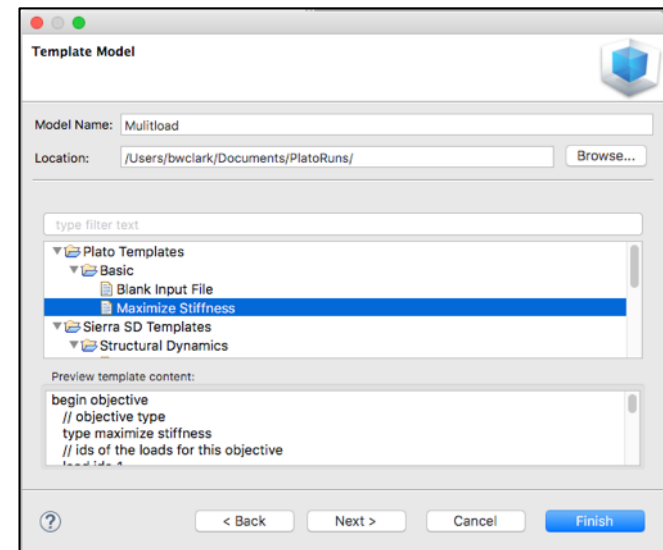
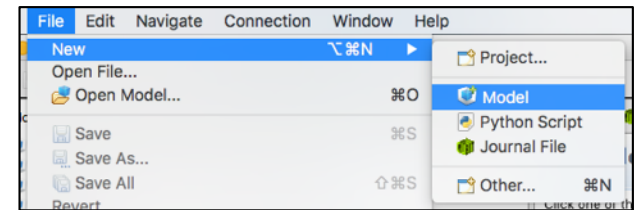


MultiLoad Tutorial

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 **Multiload** 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):

