



Training

Analysis Option: Normal Modes

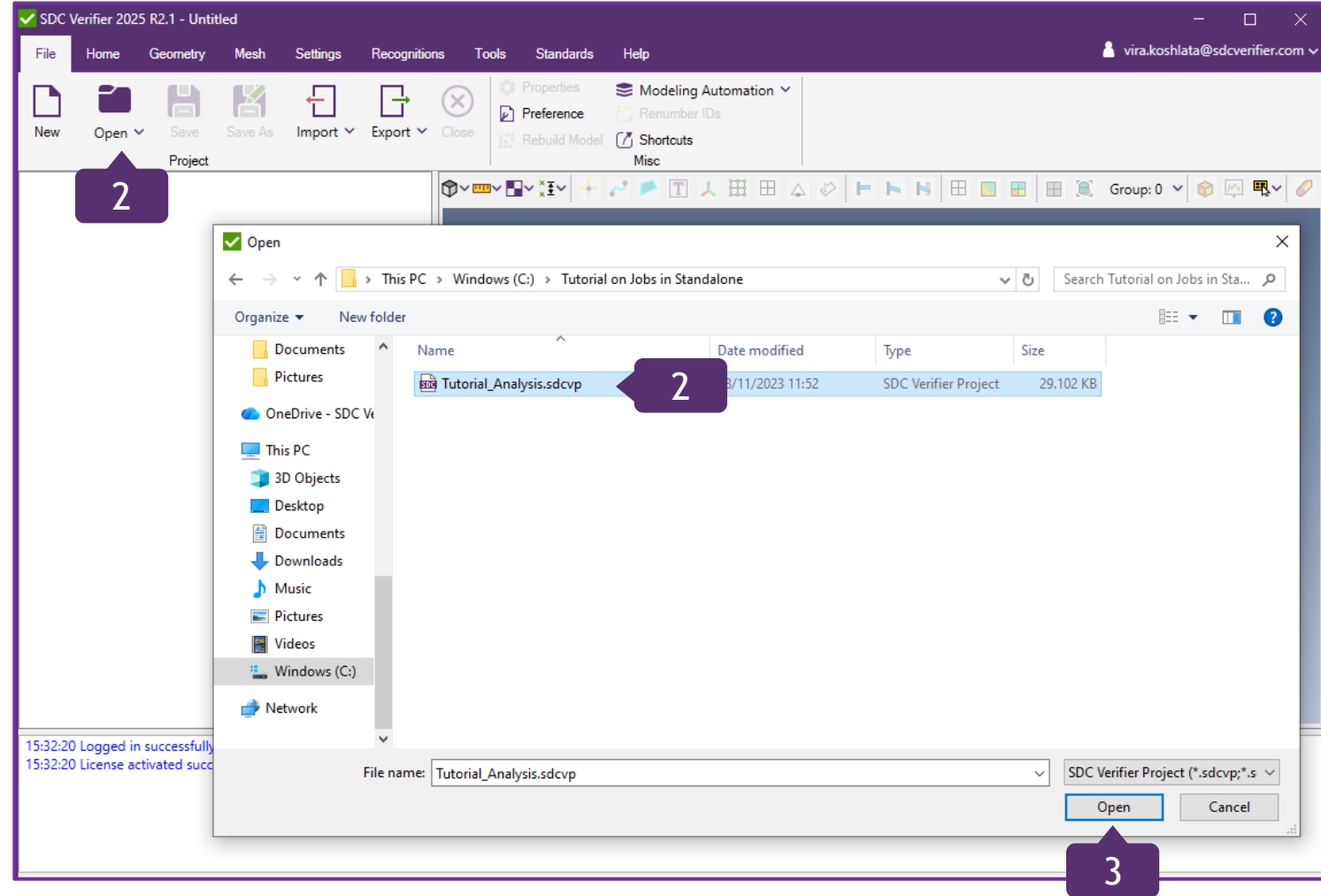
Updated on: December 9th, 2025

Tested with: SDC Verifier 2025 R2.1

- This step-by-step tutorial demonstrates how to add Normal Modes Analysis in SDC Verifier.
- Add Job detailed overview of Options;
- Normal Modes theoretical aspects;
- How to run Analysis;
- How to build Modes Tables (over loads);
- How to build Contour Plots;

Open the Starter Model

- 1 Launch SDC Verifier
- 2 Open project *Tutorial_Analysis.sdcvp*
- 3 Press *Open*



