

SIEMENS

NX Nastran Topology Optimization in Simcenter

Proprietary & Restricted Rights Notice

© 2016 Siemens Product Lifecycle Management Software Inc. All Rights Reserved. This software and related documentation are proprietary to Siemens Product Lifecycle Management Software Inc.

NASTRAN is a registered trademark of the National Aeronautics and Space Administration. NX Nastran is an enhanced proprietary version developed and maintained by Siemens Product Lifecycle Management Software Inc.

MSC is a registered trademark of MSC Software Corporation. MSC Nastran and MSC Patran are trademarks of MSC Software Corporation.

All other trademarks are the property of their respective owners.

TAUCS Copyright and License

TAUCS Version 2.0, November 29, 2001. Copyright (c) 2001, 2002, 2003 by Sivan Toledo, Tel-Aviv University, stoledo@tau.ac.il. All Rights Reserved.

TAUCS License:

Your use or distribution of TAUCS or any derivative code implies that you agree to this License.

THIS MATERIAL IS PROVIDED AS IS, WITH ABSOLUTELY NO WARRANTY EXPRESSED OR IMPLIED. ANY USE IS AT YOUR OWN RISK.

Permission is hereby granted to use or copy this program, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any derivative code must cite the Copyright, this License, the Availability note, and "Used by permission." If this code or any derivative code is accessible from within MATLAB, then typing "help taucs" must cite the Copyright, and "type taucs" must also cite this License and the Availability note. Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. This software is provided to you free of charge.

Availability (TAUCS)

As of version 2.1, we distribute the code in 4 formats: zip and tarred-gzipped (tgz), with or without binaries for external libraries. The bundled external libraries should allow you to build the test programs on Linux, Windows, and Mac OS X without installing additional software. We recommend that you download the full distributions, and then perhaps replace the bundled libraries by higher performance ones (e.g., with a BLAS library that is specifically optimized for your machine). If you want to conserve bandwidth and you want to install the required libraries yourself, download the lean distributions. The zip and tgz files are identical, except that on Linux, Unix, and Mac OS, unpacking the tgz file ensures that the configure script is marked as executable (unpack with `tar xvpf`), otherwise you will have to change its permissions manually.

Overview

You can use Simcenter to define NX Nastran topology optimization and post-process the results. This feature has its own inputs and license agreement. It works with a single overall objective (such as minimizing mass) as well as one or more constraints (such as limiting displacements at specific grids).

This feature has the following prerequisites:

- The Simulation file must be the displayed and work part, and the model must be meshed.
- The active solution must be a SOL 101 Linear Statics or SOL 103 Real Eigenvalues solution.
- The model must include at least one load and one constraint.

If you plan to use displacement constraints, you must apply a force at all grid points where the displacement constraints are applied. The force direction does not matter. The exact force magnitude is not critical, but it should be of a similar order to your actual applied loads to prevent any numerical or convergence problems.

You should use gravity loads only in connection with a compliance objective or compliance constraint. You can use moment loads only in connection with compliance constraints.

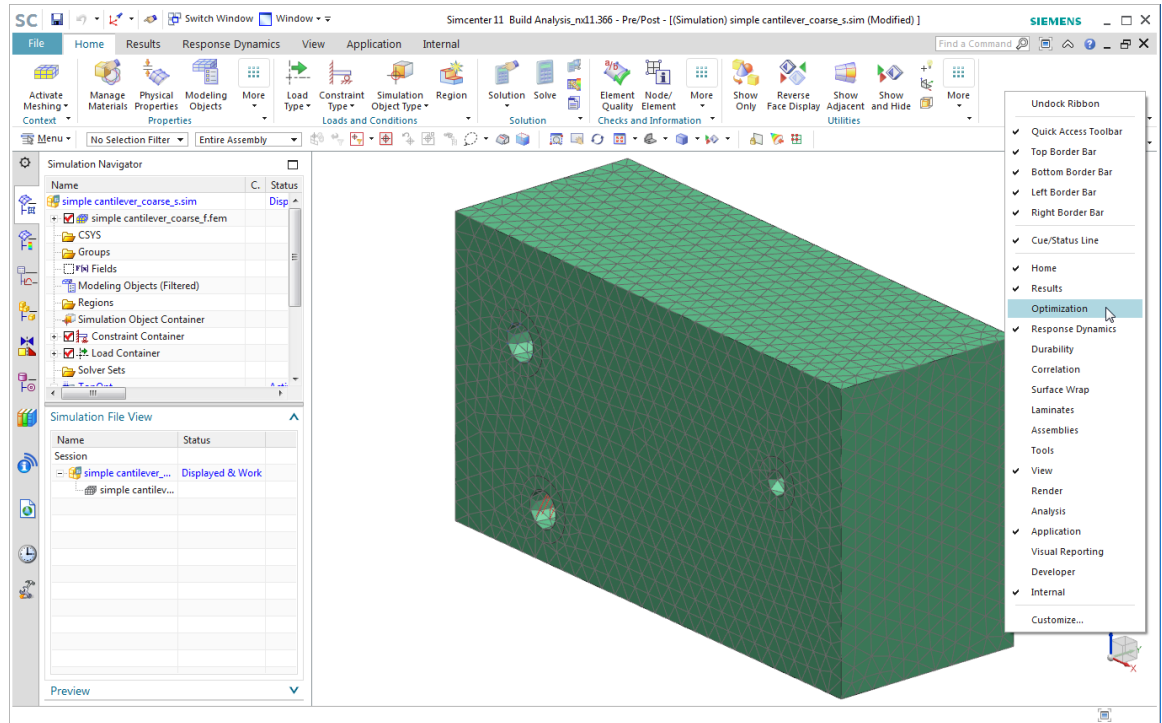
- All design elements must have the same density defined on their respective MATi entries.

