



Tutorial

Report Designer

5 Jan 2021
Version 2020.0.2

Report designer gives a possibility to completely control structure of your report and easily preview and modify it.

This tutorial demonstrates how to build reports using Report Designer:

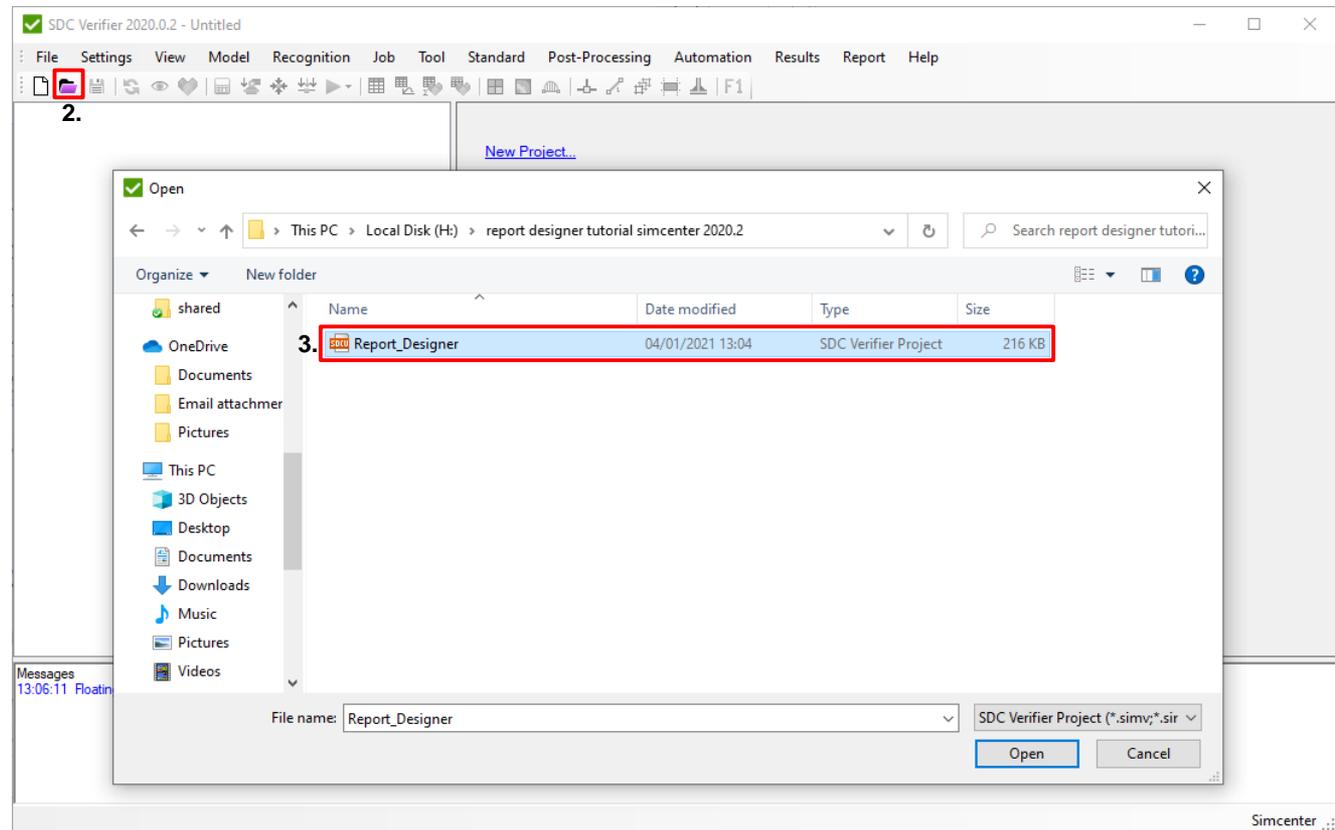
- Open predefined project;
- Model Setup Report (First Page, Preface, Materials, Properties, Fem Loads and Constraints);
- Result Report (Content items, Predefined Tables, Add Plots and Tables);
- Number Format, Legend Settings;
- Tables and Plots for Static Stress Check;

Open Project

1 Launch **SDC Verifier** 

2 Execute *File - Open Project*.

3 Project: **Report_Designer**



Predefined project



The screenshot displays the SDC Verifier interface. On the left is a project tree with the following structure:

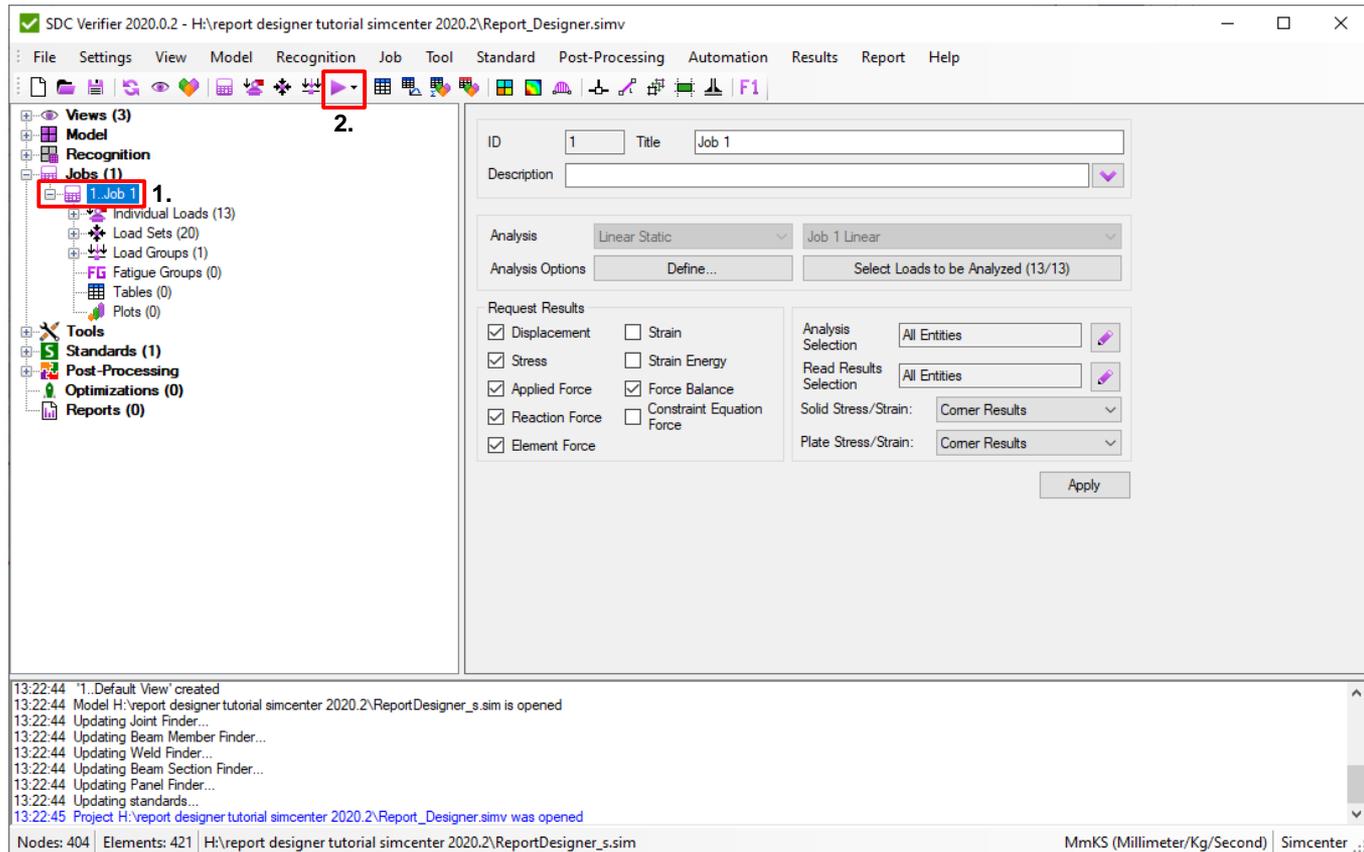
- Views (3)
- Model
- Recognition
- Jobs (1)
 - 1..Job 1
 - Individual Loads (13)
 - Load Sets (20)
 - 1..LC1s_Tip load.1
 - 2..LC1s_Tip load.2
 - 3..LC1s_Tip load.3
 - 4..LC1s_Tip load.4
 - 5..LC1s_Middle Bridge.1
 - 6..LC1s_Middle Bridge.2
 - 7..LC1s_Middle Bridge.3
 - 8..LC1s_Middle Bridge.4
 - 9..LC1s_Backside.1
 - 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)
 - Fatigue Groups (0)
 - Tables (0)
 - Plots (0)

- Standards (1)
- 1..Static Check
 - Input
 - π Constants (0)
 - Types (1)
 - Characteristic (0)
 - Classifications (0)
 - Standard Tables (0)
 - Checks (1)
 - 1..Static Stress Check

This tutorial use predefined project with the following created data: individual loads, load sets, load groups and static stress check. The focus of this tutorial is on creating report.

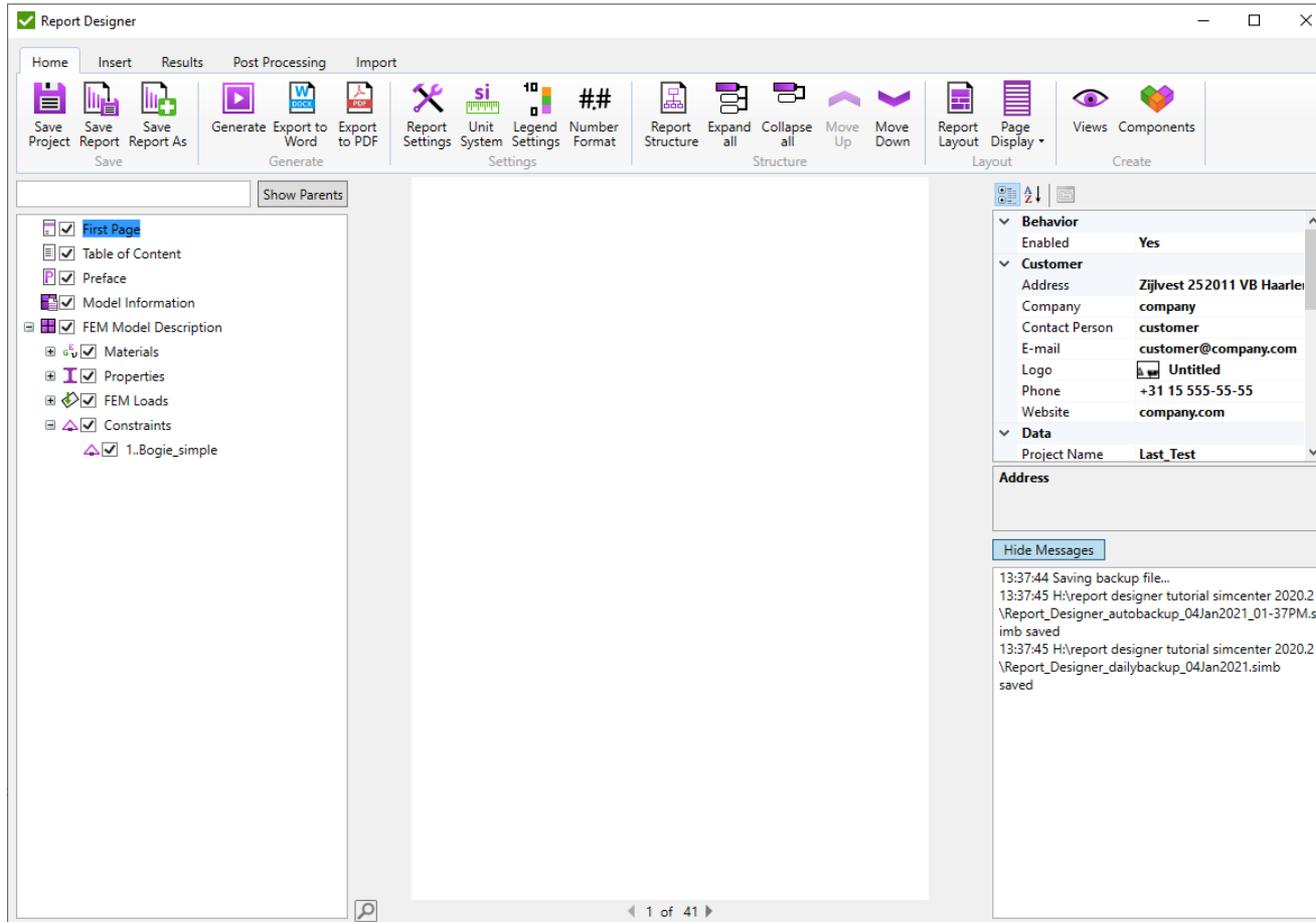
Analyze Job

- 1 Select **1..Job1** in the *Model Tree*.
- 2 Press  on toolbar to analyze job.



