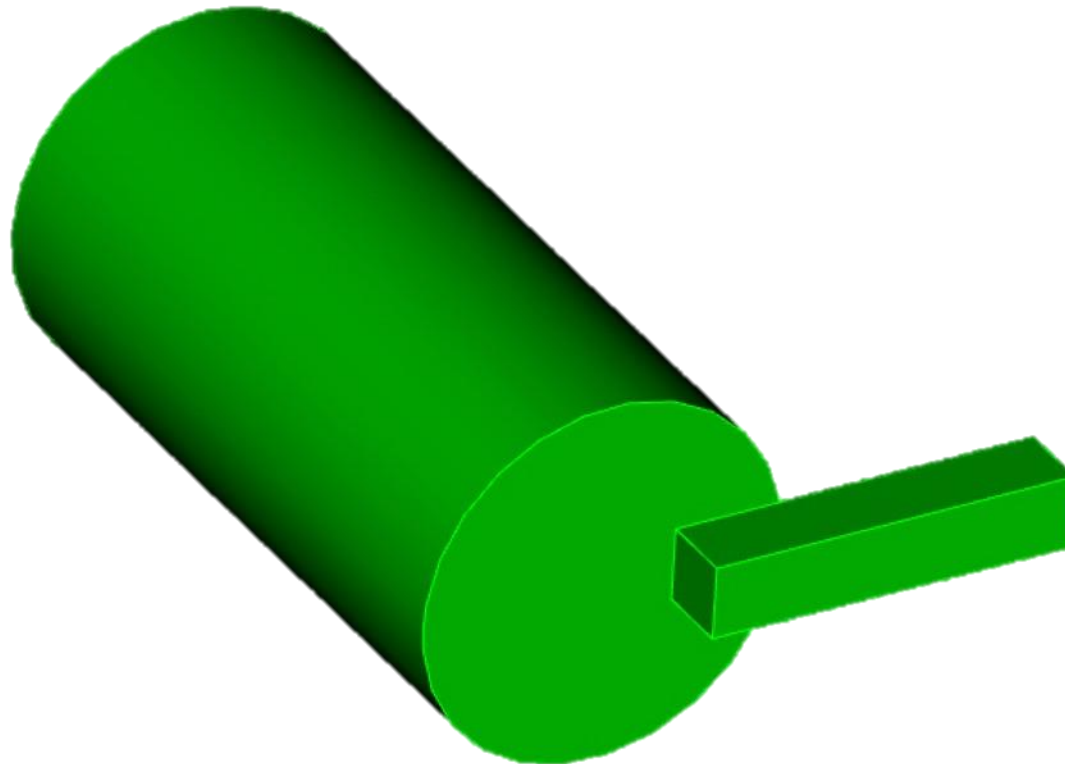


Restart Tutorial

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. SAND2020-10947 TR

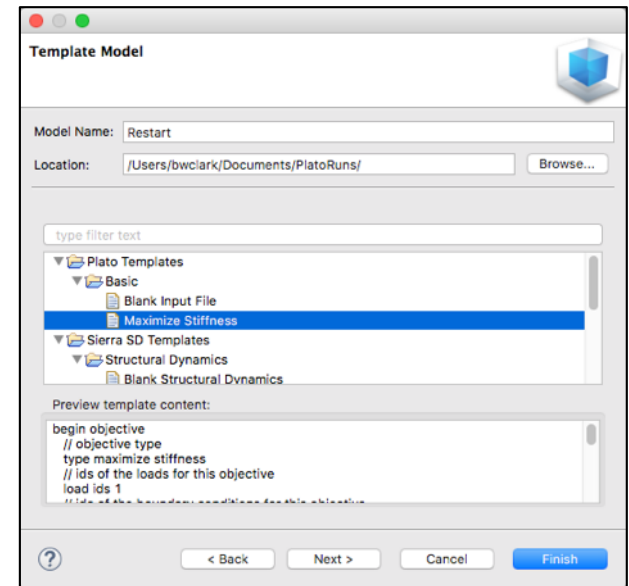
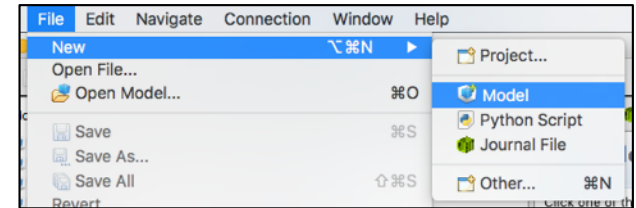
Problem Description

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** 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

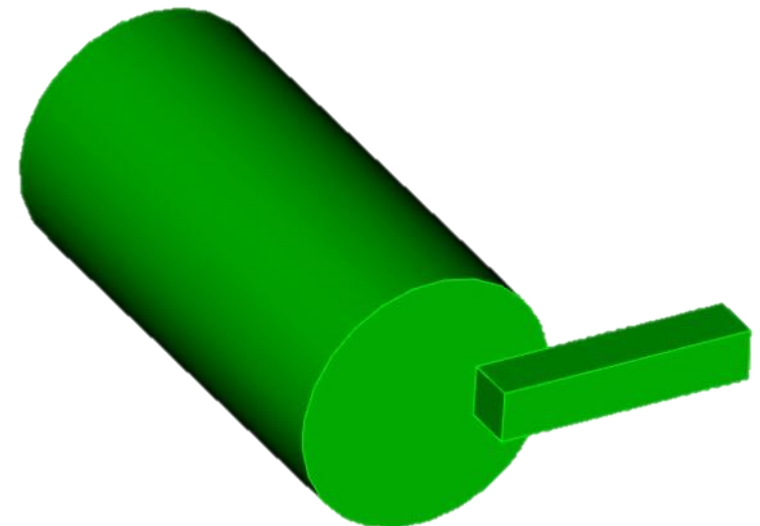
vol 2 move 7 0 11

