

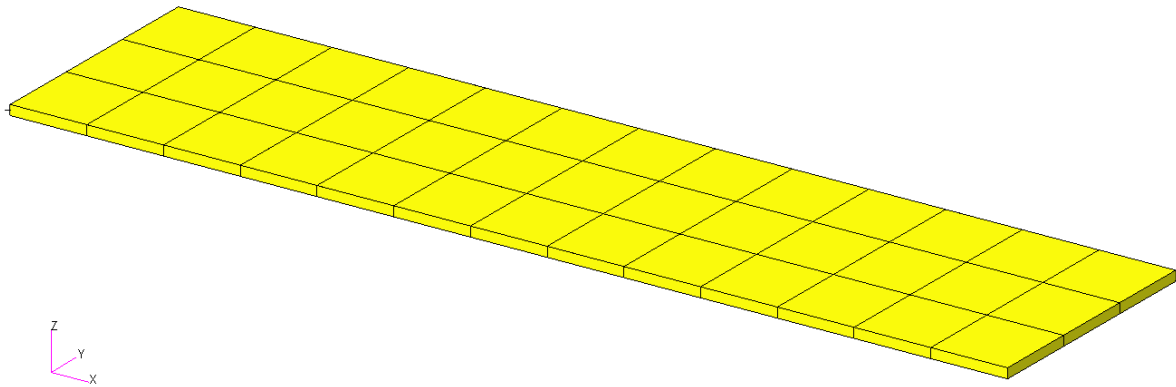
Workshop - MSC Nastran Topometry Optimization of a Cantilever Plate

AN MSC NASTRAN SOL 200 TUTORIAL

Goal: Use Nastran SOL 200 Optimization

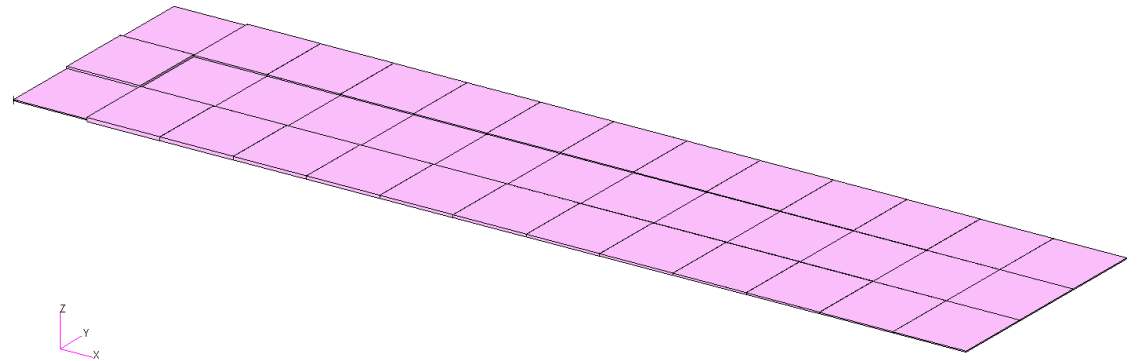
Before Optimization

- Mass: 19.5 kg



After Optimization

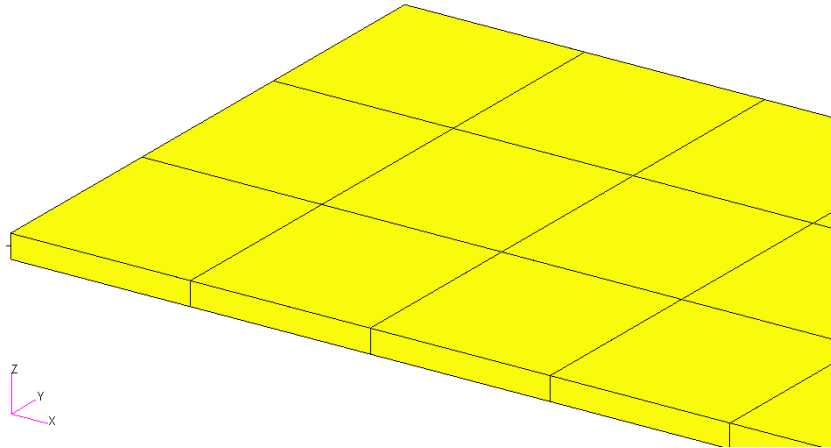
- Mass: 3.97 kg
- Vary the thickness of each element



Goal: Use Nastran SOL 200 Optimization

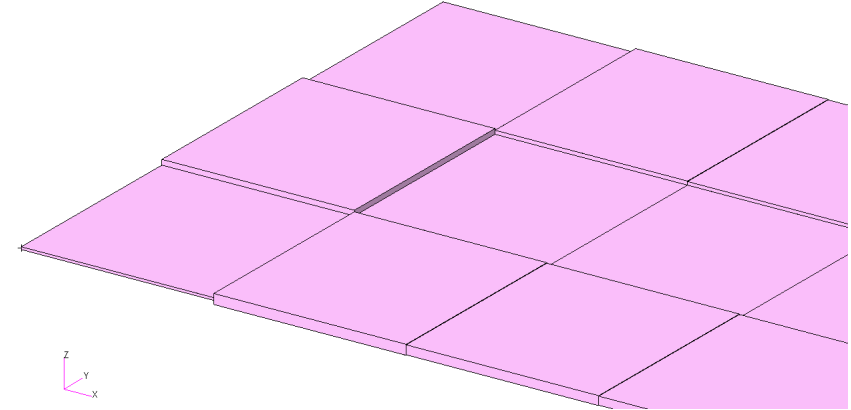
Before Optimization

- Mass: 19.5 kg



After Optimization

- Mass: 3.97 kg
- Vary the thickness of each element



Details of the structural model

Units: m, N, MPa

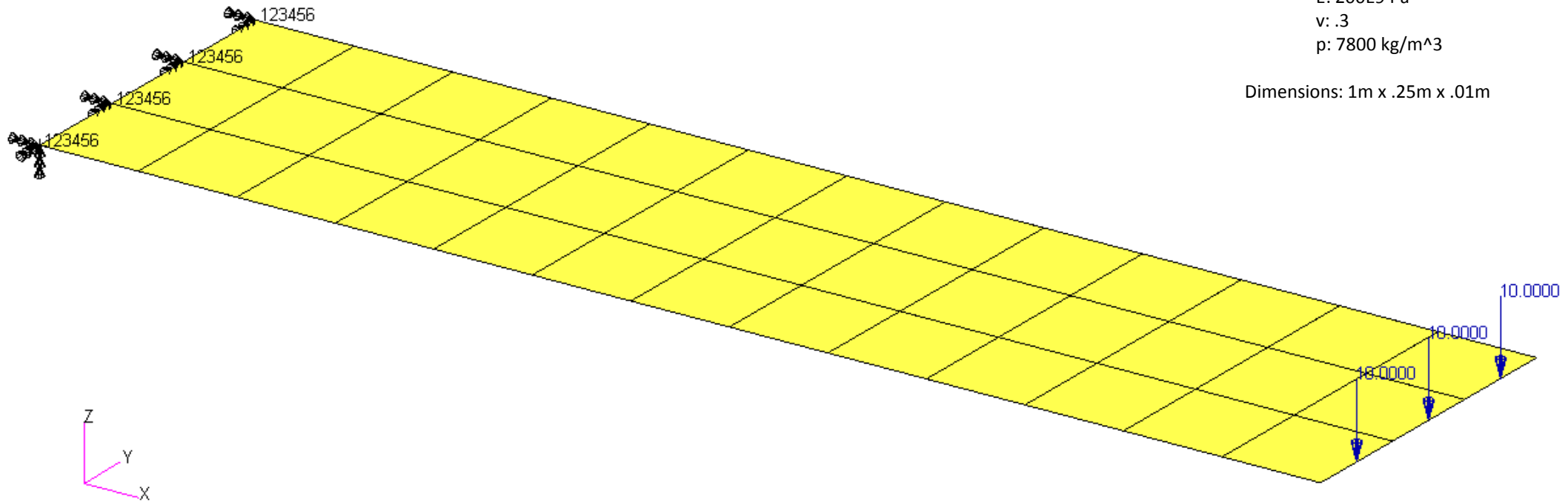
Material:

E: 200E9 Pa

v: .3

p: 7800 kg/m³

Dimensions: 1m x .25m x .01m

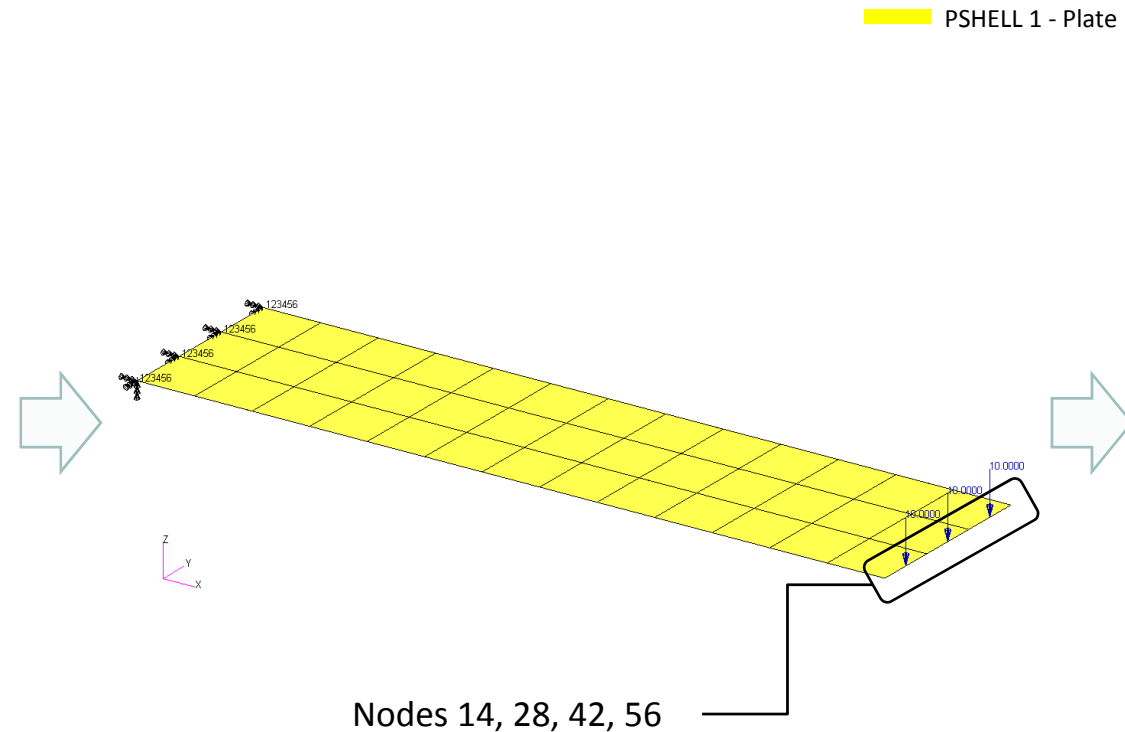


Optimization Problem Statement

Design Region/Variables

z1: Thickness (T) of PSHELL 1

$$.001 < z1$$



Design Objective

r0: Minimize weight

Design Constraints

r1: von Mises stress of PSHELL 1

$$r1 < 250E6$$

r2: Z Displacement of nodes 14, 28, 42, 56 (GRID IDs: 14, 28, 42, 56)

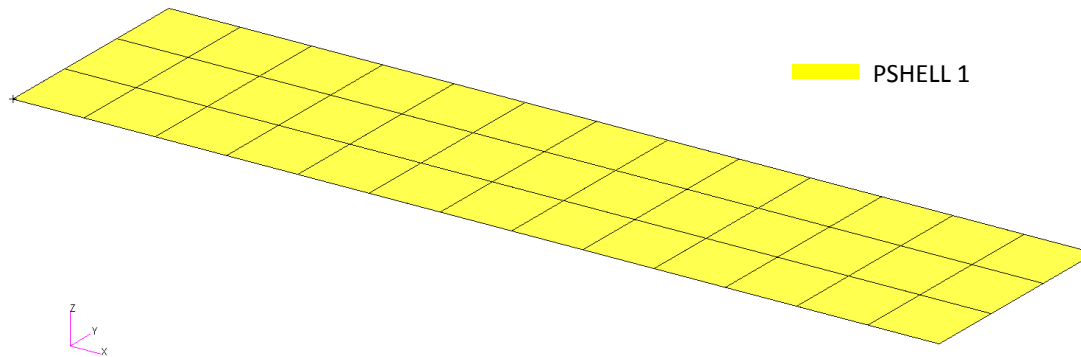
$$-.01 < r2$$

Difference Between Size and Topometry Optimization

Size Optimization

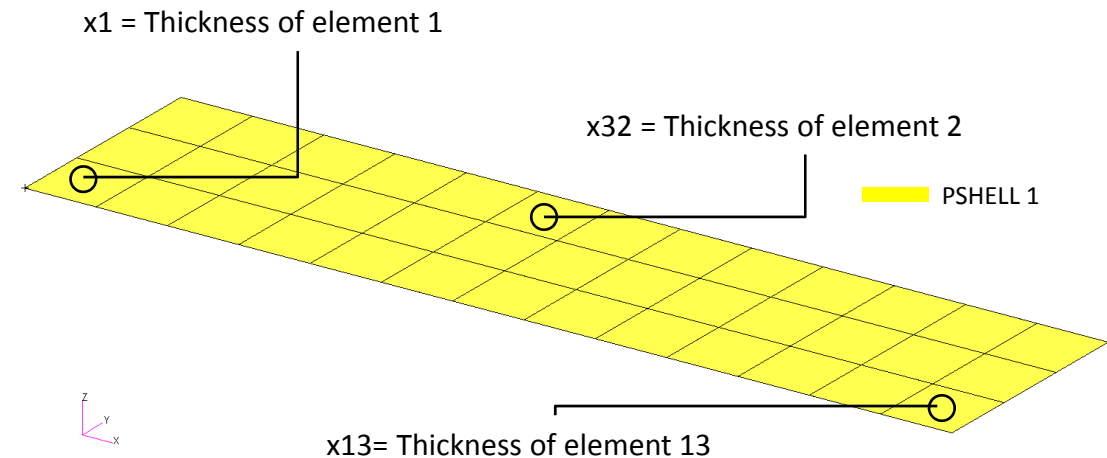
1. Select the parameter to optimize
2. One design variable (x1) is created and applies to all the elements

x1 = Thickness of every element related to PSHELL 1



Topometry Optimization

1. Select the parameter and design region
2. In the background, one design variable is automatically created for each element => Element-by-element optimization



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

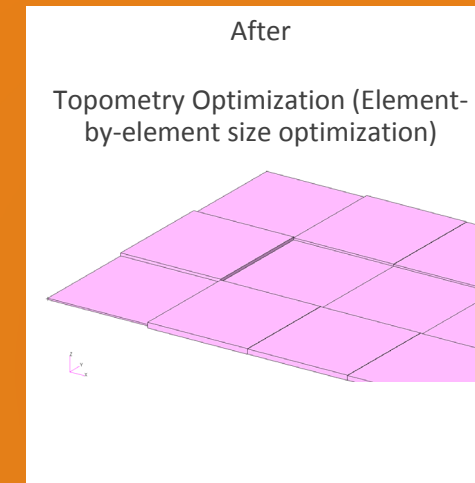
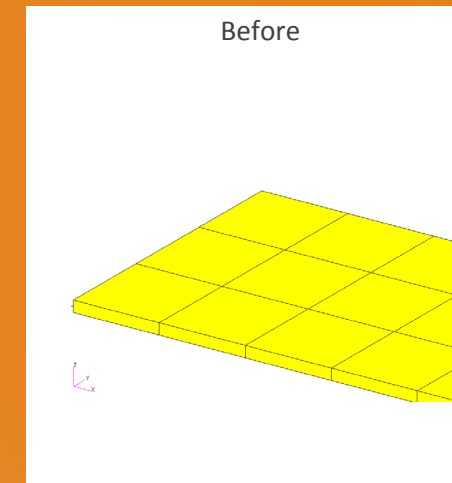
Tutorial

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 Regions/Variables
 - Design Objective
 - Design Constraints
 - Perform optimization with Nastran SOL 200
3. Review optimization results
 - .f06
 - Topometry Optimization and Structural Results

Special Topics Covered

Topometry Optimization - This type of optimization assigns a design variable for each individual element.

