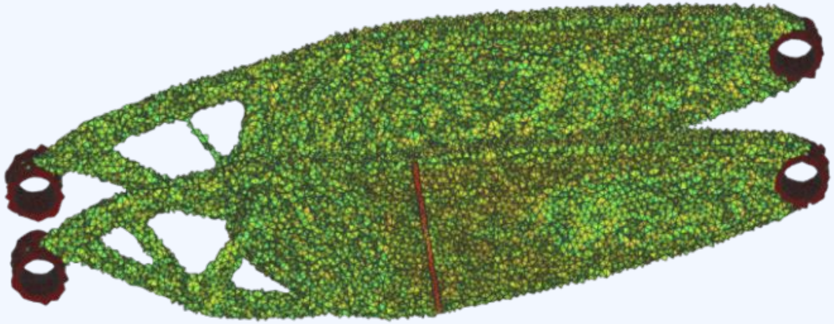


# nTopology

## Topology Optimization in nTop-
## A User Guide

# Table of Contents

# Topology Optimization in nTop

Once you have completed a structural simulation of your part, nTop offers the ability to conduct a topology optimization using the Solid Isotropic Material with Penalization (SIMP) method, if your goal is to lightweight / use less material. This document will explore the main functionalities of nTop's topology optimization capabilities.

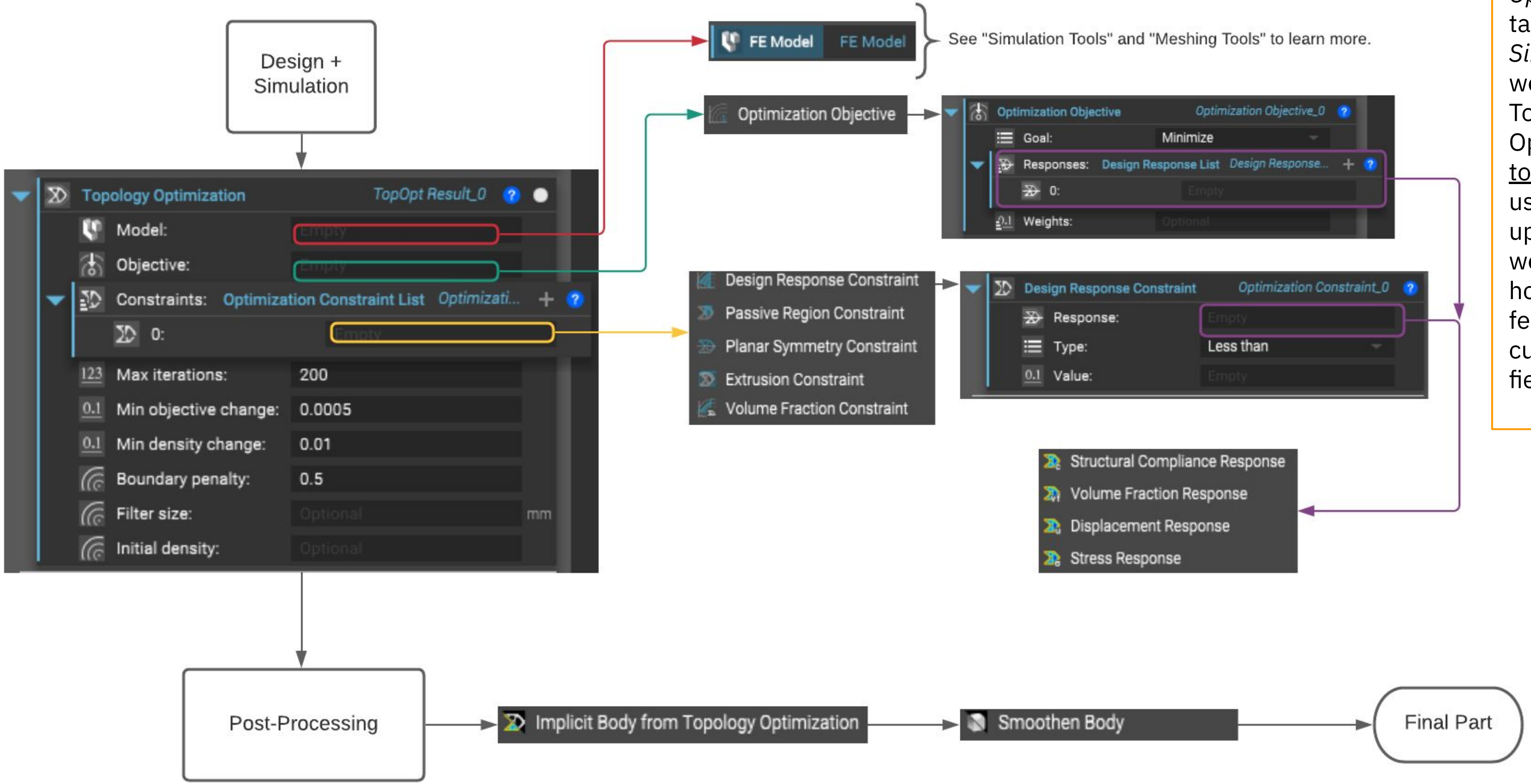
This includes a descriptive workflow of the function blocks used in TopOpt, as well as tips related to a step-by-step example.

*Why would I run topology optimization in nTop?*

- Consolidated workflow - edit your part in the same UI as your simulation and topology optimization results
- Easy post-processing (conversion, smoothen, shell, infill) of topology optimization result into suitable CAD component or manufacturable part.
- Automate your topology optimization by changing design parameters at any step in the process.
- Swift output of results including CAD file format

