

## Part 1: Creating the part

Open SciTE from the Bconverged folder. Save as a .fbd file to begin. Try pressing F10 to begin the preprocessor. Add geometry to the file as shown in Figure 1, then as try figure 2. After entering the geometry, press F10 to view it in CGX.

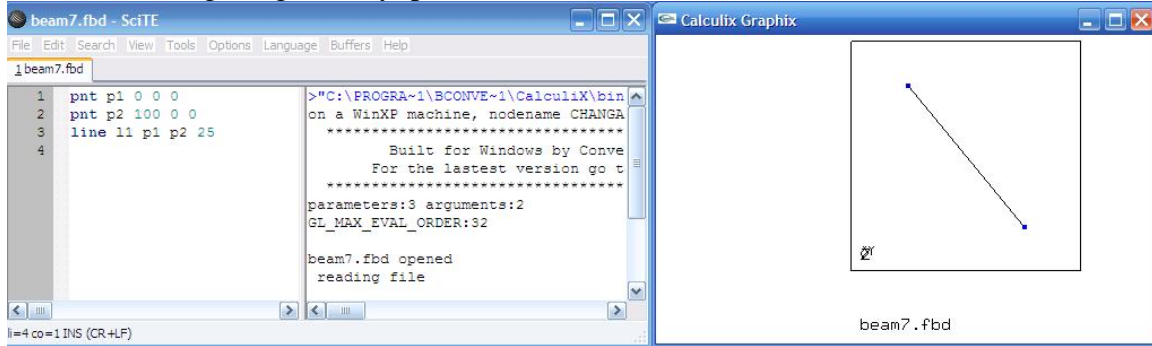


Figure 1: Line

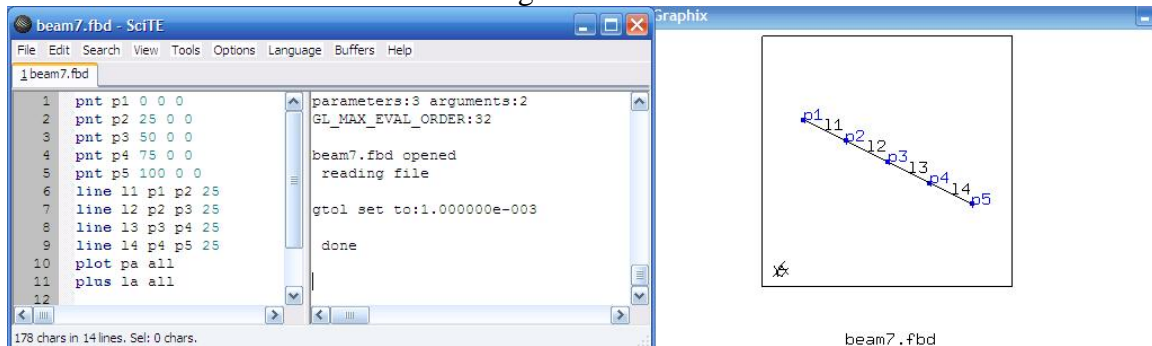


Figure 2: Segmented line

Now, try creating a surface using a sweep command. Add to the code

```

seta lines 1 11 12 13 14
sweep lines sweplines tra 0 10 0 10

```

This creates a set called lines with the above referenced lines, then sweeps that set and creates sweplines. Press F10 to view. It should look similar to figure 3.

