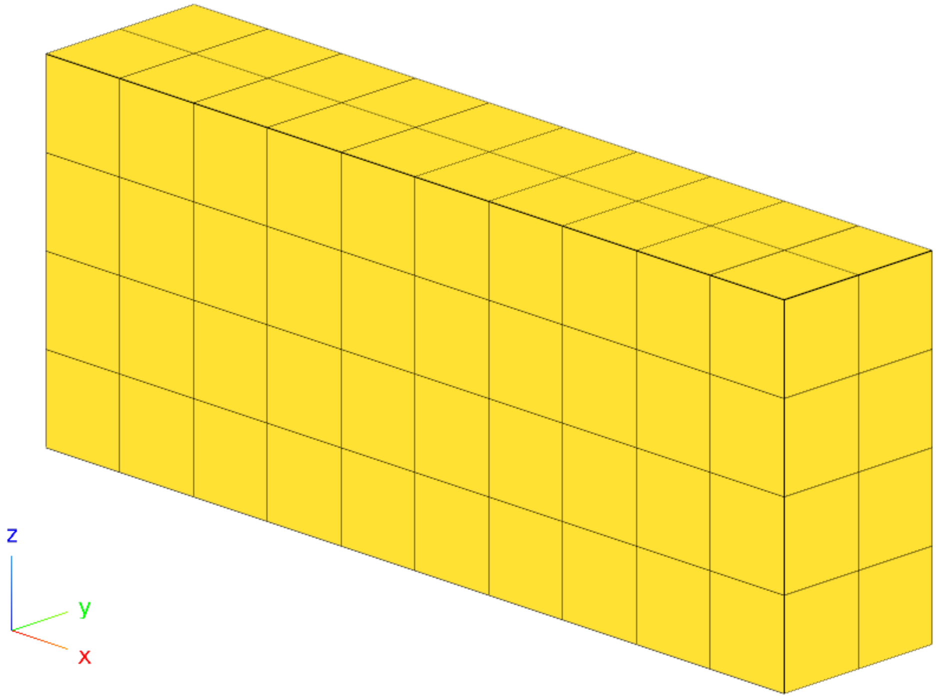
Workshop - Shape Optimization of a Cantilever Beam

AN MSC NASTRAN SOL 200 TUTORIAL

Goal: Use Nastran SOL 200 Optimization

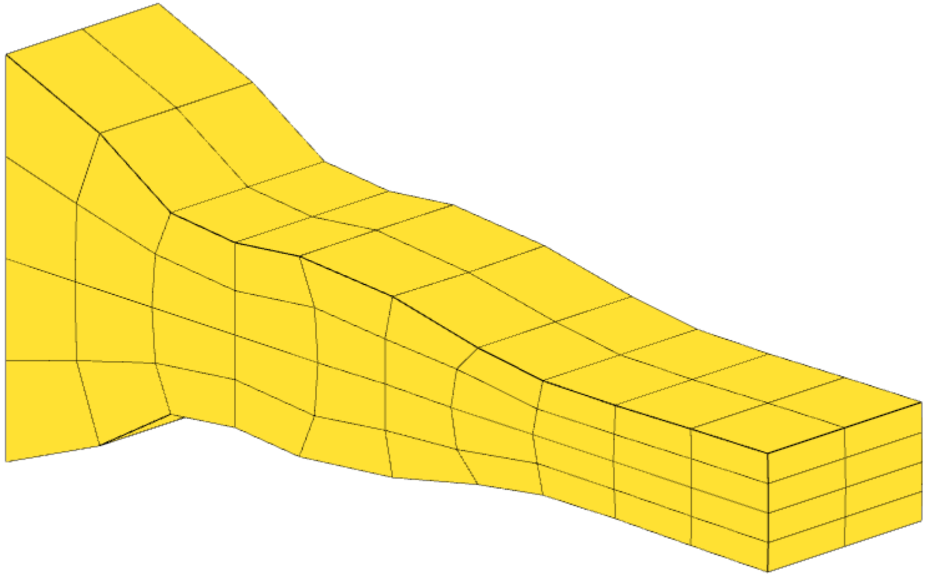
Before Optimization

- Weight: 80.0

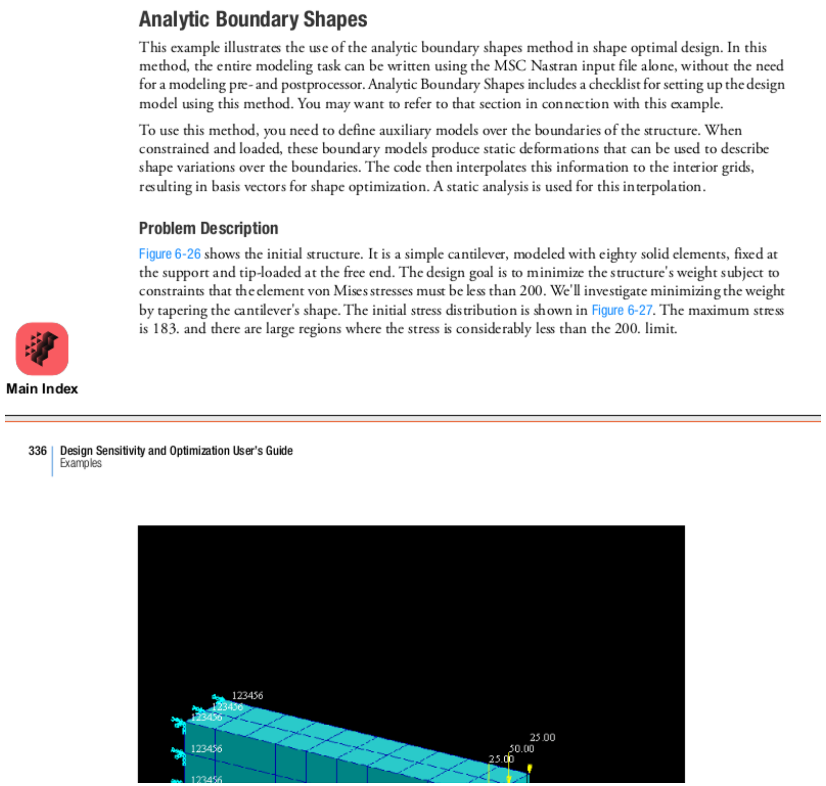


After Optimization

- Weight: 36.77



Details of the Structural Model



Analytic Boundary Shapes

This example illustrates the use of the analytic boundary shapes method in shape optimal design. In this method, the entire modeling task can be written using the MSC Nastran input file alone, without the need for a modeling pre- and postprocessor. Analytic Boundary Shapes includes a checklist for setting up the design model using this method. You may want to refer to that section in connection with this example.

To use this method, you need to define auxiliary models over the boundaries of the structure. When constrained and loaded, these boundary models produce static deformations that can be used to describe shape variations over the boundaries. The code then interpolates this information to the interior grids, resulting in basis vectors for shape optimization. A static analysis is used for this interpolation.

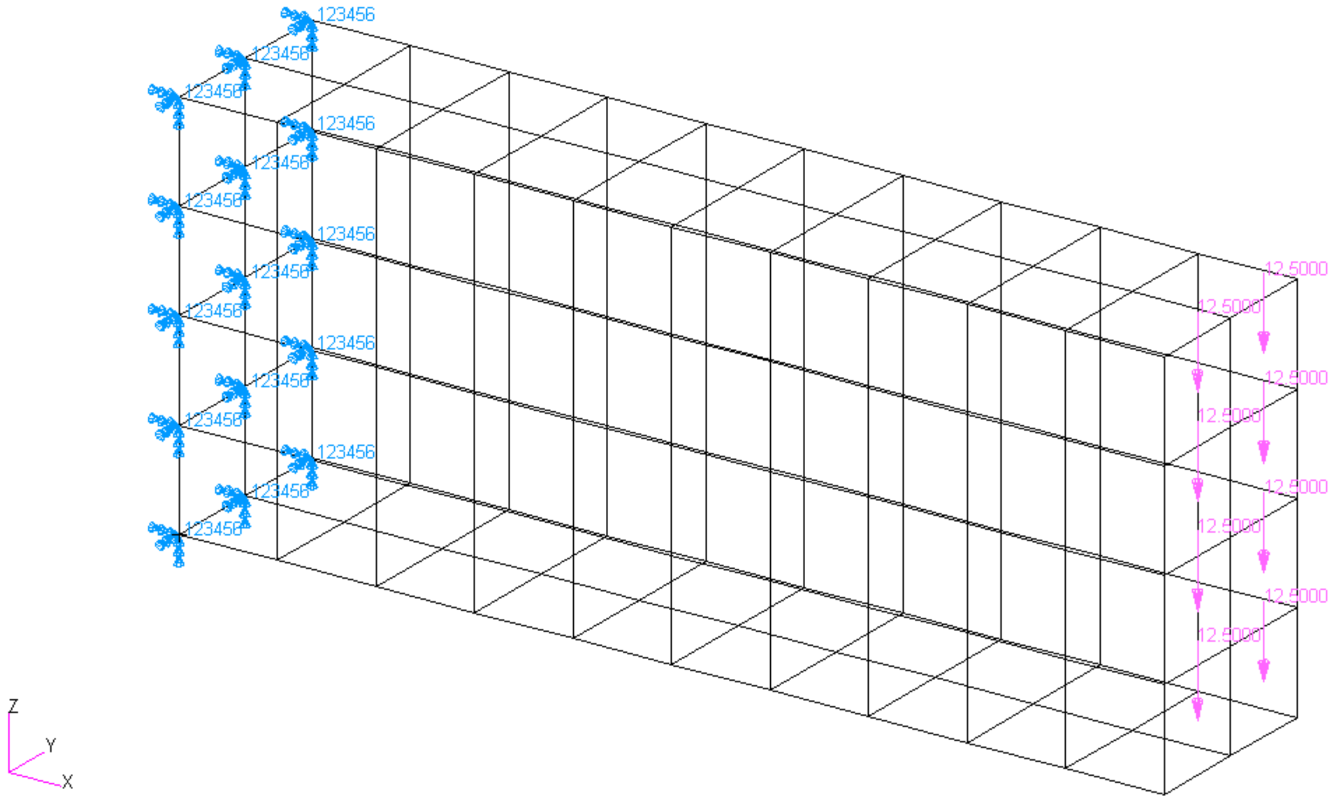
Problem Description

Figure 6-26 shows the initial structure. It is a simple cantilever, modeled with eighty solid elements, fixed at the support and tip-loaded at the free end. The design goal is to minimize the structure's weight subject to constraints that the element von Mises stresses must be less than 200. We'll investigate minimizing the weight by tapering the cantilever's shape. The initial stress distribution is shown in Figure 6-27. The maximum stress is 183. and there are large regions where the stress is considerably less than the 200. limit.

Main Index

336 | Design Sensitivity and Optimization User's Guide
Examples

MSC Nastran Design Sensitivity and Optimization User's Guide
Chapter 6 - Examples – Analytic Boundary Shapes



Optimization Problem Statement

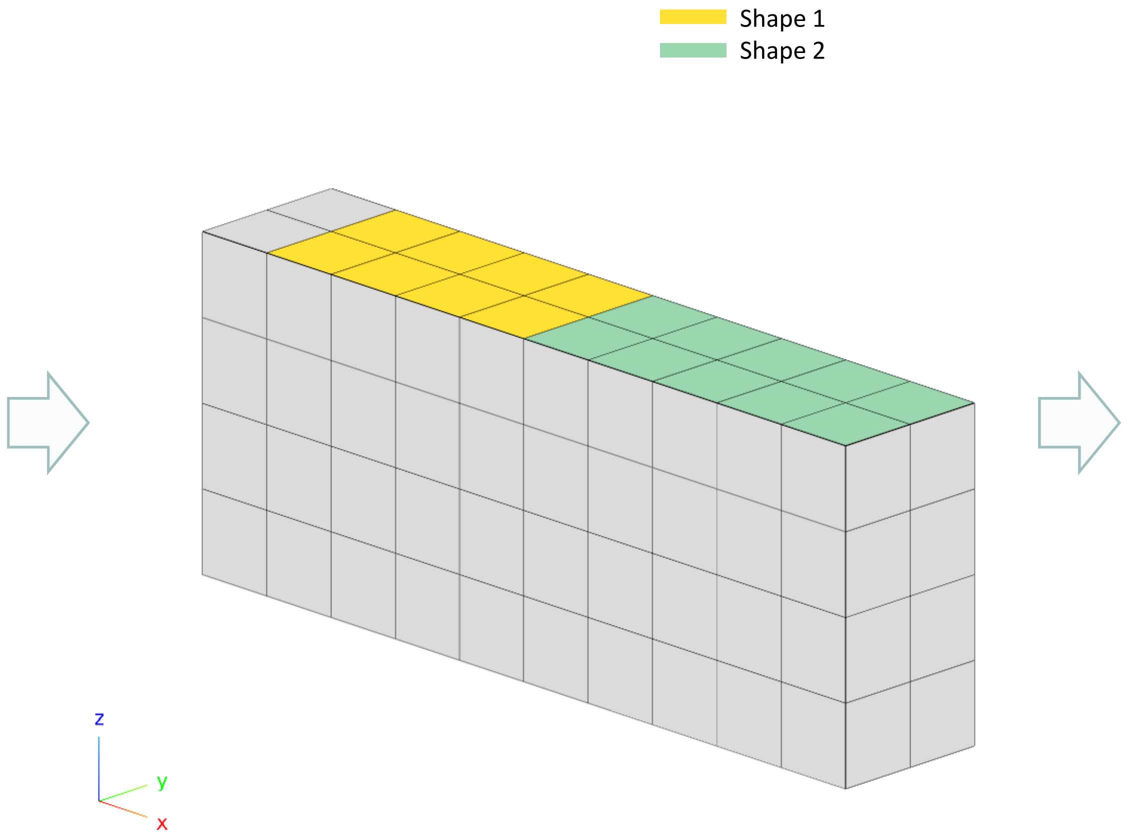
Design Variables

y1: Shape 1 – Expansion/contraction region 1
y2: Shape 2 – Expansion/contraction region 2

$8.5 < y1, y2 < 10.0$

When the initial value of y1 and y2 is 10.0, the equivalent bounds on Δyi are:

$-1.5 < \Delta y1, \Delta y2 < 0.0$



Design Objective

r0: Minimize weight

Design Constraints

r2: von Mises stress at the center of the element

r3: von Mises stress at the 1st corner of the element

…

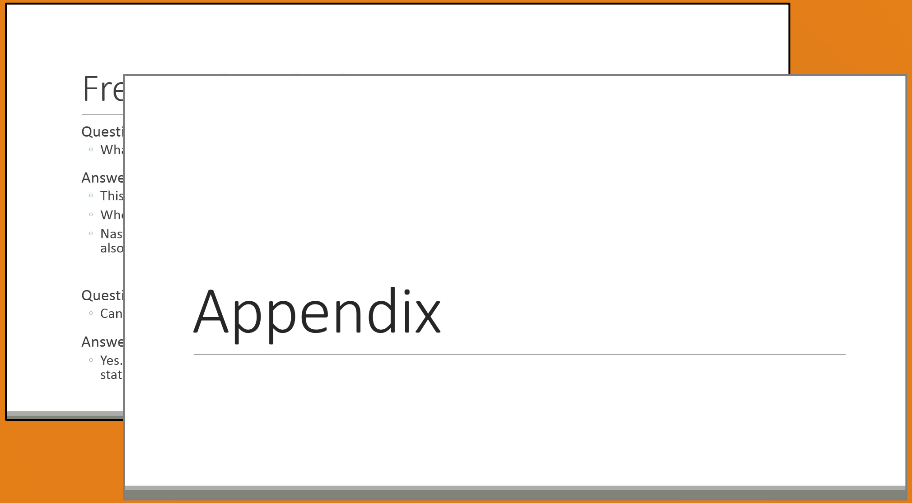
r10: von Mises stress at the 8th corner of the element

$r2, r3, \ldots, r10 < 200.0$

More Information Available in the Appendix

The Appendix includes information regarding the following:

- Frequently Asked Questions
  - How does MSC Nastran generate shape basis vectors?
  - Why the scaling factor on DVBSHAP?
  - How is the shape basis matrix constructed?
  - How to import previous BDF files?



