

# EN1740 Computer Aided Visualization and Design

Spring 2012

4/26/2012

Brian C. P. Burke

# Brown University

#### Last time:

 More motion analysis with Pro/E

#### Tonight:

- Introduction to external analysis products
  - ABAQUS



### **External Analysis**

Advanced analysis typically is done in problem-specific packages







MSC Software\* Simulating Reality, Delivering Certainty\*

> Adams Multibody Dynamics

MSC Nastran Accurate, Efficient & Affordable Finite Element Analysis



BCB – Apr. 26, 2012



#### **Structural Analysis**

**Utility and Applications** 

 Structural Analysis is used to predict the response of a solid to an applied load

- Response is typically linked to some performance criteria of component
  - Yield
  - Fatigue
  - Rupture
  - Frequency response
- Predictions need to capture correct physics

• Correct predictions (or "virtual prototyping") can have a huge impact on product development



#### **Structural Analysis**

Key Concepts

- Solids and structures are analyzed in terms of stresses and strains
  - Stress measure of the internal forces acting in a solid
  - Strains describes deformation of a solid
- Stresses and strains provide a mathematical language to quantify material response

# Brown University

### **Structural Analysis**

Finite Element Method

- FEM uses a divide and conquer approach
  - Approximate the geometry using a collection of nodes and elements
  - Satisfy equilibrium on each element
- Displacements are calculated directly
- Stresses and strains are calculated from displacements







#### **Structural Analysis**

Finite Element Method – Process layout

- Establish the geometry (CAD)
- Pre-processing
  - Material definition
  - Discretization (meshing)
  - Establish boundary conditions (loads and displacements)
  - Solver settings
- Solution
- Post-processing
  - Evaluating the results
  - Interpreting the results *MOST IMPORTANT PART!!*



### **Structural Analysis**

Finite Element Method – Packages

• Many CAD systems come with internal analysis software

- Typically very limited
- True mathematics and physics are often "package" to broaden number of potential users
- There are very strong third party packages available that specialize in this type of analysis
  - ABAQUS
  - ANSYS
  - ALGOR
  - ....many others
  - These types of codes provide the largest flexibility and versatility



**Employing Finite Element Method** 

- Using FEM assess whether the pipe section shown can support an internal pressure of 500psi
- Assume:
  - Symmetry can be employed
  - Component is 1018 steel



EN1740, S2012

BCB – Apr. 26, 2012



- Download pipe\_flange.prt from the Supporting Materials page of the web site
- Open the component in Pro/Engineer



BCB – Apr. 26, 2012

# 🛱 Brown University

### **EXERCISE** – Structural Analysis

De-feature CAD model

- We don't need this level of detail to perform the analysis we're interested in
- Best practice >
  - Wherever possible suppress instead of delete
- Suppress chamfers and rounds that are not structural
- Note: Keep Round 2





**De-feature CAD model** 

• Round 2 will definitely have a non-negligible effect on the stress in the component

• Can't get rid of this!





#### Symmetry

- When possible, symmetry can aide an analysis substantially
  - Ease application of BC's
  - Reduces computation time required
- Cut the model along the two datums shown





#### Symmetry

- This should be all that's left
- If you're here, you're ready to export the model

PIPE_FLANGE (Active) - Pro/ENGINEER Education Edition	×
Ele Edit Yew Insert Analysis Info Applications Iools Window Help	
▯ềề฿฿฿฿฿๛๛๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚๚	
<ul> <li>Select start point.</li> <li>When constraint is displayed: Right click to disable it. Press Shift + right click to lock it. Use Tab key to toggle active constraint.</li> <li>Select a sketch. (If an internal sketch is preferred, the "Edit" option can be found in the Placement panel.)</li> </ul>	< 0 >
Bow Setting Setting Setting Setting Setting Set a Setting Set a Se	
Smart	



