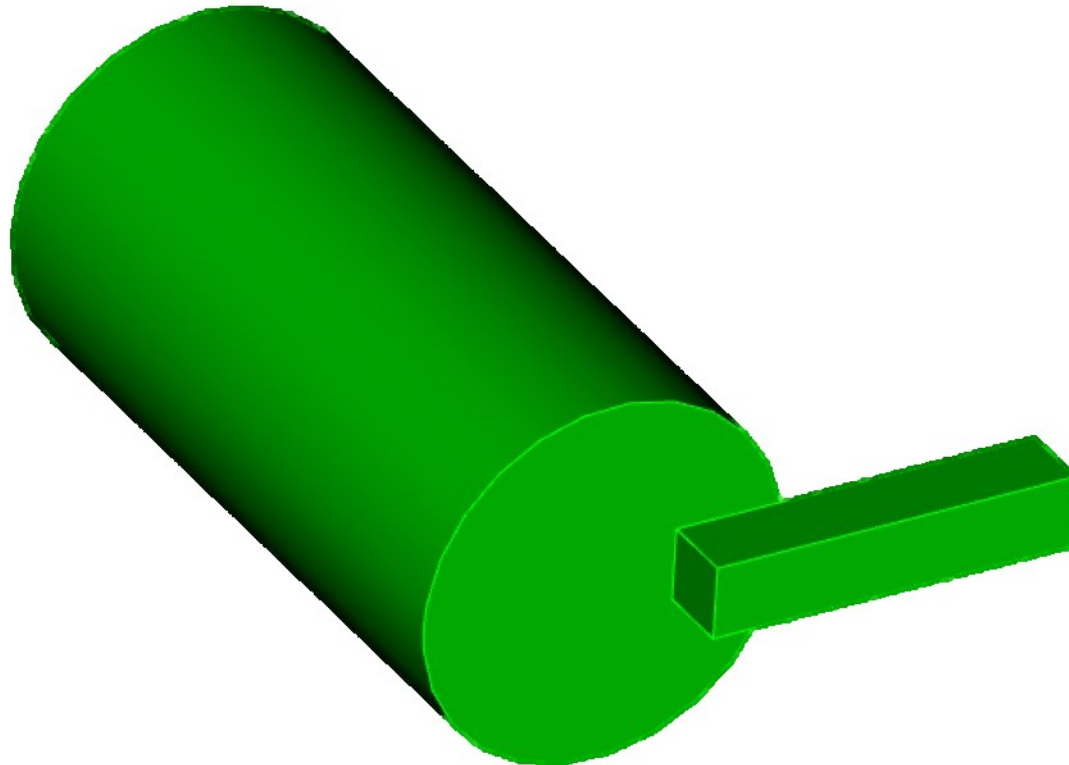


Prune and Refine Tutorial

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.

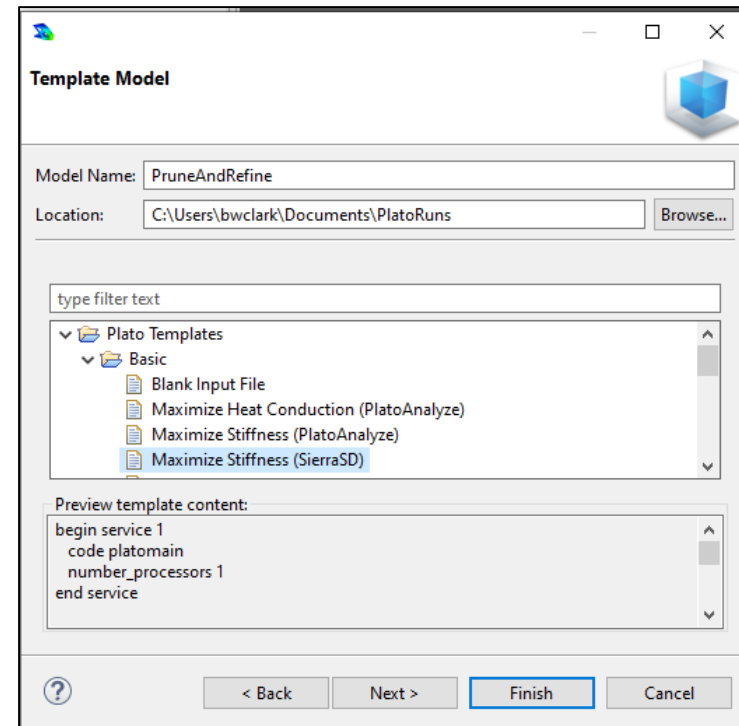
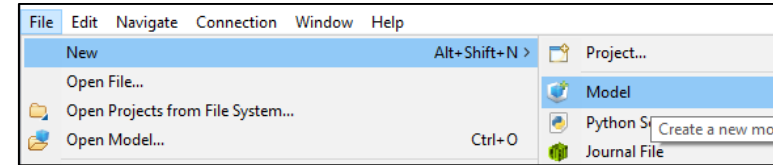
Problem Description

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 **PruneAndRefine** 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

webcut volume 1 with sheet extended from surface 1

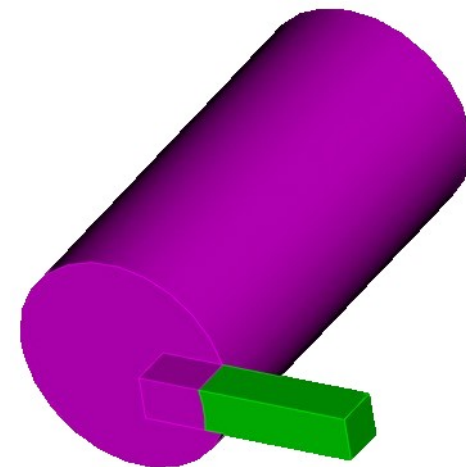
imprint all

merge all

```

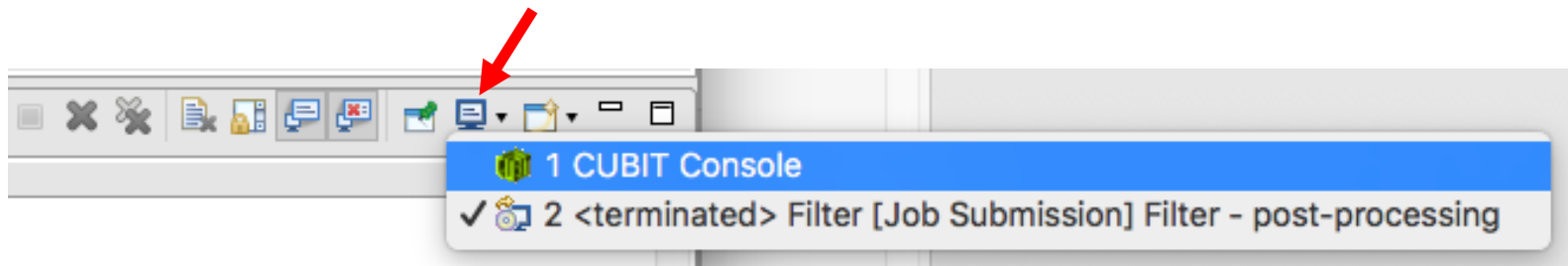
CUBIT Console
