FINITE ELEMENT ANALYSIS OF A PLANAR TRUSS

Instructor: Professor James Sherwood

Revised: Dimitri Soteropoulos

Programs Utilized: Abaqus/CAE 6.11-2

Problem Description:

This tutorial explains how to build and analyze a planar truss. The pre-processing program used is Abaqus/CAE, and Abaqus command is used for the analysis.

The geometry of the truss section is shown in Figure 1.

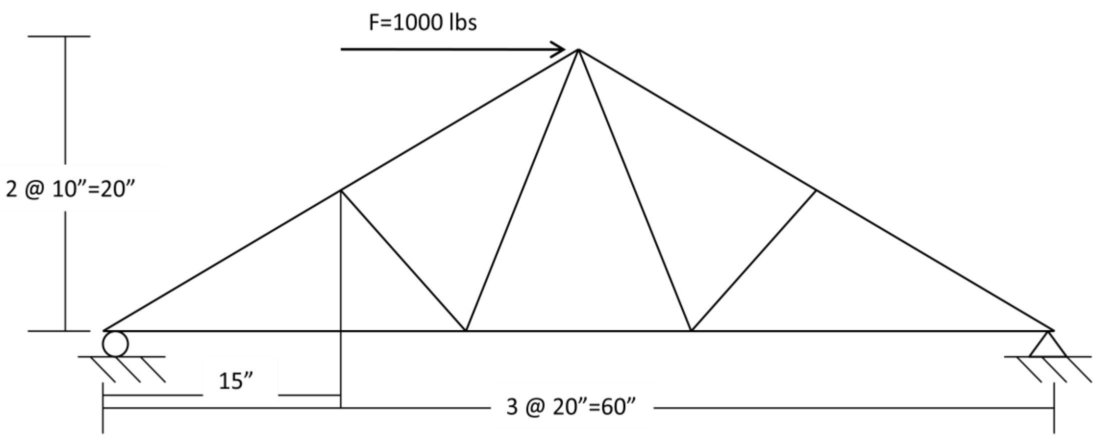


Figure 1. Truss Dimensions and BC’s

As seen in Figure 1, a pin connection will be used to constrain the right side of the truss, and a roller to constrain the left side.

Creating the Model Geometry

- Go to the **Start Menu** and open **Abaqus CAE**
- You may be prompted with an **Abaqus/CAE 6.11 Start Session** box (Figure 1). Close this box by clicking the **X** in the top right hand corner.

Figure 1. Abaqus/CAE 6.11 Start Sesssion box.

- Once the **Start Session** box is exited, the Abaqus/CAE Viewport should look similar to Figure 2. (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)

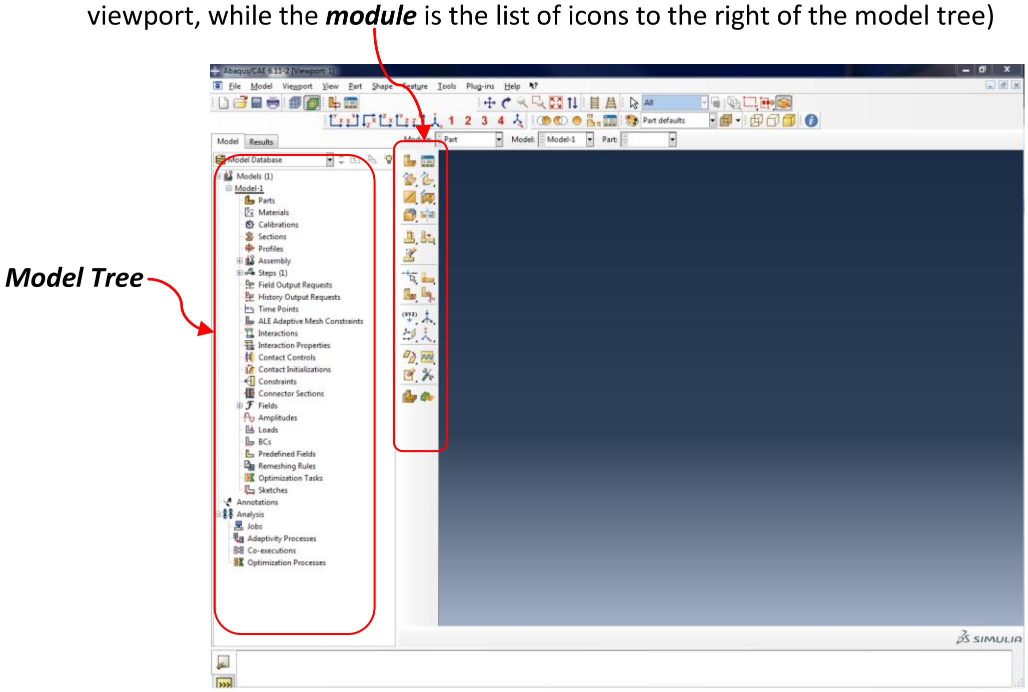


Figure 2. Abaqus/CAE Viewport

- To create the model geometry of the planar truss, a series of points and lines must be generated.

- Using the left mouse button, double click **Parts** in the model tree and the **Create Part** (Figure 3a) dialog box appears. Enter a new name for the part (TRUSS), and under the Base Feature tab choose **Wire** for shape (as in wireframe) and **Planar** for type. The **Create Part** dialog box should look identical to Figure 3b.
- Click **Continue**… and the graphics window will change to a set of gridlines.

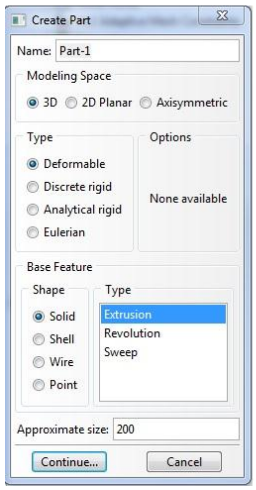


Figure 3a. Create Part Dialog Box

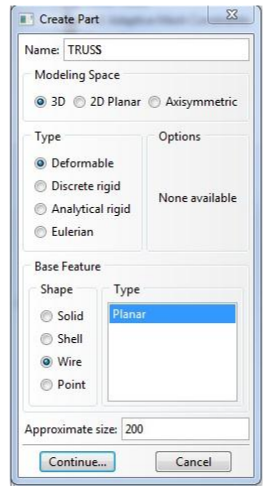


Figure 3b. Create Part Dialog Box (Truss)

- For the first step in generating the model geometry, seven isolated points must be created. Click the **Create Isolated Points** icon  in the module. (Remember, the module is the series of icons to the right of the model tree)
- On the bottom of the Viewport, a **Pick a point—or enter X,Y:** option can be seen. This "enter X,Y" option will be used to enter the x and y coordinates of generated points.
- Enter the points given in Table 1; press enter on your keyboard after each point entry. You will see each point appear on the screen after it has been entered.

Table 1. Point Coordinates

Point	X	Y
1	0	0
2	20	0
3	40	0
4	60	0
5	15	10
6	30	20
7	45	10

- After all the points have been entered, click the red **X** button once. The **X** button is located in the bottom left hand corner of the viewport.
- If all the points cannot be seen in the Viewport, click **F6** on your computer keyboard to auto-fit the window to the contents. After auto fitting, the screen should look similar to Figure 4.

