

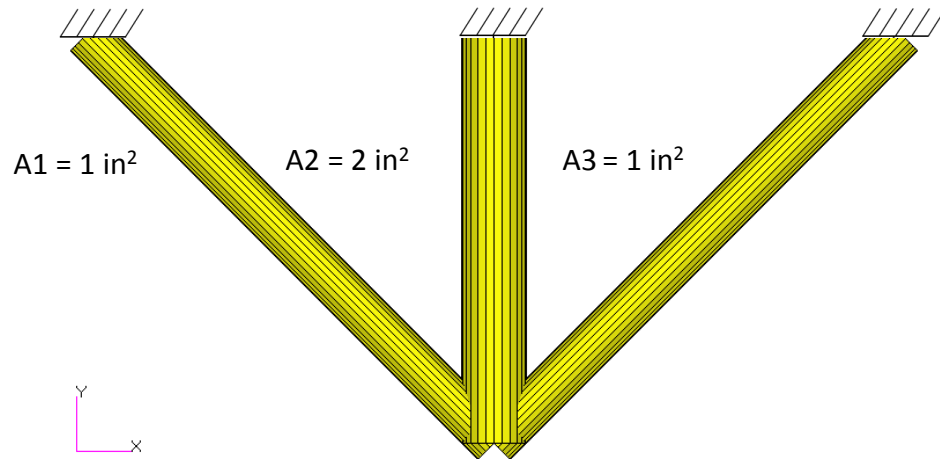
Workshop - Structural Optimization of a 3 Bar Truss

AN MSC NASTRAN SOL 200 TUTORIAL

Goal: Use Nastran SOL 200 Optimization

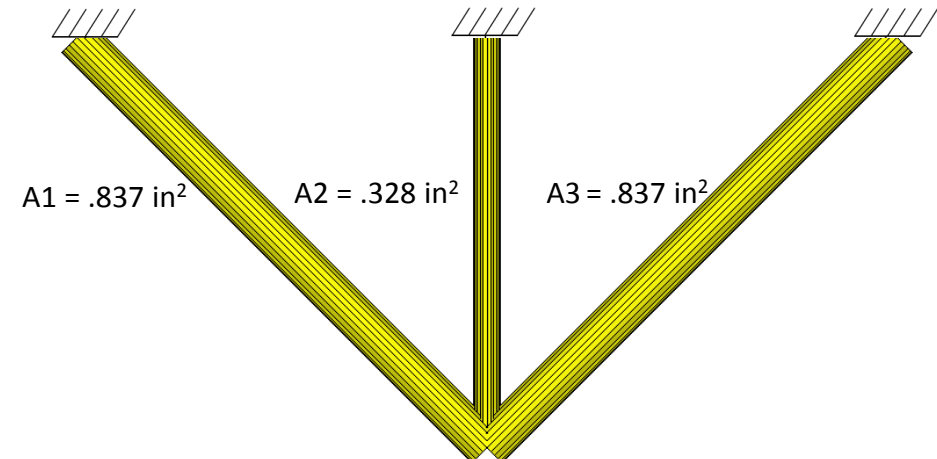
Before Optimization

- 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.

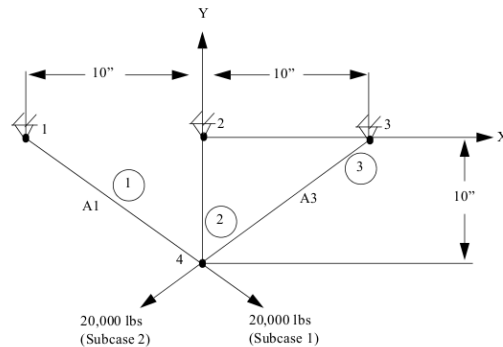
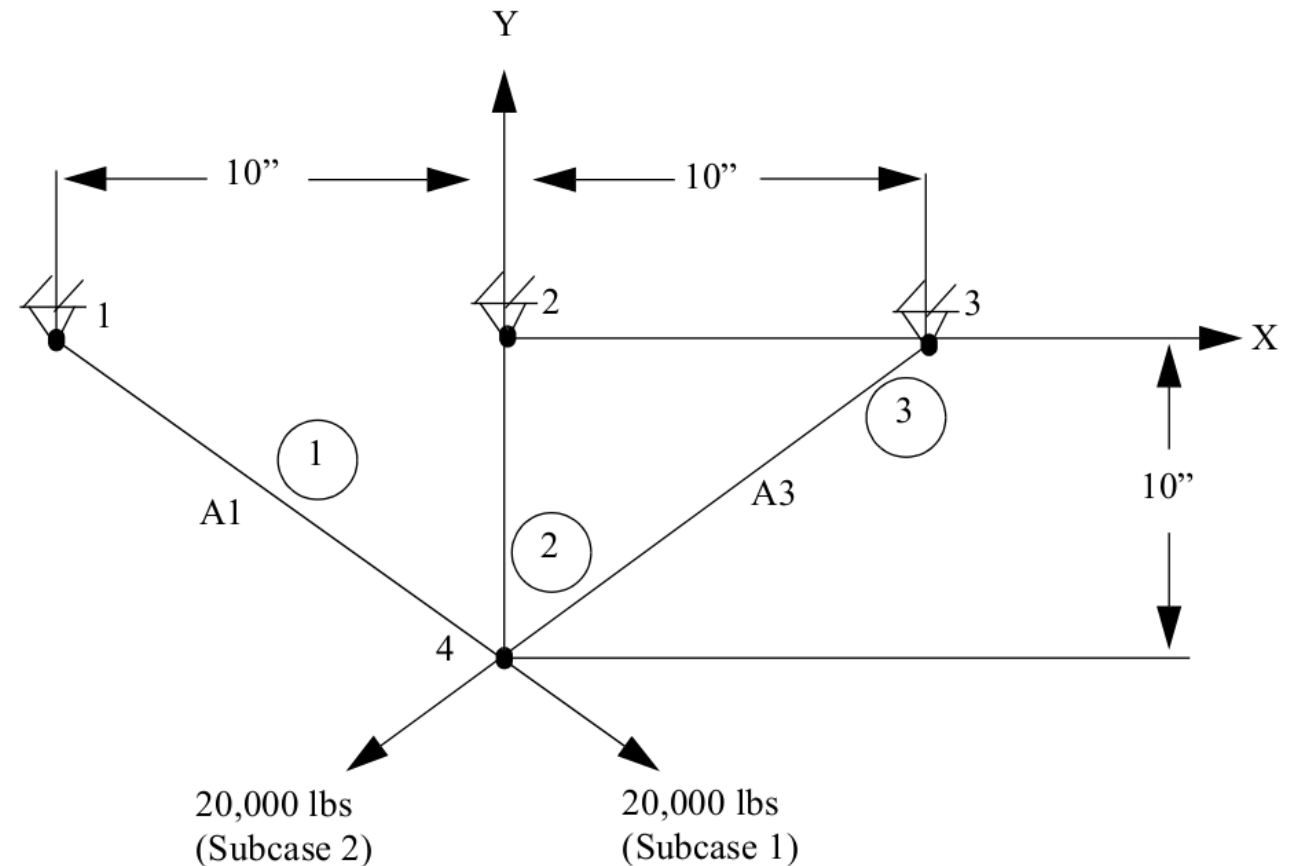


Figure 8-1 Three-Bar Truss

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 static 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

