



Get started with SDC Verifier

13 Jan 2020
version 5.3

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

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

- ▶ Creating new project;
- ▶ Create Individual Loads, Combinations and Envelop;
- ▶ Define Views;
- ▶ Model Setup report;
- ▶ Calculation report;
- ▶ Open as template feature;

Create new project

1 Launch **SDC Verifier 5.3 for Simcenter** 

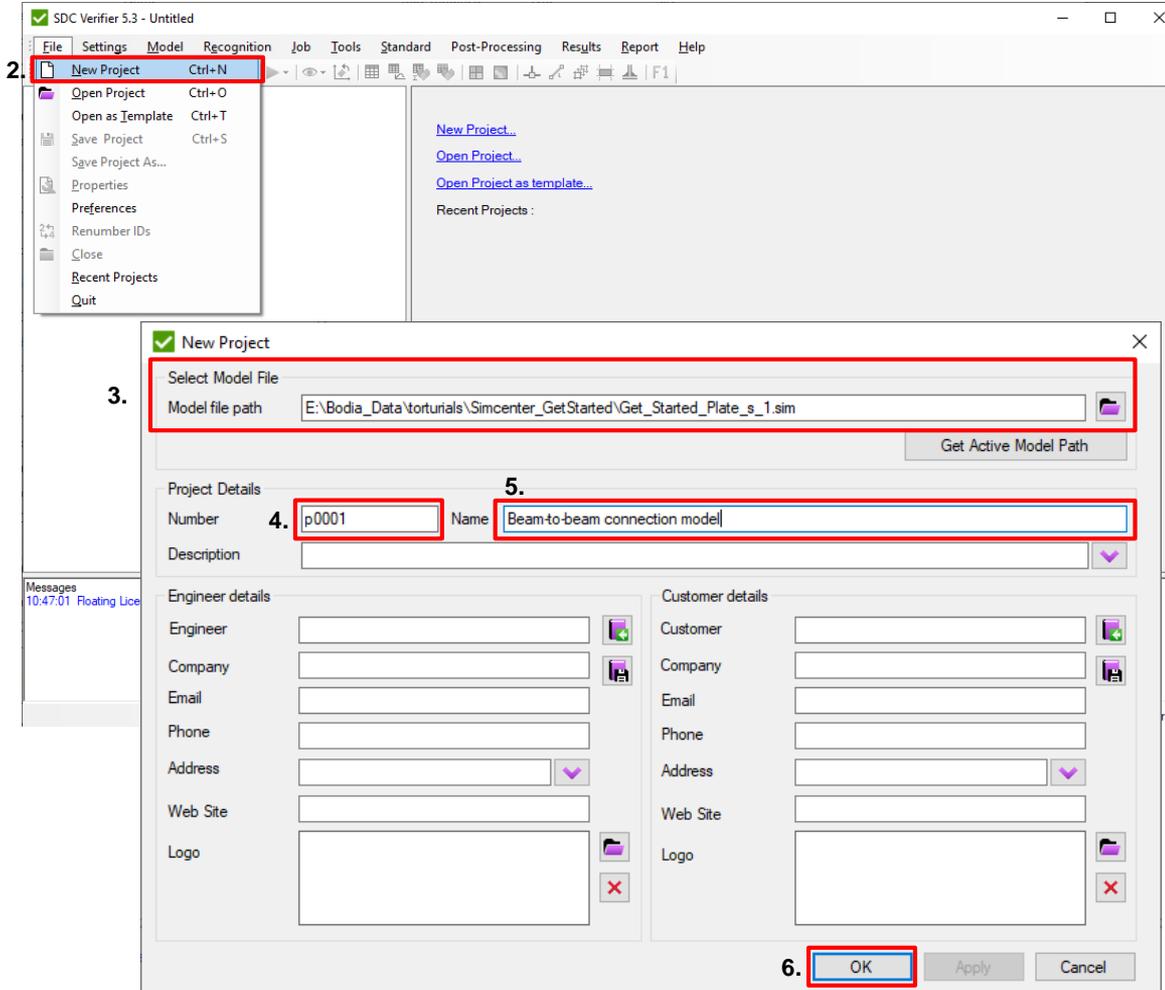
2 Execute *File - New Project*.

3 Press  and select ***Get_Started_Plate_s_1.sim*** model.

4 Number: **p0001**

5 Name: **Beam-to-beam connection model**

6 Press **OK**



Job explanation

1 Title: **Linear Static Analysis.**

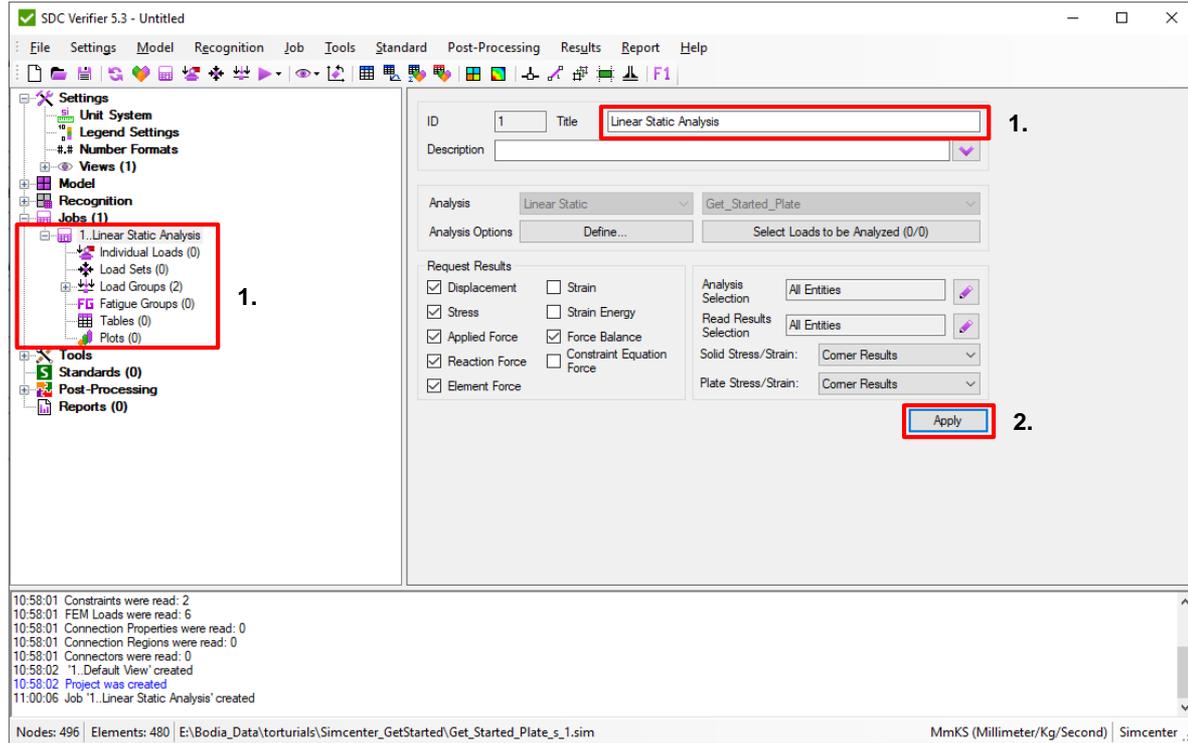
2 Press *Apply*.

Job – 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)



Create individual loads.

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

2 Select *FEM Loads* with IDs 1-3 and *Constraints* with ID 1.

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).

SDC Verifier 5.3 - Untitled

File Settings Model Recognition Job Tools Standard Post-Processing Results Report Help

Settings
Unit System
Legend Settings
Number Formats
Views (1)
Model
Recognition
Jobs (1)
1. Linear Static Analysis
Individual Loads (0) 1.
Load Sets (0)
Load Groups (2)
Fatigue Groups (0)
Tables (0)
Plots (0)
Tools
Standards (0)
Post-Processing
Reports (0)

Calculation Loads From Existing Results (based on Result Cases)

FEM Loads	Constraints
1. FORCE 1_Node ID Table	1.SPC1_4
2. FORCE 2_Node ID Table	2.SPC SS. SID 1
3. FORCE 3_Node ID Table	
4. Stat SS - Load SID 1	
5. Stat SS - Load SID 2	
6. Stat SS - Load SID 3	

3. Create

10:58:01 Connection Regions were read: 0
10:58:01 Connectors were read: 0
10:58:02 '1.Default View' created
10:58:02 Project was created
11:00:06 Job '1..Linear Static Analysis' created
11:55:30 Saving backup file...
11:55:30 E:\Bodia_Data\tutorials\Simcenter_GetStarted\Get_Started_Plate_s_1_autobackup_13Jan2020_11-55AM simb saved
11:55:30 E:\Bodia_Data\tutorials\Simcenter_GetStarted\Get_Started_Plate_s_1_dailybackup_13Jan2020 simb saved

Nodes: 496 | Elements: 480 | E:\Bodia_Data\tutorials\Simcenter_GetStarted\Get_Started_Plate_s_1.sim MmKS (Millimeter/Kg/Second) | Simcenter

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 how to create individual loads based on Result Cases. 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 Result Cases.

