ABAQUS Tutorial

3D Modeling

This exercise intends to demonstrate the steps you would follow in creating and analyzing a simple solid model using ABAQUS CAE.

Introduction

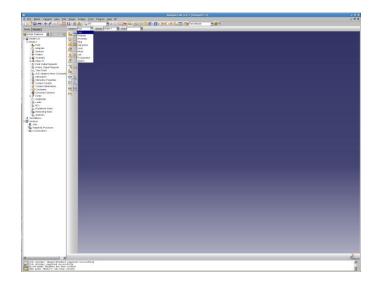
A solid undergoes thermal expansion due to the application of heat along with deformation due to applied load.

Model Definition

Consider a thin aluminum cylinder of length 1 m and inner and outer radii 0.2 m & 0.21 m respectively. The cylinder is kept fixed at one end and at the other end a tensile load of 200 kPa is applied. The fixed end of the cylinder is at 273.15 K (the ambient temperature) and the free end at 274.15 K (all other sides are insulated). The cylinder expands due to the heat flow.

The various functions within ABAQUS are organized into modules and we are going to use these modules to define the steps in our procedure.

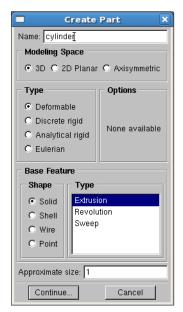
- 1. >module load abaqus/6.9-2
- 2. >abaqus cae
- 3. Once you start ABAQUS CAE select **Create Model Database** to create a new model.
- 4. The default module that opens up is the Part Module.



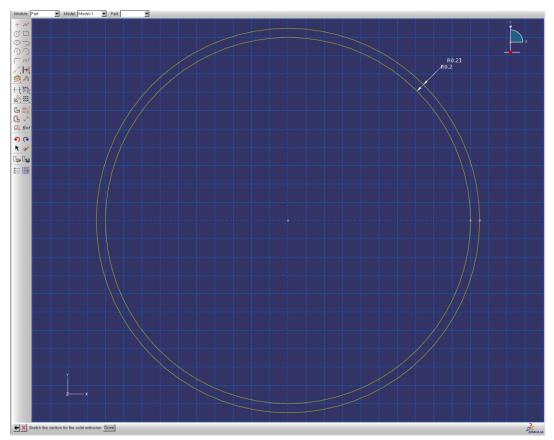
Part Module:

This module allows you to create the geometry required for the problem. To create a 3-D geometry you first create a 2-D profile and then manipulate it to obtain the solid geometry.

- 1. From the **Part Toolbox** on the left of the viewport select **Create Part**.
- You can name the part as cylinder or anything else you like. We are going to create a deformable solid shape in the 3-D modeling space through extrusion so we do not change the default selections.
- 3. Enter 1 as the approximate size and click **Continue**.



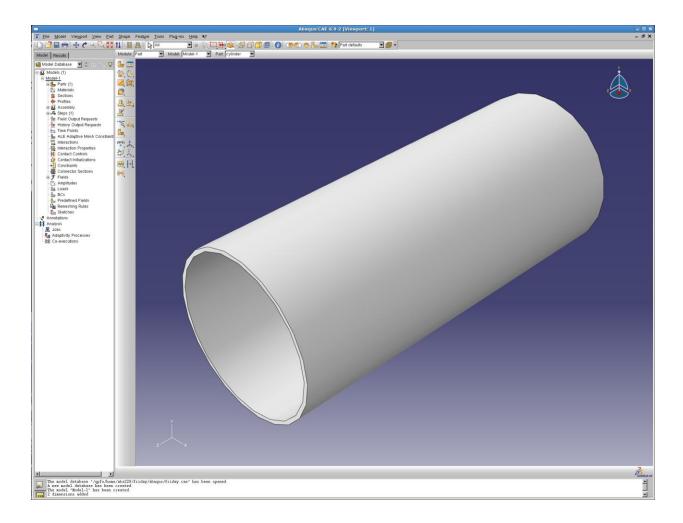
4. Click Create Circle Center and Perimeter on the drawing toolbox and enter 0, 0 as the center point in field below the viewport and press Enter. Enter the perimeter point as 0.21, 0 and press Enter to complete the circle. Similarly make another circle with the same center and the perimeter point as 0.2, 0. Press Esc to exit the circle definition and then press Done.