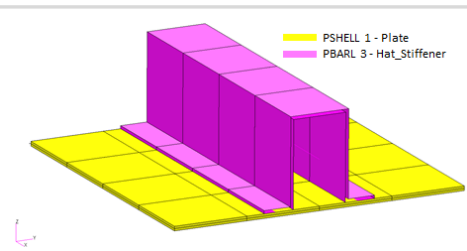


SOL 200 Web App Capabilities

Benefits

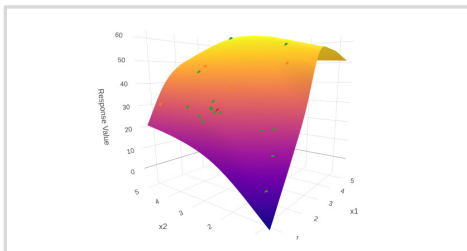
- 200+ error validations (real time)
- Web browser accessible
- Automated creation of entries (real time)
- Automatic post-processing
- 76 tutorials

Capabilities



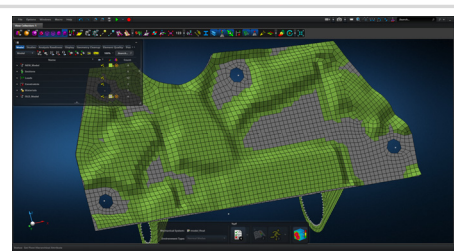
Web Apps for SOL 200

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



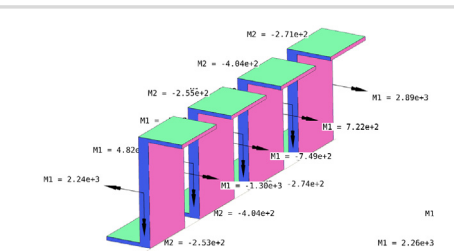
Machine Learning Web App

Bayesian Optimization for nonlinear response optimization (SOL 400)



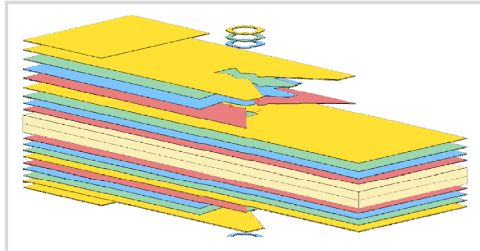
MSC Apex Post Processing Support

View the newly optimized model after an optimization



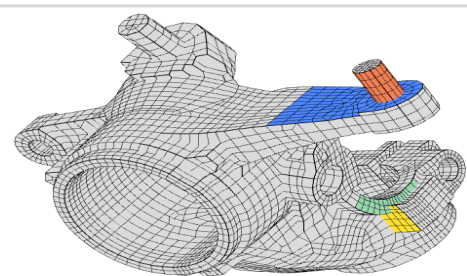
Beams Viewer Web App

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



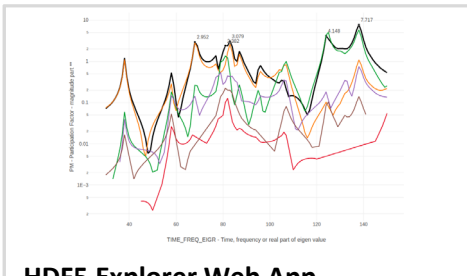
Ply Shape Optimization Web App

Spread plies optimally and generate new PCOMPG entries



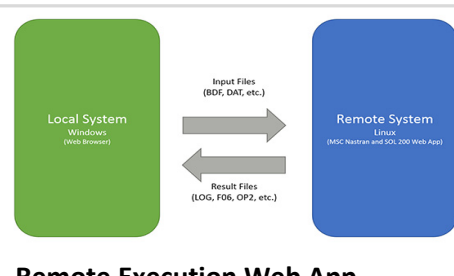
Shape Optimization Web App

Use a web application to configure and perform shape optimization.



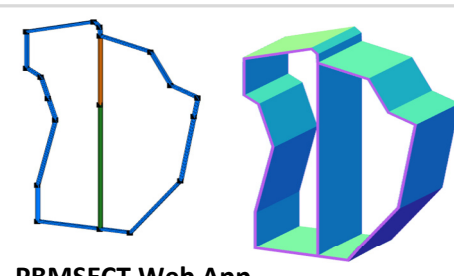
HDF5 Explorer Web App

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



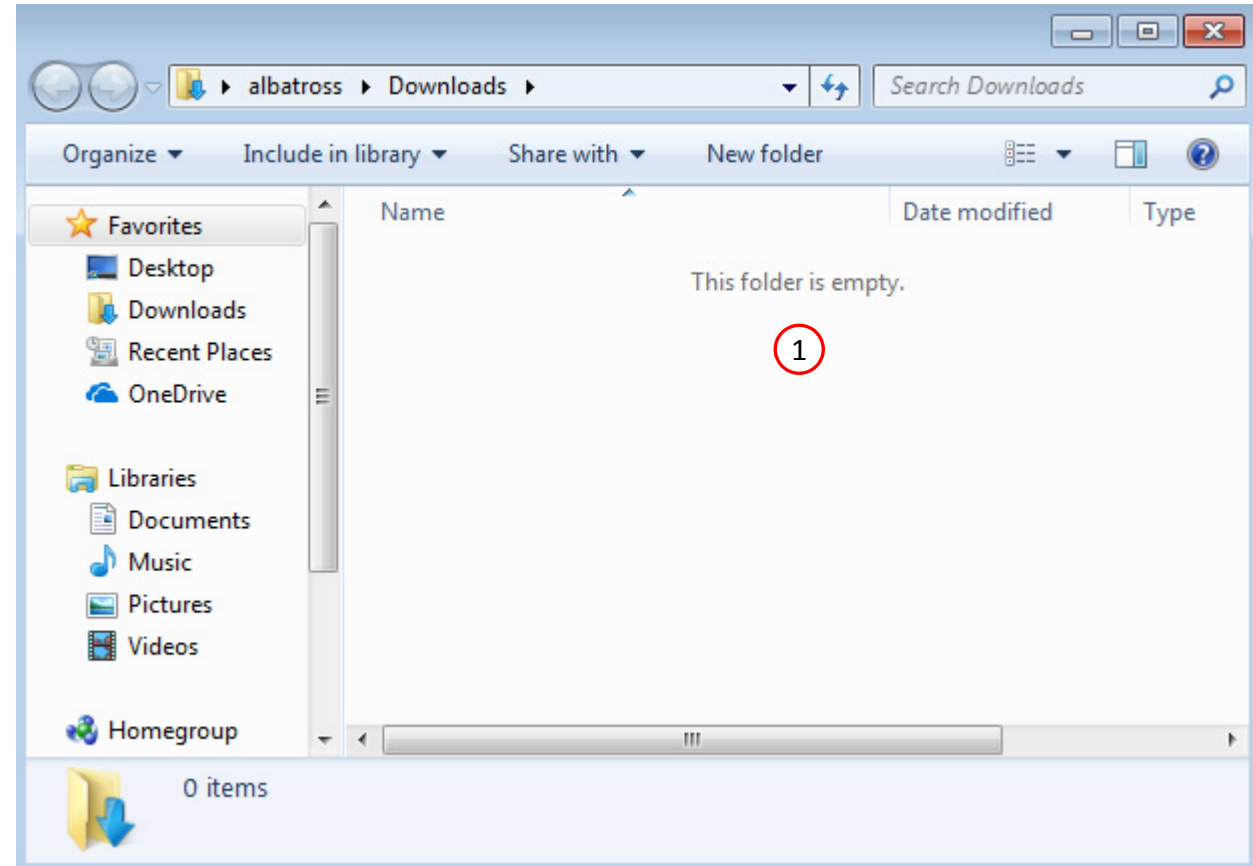
Stacking Sequence Web App

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.



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.

