Pre/Post Web App Workshop - Normal Modes, Statics, and Frequency Response Analysis

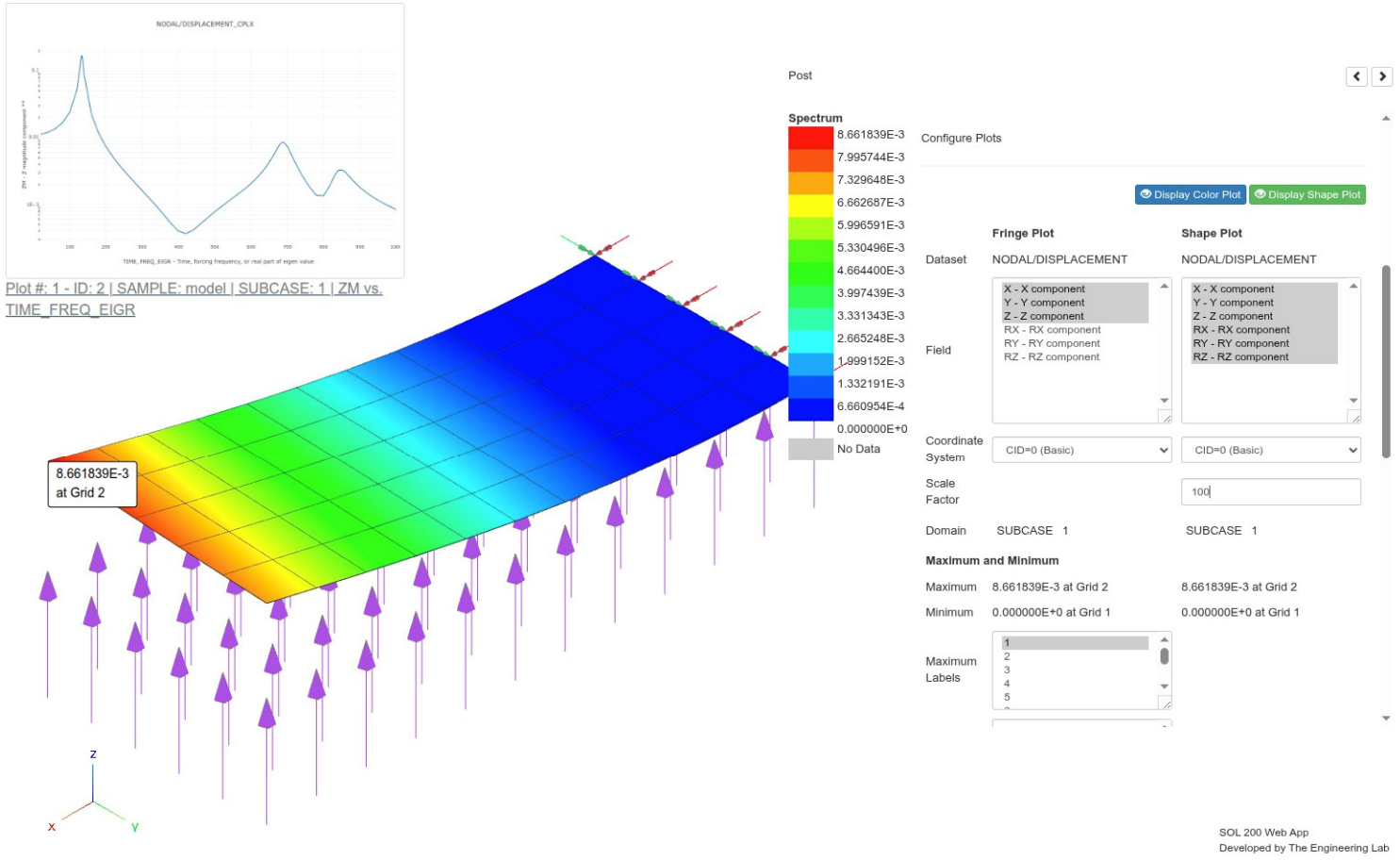
AN MSC NASTRAN TUTORIAL

Questions? Email: christian@ the-engineering-lab.com

Goal: Configure a normal modes, statics, and frequency response analysis with MSC Nastran and review results

Agenda

- Units
- Meshing
- Supports
- Normal Modes Analysis
- Linear Statics Analysis
- Saving Progress
- Frequency Response Analysis



Details of Analysis Model

- This exercise is a recreation of workshop 6 from the NAS 102 training course. A description of the analysis model is shown to the right.

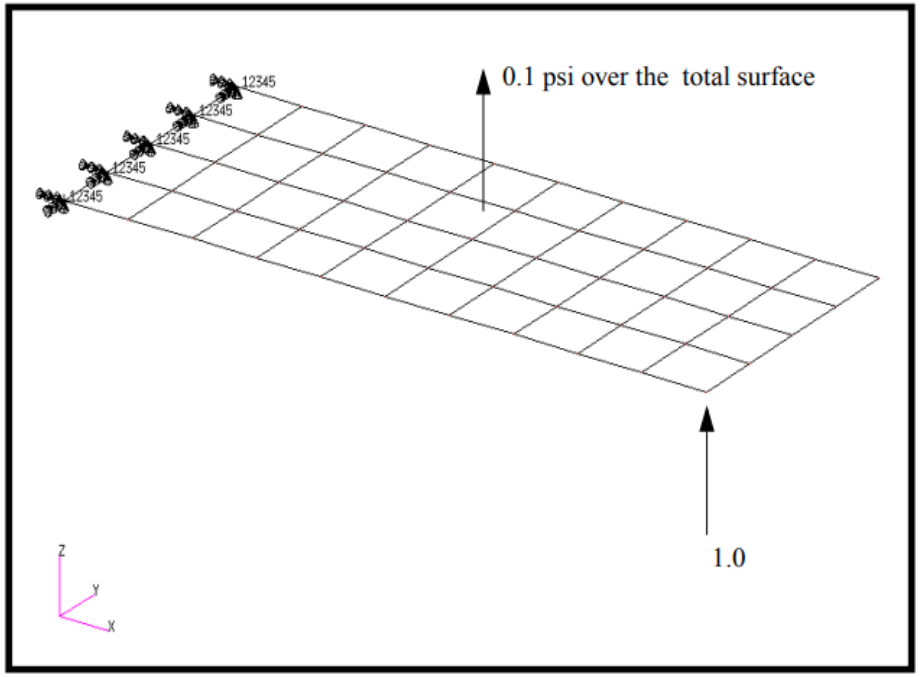
WORKSHOP 6 *Modal Frequency Response Analysis*

Model Description:

Using the modal method, determine the frequency response of the flat rectangular plate, created in Workshop 1, excited by a 0.1 psi pressure load over the total surface of the plate and a 1.0 lb. force at a corner of the tip lagging 45°. Use a modal damping of $\xi = 0.03$. Use a frequency step of 20 hz between a range of 20 and 1000 hz; in addition, specify five evenly spaced excitation frequencies between the half power points of each resonant frequency between the range of 20-1000 hz.

Below is a finite element representation of the flat plate. It also contains the loads and boundary constraints.

Figure 6.1-Loads and Boundary Conditions



Bulk Data File

This workshop will detail how to construct the case control section and each bulk data entry shown in the images to the right.

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

model.bdf

```
1  SOL 111
2  CEND
3  SET 1 = 2,3,16
4  $ Dynamic Solution Conditions
5  FREQUENCY=1
6  METHOD=1
7  SDAMPING=2
8  $ Output Control
9  ECHO=NONE
10 $ Physical Set Output Requests
11 DISPLACEMENT(SORT2)=1
12 $ Loads and BCs
13 SPC=1
14 SUBCASE 1
15   $ Loads and BCs
16   DLOAD=9001
17 BEGIN BULK
```

M O D E L S U M M A R Y

ENTRY NAME	NUMBER OF ENTRIES
CQUAD4	55
DLOAD	1
EIGRL	1
FORCE	1
FREQ1	1
FREQ4	1
GRID	72
HDF5OUT	1
MAT1	1
PARAM	2
PLOAD2	1
PSHELL	1
RLOAD2	2
SPC1	1
TABDMP1	1
TABLED1	1

Efficient Strategy to Building Nastran Bulk Data Files (BDF)

A. Start with a BDF template

B. For any Nastran keyword, use this procedure:
   1. Search
   2. Configure
   3. Confirm

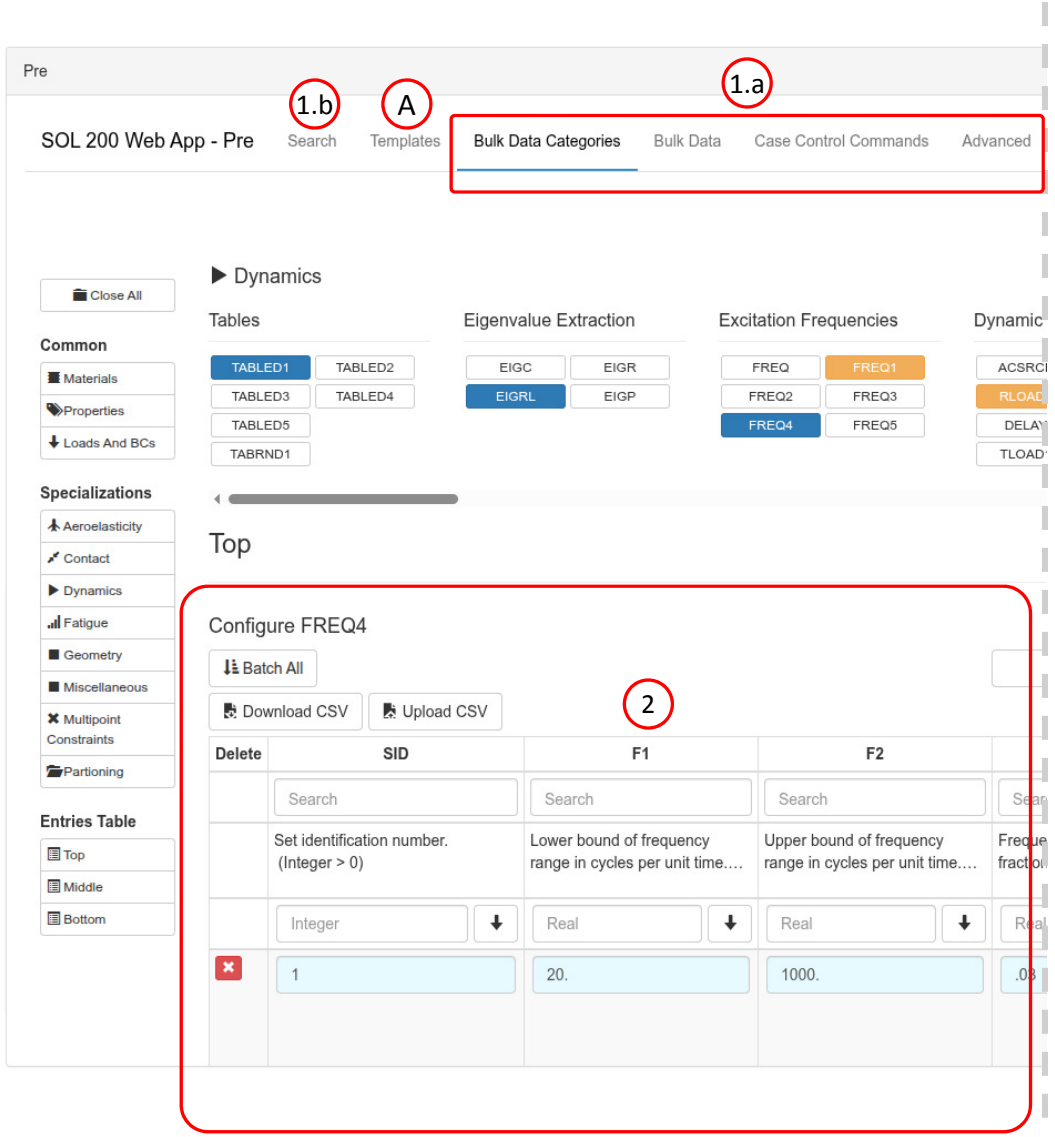
Efficient Strategy to Building Nastran Bulk Data Files (BDF)

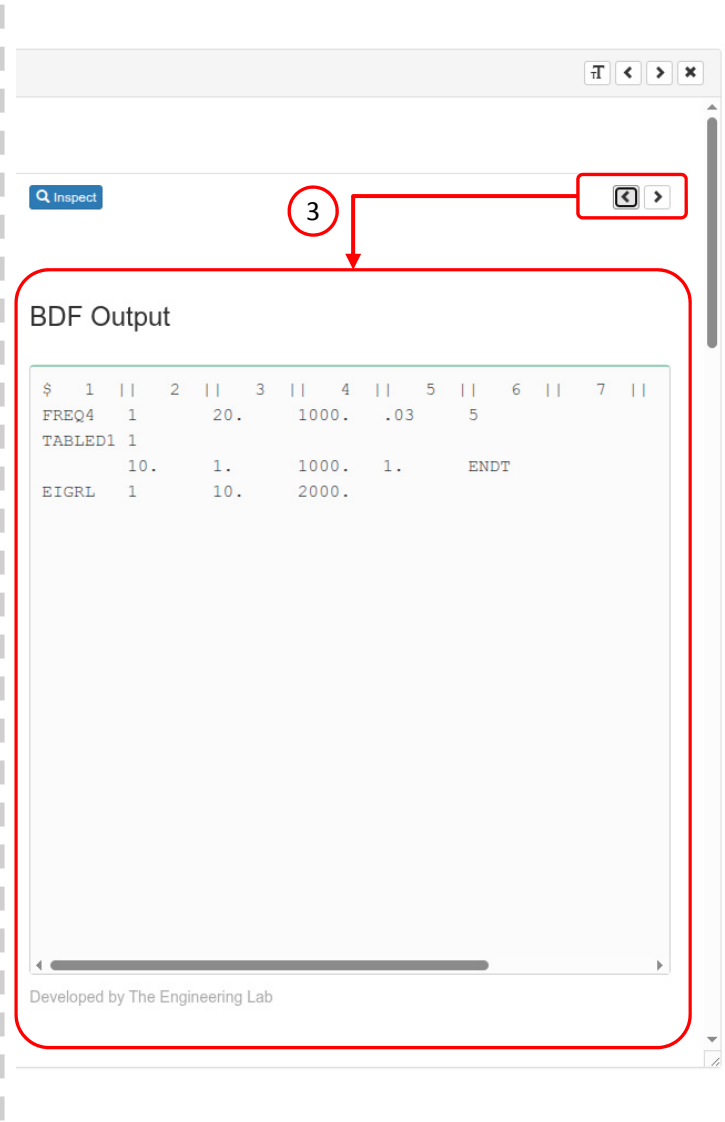
A. A Templates section contains pre-configured BDFs and serve as a starting point for common analysis types such as linear statics, normal modes, modal frequency response, etc.

B. For any Nastran keyword, use this procedure:
   1. Search
      a. Keywords are organized into a limited number of sections: Bulk Data Categories, Bulk Data, Case Control Commands and Advanced.
      b. A global keyword Search tool is also available.
   2. Configure

      Keywords are created and configured in the indicated table.
   3. Confirm

      The most critical step is confirming the keyword is correctly configured. The BDF Output section displays the final keyword and is useful for verification.





Contact me

- Nastran SOL 200 training
- Nastran SOL 200 questions
- Structural or mechanical optimization questions
- Access to the SOL 200 Web App

christian@ the-engineering-lab.com

Tutorial

Units

Units

The following system of units is used. All inputs are in this system of units, including material properties, loads, lengths, etc.

Length	in
Mass	lbf**
Density	lbf/in^3
	PARAM, WTMASS, .00259*
Force	lbf

* Since weight density is used (lbf/in^3 or weight/volume), the mass matrix will have units of weight (lbf) and is not valid/consistent. The conversion factor WTMASS=.00259 converts weight (lbf) to mass (slinch) and ensures the mass matrix has units of mass (slinch).
** The mass matrix has units of slinch, but the OUTPUT GRID POINT WEIGHT GENERATOR reports lbf.

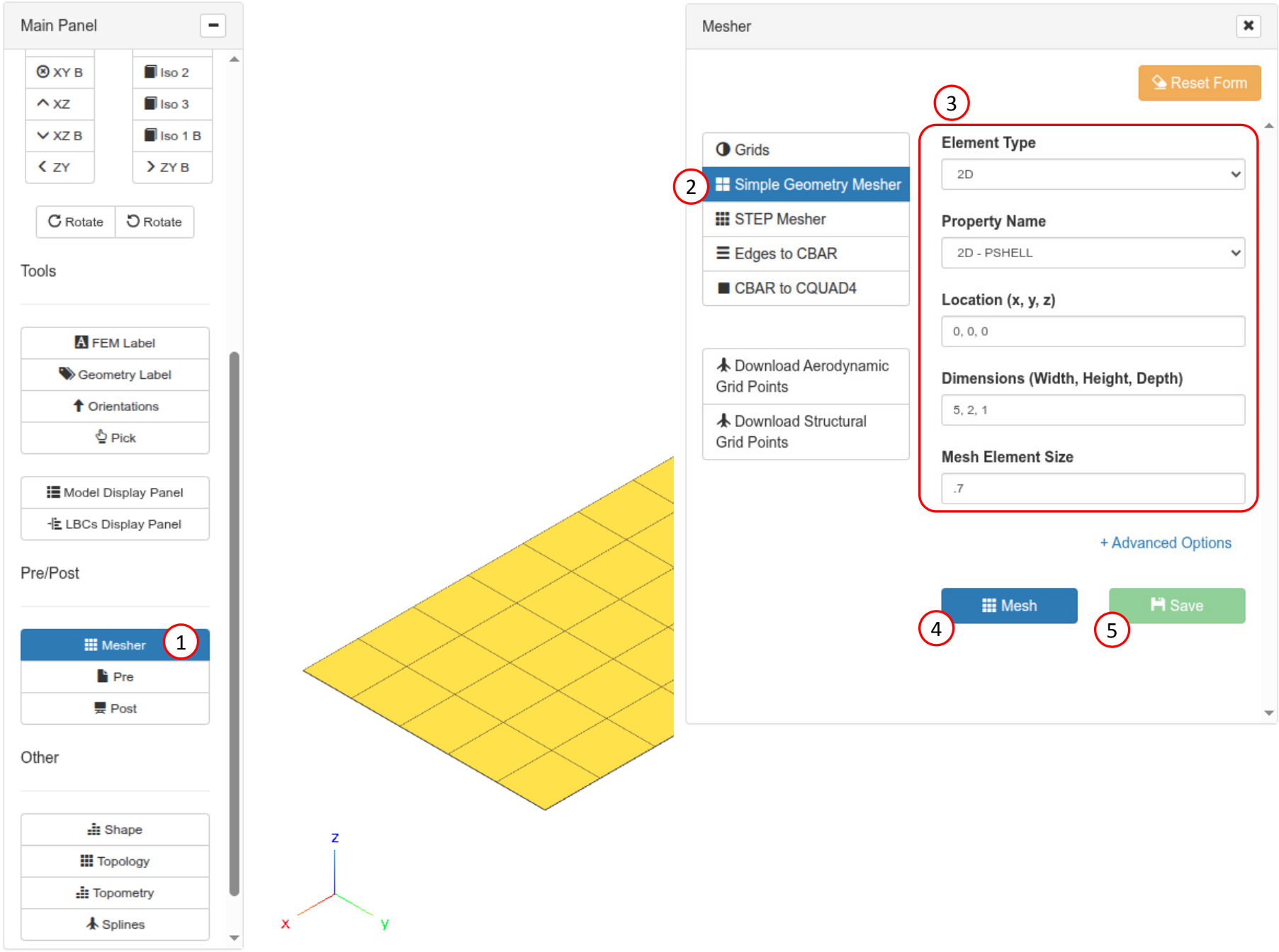
Meshing

Open the Pre/Post Web App

1. Click on the indicated link

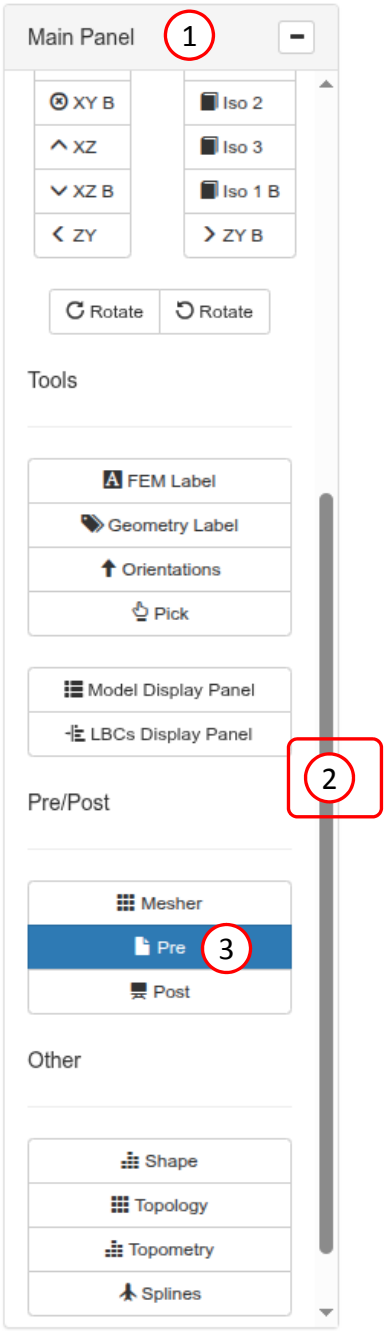
Meshing

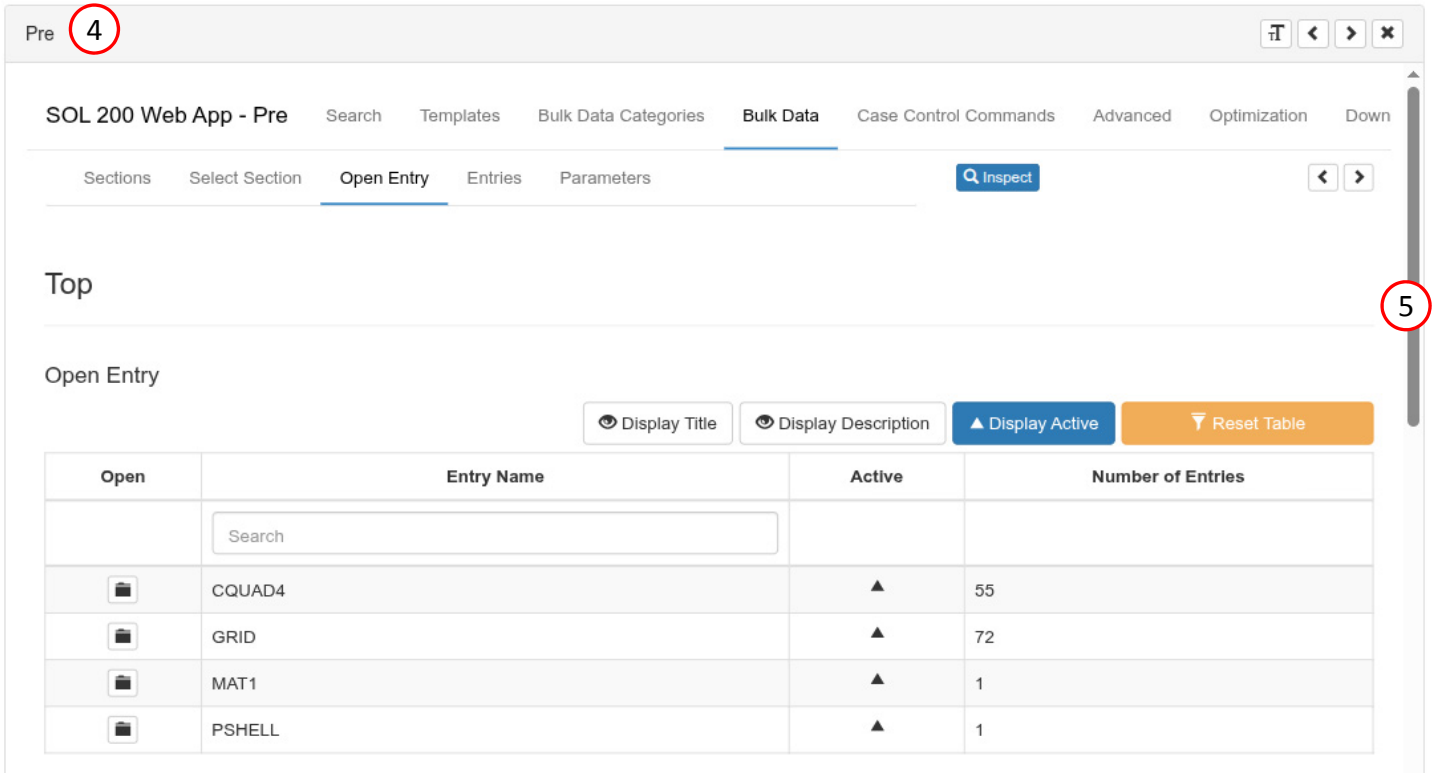
1. Click Mesher
2. Click Simple Geometry Mesher
3. Set the following
   - Element Type: 2D
   - Property Type: 2D - PSHELL
   - Location (x, y, z): 0, 0, 0
   - Dimensions (Width, Height, Depth): 5, 2, 1
   - Mesh Element Size: .7
4. Click Mesh
5. Click Save



Pre Panel

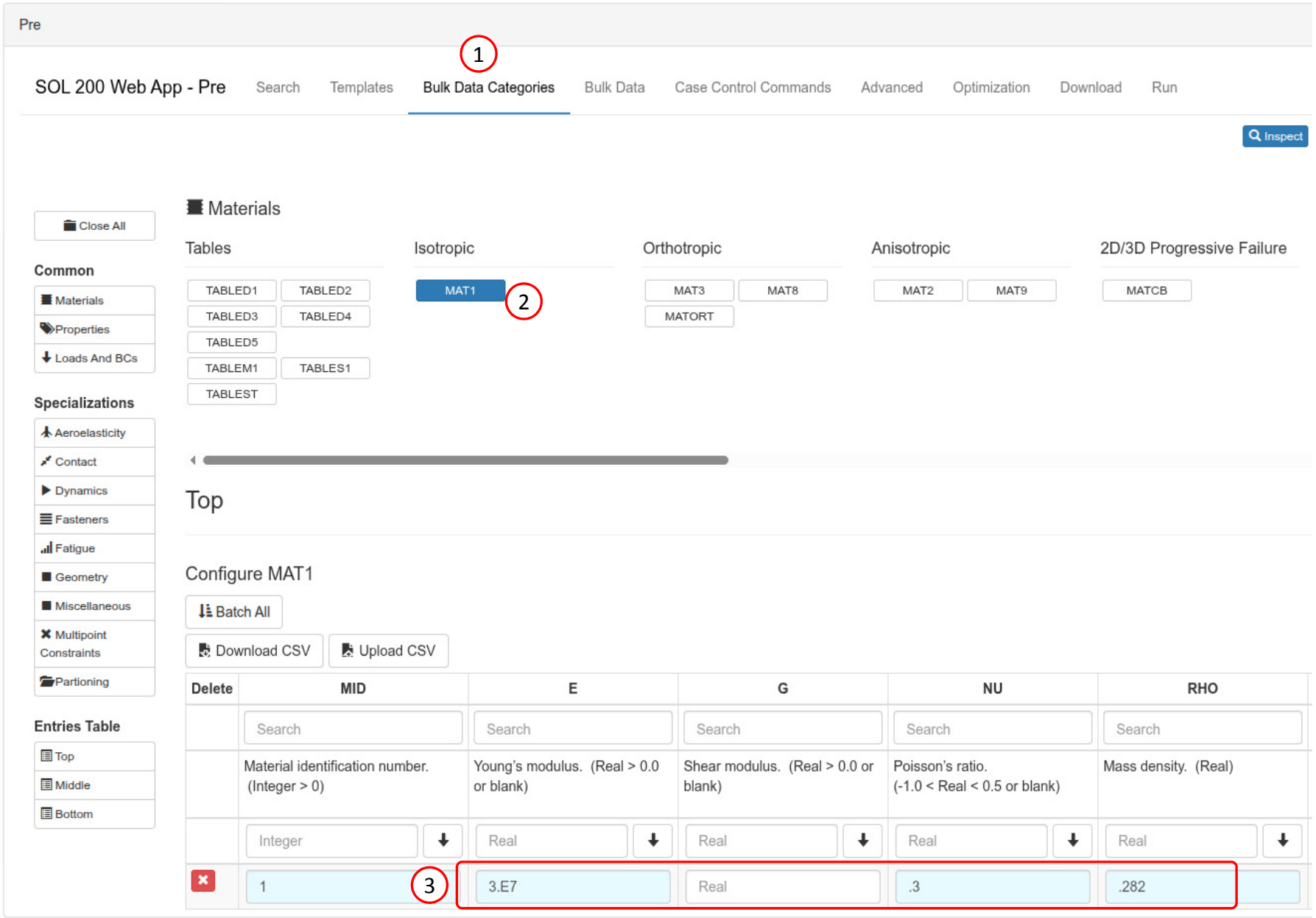
1. Refer to the Main Panel
2. Vertically scroll down
3. Click Pre
4. The Pre panel is now opened
5. The indicated scroll bar is used to scroll the panel





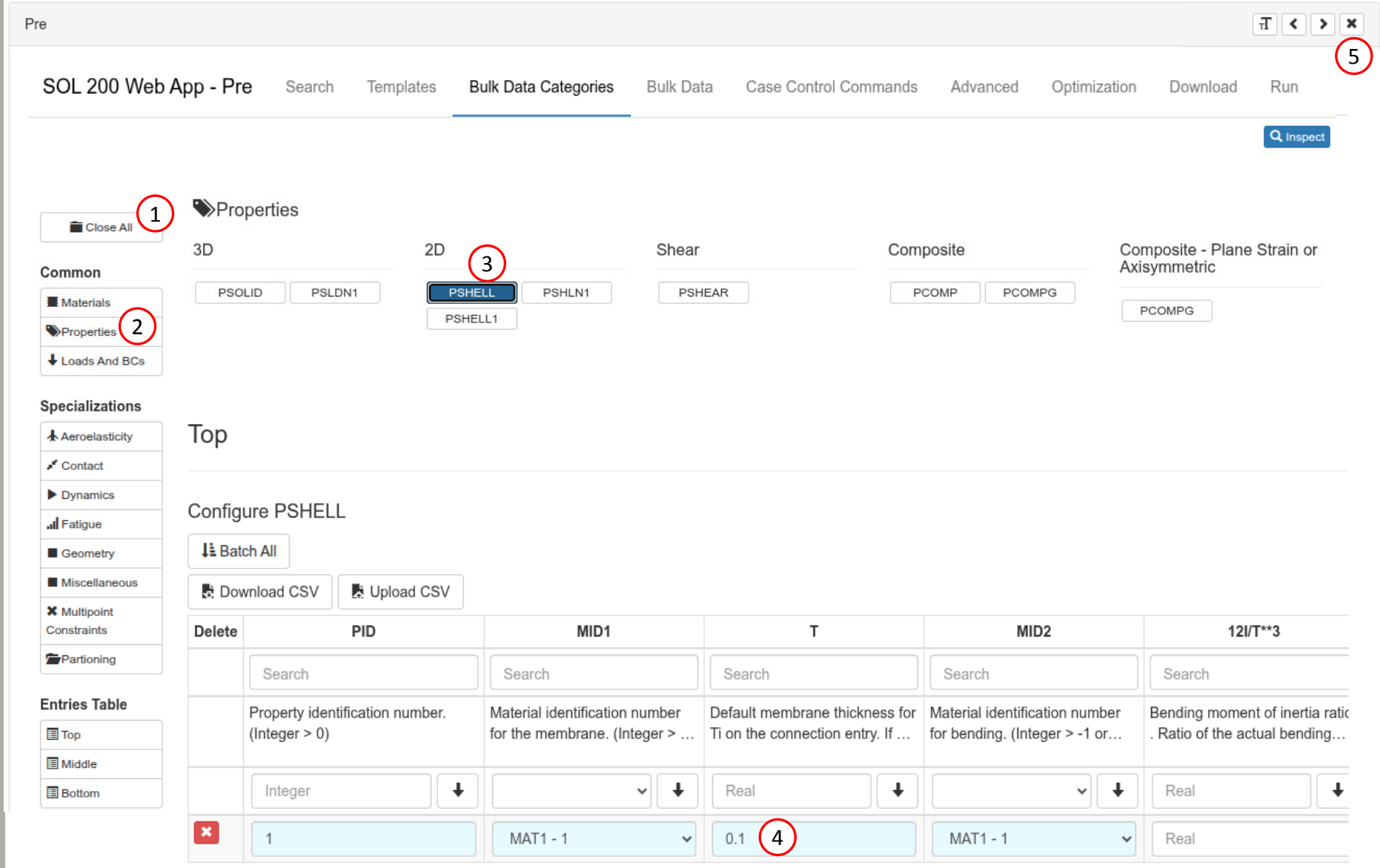
MAT1 - Isotropic Material Property Definition

1. Click Bulk Data Categories
2. Click MAT1
3. For the existing entry, supply the following values.
   - E: 3.E7
   - NU: .3
   - RHO: .282



PSHELL - Shell Element Property

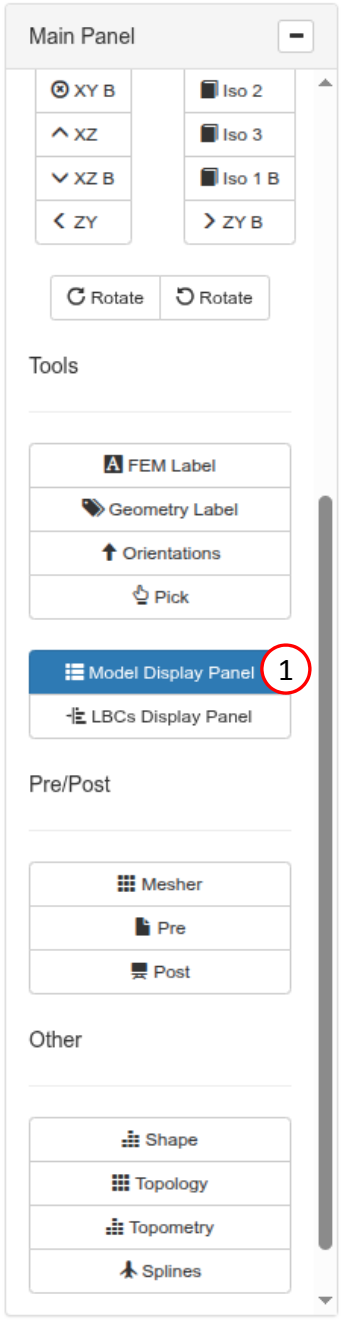
1. Click Close All
2. Click Properties
3. Click PSHELL
4. Ensure the thickness is set to 0.1
5. Click X to close the Pre panel

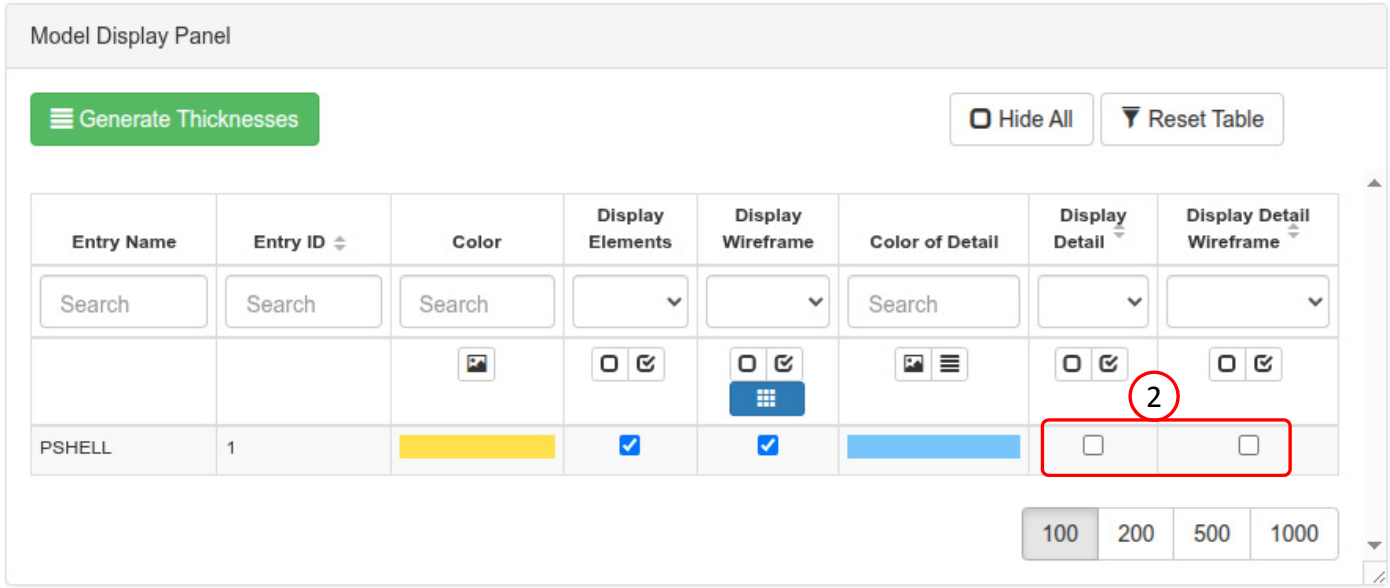


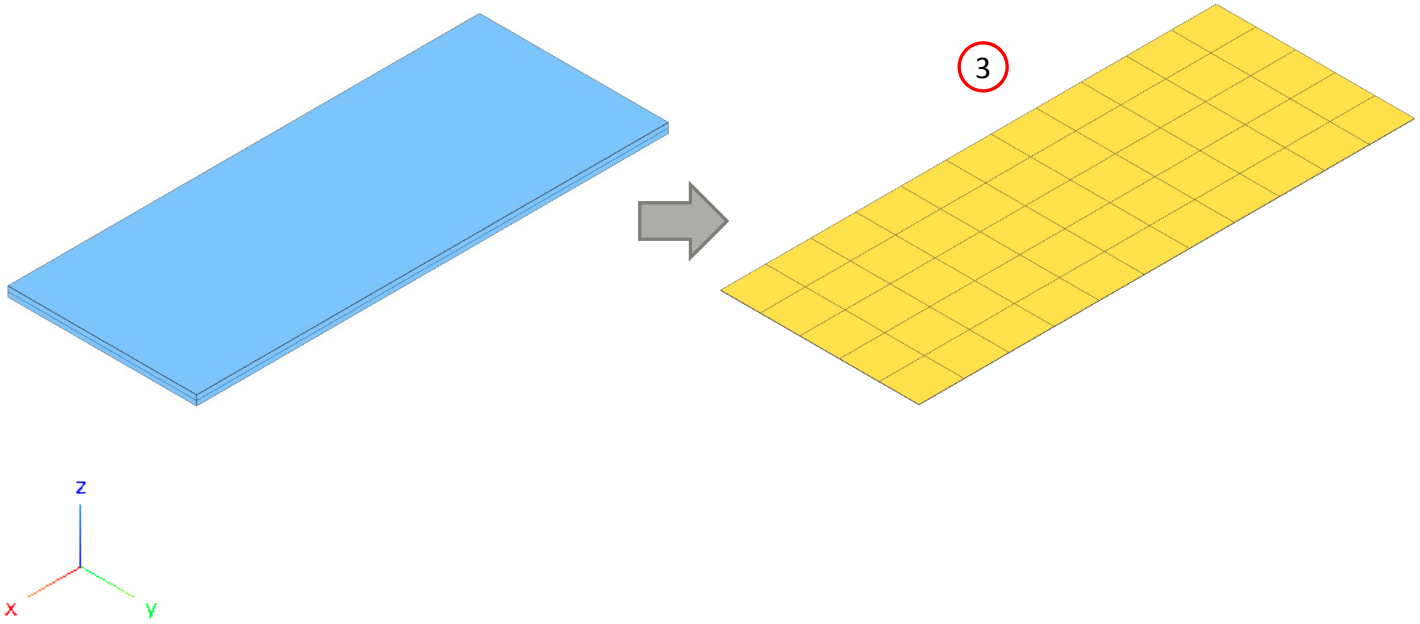
PSHELL Thickness

When the thickness is modified, the thickness is automatically displayed in the model display panel.

