Structural Optimization Analysis Guide

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

Published in the U.S.A.

Table of Contents

1. Structural Optimization Overview 1
2. Optimization Analysis Workflow 3
2.1. Create Upstream Analysis Systems 3
2.2. Attach Geometry and Launch Mechanical 5
2.3. Specify Analysis Settings 6
2.4. Define Optimization Method and Regions 24
2.5. Determine Objective and Constraints Capabilities 29
2.6. Define Design Objectives 31
2.7. Define Response Constraints 38
2.8. Define Manufacturing Constraints 48
2.9. Define Design Constraints 57
2.10. Specify Results and Solve 61
2.10.1. Topology Density 61
2.10.2. Topology Elemental Density 65
2.10.3. Structural Results 68
2.10.4. Thermal Results 71
2.11. Post Processing 74
2.12. Recreating CAD Geometry 77
2.13. Performing Design Validation 82
2.13.1. Geometry Validation 83
2.13.2. Model Validation 87
3. Design Methods 93
3.1. Shape Processing Workflows 93
3.1.1. Density Based Topology Optimization Shape Processing 94
3.1.2. Level Set Based Topology Optimization Shape Processing 96
3.1.3. Lattice Optimization Shape Processing 100
3.1.4. Shape Optimization Shape Processing 102
3.1.5. Topography Optimization Shape Processing 105
3.2. Recommendations - Optimization in Practice 108
3.2.1. Topology Optimization - Density Based 108
3.2.1.1. Topology Optimization - Density Based Solution Methodology 110
3.2.2. Topology Optimization - Level Set Based 114
3.2.3. Topology Optimization - Mixable Density 119
3.2.4. Lattice Optimization Analysis 121
3.2.5. Shape Optimization Analysis 132
3.2.6. Topography Optimization 135
3.3. Mixing Multiple Methods within an Analysis 137
3.4. Topology Optimization versus Shape Optimization 137
4. Structural Optimization Theory 139
4.1. Manufacturing Constraint Background 139
4.1.1. Member Size Minimum 139
4.1.2. Member Size Maximum 145
4.1.3. Member Size Gap Size 153
4.1.4. Pull Out Direction 157
4.1.5. AM Overhang Constraint 183
4.1.6. Housing 188
4.1.7. Complexity Index 191
5. Bibliography 195

List of Figures

4.1. Printing Angle Impact on Quality of Surface in FDM [15]. 184
4.2. Overhanging Region (red) for Shape Ω Printed in Direction $\vec{d}$. 185

List of Tables

4.1. AM Overhang Constraint Formulation 185
4.2. Setup Parameters and Technical Details 185

Chapter 1: Structural Optimization Overview

In the Ansys Mechanical application™, the **Structural Optimization** analysis is a form-finding analysis driven by mechanical and geometrical criteria. The application obtains mechanical criteria from upstream linear structural analyses (static, modal, harmonic, or thermal). The application also supports criteria from non-linear structural analyses.

Optimization Methods

The application provides the following optimization methods:

- Topology Optimization:
  - **Density Based** : This method performs optimization based on the density of each element of your model. It employs Solid Isotropic Material with Penalization (SIMP) wherein density is forced to approach 0 or 1 rather than varying continuously.
  - **Level Set**: This method directly deals with the boundary of the shape. This enables Mechanical to deliver an unambiguous solution.
  - **Mixable Density**: This optimization type is using the same mathematical background as density-based method but it has been enriched thanks to cross-fertilization with the Level-Set based technology.
- **Lattice**: This method enables you to compute an optimal variable density lattice distribution in your geometry.
- **Shape**: This method enables you to optimize the shape of your model by morphing the mesh. As opposed to topology optimization, no topological change is allowed. This method supports solid models only.
- **Topography**: This method enables you to perform shape optimization using mesh node relocations. This method supports shell models only.

Chapter 2: Optimization Analysis Workflow

The topics listed below describe the steps to create and perform an optimization analysis.

2.1. Create Upstream Analysis Systems

2.2. Attach Geometry and Launch Mechanical

2.3. Specify Analysis Settings

2.4. Define Optimization Method and Regions

2.5. Determine Objective and Constraints Capabilities

2.6. Define Design Objectives

2.7. Define Response Constraints

2.8. Define Manufacturing Constraints

2.9. Define Design Constraints

2.10. Specify Results and Solve

2.11. Post Processing

2.12. Recreating CAD Geometry

2.13. Performing Design Validation

2.1. Create Upstream Analysis Systems

An optimization analysis requires an upstream "feeder" system that provides the loading and/or boundary conditions used to create an optimized part based on the Objective (p. 31) and Response Constraint (p. 38) objects specified in the optimization analysis.

Supported Upstream Systems

The optimization analysis must be linked to (preceded by) one of the following analysis types:

- Harmonic Response
- Modal
- Static Structural/Static Structural Nonlinear
- Steady-State Thermal

- Any combination of Harmonic Response, Modal, Static Structural, and/or Steady-State Thermal.

Note:

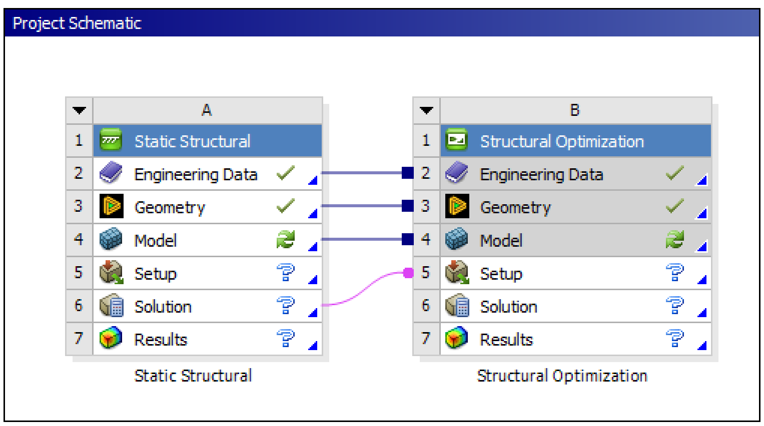
The Structural Optimization analysis supports Condensed Parts from upstream Harmonic Response, Modal, and Static Structural (linear only) analyses.

Procedure

1. Specify your upstream system or systems. Drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**. The cells between the systems should be linked as illustrated below.

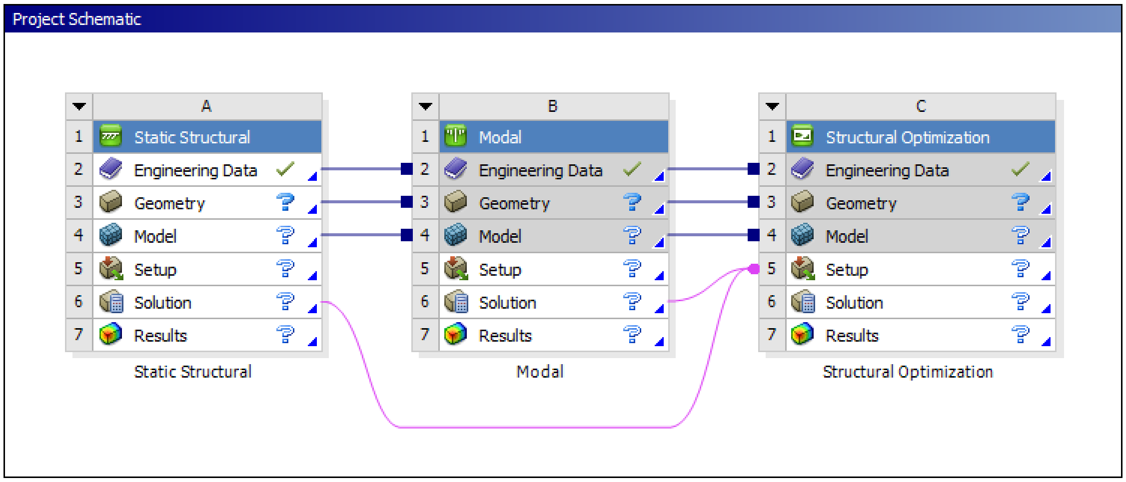
2. Link your systems as illustrated in the examples shown below. The new **Structural Optimization** system shares the **Engineering Data**, **Geometry**, and **Model** cells with the same cells as the upstream systems and the **Solution** cell links to the **Setup** cell of the **Structural Optimization** system.

   Upstream Static Structural System Example



   Multiple Upstream Systems Example

   If necessary, link the **Solution** cell of the **Static Structural** or **Modal** analysis to the **Setup** cell of the **Structural Optimization** system.



Note:

- If your upstream system is a single Static Structural analysis, Ansys recommends that you use step-based loading to improve scalability. In order to do so, you need to define your loading conditions using the **Tabular Data** window and you need to set the **Independent Variable** property to the **Step** option. This does not include the use of the Thermal Condition load.
- When you specify a **Structural Optimization** system from Mechanical, you need to make sure that you connect the systems properly. Connect the systems using the **Transfer Data From** context (right-click) menu option from the **Structural Optimization** environment. Once linked, you can disconnect the systems using the **Unlink Data From** option of the context menu.

Once you link the analyses, automatic property specifications are made in Mechanical to define the relationship between the systems.

2.2. Attach Geometry and Launch Mechanical

The procedure below assumes that you have a supported geometry file type. As needed, review the Attach Geometry/Mesh section for a description of the available methods and supported file types.

Procedure

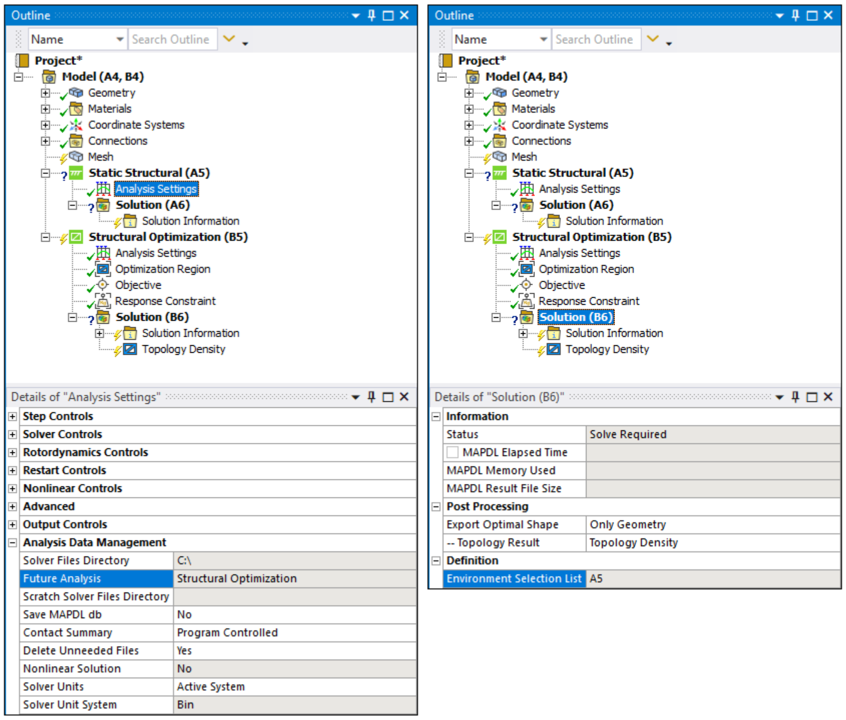
1. Right-click the **Geometry** cell of the (furthest) upstream system, select **Import** > **Browse** and select the desired file for import. Note that double-clicking on the **Geometry** cell opens Discovery Modeling.

Important:

This analysis does not support the presence of a Rigid Body.

2. Once you have your geometry specified, double-click the **Model** cell (or right-click, and select **Edit**) of any of the systems to launch the Mechanical application.

3. Once your geometry is loaded in Mechanical, review how it is presented in the project tree. Note that the **Future Analysis** property of the Analysis Data Management Category in the **Analysis Settings** object of the *upstream system* is specified as **Structural Optimization**. And, the **Environment Selection List** property of the Solution object of the optimization system shows the cell identifiers of the environments linked to the current optimization environment, as illustrated below



Note:

Also illustrated above, the application automatically inserts and assigns default values to the **Optimization Region**, **Objective**, and **Response Constraint** objects in the optimization analysis.

2.3. Specify Analysis Settings

This topic examines the application defaults for Analysis Settings properties based on the optimization method. For a general overview of the use of analysis settings, see the Establish Analysis Settings section.

Analysis Settings per Method

Density Based Optimization

For the **Topology Optimization - Density Based** method, you can modify the properties as described below.

Reload Volume Analysis

The **Reload Volume Analysis** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis and it only displays following the solution process. This category includes the **Reload Volume Fraction** property. The options for this property include **Off** and **Manual**. When you set the property to **Manual**, the **Current Reload Point** property also displays and provides the following options:

- **Initial**: Using this setting, your next solution begins from Iteration 0 (a new solution).
- **Iteration Number ##**: This setting is based on the last Iteration completed for the previous solution. Using this setting, your next solution begins by reloading the volume fractions at the specified iteration number.

Note:

When you pick a reload point that is different from the **Initial** reload point, you may receive an optimized shape that is different than the optimized shape obtained when you perform the optimization from the beginning of the analysis.

Important:

If you modify any of the following, the **Reload Volume Analysis** category disappears from the **Analysis Settings**:

- Manufacturing Constraints (You can use the **Pull Out Direction** Manufacturing Constraint for **Reload Volume Analysis** if you set the **Region of Manufacturing Constraint** property to **Exclude Exclusion**)
- Optimization Region
- **Region of Manufacturing Constraint** property
- **Region of Min Member** property
- Size property
- Any associated upstream system

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **`500`**.

- **Minimum Normalized Density**: This property requires a value greater than **`0`** and less than **`1`**. The Structural Optimization analysis uses this value to extract the permissible range of retained threshold values. The default value is **`0.001`**.

- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. This value must be less than or equal to **2**. The default value is **`0.1%`**.

  Note:

  Specifying a lower **Convergence Accuracy**, for example **`0.05%`**, is recommended if your optimization objective is to minimize Mass/Volume with Global/Local von-Mises stress constraint.

- **Initial Volume Fraction**: This property requires a value greater than 0 and less than or equal to 1. The optimization analysis uses this value as an initial estimate for the volume fraction. All optimized elements begin with this value. By default, the application uses your specified Objectives and Constraints to determine the initial estimate. A value of 1 indicates full material, whereas all other values indicate a fraction of the original material.

- **Penalty Factor (Stiffness)**: During the solution process, this property applies a penalty factor to the structural stiffness matrix in order to prevent the stiffness matrix from scaling linearly with the pseudo density. Therefore, the stiffness at each iteration, as needed, is modified using the expression: $\text{Pseudo Density}^{\text{Penalty Factor (Stiffness)}}$. The default value is **3**.

- **Region of Manufacturing Constraint**: The options for this property are **Include Exclusions** (default) and **Exclude Exclusions**. When you specify **Include Exclusions**, the application also incorporates the **Exclusion Region** (p. 24) to satisfy the **Pull Out Direction**, **Extrusion**, **Cyclic Repetition**, and **Symmetry** manufacturing constraints.

- **Region of Min Member Size**: The options for this property are **Include Exclusions** and **Exclude Exclusions** (default). When you specify **Include Exclusions**, the application also incorporates the **Exclusion Region** (p. 24) to meet the minimum member size specified through the **Member Size** manufacturing constraint.

- **Region of AM Overhang Constraint**: If your analysis includes the specifications of an **AM Overhang Constraint**, this property enables you to include or exclude the object on the **Exclusion Region** (p. 24) of the model. The options for this property are **Include Exclusions** and **Exclude Exclusions** (default). When you specify **Include Exclusions**, the application applies the **AM Overhang Constraint** to the **Exclusion Region**.

- **Filter** (Density Based method only): This property specifies the method used to calculate the pseudo density of each element. Options include:
  - **Linear**: A faster processing option than Non-Linear, this option may prefer placing material on the boundary of your design domain as well as cause the **Minimum** value of the **Member Size** property to infringe upon the boundary of the design domain.
  - **Non-Linear** (**Program Controlled** default): This option uses more advanced algorithms to calculate the pseudo densities that resolve the drawbacks of linear filtering.

Note:

Ansys recommends that you exclude all areas where loads are applied.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:
  - **No** (default): No solution data is generated.
  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).
  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.
  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

  Important:

  Strain-based results are not supported for the **Topology Optimization - Density Based** method.

  Note:

  These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Solver Controls Category

The Solver Type property is the only property for this category. The property's options include:

- **Program Controlled** (default): The application selects the **Sequential Convex Programming** solver as the default option.
- **Sequential Convex Programming**: The **Sequential Convex Programming** method is an extension of the method of moving asymptotes (MMA). The Sequential Convex Programming method requires the derivatives of all functions present in the Structural Optimization analysis.

  See the Optimization Solver Methods (p. 110) section for additional technical detail about this solver type.
- **Optimality Criteria**: The **Optimality Criteria** method can be used to solve Structural Optimization problems with a simple compliance objective that uses a volume or mass constraint.

  Note:

  The following support limitations apply to the **Optimality Criteria** method:

  - Only supports the **Compliance** (Structural) setting for the **Response Type** column of the **Objective** object worksheet.
  - Only **Volume** and **Mass** constraints are supported.
  - The **Manufacturing Constraint** is supported where only the **Minimum** property for the **Member Size** constraint subtype can be specified.

  See the Optimization Solver Methods (p. 110) section for additional technical details about this solver type.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

Level Set Based Optimization

For the **Topology Optimization - Level Set Based** method, you can modify the properties as described below.

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **`500`**.
- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **`0.1%`**.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:
  - **No** (default): No solution data is generated.
  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).
  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.
  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

  Note:

  These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

Lattice Optimization

For the **Lattice Optimization** method, you can modify the properties as described below.

Reload Volume Analysis

The **Reload Volume Analysis** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis and it only displays following the solution process. This category includes the **Reload Volume Fraction** property. The options for this property include **Off** and **Manual**. When you set the property to **Manual**, the **Current Reload Point** property also displays and provides the following options:

- **Initial**: Using this setting, your next solution begins from Iteration 0 (a new solution).
- **Iteration Number ##**: This setting is based on the last Iteration completed for the previous solution. Using this setting, your next solution begins by reloading the volume fractions at the specified iteration number.

Note:

When you pick a reload point that is different from the **Initial** reload point, you may receive an optimized shape that is different than the optimized shape obtained when you perform the optimization from the beginning of the analysis.

Important:

If you modify any of the following, the **Reload Volume Analysis** category disappears from the **Analysis Settings**:

- Manufacturing Constraints (You can use the **Pull Out Direction** Manufacturing Constraint for **Reload Volume Analysis** if you set the **Region of Manufacturing Constraint** property to **Exclude Exclusion**)
- Optimization Region
- **Region of Manufacturing Constraint** property
- **Region of Min Member** property
- Size property
- Any associated upstream system

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **500**.

- **Minimum Normalized Density**: This property requires a value greater than **0** and less than **1**. The Structural Optimization analysis uses this value to extract the permissible range of retained threshold values. The default value is **0.001**.
- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. This value must be less than or equal to **2**. The default value is **0.1%**.

  Note:

  Specifying a lower **Convergence Accuracy**, for example **0.05%**, is recommended if your optimization objective is to minimize Mass/Volume with Global/Local von-Mises stress constraint.

- **Initial Volume Fraction**: This property requires a value greater than 0 and less than or equal to 1. The optimization analysis uses this value as an initial estimate for the volume fraction. All optimized elements begin with this value. By default, the application uses your specified Objectives and Constraints to determine the initial estimate. A value of 1 indicates full material, whereas all other values indicate a fraction of the original material.
- **Region of Manufacturing Constraint**: The options for this property are **Include Exclusions** (default) and **Exclude Exclusions**. When you specify **Include Exclusions**, the application also incorporates the **Exclusion Region** (p. 24) to satisfy the **Pull Out Direction**, **Extrusion**, **Cyclic Repetition**, and **Symmetry** manufacturing constraints.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:
  - **No** (default): No solution data is generated.
  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).
  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.
  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

  Note:

  These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Solver Controls Category

The Solver Type property is the only property for this category. The property options include:

- **Program Controlled** (default): The application selects the **Sequential Convex Programming** solver as the default option.
- **Sequential Convex Programming**: The **Sequential Convex Programming** method is an extension of the method of moving asymptotes (MMA). The Sequential Convex Programming method requires the derivatives of all functions present in the Structural Optimization analysis.

  See the Optimization Solver Methods (p. 110) section for additional technical detail about this solver type.
- **Optimality Criteria**: The **Optimality Criteria** method can be used to solve Structural Optimization problems with a simple compliance objective that uses a volume or mass constraint.

  Note:

  The following support limitations apply to the **Optimality Criteria** method:

  – Only supports the **Compliance** (Structural) setting for the **Response Type** column of the **Objective** object worksheet.

  – Only **Volume** and **Mass** constraints are supported.

  See the Optimization Solver Methods (p. 110) section for additional technical details about this solver type.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

Shape Optimization

For the **Shape Optimization** method, you can modify the properties as described below.

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is `500`.

- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is `0.1%`.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:

  - **No** (default): No solution data is generated.

  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).

  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.

  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

Note:

These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

Mixable Density Based Optimization

For the **Topology Optimization - Mixable Density** method, you can modify the properties as described below.

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is `500`.
- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is `0.1%`.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:
  - **No** (default): No solution data is generated.
  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).
  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.
  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

Note:

These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

Topography Optimization

For the **Topography Optimization** method, you can modify the properties as described below.

Definition Category

The **Definition** category of the **Analysis Settings** is only available when performing a **Structural Optimization** analysis. The properties of the **Definition** category include:

- **Maximum Number of Iterations**: This property specifies the maximum number of iterations performed for the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **`500`**.

- **Convergence Accuracy**: This property specifies the convergence criteria of the Structural Optimization analysis. The solution process continues until the application achieves convergence accuracy or reaches the maximum number of iterations. The default value is **`0.1%`**.

Output Controls

The properties of the **Output Controls** category include:

- **Export Design Properties**. This property is only available for a Structural Optimization analysis when you have an upstream Modal, Static Structural, or Thermal analysis system. You use the options when the application creates solution data. You can then create results in your Structural Optimization analysis that correspond to your upstream analysis. These results enable you to examine the mechanical behavior of your optimal design (corresponding to the last accepted iteration) as well as the mechanical behavior of intermediate designs. Property options include:

  - **No** (default): No solution data is generated.

  - **On Final Design**: Solution data is created at the end of the optimization process. This corresponds to the last accepted iteration (optimal design).

  - **All Accepted Iterations**: Solution data is generated after each accepted iteration of the optimization process. That is, the solution data includes all accepted iterations.

  - **Last Accepted Iteration**: Solution data is generated during the optimization process that corresponds to the last accepted iteration. That is, the single data is updated during the optimization in accordance with the latest accepted design.

  Export Design Properties File Format: This property displays when you specify the **Export Design Properties** property. Options include **HDF5 File** (default) and **VTK File** (requires external reader).

  Note:

  These structural results are displayed on a triangle-based surface-mesh.

- **Store Results At**: Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**.

  Value. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.

See Output Controls section for additional information about the properties of this category.

Analysis Data Management Category

See the Analysis Data Management Category section of the Help for additional information about the properties of this category.

2.4. Define Optimization Method and Regions

You use the **Optimization Region** object to specify the optimization method you want to use and then select the regions of your geometry on which to perform optimization. Using the properties of the object, you define the **Design Region** and the **Exclusion Region** for the analysis. Refer to the Optimization Region object reference page for additional information about this object.

Go to a section topic:

- Select the Optimization Type (p. 24)
- Create Multiple Optimization Regions (p. 25)
- Specify Object Properties (p. 26)
- Define Additional Exclusion Regions (p. 28)

Select the Optimization Type

You use the **Optimization Type** property to select the desired optimization method.

Optimization Type Property Options

- **Topology Optimization - Mixable Density** (default)
- **Topology Optimization - Level Set Based**
- **Lattice Optimization**
- **Shape Optimization**
- **Topography Optimization**
- **Topology Optimization - Density Based**

See the Design Method Definitions (p. 93) section for more information about the requirements and limitations of each method.

Design Region Property

The properties of the **Design Region** category enable you to define the geometry as a **Geometry Selection** or a **Named Selection**. This is the region that you wish to optimize.

Exclusion Region Property

The properties of the **Exclusion Region** category enable you to specify a region (geometric entities or elements) to be excluded from optimization. You specify excluded regions using defined Boundary Conditions, Geometry Selection, or a Named Selection.

Create Multiple Optimization Regions

You can specify multiple **Optimization Regions** objects and mix the level set, mixable density, shape optimization, and topography methods together. When mixing the use of these methods, you can define as many **Optimization Region** objects as needed and attach individual manufacturing constraints to each.

Density Based optimization regions cannot be mixed with other types of optimization regions. When you are using the density-based optimization method you have two options: you can use multiple **Optimization Region** objects or you can use **Geometry Selection** or **Named Selections** to define the scope of the Manufacturing Constraints.

Important:

- Each optimization region must be uniquely scoped. The scoping of the regions cannot overlap. The application will include all the geometries of the optimization regions in the optimization process. You can use the individual optimization regions for the scoping of geometric response constraints and of manufacturing constraints.
- The **Topology Optimization - Level Set** and **Shape Optimization** options/method support 3D solid elements only and the **Topography Optimization** method supports shell elements only.

The following table outlines the methods that you can combine within an optimization problem.

	Density Based Optimization	**Lattice Optimization**	**Level Set Based Optimization**	**Mixable Density**	**Shape Optimization**	**Topography Optimization**
Density Based Optimization	X					
Lattice Optimization		X				
Level Set Based Optimization			X	X	X	X
Mixable Density			X	X	X	X
Shape Optimization			X	X	X	X
Topography Optimization			X	X	X	X

Specify Object Properties

Using the properties of the **Details** pane, specify the method optimization regions for your analysis.

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td>Design Region</td><td>Scoping Method. The options for this property include:

Geometry Selection (default): This option indicates that the design region is applied to a geometry or geometries, which are chosen using graphical selection tools. When you specify Geometry Selection for the Scoping Method, the Geometry property will appear.

In this case, use selection filters on the Graphics Toolbar to pick your geometric entities (only body and element selection), and then click Apply. Once complete, the property displays the type of geometry (Body, Element, etc.) and the number of selected geometric entities (for example: 1 Body, 12 Elements).
Named Selection: Indicates that the geometry selection is defined by a Named Selection. When you specify Named Selection for the Scoping Method, the Named Selection property will appear. This property provides a drop-down list of available user-defined Named Selections (only body-based and element-based Named Selections are supported).
</td></tr>
<tr><td>Exclusion Region</td><td>Defined By. The options for this property include:

Boundary Condition (default):

When you specify Boundary Condition for the Defined By property, a Boundary Condition property will appear. This property includes the following options:

All Boundary Conditions: Applies the locations of all loading conditions and supports from the upstream static/modal system.
All Loads: Applies the locations of all loading conditions from the upstream static/modal system.
All Supports: Applies the locations of all supports from the upstream static/modal system.
None: No boundary condition locations are applied.

Note:

Ansys strongly recommends that, at a minimum, you specify the option All Loads for any Exclusion Region.
Geometry Selection: This option indicates that the design region is applied to a geometry or geometries, which are chosen using graphical selection tools. When you specify Geometry Selection for the Scoping Method, the Geometry property
</td></tr>
</table>

Category	Properties/Options/Description
	will appear. Exclusion Region geometry selections can be scoped to bodies, faces, edges, vertices, elements, and nodes. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. When you specify **Named Selection** for the **Scoping Method**, the **Named Selection** property will appear. This property provides a drop-down list of available user-defined Named Selections. Exclusion Region Named Selections support body-, face-, edge-, vertex-, element-, and node-based Named Selections. **Exclusion Thickness**: This property displays when you set the **Optimization Type** property to **Topology Optimization - Level Set Based**, **Topology Optimization - Mixable Density**, or **Shape Optimization**. The default setting for this property is **Program Controlled** or you can enter a value manually. This option enables you to specify a thickness for the **Exclusion Region**. The default value is two times the average element size. The unit must be a length. **Exclusion Extension**: This property displays when you set the **Optimization Type** property to either **Topology Optimization - Level Set Based** or to**Topology Optimization - Mixable Density**. This option enables you to specify the extension type for the **Exclusion Region**. Options include is **Isotropic** (default) and **Orthotropic**. Use the **Isotropic** option to perform expansion uniformly in the three directions. Use the **Orthotropic** option to perform the expansion along the surface normal. The **Orthotropic** option is only available for surface bodies.
Optimization Option	**Optimization Type**: You use this property to specify the type of optimization you wish to perform. Options include: • **Topology Optimization - Density Based** • **Topology Optimization - Level Set Based** • **Lattice Optimization** (see below) • **Shape Optimization** (see below) • **Topology Optimization - Mixable Density** • **Topography Optimization** (see below) **Lattice Optimization** The following additional properties are available when you select **Lattice Optimization** as the **Optimization Type**: • **Lattice Type**: This property specifies the structure of the unit cell. • **Minimum Density**: This property specifies a minimum density in order to avoid lattice structures that are too thin. • **Maximum Density**:This property specifies a maximum density. The element will be considered as full for densities higher than the **Maximum Density**.

Category	Properties/Options/Description
	• **Lattice Cell Size**: The value of this property specifies the lattice cell size to be used when rebuilding the lattice geometry for printing. **Shape Optimization and Topography Optimization** The following additional properties are available when you select **Shape Optimization** or **Topography Optimization** as the **Optimization Type**: • **Move Limit Per Iteration**: This property enables you to define how far each node can move at each iteration. It must be defined in length units, for example one element size. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value. • **Total Move Limit**: This property enables you to define how far each node can move in total. It must be defined in length units, for example three times the element size. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value. • **Mesh Deformation Control**: This property enables you to define how much the mesh can be stretched. It is an additional control to avoid element distortion. This unit-less value is a sort of penalty factor that ranges from 0 (no control) to 1.0. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.

Define Additional Exclusion Regions

The **Optimization Region** object provides the contextual (right-click) menu option **Local Design Restriction** that enables you to specify additional design regions on your model. Inserted as a child object to the **Optimization Region** object, it provides the scoping options **Geometry Selection** and **Named Selection**. For the selected geometry, use the **Type** property to define the additional exclusion restrictions. Options include:

- **Non-Optimizable**: Do not optimize the selected region(s).
- **In-Plane Morphing**[1]: Supported for the **Shape Optimization** method and face selections only, the application does not move any nodes vertically with respect to the original face. That is, the nodes remain "in the plane."

[1] **Limitation**

If your simulation includes Symmetry and you specify the **Type** property as **In-Plane Morphing** , you must scope the **Local Design Restriction** object to each side of the symmetrical body. Otherwise, the application ignores the **In-Plane Morphing** setting. For example, if you select only one vertical face of the symmetric body (left image in the figure below), the application ignores the **In-Plane Morphing** condition. You must select both corresponding faces on either side of the symmetry (as shown in the image on the right) so that the application respects the symmetry and correctly processes the **In-Plane Morphing** condition.

The recommended use of the Local Design Restriction object includes scenarios wherein certain regions of interest, that may be disconnected, need to be retained even though no loads or boundary conditions exist on the region.

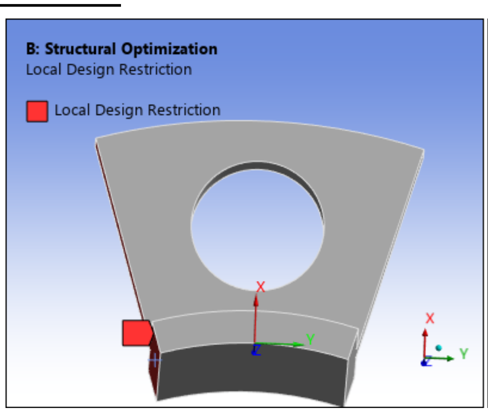
2.5. Determine Objective and Constraints Capabilities

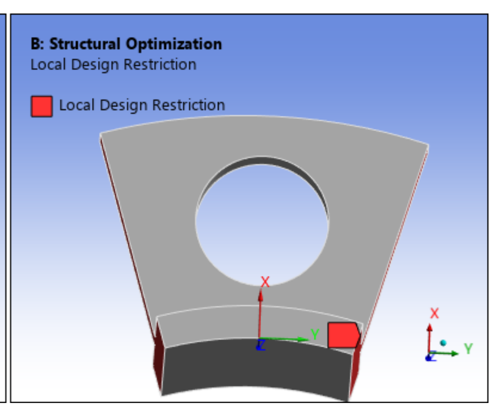
Objective and Response Constraint Capability Map

The options for the **Objective** and **Response Constraint** object depend upon the analysis you are optimizing, the response constraint you want to specify, and the optimization method you are using. Review the following table to see which Objective and Response Constraint objects are available for each method, noting the following:

- **Constraint**/**Objective** (**C**/**O**): Supported as Response Constraint and Objective.
- **C**: Supported as a Response Constraint only.
- **O**: Supported as a Objective only.

Upstream Analysis System?	Response	Density Method	Lattice Method	Level Set Method	Shape Optimization Method	Mixable Density	Topography
Geometric (default)		Y	Y	Y	Y	Y	Y
	Mass/Volume[a]	C/O	C/O	C/O	C/O	C/O	C/O
	Center of Gravity[a]	C	C	C	C	C	
	Moment of Inertia[a]	C	C	C	C	C	
	User Defined Criterion			C/O	C/O	C/O	C/O
Harmonic Response[b]	**Upstream analysis supported?**	**No**	**No**	**Yes**	**Yes**	**Yes**	**Yes**
	User-defined Criterion			C/O	C/O	C/O	C/O





Upstream Analysis System?	Response	Density Method	Lattice Method	Level Set Method	Shape Optimization Method	Mixable Density	Topography
Static Structural System	**Upstream analysis supported?**	**Yes**	**Yes**	**Yes**	**Yes**	**Yes**	**Yes**
	Compliance[c]	**C/O**	**C/O**	**C/O**	**C/O**	**C/O**	**C/O**
	Displacement[a]	**C**	**C**	**C**	**C**		
	Stress	**C/O**	**C/O**	**C/O**	**C/O**	**C/O**	
	Local Stress	**C**					
	Reaction Force[a]	**C**		**C**	**C**		
	User-defined Criterion	**C/O**		**C/O**	**C/O**	**C/O**	**C/O**
Nonlinear Static Structural System - (Nonlinear Contact only) [d]	**Upstream analysis supported?**	**Yes (Beta)**	No	No	**Yes**	**Yes**	No
	Compliance	**C/O**					
	Displacement	**C**					
	Stress	**C/O**					
	Local Stress	**C**					
	Reaction Force	**C**					
	User-defined Criterion						
	Accumulated Plastic Strain (APS)				**O**		
Nonlinear Static Structural [d]	**Upstream analysis supported?**	No	No	No	**Yes**		No
	Accumulated Plastic Strain (APS)				**O**		
Modal	**Upstream analysis supported?**	**Yes**	**Yes**	**Yes**	**Yes**	**Yes**	**Yes**
	Frequency[a]	**C/O**	**C/O**	**C/O**	**C/O**		
	User-defined Criterion			**C/O**	**C/O**	**C/O**	**C/O**
Steady-State Thermal System	**Upstream analysis supported?**	**Yes**	No	**Yes**	**Yes**	**Yes**	No
	Thermal Compliance	**O**		**C/O**	**C/O**	**C/O**	
	Temperature	**C**					
Nonlinear Steady-State Thermal System	**Upstream analysis supported?**	**Yes**	No	No	No	No	No
	Thermal Compliance	**O**					

Upstream Analysis System?	Response	Density Method	Lattice Method	Level Set Method	Shape Optimization Method	Mixable Density	Topography
(Radiation Only)	Temperature	C					

[a] This response will be deprecated in a future release. Ansys recommends that you use a **User Defined Criterion** response instead as it is more versatile.

[b] The Structural Optimization analysis only supports an upstream standalone MSUP Harmonic Response analysis.

[c] The application ignores Condensed Part objects included in the model when computing compliance.

[d] To specify a nonlinear analysis, set Large Deflection property to On in the Details of the Solver Controls category of the Analysis Settings.

2.6. Define Design Objectives

An Objective object is added by default for optimization systems. This object displays the **Worksheet** in order for you to specify the optimization goal.

Go to a section topic:

- Application (p. 31)
- Objective Worksheet Overview (p. 31)
- Objective Worksheet Properties (p. 33)

Application

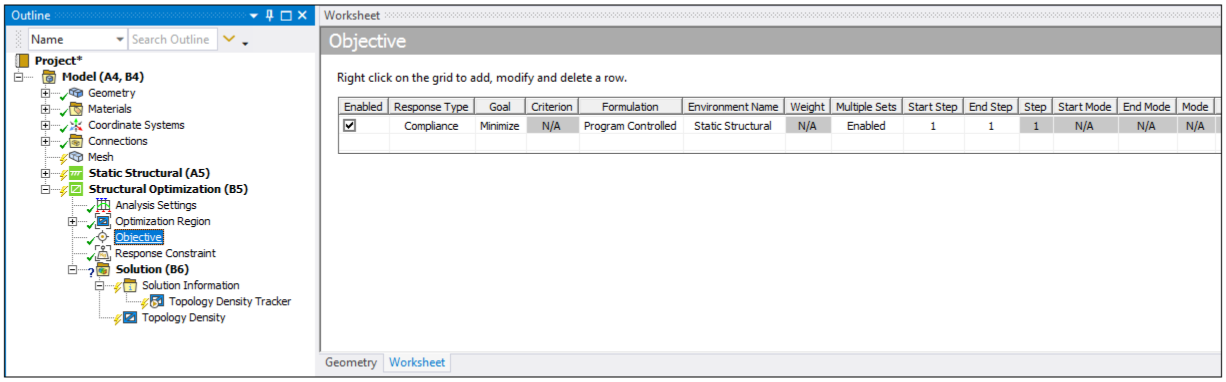
To apply an **Objective** object: On the **Environment** Context tab: select **Objective**, or right-click the **Environment** tree object or in the **Geometry** window and select **Insert** > **Objective**.

Objective Worksheet Overview

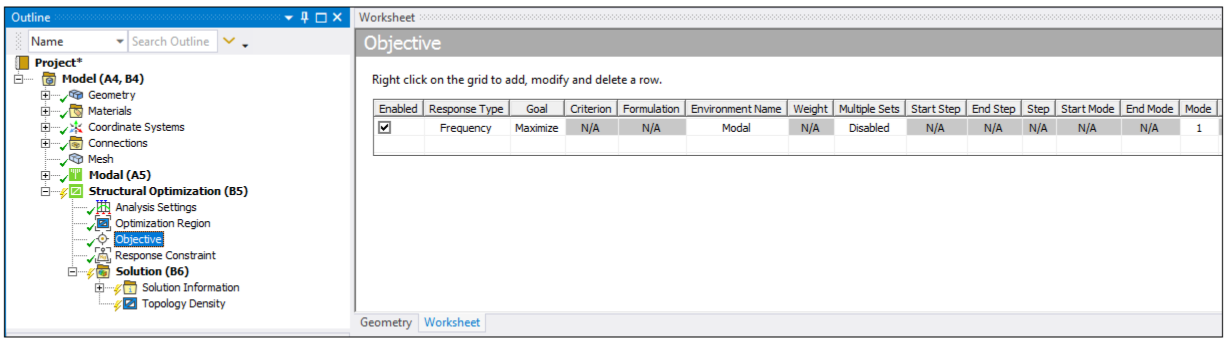
When you select the **Objective** object, the **Worksheet** displays by default, as illustrated in the images shown below. You use the **Worksheet** to specify **Response Type**, **Formulation**, **Goal**, and **Weights** for the steps/modes of the upstream analysis. A **Structural Optimization** analysis supports one or more upstream **Harmonic Response**, **Modal**, **Static Structural**, or **Steady-State Thermal** analyses. For the **Shape Optimization** method, nonlinear **Static Structural** analyses are also supported.

Whenever you link a **Harmonic Response**, **Modal**, **Static Structural**, or **Steady-State Thermal** analysis to a **Structural Optimization** analysis in the **Project Schematic**, a new row is added to the **Worksheet**. The default **Response Type** setting for a **Static Structural** analysis is **Compliance**, for a **Modal** analysis it is **Frequency**, and for a **Steady-State Thermal** analysis it is **Thermal Compliance**. There is not default setting for a **Harmonic Response** analysis.

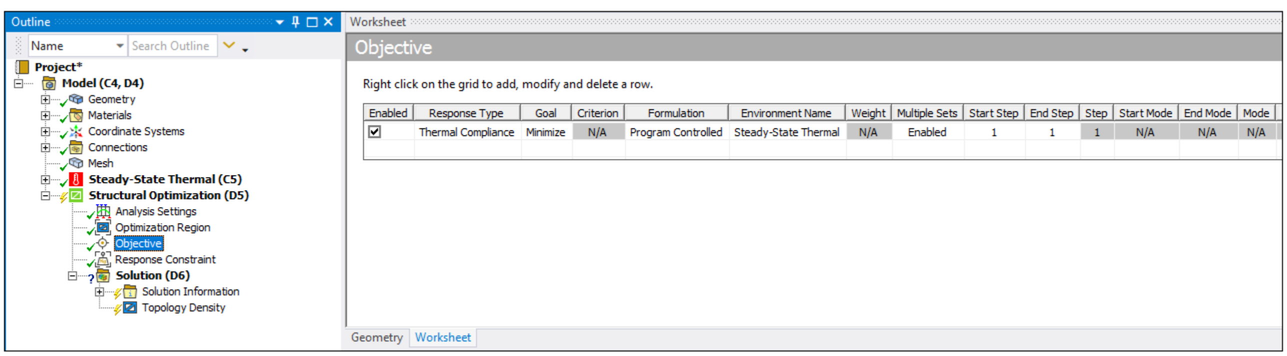
Single Static Structural System



Single Modal System



Single Thermal System



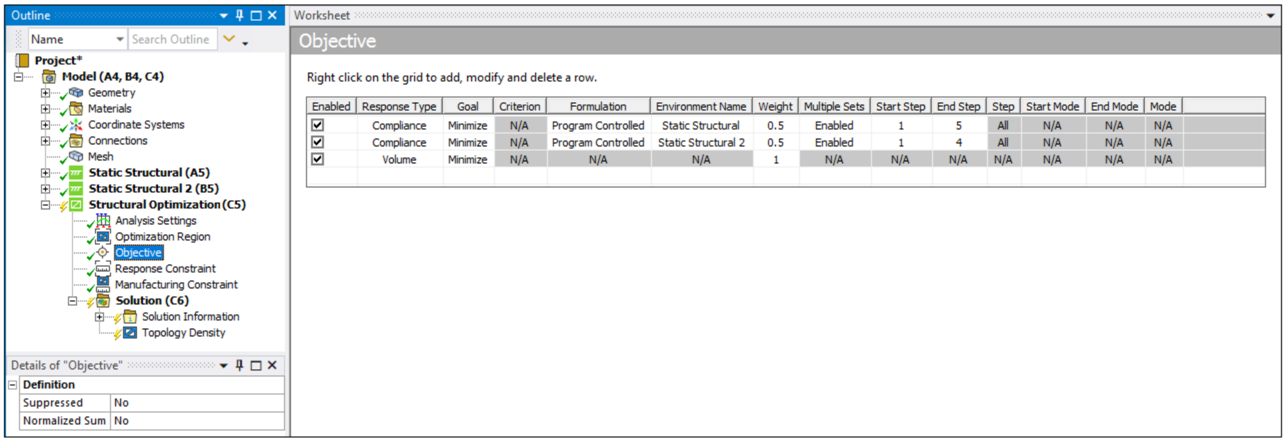
You can insert additional rows into the **Worksheet** to specify multiple response types for multiple systems and **Weight** values for a single step or multiple steps (by disabling or enabling the **Multiple Sets** option).

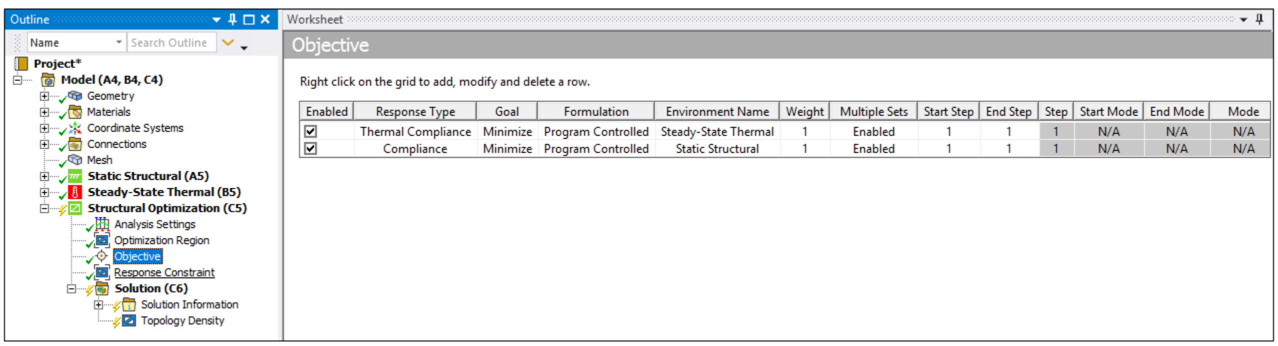
Note:

For the **Topology Optimization - Density Based** method, **Objective** objects of different types are always normalized. Here, the option only applies for Objective objects of *same* type.

For the **Topology Optimization - Level Set** method, when you have multiple **Objective** objects specified, you can choose to normalize them with their initial response to give equal weightage for all objectives. To do so, set the **Normalized Sum** property to **Yes**.

Multiple Systems





Worksheet Properties

You set the values for properties in the **Worksheet** columns to define the **Objective** object as follows:

Enabled

This option is checked by default. When checked the application includes the specifications of the row in the solution. Unchecking the row excludes it from the solution.

Response Type

The options for this column depend upon the analysis you are optimizing. Supported options include:

- Mass/Volume
- Stress
- Compliance
- Criterion

- Accumulated Plastic Strain
- Frequency
- Thermal Compliance

Review the Objective and Response Constraint Capability Map (p. 29) for a complete listing of the supported analysis types, response constraint you want to specify, and the optimization method you are using.

Note:

When you select a criterion that is scoped to Remote Point or remote boundary condition (**Remote Force** or **Remote Displacement**), the **Base Result** property options **Reaction Force** and **Reaction Moment** are not supported for the Density Based and Lattice Optimization methods.

Goal

The options for this property depend upon the selection made in the **Response Type** column.

If constraint is...	Then this entry can be set to...
Mass/Volume **Stress** **Accumulated Plastic Strain**	**Minimize**.
Frequency	**Maximize**.
Criterion **Compliance**	**Minimize** or **Maximize**[a].
Thermal Compliance	**Minimize** or **Maximize**.

[a] Maximization of Compliance is not supported for the **Density Based** method.

Note:

Generally, you set static and thermal compliance constraints to **Minimize**, however, the application also supports the **Maximize** setting.

Formulation

This column is applicable only when the **Response Type** is set to **Compliance** (Static Structural analysis), **Stress** (Static Structural analysis), or **Thermal Compliance** (Steady-State Thermal Analysis).

Compliance

For the **Density Based** optimization method, when you select **Compliance** (Static Structural analysis), this column is used to specify the formulation method by which maximum stiffness can be attained to minimize the compliance.

- **Program Controlled**: This default setting selects either the **Force** or **Displacement** formulation based on whether a force load or a displacement load exists in the Static Structural analysis.
- **Force**: If you have not applied a displacement load and a force load is applied, then the displacement, with respect to this force load, is minimized during the optimization.
- **Displacement**: If you have a non-zero displacement load and no force load is applied, then the force that leads to the given displacement is maximized during the optimization.

Note:

For the **Topology Optimization - Level Set Based** and **Shape Optimization** methods, the **Compliance** option uses a unique formula regardless of the context. That is, it executes in the presence of surface loads, acceleration, prescribed displacement, and/or thermal expansion.

Stress

When **Stress** is the specified **Response Type**, **Formulation** options include:

- **Equivalent von-Mises Stress** (default). This option is supported for density based, level set, and shape optimization methods.

$$J(\Omega) = \max_{X e \Omega} \sigma_{VM}(x)$$

- **Local Equivalent von-Mises Stress**. This option is only supported for shape optimization methods.
- **Maximum Principal Stress**. This option is supported for density based, level set, and shape optimization methods.

$$J(\Omega) = \max_{X e \Omega} \sigma_{MPS}(x)$$

- **Local Strain Energy**. This option is supported for level set and shape optimization methods.

  The strain energy computed only over the optimization region of interest.

$$J(\Omega) = \int_{\Omega} (\sigma : \varepsilon)\, dx$$

Important:

The **Local Strain Energy** calculation is exactly twice the **Compliance** if, and only if:

1. **Local Strain Energy** is computed over the entire model, and,

2. There are only loads in the upstream structural analysis.

As a result, Ansys recommends that you keep this in mind when specifying this stress option. For example, if you apply a prescribed-displacement and minimize the **Local Strain Energy**, you could experience an unwanted (but expected) disconnection.

While the **Local Strain Energy** is by nature, a scalar value, the **Equivalent von-Mises Stress** and **Maximum Principal Stress** options are elemental scalar-fields. Before being consumed by the optimizer, those two stress-norm are scalarized in a way to mimic the maximum and to stay derivable.

As summarized below, the application monitors all elements of the optimization region, however, based on the method and criterion, some elements may be ignored. This table describes the supported methods only.

Criterion	Methods[a]		
	Density Based	**Level Set Based**	**Shape Optimization**
Equivalent von-Mises Stress	Scoping elements: All elements except those lying in the Exclusion Region.	Scoping elements: All elements except those lying in the Exclusion Region.	Scoping elements: All.
Local Equivalent von-Mises Stress	Not Available	Not Available	Scoping elements: All elements except those lying in the Exclusion Region.
Maximum Principal Stress	Scoping elements: All elements except those lying in the Exclusion Region.	Scoping elements: All elements except those lying in the Exclusion Region.	Scoping elements: All.
Local Strain Energy	Not Available	Not Available	Scoping elements: All.

[a] For the mixable density and topography methods, no elements are monitored.

Thermal Compliance

For **Thermal Compliance**, this column is used to specify the formulation method by which heat transfer is maximized. Options include **Program Controlled** (default), **Thermal Load**, and **Temperature**. The **Thermal Load** option includes all thermal loads (Convection, Heat Flux, etc.) *except***Temperature**. For this **Worksheet** property, note the following behaviors:

- **Program Controlled**: This default setting selects either the **Temperature** or **Thermal Load** formulation based on whether a temperature load or a thermal load exists in the Steady-State Thermal analysis.

- **Thermal Load**: If you have not applied a Temperature load and a Thermal Load is applied, then the average temperature, with respect to this thermal load, is minimized during the optimization.

- **Temperature**: If you have Temperature load and no Thermal Load is applied, then the thermal load that leads to the given temperature is maximized during the optimization.

> **Note:**
>
> For the **Topology Optimization - Level Set Based** and **Shape Optimization** methods, the **Thermal Compliance** option uses a unique formula regardless of the context. That is, it executes in the presence of Heat Flux, Heat Flow, prescribed Displacement, Convection conditions, and/or Internal Heat Generation.

Environment Name

From the drop-down list, select the environment associated with the entries of the **Response Type** and **Goal** properties. Note that geometric criteria (mass, volume, etc) does not need to be associated to an environment.

Criterion

From the drop-down list, select the desired criterion.

Weight

The default value for this option is **1**. The **Weight** can be any real number.

If you define multiple responses, they are aggregated into a weighted sum, using:

$$f\left(\Omega_k\right) = \sum_{i-1} \alpha_i f_i\left(\Omega_k\right)$$

Where:

Ω_k is the design of the k^th iteration.

$f_i\left(\Omega_k\right)$ is the i^th Objective response.

α_i is the effective weight of the i^th Object response.

$f\left(\Omega_k\right)$ is the aggregated Objective.

When the **Normalized Sum** option is set to **No**, then $\alpha_i = \frac{\omega_i}{\sum_i \omega_i}$, where ω_i is the entered weight.

When the **Normalized Sum** option is set to **Yes**, then $\alpha_i = \frac{\frac{\omega_i}{\left|f_i\left(\Omega_0\right)\right|}}{\sum_i \frac{\omega_i}{\left|f_i\left(\Omega_0\right)\right|}}$. This indicates that it is scaled using the value at the first iteration.

Multiple Sets

The values for this option are **Enabled** (default) or **Disabled**.

- If **Enabled**, you can specify **Start Step** and **End Step** values within the boundaries of the maximum number of steps defined in the upstream Static Structural analysis.
- If **Disabled**, only the **Step** column is available to define the **Weight** for a single step.

Start Step

This option is available when the **Environment Name** column is set to either **Static Structural** or **Steady-State Thermal**. This option requires the **Multiple Sets** option to be set to **Enabled** in order to define the Start Step from the upstream solution.

End Step

This option is available when the **Environment Name** column is set to either **Static Structural** or **Steady-State Thermal**. This option requires the **Multiple Sets** option to be set to **Enabled** in order to define the **End Step** from the upstream solution.

Step

This option is supported when the **Environment Name** column is set to either **Static Structural** or **Steady-State Thermal**. This option specifies the step number used from the upstream solution. This field is read-only when the **Multiple Sets** option is set to **Enabled**, and the entry for this option can also be **Multiple** or **All**, if the **Start** and **End Step** values cover more than one step or the entire analysis from the upstream solution. Otherwise, you can specify the weight for single steps using this option.

Start Mode

Only supported when **Modal** system is selected in the **Environment Name** column. This field requires the **Multiple Sets** option to be set to **Enabled** in order to define the **Start Mode** for the selected Modal analysis solution.

End Mode

Only supported when **Modal** system is selected in the **Environment Name** column. This field requires the **Multiple Sets** option to be set to **Enabled** in order to define the **Start Mode** for the selected Modal analysis solution.

Mode

Only supported when Modal system is selected in the **Environment Name** column. This option specifies the mode number used from the upstream solution. This field is read-only when the **Multiple Sets** option is set to **Enabled** and the entry for this option can also be **Multiple** or **All**, if the **Start Mode** and **End Mode** values cover more than one mode or all of the modes from the upstream solution. Otherwise, you can specify the weight for single modes using this option.

Refer to the Objective object reference page for additional information.

2.7. Define Response Constraints

The Structural Optimization analysis requires and automatically inserts a response constraint. The available response types include:

- **Volume Constraint**
- **Mass Constraint** (default)
- **Center of Gravity Constraint**
- **Moment of Inertia**
- **Compliance Constraint** (Static Structural)
- **Displacement Constraint** (Static Structural)
- **Reaction Force Constraint** (Static Structural)
- **Global Stress Constraint** (Static Structural)
- **Local von-Mises Stress Constraint** (Static Structural)
- **Natural Frequency Constraint** (Modal)
- **Thermal Compliance** (Steady-State Thermal)
- **Temperature Constraint** (Steady-State Thermal)
- Criterion Constraint (Harmonic Response, Modal, Static Structural)

Note:

Shell Body Stress Constraints

When you apply a **Global Stress Constraint** or a **Local von-Mises Stress Constraint** to a shell body, the application optimizes the body using the elemental mean of the top and bottom surfaces.

Note:

Review the Objective and Response Constraint Capability Map (p. 29) for a complete listing of the supported analysis types, response constraint you want to specify, and the optimization method you are using.

Application

To apply a **Response Constraint**:

1. On the **Environment** Context tab, open the **Response Constraint** drop-down menu and select the desired response constraint, or, right-click the **Environment** (**Structural Optimization**) object or in the **Geometry** window and select **Insert**>**[desired Response Constraint menu option]**.
2. The application inserts the appropriate object matching the selected response option.

   Additional properties display based on the setting of the **Response** property setting, and include:

Geometric-Based Analyses

- **Mass Constraint**/**Volume Constraint**: Based on how you define the constraint, modify the percentage or the value as needed.
- **Center of Gravity Constraint**: Specify the upper and/or the lower limit (**Maximum Value**/**Minimum Value**) and desired **Axis**.
- **Moment of Inertia Constraint**: Based on how you define the constraint, modify the percentage or the value as needed and specify a desired **Coordinate System** and **Axis**.
- **Criterion Constraint**: Specify the **Criterion**, **Lower Bound**, and **Upper Bound** properties.

Harmonic Response Analyses

Criterion Constraint: Specify the **Criterion**, **Lower Bound**, and **Upper Bound** properties.

Modal Analyses

- **Natural Frequency Constraint**: Specify the values for the **Mode Number**, **Minimum Frequency**, and **Maximum Frequency** properties. Modify the **Environment Selection** property as needed.
- **Criterion Constraint**: Specify the **Criterion**, **Lower Bound**, and **Upper Bound** properties.

Static Structural Analyses

- **Global Stress Constraint**: Specify a **Stress Type**, either **Equivalent von-Mises Stress** (default), **Local Equivalent von-Mises Stress**, **Maximum Principal Stress**, or **Local Strain Energy**, and the **Maximum** stress value. Modify the **Environment Selection** property as needed.
- **Local von-Mises Stress Constraint**: Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry. Also specify the **Maximum** stress value. Modify the **Environment Selection** property as needed.
- **Displacement Constraint**: Specify the **X/Y/Z Component (Max)** properties. Modify the **Environment Selection** property as needed.
- **Reaction Force Constraint**: Specify the **Axis Selection**, **Criteria**, **Bound Type**, and **X/Y/Z Component** properties. Modify the **Environment Selection** property as needed.
- **Compliance**: Specify the maximum value. Modify the **Environment Selection** property as needed.
- **Criterion Constraint**: Specify the **Criterion**, **Lower Bound**, and **Upper Bound** properties.

Thermal Analyses

- **Temperature Constraint**: Specify the **Temperature (Abs Max)** property. Modify the **Environment Selection** property as needed.
- **Thermal Compliance**: Specify the maximum value. Modify the **Environment Selection** property as needed.

Note:

Where applicable, the application automatically specifies a (read-only) **Coordinate System** property.

Renaming Based on Definition

The **Response Constraint** object provides the context menu (right-click) option **Rename Based on Definition**. This option automatically renames the object based on your **Response** property selection. That is, it renames the object "**Mass Constraint**,""**Volume Constraint**,""**Global Stress Constraint**," or "**Natural Frequency Constraint**" accordingly. This feature supports all of the options of the **Response** property.

Details View Properties

The **Details** view for this object includes the following properties.

Category	Properties/Options/Description
Scope	**Scoping Method**: The option for this property is based upon the type of **Response Constraint** you specify. For the **Mass Constraint**, **Volume Constraint**, **Center of Gravity**, and **Moment of Inertia** response types, the available **Scoping Method** options include: • **Geometry Selection**: This option indicates that the design region is applied to a geometry or geometries (body selection only), which are chosen using the graphical selection tools. When you specify **Geometry Selection** for the **Scoping Method**, the **Geometry** property displays. In this case, use selection filters on the Graphics Toolbar to pick your geometric entities, and then click **Apply**. Once complete, the property displays the type of geometry and the number of selected geometric entities (for example: 1 Body). • **Named Selection**: This option indicates that the design region is applied to a body-based (only) Named Selection. When you specify **Named Selection** for the **Scoping Method**, the **Named Selection** property displays. This property provides a drop-down list of available user-defined Named Selections. • **Optimization Region** (default): This option indicates that the design region is applied to the specified **Optimization Region**. When **Optimization**

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td></td><td>

Region is specified for the **Scoping Method**, the **Optimization Region Selection** property also displays. This property contains a default value: **Optimization Region**.

- **All Optimization Regions**: When you have multiple **Optimization Region** objects defined, this option indicates that the constraint is applied to all of them.

For **Local von-Mises Stress Constraint**, **Displacement Constraint**, and **Reaction Force Constraint** response types, supported by a linked Static Structural analysis, and the **Temperature Constraint**, supported by a linked Steady-State Thermal analysis, the available options, as described above, include:

- **Geometry Selection**: Not restricted to body-based scoping only.
- **Named Selection**: Not restricted to body-based scoping only.

For the **Global Stress Constraint** (Static Structural only) response type, the only available option is **Optimization Region** and **All Optimization Regions**.

Note:

There is no **Scope** category for the **Natural Frequency Constraint** response type.

</td></tr>
<tr><td>Definition</td><td>

Type

This is a read-only property that indicates the object as a **Response Constraint**.

Response

The options for this property include:

- **Mass** (default)/**Volume**[a]: When you select either of these options, the **Define By** property displays. **Define By** properties include:
  - **Constant** (default): When this option is used, the **Percent to Retain** property also displays. The **Percent to Retain** property defines the upper bound (in percentage) of the Volume/Mass constraint. The default value is **50**. The entry range for this property is between **1** and **99**.

    Note:

    For the **Density Based Optimization** method, this constraint is handled as an equality constraint.

</td></tr>
</table>

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td></td><td>
<li style="list-style:none">
Range: When this option is selected, the Percent to Retain (Min) and Percent to Retain (Max) properties also display. You use these two properties to define the range, namely lower and upper bound, in percentage, of the Volume/Mass constraint. The default value for each is 50. The entry range for these properties is between 1 and 99.
Absolute Constant: When selected, the Maximum Value property also displays. The Maximum Value property defines the units-based upper bound of the Mass/Volume constraint. The default value is Free.

Note:

For the Density Based Optimization method, this constraint is handled as an equality constraint.
Absolute Range[a]: When selected, the Minimum Value and Maximum Value properties also display. You use these two properties to define the units-based range, namely lower and upper bound, of the Mass/Volume constraint. The default value for each is Free.