unite all

```

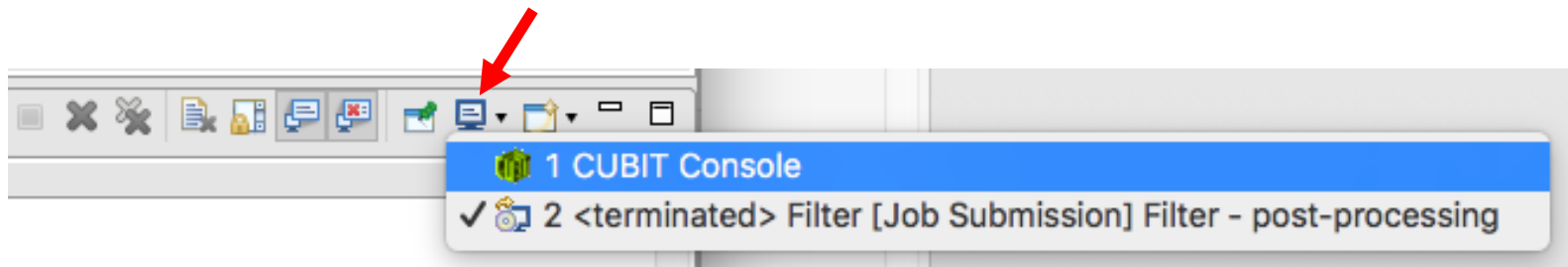
CUBIT Console
CUBIT> cylinder radius 5 height 20
brick x 10 y 2 z 2
vol 2 move 7 0 11
unite all
Successfully created cylinder volume 1
Journaled Command: cylinder radius 5 height 20

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



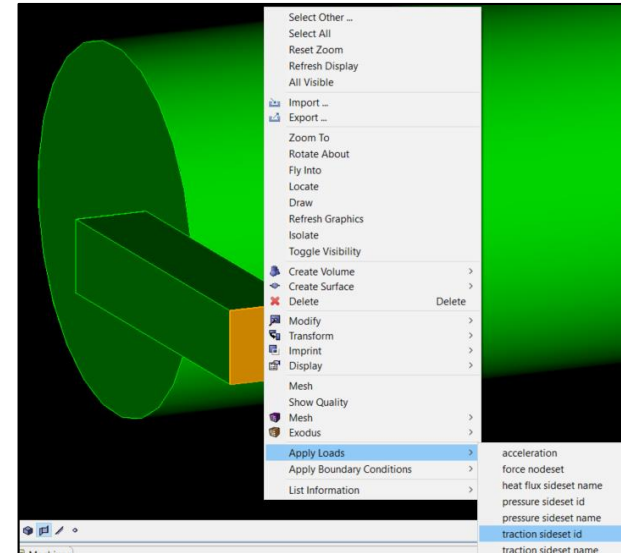
Hint: Cubit Console

- 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.