# Topology Optimization Block and Workflow

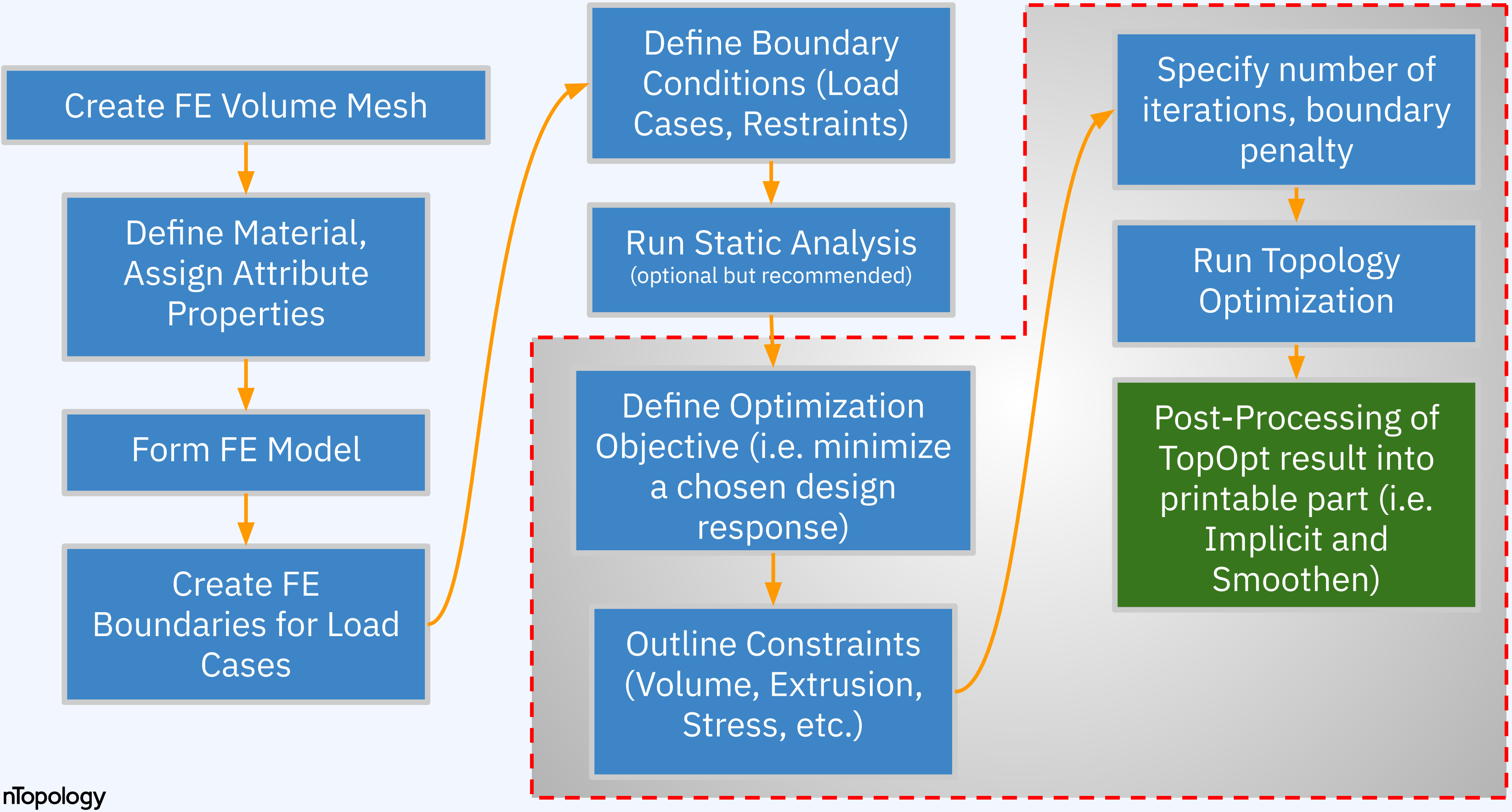
# Topology Optimization Blocks




*All blocks are found in the *Optimization* tab of the *Simulation* workflow. The Topology Optimization toolkit may be used to speed up certain workflows, however with fewer customizable fields.

# General Topology Optimization Workflow

Main Functions Explored in Slide Deck



# Getting Started with Topology Optimization - The Basics *



nTopology

*Check out the Advanced Topics section to learn more about each block.

# Setup

Setup your topology optimization in nTop using one user-friendly block.

| Block | Function | Notes |
| --- | --- | --- |
| **Topology Optimization**<br><br>Advanced Topics | Conduct topology optimization on an FE model. | TopOpt can only work with isotropic materials.<br><br>Must define:<br>● At least one response to meet (e.g. structural compliance based on load cases)<br>● At least one constraint (e.g. maximum volume of final part). |

nTopology

# Objectives

Outline the main goal of your topology optimization. What do you want to achieve with this tool? What response do you wish to maximize or minimize? Volume, stress, stiffness, etc?

| Block | Function | Notes |
|---|---|---|
| **Optimization Objective**<br><br>Advanced Topics | Decide whether to maximize / minimize a design response in your topology optimization. | Most optimization objectives aim to minimize the structural compliance (maximize the stiffness) of the part.<br><br>You may choose a different response (e.g. stress, displacement), however, keep in mind this solution will take longer to converge, since a static structural analysis is conducted at each iteration (assigning new density and sensitivity values for each element). |

# Design Responses

When running the topology optimization on your FE model, you must specify the objective to meet, and a constraint to your solution. The following responses are used to define both criteria.

More Commonly Used ↑

| Block | Function | Notes |
|---|---|---|
| **Structural Compliance Response** | List a set of boundary conditions, as part of a static analysis. | Compliance is the inverse of stiffness. Minimizing compliance, as an optimization objective, will maximize model stiffness. |
| **Stress Response** Advanced Topics | List a set of boundary conditions, as part of a static analysis. | Evaluates the maximum centroidal von Mises stress in an FE boundary, at every step of your topology optimization. |
| **Displacement Response** Advanced Topics | List a set of boundary conditions, as part of a static analysis. | Evaluates the maximum displacement / deformation of a node within a mesh of an FE boundary, at every step of your topology optimization. |
| **Natural Frequency Response (beta)** | Optionally list a set of restraint boundary conditions, as part of a natural frequency analysis. | Evaluates the a specified number of modes and attempts to maximize or minimize them. |
| **Volume Fraction Response** | Based on volume fraction of optimized geometry divided by design space volume. | Rarely used as an optimization objective. Instead, consider using the Volume Fraction Constraint block if your desire is the reduce material below a certain value. |

# Constraints - Part 1

In reality, to meet the objective of your topology optimization, you will certainly have some constraints. Perhaps you wish to limit the volume of material to less than 30% of the original part, or limit the stresses on your part by the Young's modulus of your material.

