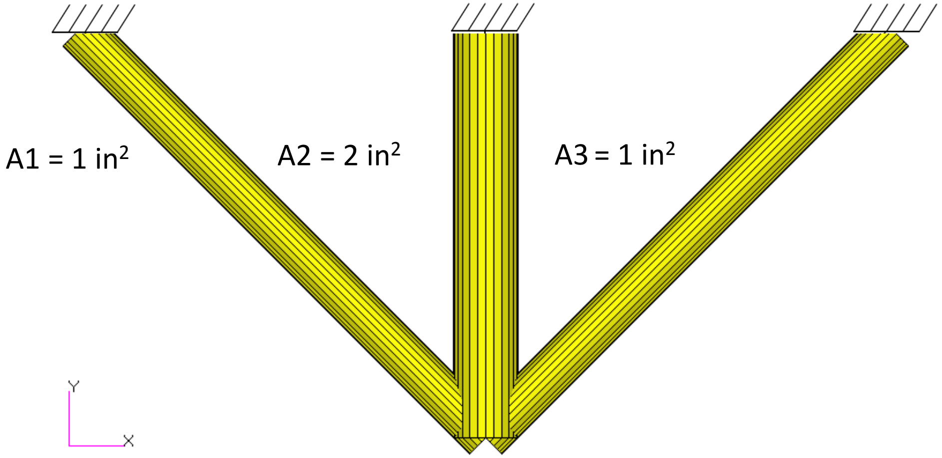
Workshop - Structural Optimization of a 3 Bar Truss

AN MSC NASTRAN SOL 200 TUTORIAL

Goal: Use Nastran SOL 200 Optimization

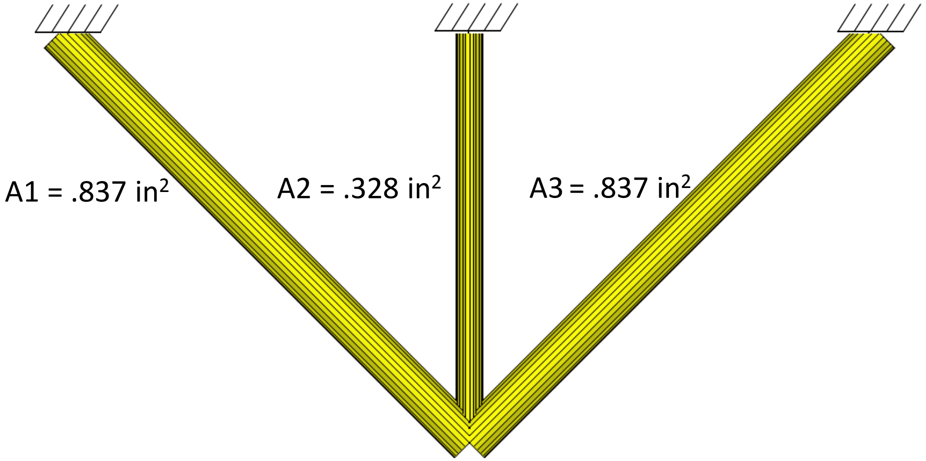
Before Optimization

- Weight: 4.82 lbs.



After Optimization

- Weight: 2.70 lbs.



Details of the Structural Model

Three-Bar Truss

A common task in design optimization is to reduce the mass of a structure subjected to several load conditions. Figure 8-1 shows a simple three-bar truss that must be built to withstand two separate loading conditions. Note that these two loads subject the outer truss members to both compressive as well as tensile loads. Due to the loading symmetry, we expect the design to be symmetric as well. As an exercise, we'll show how to enforce this symmetry using design variable linking.

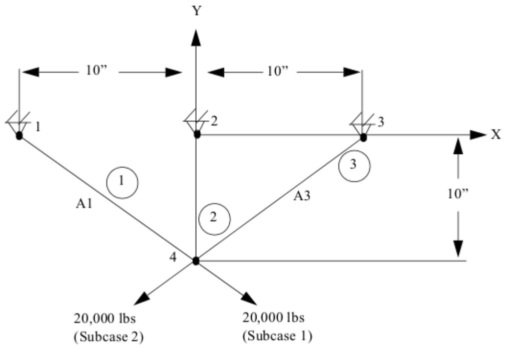
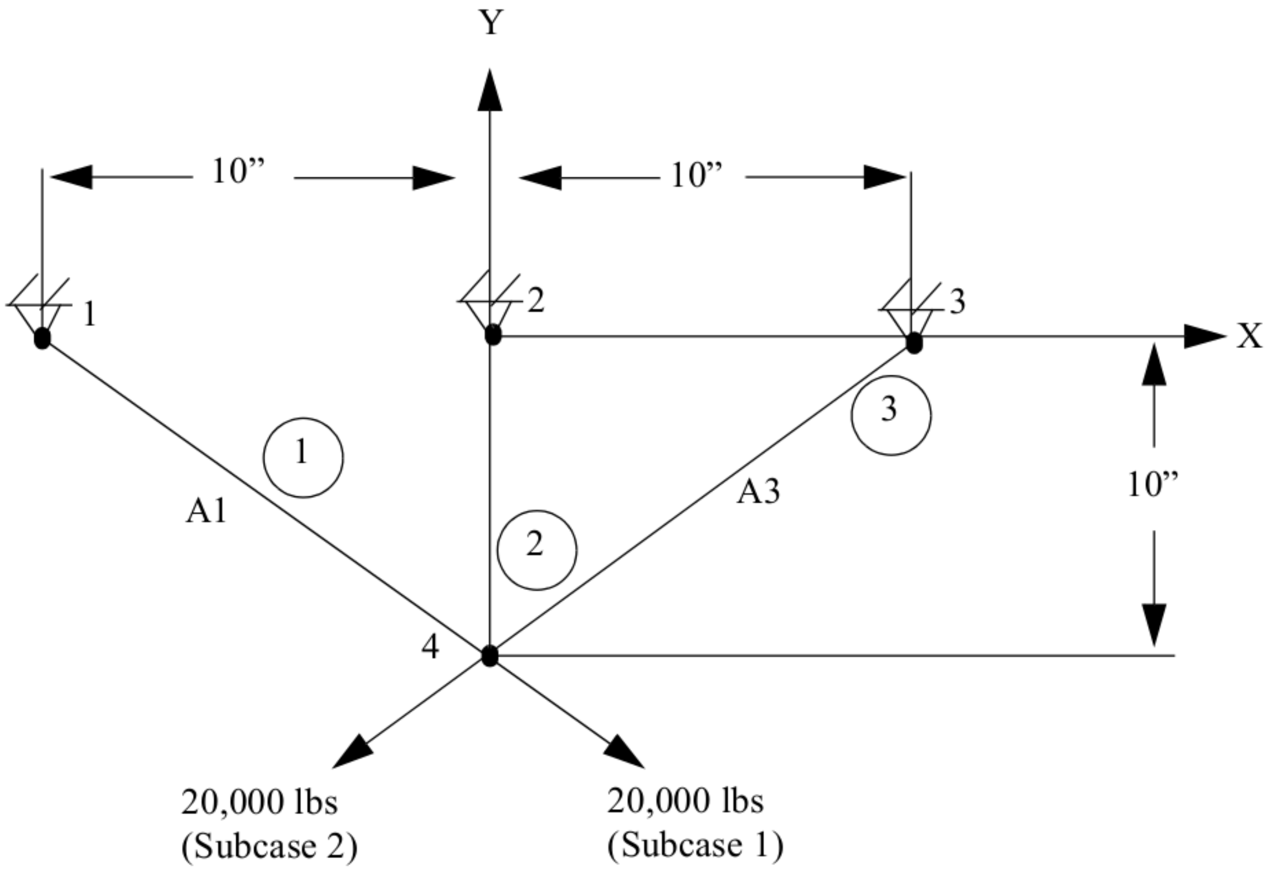


Figure 8-1 Three-Bar Truss

An important, but often overlooked consideration is that the optimization capability in MSC Nastran is multidisciplinary. That is, the final optimal design is the result of a simultaneous consideration of all analysis disciplines across all subcases. In this case, the optimal three-bar truss design will satisfy the load requirements for both statics subcases, which is to be expected. (If, for example, a normal modes or buckling subcase were to be added, the resultant design would have to not only satisfy the static strength requirements, but also constraints on eigenvalues. As an exercise you may wish to try adding an eigenvalue constraint.)

MSC Nastran Design Sensitivity and Optimization User’s Guide
Chapter 8 - Example Problems - Three Bar Truss



Optimization Problem Statement

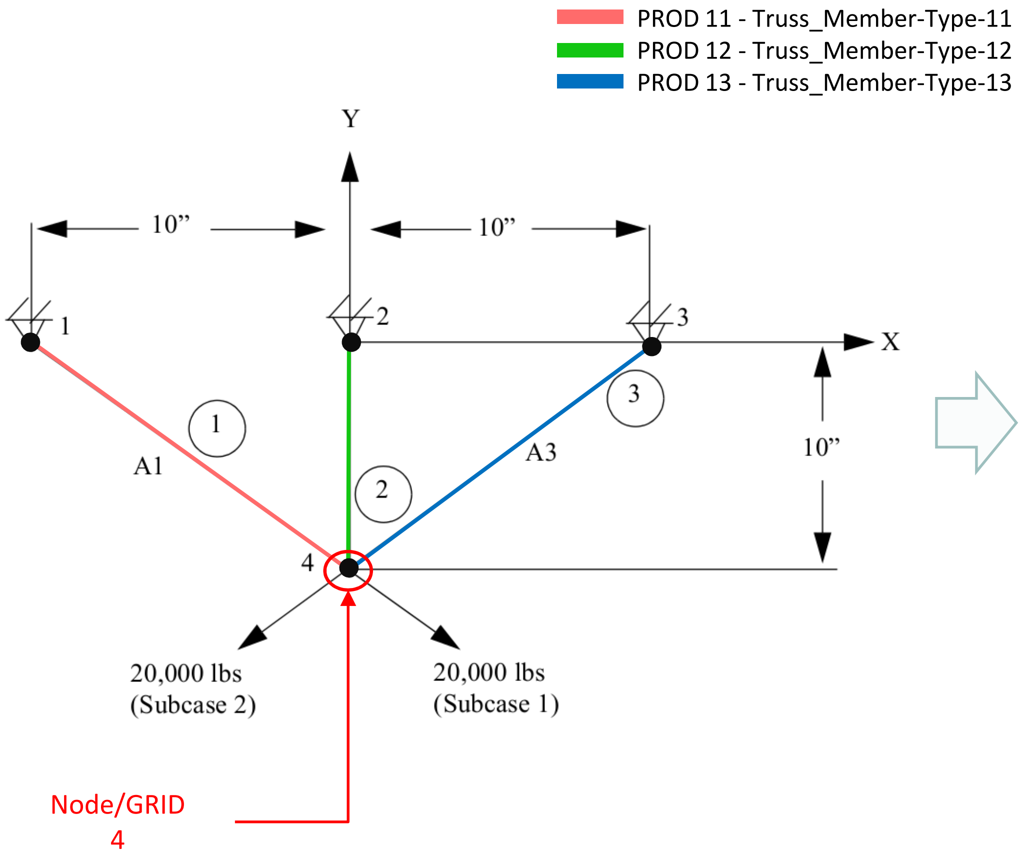
Design Variables

x1: A of PROD 11
x2: A of PROD 12
x3: A of PROD 13

.1 < x1, x2, x3 < 100.

Variable Link

x3 = x1



Design Objective

r0: Minimize weight

Design Constraints

r1: Axial stress of elements related to PROD 11, 12, 13

-15000 < r1 < 20000

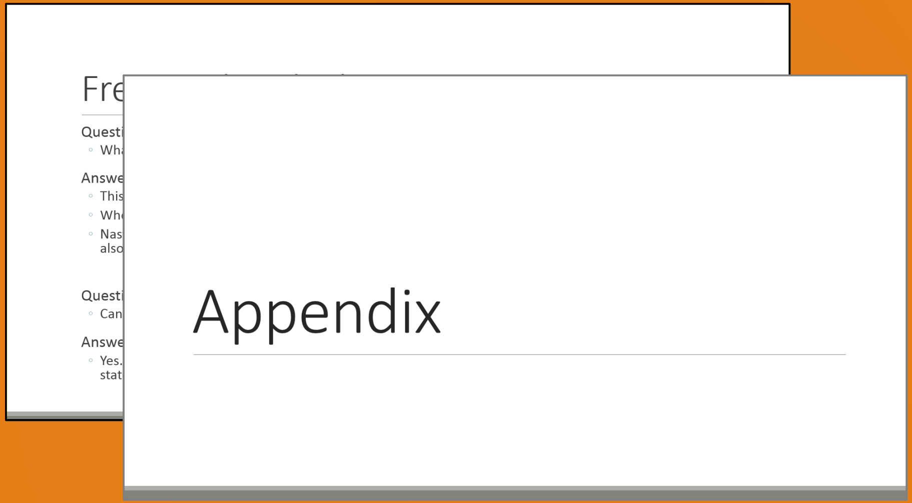
r2: x and y component of displacement for node 4

-.2 < r2 < .2

More Information Available in the Appendix

The Appendix includes information regarding the following:

- Frequently Asked Questions
  - What does this line mean, **INCLUDE './design_model.bdf'** ?
  - Can design_model.bdf be renamed?



