Workshop - MSC Nastran Topology Optimization - Minimizing mass with stress and displacement constraints

AN MSC NASTRAN SOL 200 TUTORIAL

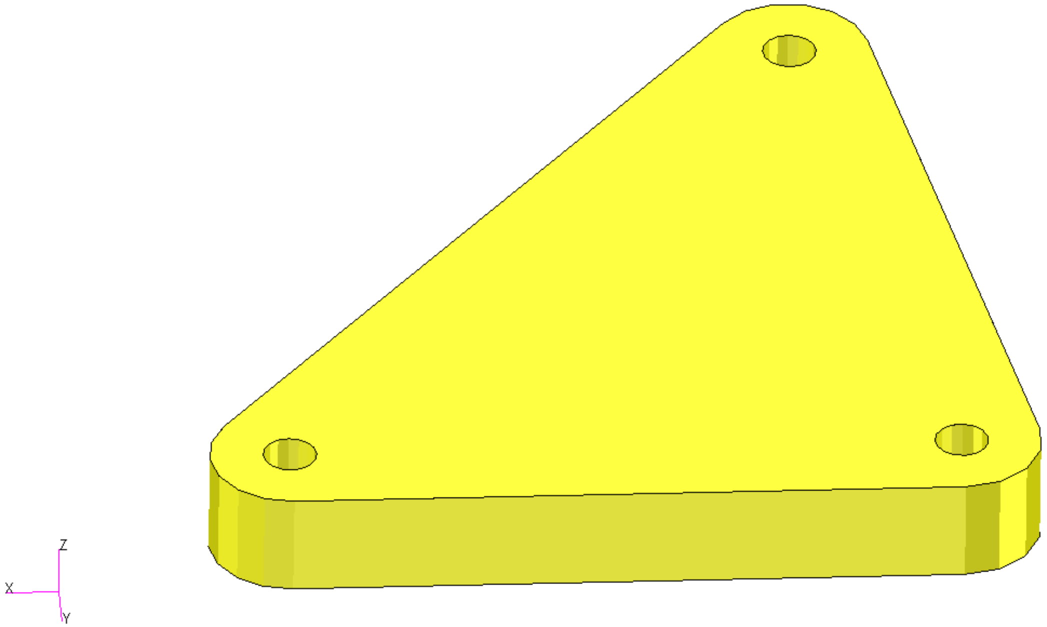
Before Starting

This example requires MSC Nastran 2017 or newer.

Goal: Use Nastran SOL 200 Optimization

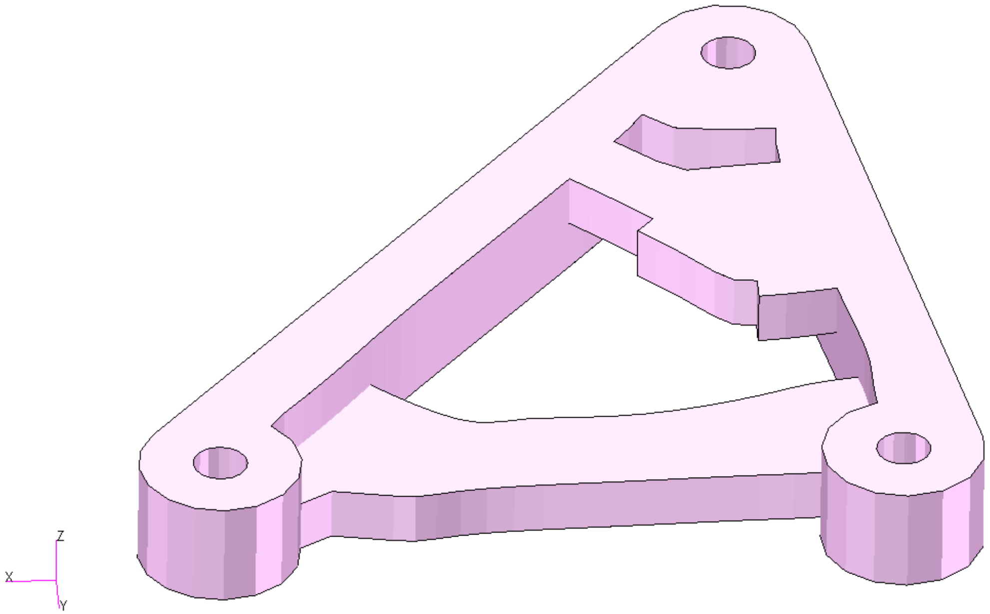
Before Optimization

- Mass: 18 kg

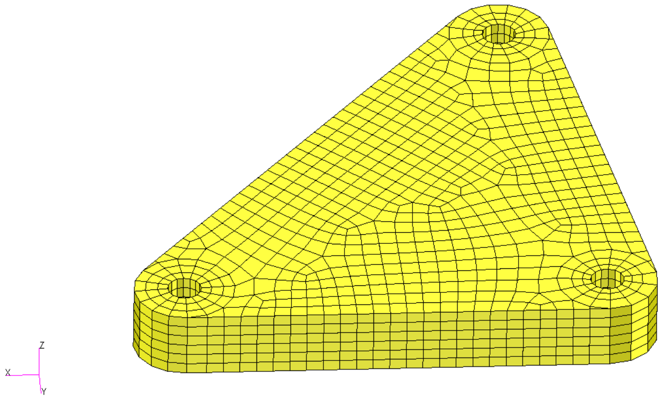


After Optimization

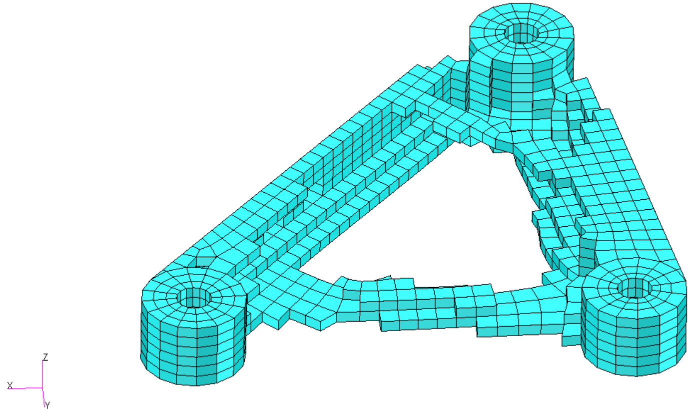
- Mass: 9.3 kg
- Prevent excessive stress and displacements



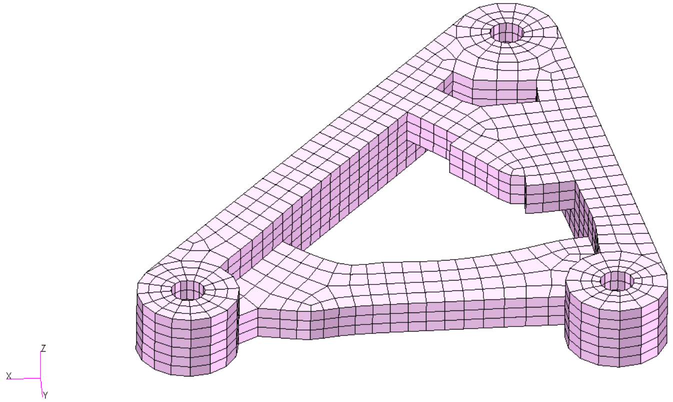
Goal: Use Nastran SOL 200 Optimization



1) Initial Design



2) Proposed Topology Solution



3) Final Design

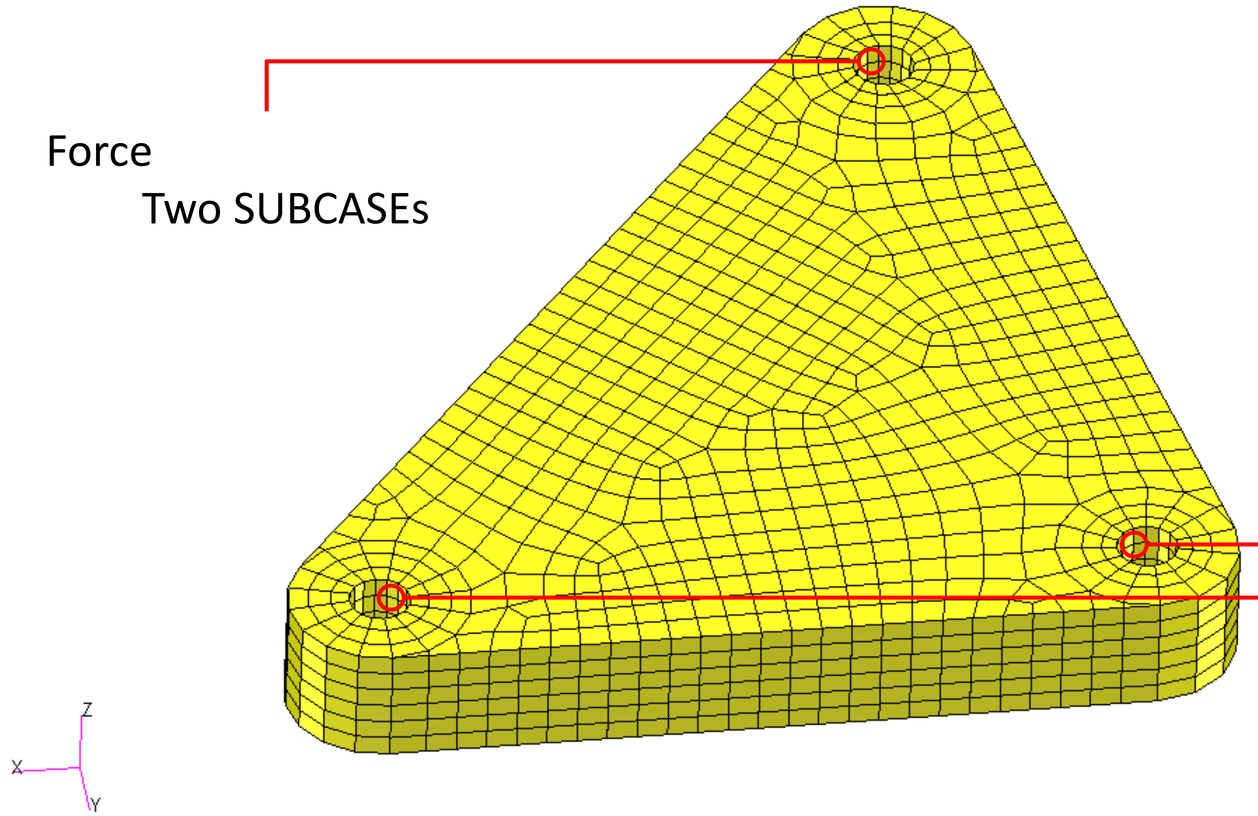
Details of the structural model

Units: m, N, MPa

Material:
E: 200E9 Pa
v: .3
p: 7850 kg/m^3

Force
Two SUBCASEs

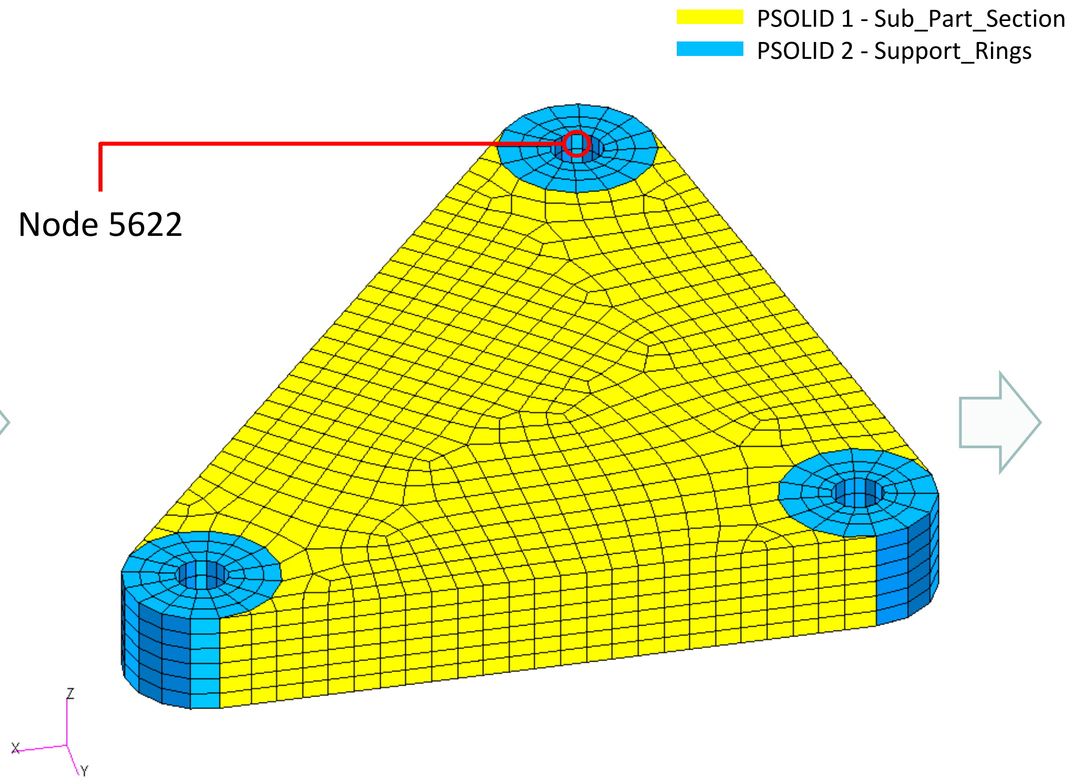
Constrained



Optimization Problem Statement

Design Region/Variables

x1: PSOLID 1



Design Objective

r0: Minimize fractional mass (FRMASS)

Design Constraints

r1: Z displacement at node 5622 (GRID 5622)

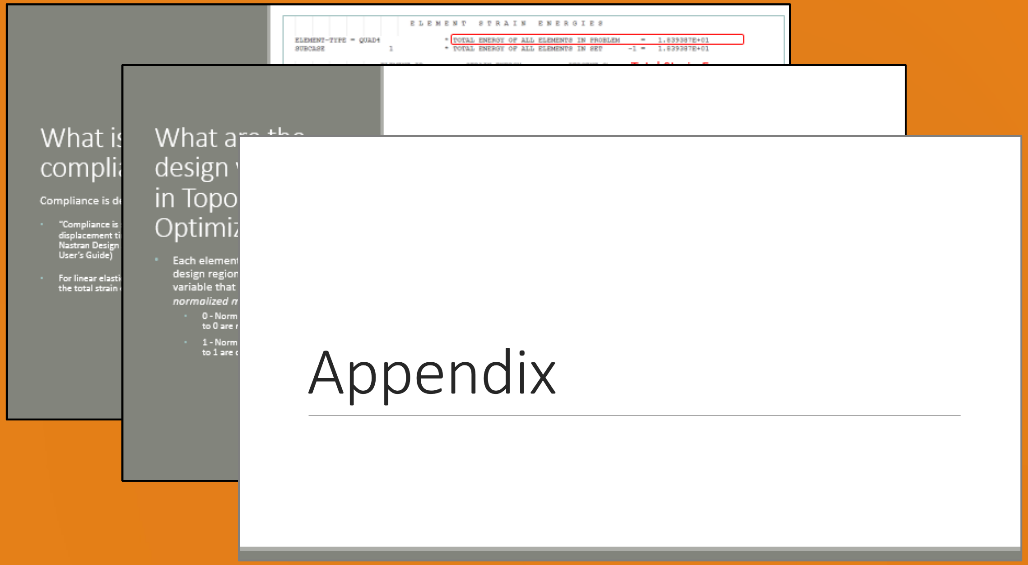
r1 < .0008 m

x1: Maximum allowable stress, 2.0E8 Pa, on design region x1

More Information Available in the Appendix

The Appendix includes information regarding the following:

- Frequently Asked Questions
  - What are the design variables in Topology Optimization?
  - What is FRMASS or Fractional Mass?
  - What is compliance?
  - How can non-critical elements be removed from the design?
- Topology Optimization Workflows
- Viewer Web App
  for Topology Optimization Post Processing



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
   - Topology Optimization and Structural Results

Special Topics Covered

Maximum Allowable Stress - A stress constraint cannot be created normally as is done in Size or Topometry optimization. A special option is available to specify a stress constraint for Topology Optimization.

