Tutorial

Get Started

Updated on: 11.04.2023

Tested with: SDC Verifier 2023R1

Ansys 2023R1

Content

SDC Verifier is a powerful extension to **Ansys Mechanical** with an advanced calculation core for checking structures according to different standards and report generation.
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:

- Launch SDC Verifier;
- Create new project;
- Create Load Sets and Load Groups;
- Define Views;
- Create Plots and Tables;
- Adjust number format;

Launch SDC Verifier

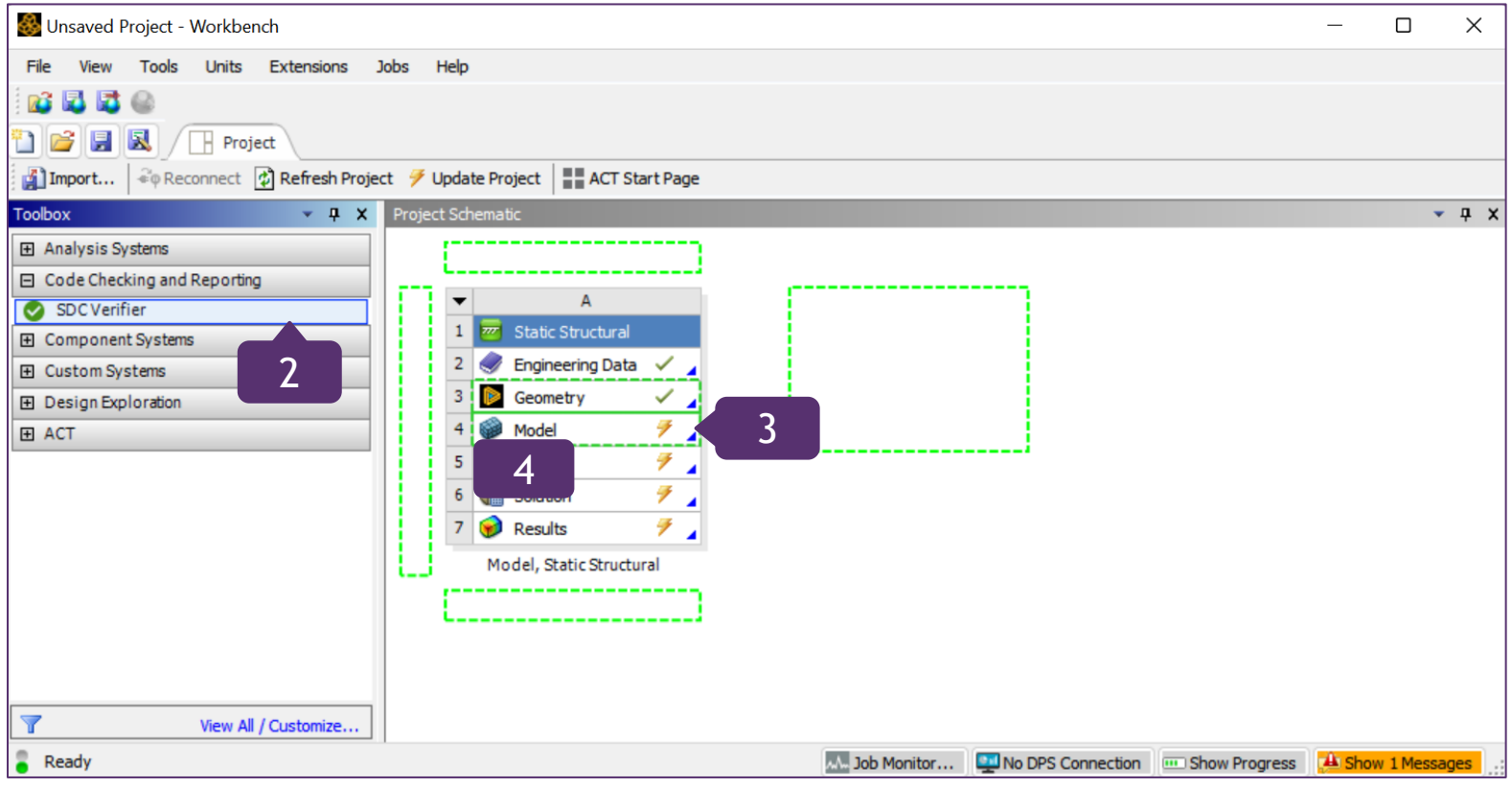
1. Open **GetStarted.wbpz** with **Ansys Workbench**

NOTE: Please make sure that correct .wbex extension is installed. See instructions HERE

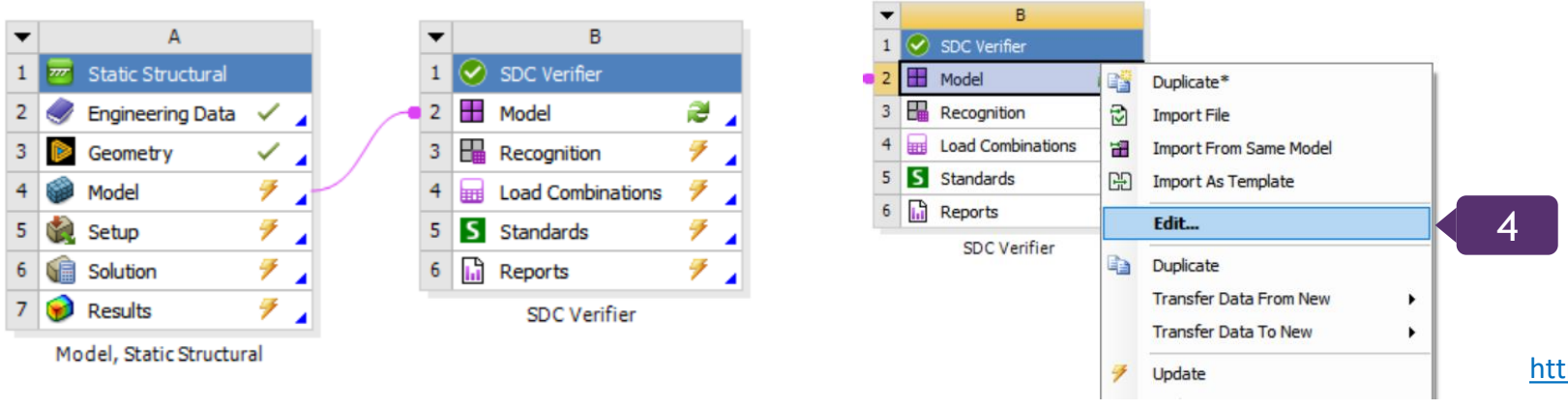
2. Drag **SDC Verifier** from Toolbox **Code Checking and Reporting** category

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

4. Double Click on
or click **Edit** in context menu



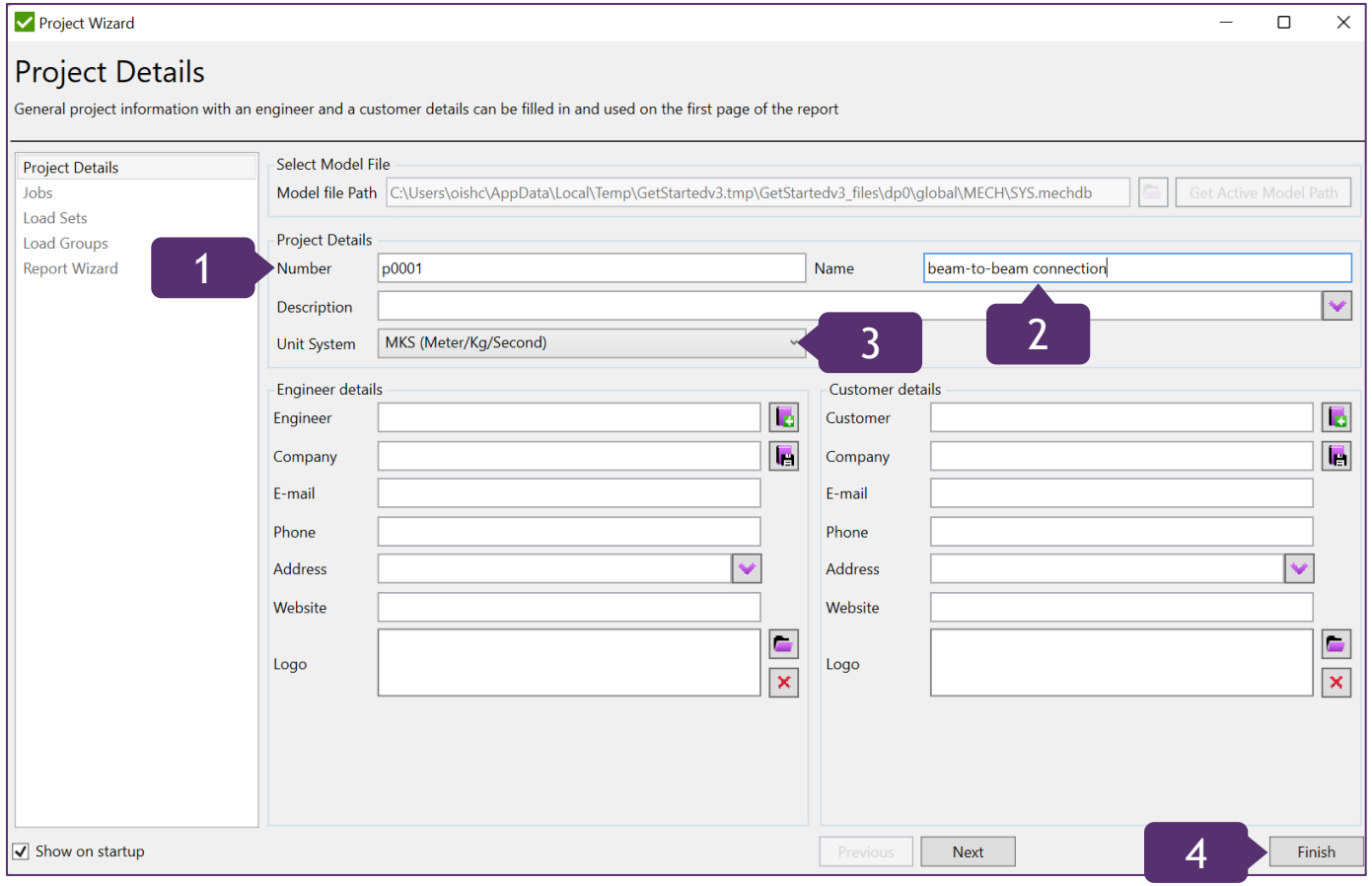
Project scheme after step 3:



https://sdcverifier.com

Create new project

1. Input project number: **p0001**
2. Type in the project *Name:* **beam-to-beam connection**
3. Make sure that **Unit System** corresponds with the one in your mode. Use MKS
4. Press Finish

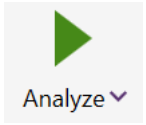


Project Wizard window also allows to set not only the Names and Details of the upcoming project but also define Jobs, Load Sets, Load Groups and Reports in the dialogue manner. These items will be presented in the following slides. Project Wizard can be skipped for the future projects.