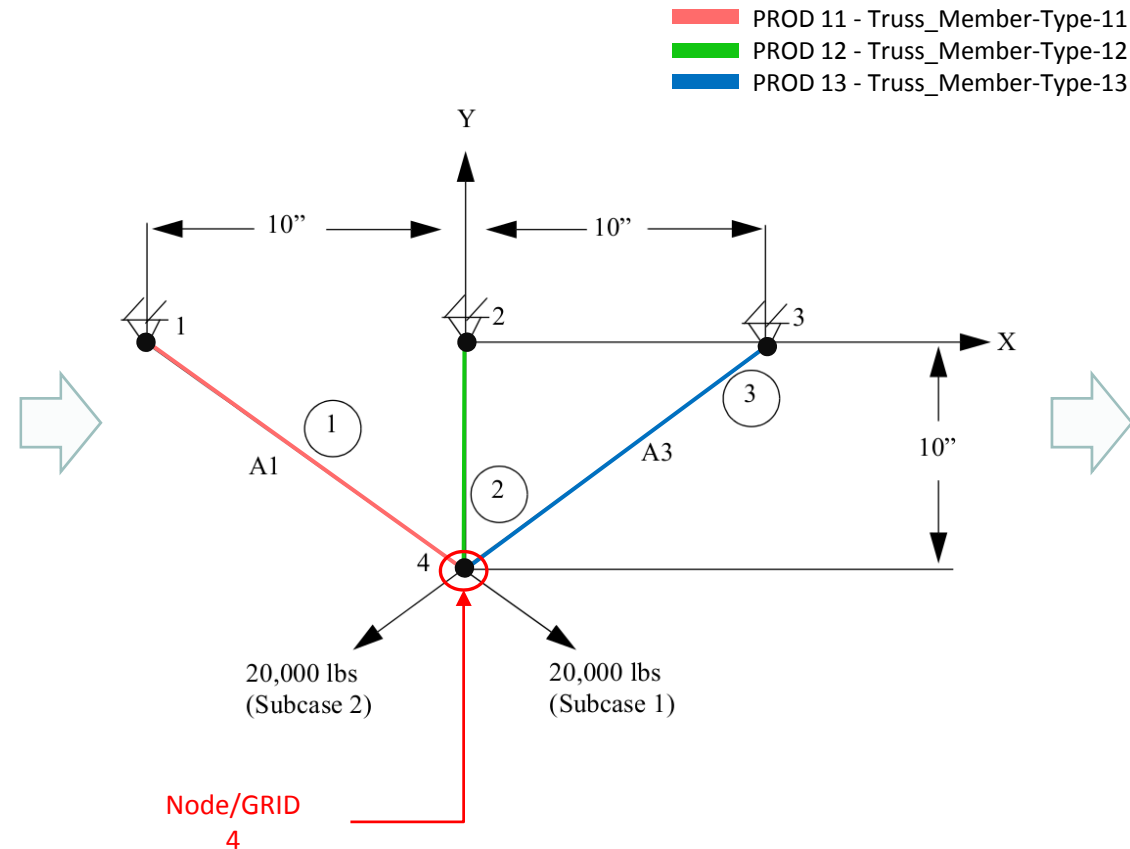
Design Variables

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

$$-15000 < r1 < 20000$$

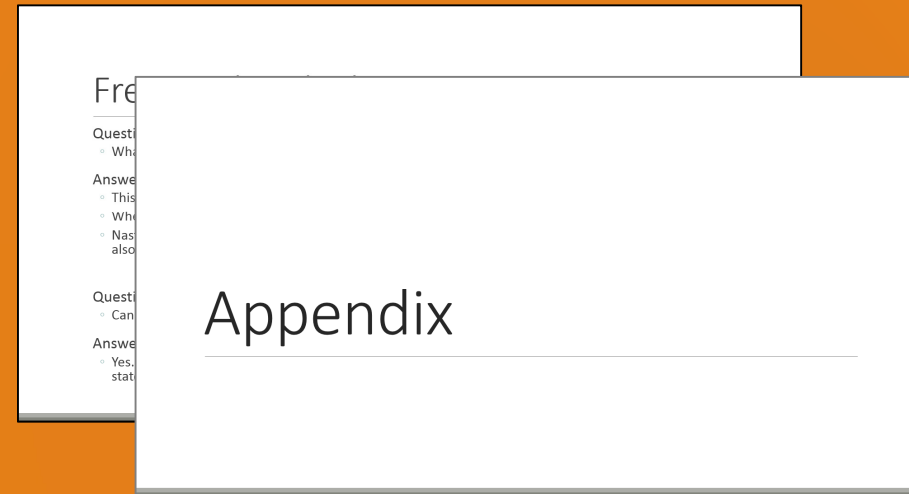
r2: x and y component of displacement for
node 4

$$-.2 < r2 < .2$$

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

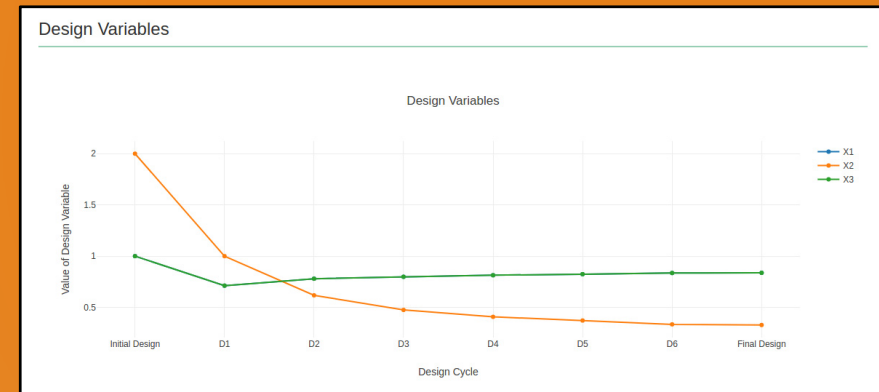
Tutorial

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.

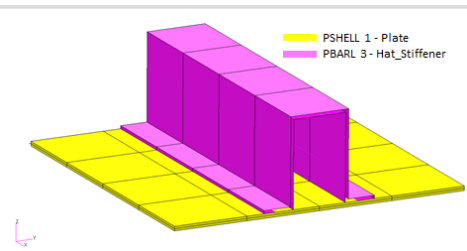


SOL 200 Web App Capabilities

Benefits

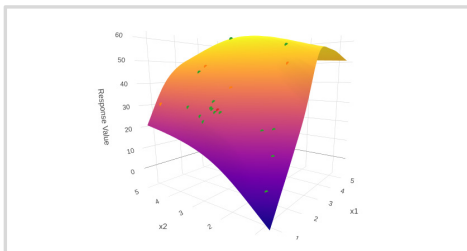
- 200+ error validations (real time)
- Web browser accessible
- Automated creation of entries (real time)
- Automatic post-processing
- 76 tutorials

Capabilities



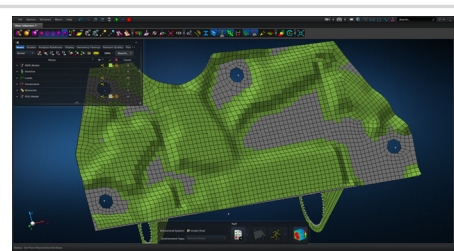
Web Apps for SOL 200

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



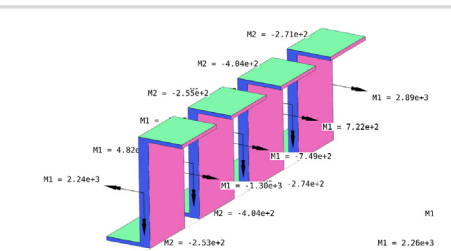
Machine Learning Web App

Bayesian Optimization for nonlinear response optimization (SOL 400)



