MSC Nastran Topology Optimization - Multidiscipline - Static Loading and Natural Frequency

AN MSC NASTRAN SOL 200 TUTORIAL
Goal: Use Nastran SOL 200 Optimization

Before Optimization
- Mass: 9.73E-06

After Optimization
- Mass: 7.05E-06 (~25% mass reduction)
- Maximize stiffness
- Maximize first natural frequency
Details of the structural model
Optimization Problem Statement

**Design Region/Variables**

x₁: PSHELL 1

**Design Equation Objective**

R₀: Minimize the sum of normalized compliance and normalized natural frequency

\[
R₀ = \frac{a₁}{36.7} + \frac{86.4}{a₂}
\]

a₁: Compliance of SUBCASE 1  
a₂: Natural frequency of mode 1 of SUBCASE 2

**Design Constraints**

r₁: Fractional mass

\[ r₁ < 0.75 \] (25% mass reduction)

r₂: The z component of displacement at node 714

\[ z < r₂ \]

r₃: The natural frequency of mode 1

\[ 20 < r₃ \]
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
- Topology Viewer

More Information Available in the Appendix
Contact me

• Nastran SOL 200 training
• Nastran SOL 200 questions
• Structural optimization questions
• Access to the MSC Nastran SOL 200 Web App

christian@the-engineering-lab.com
Tutorial
Tutorial Overview

1. Start with a .bdf or .dat file
2. Use the MSC Nastran 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

- **Multidiscipline Optimization** - This example is optimizing for static analysis and normal modes analysis.
- **Equation Objective with SUBCASE spanning responses** - The Equation Objective defined in this example is dependent on responses from multiple SUBCASEs. The $a_1$ response comes from SUBCASE 1 and $a_2$ comes from SUBCASE 2. The use of DRSPAN enables the specification of which SUBCASEs produces the necessary values.

$$ R_0 = \frac{a_1}{36.7} + \frac{86.4}{a_2} $$

Step 4 - Configure DRSPAN for Synthetic Objective and Constraints
MSC Nastran SOL 200 Web App

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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.
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.
Merge the BDF files

There are two separate BDF files. Both files contain the simply supported plate. One BDF file is configured to perform a Linear Static analysis. The second BDF file is configured to perform a Normal Modes Analysis. To perform an optimization for both static and normal modes analysis, the files must be combined.

1. Open `simply_supported_plate_modes.bdf`. Take SUBCASE 1 and move it to `simply_supported_plate_statics.bdf`. Rename SUBCASE 1 to SUBCASE 2 to avoid duplicate SUBCASEs.

2. Take the line with EIGRL at the beginning and move it to `simply_supported_plate_statics.bdf`.

3. Save `simply_supported_plate_statics.bdf` as `simply_supported_plate_combined.bdf`.

• In this tutorial a Multidisciplinary Optimization is performed. This is done by:
  1. Merging the BDF files
  2. Adding different ANALYSIS commands to each SUBCASE.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Open the Correct Page

1. Click on the indicated link

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

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Upload BDF Files

1. Click 1. Select Files and select simply_supported_plate_combined.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.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Create Design Region

1. Click on the plus (+) icons to set PSHELL 1 as a Design Region

2. The Design Region is displayed in the next table. Any element associated with PSHELL 1 will be allowed to change during the optimization.

- In traditional Size optimization, individual design variables are created. It is slightly different for Topology optimization. When a design region is set, each element in the design region has a design variable created for it. Each design variable corresponds to the Normalized Material Density of that element, see the appendix for additional details.
- 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.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Create Design Objective

1. Click on Objective
2. Click on Switch to Equation Objective

- There are 2 methods of setting an objective.
  - Method 1 – Select an objective from a given list of responses, e.g. Weight, Volume, etc.
  - Method 2 – Create an equation.
- This example uses Method 2 for the objective.
Create Design Objective