CUBIT> cylinder radius 5 height 20
brick x 10 y 2 z 2
vol 2 move 7 0 11
unite all
webcut volume 1 with sheet extended from surface 1
imprint all
merge allSuccessfully created cylinder volume 1
Journaled Command: cylinder radius 5 height 20

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



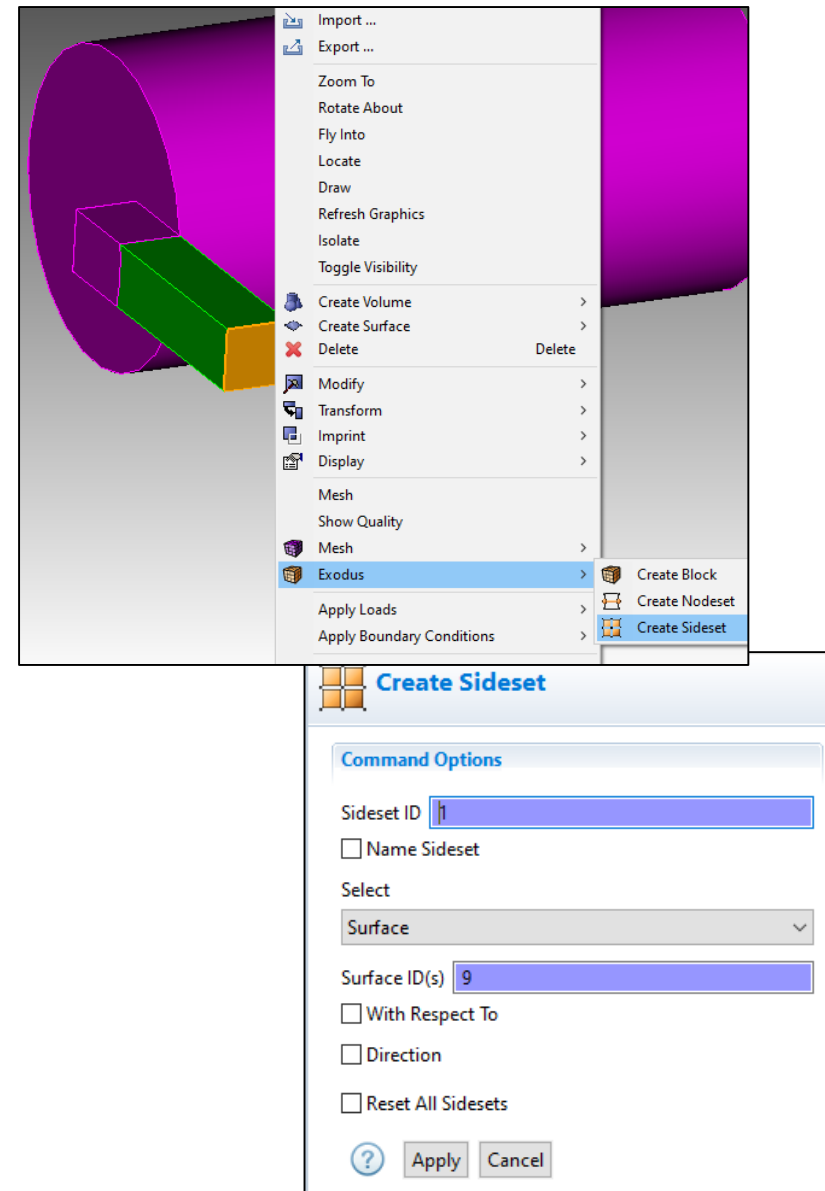
Hint: Cubit Console

- 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.



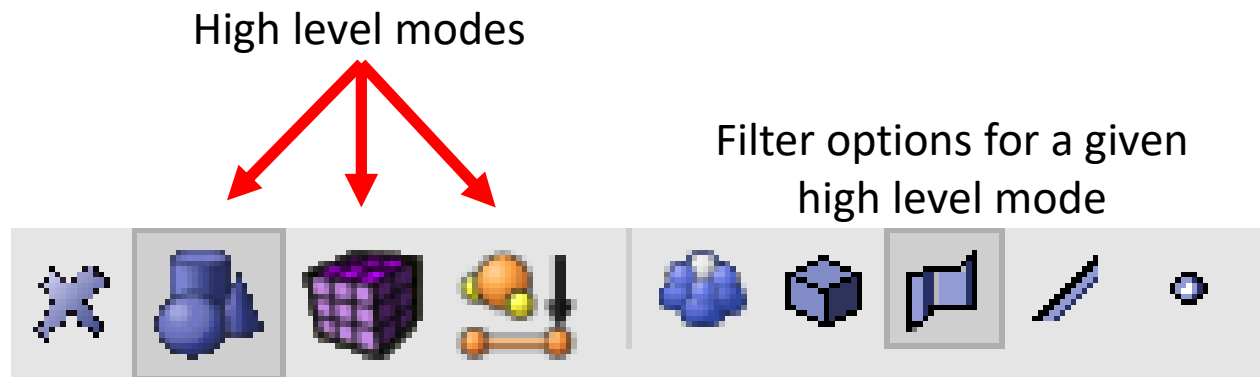
Define Load Application Location

- 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.