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

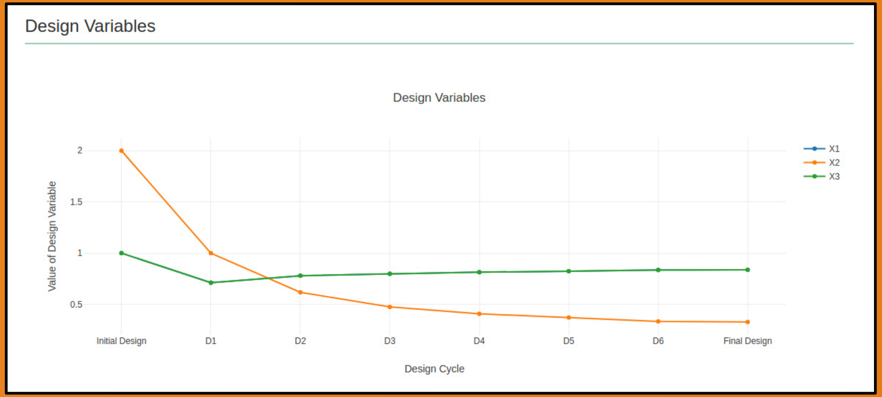
Tutorial

Tutorial Overview

1. Start with a .bdf or .dat file
2. Use the SOL 200 Web App to:
   - Convert the .bdf file to SOL 200
     - Design Variables
     - Design Objective
     - Design Constraints
   - Perform optimization with Nastran SOL 200
3. Plot the Optimization Results
4. Update the original model with optimized parameters

Special Topics Covered

Automatic Plots - After an optimization is complete and result files are created, the change during the optimization process for design variables and objective may be automatically plotted by the Nastran Web App. This tutorial describes how to create these plots. The plotting capability may also be used to plot design sensitivities.

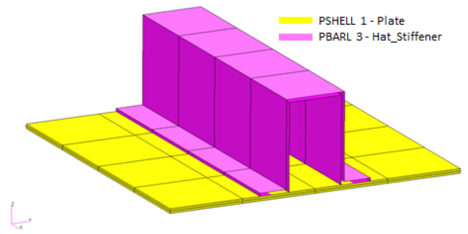


SOL 200 Web App Capabilities

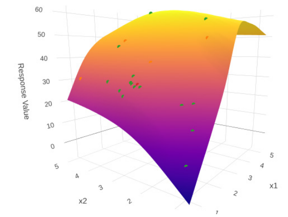
Benefits

- 200+ error validations (real time)
- Web browser accessible
- Automated creation of entries (real time)
- Automatic post-processing
- 76 tutorials

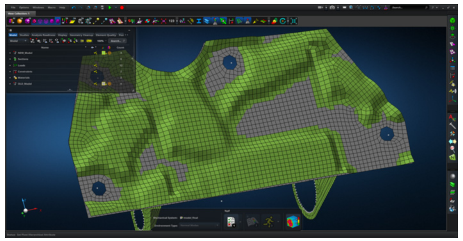
Capabilities



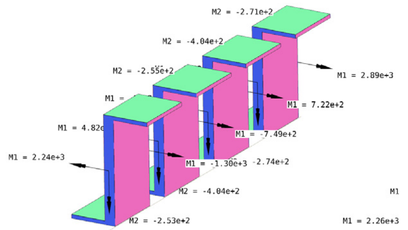
Web Apps for SOL 200
Pre/post for MSC Nastran SOL 200. Support for size, topology, topometry, topography and multi-model.



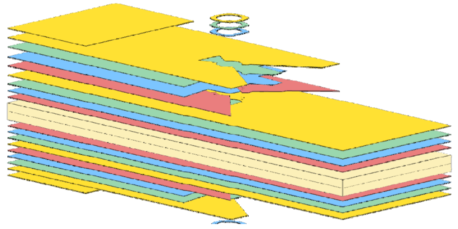
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



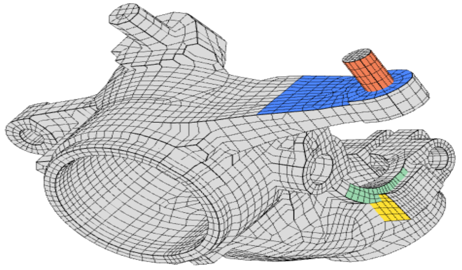
MSC Apex Post Processing Support
View the newly optimized model after an optimization



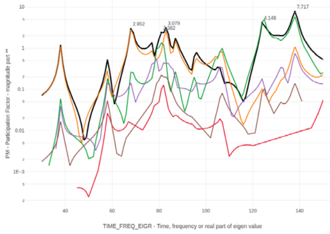
Beams Viewer Web App
Post process 1D element forces, including shear forces, moments, torque and axial forces



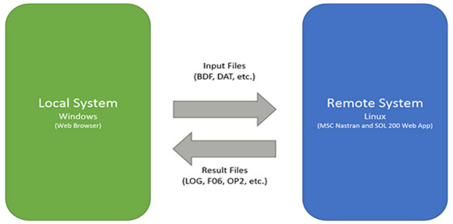
Ply Shape Optimization Web App
Spread plies optimally and generate new PCOMPG entries



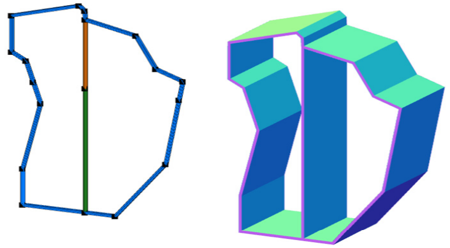
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



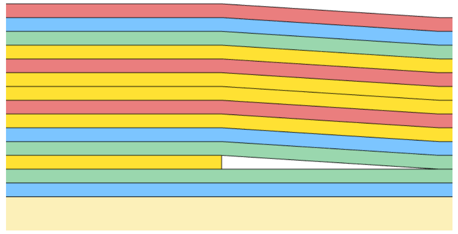
HDF5 Explorer Web App
Create XY plots using data from the H5 file



Remote Execution Web App
Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically

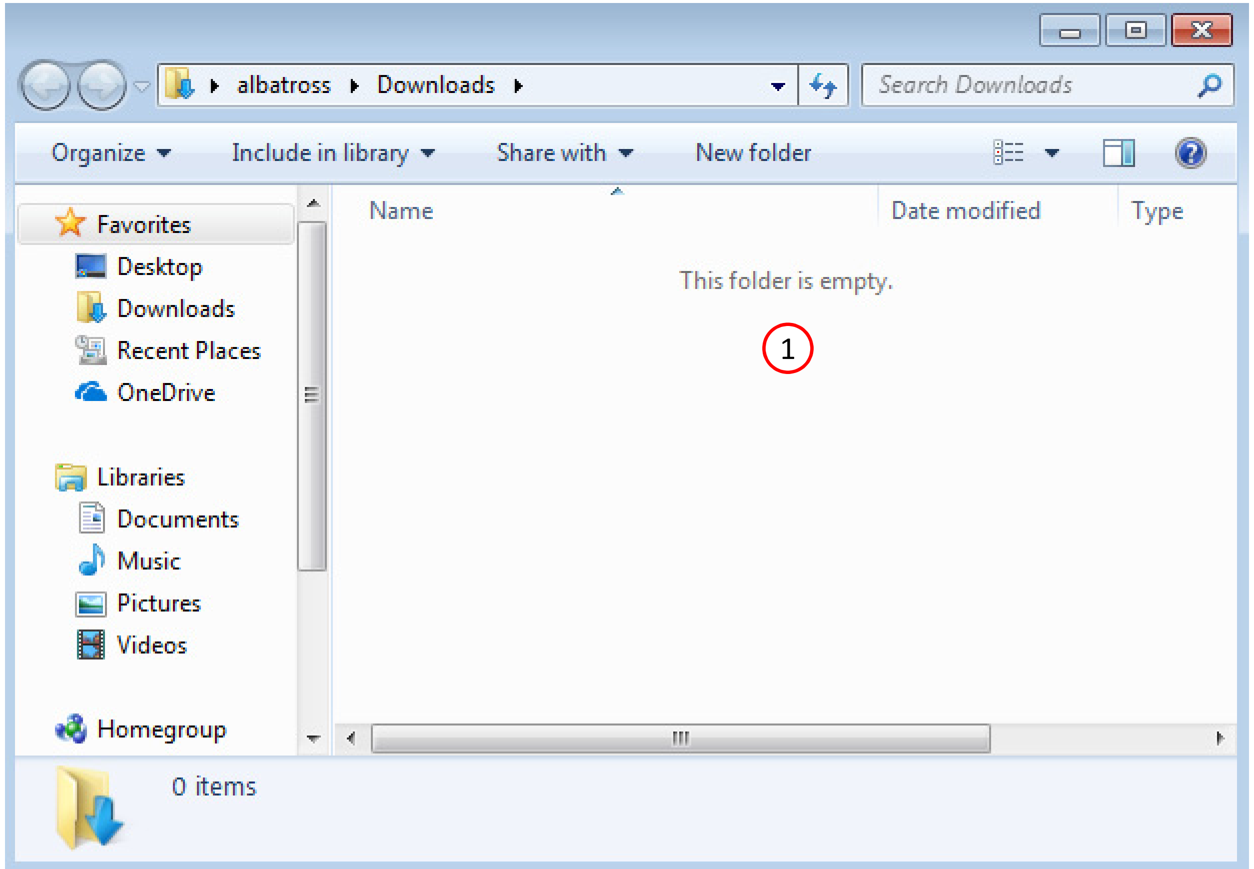


Stacking Sequence Web App
Optimize the stacking sequence of composite laminate plies

Before Starting

1. Ensure the Downloads directory is empty in order to prevent confusion with other files

- Throughout this workshop, you will be working with multiple file types and directories such as:
    - .bdf/.dat
    - nastran_working_directory
    - .f06, .log, .pch, .h5, etc.
- To minimize confusion with files and folders, it is encouraged to start with a clean directory.



Go to the User's Guide

1. Click on the indicated link

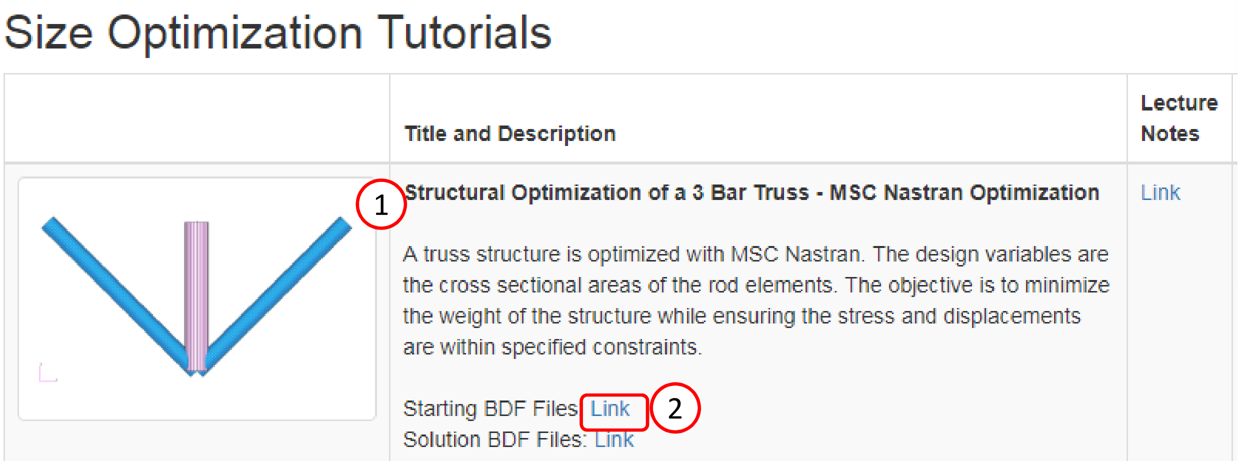
- The necessary BDF files for this tutorial are available in the Tutorials section of the User's Guide.

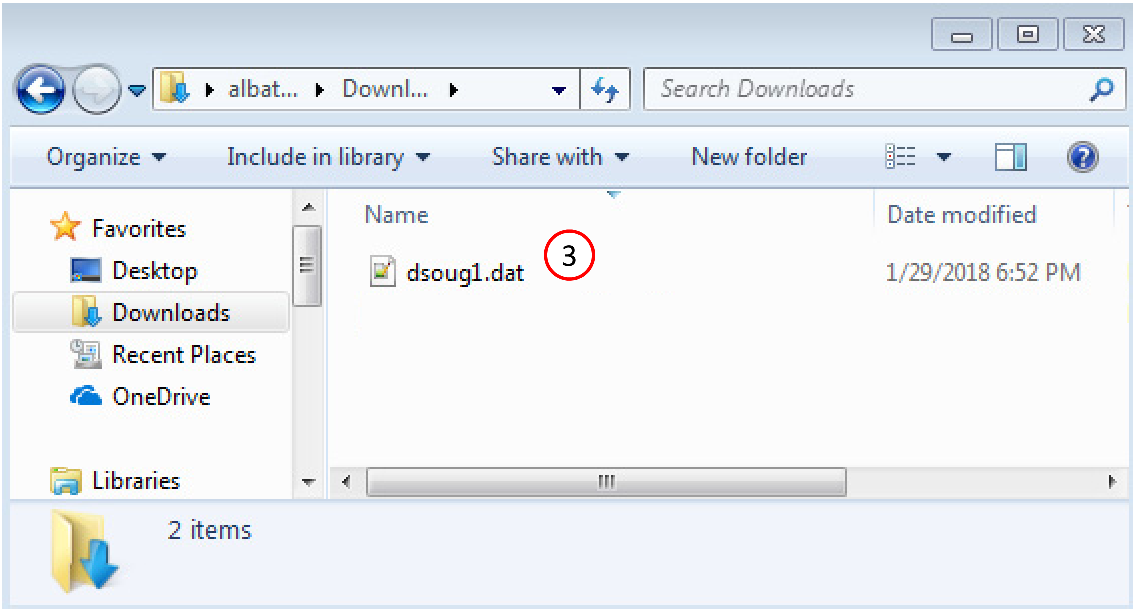
The Engineering Lab

Obtain Starting Files

1. Find the indicated example
2. Click Link
3. The starting file has been downloaded

- When starting the procedure, all the necessary BDF files must be collected together.