5. Enter the extrusion depth as 1 and press OK.

Edit Base Extru	ision 🗙
End Condition	
Type: Blind	
Depth: 1	
Options	
Note: Twist and draft cannot be	specified together.
☐ Include twist, pitch: 0	(Dist/Rev)
□ Include draft, angle: 0	(Degrees)
ОК	Cancel

6. Click **Auto-Fit View** in the toolbar above to zoom out and view all the points.



This finishes our work in the **Part** module. Select **Module: Property** from the toolbar above the viewport.

Property Module:

In this module you define the material properties for your analysis and assign those properties to the available parts.

- 1. Select Create Material from the Property Toolbox.
- 2. Enter material name as Aluminum. Click on the General tab and select Density from the drop-down menu. Type in the mass density as 2700. Click on the Mechanical tab and select Elasticity>Elastic from the drop-down menu. Enter the Young's Modulus as 70E9 and the Poisson's Ratio as 0.33. Click on the Mechanical tab and select Expansion. Edit the reference temperature to 273.15 and the expansion coefficient to 23e-6. Click on

the **Thermal** tab and select **Conductivity.** Enter the thermal conductivity as 160. Click on the **Thermal** tab and select **Specific Heat**. Enter the value as 900 and click **OK**.

Edit Material	×		Edit Material	X
Name: Aluminum		Name: Alum	num	
Description:	Edit	Description:		Edit
l				
Material Behaviors		Material B	ehaviors	
		Density		
General Mechanical Thermal Other	Delete	General	<u>M</u> echanical <u>T</u> hermal <u>O</u> ther	Delete
Density		- Density -		Elastic
Dep⊻ar		-	Plasticity •	<u>H</u> yperelastic
Regularization User Material		🗖 Use ten	Damage for D <u>u</u> ctile Metals Damage for Traction Separation Laws	Hyperfoam
User Defined Eield		Number of	Damage for Fiber-Reinforced Composites ►	Hyp <u>o</u> elastic <u>P</u> orous Elastic
User Output Variables		Data		Viscoelastic
		I D	Deformation Plasticity	Turentante
		1	<u>D</u> amping	
			Expansion	
		-	Brittle Cracking	
L				
OK	Cancel		OK	Cancel
· · · · · · · · · · · · · · · · · · ·				
Edit Material	×		Edit Material	X
Name: Aluminum	×	Name: Alum		×
	Edit	Name: Alum Description:		Edit
Name: Aluminum Description:		Description:	inum	
Name: Aluminum Description: Material Behaviors		Description: Material B	inum	
Name: Aluminum Description:		Description:	inum	
Name: Aluminum Description: Material Behaviors Density		Description: Material B Density	inum	
Name: Aluminum Description: Material Behaviors Density		Description: Material B Density Elastic	inum	
Name: Aluminum Description: Material Behaviors Density Elastic	Edit	Description: Material B Density Elastic Expansion	num ehaviors	
Name: Aluminum Description: Material Behaviors Density Elastic General Mechanical Ihermal Other		Description: Material B Density Elastic	num ehaviors Mechanical Thermal Other	
Name: Aluminum Description: Material Behaviors Density Elastic General Mechanical Thermal Other Elasticity	Edit	Description: Material B Density Elastic Expansion	num ehaviors Mechanical [Thermal Other Conductivity	Edit
Name: Aluminum Description: Density Elastic General Mechanical Inermal Other Elasterity	Edit	Description: Material B Density Elastic Expansion <u>G</u> eneral	num ehaviors Mechanical Thermal Other Conductivity Heat Generation	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: Isoti	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Eastic Expansion General Expansion Type: [sout Use use	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [sott Use use Reference	num Mechanical Thermal Other Conductivity Heat Generation Inelastic Heat Fraction Isubroutine U Latent Heat Specific Heat	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [sott Use use Reference Use ter	num Mechanical Thermal Other Conductivity Heat Generation Inelastic Heat Fraction Latent Heat Specific Heat perature-dependent data	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of	num Mechanical Thermal Other Conductivity Heat Generation Inelastic Heat Fraction Isubroutine U Latent Heat Specific Heat	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soli Use use Reference Use ter Number of Data External Expansion	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit
Name: Aluminum Description:	Edit	Description: Material B Density Elastic Expansion General Expansion Type: [soft Use use Reference Use ter Number of Data Co	num	Edit