4. Press *Create*.

3 Individual Loads will be created based on 3 Result Cases

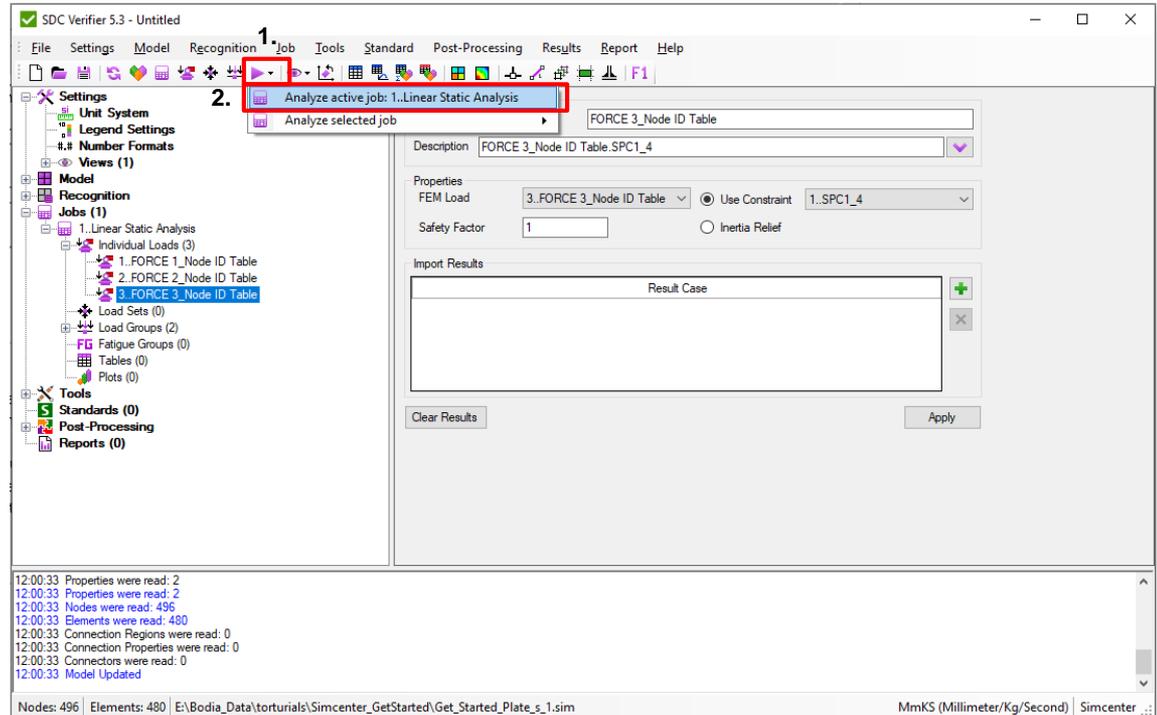
The screenshot shows the SDC Verifier 5.3 interface. On the left, the Model Tree is expanded to 'Individual Loads (3)', with a red box around the 'Individual Loads (3)' folder and a '1.' label. In the center, the 'Calculation Loads' tab is active, showing a dropdown menu with 'From Existing Results (based on Result Cases)' selected, and a '2.' label. Below this, three result cases are listed and selected with a blue highlight: 'Linear Static Analysis Linear - FORCE_1_Node ID Table (Static Step 1)', 'Linear Static Analysis Linear - FORCE_2_Node ID Table (Static Step 1)', and 'Linear Static Analysis Linear - FORCE_3_Node ID Table (Static Step 1)', with a '3.' label. At the bottom right, a 'Create' button is highlighted with a red box and a '4.' label. The bottom status bar shows 'Nodes: 496 | Elements: 480 | E:\Bodia_Data\tutorials\Simcenter_GetStarted\Get_Started_Plate_s_1.sim' and 'MmKS (Millimeter/Kg/Second) | Simcenter ...'. The bottom left shows a log of analysis steps.

Analyze Job

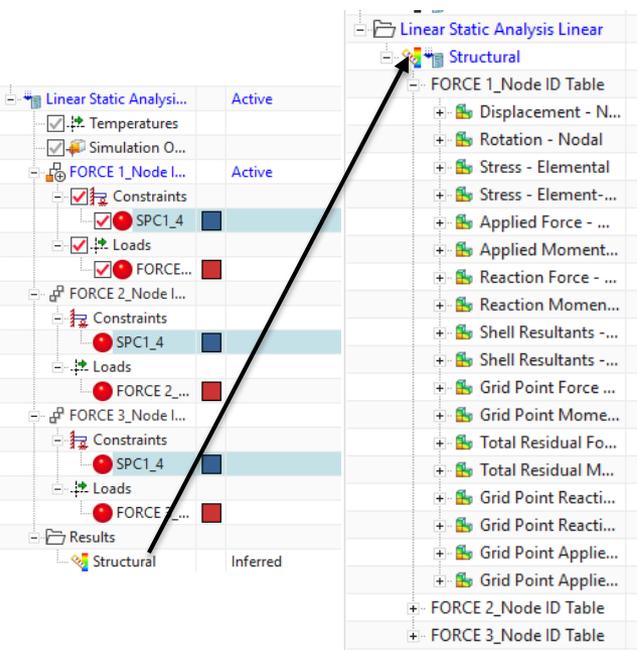
1 Press  on toolbar to analyze job.

2 Select *Analyze active job: 1..Linear Static Analysis*

Solution with 3 cases will be created and run. Result Cases will be automatically linked to analyzed Individual Loads after analysis is finished.



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*

1. Load Sets (0) is selected in the Model tree.

2. Title: All_combinations

3. Individual Loads (3) are selected in the list.

4. Right arrow button is pressed.

5. Individual Loads are selected in the Loads table.

6. Add Logic Factor button is pressed.

7. Create button is pressed.

Load Set	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

List of the created Load Sets

Edit Multiple Load Sets. Modify factors

- 1 Execute *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.

All_combinations 1	1	1	1	1.1
All_combinations 2	1	-1	1	1.1
All_combinations 3	1	1	-1	1.1
All_combinations 4	1	-1	-1	1.1
All_combinations 5	1	1	1	-1.1
All_combinations 6	1	-1	1	-1.1
All_combinations 7	1	1	-1	-1.1
All_combinations 8	1	-1	-1	-1.1

The screenshot shows the SDC Verifier 5.3 interface. The 'Load Sets' node in the Model tree is selected, and the 'Create/Edit multiple' context menu is open. The 'Create/Edit Multiple Load Sets' dialog box is displayed, showing a table of load sets. The table has columns for Safety Factor, IL1..FORCE 1, IL2..FORCE 2, and IL3..FORCE 3. The rows are labeled All_combinations.1 through All_combinations.8. The 'Factor' field is set to 1.1. The 'OK' button is highlighted.

	Safety Factor	IL1..FORCE 1, Node ID Table	IL2..FORCE 2, Node ID Table	IL3..FORCE 3, Node ID Table	
All_combinations.1	1	1	1	1.1	1.
All_combinations.2	1	-1	1	1.1	
All_combinations.3	1	1	-1	1.1	2.
All_combinations.4	1	-1	-1	1.1	
All_combinations.5	1	1	1	-1.1	
All_combinations.6	1	-1	1	-1.1	4.
All_combinations.7	1	1	-1	-1.1	
All_combinations.8	1	-1	-1	-1.1	

Create Load Group (Envelop)