Center of Gravity[a]: When this option is selected, the Max Value and Min Value properties also display and enable you to specify an upper and lower bound for the constraint. The default value is Free. You will note a value contained in the field when you select it. This is a infinite value to indicate a free state.
Moment of Inertia[a]: When you select this option, the Define By property displays. Define By properties include:

Constant (default): When this option is used, the Percent to Retain property also displays. The Percent to Retain property defines the upper bound (in percentage) of the Moment of Inertia. The default value is 50. The entry range for this property is between 1 and 99.
Range: When this option is selected, the Percent to Retain (Min) and Percent to Retain (Max) properties also display. You use these two properties to define the range, namely the lower and upper bound, in percentage, of the Moment of Inertia. The default value for each is 50. The entry range for these properties is between 1 and 99.
Absolute Constant: Specify the upper bound in the appropriate Unit system.

</td></tr>
</table>

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td></td><td>
<li style="list-style:none">
Absolute Range: Specify the upper and lower bound in the appropriate Unit system.
Natural Frequency[a]: This option is only available when there is at least one upstream Modal system. By using this property, the analysis ensures the specified mode and the range of frequencies are supported by the optimized body. When selected, the following associated properties will be shown:
Mode Number: This property defines the mode number used to create the optimized body.
Minimum Frequency: This property defines the minimum frequency for the selected mode number.
Maximum Frequency: This property defines the maximum frequency for the selected mode number.
You can use multiple Natural Frequency objects that specify different Mode Numbers and corresponding frequency ranges for each upstream Modal system.
Global Stress: This option is only available when there is at least one upstream Static Structural system. You use this property to make sure that the optimized geometry or structure always supports a specified maximum stress. When selected, the following reassociated properties also display:
Stress Type: Options include Equivalent (Von-Mises) Stress, Local Equivalent von-Mises Stress, Maximum Principal Stress, and Local Strain Energy. These options have optimization method support requirements. See the Worksheet Properties (p. 33) topic in the Objective (p. 31) section for more information about the stress options.
Maximum: Enter a stress value as a Constant or using Tabular Data entries.
Local von-Mises Stress: This option is only available when there is at least one upstream Static Structural system. You use this property to make sure that the geometry or structure always supports a specified maximum stress using the Maximum property that also displays when you select the Local von-Mises Stress option. You specify the stress value of the Maximum property as either a Constant (default) or using Tabular Data entries (via fly-out menu). The application supports multiple Local von-Mises Stress constraints. You can apply this constraint on supported elements that may or may not be included in the Optimization Region.
Displacement[a]: This option is only available when there is at least one upstream Static Structural system. You use this property to
</td></tr>
</table>

Category	Properties/Options/Description
	make sure that the optimized geometry or structure always support a specified maximum displacement using the **X/Y/Z Component (Max)** properties that also display when you select the **Displacement** option. A read-only **Coordinate System** property also displays and is automatically set to **Nodal Coordinate System**. You specify the displacement value of the **X/Y/Z Component (Max)** properties as either a **Constant** (default), **Free**, or using **Tabular Data** entries (via fly-out menu). The application supports multiple **Displacement** constraints. **Important:** If you apply a **Displacement** to more than one node, the absolute value for the constraint is met and negative numbers are no longer allowed. For example, if you enter a value of `100N`, the constraint is satisfied if it meets a value between `-100` and `100`. • **Reaction Force**[a]: This option is only available when there is at least one upstream Static Structural system. You use this constraint to make sure that the optimized geometry or structure always support a specified maximum reaction force. The application supports multiple **Reaction Force** constraints. During the solution process, the application calculates a reaction force for each node used in the **Reaction Force** constraint (if scoped to more than one node or a vertex, edge, face, or body). Based on the **Criteria** property setting, the reaction forces are either summed or normalized. Neither of these calculated values can exceed the entries you make in the Component properties for the specified direction(s). **Reaction Force** has the following distinct properties: – **Axis Selection**: Options include **All** (default), **X Axis**, **Y Axis**, and **Z Axis**. – **Criteria**: Options include **Sum** (default) and **Absolute Maximum** (when scoped to more than one node or a vertex, edge, face, or body). **Note:** For legacy databases, release 2019 R1 or earlier, that include **Reaction Force** constraints, the default setting for this property is **Absolute Maximum**.

Category	Properties/Options/Description
	– **Bound Type**: Options include **Upper Bound** (default) and **Lower Bound**. – **X/Y/Z Component (Sum or Max)**: Component entries are either **Constant** or based on **Tabular Data** entries. When the **Criteria** property is set to **Sum**: Positive values are treated as upper (maximum) bounds. Therefore, the constraint is satisfied if the constraint value is less than the value you specify. Negative values are considered as lower (minimum) bounds. Therefore, the constraint is satisfied if the constraint value is greater than the value you specify. A read-only **Coordinate System** property also displays and is automatically set to **Nodal Coordinate System**. **Important:** If you apply a **Reaction Force** to more than one node, and the **Criteria** property is set to **Absolute Maximum**, the absolute value for the constraint is met and negative numbers are no longer allowed. For example, if you enter a value of `100N`, the constraint is satisfied if it meets a value between `-100` and `100`. • **Temperature**: This option is only available when the upstream system is Steady-State Thermal. You use this constraint to put an upper bound on the temperatures using **Temperature (Abs Max)** property. This value can be define as a **Constant** or using **Tabular Data**. • **Compliance**: This option is only available when there is at least one upstream Static Structural system. You use this property to make sure that the optimized geometry or structure is stiff enough. When selected, the **Maximum** property also displays. Enter a value in the **Maximum** property as a **Constant** or using **Tabular Data** entries. When selected, the **Compliance Limit** property also displays. The **Compliance Limit** property enables you to specify an upper boundary on the **Compliance** value. • **Criterion**: This option is available when there is at least one upstream Static Structural system. The **Criterion** constraint enables you to evaluate relative displacements, such as the difference between the displacements of two nodes. And it enables you to make sure that the value of a certain criterion is above or below a

Category	Properties/Options/Description
	given boundary value or that it is within a given range. When selected, the following additional properties need to be specified: – **Criteria**: This property displays a drop-down list of available Primary Criterion and Composite Criterion objects evaluated in the upstream Static Structural analysis. – **Lower Bound**: Specify this value or set to **Free** (default). – **Upper Bound**: Specify this value or set to **Free** (default). **Note:** When you select a Criterion that is scoped to Remote Point or remote boundary condition (Remote Force or Remote Displacement), the **Base Result** property options **Reaction Force** and **Reaction Moment** are not supported for the Density Based and Lattice Optimization methods. **Suppressed** Include (**No**, default) or exclude (**Yes**) the response constraint. **Environment Selection** The application displays this property when you select the **Global Stress**, **Local von-Mises Stress**, **Natural Frequency**, **Displacement**, **Reaction Force**, or **Temperature** options for the **Response** property. The entry depends upon your upstream analysis type. Per the upstream system, the default entry is **All Structural**, **All Modal**, or **All Steady-State Thermal**. Also included in the drop-down list are the specific upstream systems. You can select from one of these systems to specify individual values for stress, frequency, etc. **Note:** If your Structural Optimization analysis includes multiple upstream analyses, any constraint that sets the **Environment Selection** property to **All Static Structural** or **All Steady State Thermal**, the application only applies the minimum number of steps as determined from the upstream analyses. That is, whichever upstream system has the least number of load steps specified, that is the value the application uses. Selecting a specific analysis from the property drop-down list applies the constraint for all load steps of the selected upstream analysis.

Category	Properties/Options/Description
Location and Ori-entation	When you specify the **Response** property as **Center of Gravity** or **Moment of Inertia**, the **Axis** property displays in order to specify a desired axis to constrain. Options include **X-Axis**, **Y-Axis**, and **Z-Axis**. In addition, for the **Moment of Inertia** option, a **Coordinate System** property displays so that you can specify the appropriate Cartesian coordinate system for the constraint.

[a] Ansys recommends that you use User Defined Criterion to define your criterion of interest and then specify the criterion in the Objective (p. 31) or **Response Constraint** object of your optimization analysis.

Refer to the Response Constraint object reference page for additional information.

2.8. Define Manufacturing Constraints

It is important to understand that a optimization solution could create unmanufacturable designs. As a result, any change to the manufacturing process due to an unintended design could undermine the integrity of the original design. Therefore, you (the designer), should apply and specify manufacturing constraints based on your manufacturing process. The **Manufacturing Constraint** condition, when applied to a optimization system, helps to alleviate design problems by enabling you to specify manufacturing limitations.

Jump to a section topic:

- Constraint Types (p. 48)
- Subtype Requirements and Restrictions (p. 49)
- Application (p. 50)
- Details Pane Properties (p. 50)

Constraint Types

The application supports manufacturing constraints, per method, as shown here.

Density Based	Level Set Based	Shape Optimization	Mixable Density
Member Size (Minimum and Maximum Thickness) **Pull Out Direction** (1-sided and 2-sided) **Extrusion**	**Member Size** (Minimum and Maximum Thickness and Minimum Gap) **Pull Out Direction** (1-sided, 2-sided, Stamping, and No-hole) **Am Overhang Constraint**	**Member Size** (Maximum Thickness) **Extrusion**	**Member Size** (Minimum and Maximum Thickness) **Pull Out Direction** (1-sided and 2-sided) **Extrusion**

Density Based	Level Set Based	Shape Optimization	Mixable Density
Am Overhang Constraint	**Housing** **Complexity Index**		**Am Overhang Constraint** **Housing**

Requirements and Restrictions

Review the following for the density based and level set methods.

Density Based and Mixable Density Methods

Note the following requirements and restrictions when you are using the **Topology Optimization - Density Based** or **Topology Optimization - Mixable Based** optimization method. The restrictions only apply when one of the manufacturing constraint types is scoped to an **Optimization Region** or if it has an overlapping region.

- If you specify an **AM Overhang Constraint** manufacturing constraint, in combination with:
  - **Symmetry**, the **Build Direction** must be in the symmetry plane.
  - **Cyclic Repetition**, the **Build Direction** must be parallel to the **Axis** specified for the **Cyclic Repetition**.
  - **Pattern Repetition**, the Build Direction must be perpendicular to the Axis specified for the **Pattern Repetition**.
- If you specify an **Extrusion** and a **Cyclic Repetition** Design Constraint, the axis of rotation of cyclic constraint must be in the same as the extrusion direction.
- If you specify an **Extrusion** and a **Symmetry** design constraint, the extrusion direction must be in the symmetry plane.
- If you specify a **Pull Out Direction** and a **Symmetry** design constraint, the pull out direction must be in the symmetry plane.
- If you specify the **Pattern Repetition** design constraint, in combination with:
  - **Extrusion**, the pattern direction must be perpendicular to the extrusion direction.
  - **Pull Out Direction**, the pull-out direction must be perpendicular to the pattern direction.
- The combination of the following constraint types is not supported:
  - **Extrusion** and **Uniform**.
  - **Pull Out Direction** and **Uniform**.
  - **AM Overhang Constraint** and **Uniform**.
  - **AM Overhang Constraint** and **Member Size**, **Extrusion**, or **Pull Out Direction**.

Level Set Based Method

Note the following requirements and restrictions when you are using the **Topology Optimization - Level Set Based** optimization method. The restrictions only apply when one of the manufacturing constraint types is scoped to an **Optimization Region** or if it has an overlapping region.

- When you specify a **Member Size** manufacturing constraint and set the **Minimum** property to **Manual**, the application performs two optimizations. The first one does not consider the manufacturing constraint in the solution calculation. However, if the constraint's **Minimum** value is exceeded at the end of this first run, then a second optimization run is executed using the constraint specifications. This logic makes sure that the optimization does not become trapped in an irrelevant local minimum.
- If you specify an **AM Overhang Constraint** manufacturing constraint, in combination with:
  - **Symmetry**, the **Build Direction** must be in the symmetry plane.
  - **Cyclic Repetition**, the **Build Direction** must be parallel to the **Axis** specified for the **Cyclic Repetition**.
  - **Pattern Repetition**, the Build Direction must be perpendicular to the Axis specified for the **Pattern Repetition**.
- If you specify a **Pull Out Direction** and a **Symmetry** design constraint, the pull out direction must be in the symmetry plane.
- If you specify the **Pattern Repetition** design constraint, in combination with **Pull Out Direction**, the pull-out direction must be perpendicular to the pattern direction.

Application

The analysis can include only one **Manufacturing Constraint** object.

1. To add the object, either look on the **Environment** Context tab and select **Manufacturing Constraint** > [manufacturing constraint type] or right-click the **Environment** object or within the **Geometry** window and select **Insert** > [manufacturing constraint type].
2. Based on the selected constraint type, specify **Details** properties as required.

Details Properties

The **Details** pane for this object includes the following properties.

Category	Fields/Options/Description
Scope	**Scoping Method**: Based upon the type of **Manufacturing Constraint** you have inserted into the tree, one or more of the following options is available for this property: • **Geometry Selection**: This option indicates that the design region is applied to a geometry or geometries, which are chosen using the graphical selection tools. When you specify **Geometry Selection** for the **Scoping Method**, the **Geometry** property displays.

<table>
<tr><th>Category</th><th>Fields/Options/Description</th></tr>
<tr><td></td><td>In this case, use selection filters on the Graphics Toolbar to pick your geometric entities (body and element selection only), and then click Apply. Once complete, the property displays the type of geometry (Body, Element, etc.) and the number of selected geometric entities (for example: 1 Body, 12 Elements).

• Named Selection: This option indicates that the design region is applied to a Named Selection. When you specify Named Selection for the Scoping Method, the Named Selection property displays. This property provides a drop-down list of available user-defined Named Selections (only body-based and element-based Named Selections are supported).

• Optimization Region: This option indicates that the design region is applied to the specified Optimization Region. When you select Optimization Region for the Scoping Method, the Optimization Region Selection property displays and is automatically set to Optimization Region (the only option).

• All Optimization Regions: When you have multiple Optimization Regions, this option indicates that the constraint is applied to all defined Optimization Regions.</td></tr>
<tr><td>Definition</td><td>Type: This is a read-only property that indicates the object as a Manufacturing Constraint.

Subtype: This is a read-only property that displays the type of Manufacturing Constraint you selected. Options include:

• Member Size: This subtype provides options to specify the minimum thickness, the maximum thickness, and/or a gap size of the connected parts in the final design.

• Pull Out Direction: This subtype is used for mold-based manufacturing processes. It enables you to specify the direction to remove the model from the mold in a manner that ensures the integrity of the model.

When selected, the Pull Out Option property displays. This property has the following options:

– None (default): The pull out direction is specified using location and orientation only.

– Stamping: Selecting this option instructs the application to create a design that is more compatible with a stamping or forging process. Like the design of a plate, the final design will not have any perforations along the pull-out direction.</td></tr>
</table>

Category	Fields/Options/Description
	– **No-Hole**: Selecting this option makes sure that the design will not have any perforations. It typically aims to ease the filling stage during the casting process. **Note:** – For the **Density Based** method only, if your analysis specifies a Tetrahedrons Mesh Method (SOLID187) and you are also defining a **Pull Out Direction**, Ansys recommends that you also include the **Manufacturing Constraint** > **Member Size**. And, you need to manually specify the **Minimum** property of the **Member Size** to at least four times the Tetrahedron element size. – You may experience an **Exclusion Region** specification that is not consistent (the shapes differ) with a **Pull Out Direction** constraint. For the **Topology Optimization – Level Set Based** method, the application automatically reshapes the **Exclusion Region** in order to comply with the pullout direction. • **Extrusion**: Using this manufacturing constraint type, you can make sure that the resulting cross section of your final design is kept constant along the selected plane. For each element of the **Optimization Region**, the application requires at least two corner nodes to lie on the **Axis** specified for the **Extrusion**. • The **AM Overhang Constraint** is used for additive printing. It creates an **Overhang Angle** constraint that uses the input of **Overhang Angle** and **Build Direction** to create self-supporting structures. A structure optimized using **AM Overhang Constraint** can then be 3D printed without (or with reduced) supports. If the application is not able to build supports for all exclusions, it creates as many as possible and issues a warning. **Note:** See the LPBF Simulation Guide for details about performing additive manufacturing simulations. **Important:** **For the Density-based Method** Note the following restrictions and requirements. The **AM Overhang Constraint**: – Can be specified only once in a Structural Optimization analysis.

Category	Fields/Options/Description
	– Cannot be used in combination with the Manufacturing Constraints **Member Size** (with **Maximum Member Size** defined), **Extrusion**, or **Pull Out Direction**. – If used with **Symmetry** Manufacturing Constraint, the Build Direction of the AM Overhang constraint must be in the symmetry plane. – If used with the **Cyclic Repetition** Design Constraint, the Build Direction of the AM Overhang constraint must be parallel to the **Axis** selection of the Cyclic Repetition constraint. • **Housing**: This manufacturing constraint enables you to create a watertight design that encloses a given set of faces. Topology optimization often generates designs that include holes and perforations. Using this manufacturing constraint, you can create a container to house a given liquid. Ansys recommends that you have at least three layers of elements above the selected faces. The surrounding edges of the selected faces will be automatically set as the **Exclusion Region**. • **Complexity Index**: Use the **Complexity Index** manufacturing constraint to control the complexity of optimized designs created during the process. Using the associated **Maximum Value** property to specify a limit for this qualitative criterion, which is based on a ratio between the volume and the model's perimeter, you can minimize the creation of overly complex structures. An entry value of `1` in the **Maximum Value** property forces the application to produce a bulky sphere-like shape. As you increase the value of this entry, you enable the optimization process to reduce materials in many free form ways, therefore making the manufacture of the design more difficult. An entry range of `2`-`10` is suggested.
Member Size	**Density Based Method** When the **Member Size** constraint type is selected for the **Topology Optimization - Density Based** method, the following associated properties display in the **Member Size** category of the **Details** pane. • **Minimum**: For the density based optimization method, the options include **Program Controlled** (default) and **Manual**. Using the **Program Controlled** setting, the application automatically sets the minimum size at 2.5 times the mesh element size. **Min Size** property: You display the property by setting the **Minimum** property to **Manual**. The application computes the default value using the mesh size of the generated mesh. This value can simplify the Structural Optimization solution run. The **Program Controlled** setting is applicable even when no **Member Size** is added to the Structural Optimization analysis.

Category	Fields/Options/Description
	• **Maximum**: The options include **Program Controlled** (default) and **Manual**. **Max Size** property: You display the property by setting the **Maximum** property to **Manual**. The application does not specify a default value for this property. This is a required entry when you wish to specify a manufacturing process constraint such as casting, extrusion of parts, etc. and when you wish to specify the maximum member size of connected parts in the final design. **Level Set Based & Mixable Density Methods** When the **Member Size** constraint type is selected for the **Topology Optimization - Level Set Based** or the **Topology Optimization - Mixable Density** methods, the following associated properties display in the **Member Size** category of the Details pane. • **Minimum**: Options include **Free** (default) and **Manual**. **Min Size** property: You display this property by setting the **Minimum** property to **Manual**. You use this property to manually enter a desired minimum member size length. • **Maximum** (not supported for mixable density method): Options include **Free** (default) and **Manual**. **Max Size** property: You display this property by setting the **Maximum** property to **Manual**. You use this property to manually enter a desired maximum member size length. **Note:** For the **Topology Optimization - Level Set Based** optimization method, the application automatically increases the value four mesh-element sizes if ever the limit is too small. • **Gap Size** (not supported for mixable density method): The purpose of the **Gap** property is to keep a minimum distance between members. The formulation of this constraint is based on an approximation that aims to limit the amount of material within multiple test regions (one per mesh node). Options include **Free** (default) and **Manual**.

Category	Fields/Options/Description
	Value property: You display this property by setting the **Gap Size** property to **Manual**. You use this property to manually enter a desired member size gap length. **Note:** For the **Topology Optimization - Level Set Based** optimization method, Ansys recommends that you: 1. Use the minimum gap constraint in combination with Maximum Member Size constraint and, 2. Specify a limit that is smaller than two times the limit for the Maximum Member Size (minGap 2*MaxThick).
Location and Orientation	When one of the following subtypes is selected, their associated properties display in the **Location and Orientation** category of the **Details** view. **Pull Out Direction** When this constraint type is selected, the following associated properties display: • **Coordinate System**: Specify the appropriate Cartesian coordinate system for material removal. • **Axis**: Specify the removal axis. Options include: **X-Axis**, **Y-Axis**, **Z-Axis**. • **Direction**: Specify the removal direction based on the above axis. Options include: **Along Axis**, **Opposite to Axis**, or **Both Direction**. The **Pull Out Direction** constraint satisfies the criteria that there is no concave shape inside of the die so that the part cannot be trapped. This makes sure that the die can be successfully separated from a part after forming. For the options **Along Axis** and **Opposite to Axis** only the direction of the coordinate system is relevant. For Density Based optimization, for the option **Both Directions** both the origin and axis selection of the coordinate system is important. The **Pullout Constraint** is applied from the normal plane (normal to the coordinate system axis selection) at the origin and along and opposite to the direction specified by the coordinate system axis. For Level Set Based optimization, also for **Both Directions**, only the direction is relevant.

Category	Fields/Options/Description
	Extrusion When this constraint type is selected, the following associated properties display: • **Coordinate System**: Specify the appropriate Cartesian coordinate system for the extrusion. • **Axis**: Specify the extrusion axis. Options include: **X-Axis**, **Y-Axis**, **Z-Axis**. **AM Overhang Constraint** When this constraint type is selected, the following associated properties display: • **Coordinate System**: You use this property to specify the appropriate Cartesian coordinate system for the overhang angle. By default, this property is set to the Global Coordinate System. You can specify a user-defined Coordinate System as desired. • **Build Direction**: You use this property to specify the direction that you would like the overhang constraint to be applied. Options include **+X Axis**, **+Y Axis**, **+Z Axis** (default), **-X Axis**, **-Y Axis**, and **-Z Axis**. • **Overhang Angle**: You use this property to specify the degree to which the constraint should be applied. The default setting is **45°**. **Note:** For the density based optimization method, the angle should be kept between **27°** and **60°**. **Housing** When this manufacturing constraint type is selected, the following associated properties display: • **Scoping Method**: Specify how you want to select the region, either using **Geometry Selection** or using a **Named Selection**. • **Geometry**: Select your geometry or mesh entities that you want to use to create a watertight cavity. Only node, edge, or face selections are supported. • **Named Selection**: Select a geometry-based or mesh-based Named Selection to be used to create a watertight cavity. Only Named Selections defined by nodes, edges, or faces are supported.

Refer to the Manufacturing Constraint object reference page for additional information.

2.9. Define Design Constraints

Like the options of the Manufacturing Constraint capability, the options of the **Design Constraint** capability, as listed below, enable you to impose specific design requirements.

- **Cyclic Repetition**: This option enables you to control how the sectors are repeated, at the required times, along the specified axis and yields a design that is symmetric with respect to an axis of rotation.
- **Symmetry**: This option enables you to enforce a design that is symmetric with respect to a user-defined plane.
- **Uniform**: This option enforces a design that has a constant Topology Density on all planes parallel to a user-defined plane.
- **Pattern Repetition**: This option enforces a design that has a repetitive pattern. The pattern is repeated along the specified axial plane with a user-defined offset value.

Jump to a section topic:

Supported Optimization Methods (p. 57)
Subtype Requirements and Restrictions for Density Based Method (p. 57)
Application (p. 58)
Details Pane Properties (p. 59)

Supported Optimization Methods

The **Design Constraint** options are supported for the following methods only:

Density Based	Lattice Optimization	Level Set Based	Mixable Density	Shape Optimization	Topography Optimization
Cyclic Repetition **Symmetry** **Uniform** **Pattern Repetition**	**Pattern Repetition**	**Cyclic Repetition** **Symmetry** **Pattern Repetition**	**Cyclic Repetition** **Symmetry** **Pattern Repetition**	**Cyclic Repetition** **Symmetry**	Not Applicable

Requirements and Restrictions

Note the following requirements and restrictions for design constraint types. The restrictions only apply when one of the types is scoped to an **Optimization Region** or if it has an overlapping region.

- Only one **Uniform** constraint can be specified per analysis.
- Only one **Cyclic Repetition** constraint can be specified per analysis.

- If you specify two **Symmetry** constraints, the symmetry planes must be perpendicular to one another.
- If you specify a **Symmetry** and a **Cyclic Repetition**, the given symmetry plane must be perpendicular to the axis of rotation.
- If you specify a **Symmetry** and a **Uniform**, the symmetry plane must be parallel to the plane of the uniform constraint.
- If you specify two **Pattern Repetition** constraints, the axis planes must be perpendicular to one another.
- The combination of the following subtypes is not supported:
  - **Cyclic Repetition** and **Uniform**
  - **Extrusion** and **Uniform**
  - **Pull Out Direction** and **Uniform**
- Note the following restrictions when you specify a **Manufacturing Constraint** and a **Design Constraint** with the **Subtype** combinations:
  - For **Symmetry** and **Extrusion**, the extrusion direction must be in the symmetry plane.
  - For **Symmetry** and **Pull Out Direction**, the pull out direction must be in the symmetry plane.
  - For **Cyclic Repetition** and **Extrusion**, the axis of rotation of cyclic constraint must be in the same as the extrusion direction.
  - For **Cyclic Repetition** and **Pull Out Direction**, the pull out direction and the cyclic axis of rotation must be the same.
- If you specify **Pattern Repetition**, in combination with:
  - **Symmetry**, the pattern direction must be in the symmetry plane.
  - **Cyclic Repetition**, the pattern direction must be the same as the rotation axis.
  - **Extrusion**, the pattern direction must be perpendicular to the extrusion direction.
  - **Pull Out Direction**, the pull-out direction must be perpendicular to the pattern direction.
  - **Uniform**, the plane of the uniform constraint must be perpendicular to the pattern direction.

Application

The analysis can include only one **Design Constraint** object.

1. To add the object, either look on the **Environment** Context tab and select **Design Constraint** > **[Subtype]** or right-click the **Environment** object or within the **Geometry** window and select **Insert** > **[Subtype]**.

2. Based on the selected **Subtype**, specify properties as required.

Details Properties

The **Details** pane for this object includes the following properties.

Category	Properties/Options/Description
Scope	**Scoping Method**: Based upon the type of **Design Constraint** you have inserted into the tree, one or more of the following options is available for this property: • **Geometry Selection**: This option indicates that the design region is applied to a geometry or geometries, which are chosen using the graphical selection tools. When you specify **Geometry Selection** for the **Scoping Method**, the **Geometry** property displays. In this case, use selection filters on the Graphics Toolbar to pick your geometric entities (body and element selection only), and then click **Apply**. Once complete, the property displays the type of geometry (Body, Element, etc.) and the number of selected geometric entities (for example: 1 Body, 12 Elements). • **Named Selection**: This option indicates that the design region is applied to a Named Selection. When you specify **Named Selection** for the **Scoping Method**, the **Named Selection** property displays. This property provides a drop-down list of available user-defined Named Selections (only body-based and element-based Named Selections are supported). • **Optimization Region**: This option indicates that the design region applied to the specified **Optimization Region**. When you select **Optimization Region** for the **Scoping Method**, the **Optimization Region Selection** property displays. • **All Optimization Regions**: When you have multiple Optimization Regions, this option indicates that the constraint is applied to all defined Optimization Regions.
Definition	**Type**: This is a read-only property that indicates the object as a **Design Constraint**. **Subtype**: This property is a read-only field and it displays the type of Design Constraint you selected from the **Design Constraint** drop-down menu. • **Cyclic Repetition** • **Symmetry** • **Uniform** • **Pattern Repetition**

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td>Location and Orientation</td><td>When a subtype is selected, their associated properties display in the Location and Orientation category of the Details:

Cyclic Repetition

When this subtype is selected, the following associated properties display:
Number of Sectors: This property specifies the appropriate number of sectors.
Coordinate System: Specify an appropriate Cartesian or Cylindrical coordinate system for the cyclic model.
Axis: Specify the appropriate axis. Options include: X-Axis, Y-Axis, Z-Axis. Only the Z-Axis option is supported for a Cylindrical coordinate system.
Symmetry

When this subtype is selected, the following associated properties display:
Coordinate System: Specify the appropriate Cartesian coordinate system for the symmetry model.
Axis: Specify the plane for the symmetry model. Options include: YZ Plane, XZ Plane, and XY Plane.
Uniform

When this subtype is selected, the following associated properties display:
Coordinate System: Specify the appropriate Cartesian coordinate system for the uniform constraint.
Axis: Specify the appropriate plane. Options include: YZ Plane, XZ Plane, and XY Plane.
Pattern Repetition

When this subtype is selected, the following associated properties display:
Coordinate System: Specify the appropriate Cartesian coordinate system for the repetitive pattern.
Axis: Specify the appropriate axis. Options include: X-Axis, Y-Axis, and Z-Axis.
Offset: Specify the appropriate offset after which the pattern is repeated.</td></tr>
</table>

2.10. Specify Results and Solve

The optimization analysis supports the following results. Refer to the individual sections for more information.

- Topology Density (p. 61) (inserted automatically)
- Topology Elemental Density (p. 65)
- Structural Results (p. 68): Supported structural results include Deformation, Stress, and Strain results. To evaluate these results, you must create solution data using the **Export Design Properties** property. See the Specify Analysis Settings (p. 6) section for more information about the settings of the **Export Design Properties** property.
- Thermal Results (p. 71): Supported thermal results include Temperature and Heat Flux (Directional and Total). To evaluate these results, you must create solution data using the **Export Design Properties** property. See the Specify Analysis Settings (p. 6) section for more information about the settings of the **Export Design Properties** property.
- User Defined Results: User Defined Results are available using the context (right-click) option on the **Worksheet** for the Available Solution Quantities.
- Topology Density Tracker Result Plot Tracker (inserted automatically)

Note:

The Mechanical APDL solver supports the use of the Commands (APDL) object for optimization analyses. This support includes all iterations of the optimization analysis and for all load steps.

Important:

If you are using the Remote Solve Manager (RSM) for your solution, the density trackers do not update during the solution process. In order to see an optimization update for your model, you need to select the tracker object, right-click, and select the option **Evaluate All Results**. This action tells the remote machine to read the appropriate local file and display the current results.

2.10.1. Topology Density

The **Topology Density** result produces nodal averaged results. Go directly to a section topic using the following links:

- Application (p. 62)
- Display Limitations (p. 62)
- Result Smoothing (p. 62)
- Specify Properties (p. 62)

- Shared Topology Display Issue (p. 65)

Application

One **Topology Density** object is added automatically to the optimization analysis system. You can add additional objects by selecting **Topology Density** from the **Results** group on the **Solution** Context tab or by right-clicking the **Solution** folder (or in the **Geometry** window) and selecting **Insert**>**Topology Density**.

Note:

You can further analyze your optimized model, through continued simulation or by performing a design validation by exporting your results and making them available to a new downstream system.

The Solution object property **Export Topology (STL file)** enables you to automatically export your results in Standard Tessellation Language (STL) and in Part Manager Database (PMDB) file format, archive the files in zip file format, and then place the zipped file in the Solver Files Directory. This option is set to **Yes** by default.

In order to make the optimized results available to a downstream system, you need to create the new system on the Workbench Project Schematic and link the **Results** cell of your **Structural Optimization** analysis to the **Geometry** cell of a new downstream system, either a Geometry component system or the Geometry cell of another analysis system. Refer to the Design Validation (p. 82) section for additional details about this process.

Display Limitation

This result type does not support the display options available from the Geometry drop-down menu on the **Result** Context tab and that include the following views: Exterior, IsoSurfaces, Capped IsoSurfaces, and Section Planes.

Result Smoothing

The **Topology Density** result offers the **Results** group option Smoothing from the Solution Context Tab. You can also insert a **Smoothing** object using the context (right-click) menu options **Insert** > **Smoothing**. This result generates an STL (Stereolithography) file based on the **Topology Density** result that you can need modify to move nodes of the geometry to refine your part and as desired, save for use in downstream validation systems. Multiple Smoothing objects can be added for each **Topology Density** result.

Note:

Smoothing is not supported on the Linux platform.

Specify Properties

Using the properties of the **Details** pane, define results.

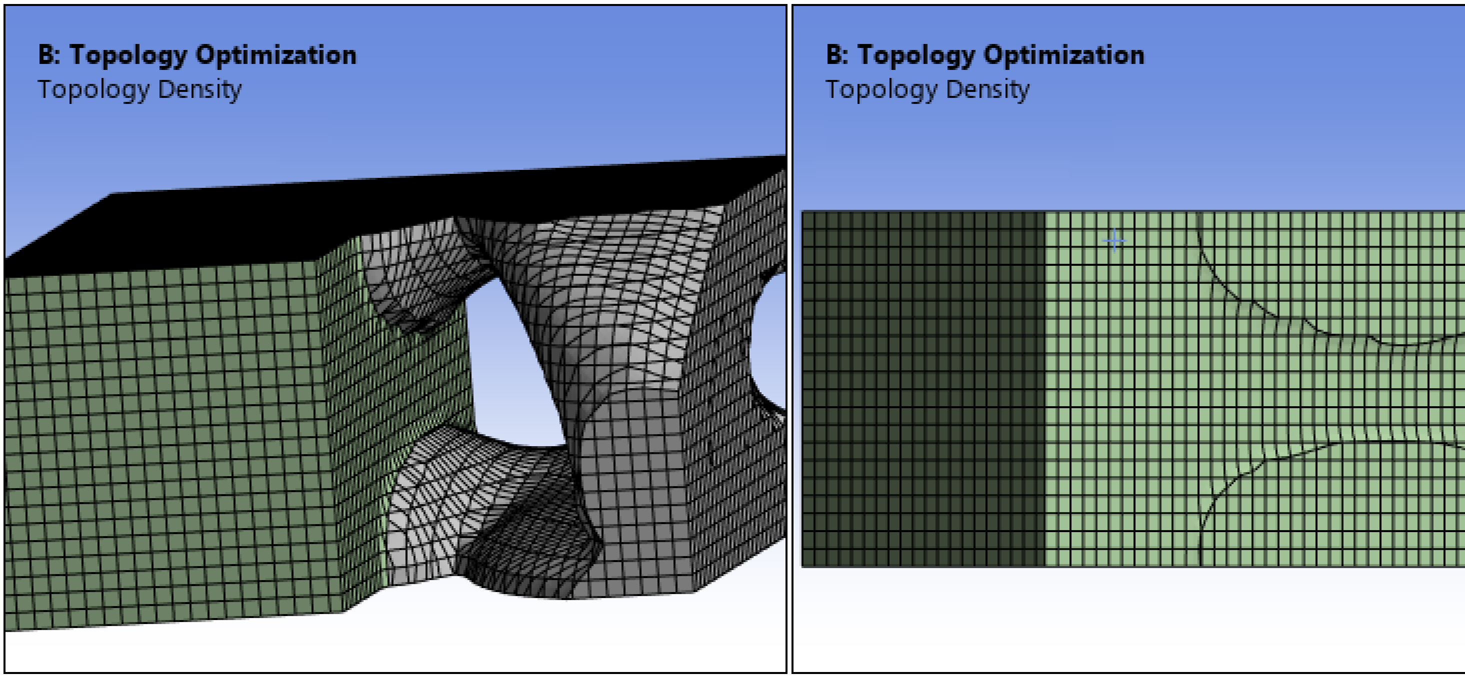
Category	Properties/Options/Description
Scope	**Scoping Method**. The options for this property include: • **Optimization Region** (default): This option indicates that the design region is applied to the specified Optimization Region. When you select **Optimization Region** for the **Scoping Method**, the **Optimization Region** property displays. • **Geometry Selection**: This option indicates that the design region is applied to a geometry or geometries, which are chosen using the graphical selection tools. When you specify **Geometry Selection** for the **Scoping Method**, the **Geometry** property displays. In this case, use selection filters on the Graphics Toolbar to pick your geometric entities (body and element selection only), and then click **Apply**. Once complete, the property displays the type of geometry (Body, Element, etc.) and the number of selected geometric entities (for example: 1 Body, 12 Elements). • **Named Selection**: This option indicates that the design region is applied to a Named Selection. When you specify **Named Selection** for the **Scoping Method**, the **Named Selection** property displays. This property provides a drop-down list of available user-defined Named Selections (only body-based and element-based Named Selections are supported).
Defini-tion	**Type**: Read-only field that describes the object - **Topology Density**. **By**: Read-only field that displays "Iteration". **Iteration**: The default setting is **Last**. You can specify an iteration number to obtain results for the specified iteration (displayed in the **Result** category). **Note:** The animation of Topology Density results occurs over all iterations for which the intermediate results are computed as well as saved during solution. The intermediate results are computed based on the setting of the **Store Results At** property of the Output Controls (**Analysis Settings** object) and the intermediate results are saved to disk based on the setting of the **Max Num of Intermediate Files** property. **Retained Threshold**: This property is controlled by a slider that represents the range from minimum to maximum for the result. The default value is `0.5`. The supported range is `0.01` to `0.99` (greater than zero and less than 1). Once you evaluate the result, use the slider to view the optimized topology in the graphics view. The application computes and displays the values for the Original Volume, Final Volume, Percent Volume of Original, Original Mass, Final Mass, and Percent Mass of Original properties.

Category	Properties/Options/Description
	Exclusions Participation: **Yes** (default) or **No**. When set to **Yes**, the application uses the excluded elements to compute the Original Volume, Final Volume, Percent Volume of Original, Original Mass, Final Mass, and Percent Mass of Original properties. When set to **No**, excluded elements are not considered. **Suppressed**: Include (**No**, default) or exclude (**Yes**) the result.
Results	**Minimum**: Read-only field that displays minimum density value of the result. **Maximum**: Read-only field that displays maximum density value of the result. **Original Volume**: Read-only field that displays, per scoping, the computed original volume. **Final Volume**: Read-only field that displays, per scoping and the Retained Threshold setting, the optimized volume. **Percent Volume of Original**: Read-only field that displays the value of the **Final Volume** divided by the **Original Volume**. **Original Mass**: Read-only field that displays, per scoping, the computed original mass. **Final Mass**: Read-only field that displays, per scoping and the Retained Threshold setting, the optimized mass. **Percent Mass of Original**: Read-only field that displays the value of the **Final Mass** divided by the **Original Mass**. **Note:** If elements are excluded from the optimization, then volume/mass contribution from those elements will not be included in the **Original Volume**/**Original Mass** computation.
Visibility	**Show Optimized Region**: This property is used to control graphical view changes only. The options for this property include: • **All Regions**: This option displays all of the regions selected by the **Scoping Method**. Three color bands are shown: Remove (Red), Marginal (Yellow), and Keep (Gray). Remove indicates a **Retained Threshold** value of 0 to 0.4, Marginal indicates a value of 0.4 to 0.6, and Keep indicates a value greater than 0.6. • **Retained Region** (default): When **Retained Region** is selected, then the **Retained Threshold** value is used from the details view to show the region which must be kept. • **Removed Region**: This option displays what will be removed.

Category	Properties/Options/Description
Informa-tion	**Iteration Number**: Read-only field that displays the converged iteration step number.

Shared Topology Display Issue

If you are using the **Density Based**, **Level Set**, or **Mixable Density** optimization methods, there is a known graphical display issue when you optimize a body that is connected to another unoptimized body using Shared Topology. The unoptimized body may display modifications that do not actually exist, as illustrated below.



Because the optimized and non-optimized bodies share nodes at the contact region, the non-optimized body appears modified at the contact surfaces. However, no actual change has taken place to the non-optimized body.

2.10.2. Topology Elemental Density

The **Topology Elemental Density** result produces element-based result values. Go directly to a section topic using the following links:

- Application (p. 66)
- Display Limitations (p. 66)
- Specify Properties (p. 66)

Application

You can add the object (or objects) by selecting **Topology Elemental Density** from the Results group on the **Solution** Context tab or by right-clicking the **Solution** folder (or in the Geometry window) and selecting **Insert** > **Topology Elemental Density**.

Note:

You can further analyze your optimized model, through continued simulation or by performing a design validation by exporting your results and making them available to a new downstream system.

The Solution object property **Export Topology (STL file)** enables you to automatically export your results in Standard Tessellation Language (STL) and in Part Manager Database (PMDB) file format, archive the files in zip file format, and then place the zipped file in the Solver Files Directory. This option is set to **Yes** by default.

In order to make the optimized results available to a downstream system, you need to create the new system on the Workbench Project Schematic and link the **Results** cell of your **Structural Optimization** analysis to the **Geometry** cell of a new downstream system, either a Geometry component system or the Geometry cell of another analysis system. Refer to the Design Validation (p. 82) section for additional details about this process.

Display Limitation

This result type does not support the following display options available from the **Geometry** drop-down menu on the **Result** Context tab: Exterior, IsoSurfaces, Capped IsoSurfaces, and Section Planes.

Specify Properties

Using the properties of the **Details** pane, define results.

Category	Properties/Options/Description
Scope	**Scoping Method**. The options for this property include: • **Optimization Region** (default): This option indicates that the design region is applied to the specified Optimization Region. When you select **Optimization Region** for the **Scoping Method**, the **Optimization Region** property displays. • **Geometry Selection**: This option indicates that the design region is applied to a geometry or geometries, which are chosen using the graphical selection tools. When you specify **Geometry Selection** for the **Scoping Method**, the **Geometry** property displays. In this case, use selection filters on the Graphics Toolbar to pick your geometric entities (body and element selection only), and then click **Apply**. Once complete, the property displays the type of geometry (Body, Element, etc.) and the number of selected geometric entities (for example: 1 Body, 12 Elements).

Category	Properties/Options/Description
	• **Named Selection**: This option indicates that the design region is applied to a Named Selection. When you specify **Named Selection** for the **Scoping Method**, the **Named Selection** property displays. This property provides a drop-down list of available user-defined Named Selections (only body-based and element-based Named Selections are supported).
Definition	**Type**: Read-only field that describes the object - **Topology Elemental Density**. **By**: Read-only field that displays "Iteration." **Iteration**: The default setting is **Last**. You can specify an iteration number to obtain results for the specified iteration (displayed in the **Result** category). **Note:** The animation of Topology Density results occurs over all iterations for which the intermediate results are computed as well as saved during solution. The intermediate results are computed based on the setting of the **Store Results At** property of the Output Controls (**Analysis Settings** object) and the intermediate results are saved to disk based on the setting of the **Max Num of Intermediate Files** property. **Retained Threshold**: This property is controlled by a slider that represents the range from minimum to maximum for the result. The default value is `0.5`. The supported range is `0.01` to `0.99` (greater than zero and less than 1). Once you evaluate the result, use the slider to view the optimized topology in the graphics view. The application computes and displays the values for the Original Volume, Final Volume, Percent Volume of Original, Original Mass, Final Mass, and Percent Mass of Original properties. **Exclusions Participation**: **Yes** (default) or **No**. When set to **Yes**, the application uses the excluded elements to compute the Original Volume, Final Volume, Percent Volume of Original, Original Mass, Final Mass, and Percent Mass of Original properties. When set to **No**, excluded elements are not considered. **Suppressed**: **Yes** or **No** (default).
Results	**Minimum**: Read-only field that displays minimum value of the result. **Maximum**: Read-only field that displays maximum value of the result. **Original Volume**: Read-only field that displays, per scoping, the computed original volume. **Final Volume**: Read-only field that displays, per scoping and the Retained Threshold setting, the optimized volume. **Percent Volume of Original**: Read-only field that displays the value of the **Final Volume** divided by the **Original Volume**.

Category	Properties/Options/Description
	Original Mass: Read-only field that displays, per scoping, the computed original mass. **Final Mass**: Read-only field that displays, per scoping and the Retained Threshold setting, the optimized mass. **Percent Mass of Original**: Read-only field that displays the value of the **Final Mass** divided by the **Original Mass**.
Visibility	**Show Optimized Region**: This property is only used to control graphical view changes. The options for this property include: • **All Regions**: This option displays all of the regions selected by the Scoping Method. Three color bands display: Remove (Red), Marginal (Yellow), and Keep (Gray). Remove indicates a Retained Threshold value of 0 to 0.4, Marginal value indicates a value of 0.4 to 0.6, and Keep value indicates a value greater than 0.6. • **Retained Region** (default): When Retained region is selected, then the Retained Threshold value is used from the details view to show the region which must be kept. • **Removed Region**: This option displays what will be removed.
Information	**Iteration Number**: Read-only field that displays the converged iteration step number.

2.10.3. Structural Results

The **Structural Optimization** analysis supports certain structural results, including Deformation, Stress, and Strain, etc.

Important:

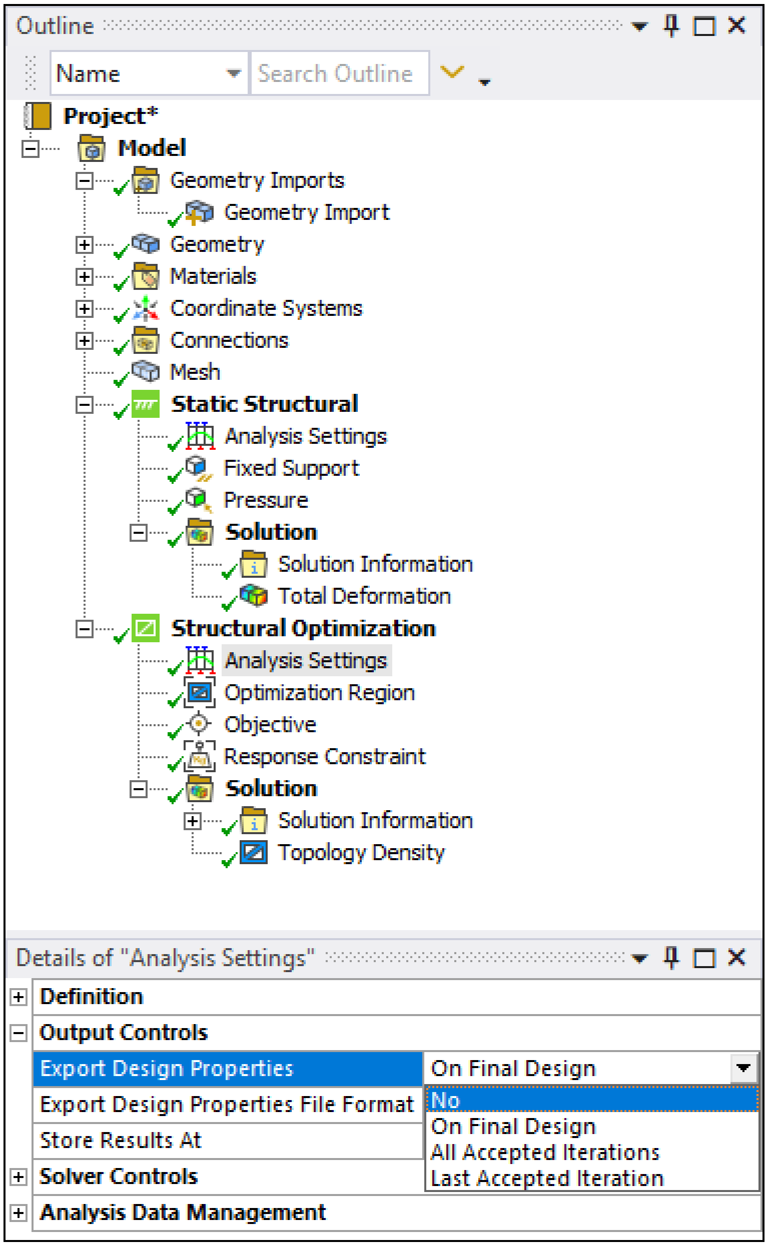
Stress- and **strain**-based results are not supported for the **Topology Optimization - Density Based** method.

Note:

Optimized structural results are displayed on a triangle-based surface-mesh.

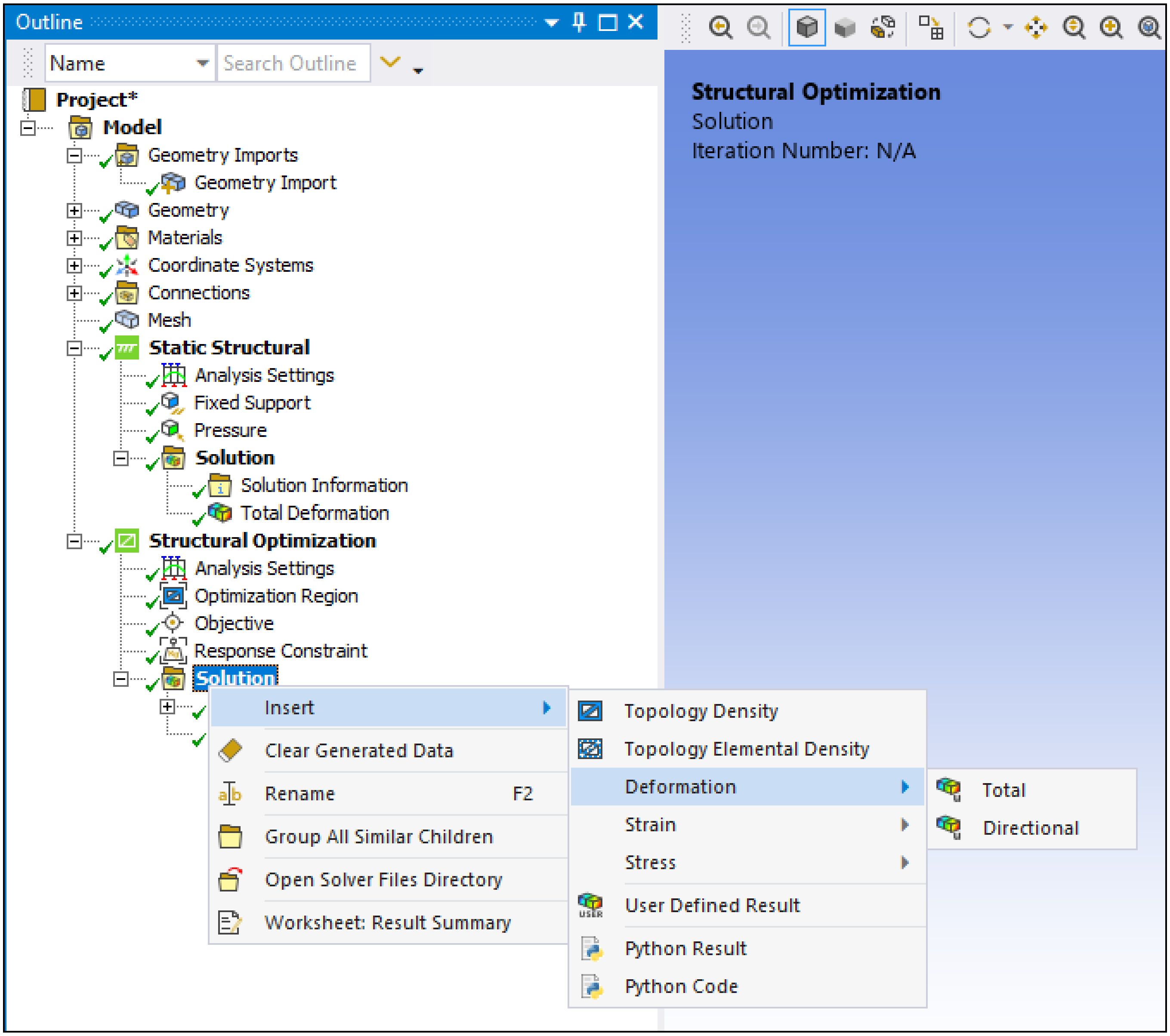
Application

1. Set the **Export Design Properties** property (shown below) to a desired option. This property is included in the Output Controls category of the **Analysis Settings** object. See the Specify Analysis Settings (p. 6) section for a description of the options based on the selected optimization method. This property is available for all optimization methods.

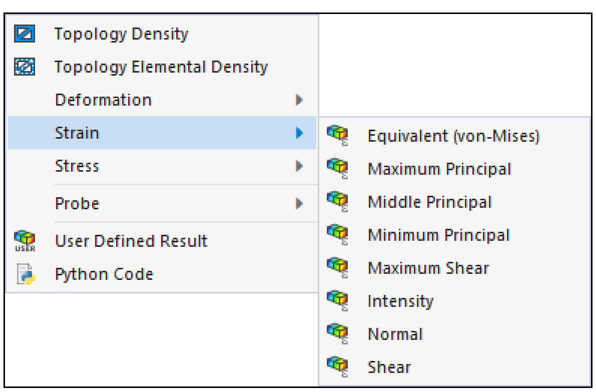


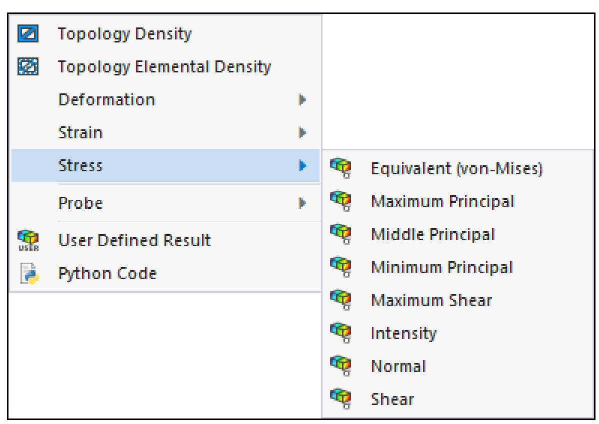
2. Insert results from the appropriate menu (Deformation, Stress, or Strain) on the Solution tab or using the context (right-click) menu options on the **Solution** object.

Deformation Result Menu



Stress and Strain Result Menus





Once you have specified a result, you specify the properties as you normally would (see results object reference), except there are the following properties with specific requirements for **Structural Optimization**:

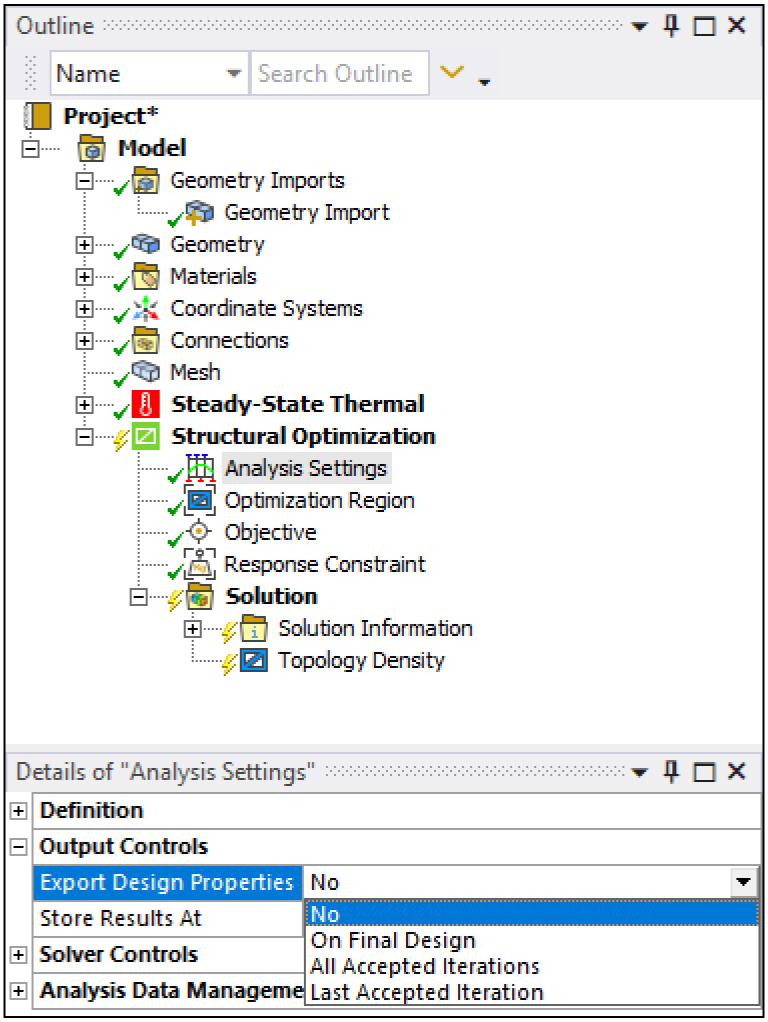
- **Environment Selection**: You use this property to select the upstream system from which the result obtains data.
- **Set Number**: This option displays the contour result for a given **Result Set** contained in the result file. By default, this value is the last set. If only one set is available, then that is the specified **Result Set**. For a solution that includes load steps and modes, you specify the desired set using the **Set Number** property (see Result Set Listing).
- **Iteration**: This property displays the result set from which the result was obtained. Contours for the result are based on the data of the iterations. By default, this value is the last iteration. If only one iteration is available, then that is the specified **Iteration**. To obtain other iterations, set the **Export Design Properties** property to **All Accepted Iterations**.

2.10.4. Thermal Results

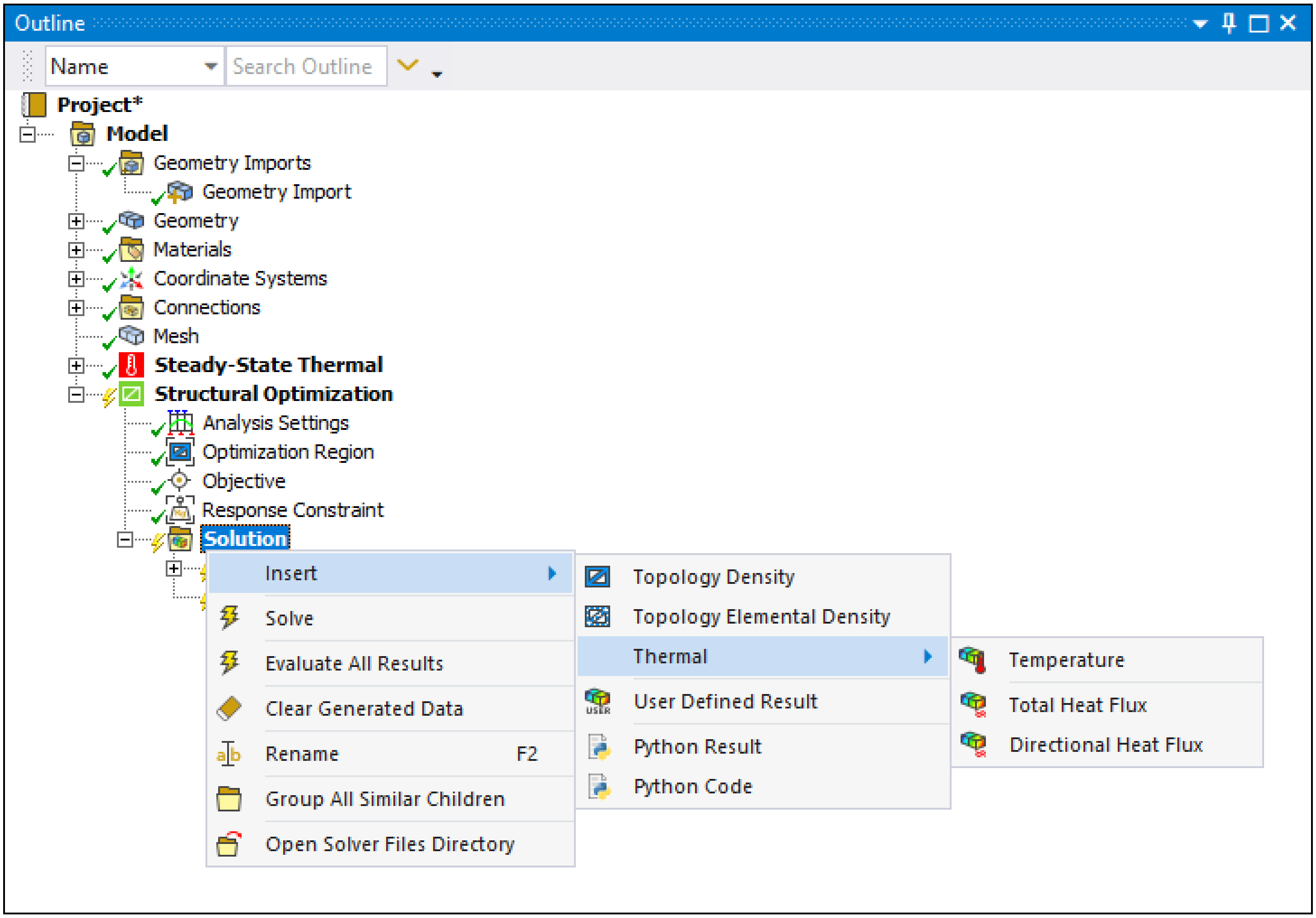
The **Structural Optimization** analysis supports certain thermal results, including Temperature, Directional Heat Flux, and Total Heat Flux.

Application

1. Set the **Export Design Properties** property (shown below) to a desired option. This property is included in the Output Controls category of the **Analysis Settings** object. See the Specify Analysis Settings (p. 6) section for a description of the options based on the selected optimization methods. This property is available for all optimization methods.
2. To evaluate thermal results available, you need to create solution data using the options the **Export Design Properties** property (shown below) of the Output Controls category of the **Analysis Settings** object. See the Specify Analysis Settings (p. 6) section for a description of the options based on the selected optimization methods. This property is available for all optimization methods.



Once specified, you can create temperature and heat flux results. Insert results from the **Thermal** menu on the Solution tab or using the using the context (right-click) menu options of the **Solution** object.



