

RBAR Tutorial

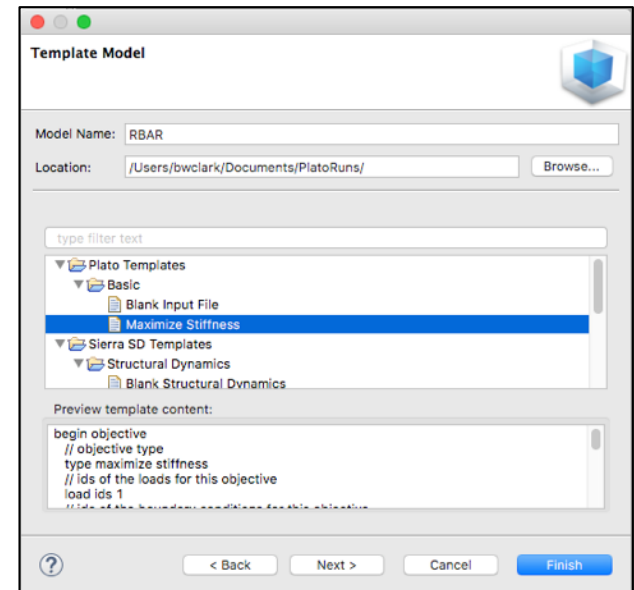
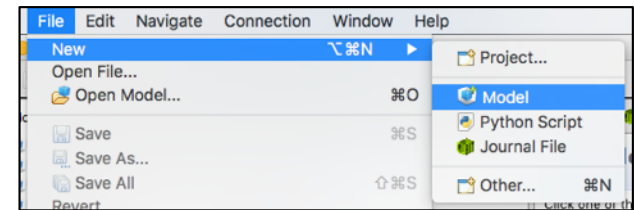
Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525. SAND2020-10947 TR

RBAR Defined

RBAR is an element type specific to the Sierra-SD physics code and as a result this tutorial is only applicable when using Sierra-SD (Salinas). It is essentially a “rigid” bar element that has no extension, torsion, or bending. It is often a convenient way to transfer or apply loads.

Create a New Model

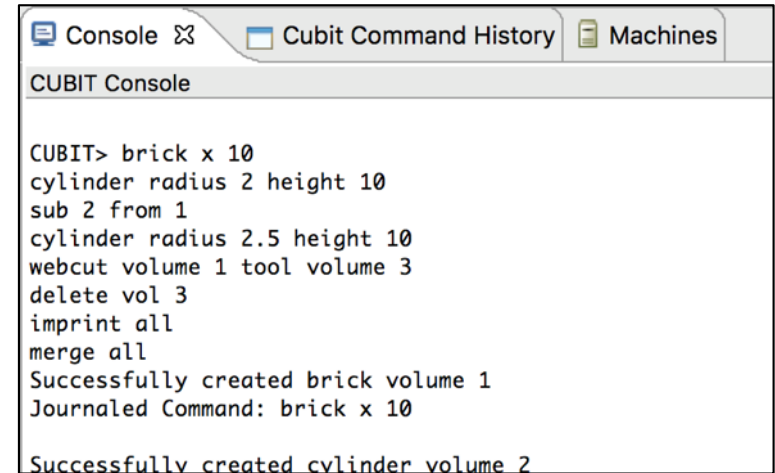
- Choose **File->New->Model** in the menu
- Choose **New Model** then **Next**
- Choose **Create From Template** then **Next**
- Enter **RBAR** as the **Model Name**
- Choose the **Plato Templates->Basic->Maximize Stiffness** template and then **Finish**



Create the Design Domain

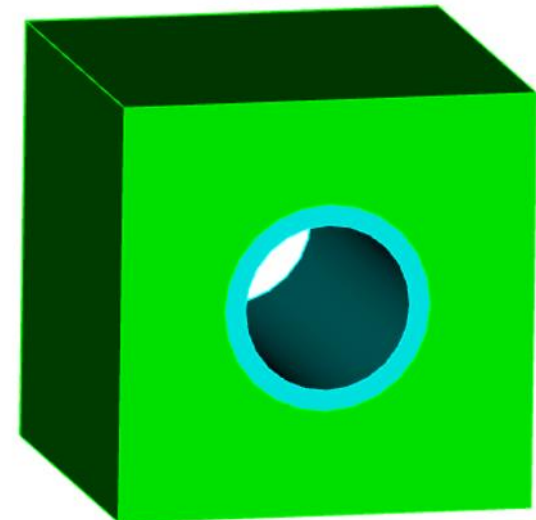
- At the **CUBIT** command prompt copy and paste the following commands (see Hint on next slide):

```
brick x 10
cylinder radius 2 height 10
sub 2 from 1
cylinder radius 2.5 height 10
webcut volume 1 tool volume 3
delete vol 3
imprint all
merge all
```