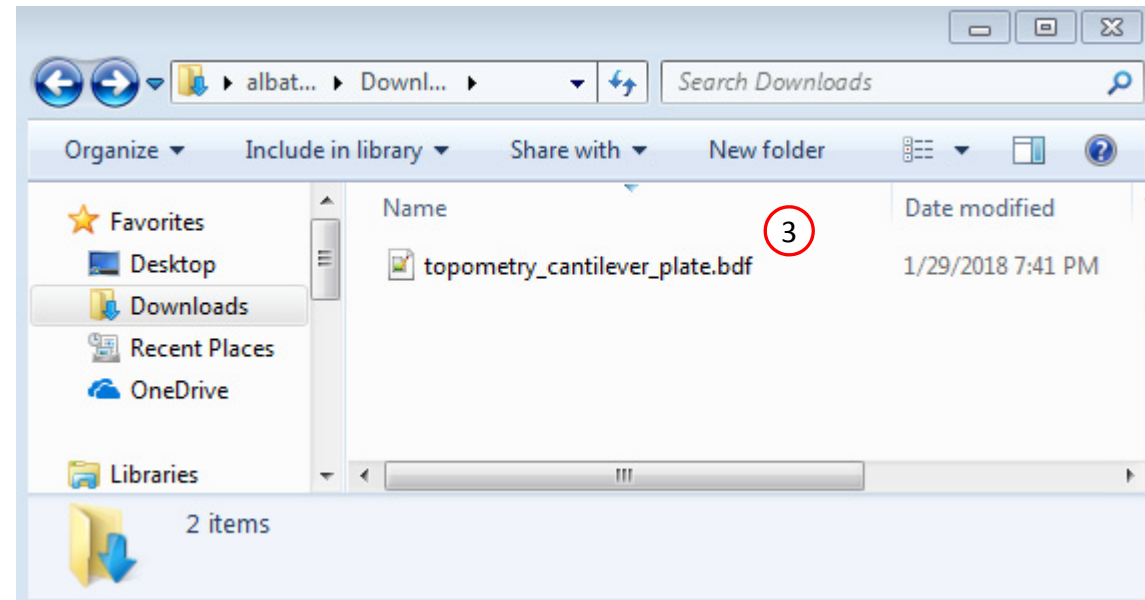
1

MSC Nastran Topometry Optimization of a Cantilever Plate

This tutorial is an introduction to Topometry Optimization. A simple cantilever plate is used to demonstrate element-by-element optimization of thickness.

Starting BDF Files: [Link](#) 2
Solution BDF Files: [Link](#)

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

Upload BDF Files

1. Click 1. Select Files and select topometry_cantilever_plate.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.

Step 1 - Upload .BDF Files

The screenshot shows a two-step process for uploading files. Step 1, '1. Select files', is highlighted with a red circle and shows a file named 'topometry_cantilever_plate.bdf' selected. Below this, a green progress bar indicates 'Inspecting: 100%'. Step 2, '2. Upload files', is also highlighted with a red circle and shows a green progress bar indicating 'Uploading: 100 %'. At the bottom, there is a checkbox labeled 'List of Selected Files' which is currently unchecked.

1. Select files topometry_cantilever_plate.bdf

Inspecting: 100%

2. Upload files

Uploading: 100 %

☐ List of Selected Files





Create Design Region

1. Click Topometry
2. Click on the plus (+) icon to set the thickness (T) of PSHELL 1 as a Design Region
3. The new Design Region is added to the table, no further edit is necessary

- Suppose the goal is to vary the thickness. In traditional Size optimization, the thickness can be a set a single design variable. With Topometry optimization, when the design region is set, each element in the region is given its own independent thickness design variable.
- If PSHELL 1 has 500 elements associated and is configured as a design region, then there will be 500 design variables created.
- 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 TOMVAR	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"/>
2 	T	Thickness	PSHELL	1	.01
	E	Young's modulus	MAT1	1	2.+11
	NU	Poisson's ratio	MAT1	1	.3
	RHO	Mass density	MAT1	1	7800.

5 10 20 30 40 50
Number of Visible Rows 5

Step 2 - Adjust TOMVAR Entries

+ Options

 Delete Visible Rows

	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"/>
	z1	3 	T	Thickness	PSHELL	1	.01	.001	Upper	Examples: -2.0, 1.0, THRU, 10.0, B'

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 scalar response. A response such as weight and volume are single responses 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.

Objective Equation Objective

1

Step 1 - Select an objective

Select an analysis type

SOL 101 - Statics

Select a response

		Response Description ▾	Response Type ▾
		<input type="text" value="Search"/>	<input type="text" value="Search"/>
2	+	Weight	WEIGHT
	+	Volume	VOLUME
	+	Displacement	DISP
	+	Strain	STRAIN
	+	Element Strain Energy	ESE

« 1 2 3 4 5 »

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. Type 's' into the search bar
3. Click on the plus (+) icon for Displacement
4. Click on the plus (+) icon for Stress
5. Configure the following for r1:
 - ATTA: 3 - T3 (Z component)
 - ATTi: 14, 28, 42, 56 (Nodes 14, 28, 42 and 56)
 - Lower Allowed Limit: -.01
6. Configure the following for r2:
 - Property Type: PSHELL
 - ATTA: 11 - von Mises
 - ATTi: 1 (PSHELL 1)
 - Upper Allowed Limit: 250.E6

- The constraints are defined normally as would be done in a size optimization.

Step 1 - Select constraints

Select an analysis type

SOL 101 - Statics

Select a response

	Response Description	Response Type
	s	Search
3	+	Displacement
	+	Strain
	+	Element Strain Energy
4	+	Stress
	+	Fatigue, pseudo-static fatigue analysis

« 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		3 - T3 (Rectangular z, Cylindrical z)		14, 28, 42, 56	-.01	Upper
×	r2	✓	STRESS	PSHELL	11 - von Mises or maximum shear		1	Lower	250.E6

Configure Optimization Settings

1. Click Settings
2. Set P2 to 12 – Print constraints and responses

- The P2 setting controls the output of the following information to the F06 file: objective, constraints, responses, properties and design variables.
- This is a topology optimization and will generate a large amount of property and design variable data in the F06 file. To make the F06 file size manageable, the design variable information is omitted by using the P2=12 option. When the results are viewed, note that the objective and constraint information is plotted, but the design variable history is not plotted due to the P2=12 option.
- If this is a combined size and topology optimization, P2 should be set to 15. If this is a pure size optimization, P2 should be set to 15.

1

Optimization Settings

Parameter ↕	Description ↕	Configure ↕
Search	Search	Search
APRCOD	Approximation method to be used	<input type="checkbox"/> 2 - Mixed Method ▼
CONV1	Relative criterion to detect convergence	<input type="checkbox"/> Enter a positive real number
CONV2	Absolute criterion to detect convergence	<input type="checkbox"/> Enter a positive real number
DELX	Fractional change allowed in each design variable during any optimization cycle	<input type="checkbox"/> Enter a positive real number
DESMAX	Maximum number of design cycles to be performed	<input checked="" type="checkbox"/> 20
DISBEG	Design cycle number for discrete variable processing initiation	<input type="checkbox"/> Enter a positive integer
GMAX	Maximum constraint violation allowed at the converged optimum	<input type="checkbox"/> Enter a positive real number
P1	Print items, e.g. objective, design variables, at every n-th design cycle to the .f06 file	<input checked="" type="checkbox"/> 1
P2	Items to be printed to the .f06 file	<input checked="" type="checkbox"/> 12 - Print constraints and responses ▼
TCHECK	Topology Checkerboarding	<input type="checkbox"/> -1 - Automatic selection (Default) ▼
TDMIN	Minimum diameter of members in topology optimization	<input type="checkbox"/> Enter a positive real number
TREGION	Trust Region	<input type="checkbox"/> 1 - Trust Region On ▼

