# Helmholtz Filter Tutorial

Helmholtz Filter Tutorial

# Problem Description

- We will be demonstrating the advantage of the Helmholtz filter over the kernel filter on a problem that demonstrates one of the pathologies of the kernel filter.

- The demonstration will be on a simple, axial loaded, mechanical problem.

- The green and magenta volumes are connected at the green surface on the magenta volume.

- The magenta and cyan volumes are connected at the magenta surface on the cyan volume.

- The portion of the optimizable region that protrudes into the fixed region will create an undesirable pathology when the kernel filter is used.



Load

Fixed

Fixed Regions

Optimizable Region with portion protruding into fixed region
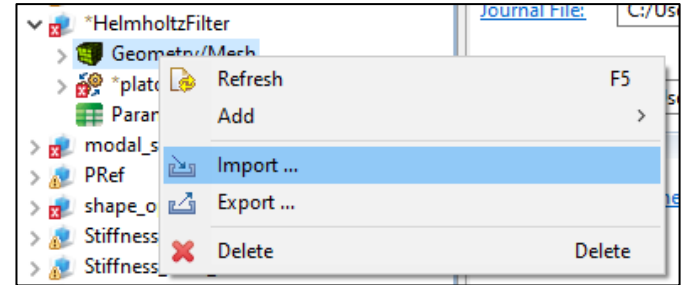
Helmholtz Filter Tutorial
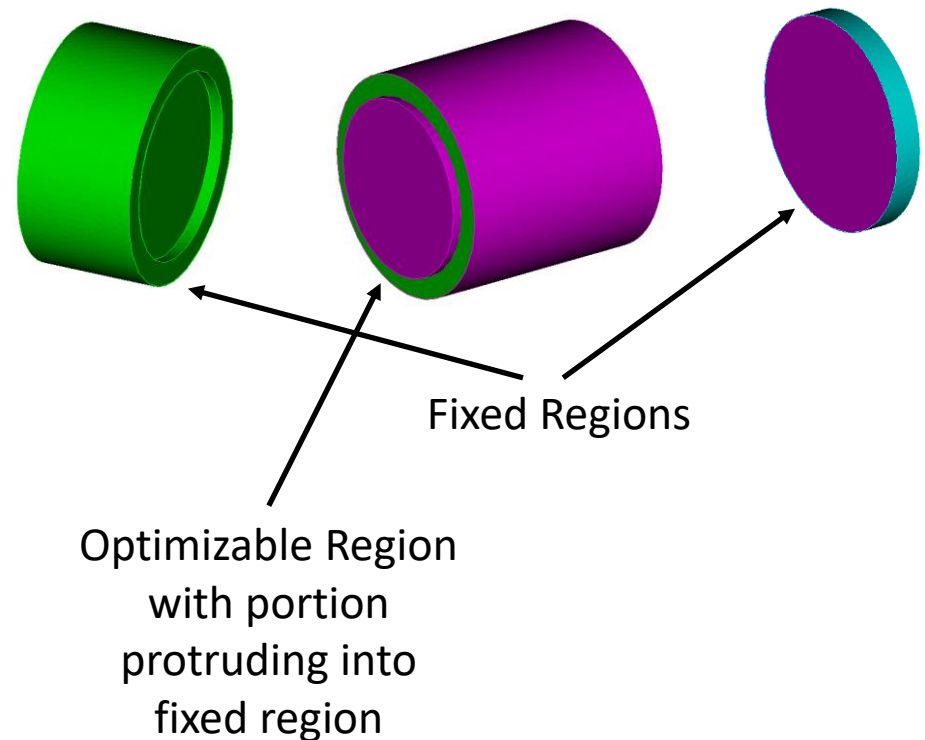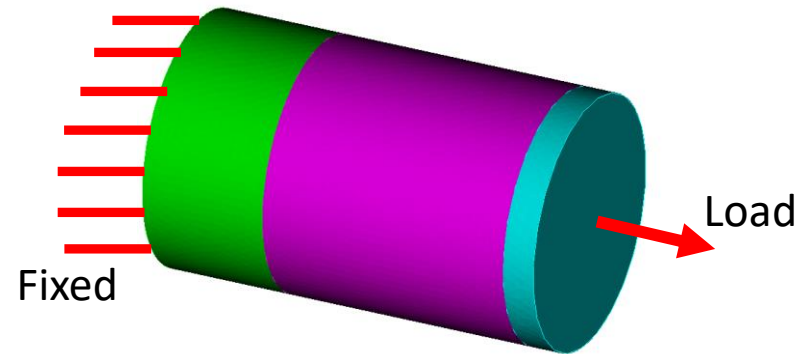
# Create a New Model



