Abaqus/CAE Truss Tutorial (Version 6.12)

Problem Description:

Given the truss structure shown below with pinned supports at the wall and 1kN applied load; solve for displacements of the free node and the reaction forces of the truss structure. The truss material is steel with E = 210 GPa and v = 0.25.

This sample problem is similar to the lecture note example.



Analysis Steps

 Start Abaqus and choose to create a new model database



2. In the model tree double click on the "Parts" node (or right click on "parts" and select Create)



- 3. In the Create Part dialog box (shown above) name the part and
 - a. Select "2D Planar"
 - b. Select "Deformable"
 - c. Select "Wire"
 - d. Set approximate size = 1
 - e. Click "Continue..."



4. Create the geometry shown below (details not discussed here)

5. Double click on the "Materials" node in the model tree



a. Name the new material and give it a description

- b. Click on the "Mechanical" tab \rightarrow Elasticity \rightarrow Elastic
- c. Define Young's Modulus and Poisson's Ratio (use base SI units)
 - i. WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct values are specified
- d. Click "OK"

lame: Stee	l		
escription:	Linear Isotropic Steel (Standard SI units)		1
Material Be	chaviors		
			_
<u>G</u> eneral	Mechanical Thermal Other		%
	Elasticity	<u>E</u> lastic	
		AC .	
	Plasticity	<u>Hyperefastic</u>	
	Plasticity Damage for Ductile Metals	<u>H</u> yperĕlàstic Hyper <u>f</u> oam	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws	<u>H</u> yperēfastic Hyper <u>f</u> oam <u>L</u> ow Density Foam	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites	<u>H</u> yperèl ^a stic Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers	<u>Hyperel¹astic</u> Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers Deformation Plasticity	<u>Hyperellastic</u> Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic Viscoelastic	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers Deformation Plasticity Damping	<u>Hyperellastic</u> Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic <u>V</u> iscoelastic	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers Deformation Plasticity Damping Expansion	<u>Hyperellastic</u> Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic <u>V</u> iscoelastic	
	Plasticity Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites Damage for Elastomers Deformation Plasticity Damping Expansion Brittle Cracking	<u>HyperePastic</u> Hyper <u>f</u> oam Low Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic <u>V</u> iscoelastic	
	Plasticity • Damage for Ductile Metals • Damage for Traction Separation Laws • Damage for Fiber-Reinforced Composites • Damage for Elastomers • Deformation Plasticity • Damping • Expansion • Brittle Cracking • Eos •	<u>HyperePastic</u> Hyper <u>f</u> oam <u>L</u> ow Density Foam Hyp <u>o</u> elastic <u>P</u> orous Elastic <u>V</u> iscoelastic	

Edit	Material				23
lame:	Steel				
escrip	tion: Linear Iso	tropic Steel (Standa	d SI units)		1
Mater	ial Behaviors				
Elastic	:				
<u>G</u> ene	ral <u>M</u> echanic	al <u>T</u> hermal <u>O</u> th	er		*
Elastic					
Type:	Isotropic	-			 Suboptions
🔳 Us	e temperature-	lependent data			
Num	ber of field varia	bles: 0 🚔			
Modu	ıli time scale (fo	r viscoelasticity):	ng-term	-	
No.	compression				
🔳 No	tension				
Data	а				
	Young's Modulus	Poisson's Ratio			

- 6. Double click on the "Sections" node in the model tree
 - a. Name the section "HorizontalBar" and select "Beam" for both the category and "Truss" for the type
 - b. Click "Continue..."
 - c. Select the material created above (Steel)
 - d. Set cross-sectional area = 0.001 (base SI units, m²)
 - e. Click "OK"

Model Results Material Library		
😂 Model Database 💿 🌲 😨		
🖻 🏭 Models (1)	Create Section	
<u>Model-1</u> <u>Parts (1)</u>	Name: Horizontal Bar	💠 Edit Section
Materials (1) Calibrations	Category Type	Name: HorizontalBar
· B Sections	Solid Beam	Type: Truss
🖲 🎎 Assembly	Shell Iruss	
en Gan Steps (1)	Beam	Material: Steel
Held Output Requests	Fluid	Cross-sectional area: 0.001
- ाme Points ≣ - ∰ ALE Adaptive Mesh Constraints	Other	Temperature variation: Constant through thickness
법 Interactions 묩 Interaction Properties	Continue Cancel	OK Cancel
The second second and a second s		