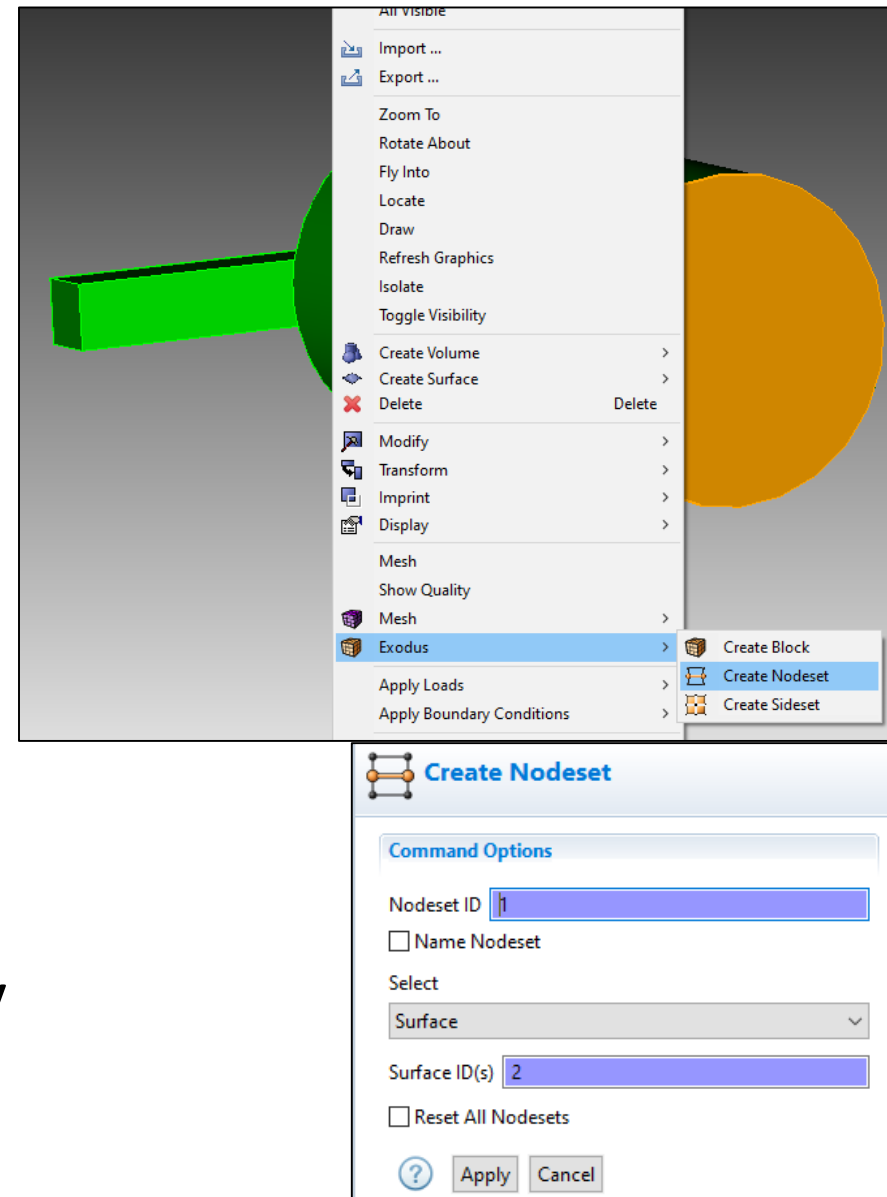
Hint: Entity Selection

- 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.



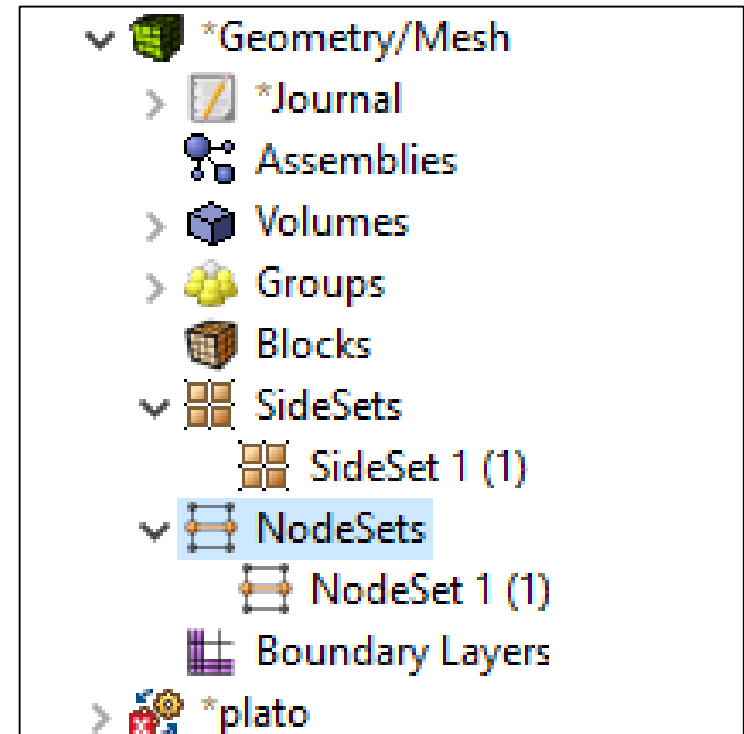
Define BC Application Location

- 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.

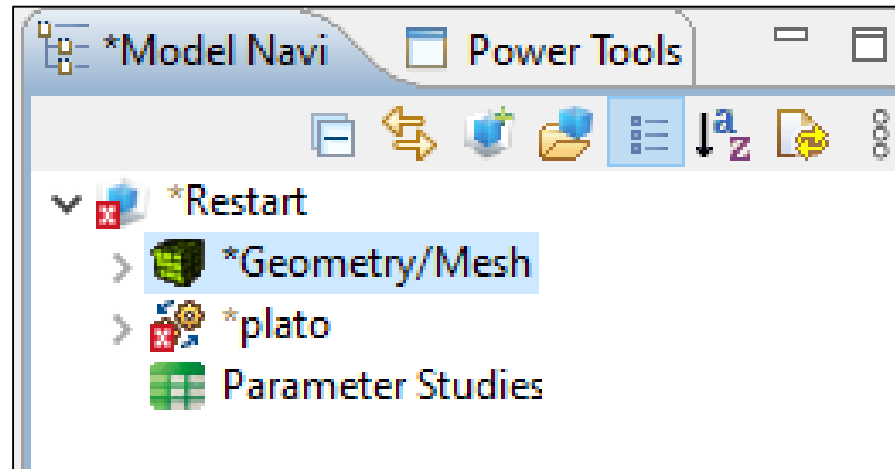


Sidesets and Nodeset in the Model Tree

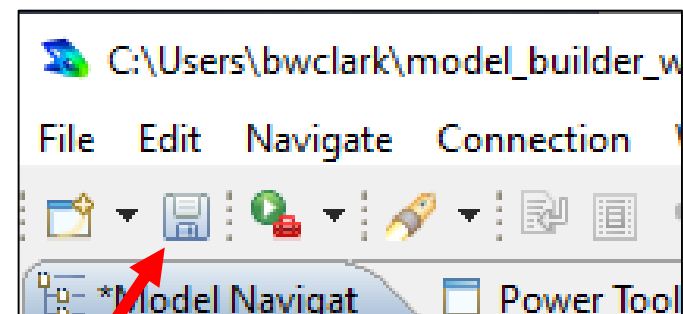
- The sideset and nodeset we just created can be seen in the Model tree under the “Geometry/Mesh” node.



