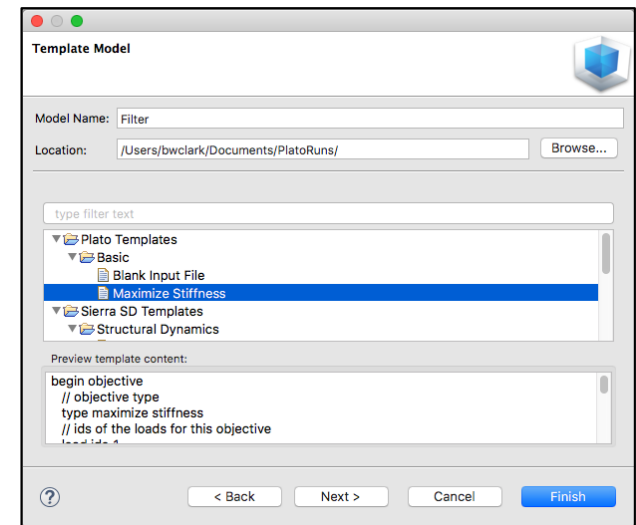
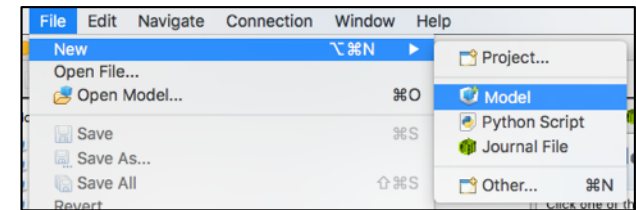


# Filter Tutorial

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

# Create a New Model

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



# Create the Design Domain

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

brick x 10 y 3 z 0.1

brick x 1 y 1 z 0.1

volume 2 move 5.5 0 0

imprint volume all

delete volume 2

- Note the surface created from the imprint (highlighted in orange).

```

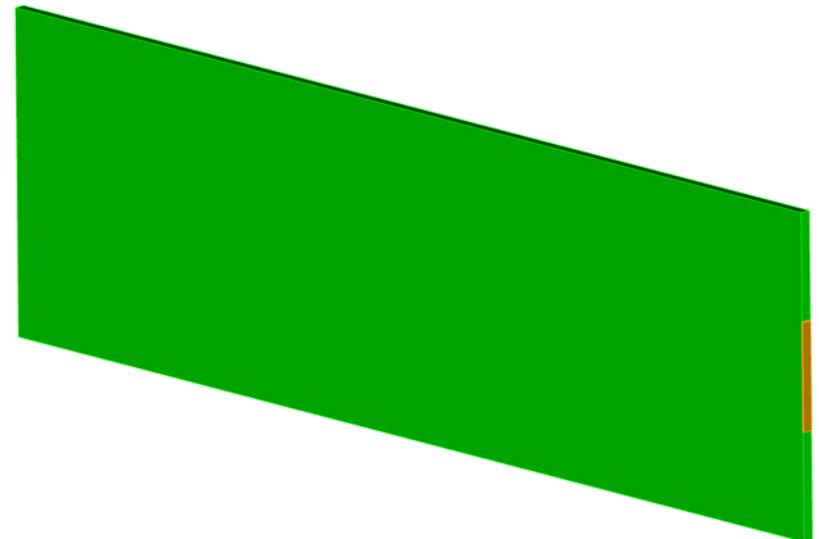
Console  Machines
CUBIT Console

CUBIT> brick x 10 y 3 z .1
Successfully created brick volume 1
Journaled Command: brick x 10 y 3 z 0.1

CUBIT> brick x 1 y 1 z .1
Successfully created brick volume 2
Journaled Command: brick x 1 y 1 z 0.1

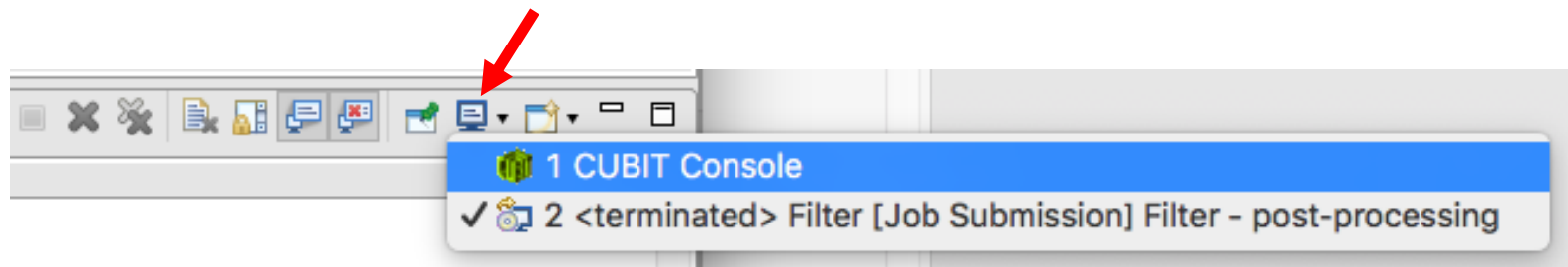
CUBIT> vol 2 move 5.5 0 0
Journaled Command: volume 2 move 5.5 0 0

CUBIT> imprint all
Preserving undo information done
  