Apply a Traction Load

- Select and right-click the surface (see Hint on next slide) on the end of the rectangular arm and choose **Apply Loads->traction sideset id** (Surface is highlighted in orange).
- In the **Settings** panel enter “**0 -1e5 0**” in the **value** box and hit the tab key. Enter “**1**” in the “**load id**” box and hit the return key.



traction sideset

Summary

1

value

0 -1e5 0

load id

1

Located in: loads

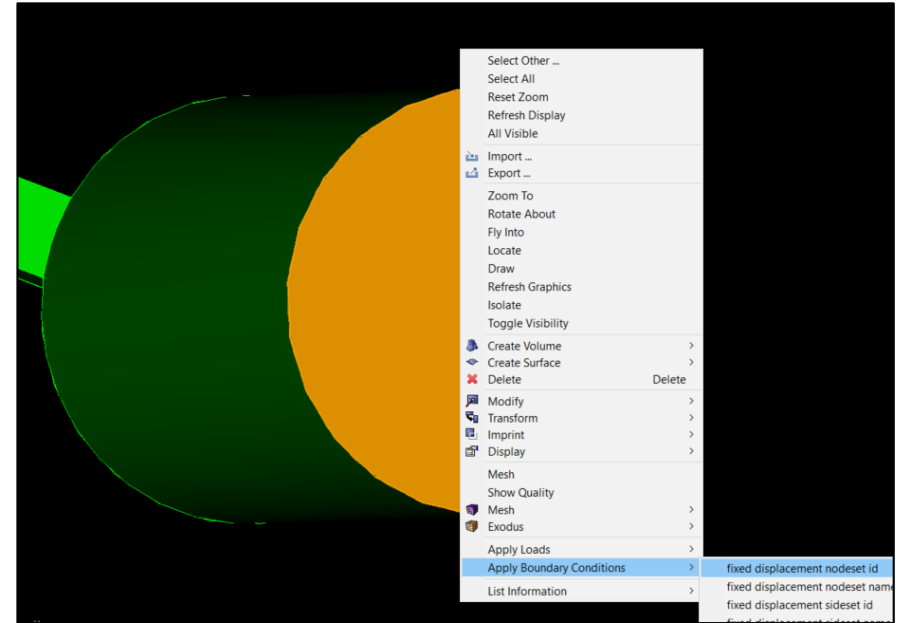
Hint: Entity Selection

- 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.



Create a Fixed BC