For information about the types of elements that you can use as design variables, as well as the parameter descriptions see NX Nastran Topology Optimization in the *NX Nastran 11 Release Guide*.

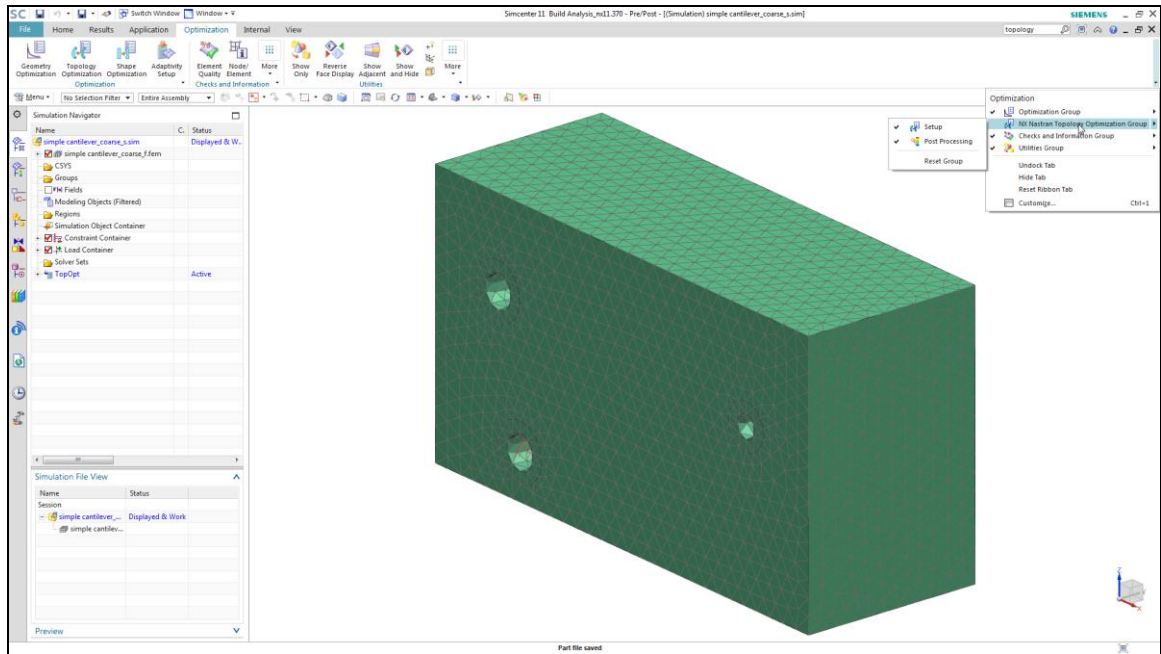
Displaying the NX Nastran Topology Optimization group

By default, the NX Nastran Topology Optimization commands are not displayed. To display them:

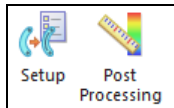
1. If the **Optimization** tab does not appear when the Simulation file is open, right-click the ribbon and select **Optimization**.



2. In the **Optimization** tab, click the arrow on the far right side of the ribbon, and then select **NX Nastran Topology Optimization Group**.



The **NX Nastran Topology Optimization** group appears in the **Optimization** tab:



Defining the topology optimization

To define the topology optimization, you must:

- Specify the elements to include in the optimization.

You can optionally group the elements into design variable sets. By specifying a different initial fill value for each set, you can identify elements whose density you know will be reduced (such as the corners of a block), thereby speeding the computations.

- Specify your design objective.

For a SOL 101 Linear Statics solution, you can minimize or maximize the mass or compliance. For a SOL 103 Real Eigenvalues solution, you can minimize or maximize the mass or eigenvalue.

- Specify the design constraints.

For a SOL 101 Linear Statics mass objective, you can specify either displacement or compliance constraints. For a compliance objective, you can specify only a mass constraint.