1. Click Model Display Panel
2. Unmark the indicated checkboxes
3. The thickness is now hidden



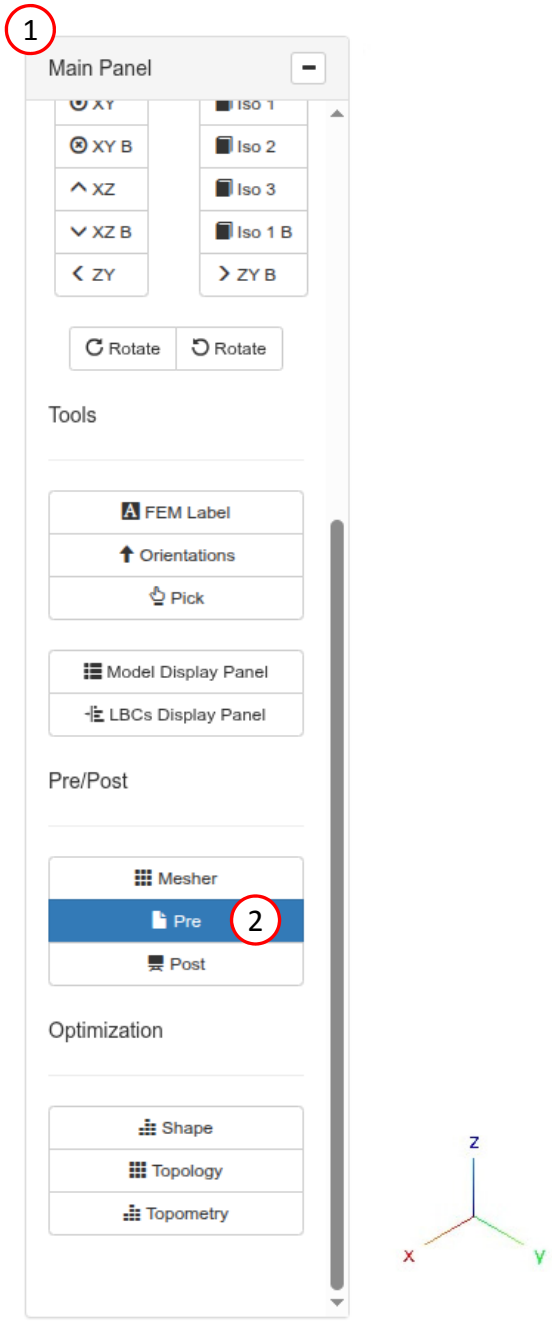




Supports

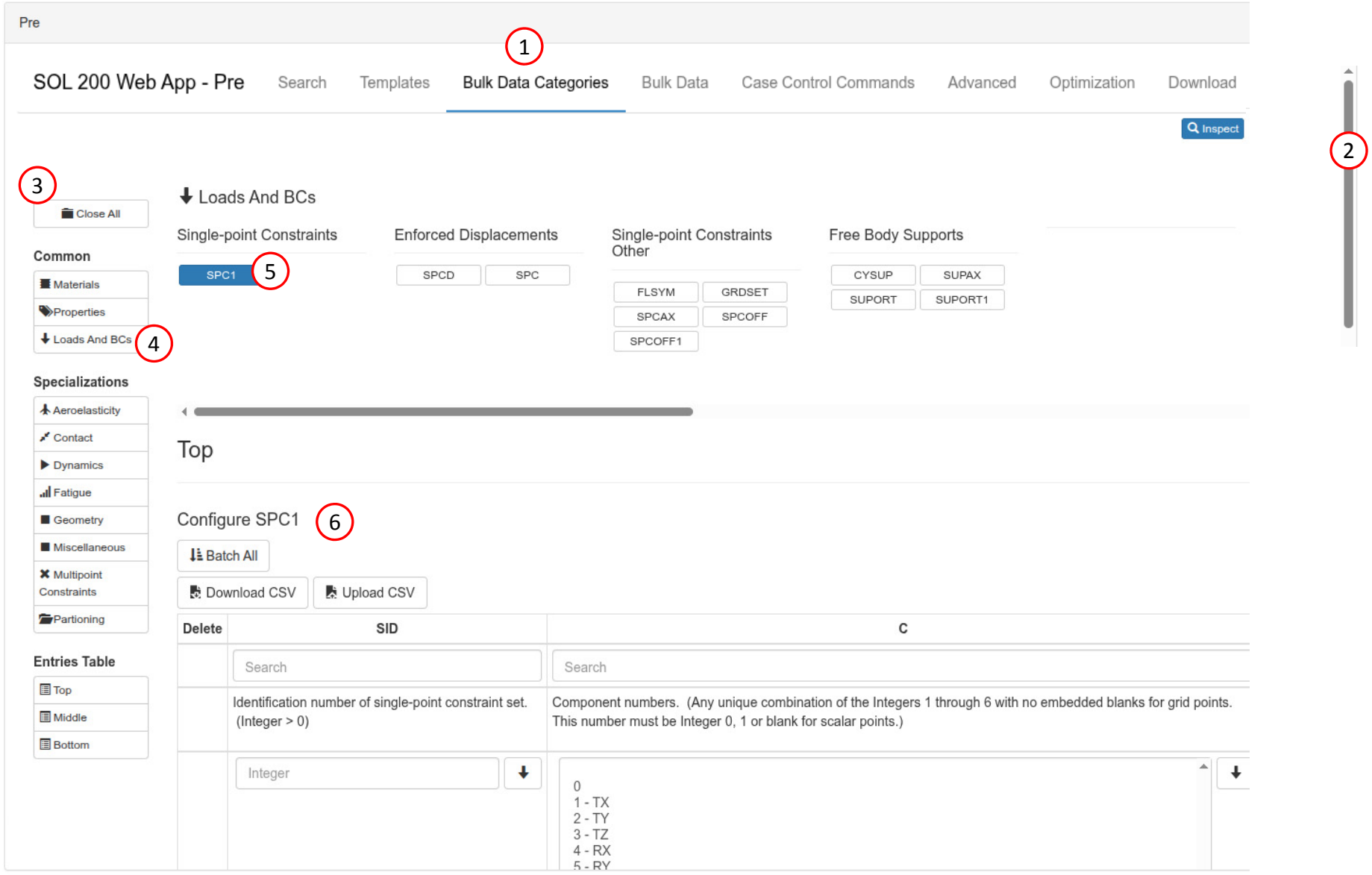
Pre

1. Refer to the Main Panel
2. Click Pre



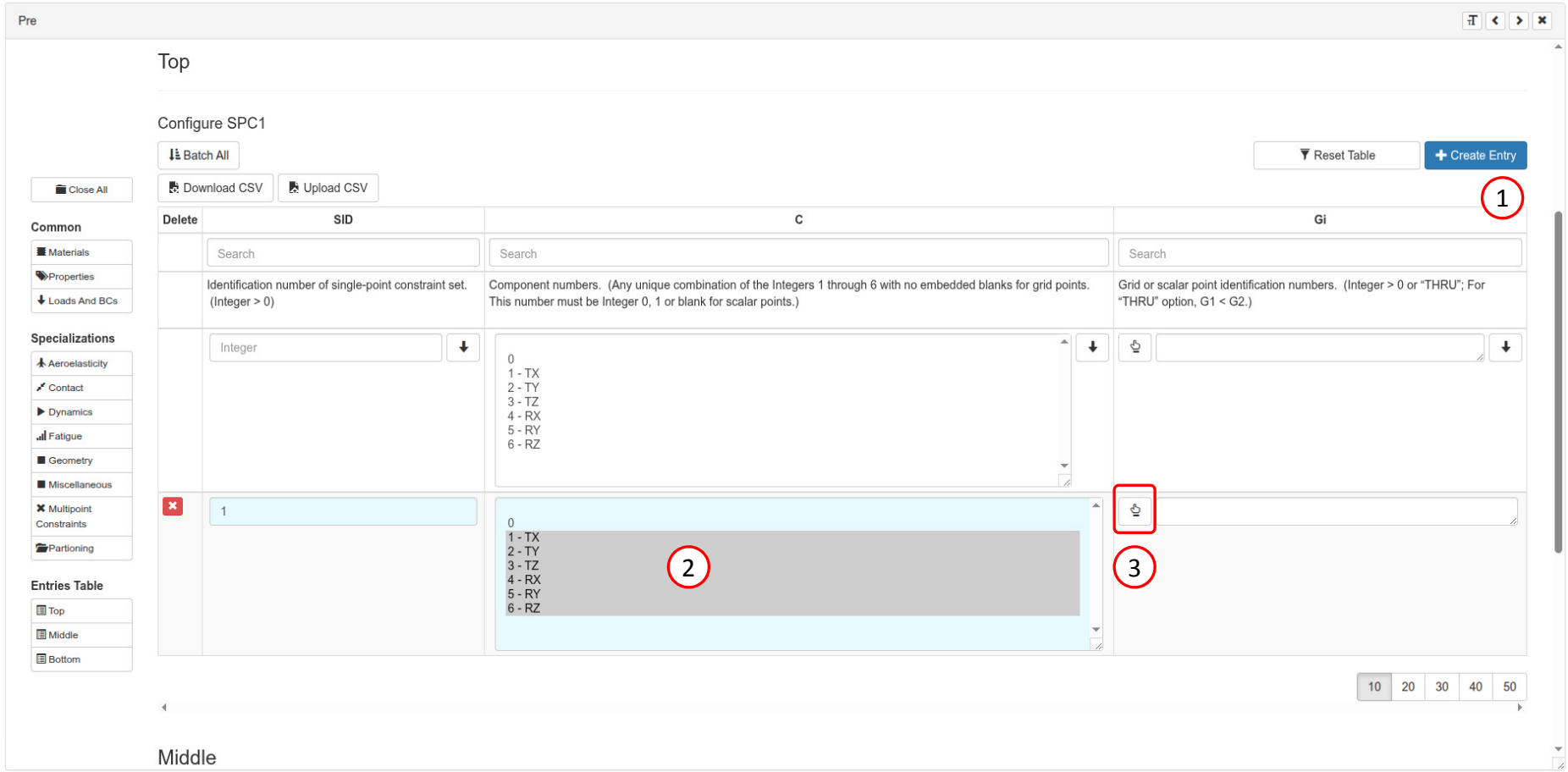
SPC1 - Single-Point Constraint

1. Click Bulk Data Categories
2. Recall the indicated scroll bar is used to scroll the panel
3. Click Close All
4. Click Loads and BCs
5. Click SPC1
6. The table for SPC1 is now visible



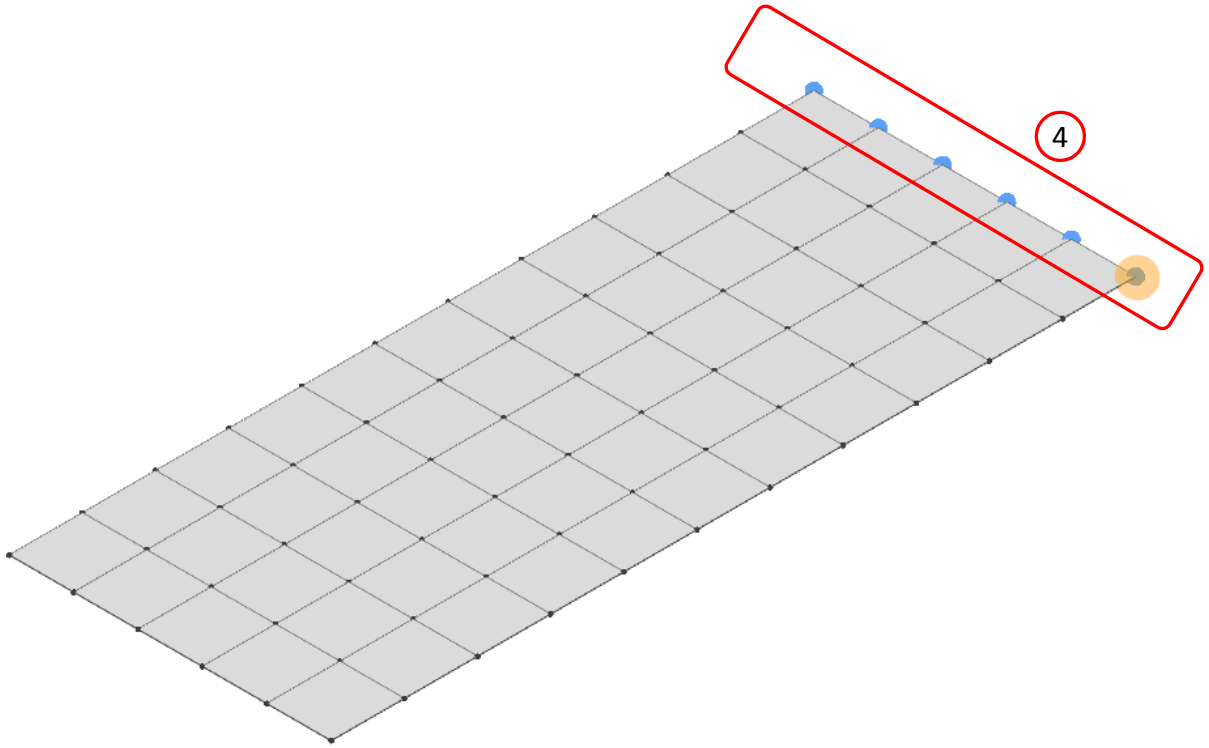
SPC1 - Single-Point Constraint

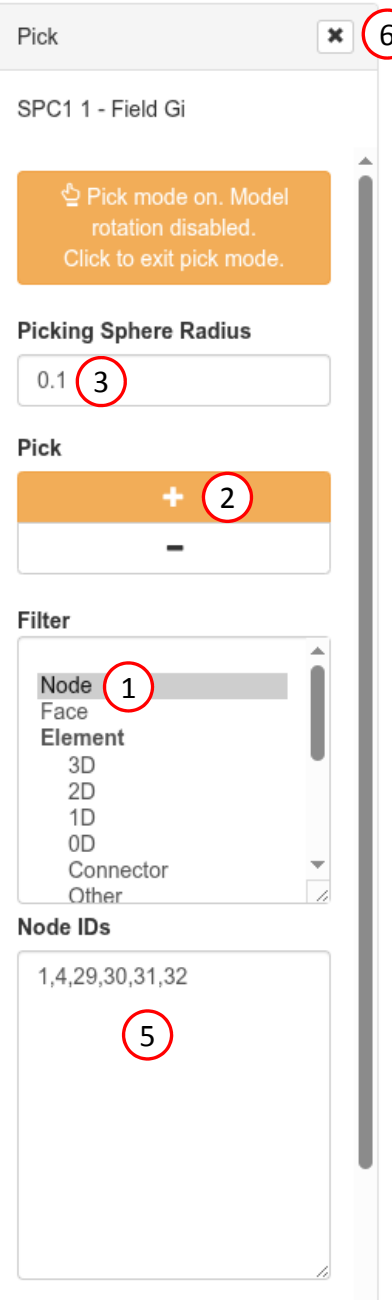
1. Click Create Entry
2. For the existing entry, supply the following values.
   - C: Select options 1, 2, 3, 4, 5, 6
     - Use the left mouse click and press and hold the CTRL key to select multiple options.
3. Click the Pick button



SPC1 - Single-Point Constraint

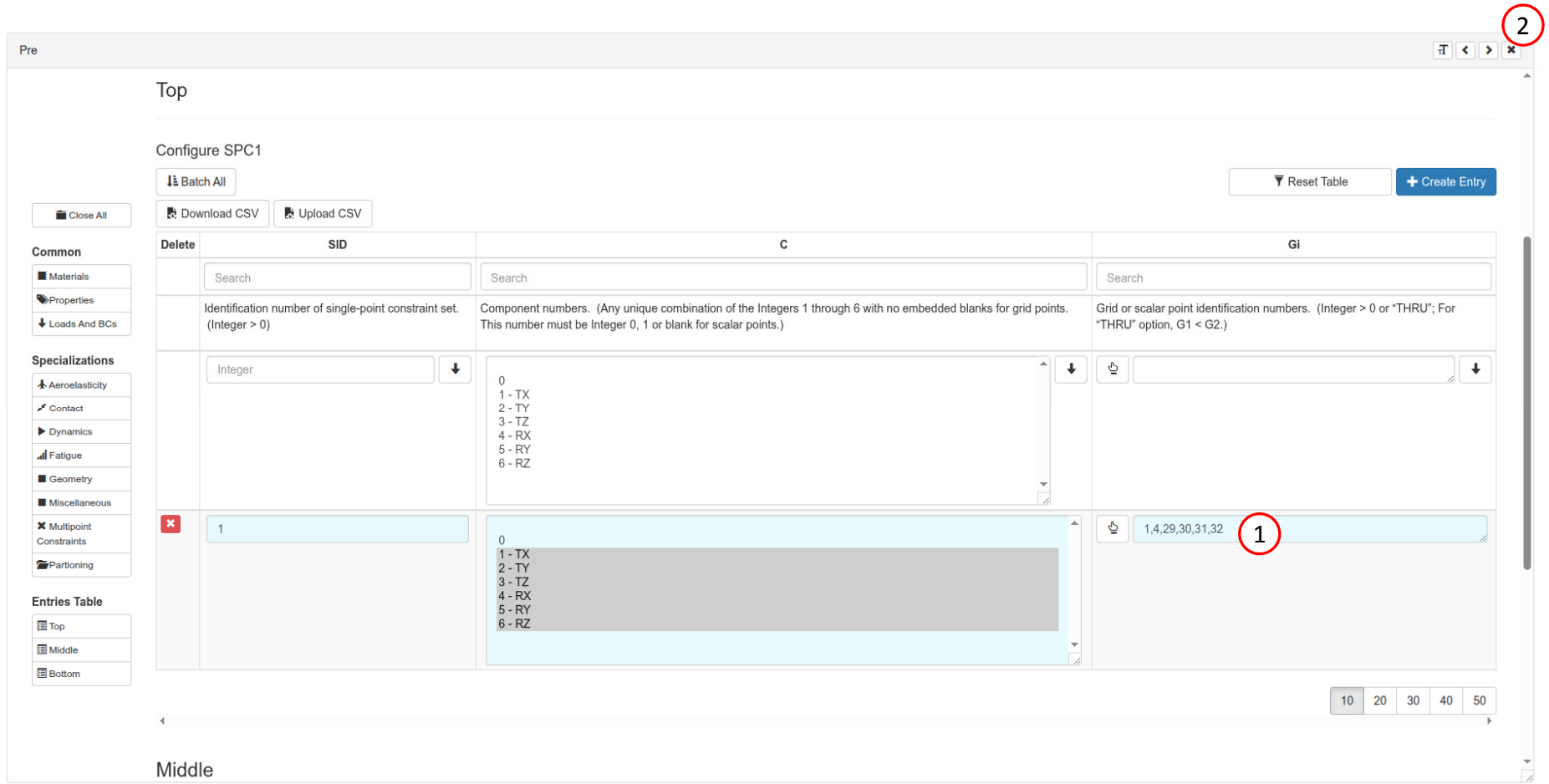
1. Set Filter to Node
2. Click +
3. Set Picking Sphere Radius to 0.1
4. Move the mouse cursor to the indicated region. Click and hold the left mouse button, then drag the picking sphere to select
5. The selected node IDs are stored
6. Click X





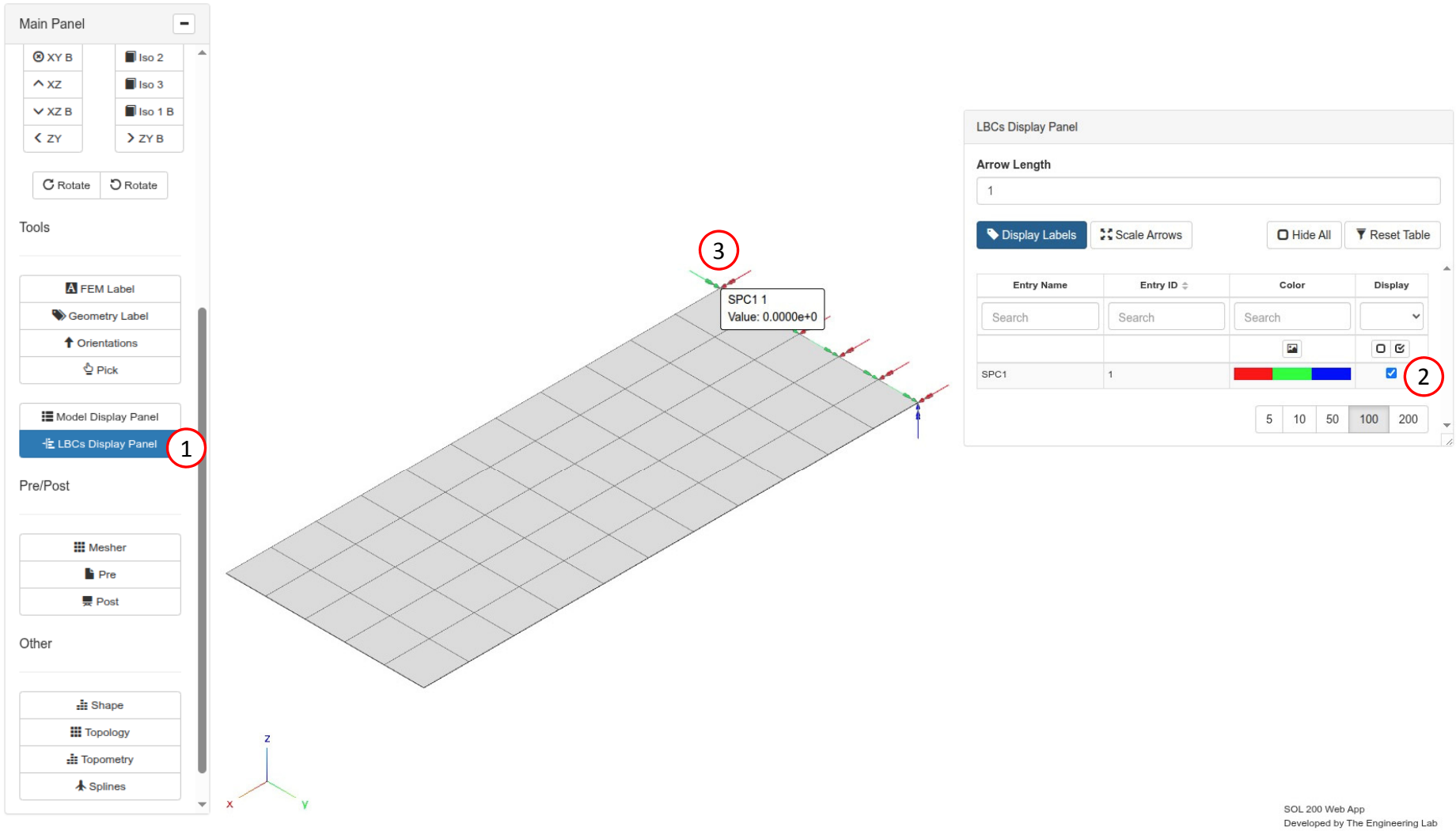
SPC1 - Single-Point Constraint

1. The selected node IDs are now stored
2. Click X



LBCs Display Panel

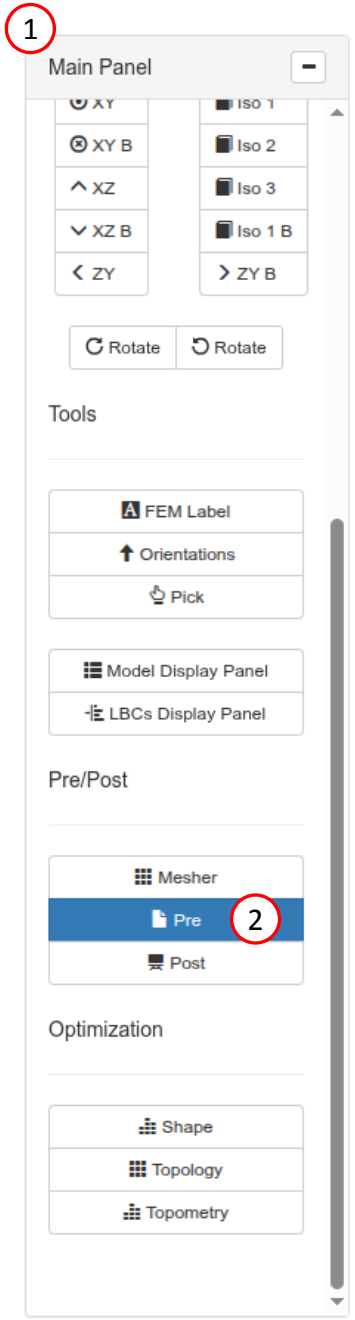
1. Navigate to Main Panel > LBCs Display Panel
2. Mark the checkbox for Entry Name=SPC1
3. Inspect the boundary condition that was created



Normal Modes Analysis

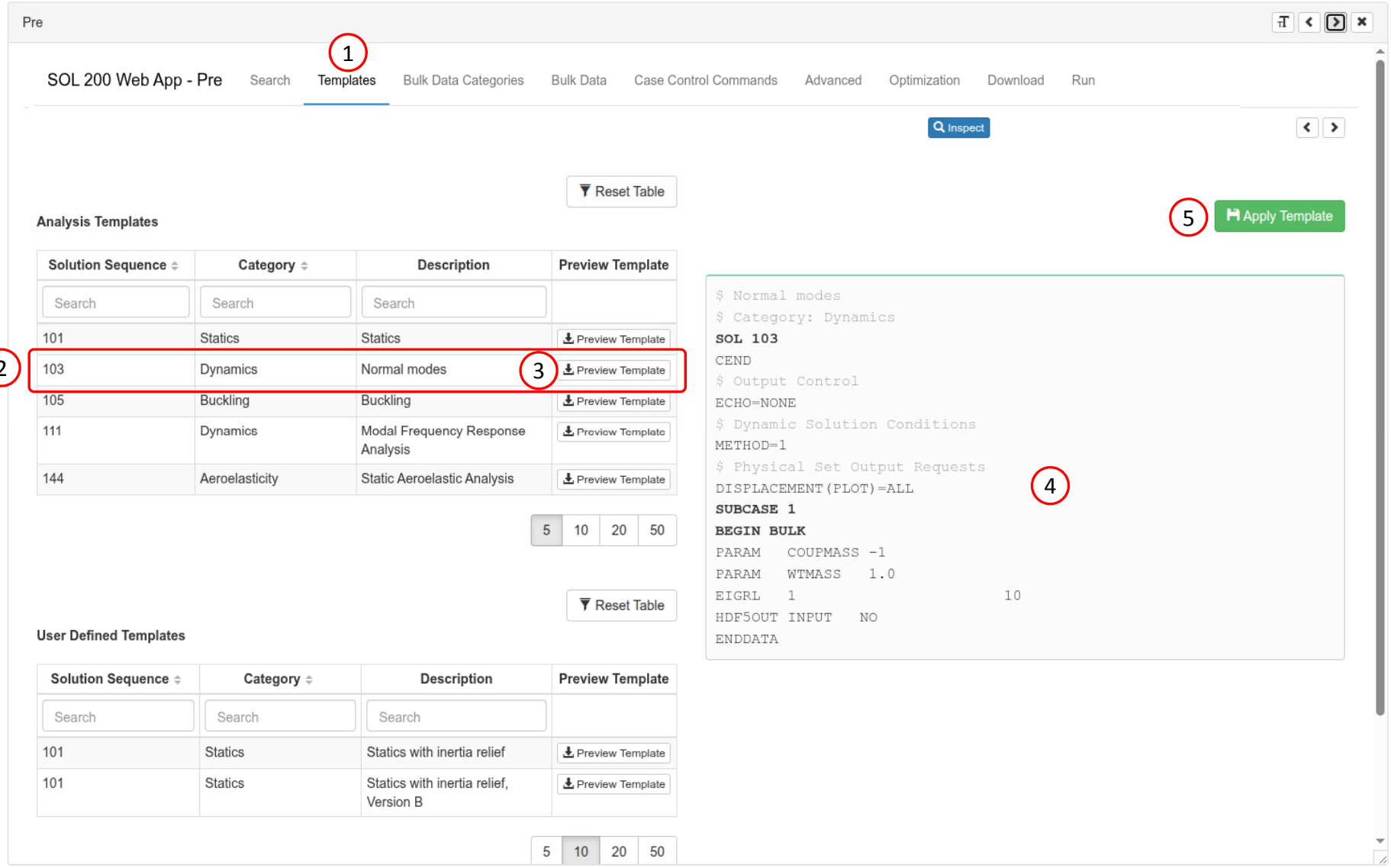
Pre

1. Refer to the Main Panel
2. Click Pre



Template

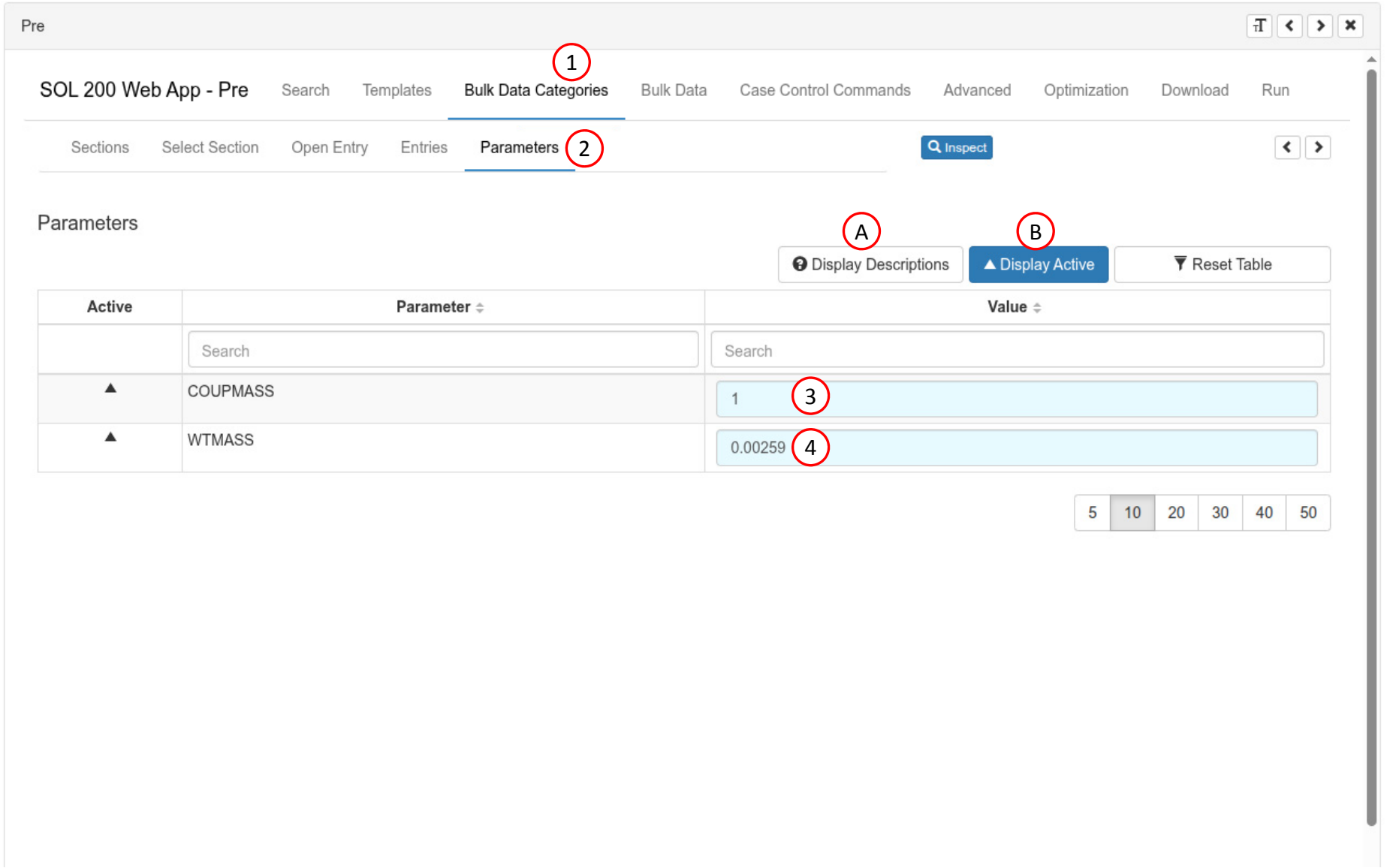
1. Click Templates
2. Locate the row for SOL 103 or Normal Modes Analysis
3. Click Preview Template
4. A preview is displayed of the bulk data file that contains a configuration for a normal modes analysis
5. Click Apply Template



Units

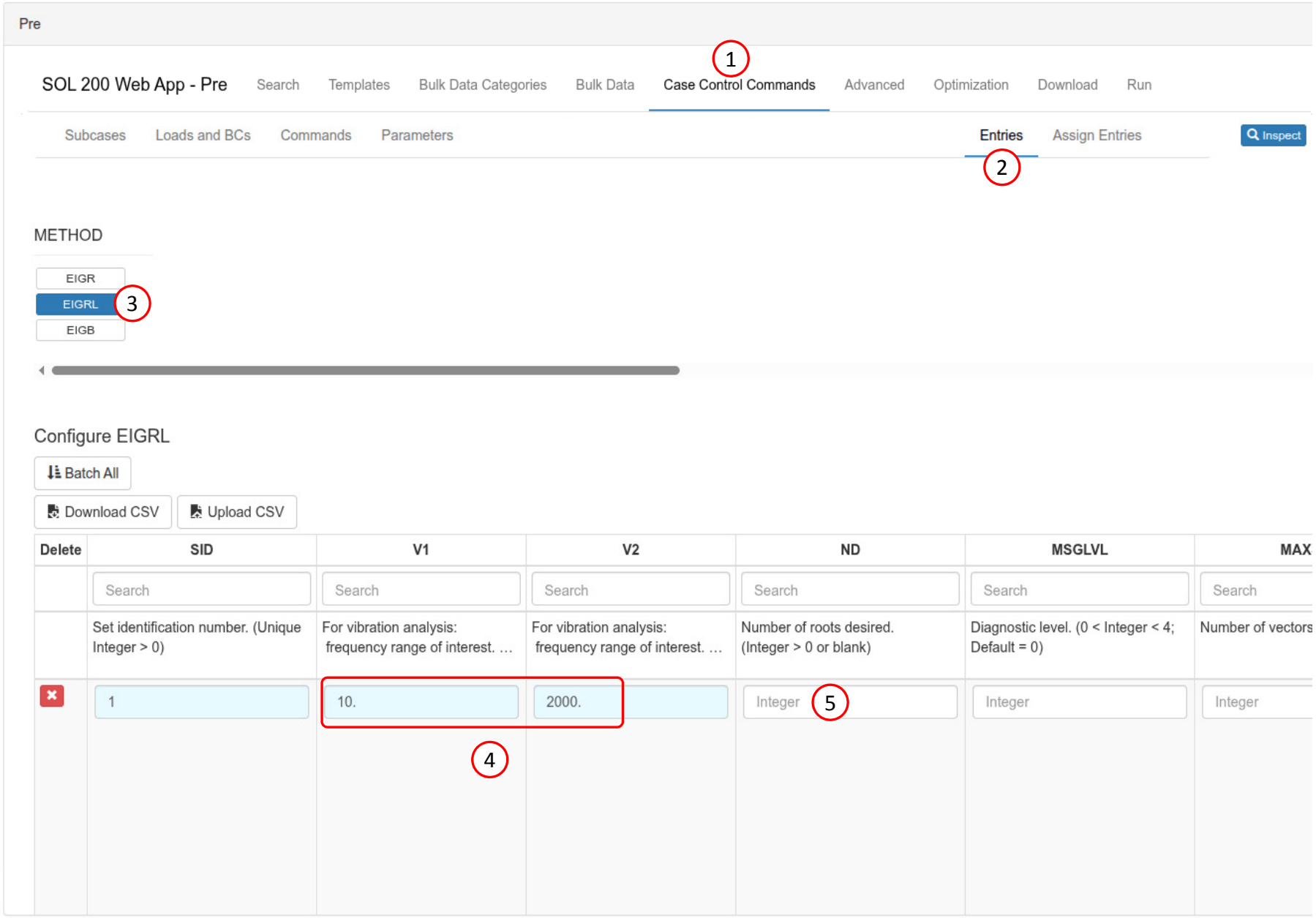
1. Click Bulk Data
2. Click Parameters
3. For COUPMASS, use the following value: 1
4. For WTMASS, use the following value: 0.00259

A. When Display Descriptions is clicked, descriptions for each parameter are listed.
B. The Display Active button will display all the parameters or only the parameters in use.



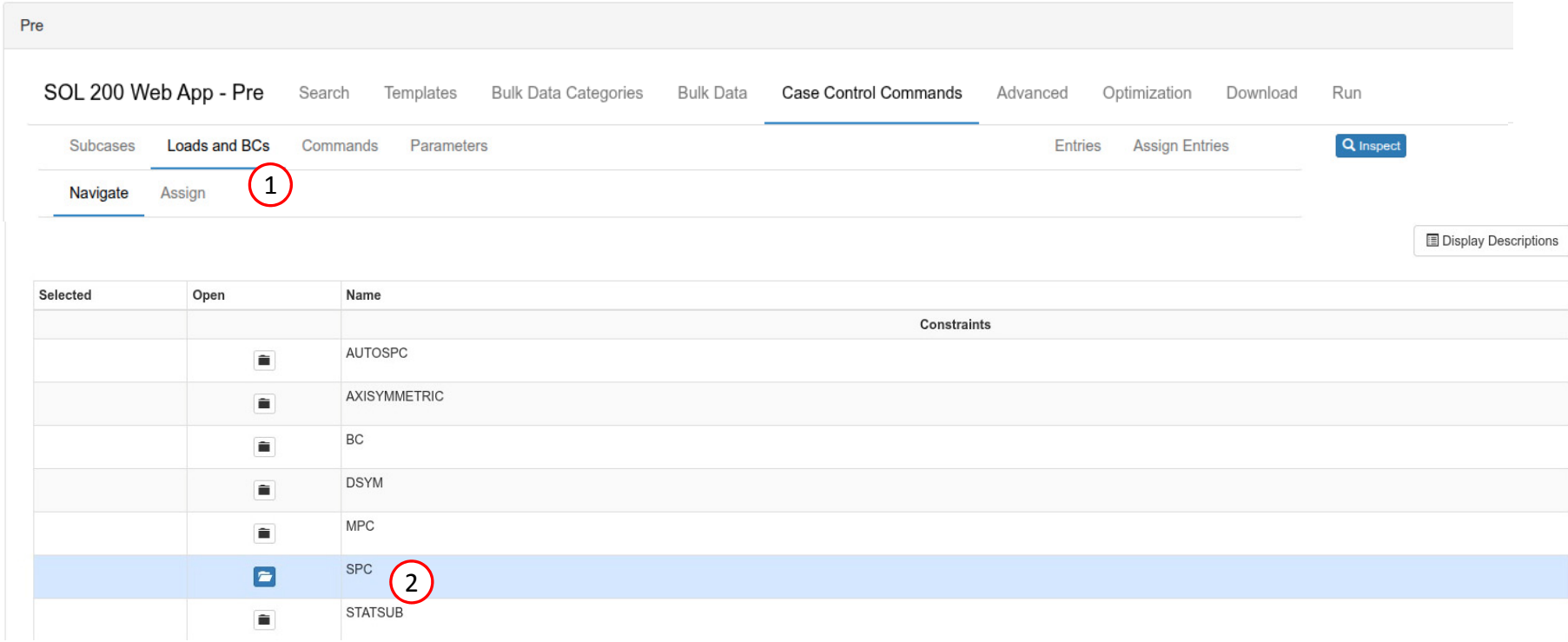
EIGRL - Real Eigenvalue Extraction Data, Lanczos Method

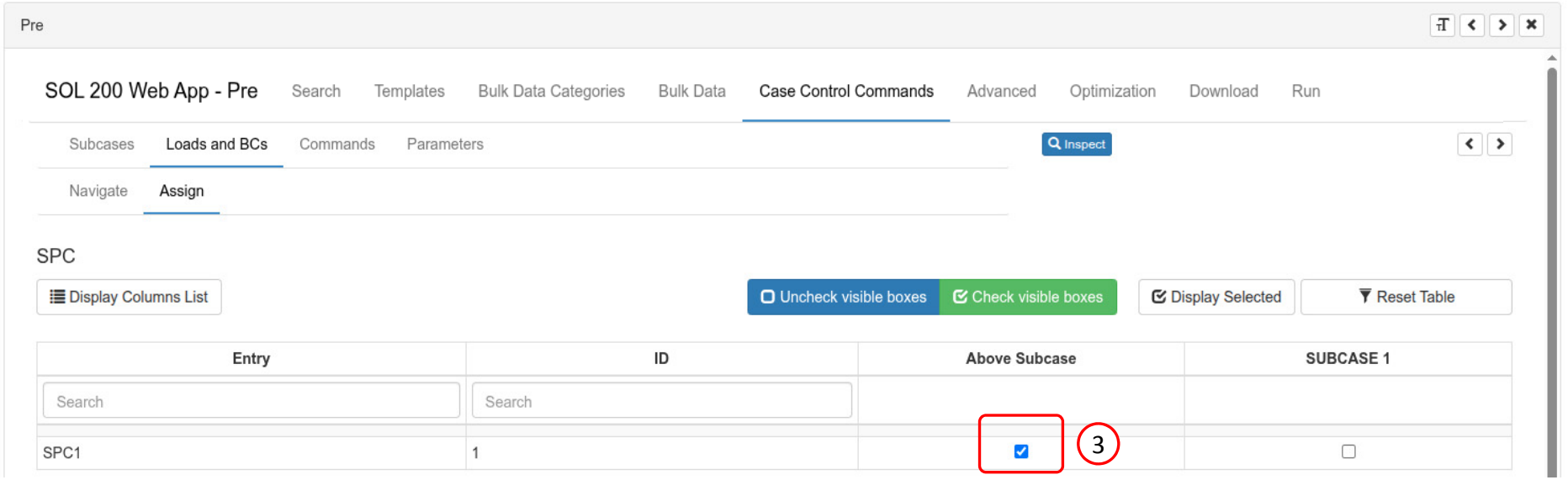
1. Click Case Control Commands
2. Click Entries
3. Click EIGRL
4. For the existing entry, supply the following values.
   - V1: 10.
   - V2: 2000.
5. Ensure field ND is blank/empty



SPC - Single Point Constraint Set Selection

1. Click Loads and BCs
2. Select SPC
3. For SPC1 ID=101 select the checkbox in column Above Subcase



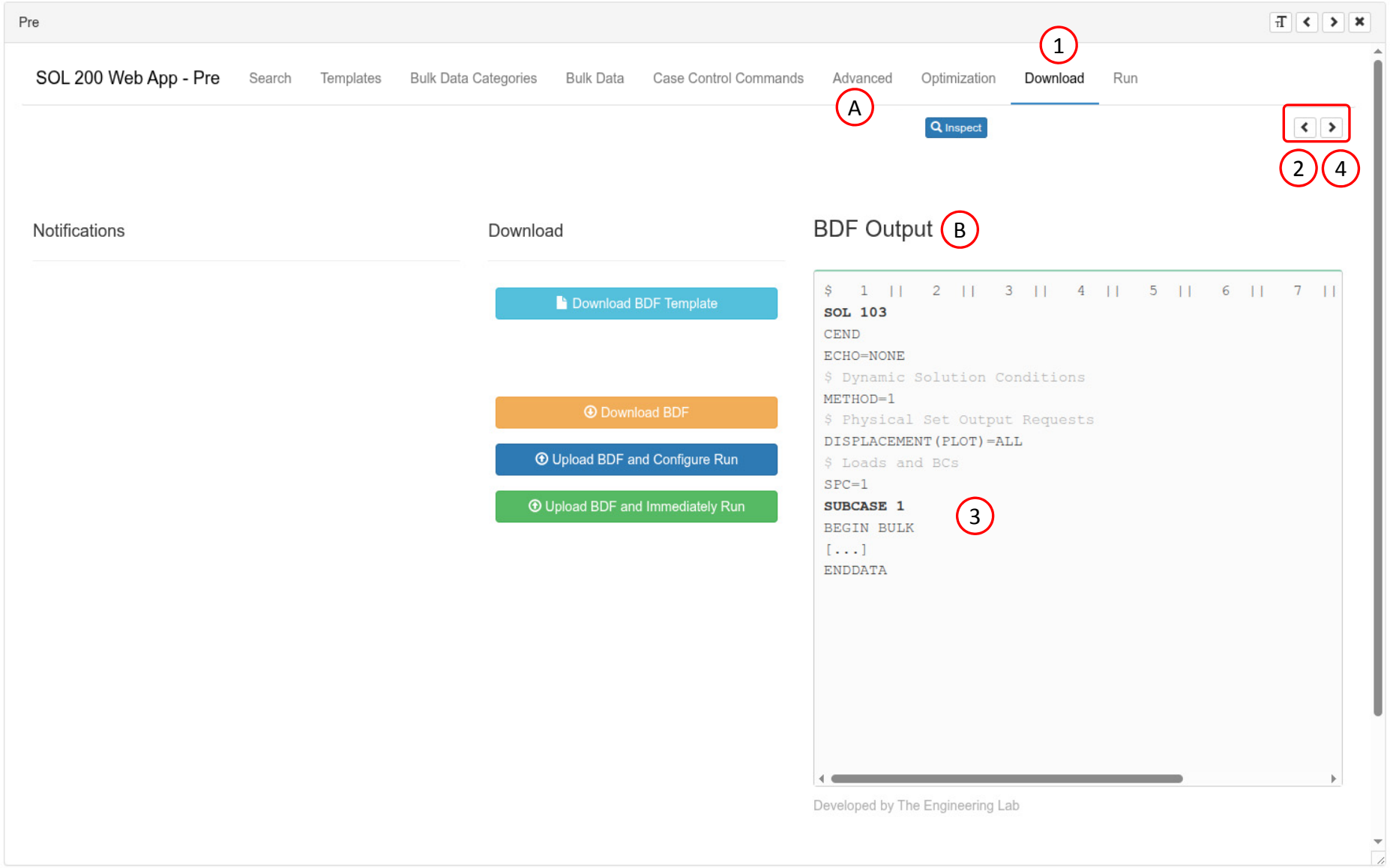


BDF Output

1. Click Download
2. Use the indicated buttons to display the BDF Output section
3. The head of the bulk data file is displayed. The head includes the following sections:
   - NASTRAN Statements
   - File Management Statements
   - Executive Control Statements
   - Case Control Commands
4. Use the indicated buttons to hide the BDF Output section

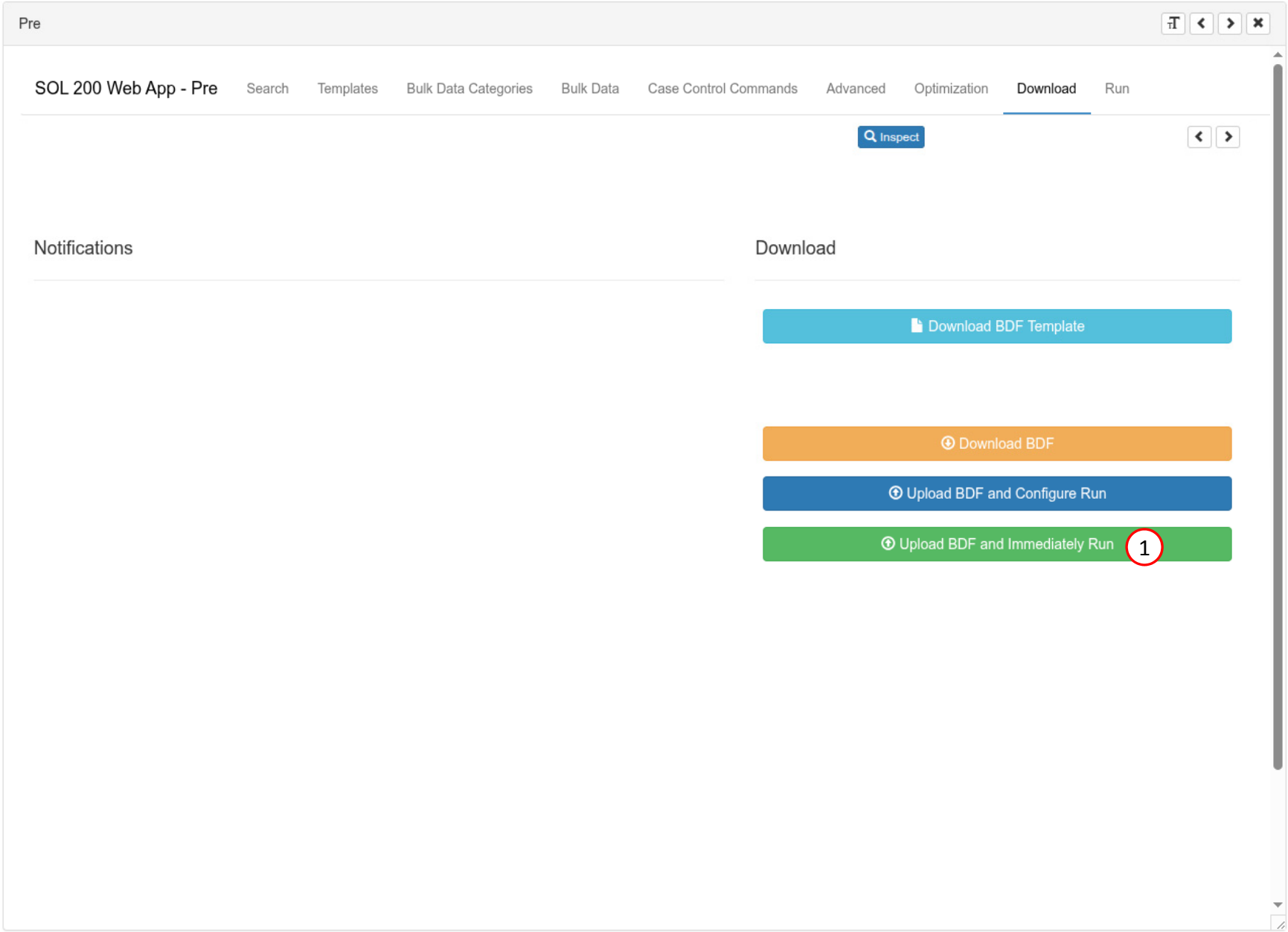
A. Optional - NASTRAN Statements, File Management Statements and Executive Control Statements are accessible by clicking Advanced Sections

B. The BDF Output section is ideal for learning and debugging MSC Nastran models. Any changes made in the interface are immediately updated in the BDF Output section.