| Block | Function | Notes |
|---|---|---|
| **Design Response Constraint**<br><br>Advanced Topics | Choose a design response that must be restricted / constrained in the topology optimization.<br>- Stress<br>- Displacement<br>- Natural Frequency<br>- Volume Fraction | You may choose any of the design responses to constrain, ensuring the solution meets and falls under or exceeds your quantitative restriction (dependent on your choice).<br><br>For example:<br>● limiting the final volume of your part to 30% or less (with an input of 0.3),<br>● limiting the stress in a particular FE region to 30 MPa or less,<br>● limiting the displacement of a node in a particular FE region to 2 mm or less,<br>● limiting the first 3 natural frequencies to remain outside of the [80,100]Hz range |

# Constraints - Part 2

More Commonly Used

| Block | Function | Notes |
|---|---|---|
| **Volume Fraction Constraint** | Choose a volume fraction of your design space. | This block will constrain your topology optimized part to less than or equal to your specified volume fraction value (e.g. 0.3 or 30%). |
| **Planar Symmetry Constraint** Advanced Topics | Specify a symmetry plane for the topology optimization. | The model should be inherently symmetric for this to work. Planes should be orthogonal to the model and placed through the centroid of the FE model. |
| **Passive Region Constraint** Advanced Topics | FE region to you wish to remain unaltered through topology optimization process. | Specify any FE region. By default, the FE regions for the boundary conditions are set to be passive regions. |
| **Overhang Constraint** Advanced Topics | Define an overhang constraint to be enforced in the optimization result | For more information: https://support.ntopology.com/hc/en-us/articles/360060738793-How-to-use-the-overhang-constraint-in-Topology-Optimization |
| **Pattern Repetition Constraint** | Constraint the topology optimization process to maintain a repeated pattern defined by a Cell Map | For more information: https://support.ntopology.com/hc/en-us/articles/4402937450003--nTopology-3-5-What-s-New#Pattern%20Constraint%20for%20Topology%20Optimization |
| **Extrusion Constraint** | Define a geometric extrusion constraint to be enforced in the optimization result. | To accurately use this constraint, the user must create a curve (orthogonal to the full design space - starting at/before and ending at/after) through which all tet elements are fixed to a high density threshold. |

nTopolog

# Post-Processing

The raw results of a topology optimization are not practical nor usable, so post-processing will be required to get a component ready for DfAM or further implicit modeling.

Use Blocks In Order

| Block | Function | Notes |
|---|---|---|
| **Implicit Body from Topology Optimization**<br><br>Advanced Topics | Create an implicit body from the TopOpt solution. | Since most operations in nTop work with implicits, if you wish to apply any additional design changes to your final part (e.g. boolean operations), it is imperative convert the result to an implicit body. |
| **Smoothen Body**<br><br>Advanced Topics | Smoothen your implicit body using a Gaussian image kernel. | Choose a grid size for the body, which is blurred using the convolution method.<br><br>Tip: Start with 2x greater than the element feature size, then iterate down slightly until you output a more optimal part.<br><br>Choose the number of times you would like to smoothen the body. Start with 2 iterations (running the spatial filter twice), then change as per the output. |

# Topology Optimization Common Questions

Pg 15

# FAQs

- **Can I apply multiple forces?**
  - Yes! Add more forces to your boundary condition list, which is appended to your design response (in the case of a stress, displacement). You may choose from a Point Force, an Edge Force, etc (as explored in the *Simulation* workflow slides).

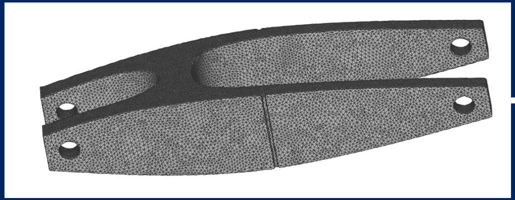
- **Can I have multiple objectives?**
  - Yes! If you wish to optimize more than one objective, you may add more than one design response (however, all design responses must be minimized or maximized, cannot do both). You may assign weights to each objective with a scalar list of weights (see Advanced Topics in the *Objectives* slide).

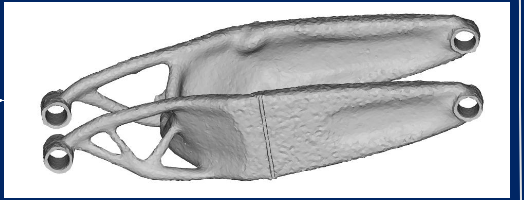
- **Can I have multiple solutions? Can I run a single TopOpt for different loading conditions?**
  - You may have multiple topology optimization solutions if you duplicate the block, and apply different optimization objectives / constraints to each solution. Then, you can compare each solution and potentially run a static structural analysis on each solution. You may also have different design responses for various loading conditions, which can be incorporated into your optimization objective as well (e.g. forces imposed on a car frame during a head-on collision and during a fender-bender). With this, you can run one single topology optimization, to meet all your objectives.

- **Can I apply manufacturing constraints?**
  - Yes, create an optimization constraint to restrain the solution based on your manufacturing constraints. For example, a restriction on the amount of material you can work with is a valid constraint that can be denoted as a volume fraction constraint.

- **What about thermal optimization?**
  - nTop does not currently offer thermal optimization, but the dev. team is certainly aware of the demand for this tool.

# Tips and Basic Examples of Topology Optimization Workflows







*It is encouraged to follow through with the next few slides with the example .ntop file provided.*

# Starting Off with a Topology Optimization

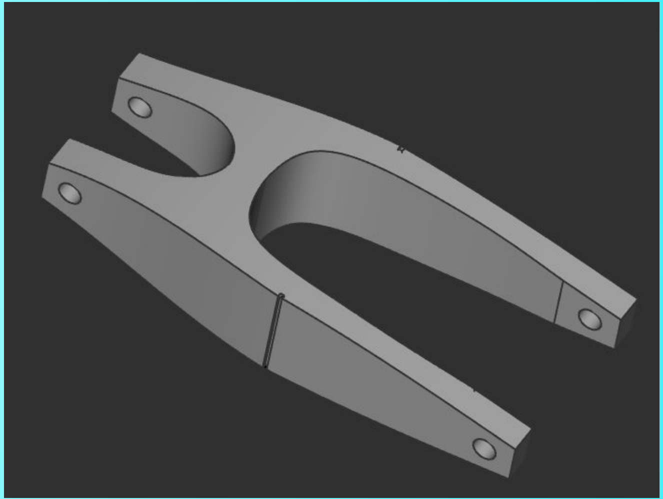
Follow this checklist to see whether you are prepared to run a topology optimization on your part:

- ❑ An FE Volume Mesh of my part was created.
- ❑ Material properties were assigned to the FE Volume Mesh to form an FE Model.
- ❑ Selected all the necessary FE boundaries / regions to localize all my boundary conditions. Created any tie constraints and FE connectors, if necessary, and assigned these to the model.
- ❑ My forces, restraints and other boundary conditions were grouped into one or multiple boundary condition lists (ideally set as variable(s) for easy reference in each design response). These boundary conditions were appropriately assigned to the chosen FE boundaries / regions.
- ❑ A Static Structural analysis was run, such that I could validate the initial performance of my part under the tabulated loading conditions. The results from the analysis seem accurate to continue (i.e. stresses not surpassing the elastic modulus of your material, etc.).
- ❑ Double-checked all the results by outputting a Displacement Point Map and a Von Mises Stress Point Map. Save these in a block, as they could be useful later on (for the comparison of the analysis results from the TopOpt). You may now proceed onto a topology optimization.

# Useful Tips to Remember when Working in nTop

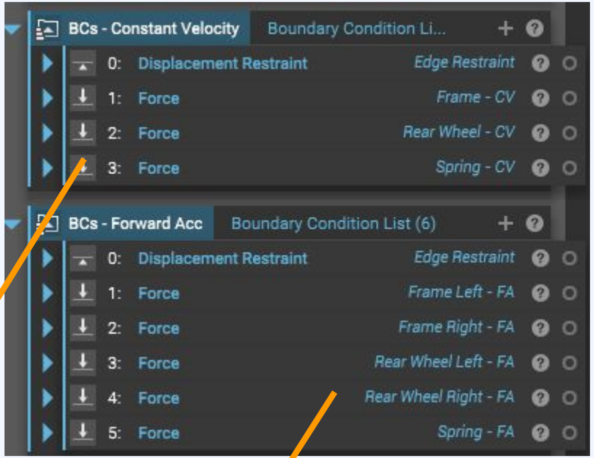
- Double-click on an input field to list all compatible input blocks
- Ensure units are appropriately appended to all values (e.g. Pa for stresses, mm for lengths, etc.)
- Start using variables when your blocks are stacked with 3+ levels
- Use descriptive variable names for easy referencing
- Use comments to section parts of your workflow (e.g. section for simulation blocks, section for topology optimization blocks) - better organization of your notebook

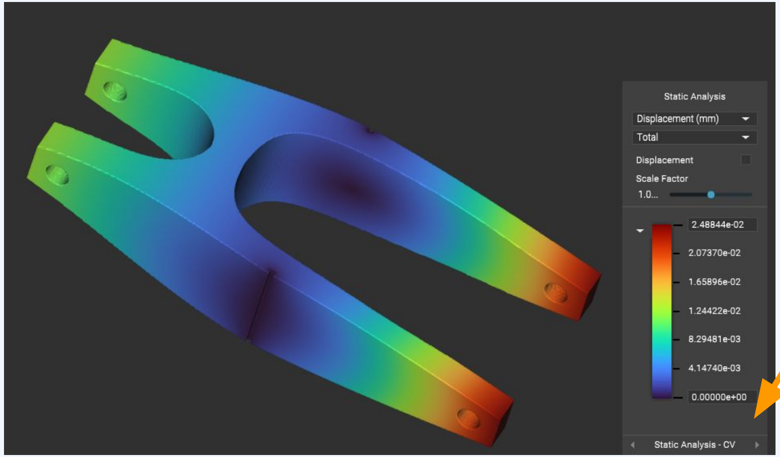
# Let’s Dive Into An Example Now …

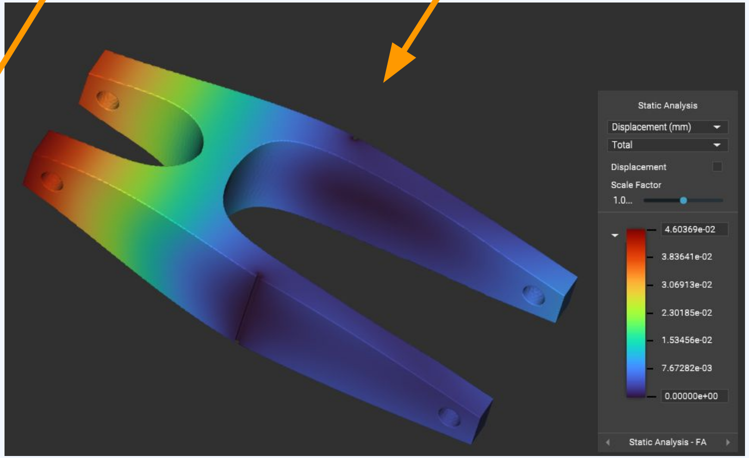


# Swingarm Workflow - Load Cases

- Created two sets of boundary conditions (BCs): when the motorcycle is operating at constant velocity, and when there is forward acceleration
- We can use both sets of BCs for one topology optimization (shown in further slides)
  - The displacement restraint for both BCs MUST be the SAME - one TopOpt cannot have differing restraints between BCs
- Ran a static analysis for each boundary condition, and extracted the displacement and stress point maps



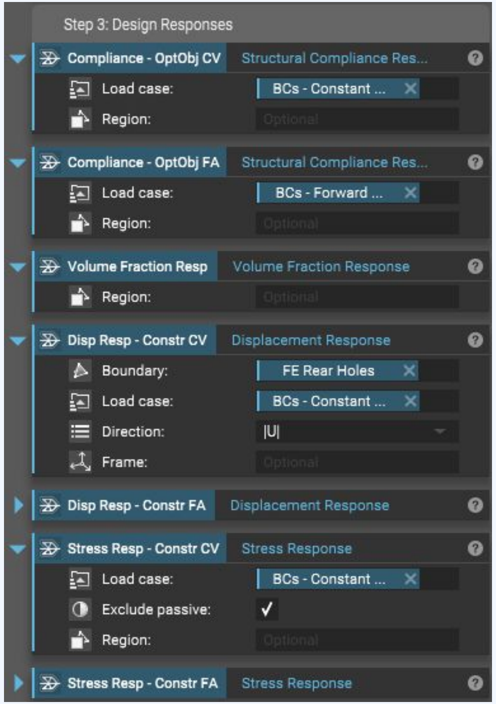



# Swingarm Workflow - Design Responses

Here are some tips to apply when initializing your design responses:

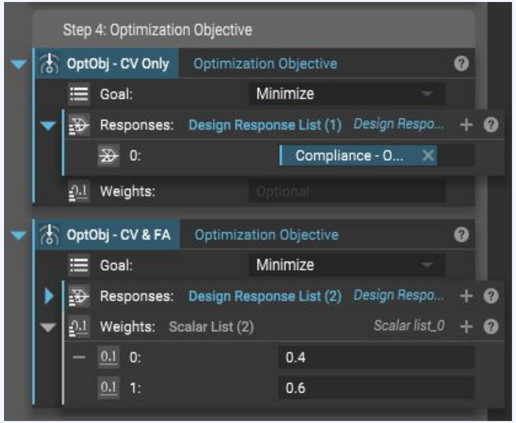
- **Structural Compliance Response**
  - This is the most commonly used application in a TopOpt workflow. To see if your TopOpt is rendering potentially accurate results, use this block in conjunction with the “Volume Fraction Constraint” block, as a test.
- **Stress Response / Displacement Response**
  - Select a loaded region or a passive region (for the FE Boundary) to achieve the best performance
  - Created individual responses for each boundary condition set (forward acceleration and constant velocity)
- **Adding Design Responses**
  - It is possible to include multiple design responses, if your design warrants this
  - This will be done in the Optimization Objective Block (see next slide)
  - Possible to have different load cases for different design responses, as to observe a different response to various boundary conditions (program will run a static analysis for each unique boundary condition set)



# Swingarm Workflow - Optimization Objective

Here are some tips to apply when outlining your optimization objectives:

- Your topology optimization can exclusively either minimize or maximize a design response (or a multitude of design responses based on optional weighting)
    - i.e. You cannot maximize one design response and minimize another design response within the same topology optimization
- Weightings for your design responses (from 0 to 1, with the sum of the weightings equalling to 1) can be used under any of the following conditions:
    - One design response is less important in minimizing / maximizing versus another design response
    - I have multiple design responses which I am attempting to optimize (e.g. displacement and stress)
    - I wish to assign a weighting to different load cases (depending upon their prevalence during the part's operations), and I wish to evaluate the same design response with different load cases (e.g. two structural compliance responses for constant velocity and forward acceleration load cases in the swingarm example)



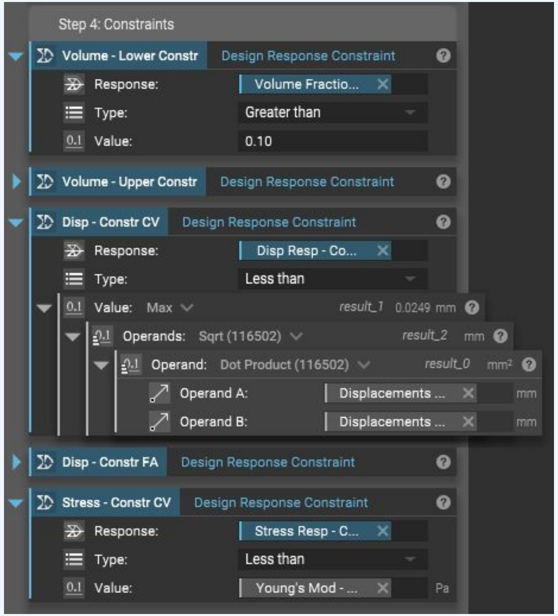
# Swingarm Workflow - Design Response Constraints

Here are some tips to apply when constraining your TopOpt solution:

- **Volume Fraction Constraint**
  - Specify any value between 0 and 1.
  - Can have an upper limit and lower limit to your volume fraction by choosing the appropriate "Type" in the dropdown input field

- **Stress Constraint**
  - Conduct a static analysis of your FE Model using the same BCs as your TopOpt, and extract the maximum von Mises stress using the "Von Mises Stress Point Map", to constrain your maximum stress values
  - Can also constrain stress values by the Young's Modulus of your material

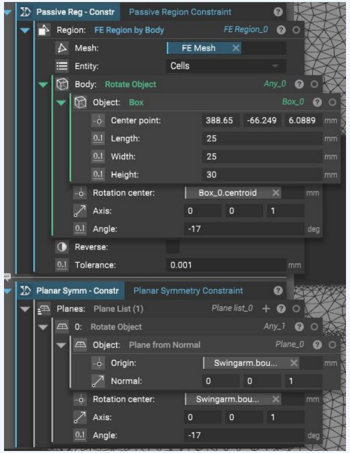
- **Displacement Constraint**
  - Same as the stress constraint, can select the maximum displacement from the initial static analysis by taking the square root of the dot product of the displacement point map vectors, or input your desired displacement



# Swingarm Workflow - Constraints

Here are some tips to apply when constraining your TopOpt solution:

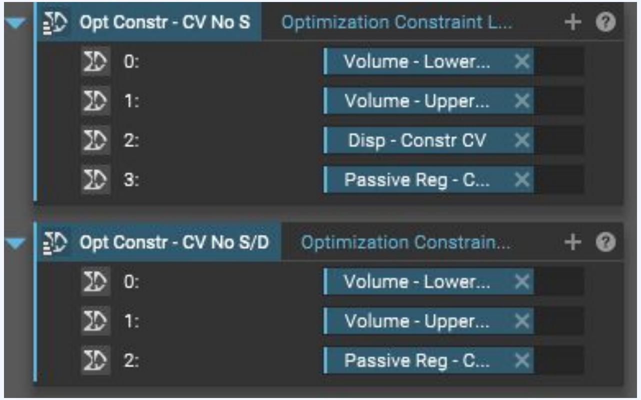
- **Planar Symmetry Constraint**
  - Ensure the FE mesh is near symmetrical, and place the plane at the centroid of the FE model (the model should be initially symmetric as well).
  - If the FE mesh is not symmetric, ensure you ‘Remesh Surface’ before creating a volume mesh of your model.
  - Easy workaround: split part in half, FE mesh the half, mirror the FE mesh, then merge the two FE meshes together
- **Extrusion Constraint**
  - Ensure the curve is normal to the design, and passes through the entirety of the design space (extend the curve by adjusting the end points as necessary)
  - Understand the use case of your part: the extrusion constraint cannot be applied to the swingarm, since it is not easily extrudable
    - This constraint is most often applied to molds (parts without perforations, etc.) and three-axis CNC machined parts
- **Passive Region Constraint**
  - Use a FE Region by Body block to select the regions you wish to keep constant during your topology optimization (flood fills nearby cells based on the ‘Body’ input)