For a SOL 103 Real Eigenvalues mass objective, you can specify only an eigenvalue constraint. For an eigenvalue objective, you can specify only a mass constraint.

You can optionally group displacement or eigenvalue constraints into sets so that you can specify a different constraint bound for each set.

After you have completed these tasks, you can optionally modify the default parameters and outputs. The output can consist of a listing window, text file, text modeling object, or any combination of those.

When you initially define your topology optimization, the **NX Nastran 11 Topology Optimization** dialog box guides you through these tasks step by step. When editing a topology optimization definition, you can immediately go to specific pages on that dialog box to make any changes. In all cases, your choices are limited based on your previous selections. For example, the options on the **Constraints** page are determined by your selections on the **Objective** page.

In addition to defining your design variables, objective, and constraints using the **NX Nastran 11 Topology Optimization** dialog box, you must also edit the solution as indicated in the following table.

	SOL 101 Linear Statics	SOL 103 Real Eigenvalues
Enable the DISPLACEMENT and GPFORCE output requests	Required	Optional
Enable the ESE output request and set the threshold to 0.0	Required	Required
Enable the EKE output request and set the threshold to 0.0	Optional	Required
Embed the topology optimization output into the NX Nastran input file	Required	Required

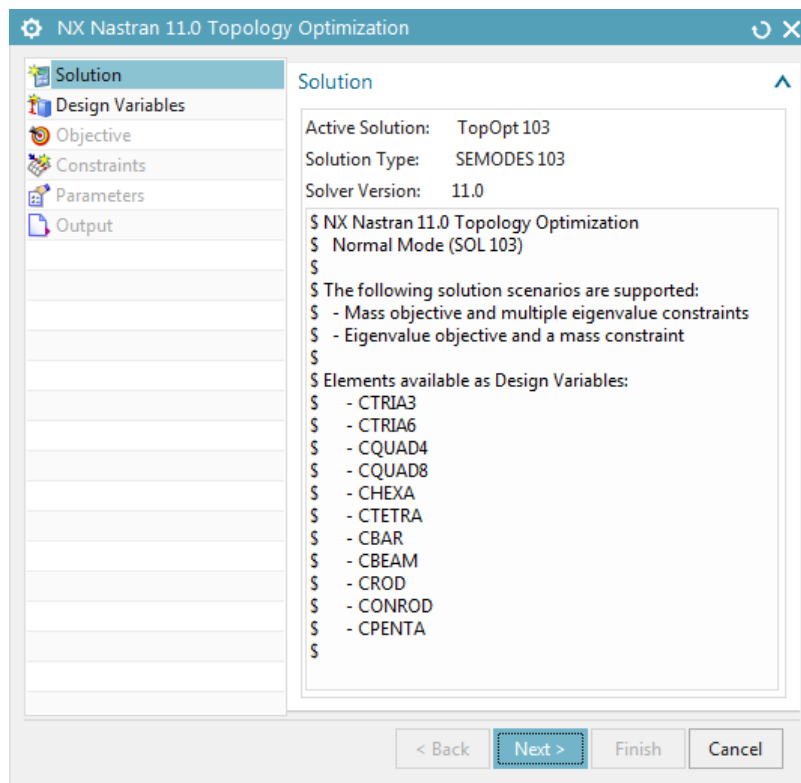
Set up NX Nastran topology optimization

1. Choose **Optimization** tab→**NX Nastran Topology Optimization** group→

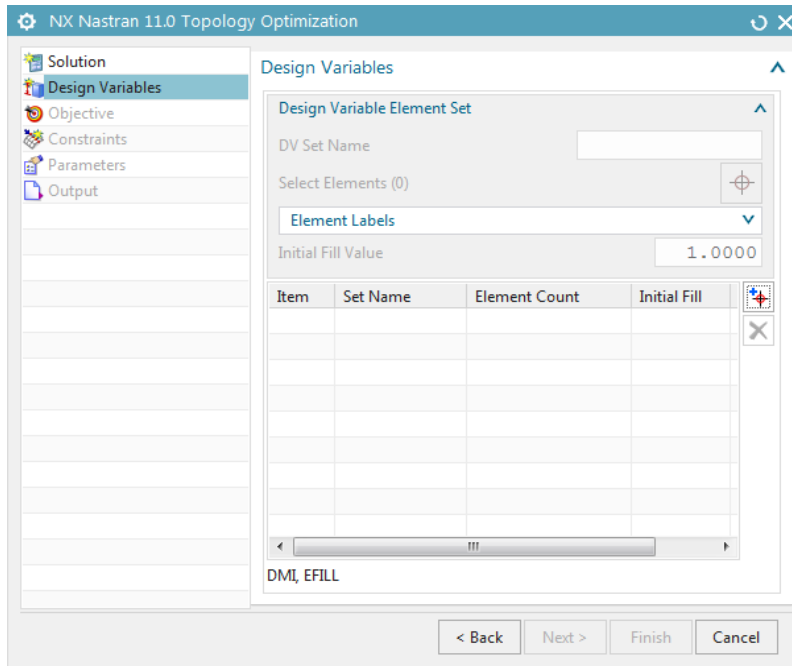
Setup 

The **Solution** page displays the name of the active solution, the solution type, and the solver version number.

