

# FINITE ELEMENT ANALYSIS OF A BRIDGE SUPERSTRUCTURE

Instructor: Professor James Sherwood

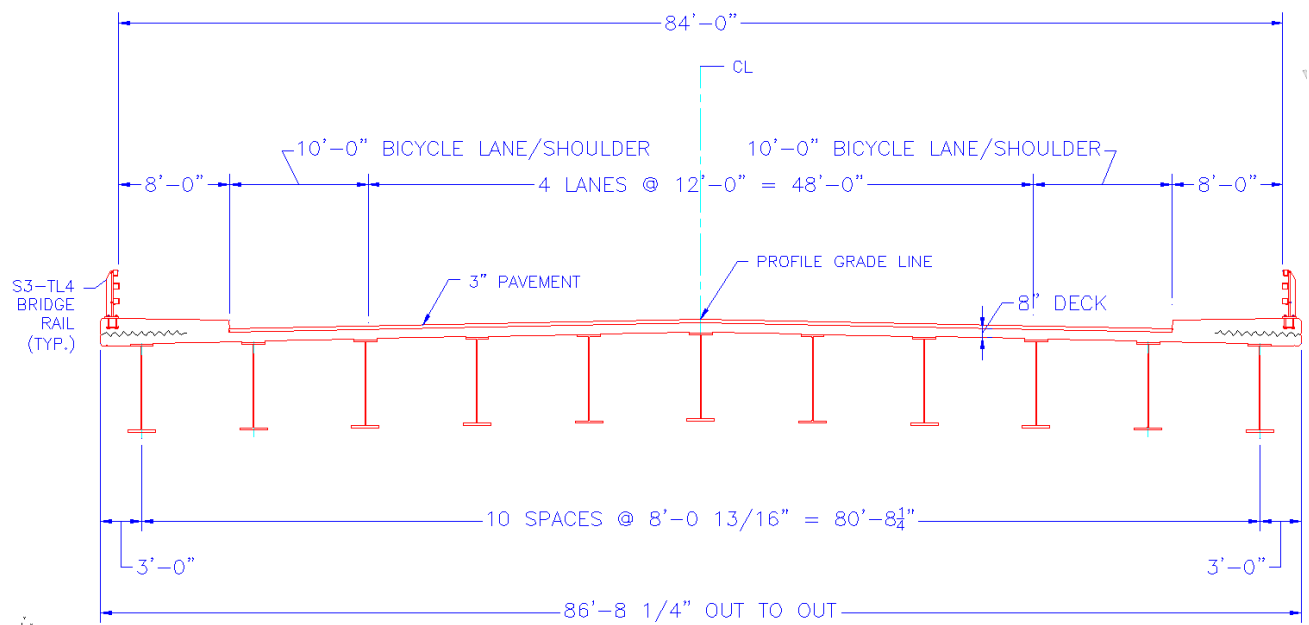
Authors: Christopher Reynolds, Ana Gouveia

Programs Utilized: ABAQUS CAE 6.12-3

## Problem Description:

This tutorial shows how to build and analyze the propped cantilever beam shown in Figure 1. The finite element model of the beam will be constructed using Abaqus/CAE. Three different loads are considered:

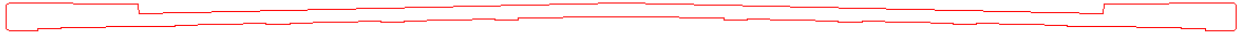
<b>Material:</b>	Steel
<b>Load case 1:</b>	Distributed
<b>Load case 2:</b>	Lumped
<b>Load case 3:</b>	Consistent



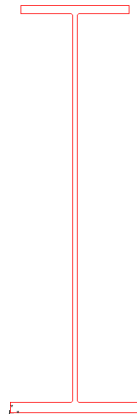
**Figure 1. Schematic of Beam Dimension and Loading**

## Importing the Model Geometry from AutoCAD 2014


- Open the sketch of the cross section in AutoCAD 2014.
- Simplify the geometry by removing unnecessary components. For analysis it is convenient to remove the guardrails and pavement layer from the sketch.
- Separate the sketch into drawings of the deck and an individual I-beam.
- Ensure the geometry of each part is closed. A path should be able to be traced around the sketch from any starting point, all the way back to that point, without any gaps.
- Also ensure the sketch does not contain overlapping lines. This will reduce the steps required to seed edges later in the tutorial.
- The geometry as prepared for importing should look something like shown below in Figures 2 and 3.

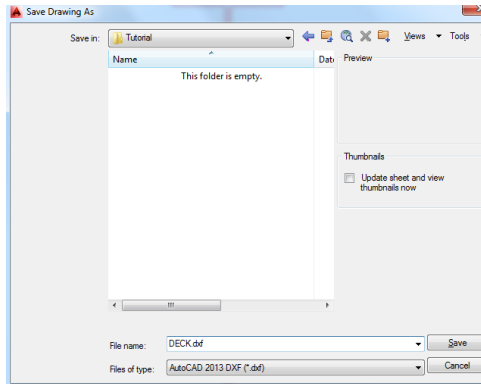


**Figure 2.** AutoCAD Deck Geometry for Import



**Figure 3.** AutoCAD Beam Geometry for Import

- With **ONLY** the sketch of the deck open, click  in the upper left hand corner of the screen and select “Save Drawing As”. Change the pull-down *Files of Type* to AutoCAD 2013 DXF (\*.dxf) and give the file a name, e.g. DECK.dxf
  - The *Save Drawing As* box should look like Figure 4.
  - For a 2-D sketch, DXF is a good choice for file type. To import a 3-D model (not covered in this tutorial) .SAT or .STEP file types may work better.



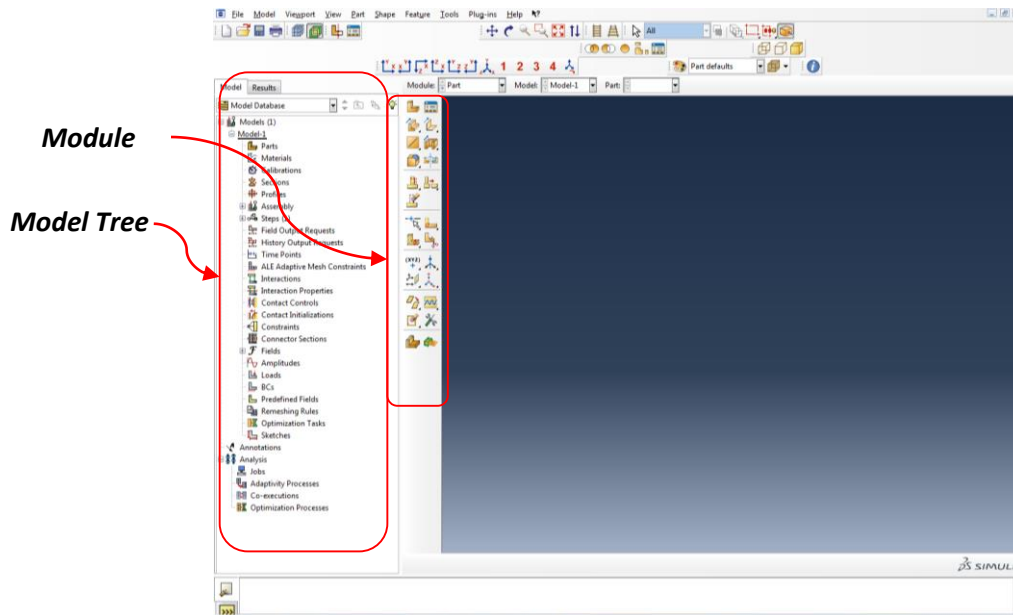
**Figure 4. Saving Sketch as .DXF File**

- Repeat the previous step with only the sketch of the I-beam open. Name the file BEAM.dxf.
- At this point in the tutorial, AutoCAD 2014 may be closed.
- Go to the **Start Menu** and open **Abaqus CAE**
- You may be prompted with an **Abaqus/CAE 6.12 Start Session** box (Figure 5). Close this box by clicking the **X** in the top right hand corner.



**Figure 5. Abaqus/CAE 6.12 Start Session**

- Once the **Start Session** box is exited, the Abaqus/CAE Viewport should look similar to Figure 6. (Please note the **model tree** is the series of functions listed on the left hand side of the viewport, while the **module** is the list of icons to the right of the model tree)

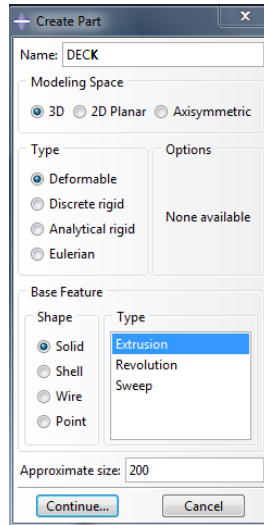


**Figure 6.** Abaqus/CAE Viewport



- Click **File**→**I**mport→**S**ketch. The Import Sketch box should open. Change the File Filter to AutoCAD DXF (\*.dxf) and locate the DECK file created earlier. Click OK.
- Repeat the previous step, importing the BEAM dxf file.
- The sketches tab under the model tree should now have two contents: DECK and BEAM

### Creating Parts

- A model of this structure will be constructed using solid elements. Parts will be created by extruding the imported sketches.
- Using the left mouse button, double click **Parts** in the model tree and the **Create Part** dialog box appears. Enter a new name for the part (DECK). The **Create Part** dialog box should look identical to Figure 7.
- Click **Continue...** and the graphics window will change to a set of gridlines.

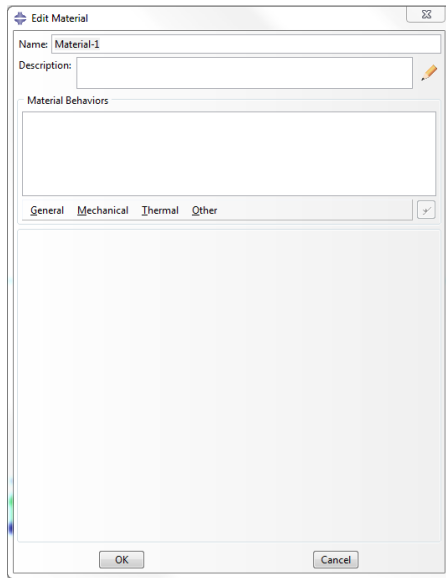


**Figure 7.** Create Part Dialog Box

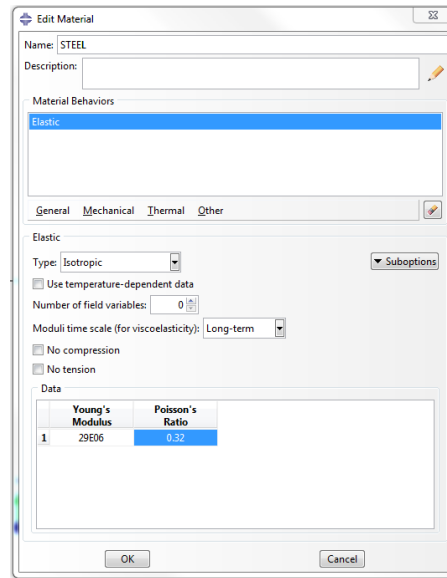
- Click the **Add Sketch** icon  in the module. (Remember, the module is the series of icons to the right of the model tree)
- The Select Sketch dialogue box will appear. Chose “DECK” and click OK.
- At this point, there is an option to scale the drawing. The drawings in this tutorial were not scaled because the AutoCAD drawings were created in inches, and it is convenient to leave the unit unchanged.
- Click  in the prompt at the bottom of the screen. The Edit Base Extrusion dialogue box appears.
- Enter the length of the bridge next to Depth:
- Please note there is no dropdown menu or feature in Abaqus that sets specific units. All of the dimensions have been input in inches; therefore the depth should also be entered in inches. Click OK.
- A 3-D model of the deck should now be visible.
- Repeat the steps for creating a part to create the I-beam.

