



Tutorial

Get started with SDC Verifier

Updated on: 30 May 2023

Tested with: SDC Verifier 2023 R1

Simcenter Femap with Nastran 2022.2 MP2

SDC Verifier is a powerful FEMAP add-on that helps verify structures according to standards and generates full calculations reports.

The goal of **SDC Verifier** is to automate all possible routine work and speed up a verification of the engineering projects significantly.

This step-by-step tutorial is designed to *get* you *started* with the main SDC Verifier features.

You will learn how to:

- Create new project;
- Create Individual Loads, Load Sets, and Load Groups;
- Analyze Job;
- Create Views;
- Create Model Setup Report (Wizard);
- Create Calculation Report (Wizard);
- Open as template feature;

Create New Project

1 Launch SDC Verifier for FEMAP 

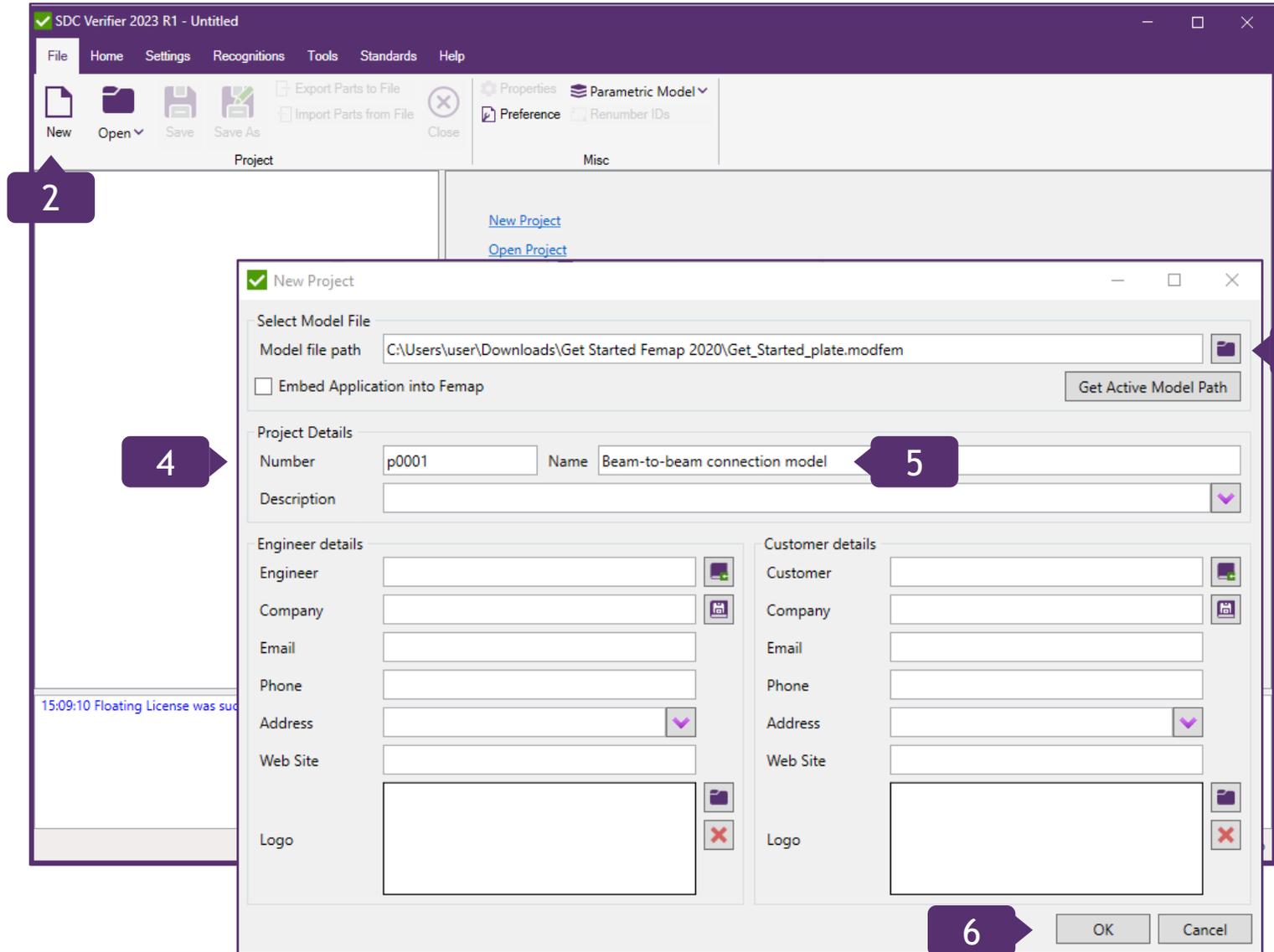
2 Execute *File - New*

3 Press  and select *Get_Started_plate* Femap model

4 Number: p0001

5 Name: Beam-to-beam connection model

6 Press *OK*



1 Select 1..Job 1

2 Title: Linear Static Analysis

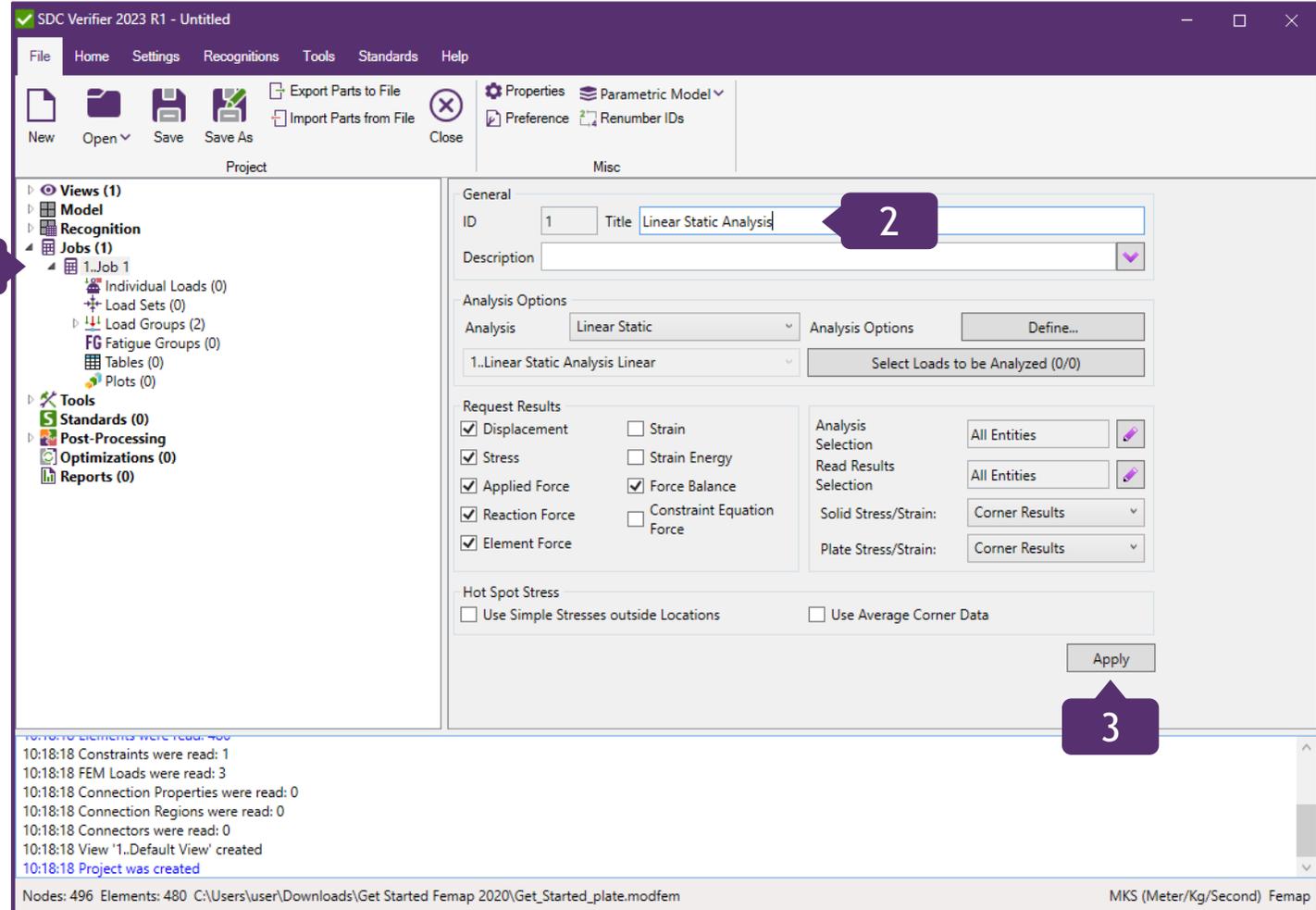
3 Press *Apply*

Job is a calculation set that contains analysis with options, boundary conditions, load combinations, envelopes and tables/plots.

Individual Loads = FEM load + Constraint.
Boundary condition + Output Set;

Load Sets - combination of Individual Loads with factors;

Load Groups (envelop - worst results among loads)



FEM Load



Constraint



Individual Load

Create Individual Loads

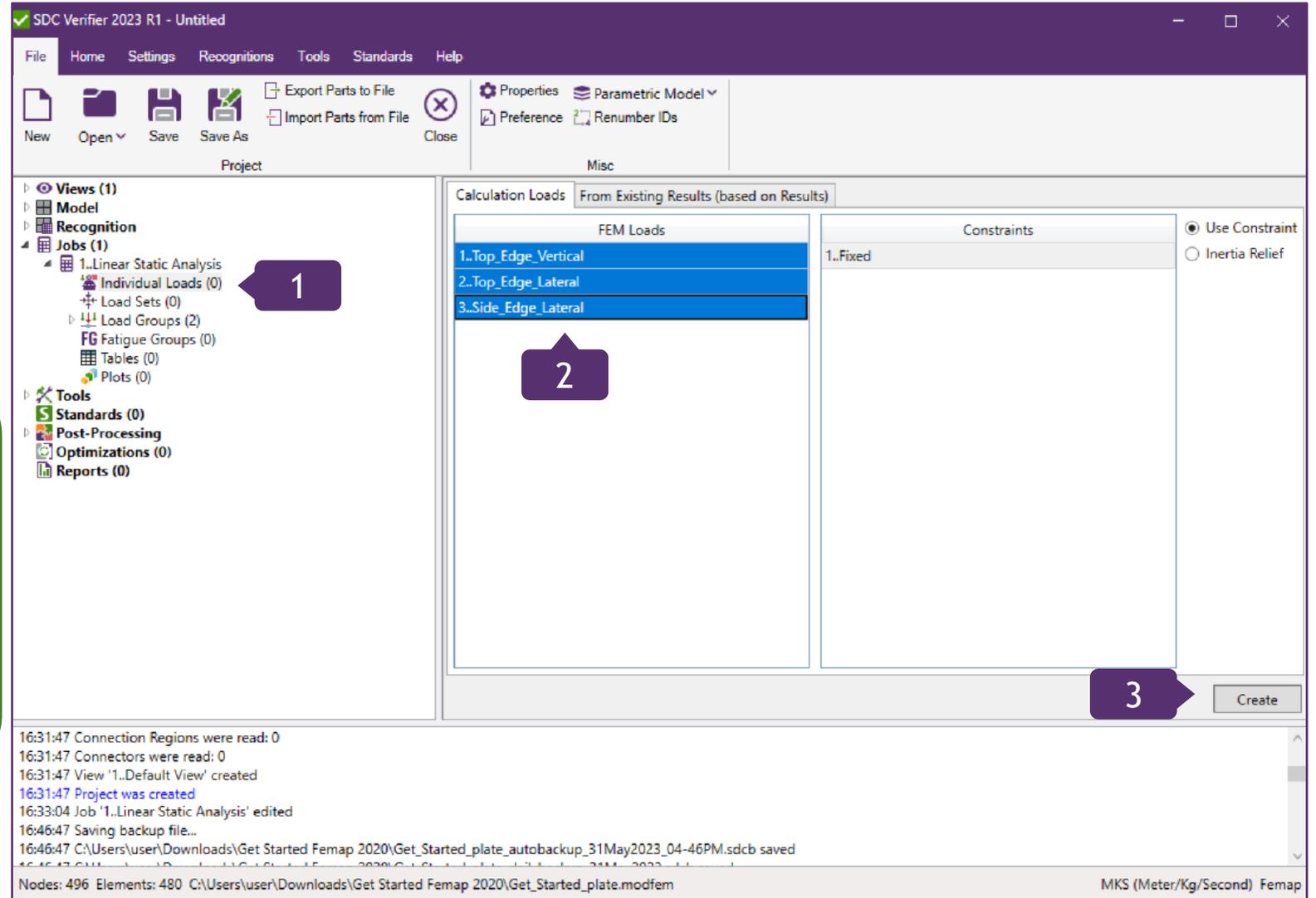
1 Select *Individual Loads* in the *Model Tree*

2 Select all *FEM Loads*

3 Press *Create*

Individual Loads will be created automatically from combinations of all selected FEM Loads and Constraints. In our case 3 Individual loads.

If the model already contains Output Sets it is possible to create Individual Loads based on results without boundary conditions (see next slide).



Note: Use option “Inertia Relief” to create Individual Loads, based on FEM Loads only (without constraint).

Create Individual Loads from Existing Results

This slide demonstrates alternative method of how to create Individual Loads based on Output Sets. Tutorial model does not contain any results yet; steps from this slide should be skipped.