Tip: To view all of the information about the supported scenarios and elements, enlarge the dialog box until the scroll bars disappear. You might have to enlarge the dialog box to view all of the information on other pages as well.



2. Click **Next**.



3. Click **Add New Design Variable Set** .




4. (Optional) In the **DV Set Name** box, type a name for this set.

5. Click **Select Elements** .

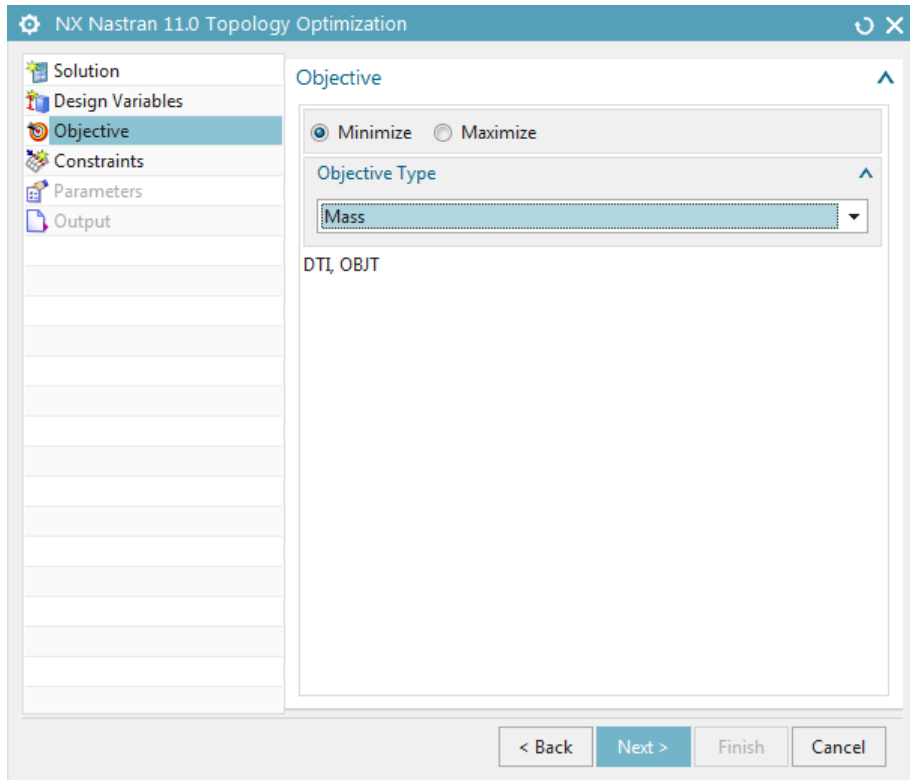
6. In the graphics window, select the elements to include in the topology optimization.

Alternatively, you can specify elements by their label name. You can also define an initial fill value between 1.0 and 0.0001.

Tips:

- In the top menu bar, you can click **Select All**  to select all of the elements, or you can select elements to omit and then click **All But Selected** . If those options do not appear in the top menu bar, click the arrow on the far right side of the bar, select **Selection Group**, and then select the commands that you want to display.
- To create multiple sets of design variables, repeat steps 3–6. If you create multiple design variable sets, each selected element must be in only one set. To delete an element set, select the set in the list and then click **Remove** .

7. Click **Next**.



8. In the **Objective** group, click **Minimize** or **Maximize** to either minimize or maximize your selected objective type.
9. In the **Objective Type** list, select your optimization objective.

