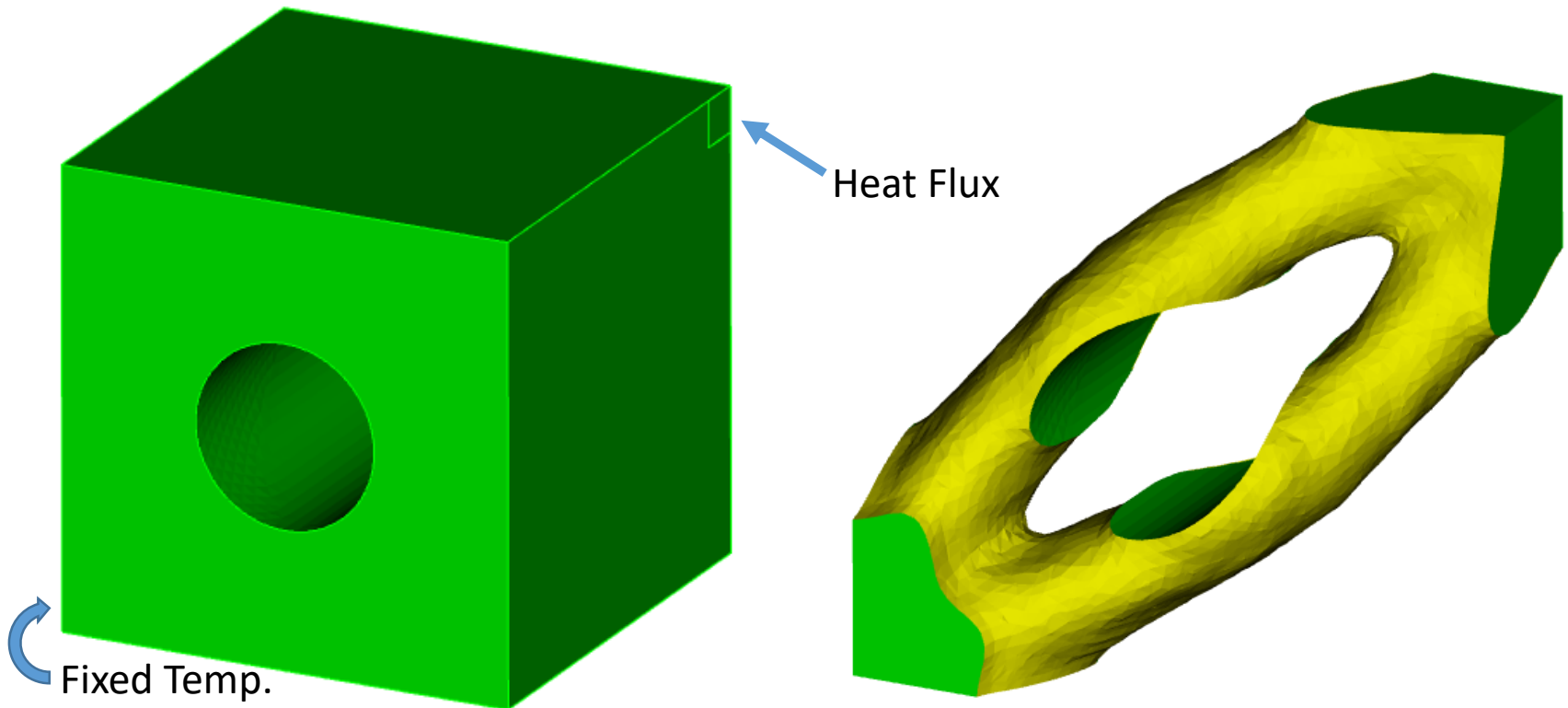


# Maximize Heat Conduction Using Plato Analyze—Tutorial

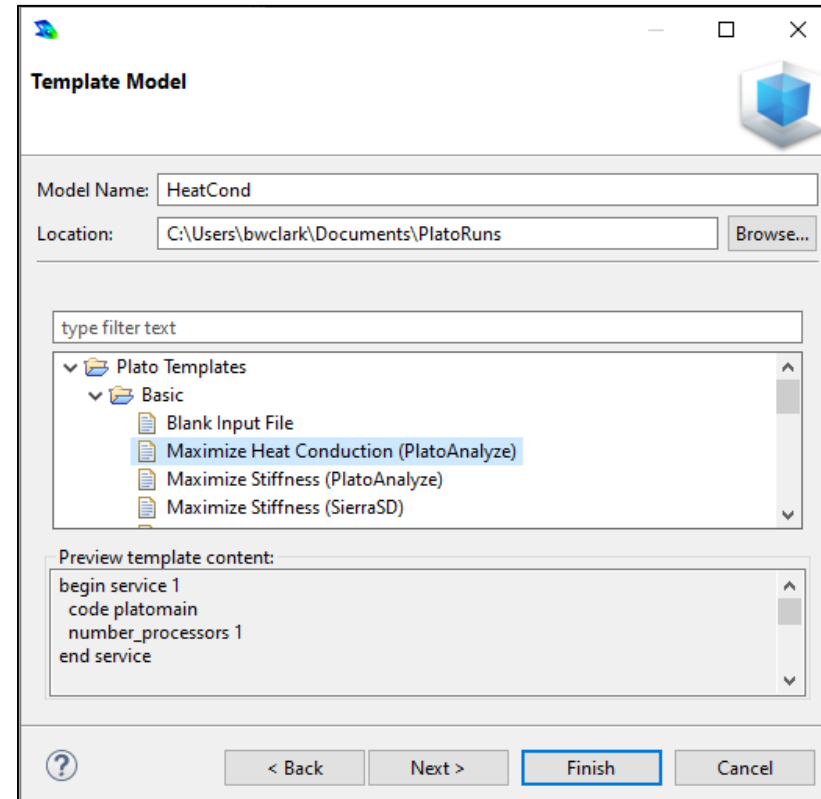
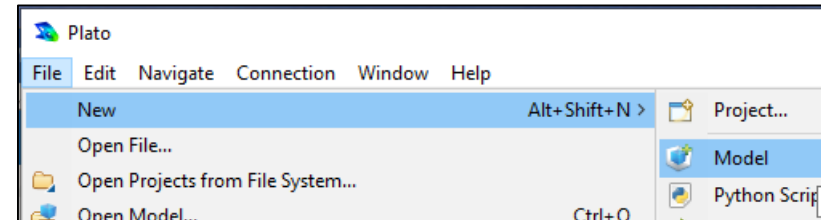
# Problem Description

In this problem we will design a geometry that maximizes heat conduction between a surface being held at constant temperature and a surface at which there is a constant heat flux. Another way to think of this problem is that we want to minimize the temperature difference between the two surfaces.