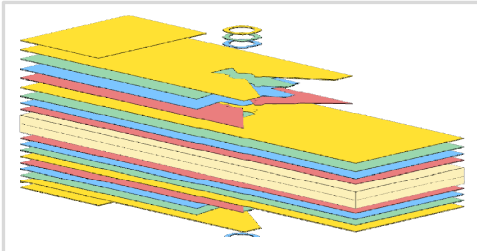
MSC Apex Post Processing Support

View the newly optimized model after an optimization



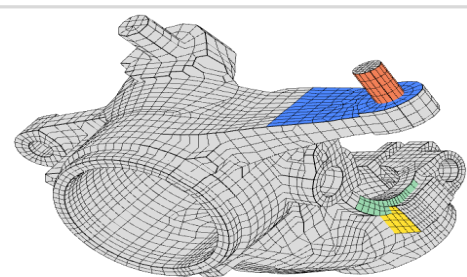
Beams Viewer Web App

Post process 1D element forces, including shear forces, moments, torque and axial forces



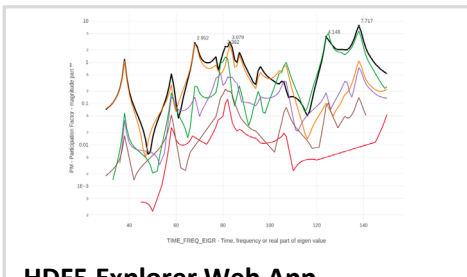
Ply Shape Optimization Web App

Spread plies optimally and generate new PCOMPG entries



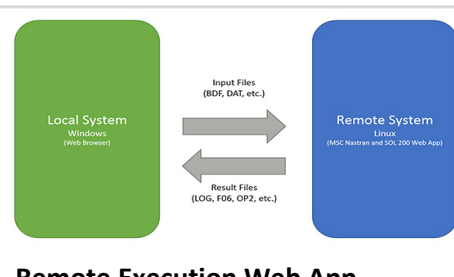
Shape Optimization Web App

Use a web application to configure and perform shape optimization.



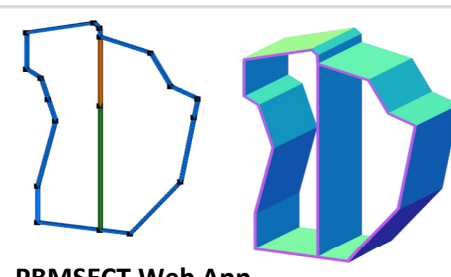
HDF5 Explorer Web App

Create XY plots using data from the H5 file



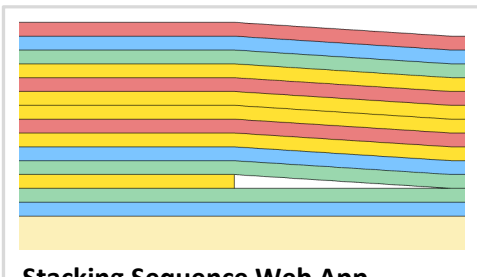
Remote Execution Web App

Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



PBMSECT Web App

Generate PBMSECT and PBRSECT entries graphically



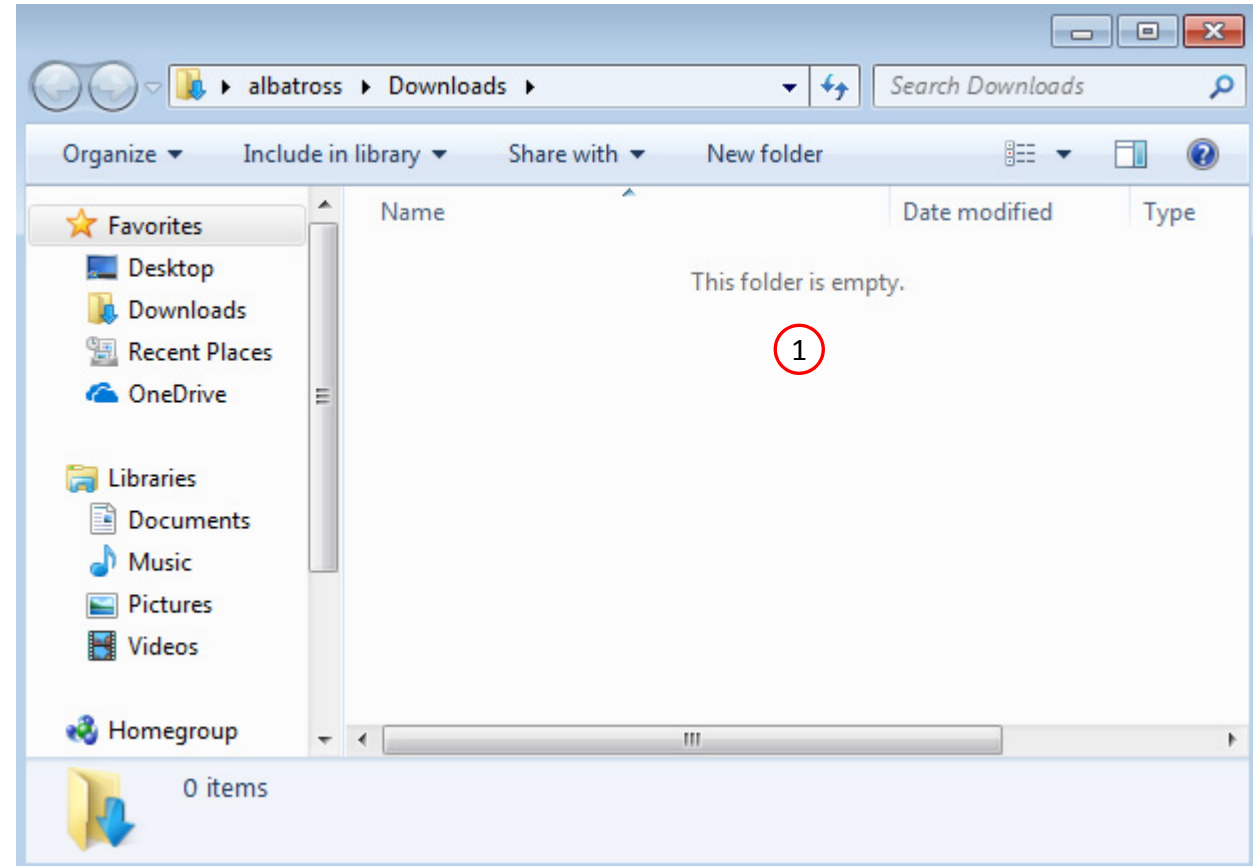
Stacking Sequence Web App

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 screenshot displays the SOL 200 Web App interface. At the top, it says "SOL 200 Web App" and "Select a web app to begin". Below this, there are five main categories of web apps, each with a representative image and a label:

- Optimization for SOL 200**: Shows a 3D model of a mechanical part with "Before" and "After" states.
- Multi Model Optimization**: Shows a 3D model of a mechanical part with a graph of multiple optimization curves.
- Machine Learning | Parameter Study**: Shows four small plots representing different machine learning or parameter study results.
- HDF5 Explorer**: Shows a line graph with multiple data series.
- Remote Execution**: Shows a diagram of data flow between a "Remote System" and a "Local System", with "Input Files" and "Results Files" labels.

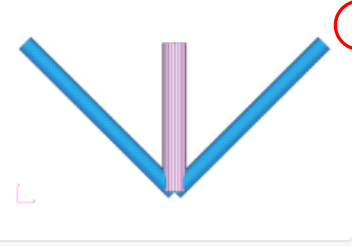
At the bottom center, there is a red-bordered button labeled "Tutorials and User's Guide" with a circled "1" next to it. Below this button, the text "Full list of web apps" is visible.