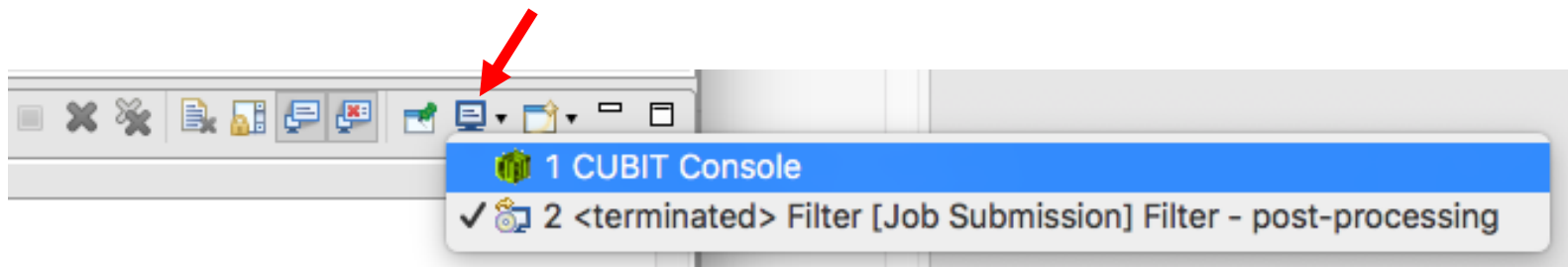
The screenshot shows the CUBIT software interface with three tabs: 'Console', 'Cubit Command History', and 'Machines'. The 'Console' tab is active, displaying the following text:

```
CUBIT> brick x 10
cylinder radius 2 height 10
sub 2 from 1
cylinder radius 2.5 height 10
webcut volume 1 tool volume 3
delete vol 3
imprint all
merge all
Successfully created brick volume 1
Journaled Command: brick x 10
Successfully created cylinder volume 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.



Create the Mesh and RBARs

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

create vertex 0 0 0

vol all size .5

mesh vol all

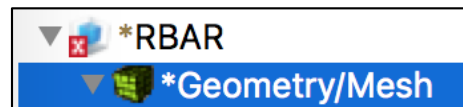
mesh vert 19

block 1 joint vert 19 spider surf 10

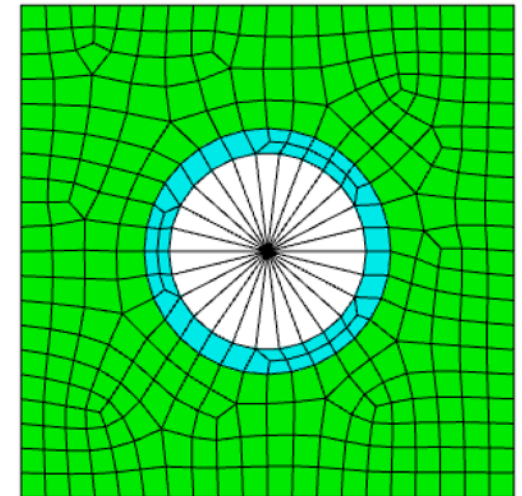
block 2 vol 4

block 3 vol 1

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

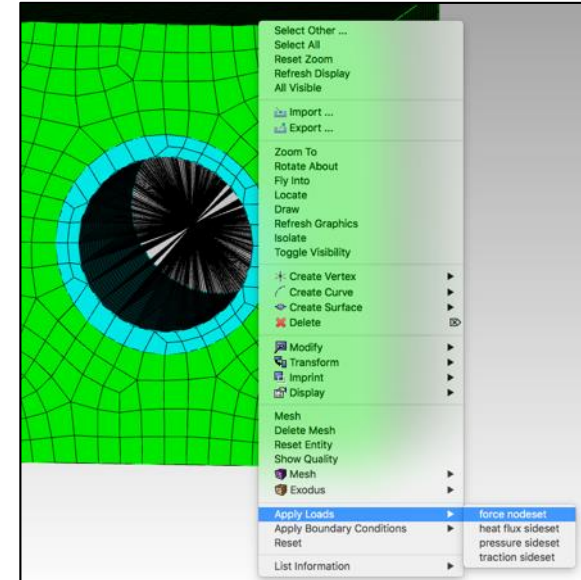


- The “block ... joint” command creates the block with RBAR elements connecting the point 0,0,0 and every node on the interior of the cylindrical surface.