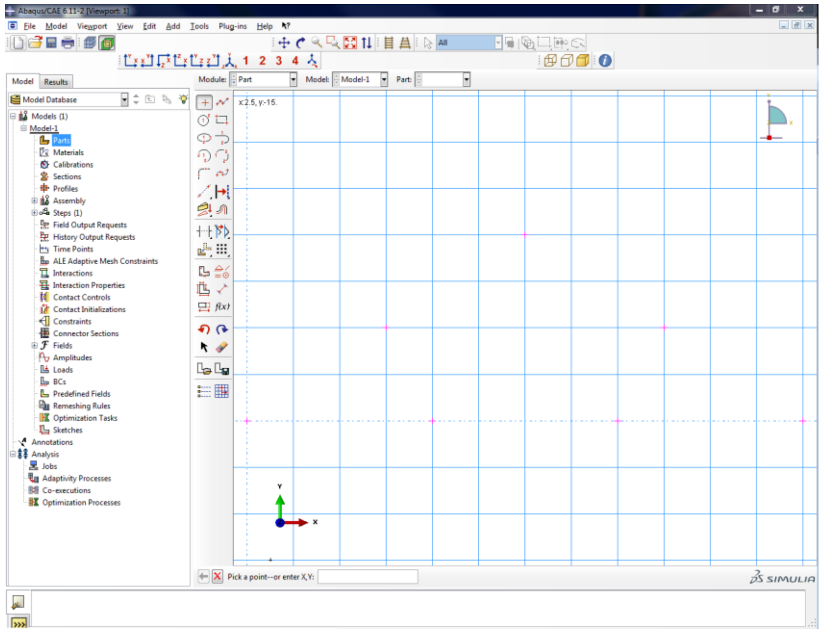


Figure 4. View of Seven Points

- The last step in creating the model geometry is to generate lines between the points, thus completing the truss network.
- Click the **Create Lines: Connected** icon in the module, it is located directly to the right of the **Create Isolated Points** icon.
- With the cursor, click two individual points to create a line between them. A total of 11 lines should be generated in this model. (Please note, when the last line is created, the line feature is exited by clicking the **Esc** key on your computer keyboard.)
- NOTE: If you accidentally create an unwanted line, you can select **Edit** > **Delete** from the dropdown menu at the top of the screen and use the mouse to select a line to delete.
- After the lines have been generated, click **Done.**
- Sketch mode will automatically be exited, and the model geometry should look identical to the truss shown in Figure 5.

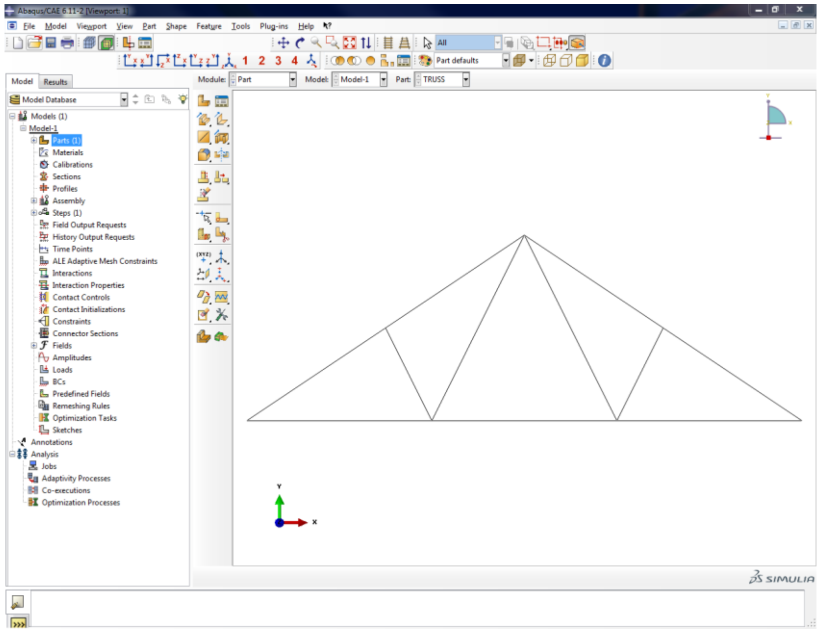


Figure 5. Final Model Geometry

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 6a). Enter a **Name** for the material (STEEL), and click the **Mechanical** tab, highlight **Elasticity** and click **Elastic.** Enter values of **Young's Modulus** = 30E06 psi, and **Poisson's Ratio** = 0.3. After the material properties have been entered, the **Edit Material** dialog box should look identical to Figure 6b.
- Click **OK**.

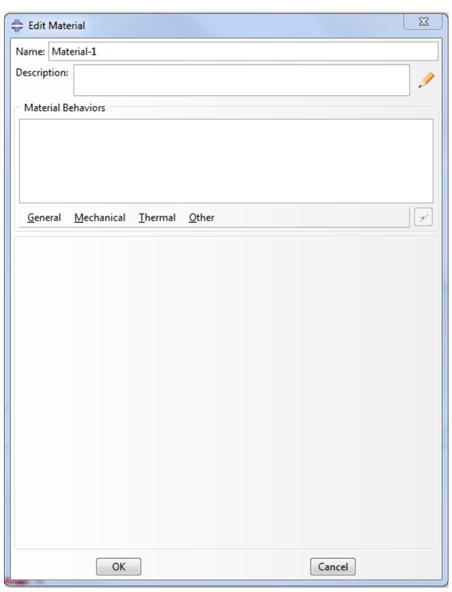


Figure 6a. Edit Material Dialog Box

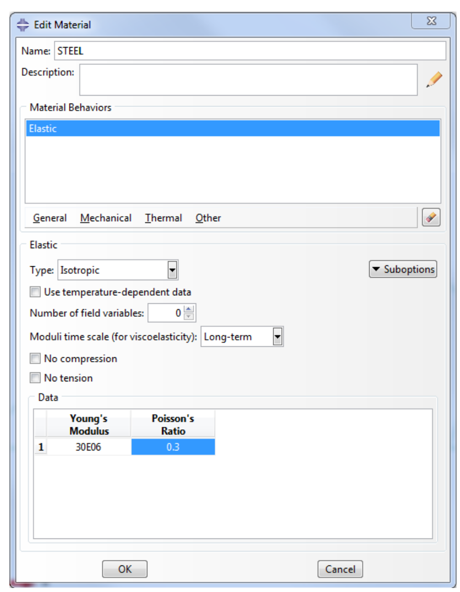


Figure 6b. Edit Material Dialog Box (STEEL)

- 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 respective 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 and dictate what units should be used for the dimensions of the structure.
- At this point in preprocessing, the model should be saved. Click **File** then click **Save.** Name the file Truss 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 truss section in Abaqus, double click **Sections** in the model tree and the **Create Section** dialog box will appear (Figure 7a). Enter a **Name** for the section (TRUSS), and choose **Beam** under the Category Tab, and **Truss** under the Type tab. Your Create Section dialog box should look identical to that in Figure 7b.
- Click **Continue**…

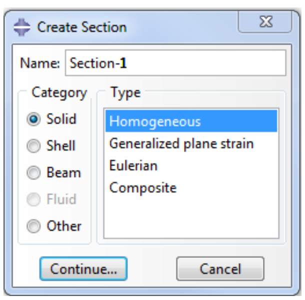


Figure 7a. Create Section Dialog Box

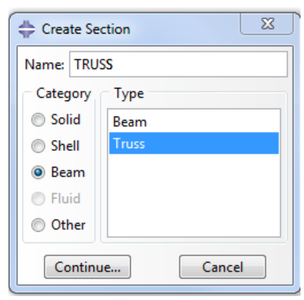


Figure 7b. Create Section Dialog Box Truss

- The **Edit Section** dialog box will then appear where a **Material** and **Cross-Sectional area** can be defined for this section. Because only one material has been created, the **Material** is defaulted to **STEEL**. If multiple materials had been created, the dropdown menu could be used to prescribe a different material to this section.
- Enter **12.566** for the **Cross-Sectional area** (This value accounts for a 2-in. radius rod, i.e. πr^2). The Edit section dialog box should look identical to that in Figure 8.
- Click **OK**.

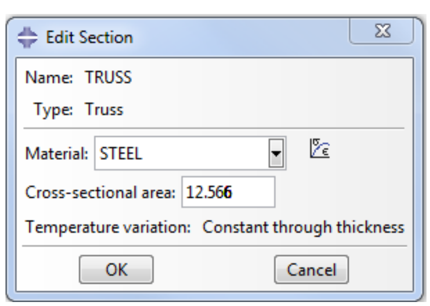


Figure 8. Edit Section Dialog Box (TRUSS)

Assigning Sections

- Now that the truss sections have been created, they 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 **TRUSS**, further expanding the model tree (Figure 9).

