

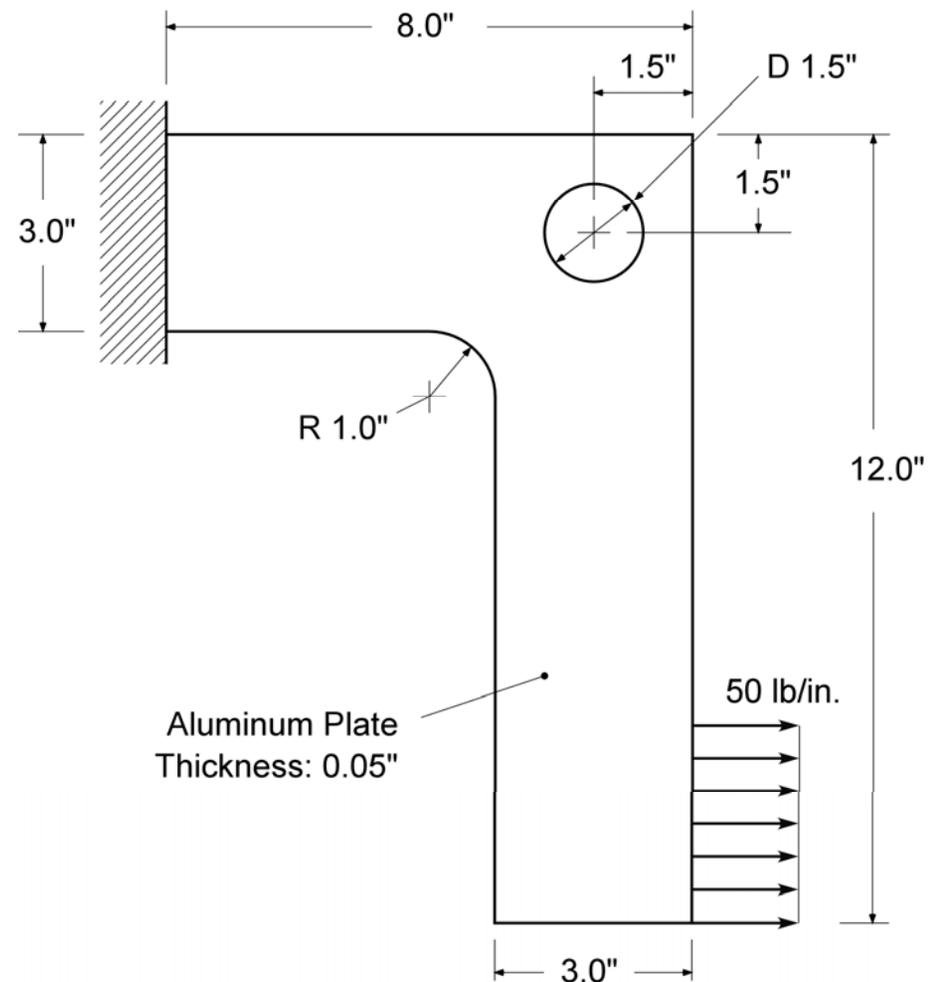
# ABAQUS/CAE Tutorial: Analysis of an Aluminum Bracket

Hyonny Kim

last updated: August 2004

In this tutorial, you'll learn how to:

1. Sketch 2D geometry & define part.
2. Define material properties.
3. Apply loads and boundary conditions.
4. Mesh.
5. Run analysis.
6. View results.



# Helpful Tips Before Getting Started

Use **Exceed 9.0** or equivalent PC terminal software.

## HELP

Online help manuals: *abaqus\_aae doc &* - there is a “book” for CAE: “ABAQUS/CAE User's Manual”. Context sensitive help is also available within CAE.

CAE creates the .inp file which you can edit and run by the command line, or you can submit jobs from within CAE. Other files are .cae (CAE model file), .odb, .dat, .log, .msg, and .sta. The .dat is the text output file that will contain results. The .odb file is the binary output file that will be read during post-processing to view graphical results. The .log file keeps a text record of all processes and is useful for checking the status of the analysis. The .msg lists the progress of the analysis, as well as provides some messages about why an analysis might have crashed (this information is often within the .dat file as well). The .sta file is a summary of the information contained in the .msg file, and is useful for monitoring the status of long-running jobs during their computation.

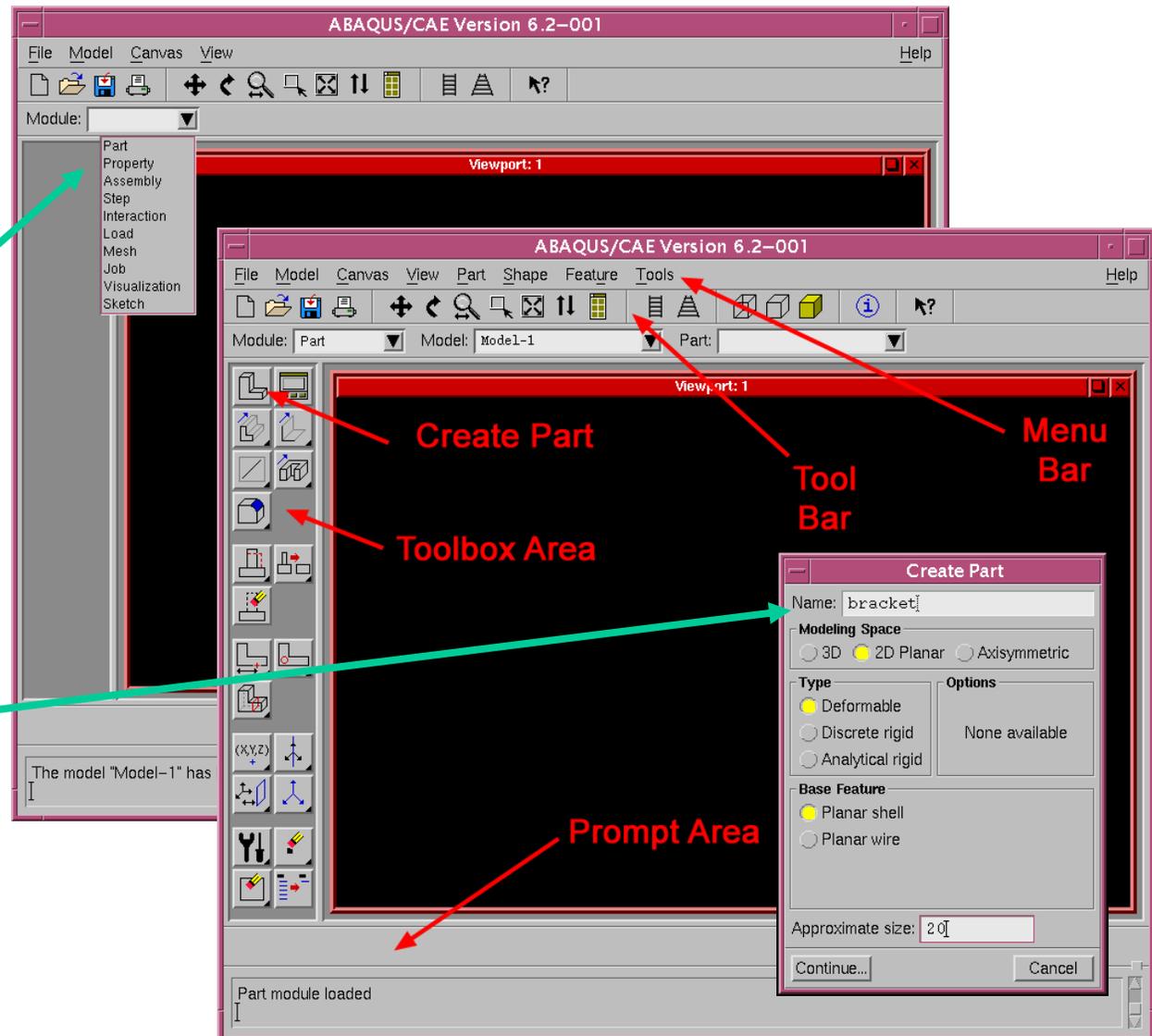
## MOUSE

Use of the Mouse:

- button 1 (left) is select, button 2 (right) gives menu, button 3 (middle, if available is “enter” or “done”)
- multiple items can be selected by: “dragging” a window or holding the SHIFT key while picking
- items can be de-selected by holding the CTRL key.