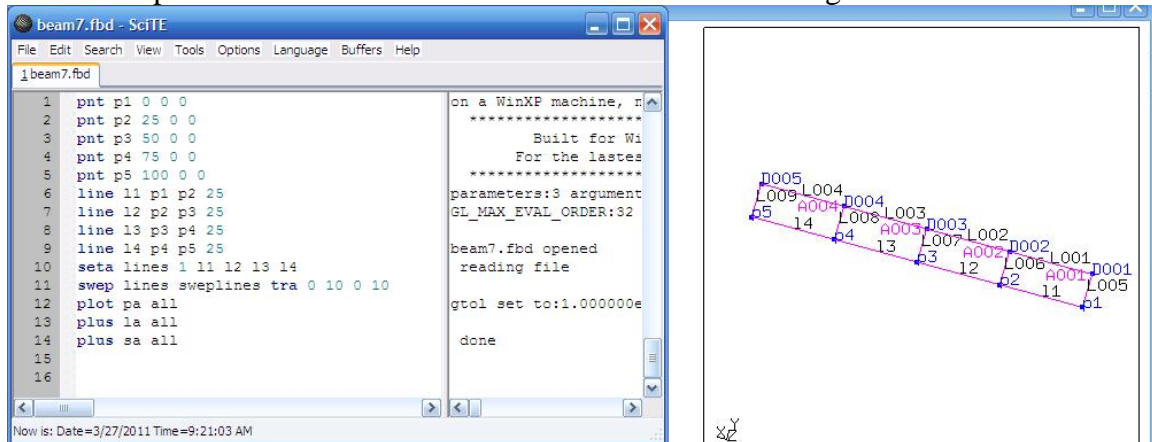


Figure 3: Surfaces

Next, use another sweep to create a 3D object. Add to the code

```
seta surfaces s A001 A002 A003 A004  
sweep surfaces swepsurfaces tra 0 0 1 1
```

Now, you should have a 3D object similar to Figure 4.