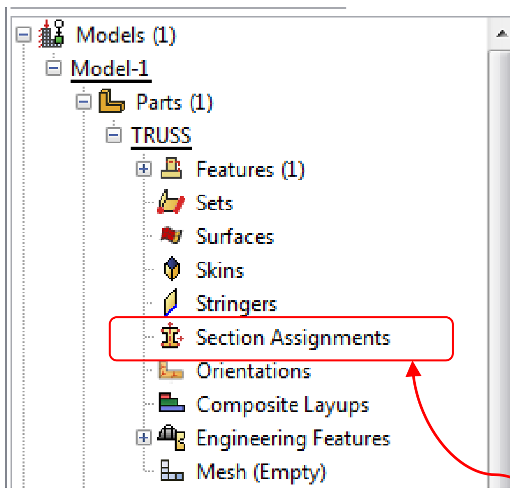


Figure 9. Model Tree Expansion (Parts)

- After the model tree has been expanded, double click **Section Assignments**. Use the cursor to select all of the lines by holding down the left mouse key and draging the cursor around the truss to create a box around the whole model. If this drag has been done correctly, the model will change color from grey to red. Click **Done**.
- The **Edit Section Assignment** dialog box will appear (Figure 10). Because only one section has been created, the dropdown menu defaults to the **TRUSS** section. If multiple sections had been created, the dropdown menu could be used to assign different sections to different parts of the geometry.
- Click **OK**. The model now should now turn to a blue color.

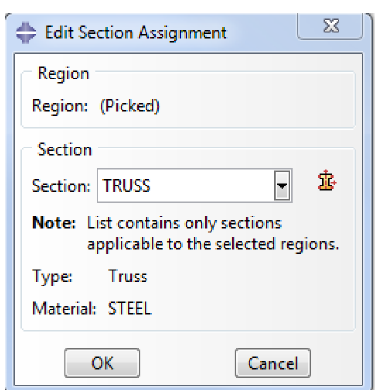


Figure 10. Edit Section Assignment Dialog Box (TRUSS)

- Finally, 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 11).

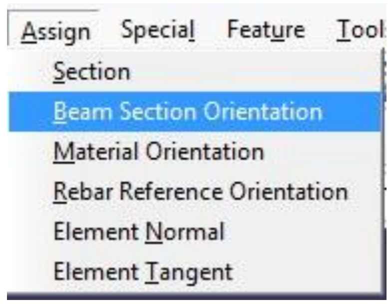


Figure 11. Beam Section Orientation Drop down Menu

- 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, hit **Enter.** The model should look identical to Figure 12.

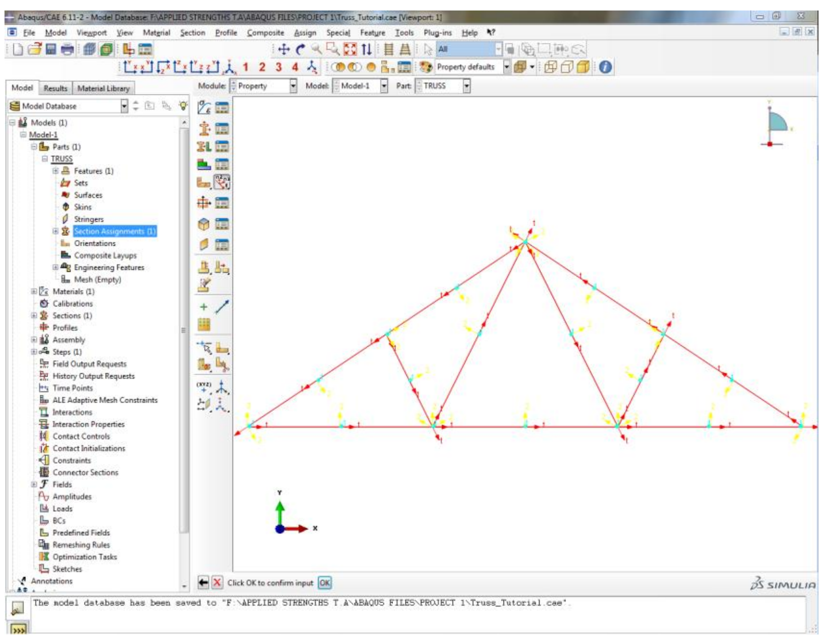


Figure 12. Beam Section Orientation

- Hit **OK**.
- Click **Done**. The model should turn back to a blue color.
- This last step is used to define the orientation of a beam in space. Because the current model is using truss elements, this orientation is not critical, so we accept the default definition.

However, if we were using beam elements, then we would need to be careful in defining this orientation.

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 Part** icon  in the mesh module and you will be prompted by the **Global Seeds** dialog box (Figure 13a). Change the **Approximate global size** to 20 and click **Apply**. The **Global Seeds** dialog box should look identical to Figure 13b.
- Click **OK**.

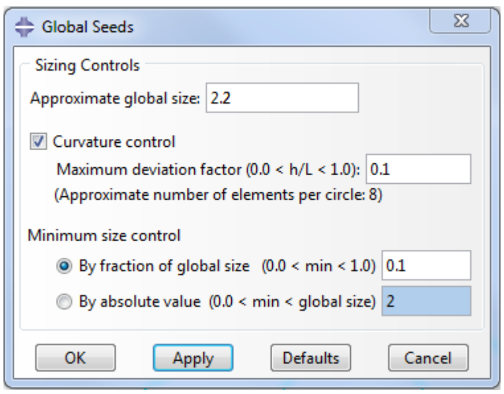


Figure 13a. Global Seeds Dialog Box

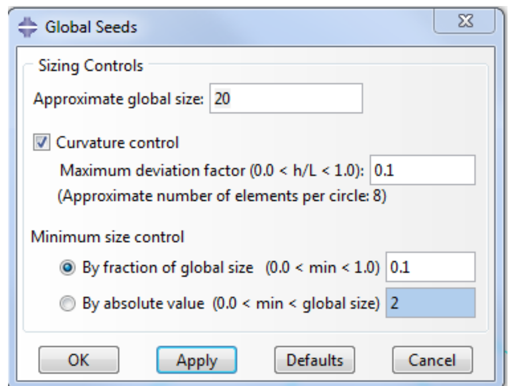


Figure 13b. Global Sees Dialog Box (20)

- Now that the part has been seeded, a mesh can be generated. Click the **Assign Element Type** icon  in the mesh module.
- Using the cursor drag the mouse while holding the left mouse key down to create a box around the whole geometry. If this drag is done correctly, then the part will turn from pink to red (Figure 14).

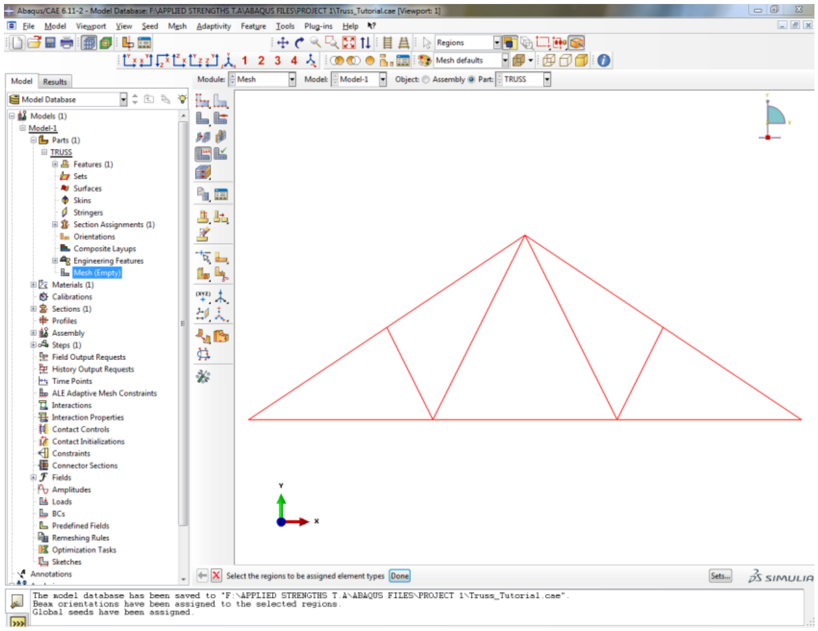


Figure 14. Assign Element Type

- Click **Done**. You will be immediately prompted by the **Element Type** dialog box. Under the **Family** category scroll down and choose **Truss**. Your **Element type** dialog box should look identical to Figure 15.