Save the Geometry/Mesh

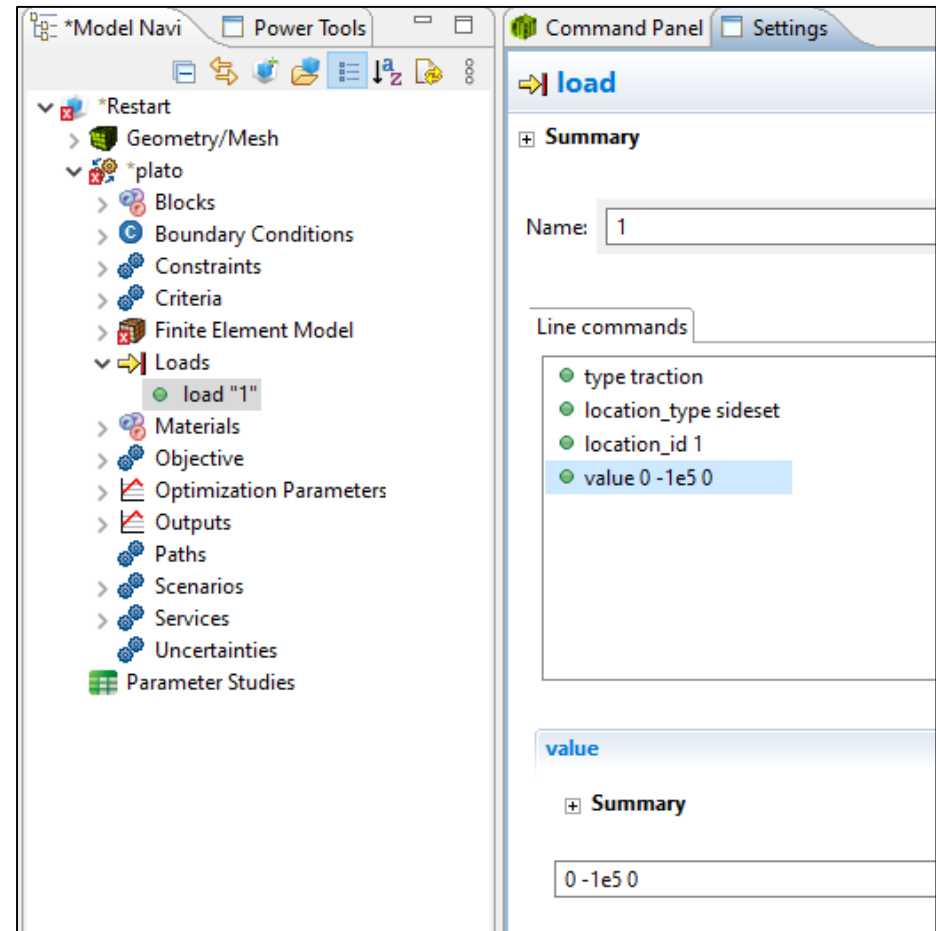


- Click on the “Geometry/Mesh” node in the tree and then click the Save icon in the toolbar.



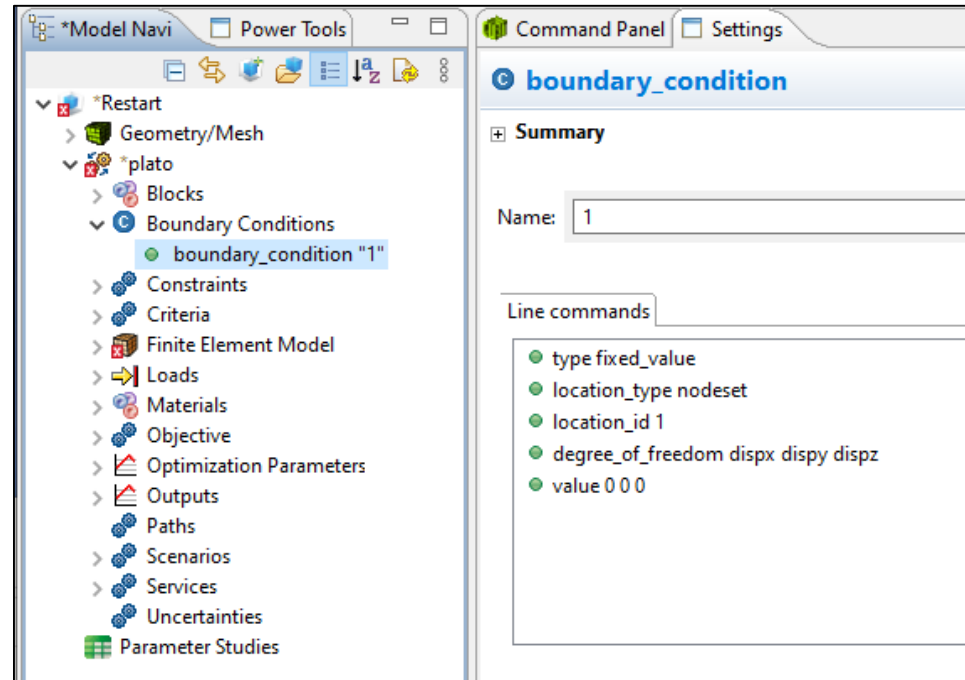
Create a Traction Load

- 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.