Edit Material	X
Name: Aluminum	
Description:	Edit
r Material Behaviors	
Density Elastic Expansion Conductivity	
General Mechanical Thermal Other Conductivity Conductivity East Generation Type: Isotropic Inelastic Heat Fraction Use temperature-depe Joule Heat Fraction	Delete
Number of field variables Latent Heat Data	
Conductivity 1 180	
ОК	Cancel

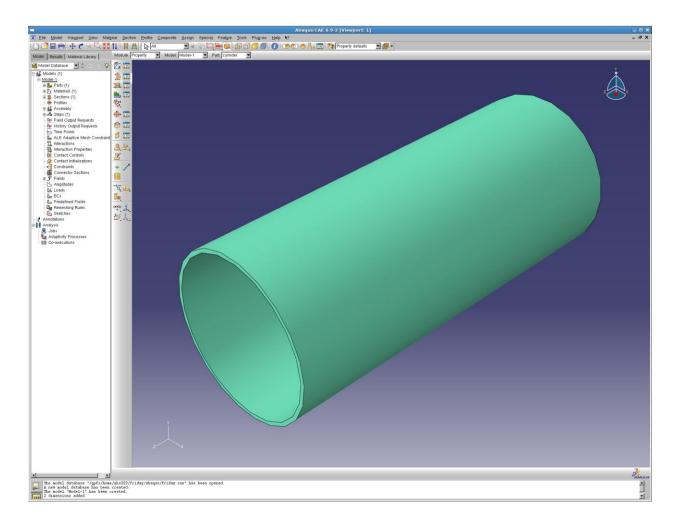
 Select Create Section from the property toolbox. Name the section as you like. We need a solid homogeneous section for our problem. Click Continue. Select the material as Aluminum and click OK.

	Create Section 🛛 🗙	
Name: Alum	inum_hom[
Category	Туре	Edit Section
 Solid Shell Beam Other 	Homogeneous Generalized plane strain Eulerian Composite	Name: Aluminum_hom Type: Solid, Homogeneous Material: Aluminum Create
Continu	Ie Cancel	OK

Click Assign Section on the property toolbox and select the part from the viewport. Click
 Done below. Select the section you had created and click OK.

Edit Section Assignment X
Section
Section: Aluminum_hom Create
Note: List contains only sections applicable to the selected regions.
Type: Solid, Homogeneous
Material: Aluminum
Region
Region: (Picked)
OK Cancel

Our work in the Property module is done and we select the **Assembly Module** from the toolbar above the viewport.



Assembly Module:

This module allows you to assemble together parts that you have created. Even if you have a single part you need to include it in your assembly.

- 1. Select Instance Part from the Assembly Toolbox.
- Select the part you have created from the parts list and then select Instance type: Independent. Click OK.

