



Tutorial

Get Started Tutorial

ANSYS[®]

10 Jan 2020
version 5.3

SDC Verifier is a powerful Ansys 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:

- ▶ Launch SDC Verifier;
- ▶ Creating new project;
- ▶ Create Combinations and Envelope;
- ▶ Define Views;
- ▶ Model Setup report;
- ▶ Calculation report;

Launch SDC Verifier

1

Open in Ansys Workbench
GetStarted.wbpz

2

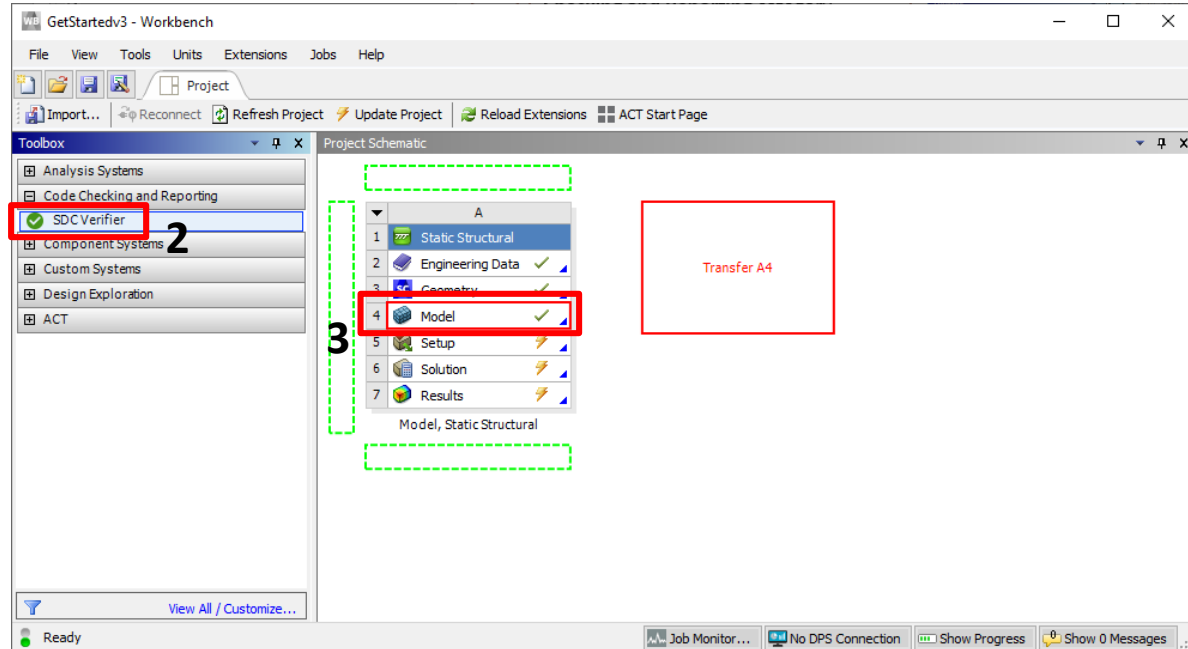
Drag **SDC Verifier** from Toolbox **Code Checking and Reporting** category

3

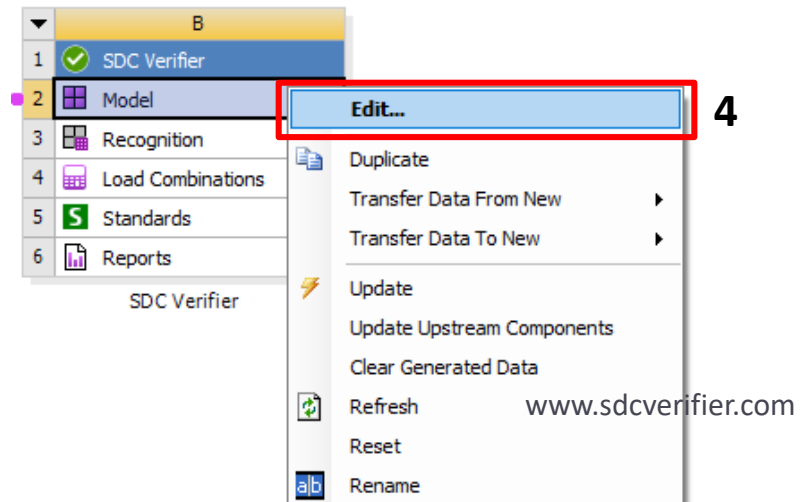
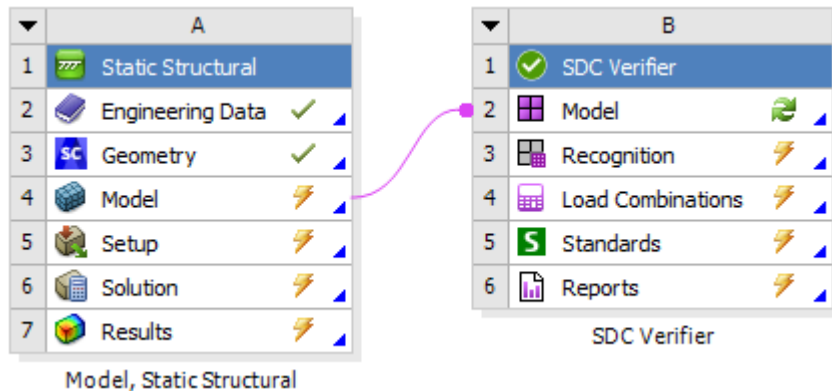
Drop on **Model** Task of **Static Structural** Task Group.

4

Double Click on **Model** or in context menu click **Edit**



After step 3 Project Schematic:



Create new project

1

Number: **p0001**

2

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

3

Press *Next*

Project Wizard

Project Details

General project information with an engineer and a customer details can be filled in and used on the first page of the report

