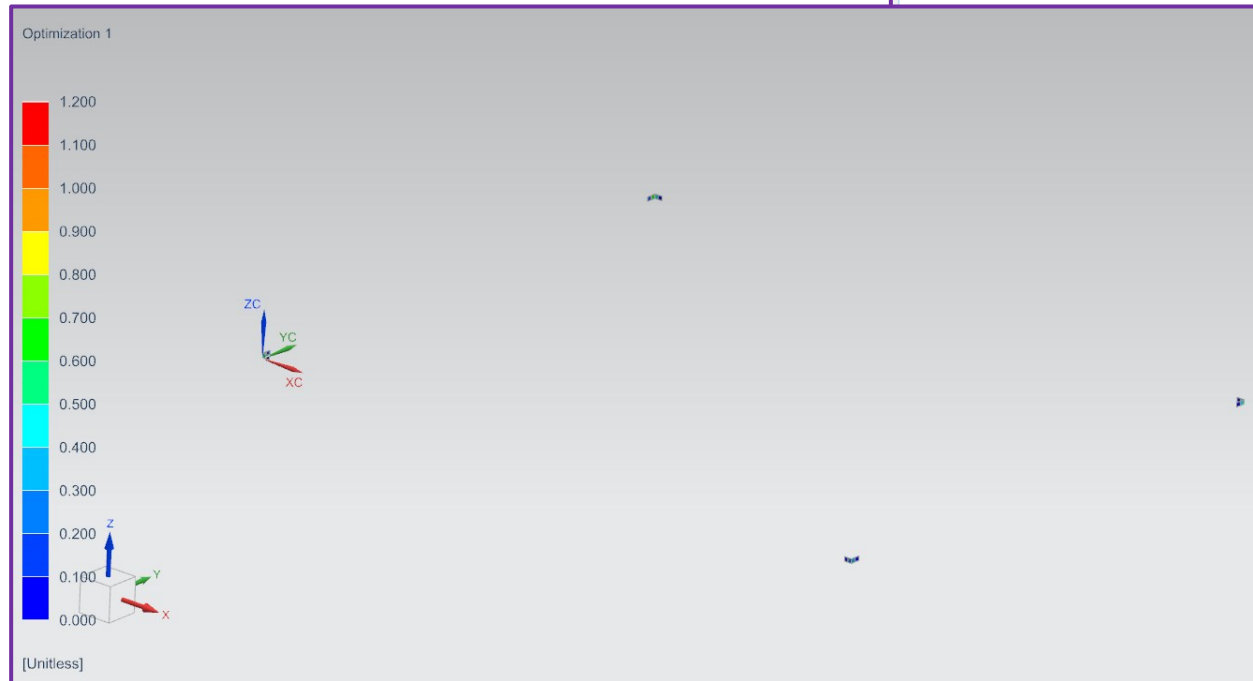
2

Press  to plot optimal result on the model

3

Press *OK*

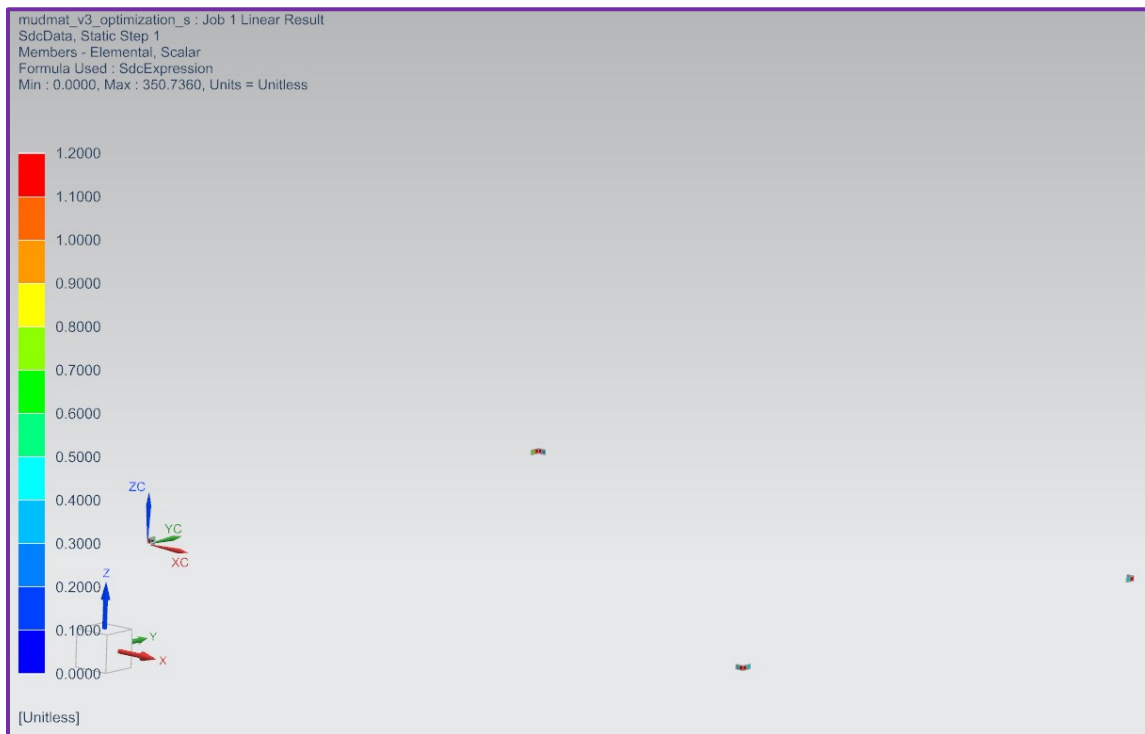
The Result



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.

