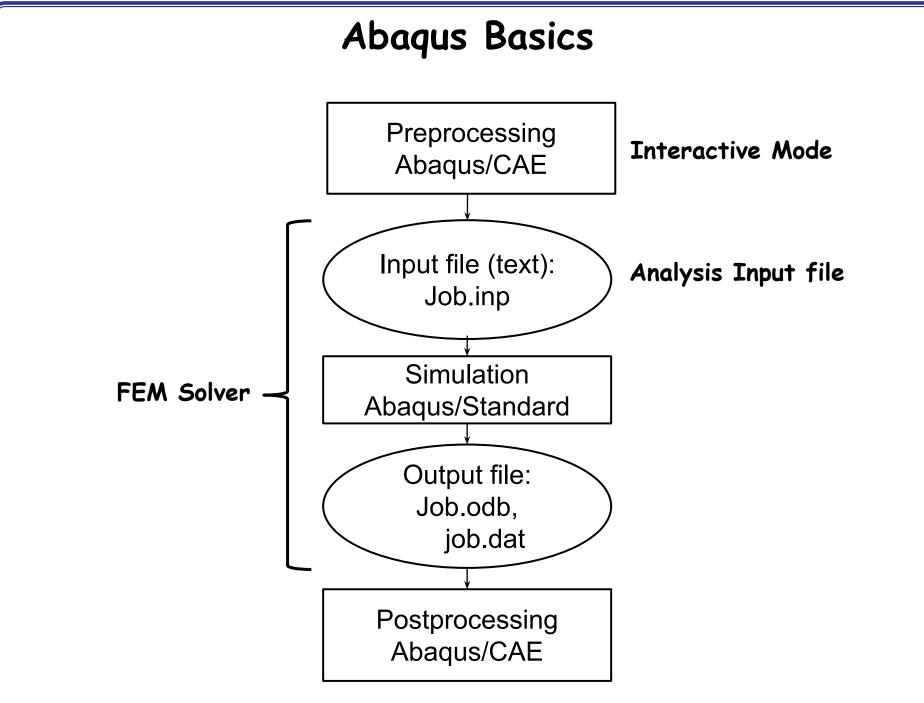
Finite Element Analysis

Using Abaqus

Instructor: Nam-Ho Kim (nkim@ufl.edu)



Methods of Analysis in ABAQUS

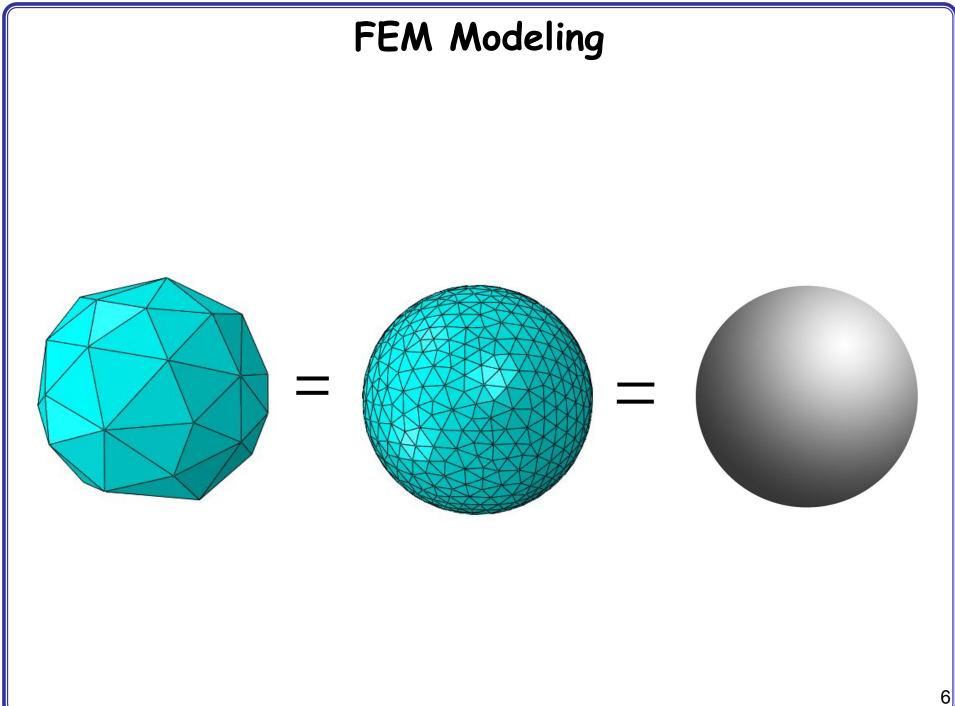
- Interactive mode
 - Create an FE model and analysis using GUI
 - Advantage: Automatic discretization and no need to remember commands
 - Disadvantage: No automatic procedures for changing model or parameters
- Python script
 - All GUI user actions will be saved as Python script
 - Advantage: Users can repeat the same command procedure
 - Disadvantage: Need to learn Python script language

