## **Abaqus/CAE Axisymmetric Tutorial**

## **Problem Description**

A round bar with varying diameter has a total load of 1000 N applied to its top face. The bottom of the bar is completely fixed. Determine stress and displacement values in the bar resulting from the load.



🗖 🌲 🖻 🗞 🍟

Model Results

🚝 Model Database

🕀 🎎 Models (1)

## **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. Axisymmetric
  - b. Deformable
  - c. Shell
  - d. Approximate size = 0.2
- 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

Edit Material

- Colt Plate	erial	×
Name: Steel		
Description:	Linear Isotropic Steel (SI units)	
Material I	Behaviors	
General	at the standard state	
General	Mechanical Inermal Other	Delete
General	Elasticity	Elastic
	Mechanical Inermal Other	Elastic Hyperelastic
	Mechanical Inermal Other Elasticity Plasticity Damage for Ductile Metals	Elastic Hyperelastic Hyperfoam
	Mechanical Infermal Other           Elasticity         Plasticity           Damage for Ductile Metals         Plasticity           Damage for Traction Separation Laws         Plasticity	Elastic Hyperelastic Hyperfoam Hypogelastic
	<u>Mechanical Inermal Other</u> <u>Elasticity</u> <u>Plasticity</u> Damage for Ductile Metals Damage for Traction Separation Laws Damage for Fiber-Reinforced Composites	Elastic Hyperelastic Hyperfoam Hypoelastic Porous Elastic
	Mechanical         Inermal         Other           Elasticity         Plasticity         Plasticity           Damage for Ductile Metals         Damage for Traction Separation Laws         Damage for Fiber-Reinforced Composites           Damage for Piber-Reinforced Composites         Deformation Plasticity         Deformation Plasticity	Elastic Hyperelast Hypefoam Hypegelastic Porous Elastic Viscoelastic
	Mechanical         Inermal         Other           Elasticity         Plasticity         Plasticity           Damage for Dyctile Metals         Plasticity         Plasticity           Damage for Traction Separation Laws         Damage for Fiber-Reinforced Composites         Peformation Plasticity           Demping         Straaschen         Straaschen         Straaschen	Elastic Hyperelastic Hypefoam Hypegelastic Porous Elastic Viscoelastic
	Mechanical         Unermal         Other           Elasticity         Plasticity         Plasticity           Damage for Dyctile Metals         Plasticity         Plasticity           Damage for Traction Separation Laws         Damage for Fiber-Reinforced Composites         Plasticity           Deformation Plasticity         Demping         Expansion         Brittle Cracking	Elastic Hyperelastic Hyperfoam Hypogelastic Porous Elastic Viscoelastic

Name: Steel		
Description: Linear Isotrop	ic Steel (SI units)	
Material Behaviors		
Elastic		
<u>G</u> eneral <u>M</u> echanical	<u>T</u> hermal <u>O</u> ther	Delete
Elastic		
Type: Isotropic	•	✓ Suboptions
Use temperature-dependent data		
Number of field variables: 0		
Moduli time scale (for viscoelasticity): Long-term		
🗖 No tension		
Data		
Young's Modulus	Poisson's Ratio	
1 210e9	0.25	
		المحمورين ويتحمون بمسمعهم

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

Model Results Model Database	Create Section           Name:           AxisymmetricProperties	
Models (1)     Model-1     Parts (1)     Perts (1)     Perts (1)     Perts (1)     Perts (1)     Pertine     Profiles     Assemb Sections	Category     Type       Solid     Homogeneous       Shell     Generalized plane strain       Beam     Composite	Edit Section         Name: AxisymmetricProperties         Type: Solid, Homogeneous         Material: Steel         Plane stress/strain thickness: 1
€ of Steps (1)	Continue Cancel	OK Cancel

- 7. Expand the "Parts" node in the model tree and double click on "Section Assignments"
  - a. Select the surface geometry in the viewport
  - b. Select the section created above (AxisymmetricProperties)