Report Designer Interface

Toolbar – contains main functions



Report Structure – displays structure of the report

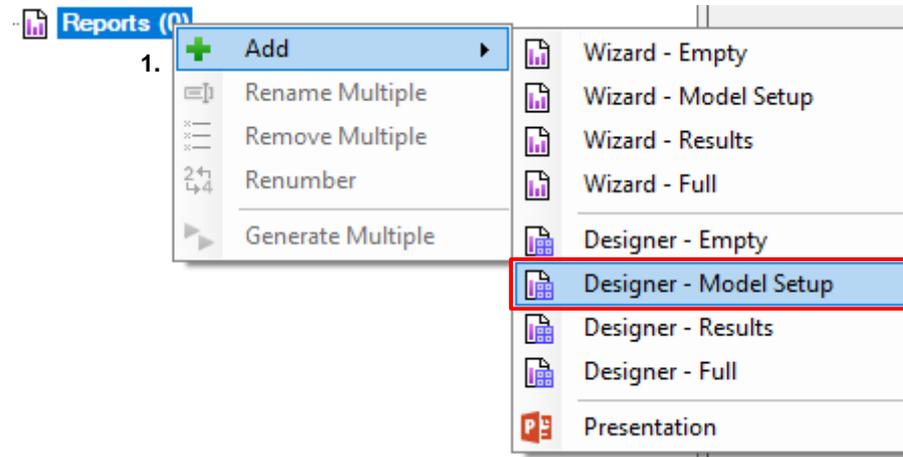
Displays properties of selected item. It is possible to modify them.

Report document

Add Model Setup Report

1

Execute Reports – Add – Designer - Model Setup from report context menu



There are 4 templates of reports:

Empty – only first page and preface items are included;

Model Setup – description of model data (materials, properties, components, boundary conditions) is included;

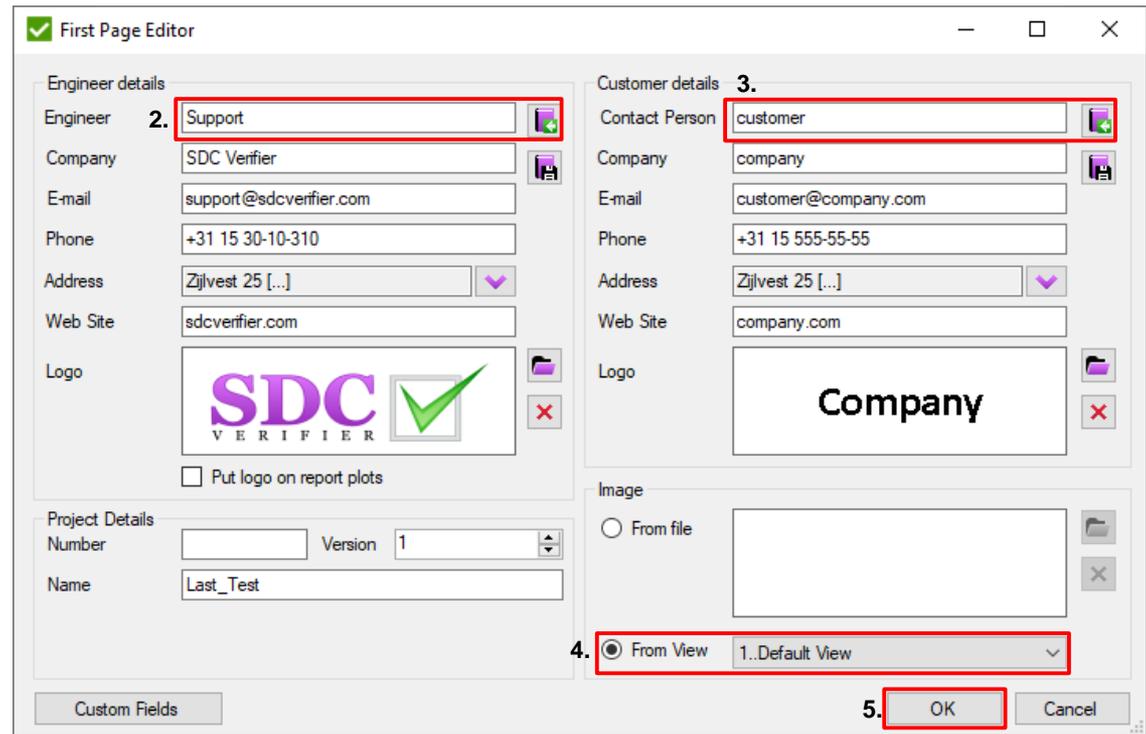
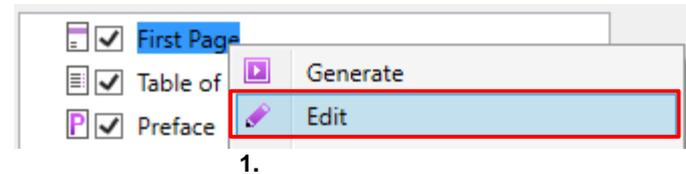
Results – for each load extreme displacement tables, stress and displacement plots are included. Predefined tables: sum of reaction forces, stresses/displacements summary tables;

Full – Model Setup + Results + all tables created in Job.

Editing First Page

- 1 Execute *Edit* from First Page context menu
- 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*.

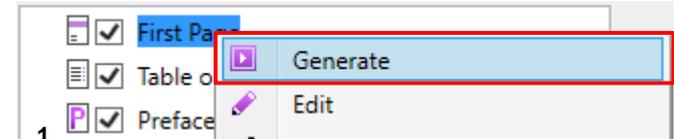
Note: For engineer and customer the default data from library is used. It is possible to fill in your data and store it to the library and reuse it in future projects.



Generate First Page

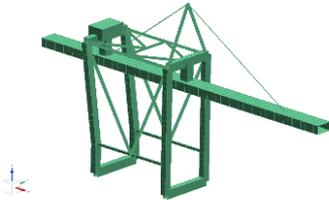
1

Execute *Generate* from First Page context menu



Model Setup

Report Designer Tutorial



Prepared by:
SDC Verifier
+31 15 30-10-310
sdoverifier.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Prepared for:
company
+31 15 555-55-55
company.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Engineer: Support
Customer: customer
Project Number:
Version: 1
Date: 04/01/2021

Model Setup

Report Designer Tutorial



Prepared by:
SDC Verifier
+31 15 30-10-310
sdoverifier.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Prepared for:
company
+31 15 555-55-55
company.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Engineer: Support
Customer: customer
Project Number:
Version: 1
Date: 04/01/2021



Company name and logo from engineer and customer are used in footer. All pages except first one have the footer.

Report in Report designer

Exported report to Microsoft Word

Generate Preface item

1 Select **Preface** item in report structure

2 Execute *Generate* from context menu

Preface

This document is generated with SDC Verifier 2020.0.2 and calculated with Simcenter v2020.2
Model File: H:\report designer tutorial simcenter 2020.2\ReportDesigner_s_sim
Project File: H:\report designer tutorial simcenter 2020.2\Report_Designer.simv
Report Profile: 1..Model Setup
Generation on: 1/4/2021 2:34:35 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

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

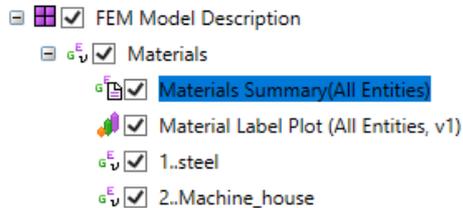
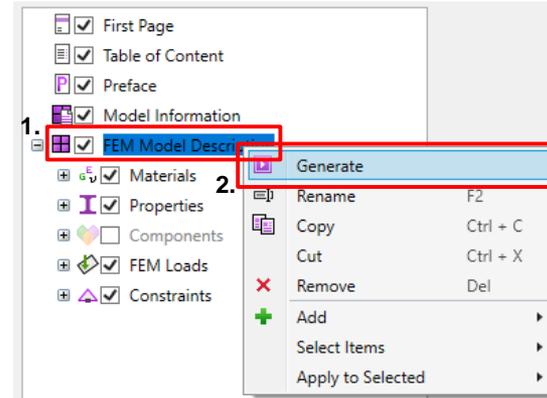
Description on current unit system. It has an influence on calculations according to some standards.

Introduction section – how to use and navigate in the report. By default is OFF.

Generate Model Setup items

1 Select **FEM Model Description** item in report structure

2 Execute *Generate* from context menu



FEM Model Description

This paragraph shows detailed or brief model overview.

Materials

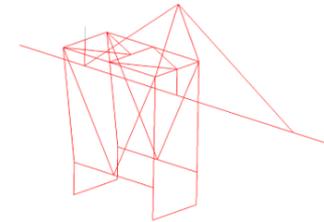
This paragraph contains materials information.

Materials Summary(All Entities)

Title	Elements	Mass [kg]	Mass Density [kg/m ³]	Gravity Center [m]
1..steel	420	1937142.8	9.812.50	[-13.85; 0.00; 34.29]
2..Machine_house	1	79999.9	333.33	[-39.48; 0.00; 52.00]
Overall	421	2017142.7		[-14.70; 0.00; 35.00]

1..steel

Property	Value
Elements	420
Mass [kg]	1937143.0
Gravity Center [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+8
Yield Stress [kPa]	0.24e+8



Material Summary – mass and gravity center overview over materials

Detailed Material description with plot

Material Options

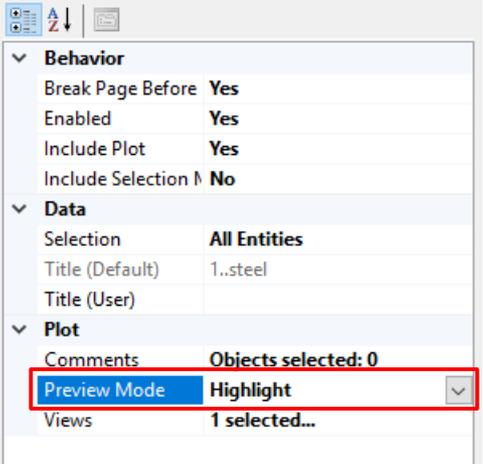
1

Select Material: **1..Steel** in report structure

2

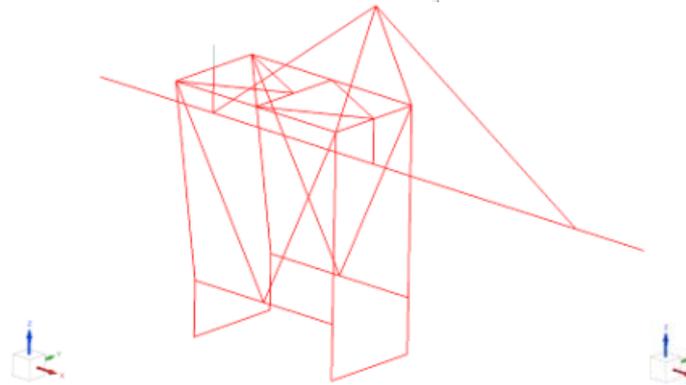
Preview Mode: **Display Only Selected**

2.



Behavior	
Break Page Before	Yes
Enabled	Yes
Include Plot	Yes
Include Selection	No
Data	
Selection	All Entities
Title (Default)	1..steel
Title (User)	
Plot	
Comments	Objects selected: 0
Preview Mode	Highlight
Views	1 selected...

It is possible to exclude plot using option – Include Plot.
Using View option it is possible to change display options for the plot.