# ABAQUS/CAE: Getting Started, Create Part

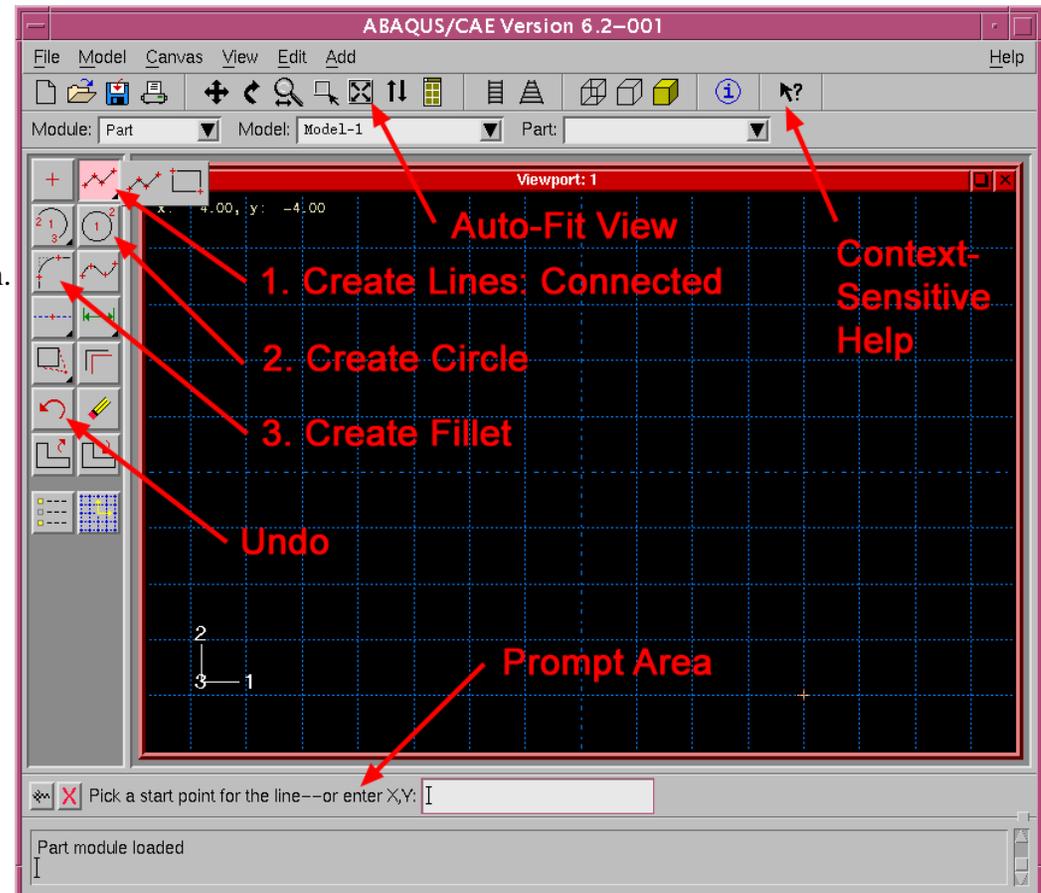
- To run ABAQUS/CAE, first go to the directory you wish your files to be located, then type: `abaqus_aae cae`  
or  
`/usr/site/aae/bin/abaqus_aae cae`
- click **Create Model Database**
- In the **Module** dropdown box, select **Part** (this takes about 30 seconds for the program to initialize)
- Note the locations of: **Tool Bar**, **Menu Bar**, **Toolbox Area**, **Prompt Area**. These will be referred to repeatedly in the future.
- In the *Toolbox Area*, click **Create Part** button. The *Create Part* window will pop up.
- Enter in name, e.g., *bracket*
- Under *Modeling Space*, choose **2D Planar**
- Base Feature*, **Shell**
- Approximate size*: **20**
- click **Continue...**



# Sketch Part

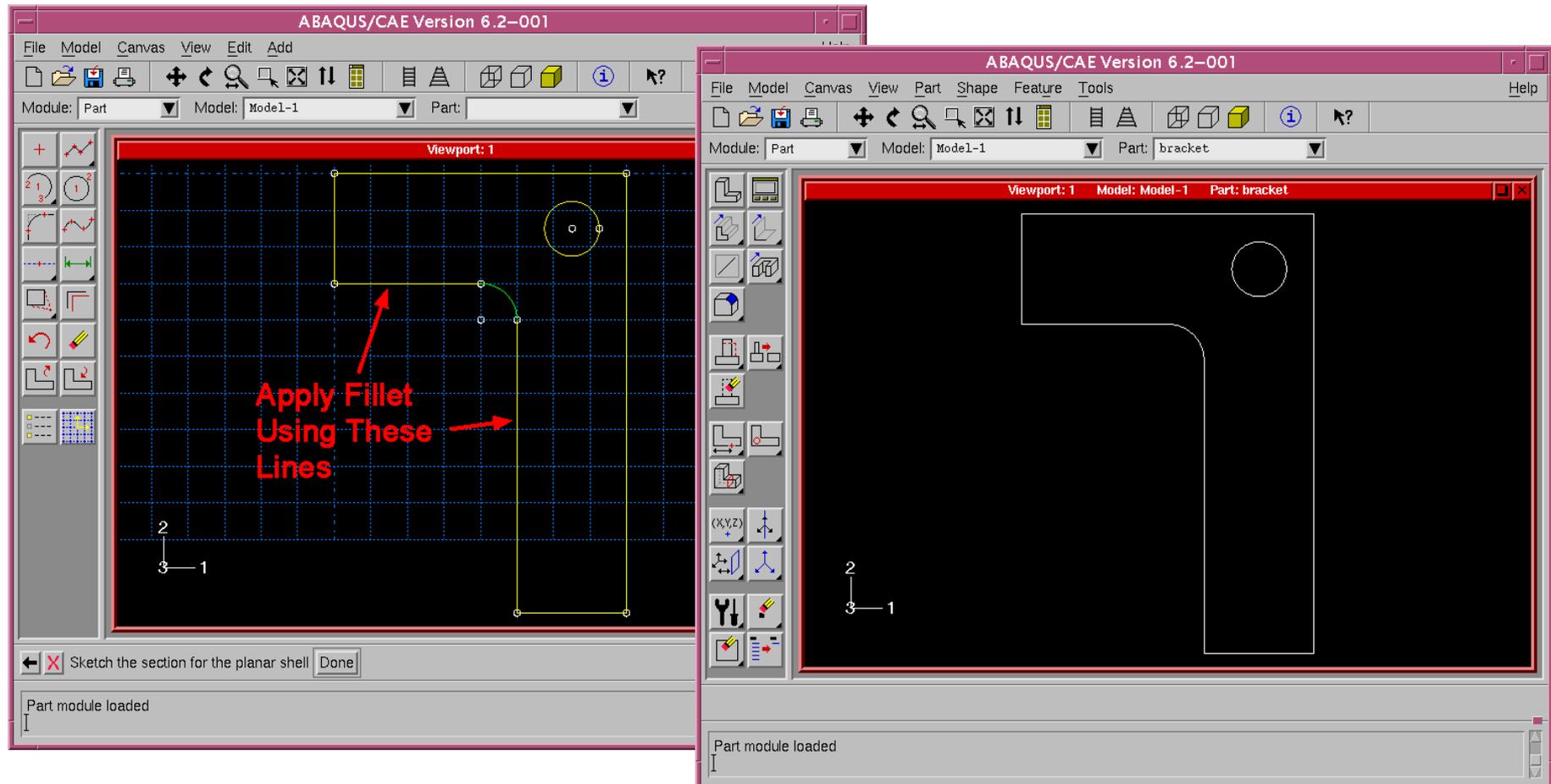
- The window will change to that shown at right. Note the buttons pointed out.
    - Buttons with a dark triangle will provide more button choices when clicked and held. Try it.
    - Float your mouse pointer over buttons, it will give a pop-up description.
    - Context-Sensitive Help is available. Click the help button, then the item you want more info on.
- Click **Create Lines** button. Note it is highlighted when active. In prompt area, enter in the coordinates:
    - 0, 0 <enter>
    - 8, 0 <enter> (it is ok if point is be out of view)
    - 8, -12 <enter>
    - 5, -12 <enter>
    - 5, -3 <enter>
    - 0, -3 <enter>
    - click on point 1 (box will appear on it). Finished product will be yellow outline of the bracket.

