



# Tutorial

## **Report Designer**

**ANSYS<sup>®</sup>**

16 Oct 2020  
Version 2020

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

This tutorial demonstrates how to build reports using the 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.

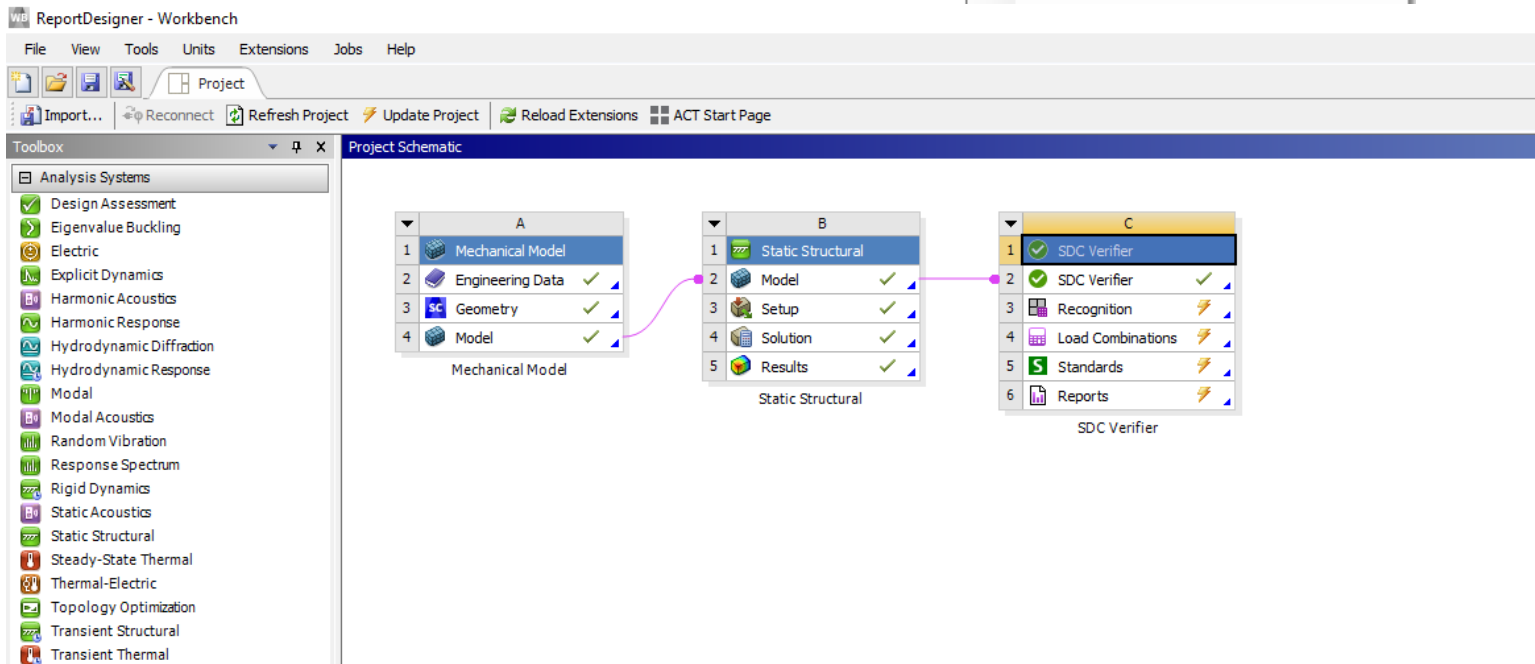
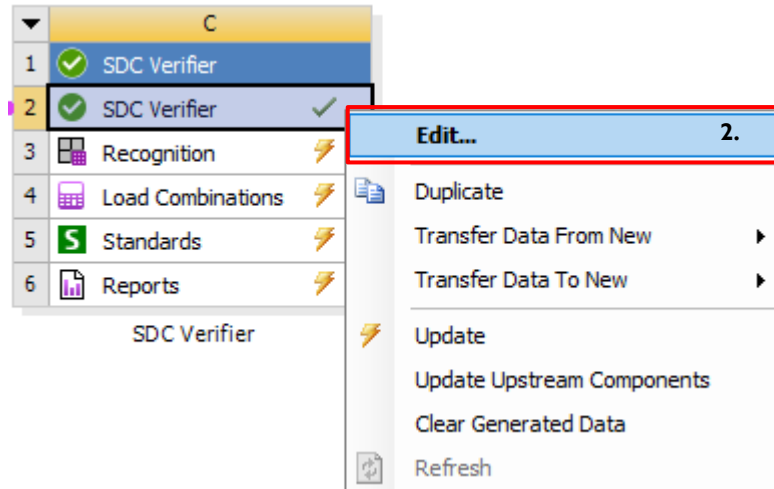
# Open Project

1

Open in Ansys Workbench  
Report Designer.wbpj

2

Double Click on **SDC Verifier**  
or execute from **Edit** context menu



# Predefined project

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

- Views (1)
- Model
- Recognition
- Jobs (1)
  - 1..Static Structural
    - 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)
- Tools
- Standards (1)
- Post-Processing
- Optimizations (0)
- Reports (0)

The central 3D model shows a bridge structure with a main span and a long approach span. A scale bar at the bottom indicates dimensions of 0.00, 15.00, 30.00, and 45.00.

On the right, a detailed view of the 'Standards (1)' tree is shown:

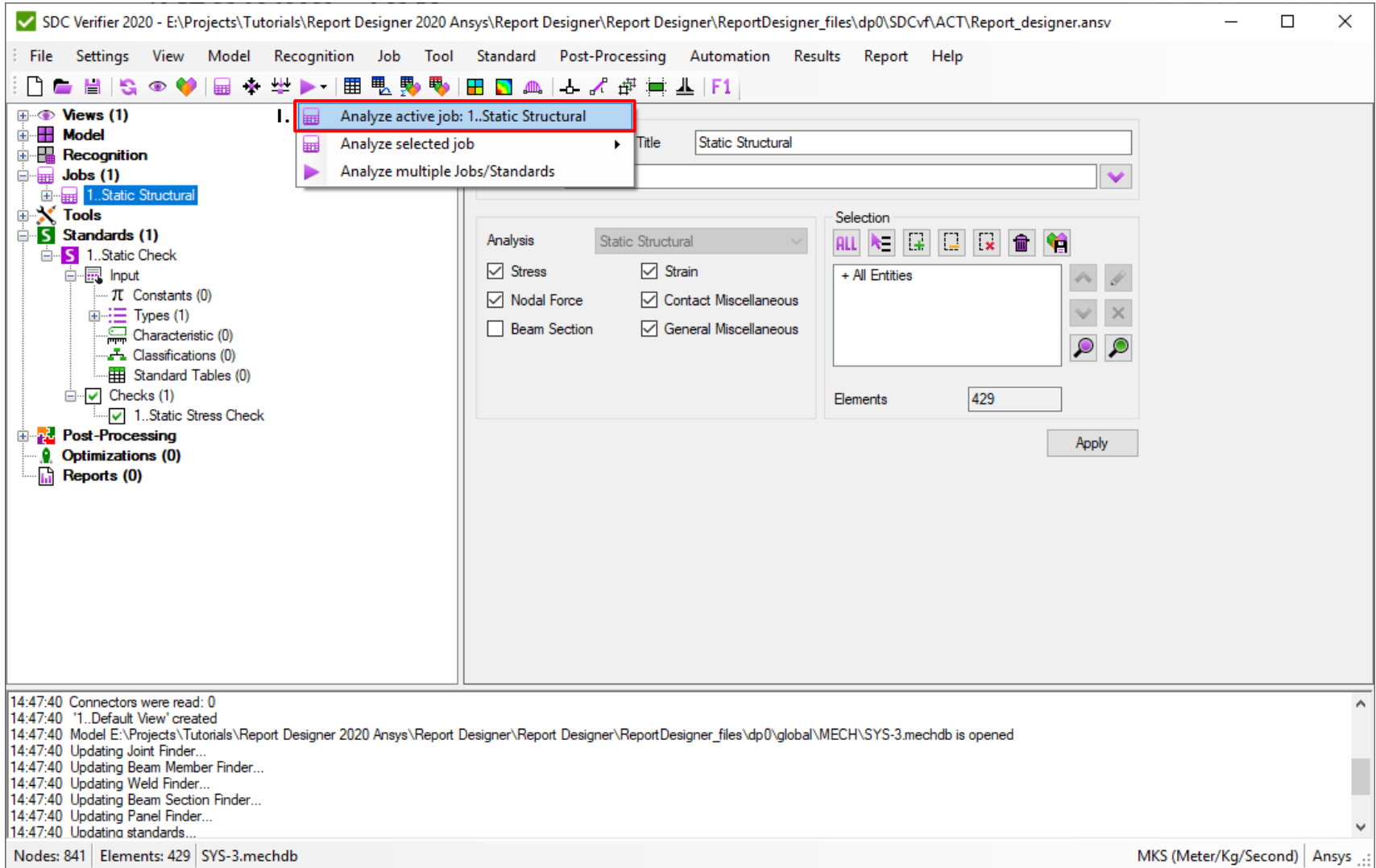
- Standards (1)
  - 1..Static Check
    - Input
      - $\pi$  Constants (0)
      - Types (1)
      - Characteristic (0)
      - Classifications (0)
      - Standard Tables (0)
    - Checks (1)
      - 1..Static Stress Check

This tutorial uses 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

Press  and select **Analyze active job:**  
**1..Static Structural**



The screenshot shows the SDC Verifier 2020 software interface. The main window displays the 'Job' menu with the 'Analyze active job: 1..Static Structural' option highlighted. The left sidebar shows a tree view of the project structure, including 'Views (1)', 'Model', 'Recognition', 'Jobs (1)', 'Tools', 'Standards (1)', 'Post-Processing', 'Optimizations (0)', and 'Reports (0)'. The 'Jobs (1)' section is expanded, showing '1..Static Structural'. The main panel shows the 'Analysis' settings for 'Static Structural', including checkboxes for 'Stress', 'Strain', 'Nodal Force', 'Contact Miscellaneous', 'Beam Section', and 'General Miscellaneous'. The 'Selection' panel shows '+ All Entities' and 'Elements: 429'. The status bar at the bottom indicates 'Nodes: 841 | Elements: 429 | SYS-3.mechdb' and 'MKS (Meter/Kg/Second) | Ansys ...'.

SDC Verifier 2020 - E:\Projects\Tutorials\Report Designer 2020 Ansys\Report Designer\Report Designer\ReportDesigner\_files\dp0\SDCvf\ACT\Report\_designer.ansv

File Settings View Model Recognition Job Tool Standard Post-Processing Automation Results Report Help


Views (1)  
Model  
Recognition  
Jobs (1)  
1..Static Structural  
Tools  
Standards (1)  
1..Static Check  
Input  
Constants (0)  
Types (1)  
Characteristic (0)  
Classifications (0)  
Standard Tables (0)  
Checks (1)  
1..Static Stress Check  
Post-Processing  
Optimizations (0)  
Reports (0)

1. Analyze active job: 1..Static Structural  
Analyze selected job  
Analyze multiple Jobs/Standards

Static Structural

Analysis: Static Structural

Stress ☒ Strain ☒  
Nodal Force ☒ Contact Miscellaneous ☒  
Beam Section ☐ General Miscellaneous ☒


Selection: ALL   
+ All Entities  
Elements: 429  
Apply

14:47:40 Connectors were read: 0  
14:47:40 '1..Default View' created  
14:47:40 Model E:\Projects\Tutorials\Report Designer 2020 Ansys\Report Designer\Report Designer\ReportDesigner\_files\dp0\global\MECH\SYS-3.mechdb is opened  
14:47:40 Updating Joint Finder...  
14:47:40 Updating Beam Member Finder...  
14:47:40 Updating Weld Finder...  
14:47:40 Updating Beam Section Finder...  
14:47:40 Updating Panel Finder...  
14:47:40 Updating standards...

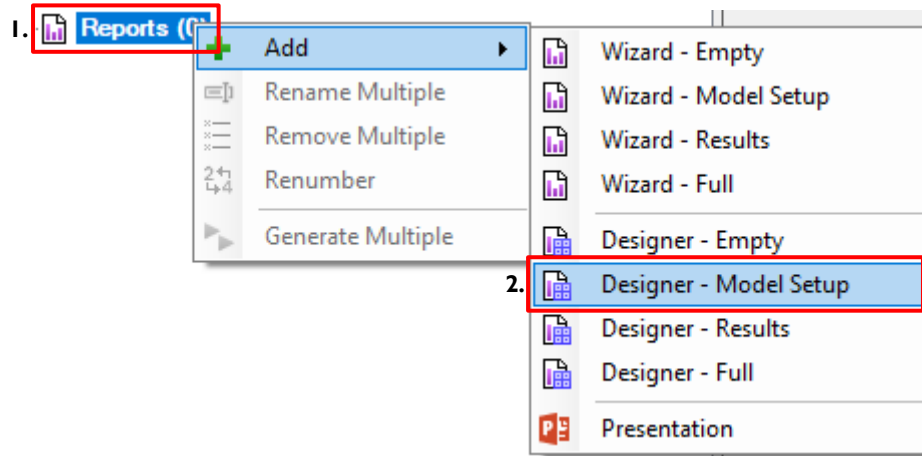
Nodes: 841 | Elements: 429 | SYS-3.mechdb

MKS (Meter/Kg/Second) | Ansys ...