Add Job: Normal Modes

1 Execute right click on *Job (1)* and select *Add*

2 Title: *Normal Modes 1*

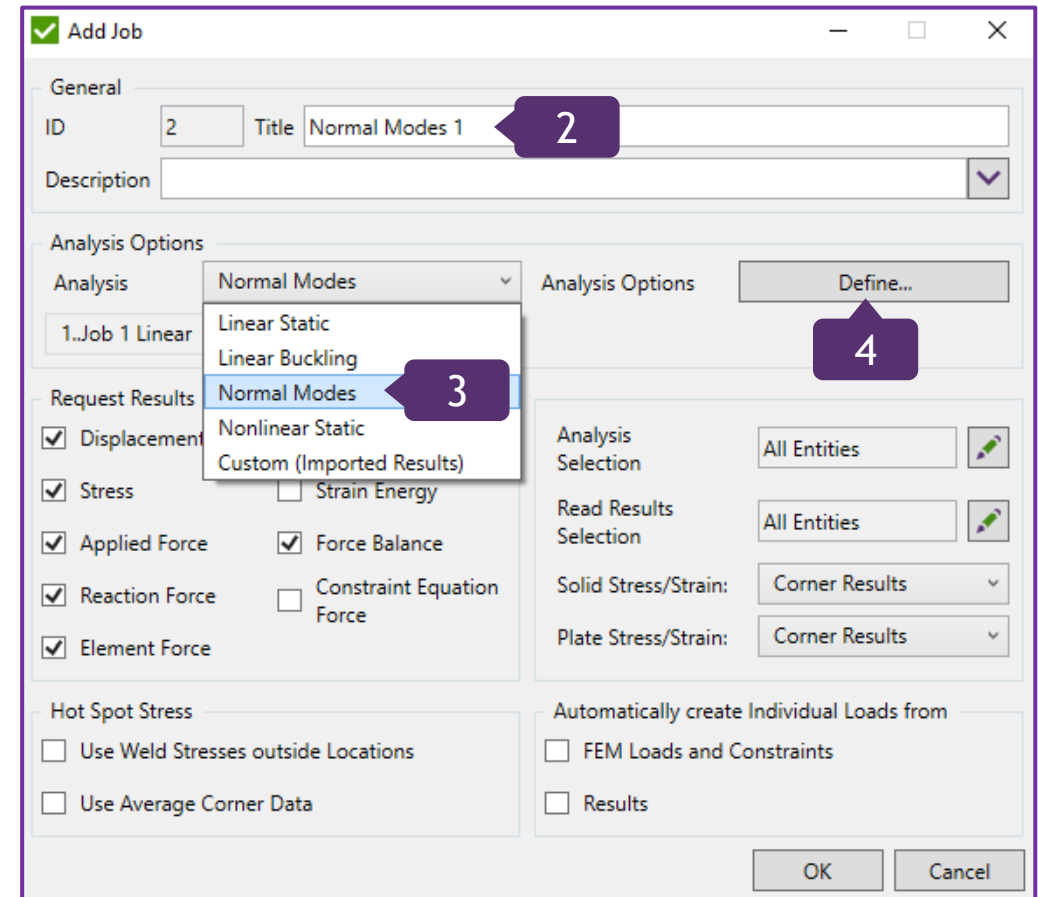
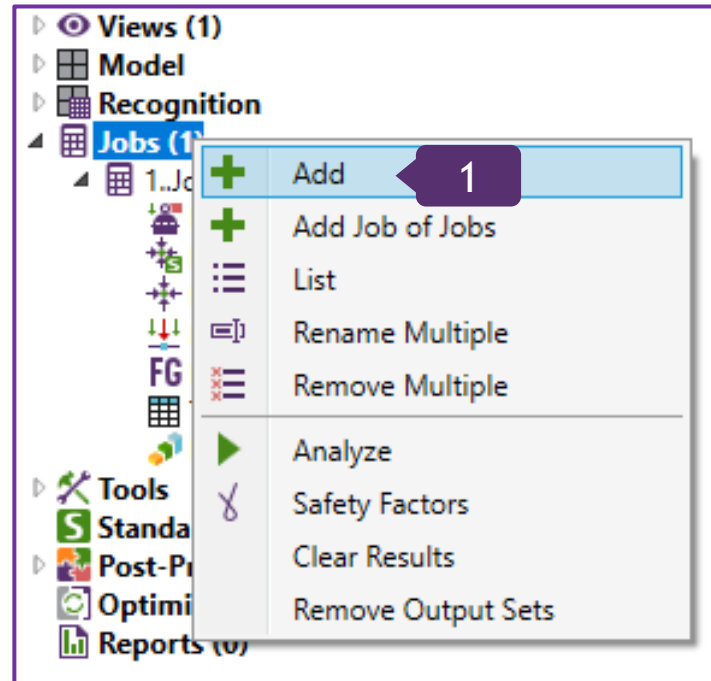
3 Analysis: select *Normal Modes*

4 Press *Define...*

Normal Modes = the natural frequencies and associated deformation patterns of a structure.

They are essential for understanding vibration behavior and for performing any Dynamic Analysis.

Job is the calculation set that contains loads and their combinations to be calculated with the defined calculation options.



Add Job: Normal Modes (Continuation)