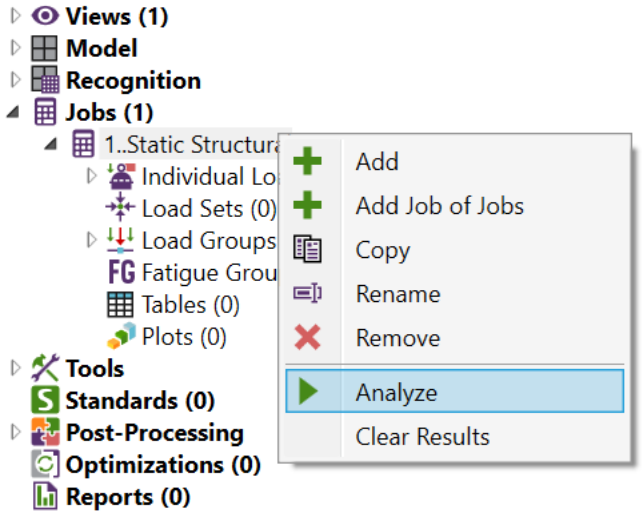
This window also allows to enter Engineer and Customer details that will be displayed in the reports. Can be edited later

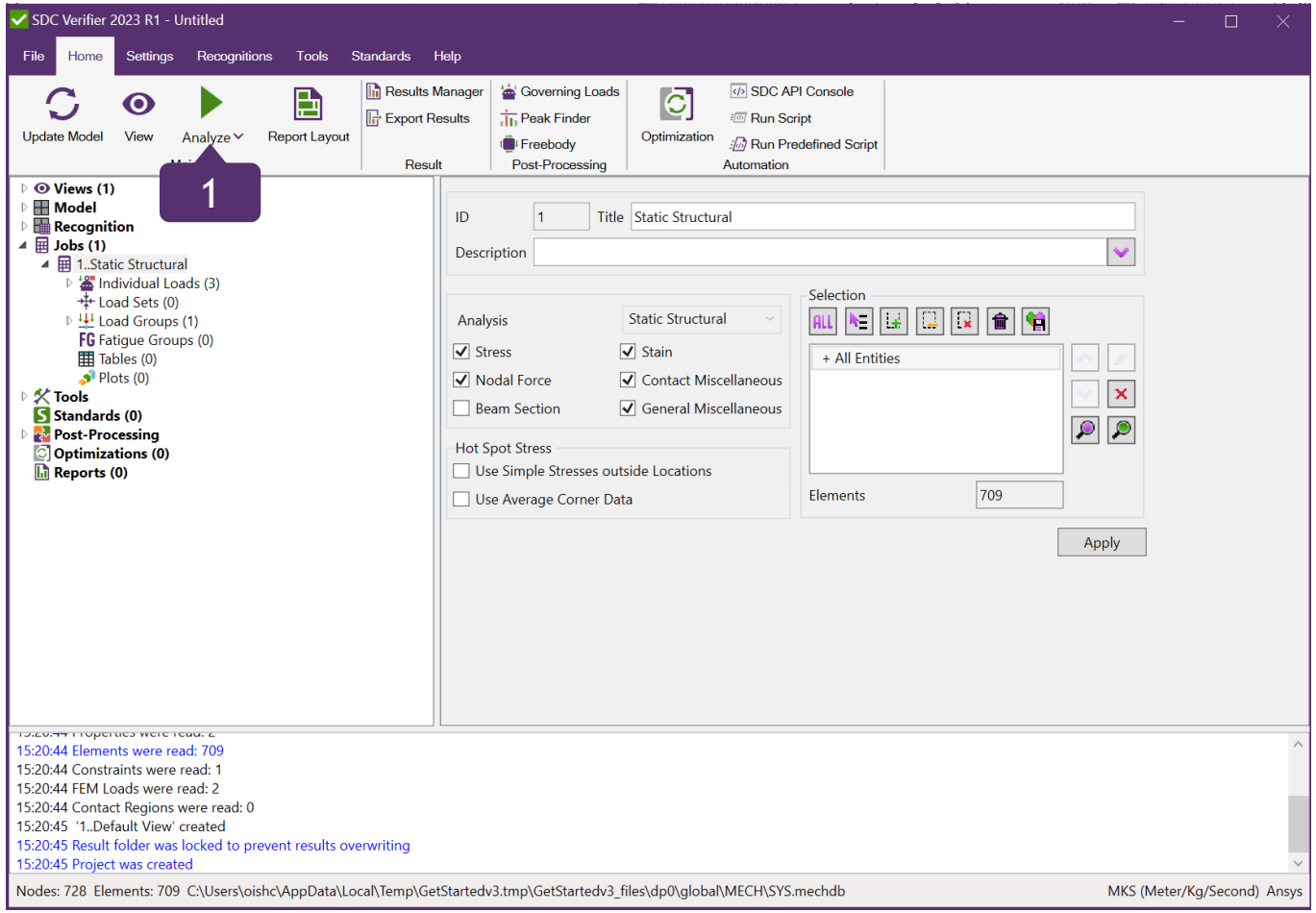
Analyze the Job

1 Click **Analyze** on the **Home** tab



Alternatively, execute right click on the *Active Job* in the *Tree* and press **Analyze**

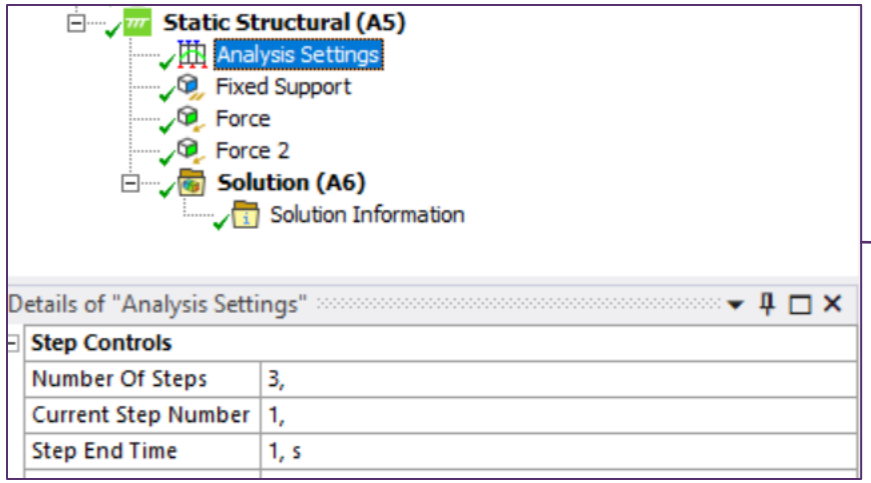


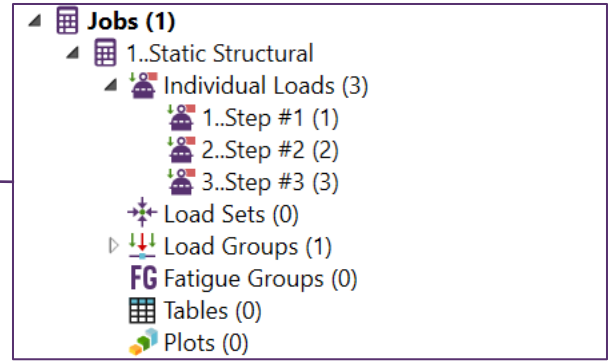


Job Explanation

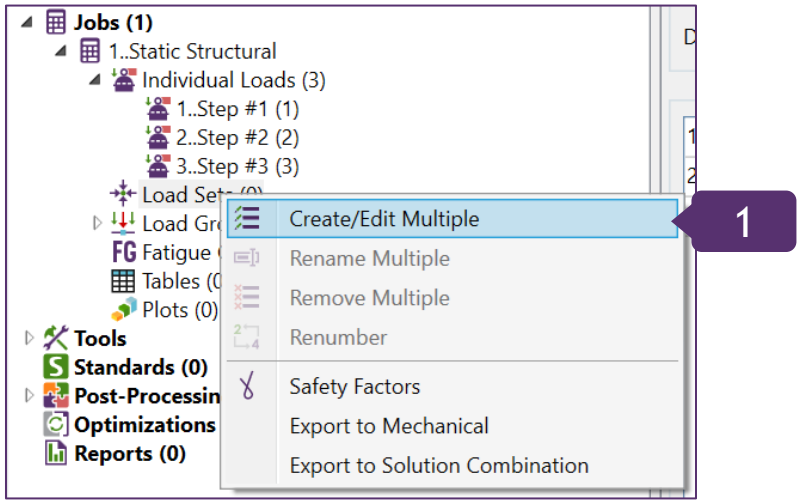
Individual Loads - single load results, each load will be created automatically from steps in Ansys Mechanical.
In our case we have 3 Individual loads.

Load sets - linear combinations of individual loads multiplied with factors.