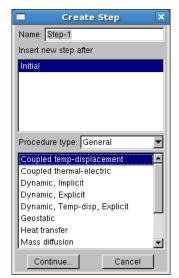
Create Instance 🗙			
Parts			
cylinder			
Instance Type			
C Dependent (mesh on part)			
Independent (mesh on instance)			
Notes To show a Dependent instance!			
Note: To change a Dependent instance's mesh, you must edit its part's mesh.			
Auto-offset from other instances			
OK Apply Cancel			

Select **Module: Step** from the toolbar above.

Step Module:

This module allows you to select the kind of analysis you want to perform on your model and define the parameters associated with it. You can also select which variables you want to included in the output files in this modules. You apply loads over a step. To apply a sequence of loads create several steps and define the loads for each of them.

- 1. Select Create Step from the Step Toolbox.
- Name the step as you want and select Coupled temp-displacement as the procedure.
 Click Continue.



- The edit step dialog box lets you choose the solution technique, the solver type and define the time stepping strategy.
- 4. Under **Basic** change the Response to **Steady-state** and click **OK**.

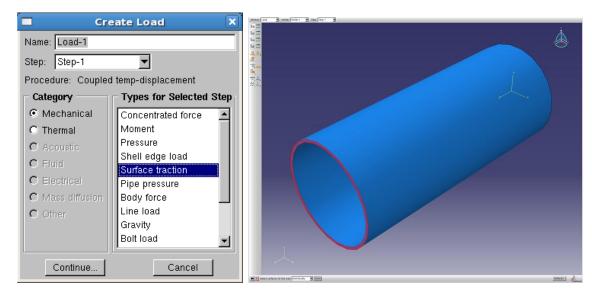
Edit Step	X)
Name: Step-1	
Type: Coupled temp-displacement	
Basic Incrementation Other	
Description:	
Response: 💿 Steady-state C Transient	L
Time period: 1	L
NIgeom: C Off (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)	l
Automatic stabilization: None	
	L
Include creep/swelling/viscoelastic behavior	L
	L
	L
	L
OK	

The **Interaction Module** allows you to set up interactions (contact, film), constraints, connectors, fasteners and wire feature between parts. Our problem does not involve any of these features but it will be a good idea to explore this module on your own at a later time. Select **Module: Load** from the toolbar above.

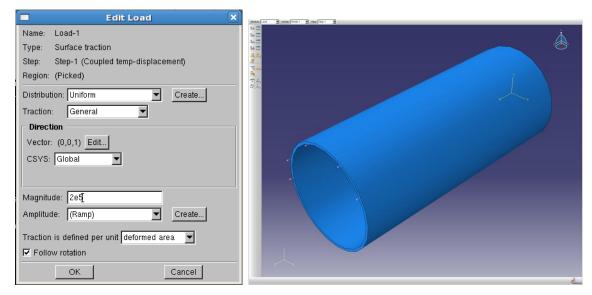
Load Module:

The Load Module is where you define the loads and boundary conditions for your model for a particular step (indicated in the toolbar above). You can even define loads and boundary conditions as fields like electric potential, acoustic pressure, etc.

Select Create Load from the Load Toolbox. Select Surface Traction and click Continue.
 Select the top face of the cylinder (z=1) (it gets highlighted in red) and click Done.



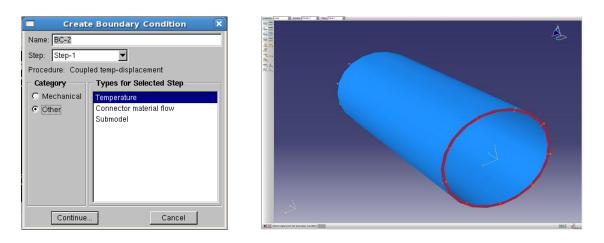
Change the Traction type to General. Click on the Edit tab under Direction in the dialog box. Enter the starting point of the direction vector as (0, 0, 0) and the end point as (0,0, 1). Enter the Magnitude as 2e5 and click OK.



