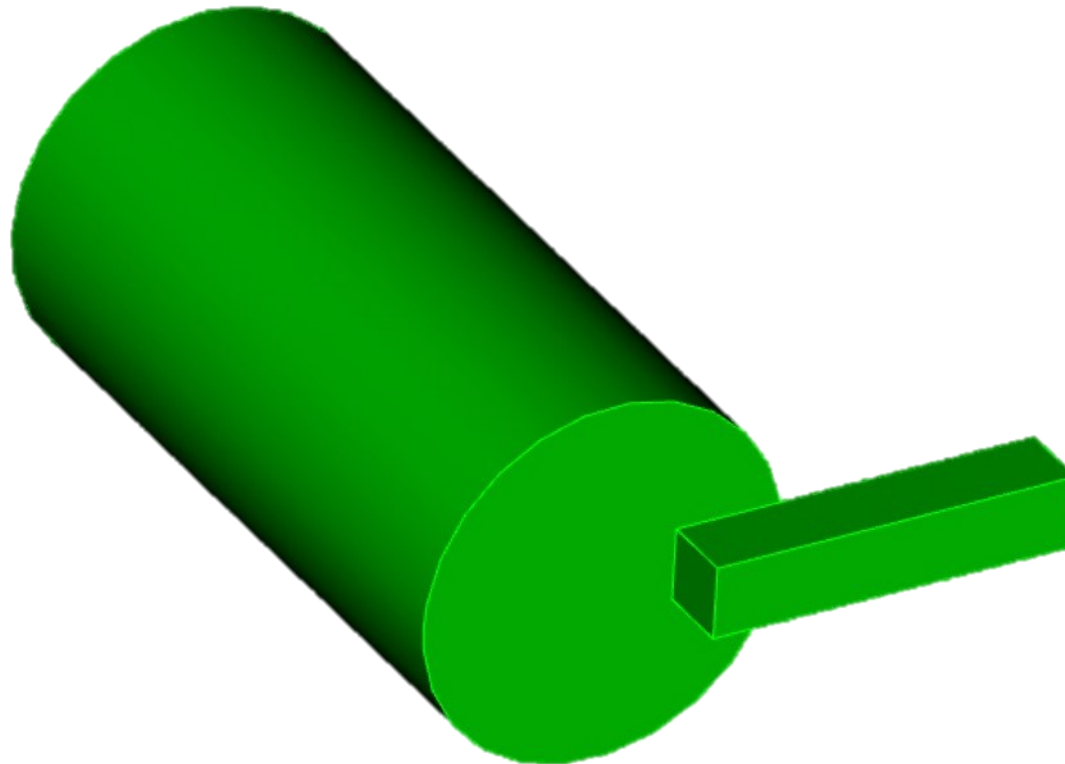


Restart Tutorial

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

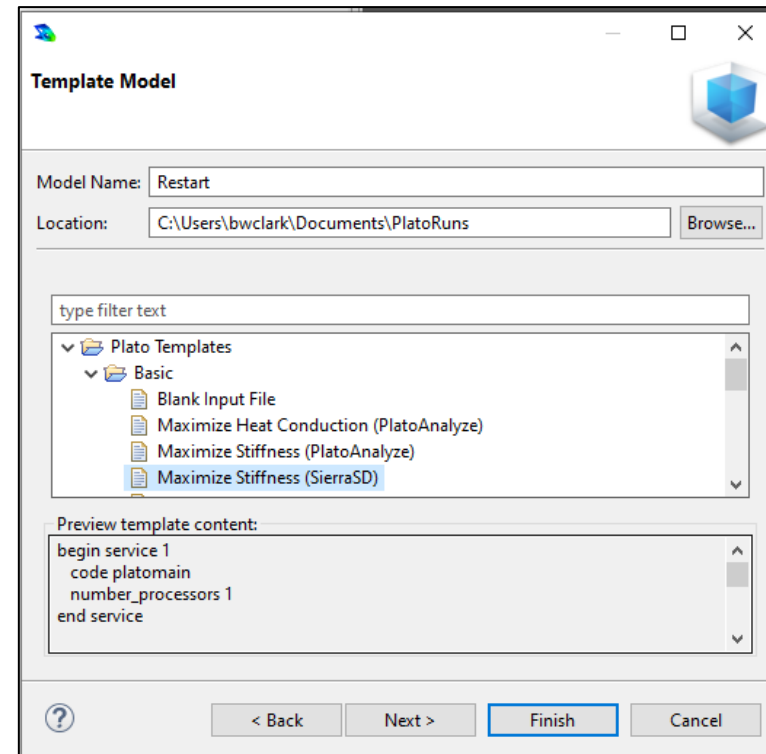
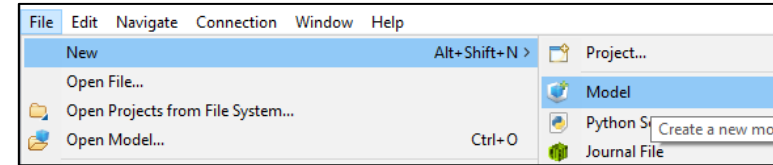
Problem Description

The problem will be a cylinder under a torsional-type loading condition using the geometry shown below as the design domain.



Create a New Model

- Choose **File->New->Model** in the menu
- Choose **New Model** then **Next**
- Choose **Create From Template** then **Next**
- Enter **Restart** as the **Model Name**
- Choose the **Plato Templates->Basic->Maximize Stiffness (SierraSD)** template and then **Finish**



Create the Design Domain

- At the **CUBIT** command prompt copy and paste the following commands (see Hint on next slide):