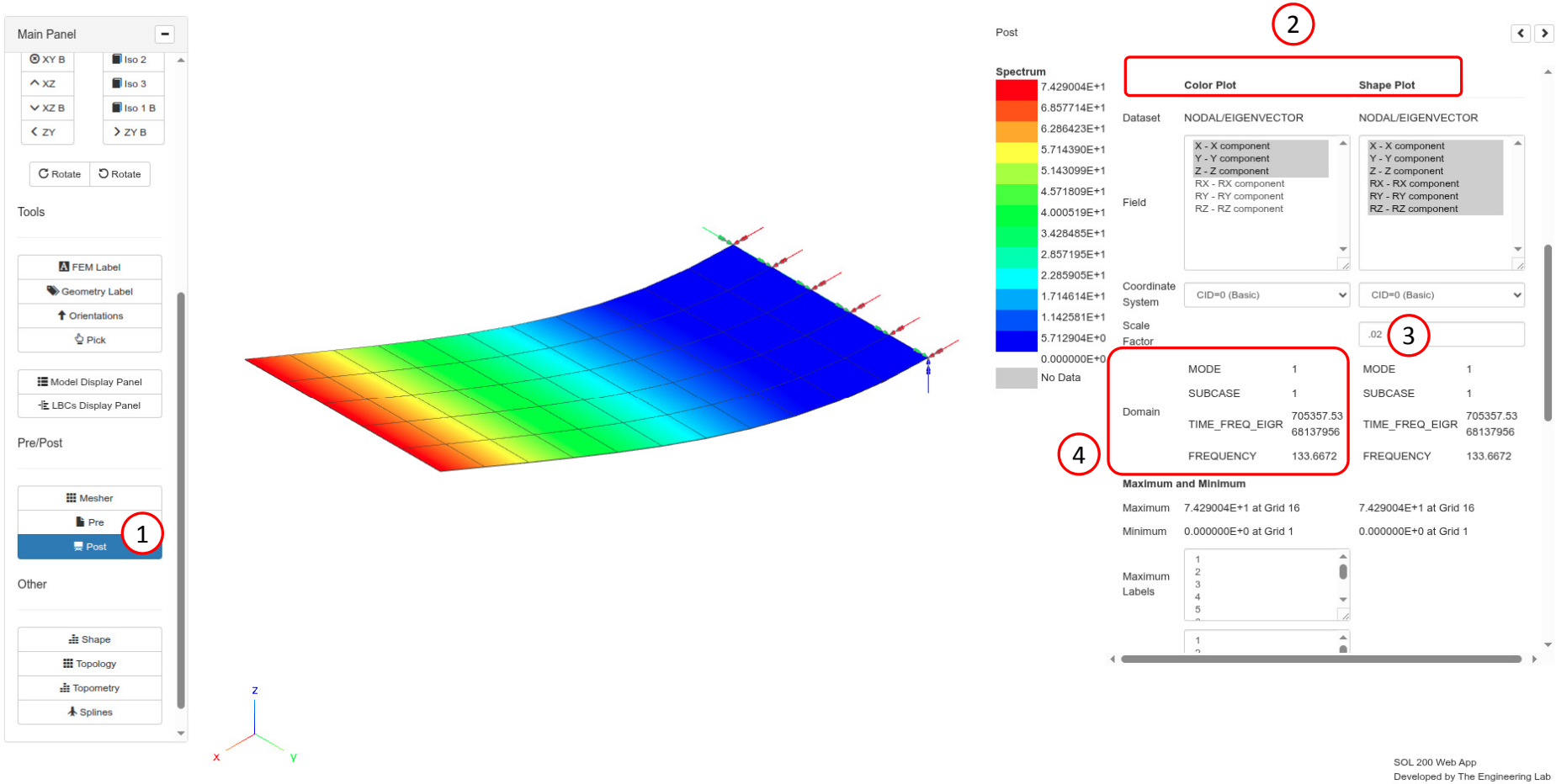
Run

1. Click Upload BDF and Immediately Run



Post

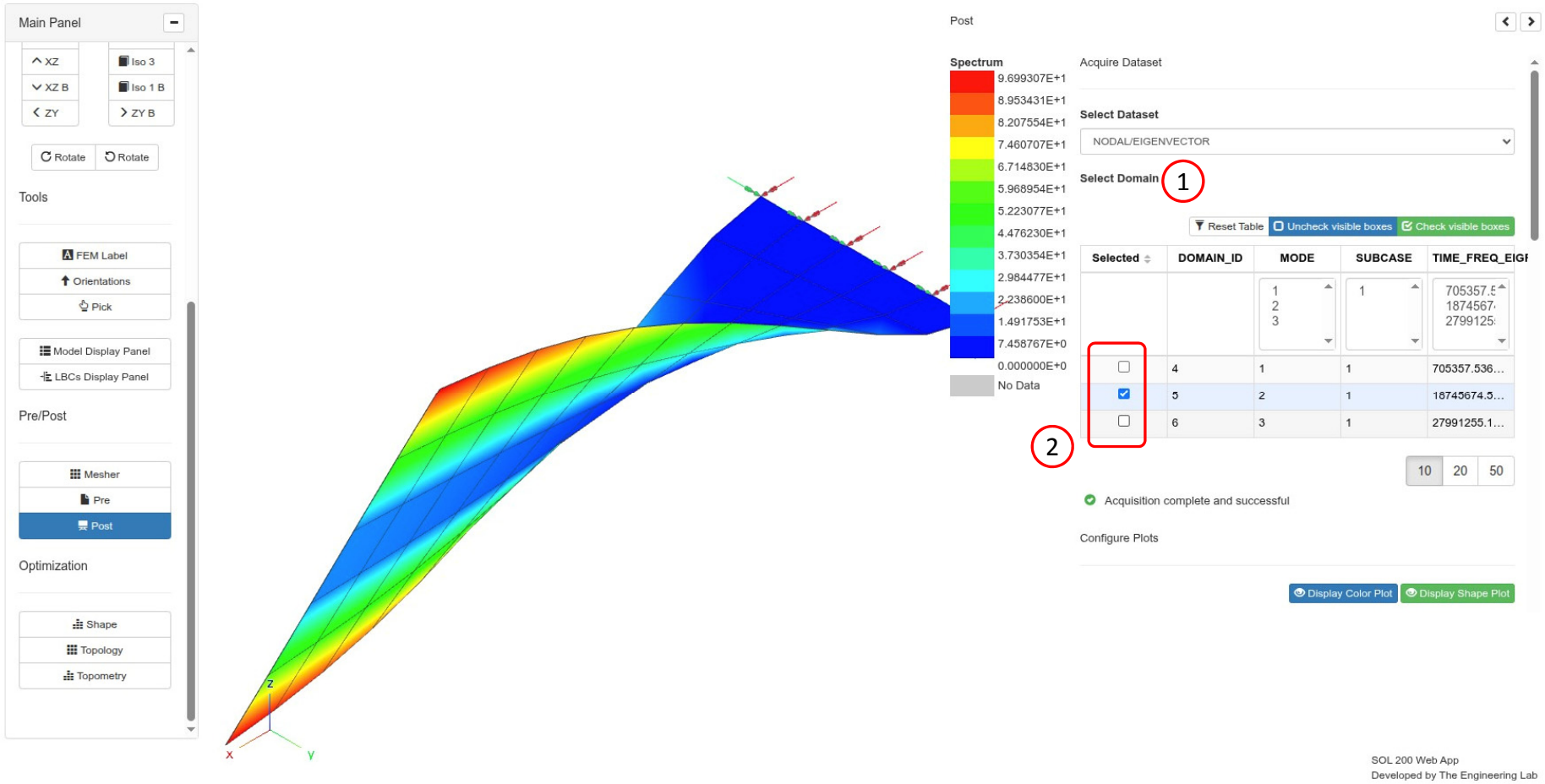
1. After the analysis, the Post panel is automatically opened. The Post panel may be closed or opened by clicking on the indicated button.
2. Vertically scroll to sections Color Plot and Shape Plot
3. Set the Scale Factor to .02
4. The MODE number, SUBCASE ID, eigenvalue (TIME_FREQ_EIGR) and natural frequency (FREQUENCY) are listed



Post

1. Vertically scroll to section Select Domain
2. Select different checkboxes to see the corresponding mode shape

Note that the table displays eigenvalues. There is a goal to find the corresponding natural frequencies.

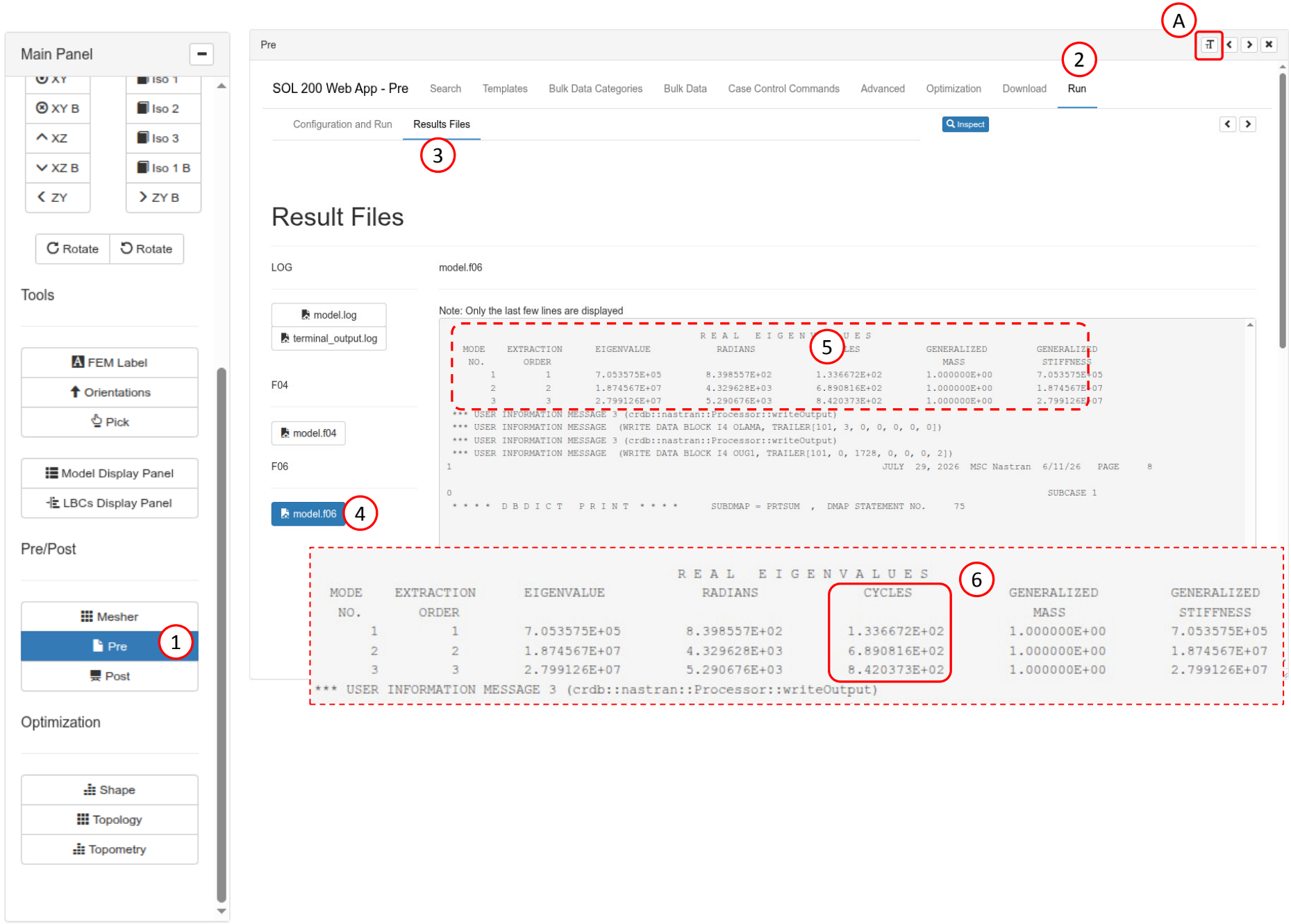


F06 > List of Natural Frequencies

1. Click Pre
2. Click Run
3. Click Results Files
4. In the left hand side, click model.f06 to display the contents of the file.
5. Vertically scroll until table REAL EIGNEVALUES is displayed.
6. The natural frequencies for modes 1, 2 and 3 are display. Record the following natural frequencies for modes 1, 2, and 3, respectively:
   - 133.6672 Hz, 689.0816 Hz, 842.0373 Hz

   Later on these natural frequencies will be used.

A. If needed, the font size may be increased by clicking the indicated button. Alternatively the web browser's zoom functionality may be used.

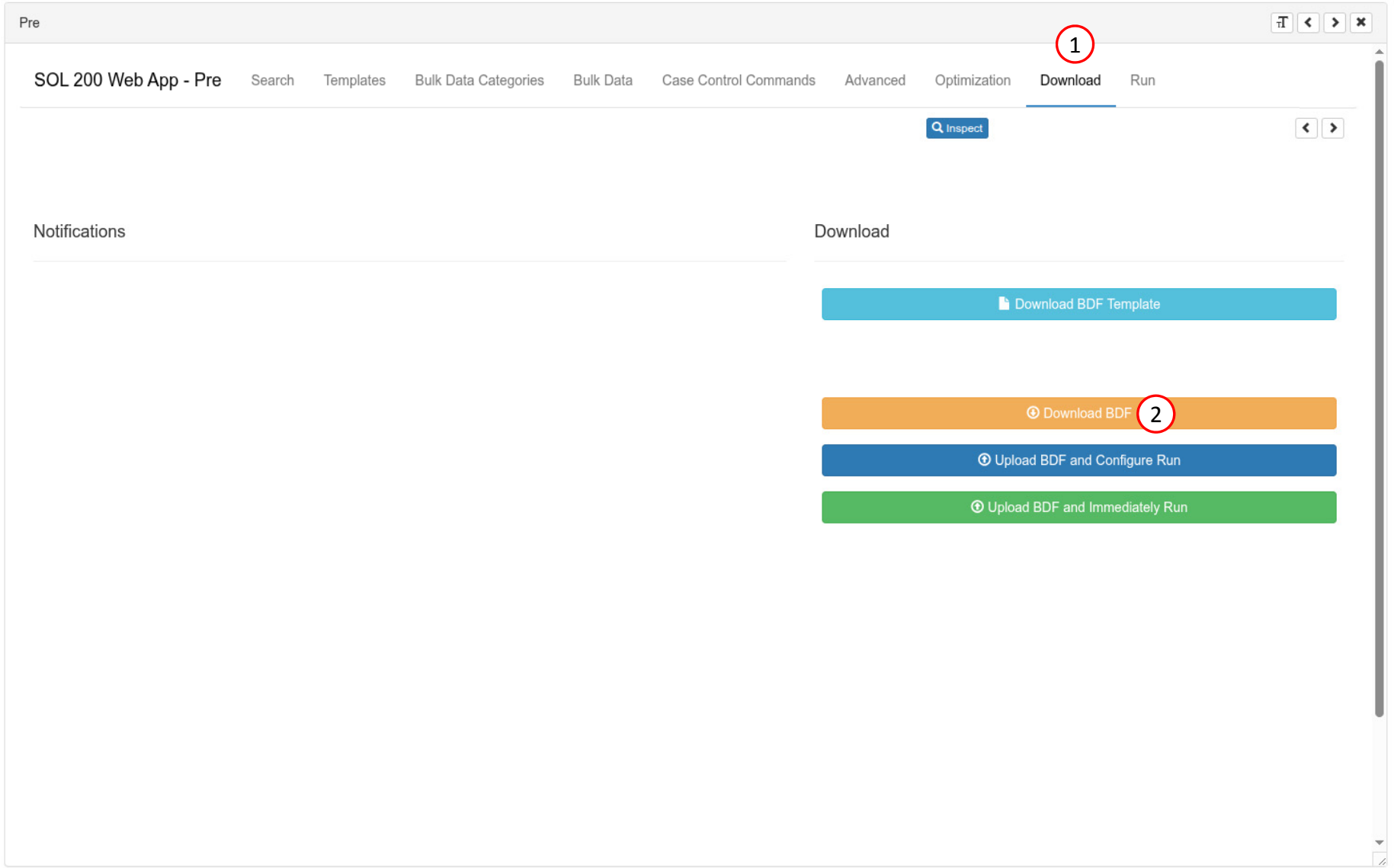


Questions? Email: christian@ the-engineering-lab.com

Download

1. Click Download
2. Click Download BDF

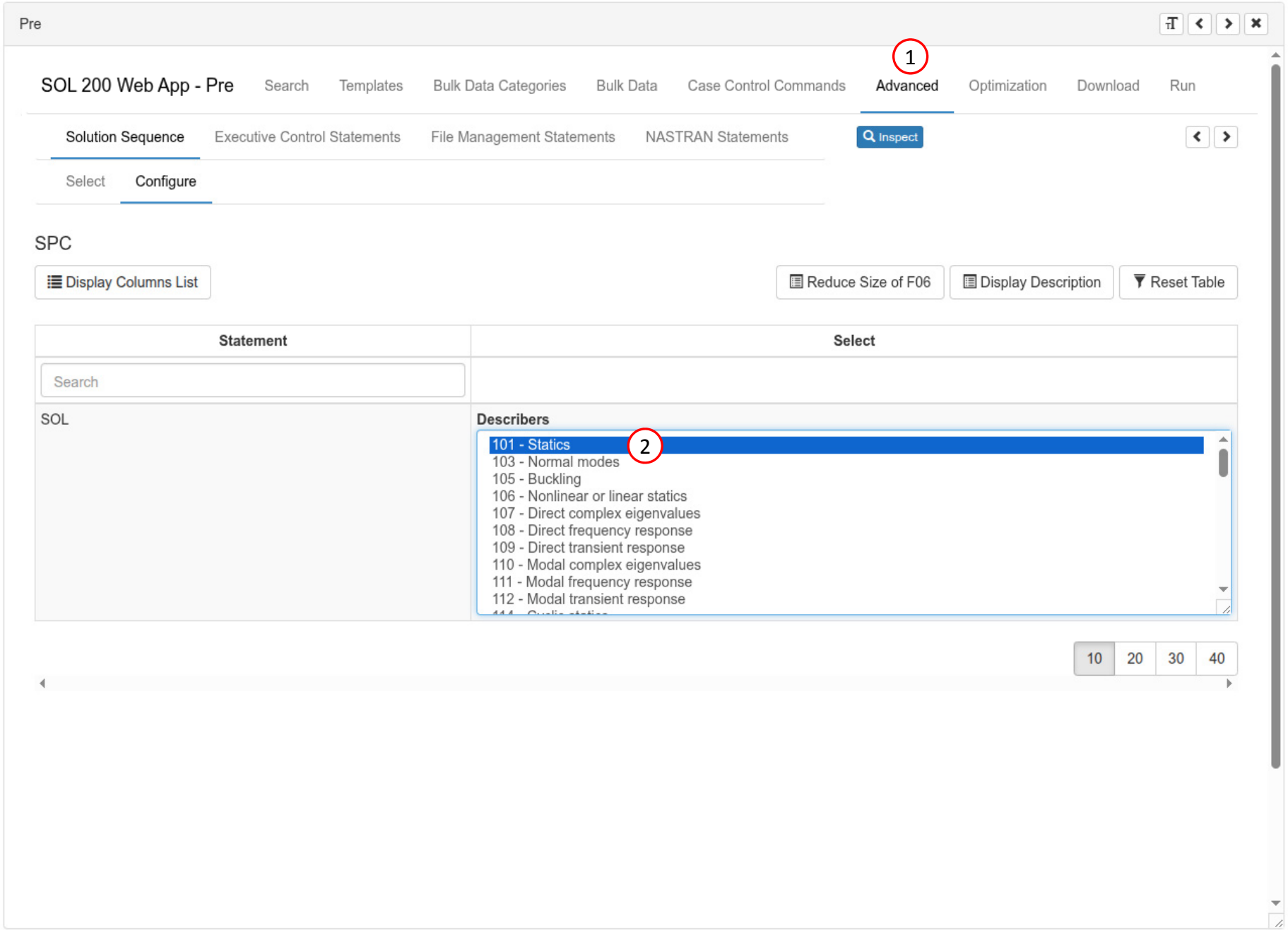
The Nastran model is now downloaded and saved to your desktop.



Linear Statics Analysis

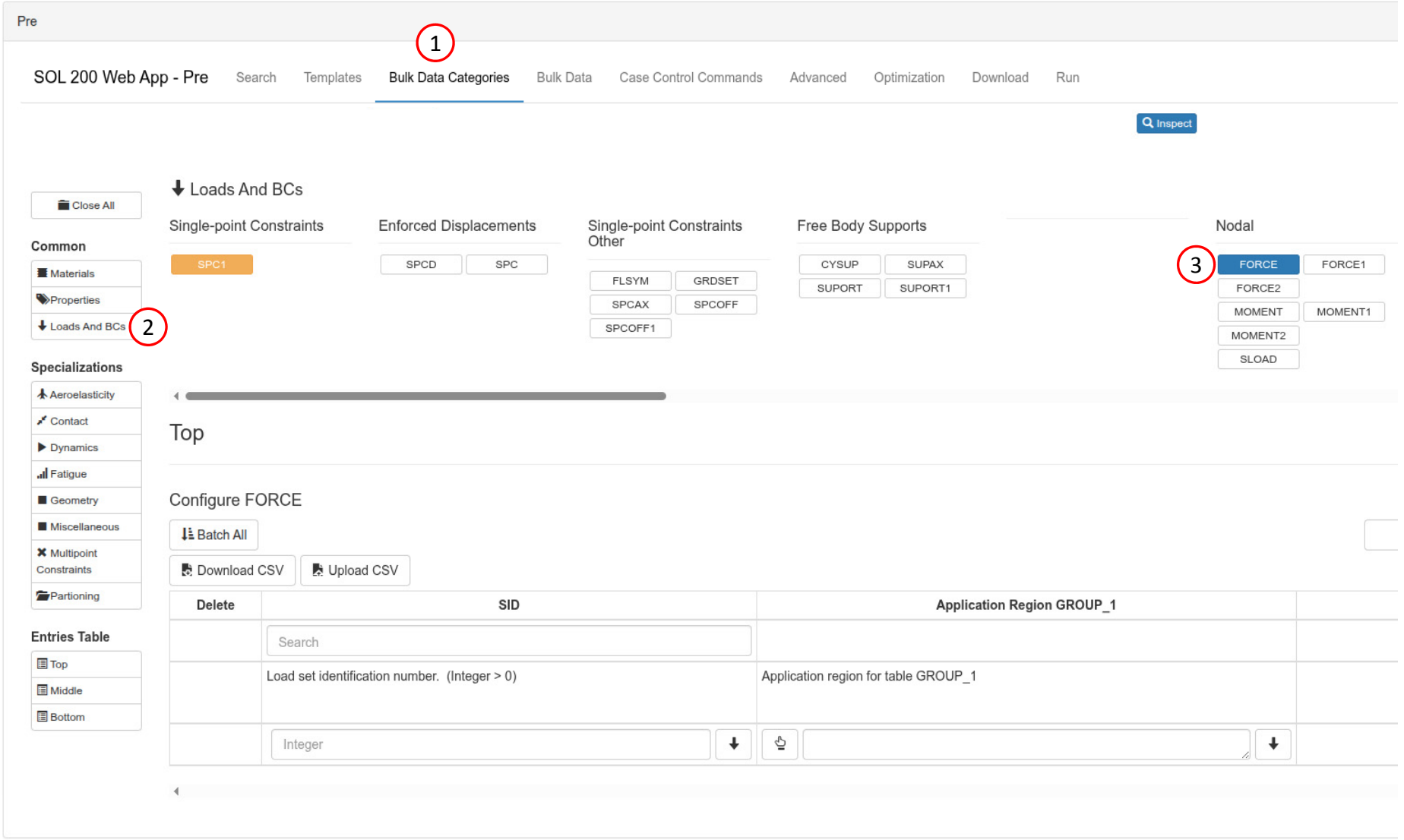
SOL 101 - Statics

1. Click Advanced
2. Select: 101 - Statics



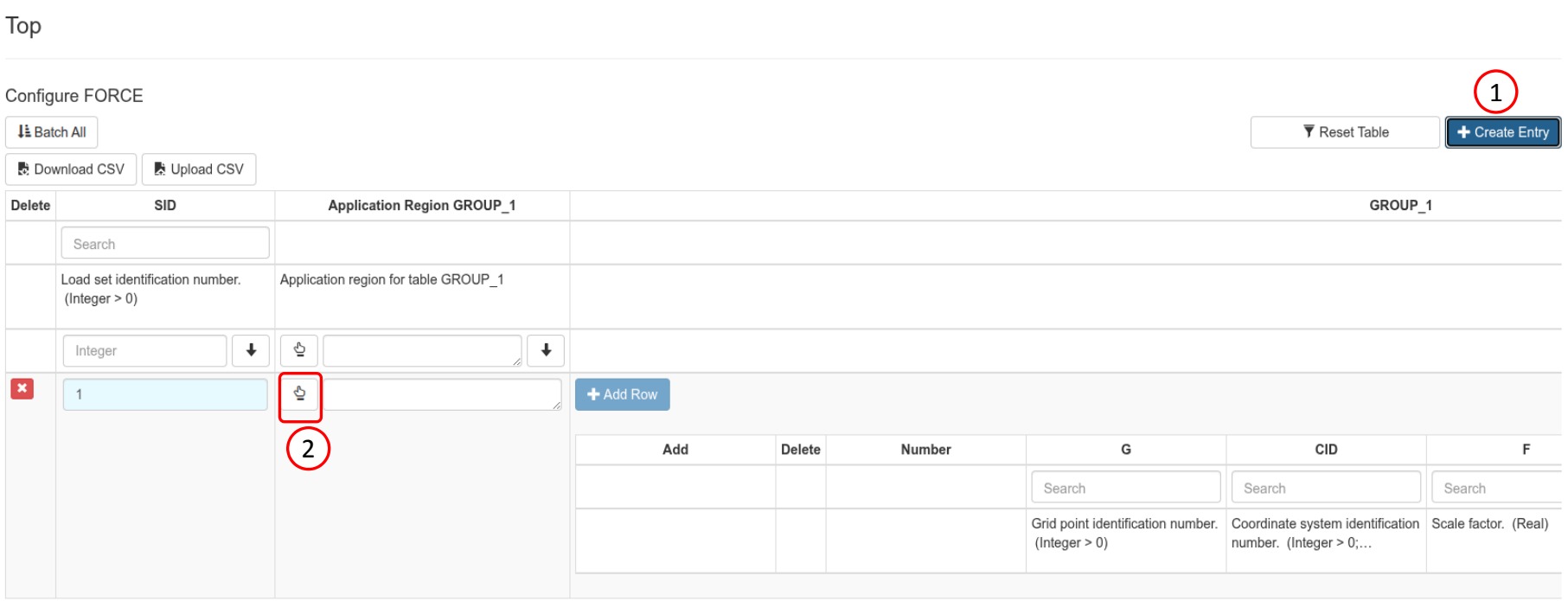
FORCE - Static Force

1. Click Bulk Data Categories
2. Click Loads and BCs
3. Click FORCE



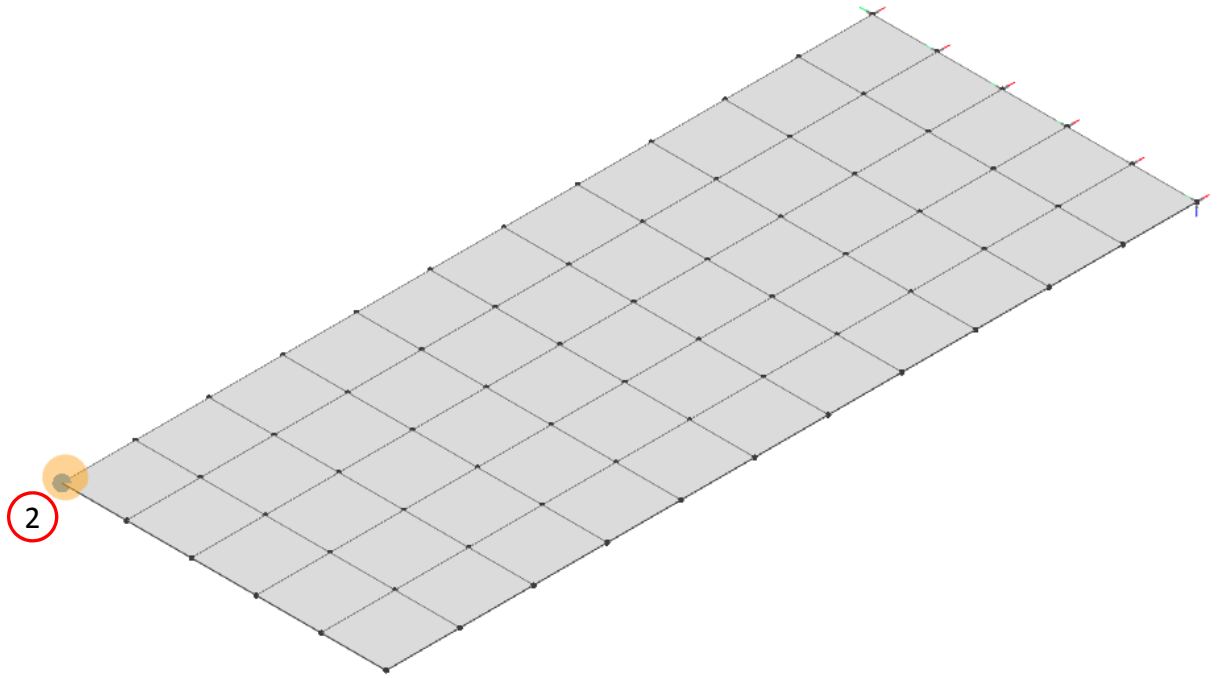
FORCE - Static Force

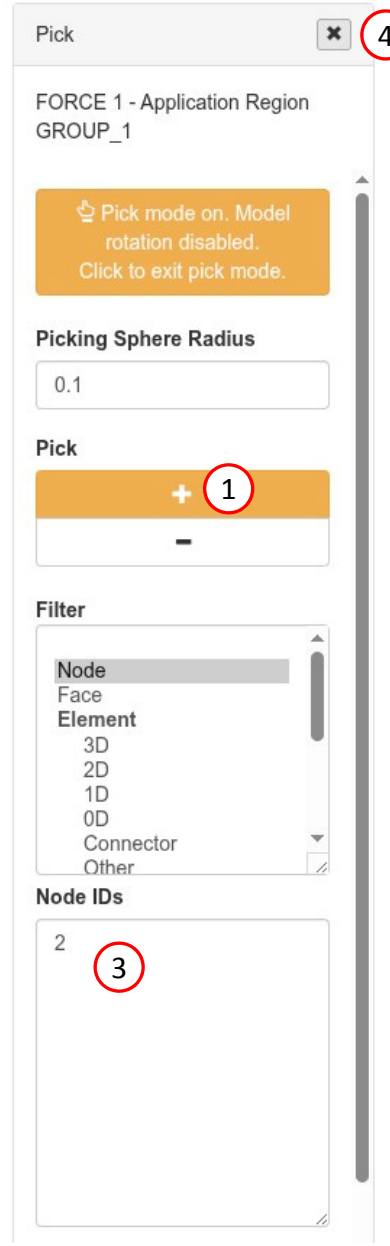
1. Click Create Entry
2. Click the Pick button



FORCE - Static Force

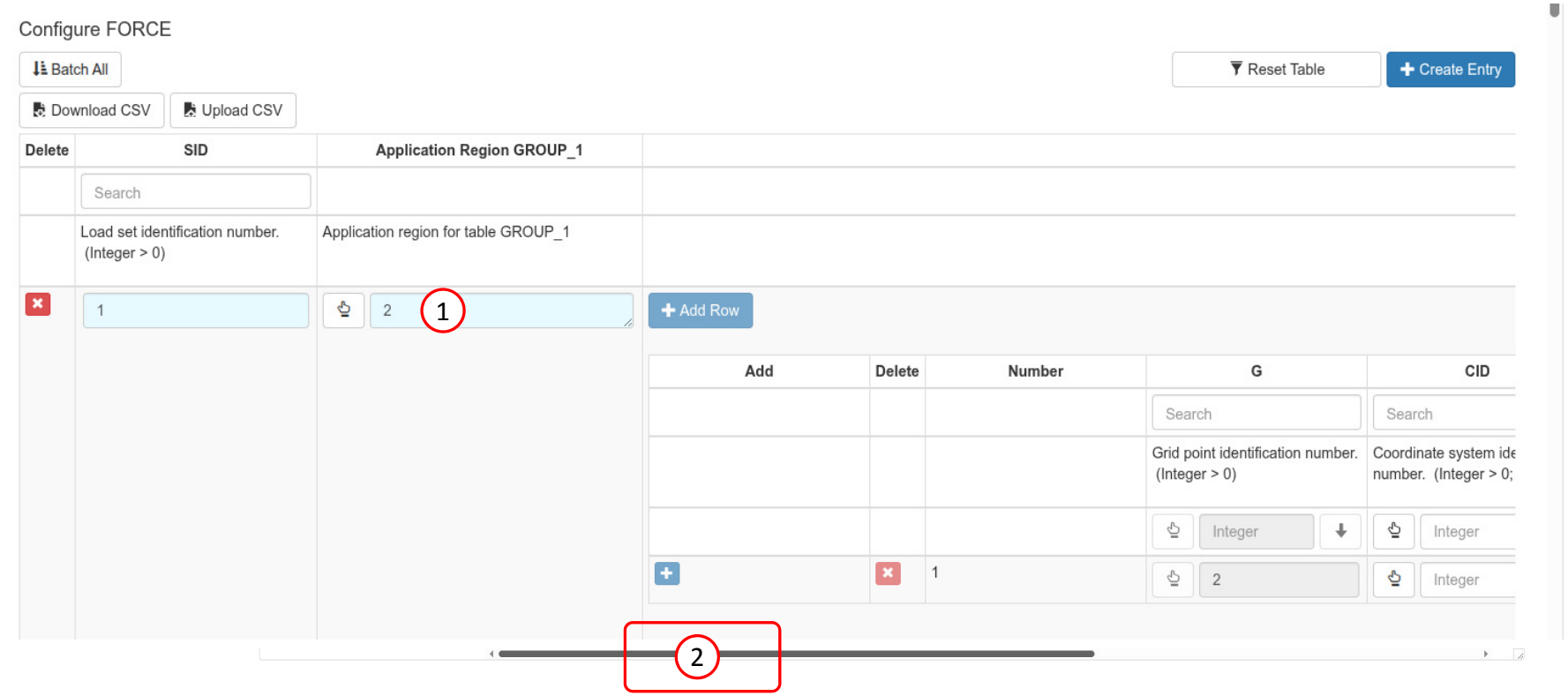
1. Click +
2. Move the mouse cursor to the indicated region. Click and hold the left mouse button, then drag the picking sphere to select
3. The selected node ID is stored
4. Click X

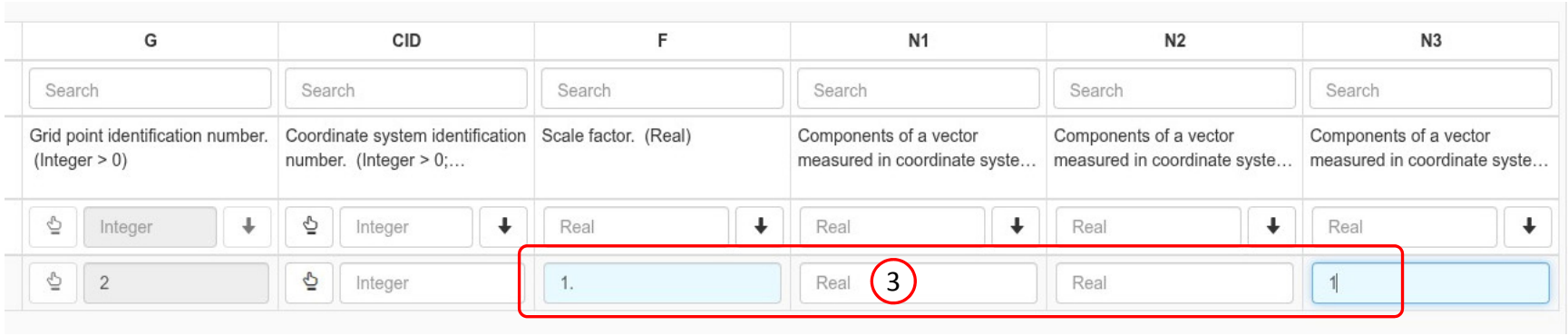




FORCE - Static Force

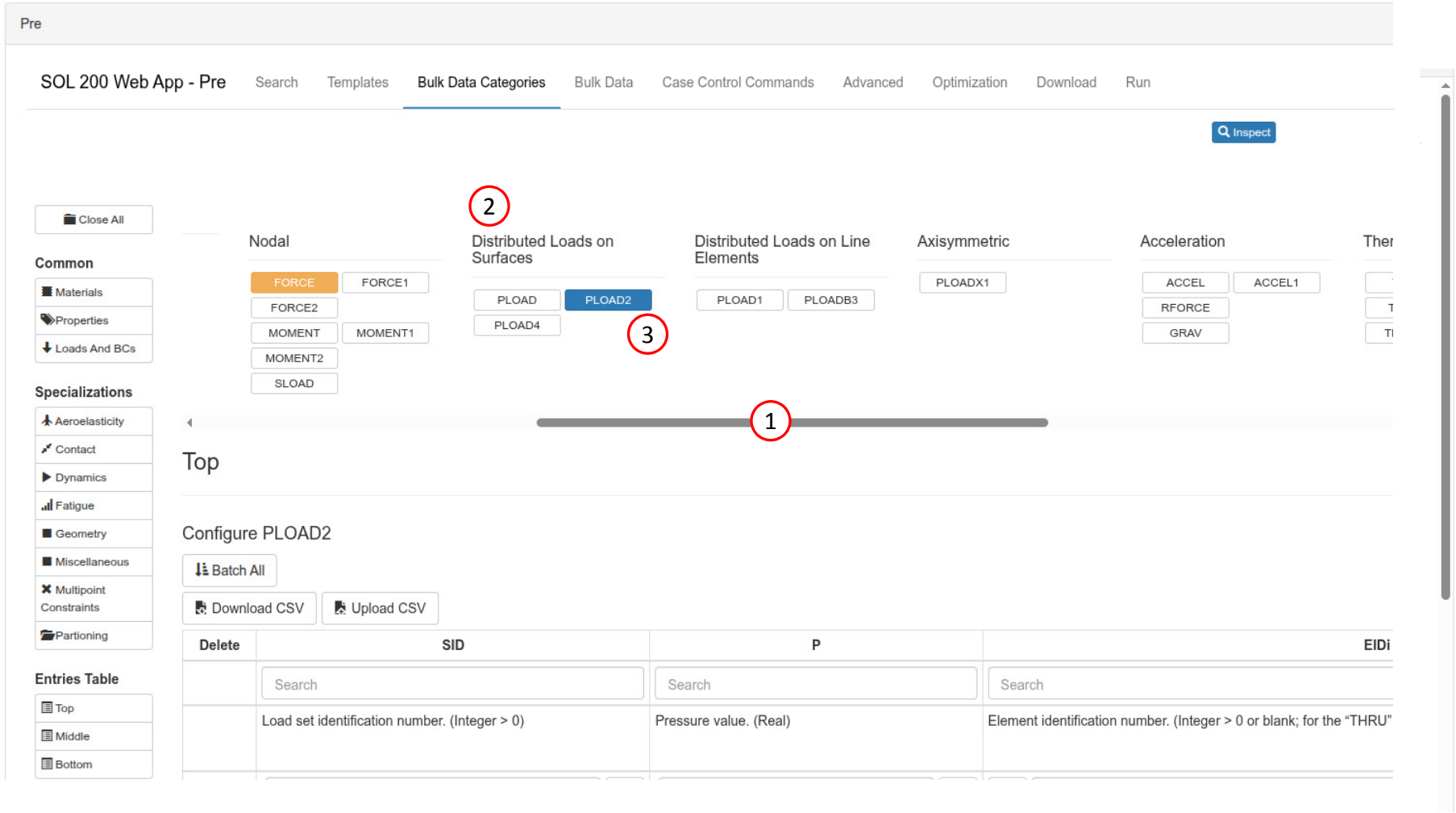
1. The selected node ID is now stored
2. Use the horizontal scroll bar to find columns G, CID, F, N1, N2, N3
3. Supply the following values
   - In row 1 of the table, supply the following values.
     - F: 1.
     - N3: 1.