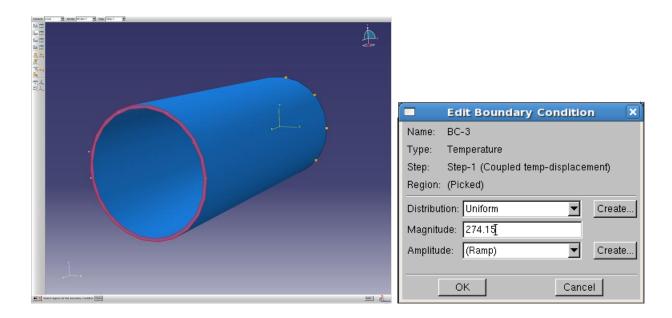
 Select Create Boundary Condition from the Load Toolbox. Select Symmetry / Antisymmetry / Encastre and click Continue. Select the bottom face (z=0) of the cylinder and click Done. Select Pinned (U1=0, U2=0, U3=0) and click OK.

Creat	e Boundary Condition	
Name: BC-1		
Step: Step-1	•	
	ed temp-displacement	
Category	- Types for Selected Step	
Mechanical	Symmetry/Antisymmetry/En	castre
C Other	Displacement/Rotation	
	Velocity/Angular velocity Connector displacement	
	Connector velocity	
	l í	
Continue.	Cancel	A a constant of the lacost product from the first state of the state o
		Edit Boundary Condition
	Name:	
	Туре:	Symmetry/Antisymmetry/Encastre
	Step:	Step-1 (Coupled temp-displacement)
	Regior	: (Picked)
	O XS	/MM (U1 = UR2 = UR3 = 0)
	C YS	/MM (U2 = UR1 = UR3 = 0)
	O ZS	/MM (U3 = UR1 = UR2 = 0)
	O XA	SYMM (U2 = U3 = UR1 = 0; Abaqus/Standard only)
	O YA	SYMM (U1 = U3 = UR2 = 0; Abaqus/Standard only)
		SYMM (U1 = U2 = UR3 = 0; Abaqus/Standard only)
	······	NED (U1 = U2 = U3 = 0)
		CASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)
		OK Cancel

 Again select Create Boundary Condition from the Load Toolbox. Switch Category to Other and select Temperature and click Continue. Select the bottom face of the cylinder and press Done. Enter the magnitude as 273.15 and click OK. Similarly put the top face at 274.15.



	Edit Boundary Condition	n 🗙
Name:	BC-2	
Туре:	Temperature	
Step:	Step-1 (Coupled temp-displacen	nent)
Region:	(Picked)	
Distribu	tion: Uniform	Create
Magnitu	ıde: 273.15	
Amplitu	de: (Ramp) 💌	Create
	OK Cance	el

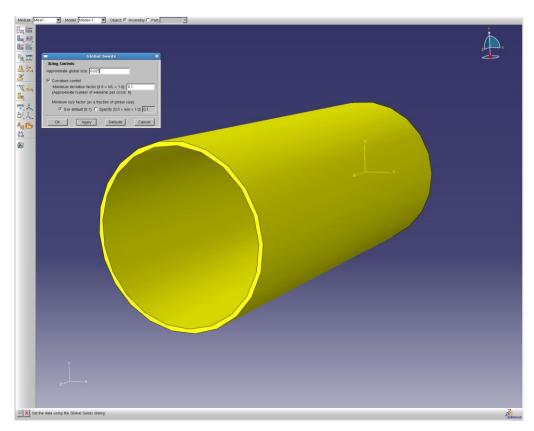


Now that we have defined the loads and the boundary conditions we move on to mesh the geometry. Select **Module: Mesh** from the toolbar above the viewport.