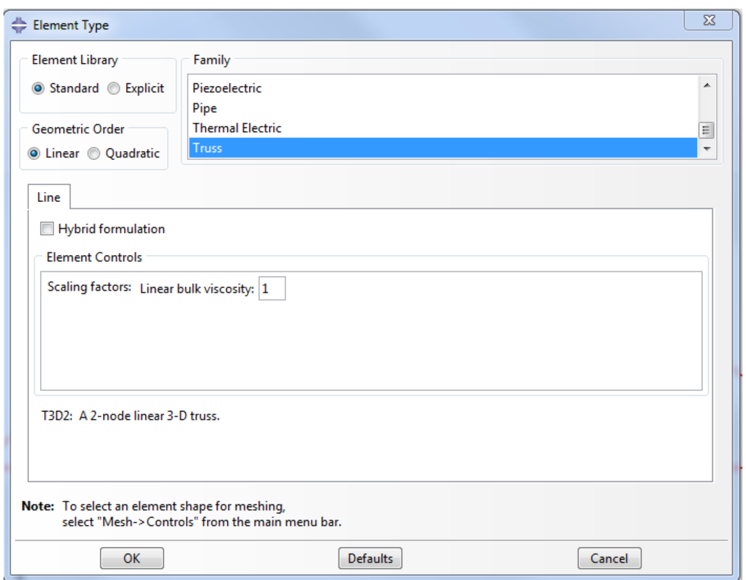


Figure 15. Element Type (Truss)

- Click **OK**.
- Click **Done**
- Note: The T3D2 element type code implies: T=Truss element class, 3D=three-dimensional and 2=two noded element.

- 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 (Figure 16).

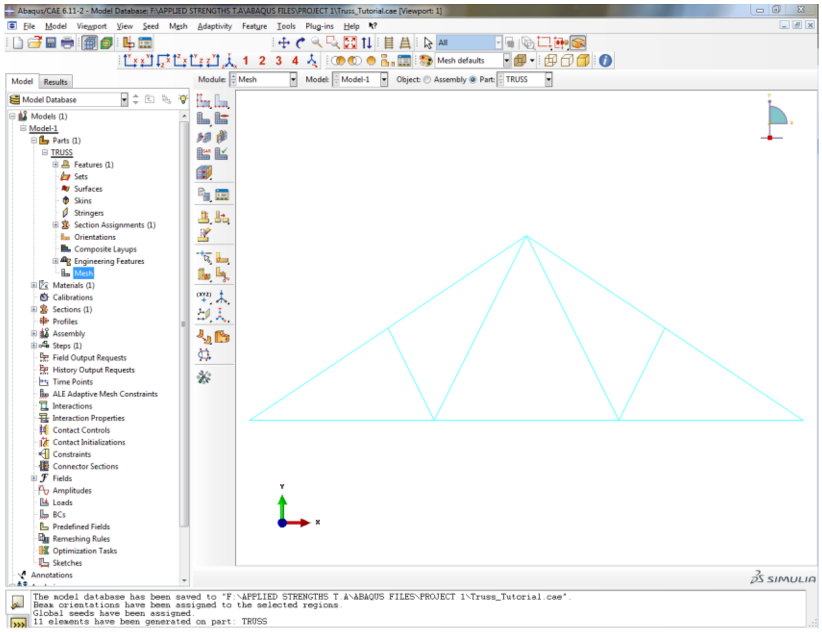


Figure 16. Meshed Geometry

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 17.

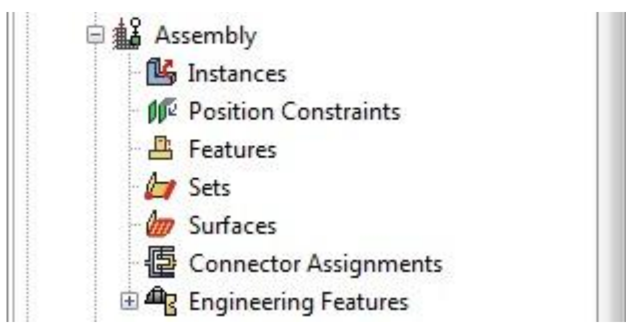


Figure 17. Model Tree Expansion (Assembly)

- 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 18).

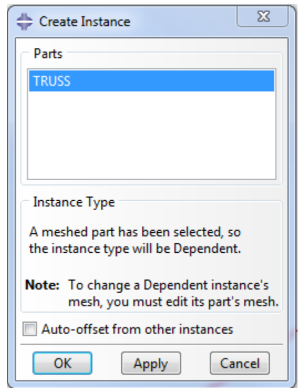


Figure 18. Create Instance Dialog Box

- The **TRUSS** 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 19).

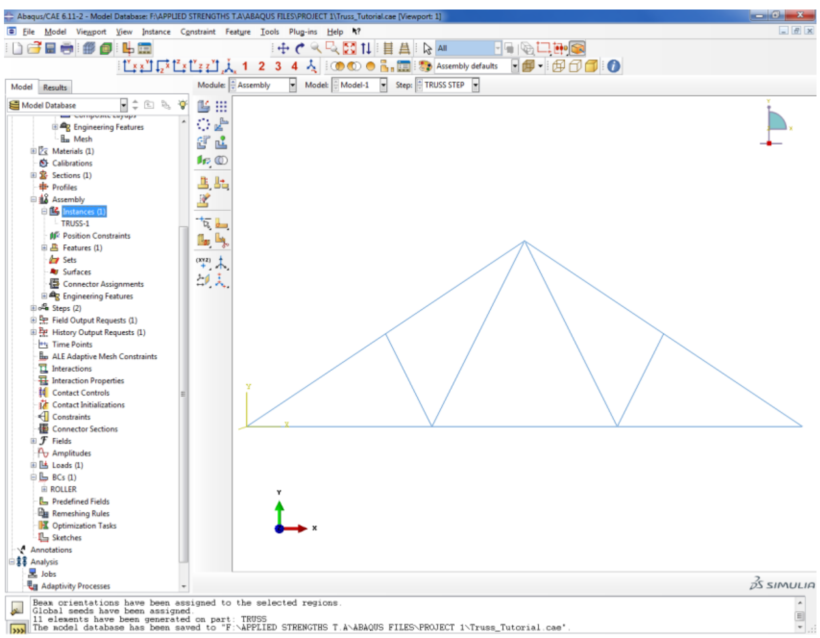


Figure 19. Create Instance

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 20a). Create a **Name** for the step called **TRUSS STEP**. Under **Procedure type** choose **General** > **Static, General**. The **Create Step** dialog box should look identical to Figure 20b.

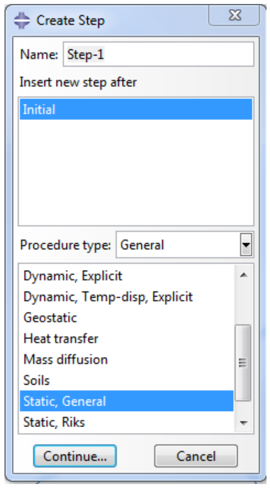


Figure 20a. Create Step Dialog Box

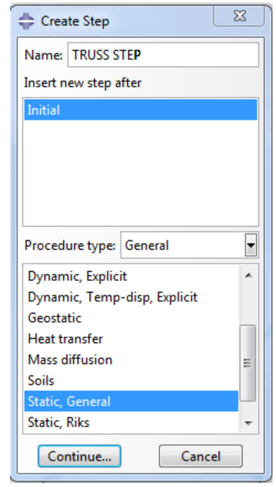


Figure 20b. Create Step Dialog Box (TRUSS STEP)

- Click **Continue**…, and the Edit Step dialog box will immediately appear (Figure 21).