Open the Correct Page

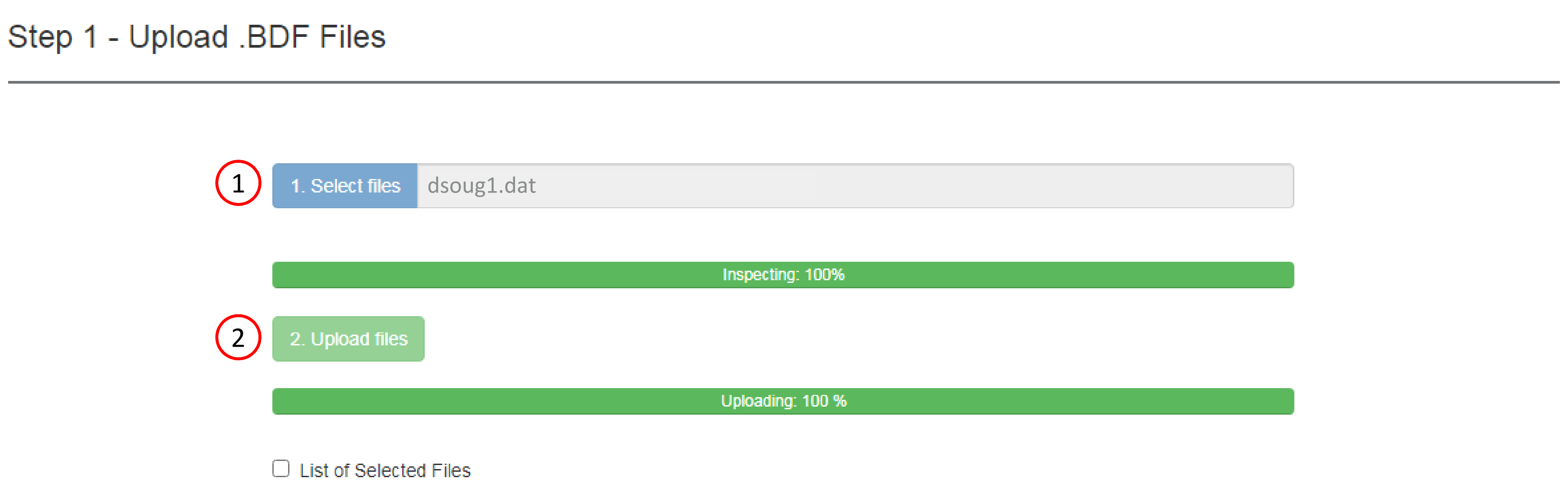
1. Click on the indicated link

- MSC Nastran can perform many optimization types. The SOL 200 Web App includes dedicated web apps for the following:
  - Optimization for SOL 200 (Size, Topology, Topometry, Topography, Local Optimization, Sensitivity Analysis and Global Optimization)
  - Multi Model Optimization
  - Machine Learning
- The web app also features the HDF5 Explorer, a web application to extract results from the H5 file type.

The Engineering Lab

Upload BDF Files

1. Click 1. Select Files and select dsoug1.dat
2. Click Upload Files



- The process starts by uploading all the necessary BDF files. The BDF files can be files of your own or files found in the Tutorials section of the User's Guide.

Create Design Variables

1. Click on the plus (+) icons to set the 3 areas as design variables
2. Specify the lower bound as .01 for design variables x1, x2, and x3
3. Specify the upper bound as 100. for design variables x1, x2, and x3

- Each step has hidden functionality for advanced users. The visibility is controlled by clicking + Options.
- If the property entry, e.g. PSHELL, was given a name in Patran, e.g. Car Door, the name can be shown by marking the checkbox titled Entry Name.

Step 1 - Select design properties

+ Options

Create DVXREL1	Property	Property Description	Entry	Entry ID	Current Value
	Search	Search	Search	Search	Search
+	A	Area of the rod	PROD	11	1.0
+	A	Area of the rod	PROD	12	2.0
+	A	Area of the rod	PROD	13	1.0
+	E	Young's modulus	MAT1	1	1.0E+7
+	NU	Poisson's ratio	MAT1	1	0.33

1

Step 2 - Adjust design variables

Delete Visible Rows

+ Options

	Label	Status	Property	Property Description	Entry	Entry ID	Initial Value	Lower Bound	Upper Bound	Allowed Discrete Values
	Search	Search	Search	Search	Search	Search	Search	Search	Search	Search
✖	x1	✔	A	Area of the rod	PROD	11	1.0	.01	100.	Examples: -2.0, 1.0, THRU, 10.0,
✖	x2	✔	A	Area of the rod	PROD	12	2.0	.01	100.	Examples: -2.0, 1.0, THRU, 10.0,
✖	x3	✔	A	Area of the rod	PROD	13	1.0	.01	100.	Examples: -2.0, 1.0, THRU, 10.0,

2 3

Create Design Variables

1. Click Create DLINK
2. Set the Dependent Design Variable as x3
3. Set the Equation as x1

- It is important to verify the Equation is configured properly. For example, the variable x3 is initially equal to 1.0. When the Equation is configured, it should also produce an initial value of 1.0. The resulting value of the Equation is displayed on the column titled Value of Equation and can be used to validate the Equation is configured properly.

Step 3 - Create variable links

(1) + Create DLINK

+ Options

	Status	Dependent Design Variables	Equation (Independent Design Variables)	Value of Equation
	Search	Search	Search	Search
✖	✔	x3 (2)	x1 (3)	1.

DLINK Entries

DLINK entries are used to create linear relationships between variables.

The SOL 200 Web App allows multiple variations of inputting the linear relationships.

Ultimately, the relationships result in one specific format. To the right are the equivalent linear relationships for the image shown above.

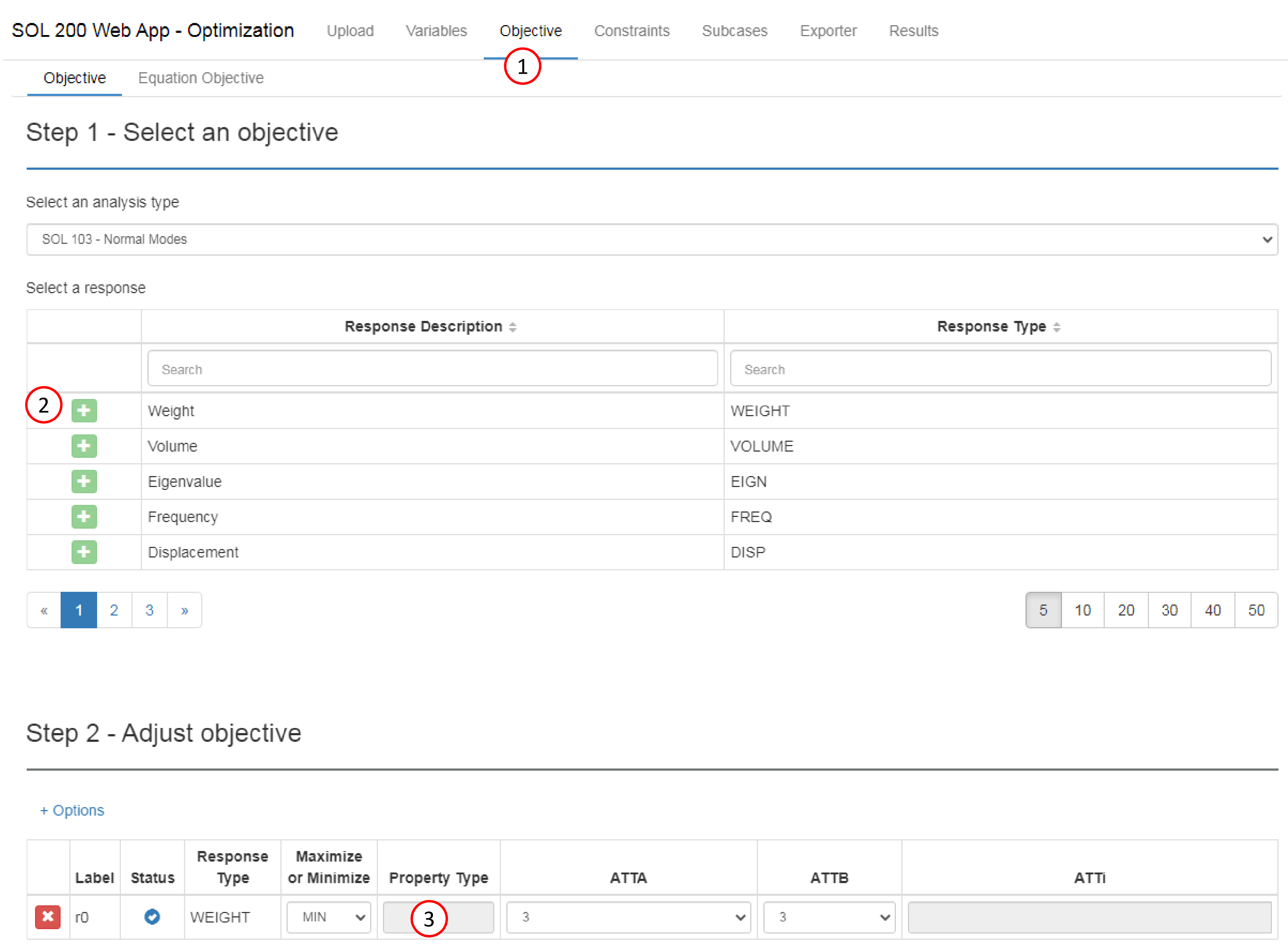
x3 = 0.0 + x1 * 1.0

The right of the expression can also have additional variables. For example, x2 = 1.5 + x1 * 1.0 + y2 * -3.5 + ...

Create Design Objective

1. Click Objective
2. Select the plus (+) icon for weight
3. The objective has been set to minimize the weight, no further modification is necessary

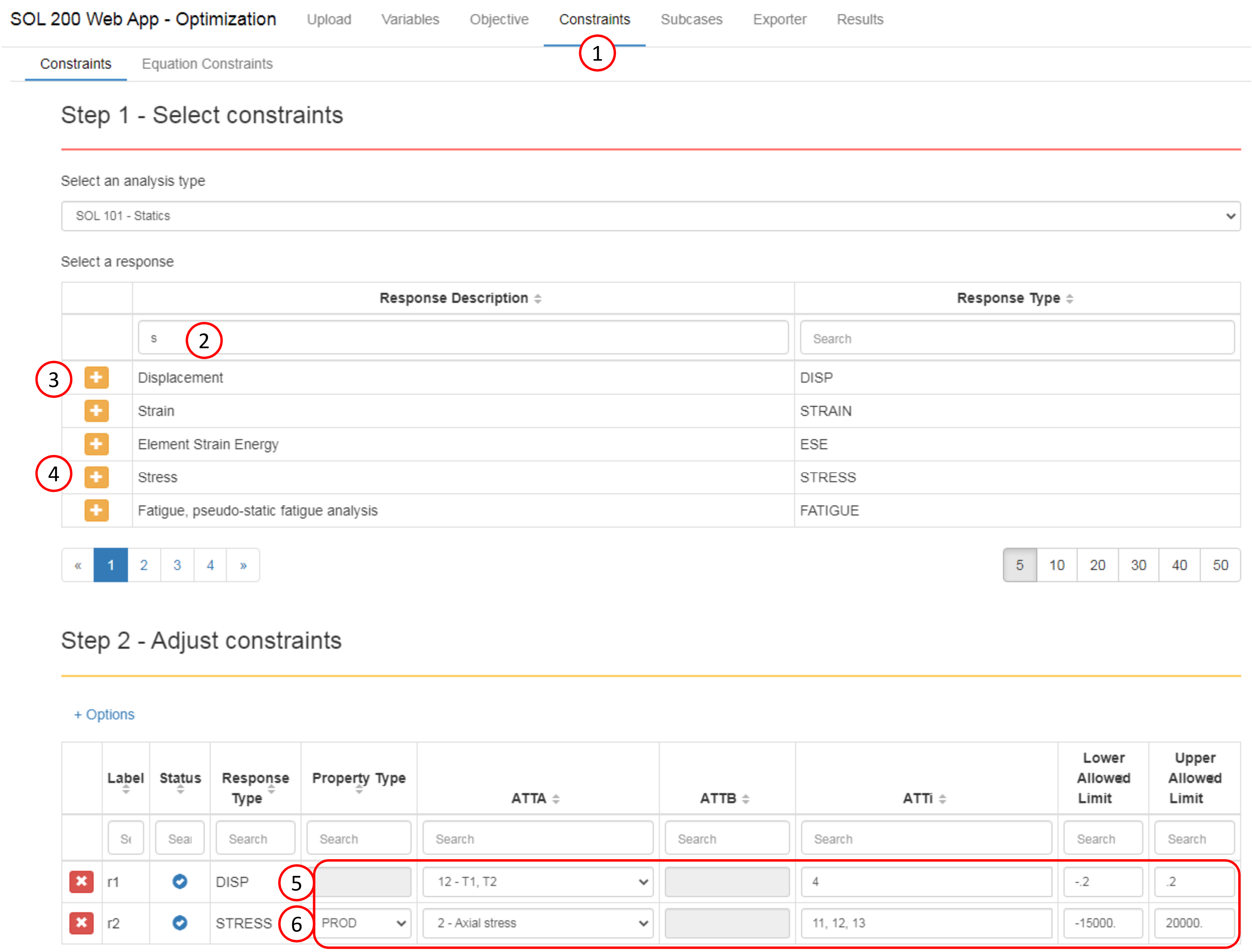
- The objective must always be a single and global response. A response such as weight and volume are single responses, are independent of load case, and can be used as an objective. Other responses require special care when set as an objective. For example, if the objective is stress, only the stress of a single component, e.g. von Mises, of a single element, of a single load case may be used.



