



Tutorial

Get Started

Updated on: 11.04.2023

Tested with: SDC Verifier 2023R1

Ansys 2023R1

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

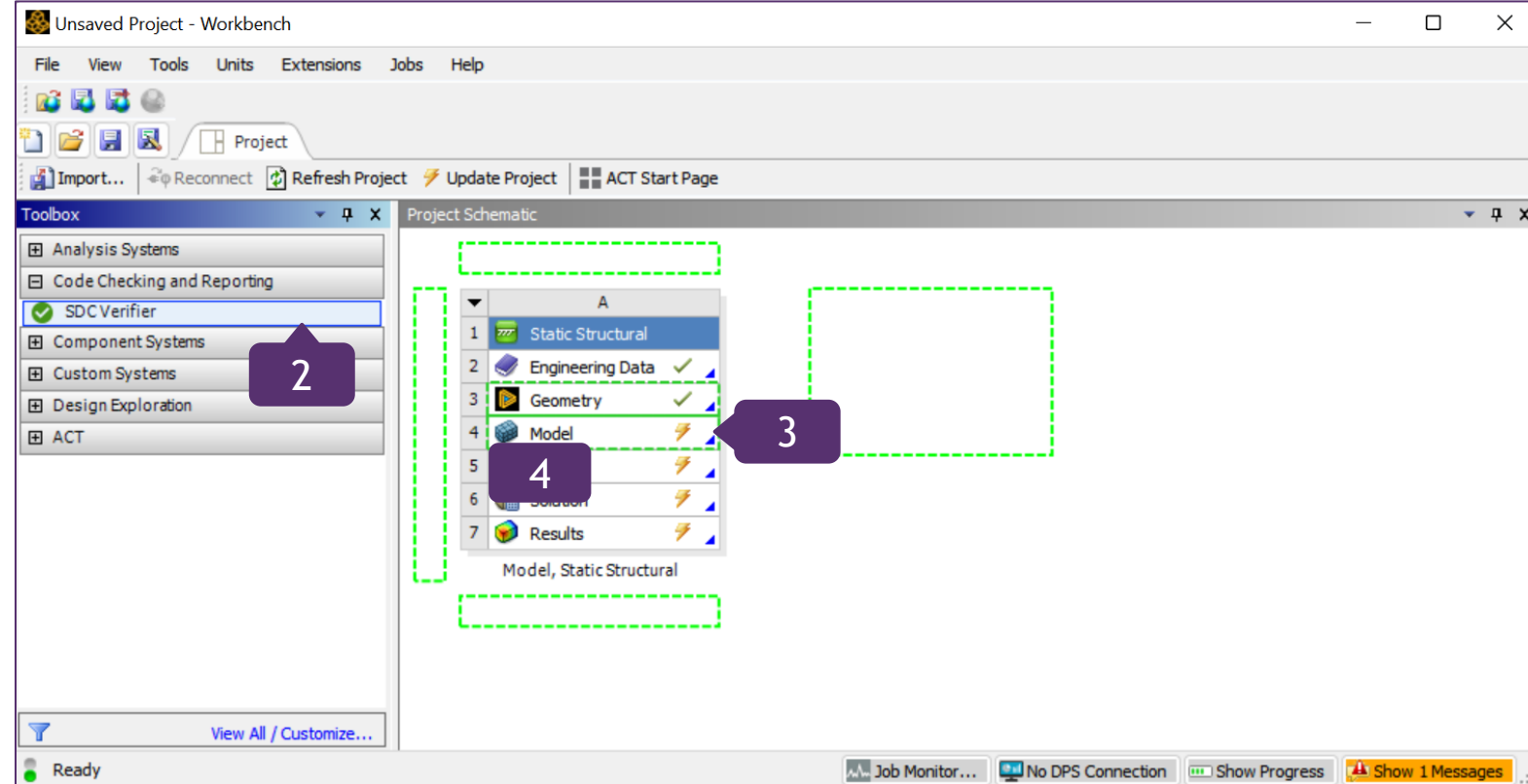
Drag **SDC Verifier** from Toolbox **Code Checking and Reporting** category

3

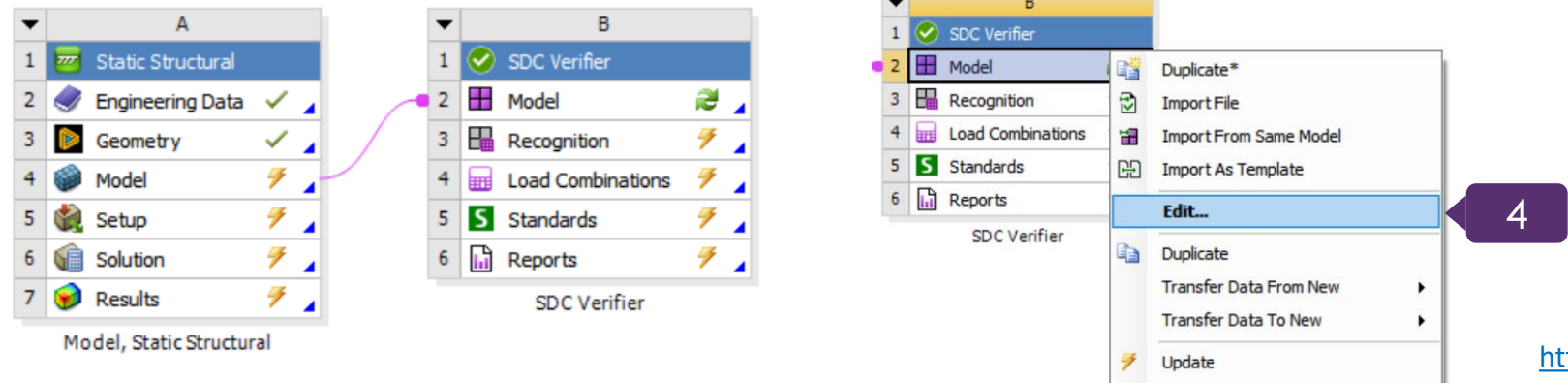
Drop on **Model** Task of **Static Structural** Task Group.

4

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



Project scheme after step 3:



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.