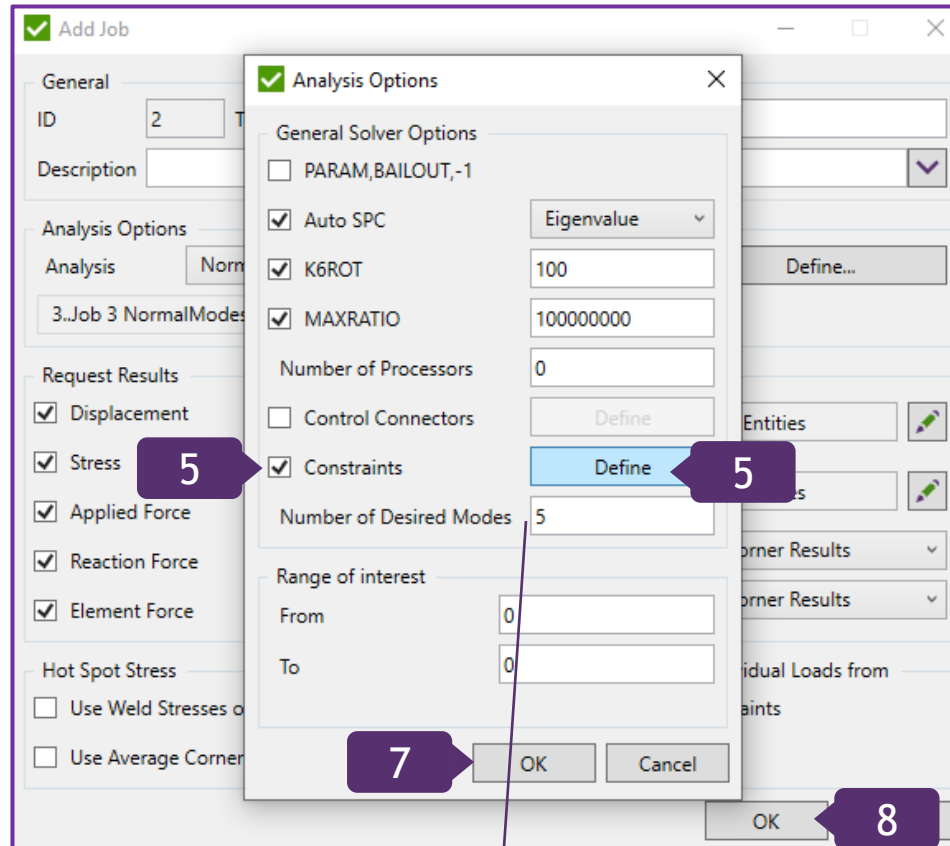
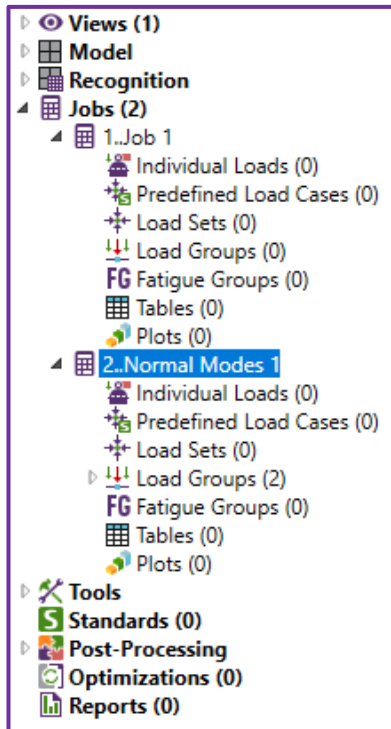
5 Activate *Constraints* and press *Define*

6 Select *1..Pinned* and press *OK*

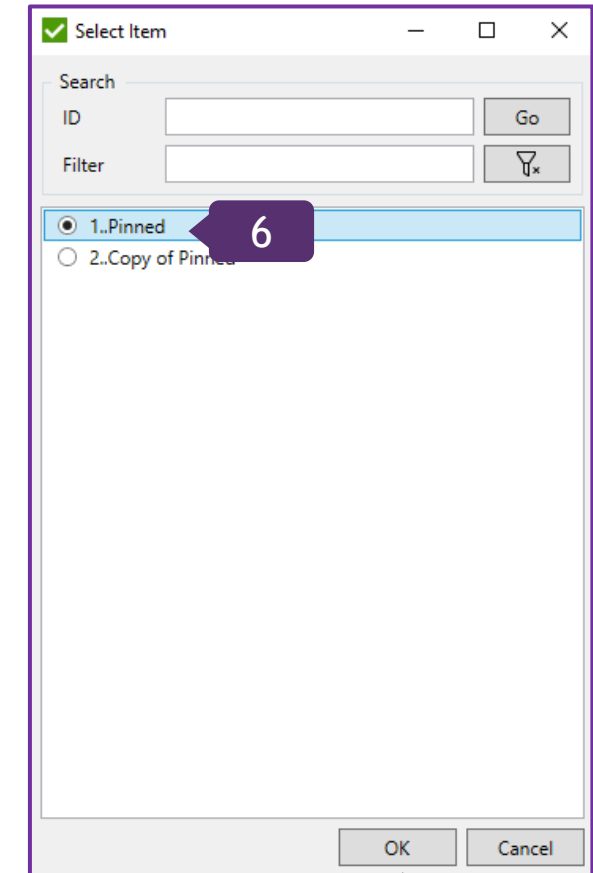
7 Press *OK*

8 Press *OK*

Normal Modes 1
Job added to the
Model tree



A number of Desired Modes can be adjusted
according to User's requirements/objectives.



Add Job

General

ID: 2 Title: Normal Modes 1

Description: [Dropdown]

Analysis Options

Analysis: Normal Modes Analysis Options: Define...

1..Job 1 Linear

Request Results

☒ Displacement ☐ Strain

☒ Stress ☐ Strain Energy

☒ Applied Force ☒ Force Balance

☒ Reaction Force ☐ Constraint Equation Force

☒ Element Force

Hot Spot Stress

☐ Use Weld Stresses outside Locations

☐ Use Average Corner Data

Analysis Selection

Analysis Selection: All Entities

Read Results Selection: All Entities

Solid Stress/Strain: Corner Results

Plate Stress/Strain: Corner Results

Automatically create Individual Loads from

☐ FEM Loads and Constraints

☐ Results

OK Cancel

Request Results allows defining what category of results has to be analyzed. By default, Displacement, Stress, Applied, Reaction, Element Force and Force Balance are requested.

Analysis Selection defines what elements the analysis will be run.

Read Results Selection - the elements for which the results have to be read. If only the portion of the model has to be checked - reduction of the result selection to that portion will improve memory usage and calculation performance.

Solid Stress/Strain - read centroid or corner results for solid elements;

Plate Stress/Strain - read centroid or corner results for plate elements.

Use Weld Stresses outside Locations - display weld stress values for all elements that are not used in Hot Spot Locations defined in Weld Finder. Display 0 if turned off;

Use Average Corner Data - use averaged top/bottom stresses from all attached elements at the element corner where hot spot is calculated.

FEM Loads and Constraints - create Individual Loads for the calculations from the combinations of all FEM Loads and all Constraints;

Results - create the Individual Loads from all existing results by one result per load.