Contact me

- Nastran SOL 200 training
- Nastran SOL 200 questions
- Structural or mechanical optimization questions
- Access to the SOL 200 Web App

christian@ the-engineering-lab.com

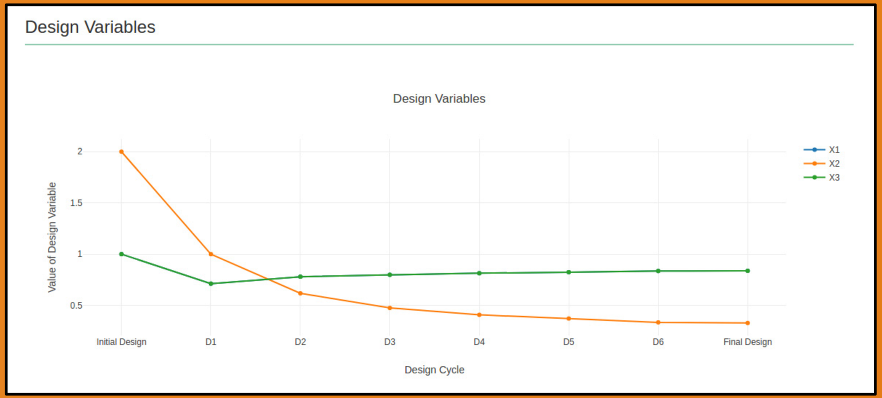
Tutorial

Tutorial Overview

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

Special Topics Covered

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



SOL 200 Web App Capabilities

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

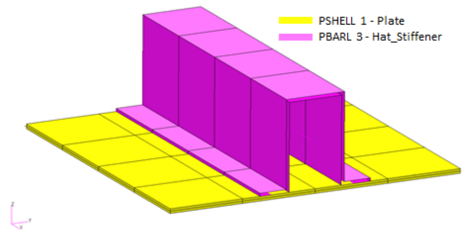
Compatibility

- 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.

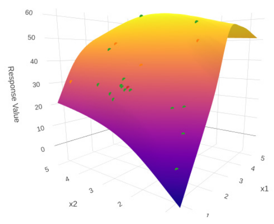
Benefits

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

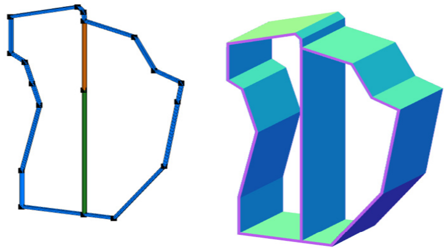
Web Apps



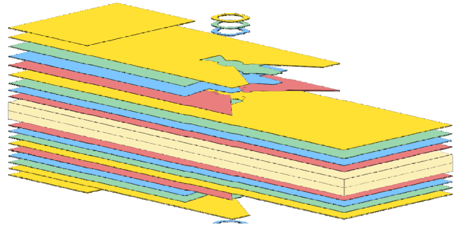
Web Apps for MSC Nastran SOL 200
Pre/post for MSC Nastran SOL 200. Support for size, topology, topometry, topography, multi-model optimization.



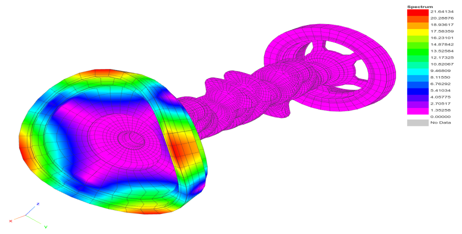
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



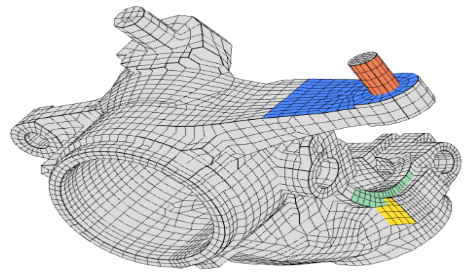
PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically



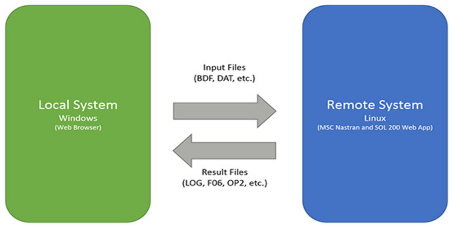
Ply Shape Optimization Web App
Optimize composite ply drop-off locations, and generate new PCOMPG entries



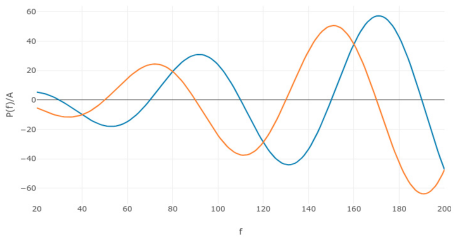
Post-processor Web App
View MSC Nastran results in a web browser on Windows and Linux



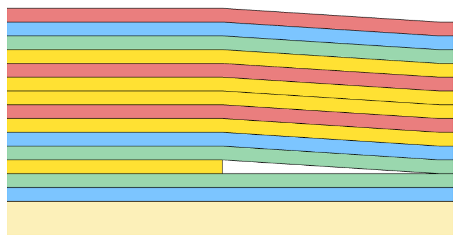
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



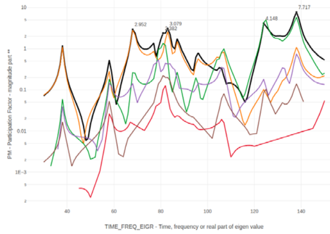
Remote Execution Web App
Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



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



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

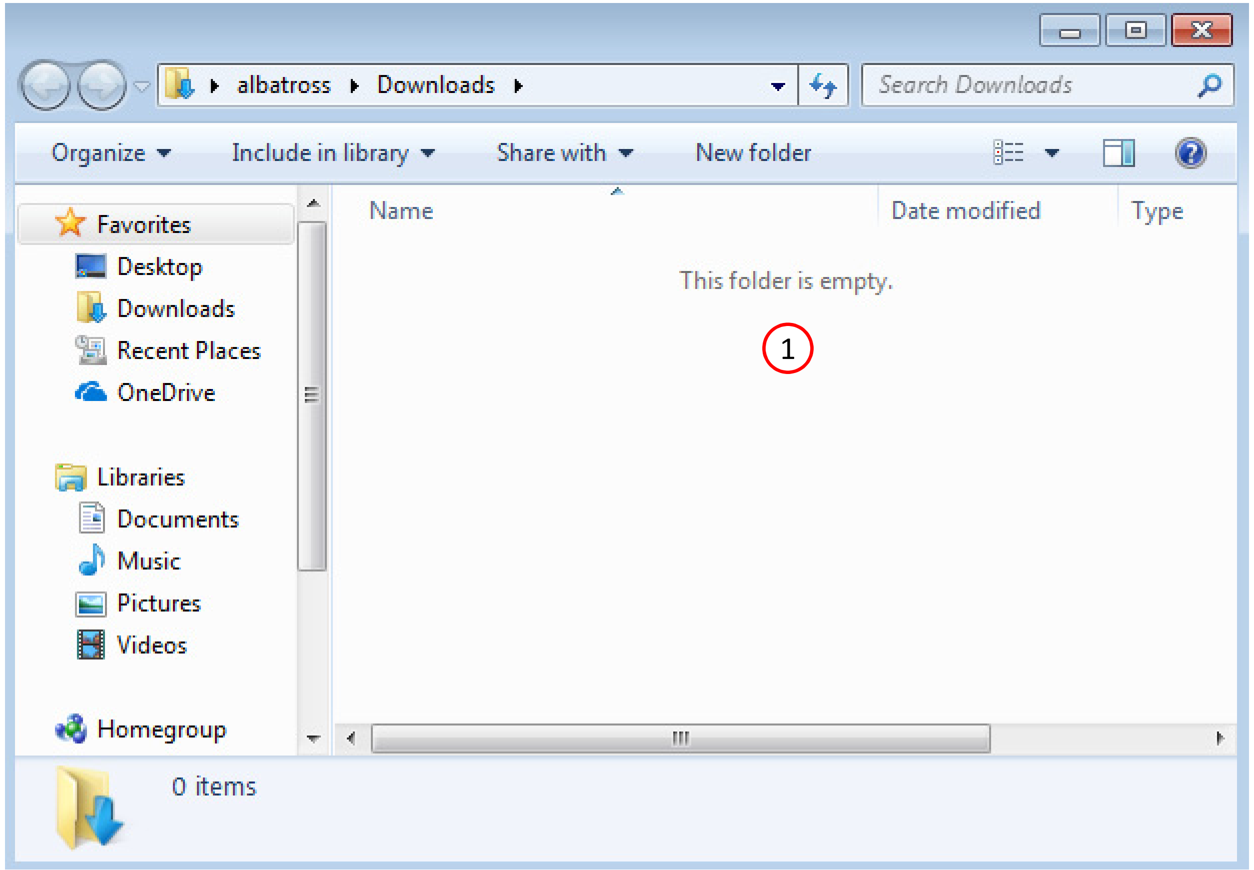


HDF5 Explorer Web App
Create graphs (XY plots) using data from the H5 file

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

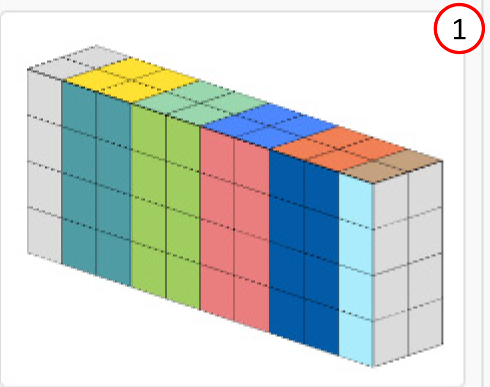
- The necessary BDF files for this tutorial are available in the Tutorials section of the User's Guide.

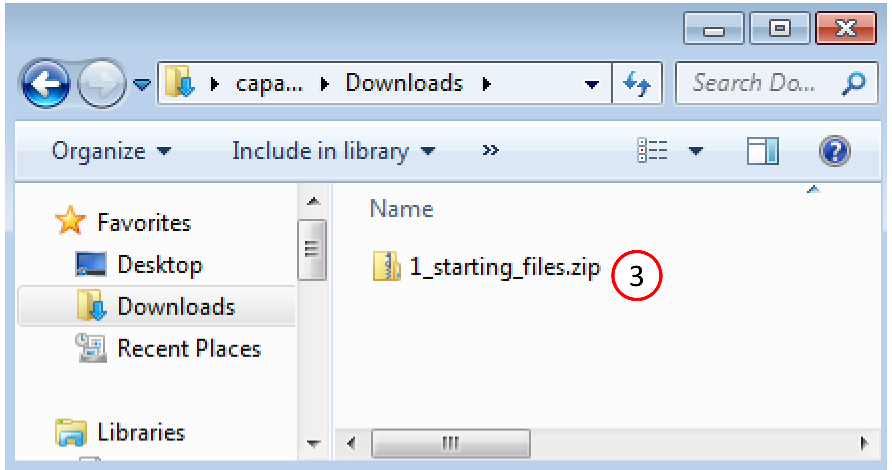
The Engineering Lab

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.

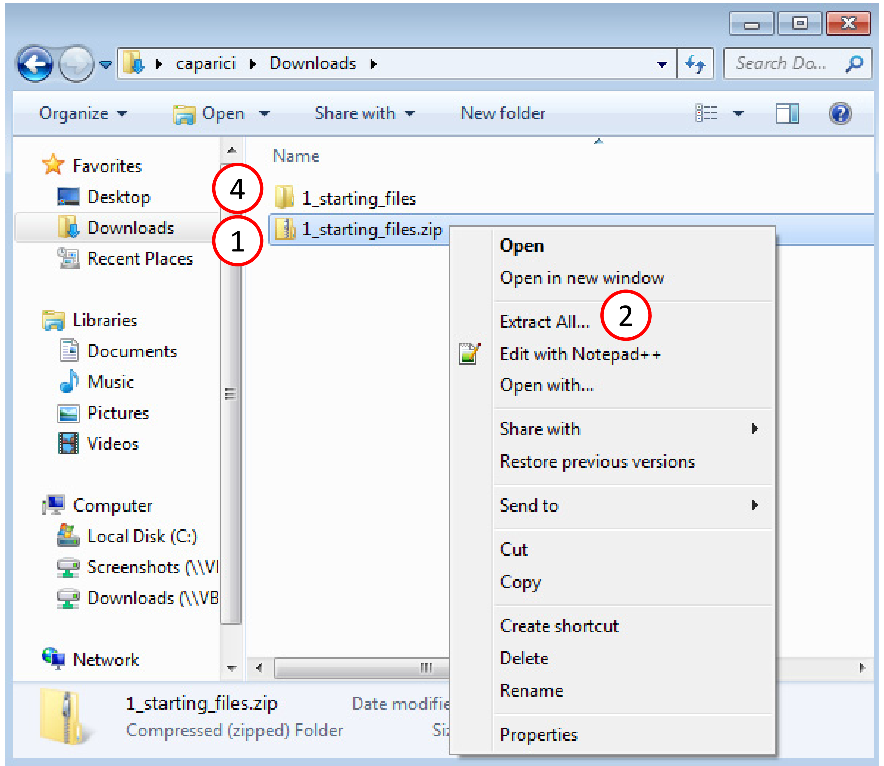
	Title and Description
	1 **Shape Optimization of a Cantilever Beam** This tutorial is an introduction to MSC Nastran's Shape Optimization capability. A cantilever beam is configured for a shape optimization. The goal is to minimize the mass while satisfying stress constraints. Specified regions of the beam are allowed to expand or contract and define the shapes that will vary during the optimization. This tutorial discusses the following concepts: auxiliary models, shape basis vectors, scaling shape basis vectors, configuring variable bounds, strategies to prevent mesh distortions, results interpretation, updating the model, and more. Starting BDF Files: Link 2 Solution BDF Files: Link