Export Geometry

- File > Save a Copy...
- Select STEP as the file type
- Keep the same file name

Eile Edit View Insert Analysis Info Applications Iools	: <u>Window</u> Help
New         Ctrl+N                ⊘pen         Ctrl+O           Set Working Directory              ∑lose Window	≗⊑≌m∷· ⊠⊁%९९Q७≮⊄⊜∎ ∎∎□□
Save Contro	Save a Copy
Backup Copy From	G C ← C ≪ inclass_examples → proe → inclass → abaqus → ← 4 Search
Integrate Save a copy of the ac	📲 Organize 🗸 🏢 Views 🗸 🎢 Tools 🗸 😥
Rename Security + Erase + Delete + Instance Operations +	Common Folders     ong       Image: Common Folders     ong       Image: Common Folders     Image: Common Folders       Image: Common Folders     <
Declare	er Network Neighborhood
Print  Quick Print  Seng To  1 T:\Brown_Stuff\\pipe_flange.prt 2 lws://egr-prd-eng-01/spring/layer00.prt 3 T:\Brown_Stuff\teaching\\pixt-crank.asm	
Exit	
L	Folder Tree       Model Name       PIPE_FLANGE.PRT
	Folder Tree     New Name     pipe_flange       Model Name     PIPE_FLANGE.PRT     Type     STEP (*.stp)       Type     STEP (*.stp)     STEP (*.stp)
< > > < >	

BCB – Apr. 26, 2012



Export Geometry

- From the Export STEP options dialog box
  - Un-check Shells
  - Make sure Solids
     are checked
  - Keep all other default preferences
- Click OK

Export STEP
Export
🔲 Wireframe Edges
Surfaces
🗹 Solids
Datum Curves and Points
Facets
Customize Layers
Quilts ALL
Coordinate System
📐 Default 💽
OK Cancel Options





#### **Open ABAQUS**

- ABAQUS is a third party Finite Element Analysis package
- Note the GUI layout > Very similar to Pro/E
  - Model Tree
  - Command Icons
- Click Create Model Database to get started



# Brown University

### **EXERCISE** – Structural Analysis

Import our geometry

- File > Import > Part
- Navigate over to our STEP file
- Click OK

Ele         Model         Viewport         Yiew         Bart         Shape         Feature         Iools         Plug-ins         Help         N?         -           New	T X
: [ New Children Line Childre	2423
VIDTV VIDTV	
Network ODB Connector	
Close ODB	
MC Set Work Directory	
Save As	
Save Options	
Import Sketch	
Export Part	
Run Script Assembly	
Print Ctrl+P	
Abaqus PDE	
<u>1</u> T://abaqus/pipe_flange.cae Import Part	
🛗 ALE Adaptive Mesh Constraint Directory: 🛄 abaqus 🛛 🔛 🟠 🦯 🧖 📅 📴 💷 💷	
🙀 Contact Controls	
Constraints	
⊕ <i>F</i> Fields	
The sector	
Loos	
Predefined Fields	
V Annotations	
回 衾事 Analysis	
Adaptivity Processes	
Eile Name:     pipe_flange.stp	
File Filter: STEP (*.stp*,*.step*)	
2	
	IULIA
A new model database has been created. The model "Model-1" has been created.	
	11

BCB – Apr. 26, 2012



Import our geometry

- Create Part... Dialog pops up
  - On Name Repair dialog check
     Convert to precise representation
- Keep the defaults on the other pages, but look over what is available
- Click OK
  - STEP is loaded
  - This will take a minute

	Create Part from STEP File
Create Part from STEP File	Name - Repair Part Attributes Scale
Create Part from STRP File         Name         Name         Part name:         pipe_flange         Repair Options         Image         Convert to precise representation	Scale  Do not scale  Use transform from file, including scale  Multiply all lengths by 1  Create Part from STEP File  Name - Repair Part Attributes Scale
Solid Shell Wire      Part Filter     Import all parts     Import part number 1	Modeling Space         ③ 3D       2D Planar         Axisymmetric         Type         Options         Deformable         Discrete rigid         Eulerian
	OK Cancel

BCB – Apr. 26, 2012



Get familiar with the interface

• POSSIBLE THE MOST USED FUNCTION – Spin, Pan, Zoom is DIFFERENT!

