Workshop - MSC Nastran Topology Optimization
Mirror Symmetry Constraints

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

Before Optimization
  ◦ Mass: 67.

After Optimization
  ◦ Mass: 27.8 (~60% mass reduction)
  ◦ Mirror Symmetry Constraints
Details of the structural model
Optimization Problem Statement

Design Region/Variables

x1: PSHELL 1

Restrictions:
- Mirror Symmetry Constraints
  - Symmetry about the ZX plane of coordinate system 1

Design Objective

r0: Minimize compliance

Design Constraints

r1: Fractional mass

r1 < .4 (60% mass reduction)
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
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

Mirror Symmetry Constraints - The Topology Optimization solution must be symmetric, constraints may be imposed to achieve this.
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.

MSC Nastran SOL 200 Web App

Select an optimization type to begin

• Size and Topometry
• Topology
• Multi Model

Tutorials are available in the User's Guide

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

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 topoug1a.dat
2. Click 2. 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. Click + Options
3. Mark the checkboxes for the following:
   1. Symmetry Constraint Columns
4. Set the following for the design region
   1. Use Symmetry Constraints: Yes
   2. Coordinate System ID: 1
   3. Symmetry Planes: ZX

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.

The plane of symmetry is also defined.
Create Design Objective

1. Click on Objective
2. The objective with label r0 is automatically created. The objective is to minimize the Compliance.

- The objective is to maximize the stiffness. This is achieved by programing an objective to minimize the compliance. The compliance is twice the total strain energy. Refer the appendix for additional details on compliance.

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

1. Click Constraints
2. The constraint r1 is automatically created
3. Configure the following for r1
   1. Upper Allowed Limit: .4
      (Retain 40% of the material / 60% mass reduction)

• The FRMASS defines the amount of mass to keep in the final design. In this example, the FRMASS should be less than .4. The problem is then posed in this way: With 40% of the original mass, determine the stiffest design.
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.
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
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.
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 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.
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.

Nastran SOL 200 questions? Email me: christian@the-engineering-lab.com
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 .3 are displayed. The threshold can be modified.
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
Results

Before Optimization

After Optimization
- Mass: 27.8 (~60% mass reduction)
- Mirror Symmetry Constraints

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

\[
\frac{p_i}{p_0} = \frac{p_{ai}}{p_0}
\]

- \( p_{ai} \): 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)
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.

\[
FRMASS = \sum p_i \cdot p_0 \cdot v_i
\]

- \(p_0\): 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

<table>
<thead>
<tr>
<th>ELEMENT-ID</th>
<th>STRAIN-ENERGY</th>
<th>PERCENT OF TC</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>8.0591E+02</td>
<td>8.9455</td>
</tr>
<tr>
<td>32</td>
<td>8.0591E+02</td>
<td>8.9455</td>
</tr>
<tr>
<td>33</td>
<td>8.0591E+02</td>
<td>8.9455</td>
</tr>
<tr>
<td>40</td>
<td>8.0591E+02</td>
<td>8.9455</td>
</tr>
</tbody>
</table>

**Total Strain Energy**

**Compliance**

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.
   - \texttt{ESE(THRESH=.99)=ALL}