Create Design Constraints

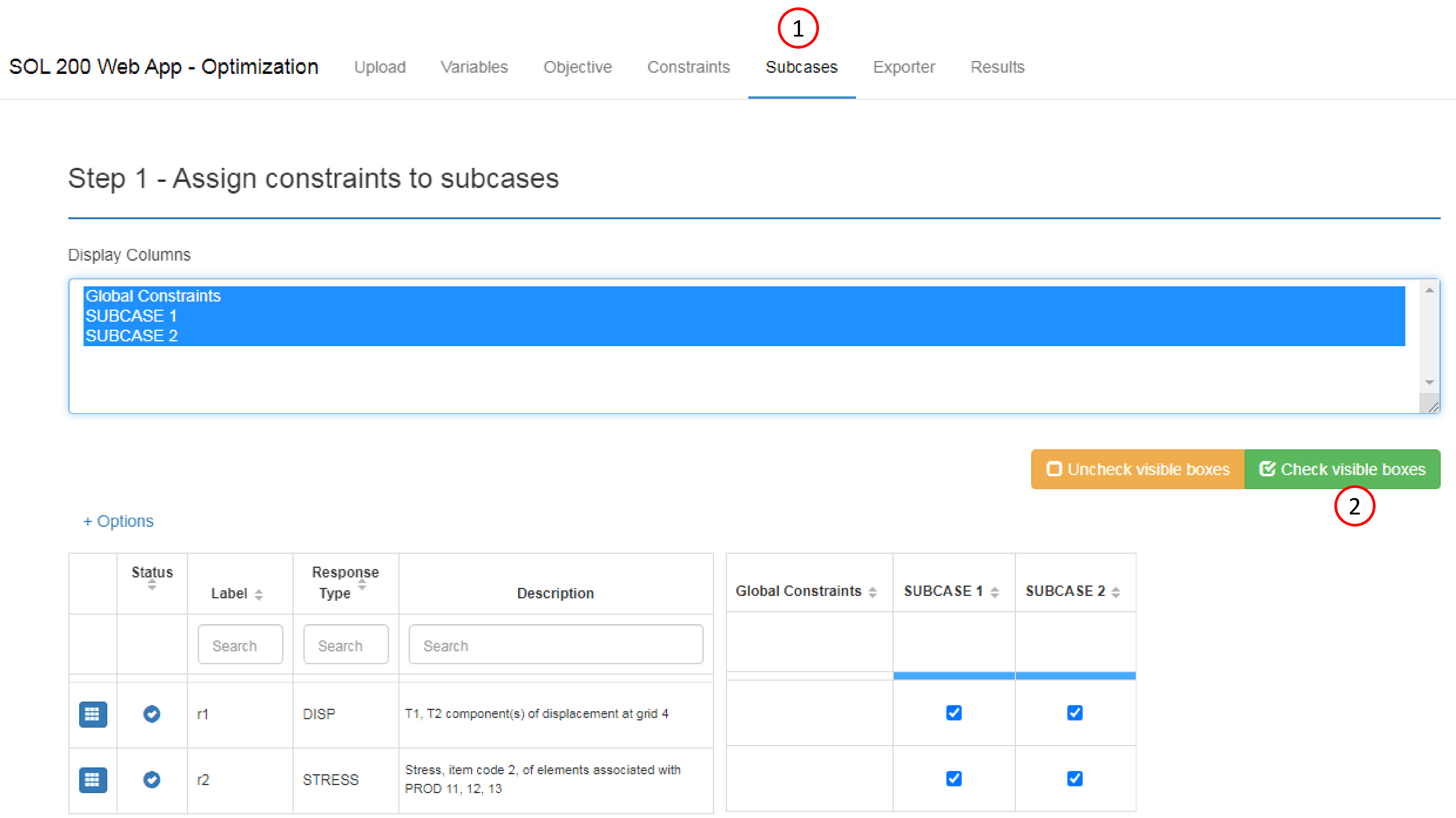
1. Click Constraints
2. In the search box, type ‘s’
3. Select the plus(+) icon for Displacement to create a displacement constraint
4. Select the plus(+) icon for Stress to create a stress constraint
5. Configure the following for r1
   - ATTA: 12 - T1, T2
   - ATTi: 4 (node 4)
   - Lower Allowed Limit: -.2
   - Upper Allowed Limit: .2
6. Configure the following for r2
   - Property Type: PROD
   - ATTA: 2 - Axial Stress
   - ATTi: 11, 12, 13 (PID 11, 12, 13)
   - Lower Allowed Limit: -15000.
   - Upper Allowed Limit: 20000.

- The r1 label is configured as follows: T1, T2 (x, y) component(s) of displacement at grid 4. The label r1 corresponds to 2 displacement responses, so 2 values are constrained.
- The r2 label is configured as follows: The axial stress of elements associated with PROD 11, 12, 13. PROD 11, 12 and 13 have a total of 3 elements associated, so 3 stress quantities are constrained.



Assign Constraints to Load Cases (SUBCASES)

1. Click Subcases
2. Click Check visible boxes



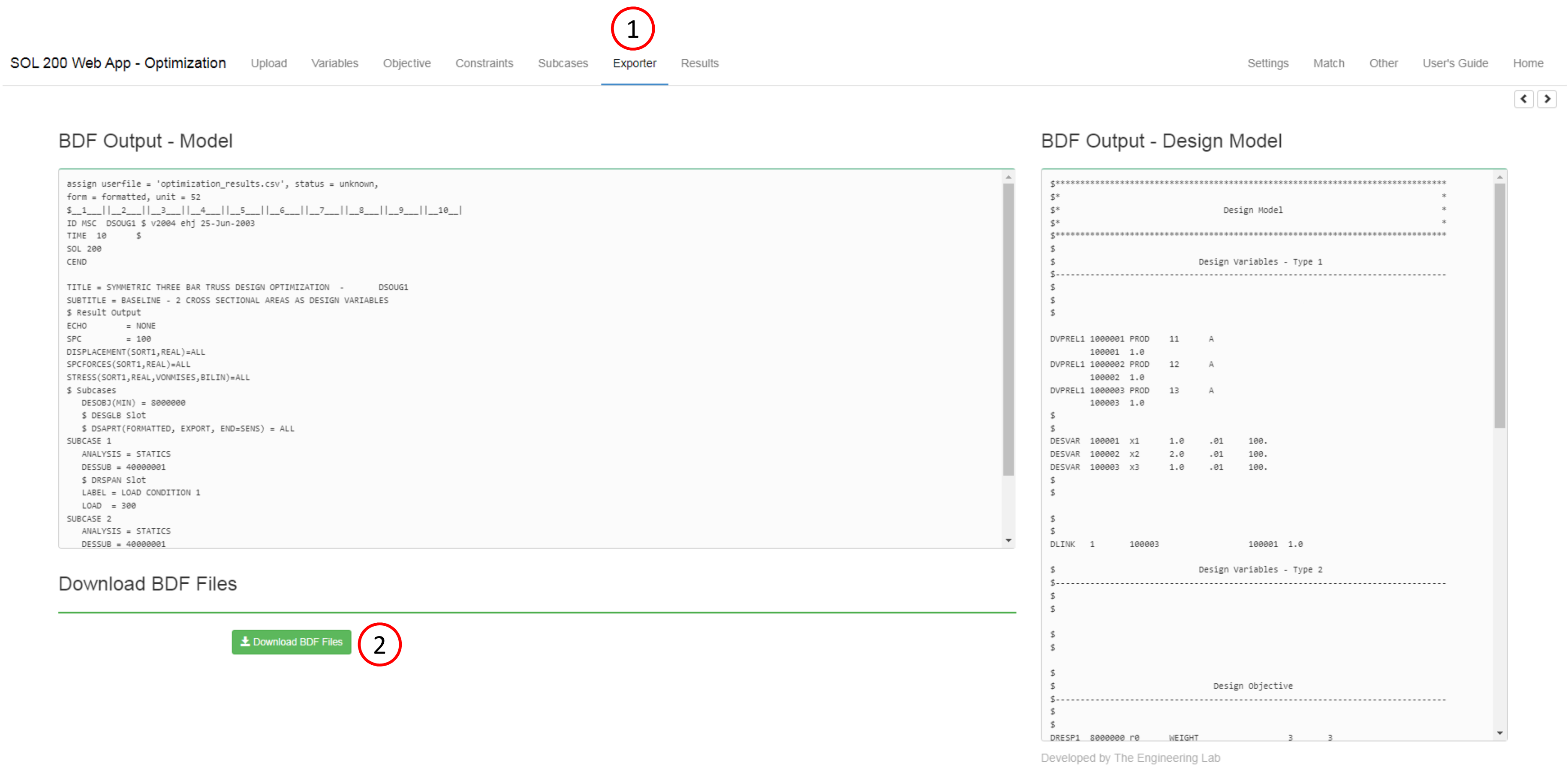
- The r1 and r2 constraints have been assigned to SUBCASE 1 and SUBCASE 2
- When hundreds of SUBCASEs must be configured, the following options expedite the process:

Uncheck visible boxes

Check visible boxes

Export New BDF Files

1. Click on Exporter
2. Click on Download BDF Files

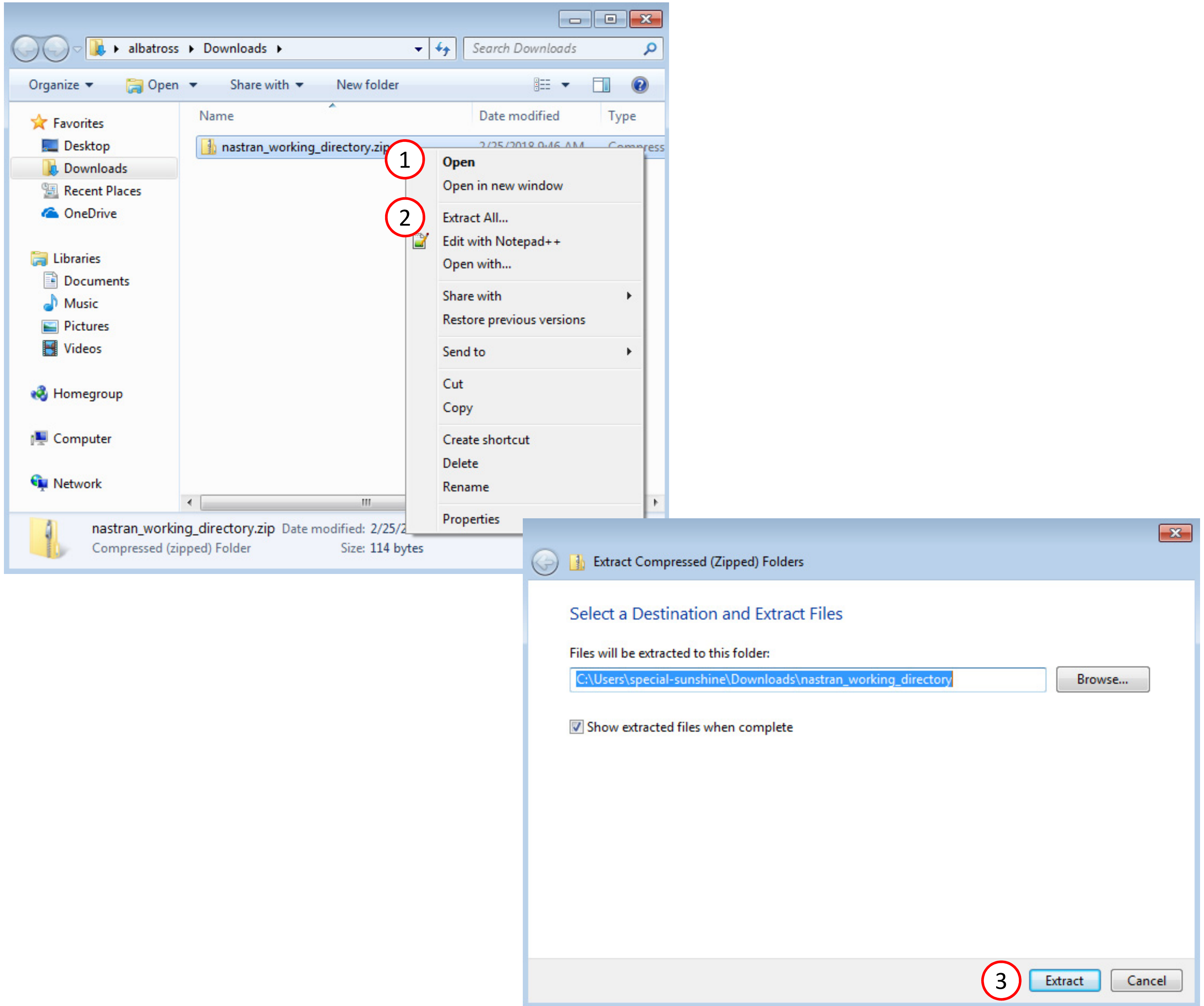


- When the download button is clicked a new file named “nastran_working_directory” is downloaded. If the file already exists in your local folder, the folder name is appended with a number, e.g. “nastran_working_directory (1).zip”

Perform the Optimization with Nastran SOL 200

1. A new .zip file has been downloaded
2. Right click on the file
3. Click Extract All
4. Click Extract on the following window

- Always extract the contents of the ZIP file to a new, empty folder.



