



Tutorial

# Report Designer

Updated on: February 20th 2024

Tested with: SDC Verifier 2023 R2

Simcenter 3D 2306

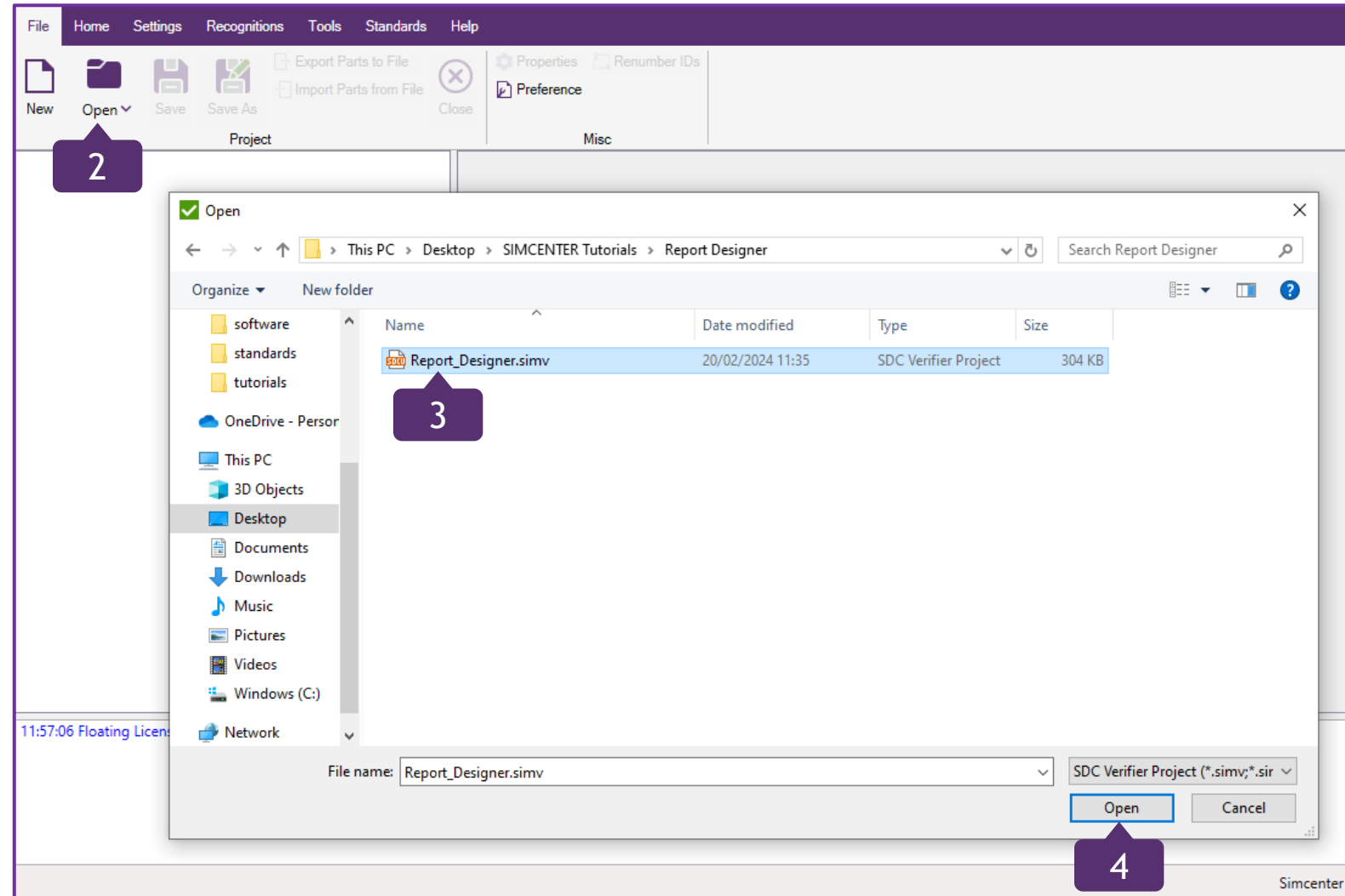
Report Designer gives a possibility to completely control the structure of your Report, easily preview, and modify it.

This tutorial demonstrates how to build Reports, using the Report Designer:

- Open a predefined project;
- Model Setup Report (First Page, Preface, Materials, Properties, Fem Loads and Constraints);
- Result Report (Content items, Predefined Tables, Add Plots and Tables);
- Number Format, Legend Settings;
- Tables and Plots for Static Stress Check.

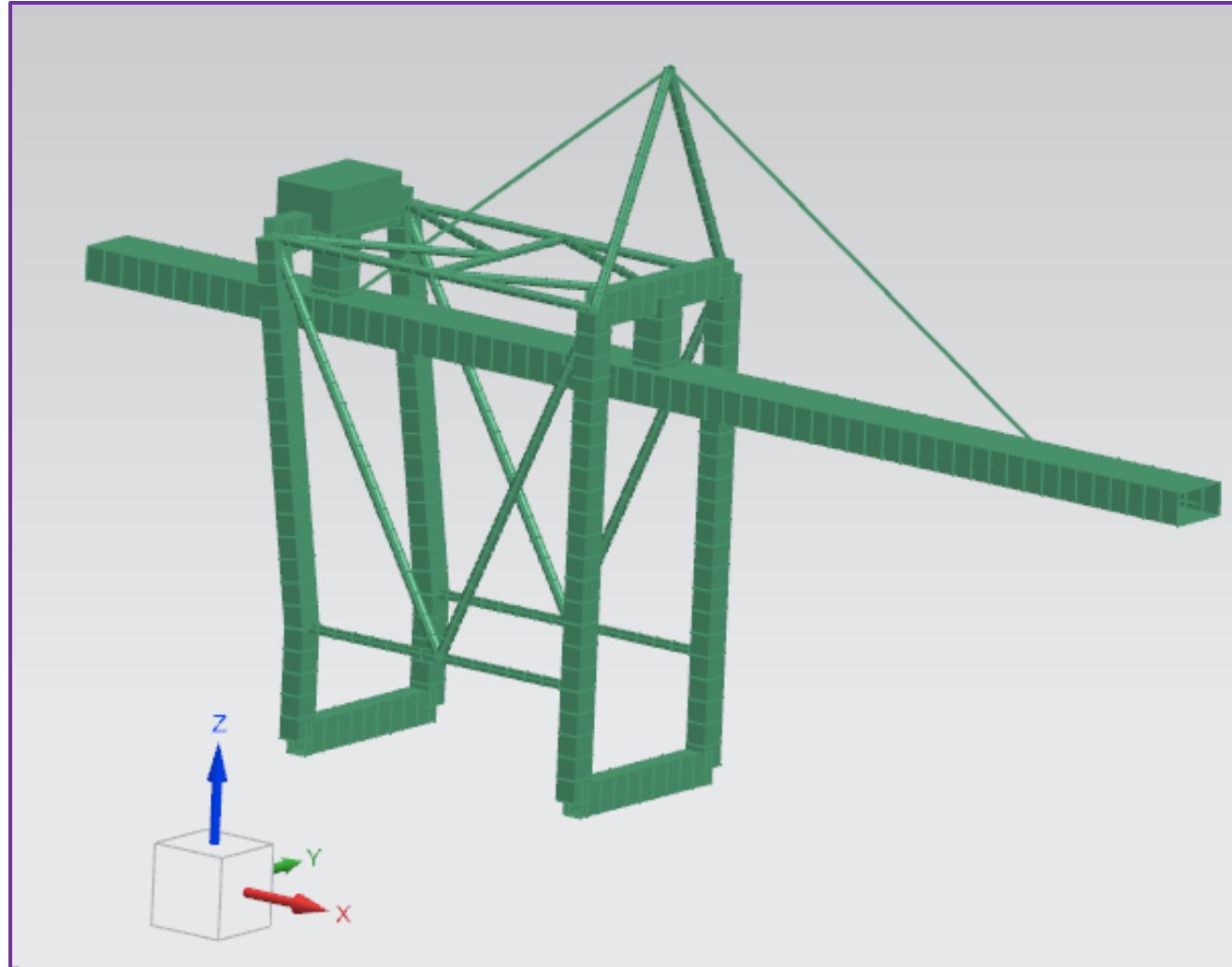
# Open the Starter Model

- 1 Launch SDC Verifier for Simcenter 3D
- 2 In *File* section, press *Open*
- 3 Select a project *Report\_Designer.simv*
- 4 Press *Open*



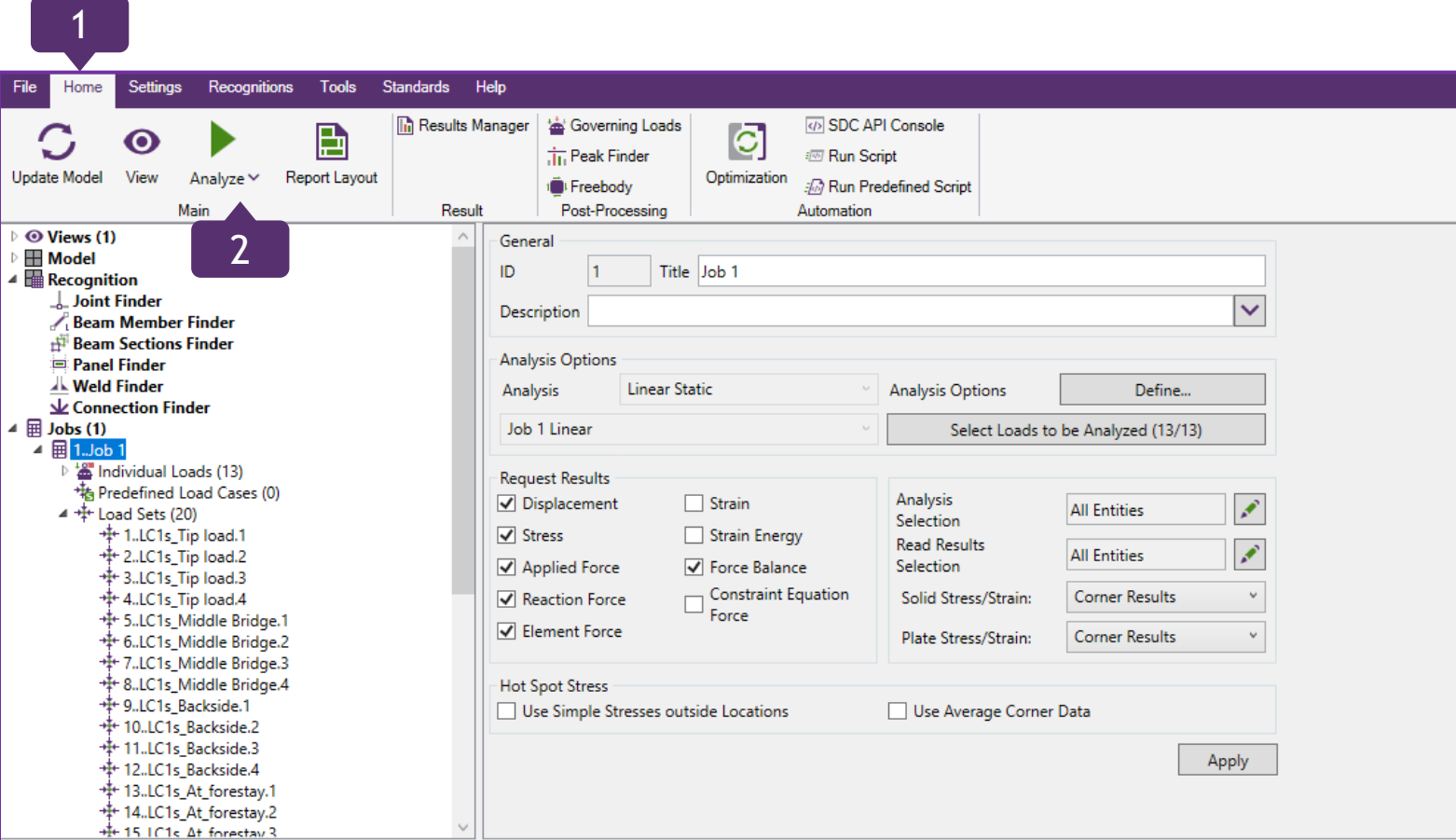
- Views (1)
- Model
- Recognition
- Jobs (1)
  - 1..Job 1
    - Individual Loads (13)
    - Predefined Load Cases (0)
    - Load Sets (20)
      - 1..LC1s\_Tip load.1
      - 2..LC1s\_Tip load.2
      - 3..LC1s\_Tip load.3
      - 4..LC1s\_Tip load.4
      - 5..LC1s\_Middle Bridge.1
      - 6..LC1s\_Middle Bridge.2
      - 7..LC1s\_Middle Bridge.3
      - 8..LC1s\_Middle Bridge.4
      - 9..LC1s\_Backside.1
      - 10..LC1s\_Backside.2
      - 11..LC1s\_Backside.3
      - 12..LC1s\_Backside.4
      - 13..LC1s\_At\_forestay.1
      - 14..LC1s\_At\_forestay.2
      - 15..LC1s\_At\_forestay.3
      - 16..LC1s\_At\_forestay.4
      - 17..LC1s\_at\_hinge\_point.1
      - 18..LC1s\_at\_hinge\_point.2
      - 19..LC1s\_at\_hinge\_point.3
      - 20..LC1s\_at\_hinge\_point.4
    - Load Groups (1)
    - FG Fatigue Groups (0)
    - Tables (0)
    - Plots (0)
- Tools
- Standards (1)
  - 1..Static Check
    - Input
    - Checks (1)
      - 1..Static Stress Check
- Post-Processing
- Optimizations (0)
- Reports (0)

This tutorial presents a predefined project with the following created data: Individual Loads, Load Sets, Load Groups and Static Stress Check. The tutorial focuses on creating Reports.



1 Go to *Home* section on the Ribbon

2 Press  on the toolbar to analyze Job



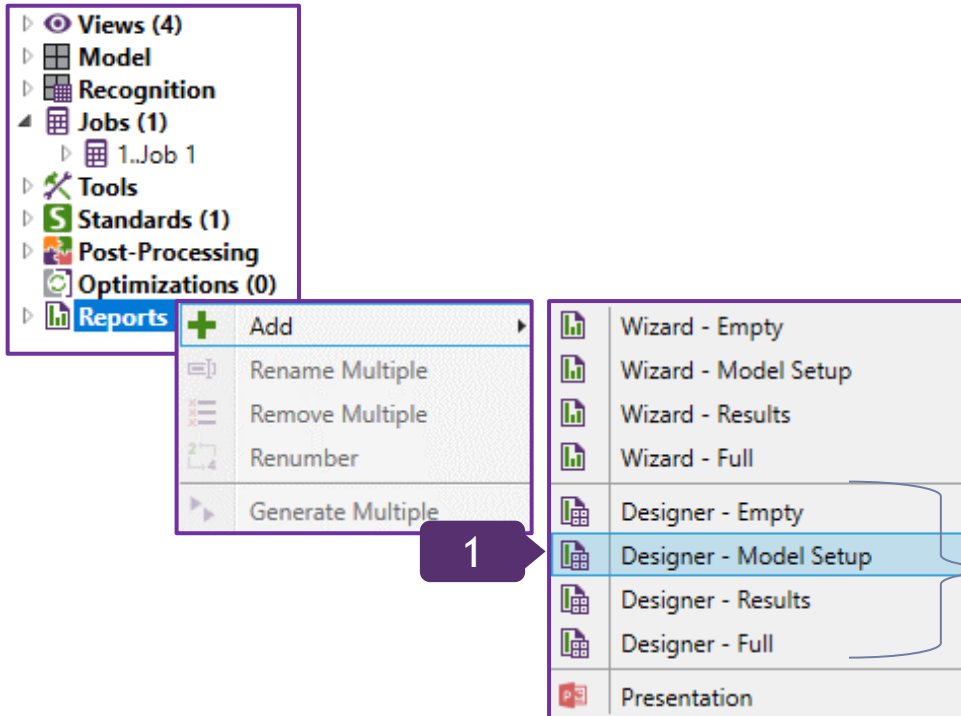
14:59:49 Job 1 analysis started  
14:59:49 Analysis options:  
Simple analysis loads: 13  
Inertia relief analysis loads: 0  
Not included loads: 0  
Skipped loads: 20  
15:00:10 Job 1 analysis finished

Nodes: 404 Elements: 421 C:\Users\user\Desktop\SIMCENTER Tutorials\Report Designer\ReportDesigner\_s.sim MmKS (Millimeter/Kg/Second) Simcenter

Job 1 analysis started and finished.

1

In the *Model Tree*, execute right click on *Reports* => *Add*, and select *Model Setup*



There are 4 templates of reports:

Empty - only first page and preface items are included;

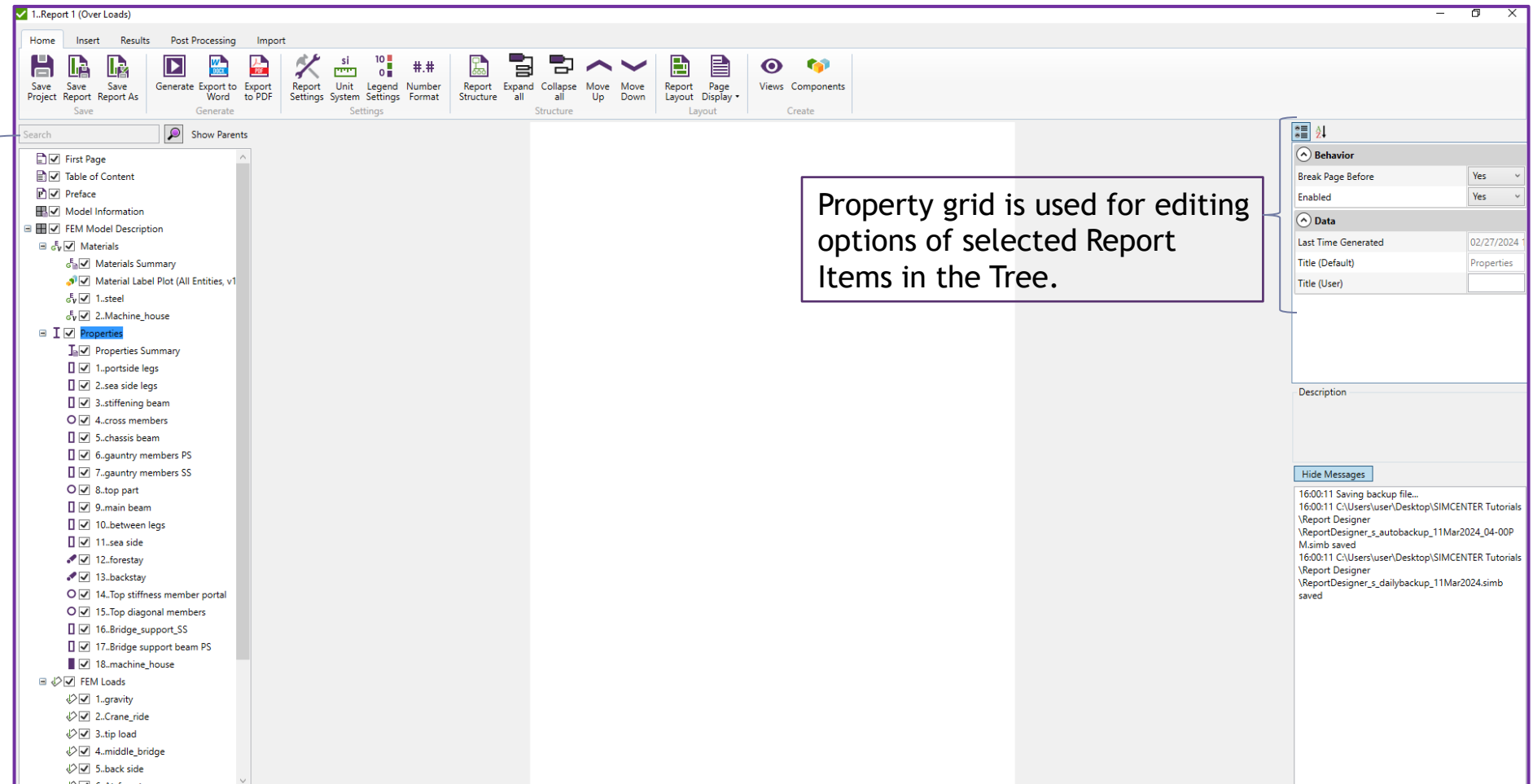
Model Setup - description of model data (materials, properties, components) is included;

Results - for each load extreme displacement tables, stress and displacement plots are included. Predefined tables: sum of reaction forces, stresses/displacements summary tables;

Full - Model Setup + Results + all tables created in Job

# Report Designer Interface (Components)

The Ribbon contains main functions.



