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.

me: Part-1		
Aodeling S	pace	
) 3D 🔘 2	D Planar	Axisymmetric
ype		Options
Deforma	ble	
Discrete	rigid	
Analytica	al rigid	None available
Eulerian		
ase Feature		
Shape	Туре	
Solid	Extrus	ion
Shell	Revolu	ution
Wire	Sweep	2
O Point		
provimate	size: 200	



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.

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

Table 1. Point Coordinates

- 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.

Abaqus/CAE 6.11-2 [Viewport: 1]				_ 6 ×
Eile Model Viewport View Edit Add	Icols Plug-ins Help N?			_ <i>8</i> ×
i 🗋 📑 🚍 🥽 i 🎒 🚺	🕂 🥐 🔍 🔀 🚺 📋 🚊 🗌	🕞 All 🚽 🖬 🚱	口 m c	
1 [×] x x 1 + x 1 ² x	1 2 3 4 Å		· # 8 6 • • • •	
Model Results	Module: Part Model: Model-1 Part	-		
				× ×
Model Database v v v v v v v v v v v v v v v v v v v	+ ** x2.5, y-15.			
Anders (1) Models (2) Models (2) Models (3) Models (4) Models Models (4) Models (4) Models Models Models (4) Models Model				
	Pick a pointor enter X,Y:			ZS SIMULIA
لم •	-			

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 **<u>E</u>dit** > <u>**Delete**</u> 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.

⇔ Edit Mate	rial					x
Name: Mat	erial-1					
Description:						1
Material B	ehaviors					
General	Mechanical	Thermal	Other			
		_		_	_	
_	OK			Cano	el	

Figure 6a. Edit Material Dialog Box

lame: STEFI		
Description:		د ا
Material Behaviors		
Elastic		
<u>G</u> eneral <u>M</u> echanical	<u>T</u> hermal <u>O</u> ther	
Elastic		
Type: Isotropic		 Suboptions
Use temperature-de	pendent data	
Number of field variable	es: 0 🚔	
Moduli time scale (for y	iscoelasticity): Long-term	
No compression		
No tension		
Data		
Young's	Poisson's	
1 30E06	0.3	

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...



💠 Create See	ction 🛛 🕅
Name: TRUS	S
Category	Туре
Solid	Beam
Shell	Truss
Beam	
Fluid	
Other	
Continu	Cancel

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**.

💠 Edit Section 🛛 🕅
Name: TRUSS Type: Truss
Material: STEEL 🔽
Cross-sectional area: 12.566
Temperature variation: Constant through thickness
OK Cancel

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.

Edit Section Assignment
Region
Region: (Picked)
Section
Section: TRUSS 👻 🗷
Note: List contains only sections applicable to the selected regions.
Type: Truss
Material: STEEL
OK Cancel

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 <u>Assign</u>. Using the left mouse button, click <u>Assign</u> and click <u>Beam Section Orientation</u> (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.

Global Seeds
Sizing Controls
Approximate global size: 2.2
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0)
By absolute value (0.0 < min < global size) 2
OK Apply Defaults Cancel



🖨 Global Seeds 🛛 🔯
Sizing Controls
Approximate global size: 20
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0) 0.1
◎ By absolute value (0.0 < min < global size) 2
OK Apply Defaults Cancel

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.

💠 Element Type		×
Element Library	Family	
Standard	Piezoelectric	•
Geometric Order	Thermal Electric	
🖲 Linear 💿 Quadratic	Truss	-
Line		
Hybrid formulation Element Controls Scaling factors: Linear	bulk viscosity: 1	
T3D2: A 2-node linear 3-	-D truss.	
Note: To select an element select "Mesh->Contr	t shape for meshing, rols" from the main menu bar.	
ОК	Defaults Cancel	

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).

+ Create Instance
Parts
TRUSS
Instance Type
A meshed part has been selected, so the instance type will be Dependent.
Note: To change a Dependent instance's mesh, you must edit its part's mesh.
Auto-offset from other instances
OK Apply Cancel

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).

🐡 Edit Step	X
Name: TRUSS STEP	
Type: Static, General	
Basic Incrementation Other	
Description:	
Time period: 1	
Nlgeom: © On of large displacements and affects subsequent steps.)	
Automatic stabilization: None	
Include adiabatic heating effects	
OK Cancel	



• 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.