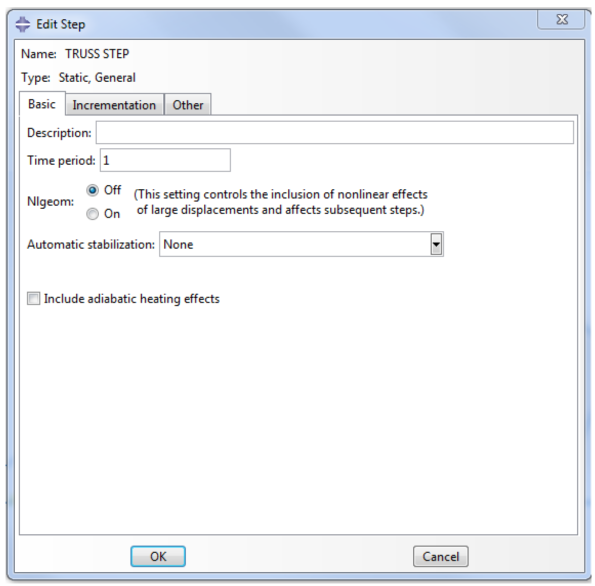


Figure 21. Edit Step Dialog Box

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

Apply a Load Boundary Condition

- A force of F_x=1000 lbs will be applied to the top node. Double click the **Loads** icon 

in the model tree. The **Create Load** dialog box will appear (Figure 22a).
- Create a name for the load called **LOAD** and ensure that **TRUSS STEP** is selected for the **Step**. Choose **Mechanical** under the **Category** option and **Concentrated force** under the **Types** for **Selected Step** option. The **Create Load** dialog box should look identical to that in Figure 22b.

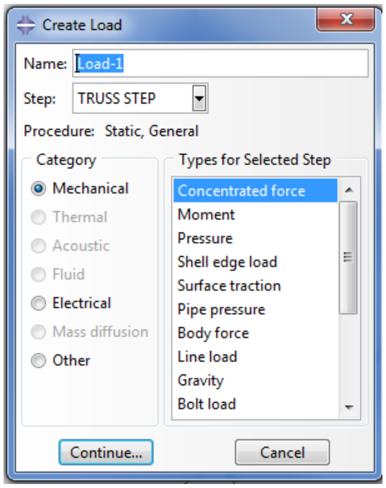


Figure 22a. Create Load Dialog Box

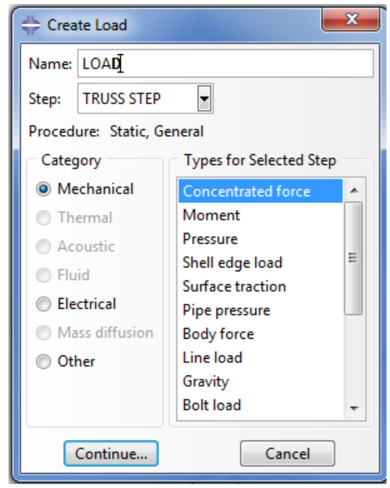


Figure 22b. Create Load Dialog Box (LOAD)

- Click **Continue**…
- At this point all of the available nodes on which the concentrated force can be applied will turn the color yellow. Using the cursor click on the top node of the structure. If this selection is done correctly, the node color will turn from yellow to red (Figure 23).

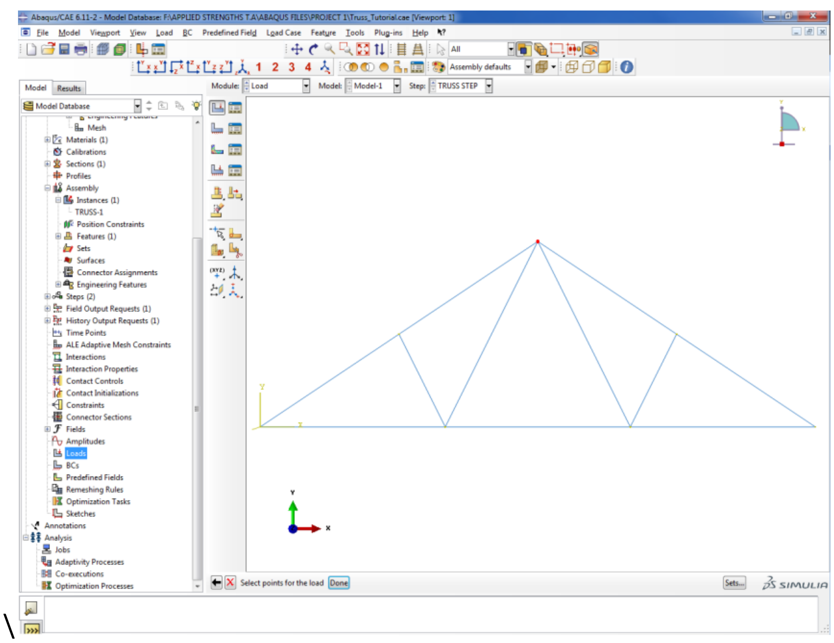


Figure 23. Concentrated Force on Top Node

- Click **Done**. The **Edit Load** dialog box will immediately appear (Figure 24a). Enter a value of 1000 into the **CF1** option. This entry will apply a load of 1000 lbs in the positive X direction. The **Edit Load** dialog box should look identical to that in Figure 24b.

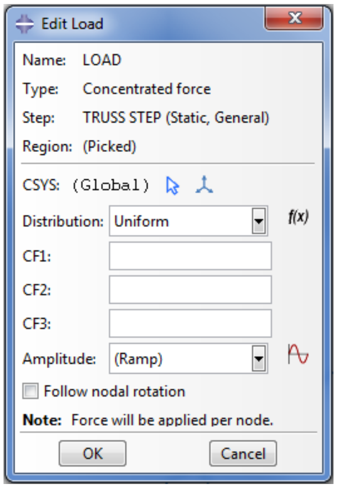


Figure 24a. Edit Load Dialog Box

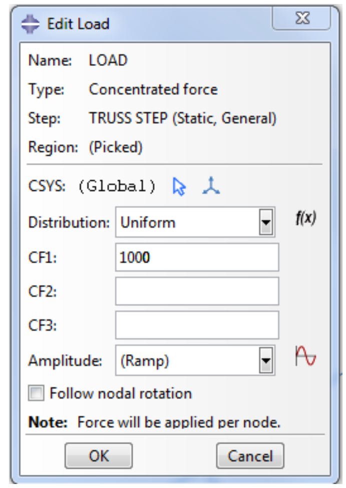


Figure 24b. Edit Load Dialog Box (1000)

- Click **OK.** If this step was done properly, then a small yellow arrow will appear at the node where the force was applied and points in the positive X direction (Figure 25).