```



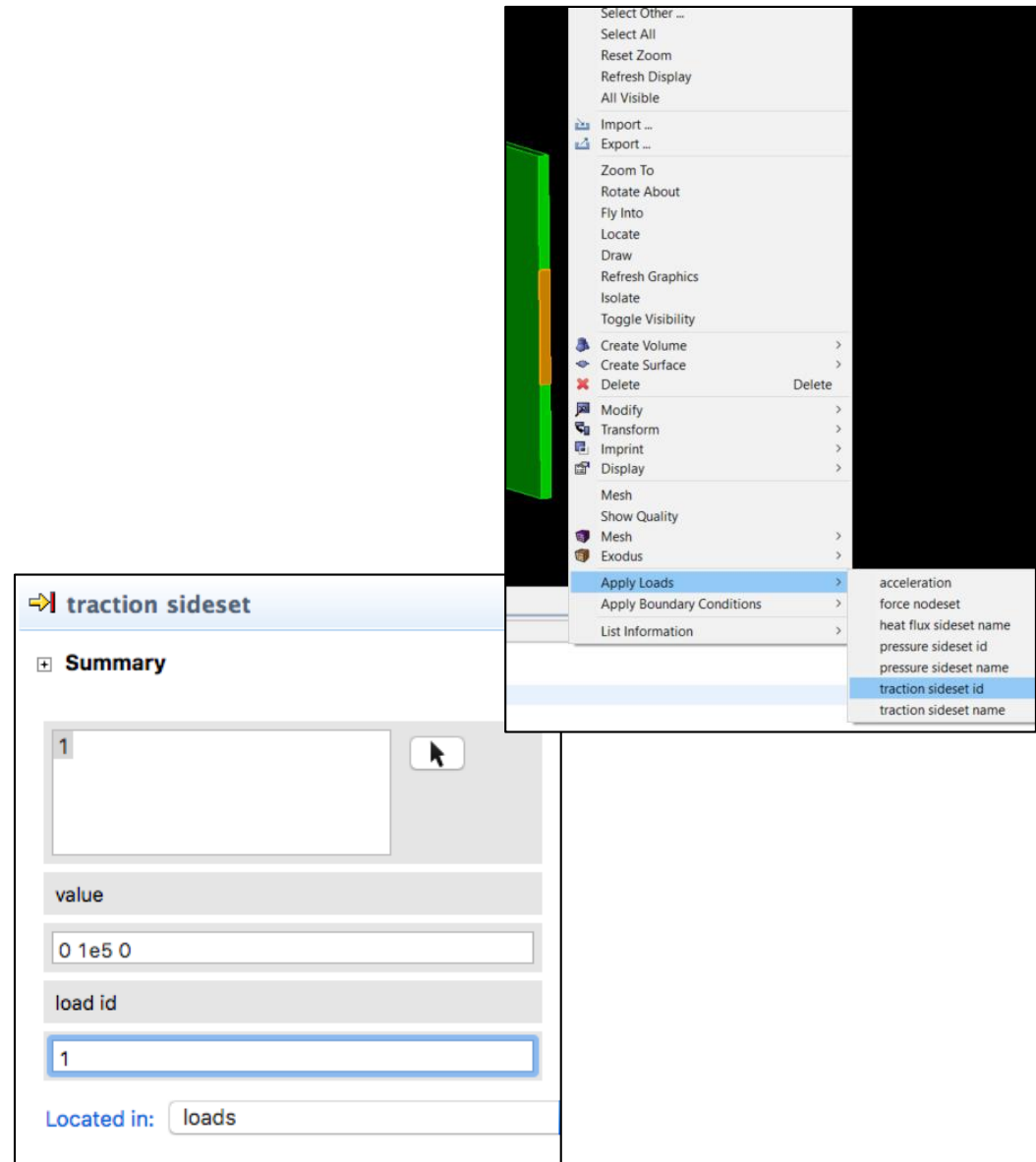
# Hint: Cubit Console

- 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.



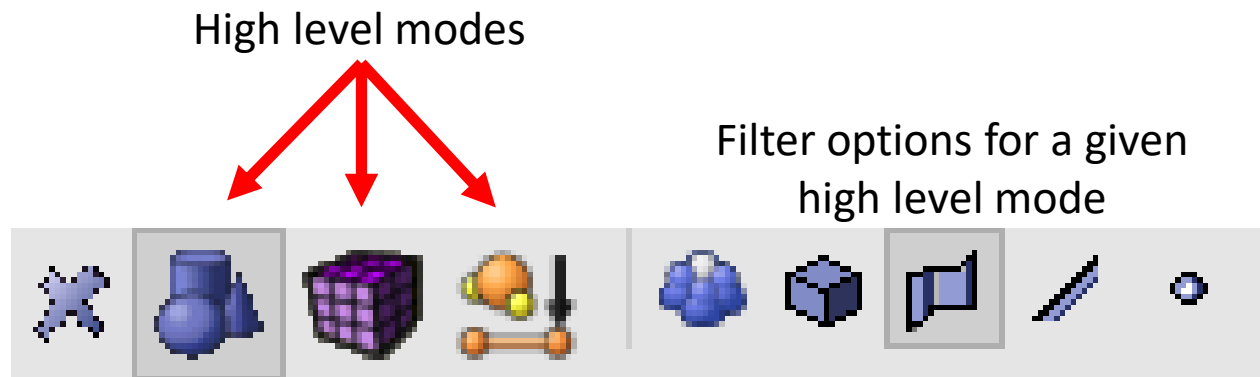
# Apply a Traction Load

- Select and right-click the middle right surface (see Hint on next slide) and choose **Apply Loads->traction sideset id**
- In the **Settings** panel enter “0 1e5 0” in the “**value**” box and hit the tab key. Then enter “1” in the “**load id**” box and hit the return key.