Methods of Analysis in ABAQUS

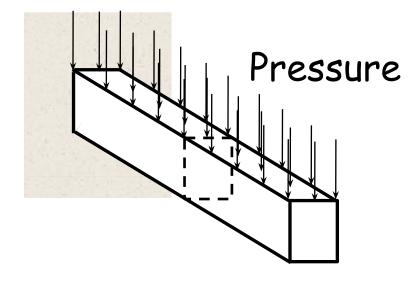
- Analysis input file
 - ABAQUS solver reads an analysis input file
 - Possible to manually create an analysis input file
 - Advantage: Users can change model directly without GUI
 - Disadvantage: Users have to discretize model and learn ABAQUS input file grammar

Components in ABAQUS Model

- Geometry modeling (define geometry)
- Creating nodes and elements (discretization)
- Element section properties (area, moment of inertia, etc)
- Material data (linear/nonlinear, elastic/plastic, isotropic/orthotropic, etc)
- Loads and boundary conditions (nodal force, pressure, gravity, fixed displacement, joint, relation, etc)
- Analysis type (linear/nonlinear, static/dynamic, etc)
- Output requests

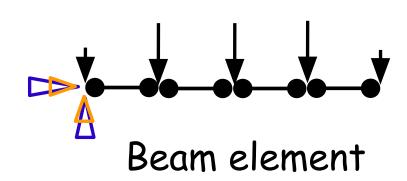


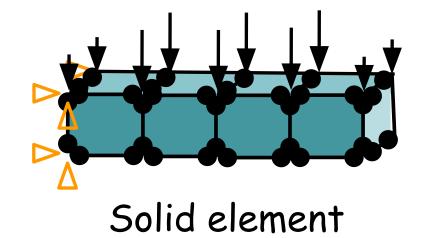
FEM Modeling





- Which element type?
 - Section properties
 - Material properties
 - Loads and boundary conditions
 - Output requests





FEM Modeling

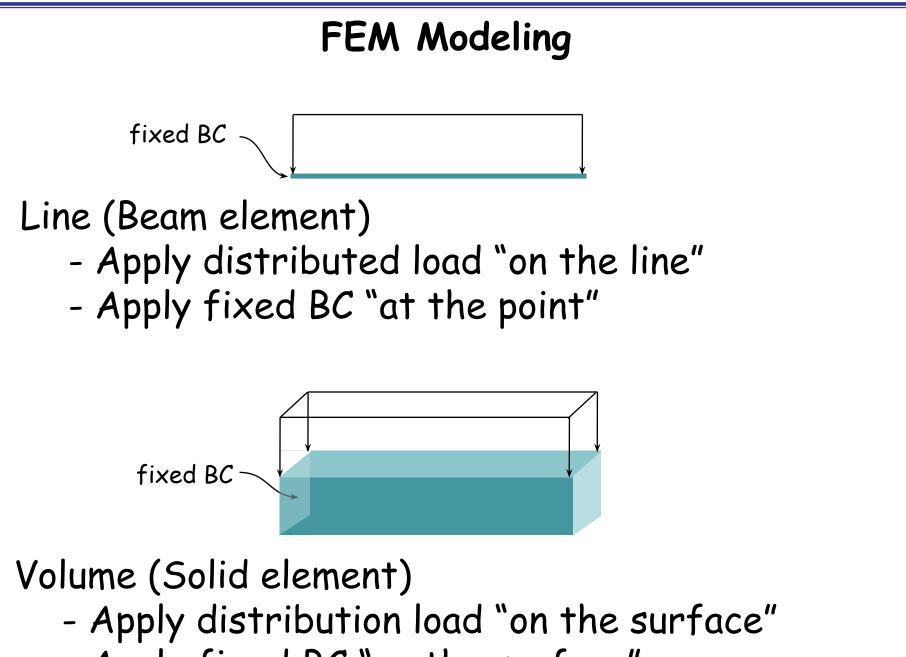
Line (Beam element)

