



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

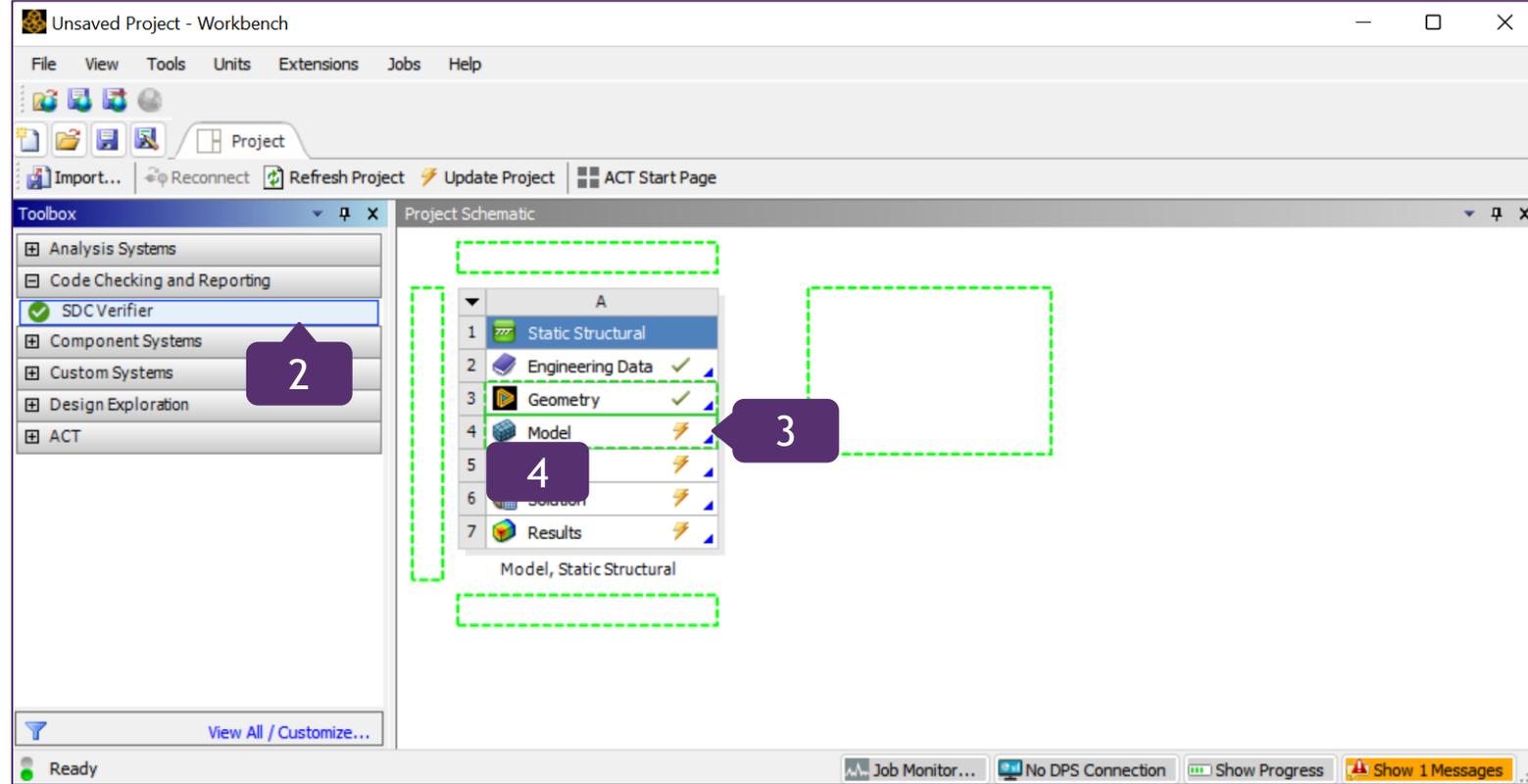
Drag **SDC Verifier** from Toolbox **Code Checking and Reporting** category

3

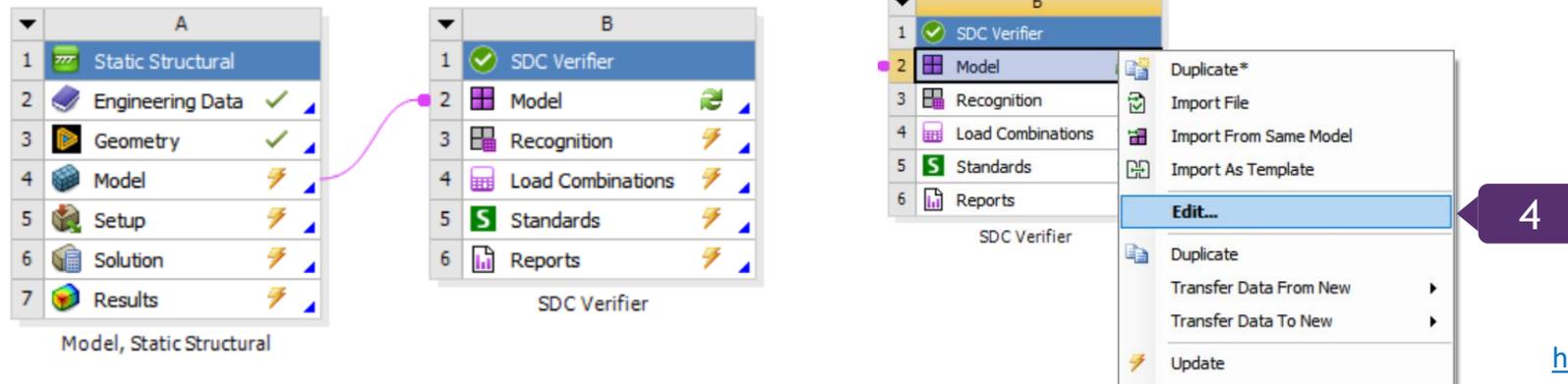
Drop on **Model** Task of **Static Structural** Task Group.