- Choose **File->New->Model** in the menu

- Choose **New Model** then **Next**

- Choose **Create From Template** then **Next**

- Enter **HelmholtzFilter** as the **Model Name**

- Choose the **Plato Templates->Basic->Maximize Stiffness (PlatoAnalyze)** template and then **Finish**
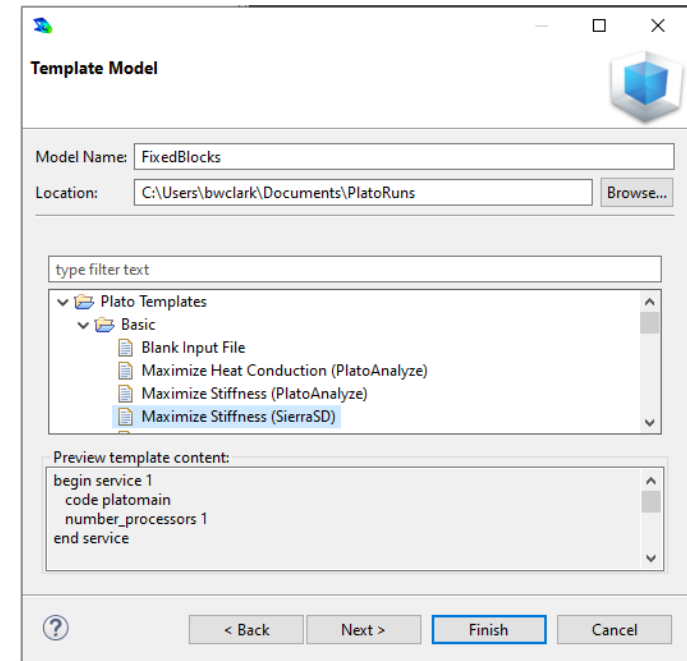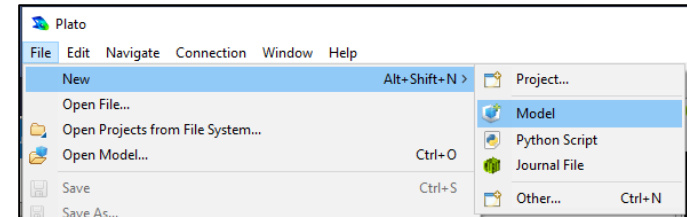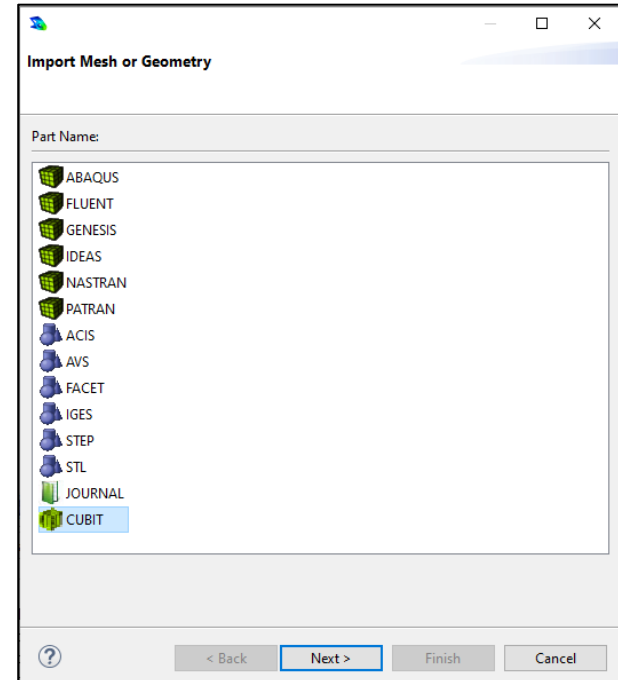
# Create the Design Domain

- Right-click on the **Geometry/Mesh** node in the Model Navigator tree and choose **Import**.
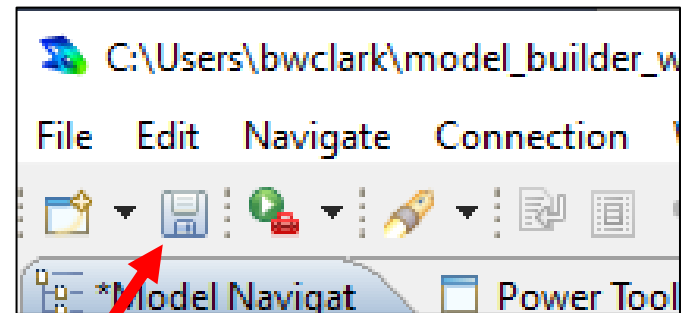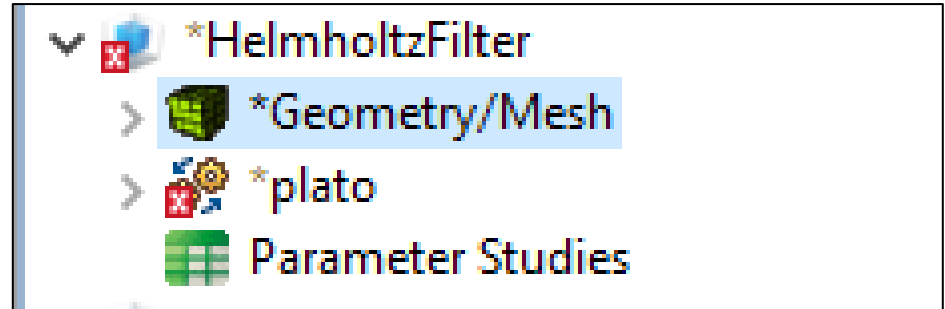
- Choose **CUBIT** and click **Next**.

- Browse for the file called **helmholtz_filter.cub** and click **Finish**.

# Save the Geometry/Mesh



- Click on the **Geometry/Mesh** node in the tree and then click the Save icon in the toolbar.

# Delete the "device_ids" Parameter

- Click on **HelmholtzFilter->plato->Services->service 2** in the tree to bring up the settings for **service 2** in the settings panel.

- Right-click on **device_ids 1** and choose **delete**.