SOL 200 Web App Capabilities

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

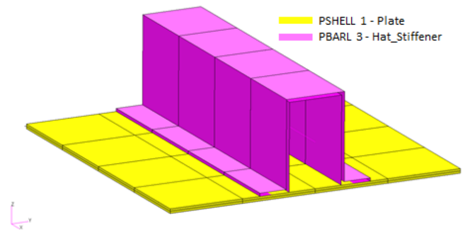
Compatibility

- 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.

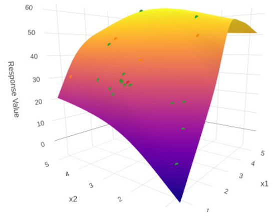
Benefits

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

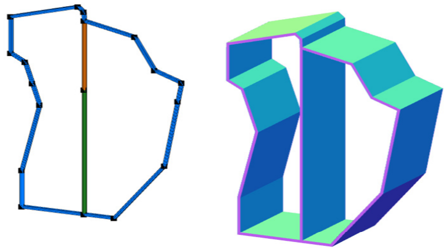
Web Apps



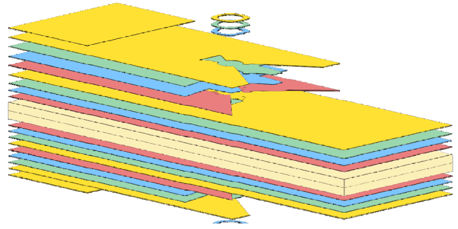
Web Apps for MSC Nastran SOL 200
Pre/post for MSC Nastran SOL 200. Support for size, topology, topometry, topography, multi-model optimization.



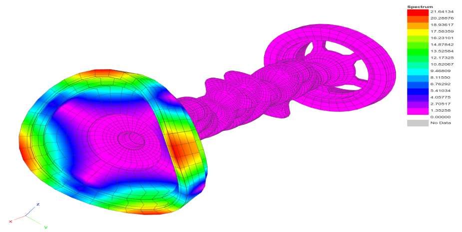
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



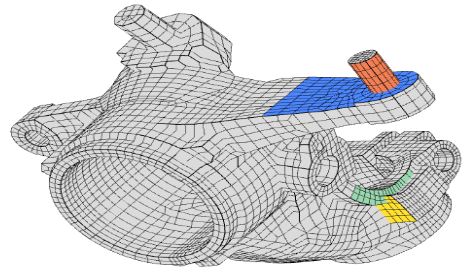
PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically



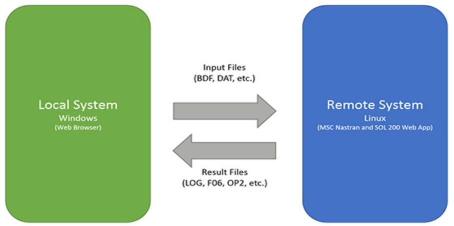
Ply Shape Optimization Web App
Optimize composite ply drop-off locations, and generate new PCOMPG entries



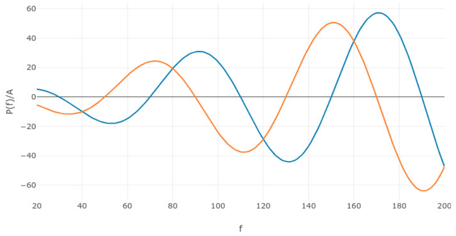
Post-processor Web App
View MSC Nastran results in a web browser on Windows and Linux



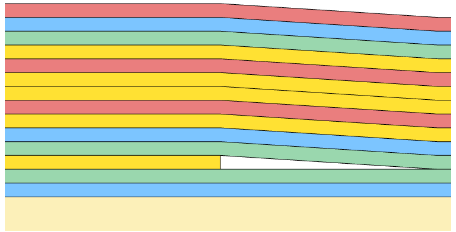
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



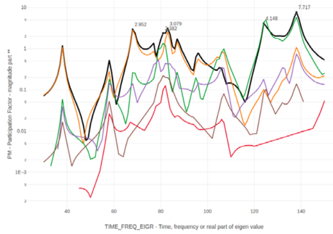
Remote Execution Web App
Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



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



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

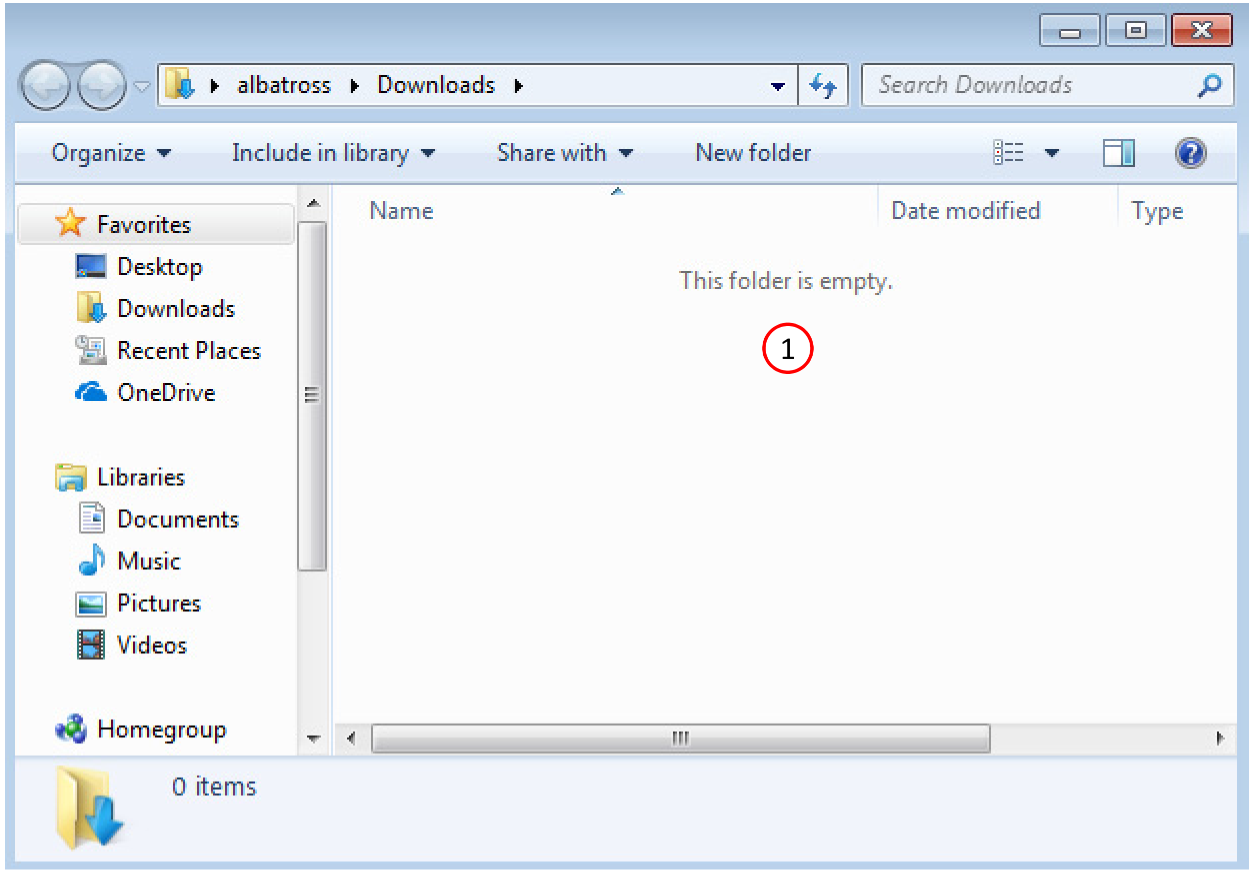


HDF5 Explorer Web App
Create graphs (XY plots) using data from the H5 file

Before Starting

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

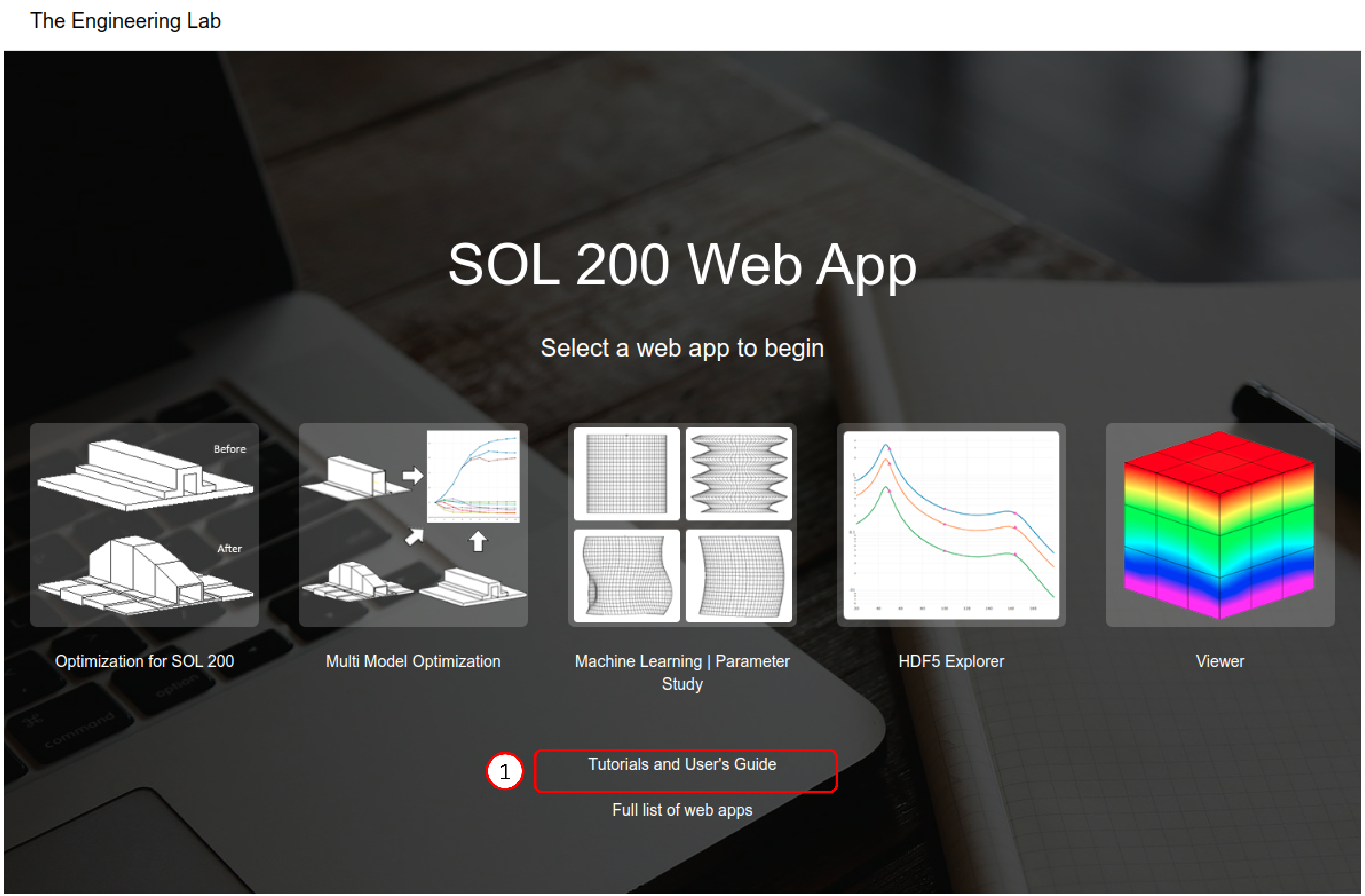
- Throughout this workshop, you will be working with multiple file types and directories such as:
  - .bdf/.dat
  - nastran_working_directory
  - .f06, .log, .pch, .h5, etc.
- To minimize confusion with files and folders, it is encouraged to start with a clean directory.



Go to the User's Guide

1. Click on the indicated link

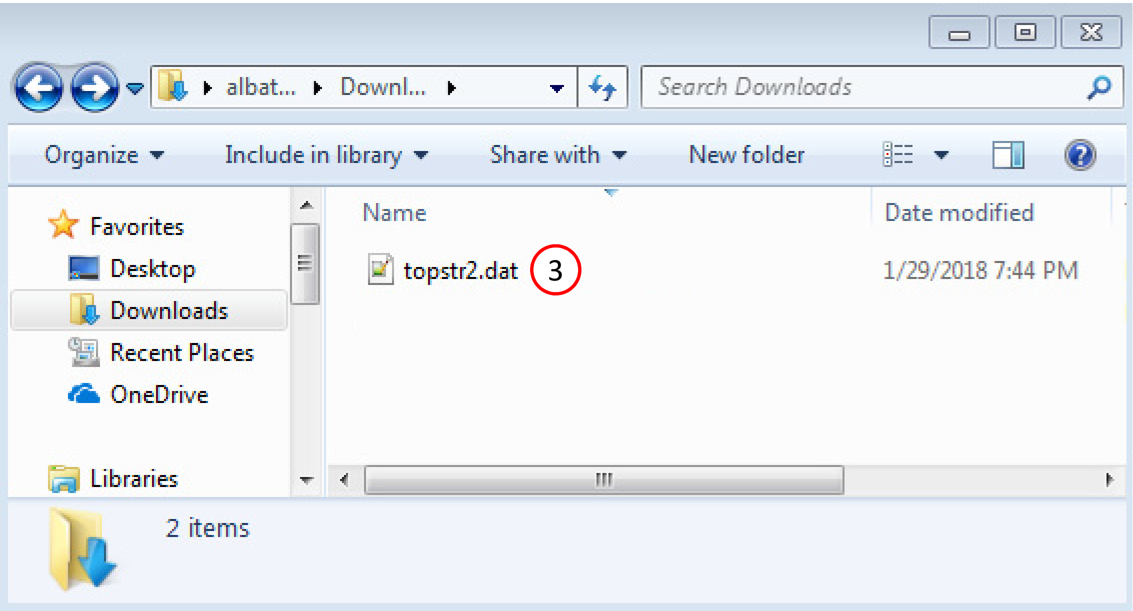
- The necessary BDF files for this tutorial are available in the Tutorials section of the User's Guide.