1

Execute right click on *First Page*, and select *Edit*

2

Press  and select *Support Engineer* from the library

3

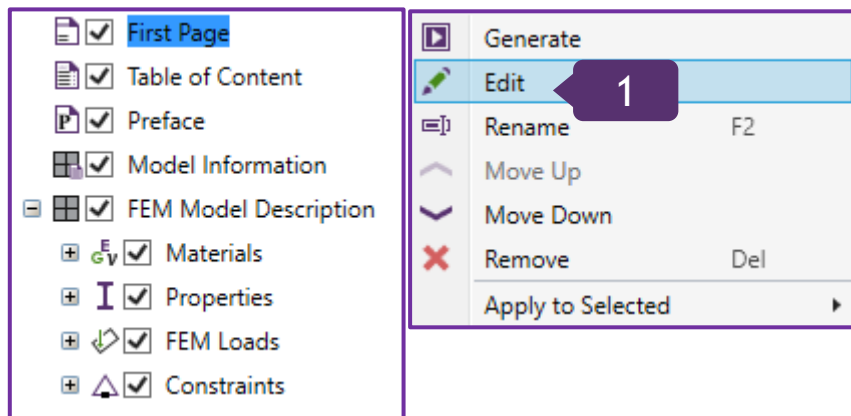
Press  and select *Customer* from the library

4

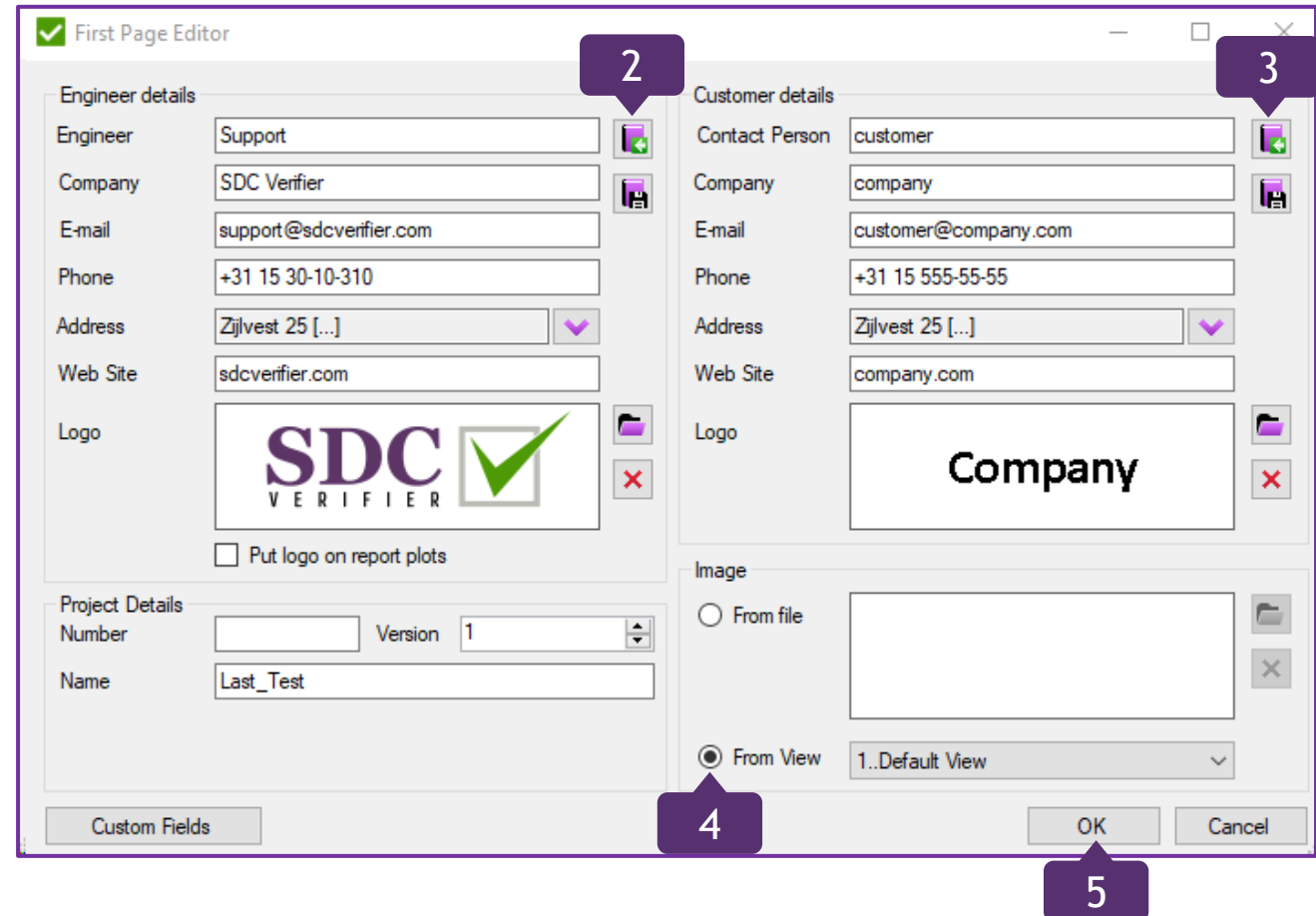
Image - From View: Selected

5

Press *OK*



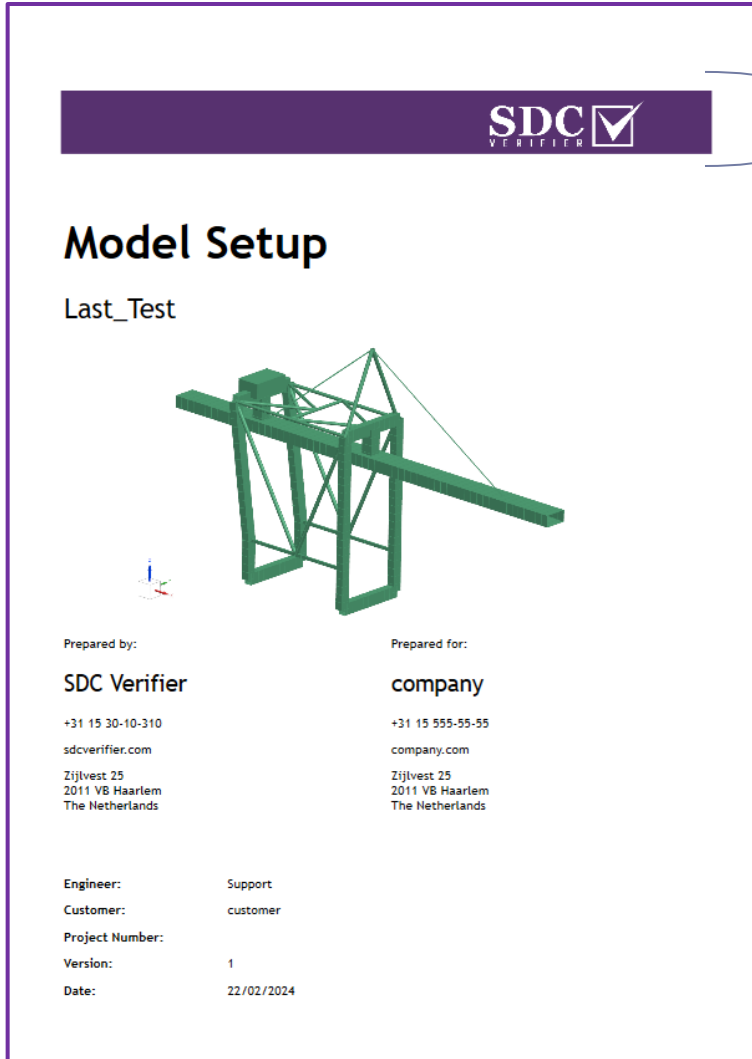
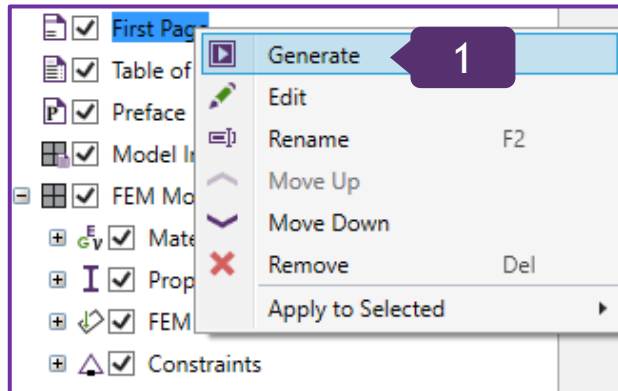
For the customer, the default data from the library is used. It is possible to fill in the relevant data and store it to the library, reusing it in future projects.





1

Execute right click on *First Page*, and select *Generate*

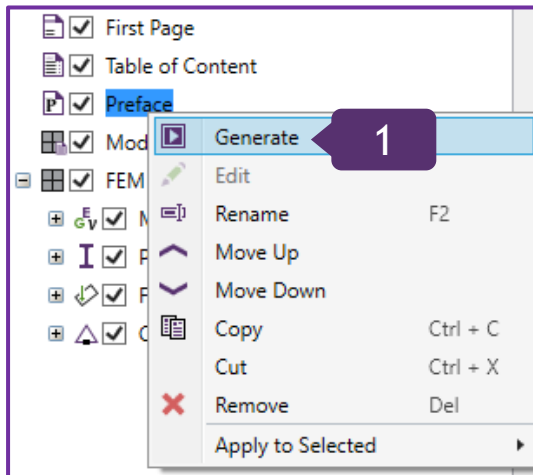


Company's name and logo are used in a footer. All pages, except for the first one have the footer.

Report layout in Designer does not contain headers and footers, as they are inserted when exported to Word Document.

1

Execute right click on *Preface*, and select *Generate*



In first paragraph, you can find what versions of SDC Verifier and Simcenter 3D were used. Full path to the model and project files are based on what profile the report was generated.

Description on Current Unit System has an influence on calculations according to certain standards.

## Preface

This document is generated with SDC Verifier 2023.2 and calculated with Simcenter v2306.0.0  
Model File: C:\Users\user\Desktop\SIMCENTER Tutorials\Report Designer\ReportDesigner\_s.sim  
Project File: C:\Users\user\Desktop\SIMCENTER Tutorials\Report Designer\Report\_Designer.simv  
Report Profile: 1..Model Setup  
Generation on: 2/23/2024 4:02:37 PM

### Unit System

Current Unit System = MmKS (Millimeter/Kg/Second). It is used in calculations for the following standards: API RP 2A, ISO 19902, Norsok N004, DIN 15018, FEM 1.001 and Eurocode3.

Dimensions		Value
Length		Millimeter
Mass		Kilogram
Time		Second
Force		mN (Millinewton)
Stress		kPa

**Behavior**

Break Page Before: Yes

Enabled: Yes

**Data**

Last Time Generated:

Title (Default): Preface

Title (User):

**Options**

Include Introduction: No

Description:

Introduction section is an option on how to use and navigate in the Report. By default, it is OFF.

1

Execute right click on *FEM Model Description*, and select *Generate*

Detailed Material description with plots.

## FEM Model Description

This paragraph shows detailed or brief model overview.

### Materials

This paragraph contains materials information.

#### Materials Summary

Calculated for the CSys "0..Base".

Title	Elements in selection / in model	Mass [kg]	Mass Density [kg/mm^3]	Gravity Center [mm]
1..steel	420	1937143.0	0.00	[-13846.06; 0.00; 34294.77]
2..Machine_house	1	79999.9	0.00	[-35480.00; 0.00; 52000.00]
Overall	421	2017143.0		[-14704.06; 0.00; 34996.96]

Material Summary is a mass overview over materials.

- ☒ First Page
- ☒ Table of Content
- ☒ Preface
- ☒ Model Information
- ☒ **FEM Model Description**
- ☒ Materials
- ☒ Properties
- ☒ FEM Loads
- ☒ Constraints

- Generate** 1
- Edit
- Rename F2
- Move Up
- Move Down
- Copy Ctrl + C
- Cut Ctrl + X
- Remove Del
- Add
- Select Items
- Apply to Selected

## FEM Loads

This paragraph contains information about applied loads to model.

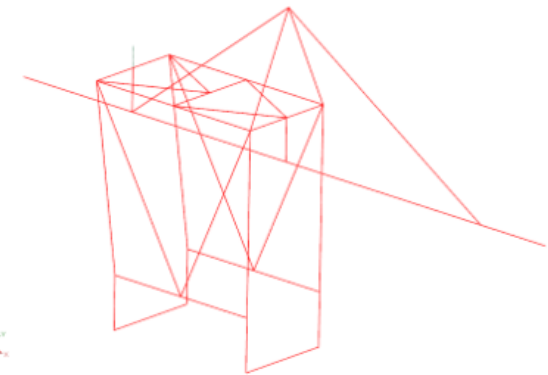
### 1..gravity

Definition	Load Type	Applied On	Values
gravity	Gravity	Full model	Fx: 0; Fy: 0; Fz: -9810; Fx: 0; Fy: 0; Fz: -9810;



### 1..steel

Property	Value
Elements	420
Mass [kg]	1937143.0
Gravity Center (CSys: 0) [mm]	[-13846.06; 0.00; 34294.77]
Young Modulus [kPa]	2.10e+8
Shear Modulus [kPa]	0
Poisson Ratio	0.30
Shear [kPa]	0
Mass Density [kg/mm^3]	9.813e-06
Tensile Strength [kPa]	0.36e+6
Yield Stress [kPa]	0.24e+6



### 18..machine\_house

Property	Value	Property	Value
Type / Elements	Beam / 1	Material	2..Machine_house
Mass [kg]	79999.9	Gravity Center, [mm]	[-35480.00; 0.00; 52000.00]
End A Shape		End B Shape	
Area, [mm^2]	60000000.00	Area, [mm^2]	0
I1, [mm^4]	1800000000000001.00	I1, [mm^4]	0
I2, [mm^4]	5000000000000000.00	I2, [mm^4]	0
I12, [mm^4]	-2.762e-03	I12, [mm^4]	0
Torsion Constant, [mm^4]	450603500240720.00	Torsion Constant, [mm^4]	0
Y Shear Area, [mm^2]	0	Y Shear Area, [mm^2]	0
Z Shear Area, [mm^2]	0	Z Shear Area, [mm^2]	0
Nonstructural Mass, [kg]	0	Nonstructural Mass, [kg]	0
Perimeter, [mm]	0	Perimeter, [mm]	0
Warping Constant, [mm^6]	34870477240910000000.00	Warping Constant, [mm^6]	0
Y Neutral Axis Offset A, [mm]	0	Y Neutral Axis Offset B, [mm]	0
Z Neutral Axis Offset A, [mm]	0	Z Neutral Axis Offset B, [mm]	0

1

In the *Report Structure*, expand *FEM Model Description* => *Materials*

2

Select *1..steel*

3

In *Plot* section, Preview Mode: *Display Only Selected*

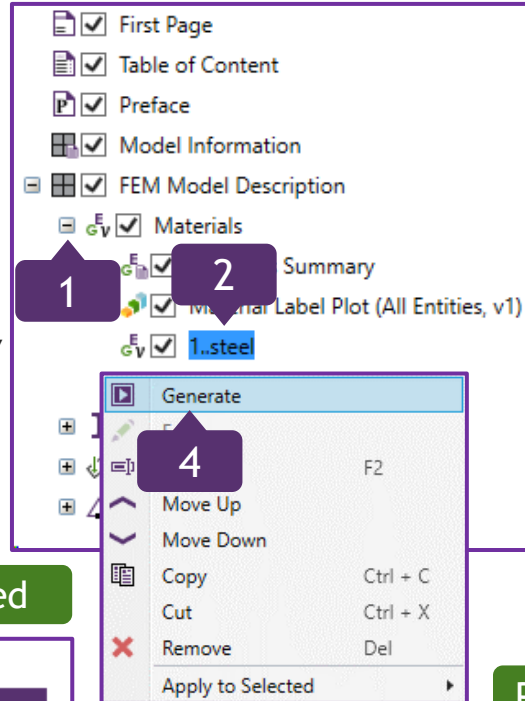
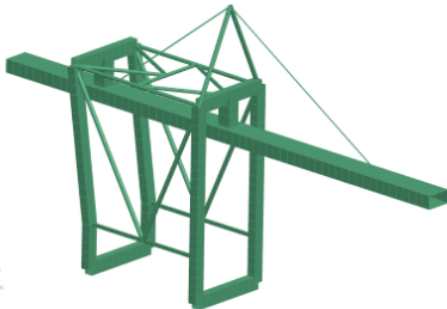
4

Execute right click on *1..steel* and press *Generate*

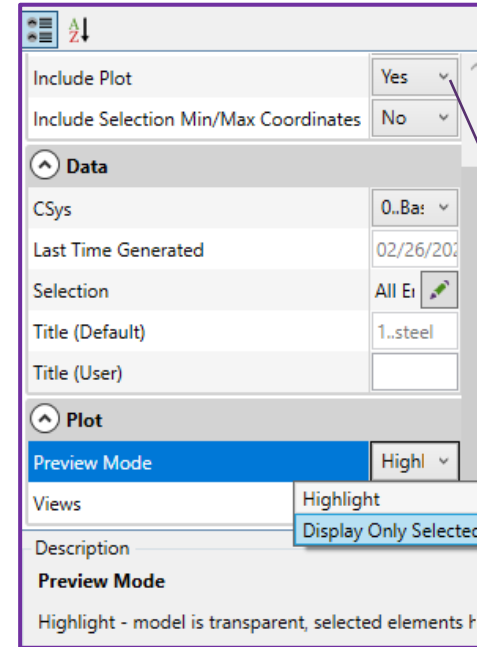
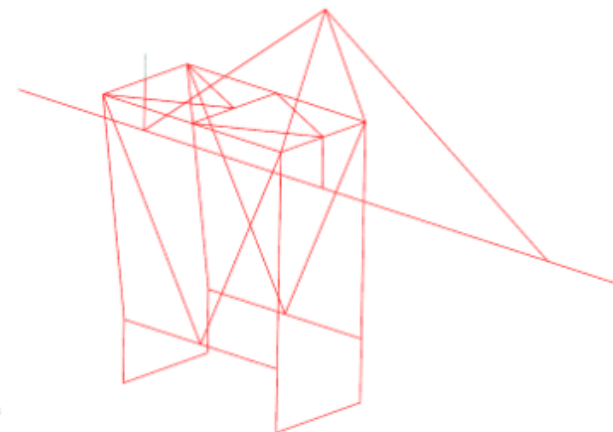
Preview Mode: Display Only Selected

## 1..steel

Property	Value
Elements	420
Mass [kg]	1937143.0
Gravity Center (CSys: 0) [mm]	[-13846.06; 0.00; 34294.77]
Young Modulus [kPa]	2.10e+8
Shear Modulus [kPa]	0
Poisson Ratio	0.30
Shear [kPa]	0
Mass Density [kg/mm <sup>3</sup> ]	9.813e-06
Tensile Strength [kPa]	0.36e+6
Yield Stress [kPa]	0.24e+6



Preview Mode: Highlight



It is possible to exclude a plot using option - Include Plot: No option.