- This will cause this service to run on the default device.
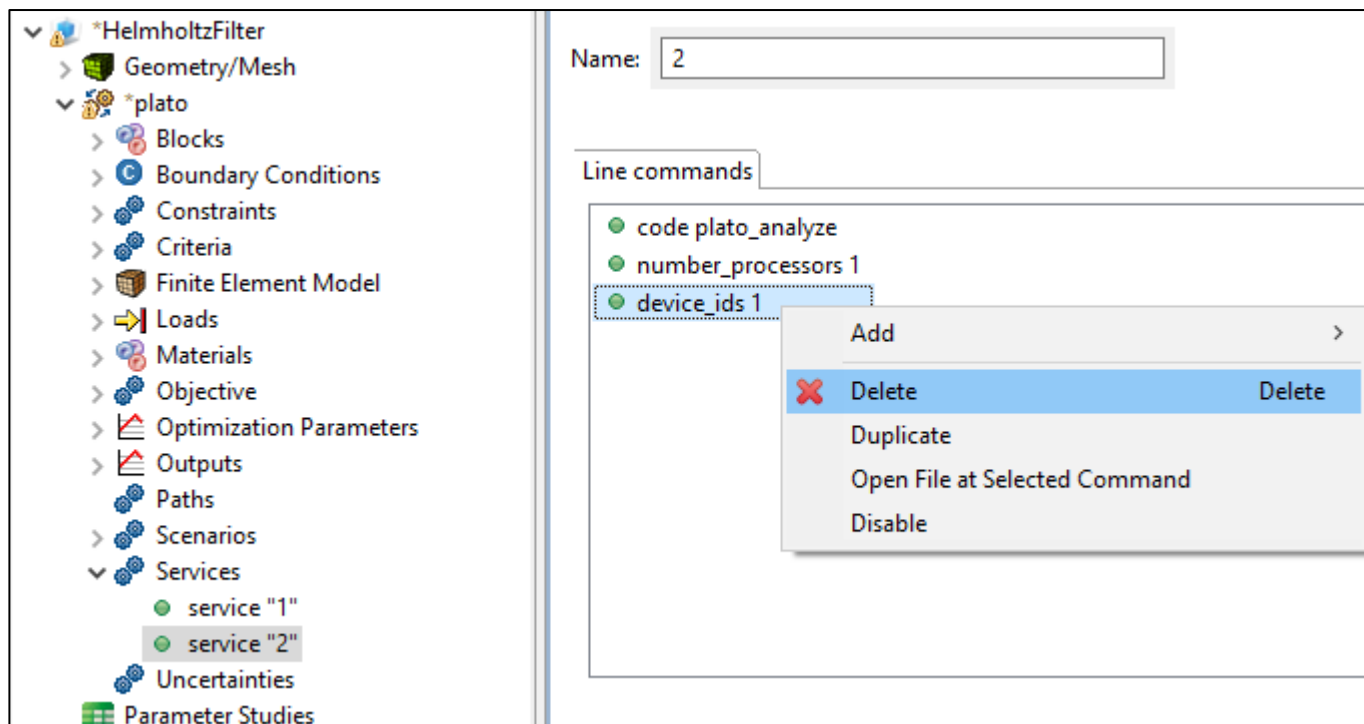
# Add Solver Tolerance Setting

- Click on **HelmholtzFilter->plato->Scenarios->scenario 1** in the tree to bring up the scenario settings.

- Right click anywhere in the settings window and choose **Add->tolerance**.

- Set the **tolerance** value to be **1e-9**.
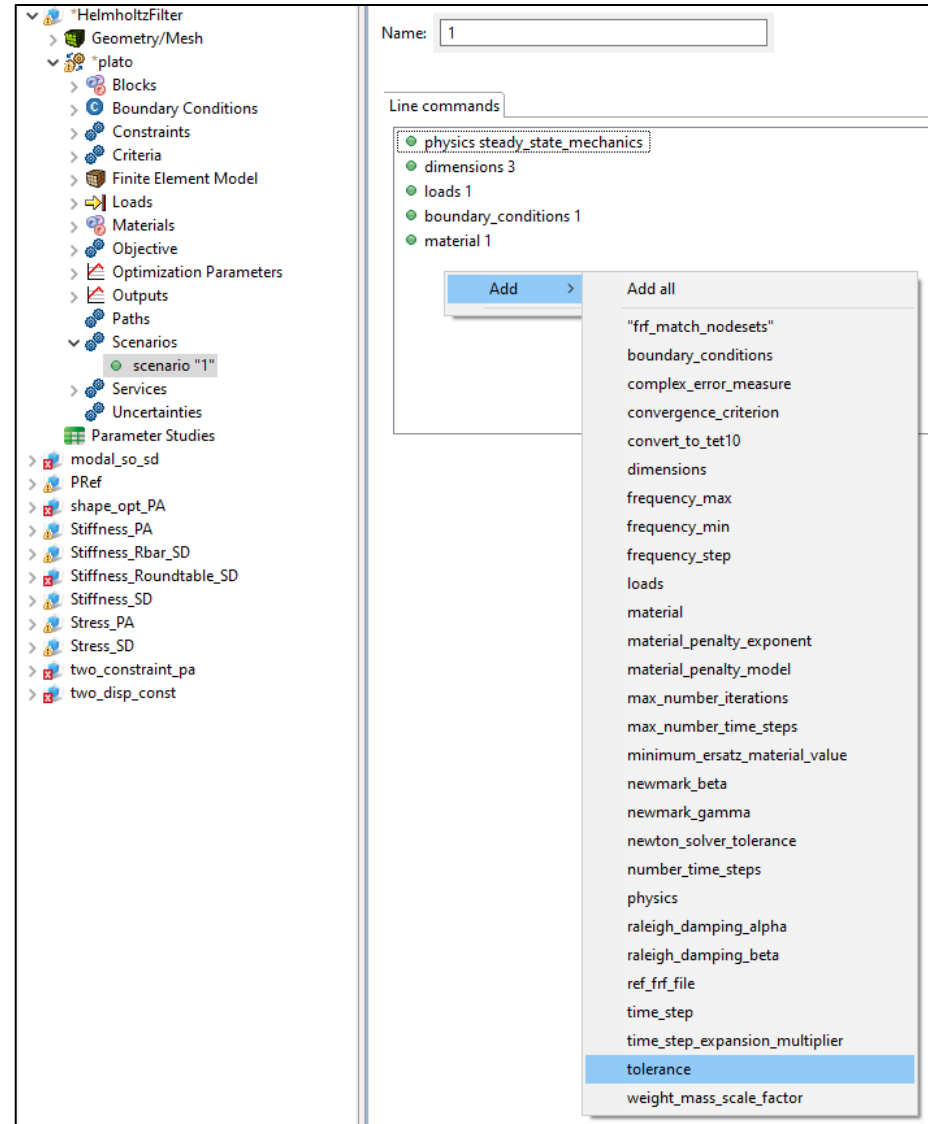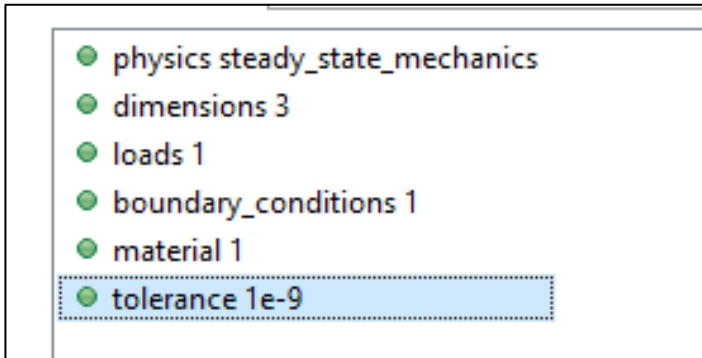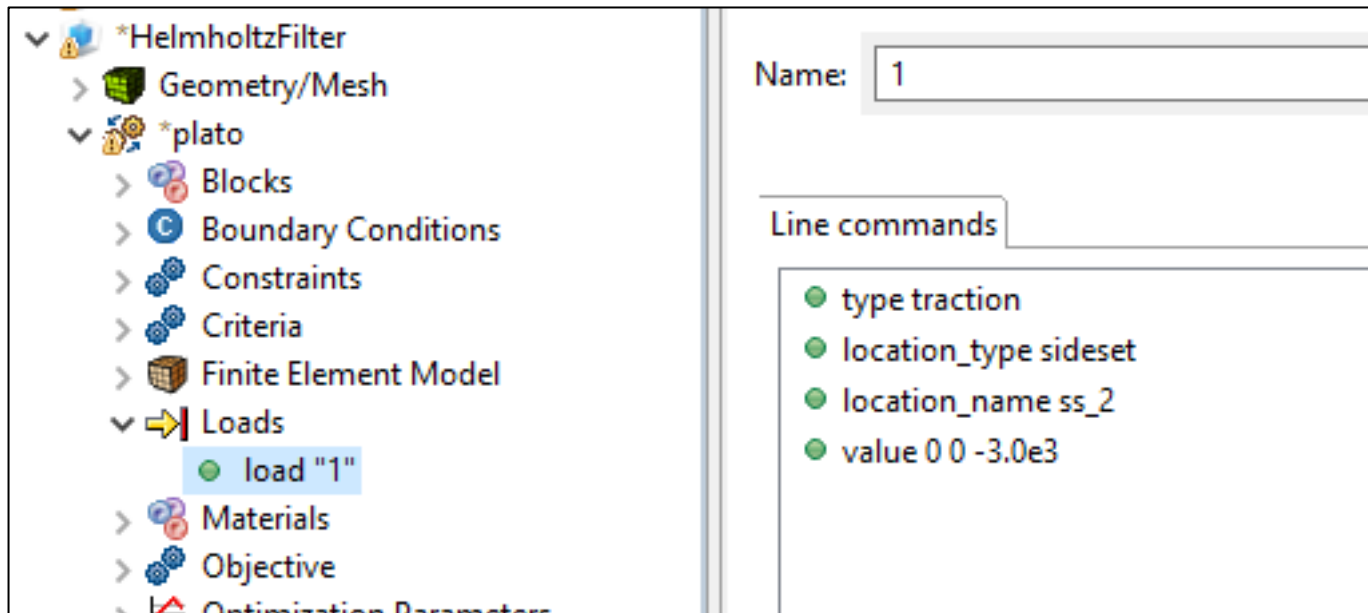
# Modify Fixed Boundary Condition