# Add Model Setup Report

1 Press right mouse button  **Reports (0)**

2 Execute *Add – 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;

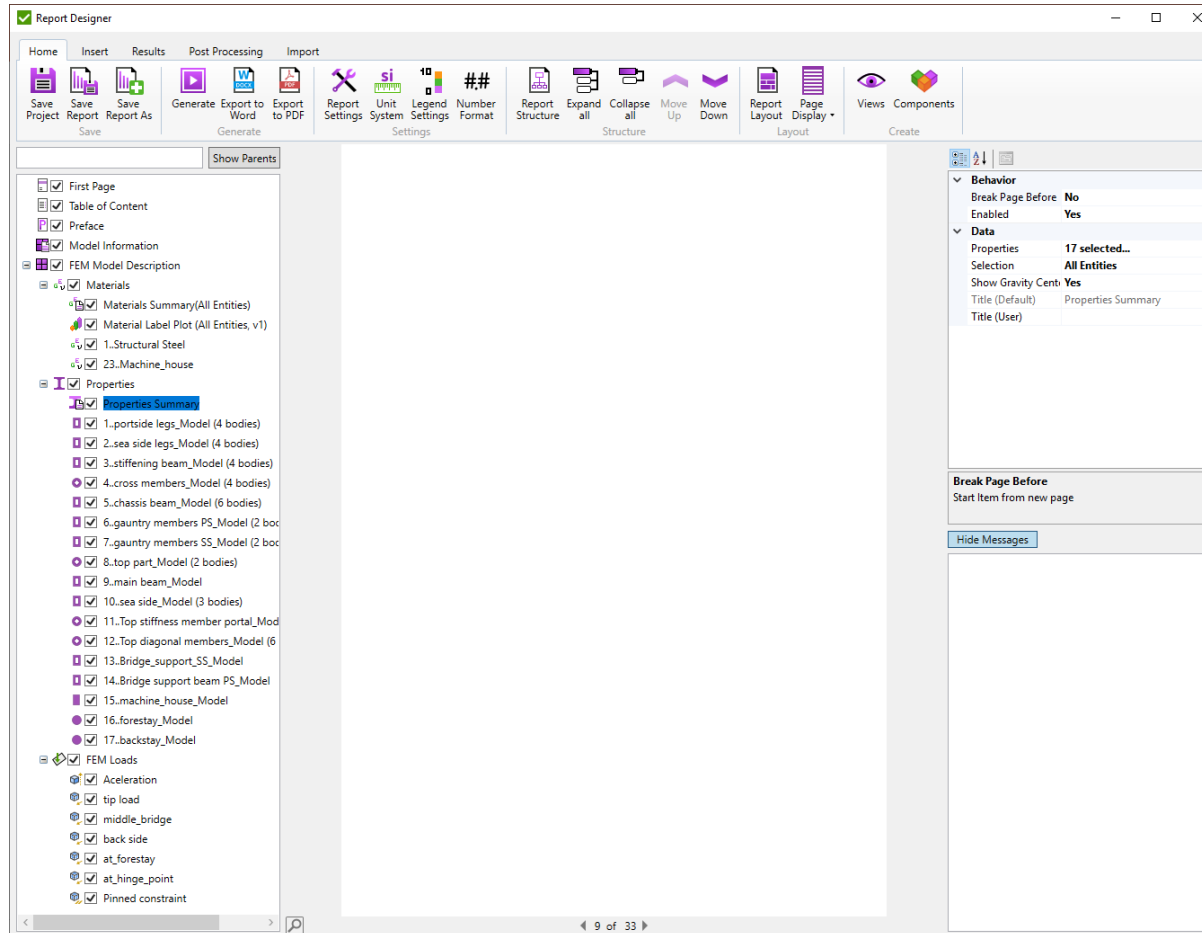
Results – for each load extreme displacement tables, stress and displacement plots are included. Predefined tables: sum of reaction forces, stresses/displacements summary tables;

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

# Report Designer Interface (Components)

Toolbar contains main functions

Report Structure –  
displays structure  
of the report





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

Report document

# Editing First Page

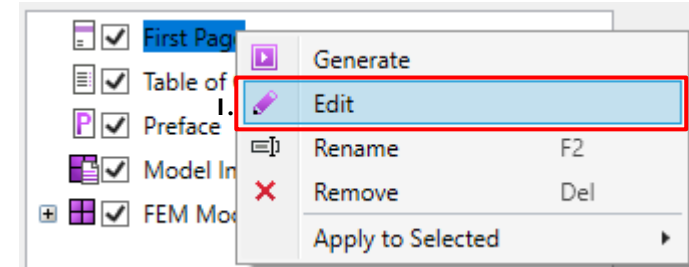
1 Execute *Edit* from First Page context menu


2 Press  and select Support Engineer from the library

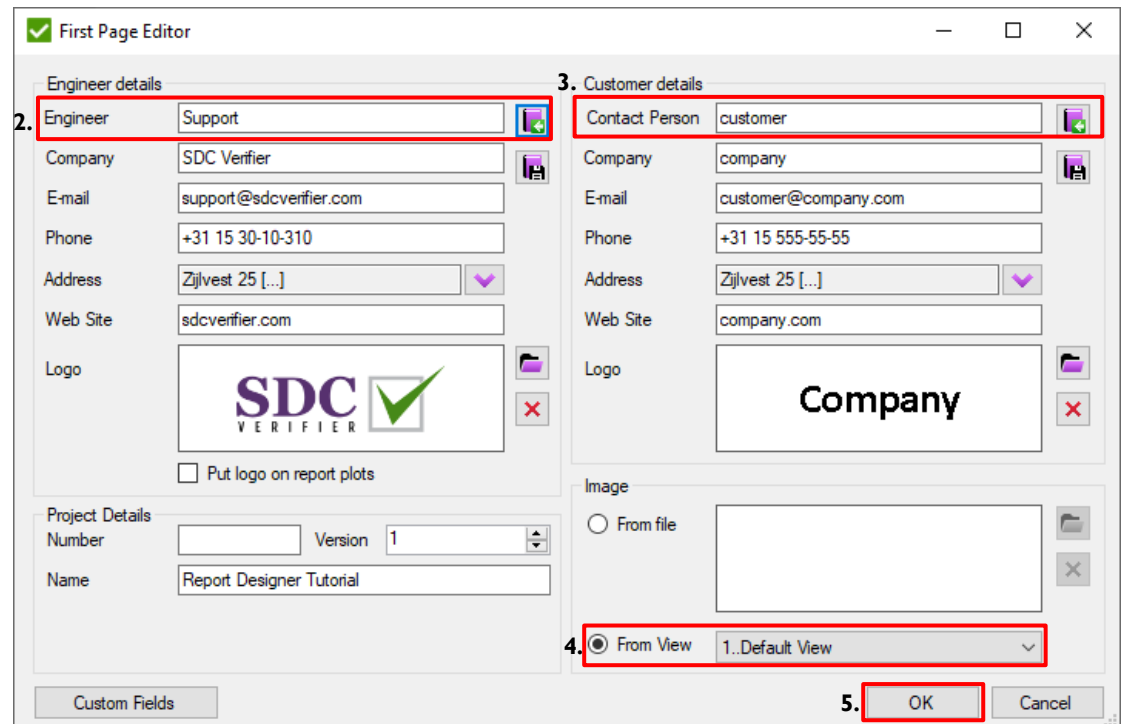
3 Press  and select Customer from the library

4 Image – From View: **Selected**

5 Press *OK*.



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

A screenshot of the 'First Page Editor' dialog box. The dialog is divided into several sections. The 'Engineer details' section on the left has fields for 'Engineer' (Support), 'Company' (SDC Verifier), 'E-mail' (support@sdcverifier.com), 'Phone' (+31 15 30-10-310), 'Address' (Zijvest 25 [...]), 'Web Site' (sdcverifier.com), and 'Logo' (SDC Verifier logo). The 'Customer details' section on the right has fields for 'Contact Person' (customer), 'Company' (company), 'E-mail' (customer@company.com), 'Phone' (+31 15 555-55-55), 'Address' (Zijvest 25 [...]), 'Web Site' (company.com), and 'Logo' (Company logo). At the bottom, there are 'Project Details' fields for 'Number' and 'Version' (1), and a 'Name' field (Report Designer Tutorial). There is also a 'Put logo on report plots' checkbox. The 'Image' section at the bottom right has a radio button for 'From file' and a dropdown for 'From View' (1..Default View). The 'OK' button is highlighted with a red box.



# Generate First Page

1

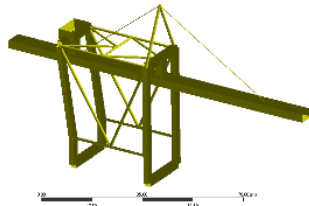
Execute *Generate* from First Page context menu



## Model Setup

Report Designer Tutorial

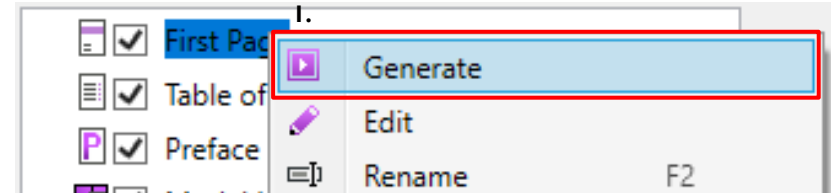
10/10/2020



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

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

Engineer: Support  
Customer: customer  
Project Number:  
Version: 1  
Date: 16/10/2020



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

Report in designer does not contain headers and footers, they are inserted when export to Word Document.

# Generate Preface item

1 Select **Preface** item in report structure

2 Execute *Generate* from context menu

In first paragraph you can find what versions of SDC Verifier and ANSYS 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.

## Preface

This document is generated with SDC Verifier 20.0 and calculated with Ansys v2019 R2  
Model File: E:\Projects\Tutorials\Report Designer 2020 Ansys\Report Designer\Report Designer  
IReportDesigner\_files\dp0\global\MECH\SYS-3.mechdb  
Project File: E:\Projects\Tutorials\Report Designer 2020 Ansys\Report Designer\Report Designer  
IReportDesigner\_files\dp0\SDCvfiACTIReport\_designer.ansv  
Report Profile: 1..Model Setup  
Generation on: 10/16/2020 5:37:44 PM

### Unit System

Current Unit System = MKS (Meter/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		Meter
Mass		Kilogram
Time		Second
Force		Newton
Stress		Pa

# Exclude Item

1

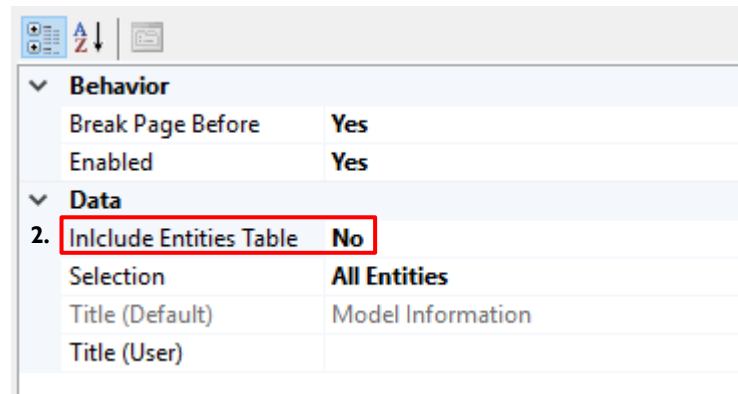
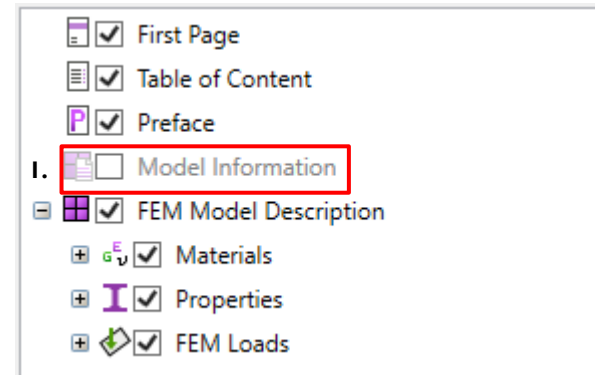
Select **Model Information** item in report structure

2

Set option Enabled: **No** in property toolbox

Model Information (with all sub items) are excluded from the report. The Model Information item will not be generated but remains in the report structure.

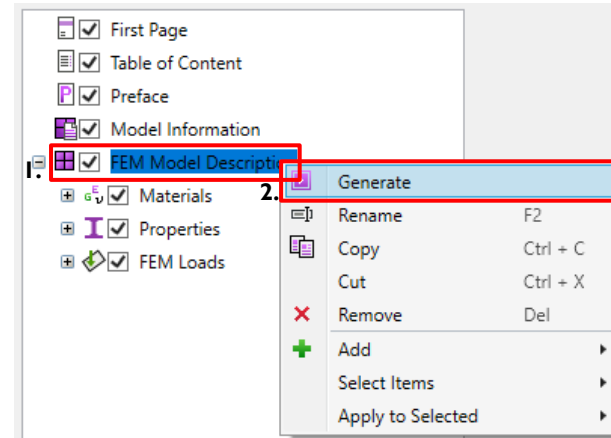
Alternatively, it is possible to delete item from the context menu or using DEL button on keyboard.