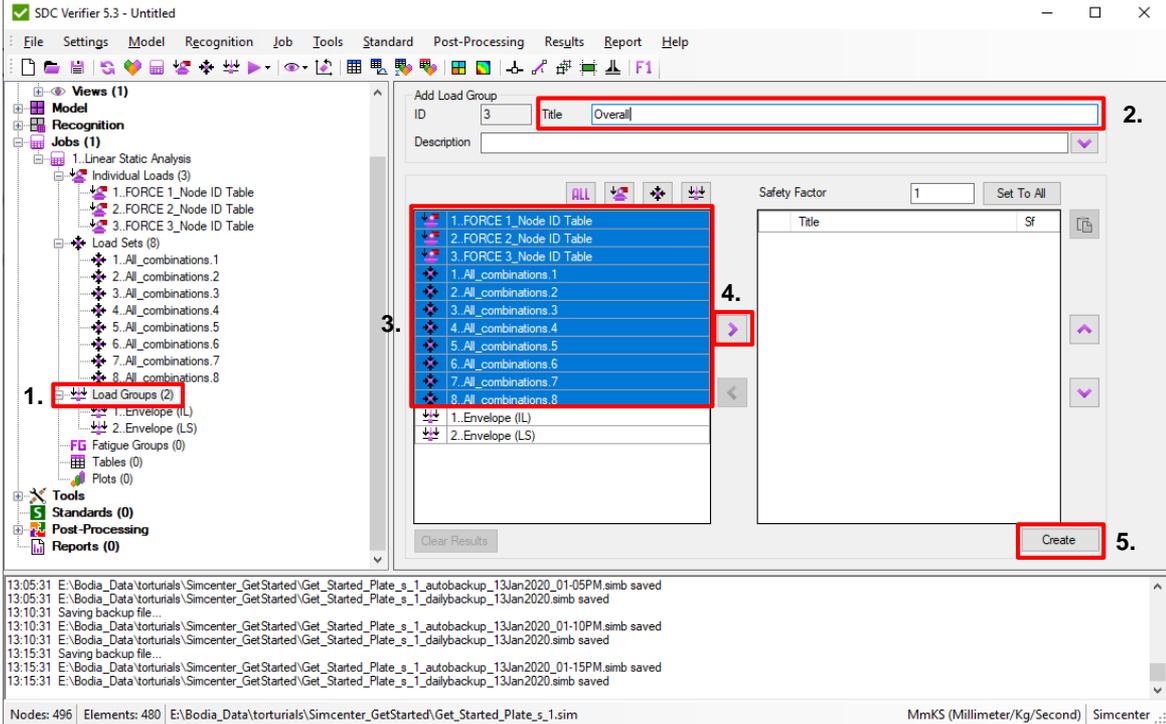
1 Activate *Load Groups* node in the *Model* tree.

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*.



The screenshot shows the SDC Verifier 5.3 interface. On the left, the 'Model' tree is expanded to 'Load Groups (2)', with the 'Load Groups (2)' node highlighted by a red box and labeled '1.'. The 'Add Load Group' dialog is open on the right. The 'Title' field contains 'Overall' and is highlighted by a red box and labeled '2.'. The 'Description' field is empty. Below the dialog, a list of load items is shown, with the first three items (1.FORCE_1_Node ID Table, 2.FORCE_2_Node ID Table, 3.FORCE_3_Node ID Table) and the last three items (7.All_combinations.7, 8.All_combinations.8) selected and highlighted in blue. A red box and arrow labeled '3.' points to the selection, and another red box and arrow labeled '4.' points to the right arrow button. The 'Create' button at the bottom right of the dialog is highlighted by a red box and labeled '5.'. The 'Safety Factor' is set to 1. The status bar at the bottom shows 'Nodes: 496 | Elements: 480 | E:\Bodia_Data\torturials\Simcenter_GetStarted\Get_Started_Plate_s_1.simb | MmKS (Millimeter/Kg/Second) | Simcenter ...'

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.

Create 2 general Views for plots

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

2. Execute *View – Add* from context menu

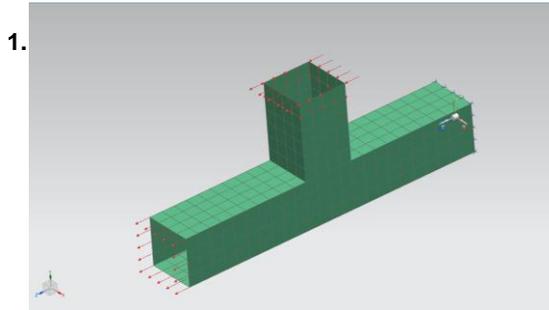
3. Title: **Front View**. Press *OK*.

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

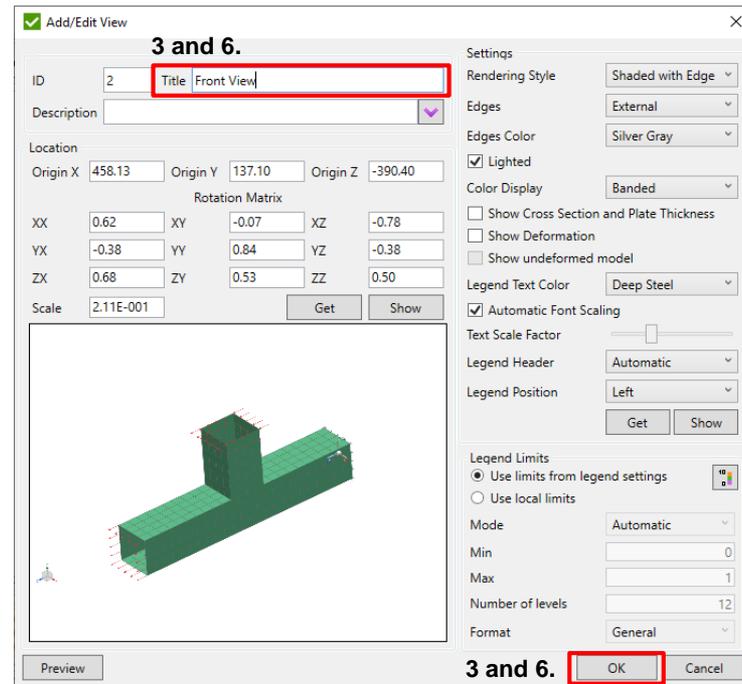
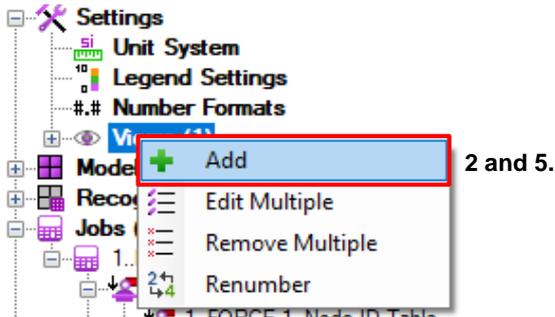
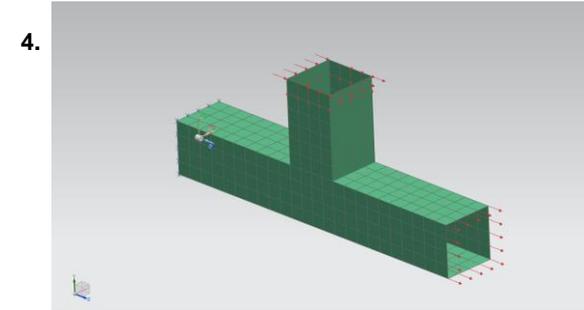
5. Execute *View – Add* from context menu

6. Title: **Back View**. Press *OK*.

Front View



Back View



Create 2 detailed Views

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

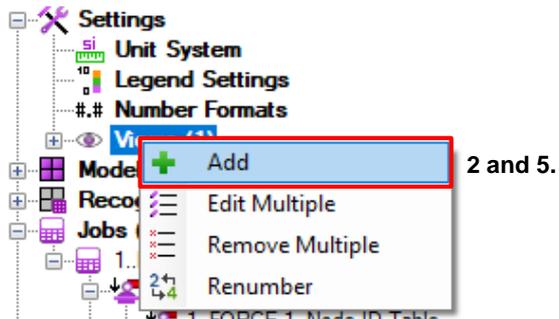
2. Execute *View – Add* from context menu

3. Title: **Front Detail**. Press *OK*.

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

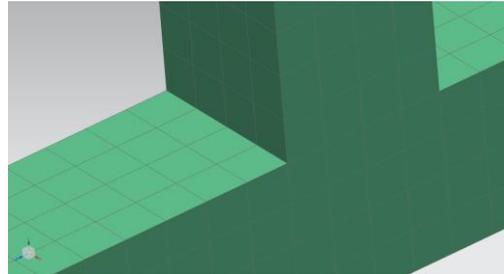
5. Execute *View – Add* from context menu

6. Title: **Back Detail**. Press *OK*.



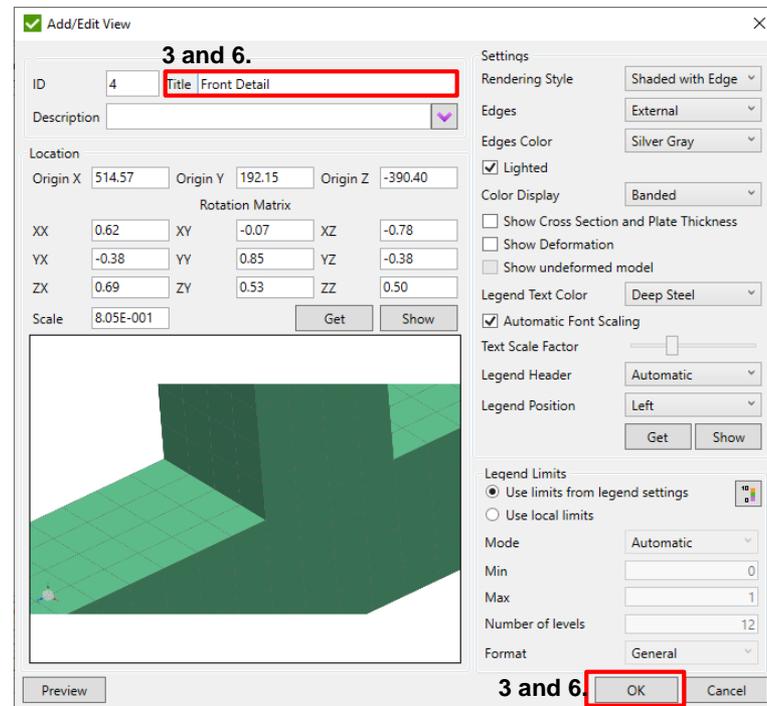
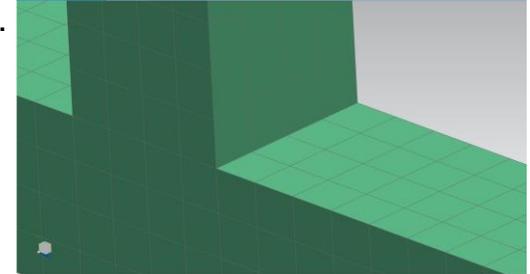
Front Detail

1.