1. Scroll down to section Step A - Optional - Create additional responses
2. Ensure the analysis type is set to SOL 101 - Statics
3. Type in ‘comp’ to the search box
4. Click the plus (+) icon to set compliance as a response type
5. Set the analysis type as SOL 103 - Normal Modes
6. Clear the search box.
7. Click the plus (+) icon to set Frequency as a response type

- This example performs a multidisciplinary optimization for statics and modes. The objective will be a combination of compliance and natural frequency. The responses necessary for compliance and natural frequency are created on this page.
Create Design Objective

1. Scroll to section Step B - Optional - Adjust responses

2. For response a2, ensure ATTA is set to 1 which represents mode 1

- a1 is read as the compliance. Compliance is twice the total strain energy. Refer to the Appendix for more information about compliance.
- a2 is the natural frequency for mode 1.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Create Design Objective

1. Scroll to section Step 1 - Adjust equation objective

2. Type this equation into the box
   • \( \frac{a1}{1.0} + \frac{2.0}{a2} \)

The values 1.0 and 2.0 will be replaced later in this tutorial.

• The objective is a combination of a1 and a2. The objective must be conditioned properly such that if one response is larger than the other, e.g. \( a1=0.01 \) and \( a2=2000000 \), a change in the smaller of the two is still detected by the optimizer. This is addressed by normalizing both a1 and a2. For now, it is not known which values are necessary to normalize a1 or a2, so values 1 and 2 are used for now.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Create Design Constraints

1. Click Constraints
2. Set the analysis type to SOL 101 - Statics
3. Select the plus(+) icon for Displacement
4. Configure the following for r2
   - ATTA: 3 (3 - T3 or Z component)
   - ATTi: 714 (node 714)
   - Lower Allowed Limit: -5.0
5. Configure the following for r1
   1. Upper Allowed Limit: .75

- By default, an FRMASS constraint has been created automatically.
- 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.

Nastran SOL 200 Web App - Topology

Step 1 - Select constraints

Select an analysis type

SOL 101 - Statics

Select a response

<table>
<thead>
<tr>
<th>Response Description</th>
<th>Response Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>WEIGHT</td>
</tr>
<tr>
<td>Volume</td>
<td>VOLUME</td>
</tr>
<tr>
<td>Displacement</td>
<td>DISP</td>
</tr>
<tr>
<td>Strain</td>
<td>STRAIN</td>
</tr>
<tr>
<td>Element Strain Energy</td>
<td>ESE</td>
</tr>
</tbody>
</table>

Step 2 - Adjust constraints

+ Options

<table>
<thead>
<tr>
<th>Label</th>
<th>Status</th>
<th>Response Type</th>
<th>Property Type</th>
<th>ATTA</th>
<th>ATTB</th>
<th>ATTi</th>
<th>Lower Allowed Limit</th>
<th>Upper Allowed Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>r1</td>
<td></td>
<td>FRMASS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>.75</td>
</tr>
<tr>
<td>r2</td>
<td></td>
<td>DISP</td>
<td>3 - T3 (Rectangular z, Cylindrical z)</td>
<td>3</td>
<td>714</td>
<td></td>
<td></td>
<td>-5.0</td>
</tr>
</tbody>
</table>

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Create Design Constraints

1. Set the analysis type to SOL 103 - Modes
2. Select the plus(+) icon for Frequency
3. Configure the following for r3
   - ATTA: 1 (mode 1)

Previously, a displacement constraint was created and will be assigned to the statics subcase.
On this page, a constraint on the natural frequency is created and will be assigned to the modes subcase.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Assign Constraints to Load Cases (SUBCASES)

1. Click Subcases
2. Mark the checkbox
3. Mark the checkbox
4. Set the analysis type as Normal Modes for SUBCASE 2
5. Mark the checkbox

- r1 or FRMASS constraint has been assigned to Global Constraints
- r2 or DISP constraint has been assigned to SUBCASE 1, the statics subcase
- r3 or FREQ constraint has been assigned to SUBCASE 2, the modes subcase
Configure DRSPAN

The Equation Objective is in terms of $a_1$ and $a_2$, where $a_1$ is a statics response and $a_2$ is a normal modes response. The DRSPAN functionality in Nastran is used to specify which SUBCASE generates $a_1$ and $a_2$.