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

Engineer

Company

E-mail

Phone

Address

Website

Logo

Customer details

Customer

Company

E-mail

Phone

Address

Website

Logo

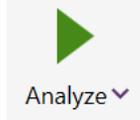
Show on startup

Previous Next Finish

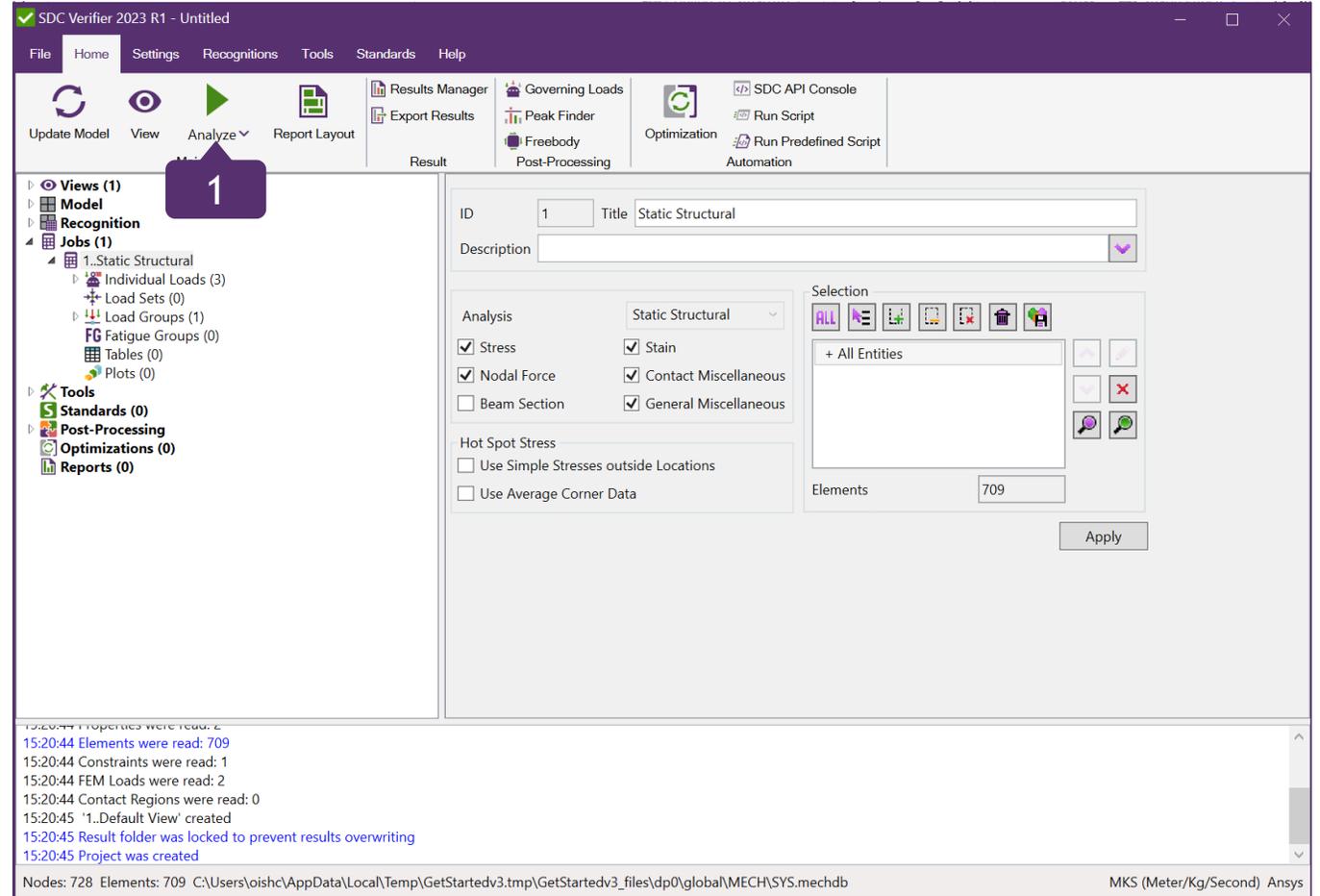
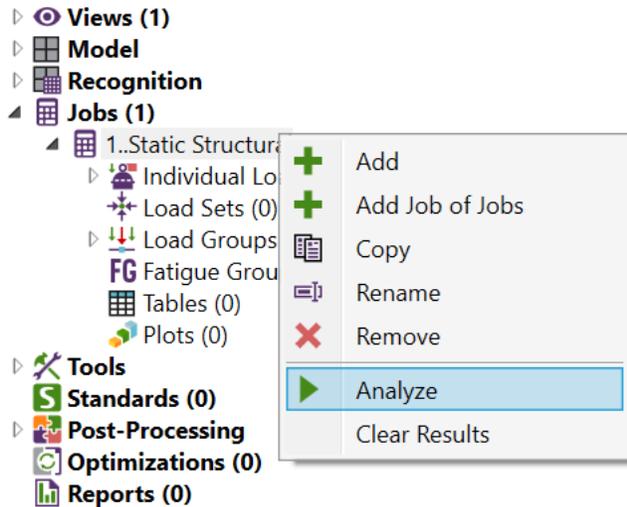
This window also allows to enter Engineer and Customer details that will be displayed in the reports. Can be edited later

1

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

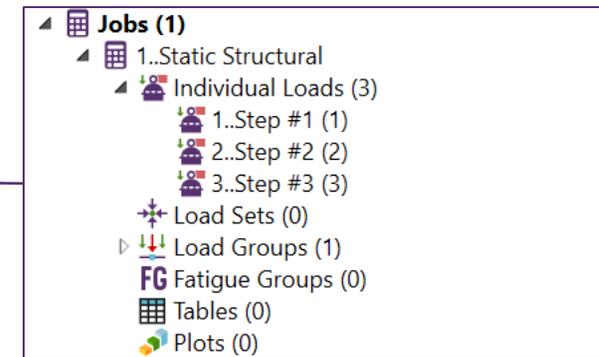
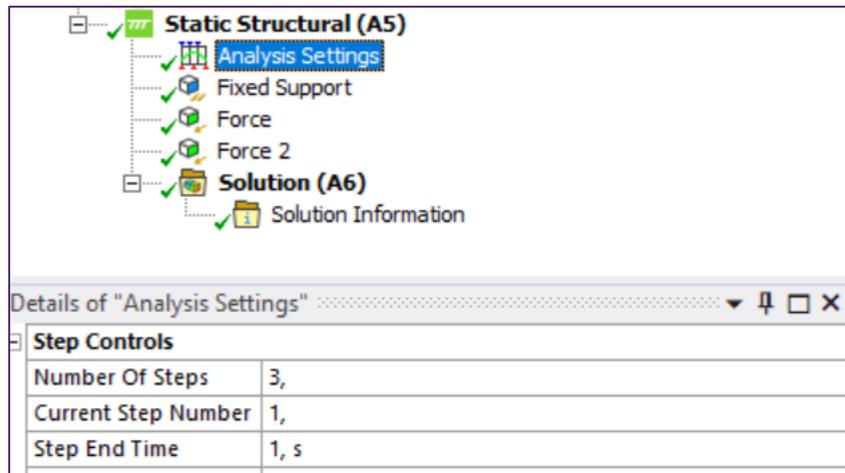