- Click on **HelmholtzFilter->plato->Boundary Conditions->boundary_condition 1** in the tree.
- Change the **location_type** to **sideset**.
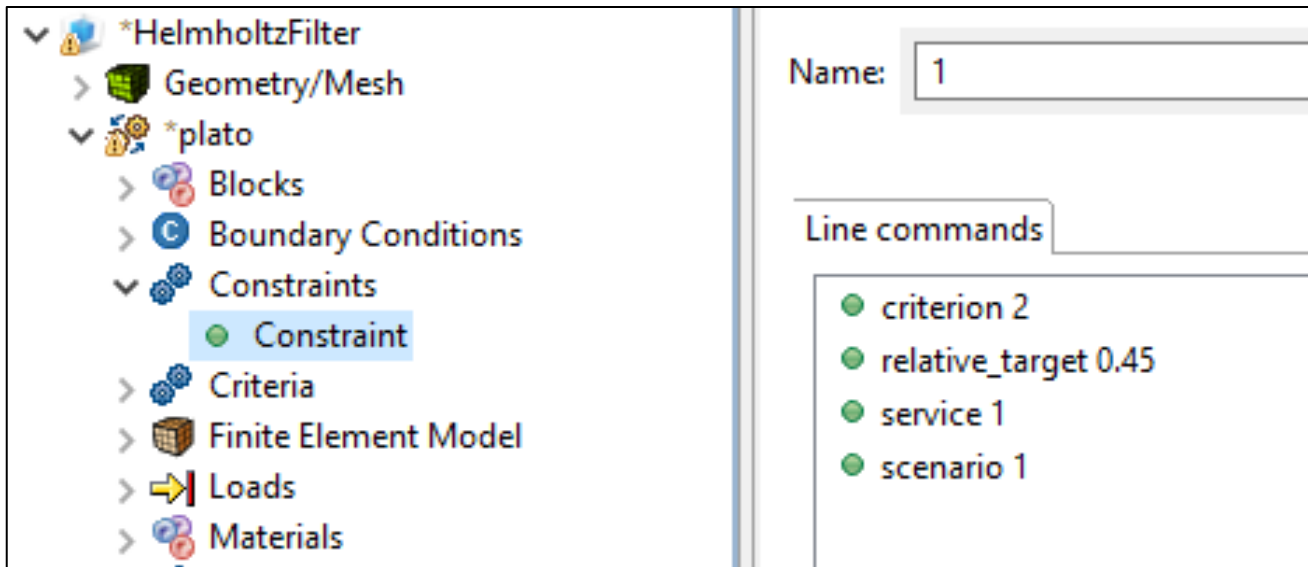- Change the **location_name** to **ss_1**.
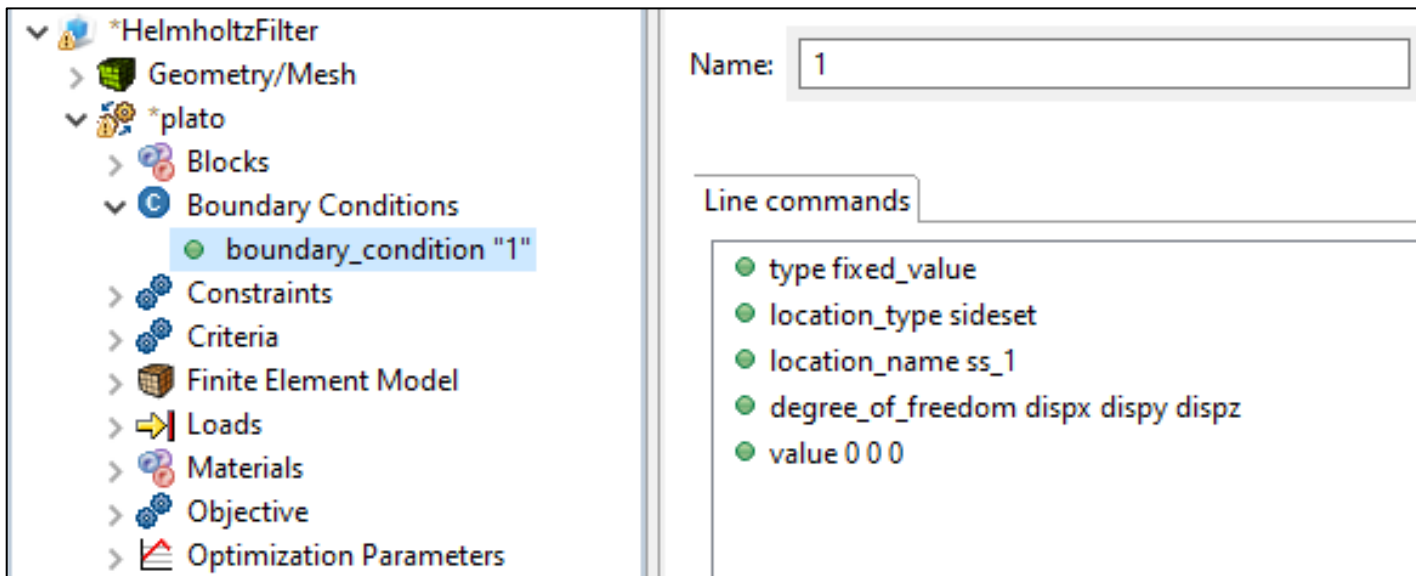
# Modify the Load