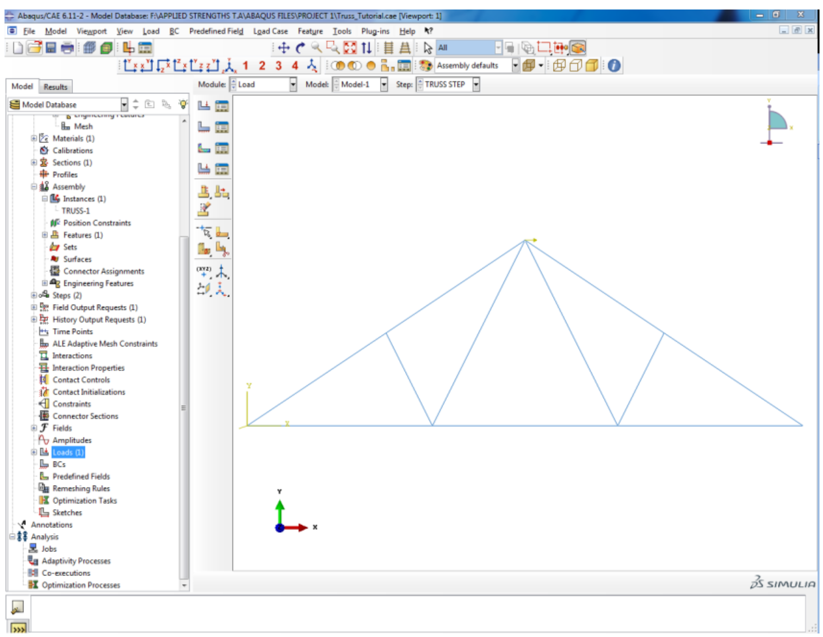


Figure 25. 1000-lb Load

Apply Constraint Boundary Conditions

- Two different boundary conditions must be applied to this model. As shown in Figure 1, the left side of the structure is constrained by a roller support, while the right side is constrained by a pinned support.
- Double click the **BCs** icon in the model tree and the **Create Boundary Condition** dialog box will appear (Figure 26a). Create a **Name** for the boundary condition called **ROLLER**, under the **Category** option choose **Mechanical**, and choose **Displacement/Rotation** under the **Types for Selected Step** option. The Create Boundary Condition dialog box should look identical to that in Figure 26b.

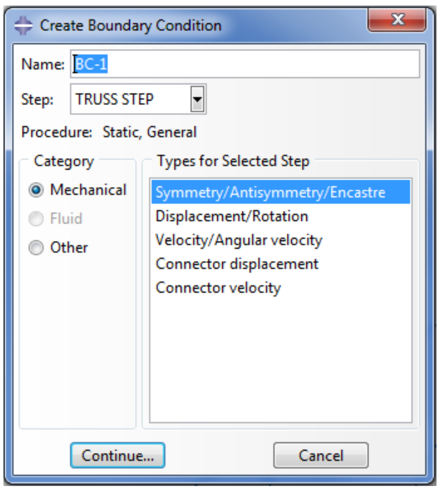


Figure 26a. Create Boundary Condition

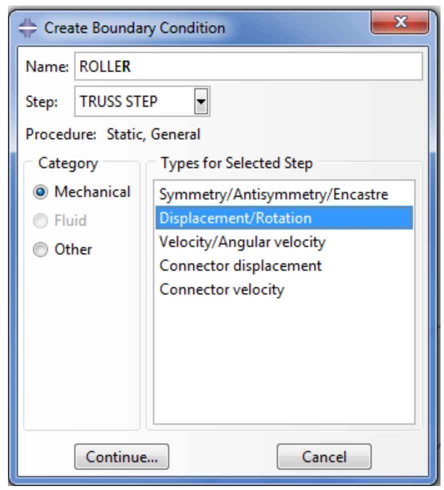


Figure 26b. Create Boundary Condition (ROLLER)

- Click **Continue**…
- All of the available nodes on which the boundary condition can be applied will turn the color yellow. Using the cursor, click bottom left node of the structure. If this selection is done correctly, then the node color will turn from yellow to red (Figure 27).

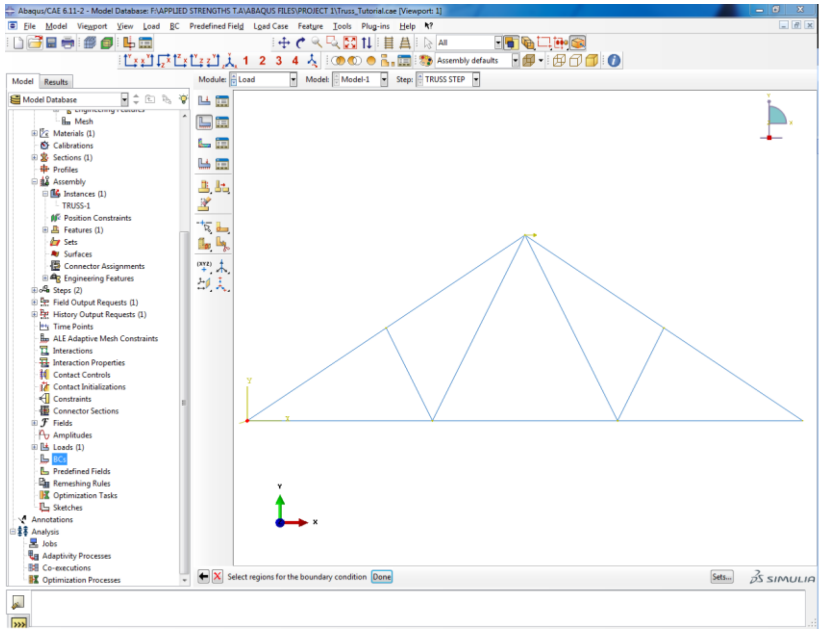


Figure 27. Boundary Condition (Roller)

- Click **Done**.
- The **Edit Boundary Condition** dialog box will immediately appear (Figure 28a). Check the boxes next to the **U2** and the **U3** options. This will constrain the structure in the Y and Z directions but allow movement in the X direction. Note that the default value for the respective displacements is zero. A nonzero displacement BC can be defined if needed.

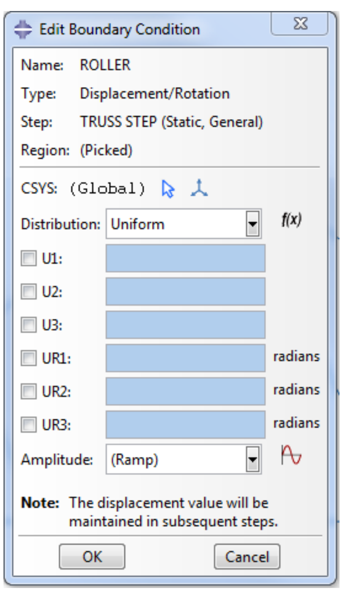


Figure 28a. Edit Boundary Condition

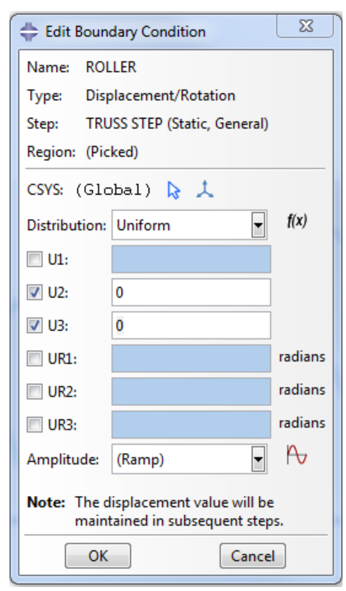


Figure 28b. Edit Boundary Condition (U2 U3)

- Click **OK**.

- Repeat the same procedure to create the **PINNED** constraint boundary condition. However, name this boundary condition **PINNED** and apply the boundary condition to the bottom right node of the structure. Finally in the **Edit Boundary Condition** dialog box, check **U1 U2** and **U3** to constrain the X, Y, and Z directions, respectively.

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 **TRUSS_TUTORIAL**. 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 29.

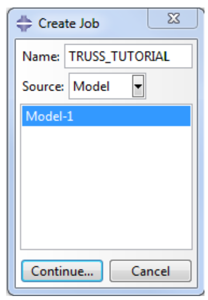


Figure 29. Create Job Dialog Box (TRUSS_TUTORIAL)

- Click **Continue**…
- The **Edit Job** dialog box will immediately appear (Figure 30).

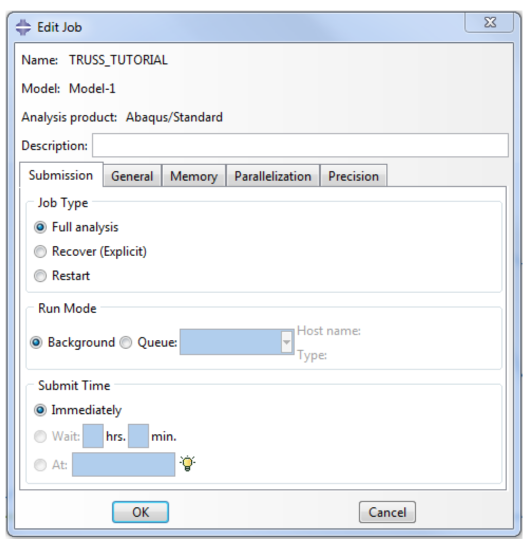


Figure 30. Edit Job Dialog Box

- Accept the default values and click **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 31).