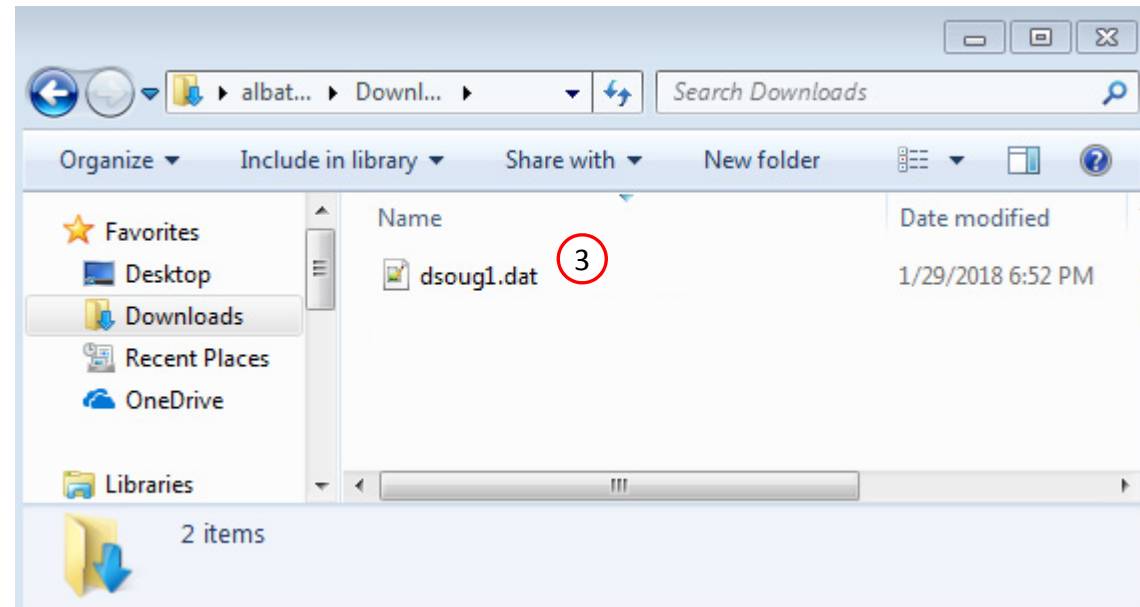
Obtain Starting Files

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.

Size Optimization Tutorials

| | Title and Description | Lecture Notes |
|--|--|----------------------|
|  | <p>1 Structural Optimization of a 3 Bar Truss - MSC Nastran Optimization</p> <p>A truss structure is optimized with MSC Nastran. The design variables are the cross sectional areas of the rod elements. The objective is to minimize the weight of the structure while ensuring the stress and displacements are within specified constraints.</p> <p>Starting BDF Files: Link 2 Solution BDF Files: Link</p> | Link |



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 screenshot displays the SOL 200 Web App interface. At the top, it says "SOL 200 Web App" and "Select a web app to begin". Below this, there are five main categories of web apps, each with a representative image:

- Optimization for SOL 200**: Shows a 3D model of a mechanical part with "Before" and "After" states. A red circle with the number "1" is placed over this icon.
- Multi Model Optimization**: Shows a 3D model and a line graph.
- Machine Learning | Parameter Study**: Shows four small plots representing different data sets or models.
- HDF5 Explorer**: Shows a line graph with multiple data series.
- Remote Execution**: Shows a diagram of data flow between a "Remote System" and a "Local System", with "Input Files" going up and "Results Files" going down.

At the bottom of the interface, there are two links: "Tutorials and User's Guide" and "Full list of web apps".

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.

Step 1 - Upload .BDF Files

1. Select files dsoug1.dat

Inspecting: 100%

2. Upload files

Uploading: 100 %

☐ List of Selected Files






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 ⚙ | |
|----------------|---|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|--------|
| | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | |
| 1 { |  | 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 |

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 |
|--|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|
| | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="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, |

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 ▾ |
|--|-------------------------------------|-------------------------------------|---|-------------------------------------|
| | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="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.

The right of the expression can also have additional variables. For example, $x_2 = 1.5 + x_1 * 1.0 + y_2 * -3.5 + \dots$

$$x_3 = 0.0 + x_1 * 1.0$$

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.

Step 1 - Select an objective

Select an analysis type

SOL 103 - Normal Modes

Select a response



| | Response Description ▾ | Response Type ▾ |
|---|-------------------------------------|-------------------------------------|
| | <input type="text" value="Search"/> | <input type="text" value="Search"/> |
| 2  | Weight | WEIGHT |
|  | Volume | VOLUME |
|  | Eigenvalue | EIGN |
|  | Frequency | FREQ |
|  | Displacement | DISP |

« 1 2 3 »

5 10 20 30 40 50

Step 2 - Adjust objective

+ Options

| | Label | Status | Response Type | Maximize or Minimize | Property Type | ATTA | ATTB | ATTi |
|---|-------|---|---------------|----------------------|---------------|------|------|------|
|  | r0 |  | WEIGHT | MIN ▾ | 3 | 3 ▾ | 3 ▾ | |

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.

1

Step 1 - Select constraints

Select an analysis type

SOL 101 - Statics

Select a response

| | | Response Description ▾ | Response Type ▾ |
|---|---|---|-----------------|
| | | s 2 | Search |
| 3 | + | Displacement | DISP |
| | + | Strain | STRAIN |
| | + | Element Strain Energy | ESE |
| 4 | + | Stress | STRESS |
| | + | Fatigue, pseudo-static fatigue analysis | FATIGUE |

« 1 2 3 4 »

5 10 20 30 40 50

Step 2 - Adjust constraints

+ Options

| | Label ▾ | Status ▾ | Response Type ▾ | Property Type ▾ | ATTA ▾ | ATTB ▾ | ATTi ▾ | Lower Allowed Limit | Upper Allowed Limit |
|---|---------|----------|-----------------|-----------------|------------------|--------|------------|---------------------|---------------------|
| | St | Seal | Search | Search | Search | Search | Search | Search | Search |
| ✖ | r1 | ✓ | DISP 5 | | 12 - T1, T2 | | 4 | -.2 | .2 |
| ✖ | r2 | ✓ | STRESS 6 | PROD | 2 - Axial stress | | 11, 12, 13 | -15000. | 20000. |

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

Step 1 - Assign constraints to subcases

Display Columns

Global Constraints
SUBCASE 1
SUBCASE 2

☐ Uncheck visible boxes

☒ Check visible boxes

+ Options

| | Status | Label | Response Type | Description | Global Constraints | SUBCASE 1 | SUBCASE 2 |
|--|--------|-------------------------------------|-------------------------------------|--|--------------------|-----------|-----------|
| | | <input type="text" value="Search"/> | <input type="text" value="Search"/> | <input type="text" value="Search"/> | | | |
| | | r1 | DISP | T1, T2 component(s) of displacement at grid 4 | | | |
| | | r2 | STRESS | Stress, item code 2, of elements associated with PROD 11, 12, 13 | | | |

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”

SOL 200 Web App - Optimization Upload Variables Objective Constraints Subcases **Exporter** Results Settings Match Other User's Guide Home

BDF Output - Model

```
assign userfile = 'optimization_results.csv', status = unknown,
form = formatted, unit = 52
$_1_||_2_||_3_||_4_||_5_||_6_||_7_||_8_||_9_||_10_
ID MSC DSOU01 $ v2004 ehj 25-Jun-2003
TIME 10 $
SOL 200
CEND

TITLE = SYMMETRIC THREE BAR TRUSS DESIGN OPTIMIZATION - DSOU01
SUBTITLE = BASELINE - 2 CROSS SECTIONAL AREAS AS DESIGN VARIABLES
$ Result Output
ECHO = NONE
SPC = 100
DISPLACEMENT(SORT1,REAL)=ALL
SPCFORCES(SORT1,REAL)=ALL
STRESS(SORT1,REAL,VONMISES,BELTM)=ALL
$ Subcases
DESOR3(MIN) = 8000000
$ DESGLB Slot
$ DSAPRT(FORMATTED, EXPORT, END=SENS) = ALL
SUBCASE 1
ANALYSIS = STATICS
DESSUB = 40000001
$ DRSPAN Slot
LABEL = LOAD CONDITION 1
LOAD = 300
SUBCASE 2
ANALYSIS = STATICS
DESSUB = 40000001
```