Obtain Starting Files

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

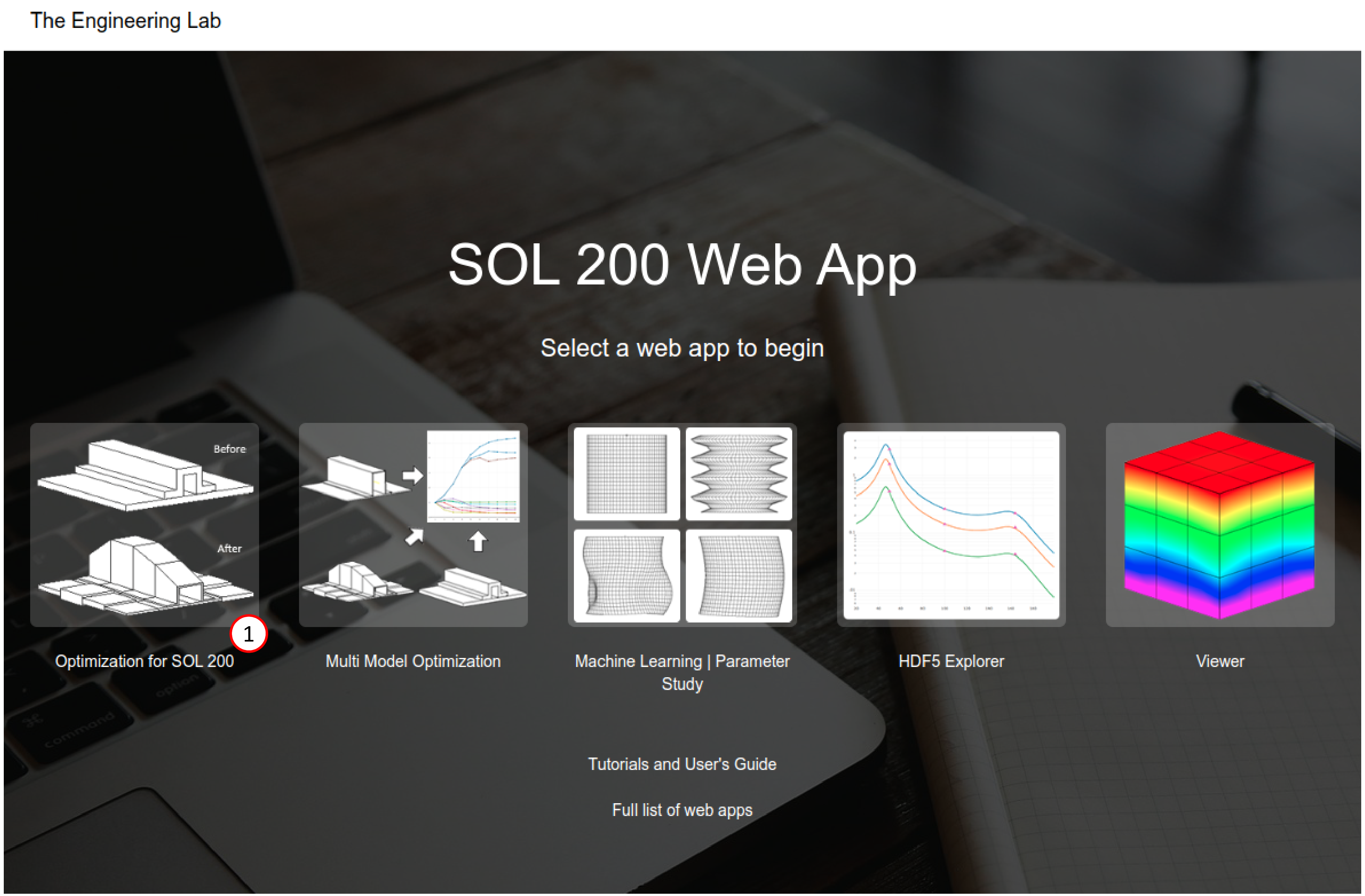
- When starting the procedure, all the necessary BDF files must be collected together.



Open the Correct Page

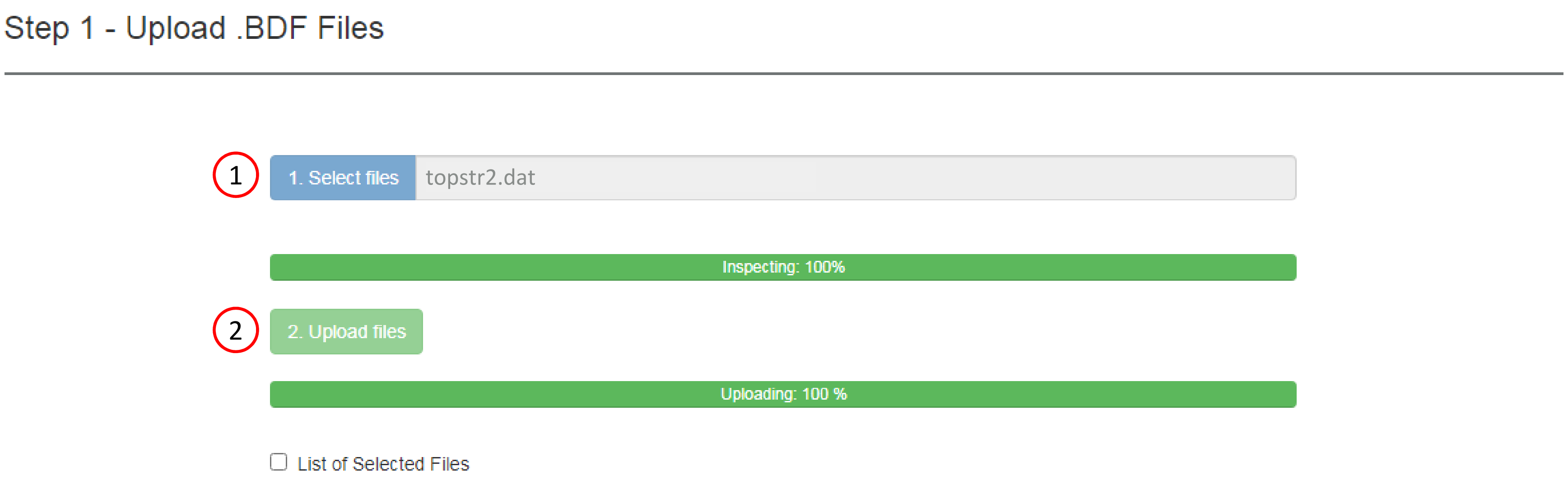
1. Click on the indicated link

- 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 topstr2.dat
2. Click Upload Files

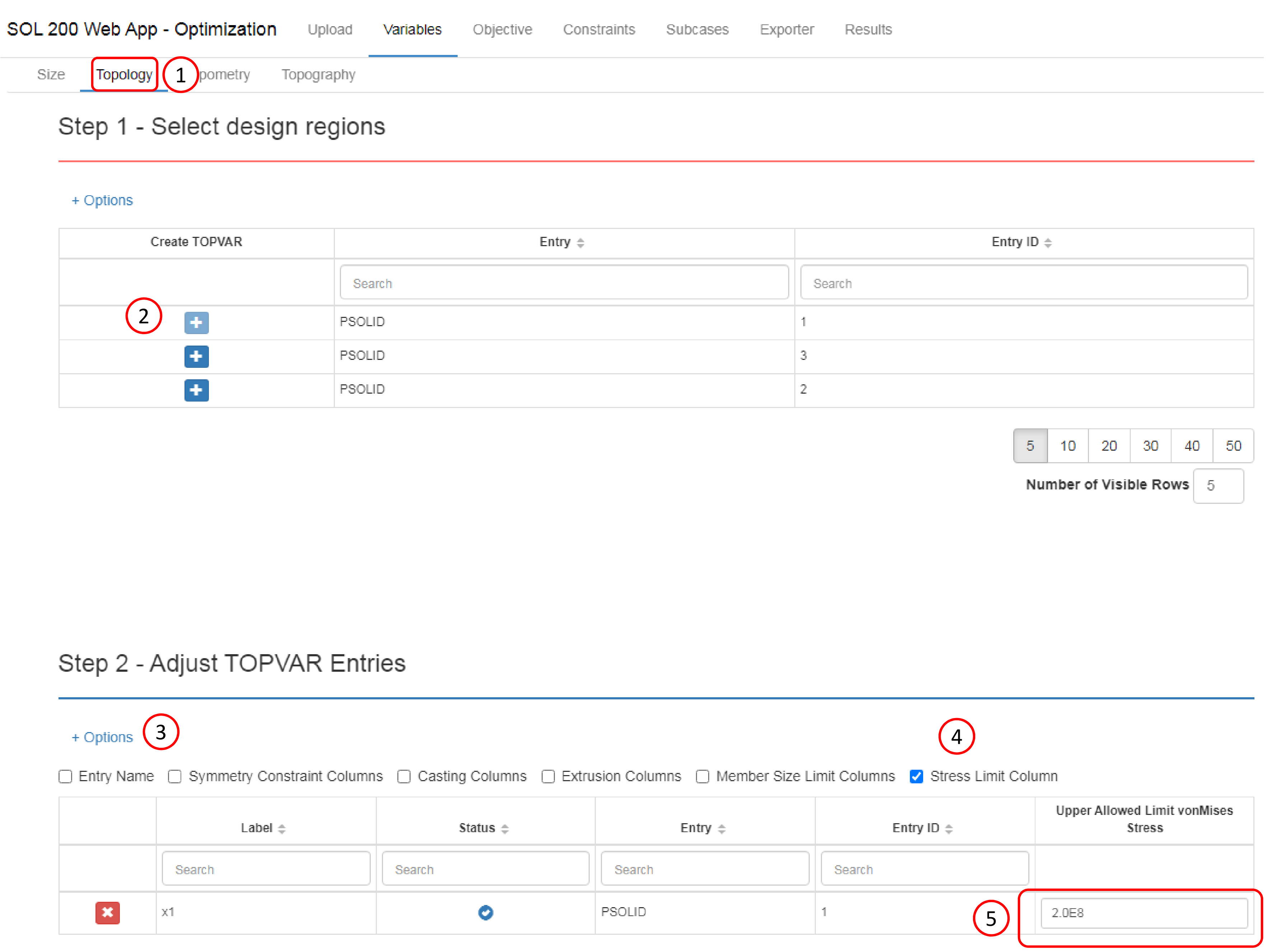


- The process starts by uploading all the necessary BDF files. The BDF files can be files of your own or files found in the Tutorials section of the User's Guide.

Create Design Region

1. Click Topology
2. Click on the plus (+) icons to set PSOLID 1 as a Design Region
3. Click + Options
4. Mark the checkbox for Show Stress Limit Column
5. Set the following for the design region
    - Upper Allowed Limit von Mises Stress: 2.0E8

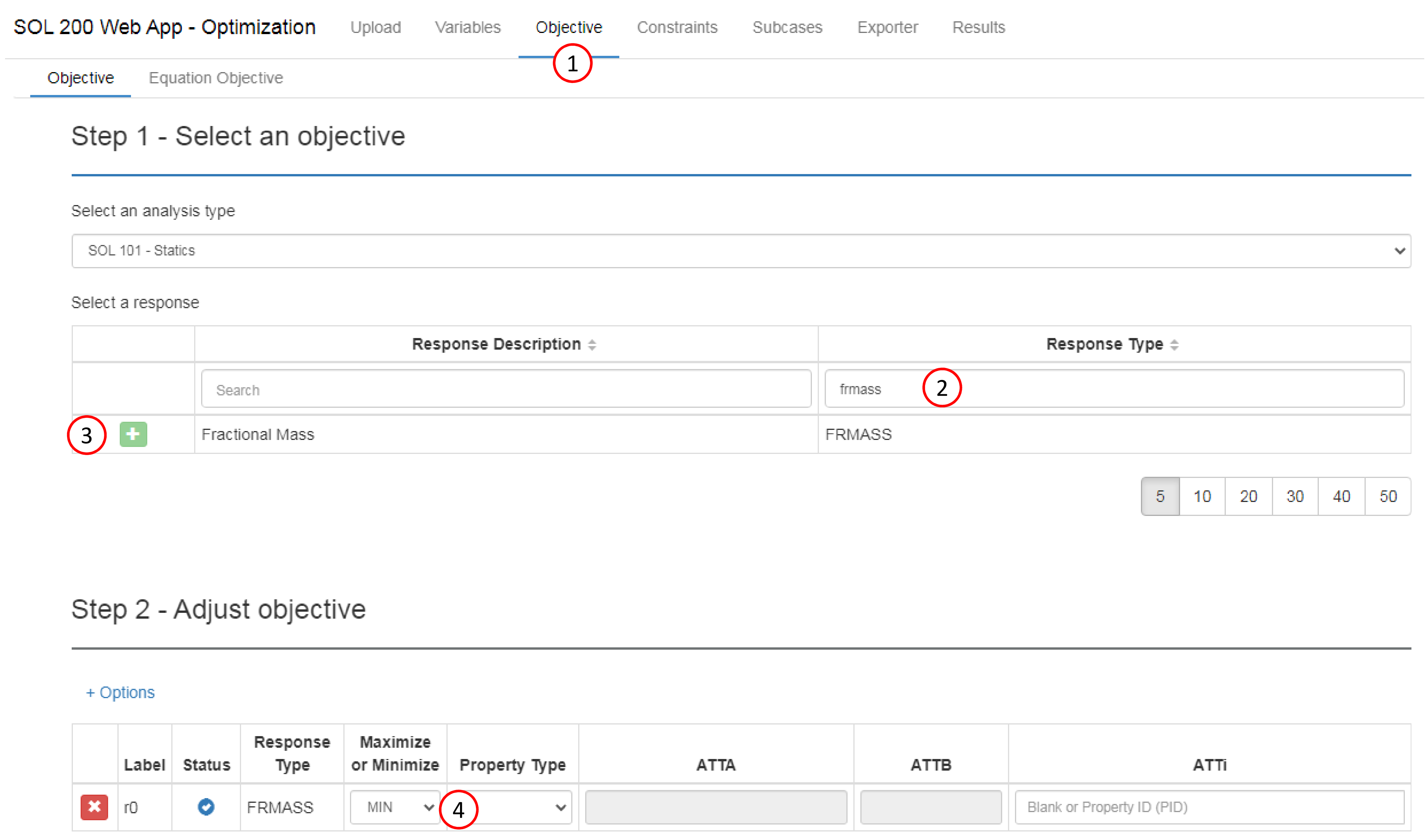
- When a topology design region is set, one topology variable is created for each element in the design region. Each topology variable controls the density and stiffness for the respective element, see the appendix for additional details.
- If PSOLID 1 has 500 elements associated and is configured as a design region, then there will be 500 design variables created.
- Once the stress constraint is specified, it applies for all design regions. The von Mises stress is constrained in this example. It should be noted that the stress of each element is not constrained. Instead, a single equivalent stress for the entire model and subcase is constrained.



Create Design Objective

1. Click on Objective
2. Type ‘frmass’ in the search box
3. Select the plus(+) icon for Fractional Mass
4. The objective with label r0 is created. The objective is to minimize (MIN)

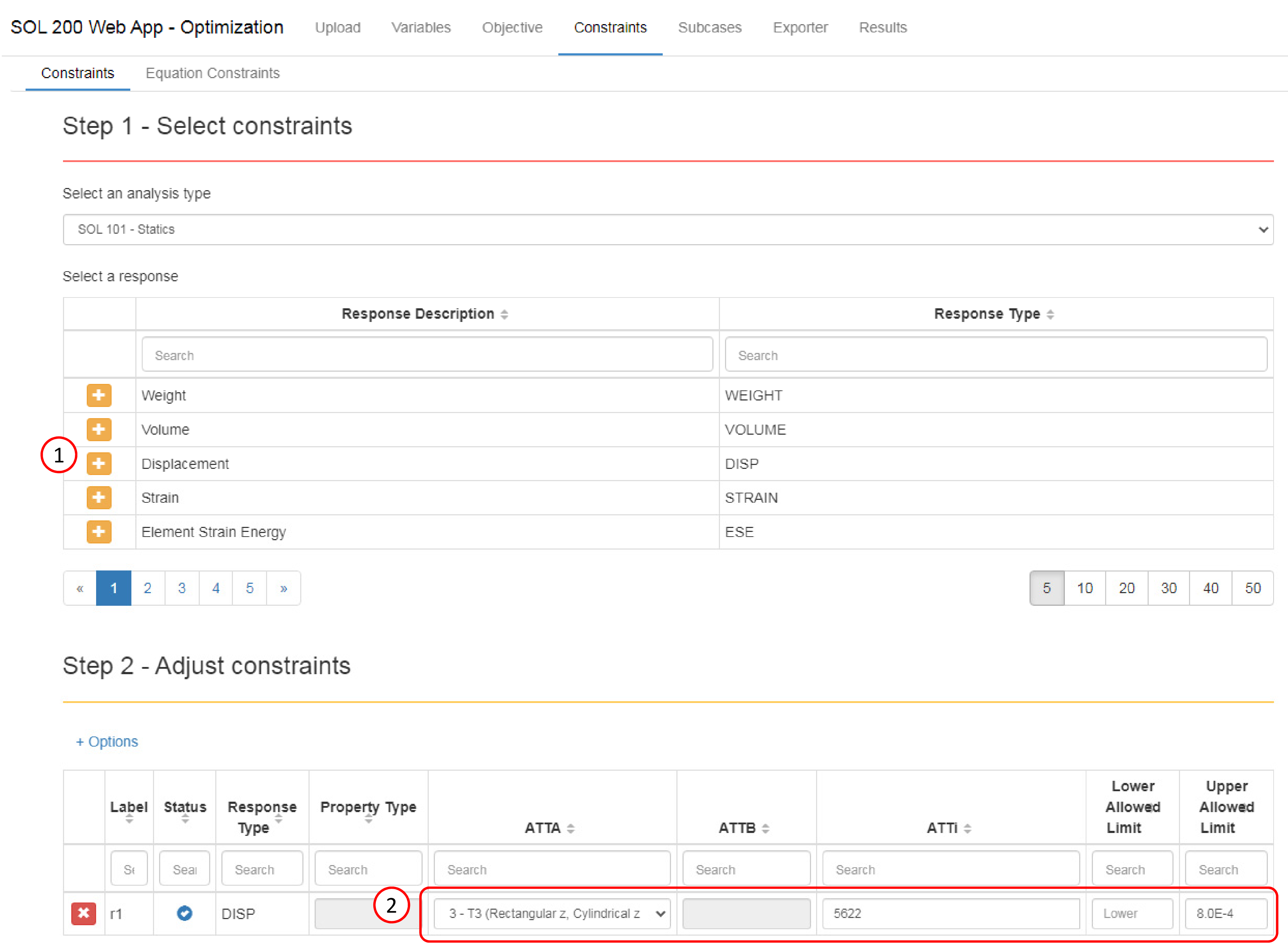
- Compliance is equal to twice the total strain energy. By minimizing the compliance/strain energy, the stiffness of the model is being maximized. See the appendix for additional details regarding compliance.