Once you have specified a result, you specify the properties as you normally would (see results object reference), except there are the following properties with specific requirements for **Structural Optimization**:

- **Environment Selection**: You use this property to select the upstream system from which the result obtains data.
- **Set Number**: This option displays the contour result for a given **Result Set** contained in the result file. By default, this value is the last set. If only one set is available, then that is the specified **Result Set**. For a solution that includes load steps and modes, you specify the desired set using the **Set Number** property (see Result Set Listing).
- **Iteration**: This property displays the result set from which the result was obtained. Contours for the result are based on the data of the iterations. By default, this value is the last iteration. If only one iteration is available, then that is the specified **Iteration**. To obtain other iterations, set the **Export Design Properties** property to **All Accepted Iterations**.

2.11. Post Processing

Once your analysis is prepared and you are ready to begin the solution process, the application enables you to view the progress of different solution elements, including response convergence charts, using the output features of the Solution Information object.

Go directly to a section topic using the following links:

- Interrupting the Solution (p. 74)
- Stopping the Solution (p. 75)
- Reviewing Results (p. 75)
- Topology Density Tracker Display Feature (p. 75)

Important:

- If your upstream system is a single Static Structural analysis, it is recommended that you use step-based loading to improve scalability. In order to do so, you need to define your loading conditions using the **Tabular Data** window and you need to set the **Independent Variable** property to the **Step** option. This does not include the use of the Thermal Condition load.
- If you are using the Remote Solve Manager (RSM) for your solution, the density trackers do not automatically update during the solution process. In order to see an optimization update for your model, you need to select the tracker object, right-click, and select the option **Evaluate All Results**. This action tells the remote machine to read the appropriate local file and display the current results.
- For a Modal analysis: your Structural Optimization analysis will terminate prior to completion if every iteration of the optimization run is not able to extract the maximum number of modes specified from the modal solver.
- The solver unit system specified in the analysis settings of the Static Structural or Modal analysis needs to match the unit system specified in the Structural Optimization analysis. If not, the optimization run presents a unit system mismatch error.

For a general overview of the solution process, see the Solve section of the Help.

Interrupting the Solution

After you have started the solution process, you can interrupt the solution using the **Interrupt Solution** button on the solve dialog box. The state of the optimization system will change and you will receive a green check mark on the dialog box, even if the solution is not converged completely. The following message will display:

> "The solution ran for *n* iterations and aborted as you requested. Examine the convergence plots to determine if this is an acceptable solution."

Stopping the Solution

You can stop the solution process using the **Stop Solution** button on the solve dialog box. The application immediately halts the solution, does not write results data and issues the following message:

"The solution process was aborted as you requested."

Reviewing Results

For the Density Based optimization type, you can increase and decrease the boundary of the shape displayed by the tracker using the **Retained Threshold** property. This property provides a scalable slider feature to increase and decrease the displayed threshold of the removed/retained elements on the model.

For the Level Set Based optimization type, the use of the **Retained Threshold** property is not relevant. The shape is clearly and unambiguously defined, removing the need for any interpretation.

See the Topology Density (p. 61) and the Topology Elemental Density (p. 65) sections for descriptions of the purpose and use of the results specific to a the analysis.

Note:

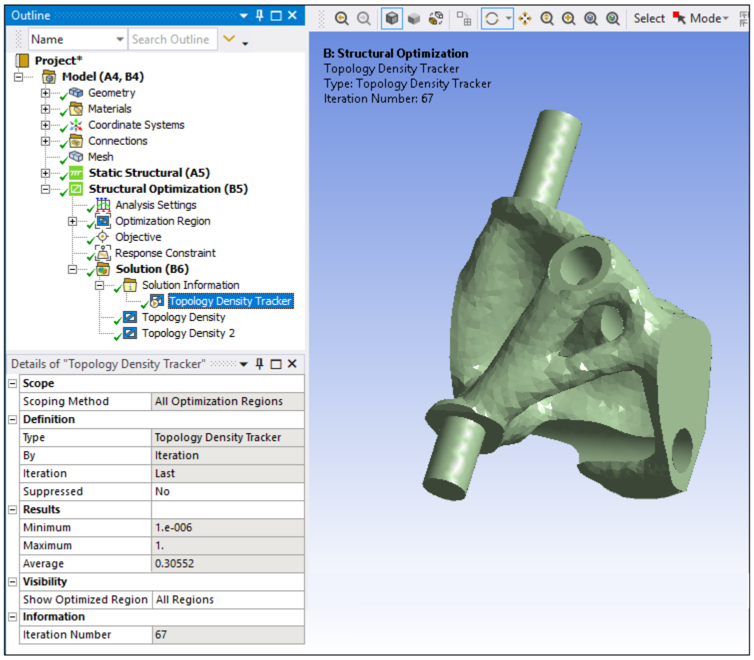
You can further analyze your optimized model, through continued simulation or by performing a design validation by exporting your results and making them available to a new downstream system.

The Solution object property **Export Topology (STL file)** enables you to automatically export your results in Standard Tessellation Language (STL) and in Part Manager Database (PMDB) file format, archive the files in zip file format, and then place the zipped file in the Solver Files Directory. This option is set to **Yes** by default.

In order to make the optimized results available to a downstream system, you need to create the new system on the Workbench Project Schematic and link the **Results** cell of your **Structural Optimization** analysis to the **Geometry** cell of a new downstream system, either a Geometry component system or the Geometry cell of another analysis system. Refer to the Design Validation (p. 82) section for additional details about this process.

Topology Density Tracker Display Feature

In addition, you can open the **Solution Information** object and select its child object, the **Topology Density Tracker** object (inserted automatically), as illustrated below.

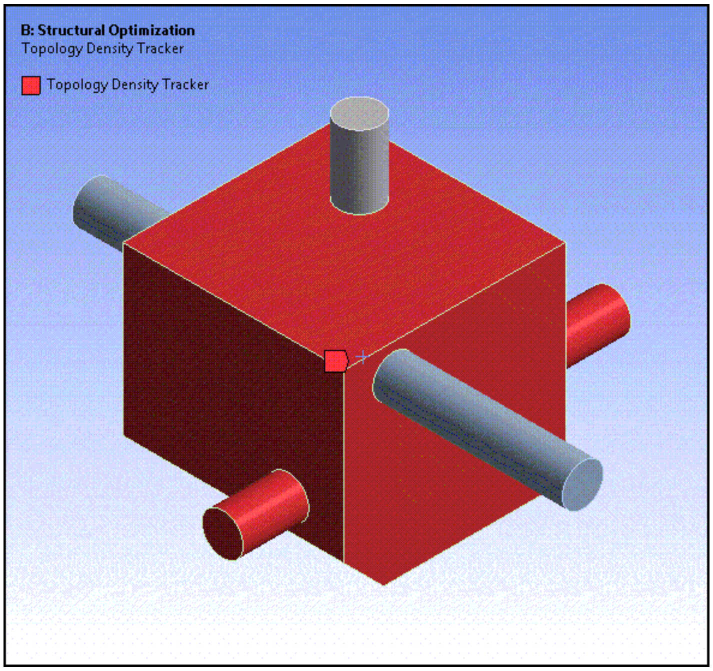


You can select this object to view the optimization of the model during the solution. The content of the display is determined by the setting of the **Show Optimized Region** property. For the density-based optimization method, the options include: **Retained Region** (default), **Removed Region**, and **All Regions**. Note that the **All Regions** option displays only color changes on the model.

Note the following behaviors of the tracker and its properties:

- The display is only available when the Topology Density Tracker object is selected during the solution.
- If the **Topology Density Tracker** object is not selected during the solution process, by default, the application displays the result of the final iteration.
- The **Iteration Number** property of the Topology Density Tracker object indicates the iteration number of the result currently displayed in the Geometry window based on the setting of the **Update Interval** property of the **Solution Information** object.

An example solution for a **Topology Density Tracker** is shown in the following animation.



You can also insert and select a **Topology Elemental Density Tracker** (p. 65) object under the Solution Information object to view elemental optimization of the model during the solution.

2.12. Recreating CAD Geometry

You can insert a downstream **Geometry** system in order to transfer your faceted geometry to Discovery Modeling to recreate your CAD geometry using the reverse engineering feature.

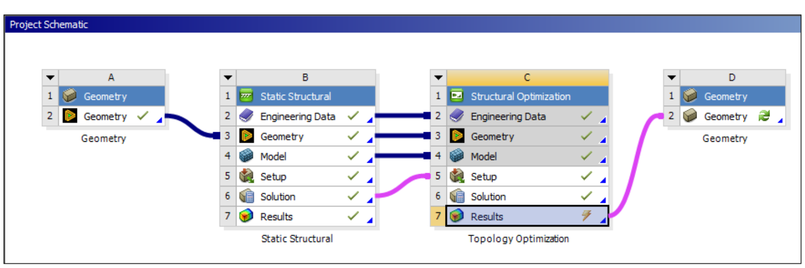
Important:

The capability to recreate a downstream geometry is not supported for 2D plane, shell, and multi-body geometries.

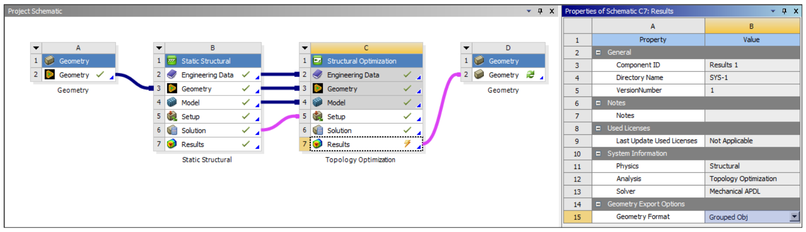
Application

This workflow assumes that you have a working knowledge of **Workbench** and its component systems. To create a new geometry from your optimized analysis:

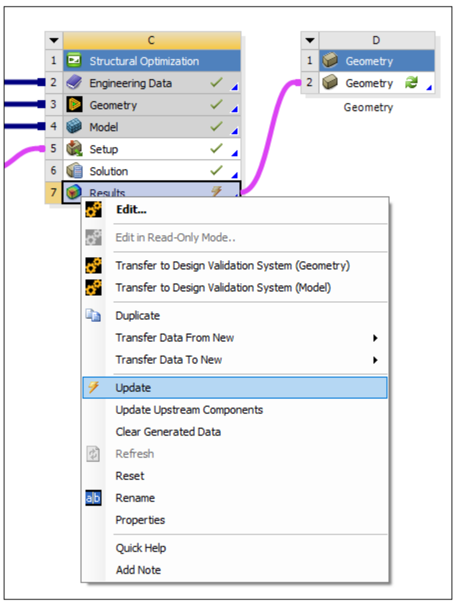
1. Place a new **Geometry** system on the Project Schematic as illustrated and link the **Results** cell of the **Structural Optimization** system to the **Geometry** cell of your new system.

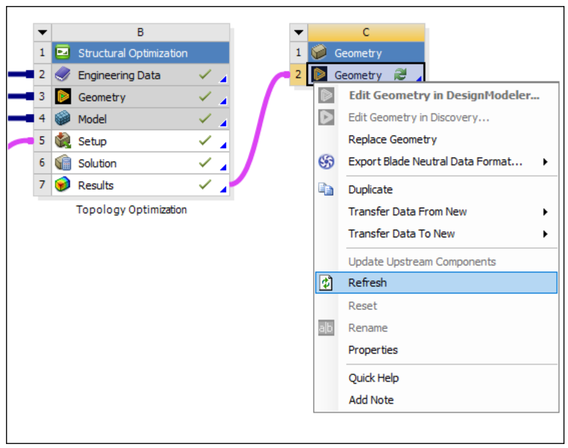


2. Select the **Results** cell of the **Structural Optimization** system. In the properties pane, set the **Geometry Format** property to **Grouped Obj** (default).

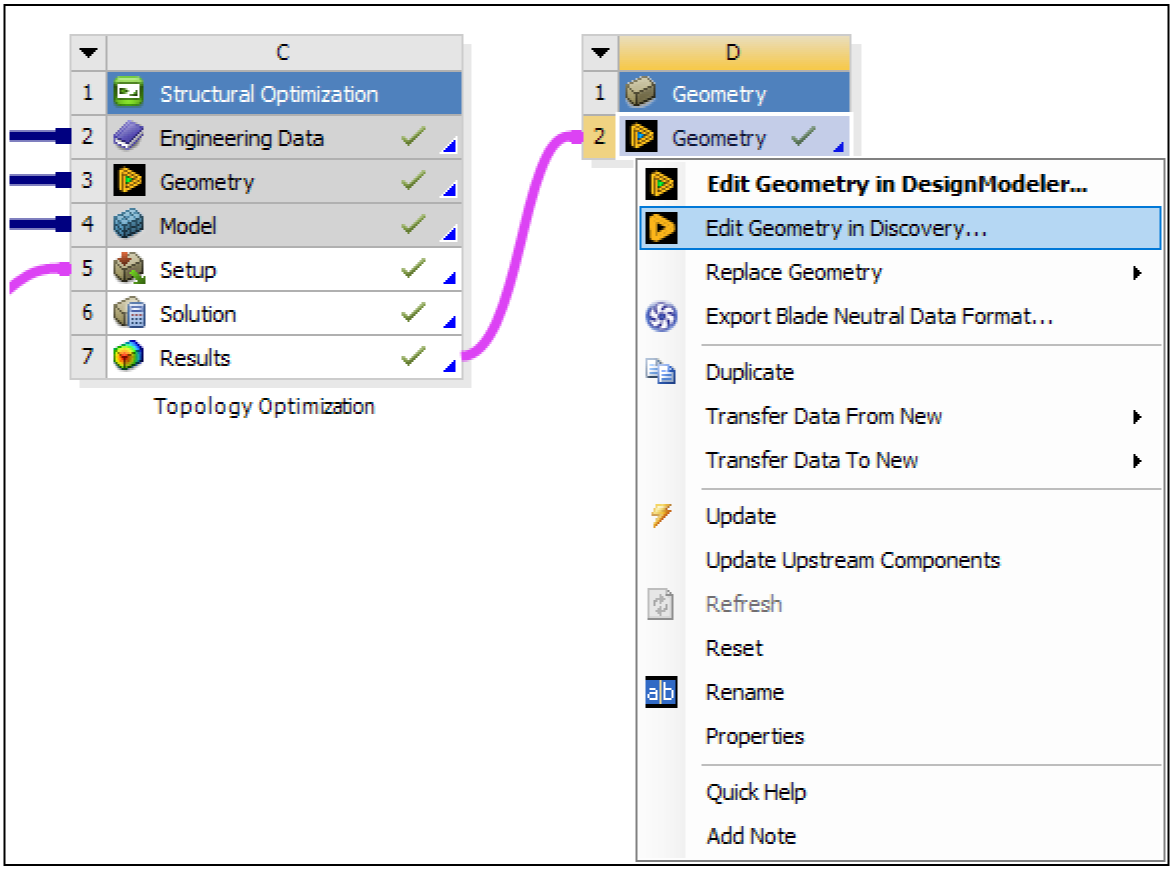


3. Right-click the **Results** cell of the **Structural Optimization** system and select **Update**. Once complete, refresh the **Geometry** cell of the new system.





4. Open the geometry in Discovery Modeling and use the reverse engineering feature to specify your geometry.

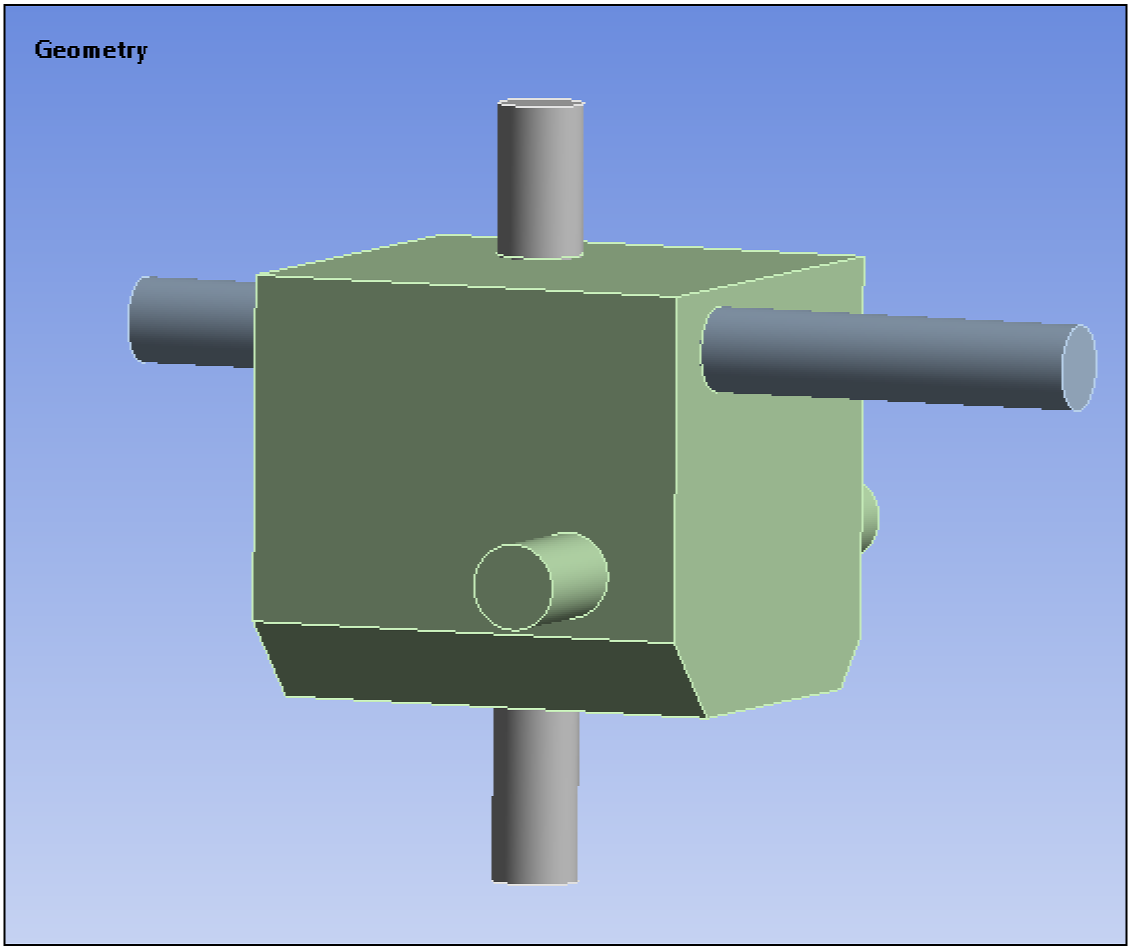


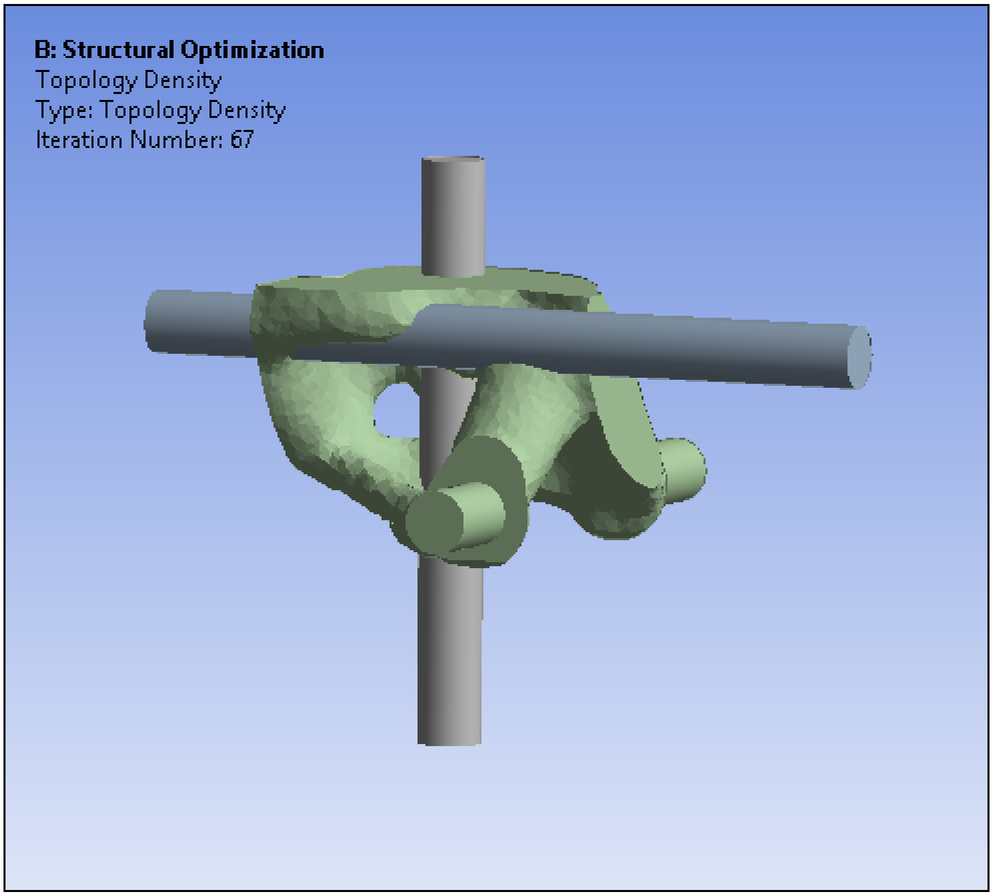
Note:

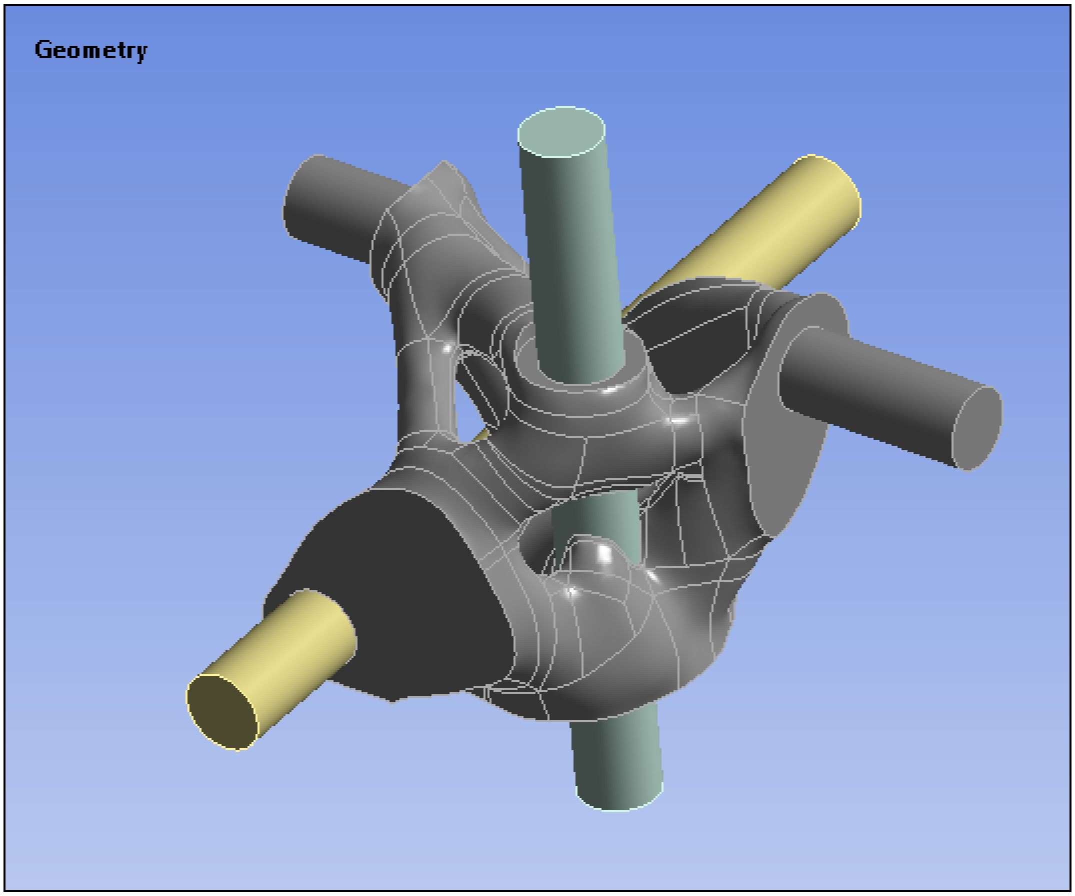
For a **Topology Result** (p. 61), you can create and export an STL file by 1) using the **Export Topology (STL file)** property on the solution object, or 2) using a Smoothing object. An STL file generated from a **Smoothing** object is specifically designed for the reverse engineering capability.

Example

The following sequence shows an example of the original geometry, the optimized geometry, and the recreated geometry.


Geometry

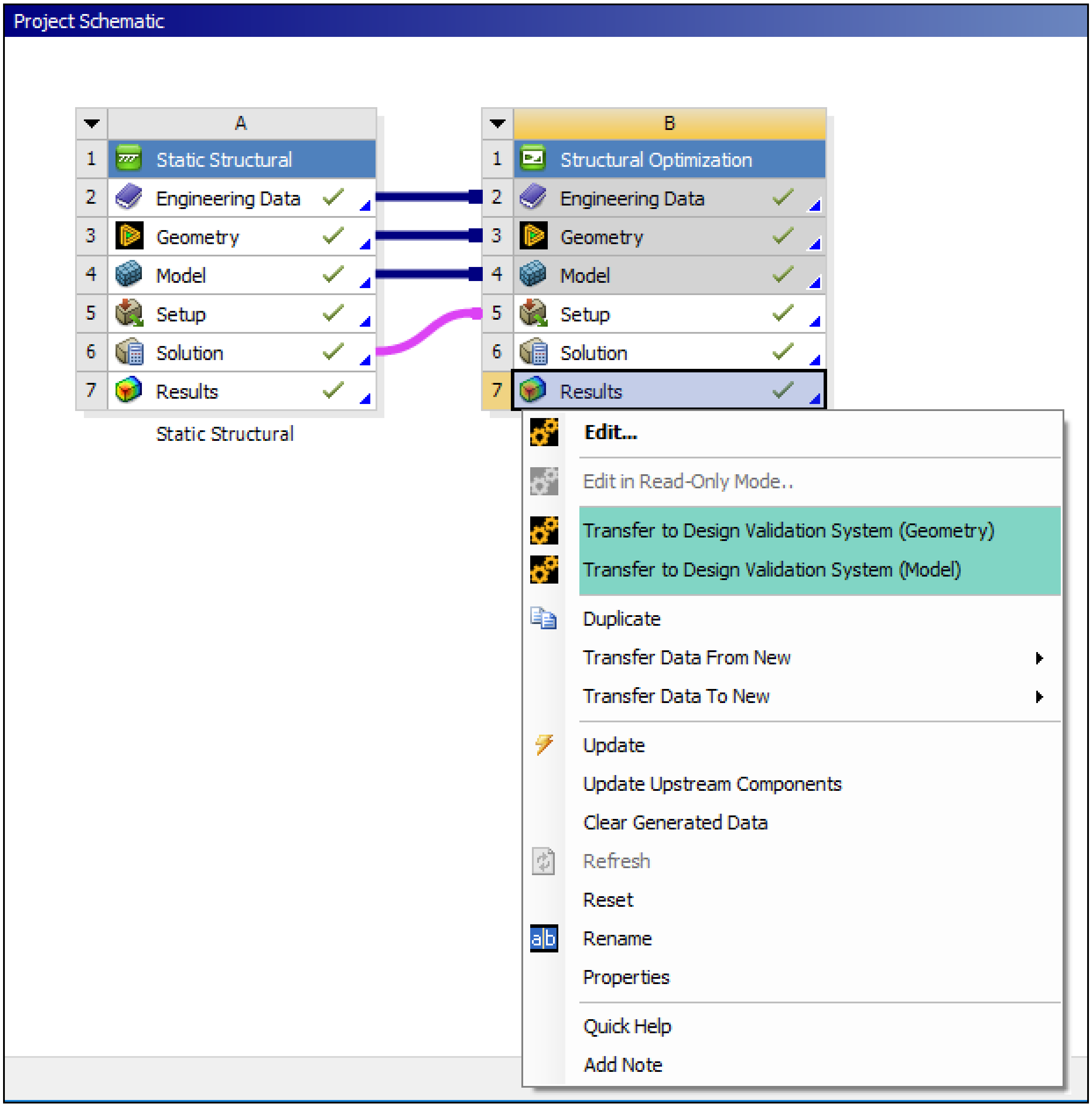

B: Structural Optimization
Topology Density
Type: Topology Density
Iteration Number: 67



2.13. Performing Design Validation

Once you have completed your optimization analysis, you can validate your optimized design in a downstream analysis system. In order to perform a validation, your optimization analysis must be in a solved state. Two validation options are available:

- **Transfer to Design Validation System (Geometry)** (p. 83): This option enables you to first revise your geometry in a CAD application. No environmental conditions are transferred to the downstream system.
- **Transfer to Design Validation System (Model)** (p. 87): This option enables you to transfer scoping downstream.



2.13.1. Geometry Validation

Design validation using the **Transfer to Design Validation System (Geometry)** option transfers data to the downstream system's **Geometry** cell. The benefit of this option is that you can revise the geometry in a CAD application. Follow the steps below to validate your simulation.

Note:

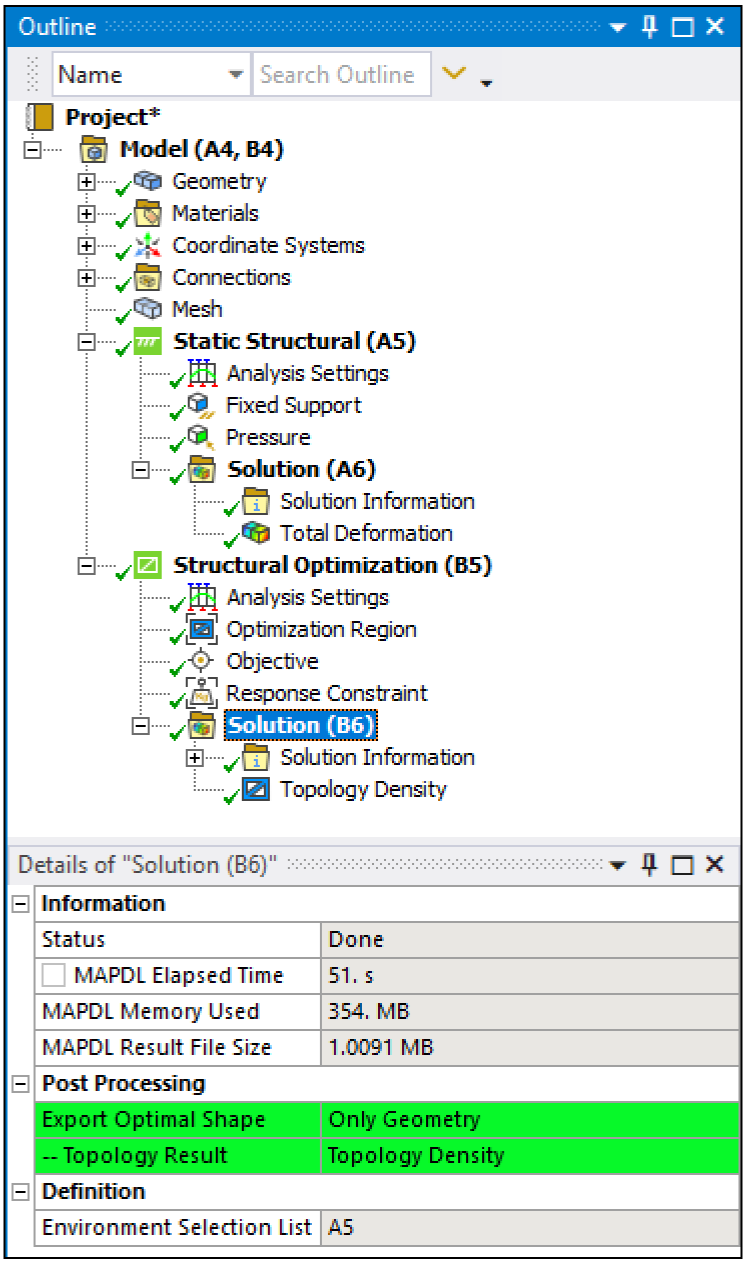
As desired, you can use Non-Ansys tools to validate your design. You will need to retrieve the STL file of the optimized design in order to import it into the geometry modeling software of your choice.

Create Validation System

To validate your optimized topology, actions are required in Mechanical as well as Workbench.

File Preparation in Mechanical

The optimized geometry file - in Standard Tessellation Language (STL) - is created from the selected **Topology Density** result picked using the **--Topology Result** property of the **Solution** object of the optimization system in Mechanical. The **--Topology Result** property only appears when the property **Export Optimal Shape** is set to **Only Geometry**.



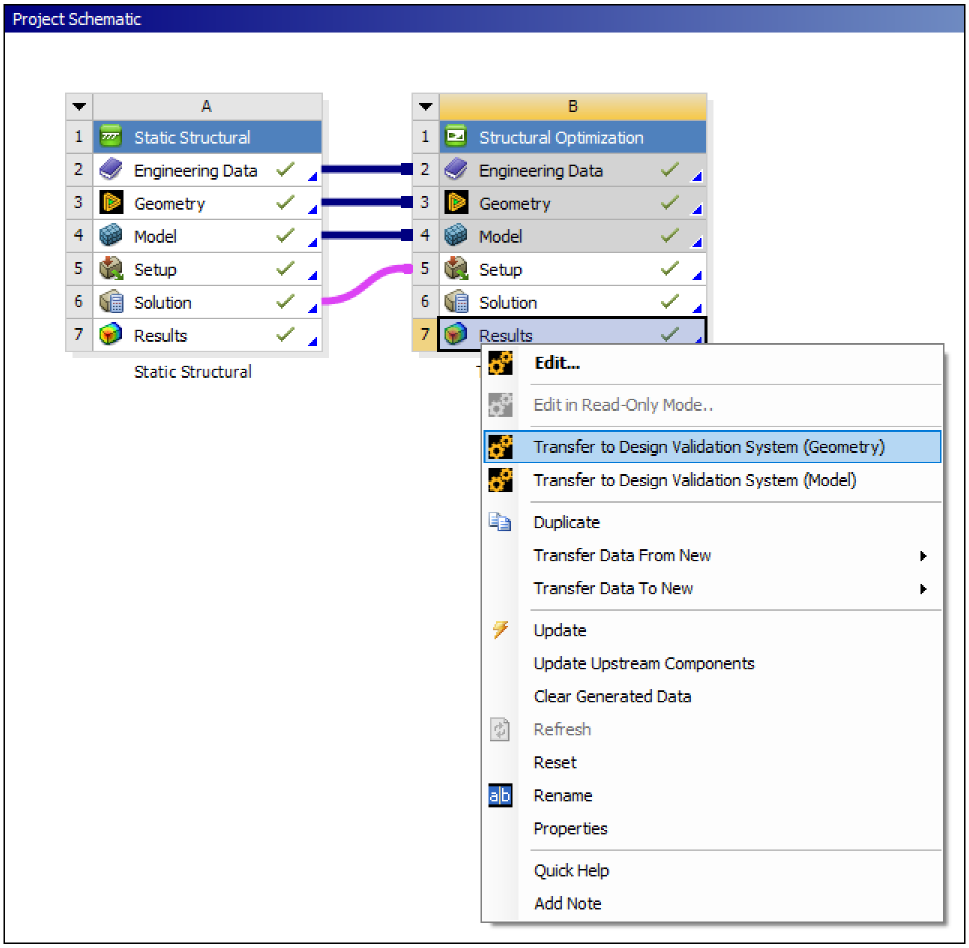
By having both geometries available in the **Geometry** cell (in Workbench), you can perform actions such as overlaying or preserving sections of the geometry or sweep surfaces in order to create additional material around selected regions such as bolt holes.

Note:

If you decide to use Discovery Modeling to adjust the optimized geometry, check the **Additive Manufacturing** section (as well as the **Designing, Repairing problems**, and **Preparing designs for analysis** sections) in the Discovery Modeling Help for the tools you can use to simplify and prepare the optimized geometry in the new system.

Create Design Validation System in Workbench

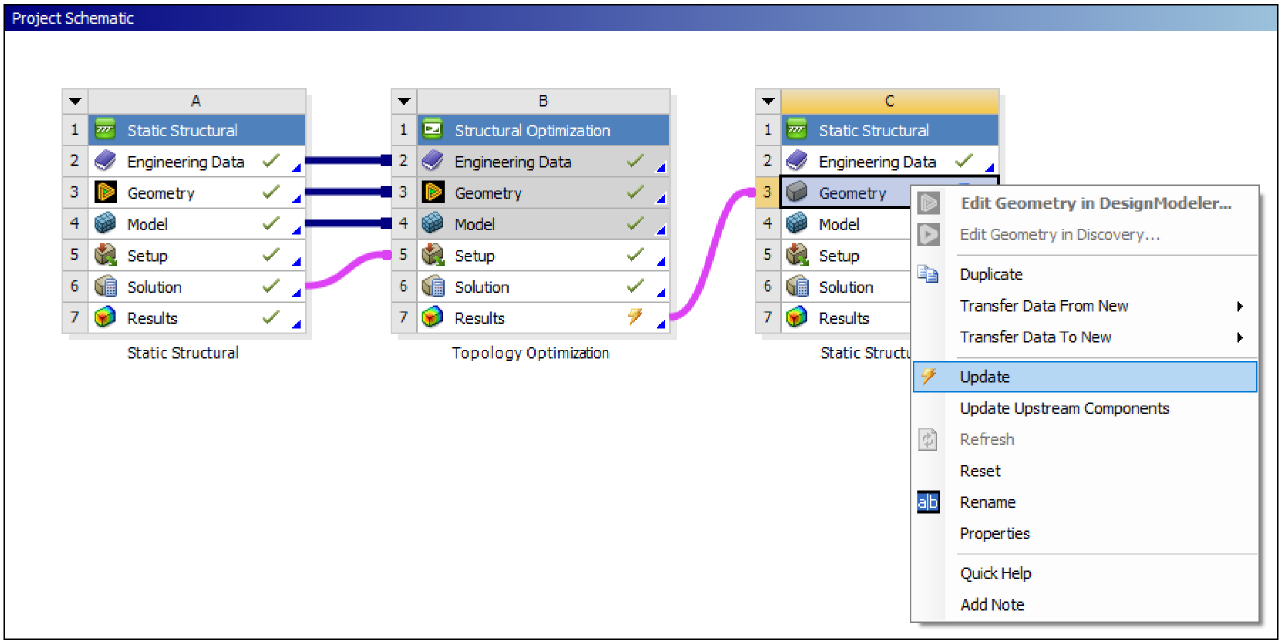
Once you have specified the desired result to export and solved the analysis, return to Workbench. As shown below, the highlighted context menu option **Transfer to Design Validation System (Geometry)** becomes available to transfer the **Results** cell of your completed analysis to either the **Geometry** cell or the **Model** cell of a newly created system. To begin this process, right-click the Structural Optimization's **Results** cell and select the **Transfer to Design Validation System (Geometry)** option from the menu.



Note:

The operation is the same if you have multiple upstream systems.

Once you select the option, Workbench creates a new Mechanical system of the same type that is upstream of the optimization system and send the original geometry and the optimized geometry to the **Geometry** cell of the new system. Next, **Update** the **Geometry** cell of the new system to update the **Results** cell of the optimization system (which changed to out of date after the new system is created and linked) and the **Geometry** cell of the new system. The **Geometry** cell of the new system becomes up-to-date after the action is complete.



If you are working with two upstream systems, you simply need to **Update** the first newly created system. All other downstream systems share **Engineering Data**, **Geometry**, and **Model** cell data. Once updated, you can validate all of the systems in one Mechanical session.

Note:

Even though the **Geometry** cell of the new system is up-to-date, first simplify the optimized geometry using Discovery Modeling before attempting to open up the geometry in Mechanical. Attempting to open the unsimplified optimized design from the STL file in Mechanical will take a long time and will lead to issues due to the use of facets.

2.13.2. Model Validation

Design validation using the **Transfer to Design Validation System (Model)** option to transfer data to the downstream system's **Model** cell. The benefit of this option is that you can automatically transfer all scoping, loading conditions, etc. to the new system. Follow the steps below to validate your simulation.

Note:

As desired, you can use non-Ansys tools to validate your design. You will need to retrieve the STL file of the optimized design in order to import it into the geometry modeling software of your choice.

Create Validation System

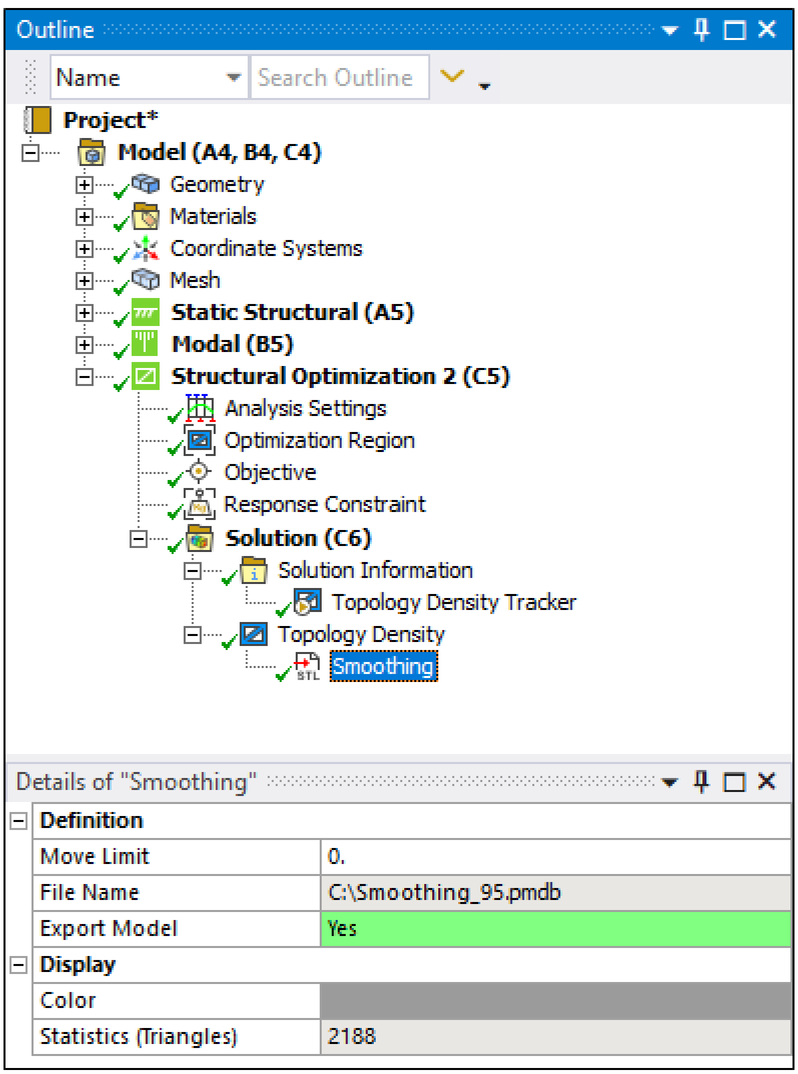
To validate your optimized topology, actions are required in Mechanical as well as Workbench.

Important:

- The design validation process does not support Instancing.
- For projects saved prior to release 2025 R1 that are using non-associative CAD import, the automatic propagation of scoping for the downstream validation system will not occur. A debug option enables you to correct this situation.

File Preparation in Mechanical

The design validation process requires that you specify a Smoothing object for your desired **Topology Density** (p. 61) result in Mechanical. Once you insert the **Smoothing** object, you need to set the **Export Model** property to **Yes** to make the result available for validation. Evaluate this result object once complete.



The application creates a `.pmdb` file for export. An example is shown in the **File Name** property above. This file is placed in the working project directory and will be used in the creation of the new design validation system in Workbench.

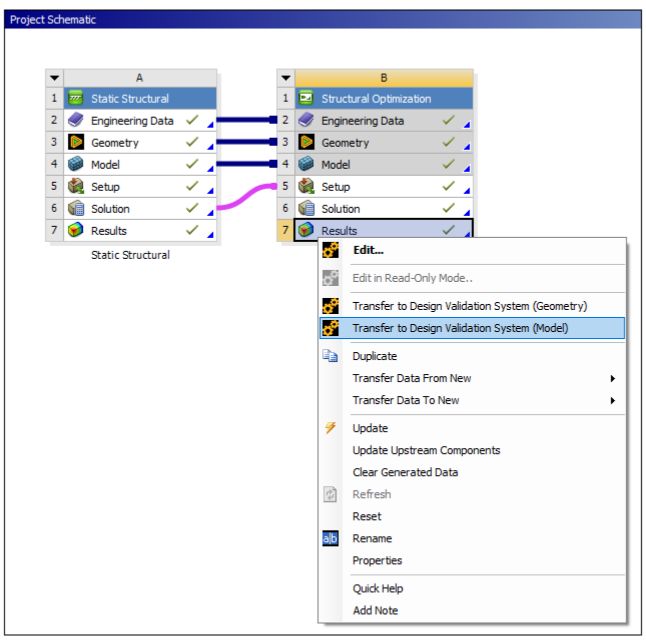
Note:

The **Smoothing** feature is not supported for:

- The Linux platform. However, you can create a design validation system on Windows and then solve it on Linux.
- 2D analyses and analyses that include a 3D Surface Body (Shell).

Create Design Validation System in Workbench

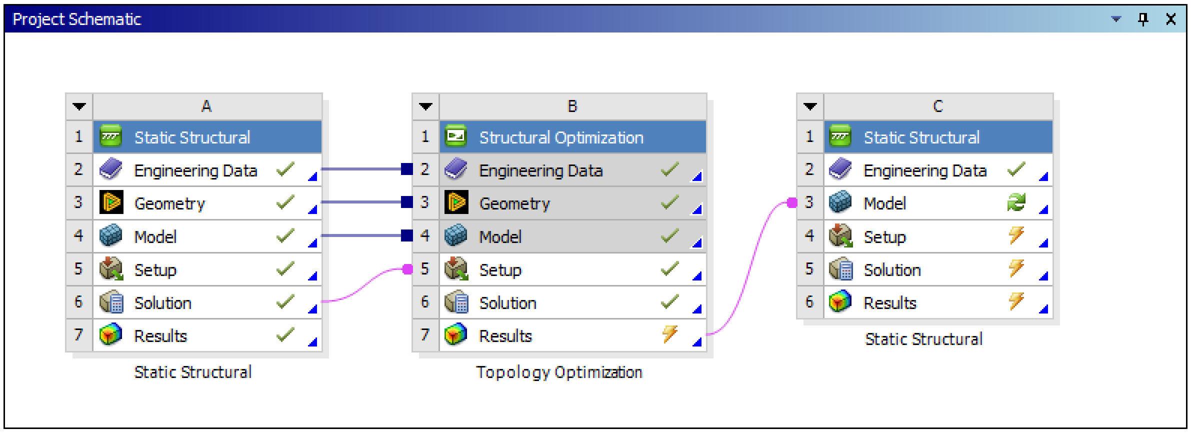
Once you have specified the desired result to export and solved the analysis, return to Workbench. As shown below, the highlighted context menu option **Transfer to Design Validation System (Model)** becomes available to transfer the **Results** cell of your completed analysis to either the **Geometry** cell or the **Model** cell of a newly created system. To begin this process, right-click the Structural Optimization's **Results** cell and select the **Transfer to Design Validation System (Model)** option from the menu.



Note:

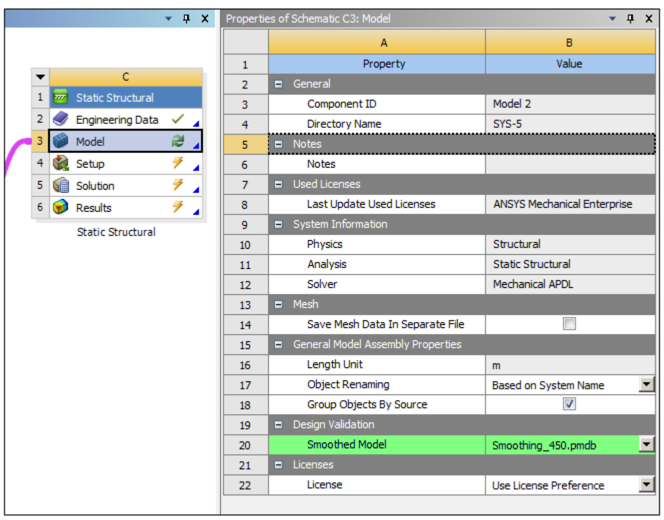
The operation is the same if you have multiple upstream systems.

Workbench creates and links a new Mechanical system of the same type that is upstream of the Structural Optimization system.

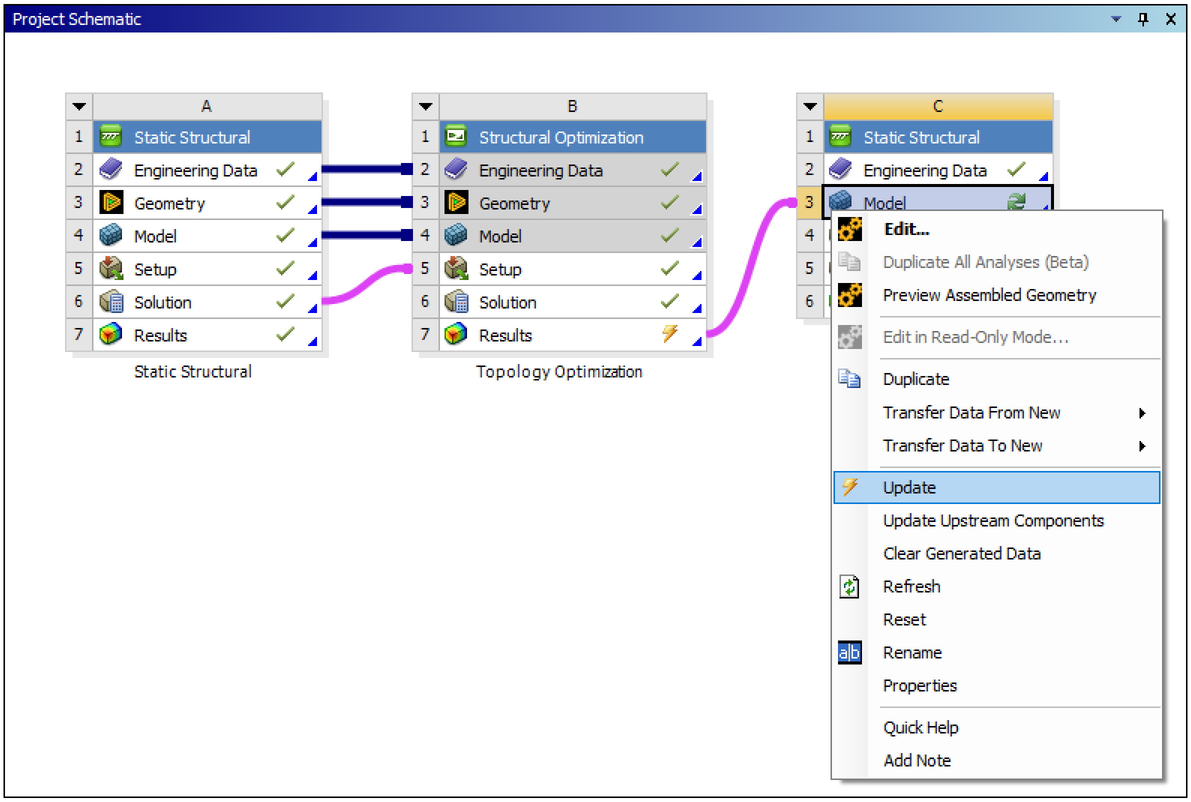


Update New System

As pointed out above, by default, the application creates an `.pmdb` file for export. In Workbench, this application generated file becomes available in the drop-down list of the **Smoothed Model** property of the **Model** cell properties on the Workbench Project Schematic. The **Smoothed Model** drop-down menu can contain multiple `.pmdb` files based on the number of **Smoothing** objects you specify for export in Mechanical. Multiple files are often created for **Topology Density** results that have different **Retained Threshold** settings.



Next, update the **Model** cell of the new system: right-click and select **Update**. This updates the **Results** cell of the Structural Optimization system (which changed to out-of-date after the new system is created and linked) and the **Model** cell of the new system.



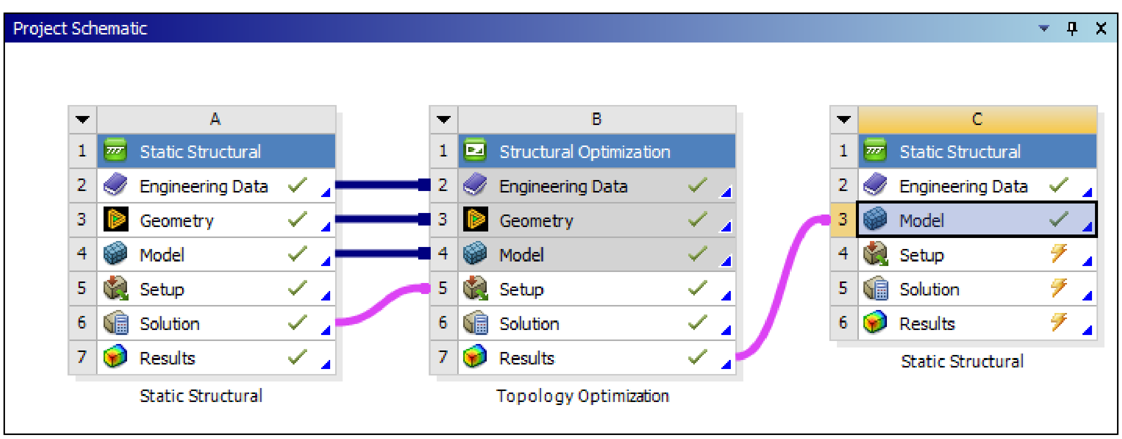
Important:

It is possible that you will receive an error during the update. This is a result of a mesh limitation. Return to Mechanical and specify a Patch Independent Mesh Method and update the new system again.

If you are working with two upstream systems, you simply need to **Update** the first newly created system. All other downstream systems share **Engineering Data** and **Model** cell data. Once updated, you can validate all of the systems in one Mechanical session.

Analyze Optimized Design

Once the **Model** cell of your new system is up-to-date, you can open it in Mechanical and analyze the newly optimized topology.



If you decide to use Discovery Modeling to adjust the optimized geometry, check the **Additive Manufacturing** section (as well as the **Designing**, **Repairing problems**, and **Preparing designs for analysis** sections) in the Discovery Modeling Help for the tools you can use to simplify and prepare the optimized geometry in the new system.

Chapter 3: Design Methods

This chapter examines how each method processes the shapes, the rules you must follow for each method, as well as other additional general process descriptions.

3.1. Shape Processing Workflows

3.2. Recommendations - Optimization in Practice

3.3. Mixing Multiple Methods within an Analysis

3.4. Topology Optimization versus Shape Optimization

3.1. Shape Processing Workflows

Structural optimization is by nature a large-scale constrained optimization, where the number of degrees of freedom (DOF) ranges from 1e4 to more than 1e9 and is solved using gradient-based algorithms.

Despite the qualitative differences between the optimization methods, each method follows the same analytical workflow when performing an analysis, including:

1. **Shape Description**: For the Shape Optimization and Topography Optimization methods the shape is explicitly defined using the finite element mesh. The Topology Optimization and Lattice Optimization methods use implicit descriptions through either density fields (Lattice and Solid Isotropic Material with Penalization method (SIMP) topology optimization) or level-set functions (Level Set-based topology optimization).

2. **Shape Evaluation**: Methods using implicit shape description require dedicated treatment for finite element analysis.

3. **Shape Derivative Computation**: Based on the shape description, the application selects how to best compute the shape derivative, that is, the desired sensitivity for each specified criterion with respect to the corresponding degrees of freedom of the optimization problem.

4. **Shape Update**: The application performs final geometry modifications based on the selected shape description.

See the following sections for the shape processing workflows of each method:

3.1.1. Density Based Topology Optimization Shape Processing

3.1.2. Level Set Based Topology Optimization Shape Processing

3.1.3. Lattice Optimization Shape Processing

3.1.4. Shape Optimization Shape Processing

3.1.5. Topography Optimization Shape Processing

3.1.1. Density Based Topology Optimization Shape Processing

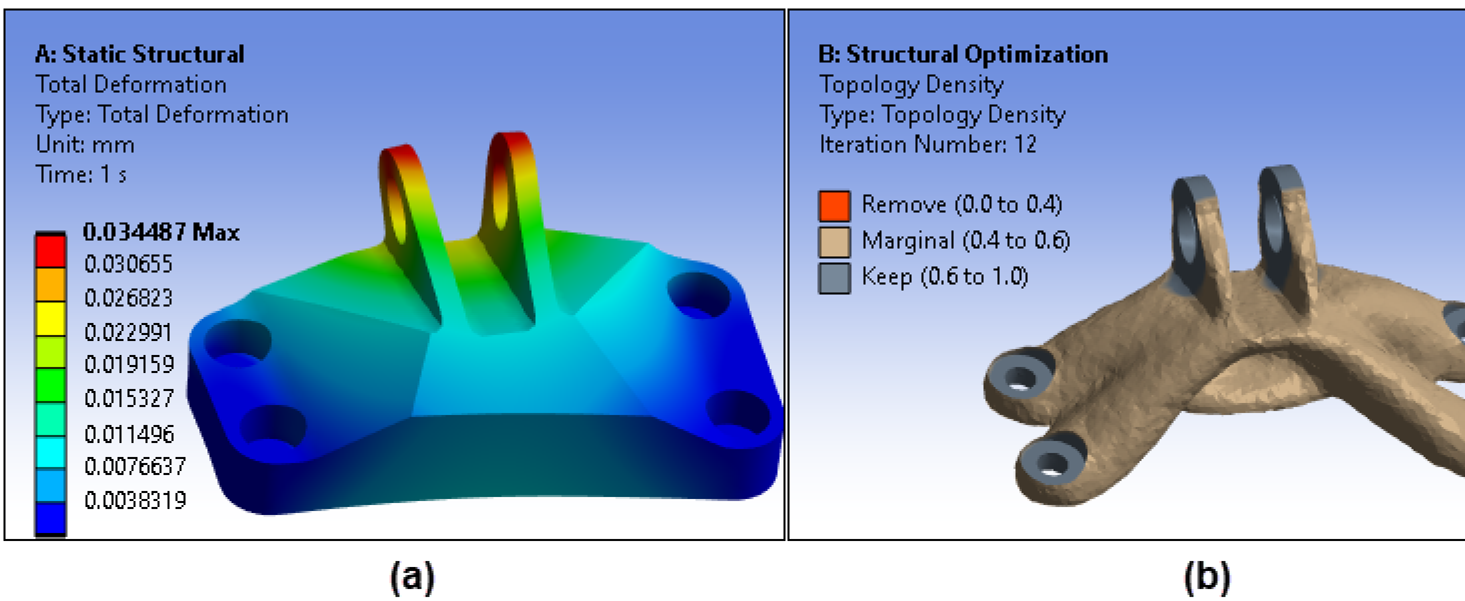
The density-based method belongs to the family of topology optimization technology. Adopting the logic of immersed boundary methods, this method aims to optimize the distribution of material within a working domain by varying a density field ranging from 0 to 1. Topology optimization technology, which first appeared in the framework of the homogenization method for composites [1 (p. 195)] [3 (p. 195)]. Simplified approaches, such as Solid Isotropic Material with Penalization method (SIMP), followed and were devised to force the density to approach either 0 or 1, making them applicable for standard solid structures.

Shape Description

You can characterize the density-based approach as a large-scale parametric optimization where the DOF is the “density.” That is, a unit-less variable ranging from 0 to 1, where:

- A value of zero (0) virtually removes elements to create virtual voids.
- A value of one (1) specifies an element with a material.

As a result, this method does not deliver a shape but an optimized density-field. To create an unambiguous design, the application forces the density to approach 0 or 1 and then removes any intermediate densities. Here is an example of a part defined with a density-threshold of 0.5 (default).



Where:

(a) is the optimal distribution of density (over the working domain).

(b) is the resulting shape (while drawing the iso-surface corresponding to 0.5 – density)

Shape Evaluation

For shape evaluation using the density-field, $\rho(x)$, it is critical to properly update the material properties. Specifically, the application updates:

- Young modulus and mass for static structural and modal analyses.
- Conductivity and mass for thermal analyses.

To force solutions to approach either zero (0) or one (1), Ansys uses the following widely documented interpolation schemes:

$M = M_{void} + \rho\left(M_{solid} - M_{void}\right)$ (linear interpolation for the mass).

$K = K_{void} + \rho^{p}\left(K_{solid} - K_{void}\right)$ (power-law interpolation for stiffness [BS2003]).

Note:

For the above calculations, $M_{\text{void}} << M_{\text{solid}}$ and $K_{\text{void}} << K_{\text{solid}}$ correspond to the material properties of a weak material representing the void.

Shape Derivative

The shape derivative used in gradient-based optimizers for the density-based method is calculated from the derivative with respect to the density. Furthermore, for:

- Geometric criteria, the computation is analytical.
- Physics-based criteria, the adjoint approach is used.

Shape Update

By adding the descent direction $d\rho^{*}(x)$, derived from the gradient-based optimizer, to the current density field $\rho^{n}(x)$, the application computes a new density field $\rho^{n+1}(x)$. The calculation is represented as:

$$\rho^{n+1}(x) = \rho^{n}(x) + d\rho^{*}(x)$$

representing the new shape: Ω^{n+1}.

Summary

Degrees of freedom for this method are based on the density of the elements or the density at the nodes.

Strengths

Enables you to easily manage topological changes.

It is based on the development of light machinery and enables fast convergence.