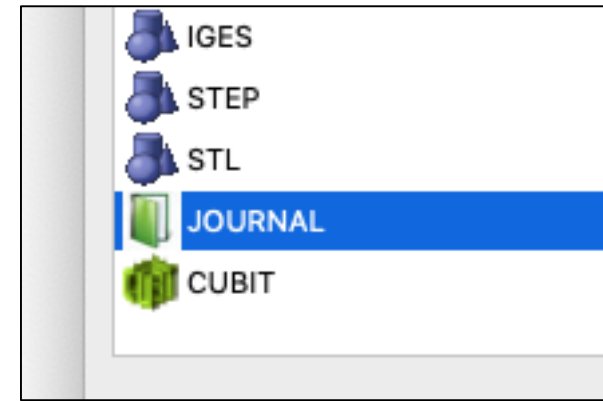
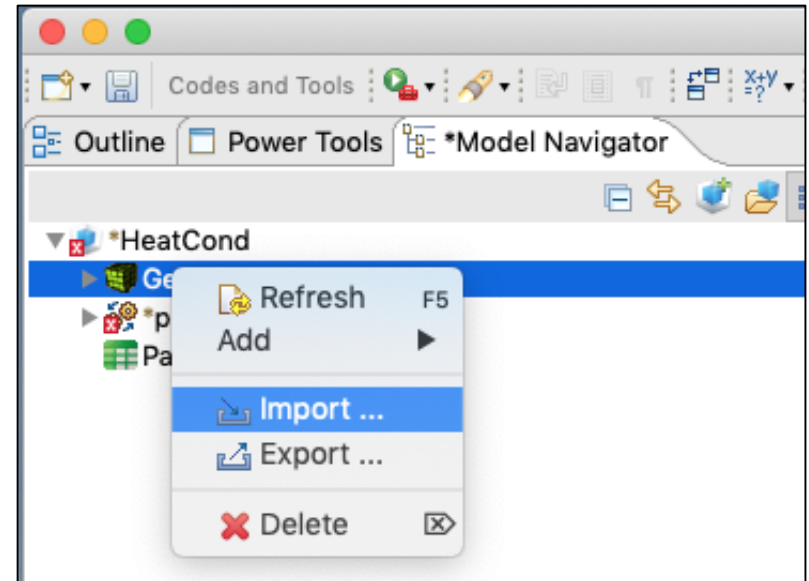
# Create a New Model

- Choose **File->New->Model** in the menu
- Choose **New Model** then **Next**
- Choose **Create From Template** then **Next**
- Enter **HeatCond** as the **Model Name**
- Choose the **Plato Templates->Basic->Maximize Heat Conduction (PlatoAnalyze)** template and then **Finish**