### Defining Material Properties

- To define material properties for this model, double click on **Materials** in the model tree and the Edit Material dialog box will appear (Figure 8a). Enter a **Name** for the material (STEEL), and click the **Mechanical** tab, highlight **Elasticity** and click **Elastic**. Enter values of **Young’s Modulus** = 29E06 psi, and **Poisson’s Ratio** = 0.32. After the material properties have been entered, the **Edit Material** dialog box should look identical to Figure 8b.
- Click **OK**.



**Figure 8a.** Edit Material Dialog Box

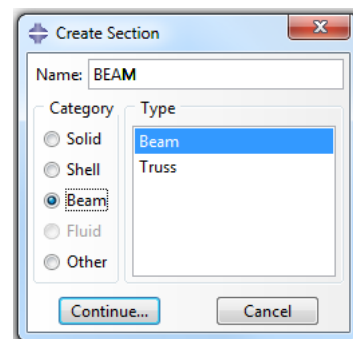
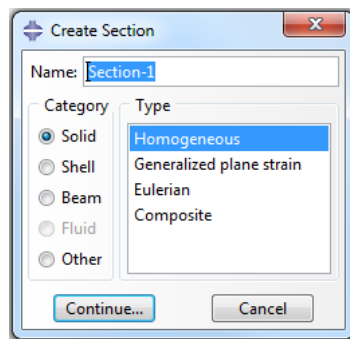


**Figure 7b.** Edit Material Dialog Box (STEEL)

- Repeat the two steps above to define a second material: CONCRETE. Enter values of **Young's Modulus** = 3.6E06 psi, and **Poisson's Ratio** = 0.15.
- Young's Modulus units should be entered in psi (pounds per square inch). The units chosen for the definition of the material properties should be consistent with the dimensions of the structure.
- At this point in preprocessing, the model should be saved. Click **File** then click **Save**. Name the file **BRIDGE SUPERSTRUCTURE TUTORIAL**. The file will save as a Model Database (\*.cae\*) file. It may be of interest to save the file after each section of this tutorial.

## Creating Sections

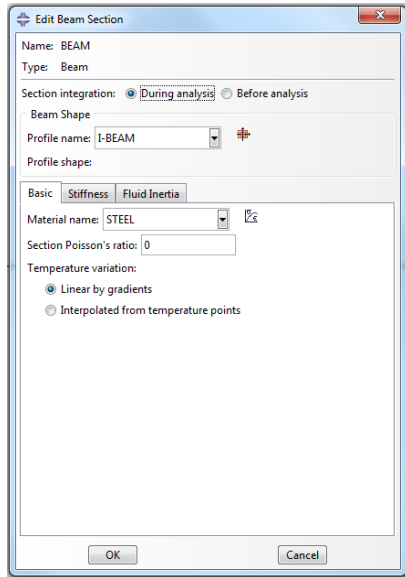
- To create a beam section in Abaqus, double click **Sections** in the model tree and the **Create Section** dialog box will appear (Figure 10a). Enter a **Name** for the section (BEAM), and choose **Beam** under the Category Tab, and **Beam** under the Type tab. Your **Create Section** dialog box should look identical to that in Figure 10b.
- Click **Continue...**



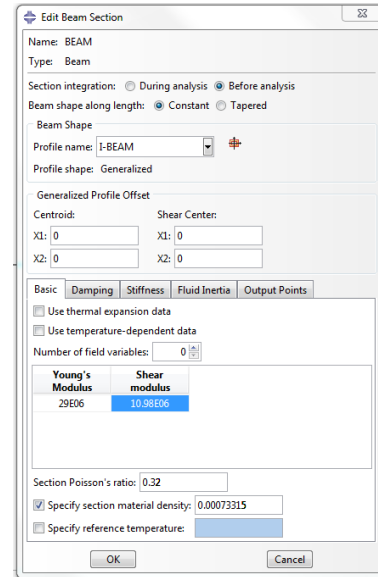
**Figure 10a.** Create Section Dialog Box

**Figure 10b.** Create Section Dialog Box (BEAM)

- The **Edit Beam Section** dialog box will immediately appear (Figure 11a). Click **Before analysis** under the **Section integration:** option. Since only one profile has been created the drop down options for **Profile name:** is set to **I-BEAM**. Under the Basic tab, enter a value of **Young's Modulus** = 29E06 psi and **Shear Modulus** = 10.98E06 psi. Enter 0.32 for a **Section Poisson's ratio**. Check the box to the left of **Specify section material density:** and enter a value of 0.00073315. The **Edit Beam Section** dialog box should look similar to that in Figure 11b.



**Figure 11a.** Edit Beam Section Dialog Box



**Figure 11b.** Edit Beam Section Dialog Box (I-BEAM)

- Click **OK**.

### Assigning Sections

- Now that the beam section has been created, it can be assigned to the geometry. In the model tree, click the + to the left of the **Parts** icon, this will further expand the model tree's options. Next, click the + to the left of the part called **I-BEAM**, further expanding the model tree (Figure 12).

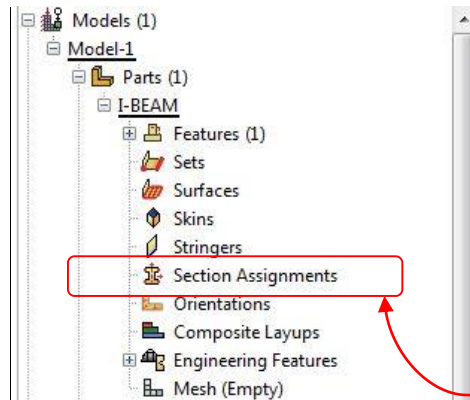


Figure 12. Model Tree Expansion (Parts)

- After the model tree has been expanded, double click **Section Assignments**. Using the cursor draw a box around the complete geometry. If this has been done correctly the model will turn from a grey line to a red line (Figure 13).

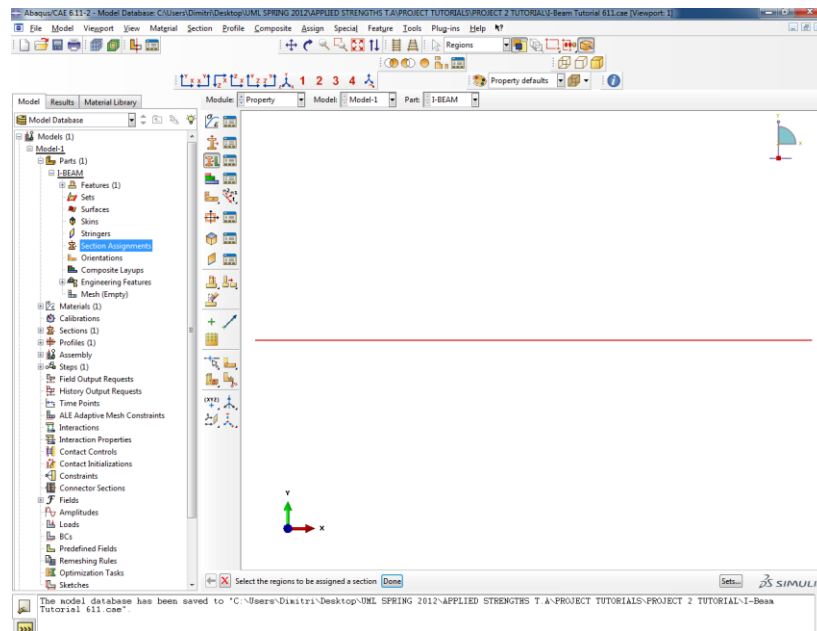
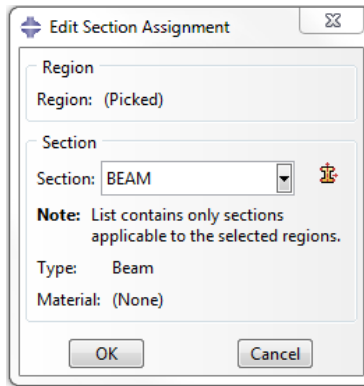


Figure 13. Assigning Beam Sections

- Click **Done**.
- The Edit Section Assignment dialog box will immediately appear (Figure 14).
- Click **OK**. If this was done correctly the model should turn a blue color.

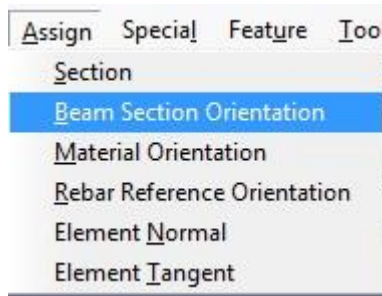




**Figure 14.** Edit Section Assignment Dialog Box

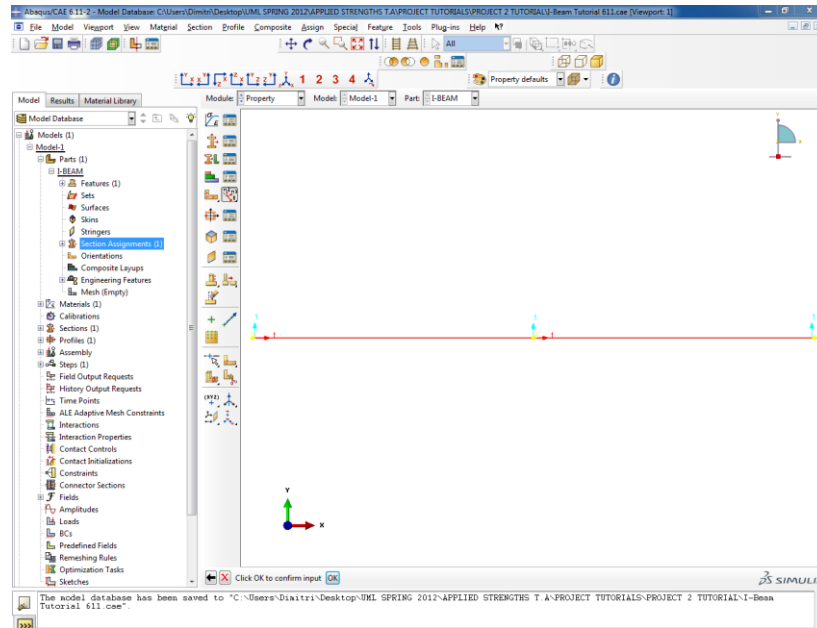
### Assigning a Beam Section Orientation

- A **Beam Section Orientation** must be assigned. In the toolbar at the top of the Viewport, there is a dropdown menu labeled **Assign**. Using the left mouse button, click **Assign** and click **Beam Section Orientation** (Figure 15).