- f. Repeat for the "AngledBar"
 - i. Cross-sectional area=0.00125
- 7. Expand the "Parts" node in the model tree, expand the node of the part just created, and double click on "Section Assignments"
 - a. Select the horizontal portion of the geometry in the viewport
 - b. Click "Done"
 - c. Select the "HorizontalBar" section created above
 - d. Click "OK"

Model Results Material Library	
🚝 Model Database 💽 🌲 🐑	📥 Edit Section Assignment
□	Region
Parts (1) Truss	Region: (Picked)
⊕ Features (1)	Section
- 😻 Surfaces - 🕅 Skins	Section: HorizontalBar 🔽 🏝
- Ø Stringers - ☎ Section Assignments - ‱ Orientations	Note: List contains only sections applicable to the selected regions.
Composite Layups Genering Features Hendering Methyle	Type: Truss Material: Steel
⊕ ∰ Materials (1)	
Calibrations Sections (2)	OK Cancel
Profiles	

e. Repeat for the angled portion of the geometry

- 8. Expand the "Assembly" node in the model tree and then double click on "Instances"
 - a. Select "Dependent" for the instance type
 - b. Click "OK"

Model Results Material Library		
Se Model Database	٠ <mark>``</mark>	Create Instance
□ 🍰 Models (1) in Model-1	<u>^</u>	Parts
🗄 🦫 Parts (1)		Truss
🕀 🖉 Materials (1)		
👏 Calibrations		
🗄 🥸 Sections (2)		
Profiles		
🛱 🏭 Assembly		
- 🗳 Instances		Instance Type
🗰 Position constraints	=	Dependent (mesh on part)
- 💾 Features		Independent (mesh on instance)
🗁 🗁 Sets		Independent (mesh on instance)
🛛 💐 Surfaces		Note: To change a Dependent instance's
Connector Assignments		mesh, you must edit its part's mesh.
🗄 🖷 Engineering Features		Auto-offset from other instances
⊕ o ^t te Steps (1)		
Field Output Requests		OK Apply Cancel
📗 📄 🗮 History Output Requests		

- 9. Double click on the "Steps" node in the model tree
 - a. Name the step, set the procedure to "General", and select "Static, General"
 - b. Click "Continue..."
 - c. Give the step a description
 - d. Click "OK"

Model Results		🜩 Create Step	×
🚝 Model Database	- 🛊 🗈 🗞 🍟	Name: Load	
 ⊟ ﷺ Models (1)		Insert new step after	
□ Model-1		Initial	
🗄 🦺 Parts (1)			
🗉 🔀 Materials	(1)		
👏 Calibratio	ons		
🗉 🤹 Sections	(2)		
Profiles		Procedure type: General	
🗄 🏭 Assembly	/	Dynamic, Explicit	*
⊕o⊶ Steps (1)		Geostatic	
5號 Field Oulz Rea una co	gut Requests	Heat transfer	
History U	output Requests	Mass diffusion	E
	nts E	Static, General	
		Static, Riks	
i i interactio	Ins	L	
	n Properties	Continue	Cancel
	n Properties	Continue	Cancel
	Edit Step	Continue	Cancel
급 Interactio 뮴 Interactio	Edit Step	Continue	Cancel
	Edit Step Name: Load Type: Static, General	Continue	Cancel
	Edit Step Name: Load Type: Static, General Basic Incrementation Other	Continue	Cancel
	Edit Step	Continue	Cancel
	Edit Step	Continue	Cancel
	Edit Step Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 Off (This setting controls the)	continue	Cancel
	For perties Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 Nigeom: On of large displacements a	e inclusion of nonlinear effects and affects subsequent steps.)	Cancel
	For perties Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 NIgeom: Off (This setting controls the On of large displacements a Automatic stabilization: None	e inclusion of nonlinear effects and affects subsequent steps.)	Cancel
	For Properties Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 NIgeom: Off (This setting controls the On of large displacements a Automatic stabilization: None	continue e inclusion of nonlinear effects and affects subsequent steps.)	Cancel
	Edit Step Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 NIgeom: Off (This setting controls the On of large displacements a Automatic stabilization: None Include adiabatic heating effects	e inclusion of nonlinear effects and affects subsequent steps.)	Cancel 23
	Edit Step Edit Step Name: Load Type: Static, General Basic Incrementation Other Description: apply point load Time period: 1 NIgeom: Off (This setting controls the On of large displacements a Automatic stabilization: None Include adiabatic heating effects	e inclusion of nonlinear effects and affects subsequent steps.)	Cancel

