Workshop – Buckling Optimization of a Cantilever Beam

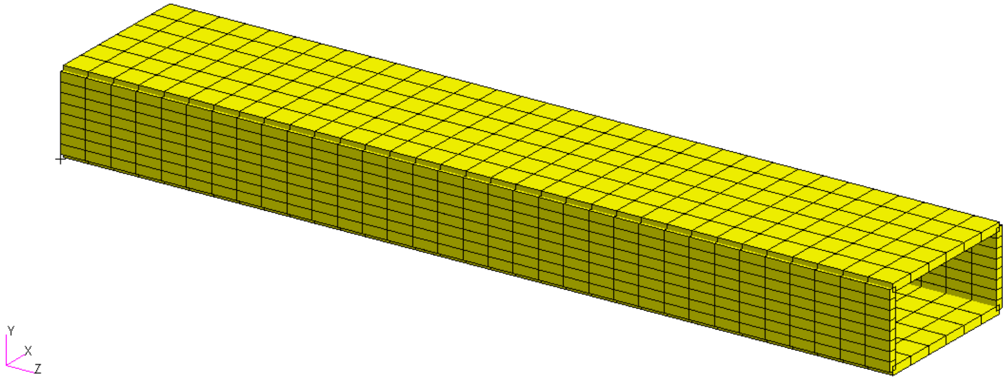
AN MSC NASTRAN SOL 200 TUTORIAL

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

Goal: Use Nastran SOL 200 Optimization

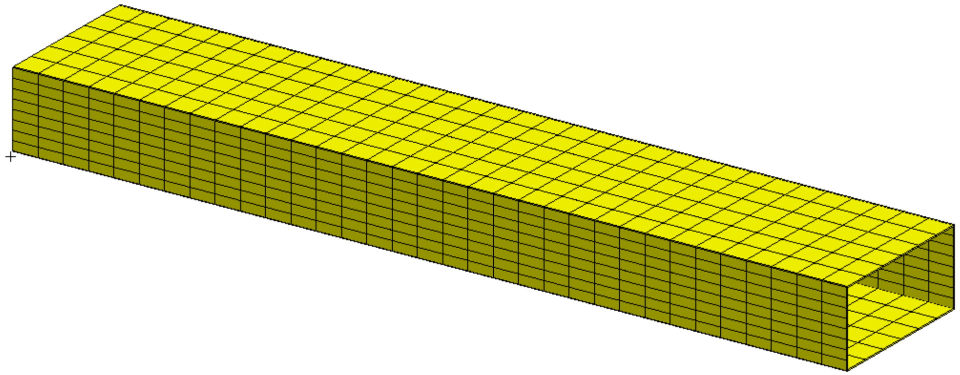
Before Optimization

- Weight: 15.12 kg
- x1 = T, thickness of wall
- = .01 m
- Load Case 1:
  - Buckling Factor 1: -242.19
  - Buckling Factor 2: 242.19

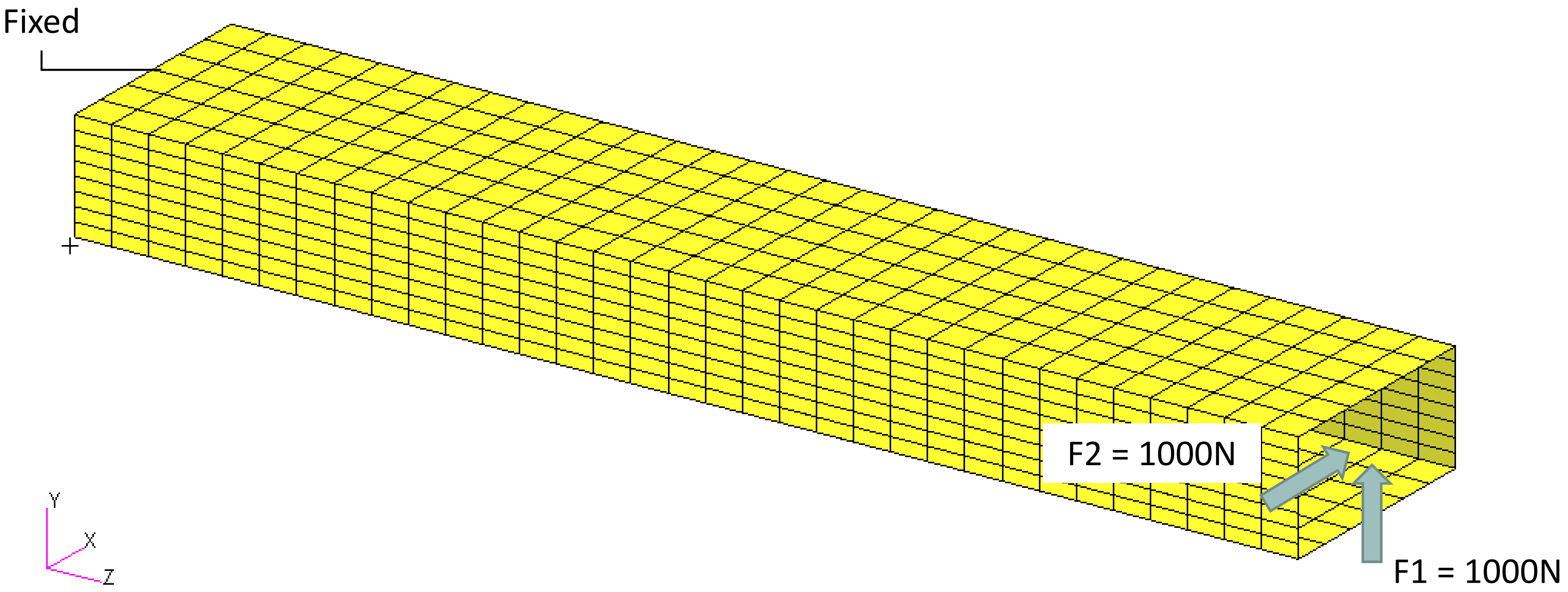


After Optimization

- Weight: 2.35 kg
- x1 = T, thickness of wall
- =.0016 m
- Load Case 1:
  - Buckling Factor 1: -1.0071
  - Buckling Factor 2: 1.0071



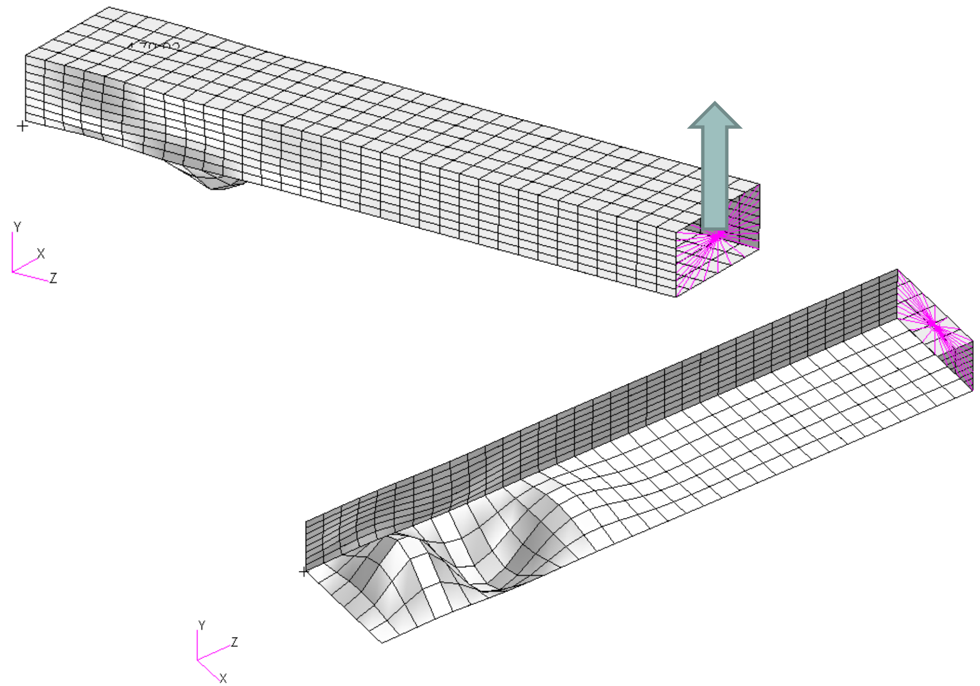
Details of the Structural Model



Buckling Factors for Load Case 1

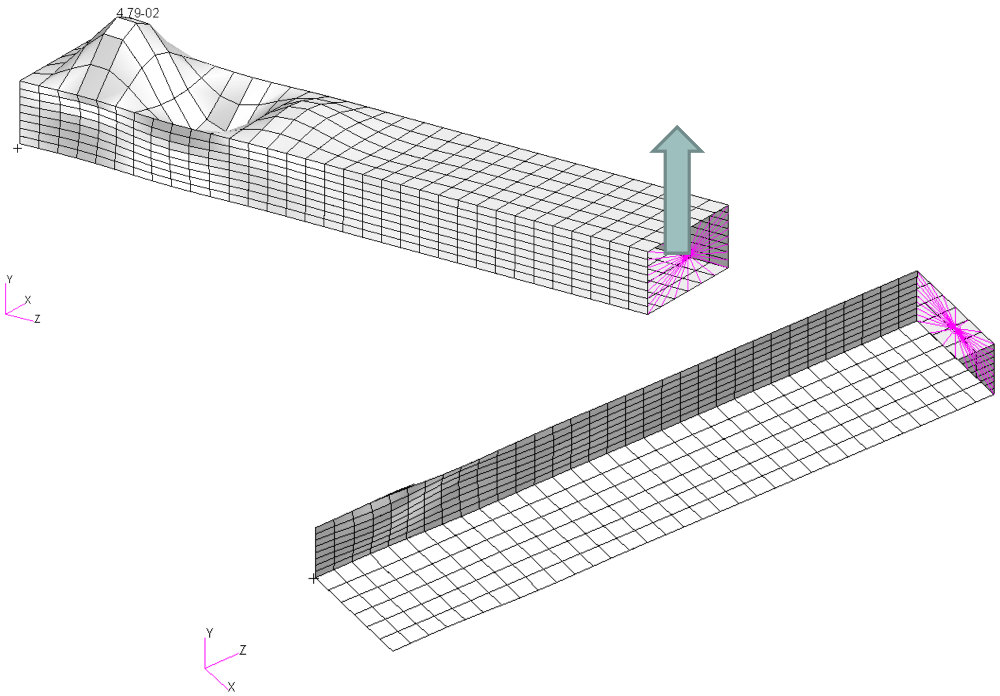
Mode 1

Buckling Factor: -242.19



Mode 2

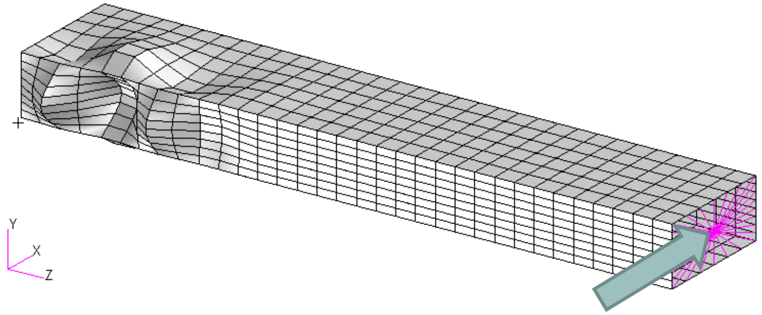
Buckling Factor: 242.19



Buckling Factors for Load Case 2

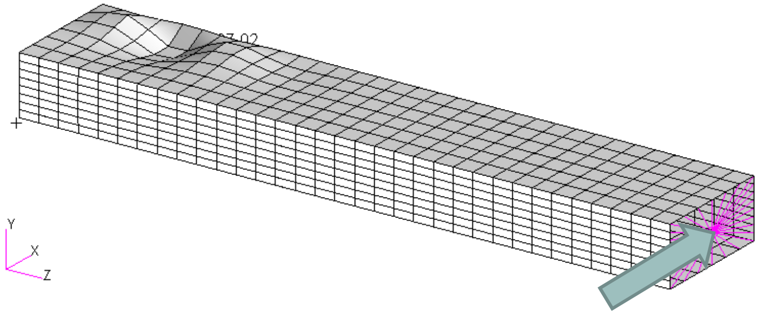
Mode 1

Buckling Factor: -1104.



Mode 2

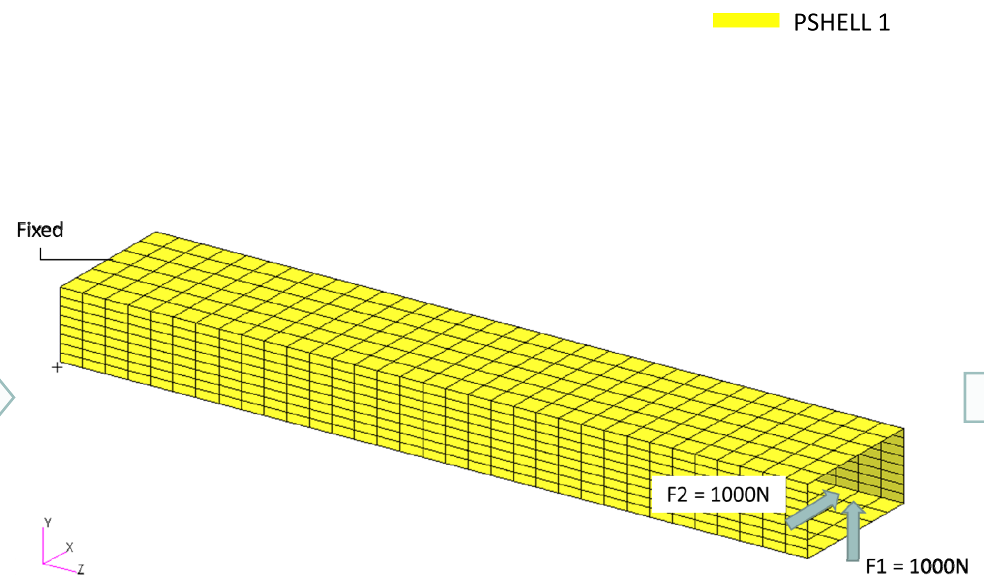
Buckling Factor: 1104.



Optimization Problem Statement

Design Variables

x1: T of PSHELL 1 | .001 < x1



Design Objective

r0: Minimize weight

Design Constraints

r1: The von Mises stress, z1, of PSHELL 1
r2: The von Mises stress, z2, of PSHELL 1

r1, r2 < 2.76E08

Design Constraints, Equation

R1: ABS(b1)
R2: ABS(b2)
1.0 < R1, R2

b1: Buckling factor of mode 1
b2: Buckling factor of mode 2

Optimization Problem Statement, SUBCASE Assignment of Constraints

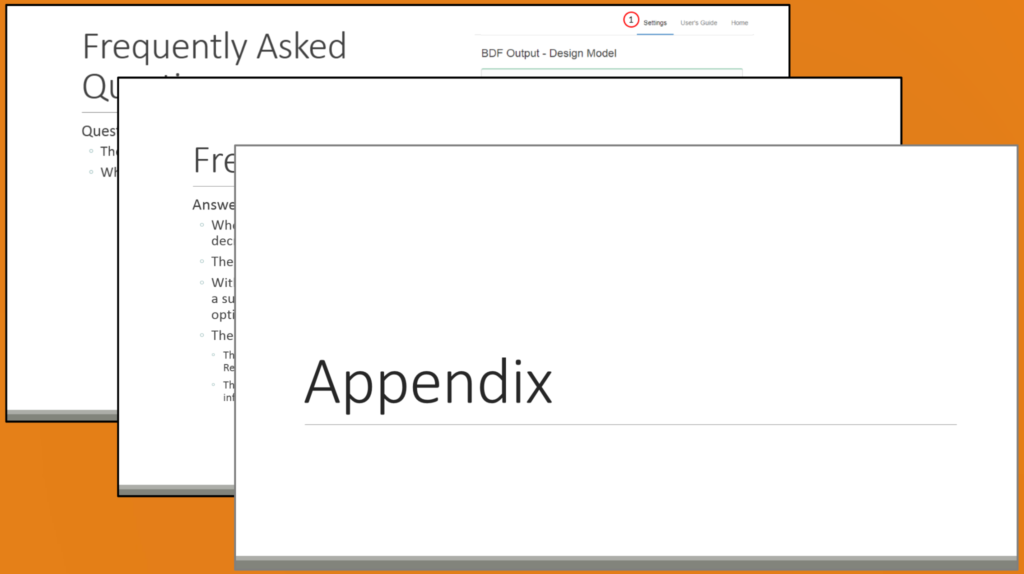
	Status	Label	Response Type	Analysis Type	Description
		Search	Search	Search	Search
		r1	STRESS	STATICS	Stress, von Mises or maximum shear at Z1, of elements...
		r2	STRESS	STATICS	Stress, von Mises or maximum shear at Z2, of elements...
		R1	Equation		
		R2	Equation		

Global Constraints	SUBCASE 1	SUBCASE 2	SUBCASE 3	SUBCASE 4
Analysis Types ➔	Statics	Statics	Buckling	Buckling
	☑	☑		
	☑	☑		
	☐	☐	☑	☑
	☐	☐	☑	☑

More Information Available in the Appendix

The Appendix includes information regarding the following:

- Frequently Asked Question
  - What is the trust region?
- An example where the Trust Region can be used
- Trust Region Visualized
- Considerations for Optimization with Buckling Constraints



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

Optimization for Buckling - This example has multiple load cases and the goal is to avoid buckling for each load case. The SOL 200 Web App facilitates the configuration of multiple SUBCASEs and allows for optimization of multiple buckling scenarios.

SOL 200 Web App Capabilities

The Post-processor Web App and HDF5 Explorer are free to MSC Nastran users.

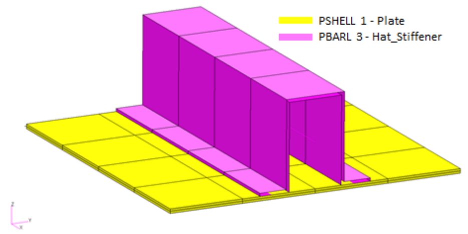
Compatibility

- Google Chrome, Mozilla Firefox or Microsoft Edge
- Windows and Red Hat Linux
- Installable on a company laptop, workstation or server. All data remains within your company.

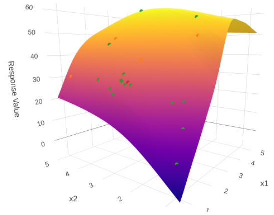
Benefits

- REAL TIME error detection. 200+ error validations.
- REALT TIME creation of bulk data entries.
- Web browser accessible
- Free Post-processor web apps
- +80 tutorials

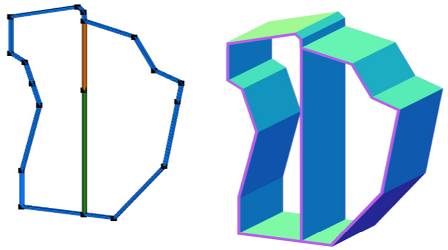
Web Apps



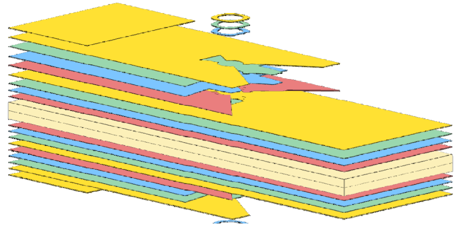
Web Apps for MSC Nastran SOL 200
Pre/post for MSC Nastran SOL 200. Support for size, topology, topometry, topography, multi-model optimization.