- Click on **HelmholtzFilter->plato->Loads->load 1** in the tree.

- Change the **location_name** to **ss_2**.

- Change the **value** to **0 0 -3.0e3**.

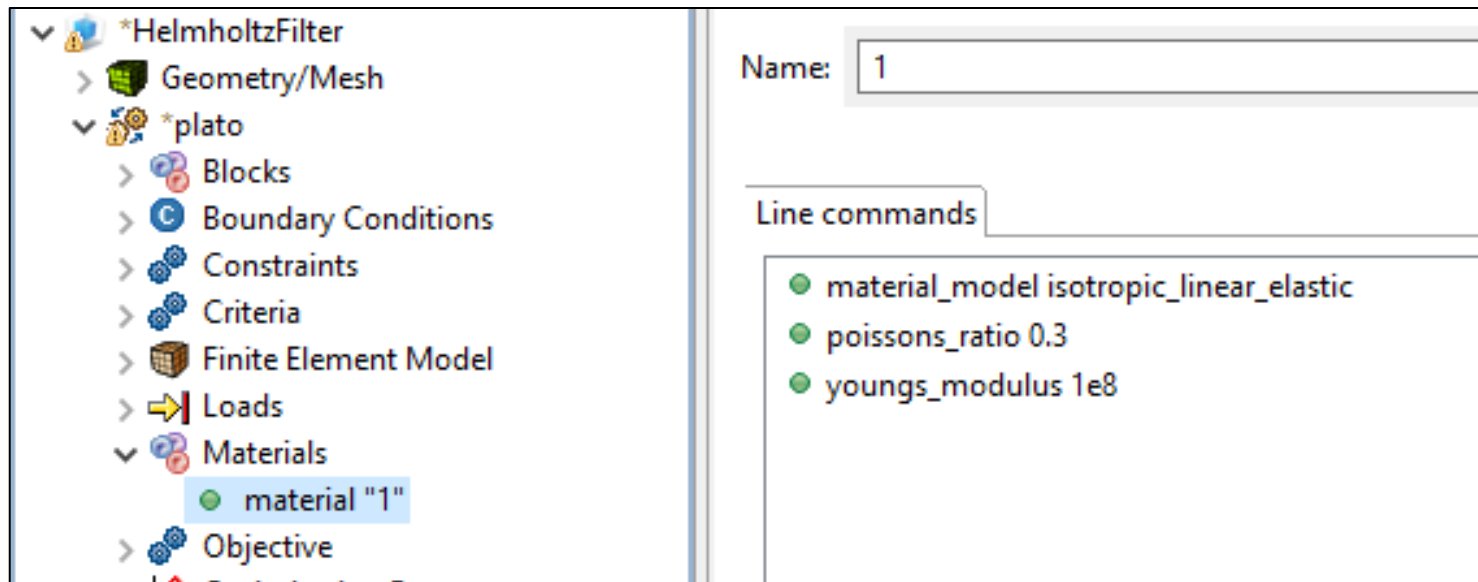# Modify the Constraint

- Click on **HelmholtzFilter->plato->Constraints->Constraint** in the tree.