# 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)

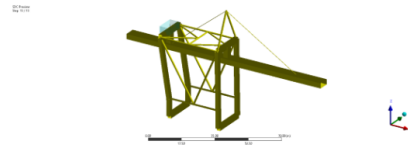
Title	Elements	Mass	Mass Density
1..Structural Steel	420	800285.7	9.812.80
23..Machine_house	1	79992.0	333.30
Summed Over Materials	420	880287.7	10.145.80
Full Model	420	880287.7	10.145.80

Material Summary – mass overview over materials

## 1..Structural Steel

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table S-110.1

Property	Value
Elements	420
Mass [kg]	800285.7
Young Modulus [Pa]	2.00e+11
Shear Modulus [Pa]	7992079923.08
Poisson Ratio	0.30
Shear [Pa]	1.00
Mass Density [kg/m³]	9812.50
Tensile Strength [Pa]	350.00e+6
Yield Stress [Pa]	240.00e+6

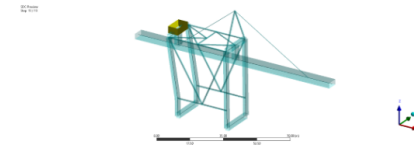


Detailed Material description with plots

## 23..Machine\_house

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table S-110.1

Property	Value
Elements	1
Mass [kg]	79992.0
Young Modulus [Pa]	2.00e+11
Shear Modulus [Pa]	7992079923.08
Poisson Ratio	0.30
Shear [Pa]	1.00
Mass Density [kg/m³]	333.30
Tensile Strength [Pa]	400.00e+6
Yield Stress [Pa]	250.00e+6



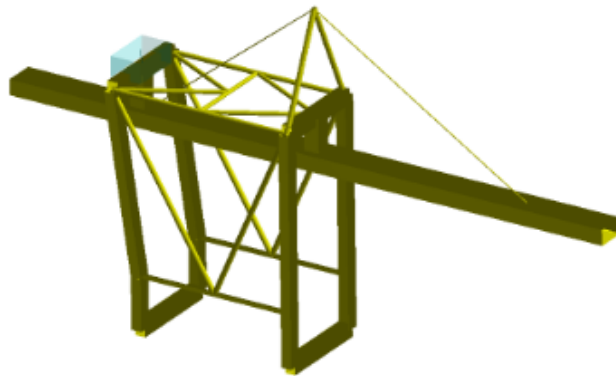
# Material Options

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

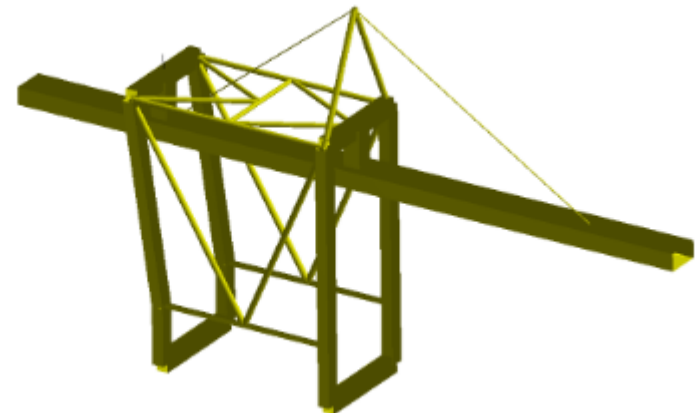
2 Preview Mode: **Display Only Selected**

It is possible to exclude a plot using option – Insert Plot.

▼ Behavior	
Break Page Before	Yes
Enabled	Yes
Include Plot	Yes
Include Selection Min/Max	No
▼ Data	
Selection	All Entities
Title (Default)	1..Structural Steel
Title (User)	
▼ Plot	
Comments	Objects selected: 0
2. Preview Mode	Highlight
Views	1 selected...




Preview Mode: Highlight




Preview Mode: Display Only Selected

# Create View

1

Press  to open View Manager

2

Press  to add View

3

Title: ***Isometric with filled edges.***

4

Locate View in Mechanical as shown on picture

5

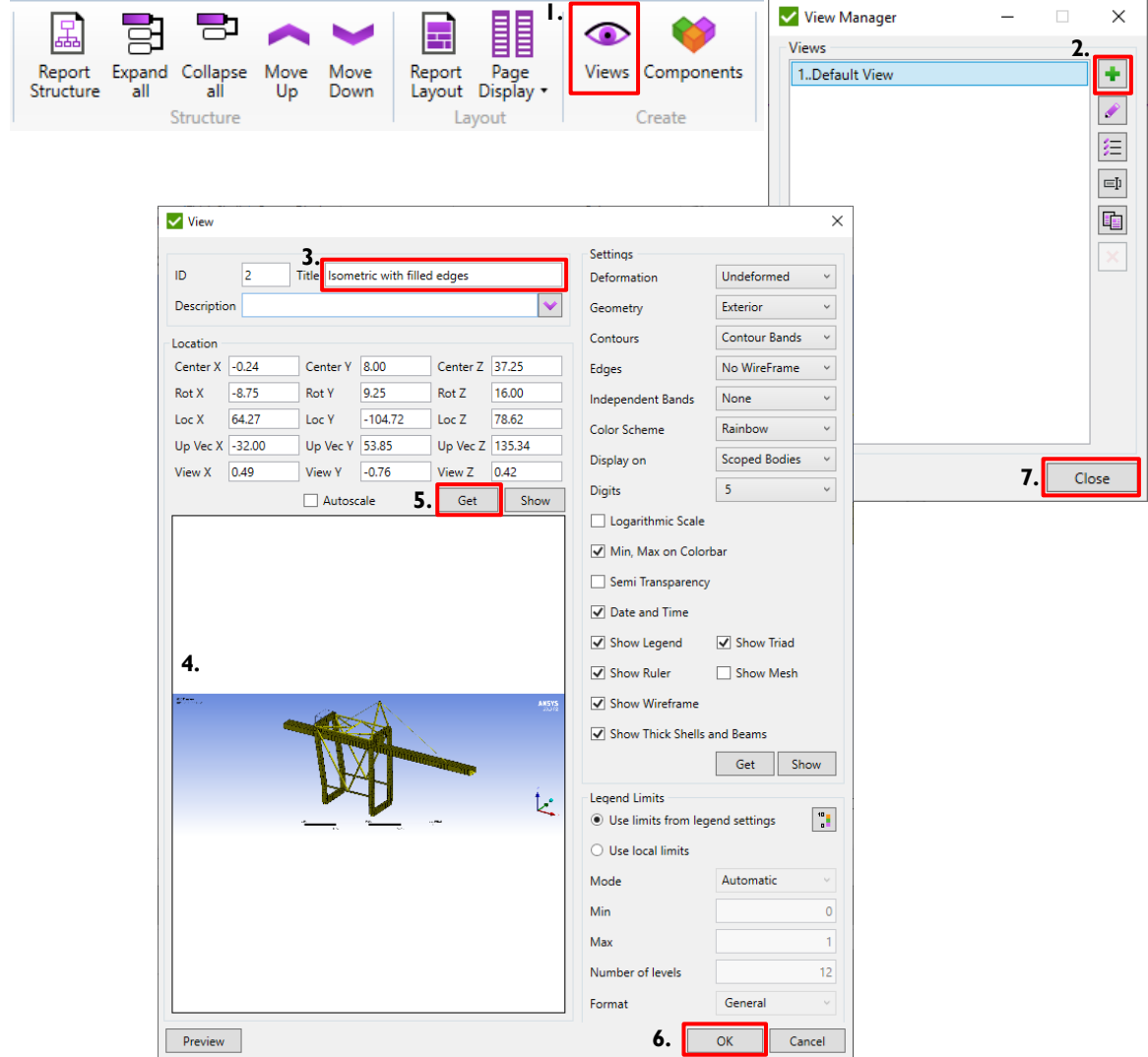
Press *Get*

6

Press *OK*

7

Press *Close*.



# Apply View to Properties

1 Select **Properties** in report structure

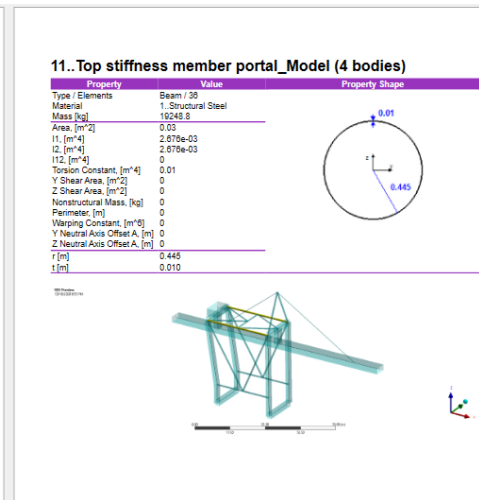
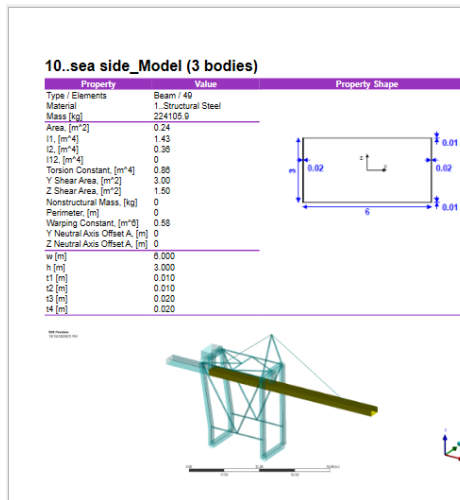
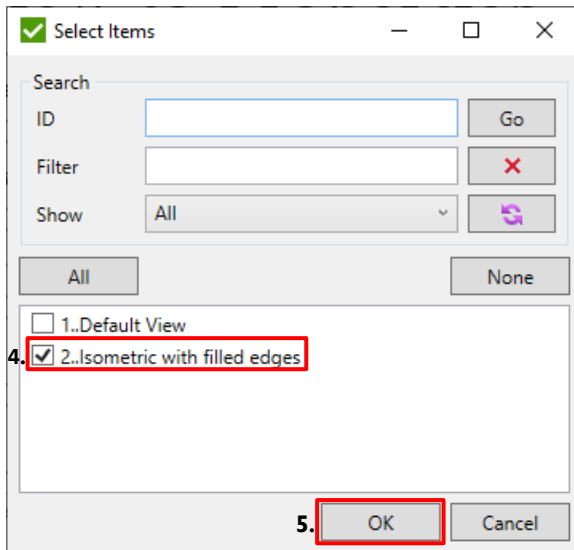
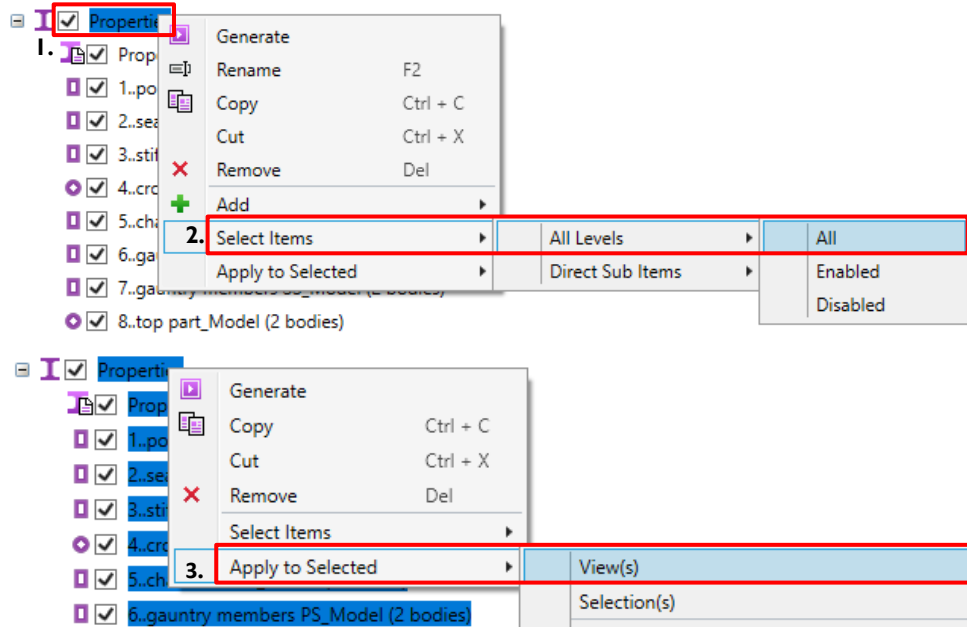
2 Execute *Select items – All Levels – All*

3 Execute *Apply to selected – Views*

4 Select *Isometric with filled edges*

5 Press *Ok*

6 Press *Generate*



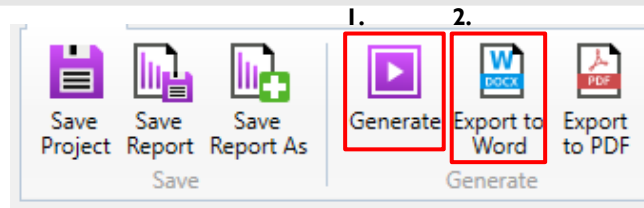
# Generate Report

1

Press  to generate report

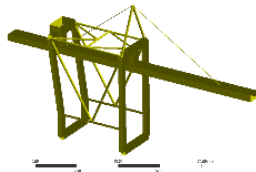
2

Press  to export report to Word



## Model Setup

Report Designer Tutorial



Prepared by:

SDC Verifier

+31 15 30-10-310

sdcoverifier.com

Zijlvest 25  
2011 VS Haarlem  
The Netherlands

Prepared for:

company

+31 15 555-55-55

company.com

Zijlvest 25  
2011 VS Haarlem  
The Netherlands

Engineer: Support

Customer: customer

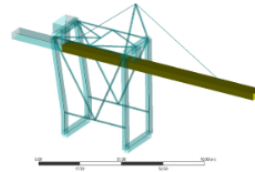
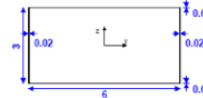
Project Number:

Version: 1

Date: 18/10/2020

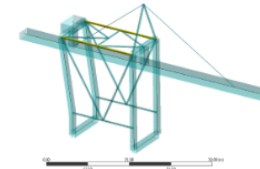
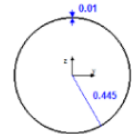
### 10..sea side\_Model (3 bodies)