- Assign section properties (area, moment of inertia)

- Assign material properties

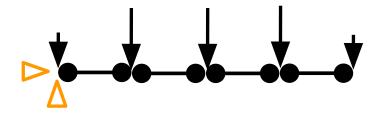
Volume (Solid element)

- Assign section properties
- Assign material properties



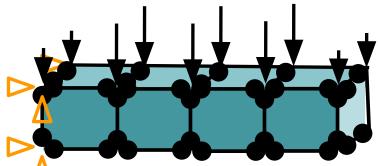
- Apply fixed BC "on the surface"

FEM Modeling



Line (Beam element)

- Discretized geometry with beam element
- Discretized BC and load on nodes

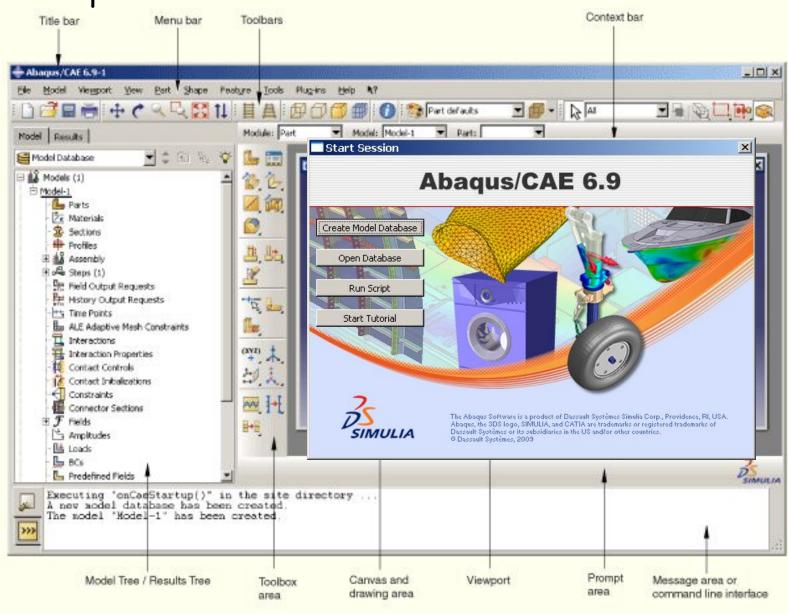


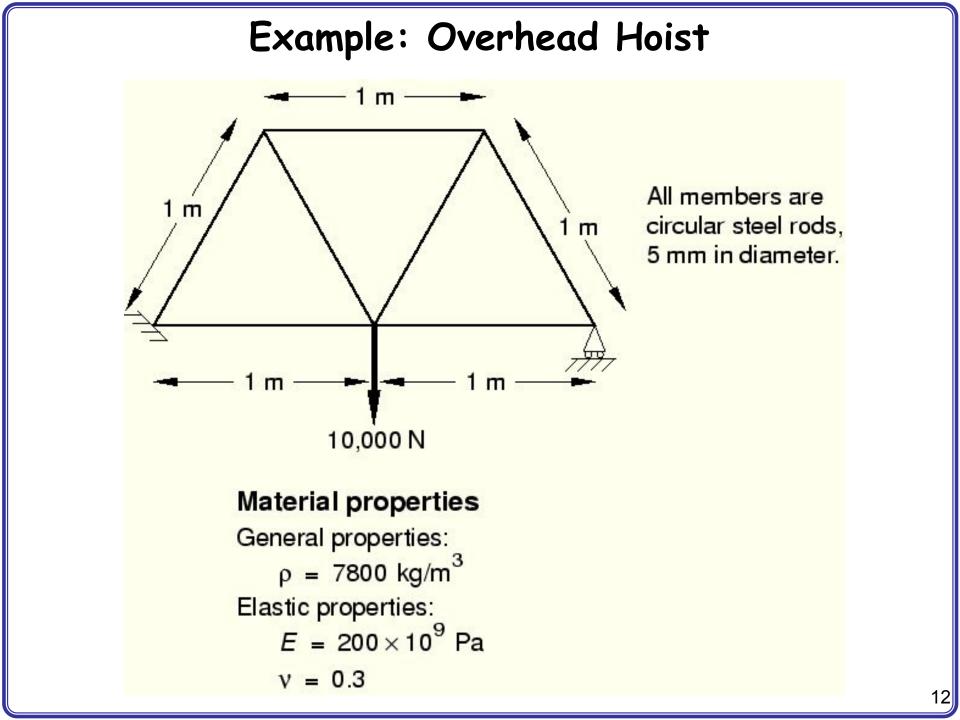
Volume (Solid etement)

- Discretized geometry with solid element
- Discretized BC and load on nodes

Start Abaqus/CAE