Project Details


Jobs

Load Sets

Load Groups

Report Wizard

Select Model File

Model file Path 


Project Details


Number Name

Description

Unit System


Engineer details

Engineer 



Company 

E-mail


Phone


Address 

Website

Logo  


Customer details

Customer 



Company 

E-mail

Phone

Address 

Website

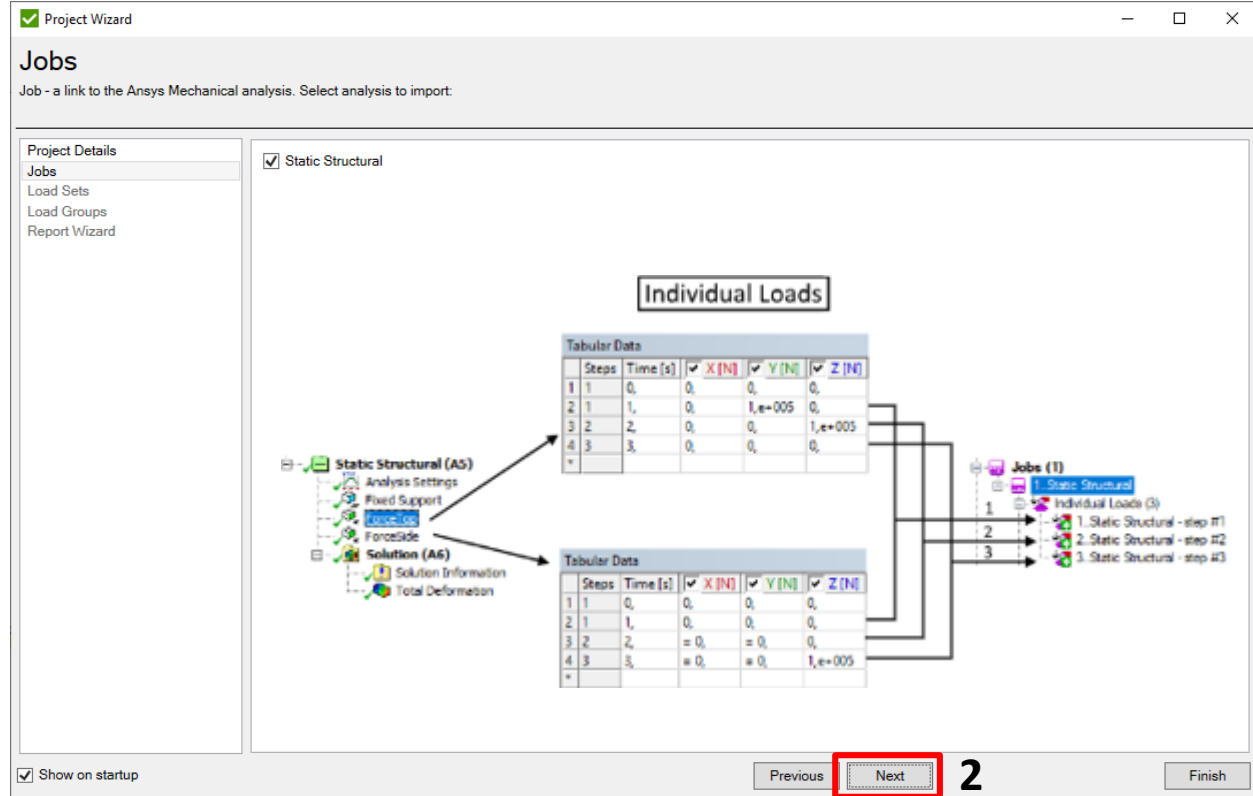
Logo  

☒ Show on startup

Create new project

1

Press *Next*



Job – the calculation set that contains loads and their combinations to be calculated with the defined calculation options. Job creation is based on analysis from Ansys Mechanical.

Individual load is a step result.

Load set is individual loads combination with factors.

Load Groups (envelope) – worst results among loads.

Job explanation

Individual Loads – each load will be created automatically from steps in Ansys Mechanical. In our case 3 Individual loads. Load sets are individual loads with factors.

The screenshot displays the Ansys Mechanical interface. On the left, the 'Static Structural (A5)' tree shows 'Analysis Settings', 'Fixed Support', 'Force', and 'Force 2'. The 'Details of "Analysis Settings"' panel is open, showing 'Step Controls' with the following values:

Property	Value
Number Of Steps	3.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled

On the right, the 'Jobs (1)' tree shows '1..Static Structural' with 'Individual Loads (3)' expanded, listing '1..Force - Step #1 (1)', '2..Force - Step #2 (2)', and '3..Force 2 - Step #3 (3)'. Below these are 'Load Sets (8)', 'Load Groups (1)', 'Fatigue Groups (0)', 'Tables (0)', and 'Plots (0)'.

The 'Edit Load Set' dialog is open at the bottom. It shows 'ID' 3, 'Title' 'All_combinations 3', and 'Safety Factor' 1. The 'Description' field is empty. The 'Loads' table is as follows:

Loads	Factor 1
1..Force - Step #1 (1)	1
2..Force - Step #2 (2)	-1
3..Force 2 - Step #3 (3)	1.1

At the bottom of the dialog, there is a 'Factor' field set to 1 and a 'Set To Selected' button. The 'Apply' button is at the bottom right.

Create load combinations

- 1 Open load_sets excel file from tutorial folder and copy data to clipboard.
- 2 Press **From Clipboard** to add load sets copied from excel
- 3 Press *Next*

Project Wizard

Load Sets

Load set is a summation of individual loads (different analysis steps) multiplied on factors;

Project Details

Jobs

Load Sets

Load Groups

Report Wizard

	Safety Factor	IL1..Force - Step #1 (1)	IL2..Force - Step #2 (2)	IL3..Force 2 - Step #3 (3)
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

Add load Sets

Count 1

From Clipboard

Factor 1

Set

Set Diagonally

Clean Selected

Clipboard

Copy

Paste

Selected Rows

Copy

Remove

☒ Show on startup

Previous **Next** Finish

	A	B	C	D	E
1		Safety Factor	IL1..Static Structural - step #1	IL2..Static Structural - step #2	IL3..Static Structural - step #3
2	All_combinations 1	1	1	1	1.1
3	All_combinations 2	1	-1	1	1.1
4	All_combinations 3	1	1	-1	1.1
5	All_combinations 4	1	-1	-1	1.1
6	All_combinations 5	1	1	1	-1.1
7	All_combinations 6	1	-1	1	-1.1
8	All_combinations 7	1	1	-1	-1.1
9	All_combinations 8	1	-1	-1	-1.1

Factor – set factor to selected cells

Clipboard – copy/paste data from/to clipboard

Selected rows – copy (default title and factors) and remove selected load sets in the list

Create Load Group (Envelope)