Property	Value	Property Shape
Type / Elements	Beam / 49	
Material	1. Structural Steel	
Mass [kg]	224105.9	
Area, [m <sup>2</sup> ]	0.24	
I1, [m <sup>4</sup> ]	1.43	
I2, [m <sup>4</sup> ]	0.36	
I12, [m <sup>4</sup> ]	0	
Torsion Constant, [m <sup>4</sup> ]	0.88	
Y Shear Area, [m <sup>2</sup> ]	3.00	
Z Shear Area, [m <sup>2</sup> ]	1.50	
Nonstructural Mass, [kg]	0	
Perimeter, [m]	0	
Warping Constant, [m <sup>6</sup> ]	0.58	
Y Neutral Axis Offset A, [m]	0	
Z Neutral Axis Offset A, [m]	0	
w [m]	6.000	
h [m]	3.000	
t1 [m]	0.010	
t2 [m]	0.010	
t3 [m]	0.020	
t4 [m]	0.020	



### 11..Top stiffness member portal\_Model (4 bodies)

Property	Value	Property Shape
Type / Elements	Beam / 30	
Material	1. Structural Steel	
Mass [kg]	19249.8	
Area, [m <sup>2</sup> ]	0.03	
I1, [m <sup>4</sup> ]	2.676e-03	
I2, [m <sup>4</sup> ]	2.676e-03	
I12, [m <sup>4</sup> ]	0	
Torsion Constant, [m <sup>4</sup> ]	0.01	
Y Shear Area, [m <sup>2</sup> ]	0	
Z Shear Area, [m <sup>2</sup> ]	0	
Nonstructural Mass, [kg]	0	
Perimeter, [m]	0	
Warping Constant, [m <sup>6</sup> ]	0	
Y Neutral Axis Offset A, [m]	0	
Z Neutral Axis Offset A, [m]	0	
r [m]	0.445	
t [m]	0.010	

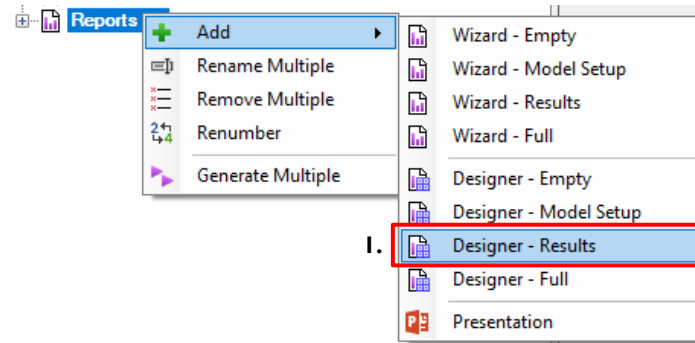




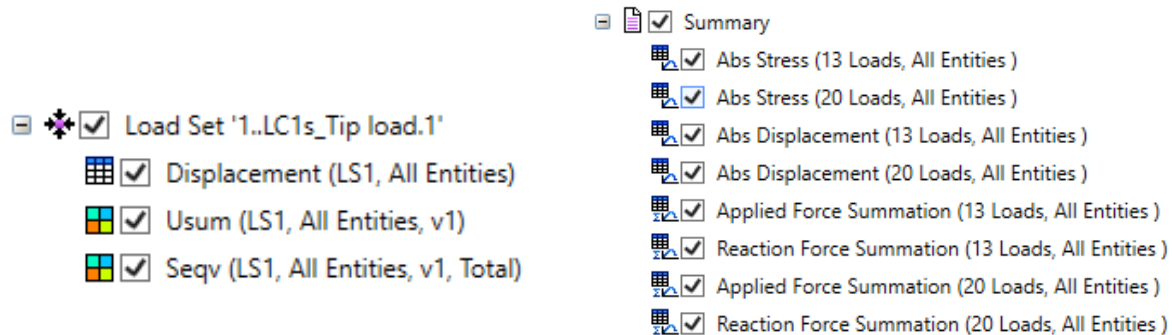
# Add Result Report

1

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



Result report includes predefined items



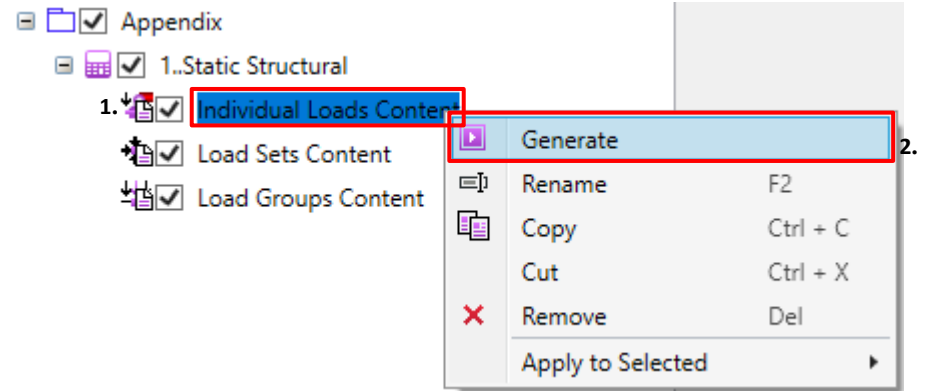
For each load extreme displacement table,  
displacement and stress plots are created

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]	Result Case
1..gravity [1]	Static Structural - step: 1 (0)
2..tip load [1]	Static Structural - step: 2 (0)
3..middle_bridge [1]	Static Structural - step: 3 (0)
4..back side [1]	Static Structural - step: 4 (0)
5..at_forestay [1]	Static Structural - step: 5 (0)
6..at_hinge_point [1]	Static Structural - step: 6 (0)
7..Trolley_ride [1]	Static Structural - step: 7 (0)
8..tip side_load [1]	Static Structural - step: 8 (0)
9..middle_bridge_side_load [1]	Static Structural - step: 9 (0)
10..back side_side_load [1]	Static Structural - step: 10 (0)
11..at_forestay_side_load [1]	Static Structural - step: 11 (0)
12..at_hinge_point_side_load [1]	Static Structural - step: 12 (0)
13..Crane_ride [1]	Static Structural - step: 13 (0)

Content shows the list of Individual loads and referenced step.

# Load Set Content

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

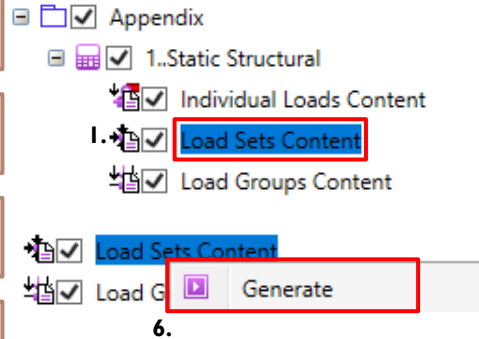
2 In Toolbox for **All Load Sets** choose **No**

3 Select **Load Sets** and press ...

4 In menu **Select Items** select all load sets what should be displayed

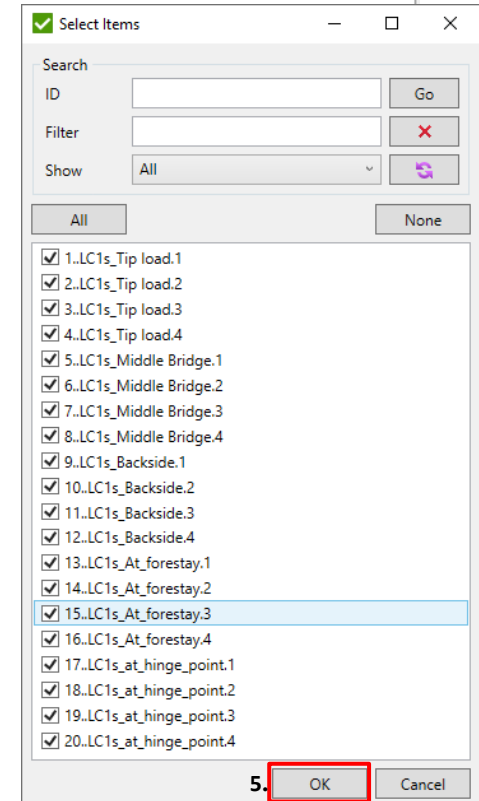
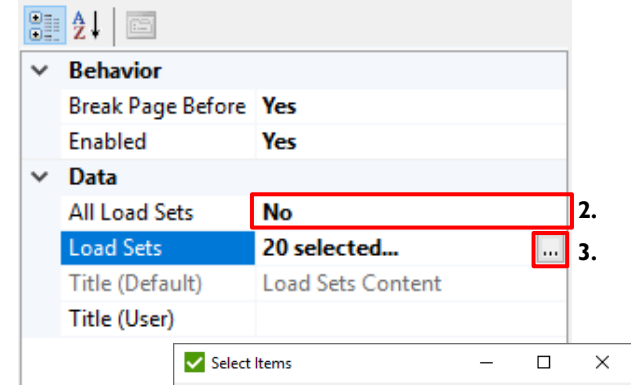
5 Press **OK**

6 Select **Load Sets Content** and Execute **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 [-1.15] 7..Trolley_ride [1.35] 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 [-1.15] 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_side_load [1.15] 13..Crane_ride [1.15]
10..LC1s_Backside.2 [1]	5	1..gravity [1.15] 4..back side [-1.15] 7..Trolley_ride [-1.15] 10..back side_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_side_load [1.15] 13..Crane_ride [1.15]
12..LC1s_Backside.4 [1]	5	1..gravity [1.15] 4..back side [-1.15]



13..LC1s_At_forestay.1 [1]	5	7..Trolley_ride [-1.15] 10..back side_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]

# Individual Load Options

1 Select **5..at\_forestay** in report structure

2 Execute *Generate* from context menu

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

<b>Behavior</b>	
Break Page Before	Yes
Enabled	Yes
<b>Data</b>	
Job	1..Static Structural
Load Type	IndividualLoad
Title (Default)	Individual Load '5..at_forestay'
Title (User)	
<b>Options</b>	
Include Load Item Content	Yes
Include Sum Of Forces	Yes
Selection	All Entities

- 1..Static Structural
  - Individual Loads
    - Individual Load '1..gravity'
    - Individual Load '2..tip load'
    - Individual Load '3..middle\_bridge'
    - Individual Load '4..back side'
    - Individual Load '5..at\_forestay'

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

Title	Value
Individual Load	5..at_forestay
Type	Imported from Result Case
Result Case	Static Structural - step: 5 (0)
Safety Factor	1

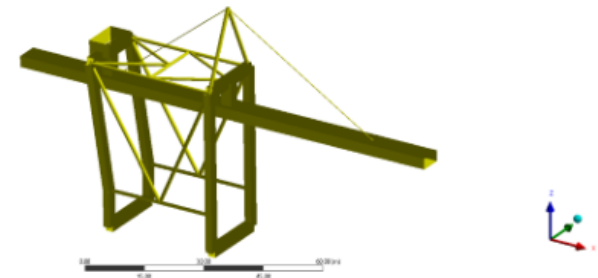
### Sum of Reaction Forces

Load	Fx [N]	Fy [N]	Fz [N]	Fsum [N]	Mx [N m]	My [N m]	Mz [N m]	Msum [N m]
All Entities	0	0	1220000	1220000	0.0	0.0	0.0	0.0

### 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
Minimum		0.00	0.00	-0.12	0.00	-0.04	0.00	-0.01
Maximum		0.03	0.00	0.02	0.12	0.04	0.13	0.01
Absolute		0.03	0.00	-0.12	0.12	-0.04	0.13	-0.01

1000 Probes  
1000000 1000000



# Number Formats

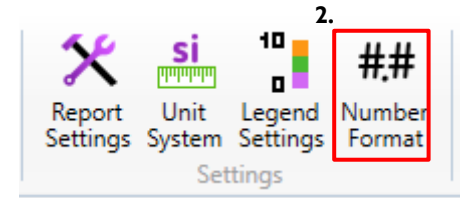
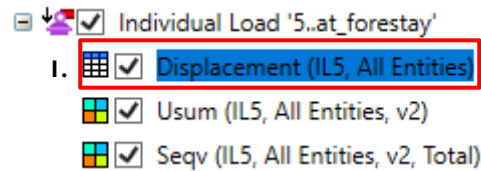
1. Generate **Displacement (All Entities)** under load **5..at\_forestay**

2. Press **##** to open Number Formats

3. Digits after decimal point: **2** for *Displacement* category and **2** for *General* category (Rotation [Rx, Ry, Rz] )

4. Press *Close*

5. Execute *Generate* from context menu



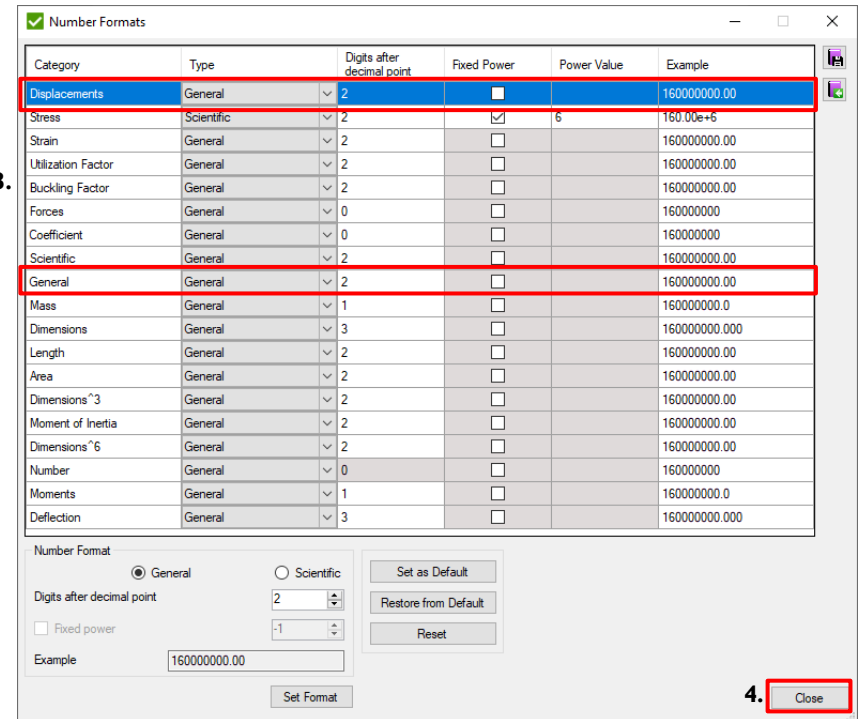
Displacement (IL5, All Entities)									
Individual Load Type	5..at_forestay Extreme			Selection Category		All Entities Displacement			
Extreme	Ux [m]	Uy [m]	Uz [m]	Usum [m]	Rx	Ry	Rz	Rsum	
Minimum	0.00	0.00	-0.12	0.00	-0.04	0.00	-0.01	0.00	
Maximum	0.03	0.00	0.02	0.12	0.04	0.13	0.01	0.13	
Absolute	0.03	0.00	-0.12	0.12	-0.04	0.13	-0.01	0.13	

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	-0.005	-0.120	0.000	-0.04	0.00	-0.01	0.00	
Maximum	0.035	0.005	0.015	0.120	0.04	0.13	0.01	0.13	
Absolute	0.035	-0.005	-0.120	0.120	-0.04	0.13	-0.01	0.13	

Digits after decimal point = 3


3.