Place in Design Stage

Used early in the design process to sketch designs.

Limitations

Can sometimes deliver an ambiguous solution, including intermediate densities that do not provide the most ideal optimized part.

Produces a less accurate evaluation than the body-fitted approach.

Tips

Use a uniform mesh to equally capture geometric details for the entire domain.

References (p. 195)

[1] MP Bendsoe, N. Kikuchi, Generating optimal topologies in structural design using a homogenization method, Computer methods in applied mechanics and engineering, 1988.

[2] MP Bendsoe, O. Sigmund, Topology optimization: theory, methods, and applications, Springer Science & Business Media, 2003.

[3] F. Murat, L. Tartar, Calcul des Variations et Homogeneisation, In Les Methodes de l Homogeneisation Theorie et Applications en Physique, Coll. Dir. Etudes et Recherches EDF, 57, Eyrolles, Paris, pp.319-369, 1985.

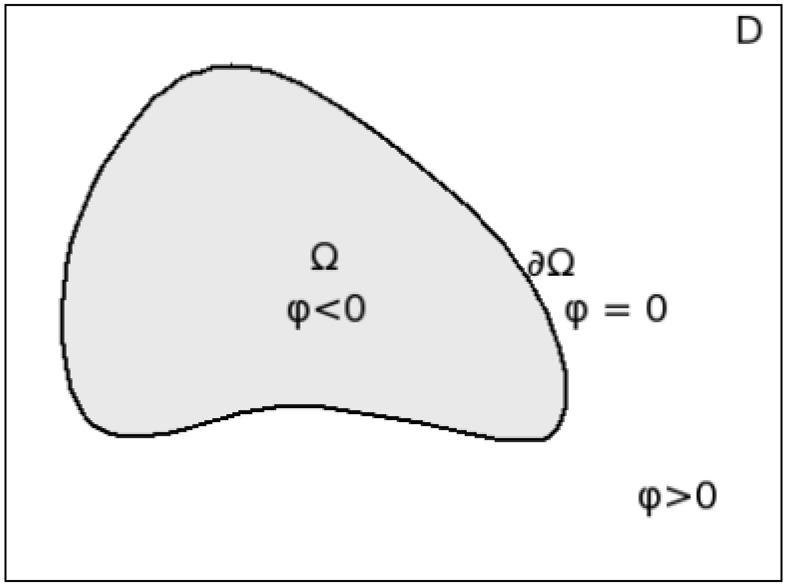
3.1.2. Level Set Based Topology Optimization Shape Processing

Another topology optimization technology method is the level-set based method. Originally used for computational fluid dynamics (CFD) to track fluid-interface it is also applied in structural optimization.

Shape Description

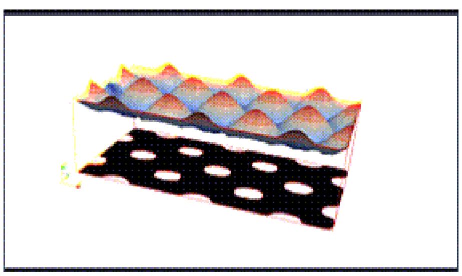
This method manages shape description through pure geometric information and defines a shape without ambiguity. That is, for the shape, defined over the working domain, using an auxiliary function, denoted as the level-set function, specifies positive, zero, or negative values, such that:

$\varphi(x) < 0, x \in \Omega$
$\varphi(x) = 0, x \in \partial\Omega$
$\varphi(x) > 0, x \in D \backslash \Omega$



Note:

- This implicit representation of the shape enables you to make topological changes without needing to detect topological modifications and reconstruct shape parametrizations.



- Provides a convenient framework for the calculation of geometric quantities, such as the exterior normal vector: $n = \frac{\nabla\varphi}{\|\nabla\varphi\|}$.

- For the level-set functions that provide the same shape description, Ansys uses the signed-distance function (SDF) [DF2012], defined as:

$$d_\Omega(x) = \begin{cases} -d(x, \partial\Omega), x \in \Omega \\ 0, x \in \partial\Omega \\ d(x, \partial\Omega), x \in D \backslash \Omega \end{cases}$$

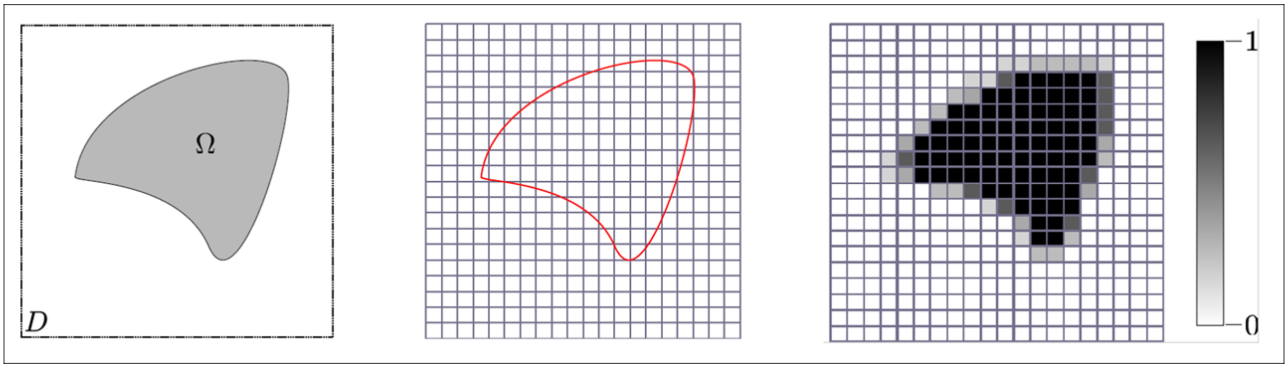
  where $d(x, \partial\Omega)$ denotes the standard Euclidian distance from a point $x \in D$ to the boundary $\partial\Omega$.

- The level-set function is discretized at vertices of the mesh and interpolated inside the elements.

Shape Evaluation

The application evaluates the criteria on a fixed mesh. However, the definition of the mechanical properties is simple and generic, that is:

- Any element lying inside of the shape has a density value of 1.
- Any element lying outside the shape is specified as a void-material.
- As to the one layer of elements cut by the zero level-set, they receive an intermediate density in accordance with the solid fraction.



Pseudo-density used for the modification of the mechanical properties.

Using the ersatz-material approach [5 (p. 195)], each material property is interpolated as $E = E_{void} + \rho(E_{solid} - E_{void})$, where $E_{void} << E_{solid}$ corresponds to the material properties of a weak material representing the void.

Shape Derivative

The application computes the shape derivative using the continuous formalism defined by Hadamard (see [5 (p. 195)]). That is, given a shape perturbation θ, the asymptotic expansion reads:

$$J(\Omega_\theta) = J(\Omega) + J'(\Omega)(\theta) + \circ(\theta)$$

Where:

Ω is current shape.

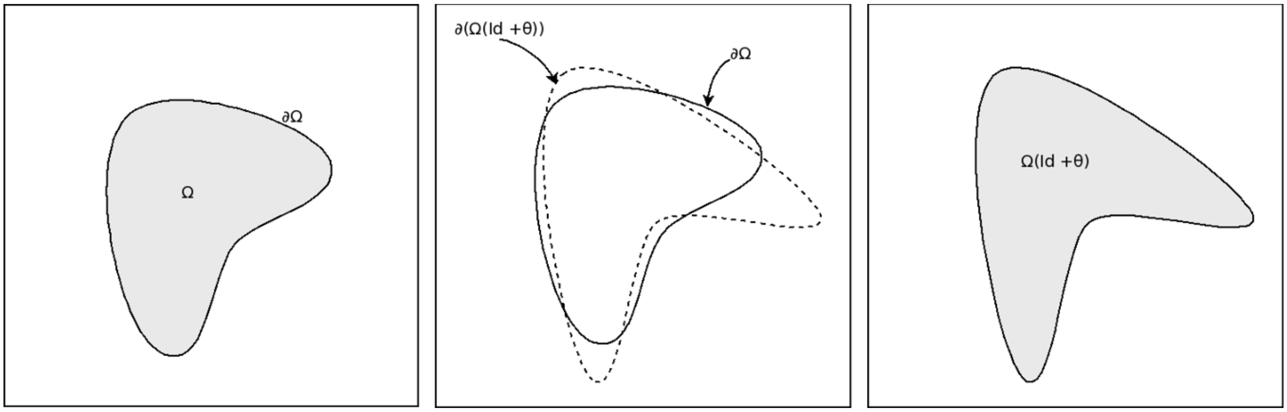
θ is the shape perturbation.

$\Omega_\theta = (id + \theta)\Omega$ is the new shape.

The shape derivative usually admits the following form:

$$J'(\Omega)(\theta) = \int_{\vartheta\Omega} j(s)n(s) \cdot \theta(s)ds$$

For the form, the integrand, $j(s)$, depends on the criterion, $J(\Omega)$, through both the solution state of the mechanical problem and some corresponding adjoint-state.



Shape perturbation by a vector field θ.

Shape Update

Given a shape perturbation θ^*, the application updates the shape by solving a transport equation for the level-set function [BDF2012]:

$$\frac{\partial\varphi(x,t)}{\partial t} + \theta^*(x) \cdot \nabla\varphi(x,t) = 0$$

Summary

The degrees of freedom for this method are based on the boundary of the shape.

Strengths

Enables you to easily manage topological changes.

It delivers an unambiguous solution.

Place in Design Stage

Used early in the design process to sketch conceptual designs.

Limitations

Produces a less accurate evaluation that the body-fitted approach.

Due to the heavy machinery, the run is sometimes more expensive compared to the density method.

Tips

Use a uniform mesh to equally capture geometric details for the entire domain.

References (p. 195)

[4] C. Dapogny, P. Frey, Computation of the signed distance function to a discrete contour on adapted triangulation, Calcolo, 2012.

[5] G. Allaire, F. Jouve, AM Toader, Structural optimization using sensitivity analysis and a level-set method, Journal of computational physics, 2004.

[6] C. Bui, C. Dapogny, P. Frey. An accurate anisotropic adaptation method for solving the level set advection equation, International Journal for Numerical Methods in Fluids, 2012.

[7] S. Osher, JA Sethian, Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations, Journal of computational physics, 1988.

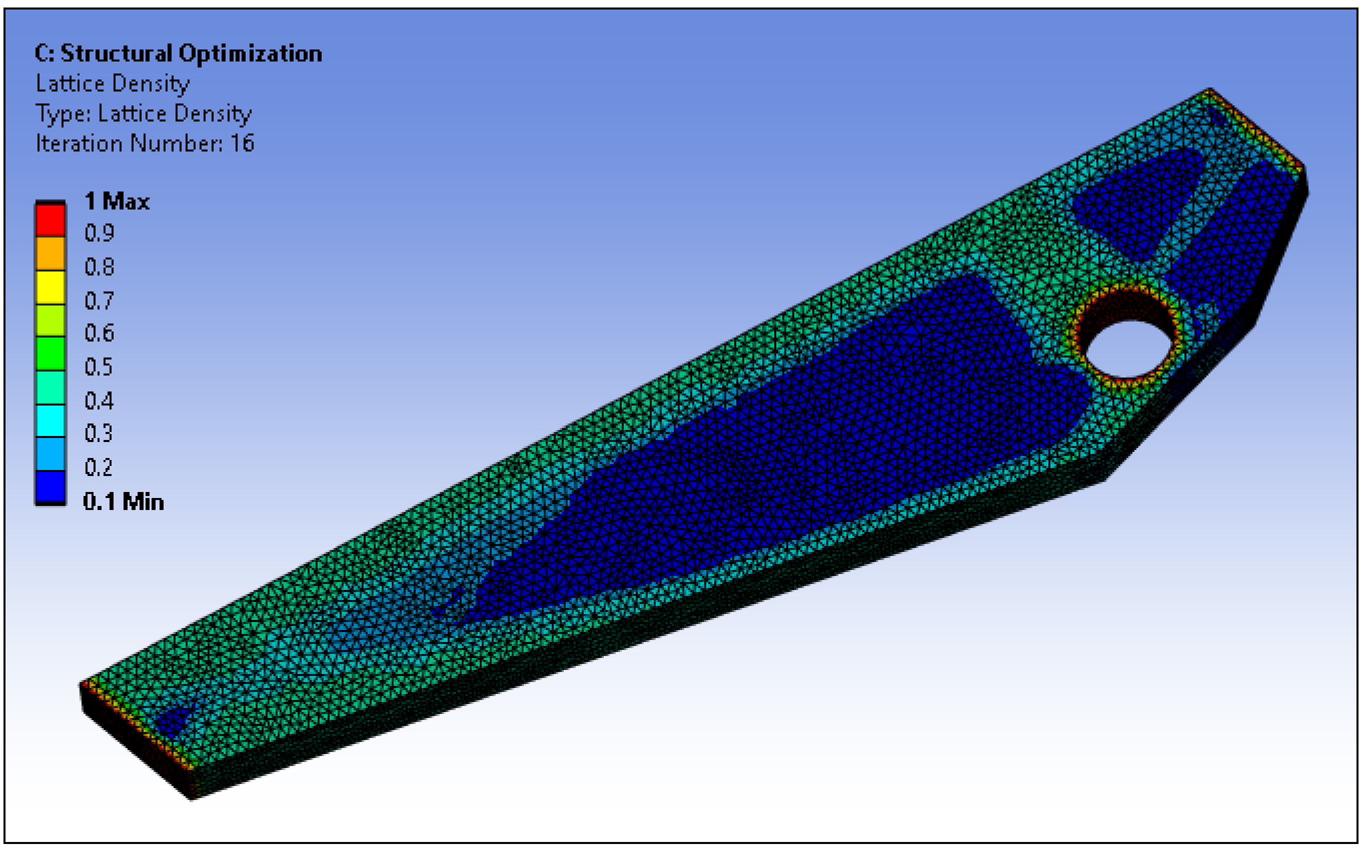
3.1.3. Lattice Optimization Shape Processing

This method enables you to compute an optimal variable density lattice distribution in your geometry.

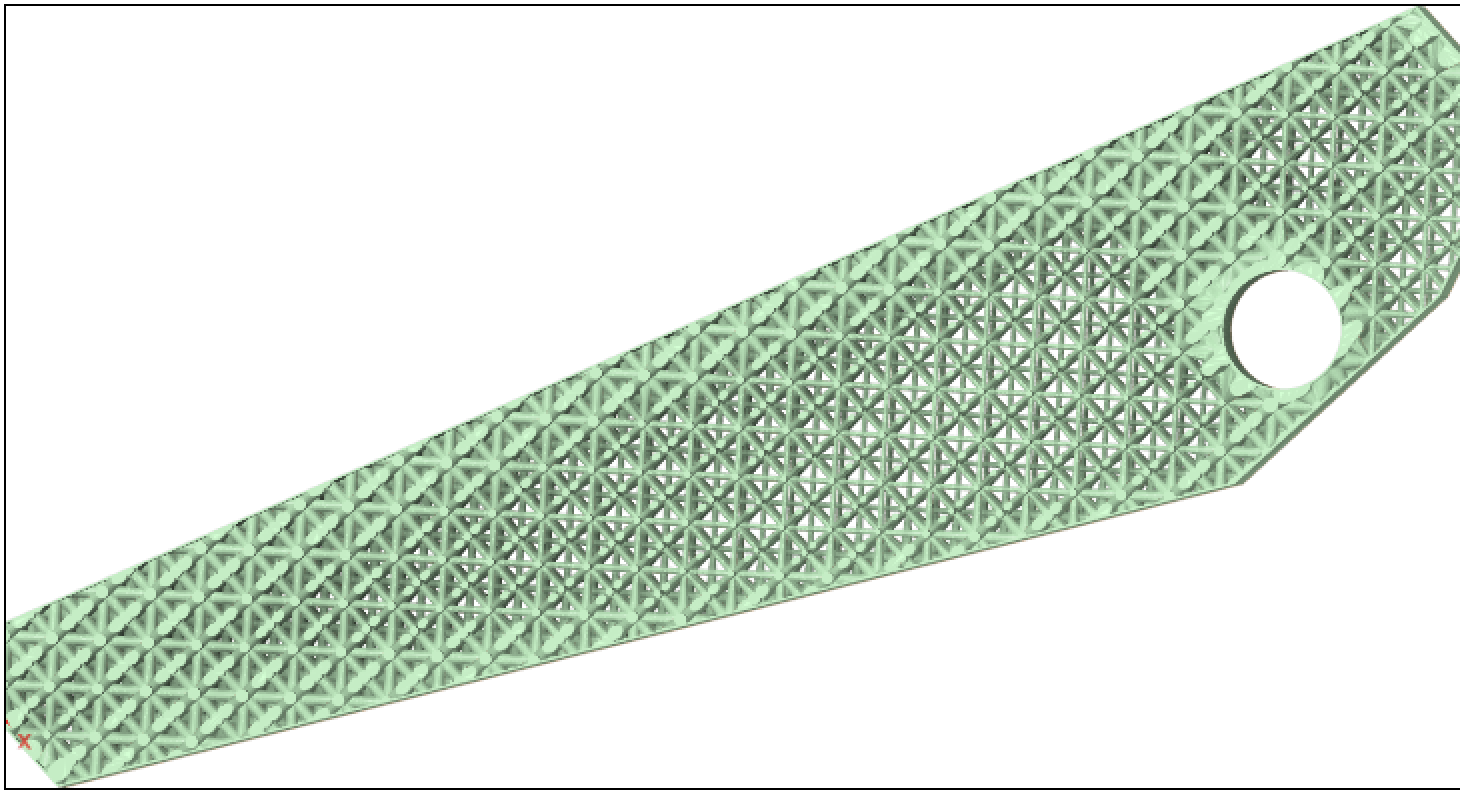
Shape Description

For the lattice method, the working domain is occupied by a lattice structure of varying density $\rho(x)$.

You need to define the cell-size as well as the minimum and maximum density value $\left(\rho_{\min} \leq \rho(x) \leq \rho_{\max}\right)$. Based on the values of $\rho(x)$ as well as other options, the product is able to reconstruct a lattice structure inside of the working domain.



Optimized Lattice Density Distribution



Lattice Reconstruction

Shape Evaluation

Depending on the lattice pattern and the density value at each element, the mechanical properties described by the elasticity tensor $E_{ijkl}(\rho(x))$ are modified via analytical formulas [11 (p. 195)], based on results extracted using the homogenization theory [12 (p. 195)].

Shape Derivative

The shape derivative used in gradient-based optimizers for the lattice method is calculated from the derivative with respect to the density. Furthermore, for:

- Geometric criteria, the computation is analytical.
- Physics-based criteria, the adjoint approach is used.

The derivatives, $\frac{\partial E_{ijkl}}{\partial \rho}$, of the elasticity tensor with respect to the density are obtained via analytical formulas [11 (p. 195)].

Shape Update

By adding the descent direction $d\rho^{*}(x)$, derived from the gradient-based optimizer, to the current density field $\rho^{n}(x)$, the application computes a new density field $\rho^{n+1}(x)$. The calculation is represented as:

$$\rho^{n+1}(x) = \rho^{n}(x) + d\rho^{*}(x)$$

Summary

Degrees of freedom for this method are the density at nodes (that will be then mapped at elements).

Strengths

The variable lattice structure is parametrized using just one field.

The computation is fast, thanks to using the homogenized properties in the finite element analysis.

Place in Design Stage

Used in the final stage of the design process.

Limitations

Some results, such as stress, may not be accurate enough, especially close to the shape boundary, due to using the homogenized properties in the finite element analysis.

This method only supports linear analyses.

Tips

The smaller you make the lattice cell size, the more accurate the finite element modelling.

References (p. 195)

[11] L. Cheng, P. Zhang, E. Biyikli, J. Bai, J. Robbins, A. To, Efficient design optimization of variable-density cellular structures for additive manufacturing: theory and experimental validation, Rapid Prototyping Journal, 2017.

[12] G. Allaire, Homogenization and two-scale convergence, SIAM Journal on Mathematical Analysis, 1992.

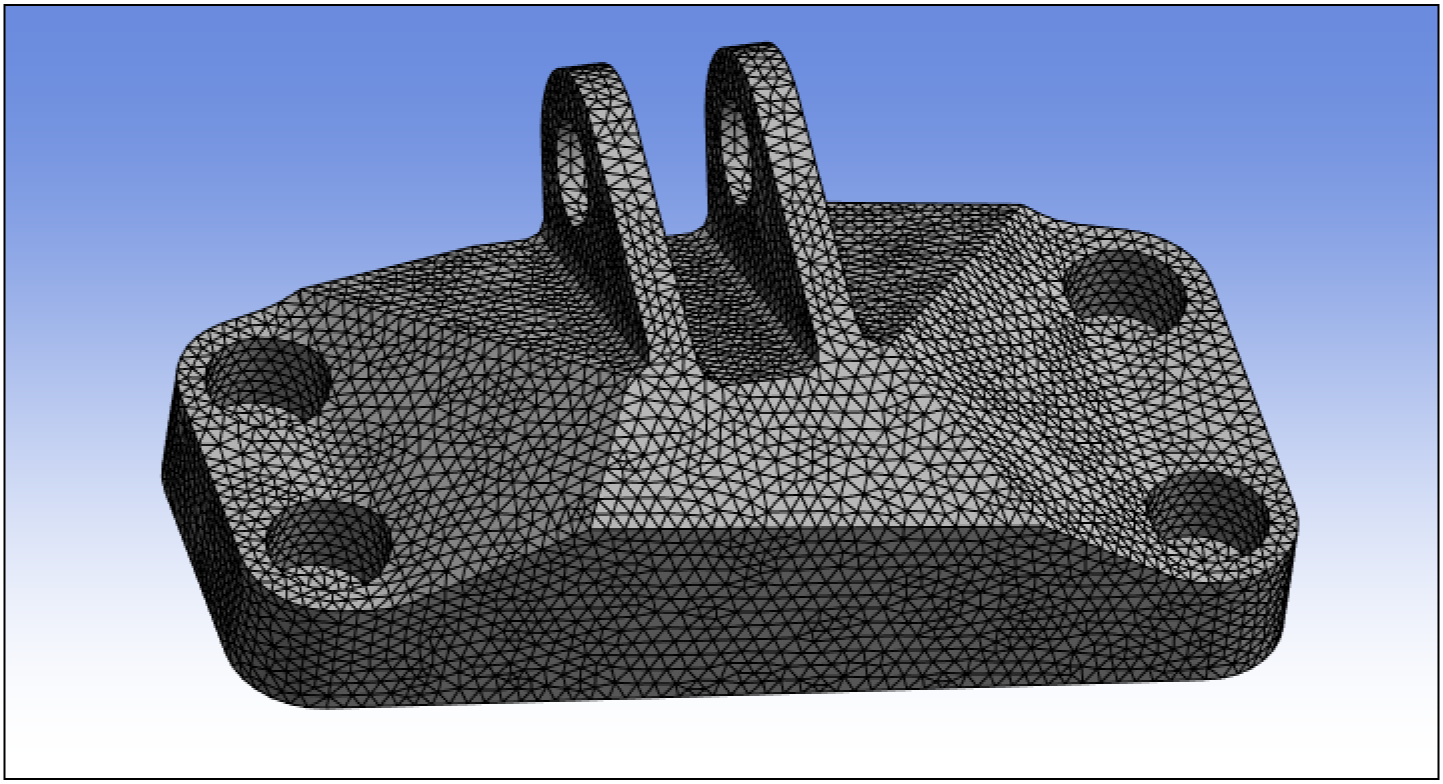
3.1.4. Shape Optimization Shape Processing

The Shape Optimization method uses morphing optimization technology to process solid elements by varying the node locations.

Shape Description

For the Shape Optimization method, the shape Ω is represented by a simplicial mesh $T = \{T_i\}_{i=1}^{N}$ composed of N tetrahedral elements T_i.

Conformal Tetrahedral Mesh



Shape Evaluation

For this method, no special treatment is required to perform shape evaluation.

Note:

An acceptable mesh quality is expected to be retained during the optimization process. However, excessive deformation of the shape may lead to large approximation errors in the finite element analysis.

Shape Derivative

The application computes the shape derivative using the continuous formalism defined by Hadamard (see [5 (p. 195)]). That is, given a shape perturbation θ, the asymptotic expansion reads:

$$J(\Omega_\theta) = J(\Omega) + J'(\Omega)(\theta) + \circ(\theta)$$

where:

Ω is current shape.

θ is the shape perturbation.

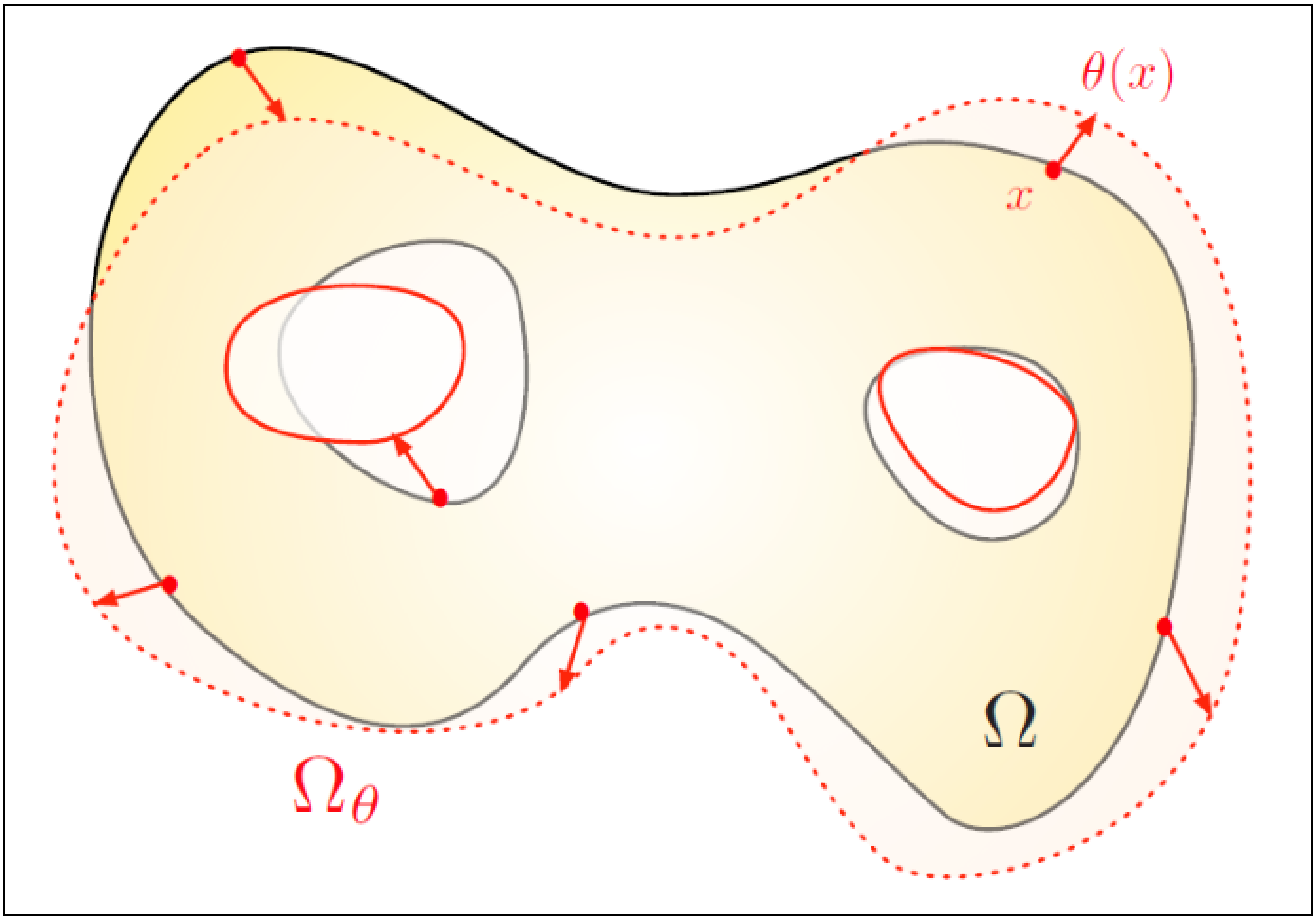
$\Omega_\theta = (id + \theta)\Omega$ is the new shape.

The shape derivative usually admits the following form:

$$J'(\Omega)(\theta) = \int_{\vartheta\Omega} j(s)n(s)\cdot\theta(s)ds$$

For the form, the integrand, $j(s)$, depends on the criterion, $J(\Omega)$, through both the solution state of the mechanical problem and some corresponding adjoint-state.

Shape Perturbation via a Vector Field $\theta(x)$



Note:

To perform larger shape modifications and to avoid distortions that may lead to poor mesh quality, the shape gradient is extended inside the whole domain by solving an auxiliary problem [8 (p. 195)].

Shape Update

The application creates the new shape by updating the position of the mesh vertices using the shape perturbation θ^*, as:

$$x_i^{n+1} = x_i^n + \theta^*(x_i^n)$$

where:

x_i^{n+1} is the position vector of vertex i at the new shape.

x_i^n is the position vector of vertex i at the current shape.

θ^* is the shape perturbation.

Specific programming is in place to preserve mesh-quality.

Summary

Degrees of freedom for this method are based on node location.

Strengths

This method accurately computes any state variable given proper mesh quality.

Compared to the topology optimization methods, no numerical trickery occurs to evaluate the shape.

This method is rather dedicated for local and decent modification, but the implementation has proved to conveniently manage large shape changes without remeshing.

Place in Design Stage

Used in the final stage of design process when local and decent shape adjustments are expected.

Limitations

Finite element approximation errors could occur due to poor-quality mesh regions or if large modifications are made.

Because shape optimization does not manages topology changes, additional programming is in place to preserve mesh-quality. This factor can sometimes lead to additional computational requirements.

Tips

Use a uniform mesh to equally capture geometric details.

References (p. 195)

[8] G. Allaire, C. Dapogny, F. Jouve, Shape and topology optimization, Handbook of numerical analysis, 2021.

[9] G. Allaire, M. Schoenauer, Conception optimale de structures, Springer, 2007.

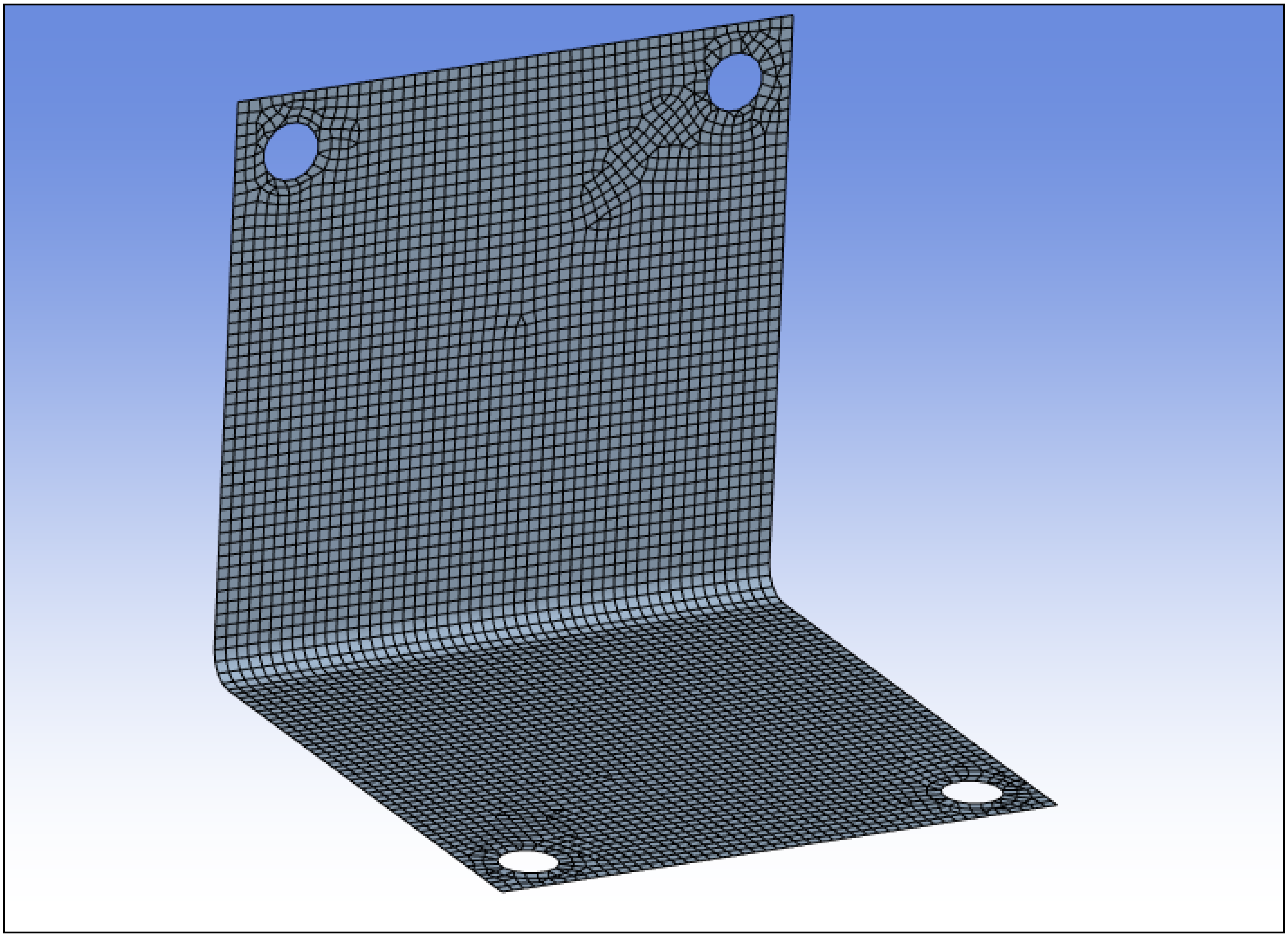
3.1.5. Topography Optimization Shape Processing

The Topography Optimization method uses morphing optimization technology to process shell elements by varying the node locations.

Shape Description

For the Topography Optimization method, the shape, Ω, is represented by a mesh, $T = \{T_i\}_{i=1}^{N}$, and composed of N shell elements T_i. This method is essentially equivalent to the Shape Optimization method for surface bodies.

Conformal Shell Mesh



Shape Evaluation

For this method, no special treatment is required to perform shape evaluation.

Note:

An acceptable mesh quality is expected to be retained during the optimization process. However, excessive deformation of the shape may lead to large approximation errors in the finite element analysis.

Shape Derivative

The application computes the shape derivative using the continuous formalism defined by Hadamard (see [AJT2004]). That is, given a shape perturbation θ, the asymptotic expansion reads:

$$J(\Omega_\theta) = J(\Omega) + J'(\Omega)(\theta) + \circ(\theta)$$

Where:

- Ω is current shape.
- θ is the shape perturbation.
- $\Omega_\theta = (id + \theta)\Omega$ is the new shape.

The shape derivative usually admits the following form:

$$J'(\Omega)(\theta) = \int_{\vartheta\Omega} j(s)n(s) \cdot \theta(s)ds$$

For the form, the integrand, $j(s)$, depends on the criterion, $J(\Omega)$, through both the solution state of the mechanical problem and some corresponding adjoint-state.

Shape Update

The application creates the new shape by updating the position of the mesh vertices using the shape perturbation θ^*, as:

$$x_i^{n+1} = x_i^n + \theta^*(x_i^n)$$

Where:

- x_i^{n+1} is the position vector of vertex i at the new shape.
- x_i^n is the position vector of vertex i at the current shape.
- θ^* is the shape perturbation.

Specific programming is in place to preserve mesh-quality.

Summary

Degrees of freedom for this method are based on node location.

Strengths

This method accurately computes any state variable given proper mesh quality.

Compared to the topology optimization methods, no numerical trickery occurs to evaluate the shape.

This method is specific to local modifications of the shape but also manages large shape changes without remeshing.

Place in Design Stage

Used in the final stage of the design process for local shape adjustments.

Limitations

Finite element approximation errors could occur due to poor-quality mesh regions or if large modifications are made.

Because shape optimization does not manage topology changes, additional programming is in place to preserve mesh-quality. This factor can sometimes lead to additional computational requirements.

Tips

Use a uniform mesh to equally capture geometric details.

References (p. 195)

[9] G. Allaire, M. Schoenauer, Conception optimale de structures, Springer, 2007.

[10] M. Shimoda, Y. Liu, A non-parametric free-form optimization method for shell structures, Structural and Multidisciplinary Optimization, 2014.

3.2. Recommendations - Optimization in Practice

The available design methods have certain prerequisites, requirements, limitations, and individual features that you need to understand. Review the following sections as needed for your analysis:

3.2.1. Topology Optimization - Density Based

3.2.2. Topology Optimization - Level Set Based

3.2.3. Topology Optimization - Mixable Density

3.2.4. Lattice Optimization Analysis

3.2.5. Shape Optimization Analysis

3.2.6. Topography Optimization

3.2.1. Topology Optimization - Density Based

Review the following requirements and limitations associated with performing a density-based optimization analysis. These limitations apply only to the optimization analysis and are not applicable to any downstream design validation systems. In addition, see the next section, Topology Optimization - Density Based Solution Methodology (p. 110), for a description of the theory behind the solution methodology used for density-based optimization analyses.

- Element Type Requirements (p. 109)
- Recommendations (p. 109)
- General Limitations (p. 109)
- Modal Analysis Limitations (p. 110)

- Linked Static Structural Analysis Limitations (p. 110)
- Linked Thermal Analysis Limitations (p. 110)

Element Type Requirements

The density-based optimization analysis supports Solid, Shell, and Plane elements in the optimization region. The application excludes any other element type used in this region from the optimization solution.

Recommendations

For a density-based optimization analysis, Ansys recommends that you:

- Use the mixable density method when you specify a **Thermal Condition** in an upstream Static Structural analysis to enable a more versatile and robust problem formulation.

General Limitations

The density-based optimization analysis method does not support:

- **Imported Plies** (Composites).
- Graphical view of Exclusions created by Direct FE boundary conditions.
- Cracks defined within the Fracture object.
- If you specify the **Solver Type** as **Optimality Criteria**, the application only supports Response constraints types of Mass and Volume and Manufacturing Constraints where only the Minimum Member Size can be specified.
- Use of Section Planes with **Topology Density** and **Topology Elemental Density** results. Note that the **User Defined Result** does support this feature.
- Pre-stressed Modal analysis.
- Large-deflection effects in a Static Structural analysis (**NLGEOM, ON**).
- Nonlinear contacts (supported contact includes **Bonded** and **No Separation**).
- Axisymmetric model when you wish to define a Global Stress Constraint (p. 38) and Local von-Mises Stress Constraint. (p. 38)
- Any Thermal system linked to the upstream Static Structural system.
- A solution run on high performance computing (HPC) using Distributed Ansys across multiple machines, when:
  - You have **Thermal Condition** included in a Static Structural analysis linked to a Structural Optimization analysis.
  - You want to optimize a Static Structural analysis that is linked to an upstream Steady-State Thermal analysis.

– A Structural Optimization that includes a **Reaction Force Constraint**.

Modal Analysis Limitations

When linked to a Modal analysis, the density-based optimization analysis does not support:

- Damping when the **Damped** property set to **Yes** (and therefore also the Campbell Diagram chart).
- **Unsymmetric** solver selection.
- **Optimality Criteria** solver selection.

Linked Static Structural Analysis Limitations

When linked to a Static Structural analysis, the density-based optimization analysis does not support the following boundary conditions:

- Joint Load.
- Fluid Solid Interface.
- EM Transducer.

Linked Thermal Analysis Limitations

For a density-based optimization analysis linked to an upstream Steady State Thermal analysis, the application does not support the use of the Ansys Remote Solve Manager (RSM) to submit jobs on a remote machine.

3.2.1.1. Topology Optimization - Density Based Solution Methodology

Review the following for the theory behind the solution methodology for density-based optimization analyses:

- Sequential Convex Programming (p. 111)
- Optimality Criteria (p. 111)
- Solution Methodology for Natural Frequencies (p. 111)
- Solution Methodology for Stress Constraints (p. 112)
- Solution Convergence Criteria (p. 113)
- Topology Optimization with Thermal Condition (p. 114)
- References Materials (p. 114)

Sequential Convex Programming

The Sequential Convex Programming method (SCP), see Zillober[1][2][3], is an extension of the method of moving asymptotes (MMA), see Svanberg[4]. The Sequential Convex Programming method requires the derivatives of all functions present in the Topology Optimization problem. MMA is a nonlinear programming algorithm that approximates a solution for a Topology Optimization problem by solving a sequence of convex and separable subproblems. These subproblems can be solved efficiently due to their special structure.

The Sequential Convex Programming method extends MMA to ensure convergence by rejecting steps that do not lead to an optimal solution of the underlying problem. The test for acceptance is done by a merit function and a corresponding line search procedure, see Zillober[5]. The goal of the merit function is to measure the progress and enable the objective function and the constraints to be combined in a suitable way.

Optimality Criteria

The Optimality Criteria method can be used to solve density-based optimization problems with a simple compliance objective that uses a volume or mass constraint. The Optimality Criteria method is an iterative solver, see Bensoe and Sigmund[6]. The Optimality Criteria method should not be used for a Modal Analysis.

Note:

The following limitations apply when using the **Optimality Criteria** Solver Type:

- Only supports the **Compliance** (Structural) setting for the **Response Type** column of the **Objective** object worksheet.
- Only **Volume** and **Mass** constraints are supported.
- The **Manufacturing Constraint** is supported where only the **Minimum** property for the **Member Size** constraint subtype can be specified.

Solution Methodology for Natural Frequencies

When performing density-based optimization analysis with supported natural frequencies, you can specify the frequency as either an objective or as a constraint. A single natural frequency or a weighted combination of several natural frequencies can be defined using the Objective object.

[1] Zillober, Ch., A globally convergent version of the method of moving asymptotes, Structural Optimization, 6(3):166-174, 1993.
[2] Zillober, Ch., A combined convex approximation — interior point approach for large scale nonlinear programming, Optimization and Engineering, 2(1):51-73, 2001.
[3] Zillober, Ch., SCPIP - an efficient software tool for the solution of structural optimization problems, Structural and Multidisciplinary Optimization, 24(5), 2002.
[4] Svanberg, K., The Method of Moving Asymptotes — a new method for structural optimization, International Journal for Numerical Methods in Engineering, 24:359-373, 1987.
[5] Zillober, Ch., Global convergence of a nonlinear programming method using convex approximations, Numerical Algorithms, 27(3):256-289, 2001.
[6] Bendsoe, M.P. and Sigmund O., Topology Optimization: Theory, Methods and Applications, Springer, Berlin, 2003.

The aim of the optimization is to maximize these frequencies according to their weights (as defined in the Worksheet).

In addition, you can add a single natural frequency as a constraint and define a lower and an upper bound on the frequency. The solver will guarantee, if possible, that this frequency lies within the specified range.

If the design objective is to optimize a frequency, then all of the repeating frequencies are optimized simultaneously.

It is important to note that the mode shapes will change during the iterative solution procedure and that there is no tracking with respect to the initial mode shape. Only the actual value of the specified natural frequency is considered. This means at the final iteration the mode shape may change dramatically in comparison to the initial shape of the optimized mode.

Because the underlying solver is sensitivity based, problems with natural frequencies have to be handled with care. The problem is not differentiable in the common sense, such as a case of multiple eigenvalues. Instead, derivatives for multiple eigenvalues have to be calculated in a special way. Since the mode shapes are not unique for multiple eigenvalues, additional effort is necessary to get sensitivities that are independent of the mode shapes. In order to obtain unique sensitivities for these eigenvalues, an additional eigenvalue problem has to be solved for each optimized element, see Seyranian[7].

Solution Methodology for Stress Constraints

When working with topological optimization for global stress constraints, and local stress constraints applied to more than one element, you can specify an upper bound $\overline{\sigma}$ on the stress that has to be satisfied by all elements. Theoretically, this requires the solution of an optimization problem with *n* stress constraints, where *n* denotes the number of optimized elements taken into account. Because the computational effort would be too great to achieve this, a relaxed reformulation has to be applied. In order to keep the complexity of the optimization problem low, a set of elements is represented by one constraint instead of individual ones. This technique divides the original design space into clusters. The maximum stress value with respect to all elements in the cluster/set *S* has to satisfy the following:

$$\max_{e \in S} \sigma_e \leq \overline{\sigma} \ ,$$

Where σ_e is the elemental mean value of the equivalent (von-Mises) stress of element *e* in set *S*. Since the maximum leads to a non-differentiable problem formulation, the *p*-norm is used to approximate the actual maximum instead. Applying the differentiable *p*-norm leads to:

$$\max_{e \in S} \sigma_e = \|\sigma_S\|_\infty \leq \|\sigma_S\|_p,$$

Where σ_S denotes the vector of all stress values of the elements in set *S*. Note that the *p*-norm overestimates the actual maximum. To stabilize the solver different regularization techniques are used in the literature. In Holmberg[8], a fixed scaling parameter is introduced. With factor: $1/\sqrt[p]{n_S}$ that leads to:

[7] Seyranian, A.P., Lund E., and Olhoff N., Multiple eigenvalues in structural optimization problems, Structural Optimization, 8:207-227, 1994.

[8] Holmberg E., Torstenfelt B., and Klarbring A., Stress constrained topology optimization, Structural and Multidisciplinary Optimization, 48(1):33-47,2013.

$$\frac{1}{\sqrt[p]{n_S}} \|\sigma_S\|_p \leq \|\sigma_S\|_\infty \leq \|\sigma_S\|_p,$$

where n_S is the number of elements in the considered set. In previous releases this approach was used. Since at the final iteration, the maximum stress of some optimized elements might be greater than the user-defined upper bound $\overline{\sigma}$ of the global/local stress constraint, the validation might fail.

To improve the accuracy of the approximation, a different regularization techniques is available. In Le[9], the *n*normalized maximum approximation is used to measure the stress value of a cluster/set. Here the *p*-Norm is also applied but instead of using a fixed factor an adaptive factor c_i is introduced. In each iteration i the factor c_i is modified. This technique leads to:

$$\lim_{i\to\infty} c_i \|\sigma_S^i\|_p = \|\sigma_S^i\|_\infty$$

Where denotes the iteration. This approach improves accuracy as well as the estimate of the stress value.

Solution Convergence Criteria

The density-based optimization solver approaches a stationary point where all constraints are satisfied within a tolerance of 0.1 percent of the defined bound. This tolerance is defined by the **Convergence Accuracy** property (see Specify Analysis Settings (p. 6)).

To simplify the notation, assume that only one constraint c exists. The optimality conditions of the Topology Optimization problem can be stated with the following equation:

$$\|\nabla L(\rho,\mu)\| = 0$$

Where $L(\rho,\mu)$ denotes the Lagrange function. The Lagrange function is defined by:

$$L(\rho,\upsilon) = f(\rho) - \mu c(\rho)$$

Where μ is the Lagrange multiplier corresponding to the constraint c, and f is the objective function to be either maximized or minimized. The solver will stop as soon as the desired tolerance is achieved, where: $\in > 0$, as defined here:

$$\|\nabla L(\rho,\mu)\| = \in$$

Because approaching this stationary point can require a large number of iterations, a relaxed convergence criterion is used. The optimization stops as soon as the following equation has three successive iterations. In this equation, ρ_i denotes the vector of pseudo densities of the $i-\text{th}$ iteration.

$$\left|\frac{f(\rho_i) - f(\rho_{i-1})}{f(\rho_i)}\right| \leq \in$$

Note that three successive iterations are considered as the underlying solver is stabilized by a line search procedure. This line search procedure might lead to small changes with respect to the pseudo

[9] Le C., Norato J., Bruns T., Ha C., Tortorelli D. Stress-based Topology Optimization for continua, Structural and Multidisciplinary Optimization, 41(4):605{620, 2010.

densities as well as small changes to the objective function. It is possible that the convergence tolerance is satisfied for one iteration but the next iteration leads to a significant improvement of the objective function. Due to the relaxed stopping criterion, the optimization might terminate too early. In this case, the optimization should be rerun with a smaller tolerance.

Topology Optimization with Thermal Condition

The optimization is influence by the thermal condition according to the following equation[10]:

Linear static equilibrium in finite element system including both mechanical and thermal loading is given by:

$$K(x)U = F^m + F^{th}(x),$$

Where:

$K(x)$ = stiffness matrix

U = displacement vector

F^m = externally applied mechanical loading

$F^{th}(x)$ = thermal load vector.

The nodal load vector due to temperature effects for the element may be written as:

$$f_e^{th} = \int_{\Omega_e} B_e^T C_e \in_e^{th} d\Omega_e$$

Here B_e is the element strain-displacement matrix, C_e is the element elasticity matrix, and $\in_e^{th}$ is the thermal strain vector for the element given by:

$$\in_e^{th} = \alpha(x_e)\Delta T_e \phi^T$$

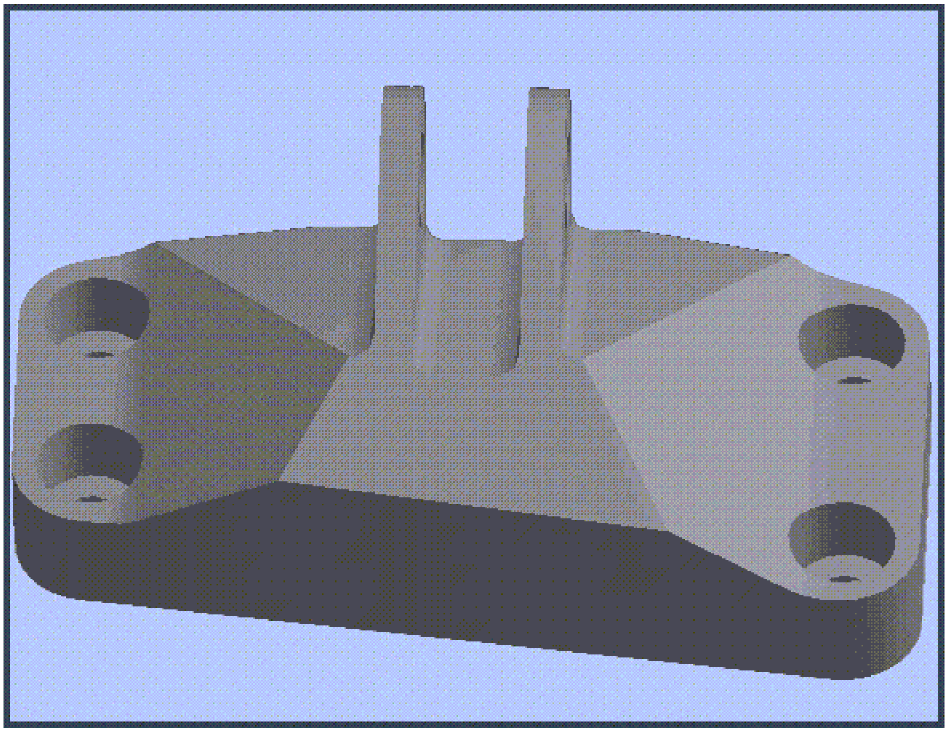
With $\alpha(x_e)$ is the thermal expansion coefficient of the material, ΔT_e is the temperature change on the element, ϕ and is [1,1,1,0,0,0] for three-dimensions and [1,1,0] for two-dimension.

References

3.2.2. Topology Optimization - Level Set Based

The **Topology Optimization - Level Set** method deals with the boundary of the shapes during the optimization process. It is a physics driven optimization that is based on a set of loads and boundary conditions provided by either a single preceding analysis or multiple preceding analyses. Using this method, the application computes an optimal shape in the design domain that can be applied to a selected region of your model and that can include specific design objectives and constraints. Here is an animated example of the Level Set topology optimization method.

[10]Joshua D. Deaton, Ramana V. Grandhi: "Stress-based Topology Optimization of Thermal Structures",10th World Congress on Structural and Multidisciplinary Optimization, 2013, Orlando, Florida, USA.



The Level Set optimization workflow has some specific considerations as compared to the other methods, as described below. Use these topics in combination with the general workflow to ensure the proper completion of your analysis.

Prerequisites and Requirements

All upstream Static Structural and Modal analyses being used in your optimization problem must already be defined before you solve the optimization analysis

Supported Mesh Elements

Only 3D elements are supported for optimization.

Geometric Analysis

For the **Objective** and the **Response Constraint** objects, the Level Set method supports the following settings for the **Response Type** and **Response** properties:

- **Mass**
- **Volume**
- **Center of Gravity**
- **Moment of Inertia**

Configuring Static Structural Analysis

Extend Compliance

For the Level Set method when you set the **Response Type**/**Response** for the **Objective** Worksheet or a **Response Constraint** object to **Compliance**, Static Structural analyses supports the combination of force-based and displacement-based loading as well as thermal loading. This context for Compliance is described by:

$$J(\Omega)=\tfrac{1}{2}\left[\int_{\Omega} A\left(e(u)-e^{th}\right):e^{th}dx+\int_{\Omega} f.udx+\int_{\Gamma_N} g.uds+\int_{\Gamma_u} \lambda_u.u_0 ds\right]$$
$$=\left[-\tfrac{1}{2}\int_{\Omega} A\left(e(u)-e^{th}\right):\left(e(u)-e^{th}\right)dx+\int_{\Omega} f.udx+\int_{\Gamma_N} g.uds\right]$$

Where:

$e(u)=\tfrac{1}{2}\left(\nabla u+\left(\nabla u\right)^t\right)$ is the total strain vector.

e^{th} is the thermal strain vector.

$\left(e(u)-e^{th}\right)$ is the elastic strain vector.

$\sigma(u)=A\left(e(u)-e^{th}\right)$ is the stress vector.

$\left(f,g\right)$ are the external loads (resp. volume and surface).

$\left(\lambda_u,u_0\right)$ are the reaction force and the prescribed displacement.

These formulas are equivalent and are based on the potential energy. The compliance is a self-adjoint response meaning that no adjoint problem must be solved. The compliance is always computed over the whole model.

Displacement-based Criterion (will be deprecated and fully replaced by the User Defined Criterion)

Context for displacement-based response:

- For a singular node selection, the response = $J(\Omega)=\left|u_k^i\right|$ (i-th node, k-axis). You can define an upper limit for each direction.
- For multiple node selection, the response = $J(\Omega)=\frac{1}{|B|}\int_B \left|u_k\right|ds$ (the average of the absolute displacement along the k-axis).

Note:

When you specify this type of Response Constraint, the **Coordinate System** property for the object is read-only and automatically set to **Nodal Coordinate System**. The application uses the **Global Coordinate System** for this setting.

Reaction Force Criterion (will be deprecated and fully replaced by the User Defined Criterion)

For a singular node selection, the response = $J(\Omega) = F_k^i$ (i-th node, k-axis). You can define an upper limit for each direction.

For multiple node selection, the response = $J(\Omega) = \sum_{i \in B} F_k^i$ (the RF along the k-axis).

Note:

When you specify this type of Response Constraint, the **Coordinate System** property for the object is read-only and automatically set to **Nodal Coordinate System**. The application uses the **Global Coordinate System** for this setting.

Review the Best Practices and Recommendations (p. 118) topic below for additional information for configuring your upstream analysis.

Configuring Static Structural Analysis

Generalized Thermal Compliance

For the Level Set method when you set the Response Type/Response for the Objective Worksheet or a Response Constraint object to Thermal Compliance, Steady-State Thermal analyses supports the combination of Heat Flux, Heat Flow, and Temperature, Convection, and Internal Heat Generation. This context for Thermall Compliance is described by:

$$J(\Omega) = \frac{1}{2}\left[-\int_{\Omega} k \nabla T . \nabla T dx - \int_{\Gamma_{convection}} h(T - T_{ambient})^2 ds + 2\int_{\Omega} f T dx + 2 \int_{\Gamma_{heatflow}} q_0 T ds \right]$$

Where:

(∇T) is the gradient of the temperature.

(k) the isotropic thermal conductivity.

$(h, T_{\alpha mb})$ the film coefficient and the ambient temperature of the convection condition.

f the internal heat generation.

q_0 the heat flux.

This is based on the potential energy. The thermal compliance is a self-adjoint response meaning that no adjoint problem must be solved. The thermal compliance is always computed over the whole model.

Configuring Modal Analysis

The Level Set method supports Frequency (Eigenfrequency) as the **Response**/**Response Type** setting. Review the Best Practices and Recommendations (p. 118) topic below for additional information for configuring your upstream analysis.

Manufacturing Constraint Definition

The Level Set method supports the following **Manufacturing Constraint Subtypes** (p. 48):

- **Member Size** (**Minimum** or **Maximum**): To properly represent the optimal shape, you should mesh your model such that Maximum Size of the Member Size is greater than four times the element average size.
- **Pull Out Direction**: You can further define the Direction for this constraint: **Along Axis** (default), **Opposite to Axis**, or **Both Directions**.

Note:

The moldability of the part could be lost during the optimization process.

- **AM Overhang Constraint** (p. 52): This constraint enables you to further define the **Build Direction** and **Overhang Angle**.

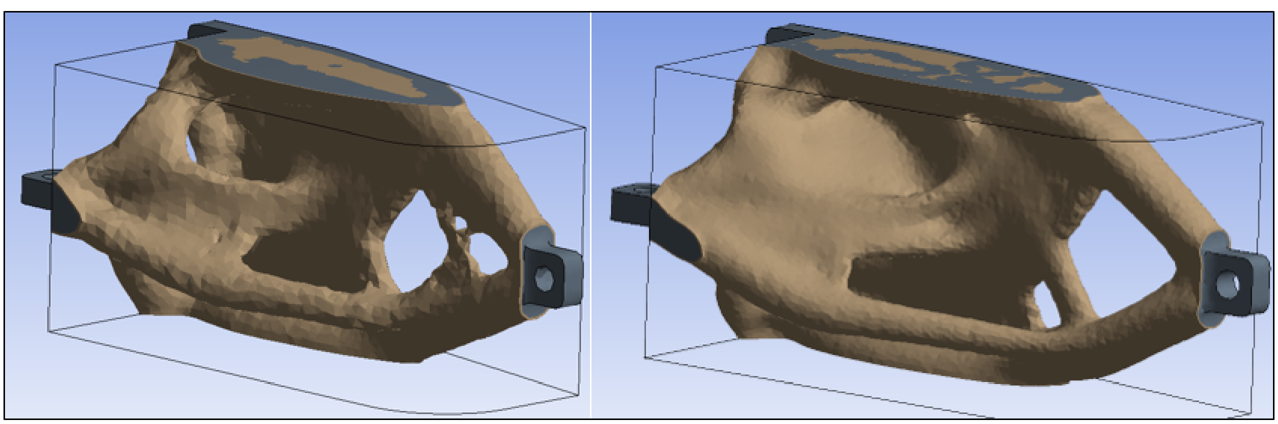
Best Practices and Recommendations

Review the following suggestions when performing this analysis.

Mesh Specifications

When specifying the mesh on your model, it is strongly recommended that you:

- Always use a uniform mesh (homogeneous element size). This enables you to capture the design with the same precision everywhere on the model. However, in the presence of thin regions, it may be necessary to refine the mesh locally in order to get at least three or four layers of elements.
- Make sure that you have a sufficiently fine mesh. If the final design shows geometric features as thick as an element size, it means that the mesh was not fine enough, as illustrated here.



This illustration displays 238,000 tetrahedrons on the left and 950,000 on the right. The feature on the left is very thin (one element size) and reaches the limit of the Level Set capability to properly capture the design. The finer mesh on the right provides an improved design.

Configuring a Static Structural Analysis

When specifying your upstream Static Structural analysis, note the following:

- A minimum stress problem can be realized by a void design (no material) if there is no stiffness constraint - if there is no mass, there is no stress. Therefore, Ansys recommends that you use stress criterion in combination with a stiffness criterion (nodal displacement, compliance, etc.).
- It is strongly encouraged that you specify an "exclusion zone" around the loading conditions (surface or node-based).
- If your solution experiences disconnected clamped parts, it may be a result of the optimization aiming to optimally distribute an amount of material. The algorithm sometimes chooses to save material by disconnecting clamped parts and/or to reinforce others. This characteristic of optimization is useful in order to identify useful and/or otherwise impractical fixed parts.

Configuring a Modal Analysis

You can control an eigenmode whose frequency always has the same ranking during the optimization process. If its ranking changes, the algorithm may face some difficulty.

3.2.3. Topology Optimization - Mixable Density

The mixable-density method is an expanded re-implementation of the density-based method, aiming to deliver the same capabilities as well as additions to these capabilities.

Furthermore, this development was motivated by the need to:

- Unify and to mutualize some components between all the optimization methods.
- Better align and streamline the density-method within the overall structural optimization framework.
- Gain consistency between all the optimization methods.

As a result, the mixable-density method:

- Delivers smoother results based on a new numerical scheme.
- Incorporates the benefits of other methods and conversely (cross fertilization process).
- Can be combined with other methods in an optimization problem enabling you to manage complex systems with dedicated requirements.

Review the following requirements and limitations associated with performing a mixable density-based optimization analysis. These limitations apply only to the optimization analysis and are not applicable to any downstream design validation systems. Go to a section topic:

- Element Type Requirements (p. 120)
- General Limitations (p. 120)
- Modal Analysis Limitations (p. 120)
- Linked Static Structural Analysis Limitations (p. 121)

Element Type Requirements

The density-based optimization analysis supports Solid elements in the optimization region. The application excludes any other element type used in this region from the optimization solution.