- Select the circular surface on the end of the cylinder that doesn't have the load applied. Right-click and choose **Apply Boundary Conditions->fixed displacement nodeset id**
- In the **Settings** panel enter “1” in the “**bc id**” box and hit return.



fixed displacement nodeset

+
Summary

1

☐

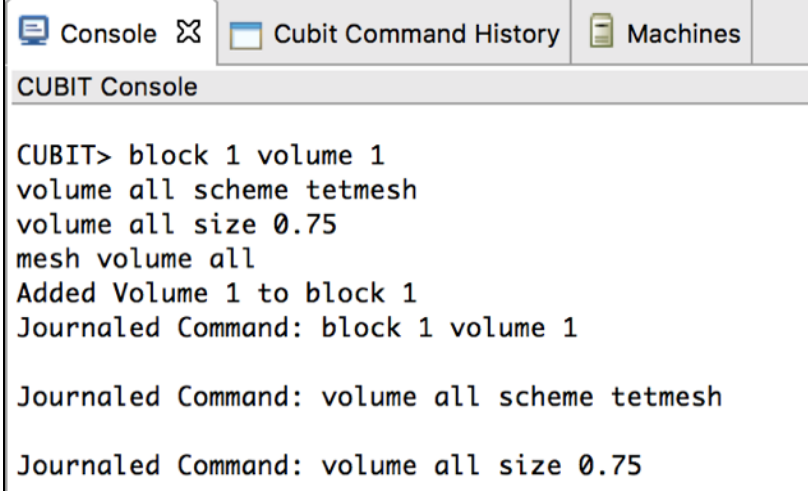
bc id

1

Located in:
boundary conditions

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

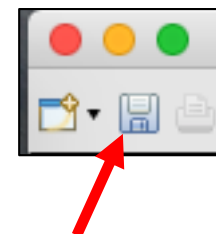
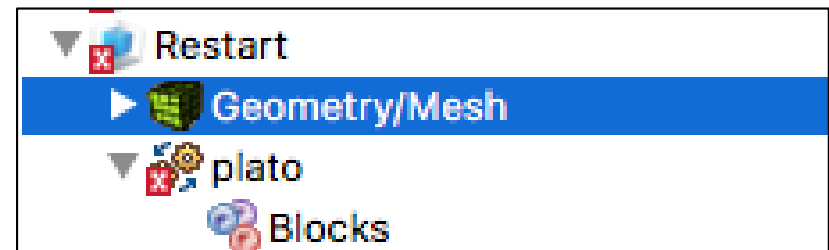


The screenshot shows the CUBIT Console window with the following text:

```
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

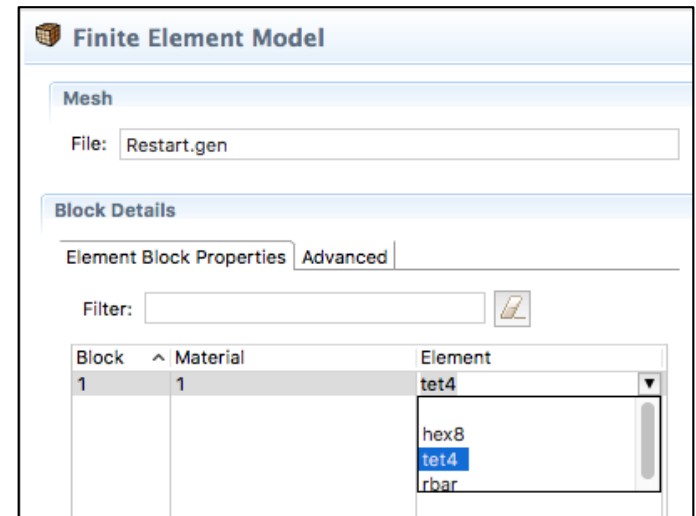
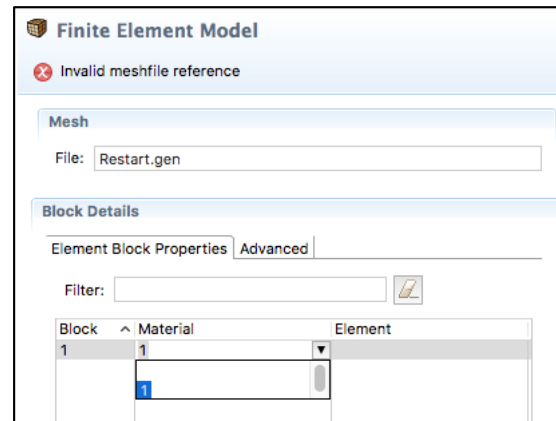