cylinder radius 5 height 20

brick x 10 y 2 z 2

vol 2 move 7 0 11

unite all

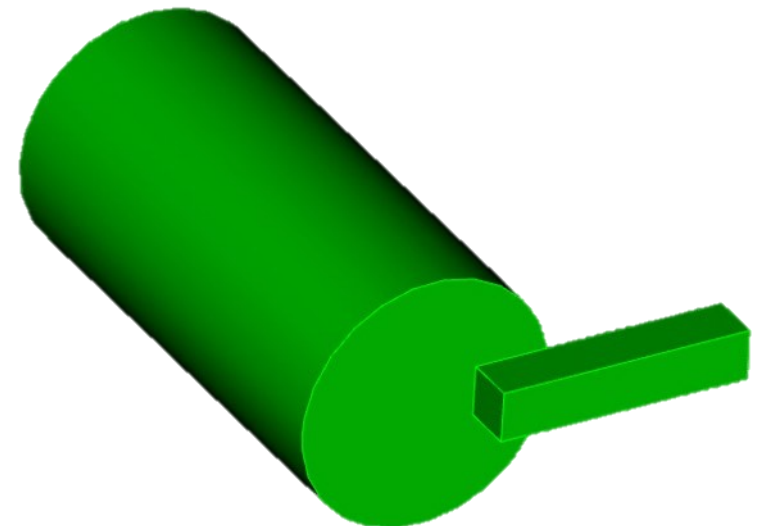
Console	Cubit Command History	Machines
---------	-----------------------	----------

```

CUBIT Console

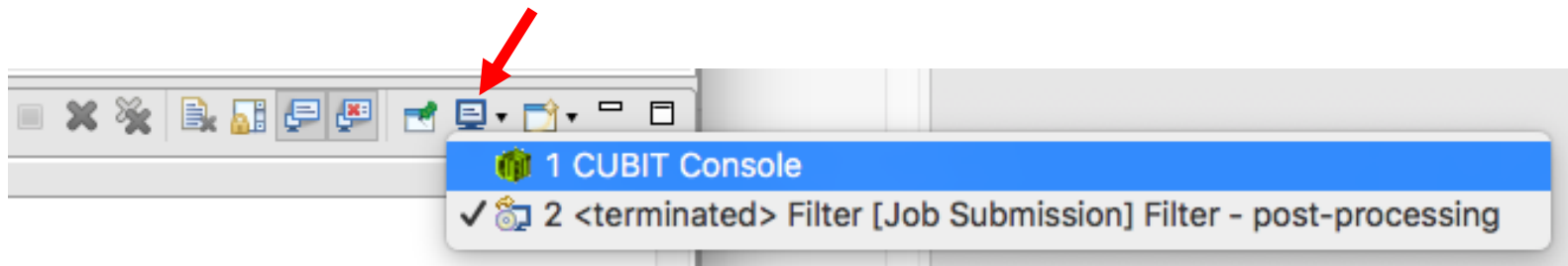
CUBIT> cylinder radius 5 height 20
brick x 10 y 2 z 2
vol 2 move 7 0 11
unite all
Successfully created cylinder volume 1
Journaled Command: cylinder radius 5 height 20

Successfully created brick volume 2
Journaled Command: brick x 10 y 2 z 2
  
```



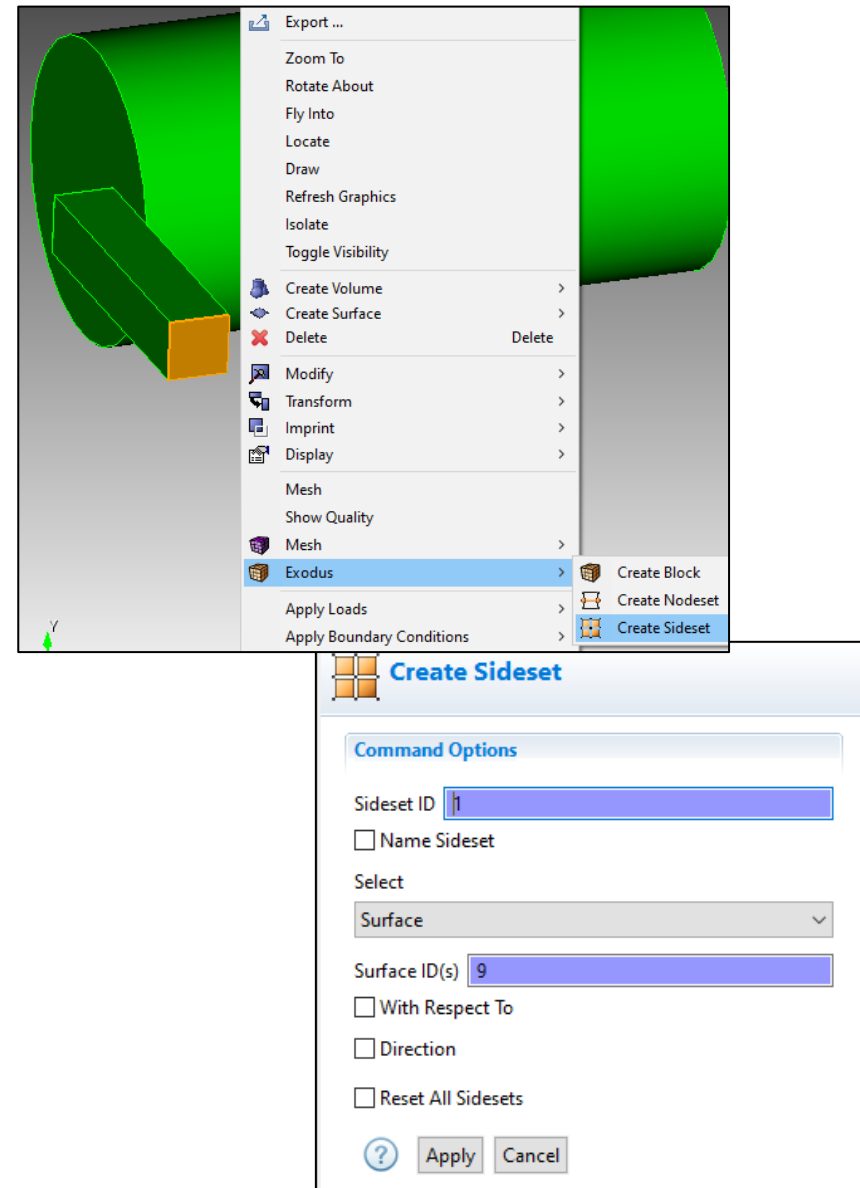
Hint: Cubit Console

- If you don't see the **CUBIT>** prompt in the console window you are just looking at the wrong console (there may be multiple to choose from loaded in Plato that use the single console window). Click on the icon at the top right of the console window that looks like a computer screen to toggle through the different consoles that are currently loaded. Or you can click on the down arrow next to the computer screen icon to see all of the currently loaded consoles and choose one from the list.