#make geometry and mesh

brick x 10

volume all size 0.25

mesh vol all

#define boundary conditions

#point load

nodeset 1 node at 0 0 5

nodeset 2 node at -2.5 -2.5 5

nodeset 2 node at 2.5 2.5 5

nodeset 3 node at 2.5 -2.5 5

nodeset 3 node at -2.5 2.5 5

#pin

nodeset 4 node at -5 -5 -5

rollers

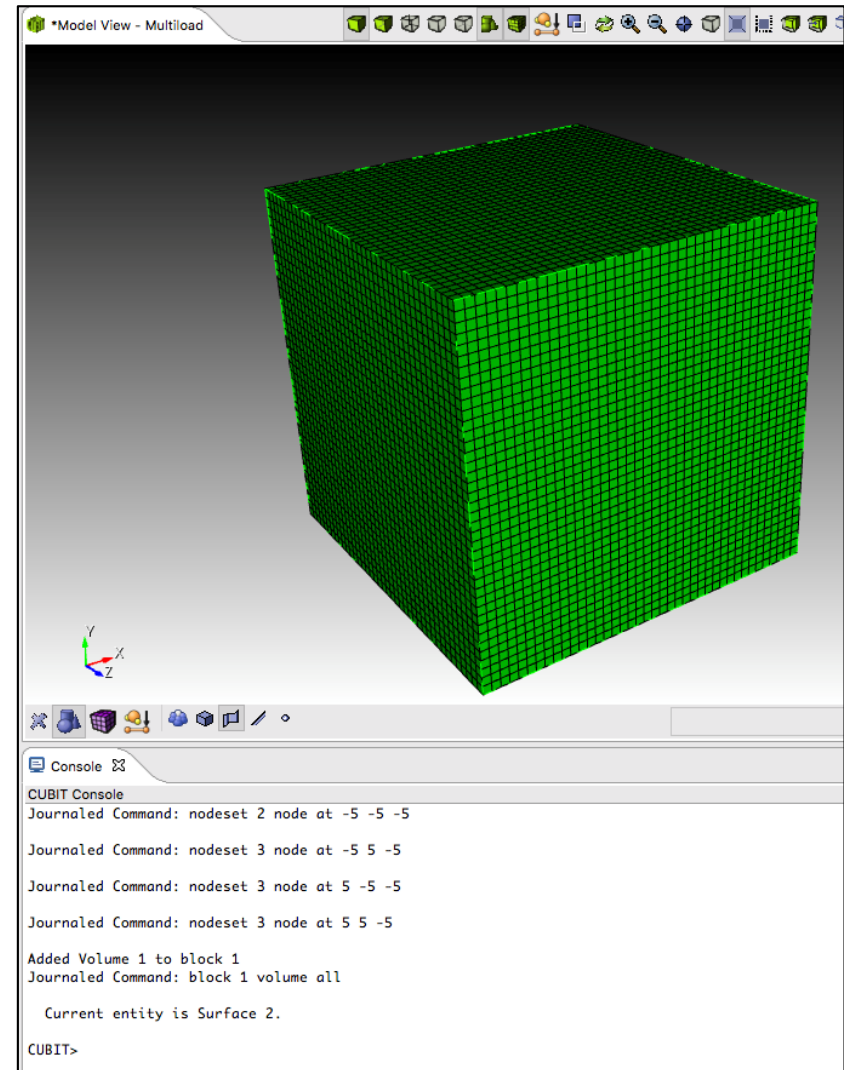
nodeset 5 node at -5 5 -5

nodeset 5 node at 5 -5 -5

nodeset 5 node at 5 5 -5

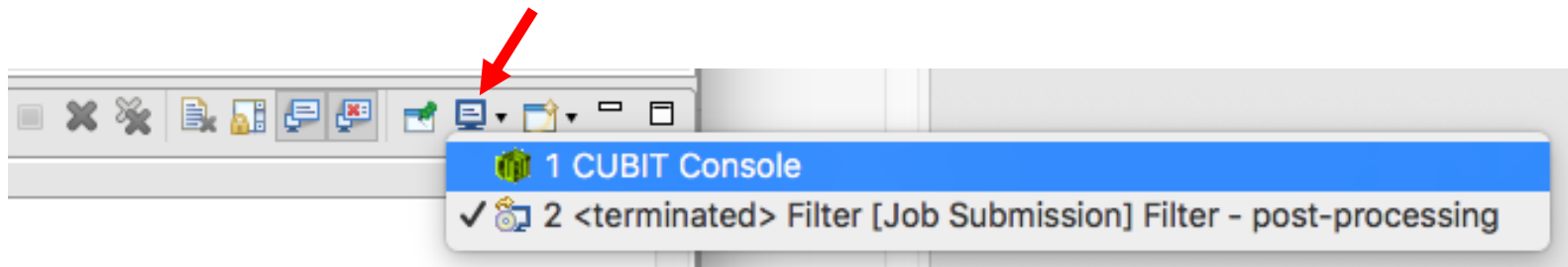
set blocks

block 1 volume all



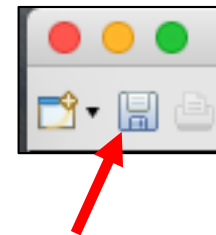
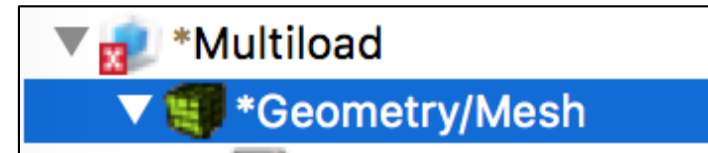
Hint: Cubit Console

- 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.