Alternatively, execute right click on the *Active Job* in the *Tree* and press **Analyze**



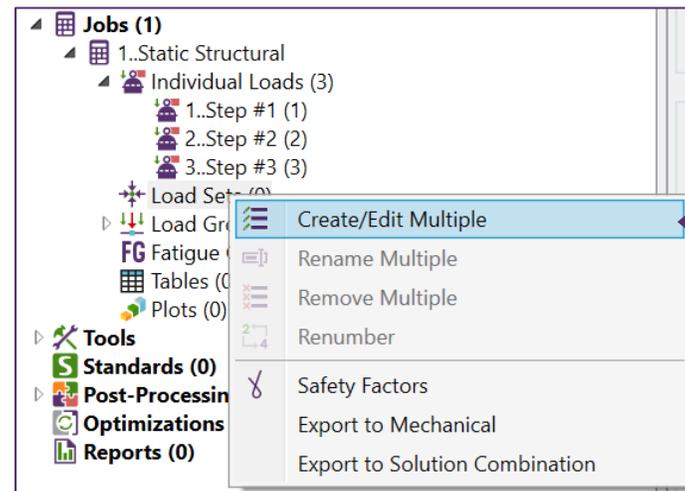
Individual Loads - single load results, each load will be created automatically from steps in Ansys Mechanical.
In our case we have 3 Individual loads.

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



1

Right click on Load Sets > Create/Edit Multiple



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

1

Right click on **Load Groups** > **Create/Edit Multiple**

2

Press Add

3

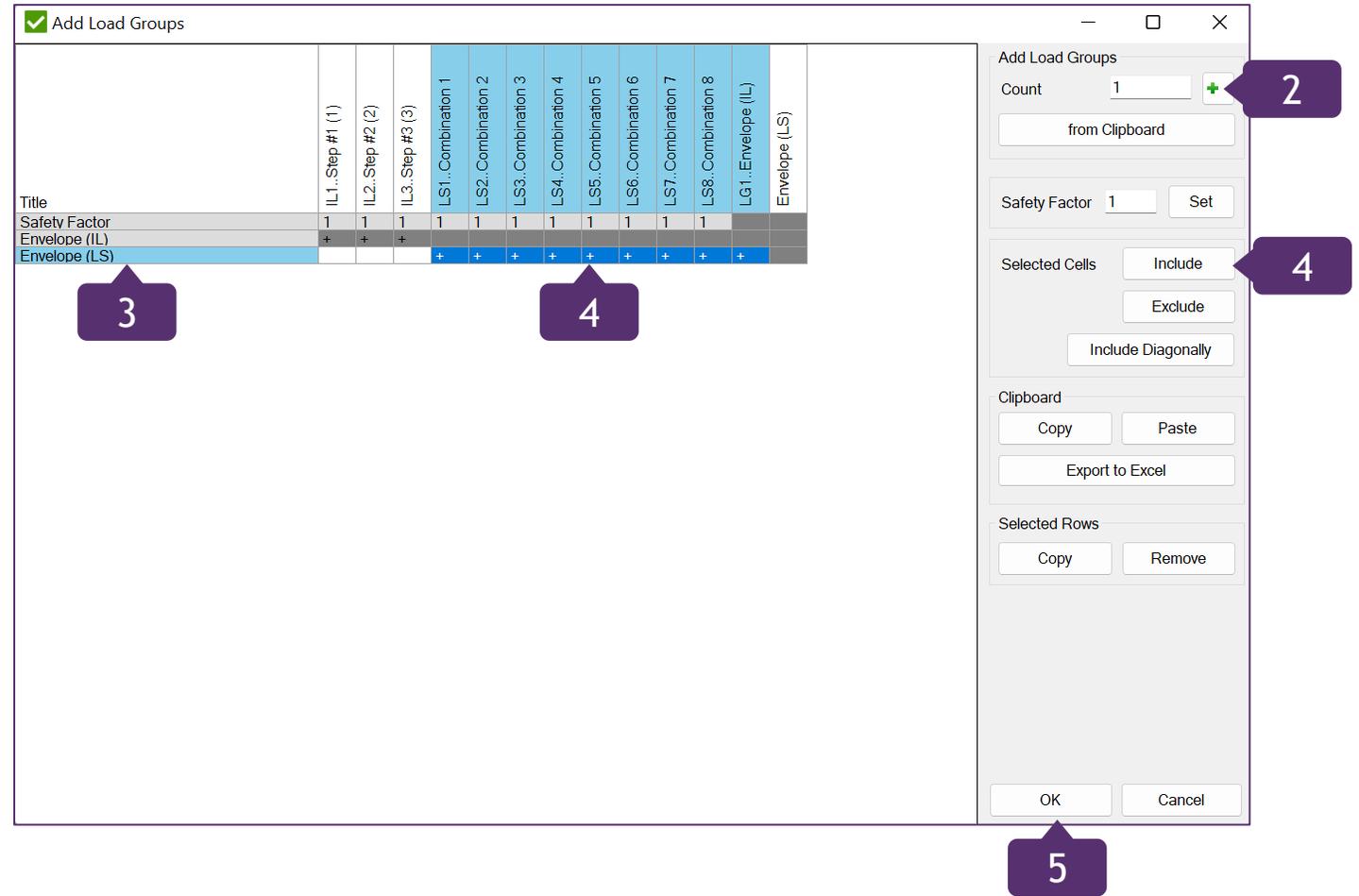
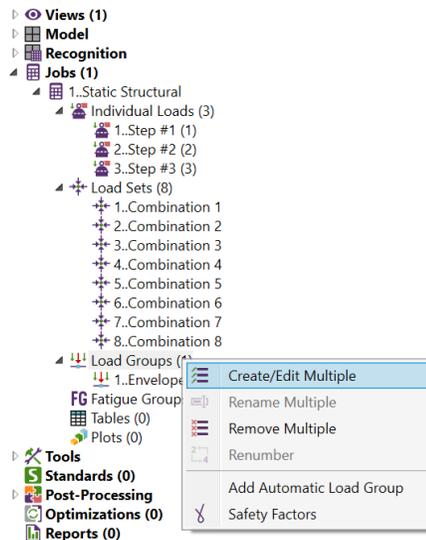
Envelope (IL) Load Group is already existing, Title the new LG: **Envelope (LG)**