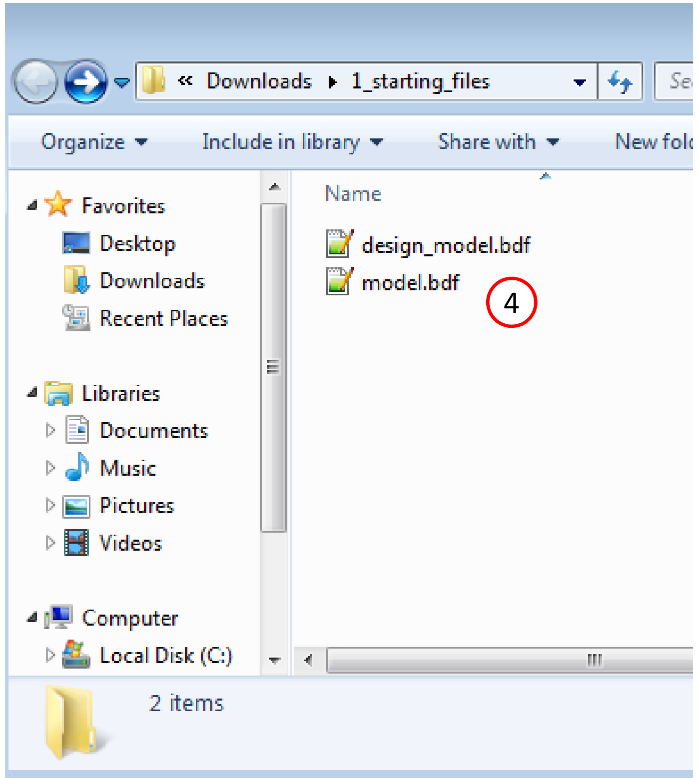


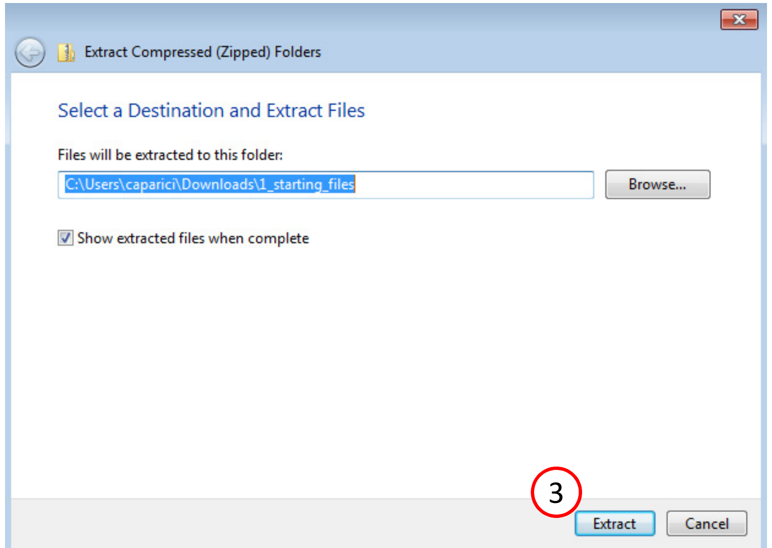
Obtain Starting Files

1. Right click on the zip file
2. Select Extract All…
3. Click Extract
4. The starting files are now available in a folder

- This example is using a previously created design model. The design model is a model that has been converted to SOL 200 and contains bulk data entries describing the optimization problem statement, e.g. variables, objective and constraints.

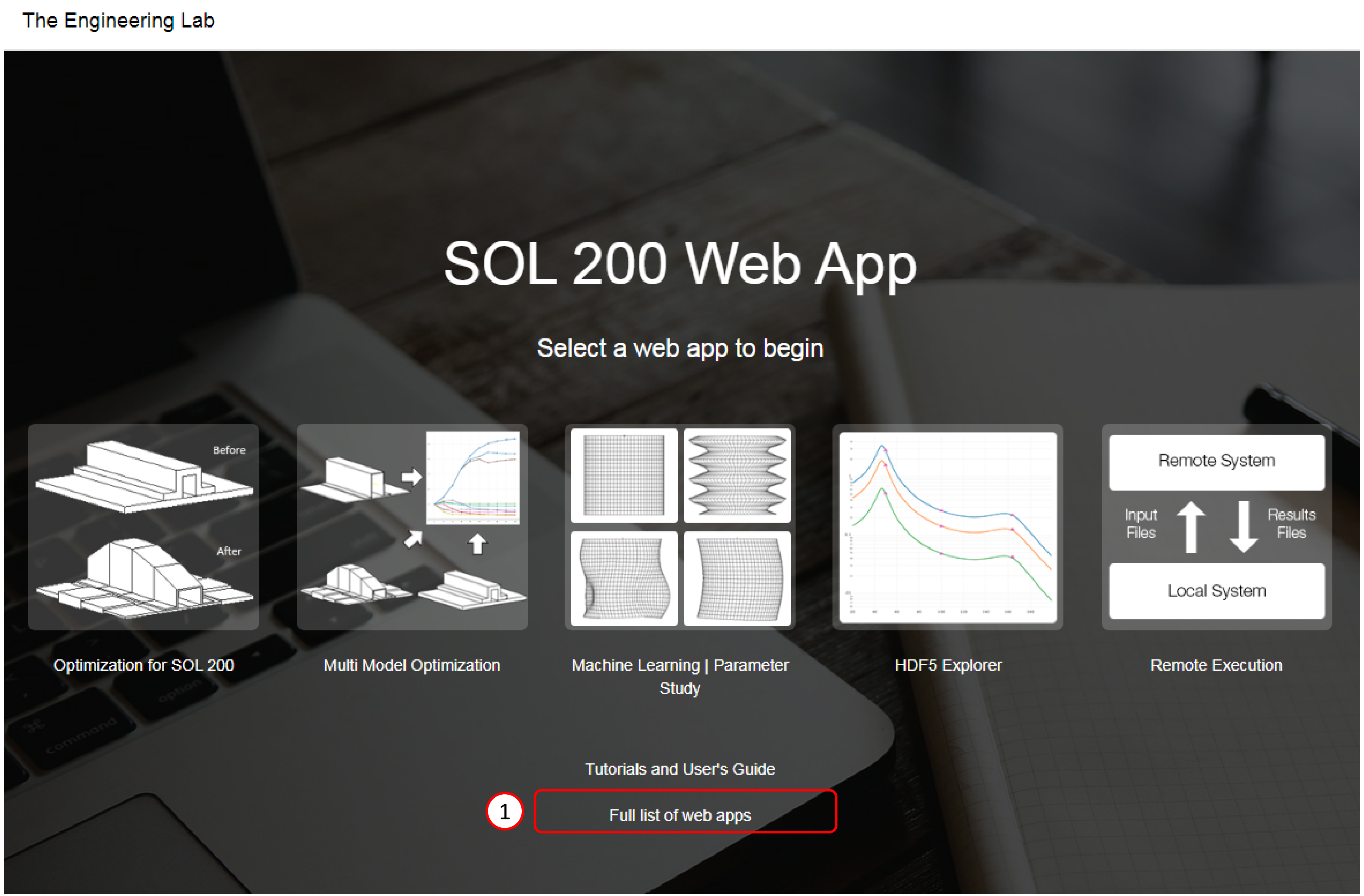






Open the Correct Page

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.

Open the Viewer

1. Navigate to the Optimization section
2. Click Viewer

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Import BDF Files

1. Click Upload BDF
2. Click Select files
3. Navigate to directory 1_starting_files
4. Select the indicated files
5. Click Open
6. Click Upload files

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

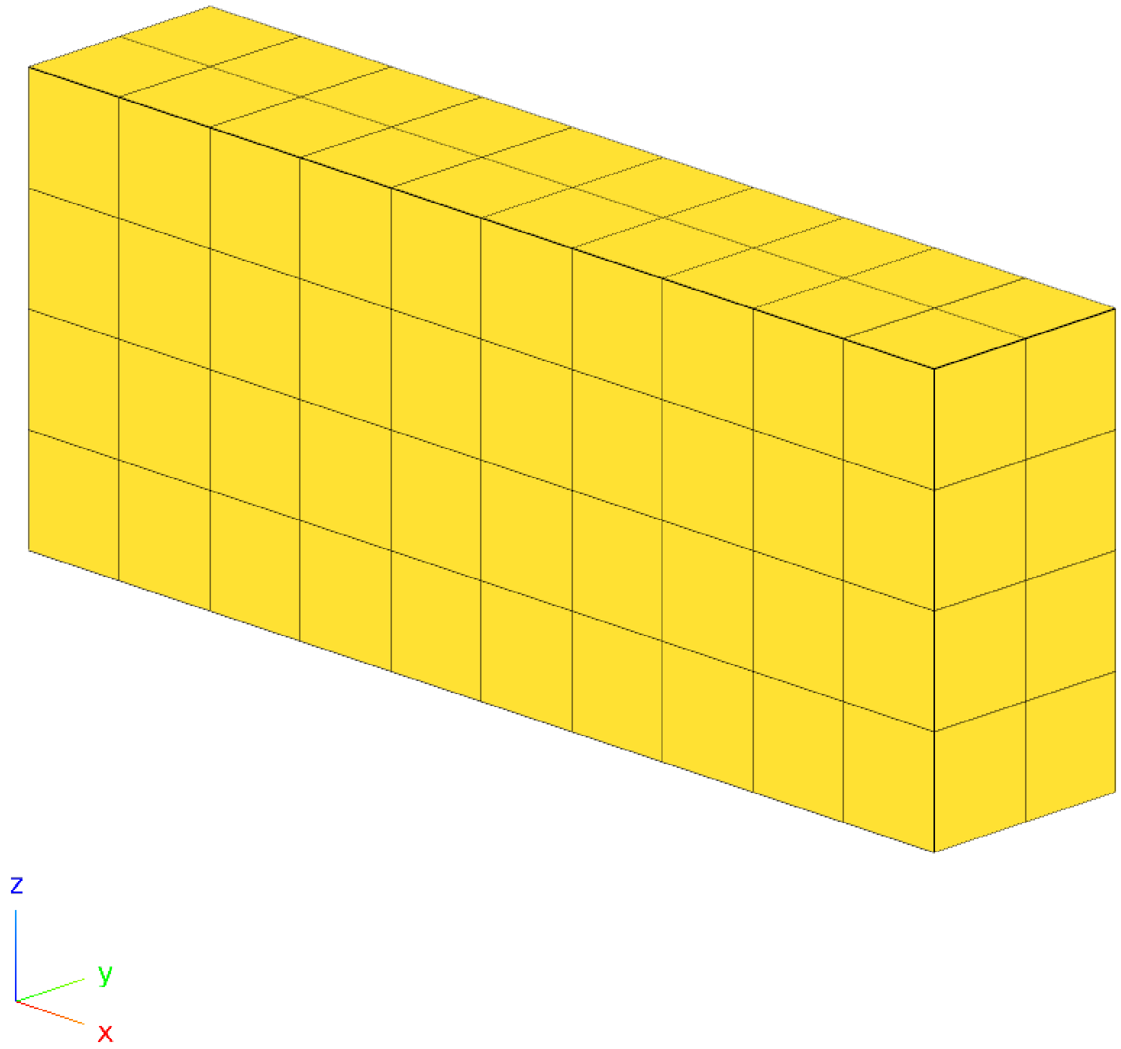
Before Continuing

Throughout this exercise, the following buttons will be useful for viewing the model

1. Center Model
2. Fit Model
3. Background Color
4. View Iso 3

The following mouse combinations will orient the model.

- Rotation: Left Mouse Click + Mouse Drag
    - After rotation, it sometimes helps to click Center Model to restore the center of rotation
- Translation: Right Mouse Click + Mouse Drag
- Zoom: Mouse Scroll Wheel



Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Before Continuing

1. The pick modes available include selecting and deselecting element faces and are accessed via the indicated buttons
2. When in a pick mode, a pick sphere appears. Left click and dragging the mouse will select or deselect the element faces.
3. To exit pick mode, click on the indicate button.
4. Alternatively, you can pick on the original pick mode button to exit pick model.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Open the Shape Panel

1. Click Shape
2. Select PSOLID ID 1

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Create Shapes

1. Click Shapes
2. Click the Toggle button to adjust the width of the panel
3. Click Add Shape
4. Click the indicated pick mode button.
5. An orange pick sphere should appear. Select the indicated element faces.
6. To deselect element faces, click the indicated button. A red pick sphere appears and may be used to deselect element faces.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Edit Shapes

1. Click the indicated pick mode button.
2. An orange pick sphere should appear. Select the indicated element faces.
3. To deselect element faces, click the indicated button. A red pick sphere appears and may be used to deselect element faces.
4. Click the indicated button to exit the pick mode.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Edit Shapes

1. Rotate the model to view the underside of the cantilever beam.
2. Click the indicated pick mode button.
3. An orange pick sphere should appear. Select the indicated element faces.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Edit Shapes

1. Click the indicated pick mode button.
2. An orange pick sphere should appear. Select the indicated element faces.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Configure Free Regions

1. Click the indicated pick mode button.
2. An orange pick sphere should appear. Select the indicated element faces.
3. Note the color of the faces is lighter, indicating these faces have a different function. The darker shaded faces indicate where the extraction or contraction will occur. The lighter shaded faces indicate where the grids may move freely.

- Primary element faces: These faces are allowed to expand or contract
- Secondary element faces: These faces are allowed to stretch

Configure Free Regions

1. Click the indicated pick mode button.
2. An orange pick sphere should appear. Select the indicated element faces.
3. Like before, note that the color of the faces is lighter indicating the grids in this region will move freely during the shape optimization.

- Primary element faces: These faces are allowed to expand or contract
- Secondary element faces: These faces are allowed to stretch

Configure Free Regions

1. Rotate the model to view the other side of the cantilever beam.
2. Click the indicated pick mode button.
3. An orange pick sphere should appear. Select the indicated element faces.

Configure Free Regions

1. Click the indicated pick mode button.
2. An orange pick sphere should appear. Select the indicated element faces.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Run MSC Nastran to Generate Shapes

1. Scroll to section 2) Generate Shapes
2. Click Run MSC Nastran
3. Continue after the status reads Complete. The duration of this MSC Nastran run will depend on the size of the model.
4. Optional – Click Display F06 Section to inspect the F06 output.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

DVBSHAP

1. After MSC Nastran is complete, the web app will update the scaling factor for each shape defined by DVBSHAP entries. The scaling factor is set in field SF1 (field 5) of the DVBSHAP entry.

- If the scaling factors are 1.0, which will happen if MSC Nastran was not executed, it is suggested that you manually supply ideal scaling factors.

File: design_shapes_psolid_1.bdf

```
$   1   ||   2   ||   3   ||   4   ||   5   ||   6   ||   7   ||   8   ||   9   ||  10   |
DVBSHAP 200001  1       1       .4847393
DVBSHAP 200002  1       2       .4900893
DESVAR  200001  y1      10.     8.5     10.
DESVAR  200002  y2      10.     8.5     10.
BNDGRID 123     1       2       3       4       5       6       7
        8       9       10      11      12      13      14      15
        16      17      18      19      20      21      22      23
        24      25      26      27      28      29      30      31
        32      33      34      35      36      37      38      39
        40      41      42      43      44      45      55      56
        57      58      59      60      61      62      63      64
        65      66      67      68      69      70      71      72
        73      74      75      76      77      78      88      89
        90      91      92      93      94      95      96      97
        98      99      100     101     102     103     104     105
        106     107     108     109     110     111     121     122
        123     124     125     126     127     128     129     130
        131     132     133     134     135     136     137     138
        139     140     141     142     143     144     145     146
        147     148     149     150     151     152     153     154
        155     156     157     158     159     160     161     162
        163     164     165
```

1

Shape Change Preview

Previewing the shape change is an important step in configuring a shape optimization. A preview of the shape change is inspected in this section.