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: C:\Users\oishc\AppData\Local\Temp\GetStartedv3.tmp\GetStartedv3_files\dp0\global\MECH\SYS.mechdb

Get Active Model Path

Project Details

Number: p0001

Name: beam-to-beam connection

Description:

Unit System: MKS (Meter/Kg/Second)

Engineer details

Engine:

Company:

E-mail:

Phone:

Address:

Website:

Logo:

Customer details

Customer:

Company:

E-mail:

Phone:

Address:

Website:

Logo:

Previous Next Finish

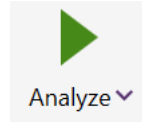
Show on startup

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

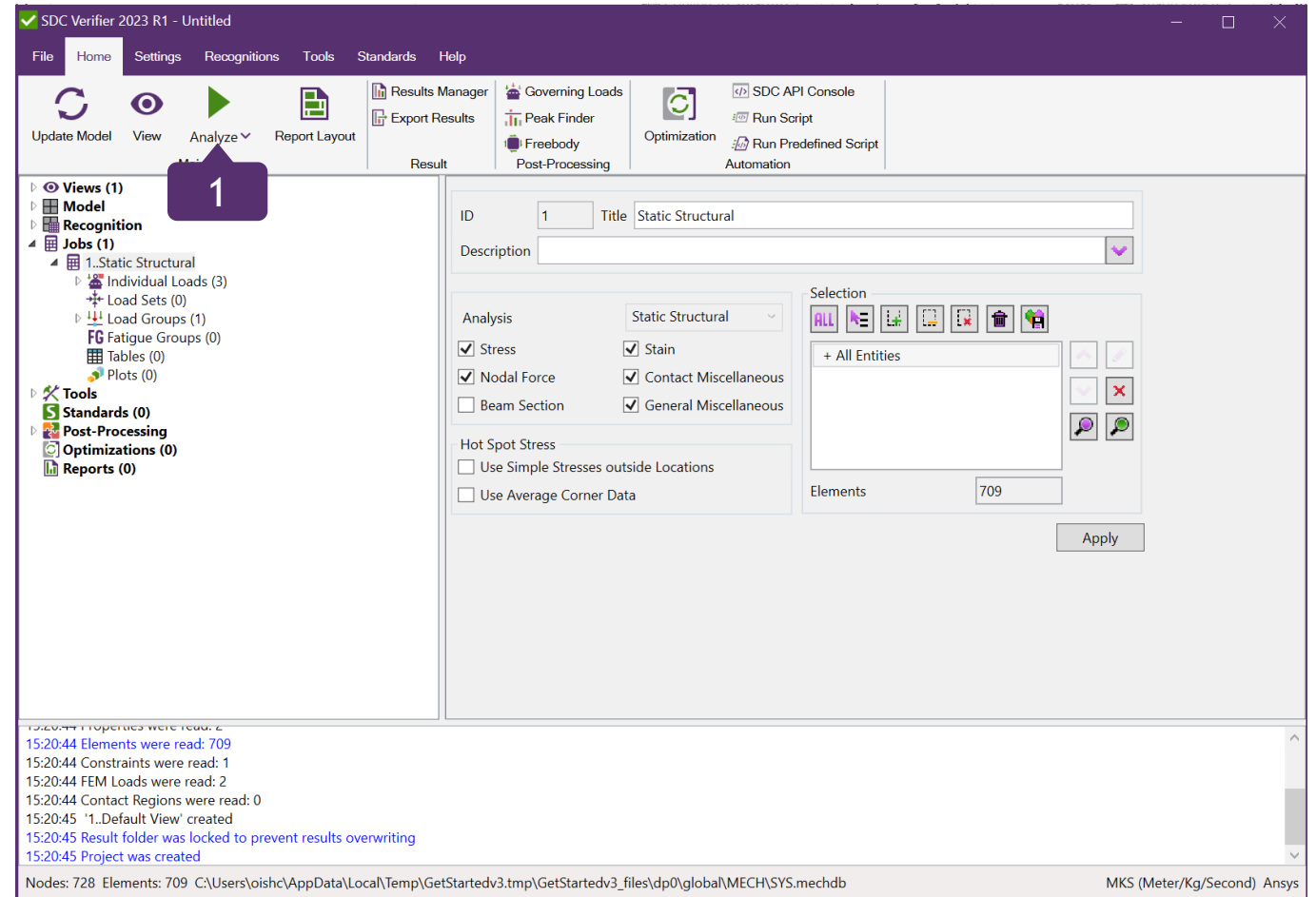
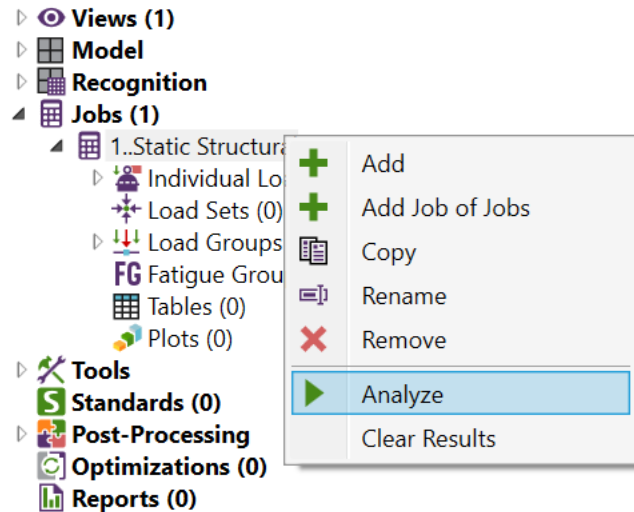
Analyze the Job

1

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

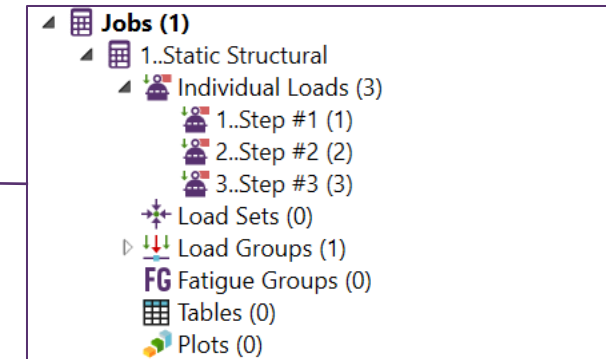
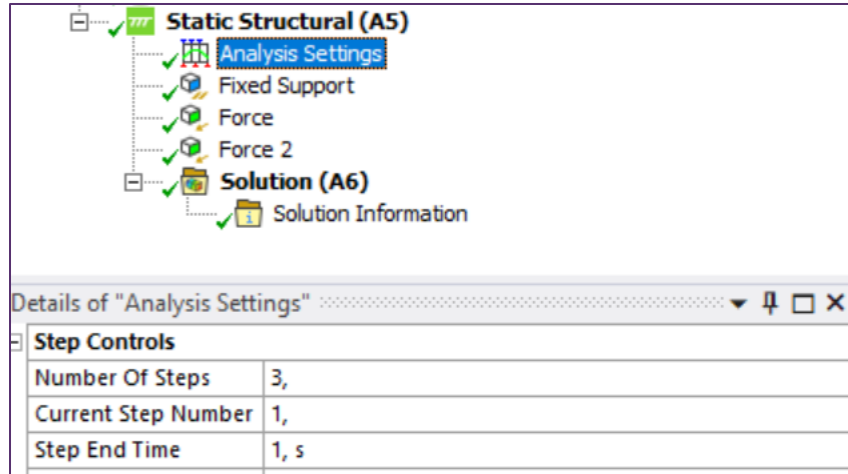


Alternatively, execute right click on the *Active Job* in the *Tree* and press **Analyze**



