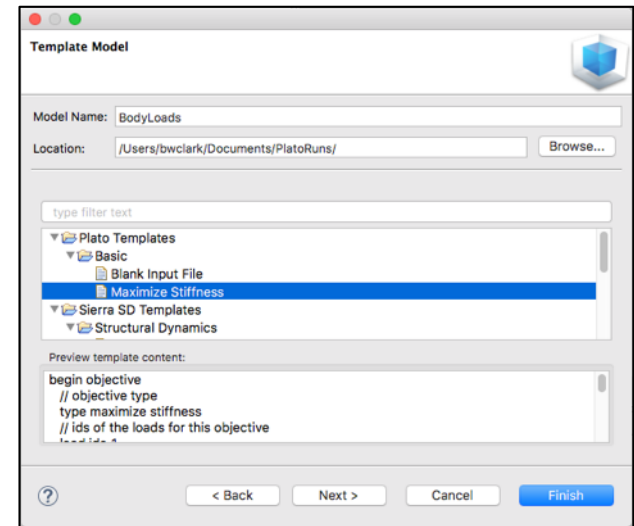
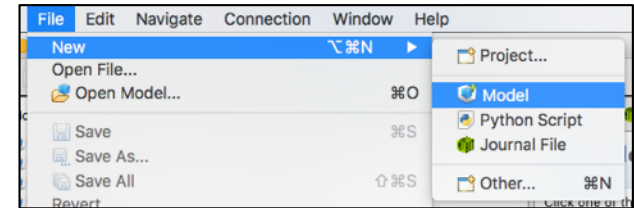


Body Loads Tutorial

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

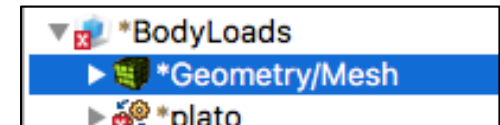
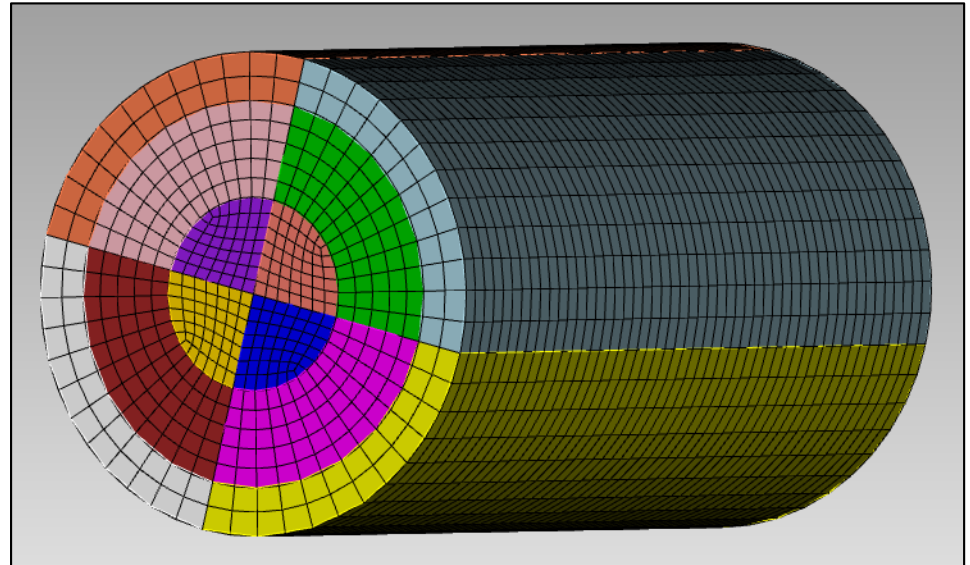
- Choose **File->New->Model** in the menu
- Choose **New Model** then **Next**
- Choose **Create From Template** then **Next**
- Enter **BodyLoads** as the **Model Name**
- Choose the **Plato Templates->Basic->Maximize Stiffness** template and then **Finish**