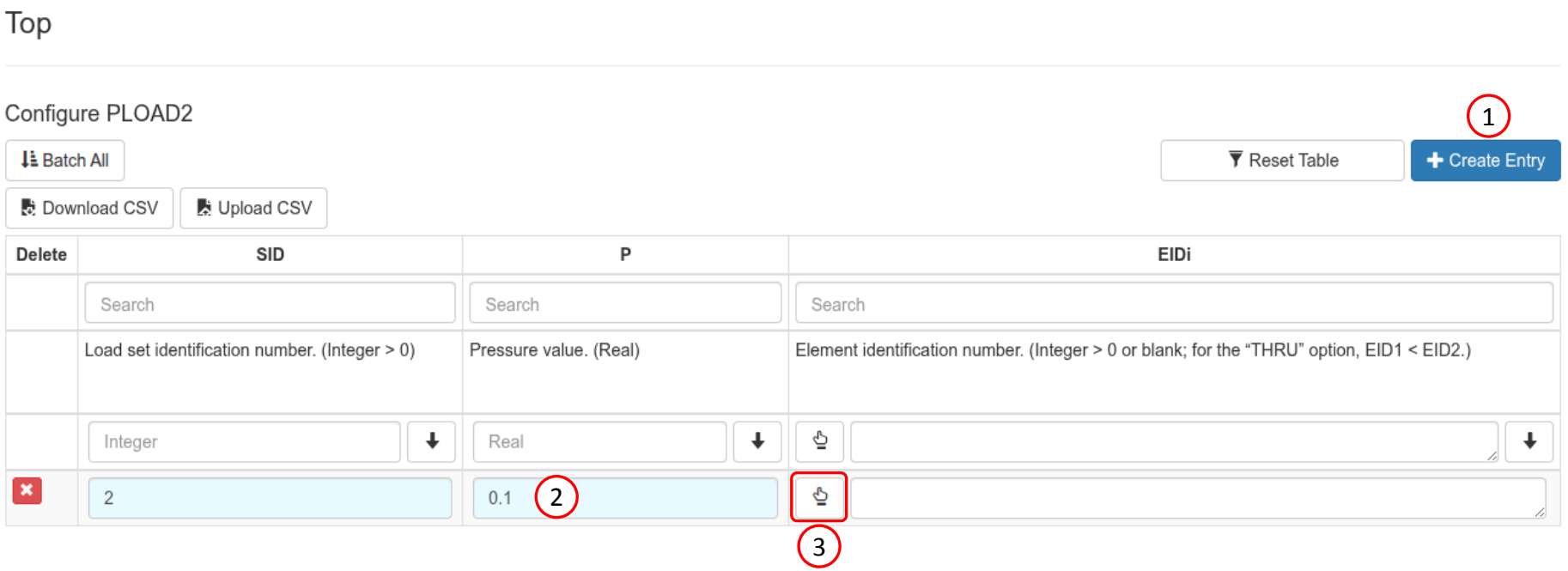
PLOAD2 - Uniform Normal Pressure Load on a Surface Element

1. Use the indicated scroll bar to scroll to the right
2. The following section is now visible: Distributed Loads on Surfaces
3. Click PLOAD2



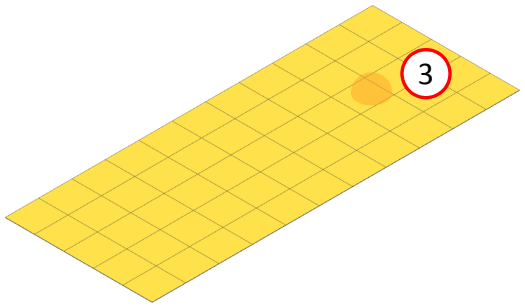
PLOAD2 - Uniform Normal Pressure Load on a Surface Element

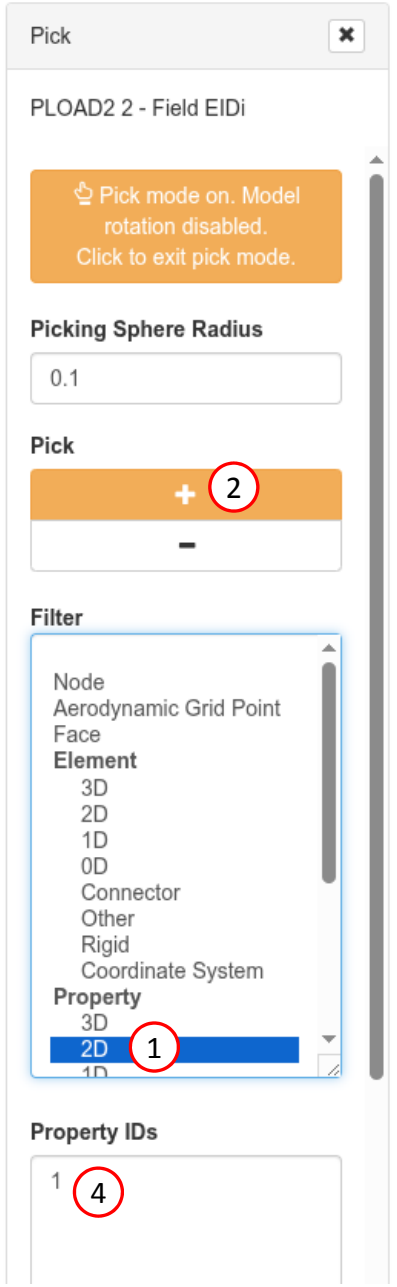
1. Click Create Entry
2. Supply the following values
   - P: 0.1
3. Click Pick

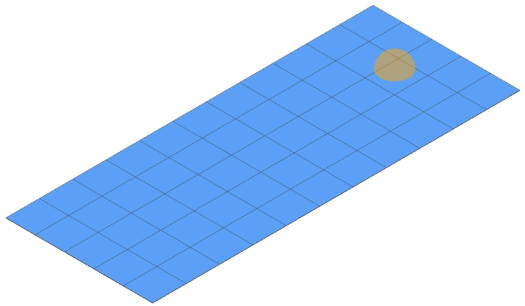


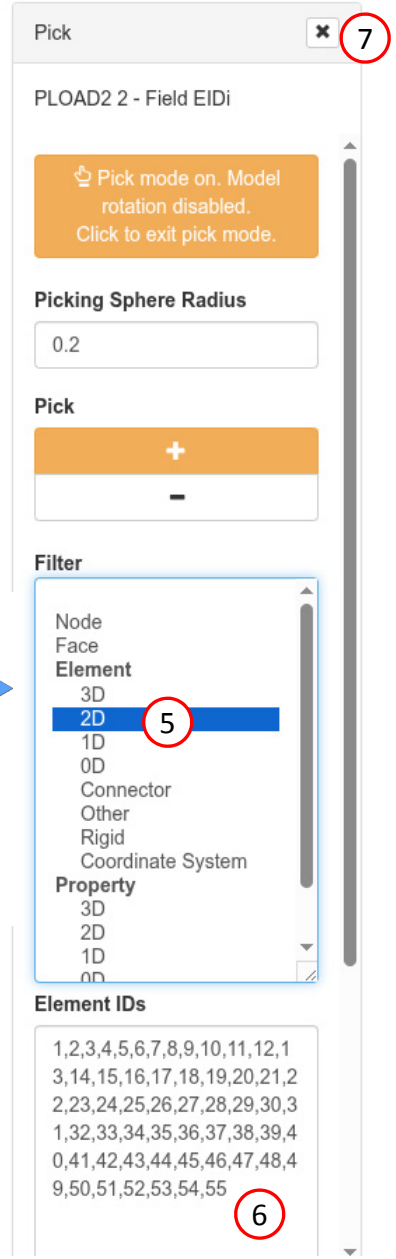
PLOAD2 - Uniform Normal Pressure Load on a Surface Element

1. Set Filter to Property > 2D
2. Click +
3. Move the mouse cursor to the indicated region. Click and hold the left mouse button, then drag the picking sphere to select.
4. The selected property ID is stored
5. Change Filer to Element > 2D
6. The element IDs associated with PSHELL ID=1 are automatically determined and stored
7. Click X





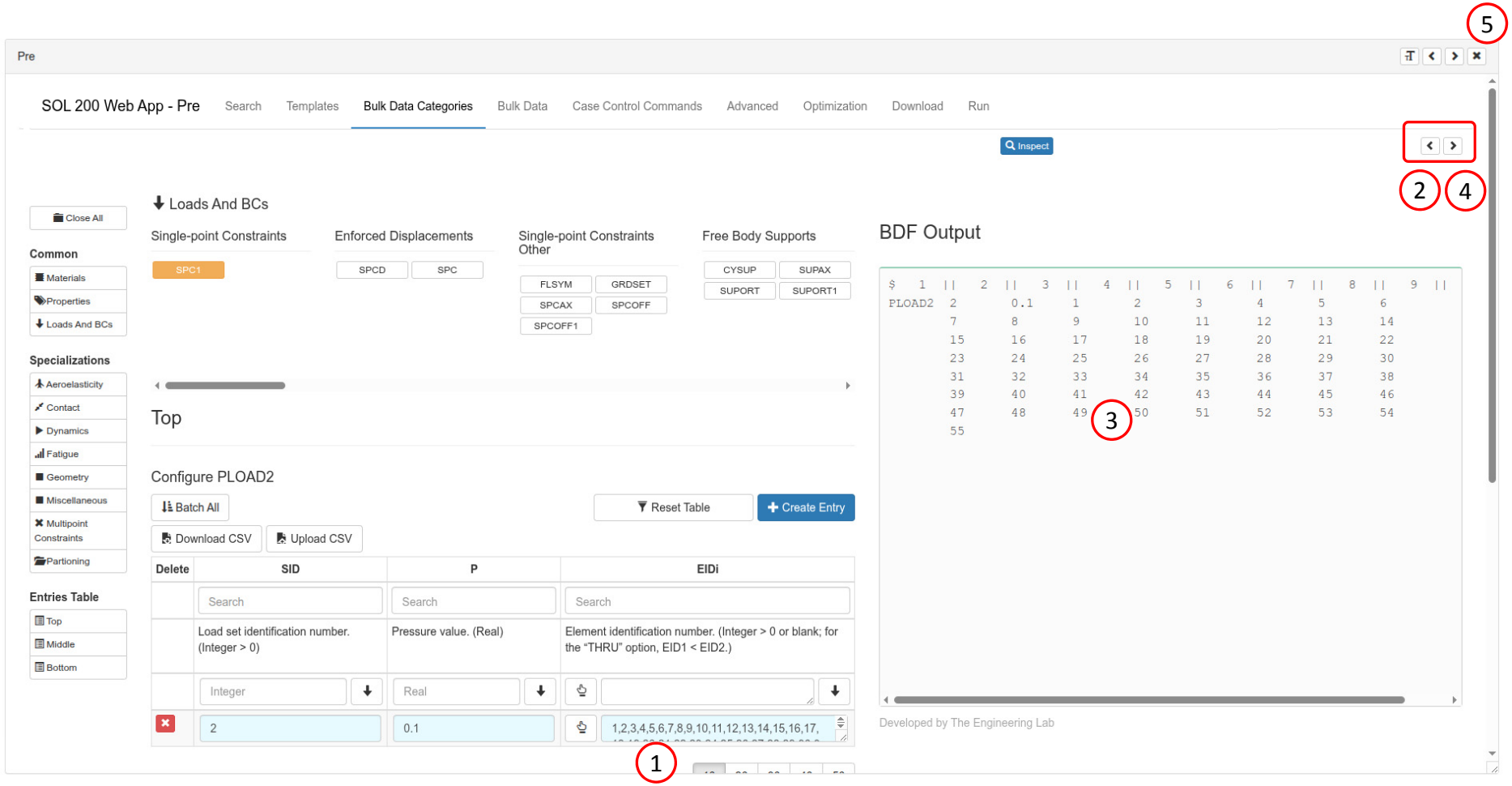




BDF Output

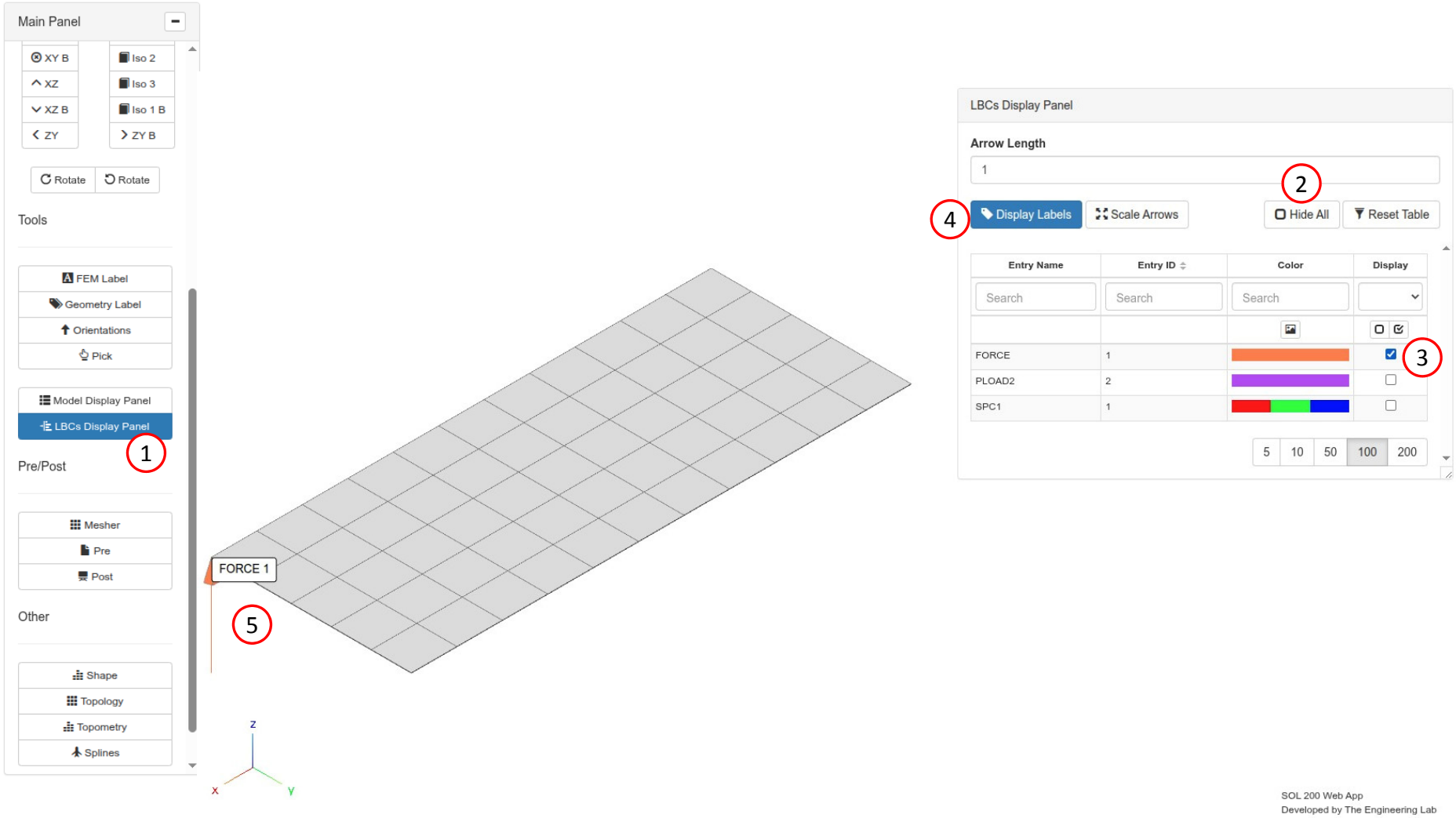
1. The selected element IDs are now stored
2. Use the indicated buttons to display the BDF Output section
3. The actual text of the bulk data entry is displayed.
4. Use the indicated buttons to hide the BDF Output section
5. Click X

The BDF Output section is ideal for learning and debugging MSC Nastran models. Any changes made in the interface are immediately updated in the BDF Output section.



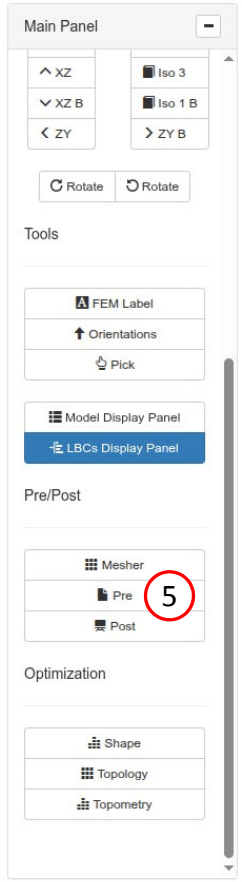
LBCs Display Panel

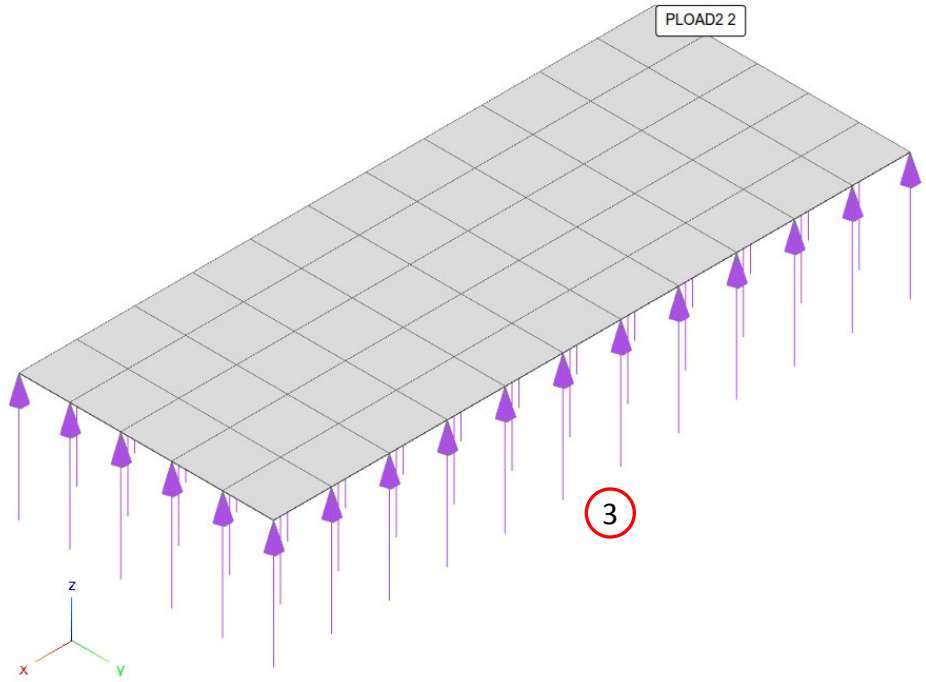
1. Click LBCs Display Panel
2. Click Hide All
3. Mark the checkbox for Entry Name=FORCE and Entry ID=1
4. Click Display Labels
5. Inspect the load

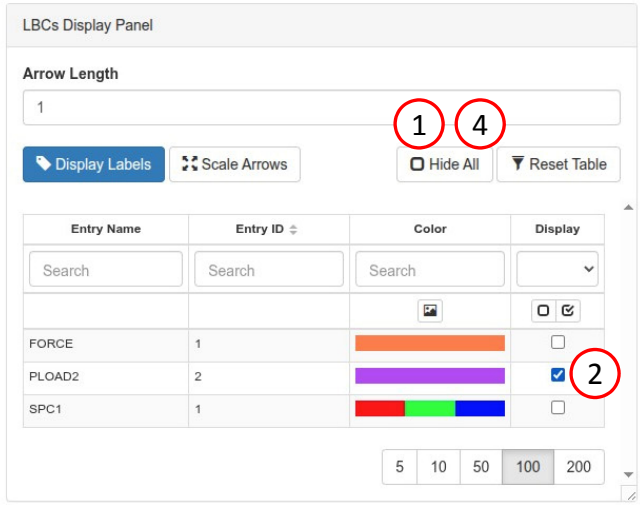


LBCs Display Panel

1. Click Hide All
2. Mark the checkbox for Entry Name=PLOAD2 and Entry ID=2
3. Inspect the load
4. Click Hide All
5. Click Pre



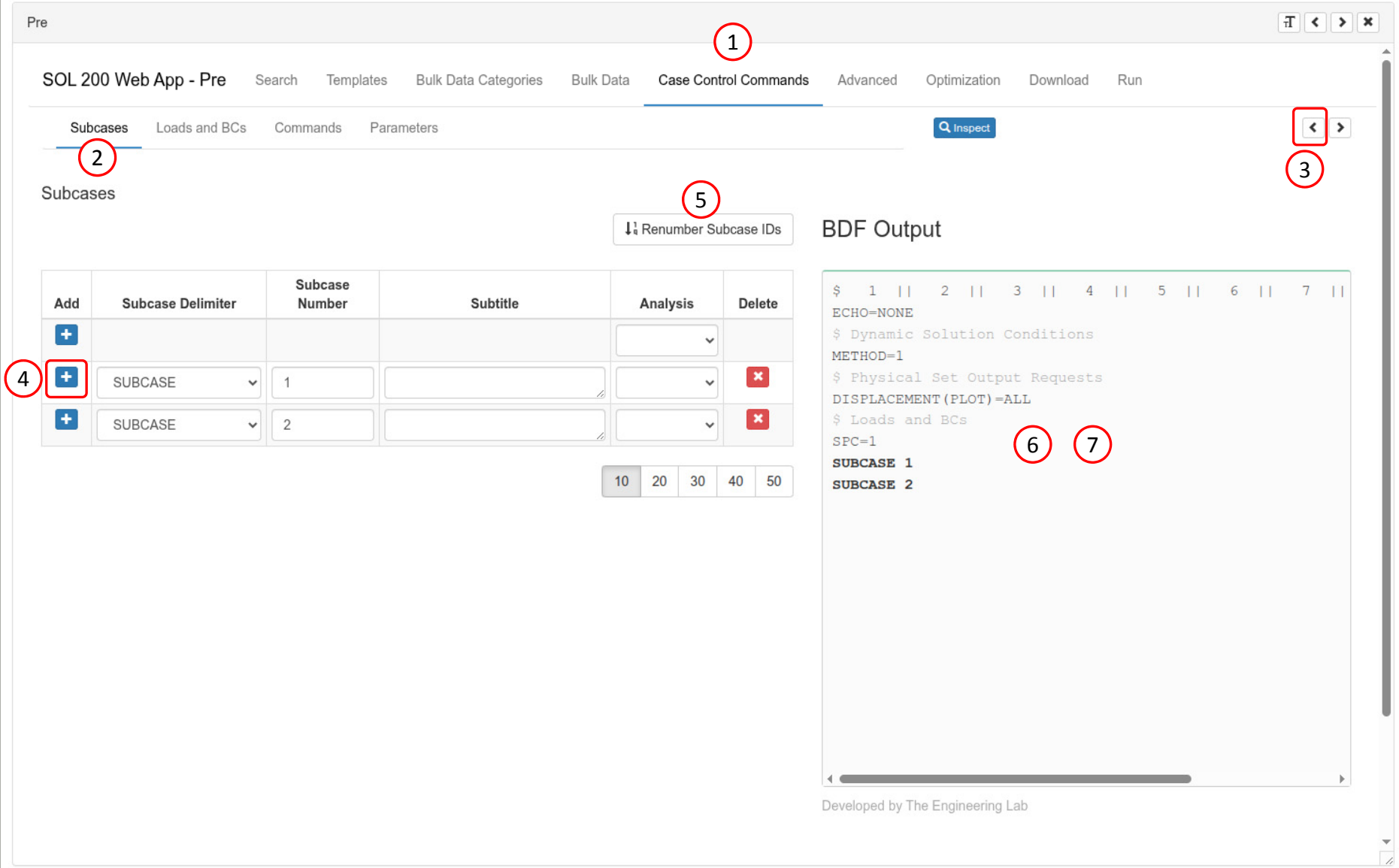




SOL 200 Web App
Developed by The Engineering Lab

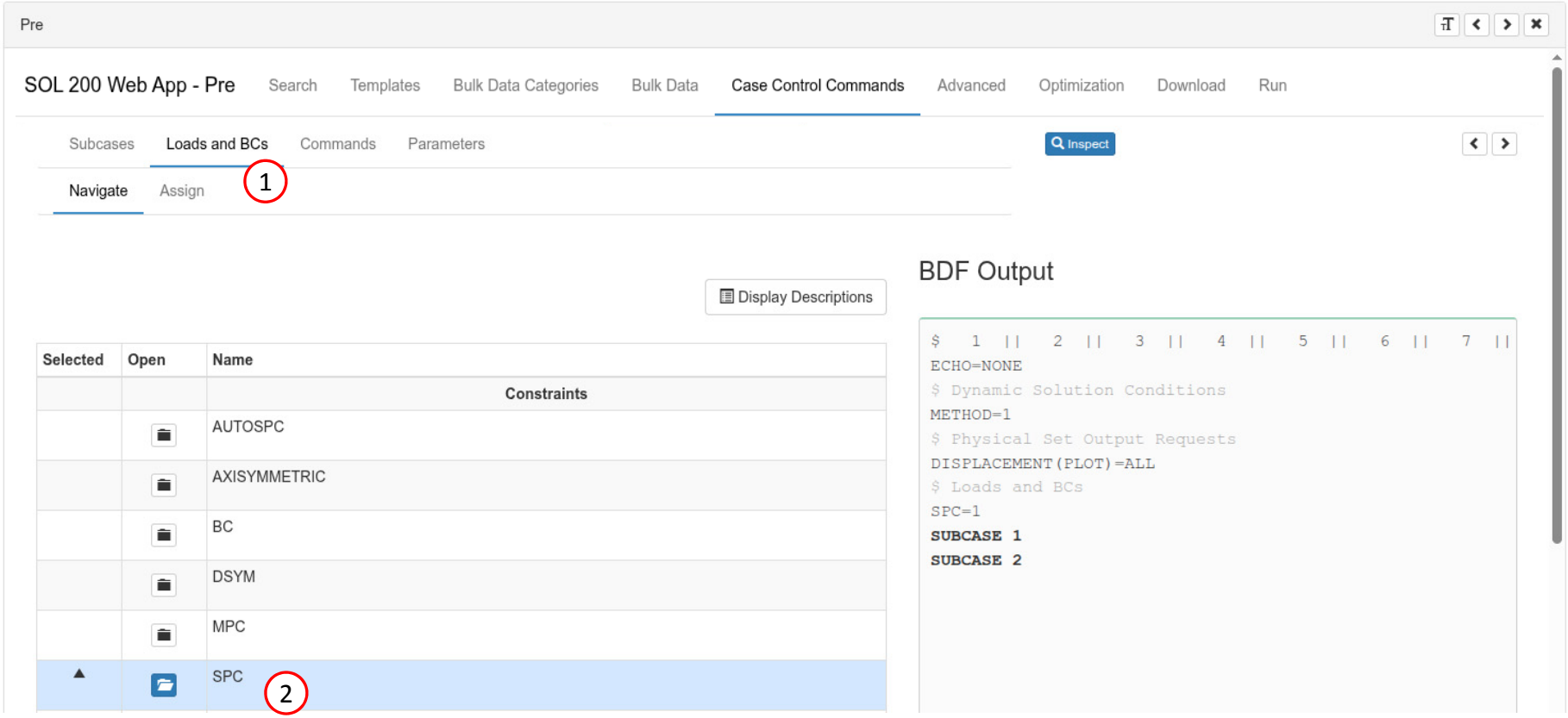
Subcases

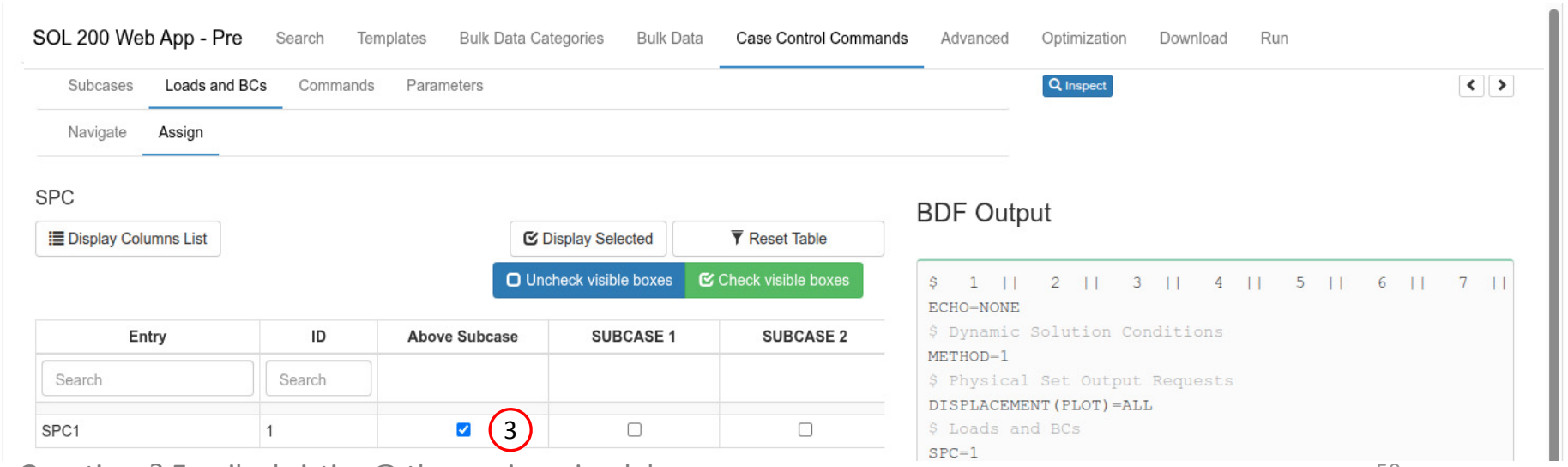
1. Click Case Control Commands
2. Click Subcases
3. Use the indicated buttons to display the BDF Output section
4. The actual text of the case control section is displayed
5. Click Add to add a new subcase
6. Click Renumber Subcase IDs
   - Subcase identification numbers must be in ascending order
7. The case control section is immediately updated with any change



SPC - Single Point Constraint Set Selection

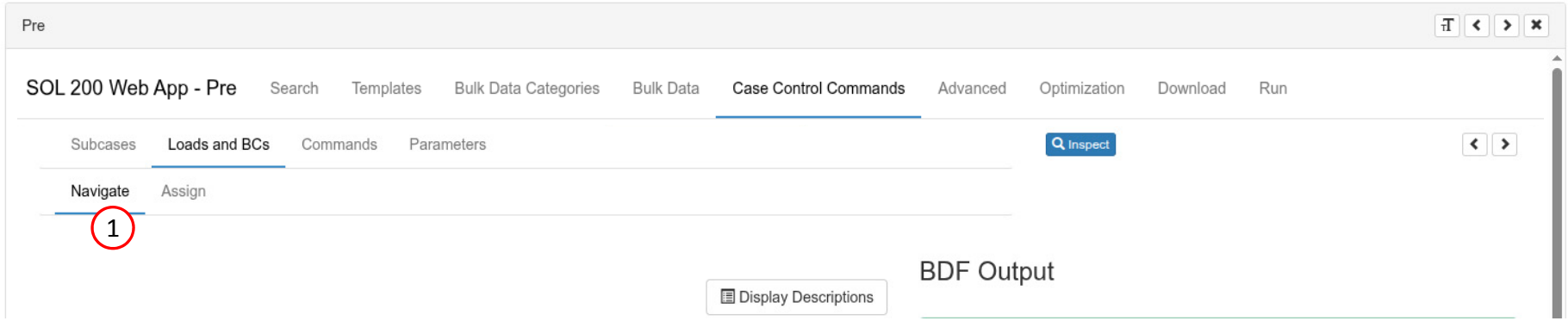
1. Click Loads and BCs
2. Select row SPC
3. Confirm SPC ID=1 is assigned to the Above Subcase column

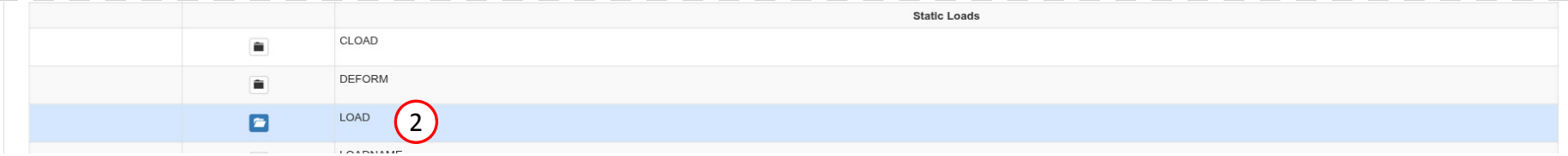


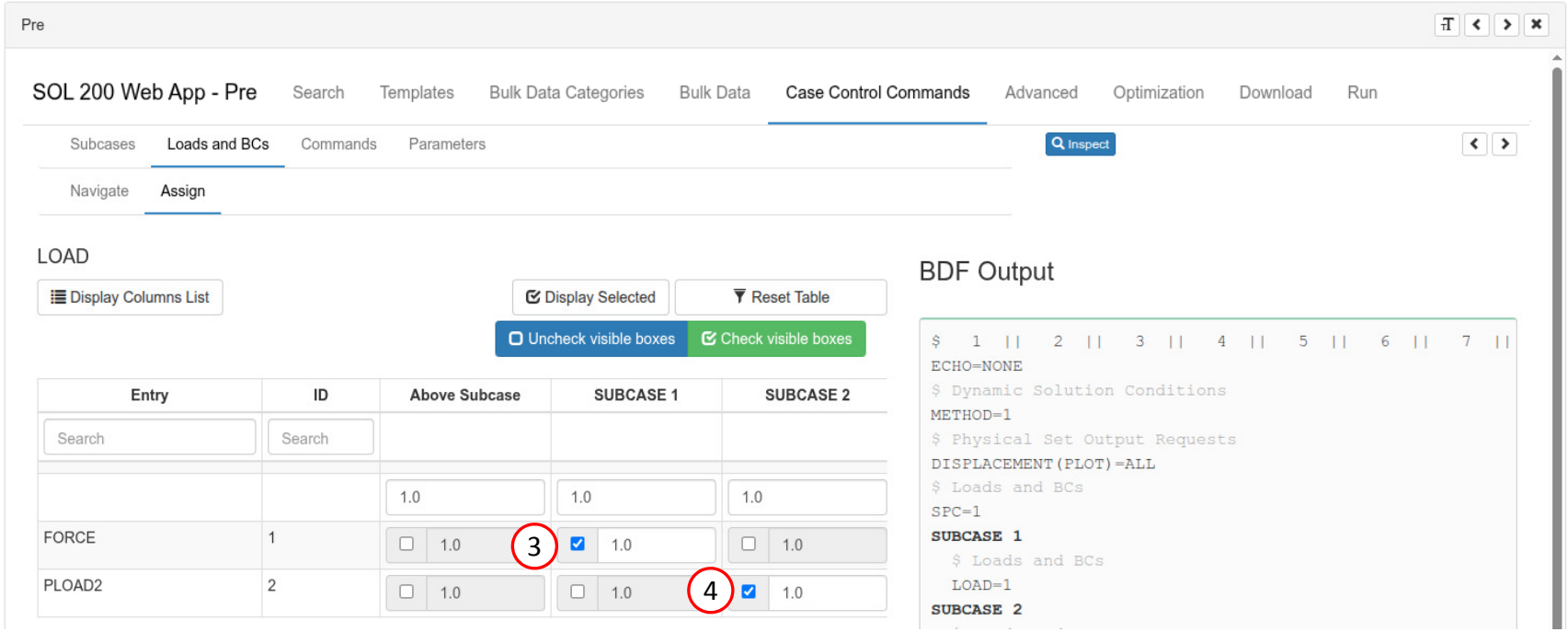


LOAD - External Static Load Set Selection

1. Click Navigate
2. Select row LOAD
3. Assign FORCE ID=1 to SUBCASE 1.
4. Assign PLOAD2 ID=2 to SUBCASE 2.

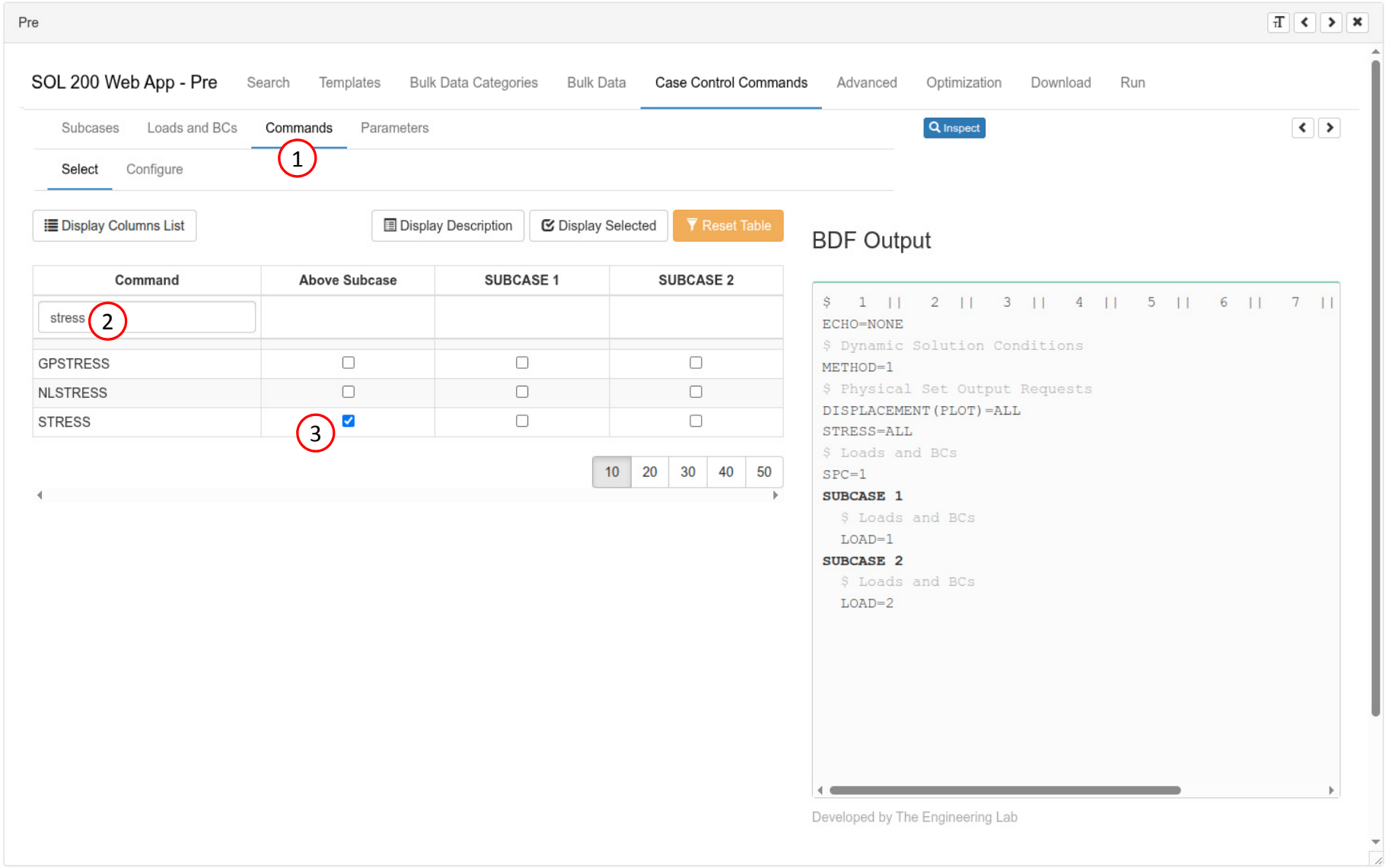






Stress Output

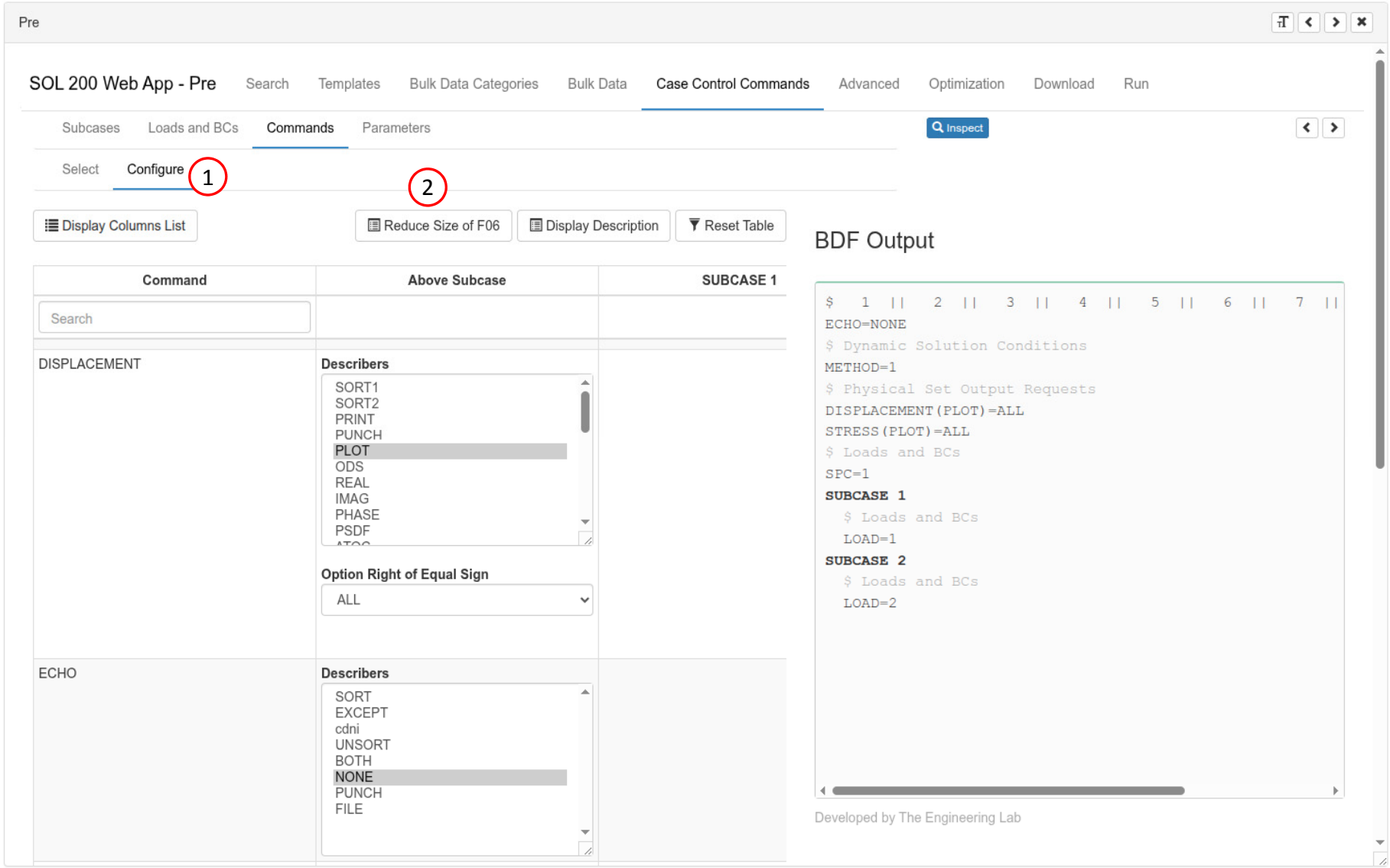
1. Click Commands
2. Search the table for stress
3. Select the indicated checkbox



Stress Output

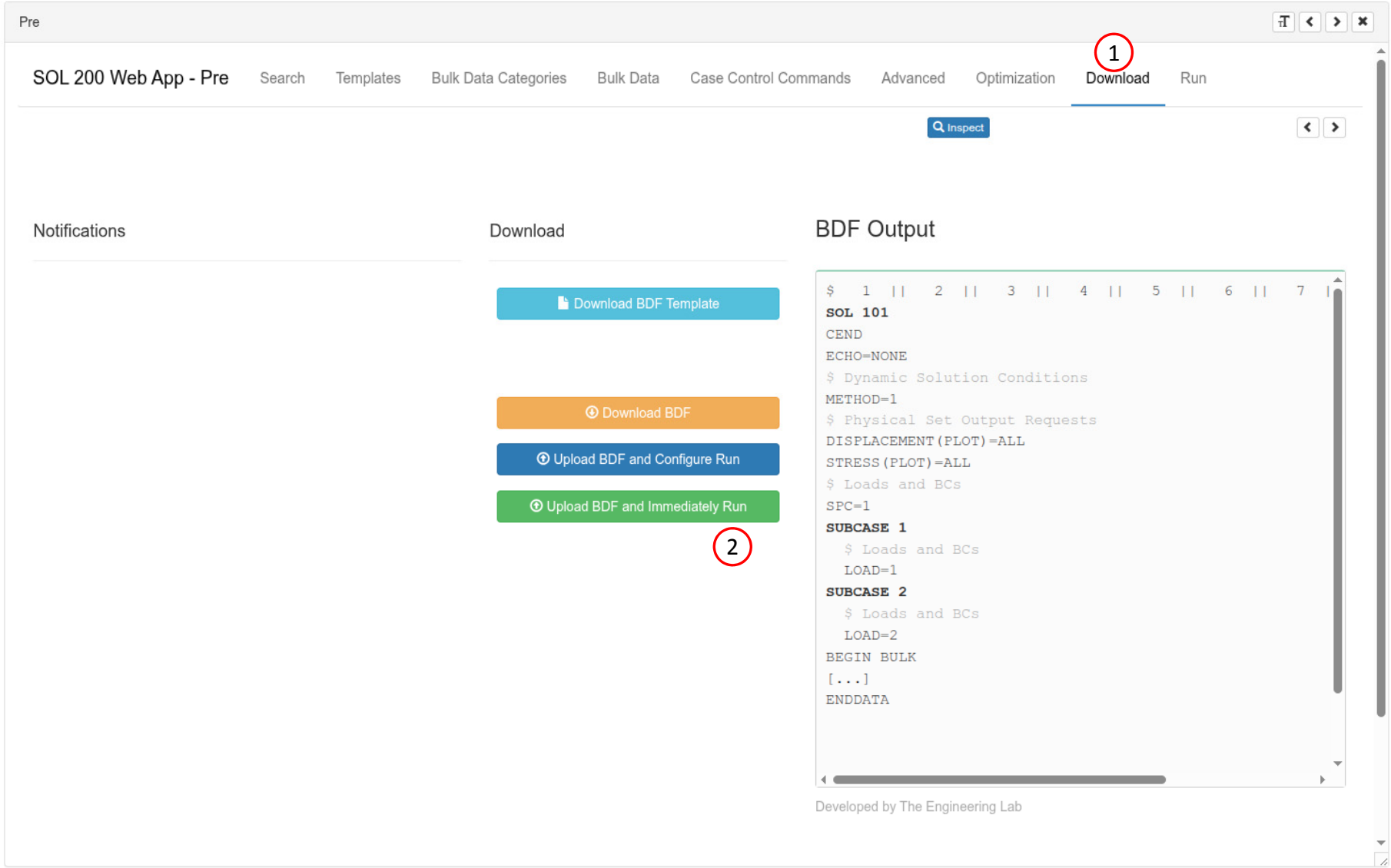
1. Click Configure
2. Click Reduce Size of F06
   - This option automatically adds ECHO=NONE and selects the PLOT describer for the commands. This option significantly reduces the size of the F06 file.

The Configure section provides complete access to configuring case control commands.