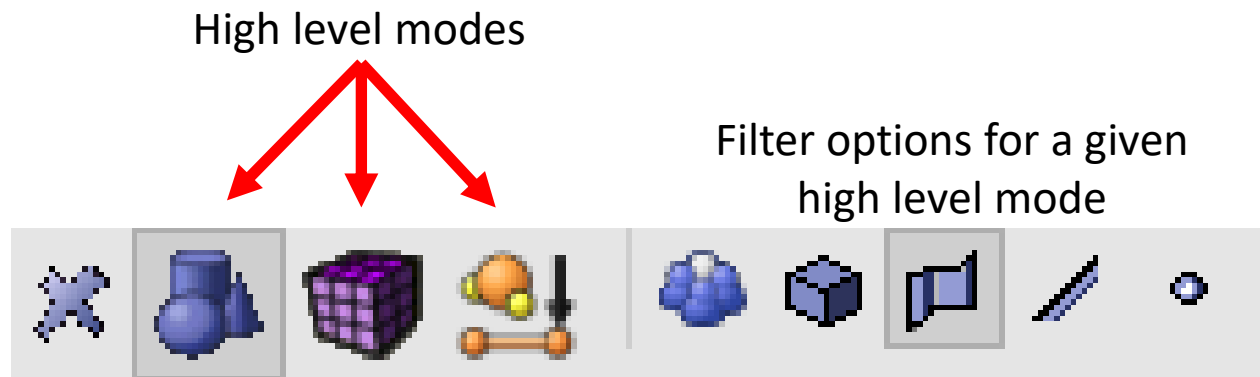
Define Load Application Location

- Select and right-click the surface (see Hint on next slide) on the end of the rectangular arm and choose **Exodus->Create Sideset** (Surface is highlighted in orange).
- In the Create Sideset panel click the “Apply” button to accept the default values and create the sideset. Note that the id of the new sidset is 1. We will use this later on when creating the actual load that is applied to this sideset.



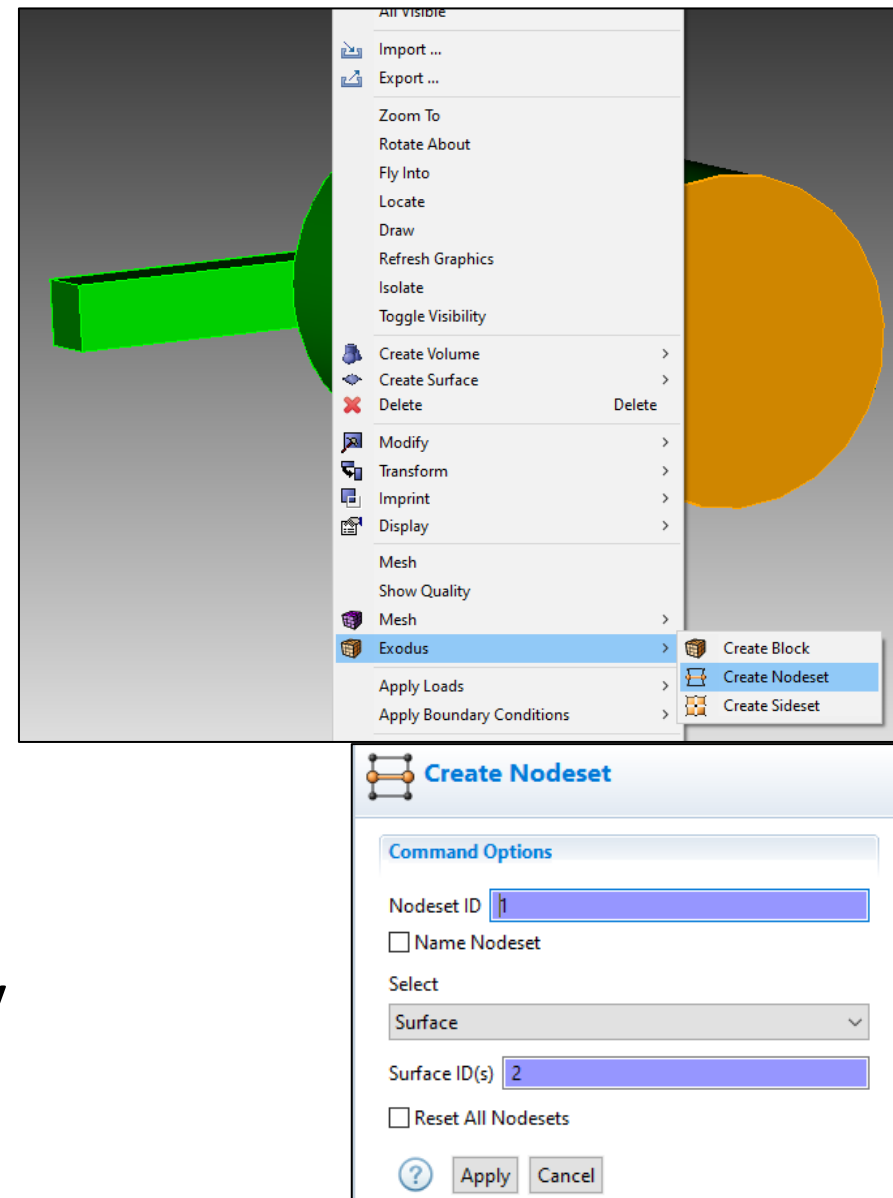
Hint: Entity Selection

- The selection filter toolbar is at the bottom left of the graphics window.
- There are 3 high level modes: Geometry, Mesh, and Exodus
- Within each high level mode there are various options for filtering what will get selected by clicks in the graphics window.



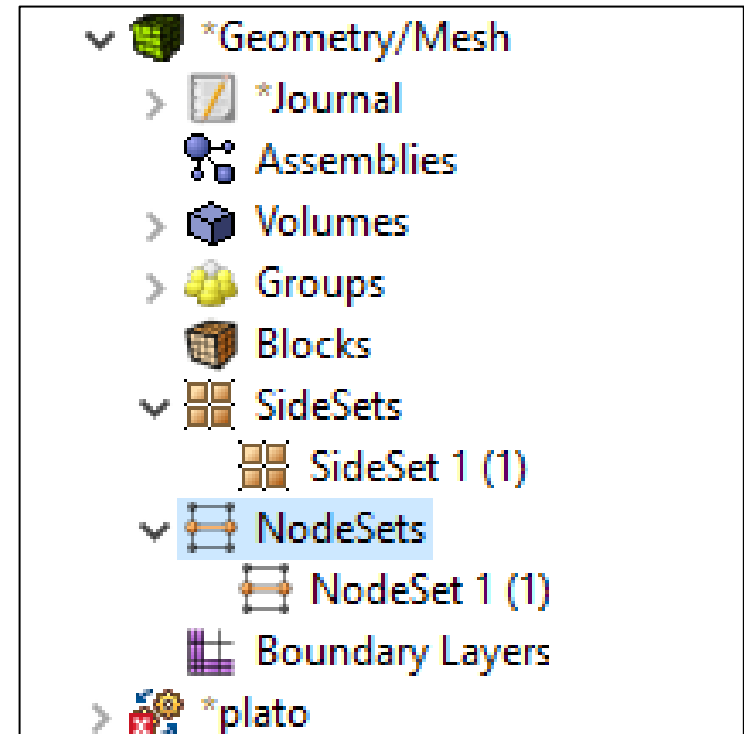
Define BC Application Location

- Select the circular surface on the end of the cylinder that doesn't have the load applied. Right-click and choose **Exodus->Create Nodeset**.
- In the "Create Nodeset" panel click the "Apply" button to accept the defaults and create the nodeset. Note the nodeset id is 1. We will use this in creating the fixed boundary condition.