Back Detail

4.



Report Wizard – Model Setup report

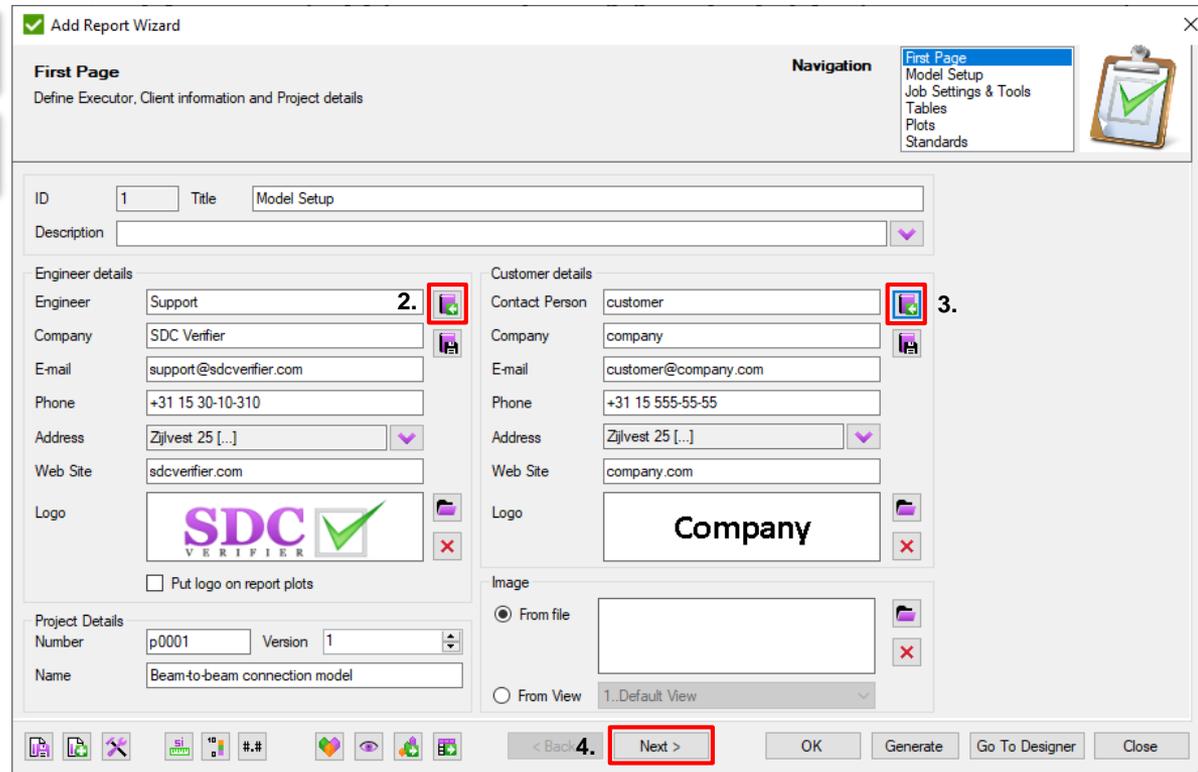
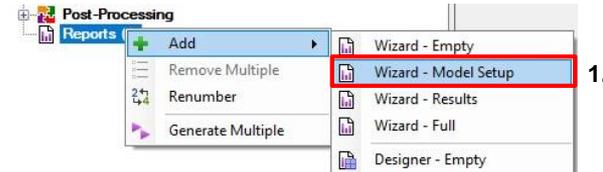
1 Execute Add -> Wizard - Model Setup from Reports in the Model tree.

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).

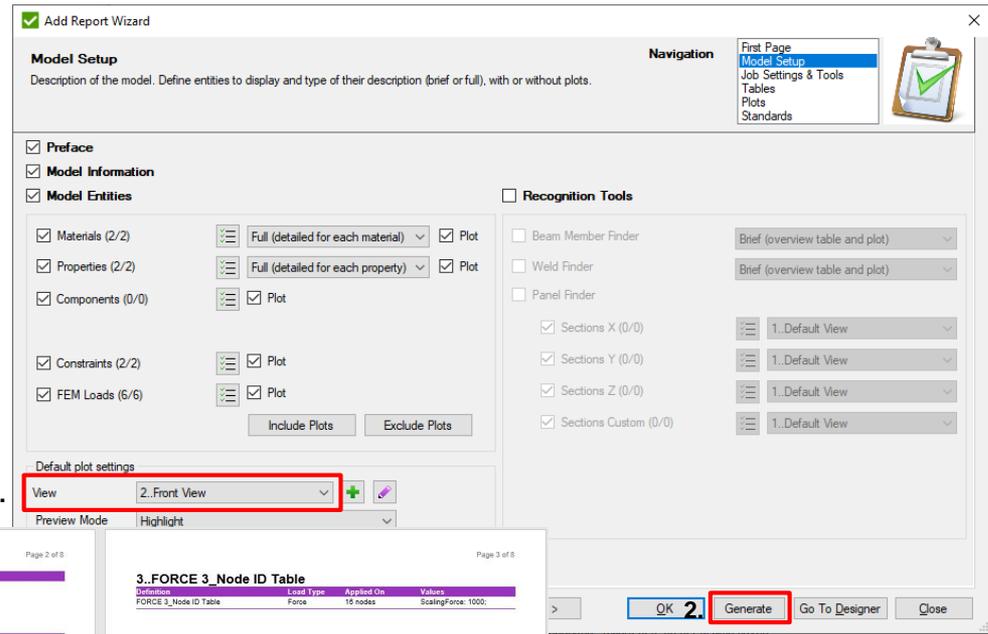


The 'Add Report Wizard' dialog box is shown. It has a 'Navigation' pane on the right with 'First Page' selected. The main area is divided into 'Engineer details' and 'Customer details' sections. In the 'Engineer details' section, the 'Engineer' field is set to 'Support' and has a file icon button highlighted with a red box and a '2.' next to it. In the 'Customer details' section, the 'Contact Person' field is set to 'customer' and has a file icon button highlighted with a red box and a '3.' next to it. At the bottom, the 'Next >' button is highlighted with a red box and a '4.' next to it. The 'Project Details' section at the bottom left shows 'Number' as 'p0001' and 'Name' as 'Beam-to-beam connection model'.

Model Setup options

1 Plot View: **Front View**

2 Press **Generate**.



Model Setup

Beam-to-beam connection model

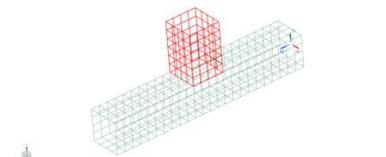
Prepared by:
SDC Verifier
+31 15 30-10-310
sdverifier.com
Zijlvest 25
2011 VB Haarlem
The Netherlands

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

Engineer: Support
Customer: customer
Project Number: p0001
Version: 1
Date: 13-01-2020

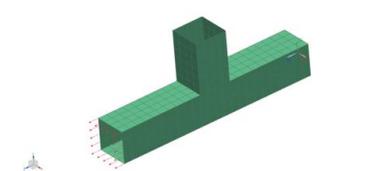
Page 2 of 8

Property	Value
Elements	55
Type	Plate
Material	2. steel_s05
Mass [kg]	18.2
Gravity Center [mm]	{0.00; 247.50; 600.00}
Thicknesses 1 [mm]	10.000
Thicknesses 2 [mm]	0
Thicknesses 3 [mm]	0
Thicknesses 4 [mm]	0
Nonstructural Mass/Area [kg/mm ²]	0
Top Fiber [mm]	0
Bottom Fiber [mm]	0
Bend Stiffness [mm ³]	0
TShearMem Thickness	0



3..FORCE 3_Node ID Table

Node ID	Node Type	Applied On	Values
FORCE_3_Node ID Table	Force	15 nodes	ScalingForce: 1000.