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*

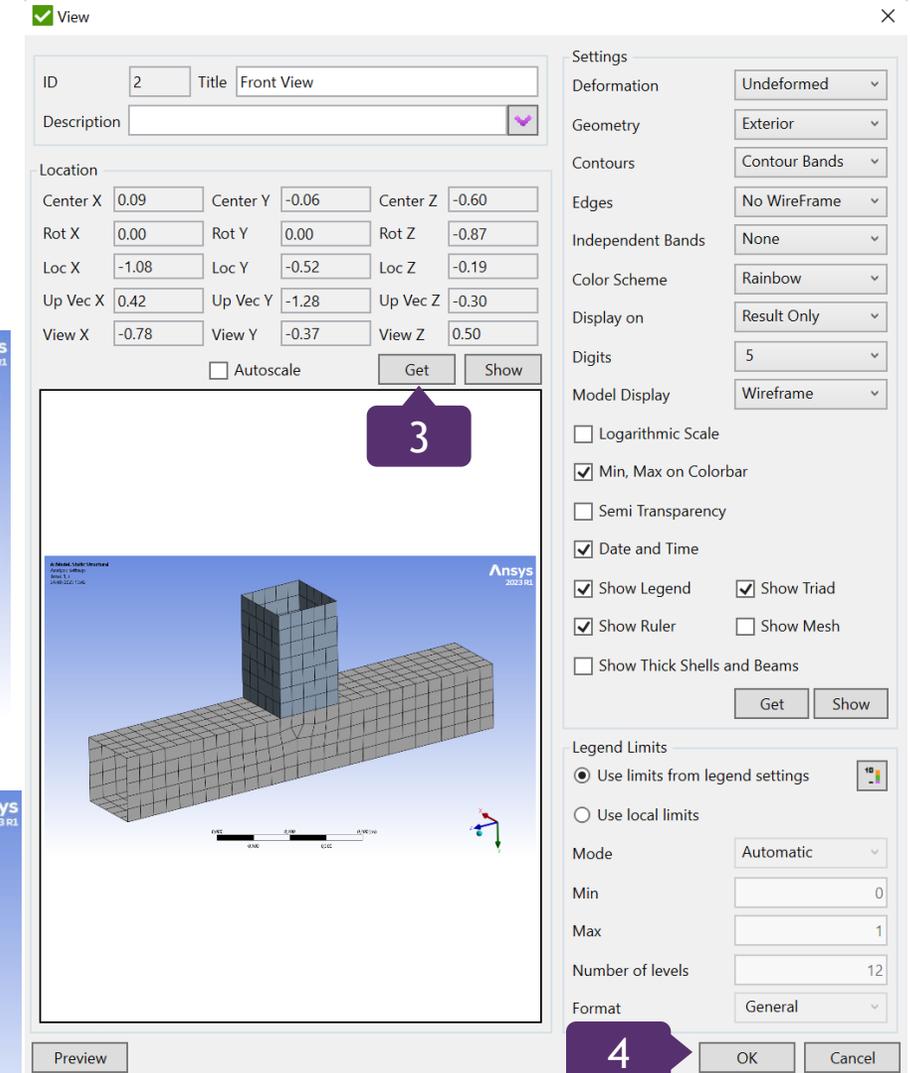
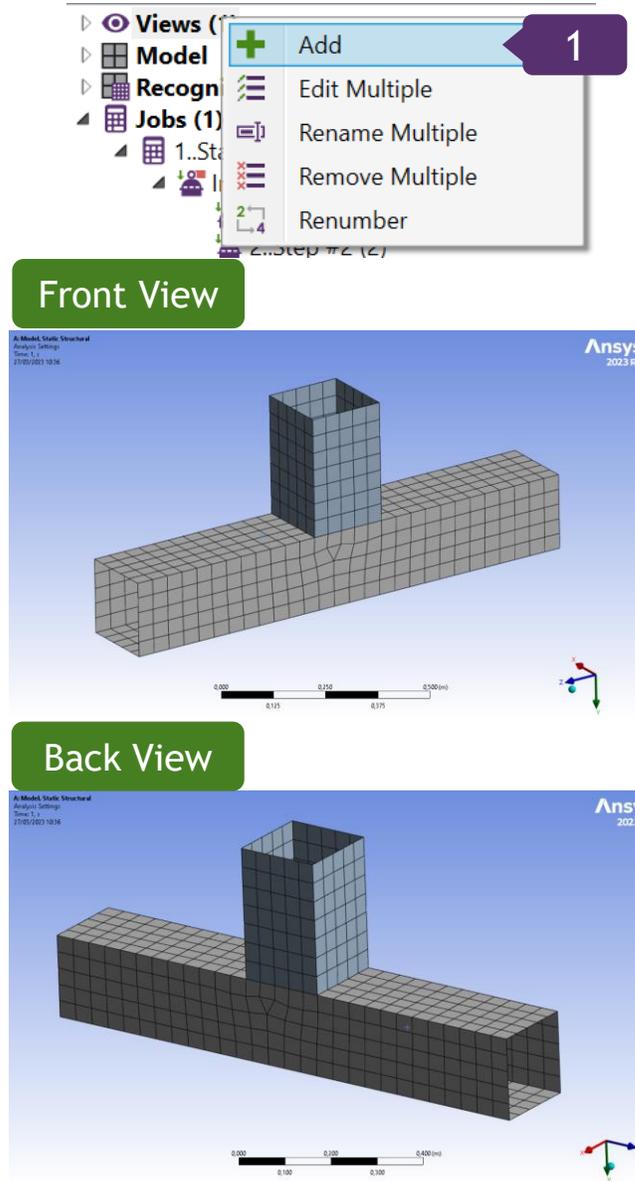
3

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.