General Limitations

The mixable density-based optimization analysis method does not support:

- **Imported Plies** (Composites).
- Graphical view of Exclusions created by Direct FE boundary conditions.
- Cracks defined within the Fracture object.
- If you specify the **Solver Type** as **Optimality Criteria**, the application only supports Response constraints types of Mass and Volume and Manufacturing Constraints where only the Minimum Member Size can be specified.
- Use of Section Planes with **Topology Density** and **Topology Elemental Density** results. Note that the **User Defined Result** does support this feature.
- Pre-stressed Modal analysis.
- Large-deflection effects in a Static Structural analysis (**NLGEOM, ON**).
- Nonlinear contacts (supported contact includes **Bonded** and **No Separation**).
- Axisymmetric model when you wish to define a Global Stress Constraint (p. 38) and Local von-Mises Stress Constraint. (p. 38)
- Any Thermal system linked to the upstream Static Structural system.
- A solution run on high performance computing (HPC) using Distributed Ansys across multiple machines, when:
  - You have **Thermal Condition** included in a Static Structural analysis linked to a Structural Optimization analysis.
  - You want to optimize a Static Structural analysis that is linked to an upstream Steady-State Thermal analysis.
  - A Structural Optimization that includes a **Reaction Force Constraint**.

Modal Analysis Limitations

When linked to a Modal analysis, the mixable density-based optimization analysis does not support:

- Damping when the **Damped** property set to **Yes** (and therefore also the Campbell Diagram chart).
- **Unsymmetric** solver selection.

Linked Static Structural Analysis Limitations

When linked to a Static Structural analysis, the mixable density-based optimization analysis does not support the following boundary conditions:

- Fluid Solid Interface.
- EM Transducer.

3.2.4. Lattice Optimization Analysis

The Structural Optimization analysis offers a **Lattice Optimization** Optimization Type. This method enables you to compute an optimal variable density lattice distribution in your geometry. Use the following lattice specific topics in combination with the general workflow to ensure the proper completion of your analysis.

- Lattice Optimization Limitations (p. 121)
- Define Lattice Type, Density, and Cell Size (p. 122)
- Specifying Constraints (p. 123)
- Defining Results (p. 123)
- Creating the Lattice Geometry (p. 123)
- Lattice Validation with a Homogenization Model (p. 129)

Lattice Optimization Limitations

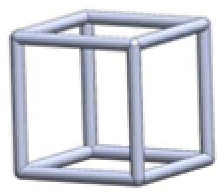
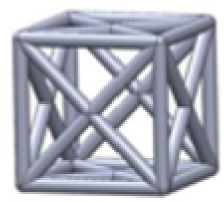
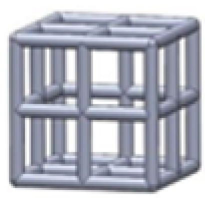
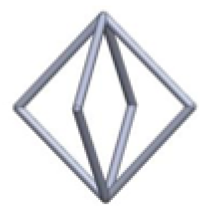
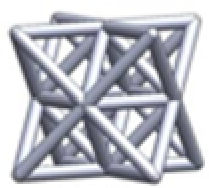
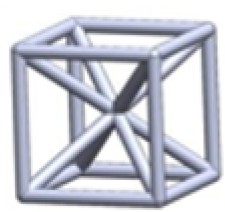
In addition to the Topology Optimization - Density Based (p. 108) method limitations, the lattice optimization has the following additional limitations. Lattice optimization does not support:

- Nonlinear Contact
- Modes with an Eigenfrequency equal to zero (based on an upstream Modal analysis)
- Thermal Loads
- Export Topology (STL file)
- Local Von-Mises Stress Constraint
- Reaction Force Constraint
- Criterion Constraint
- Member Size
- Pull Out Direction
- Extrusion
- AM Overhang Constraint

- Housing
- For the Octahedral 1 and Octahedral 2 lattice structures, note the following material property limitations:
  - A Poisson ‘s Ratio value greater than 0.32 can cause the interpolated material properties to become inaccurate.
  - A low Density value (< 0.05) can generate a negative Young's Modulus value.

Define the Optimization Region Object

Once you have specified the **Design Region** and the **Exclusion Region**, set the **Optimization Type** property to **Lattice Optimization**. Once selected, define the following lattice-specific properties.

<table>
<tr><th>Category</th><th>Properties/Options/Description</th></tr>
<tr><td>Lattice Type</td><td>Specify the Lattice Type property as one of the following:
<table>
<tr><td>Cubic (default)</td><td></td><td>Crossed</td><td></td></tr>
<tr><td>Midpoint</td><td></td><td>Octahedral 1</td><td></td></tr>
<tr><td>Octet</td><td></td><td>Octahedral 2</td><td></td></tr>
<tr><td>Diagonal</td><td></td><td></td><td></td></tr>
</table>
</td></tr>
<tr><td>Minimum Density</td><td>Specify a minimum density in order make sure the lattice structures are not too thin. The default value is <code>0</code>.</td></tr>
</table>

Category	Properties/Options/Description
Maximum Density	Specify a maximum density. The element will be considered as full for densities higher than the **Maximum Density**.
Lattice Cell Size	Specify the lattice cell size to be used when rebuilding the lattice geometry for printing.

Specifying Constraints

For this analysis, the:

- **Response Constraint** (p. 38) object supports **Mass Constraint** (default) or **Volume Constraint**, **Global Stress Constraint**, **Displacement Constraint**, **Center of Gravity Constraint**, **Moment of Inertia Constraint**, **Compliance**, and **Natural Frequency Constraint**.

- **Design Constraint** (p. 57) object supports the **Symmetry**, **Pattern Repetition**, and **Cyclic Repetition** constraints.

Defining Results

Similar to the optimization results (Topology Density (p. 61) and Topology Elemental Density (p. 65)), lattice optimization supports **Lattice Density** and **Lattice Elemental Density** results. These results produce nodal averaged results and element-based result values. A **Lattice Density** result object is inserted automatically. You can add additional objects by selecting **Lattice Density** or **Lattice Elemental Density** from the **Result** group on the **Solution** context tab or by right-clicking the **Solution** folder (or in the **Geometry** window) and selecting **Insert** > **Lattice Density**/**Lattice Elemental Density**.

Result Display Feature

A lattice analysis automatically inserts a **Lattice Density Tracker** as a child object of the **Solution Information** object that enables you to view the optimization of the model during the solution.

Results Display Limitations

These result types do not support some of the display options available from the **Geometry** drop-down menu on the **Result** Context tab, including **Exterior**, **IsoSurfaces**, and **Capped IsoSurfaces**.

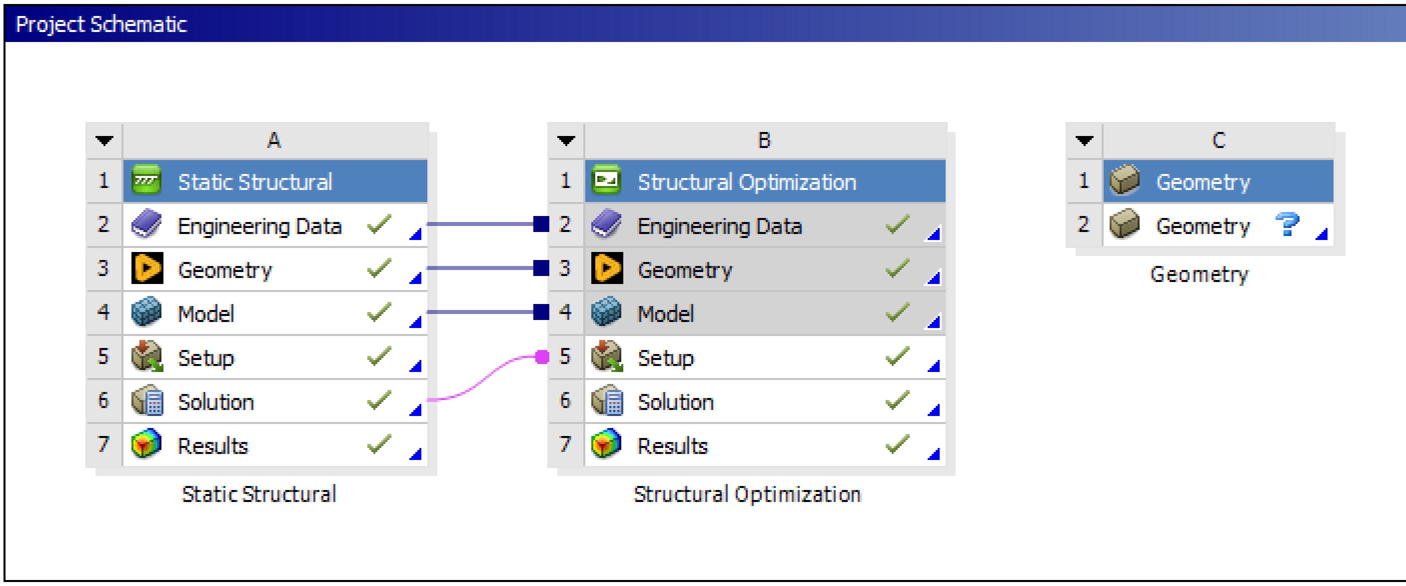
Creating the Lattice Geometry

To make the optimized results available to a downstream geometry system, you need to create the new system on the Workbench **Project Schematic** and link the **Results** cell of your lattice optimization analysis to the **Geometry** cell of a new downstream system, either a **Geometry** component system or the **Geometry** cell of another analysis system. An example of this configuration is illustrated below.

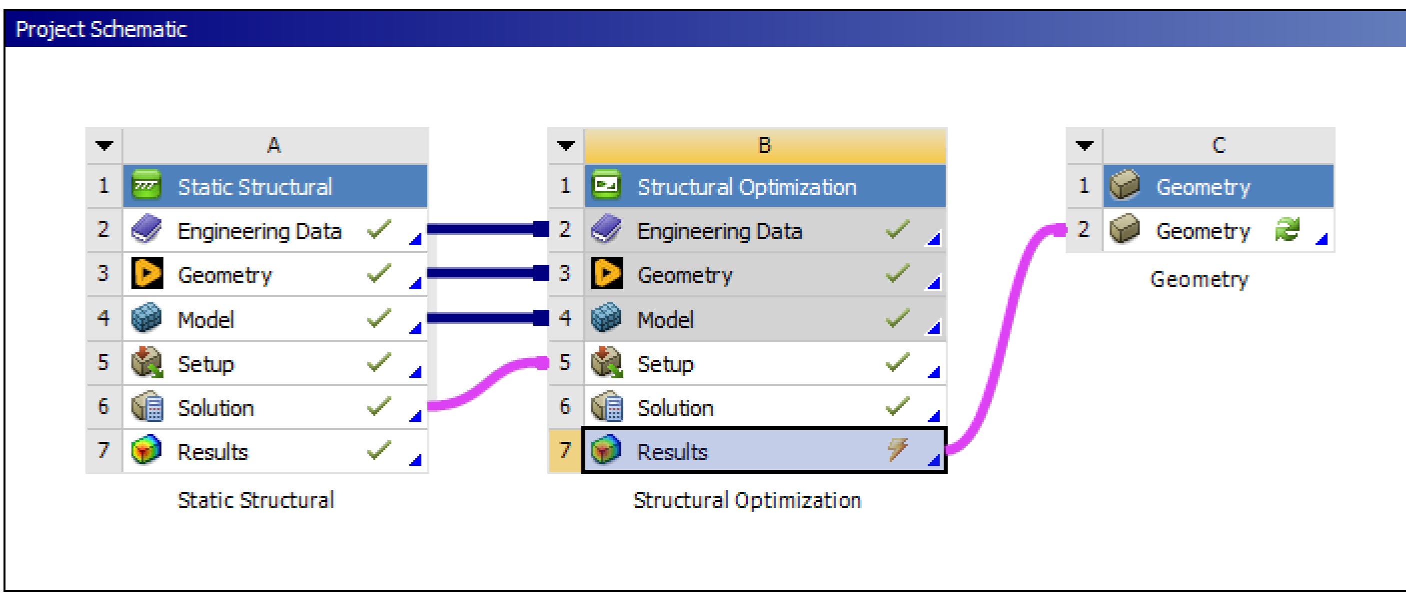
The lattice type, cell size, and density information are automatically transferred when linking a lattice optimization analysis to a **Geometry** system. Opening your new lattice geometry in the Discovery Modeling application enables you to further modify aspects of your geometry.

Procedure

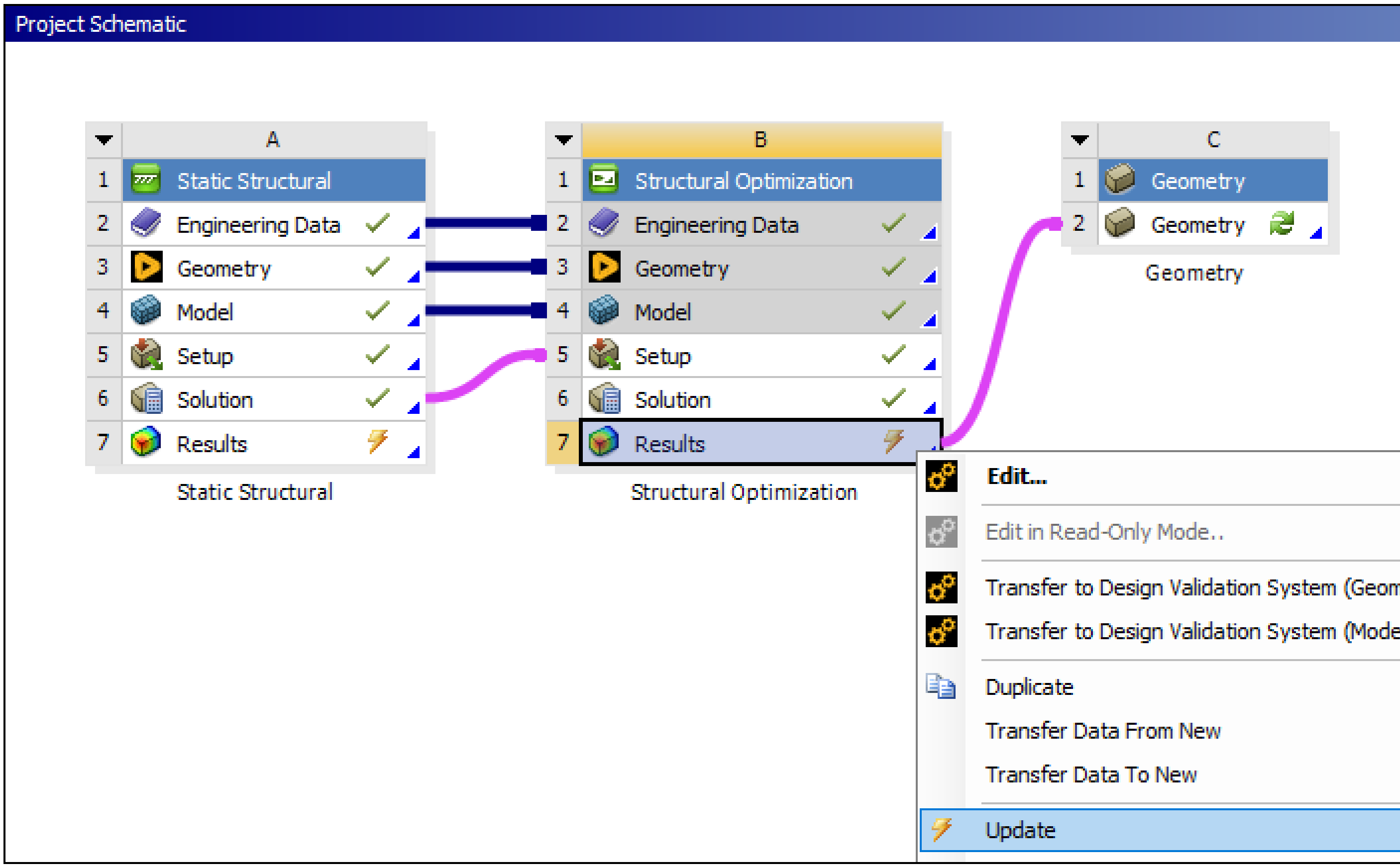
1. From the Workbench **Project Schematic**, create a new downstream **Geometry** system, as illustrated below.



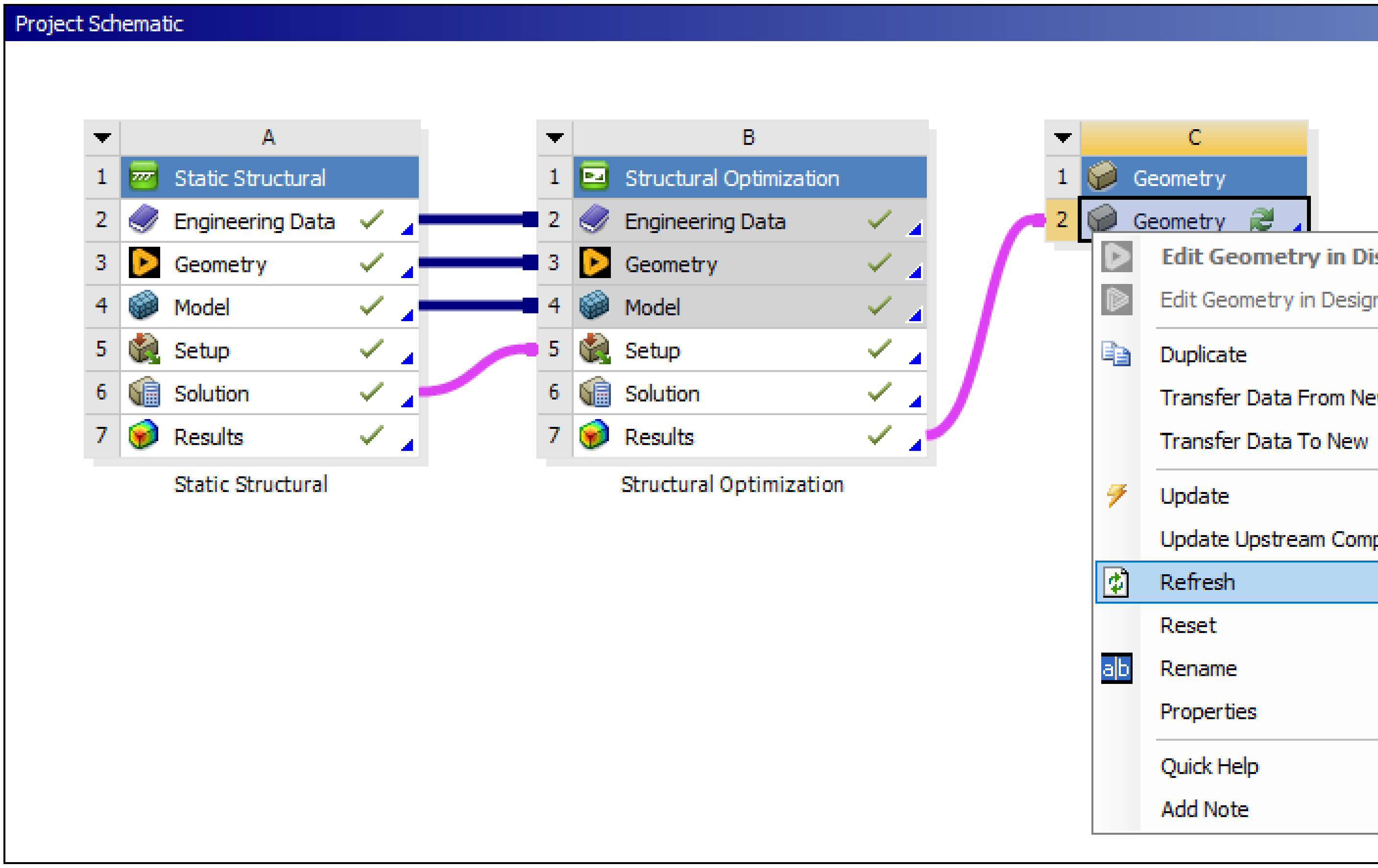
2. Link the **Results** cell of your lattice optimization analysis to the **Geometry** cell of the downstream system.



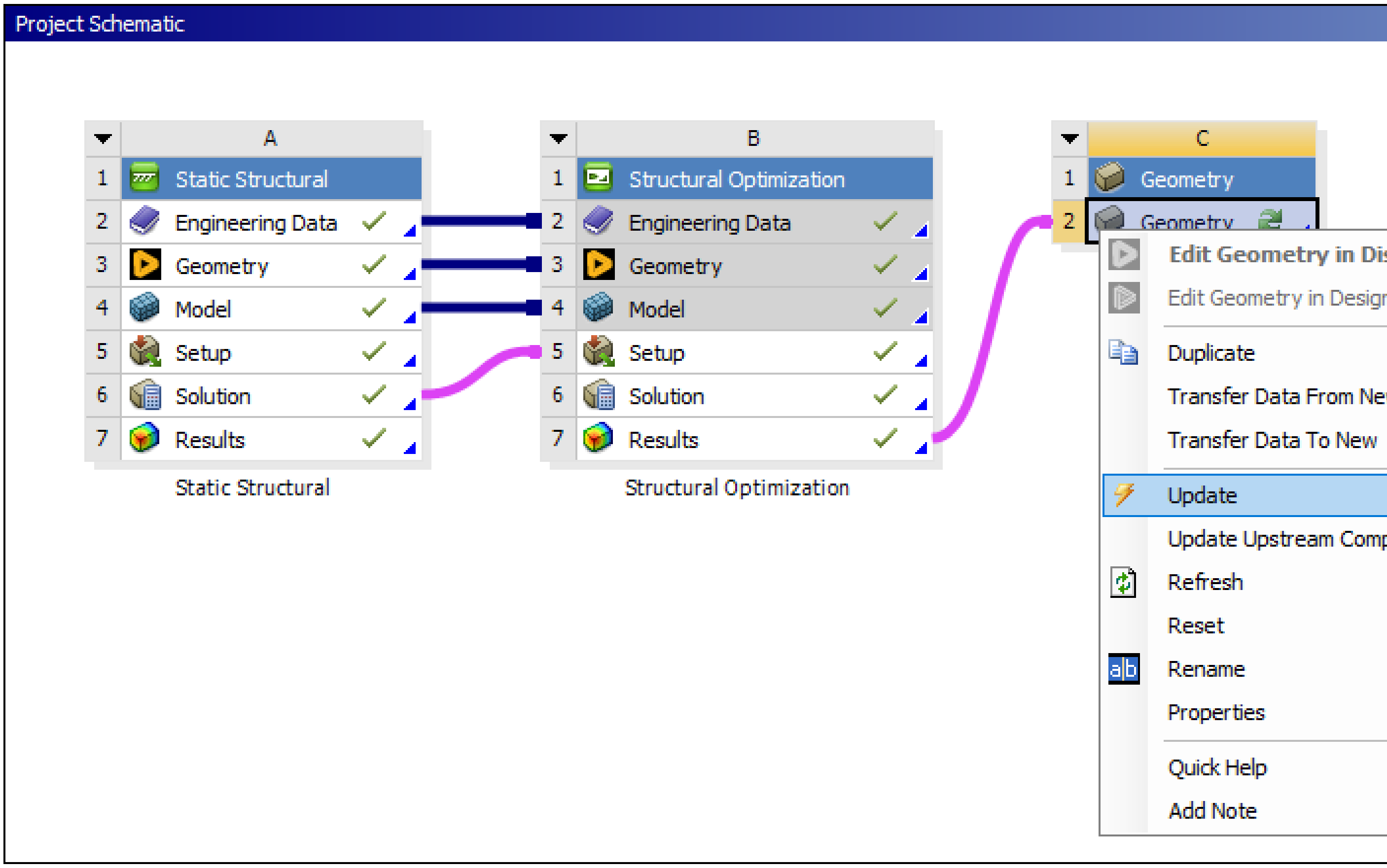
3. Right-click the **Results** cell of your lattice optimization analysis and select **Update**.



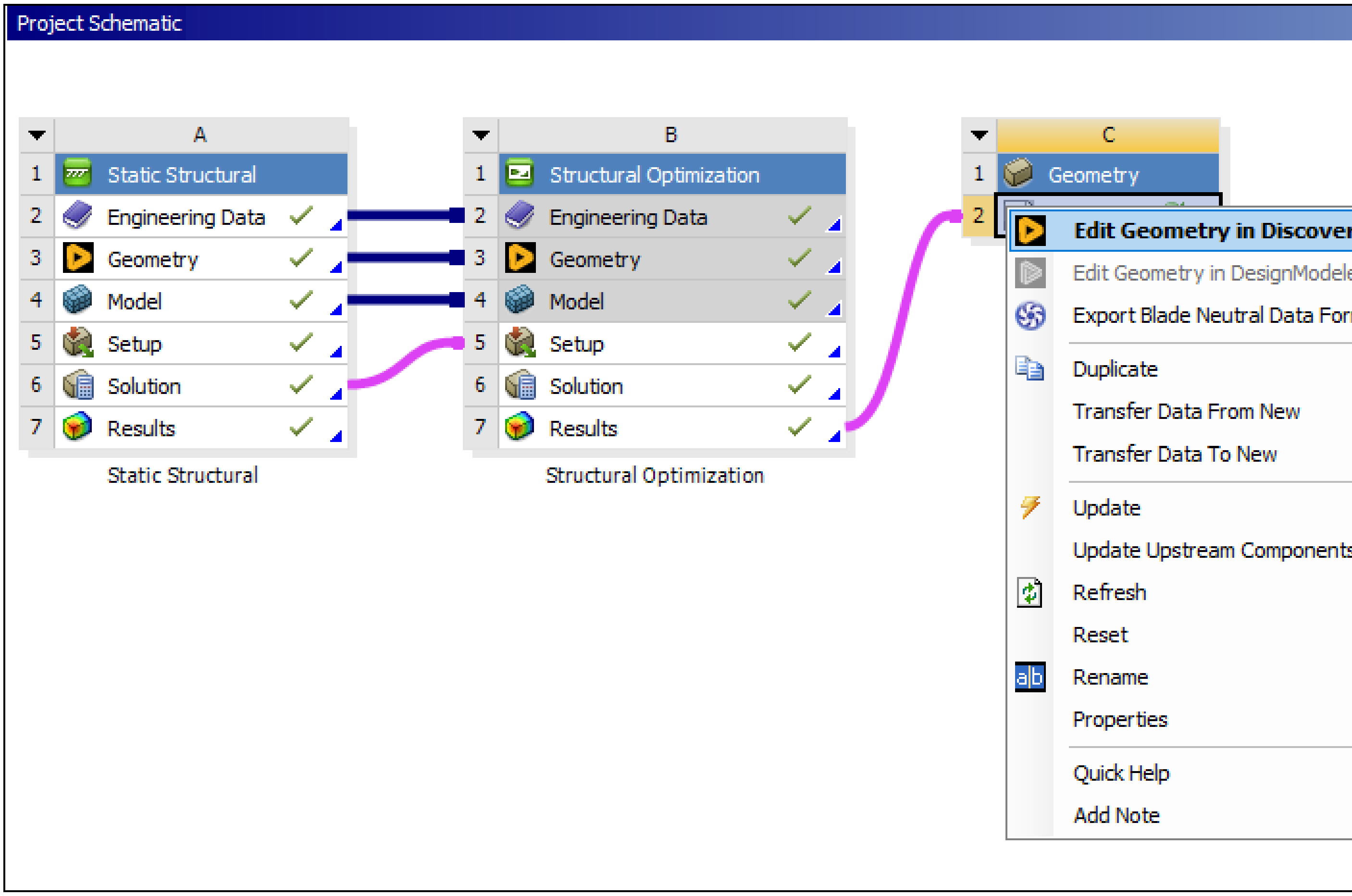
4. Right-click the **Geometry** cell of the downstream system and select **Refresh**.



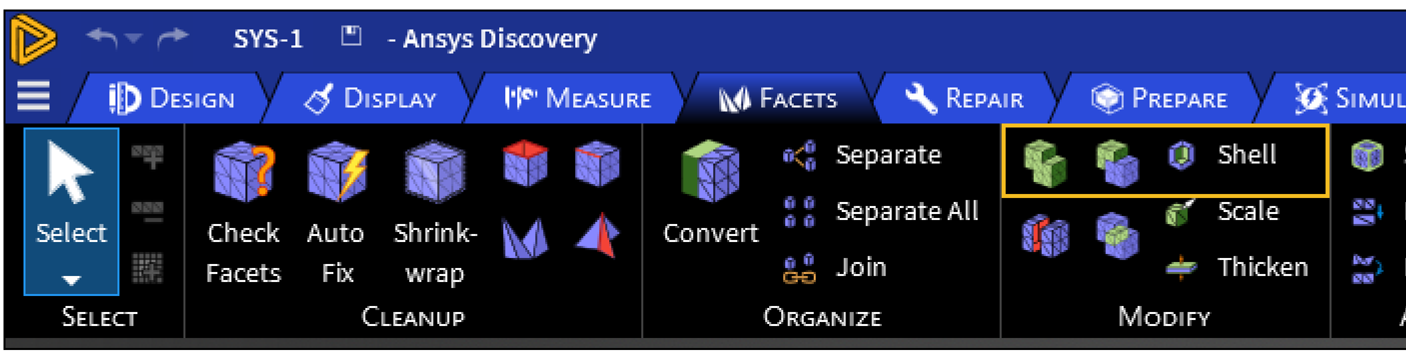
5. Right-click the **Geometry** cell of the downstream system and select **Update**.



6. Open Discovery Modeling from the **Geometry** cell of the downstream system.



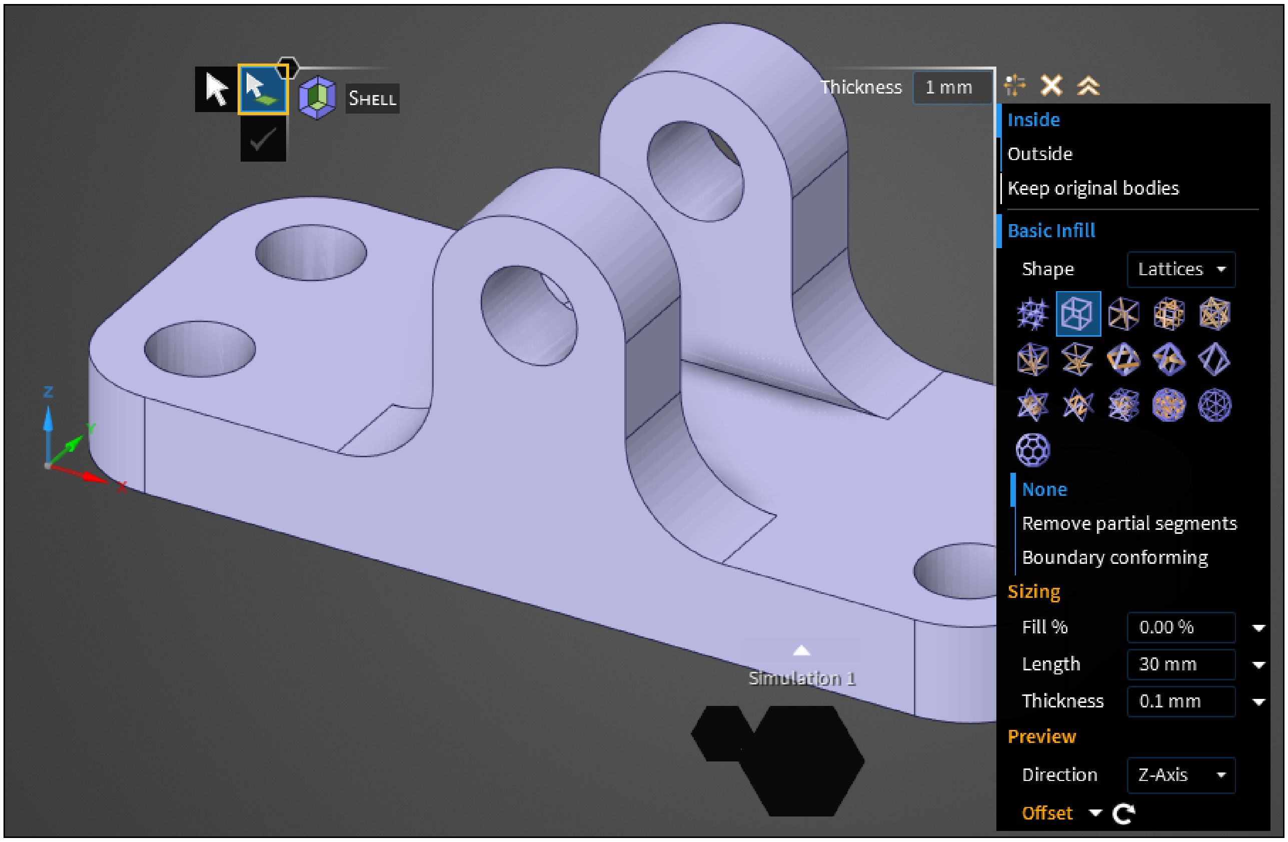
7. In SpaceClaim, select **Shell** tool from the **Facets** tab.



8. From the **Shell** Options tool, 1) choose **Basic** as the **Infill** type, and then 2) select the Solid body in the structure tree, and then 3) select Use Density Attributes. The Lattice Shape from the drop-down menu is automatically chosen. This includes the lattice type you specified in the optimization analysis in Mechanical.

Note:

Using the **Select Faces or Facets** tool (highlighted below) from the design window, you can choose to selectively exclude faces from the original body from this outer shell. This enables the lattice infill to extend all the way to the boundary of the part on that face.



9. A green check mark in the design window verifies the operation. The density distribution from the lattice optimization is now mapped onto the body and a faceted body is generated.

Lattice Validation with a Homogenization Model

For a Lattice Optimization analysis, the steps to create a downstream validation are slightly different than for the other optimization methods available in the application.

Note:

For the upstream system, the **Output Controls** property **Export Knockdown Factor** must be set to **Yes**. This is the default setting.

Duplication

When you use the **Duplicate** option on the **Solution** cell, you include all (loading, results, etc.) objects defined in the upstream system.

1. Right-click the **Solution** cell of the upstream system and select **Duplicate**. A new system is placed into the schematic. This new system includes all environmental conditions defined in the upstream system, such as loading conditions or results.

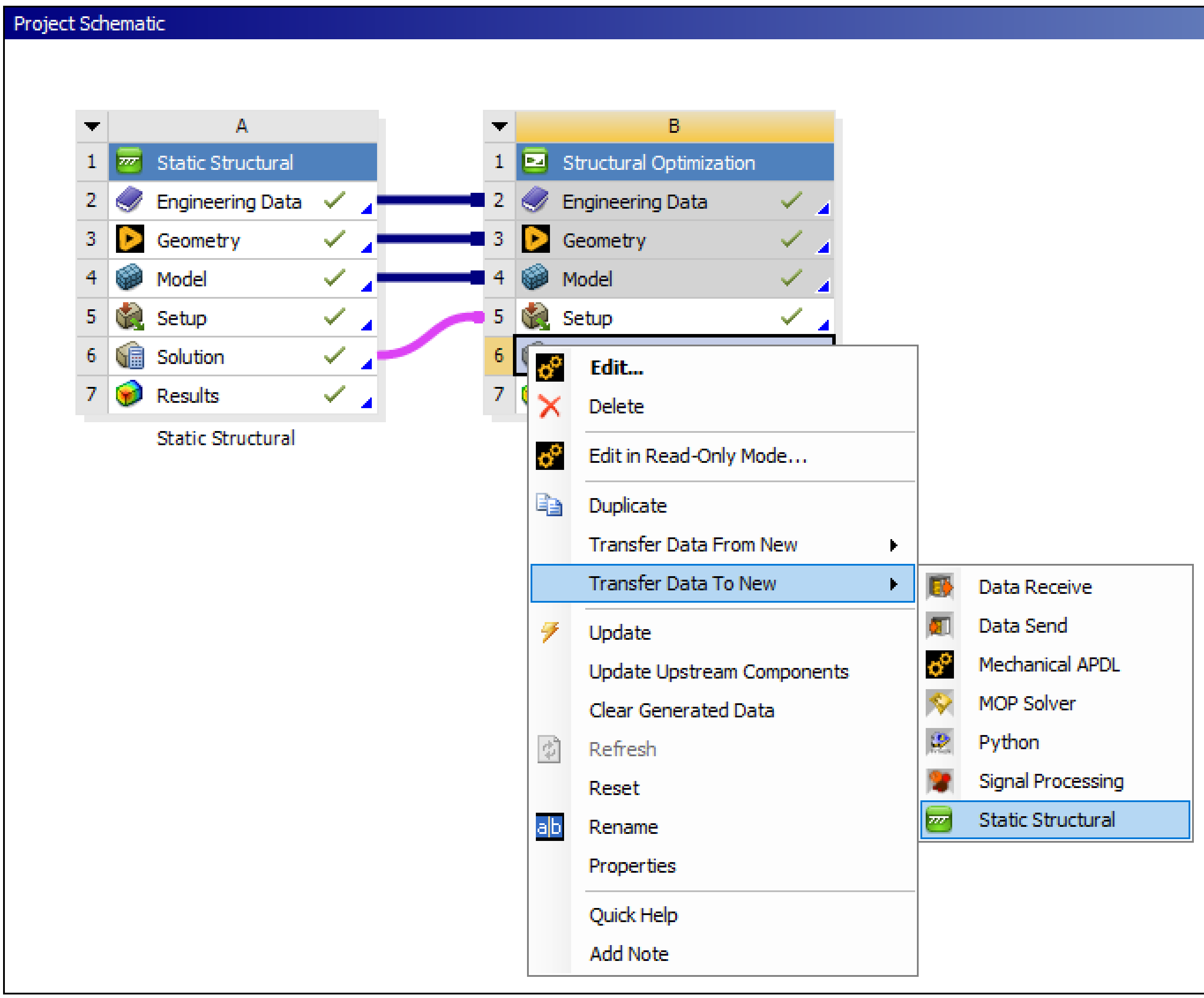
2. Drag and drop the **Solution** cell of the Structural Optimization analysis onto the **Setup** cell of the new system. The application properly links the systems together.

3. Right-click the **Setup** cell of the new system and select **Update**. Your new system is ready for a validation analysis.

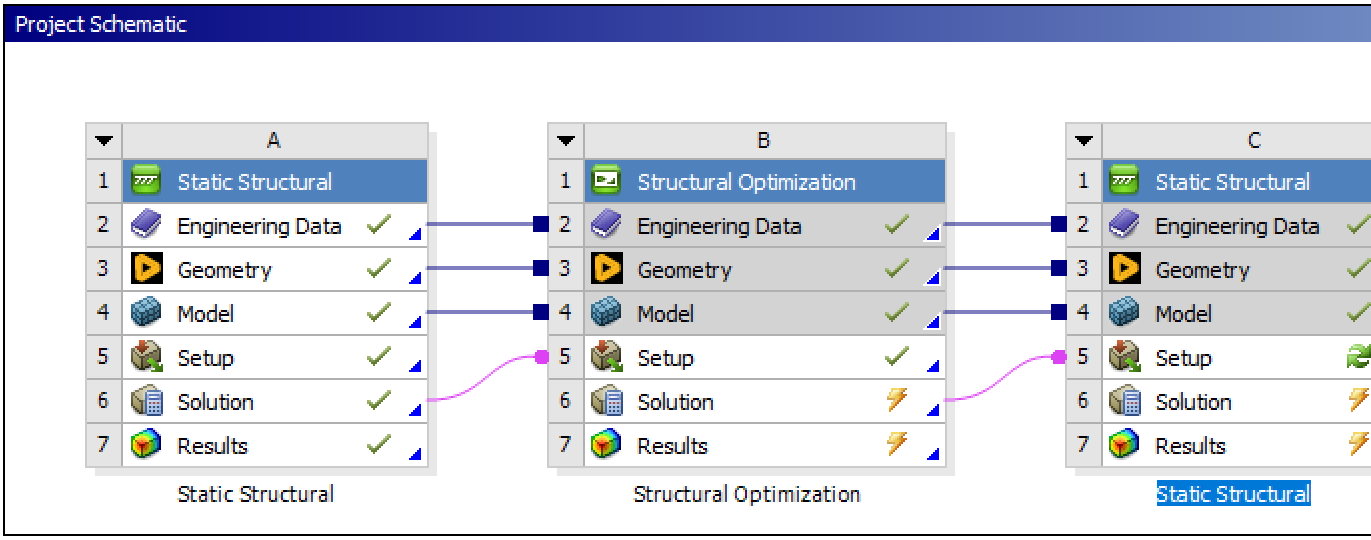
Transferring Data

You can also use the system option **Transfer Data To New**. When used, none of the environmental conditions defined in the upstream system, such as loading conditions or results are included in the new system. The steps to use this method are described below.

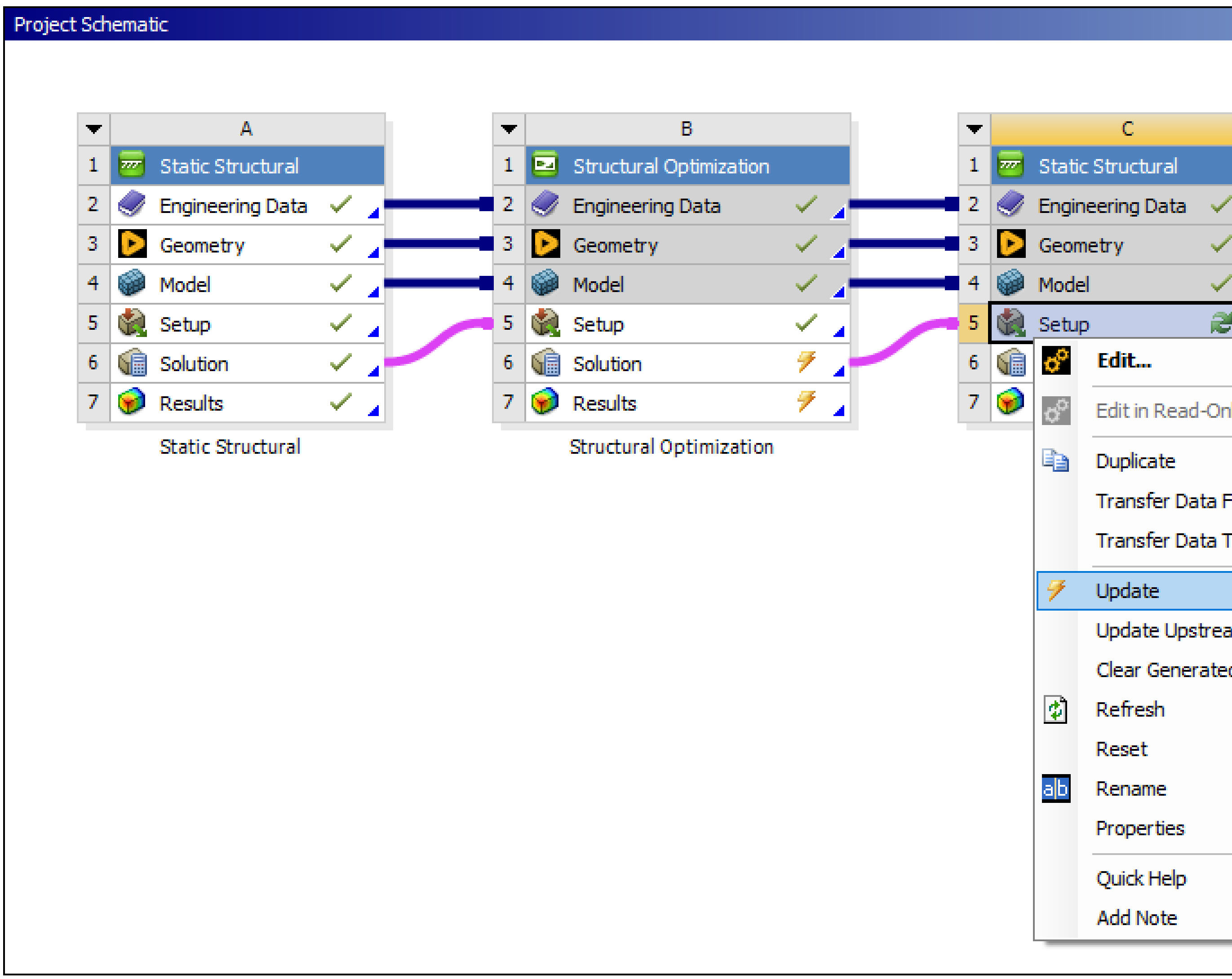
1. Right-click the **Solution** cell of the Structural Optimization analysis and select **Transfer Data to New** > Analysis System type.



A new system in inserted and is linked accordingly.



2. Right-click the **Setup** cell of the new system and select **Update**. Your new system is ready for a validation analysis.



3.2.5. Shape Optimization Analysis

Introduction

The **Shape Optimization** option of the **Optimization Type** property enables shape optimization using mesh node relocations. As with the other optimization methods, this is a physics driven optimization based on a set of loads and boundary conditions provided by either a single preceding analysis or multiple preceding analyses.

Using this method, the application computes an optimal shape in the design domain that you can apply to a selected region of your model and that also includes specific design Objectives and Constraints.

Upstream System Recommendations

Review the following to properly prepare your upstream systems. Shape Optimization has the same capability as the Level Set method. Any differences are mentioned when necessary.

Geometric Analysis

Review the Geometric Analysis (p. 115) topic in the *Prerequisites and Requirements* content in the *Topology Optimization - Level-Set Based* section for the supported **Response Type** and **Response** properties when performing a geometric analysis.

Static Structural Analysis

Review the Configuring Static Structural Analysis (p. 115) topic in the *Prerequisites and Requirements* content in the *Topology Optimization - Level-Set Based* section for the supported **Response Type**/**Response** for the **Objective** Worksheet or a **Response Constraint** object to **Compliance**. Static Structural analyses supports the combination of force-based and displacement-based loading as well as thermal loading.

In addition, when specifying your upstream Static Structural analysis, note that any surface of the optimizable body that is scoped to boundary conditions (fixed displacements, loads, bonded contacts, etc.) must be defined in the **Exclusion Region**.

Modal Analysis

This method supports Frequency (Eigenfrequency) as the **Response**/**Response Type** setting.

In addition, when specifying your upstream Modal analysis, note that you can control an eigenmode whose frequency always has the same ranking during the optimization process. If its ranking changes, the algorithm will face some difficulty.

Manufacturing Constraint

For this analysis method, only the **Member Size (Maximum)** property is available. To properly represent the optimal shape, you should mesh your model such that Maximum Size of the Member Size is greater than four times the element average size.

Application Differences

Note the following when using the **Shape Optimization** method.

Specifying the Mesh

This analysis method supports 3D tetrahedron solid elements only in the Optimization Region - all other element types are excluded.

Important:

When specifying the mesh on your model, it is strongly recommended that you:

- Always use a uniform mesh (homogeneous element size). This enables you to capture the design with the same precision everywhere on the model.
- Make sure that you have a sufficiently fine mesh.

Specifying Optimization Type

You use the **Optimization Region** object to select a region of your geometry on which to perform optimization as well as the optimization method to be used.

1. In order to scope the optimization regions using the **Shape Optimization** method, you need to first generate the mesh.
2. Specify the **Design Region**. The properties of the **Design Region** category enable you to define the geometry as a Geometry Selection or a Named Selection. This is the region that you wish to optimize.
3. Specify the **Exclusion Region**. The properties of the **Exclusion Region** category enable you to specify a region (geometric entities or elements) to be excluded from optimization. You specify excluded regions using defined Boundary Conditions, Geometry Selection, or a Named Selection.

   The surfaces scoped to boundary conditions (such as Fixed Support, Force, Bonded Contact, etc.) must be included in the scoping of the **Exclusion Region**.

   Note:

   Bordering the scoping of your defined **Exclusion Region** is a "buffer zone." The area is a transition region where the deformation is less permissive. This enables a smoother result.

   ---

4. Set the **Optimization Type** property to **Structural Optimization**. Specify the following additional properties as needed:

   - **Move Limit Per Iteration**: This property enables you to define how far each node can move at each iteration. It must be defined in length units, such as one element size.

By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.

Note:

Ansys recommends that you use a value smaller than the average element size and smaller than the **Total Move Limit** property setting.

- **Total Move Limit**: This property enables you to define how far each node can move in total. It must be defined in length units, such as three element sizes. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.
- **Mesh Deformation Control**: This property enables you to define how much the mesh can be stretched. It is an additional control to avoid element distortion. This unit-less value is a sort of penalty factor that ranges from 0 (no control) to 1.0. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.

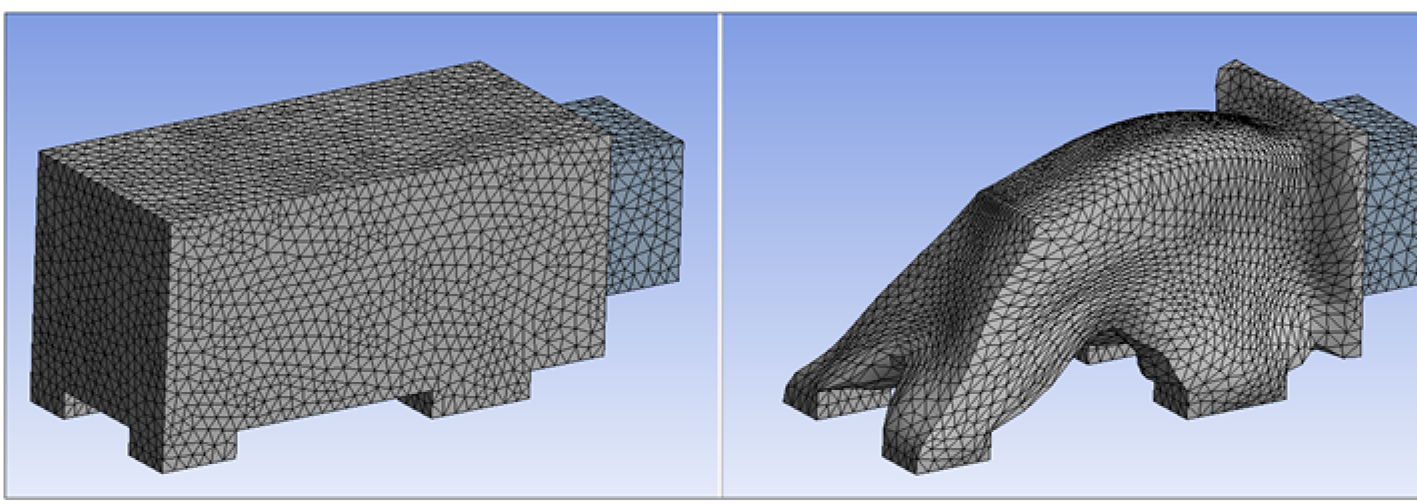
Note:

When you use the **Program Controlled** setting, a new value is computed that is based on the number of layers of elements in the mesh. As a result, the more layers you have, the more permissive the tuning. This means that the **Total Move Limit** will be higher, and the **Mesh Deformation Control** will be smaller.

Specifying Results

This method supports **Topology Density** results. The **Topology Density** object (p. 61) is added automatically to the analysis system. You can add additional objects by selecting **Topology Density** from the **Results** group on the Solution Context tab or by right-clicking the **Solution** folder (or in the **Geometry** window) and selecting **Insert**>**Topology Density**.

Here is an example of an optimized result.



3.2.6. Topography Optimization

The **Topography Optimization** option of the **Optimization Type** property enables shape optimization using mesh node relocations. As with the other optimization methods, this is a physics driven optimization based on a set of loads and boundary conditions provided by either a single preceding analysis or multiple preceding analyses.

Compared to Shape Optimization, Topography Optimization is used for shell-based models only.

Using this method, the application computes an optimal shape in the design domain that you can apply to a selected region of your model and that also includes specific design Objectives and Constraints.

Upstream System Recommendations

Review the following to properly prepare your upstream systems. Topography Optimization has the same capability as the Shape Optimization method. Any differences are mentioned when necessary.

Geometric Analysis

Review the **Geometric Analysis** topic in the **Prerequisites and Requirements** content in the Topology Optimization - Level-Set Based (p. 114) section for the supported **Response Type** and **Response** properties when performing a geometric analysis.

Only **Mass** and **Volume** are available.

Static Structural Analysis

Review the **Configuring Static Structural Analysis** topic in the **Prerequisites and Requirements** content in the Topology Optimization - Level-Set Based (p. 114) section for the supported **Response Type**/**Response** for the **Objective** Worksheet or a **Response Constraint** object.

In addition, when specifying your upstream Static Structural analysis, note that any surface of the optimizable body that is scoped to boundary conditions (fixed displacements, loads, bonded contacts, etc.) must be defined in the Exclusion Region.

Modal Analysis

This method only supports User Defined Criterion that you define in the upstream Modal analysis. This result data can then be used in the Structural Optimization analysis by the **Objective** object or as a **Response Constraint** .

In addition, when specifying your upstream Modal analysis, note that you can control an eigenmode whose frequency always has the same ranking during the optimization process. If its ranking changes, the algorithm will face some difficulty.

Manufacturing Constraint

For this analysis method, no manufacturing constraint are yet available

Application Differences

Note the following when using the Topography Optimization method.

Specifying the Mesh

This analysis method supports triangle and quadrangles.

Important:

When specifying the mesh on your model, it is strongly recommended that you:

- Always use a uniform mesh (homogeneous element size). This enables you to capture the design with the same precision everywhere on the model.
- Make sure that you have a sufficiently fine mesh.

Specifying Optimization Type

You use the **Optimization Region** object to select a region of your geometry on which to perform optimization as well as the optimization method to be used.

1. In order to scope the optimization regions using the Topography Optimization method, you need to first generate the mesh.

2. Specify the **Design Region**. The properties of the **Design Region** category enable you to define the geometry as a Geometry Selection or a Named Selection. This is the region that you wish to optimize.

3. Specify the **Exclusion Region**. The properties of the **Exclusion Region** category enable you to specify a region (geometric entities or elements) to be excluded from optimization. You specify excluded regions using defined Boundary Conditions, Geometry Selection, or a Named Selection.

   The surfaces scoped to boundary conditions (such as Fixed Support, Force, Bonded Contact, etc.) must be included in the scoping of the **Exclusion Region**.

4. Set the **Optimization Type** property to **Topography Optimization**. Specify the following additional properties as needed:

   - **Move Limit Per Iteration**: This property enables you to define how far each node can move at each iteration. It must be defined in length units, such as one element size. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.

   - **Total Move Limit**: This property enables you to define how far each node can move in total. It must be defined in length units, such as three element sizes. By default, this property is set to **Program Controlled** (equal to the size of one element or slightly larger depending the fineness of the mesh). Select the **Manual** option to change the value.

   - **Mesh Deformation Control**: This property enables you to define how much the mesh can be stretched. It is an additional control to avoid element distortion. This unit-less value is a sort of penalty factor that ranges from 0 (no control) to 1.0. By default, this property is set to **Program Controlled**. Select the **Manual** option to change the value.

Specifying Results

This method supports Topology Density (p. 61) results. The **Topology Density** object is added automatically to the analysis system. You can add additional objects by selecting **Topology Density** from the Results group on the Solution Context tab or by right-clicking the **Solution** folder (or in the **Geometry** window) and selecting **Insert** > **Topology Density**.

3.3. Mixing Multiple Methods within an Analysis

Using multiple **Optimization Region** objects, you can combine the following optimization methods within an optimization problem.

	Density Based Optimization	Lattice Optimization	Level Set Based Optimization	Mixable Density	Shape Optimization	Topography Optimization
Density Based Optimization	X					
Lattice Optimization		X				
Level Set Based Optimization			X	X	X	X
Mixable Density			X	X	X	X
Shape Optimization			X	X	X	X
Topography Optimization			X	X	X	X

3.4. Topology Optimization versus Shape Optimization

Topology Optimization methods (Density and Level Set) relate to the Immersed Boundary Method (I.B.M.) in that the shape is approximated using an auxiliary field (density-field or level-set function). Using this method, the application fixes the mesh during the optimization process and optimizes the shape by distributing the material layout within a given design domain. Topology Optimization is very convenient when you expect significant changes (hole nucleation, merge of members, etc).

Shape Optimization methods relate to a body-fitted method in that the shape is explicitly defined by the mesh. Using this method, the application optimizes the design by moving the mesh nodes. Shape Optimization is a sort of morphing without the need to define any parameter. By contrast to Topology Optimization, the calculation of certain quantities, such as stress, is more accurate. Shape Optimization is convenient when you expect moderate modifications while keeping the same topology (no hole, no merge).

Chapter 4: Structural Optimization Theory

In order to provide background about how manufacturing and design constraints are implemented and the options you can use to further improve your results, this section describes the concepts behind the use of the constraints in the application when performing a Structural Optimization analysis.

Go to a section topic:

4.1. Manufacturing Constraint Background

4.1. Manufacturing Constraint Background

Structural optimization often results in complex shapes that are difficult to realize under specific manufacturing processes. Incorporating manufacturing constraints in the setup of the optimization problem can help to reduce and even eliminate manual post-processing, speed-up the design process, and ensure optimality of the final shape.

However, manufacturing constraints also include limitations related to the numerical framework (Optimization Method, Optimization Region, Mesh, etc.). The following documentation discusses the basic details of their formulation and numerical implementation together with limitations and best-practices for efficient usage.

Note:

The description "average element size," denoted dx_{AVG}, appears frequently in the numerical implementation of manufacturing constraints. For 3D solid domains, the average element size is defined as the average of the cubic root of the volume over all elements.

Go to a section topic:

4.1.1. Member Size Minimum

4.1.2. Member Size Maximum

4.1.3. Member Size Gap Size

4.1.4. Pull Out Direction

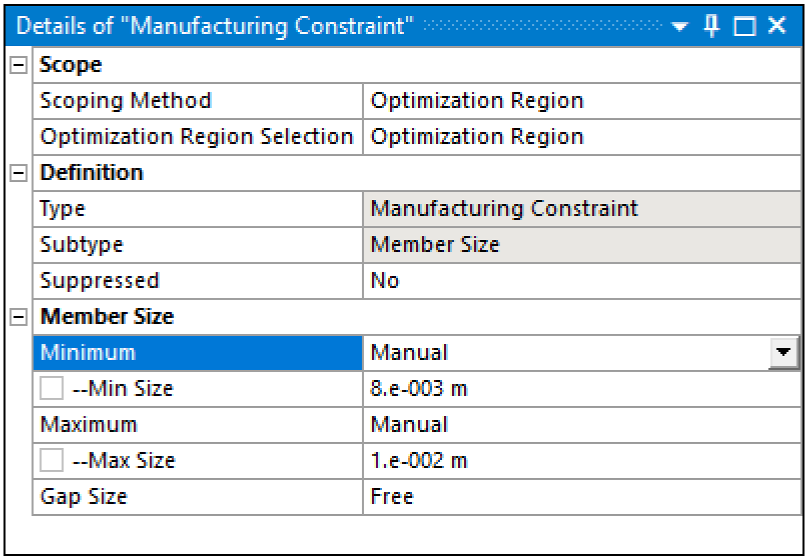
4.1.5. AM Overhang Constraint

4.1.6. Housing

4.1.7. Complexity Index

4.1.1. Member Size Minimum

The **Minimum** property prevents the formation of thin features in the optimized shape.



Details of "Manufacturing Constraint"

Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Member Size
Suppressed	No
Member Size	
Minimum	Manual
--Min Size	8.e-003 m
Maximum	Manual
--Max Size	1.e-002 m
Gap Size	Free

Industry Motivation

This specification is motivated by several industrial manufacturing processes. Examples include:

- During a casting process, you can actively avoid thin areas in the mold that can lead to failure as a result of premature solidification.

- During an additive manufacturing process, you can avoid thin members that can result in imperfections in the printing process.

Member Size Minimum Definition

In theory, the computation of the minimum member size is performed in two steps:

1. The application computes the cross-sectional thickness for every point on the boundary, that is, the distance until crossing the opposite boundary along the direction opposite to the exterior normal vector.

2. The minimum value of the distance field is defined as the minimum member size of the structure.

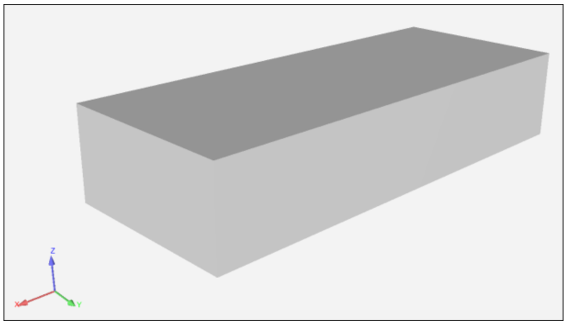
For example, in the wall-type example, the cross-sectional length equals:

- Length (X-direction)

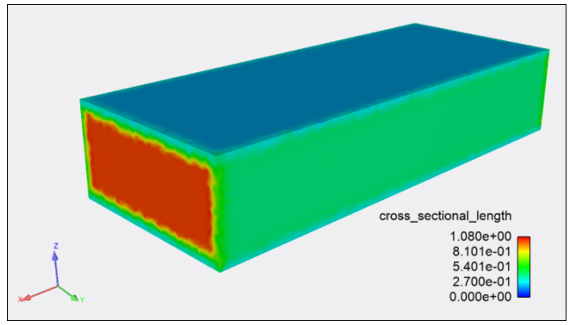
- Width (Y-direction)

- Height (Z-direction)

Plate Structure



Cross-sectional Length



This formulation has two main drawbacks:

- It is not derivable due to the minimum operator, and therefore not suitable for gradient-based optimization algorithms.
- It is not applicable for all methods, namely it requires geometric information that is not available for density-based methods.

The **Minimum** Member Size constraint is available for density-based optimization, mixable density, and level-set based optimization. The formulation depends on the structural optimization framework.