Perform the Optimization with Nastran SOL 200

1. Inside of the new folder, double click on Start MSC Nastran
2. Click Open, Run or Allow Access on any subsequent windows
3. MSC Nastran will now start

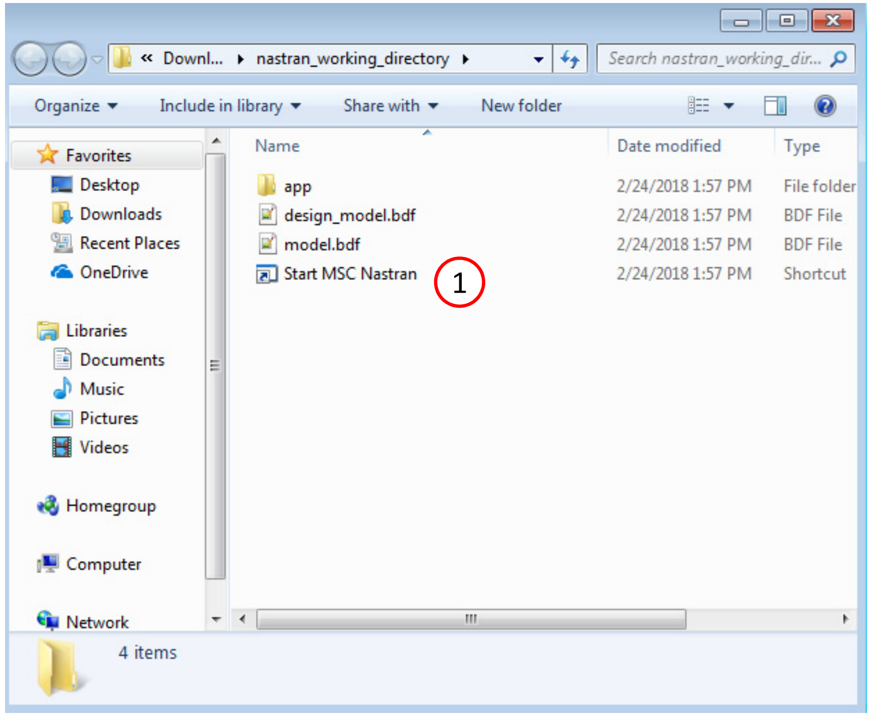
- After a successful optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- One can run the Nastran job on a remote machine as follows:
  1) Copy the BDF files and the INCLUDE files to a remote machine. 2) Run the MSC Nastran job on the remote machine. 3) After completion, copy the BDF, F06, LOG, H5 files to the local machine. 4) Click "Start MSC Nastran" to display the results.

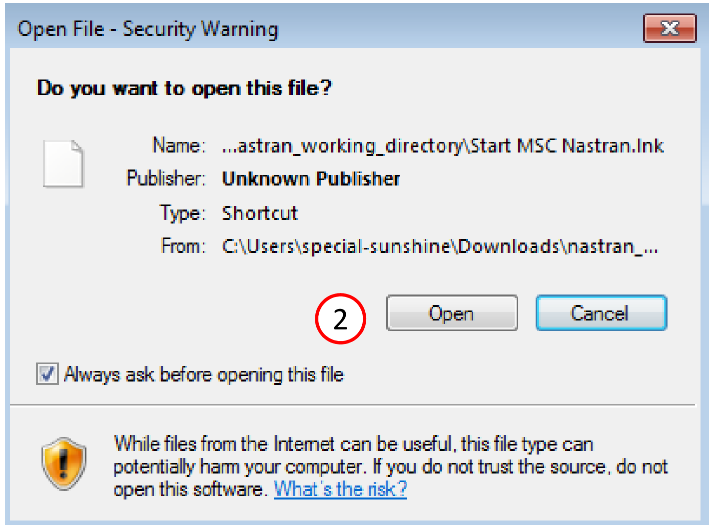
Using Linux?

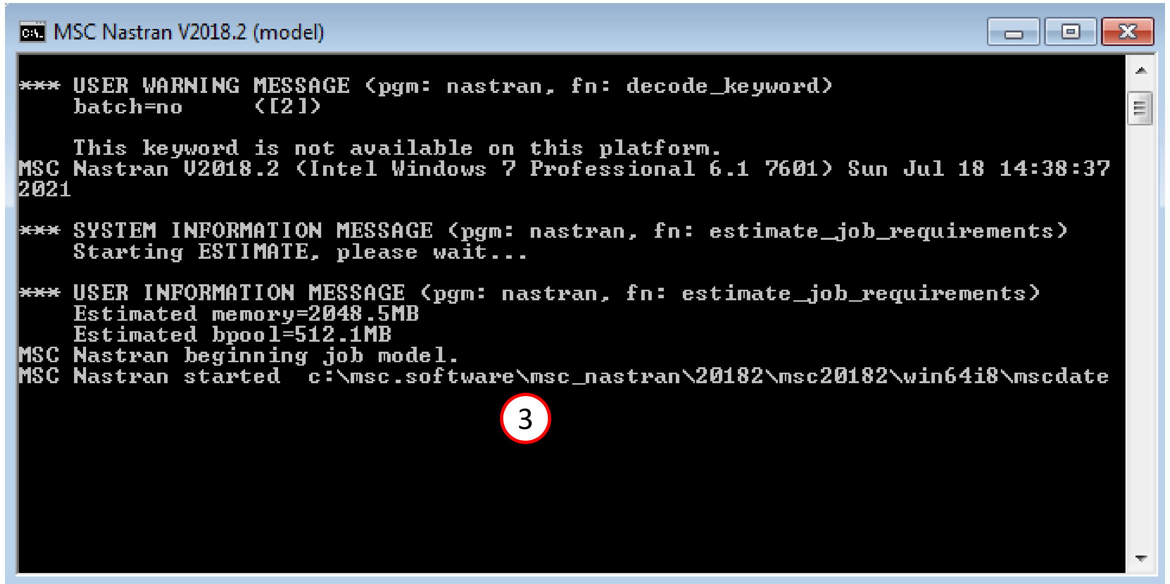
Follow these instructions:
1) Open Terminal
2) Navigate to the nastran_working_directory
   cd ./nastran_working_directory
3) Use this command to start the process
   ./Start_MSC_Nastran.sh

In some instances, execute permission must be granted to the directory. Use this command. This command assumes you are one folder level up.

sudo chmod -R u+x ./nastran_working_directory







Status

1. While MSC Nastran is running, a status page will show the current state of MSC Nastran

- The status of the MSC Nastran job is reported on the Status page. Note that Windows 7 users will experience a delay in the status updates. All other users of Windows 10 and Red Hat Linux will see immediate status updates.

SOL 200 Web App - Status

Python MSC Nastran

Status

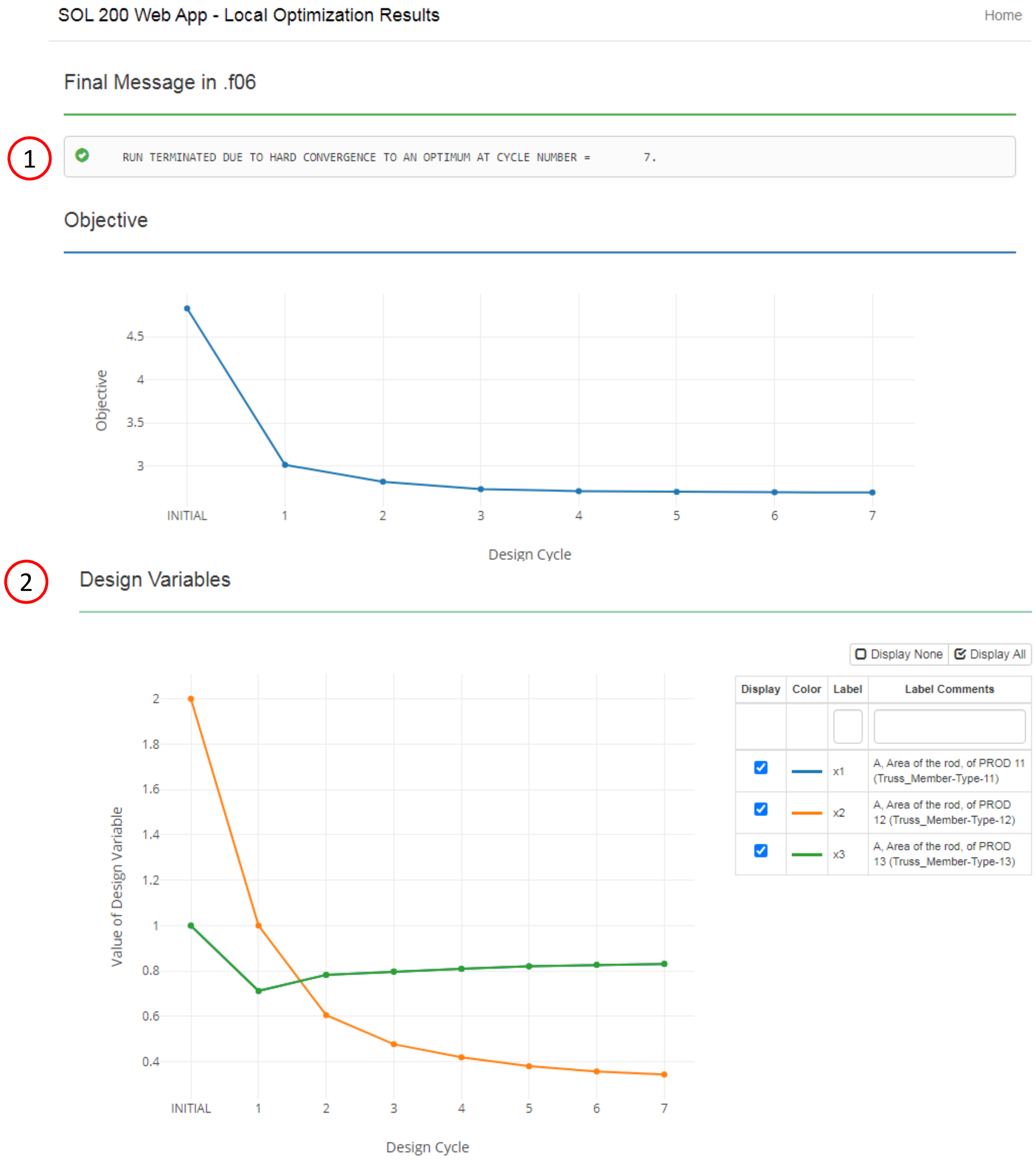
Name	Status of Job	Design Cycle	RUN TERMINATED DUE TO
model.bdf	Running	None	

Review Optimization Results

After MSC Nastran is finished, the results will be automatically uploaded.

1. Ensure the messages shown have green checkmarks. This is indication of success. Any red icons indicate challenges.
2. The final value of objective, normalized constraints (not shown) and design variables can be reviewed.

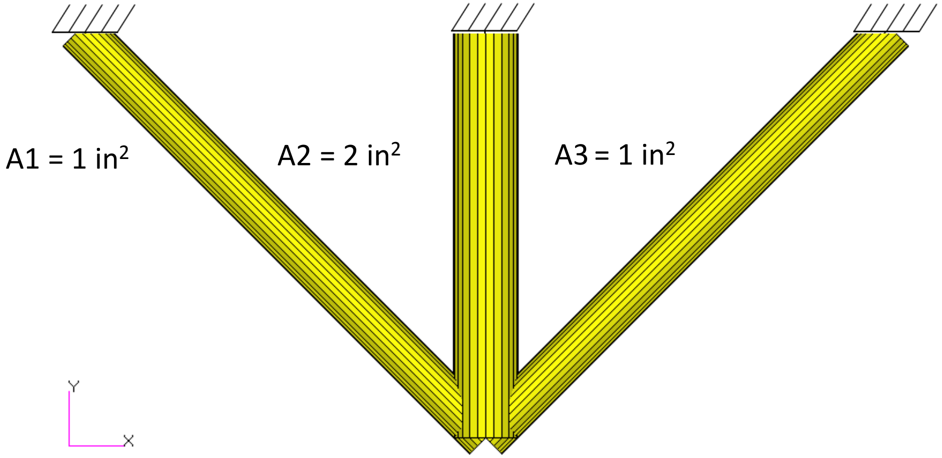
- After an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- This optimization involved 3 design variables, but the plot to the right appears to show only 2 variables. Recall that variable x3 was linked to variable x1, so both x3 and x1 change as one variable. The plot shown does show all 3 variables, but the x3 curve is overlapping the x1 curve.