The depth or height of the beam is 4.0 units of length.

1. Scroll to section 3) Preview Shape Changes
2. For Test Δy, set the value to -1.5. A negative Δy indicates contraction. A positive Δy indicates expansion.
3. The face of each shape is contracted by approximately 1.5 units of length (Δy=-1.5). The preview indicates that a mesh distortion is unlikely to occur.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Shape Change Preview

The steps on this page are optional.

1. For Test Δy, set the value to -2.5

2. The face of each shape is contracted by approximately 2.25 units of length (Δy=-2.5). The preview indicates that a mesh distortion is likely to occur.

3. Click Look Inside

4. A subset of element faces is displayed. This is useful in identifying locations of mesh distortions. As shown, the top and bottom faces have crossed each other when Δy=-2.5.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Adjustment of Variable Bounds

1. Click the toggle button 2 times to restore the width of the panel
2. Return to section 1) Select Shape Regions
3. Since it is known a $\Delta y=-1.5$ is unlikely to cause a mesh distortion, the lower bound of Δy is set to -1.5.
4. No expansion is allowed for this shape optimization, so Δy Upper Bound is set to 0.0. If expansion is desired a positive value should be used for the upper bound.

Inspect New Entries

1. Click New Entries
2. Click the toggle button 2 times to expand the width of the panel.
3. The changes that will be performed to the bulk data files are listed.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Export New BDF Files

1. Click Download
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"

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

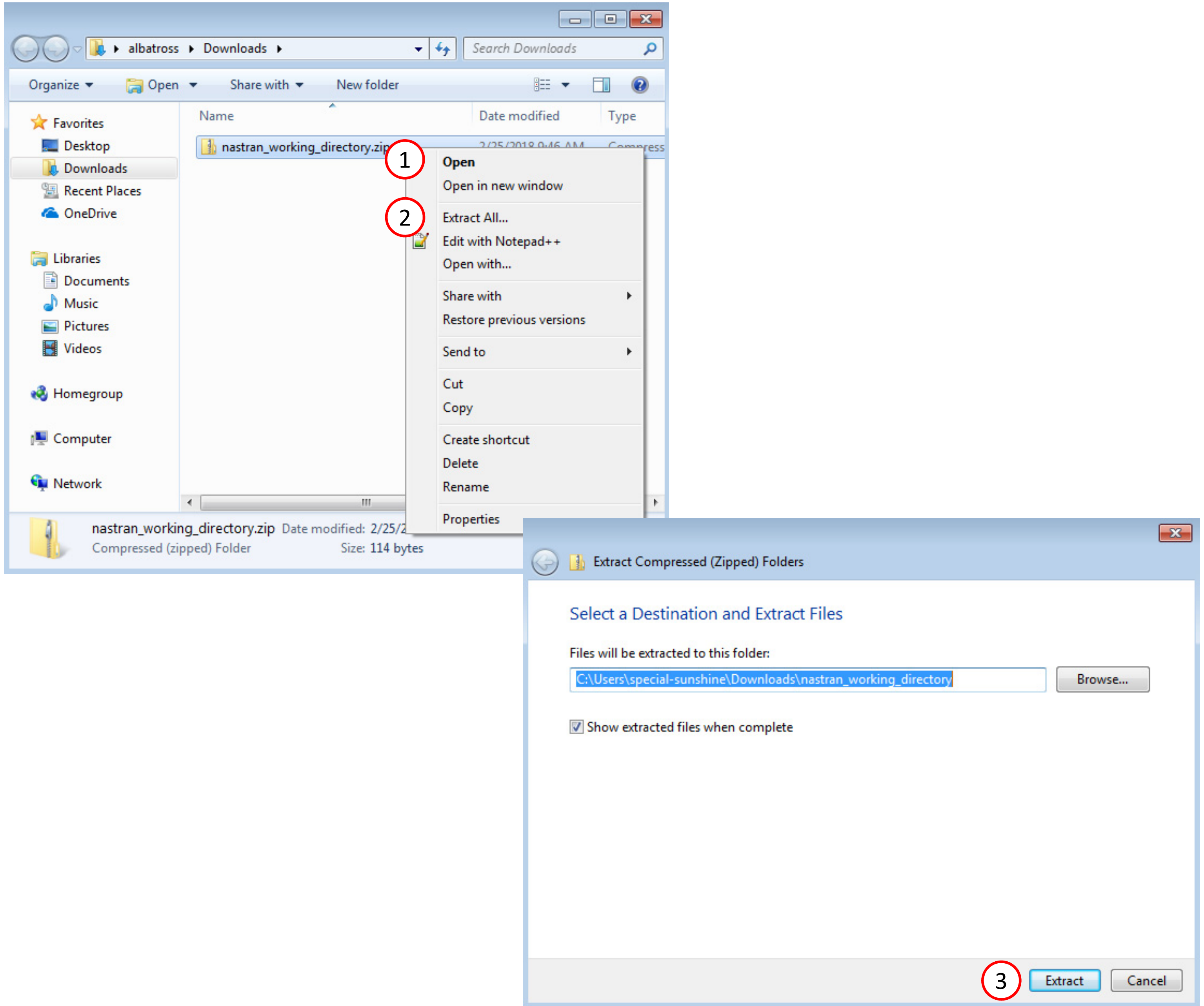
or contact

christian@ the-engineering-lab.com

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

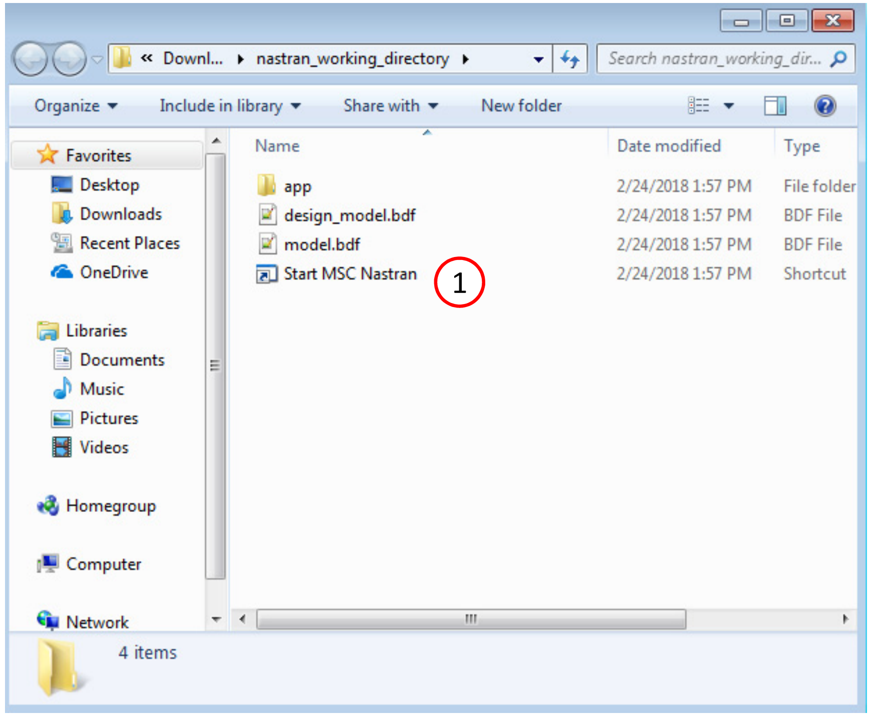
- After a successful optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- One can run the Nastran job on a remote machine as follows:
  1) Copy the BDF files and the INCLUDE files to a remote machine. 2) Run the MSC Nastran job on the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

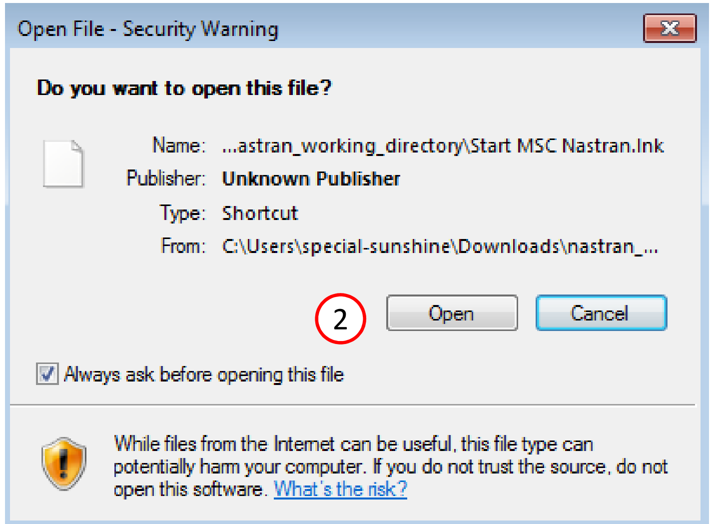
Using Linux?

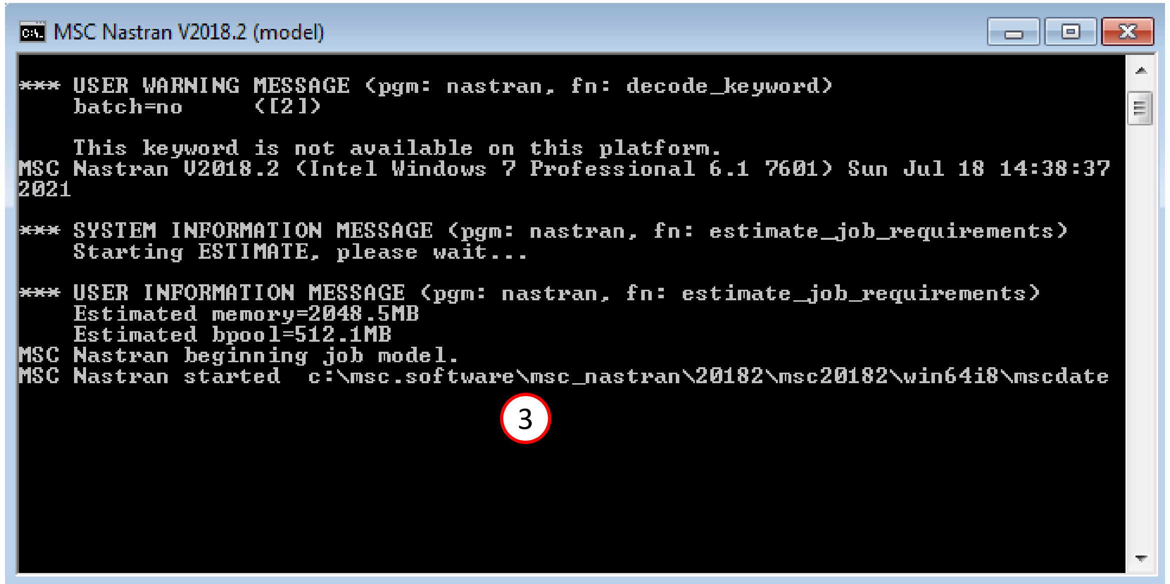
Follow these instructions:
1) Open Terminal
2) Navigate to the nastran_working_directory
 cd ./nastran_working_directory
3) Use this command to start the process
 ./Start_MSC_Nastran.sh

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 and design variables can be reviewed.

- After an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- The Normalized Constraints plot indicates the final design cycle has yielded a design that has a max normalized constraint very close to zero. Max normalized constraints that are negative or close to zero indicate a feasible design has been obtained. Feasible designs are designs that satisfy all design constraints.

SOL 200 Web App - Local Optimization Results

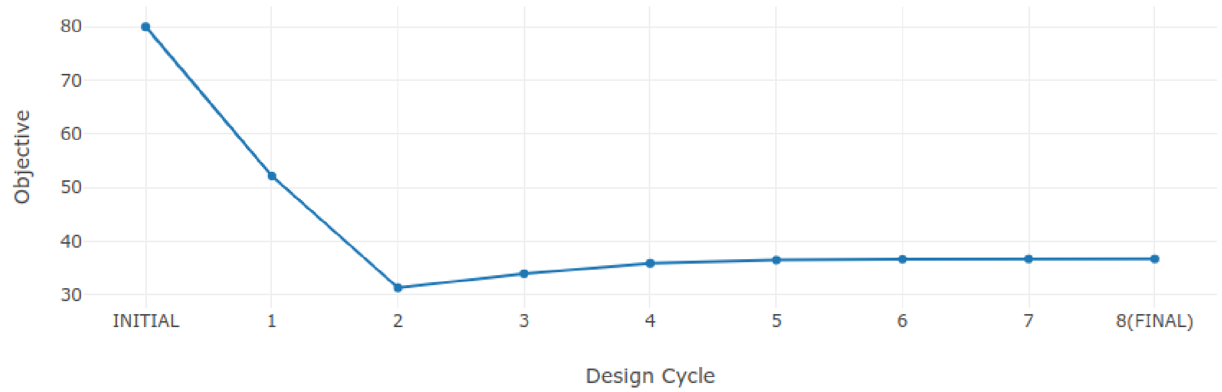
Home

Final Message in .f06

1

RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER = 8.

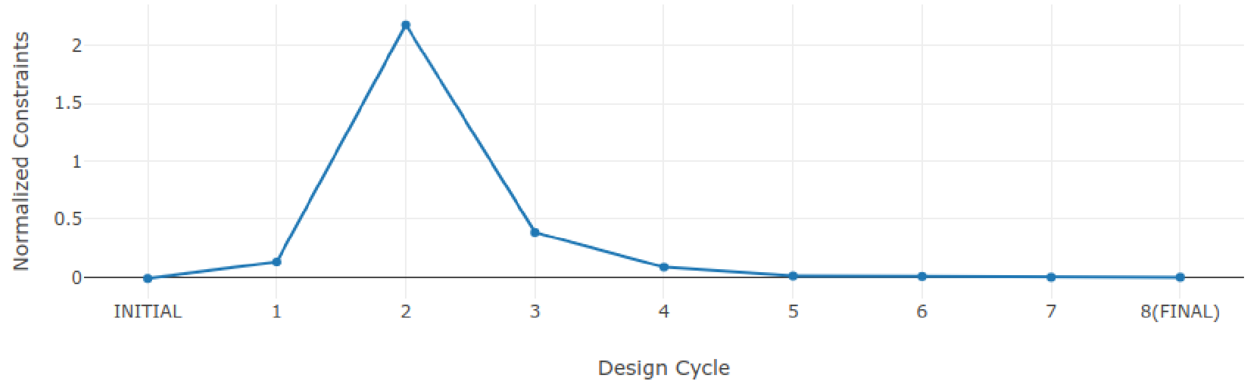
Objective



2

Normalized Constraints

+ Info



Review Optimization Results

The shape variables are interpreted in the following way.