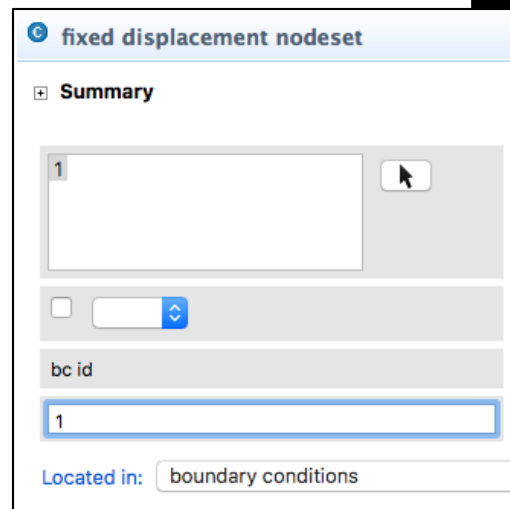
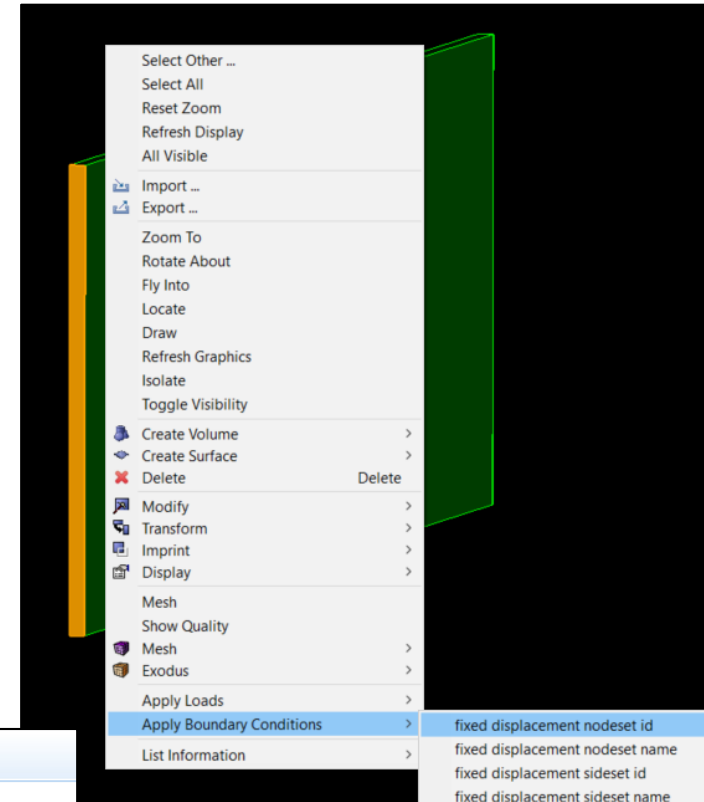
# Hint: Entity Selection

- 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.



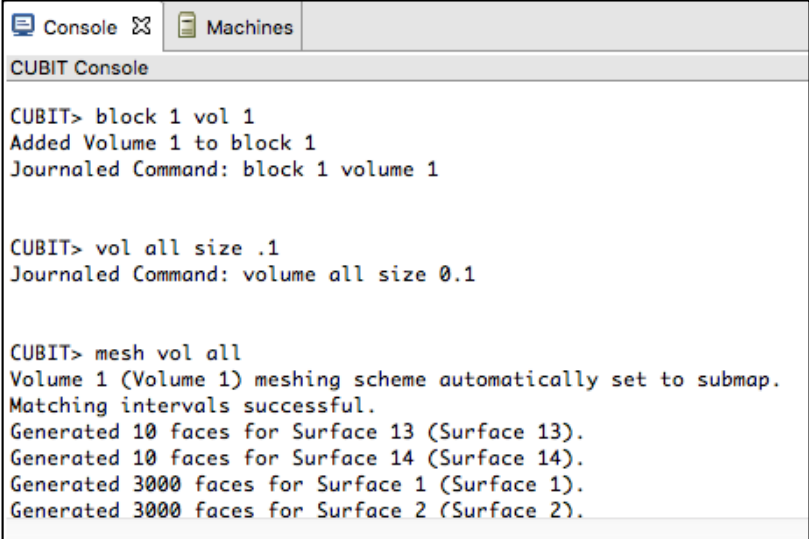
# Create a Fixed BC

- Select and right-click on the single left surface and choose **Apply Boundary Conditions->fixed displacement nodeset id**
- In the **Settings** panel enter “1” in the “**bc id**” box and hit return.