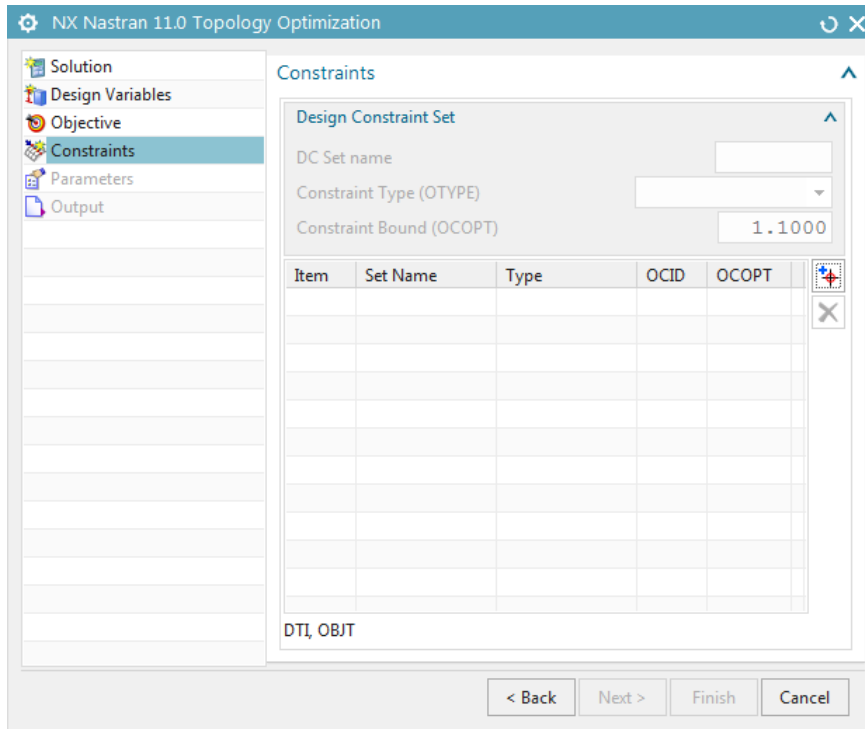
For a SOL 101 Linear Statics solution, you can select **Mass** or **Compliance**. If you select **Compliance**, select the appropriate option from the **Compliance Load Type** list (**Force/Moment** or **Gravity**).

For a SOL 103 Real Eigenvalues solution, you can select **Mass** or **Eigenvalue**. If you select **Eigenvalue**, type the appropriate value in the **Eigenvalue Number** box.

Note: The displacement objective is not available in the Simcenter **Objective Type** list. You must edit the NX Nastran input file to create a displacement objective. See NX Nastran Topology Optimization in the *NX Nastran 11 Release Guide* for the objective input description.

In addition, if you change the objective after you created constraints, you must manually remove any constraints that use an unsupported constraint type for the new objective.

10. Click **Next**.



11. Click **Add New Constraint Set** .

12. (Optional) In the **DC Set Name** box, type a name for this set.

Note: You can create multiple sets for displacement and eigenvalue constraints. Do not create multiple sets for mass or compliance constraints.

13. In the **Constraint Type (OTYPE)** list, select the appropriate option.


For a SOL 101 Linear Statics solution with a mass objective, select **Displacement** or **Compliance**. If you select **Displacement**, select the nodes to constrain in the graphics window or specify the node label. If you select **Compliance**, select the appropriate option from the **Compliance Load Type (OCID)** list (**Force/Moments** or **Gravity**).

For a SOL 101 Linear Statics solution with a compliance objective, you can select only **Mass**.

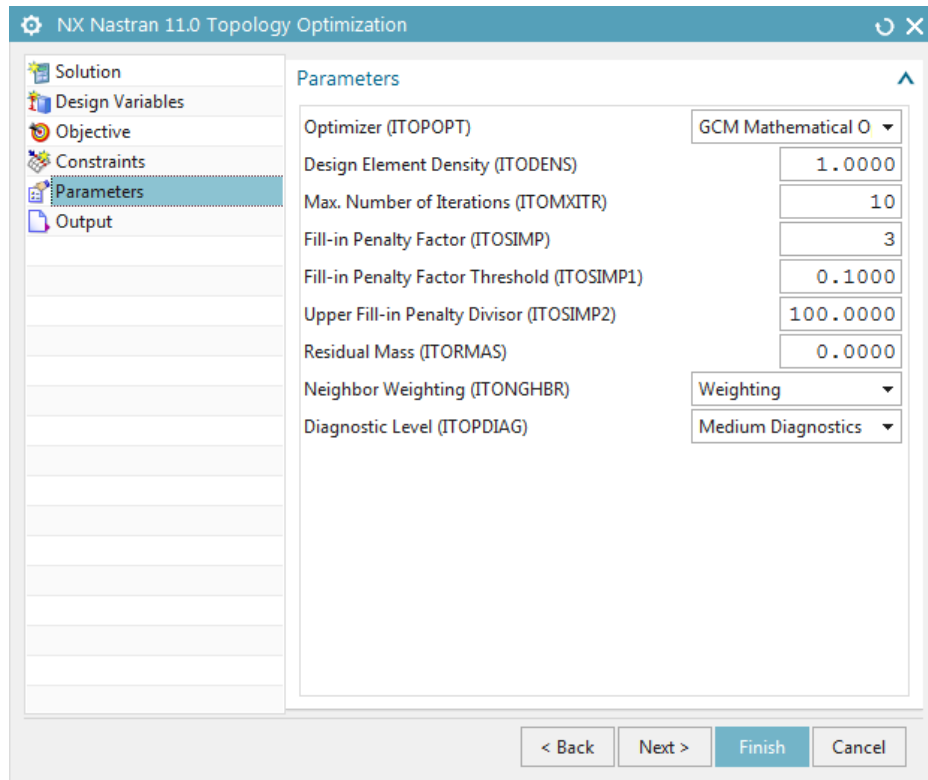
For a SOL 103 Real Eigenvalues solution with a mass objective, you can select only **Eigenvalue**. For an eigenvalue objective, you can select only **Mass**.