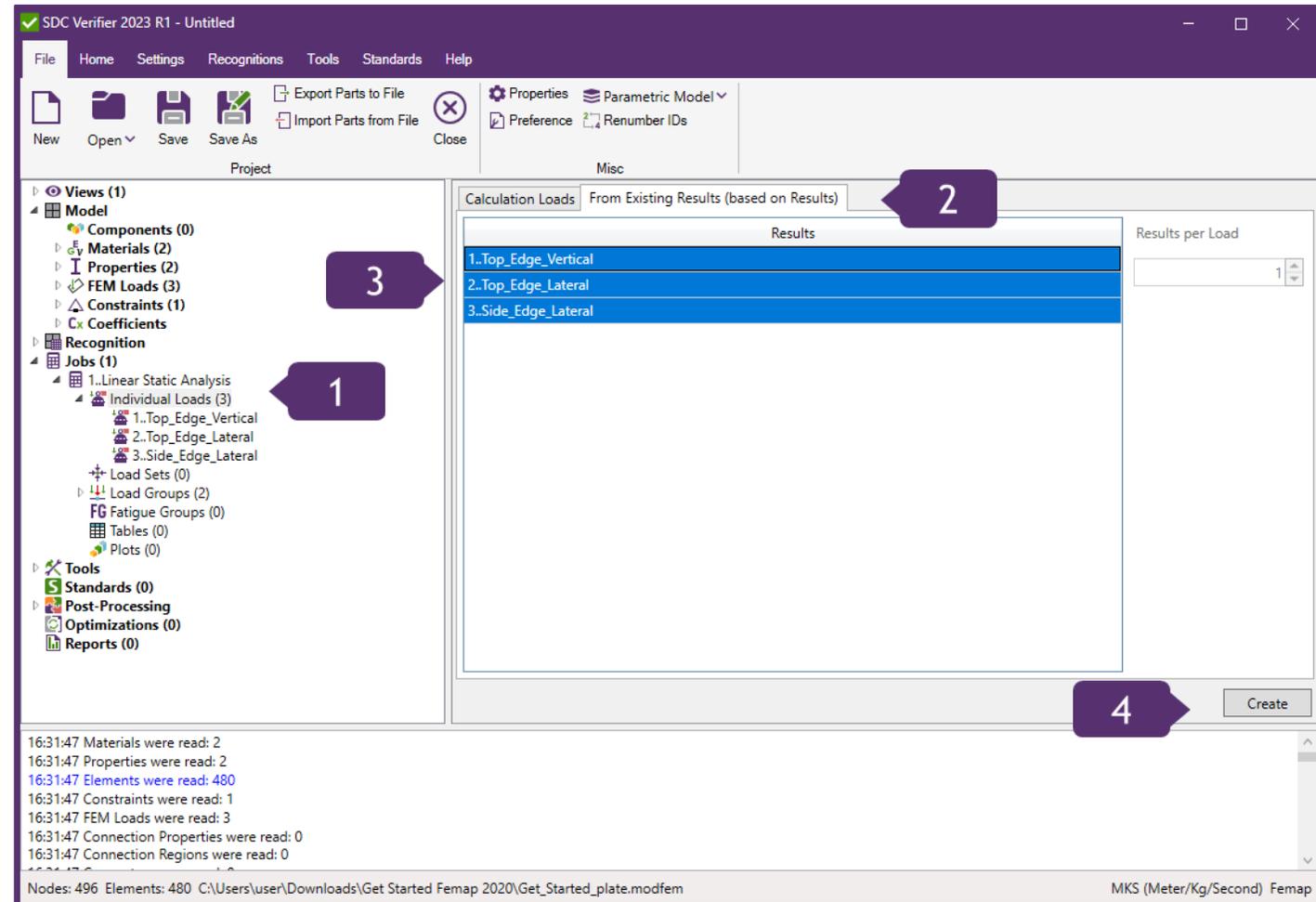
1 Activate *Individual Loads* in the *Model Tree*.

2 Select *From Existing results* option.

3 Select all Output Sets.

4 Press *Create*.

3 Individual Loads will be created based on 3 Output Sets

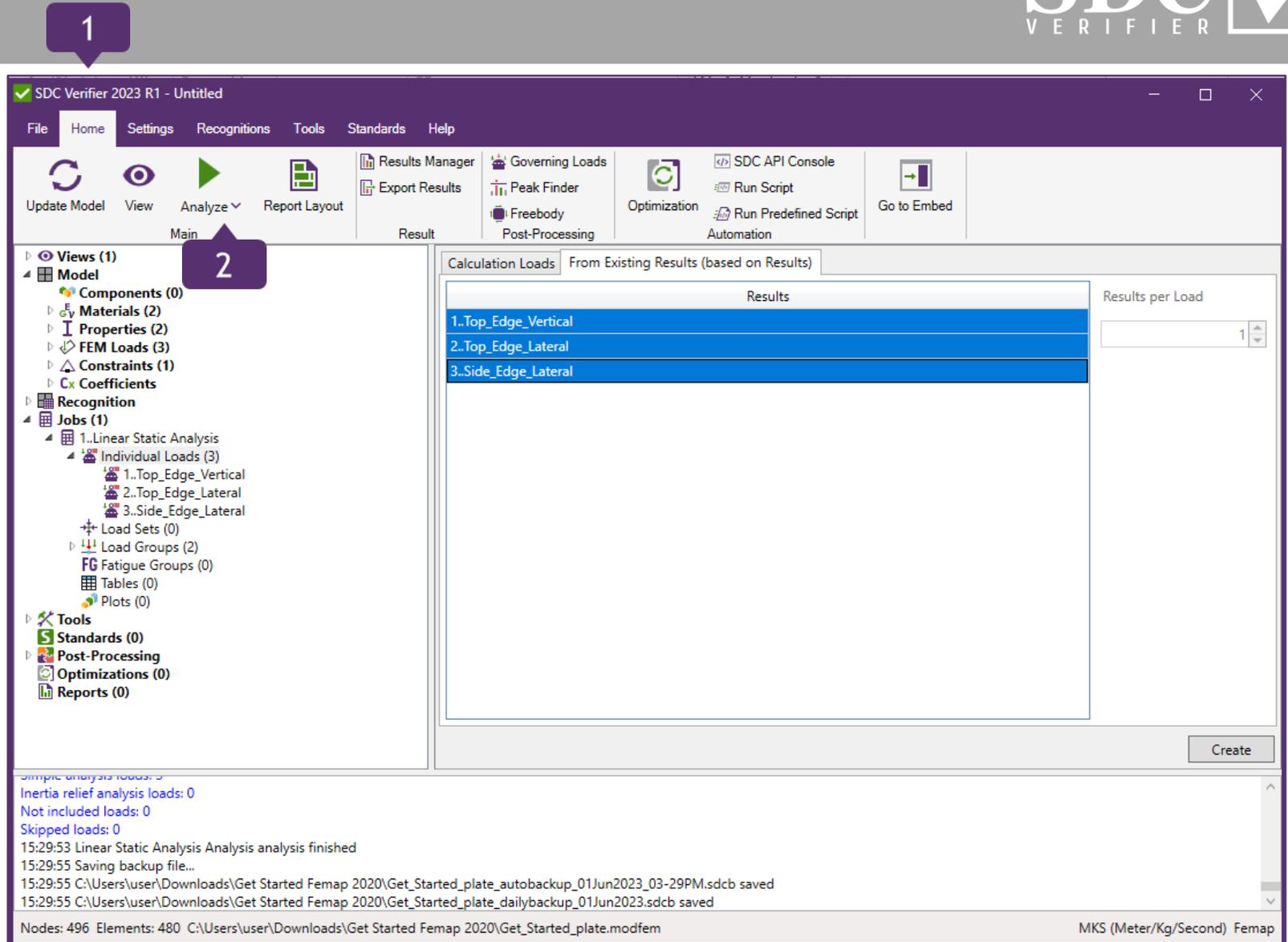


1 Go to *Home* section on the Ribbon

2 Press  on the toolbar to analyze job.

Analysis with 3 cases will be created and run. When finished, Output Sets will be linked automatically to correspondent Individual Loads.

- Analysis Set : 2..linear Static Analysis Linear
 - Solver : Simcenter Nastran
 - Type : Static
 - Integrated Solver : Simcenter Nastran
 - Options
 - Master Requests and Conditions
 - Case : 1..Top_Edge_Vertical
 - Case : 2..Top_Edge_Lateral
 - Case : 3..Side_Edge_Lateral



Note: If Individual Loads were created based on Output Sets running analysis is not required.

Create Load Combinations (Logic LS)

1 Activate *Load Sets* in the *Model* tree.

2 Title: All_combinations

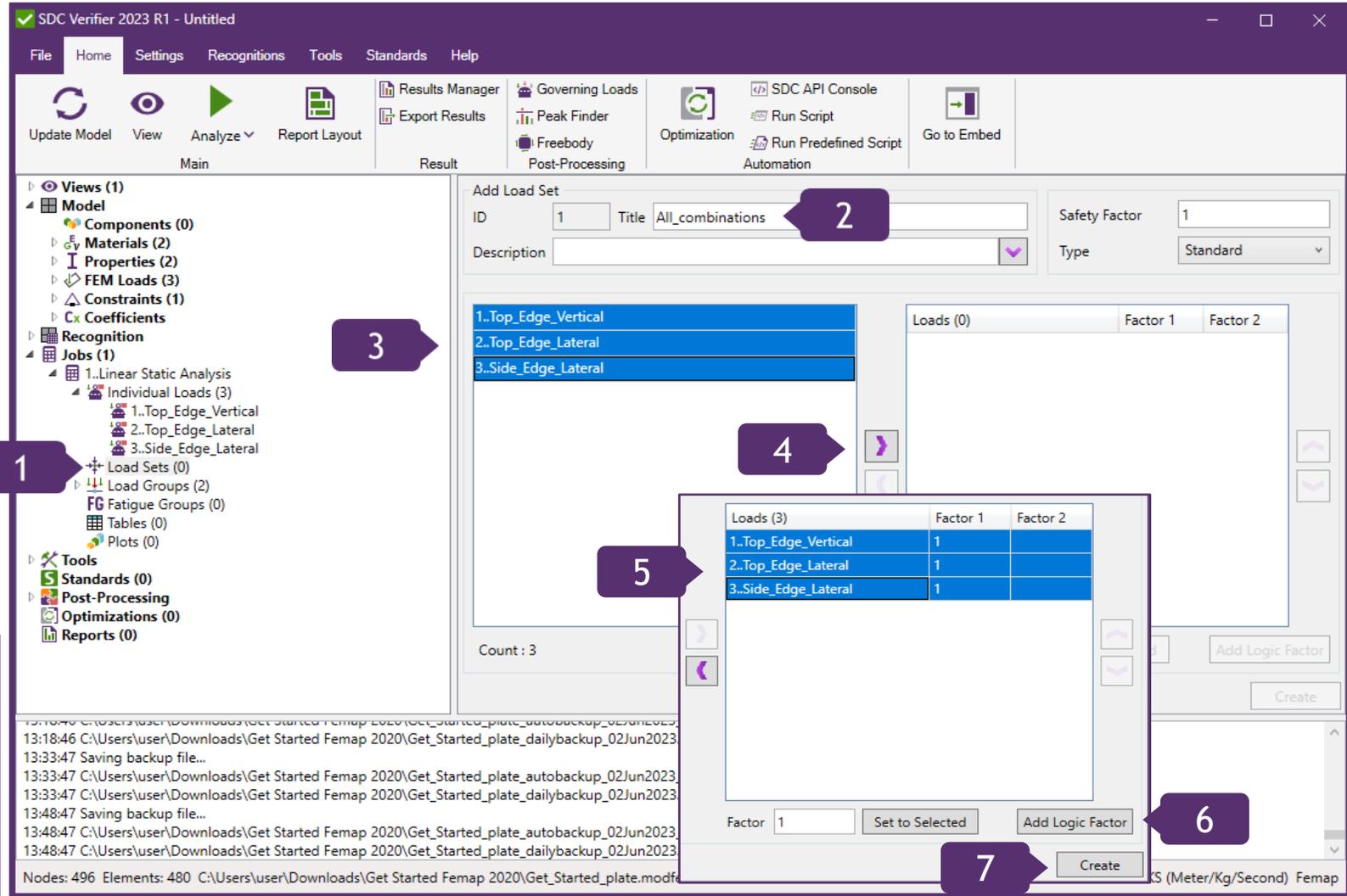
3 Select all *Individual Loads* from the list of loads.

4 Press to add items to load set.

5 Select all *Individual Loads* from the list of loads.

6 Press *Add Logic Factor* (each item will be added with positive and negative factor).

7 Press *Create*



	Safety Factor	IL1..Top_Edge_Vertical	IL2..Top_Edge_Lateral	IL3..Side_Edge_Lateral
All_combinations.1	1	1	1	1
All_combinations.2	1	-1	1	1
All_combinations.3	1	1	-1	1
All_combinations.4	1	-1	-1	1
All_combinations.5	1	1	1	-1
All_combinations.6	1	-1	1	-1
All_combinations.7	1	1	-1	-1
All_combinations.8	1	-1	-1	-1