# Define Block and Mesh Domain

- At the **CUBIT** prompt type the following commands:  
 block 1 volume 1  
 volume all size 0.1  
 mesh volume all
- Click on **Geometry/Mesh** node in the tree and then click the **Save** icon in the toolbar to save the model

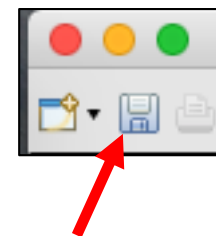
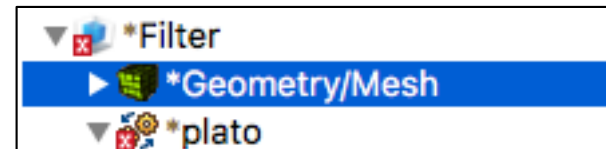


```

CUBIT Console
CUBIT> block 1 vol 1
Added Volume 1 to block 1
Journaled Command: block 1 volume 1

CUBIT> vol all size .1
Journaled Command: volume all size 0.1

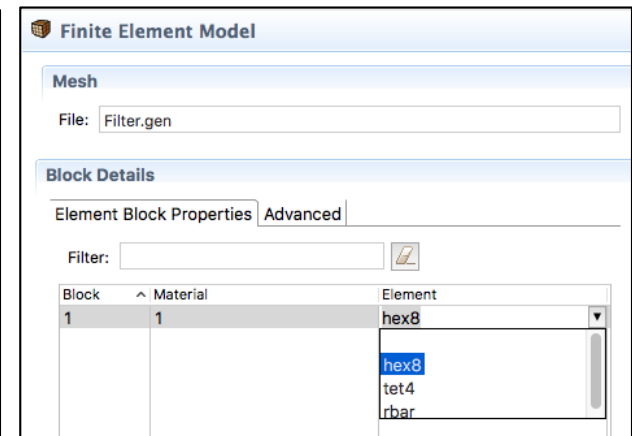
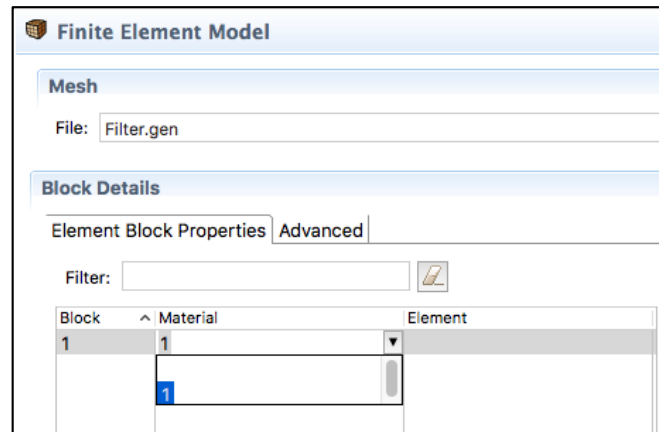
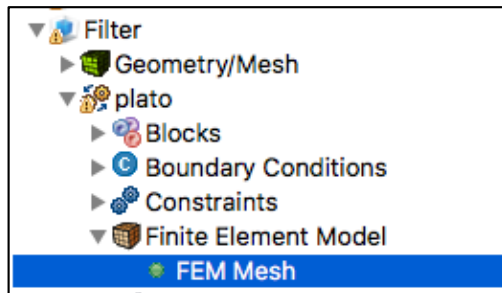
CUBIT> mesh vol all
Volume 1 (Volume 1) meshing scheme automatically set to submap.
Matching intervals successful.
Generated 10 faces for Surface 13 (Surface 13).
Generated 10 faces for Surface 14 (Surface 14).
Generated 3000 faces for Surface 1 (Surface 1).
Generated 3000 faces for Surface 2 (Surface 2).
  
```





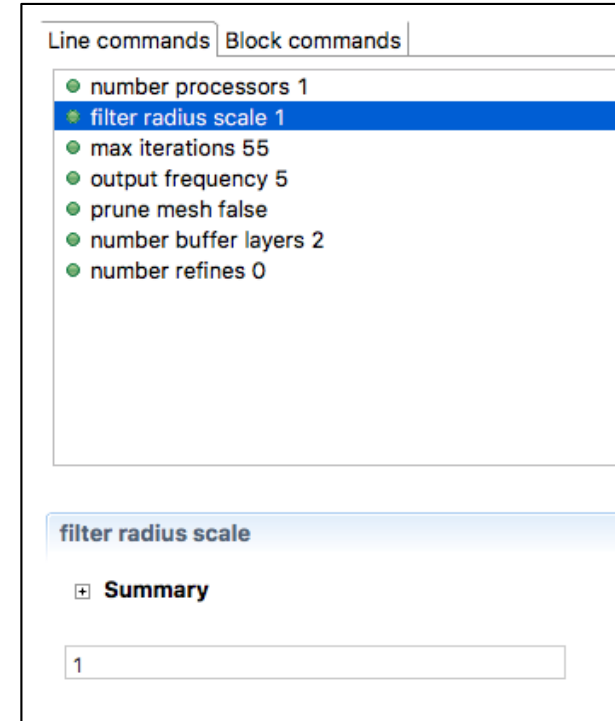
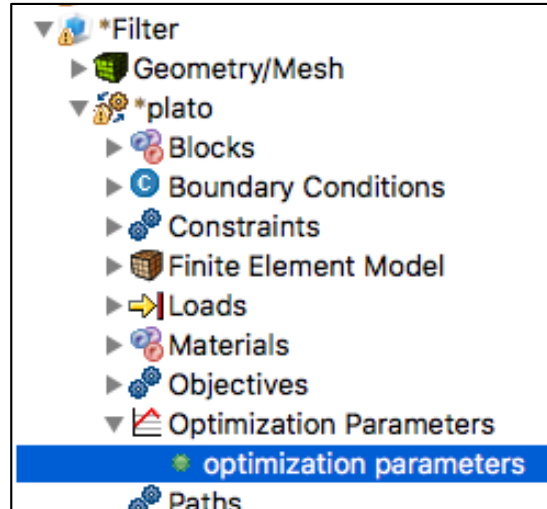
# Assign Material and Element Type