Startup window





Units

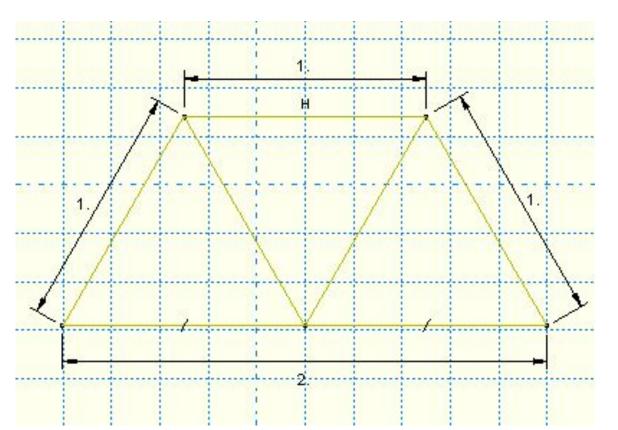
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	Ν	Ν	lbf	lbf
Mass	kg	tonne (10 ³ kg)	slug	lbf s²/in
Time	S	S	S	S
Stress	Pa (N/m²)	MPa (N/mm²)	lbf/ft ²	psi (lbf/in²)
Energy	J	mJ (10 ⁻³ J)	ft lbf	in lbf
Density	kg/m ³	tonne/mm ³	slug/ft ³	lbf s²/in ⁴

• Abaqus does not have built-in units

• Users must use consistent units

Create Part

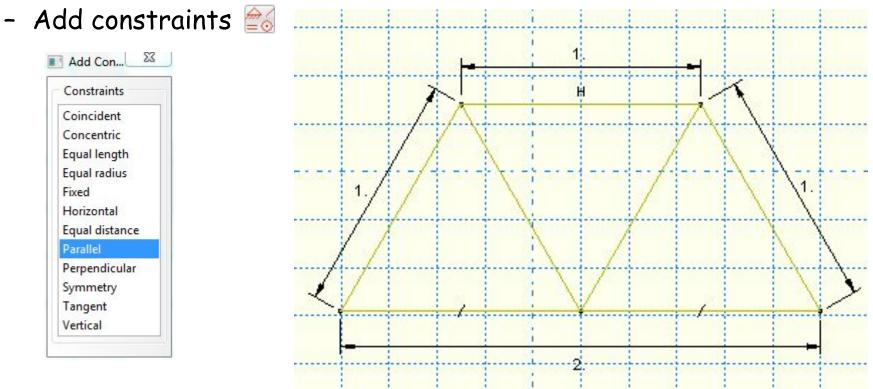
- Parts
 - Create 2D Planar, Deformable, Wire, Approx size = 4.0
 - Provide complete constrains and dimensions
 - Merge duplicate points