Create Design Constraints

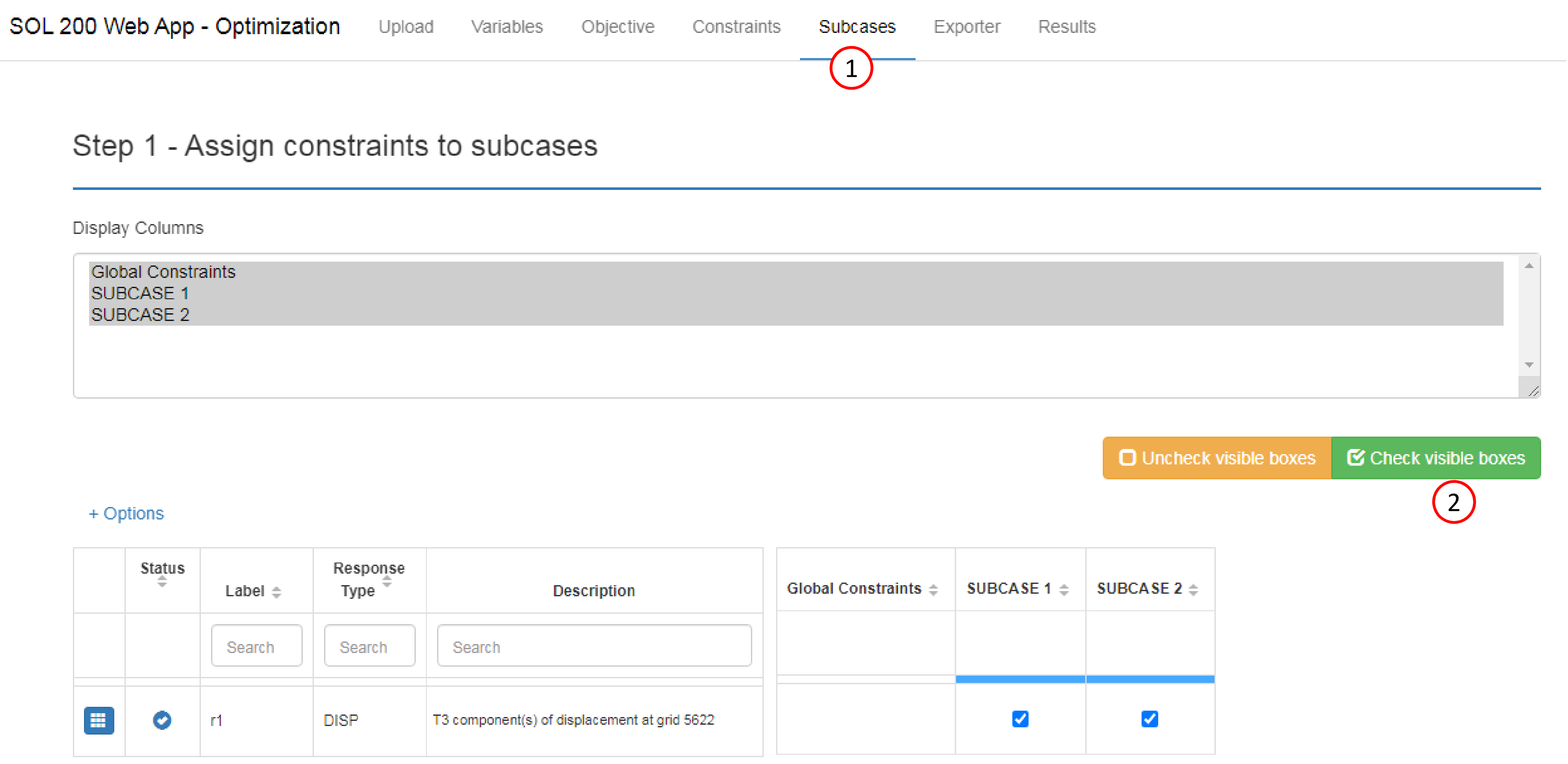
1. Click on the plus (+) icon for Displacement
2. Configure the following for r1:
   - ATTA: 3 - T3 (Z Component)
   - ATTi: 5622 (Node/GRID 5622)
   - Upper Allowed Limit: 8.0E-4

- Topology optimization works best when working with a small number of responses, e.g. Compliance, Fractional Mass, a single von Misses stress.
- In this tutorial a single displacement is constrained. The number of constraints should be kept to a minimum. For example, constraining multiple displacements at various nodes is not advised.



Assign Constraints to Load Cases (SUBCASES)

1. Click Subcases
2. Click Check visible boxes



- r1 or DISP constraint has been assigned to SUBCASE 1 and SUBCASE 2

Configure Optimization Settings

1. Click Settings
2. Set DESMAX to 100

- This example has been previously performed with a DESMAX value of 50, but the optimization ended with RUN TERMINATED DUE TO MAXIMUM NUMBER OF DESIGN CYCLES.
- A DESMAX value of 100 is used instead to allow for additional design cycles with the goal of obtaining convergence.

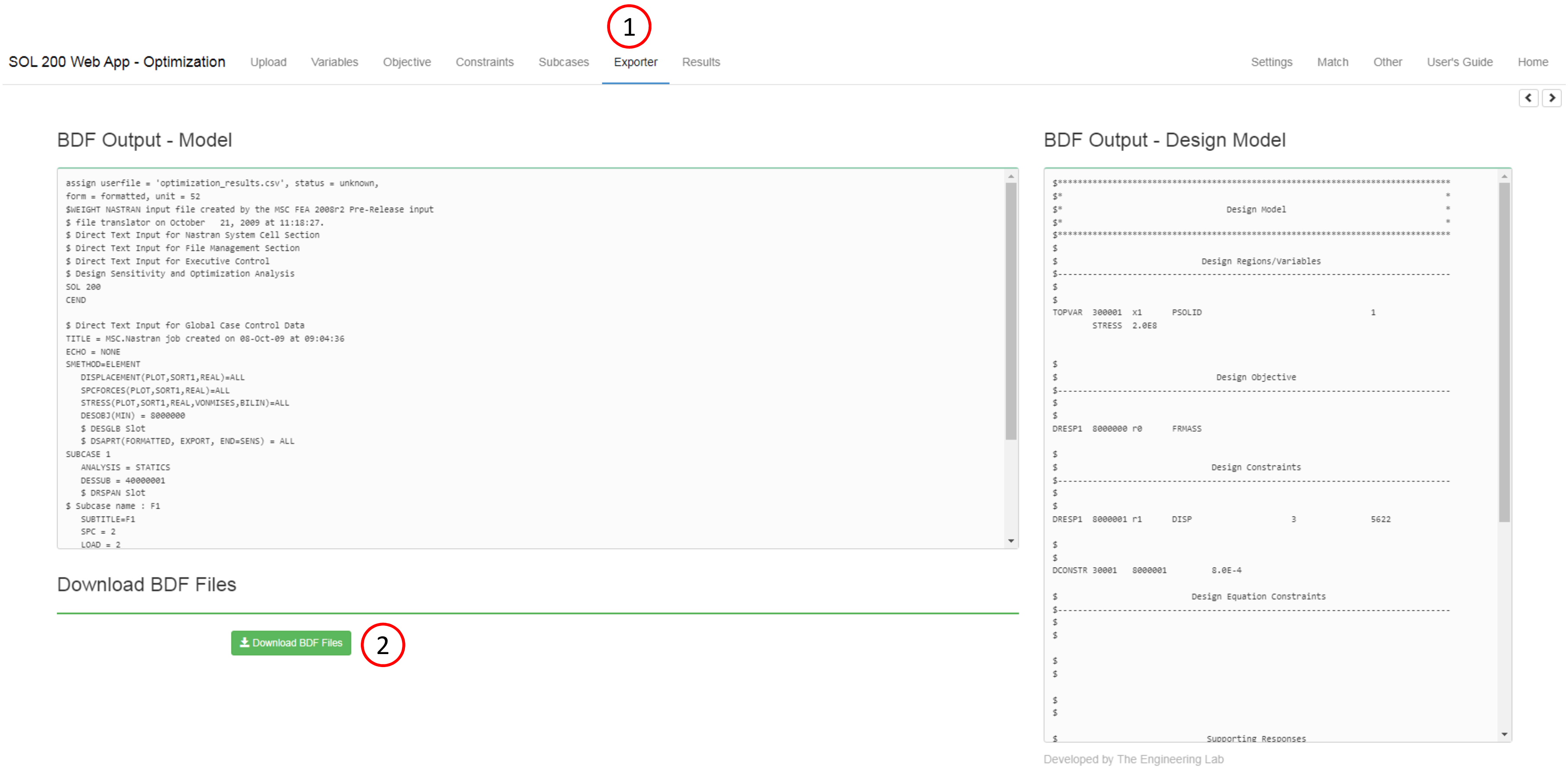
SOL 200 Web App - Optimization | Upload | Variables | Objective | Constraints | Subcases | Exporter | Results | Settings (1) | Match | Other

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	☑ 100 (2)
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	☑ 12 - Print constraints and respons
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

Export New BDF Files

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

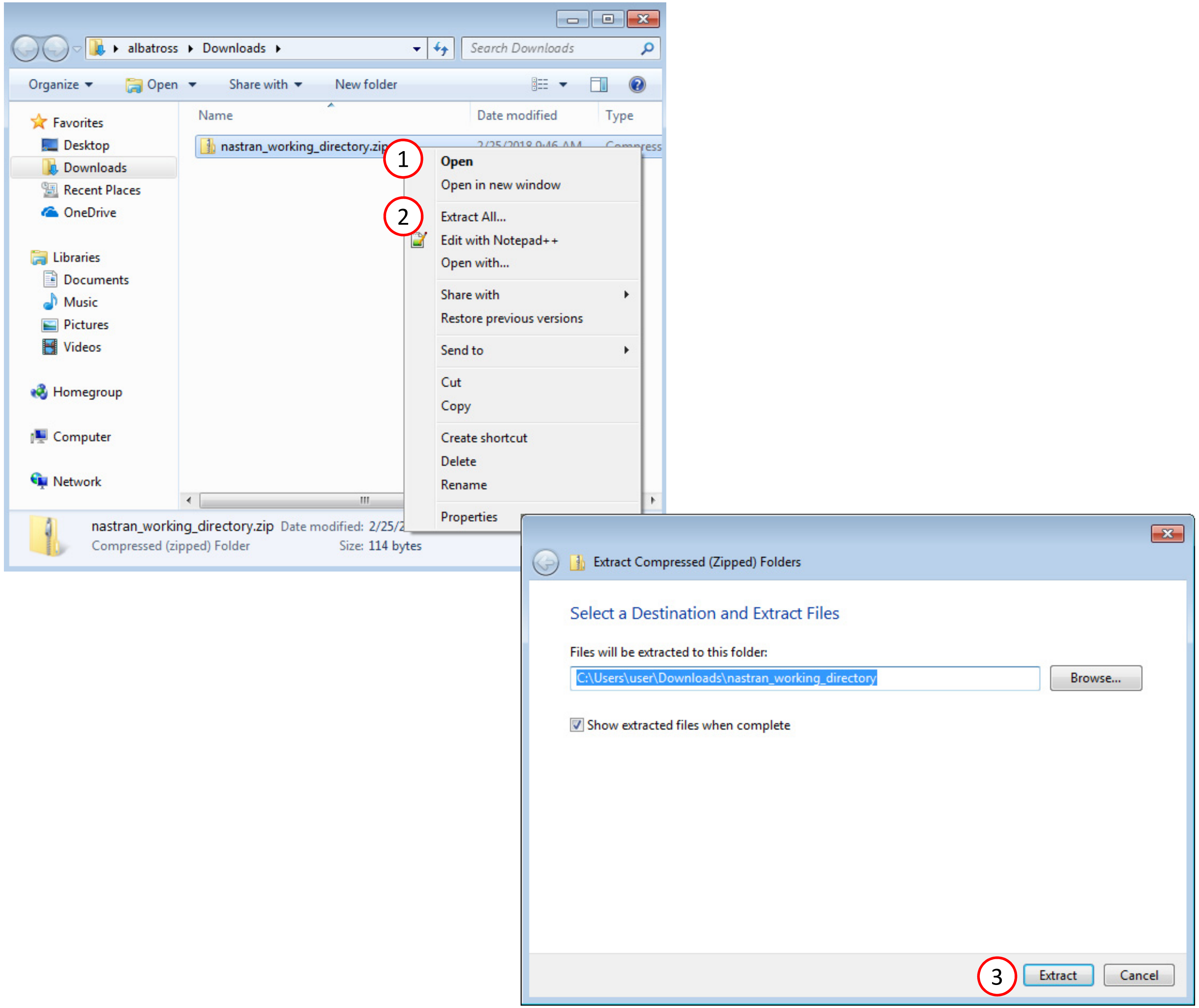


- When the download button is clicked a new file named "nastran_working_directory" is downloaded. If the file already exists in your local folder, the folder name is appended with a number, e.g. "nastran_working_directory (1).zip"

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

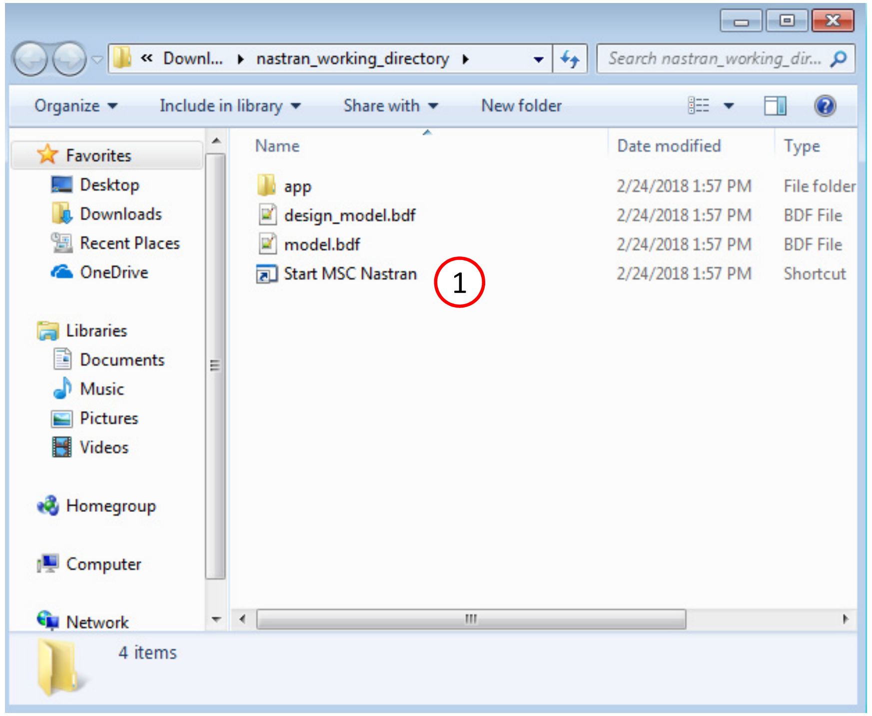
- After a successful optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- One can run the Nastran job on a remote machine as follows:
  1) Copy the BDF files and the INCLUDE files to a remote machine. 2) Run the MSC Nastran job on the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

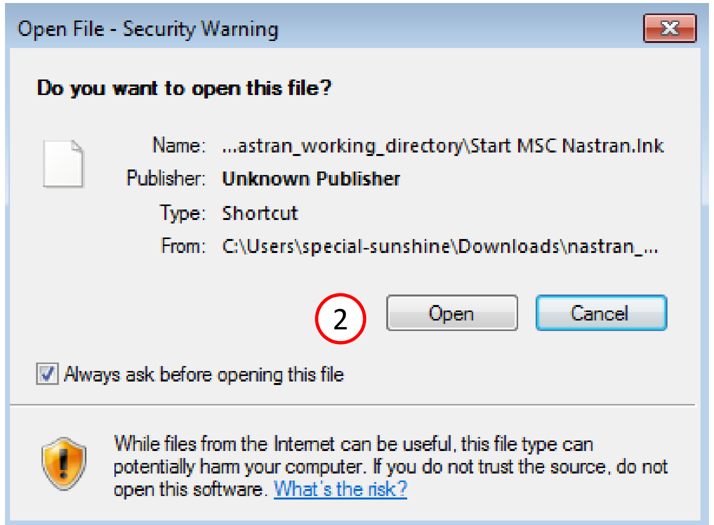
Using Linux?