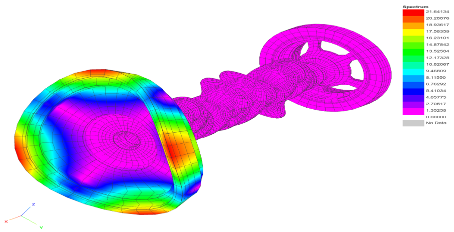
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



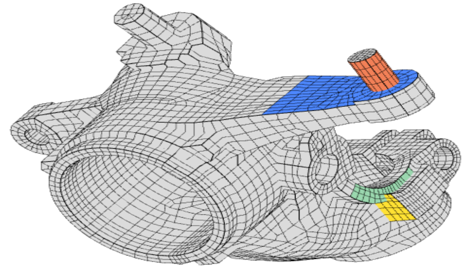
PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically



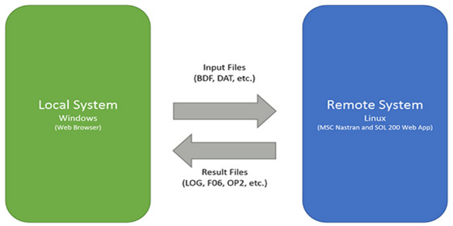
Ply Shape Optimization Web App
Optimize composite ply drop-off locations, and generate new PCOMPG entries



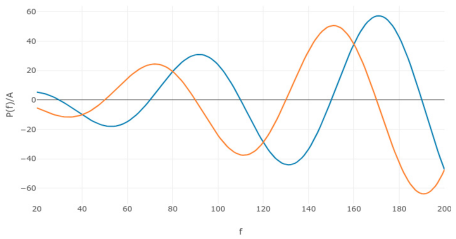
Post-processor Web App
View MSC Nastran results in a web browser on Windows and Linux



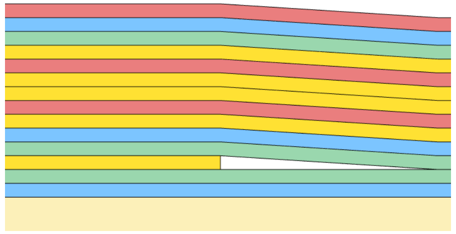
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



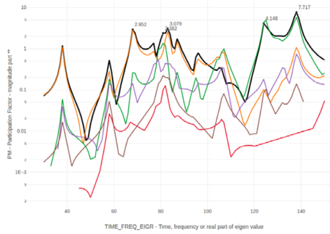
Remote Execution Web App
Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



Dynamic Loads Web App
Generate RLOAD1, RLOAD2 and DLOAD entries graphically



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

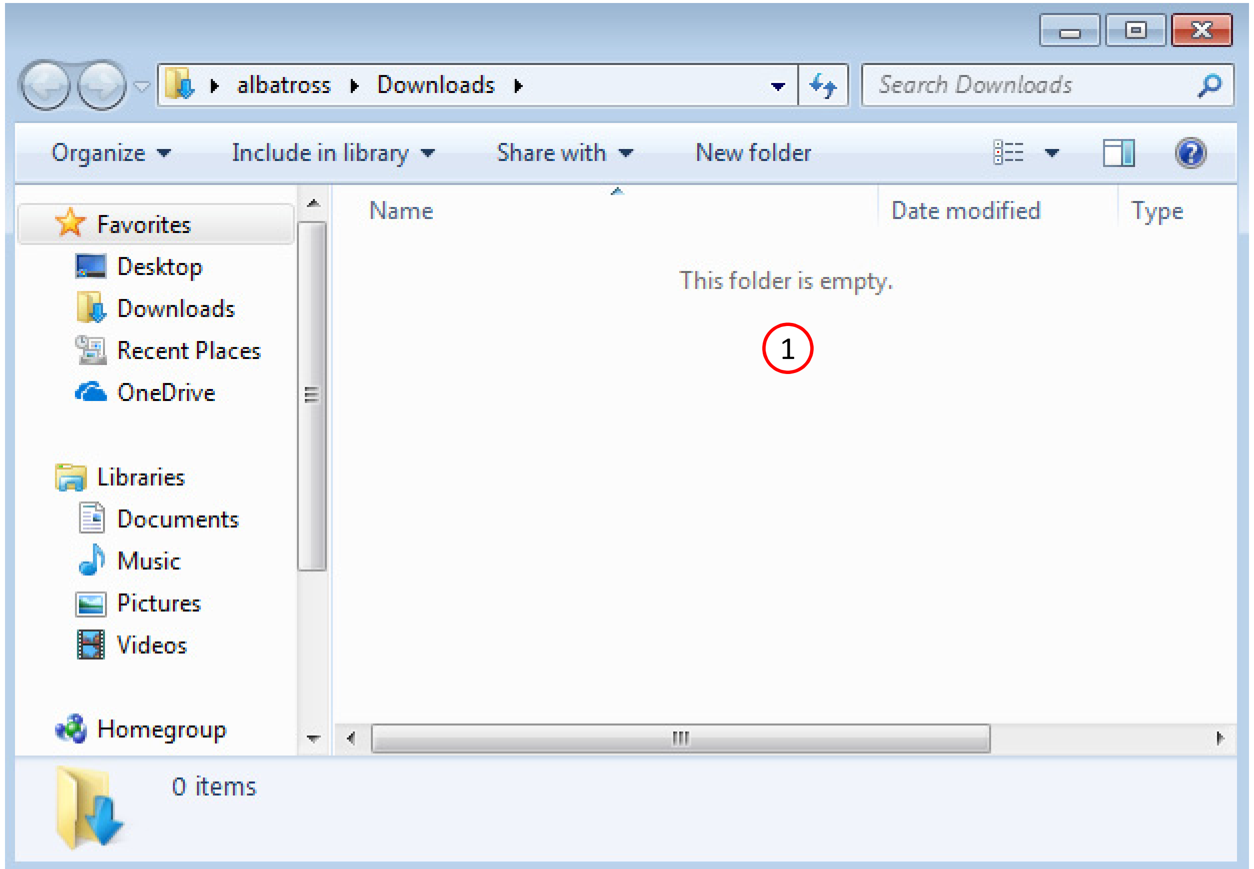


HDF5 Explorer Web App
Create graphs (XY plots) using data from the H5 file

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

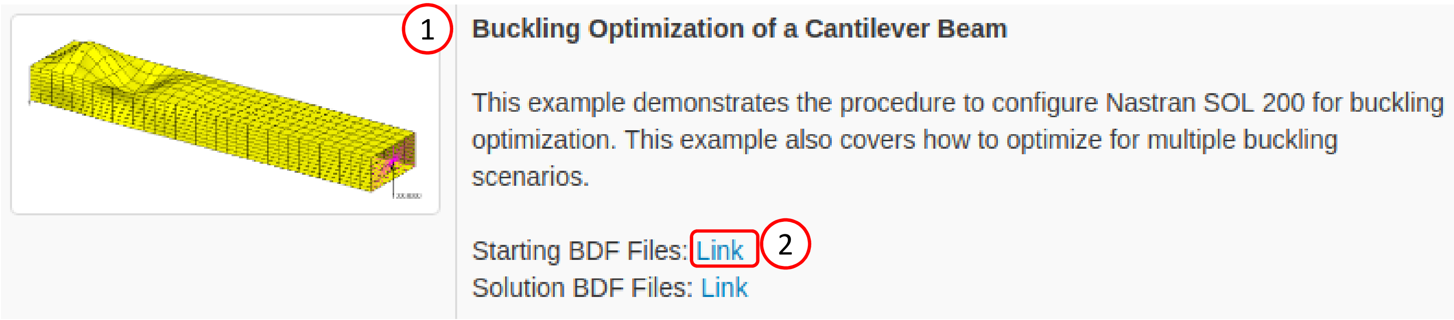
- The necessary BDF files for this tutorial are available in the Tutorials section of the User's Guide.

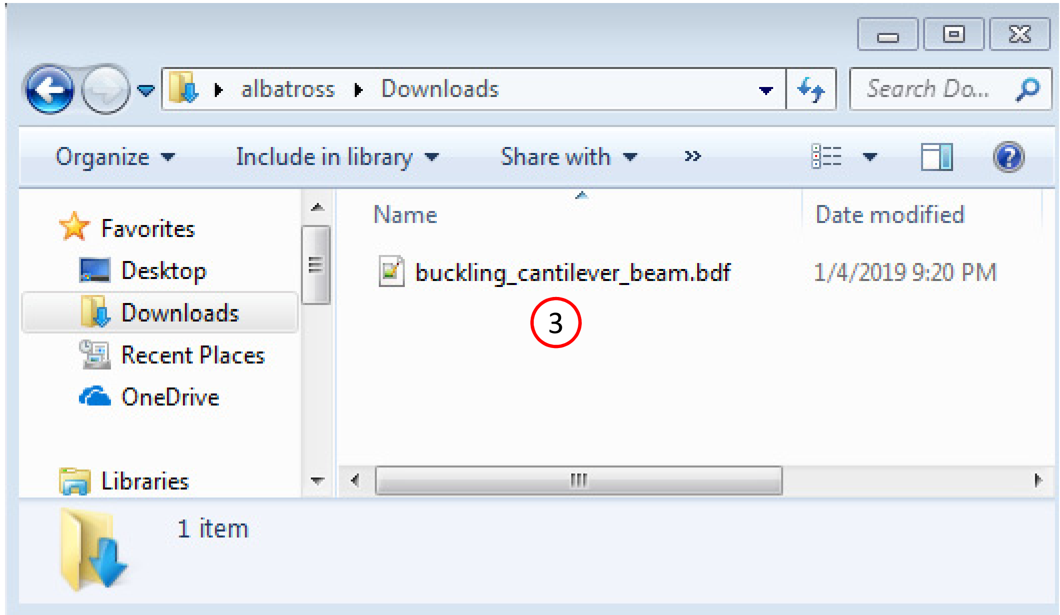
The Engineering Lab

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.

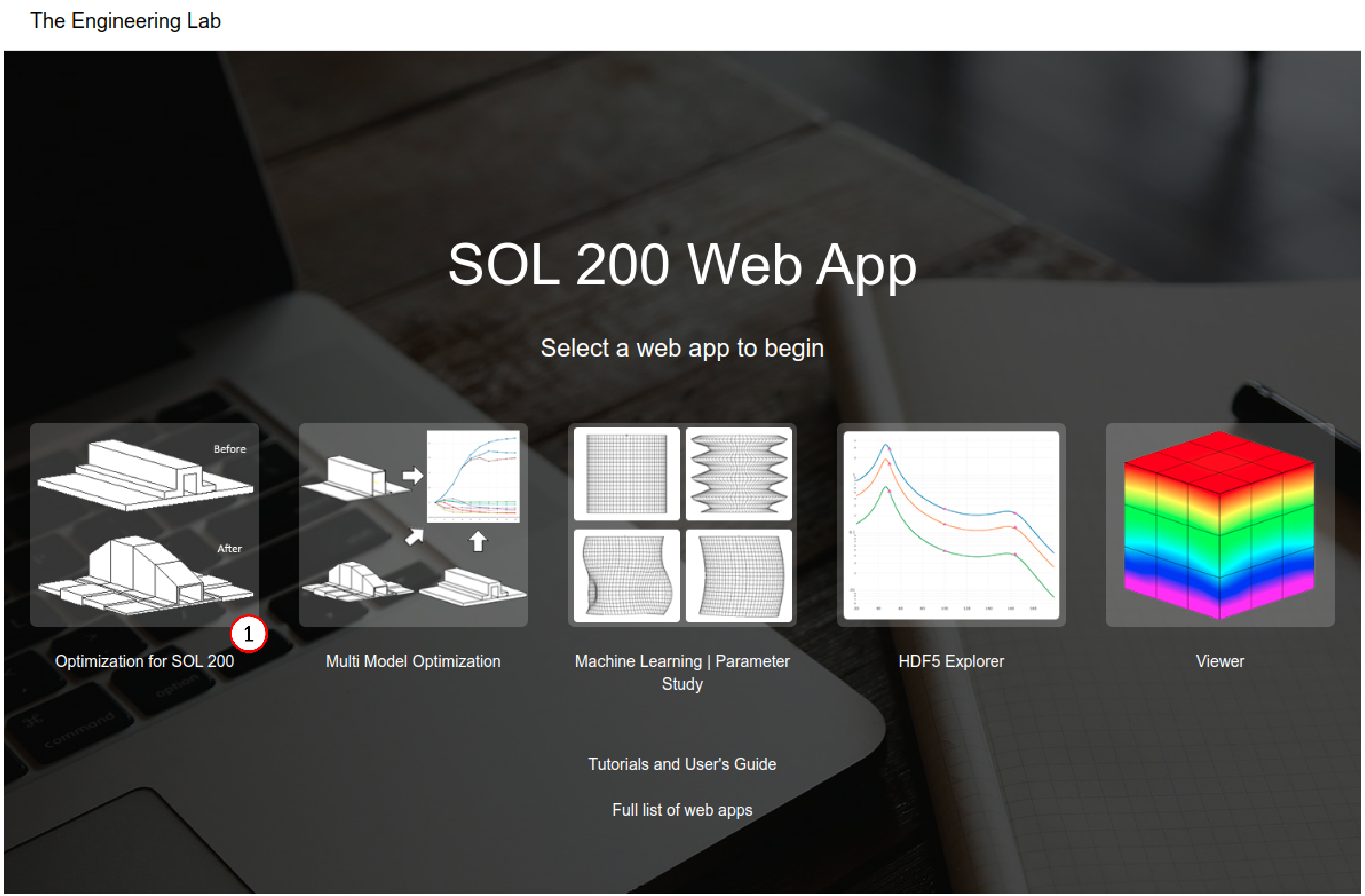




Open the Correct Page

1. Click on the indicated link

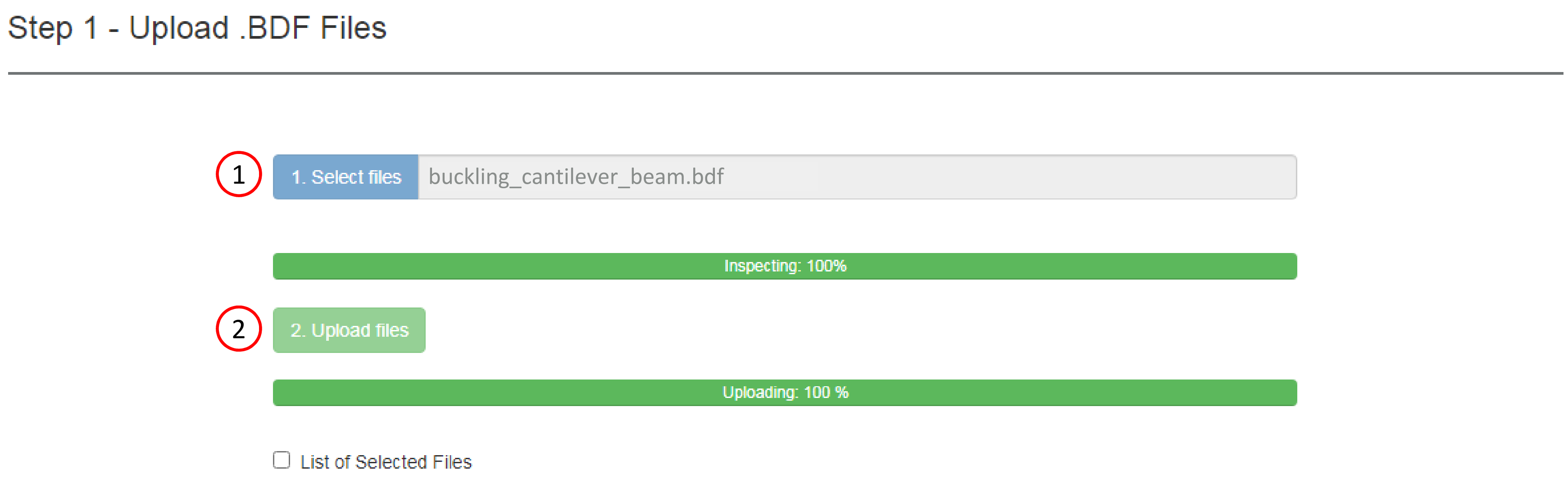
- MSC Nastran can perform many optimization types. The SOL 200 Web App includes dedicated web apps for the following:
    - Optimization for SOL 200 (Size, Topology, Topometry, Topography, Local Optimization, Sensitivity Analysis and Global Optimization)
    - Multi Model Optimization
    - Machine Learning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



Upload BDF Files

1. Click 1. Select Files and select buckling_cantilever_beam.bdf
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 thickness as a design variable
2. The x1 variable has been created for the thickness property

- The necessary design variables, as detailed in the optimization problem statement, are created.
- Each step has hidden functionality for advanced users. The visibility is controlled by clicking + Options.

SOL 200 Web App - Optimization   Upload   Variables   Objective   Constraints   Subcases   Exporter   Results

Size   Topology   Topometry   Topography

Step 1 - Select design properties

+ Options

Create DVXREL1	Property	Property Description	Entry	Entry ID	Current Value
	Search	Search	Search	Search	Search
+	E	Young's modulus	MAT1	1	6.89E+10
+	NU	Poisson's ratio	MAT1	1	.33
+	RHO	Mass density	MAT1	1	2700.
1 +	T	Thickness	PSHELL	1	1.0000-2

5   10   20   30   40   50

Number of Visible Rows 5

Step 2 - Adjust design variables

Delete Visible Rows

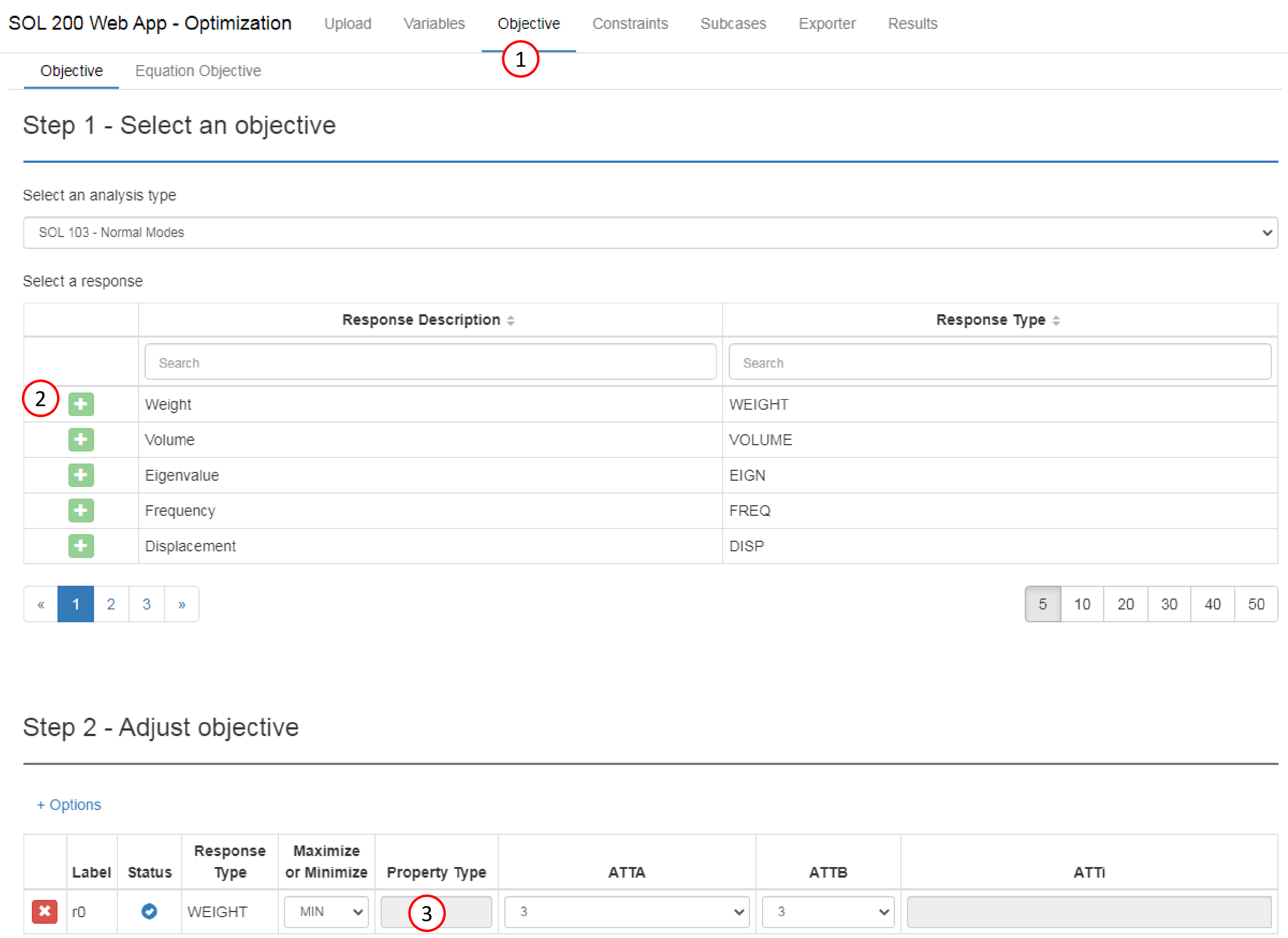
+ Options

	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	2	T	Thickness	PSHELL	1	1.0E-2	.001	Upper	Examples: -2.0, 1.0, THRU, 10.0,

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.