A list of the created Load Sets

Edit Multiple Load Sets

1 Execute *Create/Edit multiple* in the *Load Sets* node in the *Model* tree.

2 Select cells for All_combinations with IDs 1-4 in column *IL3*

3 Factor: 1.1. Press *Set*

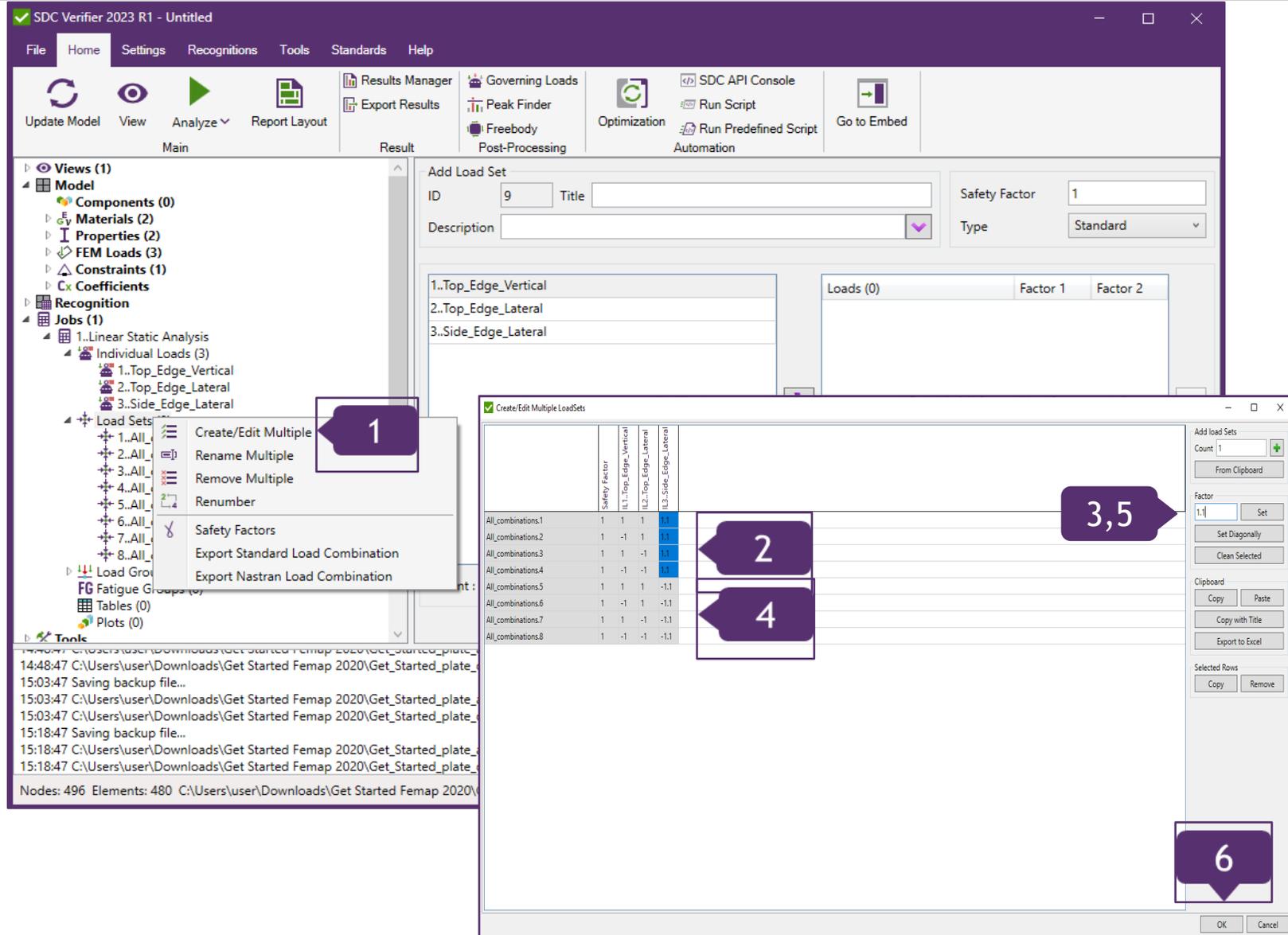
4 Select cells for All_combinations with IDs 5-8 in column *IL3*

5 Factor: -1.1. Press *Set*

6 Press *OK*.

Tip: It is possible to export/import table to excel using *Copy* and *Paste* buttons.

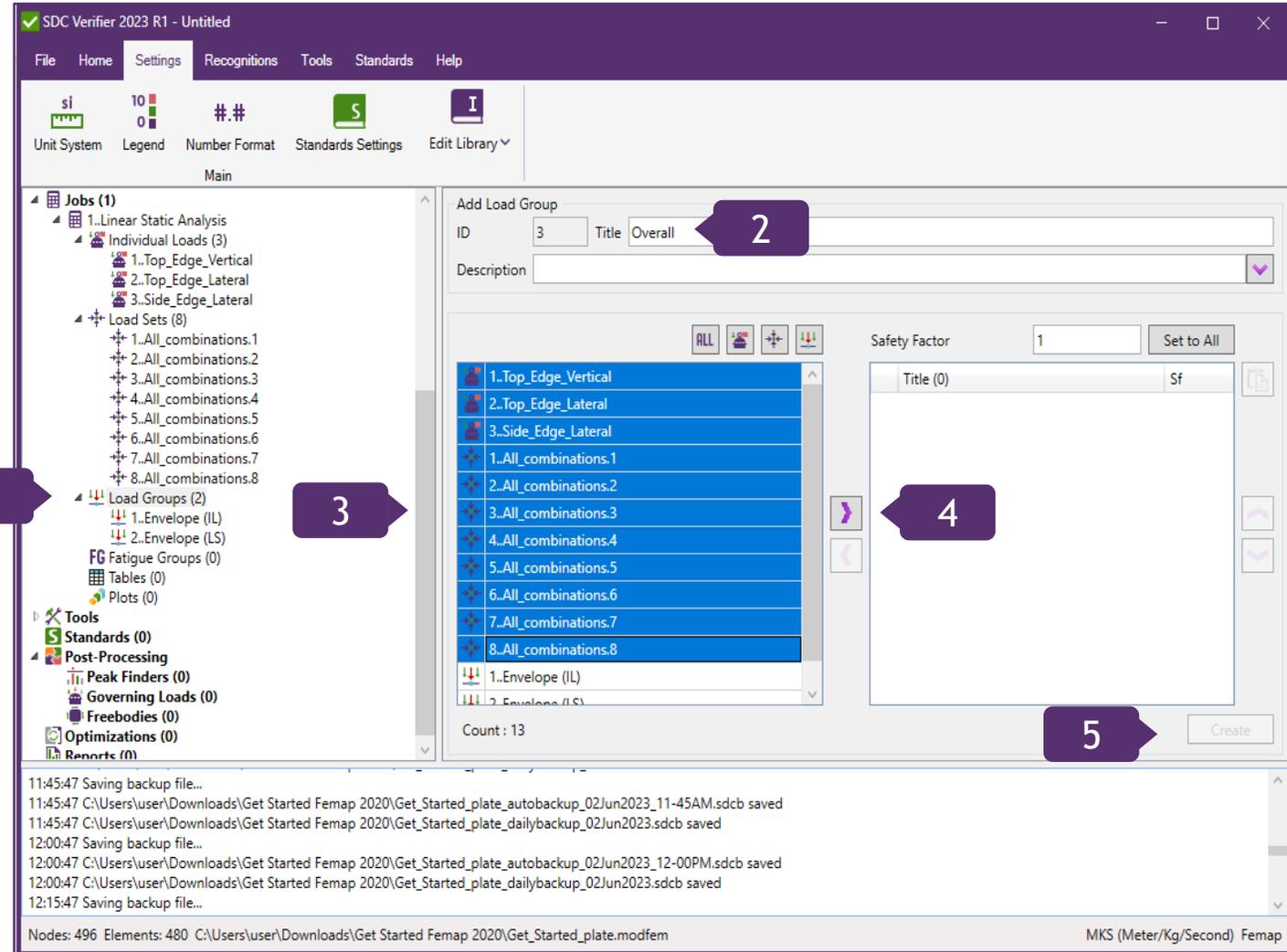
	Safety Factor	IL1..Top_Ed ge..Vertical	IL2..Top_Ed ge..Lateral	IL3..Side_Ec ge..Lateral
l_combinations.1	1	1	1	1,1
l_combinations.2	1	-1	1	1,1
l_combinations.3	1	1	-1	1,1
l_combinations.4	1	-1	-1	1,1
l_combinations.5	1	1	1	-1,1
l_combinations.6	1	-1	1	-1,1
l_combinations.7	1	1	-1	-1,1
l_combinations.8	1	-1	-1	-1,1



Create Load Group (Overall)

- 1 Activate *Load Groups* node in the *Model* tree.
- 2 Title: *Overall*
- 3 Select all Loads from the list of loads, except Envelope (IL) and Envelope (LS).
- 4 Press  to move selected Loads to Load Group.
- 5 Press *Create*.

Load Group is envelope for Individual Loads, Load Sets, and other Load Groups. It allows to determine minimum, maximum and absolute values of stresses, displacements, forces, etc.



1 Locate Model in Femap as shown on pic. Front View

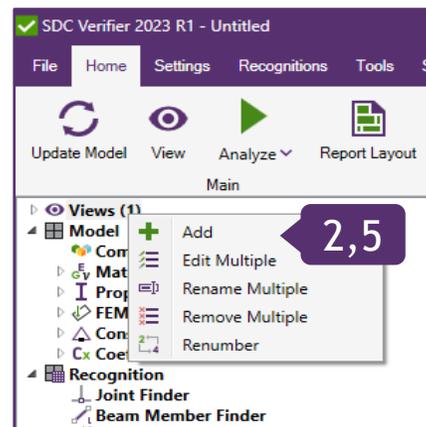
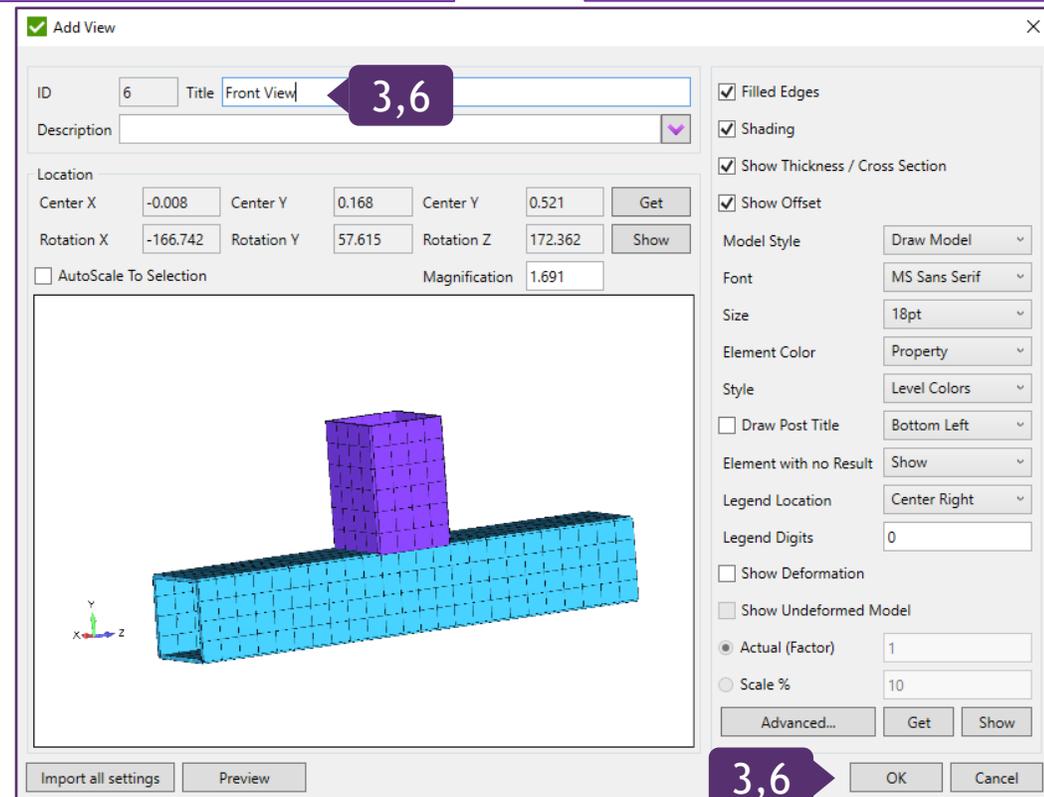
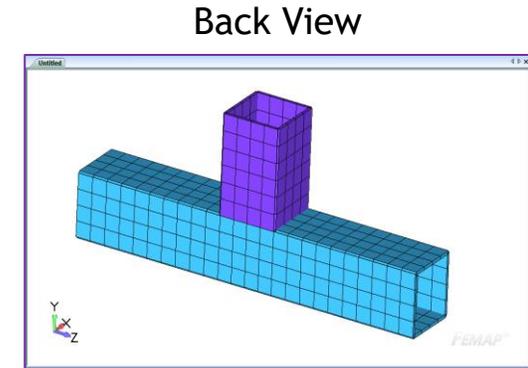
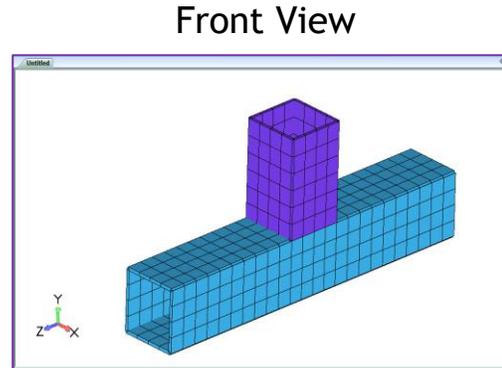
2 Execute *View - Add* from context menu

3 Title: Front View. Press *OK*

4 Locate the Model in Femap as shown on pic. Back View

5 Execute *View - Add* from context menu

6 Title: Back View. Press *OK*



1 Locate Model in Femap as shown on pic. Front Detail

2 Execute *View - Add* from context menu

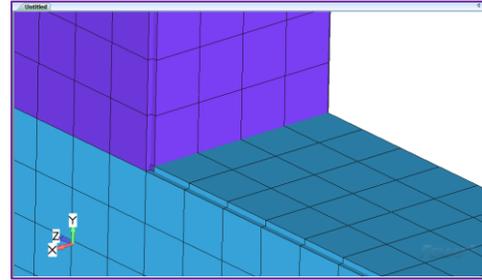
3 Title: Front Detail. Press *OK*.