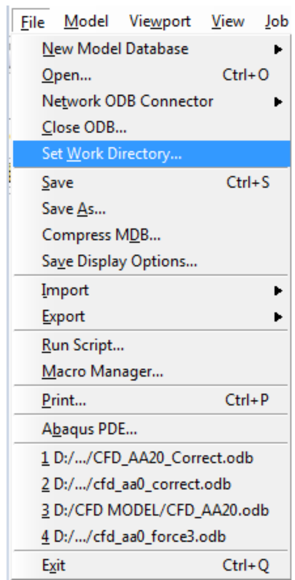


Figure 31. Set Work Directory

- The **Set Work Directory** screen will immediately appear (Figure 32). Click **Select…** and use standard Windows practice to select (and possibly create) a subdirectory.

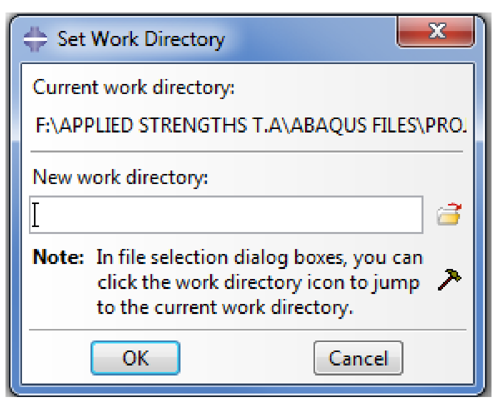


Figure 32. Set Work Directory (FOLDERS)

- 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 the **Jobs(1)** icon in the model tree.
- Right click the job called **TRUSS_TUTORIAL** 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 E:\>). To change directories in the Abaqus Command type the directory of choice followed by a colon **(F:)** 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 APPLIED STRENGTHS T.A**) then hit **Enter**.
- Now that the correct directory has been sourced in the command window type **abaqus inter j=TRUSS_TUTORIAL** and then hit enter.
- If the job has completed successfully the Abaqus prompt should look similar to Figure 33.

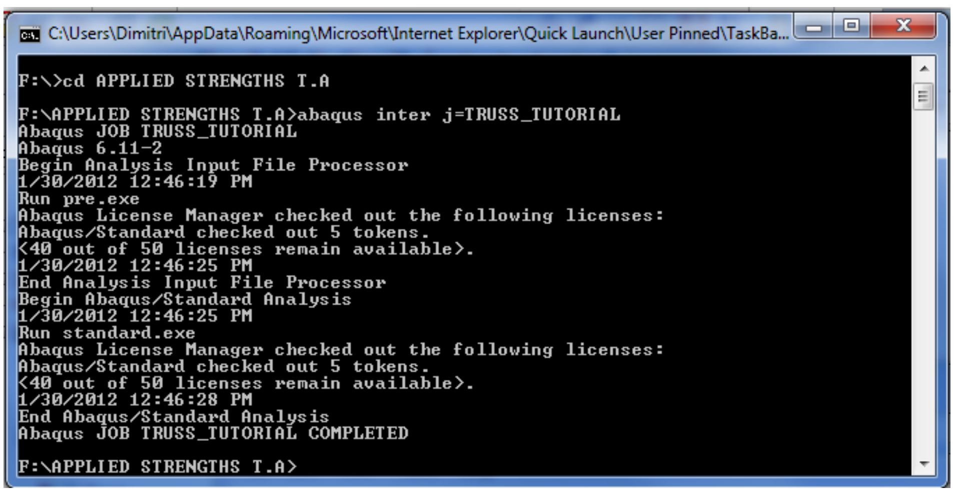


Figure 33. Abaqus Command Prompt (COMPLETED)

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 **TRUSS_TUTORIAL** and click the **Submit** option.
  If you see a warning (Figure 34) Click **OK**. The intent of this warning is to prevent the user from accidentally overwriting a previously completed analysis with the same name.

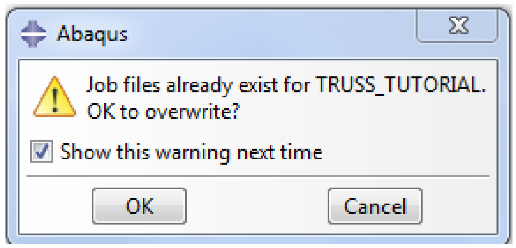


Figure 34. Abaqus Warning

- 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 (Figure 35).

Figure 35. Status Window

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 TRUSS_TUTORIAL job should still be highlighted in Abaqus/CAE model tree. **Right click** the **TRUSS_TUTORIAL** and then click **Results**.
- If this selection was done correctly, the model should turn to a green color and the truss will have rotated to an isometric view (Figure 36).

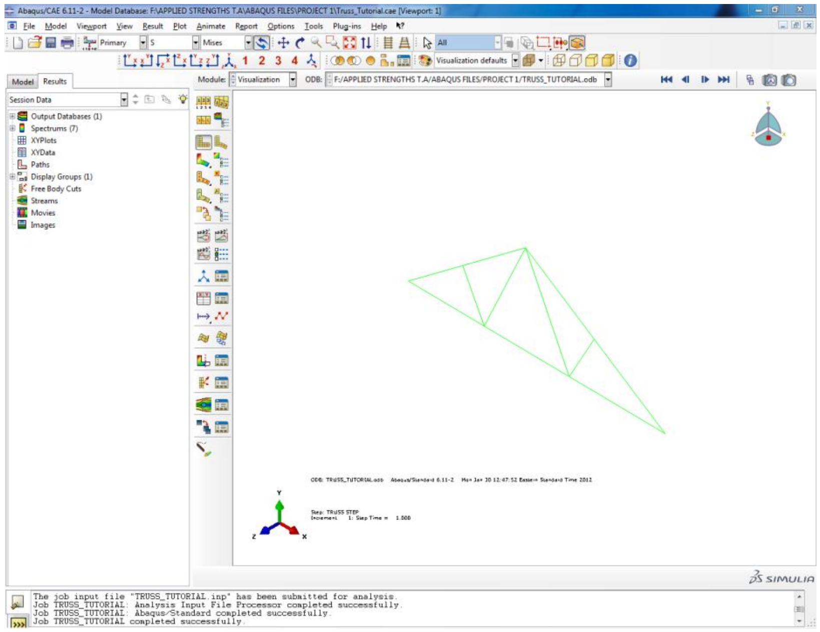


Figure 36. Analysis Results Isometric View

- To rotate the truss 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 37), and the **Apply Front View** button can be clicked to view the model in the X Y plane.

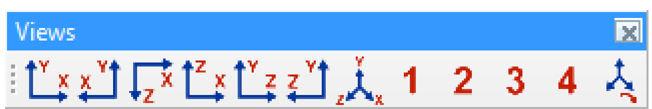


Figure 37. Views Toolbar

- 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 38.