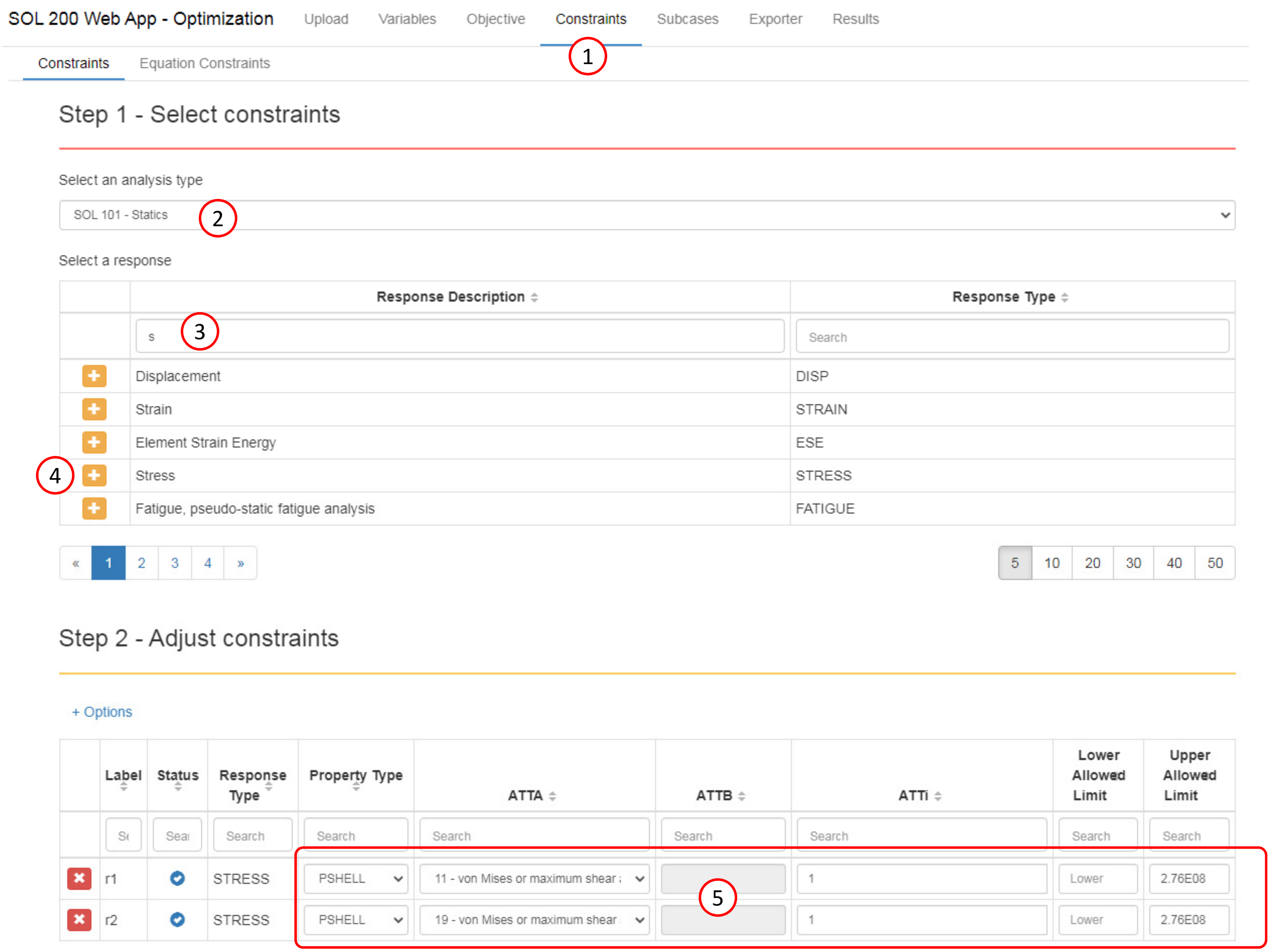


Create Design Constraints

1. Click Constraints
2. Set the analysis type to SOL 101 - Statics
3. In the search box, type ‘s’
4. Select the plus(+) icon 2 times for Stress to create 2 stress constraints
5. Configure the constraints as shown to the right

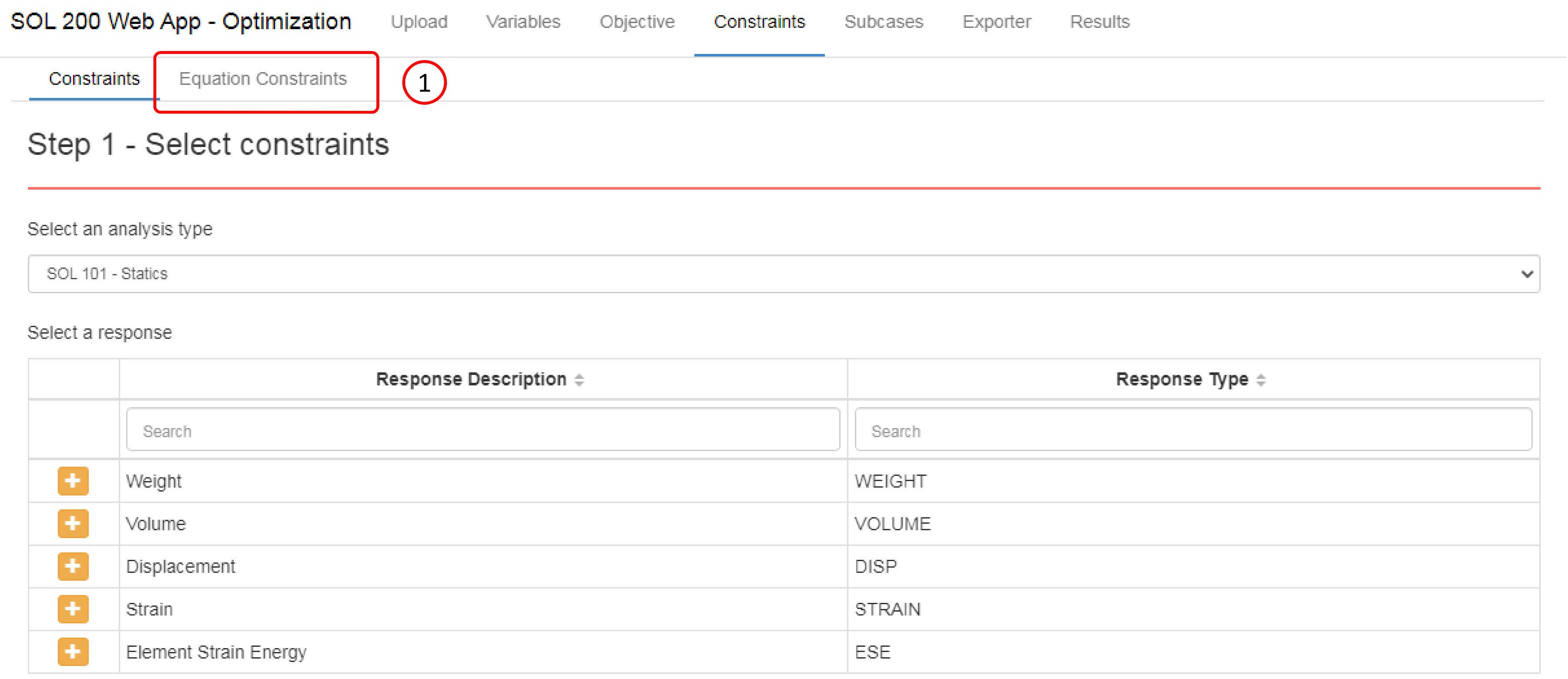
- Example: Configure the following for r1
  - Property Type: PSHELL
  - ATTA: 11 – von Mises or […]
  - ATTi: 1 (PID 3)
  - Upper Allowed Limit: 2.76E08

- Each step has a second row with empty input boxes. These boxes are search boxes and should not be confused with rows 3, 4, 5, etc. Use the search boxes to filter columns in tables with multiple rows.



Create Design Constraints

1. Click Equation Constraints



- There are 2 methods of creating a constraint.
  - Method 1 – Select a constraint from a given list of responses, e.g. Weight, Volume, etc.
  - Method 2 – Create an equation.
- This page shows the use of Method 2 to create an Equation Constraint.

Create Design Constraints

Create a responses that corresponds to the buckling load factors for modes 1 and 2. These responses value will be labeled ‘b1’ and ‘b2’

1. Scroll to Section A - Optional - Create additional responses
2. Change the analysis type to: SOL 105 - Buckling
3. Click the plus (+) icon for Buckling Eigenvalue/Factor 2 times to create 2 buckling factor responses
4. Configure the following for b1
   - ATTA: 1 (Buckling Mode 1)
5. Configure the following for b2
   - ATTA: 2 (Buckling Mode 2)

Step A - Optional - Create additional responses 1

Select an analysis type

SOL 105 - Buckling 2

Select a response

	Response Description	Response Type
	Search	Search
+	Weight	WEIGHT
3 +	Volume	VOLUME
+	Buckling Eigenvalue/Factor	LAMA
+	Weight from Particular Material or Property ID	WMPID

Step B - Optional - Adjust responses

+ Options

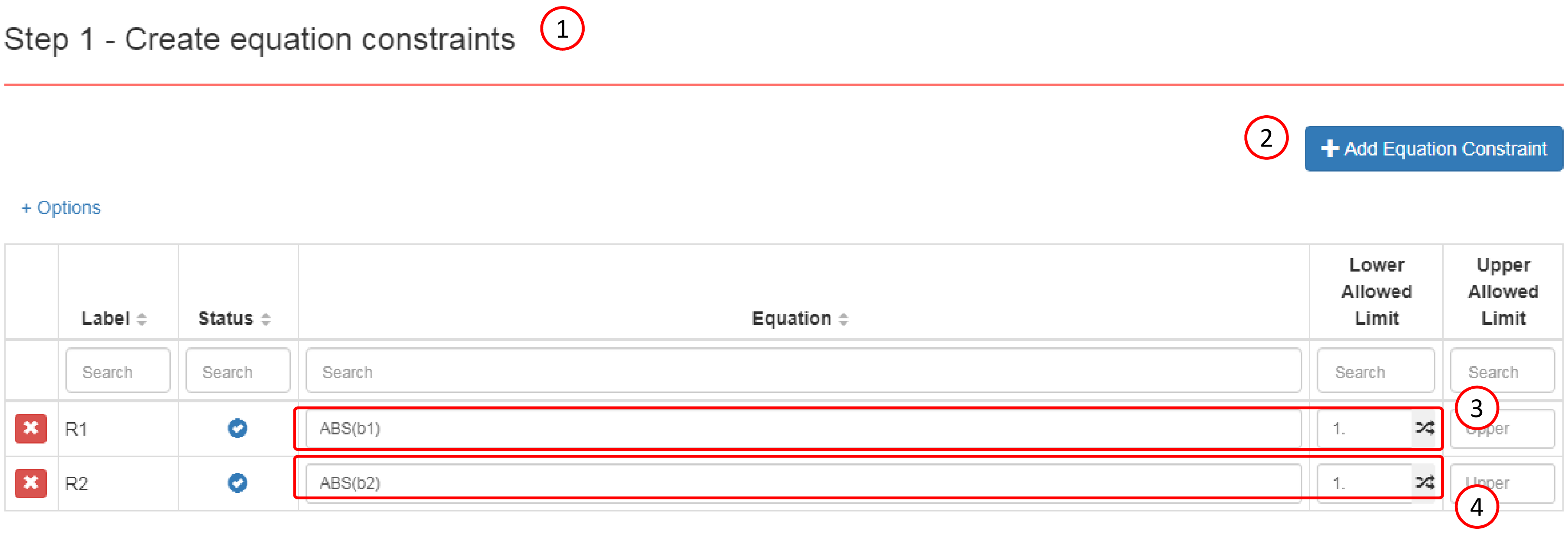
	Label	Status	Response Type	Property Type	ATTA	ATTB	ATTi
	Se	Sea	Search	Search	Search	Search	Search
✖	b1	✔	LAMA		1 4		
✖	b2	✔	LAMA		2 5		

Create Design Constraints

Create an equation constraint for each buckling mode. Since the buckling load factor can be negative, the absolute function (abs()) is used.

1. Scroll to section Step 3 - Optional - Create equation constraints
2. Click on Add Equation Constraint 2 times
3. Configure the following for R1:
    - Equation: ABS(b1)
    - Lower Allowed Limit: 1.
4. Configure the following for R2:
    - Equation: ABS(b2)
    - Lower Allowed Limit: 1.

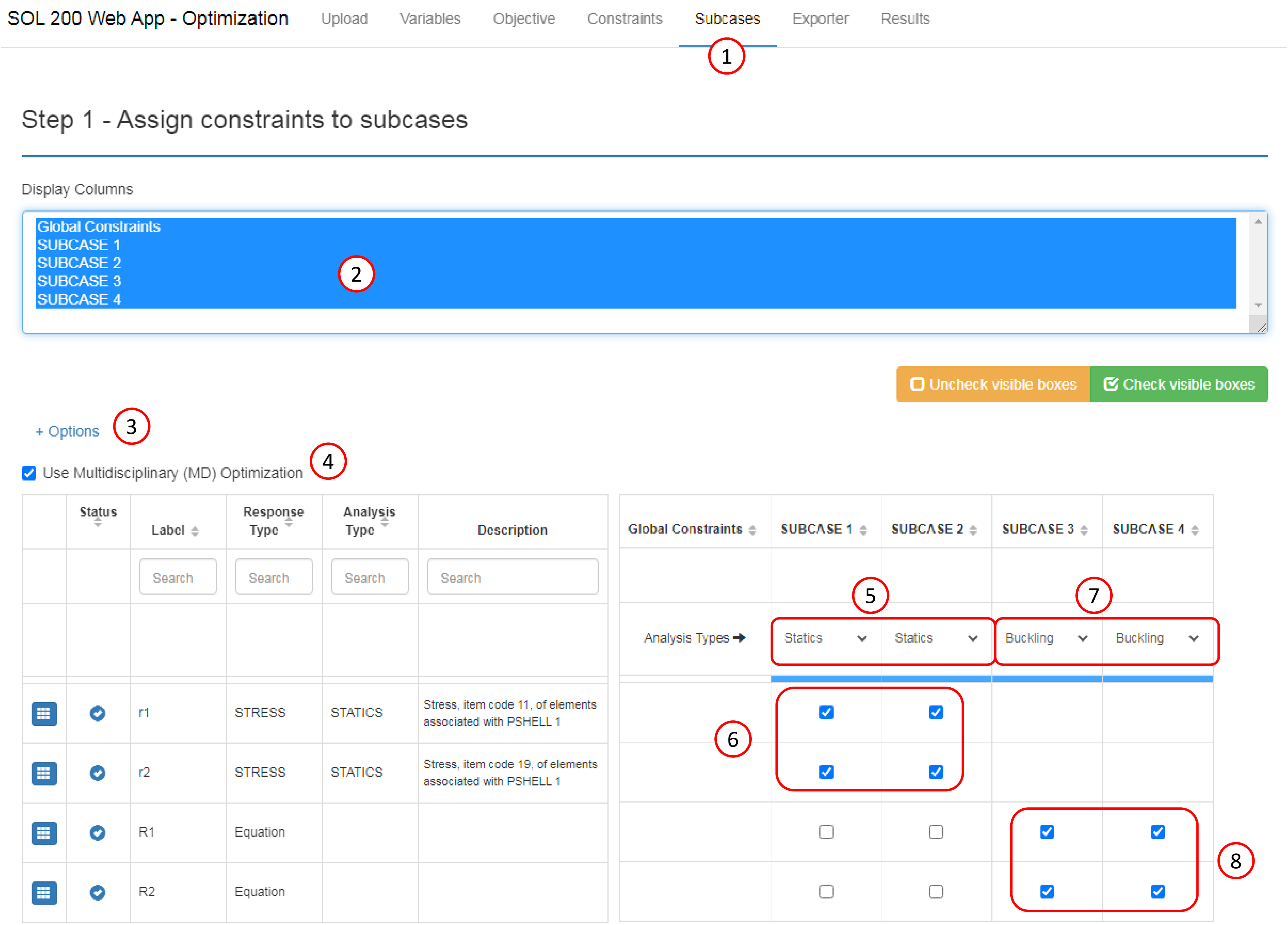
- A previous buckling analysis revealed negative buckling load factors (BLF). A negative buckling load indicates that buckling would occur if the load was reversed. To address negative BLFs, the absolute value of BLFs is constrained.



Assign Constraints to Load Cases (SUBCASES)

1. Click Subcases
2. Select each option in the select box (Hold down the Shift key on the keyboard and use the mouse to select multiple options)
3. Click + Options
4. Mark the checkbox for Use Multidisciplinary (MD) Optimization
5. For SUBCASE 1 and 2, change the analysis type to Statics
6. Mark the checkboxes to assign the stress constraints to SUBCASE 1 and 2
7. For SUBCASE 3 and 4, change the analysis type to Buckling
8. Mark the checkboxes to assign the equation constraints (buckling constraints) to SUBCASE 3 and 4

- If the SUBCASE in the Case Control Section includes that STATSUB command, then to perform an optimization, then the SUBCASE should also be configured for BUCKLING as shown on this page.



Configure Settings

1. At the top right hand corner, click on Settings
2. Ensure the checkboxes are marked and the respective values match as shown in the image to the right

- For each design cycle, the allowable change of variables is either kept constant or decreased, but never increased. An opportunity exists to adaptively vary the change of variables so that they can be increased or decreased automatically. For some optimization scenarios, the increase in allowable variable change can yield faster optimizations. In this example, the Trust Region is used. Refer to the Appendix for additional details.

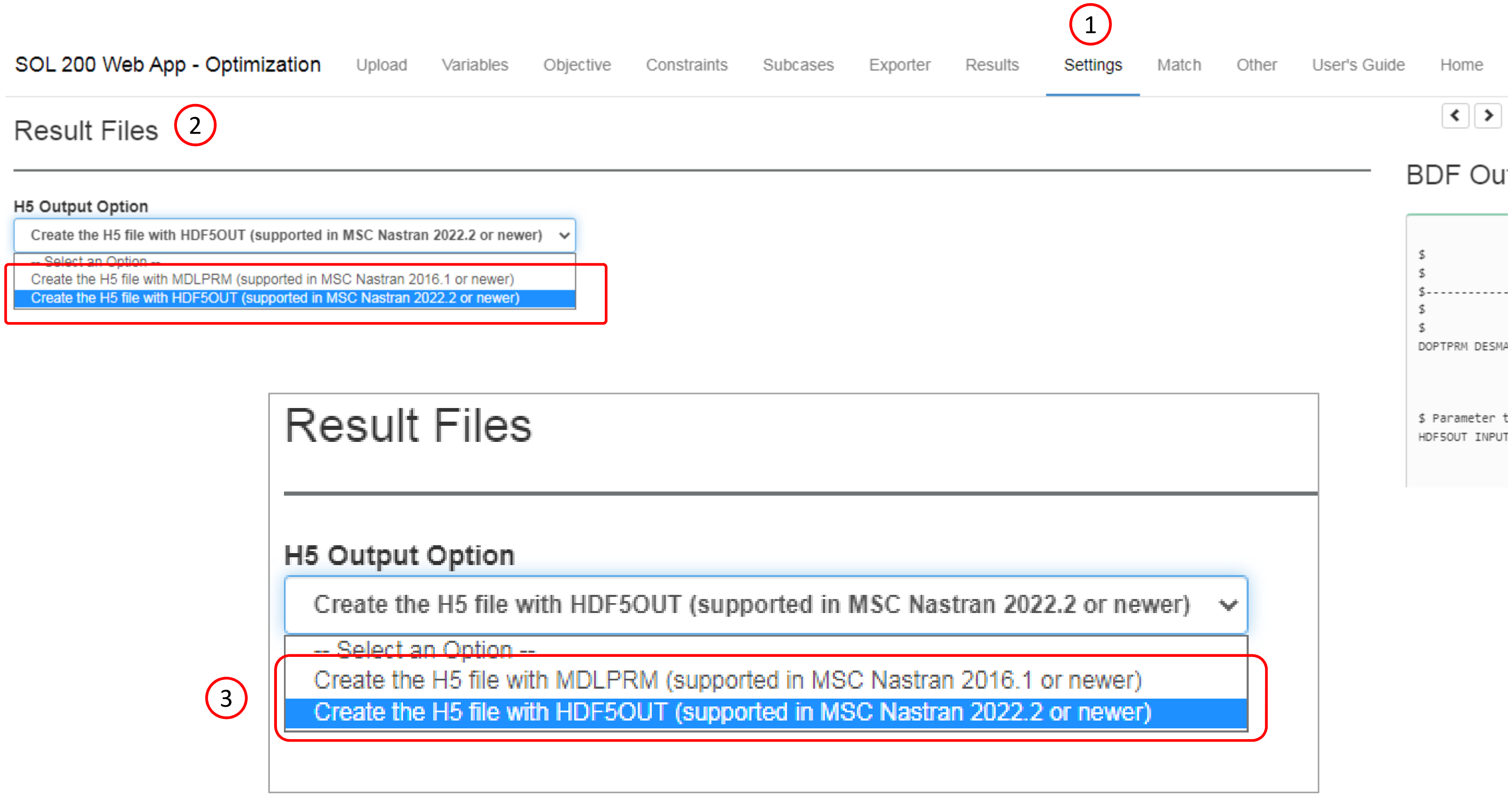
SOL 200 Web App - Optimization | Upload | Variables | Objective | Constraints | Subcases | Exporter | Results | Settings | Match | Other

Optimization Settings

Parameter	Description	Configure
Search	Search	Search
APRCOD	Approximation method to be used	☐ 2 - Mixed Method
CONV1	Relative criterion to detect convergence	☐ Enter a positive real number
CONV2	Absolute criterion to detect convergence	☐ Enter a positive real number
DELX	Fractional change allowed in each design variable during any optimization cycle	☐ Enter a positive real number
DESMAX	Maximum number of design cycles to be performed	☑ 20
DISBEG	Design cycle number for discrete variable processing initiation	☐ Enter a positive integer
GMAX	Maximum constraint violation allowed at the converged optimum	☐ Enter a positive real number
P1	Print items, e.g. objective, design variables, at every n-th design cycle to the .f06 file	☑ 1
P2	Items to be printed to the .f06 file	☑ 15 - Print objective, design variab
TCHECK	Topology Checkerboarding	☐ -1 - Automatic selection (Default)
TDMIN	Minimum diameter of members in topology optimization	☐ Enter a positive real number
TREGION	Trust Region	☑ 1 - Trust Region On

Configure Settings

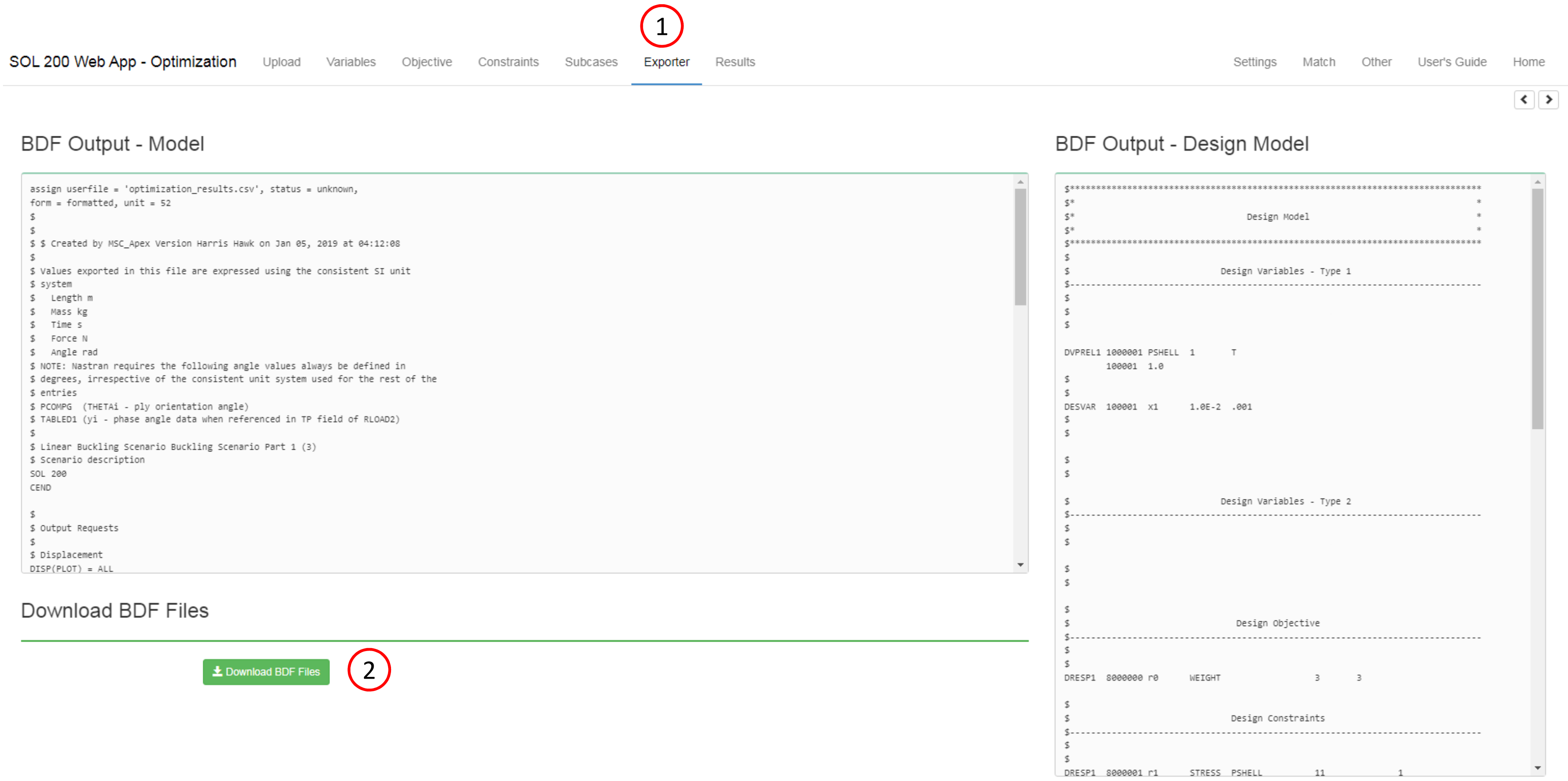
1. Click Settings
2. Scroll to section Result Files
3. Select one of the following H5 output options
    - Create the H5 file with MDLPRM
    - Create the H5 file with HDF5OUT



- The H5 file is used by the Post-processor web app to display MSC Nastran results.
- The H5 file is used by the HDF5 Explorer to create graphs (XY Plots) of MSC Nastran results.