- Change the **relative_target** to **0.45**.

# Modify the Material Properties

- Click on **HelmholtzFilter->plato->Materials->material 1** in the tree.

- Change **poissons_ratio** to **0.3**.

- Change **youngs_modulus** to **1e8**.

# Assign Material and Element Type

- Click on the **FEM Mesh** node in the tree under **plato->Finite Element Model**

- In the **Settings** panel click in the **Material** area next to Block 1 and select **"1"** from the dropdown list

- Then click in the **Element** area next to Block 1 and select "**tet4**" from the dropdown list

- Repeat this process for Block 2
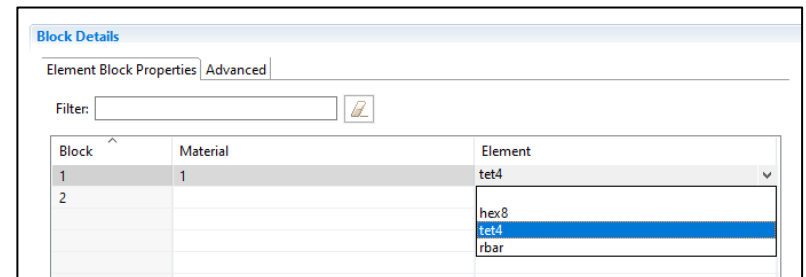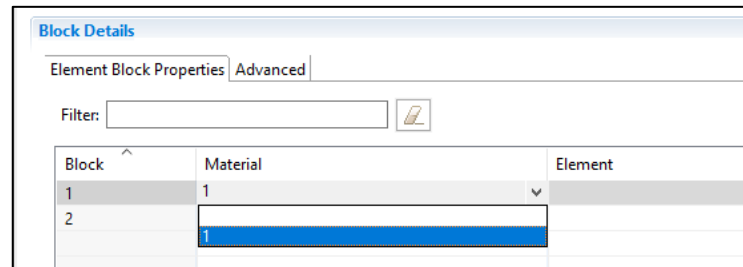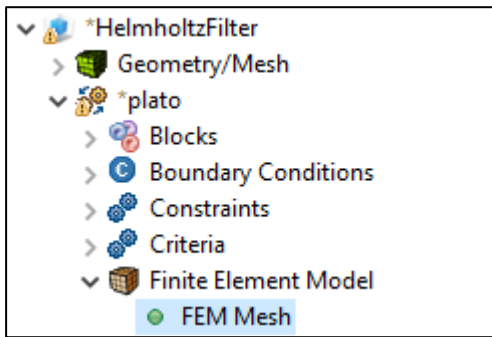
# Modify the Optimization Parameters

- Click on **HelmholtzFilter->plato->Optimization Parameters->optimization_parameters** in the tree.

- Delete **filter_radius_scale** by right-clicking it and choosing **Delete.**

- Add **filter_radius_absolute** by right-clicking anywhere in the settings parameters window and choose **Add->filter_radius_absolute.** Then set it to **0.3.**

- Add **initial_density_value** and set it to 0.45.

- Add **fixed_block_ids** and set it to **1.**
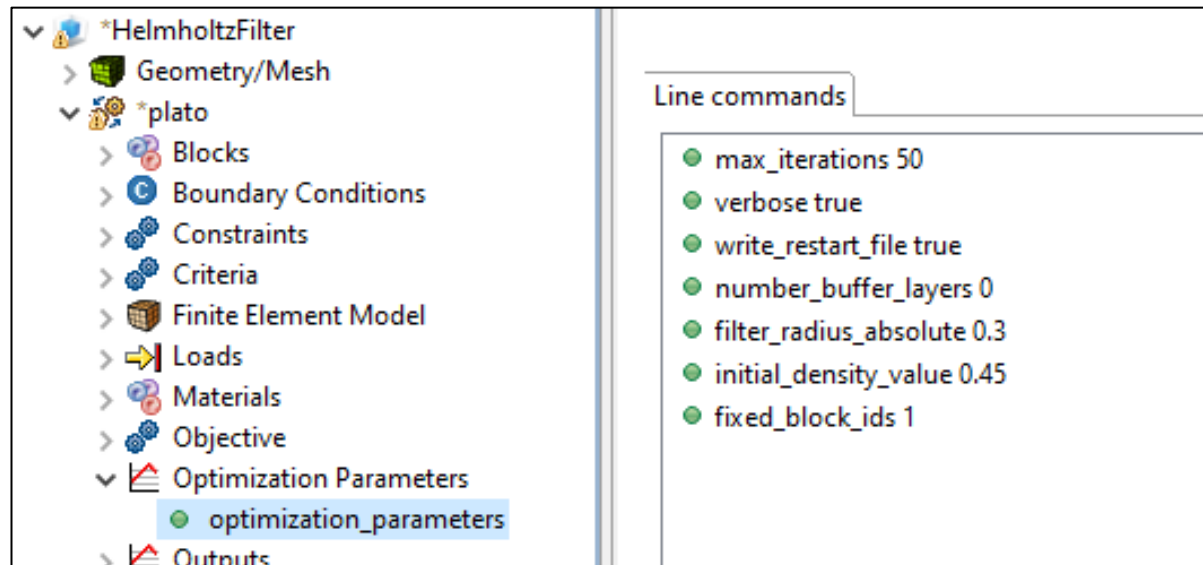
# Run the Optimization

- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view

- Choose **Plato** as the code and then choose the machine and execution template you want to use. In this tutorial we have set the problem up assuming we are using **Plato Analyze** as the physics code. Therefore, make sure you are running on a machine that has Plato Analyze installed.
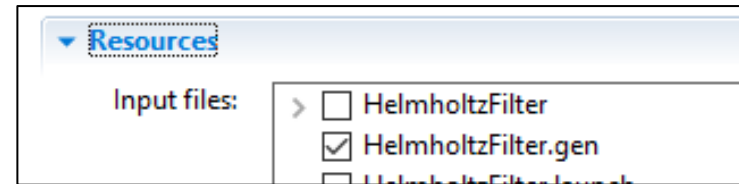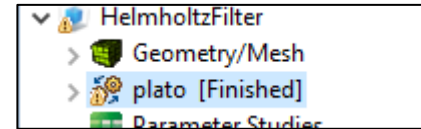
- In the **Resources** area make sure **HelmholtzFilter.gen** is checked

- In the **Prune and Refine** area make sure **Prune Mesh** is unchecked and **Number Refines** is **0**. We will not use these features in this example.