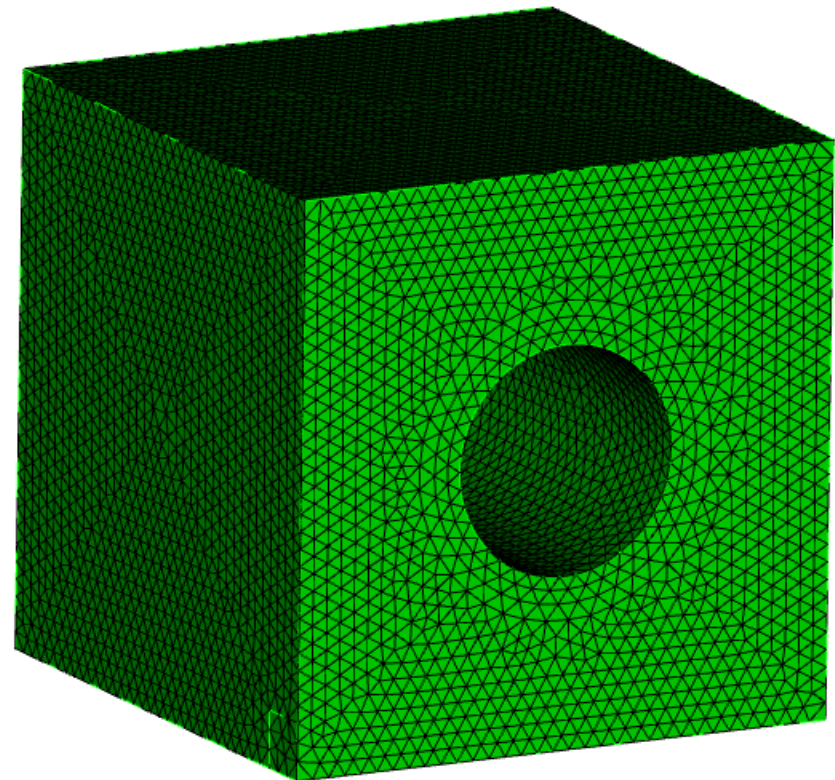
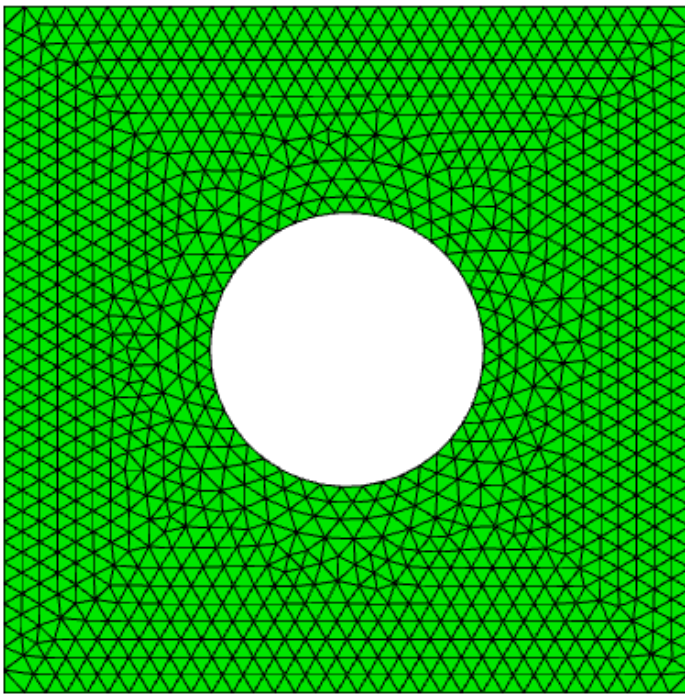
# Create the Design Domain and Mesh

- Right-click the **Geometry/Mesh** node in the Model Navigator tree and choose **Import...**
- Choose **JOURNAL** in the Import dialog box and choose **Next**.
- Browse to the file called **pa\_thermal\_build\_geom\_and\_mesh.jou** included with this tutorial and choose **Finish**. If you get a message about the model being updated in the background and asking if you want to load the new model and lose changes choose “no”.



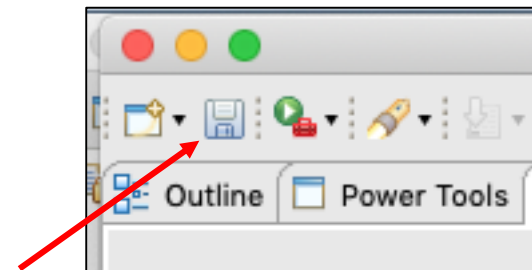
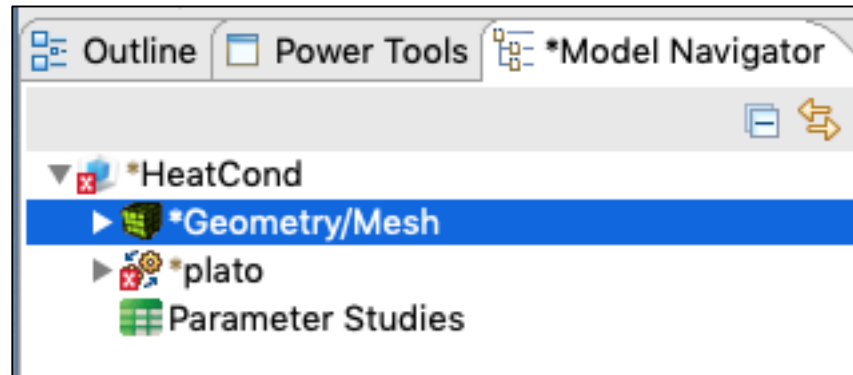
# Design Domain and Mesh

- At this point you should see the meshed CAD model of the design domain. The journal file that was imported created the cad geometry, the mesh, and then created blocks, sidesets, and nodesets on the mesh. These will be used later for setting fixed regions, applying loads, and setting fixed boundary conditions.