1. Find this section: Step A - Optional - Configure DRSPAN for equation objective and constraints
2. Set $a_1$ to SUBCASE 1
3. Set $a_2$ to SUBCASE 2

Some responses, e.g. stress, strain, frequency, etc., are subcases dependent. The Equation Objective is composed of subcase dependent responses. If there are numerous subcases, the optimizer must be told which subcases the responses originate from. This is done via the DRSPAN command and is configured on this page.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Configure Optimization Settings

1. Click Settings
2. Set DESMAX to 1

- Recall that the Equation Objective is composed of $a_1$ and $a_2$ and the goal is to find values that normalize both $a_1$ and $a_2$. This is done by obtaining the current or initial values of $a_1$ and $a_2$. In the event the values are not known, you can perform an optimization for only one design cycle. The initial values are then available as shown on the following pages.
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

A new .zip file has been downloaded

1. Right click on the file
2. Click Extract All
3. Click Extract on the following window

- Always extract the contents of the ZIP file to a new, empty folder.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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 an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- The flexibility described above enables an alternate method of starting MSC Nastran: 1) Move the BDF files to a remote machine. 2) Manually start MSC Nastran on the remote machine. 3) Move the BDF, F06 and LOG files to the local machine. 4) Click “Start MSC Nastran,” and the results will be automatically displayed.

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

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Status

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.

Nastran SOL 200 Web App - Status

<table>
<thead>
<tr>
<th>Name</th>
<th>Status of Job</th>
<th>Design Cycle</th>
<th>RUN TERMINATED DUE TO</th>
</tr>
</thead>
<tbody>
<tr>
<td>model.bdf</td>
<td>Running</td>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Review Optimization Results

After MSC Nastran is successfully complete, the results will be automatically uploaded.

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.
Obtain quantities from .f06

The previous optimization was done only for one design cycle. The intent was to obtain two quantities $a_1$ and $a_2$, the compliance and 1st natural frequency, respectively.

1. Go to the Results section
2. Click on the Responses link

- Recall that the initial values for $a_1$ and $a_2$ must be obtained. These initial values are contained in the .f06 file. The Responses app is opened and will be used to extract the initial values for $a_1$ and $a_2$. 

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Obtain quantities from .f06

1. Select the file named model.f06
2. Click Upload files
3. Enter ‘a’ into the Label filter box
4. Enter ‘INITIAL’ into the Design Cycle filter box
5. The values for a1 and a2 are now visible and will be used in the next slide

- The filter or search boxes are used to display the values for a1 and a2. The Equation Objective is currently set to \( R0 = \frac{a1}{1.0 + 2.0 / a2} \). The initial values will be used to replace 1.0 and 2.0.
Responses in Design Model

1. The previous table extracted values from the section titled Responses in Design Model
2. The value of a1 is visible
3. The value of a2 is visible

• As a side note, the values reported in the Responses app are values found in the F06 file, section RESPONSES IN DESIGN MODEL.
Update Design Objective

The previous values, 3.6788E+01 and 8.6430E+01, from the .f06 file will be used to update the Design Objective.

1. Go back to the web app and click on Objective

2. Update the equation so it reads:
   - \( \frac{a1}{3.6788E1} + \frac{8.6430E1}{a2} \)

The reader may realize that a2 is the denominator in its normalized term. This is done for the following reason. The goal is to minimize compliance, but to maximize the natural frequency, note the paradox in goals, i.e. minimization and maximization. This paradox is resolved by inverting the a2 term and placing a2 in the denominator. As the natural frequency a2 value is increased or maximized, the term “8.6430E1 / a2” goes to 0.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
### Configure Optimization Settings

1. Click Settings
2. Set DESMAX to 100

- Previously, the topology optimization was limited to only 1 design cycle. The max is changed to 100 to allow for a normal topology optimization.