Create detailed Views

1

In Views context menu click **Add**

2

Title: *Front Detail View*

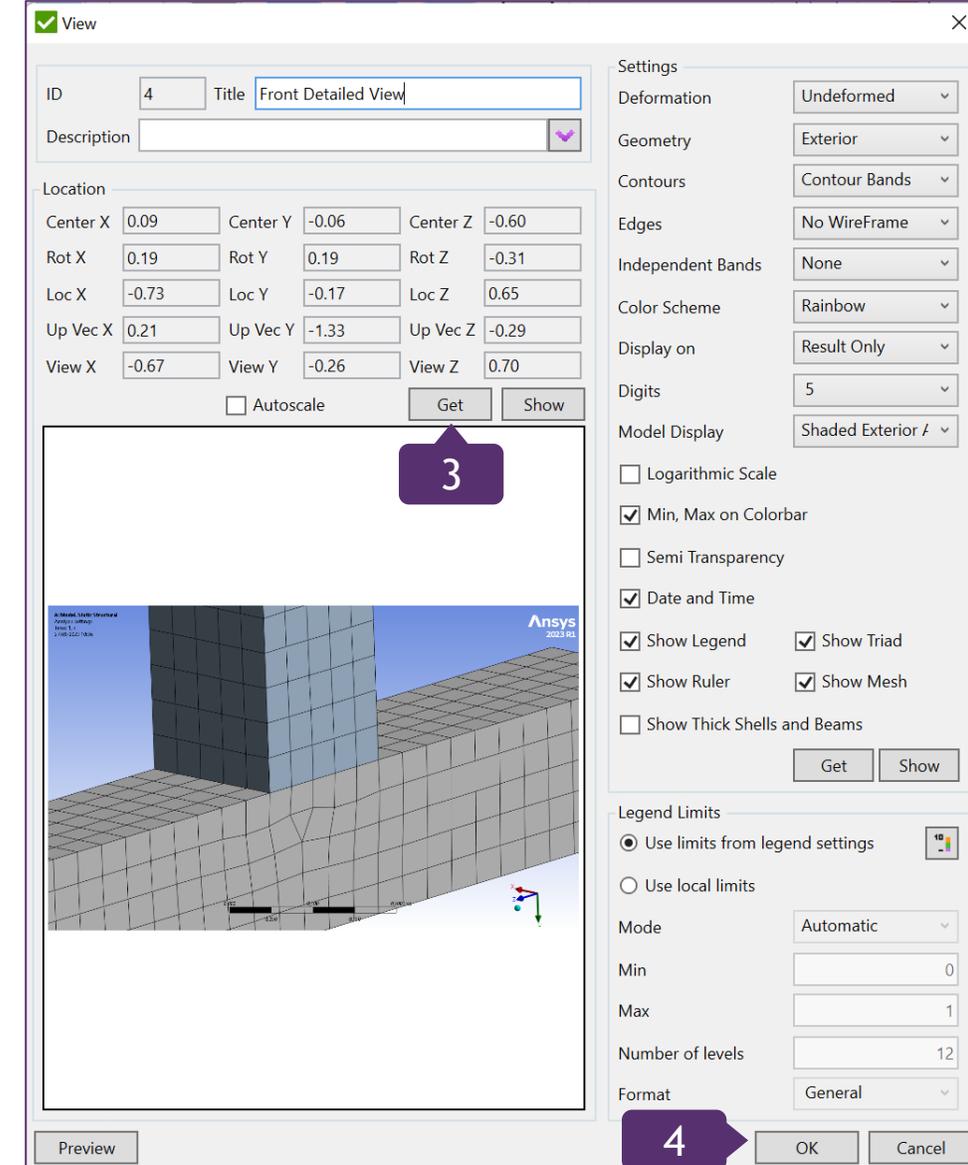
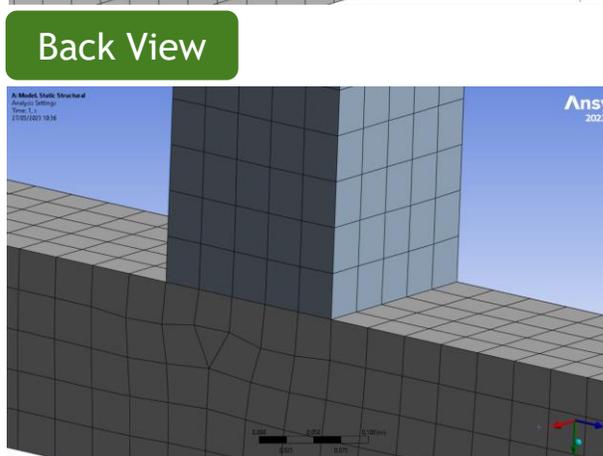
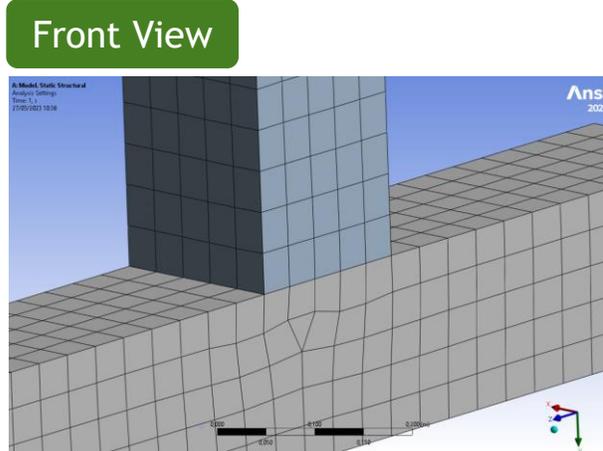
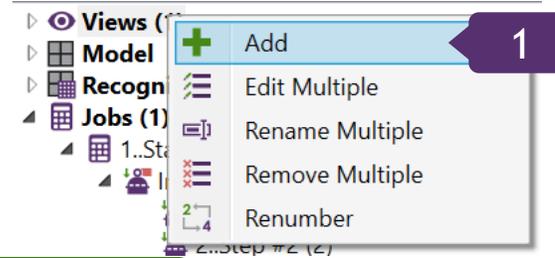
3

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.