3

# Create View

1

In *Home* section of the Report Ribbon, press **Views**

2

Press  to add View

3

Title: *Isometric with filled edges*

4

Rendering Style: *Shaded with Body Color Edges*

5

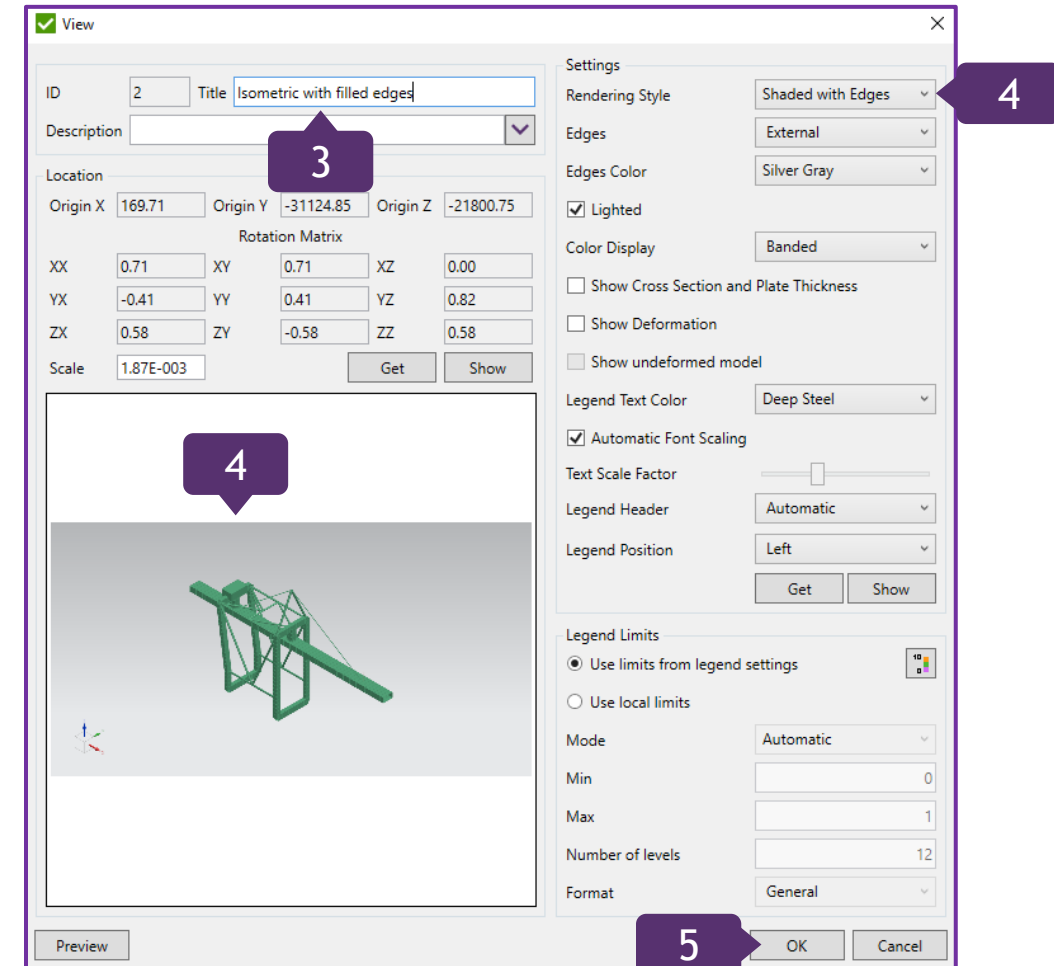
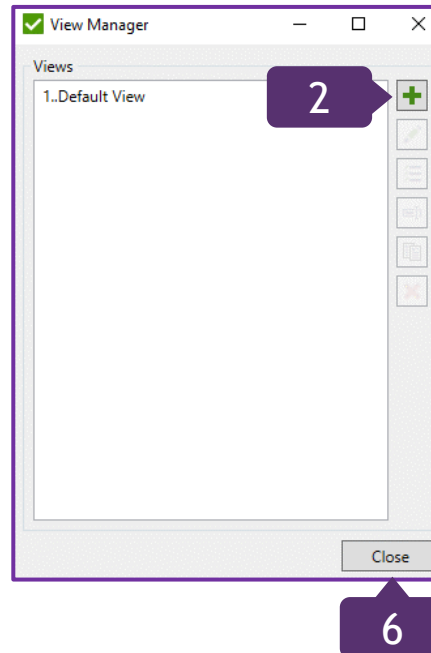
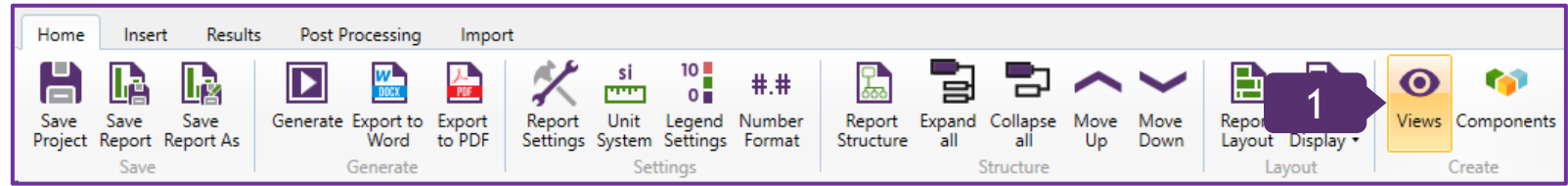
Locate View in Simcenter 3D as shown on picture

6

Press *OK*

7

Press *Close*



# Apply View to all FEM Loads

1

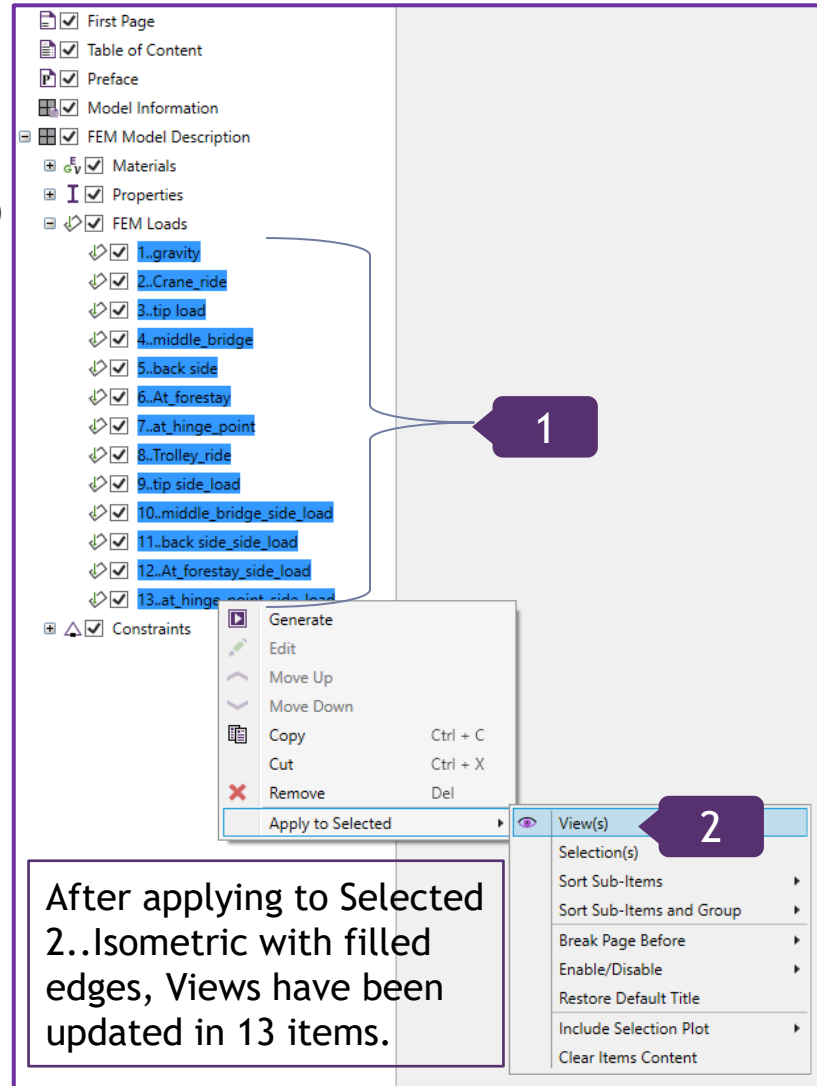
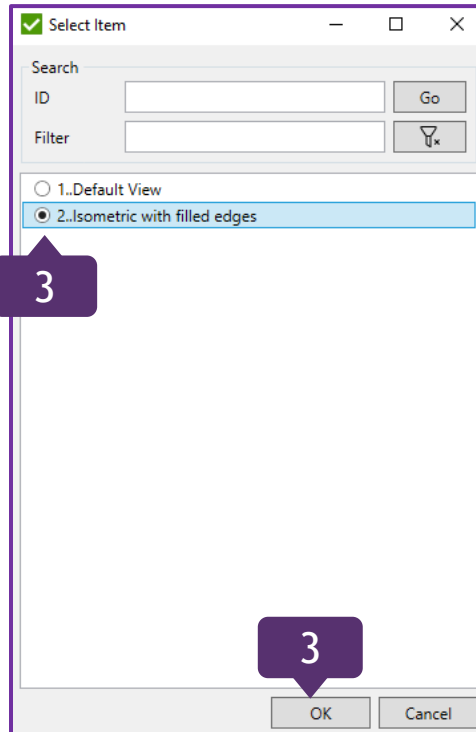
In the *Report Structure*, expand *FEM Model Description* => *FEM Loads* and select all FEM Loads

2

Execute right click on selected FEM Loads => Apply to Selected => View(s)

3

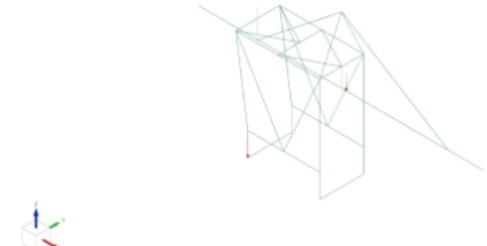
Select *2..Isometric with filled edges*; Press *OK*



After applying to Selected 2..Isometric with filled edges, Views have been updated in 13 items.

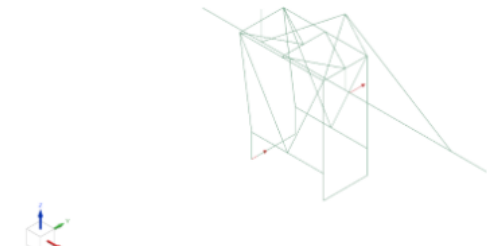
## 7..at\_hinge\_point

Definition	Load Type	Applied On	Values
at_hinge_point	Force	Nodes: 170;348;	ScalingForce: 1000;



## 13..at\_hinge\_point\_side\_load

Definition	Load Type	Applied On	Values
at_hinge_point_side_load	Force	Nodes: 170;348;	ScalingForce: 1000000;



# Report Settings Options

1

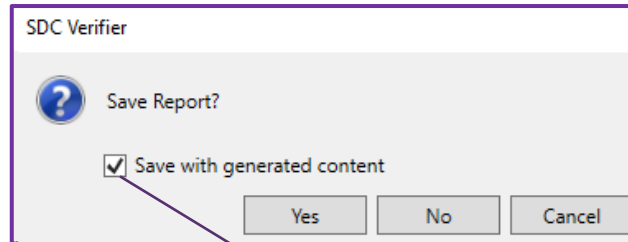
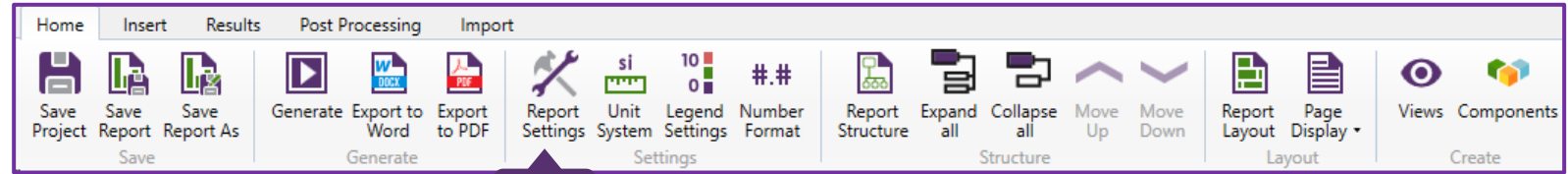
In *Home* section of the Report Ribbon, press Report Settings

2

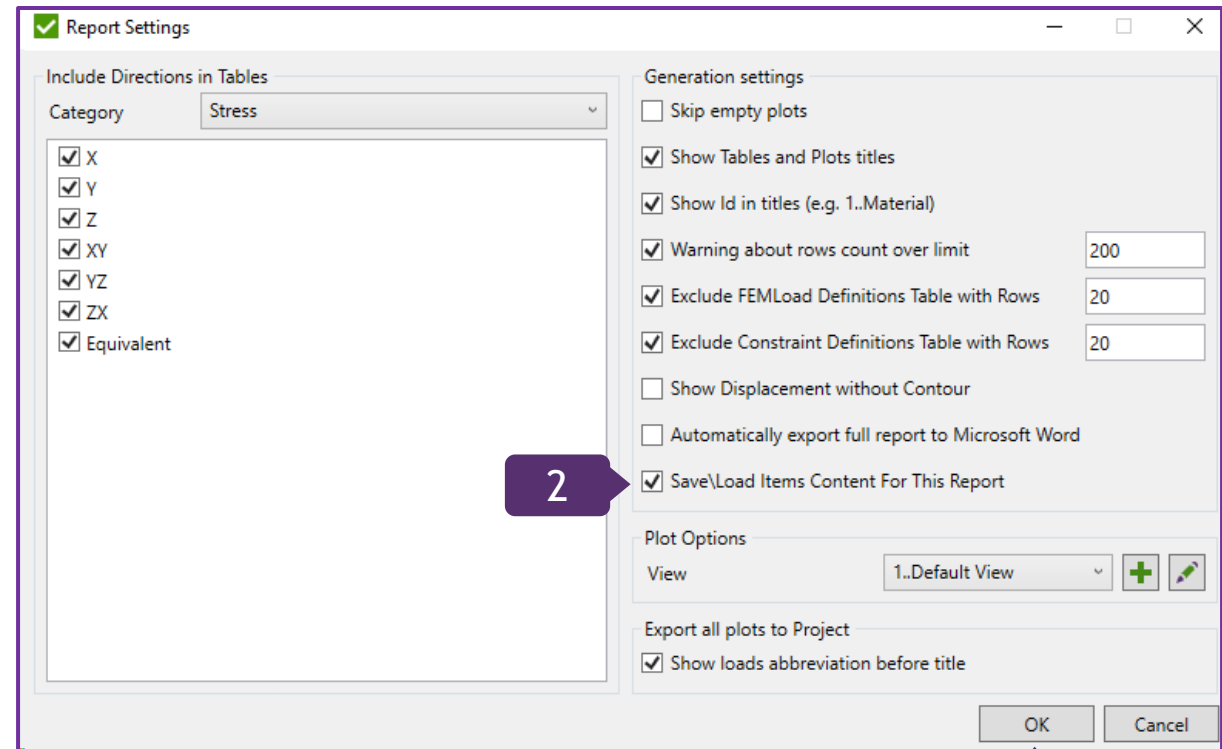
Save\Load Items Content For This Report: *ON*

3

Press *OK*



An option of saving with generated content allows to save the data without generation a report once again after closing the Report Designer window. For big-sized reports, it may be time-consuming to open, though.



3



# Generate Report

1

In *Home* section of the Report Ribbon, press *Generate*

2

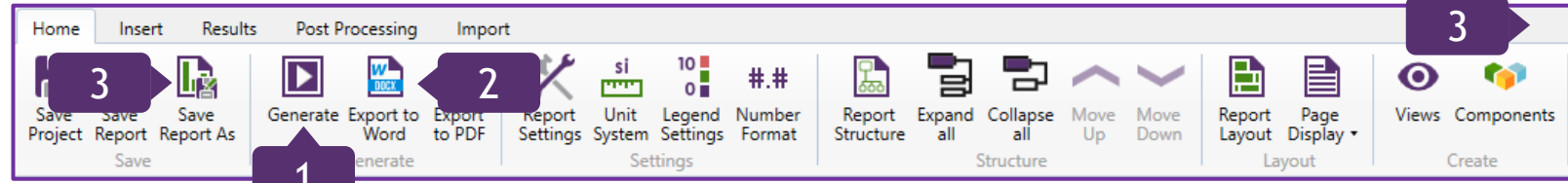
Press *Export to Word*

3

Press *Save Report As*;  
Press *OK*;  
Close the window

4

Press *Yes*



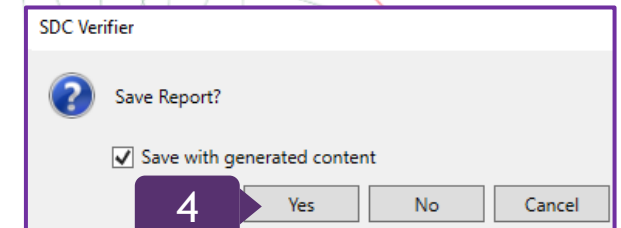
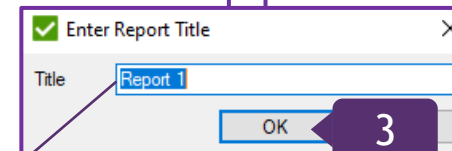
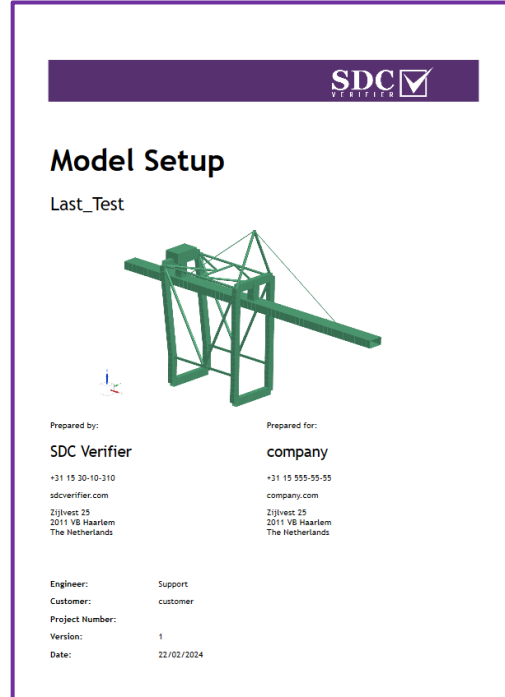
Note: By pressing Generate, the entire Report will be created.