---

**Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com**
Delete old files

1. Select old files and delete them. New files will be downloaded in the next step.

• Deleting the files is optional, but is encouraged to avoid confusion with other files and folders.
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

A new .zip file has been downloaded

1. Right click on the file
2. Click Extract All
3. Click Extract on the following window

- Always extract the contents of the ZIP file to a new, empty folder.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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 an optimization, the results will be automatically displayed as long as the following files are present: BDF, F06 and LOG.
- The flexibility described above enables an alternate method of starting MSC Nastran: 1) Move the BDF files to a remote machine. 2) Manually start MSC Nastran on the remote machine. 3) Move the BDF, F06 and LOG files to the local machine. 4) Click “Start MSC Nastran,” and the results will be automatically displayed.

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

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Status

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.

---

Nastran SOL 200 Web App - Status

<table>
<thead>
<tr>
<th>Name</th>
<th>Status of Job</th>
<th>Design Cycle</th>
<th>RUN TERMINATED DUE TO</th>
</tr>
</thead>
<tbody>
<tr>
<td>model.bdf</td>
<td>Running</td>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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.
- Note that initial value of the objective is 2.0. The Equation Objective has been configured such that the a1 and a2 terms are normalized or 1.0 each. For the initial design, R0 evaluates as follows:
  - \( R0 = \frac{a1}{3.6788E1} + \frac{8.6430E1}{a2} \)
  - \( = \frac{3.6788E1}{3.6788E1} + \frac{8.6430E1}{8.6430E1} \)
  - \( = 2.0 \)

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Review Optimization Results

1. Return to the Topology 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 Topology web app. The appendix has additional information regarding capabilities of the Topology Viewer.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Review Optimization Results

1. Under Step 1, click Select Files
2. Select the model.bdf and design_model.bdf files.
3. Click Open
4. 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. Under Step 2, click Select Files
2. Select the model.des file
3. Click Open
4. The results of the topology optimization are displayed

- When the DES file is uploaded, the topology results are automatically displayed. By default, elements with a normalized material density greater than a threshold of 0.3 are displayed. The threshold can be modified.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Review Optimization Results

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

- A normalized material density (NMD) 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 NMD close to 0 are not critical and can be removed.
- Common thresholds to use are typically in the range of .3 to .7

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Results

Before Optimization
- Mass: 9.73E-06

After Optimization
- Mass: 7.05E-06 (~25% mass reduction)
- Maximize stiffness
- Maximize first natural frequency
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
◦ Topology Viewer
What are the design variables in Topology Optimization?

The design variables in a topology optimization are normalized material densities ($p_i$) of each element.

$$ p_i = \frac{p_{ai}}{p_0} $$

$p_{ai}$ : The assumed material density of the element

$p_0$ : The true material density

$p_i$ : The normalized material density of the element

Consider one element in the model.

Suppose the original/true material density is 8000 kg/m$^3$.

If during the optimization, the normalized material density of the element is 1.0, then the assumed material density is

$$ P_{ai} = p_i \cdot p_0 $$

$$ = 1.0 \cdot 8000 $$

$$ = 8000 $$

If during the optimization, the normalized material density is .5, then the assumed material density is

$$ P_{ai} = p_i \cdot p_0 $$

$$ = .5 \cdot 8000 $$

$$ = 4000 $$
What are the design variables in Topology Optimization?

- The design variables or normalized material densities can vary between 0 and 1.
  - 1 - Normalized density values close to 1 are critical to the design
  - 0 - Normalized density values close to 0 are not critical to the design

During the optimization, the normalized material density of each element is allowed to vary between 0 and 1 (0 < \( p_i < 1 \))

\[ p_i = 1.0 \]
\[ p_i = 0.50 \]
\[ p_i = 0.0 \]
What are the design variables in Topology Optimization?

- It should be noted that during the optimization, elements are never removed. Instead, the normalized material density values are used to determine which elements should be kept or removed.
  - 1 - Normalized density values close to 1 are critical to the design
  - 0 - Normalized density values close to 0 are not critical to the design

The final values of design variables or normalized densities are plotted for each element.
What is FRMASS or Fractional Mass?