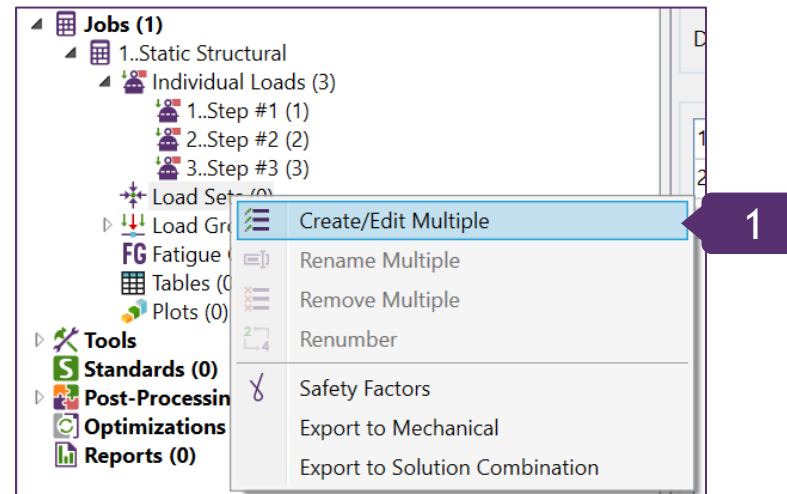
Individual Loads - single load results, each load will be created automatically from steps in Ansys Mechanical.
In our case we have 3 Individual loads.

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 |
|---|---------------|---------------|----------------------------------|----------------------------------|----------------------------------|
| | | Safety Factor | IL1..Static Structural - step #1 | IL2..Static Structural - step #2 | IL3..Static Structural - step #3 |
| 1 | | | | | |
| 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

✓ Create/Edit Multiple LoadSets

| | Safety Factor | IL1..Step #1 (1) | IL2..Step #2 (2) | IL3..Step #3 (3) |
|---------------|---------------|------------------|------------------|------------------|
| Combination 1 | 1 | 1 | 1 | 1,33 |
| Combination 2 | 1 | -1 | 1 | 1,33 |
| Combination 3 | 1 | 1 | -1 | 1,33 |
| Combination 4 | 1 | -1 | -1 | 1,33 |
| Combination 5 | 1 | 1 | 1 | -1,33 |
| Combination 6 | 1 | -1 | 1 | -1,33 |
| Combination 7 | 1 | 1 | -1 | -1,33 |
| Combination 8 | 1 | -1 | -1 | -1,33 |

Add load Sets
Count 1

2

Factor
1

Clipboard

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

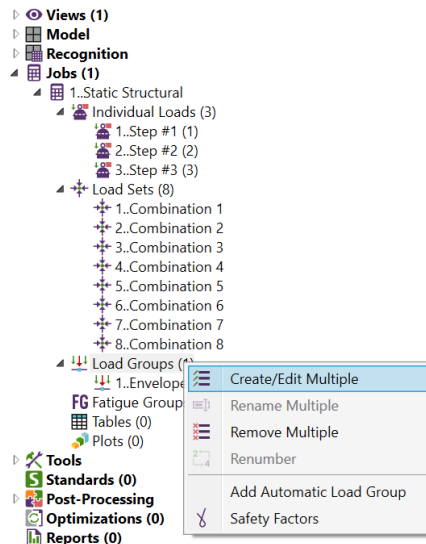
Envelope (IL) Load Group is already existing,
Title the new LG: **Envelope (LG)**

4

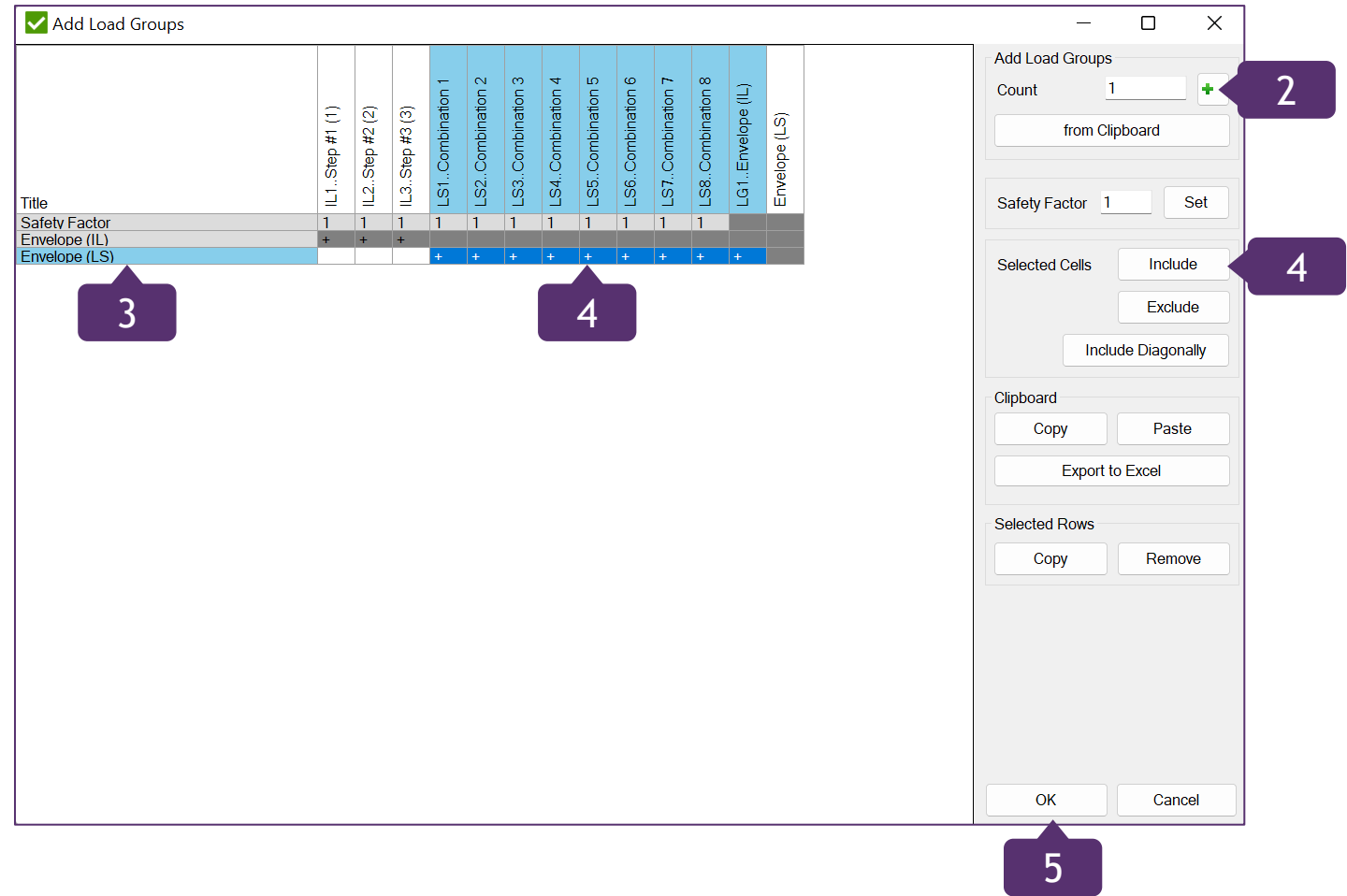
Select all cells of the LS in this new Load Group and click **Include**