## 13..backstay

Property	Value
Elements	1
Type	Rod
Material	1..steel
Mass [kg]	18272.3
Gravity Center (CSys: 0) [mm]	[-17740.00; 0.00; 56000.00]
Area [mm^2]	41200.00
Torsion Constant [mm^4]	3.287e-03
Perimeter [mm]	719.538
Coefficient for Torsion Stress	0.03
Nonstructural Mass [kg/mm]	0
Initial Tension [kPa]	0
Initial Slack [kPa]	0
Allow Tension Stress [kPa]	0
Area Moment of Inertia [mm^4]	0

## 12..forestay

Property	Value
Elements	1
Type	Rod
Material	1..steel
Mass [kg]	21426.6
Gravity Center (CSys: 0) [mm]	[22500.00; 0.00; 56000.00]
Area [mm^2]	41200.00
Torsion Constant [mm^4]	3.287e-03
Perimeter [mm]	719.538
Coefficient for Torsion Stress	0.03
Nonstructural Mass [kg/mm]	0
Initial Tension [kPa]	0
Initial Slack [kPa]	0
Allow Tension Stress [kPa]	0
Area Moment of Inertia [mm^4]	0



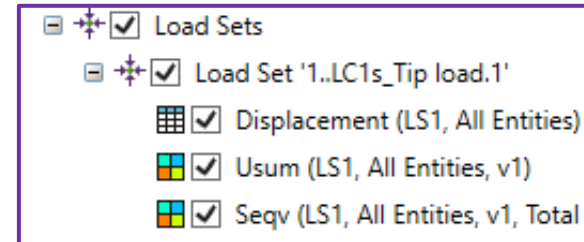
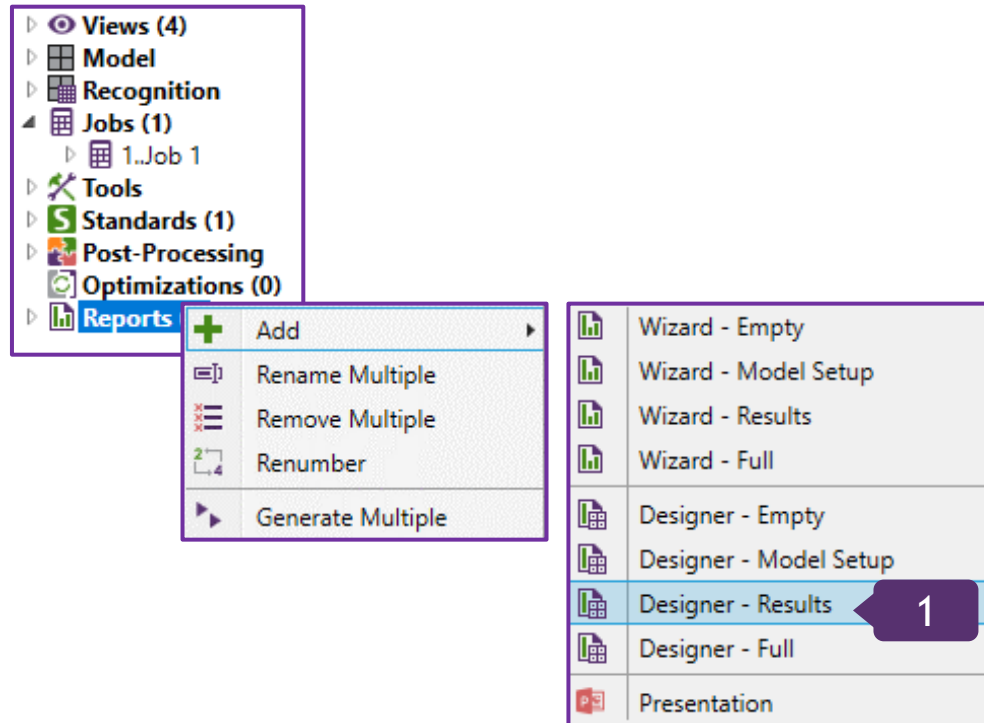
The title of the Report can be changed.



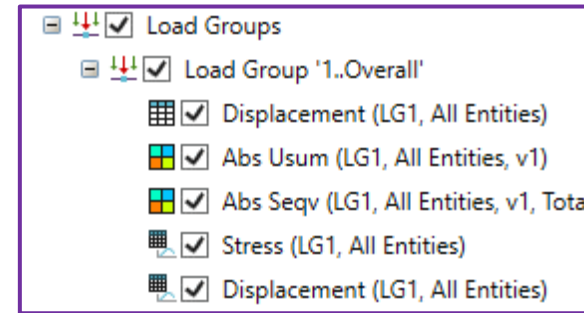
# Add Results Report

1

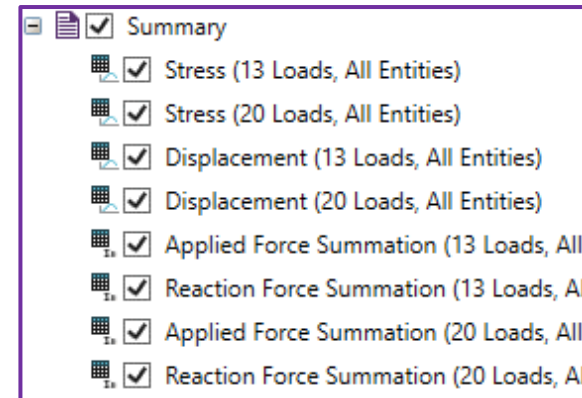
In the *Model Tree*, execute right click on *Reports => Add*, and select *Designer - Results*



For each Load extreme Displacement table, Displacement and Stress plots are created.



For Load Groups, only Displacement and Stresses over Loads tables are included.



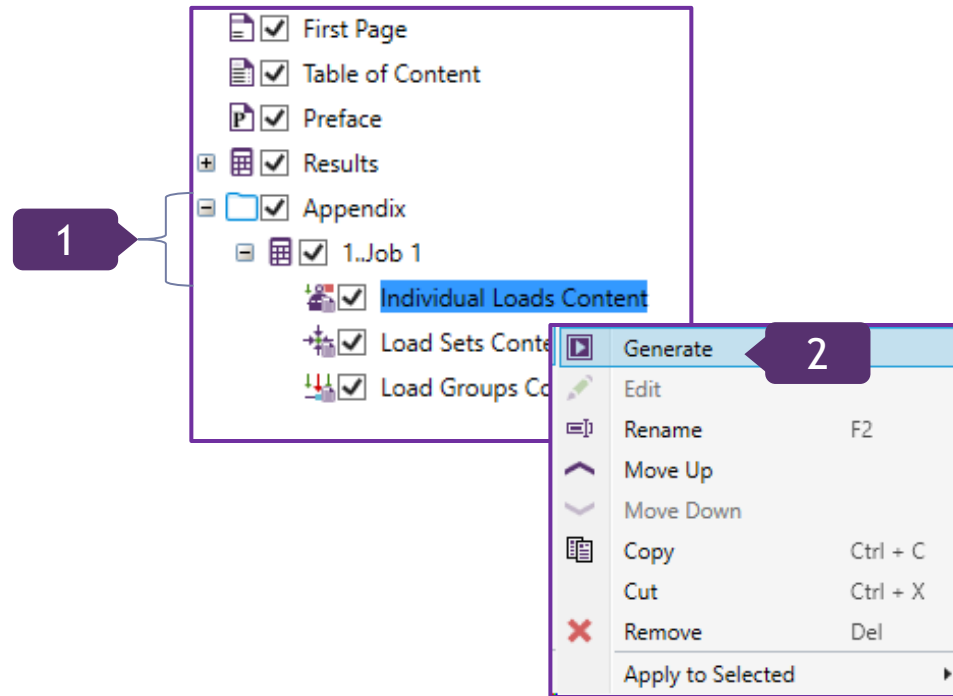
For Individual loads and Load Sets these Summary tables are included: Applied and Reaction Force Summation, Displacement and Stresses over Loads.

1

In the *Report Structure*, expand Appendix => 1..Job

2

Execute right click on *Individual Loads Content* and select *Generate*



Individual Loads Content shows what boundary conditions are for an Individual Load. If the Individual Load was created based on results, so Result Case is shown instead of FEM Load.

## Individual Loads Content

Individual Load [Safety Factor]	FemLoad / Result Case	Constraint
1..gravity [1]	1..gravity	1..Bogie_simple
2..tip load [1]	3..tip load	1..Bogie_simple
3..middle_bridge [1]	4..middle_bridge	1..Bogie_simple
4..back side [1]	5..back side	1..Bogie_simple
5..At_forestay [1]	6..At_forestay	1..Bogie_simple
6..at_hinge_point [1]	7..at_hinge_point	1..Bogie_simple
7..Trolley_ride [1]	8..Trolley_ride	1..Bogie_simple
8..tip side_load [1]	9..tip side_load	1..Bogie_simple
9..middle_bridge_side_load [1]	10..middle_bridge_side_load	1..Bogie_simple
10..back side_side_load [1]	11..back side_side_load	1..Bogie_simple
11..At_forestay_side_load [1]	12..At_forestay_side_load	1..Bogie_simple
12..at_hinge_point_side_load [1]	13..at_hinge_point_side_load	1..Bogie_simple
13..Crane_ride [1]	2..Crane_ride	1..Bogie_simple

# Load Set Content

1

In the *Report Structure*, expand Appendix => 1..Job and select *Load Sets Content*

2

In Data, All Load Sets: No

3

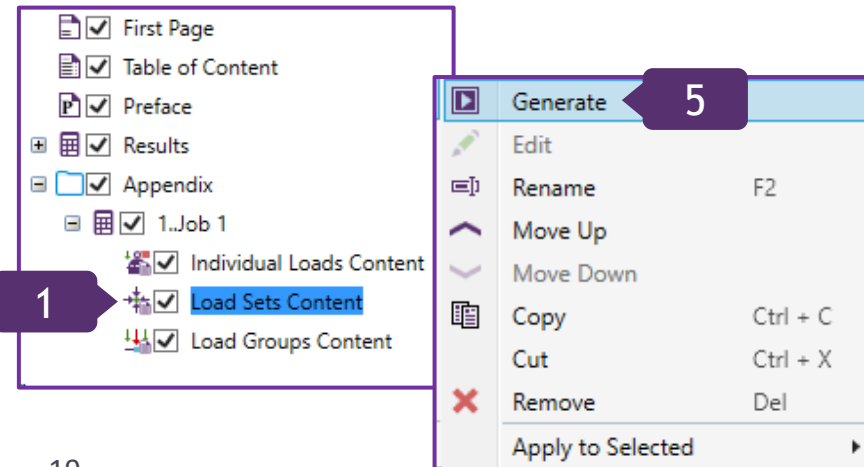
In Load Sets, press  to select Loads

4

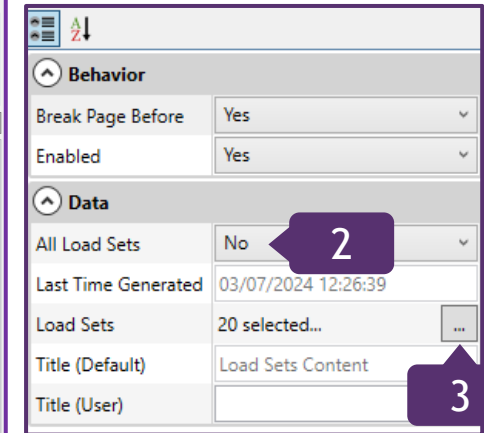
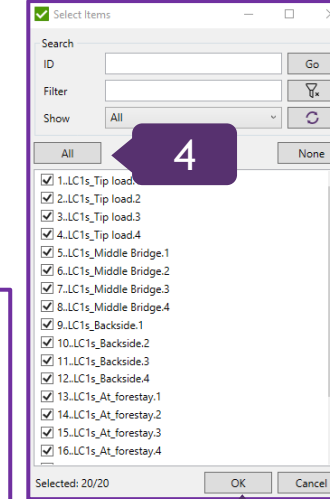
In the menu, select all Load Sets that should be displayed (press All); Press OK

5

Execute right click on *Load Sets Content* and select *Generate*



Load Sets Content		
Title [Safety Factor]	Count	Items [Partial Load Factor]
1..LC1s_Tip load.1 [1]	5	1..gravity [1.15] 2..tip load [1.35] 7..Trolley_ride [1.15] 8..tip_side_load [1.15] 13..Crane_ride [1.15]
2..LC1s_Tip load.2 [1]	5	1..gravity [1.15] 2..tip load [1.35] 7..Trolley_ride [1.15] 8..tip_side_load [-1.15] 13..Crane_ride [-1.15]
3..LC1s_Tip load.3 [1]	5	1..gravity [1.15] 2..tip load [1.35] 7..Trolley_ride [-1.15] 8..tip_side_load [1.15] 13..Crane_ride [1.15]
4..LC1s_Tip load.4 [1]	5	1..gravity [1.15] 2..tip load [1.35] 7..Trolley_ride [-1.15] 8..tip_side_load [-1.15] 13..Crane_ride [-1.15]
5..LC1s_Middle Bridge.1 [1]	5	1..gravity [1.15] 3..middle_bridge [1.35] 7..Trolley_ride [1.15] 9..middle_bridge_side_load [1.15] 13..Crane_ride [1.15]
6..LC1s_Middle Bridge.2 [1]	5	1..gravity [1.15] 3..middle_bridge [1.35] 7..Trolley_ride [1.15] 9..middle_bridge_side_load [-1.15] 13..Crane_ride [-1.15]
7..LC1s_Middle Bridge.3 [1]	5	1..gravity [1.15] 3..middle_bridge [1.35] 7..Trolley_ride [-1.15] 9..middle_bridge_side_load [1.15] 13..Crane_ride [1.15]
8..LC1s_Middle Bridge.4 [1]	5	1..gravity [1.15] 3..middle_bridge [1.35] 7..Trolley_ride [-1.15] 9..middle_bridge_side_load [-1.15] 13..Crane_ride [-1.15]
9..LC1s_Backside.1 [1]	5	1..gravity [1.15] 4..back side [1.35] 7..Trolley_ride [1.15] 10..back_side_load [1.15] 13..Crane_ride [1.15]
10..LC1s_Backside.2 [1]	5	1..gravity [1.15] 4..back side [1.35] 7..Trolley_ride [1.15] 10..back_side_load [-1.15] 13..Crane_ride [-1.15]
11..LC1s_Backside.3 [1]	5	1..gravity [1.15] 4..back side [1.35] 7..Trolley_ride [-1.15] 10..back_side_load [1.15] 13..Crane_ride [1.15]
12..LC1s_Backside.4 [1]	5	1..gravity [1.15]



13..LC1s_At_forestay.1 [1]	5	4..back side [1.35] 7..Trolley_ride [-1.15] 10..back_side_load [-1.15] 13..Crane_ride [-1.15]
14..LC1s_At_forestay.2 [1]	5	1..gravity [1.15] 5..At_forestay [1.35] 7..Trolley_ride [1.15] 11..At_forestay_side_load [1.15] 13..Crane_ride [1.15]
15..LC1s_At_forestay.3 [1]	5	1..gravity [1.15] 5..At_forestay [1.35] 7..Trolley_ride [1.15] 11..At_forestay_side_load [-1.15] 13..Crane_ride [-1.15]
16..LC1s_At_forestay.4 [1]	5	1..gravity [1.15] 5..At_forestay [1.35] 7..Trolley_ride [-1.15] 11..At_forestay_side_load [1.15] 13..Crane_ride [1.15]
17..LC1s_at_hinge_point.1 [1]	5	1..gravity [1.15] 6..at_hinge_point [1.35] 7..Trolley_ride [1.15] 12..at_hinge_point_side_load [1.15] 13..Crane_ride [1.15]
18..LC1s_at_hinge_point.2 [1]	5	1..gravity [1.15] 6..at_hinge_point [1.35] 7..Trolley_ride [1.15] 12..at_hinge_point_side_load [-1.15] 13..Crane_ride [-1.15]
19..LC1s_at_hinge_point.3 [1]	5	1..gravity [1.15] 6..at_hinge_point [1.35] 7..Trolley_ride [-1.15] 12..at_hinge_point_side_load [1.15] 13..Crane_ride [1.15]
20..LC1s_at_hinge_point.4 [1]	5	1..gravity [1.15] 6..at_hinge_point [1.35] 7..Trolley_ride [-1.15] 12..at_hinge_point_side_load [-1.15] 13..Crane_ride [-1.15]

1

Execute right click on *Load Groups Content* and select *Generate*

The screenshot shows the SDC Verifier interface. On the left, a tree view contains the following items: 'First Page', 'Table of Content', 'Preface', 'Results', 'Appendix', '1..Job 1', 'Individual Loads Content', 'Load Sets Content', and 'Load Groups Content'. The 'Load Groups Content' item is highlighted in blue. A right-click context menu is open over this item, with the 'Generate' option at the top highlighted in blue. A purple circle with the number '1' points to the 'Generate' option. Other options in the menu include 'Edit', 'Rename' (F2), 'Move Up', 'Move Down', 'Copy' (Ctrl + C), 'Cut' (Ctrl + X), 'Remove' (Del), and 'Apply to Selected'.

## Load Groups Content

Title	Count	Items [Safety Factor]
1..Overall	20	Load Set '1..LC1s_Tip load.1' [1] Load Set '2..LC1s_Tip load.2' [1] Load Set '3..LC1s_Tip load.3' [1] Load Set '4..LC1s_Tip load.4' [1] Load Set '5..LC1s_Middle Bridge.1' [1] Load Set '6..LC1s_Middle Bridge.2' [1] Load Set '7..LC1s_Middle Bridge.3' [1] Load Set '8..LC1s_Middle Bridge.4' [1] Load Set '9..LC1s_Backside.1' [1] Load Set '10..LC1s_Backside.2' [1] Load Set '11..LC1s_Backside.3' [1] Load Set '12..LC1s_Backside.4' [1] Load Set '13..LC1s_At_forestay.1' [1] Load Set '14..LC1s_At_forestay.2' [1] Load Set '15..LC1s_At_forestay.3' [1] Load Set '16..LC1s_At_forestay.4' [1] Load Set '17..LC1s_at_hinge_point.1' [1] Load Set '18..LC1s_at_hinge_point.2' [1] Load Set '19..LC1s_at_hinge_point.3' [1] Load Set '20..LC1s_at_hinge_point.4' [1]

# Individual Loads Content. Options of Property Grid

1

In the *Report Structure*, expand *Results*  
=> *1..Job 1* => *Individual Loads* and  
select *Individual Load '5..At\_forestay'*

2

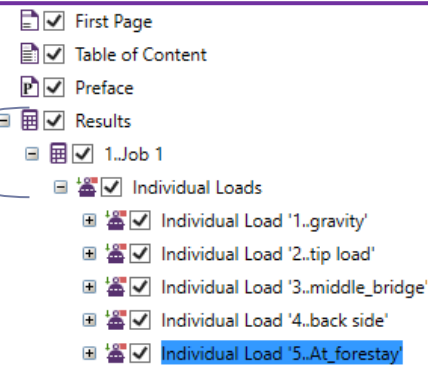
Set the options as shown on the  
screenshot

3

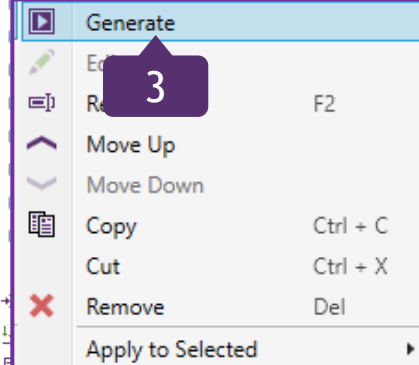
Execute right click on *Individual Load*  
*'5..At\_forestay'* and select *Generate*

Individual Load includes Content  
and Sum of Forces. It is possible to  
control what should be displayed,  
using the Options.