Results

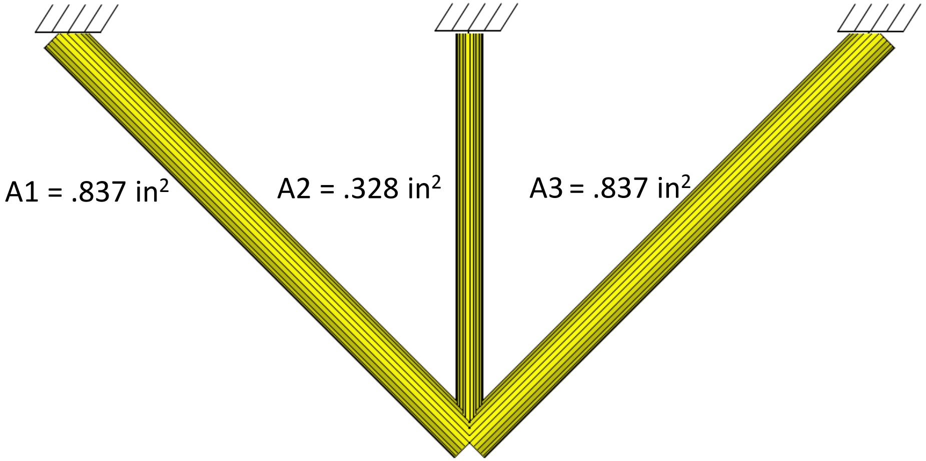
Before Optimization

- Weight: 4.82 lbs.



After Optimization

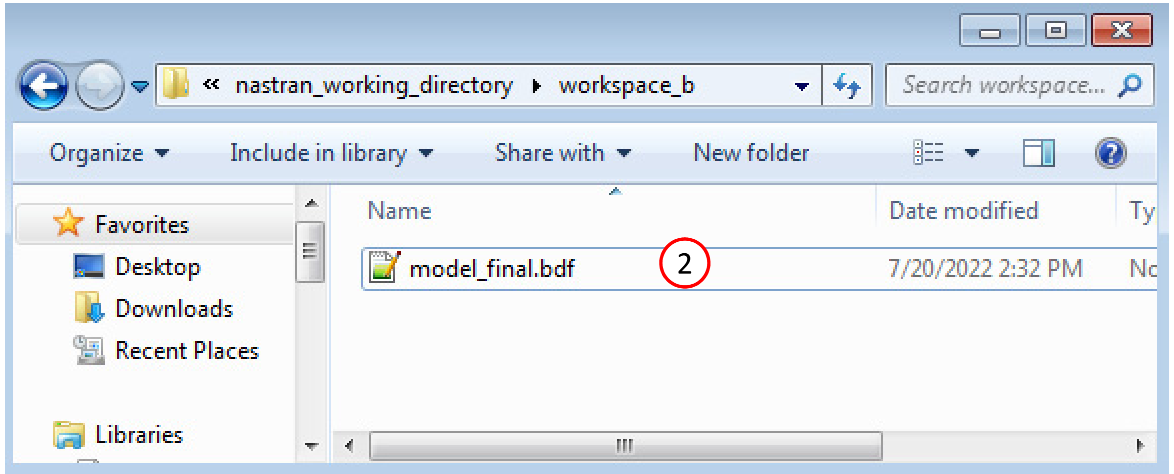
- Weight: 2.70 lbs.



Update the Original Model

1. The original input files, e.g. DAT, BDF, etc., contains the original values for the designed properties. These original values must be updated to use the new and optimized values.
2. A new BDF file has been created in nastran_working_directory/workspace_b/model_final.bdf.
3. The file model_final.bdf is a copy of the original input files but the original values for the designed properties have been updated to use the optimized values.

- If you were using multiple INCLUDE files, model_final.bdf is a combination of all INCLUDE files. The next few slides discuss an alternative method of using the PCH to BDF web app to update the values for the designed properties while preserving separate INCLUDE files.



Original Input Files

1

```
$ PROPERTY DATA
$ Elements and Element Properties for region : Truss_Member-Type-11
PROD     11       1       1.0
$ Elements and Element Properties for region : Truss_Member-Type-12
PROD     12       1       2.0
$ Elements and Element Properties for region : Truss_Member-Type-13
PROD     13       1       1.0
$ Material Record : Aluminum
$ Description of Material : Date: 17-Apr-18           Time: 17:36:49
MAT1     1        1.0E+7          0.33    0.1
$ EXTERNAL LOADS DATA
FORCE    300      4               20000.   0.8     -0.6
FORCE    310      4               20000.  -0.8     -0.6
ENDDATA
```

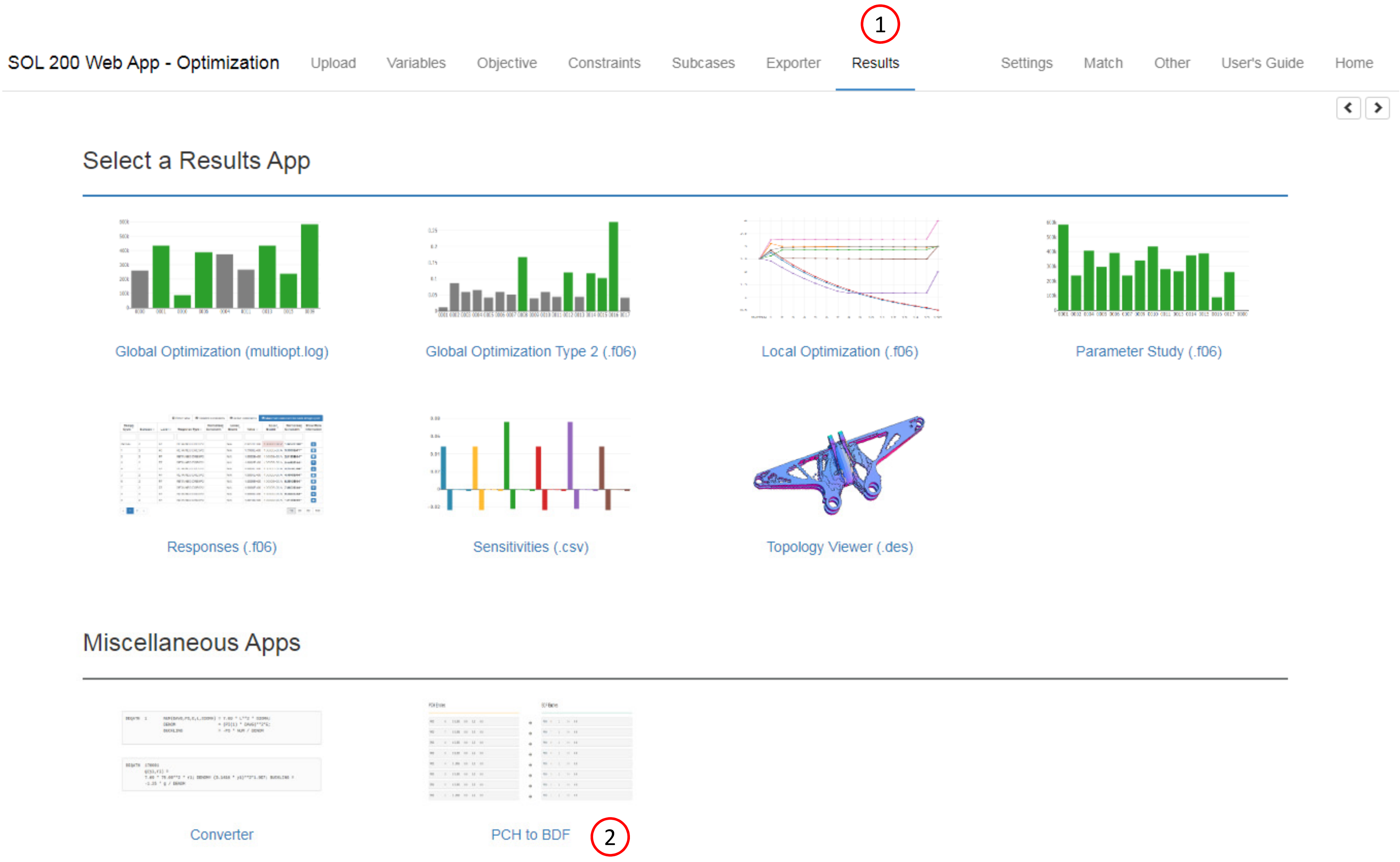
Updated BDF File (model_final.bdf)

3

```
$ PROPERTY DATA
$ Elements and Element Properties for region : Truss_Member-Type-11
PROD            11       1 .837243    0.0     0.0     0.0
$ Elements and Element Properties for region : Truss_Member-Type-12
PROD            12       1 .328299    0.0     0.0     0.0
$ Elements and Element Properties for region : Truss_Member-Type-13
PROD            13       1 .837243    0.0     0.0     0.0
$ Material Record : Aluminum
$ Description of Material : Date: 17-Apr-18           Time: 17:36:49
MAT1     1        1.0E+7          0.33    0.1
$ EXTERNAL LOADS DATA
FORCE    300      4               20000.   0.8     -0.6
FORCE    310      4               20000.  -0.8     -0.6
ENDDATA
```

Update the Original Model

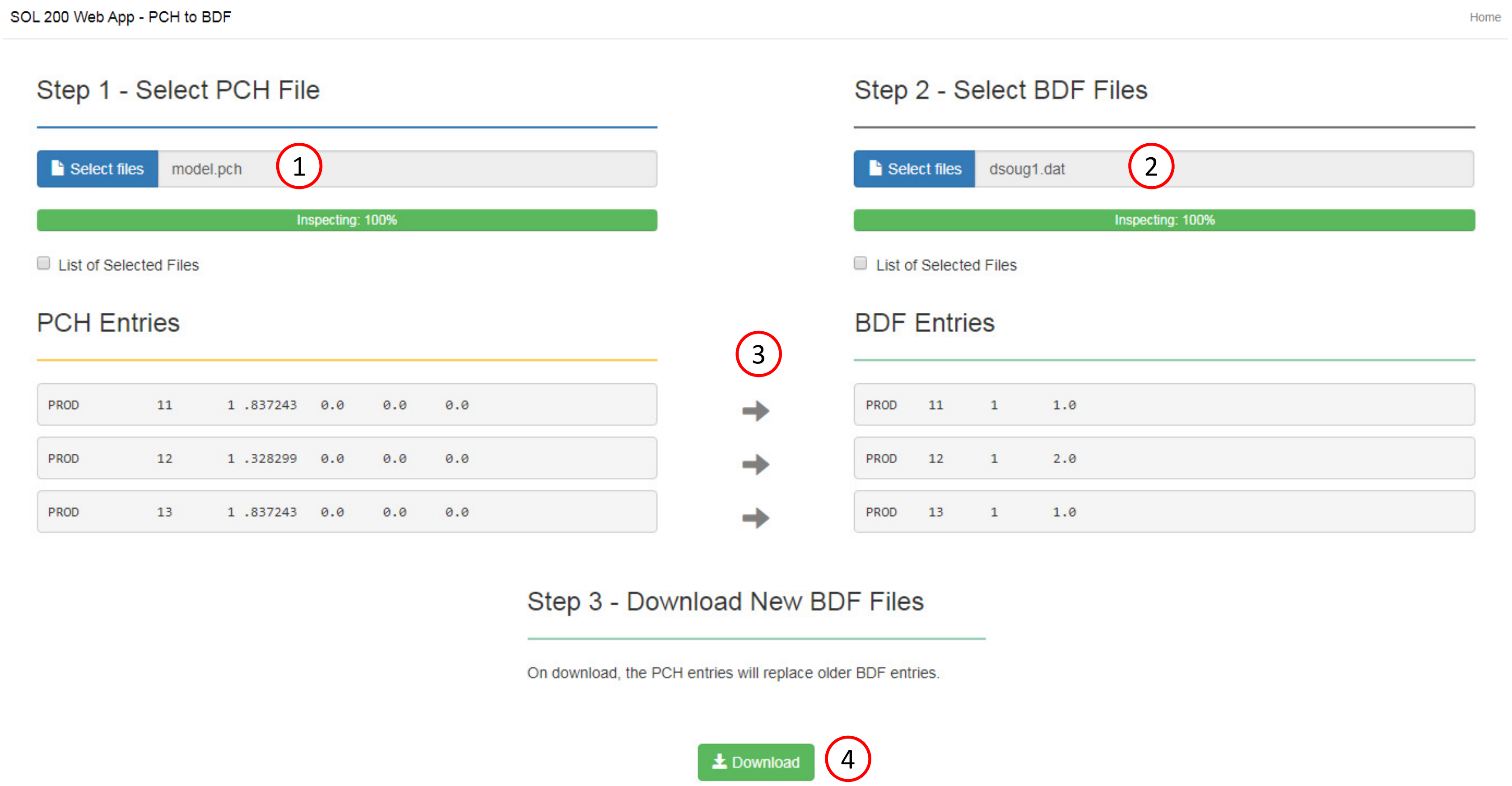
1. Click Results
2. Click PCH to BDF



Update the Original Model

The original .bdf/.dat file has old information about the properties. The properties will be updated.