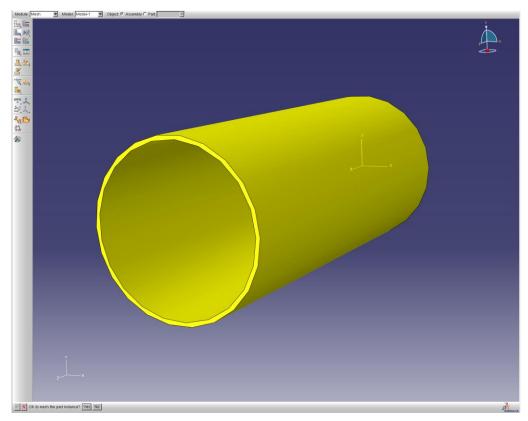
Mesh Module:

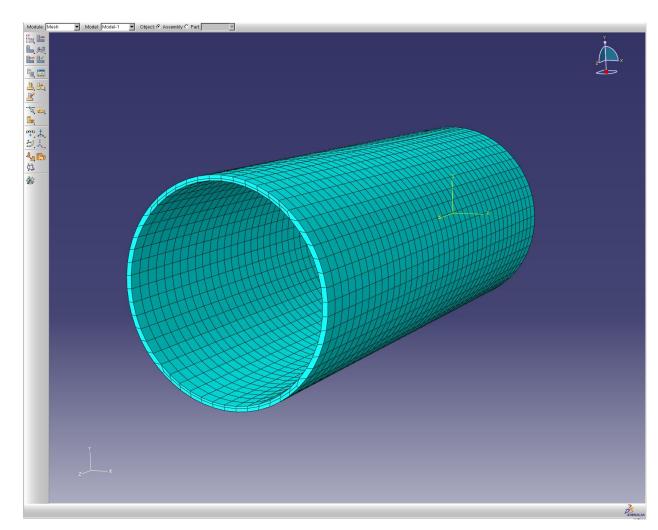
The mesh model controls how you mesh your model – the type of element, their size etc.

 Select Seed Part Instance from the mesh toolbox. Enter the approximate global size as 0.025.



2. Click on **Mesh Part Instance** and then on **Yes** to mesh the model.





3. Select **Assign Element Type** from the mesh toolbox. Under **Family** select Coupled Temperature-Displacement and switch **Geometric Order** to Quadratic. Click **OK**.

	Element Type
Element Library	Family
Standard C Explicit	Continuum Shell
	Coupled Temperature-Displacement
Geometric Order	Gasket
Linear 🖲 Quadratic	Heat Transfer
Hex Wedge Tet	
Element Controls	
Hybrid formulation	
Reduced integration	
Viscosity: • Us	se default C Specify
	se default C Yes C No
Max Degradation:	
max Degradation	e delaur o opecity

When finished select **Module: Job** from the toolbar above.

Job Module:

This module allows you to submit your model for analysis.

1. Select **Create Job** from the **Job Toolbox**. Name the job as you like. Select your model and click **Continue**.

Create Job	×
Name: cylindef	
Source: Model 💌	
Model-1	
Continue Cance	

2. You can add a description to the job, allocate memory, allot multiple processors and select precision. Use the default values and click **OK**.

Edit Job	X
Name: cylinder	
Model: Model-1	
Description:	
Submission General Memory Parallelization Precision	
_ Job Type	
• Full analysis	
C Recover (Explicit)	
O Restart	
- Run Mode	
Background C Queue: Host name:	
Туре:	
Submit Time	
C Immediately	
O Wait: hrs. min.	
C At Tip	
OK Cancel	

3. Select the **Job Manager** from the toolbox and click on the **Write Input** tab.

	jo	b Manager		×
Name	Model	Туре	Status	Write Input
cylinder	Model-1	Full Analysis	None	Data Check
				Submit
				Continue
				Monitor
				Results
				Kill
Create	Edit Copy	Rename	Delete	Dismiss

4. If you are running the job for the first time it is advisable to run **Data Check** to check the input file for errors. Click OK to overwrite the job files.