- 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



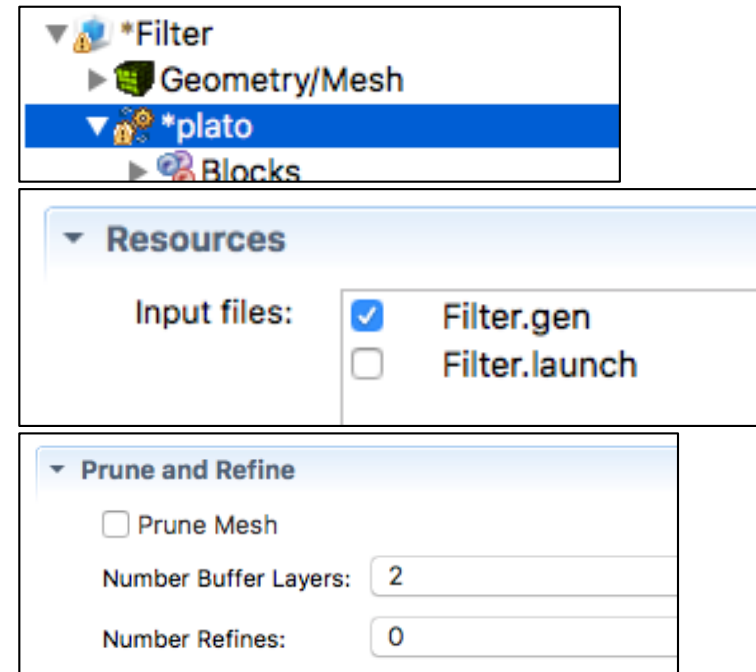
# Set the Filter Scale

- Click on the **Optimization Parameters-> optimization parameters** node in the tree
- In the **Settings** panel click on **filter radius scale** and type in **1** for the value and hit the **return** key. **Filter radius scale** specifies the scale factor that will be multiplied by the average mesh edge length to determine the filter radius.



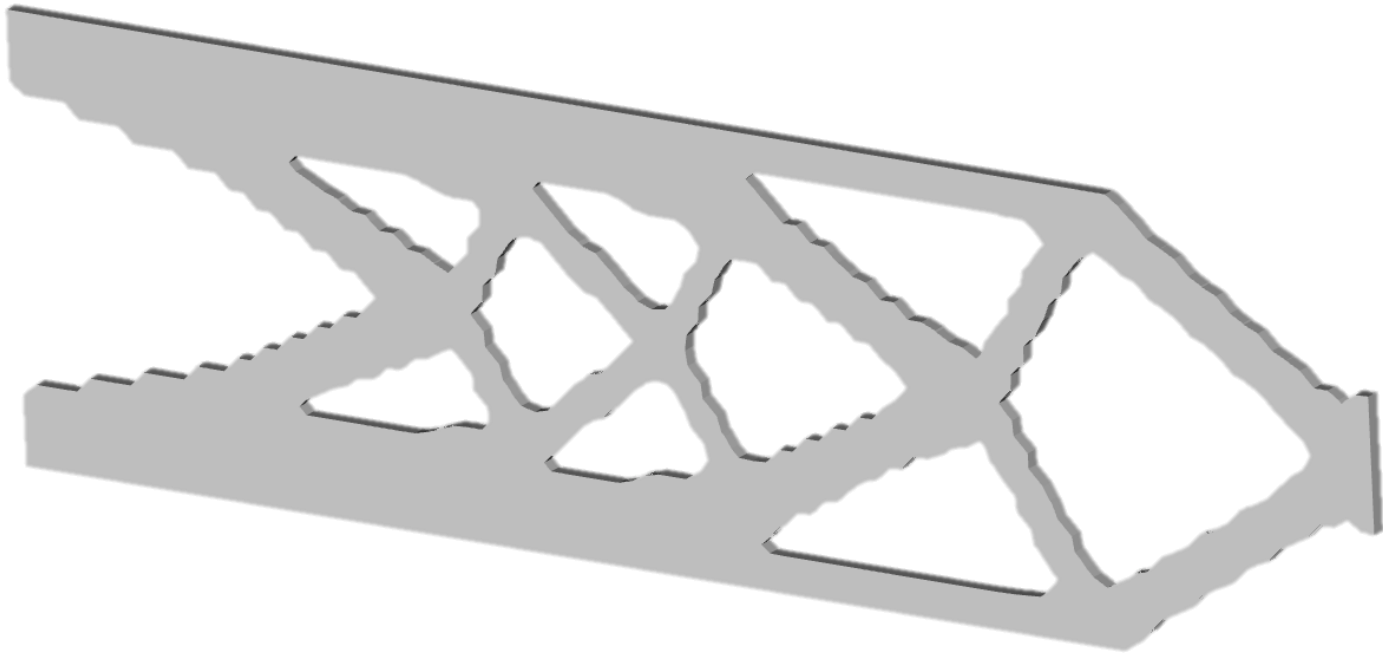
# Run the Optimization

- 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 **Filter.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



# Results for filter radius scale=1.0

- With a **filter\_scale** value of 1.0 the filter smoothing at each node will not take into account any neighbors and will have little to no effect on the result. Notice the rough edges.