Create the Design Domain

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

```
cylinder height 10 radius 2
cylinder height 10 radius 2.5
webcut volume 2 tool volume 1
delete vol 1
cylinder height 10 radius 1
webcut volume 3 tool volume 4
delete vol 4
webcut volume all with plane xplane
webcut volume all with plane yplane
imprint all
merge all
block 1 vol 6 2 9 12
block 2 vol 7 13 10 3
block 3 vol 8 14 11 5
vol all size .2
mesh vol all
```

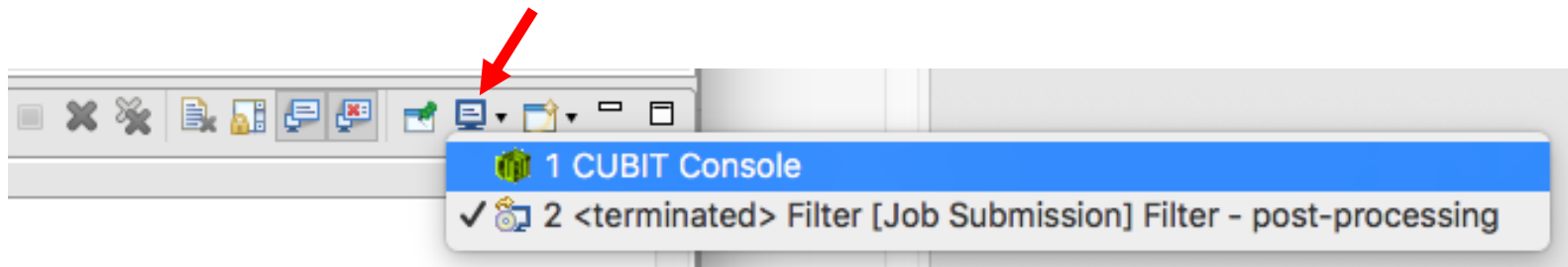


Click on **Geometry/Mesh** node in the tree and then click the **Save** icon in the toolbar to save the model



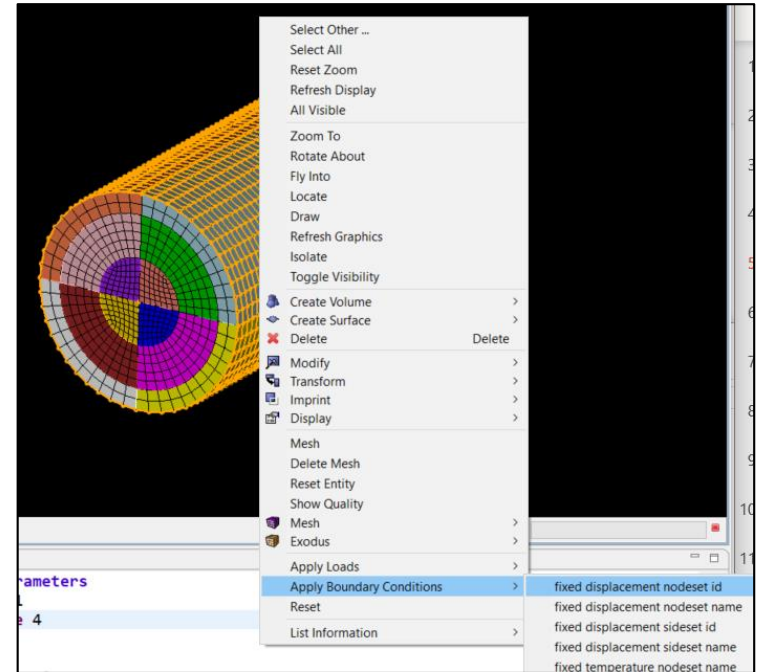
Hint: Cubit Console

- 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.