	J	ob Manager		×
Name	Model	Туре	Status	Write Input
cylinder	Model-1	Full Analysis	None	Data Check
		Abaqus	×	Submit
	<u></u> <u></u> oi	b files already exist fo K to overwrite? v this warning next tim		Continue Monitor
			ncel	Results Kill
Create	Edit Cop	y Rename	Delete	Dismiss

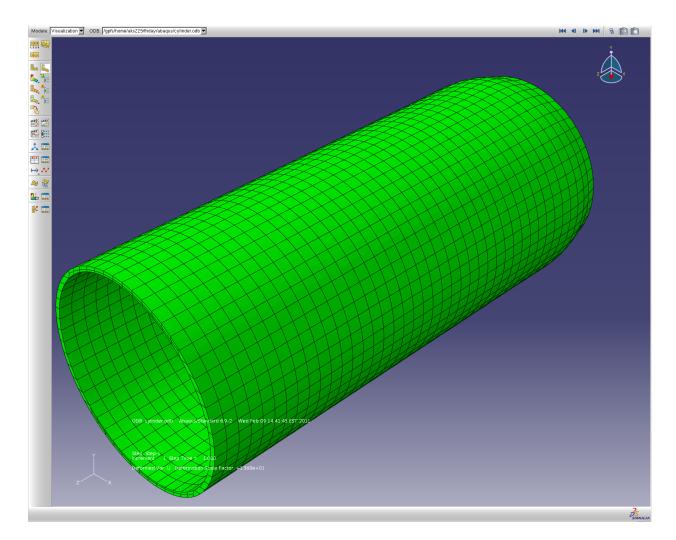
 Once the data check is completed Submit the job for analysis. Click OK to overwrite the job files. You can click Monitor to observe the progress of the solution process. You can see the errors, warnings, data and message file.

				cylinder	Monitor			2
Job: cyl	inder Statu	s: Comp	leted					
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
Log E	Errors 🛛 ! Warr	nings 🛛 C	output Dat	a File 🛛 Me	essage File	Status File	1	
Submitt	ed: Wed Feb	9 14:41:	33 2011					
Started	: Analysis In	put File F	rocessor					
Comple	ted: Analysis	Input File	Processor					
Started	Abaqus/Sta	ndard						
Comple	ted: Abaqus/S	Standard						
Comple	ted: Wed Feb	9 14:42	:12 2011					
Search					-			
Text to 1	înd:				Match ca	se 🖟 Next	얀 Previous	
						Di	smiss	

 Once the job is completed click on the **Results** tab on the job manager. This opens the Visualization Module for postprocessing.

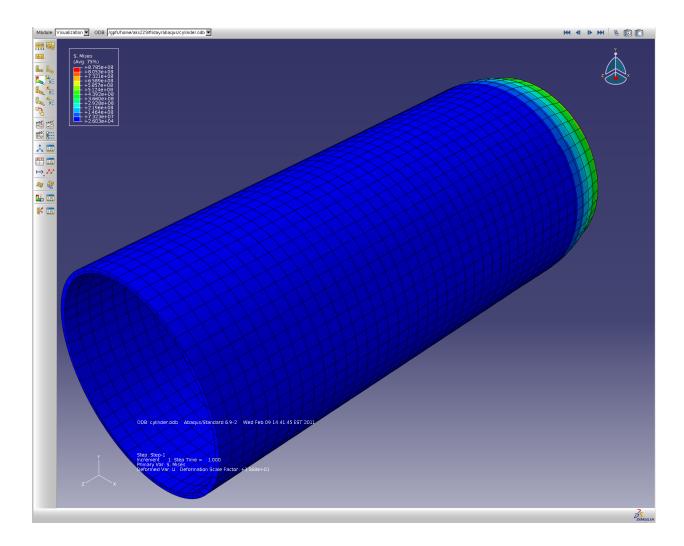
Visualization Module:

This model allows you to look at your model after deformation. You can also plot values of stress, displacement, reaction forces, etc. as contours on your model surface or as vectors or tensors.



