Abaqus Plane Stress Tutorial (ver. 6.12) using CST element

Problem Description



The steel bracket is fixed to a wall along its left side. A tensile pressure force with a total magnitude of 5000 N is applied to the right side of the bracket. The bracket contains one line of symmetry, so only half of the geometry is to be modeled. Determine the stresses in the bracket.

Analysis Steps

- 1. 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 select
 - a. 2D Planar
 - b. Deformable
 - c. Shell
 - d. Approximate size = 2
- 4. Create the geometry shown below (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→Elasticity→Elastic
- c. Define Young's Modulus and the Poisson's Ratio (use 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

Materi	ial		×
Steel			
tion: Lir	near Isotropic Steel (SI units)		
rial Be	haviors		
ral M	Aechanical Thermal Other		Delete
ral M	1echanical Ihermal Qther Elasticity ▶	Elastic	Delete
eral M	<u>fechanical</u> <u>I</u> hermal <u>Q</u> ther Elasticity ▶ Plasticity ▶	Elastic Hyperelastic	Delete
ral M	<u>fechanical</u> <u>I</u> hermal <u>Q</u> ther Elasticity Plasticity Damage for Ductile Metals ►	Elastic Hyperelastic Hyperfoam	Delete
ral M	<u>Pechanical Ihermal Other</u> Elasticity ► Plasticity ► Damage for Ductile Metals ► Damage for Traction Separation Laws ►	Elastic Hyperelastic Hyperfoam Hypgelastic	Delete
ral M	<u>Pechanical</u> <u>Ihermal</u> <u>Other</u> <u>Elasticity</u> ▶ Plasticity ▶ Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites ▶	Elastic Hyperelastic Hyperfoam Hypgelastic Porous Elastic	Delete
ral M	<u>Pechanical</u> <u>Ihermal</u> <u>Other</u> <u>Elasticity</u> ▶ Plasticity ▶ Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites ▶ Deformation Plasticity	Elastic Hyperelastic Hypefoam Hypgelastic Porous Elastic Viscoelastic	Delete
ral M	Intermal Intermal Intermal Other Intermal Intermal Intermal Inte	Elastic Hyperelastic Hypefoam Hypgelastic Porous Elastic Viscoelastic	Delete

- cuit	material					
Name:	Steel					
Descrip	tion:					
- Mate	ial Behaviors					
Elasti						
Elastic						
<u>G</u> ene	eral <u>M</u> echanica	l <u>T</u> hermal	<u>O</u> ther			
Elasti	:					
Type:	Isotropic	-				
🔲 Us	e temperature-de	ependent data				
Num	ber of field variab	les: 0				
Modu	ıli time scale (for	viscoelasticity): Long	term	T	
No	o compression					
No.	tension					
Dat	a					
	Young's Modulus	Poisson's Ratio	5			
1	210e9	0.2 9				
	Ok					Cancel

- 6. Double click on the "Sections" node in the model tree
 - a. Name the section "PlaneStressProperties" and select "Solid" for the category and "Homogeneous" for the type
 - b. Select the material created above (Steel) and set the thickness to 0.01

Ed Name Descr

Model Results
Se Model Database 💽 🖨 🗞 🍟
두 🍰 Models (1)
🖻 Model-1
🖻 🦫 Parts (1)
🕀 🎦 Materials (1)
- 🟂 Sections
Profiles
Assemb
± ⊶ Geps (1)
Barner aut Bearing

Create Section				
Name: plan	Name: plane_stress_tutoria			
Category	Туре			
Solid	Homogeneous			
Shell	Generalized plane strain Eulerian Composite			
🔘 Beam				
Fluid				
◎ Other				
Continue Cancel				

- Edit Section
Name: pl_stress_tutorial
Type: Solid, Homogeneous
Material: Steel
Plane stress/strain thickness: 0.01
OK Cancel

- 7. Click on the "Assign Section" icon
 - a. Select the surface geometry in the viewport
 - b. Be sure the section created above (plane_stress_tutorial) is selected.