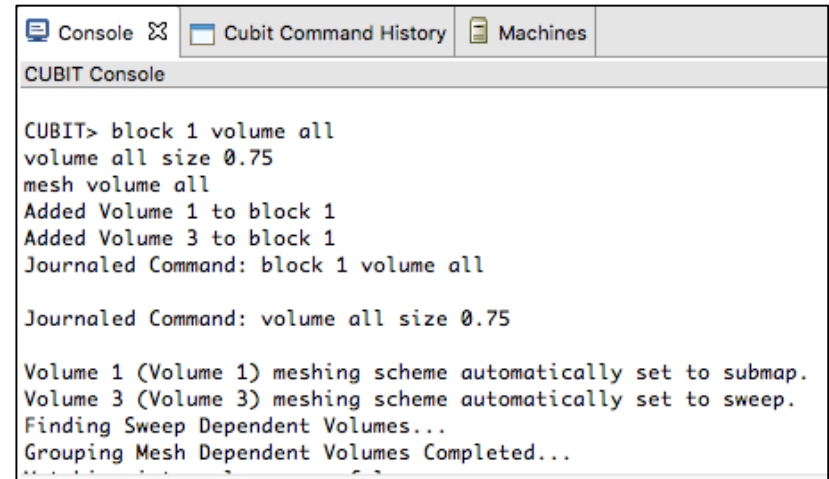
Create a Fixed BC

- 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 all
 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

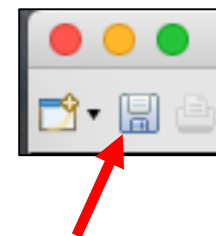
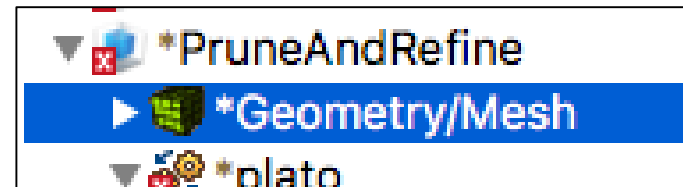


The screenshot shows the CUBIT Console window with the following text:

```
CUBIT> block 1 volume all
volume all size 0.75
mesh volume all
Added Volume 1 to block 1
Added Volume 3 to block 1
Journaled Command: block 1 volume all

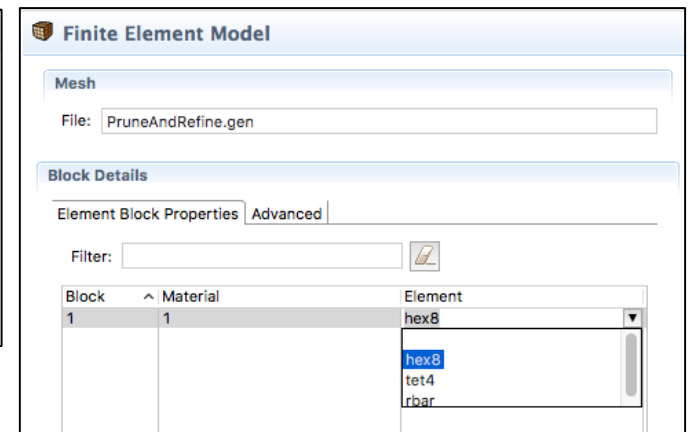
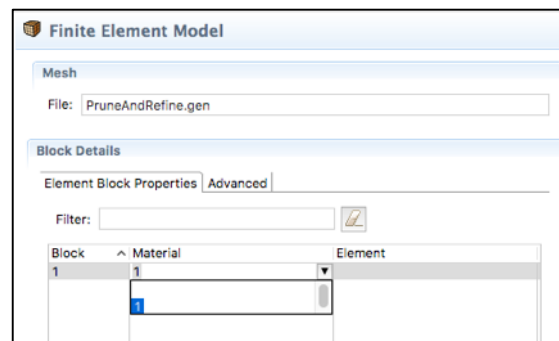
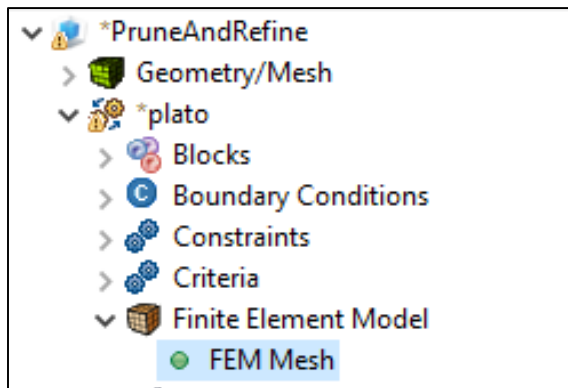
Journaled Command: volume all size 0.75

Volume 1 (Volume 1) meshing scheme automatically set to submap.
Volume 3 (Volume 3) meshing scheme automatically set to sweep.
Finding Sweep Dependent Volumes...
Grouping Mesh Dependent Volumes Completed...
```



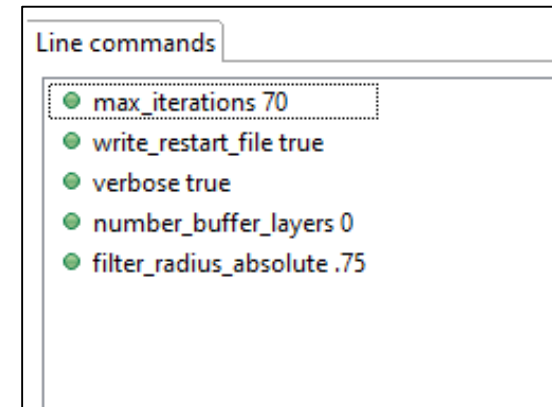
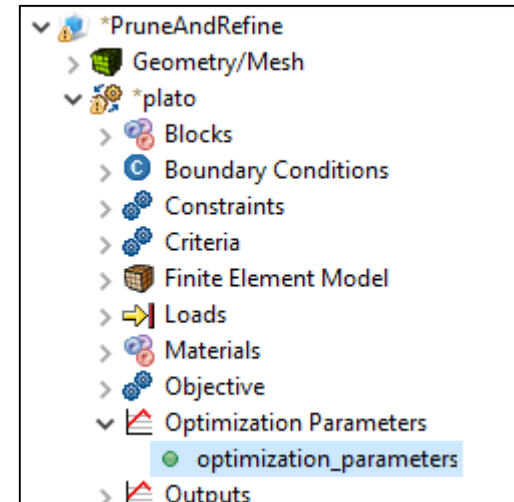
Assign Material and Element Type

- 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 **hex8** from the dropdown list



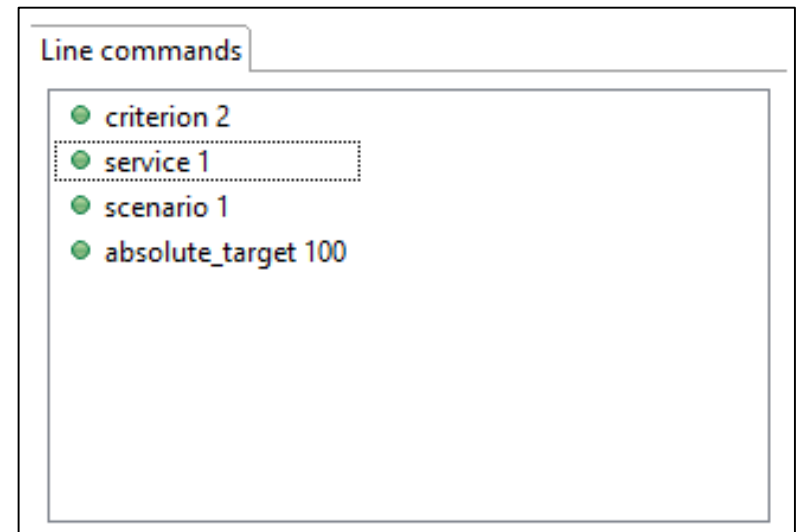
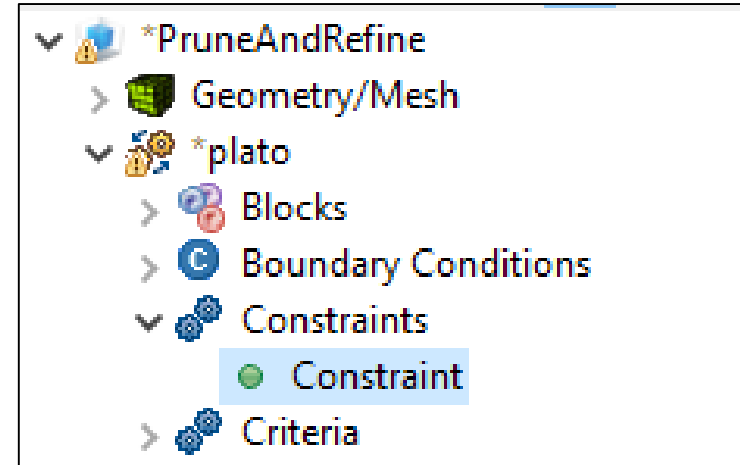
Set the Top. Opt. Parameters

- Click on the **Optimization Parameters-> optimization_parameters** node in the tree
- In the **Settings** panel right-click in the **Parameters** area and **Add** the parameter **filter_radius_absolute**. We will use the absolute filter radius value so that we can specify the radius as an absolute value rather than a scale of the average mesh edge size. We do this because when we refine the mesh our average mesh edge size will change and we want the filter radius value to stay constant.