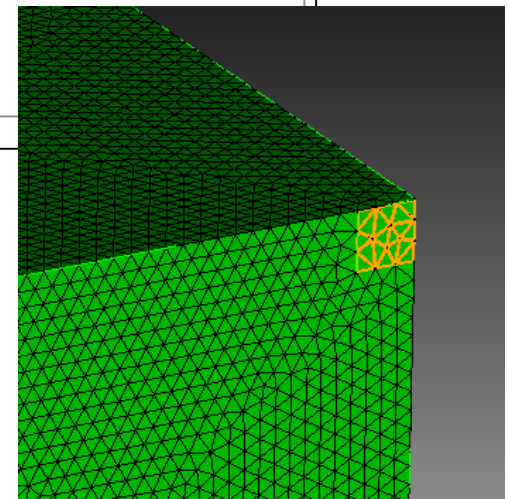
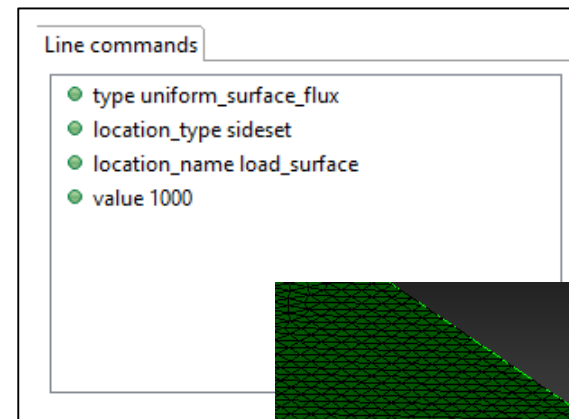
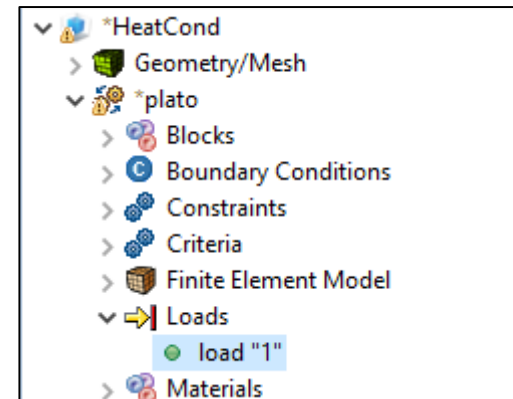
# Save the Geometry and Mesh

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



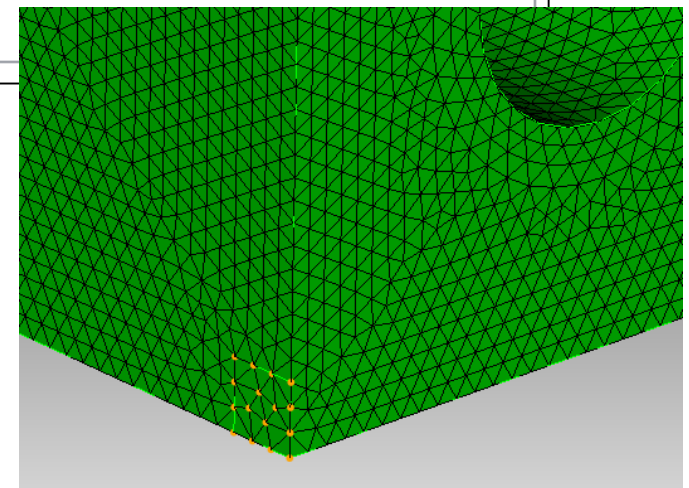
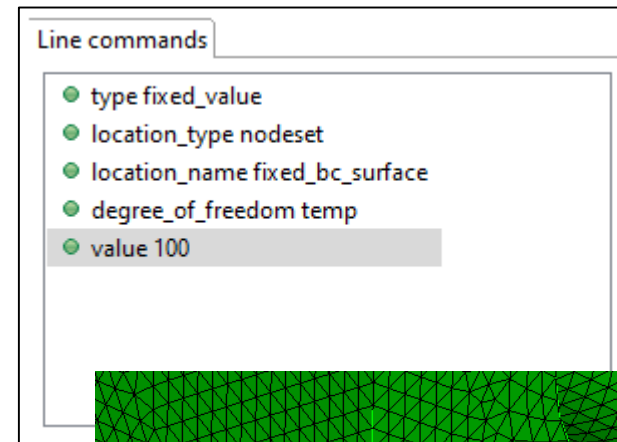
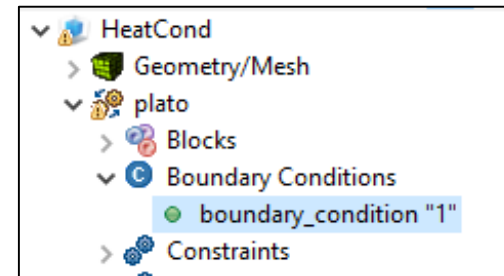
# Apply a Heat Flux Load

- Click on **plato->Loads->load "1"** in the **Model Navigator** tree. This load was already defined in the template we used for this model.
- In the **Settings** panel
  - Change **location\_name** to **load\_surface**
  - Change **value** to **1000**
- These steps added a heat flux of 1000 on the sideset named **load\_surface**. Sideset **load\_surface** can be seen highlighted in the graphics window by selecting **Geometry/Mesh->SideSets->load\_surface** in the **Model Navigator** tree. See image on right.