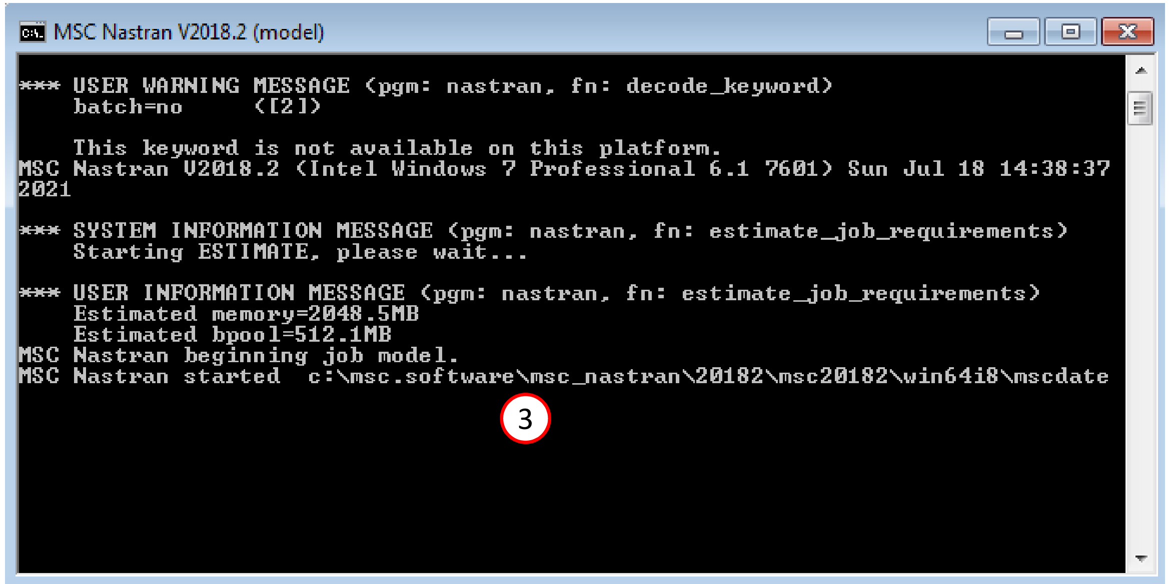
Follow these instructions:
1) Open Terminal
2) Navigate to the nastran_working_directory
   cd ./nastran_working_directory
3) Use this command to start the process
   ./Start_MSC_Nastran.sh

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

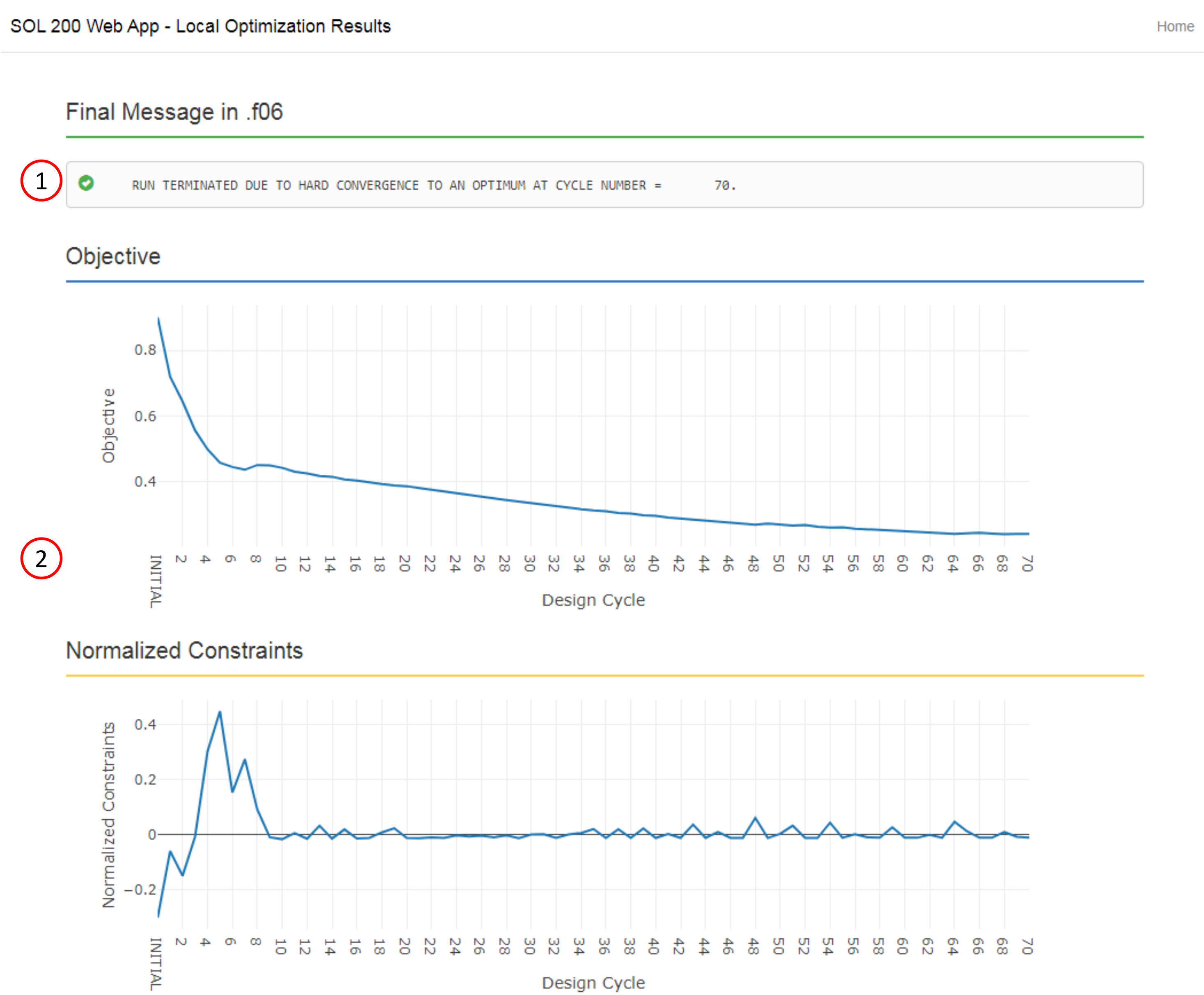
Name	Status of Job	Design Cycle	RUN TERMINATED DUE TO
model.bdf	Running	None	

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 and normalized constraints can be reviewed.

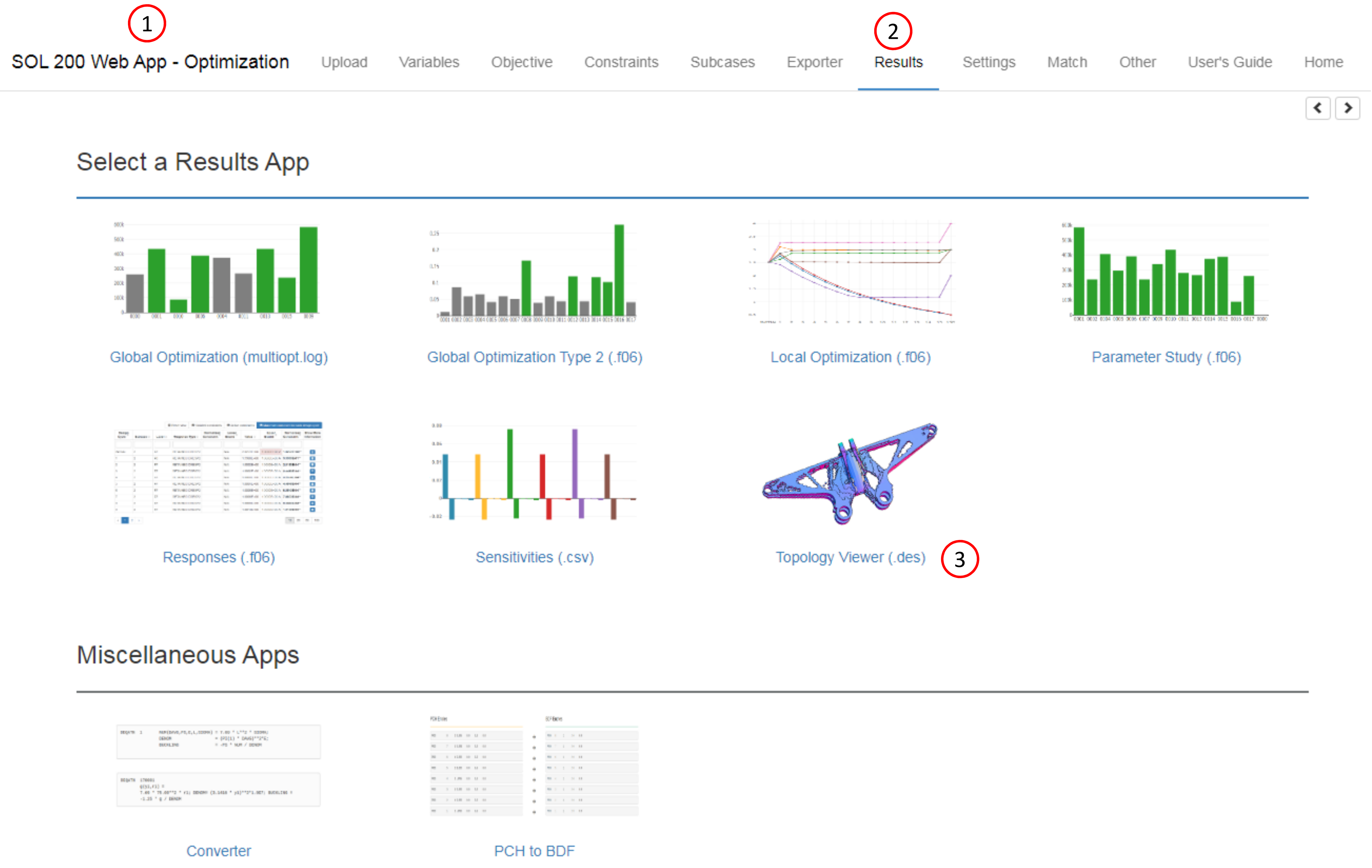
- After an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.



Review Optimization Results

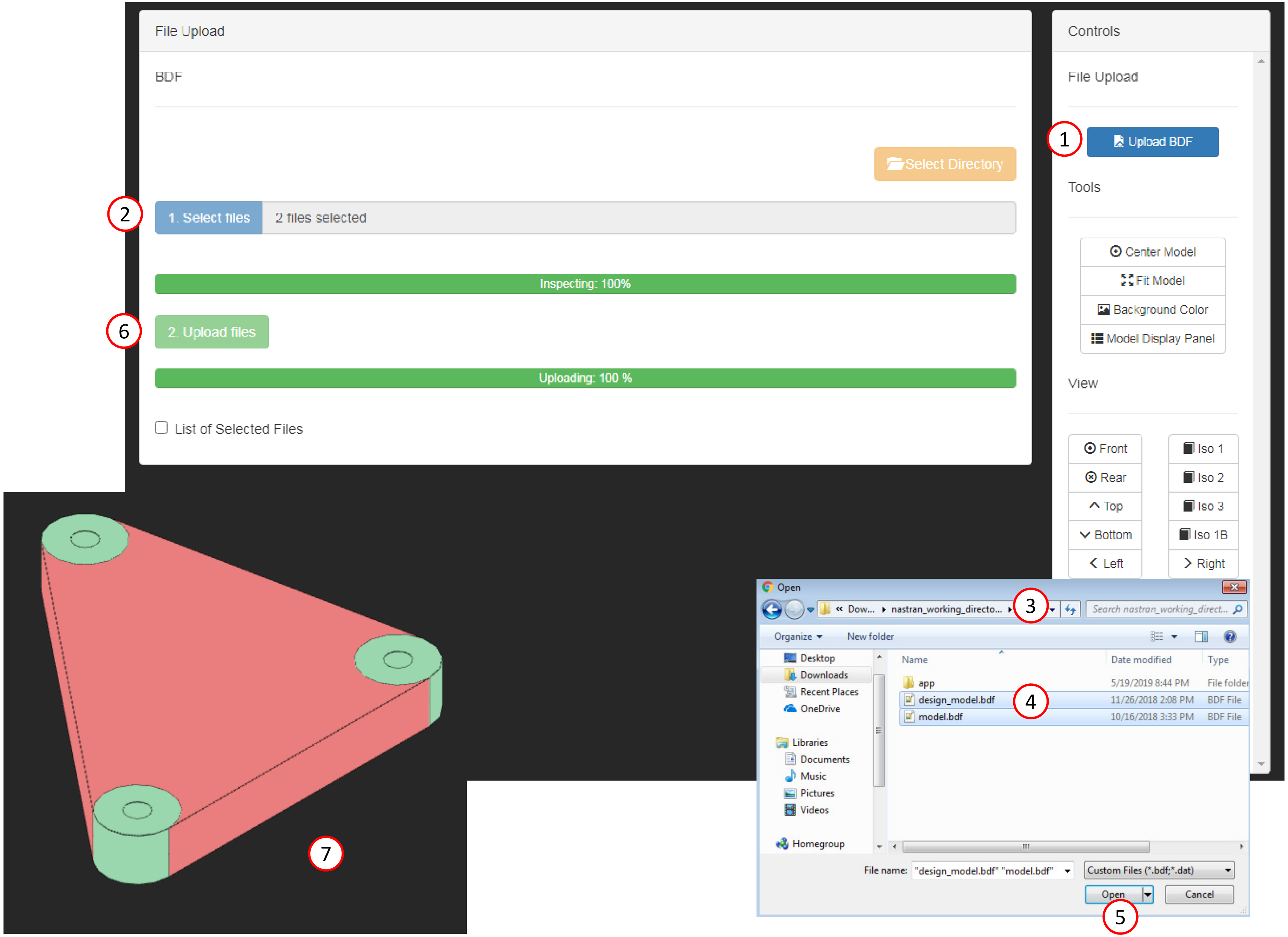
1. Return to the Optimization web app
2. Go to the Results section
3. Click Topology Viewer

- The Topology Viewer is capable of displaying topology results and is accessed from the Results section of the Optimization web app. The appendix has additional information regarding capabilities of the Topology Viewer.