BDF Output - Design Model

```
$*****
$*
$*          Design Model          *
$*
$*****
$
$          Design Variables - Type 1
$-----
$
$
$
DVPREL1 1000001 PROD 11 A
100001 1.0
DVPREL1 1000002 PROD 12 A
100002 1.0
DVPREL1 1000003 PROD 13 A
100003 1.0
$
$
DESVAR 100001 X1 1.0 .01 100.
DESVAR 100002 X2 2.0 .01 100.
DESVAR 100003 X3 1.0 .01 100.
$
$
$
DLINK 1 100003 100001 1.0
$
$          Design Variables - Type 2
$-----
$
$
$
$
$          Design Objective
$-----
$
$
DRESP1 8000000 r0 WEIGHT 3 3
```

Download BDF Files

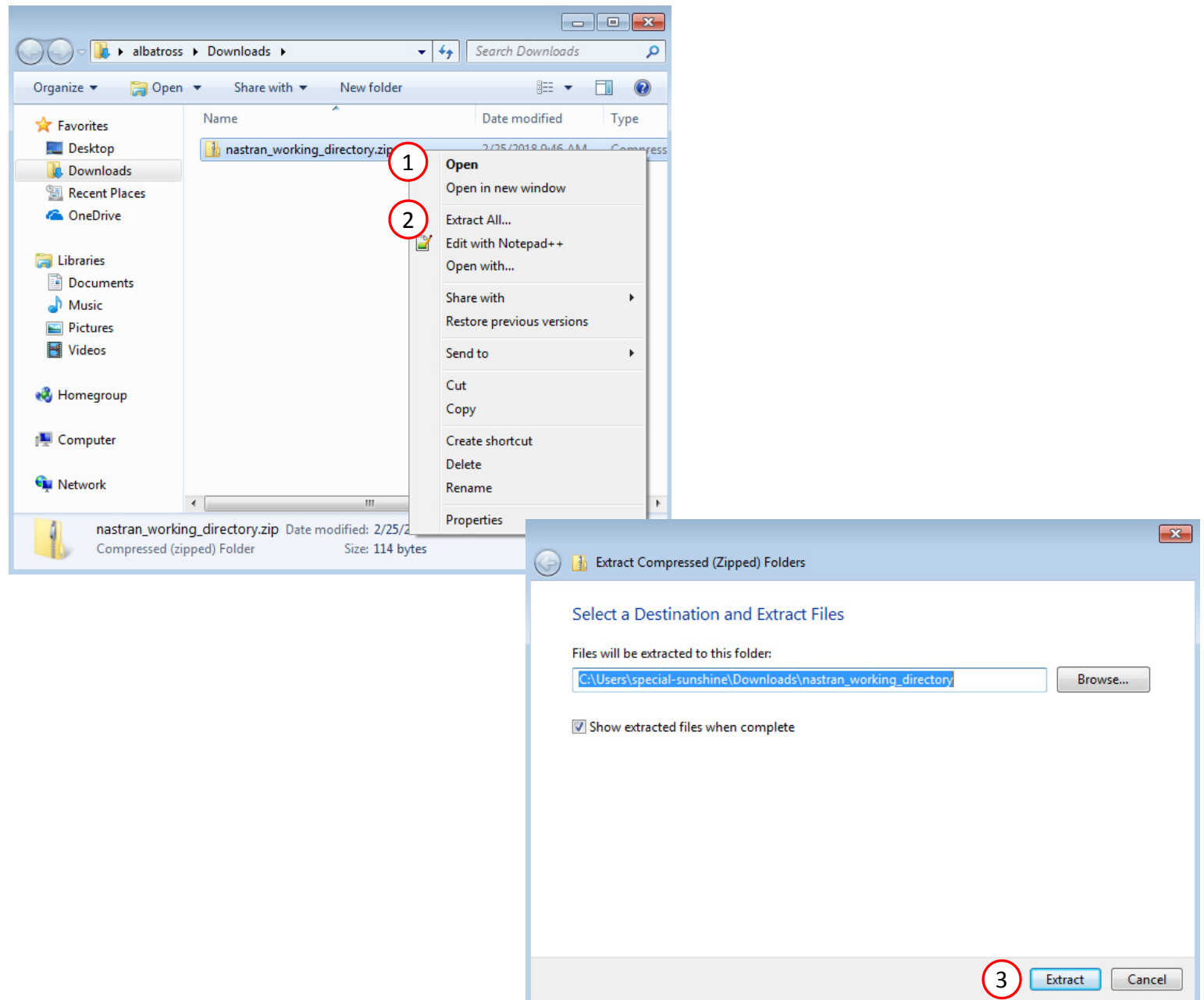
[Download BDF Files](#) 2

Developed by The Engineering Lab

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.