Preview Mode: Highlight



Preview Mode: Display Only Selected

Create View for Fem Loads

1 Press  on the toolbar to open View Manager

2 Select View: **1..Default View**

3 Press  to copy view.

4 Select: **4..Default View.(Copy)**

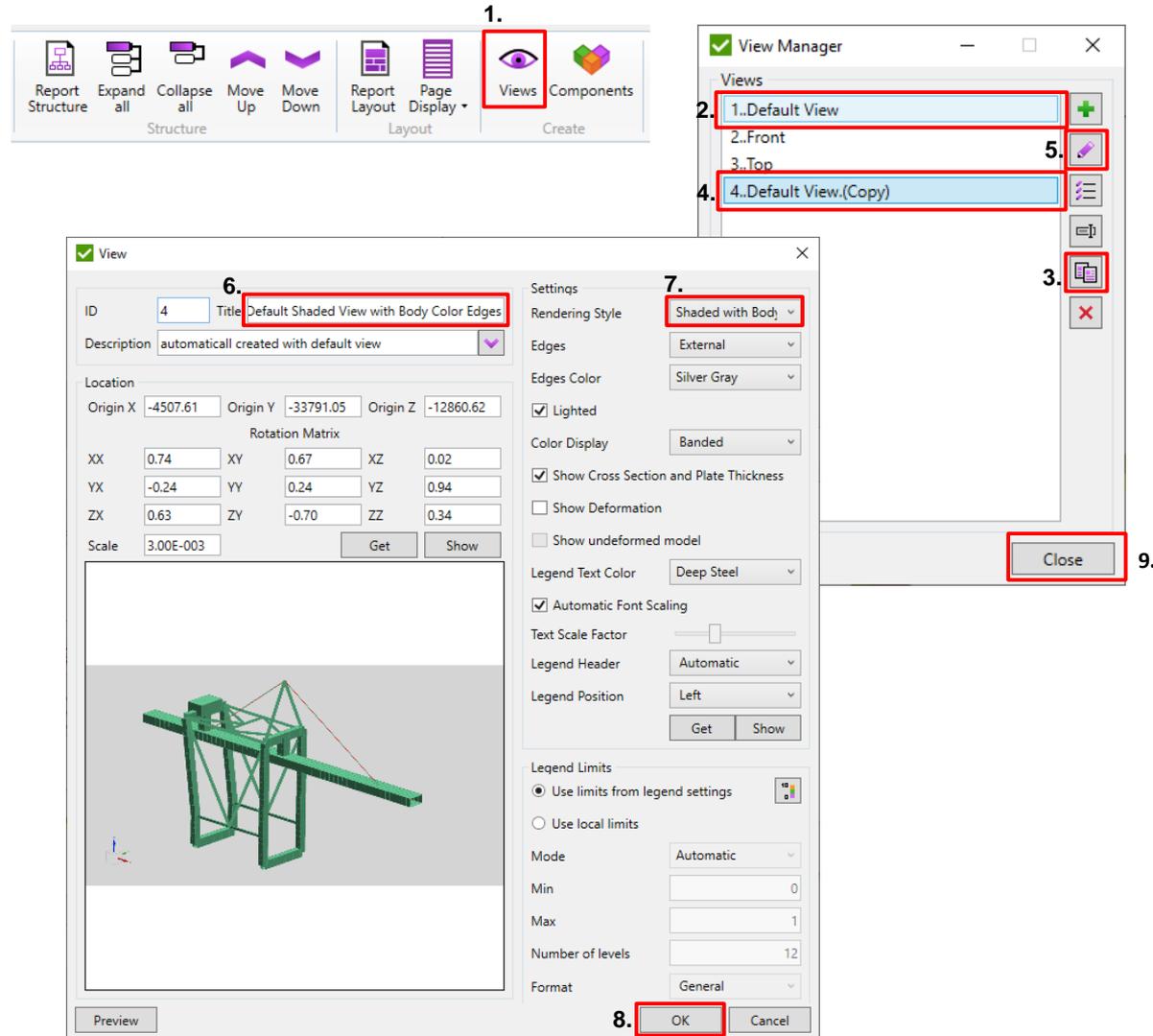
5 Press  to edit view

6 Title: **Default Shaded View with Body Color Edges**

7 **Rendering Style: Shaded with Body Color Edges**

8 Press **OK**.

9 Press **Close**.



The screenshot displays the software interface with two windows open: 'View Manager' and 'View'. The 'View Manager' window shows a list of views: '1..Default View', '2..Front', '3..Top', and '4..Default View.(Copy)'. The 'View' dialog box is open for view ID 4, showing the title 'Default Shaded View with Body Color Edges' and the rendering style 'Shaded with Body'. A 3D model of a truss structure is visible in the preview window. Numbered callouts (1-9) indicate the steps for creating and editing the view.

1.  on the toolbar to open View Manager

2. Select View: **1..Default View**

3. Press  to copy view.

4. Select: **4..Default View.(Copy)**

5. Press  to edit view

6. Title: **Default Shaded View with Body Color Edges**

7. **Rendering Style: Shaded with Body Color Edges**

8. Press **OK**.

9. Press **Close**.

Apply View to all Fem Loads

1. Select **FEM Loads** in report structure

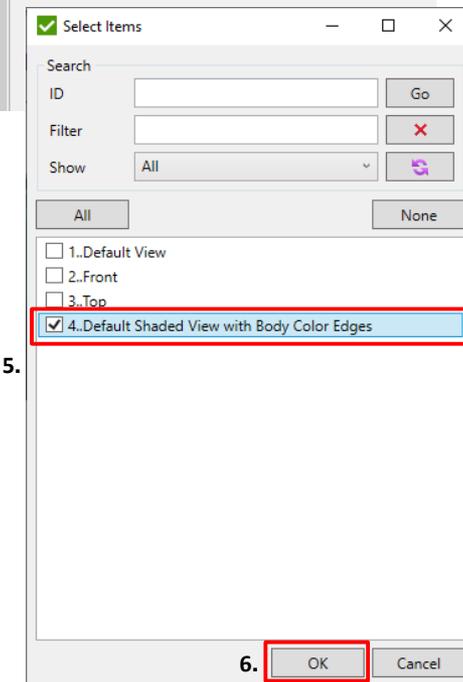
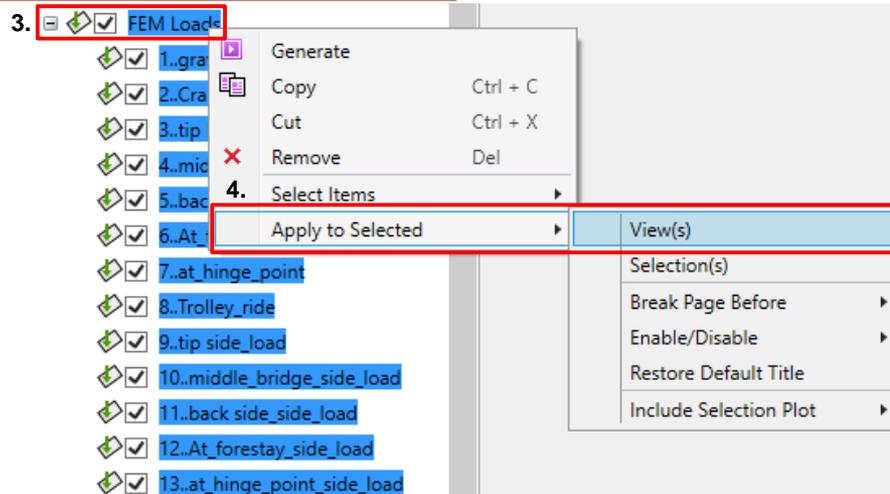
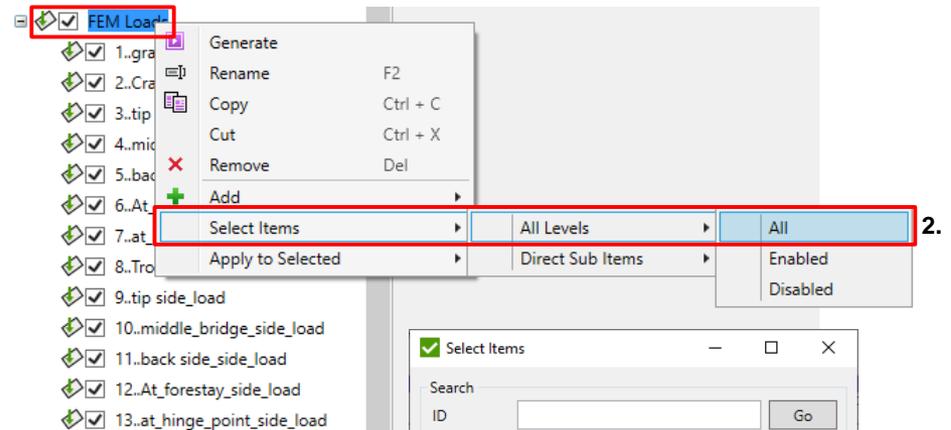
2. Execute *Select Items – All levels – All*

3. Select **FEM Loads** in report structure

4. Execute – *Apply to selected – Views*

5. Select: Default Shaded View with Body Color Edges

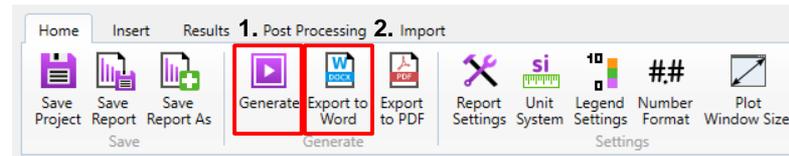
6. Press **OK**



Generate Report

1 Press to generate report

2 Press to export report to Word



Model Setup

Report Designer Tutorial



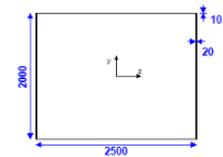
Prepared by:
SDC Verifier
+31 15 30-10-310
sdoverifier.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Prepared for:
company
+31 15 555-55-55
company.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Engineer: Support
Customer: customer
Project Number:
Version: 1
Date: 04/01/2021

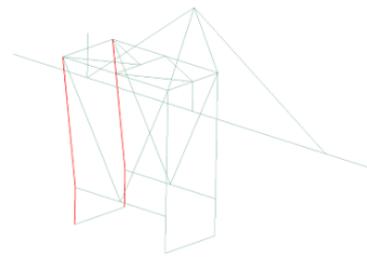
1..portside legs

Property	Value	Property Shape
Type / Elements	Beam / 44	
Material	1..steel	
Mass [kg]	112491.4	
Gravity Center [mm]	[-32416.50; 0.00; 28041.21]	
Area, [mm ²]	129200.00	
I1, [mm ⁴]	75378306666.67	
I2, [mm ⁴]	147822226666.67	
I12, [mm ⁴]	6.974e-05	
Torsion Constant, [mm ⁴]	140675346147.14	
Y Shear Area, [mm ²]	0	
Z Shear Area, [mm ²]	0	
Nonstructural Mass, [kg]	0	
Perimeter, [mm]	0	
Warping Constant, [mm ⁶]	12011140169988100.00	
Y Neutral Axis Offset A, [mm]	0	
Z Neutral Axis Offset A, [mm]	0	
w [mm]	2500.000	
h [mm]	2000.000	
t1 [mm]	10.000	
t2 [mm]	20.000	



3..tip load

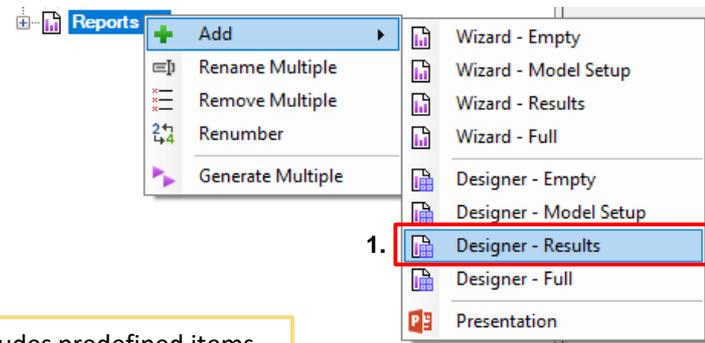
	Definition	Load Type	Applied On	Values
tip load		Force	Nodes: 325;	ScalingForce: 1000;



Add Result Report

1

Execute Report – Add – Designer- Results from report context menu



Result report includes predefined items

- Load Set '1..LC1s_Tip load.1'
 - Displacement (LS1, All Entities)
 - Usum (LS1, All Entities, v1)
 - Seqv (LS1, All Entities, v1, Total)
- Summary
 - Abs Stress (13 Loads, All Entities)
 - Abs Stress (20 Loads, All Entities)
 - Abs Displacement (13 Loads, All Entities)
 - Abs Displacement (20 Loads, All Entities)
 - Applied Force Summation (13 Loads, All Entities)
 - Reaction Force Summation (13 Loads, All Entities)
 - Applied Force Summation (20 Loads, All Entities)
 - Reaction Force Summation (20 Loads, All Entities)

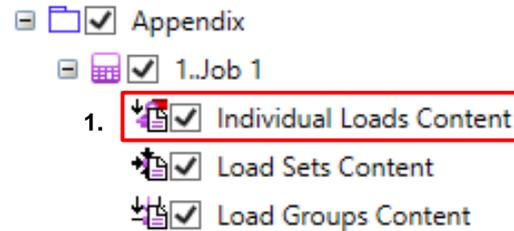
For each load extreme displacement table, displacement and stress plots are created

For individual loads and load sets the following summary tables are included: applied and reaction forces summation, displacement and stresses over loads

Individual Loads Content

1 Select **Individual Loads Content** under Appendix item in report structure

2 Execute *Generate* from context menu



Individual Loads Content

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

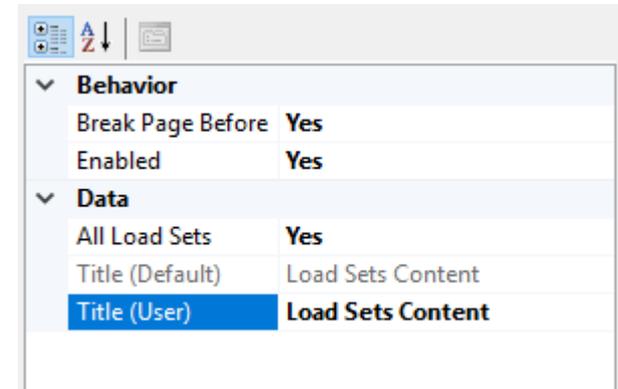
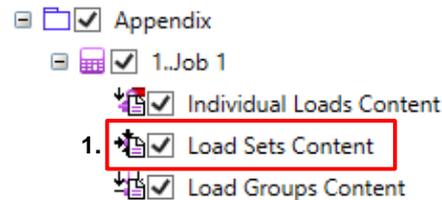
Content shows what boundary conditions are for Individual Load. If Individual Load was created based on results than Output Set is shown instead of FemLoad.