2. Search the .f06 file for the initial design's
   - \texttt{ELEMENT STRAIN ENERGIES}

3. Note the value of \texttt{TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM}.

4. Search the .f06 file for the
   - \texttt{SUMMARY OF DESIGN CYCLE HISTORY}

5. Note the value for \texttt{OBJECTIVE FROM EXACT ANALYSIS} for the initial cycle number.

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

**Summary of Design Cycle History**

- **NUMBER OF FINITE ELEMENT ANALYSES COMPLETED**: 86
- **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>1.822207E+04</td>
<td>9.11034E+03</td>
<td>2.6417E-01</td>
<td>-4.0.25929E-15</td>
</tr>
<tr>
<td>1</td>
<td>5.07653E+03</td>
<td>1.92308E+04</td>
<td>-6.16314E-01</td>
<td>9.999972E-09</td>
</tr>
<tr>
<td>2</td>
<td>8.7214S5E+03</td>
<td>1.12050E+04</td>
<td>-8.93375E-01</td>
<td>6.60427E-09</td>
</tr>
<tr>
<td>3</td>
<td>4.22080E+03</td>
<td>1.01658E+04</td>
<td>-5.84857E-01</td>
<td>1.000032E-08</td>
</tr>
<tr>
<td>4</td>
<td>3.99689E+03</td>
<td>9.76989E+03</td>
<td>-5.90918E-01</td>
<td>9.383010E-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

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

METHOD A

Objective:
◦ Minimize Compliance

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

Mass: 9.737 grams

Original Design

Max von Misses: 150 MPa
Max Displacement: 2.78 mm
1st natural Frequency: 111 Hz

FRMASS < .75
Mass: 7.186 g
Optimization B

Mass: 7.739 g

Mass: 7.739 g
Traditional Topology Optimization

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

Topology Solution          Refined Design          Verification

<table>
<thead>
<tr>
<th>Original Design</th>
<th>Mass: 9.737 grams</th>
</tr>
</thead>
<tbody>
<tr>
<td>FRMASS &lt; .9</td>
<td>Mass: 8.756 g</td>
</tr>
<tr>
<td>Optimization A</td>
<td></td>
</tr>
<tr>
<td>FRMASS &lt; .75</td>
<td>Mass: 7.186 g</td>
</tr>
<tr>
<td>Optimization B</td>
<td></td>
</tr>
<tr>
<td>FRMASS &lt; .6</td>
<td>Mass: 5.718 g</td>
</tr>
<tr>
<td>Optimization C</td>
<td></td>
</tr>
</tbody>
</table>

Mass: 9.094 g
| 1st natural Frequency: 114 Hz |
Max von Misses: 150 MPa
Max Displacement: 2.52 mm

Mass: 7.739 g
| 1st natural Frequency: 111 Hz |
Max von Misses: 150 MPa
Max Displacement: 2.78 mm

Mass: 6.119 g
| 1st natural Frequency: 109 Hz |
Max von Misses: 250 MPa
Max Displacement: 3.57 mm

Best Solution: Optimization B led to a valid and lightweight design
Latest Topology Optimization
Objective: Minimize Fractional Mass (Minimize Mass)
Constraint: Stress Constraint
Topology Viewer
Review Optimization Results

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 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

<table>
<thead>
<tr>
<th>Supported Element Types</th>
<th>Coordinate Systems Supported</th>
</tr>
</thead>
<tbody>
<tr>
<td>◦ CTRIA3</td>
<td>◦ 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.</td>
</tr>
<tr>
<td>◦ CTRIA6</td>
<td>◦ All other coordinate systems are not supported. This includes cylindrical, spherical and other cartesian systems (CID=1, 2, 3...).</td>
</tr>
<tr>
<td>◦ CTRIAR</td>
<td></td>
</tr>
<tr>
<td>◦ CQUAD4</td>
<td></td>
</tr>
<tr>
<td>◦ CQUAD8</td>
<td></td>
</tr>
<tr>
<td>◦ CQUADR</td>
<td></td>
</tr>
<tr>
<td>◦ CQUAD4</td>
<td></td>
</tr>
<tr>
<td>◦ CQUAD8</td>
<td></td>
</tr>
<tr>
<td>◦ CQUADR</td>
<td></td>
</tr>
<tr>
<td>◦ CHEXA</td>
<td></td>
</tr>
<tr>
<td>◦ CTETRA</td>
<td></td>
</tr>
<tr>
<td>◦ CPENTA</td>
<td></td>
</tr>
<tr>
<td>◦ All other elements are <strong>not</strong> supported</td>
<td></td>
</tr>
</tbody>
</table>

### 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.