2

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

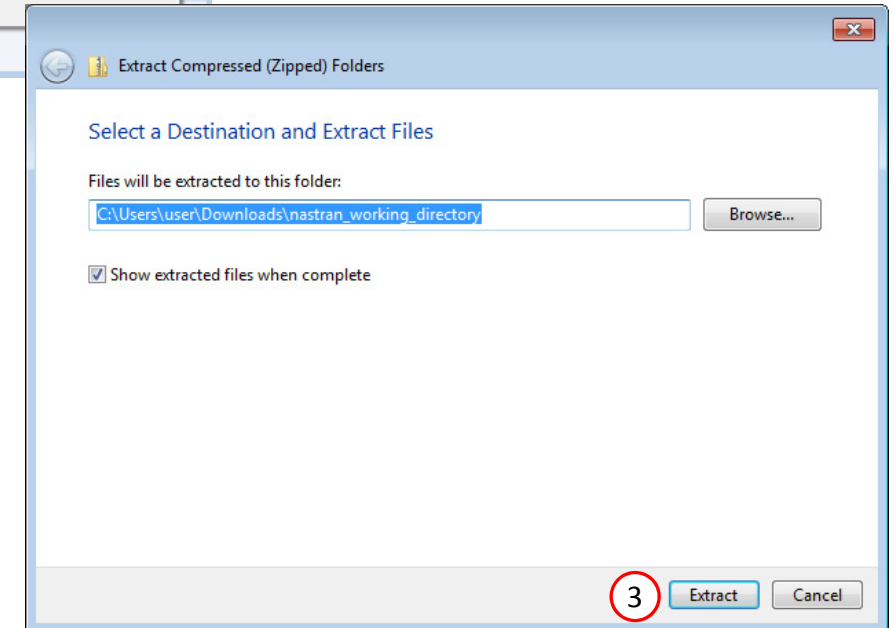
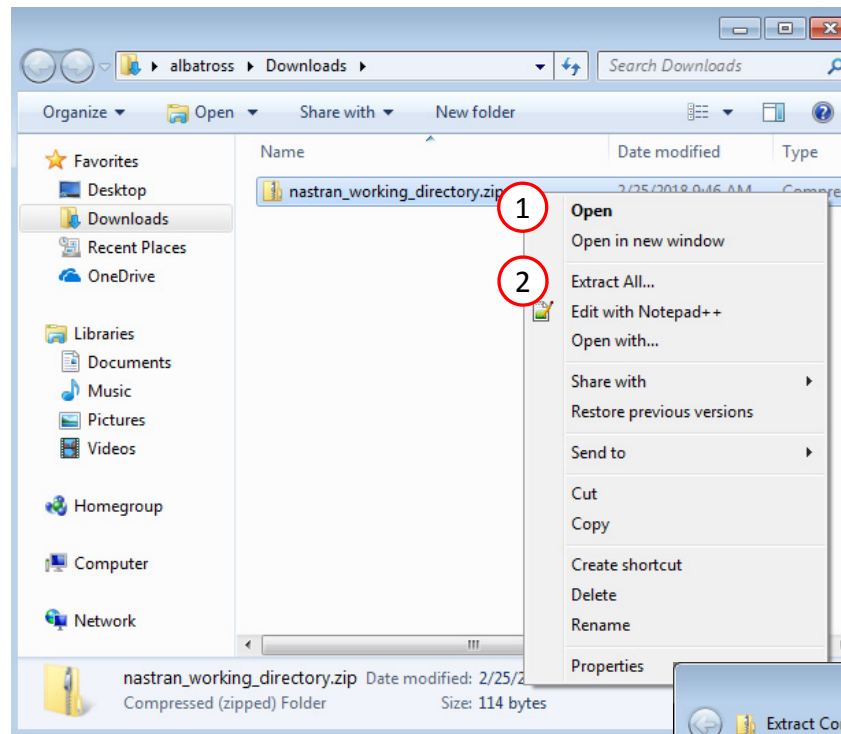
- When the download button is clicked a new file named “nastran_working_directory” is downloaded. If the file already exists in your local folder, the folder name is appended with a number, e.g. “nastran_working_directory (1).zip”

Download BDF Files

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.

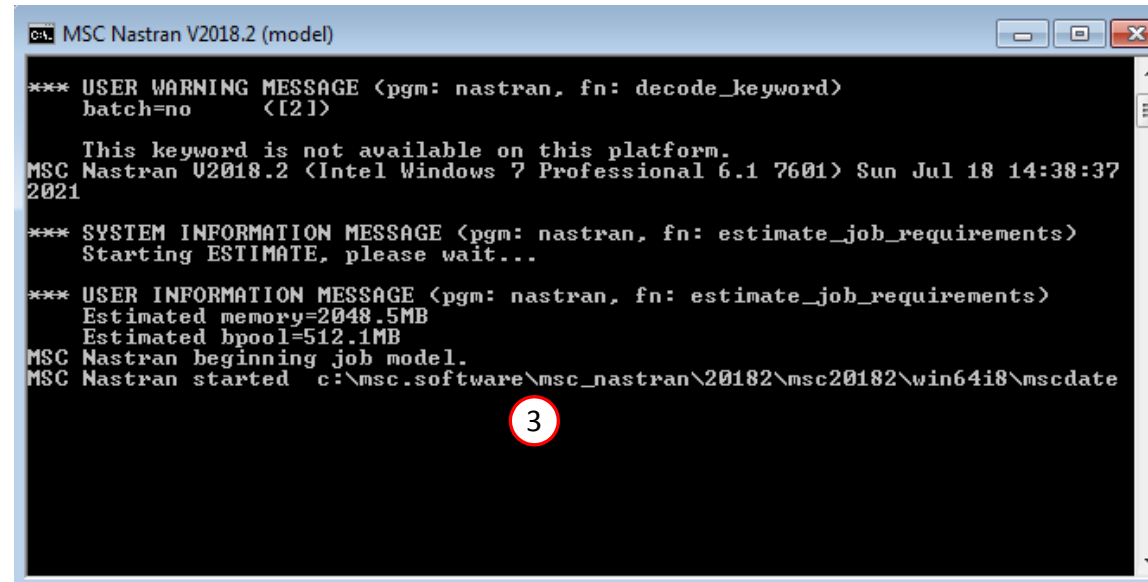
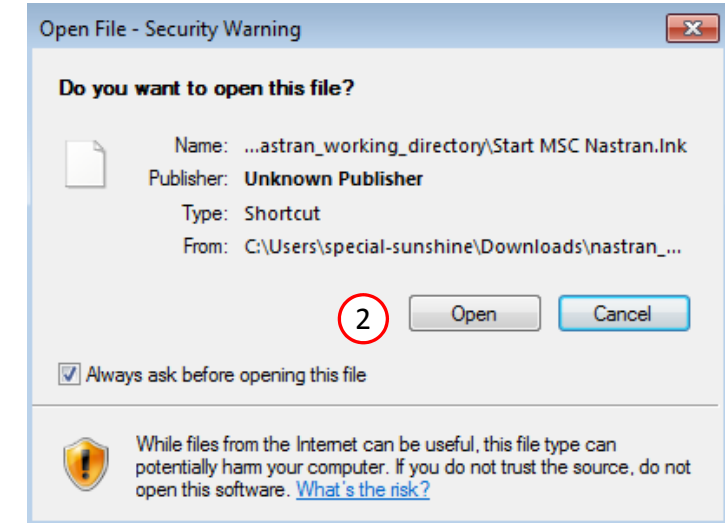
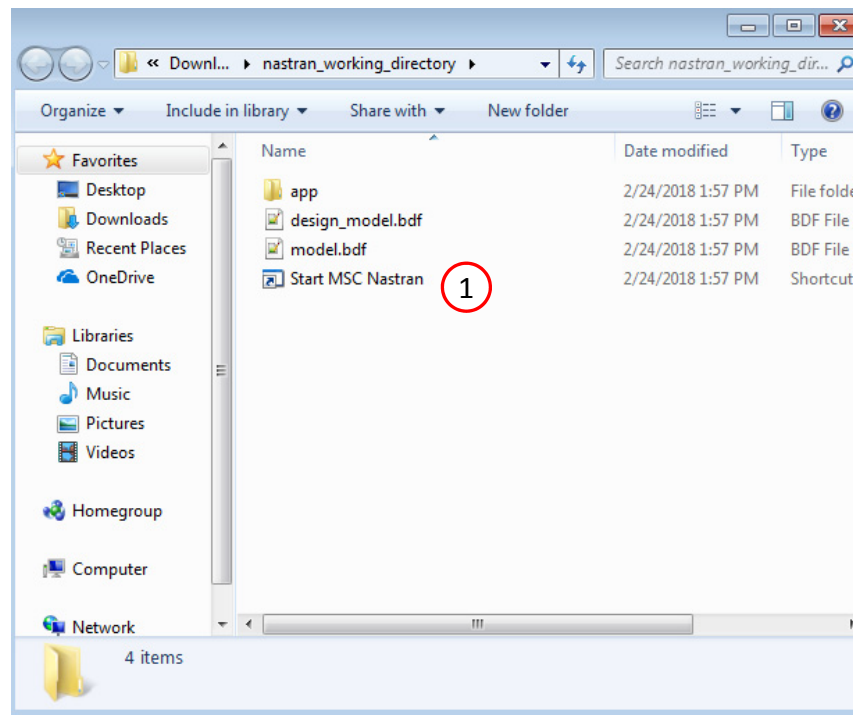
Using Linux?