1

Select **Load Sets Content** under Appendix item in report structure

2

Execute *Generate* from context menu



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..9Trolley_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..9Trolley_ride [1.15] 8..tip side_load [-1.15] 13..Crane_ride [-1.15]

Individual Load Options

1 Select 5..At_forestay in report structure

2 Execute *Generate* from context menu

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

Behavior	
Break Page Before	Yes
Enabled	Yes
Data	
Job	1..Job 1
Load Type	IndividualLoad
Title (Default)	Individual Load '5..A
Title (User)	
Options	
Include Individual Load Plot	Yes
Include Load Item Content	Yes
Include Sum Of Forces	Yes
Selection	All Entities
Plot	
View	1..Default View

- 1..Job 1
 - Individual Loads
 - Individual Load '1..gravity'
 - Individual Load '2..tip load'
 - Individual Load '3..middle_bridge'
 - Individual Load '4..back side'
 - 1. Individual Load '5..At_forestay'**
 - Individual Load '6..at_hinge_point'

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	Job 1 Linear -At_forestay
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+3	0e+3	1220000e+3	1220000e+3	0.0	0.0	0.0	0.0

Displacement (IL5, All Entities)								
Individual Load	5..At_forestay			Selection	All Entities			
Type	Extreme			Category	Displacement			
	Ux [mm]	Uy [mm]	Uz [mm]	Usum [mm]	Rx	Ry	Rz	Rsum
Minimum	-0.72	-3.70	-112.66	0.00	0.00	0.00	0.00	0.00
Maximum	29.00	3.70	12.59	112.66	0.00	0.00	0.00	0.00
Absolute	29.00	3.70	-112.66	112.66	0.00	0.00	0.00	0.00

Number Formats

1. Select **Displacement (All Entities)** table under load **5..At_forestay**
2. Press **##** to open Number Formats
3. Digits after decimal point: **3** for Displacement category
4. Press *Close*
5. Execute *Generate* from context menu

- Individual Load '5..At_forestay'
 - 1. **Displacement (IL5, All Entities)**
 - Usun (IL5, All Entities, v1)
 - Seqv (IL5, All Entities, v1, Total)

Report Settings
 Unit System
 Legend Settings
 2. ## Number Format
 Plot Window Size

Displacement (IL5, All Entities)

Individual Load Type	5..At_forestay Extreme	Selection Category			All Entities Displacement				
	Extreme	Ux [m]	Uy [m]	Uz [m]	Usum [m]	Rx	Ry	Rz	Rsum
Minimum		0.00	0.00	-0.11	0.00	0.00	0.00	0.00	0.00
Maximum		0.03	0.00	0.01	0.11	0.00	0.00	0.00	0.00
Absolute		0.03	0.00	-0.11	0.11	0.00	0.00	0.00	0.00

Digits after decimal point = 2

Displacement (IL5, All Entities)

Individual Load Type	5..At_forestay Extreme	Selection Category			All Entities Displacement				
	Extreme	Ux [m]	Uy [m]	Uz [m]	Usum [m]	Rx	Ry	Rz	Rsum
Minimum		0.000	-0.004	-0.114	0.000	0.00	0.00	0.00	0.00
Maximum		0.030	0.004	0.013	0.114	0.00	0.00	0.00	0.00
Absolute		0.030	0.004	-0.114	0.114	0.00	0.00	0.00	0.00

Digits after decimal point = 3

3. **Number Formats**

Category	Type	Digits after decimal point	Fixed Power	Power Value	Example
Displacements	General	3	<input type="checkbox"/>		160000000.000
Stress	Scientific	2	<input checked="" type="checkbox"/>	6	160.00e+6
Strain	General	2	<input type="checkbox"/>		160000000.00
Utilization Factor	General	2	<input type="checkbox"/>		160000000.00
Buckling Factor	General	2	<input type="checkbox"/>		160000000.00
Forces	Scientific	0	<input checked="" type="checkbox"/>	3	160000e+3
Coefficient	General	2	<input type="checkbox"/>		160000000.00
Scientific	Scientific	2	<input type="checkbox"/>		1.60e+8
General	General	2	<input type="checkbox"/>		160000000.00
Mass	General	1	<input type="checkbox"/>		160000000.0
Dimensions	General	3	<input type="checkbox"/>		160000000.000
Length	General	2	<input type="checkbox"/>		160000000.00
Area	General	2	<input type="checkbox"/>		160000000.00
Dimensions^3	General	2	<input type="checkbox"/>		160000000.00
Moment of Inertia	General	2	<input type="checkbox"/>		160000000.00
Dimensions^6	General	2	<input type="checkbox"/>		160000000.00
Number	General	0	<input type="checkbox"/>		160000000
Moments	General	1	<input type="checkbox"/>		160000000.0
Deflection	General	3	<input type="checkbox"/>		160000000.000

Number Format: General Scientific

Digits after decimal point: 3

Fixed power: -1

Example: 160000000.000

4. **Close**

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

Legend Settings

1. Select Equivalent Stress – **Seqv(All Entities)** under load **5..At_forestay**

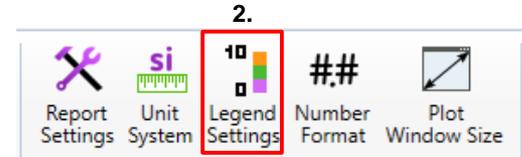
2. Press to open Legend Settings

3. Max: **180e+3** for Stress category

4. Press *Close*

5. Execute *Generate* from context menu

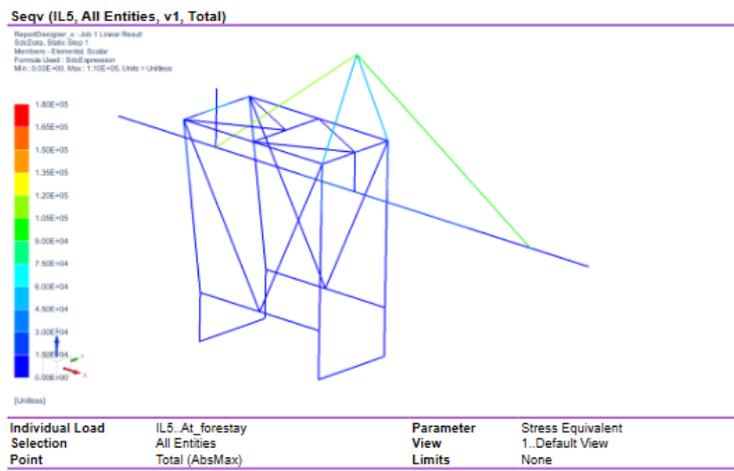
- Individual Load '5..At_forestay'
 - Displacement (IL5, All Entities)
 - Usum (IL5, All Entities, v1)
 - 1. **Seqv (IL5, All Entities, v1, Total)**



3. **Legend Settings**

Category	Mode	Min	Max	Number of Levels	Legend Format
Displacements	Automatic			12	General
Stress	Min Max	0	180e+3	12	Exponent
Strain	Automatic			12	General
Utilization Factor	Min Max	0	1.2	12	General
Buckling Factor	Min Max	0	1.2	12	General
Forces	Automatic			12	General
Coefficient	Automatic			12	General
Scientific	Automatic			12	General
General	Automatic			12	General
Mass	Automatic			12	General
Dimensions	Automatic			12	General
Length	Automatic			12	General
Area	Automatic			12	General
Dimensions^3	Automatic			12	General
Moment of Inertia	Automatic			12	General
Dimensions^6	Automatic			12	General
Number	Automatic			12	General
Moments	Automatic			12	General
Deflection	Automatic			12	General

4. **Close**



Legend Settings controls legend options for different categories. It is possible to save settings to the library and reuse in another projects.

Stress and displacement tables over loads

1 Select **Abs stress** and **Abs displacement** together, under Summary item

2 Execute *Generate* from context menu

- ☑ Summary
 - ☑ Abs Stress (13 Loads, All Entities)
 - ☑ **Abs Stress (20 Loads, All Entities)**
 - 1. ☑ Abs Displacement (13 Loads, All Entities)
 - ☑ **Abs Displacement (20 Loads, All Entities)**
 - ☑ Applied Force Summation (13 Loads, All Entities)
 - ☑ Reaction Force Summation (13 Loads, All Entities)
 - ☑ Applied Force Summation (20 Loads, All Entities)
 - ☑ Reaction Force Summation (20 Loads, All Entities)

Stress and displacement extreme flow tables give nice results overview among loads. For each direction min and max values are highlighted. Min = aqua, Max = red.

Abs Stress (20 Loads, All Entities)								
Loads Count	All Entities		Category	Stress				
Selection	Abs		Type	Extreme				
Parameter	Load	X [Pa]	Y [Pa]	Z [Pa]	XY [Pa]	YZ [Pa]	ZX [Pa]	Equivalent [Pa]
LS1	LC1s_Tip load.1	-176.40e+6			-0.02e+6			176.40e+6
LS2	LC1s_Tip load.2	-176.40e+6			0.02e+6			176.40e+6
LS3	LC1s_Tip load.3	-172.96e+6			-0.02e+6			172.96e+6
LS4	LC1s_Tip load.4	-172.96e+6			0.02e+6			172.96e+6
LS5	LC1s_Middle Bridge.1	-113.16e+6			0.00e+6			113.16e+6
LS6	LC1s_Middle Bridge.2	-113.16e+6			0.00e+6			113.16e+6
LS7	LC1s_Middle Bridge.3	-115.13e+6			0.00e+6			115.13e+6
LS8	LC1s_Middle Bridge.4	-115.13e+6			0.00e+6			115.13e+6
LS9	LC1s_Backside.1	139.42e+6			0.00e+6			139.42e+6
LS10	LC1s_Backside.2	139.42e+6			0.00e+6			139.42e+6
LS11	LC1s_Backside.3	142.59e+6			0.00e+6			142.59e+6
LS12	LC1s_Backside.4	142.59e+6			0.00e+6			142.59e+6
LS13	LC1s_At_forestay.1	-148.02e+6			-0.01e+6			148.02e+6
LS14	LC1s_At_forestay.2	-148.02e+6			0.01e+6			148.02e+6
LS15	LC1s_At_forestay.3	-144.57e+6			-0.01e+6			144.57e+6
LS16	LC1s_At_forestay.4	-144.57e+6			0.01e+6			144.57e+6
LS17	LC1s_at_hinge_point.1	148.51e+6			-0.01e+6			148.51e+6
LS18	LC1s_at_hinge_point.2	148.39e+6			0.01e+6			148.39e+6
LS19	LC1s_at_hinge_point.3	145.20e+6			-0.01e+6			145.20e+6
LS20	LC1s_at_hinge_point.4	145.09e+6			0.01e+6			145.09e+6