1 Right click on **Load Sets > Create/Edit Multiple**



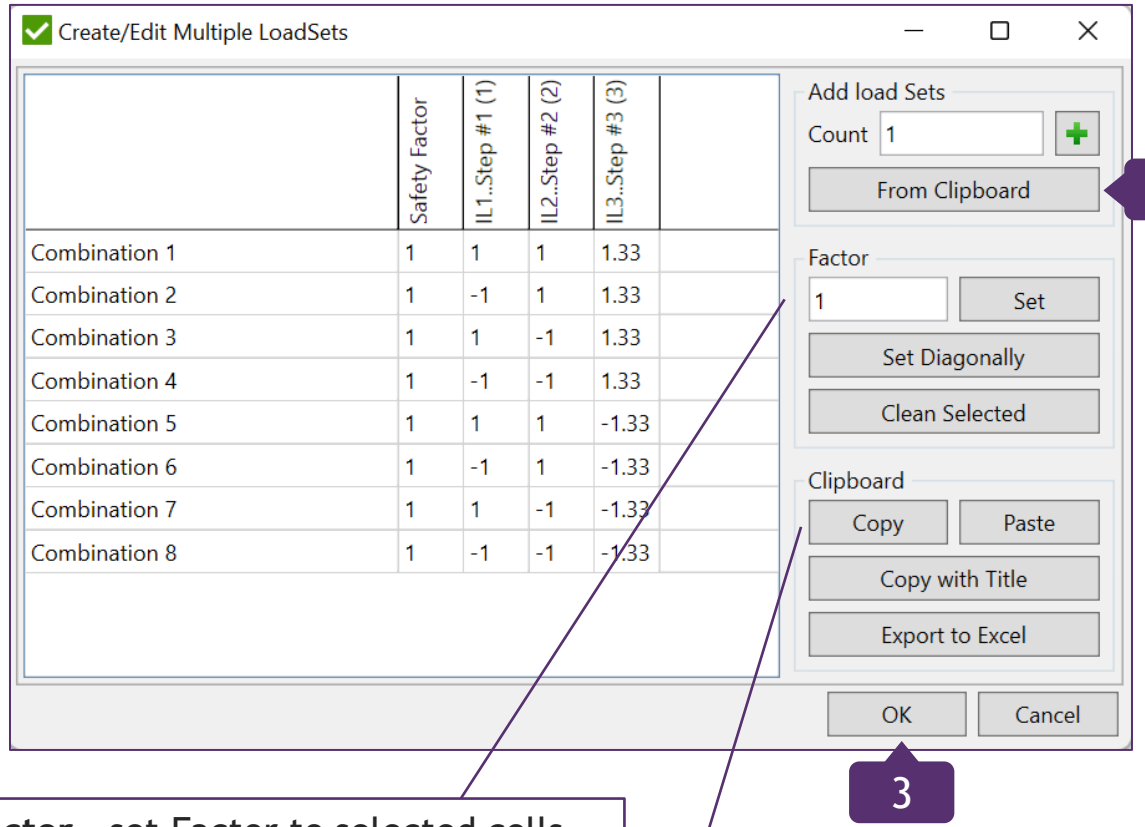
Create Load Sets

1. Open load sets Excel file from tutorial folder and copy data to clipboard
2. Press **From Clipboard** button to add load sets copied from Excel
3. Press **OK**

	A	B	C	D	E
1		Safety Factor	IL1..Static Structural - step #1	IL2..Static Structural - step #2	IL3..Static Structural - step #3
2	Combination 1	1	1	1	1,33
3	Combination 2	1	-1	1	1,33
4	Combination 3	1	1	-1	1,33
5	Combination 4	1	-1	-1	1,33
6	Combination 5	1	1	1	-1,33
7	Combination 6	1	-1	1	-1,33
8	Combination 7	1	1	-1	-1,33
9	Combination 8	1	-1	-1	-1,33

1

This window allows to Create and Edit Load Sets



2

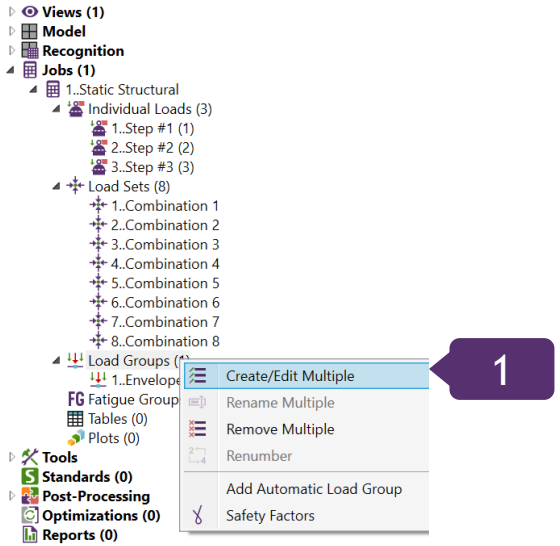
3

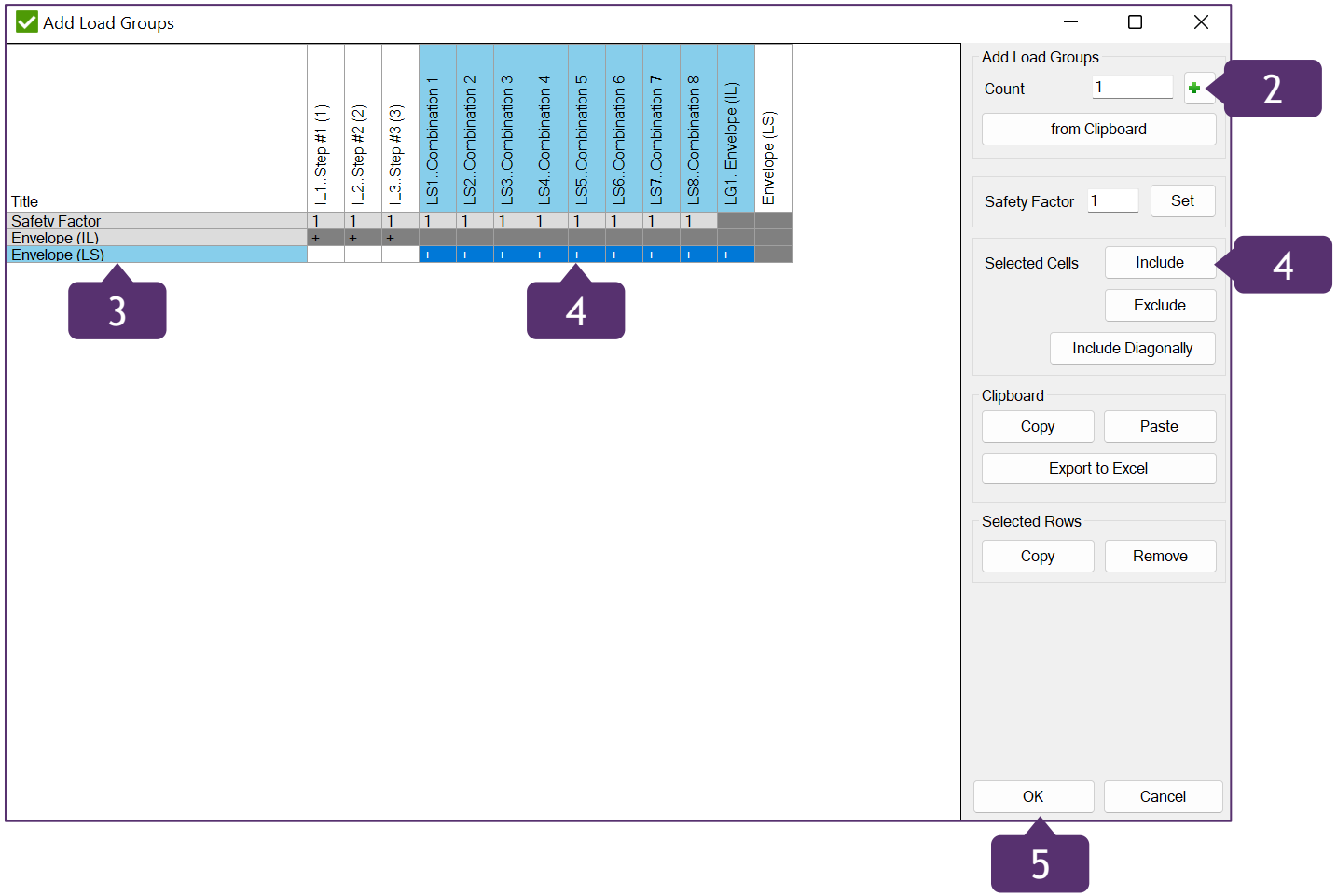
Factor - set Factor to selected cells

Clipboard - copy/paste data from/to clipboard or export to Excel file

Create Load Group

1. Right click on **Load Groups > Create/Edit Multiple**
2. Press Add 
3. Envelope (IL) Load Group is already existing, Title the new LG: **Envelope (LG)**
4. Select all cells of the LS in this new Load Group and click **Include**
5. Press **OK**



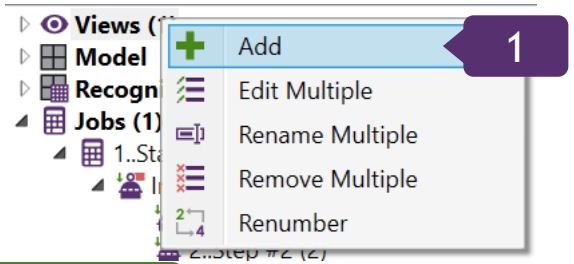


Load Group is an envelope for Individual Loads, Load Sets, or other Load Groups. It allows to determine minimum, maximum and absolute values of stresses, displacements, forces, etc.

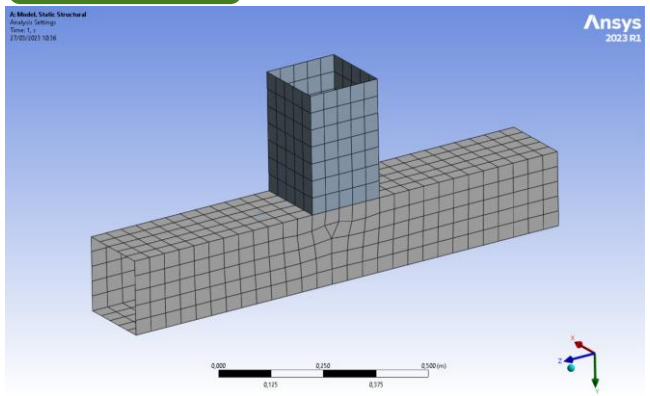
Create general Views

1. In Views context menu click **Add**
2. Title: *Front View*
3. Locate the model in Ansys Mechanical as shown on a picture *Front View*. Click **Get**
4. Press OK. Repeat the procedure with the *Back View*

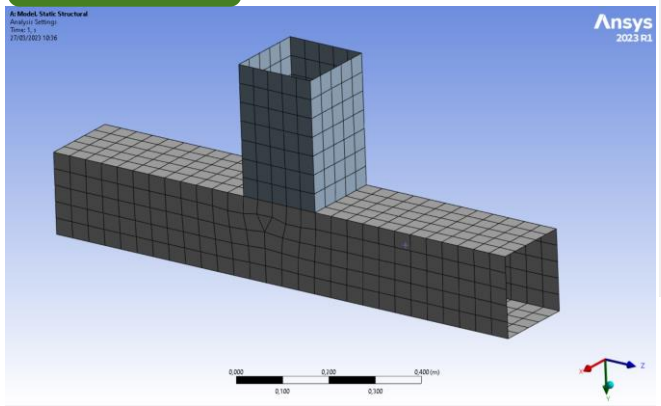
Views are a set of options that define how the plots are displayed.