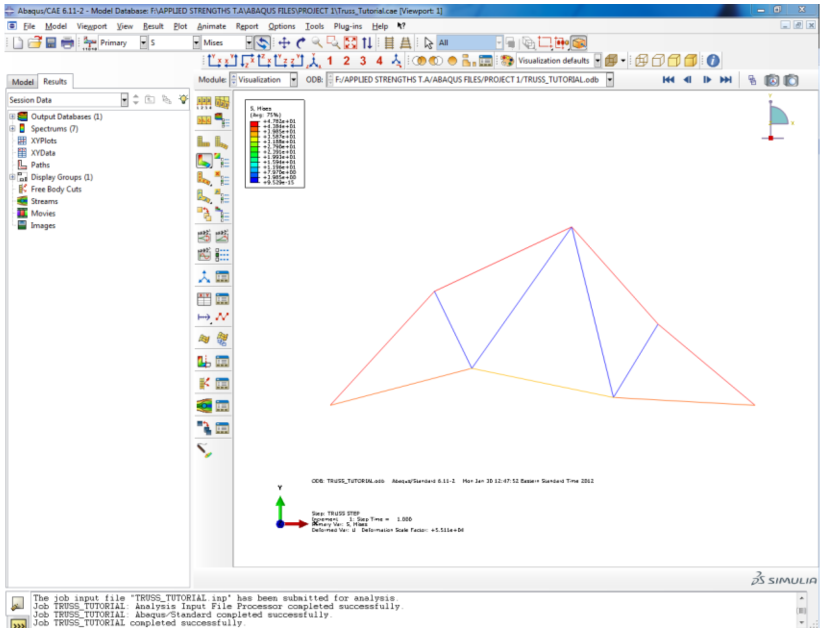
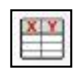


Figure 38. Deformed Shape

Obtaining Stress Values in Elements

- To obtain the stresses in an element click the **Create XY Data** icon  in the **Visualization** module. The **Create XY Data** dialog box will appear (Figure 39a). Choose the **ODB field output option** (Figure 39b).

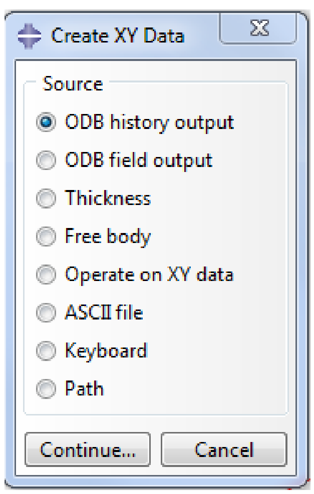


Figure 39a. Create XY Data Dialog Box

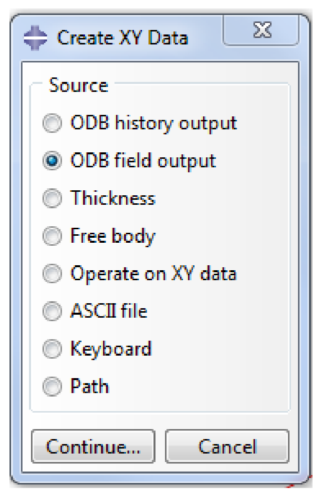


Figure 39b. Create XY Data Dialog Box (FIELD)

- Click **Continue**…
- The **XY Data from ODB Field Output** dialog box will appear (Figure 40). Under the **Variables** tab click the **black** arrowhead next to **S: Stress components**. This click will expand the selection for

more options. Scroll down and check the box next to **S11**. All other stress components should be left unchecked.

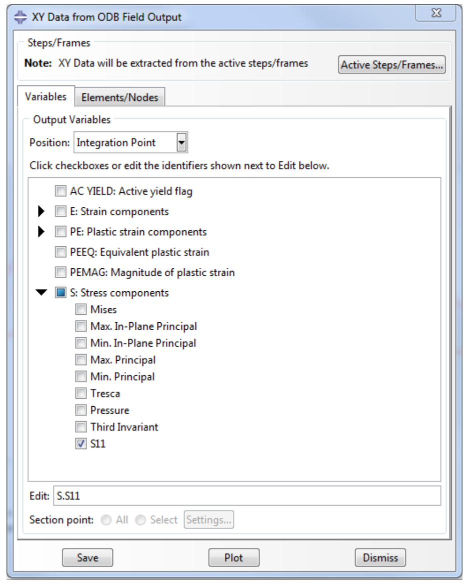


Figure 40. XY Data from ODB Field Output

- Next click the **Elements/Nodes** tab. Make sure that **Pick from viewport** is selected in the **Method** section, and click **Edit Selection**.
- While holding the **Shift** key on the keyboard, click the elements on the model for which the stresses are of interest. When the elements have been selected they will turn a red color.
- Click **Plot**.
- Click **Dismiss**.
- A plot should appear similar to that in Figure 41.

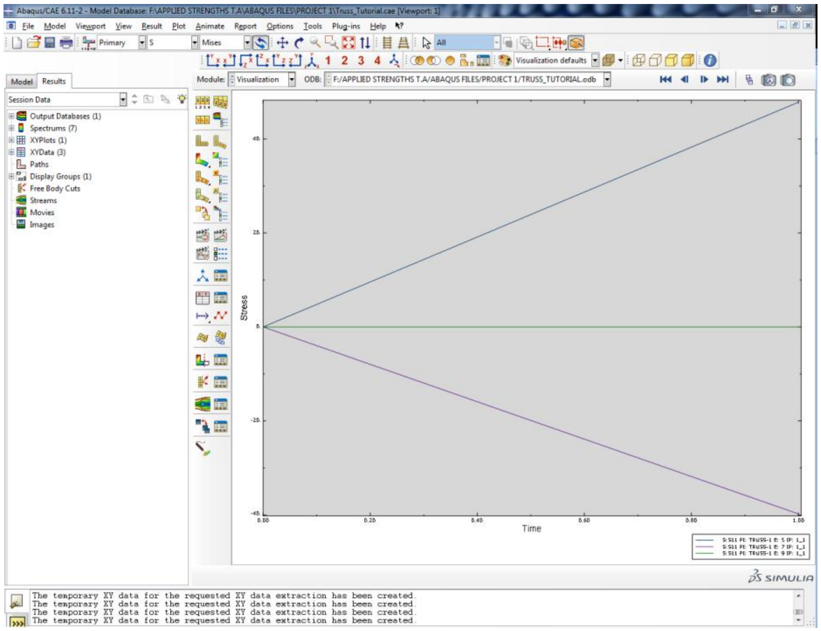


Figure 41. Field Output Plot

- To view the numerical values of the stresses in the elements, 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 42).

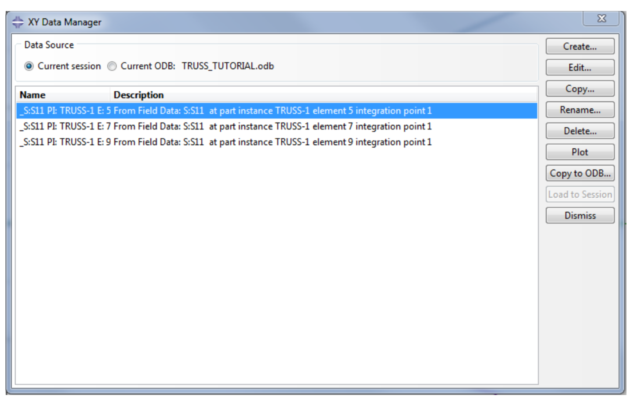


Figure 42. XY Data Manager Dialog Box

- There are stress values for three elements because three were selected in the previous step. To view the value, double click each selection and an **Edit XY Data** dialog box will appear (Figure 43).

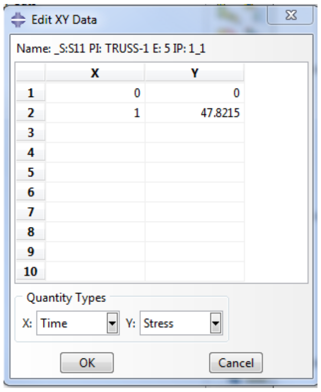


Figure 43. Edit XY Data Dialog Box

- In this dialog box the value of the stress seen by this member is listed. Multiply the stress by the area to get the force in each element.

Conclusion

- Save the file by doing either **File > Save** or clicking the disk icon (Figure 44).

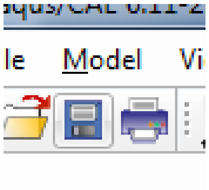


Figure 44. Save File

- Close Abaqus/CAE: **File > Exit** or **Ctrl+Q**
- This completes the Finite Element Analysis of a Planar Truss tutorial.