# Swingarm Workflow - Optimization Constraints

Here are some tips to apply when constraining your TopOpt solution:

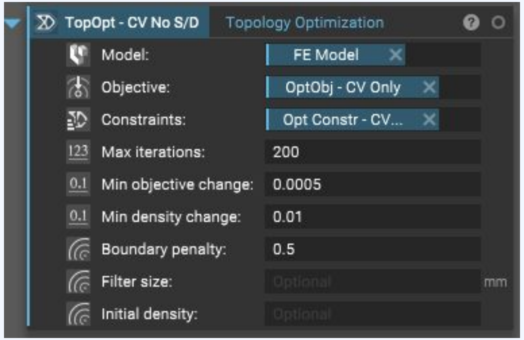
- **Optimization Constraint List**
  - Create a list of all the optimization constraints you wish to apply to your topology optimization, and set this aside as a variable
  - You may wish to apply different constraints depending upon your application (i.e. a topology optimization with no stress constraint, versus a TopOpt with a displacement and a stress constraint) and creating multiple lists will help you organize your 'DoE'
  - This list input, with multiple design constraints, can be dragged into the 'Optimization Constraint' field in the Topology Optimization block



# Swingarm Workflow - Topology Optimization

Here are some tips to apply when running the topology optimization:

- **Model, Objective, Constraints**
  - All these fields can be easily populated based on knowledge from the previous slides (FE Model, Optimization Objectives and Optimization Constraints)
- **Max iterations, Min objective change, Min density change**
  - Changing the latter two parameters will either decrease (increasing min change) or increase the time (decrease min change) it takes for your TopOpt solution to converge
  - The max iterations is the bottleneck - if the solution does not converge prior to the noted iteration, the solution may not be the most accurate optimization (as it likely did not meet the min objective, density change requirements)
  - Density change is evaluated based on the tet element density within the part through each TopOpt



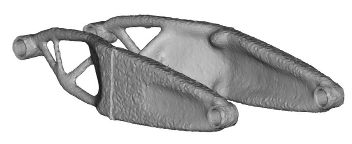

Check out this Advanced Topics section for more...

# Swingarm Workflow - Topology Optimization

Here are some tips to apply when running the topology optimization:

- **Boundary Penalty**
    - Use the boundary penalty as your use case deems appropriate (e.g. a TopOpt of a mold may require the boundary penalty to be 0, if the mold must fill the boundaries of the design space) - see advanced topics

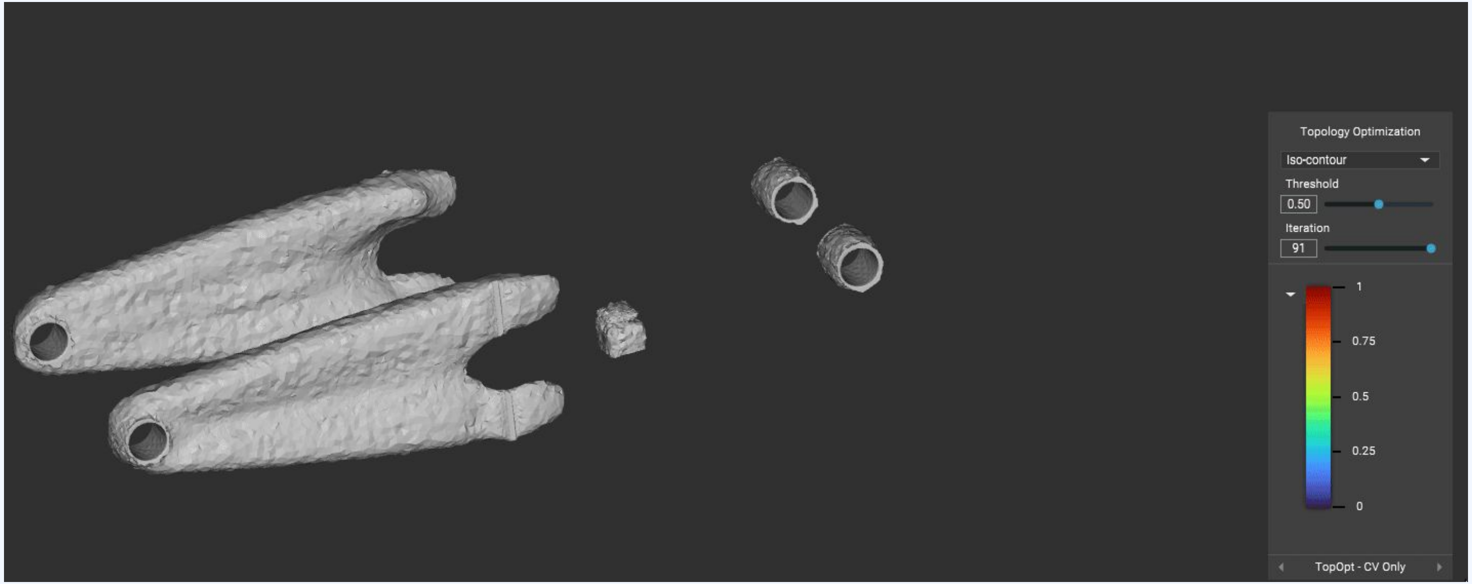
- **Filter Size**
    - Controls the minimum feature size which the topology optimization evaluates through every iteration. The smaller the filter size, the longer the solution will take to process, since more elements are being evaluated at every step.