- For either Spin, Pan or Zoom start by holding the Crtl + Alt Key
- Spin Crtl + Alt + LMB
- Pan Crtl + Alt + MMB
- Zoom Crtl + Alt + RMB
- Try this until comfortable



#### Brown University

### **EXERCISE** – Structural Analysis

Get familiar with the interface

- The ABAQUS/CAE interface is broken up into a number of modules
- Definitions established in these modules set necessary parameters and options for an analysis
- Working through these one at a time makes the setup process very orderly





Part Module

- Importing the STEP file completes everything we need in the part module
- Expand the Model Tree to see the STEP file name
- Take a look at the geometry tools in ABAQUS
  - You can create solid models right in ABAQUS
  - If you know Pro/E, don't do this



BCB – Apr. 26, 2012

### 🗿 Brown University

### **EXERCISE** – Structural Analysis

- Define material properties
  - Material > Create
  - Rename the material 1018-steel
  - For Material Behaviors select Mechanical > Elasticity > Elastic

+ Abaqus/CAE Version 6.8-4 [V	iewport: 1]					
🔄 Eile <u>M</u> odel Vie <u>w</u> port <u>V</u> iew	Material Section	<u>Profile </u> Composit	e <u>A</u> ssign	Specia <u>l</u> Feat <u>u</u>	re <u>T</u> ools Plug	ins <u>H</u> elp
	Manager			<b>-</b>	1000	90C
🎨 Property defaults 🛛 🖉 🔫	Edit 🕨					
Model Results Material Library	Copy	perty 🔽 Mod	el: Model-1	Mart: p	ipe_flange 🔽	
🚝 Model Database 🛛 韋 🛍 🍇	Delete 🕨 🕨					
			<b>4</b> 9			
Edit Material						
Name: steel-1018						
Description:						
Material Behaviors -						
Elastic						
<u>G</u> eneral <u>M</u> echanic	al <u>T</u> hermal	<u>O</u> ther				Delete

#### Brown University

### **EXERCISE** – Structural Analysis

- Define parameters
  - Leave Type as
     Isotropic
  - Enter 29.7e6 psi as Young's Modulus
  - Enter .29 as Poisson's Ratio
- NOTE: ABAQUS does *NOT* keep track of units. It's your responsibility to make sure the units are coordinated

🗖 Edit Material 🛛 🔀
Name: steel-1018
Description:
Material Behaviors
Elastic
General Mechanical Inermal Other Delete
Elastic
Type: Isotropic 🔽 Suboptions
Use temperature-dependent data
Number of field variables: 0 🝧
Moduli time scale (for viscoelasticity): Long-term
No compression
Young's Poisson's
Modulus Ratio
1 29700000 129
OK Cancel





- Define Section properties
  - A Section is a set of parameters that defines how a geometry should be treated
  - Solid, homogeneous, material, etc.
  - Sections are then assigned to geometry
- Name the section steel-1018
- Keep defaults of Solid and Homogeneous
- C<u>lick Continue...</u> BCB – Apr. 26, 2012





- In the next dialog
- Make sure steel-1018 is selected as material
- Un-check Plane
   stress/strain
- Click OK

💻 Edit	Section		
Name: sl Type: S	teel-1018 olid, Homogeneous		
Material:	steel-1018	~	Create
📃 Plane	stress/strain thickness:	1	
	ок	Can	cel

# 🟺 Brown University

### **EXERCISE** – Structural Analysis

#### **Property Module**

- Assign the Section
  - The Section properties must be applied to the geometry
- Assign > Section



EN1740, S2012

BCB – Apr. 26, 2012



**Property Module** 

- Select the region as shown
- The Edit Section Assignment dialog comes up
- Make sure the steel-1018 section is in the Section block
- Click OK