Name: Part-1	
Modeling Space	C
🔘 3D 💿 2D Planar	Axisymmetric
Туре	Options
Oeformable	
Discrete rigid	None available
Analytical rigid	None available
🖱 Eulerian	
Base Feature	
Shell	
Wire	
Point	
Approximate size: 4	

Geometry Constraint

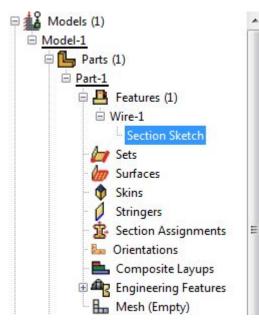
Define exact geometry



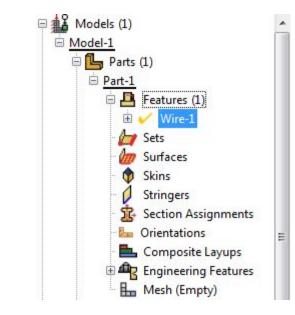
- Add dimension 📝
- Over constraint warning

Geometry Modification

- Modify geometry modeling
 - 1. Go back to the sketch



2. Update geometry



Define Material Properties

- Materials
 - Name: Steel
 - Mechanical Elasticity Elastic

Material Behaviors Elastic		
Elastic General Mechanical Ihermal Qther Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus		Edit
Elastic General Mechanical Ihermal Qther Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus		
General Mechanical Ihermal Other Elastic Type: Isotropic Suboptic Use temperature-dependent data Number of field variables: O Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Ratio Description Poisson's Ratio Poisson's Poison's Poisson's Poisson's<		
Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus		
Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus		
Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus		
Elastic Type: Isotropic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Modulus	al Thermol Other	Dulat
Type: Isotropic Suboptic Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Ratio	ai <u>I</u> hermai <u>O</u> ther	Delet
 Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Ratio 		
 Use temperature-dependent data Number of field variables: 0 Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Ratio 	Subc	ptions
Moduli time scale (for viscoelasticity): Long-term No compression No tension Data Young's Poisson's Ratio		
No compression No tension Data Young's Modulus Ratio	ables: 0 💌	
No compression No tension Data Young's Modulus Ratio	or viscoelasticity): Long-term	
No tension Data Young's Poisson's Modulus Ratio		
Young's Poisson's Modulus Ratio		
Modulus Ratio		
1 200E9 0.3		
	0.3	

Define Section Properties

- Calculate cross-sectional area using CLI (diameter = 5mm)
- Sections
 - Name: Circular_Section
 - Beam, Truss
 - Choose material (Steel)
 - Write area

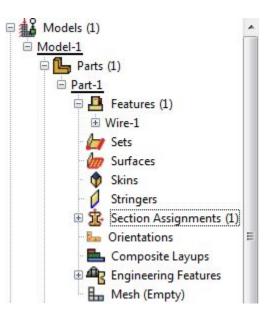
Name: Cir	cular_Section
Category	Туре
Solid	Beam
Shell	Truss
Beam	
Other	
	nue Cancel

Name: C	ircular_Sect	ion		
Type: T	russ			
Material:	Steel		Create	
Cross-sec	tional area:	1.9635E-5		
Temperat	ure variatio	n: Constant	through thickn	less

Define Section Properties

- Assign the section to the part
 - Section Assignments

- Select all wires
- Assign Circular_Section



Section		
Section:	Circular_Section	Create
	ist contains only se pplicable to the sele	
Туре:	Truss	
Material:	Steel	
Region		
Region:	(Picked)	
_	OK	Cancel

Assembly and Analysis Step

- Different parts can be assembled in a model
- Single assembly per model
- Assembly
 - Instances: Choose the frame wireframe
- Analysis Step
 - Configuring analysis procedure
- Steps
 - Name: Apply Load
 - Type: Linear perturbation
 - Choose Static, Linear perturbation