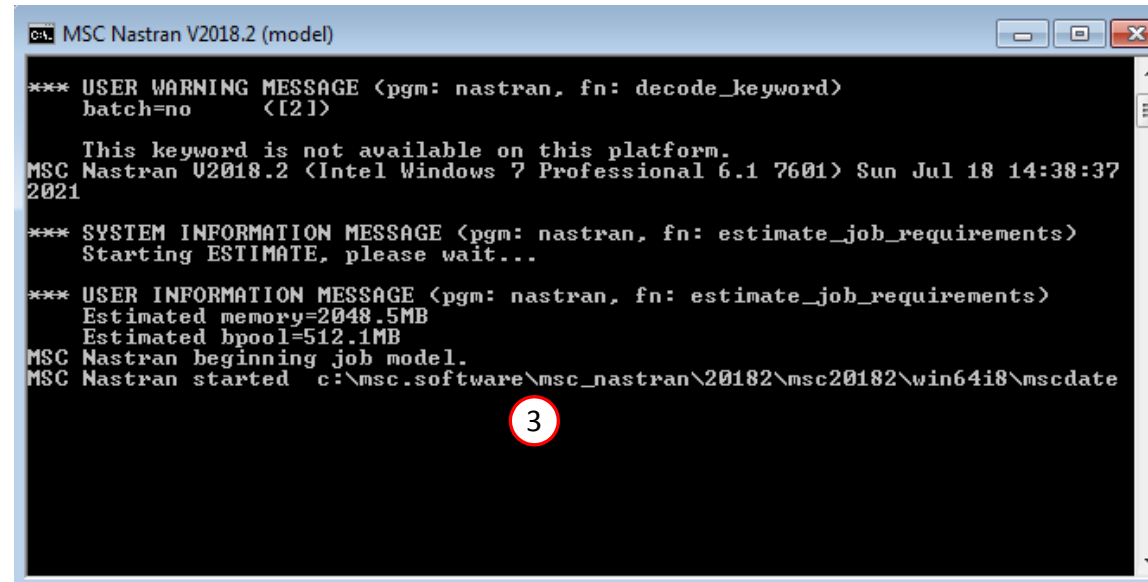
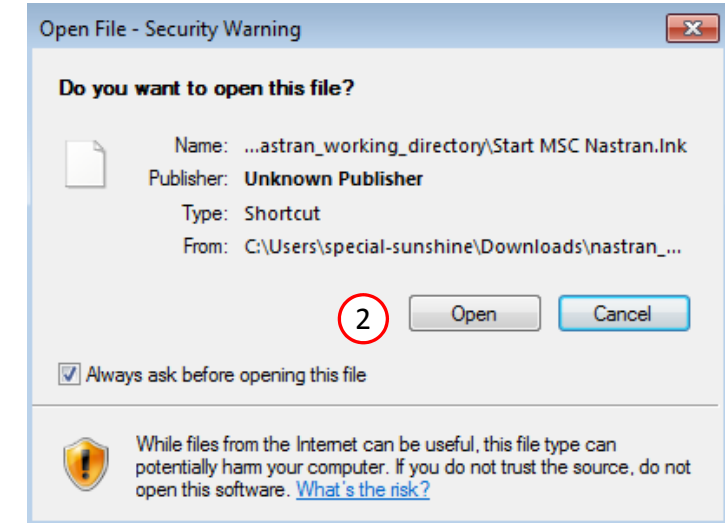
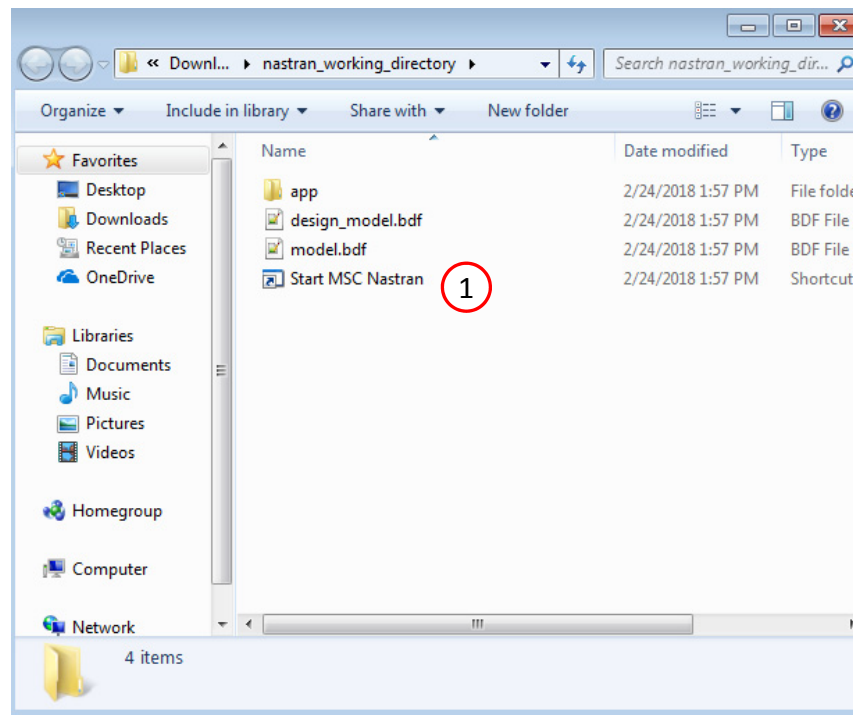
Using Linux?

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.

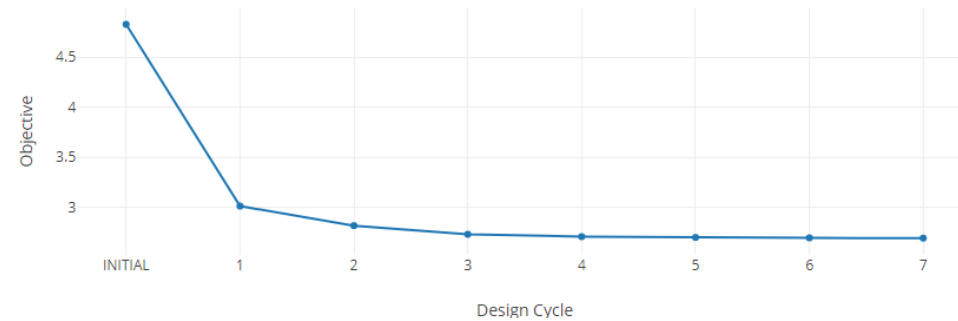
- After an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- This optimization involved 3 design variables, but the plot to the right appears to show only 2 variables. Recall that variable x3 was linked to variable x1, so both x3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

Final Message in .f06

1

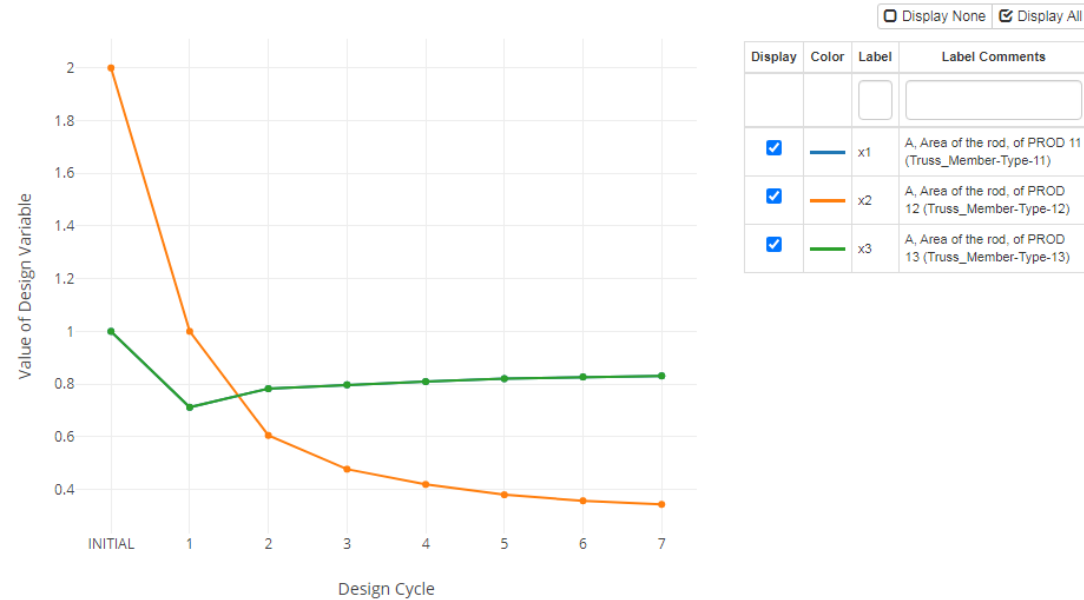
✓ RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 7.