Export New BDF Files

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

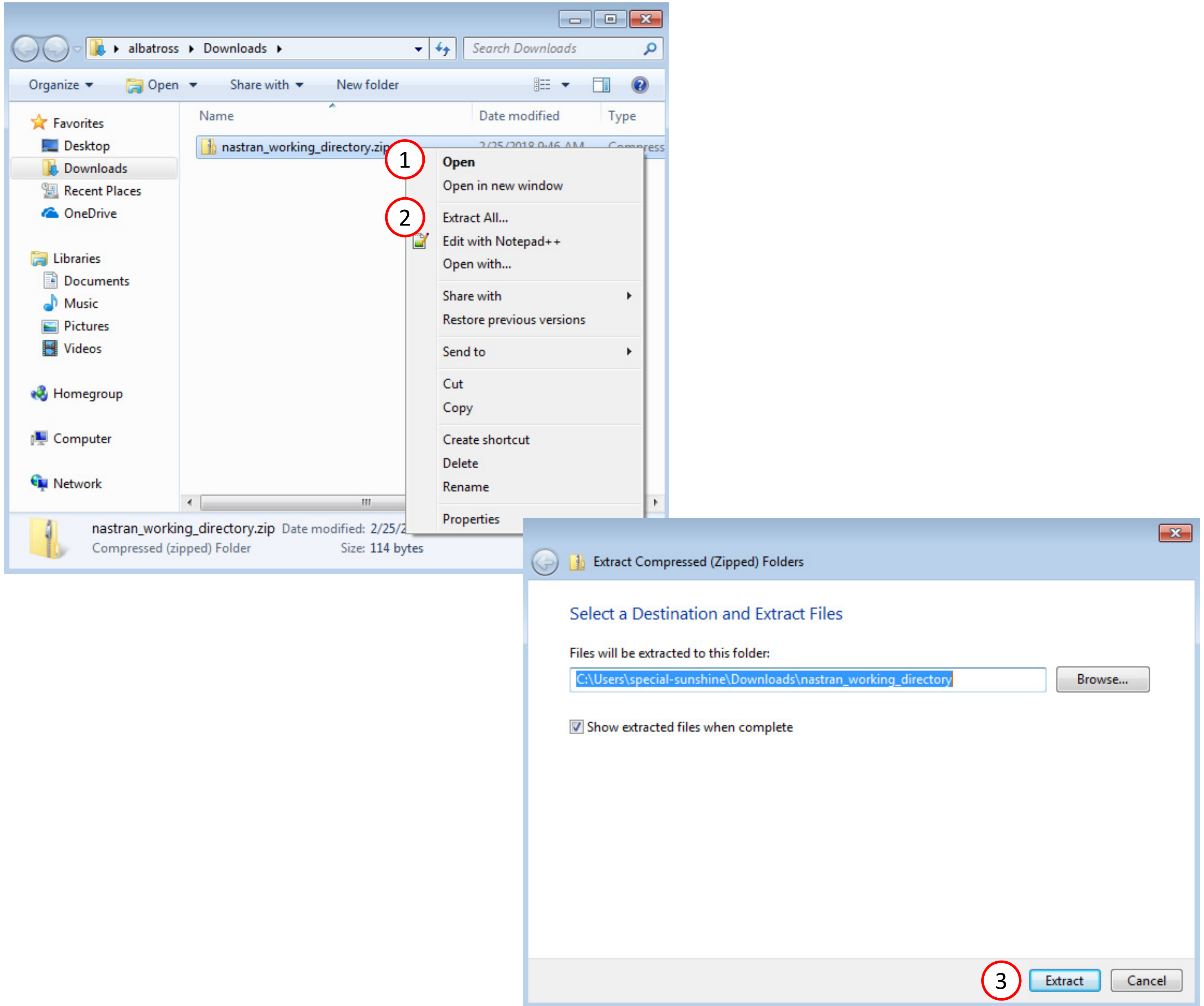


- When the download button is clicked a new file named “nastran_working_directory” is downloaded. If the file already exists in your local folder, the folder name is appended with a number, e.g. “nastran_working_directory (1).zip”

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

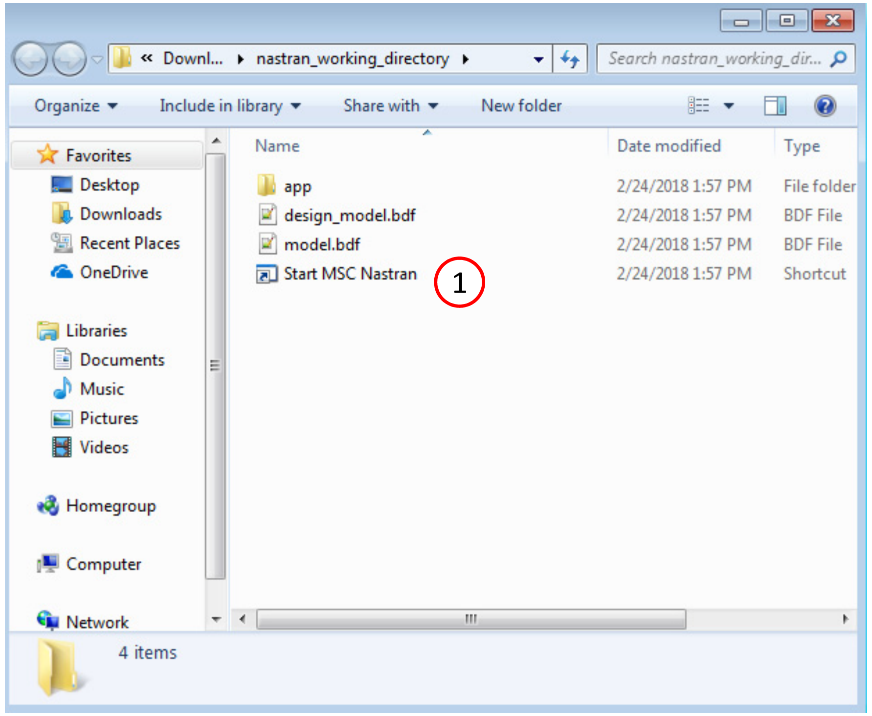
- After a successful optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- One can run the Nastran job on a remote machine as follows:
  1) Copy the BDF files and the INCLUDE files to a remote machine. 2) Run the MSC Nastran job on the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

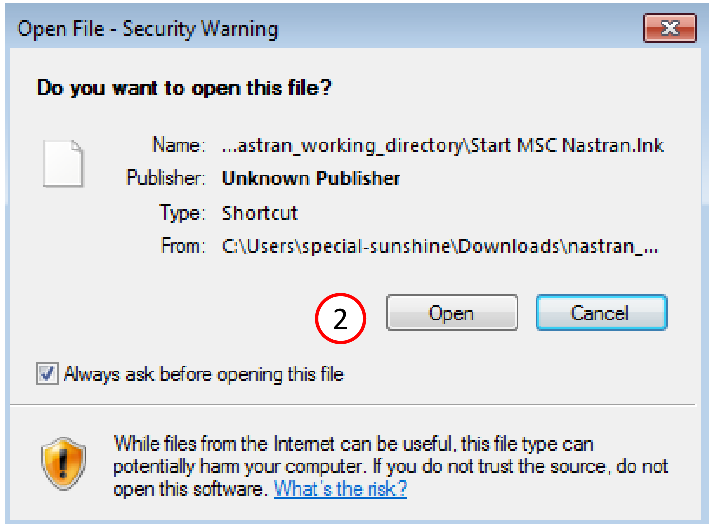
Using Linux?

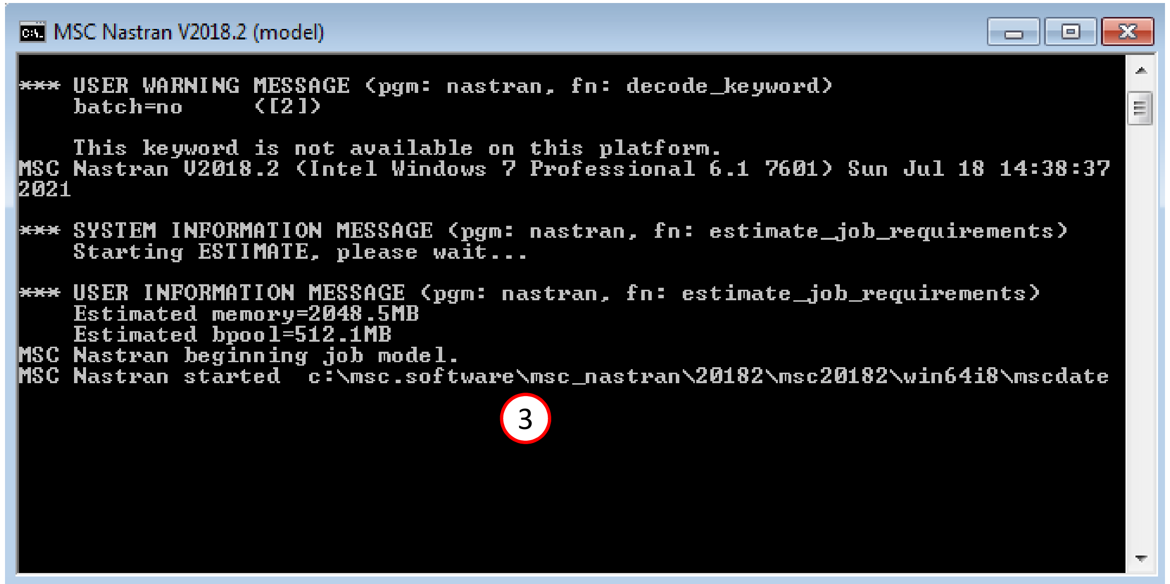
Follow these instructions:
1) Open Terminal
2) Navigate to the nastran_working_directory
cd ./nastran_working_directory
3) Use this command to start the process
./Start_MSC_Nastran.sh

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.

- The reader may realize some of the design cycles have the letter R appended, e.g. 1R, 2R, etc. Since the Trust Region was utilized, some design cycles do not meet certain reliability criteria and are rejected. In the event the Trust Region is used, but an optimum is not successfully achieved, it is recommended to try again with the Trust Region turned off. See the Appendix for more details.

SOL 200 Web App - Local Optimization Results

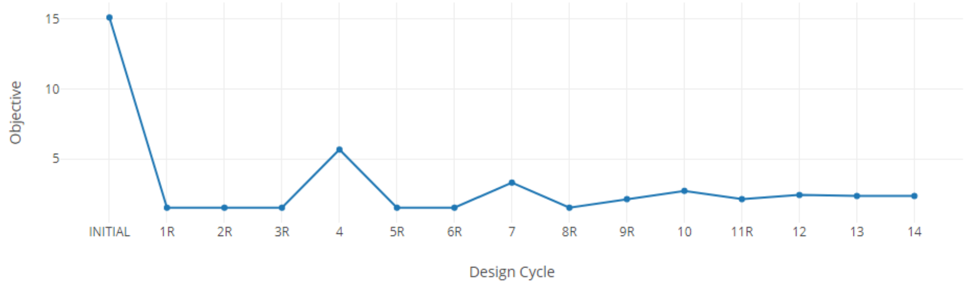
Home

Final Message in .f06

1

RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 14.

Objective

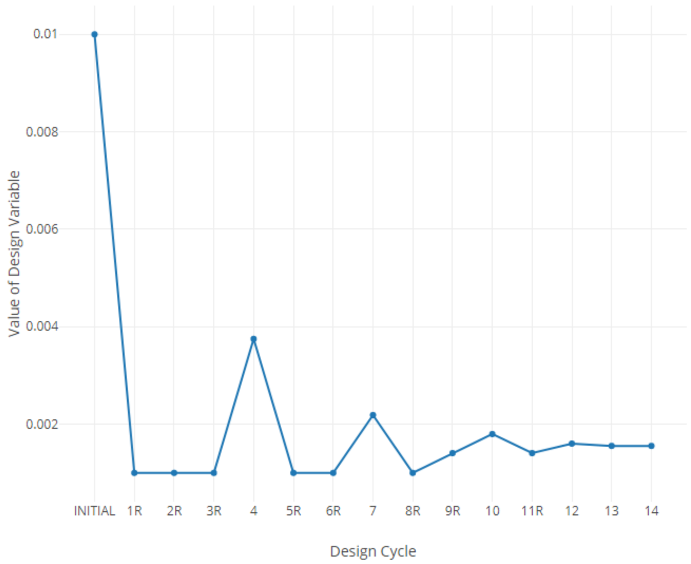


2

Design Variables

Display None | Display All

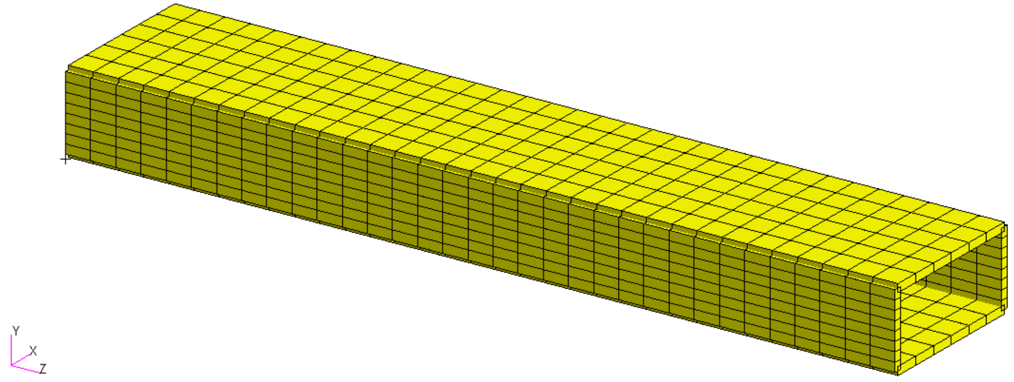
Display	Color	Label	Label Comments
☑		x1	T, Thickness, of PSHELL 1 (Constant_Thickness)



Results

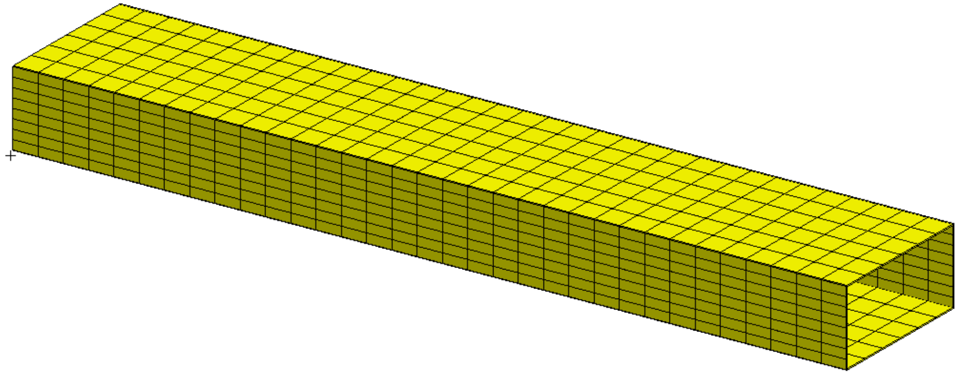
Before Optimization

- Weight: 15.12 kg
- x1 = T, thickness of wall
- = .01 m
- Load Case 1:
  - Buckling Factor 1: -242.19
  - Buckling Factor 2: 242.19



After Optimization

- Weight: 2.35 kg
- x1 = T, thickness of wall
- =.0016 m
- Load Case 1:
  - Buckling Factor 1: -1.0071
  - Buckling Factor 2: 1.0071



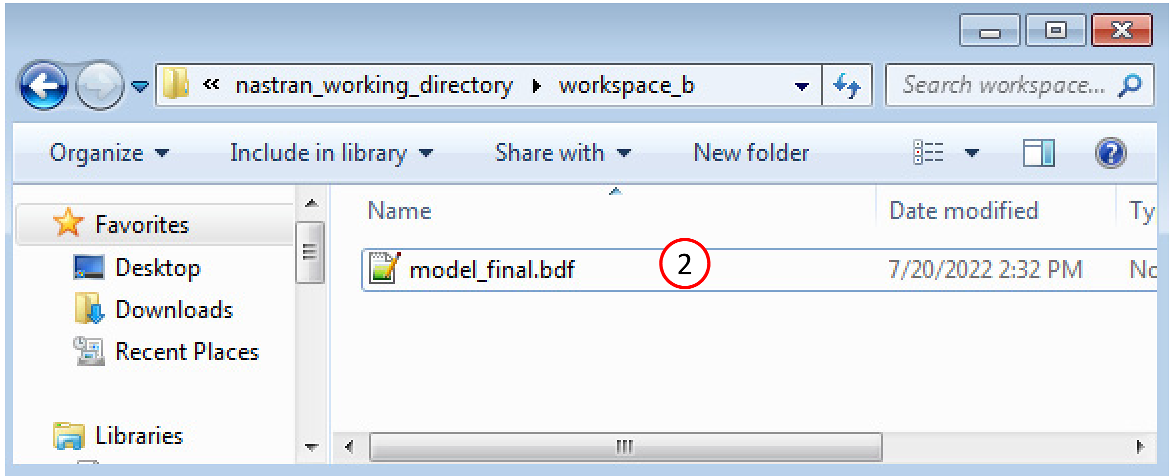
Update the Original Model

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

```
PSHELL   1       1       1.0000-21                       1
RBE3     41022           39921   1    56 1.0     123     38635   38636
+        38637   38638   38639   3     0 38839   38941   38942   38943
+        38944   38945   38946   38947   39445   39446   39447   39448
+        39449   39450   39451   39683   39684   39685   39686   39687
+        39688   39689
RBE3     41023           39922   123456  1.0     123     38635   38636
```

1

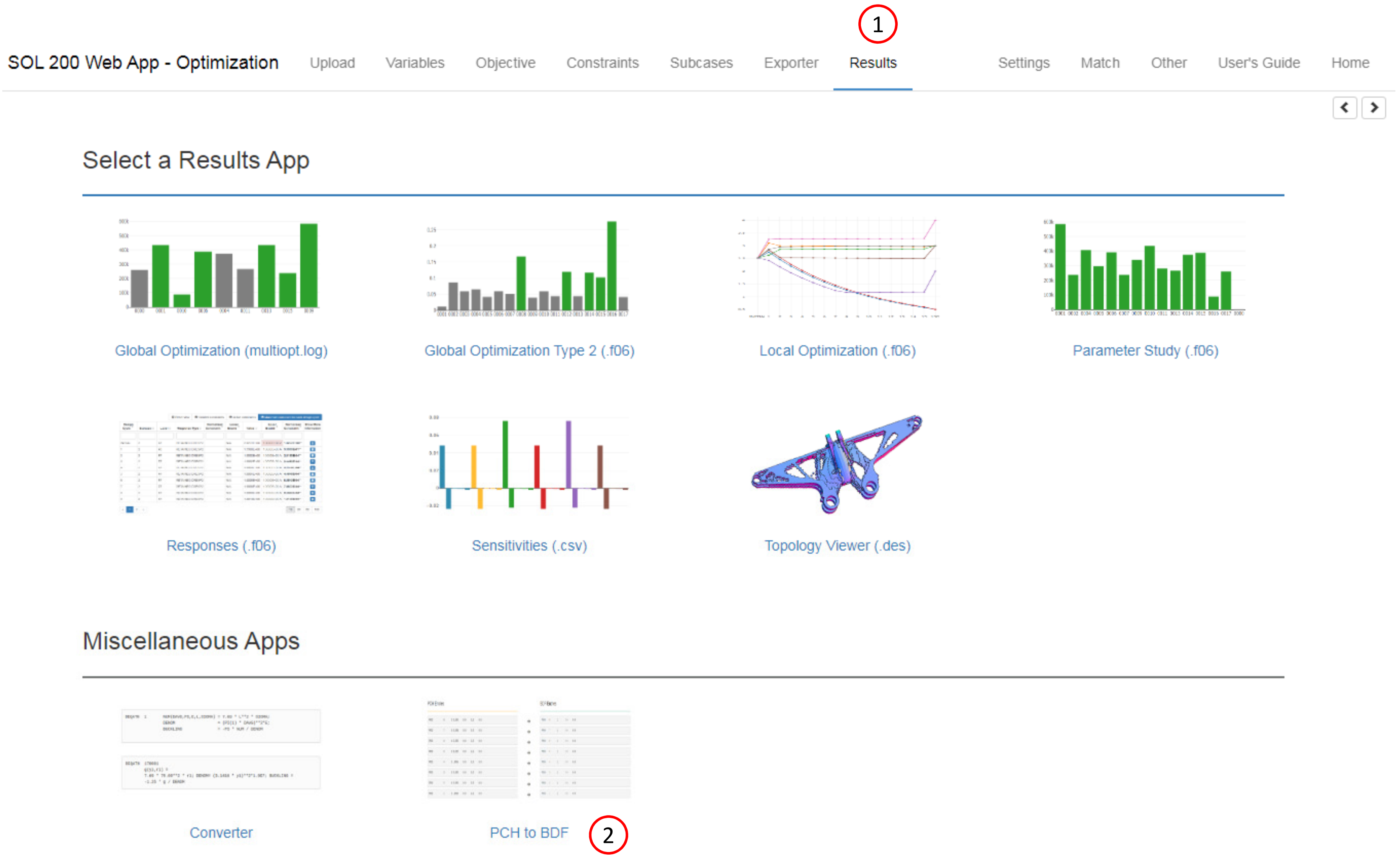
Updated BDF File (model_final.bdf)

```
PSHELL          1       1 .001554       1     1.0        1 .833333    0.0
                              0
RBE3     41022           39921        56 1.0     123     38635   38636
+        38637   38638   38639   38640   38839   38941   38942   38943
+        38944   38945   38946   38947   39445   39446   39447   39448
+        39449   39450   39451   39683   39684   39685   39686   39687
+        39688   39689
```