4 Locate the Model in Femap as shown on pic. Back Detail.

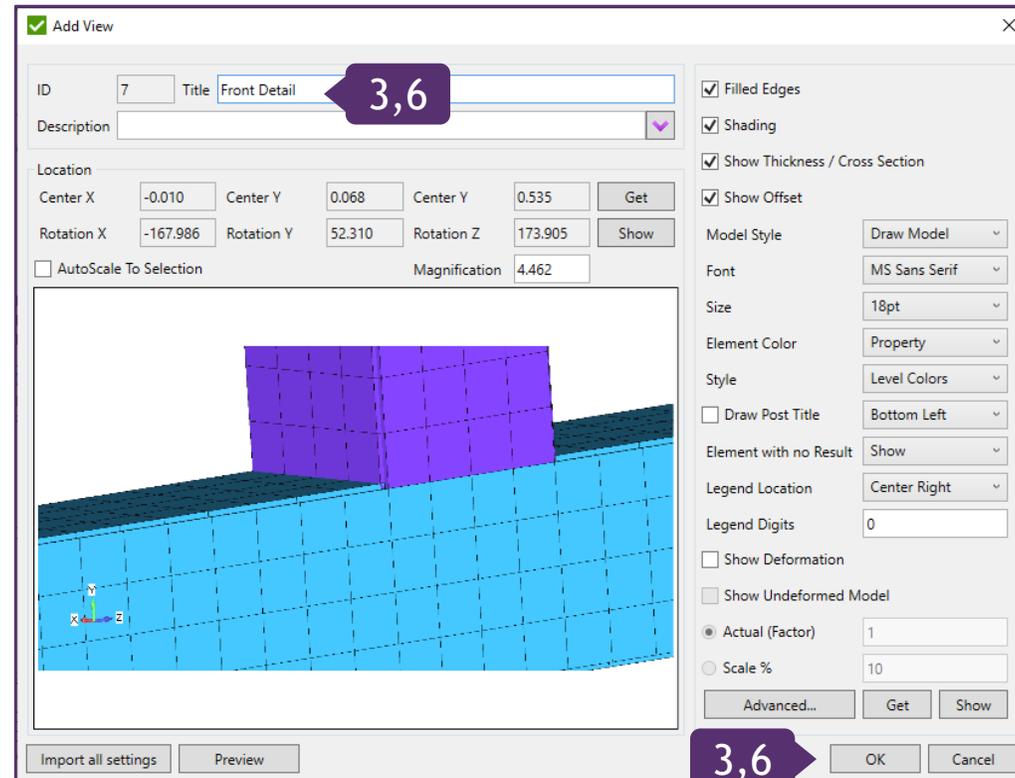
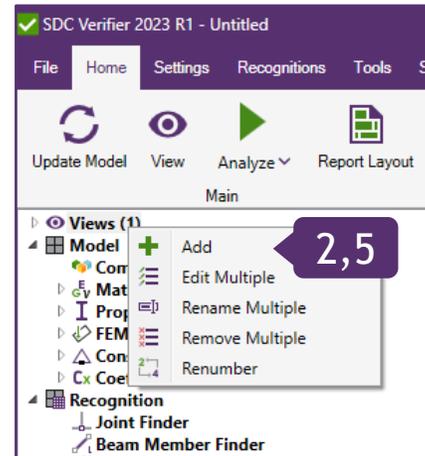
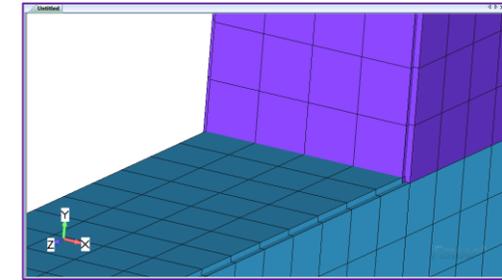
5 Execute *View - Add* from context menu

6 Title: Back Detail. Press *OK*.

Front Detail



Back Detail



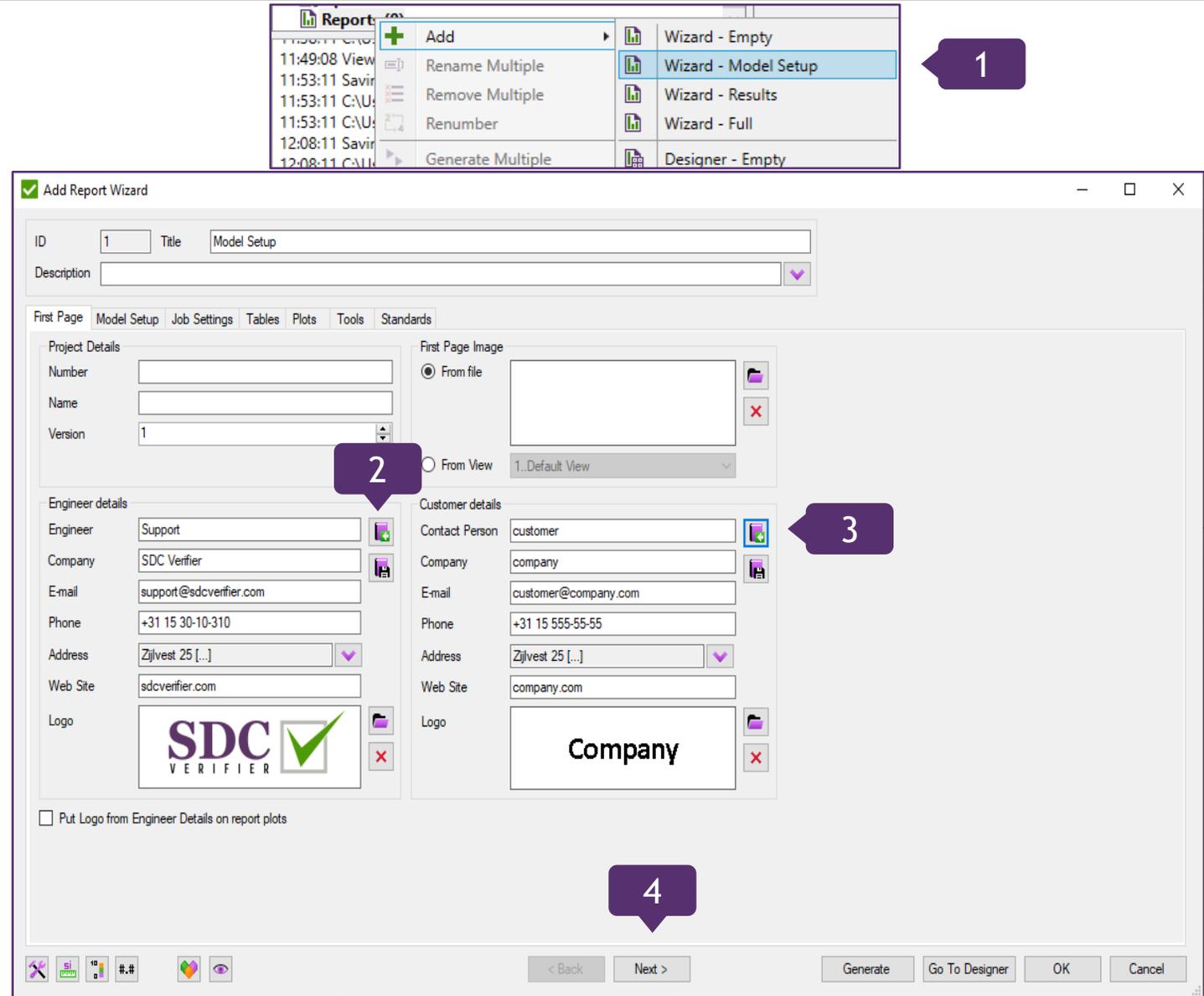
1 From *Reports* in the *Model* tree execute *Add* -> *Wizard - Model Setup*

2 Press and select *Support Engineer* from the library

3 Press and select *Customer* from the library

4 Press *Next*

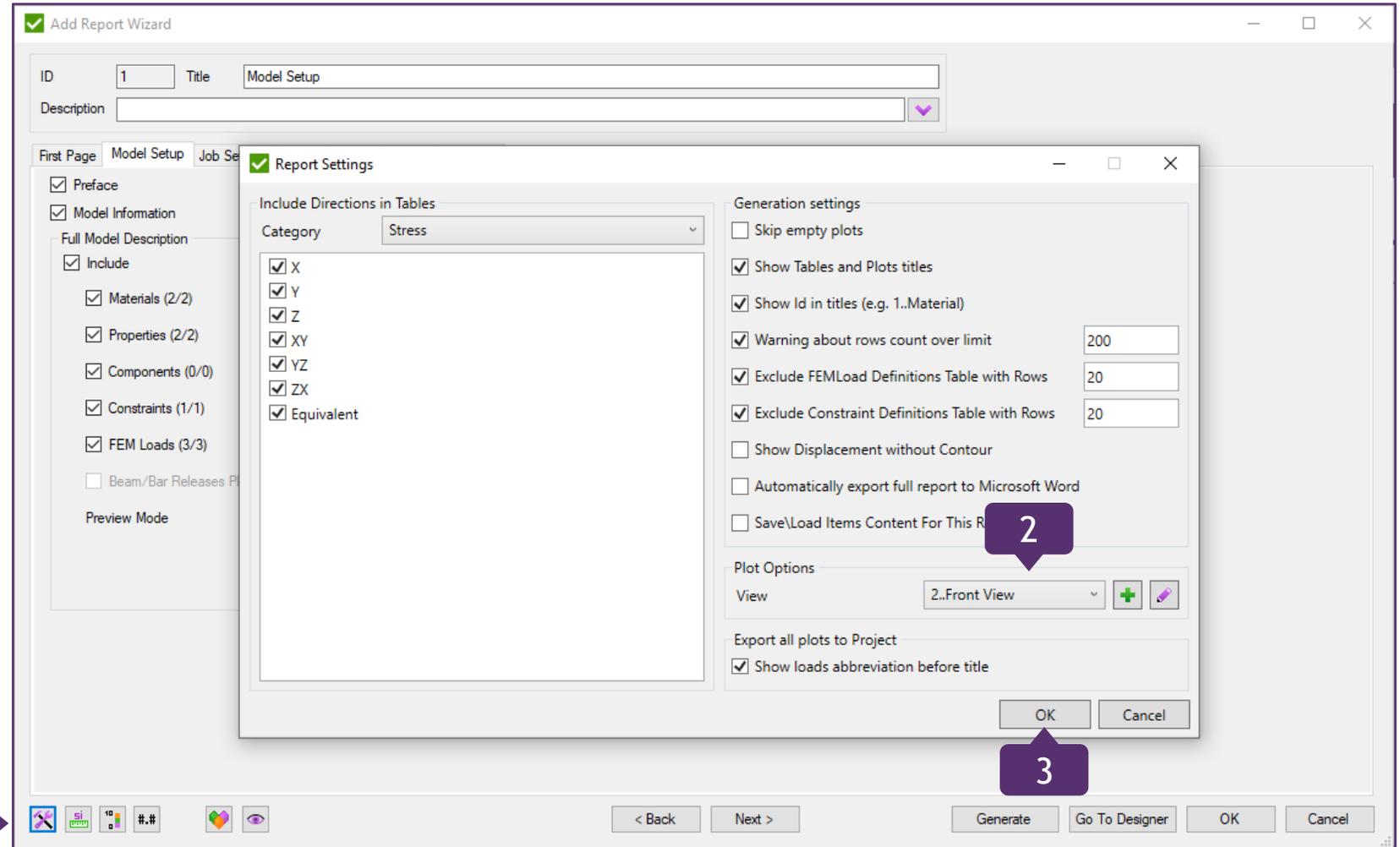
Note: Engineer and customer information is used on the report's first page and in footer (company name and logo).



1 Press Report Settings

2 In Plot Options, select 2..Front View

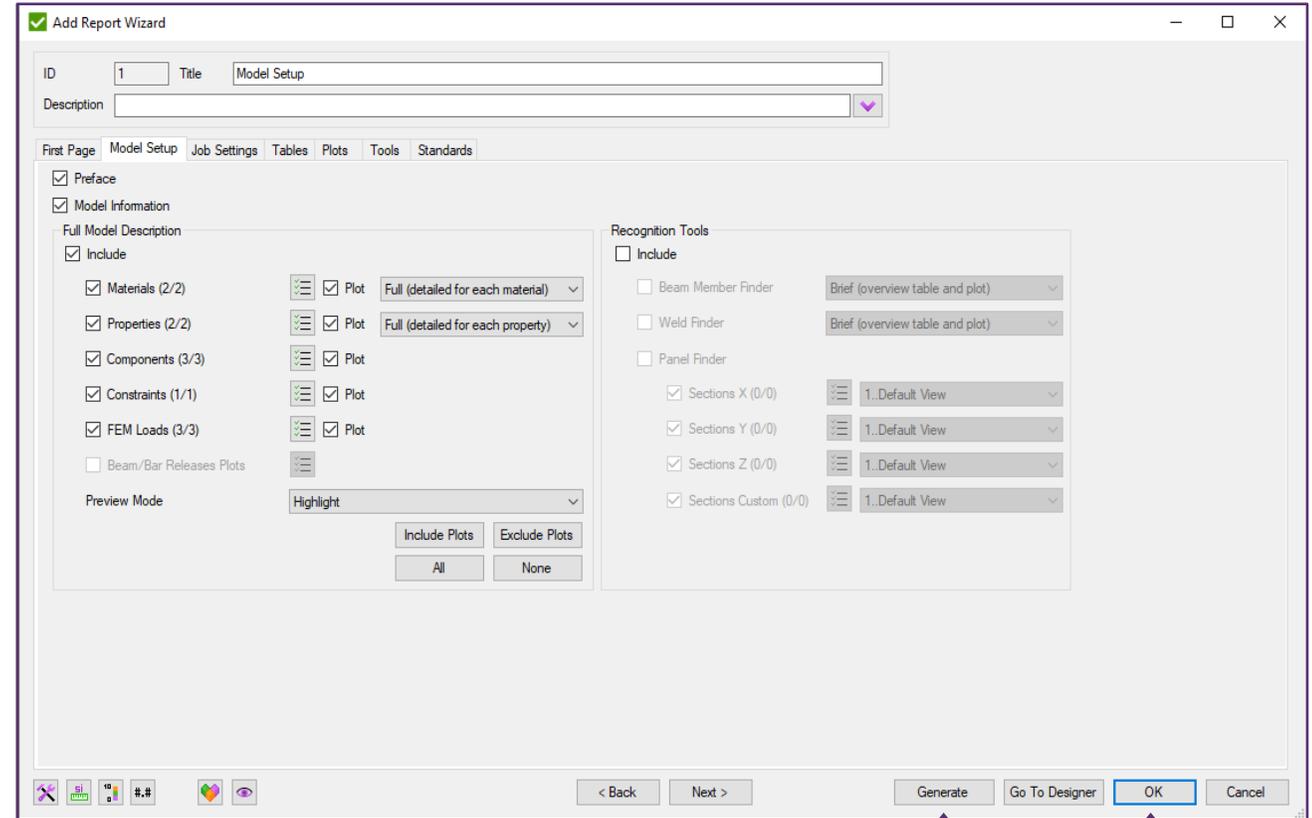
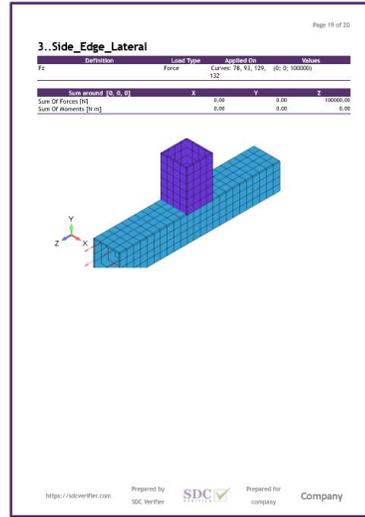
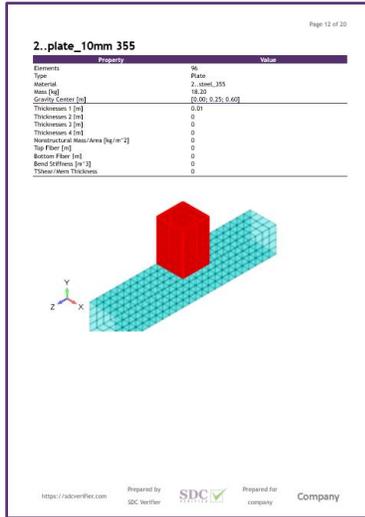
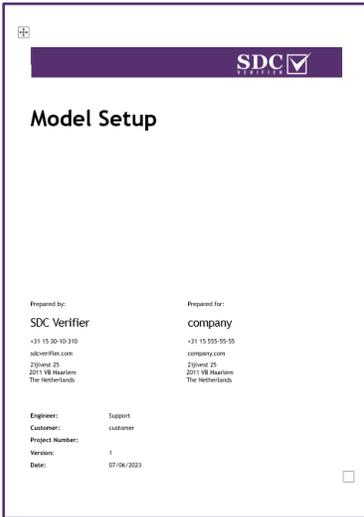
3 Press OK



Model Setup Report Generation

1 Press *Generate*

2 Press *OK*



Model setup report has been generated and opened in MS Word.
Using the *Report Designer* you can print out the report without any text editor installed.

