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.

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.

![Start Session Box](image1.png)

**Figure 1.** Abaqus/CAE 6.11 Start Session 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)

![Viewport](image2.png)

**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](image1)

![Figure 3b. Create Part Dialog Box (Truss)](image2)

• 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>
<thead>
<tr>
<th>Point</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>15</td>
<td>10</td>
</tr>
<tr>
<td>6</td>
<td>30</td>
<td>20</td>
</tr>
<tr>
<td>7</td>
<td>45</td>
<td>10</td>
</tr>
</tbody>
</table>

• 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.
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.
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**.
• 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…

• 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. \( \pi r^2 \)). The Edit section dialog box should look identical to that in Figure 8.

• Click OK.
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).

![Model Tree Expansion (Parts)](image)

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 dragging 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.

![Edit Section Assignment Dialog Box (TRUSS)](image)

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).

![Beam Section Orientation Drop down Menu](image)

**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.

![Beam Section Orientation](image)

**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](image1)
  ![Figure 13b. Global Seeds Dialog Box (20)](image2)

- 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).

![Meshed Geometry](image)

**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.

![Model Tree Expansion (Assembly)](image)

**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).
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).

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

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.
• 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).

• 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.

![Create Boundary Condition](image)

**Figure 26a. Create Boundary Condition**

![Create Boundary Condition (ROLLER)](image)

**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).
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.

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).

![Set Work Directory](image)

**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.

![Set Work Directory](image)

**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](image)

- 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).

```
The job input file "TRUSS_TUTORIAL.inp" has been submitted for analysis.
Job TRUSS_TUTORIAL: Analysis Input File Processor completed successfully.
Job TRUSS_TUTORIAL: Abaqus/Standard completed successfully.
```

![Figure 35. Status Window](image)

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.
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).

  - 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.

![XY Data from ODB Field Output](image)

**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.
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).

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