Model Results	
See Model Database 💽 🌲 😰	Edit Section Assignment
Models (1)  Model-1  Parts (1)  Parts (1)  Parts (1)  Sets  Sets  Surfaces  Skins  Stringers  Scringers  Comp Setion Assignments (1)  Comp Setion Assignments  Comp Setion	Section Section: AxisymmetricProperties Create Note: List contains only sections applicable to the selected regions. Type: Solid, Homogeneous Material: Steel  Region Region: (Picked) OK 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 lower edge of the surface in the viewport

X



- c. Create another set named "Symmetry"
- d. Select the left edge 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 top 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



- 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
	⊙ Select from list below C Preselected defaults C All C Edit variables
	S,U,RF,CF
	▶     ✓       ▶     ✓       Strains
Model Results	
Model Database 🗸 🚖 🖻 🐘 🏹	
⊡ Model-1	► Failure/Eracture
🕀 Parts (1)	Thermal
🕀 🖉 Materials (1)	
E Sections (1)	Note: Error indicators are not available when Domain is Whole Model or Interaction.
ture de la competition de la	
田 編 Assenitiv 田 品 Steps (2)	Cutout at shall be an and burned as the points.
Field Output Requests (1)	Couput at sheir, bearr, and layered section points:
F-Output-1	• Use defaults • Specify:
History 👯 History	Include local coordinate directions when available
Time Pd F-Output-1	OK Cancel
T The second difference and the second fill the second s	

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 "Symmetry/Antisymmetry/Encastre" for the type



- b. In the prompt area click on the Sets button
- c. Select the set named "Fixed"

← X Select regions for the boundary condition 🔛 Done

Region Selection	X
Eligible Sets	
Sets below may contain vertices,	edges, faces or cells.
Name	Туре
Fixed	Geometry
Symmetry	Geometry
Highlight selections in viewpo	rt
Continue	Cancel

d. Select "ENCASTRE" for the boundary condition

Sets.N

Edit E	Soundary Condition	
Name:	Fixed	
Type:	Symmetry/Antisymmetry/Encastre	
Step:	Apply Load (Static, General)	
Region:	Fixed	
C XSYM	M (U1 = UR2 = UR3 = 0)	
C YSYM	M (U2 = UR1 = UR3 = 0)	
O ZSYM	M(U3 = UR1 = UR2 = 0)	
C XASYMM (U2 = U3 = UR1 = 0; Abaqus/Standard only)		
○ YASYMM (U1 = U3 = UR2 = 0; Abaqus/Standard only)		
C ZASYMM (U1 = U2 = UR3 = 0; Abaqus/Standard only)		
C PINN	ED (U1 = U2 = U3 = 0)	
	STRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)	
	OK Cancel	

- e. Repeat the procedure for the symmetry restraint using the set named "Symmetry", select "XSYMM" 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

Region Selection	x
Eligible Sets	
Surfaces below may contain faces.	
Name	Туре
Pressure	Surface
Highlight selections in viewport	
I Highlight selections in viewport	
Continue	Cancel

- 16. In the model tree double click on "Mesh" for the Bar 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 "Axisymmetric Stress" for family
  - d. Note that the name of the element (CAX4R) and its description are given below the element controls