Review Optimization Results

1. Click Upload BDF
2. Click 1. Select files
3. Navigate to directory nastran_working_directory
4. Select the model.bdf and design_model.bdf files.
5. Click Open
6. Click 2. Upload files
7. The model is displayed



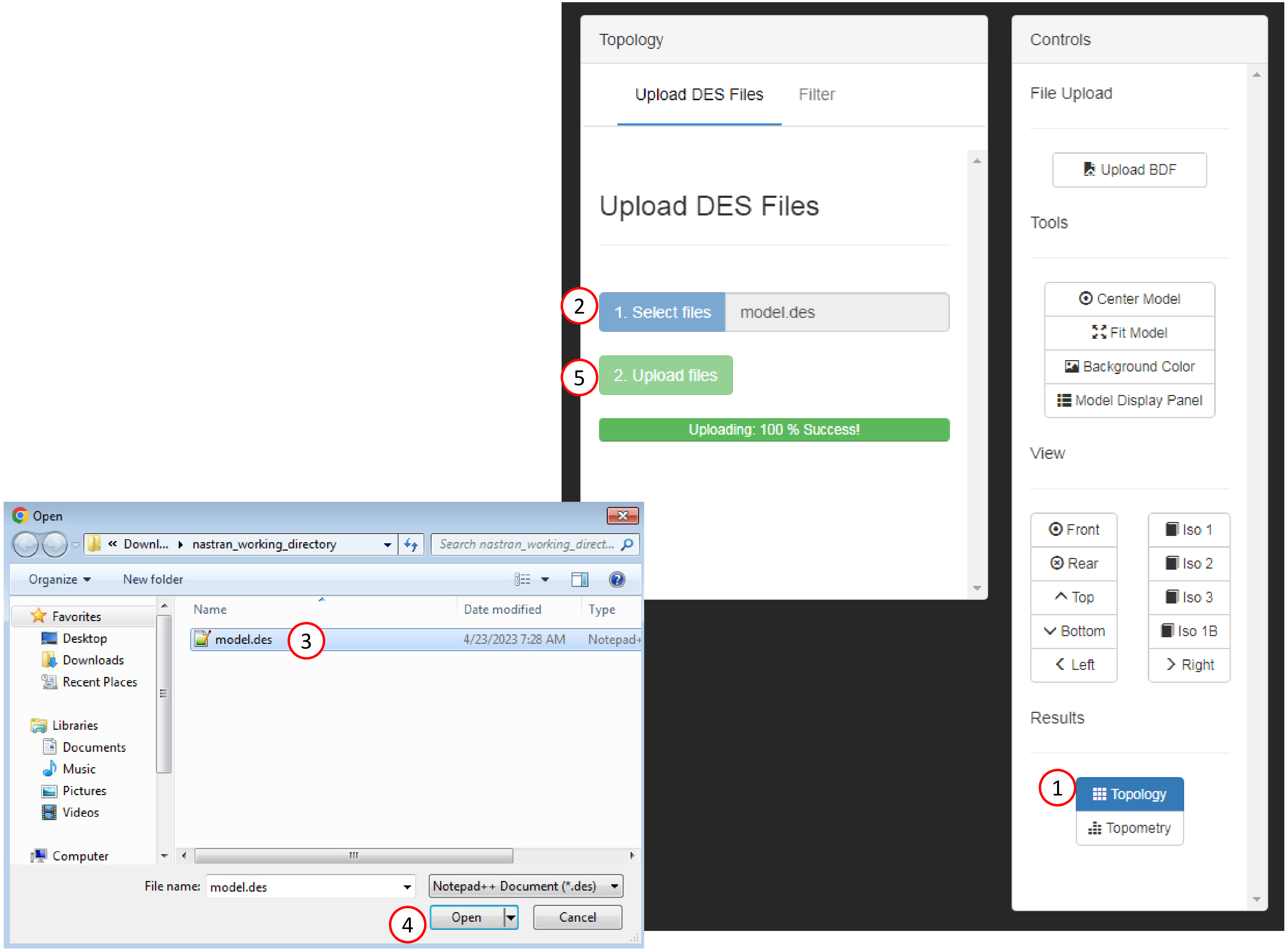
- During file upload, reading and parsing process, the web app does not report the reading progress for large files. Know that the web app parses files at a rate of 10MB every 25 seconds.

Review Optimization Results

1. Click Topology
2. Click 1. Select files
3. Select the model.des file
4. Click Open
5. Click 2. Upload files

The results of the topology optimization are now accessible within the Viewer web app.

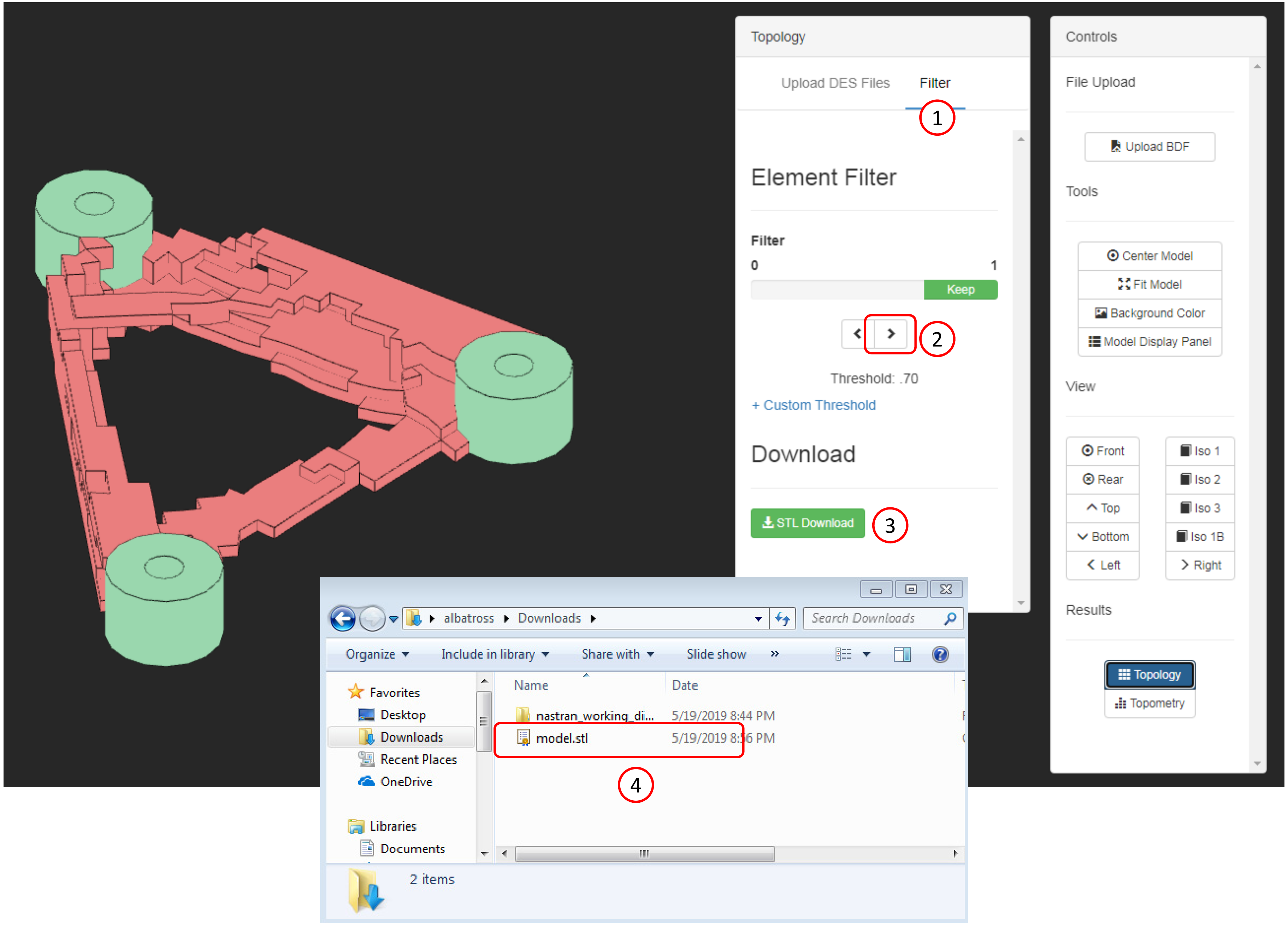
- When the DES file is uploaded, the topology results are automatically displayed. By default, elements with a topology variable value greater than a threshold of .3 are displayed. The threshold can be modified.



Review Optimization Results

1. Click Filter
2. Click the right arrow to remove elements below the threshold value
3. Click STL Download
4. The displayed model has been downloaded to an STL file and may be imported to separate CAD package or FEA pre processor

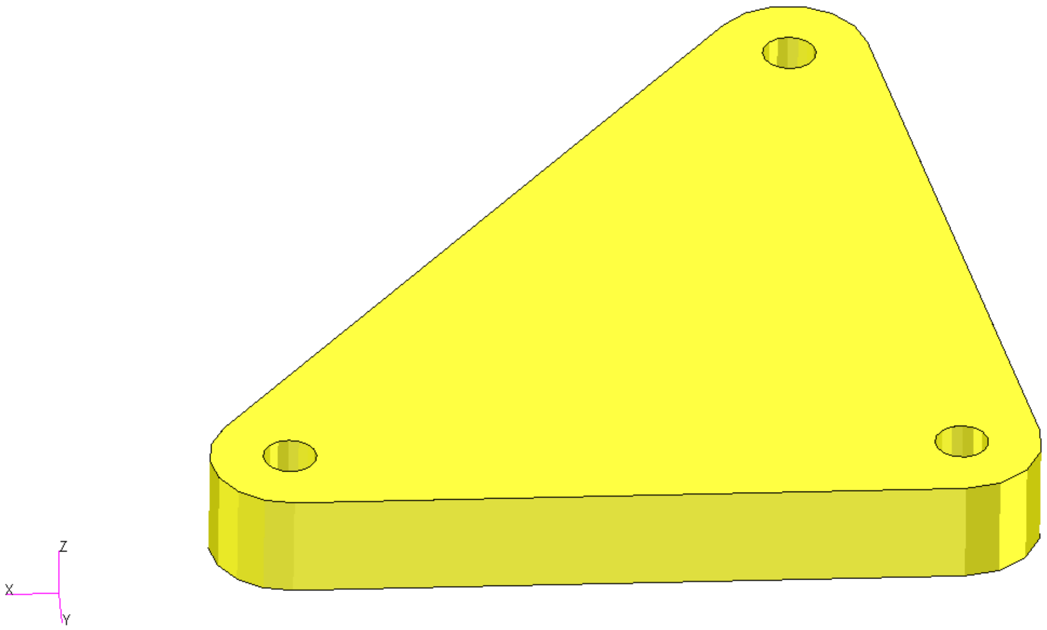
- A topology variable value close to 1 indicates the element is very important and should be kept in the design. It is not recommended to go beyond a threshold of .7 since very critical elements would be removed. Elements with an topology variable value close to 0 are not critical and can be removed.
- Common thresholds to use are typically in the range of .3 to .7



Results

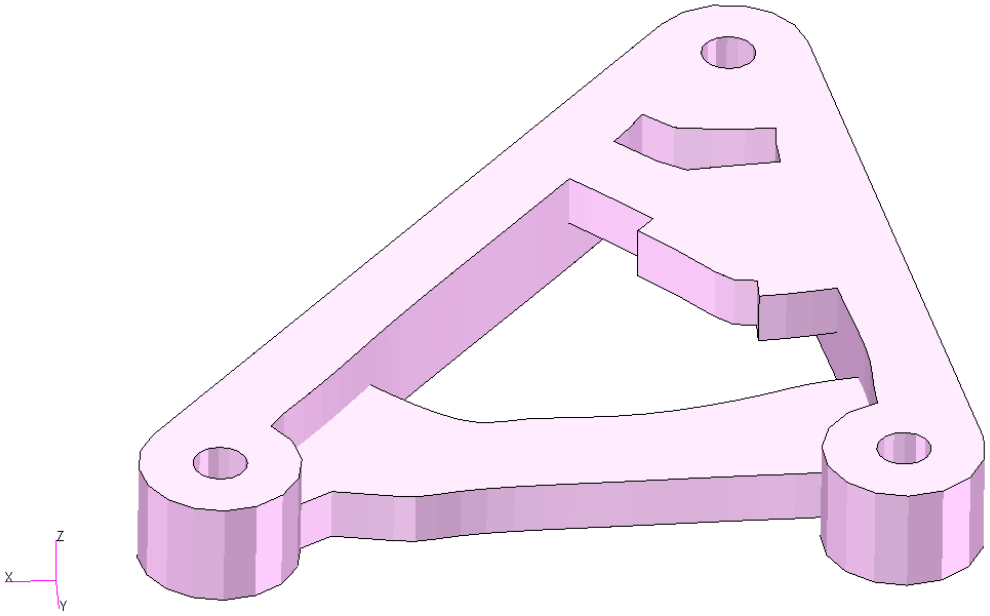
Before Optimization

- Mass: 18 kg



After Optimization

- Mass: 9.3 kg
- Prevent yield and excessive displacement



End of Tutorial

Appendix

Appendix Contents

- Frequently Asked Questions
  - What are the design variables in Topology Optimization?
  - What is FRMASS or Fractional Mass?
  - What is compliance?
  - How can non-critical elements be removed from the design?
- Topology Optimization Workflows
- Viewer Web App for Topology Optimization Post Processing

What are the design variables in Topology Optimization?

Consider the following topology optimization workflow.

1. A topology design region is selected.
   - 1 material
   - 2048 elements

2. At the start of an optimization, each element is assigned its own material (stiffness and density).
   - 2048 materials
   - 2048 elements

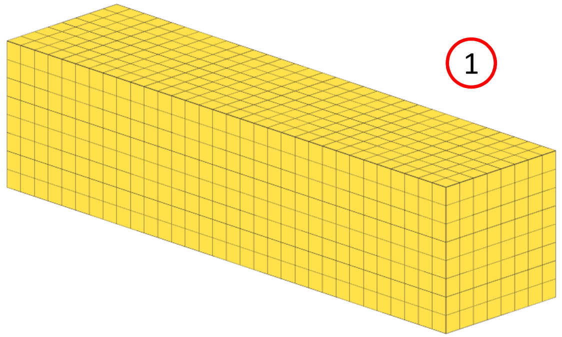
3. During the optimization, each element is given a topology variable x_i, where i is the element ID.
   - 2048 topology variables

   The topology variable x_i controls the material density and stiffness of element i via these expressions.
   - $p_i = p_0 \cdot x_i$
   - $E_i = E_0 \cdot x_i^{Penalty}$

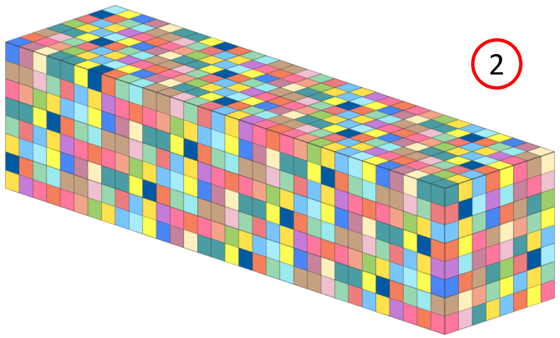
   The penalty term ranges between 2-5 and is 3 by default. The topology variable varies between 0 and 1.