Add Job Overview (Continuation)

To set Nastran options for Analysis press **Define...**

Add Job

General

ID: 2 Title: Normal Modes 1

Description:

Analysis Options

Analysis: Normal Modes Analysis Options: Define...

1..Job 1 Linear

Request Results

☒ Displacement ☐ Strain

☒ Stress ☐ Strain Energy

☒ Applied Force ☒ Force Balance

☒ Reaction Force ☐ Constraint Equation Force

☒ Element Force

Hot Spot Stress

☐ Use Weld Stresses outside Locations

☐ Use Average Corner Data

Analysis Selection: All Entities

Read Results Selection: All Entities

Solid Stress/Strain: Corner Results

Plate Stress/Strain: Corner Results

Automatically create Individual Loads from

☐ FEM Loads and Constraints

☐ Results

OK Cancel

PARAM BAILOUT -1 - used to allow under constrained models continue to run if any mechanism (free motion) is detected.

Analysis Options

General Solver Options

☐ PARAM,BAILOUT,-1

☒ Auto SPC Eigenvalue

☒ K6ROT 100

☒ MAXRATIO 100000000

Number of Processors: 0

☐ Control Connectors Define

☒ Constraints Define

Number of Desired Modes: 5

Range of interest

From: 0 To: 0

OK Cancel

AUTOSPC - allows to control the format of the AUTOSPC command to the Nastran convention (PARAM, AUTOSPC, YES). In Simcenter Nastran there is a choice to use the default method (0..Eigenvalue) or a method available in NX Nastran version 4.1 and higher (1..Singular Value Decomposition). If you use the SVD method, SDC will write a system cell to Simcenter Nastran (SVDSPC=1).

K6ROT - when activated, writes PARAM,K6ROT,(value). Specifies the stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. Default value is 100.0 and parameter is ignored by CQUADR, CTRIAR, CQUAD8, and CTRIA6 elements.

MAXRATIO - when activated, writes PARAM,MAXRATIO,(value). The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (i.e., has mechanisms). Default value is 1.0E7.

Range of Interest - frequency range of interest, *From* = Min, *To* = Max.

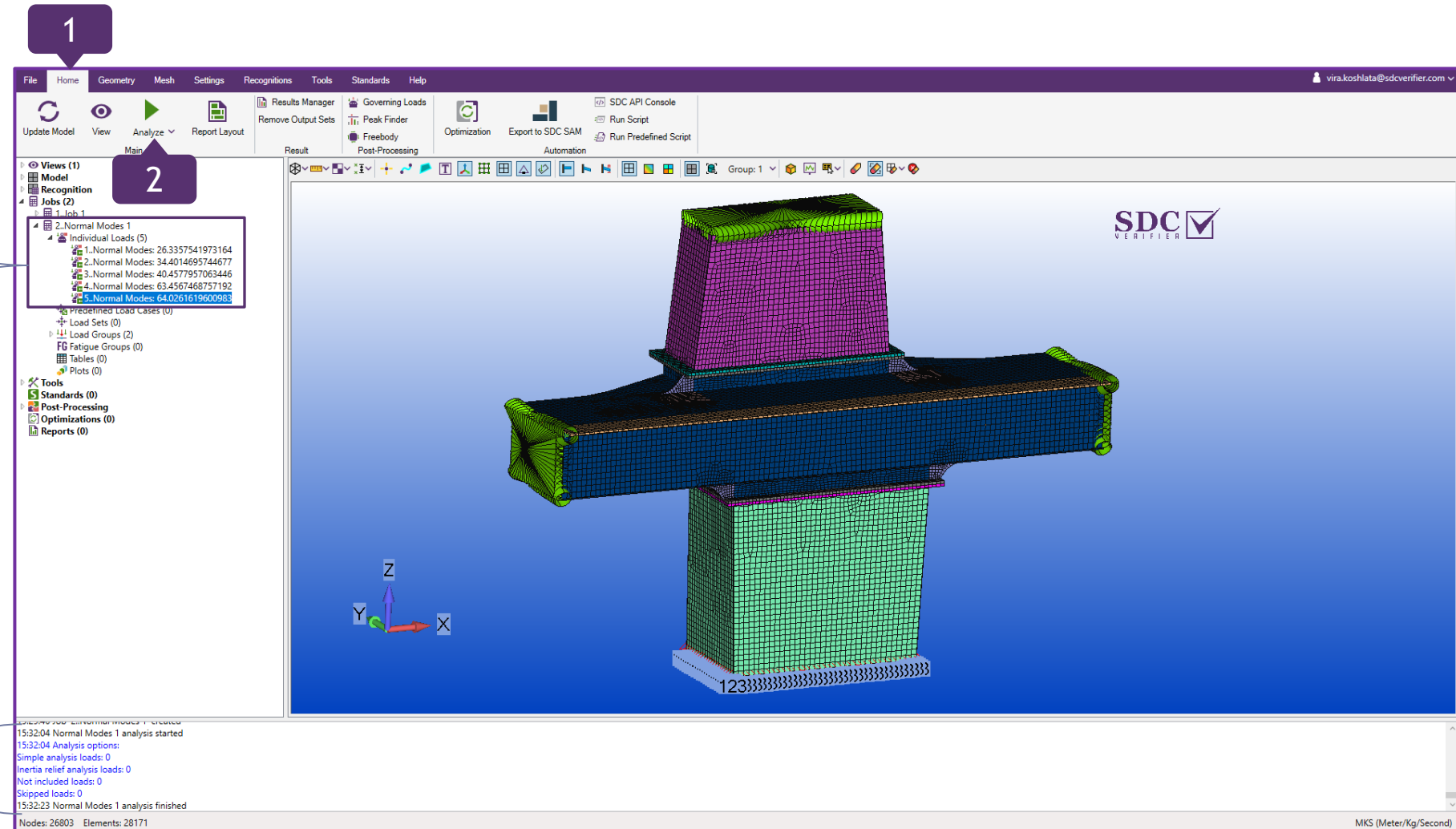
Constraints - For Normal Modes analysis, it is essential to define them. Otherwise, the results will be unrealistic.