**Figure 15.** Beam Section Orientation Drop down Menu


- Using the cursor, hold the left mouse button while dragging the cursor around the model to create a box around the whole geometry. If this drag is done correctly, the model will change color from blue to red.
- Click **Done**.
- Using the computer keyboard, enter (0.0,1.0,0.0) in the **Enter an approximate n1 direction (tangent vectors shown)** option. Hit **Enter**. The model should look identical to Figure 16.

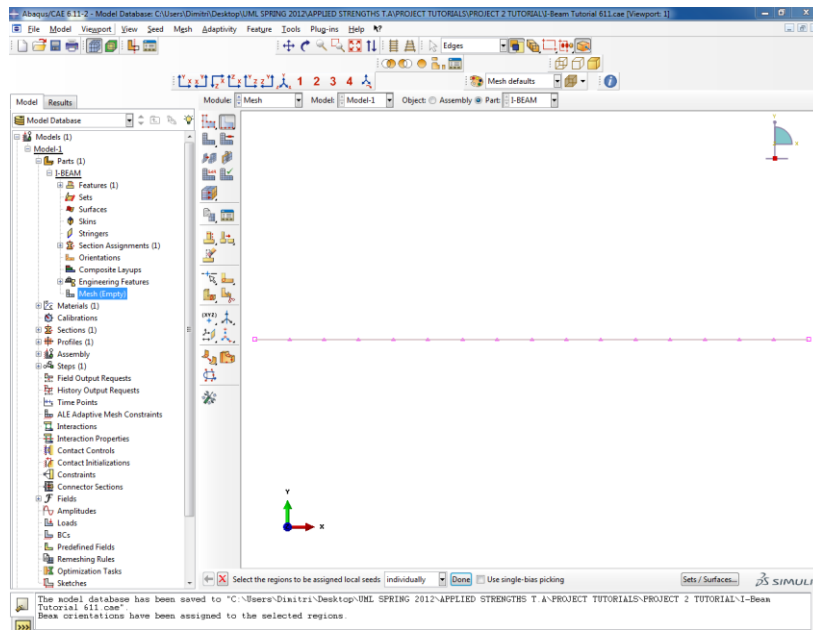


**Figure 16.** Beam Section Orientation


- Click **OK**.
- Click **Done**. The model should turn back to a blue color.

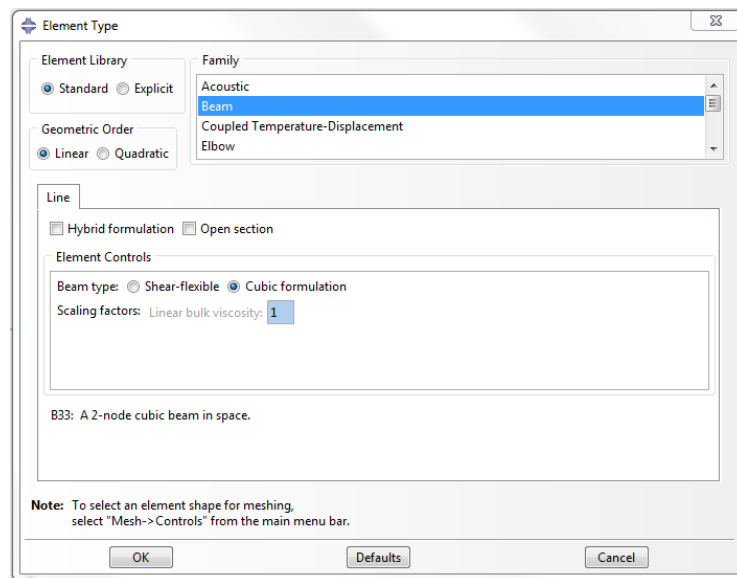
## Creating a Mesh

- To create a mesh for the model geometry, double click **Mesh (Empty)** in the model tree. If this selection is done correctly, then the geometry should change color to pink.
- The first step in creating a mesh is to seed the part, click the **Seed Edges** icon  in the mesh module.
- Using the cursor draw a box around the whole model, if this is done correctly the model will turn from a pink to a red color.
- Click **Done**.
- In the **Local Seeds** dialog box, click **By Number** in the **Method** option. Under **Sizing Controls**, enter **16** under **Number of elements**:. This will seed the selected edge to have 16 evenly spaced elements along its length.
- Click **OK**. The model will now appear to be seeded with evenly spaced pink points along its length Figure 17.




**Figure 17. Seeded Geometry (16 Elements)**

- Click **Done**.
- Now that the part has been seeded, a mesh can be generated. Click the **Assign Element Type** icon  in the mesh module.
- You will be immediately prompted by the **Element Type** dialog box. Under the **Family** category ensure that **Beam** is selected. Under **Beam type**: click **Cubic formulation**. Your **Element type** dialog box should look identical to Figure 18.



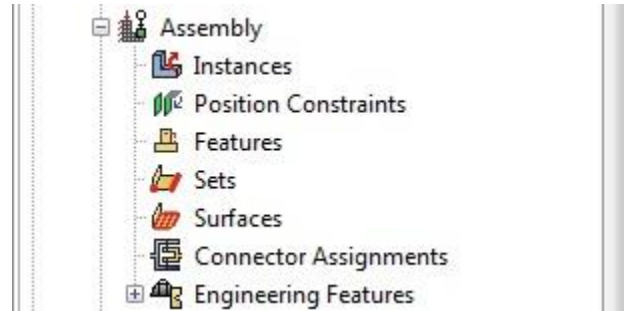
**Figure 18. Element Type (BEAM)**

- Click **OK**.

- The part is now ready to be meshed. In the mesh module, click the **Mesh Part** icon . At the bottom of the viewport you will be prompted if it is **OK to mesh the part?** Click **Yes**.
- If this procedure was done correctly, the geometry will turn blue.

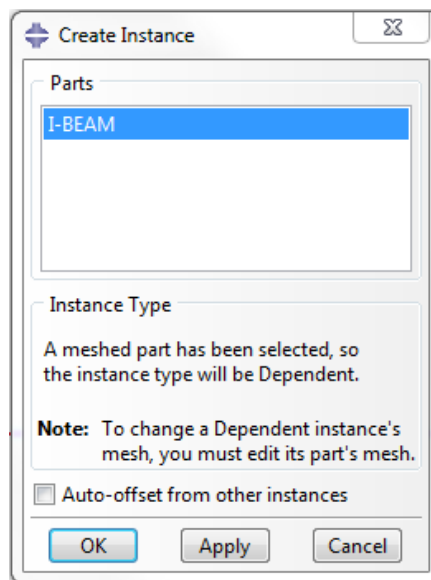
### Creating an Instance

- Now that the part has been meshed, it can be brought into the assembly. To do this task, click the **+** to the left of **Assembly** in the model tree. The model tree will expand and should look identical to Figure 19.



**Figure 19.** Model Tree Expansion (Assembly)

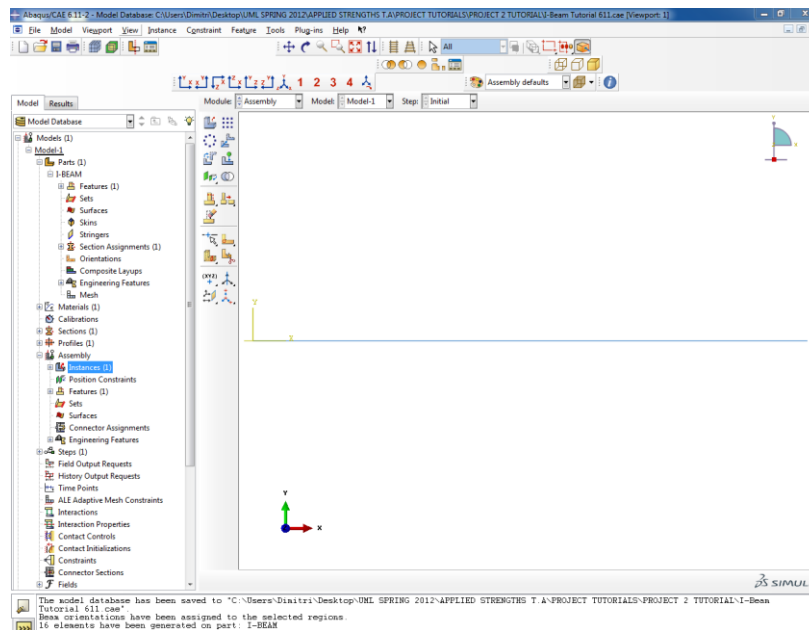
- Double click on the **Instances** icon in the expanded model tree. This feature will allow multiple parts to be brought into the assembly. The **Create Instance** dialog box will appear (Figure 20).



**Figure 20.** Create Instance Dialog Box

- The **I-BEAM** part is selected by default because only one part has been created for this tutorial. If multiple parts had been created, then this step would allow them to be entered into the assembly.

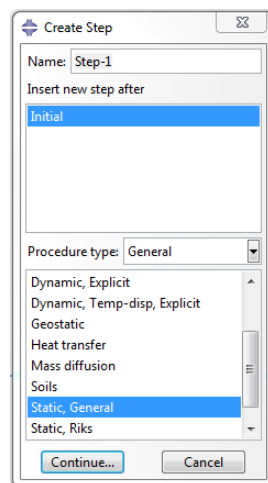
- Click **OK**. If this step was done correctly the model should turn a blue color (Figure 21).



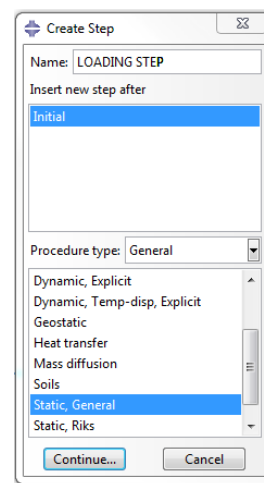
**Figure 21. Create Instance**

## Creating a Step

- A Step is where the user defines the type of loading, e.g. Static or Dynamic, and defines the boundary conditions, e.g. support constraints and forces.
- In the model tree, double click the **Steps** icon. The **Create Step** dialog box will appear (Figure 22a). Create a **Name** for the step called **LOADING STEP**. Under **Procedure type** choose **General > Static, General**. The **Create Step** dialog box should look identical to Figure 22b.