- 10. Expand the Field Output Requests node in the model tree, and then double click on F-Output-1 (F-
 - Output-1 was automatically generated when creating the step)
 - a. Uncheck the variables "Strains" and "Contact"
 - b. Click "OK"

(🖶 Edit Field Output Request
	Name: F-Output-1
	Step: Load
	Procedure: Static, General
	Domain: Whole model
	Frequency: Every n increments n: 1
	Timing: Output at exact times
	Output Variables
	Select from list below ○ Preselected defaults ○ All ○ Edit variables
	S, U, RF, CF
Model Results	
🗃 Model Database 🛛 🚽 🌲 😨	Stresses
a 🛔 Models (1)	Strains
⊟ Model-1	Displacement/Velocity/Acceleration
🗄 🕒 Parts (1)	Forces/Reactions
🗉 🖉 Materials (1)	Contact
Calibrations	
🗄 🧱 Sections (2)	Energy
Profiles	Failure/Fracture
🗄 🏙 Assembly	Thermal 🔻
	4 III •
. Initial	Note: Some error indicators are not available when Domain is Whole Model or Interacti
Field Output Requests (1)	Output for rebar
The Provide the Present (1)	Output at shell, beam, and layered section points:
Image Time Points	Use defaults Specify:
Hand I F Adaptive Mesh Constraints	
	Include local coordinate directions when available
Hinteraction Properties	OK
- 10 Contact Controls	Caricel

11. Expand the History Output Requests node in the model tree, and then right click on H-Output-1 (H-Output-1 was automatically generated when creating the step) and select Delete

ا الله Assembly ا م⊈ Steps (2)	y	
H 👷 Field Out	put Requests (1)	
🗏 📆 History C	output Requests (1)	
🗄 H-Output		Ξ
📇 Time 🗐	Switch Context Ctrl+Space	
🛄 ALE A	Edit	
🗄 🚹 Interac	Сору	
🗄 Interac	Rename	
🙀 Conta	Delete Del	
👔 Conta	Suppress	
🖳 Consti	Resume	
- 🔁 Conne	Set As Root	
${f \pm}~{m {\cal F}}~$ Fields	Expand All Under	
🕂 Ampli	Collapse All Under	
🕒 🗠 Loads	•	
🖳 📙 BCs		
- Part - Lago	المرجعين مستعمي الريعة فقيد	

- 12. Double click on the "BCs" node in the model tree
 - a. Name the boundary conditioned "Pinned" and select "Displacement/Rotation" for the type
 - b. Click "Continue..."
 - c. Select the endpoints on the left ("shift" select) and press "Done" in the prompt area
 - d. Check the U1 and U2 displacements and set them to 0
 - e. Click "OK"



- 13. Double click on the "Loads" node in the model tree
 - a. Name the load "PointLoad" and select "Concentrated force" as the type
 - b. Click "Continue..."
 - c. Select the vertex on the right and press "Done" in the prompt area
 - d. Specify CF2 = -1000
 - e. Click "OK"



- 14. In the model tree double click on "Mesh" for the Truss part, and in the toolbox area click on the "Assign Element Type" icon
 - a. Select "Standard" for element type
 - b. Select "Linear" for geometric order
 - c. Select "Truss" for family
 - d. Note that the name of the element (B21) and its description are given below the element controls
 - e. Click "OK"