Objective



2

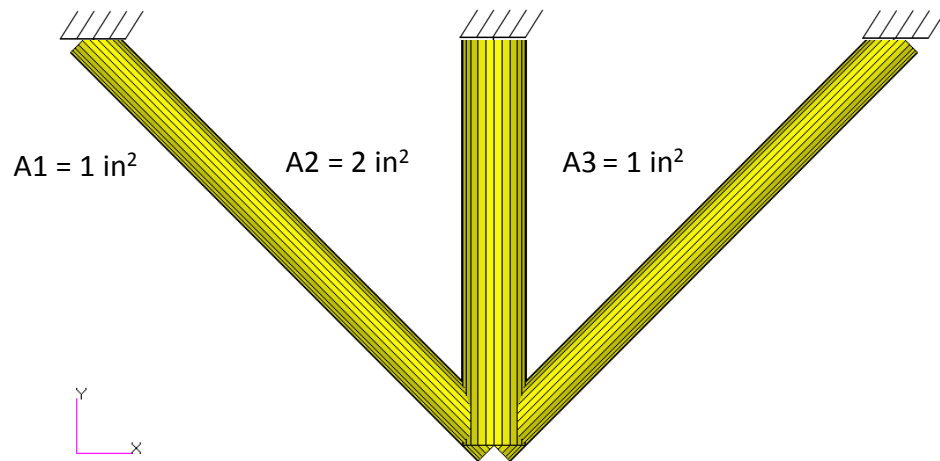
Design Variables



Results

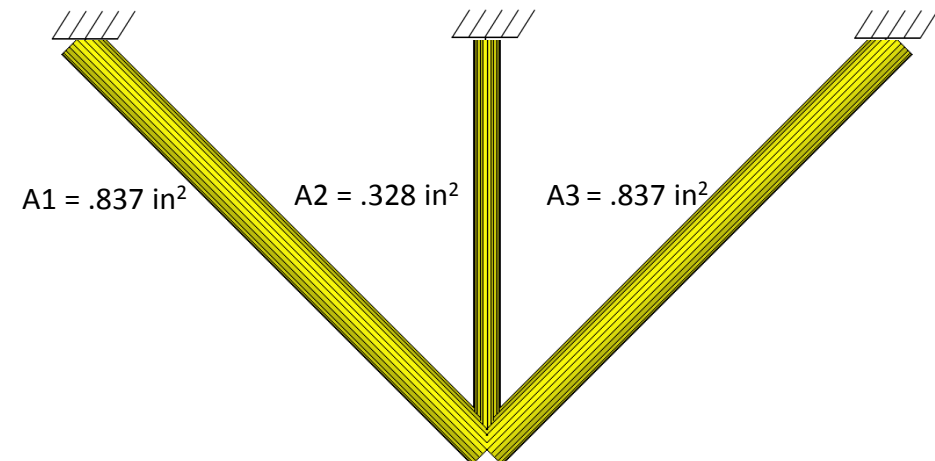
Before Optimization

- Weight: 4.82 lbs.



After Optimization

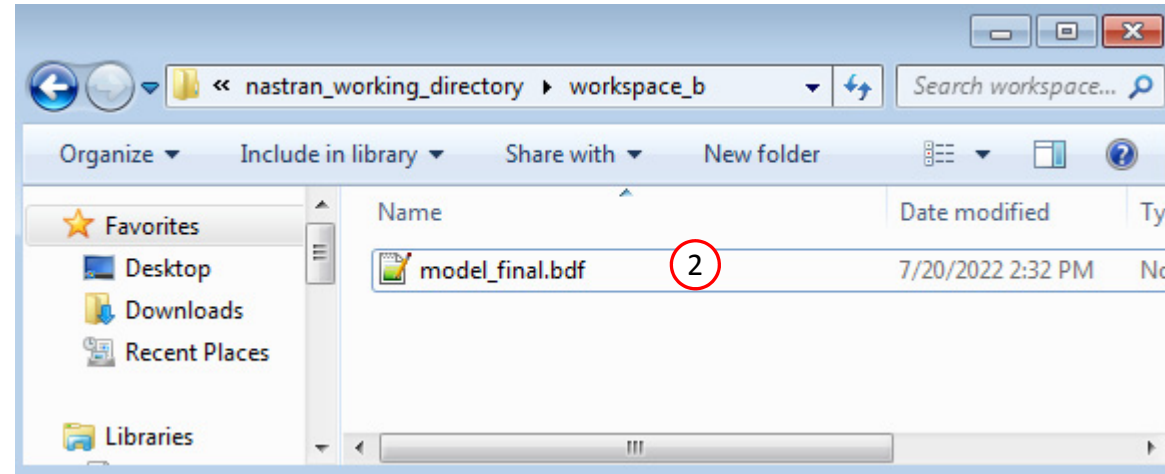
- Weight: 2.70 lbs.



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
    
```

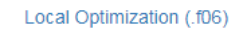
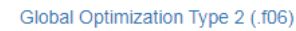
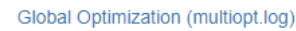
Updated BDF File (model_final.bdf)

3

```

$ PROPERTY DATA
$ Elements and Element Properties for region : Truss_Member-Type-11
PROD 11 1 1.837243 0.0 0.0 0.0
$ Elements and Element Properties for region : Truss_Member-Type-12
PROD 12 1 1.328299 0.0 0.0 0.0
$ Elements and Element Properties for region : Truss_Member-Type-13
PROD 13 1 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
    
```

1. Click Results
2. Click PCH to BDF



Converter

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

Step 1 - Select PCH File

Select files model.pch 1

Inspecting: 100%

☐ List of Selected Files

PCH Entries

| | | | | | | |
|------|----|---|---------|-----|-----|-----|
| PROD | 11 | 1 | .837243 | 0.0 | 0.0 | 0.0 |
| PROD | 12 | 1 | .328299 | 0.0 | 0.0 | 0.0 |
| PROD | 13 | 1 | .837243 | 0.0 | 0.0 | 0.0 |

Step 2 - Select BDF Files

Select files dsoug1.dat 2

Inspecting: 100%

☐ List of Selected Files

BDF Entries

| | | | |
|------|----|---|-----|
| PROD | 11 | 1 | 1.0 |
| PROD | 12 | 1 | 2.0 |
| PROD | 13 | 1 | 1.0 |

3



Step 3 - Download New BDF Files

On download, the PCH entries will replace older BDF entries.

Download

4

Update the Original Model

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