Stresses for all load sets

Abs Displacement (20 Loads, All Entities)										
Loads Count	All Entities		Category	Displacement						
Selection	Abs		Type	Extreme						
Parameter	Load	Ux [m]	Uy [m]	Uz [m]	Usum [m]	Rx	Ry	Rz	Rsum	
LS1	LC1s_Tip load.1	0.073	0.111	-0.352	0.369	0.00	0.01	0.00	0.01	
LS2	LC1s_Tip load.2	0.073	-0.111	-0.352	0.369	0.00	0.01	0.00	0.01	
LS3	LC1s_Tip load.3	0.059	0.111	-0.351	0.369	0.00	0.01	0.00	0.01	
LS4	LC1s_Tip load.4	0.059	-0.111	-0.351	0.369	0.00	0.01	0.00	0.01	
LS5	LC1s_Middle Bridge.1	-0.033	0.057	-0.079	0.098	0.00	0.00	0.00	0.00	
LS6	LC1s_Middle Bridge.2	-0.033	-0.057	-0.079	0.098	0.00	0.00	0.00	0.00	
LS7	LC1s_Middle Bridge.3	-0.044	0.057	-0.078	0.098	0.00	0.00	0.00	0.00	
LS8	LC1s_Middle Bridge.4	-0.044	-0.057	-0.078	0.098	0.00	0.00	0.00	0.00	
LS9	LC1s_Backside.1	-0.032	0.050	-0.144	0.152	0.00	-0.01	0.00	0.01	
LS10	LC1s_Backside.2	-0.032	-0.050	-0.144	0.152	0.00	-0.01	0.00	0.01	
LS11	LC1s_Backside.3	-0.043	0.050	-0.142	0.150	0.00	-0.01	0.00	0.01	
LS12	LC1s_Backside.4	-0.043	-0.050	-0.142	0.150	0.00	-0.01	0.00	0.01	
LS13	LC1s_At_forestay.1	0.059	0.100	-0.239	0.259	0.00	0.00	0.00	0.00	
LS14	LC1s_At_forestay.2	0.059	-0.100	-0.239	0.259	0.00	0.00	0.00	0.00	
LS15	LC1s_At_forestay.3	0.045	0.100	-0.238	0.259	0.00	0.00	0.00	0.00	
LS16	LC1s_At_forestay.4	0.045	-0.100	-0.238	0.259	0.00	0.00	0.00	0.00	
LS17	LC1s_at_hinge_point.1	-0.032	0.087	-0.089	0.111	0.00	0.00	0.00	0.00	
LS18	LC1s_at_hinge_point.2	-0.032	-0.087	-0.089	0.112	0.00	0.00	0.00	0.00	
LS19	LC1s_at_hinge_point.3	-0.044	0.087	-0.088	0.111	0.00	0.00	0.00	0.00	
LS20	LC1s_at_hinge_point.4	-0.044	-0.087	-0.088	0.112	0.00	0.00	0.00	0.00	

Displacements for all load sets

Reaction Forces

1. Select (LS) Reaction Forces Summation under Summary

2. Press ## to open Number Format

3. Select category Forces

4. Scientific: **ON**
 Digits after decimal point: **0**
 Fixed Power: **ON**
 Fixed Power Value: **3**

5. Press *Set Format*

- Summary
 - ✓ Abs Stress (13 Loads, All Entities)
 - ✓ Abs Stress (20 Loads, All Entities)
 - ✓ Abs Displacement (13 Loads, All Entities)
 - ✓ Abs Displacement (20 Loads, All Entities)
 - ✓ Applied Force Summation (13 Loads, All Entities)
 - ✓ Applied Force Summation (20 Loads, All Entities)
 - ✓ Reaction Force Summation (13 Loads, All Entities)
 - ✓ Reaction Force Summation (20 Loads, All Entities)

Reaction Force Summation (20 Loads, All Entities)									
Loads Count Selection	20 All Entities	Category Type				Reaction Force Expand			
Load	Fx [N]	Fy [N]	Fz [N]	Fsum [N]	Mx [N m]	My [N m]	Mz [N m]	Msum [N m]	
LS1..LC1s_Tip load.1	-223100	-835704	24403398	24412894	0.0	0.0	0.0	0.0	
LS2..LC1s_Tip load.2	-223100	835704	24403398	24412894	0.0	0.0	0.0	0.0	
LS3..LC1s_Tip load.3	223100	-835704	24403398	24412894	0.0	0.0	0.0	0.0	
LS4..LC1s_Tip load.4	223100	835704	24403398	24412894	0.0	0.0	0.0	0.0	
LS5..LC1s_Middle Bridge.1	-223100	-835704	24403394	24412892	0.0	0.0	0.0	0.0	



Reaction Force Summation (20 Loads, All Entities)									
Loads Count Selection	20 All Entities	Category Type				Reaction Force Expand			
Load	Fx [N]	Fy [N]	Fz [N]	Fsum [N]	Mx [N m]	My [N m]	Mz [N m]	Msum [N m]	
LS1..LC1s_Tip load.1	-223e+3	-836e+3	24403e+3	24413e+3	0.0	0.0	0.0	0.0	
LS2..LC1s_Tip load.2	-223e+3	836e+3	24403e+3	24413e+3	0.0	0.0	0.0	0.0	
LS3..LC1s_Tip load.3	223e+3	-836e+3	24403e+3	24413e+3	0.0	0.0	0.0	0.0	
LS4..LC1s_Tip load.4	223e+3	836e+3	24403e+3	24413e+3	0.0	0.0	0.0	0.0	
LS5..LC1s_Middle Bridge.1	-223e+3	-836e+3	24403e+3	24413e+3	0.0	0.0	0.0	0.0	

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

Add Extreme Stress Tables

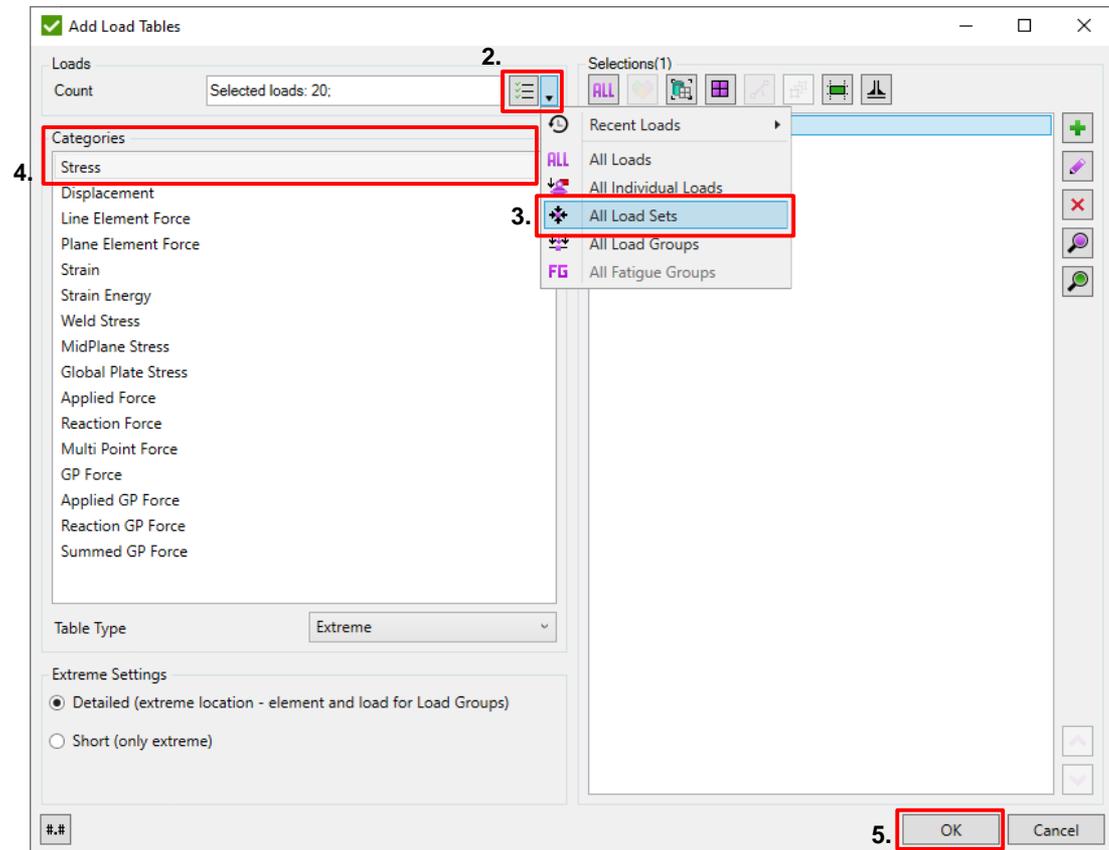
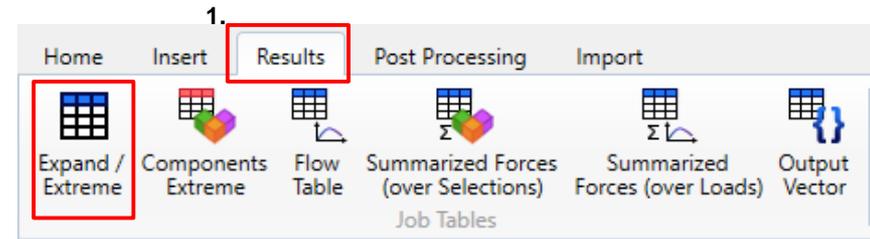
1 Press Results on the toolbar and select  to open tables window

2 Use dropdown menu for load selector

3 Select all Load Sets

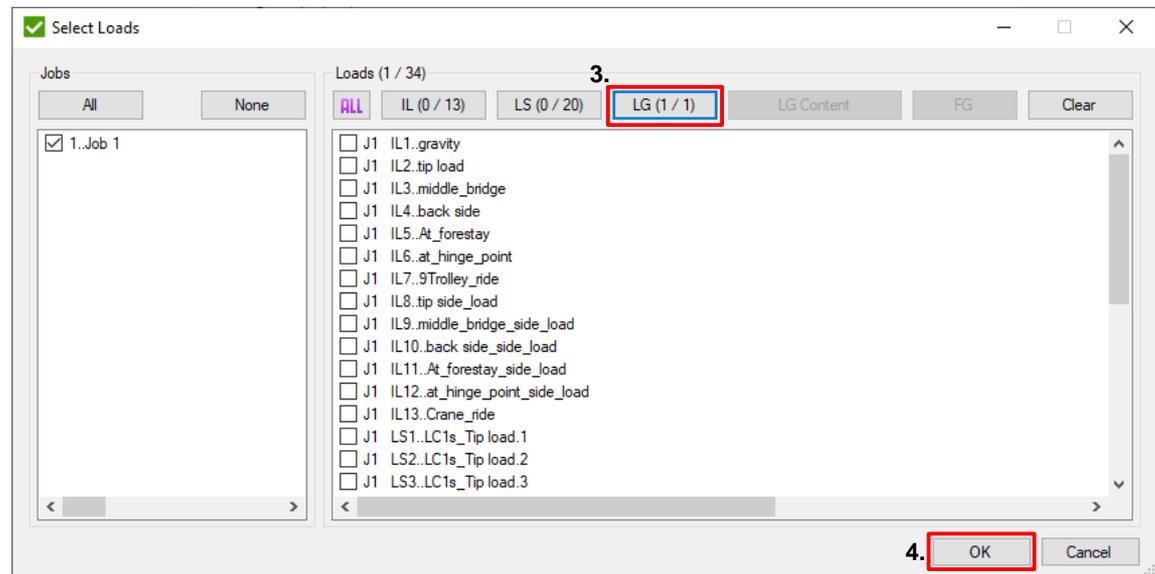
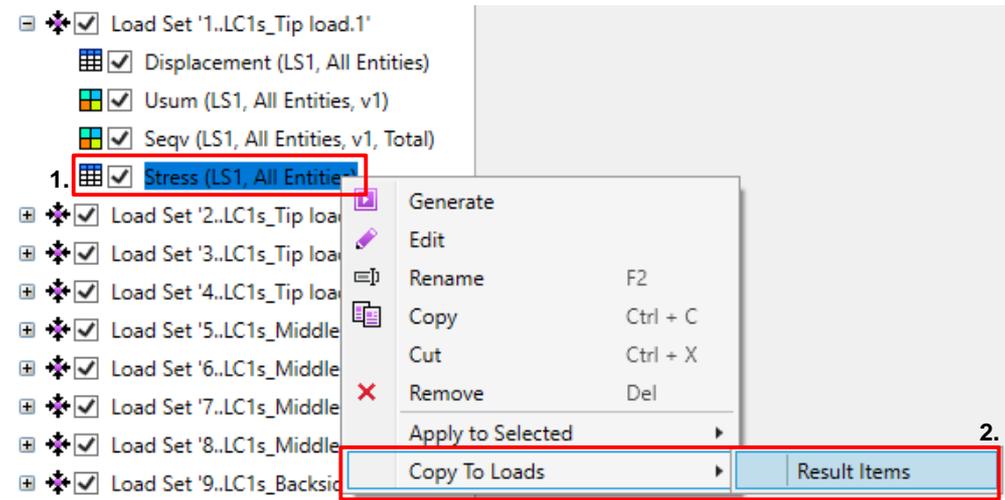
4 Categories: **Stress**

5 Press *OK*



Copy Table to Load Group

- 1 Execute **Stress Table** under Load Set
- 2 Select **Result Items** from context menu
- 3 Table Type: **Load Group**
- 4 Press **OK**