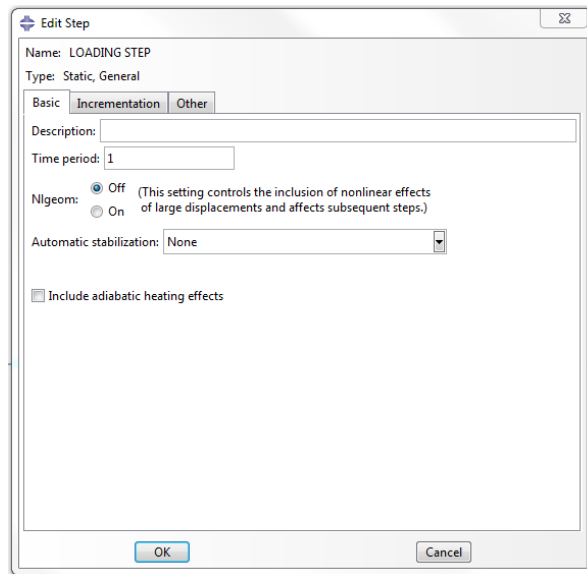


**Figure 22a. Create Step Dialog Box**



**Figure 22b. Create Step Dialog Box (LOADING STEP)**

- Click **Continue...**, and the **Edit Step** dialog box will immediately appear (Figure 23).

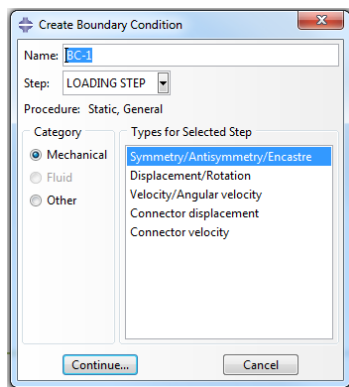


**Figure 23.** Edit Step Dialog Box

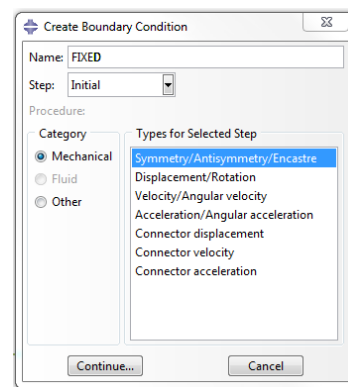
- Click **OK** to accept the default values for the various options.

### Apply Constraint Boundary Conditions

- Boundary conditions will be defined which will simulate a fixed (also known as “clamped”) beam at one end and a roller boundary condition at the opposite end.
- Double click **BCs** in the model tree and the **Create Boundary Condition** dialog box will appear (Figure 24a). Create a **Name** for the boundary condition called **FIXED**, and under the **Step** drop down menu make sure to choose **Initial**. Under the **Category** option choose **Mechanical**, and choose **Symmetry/Antisymmetry/Encastre** under the **Types for Selected Step** option. The **Create Boundary Condition** dialog box should look identical to that in Figure 24b.



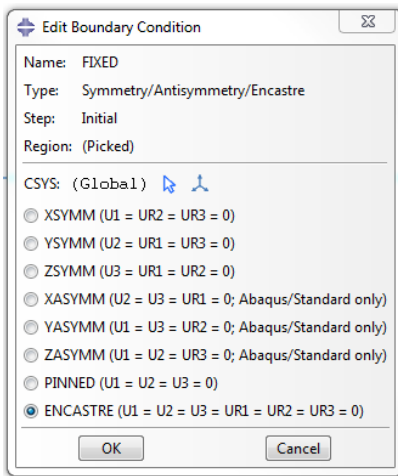
**Figure 24a.** Create Boundary Condition



**Figure 24b.** Create Boundary Condition (FIXED)

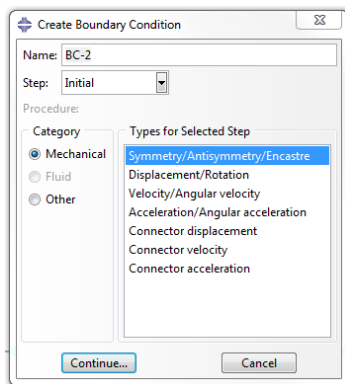
- Click **Continue...**

- In the viewport the two visible yellow points will be located at either end of the beam. Using the cursor click the yellow point at the **left** side of the beam. If this is done correctly the point will turn from a yellow to red color.
- Click **Done**.
- The **Edit Boundary Condition dialog** box will immediately appear. Click **ENCASTRE** ( $U1=U2=U3=UR1=UR2=UR3=0$ ). The Edit Boundary Condition dialog box should look identical to that in Figure 25.

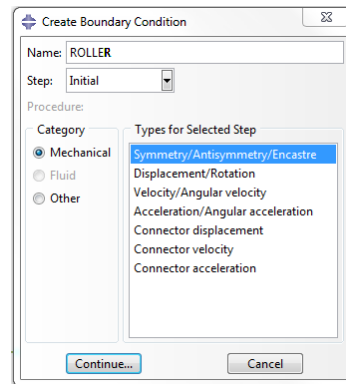


**Figure 25.** Edit Boundary Condition Dialog Box (FIXED)

- Click **OK**.
- A roller boundary condition will be applied to the right side of the beam model. Double click **BCs** in the model tree and the **Create Boundary Condition** dialog box will appear (Figure 26a). Create a **Name** for the boundary condition called **ROLLER**, and under the **Step** drop down menu make sure to choose **Initial**. Under the **Category** option choose **Mechanical**, and choose **Symmetry/Antisymmetry/Encastre** under the **Types for Selected Step** option. The **Create Boundary Condition** dialog box should look identical to that in Figure 26b.

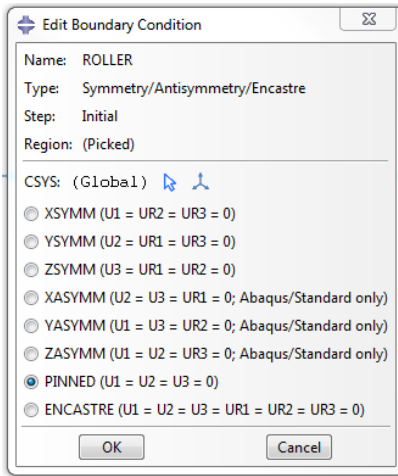


**Figure 26a.** Create Boundary Condition



**Figure 26b.** Create Boundary Condition (ROLLER)

- Click **Continue...**
- In the viewport the two visible yellow points will be located at either end of the beam. Using the cursor click the yellow point at the **right** side of the beam. If this is done correctly the point will turn from a yellow to red color.
- Click **Done**.
- The **Edit Boundary Condition dialog** box will immediately appear. Click **PINNED (U1=U2=U3= 0)**. The **Edit Boundary Condition** dialog box should look identical to that in Figure 27.



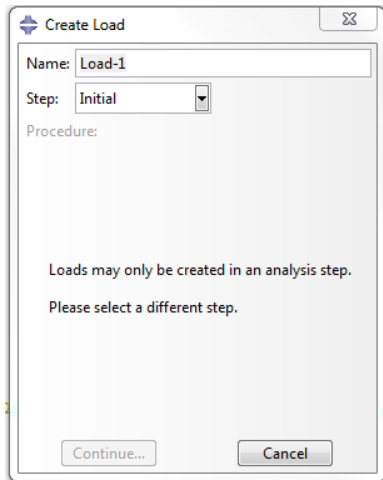
**Figure 27.** Edit Boundary Condition Dialog Box (ROLLER)

- Click **OK**.

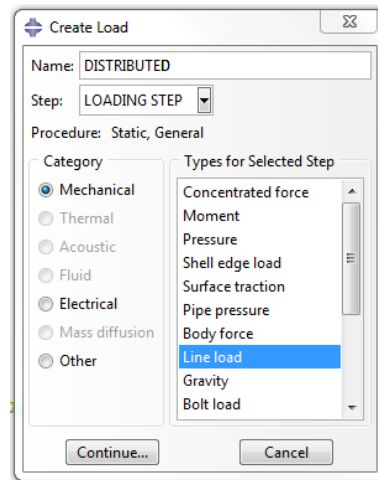
### Applying a Distributed Load to the Beam

- A 10 lb/in distributed load will be applied to the end of the beam.
- Double click **Loads** in the model tree and the **Create Load** dialog box will appear (Figure 28a). Create a **Name** for load called **DISTRIBUTED**, and under the **Step** drop down menu make sure to choose **LOADING STEP**. Under the **Category** option choose **Mechanical**, and choose **Line load** under the **Types for Selected Step** option. The **Create Load** dialog box should look identical to that in Figure 28b.



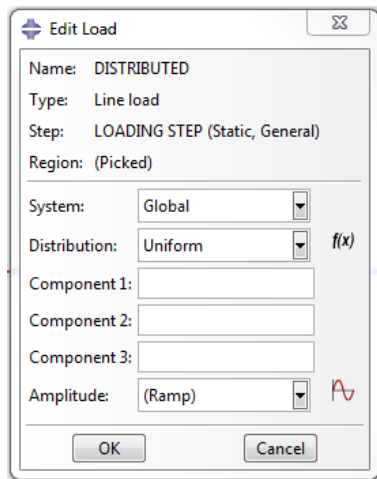


**Figure 28a.** Create Load

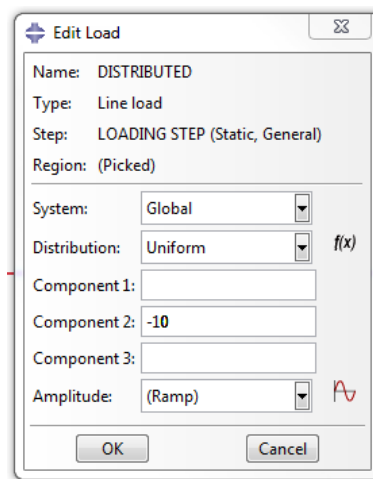


**Figure 28b.** Create Load (DISTRIBUTED)

- Click **Continue...**
- Using the cursor, click the beam. If this is done correctly the beam will turn from a blue to red color.
- Click **Done**. The **Edit Load** dialog box will immediately appear (Figure 29a). Enter a value of **-10** under the **Component 2:** option. This will prescribe a load of 10 lb/in on the selected region in the **–Y** direction. The **Edit Load** dialog box should look identical to that in Figure 29b.