- We will apply our loading to the single vertex at 0,0,0 and the RBAR elements will transfer that load to the interior of the cylindrical surface.



Apply a Force Load

- Select and right-click the vertex (see Hint on next slide) in the middle of the cylindrical surface and choose **Apply Loads->force nodeset** (the vertex is the one that all of the RBAR elements are connected to).
- 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 the return key.



force nodeset

Summary

1

value

0 0 -1e5

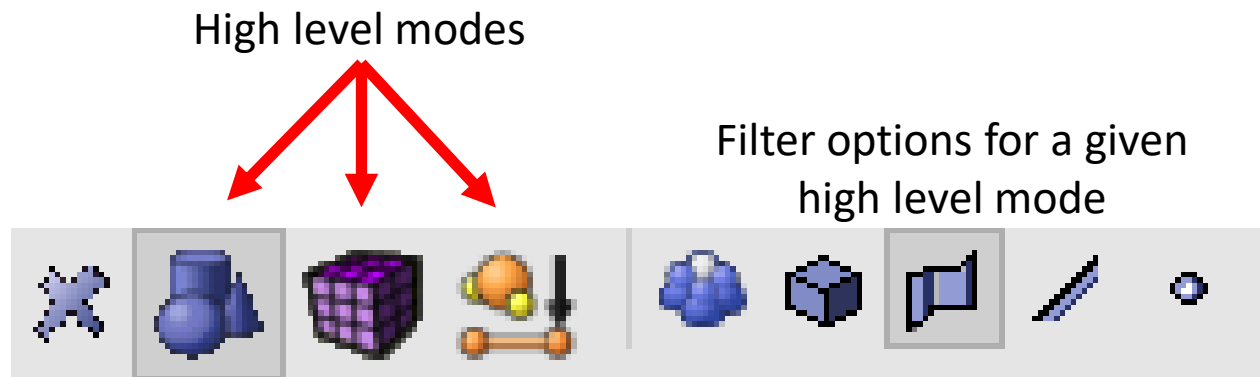
load id

1

Located in: loads

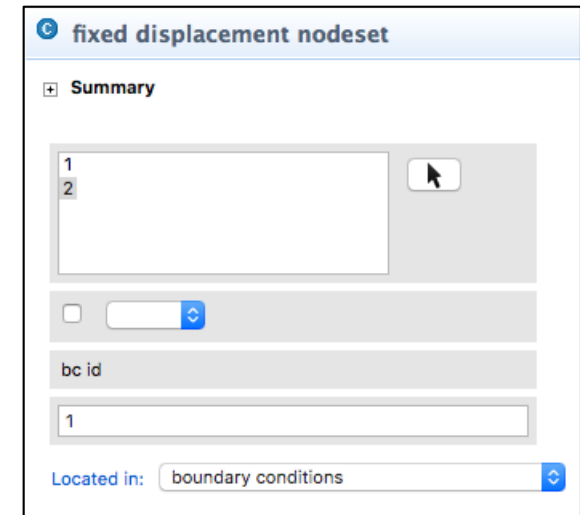
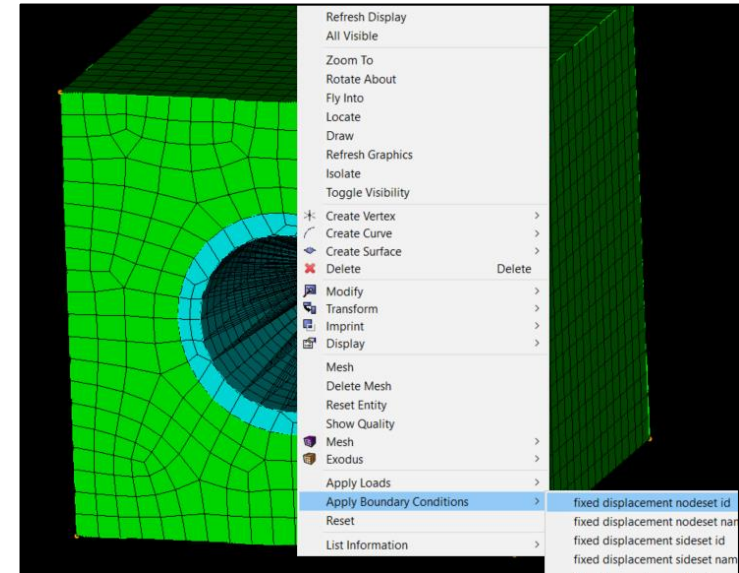
Hint: Entity Selection

- 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.



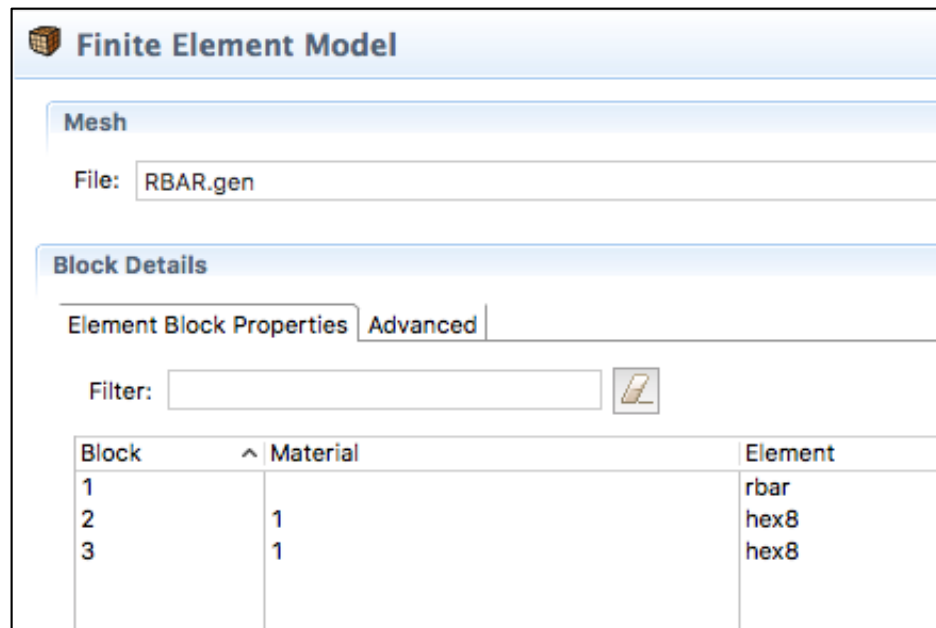
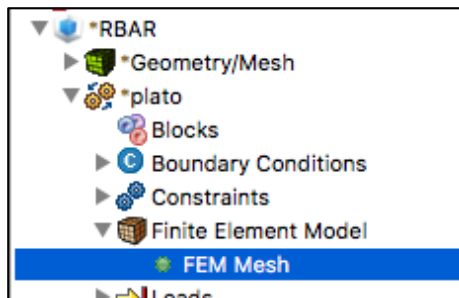
Create a Fixed BC

- Select all 8 vertices at the corners of the cube using multi-select (Ctrl-select on Windows/Linux, Command-select on Mac) and then 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.



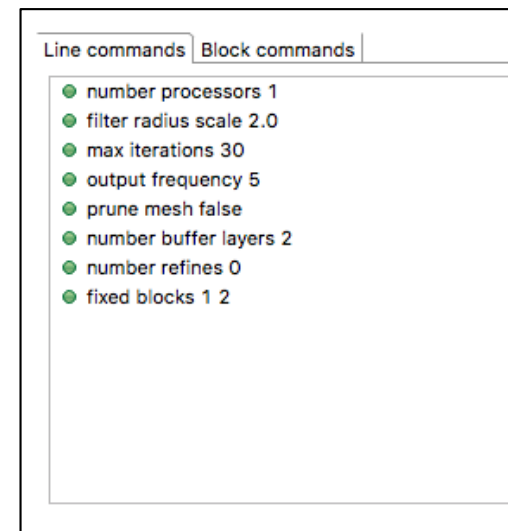