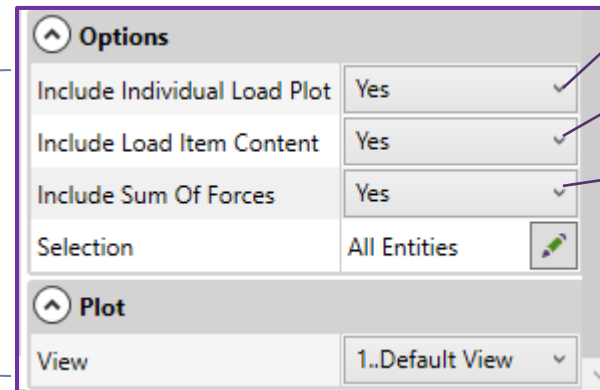
1



3

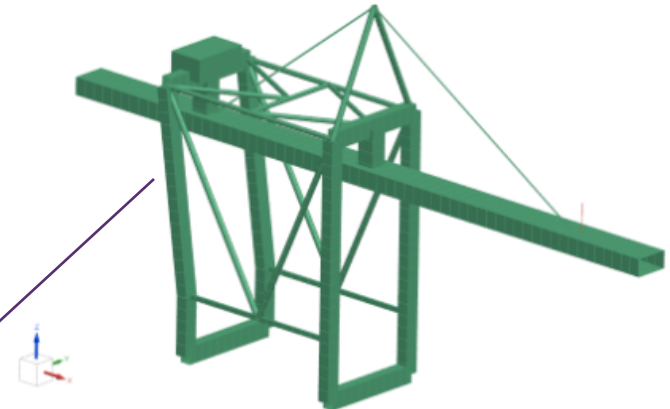


2



## Individual Load '5..At\_forestay'

At\_forestay.Bogie\_simple



Title	Value
Individual Load	5..At_forestay
FemLoad	6..At_forestay
Constraint	1..Bogie_simple
Result Case	-
Safety Factor	1

Sum of Reaction Forces								
Load	Fx [mN]	Fy [mN]	Fz [mN]	Fsum [mN]	Mx [mN mm]	My [mN mm]	Mz [mN mm]	Msum [mN mm]
Constraint '1..Bogie_simple'	0e+6	0e+6	1220e+6	1220e+6	0.0	0.0	0.0	0.0

Displacement (IL5, All Entities)								
Individual Load	5..At_forestay	Selection Category			All Entities Displacement			
Type	Extreme	Ux [mm]	Uy [mm]	Uz [mm]	Usum [mm]	Rx	Ry	Rz
Minimum		-7.24e-1	-3.70e+0	-1.13e+2	0	0.00	0.00	0.00
Maximum		2.90e+1	3.70e+0	1.26e+1	1.13e+2	0.00	0.00	0.00
Absolute		2.90e+1	3.70e+0	-1.13e+2	1.13e+2	0.00	0.00	0.00

1 Expand *Individual Load '5..At\_forestay'*;  
Execute right click on *Displacement (IL5, All Entities)*

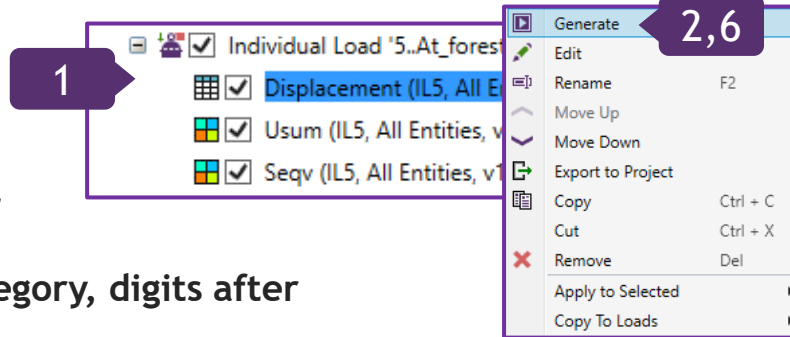
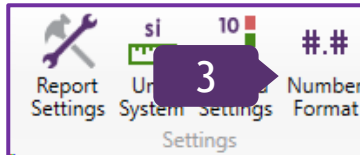
2 Press *Generate*

3 Press *Number Format*

4 For *Displacement* category, digits after decimal point: 3;  
Number Format: *General*

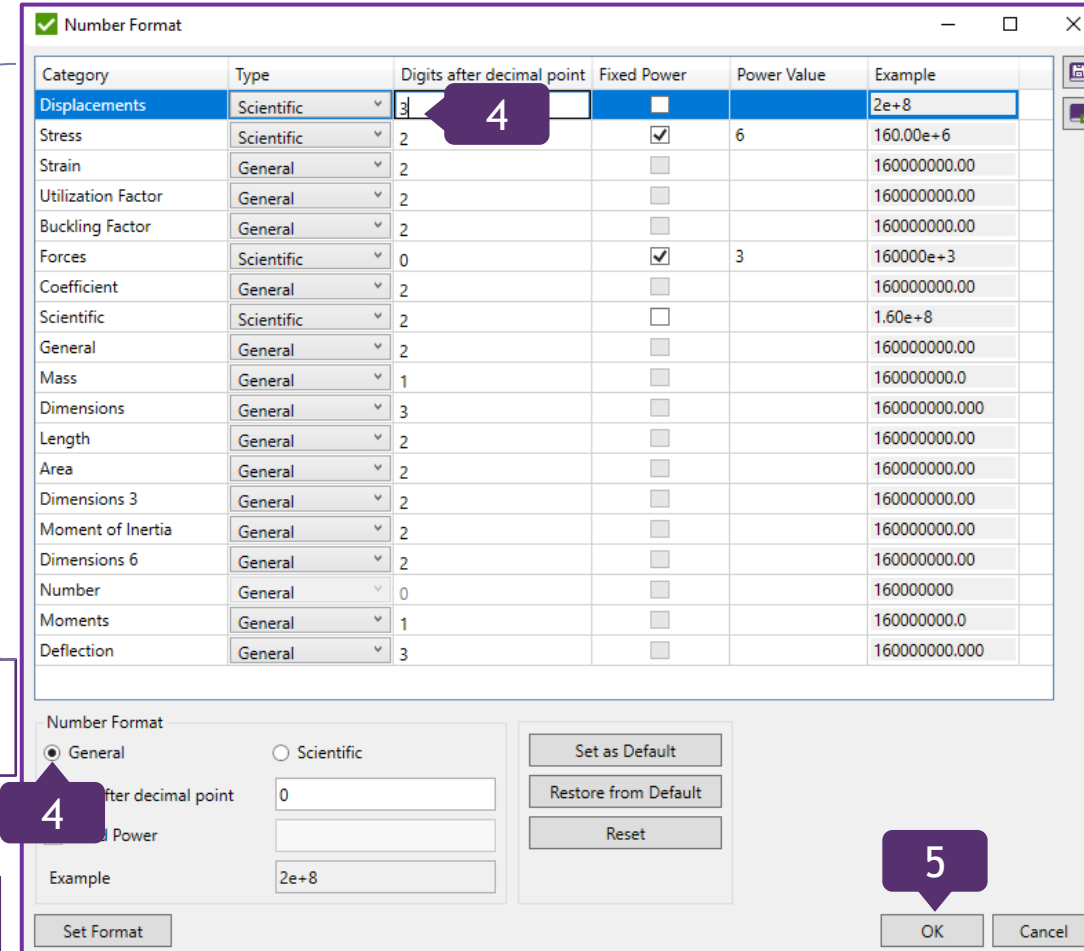
5 Press *OK*

6 Repeat step 2



Number Formats controls how numbers are displayed in tables for different categories. It is possible to save settings to library and reuse them in other projects.

In addition to Displacements, other Categories can be modified in the same Number Format window, if required.



Digits after decimal point: 0

Digits after decimal point: 3

Displacement (IL5, All Entities)									
Individual Load Type	5..At_forestay Extreme	Selection Category			All Entities Displacement				
	Extreme	Ux [mm]	Uy [mm]	Uz [mm]	Usum [mm]	Rx	Ry	Rz	Rsum
Minimum		-7e-1	-4e+0	-1e+2	0	0.00	0.00	0.00	0.00
Maximum		3e+1	4e+0	1e+1	1e+2	0.00	0.00	0.00	0.00
Absolute		3e+1	4e+0	-1e+2	1e+2	0.00	0.00	0.00	0.00

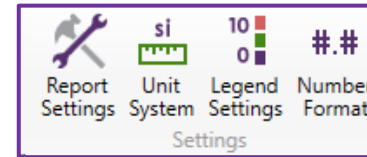
Displacement (IL5, All Entities)									
Individual Load Type	5..At_forestay Extreme	Selection Category			All Entities Displacement				
	Extreme	Ux [mm]	Uy [mm]	Uz [mm]	Usum [mm]	Rx	Ry	Rz	Rsum
Minimum		-7.238e-1	-3.699e+0	-1.127e+2	0	0.00	0.00	0.00	0.00
Maximum		2.900e+1	3.699e+0	1.259e+1	1.127e+2	0.00	0.00	0.00	0.00
Absolute		2.900e+1	3.699e+0	-1.127e+2	1.127e+2	0.00	0.00	0.00	0.00



# Reaction Forces

- 1 In *Summary*, select *Reaction Force Summation (20 Loads, All Entities)*
- 2 Press *Number Format*
- 3 Category: *Forces*
- 4 Type: *Scientific*; Digits after decimal point: 0  
Fixed Power: *ON*; Fixed Power Value: 3
- 5 Press *Set Format* and *OK*
- 6 Execute right click on *Reaction Force Summation (20 Loads, All Entities)* and select *Generate*

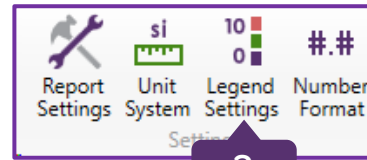
Number Format from general is changed to scientific with fixed power = 3.  
The numbers became more readable.



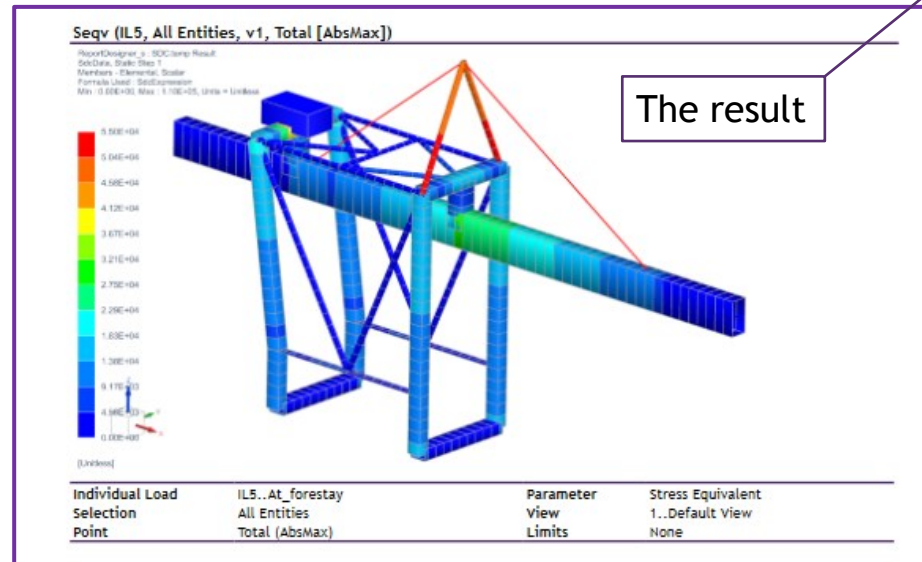
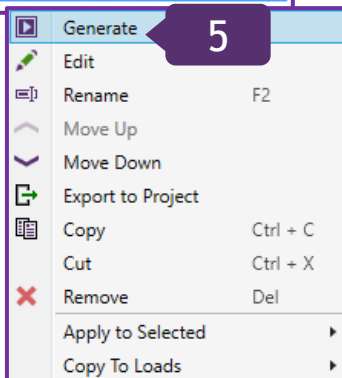
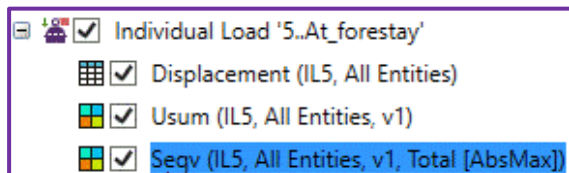
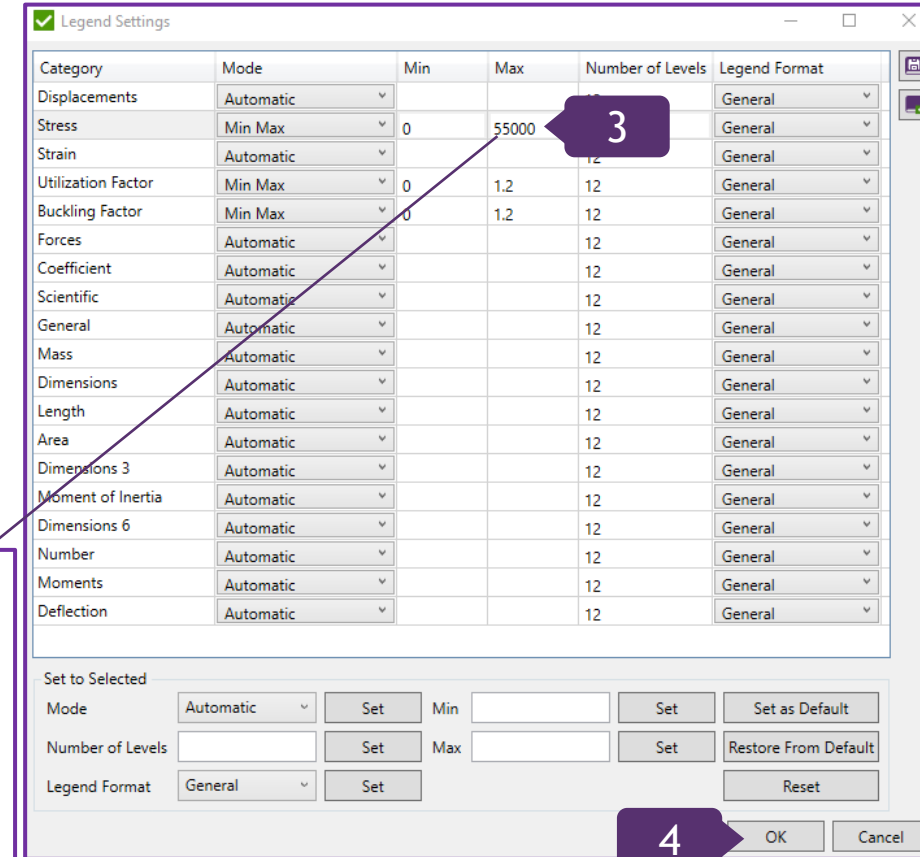
The image shows two screenshots. The left screenshot is the 'Number Format' dialog box. It has a table with columns: Category, Type, Digits after decimal point, Fixed Power, Power Value, and Example. The 'Forces' category is selected with 'Scientific' type, 0 digits, and Fixed Power checked with a value of 3. A callout '3' points to the 'Forces' row, and a callout '4' points to the 'Fixed Power' checkbox. Below the table, the 'Number Format' section shows 'Scientific' selected, with 'Digits after decimal point' set to 0. A callout '5' points to the 'Set Format' button. The right screenshot is the 'Reaction Force Summation (20 Loads, All Entities)' table. It shows a list of loads and their reaction forces (Fx, Fy, Fz, Fsum, Mx, My, Mz, Msum) in scientific notation. A callout '6' points to the 'Generate' button in the bottom right corner of the table.

Load	Fx [mN]	Fy [mN]	Fz [mN]	Fsum [mN]	Mx [mN mm]	My [mN mm]	Mz [mN mm]	Msum [mN mm]
LS1...LC1s_Tip load.1	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS2...LC1s_Tip load.2	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS3...LC1s_Tip load.3	223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS4...LC1s_Tip load.4	223100e+3	635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS5...LC1s_Middle Bridge.1	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS6...LC1s_Middle Bridge.2	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS7...LC1s_Middle Bridge.3	223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS8...LC1s_Middle Bridge.4	223100e+3	635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS9...LC1s_Backside.1	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS10...LC1s_Backside.2	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS11...LC1s_Backside.3	223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS12...LC1s_Backside.4	223100e+3	635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS13...LC1s_At_forestay.1	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS14...LC1s_At_forestay.2	-223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS15...LC1s_At_forestay.3	223100e+3	-635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS16...LC1s_At_forestay.4	223100e+3	635703e+3	24403388e+3	24412686e+3	0.0	0.0	0.0	0.0
LS17...LC1s_at_hinge_point.1	-223099e+3	-11212993e+3	26050384e+3	11511643e+3	0.0	0.0	0.0	0.0
LS18...LC1s_at_hinge_point.2	-223097e+3	-11212992e+3	26050384e+3	11511643e+3	0.0	0.0	0.0	0.0
LS19...LC1s_at_hinge_point.3	-223099e+3	-11212991e+3	26050384e+3	11511642e+3	0.0	0.0	0.0	0.0
LS20...LC1s_at_hinge_point.4	-223101e+3	-11212992e+3	26050396e+3	11511643e+3	0.0	0.0	0.0	0.0

- 1 In Individual Load '5..At\_forestay', select Seqv (IL5, All Entities, v1, Total [AbsMax])
- 2 Press Legend Settings
- 3 Stress Category: Max: 55000
- 4 Press OK
- 5 Execute right click on Seqv (IL5, All Entities, v1, Total [AbsMax]) and select Generate



Legend Settings control Legend options for different categories. It is possible to save settings to the library and reuse them in other projects.





# Stress and Displacement Tables over Loads

Stress and Displacement Extreme Flow Tables give results overview among Loads. For each direction Min and Max values are highlighted. Min = aqua, Max = red.