1 Go to *Home* section on the Ribbon

2 Press  to analyze Job

Result: 5 Individual Loads/Normal Modes with frequencies were created based on a number (5) of Desired Modes.

Normal Modes 1 analysis started and finished

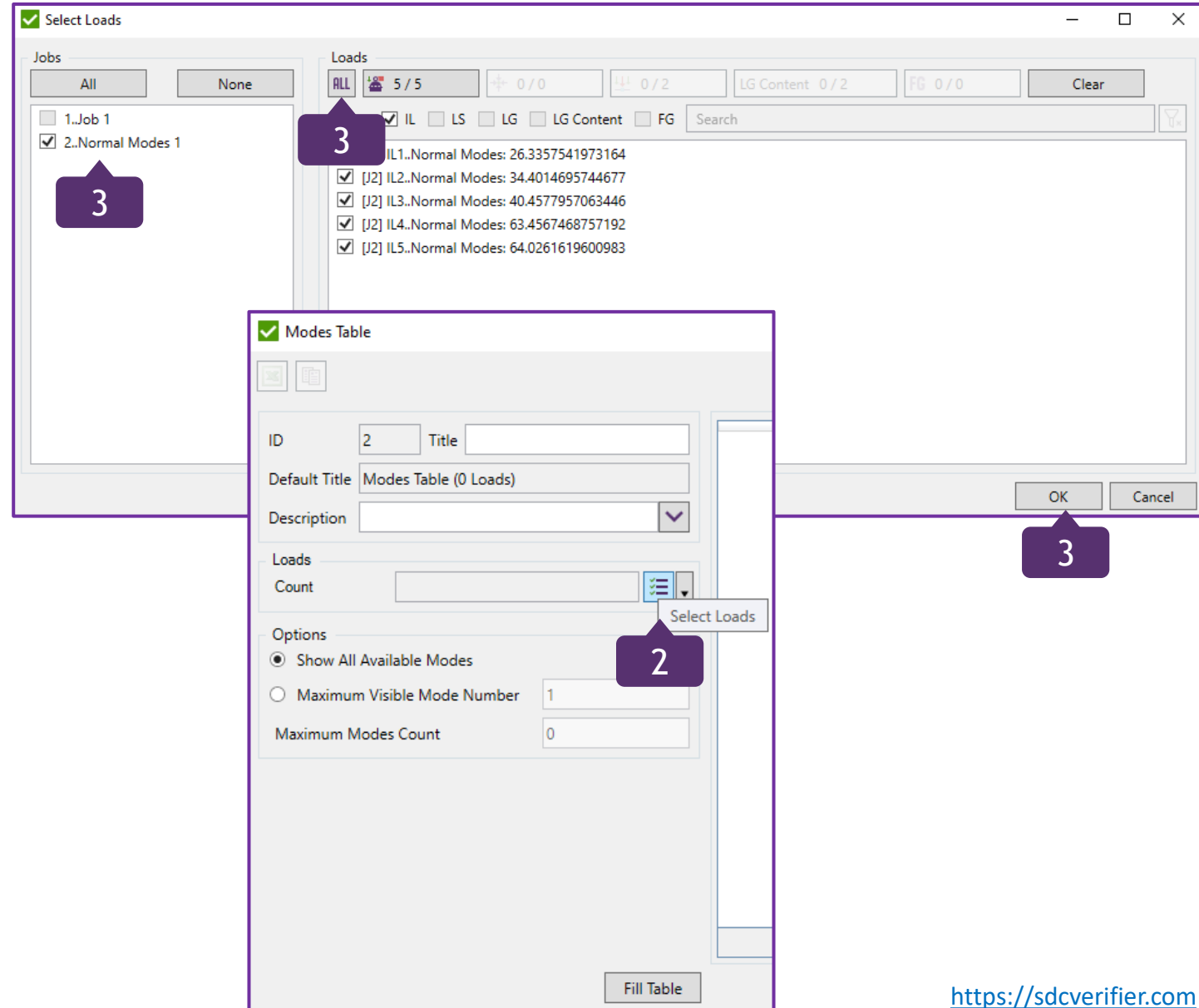
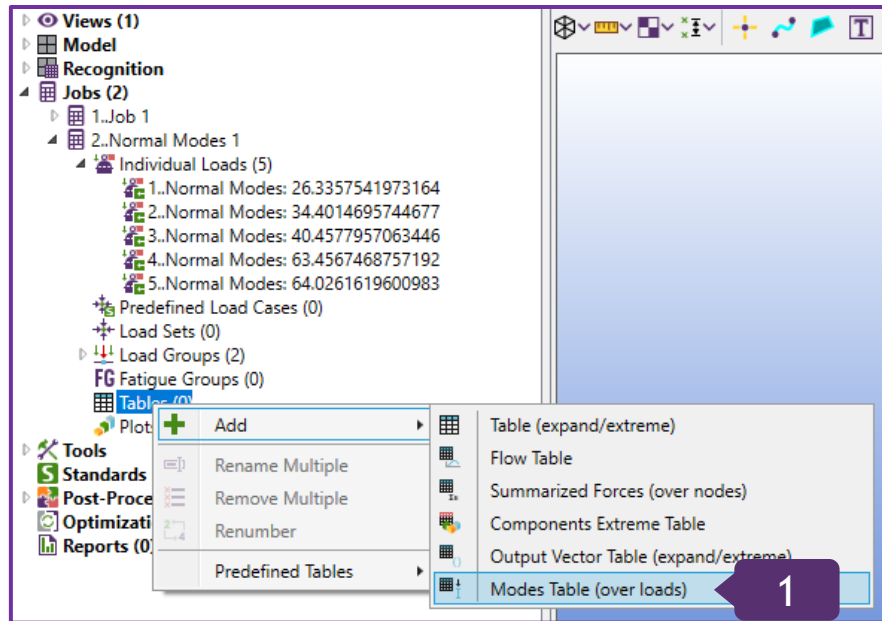