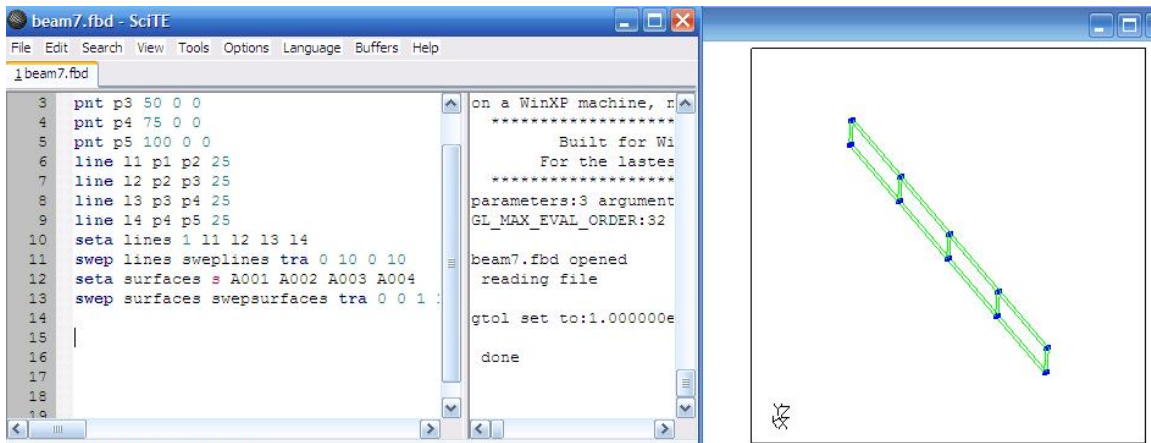


Figure 4: 3D Object

Now mesh the surface as follows:

```
elty all he8  
mesh all  
plot m all
```

This should look like figure 5

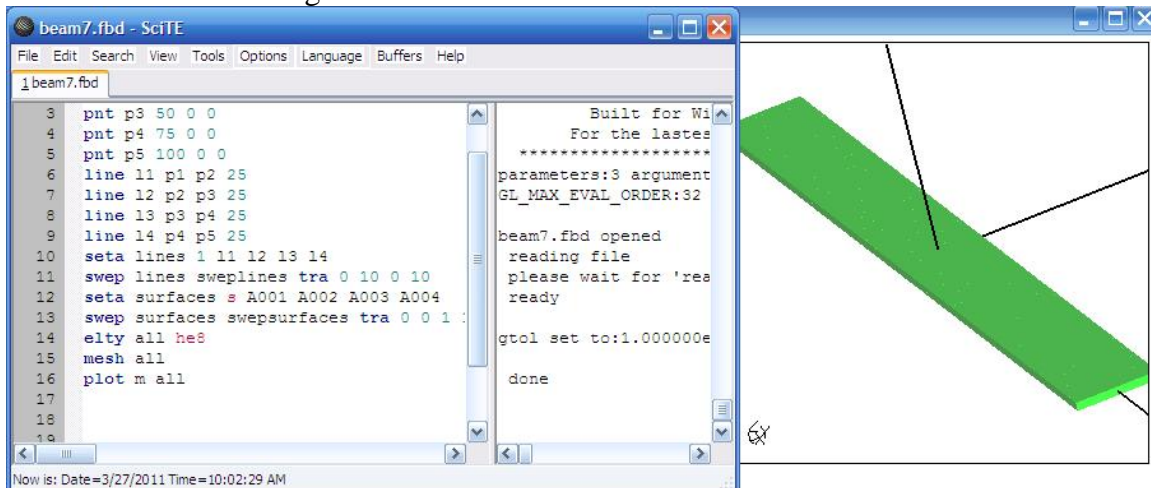
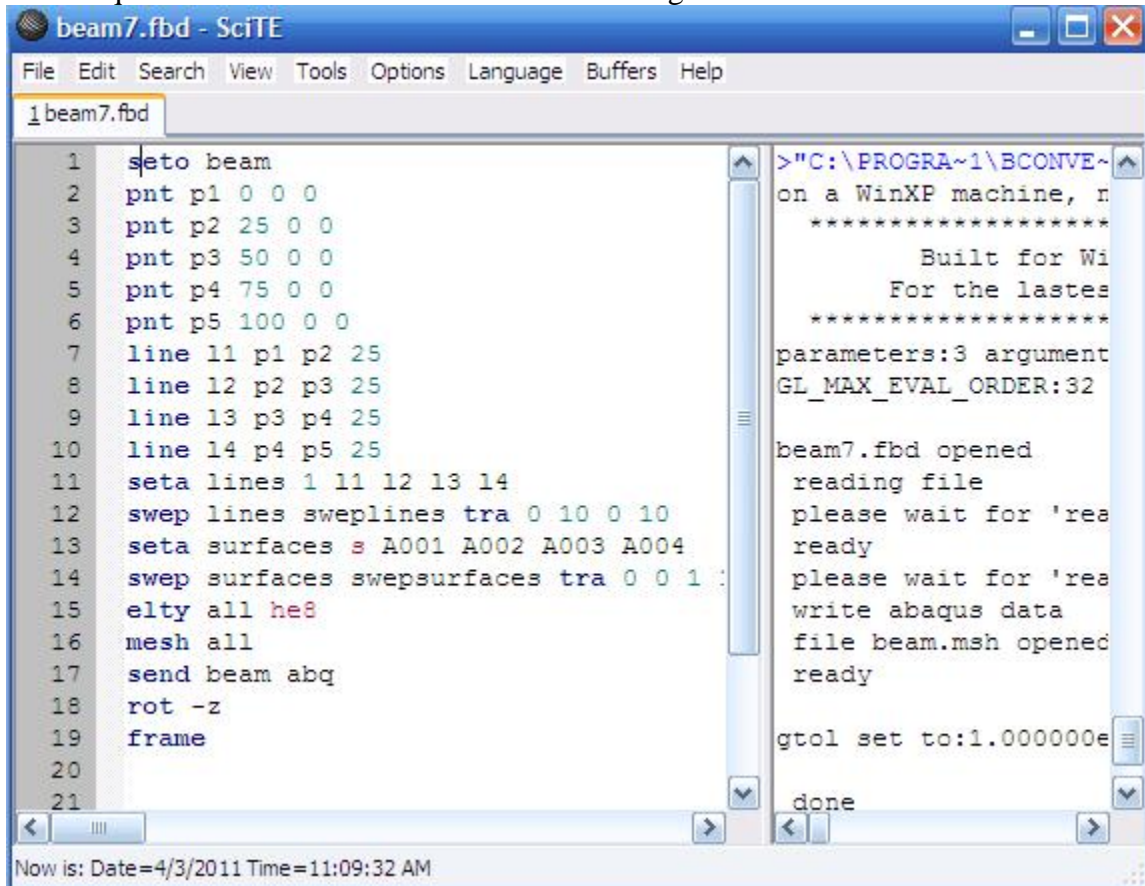


Figure 5: Meshed Beam

Bring up the menu in CGX and try changing the views under the viewing menu.

## Part 2: Meshing

Meshing, loads and BCs are applied in part 2. Modify the previous code as shown in figure 6. The command `seto` is used to open a set called `beam`. After the set is created, it is meshed and the command `send beam abq` exports the mesh created from set `beam` to the Abaqus format. This creates a text file describing the mesh.



```
beam7.fbd - SciTE
File Edit Search View Tools Options Language Buffers Help
_1 beam7.fbd
1  seto beam
2  pnt p1 0 0 0
3  pnt p2 25 0 0
4  pnt p3 50 0 0
5  pnt p4 75 0 0
6  pnt p5 100 0 0
7  line 11 p1 p2 25
8  line 12 p2 p3 25
9  line 13 p3 p4 25
10 line 14 p4 p5 25
11 seta lines 1 11 12 13 14
12 swep lines sweplines tra 0 10 0 10
13 seta surfaces s A001 A002 A003 A004
14 swep surfaces swepsurfaces tra 0 0 1 :
15 elty all he8
16 mesh all
17 send beam abq
18 rot -z
19 frame
20
21
>"C:\PROGRA~1\BCONVE~
on a WinXP machine, r
*****
        Built for Wi
        For the lastes
*****
parameters:3 argument
GL_MAX_EVAL_ORDER:32
beam7.fbd opened
reading file
please wait for 'rea
ready
please wait for 'rea
write abaqus data
file beam.msh opened
ready
gtol set to:1.000000e
done
Now is: Date=4/3/2011 Time=11:09:32 AM
```