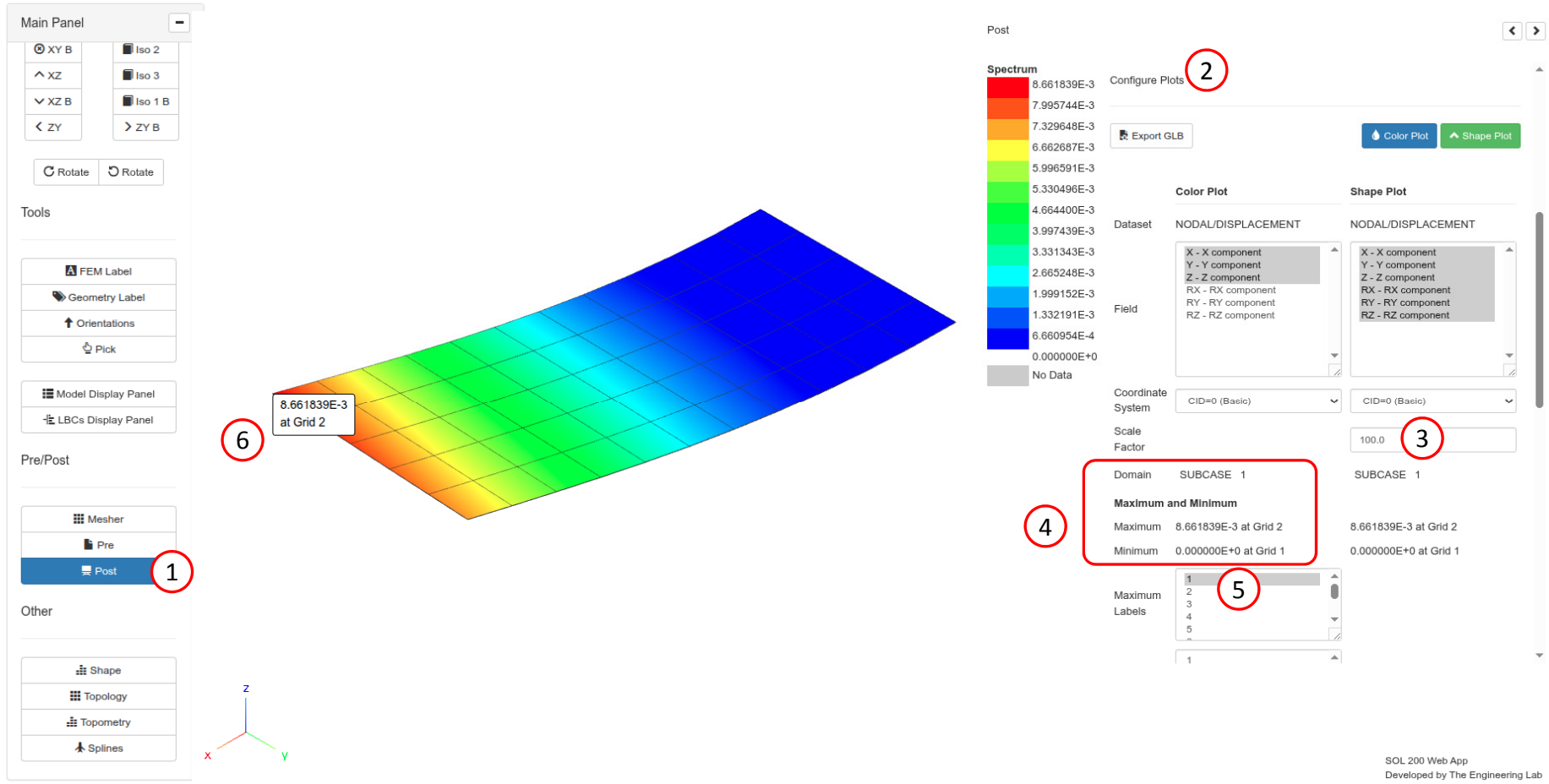
Run

1. Click Download
2. Click Upload BDF and Immediately Run



Post

1. After the analysis, the Post panel is automatically opened. The Post panel may be closed or opened by clicking on the indicated button.
2. Vertically scroll to section Configure Plots
3. Set the Scale Factor to 100.0
4. The SUBCASE ID and maximum deflection are display
5. Select the first maximum label
6. A label displays the location and the maximum displacement



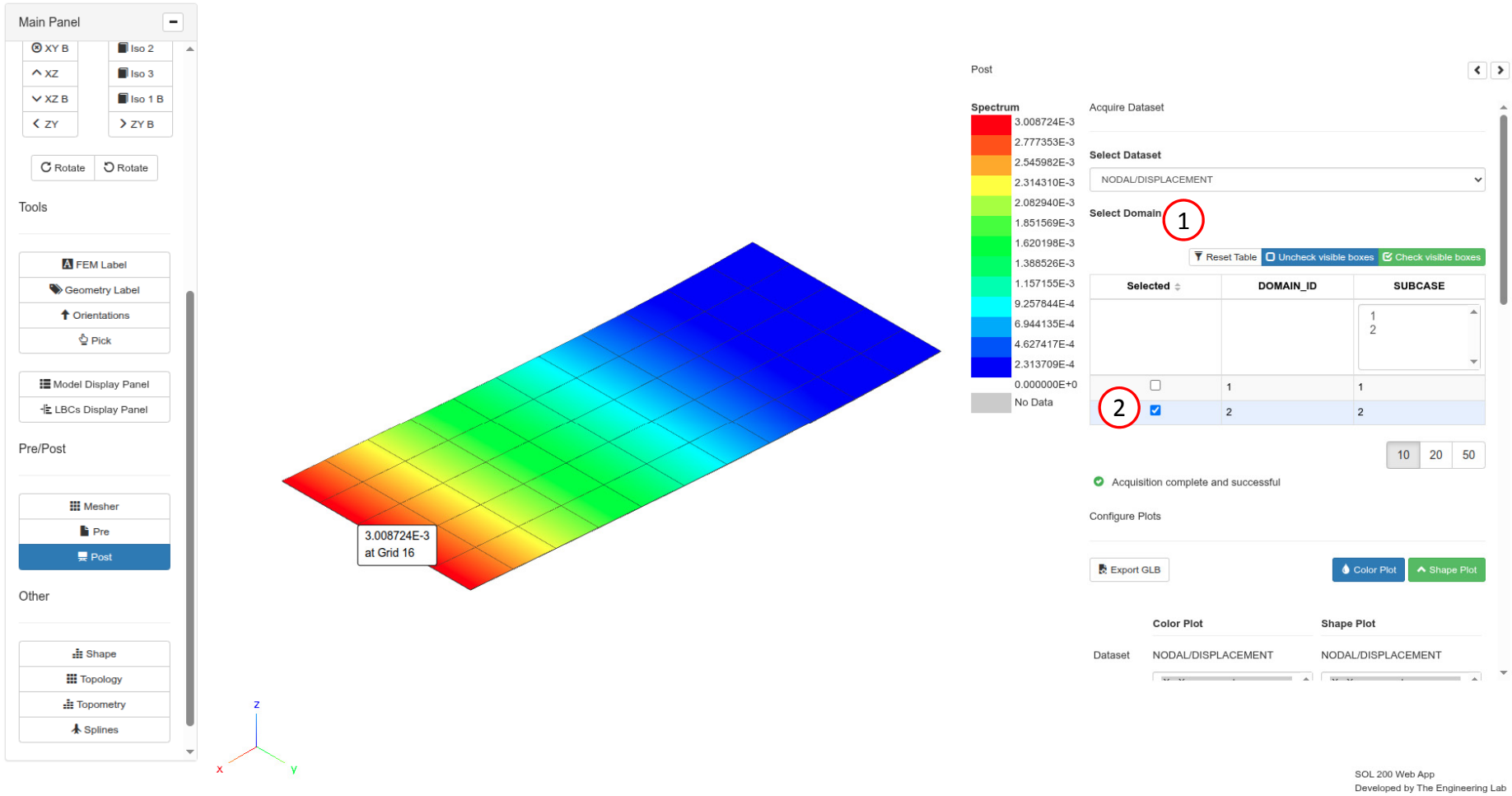
Post

1. Vertically scroll to section Select Domain
2. Select different checkboxes to see the corresponding deformation.

When performing normal modes analysis, the perturbed shape is a eigenvector/mode shape.

When performing statics analysis, the perturbed shape is a deformation.

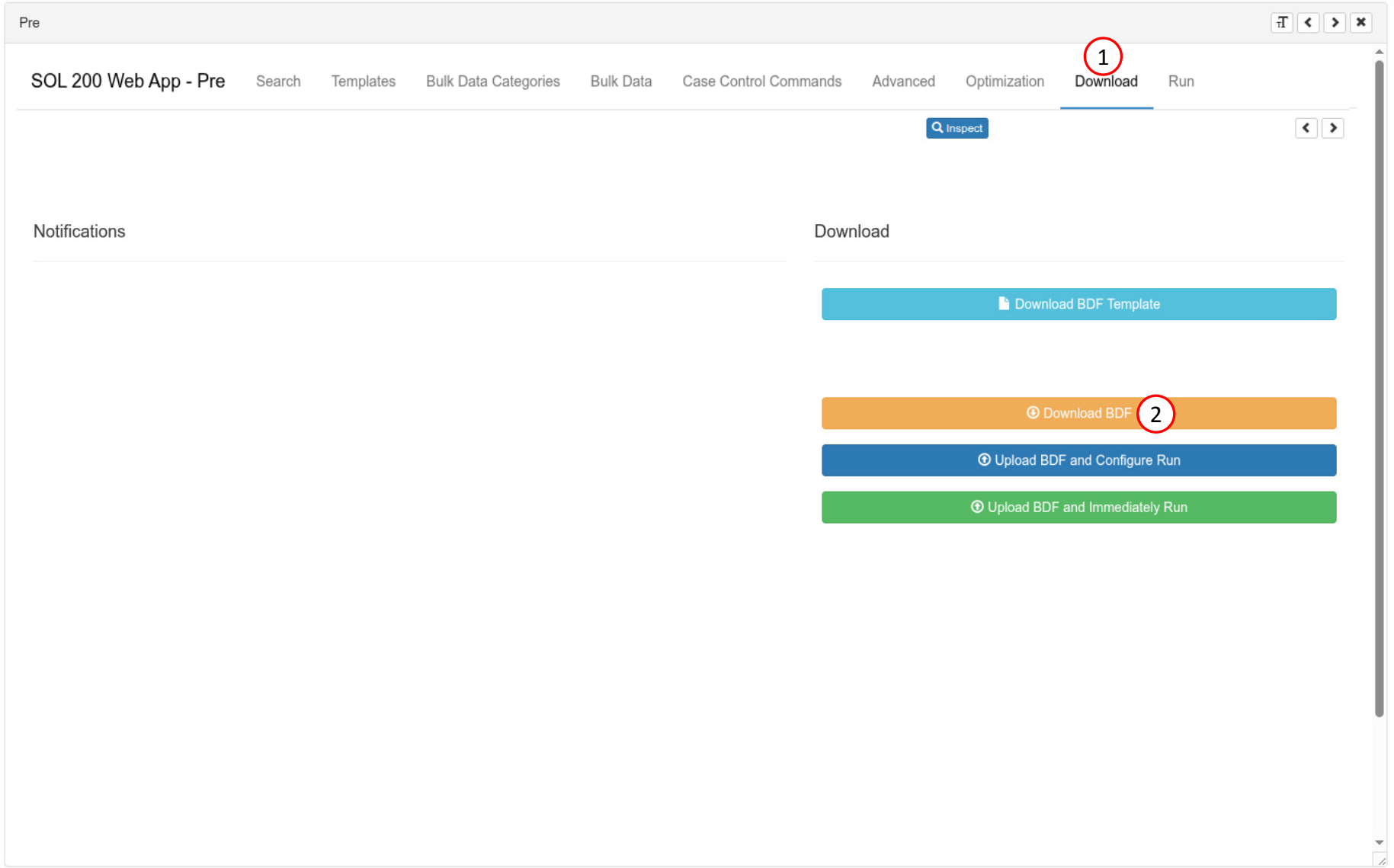
Students often use 'displacement' to refer to 'mode shapes' and is incorrect. The terms have different meaning and should be used appropriately.



Download

1. Navigate to Pre > Download
2. Click Download BDF

The Nastran model is now downloaded and saved to your desktop.



Saving Progress

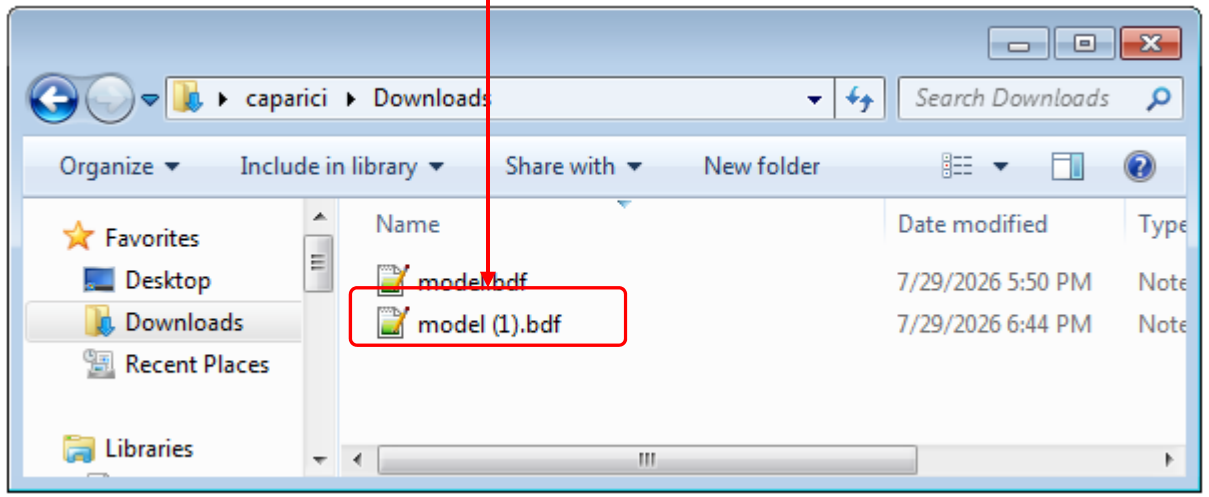
Saving Progress

The bulk data file (.bdf or BDF) contains the Nastran model and is critical to any Nastran analysis.

All work is saved or opened from this bulk data file.

1. When the button Download BDF is clicked, a .bdf file is saved to your desktop.

Keep these files in a safe location.

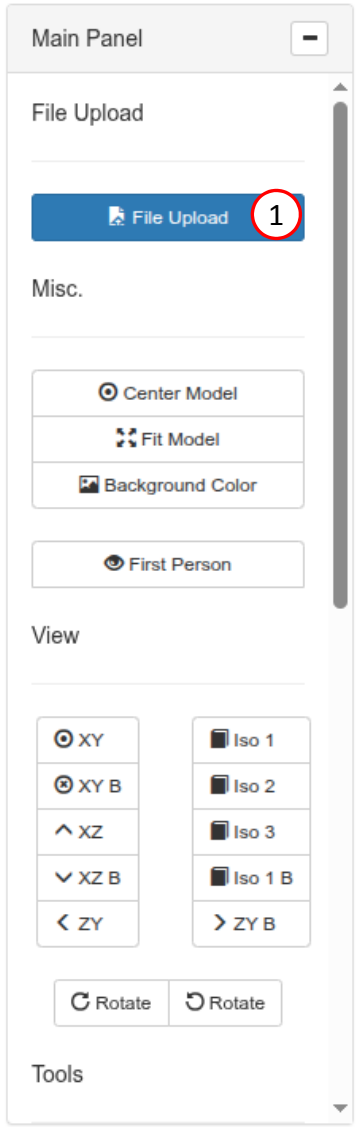


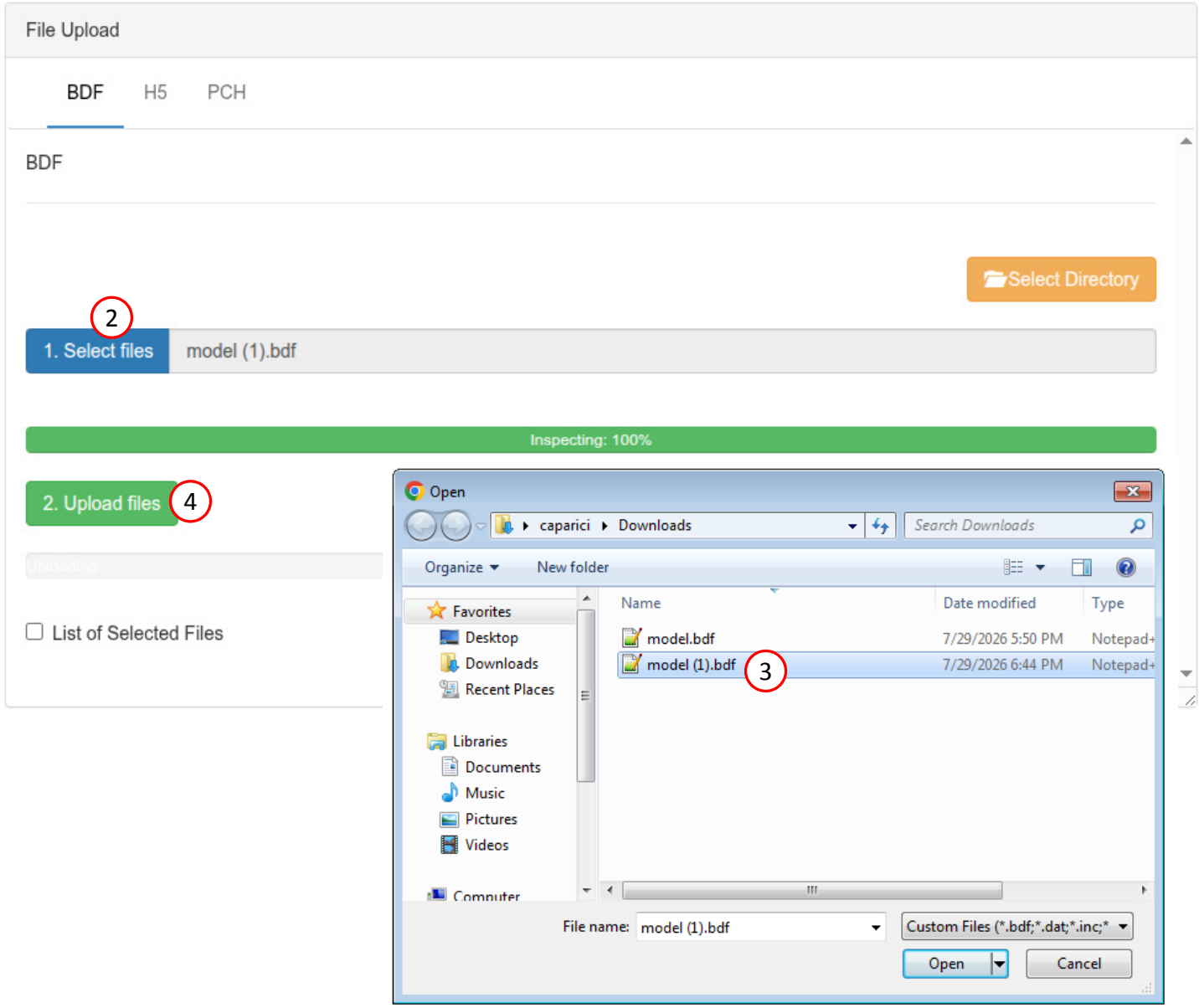
Open the Viewer

1. Click on the indicated link

Opening a saved .bdf file

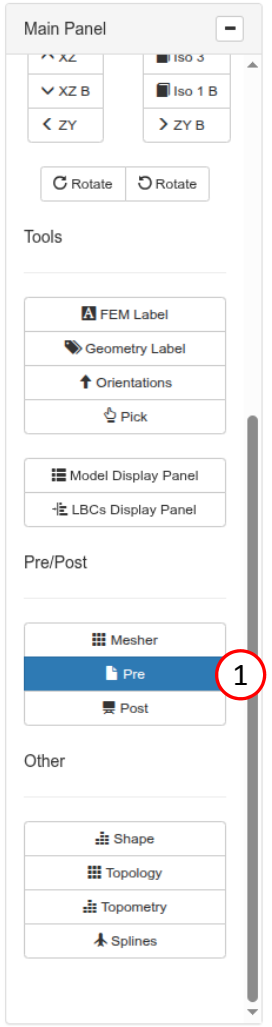
1. Click File Upload
2. Click Select files
3. Select a .bdf file
   - Example: File model (1).bdf contains he the configuration for the linear statics analysis.
4. Click Upload files

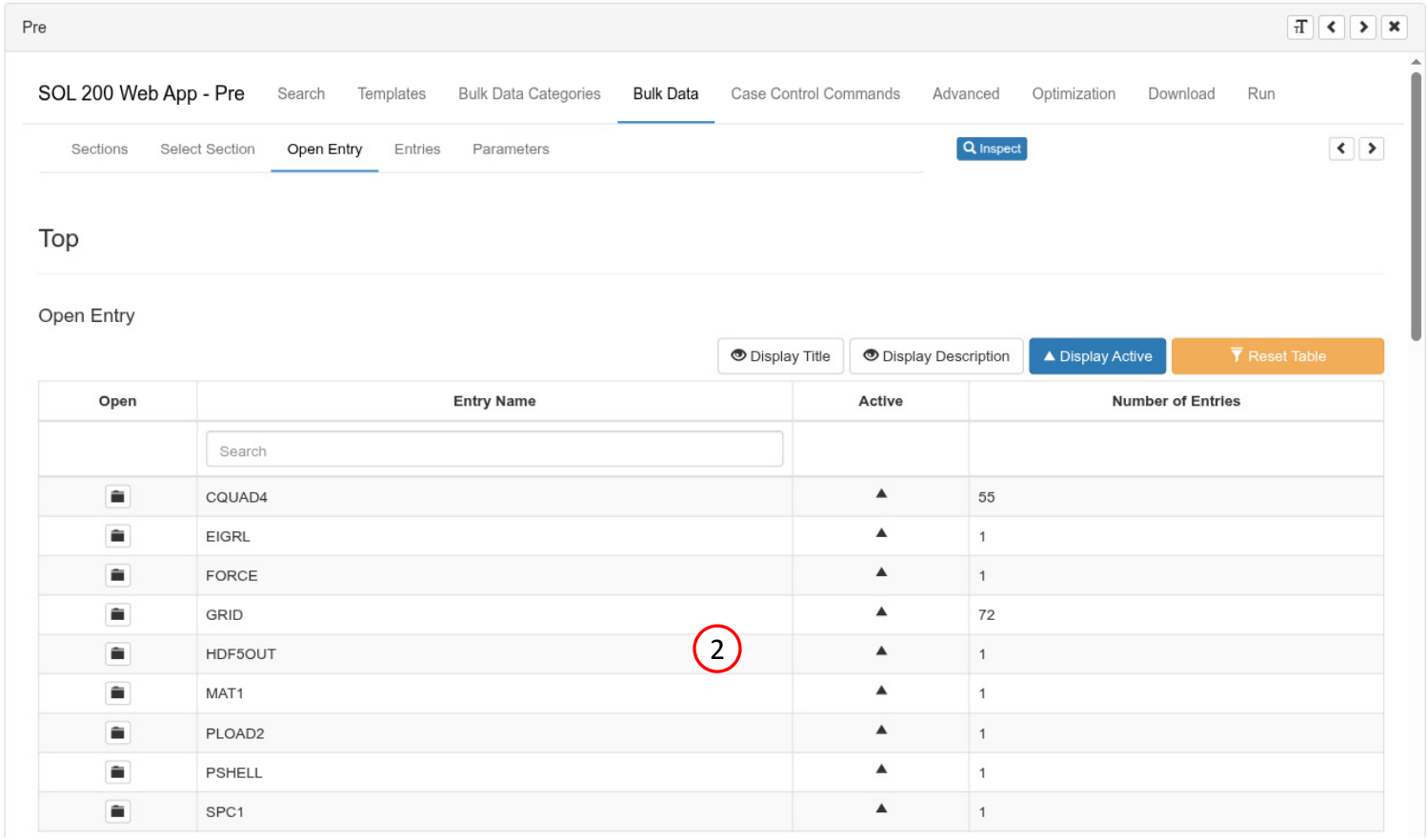




Opening a saved .bdf file

1. Click Pre
2. The previously saved .bdf is now imported to the web app

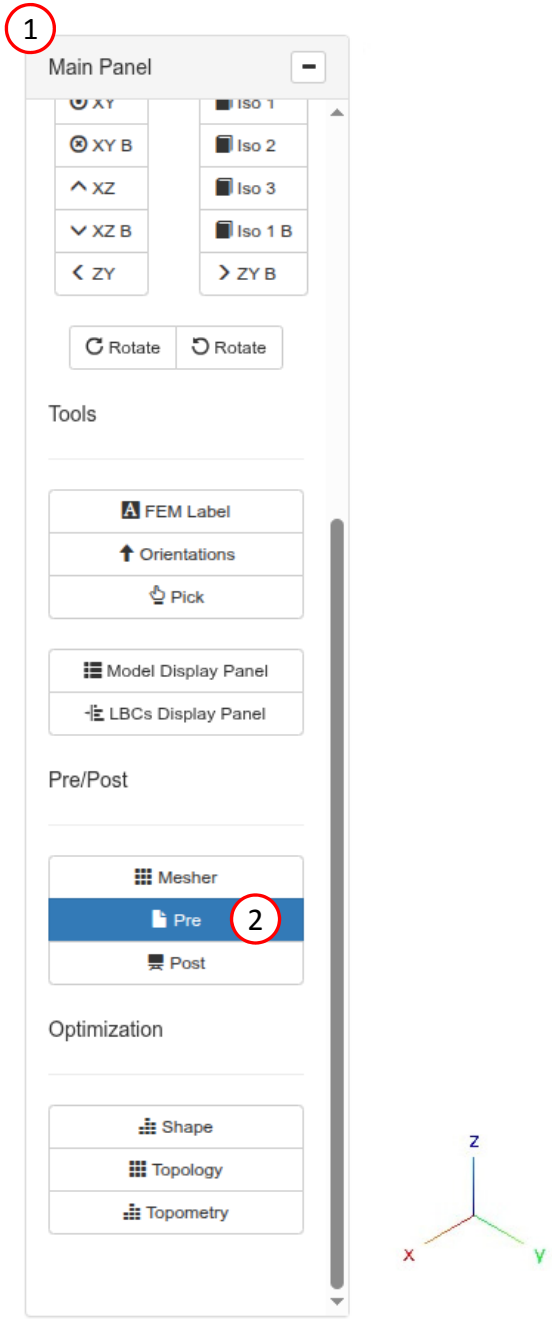




Frequency Response Analysis

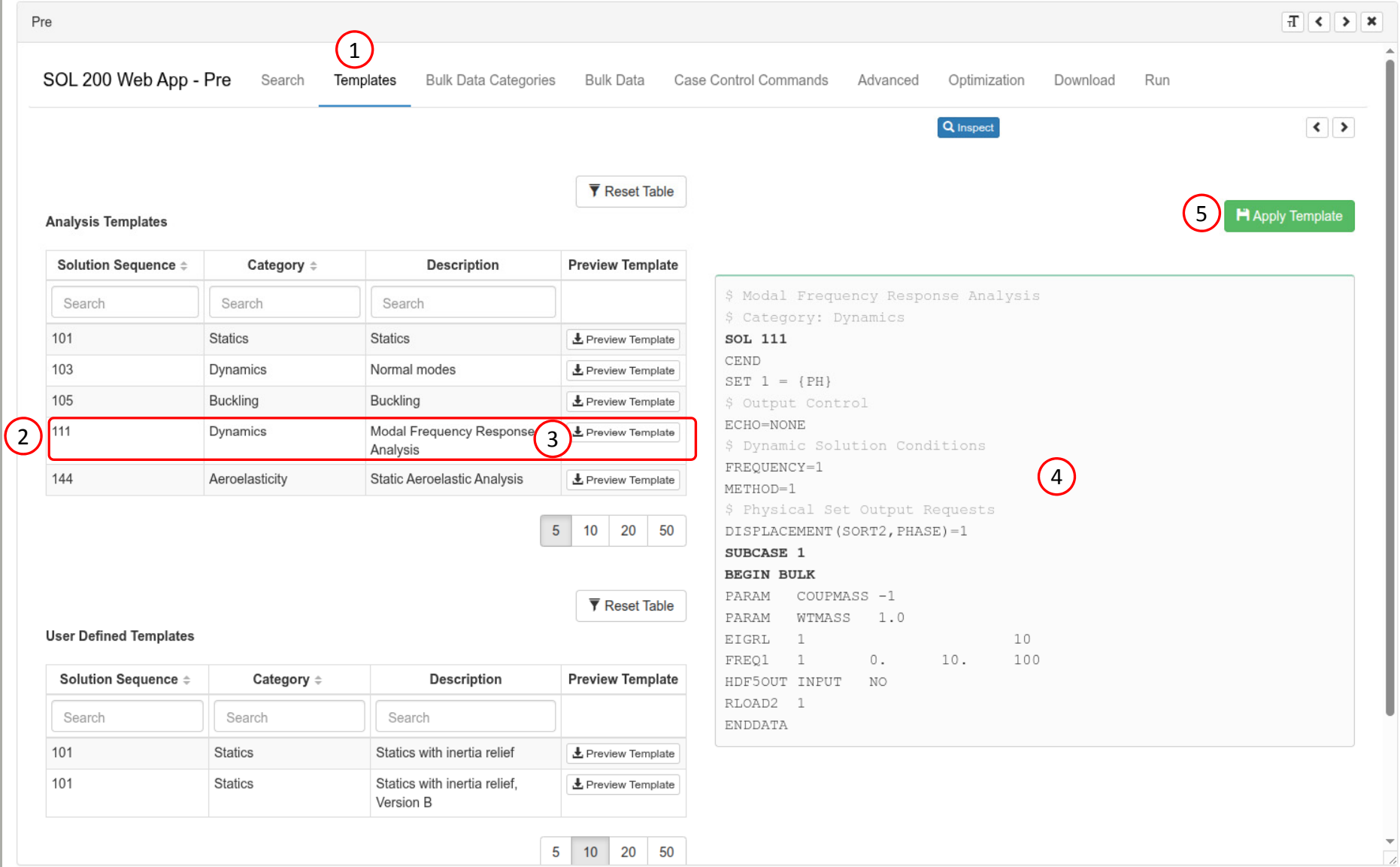
Pre

1. Refer to the Main Panel
2. Click Pre



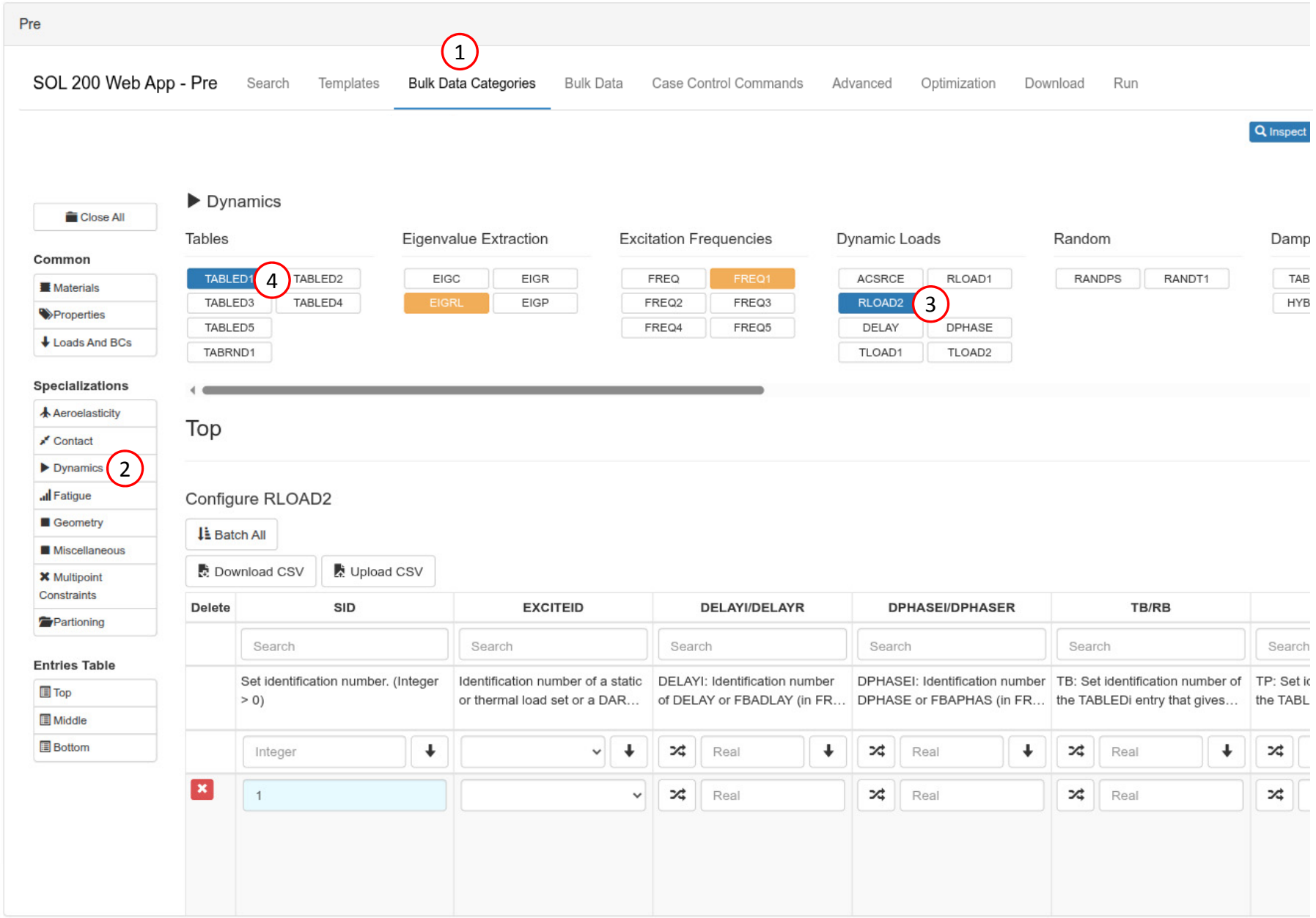
Template

1. Click Templates
2. Locate the row for SOL 111 or Modal Frequency Response Analysis
3. Click Preview Template
4. A preview is displayed of the bulk data file that contains a configuration for a modal frequency response analysis
5. Click Apply Template



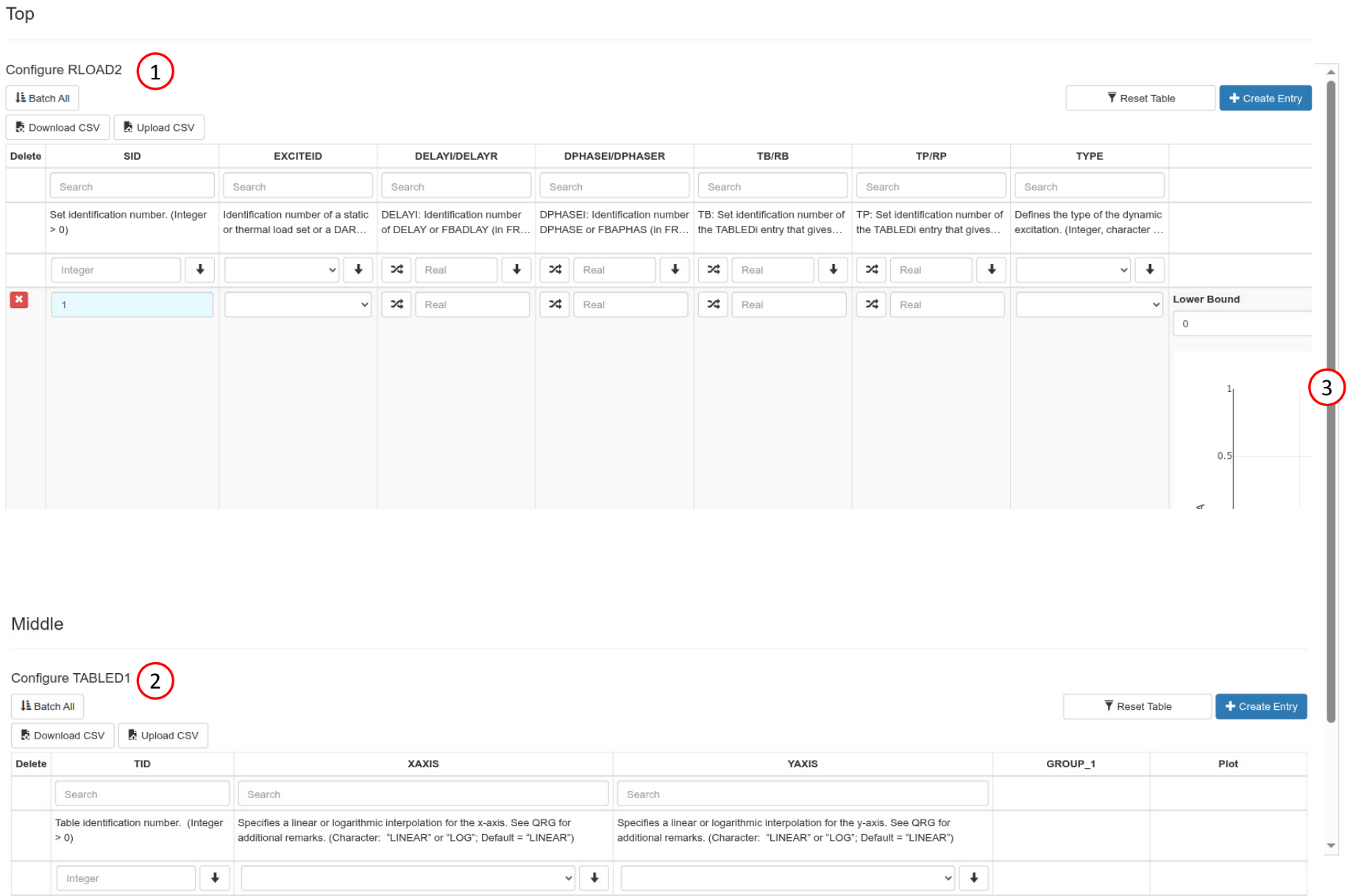
RLOAD2 - Frequency Response Dynamic Excitation, Form 2

1. Click Bulk Data Categories
2. Click Dynamics
3. Click RLOAD2
4. Click TABLED1



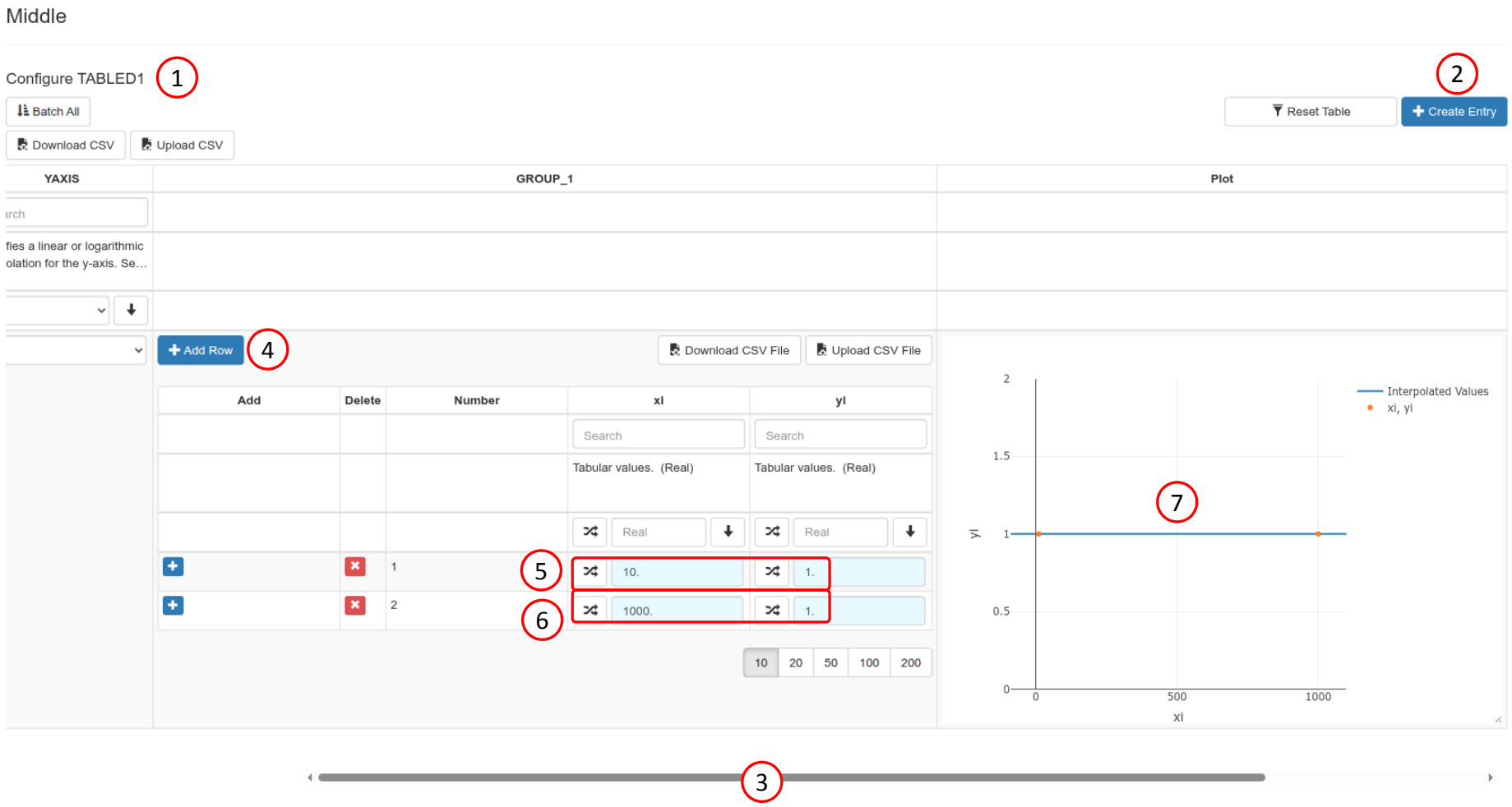
RLOAD2 - Frequency Response Dynamic Excitation, Form 2

1. The table of RLOAD2 entries is now visible
2. The table of TABLED1 entries is also visible
3. Use the indicated scroll bar to view both tables



TABLED1 - Dynamic Load Tabular Function, Form 1

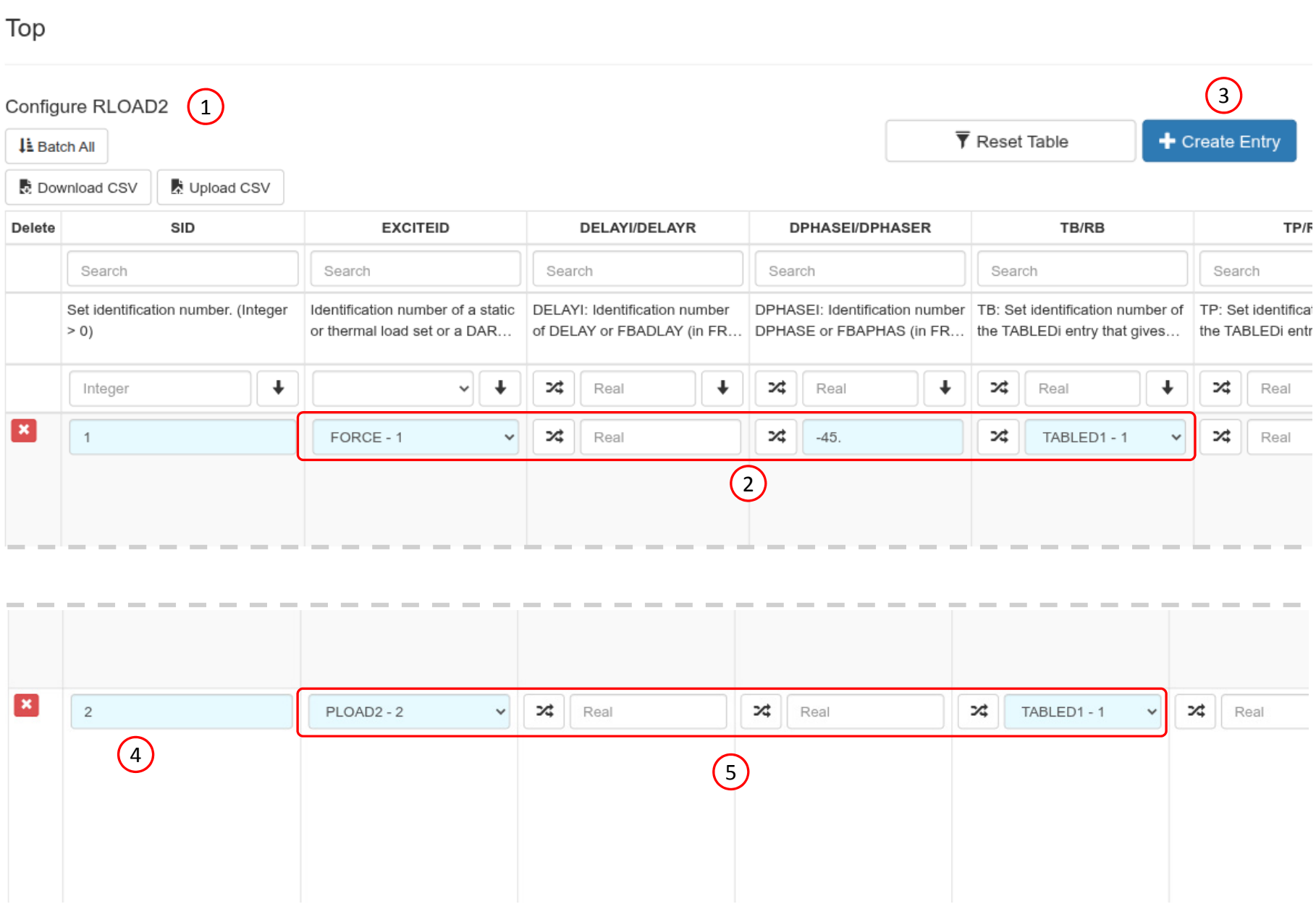
1. Scroll to the table of TABLED1 entries
2. Click Create Entry
3. Horizontally scroll until column GROUP_1 is visible
4. Click Add Row twice.
5. For row 1, supply the following values:
   - xi: 10.
   - yi: 1.
6. For row 2, supply the following values:
   - xi: 1000.
   - yi: 1.
7. The column Plot displays the interpolation function based on the supplied points (xi, yi).