5

Press OK



1



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*

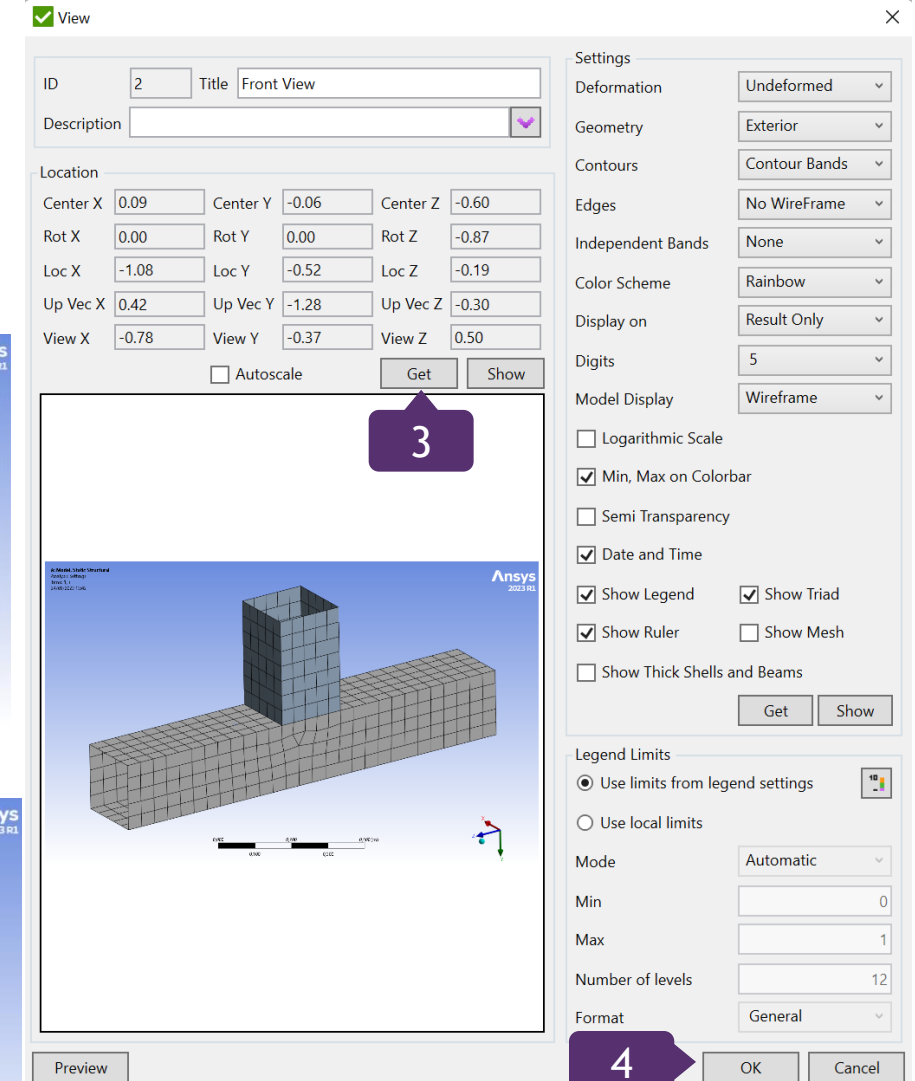
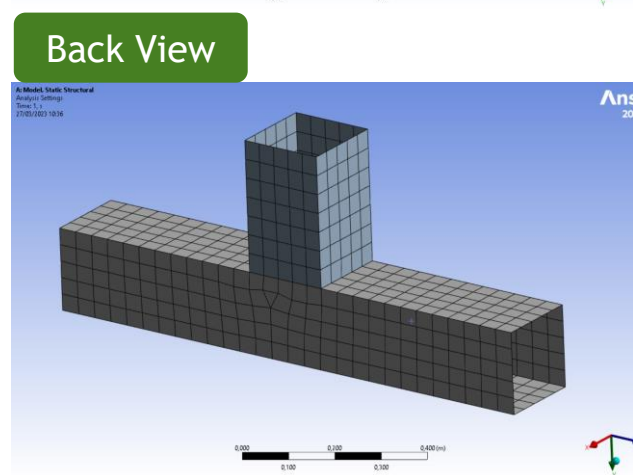
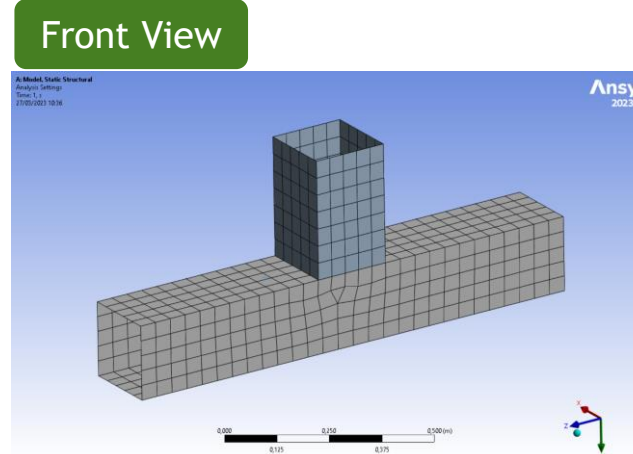
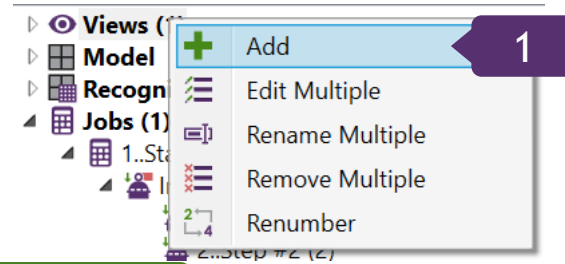
3

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.