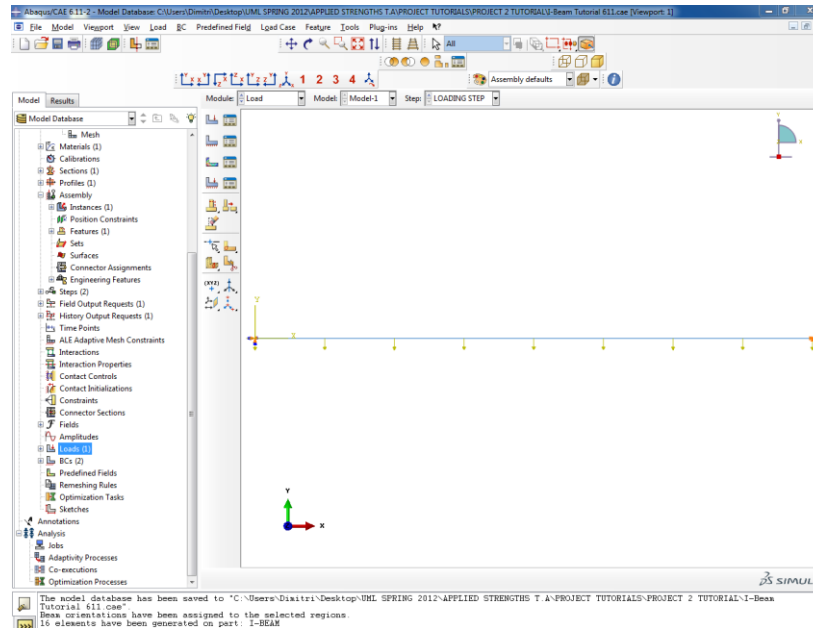


**Figure 29a.** Edit Load Dialog Box



**Figure 29b.** Edit Load Dialog Box (10lb/in)

- Click **OK**. If this was done correctly 9 evenly distributed yellow arrows will appear on the model pointing in the **–Y** direction (Figure 30).



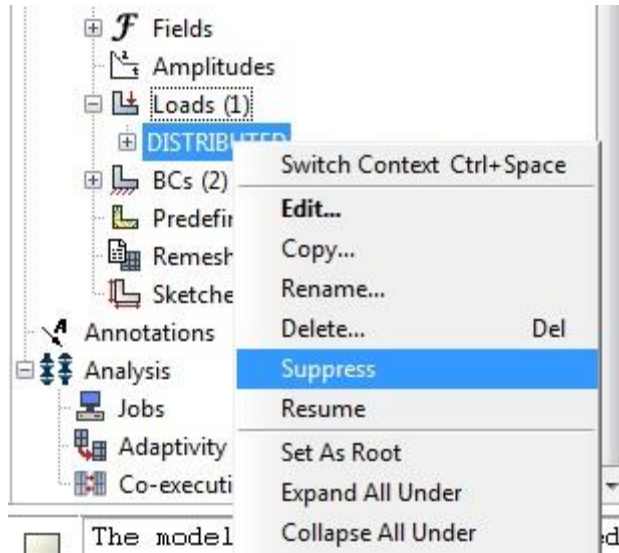
**Figure 30.** Distributed Loading Condition

- Since this project tutorial calls for three different loading conditions on the beam, the created distributed load will be “suppressed” so that multiple loads can be created with no confusion.
- In the Model Tree click the + to the left of the **Loads(1)** option. The model tree will expand and should look identical to Figure 31.



**Figure 31.** Model Tree Expansion (Loads)

- Using the cursor, right click **DISTRIBUTED** (Figure 32). Scroll over the **Suppress** option and right click.

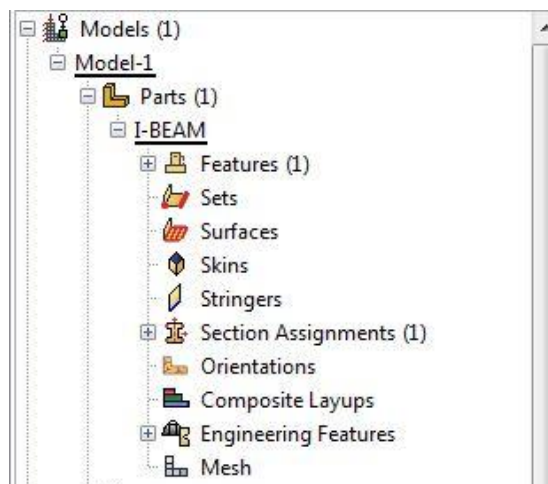


**Figure 32.** Suppress Distributed Load

- If this has been done correctly a small red **X** will appear to the left of **DISTRIBUTED** in the model tree. Also the yellow arrows pointing in the **–Y** direction in the model will disappear.

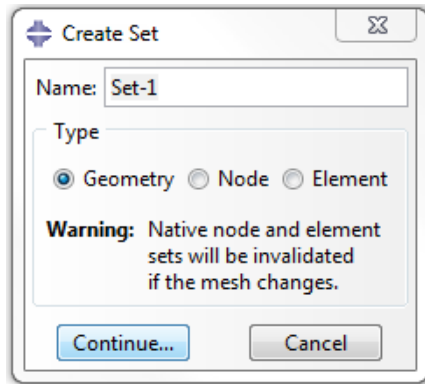
#### Applying a Lumped Load to the Beam

- Now that the distributed load has been suppressed, a lumped load will be added to the model. Since the lumped load will be only added to certain nodes of the model it would be of interest to create a **Set** so that a concentrated force can be applied to a specific set of nodes.
- Double click **Sets** in the model tree. Make sure to double click the Sets option that is underneath the I-BEAM **Part** in the model tree (Figure 33).

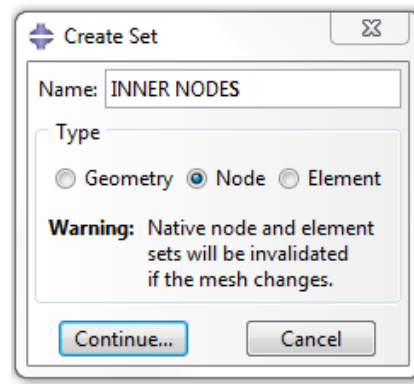


**Figure 33.** Sets Option in Model Tree

- The **Create Set** dialog box will immediately appear (Figure 34a). Create a **Name** for the set called **INNER NODES**, and under the type option make sure to choose **Node**. The Create Set dialog box should look identical to that in Figure 34b.

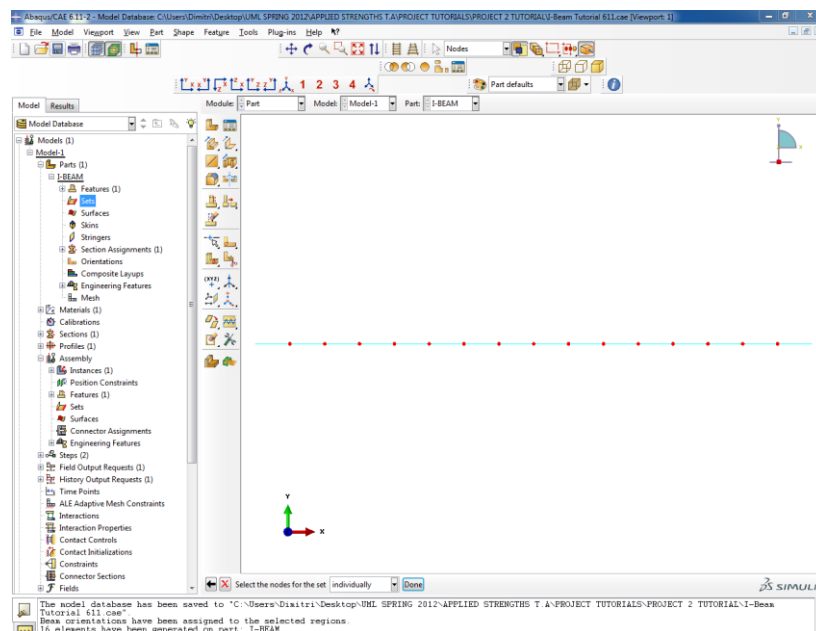


**Figure 34a.** Create Set Dialog Box



**Figure 34b.** Create Set Dialog Box (INNER NODES)

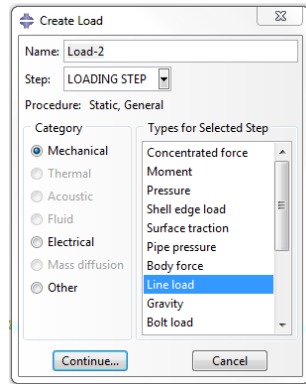
- Click **Continue...** The model will turn a turquoise color.
- Using the cursor, draw a box around the model excluding the two outer most edges (Figure 35).



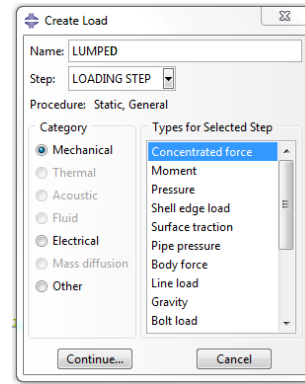
**Figure 35.** Box (excluding outer most edges)

- If this procedure has been done correctly a total of 15 nodes will turn a red color.
- Click **Done**.
- Specific concentrated forces will be applied to all of the interior nodes of the model to create a lumped load.
- Double click **Loads(1)** in the model tree and the **Create Load** dialog box will immediately appear (Figure 36a). Create a **Name** for the load called **LUMPED**, and under the **Step** drop down menu

make sure to choose **LOADING STEP**. Under the **Category** option choose **Mechanical**, and choose **Concentrated force** under the **Types for Selected Step** option. The **Create Load** dialog box should look identical to that in Figure 36b.

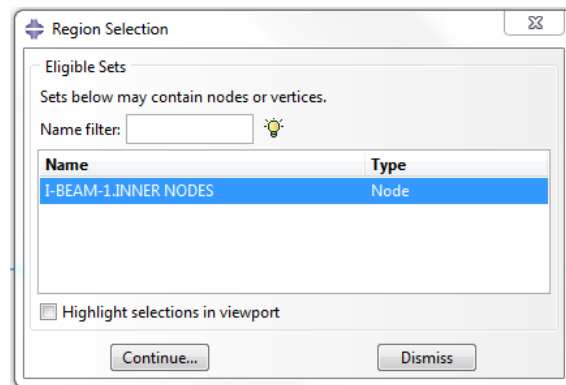


**Figure 36a.** Create Load



**Figure 36b.** Create Load (LUMPED)

- Click **Continue...**
- In the bottom right hand corner of the viewport click **Sets...** and the **Region Selection** dialog box will immediately appear. Since only one set has been created click **I-BEAM-1.INNER NODES**. The **Region Selection** dialog box should look identical to that in Figure 37.



**Figure 37.** Region Selection Dialog Box

- Click **Continue...**
- The **Edit Load** dialog box will immediately appear (Figure 38a). Enter a value of **-62.5** in the **CF2:** option. This will enter a force of 62.5 lbs in the – Y direction at each of the interior nodes in the selectedset. The **Edit Load** dialog box should look identical to that in Figure 38b.