- In the **Settings** panel delete **filter_radius_scale** by right-clicking on it and choosing **Delete**.
- Set the following parameter values by clicking on them and then entering the value in the edit field.
 - **filter_radius_absolute = .75**
 - **max_iterations = 70**



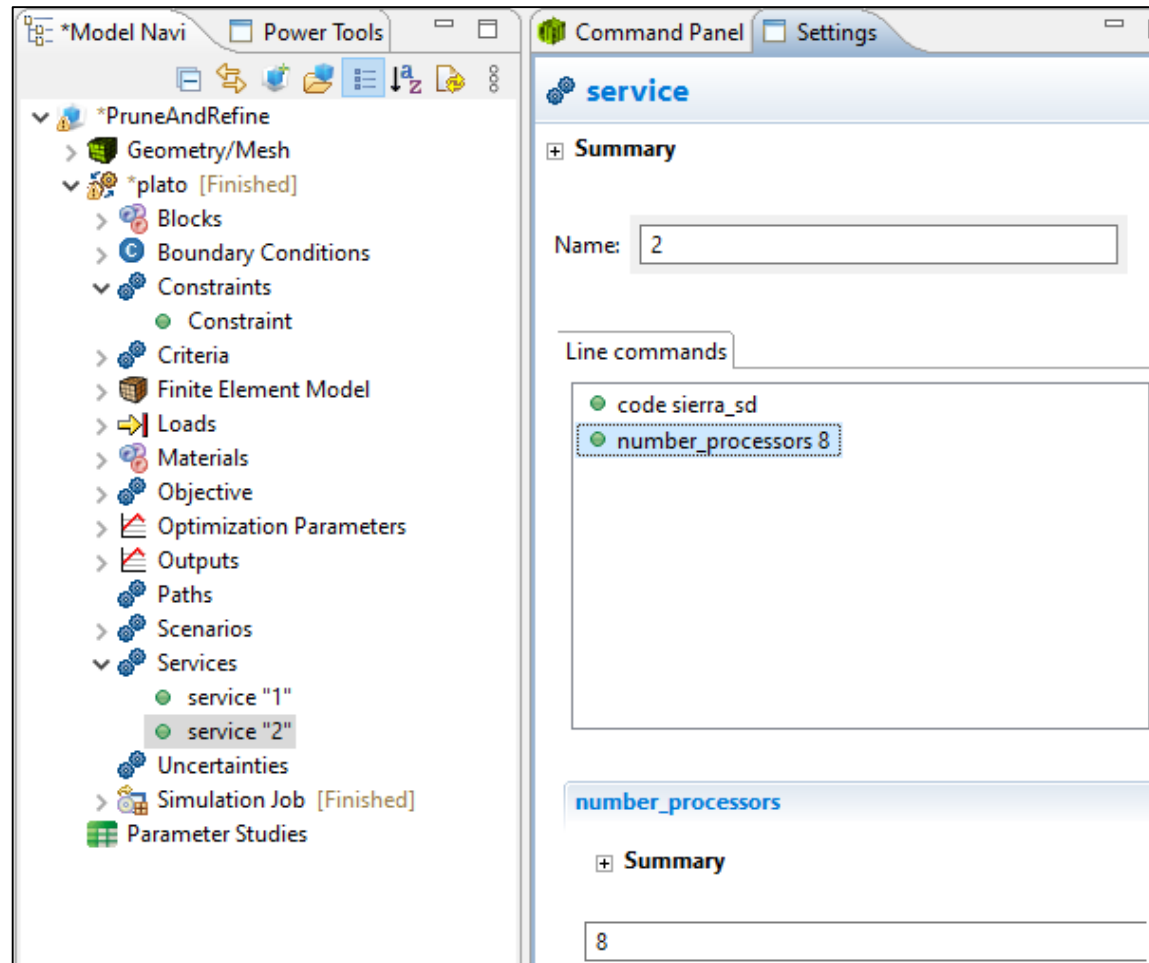
Set the Volume Fraction Constraint

- Click on the **Constraints->Constraint** node in the tree.
- In the **Settings** panel right-click in the **Parameters** area and **Add** the parameter **absolute_target**. We will use this parameter to specify an absolute target value rather than a percentage of the original design domain volume. We do this because when we prune the mesh the starting domain volume will change and we want our target volume to stay constant.
- In the **Settings** panel delete **relative_target** by right-clicking on it and choosing **Delete**.
- Set **absolute_target** to 100. Just for reference our starting design domain volume is about 1605 so we are asking for a small fraction of that



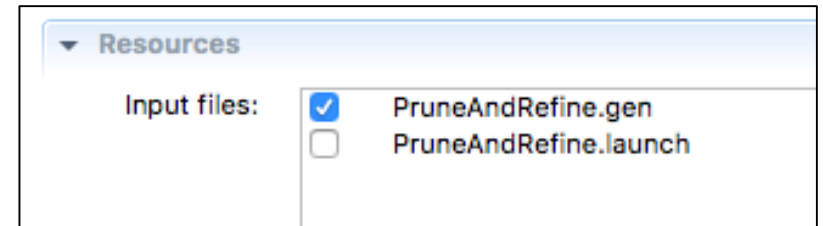
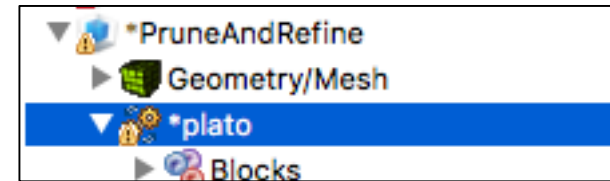
Set the Number of Processors

- Click on **Services->service "2"** in the tree
- In the **Settings** panel change the number of processors used by this sierra_sd service to 8 as shown in the right. This will speed up the optimization run.