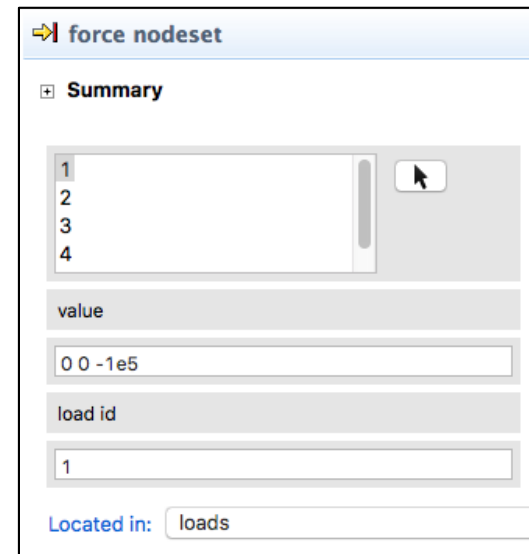
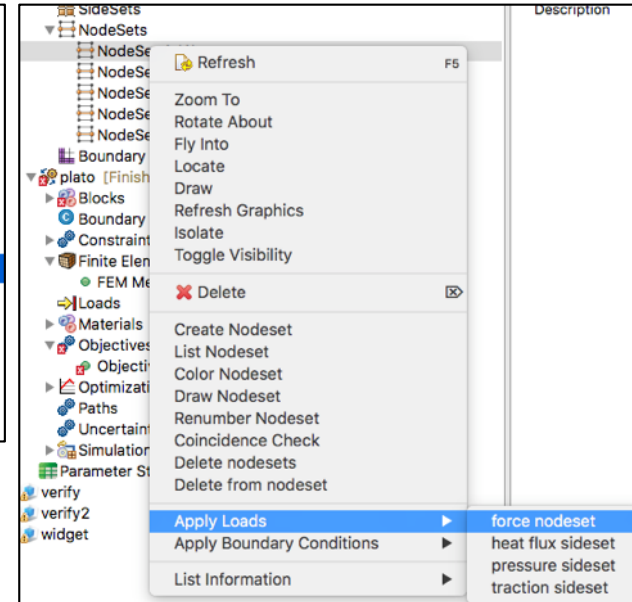
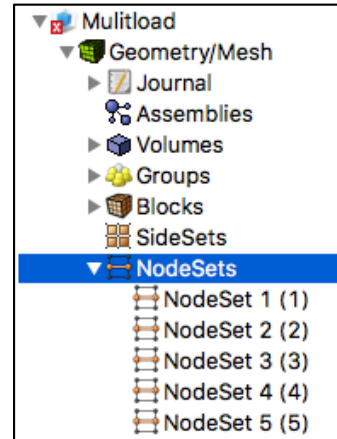
Save Geometry/Mesh

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



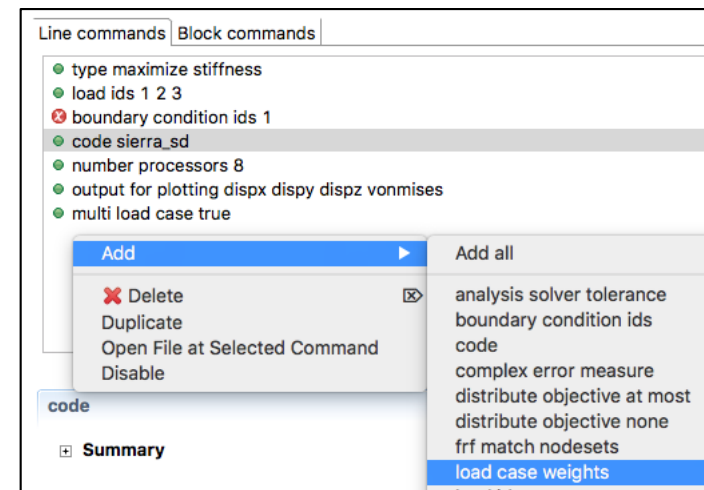
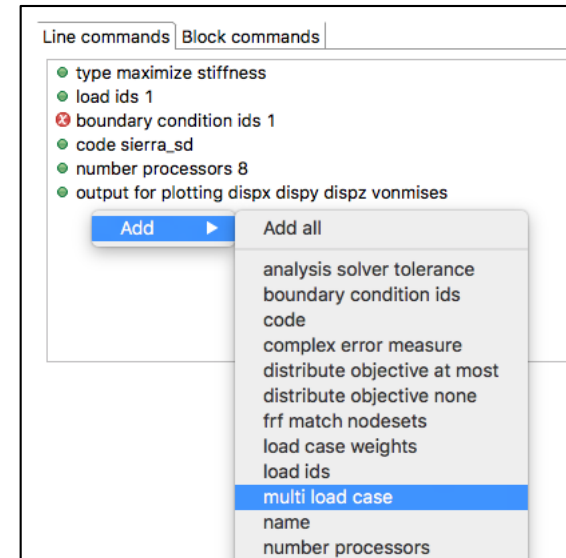
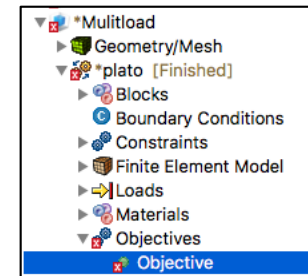
Apply Loads