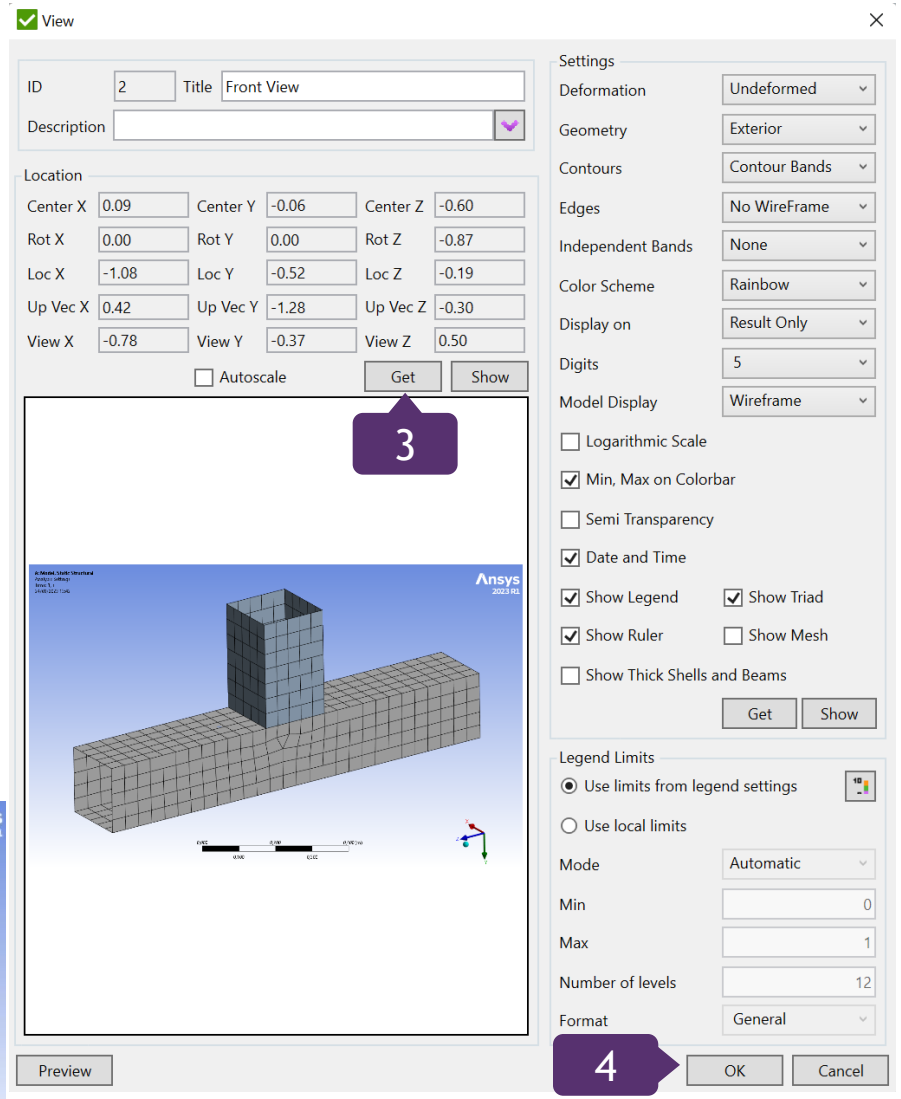


Front View



Back View

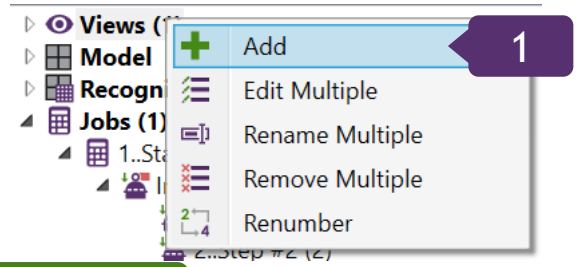




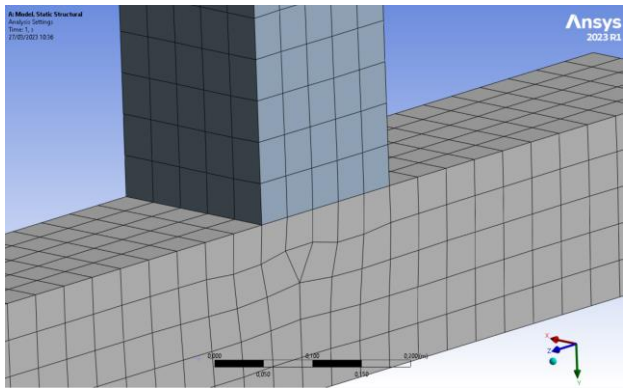
Create detailed Views

1. In Views context menu click **Add**
2. Title: *Front Detail View*
3. Locate the model in Ansys Mechanical as shown on a picture *Front Detail View*. Click **Get**
4. Press OK. Repeat the procedure with the *Back Detail View*

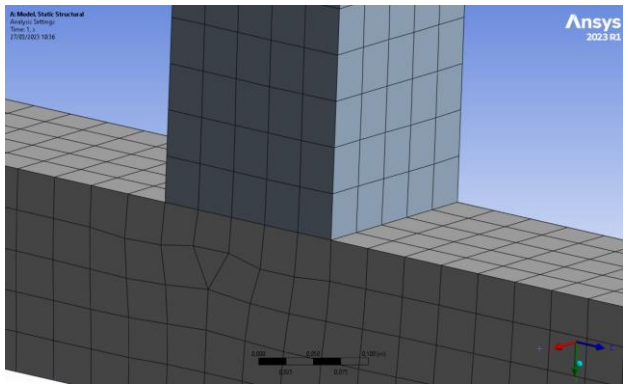
Views are a set of options that define how the plots are displayed.