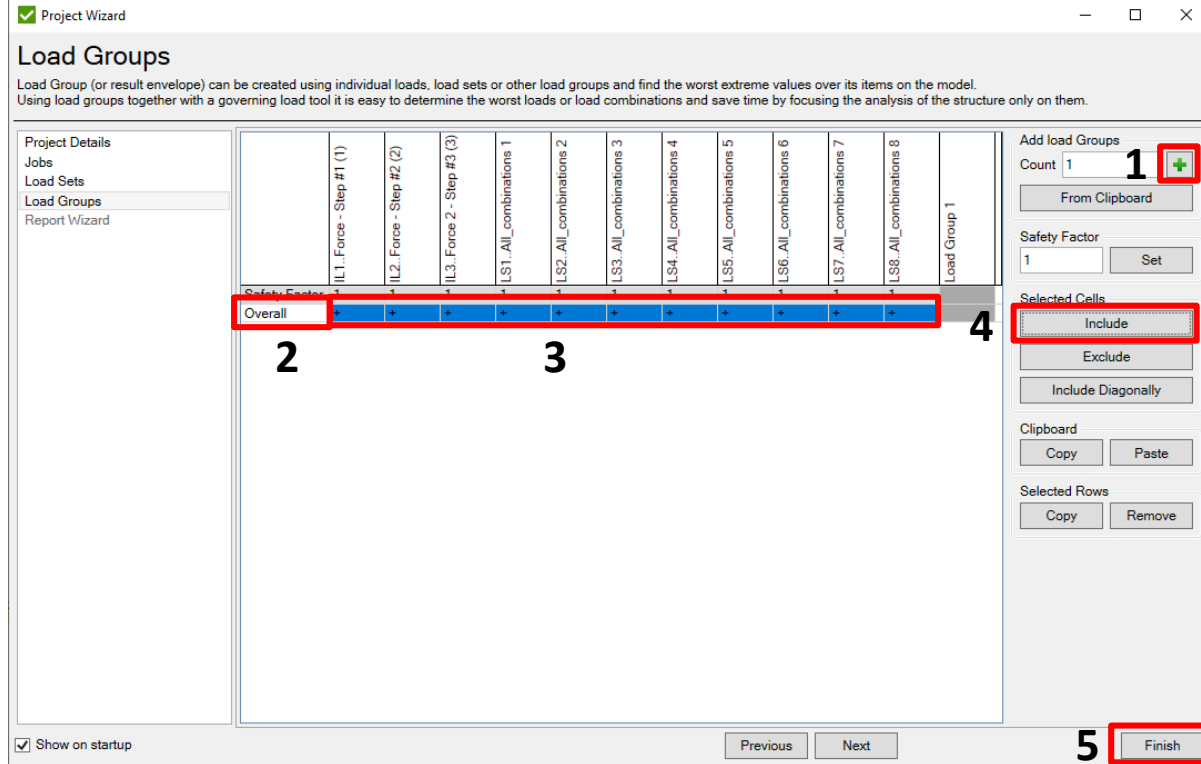
1 Press 

2 Title (first column): **Overall**

3 Select all cells in the second row except title column

4 Press *Include*

5 Press *Finish*



Project Wizard

Load Groups

Load Group (or result envelope) can be created using individual loads, load sets or other load groups and find the worst extreme values over its items on the model.
Using load groups together with a governing load tool it is easy to determine the worst loads or load combinations and save time by focusing the analysis of the structure only on them.

Project Details
Jobs
Load Sets
Load Groups
Report Wizard

	IL1. Force - Step #1 (1)	IL2. Force - Step #2 (2)	IL3. Force 2 - Step #3 (3)	LS1. All combinations 1	LS2. All combinations 2	LS3. All combinations 3	LS4. All combinations 4	LS5. All combinations 5	LS6. All combinations 6	LS7. All combinations 7	LS8. All combinations 8	Load Group 1
Safety Factor												
Overall												


2 3 4

5

✓ Show on startup

Previous Next

Finish

Add load Groups
Count 1 
From Clipboard

Safety Factor
1 Set

Selected Cells
Include
Exclude
Include Diagonally

Clipboard
Copy Paste

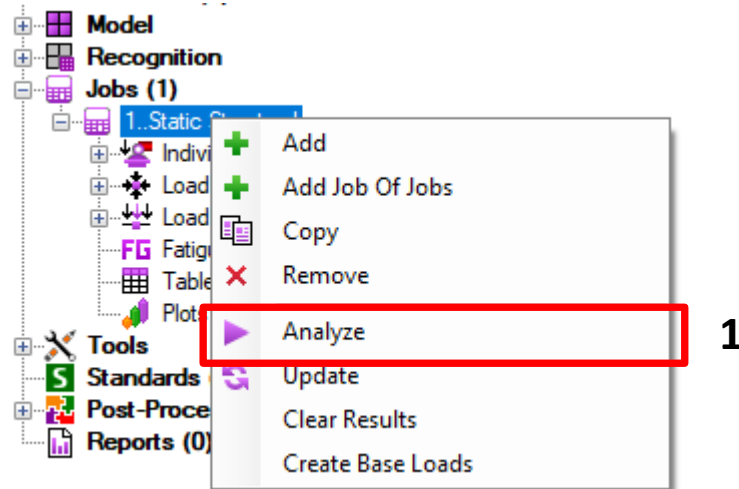
Selected Rows
Copy Remove

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

Analyze Job

1

Execute Analyze from Job “**Static Analysis**” context menu



Create 2 general Views for plots

1 Locate Model in Ansys Mechanical as shown on pic. Front View.

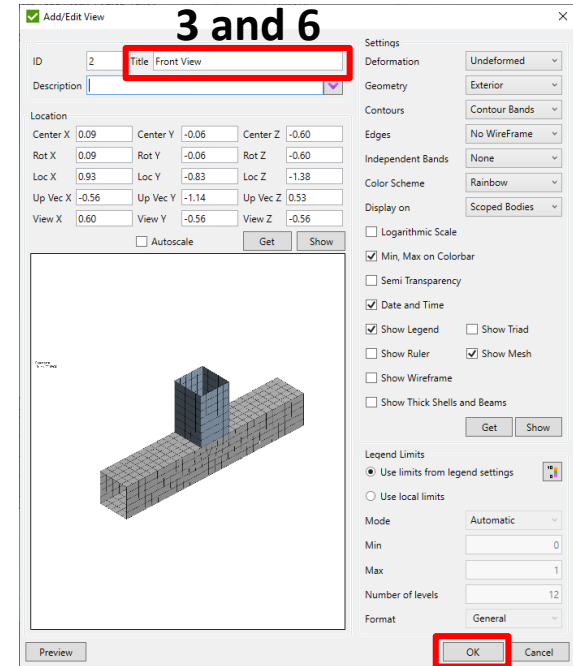
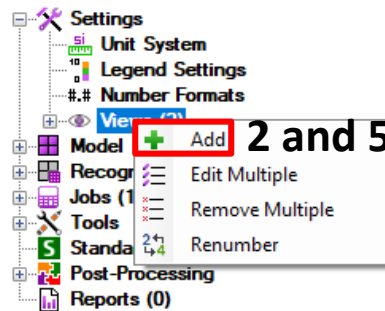
2 From View context menu execute – Add