Name: Step-1	
Insert new step a	after
Initial	
Procedure type:	Linear perturbation
Buckle	
вискіе	
Frequency	
Frequency Static, Linear pe	CALCULATION AND A DATA STOCKED
Frequency	CALCULATION AND A DATA STOCKED
Frequency Static, Linear pe	CALCULATION AND A DATA STOCKED
Frequency Static, Linear pe	CALCULATION AND A DATA STOCKED

Assembly and Analysis Step

- Examine Field Output Request (automatically requested)
- User can change the request

Name:	F-Output-1	
Step:	Step-1	
	Static, Linear perturbation	
Domain:	Whole model	
Frequency:	Every n increments n: 1	
Output Va	ariables	
Select f	rom list below 💿 Preselected defaults 🔘 All 🔘	Edit variables
CF,LE,RF,S	,U,	
🕨 🔲 St	tresses	<u>^</u>
🕨 🔳 St	trains	
• D	isplacement/Velocity/Acceleration	E
	prces/Reactions	
) DC	ontact	
Er Er	nergy	
1.0	ailure/Fracture	
🕨 📄 Fa		
Fa	m	
-	m or indicators are not available when Domain is Who	
Note: Erro	or indicators are not available when Domain is Who	
Note: Erro	or indicators are not available when Domain is Who for rebar	
Note: Erro	or indicators are not available when Domain is Who for rebar hell, beam, and layered section points:	
Note: Erro	or indicators are not available when Domain is Who for rebar	
Note: Erro Output f Output at sl	or indicators are not available when Domain is Who for rebar hell, beam, and layered section points:	

Boundary Conditions

- Boundary conditions: Displacements or rotations are known
- BCs
 - Name: Fixed
 - Step: Initial
 - Category: Mechanical
 - Type: Displacement/Rotation
 - Choose lower-left point
 - Select U1 and U2
- Repeat for lower-right corner
 - Fix U2 only

1 m		1 m
1 m	10,000 N dary Condition	— 1 m — —
Name: BC- Type: Disp	1 blacement/Rotation p-1 (Static, Linear pe	
CSYS: (Glo Distribution: VU1: VU2:		ate Create
🔄 UR3:		radians

Applied Loads

• Loads

- Name: Force
- Step: Applied Load
- Category: Mechanical
- Type: Concentrated force
- Choose lower-center point
- *C*F2 = -10000.0

Loads	Create Load		23
	Name: Load-1		
	Step: Step-1		
	Procedure: Static, L	inear perturbation	
	Category	Types for Selected Step	
	 Mechanical Thermal Acoustic Fluid Electrical Mass diffusion 	Concentrated force Moment Pressure Shell edge load Surface traction Body force Line load	ш
Edit Load	© Other	Gravity Pipe pressure Generalized plane strain Cancel	
Name: Load-1 Type: Concentrate Step: Step-1 (Stat Region: (Picked)	ed force ic, Linear perturbation		
CSYS: (Global) Distribution: Uniform	Edit Create n Cre	eate	
CF2: -10000			
Follow nodal rota Note: Force will be			

Meshing the Model

- Parts
 - Part-1, Mesh
- Menu Mesh, Element Types (side menu 🔤)
- Select all wireframes
- Library: Standard
- Order: Linear
- Family: Truss
- T2D2: 2-node linear
 2-D truss

Element Library	Family	
Standard Explicit Geometric Order	Piezoelectric Pipe Thermal Electric	Ē
🖲 Linear 🔘 Quadratic	Truss	
Line		
Element Controls	ling factor: 1	
Linear bulk viscosity sca Quadratic bulk viscosity	scaling factor: 1	