- The cubit commands entered in the previous step defined 5 nodesets that we will use for applying boundary conditions and loads. We will apply loads using nodesets 1, 2, and 3.
- Expand the **NodeSets** node in the tree as shown.
- Right-click on **NodeSet 1** and choose **Apply Loads->force nodeset**.
- In the Settings panel enter “0 0 -1e5” in the “**value**” box and hit the tab key. Then enter “1” in the “**load id**” box and hit return.
- Repeat this for nodesets **2** and **3** using the same load value but “2” and “3” for the load ids respectively.



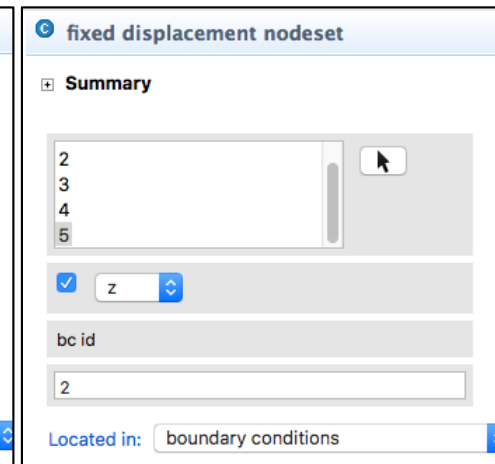
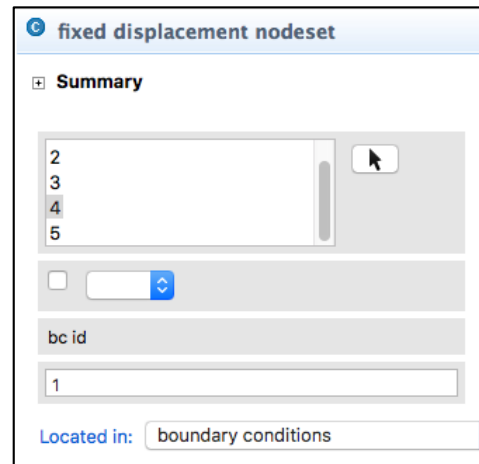
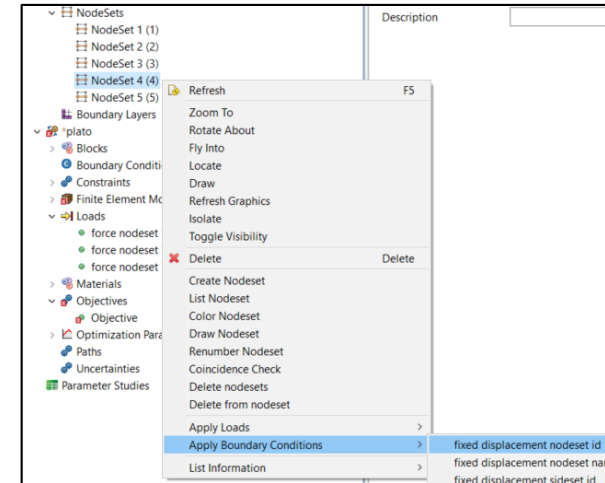
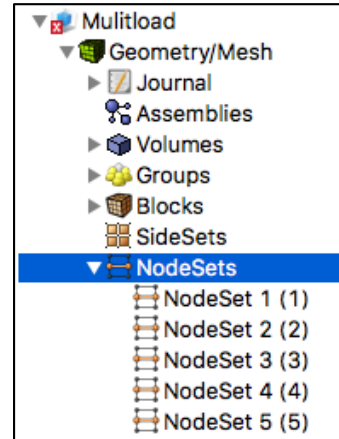
Apply Loads (cont.)

- This example will be run as a “multi-load case” problem meaning that each load case will be run independently and the results from each will be aggregated at each iteration.
- Expand the **Objectives->Objective** node in the tree as shown.
- Right-click anywhere in the parameter area in the settings panel and choose **Add->multi load case**.
- Select “**true**” for the **multi load case** value.
- Click on “**load ids**” in the parameter list and enter “**1 2 3**” for the value. This indicates that our objective will use the 3 loads we just defined.
- Finally, right-click anywhere in the parameter area in the settings panel and choose **Add->load case weights** and enter “**1 1 1**” for the load case weights value.