Run the Optimization

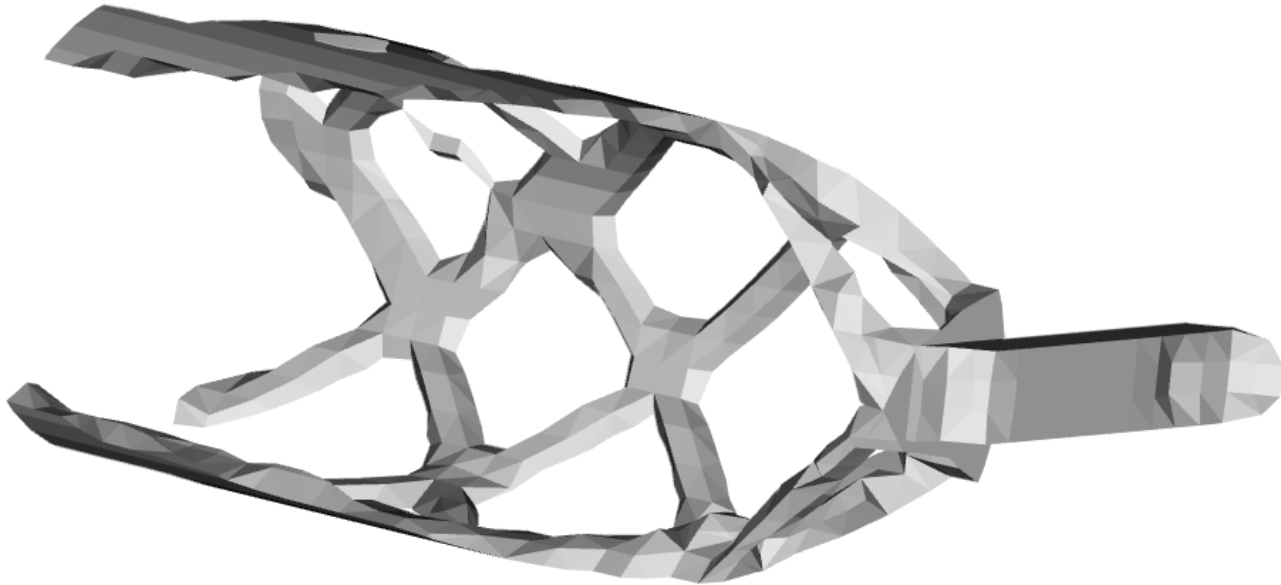
- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view
- Choose **Plato** as the code and then choose the machine and execution template you want to use



- In the **Resources** area make sure **PruneAndRefine.gen** is checked so that this file gets pushed to the working directory.
- In the **Prune and Refine** area make sure **Prune Mesh** is unchecked and **Number Refines** is **0**. For this first run we won't be doing any pruning or refining.
- Choose any other preferences and launch the job by clicking on **Submit Simulation Job** toward the top of the panel

Initial Result

The initial results should look something like that shown below. Note how rough the results look because of the coarse mesh we used. Next, we will use pruning and refining to run with a finer mesh so that we get smoother features while still keeping the problem size reasonable.



Launch the Restart Run

- Bring up the job submission panel by clicking on the **plato** node in the tree.
- Under the **Prune and Refine** area check the **Prune Mesh** box, set the **Number Buffer Layers** to **2**, **Number Refines** to **1**, and **Restart Iteration** to **65**. Plato will prune away the mesh not touched by the results in Iteration 65, leaving 2 additional buffer layers of hexes around the pruned mesh, and will then refine the pruned mesh 1 time before starting the next run.
- Resubmit the job by clicking on the **Submit Simulation Job** in the job submission area in the **Settings** panel

▼

Prune and Refine

☒ Prune Mesh

Number Buffer Layers:

2

▼

Number Refines:

1

▼

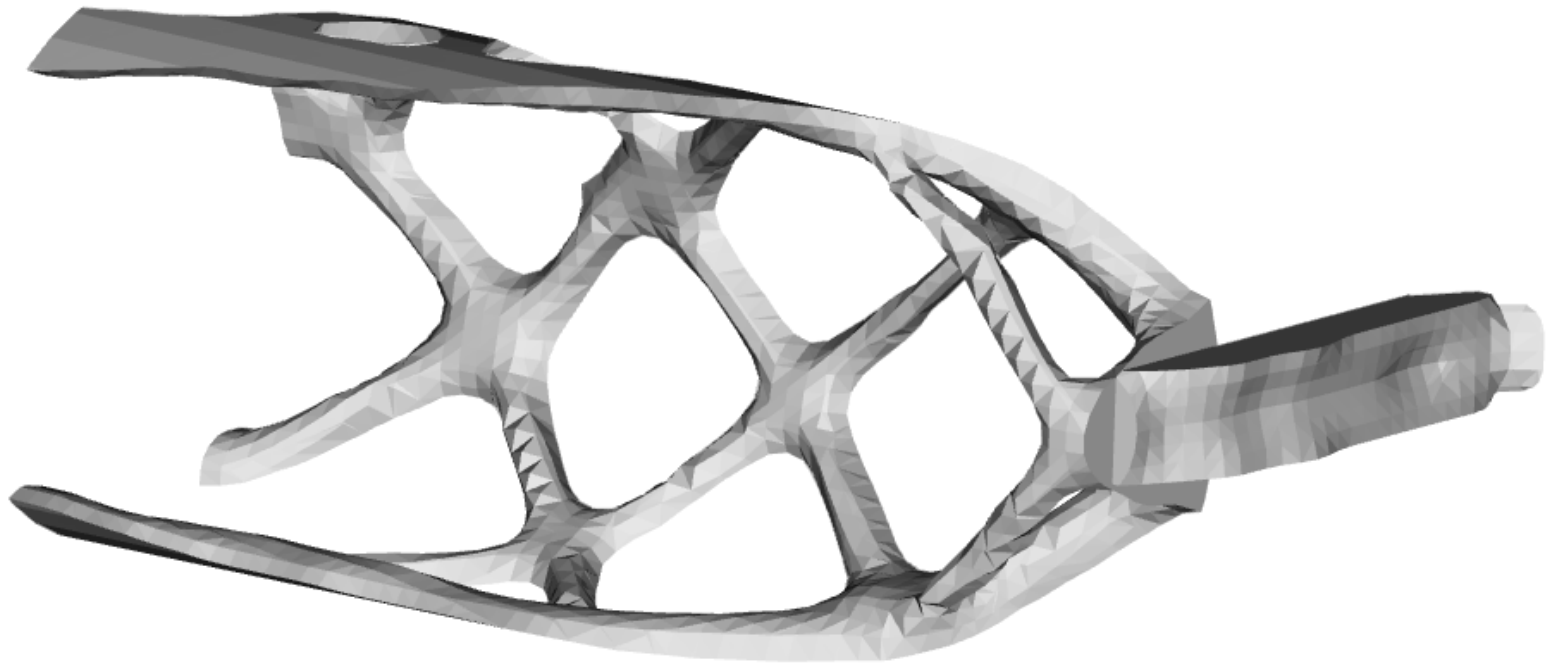
Restart Iteration:

65

▼

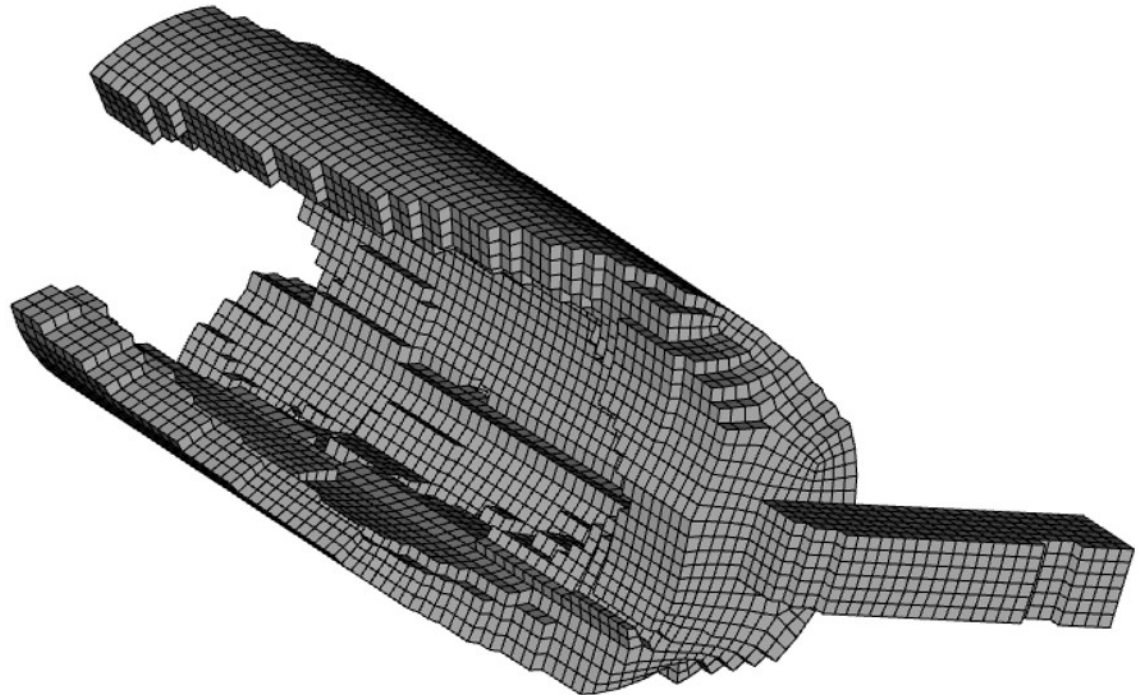
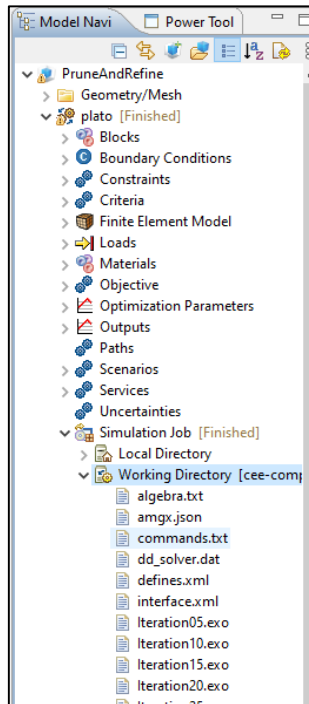
Result after 1st Prune/Refine

Your result should look something like that shown below. As you can see the features are smoother than in the first run. This is because we are using a finer mesh.