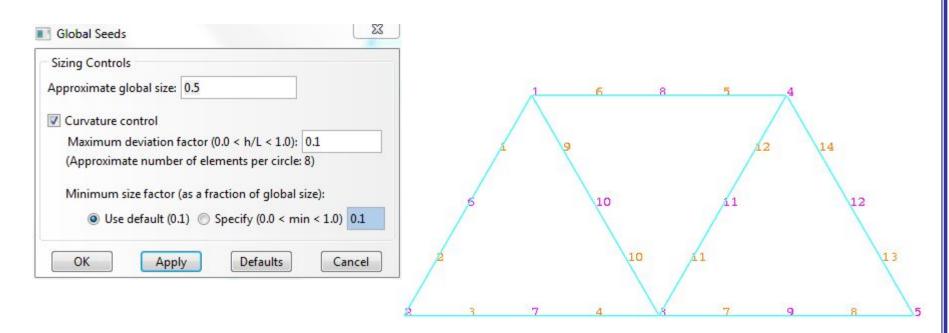
Meshing the Model

- Seed a mesh
 - Control how to mesh (element size, etc)
- Menu Seed, Part (side menu <u>E</u>)
 - Global size = 1.0
- Menu Mesh, Part, Yes (side menu)
- Menu View, Part Display Option
 - Label on

Sizing Controls		
Approximate global size:	1	
Curvature control		
Maximum deviation f	actor (0.0 < h/L < 1.0):	0.1
(Approximate numbe	r of elements per circle:	8)
Minimum size factor ((as a fraction of global s	ize):
() Llas default ()	1) 🔘 Specify <mark>(</mark> 0.0 < mi	n < 1.0) 0.1

Mesh Modification

- Menu Seed, Part (side menu 🏣)
 - Change the seed size (Global size) 1.0 to 0.5
 - Delete the previous mesh
- Menu Mesh, Part, Yes (side menu



Creating an Analysis Job 23 Create Job Name: Truss Source: Model Model-1 Jobs, Truss Data Check Edit Job Name: Truss Continue... Cancel Model: Model-1 Description: Truss under center load Continue (or, submit) Submission General Memory Parallelization Precision Job Type Full analysis Recover (Explicit) Restart Run Mode Host name: Background Queue: Type: Submit Time Immediately O Wait: min. hrs. O At: Tip... OK Cancel

Jobs

-

-

-

Monitor

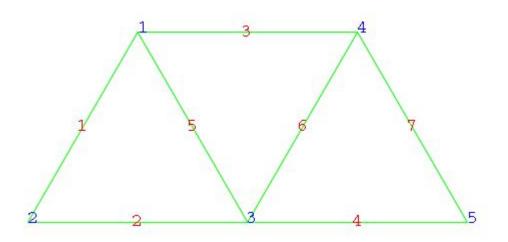
•

27

- Change "Model" tab to "Results" tab
- Menu File, Open Job.odb file
- Common Plot Option (side menu
), click on the Labels tab (Show element labels, Show node labels)

Basic	Color & Style	Labels	Normals	Other
et Fo	nt for All Model	Labels		
/ Sho	w element label	s Colo	r: 💻	
Sho	w face labels	Colo	r: 🔳	
Sho	w node labels	Colo	r: 🔲	
Sho	w node symbols	Colo	r: 🔳	
	Symbol: 🔘	Size:	Small	
	w color code se ions in this dialo		o override	
lote:	Color code sele when contours			le

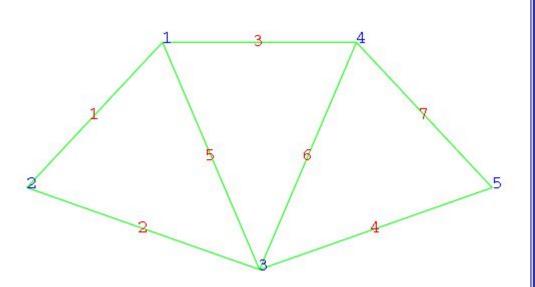
Set Font for All Model Labels...