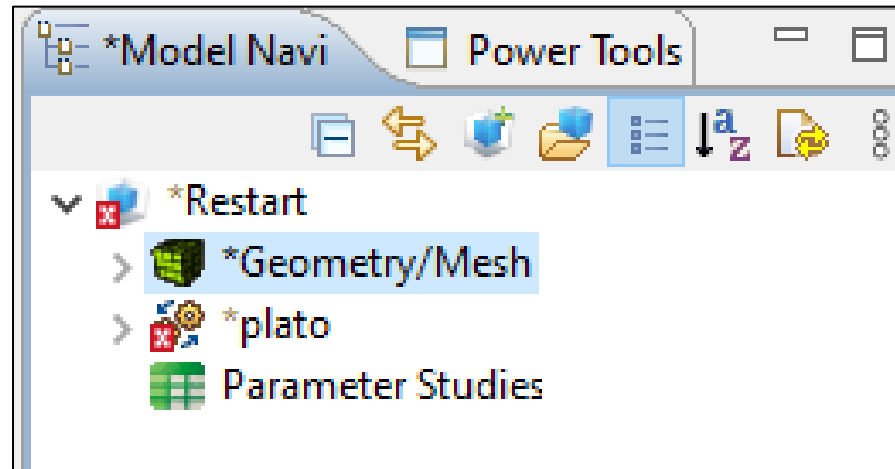


Sidesets and Nodeset in the Model Tree

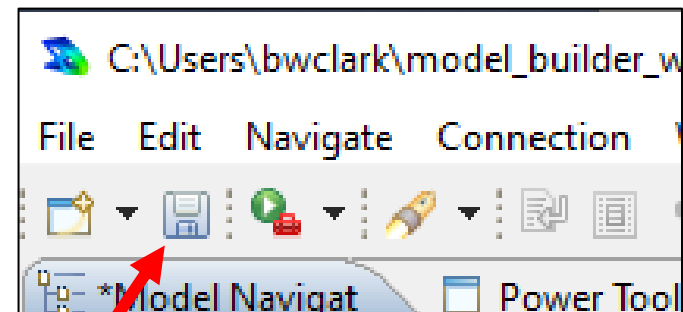
- The sideset and nodeset we just created can be seen in the Model tree under the “Geometry/Mesh” node.



Save the Geometry/Mesh

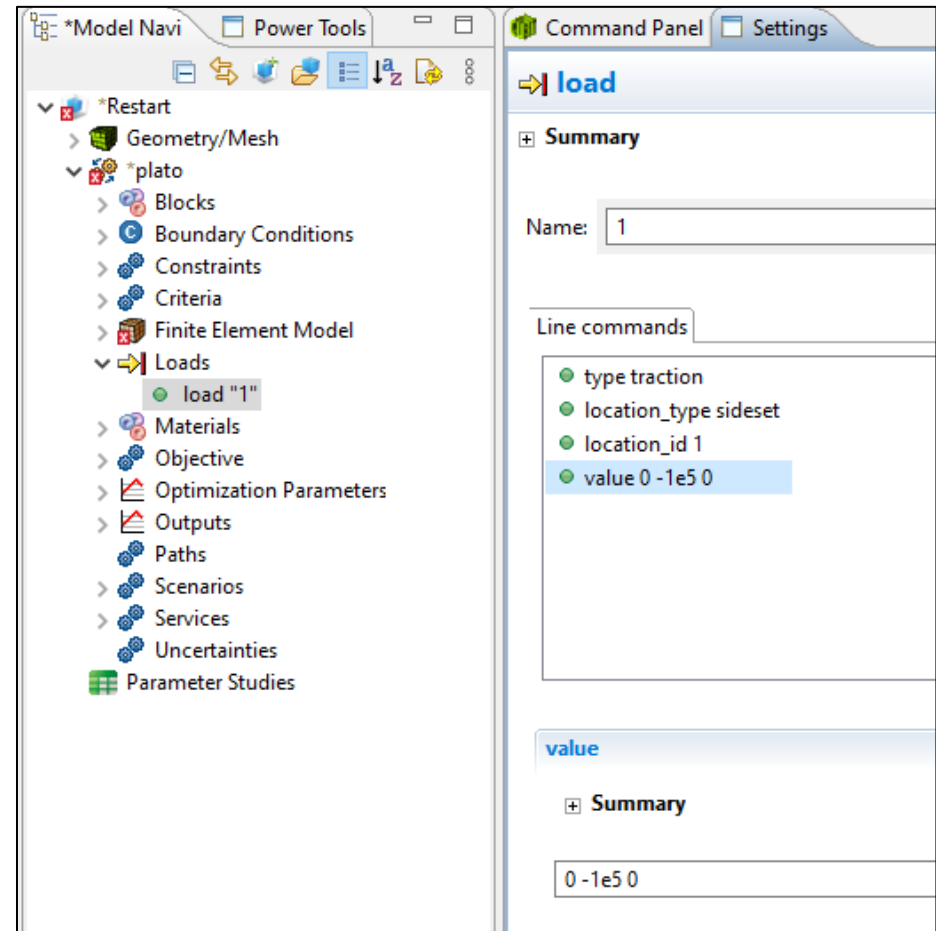


- Click on the “Geometry/Mesh” node in the tree and then click the Save icon in the toolbar.