1

In the *Report Structure*, expand *Results* => *1..Job 1* => *Summary* and select *Stress (20 Loads, All Entities)*

2

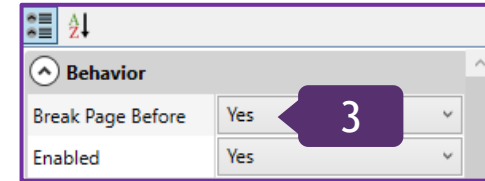
Press *Generate*

3

Break Page Before: Yes

4

Execute right click on *Displacement (20 Loads, All Entities)* and select *Generate*






Stresses for all Load Sets

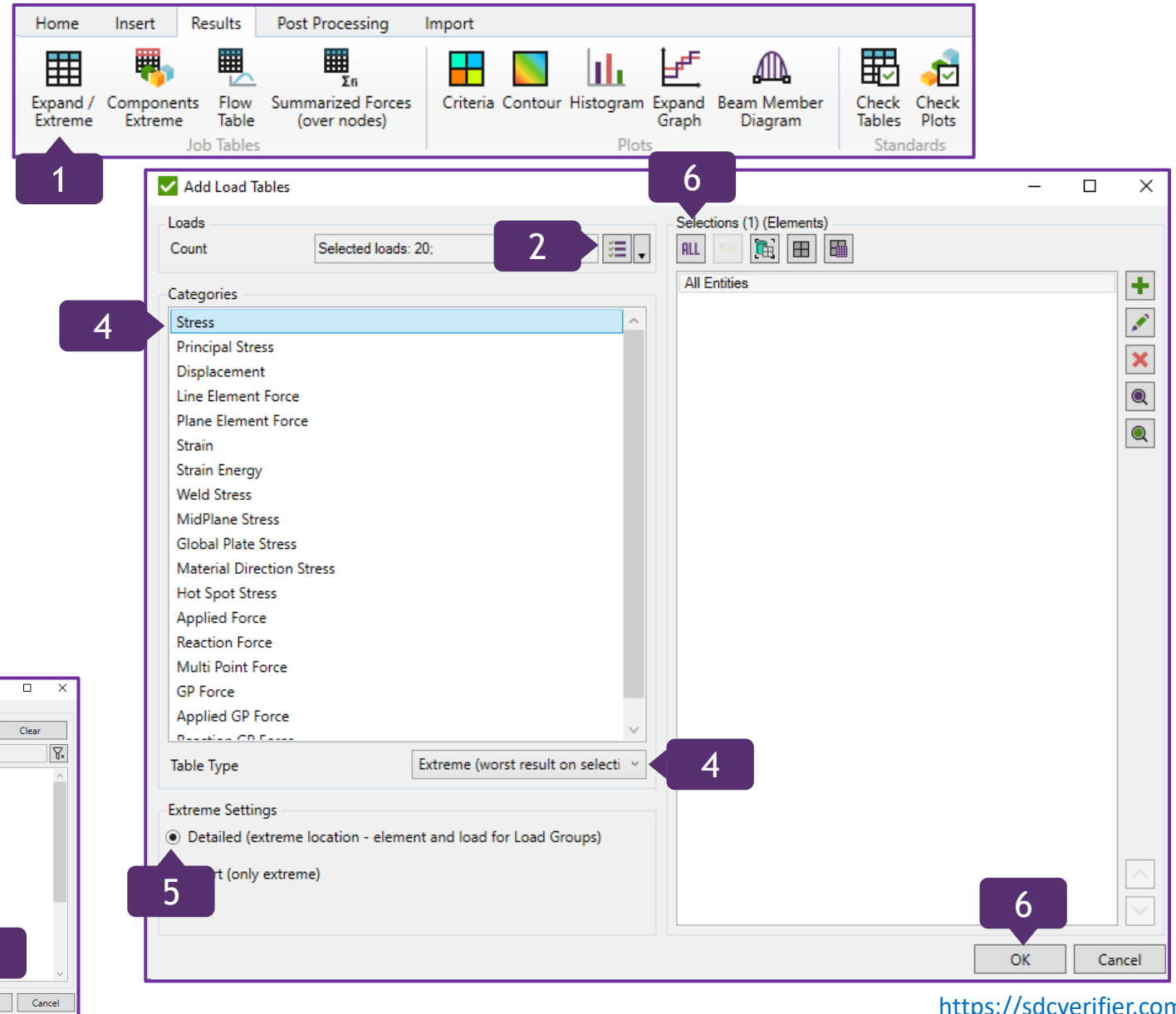
Stress (20 Loads, All Entities)							
Loads Count	20	Category		Stress			
Selection	All Entities	Type		Extreme			
Parameter	Abs						
Load	X [kPa]	Y [kPa]	Z [kPa]	XY [kPa]	YZ [kPa]	ZX [kPa]	Equivalent [kPa]
LS1...LC1s_Tip load.1	-0.18e+6			0.15e+6			0.29e+6
LS2...LC1s_Tip load.2	-0.18e+6			0.15e+6			0.29e+6
LS3...LC1s_Tip load.3	-0.17e+6			0.15e+6			0.30e+6
LS4...LC1s_Tip load.4	-0.17e+6			0.15e+6			0.30e+6
LS5...LC1s_Middle Bridge.1	-0.12e+6			0.05e+6			0.12e+6
LS6...LC1s_Middle Bridge.2	-0.12e+6			0.05e+6			0.12e+6
LS7...LC1s_Middle Bridge.3	-0.12e+6			0.05e+6			0.12e+6
LS8...LC1s_Middle Bridge.4	-0.12e+6			0.05e+6			0.12e+6
LS9...LC1s_Backside.1	-0.14e+6			0.05e+6			0.14e+6
LS10...LC1s_Backside.2	-0.14e+6			0.05e+6			0.14e+6
LS11...LC1s_Backside.3	-0.14e+6			0.05e+6			0.14e+6
LS12...LC1s_Backside.4	-0.14e+6			0.05e+6			0.14e+6
LS13...LC1s_At_forestay.1	-0.15e+6			0.12e+6			0.25e+6
LS14...LC1s_At_forestay.2	-0.15e+6			0.12e+6			0.25e+6
LS15...LC1s_At_forestay.3	-0.15e+6			0.12e+6			0.25e+6
LS16...LC1s_At_forestay.4	-0.15e+6			0.12e+6			0.25e+6
LS17...LC1s_at_hinge_point.1	-5.57e+6			0.05e+6			5.57e+6
LS18...LC1s_at_hinge_point.2	-5.57e+6			0.05e+6			5.57e+6
LS19...LC1s_at_hinge_point.3	-5.56e+6			0.05e+6			5.56e+6
LS20...LC1s_at_hinge_point.4	-5.56e+6			0.05e+6			5.56e+6

Displacements for all Load Sets

Displacement (20 Loads, All Entities)									
Loads Count	20	Category		Displacement					
Selection	All Entities	Type		Extreme					
Parameter	Abs								
Load	Ux [mm]	Uy [mm]	Uz [mm]	Usum [mm]	Rx	Ry	Rz	Rsum	
LS1...LC1s_Tip load.1	6.92e+1	9.54e+1	-3.51e+2	3.64e+2	0.00	0.00	0.00	0.00	
LS2...LC1s_Tip load.2	6.92e+1	-9.54e+1	-3.51e+2	3.64e+2	0.00	0.00	0.00	0.00	
LS3...LC1s_Tip load.3	6.32e+1	9.54e+1	-3.51e+2	3.64e+2	0.00	0.00	0.00	0.00	
LS4...LC1s_Tip load.4	6.32e+1	-9.54e+1	-3.51e+2	3.64e+2	0.00	0.00	0.00	0.00	
LS5...LC1s_Middle Bridge.1	-3.47e+1	4.98e+1	-7.94e+1	9.38e+1	0.00	0.00	0.00	0.00	
LS6...LC1s_Middle Bridge.2	-3.47e+1	-4.98e+1	-7.94e+1	9.38e+1	0.00	0.00	0.00	0.00	
LS7...LC1s_Middle Bridge.3	-3.80e+1	4.98e+1	-7.92e+1	9.38e+1	0.00	0.00	0.00	0.00	
LS8...LC1s_Middle Bridge.4	-3.80e+1	-4.98e+1	-7.92e+1	9.38e+1	0.00	0.00	0.00	0.00	
LS9...LC1s_Backside.1	-3.35e+1	4.19e+1	-1.42e+2	1.48e+2	0.00	0.00	0.00	0.00	
LS10...LC1s_Backside.2	-3.35e+1	-4.19e+1	-1.42e+2	1.48e+2	0.00	0.00	0.00	0.00	
LS11...LC1s_Backside.3	-3.69e+1	4.15e+1	-1.40e+2	1.46e+2	0.00	0.00	0.00	0.00	
LS12...LC1s_Backside.4	-3.69e+1	-4.15e+1	-1.40e+2	1.46e+2	0.00	0.00	0.00	0.00	
LS13...LC1s_At_forestay.1	5.56e+1	8.64e+1	-2.38e+2	2.53e+2	0.00	0.00	0.00	0.00	
LS14...LC1s_At_forestay.2	5.56e+1	-8.64e+1	-2.38e+2	2.53e+2	0.00	0.00	0.00	0.00	
LS15...LC1s_At_forestay.3	4.96e+1	8.64e+1	-2.38e+2	2.53e+2	0.00	0.00	0.00	0.00	
LS16...LC1s_At_forestay.4	4.96e+1	-8.64e+1	-2.38e+2	2.53e+2	0.00	0.00	0.00	0.00	
LS17...LC1s_at_hinge_point.1	-6.00e+2	1.14e+4	-1.72e+2	1.14e+4	0.00	0.00	0.00	0.00	
LS18...LC1s_at_hinge_point.2	-6.00e+2	-1.14e+4	-1.72e+2	1.14e+4	0.00	0.00	0.00	0.00	
LS19...LC1s_at_hinge_point.3	-6.04e+2	1.14e+4	-1.73e+2	1.14e+4	0.00	0.00	0.00	0.00	
LS20...LC1s_at_hinge_point.4	-6.04e+2	-1.14e+4	-1.73e+2	1.14e+4	0.00	0.00	0.00	0.00	

# Add Extreme Stress Tables

- 1 In **Results** section, press **Expand/Extreme**
- 2 In **Loads Count**, press  to select Loads
- 3 Select all **Load Sets (LS)**, excluding other options; press  and then press OK
- 4 Categories: **Stress**;  
Table Type: **Extreme (worst results on selection)**
- 5 Extreme Settings: **Detailed (extreme location - element and load for Load Group)**
- 6 Press  to add **All Entities**;  
Press **OK**



# Copy Table to Load Group

1

Expand *Load Sets* => *Load Set '20..LC1s\_at\_hinge\_point.4'* and execute right click on *Stress (LS20, All Entities)*

2

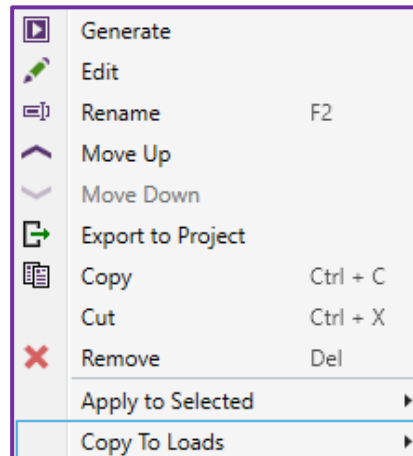
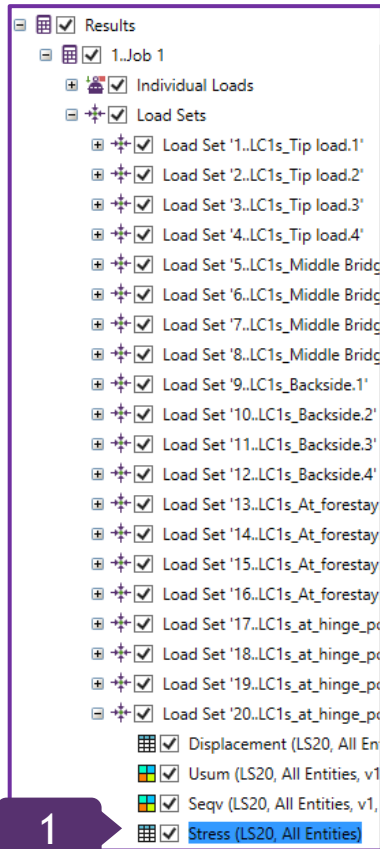
Select *Copy To Loads* and press *Result Items*

3

Select all *Load Groups (LG)*, excluding other options

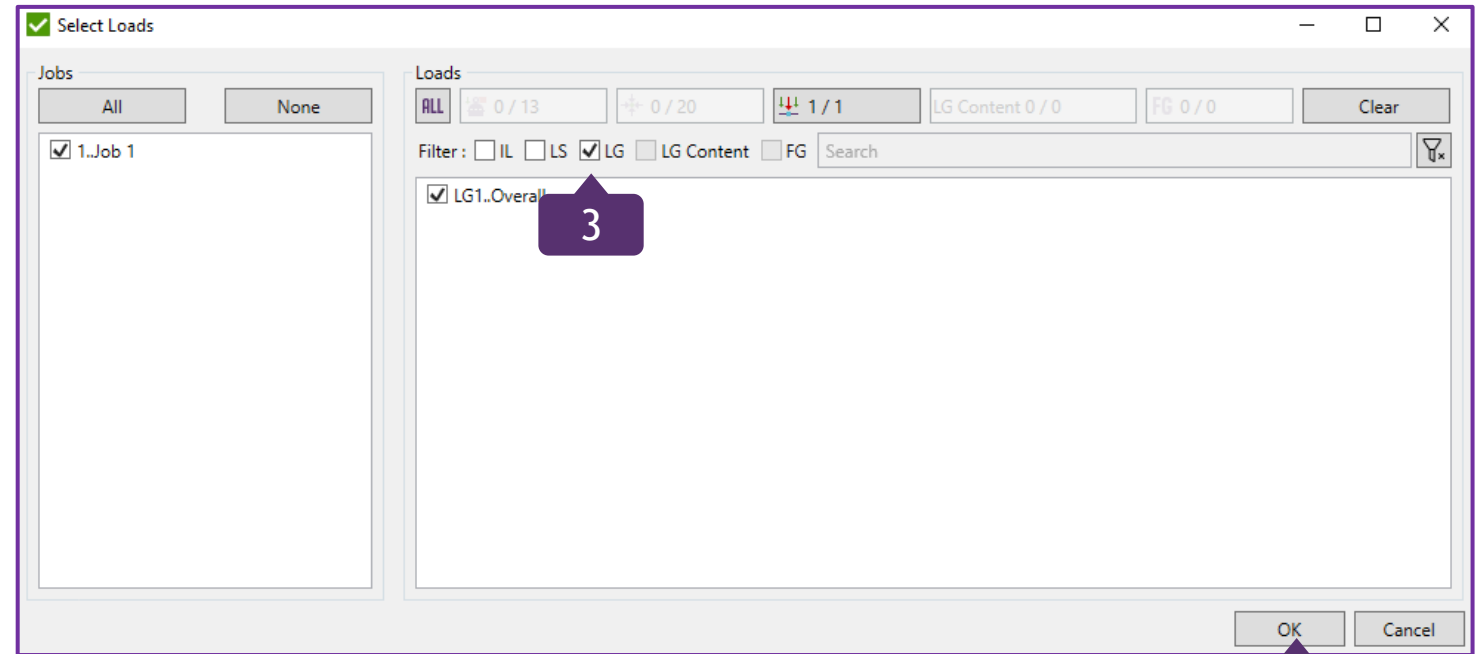
4

Press *OK*



2

Result Items



4

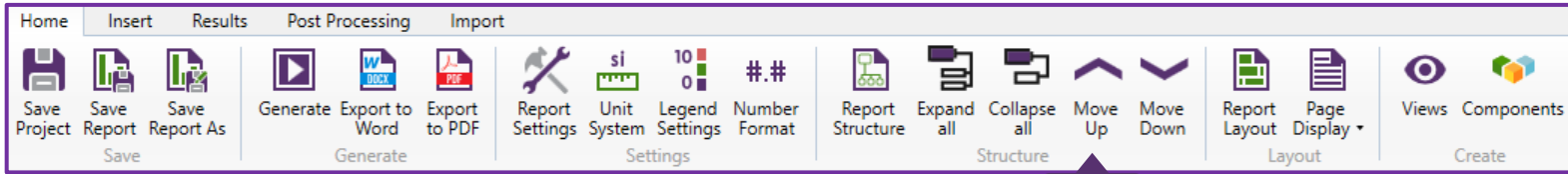
# Move Items in the Structure

1

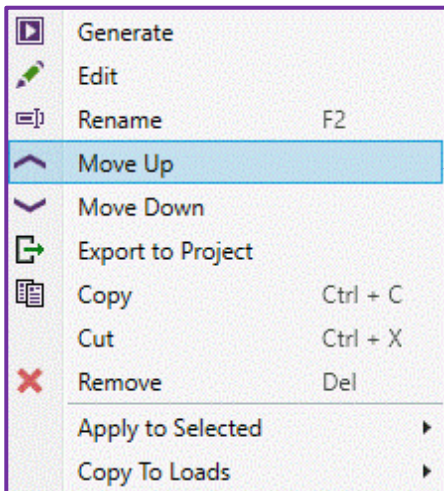
Expand *Load Groups* => *Load Group '1..Overall'* and select *Displacement (LG1, All Entities)*

2

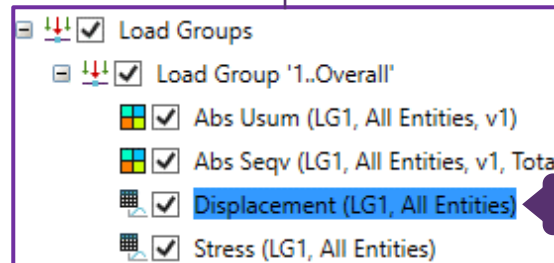
In *Home* section of the Ribbon, press *Move Up*



2

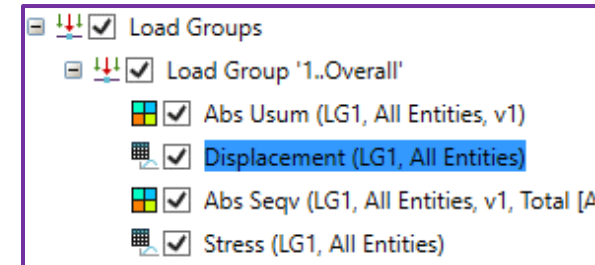


An alternative method of moving items up or down is to execute right click on Displacement (LG1, All Entities) and select Move Up/Down.



1

The result of moving the Item Up.



The navigation of Move Up and move Down functionality is also possible by using Ctrl + Up and Ctrl + Down key buttons combination.

# Copy Table for Load Group

1

Expand *Load Sets* => *Load Set '5..LC1s\_Middle Bridge.1'* and execute right click on *Stress (LS5, All Entities)*

2

Select *Copy*

3

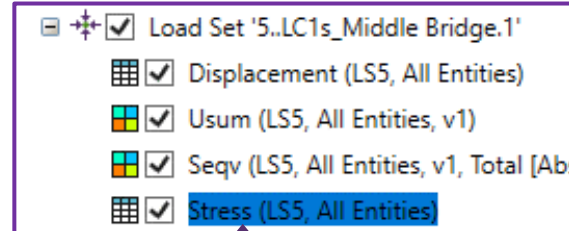
Execute right click on *Load Set '19..LC1s\_at\_hinge\_point.3'*

4

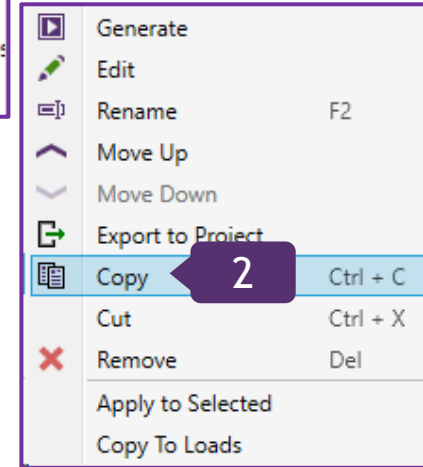
Select *Paste*

5

Insert Plot: Yes;  
View for the second Stress Table: *2..Isometric with filled edges*