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

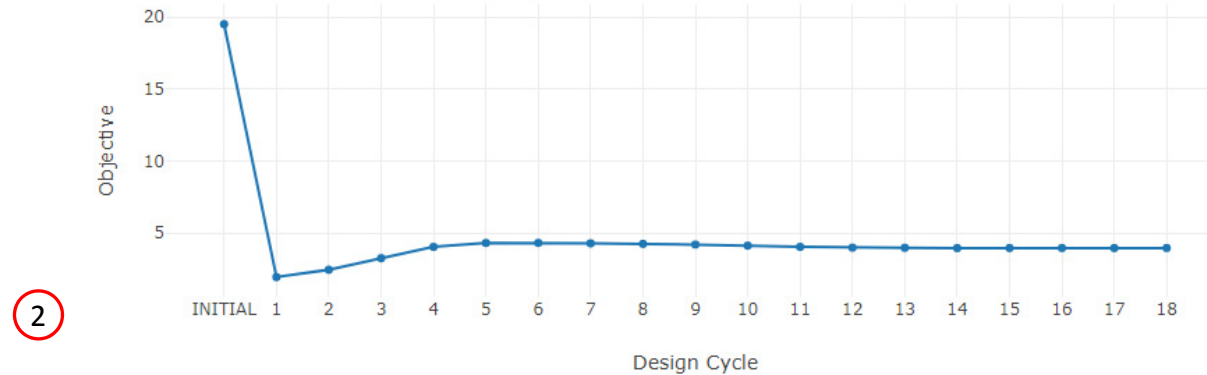
1. After MSC Nastran is finished, the results will be automatically uploaded.
2. Ensure the messages shown have green checkmarks. This is indication of success. Any red icons indicate challenges.
3. The final value of objective and normalized constraints can be reviewed.

- Note that in a Topometry optimization, hundreds or thousands of design variables can be created. In this situation, the Design Variables are not plotted and displayed. Instead, the Objective and Normalized Constraints are displayed. It is recommended that a traditional post-processor be used to review the design variable results.

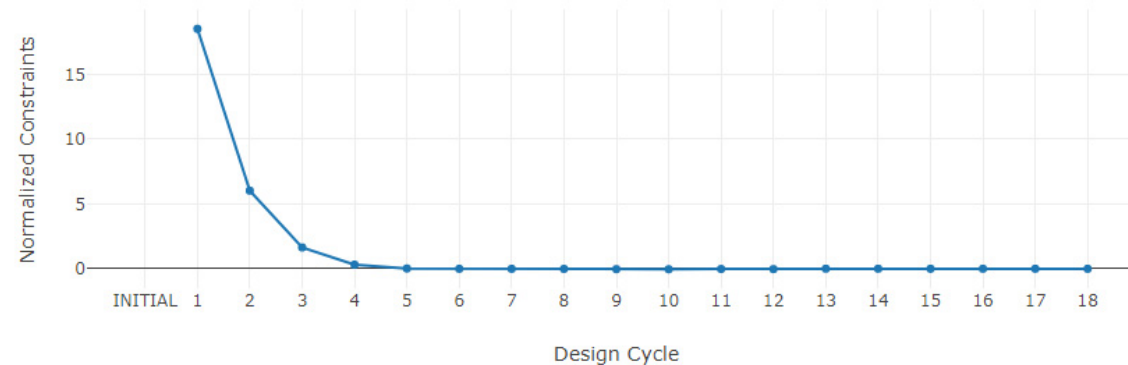
Final Message in .f06

- 1  RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 18.