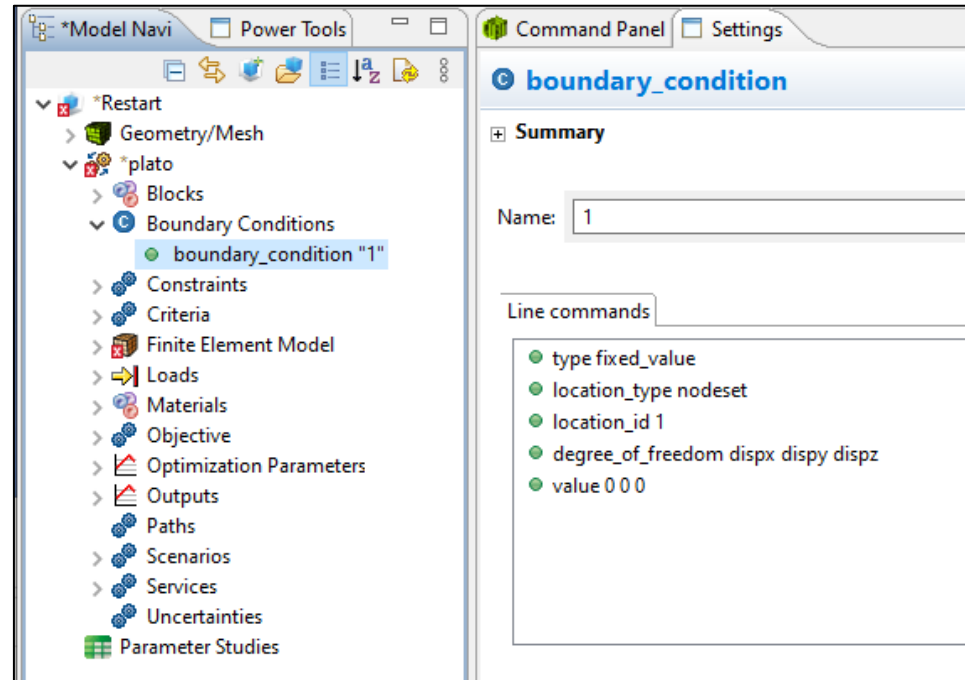
Create a Traction Load

- Expand the “plato” node in the Model tree and then expand the “Loads” node. In the template we used for this model there was already a load defined. Click on “load 1” in the tree to bring up its information in the Settings panel. The “location_id” is already set to “1” meaning it will be applied to sideset 1. Change the value to “0 -1e5 0” indicating there will be a traction load with a negative y component of 1e5 applied to sideset 1.
- Note: If the template did NOT already have a load defined, we could have created one by clicking on the “Load” hyperlink that shows up in the Settings panel when you click on the “Loads” node in the tree.



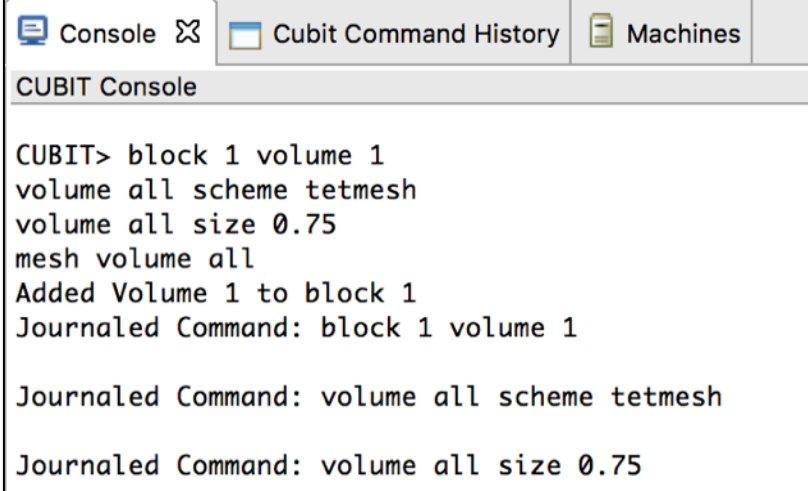
Create a Fixed BC

- Expand the “plato” node in the Model tree and then expand the “Boundary Conditions” node. In the template we used for this model there was already a boundary condition defined. Click on “boundary_condition 1” in the tree to bring up its information in the Settings panel. The “location_id” is already set to “1” meaning it will be applied to nodeset 1. The “degree_of_freedom” and “value” parameters specify that displacement in all 3 directions will be fixed so there is nothing to change.
- Note: If the template did NOT already have a boundary condition defined, we could have created one by clicking on the “Boundary condition” hyperlink that shows up in the Settings panel when you click on the “Boundary Conditions” node in the tree.



Define Blocks and Mesh Domain

- At the **CUBIT** prompt copy and paste the following commands:
 block 1 volume 1
 volume all scheme tetmesh
 volume all size 0.75
 mesh volume all
- Click on **Geometry/Mesh** node in the tree and then click the **Save** icon in the toolbar to save the model

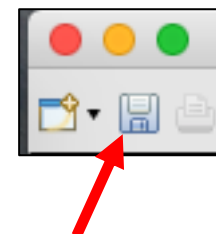
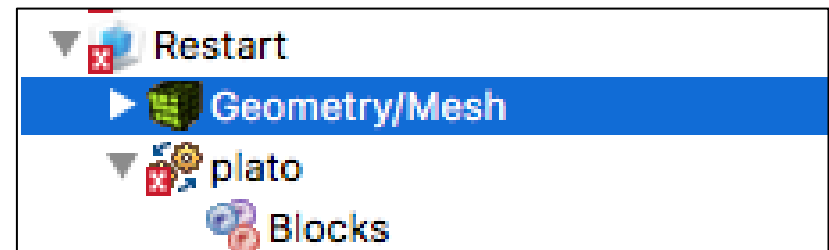


The screenshot shows the CUBIT Console window with tabs for Console, Cubit Command History, and Machines. The console displays the following commands and their journalized versions:

```
CUBIT> block 1 volume 1
volume all scheme tetmesh
volume all size 0.75
mesh volume all
Added Volume 1 to block 1
Journaled Command: block 1 volume 1

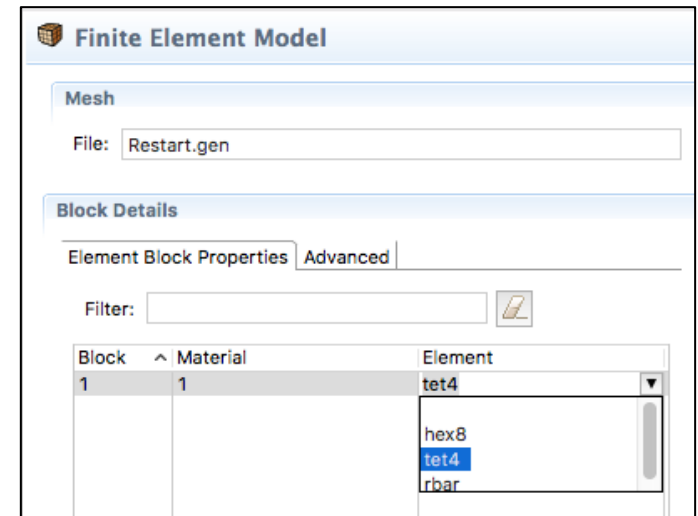
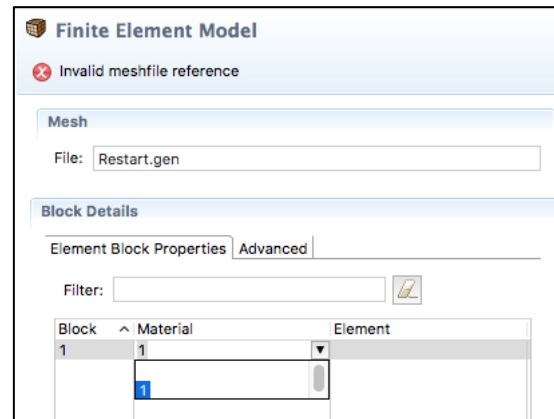
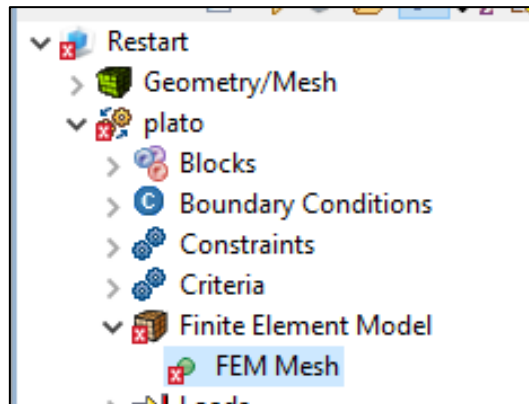
Journaled Command: volume all scheme tetmesh

Journaled Command: volume all size 0.75
```