Create detailed Views

1

In Views context menu click **Add**

2

Title: *Front Detail View*

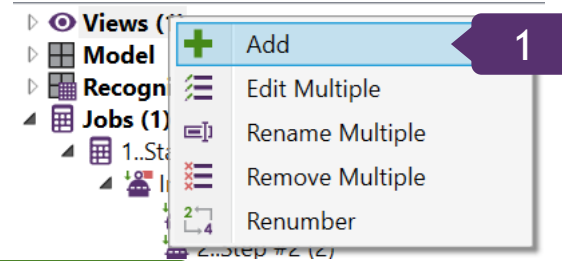
3

Locate the model in Ansys Mechanical as shown on a picture *Front Detail View*. Click **Get**

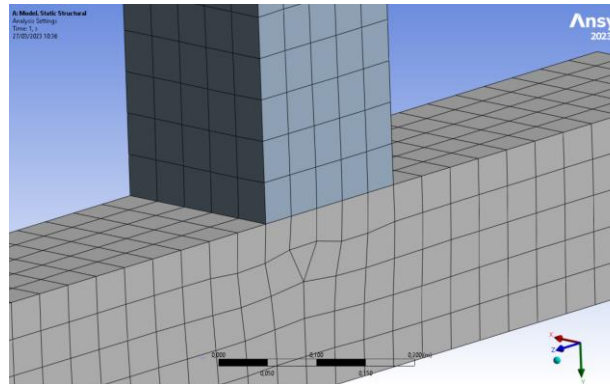
4

Press OK. Repeat the procedure with the *Back Detail View*

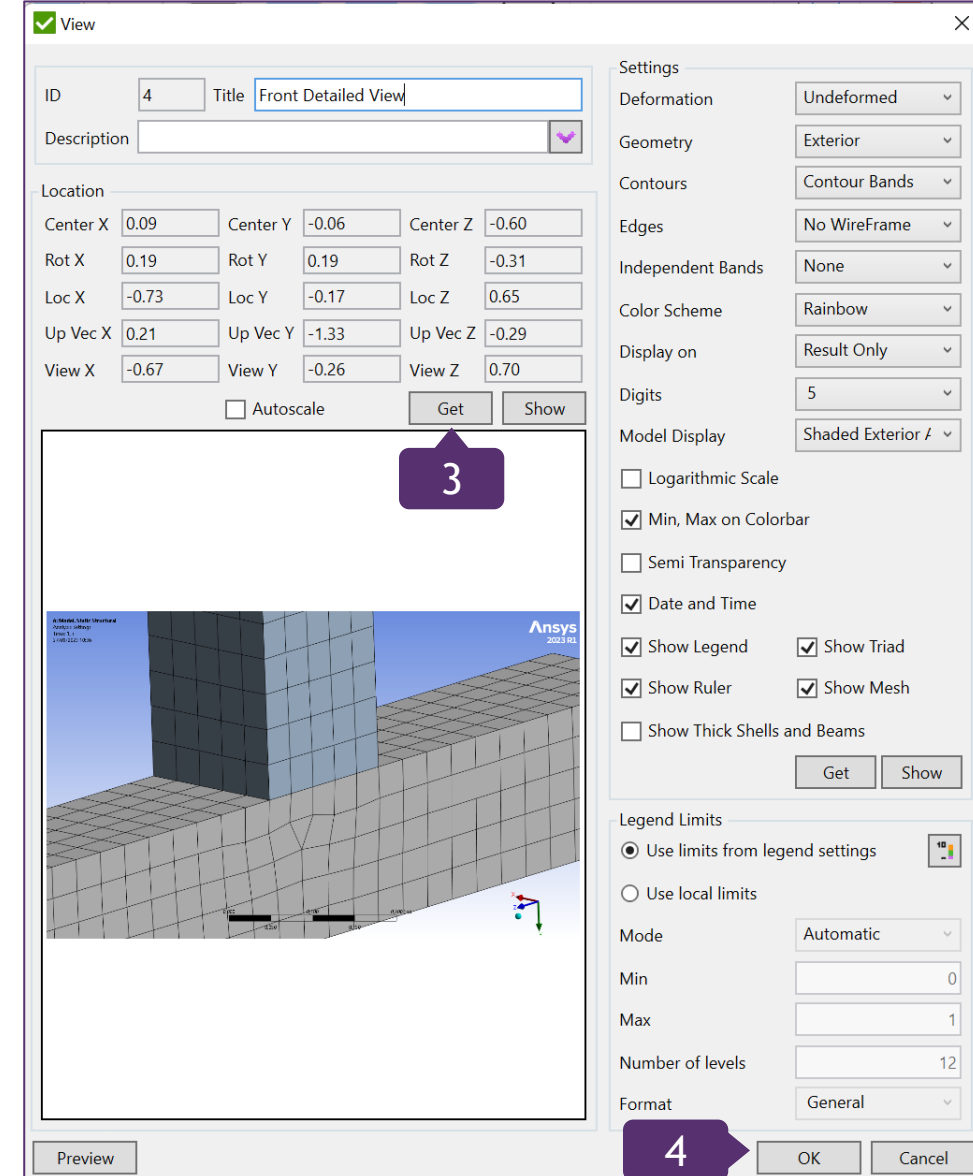
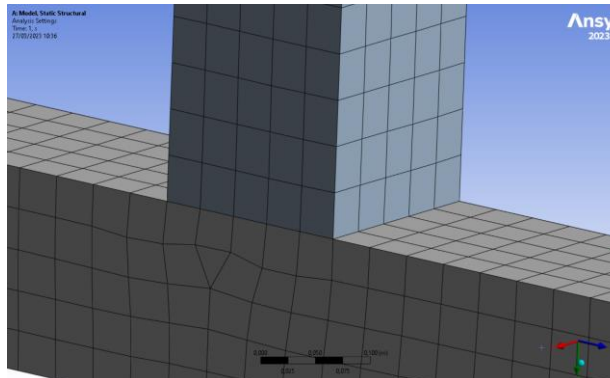
Views are a set of options that define how the plots are displayed.



