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

AN MSC NASTRAN SOL 200 TUTORIAL
Before Starting

This example requires MSC Nastran 2017 or newer.
Goal: Use Nastran SOL 200 Optimization

Before Optimization
- Mass: 18 kg

After Optimization
- Mass: 9.3 kg
- Prevent excessive stress and displacements
Goal: Use Nastran SOL 200 Optimization

1) Initial Design
2) Proposed Topology Solution
3) Final Design

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

- Units: m, N, MPa
- Material:
  - E: 200E9 Pa
  - v: .3
  - p: 7850 kg/m^3

Two SUBCASEs

Constrained

Force
Optimization Problem Statement

Design Region/Variables

x1: PSOLID 1

Design Objective

r0: Minimize fractional mass (FRMASS)

Design Constraints

r1: Z displacement at node 5622 (GRID 5622)
   \[ r1 < 0.0008 \text{ m} \]

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

Maximum Allowable Stress - A stress constraint cannot be created normally as is done in Size or Topometry optimization. A special option is available to specify a stress constraint for Topology Optimization.
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.
MSC Nastran SOL 200 Web App

Select a web app to begin

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.
Upload BDF Files

1. Click 1. Select Files and select topstr2.dat
2. Click Upload Files

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

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

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

- 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 PSOLID 1 has 500 elements associated and is configured as a design region, then there will be 500 design variables created.
- Once the stress constraint is specified, it applies for all design regions. The von Mises stress is constrained in this example. It should be noted that the stress of each element is not constrained. Instead, a single equivalent stress for the entire model and subcase is constrained.

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

1. Click on Objective
2. Click the x icon to delete the existing objective of compliance

- When the Topology web app is first opened, the objective is automatically set to minimizing the compliance. Since this tutorial’s objective is to minimize the fractional mass, the COMP objective must be removed before setting the FRMASS response as the objective.
Create Design Objective

1. Search for FRMASS
2. Click the plus (+) icon
3. The objective for FRMASS is added to the table, no further edit is necessary

• In Size Optimization, usually Weight is set as the objective. Topology optimization has special requirements, including using FRMASS instead of Weight to minimize the mass.
Create Design Constraints

1. Click Constraints

2. Click the x icon to delete the existing constraint of Fractional Mass (FRMASS)

- When the Topology Web App is first opened, the objective is automatically set to minimize COMP (compliance) and the constraint FRMASS (mass) is created. This tutorial is doing a FRMASS minimization with a stress constraint.
- The previous COMP objective was replaced with an FRMASS objective.
- On this page, the FRMASS constraint is removed.
Create Design Constraints

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

Topology optimization works best when working with a small number of responses, e.g. Compliance, Fractional Mass, a single von Misses stress.

In this tutorial a single displacement is constrained. The number of constraints should be kept to a minimum. For example, constraining multiple displacements at various nodes is not advised.

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

1. Click Subcases
2. Click Check visible boxes

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

1. Click Settings
2. Set DESMAX to 100

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

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

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.

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

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

Before Optimization
- Mass: 18 kg

After Optimization
- Mass: 9.3 kg
- Prevent yield and excessive displacement
End of Tutorial
Appendix
Appendix Contents

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

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

$p_{ci}$ : The current material density of element $i$

$p_0$ : The original material density

$p_i$ : The normalized material density of element $i$

Consider one element in the model.

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

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

\[ p_{ci} = p_i \cdot p_0 \]
\[ = 1.0 \cdot 8000 \]
\[ = 8000 \]

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

\[ p_{ci} = p_i \cdot p_0 \]
\[ = .5 \cdot 8000 \]
\[ = 4000 \]
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

- 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.

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

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

- \(p_0\): The original 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.

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

---

**Summary of Design Cycle History**

(HARD CONVERGENCE ACHIEVED)

<table>
<thead>
<tr>
<th>CYCLE</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>5.076533E+03</td>
<td>6.163140E-01</td>
<td>-4.625929E-15</td>
</tr>
<tr>
<td>1</td>
<td>5.721555E+03</td>
<td>1.121513E+04</td>
<td>4.893855E-01</td>
<td>6.604279E-09</td>
</tr>
<tr>
<td>2</td>
<td>4.220301E+03</td>
<td>1.016388E+04</td>
<td>8.848357E-01</td>
<td>1.000326E-08</td>
</tr>
<tr>
<td>3</td>
<td>3.996596E+03</td>
<td>9.769040E+03</td>
<td>5.209815E-01</td>
<td>9.283010E-09</td>
</tr>
</tbody>
</table>

---

**Element Strain Energies**

<table>
<thead>
<tr>
<th>ELEMENT-ID</th>
<th>STRAIN-ENERGY</th>
<th>PERCENT OF TC</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>8.059148E+02</td>
<td>8.5955</td>
</tr>
<tr>
<td>32</td>
<td>8.059148E+02</td>
<td>8.6955</td>
</tr>
<tr>
<td>33</td>
<td>8.059148E+02</td>
<td>8.6955</td>
</tr>
<tr>
<td>40</td>
<td>8.059148E+02</td>
<td>8.6955</td>
</tr>
</tbody>
</table>

**Total Strain Energy**

TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM = 9.111034E+03

TOTAL ENERGY OF ALL ELEMENTS IN SET  = 9.111034E+03

---

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 1.822E4 is twice the TOTAL STRAIN ENERGY of 9.11E3.

---

### 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

**METHOD A**

Objective:
- Minimize Compliance

Constraint:
- FRMASS < Upper Bound

Comments:
- Multiple optimizations at different bounds for FRMASS are recommended. 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 < 0.75 (Target Mass)

Original Design

Mass: 9.737 grams
FRMASS < .75
Mass: 7.186 g
Optimization B

Mass: 7.739 g

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

Objective: Minimize Compliance (Maximize Stiffness)
Constraint: Fractional Mass < 0.6 (Target Mass)

Original Design
Mass: 9.737 grams

Topology Solution
- FRMASS < 0.9
- Mass: 8.756 g
- Optimization A

- FRMASS < 0.75
- Mass: 7.186 g
- Optimization B

- FRMASS < 0.6
- Mass: 5.718 g
- Optimization C

Refined Design
- Mass: 9.094 g

Verification
- Max von Misses: 150 MPa
- Max Displacement: 2.52 mm
- 1st natural Frequency: 114 Hz

- Mass: 7.739 g

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

- Mass: 6.119 g

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

Best Solution: Optimization B led to a valid and lightweight design

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
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 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.
# 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.