Since the design variables or normalized material densities 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.

\[
\text{FRMASS} = \sum p_i \cdot p_0 \cdot v_i
\]

- \( p_0 \) : The true material density
- \( p_i \) : The normalized material density of the element
- \( v_i \) : Volume of element

0) Suppose this is the optimization problem statement:
- Objective: Minimize compliance
- Constraint: FRMASS < .3

1) Prior to the optimization start, each material density is reduced from 1.0 to .3, and 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 variable (normalized material density) is allowed to range between 0 and .1, but the constraint that the 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

---

**Element Strain Energies**

<table>
<thead>
<tr>
<th>Element-Type = HEXA</th>
<th>Total Energy of All Elements in Problem = 9.11103E+03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Subcase 1</td>
<td>Total Energy of All Elements in Set = -1 = 9.11103E+03</td>
</tr>
<tr>
<td>Element-ID</td>
<td>Strain-Energy</td>
</tr>
<tr>
<td>25</td>
<td>8.05918E+02</td>
</tr>
<tr>
<td>32</td>
<td>8.05918E+02</td>
</tr>
<tr>
<td>33</td>
<td>8.05918E+02</td>
</tr>
<tr>
<td>40</td>
<td>8.05918E+02</td>
</tr>
</tbody>
</table>

**Summary of Design Cycle History**

(HARD CONVERGENCE ACHIEVED)

- Number of Finite Element Analyses Completed: 56
- Number of Optimizations w.r.t. Approximated Models: 55

<table>
<thead>
<tr>
<th>Cycle Number</th>
<th>Objective from Approximate Optimization</th>
<th>Objective from Exact Analysis</th>
<th>Fractional Error of Approximation</th>
<th>Maximum Value of Constraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial</td>
<td>5.076533E+03</td>
<td>1.52</td>
<td>6.16314E-01</td>
<td>-4.625929E-15</td>
</tr>
<tr>
<td>1</td>
<td>5.721455E+03</td>
<td>1.12</td>
<td>4.893855E-01</td>
<td>-6.604279E-09</td>
</tr>
<tr>
<td>2</td>
<td>4.220391E+03</td>
<td>1.01</td>
<td>5.848357E-01</td>
<td>-1.000323E-08</td>
</tr>
<tr>
<td>3</td>
<td>3.996396E+03</td>
<td>9.769804E+03</td>
<td>5.909816E-01</td>
<td>9.383017E-09</td>
</tr>
<tr>
<td>4</td>
<td>1.022207E+04</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Total Strain Energy**

---

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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.
   1. ESE(THRESH=.99)=ALL

2. Search the .f06 file for the initial design’s
   1. ELEMENT STRAIN ENERGIES

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

4. Search the .f06 for the
   1. SUMMARY OF DESIGN CYCLE HISTORY

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.

<table>
<thead>
<tr>
<th>ELEMENT-TYPE</th>
<th>HEXA</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUBCASE</td>
<td>1</td>
</tr>
<tr>
<td>TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM</td>
<td>9.111034E+03</td>
</tr>
<tr>
<td>TOTAL ENERGY OF ALL ELEMENTS IN SET</td>
<td>-1</td>
</tr>
<tr>
<td>-</td>
<td>9.111034E+03</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ELEMENT-ID</th>
<th>STRAIN-ENERGY</th>
<th>PERCENT OF TOTAL</th>
<th>STRAIN-ENERGY-DENSITY</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>8.05918E+02</td>
<td>8.8455</td>
<td>6.447318E+03</td>
</tr>
<tr>
<td>32</td>
<td>8.05918E+02</td>
<td>8.8455</td>
<td>6.447318E+03</td>
</tr>
<tr>
<td>33</td>
<td>8.05918E+02</td>
<td>8.8455</td>
<td>6.447318E+03</td>
</tr>
<tr>
<td>40</td>
<td>8.05918E+02</td>
<td>8.8455</td>
<td>6.447318E+03</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TYPE</th>
<th>HEXA</th>
<th>SUBTOTAL</th>
<th>9.111034E+03</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>100.0000</td>
</tr>
</tbody>
</table>