RLOAD2 - Frequency Response Dynamic Excitation, Form 2

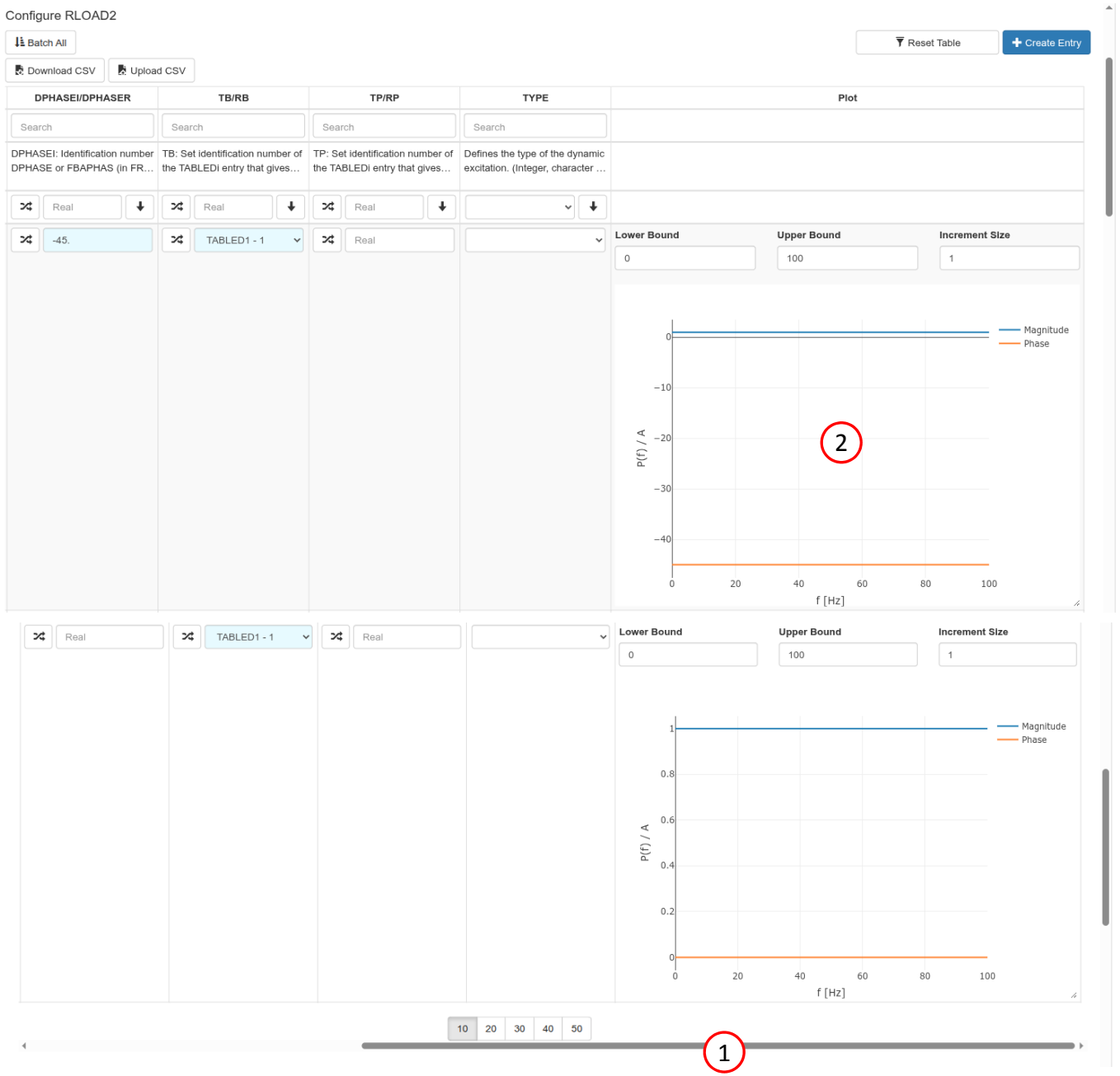
1. Scroll to the table of RLOAD2 entries
2. Supply the following values
   - EXCITEID: FORCE - 1
   - DPHASEI/DPHASER: -45.
   - TB/RB: Click the toggle button once, then select option: TABLED1 - 1

   Note the previous TABLED1 entry is now used by the RLOAD2 entry.
3. Click Create Entry
4. Vertically scroll down until the second row containing the second RLOAD2 entry is found.
5. Supply the following values
   - EXCITEID: PLOAD2 - 2
   - TB/RB: Click the toggle button once, then select option: TABLED1 - 1



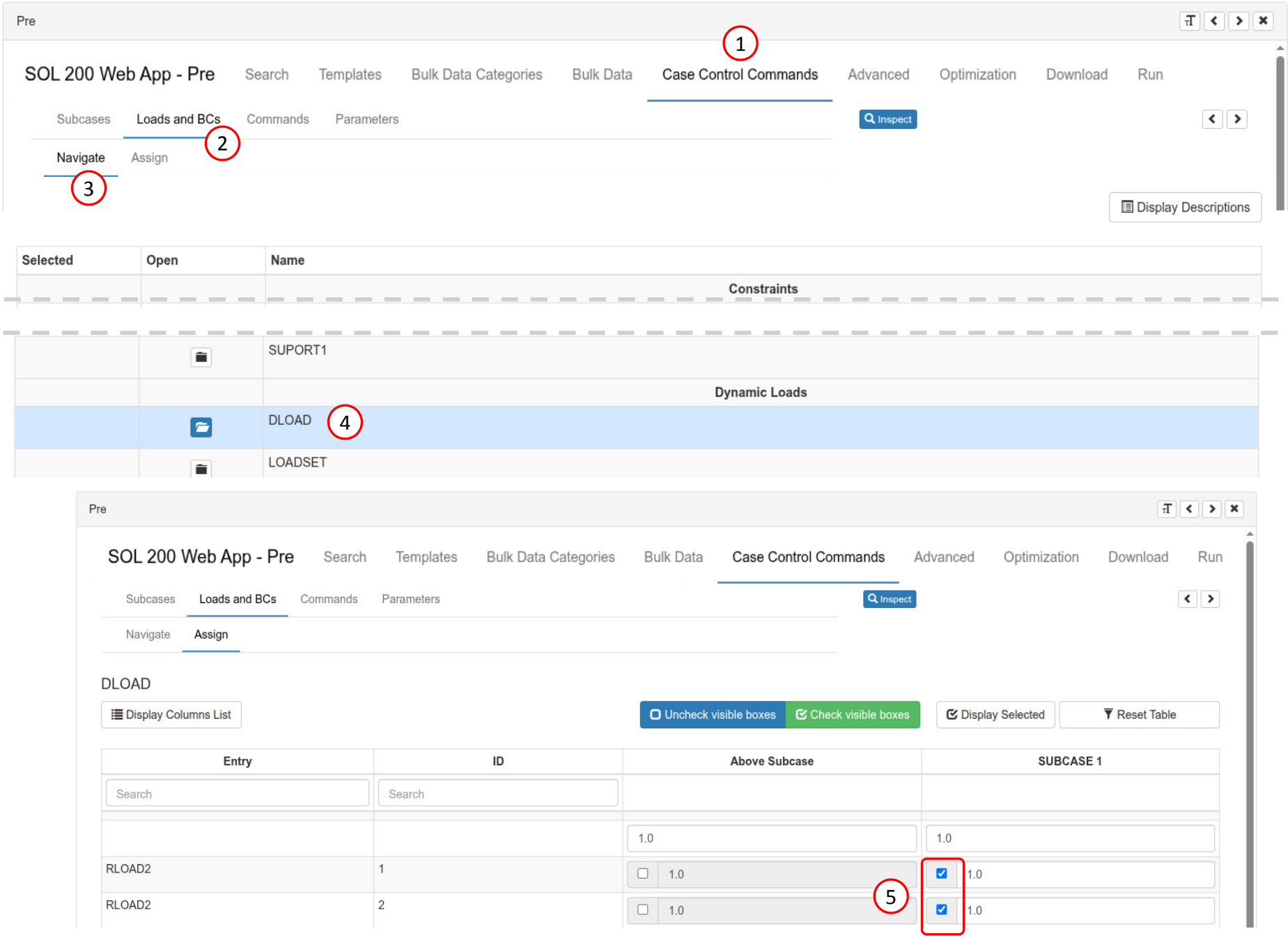
RLOAD2 - Frequency Response Dynamic Excitation, Form 2

1. Horizontally scroll until the Plot column is visible
2. The column Plot displays the temporal distribution of the load based on the values configured in the RLOAD2 and TABLED1 entries.



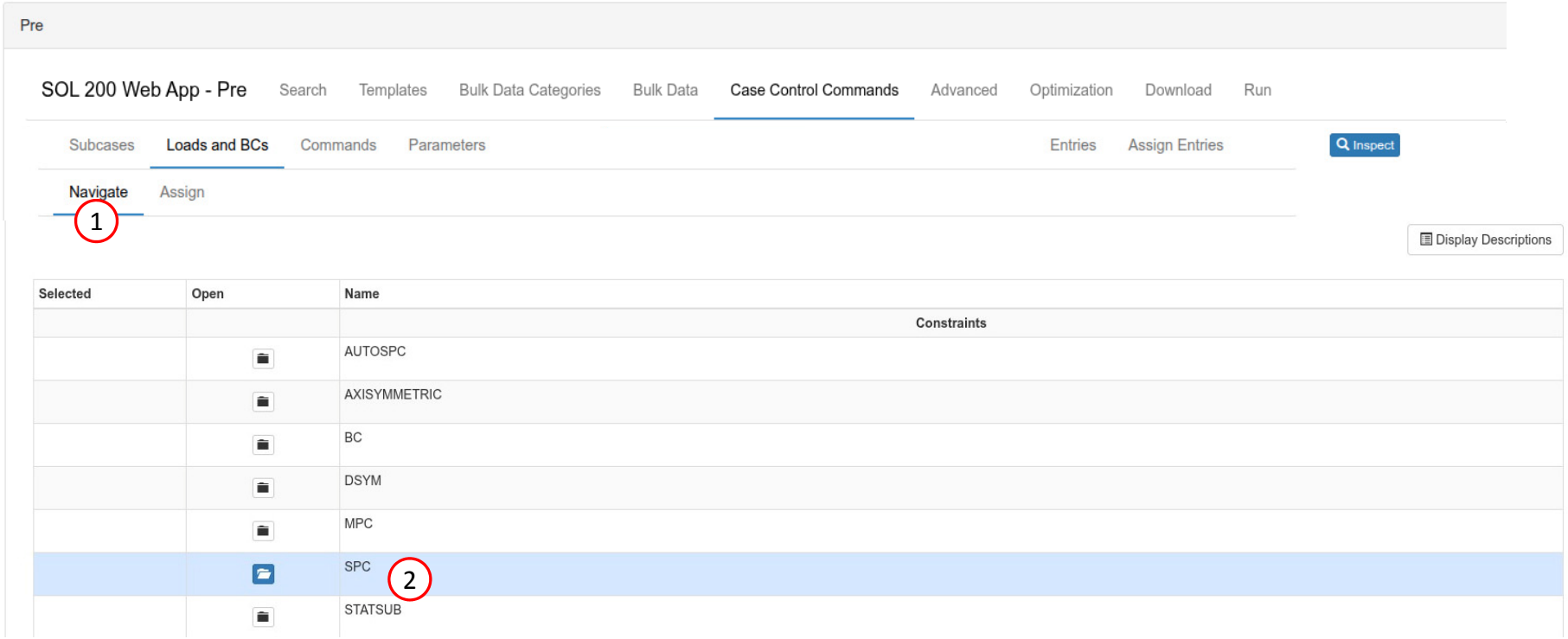
DLOAD - Dynamic Load Set Selection

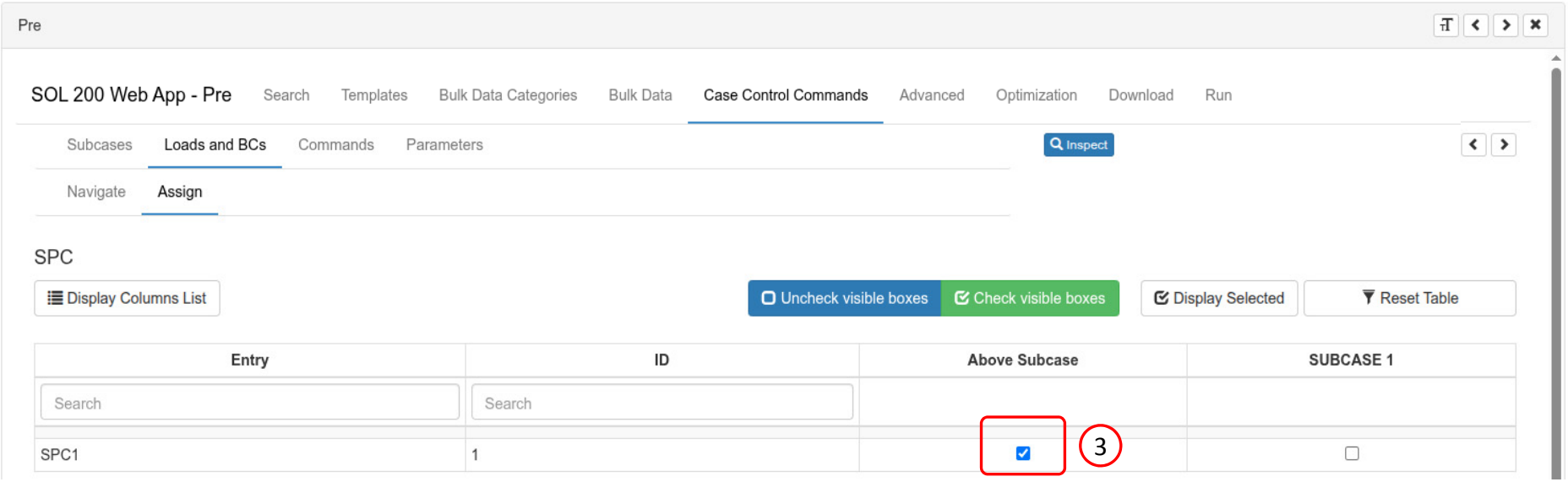
1. Click Case Control Commands
2. Click Loads and BCs
3. Click Navigate
4. Select DLOAD
5. Assign RLOAD2 ID=1 and ID=2 to SUBCASE 1.



SPC - Single Point Constraint Set Selection

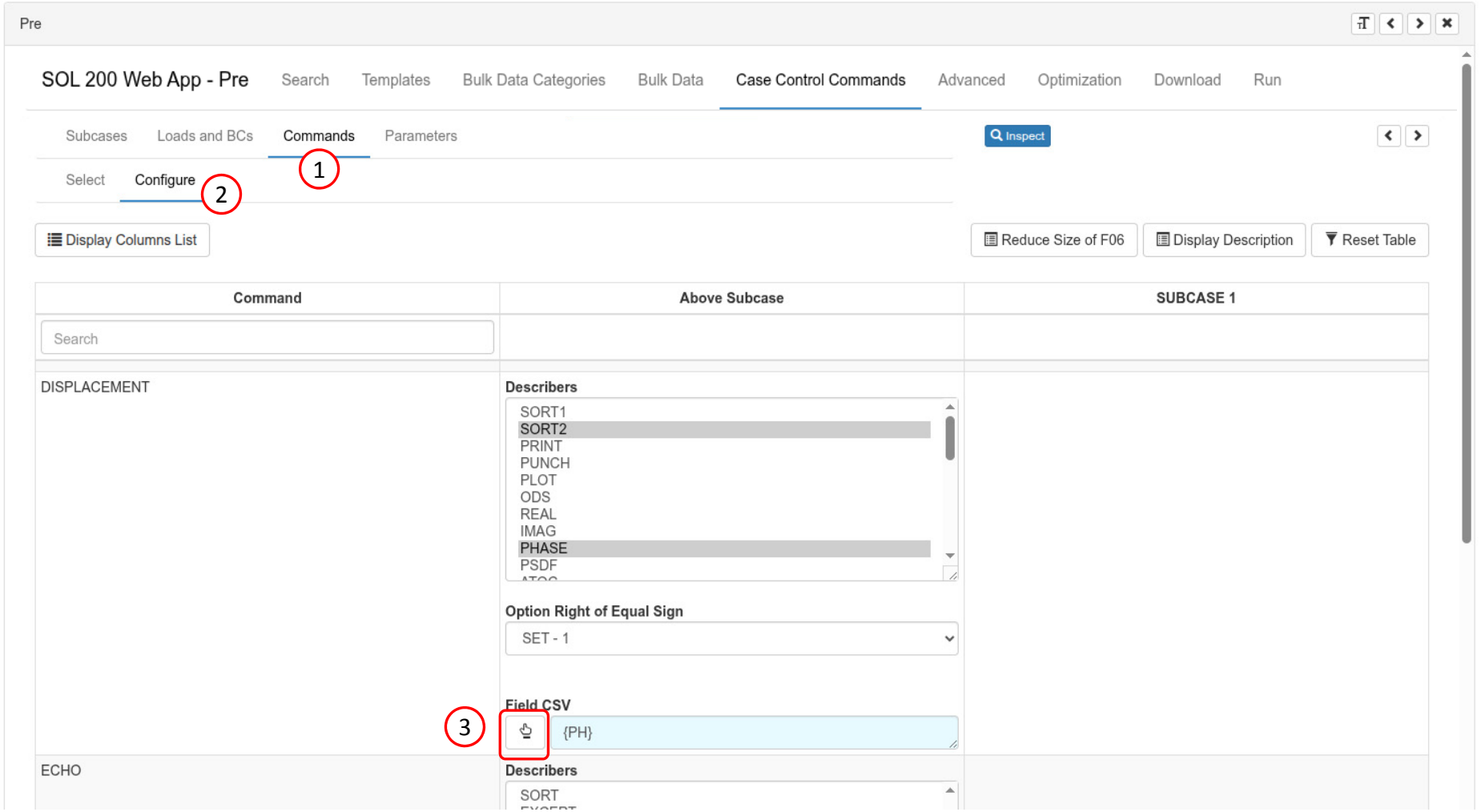
1. Click Navigate
2. Select SPC
3. For SPC1 ID=101 select the checkbox in column Above Subcase





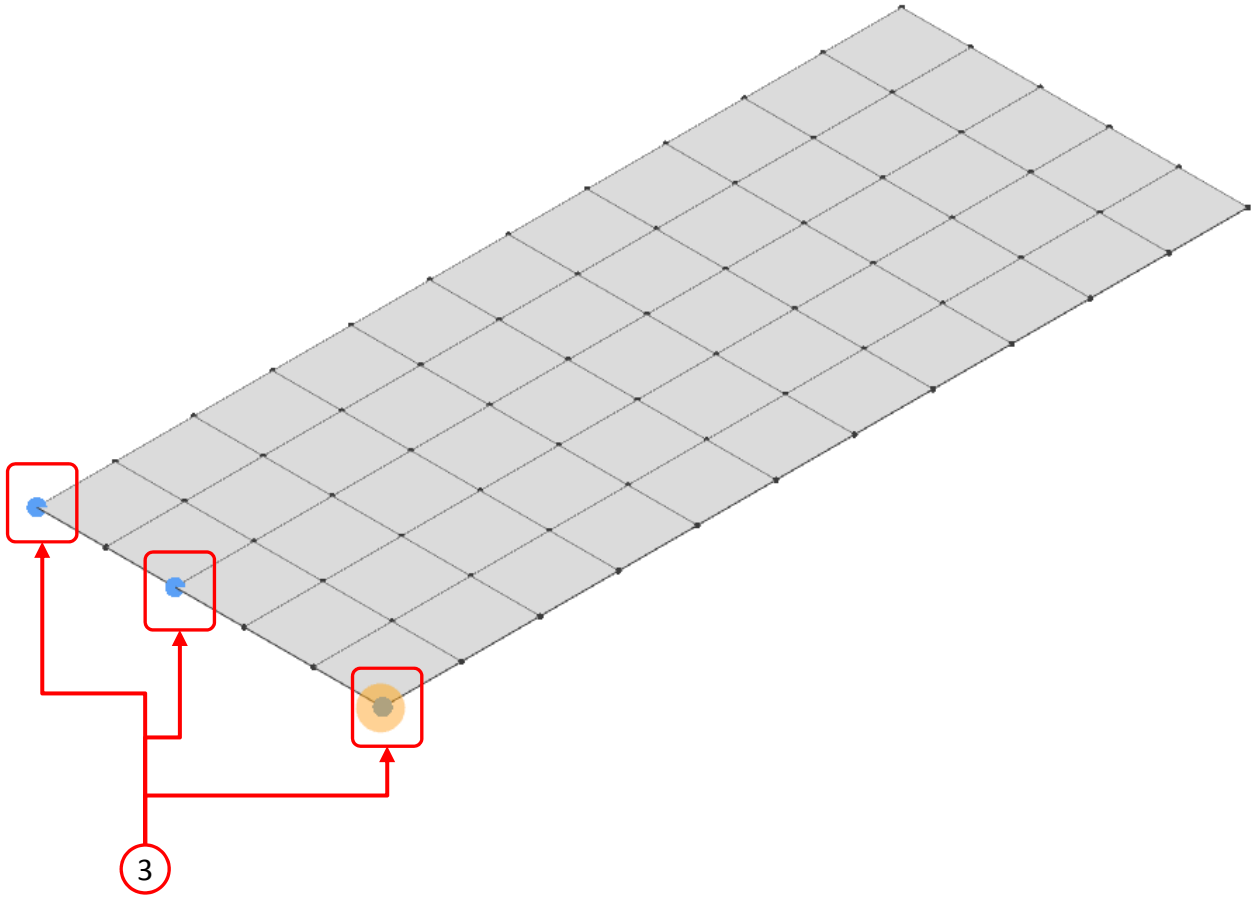
SET - Set Definition, General Form

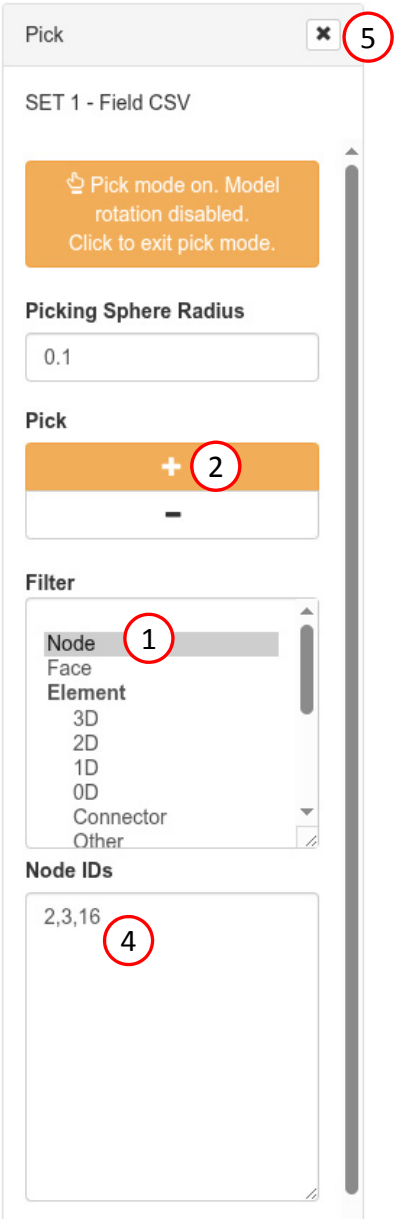
1. Click Commands
2. Click Configure
3. Click the Pick button



SET - Set Definition, General Form

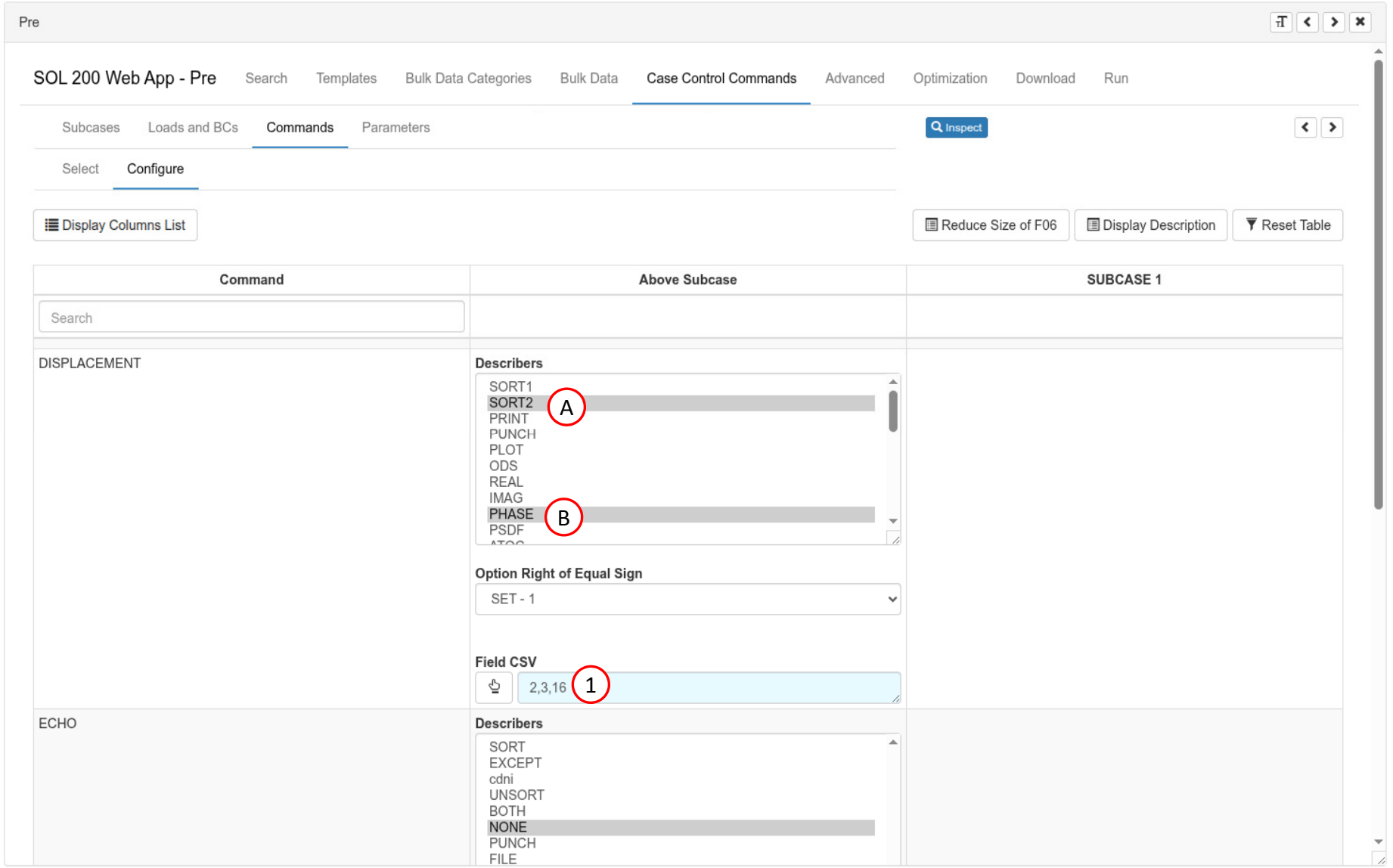
1. Set Filter to Node
2. Click +
3. Move the mouse cursor to the indicated region. Click and hold the left mouse button, then drag the picking sphere to select
4. The selected node IDs are stored
5. Click X





SET - Set Definition, General Form

1. The selected node IDs are now stored

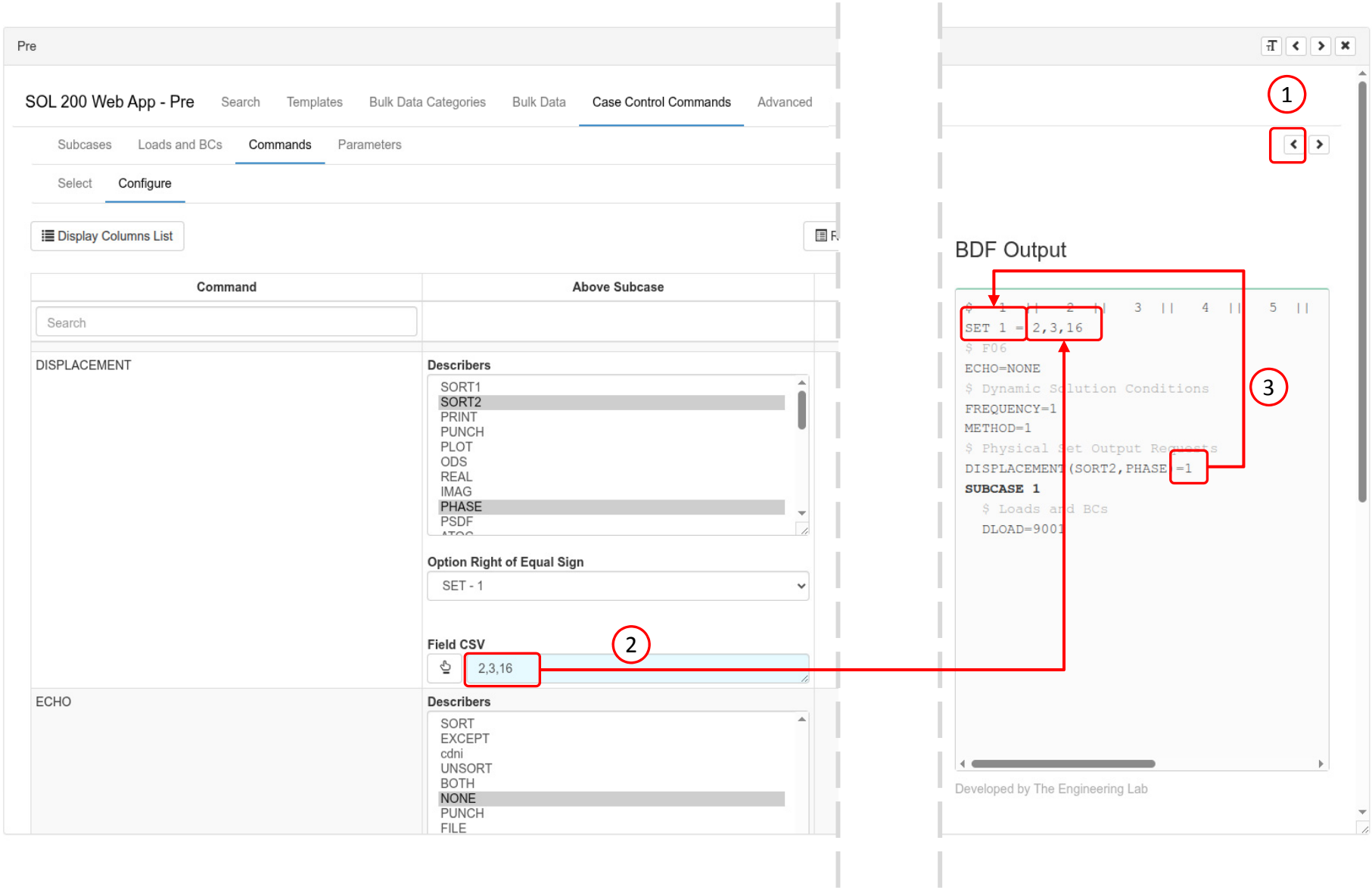


A. SORT1 and SORT2 control how the output is printed to the F06 file. SORT2 is the preferred choice when performing frequency or transient analysis.
   - SORT1: For each excitation frequency, print the results for each node.
   - SORT2: For each node, print the results for each excitation frequency.

B. The default option REAL will print real and imaginary terms. The PHASE option will print magnitude and phase terms.

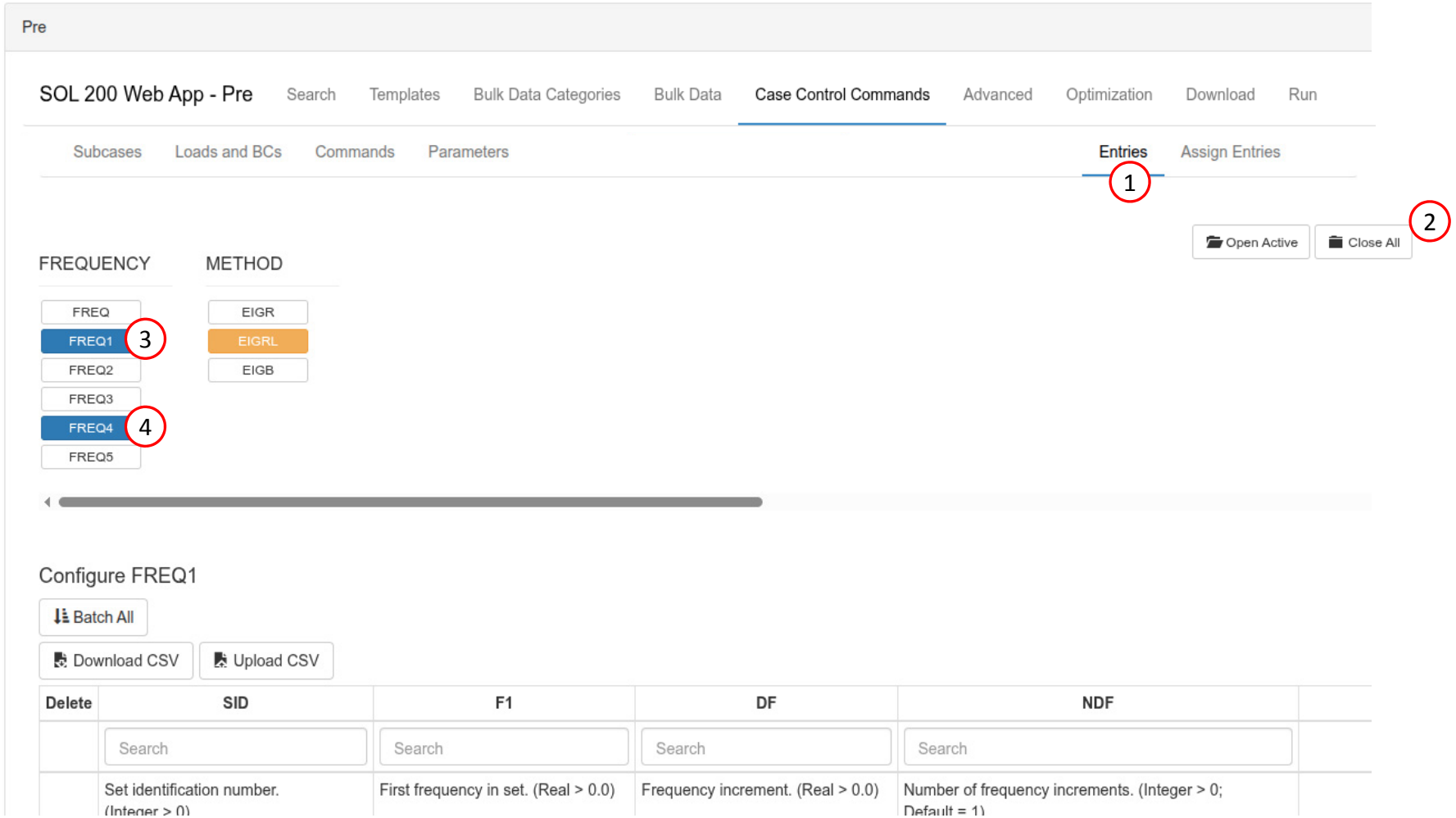
SET - Set Definition, General Form

1. Click the indicated button to display the BDF Output section
2. The selected node IDs are stored in the SET keyword
3. The DISPLACEMENT command points to the SET keyword, effectively outputting displacements only for 3 nodes



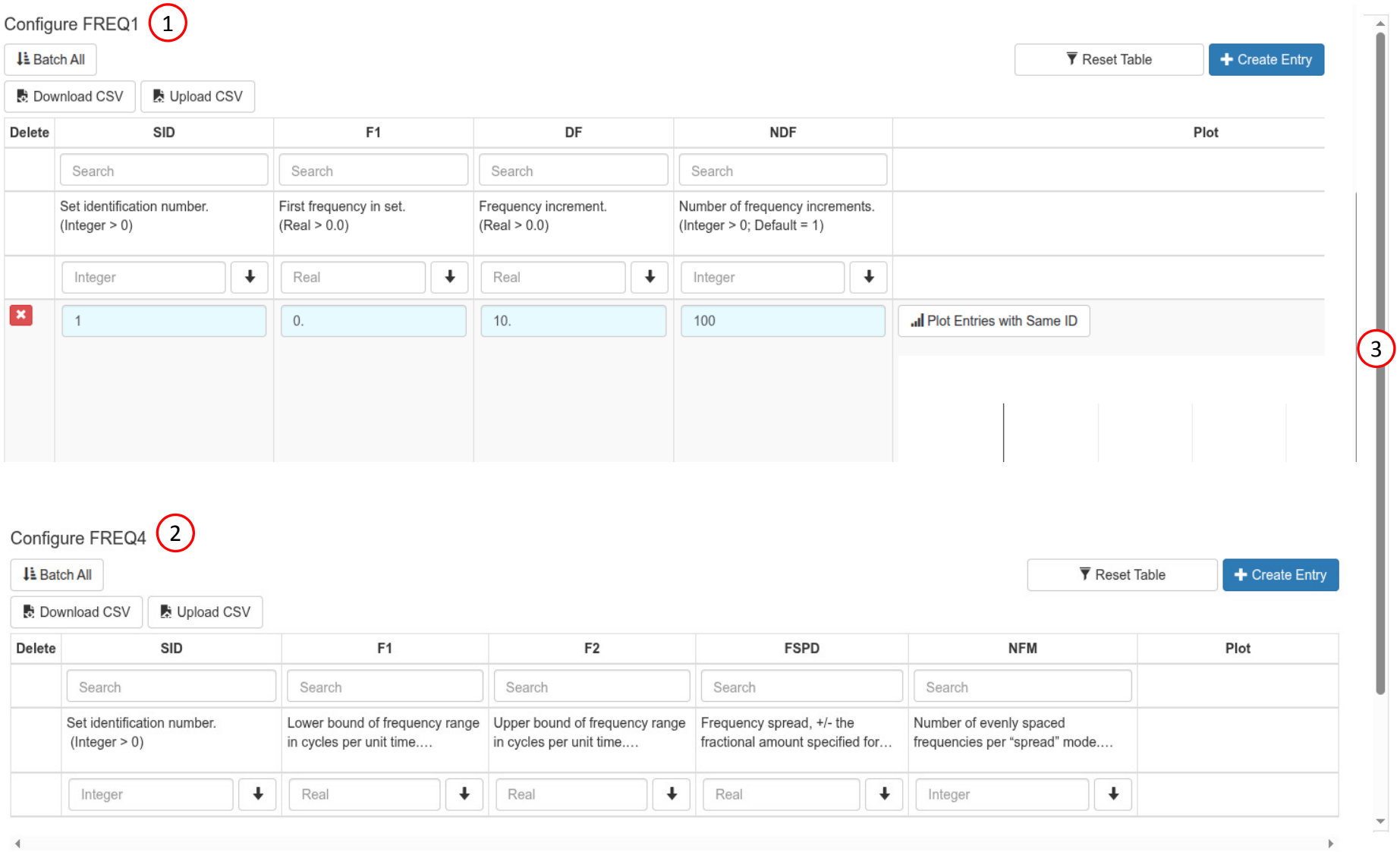
Excitation Frequencies

1. Click Entries
2. Click Close All
3. Click FREQ1
4. Click FREQ4



Excitation Frequencies

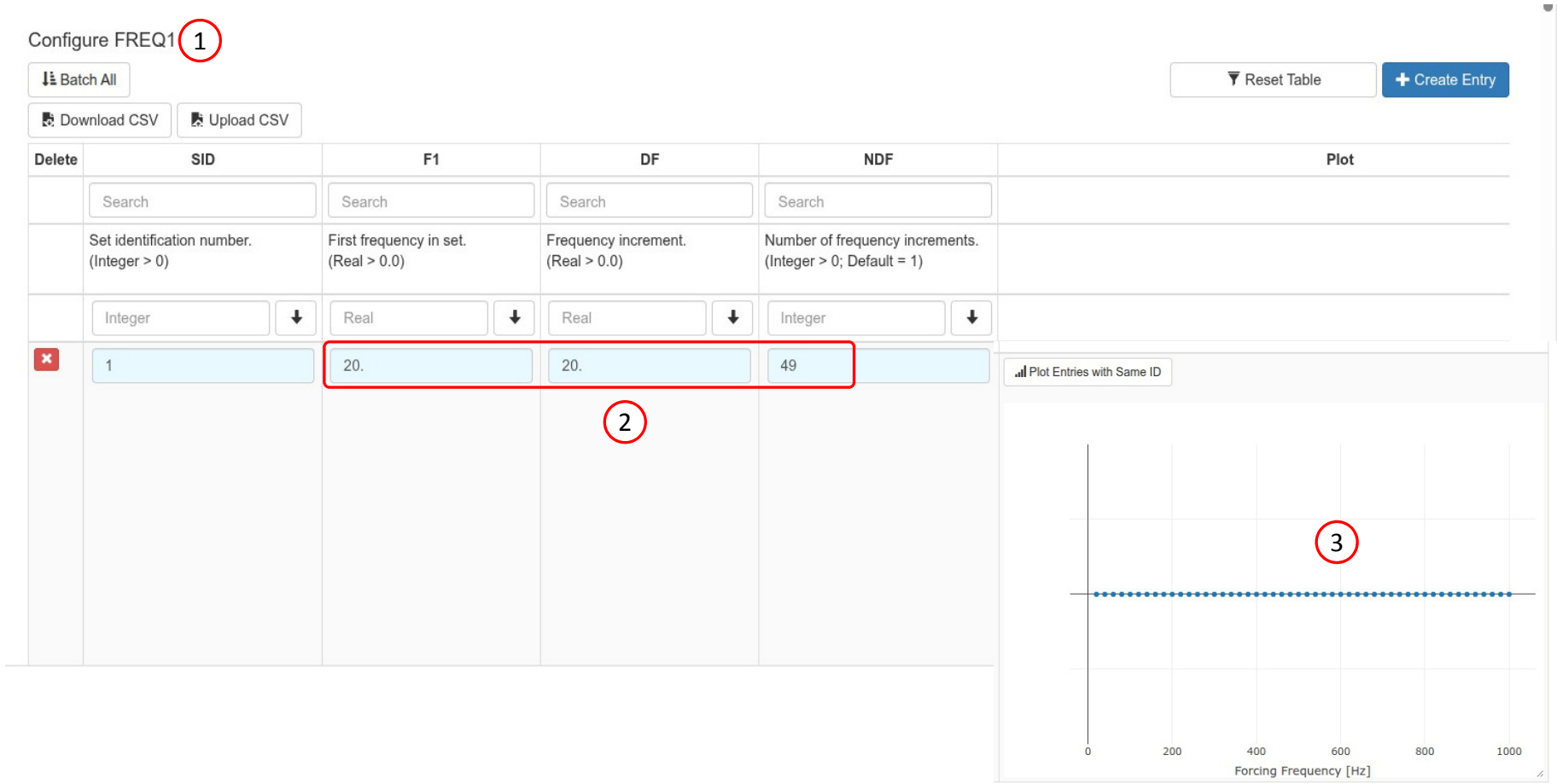
1. The table of FREQ1 entries is now visible
2. The table of FREQ4 entries is also visible
3. Use the indicated scroll bar to view both tables



FREQ1 - Frequency List, Alternate Form 1

1. Scroll to the table of FREQ1 entries
2. Supply the following values
   - F1: 20.
   - DF: 20.
   - NDF: 49
3. A plot displays the excitation frequencies

This defines excitation frequencies and are sometimes referred to as the solution frequencies or forcing frequencies.