Density and Mixable Density	Level-Set
Built-in approach based on filtering techniques [2 (p. 195)].	The theoretical definition is adopted, based on the cross-sectional thickness, and is appropriately adjusted to overcome the differentiability issue [13 (p. 195)].

Setup and Technical Specifications

The setup only requires specifying the lower limit not to violate, denoted $d_{\min}$, in terms of length units.

	Density Based	Mixable Density	Level Set Based
$d_{\min}$	$d_{\min} \geq dx_{avg}$ **Note:** $d_{\min}$ is automatically rounded-up to d_{avg}. The default value is equal to d_{avg}.	$d_{\min} \geq dx_{avg}$ **Note:** $d_{\min}$ is automatically rounded-up to d_{avg}. The default value is equal to 4 d_{avg}.	$d_{\min} \geq 0.0$

	Density Based	Mixable Density	Level Set Based
Exclusion Region	You specify whether to Include or Exclude Exclusions from the Analysis Settings.	Exclusion regions are inherently considered in the member size evaluation.	
Infeasible Regions[a]	Naturally ignored		Detected and eliminated from the constraint evaluation.
Special treatment	None		The optimization runs first without the MinMS constraint. If the solution does not meet the $d_{\min}$ requirement, then the optimization continues adding the MinMS as a constraint.
Mesh Size	The finer the mesh, the more accurate the computations related to Minimum member size.		

[a] Regions where the Minimum Member Size cannot be respected - closely spaced boundaries.

Recommendations

Note the following:

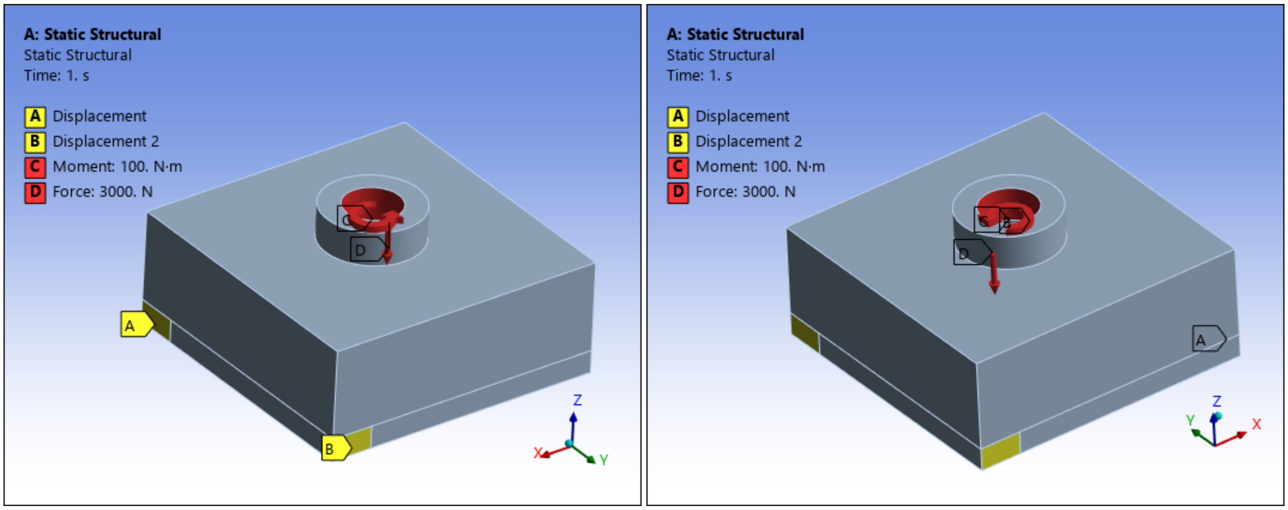
- The computational time for the Minimum Member Size constraint increases together with $d_{\min}$ and the mesh size refinement.
- As any constraint, the Minimum Member Size affects the objective performance. Namely, by increasing $d_{\min}$, the feasible domain shrinks and probably leads to smaller objective gain. Especially when an objective function with antagonistic behavior is used, that is, the volume, this tendency becomes more significant.

Example

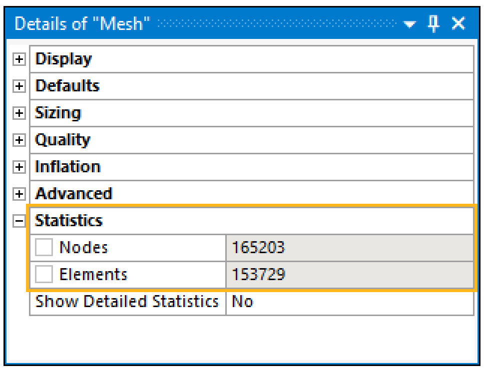
For the following example, the **Optimization Type** property is set to the **Topology Optimization – Level Set Based** option. The optimization problem consists of:

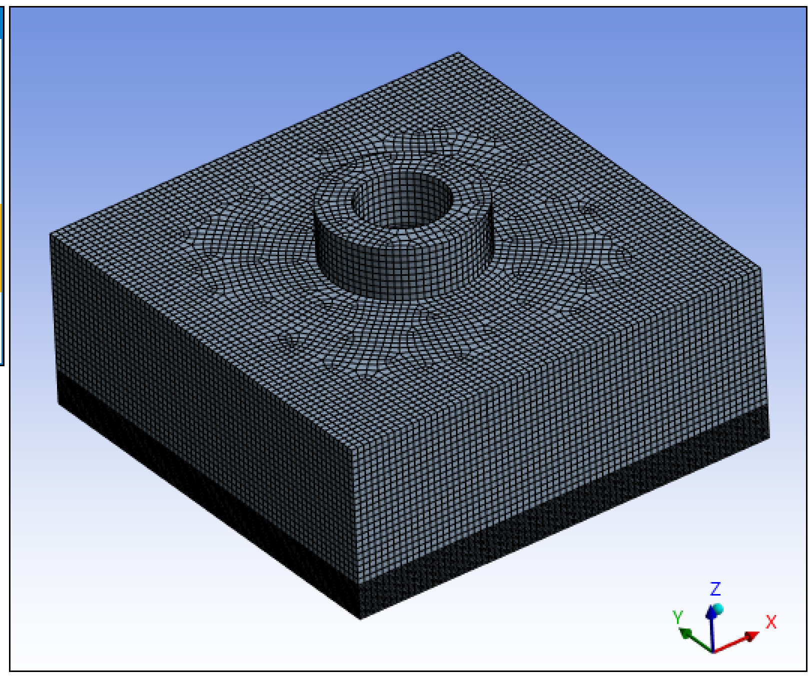
- Minimizing the structural compliance under a volume fraction constraint of 0.4.
- A Maximum Member Size constraint with d_{max} = 1.0e-02.
- A Pull-Out Direction constraint in the Z-direction.
- A Cyclic Repetition Design Constraint with 4 sectors enforced.

The upstream **Static Structural** system is specified with two displacements, a force, and a moment, as illustrated.

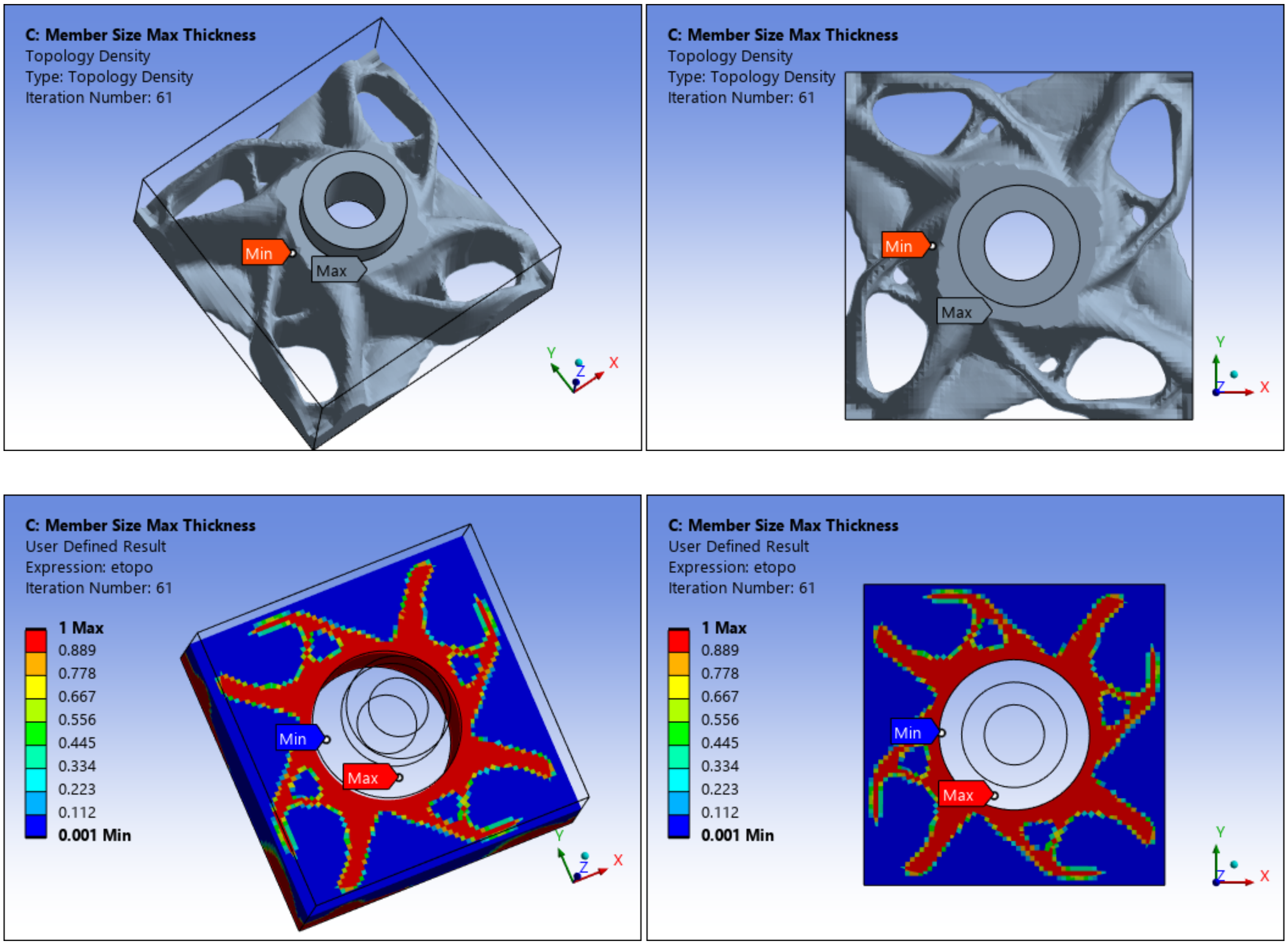


The finite element mesh includes 153,729 elements.

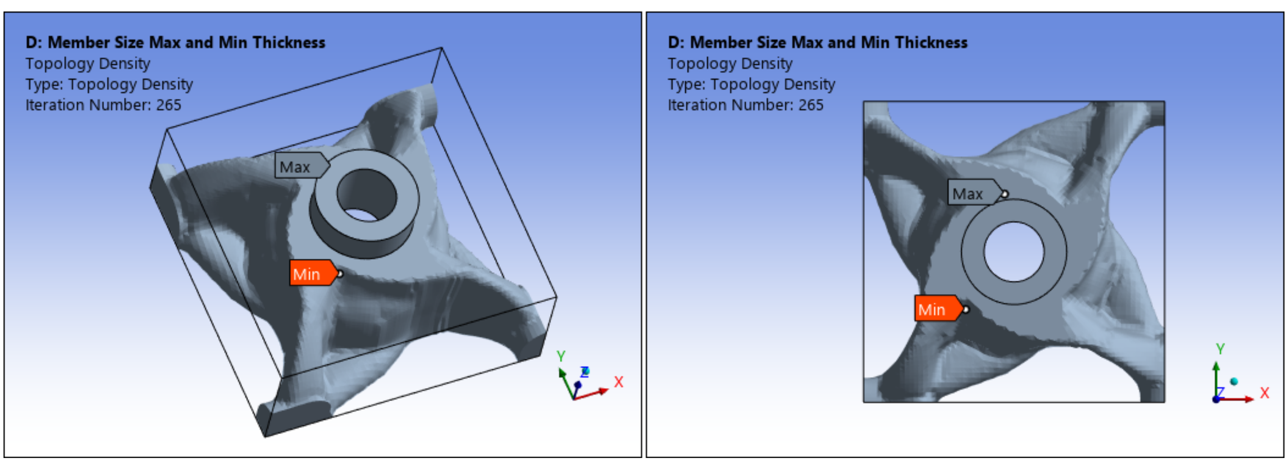


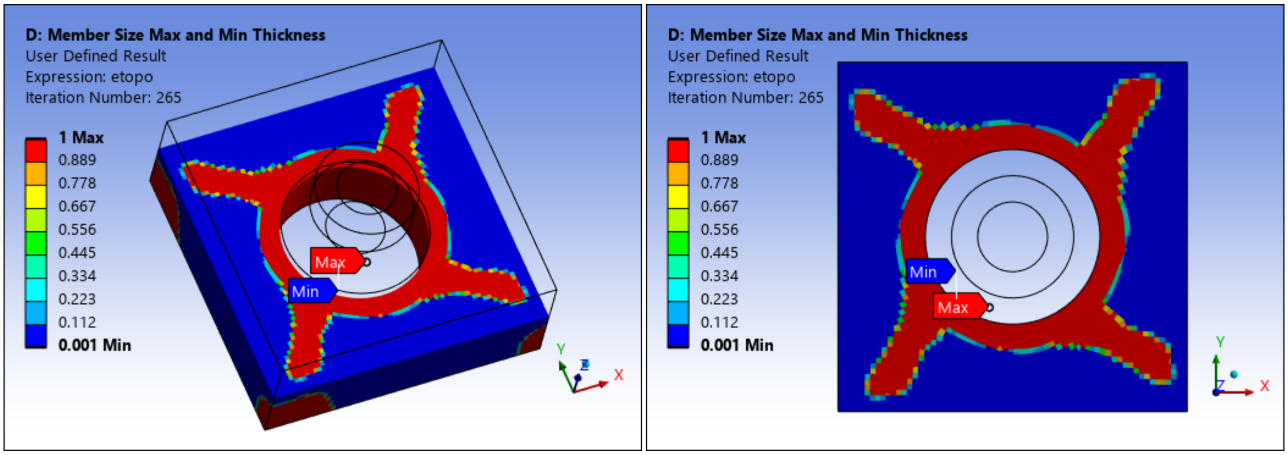


As shown from the **Topology Density** as well as a User Defined Result, the optimized result without considering an minimum member size constraint, shown here, produces slender "ribs.". Note that the **User Defined Result** shown below is a cutout of the result using the Section Plane feature.



When you add a minimum member size constraint with d_{min} = 8.0e-03, it leads to the optimized shape where slender ribs disappear.





References (p. 195)

[3] M.P. Bendsoe, O. Sigmund. Topology optimization: theory, methods, and applications, Springer Science & Business Media, 2003.

[13] G. Allaire, F. Jouve, G. Michailidis, Thickness control in structural optimization via a level set method, Structural and Multidisciplinary Optimization, 2016.

4.1.2. Member Size Maximum

The **Maximum** member size property prevents the formation of thick features in the optimized shape. It is supported for density-based optimization, level-set based optimization, mixable density, and Shape Optimization.

Details of "Manufacturing Constraint"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Member Size
Suppressed	No
Member Size	
Minimum	Manual
--Min Size	8.e-003 m
Maximum	Manual
--Max Size	1.e-002 m
Gap Size	Free

Industry Motivation

This specification is motivated by several industrial manufacturing processes. Examples include:

- During casting processes, a maximum member size is directly related to the cooling process of the parts where the distance to shape boundaries determines solidification times. Larger members require a more elaborate solidification system to remove the shrinkage porosity outside of the cast part and therefore increase manufacturing costs.

- During additive manufacturing processes, you can avoid large members that result from the distortion created during the solidification of each layer by thermal residual stress.

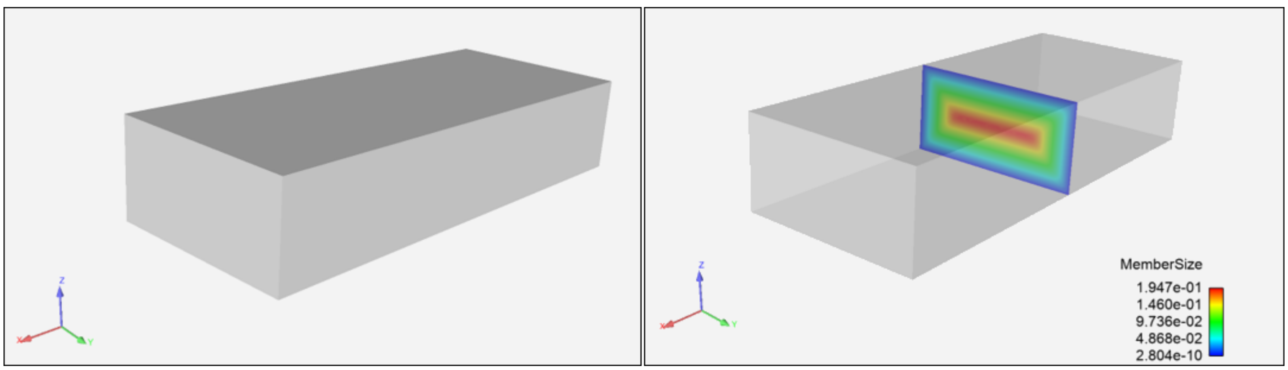
Member Size Maximum Definition

Theoretically, the computation of the maximum member size is performed in two steps:

1. The application computes the pointwise distance by selecting a point inside of the shape that it then extends to the boundary. [13 (p. 195)], [14 (p. 195)].

2. The maximum value of the distance field is defined as the maximum thickness of the structure.

It is important to note that the formulation of member size is **not directional**. The constraint is satisfied when you preserve the distance in **any direction**. For example, for the following plate, the maximum thickness corresponds to its height.

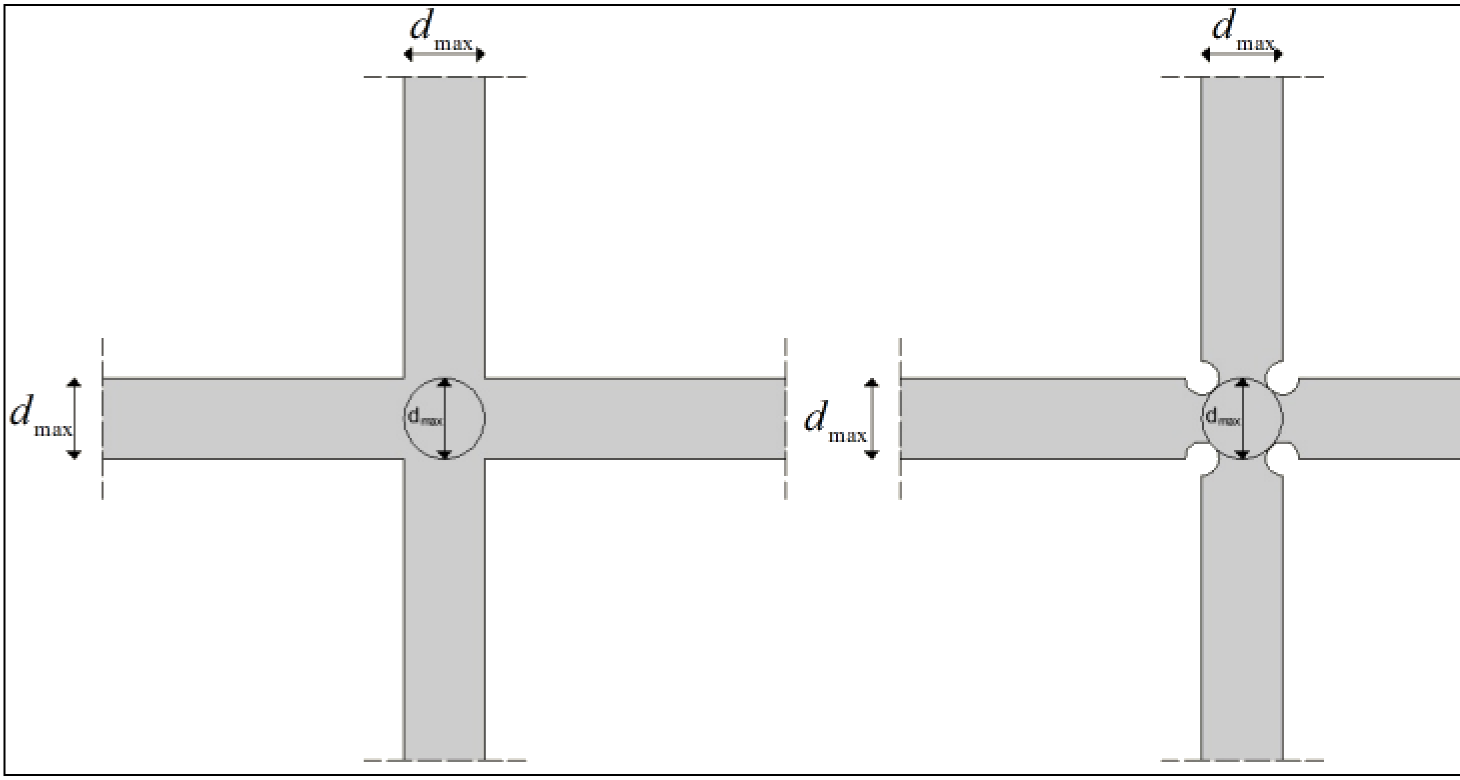


Furthermore, the above formulation presents two critical drawbacks:

- It is not derivable due to the Maximum operator and therefore is not suitable for gradient-based optimization algorithms.

- As illustrated below, it is not permissive because it tracks violation regions that should be ignored [13 (p. 195)].

However, the devised formulation is appropriately adjusted to overcome these two problems. As a result, the limit value might be violated in some limited regions of the optimized shape.



Distortions close to joints of structural members to respect the maximum member size limit (d_{max}) in a pointwise
(left): d_{max} is not respected close to joints; (right): d_{max} is respected everywhere.

Setup and Technical Specifications

The setup only requires specifying the upper limit not to exceed, denoted d_{max}, in terms of length units. However, for the sake of accuracy in the calculation of the member size, d_{max} must respect a certain ratio with respect to dx_{AVG}, as shown in the following table.

	Density Based	**Mixable Density**	**Level Set Based**	**Shape Optimization**
$d_{\max}$	$d_{\max} \geq 4.4\, dx_{AVG}$ Lower values lead the optimization process to failure.	$d_{\max} \geq 4\, dx_{AVG}$ $d_{\max}$ is automatically rounded-up to $d_{\max} \geq 4\, dx_{AVG}$ if the limit ratio is not respected.		
Exclusion Region	Using the options, Include Exclusions or Exclude Exclusions, of the Region of Manufacturing Constraint and/or Region of Min Member Size properties of the Analysis Settings object, you can specify whether	Exclusion regions are inherently considered in the member size evaluation. In case the thickness of the exclusion region exceeds $d_{\max}$, the constraint cannot be respected during the optimization and therefore, $d_{\max}$ is automatically re-adjusted to comply with the exclusion region.		

	Density Based	Mixable Density	Level Set Based	Shape Optimization
	to include or exclude exclusions.			
Retained Threshold	The member size is computed with respect to the 0.5 iso-contour of the density field. Modifying the **Retained Threshold** property changes the member size of the shape.		NA	NA

Recommendations

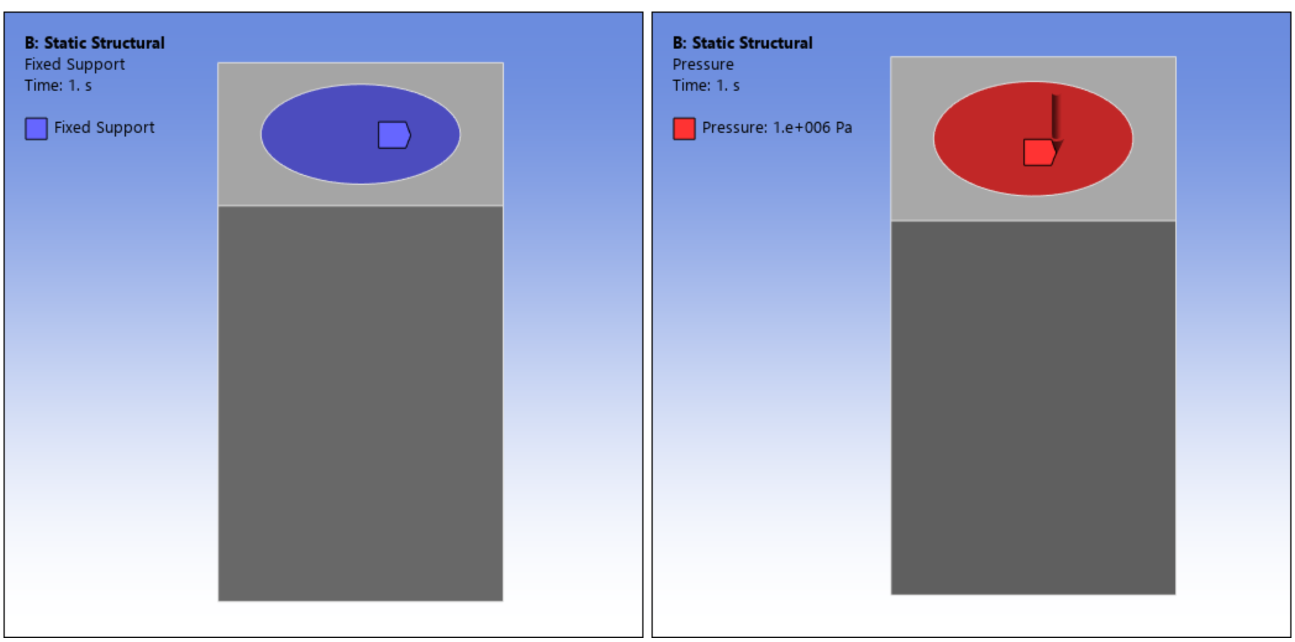
Note the following:

- The **Maximum** Member Size constraint tends to deliver more complex designs.
- Ansys recommends that you use a fine mesh if you expect additional details in the geometry.
- As with any constraint, the **Maximum** Member Size affects the objective performance. Specifically, if you reduce $d_{\max}$, the feasible domain shrinks and likely leads to a smaller objective gain.

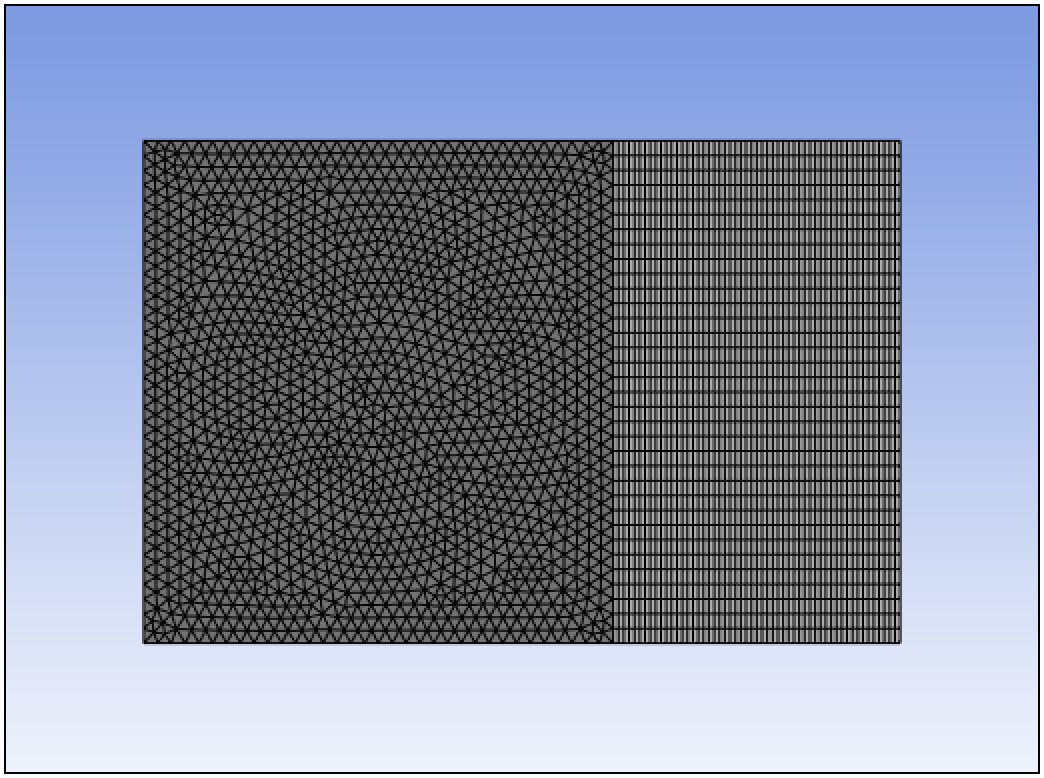
Examples

Pillar Structure

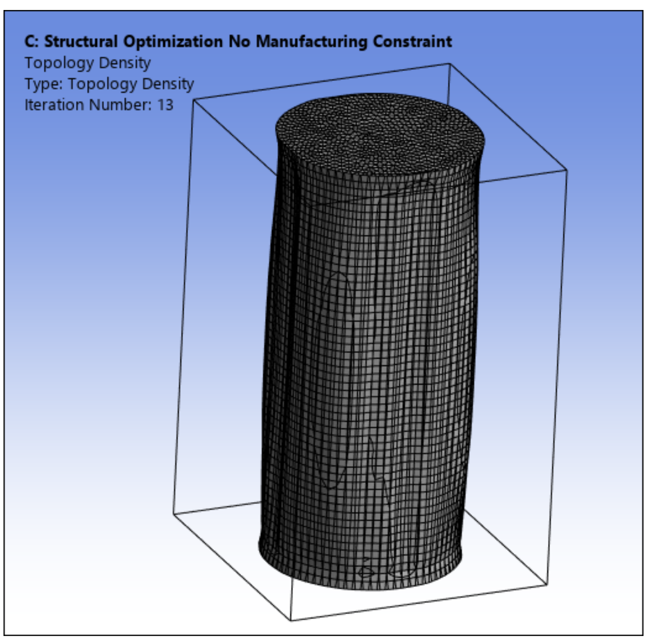
For the following example, the **Optimization Type** property is set to the **Topology Optimization – Level Set Based** option. This analysis uses a pillar structure clamped on the bottom with a vertical load applied to the top, as illustrated below. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.4.

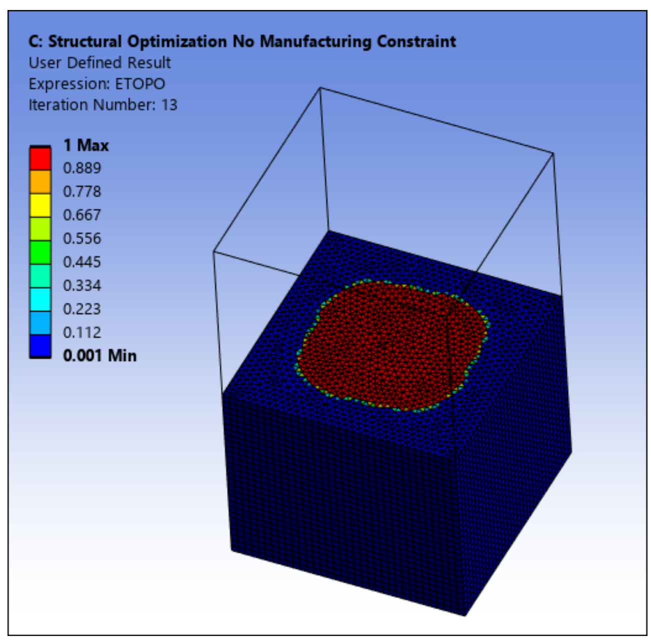


Here is the finite element mesh (144,828 elements).



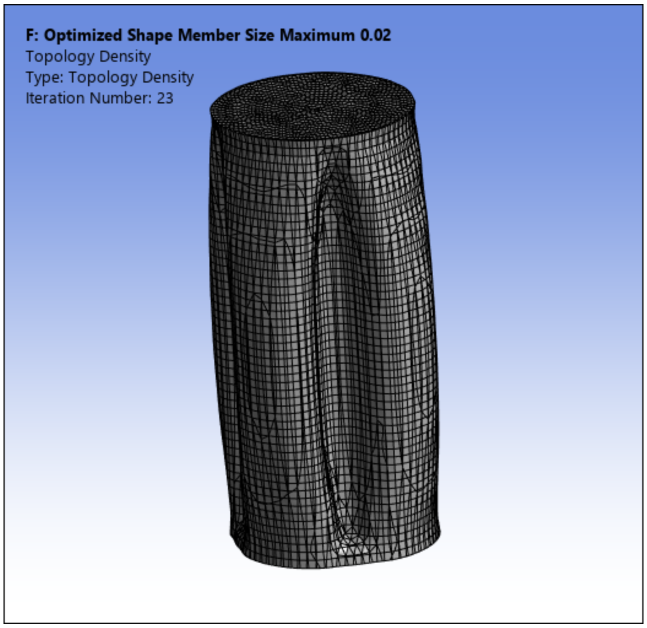
The following **Topology Density** result and **User Defined Result**, show the optimized result without considering any maximum member size constraints. As expected, the optimized shape is a cylinder. Note that the **User Defined Result** shown below is a cutout of the result using the Section Plane feature.

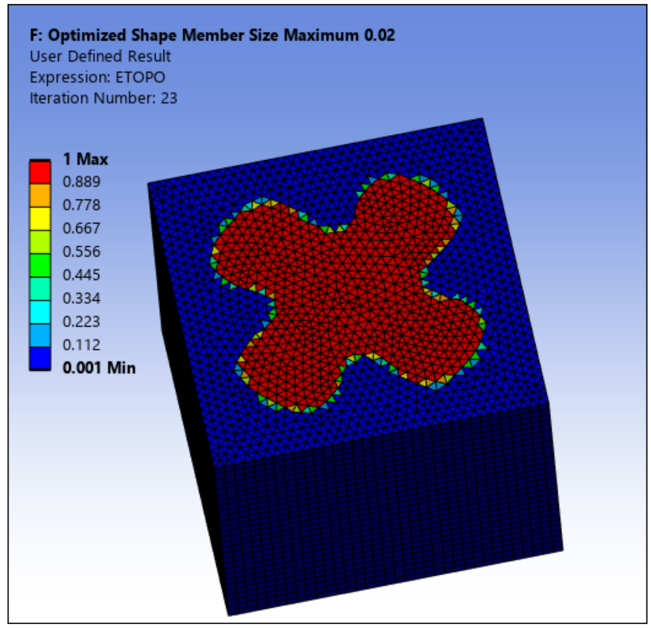




Adding an maximum member size constraint of 0.02 leads to the optimized shape shown below. One can observe the indentations that are created to satisfy the maximum member size constraint.

The expected impact of the maximum member size constraint is clearly seen, reducing the maximum distance from the points inside the shape to the structural boundary.





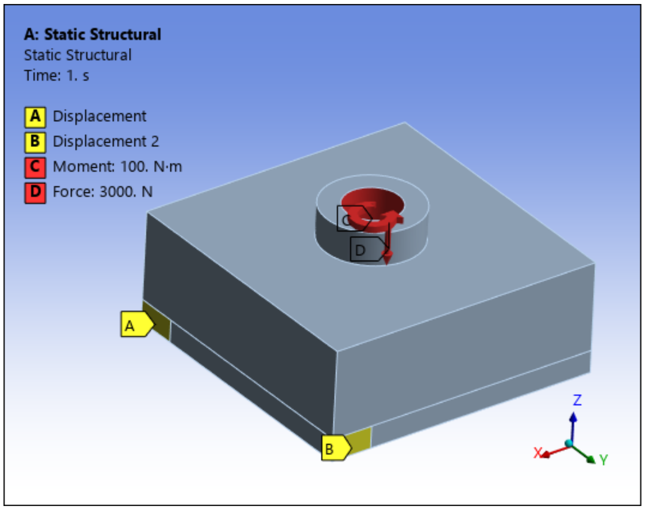
Engine Bracket

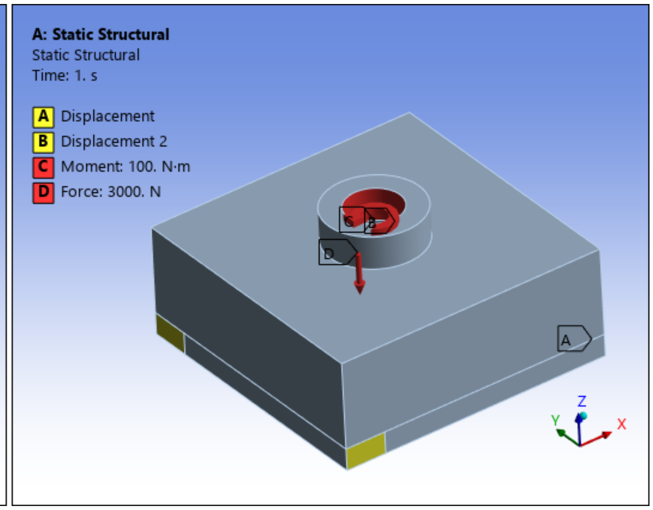
For the following example, the **Optimization Type** property is set to the **Topology Optimization – Level Set Based** option. The optimization problem consists of:

- Minimizing the structural compliance under a volume fraction constraint of 0.4.
- A Maximum Member Size constraint with d_{max} = 1.0e-02.
- A Pull-Out Direction constraint in the Z-direction.
- A Cyclic Repetition Design Constraint with 4 sectors enforced.

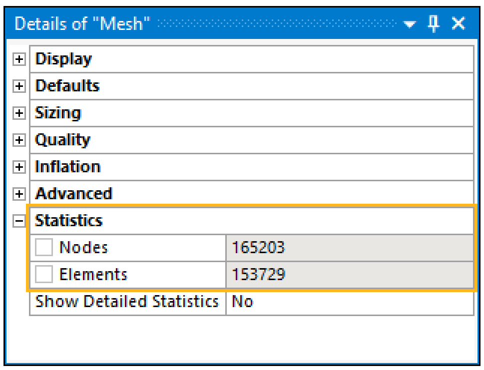
Mechanical Setup

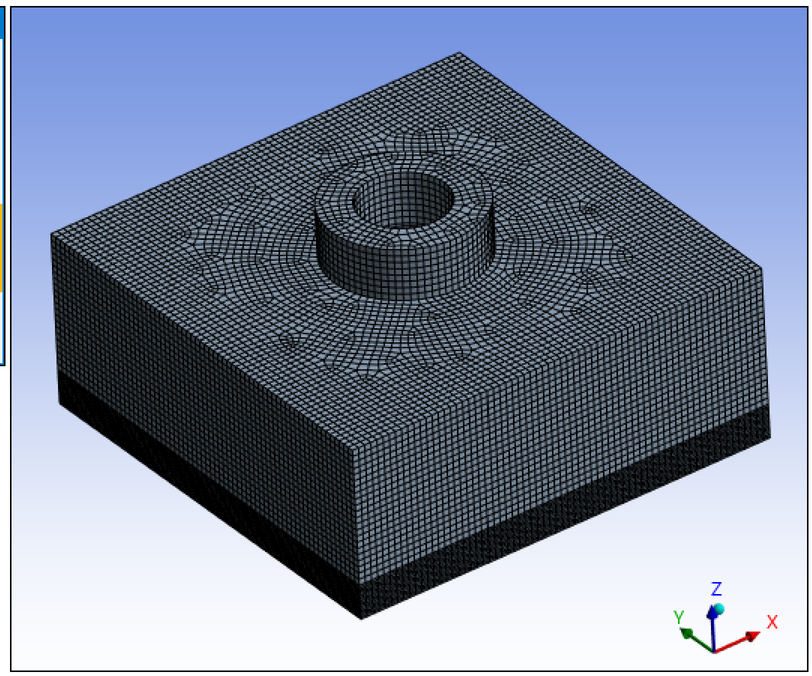
The upstream Static Structural system is specified with two displacements, a force, and a moment, as illustrated.



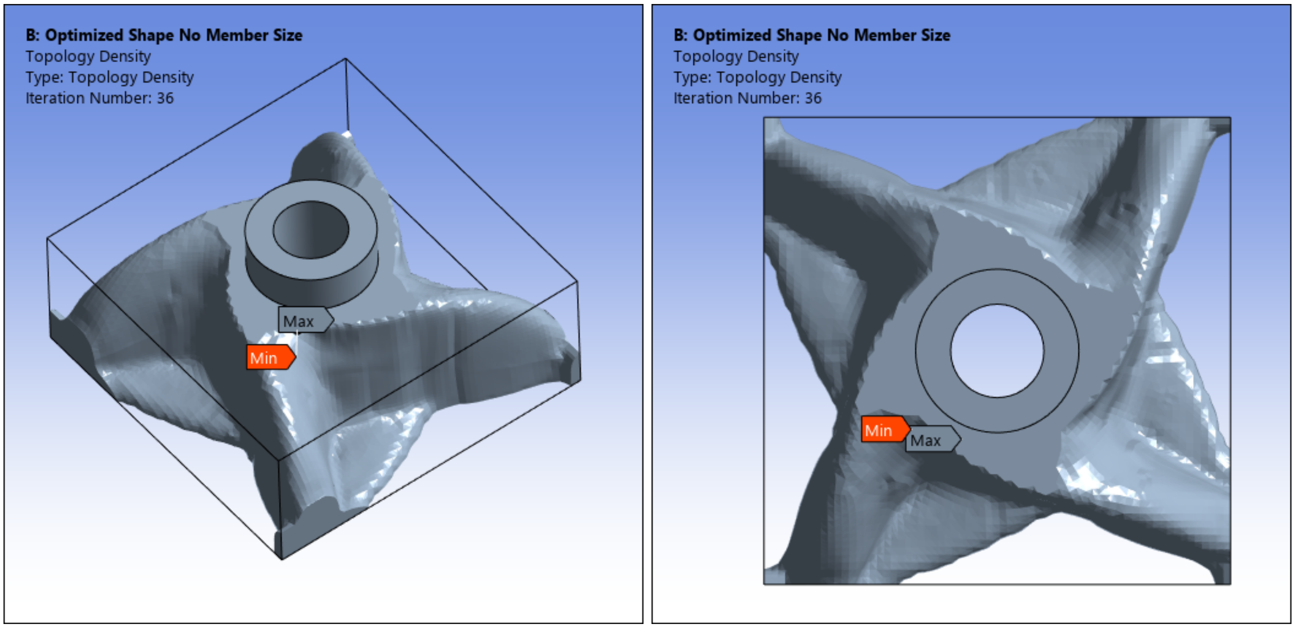


Here is the finite element mesh (153,729 elements).

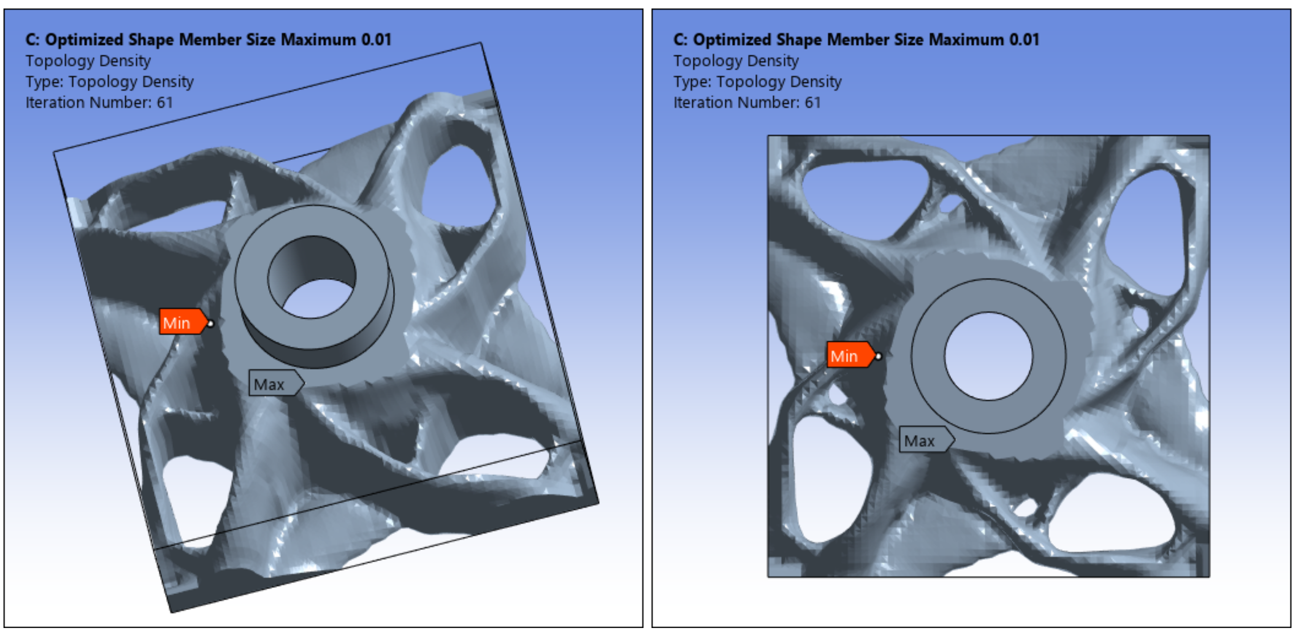




The following **Topology Density** result shows the optimized result without considering an maximum member size constraint.



And here you can see the optimized shape and cross-sectional density for d_{max} of 0.01.



References (p. 195)

[13] G. Allaire, F. Jouve, G. Michailidis, Thickness control in structural optimization via a level set method, Structural and Multidisciplinary Optimization, 2016.

[14] Fernandez, E., Yang, K.K., Koppen, S., Alarcon, P., Bauduin, S., Duysinx, P., Imposing minimum and maximum member size, minimum cavity size, and minimum separation distance between solid members in topology optimization. Computer Methods in Applied Mechanics and Engineering, 2020.

4.1.3. Member Size Gap Size

The **Gap Size** constraint prevents the formation of closely spaced features in the optimized shape. The appearance of such features is more pronounced when the maximum member size constraint is used, since the algorithm may satisfy the maximum member size constraint by placing small holes to reduce the size of large features. It is only available for the level-set based optimization method and can only be activated in conjunction with the **Maximum** member size property.

Details of "Manufacturing Constraint"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Manufacturing Constraint
Subtype	Member Size
Suppressed	No
Member Size	
Minimum	Free
Maximum	Manual
--Max Size	1.2e-002 m
Gap Size	Manual
--Value	1.2e-002 m

Industry Motivation

This specification is motivated by several industrial manufacturing processes:

1. During casting process, it is linked to the capability of constructing molds with thin features as well as with their life cycle.
2. During milling process, as well as other tooling processes, it is related to the characteristics of the tooling machine (head and bit radius, etc.).

Gap Size Definition

Theoretically, the computation of the gap size is performed in two steps:

1. The pointwise distance between structural members is computed for every point on the boundary along the exterior normal vector.
2. The minimum value of the distance field is defined as the gap size of the structure.

However, this formulation presents two main drawbacks:

- It is not derivable, due to the minimum operator and the computational procedure, therefore, it is not suitable for gradient-based optimization algorithms.
- It is quite restrictive for topology optimization since it hinders structural members from approaching each other and finally merging.

A specific formulation has been devised to overcome these problems. As a consequence of adopting a rather qualitative formulation [14 (p. 195)], closely spaced features may still appear in the optimized shape.

Setup and Technical Specifications

The setup only requires specifying the lower limit not to violate, denoted $d_{GapSize}$, in terms of length units. However, for the sake of accuracy in the calculation of the member size, $d_{GapSize}$, must respect a certain ratio with respect to $d_{\max}$, as shown below.

$d_{GapSize}$

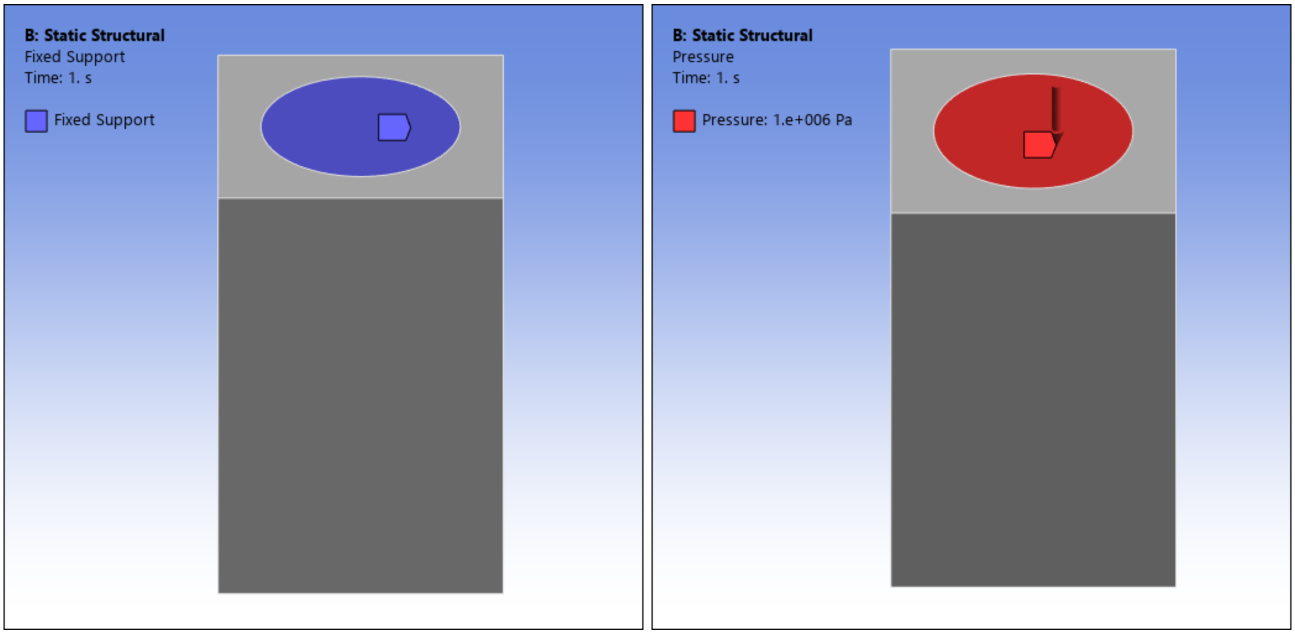
	Level Set Based
$d_{GapSize}$	$2d_{\max} \geq d_{GapSize} \geq 0.40d_{\max}$ Both relations are motivated for accuracy reasons. However: • The lower limit relation is required, that is, $d_{GapSize}$ is automatically rounded-up to 0.40dmax if the limit ratio is not respected. • The upper limit relation is rather a recommendation.

Recommendations

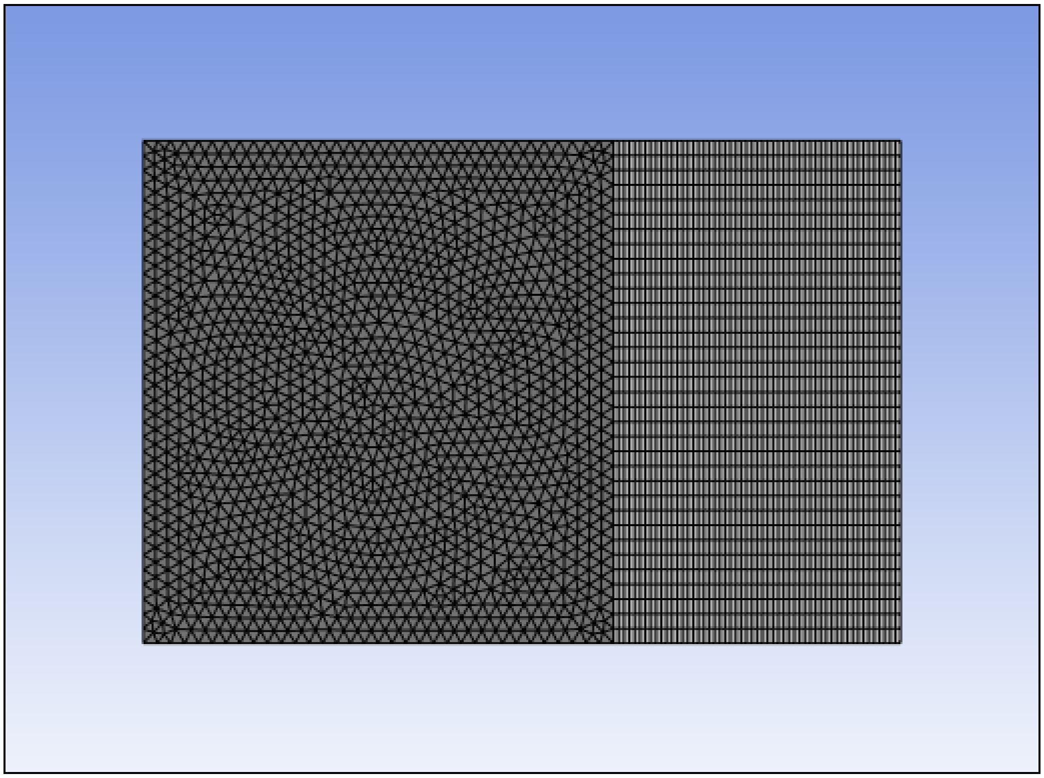
As any constraint, the **Gap Size** affects the objective performance. Namely, by increasing $d_{GapSize}$, the feasible domain shrinks and probably leads to smaller objective gain.

Example

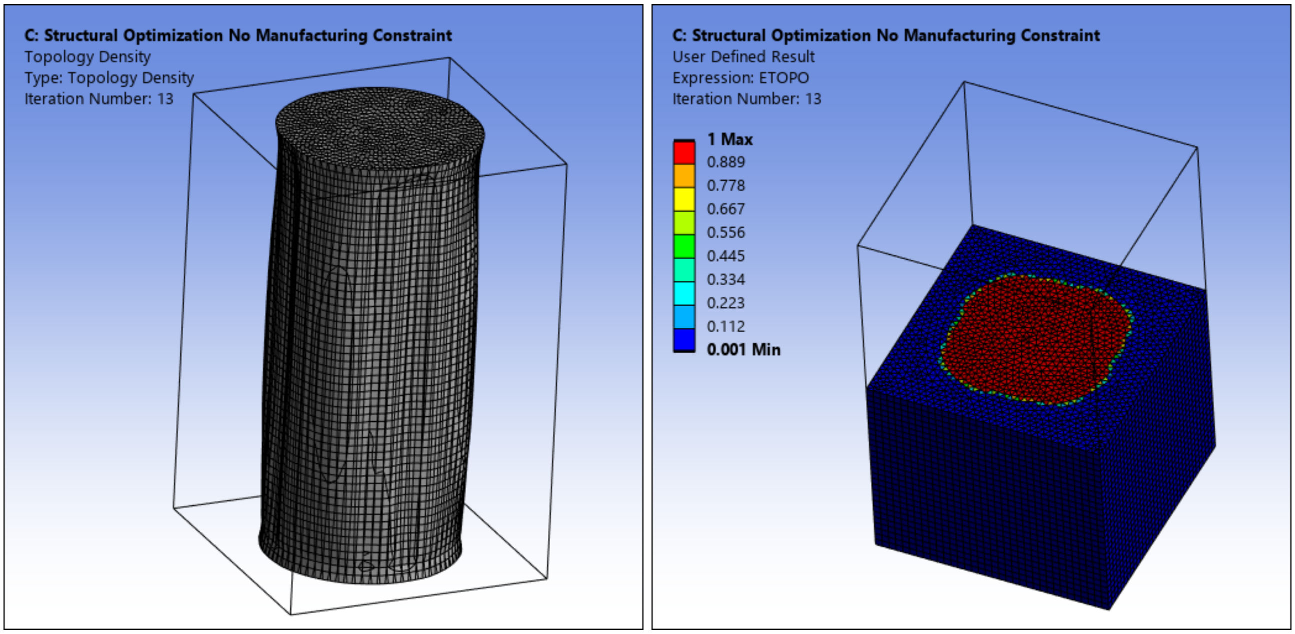
For the following example, the **Optimization Type** property is set to the **Topology Optimization – Level Set Based** option. This analysis uses a pillar structure clamped on the bottom with a vertical load applied to the top, as illustrated below. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.4.



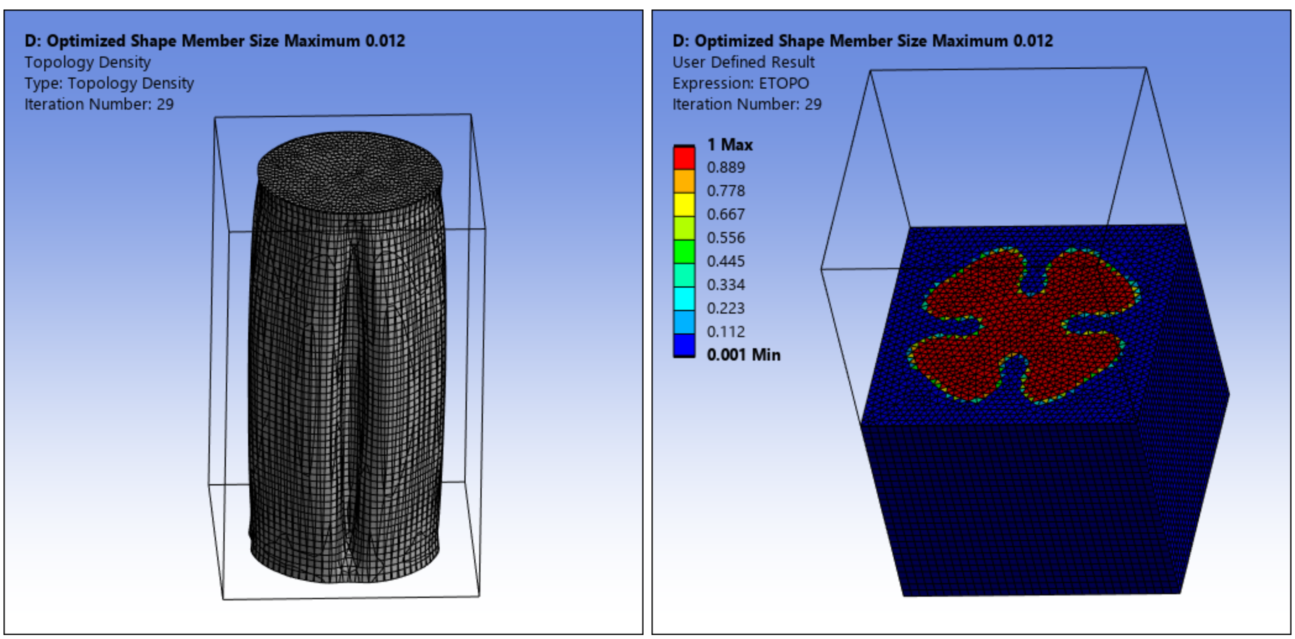
Here is the finite element mesh (144,828 elements).



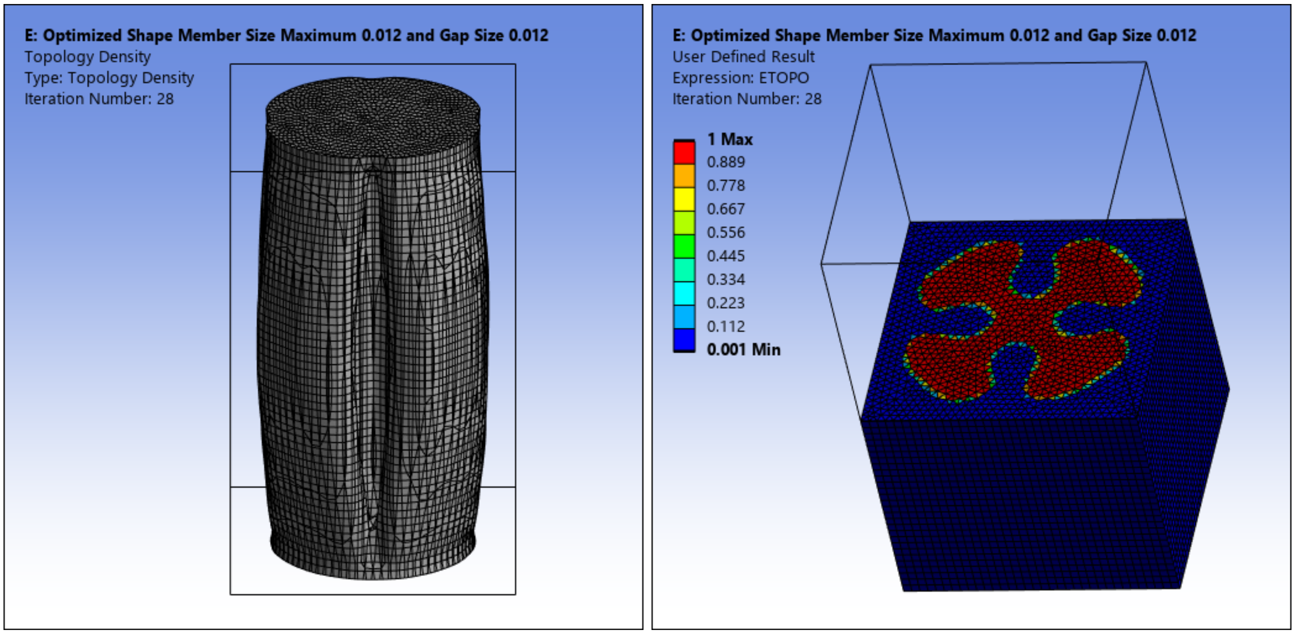
As shown from the **Topology Density** result as well as a User Defined Result, the optimized result without considering manufacturing constraints. As expected, the optimized shape is a cylinder. Note that the **User Defined Result** shown below is a cutout of the result using the Section Plane feature.