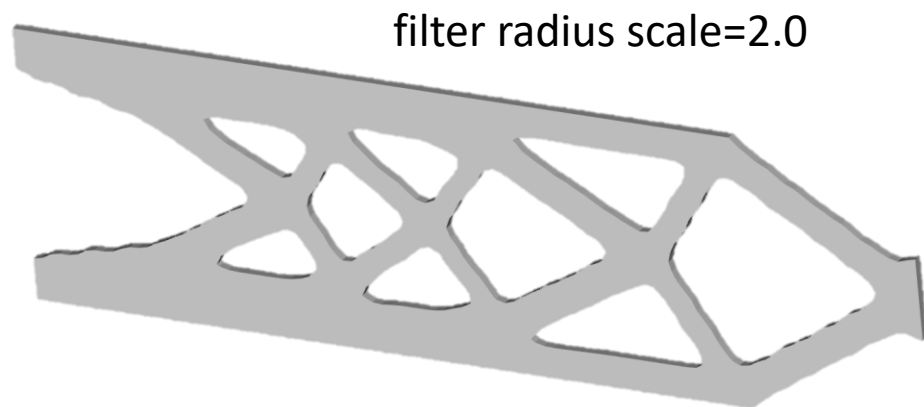
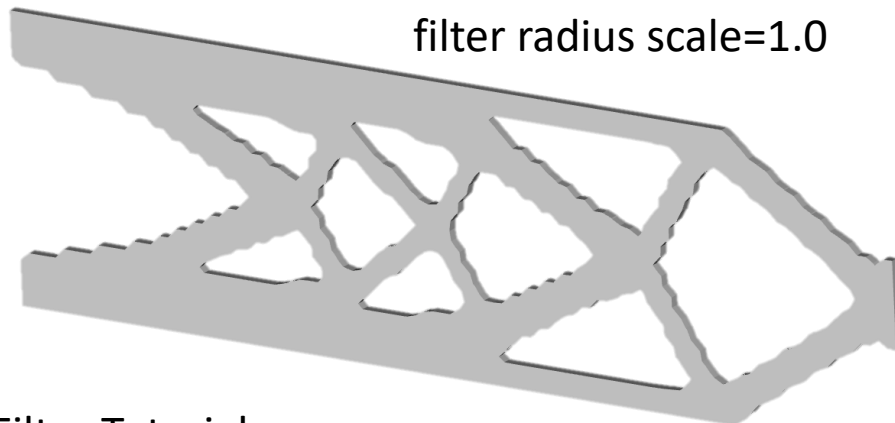


# Rerun with filter radius scale=2.0

- Change the **filter radius scale** value to **2.0**
- Resubmit the job with all of the same parameters by clicking on the **plato** node in the tree and then click on the **Submit Simulation Job** in the job submission area in the **Settings** panel

# Results for filter radius scale=2.0

- Notice how as we increase the **filter radius scale** value the results become smoother. In general, this is one of the effects of using a filter. A **filter radius scale** of **2.0** means that the filter radius of influence is 2.0 times the average mesh edge length. With the radius larger than one element width the filter actually starts having influence on neighbors and starts smoothing out local discontinuities or roughness.