3

Update the Original Model

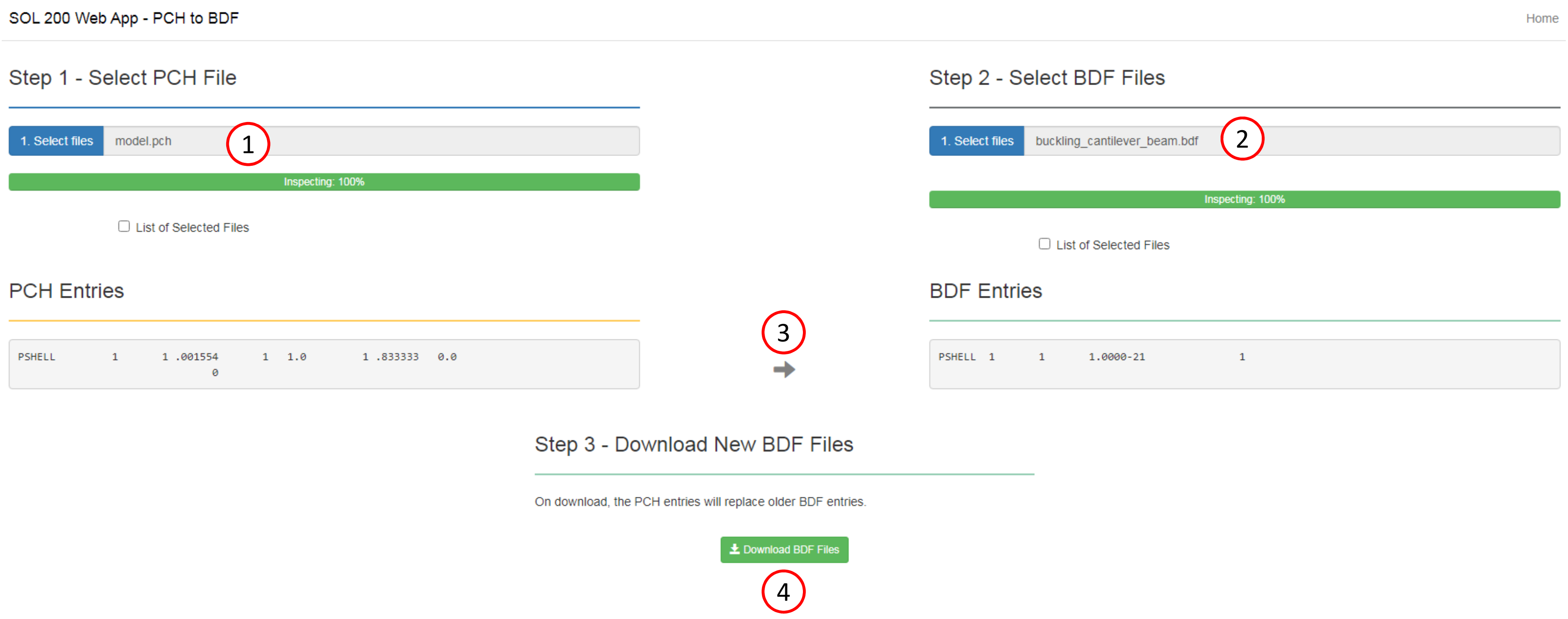
1. Click Results
2. Click PCH to BDF



Update the Original Model

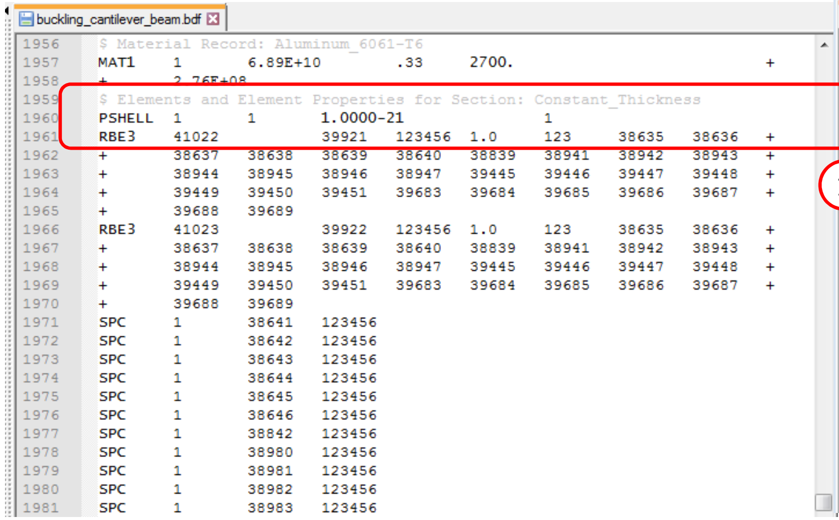
The original .bdf/.dat file has old information about the properties. The properties will be updated.

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



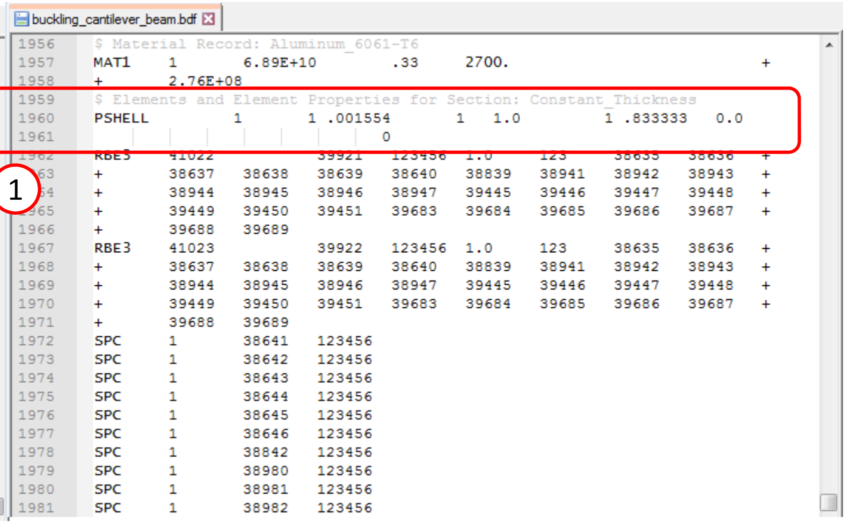
Update the Original Model

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



```
buckling_cantilever_beam.bdf
$ Material Record: Aluminum_6061-T6
MAT1    1       6.89E+10        .33     2700.                                   +
+       2.76E+08
$ Elements and Element Properties for Section: Constant_Thickness
PSHELL  1       1       1.0000-21               1
RBE3    41022           39921   123456  1.0     123     38635   38636   +
+       38637   38638   38639   38640   38839   38941   38942   38943   +
+       38944   38945   38946   38947   39445   39446   39447   39448   +
+       39449   39450   39451   39683   39684   39685   39686   39687   +
+       39688   39689
RBE3    41023           39922   123456  1.0     123     38635   38636   +
+       38637   38638   38639   38640   38839   38941   38942   38943   +
+       38944   38945   38946   38947   39445   39446   39447   39448   +
+       39449   39450   39451   39683   39684   39685   39686   39687   +
+       39688   39689
SPC     1       38641   123456
SPC     1       38642   123456
SPC     1       38643   123456
SPC     1       38644   123456
SPC     1       38645   123456
SPC     1       38646   123456
SPC     1       38842   123456
SPC     1       38980   123456
SPC     1       38981   123456
SPC     1       38982   123456
SPC     1       38983   123456
```

Original BDF/DAT File



```
buckling_cantilever_beam.bdf
$ Material Record: Aluminum_6061-T6
MAT1    1       6.89E+10        .33     2700.                                   +
+       2.76E+08
$ Elements and Element Properties for Section: Constant_Thickness
PSHELL         1       1 .001554       1     1.0       1 .833333     0.0
                              0
RBE3    41022           39921   123456  1.0     123     38635   38636   +
+       38637   38638   38639   38640   38839   38941   38942   38943   +
+       38944   38945   38946   38947   39445   39446   39447   39448   +
+       39449   39450   39451   39683   39684   39685   39686   39687   +
+       39688   39689
RBE3    41023           39922   123456  1.0     123     38635   38636   +
+       38637   38638   38639   38640   38839   38941   38942   38943   +
+       38944   38945   38946   38947   39445   39446   39447   39448   +
+       39449   39450   39451   39683   39684   39685   39686   39687   +
+       39688   39689
SPC     1       38641   123456
SPC     1       38642   123456
SPC     1       38643   123456
SPC     1       38644   123456
SPC     1       38645   123456
SPC     1       38646   123456
SPC     1       38842   123456
SPC     1       38980   123456
SPC     1       38981   123456
SPC     1       38982   123456
```

Downloaded BDF/DAT File

1

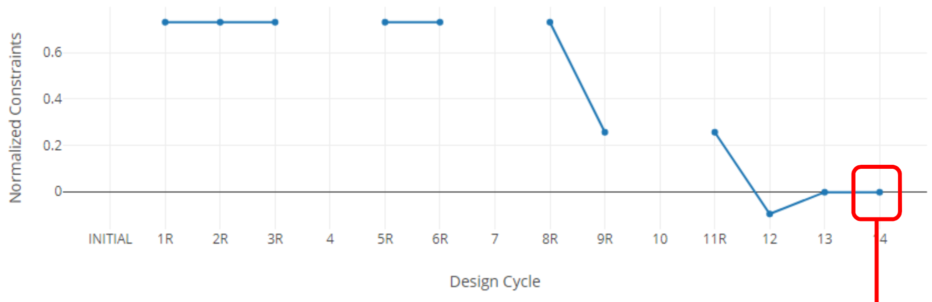
Inspection of MSC Nastran Results with the Post-processor Web App

Normalized Constraints

- All constraints are normalized. For each design cycle, the maximum normalized constraint (NC) is reported in the Normalized Constraints plot.
- The Responses web app is used to inspect the corresponding response for each maximum normalized constraint value.
  - For the final design, the maximum NC is -.0028102 and corresponds to a buckling load factor of 1.0028.

Normalized Constraints

+ Info



SOL 200 Web App - Responses

Home

Responses

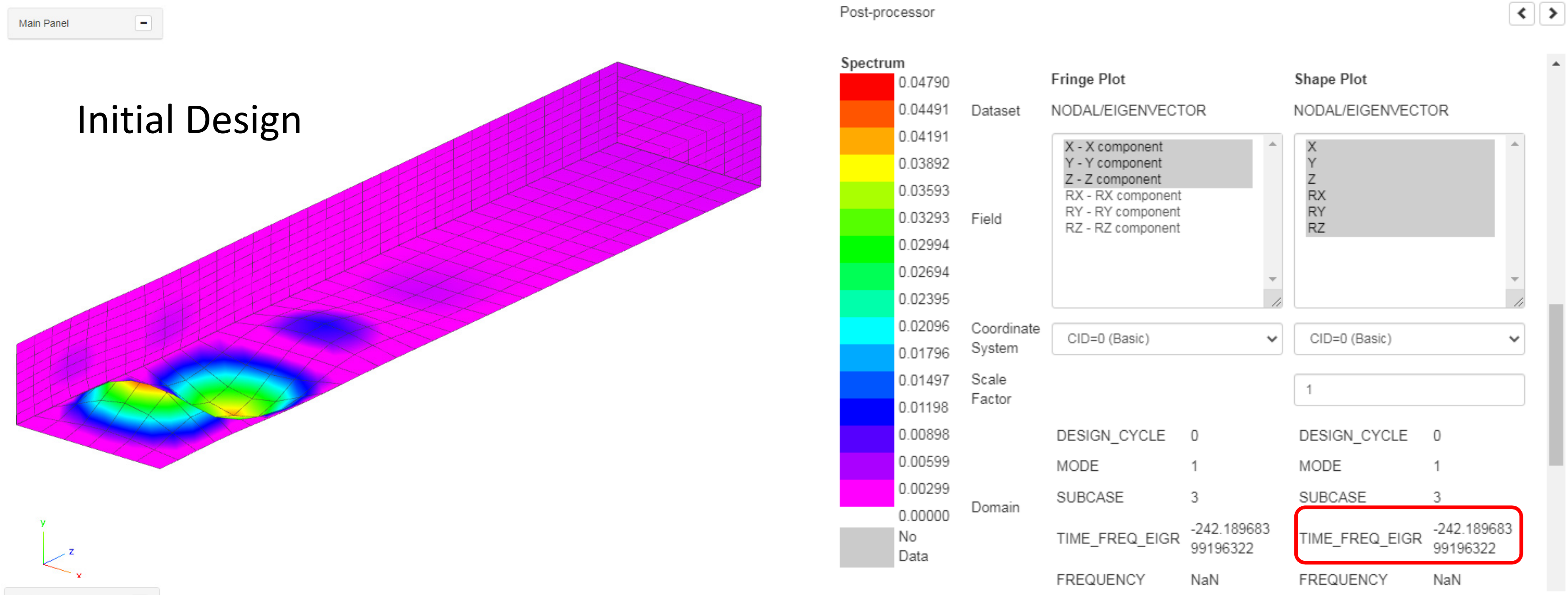
Reset view | Violated constraints | Active constraints | Maximum constraint for each design cycle

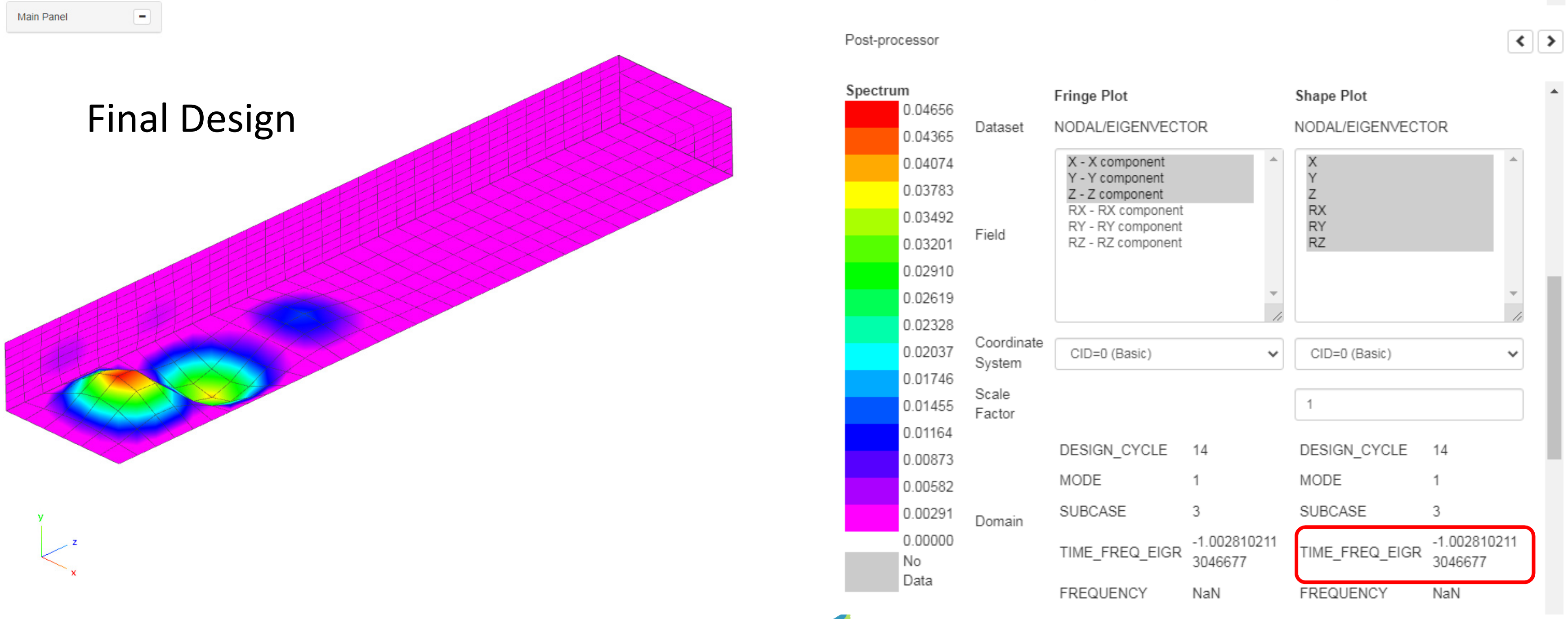
Design Cycle	Subcase	Label	Response Type	Normalized Constraint	Lower Bound	Value	Upper Bound	Normalized Constraint	Show More Information	designCycleNumber	DRESP2 ID	EQUATION ID
12	3	R2	RETAINED DRESP2	**-9.6814E-02****	1.0000E+00	1.0968E+00	N/A			12	9000002	170002
13	3	R2	RETAINED DRESP2	**-2.8102E-03****	1.0000E+00 A	1.0028E+00	N/A			13	9000002	170002
FINAL - 14	3	R2	RETAINED DRESP2	**-2.8102E-03****	1.0000E+00 A	1.0028E+00	N/A			14	9000002	170002

Post-processor Web App

- The Post-processor web app is used to inspect the MSC Nastran results.
- The buckling load factor (BLF) is the eigenvalue.
- Consider subcase 3, mode 1.
    - For the initial design, the eigenvalue, or BLF, is -242.1896.
    - For the final design, the eigenvalue, or BLF, is -1.0028.

- If F is the load applied, then the buckling load F_b is equal to F * BLF. A negative buckling load factor indicates buckling occurs when the load is reversed, i.e. -F_b = F * -BLF.
- Refer to the Post-processor web app tutorials to learn more about MSC Nastran results.



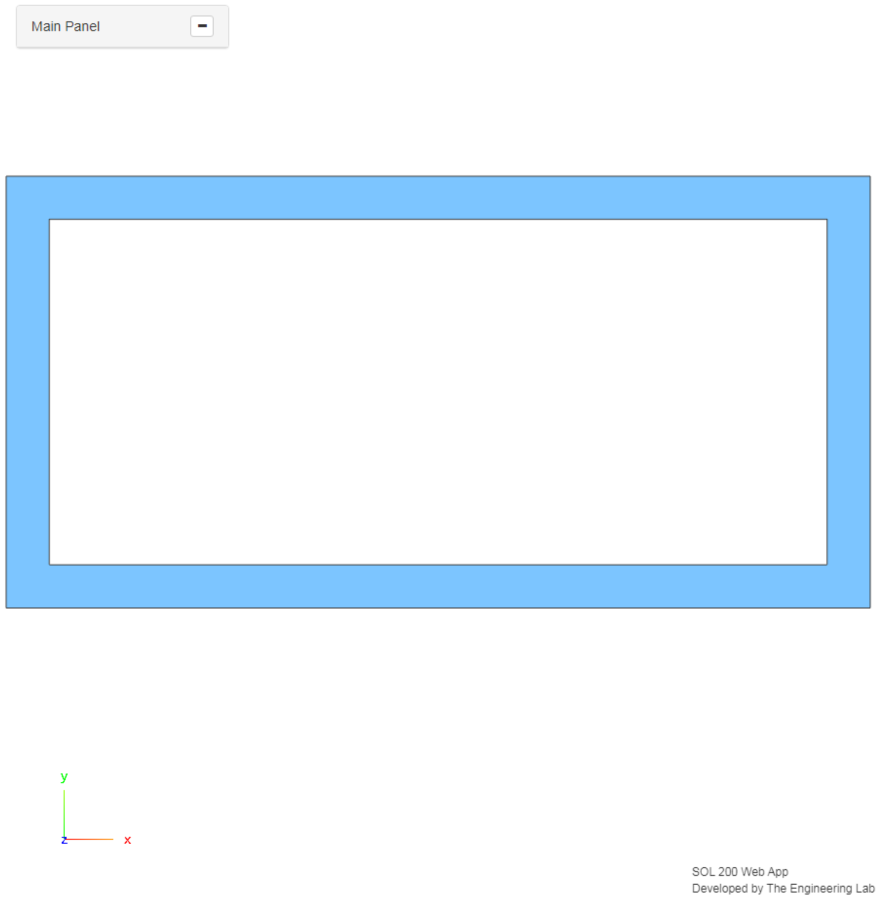


Post-processor Web App

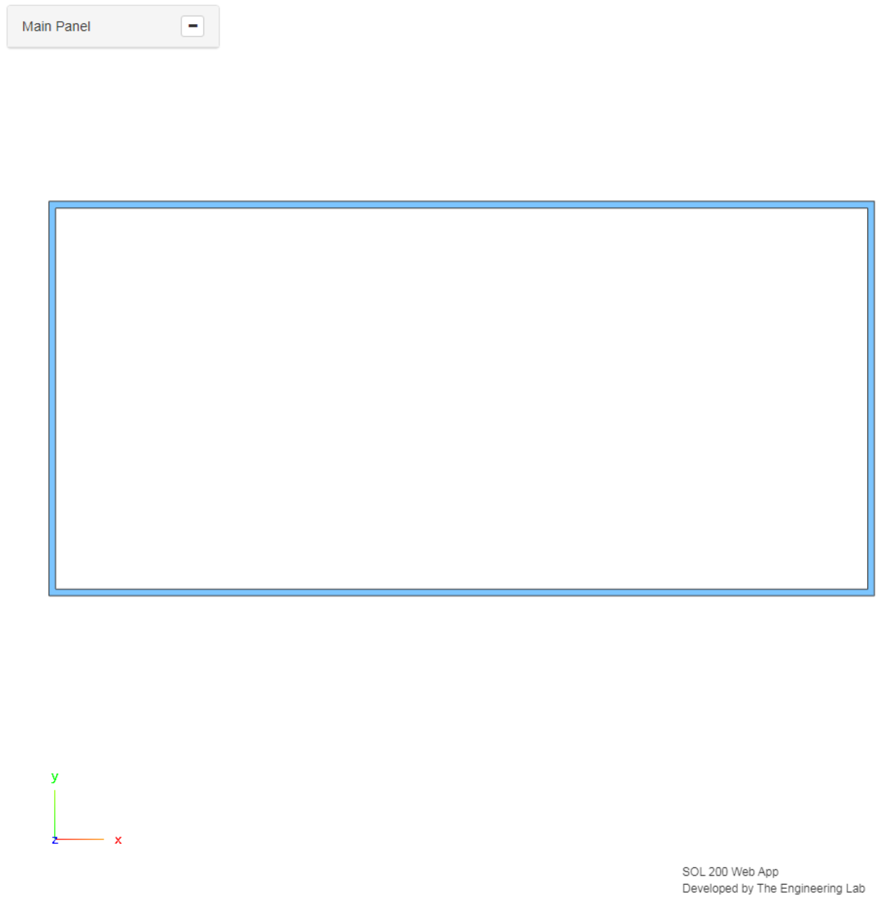
- The PSHELL thicknesses of the initial and final design are compared.