Add Displacement Plot

1

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

2

Select Category - *Displacement*
Direction - *Usum*

3

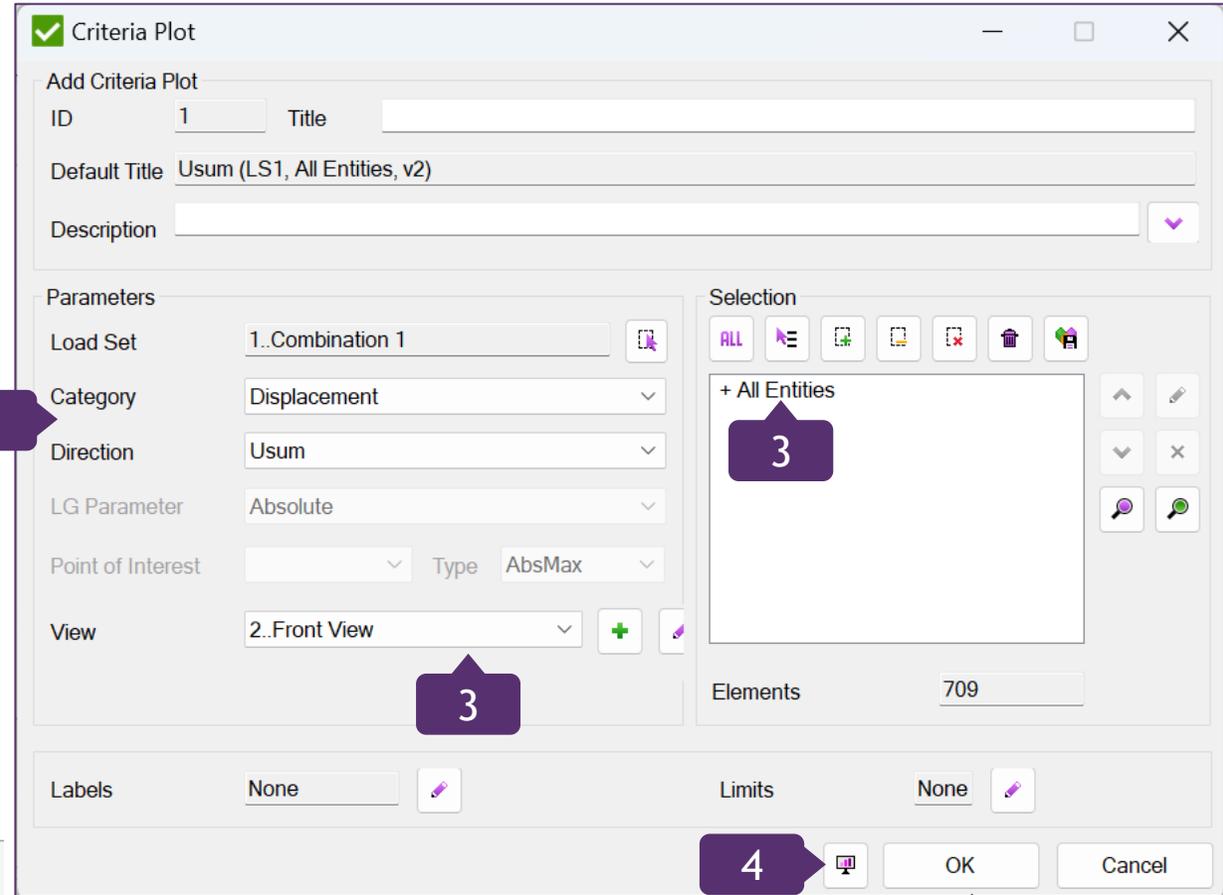
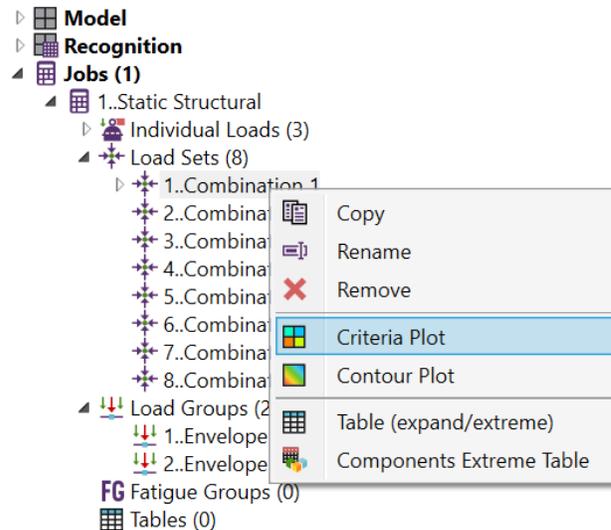
View - *2. Front View*
Selection - *All Entities*