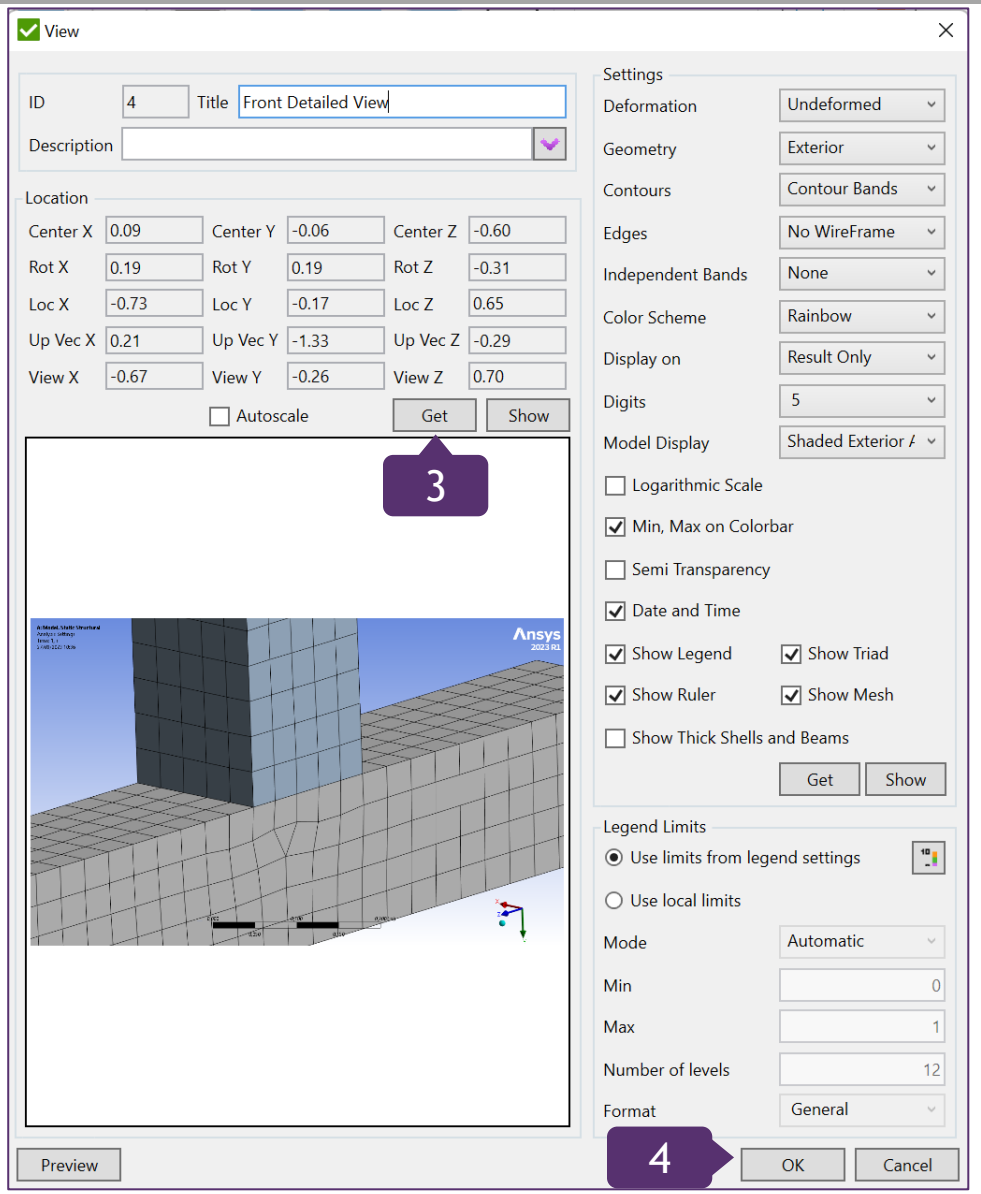


Front View



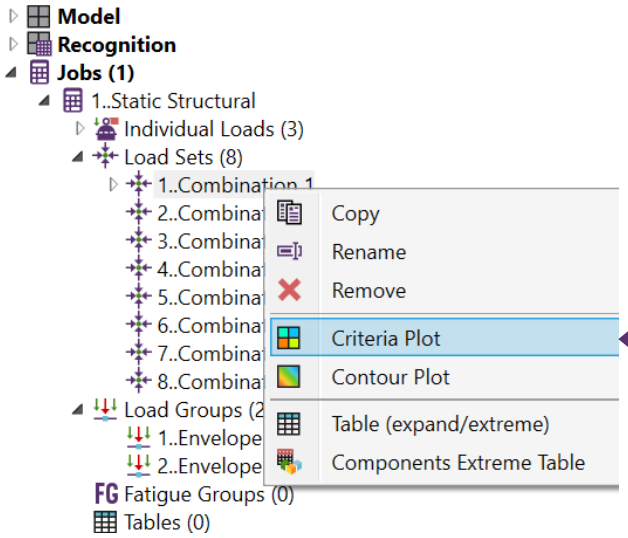
Back View

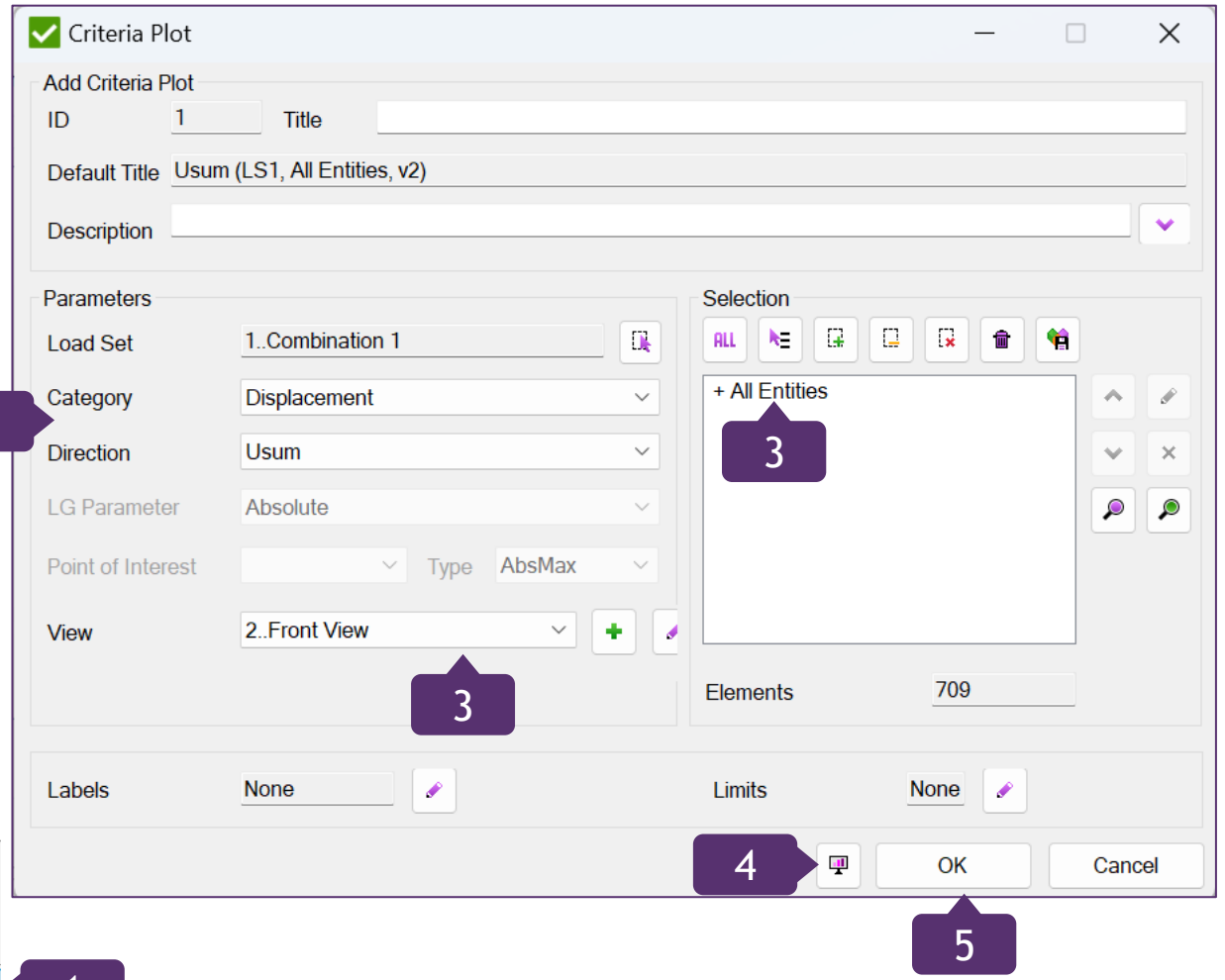




Add Displacement Plot

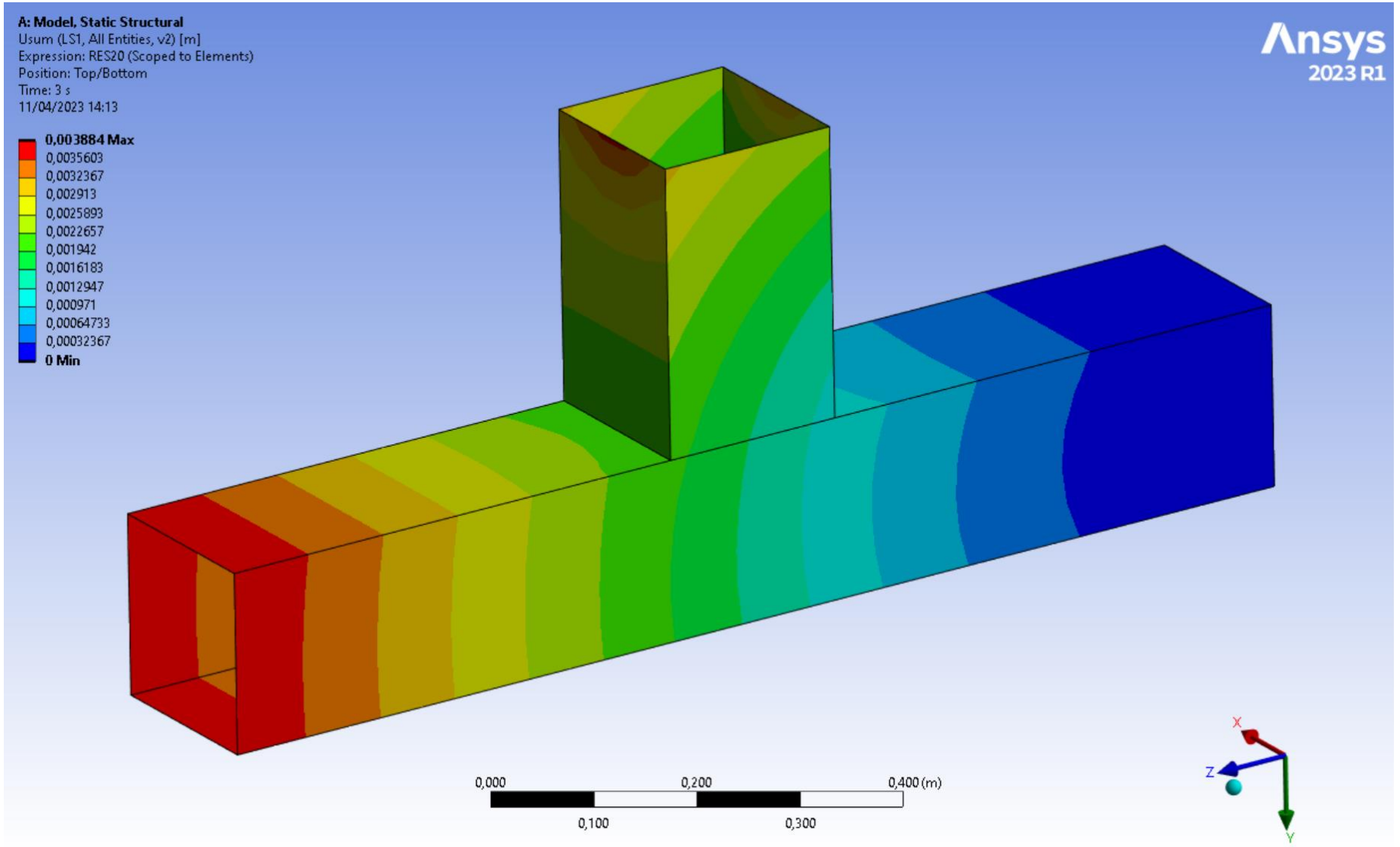
1. Right click on **LS1.. Combination 1 > Criteria Plot**
2. Select Category – *Displacement*
   Direction - *Usum*
3. View – *2. Front View*
   Selection – *All Entities*
4. Press **Preview**
   (Result on a next slide)
5. Press **OK**





Displacement Plot Result

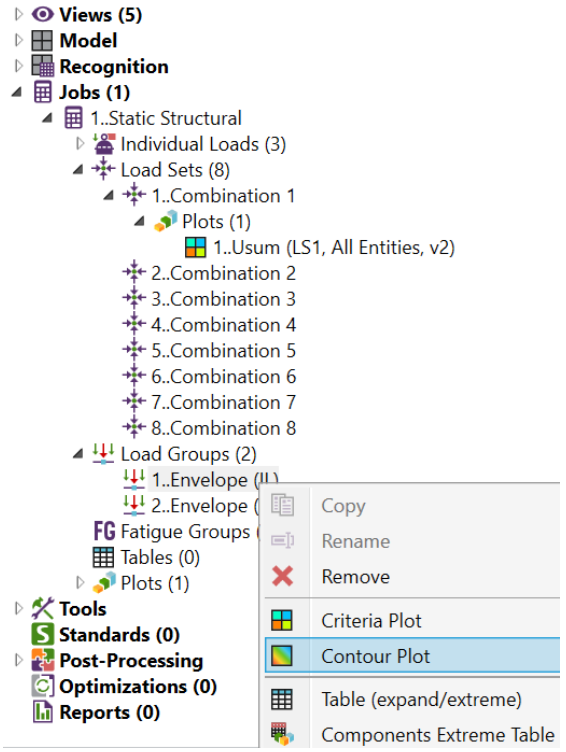
Usum Displacement plot for All entities with Front View. Defined on a previous slide.