- Refer to the Post-processor web app tutorials to learn more about MSC Nastran results.

Thickness of PSHELL



Initial Design



Final Design

End of Tutorial

Appendix

Appendix Contents

- Frequently Asked Question
  - What is the trust region?
- An example where the Trust Region can be used
- Trust Region Visualized
- Considerations for Optimization with Buckling Constraints

Frequently Asked Questions

Question:

- The trust region was used in the tutorial.
- What is the trust region?

1 Settings User's Guide Home

BDF Output - Design Model

```
$
$                              Optimization Control Settings
$------------------------------------------------------------------------------------------
$
$
DOPTPRM DESMAX  40      P1      1       P2      15      TREGION 1
```

Optimization Settings

Parameter	Description	Configure
APRCOD	Approximation method to be used	2 - Mixed Method
CONV1	Relative criterion to detect convergence	Enter a positive real number
CONV2	Absolute criterion to detect convergence	Enter a positive real number
DELX	Fractional change allowed in each design variable during any optimization cycle	Enter a positive real number
DESMAX	Maximum number of design cycles to be performed	40
DISBEG	Design cycle number for discrete variable processing initiation	Enter a positive integer
GMAX	Maximum constraint violation allowed at the converged optimum	Enter a positive real number
P1	Print items, e.g. objective, design variables, at every n-th design cycle to the .f06 file	1
P2	Items to be printed to the .f06 file	15 - Print objective, design variab
TREGION	Trust Region	2 1 - Trust Region On

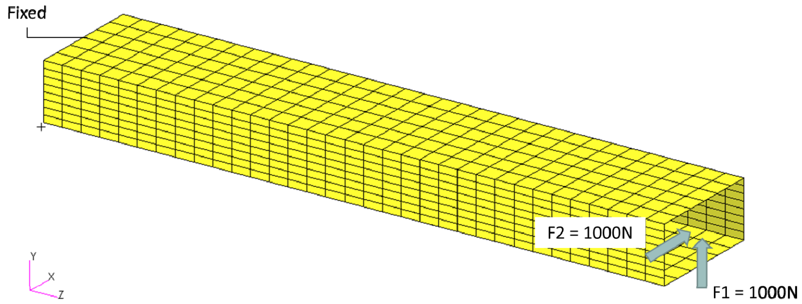
Frequently Asked Questions

Answer:

- When the Trust Region is used, the Move Limits after each design cycle are automatically increased or decreased.
- The Move Limits dictate the degree to which design variables can change during a design cycle.
- With each design cycle, a new design is found through approximate optimization and then validated by a subsequent finite element analysis. The Move Limits impact the reliability of approximate optimization.
- There are 2 benefits to using the Trust Region:
  - The Trust Region can enable faster optimizations for some, but not all optimization problems. This is achieved because the Trust Region can reduce costly finite element analysis and depend more on approximate optimization.
  - The Trust Region can help avoid scenarios where large design variable changes suddenly cause feasible designs to become infeasible designs. This is sometimes indicated by the the message: “BEST COMPROMISE TO INFEASIBLE DESIGN.”

An example where the Trust Region can be used

Consider the example of this workshop.



After design cycle 1, note the following:

- The thickness design variable has gone from .01 to .001.
- The normalized constraint has gone from -1.0 to .7 and is a change from feasible to infeasible.
- The optimization terminated with the message: RUN TERMINATED DUE TO HARD CONVERGENCE TO A BEST COMPROMISE INFEASIBLE DESIGN

Inaccuracy in Approximate Model

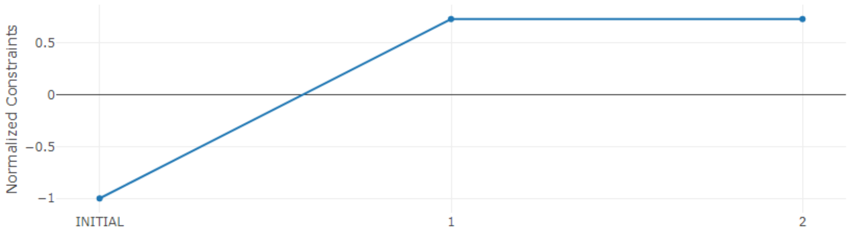
- See the F06 output on the following slide.
- For Design 1, there is a large discrepancy between the structural results from the Approximate Model and an actual Finite Element Analysis. Normally, the discrepancies are small. This large difference is likely due to a large change in the design variable from .01 to .001.

SOL 200 Web App - Local Optimization Results

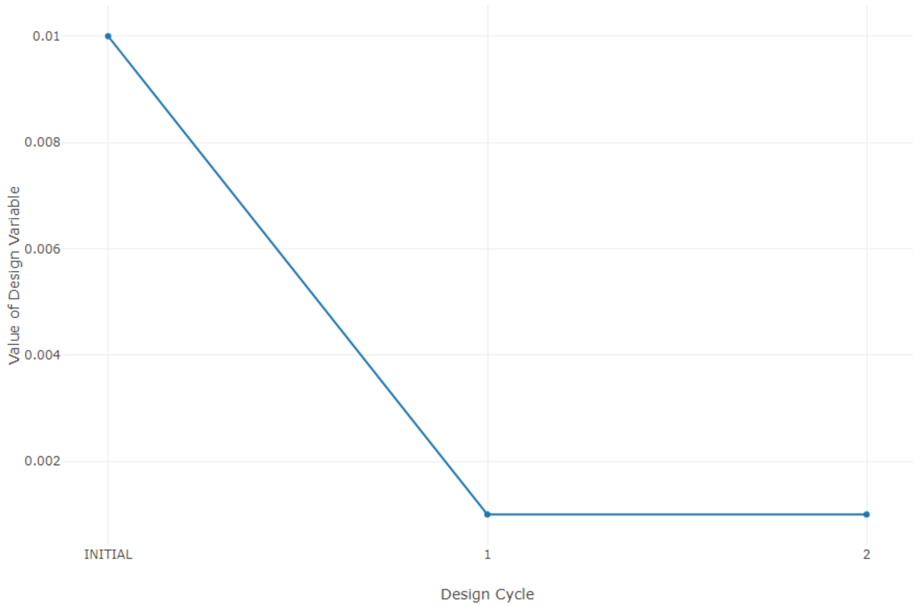
Final Message in .f06

RUN TERMINATED DUE TO HARD CONVERGENCE TO A BEST COMPROMISE INFEASIBLE DESIGN AT CYCLE NUMBER = 2.

Normalized Constraints

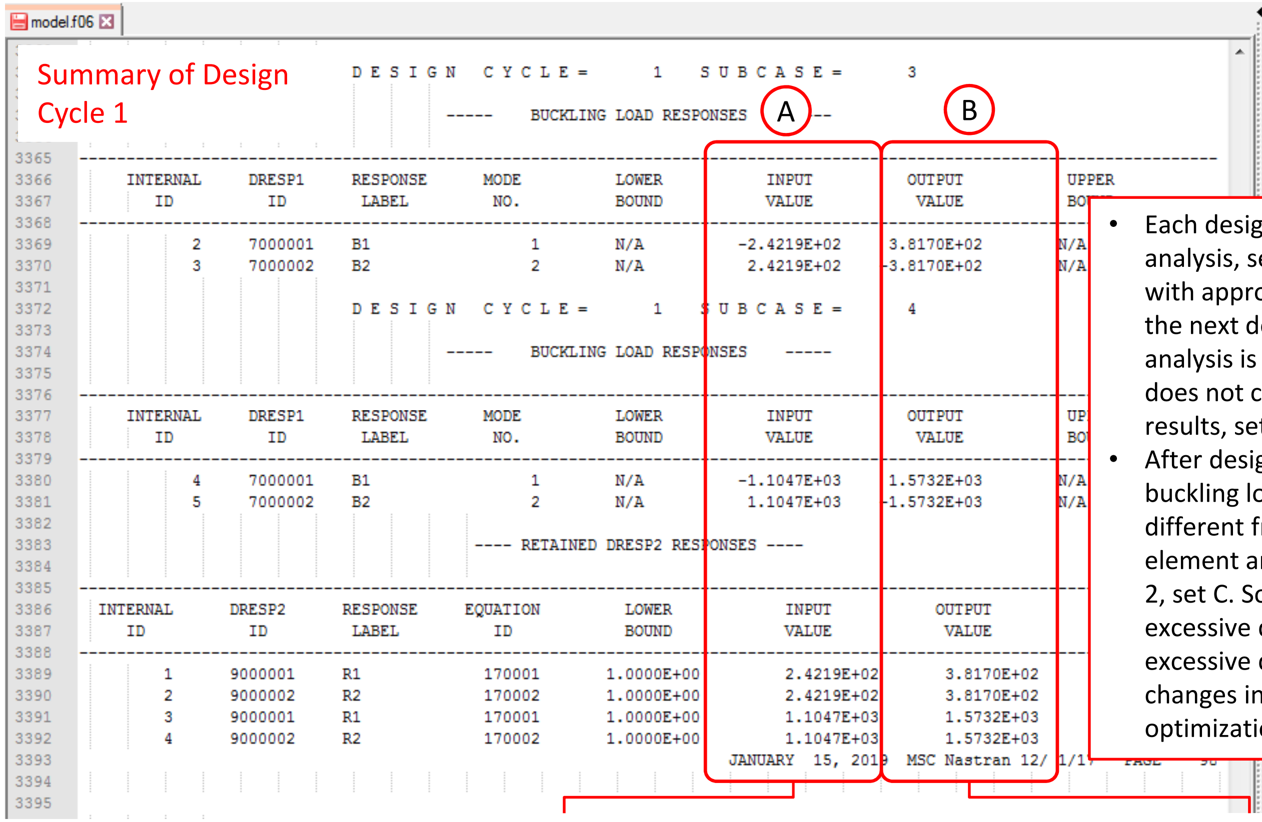


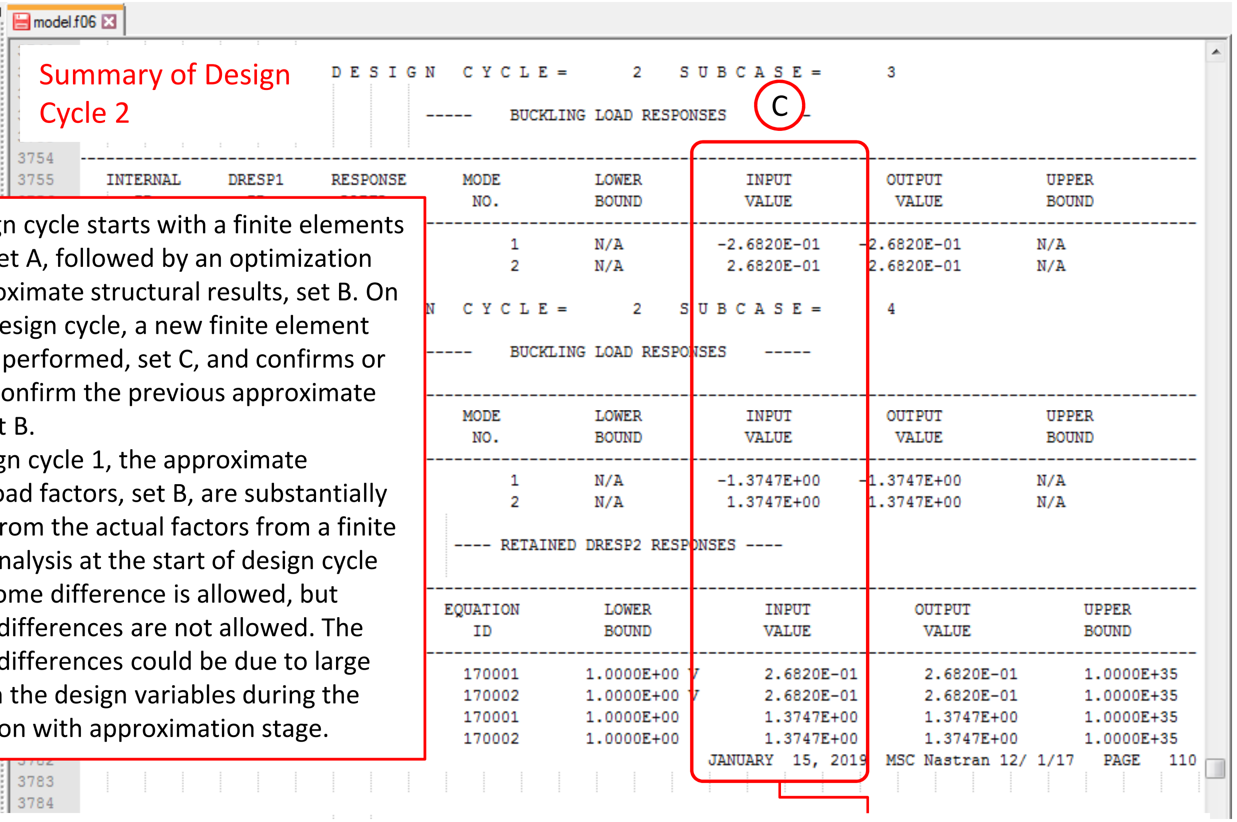
Design Variables



An example where the Trust Region can be used, Continued

F06 Output, Design Cycle Summaries Shown





- Each design cycle starts with a finite elements analysis, set A, followed by an optimization with approximate structural results, set B. On the next design cycle, a new finite element analysis is performed, set C, and confirms or does not confirm the previous approximate results, set B.
- After design cycle 1, the approximate buckling load factors, set B, are substantially different from the actual factors from a finite element analysis at the start of design cycle 2, set C. Some difference is allowed, but excessive differences are not allowed. The excessive differences could be due to large changes in the design variables during the optimization with approximation stage.

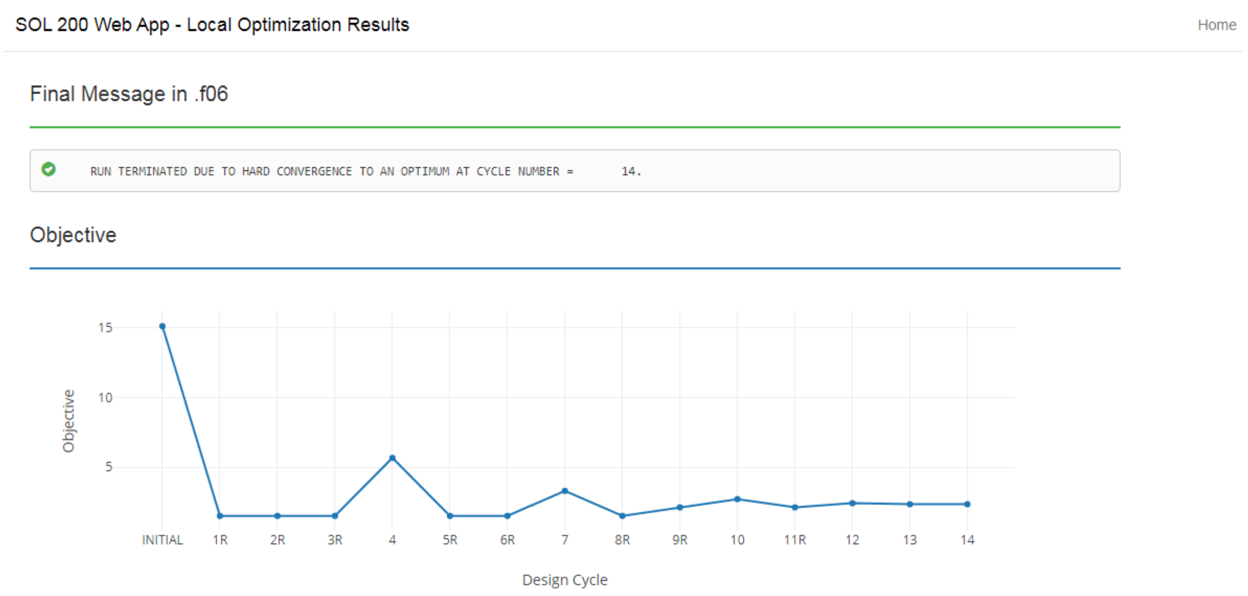
Structural Results of Design 0 based on Finite Element Analysis

Structural Results of Design 1 based on Approximate Model

Structural Results of Design 1 based on Finite Element Analysis

An example where the Trust Region can be used, Continued

After employing the Trust Region, an optimum is successfully obtained



Trust Region Visualized

The following slides are based on the optimization example shown below, and serve to visually depict the behavior of the Trust Region. This example is found in the MSC Nastran Design Sensitivity and Optimization User's Guide.