# Create a Fixed Temperature Boundary Condition

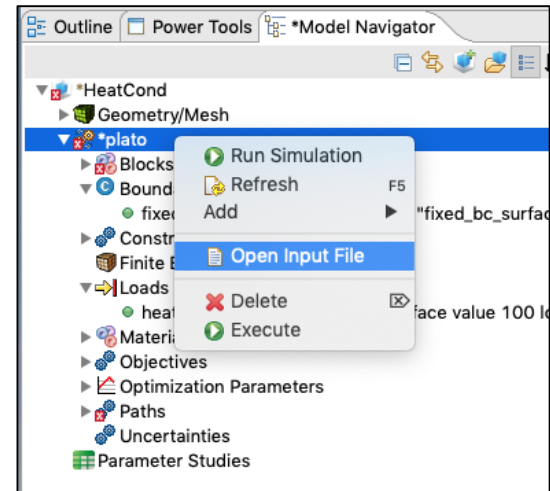
- Click on **plato**->**Boundary Conditions**->**boundary\_condition "1"** in the **Model Navigator** tree. This boundary condition was already defined in the template used to create this model.
- In the **Settings** panel
  - Change **location\_name** to **fixed\_bc\_surface**.
  - Change **value** to **100**
- These steps added a fixed temperature boundary condition to Nodeset fixed\_bc\_surface with a value of 100 and assigned it boundary condition id 1. Nodeset fixed\_bc\_surface can be seen highlighted in the graphics window by selecting **Geometry/Mesh**->**NodeSets**->**fixed\_bc\_surface** in the **Model Navigator** tree. See image on right.





# Review Input Deck

- Right-click on **plato** in the **Model Navigator** tree and select **Open Input File**
- The input deck **HeatCond.i** should come up in an editor and look something like that shown here.
- Verify the boundary condition and load we created. They are defined in **boundary\_condition** and **load** blocks and are referenced by id in the **scenario** block.



```
*HeatCond.i
begin service 1
  code platomain
  number_processors 1
end service

begin service 2
  code plato_analyze
  number_processors 1
  device_ids 1
end service

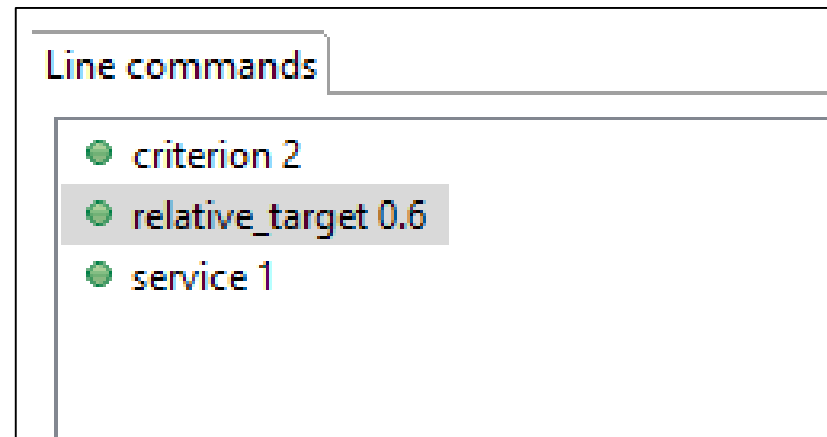
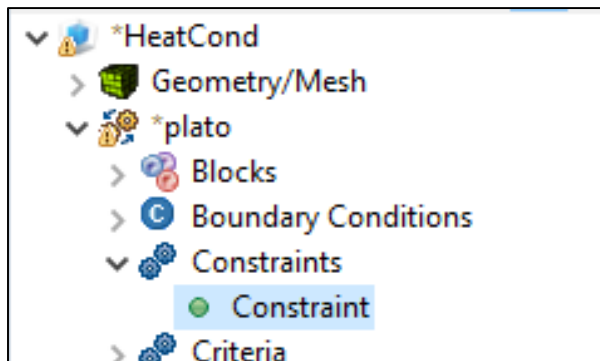
begin criterion 1
  type thermal_compliance
end criterion

begin criterion 2
  type volume
end criterion

begin scenario 1
  physics steady_state_thermal
  dimensions 3
  loads 1
  boundary_conditions 1
  material 1
```