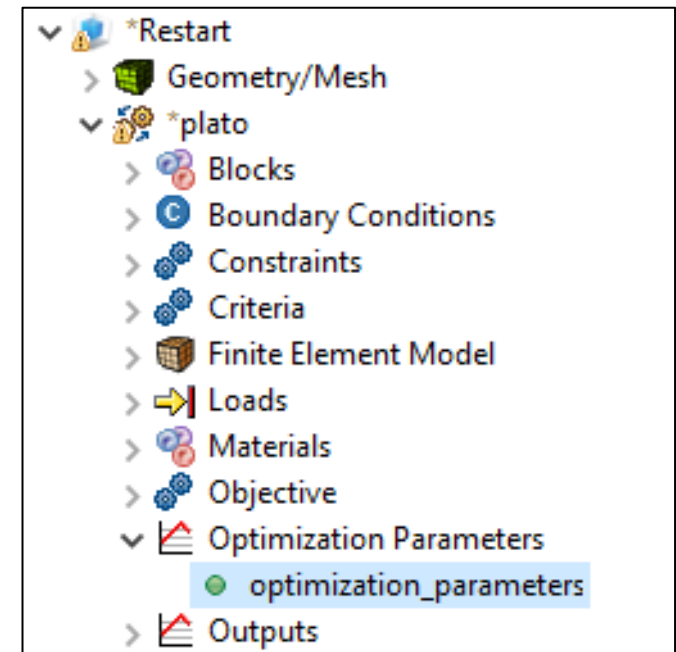
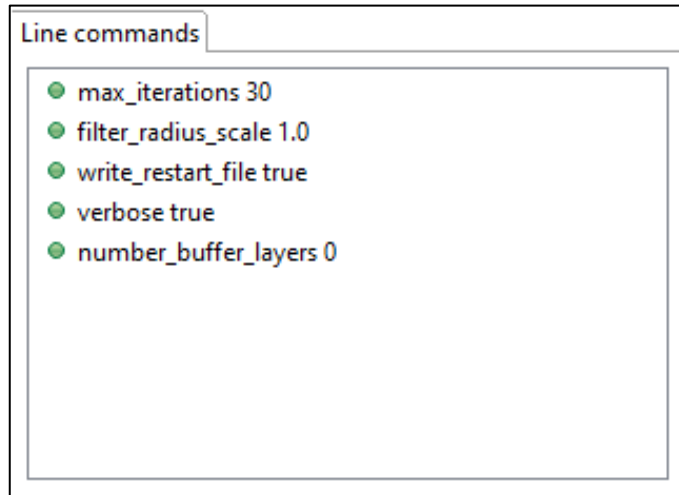
Assign Material and Element Type

- 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



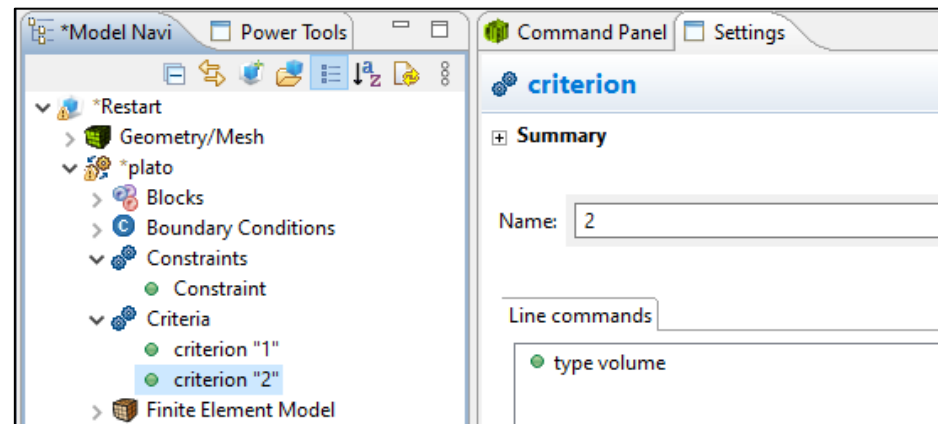
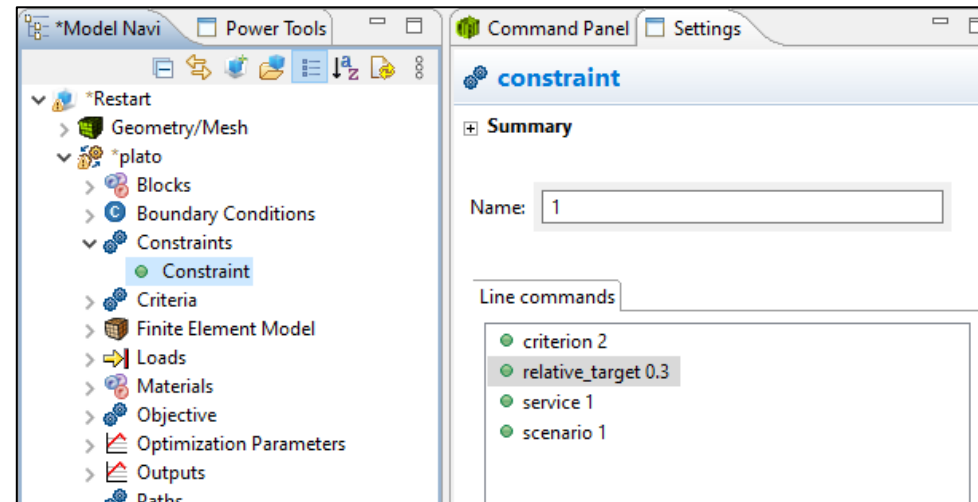
Set the Top. Opt. Parameters