Adding an maximum member size constraint of 0.012 leads to the optimized shape shown below. One can observe the indentations that are created to satisfy the maximum member size constraint. The structural compliance is equal to 1.85953 e-04 (+1.5%).

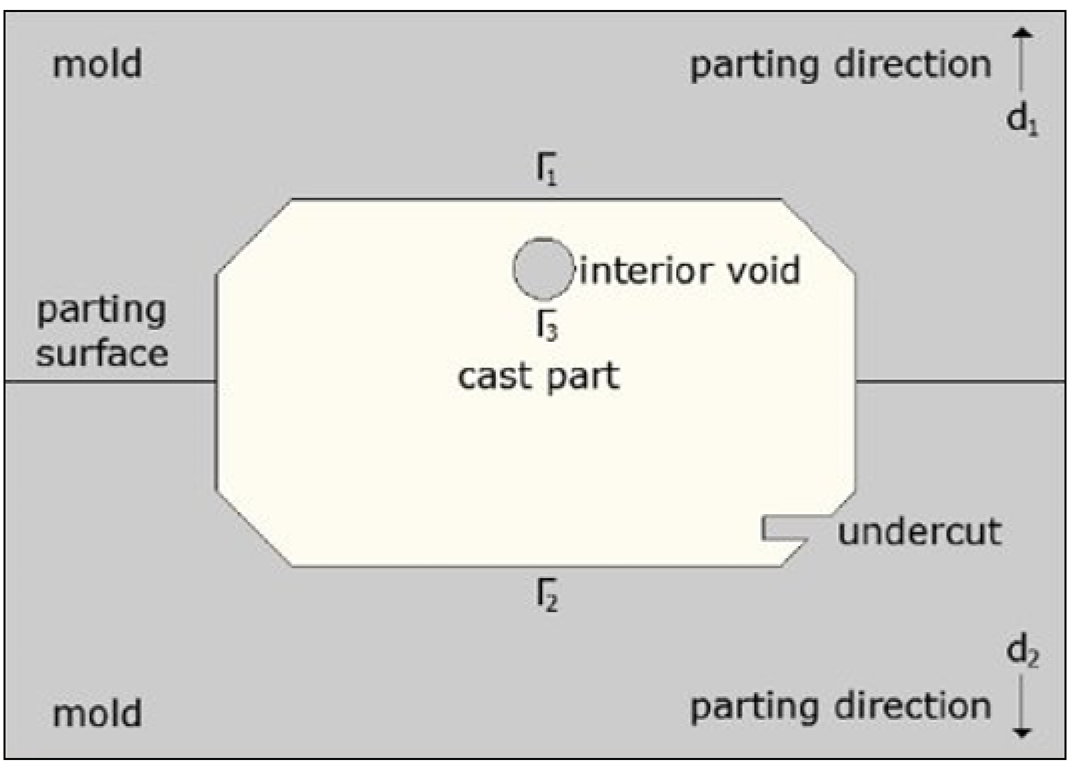


Adding a **Gap Size** constraint with limit $d_{GapSize}$ = 0.012 results in the optimized shape shown below, wherein the distance between structural members increases. The structural compliance equals to 1.92452 e-04, namely 5% less stiff due to the maximum member size and gap size constraints.



4.1.4. Pull Out Direction

The Manufacturing Constraint (p. 48) **Pull Out Direction** enables you to deliver a design that is compatible with the casting process by preventing the formation of undercuts and internal holes. This capability only requires that you specify the direction in which to remove the mold, as illustrated below, for a cast part enclosed by two molds that are removed in opposite directions [13 (p. 195)].



This constraint also provides the following **Pull Out Options**:

- **No-Hole**: This option aims to simplify the casting process. It ensures that the shape does not perforate along the specified direction and enables it to shrink perpendicular to the pull-out direction.
- **Stamping**: You use this option to create a design that is compatible with a stamping or forging process. This option makes sure the shape does not perforate along the specified pull-out direction.

Go to a section topic:

- Supported Methods (p. 158)
- Setup and Technical Details (p. 158)
- Recommendations and Observations (p. 160)
- Examples (p. 161)
- References (p. 183)

Supported Methods

The **Pull Out Direction** constraint is available for the **Mixable Density**, **Level Set**, and **Density Based** optimization methods. These methods employ a built-in approach, namely that a dedicated machinery has been devised. Only the **Level Set** method provides the **Pull Out Options** properties.

Setup and Technical Details

The basic setup requires you to specify the **Axis** and the **Direction** properties along which the **Pull Out Direction** constraint is enforced. For the Level Set Based method, additional options are available, as described below.

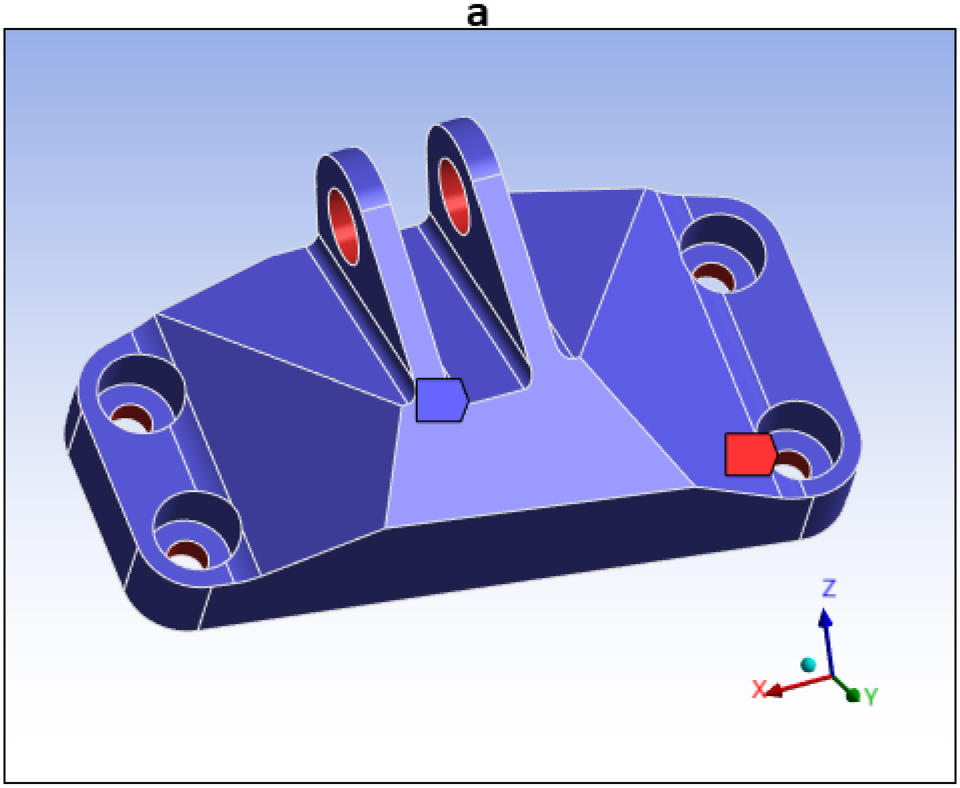
	Method	
Property	**Mixable Density/Density Based Description**	
Axis	The **Axis** property enables you to specify the axis of the Coordinate Sy	
Direction	The **Direction** property enables you to specify whether the shape bou (two-sided) directions. **Note**: For the **Density Based** method, when you specify the constraint automatically imposes a parting surface. This surface passes from the c the axis you specify in the **Axis** property.	
Stamping	NA	Imposes r pull-out c
No-Hole	NA	Imposes r direction. to the pu **Note**: The

	Method	
Property	**Mixable Density/Density Based Description**	
Exclusion Region	You can decide whether to Include or Exclude Exclusions from the **Analysis Settings**.	Exclusion automatic images b

Here is an example of the extension of the **Exclusion Region** in a **Level Set Based** Optimization using the **Pull Out Direction** constraint.

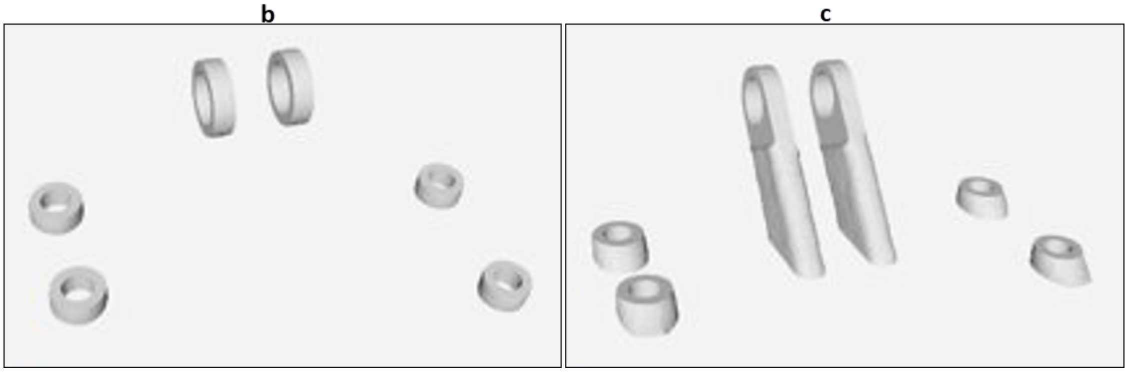
Exclusion Surfaces (a)

a



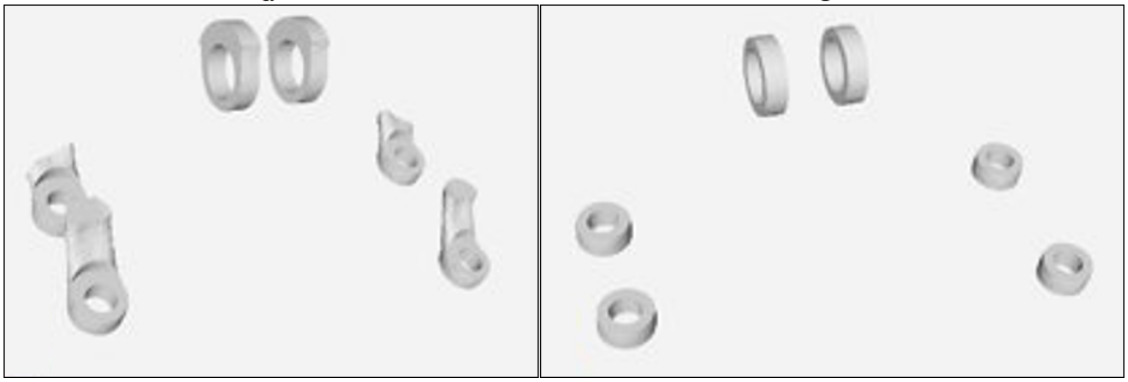
Exclusion Region (b) without **Pull Out constraint**. Extended **Exclusion Region** for (c) one-sided pull out in the +Z direction.

b c



One-sided pull out (d) in the -Z direction and (e) two-sided pull out along the Z-axis.

d e



Recommendations and Observations

As with any constraint, the **Pull Out Direction** shrinks the feasible domain and leads to smaller objective gain. With the **No-Hole** constraint, tiny perforations may still appear in the optimized shape (see the illustrations section).

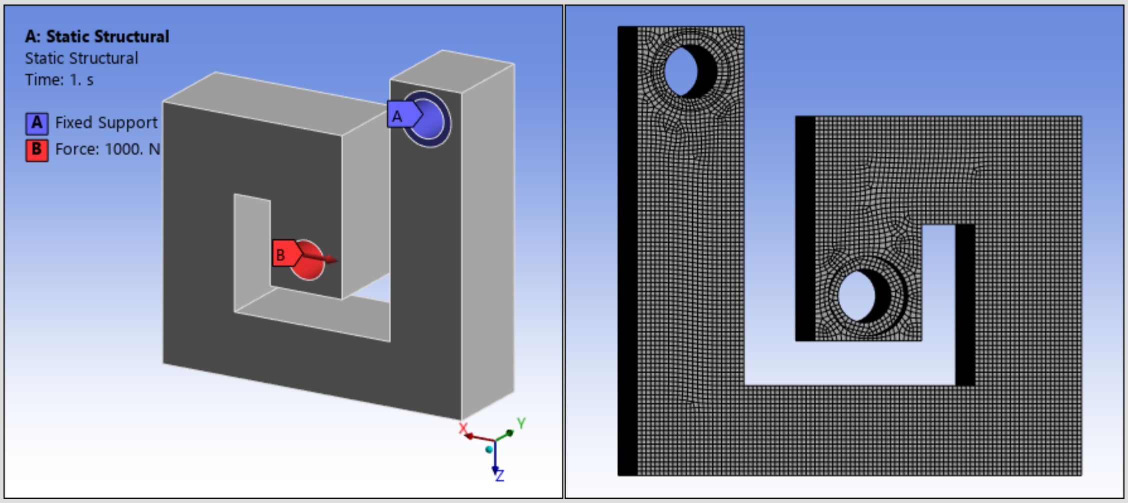
Examples

Pull Out Example

This analysis examines the effect of the **Pull Out Direction** Manufacturing Constraint on a cantilever model with a spiral shape. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.4.

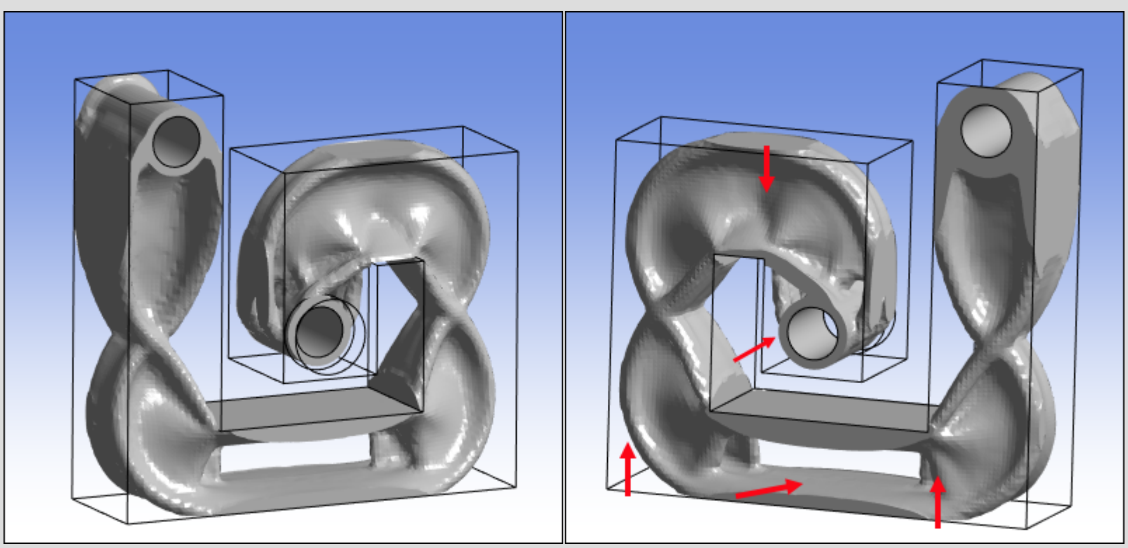
Set Up and Mesh

The environmental conditions and mesh are illustrated here.



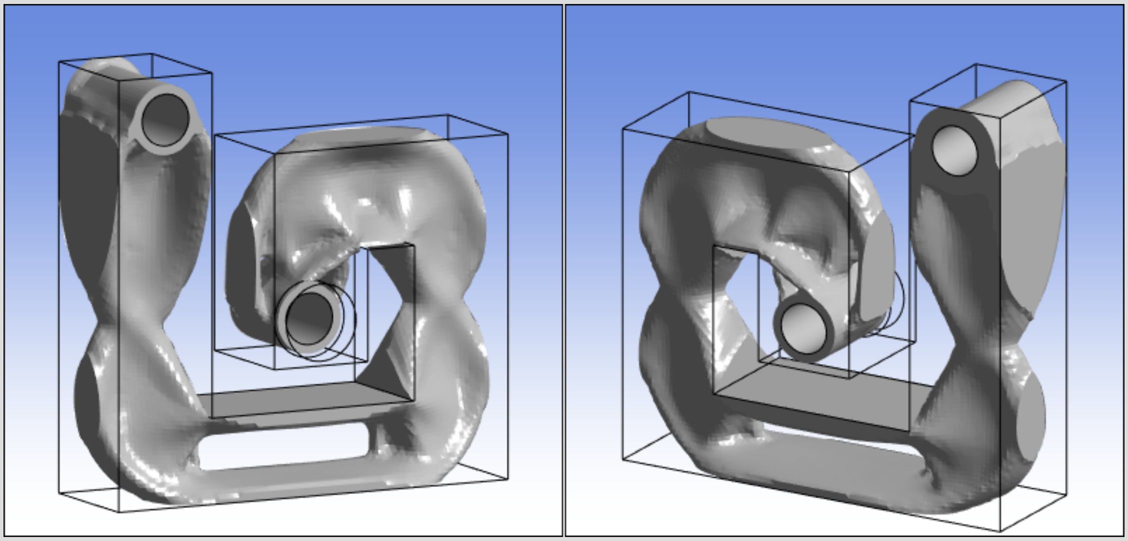
No Manufacturing Constraints

This result shows the optimized shape with no manufacturing constraints. The shape presents some cavities and undercuts along the Y-direction (indicated with red arrows). For this result, the structural compliance equals 5.22538e-02.



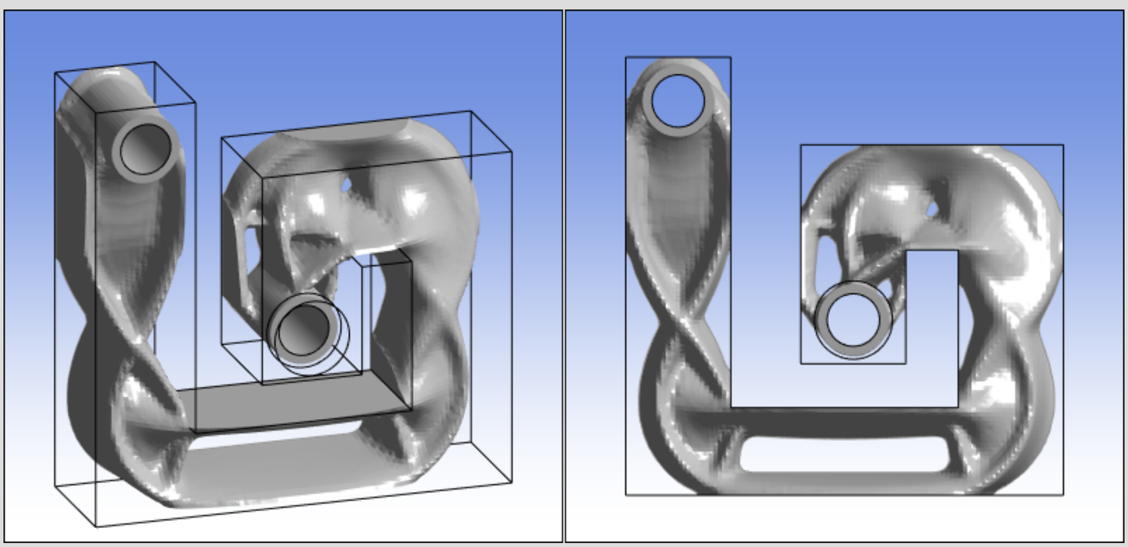
Pull Out Direction in Both Directions

Here is the optimized shape with the **Axis** property set to **Y Axis** and the **Direction** property set to **Both Directions**. Using a two-sided **Pull Out Direction** constraint along the Y axis results, the previous cavities and undercuts have disappeared. The value of the structural compliance increased slightly to 5.27043e-02 (+0,9%).



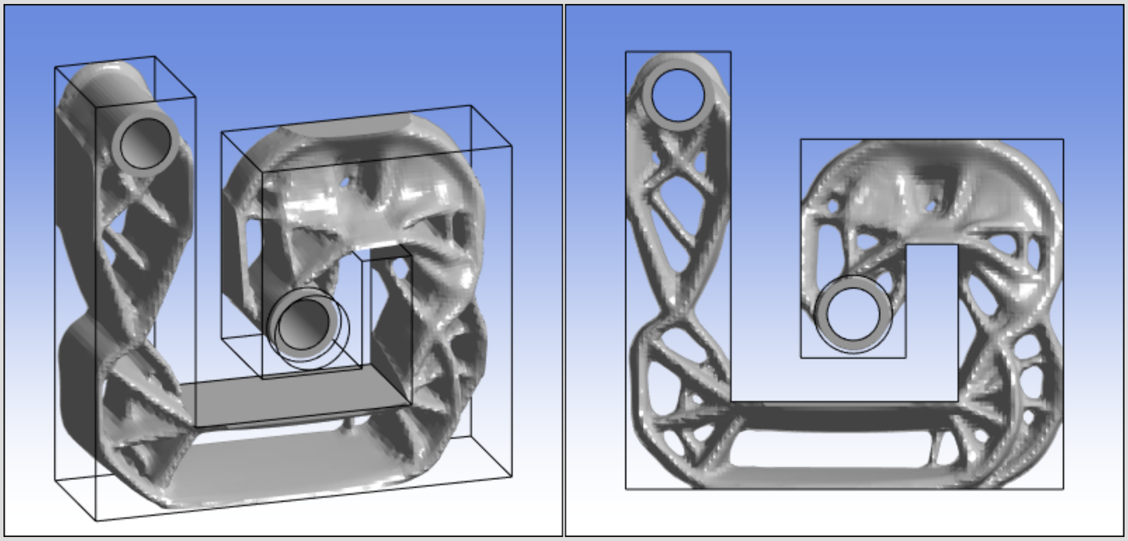
One-Sided Pull Out Direction

As illustrated below, changing the optimized shape by setting the **Axis** property set to **Y Axis** and the **Direction** property set to **Along Axis**, the shape changes dramatically because the side oriented in the -Y direction is considered as a parting surface. Disposing less freedom to evolve than in the two-sided, the value of the structural compliance increases more than previously, to 5.85545e-02 (+12.1%).



Pull Out Direction and Member Size Constraints

The above formulation includes some thick regions that may present a problem to manufacture. Here you add a **Member Size** constraint specified with the **Maximum** property set to **Manual** and the **Max Size** property set to **`4mm`** (d_{max} = 4.0e-03). Adding the constraint helps to redistribute the available material, while respecting the **Pull Out Direction** constraint. The **Max Size** property specification (d_{max} = 4.0e-03) substitutes thick regions with thin nerves. By chance, this solution is better than the initial nominal optimization.

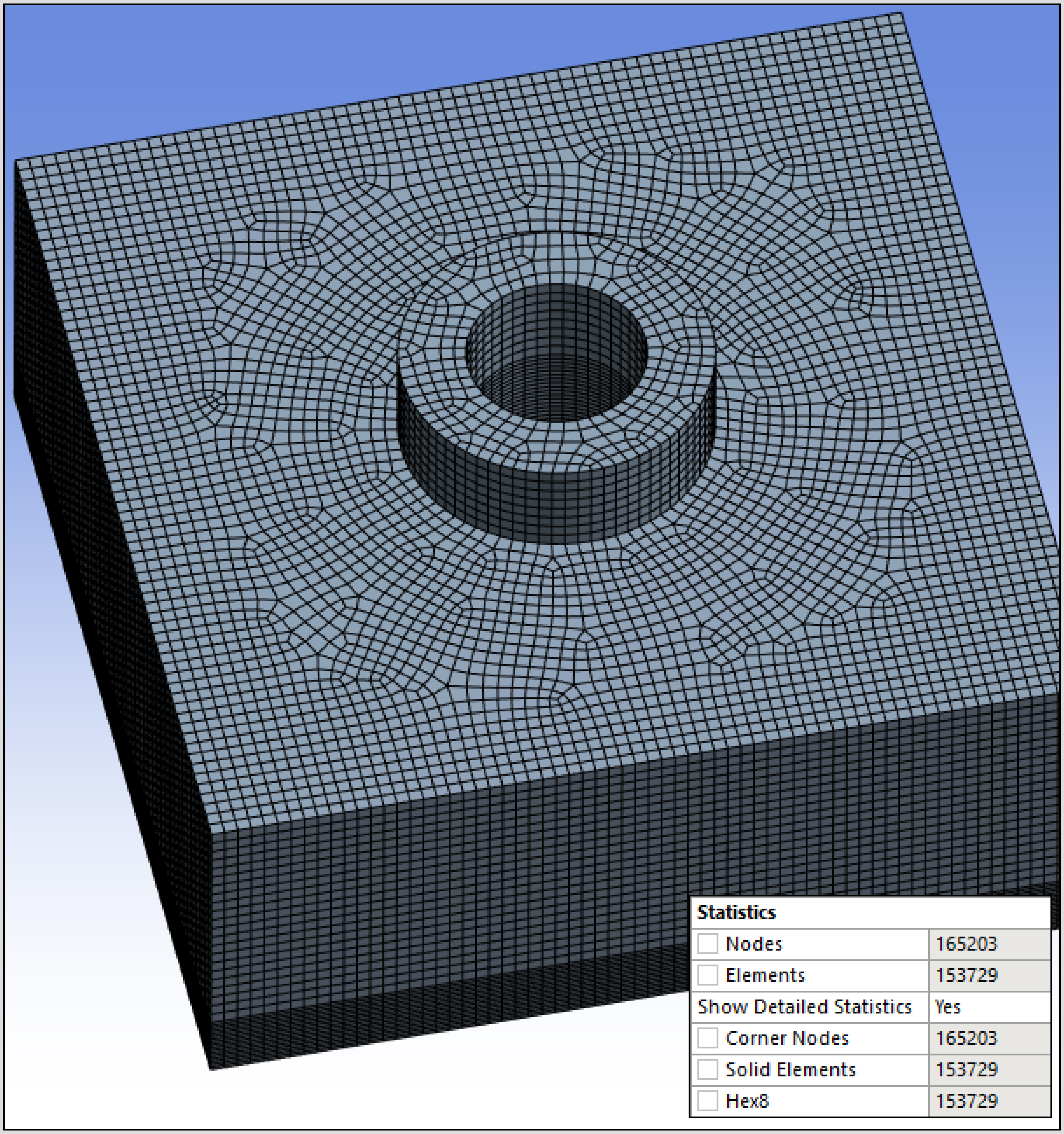


Stamping Example

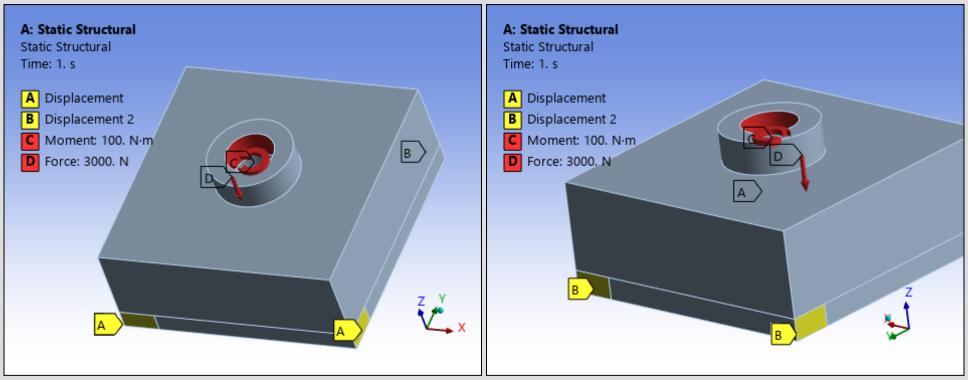
Using the **Level Set Based** optimization method, this analysis examines the effect of the **Pull Out Direction** Manufacturing Constraint on an engine bracket part. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.2.

Set Up and Mesh

The mesh and environmental conditions are illustrated here. The finite element model contains 153,729 elements.

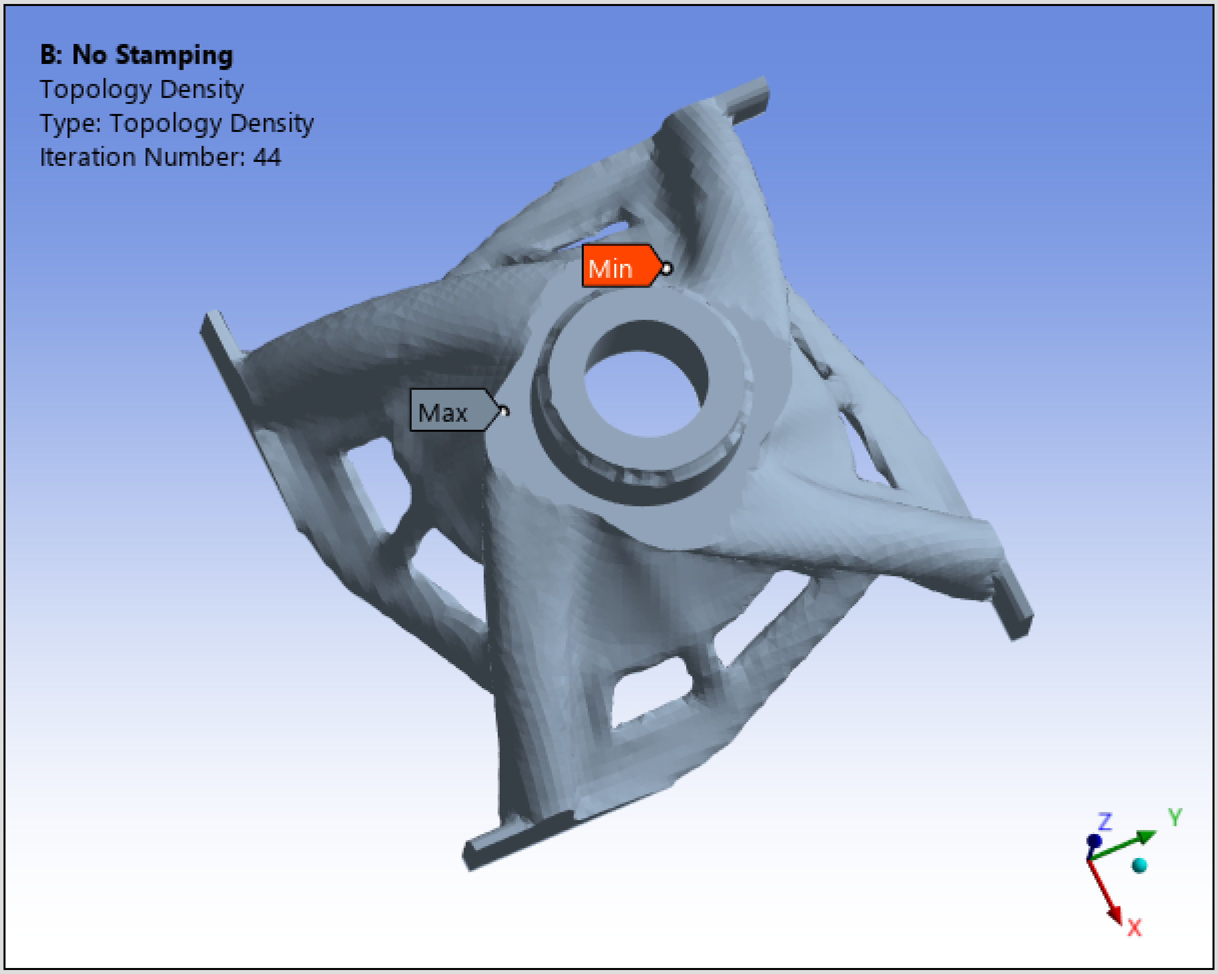


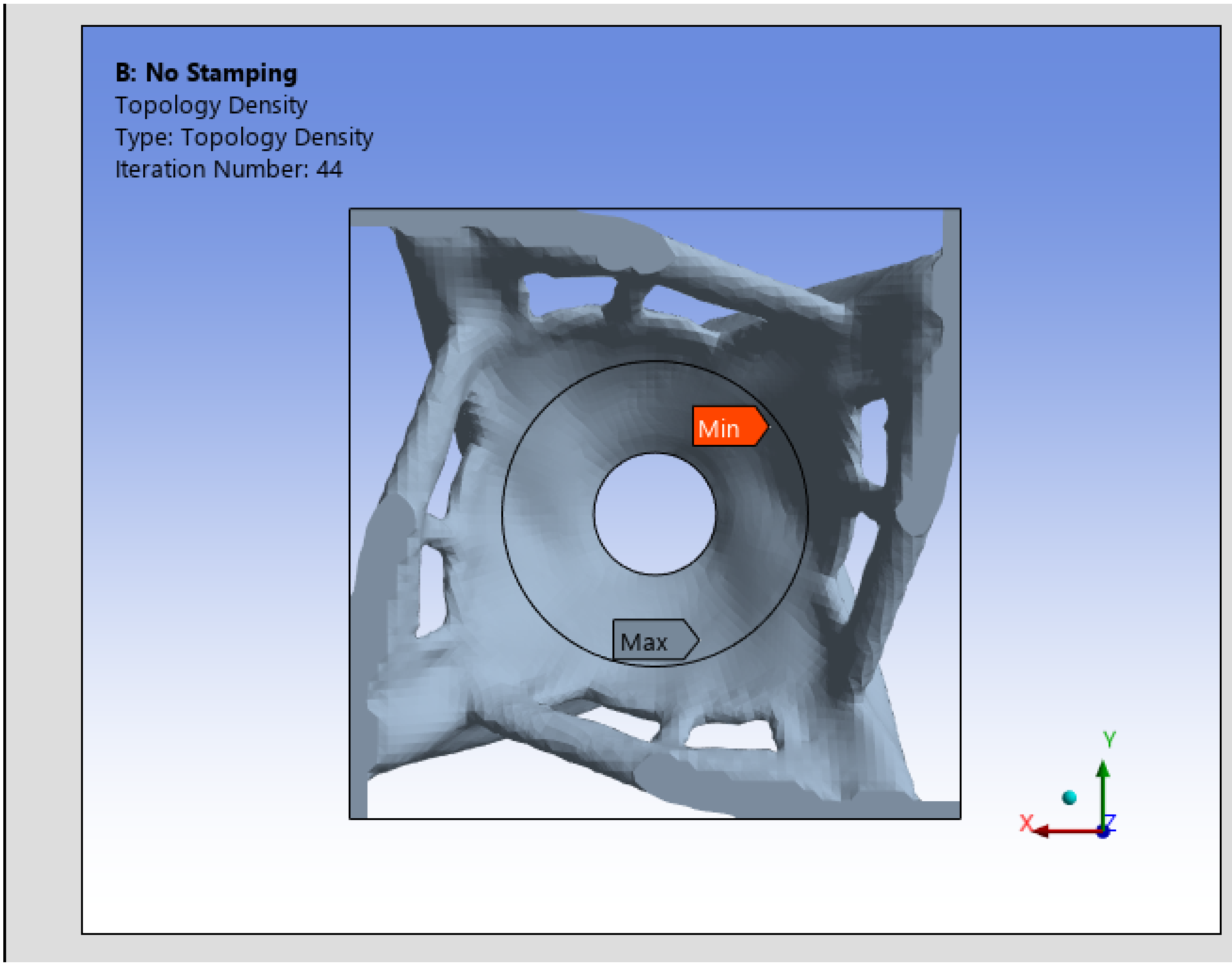
Displacements (A and B) are scoped to two faces at the left hand corner of the base. A Moment and Force (C and D) are specified on the inner surface of the bracket.


A: Static Structural
Static Structural
Time: 1. s
A Displacement
B Displacement 2
C Moment: 100. N·m
D Force: 3000. N
A: Static Structural
Static Structural
Time: 1. s
A Displacement
B Displacement 2
C Moment: 100. N·m
D Force: 3000. N

No Manufacturing Constraints

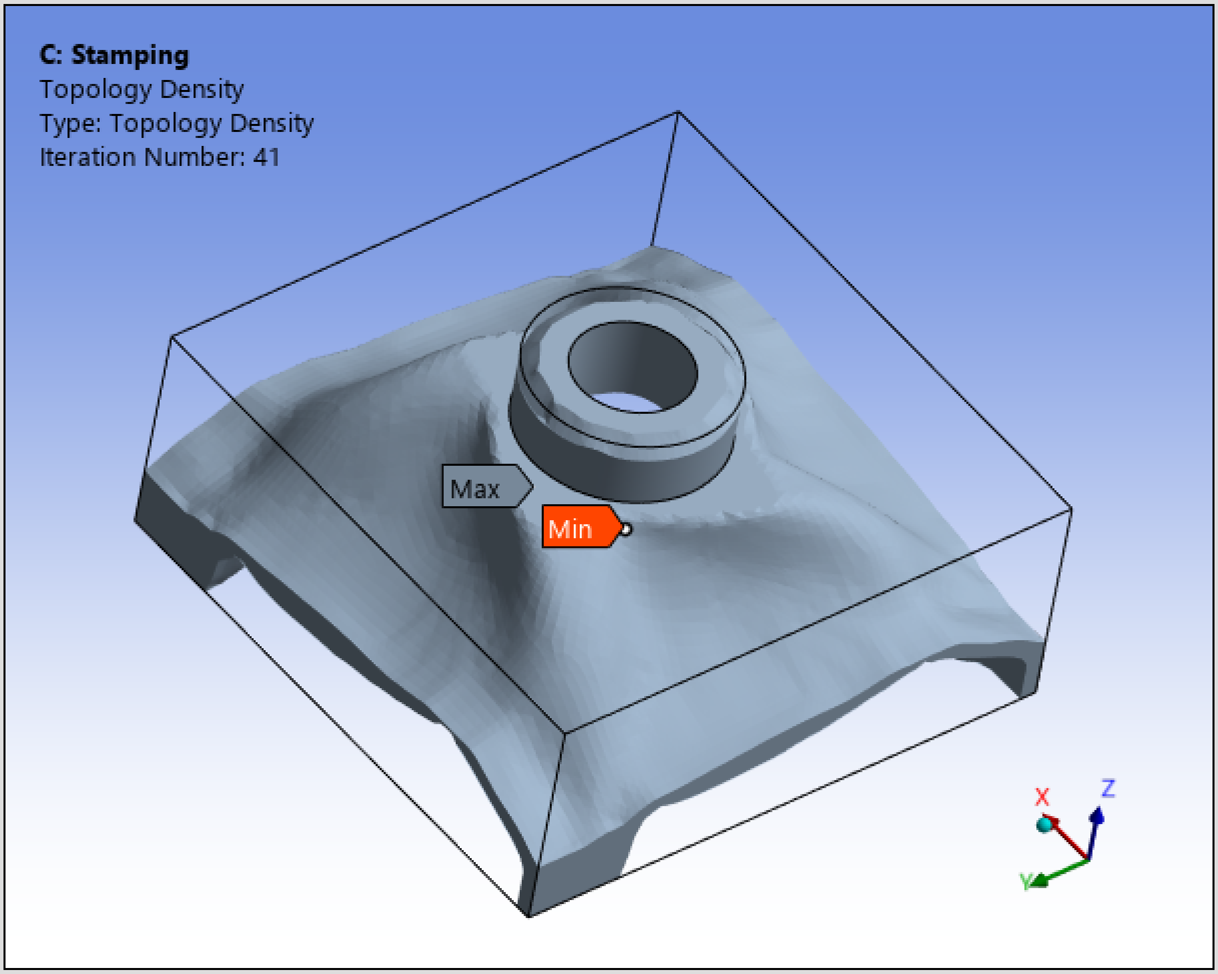
The optimized result without considering manufacturing constraints is illustrated below. The structural compliance equals 0.0159048.

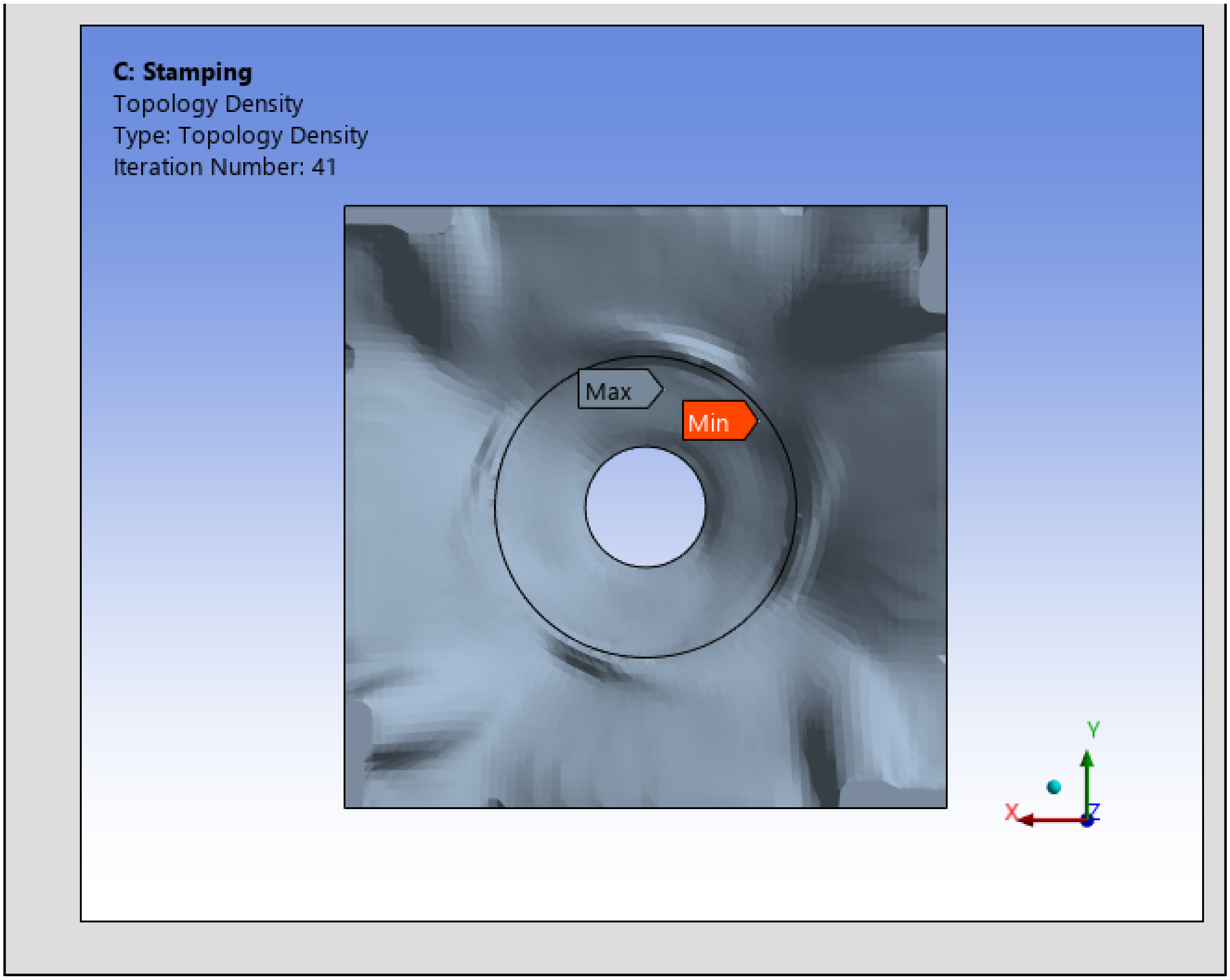



B: No Stamping
Topology Density
Type: Topology Density
Iteration Number: 44
Min
Max
Y
X
Z

Pull Out Constraint

Here, the **Pull Out Option** property is set to **Stamping** and specified in the Z-direction. This produces an optimized shape where no perforation appears in the Z-direction. The value of the structural compliance increases to 0.0176963 (+11.3%).



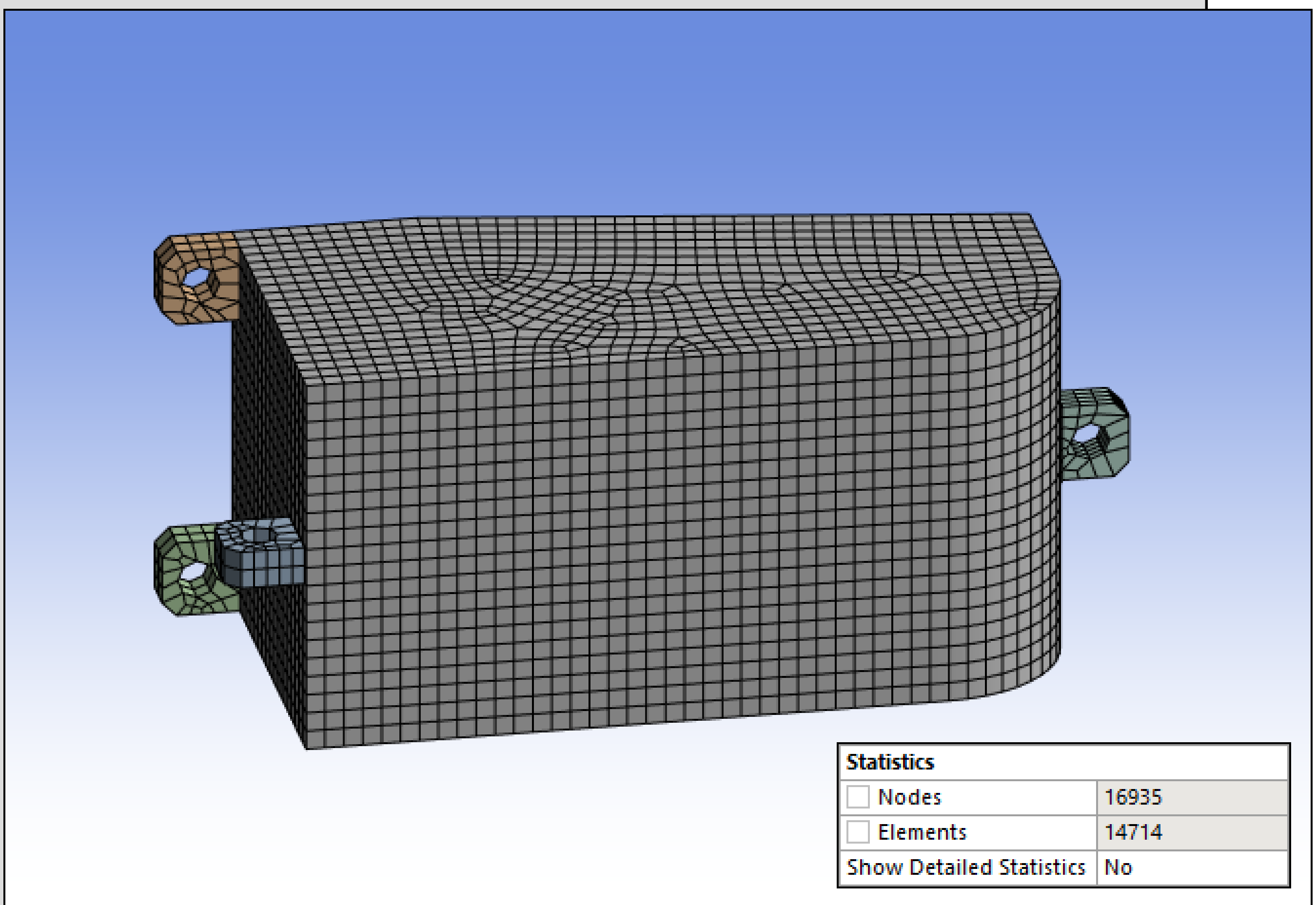

C: Stamping
Topology Density
Type: Topology Density
Iteration Number: 41
Max
Min
Y
X
Z

No Hole Example

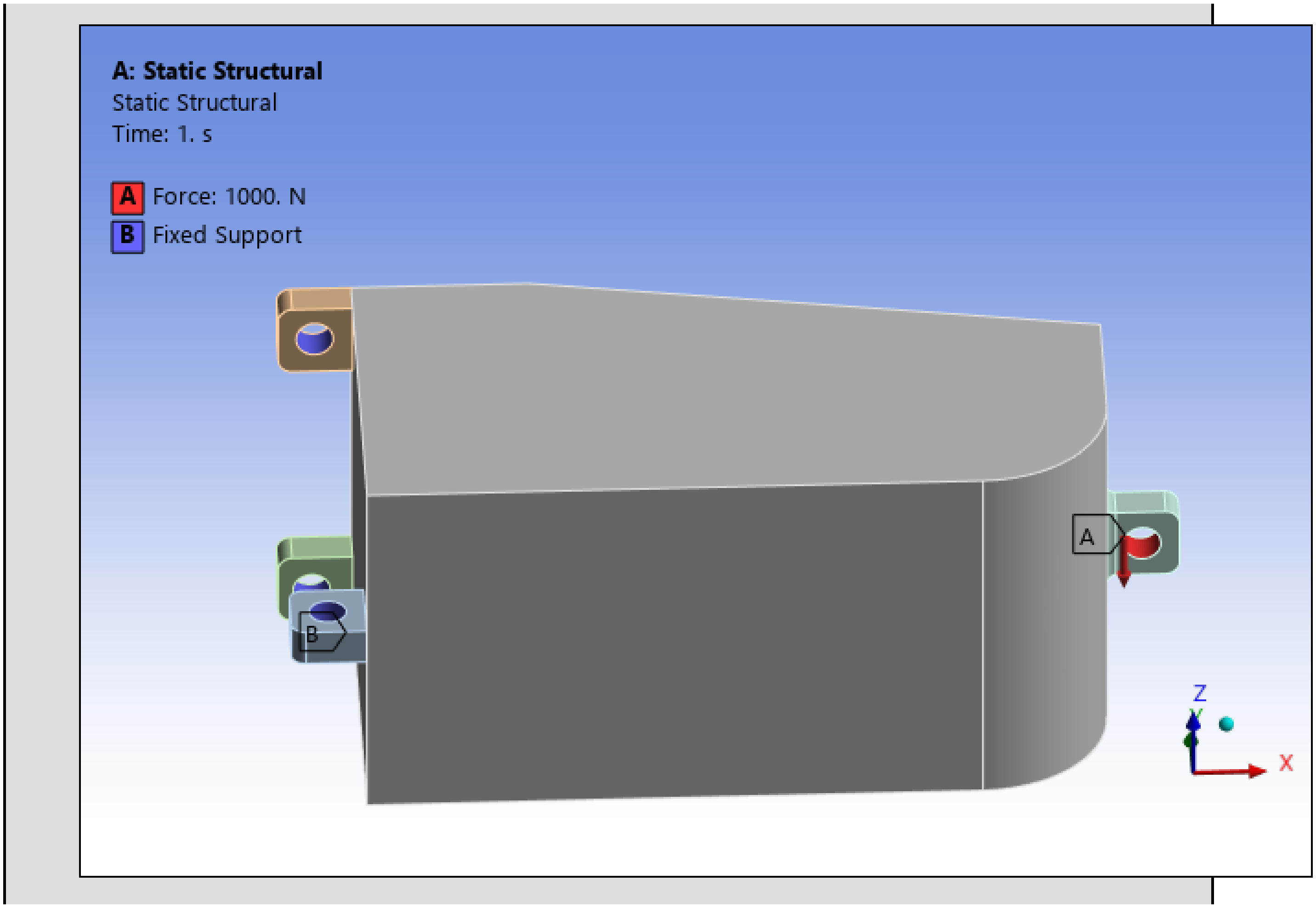
Using the **Level Set Based** optimization method, this analysis examines the effect of the **Pull Out Direction** Manufacturing Constraint on a suspending arm. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.4 and a two-sided Pull Out Direction constraint in the Y-axis.

Set Up and Mesh

The mesh and environmental conditions are illustrated here. The finite element model contains 14,714 elements.

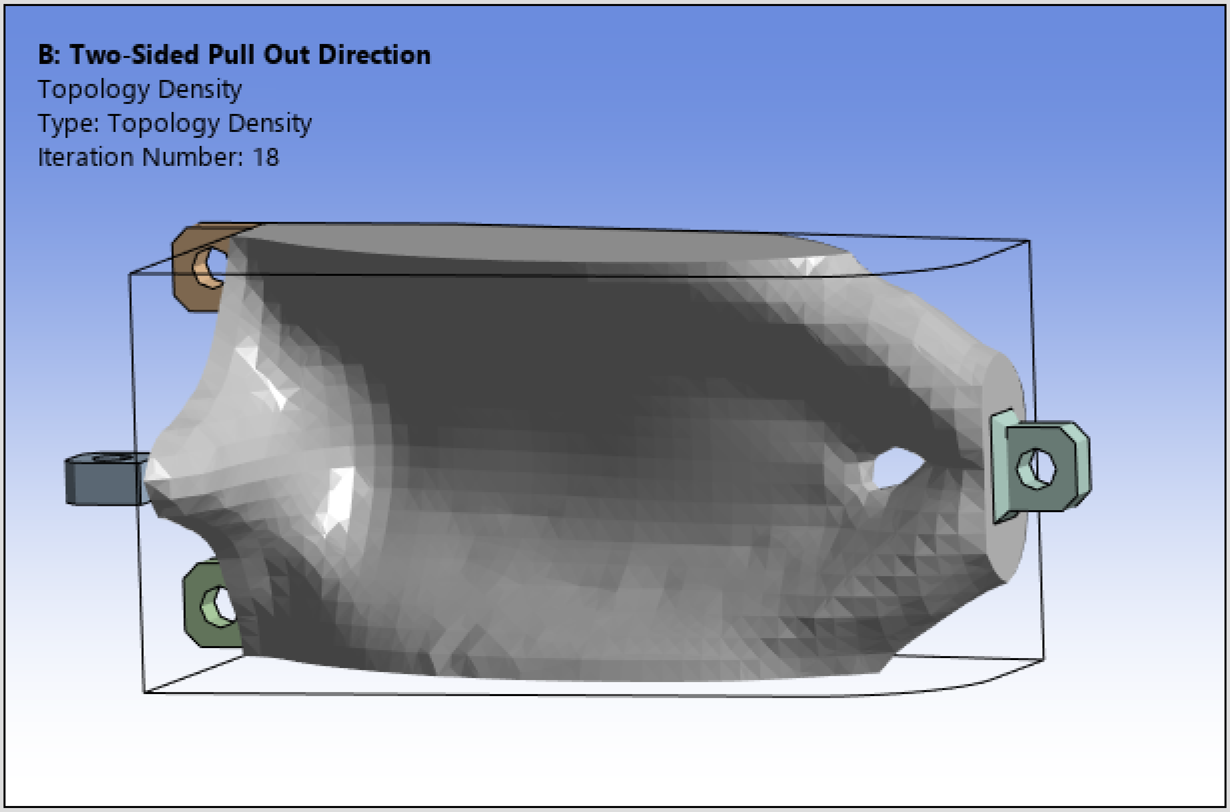


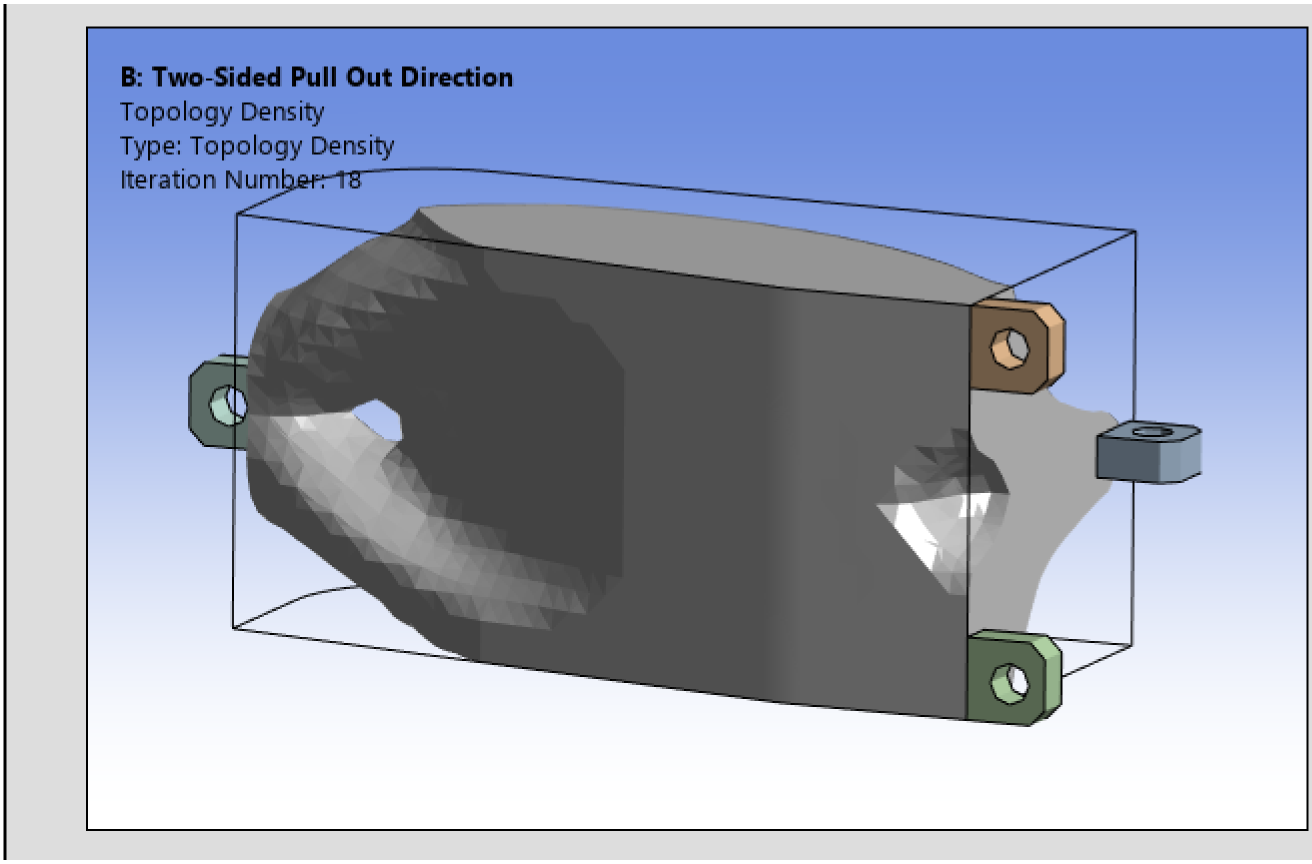
A Fixed Support (B) is scoped to the inner surface of the three parts on the right side of the model and a vertical Force (A) is applied to the inner surface of the part on the right side.


A: Static Structural
Static Structural
Time: 1. s
A Force: 1000. N
B Fixed Support
A
B
Z
Y
X

Pull Out Constraint

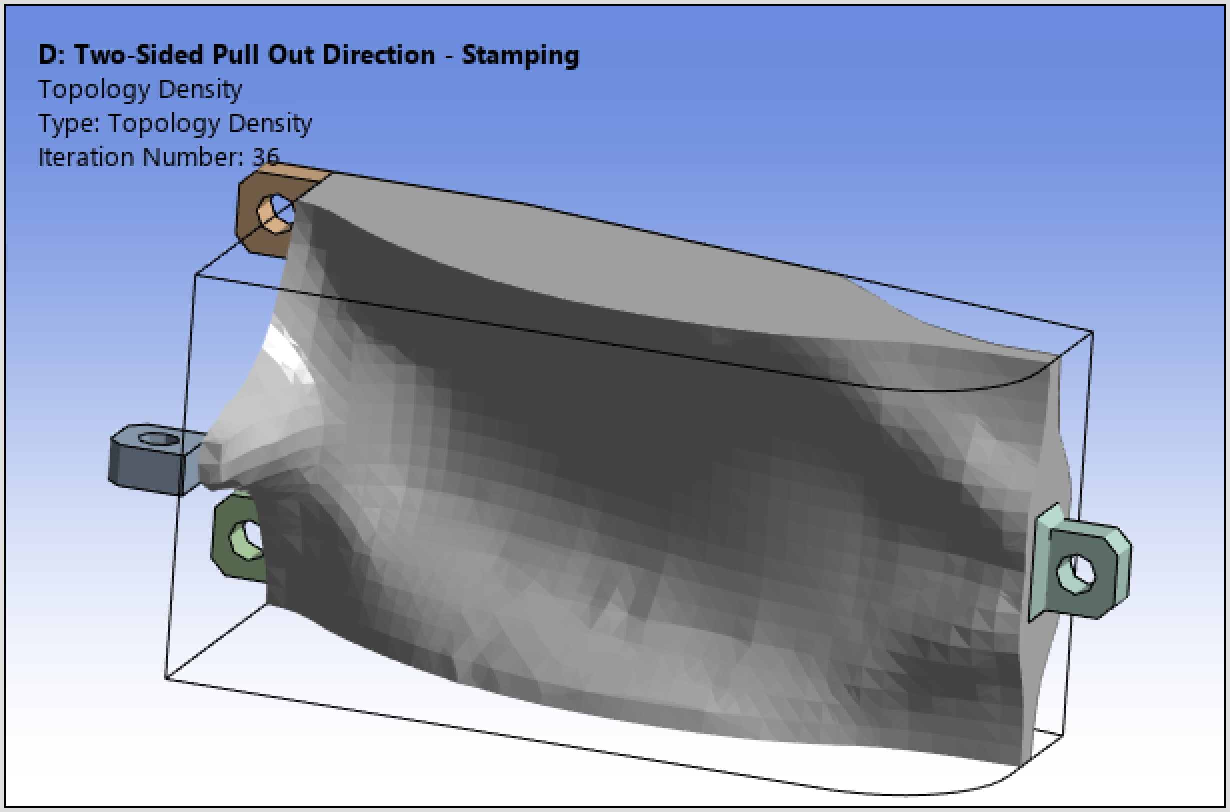
For this result, the **Pull Out Option** property is set to **None**. The optimized shape for the constraint pulled out in both directions (**Direction** property set to **Both Directions**) along the Y-axis (**Axis** property set to **Y Axis**). The structural compliance equals 1.34666e-03.