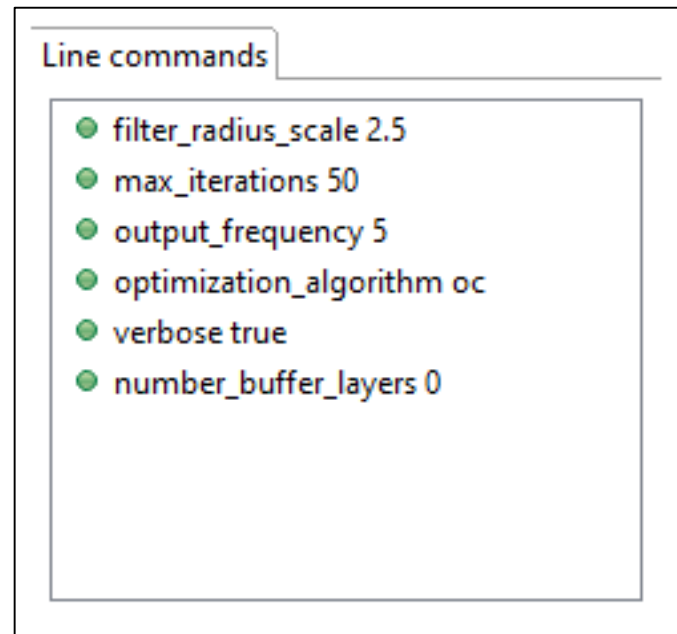
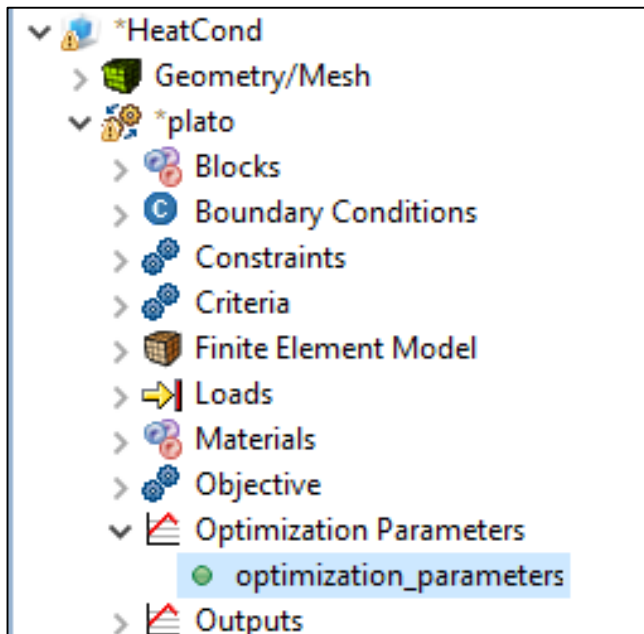
# Set the Volume Constraint

- Click on the **Constraints->Constraint** node in the tree.
- In the **Settings** panel set the **relative\_target** parameter to be **0.6**. This will indicate that we want the final design to only use 60% of the original volume.
- You can verify the constraint type by clicking on **Criteria->criteria “2”** (the one reference in the constraint) and checking that it is of type “volume”.



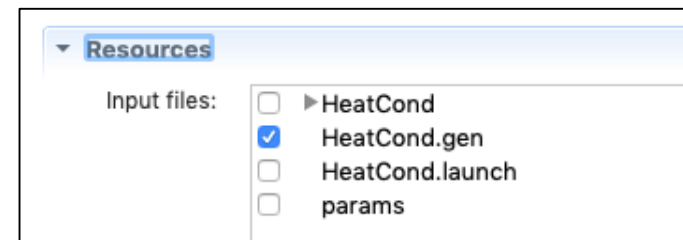
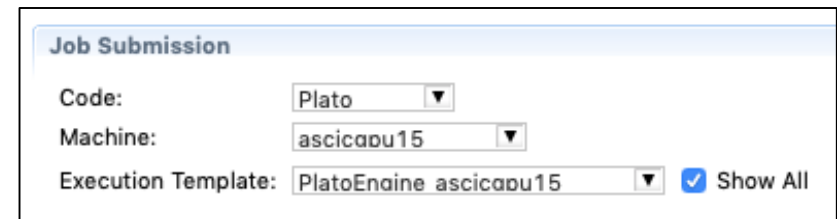
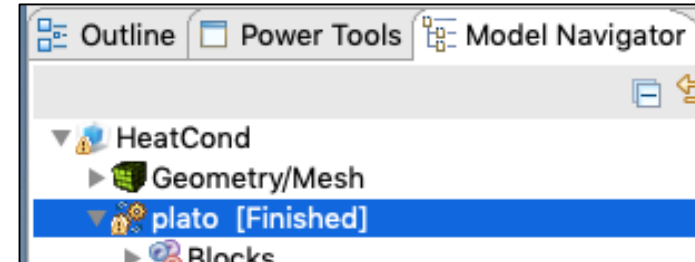
# Set the Top. Opt. Parameters

- Click on the **Optimization Parameters->optimization\_parameters** node in the tree.
- Change the **filter\_radius\_scale** to **2.5**. The larger this value the smoother the results and the larger the features.
- Change the **optimization\_algorithm** to **oc**.