Model Results	
🝯 Model Database 🚽 🌲 住 🗞 🍟	Module: 🖨 Mesh 🔻 Modi
 Models (1) Model-1 Parts (1) Truss Features (1) Sets Surfaces Skins Stringers Section Assignments (2) Orientations Composite Layups Engineering Features Mesh (Empty) Calibrations Sections (2) 	Assign Element Type
Element Type	X
Element Library Standard C Explicit Geometric Order Linear C Quadratic Family Piezoelectric Pipe Thermal Electric Truss	Ē
Line Hybrid formulation Element Controls Scaling factors: Linear bulk viscosity: 1	
T2D2: A 2-node linear 2-D truss.	
Note: To select an element shape for meshing, select "Mesh-> Controls" from the main menu bar.	
OK Defaults	Cancel

- 15. In the toolbox area click on the "Seed Edges" icon
 - a. Select the entire geometry
 - b. Choose method as "By number"
 - c. Define the number of elements along the edges as "1"
 - d. Click "OK"

	💠 Local Seeds
Module: Mesh Mod	Basic Constraints Method Bias By size Image: Single in the
Seed Edges	Sizing Controls Number of elements: 1
	Set Creation Create set with name: Edge Seeds-1
<u></u>	OK Apply Defaults Cancel

16. In the toolbox area click on the "Mesh Part" icon

a. Click "Yes" in the prompt area



OK to mesh the part? Yes No

17. In the menu bar select View → Part Display Options

- a. On the Mesh tab check "Show node labels" and "Show element labels"
- b. Click "OK"

ewport	View Seed Mesh	Adaptivity	Featur
	<u>S</u> ave		
	Pa <u>n</u> <u>R</u> otate <u>Z</u> oom In/Out <u>B</u> ox Zoom	F2 F3 F4 dule F5	: • Me
	Auto- <u>F</u> it <u>C</u> ycle Views Sp <u>e</u> cify Parallel	F6 F7	
atures (1) ts	Show <u>M</u> odel Tree Ct Toolbars	trl+T	1 44 4 4
ins ingers	View Options Graphics Options		
ingers ction Ass ientation	Light Options		
mposite gineering	Part Display Options. Features		

- 18. In the model tree double click on the "Job" node
 - a. Name the job "Truss"
 - b. Click "Continue..."
 - c. Give the job a description
 - d. Click "OK"

🜩 Create Job	
Name: Truss	
Source: Model	
Bes (1) B	
Analysis	
Real Adaptivity Processes	4
Continue Cancel	
C Detimization Processes	_
₩ Edit Job	
Name: Truss	
Model: Model-1	
Analysis product: Abaqus/Standard	
Description: Static analysis of a truss with a point load	
Submission General Memory Parallelization Precision	
_ Job Type	
Full analysis	
Recover (Explicit)	
Restart	
Run Mode	
Background Queue: Host name: Type:	
Submit Time	
Immediately	
🔿 Wait: hrs. min.	
○ At:	
OK Cancel	

- 19. In the model tree right click on the job just created (Truss) and select "Submit
 - a. While Abaqus is solving the problem right click on the job submitted (Truss), and select "Monitor"

Image: Free Amplitie Image: Free Amplitie <th>tudes (1)) witch Context Ctrl+Space dit opy ename elete Del //rite Input ata Check ubmit ontinue</th> <th>E</th> <th></th> <th>s (1) Switch Context Ctrl+Sp Edit Copy Rename Delete Write Input Data Check Submit Continue Monitor</th> <th>ace Del</th>	tudes (1)) witch Context Ctrl+Space dit opy ename elete Del //rite Input ata Check ubmit ontinue	E		s (1) Switch Context Ctrl+Sp Edit Copy Rename Delete Write Input Data Check Submit Continue Monitor	ace Del
The R 2 el K 2 el F The E	ionitor esults ill xport	n creat ted on ted on eated.	Job Tr Job Tr Job Tr Job Tr	Results Kill Export	" ha: e Pri mple 1ly.