Per the MSC Nastran Design Sensitivity and Optimization User's Guide, after each design cycle, the newest grid positions are determine by this expression.

$$\{G\}_{i+1} = \{G\}_i + [T]\{\{x\}_{i+1} + \{x\}_i\}$$

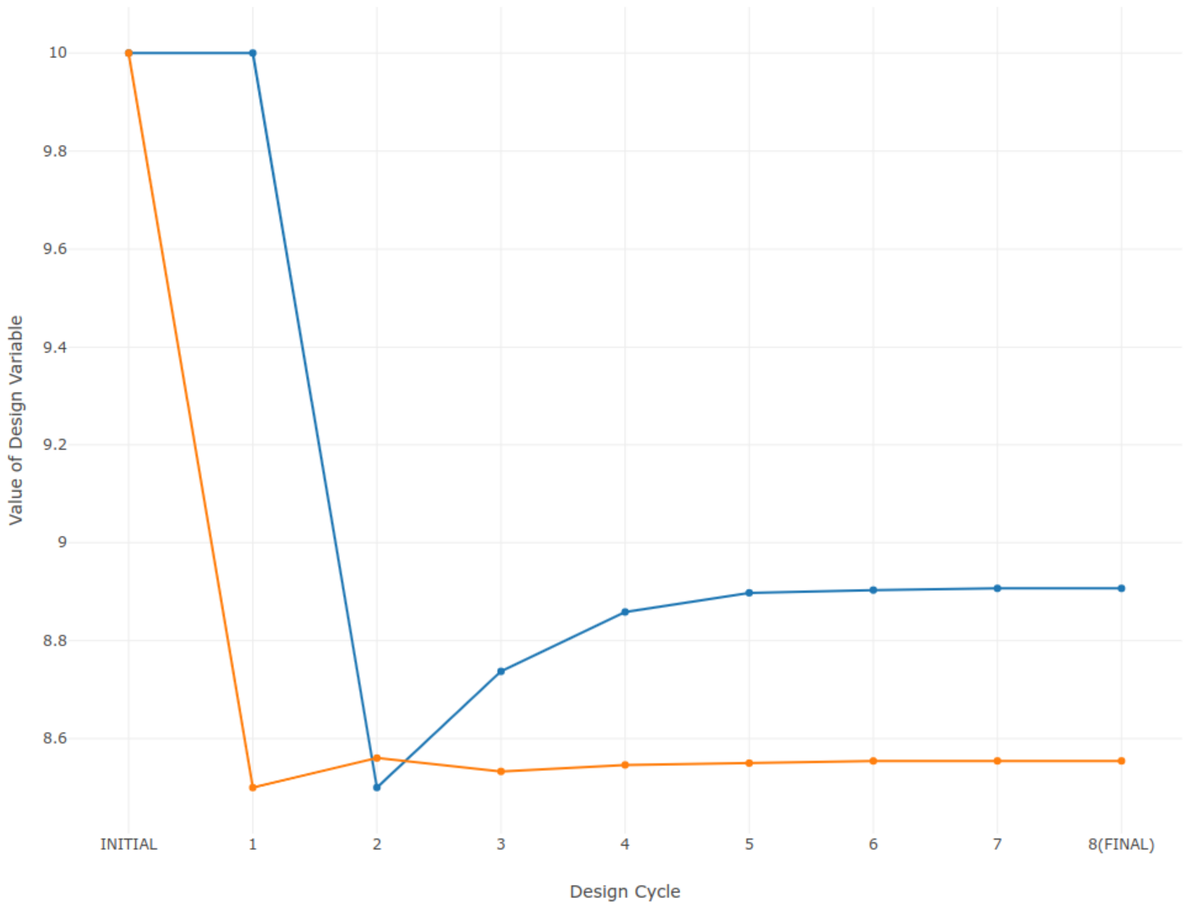
$$\{G\}_{i+1} = \{G\}_i + [T]\{\Delta x\}$$

Since this workshop expresses the variables as yi, the expression is rewritten as

$$\{G\}_{i+1} = \{G\}_i + [T]\{\Delta y\}$$

Where i is the current design cycle number. It is the values Δy that express how much expansion or contraction the shape has undergone.

Design Variables



Reset Table | Display None | Display All

Display	Color	Label	Label Comments
		Search	Search
☑	blue	y1	Shape 1 of PSOLID 1
☑	orange	y2	Shape 2 of PSOLID 1

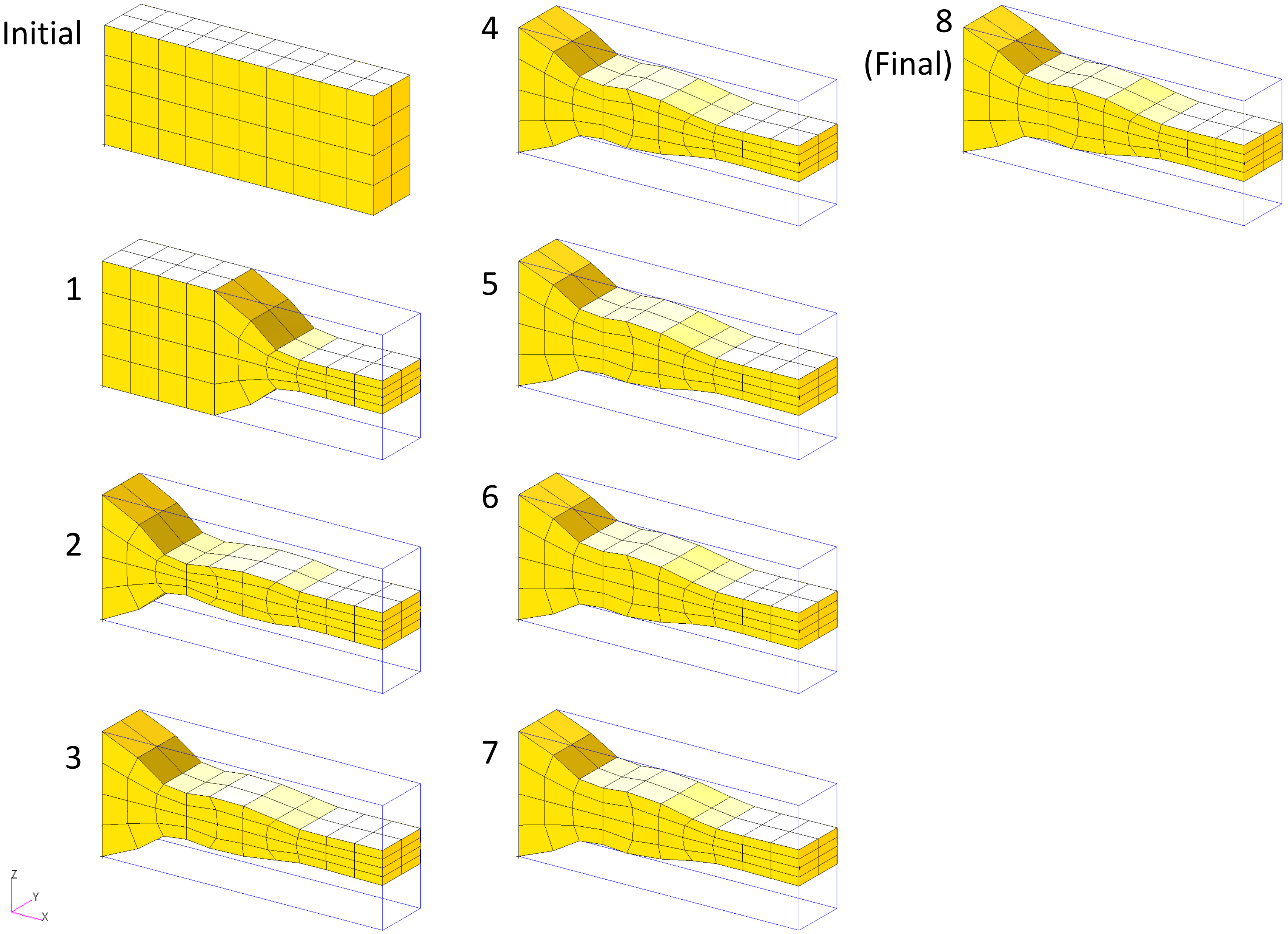
5 | 10 | 20 | 50 | 100 | 200

Review Optimization Results

1. Δy is the difference between the current and next design cycle. Δy values for shape 1 (y1) are overlaid on the variable history plot.
2. The shape change during the optimization is characterized by a series of contractions and expansions of shapes 1 and 2.

Shape Changes for Each Design Cycle

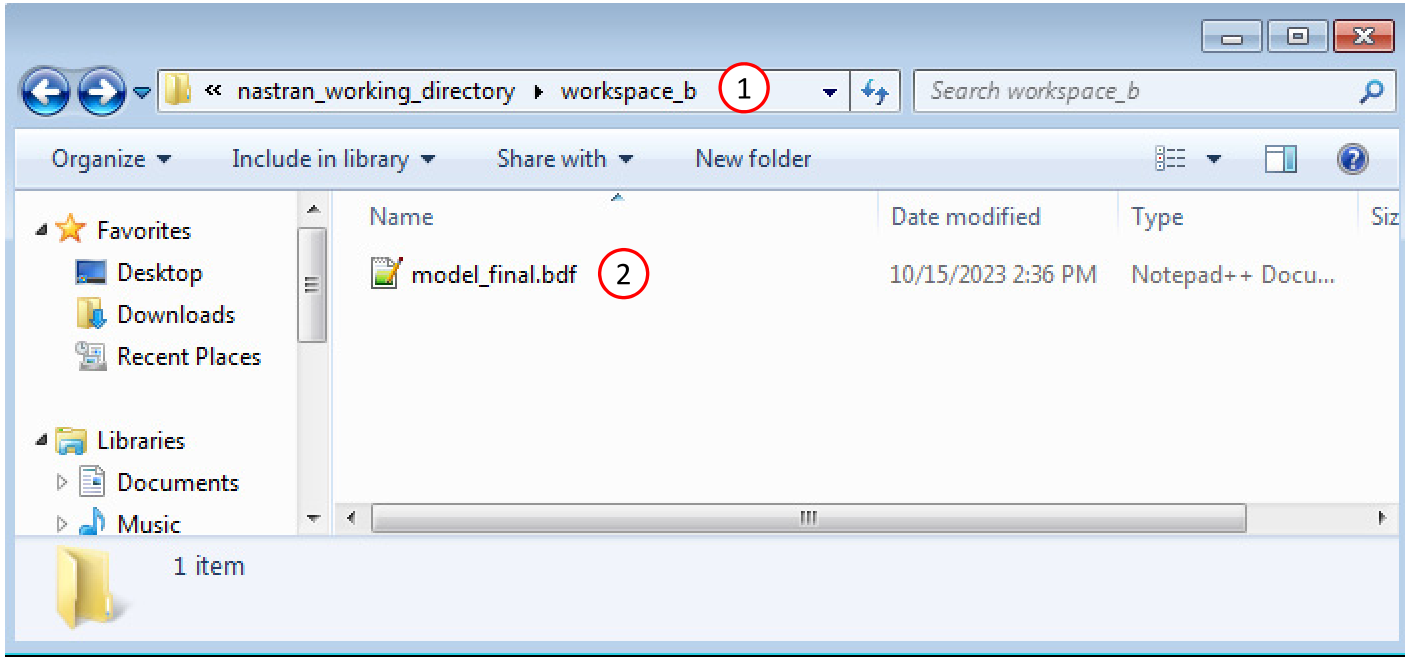
The total shape change is a linear combination of all the shapes. In this exercise, 2 shapes were defined, so the total shape change is a linear combination of 2 shapes.



New Updated BDF File

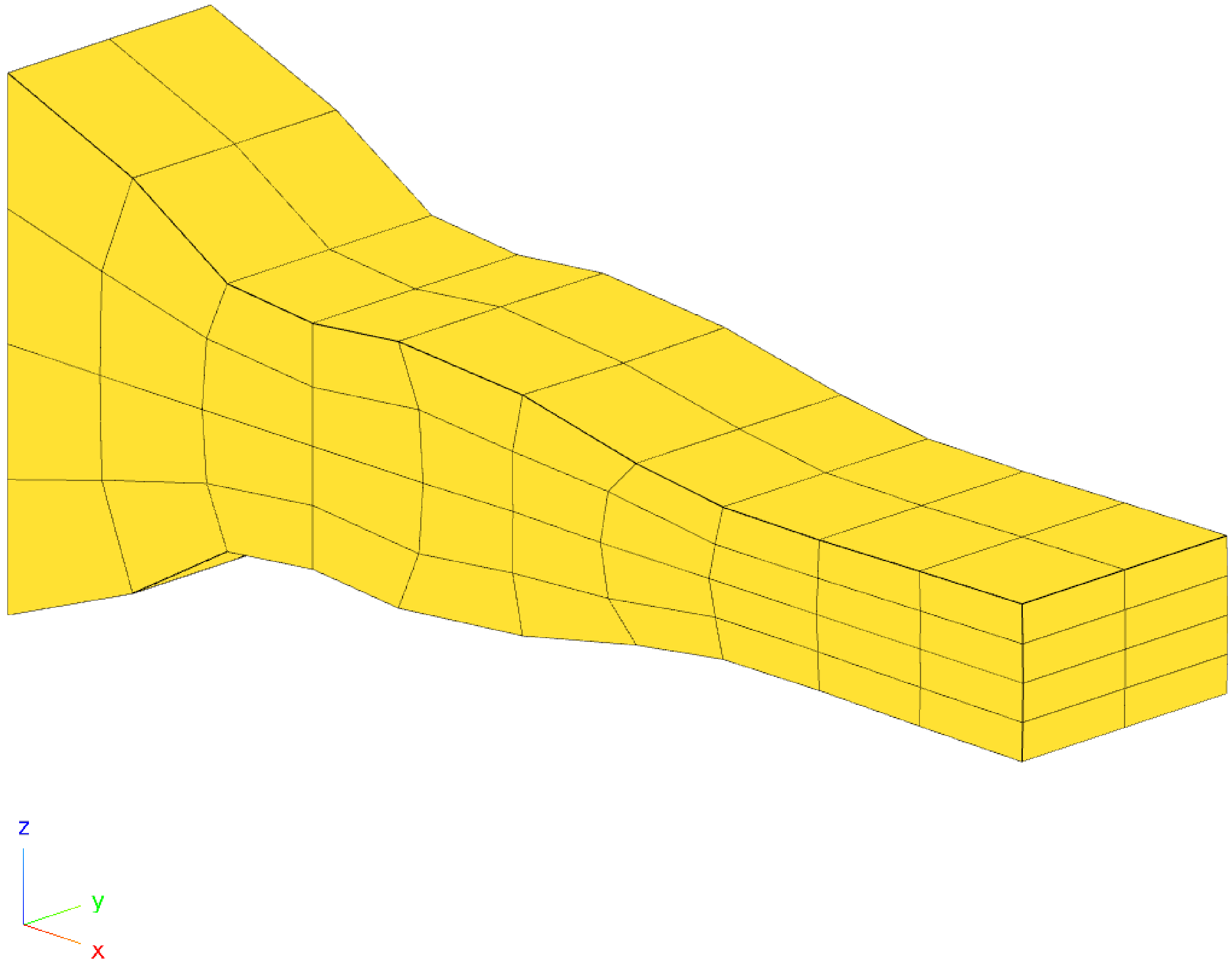
1. After the optimization, a new directory named workspace_b is created

2. This directory contains a new BDF file where the node positions have been updated to reflect the optimized shape. Specifically, the optimized GRID entries found in the file model.pch were used to replace the old GRID entries.