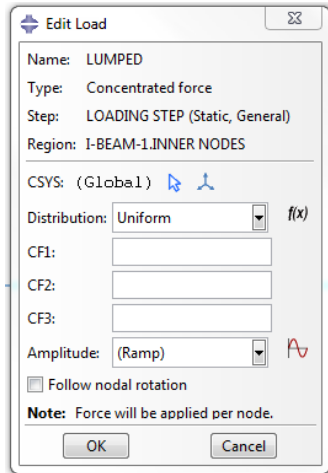


Figure 38a. Edit Load Dialog Box

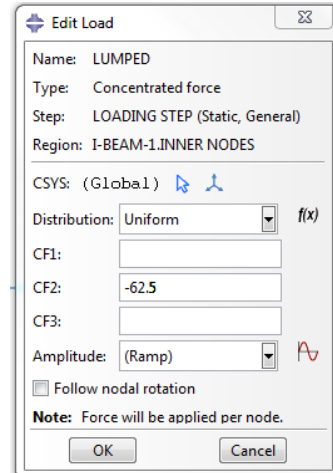


Figure 38b. Edit Load Dialog Box (LUMPED)

- Click **OK**. If this procedure has been done correctly, 15 yellow arrows will appear pointing in the – Y direction on the set of inner nodes (Figure 39).

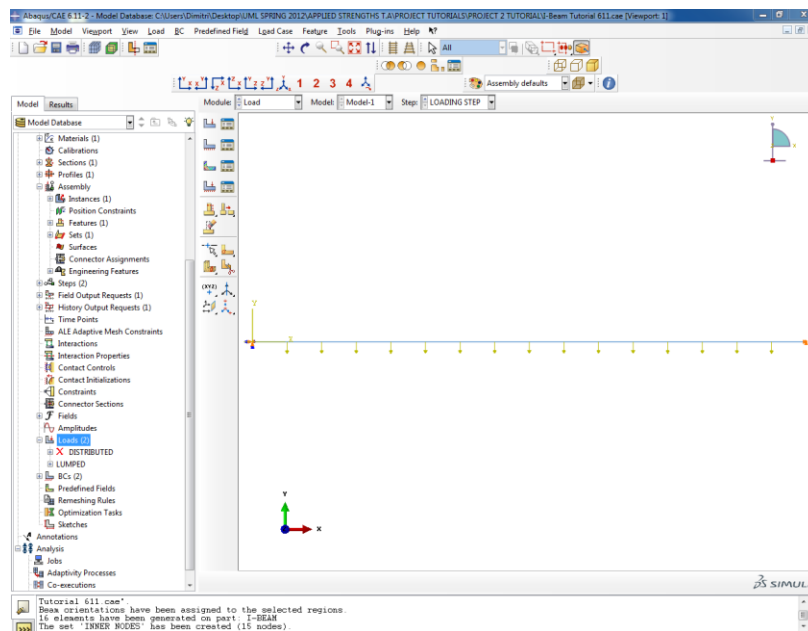
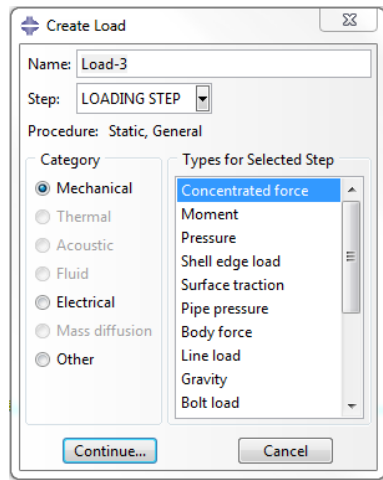


Figure 39. Lumped Loading Condition

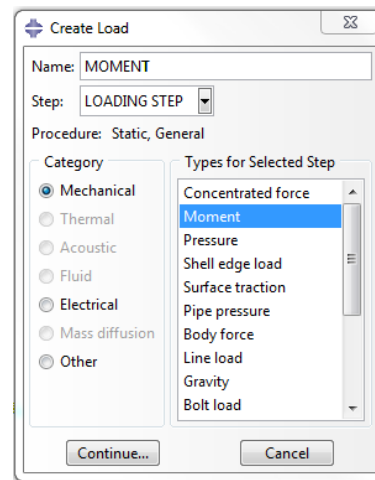
### Applying a Consistent Load to the Beam

- To apply a consistent load to the beam the lumped loading condition will be used as well as a moment about the Z axis applied to the roller end.
- Double click **Loads(2)** in the model tree and the **Create Load** dialog box will immediately appear (Figure 40a). Create a **Name** for the load called **MOMENT**, and under the **Step** drop down menu make sure to choose **LOADING STEP**. Under the **Category** option choose **Mechanical**, and

choose **Moment** under the **Types for Selected Step** option. The **Create Load** dialog box should look identical to that in Figure 40b.

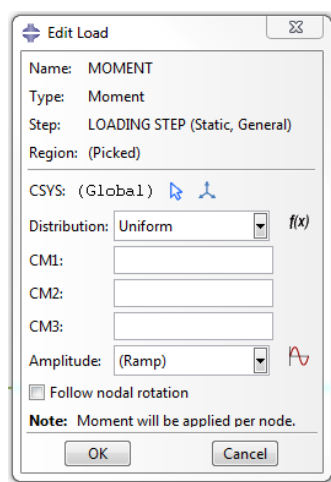


**Figure 40a.** Create Load Dialog Box

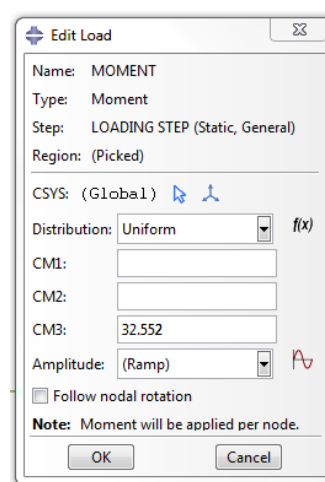


**Figure 40b.** Create Load Dialog Box (MOMENT)

- Click **Continue...**
- You may be prompted by the Region Selection Dialog Box. If so click **Dismiss**. Using the cursor, click the node on the right end of the beam. If this node has been selected correctly it will turn a red color.
- Click **Done**.
- The **Edit Load** dialog box will immediately appear (Figure 41a). Enter a value of **32.552** in the **CM3**: option. The concentrated moment is equal to  $qL^2/12$ , where  $q$  is the distributed load per inch and  $L$  is the effective length of a beam element,  $L = 100/16$  in. The **Edit Load** dialog box should look identical to that in Figure 41b.



**Figure 41a.** Edit Load Dialog Box

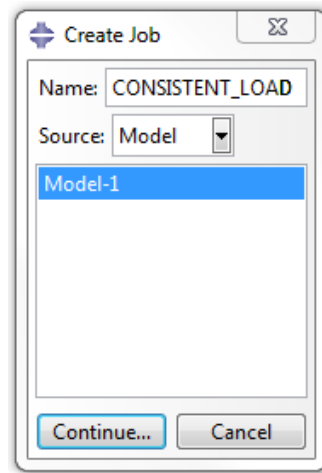


**Figure 41b.** Edit Load Dialog Box (MOMENT)

- Click **OK**.

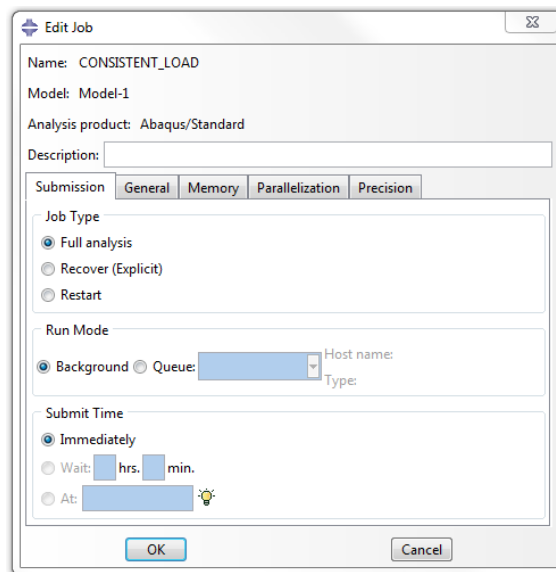
## Creating a Job

- To create a job for this model, double click the **Jobs** icon in the model tree. Up to this point, you have been preprocessing the model. A job will take the input file created by the preprocessor and process the model, i.e. perform the analysis. In the **Create Job** dialog box, create a **Name** for this job called **CONSISTENT\_LOAD**. Blank spaces are not allowed in a job name. Thus the use of the underline in the name. The Create Job dialog box should look identical to that in Figure 42.



**Figure 42.** Create Job Dialog Box (CONSISTENT\_LOAD)

- Click **Continue...**
- The **Edit Job** dialog box will immediately appear (Figure 43).



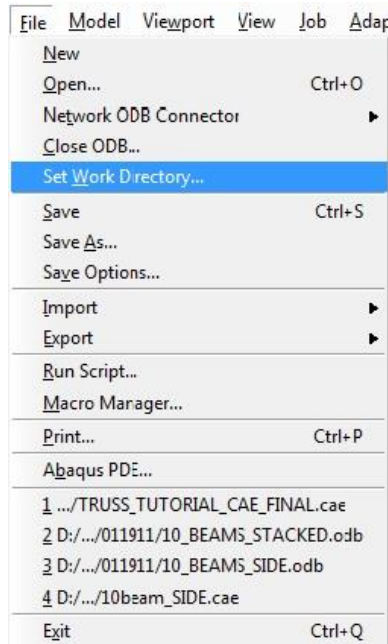
**Figure 43.** Edit Job Dialog Box



- Accept the default values by clicking **OK**.

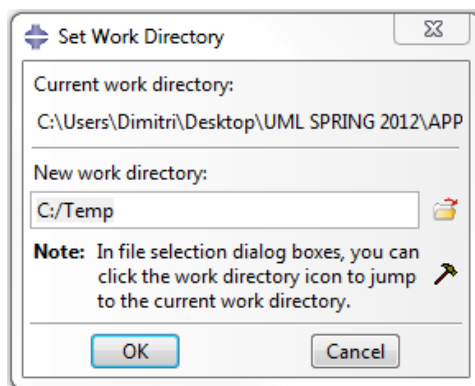
### Setting the Work Directory

- To ensure that the input files write to the correct folder, setting the work directory must be accomplished. At the top of the screen, click **File** and in the dropdown menu click **Set Work Directory...** (Figure 44).



**Figure 44.** Set Work Directory

- The **Set Work Directory** screen will immediately appear (Figure 45). Click **Select...** and use standard Windows practice to select (and possibly create) a subdirectory.



**Figure 45.** Set Work Directory (FOLDERS)

- Click **OK**.
- Click **OK**.