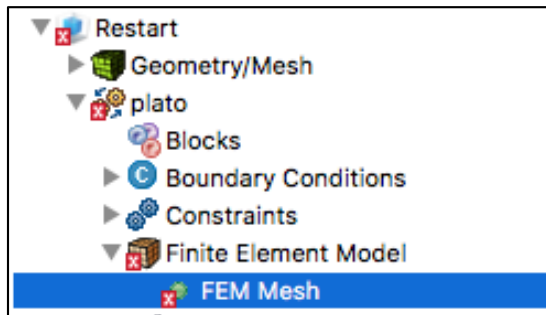
Journaled Command: volume all scheme tetmesh

Journaled Command: volume all size 0.75
```



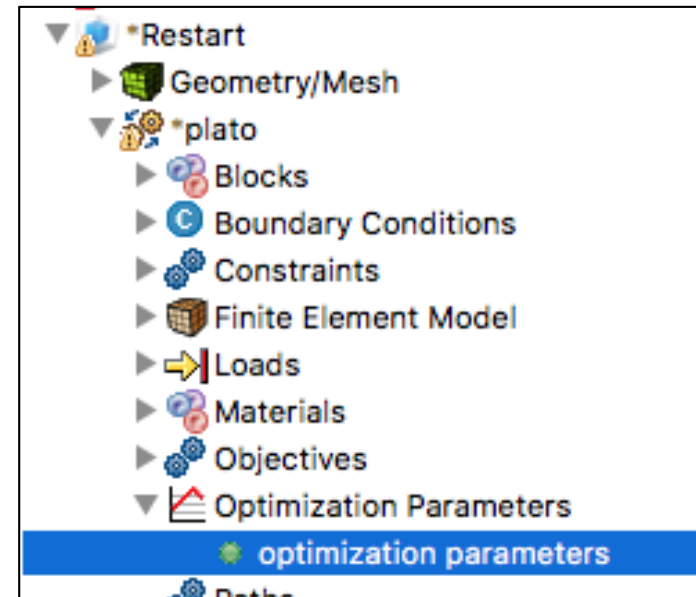
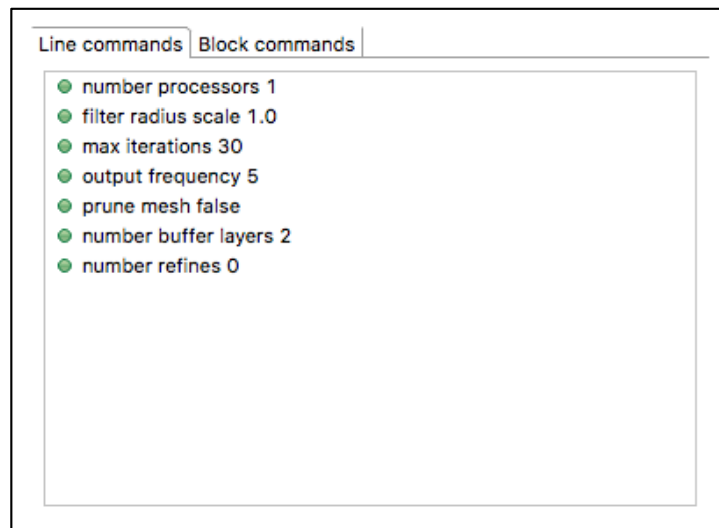
Assign Material and Element Type

- 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



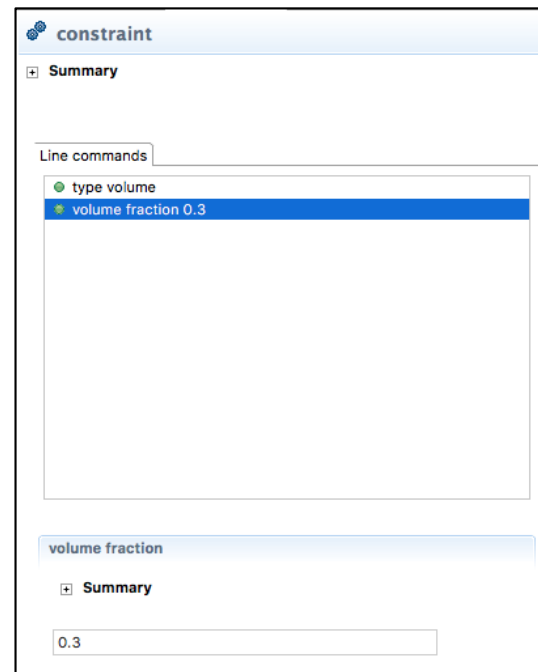
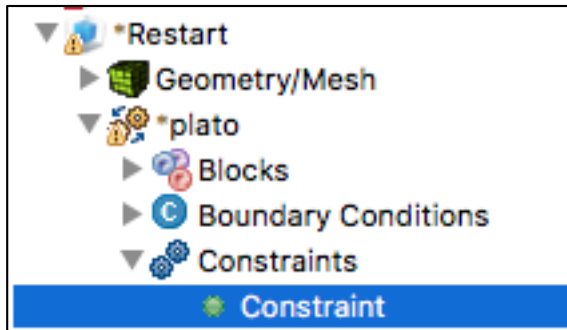
Set the Top. Opt. Parameters

- 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**



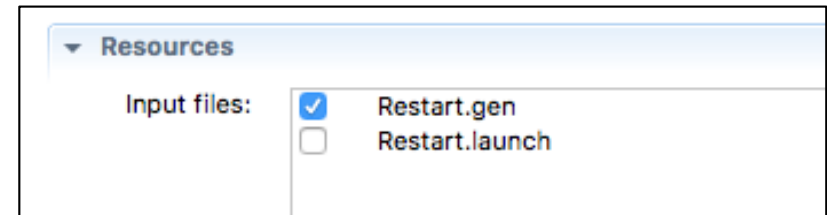
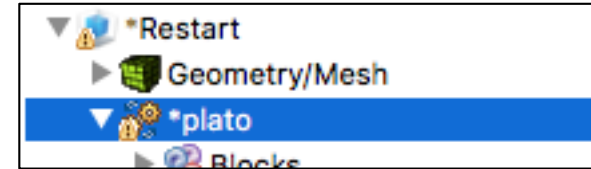
Set the Volume Constraint

- Click on the **Constraints->Constraint** node in the tree.