Objective

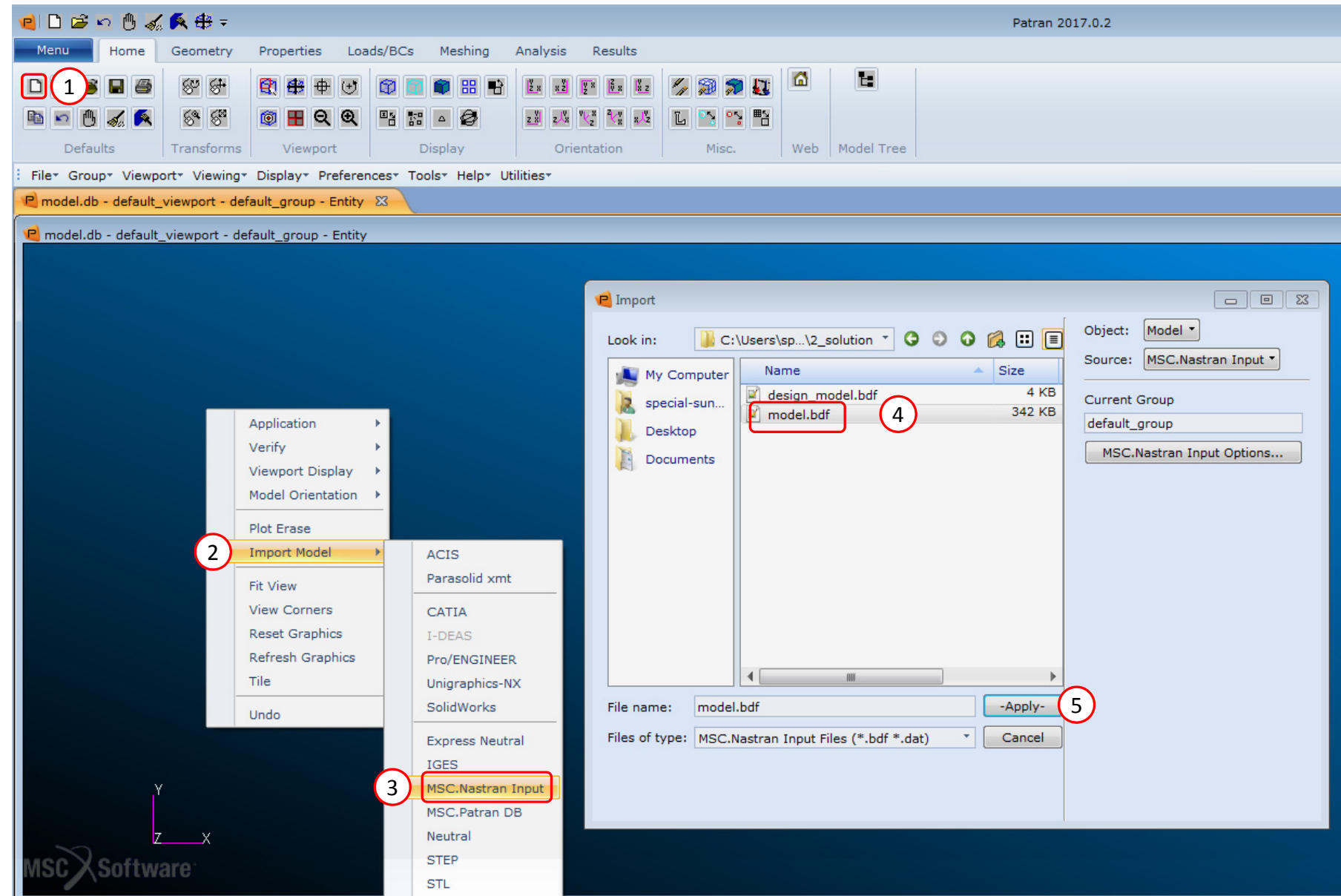


Normalized Constraints



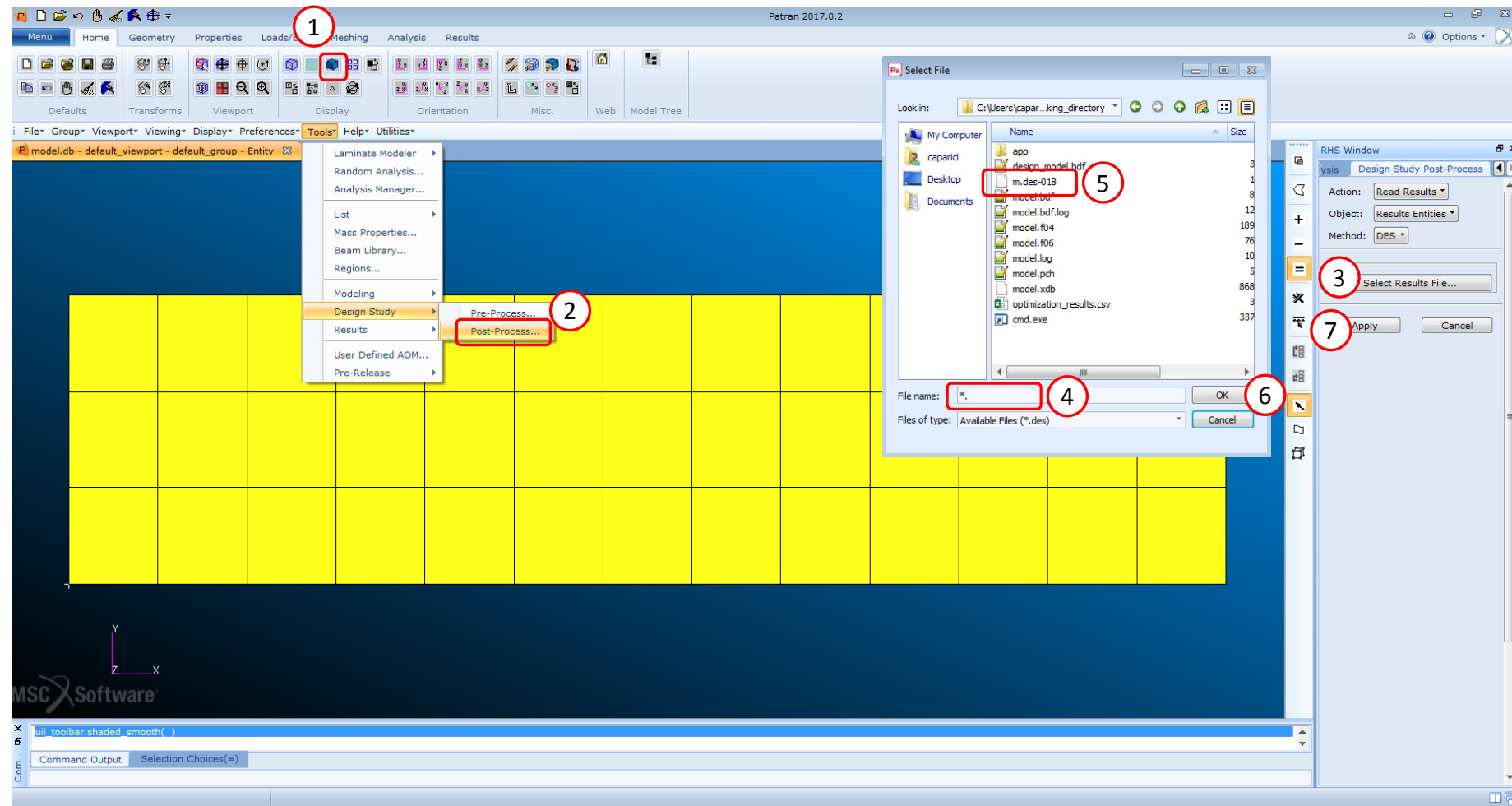
Review Optimization Results in Patran

1. Start a new Patran session
2. Right click to open a menu
3. Go to Import Model and click on MSC.Nastran Input
4. Select model.bdf (This file was used for the optimization)
5. Click Apply



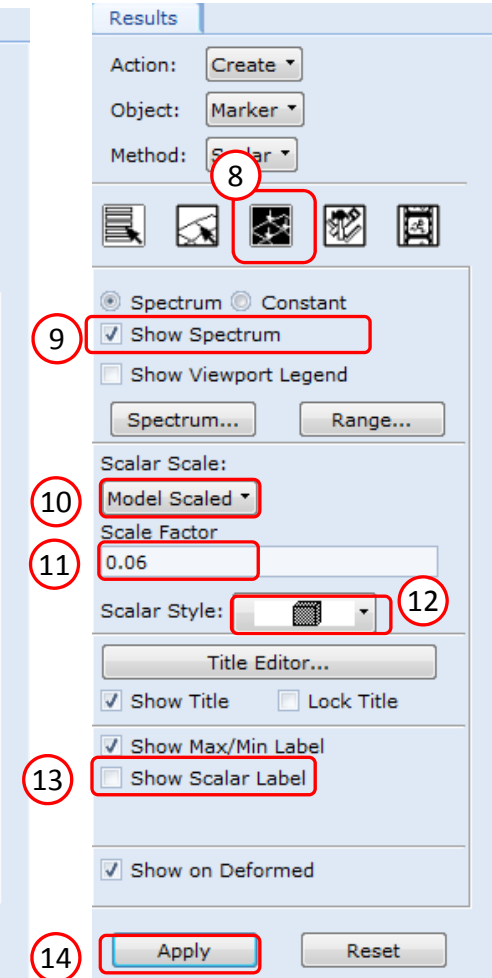
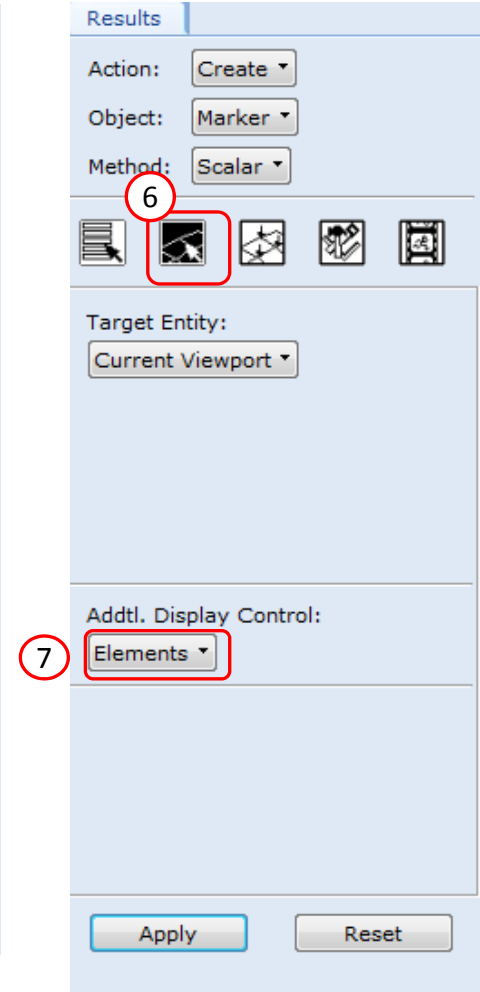
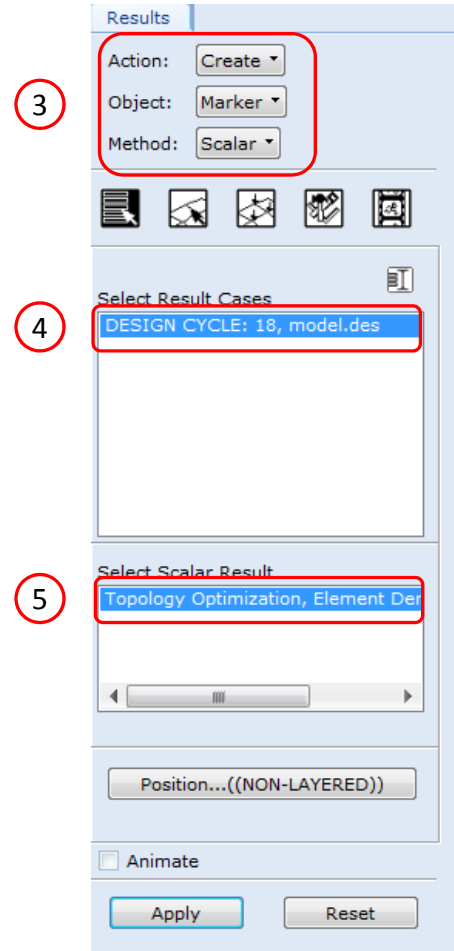
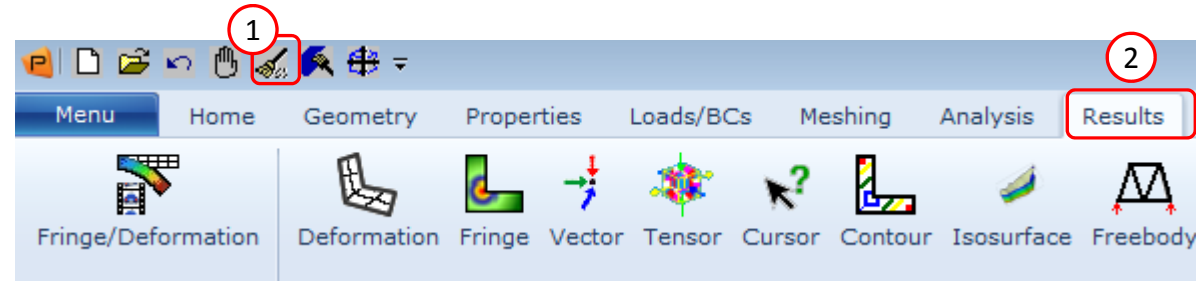
Review Optimization Results in Patran