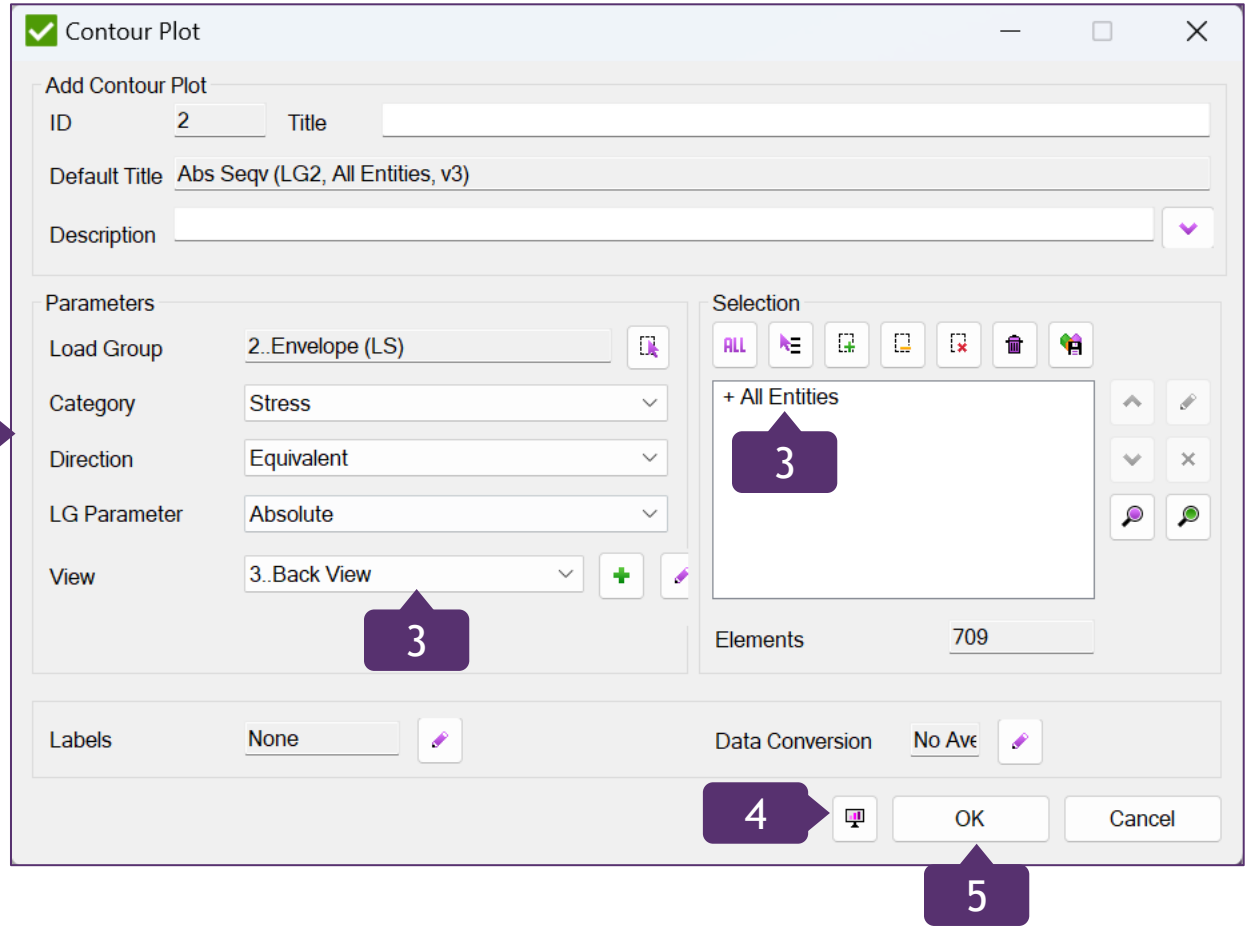


Add Stress Plot

1. Right click on **LG2.. Envelope (LS) > Contour Plot**
2. Select Category – *Stress*
   Direction - *Equivalent*
3. View – *3. Back View*
   Selection – *All Entities*
4. Press **Preview**
   (Result on a next slide)
5. Press **OK**

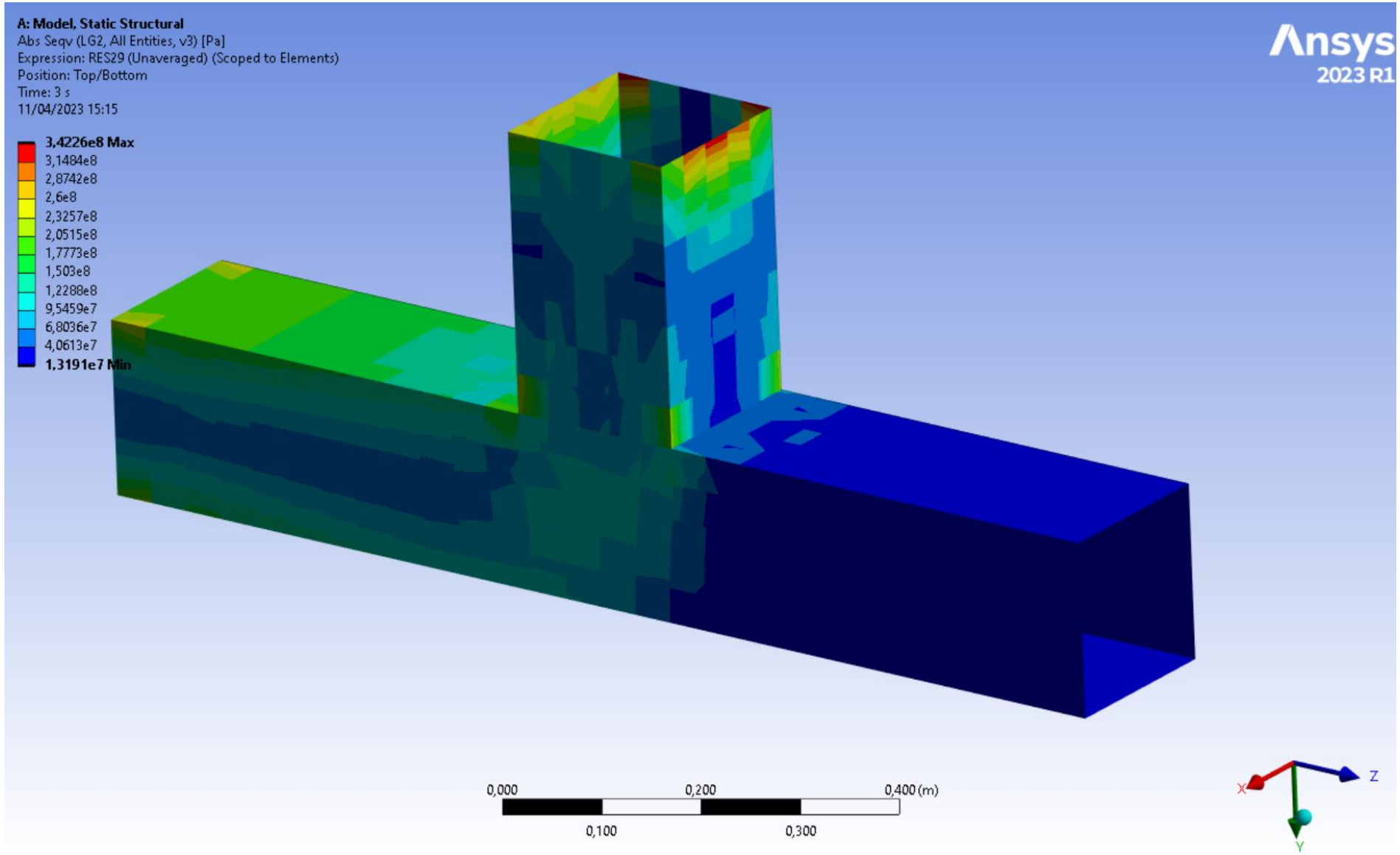
Criteria – each element is colored based on a single output value for the element.
Contour – element is colored based on the corner data.