Click **Auto-Fit View** button to re-scale image. The other buttons adjacent to this one will adjust panning, rotation, and zoom. Try them out. Dynamic viewing with mouse buttons by holding CTRL + ALT on right side of keyboard. Try it.
  - Click **Create Circle** button. Enter **6.5, -1.5** for *center*, and **7.25, -1.5** for *perimeter point*.
  - Click **Create Fillet** button... (go to next page)



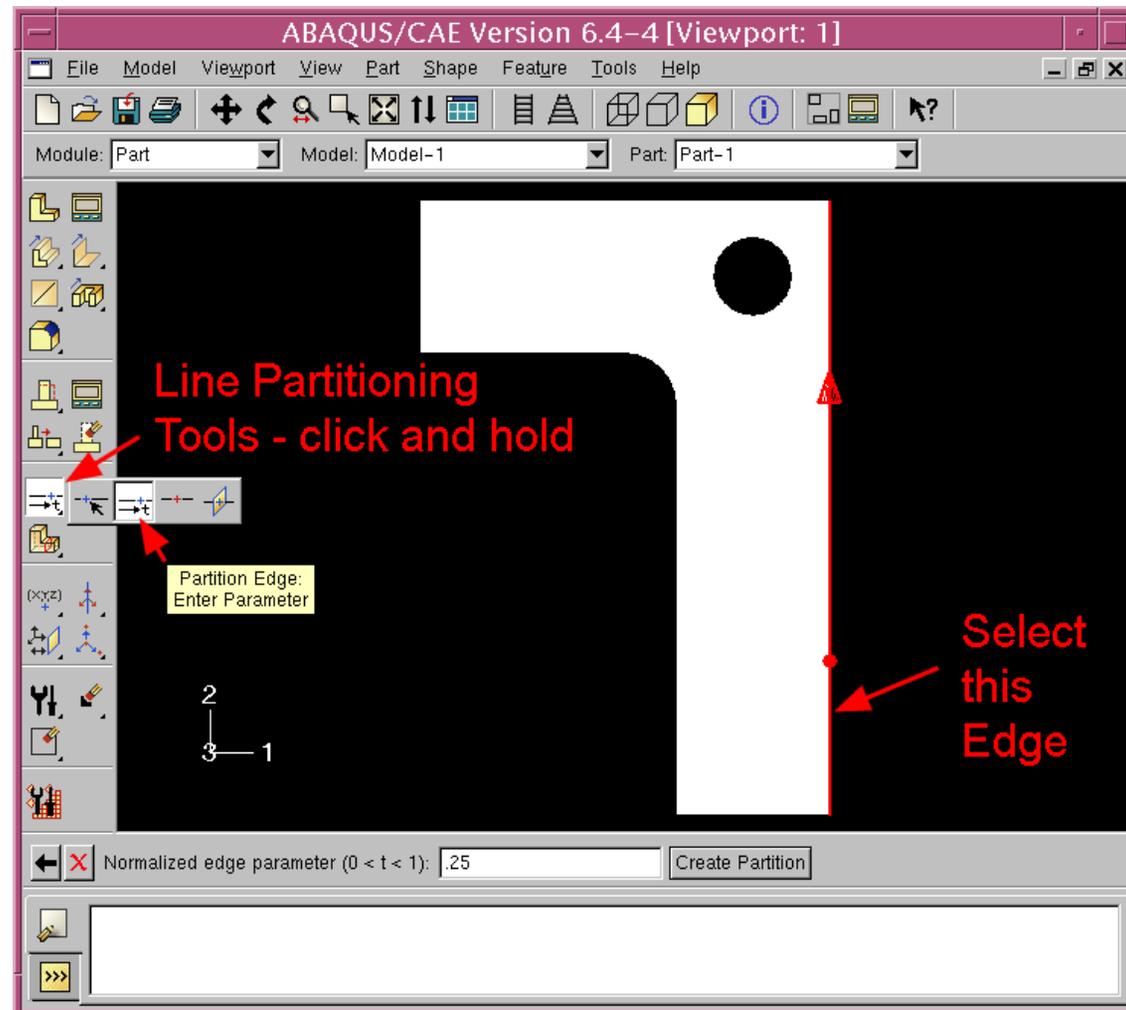
## Sketch Part – contd.

3. cont'd... enter **1.0** for *fillet radius* in the *Prompt Area*, hit enter, then Mouse click on inner two lines when prompted.
  - The **Create Fillet** button should still be highlighted. Click this again to get the screen shown below left.
  - Click the **Done** button in the **Prompt Area** at the bottom of the screen.
  - You now are returned to the **Part Module** screen. This should look like the screen below right. Note different tool buttons shown in the *Toolbox Area*.



# Partition Edge

- Click the “**Partition Edge: Enter Parameter**” button. In order to get this button, you need to click and hold over the line partitioning tools button – note the small dark triangle in the lower right corner indicates that this is an expanding button.
- You will be prompted to select an edge, select the far right-hand edge of the bracket.
- Click **Done**.
- In the Prompt Area, enter in value of **0.25** for the *Normalized edge parameter*.
- Click the **Create Partition** button to finish.
- You will see a large dot one-fourth of the way up the edge of the bracket. This partitioned edge will be used later for applying a uniform load.



# Saving and Defining Material Properties

- Save your work: in the **Menu Bar**, click **File, Save As**.
  - Under *Selection*, enter a name at the end of the path, e.g. *bracket*. Click **OK**. From now on, you can just click the **blue floppy disk icon** in the **Tool Bar**. *Save often!!!*

- Change Modules. In the **Module** drop-down box beneath the **Tool Bar**, select **Property**.

1. Click **Create Material** Button
  - enter a name, e.g. *Aluminum*, select **Mechanical** --> **Elasticity** --> **Elastic**
  - enter **10e6** for *Young's Modulus*, **0.3** for *Poisson's Ratio*.
  - click **OK**

If you want to modify the material, click the **Material Manager** button to the right of **Create Material**, select the material by name and click **Edit**, or click **Dismiss** to leave without making any changes.

1. Create Material

2. Create Section

3. Assign Section

ABAQUS/CAE Version 6.2-001 – Model Database: /home/roger/d/hyonny/FEA/558/bracket.c

Module: Property Model: Model-1 Part: bracket

Viewport: 1 Model: Model-1 Part: bracket

Edit Material

Name: Aluminum

Material Options

Elastic

General Mechanical Thermal Other Delete

Elastic

Type: Isotropic Suboptions

Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

Data

Click mouse button 3 for table options.

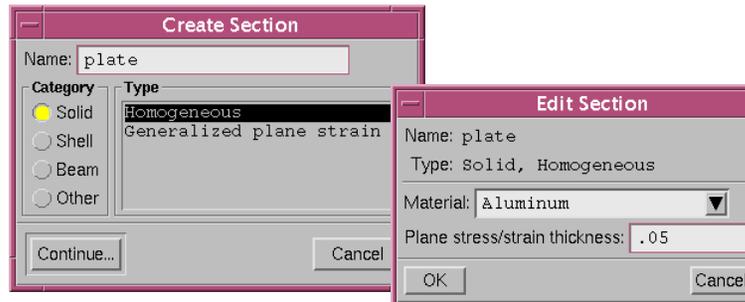
Young's Modulus	Poisson's Ratio
1e+07	0.3