- For a SOL 101 Linear Statics solution, type the appropriate value in the **Constraint Bound (OCOPT)** box.

For a SOL 103 Real Eigenvalues solution, type the appropriate values in the **Mode Number (OCID)** and **Constraint Bound (OCOPT)** boxes. For a gravity load, set **Mode Number (OCID)** to -1.

Tip: To create multiple sets of design constraints, repeat steps 11–14. To delete a constraint set, select the set in the list and then click **Remove** . If you change the objective, you must manually remove any constraint sets that use an unsupported constraint type for the new objective.

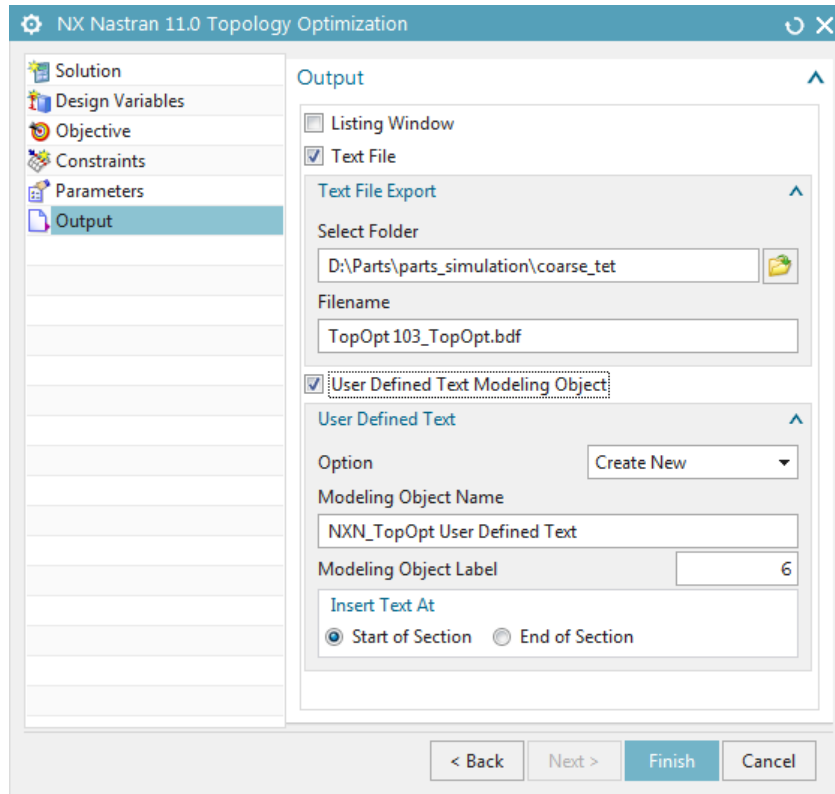
- To use the default parameters and create a .bdf file in the same folder as your Simulation file, click **Finish**. Otherwise, click **Next** and go to the next step.



- On the **Parameters** page, select or type the value to use for each parameter.

For information about these parameters, see NX Nastran Topology Optimization in the *NX Nastran 11 Release Guide*.

17. Click **Next**.



18. Select the check box for each output type that you want to create (**Listing Window**, **Text File**, **User Defined Text Modeling Object**, or any combination of those). You should select at least one.

If you select **Text File**, you must specify the folder and .bdf file name.


If you select **User Defined Text Modeling Object**, you must specify whether you want to create a new object, overwrite an existing one, or append data to an existing one. If you create a new object, you must specify its name and label. If you overwrite or append an existing object, you must select the existing object. In all cases, you must indicate whether the text should be inserted at the start or end of the section.

Note: Before you solve, you must add optimization output to the NX Nastran input file through either a text file or modeling object.

19. Click **Finish**.

If you requested a listing, it appears in the **Information** window. If you created a text file output, it is created in the specified location.

Modify the solution

1. In the **Simulation Navigator**, right-click the solution and then choose **Edit**.
2. In the list in the **Solution** dialog box, select **Case Control**.
3. Click **Edit**  (**Output Requests**).

4. In the list in the **Structural Output Requests** dialog box, select **Displacement**. Then select the **Enable DISPLACEMENT Request** check box.

This is required for a SOL 101 Linear Statics solution and optional for a SOL 103 Real Eigenvalues solution.

5. In the list, select **Grid Point Force**. Then select the **Enable GPFORCE Request** check box.

This is required for a SOL 101 Linear Statics solution and optional for a SOL 103 Real Eigenvalues solution.


If you specified a compliance objective or constraint, select **ALL** in the **Entity** list.