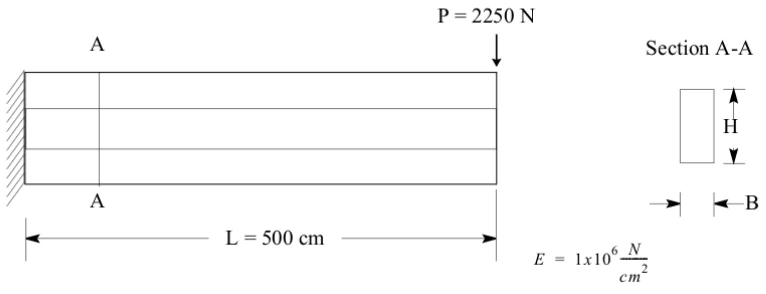


Figure 1-6 Cantilever Beam

The optimization problem statement could be written as follows

minimize

$$V = B \cdot H \cdot L$$

subject

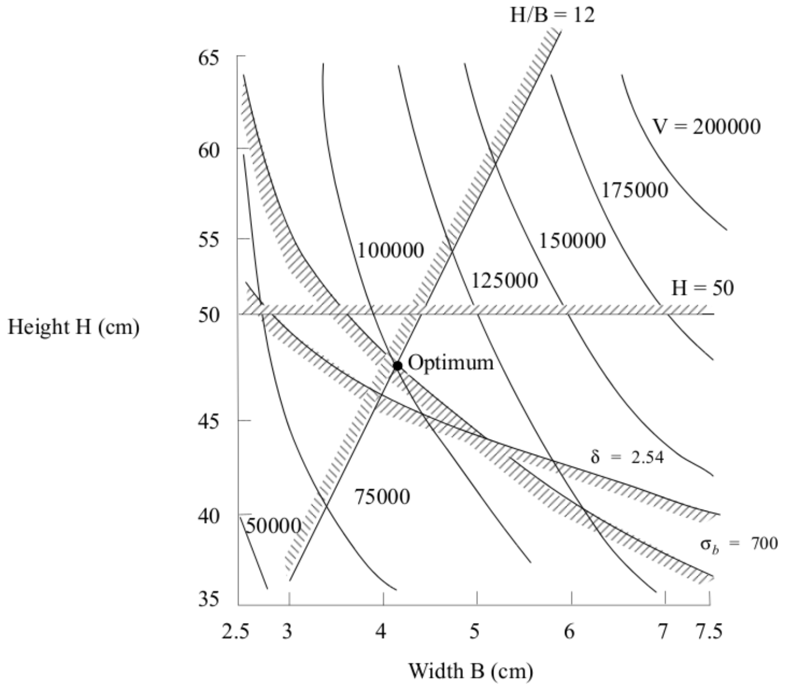
$$\sigma = \frac{Mc}{I} = \frac{6PL}{BH^2} \le 700 \frac{N}{cm^2}$$

$$\delta = \frac{PL^3}{3EI} = \frac{4PL^3}{BH^3E} \le 2.54 cm$$

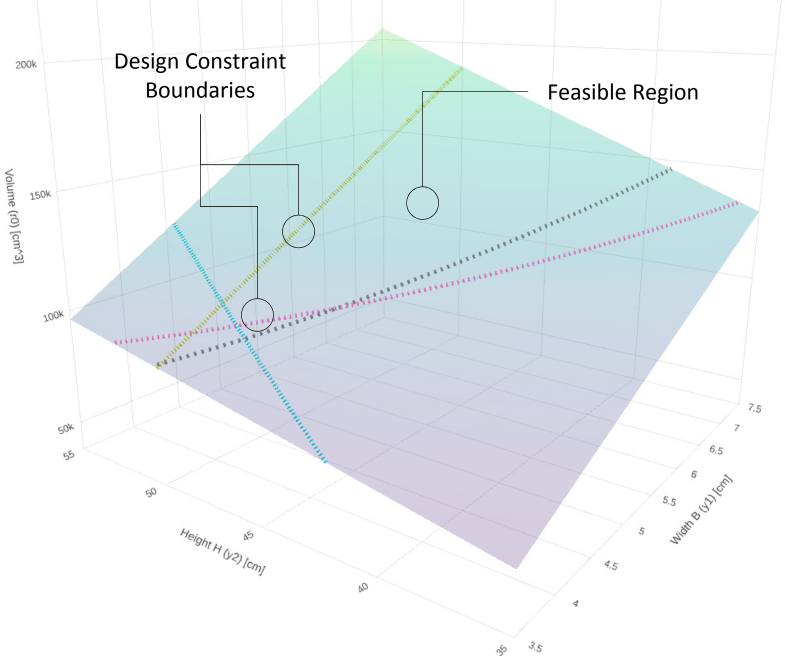
$$\frac{H}{B} \le 12$$

$$1 \le B \le 20$$

$$20 \le H \le 50$$



Design Space – 2D View

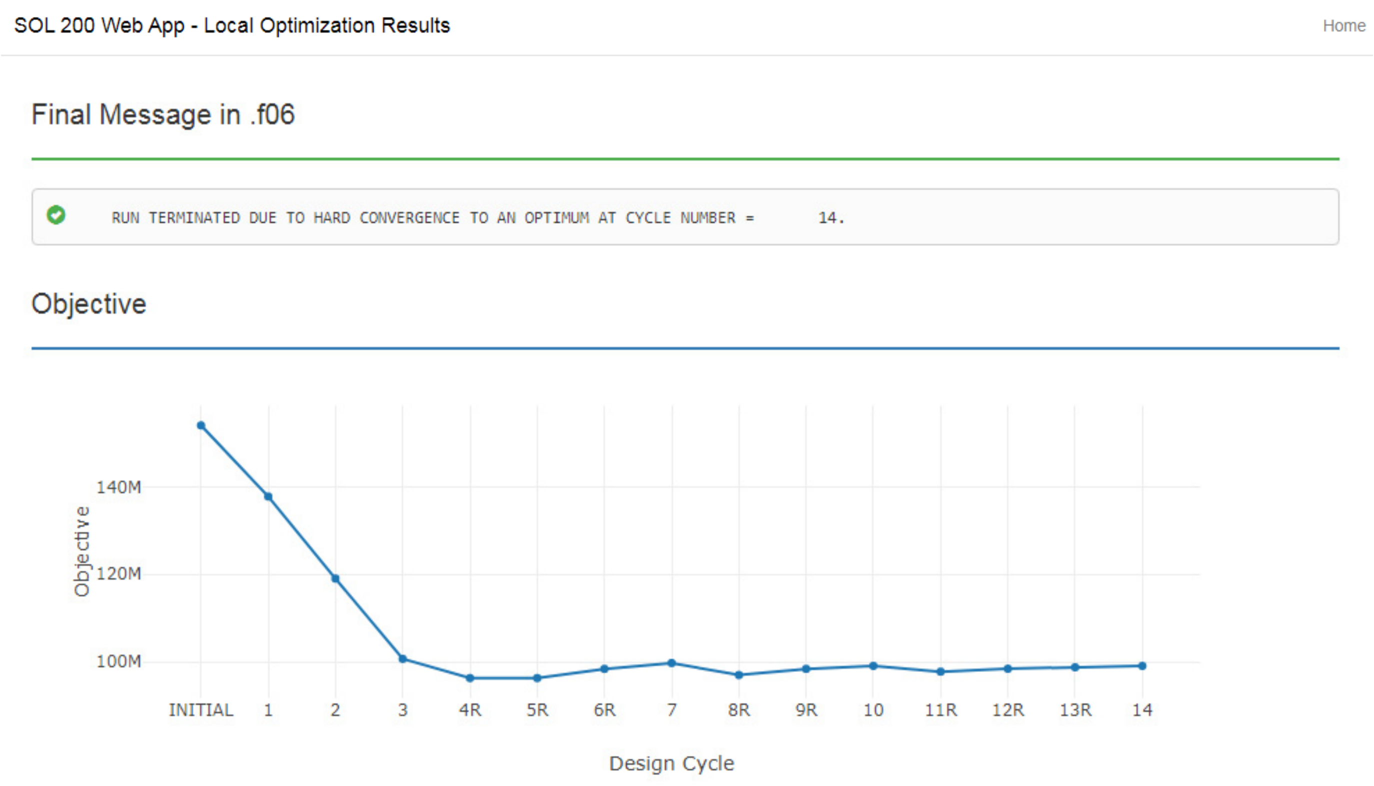


Design Space – 3D View

Trust Region Visualized, Continued
Local Optimization Results

When the Trust Region is used, the Local Optimization Results will show some design cycles marked with the "R" label.

The "R" labels indicate that the design cycle has been rejected. This usually occurs when the move limits are so large that the constraints are violated. The design cycle is rejected, a new design cycle with decreased move limits is performed and the design cycle is accepted if the constraints are within limits.



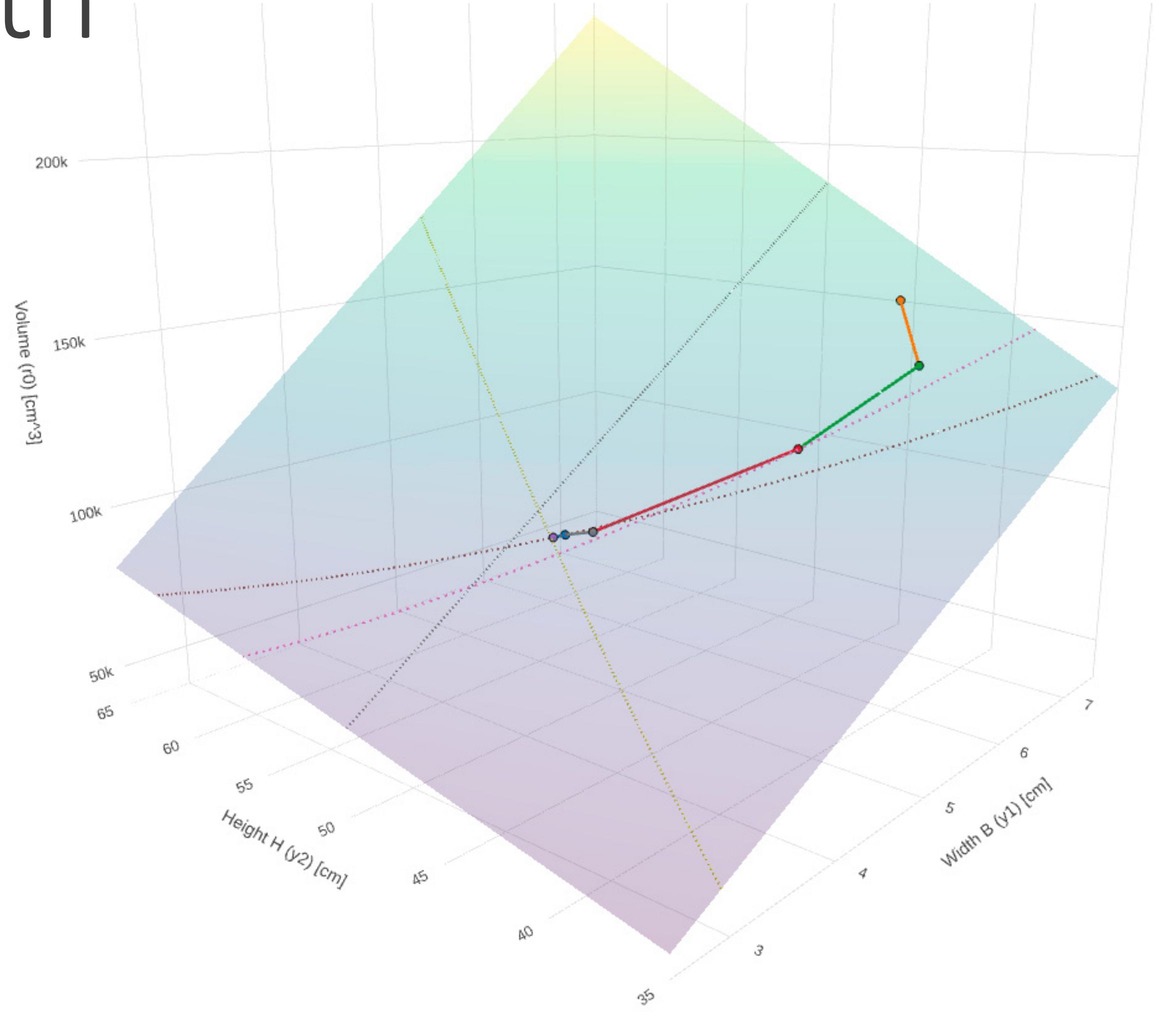
Trust Region Visualized, Continued Final Optimization Path

It should be noted that the final optimization path consists of the following design cycles:

- INITIAL
- 1
- 2
- 3
- 7
- 10
- 14

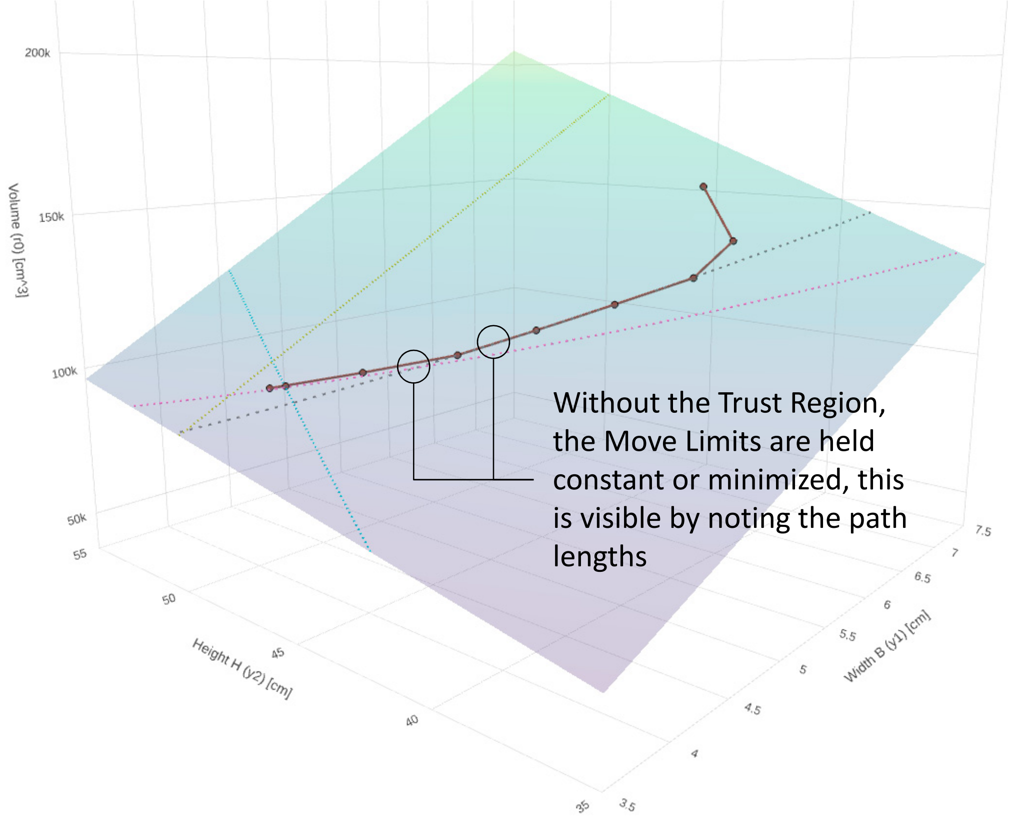
The following rejected design cycles are not part of the final optimization path and are not shown in the image to the right:

- 5R
- 6R
- 8R
- 9R
- 11R
- 12R
- 13R



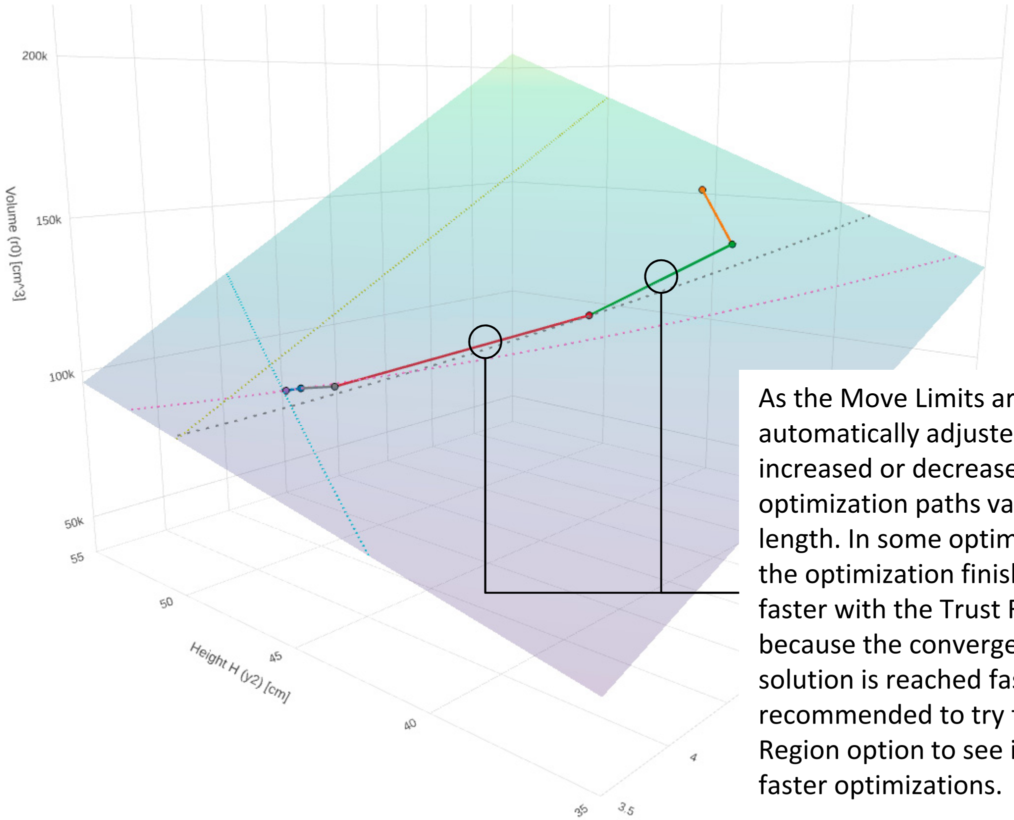
Trust Region Visualized, Continued Final Optimization Path

Trust Region Not Used



Without the Trust Region, the Move Limits are held constant or minimized, this is visible by noting the path lengths

Trust Region Used



As the Move Limits are automatically adjusted, increased or decreased, the optimization paths vary in length. In some optimizations, the optimization finishes faster with the Trust Region because the converged solution is reached faster. It is recommended to try the Trust Region option to see if it yields faster optimizations.

Considerations for Optimization with Buckling Constraints

Up and Down Behavior During Optimization

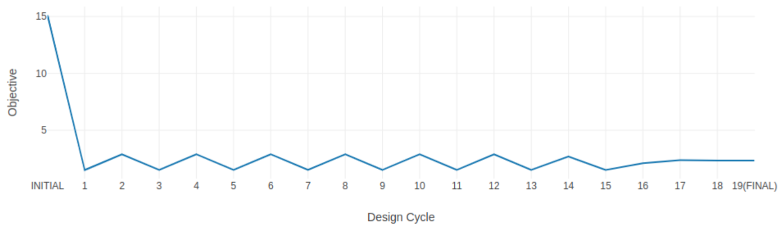
When working with buckling load factors during an optimization, the results may appear to have an *up and down* behavior.

The following slides discuss more information to help understand this up and down behavior.

Final Message in .f06

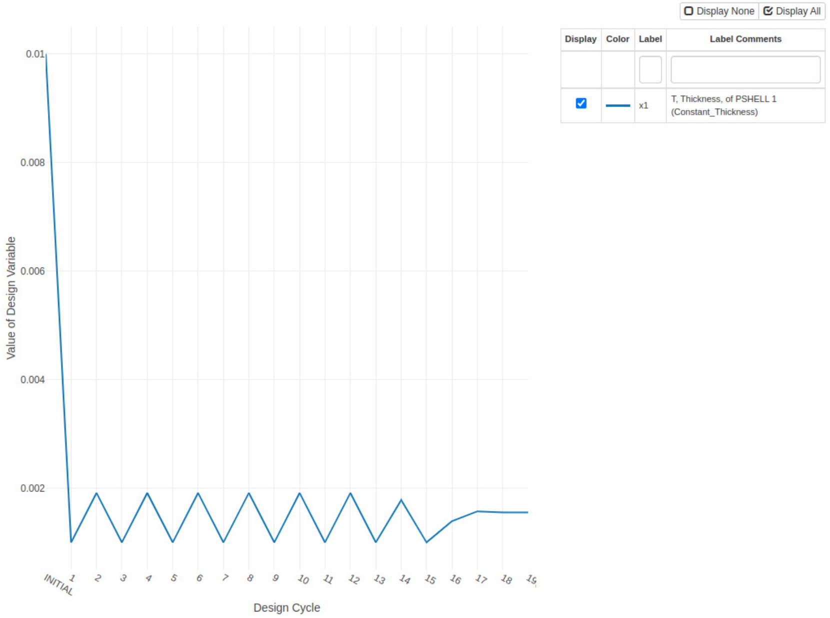
RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 19.

Objective

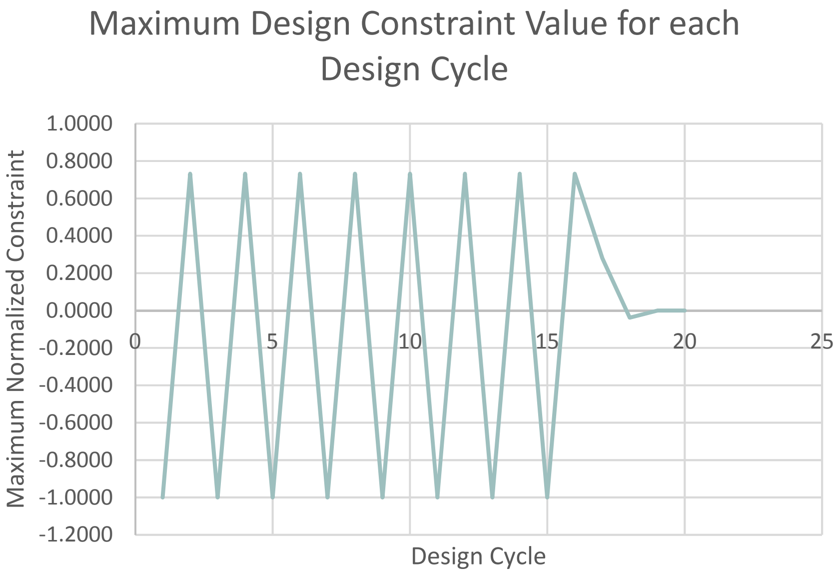


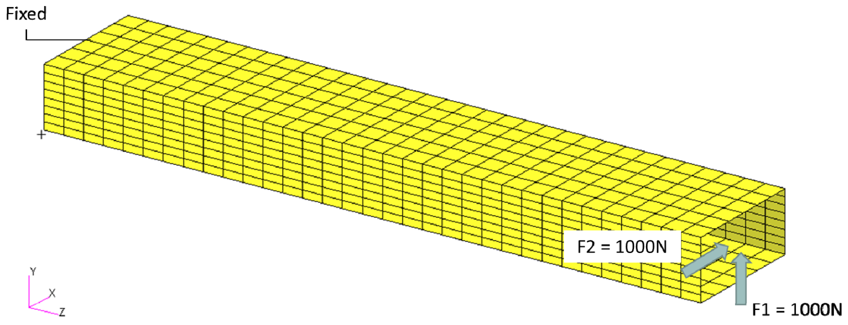
Objective

Design Variables



Design Variables





Optimization Problem Statement

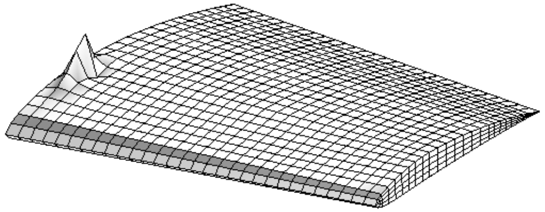
- Objective: minimize mass
- Constrain the absolute value of buckling load factor (1.0 < ABS(BLF))
- Vary the thickness of the walls
- Trust region off and not used

Response Surface of Buckling Load Factor

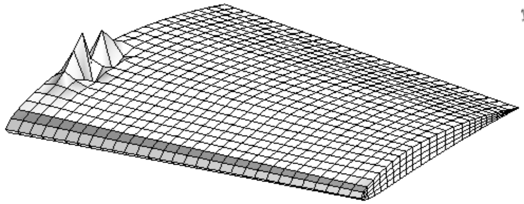
The response surface for the buckling load factor is sometimes partially rough or discontinuous.