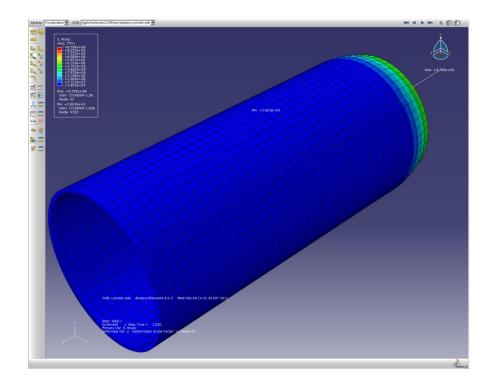
1. Select **Plot Deformed Shape** from the Visualization toolbox.

 Select Plot Contours on Deformed Shape to plot stress contours on the model surface.

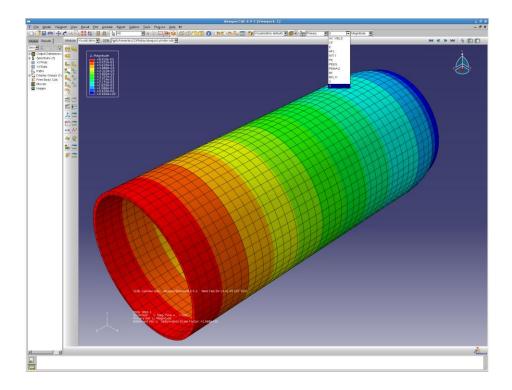


 You can see the location of the maximum & minimum stresses by selecting Contour Options>Limits>Show Location.

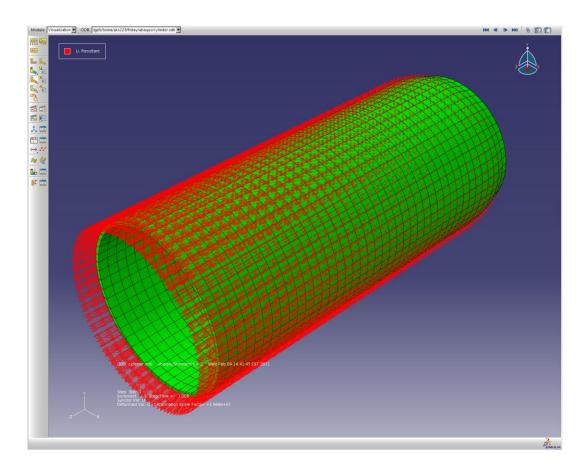
	Contour Plot Options
Basic	Color & Style Limits Other
Note:	User-defined interval values override the settings below.
- Min/	Max
Max:	 C Auto-compute (8.78462e+08)
Min:	C Auto-compute (26033.1) ✓ Show location Specify: 26033.1
Auto	-Computed Limits
Wher	auto-computing animation limits:
Use t	īrst and last frame limits 📃
0	K Apply Defaults Cancel



- 4. Select **Results>Field Output** from the main menu. This opens a dialog box that allows you to select the variable you want to plot in the viewport.
- 5. Select **U** (Spatial Displacement at nodes)>**Magnitude**>**OK** to plot the displacement contours on the model.



 To plot displacement vectors click on Plot Symbols on Deformed Shape on the toolbox.



7. You can now animate this plot by selecting Animate Harmonic.

Mouse Gestures:

Ctrl+Alt+Left Click (MB1): Rotate View Ctrl+Alt+Middle Click (MB2): Pan View Ctrl+Alt+Right Click (MB3): Magnify View Use Shift key to select multiple objects.

Note on System of Units:

ABAQUS has no built-in system of units. Specify all unit data in consistent units. Some common systems of consistent units: SI: m, N, kg, s, Pa, J, kg/m³ SI (mm): mm, N, tonne (1000 kg), s, MPa, mJ, tonne/mm³ US Unit (ft): ft, lbf, slug, s, lbf/ft², ft lbf, slug/ft³