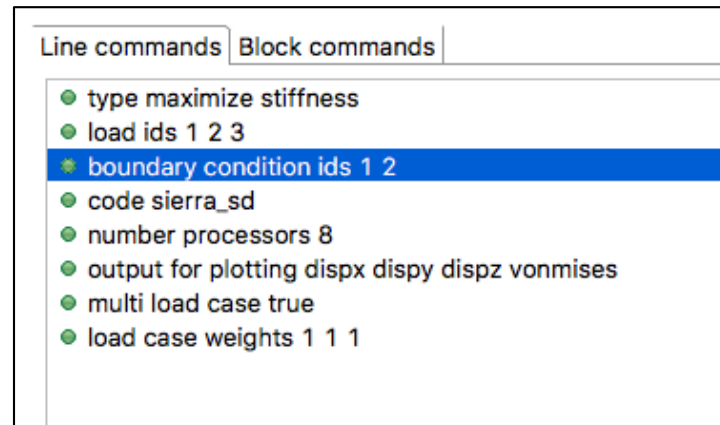
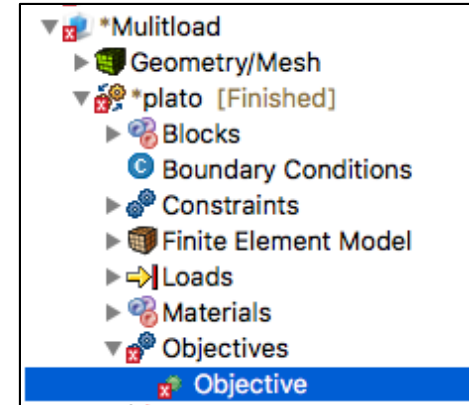
Define the boundary conditions

- Expand the **NodeSets** node in the tree as shown.
- Right-click on **NodeSet 4** and choose **Apply Boundary Conditions->fixed displacement nodeset id**.
- In the Settings panel enter “1” in the “**bc id**” box and hit return.
- Right-click on **NodeSet 5** and choose **Apply Boundary Conditions->fixed displacement nodeset id**.
- In the Settings panel select the checkbox and then choose “z” in the dropdown box indicating that nodeset 5 will only be fixed in the z direction. Then enter “2” in the “**bc id**” box and hit return



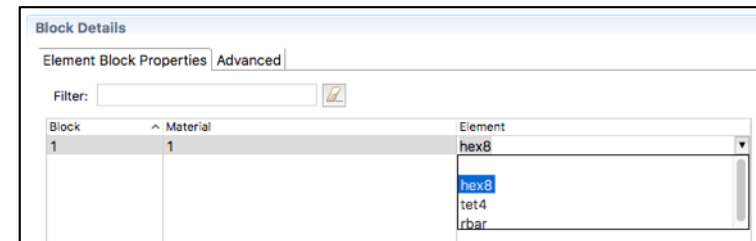
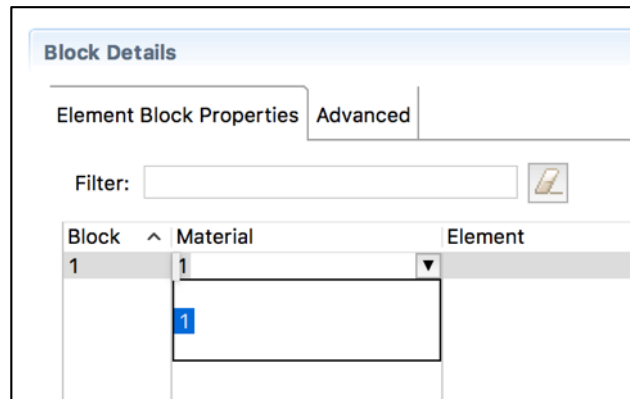
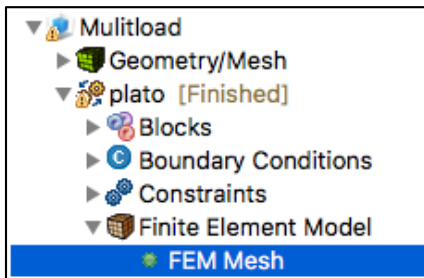
Define boundary conditions (cont.)