Property module loaded

# Assign Properties to Regions of Model

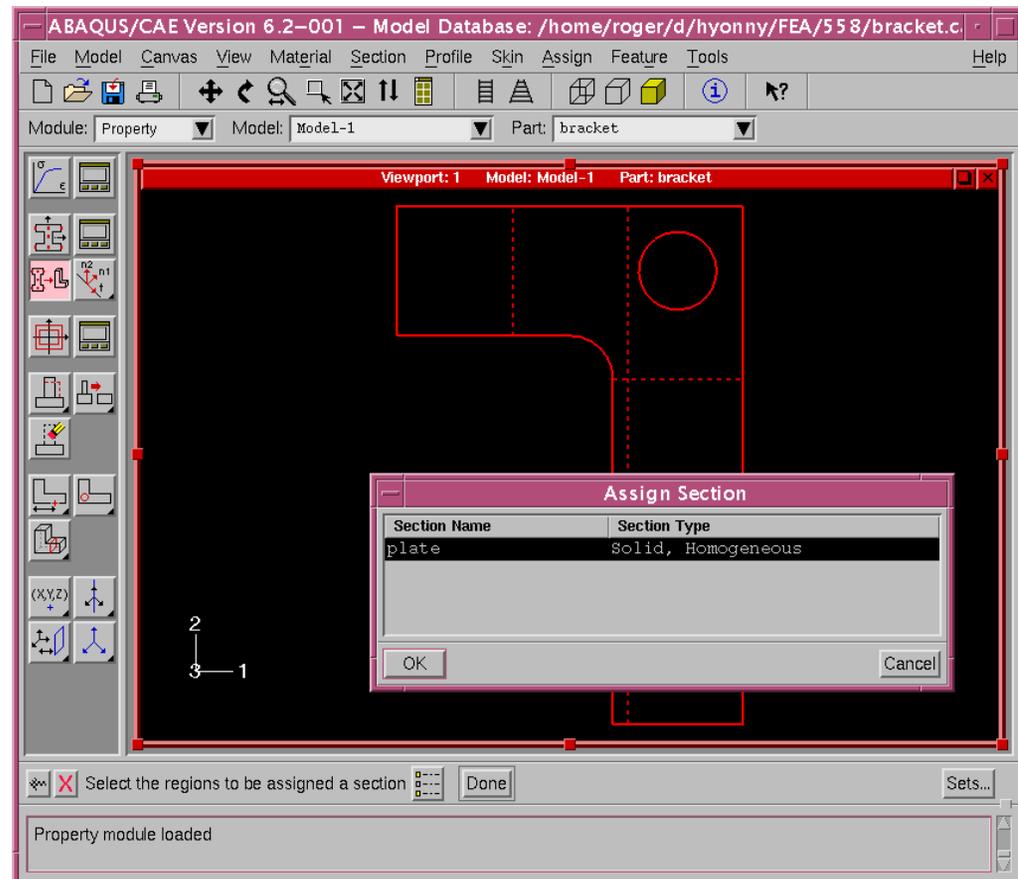
## 2. Click **Create Section** Button

- enter a name, e.g., *plate*
- choose **Solid** and **Homogeneous**
- click **Continue**
- select the material *Aluminum* (or what ever you named it, there should be only one to choose from) in the drop down box
- enter *Plane stress/strain thickness: 0.05*. Click **OK**.



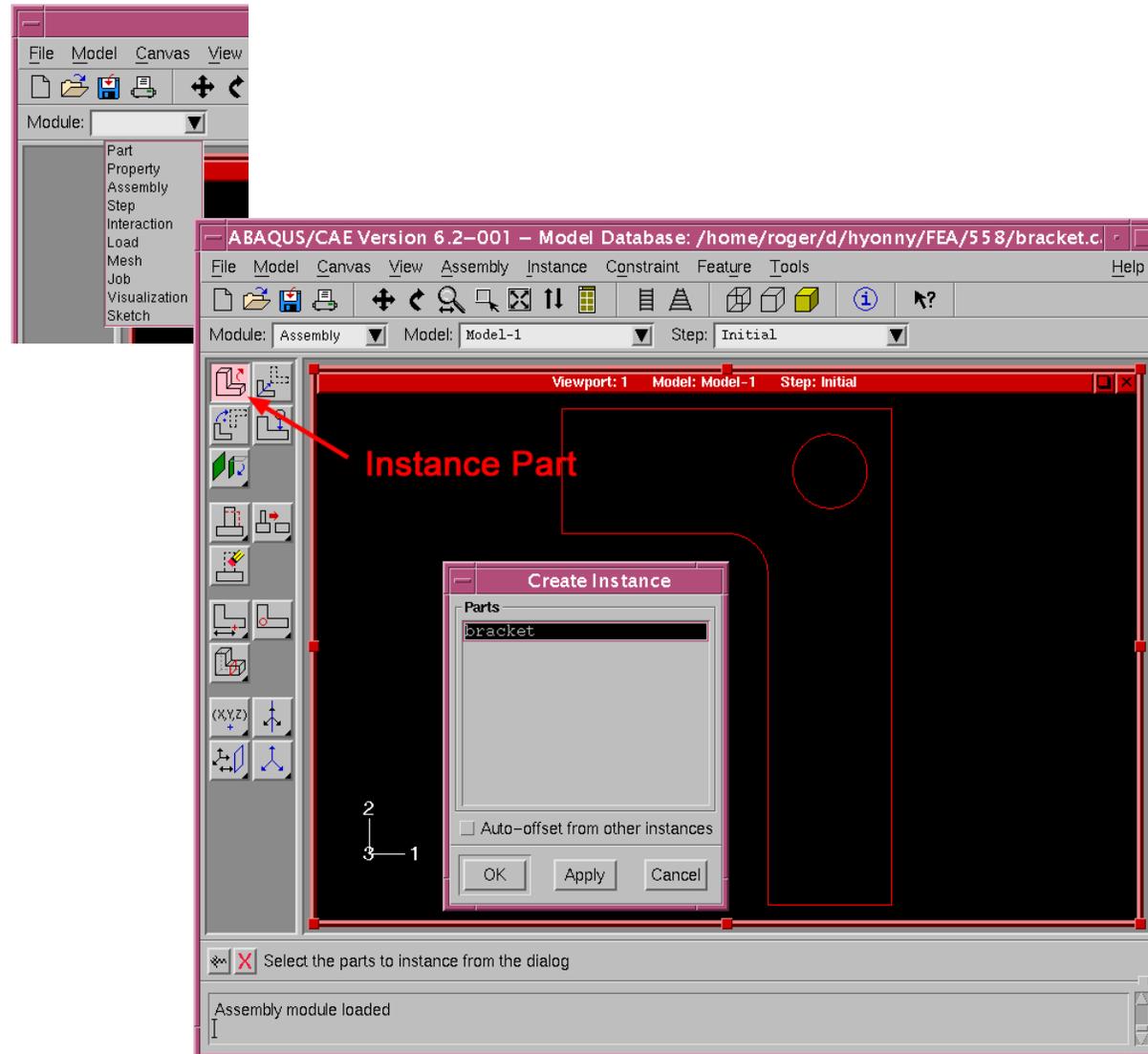
## 3. Click **Assign Section** Button

- you will be prompted to select a region. **Click on the part.**
- Click the **Done** button in the **Prompt Area** at the bottom of the screen.
- The *Assign Section* window will pop up. Select the *Section Name* you wish to assign to this region (there should be only one which you've previously named, e.g., *plate*)
- click **OK**.



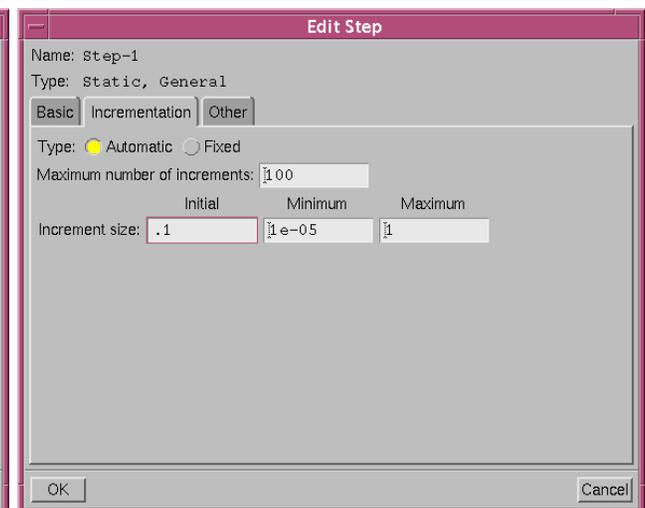
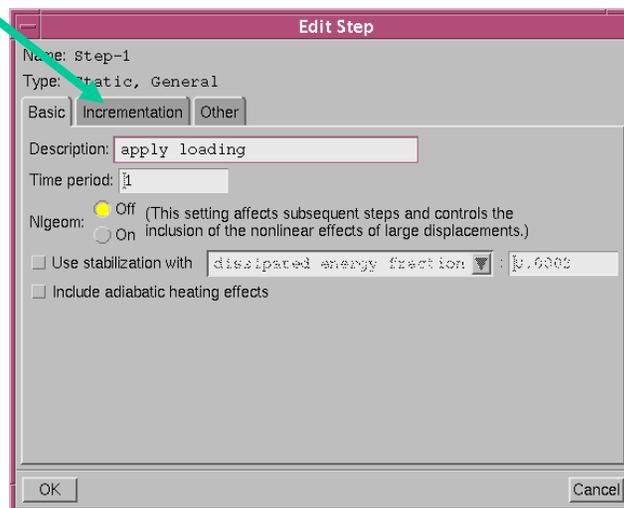
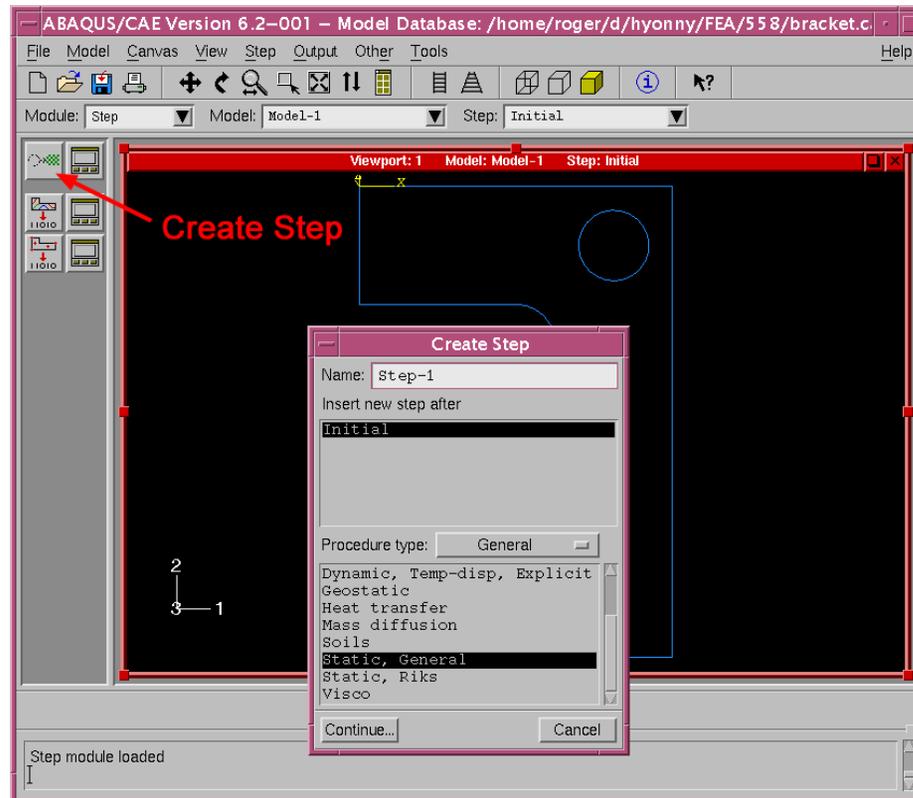
# Instance Part