Create a Fixed BC

- Select the 4 outer surfaces on the cylinders as shown (Ctrl-select on Windows/Linux, Command-select on Mac). 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

☐

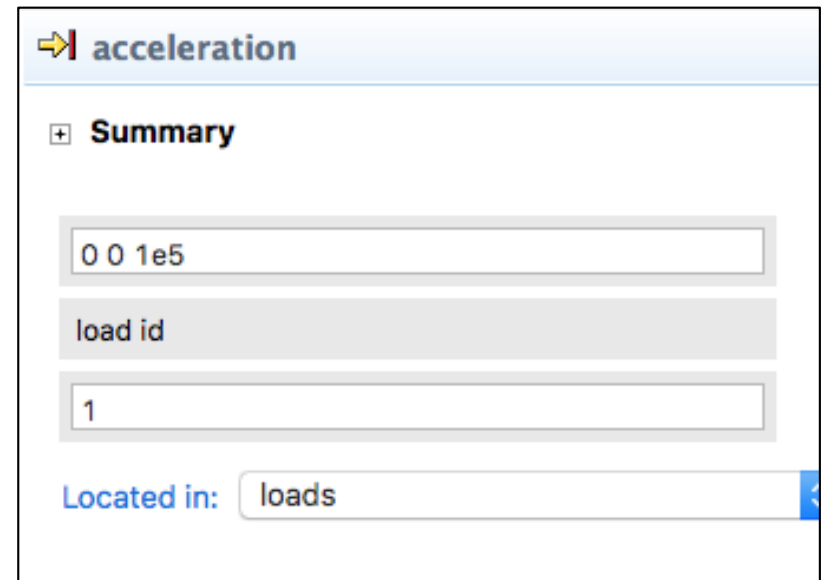
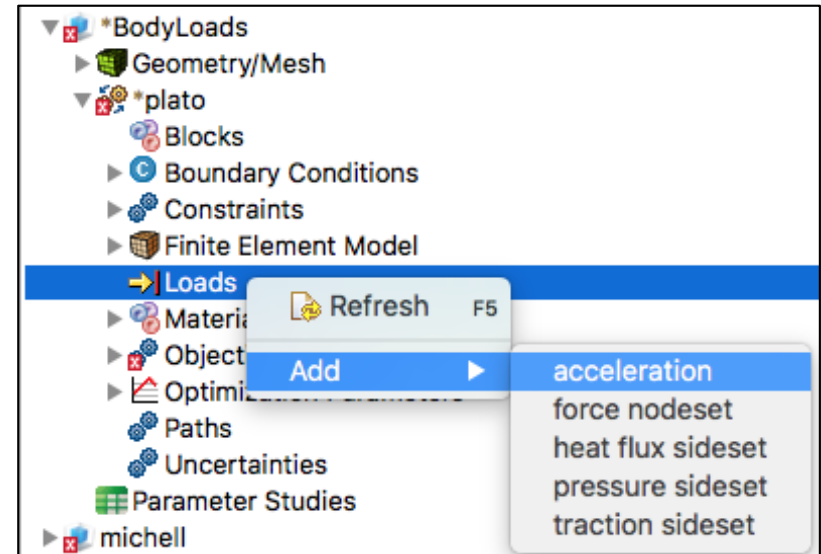
bc id