- Expand the **Objectives->Objective** node in the tree as shown.
- In the Settings panel click on “**boundary condition ids**” in the parameter list and enter “**1 2**” for the value. This indicates that our objective will be using the two boundary conditions we just set up.



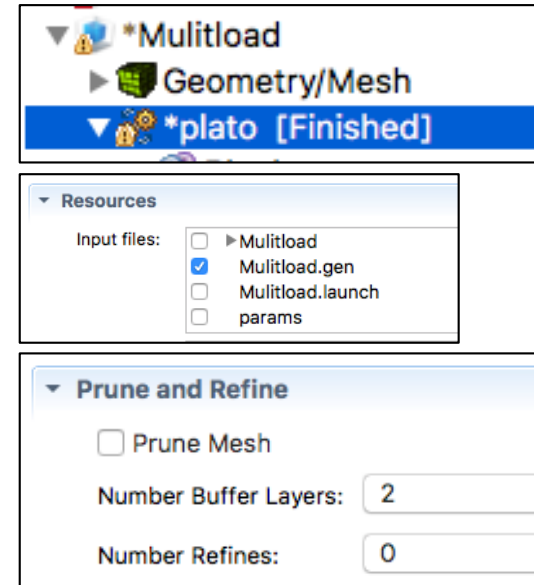
Assign Material and Element Type

- 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



Run the Optimization

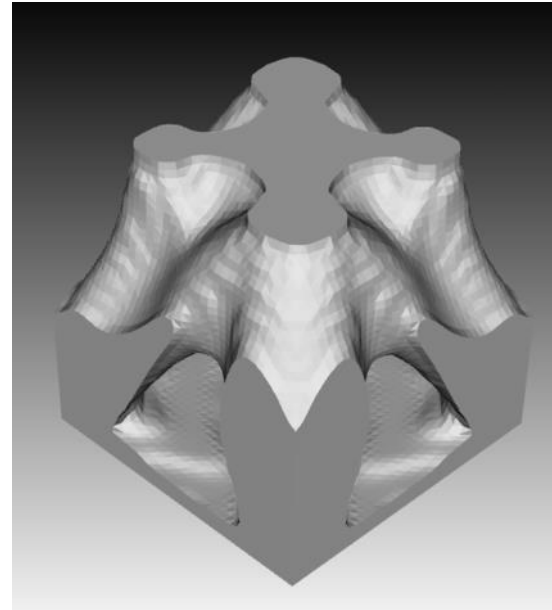
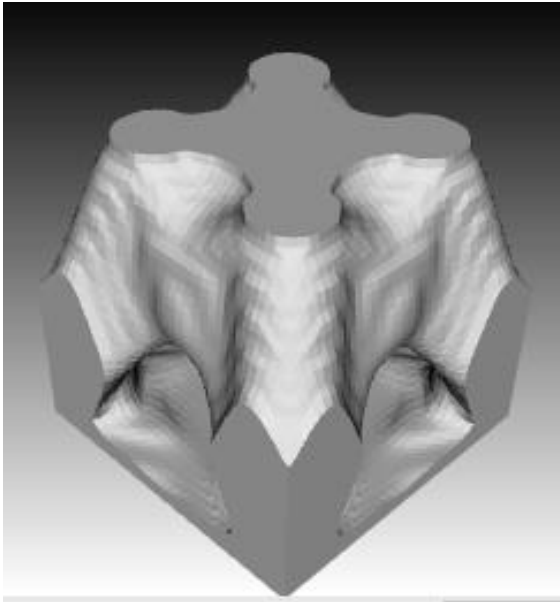
- Click on the **plato** node in the tree to bring up the job submission panel in the **Settings** view
- Choose **Plato** as the code and then choose the machine and execution template you want to use



- In the **Resources** area make sure **Multiload.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 will begin to populate after submitting the job.



- TIP: This example (left) shows where the load cases were weighted equally. To repeat using non-equal weighting on the load cases try changing “**Objectives->Objective->load case weights**” to be “**5 15 1**”. Results for this change are shown on the right.
- 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.