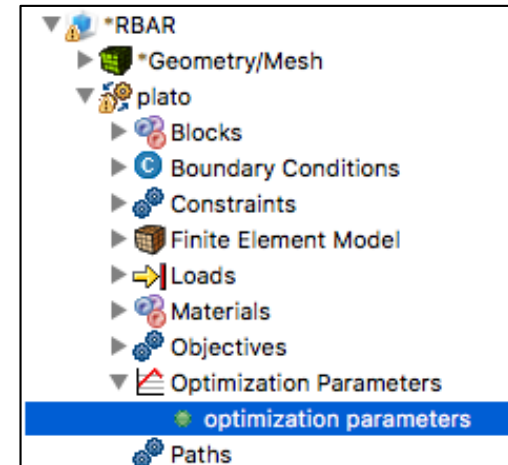
Assign Material and Element Type

- 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 Blocks 2 and 3 and select **1** from the dropdown list. Don't set a material for Block 1 since it will be the RBAR block. If the material for Block 1 is already set to 1 then just click on it and select the blank entry from the dropdown list.
- Then click in the **Element** area next to Block 1 and select **rbar** from the dropdown list.
- Then click in the **Element** area next to Blocks 2 and 3 and select **hex8** from the dropdown list.



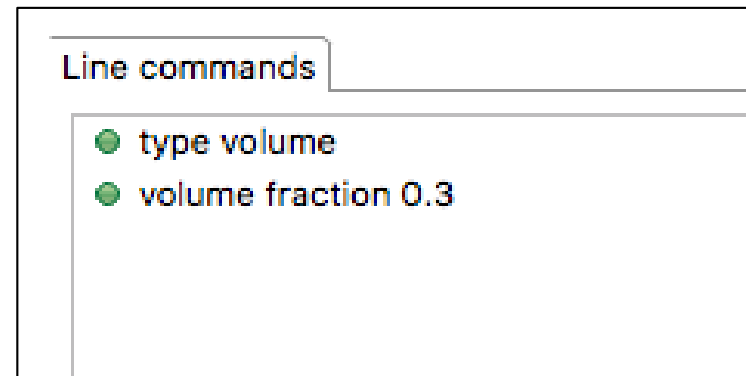
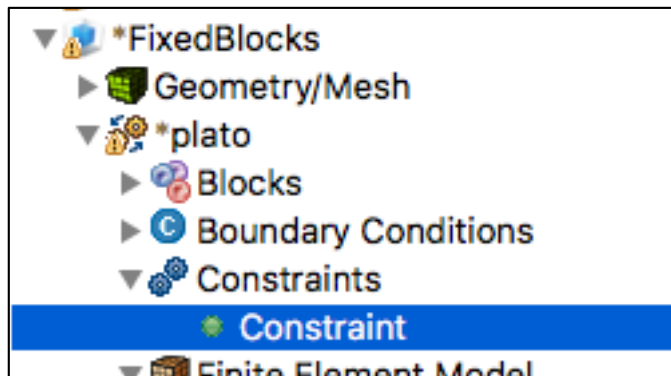
Set the Top. Opt. Parameters

- Click on the **Optimization Parameters-> optimization parameters** node in the tree
- In the **Settings** panel right-click anywhere in the **Parameters** area and choose **Add->fixed blocks**. Enter “1 2” (1 and 2 separated by a space) for the value of the **fixed blocks** parameter and hit the return key. This tells Plato to treat blocks 1 and 2 as fixed and as a result they will be