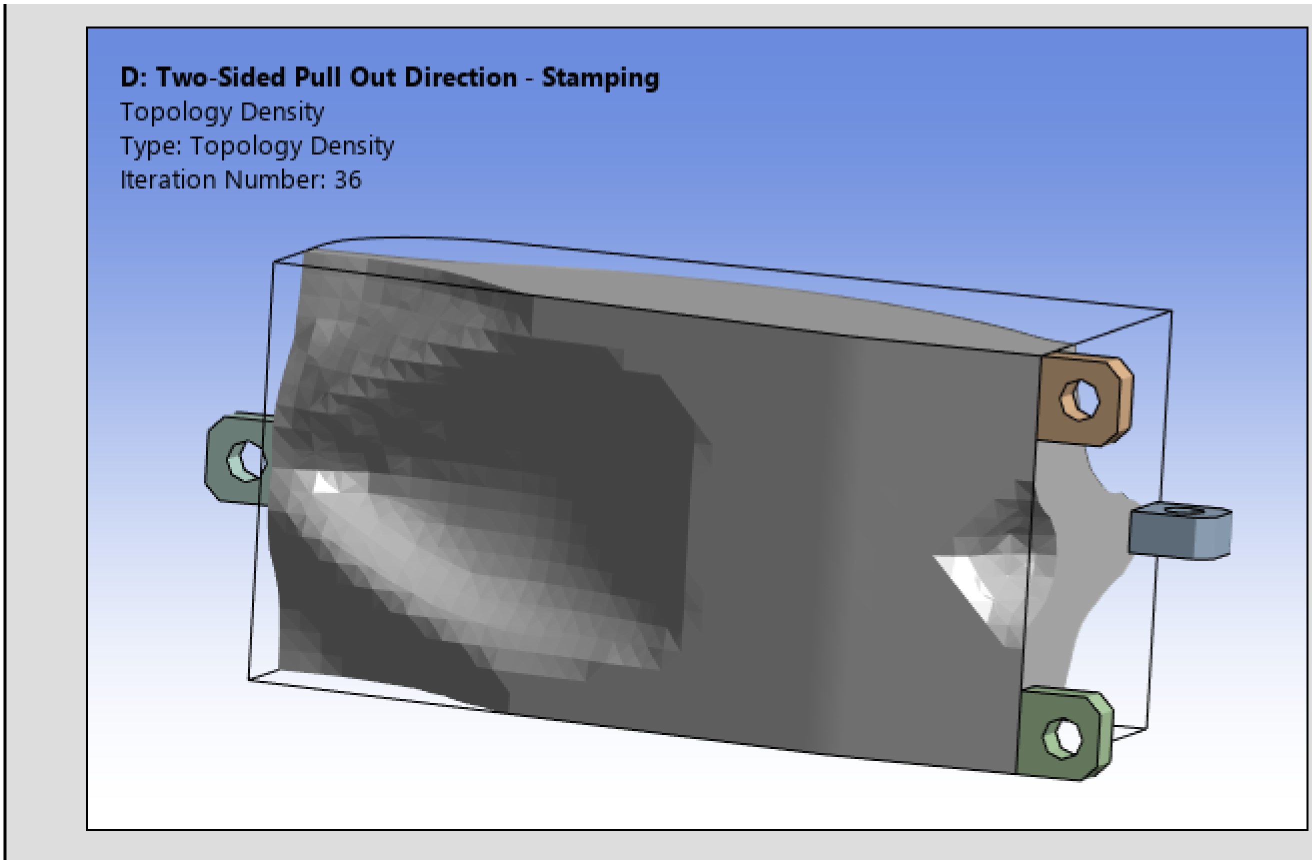



B: Two-Sided Pull Out Direction
Topology Density
Type: Topology Density
Iteration Number: 18

Pull Out Constraint Stamping

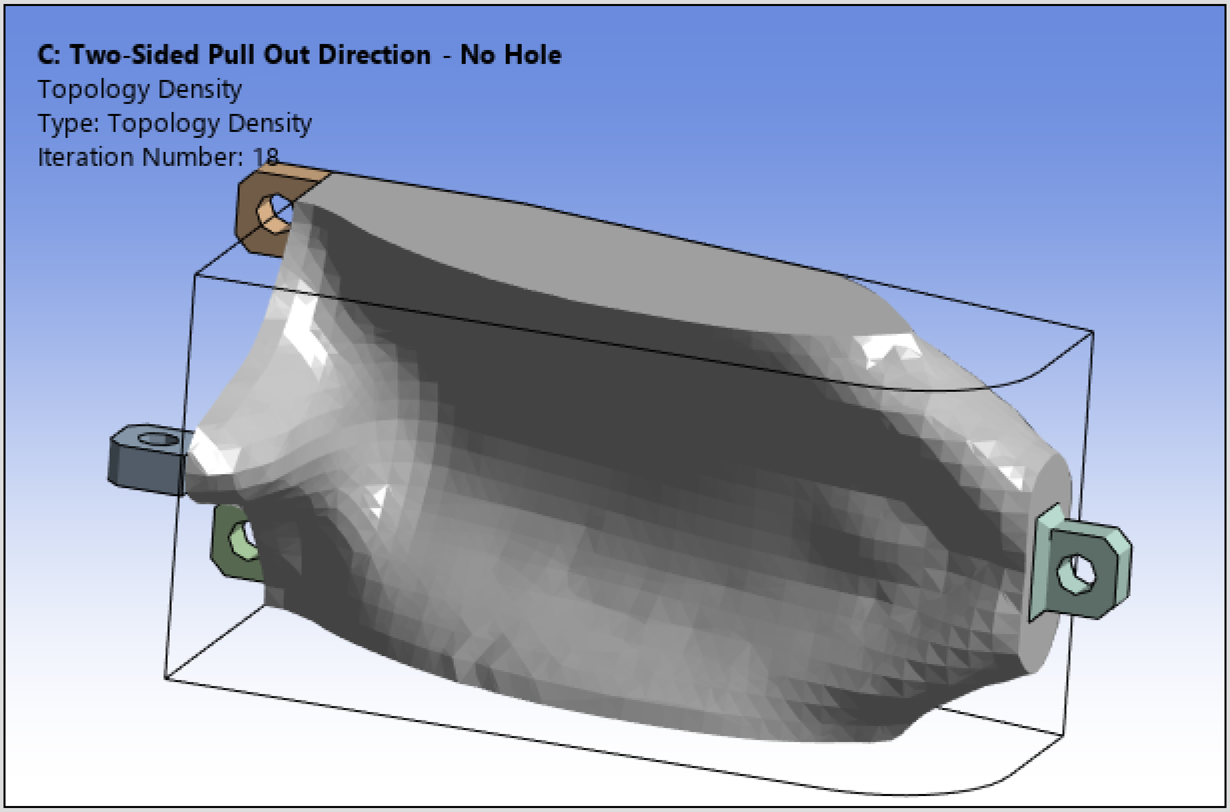
For this result, the **Pull Out Option** property is set to **Stamping**. The optimized shape for the constraint pulled out in both directions (**Direction** property set to **Both Directions**) along the Y-axis (**Axis** property set to **Y Axis**). There are now no perforations along the Y-axis. The structural compliance increases to 1.3527e-03 (+0.45%).

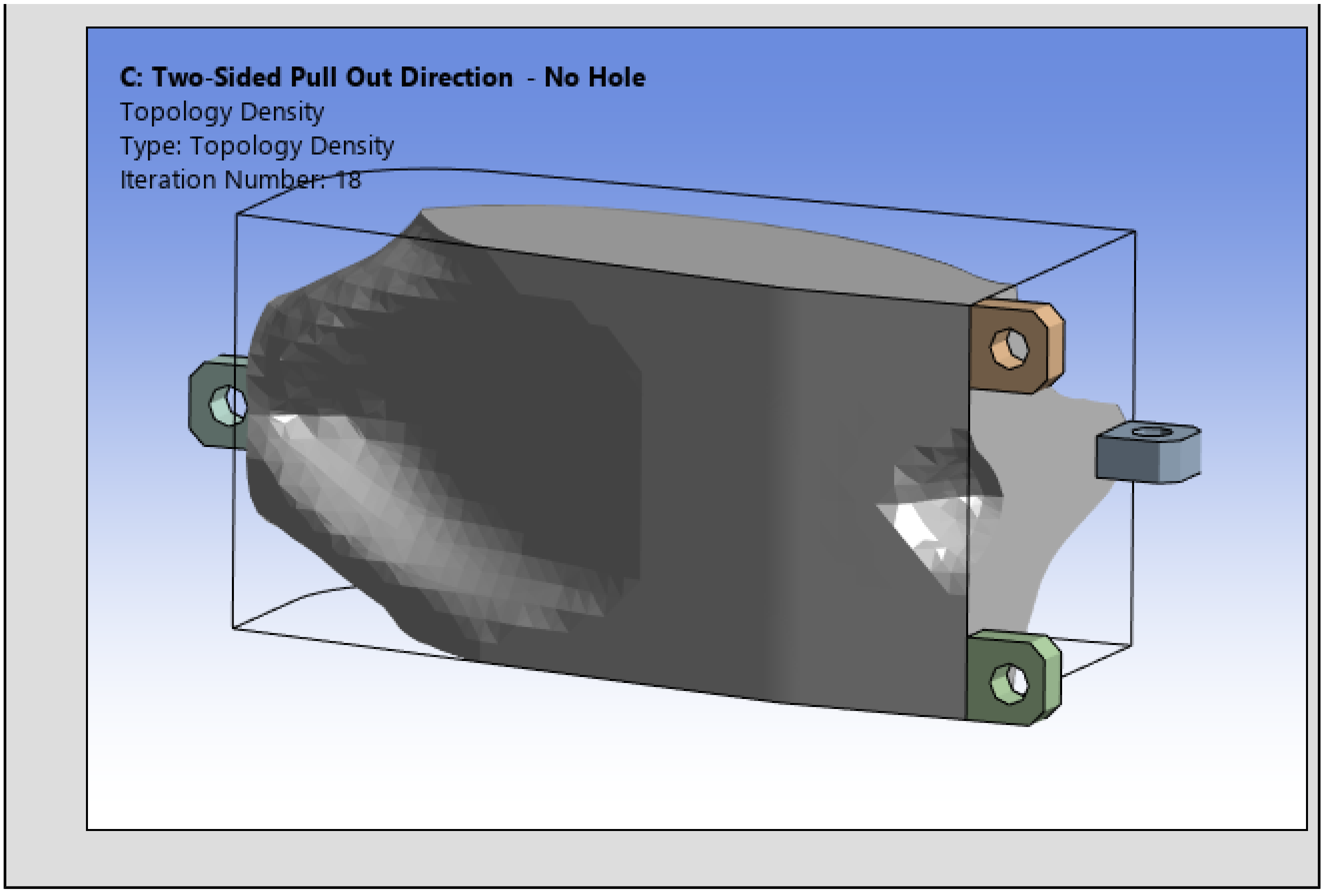



D: Two-Sided Pull Out Direction - Stamping
Topology Density
Type: Topology Density
Iteration Number: 36

Pull Out Constraint No Hole

For this result, the **Pull Out Option** property is set to **No Hole**. The optimized shape for the constraint pulled out in both directions (**Direction** property set to **Both Directions**) along the Y-axis (**Axis** property set to **Y Axis**). Compared to the default option, the small hole close to the loading disappears. More freedom to shrink perpendicular to the Y-axis is provided compared to the **Stamping** option. The structural compliance is similar to the result shown above (+0.1%).





References

[13 (p. 195)] G. Allaire, F. Jouve, G. Michailidis, Molding direction constraints in structural optimization via a level-set method, Variational Analysis and Aerospace Engineering: Mathematical Challenges for the Aerospace of the Future, 2016.

4.1.5. AM Overhang Constraint

Overhang features, or simply "overhangs," are unsupported solid features that rise in the build direction when using Additive Manufacturing (AM) processes. They are mainly related to the printing angle, but other factors, such as feature length, may also be included.

Available for the **Mixable Density**, **Density Based**, and **Level Set Based** optimization methods, the AM Overhang Constraint (p. 48) manufacturing constraint aims to create self-supporting designs that prevent the formation of these overhanging regions.

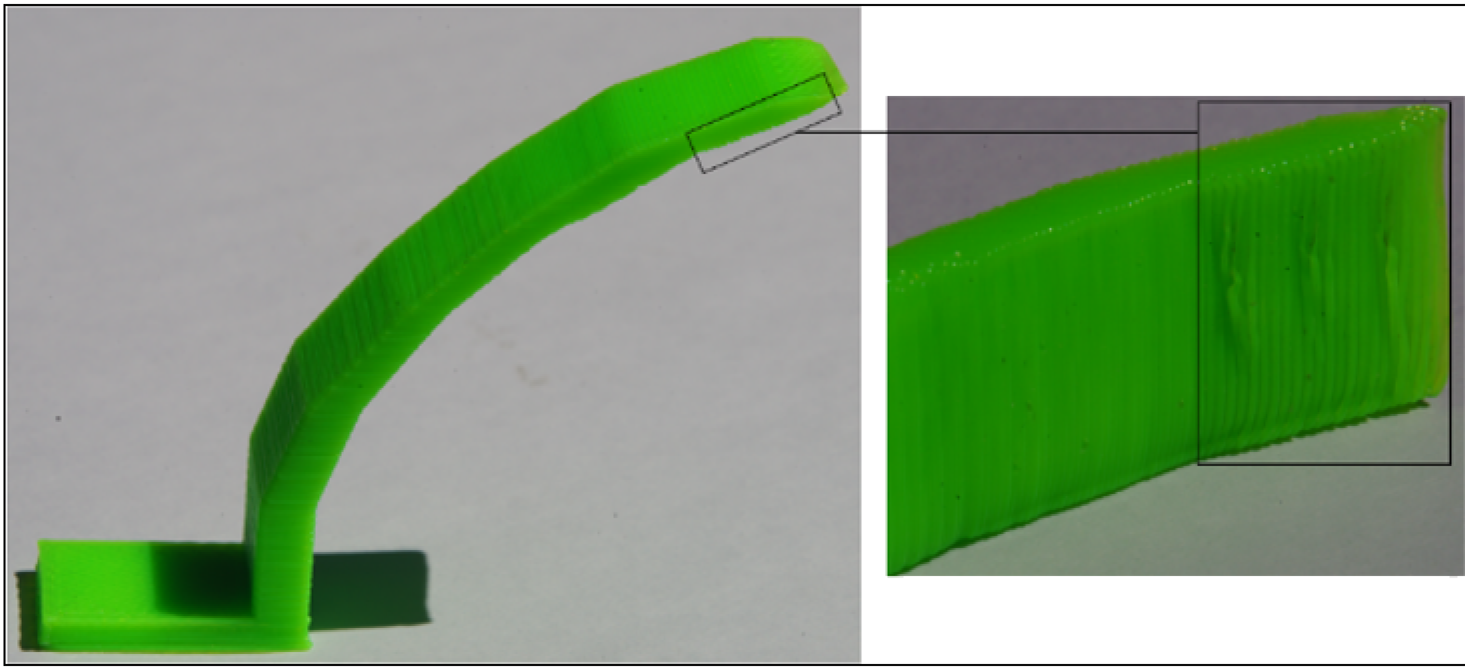
Go to a section topic:

- Overview (p. 184)
- Setup and Technical Details (p. 185)
- Recommendations and Observations (p. 186)
- Example (p. 186)

Overview

For any AM process, support structures can lead to excessive printing time and cost [15 (p. 195)]. More precisely, for metal powder bed fusion processes, overhang features can significantly impact the structural quality of the printed part. For fused deposition modeling (FDM) processes, the existence of supports may significantly deteriorate the quality of the outer surface, having a strong visual impact on the printed part, as illustrated below.

Figure 4.1: Printing Angle Impact on Quality of Surface in FDM [15 (p. 195)].



In theory, the computation of the overhang angle constraint is performed in two steps:

1. The inner product $\vec{n} \cdot \vec{d}$ between the exterior normal vector and the build direction is computed for every point of the structural boundary.

2. The minimum value of $\vec{n} \cdot \vec{d}$ shall be greater than the threshold provided by the angle $\phi_{\max} = \pi - \phi_{overhang}$. That is:

$$\vec{n} \cdot \vec{d} \geq \cos(\phi_{\max})$$

However, the above formulation presents two drawbacks:

- It is not derivable due to the minimum operator and therefore not suitable for gradient-based optimization algorithms.

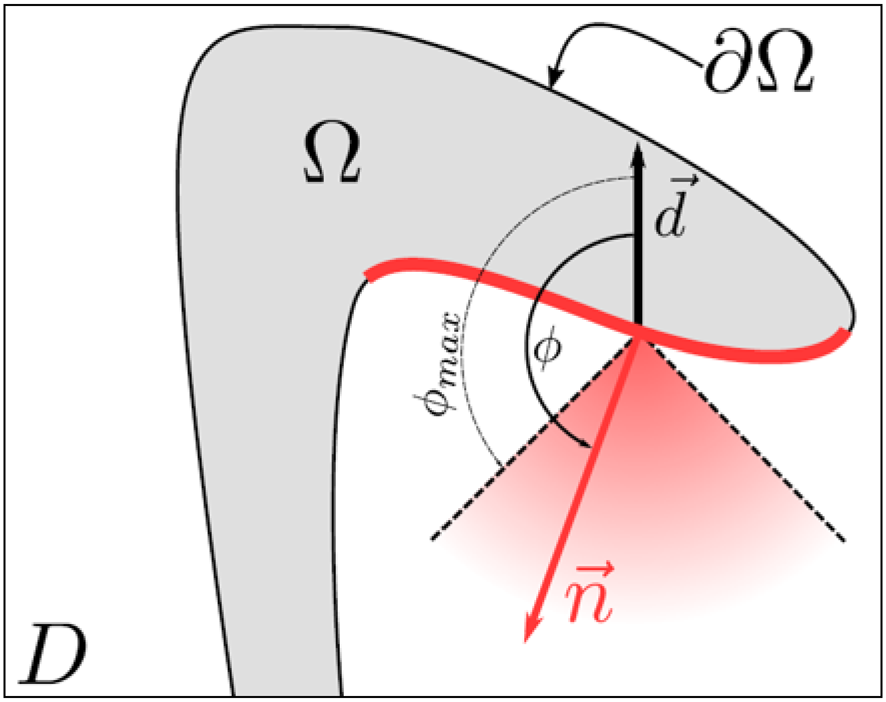
- It is not applicable for all methods, mainly because it requires geometric information that is not directly available for the density-based methods.

The devised formulation depends on the Structural Optimization framework described in the following table.

Table 4.1: AM Overhang Constraint Formulation

Mixable Density /Density Based Methods	Level Set Based Method
The built-in approach is based on filtering techniques [16 (p. 195)].	The theoretical definition is adopted and is appropriately adjusted to overcome the differentiability issue [15 (p. 195)].

Figure 4.2: Overhanging Region (red) for Shape Ω Printed in Direction $\vec{d}$.



Setup and Technical Details

The setup requires specifying a **Build Direction** ($\vec{d}$) and the **Overhang Angle** ($\phi_{overhang}$).

Table 4.2: Setup Parameters and Technical Details

	Mixable/Density Based Method	Level Set Based Method
$\phi_{overhang}$	$27° \leq \phi_{overhang} \leq 60°$.	
Exclusion Region	You can decide whether to Include or Exclude Exclusions from the Analysis Settings.	Exclusion regions are inherently considered in the overhangs evaluation.
Working Domain		All boundaries of the working domain that are not self-supported are considered as print beds. Namely, they are ignored by the constraint.

Recommendations and Observations

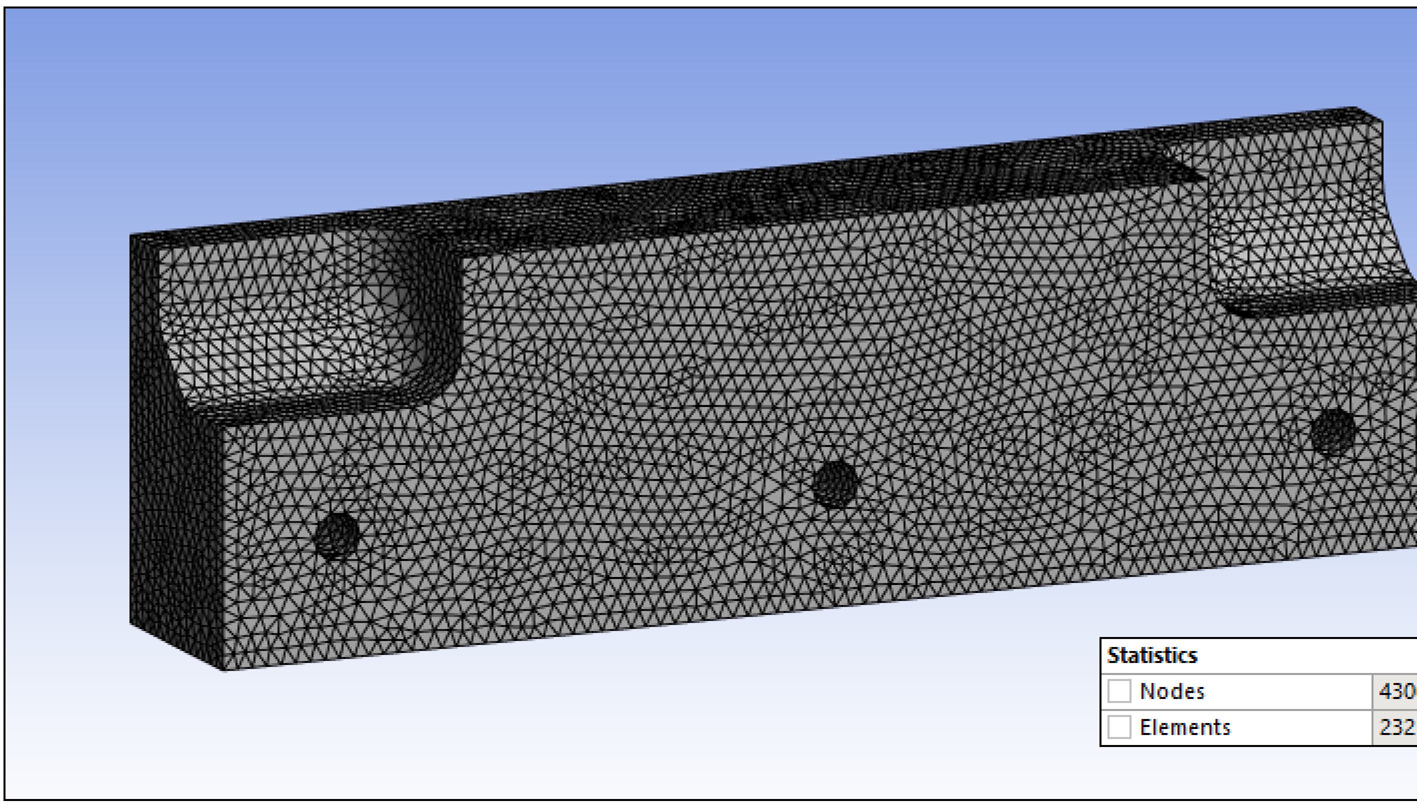
- For the **Level Set Based** optimization method, the angle limit may still be violated along sharp lines, where the normal vector is not accurately defined (see illustration for Y axis overhang (p. 188)) [16 (p. 195)]. Such shapes require linear supports during the printing process, that is, supports along some two dimensional curve of the print bed.
- As with any constraint, the **AM Overhang Constraint** affects the objective performance. Namely, by increasing the allowed overhang angle ($\phi_{overhang}$), the feasible domain shrinks and likely results in smaller objective gain.

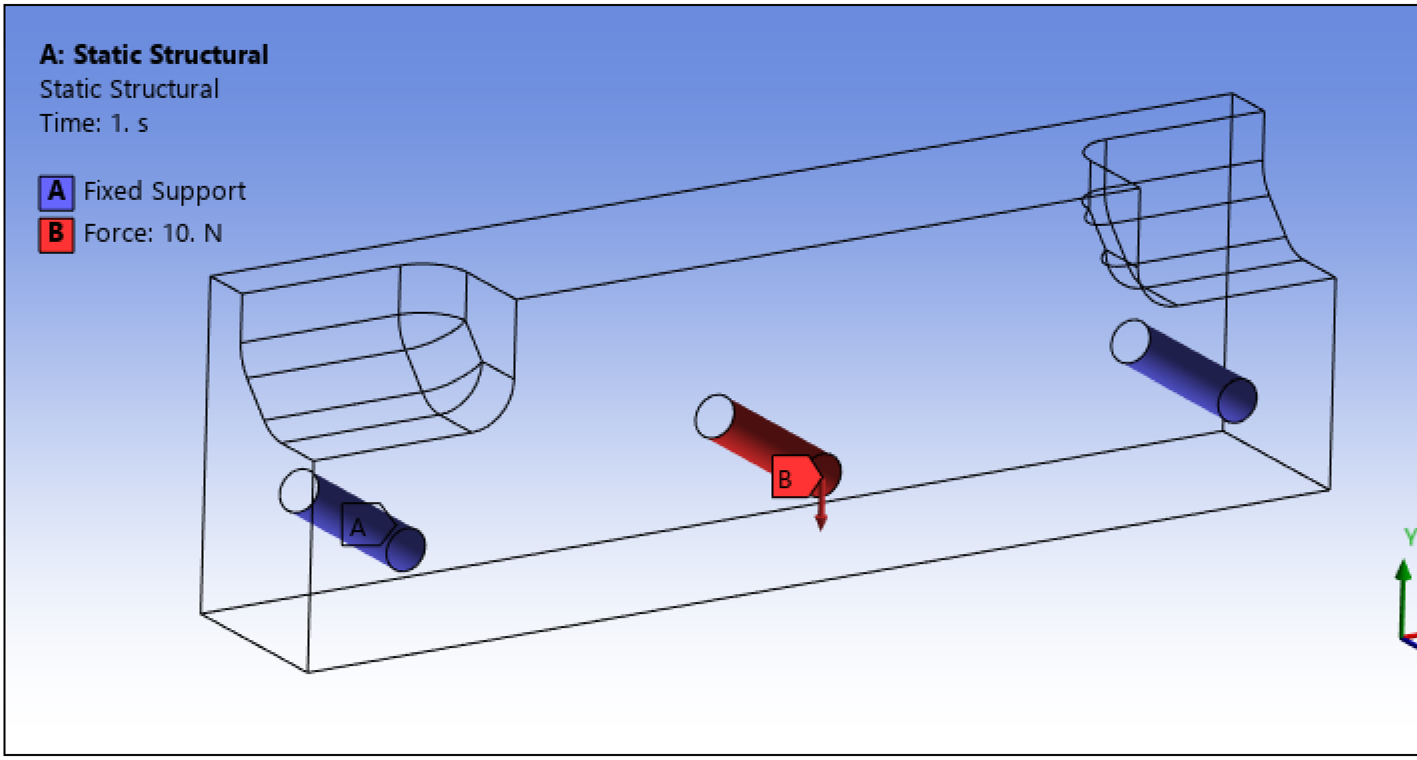
Example

This example examines the effect of an **AM Overhang Constraint** Manufacturing Constraint on an engine bracket model.

Set Up and Mesh

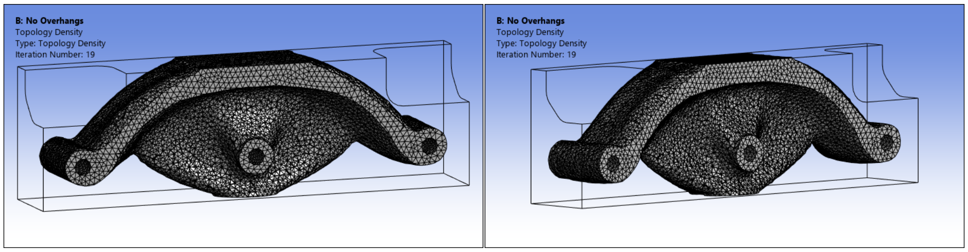
The specified mesh and the setup of the environment, shown in wireframe mode so that you can clearly see the specified boundary conditions, are illustrated below. The optimization problem is to minimize the structural compliance under a volume fraction constraint of 0.4.





No AM Overhang Constraint

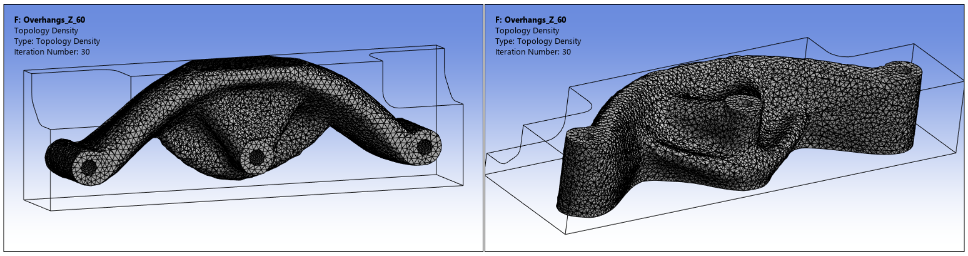
Here are illustrations of the optimized result without considering an **AM Overhang Constraint**. The structural compliance equals 1.29836e-07.



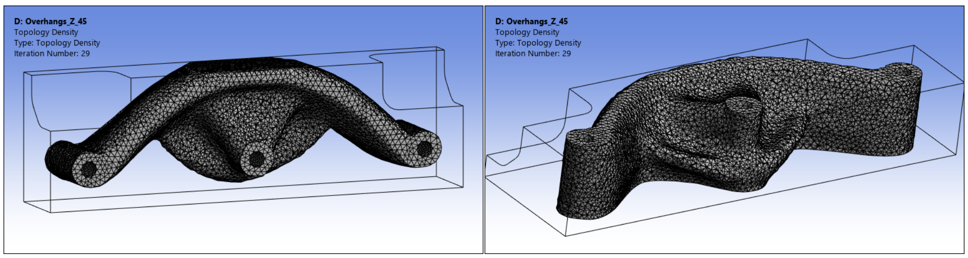
AM Overhang Constraint Z Axis

In the following, an **AM Overhang Constraint** is added for a build direction in the positive Z axis ($\vec{d} = +Z$) and for varied overhang angles ($\phi_{overhang}$ = 60°, 45°, and 30°). The structural compliance is increased by +7.4%, +6.3%, and +5.7% respectively and, as expected, increasing the overhang-angle results in less objective gain.

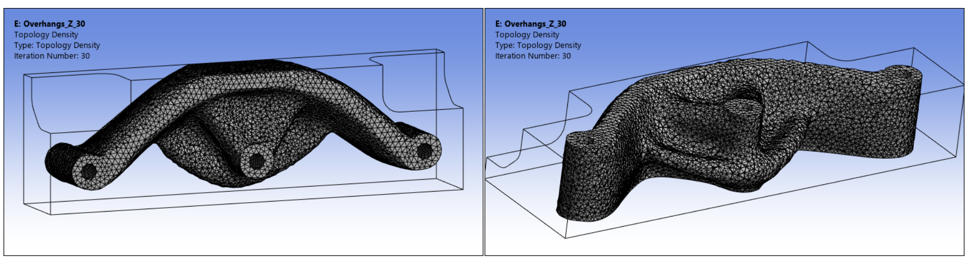
AM Overhang Constraint + Z 60°



AM Overhang Constraint + Z 45°

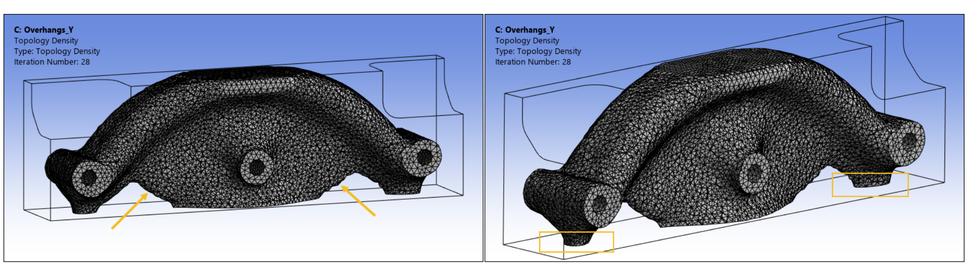


AM Overhang Constraint + Z 30°



AM Overhang Constraint Y Axis

In the following, an **AM Overhang Constraint** is added for a build direction in the positive Y axis ($\left(\vec{d} = +\mathrm{Y}\right)$) and for an overhang angle of $\phi_{overhang}$ = 45°. The structural compliance equals 1.32978e-07 (+2.4%). Overhanging regions are removed, and the shape appears to be self-supporting. However, as shown, this is achieved by forming linear overhangs and using some material as supports for the 3D printing. This indicates that the choice of build-direction is not appropriate.



4.1.6. Housing

Supported for the **Mixable Density** and **Level Set Based** optimization methods, the **Housing** constraint enables you to create watertight designs that enclose a given set of faces.

Description and Industrial Motivation

This constraint is used for several industrial applications where the structural design must not exhibit holes and perforations, such as designing a container to house a liquid region.

Setup and Technical Details

This constraint requires you to specify faces that will encapsulate a shape. The final shape must contain at least one layer of material of thickness dx_{AVG} surrounding the selected surfaces.

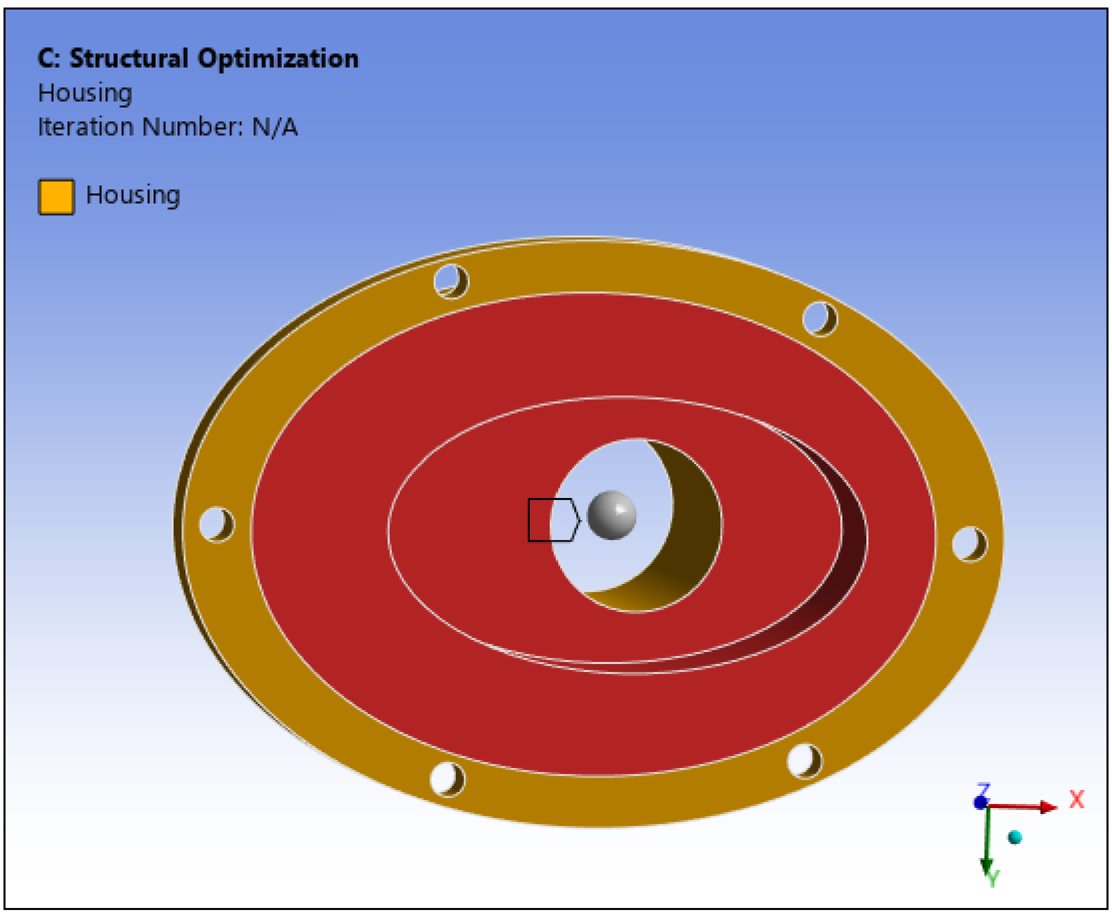
Recommendations and Observations

Ansys recommends that you have at least two layers of elements above the selected surfaces.

As with any constraint, the **Housing** constraint affects the objective performance. Due to the imposed geometric restriction, the feasible domain shrinks and probably results in smaller objective gain.

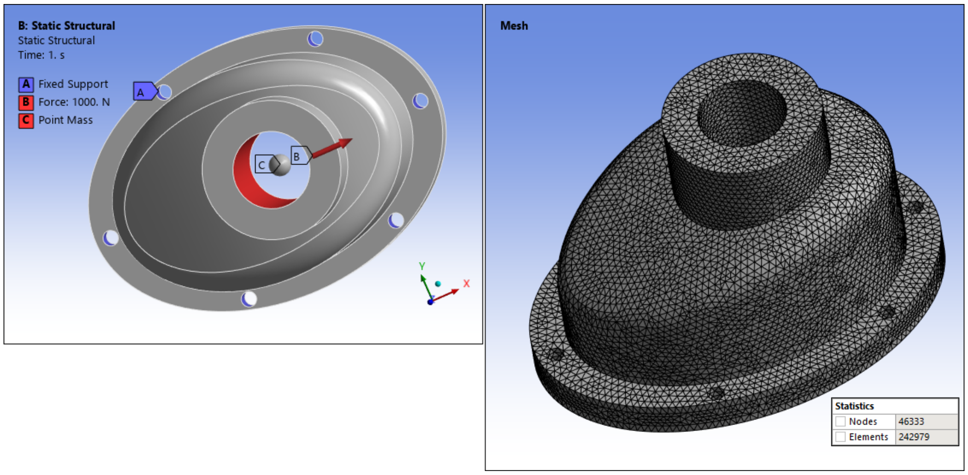
Example

This example examines a clutch housing. Using the **Level Set Based** Optimization Method, this problem consists of minimizing the structural compliance under a volume fraction constraint of 0.5. In addition, it is required that no perforations exists between the exterior and the surfaces shown below in red.



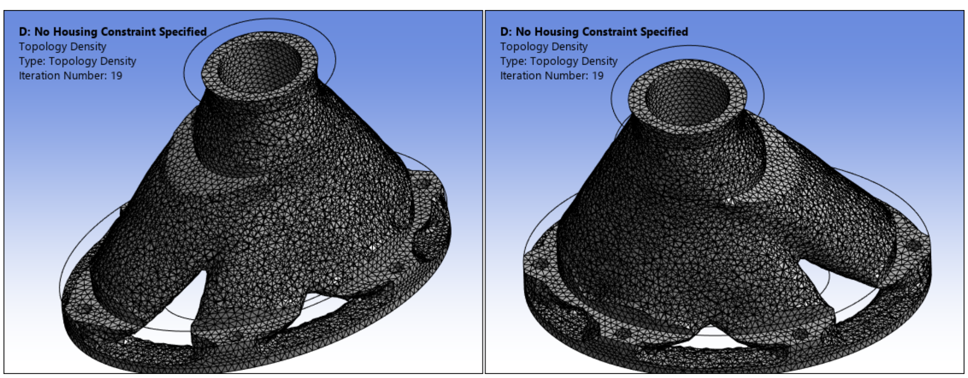
Set Up and Mesh

The specified mesh and the setup of the environment, are illustrated below. The displacement is fixed at six locations (blue) and a horizontal load is applied on the inner surface (red). The finite element model includes 242, 979 elements.



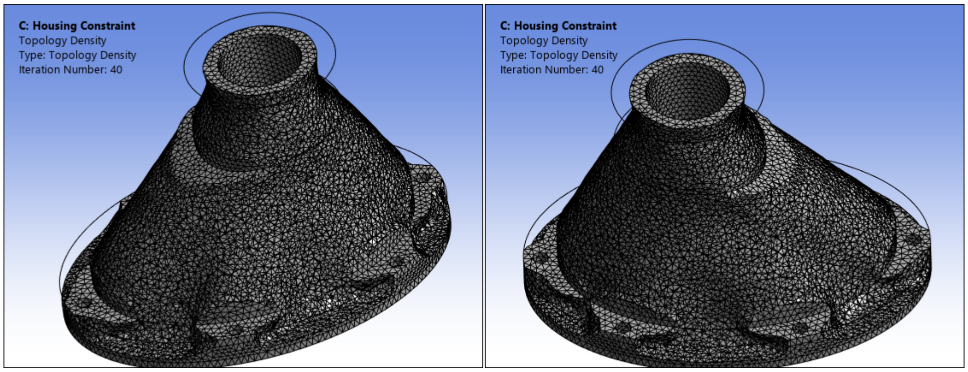
No Housing Constraint Specified

No **Housing** constraint is specified for the following result. The existing perforations violate the housing specification and the structural compliance equals 3.22359e-04.



Housing Constraint Specified

When you add a **Housing** constraint, the optimization problem results in the optimized shape shown below. As expected, the surfaces are encapsulated by the structural design and the structural compliance slightly increases to 3.25889e-04 (+1.1%).



4.1.7. Complexity Index

The **Complexity Index** Manufacturing Constraint enables you to minimize the creation of overly complex structures.

Supported Methods

The **Complexity Index** constraint is only available only for **Level Set Based** optimization method.

Industrial Motivation

This constraint helps you limit the geometric complexity of your designs which directly impacts the cost of production. Furthermore, this constraint type enables you to produce more organic, bio-mimetic designs, which facilitate aesthetically pleasing designs in development areas such as architecture and furniture manufacturing.

Setup and Technical Details

The setup for this constraint requires only the specification of a lower limit that cannot be violated, denoted by $CI_{limit} \geq 1.0$.

Note: Complexity index values are non-dimensional. In fact, the sphere is considered as a reference design where the Complexity Index is equal to 1.0 - the lowest supported value.

Recommendations and Observations

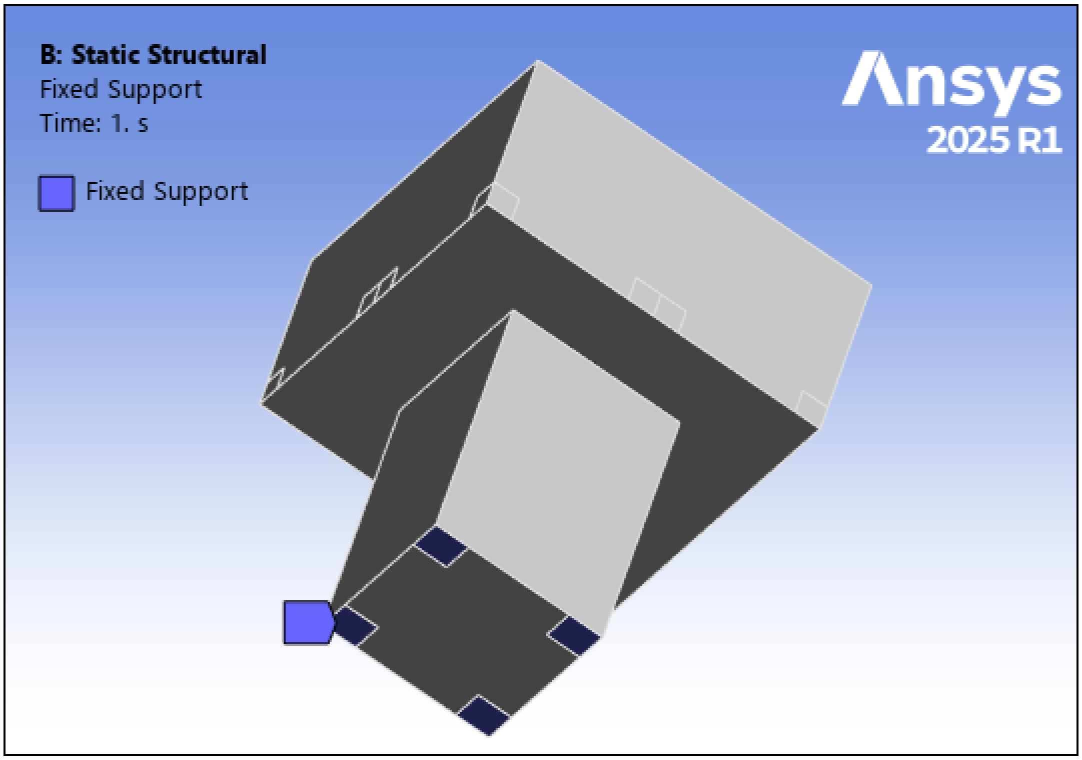
As with any constraint, the **Complexity Index** constraint affects the objective performance. Namely, by reducing the CI_{limit} the feasible domain shrinks and leads to less complex designs with a likely smaller objective gain.

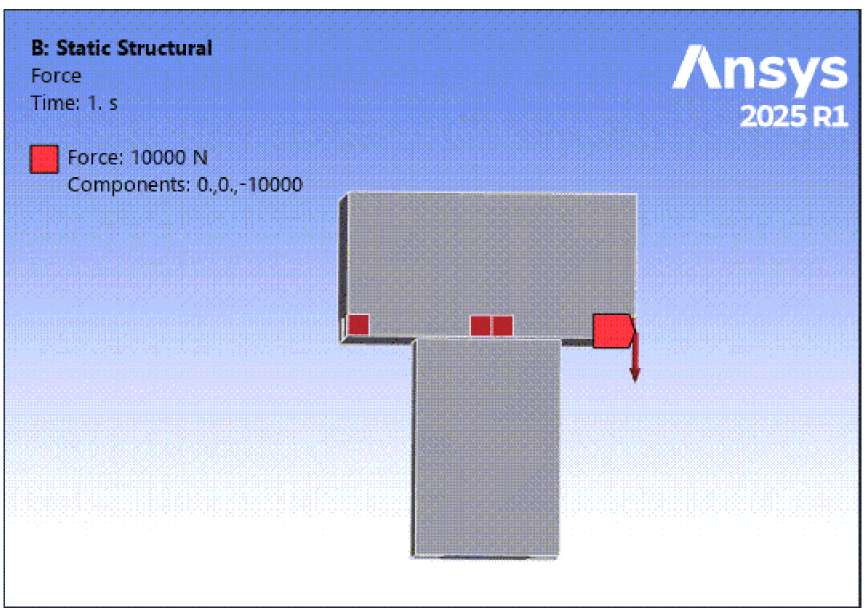
Electrical Mast Example

This analysis optimizes a simplified electrical mast. The optimization problem consists in minimizing the total volume under a compliance constraint of 0.13 Joule. A Cyclic Repetition Design Constraint in the Z-direction with 4 sectors is also enforced.

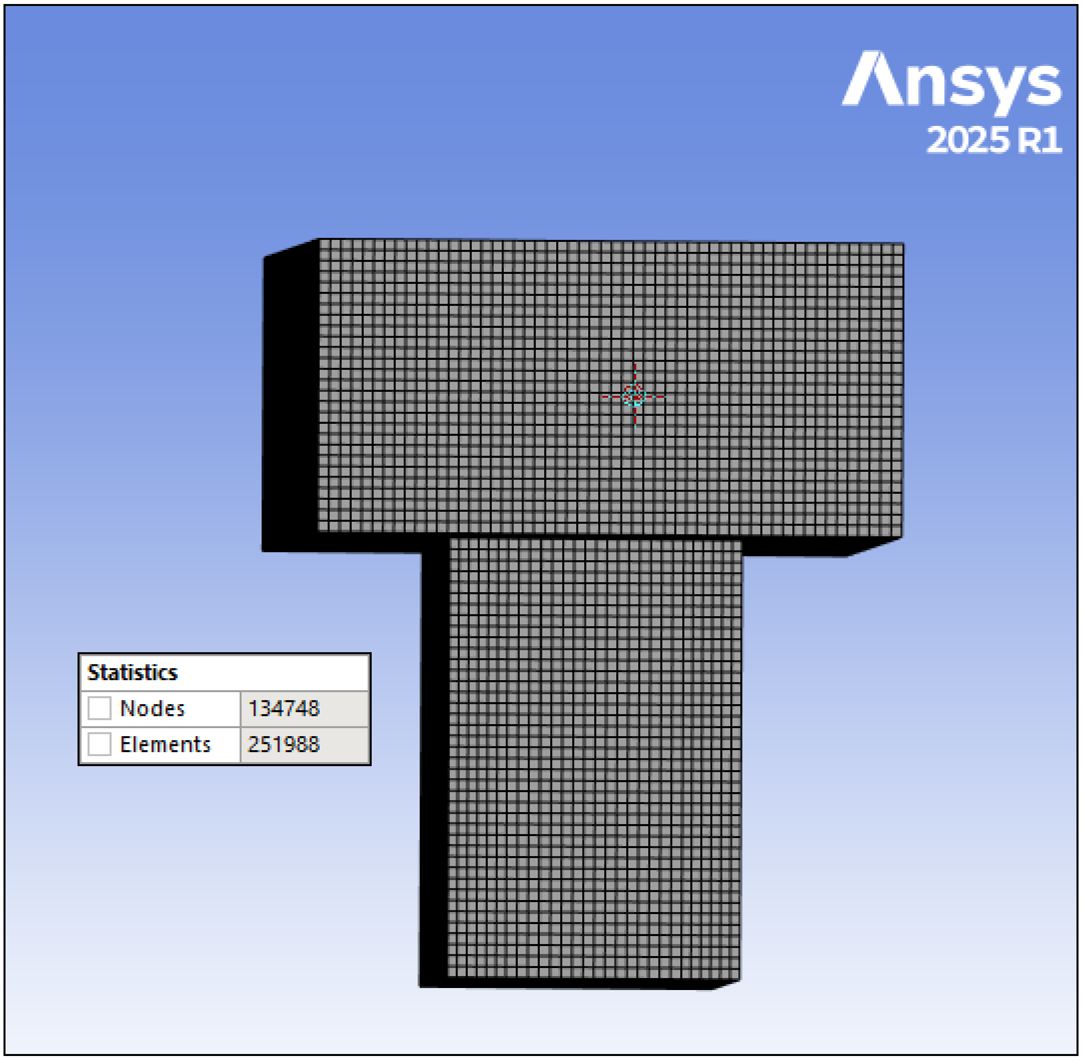
Set Up and Mesh

The specified mesh and the setup of the environment are illustrated below. Fixed supports are applied on the four faces highlighted in blue and force loads are applied on the upper portion of the model (in red). This represents forces exercised by electrical cables.

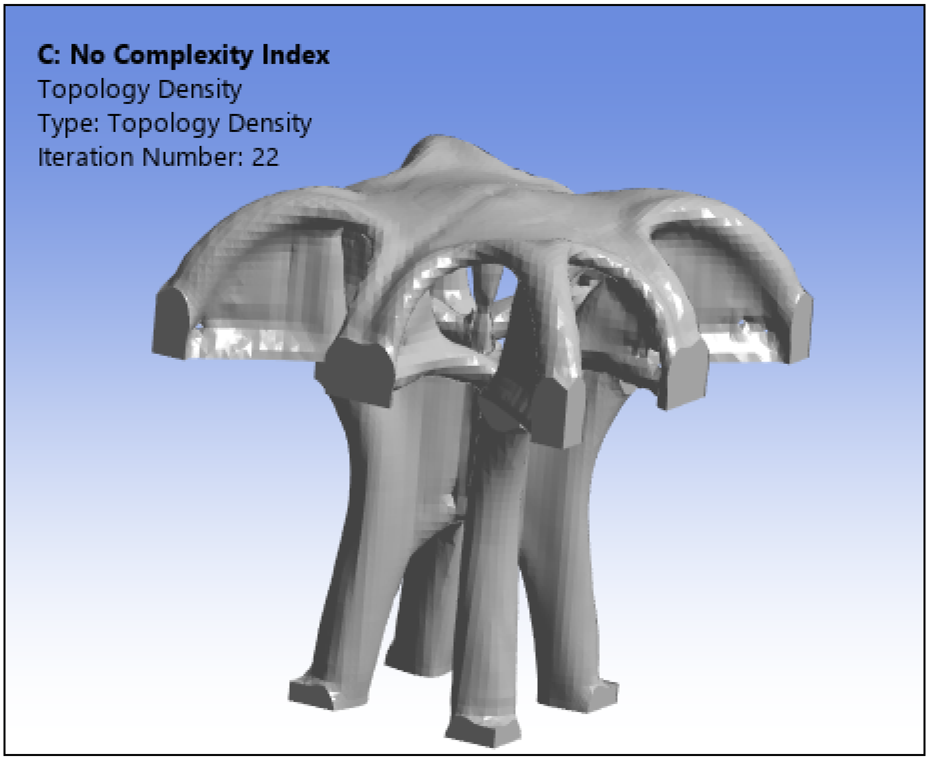


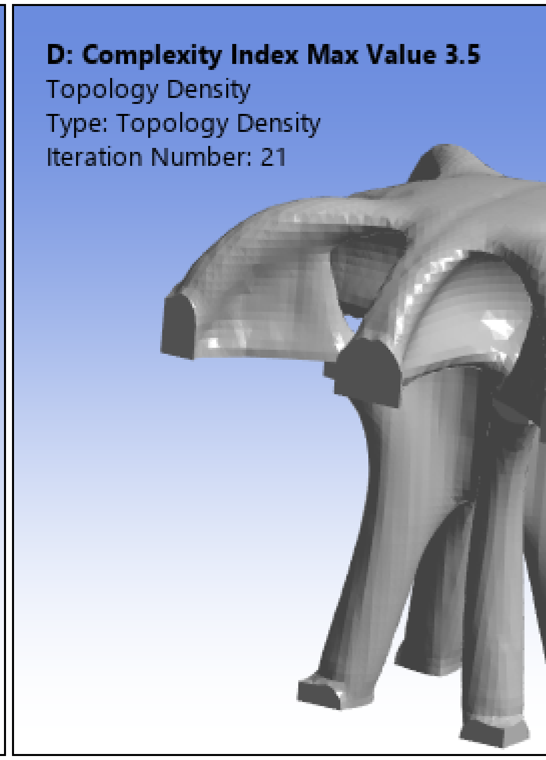


The finite element model, illustrated below, is comprised of 251.988 elements.

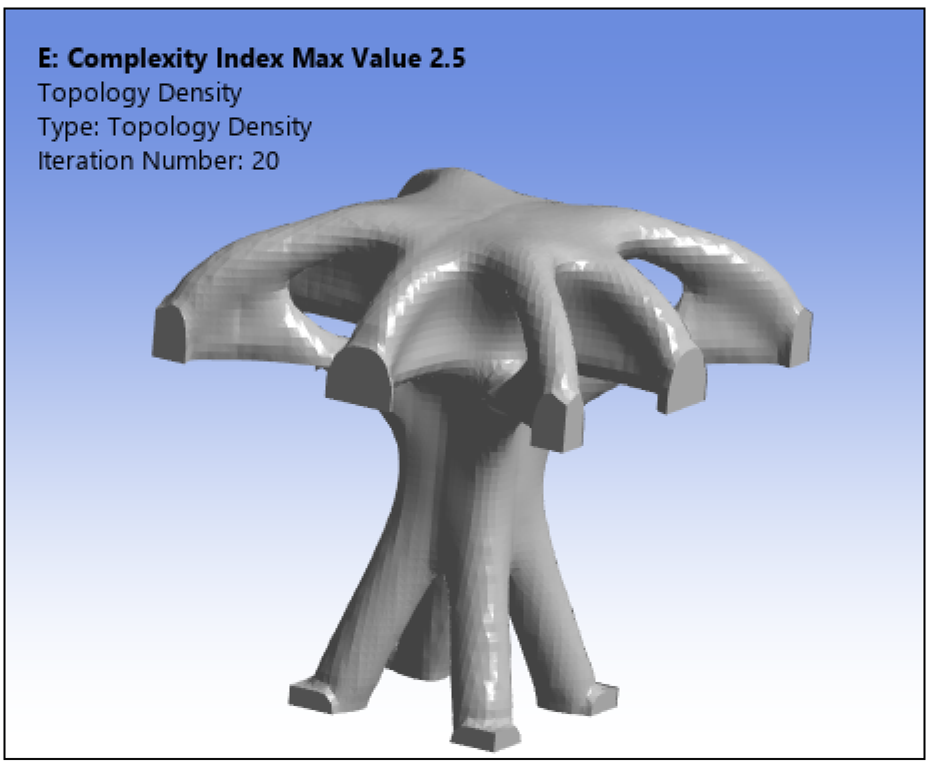


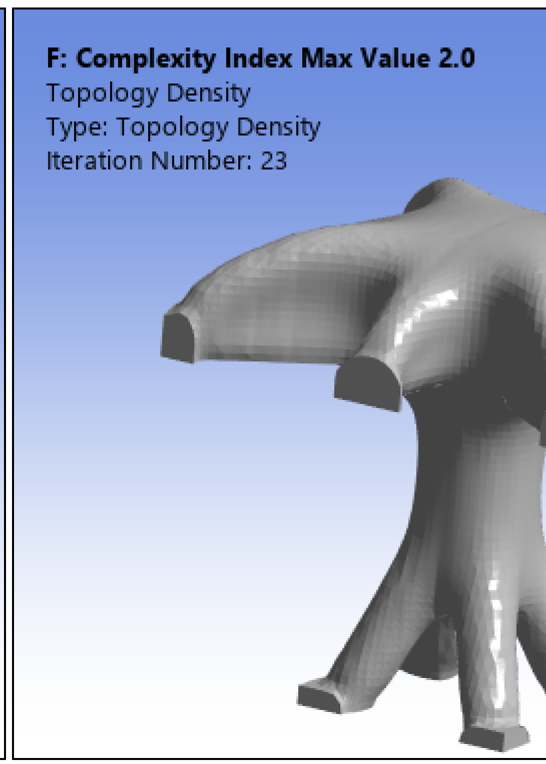
Here are the results when no Complexity Index is specified and when an index of 3.5 is specified.





Here, Complexity Index values of 2.5 and 2.0 are specified. As expected, the shape gets simplified and the total volume increases by +17%, +42,3%, and +63% correspondingly.





Chapter 5: Bibliography

[1] Bendsøe, M. P., & Kikuchi, N. (1988). Generating optimal topologies in structural design using a homogenization method. Computer Methods in Applied Mechanics and Engineering. 71(2), 197–224.

[2] Bendsøe, M. P., & Sigmund, O. (2003). Topology Optimization: Theory, Methods, and Applications. Berlin, DE: Springer.

[3] Murat, F., & Tartar, L. (1985). Calcul des Variations et Homogénéisation. D. Bergman & al. (Eds.), Les Méthodes de l'Homogénéisation: Théorie et Applications en Physique (pp. 319-369). Coll. Dir. Etudes et Recherches EDF. 57. Paris, FR: Eyrolles.

[4] Dapogny, C., & Frey, P. (2012). Computation of the signed distance function to a discrete contour on adapted triangulation. Calcolo. 49, 193–219

[5] Allaire, G., Jouve, F., & Toader. A.-M. (2004). Structural optimization using sensitivity analysis and a level-set method. Journal of Computational Physics. 194, 363–393

[6] Bui, C., Dapogny, C., & Frey, P. (2012). An accurate anisotropic adaptation method for solving the level set advection equation. International Journal for Numerical Methods in Fluids.

[7] Osher, S., & Sethian, J. A. (1988). Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. Journal of Computational Physics. 79(1), 12–49.

[8] Allaire, G., Dapogny, C., & Jouve, F. (2021). Shape and Topology Optimization. A. Bonito & R. Nochetto (Eds.), Geometric Partial Differential Equations, Part II (pp. 1–132). Handbook of Numerical Analysis. 22. Elsevier.

[9] Allaire, G., & Schoenauer, M. (2007). Conception optimale de structures. Berlin, DE: Springer.

[10] Shimoda, M., & Liu, Y. (2014). A non-parametric free-form optimization method for shell structures. Structural and Multidisciplinary Optimization. 50, 409–423.

[11] Cheng, L., Zhang, P., Biyikli, E., Bai, J., Robbins, J., & To, A. (2017). Efficient design optimization of variable-density cellular structures for additive manufacturing: theory and experimental validation. Rapid Prototyping Journal. 23(4), 660–677.

[12] Allaire, G. (1992). Homogenization and two-scale convergence. SIAM Journal on Mathematical Analysis. 23(6), 1482–1518.

[13] Allaire, G., Jouve, F., & Michailidis, G. (2016). Thickness control in structural optimization via a level set method. Structural and Multidisciplinary Optimization. 53, 1349–1382.

[14] [Fernández, E., Yang, K. K., Koppen, S., Alarcón, P., Bauduin, S., & Duysinx, P. (2020). Imposing minimum and maximum member size, minimum cavity size, and minimum separation distance between solid members in topology optimization. Computer Methods in Applied Mechanics and Engineering. 368, Article 113157.

[15] G. Allaire, C. Dapogny, R. Estevez, A. Faure, G. Michailidis, Structural optimization under overhang constraints imposed by additive manufacturing technologies, Journal of Computational Physics, Volume 351, 2017, Pages 295-328.

[16] Matthijs Langelaar, Topology optimization of 3D self-supporting structures for additive manufacturing, Additive Manufacturing, Volume 12, Part A, 2016, Pages 60-70.