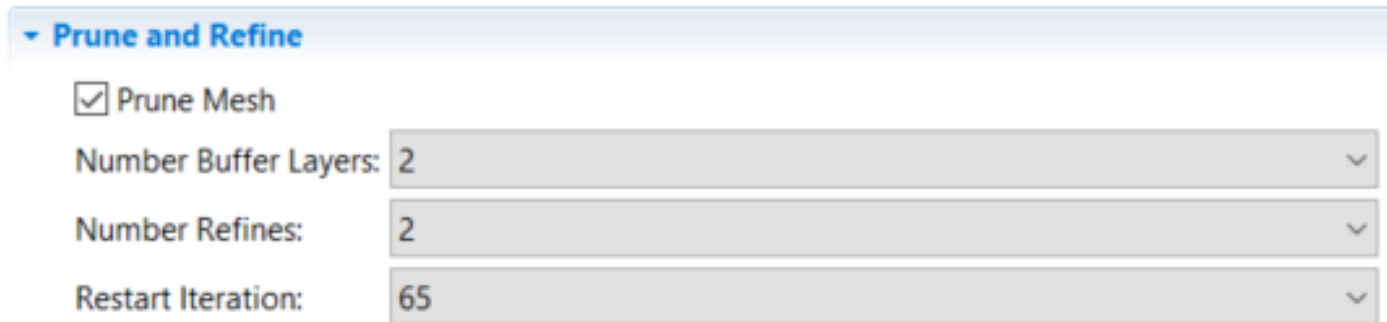
Pruned and Refined Mesh

- Let's take a look at the pruned and refined mesh used for this second run.
- Open the **Working Directory** as shown in the image below. Look for a file called **PruneAndRefine_mod.gen**. Right-click on this file and choose **Load Mesh**. The mesh will be loaded in the **Geometry/Mesh** area in the tree.
- In the **CUBIT** console type “edge vis on” and you should see something like the mesh shown below.
- If you type “list hex” at the **CUBIT** prompt you will see how many hexes are in this pruned and refined mesh. It is more than when we started but much less than if we had refined the starting mesh without first pruning.



2 Levels of Refinement

- Bring up the job submission panel by clicking on the **plato** node in the tree.
- Under the **Prune and Refine** area change **Number Refines** from **1** to **2**.
- Resubmit the job by clicking on the **Submit Simulation Job** in the job submission area in the **Settings** panel. This mesh will now grow substantially in size (roughly 8x) but the problem size is again much smaller than if we were to refine without pruning.



The screenshot shows a software interface for the 'Prune and Refine' settings. It includes a checkbox for 'Prune Mesh' which is checked, and three dropdown menus for 'Number Buffer Layers' (set to 2), 'Number Refines' (set to 2), and 'Restart Iteration' (set to 65).

▼ Prune and Refine	
<input checked="" type="checkbox"/> Prune Mesh	
Number Buffer Layers:	2
Number Refines:	2
Restart Iteration:	65

Result after 2nd Prune/Refine

Your result should look something like that shown below. You should only have to let it run about 5-10 iterations. Now the features are really starting to look nice. If you like you can again load the **PruneAndRefine_mod.gen** file (delete the existing one first) to examine the mesh that was used on this second restart.