— Edit Se	ction Assignment		x
Region			
Region:	(Picked)		
Section			
Section:	pl_stress_tutorial	• 1	
Note: Li a	ist contains only section pplicable to the selected	s I regions.	
Туре:	Solid, Homogeneous		
Material:	Steel		
Thickne	ss		
Assignm	ent: () From section (🗇 From geom	netry
	ОК	Cancel	



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



- 9. In the model tree, under the expanded "Assembly" node, double click on "Sets"
 - a. Name the set "Fixed"
 - b. Select the left edge of the surface in the viewport

X



- c. Create another set named "Symmetry"
- d. Select the two lower horizontal edges of the surface in the viewport

10. In the model tree, under the expanded "Assembly" node, double click on "Surfaces"

- a. Name the surface "PressureLoad"
- b. Select the right edge of the surface in the viewport



- 11. Double click on the "Steps" node in the model tree
 - a. Name the step, set the procedure to "General", and select "Static, General"
 - b. Give the step a description

Nar Ins Model Results	Create Step X me: Apply Load sert new step after Itial	
Model Database Image: Constraint of the second	ocedure type: General ynamic, Explicit ynamic, Temp-disp, Explicit eostatic eat transfer ass diffusion oils a stic, General ratic, Riks	Edit Step X Name: Apply Load Type: Static, General Basic Incrementation Other Description: Apply pressure load to bracket Time period: I Nigeom: © Off (This setting controls the inclusion of nonlinear effects Nigeom: © On On of large displacements and affects subsequent steps.) Automatic stabilization: None

- 12. 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"

	Edit Field Output Request	×
	Name: F-Output-1	
	Step: Apply Load	
	Procedure: Static, General	
	Domain: Whole model	
	Frequency: Every n increments n: 1	
	Timing: Output at exact times	
	Output Variables	
	⑦ Select from list below ○ Preselected defaults ○ All ○ Edit variables	
	S,U,RF,CF	
Model Results Model Database 도 한 한 한 한	 Stresses Strains Displacement/Velocity/Acceleration Forces/Reactions Contact 	•
P 👪 Models (1)	Energy	
	Failure/Fracture	
田 Se Materials (1)	Thermal	<u>ا</u> لئے :
Sections (1)	Note: Error indicators are not available when Domain is Whole Model or Interaction	-
Profiles		
🕀 🎎 Assembly	Output for rebar	
H 04 Steps (2)	Output at shell, beam, and layered section points:	
	Use defaults O Specify:	
History Witput Requests (1)	Include local coordinate directions when available	
Time Pd F-Output-1	OK Cancel	

13. 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



- 14. Double click on the "BCs" node in the model tree
 - a. Name the boundary conditioned "Fixed" and select "Displacement/Rotation" for the type
 - b. In the prompt area click on the Sets button
 - c. Select the set named "Fixed.
 - d. Check U1 and U2 to fully restrain the left edge..

		ry Condition
n	Name: Fixed	
- E	Step: Step-1	T
- 2-	Procedure: Static,	General
	Category	Types for Selected Step
	Mechanical	Symmetry/Antisymmetry/Encastre
- B	Fluid	Displacement/Rotation
l _ ⊑	Other	Velocity/Angular velocity
Annota		Connector displacement
E 🕹 Anaiys		Connector velocity
Ada		
	Continue	Cancel

← X Select regions for the boundary condition 🔛 Done Sets.

- Region Selection		
Eligible Sets		
Sets below may contain vertices, edges, faces	s, cells or nodes.	
Name filter:		
Name	Туре	
Fixed	Geometry	
symmetry	Geometry	
Highlight selections in viewport		
Continue	Dismiss	

🔶 Edit Boun	dary Condition	
Name: Fixe	d	
Type: Disp	placement/Rotation	
Step: Step	o-1 (Static, General)	
Region: (Pic	ked)	
CSYS: (Glo	obal) 🔓 🙏	
Distribution:	Uniform 👻	f(x)
V 1:	0	
U 2:	D	
UR3:		radians
Amplitude:	(Ramp)	۴v
Note: The displacement value will be maintained in subsequent steps.		
OK Cancel		

e. Repeat the procedure for the symmetry restraint using the set named "Symmetry", check U2 for the boundary condition.