- In the **Settings** panel set the **volume fraction** parameter to be **0.3**. This will indicate that we want the final design to only use 30% of the original volume.



Run the Optimization

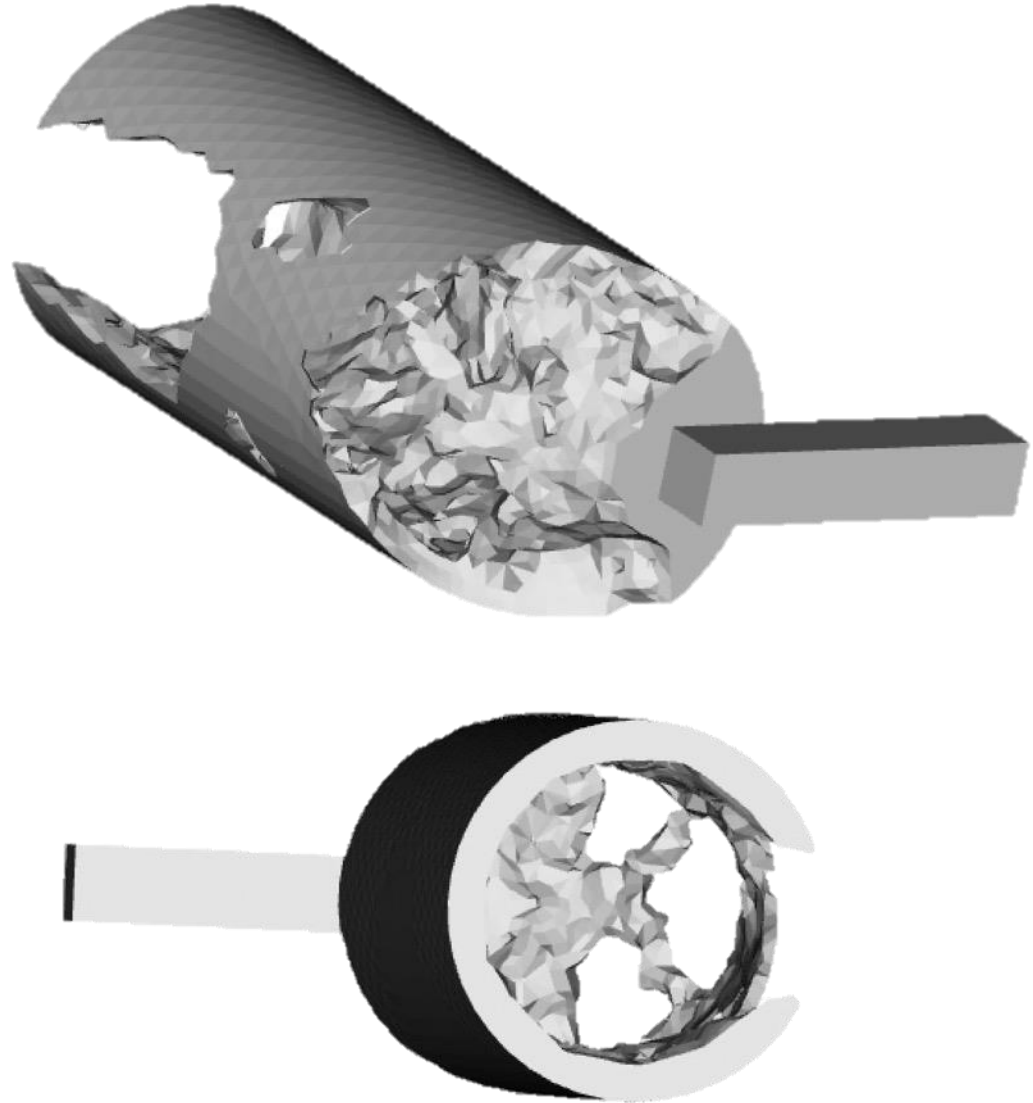
- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view
- Choose **Plato** as the code and then choose the machine and execution template you want to use