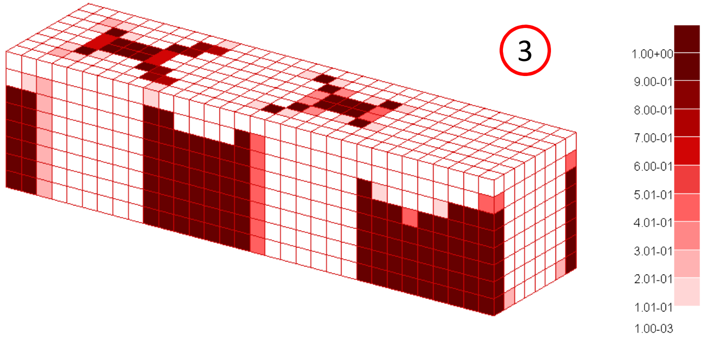
4. After the topology optimization, the user must decide which elements to keep.
   - During the topology optimization, elements are not automatically removed. It is up to the user to decide which elements to keep after the optimization.

Topology Design Region


1

Topology Optimization

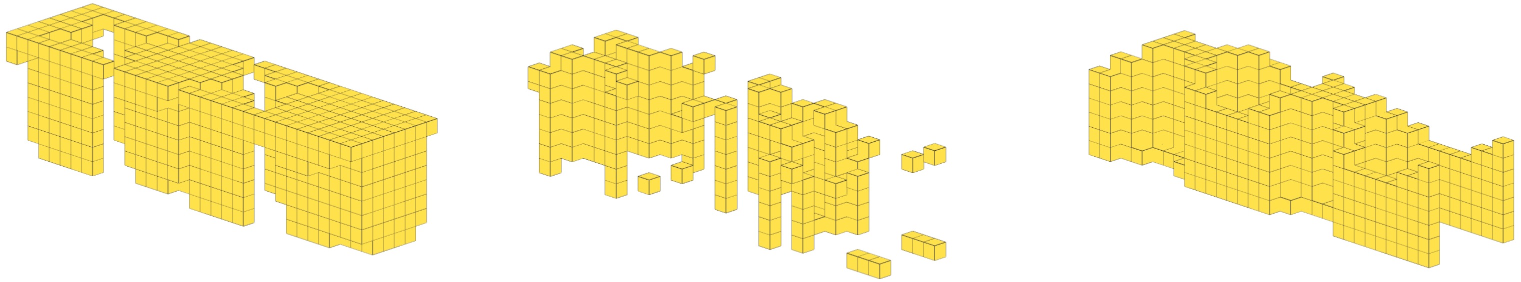

2



3

Results Post-processing

4



What are the design variables in Topology Optimization?

Many practitioners suggest keeping elements whose topology variable is in the range of 0.3 and 1.0, but do not explain the reasoning behind this suggestion.

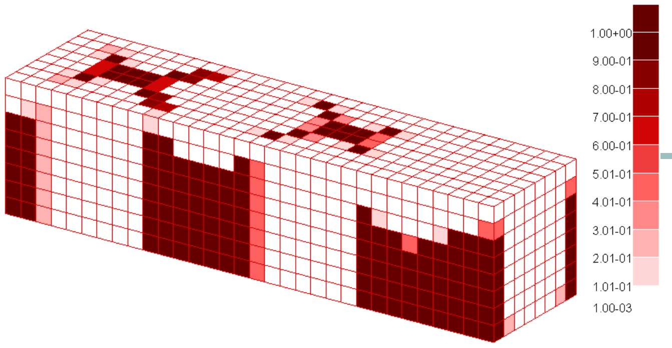
The following is an attempt to explain the suggestion.

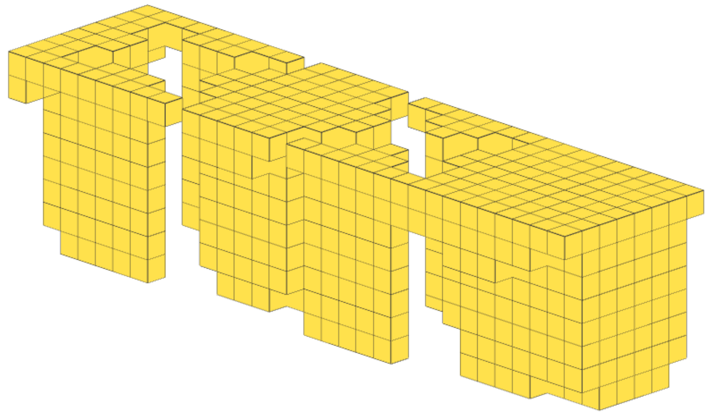
Suppose the original stiffness of the material is $E_0 = 200E9\ Pa$.

- If x_i=0.3, then
  - E_i = 200E9 Pa * 0.3^3 = 5.4E9 Pa (5.4 GPa)
  - A topology variable value of x_i=0.3 yields a stiffness on the range of wood.
- If x_i=.0056, then
  - E_i = 200E9 Pa * .0056^3 = 3.5123E4 Pa (35.123 kPa)
  - A topology variable value of x_i=.0056 yields a stiffness on the range of gelatin dessert, such as Jello.

Elements with a stiffness equivalent to Jello are negligible and may be removed from the design. Elements with a stiffness equivalent to wood are also candidates for removal. Those who use topology optimization long enough will find the suggestion of keeping elements between 0.3 and 1.0 is not absolute. With trial and error, some will find that ranges of 0.5 to 1.0 or 0.4 to 1.0 will also sometimes work. Given that the best range is often unknown, this makes topology optimization a *black art*.

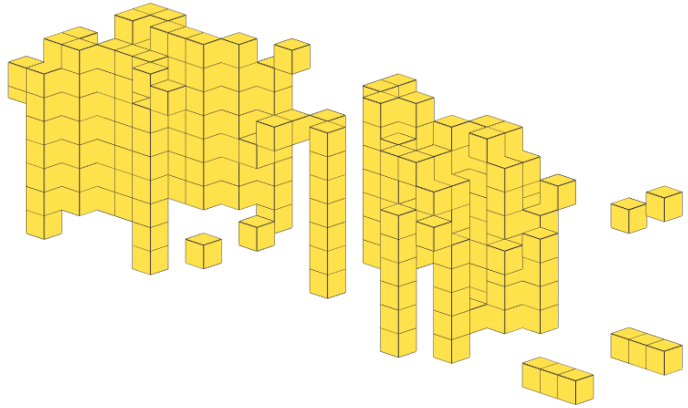
Topology Optimization Results





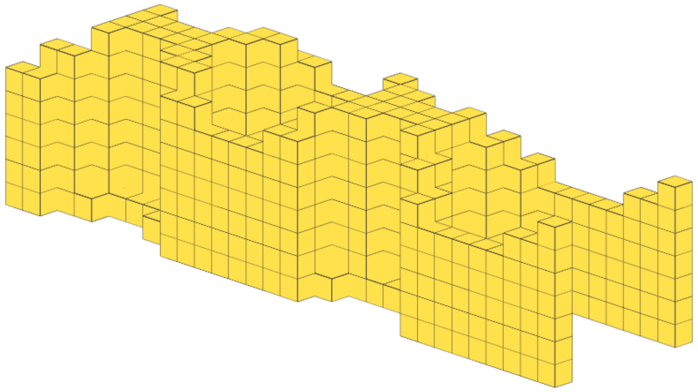
Elements in range:
0 < x_i <= .0056
0 < E_i <= 3.5123E4 Pa
Range: Jello

Do not keep



Elements in range:
.0056 < x_i <= 0.3
3.5123E4 < E_i <= 5.4E9 Pa
Range: Wood

Do not keep



Elements in range:
0.3 < x_i <= 1.0
5.4E9 Pa < E_i <= 200E9 Pa

Keep

What is FRMASS or Fractional Mass?

Since the topology variables can range between 0 and 1, the final mass will be some fraction of the original mass. This is known as the fractional mass or FRMASS.

$$\text{FRMASS} = \frac{\Sigma\, p_i \cdot v_i}{\Sigma\, p_0 \cdot v_i}$$

p_0 : The original material density

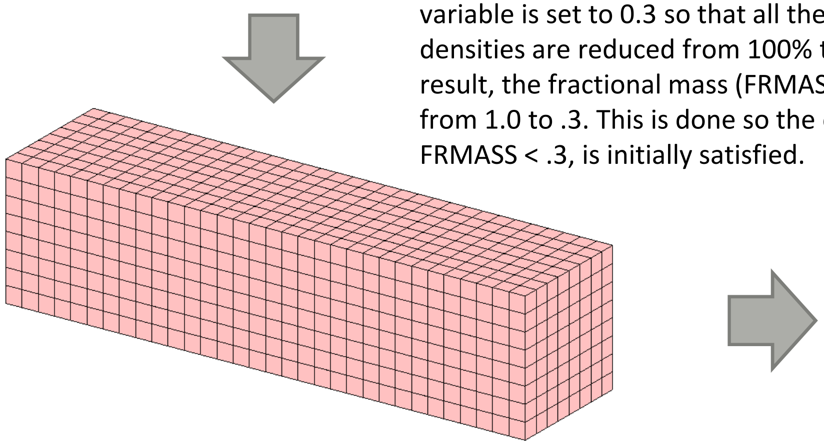
p_i : The optimized material density of the element ($p_i = p_0 \cdot x_i$)

v_i : Volume of element

0) Suppose this is the optimization problem statement:

- Objective: Minimize compliance
- Constraint: FRMASS < .3

1) Prior to the optimization start, each topology variable is set to 0.3 so that all the material densities are reduced from 100% to 30% of p_0. As a result, the fractional mass (FRMASS) is reduced from 1.0 to .3. This is done so the design constraint, FRMASS < .3, is initially satisfied.



2) During the optimization, each topology variable is allowed to range between 0 and 1.0, but the constraint on fractional mass (FRMASS < .3) should ultimately be satisfied.

What is compliance?

Compliance is defined in many ways

- “Compliance is simply the product of the displacement times the applied load” (MSC Nastran Design Sensitivity and Optimization User’s Guide)
- For linear elastic solids, the work is twice the total strain energy

```
                         E L E M E N T   S T R A I N   E N E R G I E S

ELEMENT-TYPE = HEXA              * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM     =    9.111034E+03
SUBCASE               1          * TOTAL ENERGY OF ALL ELEMENTS IN SET      -1 =    9.111034E+03

                    ELEMENT-ID          STRAIN-ENERGY           PERCENT OF TO
                            25          8.059148E+02                 8.8455
                            32          8.059148E+02                 8.8455                 6.447318E+03
                            33          8.059148E+02                 8.8455                 6.447318E+03
                            40          8.059148E+02                 8.8455                 6.447318E+03

        TYPE = HEXA      SUBTOTAL       9.111034E+03               100.0000
```

Total Strain Energy

```
                   ****************************************************************
                   S U M M A R Y   O F   D E S I G N   C Y C L E   H I S T O R Y
                   ****************************************************************

                                   (HARD CONVERGENCE ACHIEVED)

                   NUMBER OF FINITE ELEMENT ANALYSES COMPLETED           56
                   NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS     55

                          OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY
```

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		1.822207E+04		-4.625929E-15
1	5.076533E+03	1.32[illegible]	6.163140E-01	9.999972E-09
2	5.721454E+03	1.12[illegible]	4.893855E-01	6.604279E-09
3	4.220301E+03	1.016538E+04	-5.848357E-01	1.000032E-08
4	3.996396E+03	9.769504E+03	-5.909315E-01	9.983010E-09

Compliance

What is compliance? Continued

The .f06 file reports the value of compliance and strain energy. **The following applies if and only if minimizing the compliance is the design objective.**

1. Make sure this statement is in the Case Control Section of the .bdf file.

   ESE(THRESH=.99)=ALL

2. Search the .f06 file for the initial design's

   E L E M E N T S T R A I N E N E R G I E S

3. *Note the value of TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM*

4. *Search the .f06 for the*

   S U M M A R Y O F D E S I G N C Y C L E H I S T O R Y

5. Note the value for OBJECTIVE FROM EXACT ANALYSIS for the INITIAL cycle number

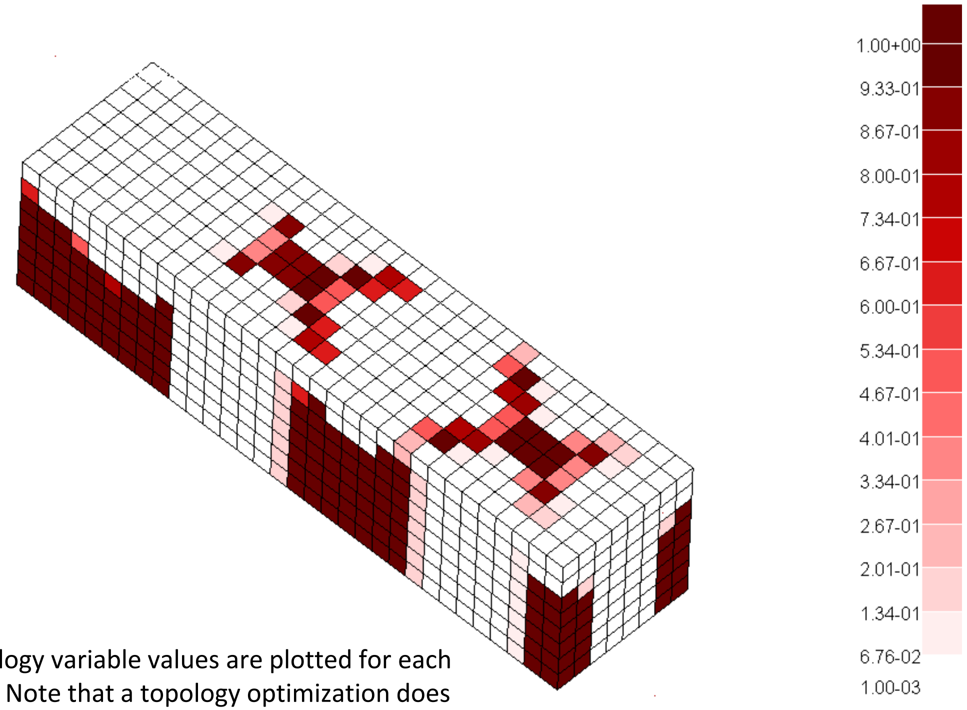
6. The Compliance of 1.8222E4 is twice the TOTAL STRAIN ENERGY of 9.11E3.

```
                           E L E M E N T   S T R A I N   E N E R G I E S

ELEMENT-TYPE = HEXA                * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM     =    9.111034E+03
SUBCASE               1            * TOTAL ENERGY OF ALL ELEMENTS IN SET      -1 =    9.111034E+03

                  ELEMENT-ID          STRAIN-ENERGY          PERCENT OF TOTAL    STRAIN-ENERGY-DENSITY
                          25          8.059148E+02                 8.8455              6.447318E+03
                          32          8.059148E+02                 8.8455              6.447318E+03
                          33          8.059148E+02                 8.8455              6.447318E+03
                          40          8.059148E+02                 8.8455              6.447318E+03

     TYPE = HEXA      SUBTOTAL        9.111034E+03               100.0000
```

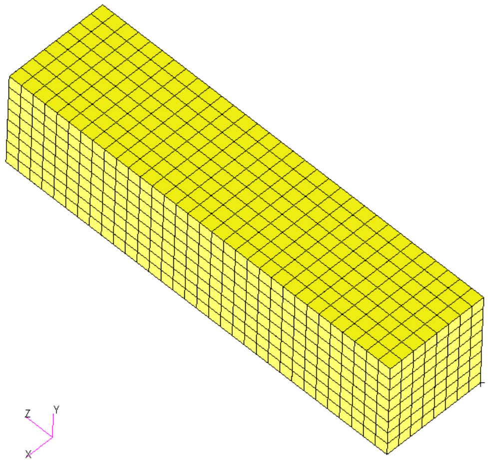
```
                   ***************************************************************
                   S U M M A R Y   O F   D E S I G N   C Y C L E   H I S T O R Y
                   ***************************************************************

                                (HARD CONVERGENCE ACHIEVED)

                NUMBER OF FINITE ELEMENT ANALYSES COMPLETED          56
                NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS    55

                          OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY
---------------------------------------------------------------------------------------------------------------
                   OBJECTIVE FROM         OBJECTIVE FROM        FRACTIONAL ERROR         MAXIMUM VALUE
CYCLE               APPROXIMATE              EXACT                   OF                      OF
NUMBER             OPTIMIZATION            ANALYSIS            APPROXIMATION            CONSTRAINT
---------------------------------------------------------------------------------------------------------------

INITIAL                                   1.822207E+04                                  -4.625929E-15

      1             5.076533E+03          1.323096E+04         -6.163140E-01             9.999972E-09

      2             5.721454E+03          1.120504E+04         -4.893855E-01             6.604279E-09

      3             4.220301E+03          1.016538E+04         -5.848357E-01             1.000032E-08

      4             3.996396E+03          9.769504E+03         -5.909315E-01             9.983010E-09
```

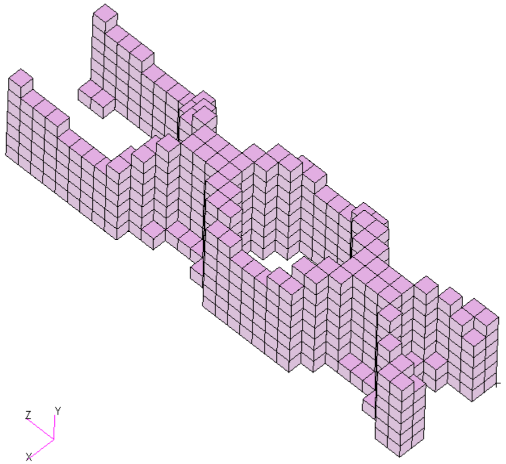
How can non-critical elements be removed from the design?