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

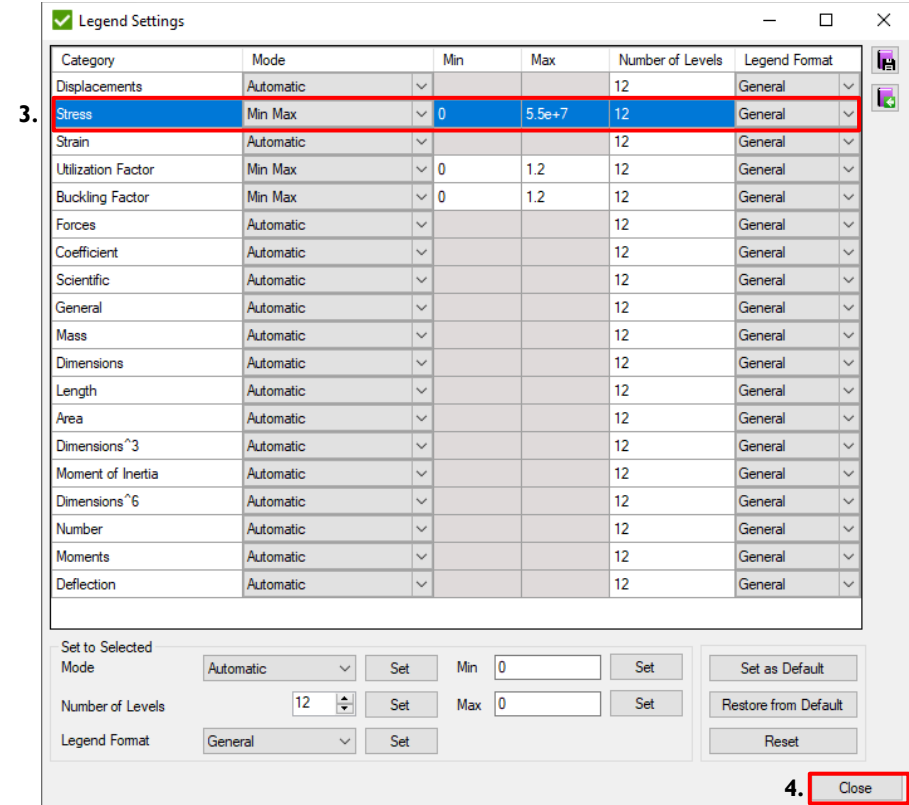
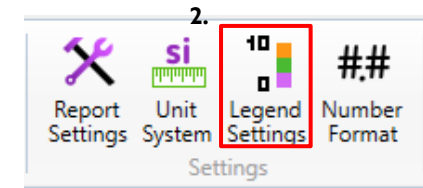
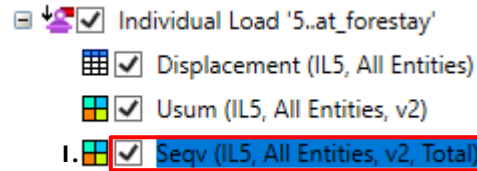
1. Select **Seqv (All Entities. V1. Total)** table under load **5..at\_forestay**


2. Press  to open Legend Settings

3. Max: **5.5e+7** for Stress category

4. Press *Close*

5. Execute *Generate* from context menu



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

Execute **Abs Displacement (LS)** under **Summary** item

2

In context menu select **Generate**

- ☑ 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 )

Abs Stress (20 Loads, All Entities )

Loads Count	20	Category	Stress
Selection	All Entities	Type	Extreme
Parameter	Abs		
Load	X [Pa]	Y [Pa]	Z [Pa]
LS1..LC1s_Tip load.1	149.13e+6		
LS2..LC1s_Tip load.2	149.13e+6		
LS3..LC1s_Tip load.3	149.81e+6		
LS4..LC1s_Tip load.4	149.81e+6		
LS5..LC1s_Middle Bridge.1	48.83e+6		
LS6..LC1s_Middle Bridge.2	48.83e+6		
LS7..LC1s_Middle Bridge.3	49.51e+6		
LS8..LC1s_Middle Bridge.4	49.51e+6		
LS9..LC1s_Backside.1	50.46e+6		
LS10..LC1s_Backside.2	50.46e+6		
LS11..LC1s_Backside.3	51.14e+6		
LS12..LC1s_Backside.4	51.14e+6		
LS13..LC1s_At_forestay.1	124.91e+6		
LS14..LC1s_At_forestay.2	124.91e+6		
LS15..LC1s_At_forestay.3	125.59e+6		
LS16..LC1s_At_forestay.4	125.59e+6		
LS17..LC1s_at_hinge_point.1	52.81e+6		
LS18..LC1s_at_hinge_point.2	52.81e+6		
LS19..LC1s_at_hinge_point.3	53.49e+6		
LS20..LC1s_at_hinge_point.4	53.49e+6		

Stresses for all load sets

Abs Displacement (20 Loads, All Entities )

Loads Count	20	Category	Displacement
Selection	All Entities	Type	Extreme
Parameter	Abs		
Load	Ux [m]	Uy [m]	Uz [m]
LS1..LC1s_Tip load.1	0.09	0.03	-0.37
LS2..LC1s_Tip load.2	0.09	-0.03	-0.37
LS3..LC1s_Tip load.3	0.08	0.03	-0.37
LS4..LC1s_Tip load.4	0.08	-0.03	-0.37
LS5..LC1s_Middle Bridge.1	0.02	-0.04	-0.08
LS6..LC1s_Middle Bridge.2	0.02	0.04	-0.08
LS7..LC1s_Middle Bridge.3	-0.03	-0.04	-0.07
LS8..LC1s_Middle Bridge.4	-0.03	0.04	-0.07
LS9..LC1s_Backside.1	-0.02	-0.05	-0.16
LS10..LC1s_Backside.2	-0.02	0.05	-0.16
LS11..LC1s_Backside.3	-0.04	-0.05	-0.16
LS12..LC1s_Backside.4	-0.04	0.05	-0.16
LS13..LC1s_At_forestay.1	0.07	-0.03	-0.25
LS14..LC1s_At_forestay.2	0.07	0.03	-0.25
LS15..LC1s_At_forestay.3	0.05	-0.03	-0.24
LS16..LC1s_At_forestay.4	0.05	0.03	-0.24
LS17..LC1s_at_hinge_point.1	0.03	-0.02	-0.09
LS18..LC1s_at_hinge_point.2	0.03	0.02	-0.09
LS19..LC1s_at_hinge_point.3	-0.02	-0.02	-0.09
LS20..LC1s_at_hinge_point.4	-0.02	0.02	-0.09

Displacements for all load sets

Stress and displacement extreme flow tables give nice results overview among loads.

# Reaction Forces

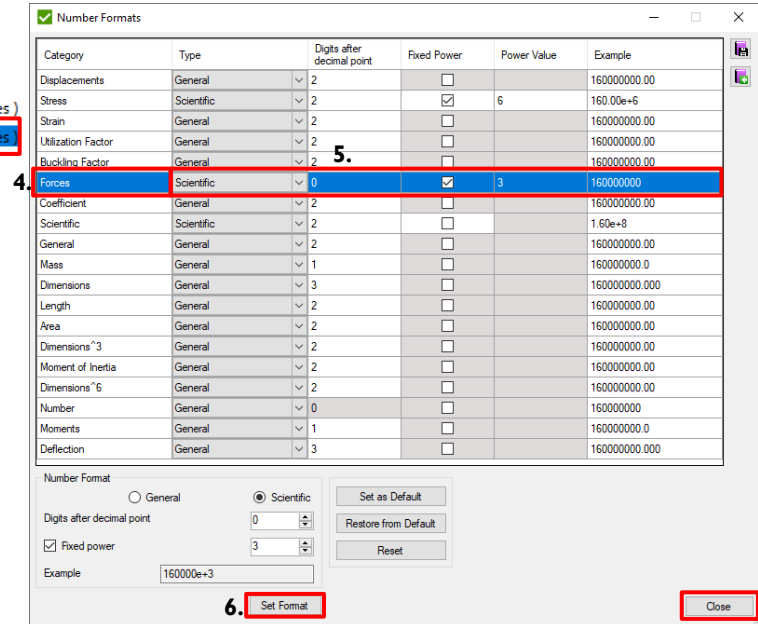
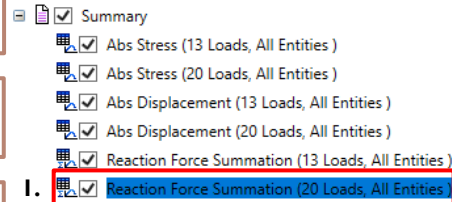
1 Select **Reaction Forces Summation** under Summary

2 Press **##** to open Number Format

3 Select category **Forces**

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

5 Press **Set Format** and **Close**



Reaction Force Summation (20 Loads, All Entities)

Loads Count	20	Category		Reaction Force					
Selection	All Entities	Type		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	197270	11576489	11580319	0.0	0.0	0.0	0.0	0.0
LS2..LC1s_Tip load.2	-223100	-197270	11576489	11580319	0.0	0.0	0.0	0.0	0.0
LS3..LC1s_Tip load.3	223100	197270	11576489	11580319	0.0	0.0	0.0	0.0	0.0
LS4..LC1s_Tip load.4	223100	-197270	11576489	11580319	0.0	0.0	0.0	0.0	0.0
LS5..LC1s_Middle Bridge.1	-223100	197270	11576488	11580318	0.0	0.0	0.0	0.0	0.0
LS6..LC1s_Middle Bridge.2	-223100	-197270	11576488	11580318	0.0	0.0	0.0	0.0	0.0
LS7..LC1s_Middle Bridge.3	223100	197270	11576488	11580318	0.0	0.0	0.0	0.0	0.0
LS8..LC1s_Middle Bridge.4	223100	-197270	11576488	11580318	0.0	0.0	0.0	0.0	0.0
LS9..LC1s_Backside.1	-223100	197270	11576488	11580318	0.0	0.0	0.0	0.0	0.0


Reaction Force Summation (20 Loads, All Entities)

Loads Count	20	Category		Reaction Force					
Selection	All Entities	Type		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	197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS2..LC1s_Tip load.2	-223e+3	-197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS3..LC1s_Tip load.3	223e+3	197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS4..LC1s_Tip load.4	223e+3	-197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS5..LC1s_Middle Bridge.1	-223e+3	197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS6..LC1s_Middle Bridge.2	-223e+3	-197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS7..LC1s_Middle Bridge.3	223e+3	197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS8..LC1s_Middle Bridge.4	223e+3	-197e+3	11576e+3	11580e+3	0.0	0.0	0.0	0.0	0.0
LS9..LC1s_Backside.1	-223e+3	197e+3	11576e+3	11580e+3	0.0	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