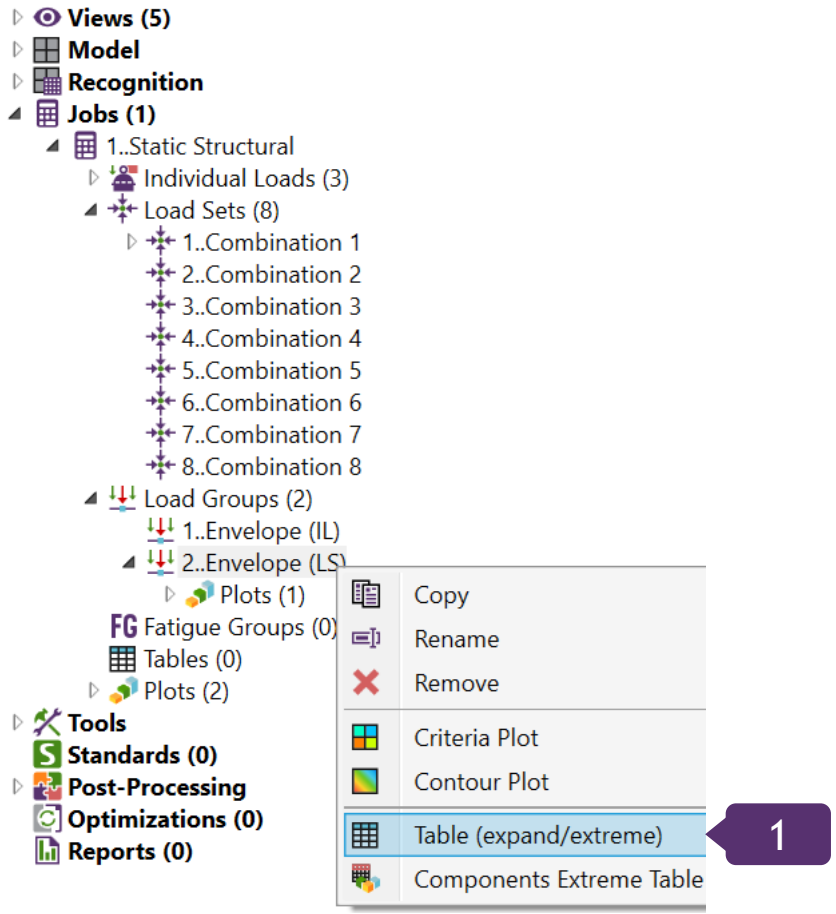
Stress Plot Result

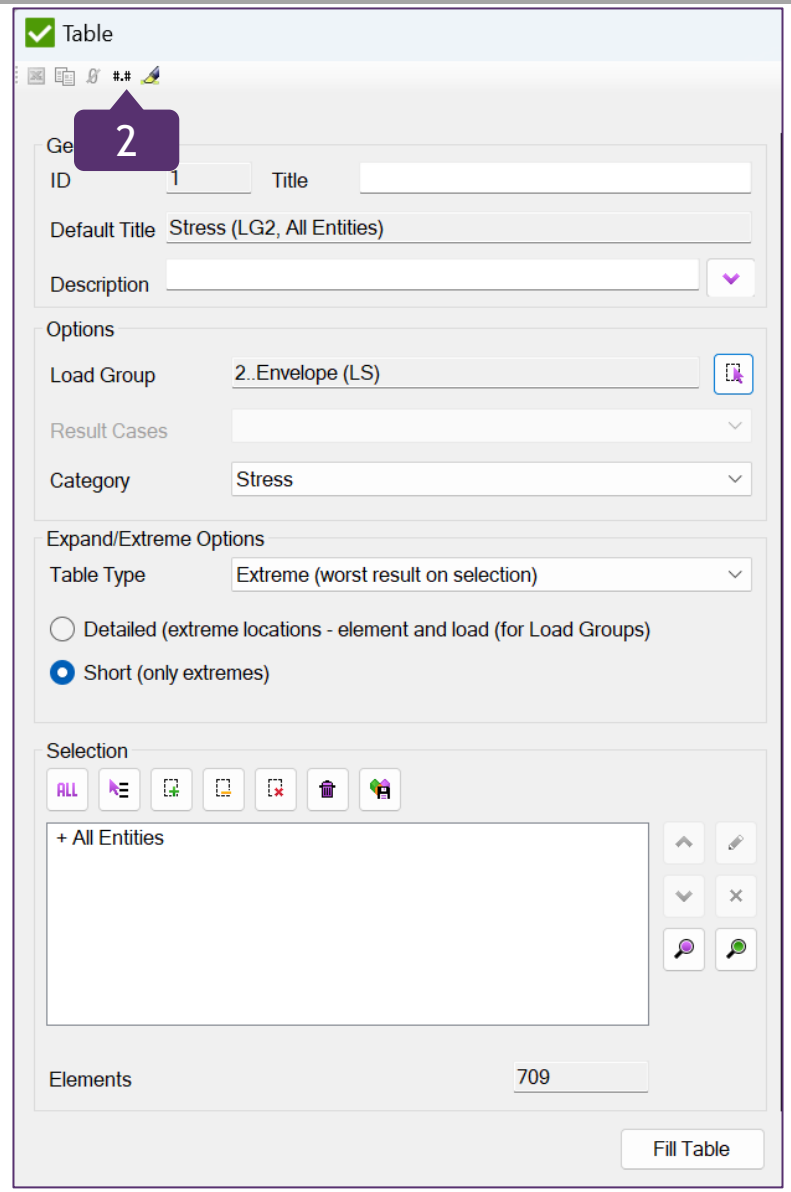
Equivalent Stress plot for All entities with Back View. Defined on a previous slide.



Add Simple Table

1. Right click on **LG2.. Envelope (LS) > Table (expand/extreme)**
2. Click **Number Format**



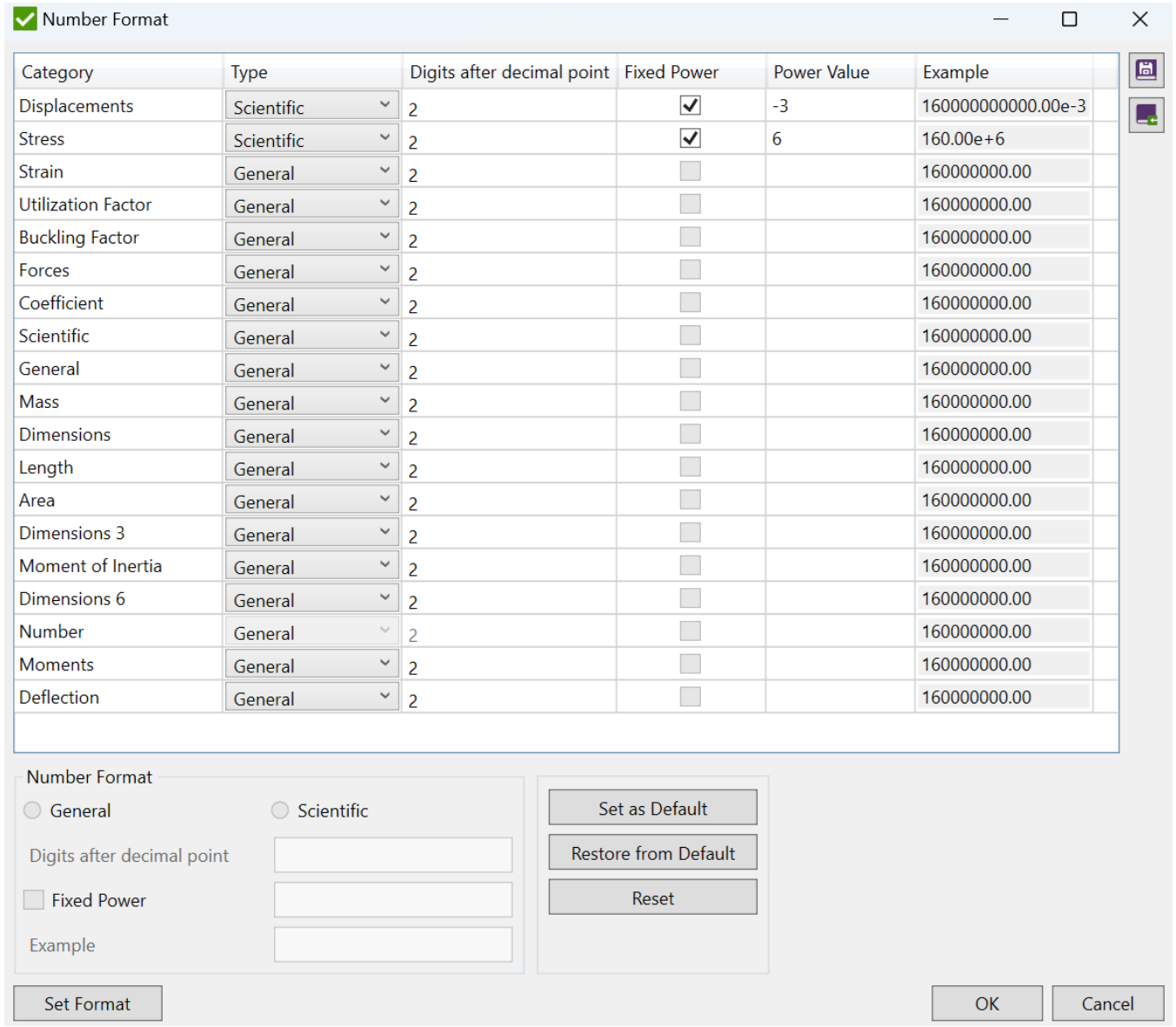


Number Format

1. For Category *Displacements* set **Scientific format > Fixed Power e-3**
2. For Category *Stress* set **Scientific format > Fixed Power e6**

FEA interfaces are unitless. It is important to adjust the number formats for better data readability.