- Click on the **Optimization Parameters->optimization_parameters** node in the tree.
- In the **Settings** panel set the following parameters to the indicated values. These parameters will specify 1) how much filtering or smoothing will take place (see Filter tutorial for more on filtering), and 2) and how many iterations we want Plato to run, respectively.
 - **filter_radius_scale = 1.0**
 - **max_iterations = 30**



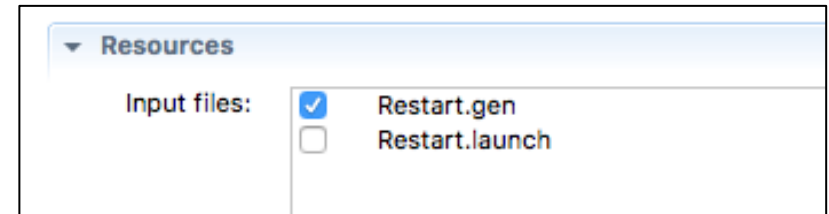
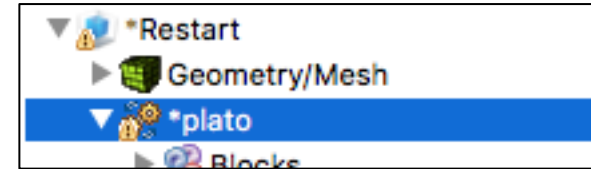
Set the Volume Fraction Constraint

- Click on the **Constraints->Constraint** node in the tree.
- Change the **relative_target** to be **0.3**. This will indicate that we want the final design to only use 30% of the original volume.
- Also, verify that the criterion being used in the constraint is a volume criterion by clicking on **Criteria->criterion "2"** in the tree (see bottom image on right).



Run the Optimization

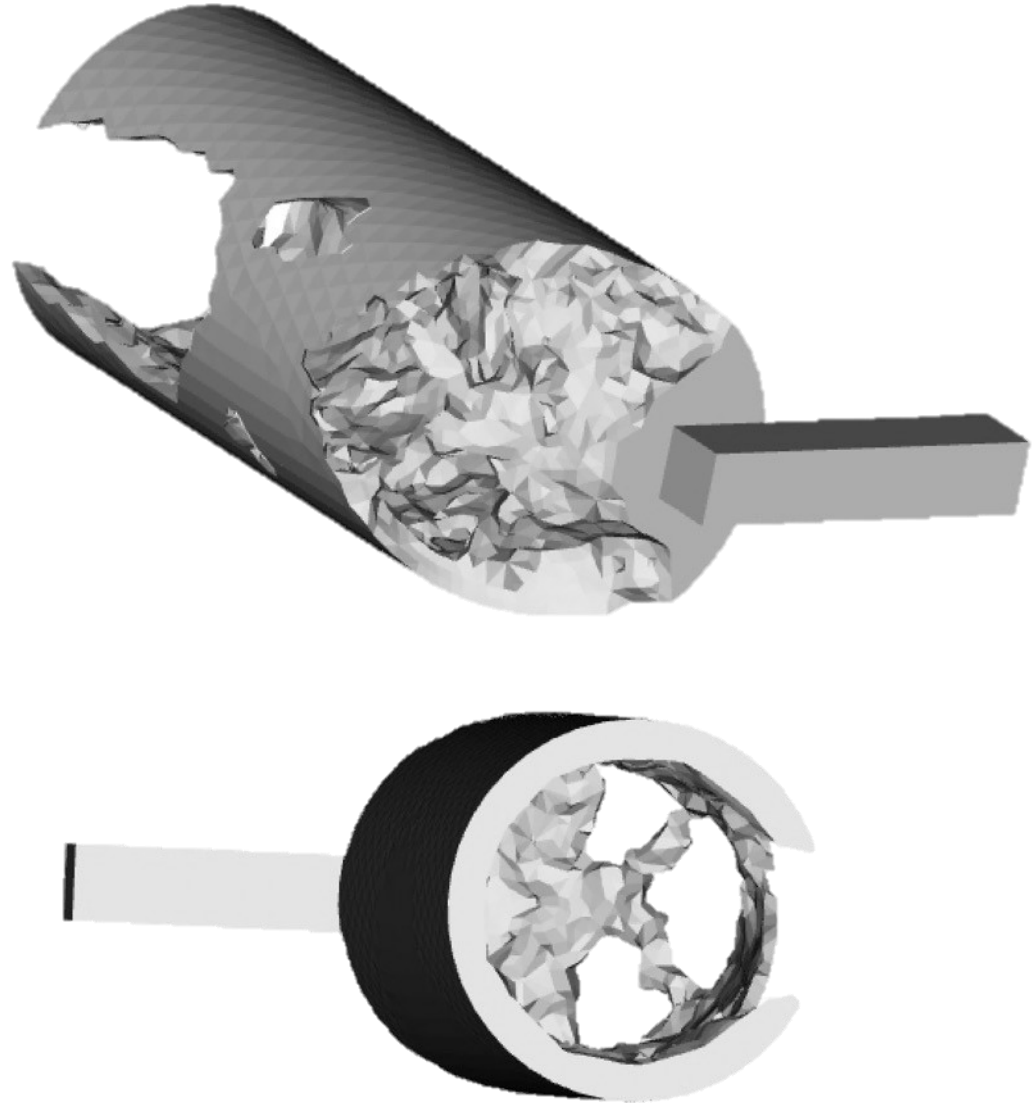
- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view
- Choose **Plato** as the code and then choose the machine and execution template you want to use



- In the **Resources** area make sure **Restart.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