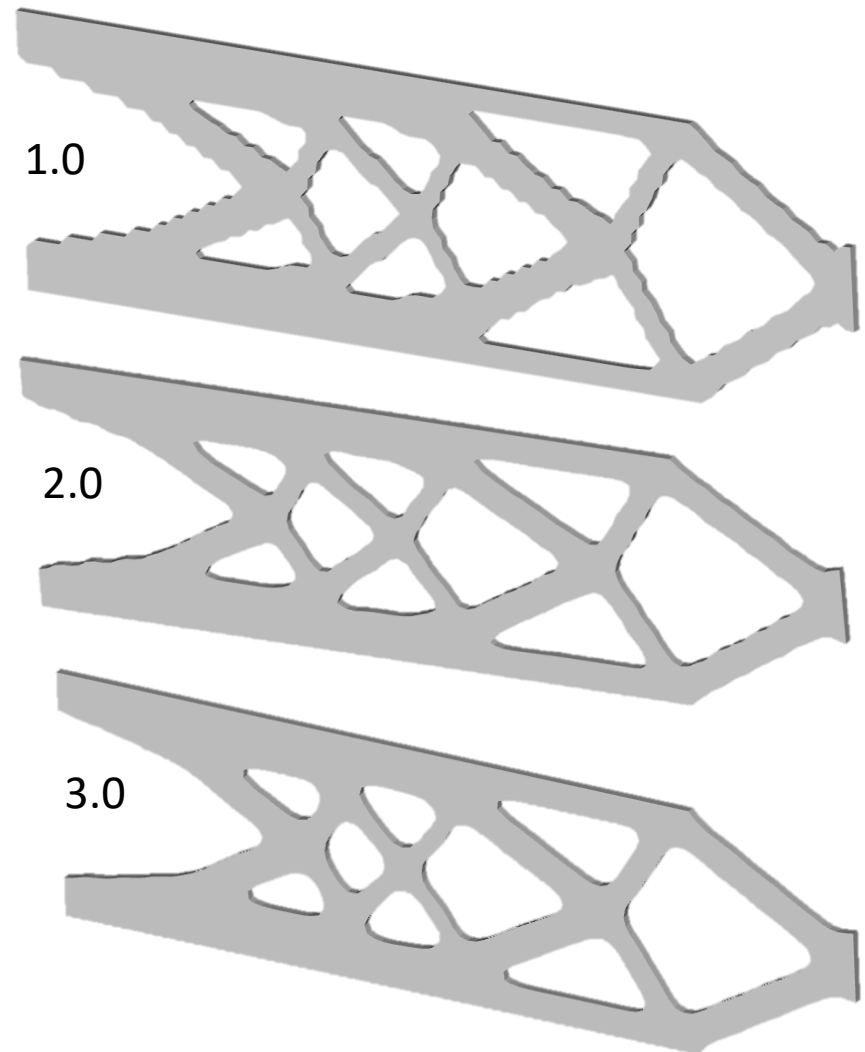


# Rerun with filter radius scale=3.0

- Change the **filter radius scale** value to **3.0**
- Resubmit the job with all of the same parameters by clicking on the **plato** node in the tree and then click on the **Submit Simulation Job** in the job submission area in the **Settings** panel

# Results for filter radius scale=3.0

- Again, increasing **filter radius scale** to **3.0** creates an even smoother result.
- With **filter radius scale** at 3.0 the filter radius is 3 elements wide and has a broader effect.



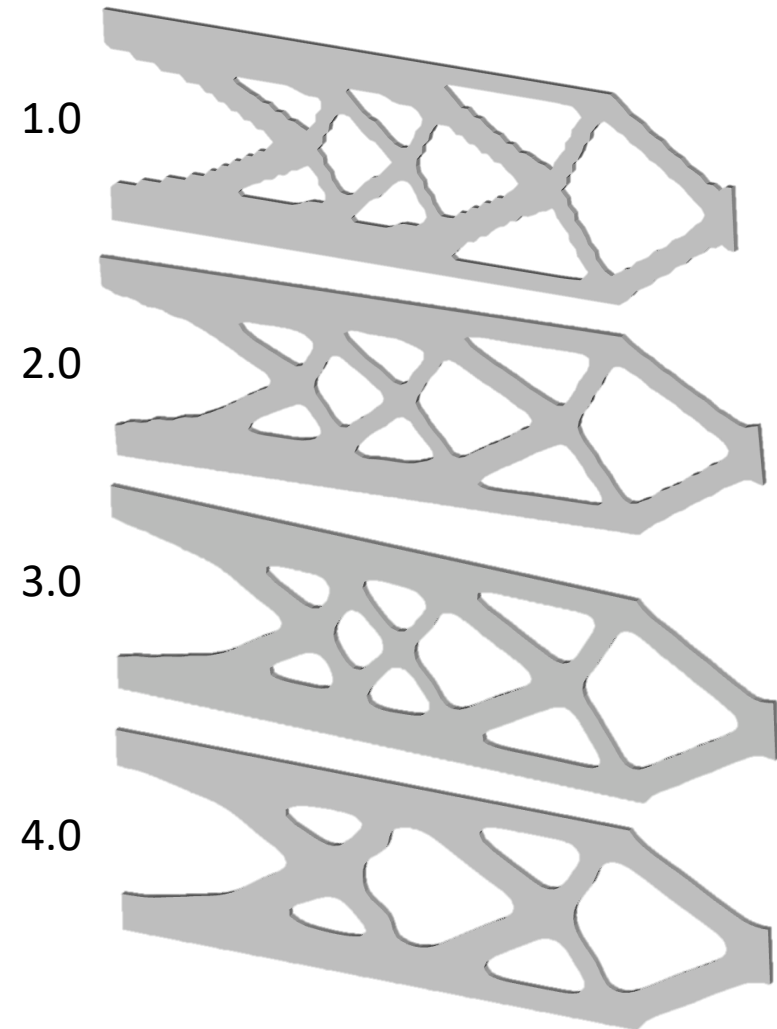


# Rerun with filter radius scale=4.0

- Change the **filter radius scale** value to **4.0**
- Resubmit the job with all of the same parameters by clicking on the **plato** node in the tree and then click on the **Submit Simulation Job** in the job submission area in the **Settings** panel

# Results for filter radius scale=4.0

- Again, increasing **filter radius scale** to **4.0** creates an even smoother result. However, now the topology has changed. You can see that the result has bulkier members and some of the thinner members are gone all together. As the filter radius gets larger it has a more global smoothing effect and will increase the size of the smallest allowable feature.