On this slide we're setting Displacements to be shown in mm, and Stresses in MPa

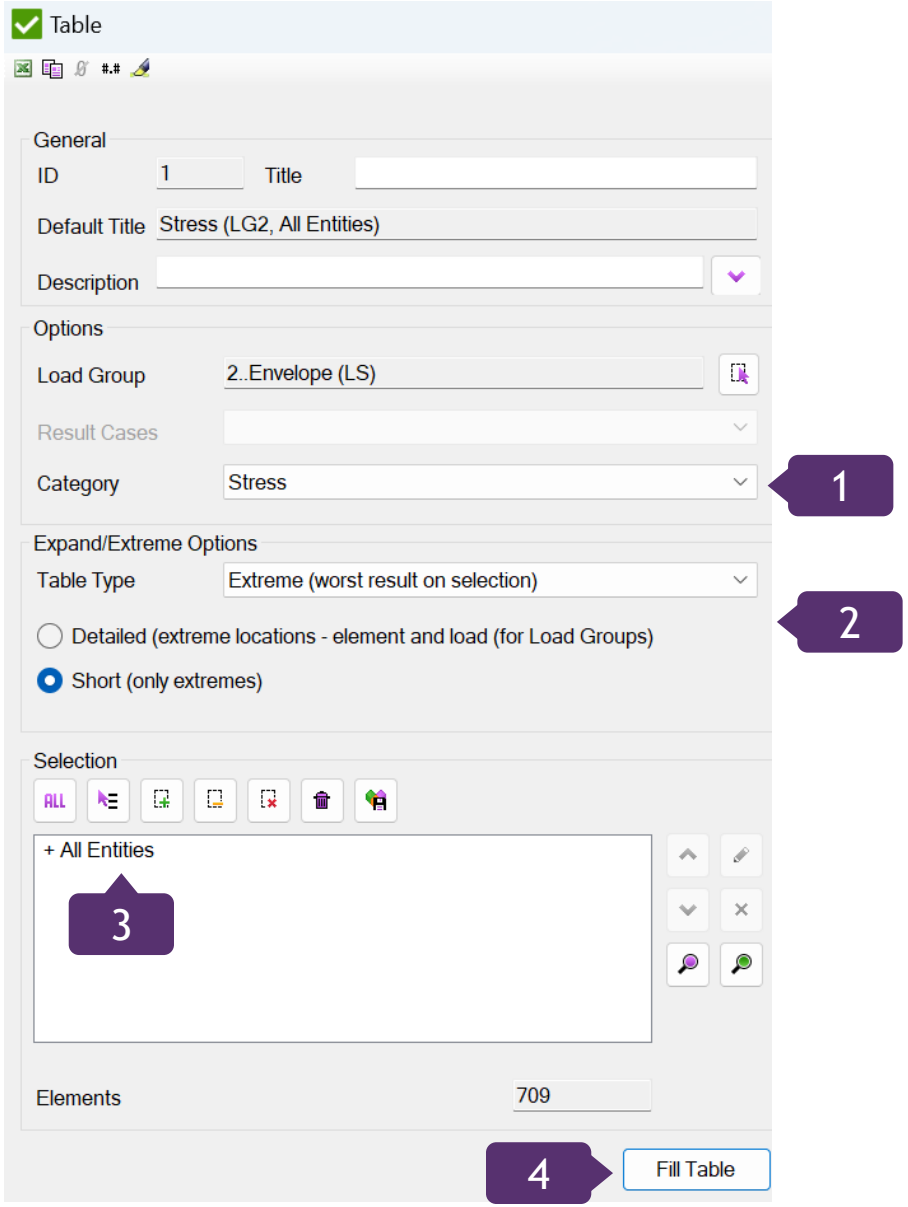


Stress Table

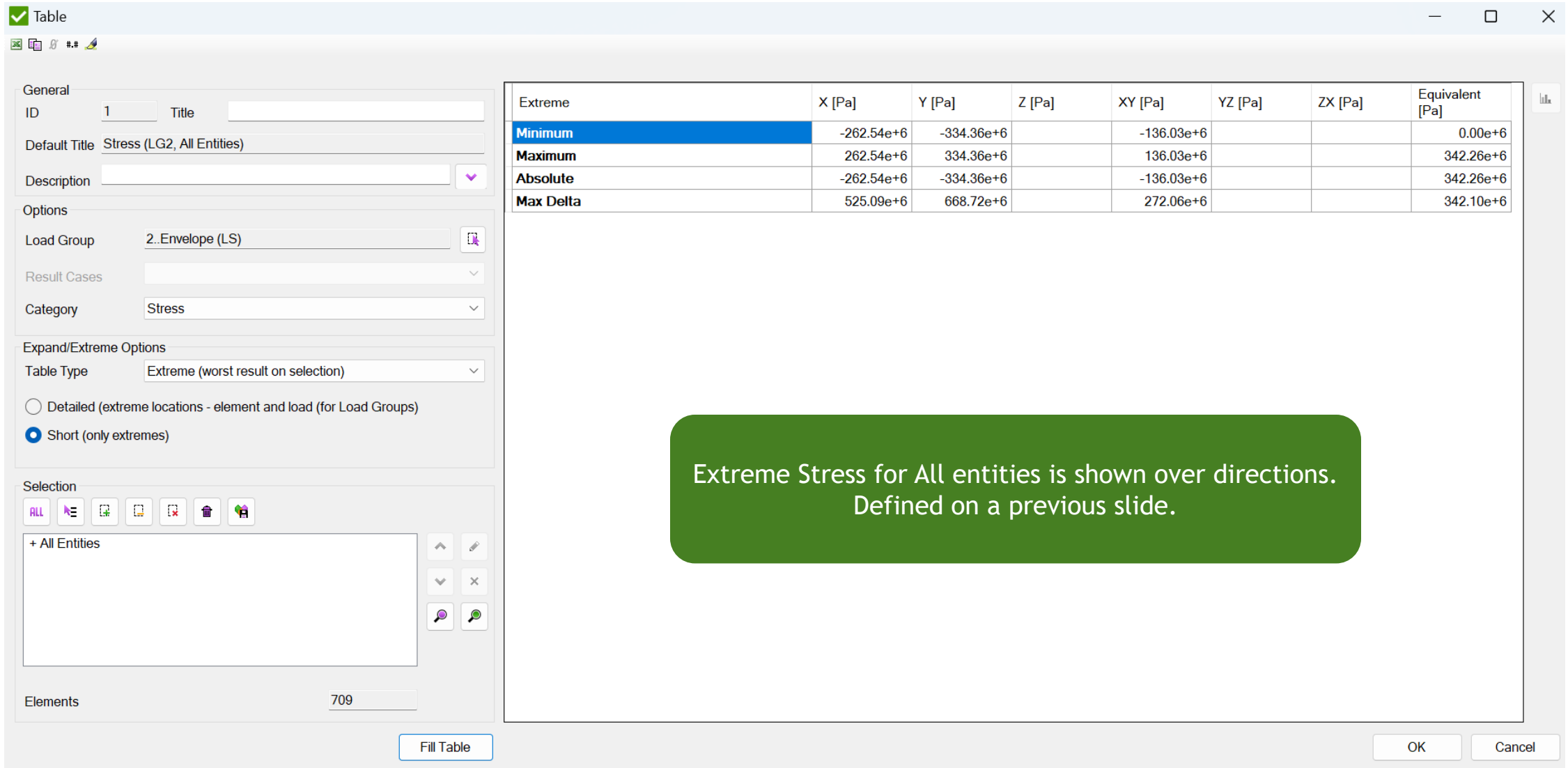
1. Select Category – *Stress*
2. Table Type – *Extreme / Short*
3. Selection – *All Entities*
4. Press Fill Table
(Result on a next slide)

An **expand table** displays the results for each Node or Element in Selection.

An **extreme table** displays minimum, maximum, absolute maximum or max delta (for load groups) results and their location over directions.



Stress Table with Results

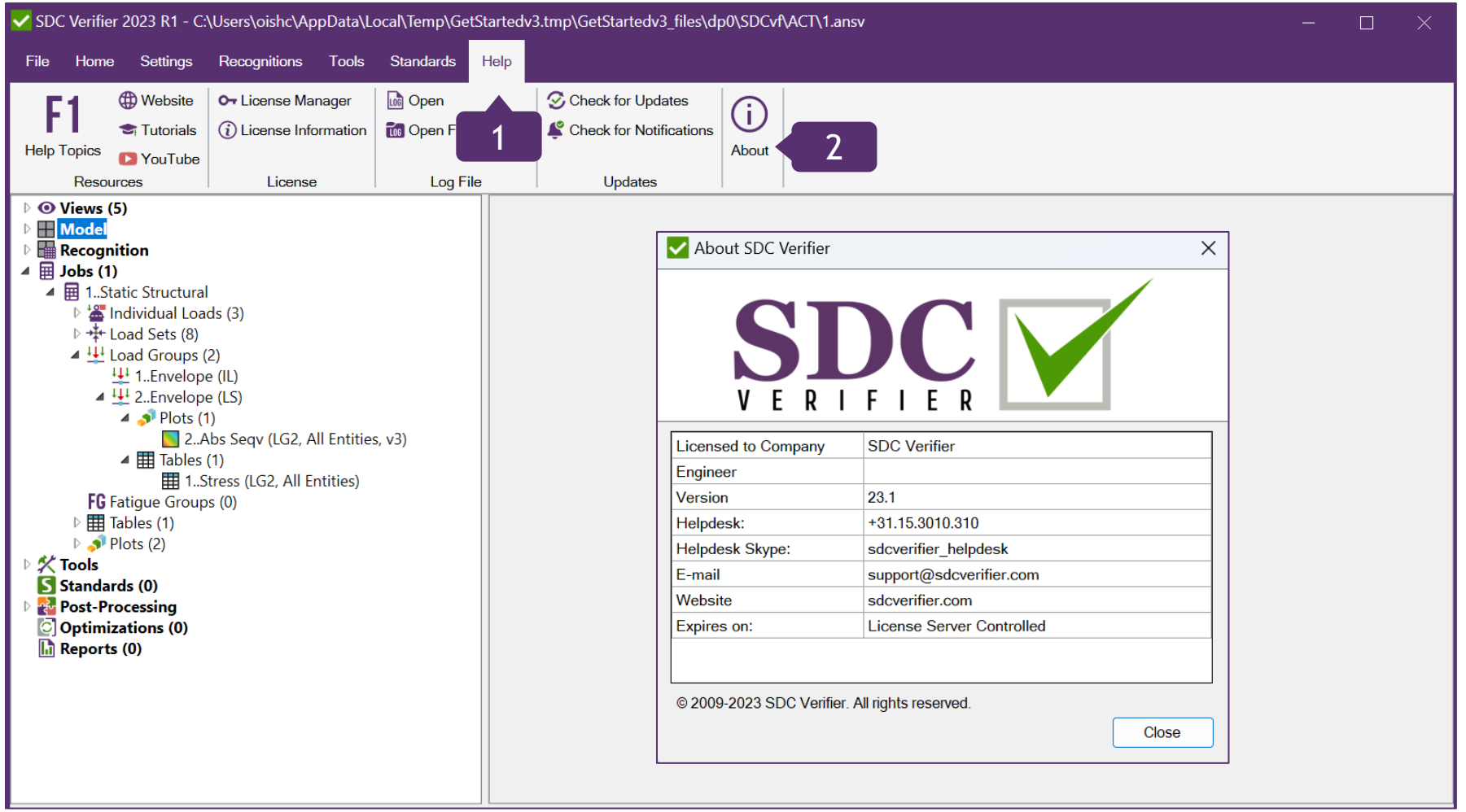


Extreme	X [Pa]	Y [Pa]	Z [Pa]	XY [Pa]	YZ [Pa]	ZX [Pa]	Equivalent [Pa]
Minimum	-262.54e+6	-334.36e+6		-136.03e+6			0.00e+6
Maximum	262.54e+6	334.36e+6		136.03e+6			342.26e+6
Absolute	-262.54e+6	-334.36e+6		-136.03e+6			342.26e+6
Max Delta	525.09e+6	668.72e+6		272.06e+6			342.10e+6

Extreme Stress for All entities is shown over directions. Defined on a previous slide.

Support

1. Go to **Help** tab in the main window
2. Click **About**



Should you need any help or assistance with SDC Verifier do not hesitate to contact us.

by phone: +31 15 30-10-310:
by email: support@sdcverifier.com
by Skype: sdcverifier_helpdesk