Front View



Back View



Add Displacement Plot

1

Right click on **LS1.. Combination 1** >
Criteria Plot

2

Select Category - *Displacement*
Direction - *Usum*

3

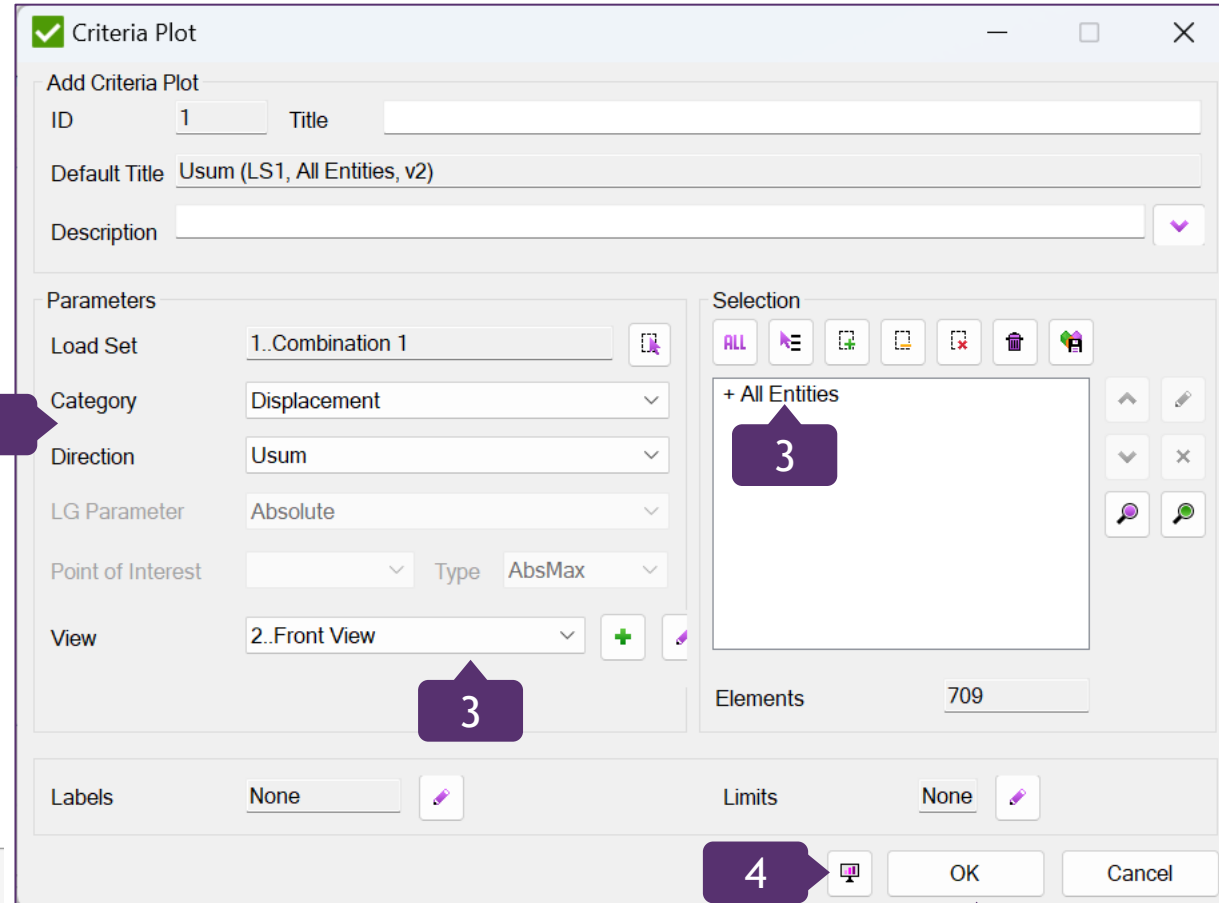
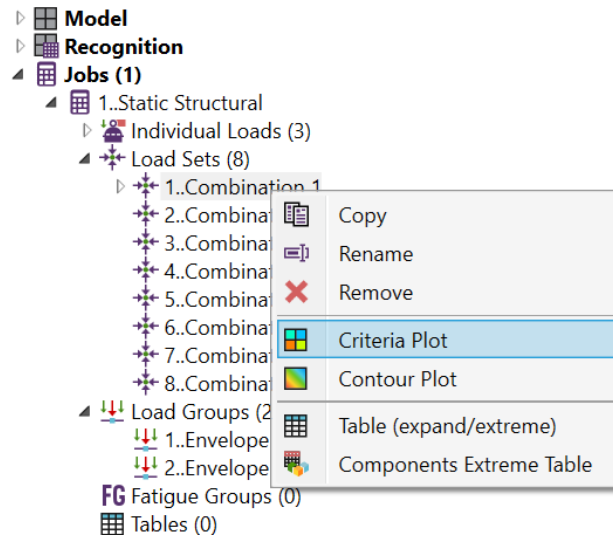
View - *2. Front View*
Selection - *All Entities*

4

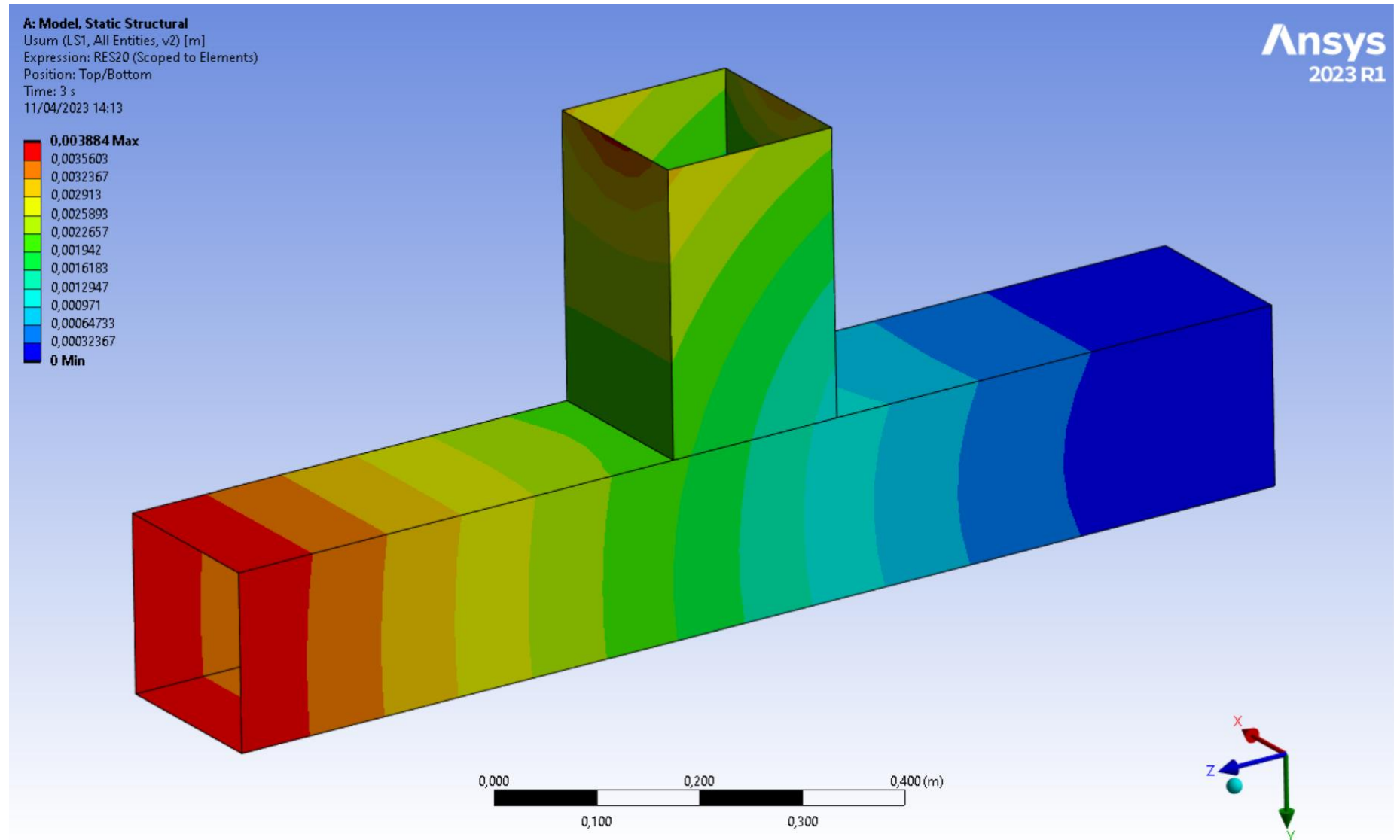
Press **Preview**
(Result on a next slide)