1. Click Smooth Shading
2. Go to Tools > Design Study and click on Post-Process
3. Click Select Results File
4. Type this in the File name input box and press the Enter key: *.
 - This will display all the files in the directory
5. Select one of the following result files:
 - m.des-000i
 - This file is created when this entry is in the BDF file:
PARAM DESPCH1 -1
 - model.des
 - This file is created when this entry is not in the BDF file:
PARAM DESPCH1 -1
6. Click OK
7. Click Apply



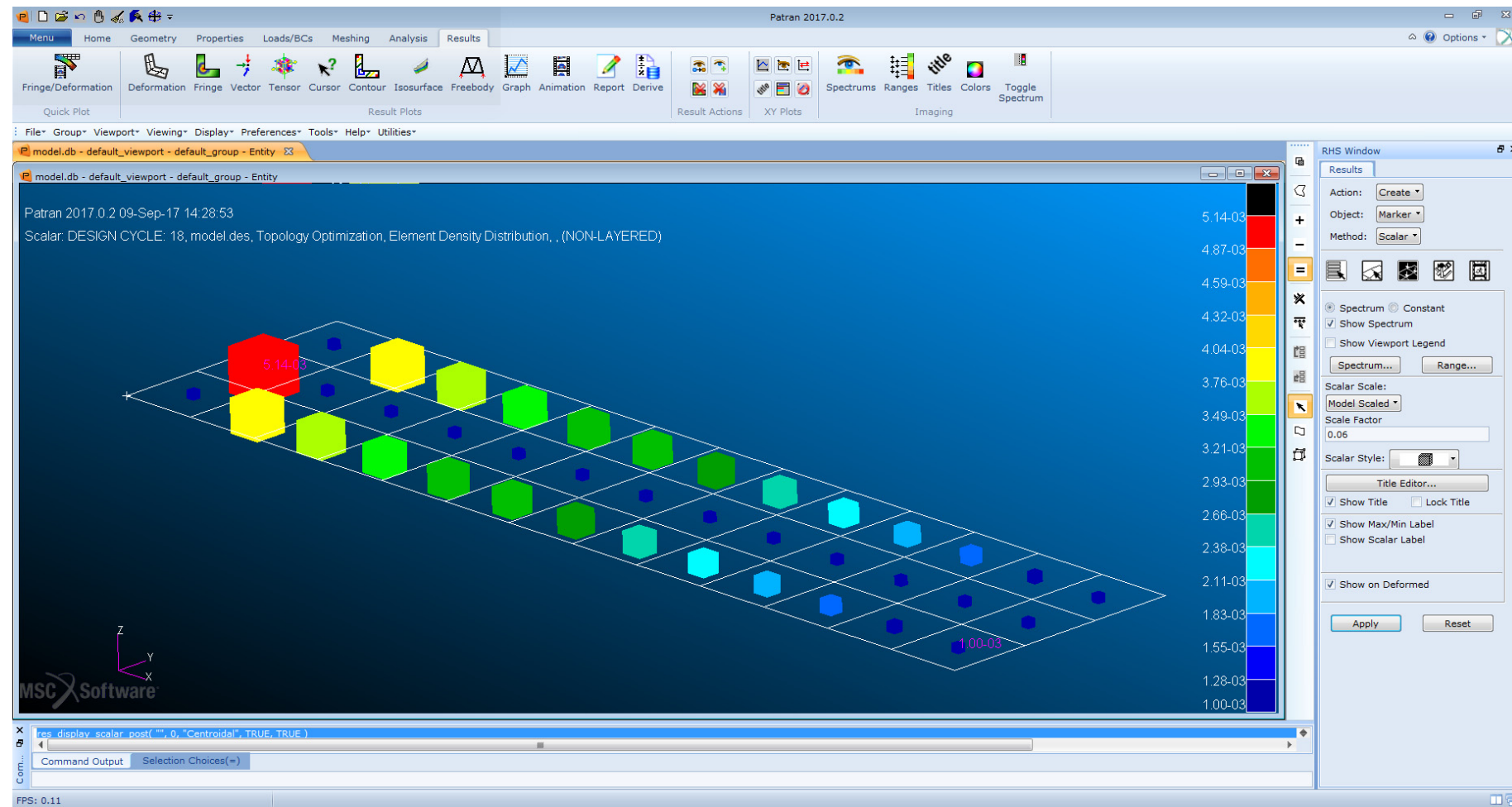
Review Optimization Results in Patran

1. Click the clear icon
2. Click Results
3. Set the following:
 1. Action: Create
 2. Object: Marker
 3. Method: Scalar
4. Select: DESIGN CYCLE: 18, model.des (The row should be highlighted blue)
5. Select Topology Optimization, Element D... (The row should be highlighted blue)
6. Click Target Entities
7. Set to Elements
8. Click Display Attributes
9. Mark the checkbox for Show Spectrum
10. Change to Model Scaled
11. Set Scale Factor to .06
12. Change Scalar Style to the shaded cube
13. Uncheck the box for Show Scalar Label
14. Click Apply



Review Optimization Results in Patran

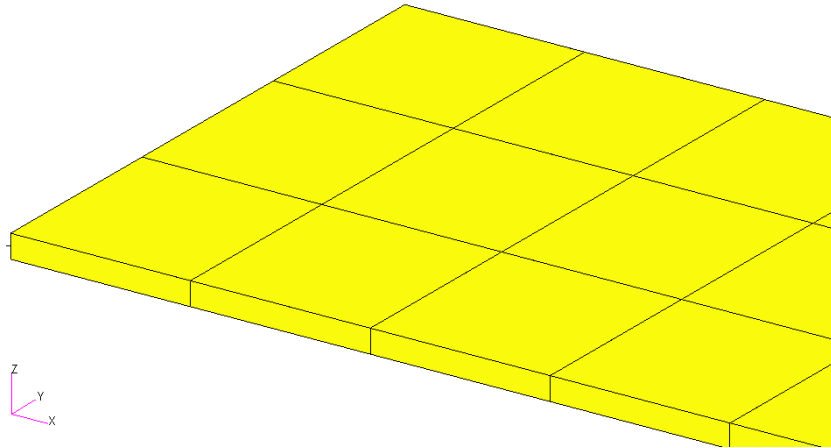
1. The plot shows the new thickness distribution of each element. Plot not to scale.



Results

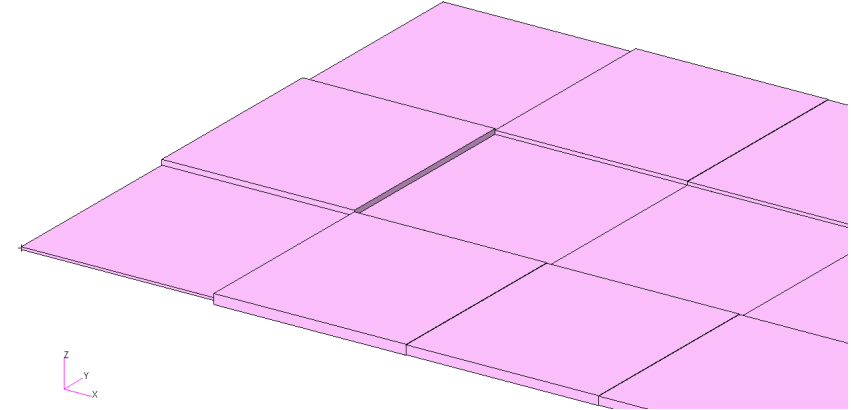
Before Optimization

- Mass: 19.5 kg



After Optimization

- Mass: 3.97 kg
- Vary the thickness of each element



End of Tutorial