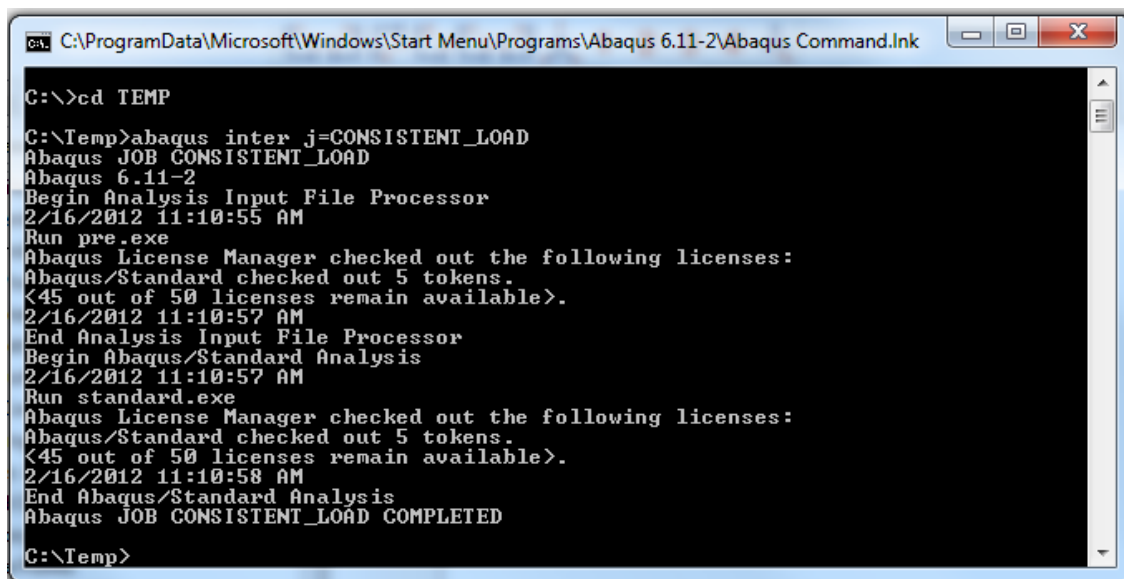
## Writing the Input File (.inp)

- To write the input file of the job that was created, first click the + next to **Jobs(1)** in the model tree.
- Right click the job called **CONSISTENT\_LOAD** and click the **Write Input** option. This choice will write an input file (.inp) of this model to the work directory.
- It may be helpful to go to the folder on the computer to which the work directory is set to ensure that the input file was written there.

## Model Analysis (ABAQUS Command)

### Method #1

- Go to the **Start Menu** and open **Abaqus Command**
- Abaqus is set to a default directory (Example K:\>). To change directories in the Abaqus Command type the directory of choice followed by a colon (**C:**) then hit **Enter**.
- To access a specific directory within that drive type **cd** followed by the specific folder name in that directory (e.g., **cd TEMP**) then hit **Enter**.
- Now that the correct directory has been sourced in the command window type **abaqus inter j=CONSISTENT\_LOAD** and then hit **Enter**.
- If the job has completed successfully the Abaqus prompt should look similar to Figure 46.

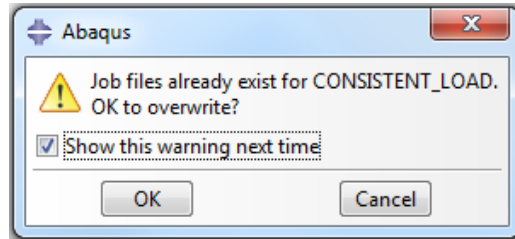
A screenshot of a Windows command prompt window titled "C:\ProgramData\Microsoft\Windows\Start Menu\Programs\Abaqus 6.11-2\Abaqus Command.Ink". The window shows the following text:

```
C:\>cd TEMP
C:\Temp>abaqus inter j=CONSISTENT_LOAD
Abaqus JOB CONSISTENT_LOAD
Abaqus 6.11-2
Begin Analysis Input File Processor
2/16/2012 11:10:55 AM
Run pre.exe
Abaqus License Manager checked out the following licenses:
Abaqus/Standard checked out 5 tokens.
<45 out of 50 licenses remain available>.
2/16/2012 11:10:57 AM
End Analysis Input File Processor
Begin Abaqus/Standard Analysis
2/16/2012 11:10:57 AM
Run standard.exe
Abaqus License Manager checked out the following licenses:
Abaqus/Standard checked out 5 tokens.
<45 out of 50 licenses remain available>.
2/16/2012 11:10:58 AM
End Abaqus/Standard Analysis
Abaqus JOB CONSISTENT_LOAD COMPLETED
C:\Temp>
```

Figure 46. Abaqus Command Prompt (COMPLETED)

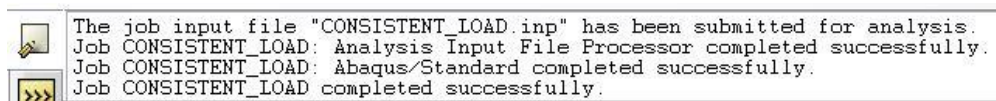
## Method #2

- An alternative method for submitting an \*.inp file for processing by Abaqus can be accomplished with Abaqus/CAE
- Right click the job called **CONSISTENT\_LOAD** and click the **Submit** option.
- If you see a warning:



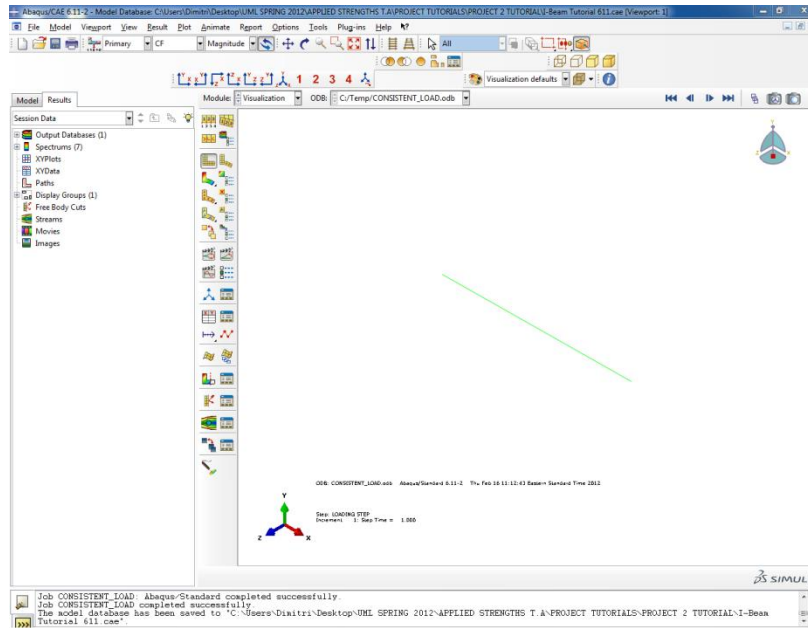
Click **OK**. The intent of this warning is to prevent the user from accidentally overwriting a previously completed analysis with the same name.

- The model will now be submitted for analysis by Abaqus and the progress can be viewed in the status window at the bottom of the screen.



## Postprocessing using ABAQUS CAE

- After the analysis has successfully completed in the Abaqus Command window using method #1 or using Method #2, return to view the Abaqus/CAE viewport.
- Because the last step of creating the model was to create a job/write (and possibly submit) an input file, the **CONSISTENT\_LOAD** job should still be highlighted in Abaqus/CAE model tree. Right click the **CONSISTENT\_LOAD(Completed)** and then click **Results**.
- If this selection was done correctly, the model should turn to a green color and the model will have rotated to an isometric view (Figure 47).




**Figure 47. Analysis Results Isometric View**

- To rotate the beam back into the X Y plane for viewing, click **View** in the toolbar at the top of the screen. Next, Click **Toolbars** and make sure the option **Views** has a check mark to the left of it. If not, then click it.
- The **Views** toolbar will appear (Figure 48), and the **Apply Front View** button can be clicked to view the model in the X Y plane.



**Figure 48. Views Toolbar**

- To view the deformed shape of the model, click the **Plot Contours on Deformed Shape** icon  in the **Visualization** module. The model should look similar to that in Figure 49.

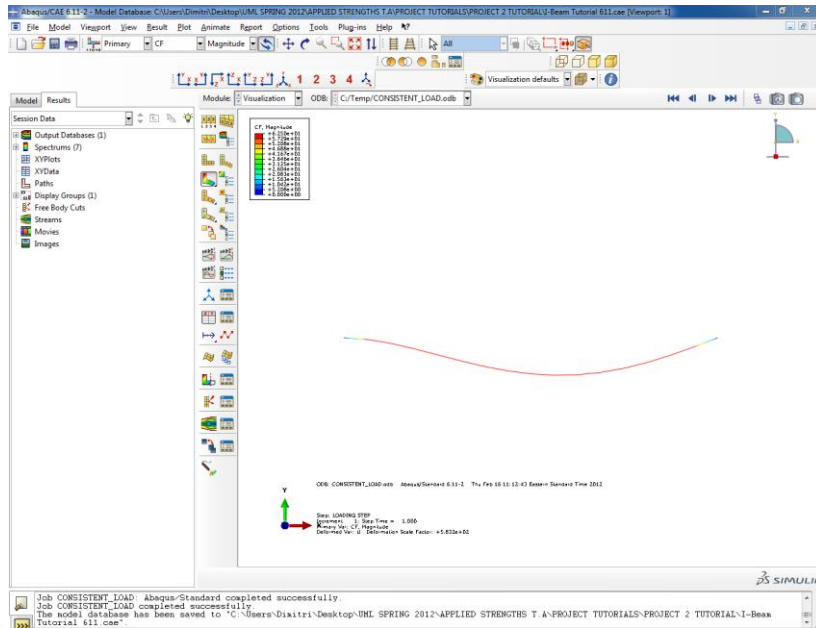


Figure 49. Deformed Shape

### Obtaining Displacement Plots

- To obtain a plot of the nodal displacement a path must be created. Double click **Paths** in the results tree and the **Create Path** dialog box will appear (Figure 50a). Create a **Name** for the path called **LENGTH** and ensure that **Edge list** is selected under **Type**. The **Create Path** dialog box should look identical to that in Figure 50b.