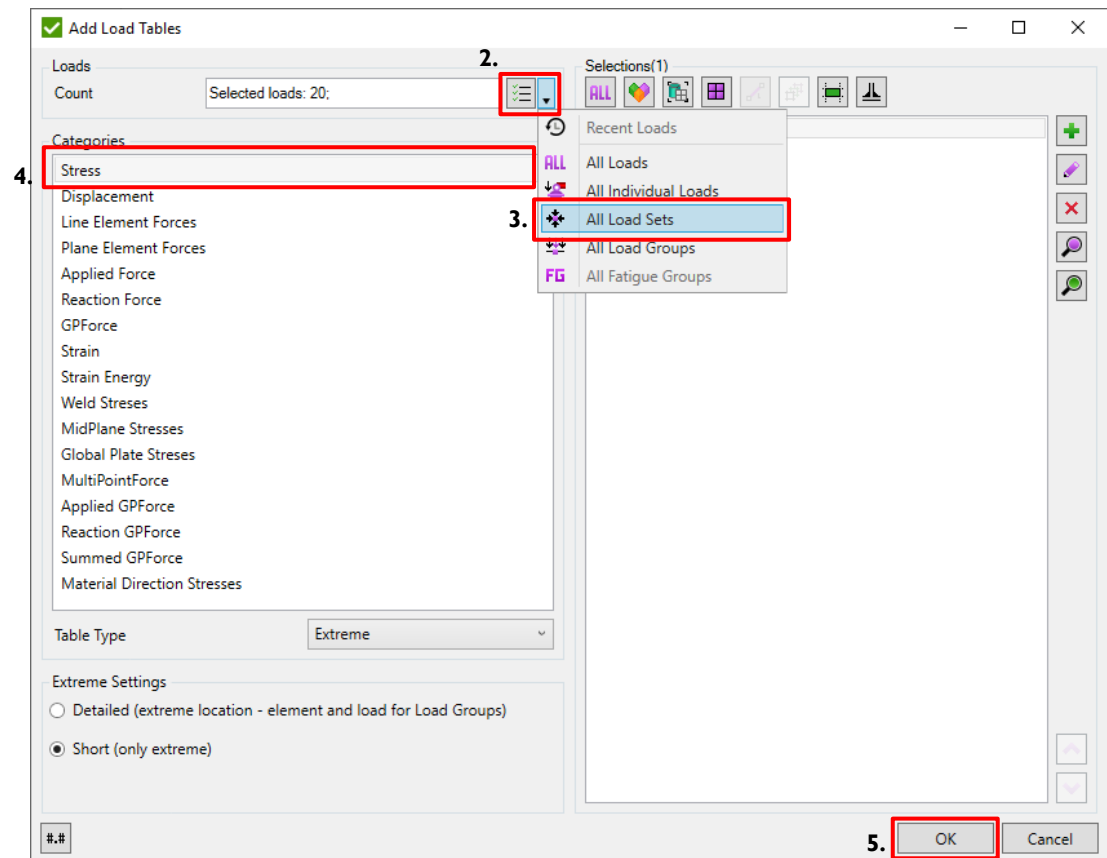
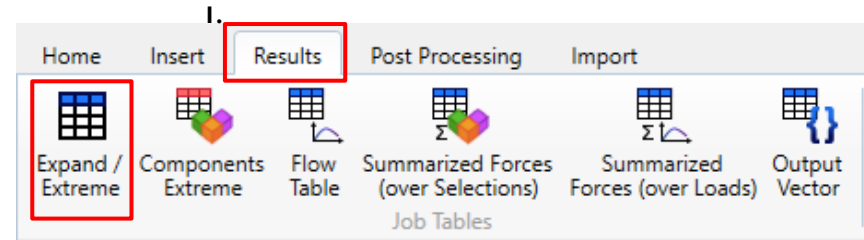
1 Press Results on the toolbar and select  to open tables window

2 Use dropdown menu for load selector

3 Select all Load Sets

4 Categories: **Stress**

5 Press **OK**



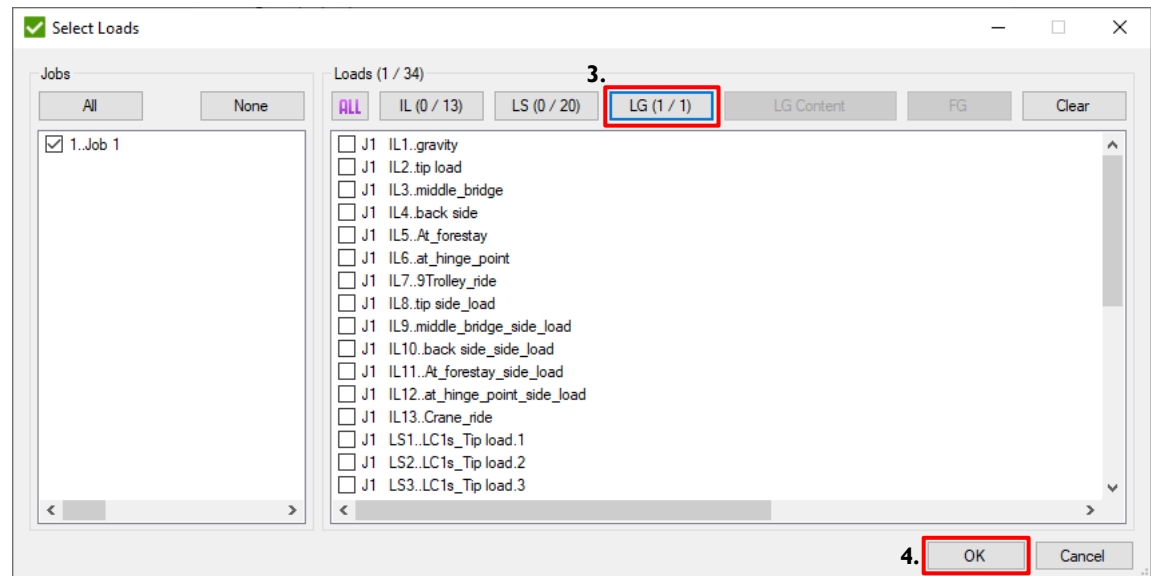
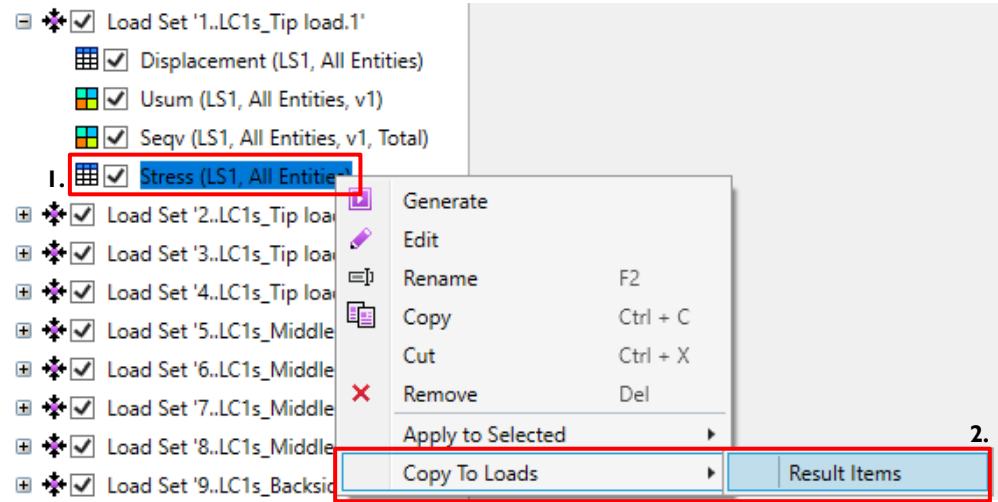
# Copy Table to Load Group

1 Execute **Stress Table** under Load Set

2 Select **Result Items** from context menu

3 Table Type: **Load Group**

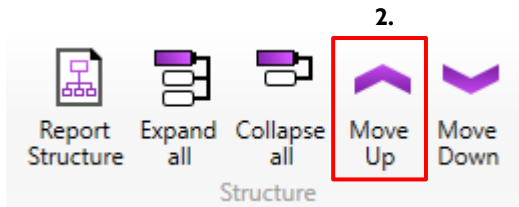
4 Press **OK**



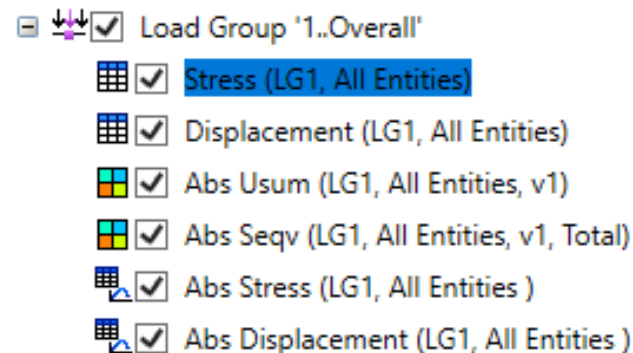
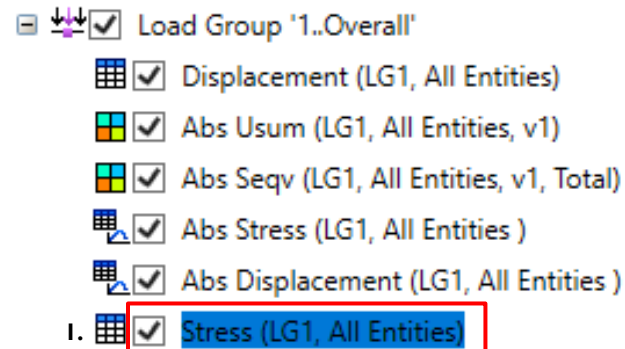
# Move item in the structure

1 Select **Stress (All Entities)**

2 Press *Move Up* item



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

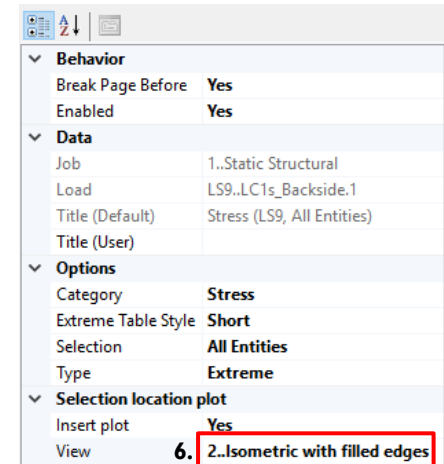
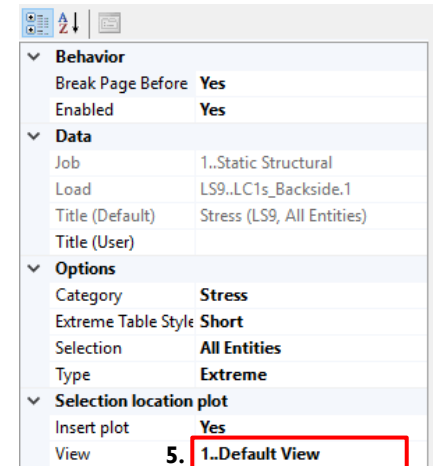
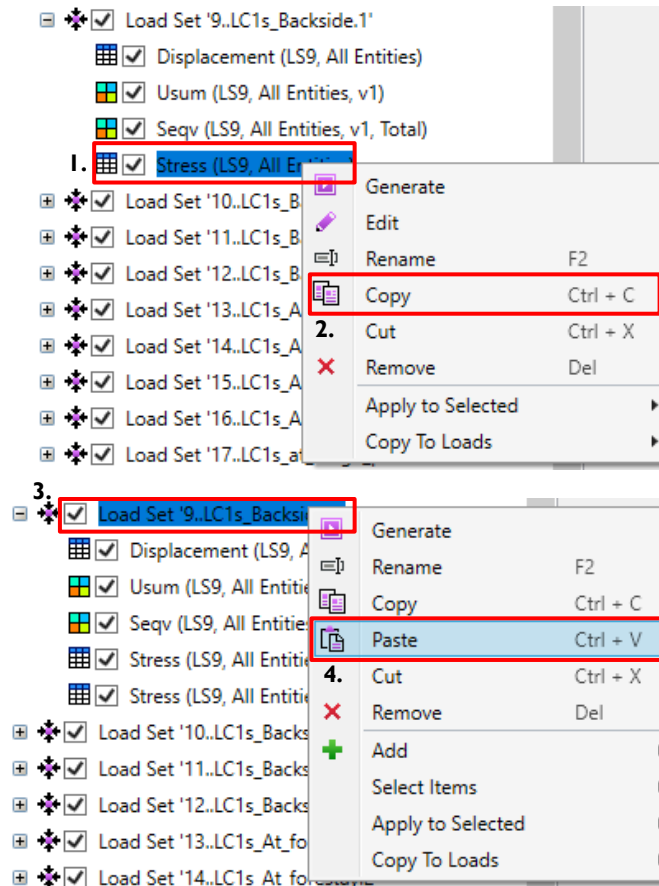
2 Select **Copy**