		📑 Element Type		×
		Element Library	Family	
		• Standard • Explicit	Acoustic	3
Model Results		Geometric Order	Axisymmetric Stress Cohesive	
Se Model Database 💽 🌲 😵		C Linear C Quadratic	Coupled Temperature-Displacement	-
🖃 🏭 Models (1)		Quad Tri		
🗍 Model-1		Element Controls		5U
🗄 🖫 Parts (1)		Hybrid formulation		
🖻 Bracket		Reduced integration		
🗄 📇 Features (1)		Incompatible modes		
by Sets		Hourglass stiffness:	🖲 Use default 🖸 Specify	
Jurfaces	II. =	Second-order accuracy:	C Yes 💿 No	
Skins		Distortion control:	O Use default O Yes O No	
Stringers			Length ratio: 0.1	
T R Section Assignments (1)	man P B	Hourglass control: 💿 U:	se default C Enhanced C Relax stiffness C Stiffness C Viscous C Combined	
Composite Lavuns	S4R		Stiffness-viscous weight factor: 0.5	
		Displacement hourglass s	caling factor: 1	
B Mach (Empty)	Assign	tin na hullu sinnaitu anali		
		Linear baix viscosicy scali		
$\square \square $	<u>1: 1-</u>	Quadratic bulk viscosity s	caling Factor: 1	
		CAX4R: A 4-node bilinear	axisymmetric quadrilateral, reduced integration, hourglass control.	
الاغتماميرية 🛒 ۲۲۹ والعصمينية محمد معامدته مرياميا	الى دەرىلىسى مىر كانىل، ا	a second and the second se	هم ومصفقه من الصورة ما ومصفى إردي معدودهما المعرة فرض والمناطق والدين المراجر الما معروفة فراعه معتم والمعتقرين المراجل المستقد فا	~~~

17. In the toolbox area click on the "Assign Mesh Controls" icon

- a. Change the element shape to "Quad"
- b. Change the Algorithm to "Medial axis" for a more structured mesh

Assi See Mesh Co	gn ontrols

Mesh Controls	<u>×</u>
Element Shape     O Quad	-dominated C Tri
C As is C Free C Structured C Structured C Sweep C Multiple	Algorithm  Medial axis  Minimize the mesh transition  Tip  Advancing front  Use mapped meshing where appropriate
ОК	Cancel

-

- 18. In the toolbox area click on the "Seed Part" icon
  - a. Set the approximate global size to 0.005

	Global Seeds
	☐ Sizing Controls
	Approximate global size: 0.005
	Curvature control
	Maximum deviation factor $(0.0 < h/L < 1.0)$ : 0.1
	(Approximate number of elements per circle: 8)
	Minimum size factor (as a fraction of global size):
	O Use default (0.1)
L	· · · · ·
	OK Apply Defaults Cancel

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



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

		Name: Bar
		Model: Model-1
		Description: Static analysis of axisymmetric bar
		Submission General Memory Parallelization Precision
		Job Type
		C Full analysis
		C Recover (Explicit)
	Create Job 🔀	C Restart
	Name: Bar	Run Mode
	Source: Model	Background C Queue:
	Model-1	Type:
Predefined Fields		Submit Time
		© Immediately
		C Wait: hrs. min.
Annotations		C At: Tip
Ad Jobs Processes	Continue Cancel	OK

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



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



- 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

\_\_\_\_

🗧 <u>F</u> ile <u>M</u> o	del Vi	iewport	⊻iew	<u>R</u> esult	<u>P</u> lot	Animate	R
	1.5	Create					
	1 2	Next				Ctrl+Tab	
		Previou	15		Shift	:+Ctrl+Tab	
		Cascad	le				
Model Resu	ılts	Tile <u>H</u> or	rizontall	y			
Session Data	_	Tile <u>V</u> er	tically				
		<u>D</u> elete	Current				
🗄 🔁 Output	Data	Annota	ition <u>M</u> a	nager			
🖽 🗧 Spectru	ıms (	Create	Annota	tion			
YVData		<u>E</u> dit An	notatior	ns			
Paths		Viewpo	rt Anno	tation Opl	ions		
E Display	Grou	Linked	Viewpor	ts	Νζ		
Movies	~	1 Viewp	oort: 1	ODB: C:	/Temp/	Bridge.odb	

Viewpo	Viewport Annotation Options					
General Triad Legend Title Block State Block						
Visibili	t <b>y</b> —	,				
I Show	compas	s				
Show	v triad					
Show	legend					
Show	/ title blo	CK				
Show	/ state D	юск 				
Show text and arrows						
OK	#	pply	Defaults	Cancel		