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.

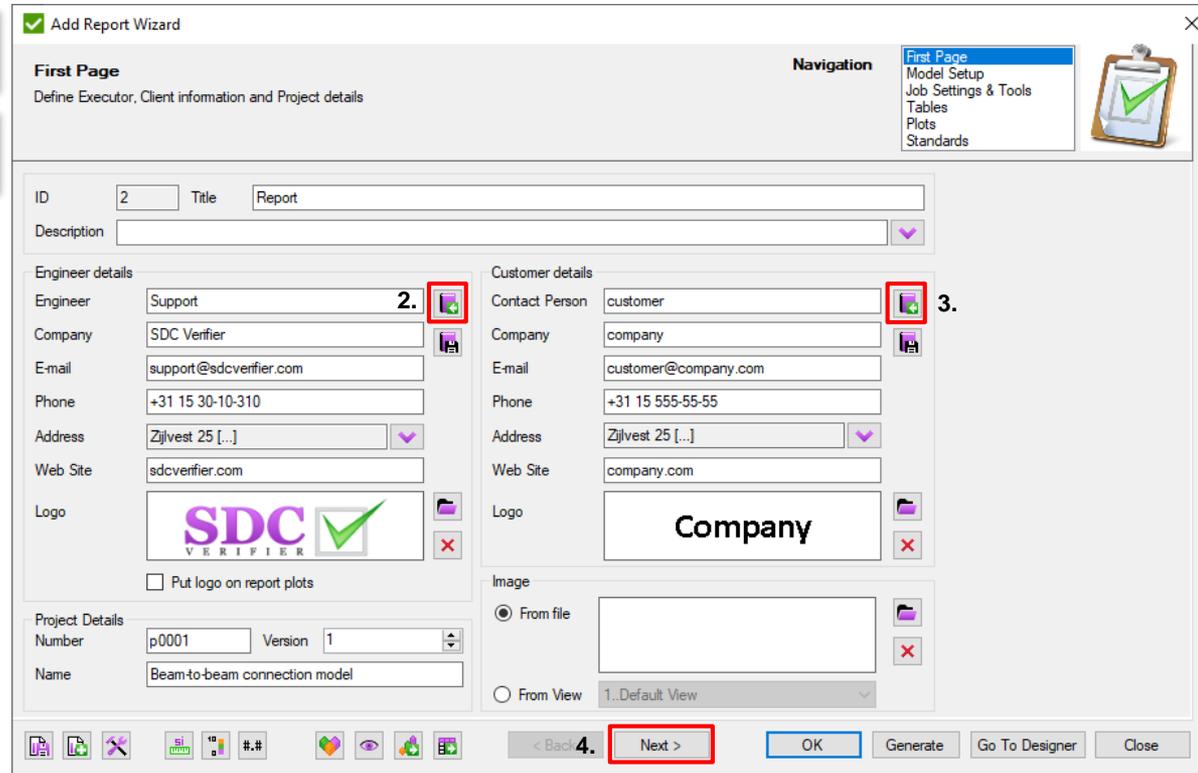
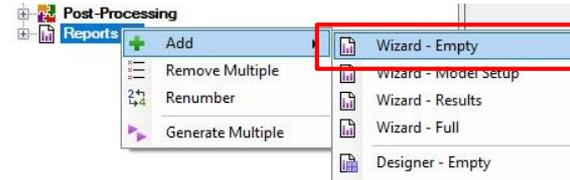
Create calculation report

1 Execute Add -> Wizard - Empty on Reports in the Model tree.

2 Press  and select Support Engineer from the library

3 Press  and select Customer from the library

4 Press Next 2 times.



The 'Add Report Wizard' dialog box is shown. It has a 'First Page' tab selected in the 'Navigation' pane. The 'First Page' contains the following fields:

- ID: 2
- Title: Report
- Description: (empty)
- Engineer details:
 - Engineer: Support (selected from a dropdown, indicated by a red box and '2.')
 - Company: SDC Verifier
 - E-mail: support@sdcverifier.com
 - Phone: +31 15 30-10-310
 - Address: Zijlvest 25 [...]
 - Web Site: sdcverifier.com
 - Logo: SDC Verifier logo (indicated by a red box and '3.')
 - Put logo on report plots
- Customer details:
 - Contact Person: customer (selected from a dropdown, indicated by a red box and '3.')
 - Company: company
 - E-mail: customer@company.com
 - Phone: +31 15 555-55-55
 - Address: Zijlvest 25 [...]
 - Web Site: company.com
 - Logo: Company logo
- Image:
 - From file
 - From View: 1..Default View
- Project Details:
 - Number: p0001
 - Version: 1
 - Name: Beam-to-beam connection model

At the bottom, there are buttons: '< Back 4.', 'Next >' (highlighted with a red box), 'OK', 'Generate', 'Go To Designer', and 'Close'.