- Choose any other preferences and launch the job by clicking on **Submit Simulation Job** toward the top of the panel

# Result with the Kernel Filter



The default filter is the kernel filter. Notice that for this problem where we have non-connected mesh that is very close in proximity between the fixed and optimizable regions (between the green and magenta volumes) the kernel filter results in material in areas that are not in the load path. This is a pathology that we would like to avoid and the Helmholtz filter accomplishes this.
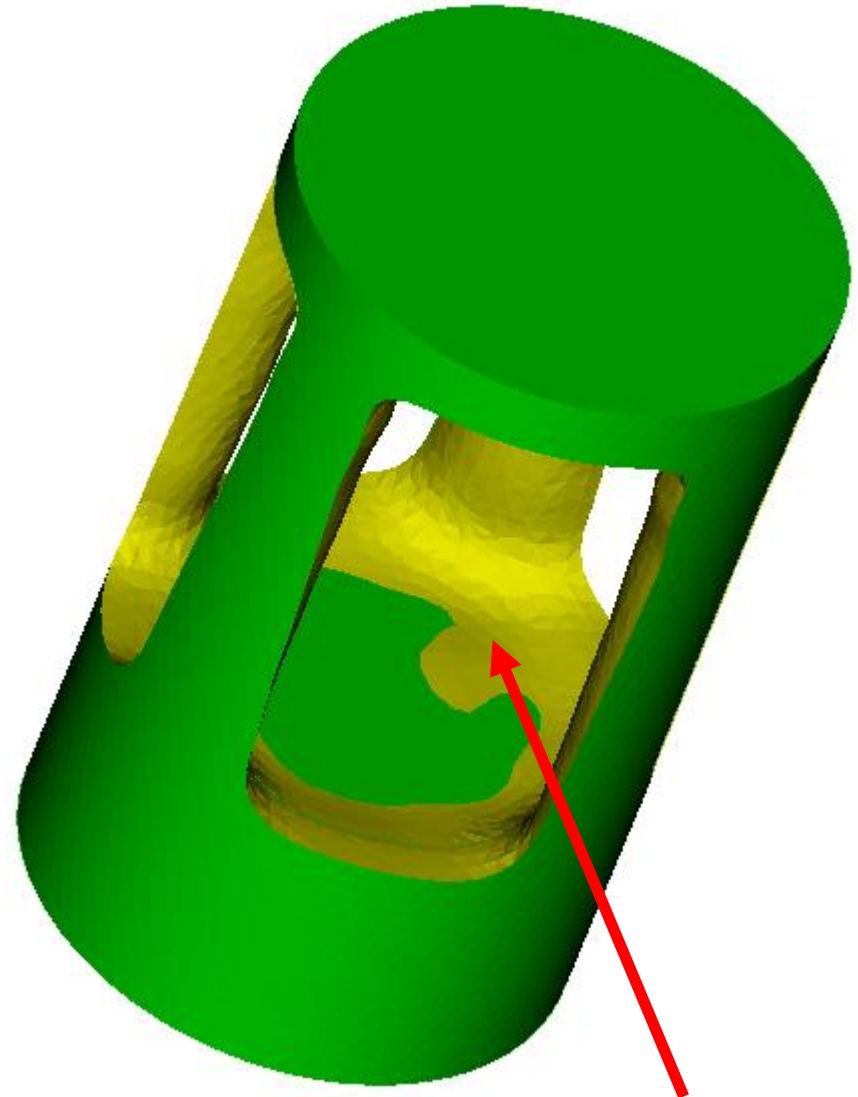
Pathological material

# Modify the Filter

- Click on **HelmholtzFilter->plato->Optimization Parameters->optimization_parameters** in the tree.