- Change Modules. In the **Module** drop-down box beneath the **Tool Bar**, select the **Assembly**.
  - note, the *Canvas* (main working graphical window) will be blank.
- Click the **Instance Part** button. The *Create Instance* window will pop up.
- Select the part you wish to instance, e.g., the part we named *bracket* previously. A red outline of the bracket will show.
- Click **OK**.



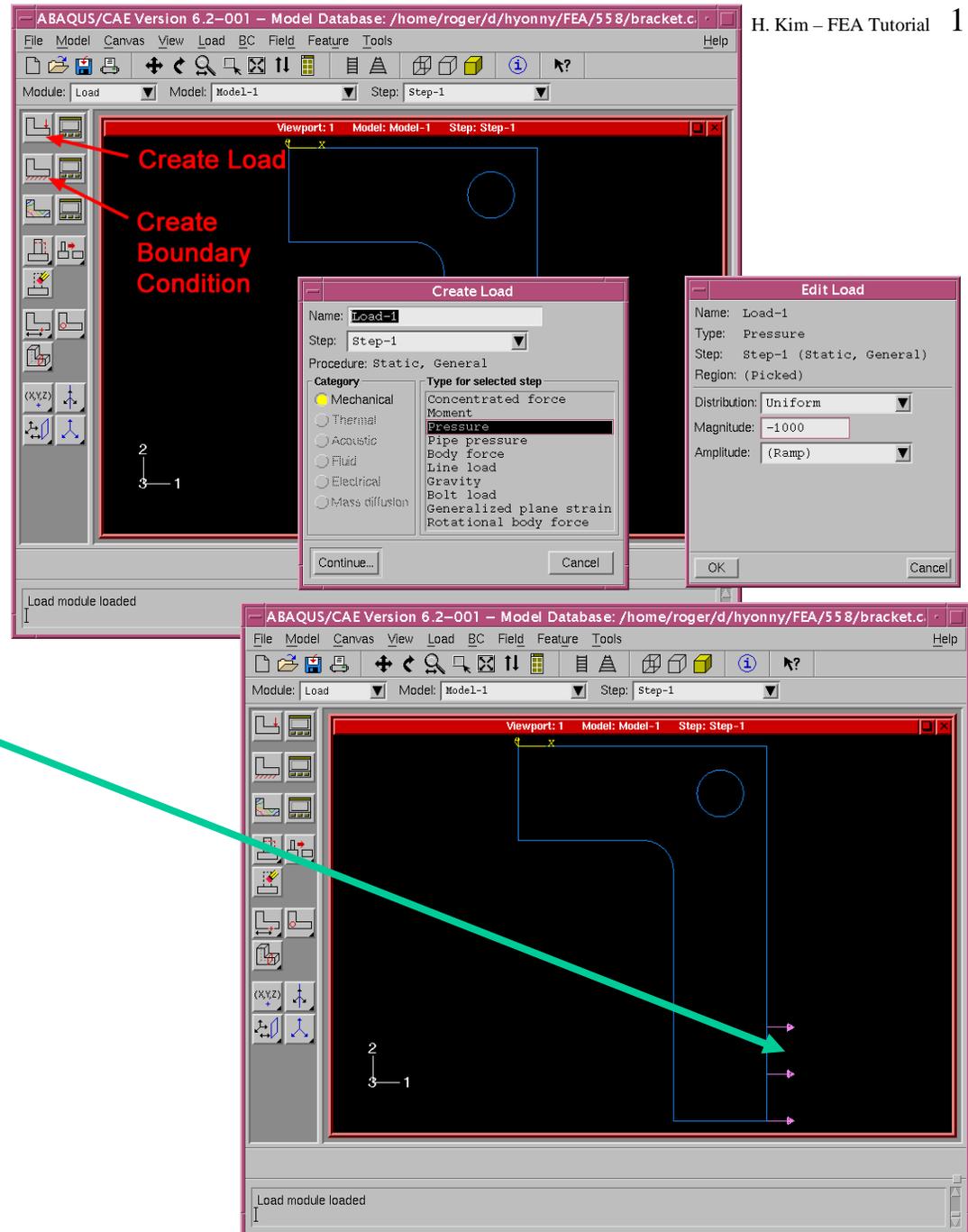
# Step

- Change Modules. Select the **Step** module.
- Click the **Create Step** button.
  - *Create Step* window pops up
  - enter a name – use the default name *Step-1*.
  - be sure *Procedure type* is set to **General**, and **Static**, **General** is highlighted in the list. Click **Continue**.
- *Edit Step* window pops up, with the *Basic* tab active.
  - enter in a *Description*, e.g., *apply loading*
- Click the *Incrementation* tab.
  - under *Increment Size*, enter value of 0.1 for *Initial*. Leave the rest the same. Full load corresponds to an Increment value of 1 (when *Time Period* is set to 1 under the *Basic* tab). Setting *Initial* to 0.1 forces ABAQUS to start the analysis by applying 1/10 of the full load. This can also be left to default 1 value and the software will auto-select.
  - click **OK**.