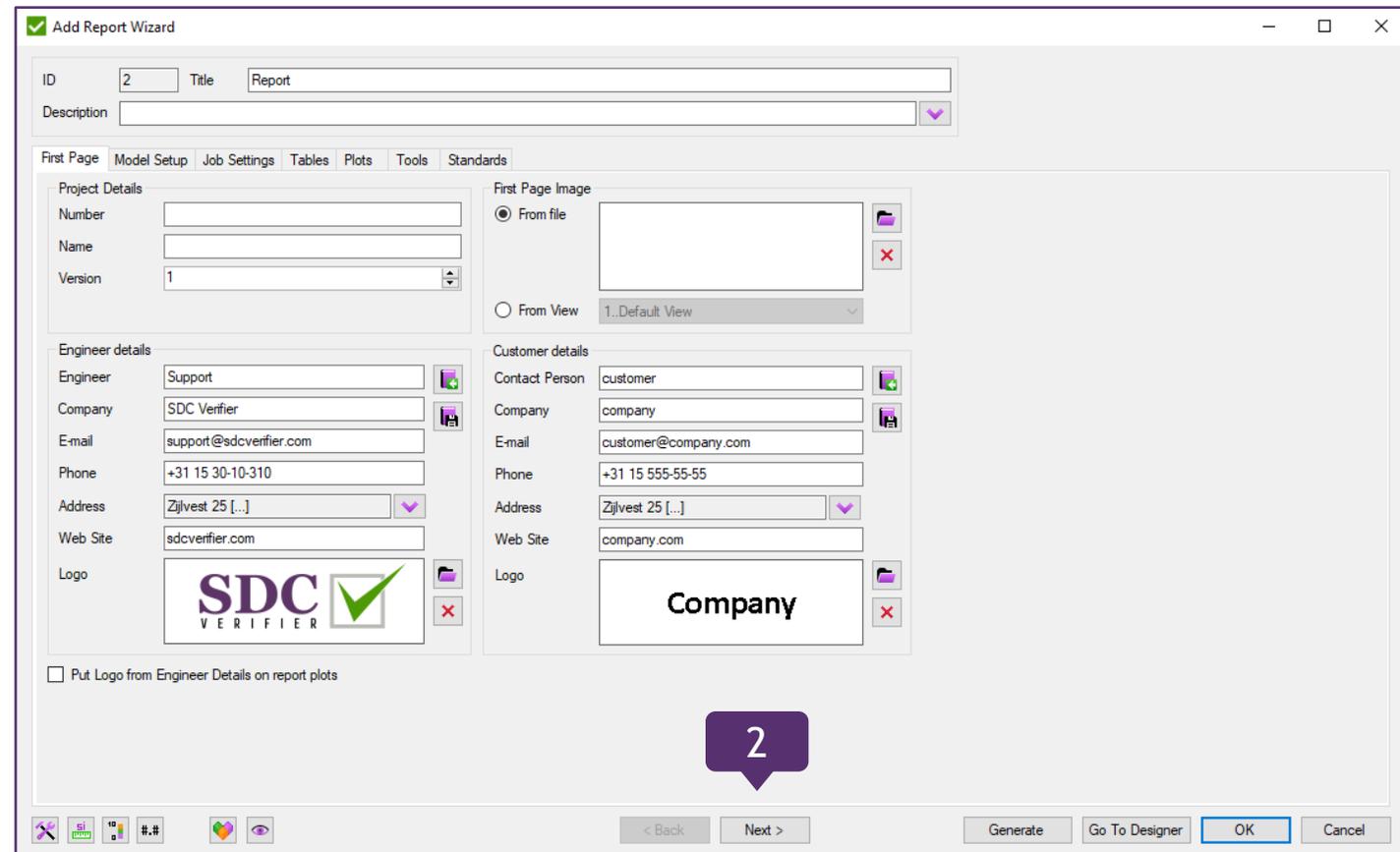
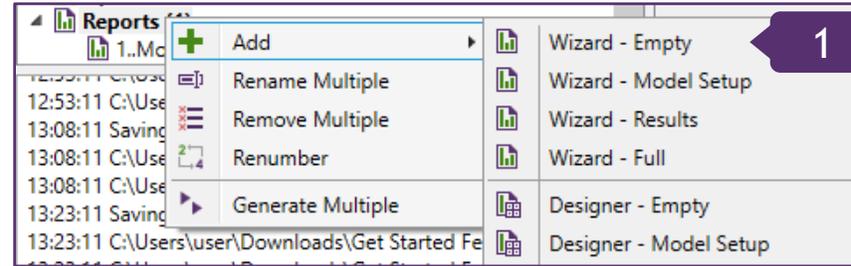
1

2

Create Calculation Report

1 From *Reports* in the *Model* tree, Execute *Add -> Wizard - Empty*

2 Press *Next* twice



1 Jobs: ON

2 Include Sum of Forces: ON

1

2

Add Report Wizard

ID: 2 Title: Report
Description: [Empty]

First Page | Model Setup | **Job Settings** | Tables | Plots | Tools | Standards

Include Jobs

- 1..Linear Static Analysis

Job Settings

- Job Description
- Plot Individual Load
- Modes Table for Individual Loads and Load Sets
- Include Contents
 - Individual Loads Content
 - Load Sets Content
 - Load Groups Content
 - Fatigue Groups Content

Automatically sort tables and plots by result category

Advanced Tables (Overall)

- Include Sum of Forces
 - Individual Loads Applied Forces
 - Individual Loads Reaction Forces
 - Load Sets Applied Forces
 - Load Sets Reaction Forces
- Absolute Maximum Displacement
 - Individual Load
 - Load Set
 - Load Group
- Absolute Maximum Stresses
 - Individual Load
 - Load Set
 - Load Group

Advanced Tables (For Each Load)

- Include Sum Of Forces
 - Individual Load Reaction Forces
 - Load Set Reaction Forces
- Stress Over All Properties
 - Individual Load
 - Load Set
 - Load Group
- Stress Over All Components
 - Individual Load
 - Load Set
 - Load Group

[All] [None]

< Back Next > Generate Go To Designer **OK** Cancel

Add Displacements Plots

1 Click on *Plots* in the Navigation list

2 Select All loads from the list

3 Press  to add contour plot

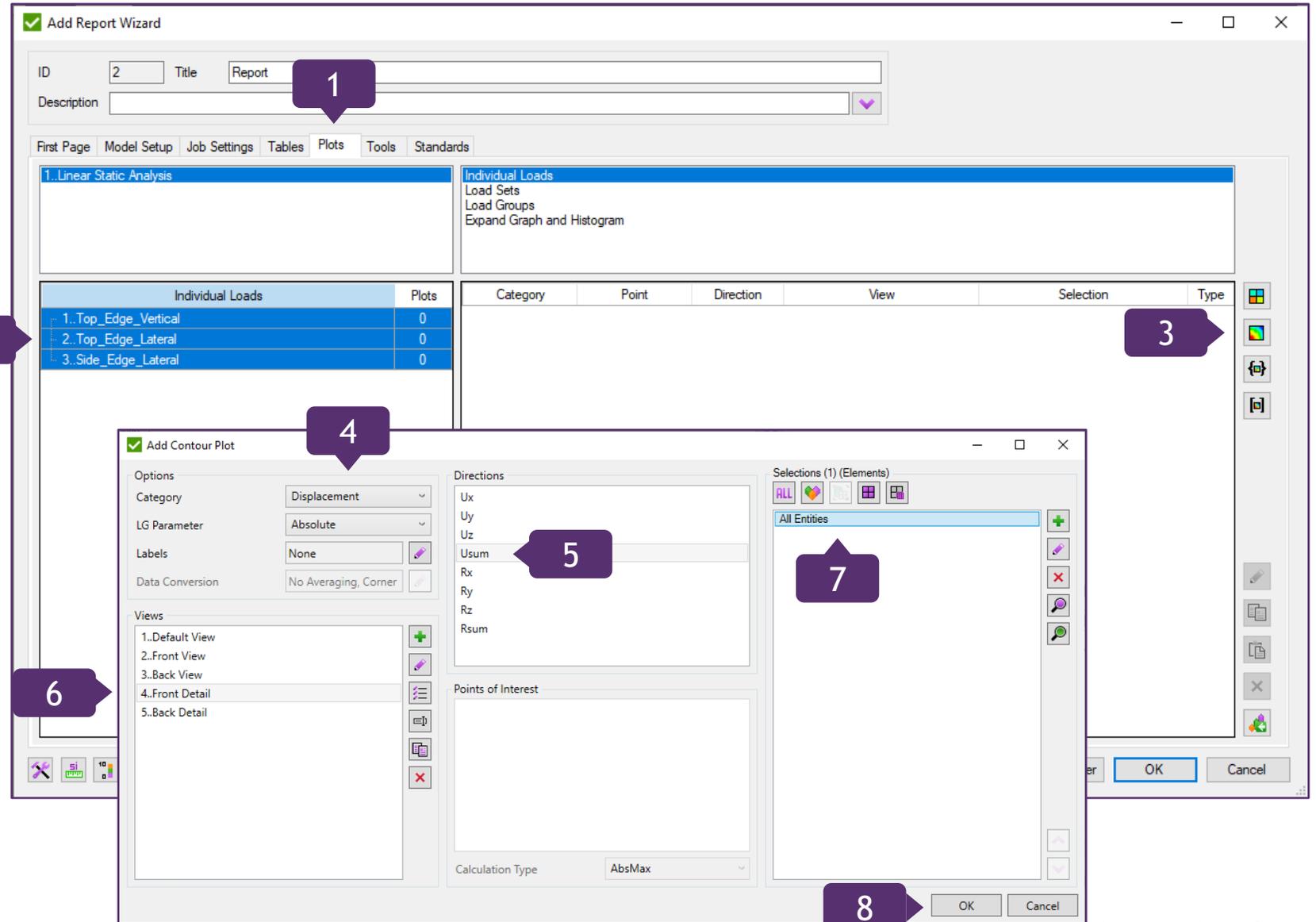
4 *Category*: Displacement

5 *Direction*: Usum

6 *Views*: Front Detail

7 *Selection*: All Entities

8 Press *OK*



Add Stress Plots

1 Press  to add contour plots

2 *Category: Stress*

3 *Direction: Equivalent*

4 Select 4 created Views

5 *Selection: All Entities*

6 Press *OK*

The screenshot shows the 'Add Report Wizard' window with the 'Plots' tab selected. The 'Individual Loads' table is expanded to show three views: '1..Top_Edge_Vertical', '2..Top_Edge_Lateral', and '3..Side_Edge_Lateral', each with a count of 1. The 'Category' dropdown is set to 'Displacement', 'Point' is 'Usum', and 'View' is '4..Front Detail'. The 'Selection' is 'All Entities'. A purple callout '1' points to the 'Add' icon in the top right corner of the wizard.

The 'Add Contour Plot' dialog is open in the foreground. The 'Options' section has 'Category' set to 'Stress', 'LG Parameter' to 'Absolute', 'Labels' to 'None', and 'Data Conversion' to 'No Averaging, Corner'. The 'Directions' list includes X, Y, Z, XY, YZ, ZX, and 'Equivalent', with 'Equivalent' selected and a purple callout '3' pointing to it. The 'Views' list includes '1..Default View', '2..Front View', '3..Back View', '4..Front Detail', and '5..Back Detail', with '2..Front View', '3..Back View', '4..Front Detail', and '5..Back Detail' selected. A purple callout '4' points to this list. The 'Selections (1) (Elements)' list contains 'All Entities', with a purple callout '5' pointing to it. The 'Calculation Type' is 'AbsMax'. A purple callout '2' points to the 'Add Contour Plot' title bar. A purple callout '6' points to the 'OK' button at the bottom right.