Optimized Shape

1. The Viewer is used to import the new file model_final.bdf. The shape is confirmed to have changed.



Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

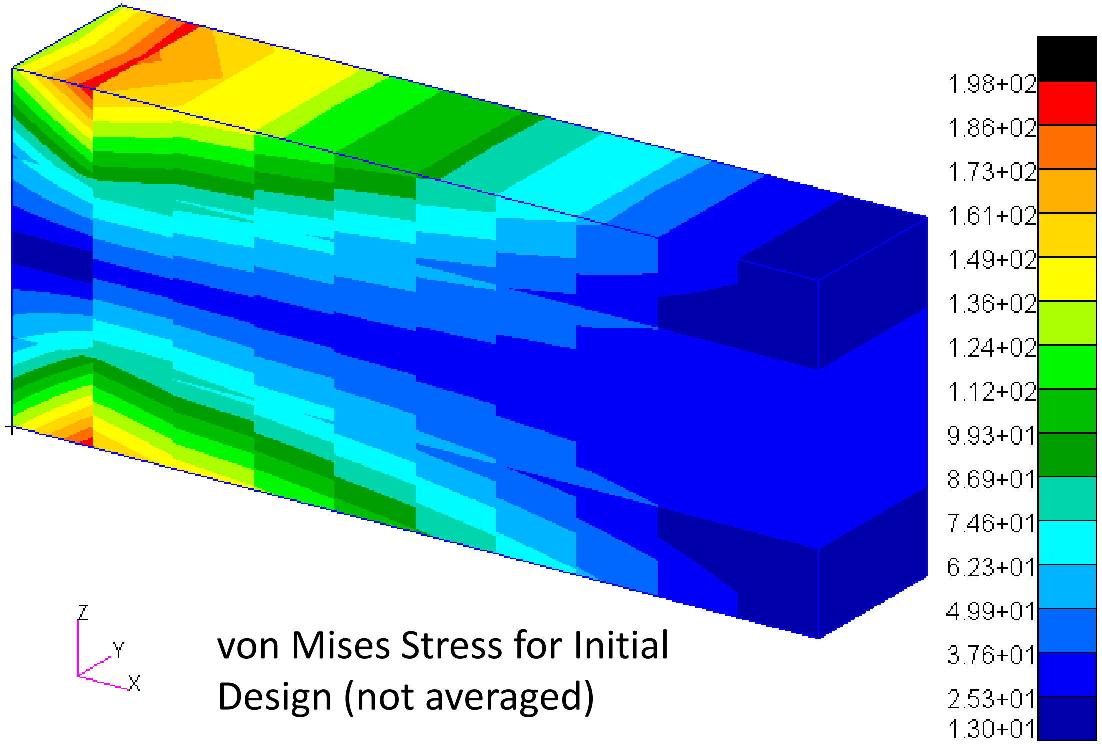
or contact

christian@ the-engineering-lab.com

Results

Before Optimization

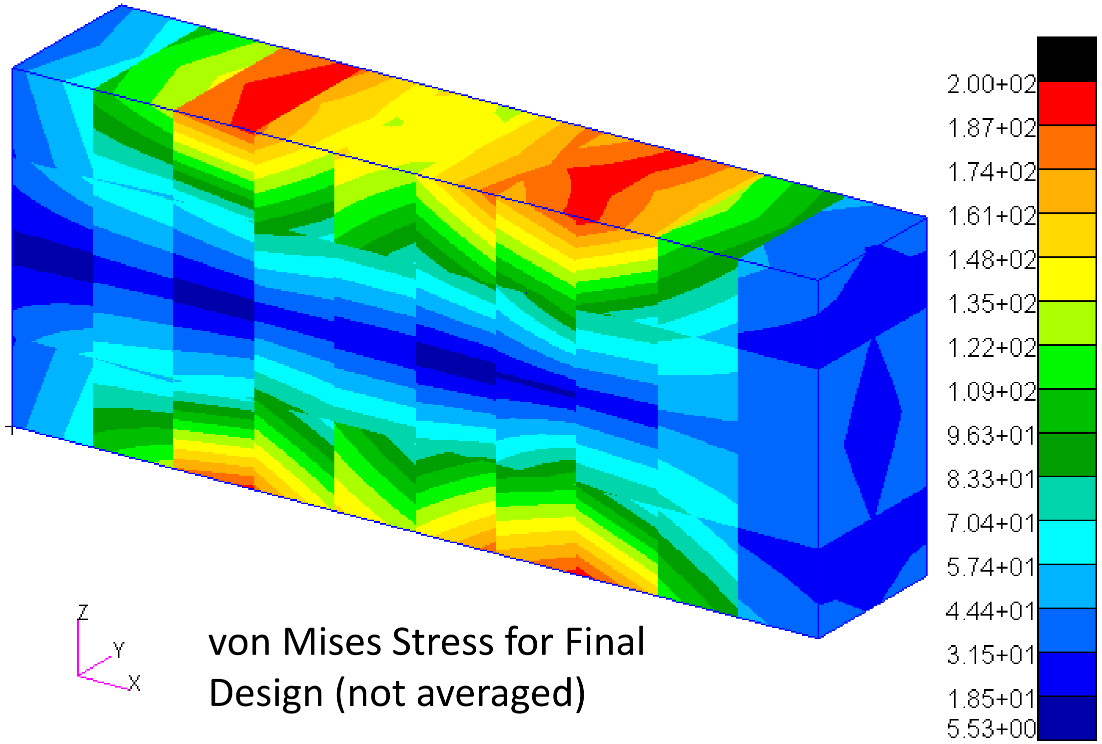
- Weight: 80.0



von Mises Stress for Initial Design (not averaged)

After Optimization

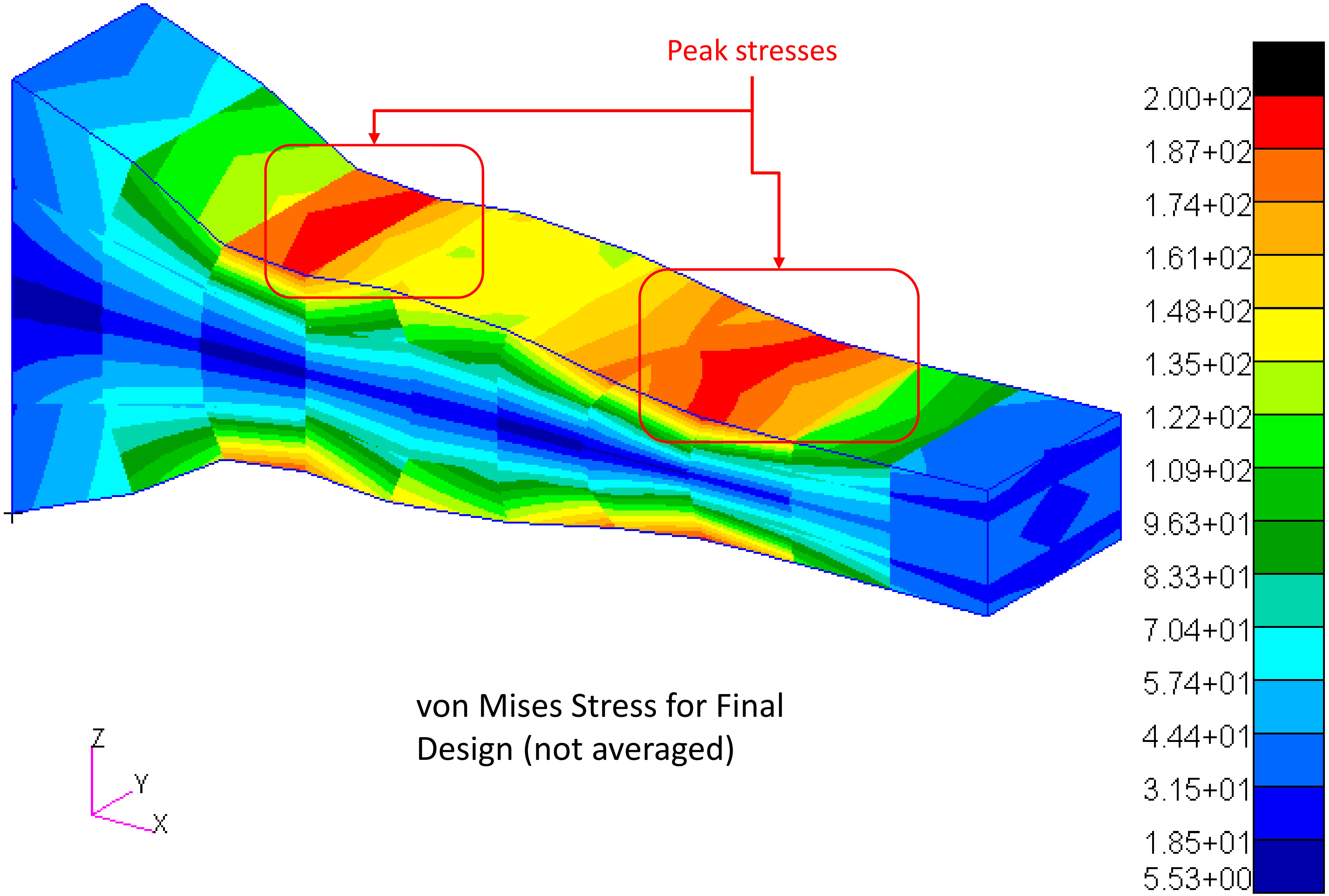
- Weight: 36.77



von Mises Stress for Final Design (not averaged)

Results

The initial design had peak stresses at the fixed end of the cantilever beam. After a shape optimization, peak stresses are found at various locations on the beam. This is indication that material has been distributed more effectively while satisfying the stress constraints.



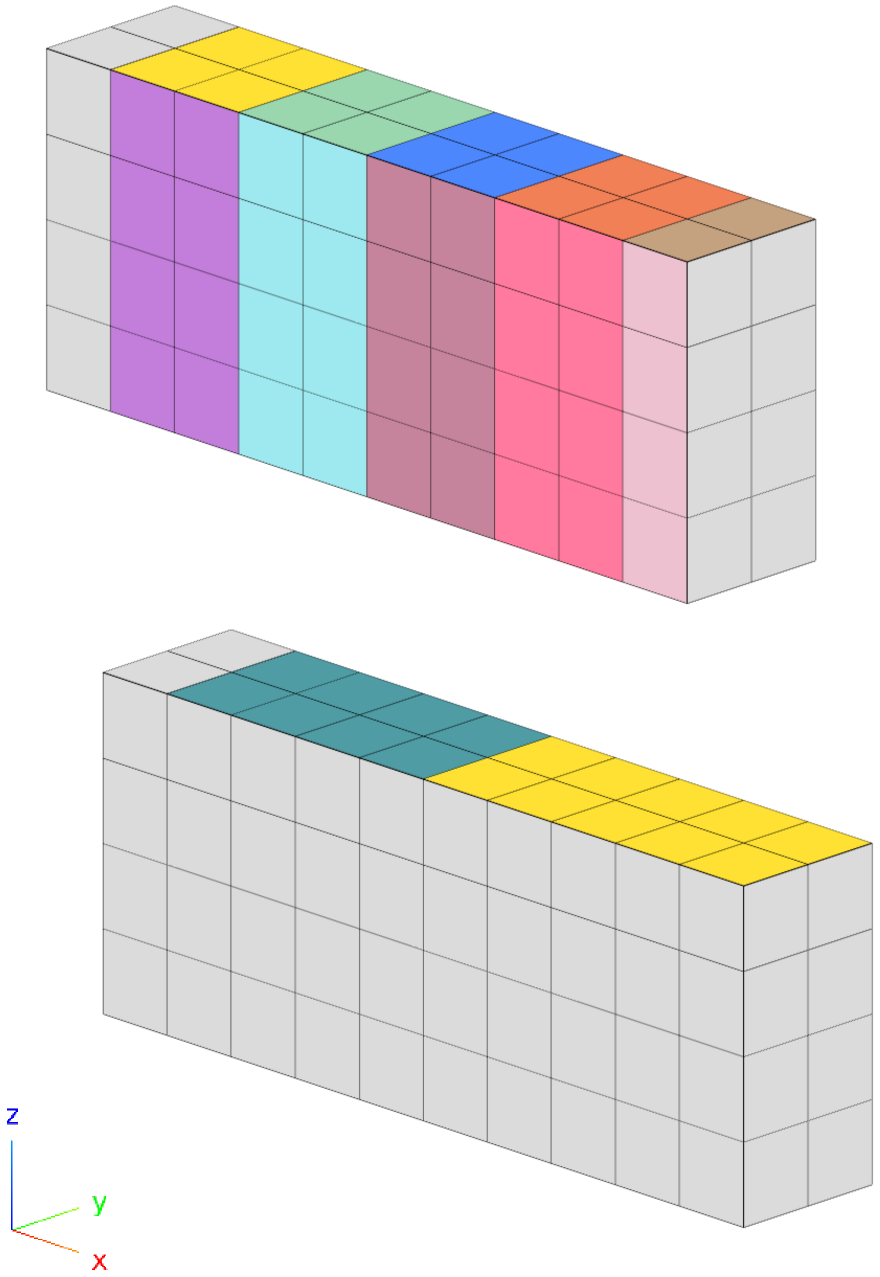
von Mises Stress for Final Design (not averaged)

Final Comments

This workshop involved creating 2 shapes that describe shape changes in the z direction.

This workshop was repeated with up to 17 shapes as shown to the right. Shapes 3-12 and describe shape changes in the z direction. Shapes 13-17 describe shape changes in the y direction. Shapes 18 and 19 are similar to the two original shapes that were created in this workshop.

The reader is encouraged to repeat this workshop but with additional shapes.



Content only available to professional engineers and students.