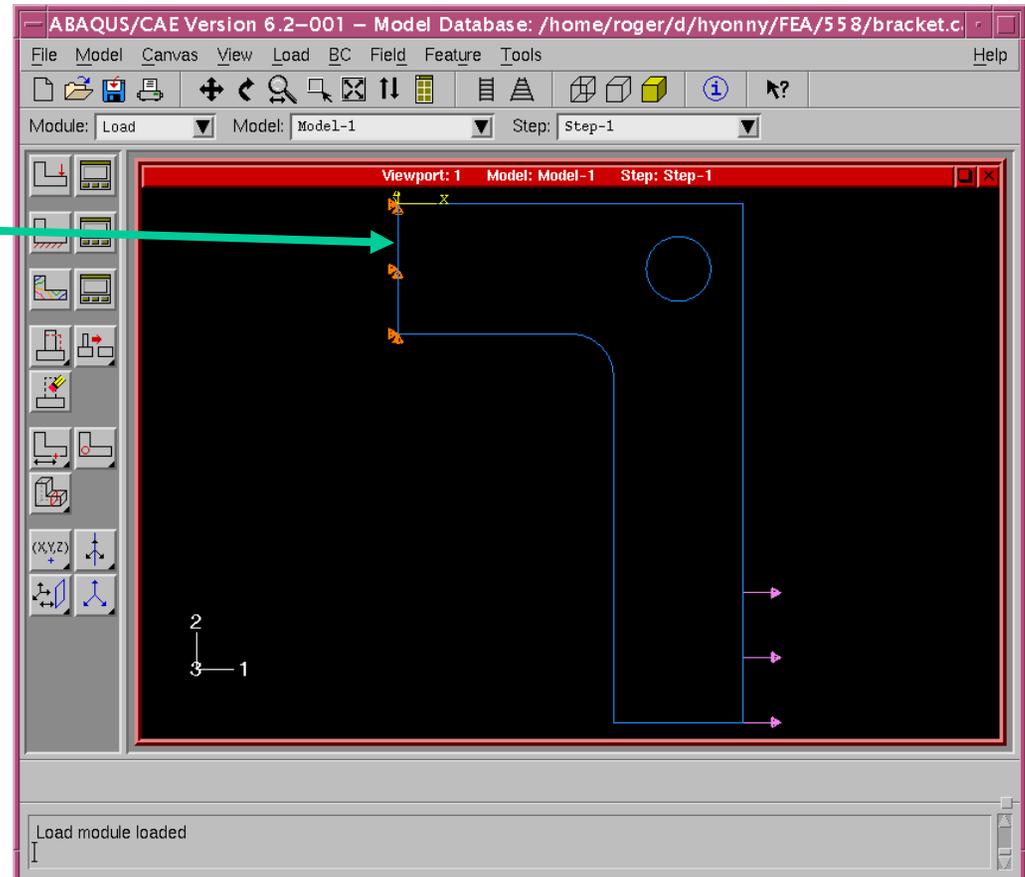
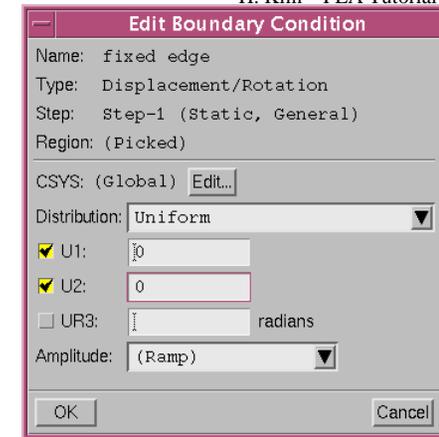
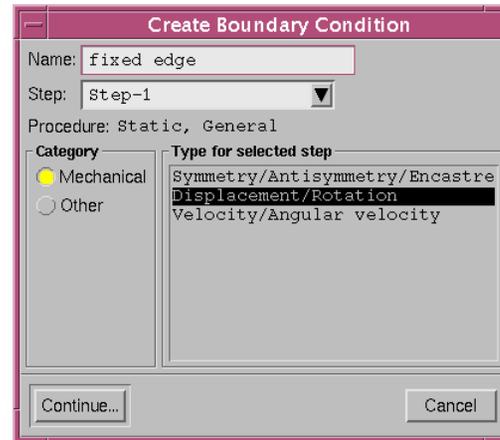
# Load

- Change Modules. Select the **Load** module.
- Click the **Create Load** button.
  - the *Create Load* window pops up.
  - enter a *Name*, e.g., *Load-1* is the default name
  - be sure **Step-1** (or what name you have given it) is selected in the *Step* drop down box.
  - be sure **Mechanical** is selected in *Category*
  - under *Type for selected step*, choose **Pressure**
  - click **Continue**
  - Upon prompting to *Select surfaces*, mouse-pointer click on the lower portion of the right edge of the bracket, the region we partitioned previously. It will highlight red.
  - click **Done**.
  - in the *Edit Load* window that pops up, be sure to have *Distribution* set to **Uniform**, enter value of **-1000** in *Magnitude*, and be sure that **(Ramp)** is selected under *Amplitude*. This is a 1000 psi traction.
  - click **OK**.
  - You should get the image shown to the right. If your arrows are in the wrong direction, you need to go back and be sure to specify a *negative* pressure.



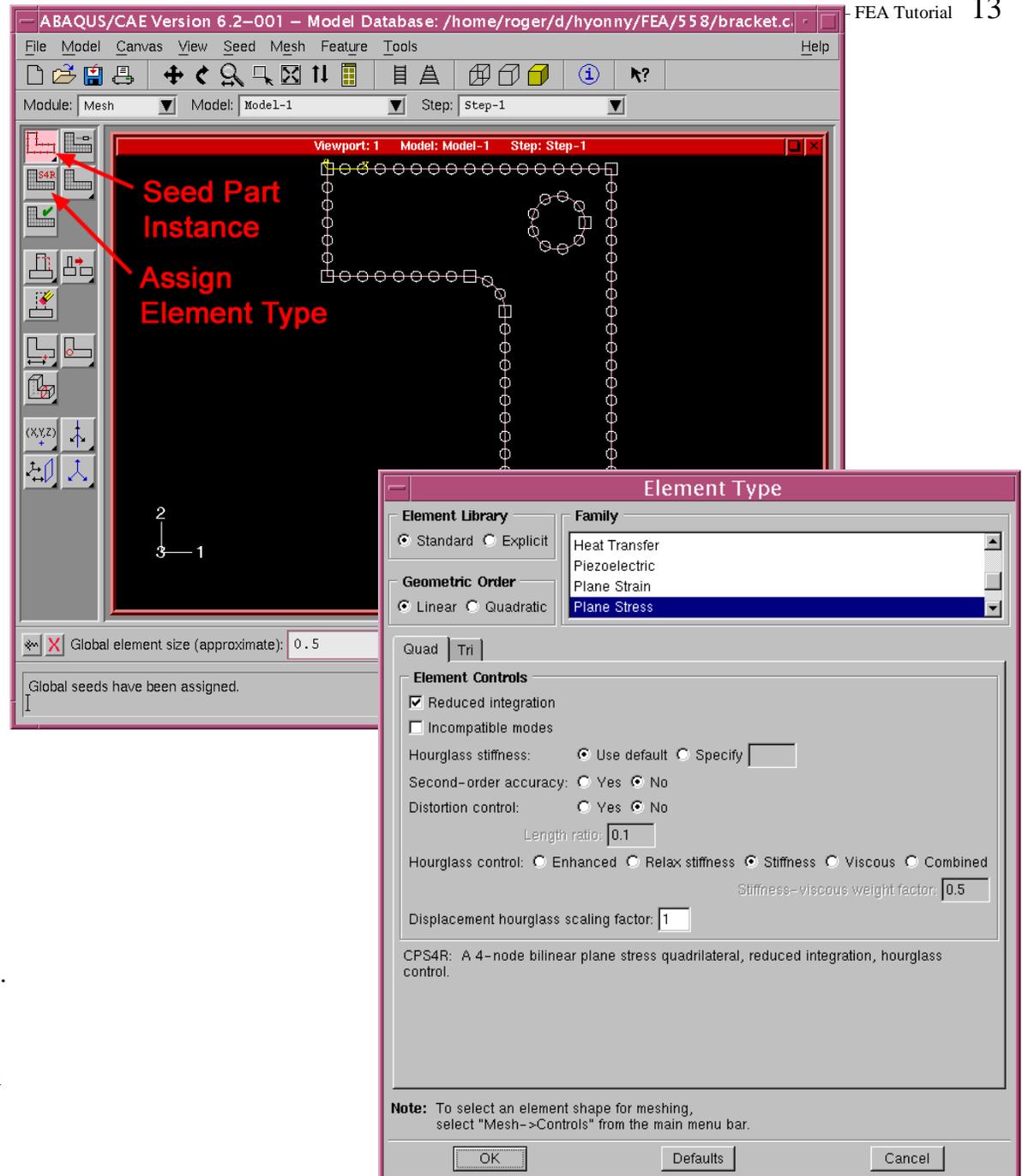
# Boundary Conditions

- Click the **Create Boundary Condition** button.
  - the *Create Boundary Condition* window pops up.
  - enter a name, e.g., *fixed edge*
  - under *Category*, be sure that **Mechanical** is selected.
  - under *Type for selected step*, choose **Displacement/Rotation**
  - click **Continue**.
  - upon prompt to select regions, mouse-pointer click on the upper left vertical edge of the bracket. It will highlight red.
  - click **Done**.
  - *Edit Boundary Condition* window pops up.
  - be sure **Uniform** is selected in *Distribution*.
  - check-mark (click) boxes for **u1** and **u2**, and leave the default values of 0.
  - click **OK**.
  - You should get the image shown to the right