6. In the list, select **Strain Energy**. Then select the **Enable ESE Request** and **Threshold** check boxes, and type **0.0** in the **Threshold** box.
7. In the list, select **Kinetic Energy**. Then select the **Enable EKE Request** and **Threshold** check boxes, and type **0.0** in the **Threshold** box.

This is required for a SOL 103 Real Eigenvalues solution and optional for a SOL 101 Linear Statics solution.

8. Click **OK** in the **Structural Output Requests** dialog box.
9. In the list on the **Solution** dialog box, select **Bulk Data**.

If you created a modeling object when you specified the output for the topology optimization definition, select that object in the **User Defined Text** list.

If you created a text file, click **Create Modeling Object**  (**User Defined Text**). In the **User Defined Text** dialog box, in either the **Text to Insert at Start of Section** or **Text to Insert at End of Section** group, select **Merged** in the **Text From File** list. Then specify the location of the .bdf file.

10. Click **OK** on all dialog boxes.



Post-processing the results

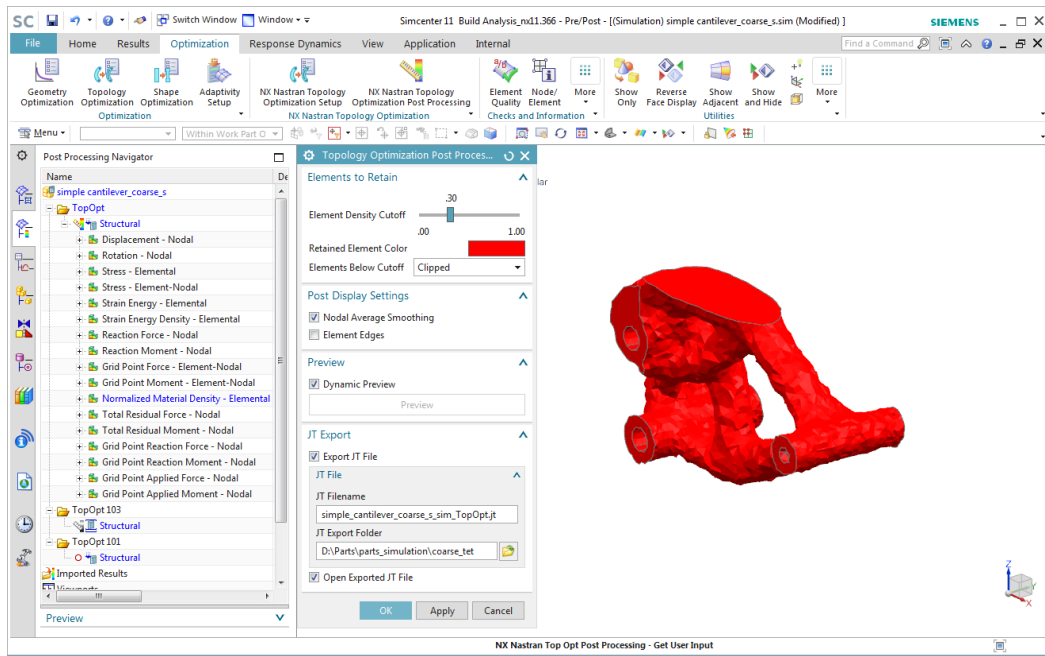
Once NX Nastran has completed the topology optimization solution, you will load the results into the **Post Processing Navigator**. The results will include the **Normalized Material Density** result type. This result type represents element densities between 0 and 1. The closer the value is to 1, the more important the element is to the objective. The optimum element density is displayed automatically. You can examine how increasing or decreasing the element density cutoff modifies the topology.

You can optionally modify the displayed results in the graphics window. For example, you can exclude the elements with values below the density cutoff, or make them translucent. You can also export a JT file that you can use with various CAD systems.

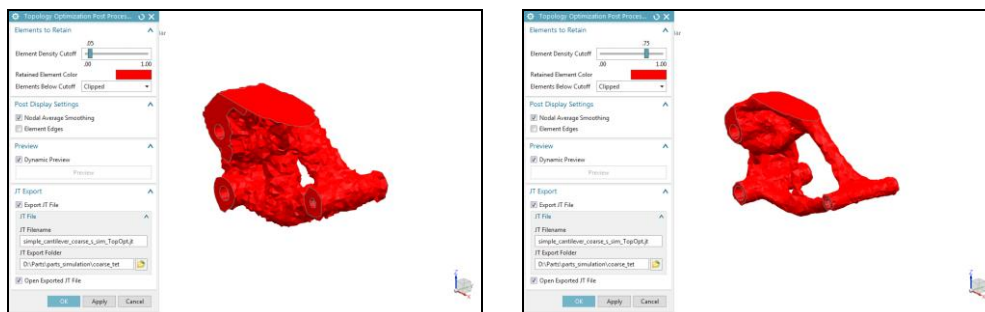
The topology optimization statistical results appear in the .f06 file. They include the number of iterations at which the optimum is reached, the objective and constraint values during the iterations, and a graphical iteration diagnostics about the design variable distribution.