- 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

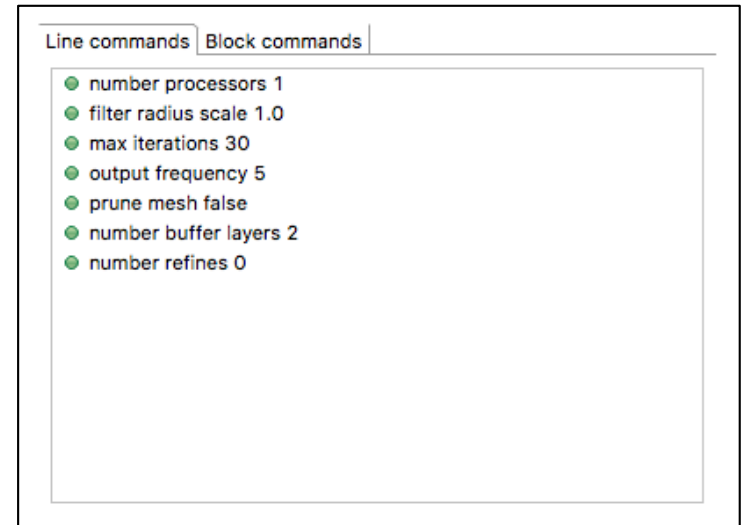
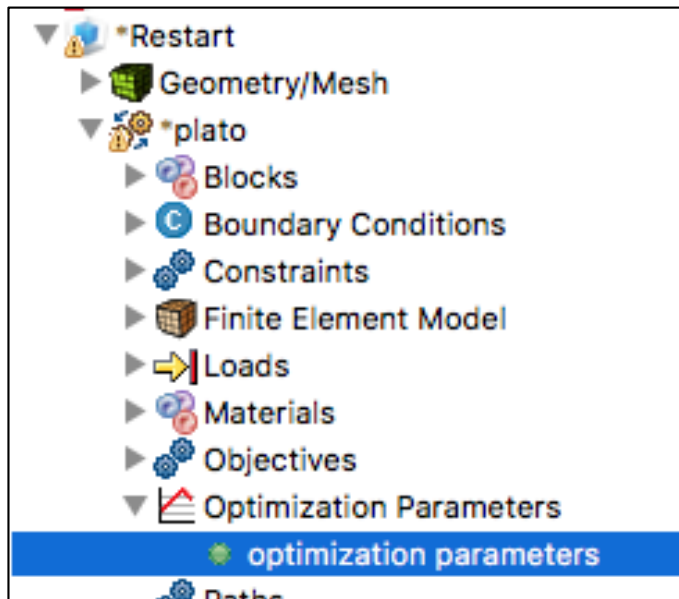
Initial Results

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.



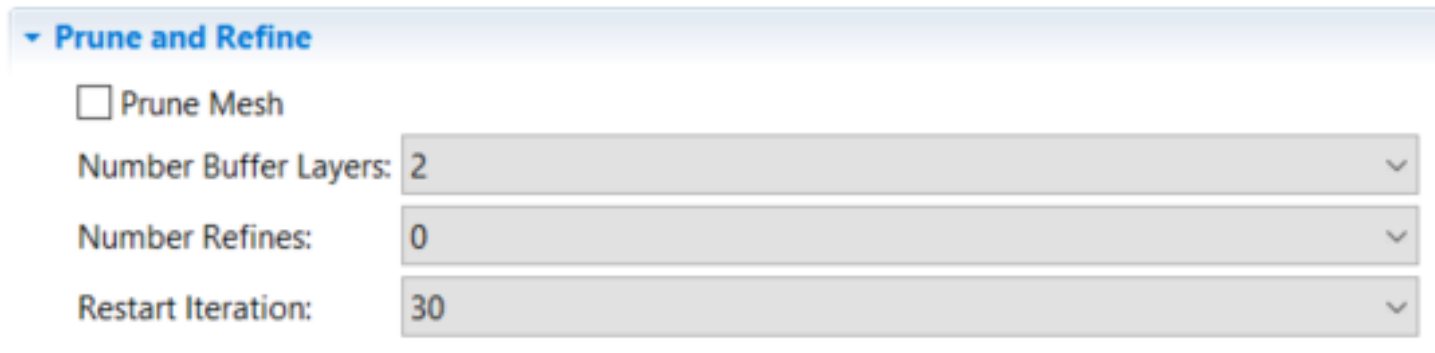
Change Parameters

- 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.

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 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.