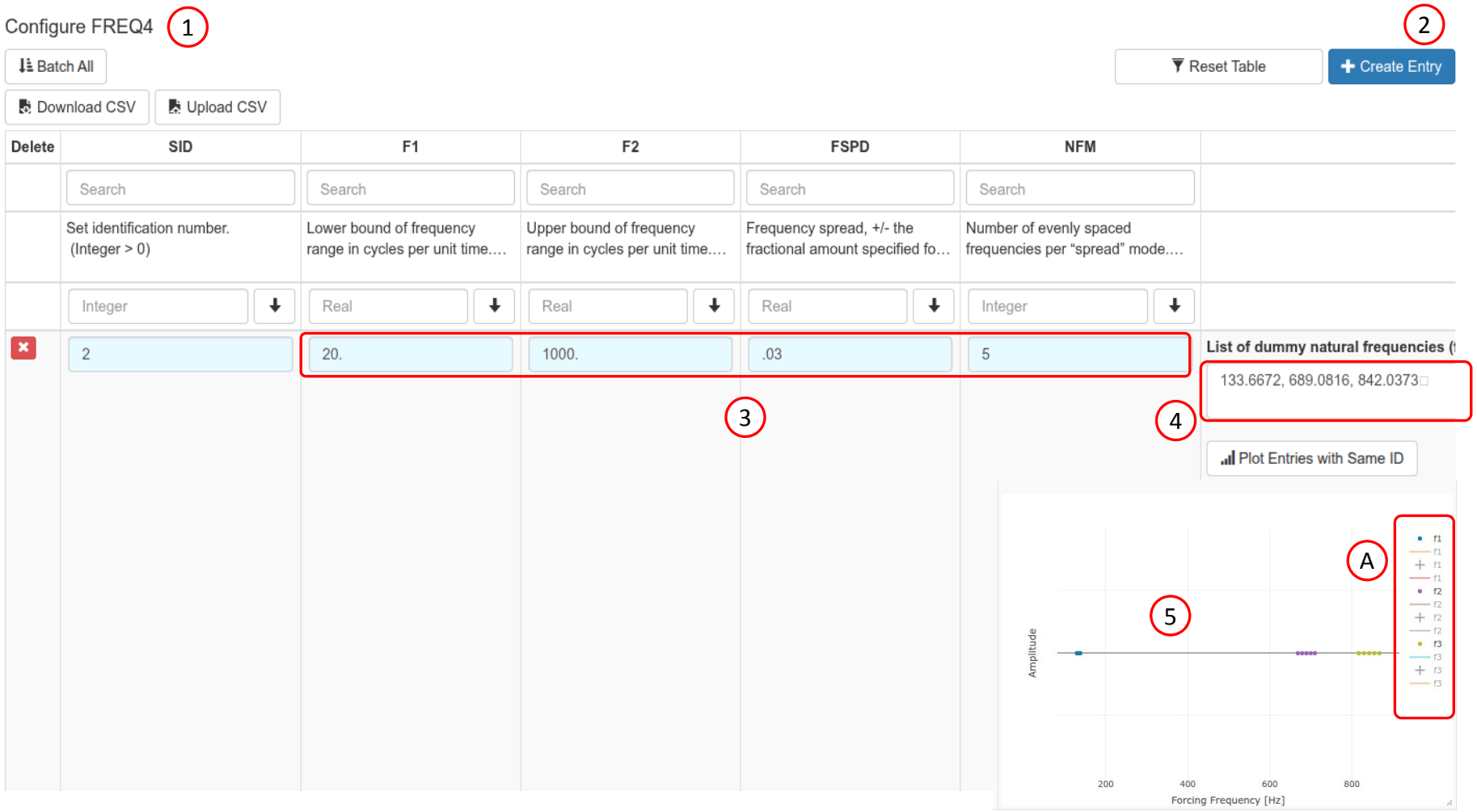
FREQ4 - Frequency List, Alternate Form 4

1. Select row FREQ4
2. Click Create Entry
3. Supply the following values
   - F1: 20.
   - F2: 1000.
   - FSPD: .03
   - NFM: 5
4. Supply the following values
   - List of dummy natural frequencies:

     133.6672, 689.0816, 842.0373

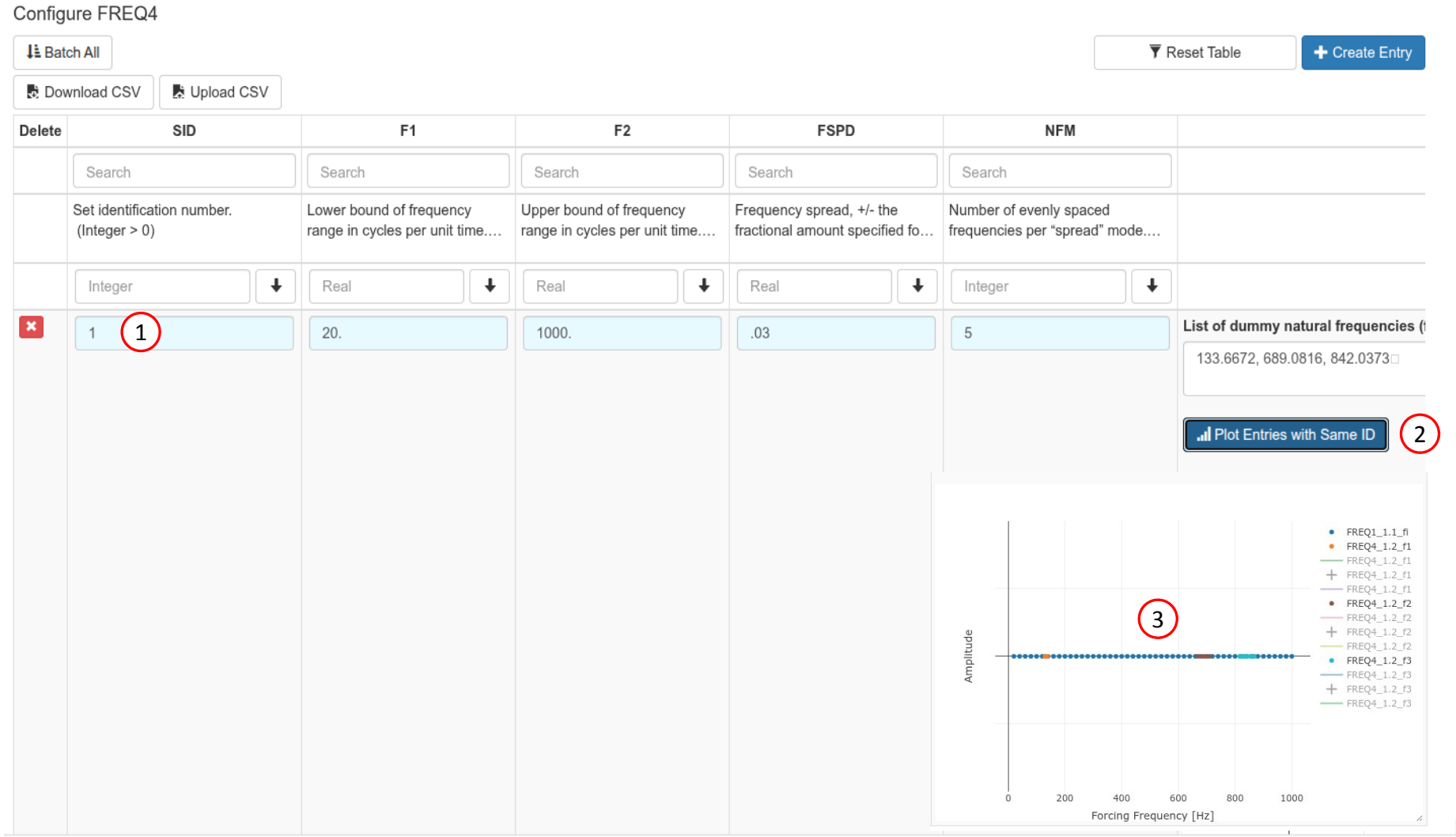
     These are the natural frequencies from the normal modes analysis. Alternatively you can supply dummy test frequencies of your liking.
   - These dummy or test frequencies are referred to as f1, f2, and f3, respectively.
5. The column Plot displays the corresponding excitation frequencies.

A. There is additional information that may be displayed for FREQ4 entries. This will not be covered on this page, but is discussed in more detail in the Appendix, section *How to display additional information for FREQ4 entries*.



FREQ4 - Frequency List, Alternate Form 4

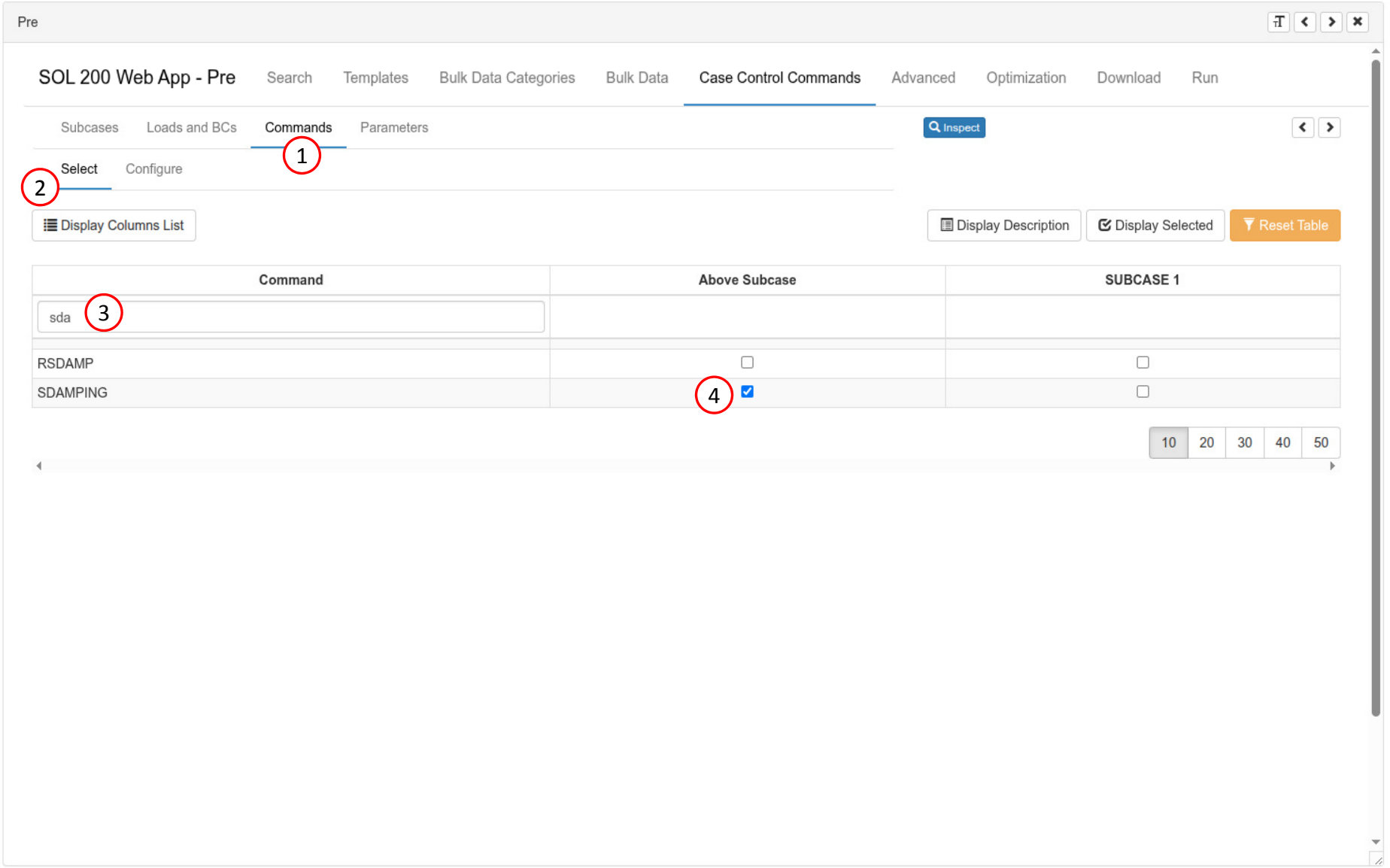
1. Update the following value
   - SID: 1
2. Click Plot Entries with Same ID
3. All the excitation frequencies for any entry FREQ, FREQ1, ..., FREQ5 with the same ID are now displayed in a single plot. The frequencies from FREQ1 ID=1 and FREQ4 ID=1 are now plotted.



SDAMPING - Structural Damping

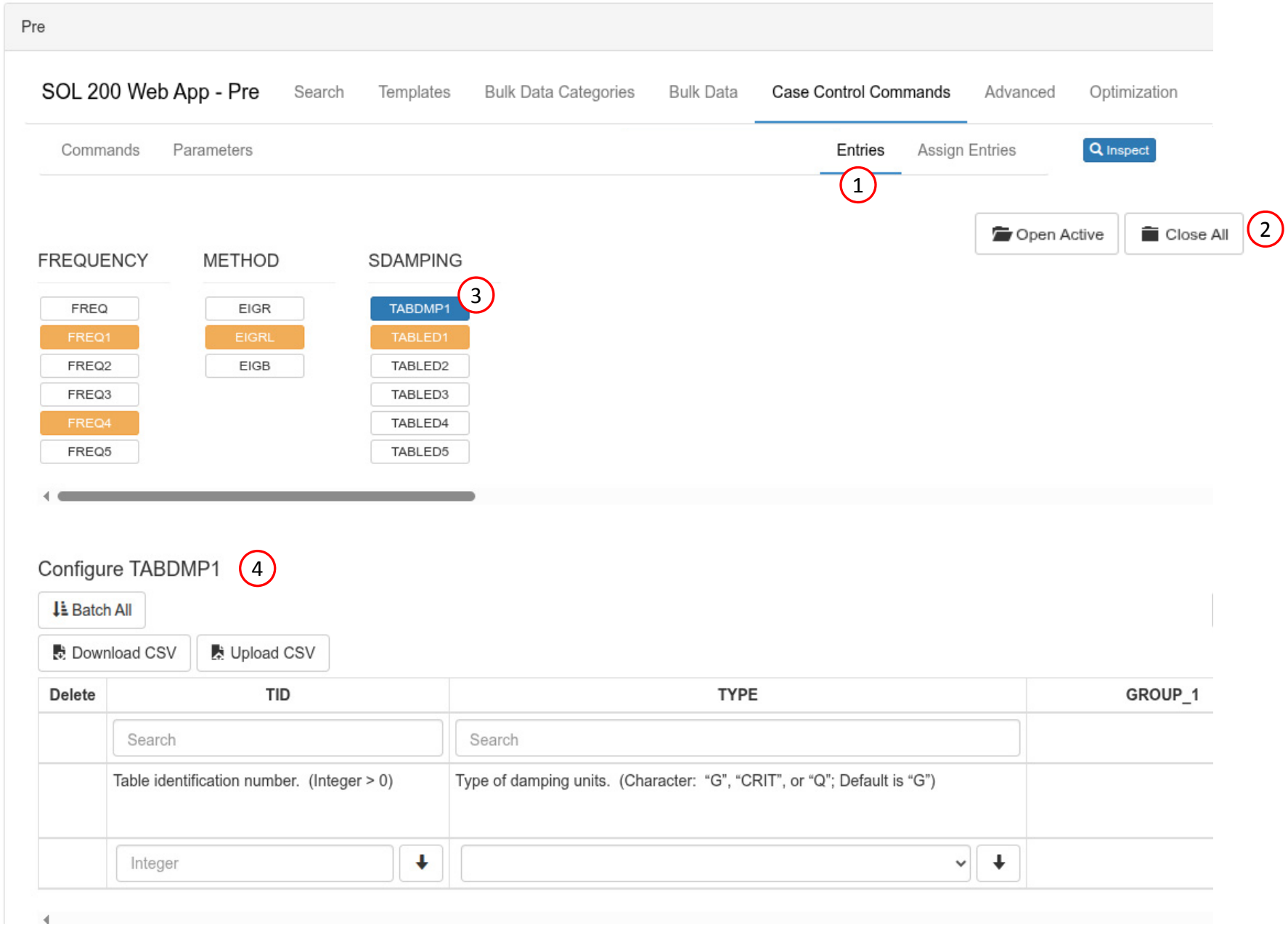
1. Click Commands
2. Click Select
3. Search the table for SDA or SDAMPING
4. For row SDAMPING, mark the indicated checkbox

Structural damping (SDAMPING) is now on for the above subcase level.



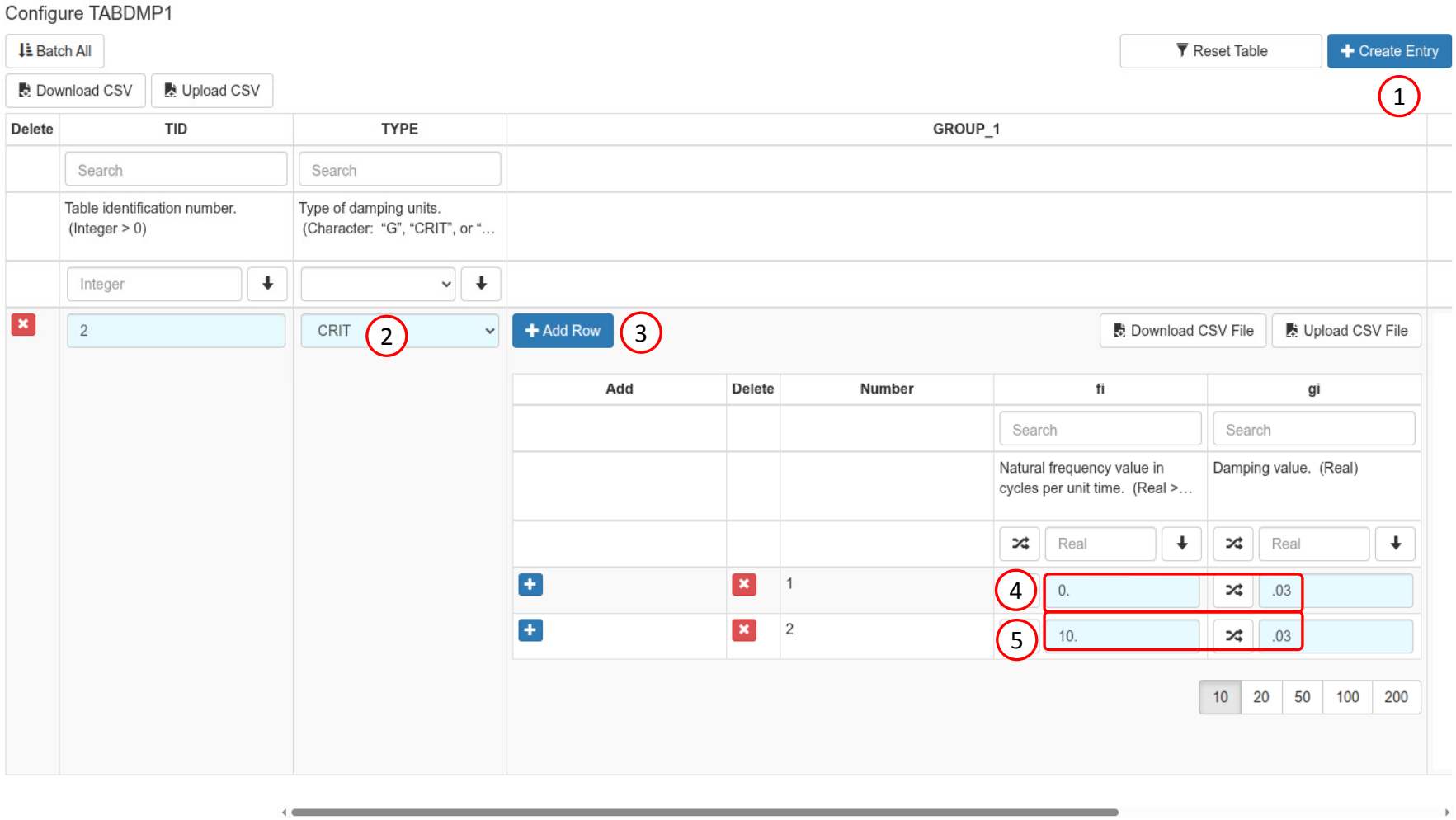
TABDMP1 - Modal Damping Table

1. Click Entries
2. Click Close All
3. Click TABDMP1
4. The table of TABDMP1 entries is now visible



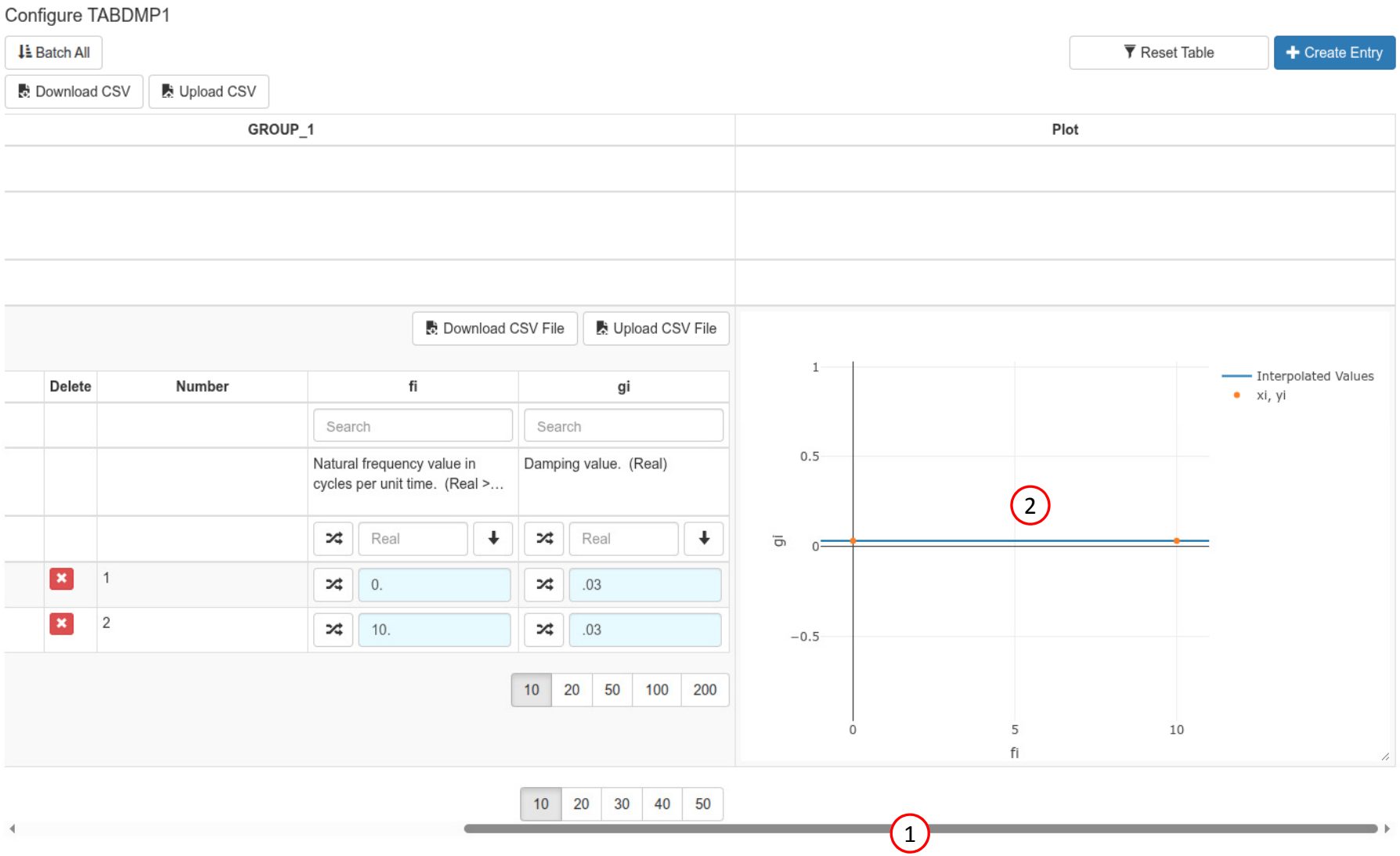
TABDMP1 - Modal Damping Table

1. Click Create Entry
2. Supply the following values
   - TYPE: CRIT
3. Click Add Row 2 times
4. For row 1, supply the following values:
   - fi: 0.
   - gi: .03
5. For row 2, supply the following values:
   - fi: 10.
   - gi: .03



TABDMP1 - Modal Damping Table

1. Use the indicated scroll bar to display the column Plot.
2. The Plot column displays the interpolation function based on the supplied points (fi, gi) (Not shown)

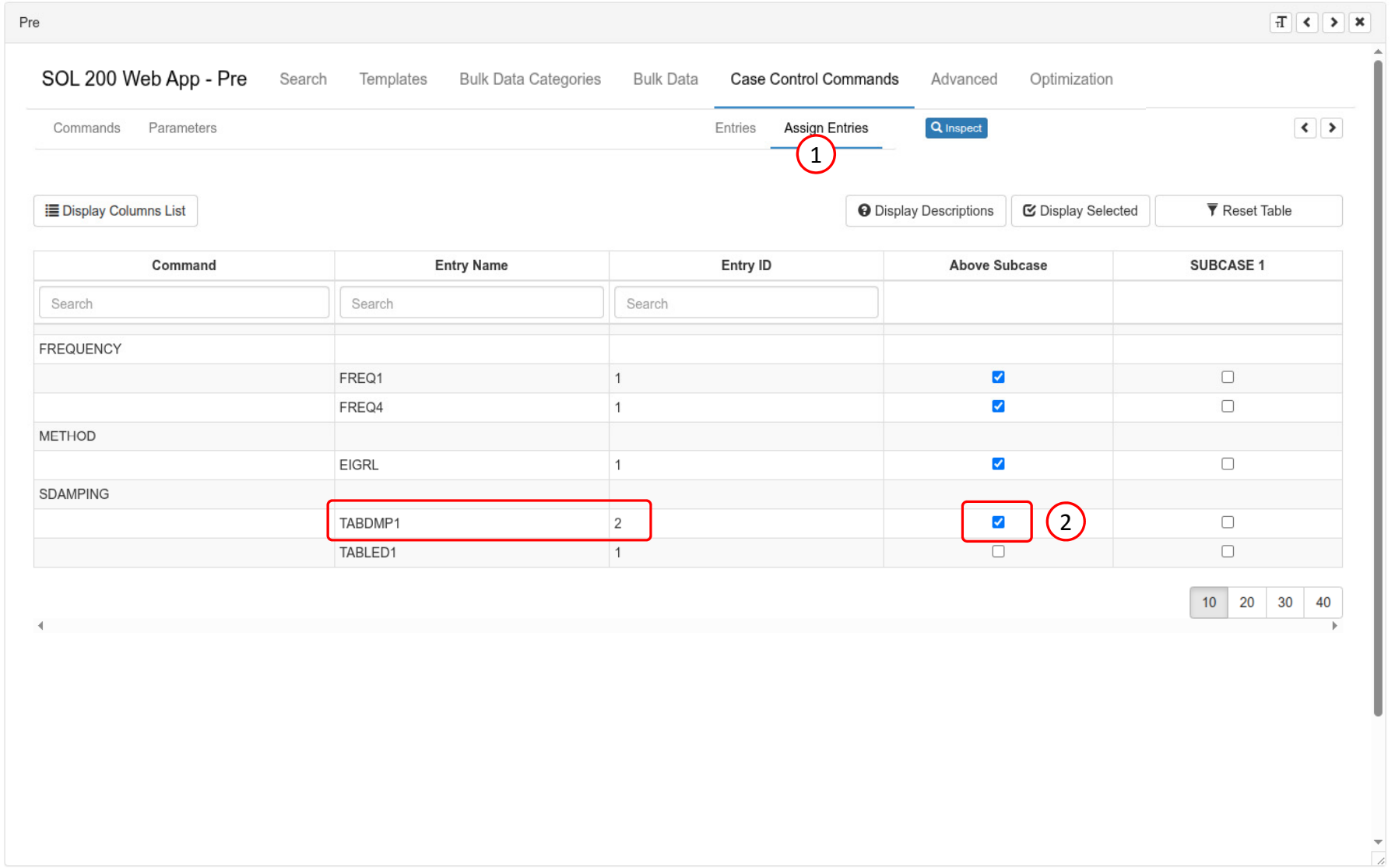


TABDMP1 - Modal Damping Table

1. Click Assign Entries
2. Find the row corresponding to TABDMP1 ID=2
3. Mark the corresponding checkbox

The TABDMP1 entry has been assigned to command SDAMPING and the above subcase level. Structural damping is now based on the interpolation function defined in the TABDMP1 entry.

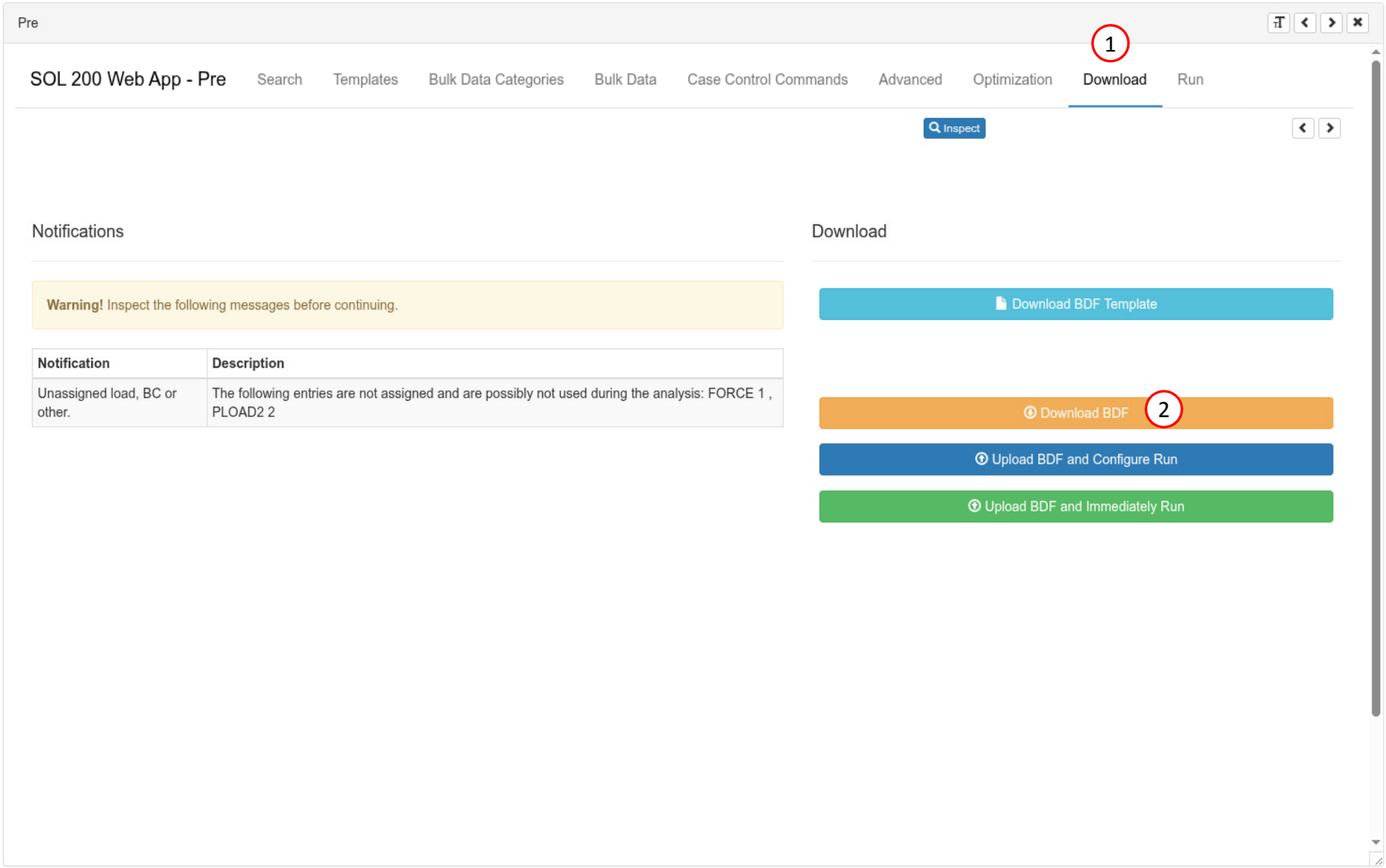
When assigned to the above subcase level, that assignment also applies to subsequent subcases, e.g. subcase 1, subcase 2, etc.



Download

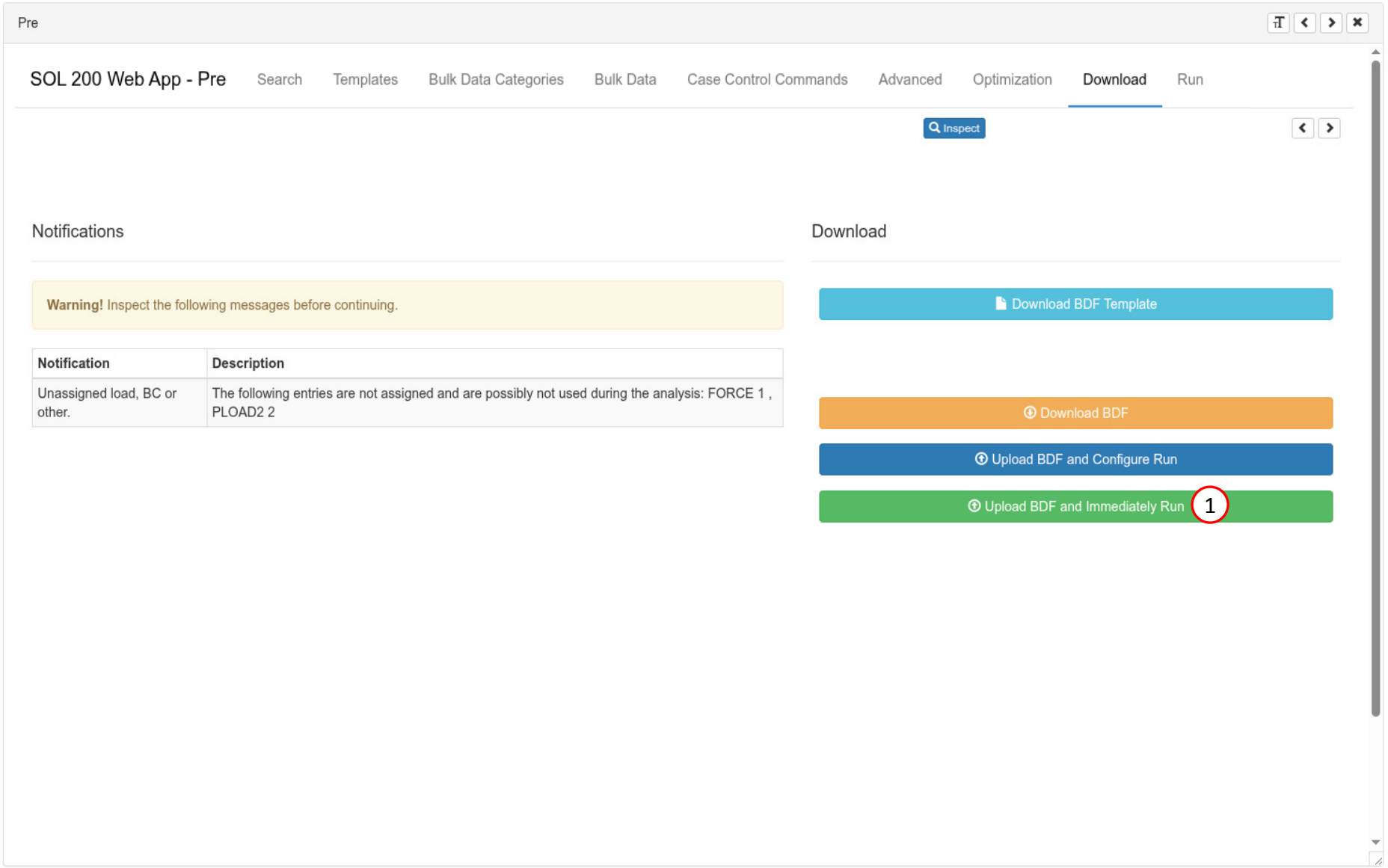
1. Navigate to Pre > Download
2. Click Download BDF

The Nastran model is now downloaded and saved to your desktop.



Run

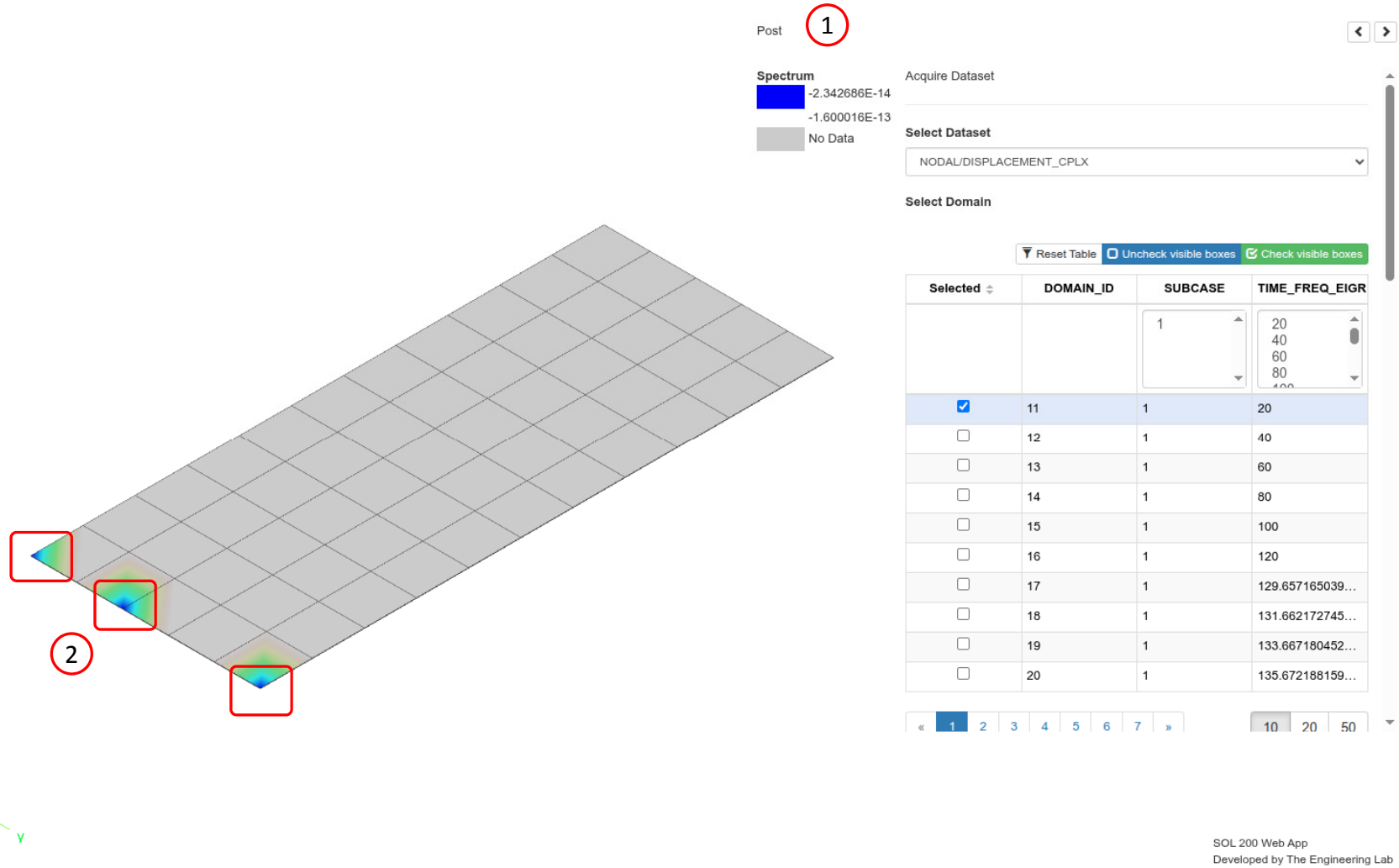
1. Click Upload BDF and Immediately Run



Post

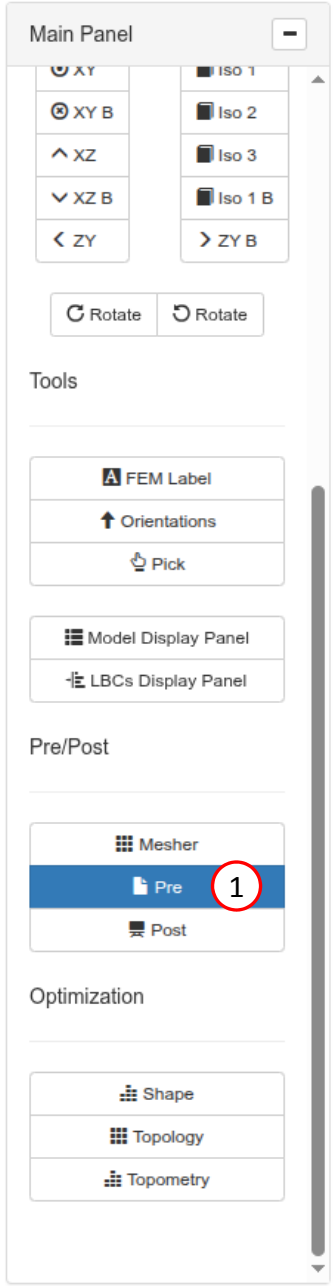
1. After the job is finished, the Post app is automatically opened.
2. Earlier in this exercise, the SET command was defined to output the displacement at only 3 nodes.

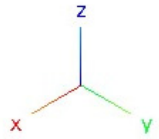
Graphs, or XYPLOTs, showing displacement vs. frequency are desired. The following slides detail how to construct graphs for the 3 nodes.



Pre

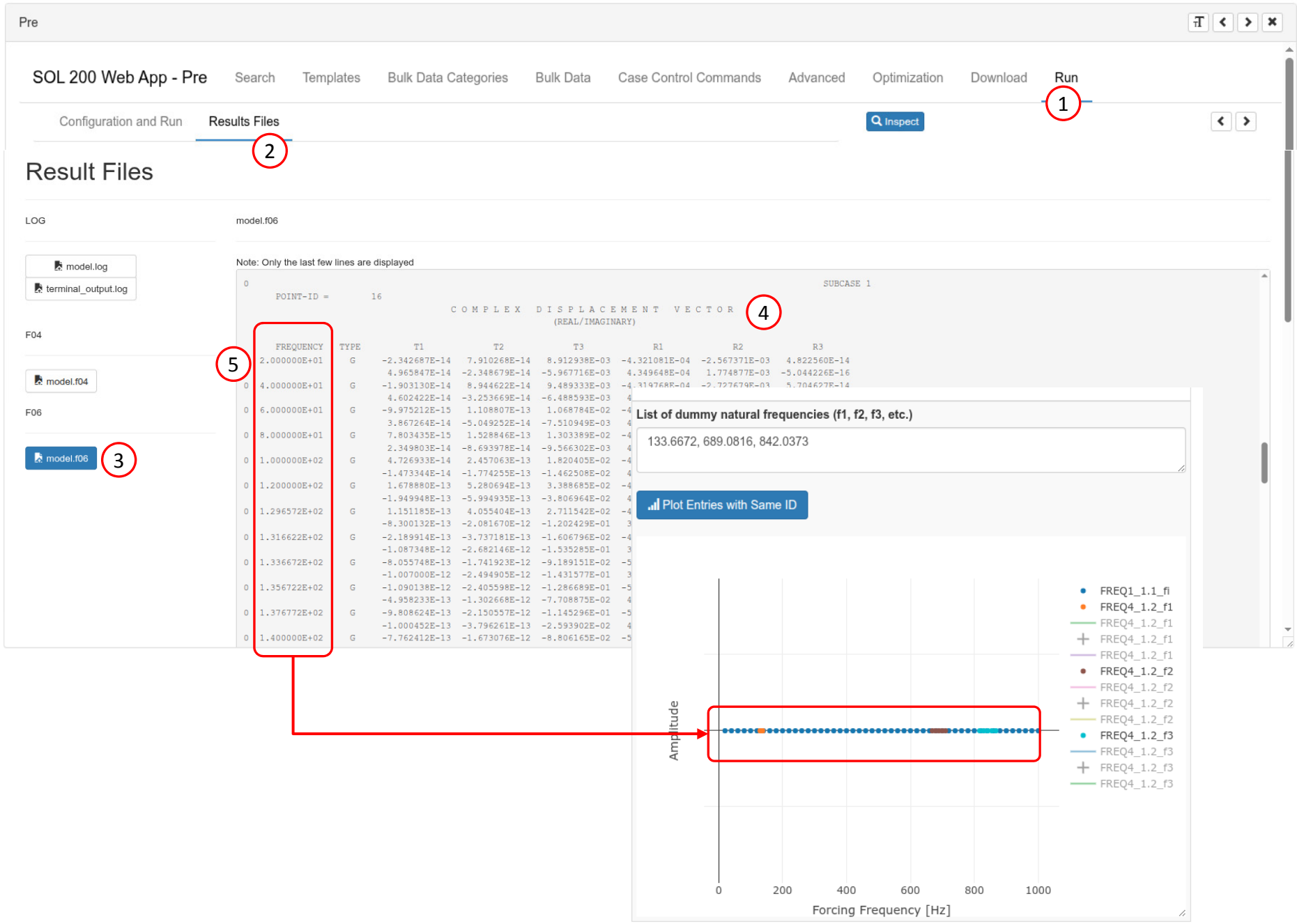
1. Click Pre





F06 - Excitation Frequencies

1. Click Run
2. Click Result Files
3. In the left hand side, click model.f06 to display the contents of the file.
4. Vertically scroll until table COMPLEX DISPLACEMENT VECTOR is displayed.
5. The excitation frequencies in the vertical column are identical to the points shown in the plot for FREQ1 and FREQ4.