1 Press  to add output criteria plots

Add Report Wizard

ID: 2 Title: Report

Description:

First Page | Model Setup | Job Settings | Tables | **Plots** | Tools | Standards

1..Linear Static Analysis

Individual Loads

Individual Loads	Plots
1..Top_Edge_Vertical	5
2..Top_Edge_Lateral	5
3..Side_Edge_Lateral	5

Individual Loads

- Load Sets
- Load Groups
- Expand Graph and Histogram

Category	Point	Direction	View	Selection	Type
Displacement		Usun	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	2..Front View	All Entities	
Stress	No Averaging, C...	Equivalent	3..Back View	All Entities	
Stress	No Averaging, C...	Equivalent	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	5..Back Detail	All Entities	

1

< Back Next > Generate Go To Designer **OK** Cancel

Add Output Vector Plots (Continuation)

2 Press *Select* button

3 Select option: *By Category*

4 Select: *Element Stress*

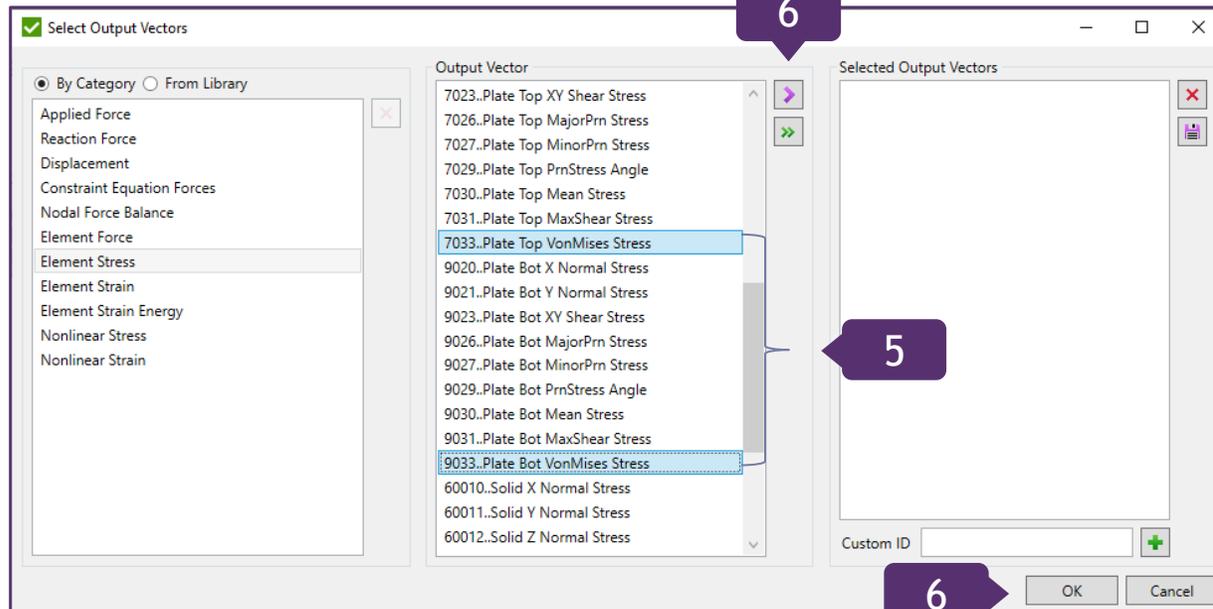
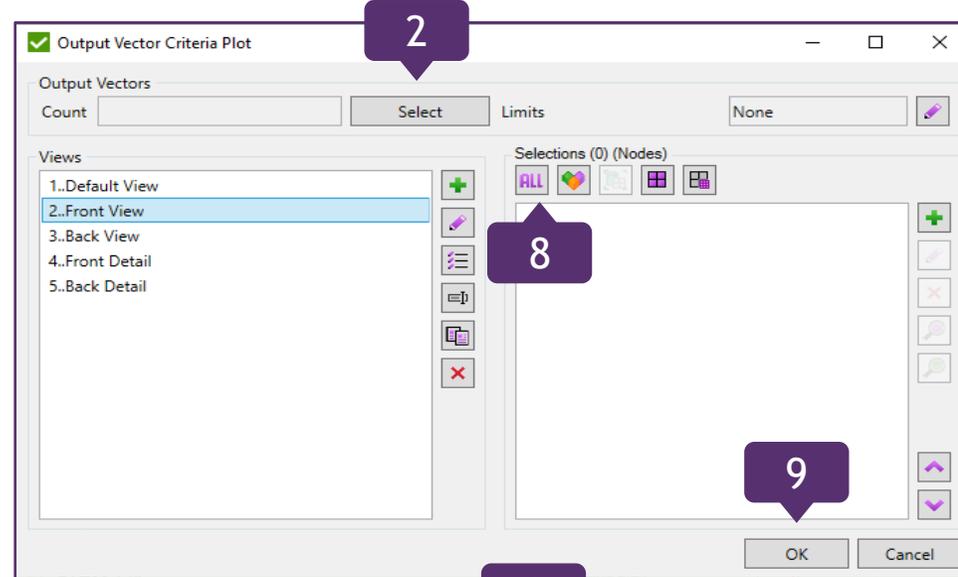
5 Select Vectors: *7033 and 9033*

6 Press  and *OK*

7 Views: *2..Front View*

8 Add  Entities

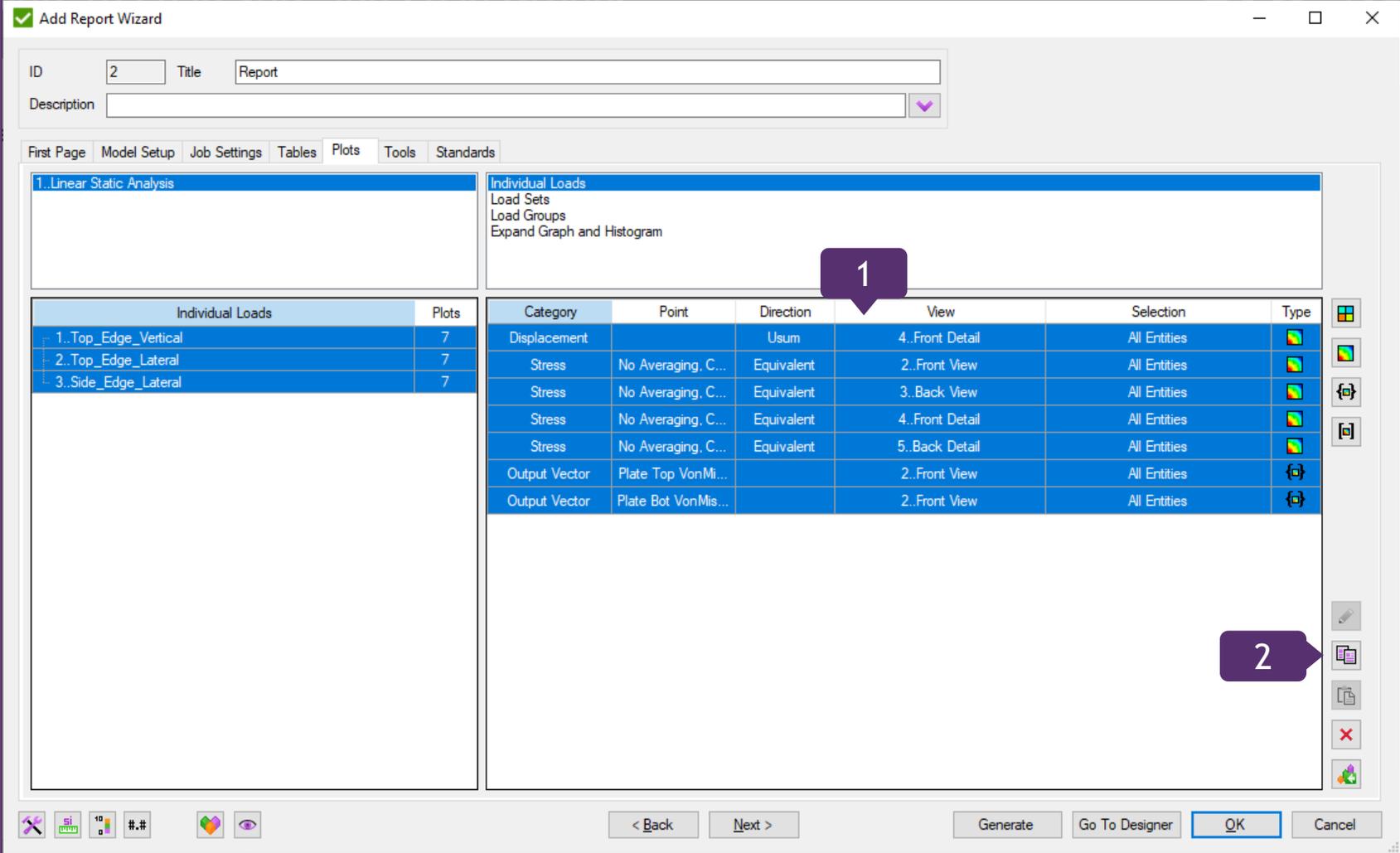
9 Press *OK*



Copy Plots to Load Sets and Load Groups

1 Select all plots from the list

2 Press Copy 



Add Report Wizard

ID: 2 Title: Report Description:

First Page Model Setup Job Settings Tables **Plots** Tools Standards

1..Linear Static Analysis

Individual Loads
Load Sets
Load Groups
Expand Graph and Histogram

Category	Point	Direction	View	Selection	Type
Displacement		Usun	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	2..Front View	All Entities	
Stress	No Averaging, C...	Equivalent	3..Back View	All Entities	
Stress	No Averaging, C...	Equivalent	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	5..Back Detail	All Entities	
Output Vector	Plate Top VonMi...		2..Front View	All Entities	
Output Vector	Plate Bot VonMis...		2..Front View	All Entities	

Individual Loads Plots

Individual Loads	Plots
1..Top_Edge_Vertical	7
2..Top_Edge_Lateral	7
3..Side_Edge_Lateral	7

1

2

< Back Next > Generate Go To Designer **OK** Cancel

Copy Plots to Load Sets and Load Groups (Continuation)

3 Plot Type: *Load Sets*

4 Select all Load Sets

5 Press  to paste

1..Linear Static Analysis

Load Sets	Plots
1..All_combinations.1	5
2..All_combinations.2	5
3..All_combinations.3	5
4..All_combinations.4	5
5..All_combinations.5	5
6..All_combinations.6	5
7..All_combinations.7	5
8..All_combinations.8	5

Individual Loads
Load Sets
Load Groups
Expand Graph and Histogram

Category	Point	Direction	View	Selection	Type
Displacement		Usum	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	2..Front View	All Entities	
Stress	No Averaging, C...	Equivalent	3..Back View	All Entities	
Stress	No Averaging, C...	Equivalent	4..Front Detail	All Entities	
Stress	No Averaging, C...	Equivalent	5..Back Detail	All Entities	

< Back Next > Generate Go To Designer **OK** Cancel

Copy Plots to Load Sets and Load Groups (Continuation)

6 Plot Type: *Load Groups*

7 Select 3..Overall Load Group

8 Press  to paste

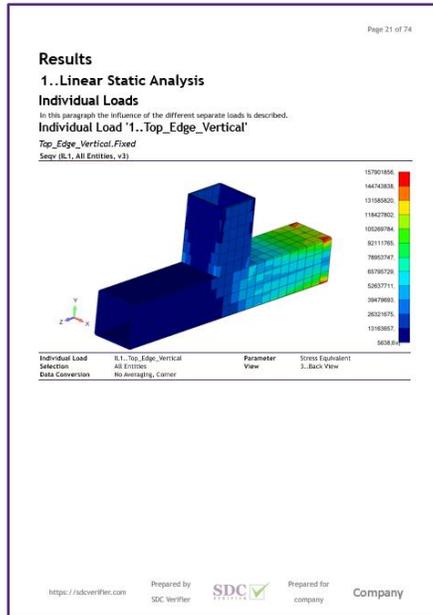
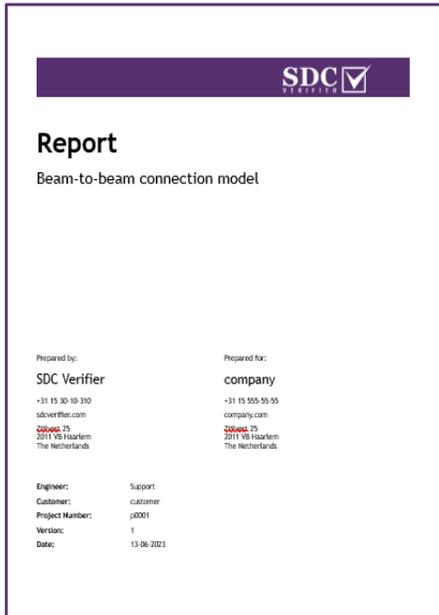
1..Linear Static Analysis

Load Groups	Plots
1..Envelope (IL)	0
2..Envelope (LS)	0
3..Overall	5

Category	Point	Parameter	Direction	View	Selection	Type
Displacement		Absolute	Usum	4..Front Detail	All Entities	
Stress	No Averaging, C...	Absolute	Equivalent	2..Front View	All Entities	
Stress	No Averaging, C...	Absolute	Equivalent	3..Back View	All Entities	
Stress	No Averaging, C...	Absolute	Equivalent	4..Front Detail	All Entities	
Stress	No Averaging, C...	Absolute	Equivalent	5..Back Detail	All Entities	

1 Press *Generate*

2 Press *OK*



✓ Add Report Wizard

ID: 2 Title: Report
Description: []

First Page | Model Setup | Job Settings | Tables | Plots | Tools | Standards

1..Linear Static Analysis

Individual Loads
Load Sets
Load Groups
Expand Graph and Histogram

Load Groups	Plots	Category	Point	Parameter	Direction	View	Selection	Type
1..Envelope (IL)	0	Displacement		Absolute	Usun	4..Front Detail	All Entities	[]
2..Envelope (LS)	0	Stress	No Averaging, C...	Absolute	Equivalent	2..Front View	All Entities	[]
3..Overall	5	Stress	No Averaging, C...	Absolute	Equivalent	3..Back View	All Entities	[]
		Stress	No Averaging, C...	Absolute	Equivalent	4..Front Detail	All Entities	[]
		Stress	No Averaging, C...	Absolute	Equivalent	5..Back Detail	All Entities	[]