1. Select the model.pch file
2. Select the original file: dsoug1.dat
3. A summary of updates that will be performed are shown
4. Click Download and a new updated BDF file is downloaded



Update the Original Model

1. Note the entries have been updated with the optimized properties

dsoug1.dat

```
STRESS(SORT1,REAL,VONMISES,BILIN)=ALL

$ Subcases
SUBCASE 1
   LABEL = LOAD CONDITION 1
   LOAD  = 300
SUBCASE 2
   LABEL = LOAD CONDITION 2
   LOAD  = 310
BEGIN BULK
param, post, 1
$
$------------------------------------------------------------------------
$ ANALYSIS MODEL
$------------------------------------------------------------------------
$
$ GRID DATA
$       2       3       4       5       6       7       8       9       10
GRID    1               -10.0   0.0     0.0
GRID    2               0.0     0.0     0.0
GRID    3               10.0    0.0     0.0
GRID    4               0.0     -10.0   0.0
$ SUPPORT DATA
SPC1    100     123456  1       THRU    3
$ ELEMENT DATA
CROD    1       11      1       4
CROD    2       12      2       4
CROD    3       13      3       4
$ PROPERTY DATA
$ Elements and Element Properties for region : Truss_Member-Type-11
PROD    11      1       1.0
$ Elements and Element Properties for region : Truss_Member-Type-12
PROD    12      1       2.0
$ Elements and Element Properties for region : Truss_Member-Type-13
PROD    13      1       1.0
$ Material Record : Aluminum
$ Description of Material : Date: 17-Apr-18           Time: 17:36:49
MAT1    1       1.0E+7          0.33    0.1
$ EXTERNAL LOADS DATA
FORCE   300     4               20000.  0.8     -0.6
FORCE   310     4               20000.  -0.8    -0.6
ENDDATA
```

Original BDF/DAT File

dsoug1.dat

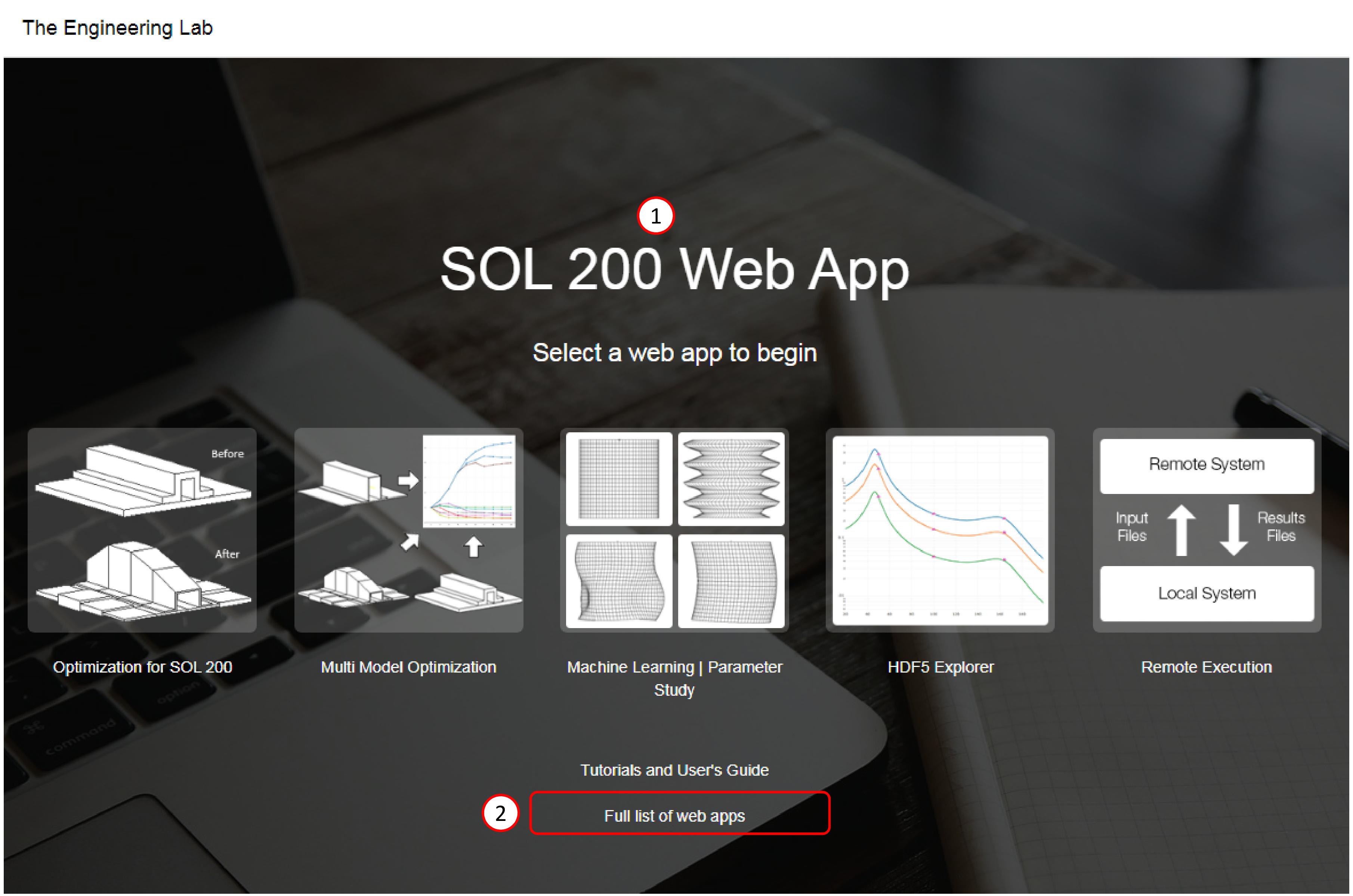
```
STRESS(SORT1,REAL,VONMISES,BILIN)=ALL

$ Subcases
SUBCASE 1
   LABEL = LOAD CONDITION 1
   LOAD  = 300
SUBCASE 2
   LABEL = LOAD CONDITION 2
   LOAD  = 310
BEGIN BULK
param, post, 1
$
$------------------------------------------------------------------------
$ ANALYSIS MODEL
$------------------------------------------------------------------------
$
$ GRID DATA
$       2       3       4       5       6       7       8       9       10
GRID    1               -10.0   0.0     0.0
GRID    2               0.0     0.0     0.0
GRID    3               10.0    0.0     0.0
GRID    4               0.0     -10.0   0.0
$ SUPPORT DATA
SPC1    100     123456  1       THRU    3
$ ELEMENT DATA
CROD    1       11      1       4
CROD    2       12      2       4
CROD    3       13      3       4
$ PROPERTY DATA
$ Elements and Element Properties for region : Truss_Member-Type-11
PROD          11       1 .837243   0.0     0.0     0.0
$ Elements and Element Properties for region : Truss_Member-Type-12
PROD          12       1 .328299   0.0     0.0     0.0
$ Elements and Element Properties for region : Truss_Member-Type-13
PROD          13       1 .837243   0.0     0.0     0.0
$ Material Record : Aluminum
$ Description of Material : Date: 17-Apr-18           Time: 17:36:49
MAT1    1       1.0E+7          0.33    0.1
$ EXTERNAL LOADS DATA
FORCE   300     4               20000.  0.8     -0.6
FORCE   310     4               20000.  -0.8    -0.6
ENDDATA
```

Downloaded BDF/DAT File

Beams Viewer

1. Navigate to the homepage
2. Click on the indicated link

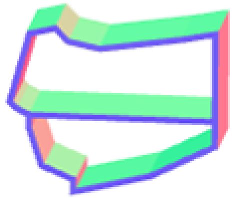


Beams Viewer

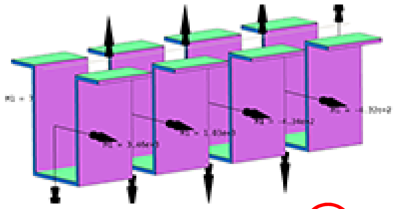
1. Click the icon titled Beams Viewer to open the Beams Viewer