- 15. Double click on the "Loads" node in the model tree
 - a. Name the load "Pressure" and select "Pressure" as the type
 - b. Select surface named "Pressure"
 - c. For the magnitude enter -5e6



i. Note that because we have been using standard SI units the load applied is $-5x10^6$ N/m² which is

a total of -2500 N distributed across the right edge of the surface $\left(\frac{-2500N}{(0.05m)(0.01m)}\right)$

- Region Selection	×
Eligible Surfaces	
Surfaces below may contain faces.	
Name filter:	
Name	Туре
Pressure	Surface
Highlight selections in viewport Continue	Dismiss
Amplitude: (Ramp)	
OK Cancel	Your model, up

Create Load	×
Name: Pressure	
Step: Apply Load	v
Procedure: Static, G	eneral
Category	Types for Selected Step
Mechanical	Concentrated force
C Thermal	Moment
C Acoustic	Pressure
C Eluid	Shell edge load
C Electrical	Surface traction
	Pipe pressure
Mass diffusion	Body force
C Other	Gravitu
	Bolt load
Continue	Cancel

our model, upon application of BC and load should look similar to the figure below.



- 16. In the model tree double click on "Mesh" for the Bracket 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 "Plane Stress" for family
 - d. Pick the tab "Tri" and note the name of the CST element "CPS3" in the description window!

	1	Element Type	· Patta		
		Element Library Standard Explicit	Family Plane Strain		•
		Geometric Order	Pore Fluid/Stress Thermal Electric		E
Model Results		Quad Tri			
Models (1) Model-1 Parts (1) Bracket Features (1) Sets Surfaces Skins Comparison Skins S	L. 6.0	Element Controls Viscosity: Second-order accuracy: Distortion control: Element deletion:	 Use default Specify Yes No Use default Yes No Length ratio: 0.1 Use default Yes No 		A E
Scringers Section Assignments (1) Section Assignments (1) Section Assignments (1) Sections (1) Sections (1) Sections (1) Sections (1)	Assign Element Type	Note: To select an element select "Mesh->Contro	shape for meshing, ols" from the main menu bar. Defaults	Cancel]

- 17. In the toolbox area click on the "Assign Mesh Controls" icon
 - a. Modify the element shape "Tri" and click OK.

Assign 1esh Controls

> Mesh Controls	
Element Shape	
🔘 Quad 🔘 Quad	-dominated 🔍 Tri
Technique	Algorithm
🔘 As is	Use mapped meshing where appropriate
Free	
Structured	
🔘 Sweep	
Multiple	
ОК	Defaults Cancel

- 18. In the toolbox area click on the "Seed Part" icon
 - a. Set the approximate global size to 0.02

€න් Part

🕂 Global Seeds 📃 💌
Sizing Controls
Approximate global size: 0.02
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) 0.0035
OK Apply Defaults Cancel

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

E.	
Det m	ШY
Mes	h Part
Mesl	h Part

- 20. In the model tree double click on the "Job" node
 - a. Name the job "Bracket"
 - b. Give the job a description

		Edit Job	×
		Name: Bracket	
		Model: Model-1	
		Description: Static analysis of a bracket	
		Submission General Memory Parallelization Precision	
		Job Type	
		Full analysis	
		C Recover (Explicit)	
	🗌 Create Job 🛛 🗶	C Restart	
	Name: Bracket	└── Run Mode ─────	
	Source: Model	Background C Queue: Host name: Type:	
Predefined Fields		Submit Time	
Remeshing Rules		• Immediately	
L Sketches		O ∀ait; hrs. min.	
Annotations			
Add Jobs	Continue Cancel	OK Cancel	

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

Am E Loz E BC C Am B C E Job C Amote C Am	plitudes ads (1) s (2) Switch Context Ctrl+Space Edit Copy Rename Delete Del Write Input Data Check			Connector settions ields implitudes .oads (1) GS (2) Switch Context Ctrl+Space Edit Copy Rename Delete Delet Write Input Data Check	
▲	Submit		Ⅰ	Submit	н
Image: A mean of the second se	Continue K Monitor Results Kill	b e e	The The Job Job Job	Continue Monitor Results Kill	ee ge at ar ce