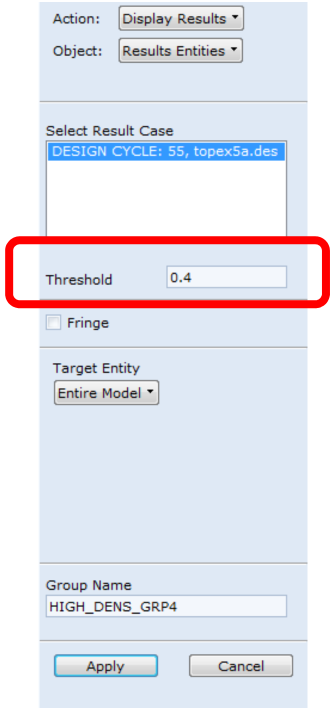
- Use the threshold to suppress non-critical elements
- The threshold means: *'Keep every element that has a topology variable value greater than the threshold'*
- Recall from before:
  - 0 - Topology variable values close to 0 are not critical to the design
  - 1 - Topology variable values close to 1 are critical to the design



The topology variable values are plotted for each element. Note that a topology optimization does not automatically remove elements. It is up to the user to manually decide which elements to keep.







Topology Optimization Workflows

There are 2 common optimization problem statements for topology optimization

METHOD A

Objective:

- Minimize Compliance
  - Minimize average compliance when multiple load cases are involved.

Constraint:

- FRMASS < Upper Bound

Comments:

- Multiple optimizations at different bounds for FRMASS are necessary. The best solution is selected from the multiple optimizations.

METHOD B

Objective:

- Minimize FRMASS

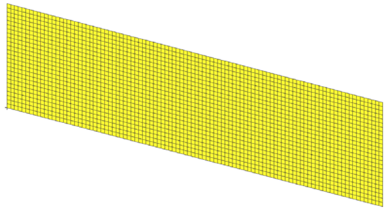
Constraint:

- Von Mises Stress < Upper Bound

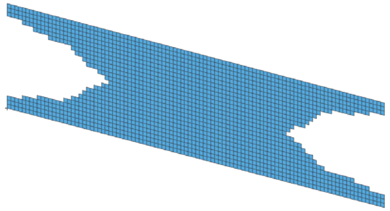
Traditional Topology Optimization

Objective: Minimize Compliance (Maximize Stiffness)
Constraint: Fractional Mass < .## (Target Mass)

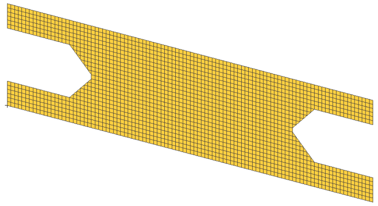
Original Design



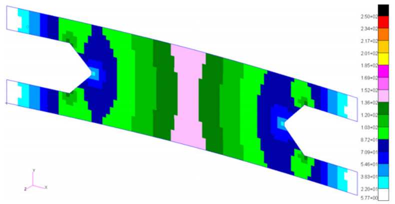
Mass: 9.737 grams



FRMASS < .75
Mass: 7.186 g
Optimization B



Mass: 7.739 g



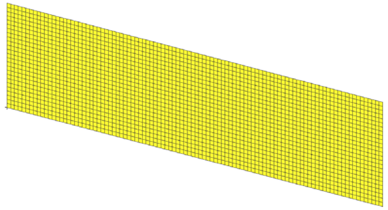
Max von Misses: 150 MPa
Max Displacement : 2.78 mm

1st natural Frequency: 111 Hz

Traditional Topology Optimization

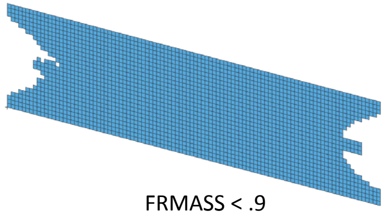
Objective: Minimize Compliance (Maximize Stiffness)
Constraint: Fractional Mass < .## (Target Mass)

Original Design

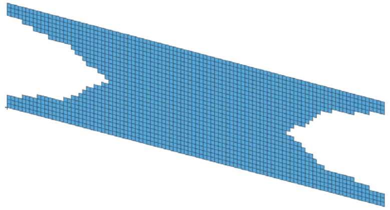


Mass: 9.737 grams

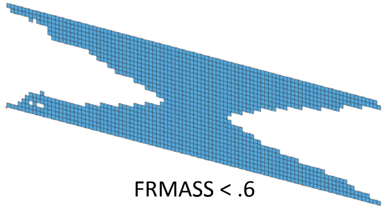
Topology Solution



FRMASS < .9
Mass: 8.756 g
Optimization A

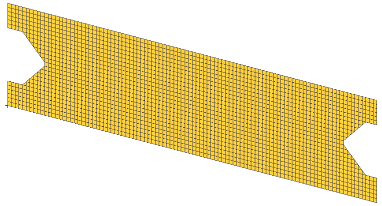


FRMASS < .75
Mass: 7.186 g
Optimization B

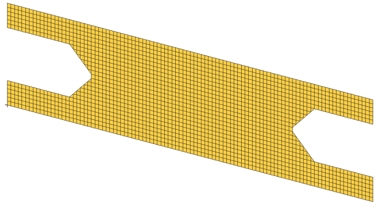


FRMASS < .6
Mass: 5.718 g
Optimization C

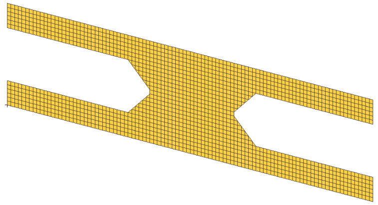
Refined Design



Mass: 9.094 g

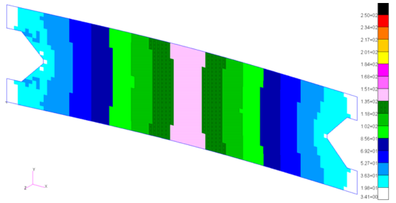


Mass: 7.739 g



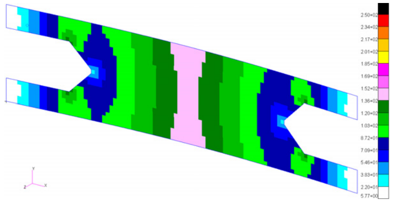
Mass: 6.119 g

Verification



Max von Misses: 150 MPa
Max Displacement: 2.52 mm

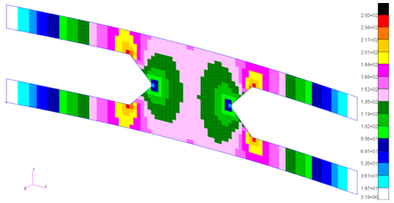
1st natural Frequency: 114 Hz



Max von Misses: 150 MPa
Max Displacement : 2.78 mm

1st natural Frequency: 111 Hz

Best Solution: Optimization B led to a valid and light weight design



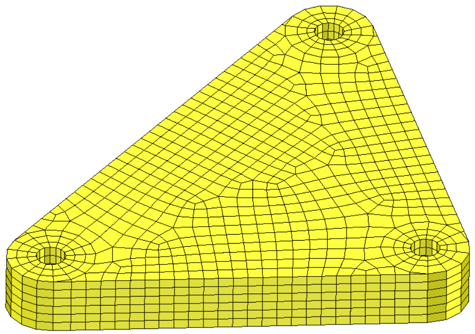
Max von Misses: 250 MPa
Max Displacement : 3.57 mm

1st natural Frequency: 109 Hz

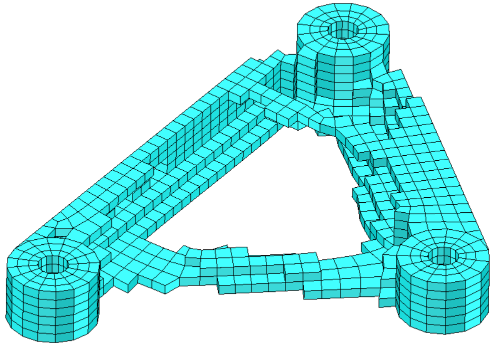
Latest Topology Optimization

Objective: Minimize Fractional Mass (Minimize Mass)
Constraint: Stress Constraint

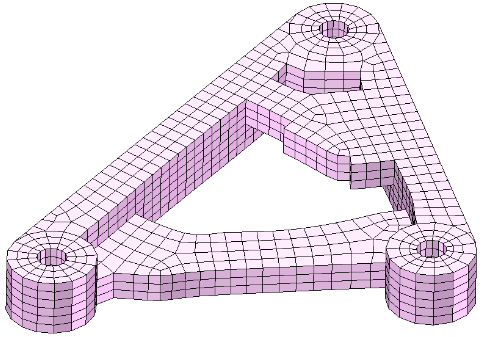
Original Design



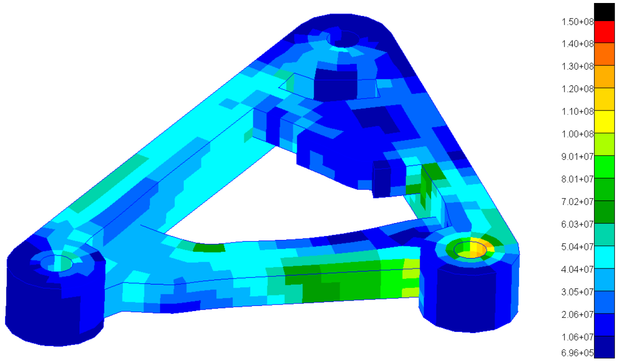
Topology Solution



Refined Design



Verification



Viewer Web App for Topology Optimization Post Processing

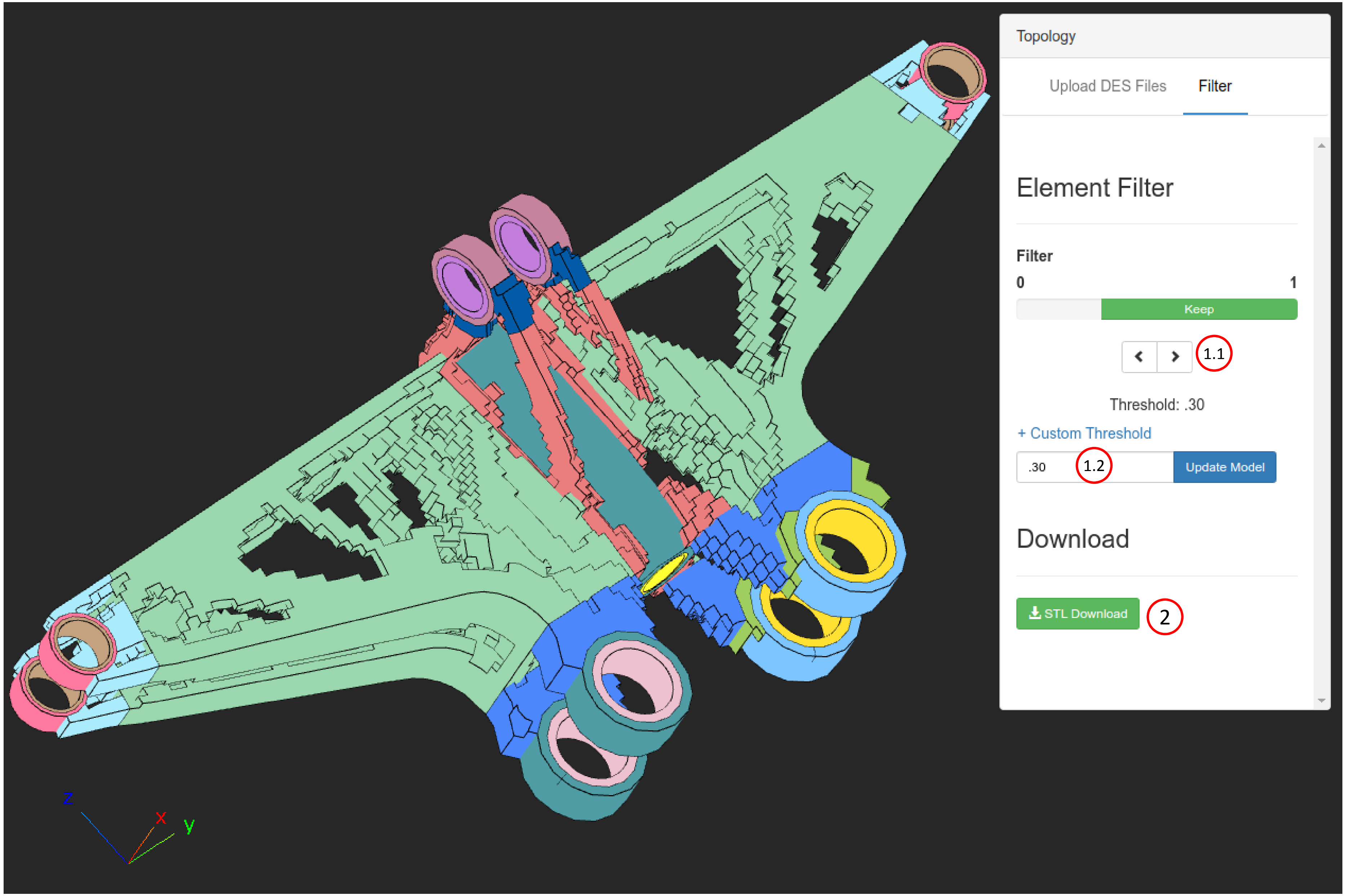
Review Optimization Results

The Viewer web app supports post processing topology optimization results.

Filtering of topology optimization results is controlled by one of 2 different ways:

1. The arrows can be used to move the threshold to values of 1.0, .3, .4, .5, .6 and .7
2. If a specific threshold is necessary, do the following:
    1. Click Custom Threshold
    2. Supply the custom threshold
    3. Click Update Model

1. STL Download – This downloads an STL file containing the model as displayed. This is useful for moving the topology results to a CAD package or FEA pre processor



Viewer
Supported Capabilities

Supported Element Types for Post-processing Topology Optimization Results

- CTRIA3
- CTRIA6
- CTRIAR
- CQUAD4
- CQUAD8
- CQUADR
- CQUAD4
- CQUAD8
- CQUADR
- CHEXA
- CTETRA
- CPENTA
- All other elements are not supported

STL Download/Export is Supported

Performance

- When uploading BDF or DES files, there are many operations performed, e.g. reading, parsing, and displaying data. This is the first release of the Viewer and future improvements to performance will be made. At the time of writing this, the viewer is capable of fully parsing and displaying 10MB of BDF files every 25 seconds. The viewer does not provide a progress bar regarding the parsing process, so it was best to document here the expected parsing rate.