1

Located in: boundary conditions

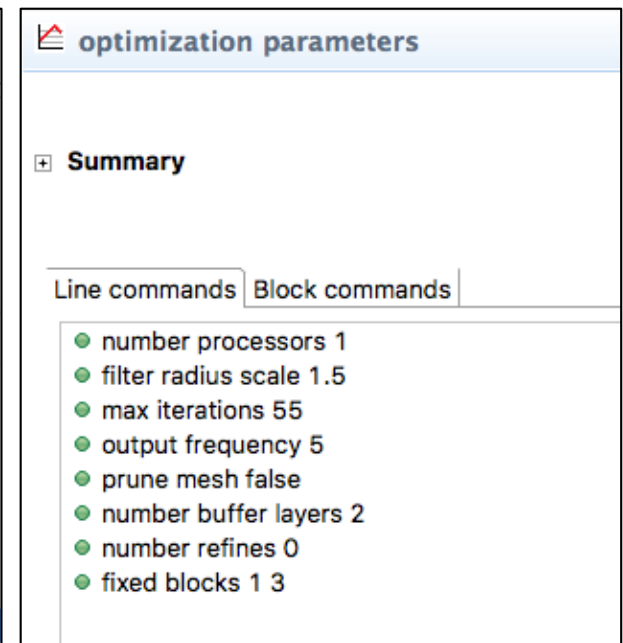
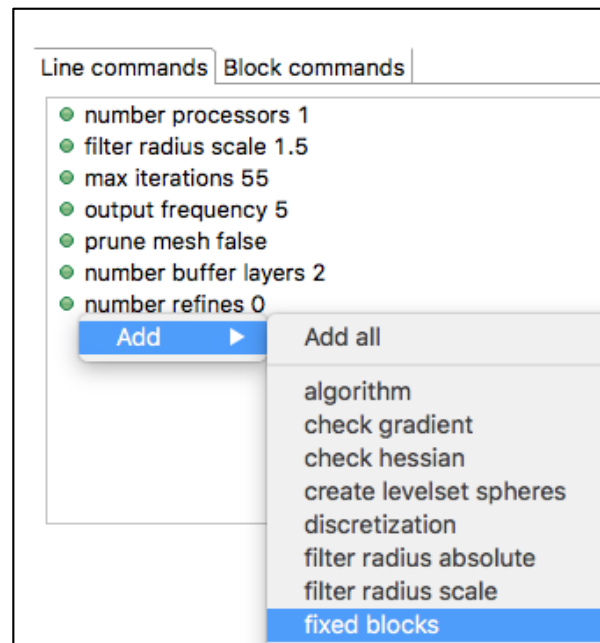
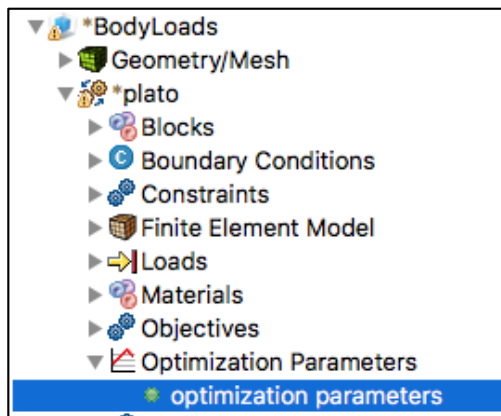
Apply a Body/Acceleration Load

- Right-click on **Loads** in the tree and choose **Add->acceleration**.
- Enter “**0 0 1e5**” in the first edit box (specifying 1e5 acceleration in z direction) and hit the tab key. Enter “**1**” in the “**load id**” edit box and hit return.



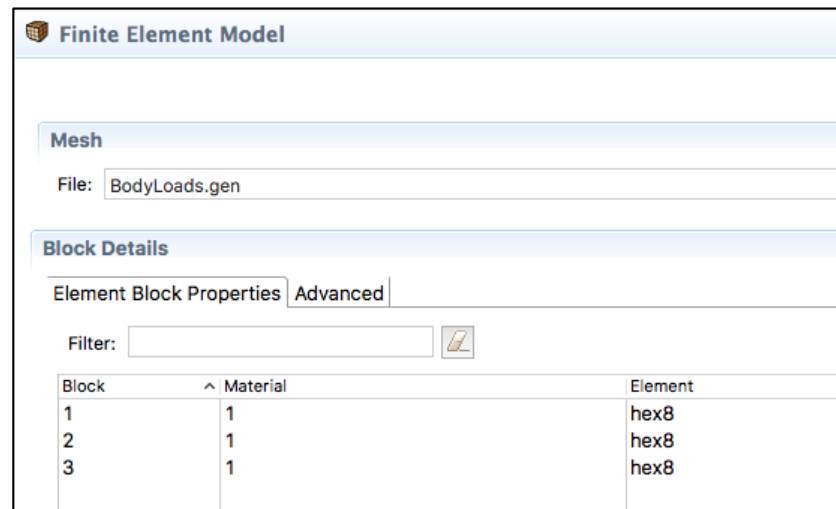
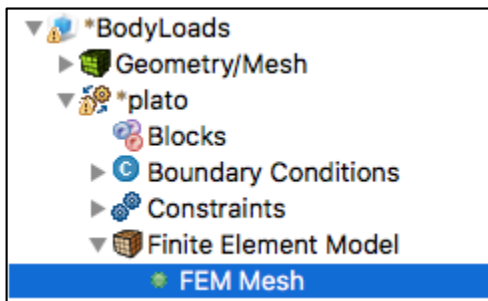
Specify Fixed Blocks

- Click on **Optimization Parameters**->**optimization parameters** in the tree.
- Right-click in the parameter window in the **Settings** panel and choose **Add->fixed blocks**.
- Enter “**1 3**” for the fixed blocks value specifying that blocks 1 and 3 are fixed.