SOL 200 Web App - List of Web Apps

Beams



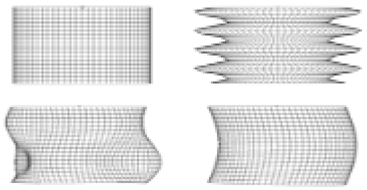
PBMSECT



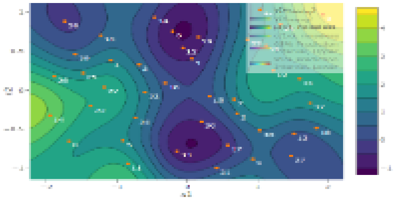
Beams Viewer

1

Machine Learning



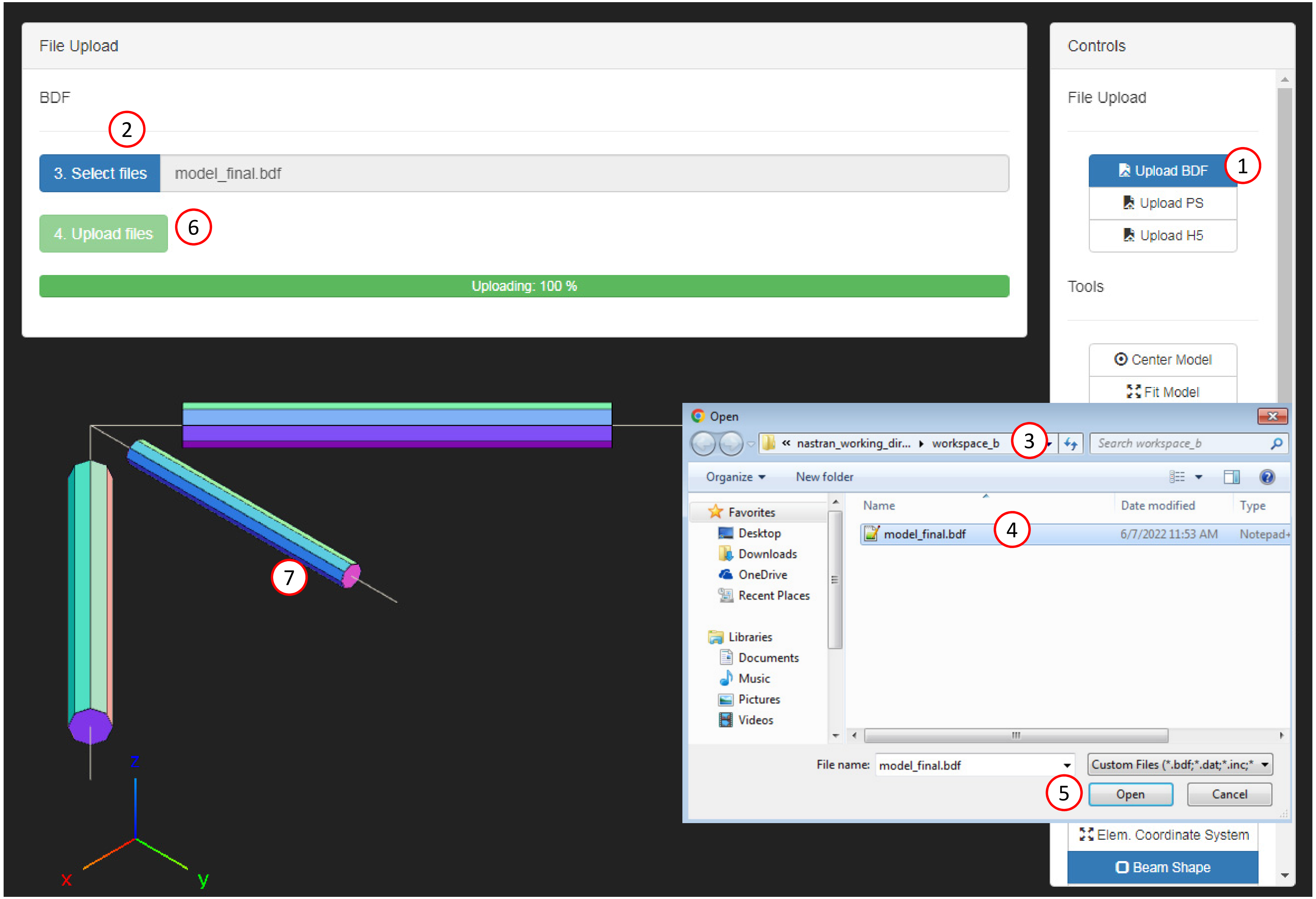
Machine Learning



Prediction Analysis

Beams Viewer

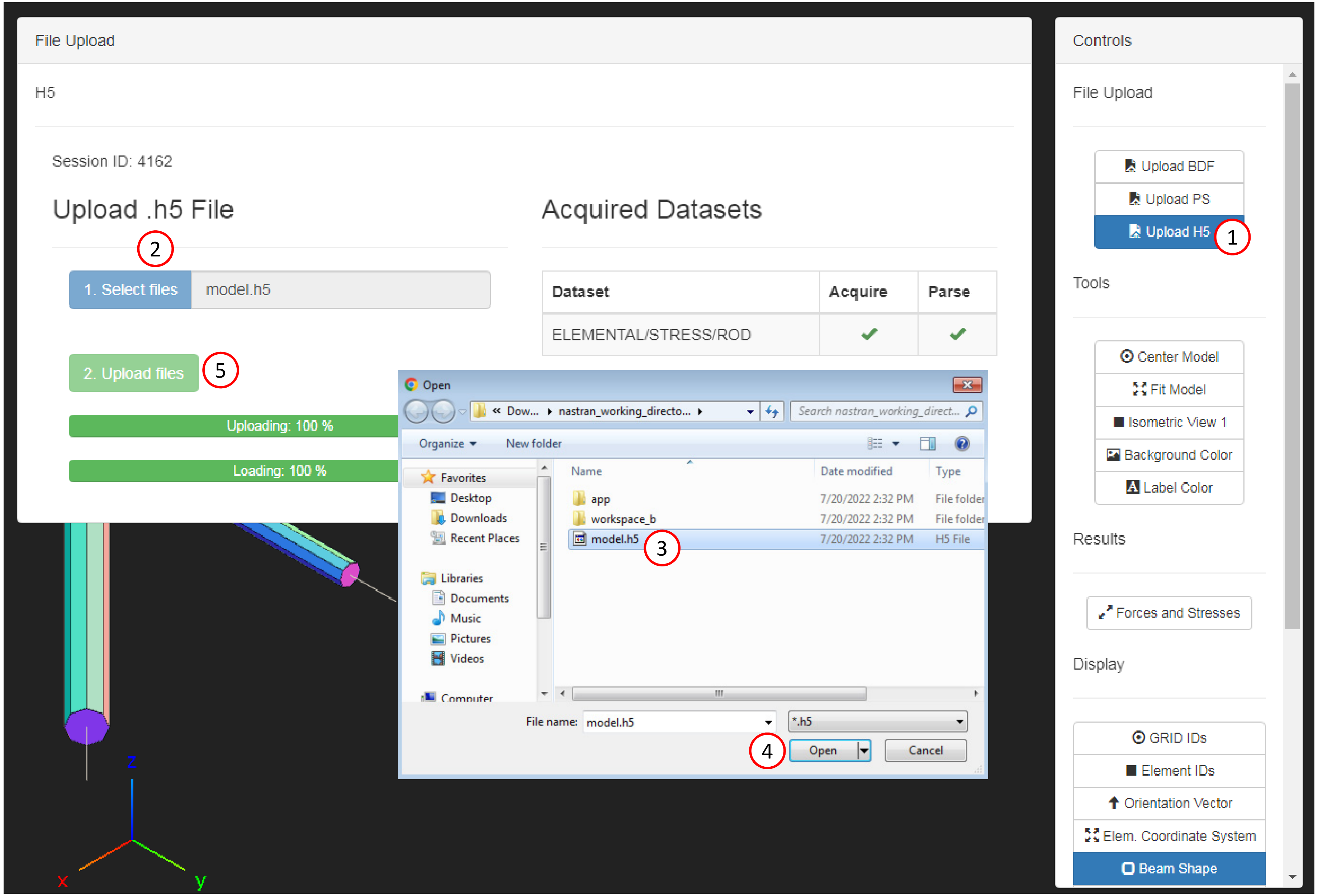
1. Click Upload BDF
2. Click Select files
3. Navigate to the directory workspace_b
4. Select model_final.bdf
5. Click Open
6. Click Upload files
7. The MSC Nastran model has been uploaded to the Beams Viewer



Upload the H5 File

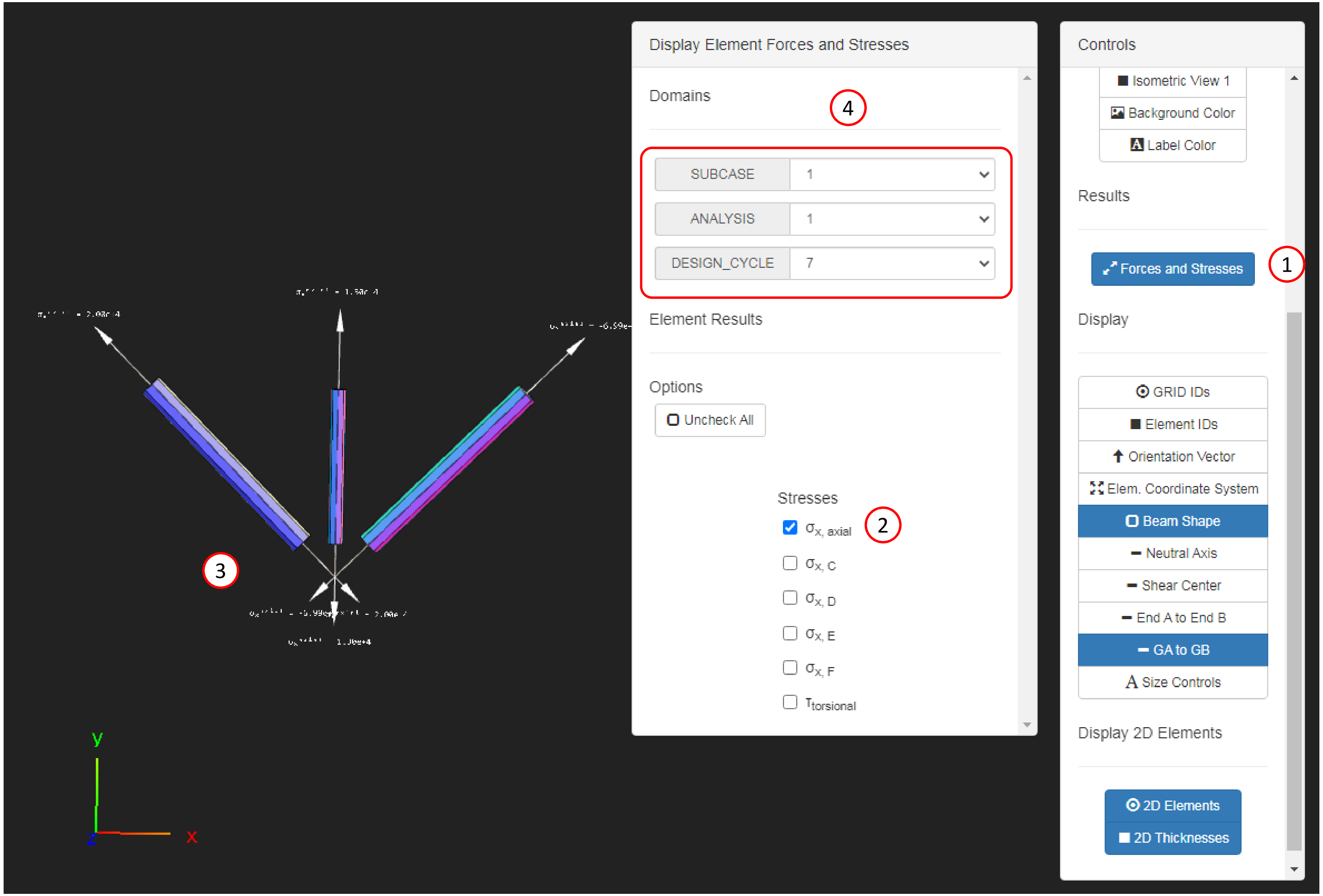
If an H5 file was created, via MDLPRM,HDF5 or HDF5OUT, the H5 file may be uploaded to the Beams Viewer.

1. Click Upload H5
2. Click Select files
3. Select file nastran_working_directory/model.h5
4. Click Open
5. Click Upload files



Display Internal Element Moments

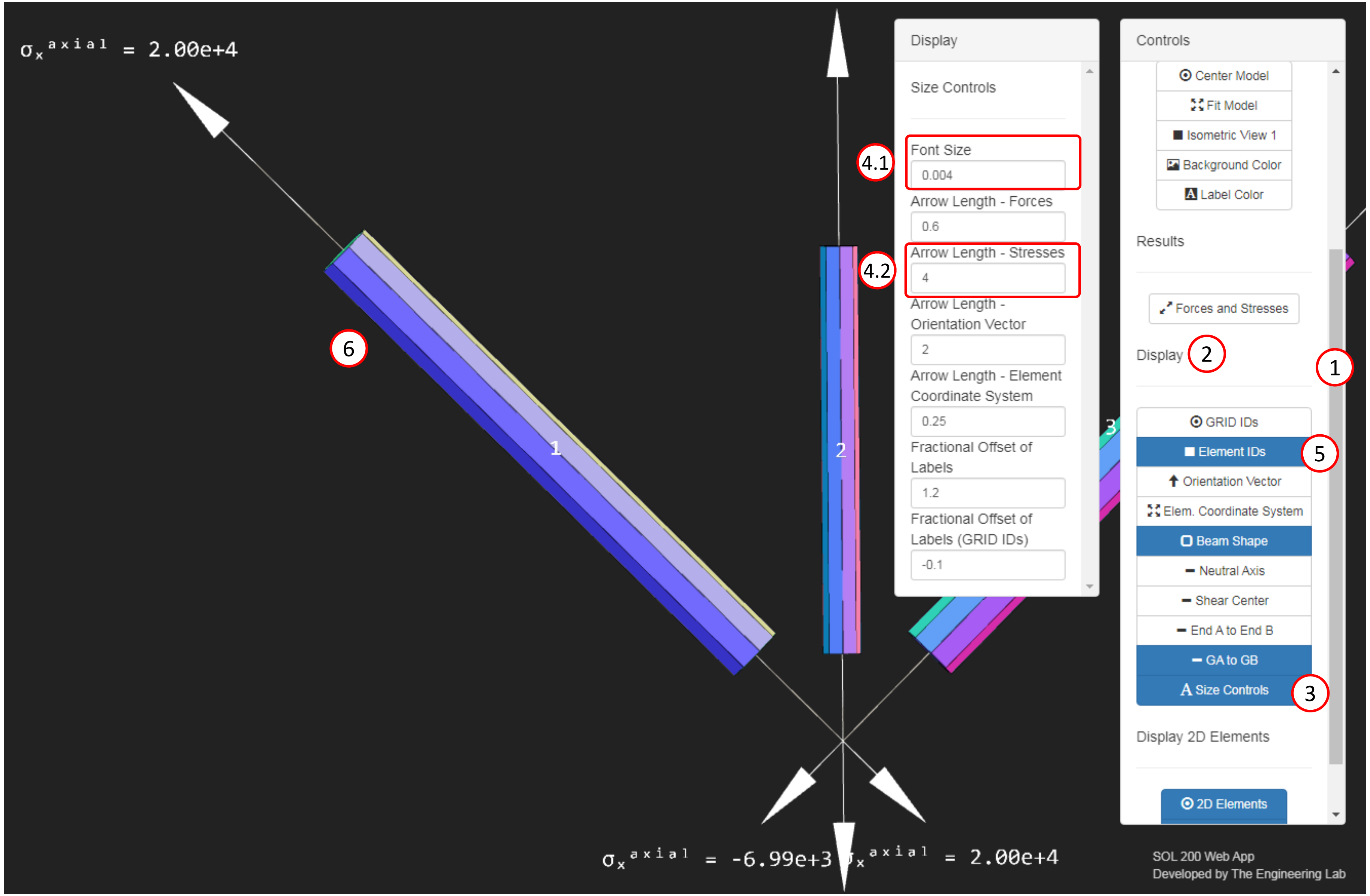
1. Click Forces and Stresses
2. Mark the check boxes for $\sigma_{x,\ axial}$, which corresponds to the axial stress
3. Axial stress arrows are now displayed on the 1D elements
4. Select the following:
    1. SUBCASE: 1
    2. ANALYSIS: 1
    3. DESIGN_CYCLE: 7



Adjust the Size of Labels

1. Use the vertical scroll bar
2. Locate the Display section
3. Click Size Controls
4. In the new panel, configure the following values:
    1. Font Size: 0.004
    2. Arrow Length – Stresses: 4
5. Click Element IDs
6. Rotate the model until element 1 is visible

- Recall the design constraint for axial stress had an upper bound of 20,000 units of stress. The optimizer has varied the cross section area of the 1D elements such that the design constraint is not violated. Inspection of the final axial stress shows the axial stress is no greater than 20,000 and is equal to 20,000. This optimization has been a success.



End of Tutorial

Appendix

Appendix Contents

- Frequently Asked Questions
  - What does this line mean, **INCLUDE './design_model.bdf'** ?
  - Can design_model.bdf be renamed?

Frequently Asked Questions

Question:

- What does this line mean, **INCLUDE './design_model.bdf'** ?

```
   LOAD   = 310
BEGIN BULK
INCLUDE './design_model.bdf'

param, post, 1
$
$
```

Answer:

- This is contained in the file model.bdf.
- When you perform the optimization, you first select model.bdf.
- Nastran will read each line in the model.bdf file. Once Nastran reads the line with INCLUDE, Nastran will also take all the text contained in design_model.bdf and make it part of the optimization.

Question:

- Can design_model.bdf be renamed?

Answer:

- Yes. 1) Rename the file. Before: design_model.bdf After: renamed_file.bdf . 2) Update the INCLUDE statement. Before: INCLUDE './design_model.bdf' After: INCLUDE './renamed_file.bdf'