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





Static analysis of axisymmetric bar ODB: Bar.odb Abaqus/Standard Student Edition 6.8-2 Sun Jan 04 21:59:45 Pacific Standard Time 2009 Step: Apply Load, Apply pressure load to the top of the bar Increment 1: Step Time = 1.000 Primary Var: S, Mises Deformed Var: U Deformation Scale Factor: +6.393e+03



## 25. To determine the stress values, from the menu bar click Tools $\rightarrow$ Query

- a. Check the boxes labeled "Nodes" and "S, Mises"
- b. In the viewport mouse over the element of interest
- c. Note that Abaqus reports stress values from the integration points, which may differ slightly from the values determined by projecting values from 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
- d. Click on an element to store it in the "Selected Probe Values" portion of the dialogue box



- 26. To change the output being displayed, in the menu bar click on Results → Field Output
  - a. Select "Spatial displacement at nodes"
    - i. Invariant = Magnitude

	📑 Field Output		×		
	Step/Frame				
	Step: 1, Apply Load				
	Frame: 0 Step/Frame				
	Primary Variable	Deformed Variable Symbol Variable Status Variable	1		
	Output Varia	able			
	List only va	riables with results:			
	Name	Description (* indicates complex)			
	CF	Point loads at nodes			
	RF	Reaction force at nodes			
	S	Stress components at integration points			
<u>P</u> lot <u>A</u> nimate R <u>e</u> o/Frame	Magnitude	U1 U2			
ve Steps/Frames tion <u>P</u> oints d Qutput	Section Points				
or\$Output ions	OK	Apply	Cancel		

- 27. To create a text file containing the stresses and reaction forces (including total), in the menu bar click on Report → Field Output
  - a. For the output variable select (Von) Mises

Result Ste Act Sec Fiel Hist

- 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: 1, Apply Load       Frame: 1     Step/Frame       Variable     Setup	Step/Frame       Step: 1, Apply Load       Frame: 1     Step/Frame       Variable     Setup
Report Options T	Output Variables         Position:       Integration Point         Click checkboxes or edit the identifiers shown next to Edit below.         Image: Signal Stress components         Imag	File         Name:       bar.rpt         Image:       Select         Image:       Single table for all field output variables         Image:       Separate table for each field output variable         Sort by:       Element Label         Image:       Number         Image:       No limit         Image:       Singlificant digits:         Image:       Image:         Image:       Image:
<u>X</u> Y Field Output	OK Apply Defaults Cancel	OK Apply Defaults Cancel

- a. Back on the Variable tab change the position to "Unique Nodal"
- b. Uncheck the stress variable, and select the RF1 reaction force
- c. On the Setup tab, check the "Column totals" option
- d. Click OK

Report Field Output	Report Field Output
Step/Frame	Step/Frame
Step: 1, Apply Load	Step: 1, Apply Load
Frame: 1 Step/Frame	Frame: 1 Step/Frame
Variable Setup	Variable Setup
Output Variables	File
Position: Unique Nodal	Name: bar.rpt Select
Click checkboxes or edit the identifiers shown next to Edit below.	Append to file
CF: Point loads	🗆 Output Format
▼ 🔽 RF: Reaction force	Layout: 💿 Single table for all field output variables
Magnitude	C Separate table for each field output variable
RF1 ₩ RF2	Sort by: Node Label
S: Stress components	Ascending O Descending
U: Spatial displacement	Page width (characters):  No limit C Specify: 80
	Number of significant digits: 6 🚔
	Number format: Engineering
Edit: RF.RF2	Data
Section point: C All C Select Settings,	Write: 🔽 Field output 🔽 Column totals 🔽 Column min/max
OK Apply Defaults Cancel	OK Apply Defaults Cancel

- 28. Open the .rpt file with any text editor
  - a. One thing to check is that the total reaction force is equal to the applied load.