5

Press **OK**



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

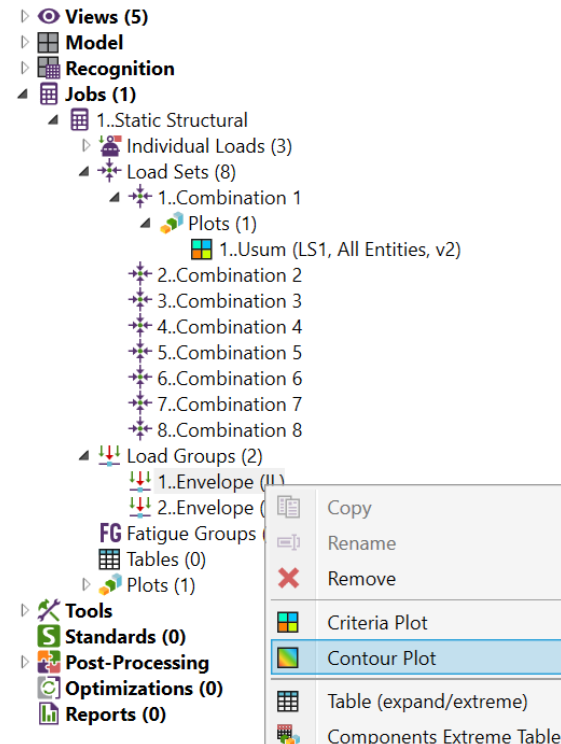
Press **Preview**
(Result on a next slide)

5

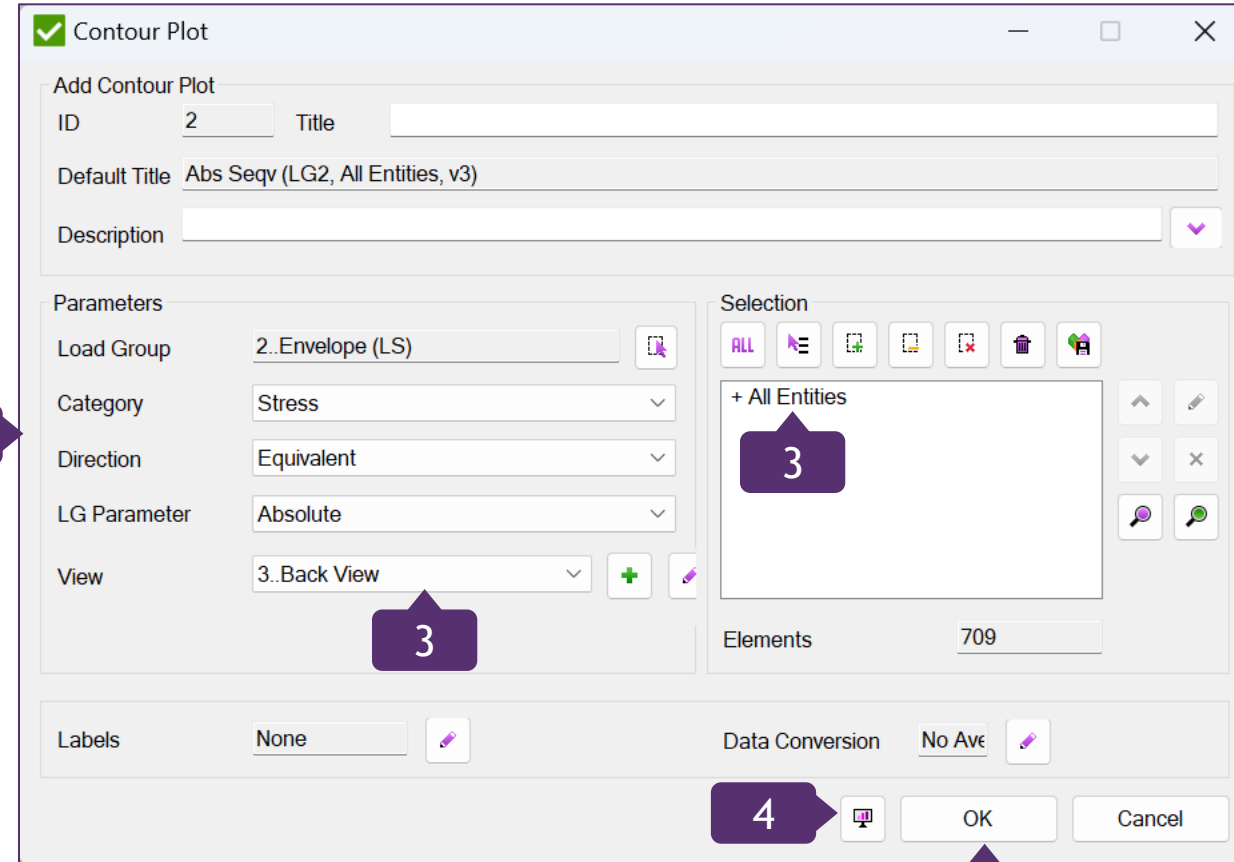
Press **OK**

Criteria - each element is colored based on a single output value for the element.

Contour - element is colored based on the corner data.