# Mesh Refinement

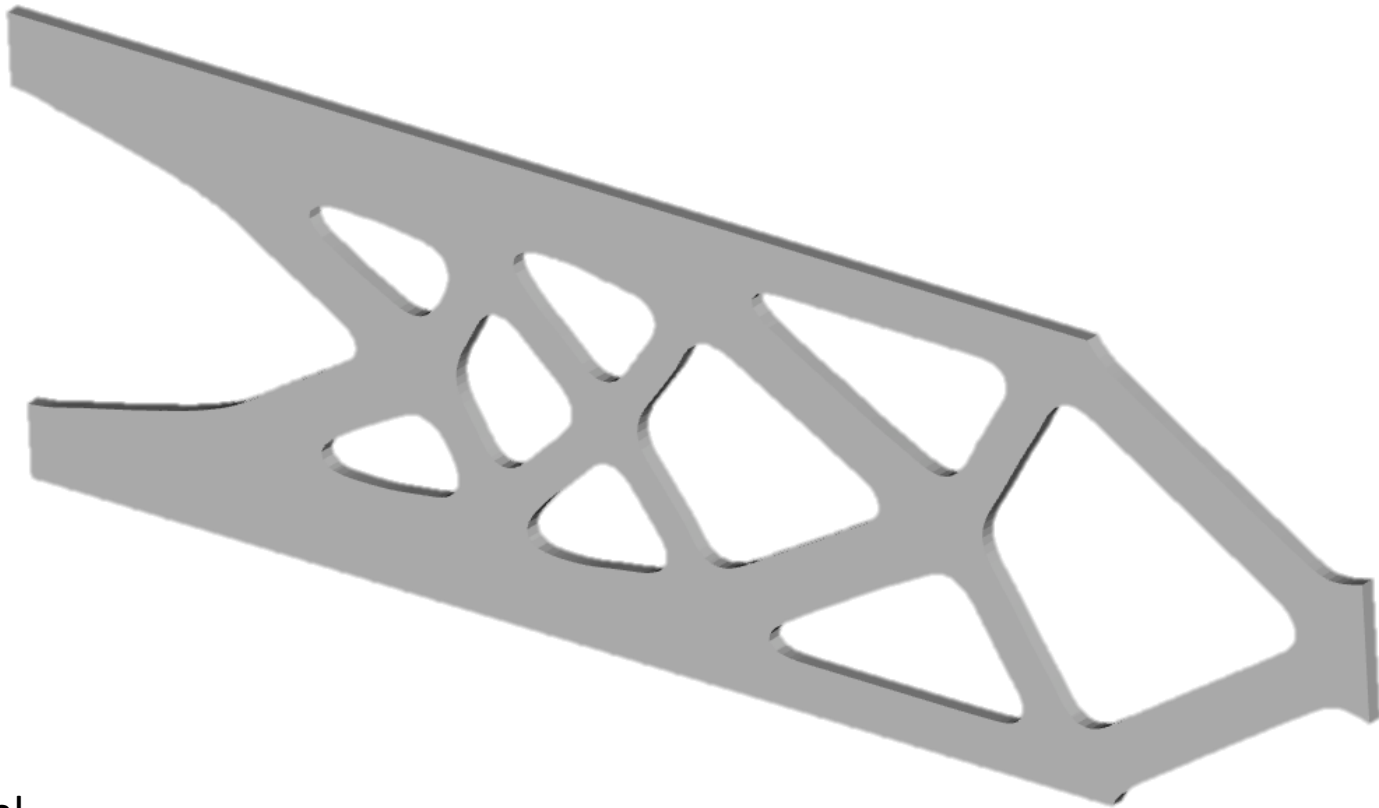
- So, what if you want to resolve fine features in your design but the filtering is limiting how small the features can be?
- The solution is to use a finer mesh. The finer the mesh, the finer the features that can be resolved in the design.
- Let's keep our **filter radius scale** at 4.0 but use a finer mesh.

# “Refine” the Mesh

- Click on the **Filter** node in the tree under **Geometry/Mesh** to make sure your commands apply to our starting mesh and not one of the result meshes.
- In the **CUBIT** console window type “**edge vis on**” and hit return to turn the mesh edge visibility back on.
- Type the following commands at the **CUBIT** prompt:
  - delete mesh
  - curve 12 interval 1
  - volume all size 0.05
  - mesh volume all
- The “curve...interval” command forces CUBIT to keep one mesh interval through the thickness so that we are only refining the mesh in the XY plane. This will help keep the problem size down.
- Now, rerun the optimization

# Results from Mesh Refinement

- The image below shows how by refining the mesh we were able to maintain the smaller features while still filtering at a radius of 4 elements wide.



# filter radius scale vs. filter radius absolute

- If you were paying close attention to what we just did with mesh refinement you noticed that we kept the filter radius at 4 elements wide but we changed the size of our elements (mesh refinement) from 0.1 to 0.05. This means that the actual filter radius value went from 0.4 to 0.2. This still allowed us to get the smoothness and fine features we wanted. But it highlights the fact that **filter radius scale** provides a radius that is relative to mesh edge length. If you want to specify an absolute filter radius value you can use the **filter radius absolute** command instead. When using progressive mesh refinement it is often a good idea to use an absolute filter radius so that the feature size in the design stays roughly the same.

# Specifying an Absolute Filter Radius

- To use **filter radius absolute** instead of **filter radius scale** first bring up the topology optimization parameters in the settings panel by clicking on **optimization parameters** in the tree.
- Right-click on **filter radius scale** in the settings panel and choose **Delete**.
- Right-click somewhere in the blank part of the parameters window and choose **Add** and then **filter radius absolute**.
- Then enter the desired value for **filter radius absolute**.