F06 - Displacements

1. Recall that displacements are output for the following grid IDs: 2, 3, 16
2. The f06 file lists the displacements at these grid IDs

BDF Output

```
$   1   ||   2   ||   3   ||   4   ||
SET 1 = 2,3,16
ECHO=NONE
$ Dynamic Solution Conditions
FREQUENCY=1
METHOD=1
SDAMPING=2
$ Physical Set Output Requests
DISPLACEMENT(SORT2,PHASE)=1
$ Loads and BCs
SPC=1
SUBCASE 1
  $ Loads and BCs
  DLOAD=9001
```

(1)

model.f06

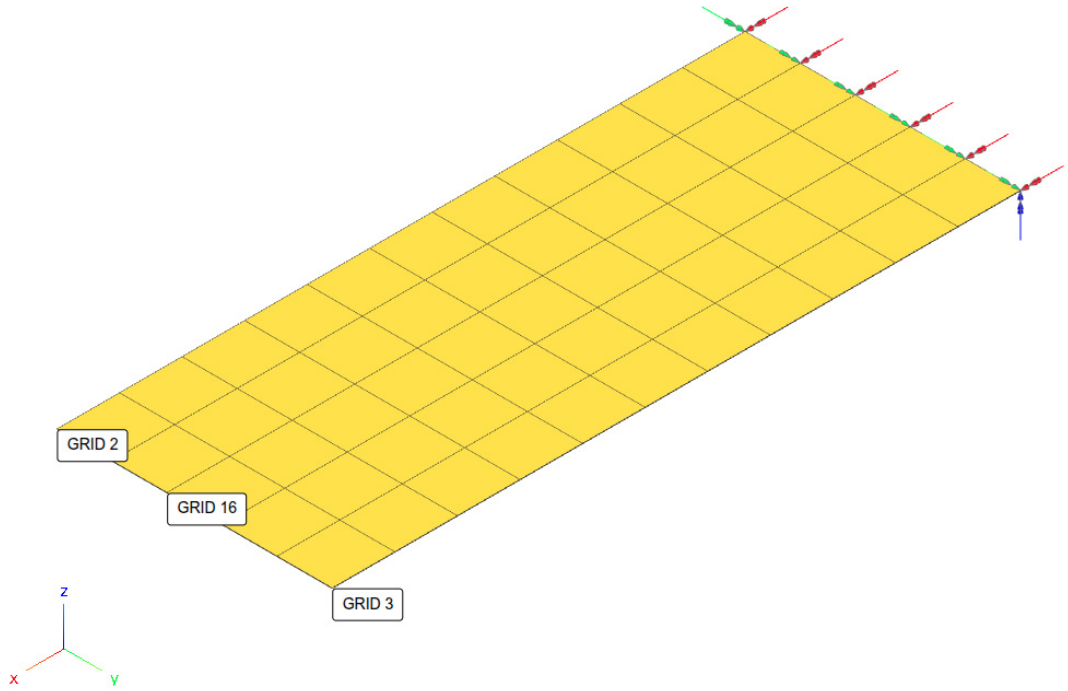
Note: Only the last few lines are displayed

0 SUBCASE 1

POINT-ID = 16 (2)

C O M P L E X D I S P L A C E M E N T V E C T O R
(MAGNITUDE/PHASE)

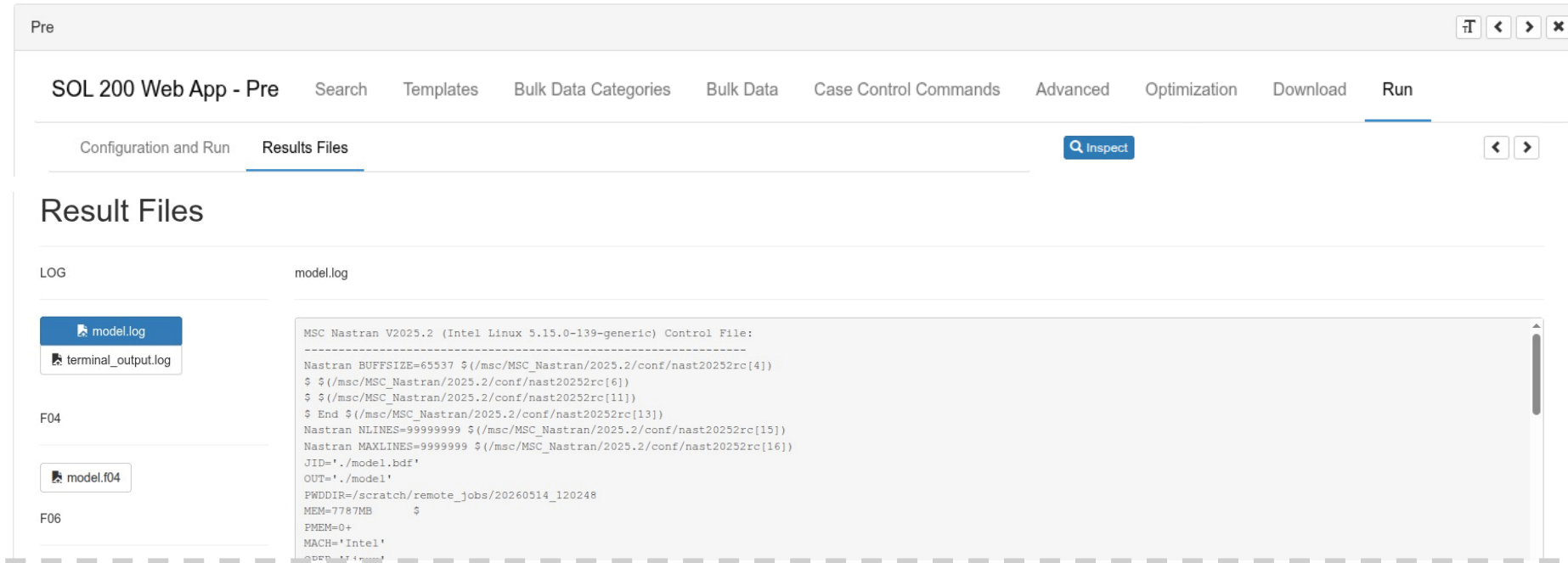
	FREQUENCY	TYPE	T1	T2	T3	R1	R2	R3
0	2.000000E+01	G	5.490703E-14	8.251583E-14	1.072633E-02	6.131165E-04	3.121151E-03	4.822823E-14
			115.2561	343.4630	326.1955	134.8112	145.3431	359.4007
0	4.000000E+01	G	4.980381E-14	9.518016E-14	1.149562E-02	6.138920E-04	3.335661E-03	5.762573E-14
			112.4654	340.0108	325.6366	134.7221	144.8585	351.8679
0	6.000000E+01	G	3.993843E-14	1.218360E-13	1.306309E-02	6.150125E-04	3.772427E-03	7.885150E-14
			104.4636	335.5166	324.9021	134.6265	144.2372	342.7582
0	8.000000E+01	G	2.475987E-14	1.758756E-13	1.616775E-02	6.160002E-04	4.637226E-03	1.236888E-13
			71.6292	330.3747	323.7230	134.5235	143.2137	333.9327
0	1.000000E+02	G	4.951226E-14	3.030699E-13	2.335124E-02	6.151607E-04	6.637636E-03	2.312709E-13
			342.6882	324.1668	321.2217	134.4235	140.9160	325.3615
0	1.200000E+02	G	2.573117E-13	7.989054E-13	5.096682E-02	6.015903E-04	1.432464E-02	6.541004E-13

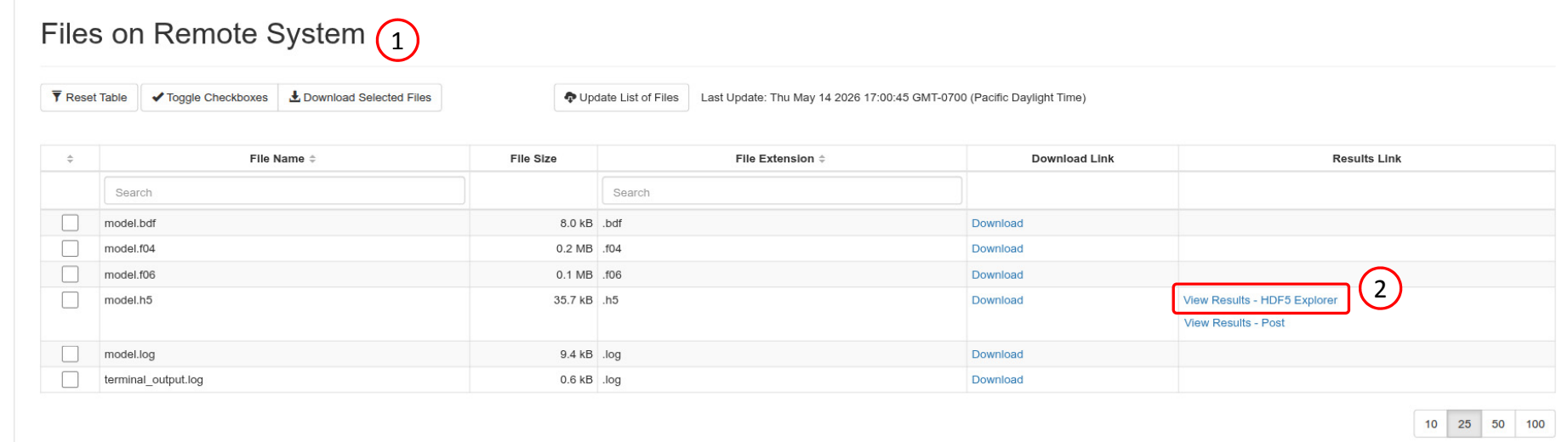


Displacement vs. Frequency Graphs/Plots

Graphs are desired that show Displacement vs. Frequency.

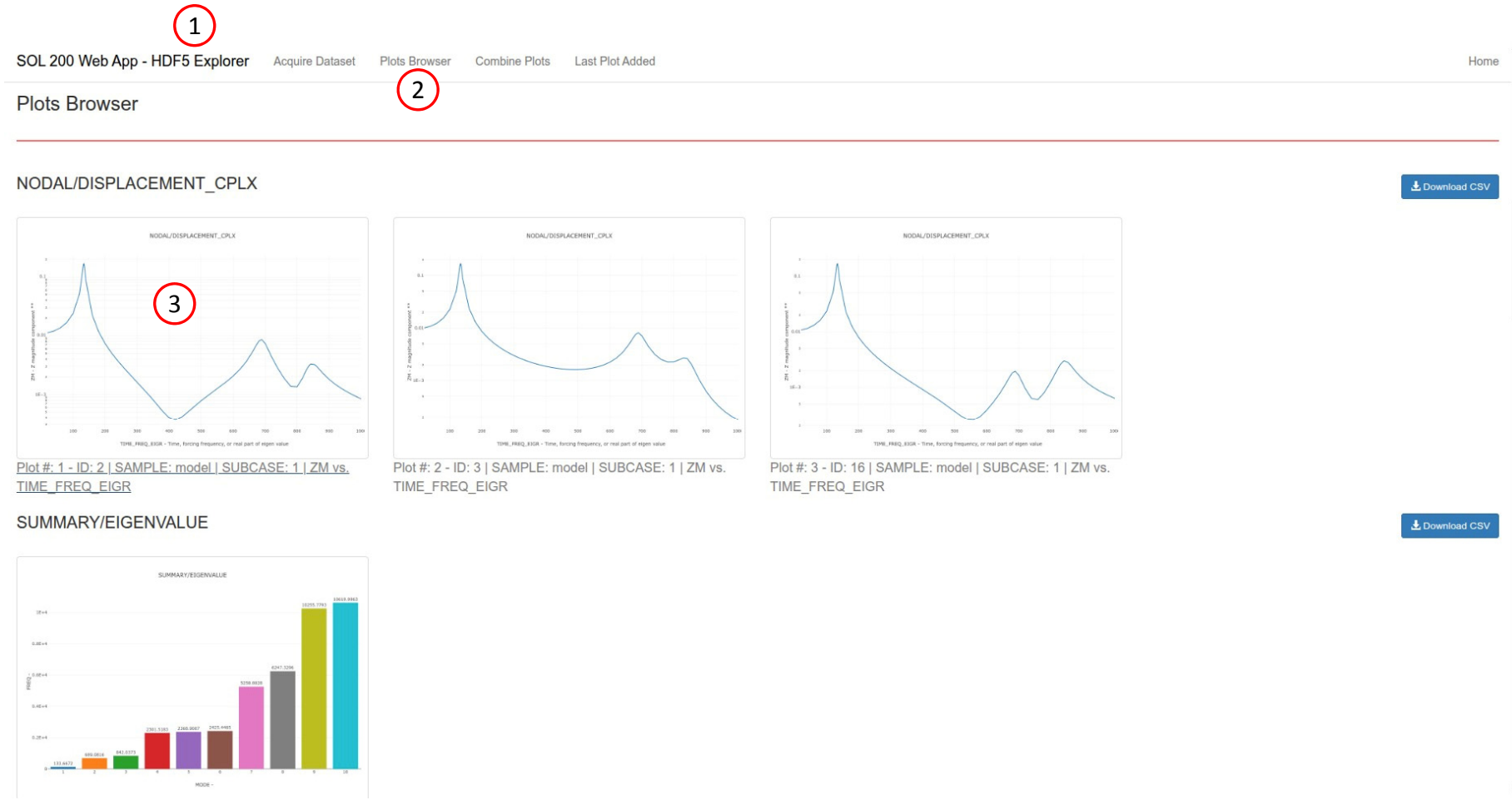
1. Scroll to the section titled Files on Remote System
2. For the row listing file model.h5, click View Results - HDF5 Explorer





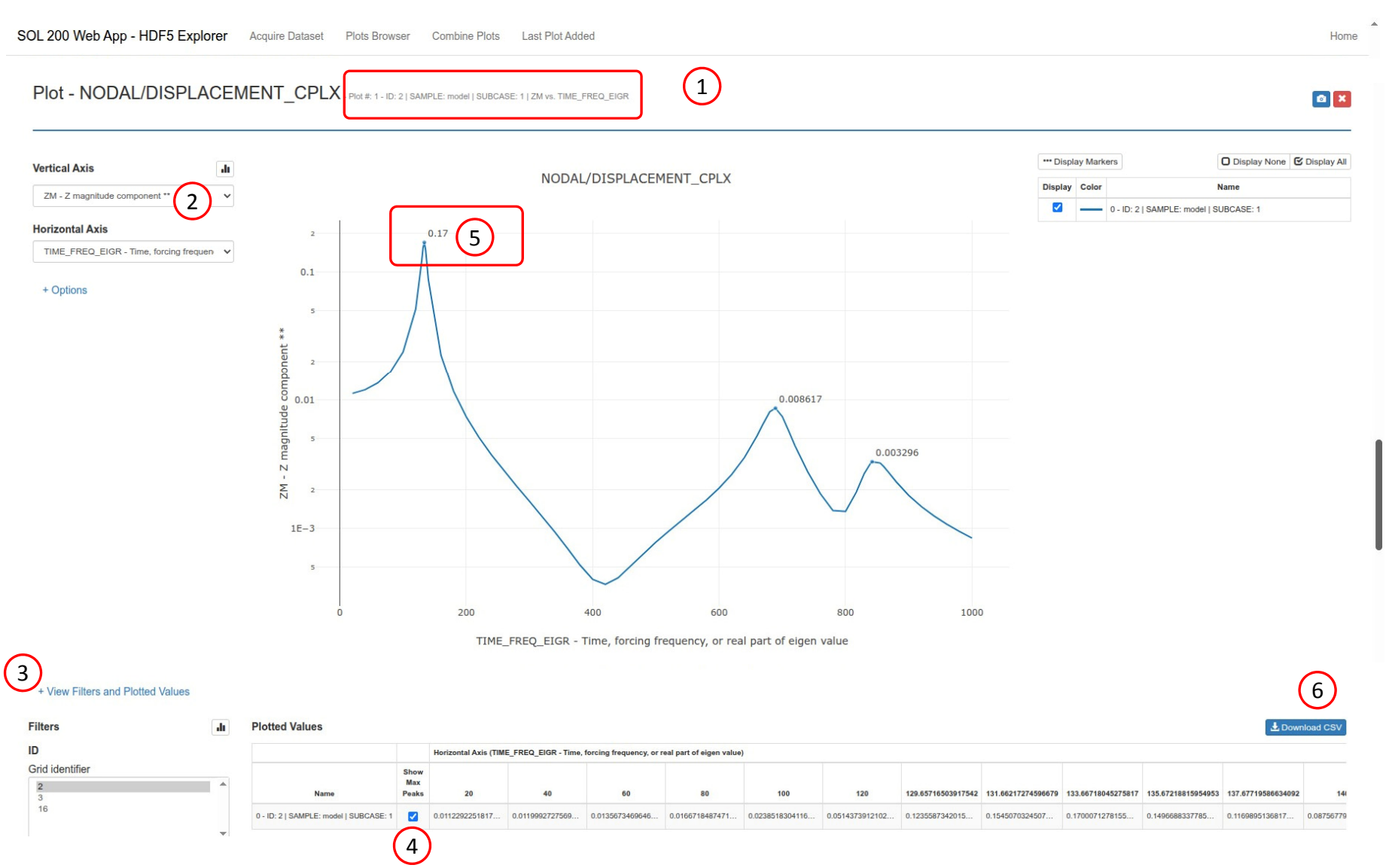
Displacement vs. Frequency Graphs/Plots

1. The HDF5 Explorer is opened. Wait until loading is complete.
2. Click Plots Browser
3. Click the indicated plot
   - This graph displays displacement vs. frequency for GRID ID 2.



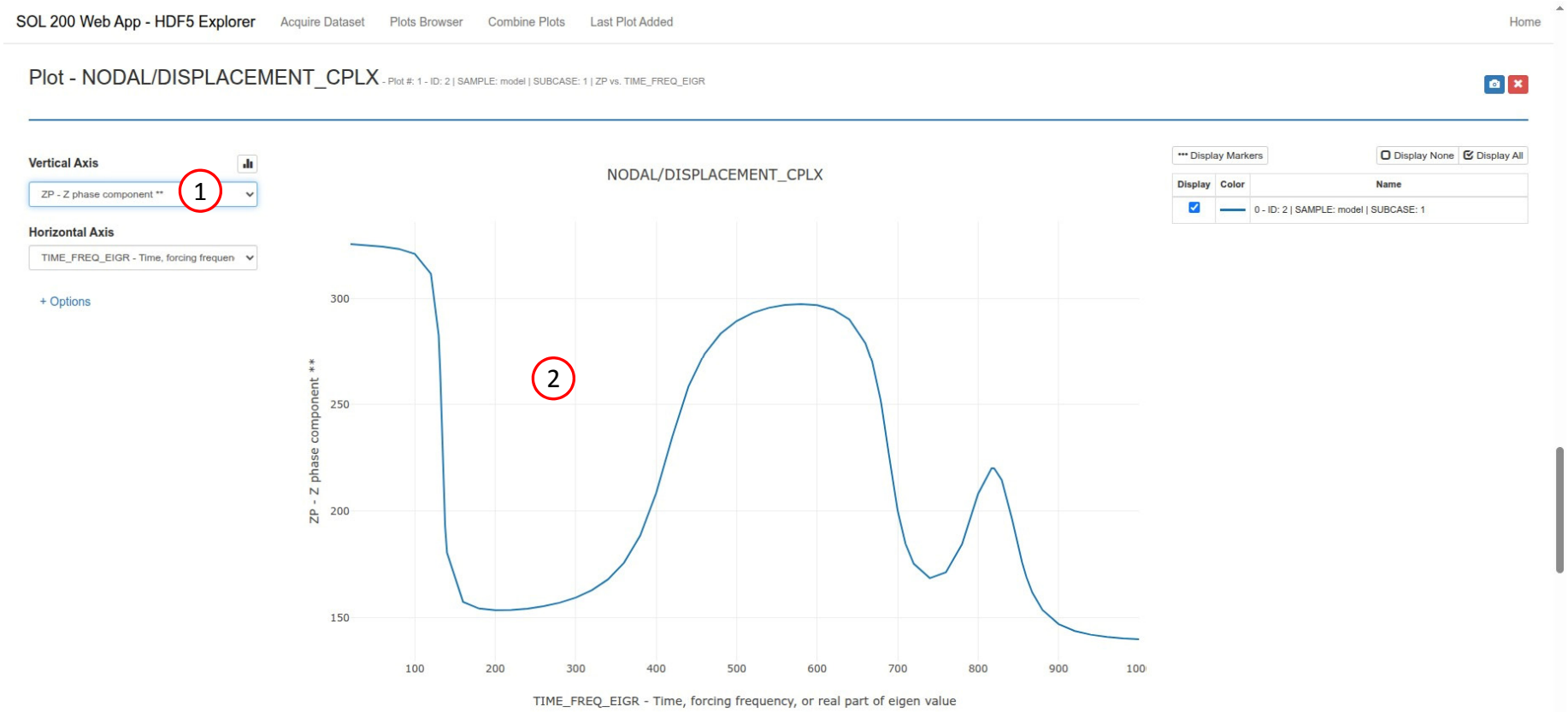
Displacement vs. Frequency Graphs/Plots

1. The plot for displacement vs. frequency is displayed at grid 2
2. Currently, the ZM component is displayed
3. Click View Filters and Plotted Values
4. Mark he indicated checkbox
5. Maximum labels are now visible on the graph
6. Click Download CSV
   - A CSV file is downloaded and contains the values used to create the plot



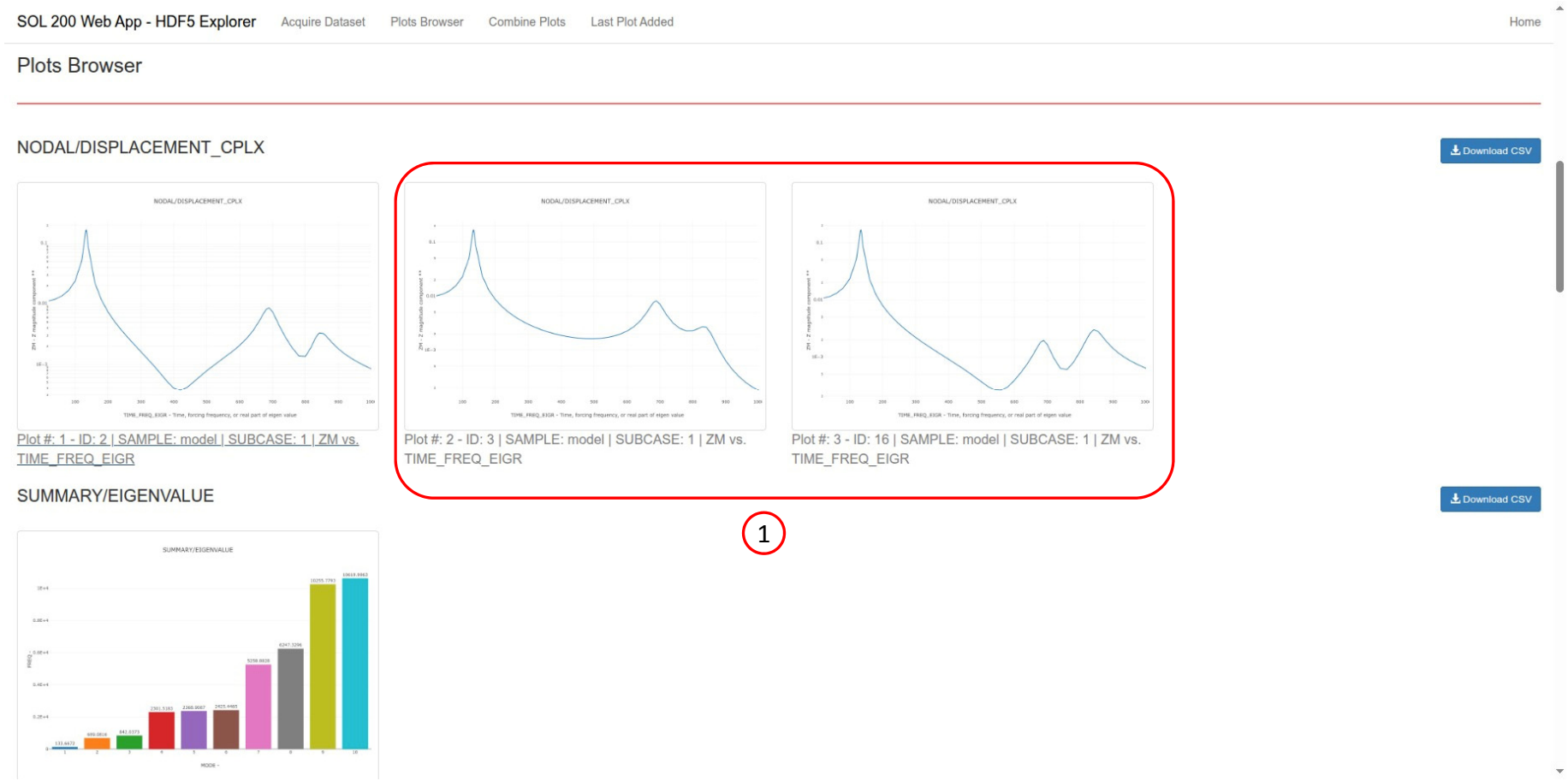
Displacement vs. Frequency Graphs/Plots

1. Set Vertical Axis to ZP - Z phase component
2. The graph is updated to show the ZP component



Displacement vs. Frequency Graphs/Plots

1. Repeat the previous steps for node 3 and 16.



Final Comments - MODEL SUMMARY

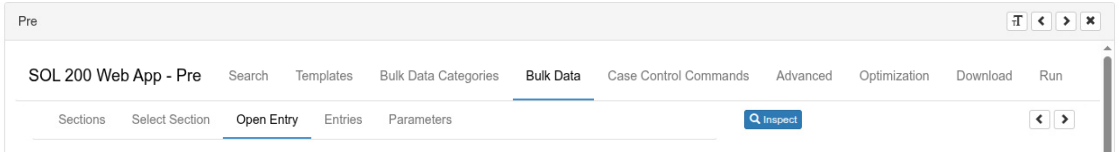
These steps are optional.

A. The F06 file contains a MODEL SUMMARY table and lists bulk data entries Nastran has read from the BDF file.

B. The web app has a similar table. Navigate to Pre > Bulk Data > Open Entry.

C. Click Display Active.

D. A listing is displayed of bulk data entries that have been defined



M O D E L S U M M A R Y

ENTRY NAME	NUMBER OF ENTRIES
CQUAD4	55
DLOAD	1
EIGRL	1
FORCE	1
FREQ1	1
FREQ4	1
GRID	72
HDF5OUT	1
MAT1	1
PARAM	2
PLOAD2	1
PSHELL	1
RLOAD2	2
SPC1	1
TABDMP1	1
TABLED1	1

A

Open Entry B

C

Display Title | Display Description | Display Active | Reset Table

Open	Entry Name	Active	Number of Entries
	Search		
	SET	▲	1
	CQUAD4	▲	55
	DLOAD	▲	1
	EIGRL	▲	1
	FORCE	▲	1
	FREQ1	▲	1
	FREQ4	▲	1
	GRID	▲	72
	HDF5OUT	▲	1
	MAT1	▲	1
	PLOAD2	▲	1
	PSHELL	▲	1
	RLOAD2	▲	2
	SPC1	▲	1
	TABDMP1	▲	1
	TABLED1	▲	1

D

BDF Templates

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Conclusion

The goal of this exercise was to demonstrate how to configure a normal modes, linear statics and modal frequency response analysis.

Constructing BDFs is an extensive process. A secondary goal was to detail an efficient strategy to build Nastran Bulk Data Files (BDF).

- The procedure below was used to create numerous keywords that define materials, properties, loads, etc.:
  1. Search
  2. Configure
  3. Confirm
- Towards the end of the exercise, a BDF template was downloaded and is a nearly complete BDF that avoids needing to re-define a majority of the BDF.

End of Tutorial

Appendix

Appendix

- How to display additional information for FREQ4 entries

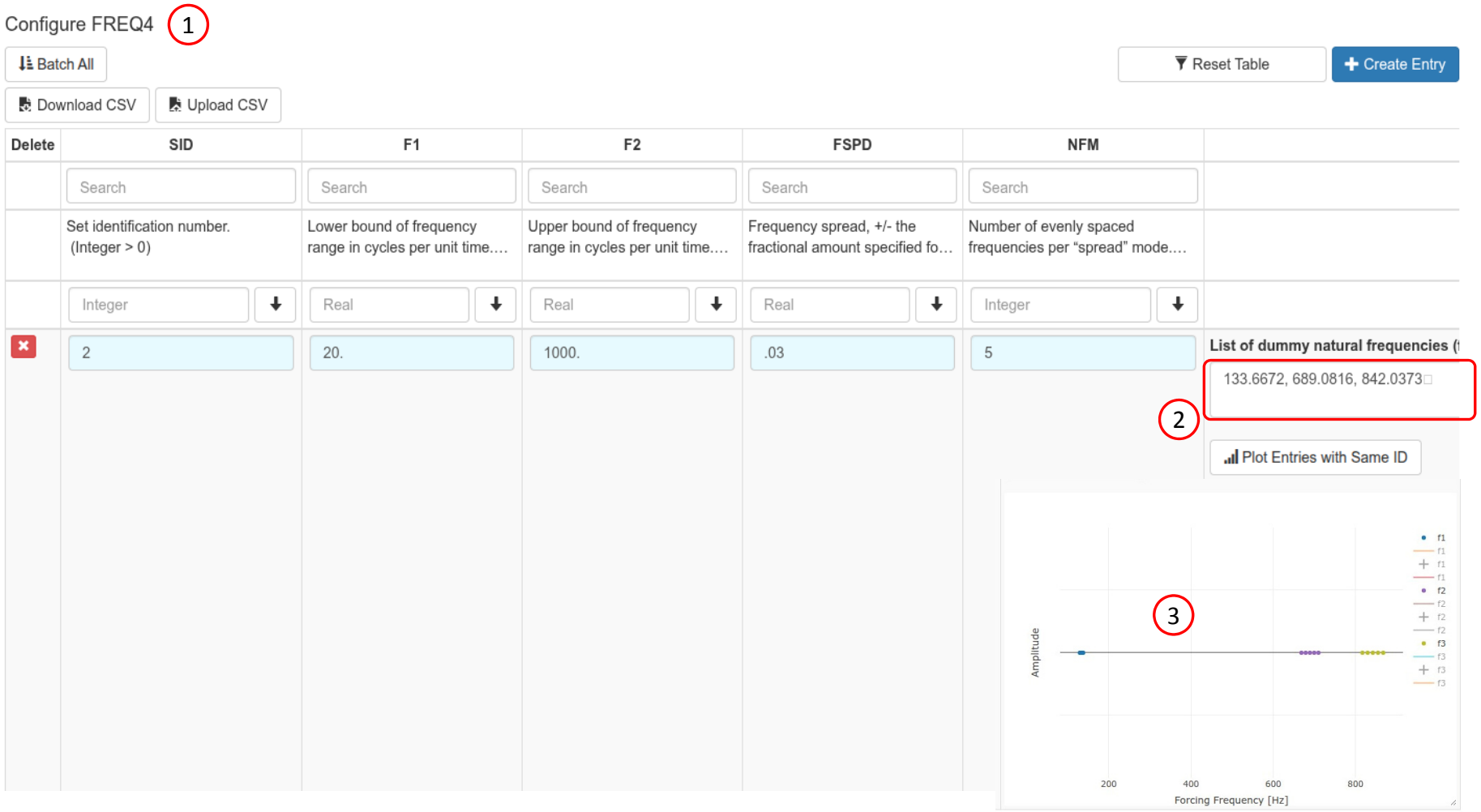
How to display additional information for FREQ4 entries

FREQ4 - Frequency List, Alternate Form 4

1. Refer to the table Configure FREQ4
2. Supply the following values
   - List of dummy natural frequencies:

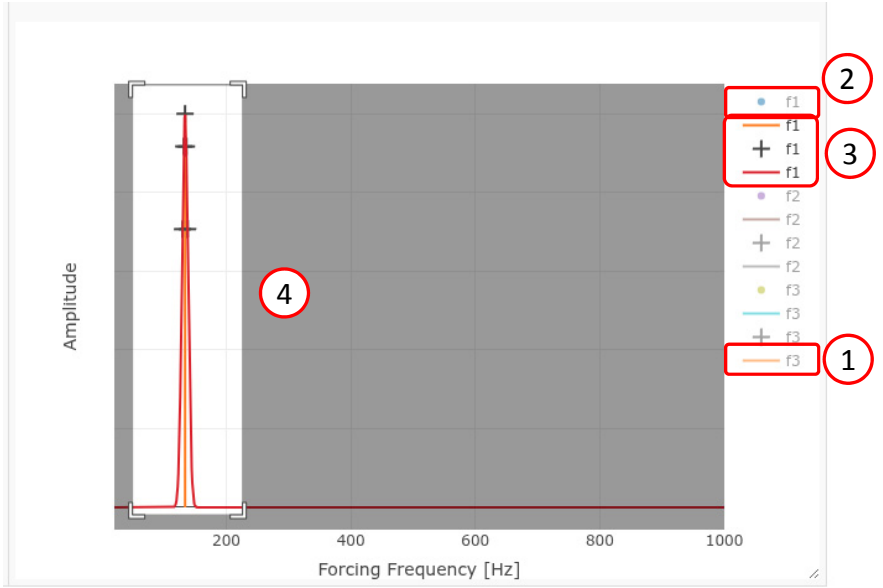
     133.6672, 689.0816, 842.0373

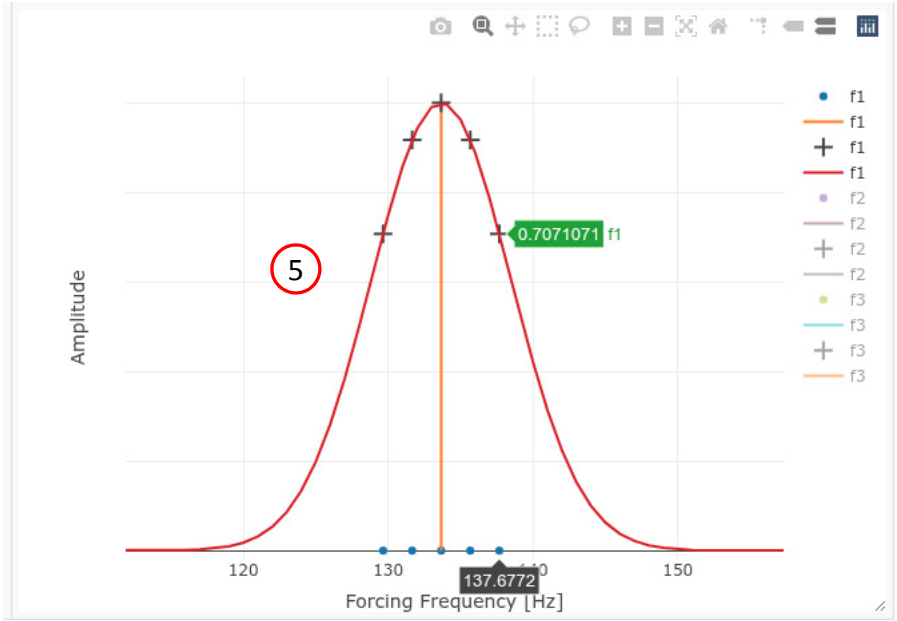
     These are the natural frequencies from the normal modes analysis. Alternatively you can supply dummy test frequencies of your liking.
   - These dummy or test frequencies are referred to as f1, f2, and f3, respectively.
3. The column Plot displays the corresponding excitation frequencies.



FREQ4 - Frequency List, Alternate Form 4

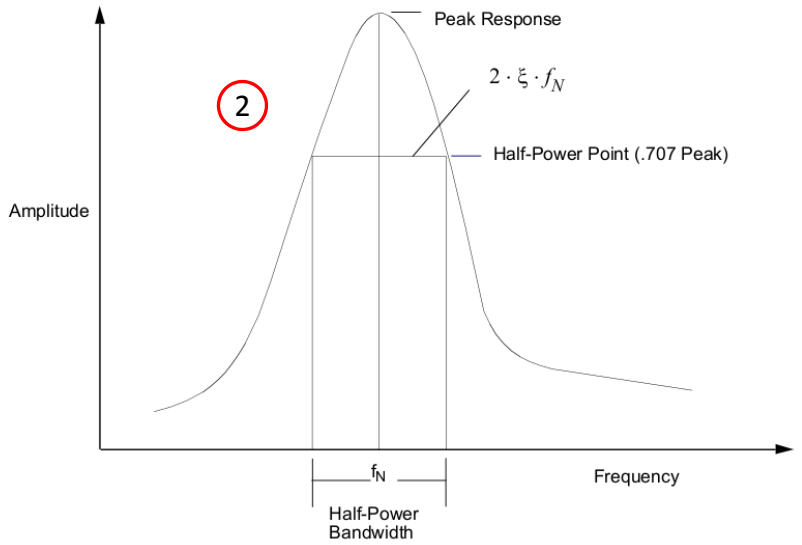
1. Double click the indicated row
   - This will toggle the visibility so only one row is visible. Double clicking again will display all rows.
2. Double click the indicated row
3. Individually click the indicated rows
4. Click, drag and select the indicated region
5. The plot is now updated
   - The excitation frequencies for natural frequency f1 are displayed.

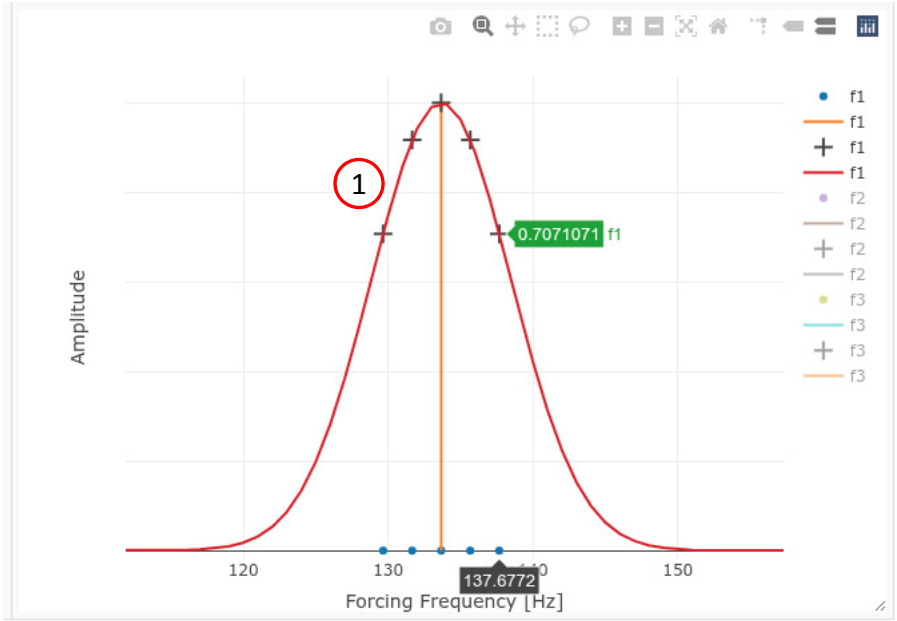




FREQ4 - Frequency List, Alternate Form 4

1. Five evenly spaced excitation frequencies between the half power points of resonant frequency f1 are shown
2. This plot in the web app aligns with the plot shown in the Quick Reference Guide, entry FREQ4



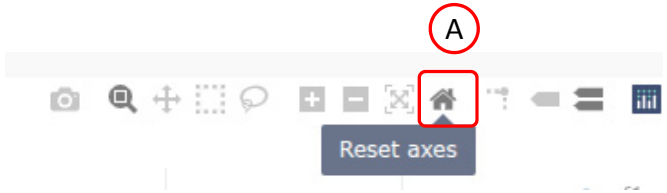


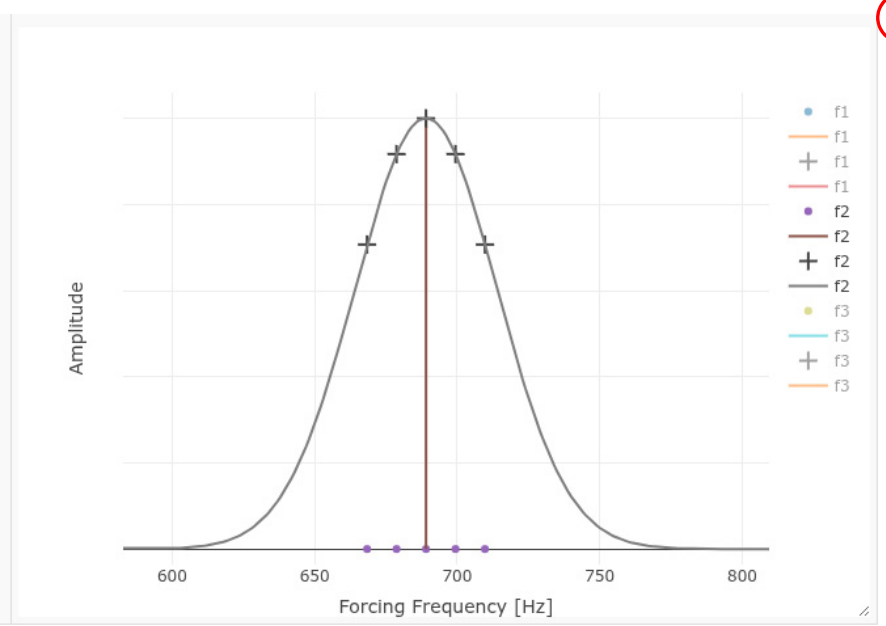
Excitation Frequencies for f1

FREQ4 - Frequency List, Alternate Form 4

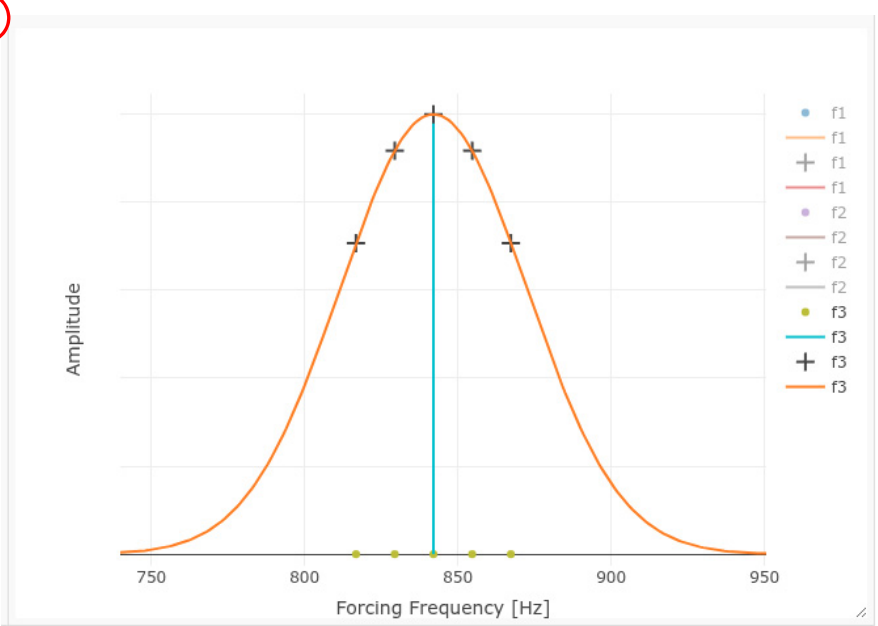
1. Repeat the previous steps for frequency f2 and f3.

A. If needed, click the Reset Axes button to restore the plot to a default state.





Excitation Frequencies for f2



Excitation Frequencies for f3