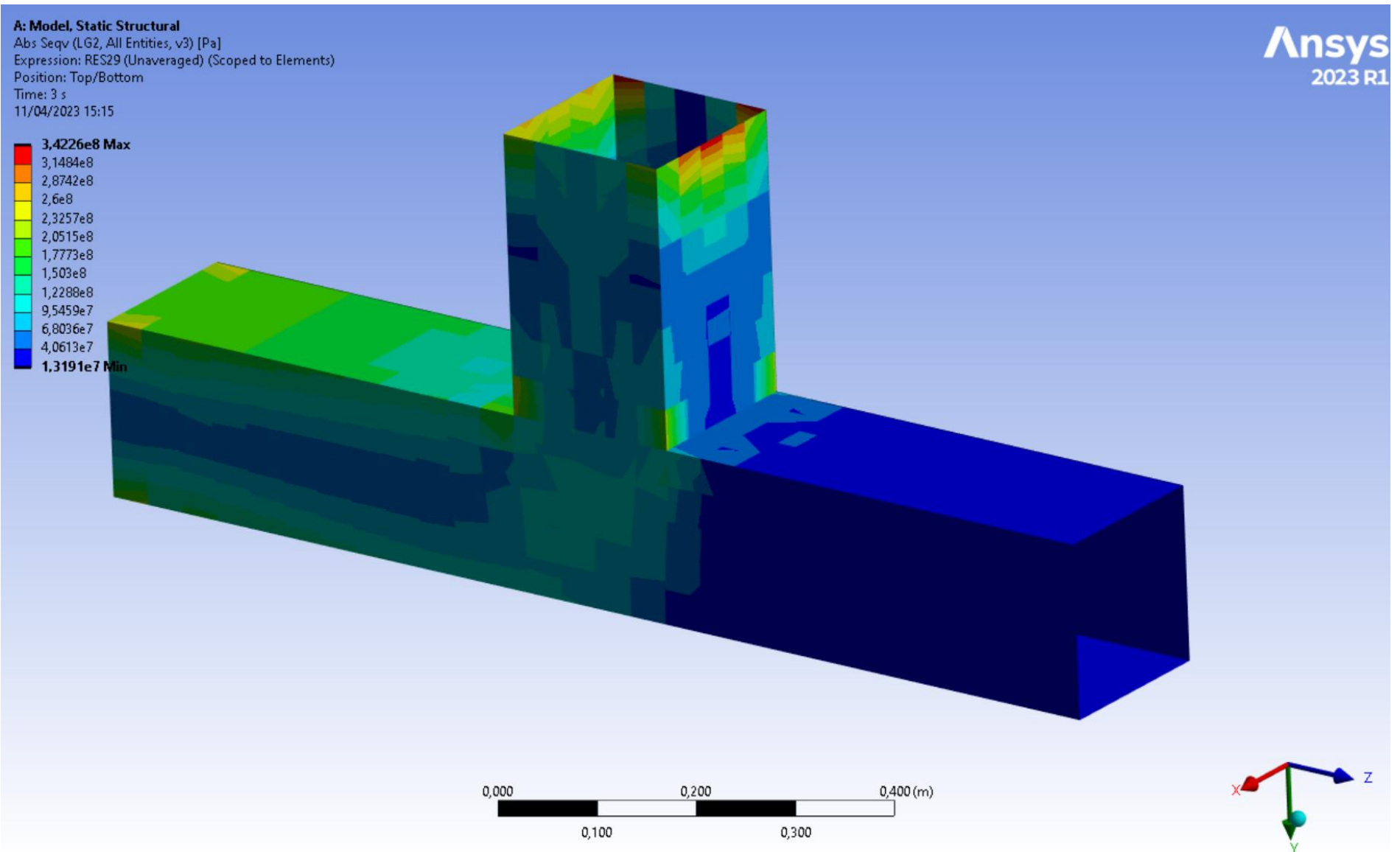
2



1

4

5



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

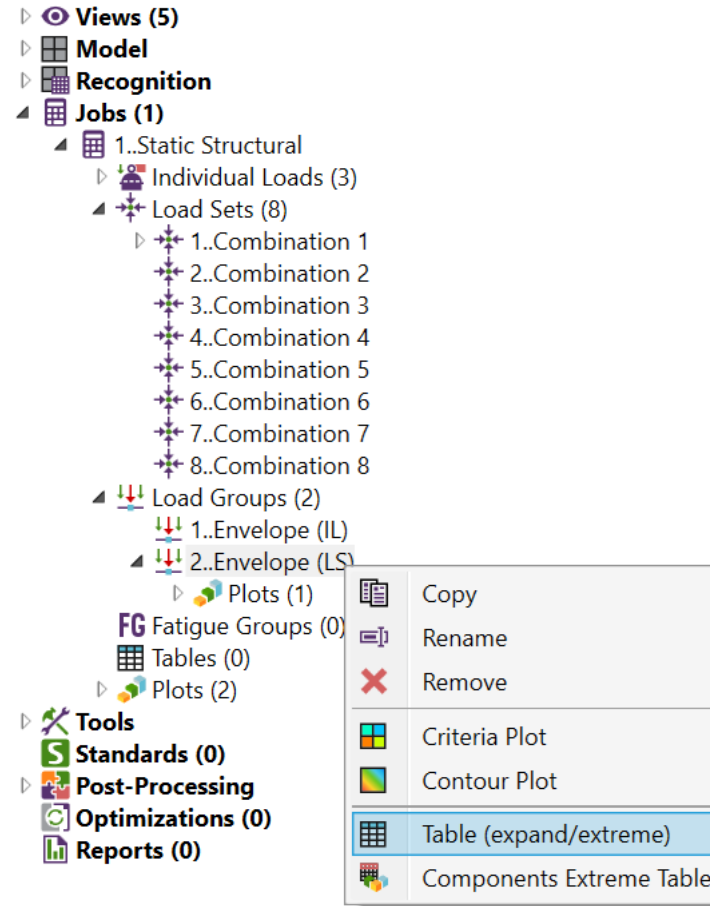
Add Simple Table

1

Right click on **LG2.. Envelope (LS)** >
Table (expand/extreme)

2

Click **Number Format**



The 'Table' configuration dialog box is shown with the following fields and options:

- Ge ID:** 1 (Highlighted with callout '2')
- Title:** [Empty field]
- Default Title:** Stress (LG2, All Entities)
- Description:** [Empty field]
- Options:**
 - Load Group:** 2..Envelope (LS)
 - Result Cases:** [Empty field]
 - Category:** Stress
- Expand/Extreme Options:**
 - Table Type:** Extreme (worst result on selection)
 - ☐ Detailed (extreme locations - element and load (for Load Groups))
 - ☒ Short (only extremes)
- Selection:**
 - Buttons: ALL, [Filter], [Reset], [Save], [Delete], [New]
 - + All Entities** [List area]
 - Elements:** 709
- Fill Table** button

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

Number Format

| Category | Type | Digits after decimal point | Fixed Power | Power Value | Example |
|--------------------|------------|----------------------------|-------------------------------------|-------------|-------------------|
| Displacements | Scientific | 2 | <input checked="" type="checkbox"/> | -3 | 16000000000.00e-3 |
| Stress | Scientific | 2 | <input checked="" type="checkbox"/> | 6 | 160.00e+6 |
| Strain | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Utilization Factor | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Buckling Factor | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Forces | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Coefficient | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Scientific | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| General | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Mass | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Dimensions | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Length | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Area | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Dimensions 3 | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Moment of Inertia | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Dimensions 6 | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Number | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Moments | General | 2 | <input type="checkbox"/> | | 160000000.00 |
| Deflection | General | 2 | <input type="checkbox"/> | | 160000000.00 |

Number Format

☐ General ☐ Scientific

Digits after decimal point

☐ Fixed Power

Example

Set as Default

Restore from Default

Reset

Set Format

OK Cancel

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.

The screenshot shows the 'Table' configuration window in SDC Verifier. It is divided into several sections: 'General', 'Options', 'Expand/Extreme Options', 'Selection', and 'Elements'. Callout 1 points to the 'Category' dropdown menu, which is set to 'Stress'. Callout 2 points to the 'Table Type' dropdown menu, which is set to 'Extreme (worst result on selection)'. Callout 3 points to the 'Selection' area, which shows '+ All Entities' selected. Callout 4 points to the 'Fill Table' button at the bottom right. The 'Elements' field at the bottom shows '709'.

Table

General

ID 1 Title

Default Title Stress (LG2, All Entities)

Description

Options

Load Group 2..Envelope (LS)

Result Cases

Category Stress

Expand/Extreme Options

Table Type Extreme (worst result on selection)

☐ Detailed (extreme locations - element and load (for Load Groups))

☒ Short (only extremes)

Selection

ALL

+ All Entities

Elements 709

Fill Table

SDC
VERIFIER

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

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