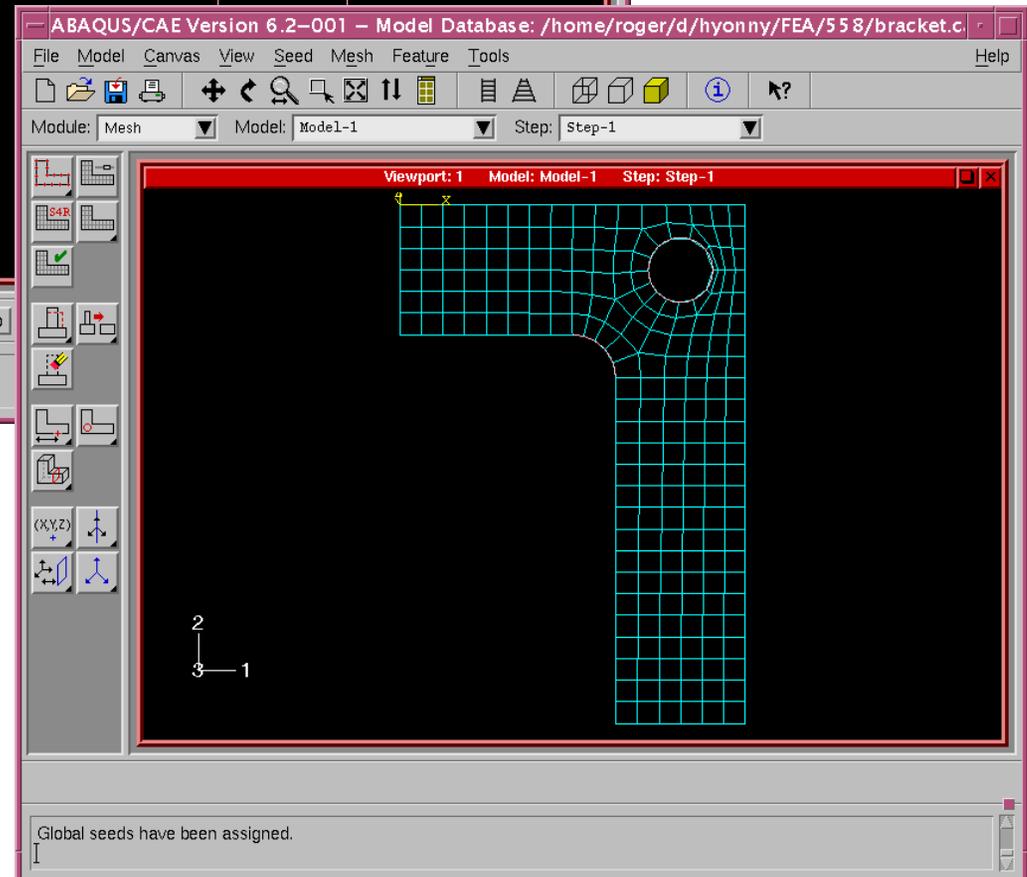
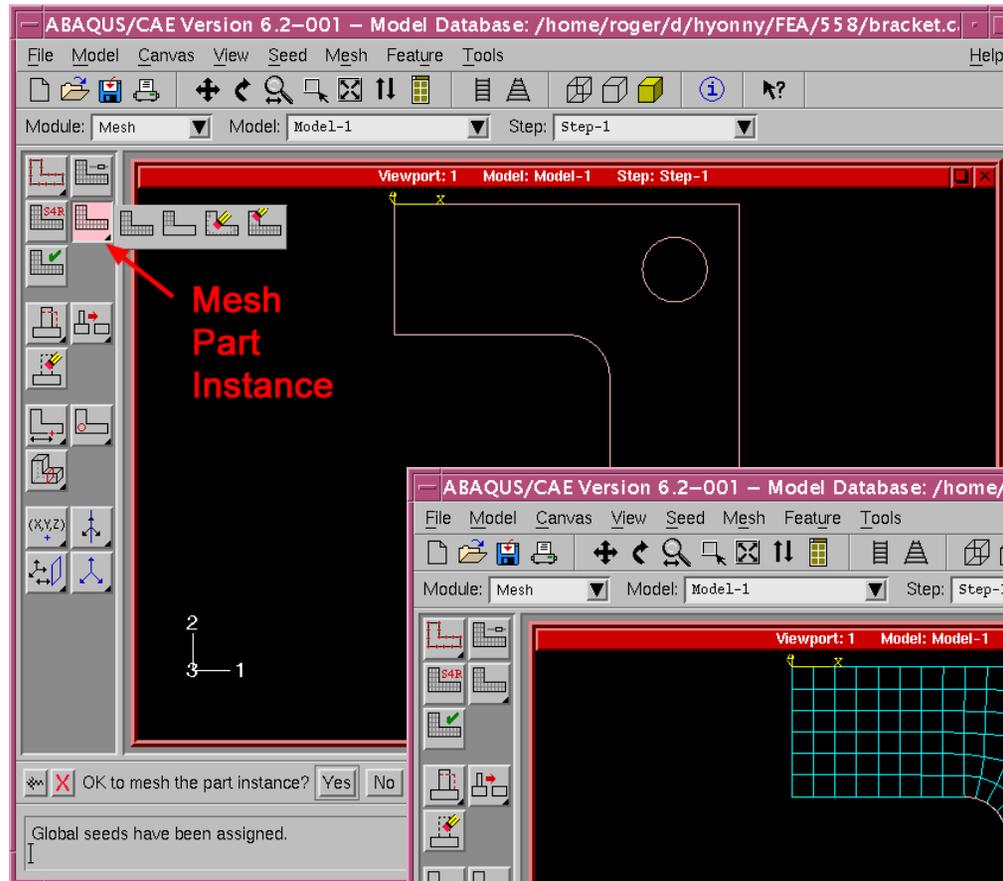
# Seed Mesh

- Change Modules. Select the **Mesh** module.
- Click the **Seed Part Instance** button. This is an expandable button. There are many other functions within this button that are useful for controlling mesh size.
  - In the *Prompt Area*, enter a *Global element size* value of **0.5**.
  - Hit enter and you will see circular symbols indicating nodal locations along the part edges.
- Click the **Assign Element Type** button.
  - the *Element Type* window pops up.
  - choose **Standard** in *Element Library*
  - **Plane Stress** in *Family*
  - **Linear** in *Geometric Order*
  - within the *Quad* tab, choose **Reduced Integration** in *Element Controls*. Leave everything else at default values.
  - the text in the lower box should indicate a *CPS4R* element identification. This is a 4-node reduced integration quadrilateral element.
  - click **OK**.



# Mesh

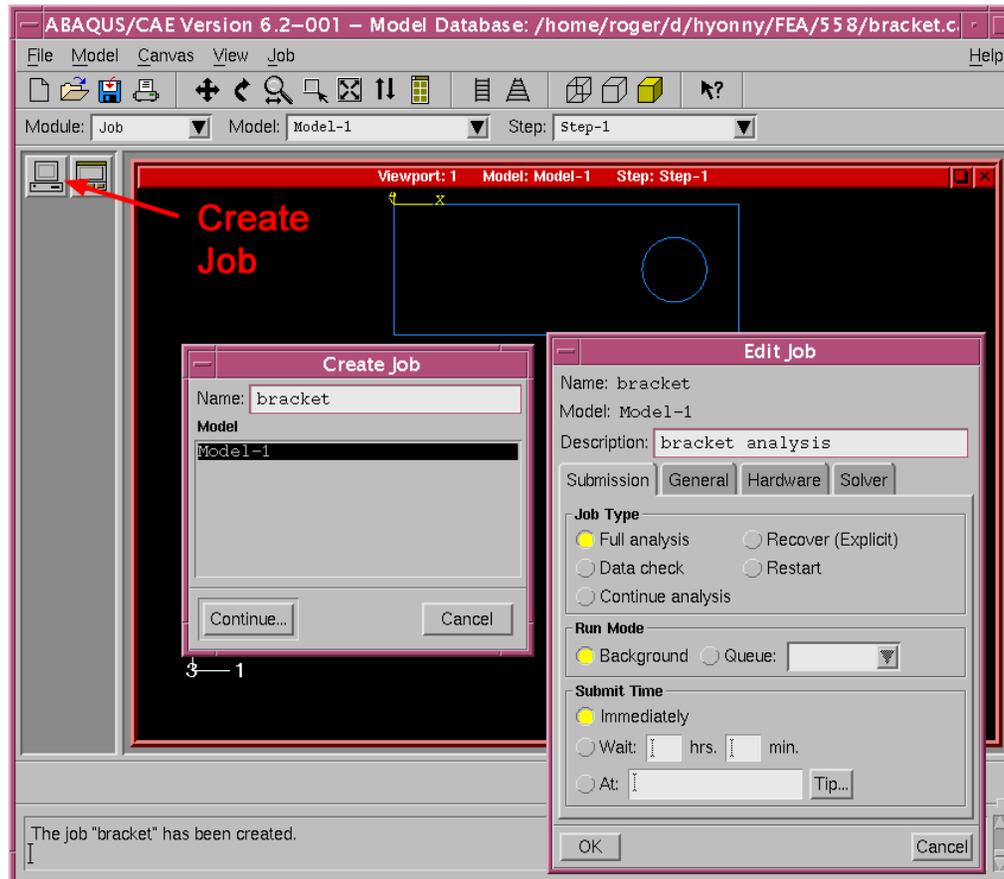
- Click **Mesh Part Instance** button. Note this button has many other functions within it (click-hold mouse button down on this button) such as deleting mesh and meshing regions of a part.
- Click **Yes** in the *Prompt Area*.
- Your mesh should look like the image shown to the right.