< Back Next > Generate Go To Designer OK Cancel

1

2

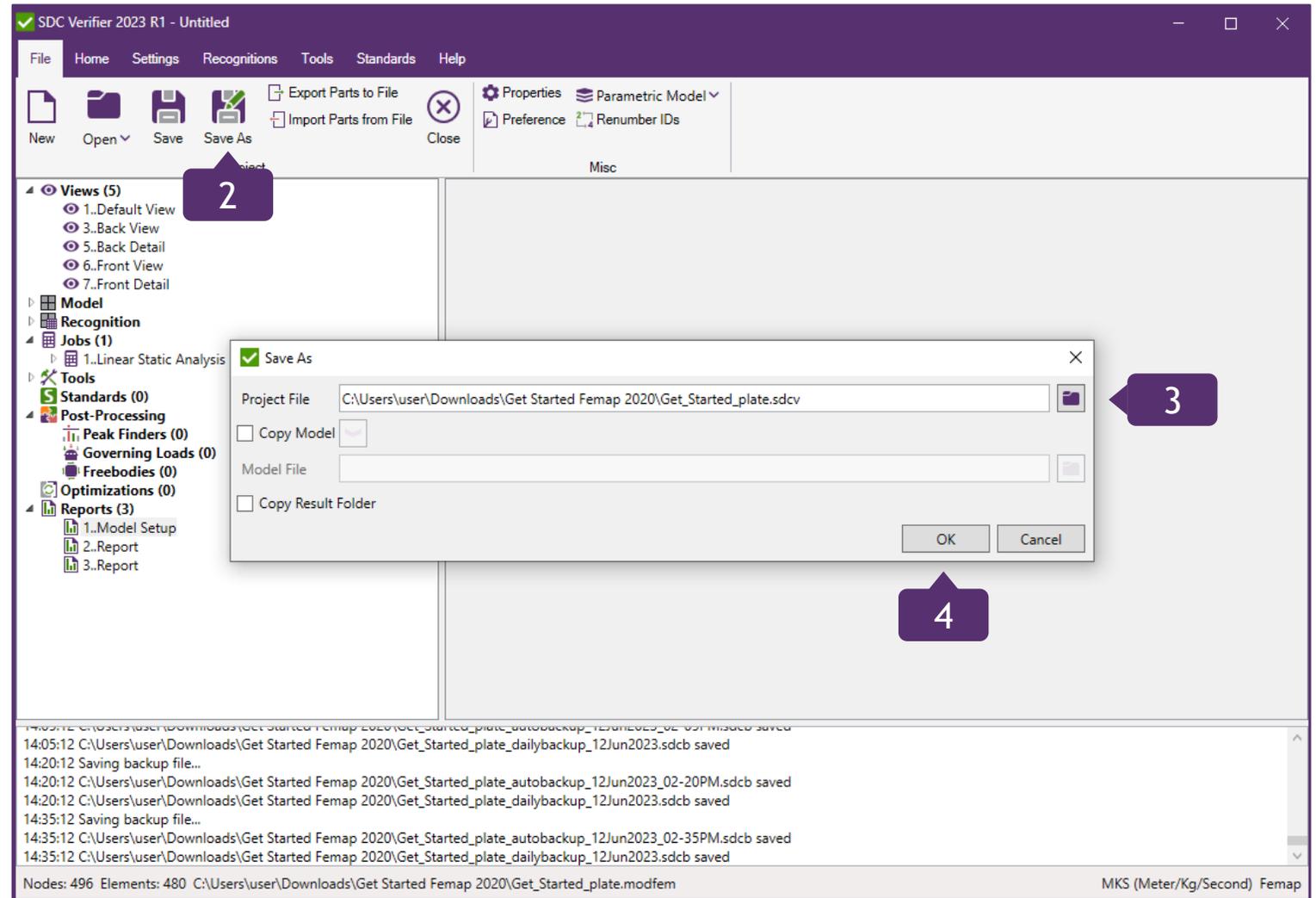
Save SDC Verifier Project

1 Go to *File* section

2 Press *Save As*

3 Press to browse location and define the filename

4 Press *OK*



Open As Template feature allows to reuse project for similar Femap model. In our case, a solid model with same boundary conditions

1 Go to *File* section

2 In *Open* folder execute *Open As Template*

3 *Template Project: Get_Started plate.sdcv*

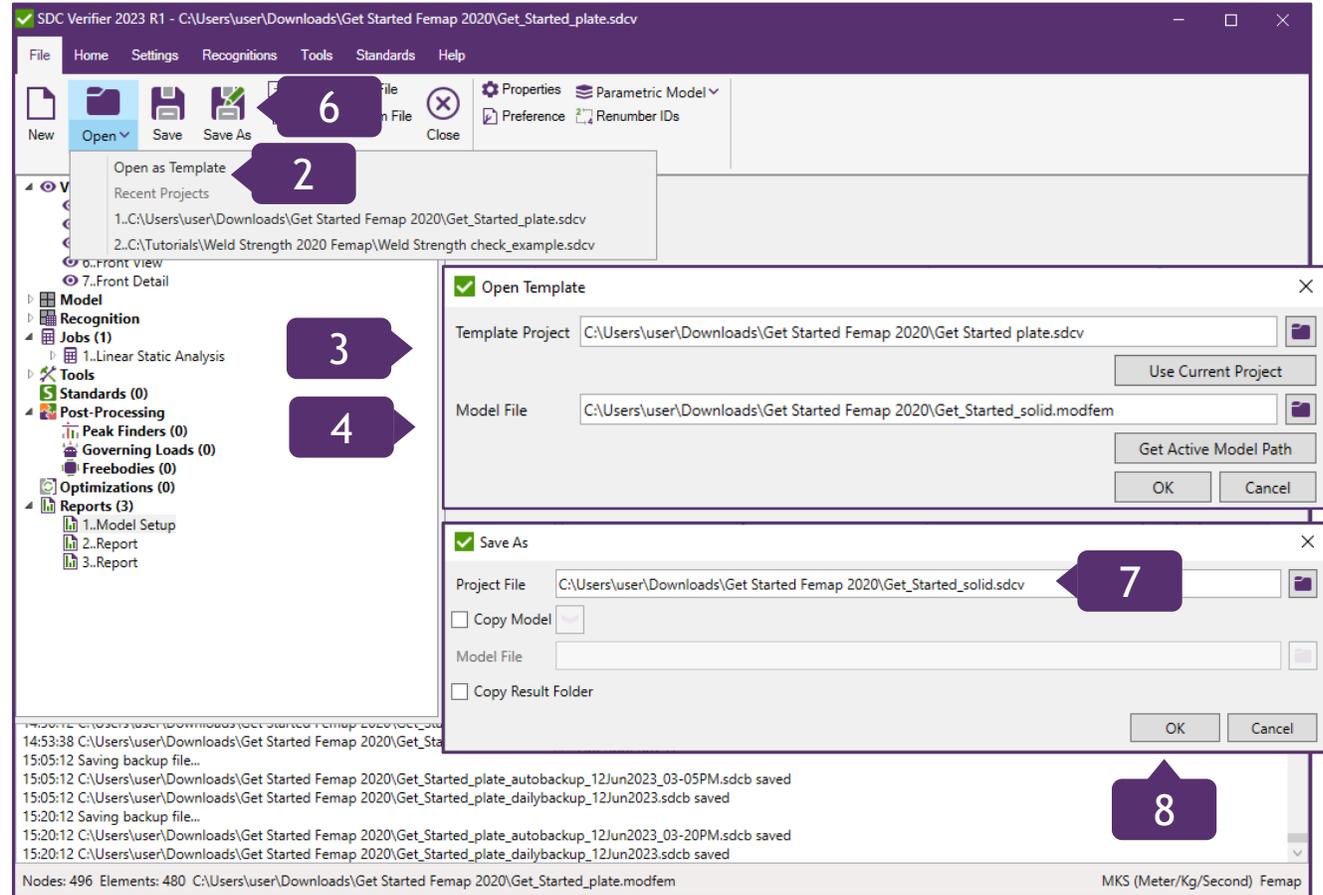
4 *Model File: Get_Started_solid.modfem*

5 Press *OK*

6 In *File* section, execute *Save As*

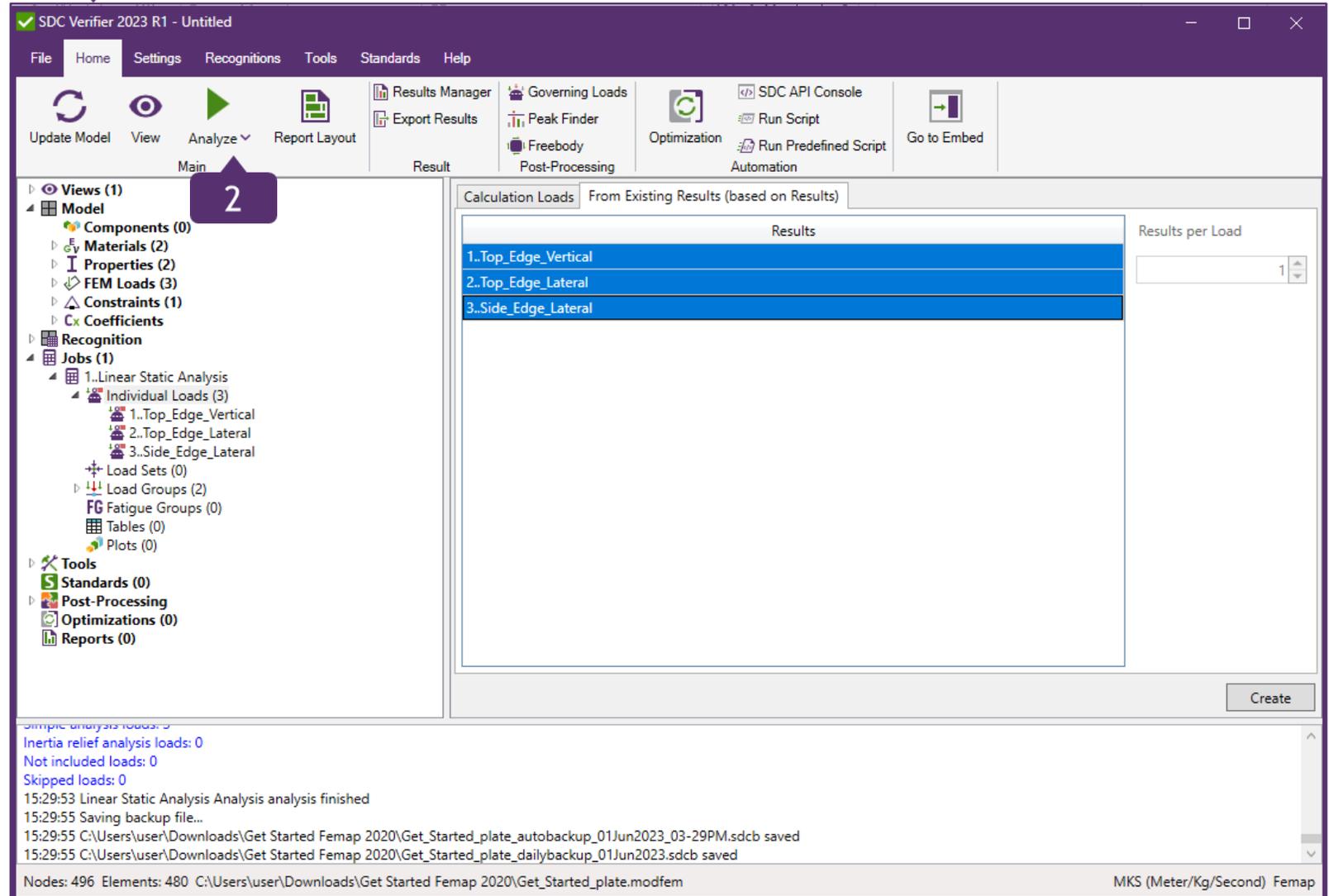
7 *Project File: Get_Started_solid.sdcv*