- 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

📑 Brack	Bracket Monitor							
Job: Brad	:ket Status: (Completed						
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
L								
L								
		,						
Log E	rrors Warning	s Outpu	it					
Complet	ed: Abeque/Ster	vdərd						
Complet	ea, moaqusyotai	luaru						
Complet	ed: Thu Jul 24 1	4:17:31 20)08					•
	[Kill				Di	smiss	

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

	Amplitudes		`11		
🕀 陆 Loads (1)					
∎	BCs (2)				
- L	n				
- B	Switch Context Ctrl+Sp	ace			
L L	Edit				
Anno	Сору				
🗄 🗱 Anal	Rename				
🖹 📮 🔤 🔤	Delete	Del			
Br	Write Input				
_ ~ ₩ # A	Data Check		-		
	Submit				
The	Continue		eeı		
Joint Loint	Monitor		ge 11 t		
Jo.	Results		are		
Joi	Kill		ce:		

- 23. 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

🗐 Eile Model	Vie	ewport	⊻iew	<u>R</u> esult	<u>P</u> lot	<u>A</u> nimate	R
: 🗅 🏞 🗖 🖥		<u>C</u> reate	,				
·		Next				Ctrl+Tab	
		Previo	us		Shif	t+Ctrl+Tab	
		Ca <u>s</u> ca	de				
Model Results		Tile <u>H</u> a	rizontall	y			
Session Data		Tile <u>V</u> e	rtically				
🖂 🥌 Quideau de Dach		<u>D</u> elete	Current				
🕀 🥁 Output Dat		Annota	ation <u>M</u> a	nager			
	`	Create	e <u>A</u> nnota	tion			
XYData		<u>E</u> dit Ar	nnotatior	ns			
Paths		Viewpo	ort Anno	tation <u>O</u> pl	tions		
🗄 🔚 Display Gro	ι	Linked	Viewpor	ts	NG		
Movies	~	<u>1</u> View	port: 1	ODB: C:	:/Temp/	/Bridge.odb	



- 24. Display the deformed contour of the (Von) Mises stress
 - a. In the toolbox area click on the "Plot Contours on Deformed Shape" icon



25. Request displacement plot along x-dir. With the undeformed shape superimposed.

	Vie <u>w</u> port	<u>V</u> iew	<u>R</u> esult	<u>P</u> lot	<u>A</u> nimate	R <u>e</u> po
	Prir	mary	▼ U		U 1	-
U. U1						
+1.167e-05 +1.070e-05 +9.727e-06 +8.754e-06 +7.781e-06 +6.809e-06 +5.836e-06 +3.83e-06 +3.891e-06 +2.918e-06 +1.458-06 +2.918e-06 +1.458-06 +9.727e-07 +9.727e-07 +0.000e+00						
			\rightarrow			

The image was obtained using the "superimpose options" icon and setting the parameters as shown. Note the clear stretching of the right side of the bracket and the imposition of symmetry BC preventing the lower horizontal segement from moving in the vertical direction.

123	- Superimpose Plot Options									
SUP 🛃 🔂 💠 🗤 🗤	Basic	Color & Style	Labels	Normals	Other					
Superimpose Options	Ren	der Style		Visible Edges	-					
	© ₩	/ireframe 🔘 Hid	den	🖱 All edges						
	I I Fi	lled 💿 Sha	ded	Exterior ed	lges					
			(Feature ed	lges					
👢 📕 🛄 📕			(Free edges	5					
			(🖱 No edges						
				÷ġŕ						
			_							
	0	К Арр	ly	Defaults	Cancel					

Oiscontinuities

- 26. Activate the icon labeled "Results Options" shown on the right.
 - a. Uncheck the box "Average element output at nodes."
 - b. Change the stress output to S11.



c. Note the stress plot now shows each element acquiring a single color, denoting the Constant Stress nature of CST element!



Apply

Cancel

"Averaging" creates average stress values at the nodes based on the number of elements sharing the node (and stresses corresponding to those elements). It then creates a color gradient between values of adjacent nodes, thus the reason for "smooth" representation of colors. Averaged results can be useful for investigating mesh convergence at a point of interest where a node can be placed.