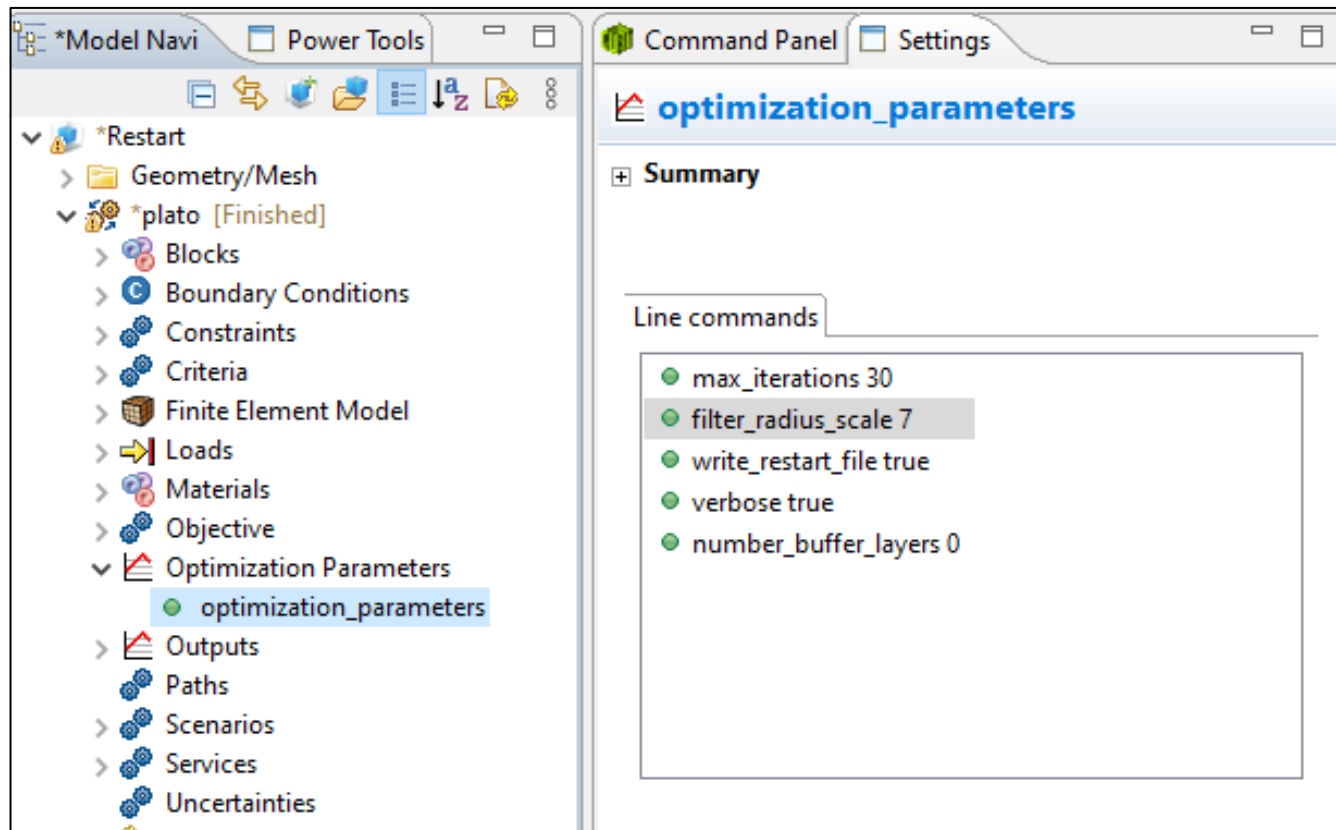
Initial Results

The initial results should look something like that shown here. Note how rough the results look because of the low **filter_radius_scale** value we used. We will restart the problem from the last iteration with a larger filter radius to smooth the design out.



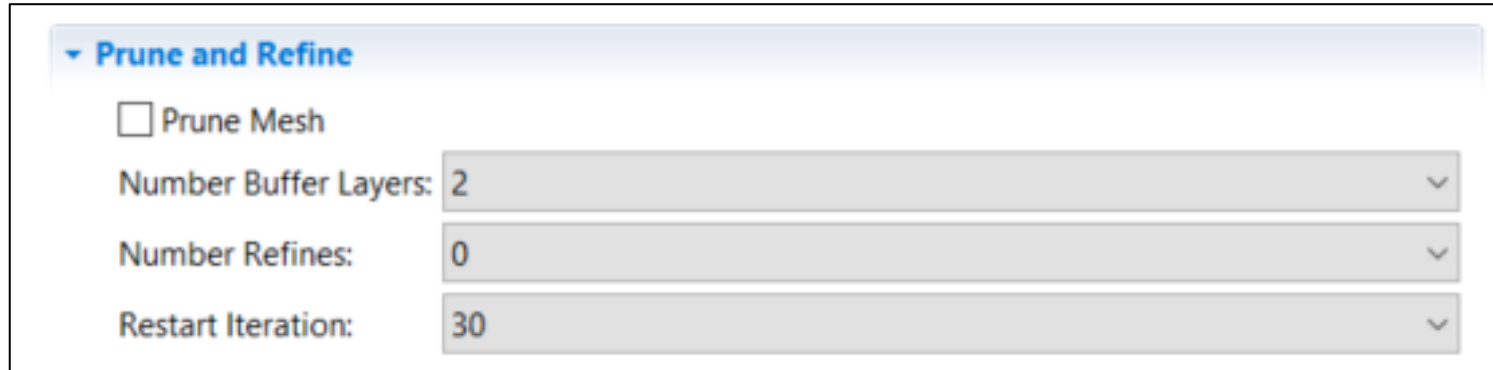
Change Parameters

- First, click on the **Optimization Parameters->optimization_parameters** node in the tree and change the **filter_radius_scale** parameter to **7.0**. This will cause the filtering/smoothing to be very aggressive.



Launch the Restart Run

- Bring up the job submission panel by clicking on the **plato** node in the tree. Under **Prune and Refine**, change the restart iteration to 30. Click on **Submit Simulation Job** to rerun the job. Plato will use the results from the iteration you specified as the initial guess for the next run and it will use the new **filter_radius_scale** value we specified. Results from the previous run will be overwritten.

A screenshot of a software interface showing the 'Prune and Refine' settings panel. The panel has a light blue header with a downward arrow and the text 'Prune and Refine'. Below the header, there is a checkbox labeled 'Prune Mesh' which is currently unchecked. Underneath the checkbox are three input fields, each with a label on the left and a value in a grey box on the right, followed by a small downward arrow. The first field is 'Number Buffer Layers' with the value '2'. The second field is 'Number Refines' with the value '0'. The third field is 'Restart Iteration' with the value '30'.

▼ Prune and Refine	
<input type="checkbox"/> Prune Mesh	
Number Buffer Layers:	2
Number Refines:	0
Restart Iteration:	30

Results after Restarting

Your result should look something like that shown below. Because we used such a large filter radius some of the previous fine features were filtered or smoothed out and the remaining features are very smooth. Any parameter can be changed when doing a restart—even the number of processors. This restart feature allows you to incrementally explore changes to input parameters and interact with the optimizer.