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

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

5 From View context menu execute – Add

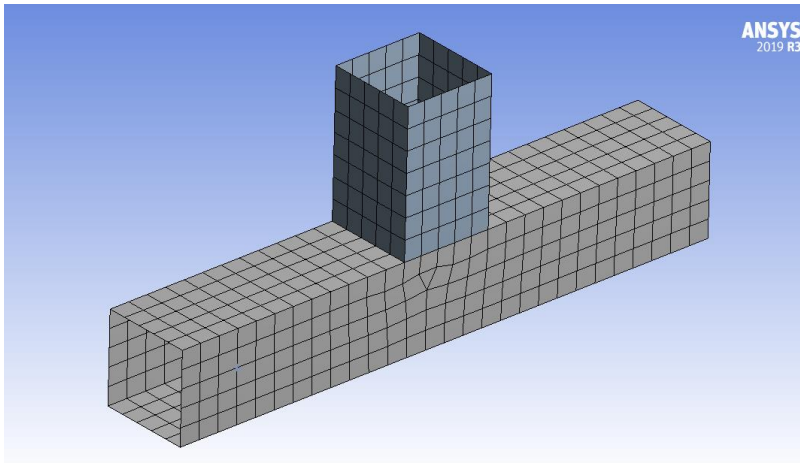
6 Title: **Back View**. Press **OK**.



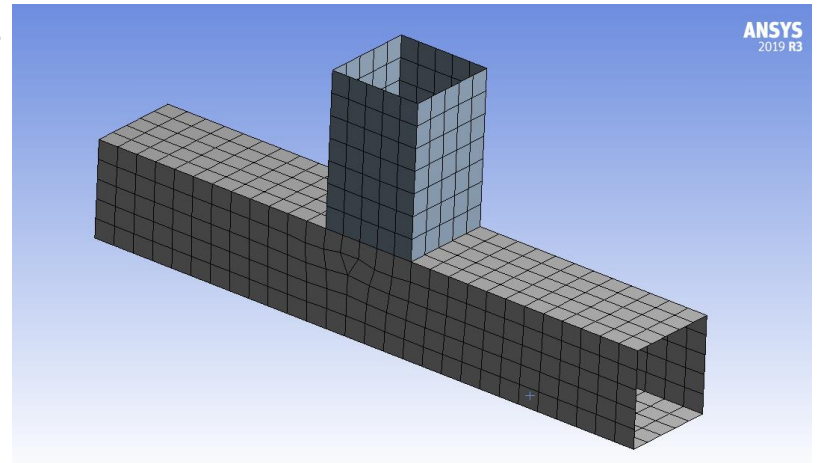
Front View

Back View

1



4



Create 2 detailed Views

1 Locate Model in Ansys Mechanical as shown on pic. Front Detail.

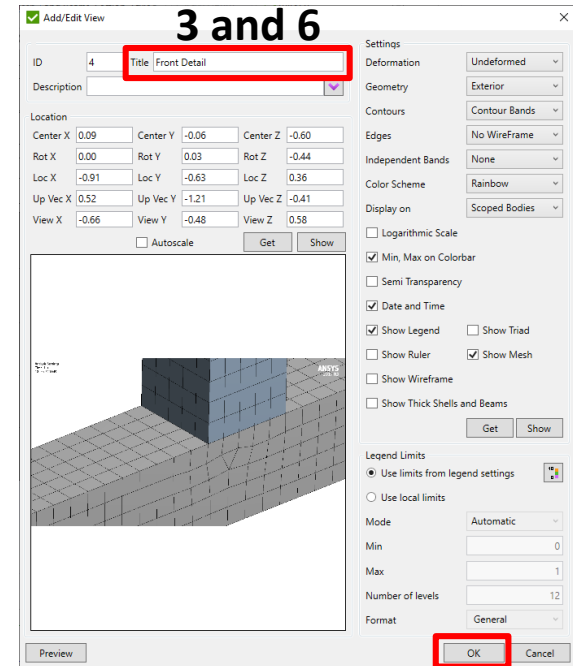
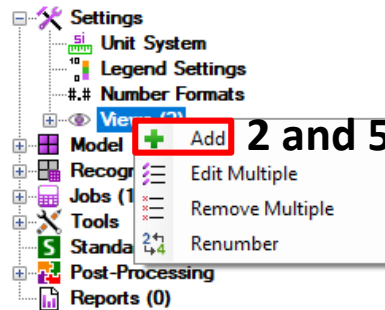
2 From View context menu execute – Add

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

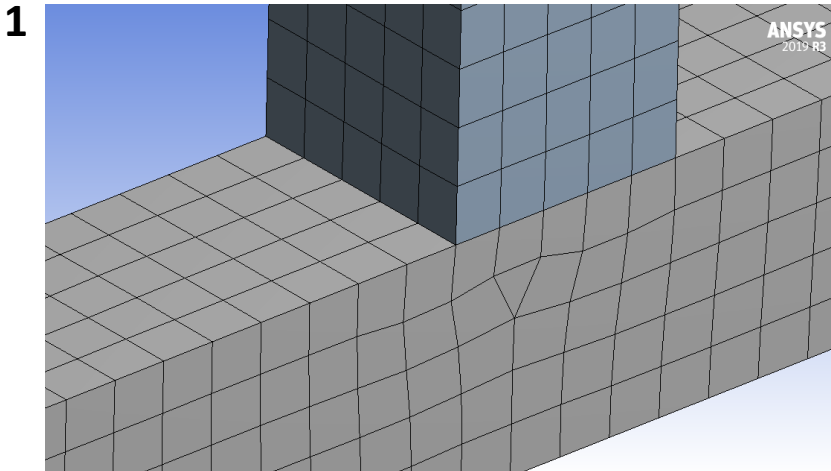
4 Locate the Model in Ansys Mechanical as shown on pic. Back Detail.

5 From View context menu execute – Add

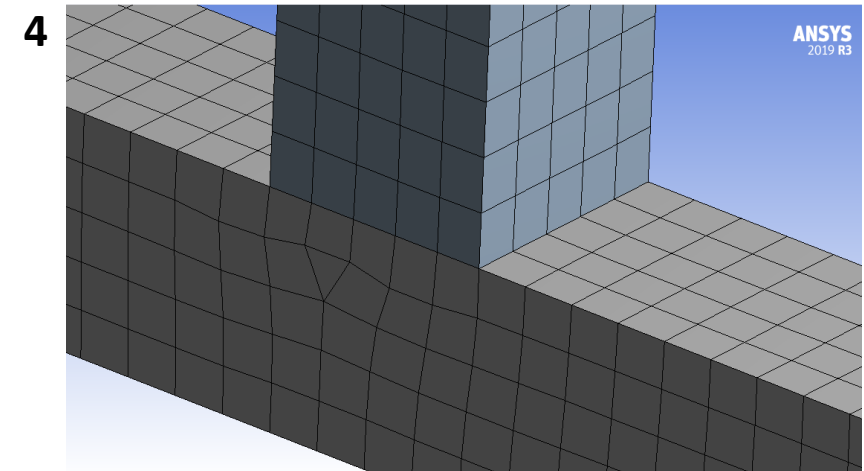
6 Title: **Back Detail**. Press *Get and OK*.