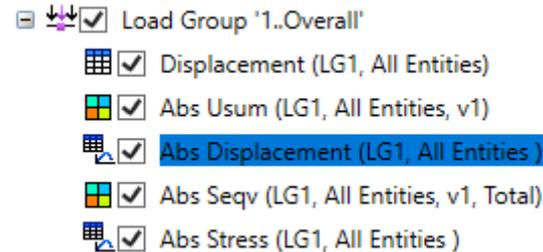
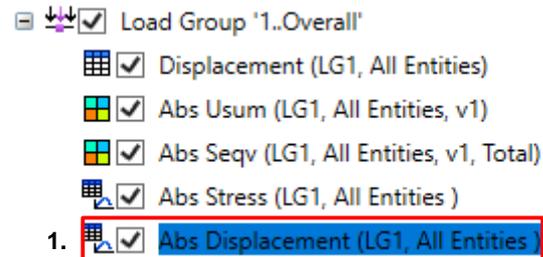
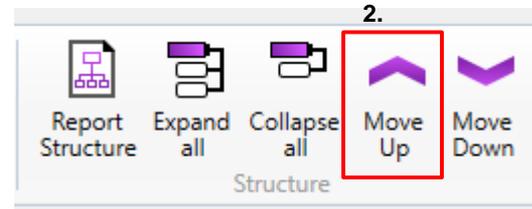


Move item in the structure

1 Select **Abs Displacement (LG1, All Entities)**

2 Press  twice to move item up

Move up and move down is possible using Ctrl + Up and Ctrl + Down



Add plots for Load Group

1 Execute **Stress Table** under **Load Set** tree

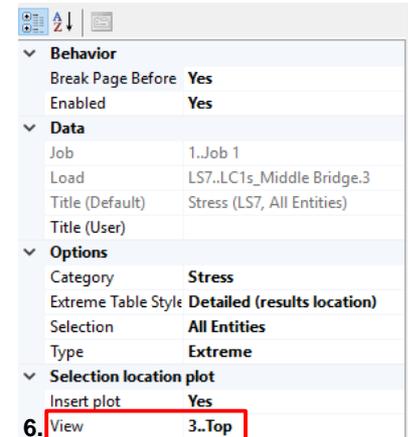
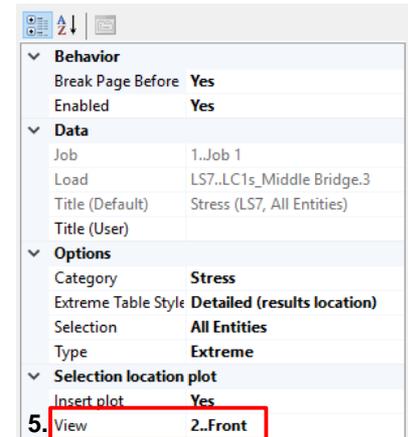
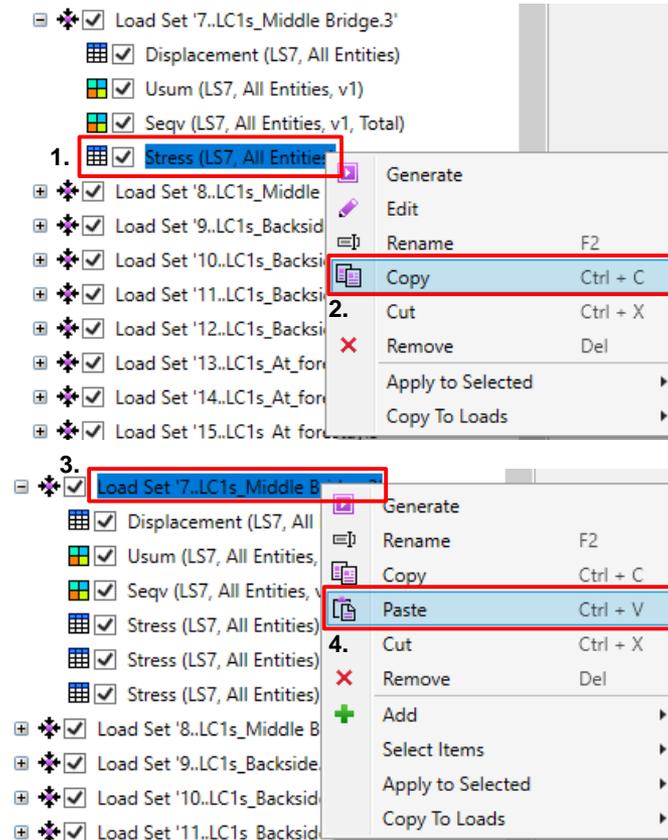
2 Select *Copy*

3 Execute **Load Set**

4 In context menu select *Paste* (twice)

5 For second **Stress Table** set View = **2..Front** in Property Grid

6 For third Stress Table set View = **3..Top** in Property Grid



Add table for Static Stress Check

1. Select **Check Tables** from Toolbar

2. Select **Static Stress Check**

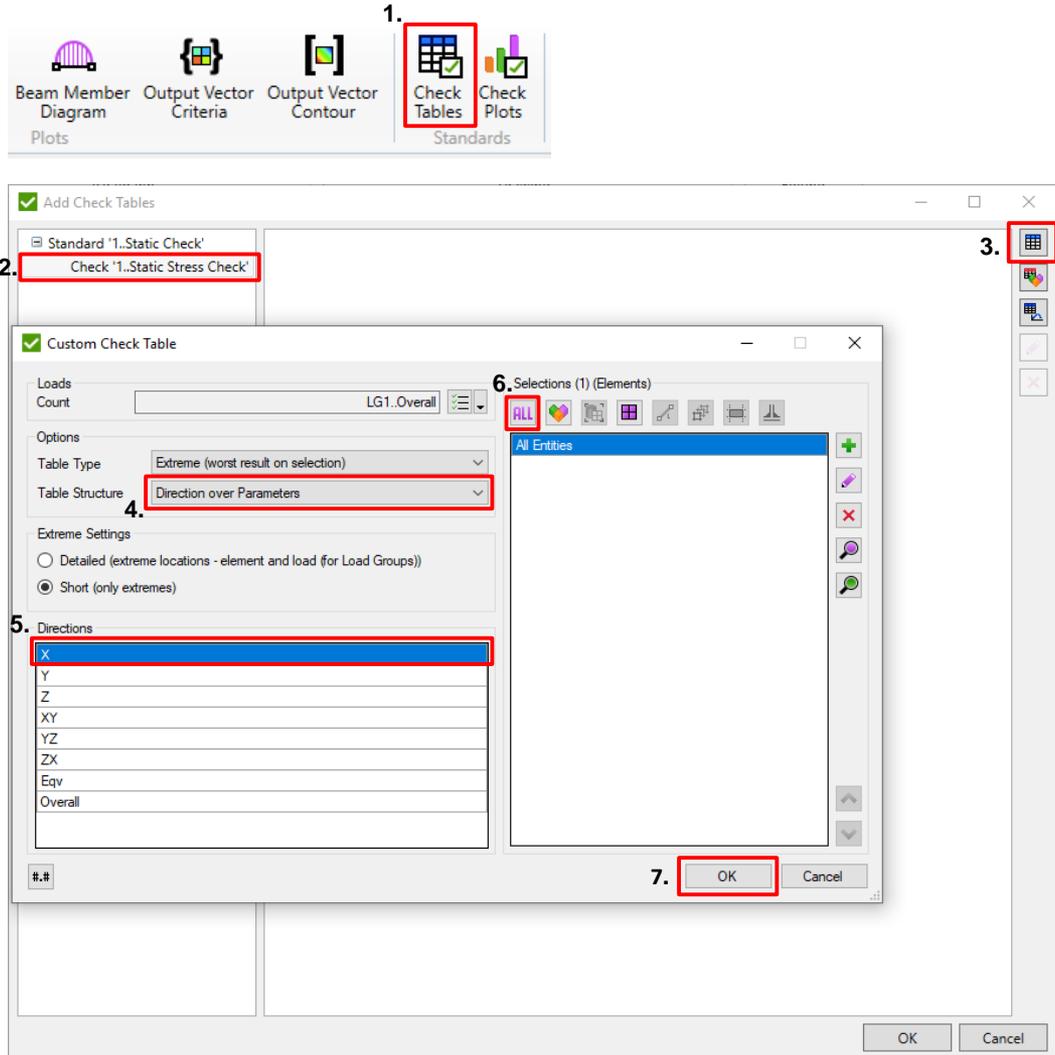
3. Select 

4. Select **Direction over Parameters**

5. Direction: **X**

6. Press  to add full model selection

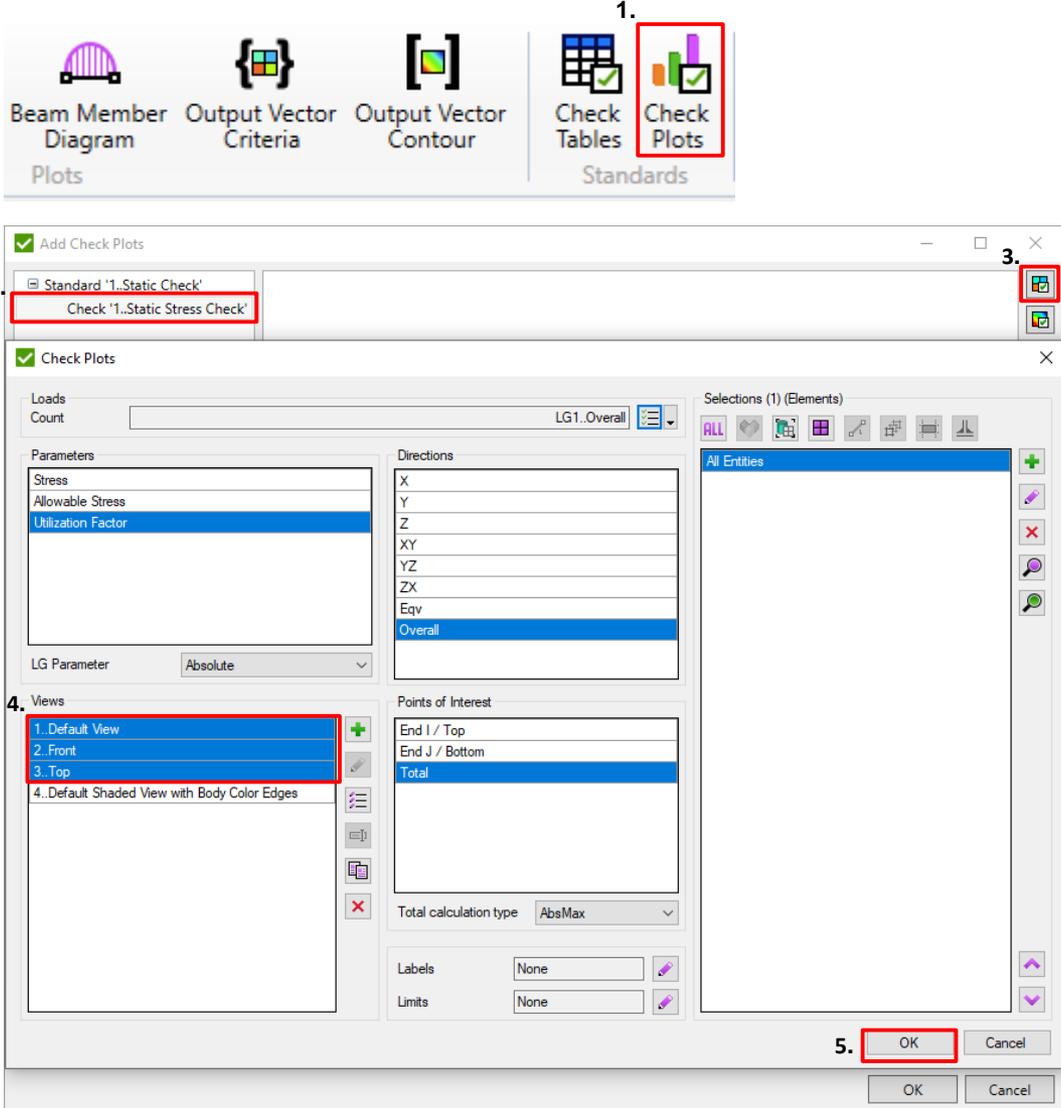
7. Press **OK**



The screenshot displays the SDC Verifier interface with two dialog boxes open. The top dialog, 'Add Check Tables', has a toolbar with icons for 'Beam Member Diagram Plots', 'Output Vector Criteria', 'Output Vector Contour', 'Check Tables', and 'Check Plots Standards'. The 'Check Tables' icon is highlighted with a red box and labeled '1.'. Below the toolbar, a list of check tables includes 'Standard '1..Static Check'' and 'Check '1..Static Stress Check'', with the latter highlighted by a red box and labeled '2.'. The bottom dialog, 'Custom Check Table', has several sections: 'Loads' (Count: LG1..Overall), 'Options' (Table Type: Extreme (worst result on selection), Table Structure: Direction over Parameters, highlighted with a red box and labeled '4.'), 'Extreme Settings' (Short (only extremes) selected), and 'Directions' (X selected, highlighted with a red box and labeled '5.'). The 'Selections (1) (Elements)' section shows 'All Entities' selected, with the 'ALL' button highlighted by a red box and labeled '6.'. The 'OK' button at the bottom right is highlighted with a red box and labeled '7.'. The 'Add Check Tables' dialog also has a 'Check Tables' icon highlighted with a red box and labeled '3.' in its top right corner.