Build Modes Table (over loads)

1 Execute right click on Tables → Add → Modes Table (over loads)

2 In Loads Count, select Loads

3 2.. Normal Modes 1 → Press All; Press OK



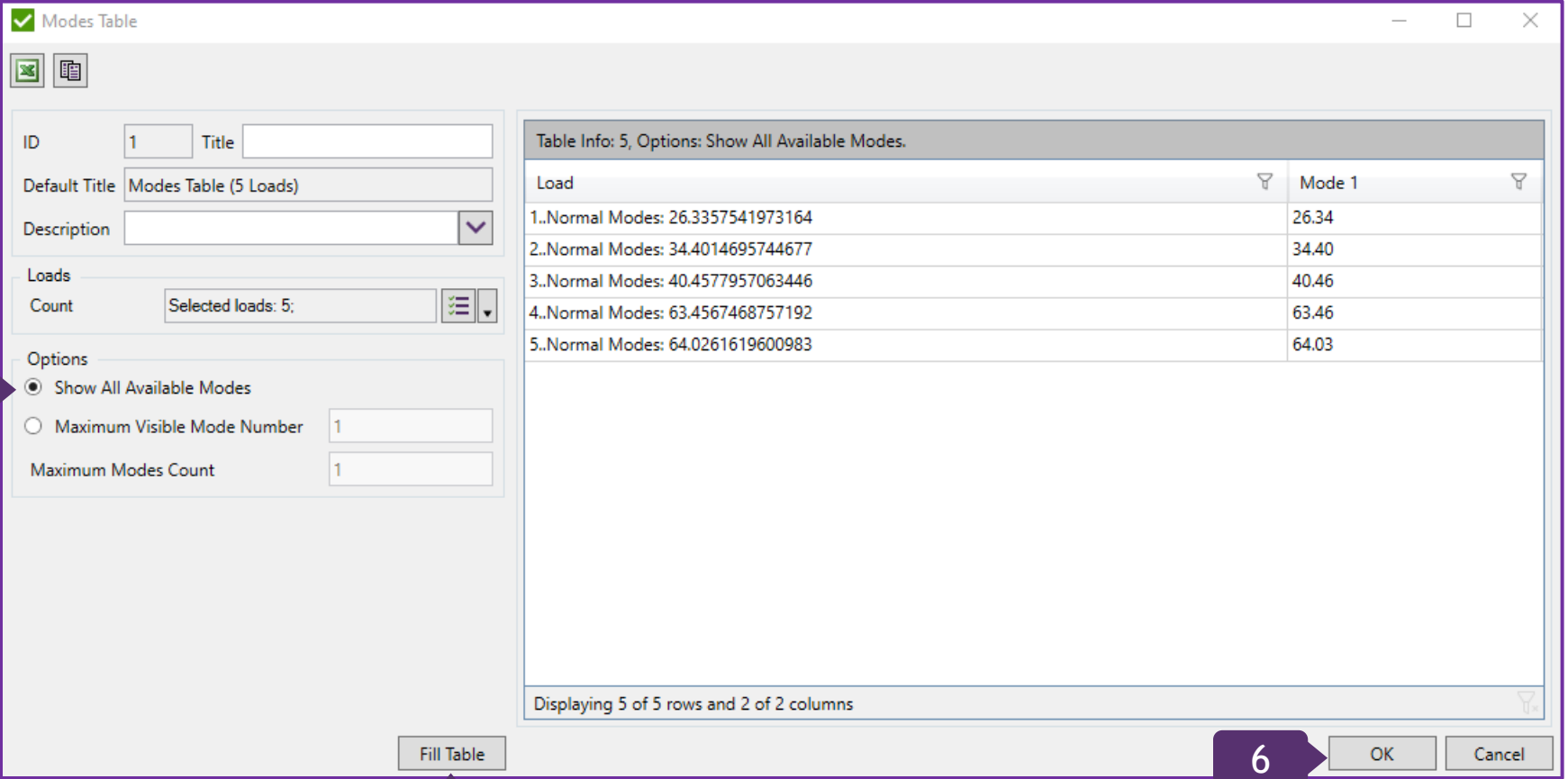
Build Modes Table (over loads) (Continuation)

4 Show All Available Modes: *ON*

5 Press *Fill Table*

6 Press *OK*

4



✓ Modes Table

ID 1 Title

Default Title Modes Table (5 Loads)

Description

Loads

Count Selected loads: 5;

Options

☒ Show All Available Modes

☐ Maximum Visible Mode Number 1

Maximum Modes Count 1

Fill Table

6

Load	Mode 1
1..Normal Modes: 26.3357541973164	26.34
2..Normal Modes: 34.4014695744677	34.40
3..Normal Modes: 40.4577957063446	40.46
4..Normal Modes: 63.4567468757192	63.46
5..Normal Modes: 64.0261619600983	64.03