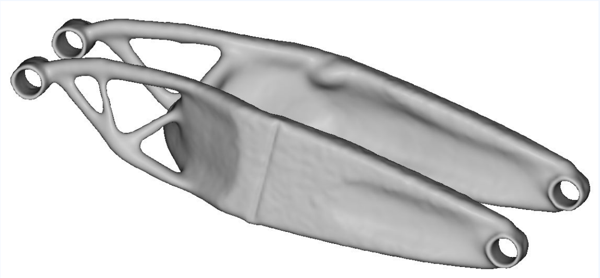
- **Initial Density**
    - Optionally, can add an initial density (for the tet elements) for which the desired volume constraint can be satisfied
    - This parameter does not affect your volume constraint - you can adjust the densities of your tet elements during post-processing to achieve a usable part

# Swingarm Workflow - Topology Optimization

Check out how you can change the density threshold for your tet elements in your topology optimization solution, and how to use the sidebar:



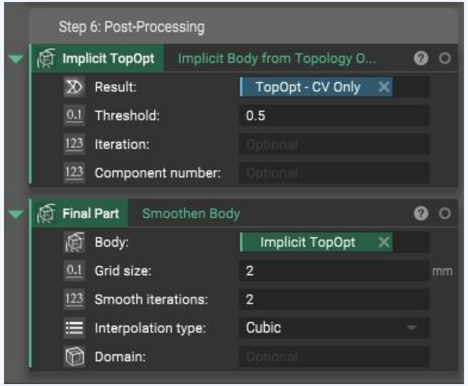
# Swingarm Workflow - Post-Processing

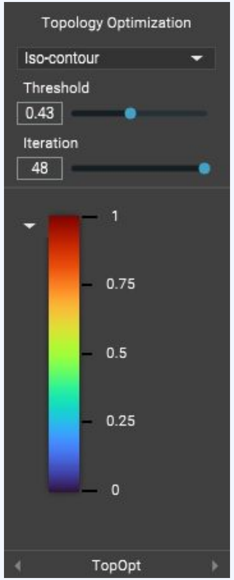


Here are some tips to apply when post-processing your topology optimization results:

- **Implicit Body from Topology Optimization**
  - Choose the tet element density threshold which you found to represent a usable part, from your TopOpt solution (i.e. if tet elements above a certain threshold do not create a connected model, you may lower this density value)
  - Can specify an iteration of the TopOpt solution to choose a desired model to convert to an implicit body
  - Can specify a component number from the density field to assign a density threshold for the implicit body, as well

- **Smoothen Body**
  - Choose a Grid Size twice the element feature size of the implicit body, and iterate down until a desirable model is achieved (may hit a 'Resolution Cap' if the grid size is too small)
  - Follow a 'Cubic' interpolation type for more optimal results

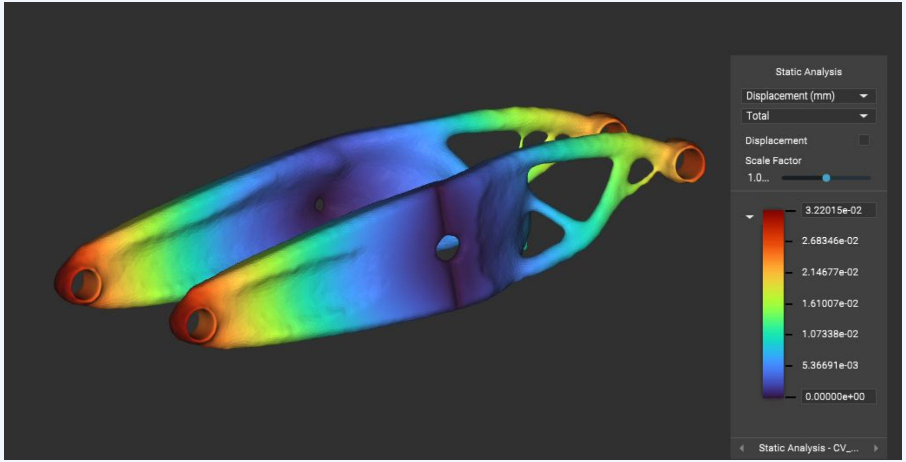


# Swingarm Workflow - Post-Processing

Here are some tips to apply when post-processing your topology optimization results:

- **Conduct a post-static structural analysis on your part with the original load cases!**
    - This will help you identify any issues with your TopOpt design decisions (e.g. the chosen element density threshold, if your part is a connected model when creating your FE mesh, etc.)




**You have completed a topology optimization!**

# Advanced Topics - Topology Optimization

# Setup

**Block** | **Advanced Topics**

**Topology Optimization**

Back to Basics

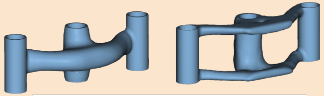
How can I control the convergence of my solution?

- Editing the maximum number of iterations - reducing this value will speed up the output of the result, however, it may not yield an accurate solution (outputting a result before the solution converges);
- Altering the minimum objective change (with relation to the optimization objective design response) - increasing this value will speed up the optimization;
- Choosing a minimum element density change (based on present tet elements) between consecutive optimization iterations - increasing this value will speed up the optimization.

What can I do with the 'Optional' fields?

- Specify a filter size for the final TopOpt mesh (likely excluding tet elements smaller than the stated feature size);
- Select an initial density field (as a scalar value) when viewing tet elements which meet your volume constraint criterion.

What is a "Boundary Penalty"?




Previously known as "Filter Boundary", the TopOpt solution can either:

- Fully restrict material (i.e. tet elements) to the boundaries of the design space (e.g. where no loads / constraints are applied) ← NEUMANN METHOD (assigned a value of 0)
  ... OR ...
- the solution could intuitively follow the load paths created by the boundary conditions, thus not penalizing the boundaries of the design space with material ← DIRICHLET METHOD (value of 1).

A mix of both methods can be implemented with a value ranging between 0 and 1.

nTopology

# Objectives

| Block | Advanced Topics |
| --- | --- |
| **Optimization Objective**<br><br>Back to Basics | <u>Under what conditions would I maximize a design response?</u><br><br>• When your experiment aims to identify the limits at which your part will operate under your designated boundary conditions<br>• Force inverters<br>• Applying an axial force in one direction to maximize the displacement / stress of a node along a different axis<br>• Maximizing the moment of inertia and eigenfrequencies (modal analysis) of a part are design responses in the development pipeline |
| | <u>What do I do if I have multiple design responses?</u><br><br>You may assess a *scalar weighting* to each design response, dependent upon how "important" it is for your part to meet one objective over another!<br><br>Do this by creating a list of scalars of length equivalent to the number of design response objectives. Tip: Have all the weightings sum to 1, with values varying between 0 and 1. |