🐥 Edit Load	🖶 Edit Load 🛛 🕅
Name: LOAD	Name: LOAD
Type: Concentrated force	Type: Concentrated force
Step: TRUSS STEP (Static, General)	Step: TRUSS STEP (Static, General)
Region: (Picked)	Region: (Picked)
CSYS: (Global) 😓 🙏	CSYS: (Global) 🔈 🙏
Distribution: Uniform f (x)	Distribution: Uniform 💌 f(x)
CF1:	CF1: 1000
CF2:	CF2:
CF3:	CF3:
Amplitude: (Ramp)	Amplitude: (Ramp) 🔽 🏠
Follow nodal rotation	Follow nodal rotation
Note: Force will be applied per node.	Note: Force will be applied per node.
OK Cancel	OK Cancel

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 Bounda	ry Condition
Name: BC-1	
Step: TRUSS ST	EP
Procedure: Static	, General
Category	Types for Selected Step
Mechanical	Symmetry/Antisymmetry/Encastre
Fluid	Displacement/Rotation
Other	Velocity/Angular velocity
	Connector displacement
	Connector velocity
Continue	Cancel





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.

💠 Edit Boun	dary Condition)
Name: ROL	LER	
Type: Disp	lacement/Rotation	
Step: TRU	ISS STEP (Static, General)	
Region: (Pic	ked)	
CSYS: (Glo	bal) 🔓 🙏	
Distribution:	Uniform 💌	f(x)
🔲 U1:		
🔲 U2:		
🔲 U3:		
UR1:		radians
UR2:		radians
UR3:		radians
Amplitude:	(Ramp)	₽v
Note: The d maint	isplacement value will be ained in subsequent step	15.
ОК	Cance	

🖶 Edit Boundary Condition ΣX Name: ROLLER Type: Displacement/Rotation TRUSS STEP (Static, General) Step: Region: (Picked) CSYS: (Global) 😓 🙏 Distribution: Uniform f(x) • 🔲 U1: V U2: 0 V U3: 0 UR1: radians UR2: radians radians UR3: Ð Amplitude: (Ramp) -Note: The displacement value will be maintained in subsequent steps. ОК Cancel

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).

🚔 Edit Job	X
Name: TRUSS_TUTORIAL	
Model: Model-1	
Analysis product: Abaqus/Standard	
Description:	
Submission General Memory Parallelization Precision	
Job Type	
Full analysis	
Recover (Explicit)	
© Kestart	
Run Mode	
Background Queue: Type:	
Submit Time	
Immediately	
🔘 Wait: hrs. min.	
At:	
OK	

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** <u>W</u>ork **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.

💠 Set Work Directory
Current work directory: F:\APPLIED STRENGTHS T.A\ABAQUS FILES\PRO.
New work directory:
Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.
OK Cancel

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.

💠 Abaqus	23
Job files already exist OK to overwrite?	for TRUSS_TUTORIAL.
Show this warning next	time
ОК	Cancel

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).

	a succession of the second s		
E File Model Viewport View Result Plo	t Animate Report Options Iools Plug-ins Help %?		
	x 🔯 2, 1, 2, 3, 4, 3, 1, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0,		
Model Results	Module: Visualization ODB: F/APPLIED STRENGTHS T.A/ABAQUS FILES/PROJECT L/TRUSS_TUTORIAL.odb	× •••	
Section Data I Construction I Constructio I Construction I Construction I Construction I Constru			
			35 SIMULIA
The job input file "TRUSS_TUTO	ORIAL inp" has been submitted for analysis.		*

Figure 36. Analysis Results Isometric View

- To rotate the truss back into the X Y plane for viewing, click <u>View</u> 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).

🜩 Create XY Data 🛛 🛛	💠 Create XY Data 🛛 🖾
Source	Source
ODB history output	ODB history output
ODB field output	ODB field output
Thickness	Thickness
Free body	Free body
Operate on XY data	Operate on XY data
ASCII file	○ ASCII file
Keyboard	C Keyboard
Path	Path
Continue Cancel	Continue Cancel

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.

TY Data from ODB Field Output
Steps/Frames
Note: XY Data will be extracted from the active steps/frames Active Steps/Frames
Variables Elements/Nodes
Output Variables
Position: Integration Point
Click checkboxes or edit the identifiers shown next to Edit below.
AC YIELD: Active yield flag
E: Strain components
E: Plastic strain components
PEEQ: Equivalent plastic strain
PEMAG: Magnitude of plastic strain
▼ ■ S: Stress components
Mises
Max. In-Plane Principal
Min. In-Plane Principal
Max. Principal
Min. Principal
Tresca
Pressure
Third Invariant
<u>▼</u> 511
Edit: S.S11
Section point: All Select Settings
Save Plot Dismiss

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).

	X	Y	
1	0	0	
2	1	47.8215	
3			
4			
5			
6			
7			
8			
9			
10			
Quantit	v Types		
Quantity	y types		

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.