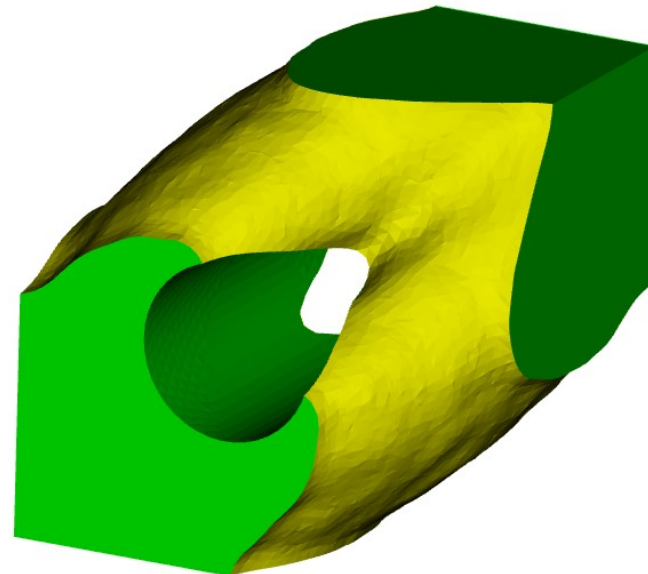
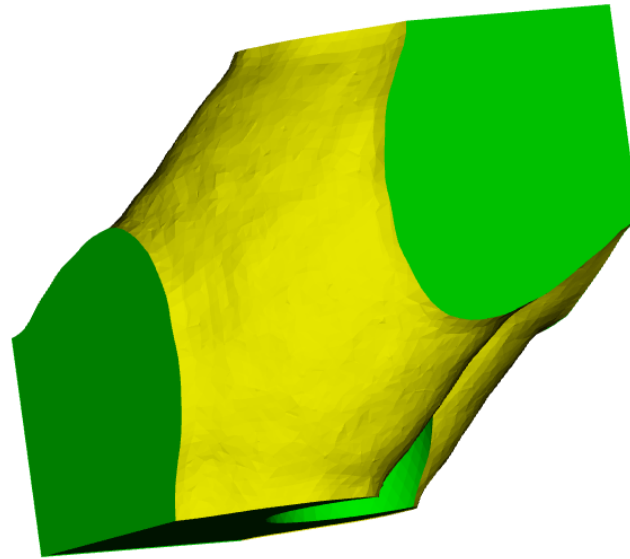
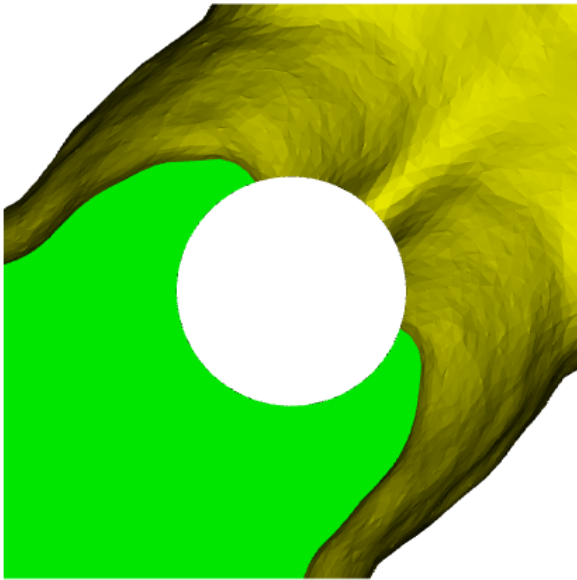
# Run the Optimization

- 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. If running on Sandia's CEE LAN choose one of the ascicgpu machines as shown here.