4

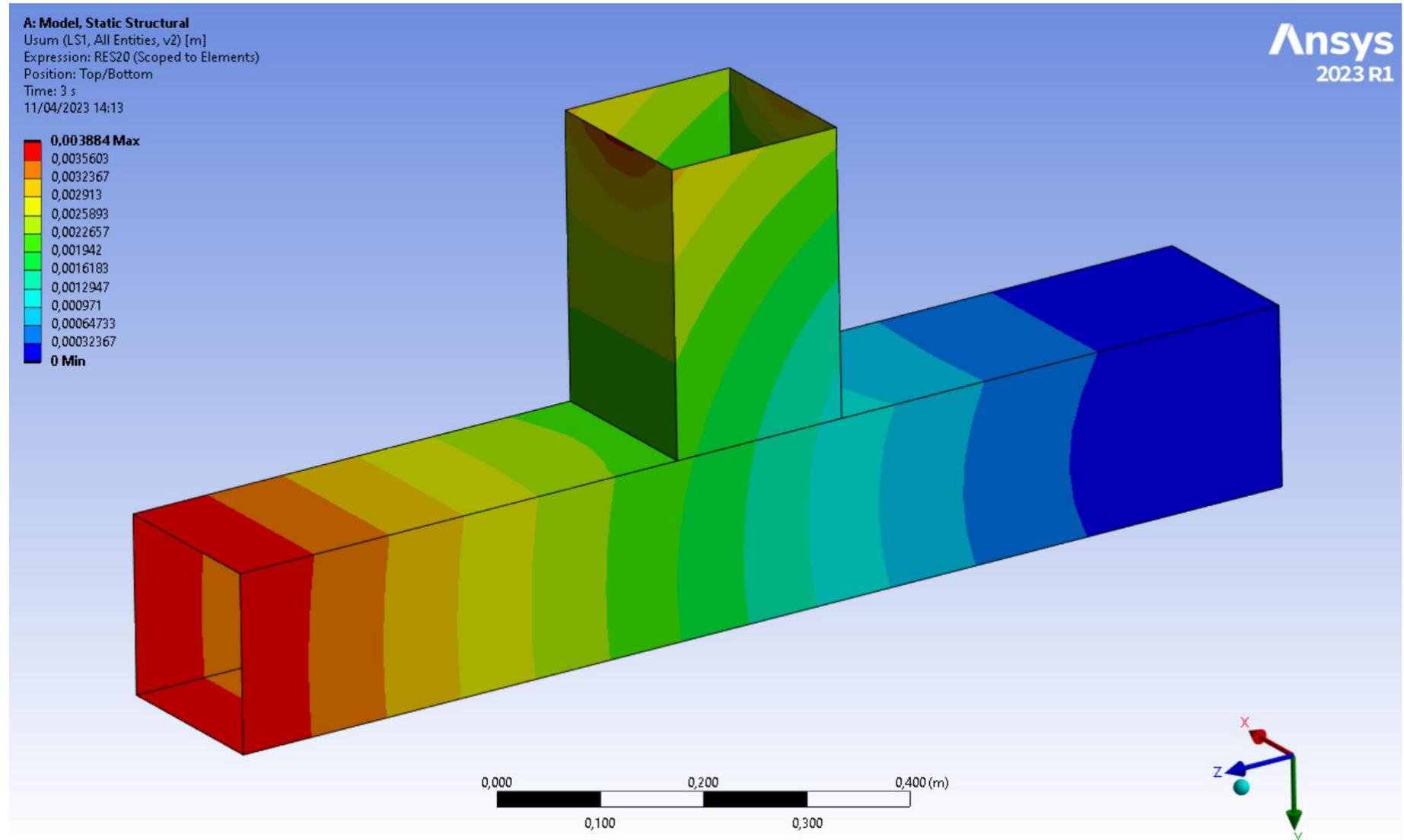
Press **Preview**
(Result on a next slide)