For access, visit

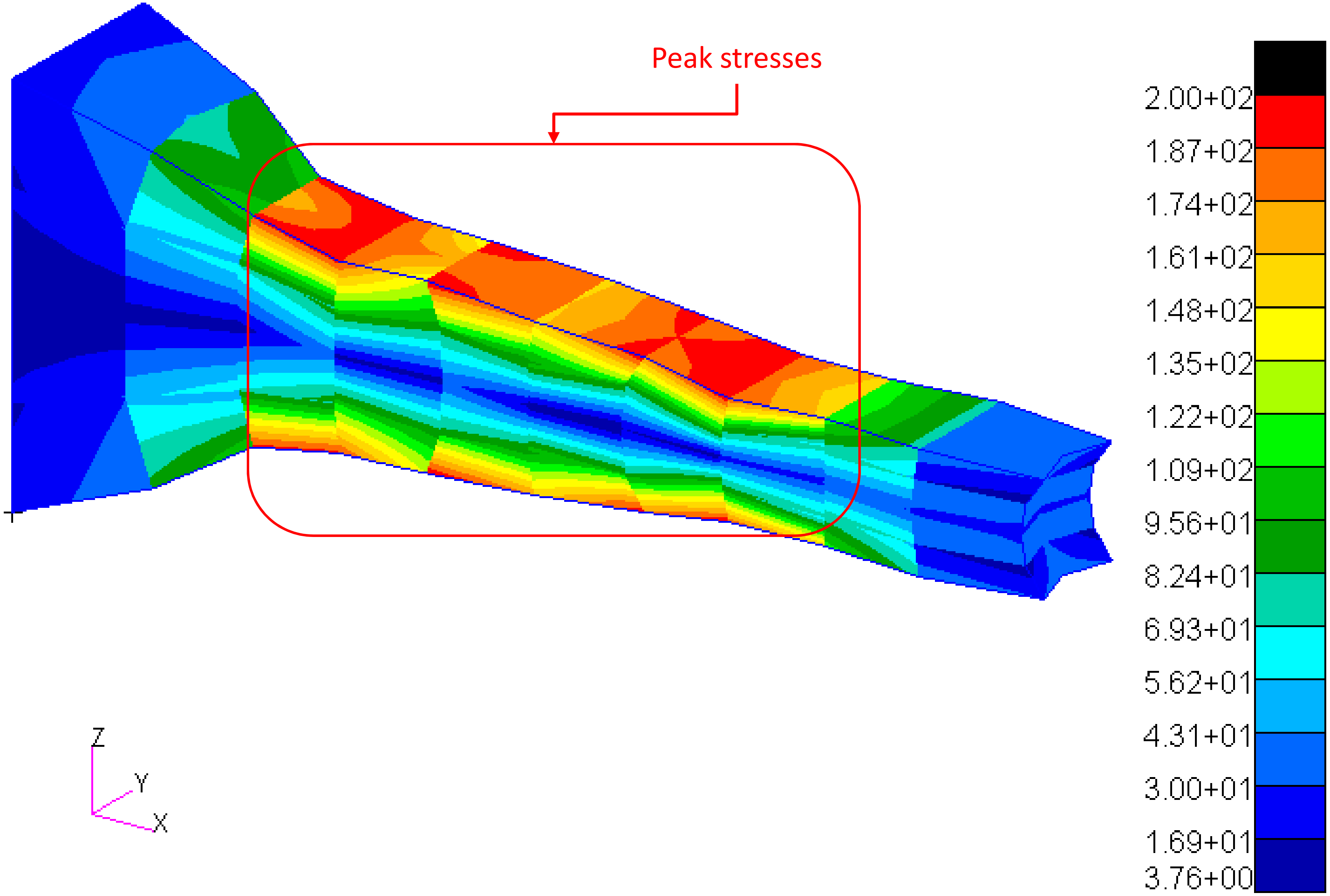
the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Final Comments

When more shapes are used, this allows for a larger region with constant stress field on the boundary of the beam and yields greater mass savings.



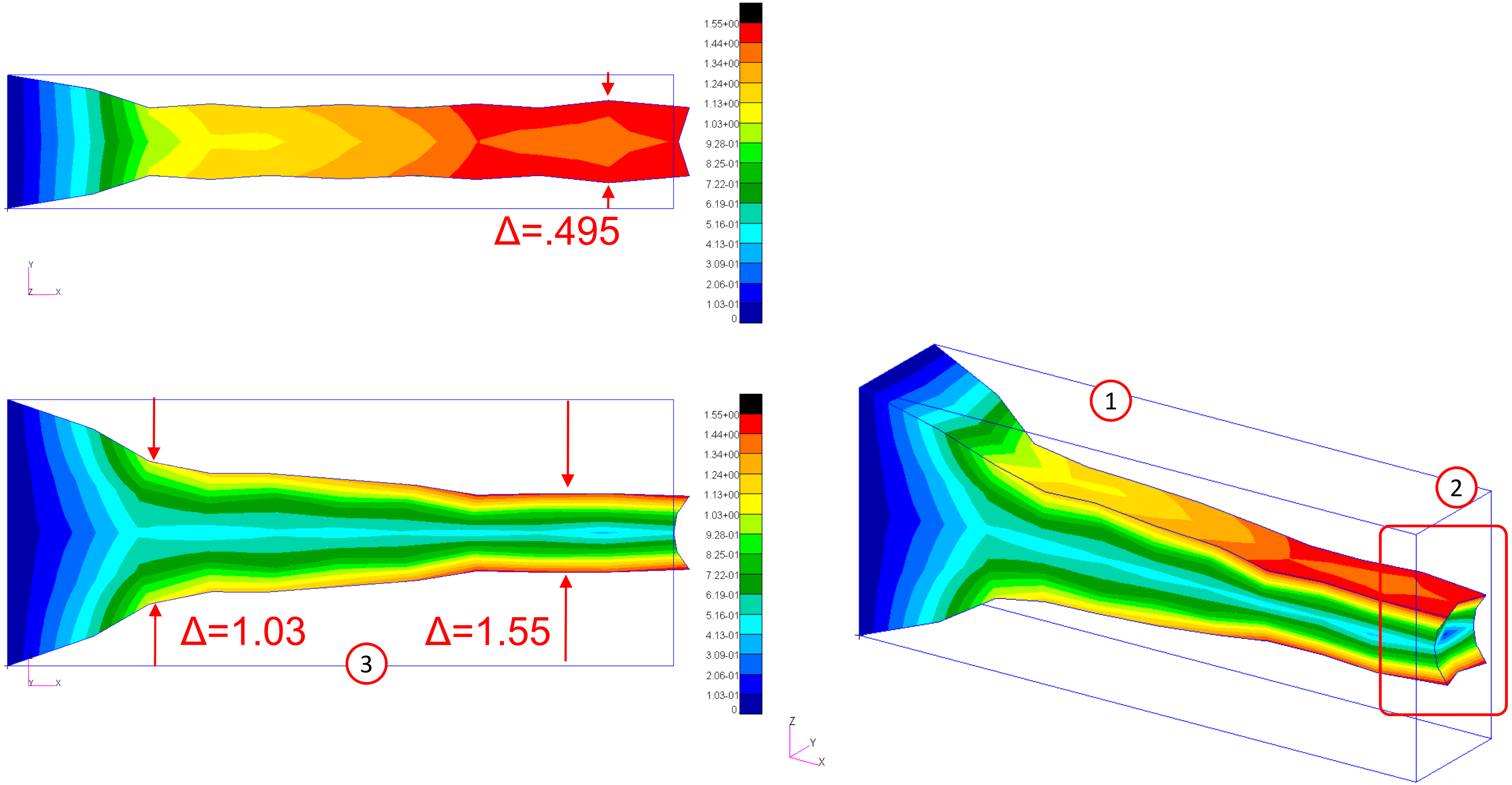
Final Comments

The shape change resulting from a shape optimization, when 17 shapes are used, is displayed to the right. The fringe plot represents the shape change.

1. Since shapes were created that allow for shape changes in the y and z direction, the optimized beam does reflect shape changes in the y and z directions.

2. After a shape optimization, in a majority of cases, the final mesh must be updated due to mesh distortions.

3. The approximate shape change values Δ=1.03, Δ=1.55 and Δ=.495 are used to update the original geometry and mesh.

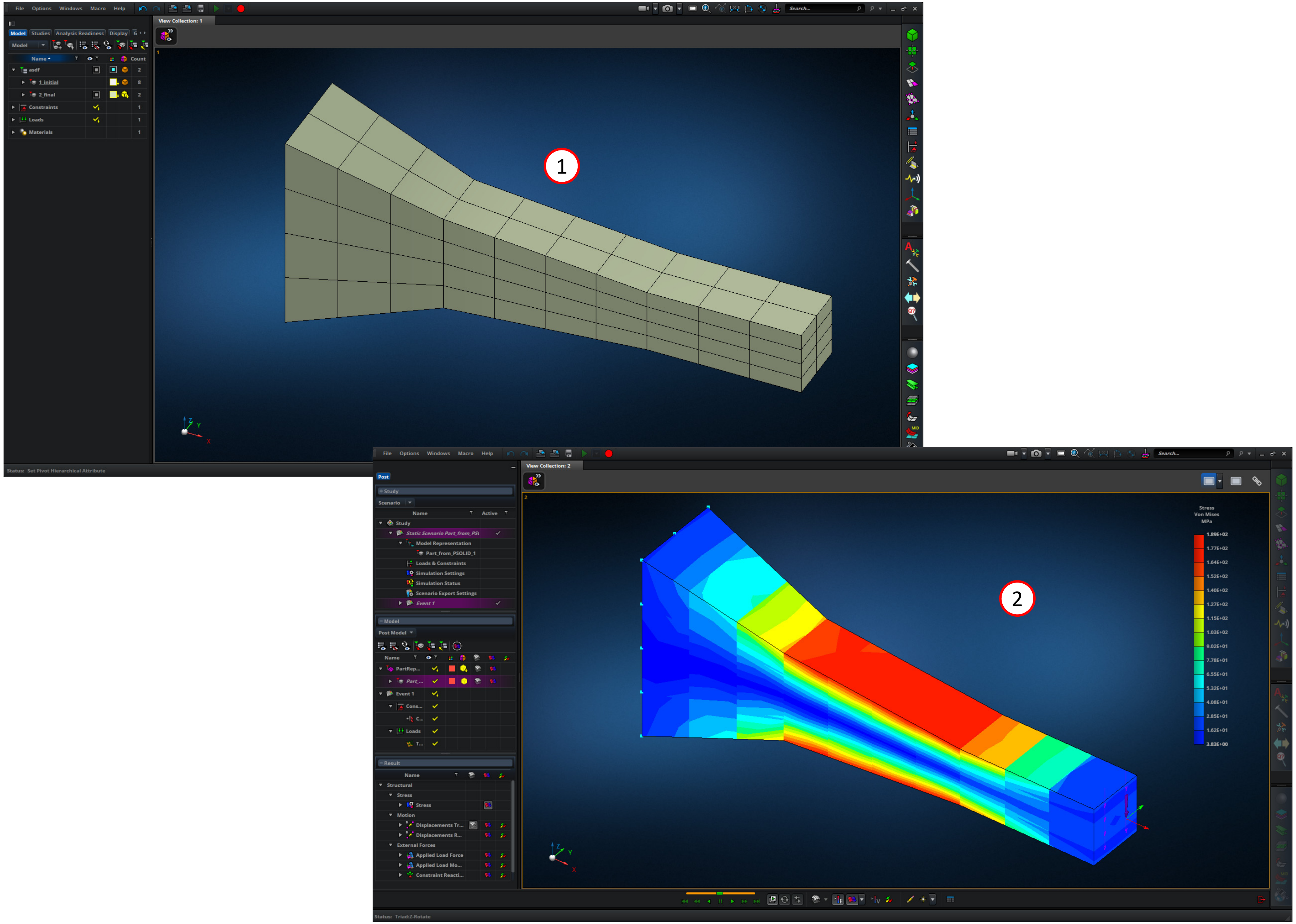


Final Comments

1. MSC Apex is used to update the original geometry and mesh.

2. A stress analysis yields a maximum stress of 189, which is well under the stress upper bound of 200.

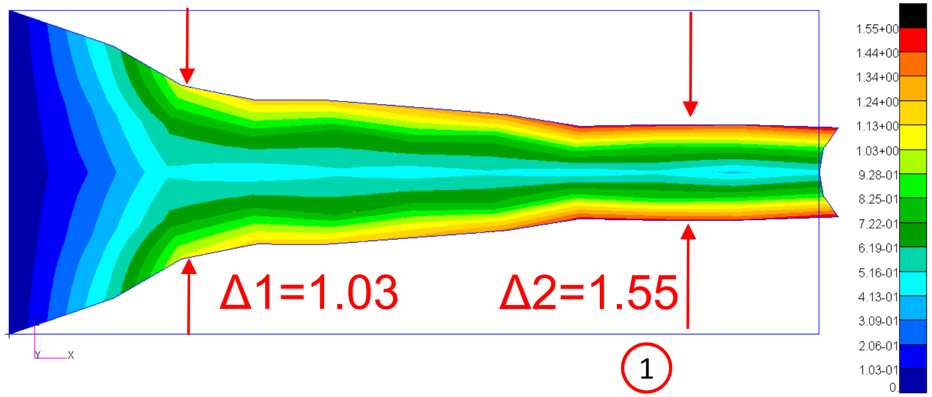
Since the mass has been reduced and the stress constraints are satisfied, this is a successful shape optimization.

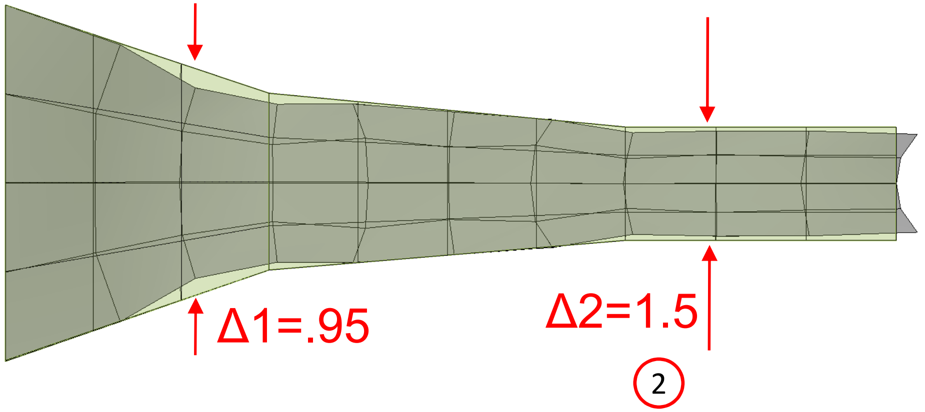


Final Comments

When manually updating the geometry and mesh, the updated mesh should envelope the mesh generated by shape optimization.

1. Note that the shape optimization may suggest shape changes of Δ=1.03, Δ=1.55 .
2. A conservative approach is taken and slightly reduced shape changes of Δ=0.95, Δ=1.5 are used. The reduction is done to ensure the shape optimization mesh is contained within the new mesh. If the shape change applied is too aggressive, i.e. the updated mesh does not envelope the shape optimization mesh, the structure might have too little material to satisfy the constraints.





Shape Optimization Mesh
Manually Updated Mesh

Table of Shape Change Values to Consider

	Shape Change from Shape Optimization	Conservative Shape Change Applied	Aggressive Shape Change Applied
Δ1	1.03	.95	1.1
Δ2	1.55	1.5	1.8
Comments	These values are obtained from the shape optimization.	These values are the shape optimization values but reduced at the engineer's discretion.	These values are greater than the values obtained from the shape optimization. These shape change values could leave the structure with too little material to satisfy the constraints.