3 Execute **Load Set**

4 In context menu select **Paste**

5 For first **Stress Table** set View **1..Default View** in Property Grid

6 For second **Stress Table** set View = **2.. Isometric with filled edges**



# 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

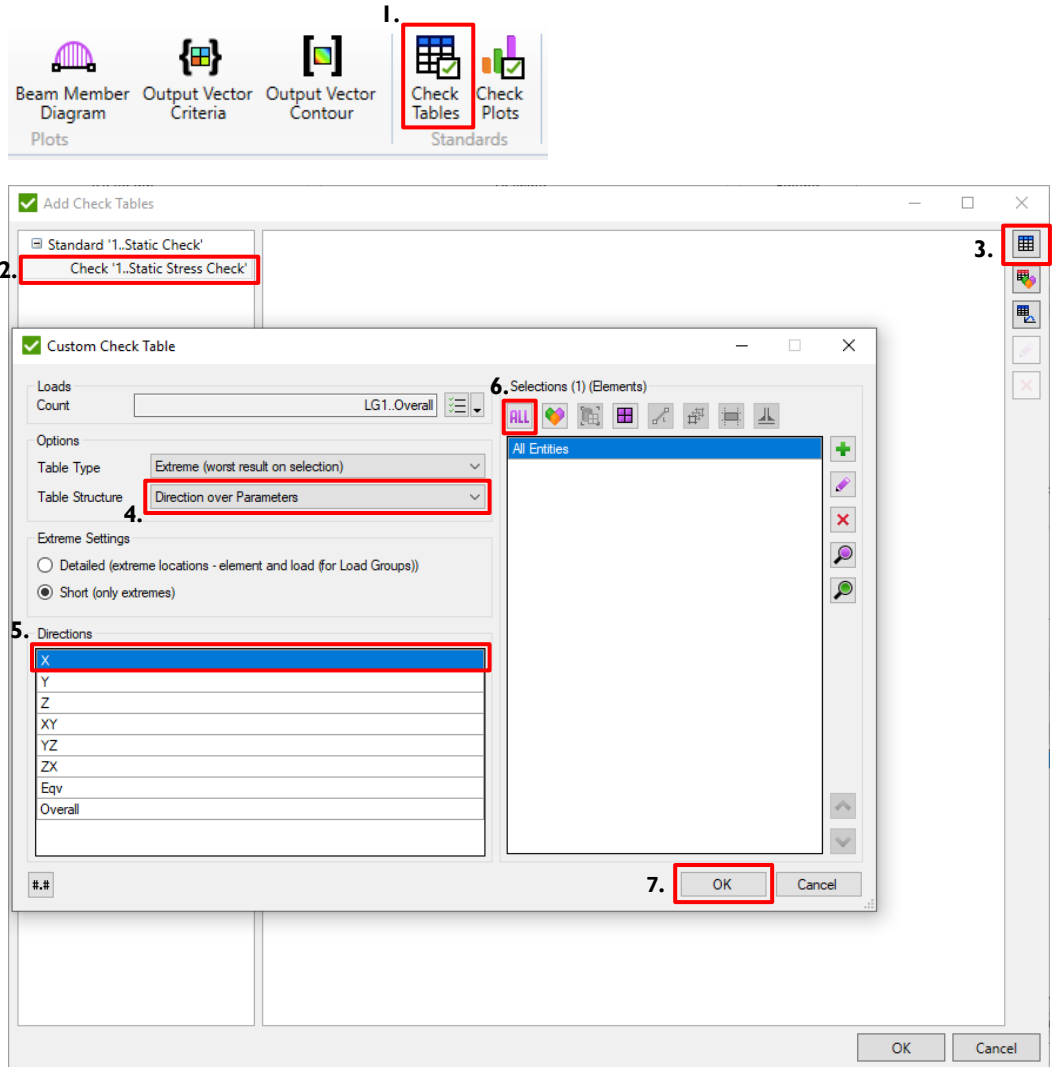
Direction: **X**

6

Press  to add full model selection

7

Press **OK**



# Add Plot for Static Stress check

1

Select **Check Plots** on the Toolbar

2

Select **Static Stress Check**

3

Press **Check Plots**

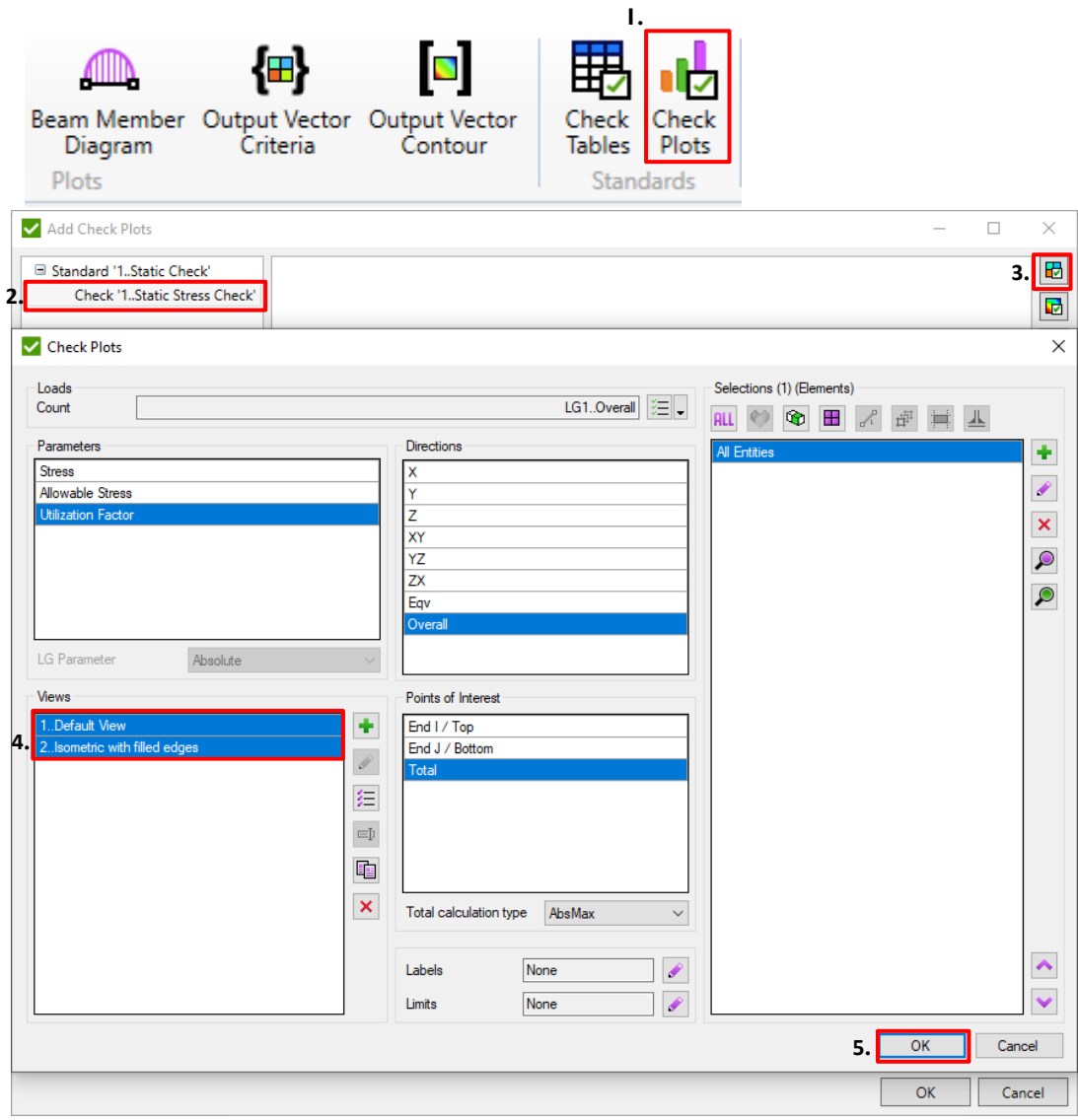


4

Select Views with IDs 1-2

5

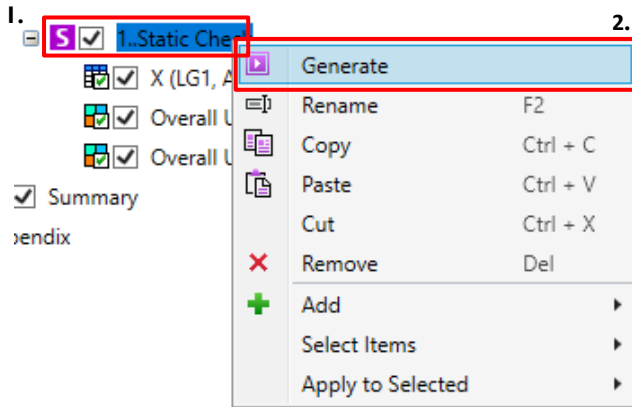
Press **OK**



# Generate Static Stress Check results

1. Select **Static Stress Check**

2. Execute *Generate* from context menu



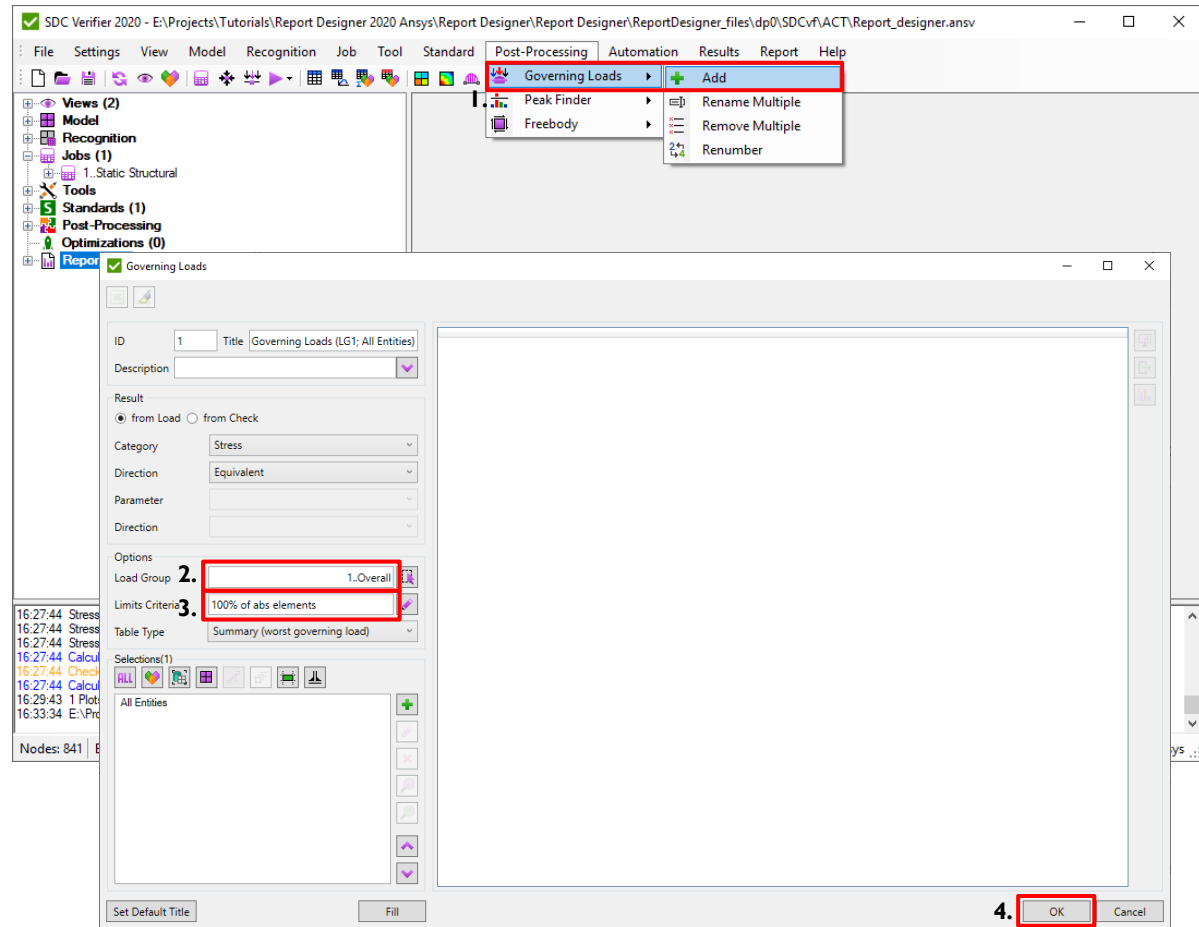
# Add Governing Loads

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

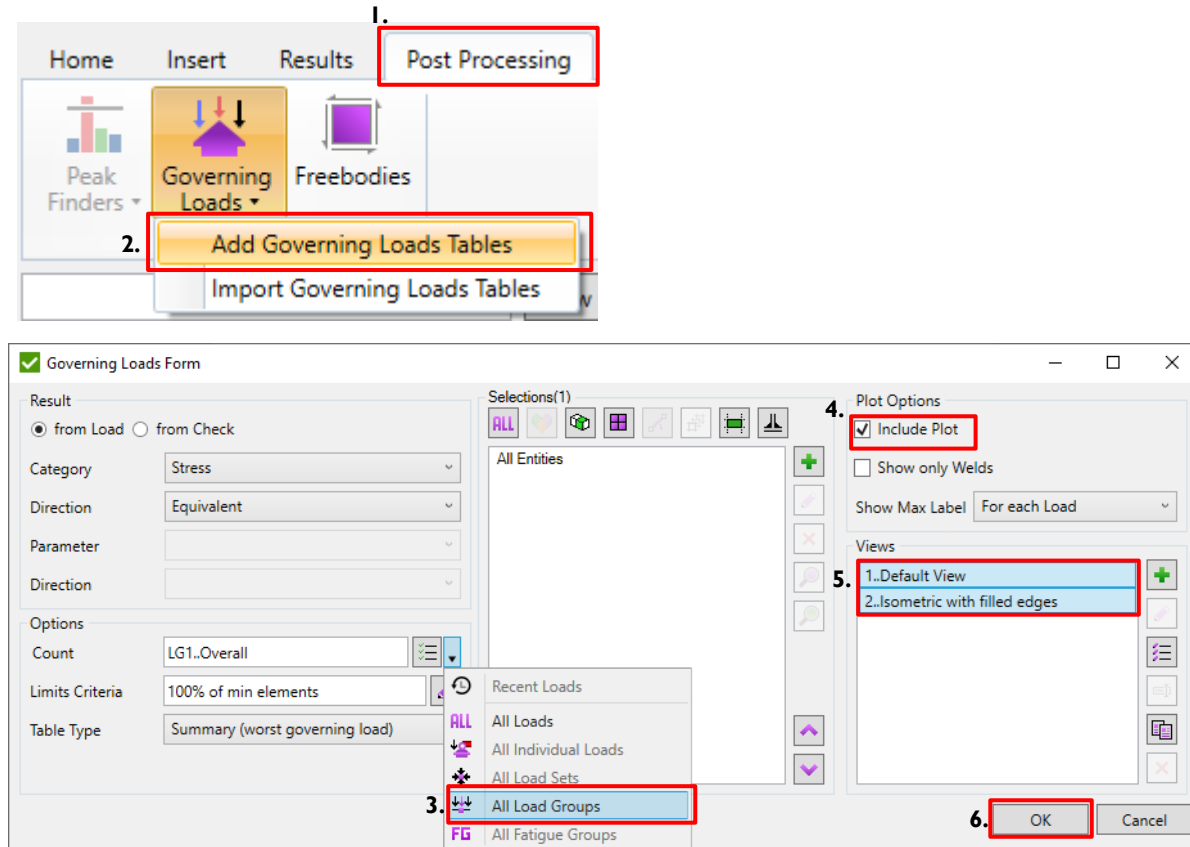
2. Select **Add Governing Load Tables**

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

4. Press **Include Plot**