# Create Job

## SAVE YOUR WORK!!!

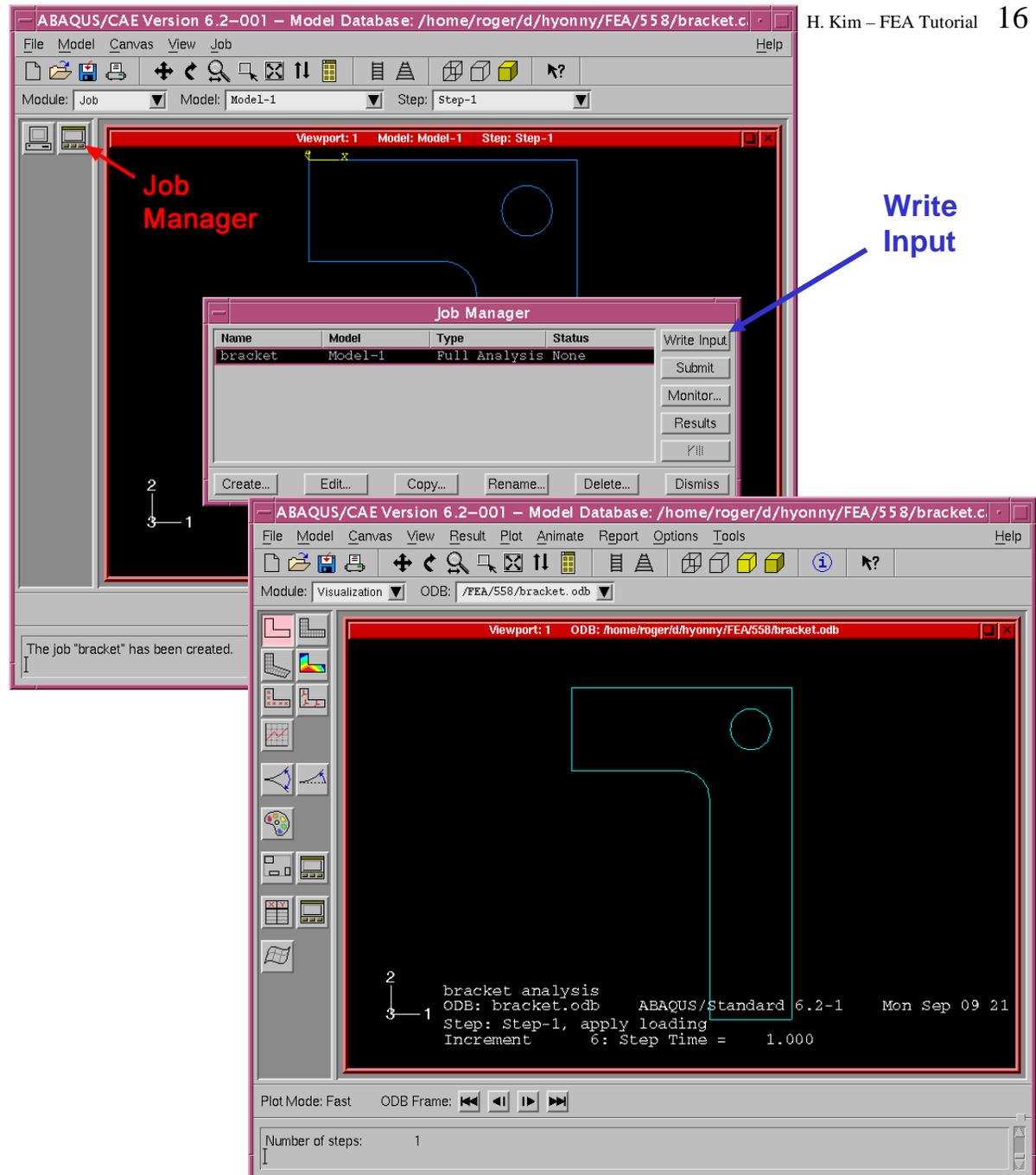
- Change Modules. Select the **Job** module.
- Click **Create Job** button.
- Enter a name, e.g., *bracket*
- Click **Continue**.
- In the *Edit Job* window that pops up, enter a *Description*, e.g., *bracket analysis*
- Check that **Full analysis**, **Background**, and **Immediately** are selected.
- Click **OK**.



The job "bracket" has been created.

# Submit Job

- Click **Job Manager** button.
- In *Job Manger* window pops up, check that your job is selected, then click **Submit**.
- To run your model in Unix Server, click **Write Input**. It takes few seconds to write “job name.inp”.
- Then:
  1. *Save you job*
  2. *Close ABAQUS/CAE*
  3. *Type “abaqus job = job name”*
  4. *Enter ↵*
  5. Then, go to slide 18-Result (b) for visualizing results
- Under *Status*, it will read:
  1. *Sumbitted*
  2. *Running*
  3. *Completed*
- Click **Results**.
- The Visualization module will run and the part in outline will be shown. It should look like the image to the right.



# Results (a) - Visualization

- Click the **Plot Contours** button.
- A colorful plot of Von Mises stresses appears.
- Color control can be adjusted by clicking the **Contour Options** button and adjusting parameters.
- To select the scalar field quantity plotted, in the *Menu Bar*, select **Result, Field Output**, then choose the stress component you wish to plot, e.g., S11, or U1.
- Click **OK**.
- Strains, Spatial Displacements, etc., can be selected through **Field Output**.

The screenshot displays the ABAQUS/CAE interface. The main viewport shows a contour plot of Von Mises stress (S, Mises) on a bracket model. The color scale ranges from blue (low stress) to red (high stress). The maximum stress is  $+2.039e+04$  and the minimum is  $+5.154e+01$ . The average critical stress is 75%.

The **Field Output** dialog box is open, showing the following settings:

- Step/Frame:** Step: 1, Step-1; Frame: 6
- Primary Variable:** Deformed Variable
- Output Variable:** List only variables with results: at integration points
- Name/Description Table:**

Name	Description
AC YIELD	Active yield flag at integ
CF	Point loads at nodes
E	Strain components at integ
PE	Plastic strain components
PEEQ	Equivalent plastic strain
PEMAG	Magnitude of plastic strain
RF	Reaction force at nodes
<b>S</b>	<b>Stress components at integ</b>
U	Spatial displacement at nodes
- Invariant/Component Table:**

Invariant	Component
Mises	<b>S11</b>
Max. In-Plane Principal	S22
Min. In-Plane Principal	S33
Out-of-Plane Principal	S12
Max. Principal	
Mid. Principal	

The **Field Output** dialog box also includes an **OK** button, an **Apply** button, and a **Cancel** button.

## Results (b) - After Run Model on Unix Server

- Run ABAQUS/CAE or ABAQUS/VIEWER.
- Open “job name.odb”.
- Click the **Plot Contours** button.
- A colorful plot of Von Mises stresses appears.
- Color control can be adjusted by clicking the **Contour Options** button and adjusting parameters.
- To select the scalar field quantity plotted, in the *Menu Bar*, select **Result, Field Output**, then choose the stress component you wish to plot, e.g., S11, or U1.
- Click **OK**.
- Strains, Spatial Displacements, etc., can be selected through **Field Output**.