Displaying 5 of 5 rows and 2 of 2 columns

OK Cancel

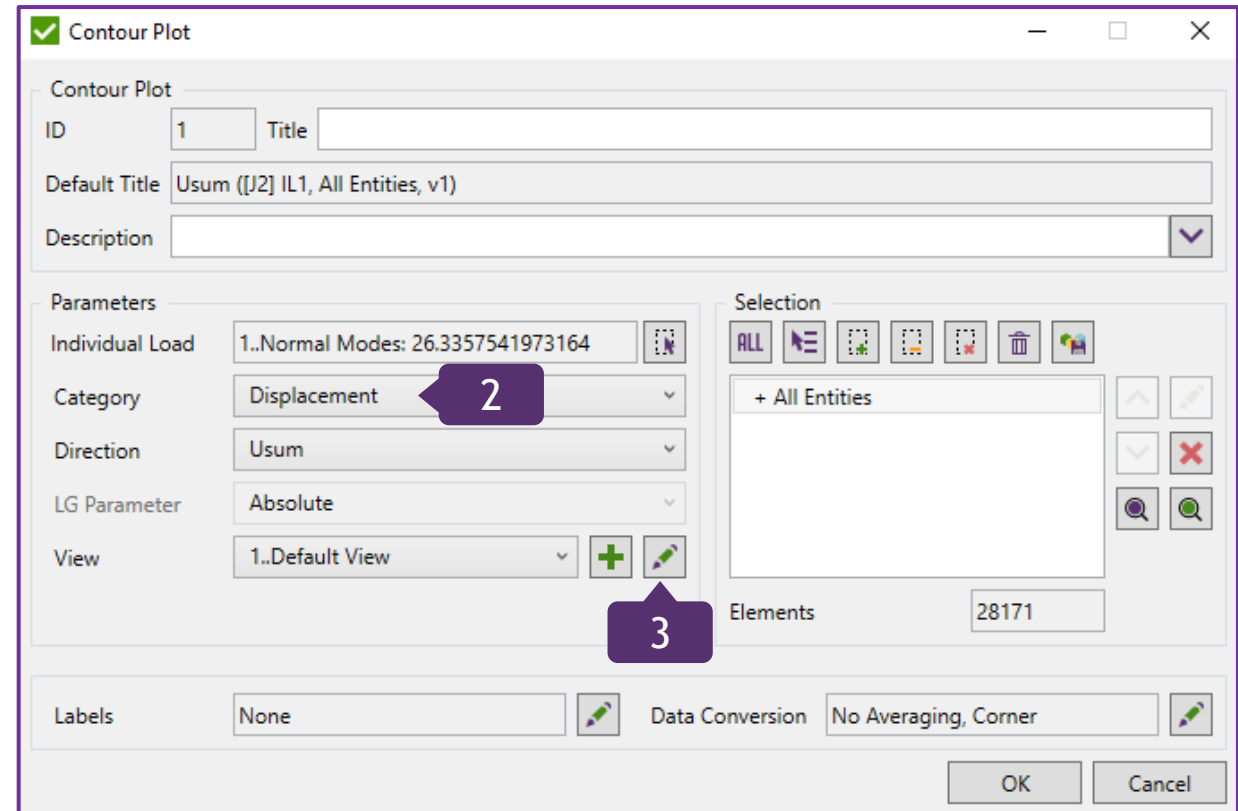
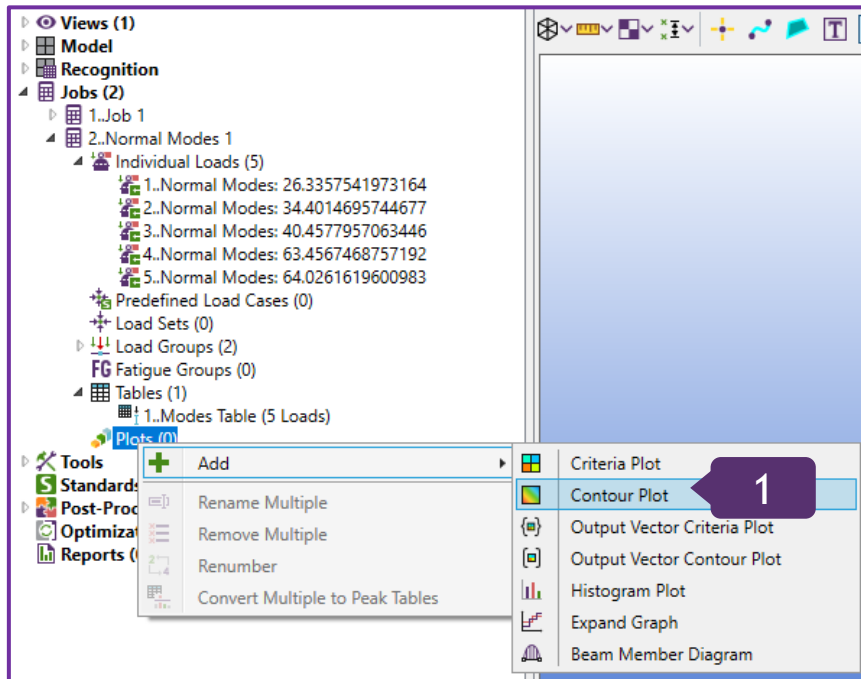
5

Add Contour Plot

1 Execute right click on Plots → Add → Contour Plot

2 Category: Displacement

3 In View, press edit button



Add Contour Plot (Continuation)