8 Press *OK*



1 Go to *Home* section on the Ribbon

2 Press  on the toolbar to analyze job.



Generate a Report for a Solid Model

1 Select 2..Report in the Model tree

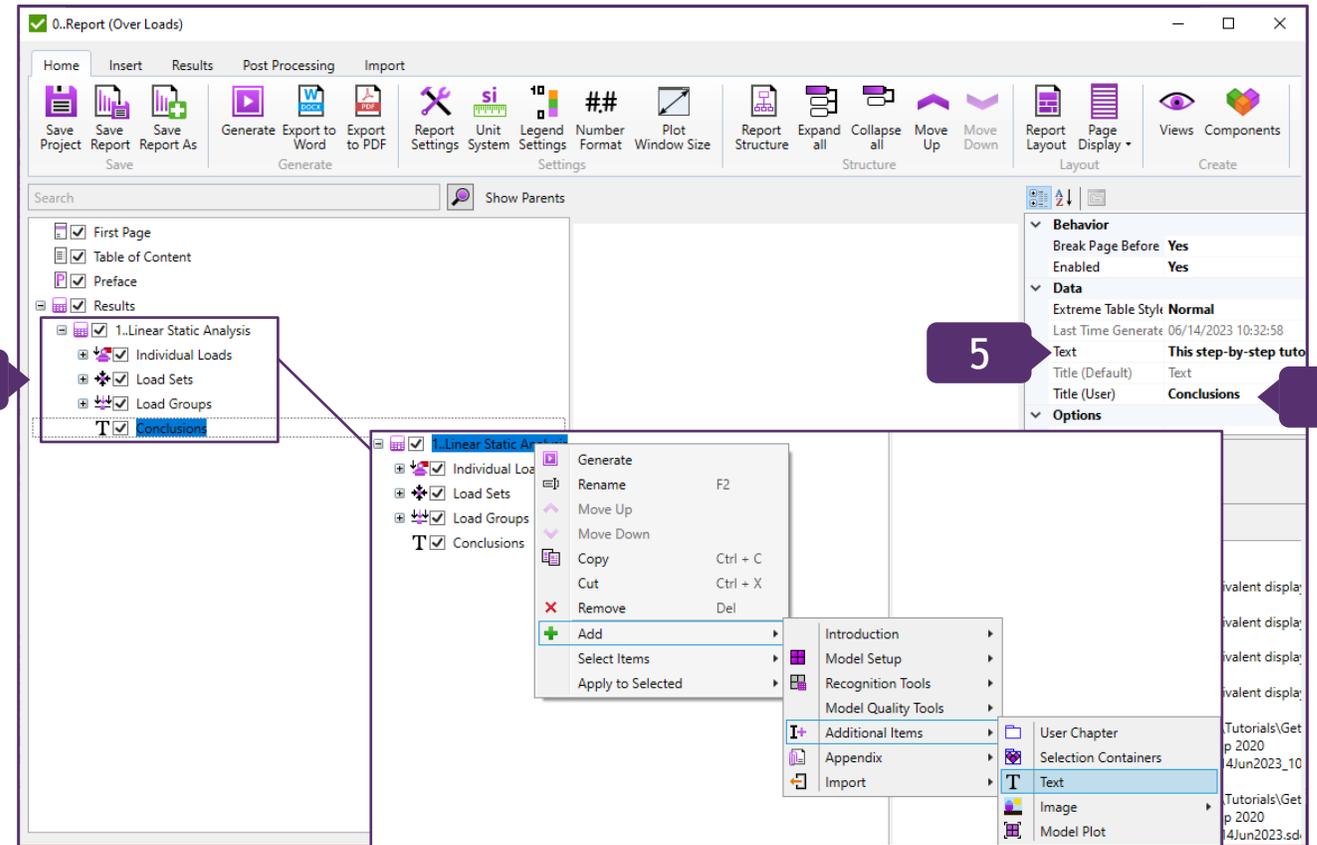
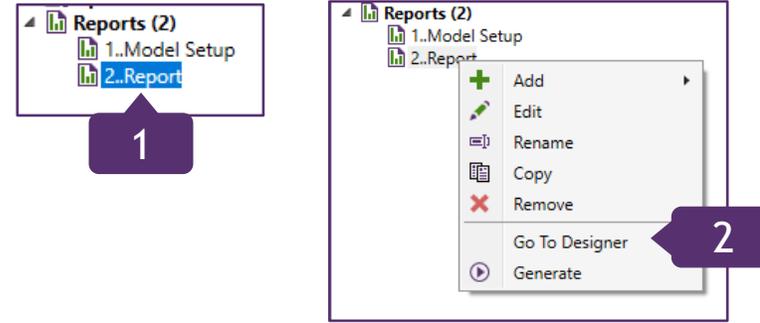
2 Execute Go To Designer from the context menu

3 Expand Linear Static Analysis => Add
=> Additional Items => Text

4 Title (User): Conclusions

5 Text: This step-by-step tutorial is designed to get you started with the main SDC Verifier features

Report Wizard guides you through the steps of building and running the report. Whereas the Report Designer gives added ability to design your own reports.



Generate a Report for a Solid Model

1 Expand Individual Loads => Expand Individual Load 1..Top_Edge_Vertical

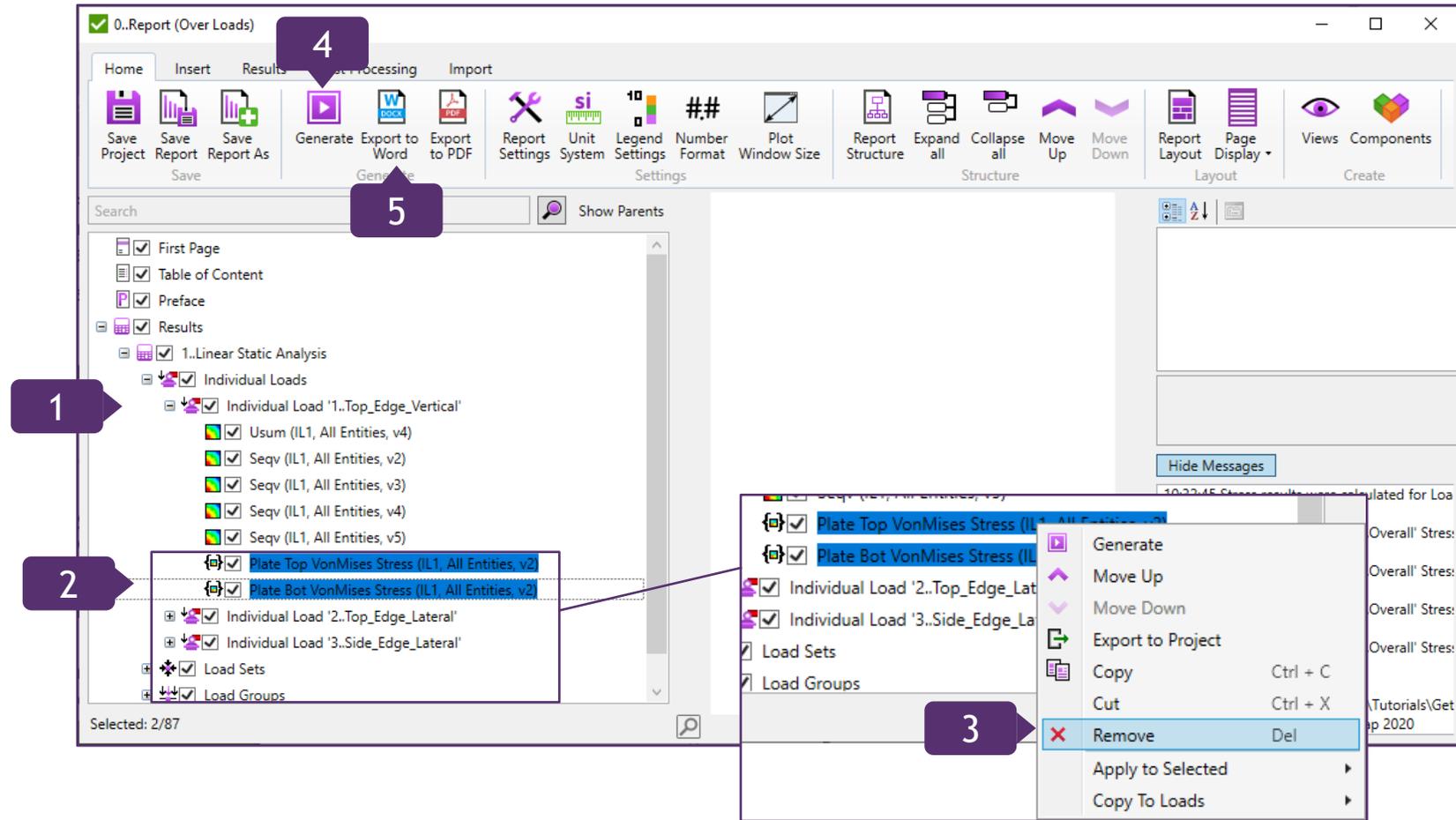
2 Select Plate Top VonMises Stress and Plate Bot VonMises Stress plots

3 Remove selected plots

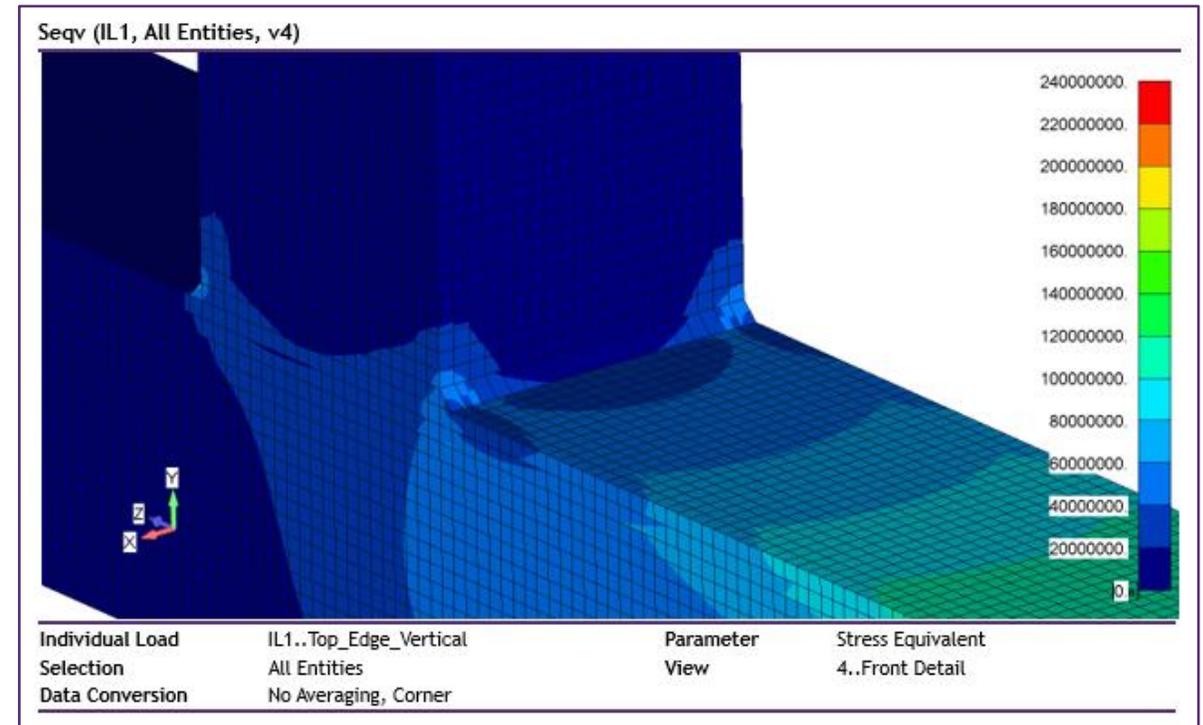
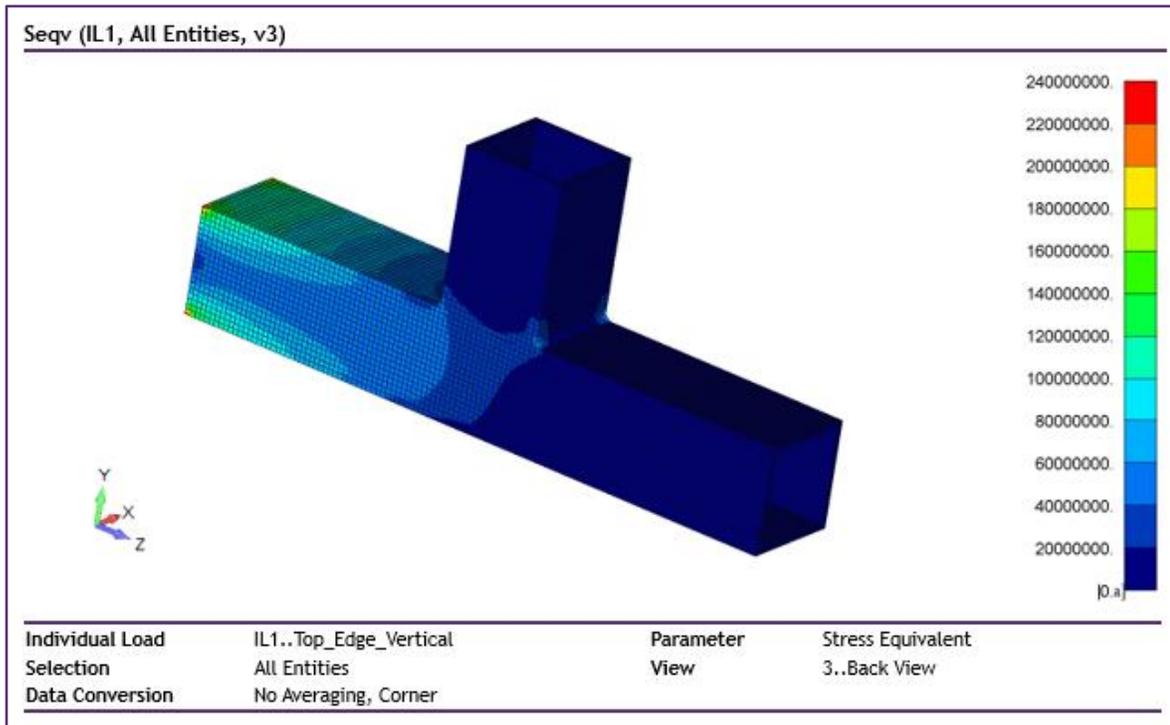
4 Press  to generate report

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

Remove Output Vector Plots that were created for plate model, as they do not exist for the solid model.

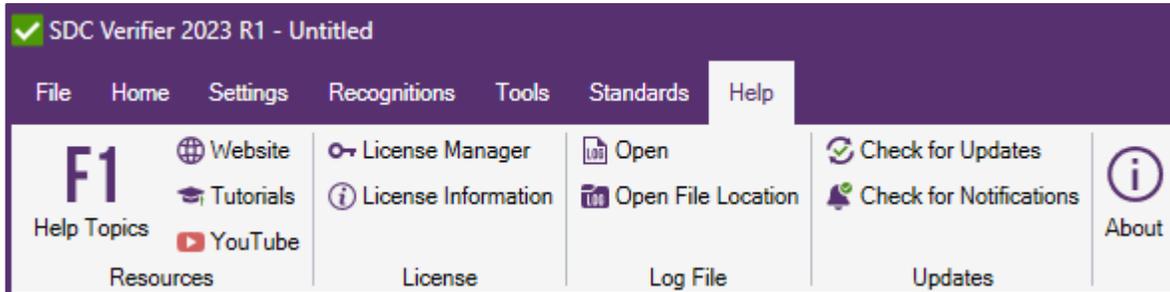


Equivalent Stress plots are automatically displayed for solid model.



1

On the Ribbon, go to *Help* section, where informational resources, tutorials, License Information, Checks for Updates etc. can be found.



In order to clarify any queries or resolve issues that might arise throughout SDC Verifier usage, customers can contact our Support Team by the following communication channels:

Skype: [sdcverifier_helpdesk](https://www.skype.com/people/sdcverifier_helpdesk);
phone: +31 15 30-10-310;
email: support@sdcverifier.com

Additionally, if the screen demonstration is required, we offer a possibility to schedule a Teams meeting call with our Engineering or Development Team. In this case, we will need the customer's email address to agree on the date and time, and send the customer a meeting invitation.