- Add a **filter_type** parameter and set it to **helmholtz.**

- Add a **boundary_sticking_penalty** parameter and set its value to **0**. This will tell the Helmholtz filter not to avoid the phenomenon where material is pushed toward the boundary of the design domain. The kernel has this undesirable effect and the Helmholtz filter by default (boundary_sticking_penalty=1) will avoid this but for the purposes of comparing with the kernel filter we will set the value to 0.
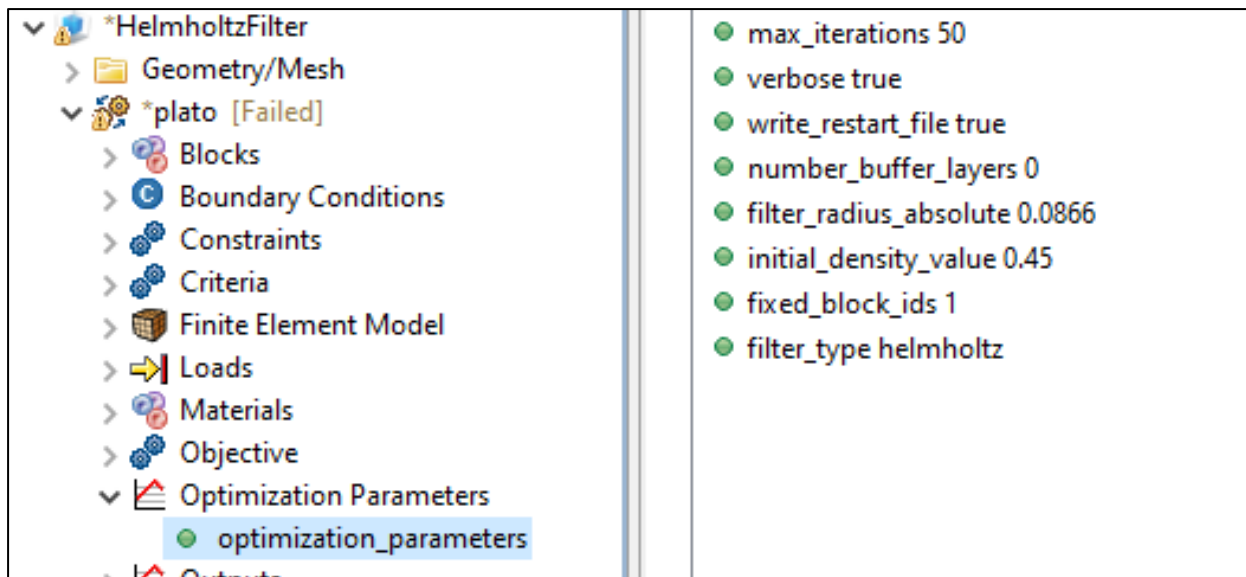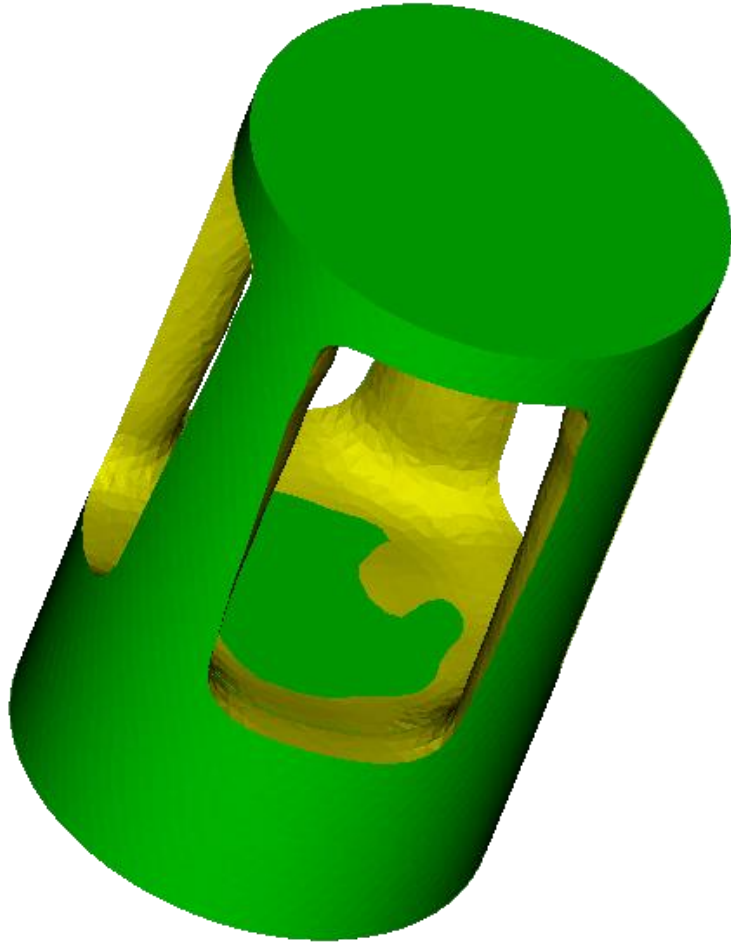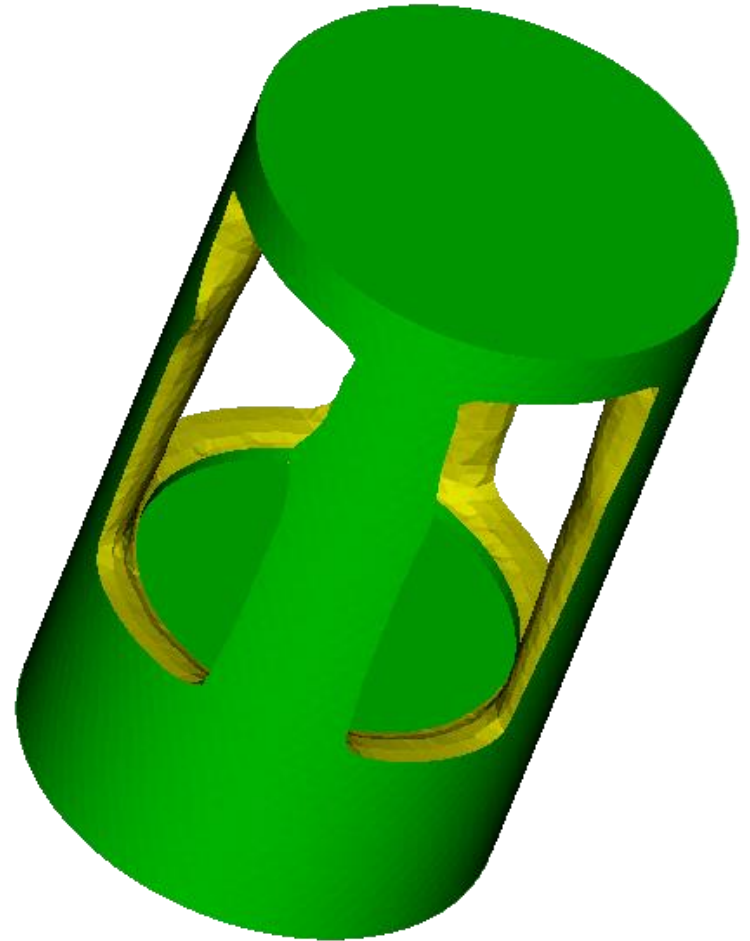
# Re-run the Optimization

- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view

- Choose any other preferences and launch the job by clicking on **Submit Simulation Job** toward the top of the panel

# Result with Helmholtz Filter



Kernel Filter                              Helmholtz Filter

Helmholtz Filter Tutorial