- Deformation scale
- Common Plot Option (side menu
), click on the Basic tab,

 Deformation Scale Factor area

Basic	Color & Style	Labels	Normals	Other
© W © Fil Defo O Au O Ur	ler Style ireframe () Hid led () Sha rmation Scale Fa uto-compute (42 hiform () Nonu e: (42.84	ded ctor 2.84)	 All Ext Fea Free No 	e Edges edges erior edges ature edges e edges edges Tip



- Tools, XY Data, Manager
 - Position: Integration Point
 - Stress components, S11 (Try with displacements and reaction)

	Operate on XY data
11 (Try	ASCII file
d	Keyboard
	Path
XY Data from ODB Field Output	Cance
Steps/Frames	Cance
Note: XY Data will be extracted from the active steps/frames Activ	ve Steps/Frames
Variables Elements/Nodes	
Output Variables	
Position: Integration Point	
Click checkboxes or edit the identifiers shown next to Edit below.	
E: Strain components	
▼	
Mises	
Max. In-Plane Principal	
Min. In-Plane Principal	
Max. Principal	
Min. Principal	
Tresca	
Pressure	
Third Invariant	
S11	
Edit: S.S11	
Section point: O All O Select Settings	
Save	Dismiss
Plot	DISITISS

Create XY Data

ODB history output
 ODB field output

Source

Thickness

X

- Click on the Elements/Nodes tab
- Select Element/Nodes you want to see result and save
- Click Edit

XY Data Manager Data Source

Name

see l'esuit unu suve	XY Data from OI	ODB Field Output				
Click Edit to see the result	Steps/Frames Note: XY Data will be extracted from the active steps/frames Active Steps/Frames					
	Variables Eleme Selection Method Pick from viewp Element labels Element sets Internal sets	ls I				
XY Data Manager						
Data Source		Create				
Ourrent session Current ODB: Truss.odb		Edit				
lame Description		Сору				
S11 PI: PART-1-1 E: 7 From Field Data: S:S11 at part instance PART-1-1 element 7 integration	n point 1	Rename				
		Delete				
		Plot				
		Copy to ODB ht items in viewport				
		Load to Session				
	1	Dismiss Plot Dismiss				

- Report, Field Output
 - Position: Integration Point
 - Stress components, S11 (Try with displacements and reaction)
 - Default report file name is "abaqus.rpt"
 - The report file is generated in "C:\temp" folder

<u>File Edit Format View H</u> elp	р		
Output sorted by columr	n "Element L	abel".	
Field Output reported a 1-1	at integrati	on points for part: F	PART-
Element Label	Int Pt	5.511 @Loc 1	
1 2 3	1 1 1	-294.041E+06 147.021E+06 -294.041E+06	
4 5 6	1 1 1	147.021E+06 294.041E+06 294.041E+06	
7	1	-294.041E+06	

85 ³ 7.53	ame					
Step: 1,	Step-1					
Frame:	1 Step/F	rame				
Variable	Setup]				
File						
Name:	abaqus.	rpt				Select
🔽 App	end to fil	e				
Outpu	t Format					
·		le table for	all field	output varial	oles	
Layout				field output		
	· · · ·		ior cacin	neid odiput	Variabi	c
Sort by	Node L	abel				
			ending			
0	Ascendin	y o best			100000	
			No lim	nit 🔘 Specify	y: 80	
Page w	idth (cha	racters):	1		y: 80	
Page w Numbe	idth (cha er of signi	racters): @	ts: 6 🚔		y: 80	-
Page w Numbe	idth (cha er of signi	racters):	ts: 6 🚔		y: 80	
Page w Numbe	idth (cha er of signi	racters): @	ts: 6 🚔		y: 80	
Page w Numbe Numbe Data	idth (cha er of signi er format:	racters): @ ficant digit Engineeri	ing ႃ			min/max
Page w Numbe Numbe Data	idth (cha er of signi er format:	racters): @ ficant digit Engineeri	ing ႃ]		min/max

Save

- Save job.cae file
- Menu, File, Save As...
 - job.cae file is saved

job.jnl file is saved as well (user action history, python code)

Save Mo	del Database As					×
Directory:	🚞 Temp	• 🗈	🖾 プ /	• 📥 📴 🖣	. iiii (1
input fil	port les					
<u>F</u> ile Name:	Truss.cae					<u>0</u> K
File F <u>i</u> lter:	Model Database (*.	.cae*)				Cancel