The buckling load factor (BLF) is equivalent to this expression

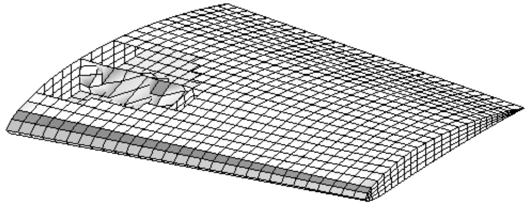
1. BLF = MIN(r1, r2, r3, r4, r5, …, ri)
   - MIN: Take the minimum value
   - r1: Eigenvalue of mode 1
   - r2: Eigenvalue of mode 2
   - …
   - ri: Eigenvalue of mode i



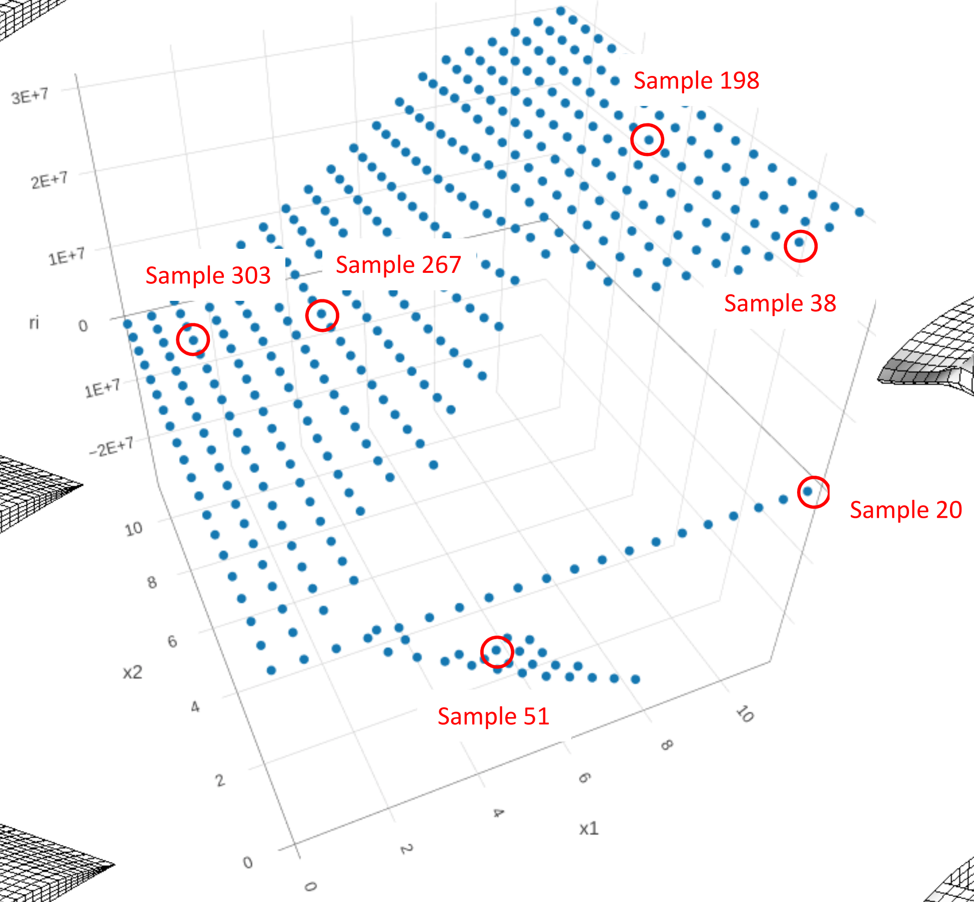
Sample 267
(3.474368, 7.526632)

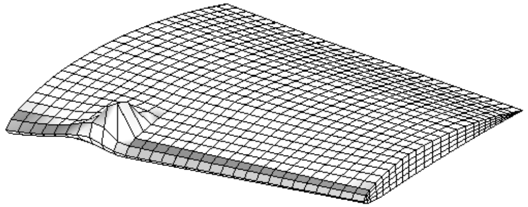


Sample 303
(1.158789,8.684421)

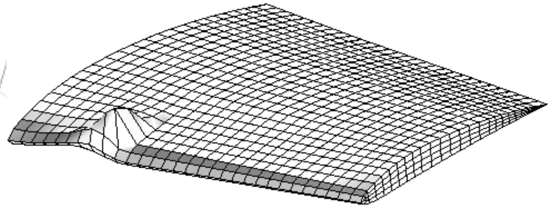


Sample 51
(5.789947,1.158789)

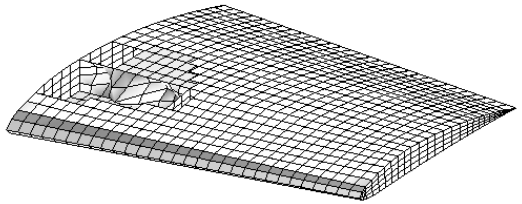




Sample 198
(9.842211,5.211053)



Sample 38
(9.842211,5.211053)



Sample 20
(11.,.001)

For the FE model shown, 400 different configurations of the FE model were evaluated and the corresponding BLFs were collected and a scatter plot was created. x1 and x2 correspond to the thickness of the skin and internal stiffeners. Response ri is the BLF.

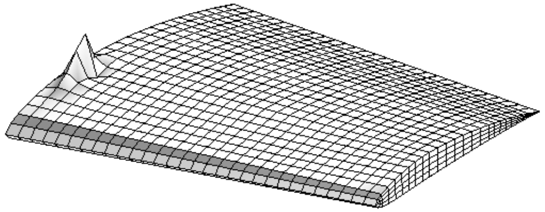
Response Surface of Buckling Load Factor, Continued

The scatter plot shown to the right is the response surface for buckling load factor across the design space. Note the following:

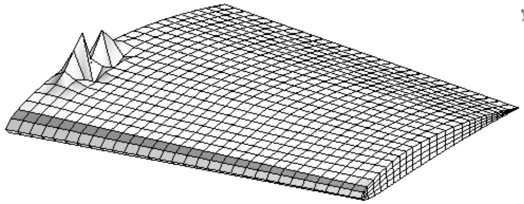
1. Partially Rough Surface – The response surface for different areas of the design space has different characteristics, i.e. flat versus steep surfaces when going from sample 198 to sample 267. This difference is due to the BLF corresponding to the eigenvalues of different mode shapes.

2. Discontinuous Surface – Since the BLF can include negative values, the response surface is discontinuous, e.g. when going from sample 38 to sample 51. In other instances, say going from sample 38 to sample 20, the BLF values vary significantly.

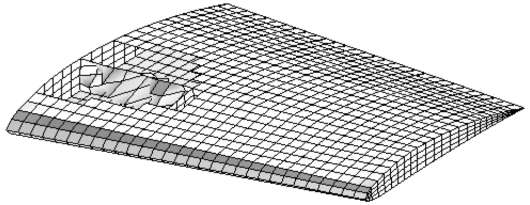
As the optimizer transverses the design space, the optimizer will encounter the roughness and sometimes discontinuous regions of the response surface, hence, the optimization results appear to go up and down. The optimizer in MSC Nastran is a gradient based optimizer and requires the response surface to be continuous, which is why the optimizer might terminate when the optimizer attempts to move between discontinuous regions.



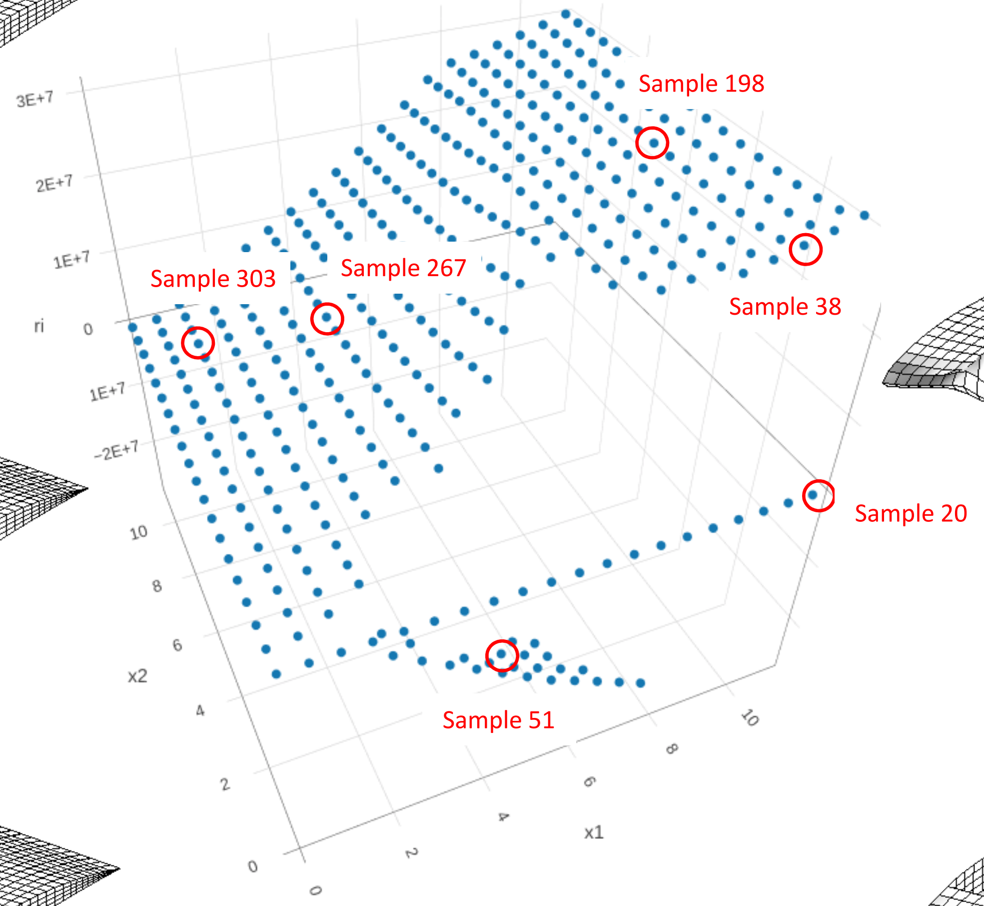
Sample 267
(3.474368, 7.526632)



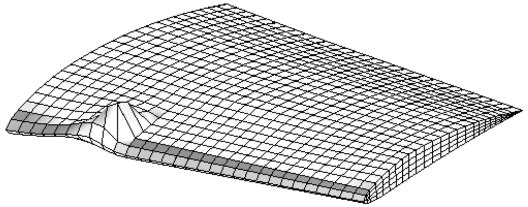
Sample 303
(1.158789,8.684421)



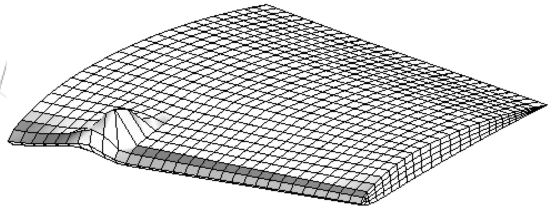
Sample 51
(5.789947,1.158789)



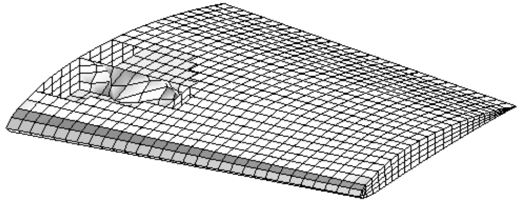
For the FE model shown, 400 different configurations of the FE model were evaluated and the corresponding BLFs were collected and a scatter plot was created. x1 and x2 correspond to the thickness of the skin and internal stiffeners. Response ri is the BLF.



Sample 198
(9.842211,5.211053)



Sample 38
(9.842211,5.211053)

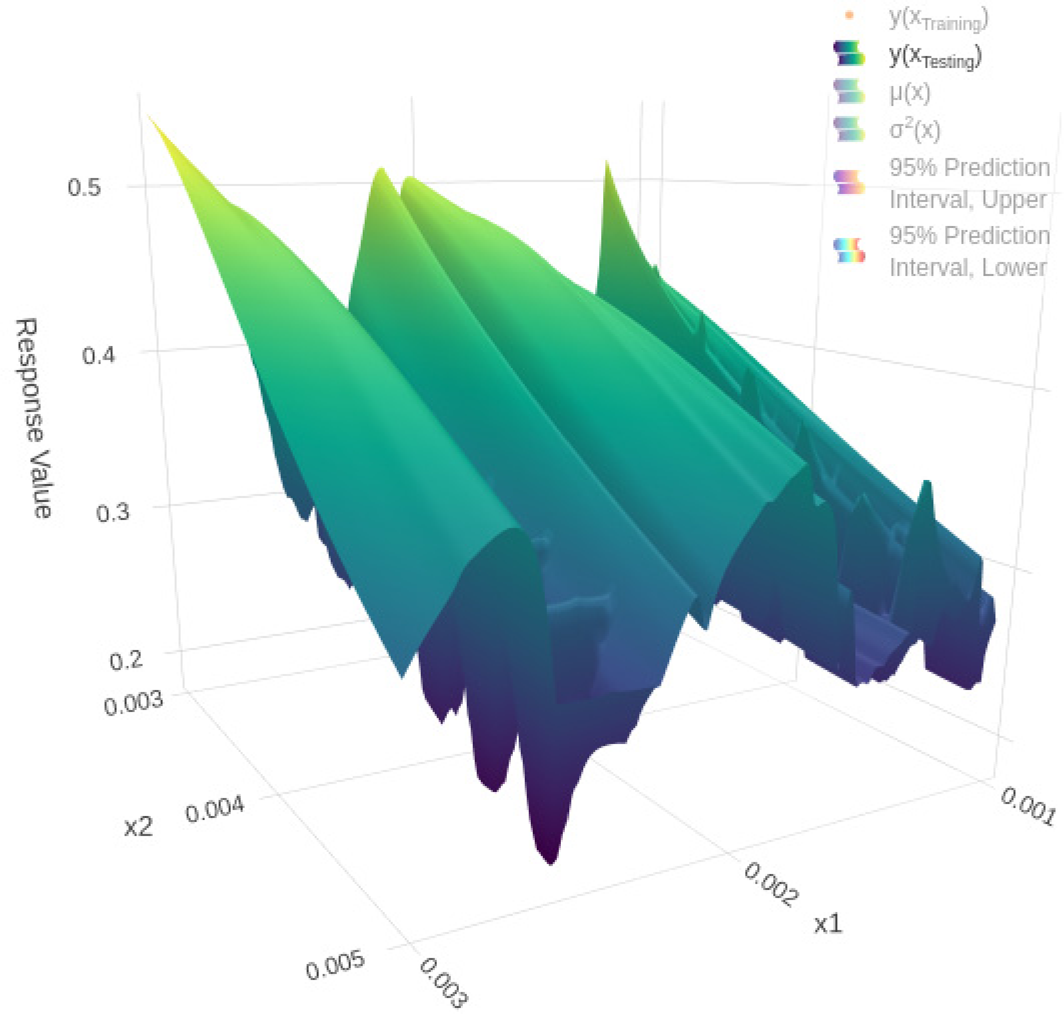


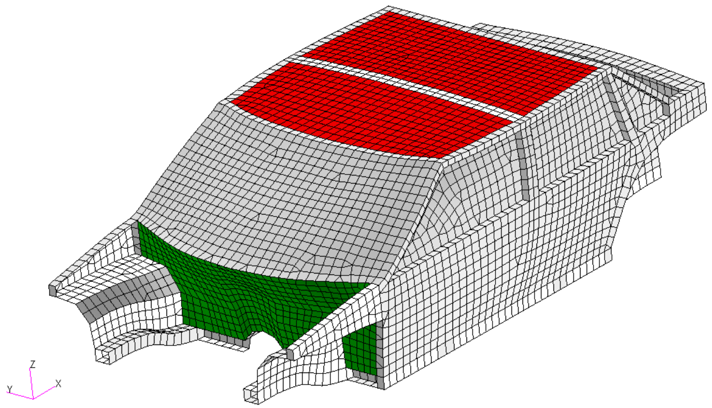
Sample 20
(11.,.001)

Another Example of a Rough Response Surface

Shown to the right is a different example of a response surface that is very rough. The response surface corresponds to the maximum value of sound pressures. In many cases, the use of MAX or MIN will cause rough or discontinuous in the response surface.

Be careful when using the MIN or MAX of multiple values during an optimization.





Optimization Problem Statement

- Minimize the maximum sound pressure level in dB, magnitude, (DBM) at grid 30606
- Vary the thickness of the roof and dash

Internal Source:

- 20200818_gp_modeling_survey_of_sol_200_design_models/z_nascar_2/
- 20220731_response_surface_of_buckling_load_factor

How to handle negative buckling factors during an optimization?

If you have negative BLFs there are 2 methods to handle negative BLFs.

1. For the optimization, consider using the absolute value, e.g. ABS(BLF). Refer to figures 1 and 2. This tutorial used the absolute value of the BLF.

2. Use the EIGRL or EIGR bulk data entries to restrict the range of BLFs. Refer to Example Entries.
    1. Entry EIGRL 10 is configured to consider any BLF between -5E6 and 5E6. Since fields 4 and 5 are blank, the default values of -5E6 and 5E6 are used.
    2. Entry EIGRL 11 is configured to consider only BLFs greater than .00001
    3. Entry EIGRL 11 is configured to consider BLFs between .00001 and 10000.

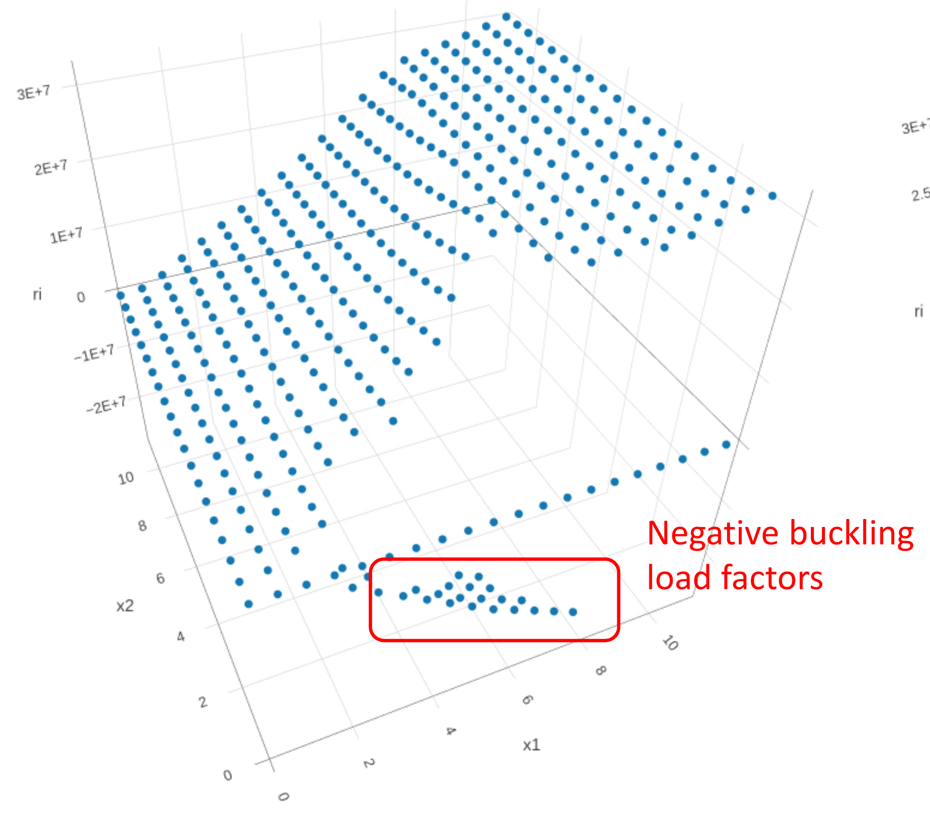


Figure 1: BLF = MIN(r1, r2, …, ri)

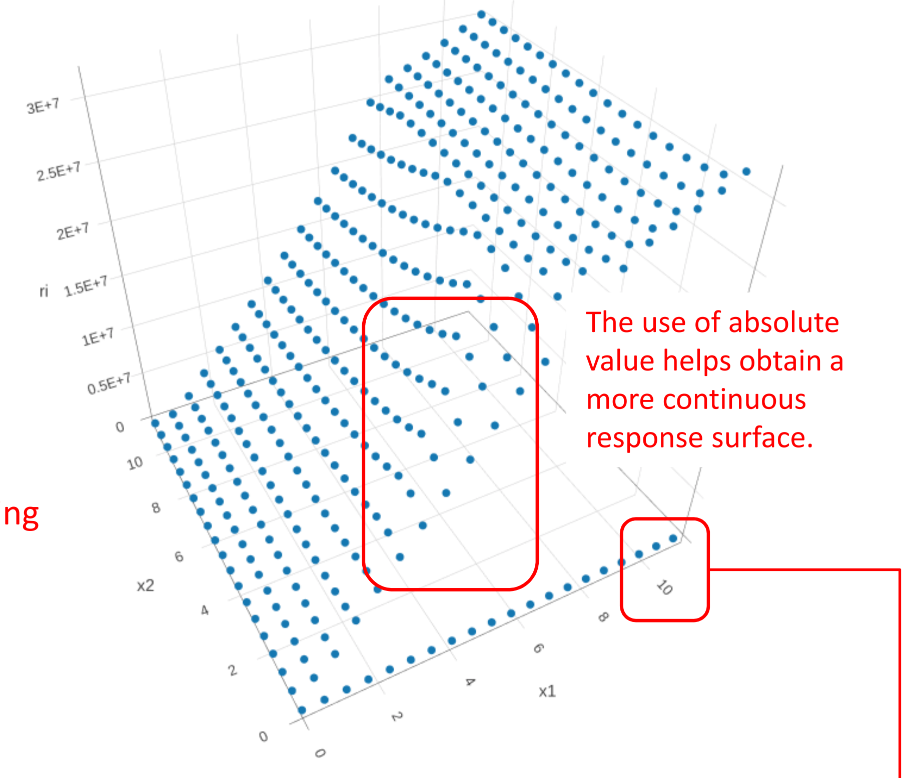


Figure 2: ABS(BLF) = ABS(MIN(r1, r2, …, ri))

Example Entries

```
EIGRL    10                              100
EIGRL    11        .00001                100
EIGRL    12        .00001   10000.       100
```

While this region of the design space is discontinuous, in some cases the optimizer never travels to these discontinuous regions and successfully converges elsewhere in the design space