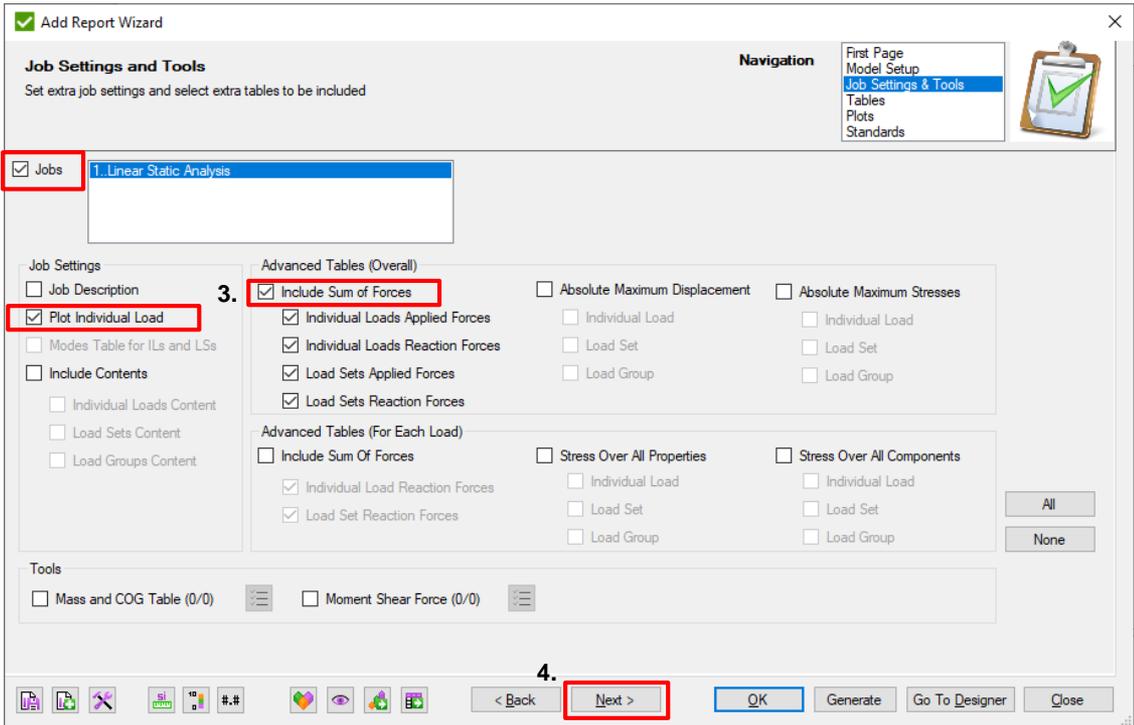
Predefined Job Tables

1 Jobs: **ON**

2 Plot Individual Loads: **ON**

3 Include Sum of Forces: **ON**

4 Press **Next**.



1. Jobs

2. Plot Individual Load

3. Include Sum of Forces

4. **Next >**

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*

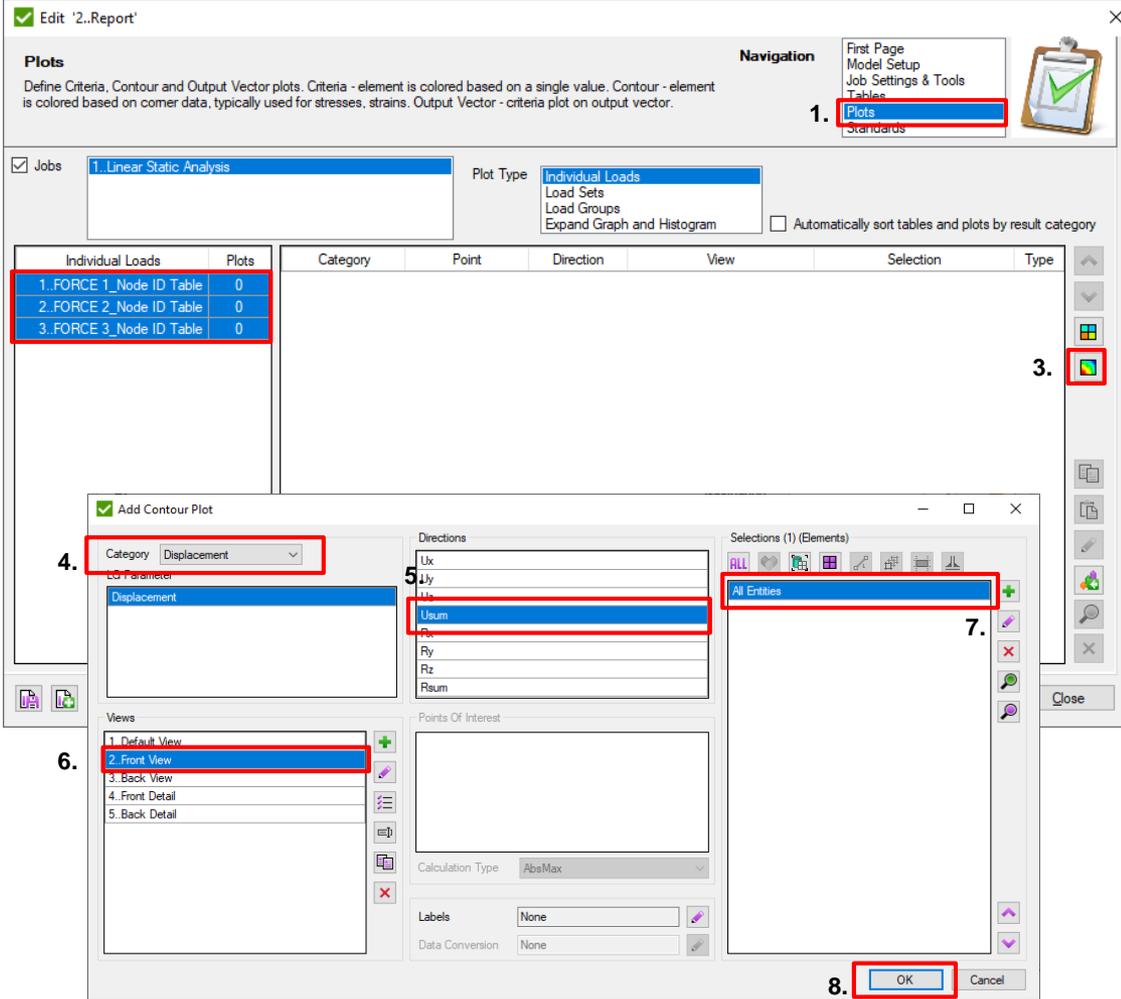
5 *Direction: Usum*

6 *Views: Front View*

7 *Selection: All Elements*

8 Press *OK*.

2.



The screenshot shows the 'Edit Report' dialog box with the 'Plots' section selected. The 'Jobs' list contains '1. Linear Static Analysis'. The 'Plot Type' is set to 'Individual Loads'. A table lists three force nodes, each with a value of 0. The 'Add Contour Plot' dialog is open, showing the 'Category' set to 'Displacement', the 'Direction' set to 'Usum', and the 'Views' list with '2. Front View' selected. The 'Selections' list is set to 'All Entities'. The 'OK' button is highlighted.

Individual Loads	Plots
1. FORCE 1_Node ID Table	0
2. FORCE 2_Node ID Table	0
3. FORCE 3_Node ID Table	0

Category	Point	Direction	View	Selection	Type

4. Category: Displacement

5. Direction: Usum

6. Views: Front View

7. Selection: All Elements

8. OK

Add stress plots

1 Select All Loads from the list.

2 Press  to add criteria plots.

3 *Category: Stress*

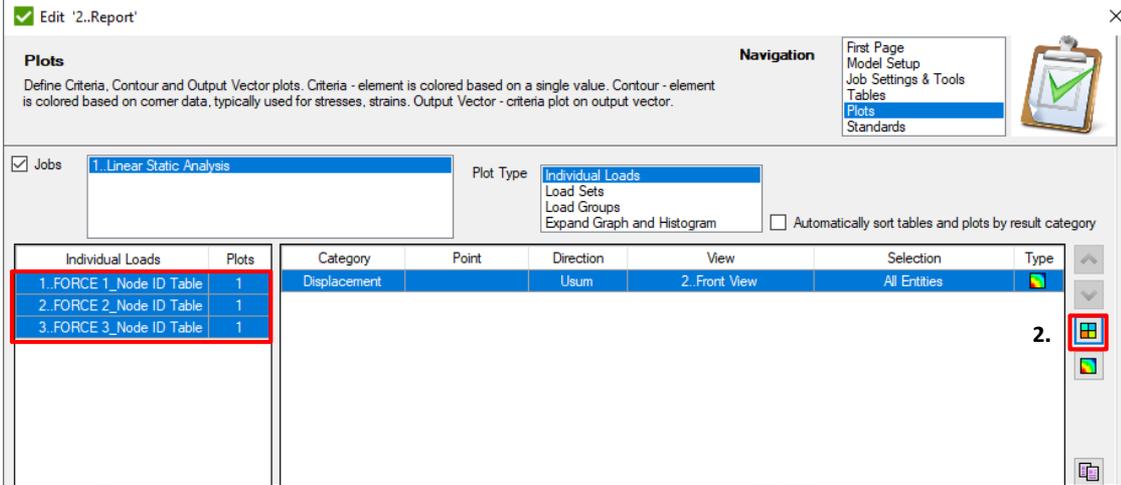
4 *Direction: Equivalent.*

5 Select 4 Views: ID from 2 to 5

6 *Selection: All Elements*

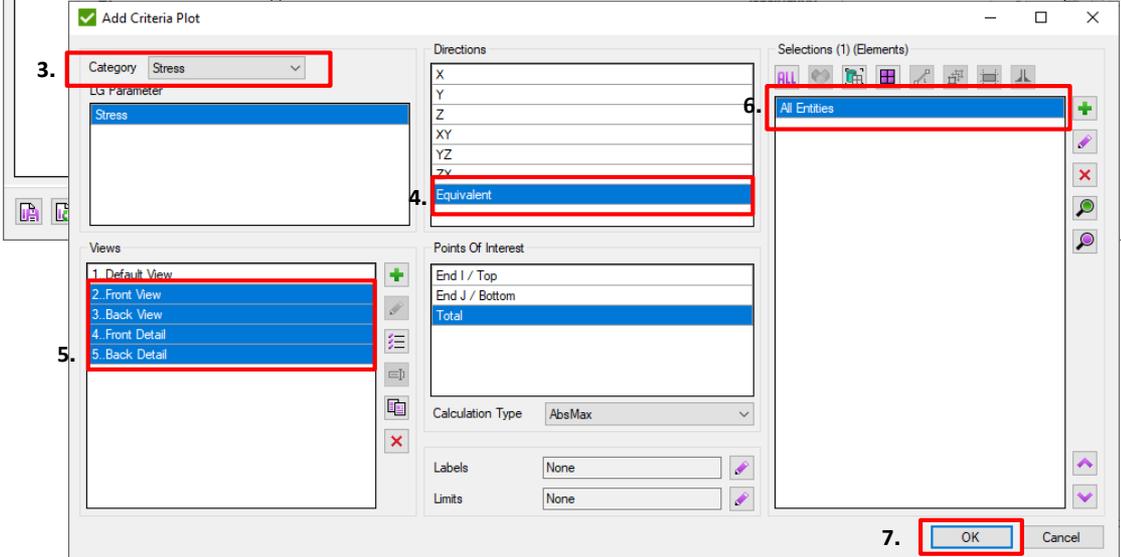
7 Press *OK*.

1.



Individual Loads	Plots	Category	Point	Direction	View	Selection	Type
1..FORCE 1_Node ID Table	1	Displacement		Usum	2. Front View	All Entities	2.
2..FORCE 2_Node ID Table	1						
3..FORCE 3_Node ID Table	1						

3.



4. Equivalent

6. All Entities

5.

7. OK

Copy plots to Load Sets and Load Groups

- 1 Select all plots from the list.
- 2 Press to copy plot to clipboard.
- 3 Plot Type: **Load Sets**
- 4 Select all loads sets.
- 5 Press to paste.
- 6 Plot Type: **Load Groups**
- 7 Select **Overall** Load Group.
- 8 Press to paste.