Post-process the results

1. In the **Simulation Navigator**, under the solution node, double-click **Results**.
2. In the **Post Processing Navigator** , expand the solution and **Structural** nodes, and double-click **Normalized Material Density – Elemental**.
3. Choose **Optimization** tab→**NX Nastran Topology Optimization** group→**Post Processing** .

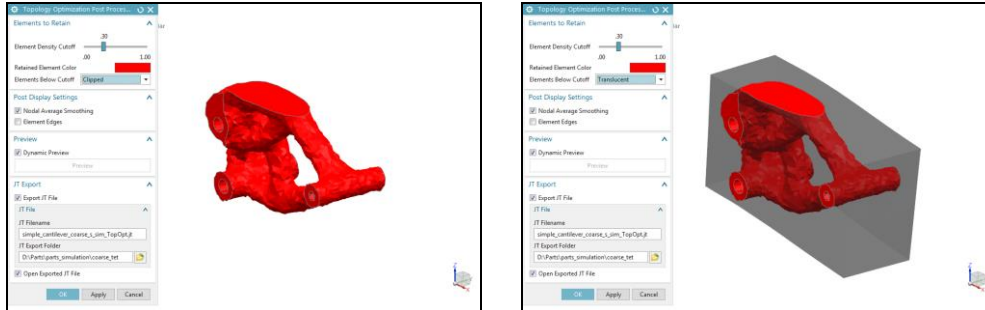


4. To add or remove elements based on their element density, move the **Element Density Cutoff** slider to the left or right.

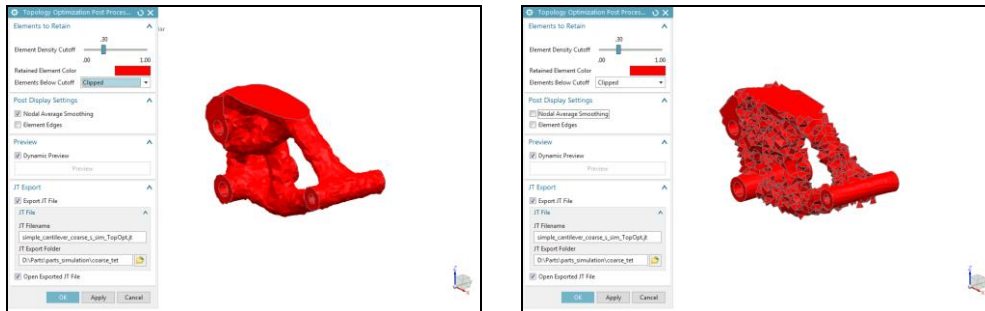


Tip: If the **Dynamic Preview** check box is selected, any changes that you make in the **Topology Optimization Post Processing** dialog box are immediately reflected in the graphics window. If that check box is cleared, you must click **Preview** to see the change.

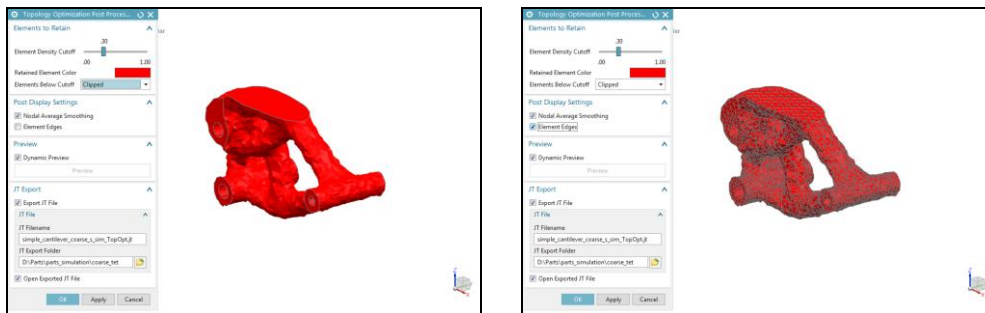
- To change the color of the retained elements, click the **Retained Element Color** box and specify the desired color.
- In the **Elements Below Cutoff** list, select **Clipped** to omit the elements that fall below the element density cutoff threshold, or select **Translucent** to display them translucently.



- To smooth the displayed elements, select the **Nodal Average Smoothing** check box.




- To display of the element edges, select the **Element Edges** check box.



- To export a JT file of the results, select the **Export JT File** check box and then specify the JT file name and folder. You can optionally select the **Open Exported JT File** check box to open the file after you export it.

10. Click **OK**.

Displaying topology optimization results modifies the post view settings. To restore the initial settings, choose **Results** tab→**Return to Model** . You can then access the results again with the initial settings. You can also apply any template that you created.