Front Detail





Back Detail



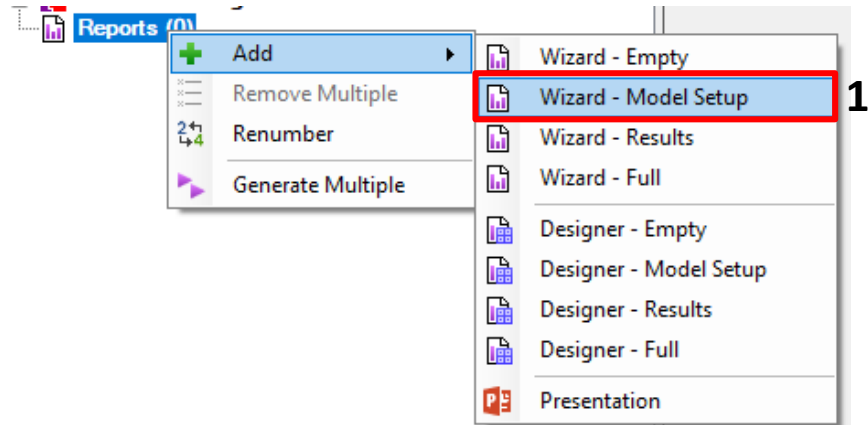
Report Wizard – Model Setup report

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

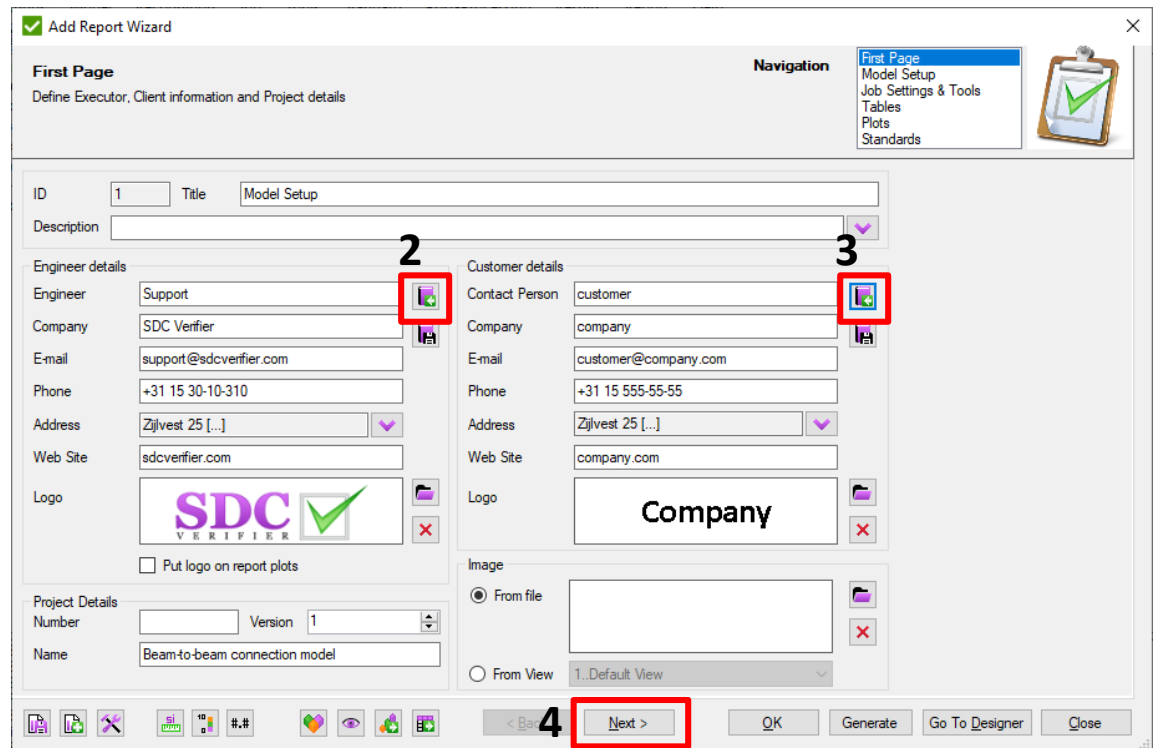
2 Press  and select Support Engineer from the library

3 Press  and select Customer from the library

4 Press Next



An engineer and a customer information is used on the report's first page and in footer (company name and logo).



The screenshot shows the 'Add Report Wizard' dialog box. The 'First Page' tab is active, showing fields for Engineer details, Customer details, and Project Details. The 'Engineer' field is set to 'Support' and the 'Customer' field is set to 'customer'. The 'Project Details' field is set to 'Beam-to-beam connection model'. The 'Next >' button is highlighted with a red box and the number '4'.

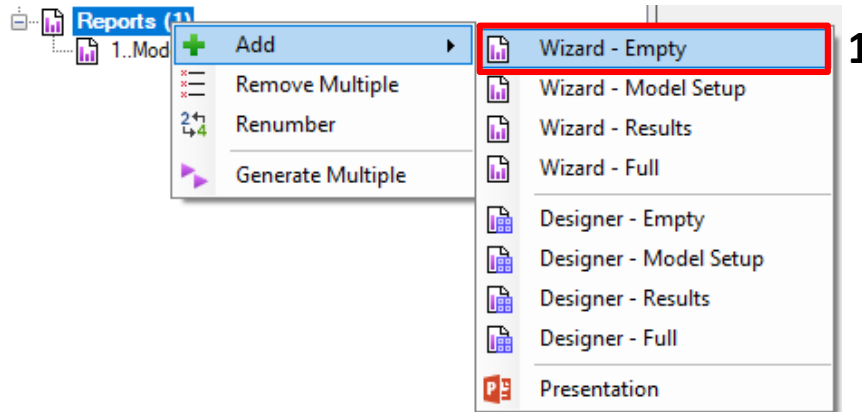
Press *Generate*.