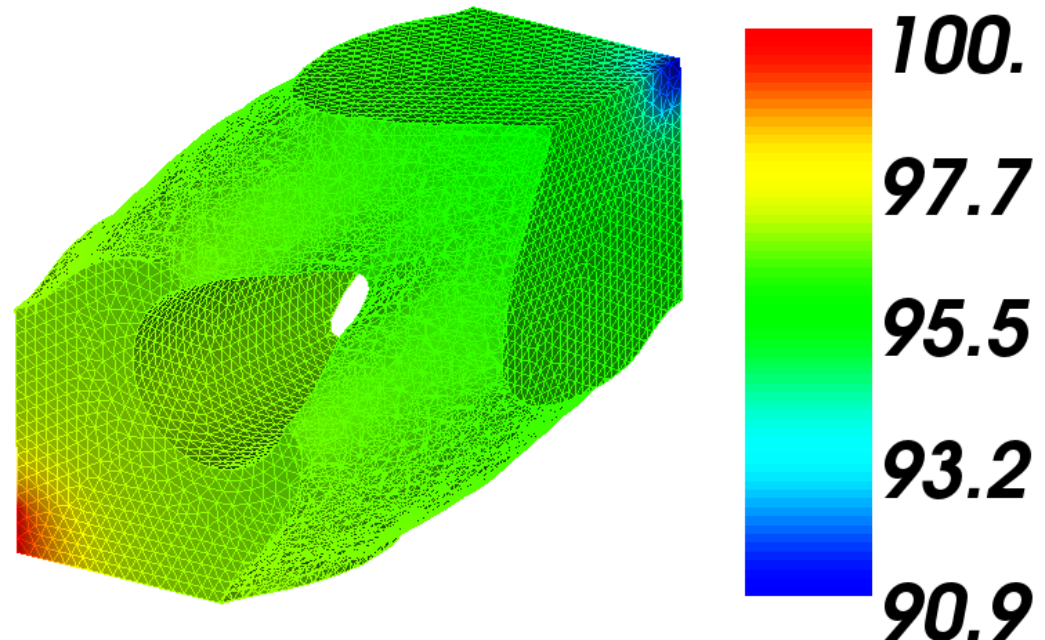
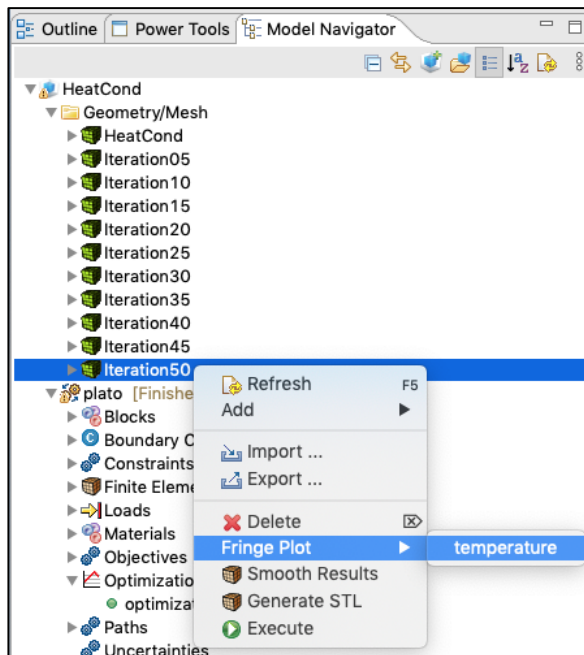
- In the **Resources** area make sure **HeatCond.gen** is checked
- Choose any other preferences and launch the job by clicking on **Submit Simulation Job** toward the top of the panel

# Results



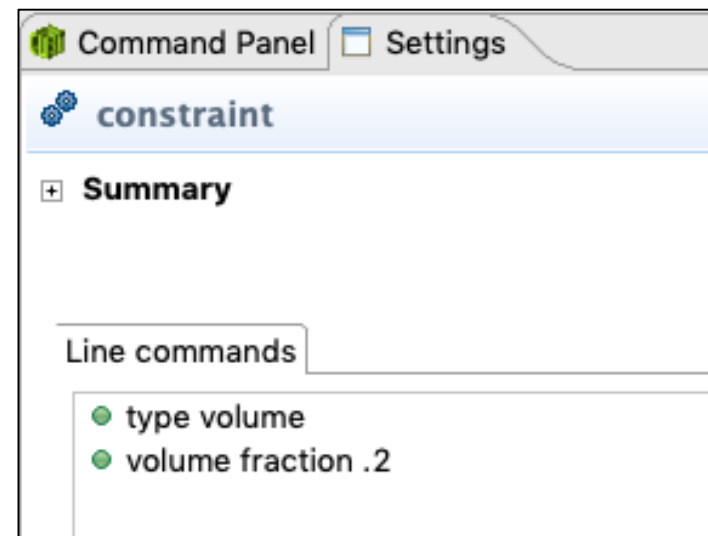
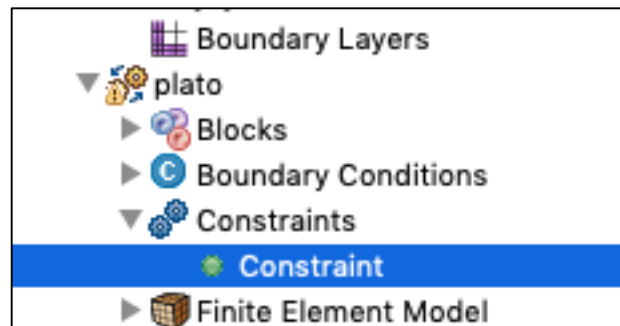
# Results

Right-click on the last Iteration under Geometry/Mesh and choose **Fringe Plot->temperature\_plato\_analyze\_2**. This plots the temperature on the resulting design. Notice that for 60% of the material the design is able to keep the two extreme temperatures within about 9 degrees of each other. Next we will try reducing the amount of material to see how it affects the resulting temperature range.



# Update the Volume Budget and Re-run

- Click on the **Constraints->Constraint** node in the tree.
- In the **Settings** panel set the **relative\_target** parameter to be **0.2**. This will indicate that we want the final design to only use 20% of the original volume.
- Now, rerun the problem by clicking on **plato** in the tree and then **Submit Simulation Job** in the settings panel



# Results for 20% Volume Budget

Plot the temperature for the new results and note that with less material less heat is transferred, and the two extreme temperatures now differ by about 14 degrees.