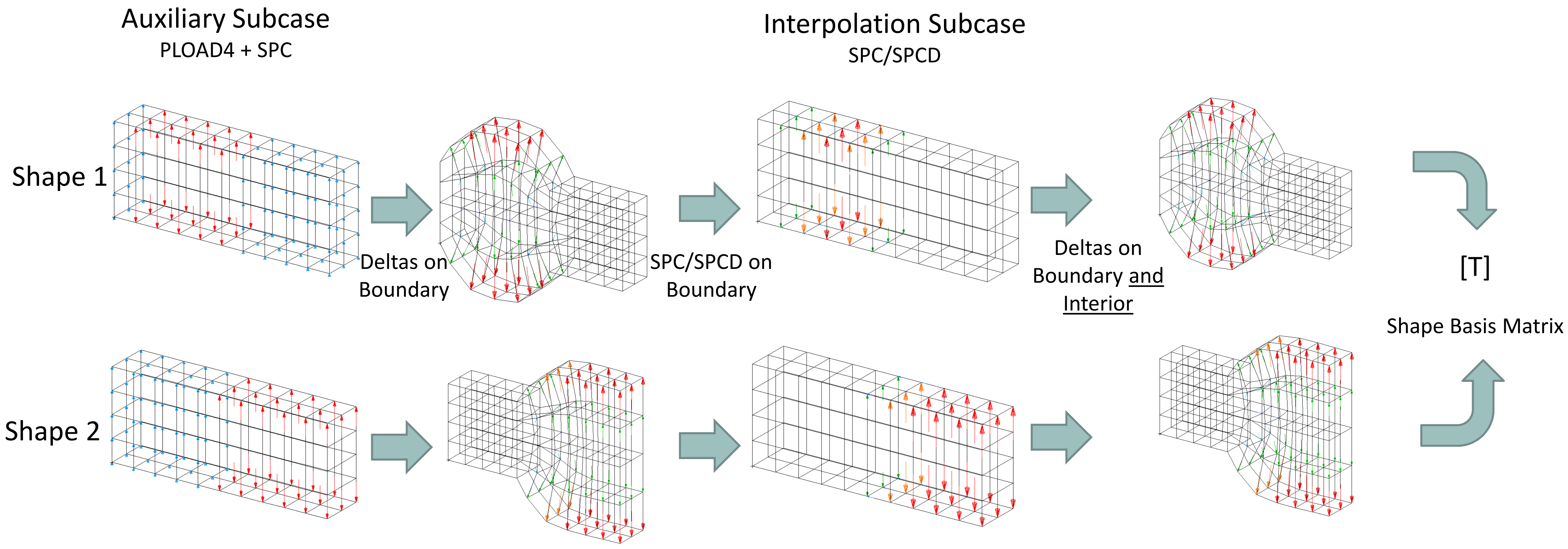
End of Tutorial

Appendix

Appendix Contents

- Frequently Asked Questions
  - How does MSC Nastran generate shape basis vectors?
  - Why the scaling factor on DVBSHAP?
  - How is the shape basis matrix constructed?
  - How to import previous BDF files?

How does MSC Nastran generate shape basis vectors?



Why the scaling factor on DVBSHAP?

1. Scaling factors were specified on the DVBSHAP entry

Each shape should describe a unit change

File: design_shapes_psolid_1.bdf

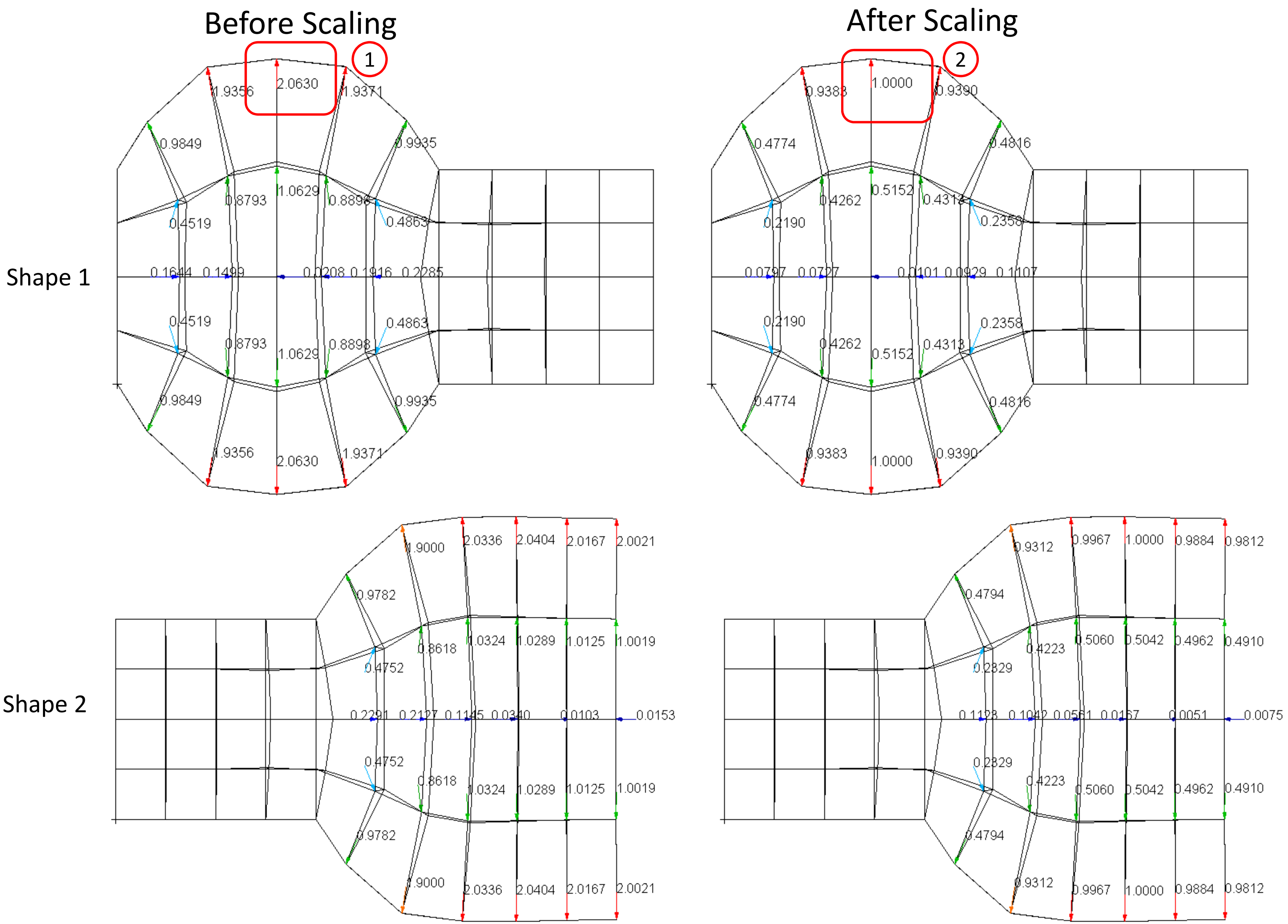
```
$   1   ||   2   ||   3   ||   4   ||   5   ||   6   ||   7   ||   8   ||   9   ||  10   |
DVBSHAP 200001  1       1       .4847393
DVBSHAP 200002  1       2       .4900893
DESVAR  200001  y1      10.     8.5     10.
DESVAR  200002  y2      10.     8.5     10.
BNDGRID 123     1       2       3       4       5       6       7
        8       9       10      11      12      13      14      15
        16      17      18      19      20      21      22      23
        24      25      26      27      28      29      30      31
        32      33      34      35      36      37      38      39
        40      41      42      43      44      45      55      56
        57      58      59      60      61      62      63      64
        65      66      67      68      69      70      71      72
        73      74      75      76      77      78      88      89
        90      91      92      93      94      95      96      97
        98      99      100     101     102     103     104     105
        106     107     108     109     110     111     121     122
        123     124     125     126     127     128     129     130
        131     132     133     134     135     136     137     138
        139     140     141     142     143     144     145     146
        147     148     149     150     151     152     153     154
        155     156     157     158     159     160     161     162
        163     164     165
```

1

Why the scaling factor on DVBSHAP?

1. For shape 1, the largest value was 2.0830 before scaling.
2. After scaling, the largest value is now 1.0.

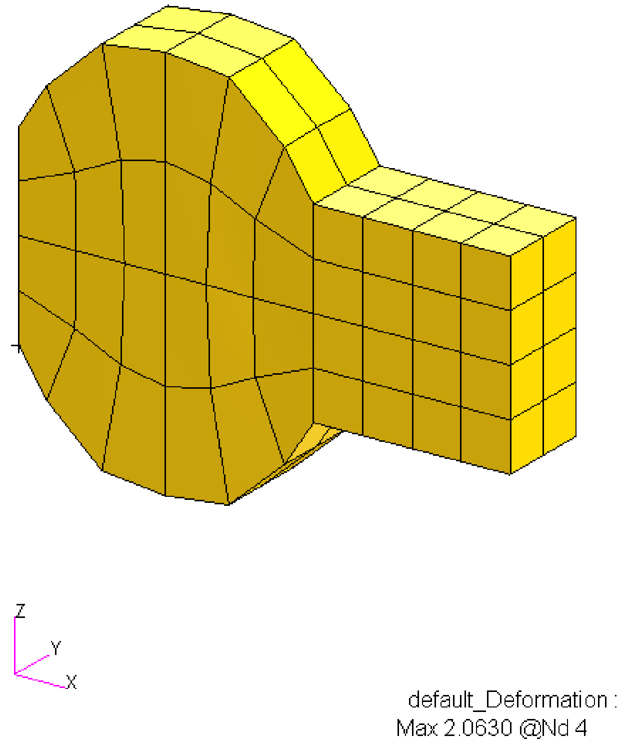
Each shape should describe a unit change



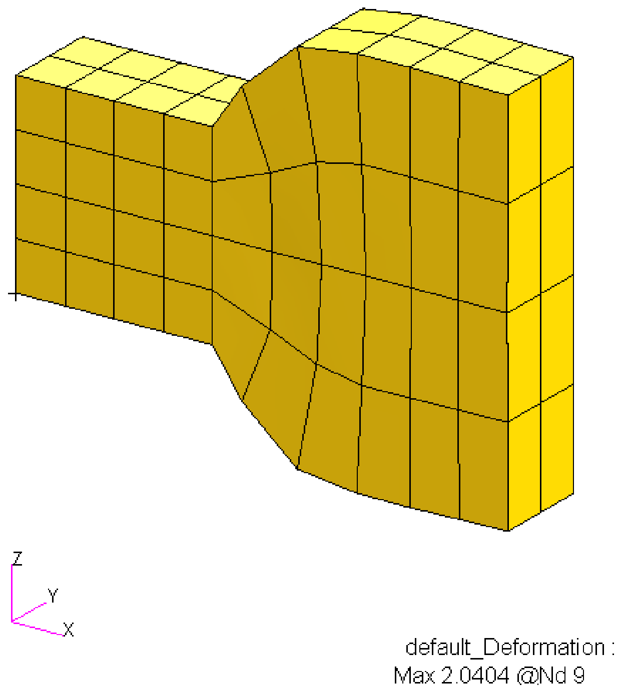
Why the scaling factor on DVBSHAP?

For each shape's auxiliary subcase, find the max resultant. The scaling factor for the shape is 1 divided by the max resultant.

Shape 1



Shape 2



1 / 2.0630 ≈ .4847393

1 / 2.0404 ≈ .4900893

```
DVBSHAP 200001   1        1         .4847393
DVBSHAP 200002   1        2         .4900893
```

How is the Shape Basis Matrix Constructed?

1. Let T1 be shape basis vector 1 yielded by the interpolation subcase. The entire vector is multiplied by the scaling factor so that the shape basis vector yields a one unit change.

2. Let T2 be shape basis vector 2 yielded by the interpolation subcase. The entire vector is multiplied by the scaling factor so that the shape basis vector yields a one unit change.

Since [T] expresses unit changes, Δy in the expression below will have units of length.

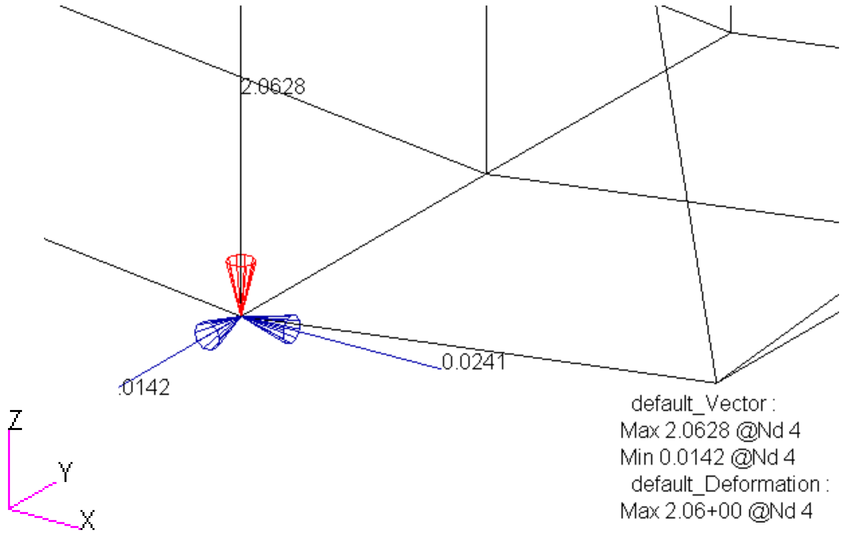
$$\{G\}_{i+1} = \{G\}_i + [T]\{\Delta y\}$$

If [T] is not scaled, then Δy will not be in units of length, which will make it difficult to interpret the design variables.

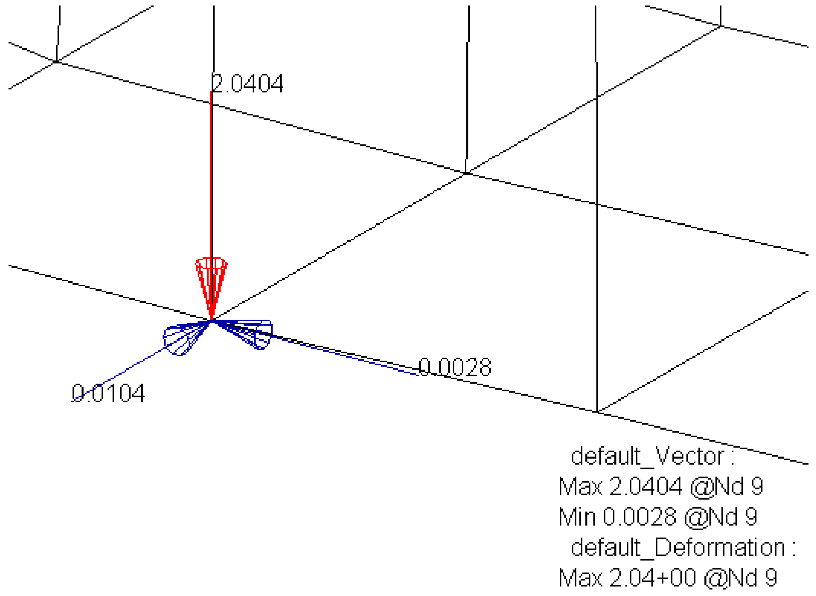
This model has 165 grids. With an x, y and z components defined for each grid. Each shape basis vector will have 165 x 3 = 495 elements. To the right, only 3 elements are displayed for basis vectors 1 and 2.

It should be noted that [T] is automatically updated after design cycle if geometric or analytic boundary shapes are defined. This workshop defined analytic boundary shapes. Shapes defined via manual grid variation or direct input of shapes do not update [T] after each design cycle and often will yielded distorted meshes.

Shape 1



Shape 2



$$T_1 = \begin{bmatrix} \vdots \\ .0241 \\ .0142 \\ 2.0628 \\ \vdots \end{bmatrix} \times .4847393$$

Elements of shape basis vector 1 for GRID 4

$$T_2 = \begin{bmatrix} \vdots \\ .0028 \\ .0104 \\ 2.0404 \\ \vdots \end{bmatrix} \times .4900893$$

Elements of shape basis vector 2 for GRID 9

How to import and edit previous files

Import

1. Return to the window or tab that has the Viewer opened
2. Refresh the web page to start a new session

- Refreshing the page is only required when the *Select files* button is disabled.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Import

1. Click Upload BDF
2. Click Select Files
3. Navigate to the folder named nastran_working_directory
4. Select all the BDF files
5. Click Open
6. Click Upload files

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com

Import

1. Click Shape

Previous shapes may now be reconfigured or new shapes may be added.

Content only available to professional engineers and students.

For access, visit

the-engineering-lab.com

or contact

christian@ the-engineering-lab.com