5. Select Views with IDs 1-2

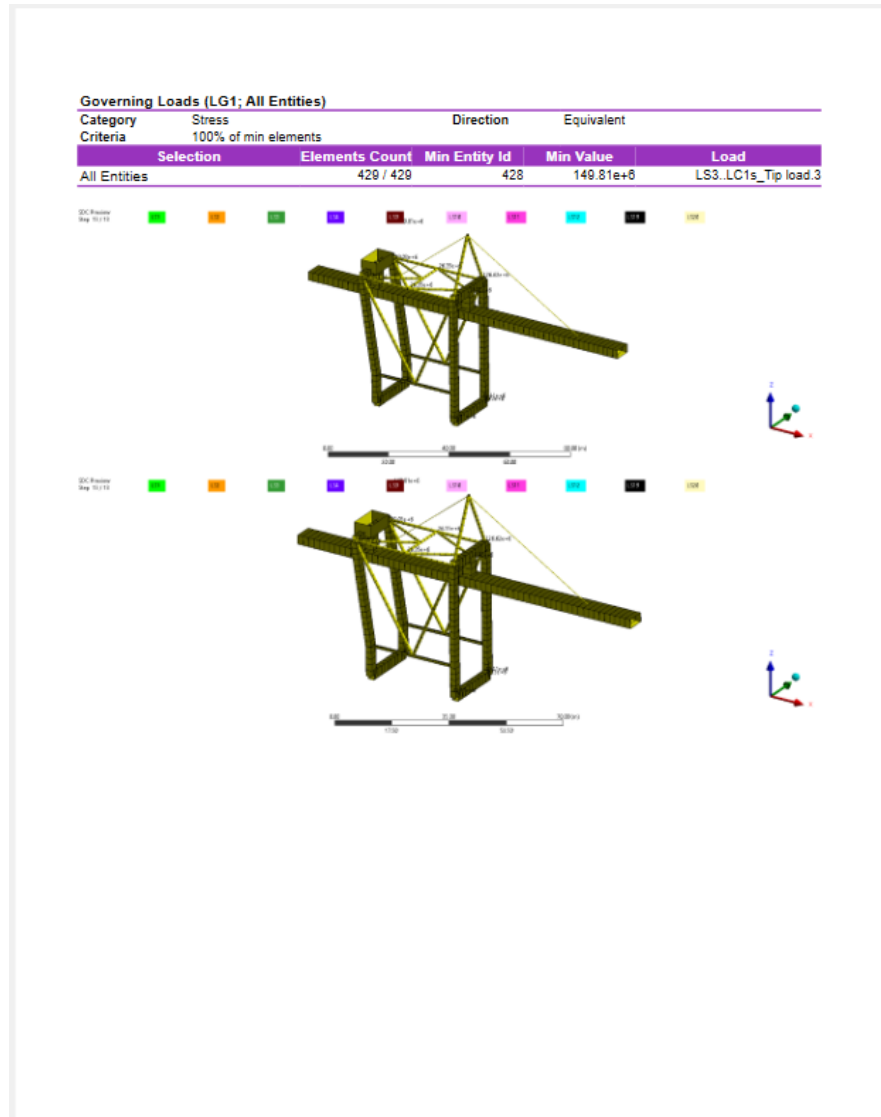
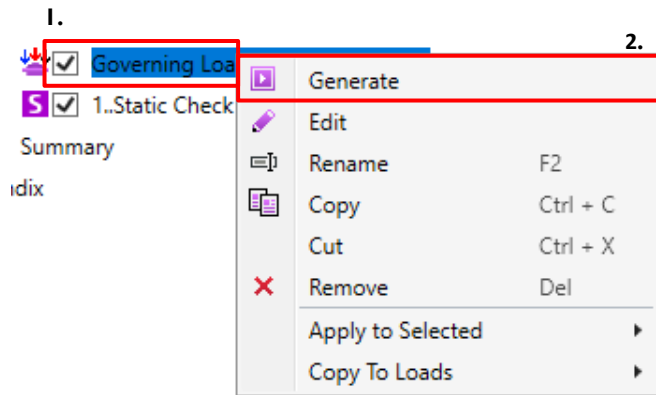
6. Press **OK**



# Generate Governing Loads results

1. Select **Governing Loads**

2. Execute *Generate* from context menu



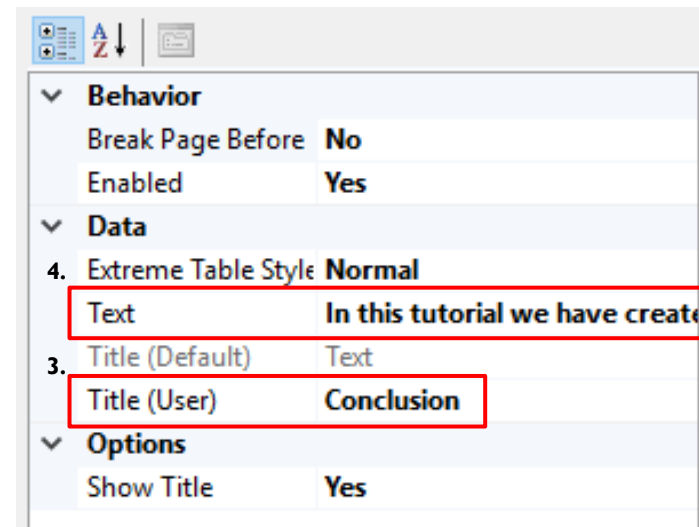
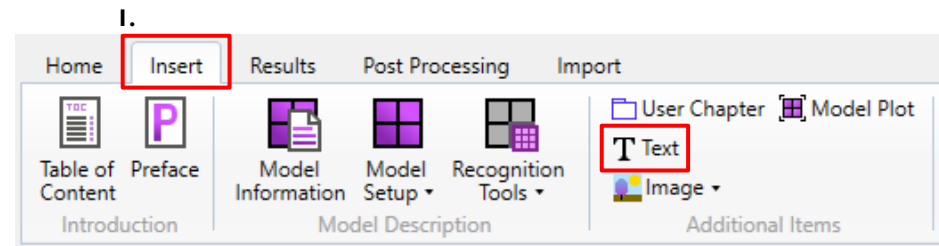
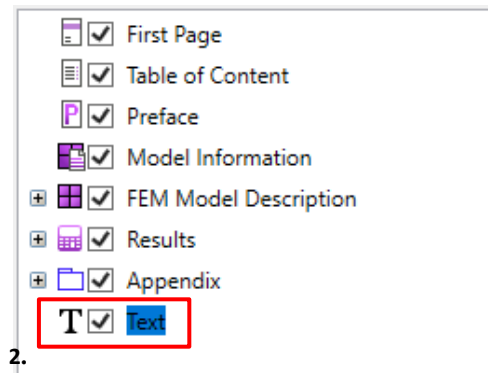
# Add Conclusion

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**

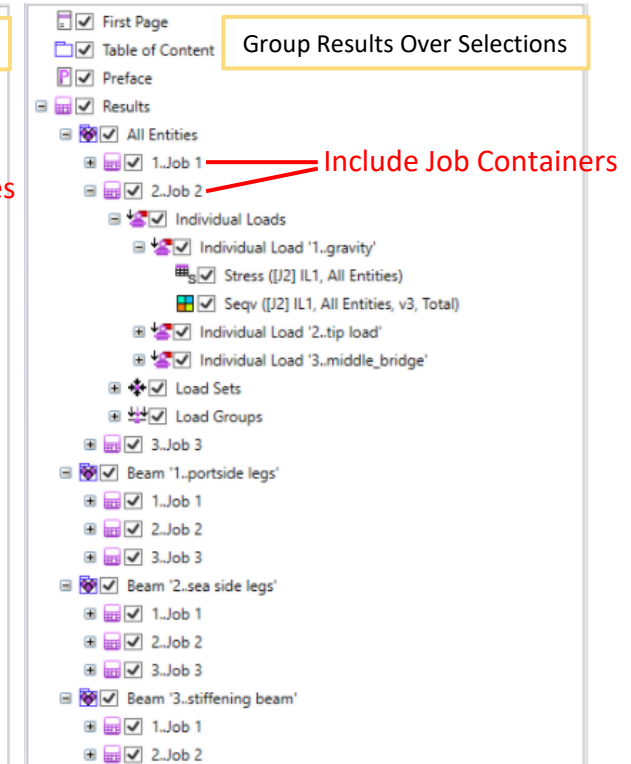
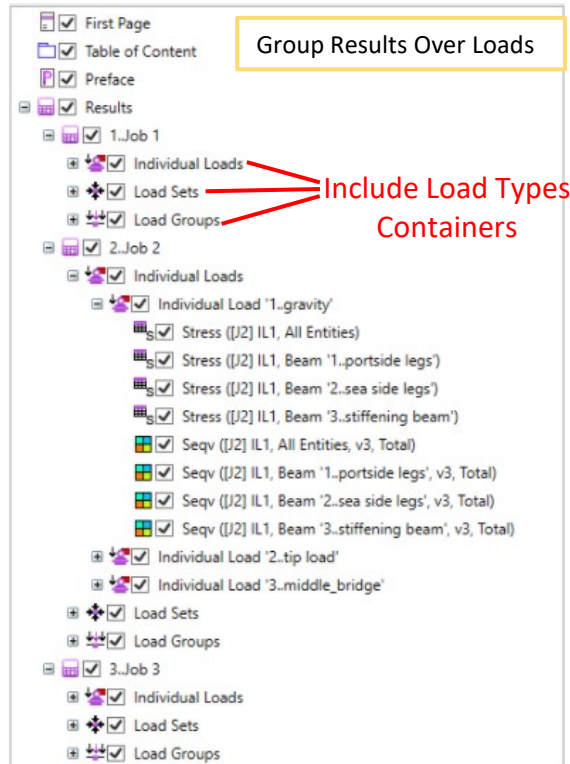
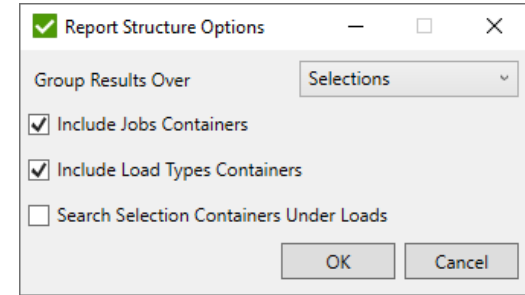
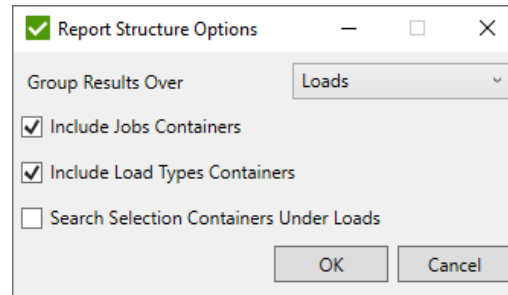
4. Text: **In this tutorial we have created 2 reports using Report Designer**



# 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

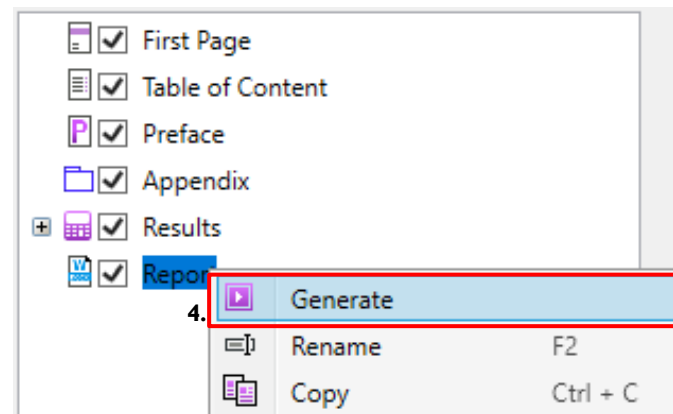
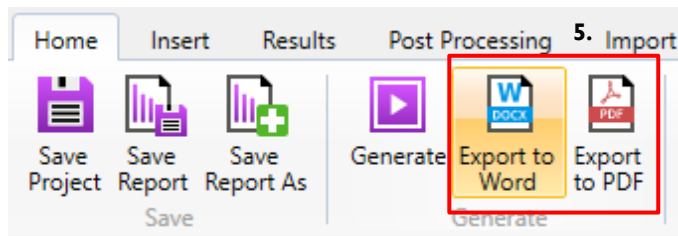
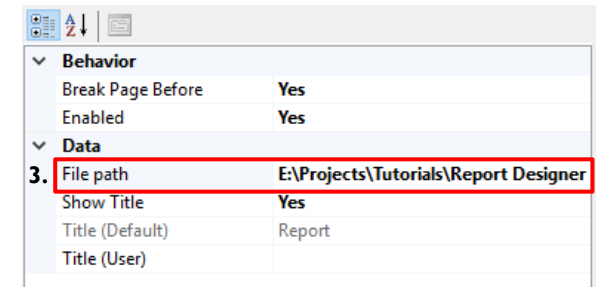
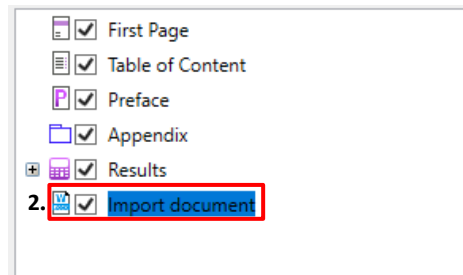
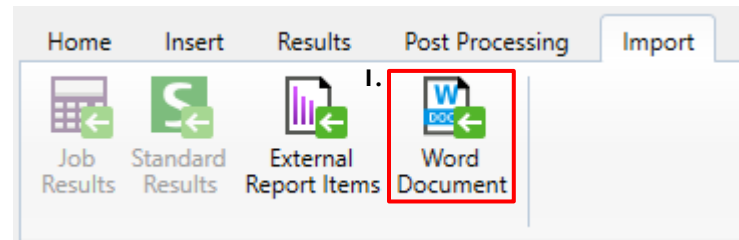
In display properties set the file path

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