# Design Responses

| Block | Advanced Topics |
| --- | --- |
| **Stress Response**<br><br>Back to Basics | • Longer solution time → a full-body static structural simulation is conducted at each time step to determine the stresses imposed upon the part.<br>• Setting the loaded regions to be passive can ensure a more optimal solution. |
| **Displacement Response**<br><br>Back to Basics | • Longer solution time → a full-body static structural simulation is conducted at each time step to determine the individual node deformations.<br>• Setting the loaded regions to be passive can ensure a more optimal solution.<br>• You may also select a direction for which you would like to track the displacement (e.g. the total magnitude, or simply in the x-y axis, etc.). |

# Constraints - Part 1

| Block | Advanced Topics |
| --- | --- |
| **Design Response Constraint**<br><br>Back to Basics | Cases where you may want a solution which meets or exceeds your quantitative restriction are when you want to include an upper and lower bound to your constraint.<br><br>Since multiple optimization constraints may be included in your topology optimization, your volume limit may be 30% or less as an upper limit, and 10% or more as a lower limit (i.e. $0.1 \leq V \leq 0.3$). Same goes for the displacement and stress response constraints.<br><br>This tool ensures your optimization objective converges to an ideal solution. Rarely is structural compliance used as an optimization constraint, as it is more often chosen as the objective. |

# Constraints - Part 2

| Block | Advanced Topics |
|---|---|
| **Planar Symmetry Constraint**<br><br>Back to Basics | A *symmetric mesh* about the plane which is splitting the part offers a more optimal solution.<br><br>To create a symmetric mesh:<br>● Cut the design space in half<br>● FE mesh this half<br>● Mirror the FE mesh about your plane to create the second half of your design space<br>● Merge the FE meshes to tie both halves back together. |
| **Passive Region Constraint**<br><br>Back to Basics | During post-processing with the *Smoothen Body* block, regions which are designated to be passive are smoothened as well. |
| **Extrusion Constraint**<br><br>Back to Basics | To best use this block with a fully extrudable part, set the boundary penalty of your TopOpt to 0 (Neumann criteria). This will help force tet elements to follow the extrusion curve more appropriately. |

# Post-Processing

| Block | Advanced Topics |
|---|---|
| **Implicit Body from Topology Optimization**<br><br>Back to Basics | You may specify an element density threshold for the implicit body, which is different from the volume fraction (i.e. increasing the density threshold will increase the number of tet elements in select regions, while keeping the volume fraction constant).<br><br>A higher density threshold will remove more material, while a lower one will keep more material. Select your density threshold by comparing results at different thresholds, and use a smaller filter size to see the difference between various density thresholds.<br><br>You may have the implicit body reflect a chosen iteration of the TopOpt, instead of the final solution (by default).<br><br>You may also specify a component number (i.e. FE component in your part) which will apply this function on a component of your topology optimization (if you have multiple FE components connected by tie constraints). |
| **Smoothen Body**<br><br>Back to Basics | Choose an interpolation type between grids (cubic offers the best representation).<br><br>Optionally, you may apply the smoothing operation to a single domain (i.e. region by bounding box), if you wish to smoothen some areas of your design space differently from other areas. |

# Best Practices - Topology Optimization

Pg 39 - Best Practices

# Best Practices Running a Topology Optimization

Here are a few practices to follow when running a topology optimization. These tips will help you output a very useful TopOpt result.

- ❑ TopOpts with bending and torsional loads return a more unique and interesting result, compared to TopOpts with simply tension / compression loads.
- ❑ Always keep in mind the printability of your part - ensure there are no floating elements in your final smoothened body
    - ❑ You can mesh your part then *Filter Mesh* to identify and remove any floating elements (dependent on their volume)

# Q/A - Further topics to explore

Pg 41 - Questions

nTopology

# Current Topology Optimization Questions to Resolve

- *Once a question has been answered, the answer should be documented in this slide deck*

1. What is the best way to select the threshold value (ie. in the HUD, and "Implicit from TopOpt" when reconstructing geometry, so that it satisfies all constraints? Need to run another simulation?
    a. What does the .5 threshold value mean in relation to constraints? I.e., does the 50% mark mean all constraints are satisfied at the desired value, within a tolerance?

*************************

2. What would be the accuracy of a TopOpt with multiple different load cases (i.e. a part which observes various loading conditions at different times) versus merging TopOpt results for each respective loading condition (i.e. merging the smoothened bodies of one TopOpt with another)? Assuming that the TopOpt treats every loading condition with equal weighting, would I be able to change the weighting of a loading condition (i.e. dependent upon how often one condition occurs over another), within one TopOpt?
    - Merging combined results may not satisfy both constraints. E.g. You cannot guarantee a 50% volume reduction constraint from two distinct studies will be satisfied when the results are merged.
3. When would a user attempt to *maximize* one of our design responses? Is there an example of this? Most TopOpts attempt to minimize an optimization objective, with the large majority focusing on structural compliance.
    - Maximizing displacement but minimizing peak stress.
    - Could potentially maximize moment of inertia or center of mass away from a certain point.
    - Maximize eigenfrequencies

4. How is the volume fraction response used as an Optimization Objective compared to using it as a Design Response Constraint? It's more intuitive to use it as a constraint, not sure how it fits in as an optimization objective.
    - Volume Fraction Response can be used as an objective or a constraint

5. Same goes for the Displacement Response and the Stress Response, as an Optimization Objective. Since we do not specify a value for these responses, would the process involve a simulation at each step of the TopOpt determining if these parameters have been minimized? What is the algorithm involved in minimizing these responses? Do we have examples of a situation where, for example, a Displacement Response is used on a particular boundary as an optimization objective?
    - Each iteration has a new density assigned to it, and a FE simulation is performed each time.

6. When would the boundary conditions differ between optimization objectives and design response constraints? Why don't we have a universal list of boundary conditions in the TopOpt block, instead of appending our boundary conditions to each design response?
    - Each BC list corresponds to a different static analysis - this provides the most flexibility. e.g. Simultaneously considering flexibility and stiffness for different areas.

7. Do you require a symmetric *mesh* to run the planar symmetry constraint? If not, would the results be more accurate with a symmetric mesh (obviously can only be applied to a part which is symmetric about a chosen plane)?
    - Best practice is to split the domain (in CAD), mesh from CAD, Remesh, mirror the mesh, then merge the mesh.

8. What's the difference between using an extrusion constraint and setting the boundary penalty of a TopOpt to 0? I see setting the penalty to 0 renders more optimal results for an extrusion constraint-based TopOpt, but doesn't a low boundary penalty already ensure most tet elements remain on the boundaries / edges of the design space?