1. Select all plots from the list.

Individual Loads	Plots
1. FORCE_1_Node ID Table	5
2. FORCE_2_Node ID Table	5
3. FORCE_3_Node ID Table	5

2. Press to copy plot to clipboard.

Category	Point	Direction	View	Selection	Type
Displacement	Usun	Equivalent	2. Front View	All Entities	
Stress	Total(AbsMax)	Equivalent	2. Front View	All Entities	
Stress	Total(AbsMax)	Equivalent	3. Back View	All Entities	
Stress	Total(AbsMax)	Equivalent	4. Front Detail	All Entities	
Stress	Total(AbsMax)	Equivalent	5. Back Detail	All Entities	

3. Plot Type: **Load Sets**

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

4. Select all loads sets.

Category	Point	Direction	View	Selection	Type
Displacement	Usun	Equivalent	2. Front View	All Entities	
Stress	Total(AbsMax)	Equivalent	2. Front View	All Entities	
Stress	Total(AbsMax)	Equivalent	3. Back View	All Entities	
Stress	Total(AbsMax)	Equivalent	4. Front Detail	All Entities	
Stress	Total(AbsMax)	Equivalent	5. Back Detail	All Entities	

5. Press to paste.

6. Plot Type: **Load Groups**

Load Groups	Plots
1. Envelope (LL)	0
2. Envelope (RS)	0
3. Overall	5

7. Select **Overall** Load Group.

Category	Point	Parameter	Direction	View	Selection	Type
Displacement	Absolute	Usun	Equivalent	2. Front View	All Entities	
Stress	Absolute	Equivalent	Equivalent	2. Front View	All Entities	
Stress	Absolute	Equivalent	Equivalent	3. Back View	All Entities	
Stress	Absolute	Equivalent	Equivalent	4. Front Detail	All Entities	
Stress	Absolute	Equivalent	Equivalent	5. Back Detail	All Entities	

8. Press to paste.

Result Report Generation

- Press to save the report profile
- Press *Generate*.

Edit '2..Report'

Plots
Define Criteria, Contour and Output Vector plots. Criteria - element is colored based on a single value. Contour - element is colored based on corner data, typically used for stresses, strains. Output Vector - criteria plot on output vector.

Navigation
First Page
Model Setup
Job Settings & Tools
Tables
Plots
Standards

Jobs: 1. Linear Static Analysis Plot Type: Individual Loads, Load Sets, Load Groups, Expand Graph and Histogram Automatically sort tables and plots by result category

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

Category	Point	Parameter	Direction	View	Selection	Type
Displacement	Total(AbsMax)	Absolute	Usym	2. Front View	All Entities	
Stress	Total(AbsMax)	Absolute	Equivalent	2. Front View	All Entities	
Stress	Total(AbsMax)	Absolute	Equivalent	3. Back View	All Entities	
Stress	Total(AbsMax)	Absolute	Equivalent	4. Front Detail	All Entities	
Stress	Total(AbsMax)	Absolute	Equivalent	5. Back Detail	All Entities	

1.

2.

Report

Beam-to-beam connection model

Prepared by: SDC Verifier
+31 15 30-10-010
sdcverifier.com
2016-01-25
2011 VG Houtster
The Netherlands

Prepared for: company
+31 15 305-05-05
company.com
2016-01-25
2011 VG Houtster
The Netherlands

Engineer: Support
Customer: customer
Project Number: p0001
Version: 1
Date: 14-01-2020

1..Linear Static Analysis Individual Loads

In this paragraph the influence of the different response loads is described.
Individual Load '1..FORCE 1_Node ID Table'
FORCE 1_Node ID Table: SPC1_4

1.

Individual Load Selection	Point	Parameter	View	Stress Equations	Element
1..FORCE 1_Node ID Table	All Entities	Force	Line	None	None

Applied Force Summation (2 Loads, All Entities)

Load	Fx [kN]	Fy [kN]	Fz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
S1..1 FORCE 1_Node ID Table	0.0	0.0	10000000.0	0.0	0.0	0.0
S2..2 FORCE 2_Node ID Table	0.0	0.0	10000000.0	0.0	0.0	0.0
S3..3 FORCE 3_Node ID Table	0.0	0.0	10000000.0	0.0	0.0	0.0

Reaction Force Summation (2 Loads, All Entities)

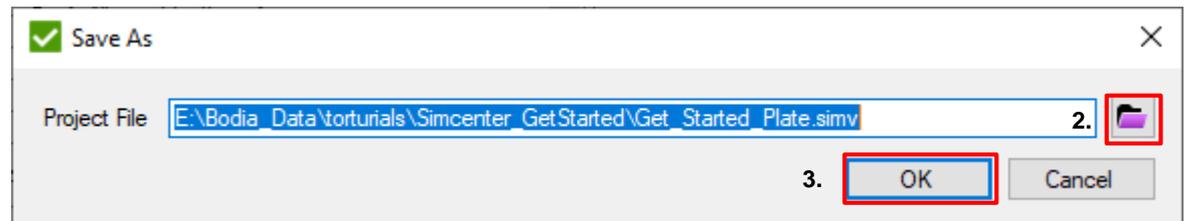
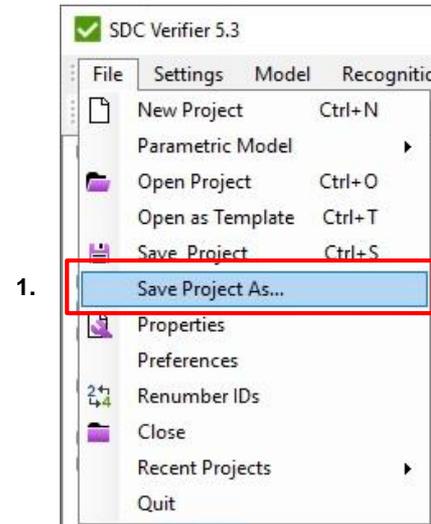
Load	Fx [kN]	Fy [kN]	Fz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
S1..1 FORCE 1_Node ID Table	0.0	0.0	-10000000.0	0.0	0.0	0.0
S2..2 FORCE 2_Node ID Table	-0.1	-0.1	10000000.0	0.0	0.0	0.0
S3..3 FORCE 3_Node ID Table	-0.1	0.1	-10000000.0	0.0	0.0	0.0

Save SDC project

1 Execute *File - Save Project As*

2 Press  to browse location and define the filename

3 Press *OK*



Open As Template – solid model

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

1 Execute *File - Open as Template*.

2 *Template Project: Get_Started_Plate.simv*

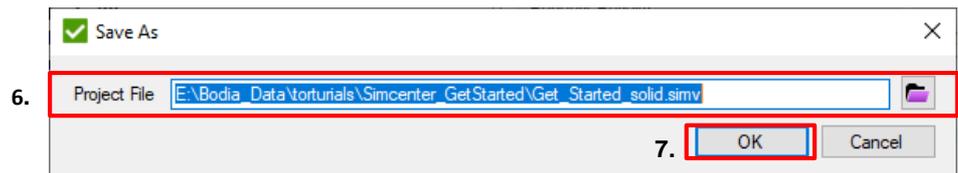
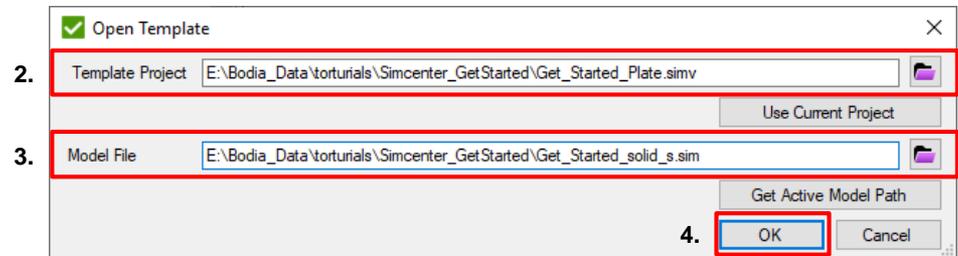
3 *Model File: Get_Started_solid_s.sim*