```
14 STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
15
16 $ Subcases
17 SUBCASE 1
18 LABEL = LOAD CONDITION 1
19 LOAD = 300
20 SUBCASE 2
21 LABEL = LOAD CONDITION 2
22 LOAD = 310
23 BEGIN BULK
24 param, post, 1
25 $
26 $-----
27 $ ANALYSIS MODEL
28 $-----
29 $
30 $ GRID DATA
31 $ 2 3 4 5 6 7 8 9 10
32 GRID 1 -10.0 0.0 0.0
33 GRID 2 0.0 0.0 0.0
34 GRID 3 10.0 0.0 0.0
35 GRID 4 0.0 -10.0 0.0
36 $ SUPPORT DATA
37 SPC1 100 123456 1 THRU 3
38 $ ELEMENT DATA
39 CROD 1 11 1 4
40 CROD 2 12 2 4
41 CROD 3 13 3 4
42 $ PROPERTY DATA
43 $ Elements and Element Properties for region : Truss_Member-Type-11
44 PROD 11 1 1.0
45 $ Elements and Element Properties for region : Truss_Member-Type-12
46 PROD 12 1 2.0
47 $ Elements and Element Properties for region : Truss_Member-Type-13
48 PROD 13 1 1.0
49 $ Material Record : Aluminum
50 $ Description of Material : Date: 17-Apr-18 Time: 17:36:49
51 MAT1 1 1.0E+7 0.33 0.1
52 $ EXTERNAL LOADS DATA
53 FORCE 300 4 20000. 0.8 -0.6
54 FORCE 310 4 20000. -0.8 -0.6
55 ENDDATA
56
```

Original BDF/DAT File

Downloaded BDF/DAT File

Beams Viewer

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

1

SOL 200 Web App

Select a web app to begin



Optimization for SOL 200



Multi Model Optimization



Machine Learning | Parameter Study



HDF5 Explorer



Remote Execution

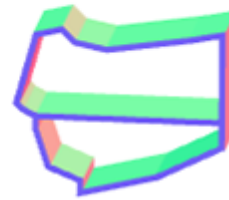
Tutorials and User's Guide

2 Full list of web apps

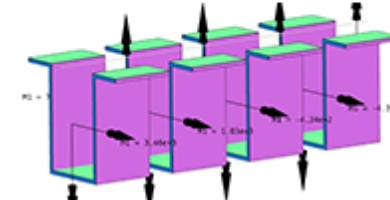
Beams Viewer

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

Beams

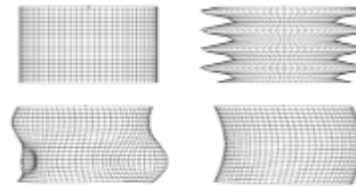


PBMSECT

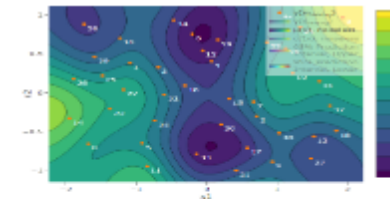


Beams Viewer

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

The screenshot displays the Beams Viewer interface. At the top, the 'File Upload' section shows a 'BDF' file named 'model_final.bdf' being uploaded, with a progress bar indicating 'Uploading: 100 %'. The 'Controls' panel on the right includes buttons for 'Upload BDF', 'Upload PS', and 'Upload H5', as well as 'Center Model' and 'Fit Model' options. Below the upload section, a 3D model of a beam structure is shown, with a coordinate system (x, y, z) at the bottom left. An 'Open' file dialog is overlaid on the right, showing the file 'model_final.bdf' selected in the 'workspace_b' directory. The dialog includes a search bar, a list of files, and an 'Open' button.

File Upload

BDF

3. Select files model_final.bdf

4. Upload files

Uploading: 100 %

Controls

File Upload

Upload BDF

Upload PS

Upload H5

Tools

Center Model

Fit Model

Open

nastran_working_dir... workspace_b

Search workspace_b

Organize New folder

Favorites

Desktop

Downloads

OneDrive

Recent Places

Libraries

Documents

Music

Pictures

Videos

Name Date modified Type

model_final.bdf 6/7/2022 11:53 AM Notepad+

File name: model_final.bdf Custom Files (*.bdf;*.dat;*.inc;*)

Open Cancel

Elem. Coordinate System

Beam Shape

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

File Upload

H5

Session ID: 4162

Upload .h5 File

1. Select files **2** model.h5

2. Upload files **5**

Uploading: 100 %

Loading: 100 %

Acquired Datasets

| Dataset | Acquire | Parse |
|----------------------|---------|-------|
| ELEMENTAL/STRESS/ROD | ✓ | ✓ |

Open

File name: model.h5 **4** *.h5

Open Cancel

File explorer contents:

| Name | Date modified | Type |
|-------------------|-------------------|-------------|
| app | 7/20/2022 2:32 PM | File folder |
| workspace_b | 7/20/2022 2:32 PM | File folder |
| model.h5 3 | 7/20/2022 2:32 PM | H5 File |

Controls

File Upload

Upload BDF

Upload PS

Upload H5 **1**

Tools

Center Model

Fit Model

Isometric View 1

Background Color

Label Color

Results

Forces and Stresses

Display

GRID IDs

Element IDs

Orientation Vector

Elem. Coordinate System

Beam Shape

Display Internal Element Moments

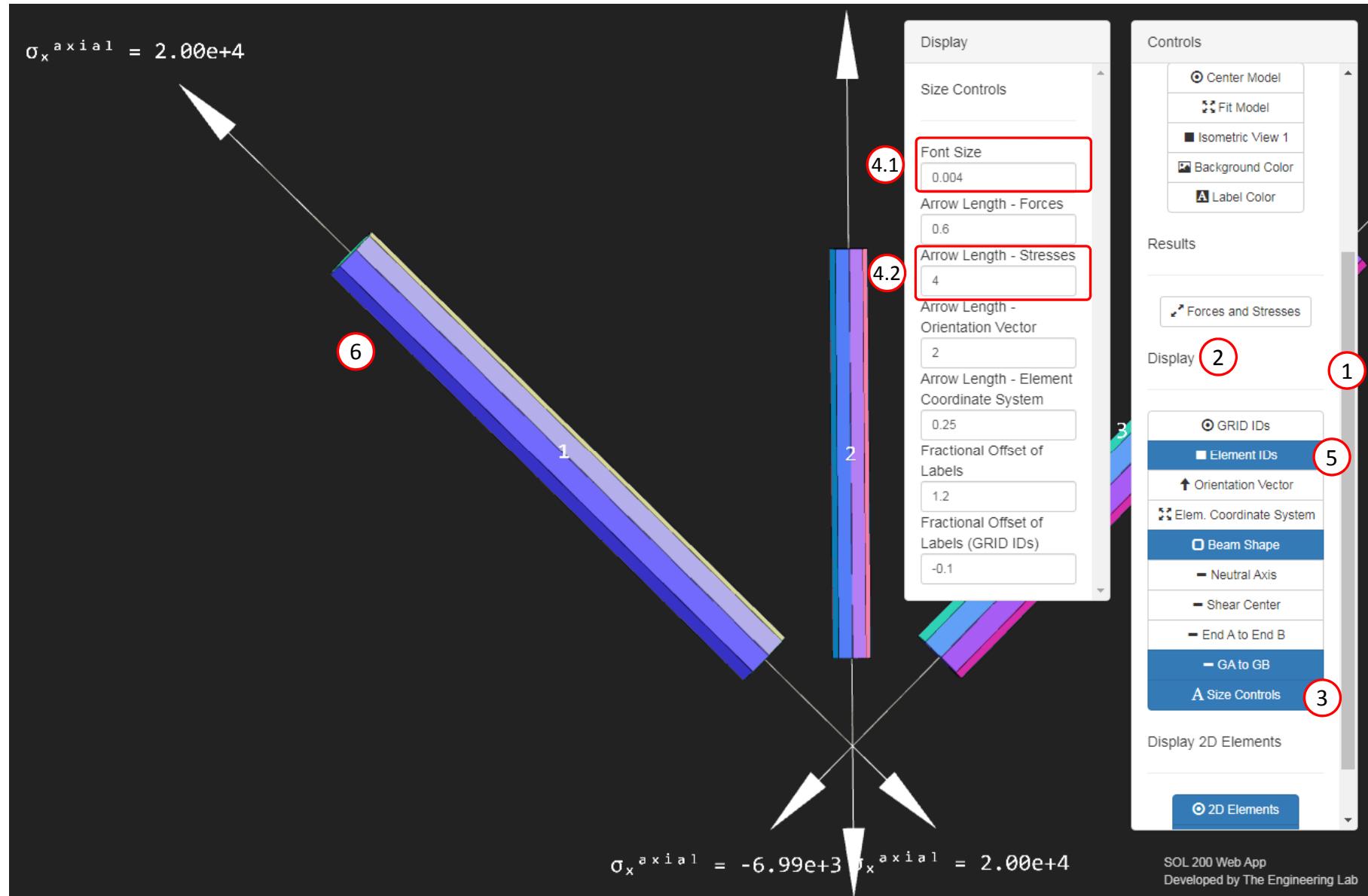
1. Click Forces and Stresses
2. Mark the check boxes for $\sigma_{x, \text{axial}}$ which corresponds to the axial stress
3. Axial stress arrows are now displayed on the 1D elements
4. Select the following:
 1. SUBCASE: 1
 2. ANALYSIS: 1
 3. DESIGN_CYCLE: 7

The screenshot shows the Hexagon software interface with the 'Display Element Forces and Stresses' panel open. The 'Domains' section has SUBCASE: 1, ANALYSIS: 1, and DESIGN_CYCLE: 7 selected. The 'Stresses' section has the checkbox for $\sigma_{x, \text{axial}}$ checked. The 'Results' section has the 'Forces and Stresses' button selected. The 'Display' section has 'Beam Shape' selected. The 'Display 2D Elements' section has '2D Elements' selected. The main view shows a 3D model of a structure with axial stress arrows displayed on the 1D elements.

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
$
c
```

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'