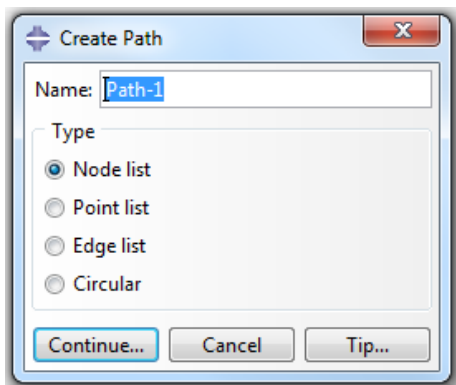


Figure 50a. Create Path Dialog Box

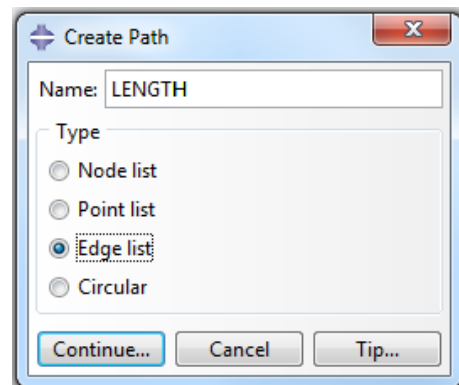
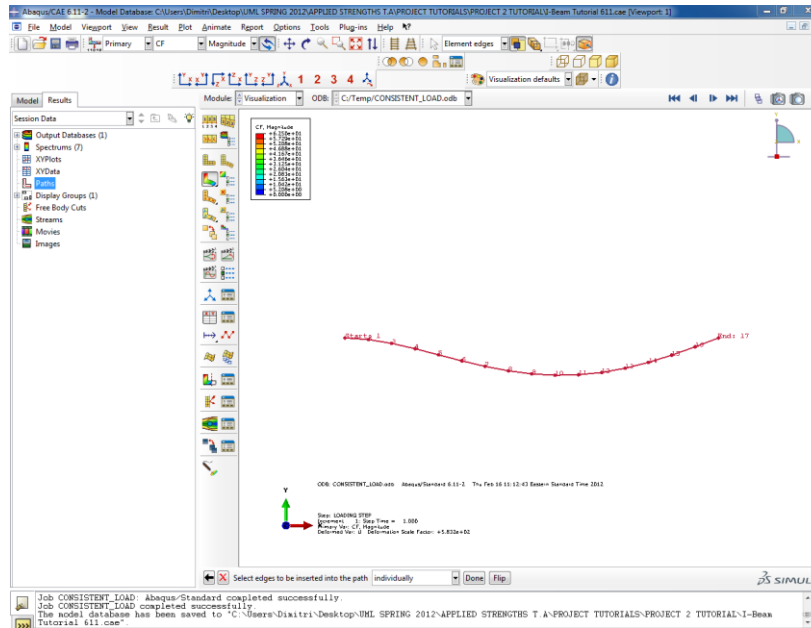



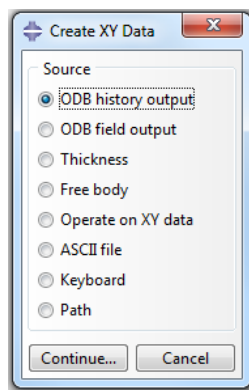
Figure 50b. Create Path Dialog Box (LENGTH)

- Click **Continue...**
- The **Edit Edge List Path** dialog box will immediately appear. Click **Add Before...** and the dialog box will disappear. At this click all of the elements on the beam starting from the left end. As each element is selected a red line will connect appear on that particular element. The model geometry should look identical to that in Figure 51.

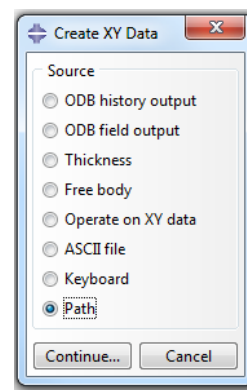


**Figure 51.** Path of Elements

- Click **Done**. The **Edit Edge List Path** dialog box will immediately appear. Click **OK**.
- Now that the path has been created, click the **Create XY Data** icon  in the **Visualization** module. The **Create XY Data** dialog box will appear (Figure 52a). Under **Source**, choose the **Path** option (Figure 52b).

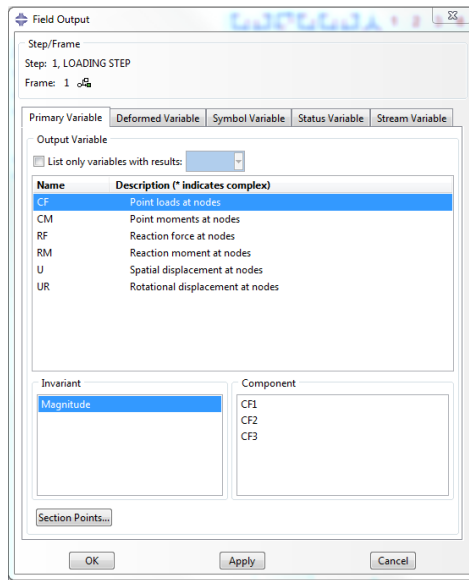


**Figure 52a.** Create XY Data Dialog Box

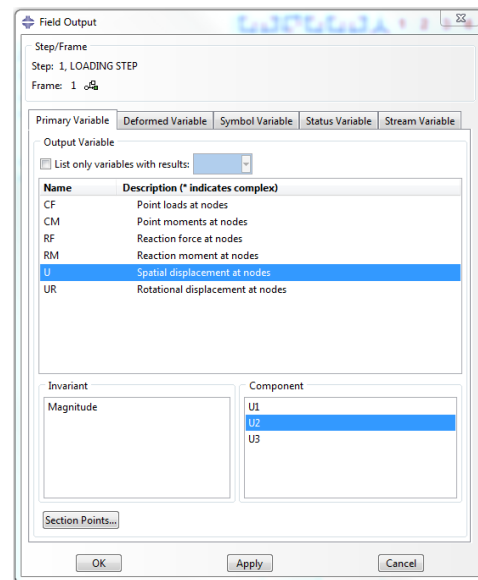


**Figure 52b.** Create XY Data Dialog Box (PATH)

- Click **Continue...**
- The **XY Data from Path** dialog box will appear. Under the **X Values** option click **True Distance**, under the **Y Values** option click **Field Output...** and the **Field Output** dialog box will immediately appear (Figure 53a). Under the **Primary Variable** tab click **U Spatial displacements at nodes**, and under the **Component** option click **U2**. The **Field Output** dialog box should look identical to that in Figure 53b.

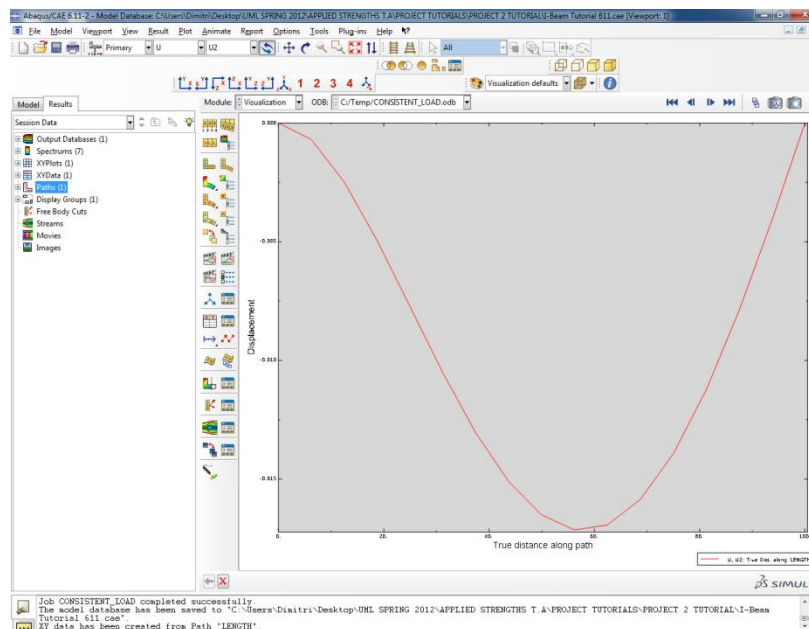


**Figure 53a.** Field Output Dialog Box




**Figure 53b.** Field Output Dialog Box (U2)

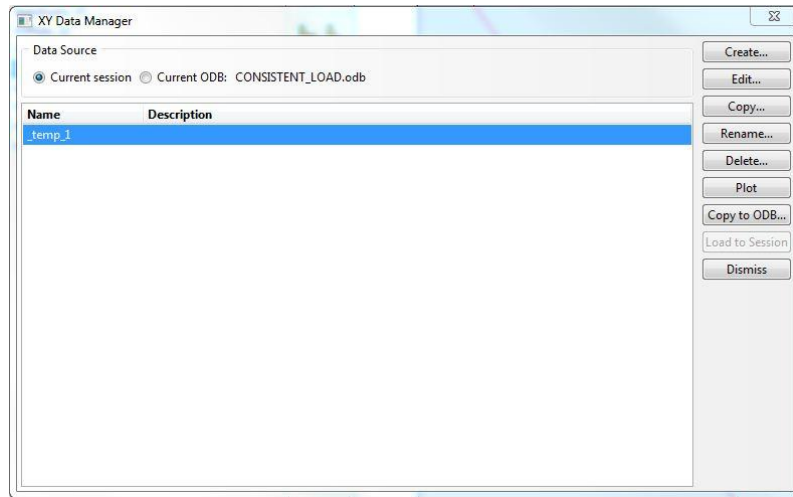
- Click **OK**. The **Field Output** dialog box will disappear and the **XY Data from Path** dialog box will reappear. Click **Plot**. The viewport should change from viewing the deformed shape of the model to a plot (Figure 54).



**Figure 54.** Plot of Y deflection (U2)

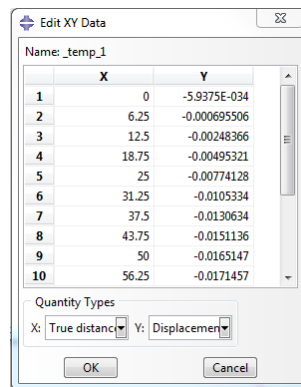
- Click **Cancel**.

- To view the numerical values of the Y displacement of the beam, click the **XY Data Manager** icon  in the **Visualization** module. This option is located directly to the right of the **Create XY Data** option.
- The **XY Data Manager** dialog box will appear (Figure 55).



**Figure 55.** XY Data Manager Dialog Box

- Double Click **\_temp\_1** and an **Edit XY Data** dialog box will appear (Figure 56). These are the X and Y values which generate the plot. The values can be cut and pasted into Excel for further post processing.



**Figure 56.** Edit XY Data Dialog Box

- Click **OK** in the **Edit XY Data** dialog box. Click **Dismiss** in the **XY Data Manager** dialog box.
- This tutorial completes the post processing for the **Consistent Load** of the project. To complete both the **Distributed** and **Lumped** loading return to the model tree by clicking the **Model** tab at the top left of the screen.
- Earlier in the tutorial the **DISTRIBUTED** load was suppressed. To complete the analysis using the **DISTRIBUTED** load, **Suppress** both the **LUMPED** and **MOMENT** loads and **Resume** the

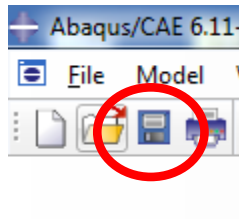


**DISTRIBUTED** loading condition. (To resume the distributed load right click DISTRIUBTED in the model tree and click resume).

- Create a new Job called **DISTRIBUTED\_LOAD** and complete the post processing as needed. Likewise, suppress the DISTRIBUTED and MOMENT loads and create a new Job called **LUMPED\_LOAD** and complete the post processing as needed.

### Conclusion

- Save the file by doing either **File > Save** or clicking the disk icon



- Close Abaqus/CAE: **File > Exit** or **Ctrl+Q**
- This completes the **Finite Element Analysis of a Propped Cantilever Beam Tutorial**.