5

Press **OK**



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



1

Right click on **LG2.. Envelope (LS)** > **Contour Plot**

2

Select Category - *Stress*
Direction - *Equivalent*

3

View - *3. Back View*
Selection - *All Entities*

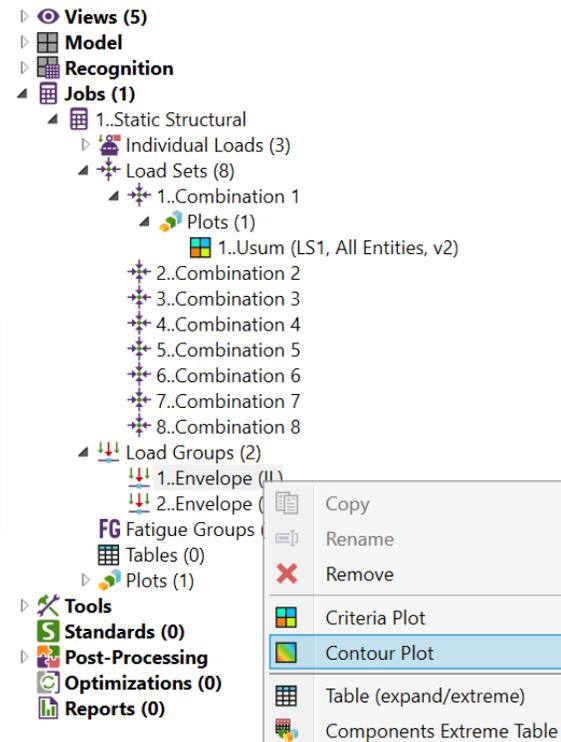
4

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

5

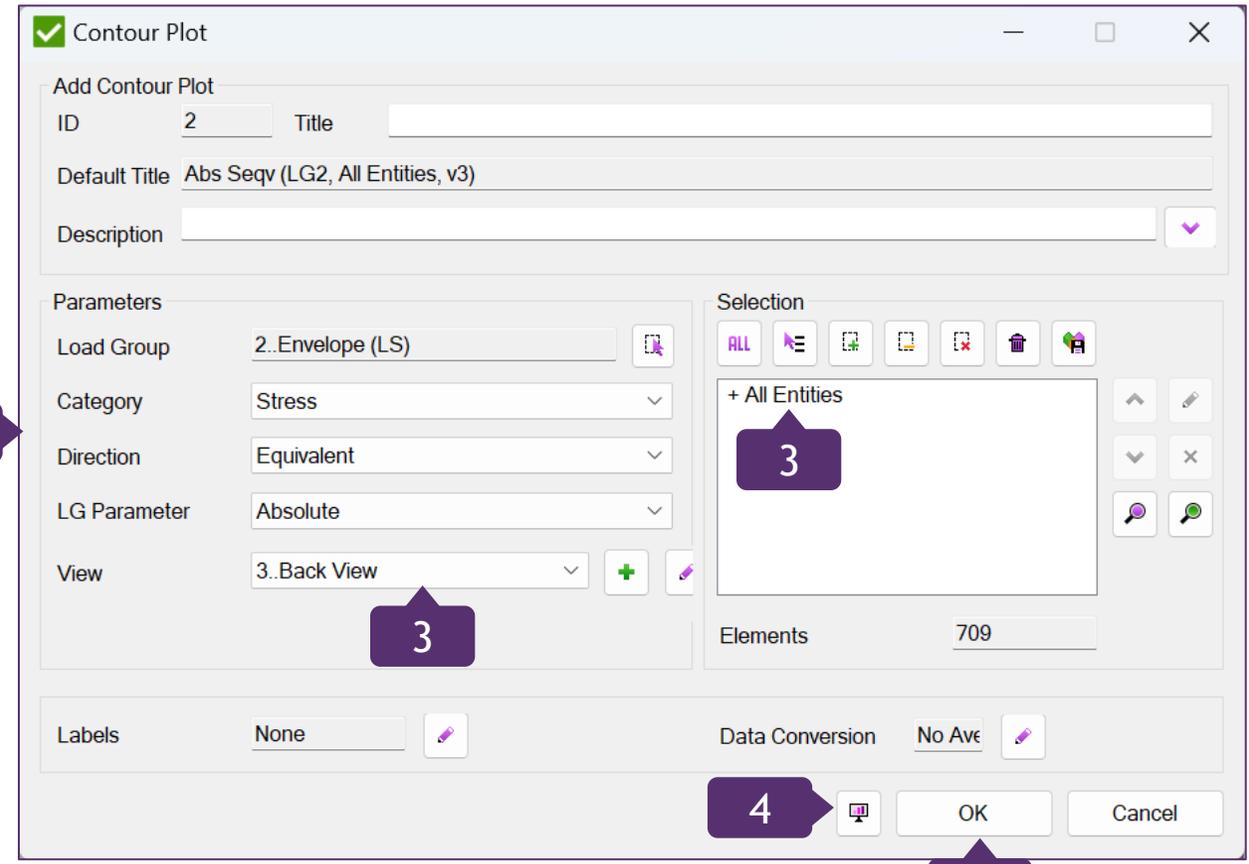
Press **OK**

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



2

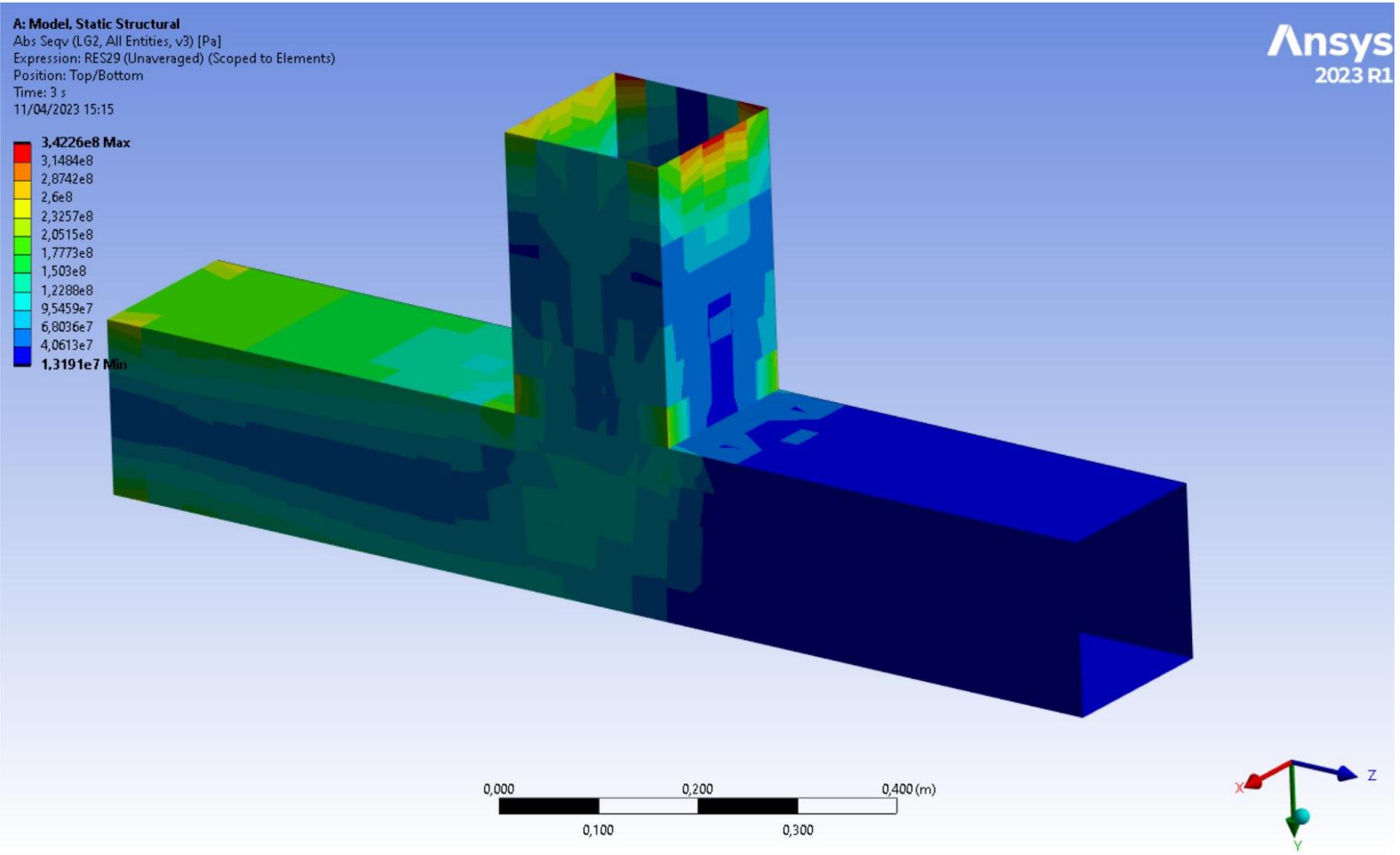
1



3

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**

Views (5)
Model
Recognition
Jobs (1)
 1..Static Structural
 Individual Loads (3)
 Load Sets (8)
 1..Combination 1
 2..Combination 2
 3..Combination 3
 4..Combination 4
 5..Combination 5
 6..Combination 6
 7..Combination 7
 8..Combination 8
 Load Groups (2)
 1..Envelope (IL)
 2..Envelope (LS)
 Plots (1)
 FG Fatigue Groups (0)
 Tables (0)
 Plots (2)
Tools
Standards (0)
Post-Processing
Optimizations (0)
Reports (0)

Context Menu:
Copy
Rename
Remove
Criteria Plot
Contour Plot
Table (expand/extreme)
Components Extreme Table

Table

Get 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 Entities
Elements: 709
Fill Table

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

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.

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 Entities

Elements: 709

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.

Fill Table OK Cancel

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