4 Press *OK*.

5 Execute *File - Save Project as*

6 *Project File: Get_Started_solid.simv*

7 Press *OK*.



Generate report for Solid model

1 Select *Report* in the *Model* tree

2 Execute *Go To Designer* from the context menu

3 Add *Text* item from *Toolbox*

4 Title: **Conclusions**

5 Text:

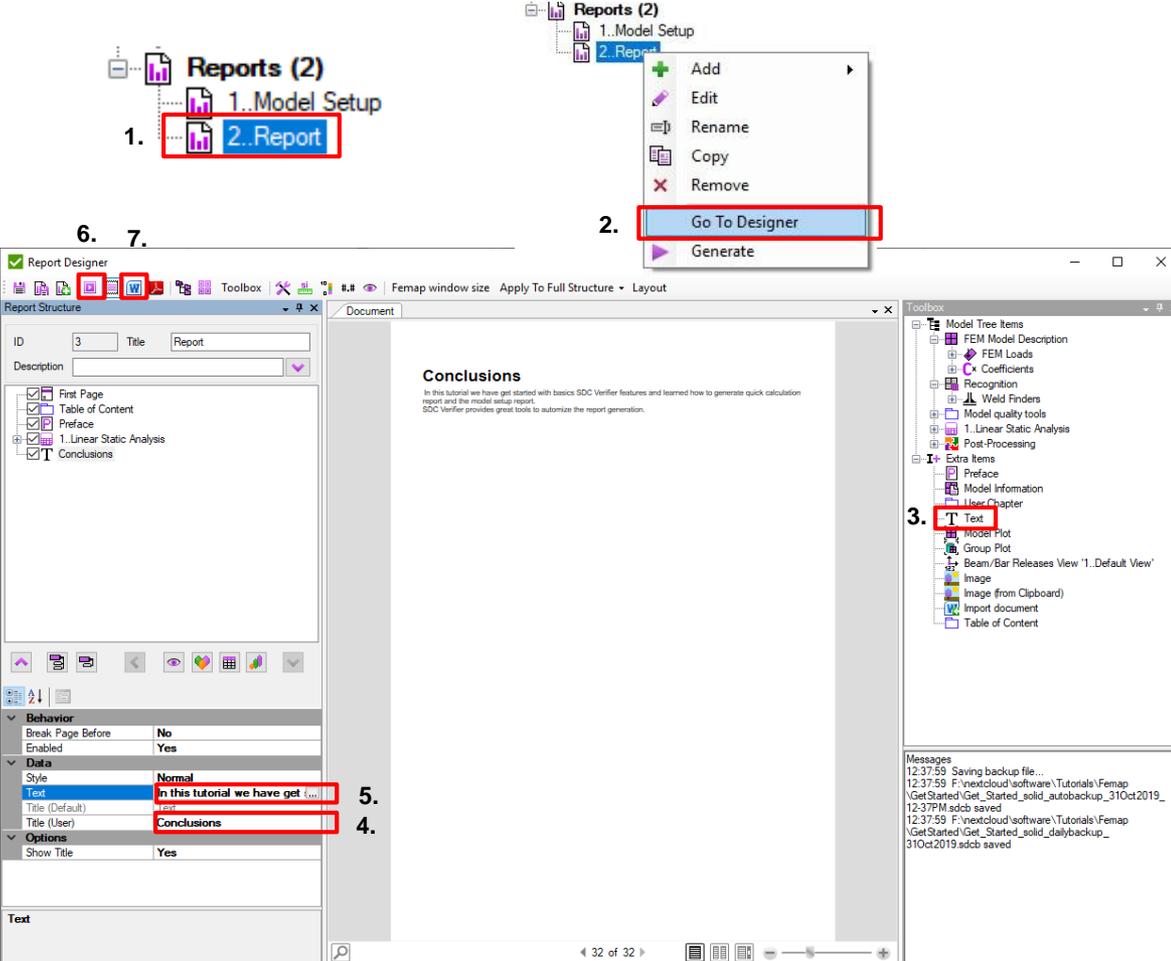
6 Press  to generate report

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

Note:

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.



1. **2..Report**

2. **Go To Designer**

6. 7.

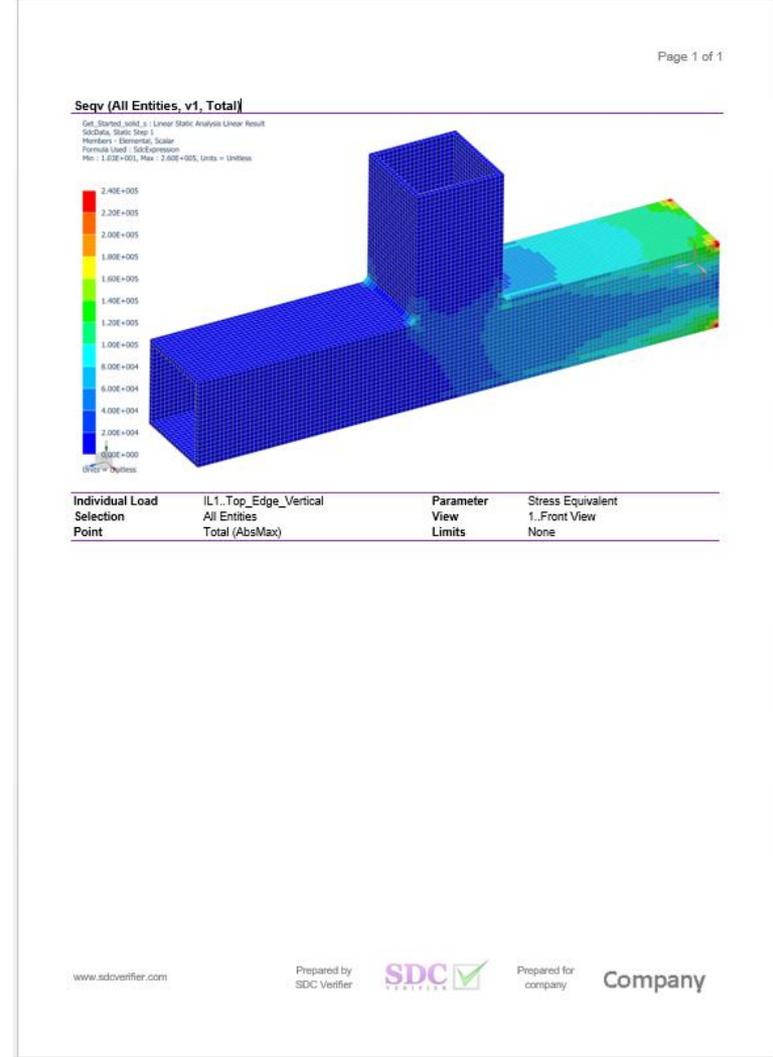
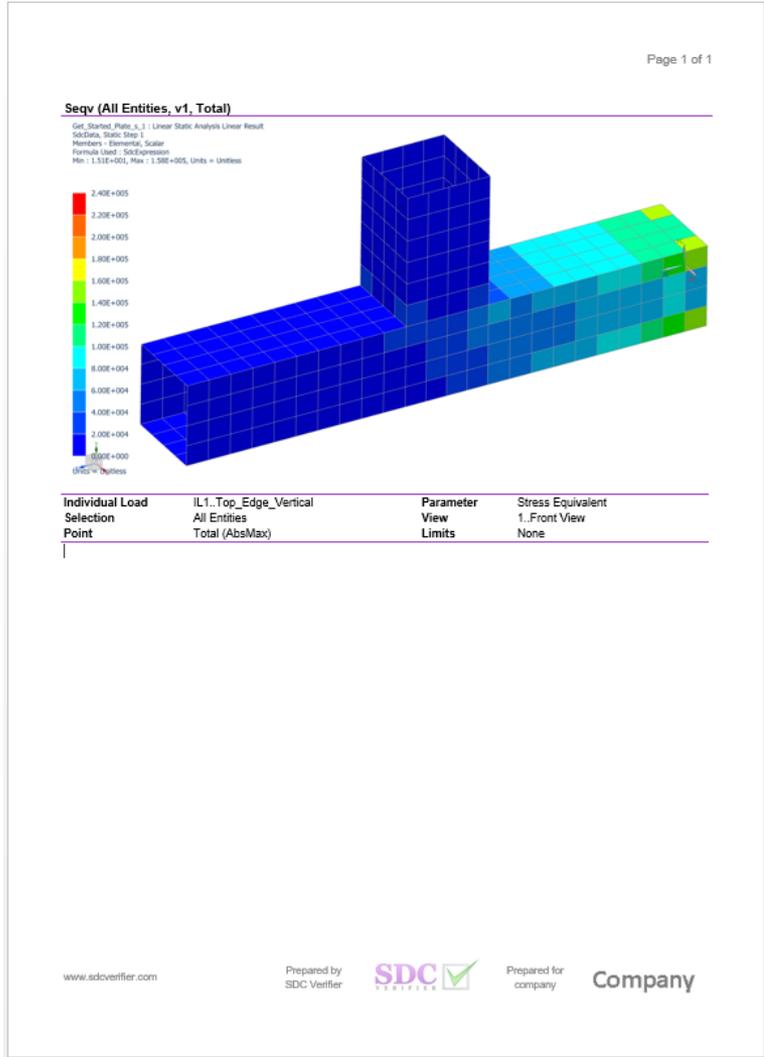
3. **Text**

5. **in this tutorial we have get**

4. **Conclusions**

Comments. Report on the volume model

Equivalent Stress plot is automatically displayed correct for plate and solid model.



Quick Support – remote access to your PC



1 Select *Help - Quick Support*

2 Contact helpdesk by Team Viewer

3 Tell your ID to SDC specialist

Quick Support allows to get remote access to your PC by SDC Verifier specialist to solve your problem. All you need is to send your ID to us:
by Skype: `sdcverifier_helpdesk`;
by phone: `+31 15 30-10-310`;
by email: support@sdcverifier.com

Quick Support tool is standalone program.
Team Viewer is not required to be installed.