Figure 6: Modified Code

To apply BC, we use the code `plot n all` to plot the nodes, and select using the `qadd fixed` command. Then use the command `send fixed abq spc 123` to apply the BC DOF and create a `.bou` file. Now select the load face, and apply a force of 10 using `qadd load`, then send the load to file as such:

```
qadd load
```

```
done
send load abq pres 10
please wait for 'ready'
write file: load.dlo
ready
ready
```

### **Part 3: Writing an input file**

Open SCITE and create a new .inp file. Write the following:

```
*heading
model: Calculx Beam Input File
*include,input=beam.msh
*boundary
*include,input=fixed_123.bou
*material,name=el
*elastic
30000000, 0.3
*solid section,elset=ebeam,material=el
*step
*static
*dload
*include,input=load.dlo
*node file
u
*el file
```

This code was explained well in the tutorial, so please reference that for a complete explanation of how it works.

With this .inp file open, solve using Tools->Solve. This will run the above code and output a file beam.frb. Enter into post processing mode. You can display the von mises stresses or other pertinent information.

### **Step 4: Post processing**

There are various ways to display the results from the solver. This section details how to display stresses, displacements, and other information. Opening up the post processor, you will find something like figure 7.

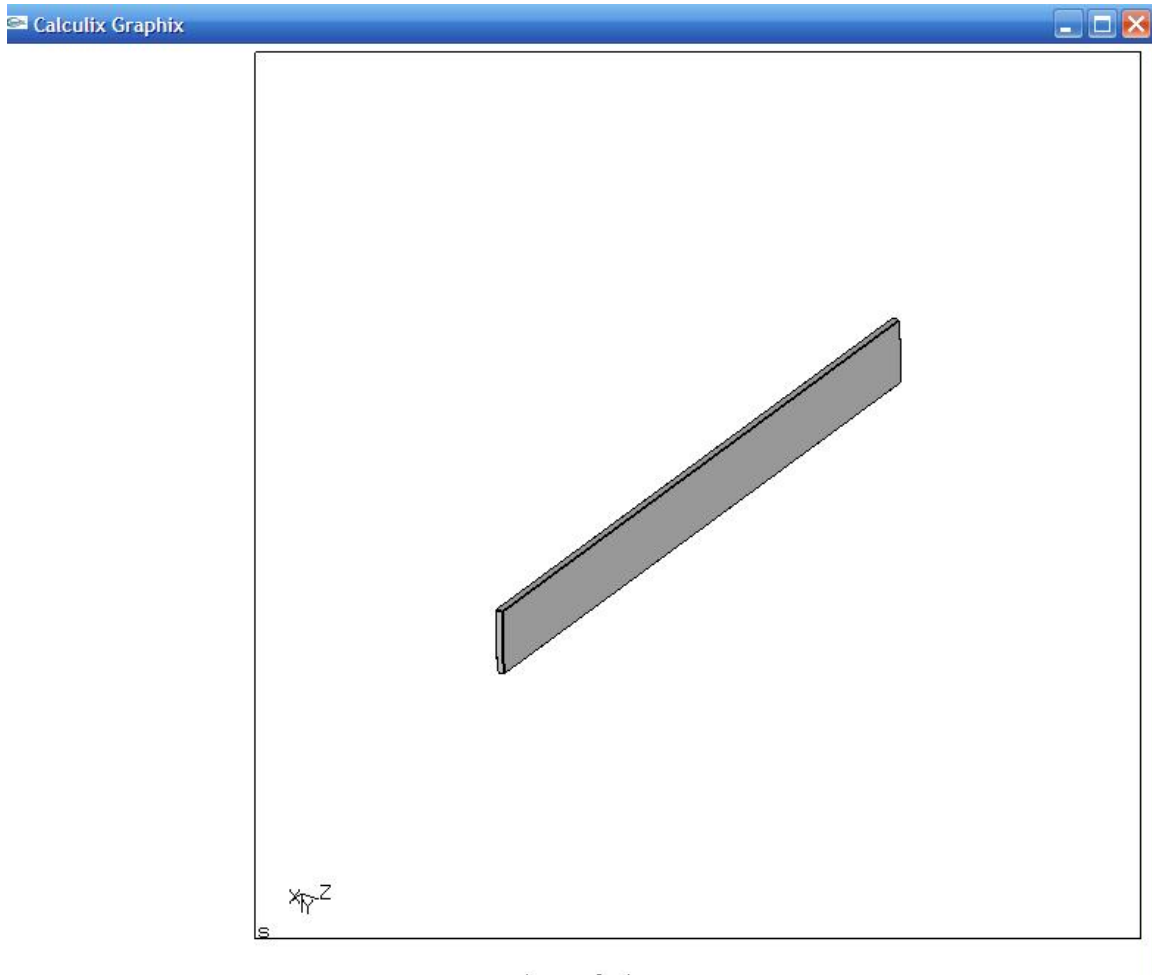


Figure 7: Grey Beam

Choose from the menu disp or stress, and then choose the entity to display. When you chose to display the displacement, it will likely be small. Use the command `scal d 10000` to scale up the displacement. You can also set up animations using the animate menu. Check out the movi command for making a .gif file of the animation. The cut and graph commands can be of great value. To graph a number of nodes, simply select graph and length, then select the nodes with the left mouse button, and exit with the right mouse button. The graph should have been written to a .png file such as in figure 8. This concludes the tutorial.

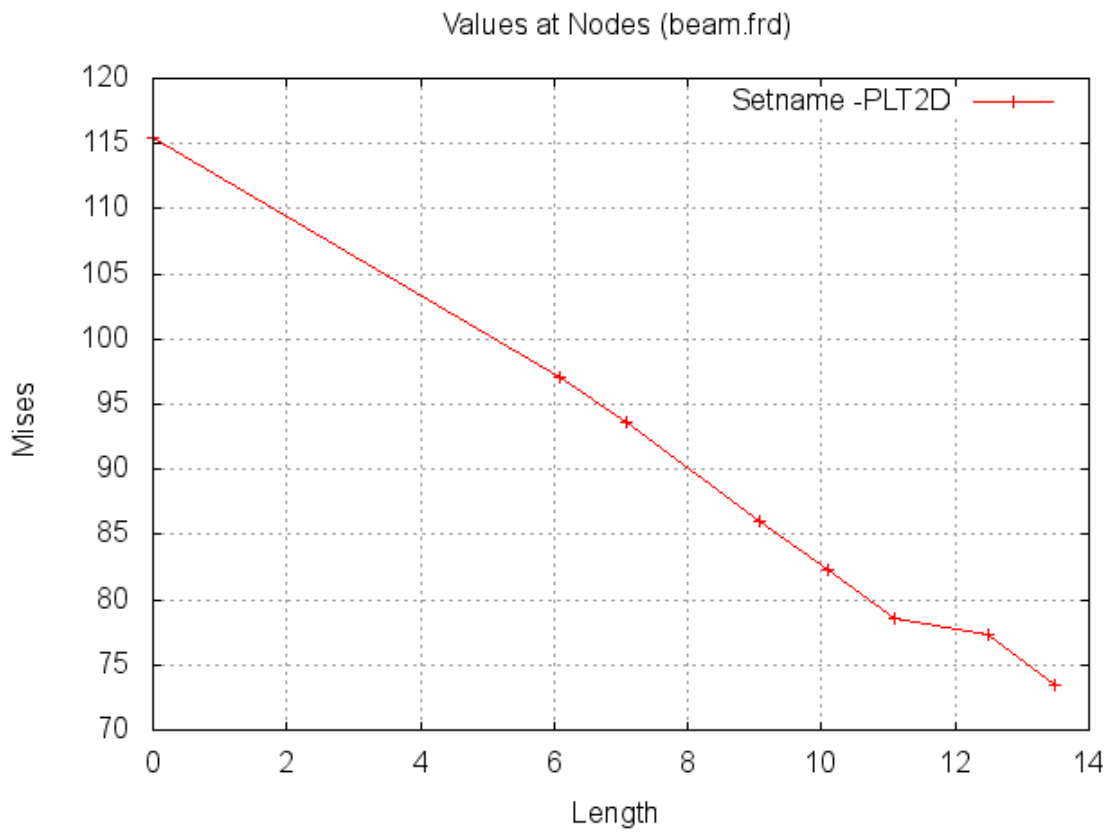


Figure 8: Plot of stresses.