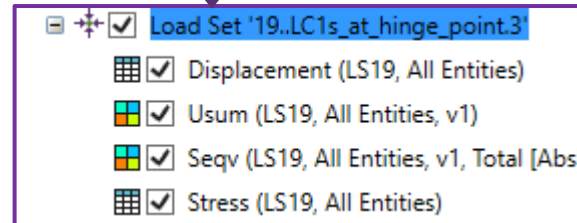


1

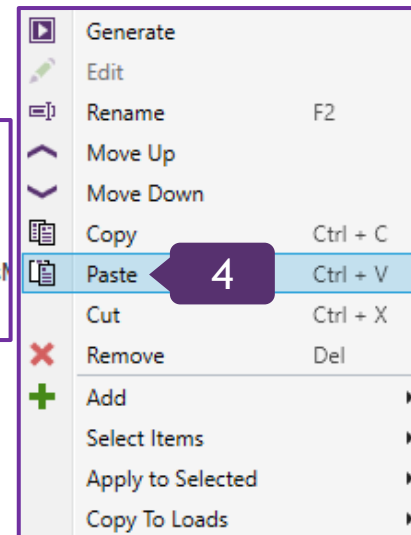


2

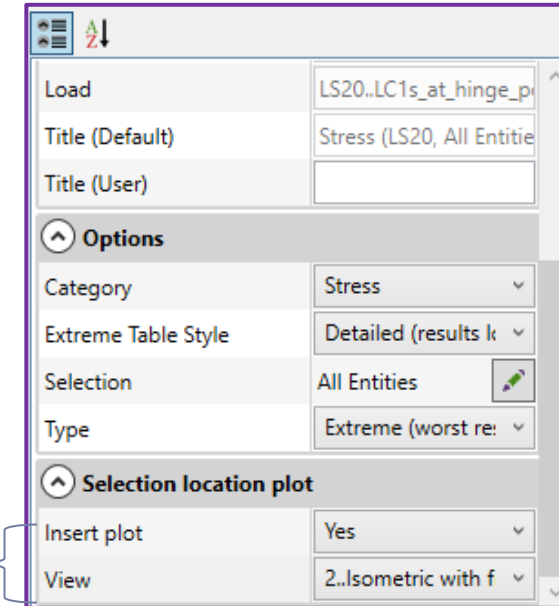
3



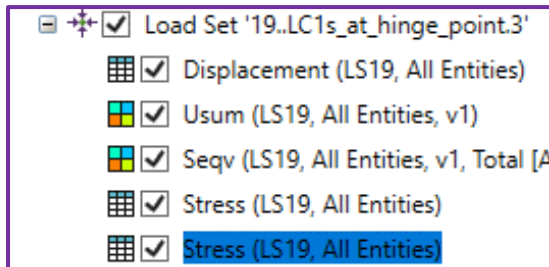
4





5

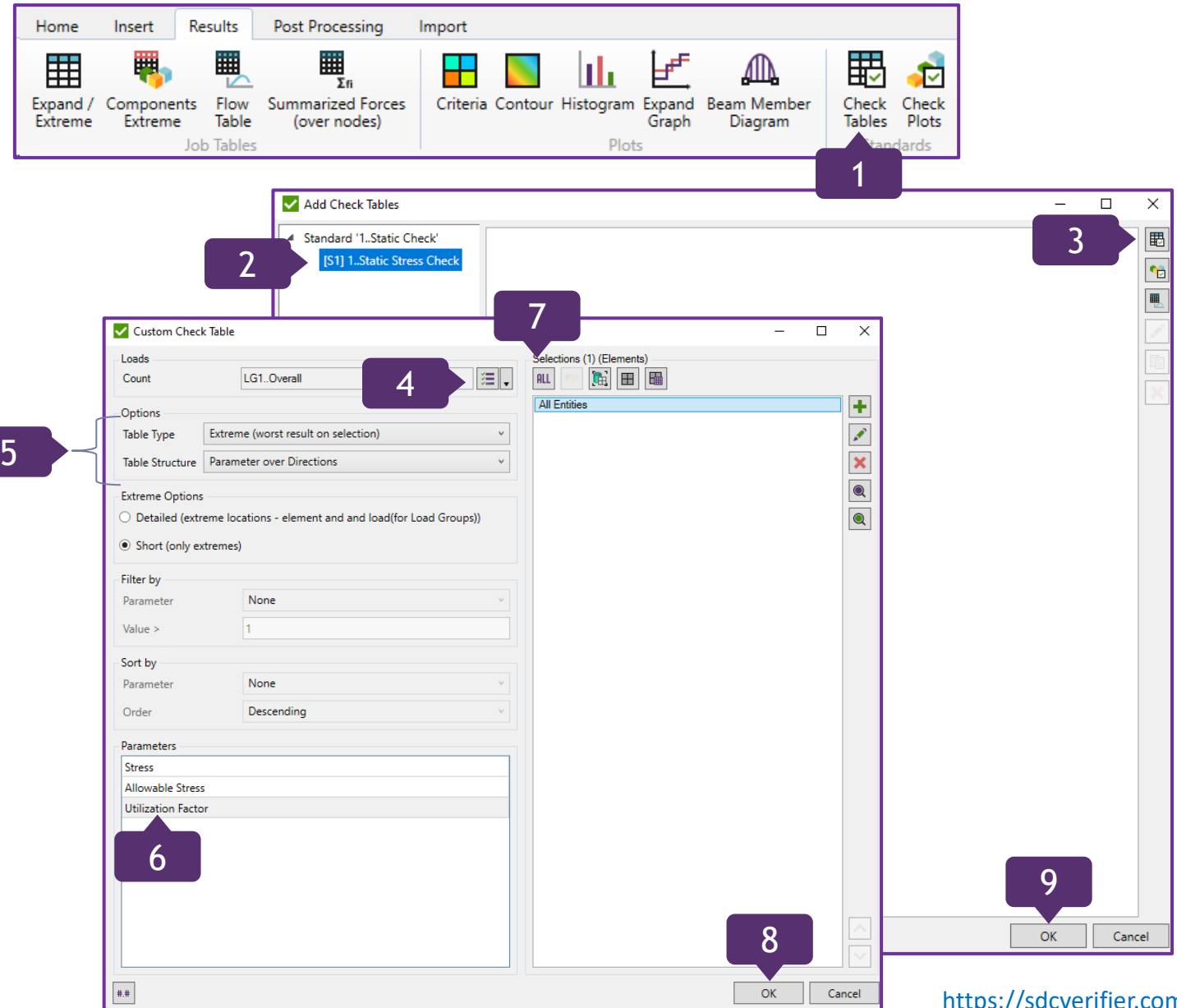


The result





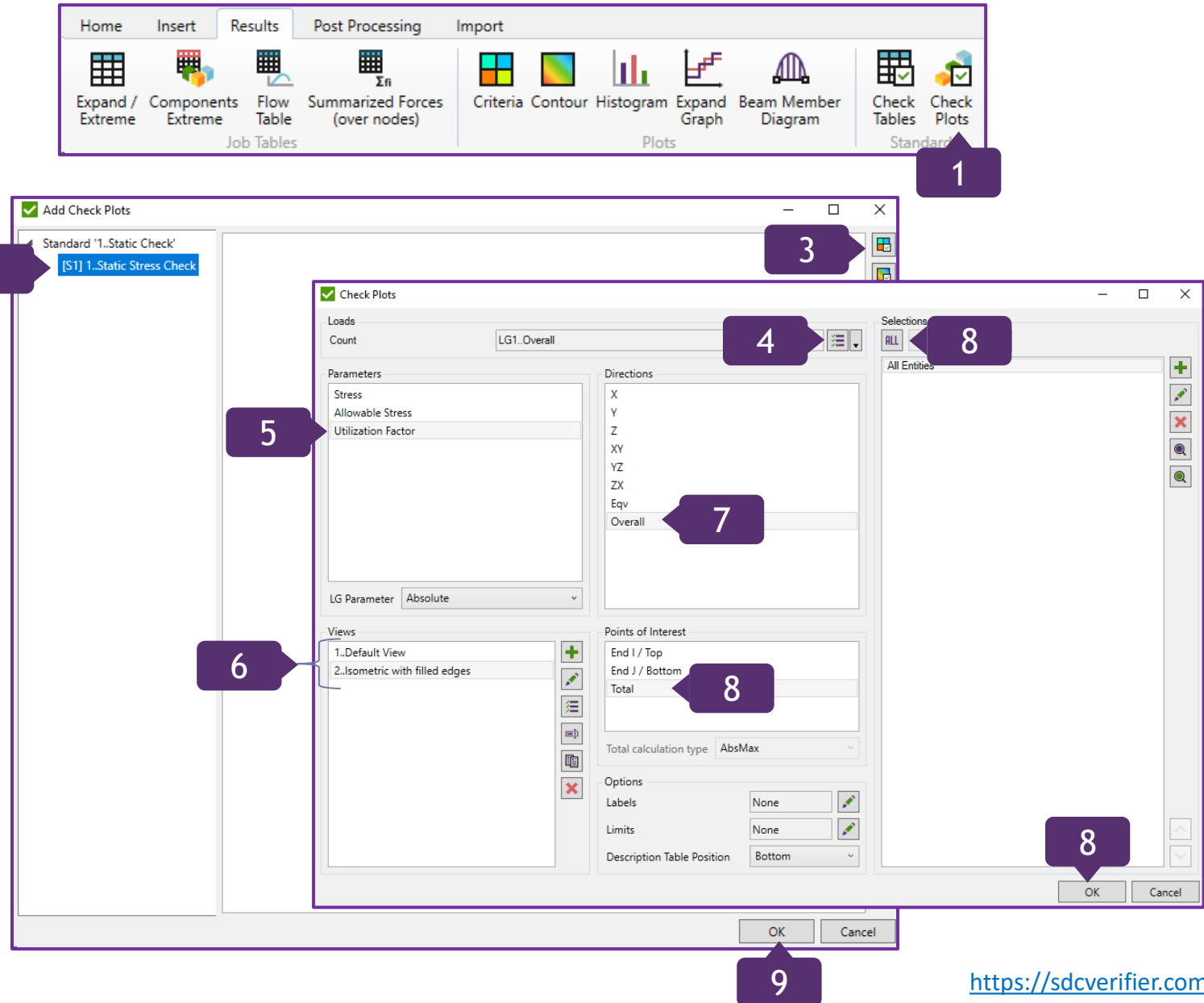
# Add Table for Static Stress Check

- 1 In Results section, select *Check Tables*
- 2 Expand *Standard '1..Static Check'* and select *[S1] 1..Static Stress Check*
- 3 Press  to add *Check Table*
- 4 In Loads Count, select *LG1..Overall*
- 5 Table Type: *Extreme (worst result on selection)*; Table Structure: *Parameter over Directions*
- 6 Parameters: *Utilization Factor*
- 7 Press  to add *All Entities* selection
- 8 Press *OK*
- 9 Press *OK*



# Add Plot for Static Stress Check

- 1 In Results section, select *Check Plots*
- 2 Expand *Standard '1..Static Check'* and select *[S1] 1..Static Stress Check*
- 3 Press  to add *Criteria Plot*
- 4 In Loads Count, select *LG1..Overall*
- 5 Parameters: *Utilization Factor*
- 6 Views: *1..Default View* and *2..Isometric with filled edges*
- 7 Directions: *Overall*;  
Points of Interest: *Total*
- 8 Press  to add *All Entities* selection;  
Press *OK*
- 9 Press *OK*





# Generate Static Stress Check Results

1 In the *Report Structure*, expand *Results* => 1..Job 1 => Summary => Load Group '1..Overall'

2 Execute right click on 1..Static Check and select *Generate*

3 Break Page before: No

4 Execute right click on Abs Overall Utilization Factor (LG1, All Entities, v1, Total) and select *Generate*

Behavior

Break Page Before No

Enabled Yes

Data

Include Safety Factors No

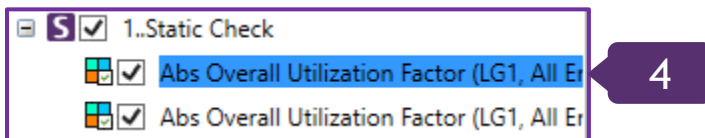
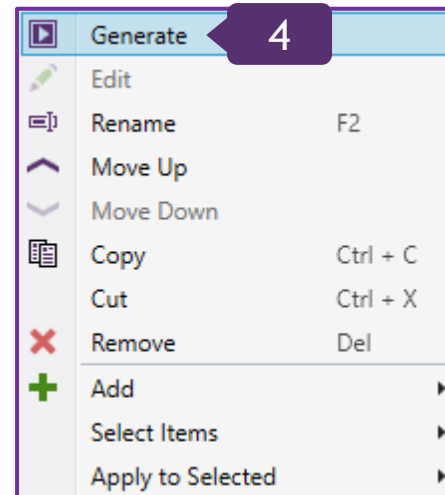
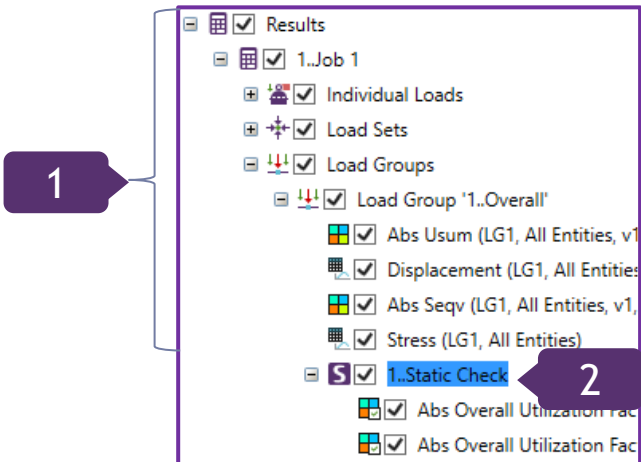
Last Time Generated 03/06/2024 16:17:52

Title (Default) 1..Static Check

Title (User)

Plot

Check on not covered elements No



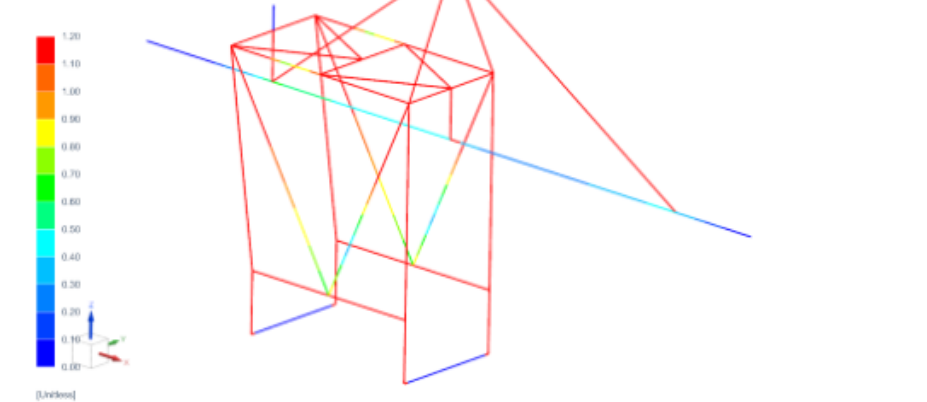
## 1..Static Check

### Unit System

Current Unit System = MmKS (Millimeter/Kg/Second). It is used in calculations for the following standards: API RP 2A, ISO 19902, Norsok N004, DIN 15018, FEM 1.001 and Eurocode3.

### Abs Overall Utilization Factor (LG1, All Entities, v1, Total)

Report Designer: s - SDC Temp Result  
Data: Static Step 1  
Standard: Eurocode3, Scalar  
Formula Used: StdExpression  
Min: 0.00, Max: 23.21, Units: Unitless



Check	[S1] 1..Static Stress Check	Point	Total
Load Group	LG1..Overall	Parameter	Absolute Overall Utilization Factor
Selection	All Entities	View	1..Default View



# Add Governing Loads

1

In Post Processing section of the Ribbon,  
press **Governing Loads**;  
Select **Add Governing Loads Table**

2

Result: *from Load*


3

Category: *Stress*;  
Direction: *Equivalent*

4

Press  to select Loads;  
Select **LG1..Overall** and press **OK**

5

Press  and in Result Criteria select  
**Abs**; press **OK**

6

Selections: *All Entities*

7

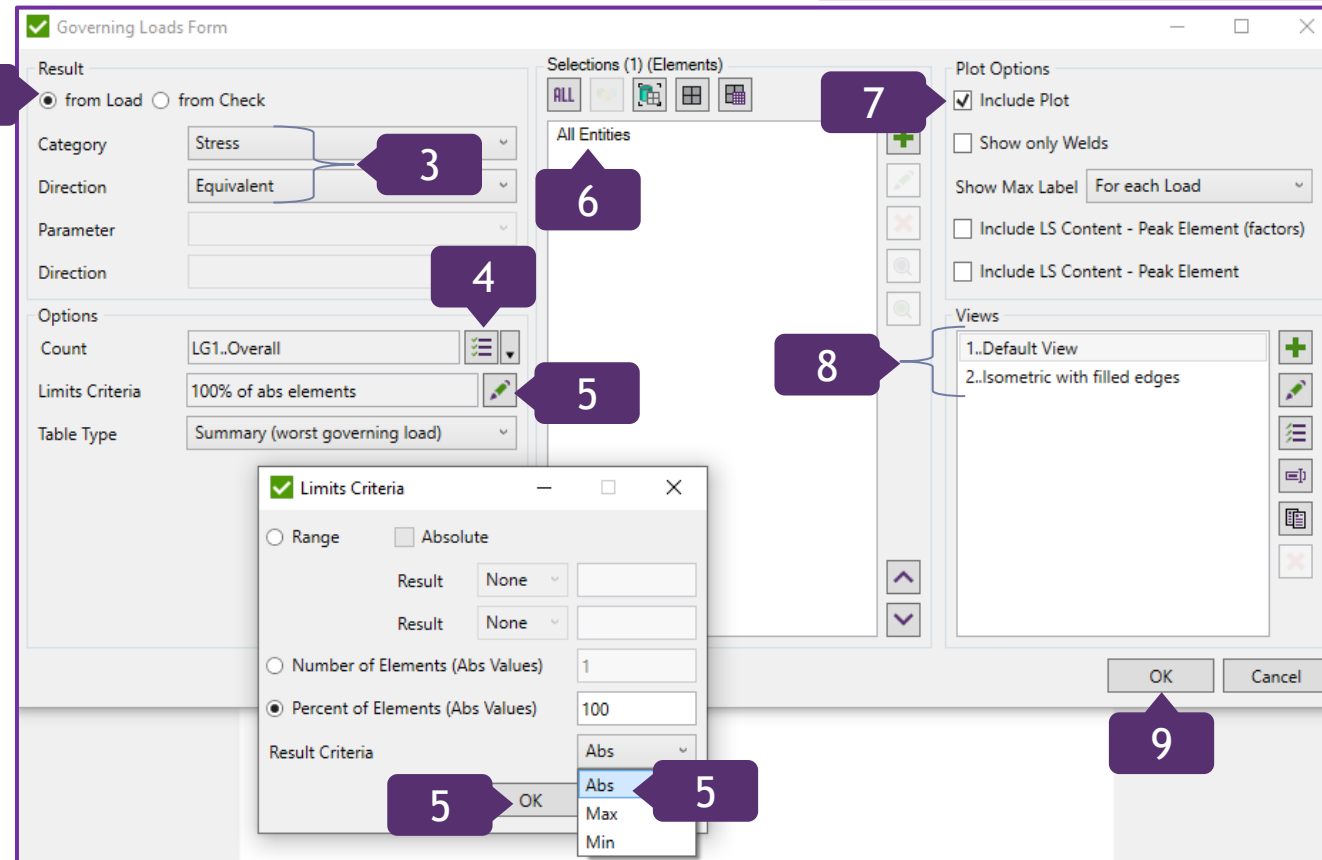
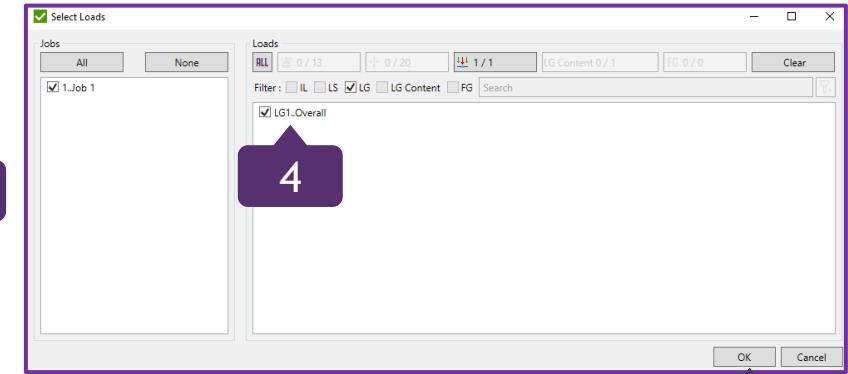
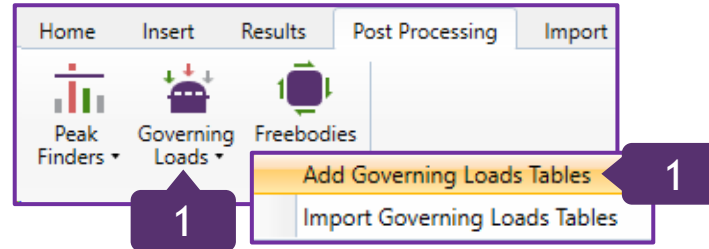
Plots Options: Include Plot is **ON**