Create calculation report

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

2 Press *Next* twice



The screenshot shows the 'Add Report Wizard' dialog box, 'First Page'. The title bar says 'Add Report Wizard'. The 'First Page' section has the subtitle 'Define Executor, Client information and Project details'. The 'Navigation' pane on the right shows 'First Page' as the active tab, with other tabs like 'Model Setup', 'Job Settings & Tools', 'Tables', 'Plots', and 'Standards'. The main form contains fields for 'ID' (1), 'Title' (Model Setup), and 'Description'. Below these are sections for 'Engineer details' and 'Customer details'. The 'Engineer details' section includes fields for 'Engineer' (Support), 'Company' (SDC Verifier), 'Email' (support@sdcsverifier.com), 'Phone' (+31 15 30-10-310), 'Address' (Zijvest 25 [...]), 'Web Site' (sdcsverifier.com), and a 'Logo' field with the SDC Verifier logo. The 'Customer details' section includes fields for 'Contact Person' (customer), 'Company' (company), 'Email' (customer@company.com), 'Phone' (+31 15 555-55-55), 'Address' (Zijvest 25 [...]), 'Web Site' (company.com), and a 'Logo' field with the text 'Company'. There is a checkbox 'Put logo on report plots'. The 'Project Details' section includes 'Number' and 'Version' (1) fields, and a 'Name' field (Beam-to-beam connection model). At the bottom, there are 'From file' and 'From View' radio buttons, with 'From file' selected. The 'Image' field is empty. The 'Next >' button is highlighted with a red rectangle. A large number '2' is placed to the left of the 'Next >' button. Other buttons at the bottom include '< Back', 'OK', 'Generate', 'Go To Designer', and 'Close'.

Predefined Job Tables

1

Jobs: **ON**

2

Include Sum of Forces: **ON**

3

Press *Next*

Add Report Wizard

Job Settings and Tools
Set extra job settings and select extra tables to be included

Navigation

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

1 ☒ Jobs 1..Static Structural

2 ☒ Include Sum of Forces

Job Settings

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

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

Tools

- ☐ Mass and COG Table (0/0)
- ☐ Moment Shear Force (0/0)

3 Next >

OK Generate Go To Designer Close

Add tables

1 Click on *Tables* in the Navigation list.

2 Table Type: Load Groups.

3 Press  to add tables.

4 *Table Type* – **Extreme**;

5 *Categories*: **Stress, Displacement**;

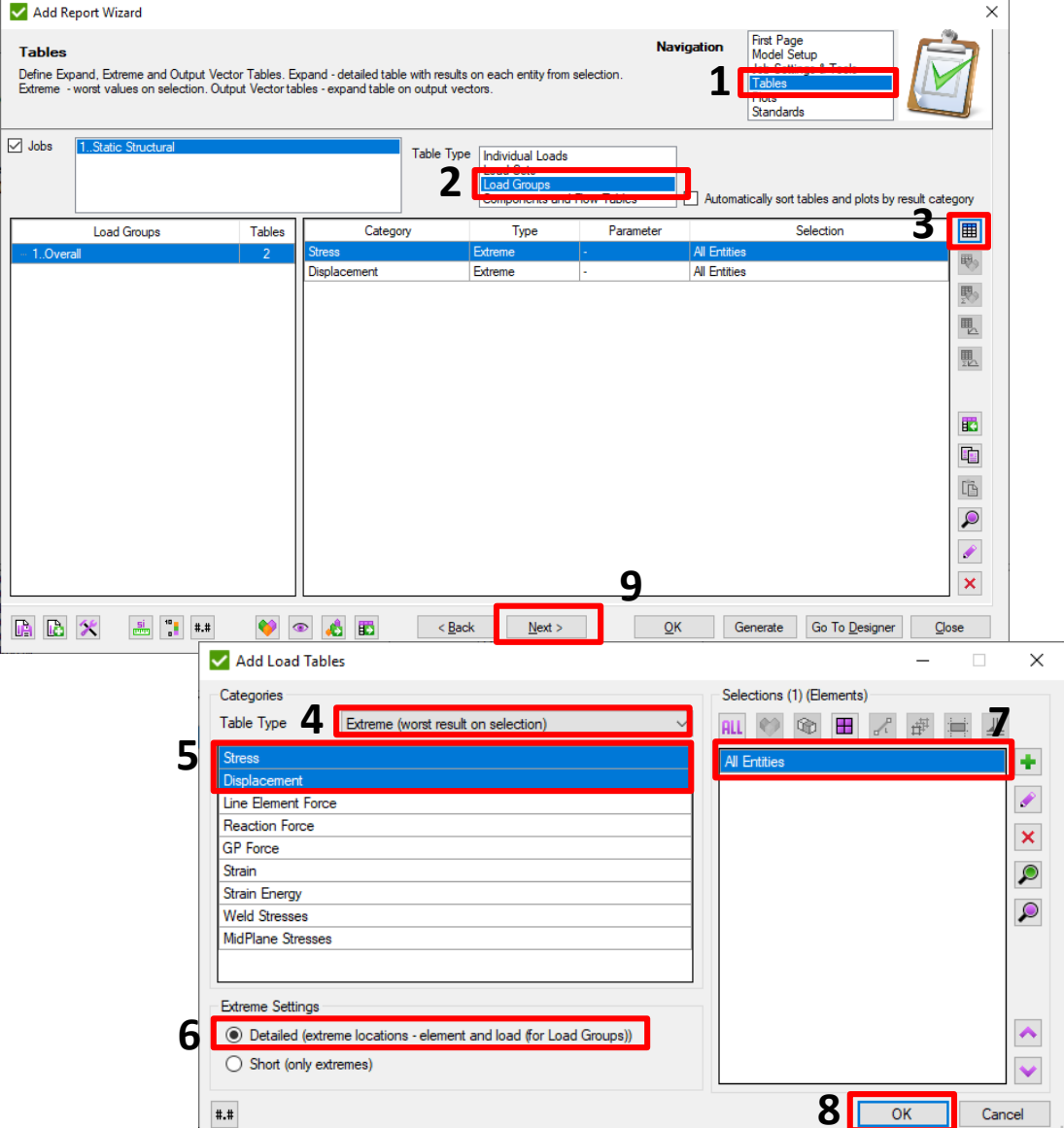
6 *Extreme settings*: **Detailed**;

7 *Selection*: **All Entities**;

8 Press **OK**.

9 Press *Next*.

In **Advanced Table Type** it is possible to create tables on multiple loads or selections (e.g. extreme table over all loads)



Add Report Wizard

Navigation

- 1 First Page
- 2 Model Setup
- 3 **Tables**
- 4 Results and Tables
- 5 Standards

Tables

Define Expand, Extreme and Output Vector Tables. Expand - detailed table with results on each entity from selection. Extreme - worst values on selection. Output Vector tables - expand table on output vectors.