respected as part of the final result. Block 1, the RBAR block, must be set as fixed so the optimizer does not try to perform optimization on the RBAR elements. As they are not volume elements removing or adding material does not make sense.
- In the **Parameters** area set the following parameter values by clicking on them and then entering the value in the edit field:
 - **filter radius scale = 2.0**
 - **max iterations = 30**



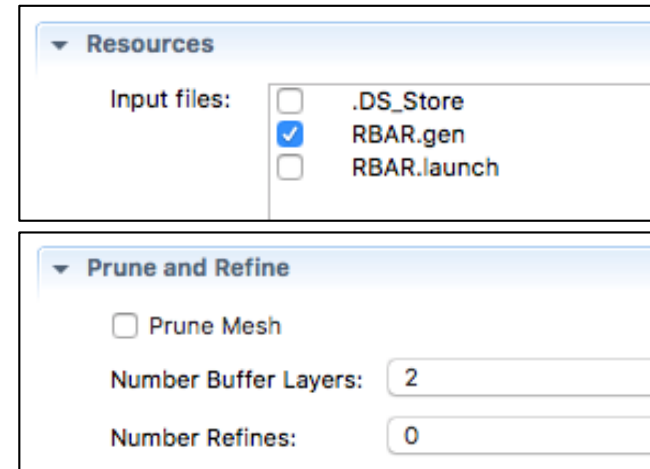
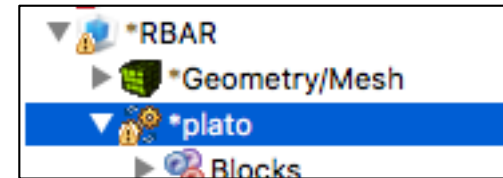
Set the Volume Fraction

- 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 **RBAR.gen** is checked. This is the mesh file that will be used in the run.
- In the **Prune and Refine** area make sure **Prune Mesh** is unchecked and **Number Refines** is **0**. We will not be using pruning and refining in this example.
- Choose any other preferences and launch the job by clicking on **Submit Simulation Job** toward the top of the panel

Results

The results should look something like that shown below. 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.