**Summary of Design Cycle History**

- Number of Finite Element Analyses Completed: 86
- Number of Optimizations W.R.T. Approximate Models: 55

<table>
<thead>
<tr>
<th>CYCLE NUMBER</th>
<th>OBJECTIVE FROM APPROXIMATE OPTIMIZATION</th>
<th>OBJECTIVE FROM EXACT ANALYSIS</th>
<th>FRACTIONAL ERROR OF APPROXIMATION</th>
<th>MAXIMUM VALUE OF CONSTRAINT</th>
</tr>
</thead>
<tbody>
<tr>
<td>INITIAL</td>
<td>1.022207E+04</td>
<td>-4.625929E-15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>5.076533E+03</td>
<td>1.523096E+04</td>
<td>-6.163140E-01</td>
<td>9.999972E-09</td>
</tr>
<tr>
<td>2</td>
<td>8.721545E+03</td>
<td>1.120504E+04</td>
<td>-4.893855E-01</td>
<td>6.694279E-09</td>
</tr>
<tr>
<td>3</td>
<td>4.220301E+03</td>
<td>1.016383E+04</td>
<td>-5.848387E-01</td>
<td>1.000032E-08</td>
</tr>
<tr>
<td>4</td>
<td>3.596596E+03</td>
<td>9.769804E+03</td>
<td>-5.905815E-01</td>
<td>9.338010E-09</td>
</tr>
</tbody>
</table>

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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 normalized density greater than the threshold'.
- Recall from before:
  - 0 - Normalized density values close to 0 are not critical to the design.
  - 1 - Normalized density values close to 1 are critical to the design.

The normalized densities are plotted for each element. Note that all the elements are present.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Topology Optimization Workflows
There are 2 common optimization problem statements for topology optimization

<table>
<thead>
<tr>
<th>METHOD A</th>
<th>METHOD B</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Objective:</strong></td>
<td><strong>Objective:</strong></td>
</tr>
<tr>
<td>◦ Minimize Compliance</td>
<td>◦ Minimize FRMASS</td>
</tr>
<tr>
<td><strong>Constraint:</strong></td>
<td><strong>Constraint:</strong></td>
</tr>
<tr>
<td>◦ FRMASS &lt; Upper Bound</td>
<td>◦ Von Mises Stress &lt; Upper Bound</td>
</tr>
<tr>
<td><strong>Comments:</strong></td>
<td><strong>Comments:</strong></td>
</tr>
<tr>
<td>◦ Multiple optimizations at different bounds for FRMASS are necessary. The best solution is selected from the multiple optimizations.</td>
<td></td>
</tr>
</tbody>
</table>
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

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Traditional Topology Optimization

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

Original Design

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 lightweight design

- Max von Misses: 250 MPa
  - Max Displacement: 3.57 mm
  - 1st natural Frequency: 109 Hz

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Latest Topology Optimization

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

Original Design → Topology Solution → Refined Design → Verification

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
Topology Viewer
The Topology Viewer’s purpose is to only display topology results. Given this, the amount of functionality is streamlined and limited to the most critical operations.

1. The threshold can be modified in 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

2. 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

3. This icon allows the model to be re-centered in the view.

4. This icon changes the background color between black and white.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
# Topology Viewer

## Supported Capabilities

### Supported Element Types
- CTRIA3
- CTRIA6
- CTRIAR
- CQUAD4
- CQUAD8
- CQUADR
- CQUAD4
- CQUAD8
- CQUADR
- CHEXA
- CTETRA
- CPENTA

- All other elements are **not** supported

### Coordinate Systems Supported
- Only the basic coordinate system (CID=0) is supported for GRIDs. This is a rectangular Cartesian system and is also known as the default coordinate system.
- All other coordinate systems are **not** supported. This includes cylindrical, spherical and other cartesian systems (CID=1, 2, 3...).

### 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 Topology 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.