4

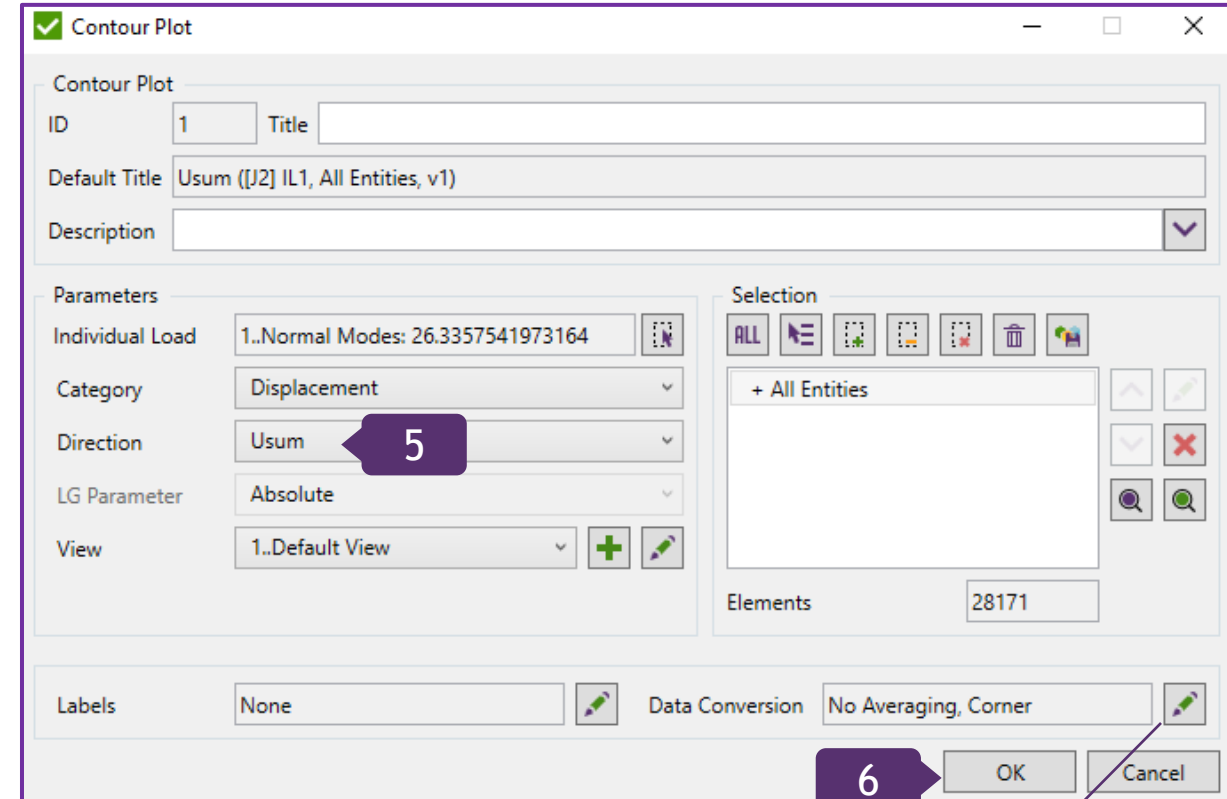
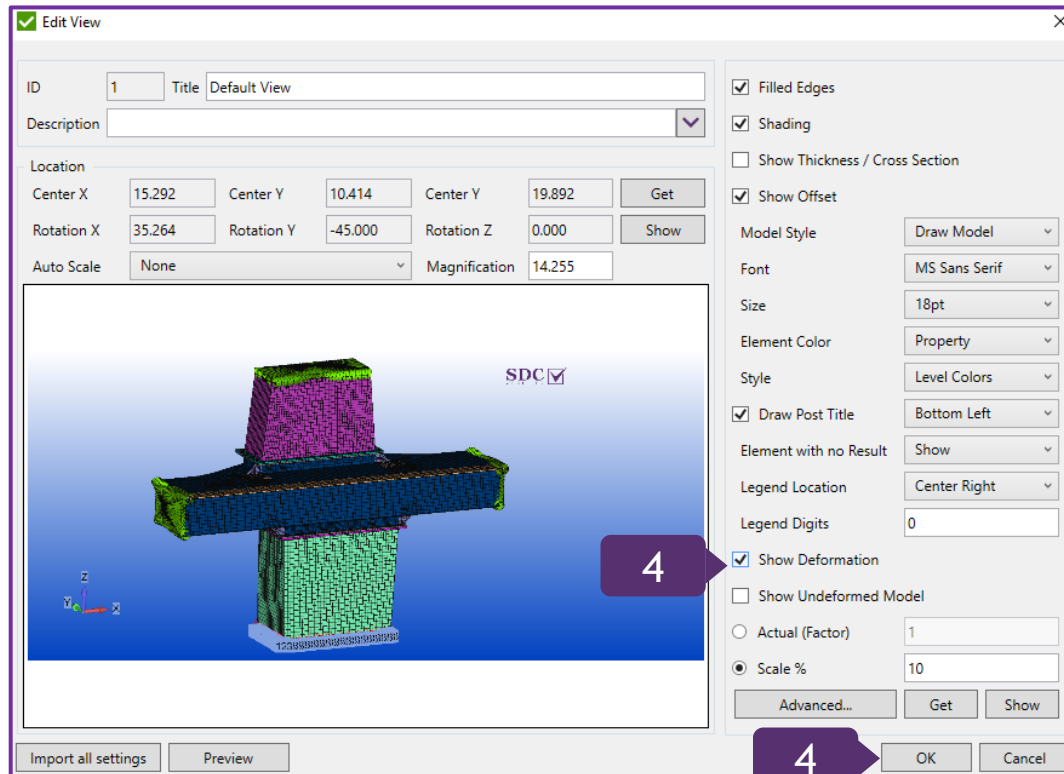
Show Deformation: *ON*;
Press *OK*

5

Direction: *Usum*

6

Press *OK*



Possible to adjust according to
User's requirements.