☒ Jobs: 1. Static Structural

Table Type: Individual Loads

2 **Load Groups**

☐ Automatically sort tables and plots by result category

Load Groups	Tables	Category	Type	Parameter	Selection
1. Overall	2	Stress	Extreme	-	All Entities
		Displacement	Extreme	-	All Entities

3 

9

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

Add Load Tables

Categories

Table Type: 4 **Extreme (worst result on selection)**

5

- Stress**
- Displacement**
- Line Element Force
- Reaction Force
- GP Force
- Strain
- Strain Energy
- Weld Stresses
- MidPlane Stresses

6

Extreme Settings

- ☒ Detailed (extreme locations - element and load (for Load Groups))
- ☐ Short (only extremes)

Selections (1) (Elements)

7


- All Entities**

8 **OK** Cancel

Add displacement plots

1 Click on *Plots* in the Navigation list.

2 Select All Loads from the list.

3 Press  to add contour plot.

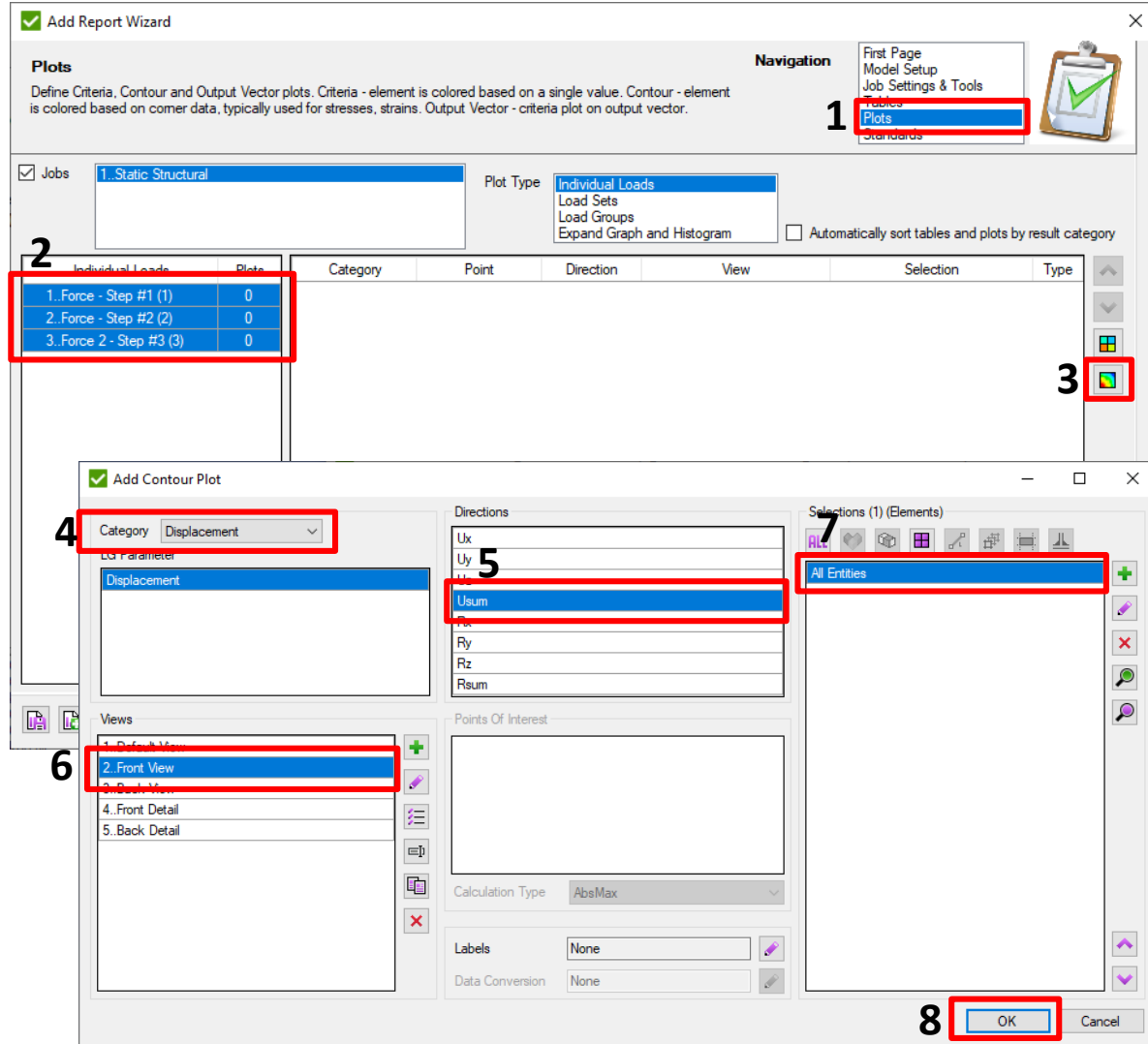
4 *Category: Displacement*

5 *Direction: Usum*

6 *Views: Front Detail*

7 *Selection: All Entities*

8 Press *OK*.



Add Report Wizard

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
1 First Page
Model Setup
Job Settings & Tools
Plots
Standards

☒ Jobs: 1.Static Structural

Plot Type: Individual Loads
Load Sets
Load Groups
Expand Graph and Histogram

☐ Automatically sort tables and plots by result category

Individual Loads	Plots	Category	Point	Direction	View	Selection	Type
1..Force - Step #1 (1)	0						
2..Force - Step #2 (2)	0						
3..Force 2 - Step #3 (3)	0						

Add Contour Plot

Category: Displacement

LD Parameter: Displacement

Directions: Ux, Uy, Uz, Rx, Ry, Rz, Rsum

5 Usum

Views: 1..Front View, 2..Front View, 3..Back View, 4..Front Detail, 5..Back Detail

6 2..Front View

Points Of Interest

Calculation Type: AbsMax

Labels: None

Data Conversion: None


Selections (1) (Elements): All Entities

7 All Entities

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 Entities**

7 Press *OK*.


Add Report Wizard

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. Static Structural Plot Type: Individual Loads
Load Sets
Load Groups
Expand Graph and Histogram

☐ Automatically sort tables and plots by result category

Individual Loads	Plots	Category	Point	Direction	View	Selection	Type
1..Force - Step #1 (1)	1	Displacement		Usum	2..Front View	All Entities	
2..Force - Step #2 (2)	1						
3..Force 2 - Step #3 (3)	1						