Add Plot for Static Stress check

- 1 Select **Check Plots** on the Toolbar
- 2 Select **Static Stress Check**
- 3 Press **Check Plots** 
- 4 Select Views with IDs 1-3
- 5 Press **OK**



The screenshot shows the 'Add Check Plots' dialog box in the software. The 'Check Plots' section is active, and the 'Check '1..Static Stress Check'' option is selected. The 'Views' list shows '1..Default View', '2..Front', and '3..Top' selected. The 'OK' button is highlighted.

1. **Check Plots** (on the toolbar)

2. **Check '1..Static Stress Check'** (in the dialog)

3. **Check Plots** (icon in the dialog)

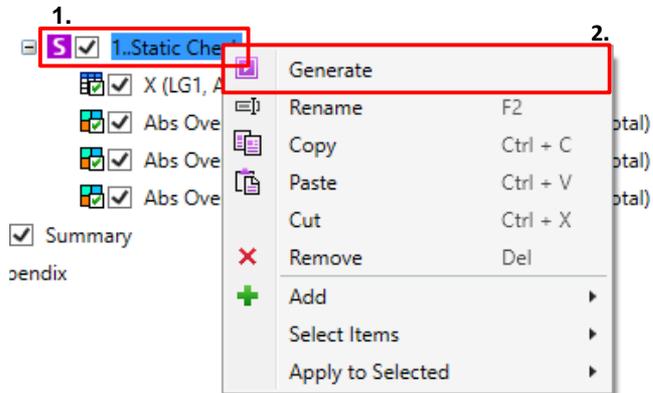
4. **Views** (list in the dialog)

5. **OK** (button in the dialog)

Generate Static Stress Check results

1. Select **Static Stress Check**

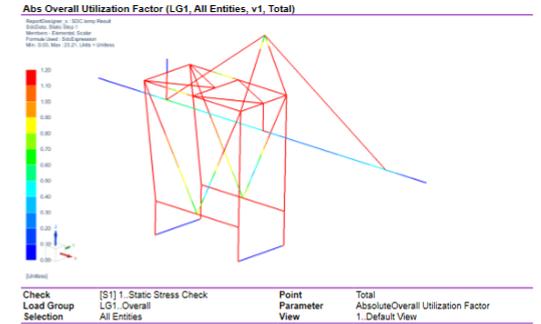
2. Execute *Generate* from context menu



1.. Static Check

Unit System
Current Unit System = Mm/Kg (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.

X (LG1, All Entities)			
Standard	1..Static Check	Check Selection	[S1] 1..Static Stress Check All Entities
Load Group	LG1..Overall		
Extreme	Stress [KPa]	Allowable Stress [KPa]	Utilization Factor
Minimum	-5.57e+8	0.24e+0	0.00
Maximum	5.51e+8	0.24e+0	23.21
Absolute	-5.57e+8	0.24e+0	23.21



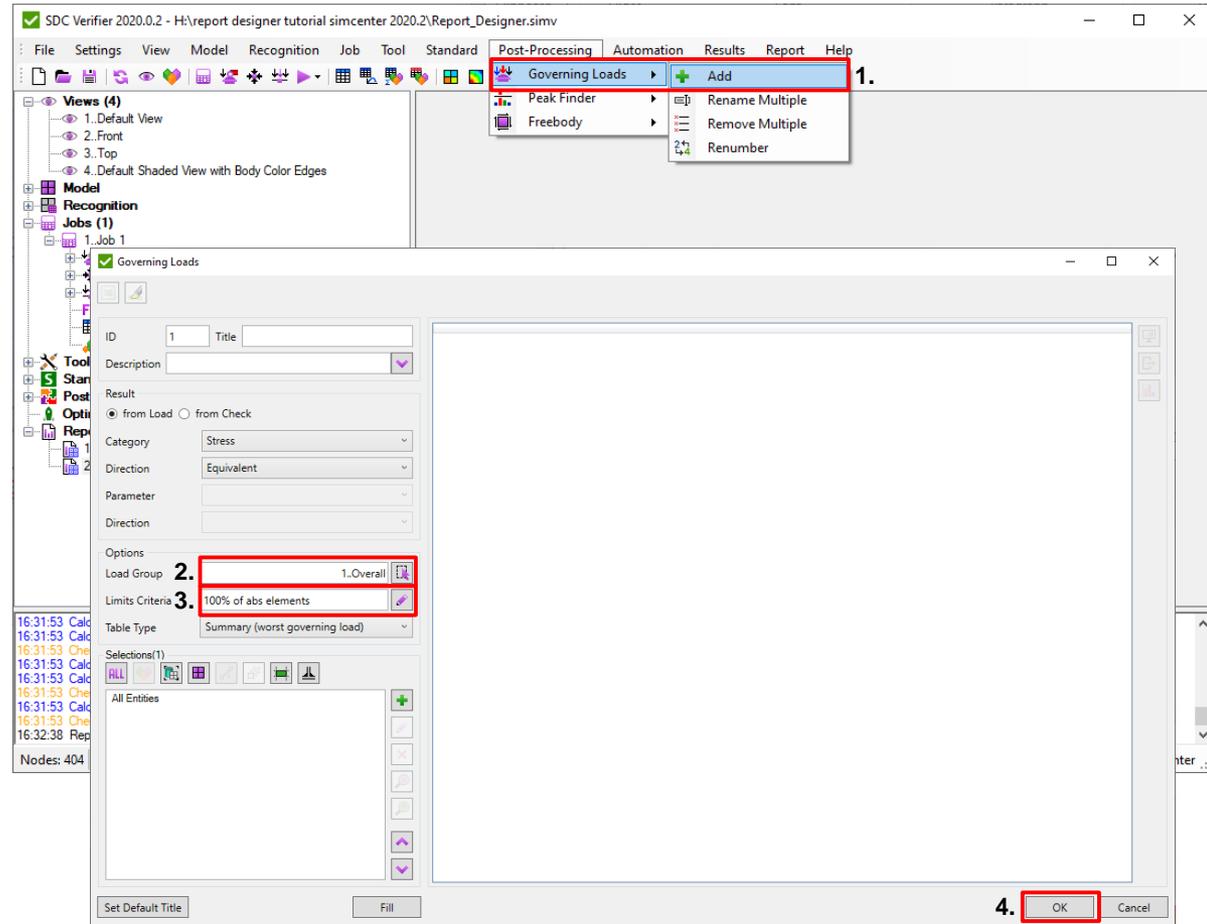
Add Governing Loads

1 Select *Post-Processing- Governing Loads - Add*

2 Load Group **1.Overall**

3 Limits Criteria **100% of abs elements**

4 Press *OK*



Add table for Governing Loads

1. Select **Post-Processing** from Toolbar

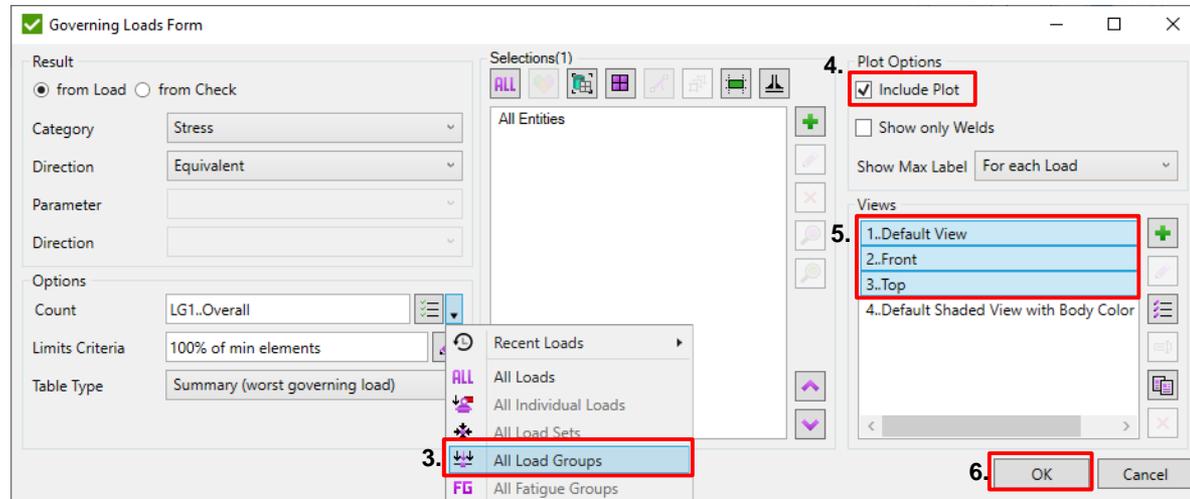
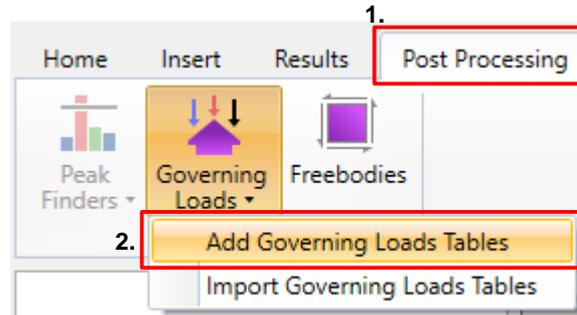
2. Select **Add Governing Load Tables**

3. In dropdown menu select **All Load Groups**

4. Press **Include Plot**

5. Select Views with IDs 1-3

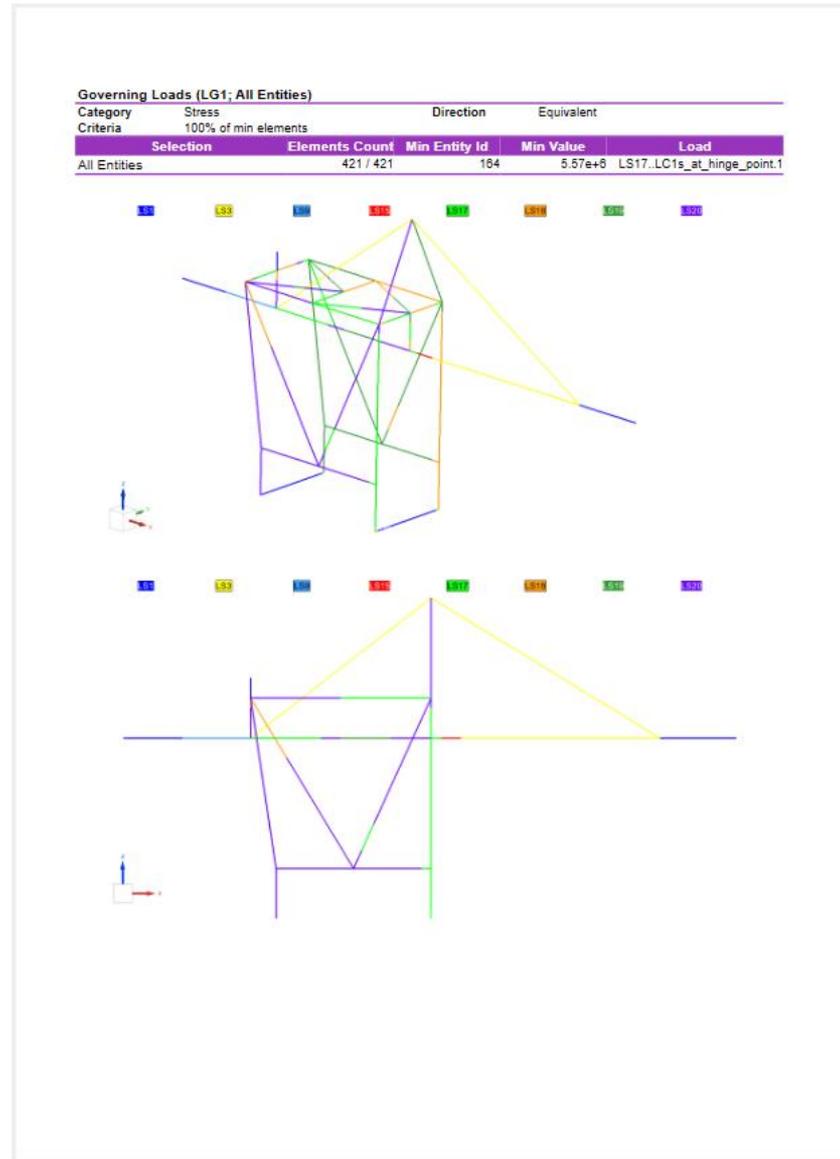
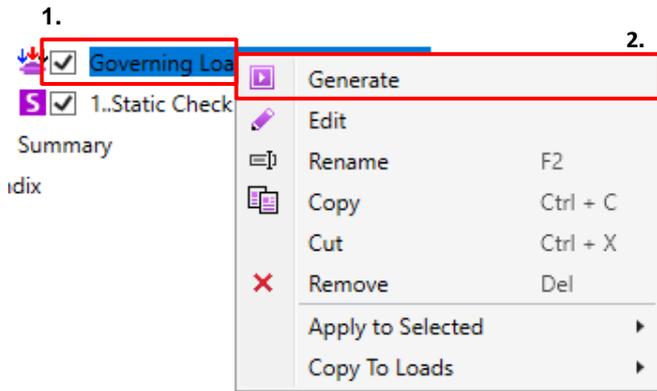
6. Press **OK**