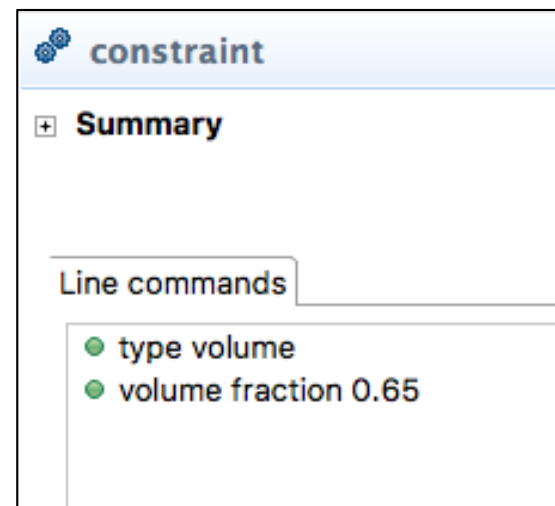
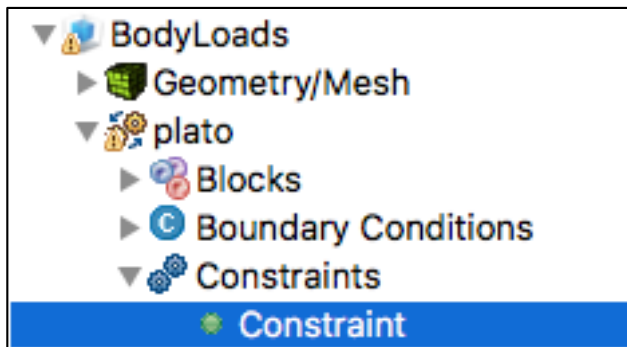
Assign Material and Element Type

- 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.
- Repeat this for Blocks 2 and 3.



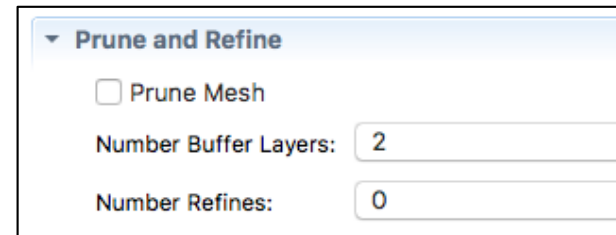
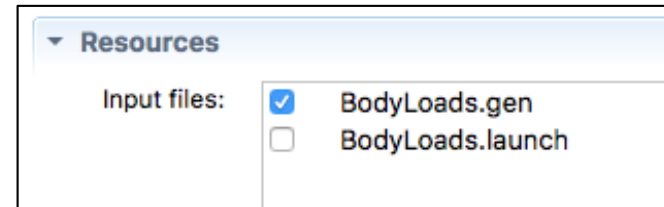
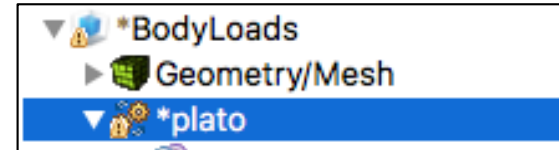
Set the Volume Constraint

- Click on the **Constraints->Constraint** node in the tree.
- In the **Settings** panel set the **volume fraction** parameter to be **0.65**. This will indicate that we want the final design to only use 65% of the original volume. Remember that this includes the fixed blocks. In this example the fixed blocks took up about 50% 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 **BodyLoads.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

- Results should begin to populate under the “Geometry/Mesh” header with each requested iteration.



- Your results should look something like this after about 30 iterations. The coarseness or roughness of the result is related to mesh resolution. See the “Prune and Refine” tutorial to learn how to efficiently generate smooth results.