Add Criteria Plot

Category: Stress

ES Parameter: Stress

Directions:
X
Y
Z
XY
YZ
ZY
Equivalent

Points Of Interest:
End I / Top
End J / Bottom
Total

Calculation Type: AbsMax

Labels: None
Limits: None


Views:
1. Default View
2. Front View
3. Back View
4. Front Detail
5. Back Detail

Selections (1) (Elements)
All Entities

OK Cancel

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**

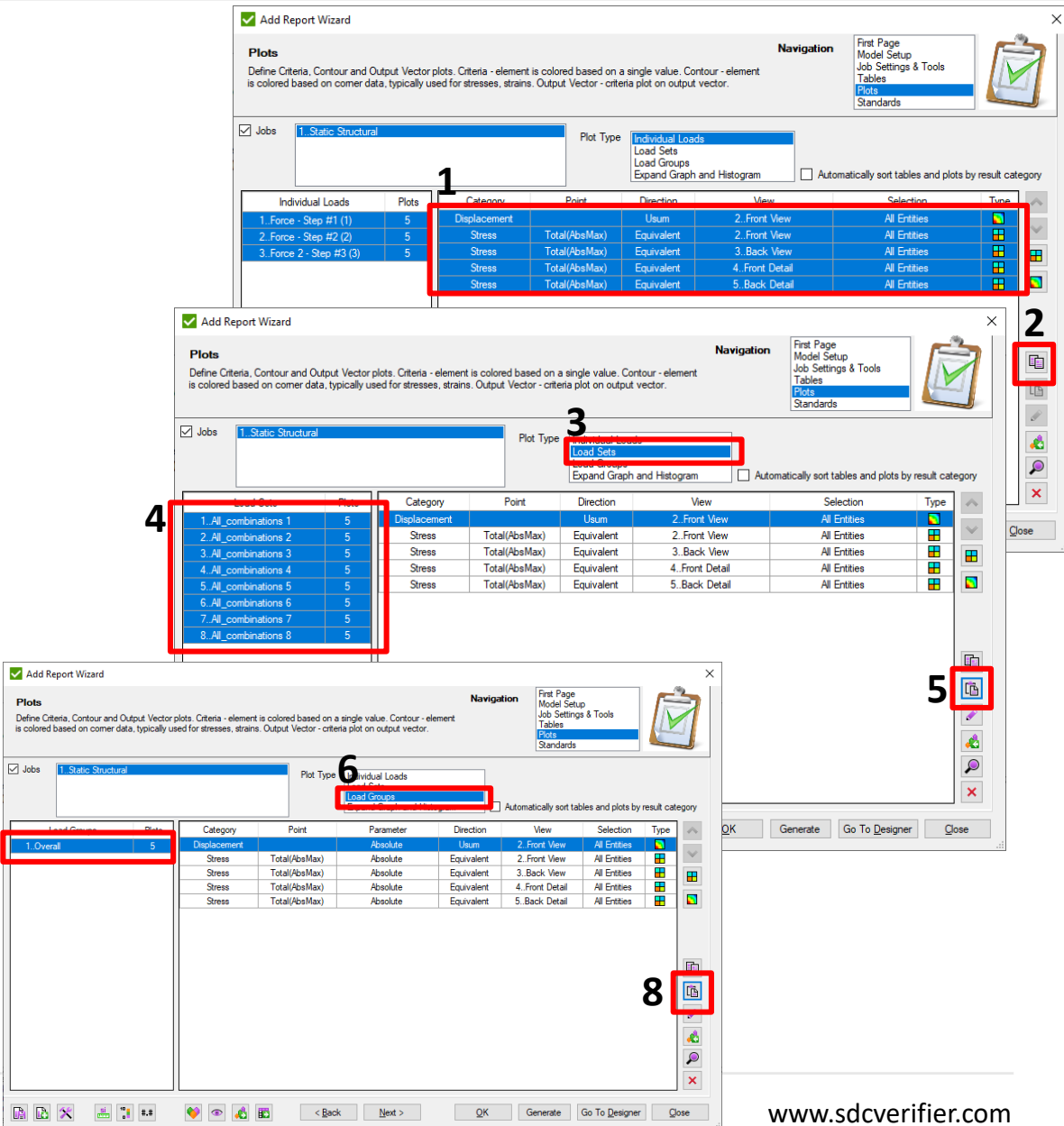
4 Select all loads sets.

5 Press  to paste.

6 Plot Type: **Load Groups**

7 Select all loads groups.

8 Press  to paste.



The screenshot shows the 'Add Report Wizard' dialog in SDC Verifier. The 'Plots' section is active, and the 'Jobs' list shows '1. Static Structural'. The 'Plot Type' dropdown is set to 'Individual Loads'. The 'Plots' table lists the following plots:

Plot	Category	Point	Direction	View	Selection	Type
1. Force - Step #1 (1)	Displacement	Usun	2. Front View	All Entities		
2. Force - Step #2 (2)	Stress	Total(AbsMax)	Equivalent	2. Front View	All Entities	
3. Force - Step #3 (3)	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	

The 'Load Sets' table is also shown, listing the following load sets:

Load Set	Category	Point	Direction	View	Selection	Type
1. All combinations 1	Displacement	Usun	2. Front View	All Entities		
2. All combinations 2	Stress	Total(AbsMax)	Equivalent	2. Front View	All Entities	
3. All combinations 3	Stress	Total(AbsMax)	Equivalent	3. Back View	All Entities	
4. All combinations 4	Stress	Total(AbsMax)	Equivalent	4. Front Detail	All Entities	
5. All combinations 5	Stress	Total(AbsMax)	Equivalent	5. Back Detail	All Entities	
6. All combinations 6						
7. All combinations 7						
8. All combinations 8						

The 'Load Groups' table is also shown, listing the following load groups:

Load Group	Category	Point	Parameter	Direction	View	Selection	Type
1. Overall	Displacement	Absolute	Usun	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	

The 'Add Report Wizard' dialog has a 'Navigation' pane on the right with the following items: First Page, Model Setup, Job Settings & Tools, Tables, Plots, and Standards. The 'Plots' item is selected. The 'Add Report Wizard' dialog has a 'Jobs' list with '1. Static Structural' selected. The 'Plot Type' dropdown is set to 'Individual Loads'. The 'Plots' table is shown. The 'Load Sets' table is shown. The 'Load Groups' table is shown. The 'Add Report Wizard' dialog has a 'Generate' button and a 'Go To Designer' button.



2



Quick Support – remote access to your PC

1 Select *Help - Quick Support*

2 Contact helpdesk by phone or skype

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](https://www.skype.com/partner/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.