Generate Governing Loads results

1. Select **Governing Loads**

2. Execute *Generate* from context menu



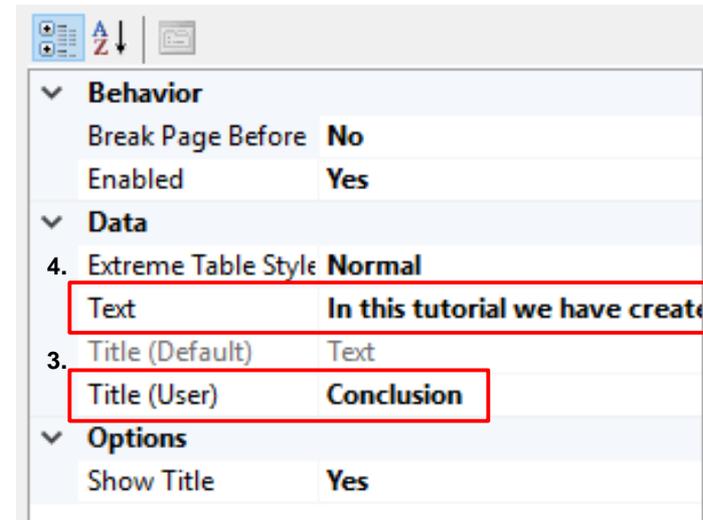
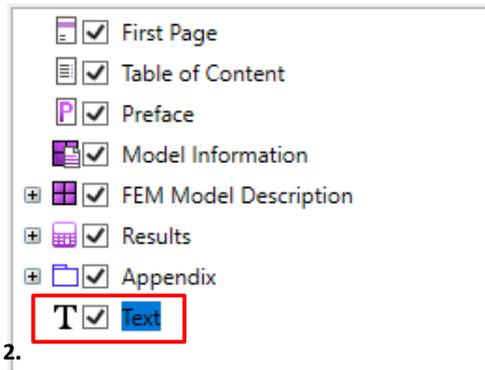
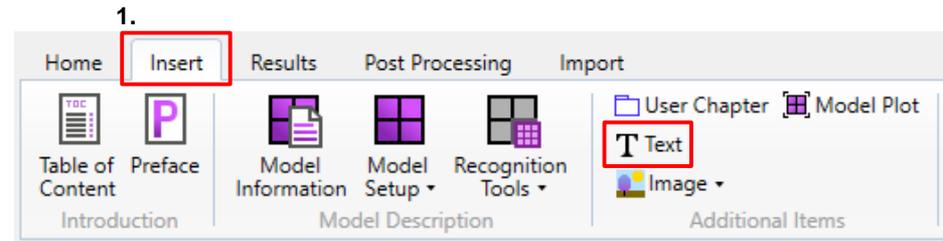
Add Conclusion

1. Select **Insert** on the Toolbar and click on *Text* item

2. Select **Text** in model tree

3. In display properties set the Title: **Conclusion**

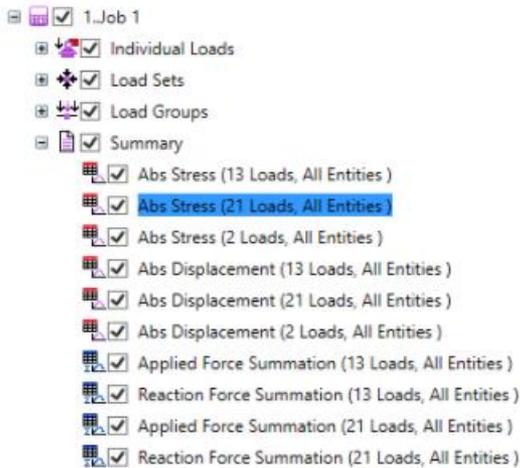
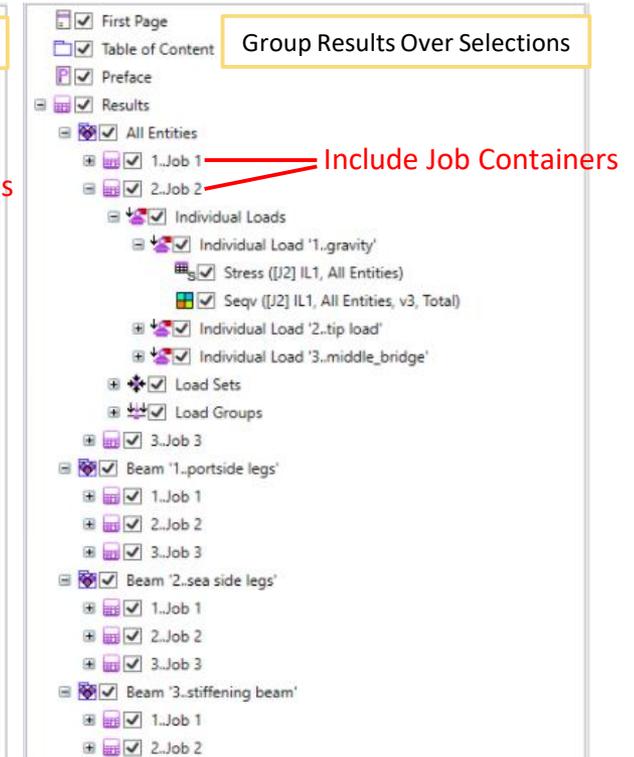
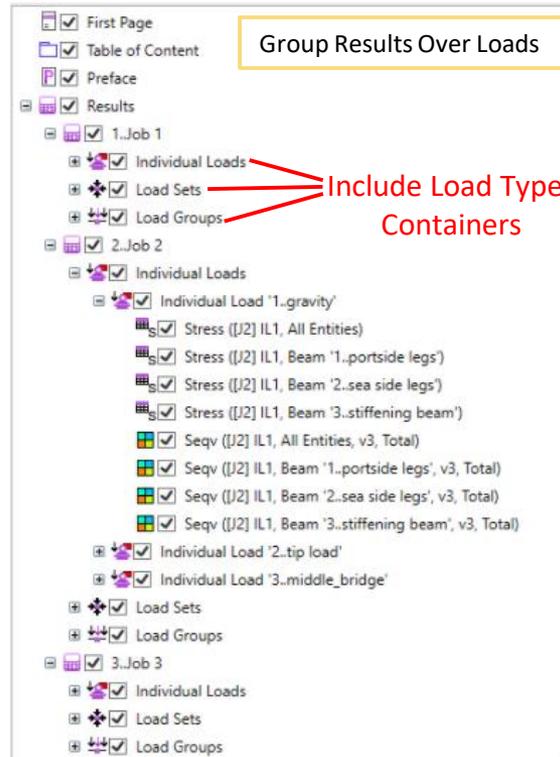
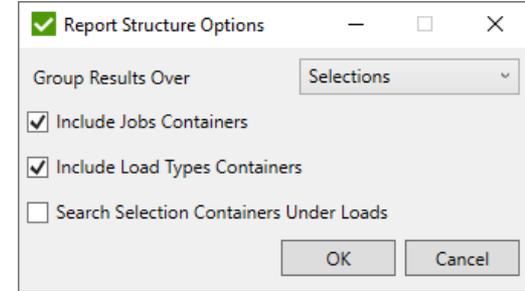
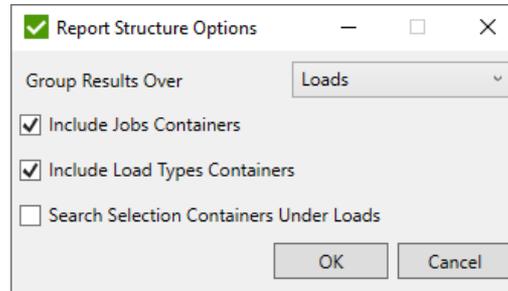
4. Text: **In this tutorial we have created 2 reports using Report Designer**



Report Structure

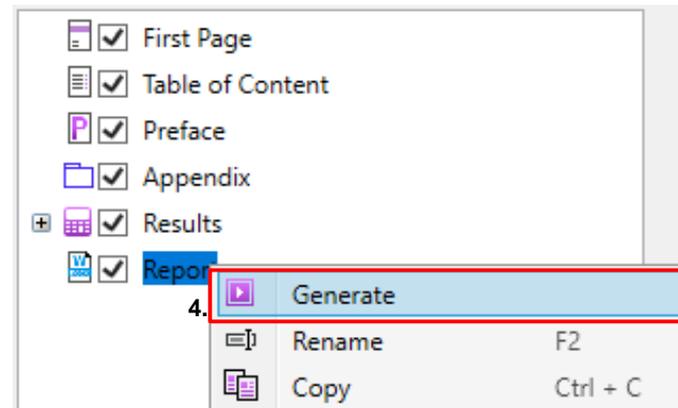
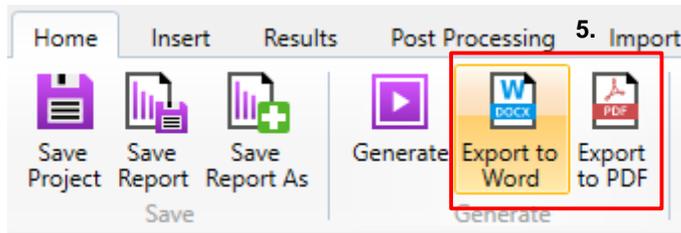
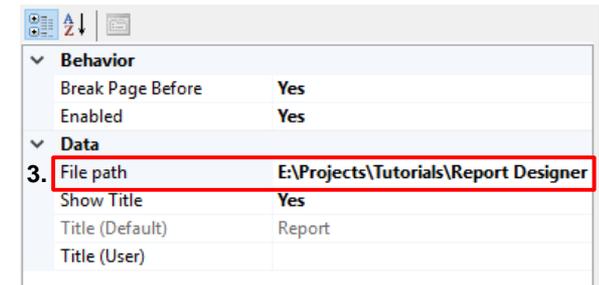
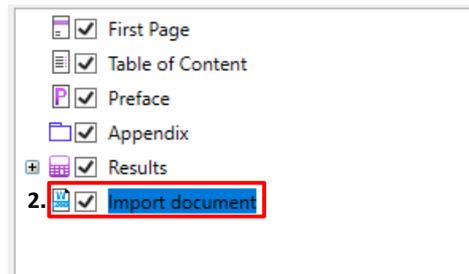
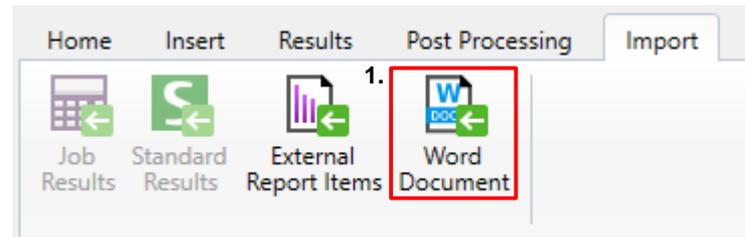
When Table/Plot is edited and load/selection is changed, the item is moved under correspondent Load/Selection automatically. Moreover, when the item is dropped under Load/Selection its load/selection is updated as well.

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



Import from word document

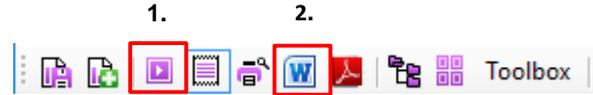
- 1 Select **Import** on the Toolbar and click on *Word Document*
- 2 Select **Import document** in report structure
- 3 In display properties set the file path
- 4 *Generate*
- 5 Your word file will be displayed after report is exported to word or PDF



Generated report

1 Press to generate report

2 After generation is finished press to export generated report to Word



Model Setup

Report Designer Tutorial



Prepared by:
SDC Verifier
+31 15 30-10-310
sdcoverifier.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Prepared for:
company
+31 15 555-55-55
company.com
Zijvest 25
2011 VB Haarlem
The Netherlands

Engineer: Support
Customer: customer
Project Number:
Version: 1
Date: 04/01/2021