8

Views: *1..Default View* and  
*2..Isometric with filled edges*

9

Press **OK**



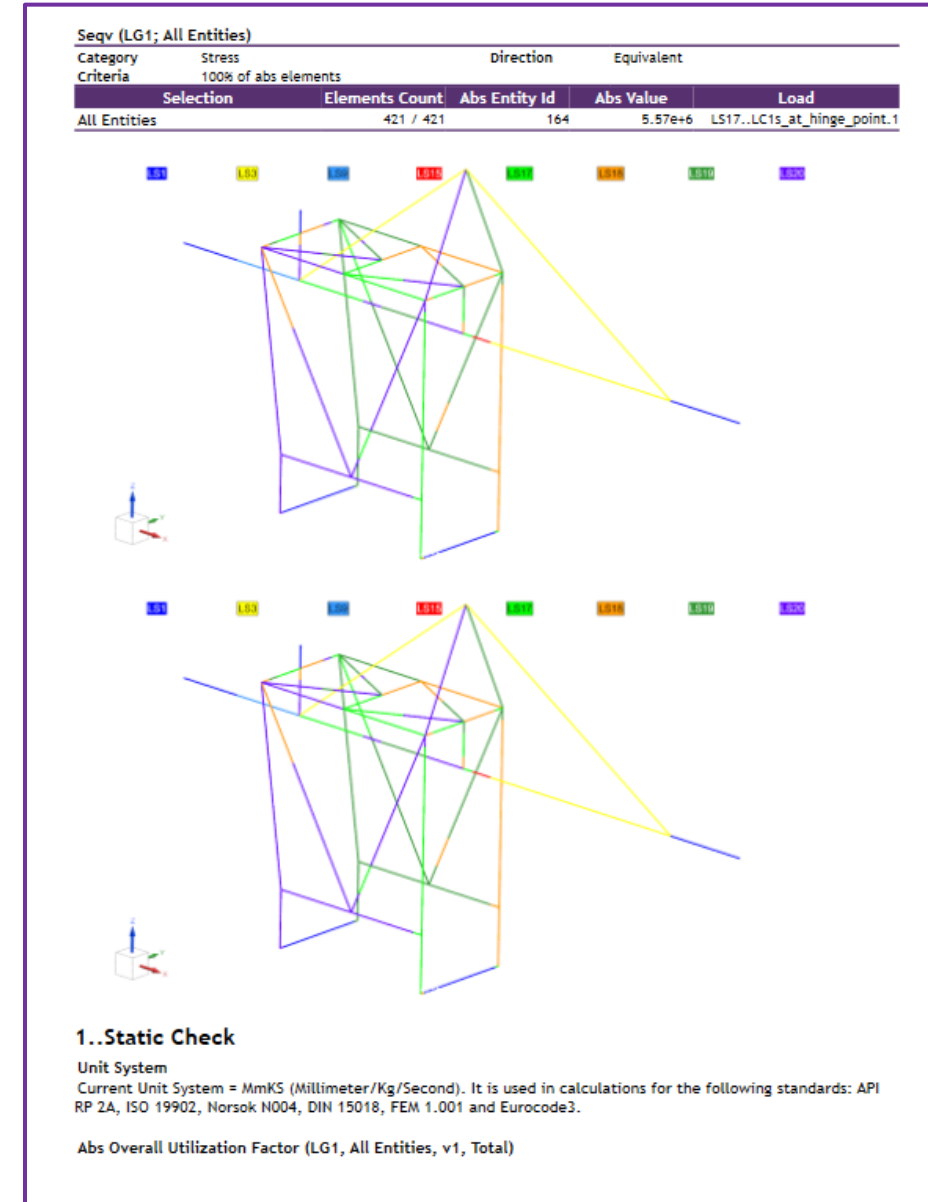
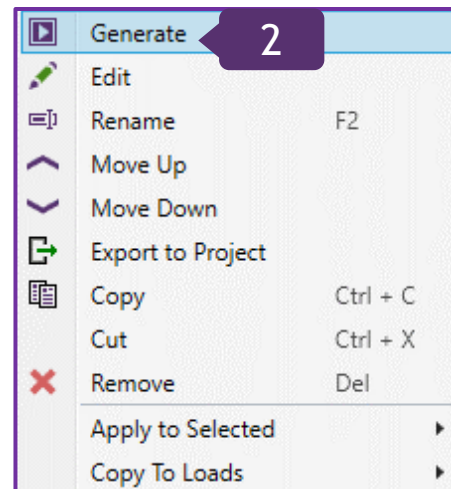
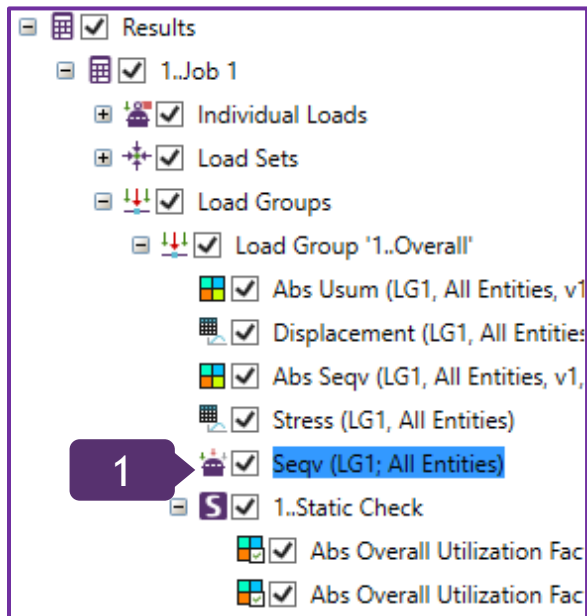
# Generate Governing Loads Results

1

In the *Report Structure => Results => 1..Job 1*  
*=> Summary => Load Group '1..Overall', select*  
*Seqv (LG1; All Entities)*

2

Execute right click on *Seqv (LG1; All*  
*Entities)* and select *Generate*



# Add Conclusion

1 In Insert section of the Ribbon, press *Text*

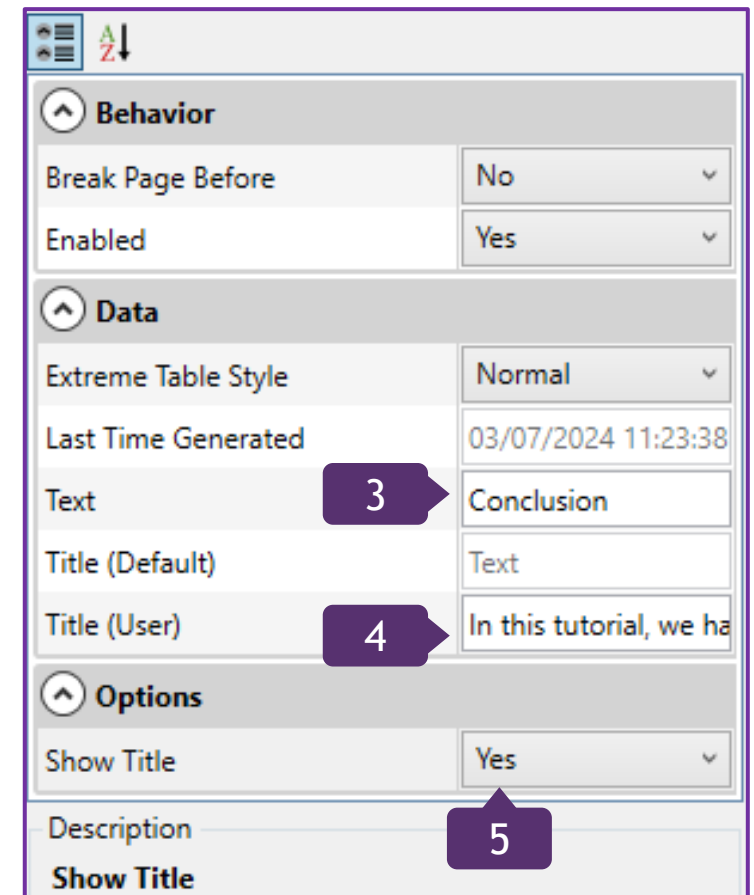
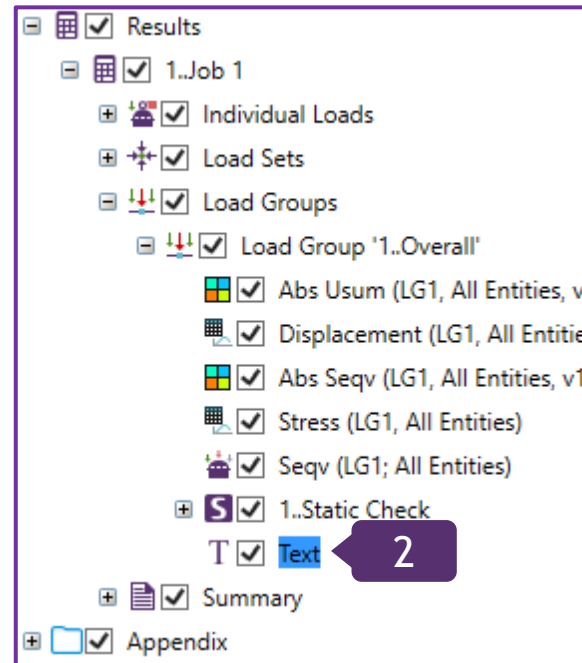
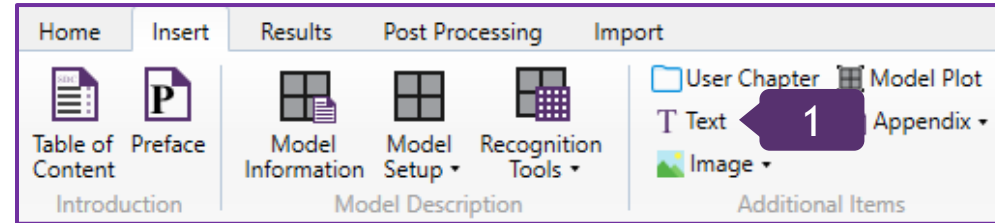
2 In the *Report Structure* => *Results* => *1..Job 1* => *Summary* => Load Group '1..Overall', select *Text*

3 Text: *Conclusion*

4 Title (User): *In this tutorial, we have described the Report functions*

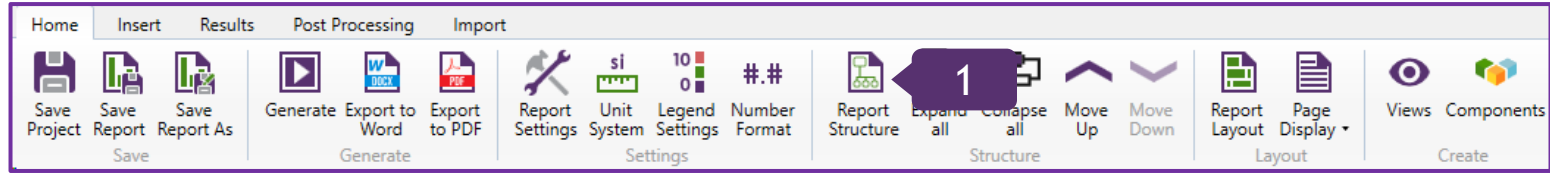
5 Show Title: *Yes*

The item Text will be inserted under the previously chosen Item.



1

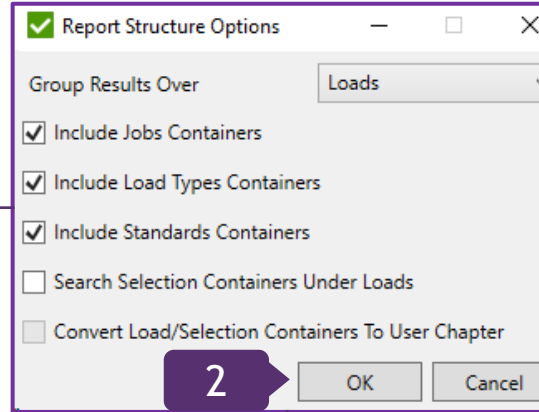
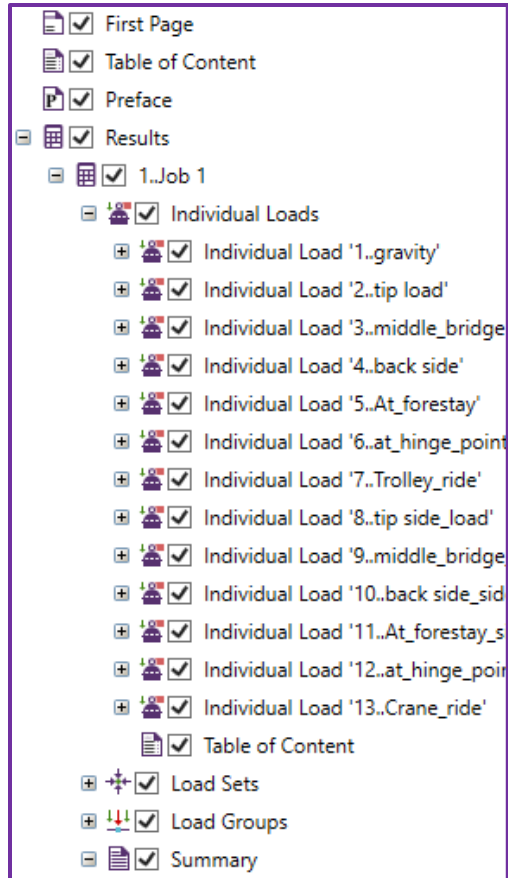
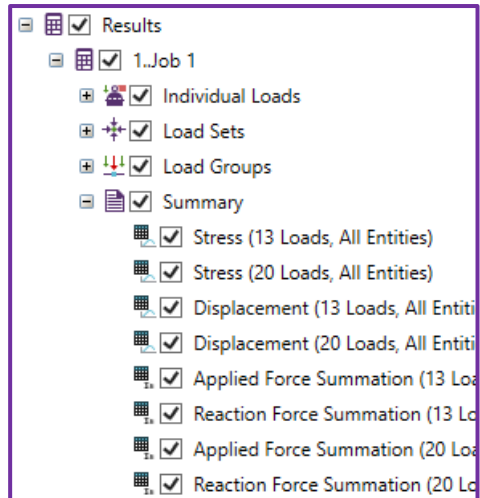
In Home section of the Ribbon, press **Report Structure**



2

Select the required options and press **OK**

When Table/Plot is edited and Load/Selection is changed, the item is moved under correspondent Load/selection automatically. Moreover, when the item is dropped under Load/Selection, its Load/Selection is updated as well.



Include Load Types Containers

Include Job Containers

Tables/Plots with multiple loads of the same Job are placed under Job Summary Chapter (for loads from different Jobs in Summary under Result chapter).

# Import from Word Document

1

In Import section of the Ribbon, press *Word Document*

2

In the *Report Structure => Results => 1..Job 1 => Summary => Load Group '1..Overall'*, select *Import document*

3

In Home section of the Ribbon, press *Export to Word*

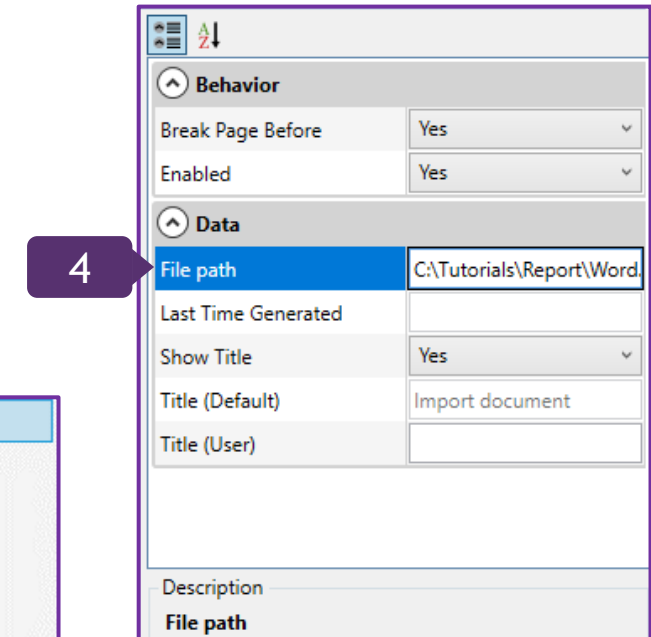
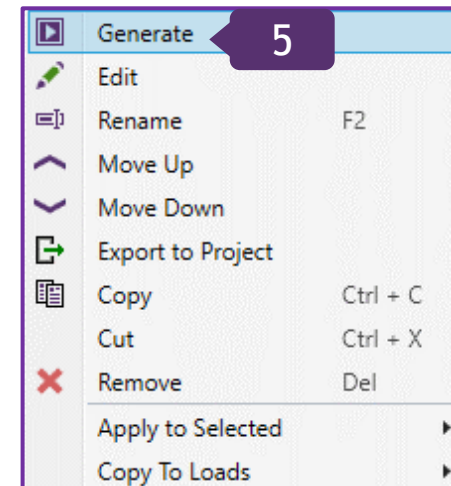
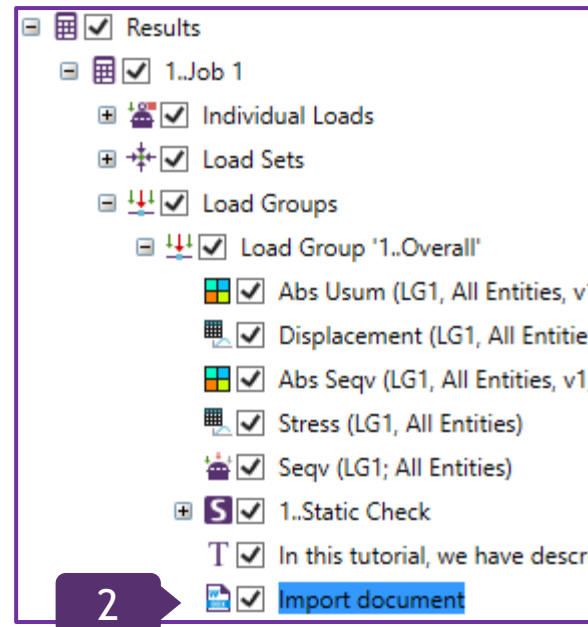
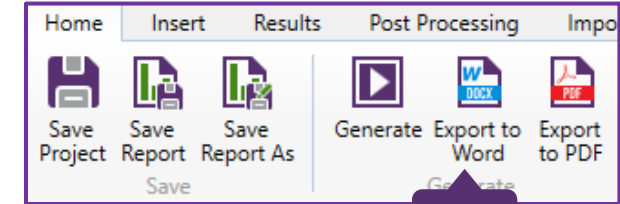
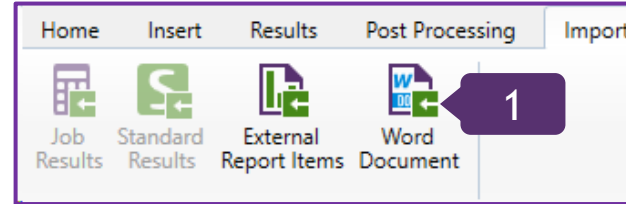
4

In display properties, set the file path

5

Press *Generate*

Note: the word file will be displayed after Report is exported to Word or PDF.



# Generated Report

- 1 Execute right click on *First Page* and select *Edit*
- 2 Select *Default View* and press *OK*
- 3 In Home section of the Ribbon, press *Generate*
- 4 After generation is finished, press *Export to Word*