- b. In the Monitor window check that there are no errors or warnings
 - i. If there are errors, investigate the cause(s) before resolving
 - ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	1	1	0	1	1	1	1	1
.og I	Frrors Warning	gs Out	put Data F	ile Messa	age File S	tatus File		
Compl Compl	eted: Abaqus/St eted: Tue Oct 04	andard 11:26:24	2011					
Search	Text			Ma	tch case	l Nevt û Pre	evious	

20. In the model tree right click on the submitted and successfully completed job (Truss), and select "Results"



- 21. In the menu bar click on Viewport → Viewport Annotations Options
 - a. Uncheck the "Show compass option"
 - b. The locations of viewport items can be specified on the corresponding tab in the Viewport Annotations Options
 - c. Click "OK"

vport View Result	Plot	Animate	Rep	General	Iriad	Legend	Title Block	State
Create			F	Visibility	/			
Next		Ctrl+Tab	-3	Show	compa	SS		
Previous	Shift	+Ctrl+Tab		Show	triad			
Cascade			:	Show	legend			
Tile Horizontally				Show	title blo	ock		
Tile Vertically			3 1	Show	state bl	ock		
Delete Current			114	Show	text and	d arrows		
Annotation Manager			- 1	Set all o	n Set a	all off		
Create Annotation			14 m					
Edit Annotations			11					
Viewport Annotation O	ptions	N						
Linked Viewports		43	11					
1 Viewport: 1 ODB: C:	/Temn/	Trussodh	- 3					

- 22. Display the deformed contour of the (Von) Mises stress overlaid with the undeformed geometry
 - a. In Abaqus 6.11 the default display field output is (Von) Mises stress
 - b. In the toolbox area click on the following icons
 - i. "Plot Contours on Deformed Shape"
 - ii. "Allow Multiple Plot States"
 - iii. "Plot Undeformed Shape"

Fall 2012



- 23. In the toolbox area click on the "Common Plot Options" icon
 - a. Note that the Deformation Scale Factor can be set on the "Basic" tab
 - b. On the "Labels" tab check "Show element labels", "Show node labels", and "Show node symbols"
 - i. *Note* the default label colors may need to be adjusted for visibility. They can be changed in this option box by clicking on the color and picking a new one.
 - c. Click "OK"



ME 455/555 Intro to Finite Element Analysis



- 24. To determine the local stress values use the probe feature.
 - a. This can be found in two locations
 - i. In the menu bar click Tools \rightarrow Query \rightarrow Probe Values
 - ii. In the vertical icon tool bar



- b. In the viewport mouse over the element of interest
- c. Click on an element to store it in the "Selected Probe Values" portion of the dialogue box
- d. Note that Abaqus reports stress values from the integration points, which may differ slightly from the values determined by projecting values from the surrounding integration points to the nodes
 - i. The minimum and maximum stress values contained in the legend are from the stresses projected to the nodes
 - ii. The probe can be set to query either the elements or nodes.
- e. Click "Cancel" and there is no need to save the queried points at this time.

1, Load			Frame: 1		
output variable	e for Probe: S ,	, Mises (N	ot averaged)		
lues					
from viewport	🔘 Key-in lab	oel 🔘 Sel	ect a display group		
· · · ·				·· - D1	
lements 💌 C	omponents:	Selected	Position: Int	egration Pt	
Attached node	25:				
Part Instance	Element ID	Туре	Attached nodes	S, Mises	
RUSS-1	2	T2D2		1.44222e+06	
		T2D2		15++06	
rruss-1	1	1202		1.56+00	
rruss-1	1	1202		1.52+00	
rruss-1	1	1202		1.50+00	
	output variable lues from viewport lements v C Attached node 'art Instance	output variable for Probe: S, lues from viewport Key-in lat lements Components: Attached nodes: Part Instance Element ID	output variable for Probe: S, Mises (N lues from viewport () Key-in label () Sel lements () Components: Selected Attached nodes: Part Instance Element ID Type	output variable for Probe: S, Mises (Not averaged) lues from viewport Key-in label Select a display group lements Components: Selected Position: Int Attached nodes: Part Instance Element ID Type Attached nodes	output variable for Probe: S, Mises (Not averaged) lues from viewport Key-in label Select a display group lements Components: Selected Position: Integration Pt Attached nodes: Attached nodes: Attached nodes Attach

- 25. To change the displayed output, either:
 - a. Change the field output using the tool bar option shortcut



- b. Use the menu bar
 - Click on Results → Field
 Output
 - ii. Select "U Spatial displacement at nodes"
 - iii. Component = U2
 - iv. Click "OK"



rimary Variable	Deformed Variable	Symbol Variable	Status Variable	Stream Variable
Output Variable	bles with results:		•	
Name	Description (* indic	ates complex)		
CF	Point loads at no	des		
RF	Reaction force at	nodes		
S	Stress componer	nts at integration po	oints	
U	Spatial displacen	nent at nodes		
U	Spatial displacen	nent at nodes		
U	Spatial displacen	c Componer	nt	
U Invariant Magnitude	Spatial displacen	Componer	nt	
U Invariant Magnitude	Spatial displacen	Componer U1 U2	nt	

26. To create a text file containing the stresses, vertical

displacements, and reaction forces (including the total), in the menu bar click on Report → Field Output

- a. For the output variable select (Von) Mises
- b. On the Setup tab specify the name and the location for the text file
- c. Uncheck the "Column totals" option
- d. Click "Apply"



Report Field Output	Report Field Output
Step/Frame	Step/Frame
Step: 1, Load	Step: 1, Load
Frame: 1 Step/Frame	Frame: 1 Step/Frame
Variable Setup	Variable Setup
Output Variables	File
Position: Integration Point	Name: Truss_output.rpt Select
Click checkboxes or edit the identifiers shown next to Edit below.	Append to file
▼ ■ S: Stress components	Output Format
Max. In-Plane Principal	Layout: Single table for all field output variables
Min. In-Plane Principal	Separate table for each field output variable
Max. Principal	Sort by: Element Label 🔻
Min. Principal	Ascending Descending
Pressure	Page width (characters): No limit Specify: 80
Third Invariant	Number of significant digits: 6
	Number format: Engineering 💌
Edit: S.Mises	Data
Section point: O All O Select Settings	Write: 🖉 Field output 🔲 Column totals 📝 Column min/max
OK Apply Defaults Cancel	OK Apply Defaults Cancel

- e. Back on the Variable tab change the position to "Unique Nodal"
- f. Uncheck the stress variable, and select the U2 spatial displacement
- g. Click "Apply"

Fall 2012

🐥 Report Field Output
Step/Frame
Step: 1, Load
Frame: 1 Step/Frame
Variable Setup
Output Variables
Position: Unique Nodal 👻
Click checkboxes or edit the identifiers shown next to Edit below.
CF: Point loads
RF: Reaction force
S: Stress components
▼ ■ U: Spatial displacement
Magnitude
Edit: U.U2
Section point: O All O Select Settings
OK Apply Defaults Cancel

- h. On the Variable tab, uncheck Spatial displacement and select the RF2 reaction force
- i. On the Setup tab, check the "Column totals" option
- j. Click "OK"

Report Field Output	Report Field Output
Step/Frame Step: 1, Load Frame: 1 Step/Frame Variable Setup	Step/Frame Step: 1, Load Frame: 1 Step/Frame Variable Setup
Output Variables Position: Unique Nodal Click checkboxes or edit the identifiers shown next to Edit below.	File Name: Truss_output.rpt Select Append to file
 CF: Point loads RF: Reaction force Magnitude RF1 RF2 S: Stress components U: Spatial displacement 	 Output Format Layout: Single table for all field output variables Separate table for each field output variable Sort by: Node Label Ascending
Edit: RF.RF2 Section point: All Select Settings OK Apply Defaults Cancel	Data Write: Image: Field output OK Apply Defaults Cancel

- 27. Open the .rpt file with any text editor (such as Notepad)
 - a. One thing to check is that the total downward reaction force is equal to the applied load (1,000 N)
 - b. Each time "Apply" is pressed the .rpt file is updated as long as the "Append to file" option is checked under "Setup"

Truss_output.rpt - Notepad	
File Edit Format View Hel	p
Field Output Report, wr	itten Tue Oct 04 16:57:45 2011
Source 1	
ODB: C:/Temp/Truss.c Step: Load Frame: Increment	odb 1: Step Time = 1.000
LOC 1 : Nodal values fr	om source 1
Output sorted by columr	n "Node Label".
Field Output reported a	at nodes for part: TRUSS-1
Node Label	RF.RF2 @Loc 1
1 2 3	-0. 0. 1.E+03
Minimum	0.
At Node Maximum	2 1.E+03
At Node	3
بالاستحدين بالمعج ريسم م	أكروسية المجاجرة والمتوسور والمتحاصية في والاراد والمحاص والمسارين