BCB – Apr. 26, 2012



Assembly Module

# • Switch to the Assembly Module

- Create an Instance of the pipe\_flange
  - Instances allow for multiple uses of geometry in assembly
  - If there's only one part, this isn't a meaningful step
- Click on Instance Part icon
- Leave Instance Type as Dependent
- Click OK



BCB – Apr. 26, 2012



Step Module

- Switch to the Step Module
- Create a new Load Step
  - Step > Create
- For Procedure type select Static, General
- Click Continue
- We'll use the defaults for the step > Click OK



BCB – Apr. 26, 2012

# 🟺 Brown University

### **EXERCISE** – Structural Analysis

#### Step Module

- Select what variable should be written to output
  - Field Output Requests - written at the end
  - History Output Requests - written at sub-steps
- Click Field Output Requests > Create
- Use pressure as the name for the results set
- Click Continue...



# 🗿 Brown University

#### **EXERCISE** – Structural Analysis

#### Step Module

- There's a lot of options here let's stay general
  - Domain Whole model
  - Frequency Last increment
  - Use Preselected defaults for the variable selection

Edit Field Output Request				
Name: pressure Step: Step-1 Procedure: Static, General				
Domain: Whole model  Frequency: Last increment Timing: Output at exact times Output Variables Select from list below  Preselected defaults  All  Edit variables				
CDISP,CF,CSTRESS,LE,PE,PEEQ,PEMAG,RF,S,U,				
Note:       Error indicators are not available when Domain is Whole Model or Interaction         Output for rebar       Output at shell, beam, and layered section points:         Output at shell, beam, and layered section points:       Output at shell, beam, and layered section points:         Output defaults       Specify:         Include local coordinate directions when available       OK	٦.			

BCB – Apr. 26, 2012



#### Load Module

- In this module we establish loads and boundary conditions
- BC > Create
- Rename if desired
- Step is Step-1
- Category > Check
   Mechanical
- Types for Selected Step
   Displacement/Rotation
- Click Continue...



BCB – Apr. 26, 2012

# 🗿 Brown University

### **EXERCISE** – Structural Analysis

#### Load Module

- Select the highlighted regions
- Click Done
- Specify the following BC's:
  - Displacement u3=0
  - Rotations about u1, u2=0
- Click OK

Edit Boundary Condition	
Name: BC-1	
Type: Displacement/Rotation 41.4	ecture\inclass_examples\proe\inclass\abaqus\pipe_flange.cae [Viewport: 1]
Step: Step-1 (Static, General)	
Region: (Picked)	
CSYS: (Global) Edit Create	Step: Step-1 V
Distribution: Uniform 🛛 🔽 Create	
U1:	z Contraction and a second sec
U2:	
☑ U3: 0	
UR1: 0 radians	
UR2: 0 radians	
UR3: radians	
Amplitude: (Ramp) 🔽 Create	
Note: The displacement value will be maintained in subsequent steps.	
OK Cancel	
Select regions for the boladary condition Don	
T:\Brown_Stuff\teaching\en174\Lecture\inclass_examples\propriet T:\Brown_Stuff\teaching\en174\Lecture\inclass\propriet T:\Brow	<pre>ce\inclass\abaqus\pipe_flange.cae". ce\inclass\abaqus\pipe_flange.cae". coe\inclass\abaqus\pipe_flange.cae". coe\incl</pre>

BCB – Apr. 26, 2012

### 🗿 Brown University

### **EXERCISE** – Structural Analysis

#### Load Module

- Select the opposite side
- Click Done
- Specify the following BC's:
  - Displacement u2=0
  - Rotations about u1, u3=0
- Click OK

🔲 Edit Bou	undary Condition	×		
Name: BC-2 Type: Disp Step: Step Region: (Pick	2 Ilacement/Rotation o-1 (Static, General) ked)		Lecture\inclass_examples\proe\inclass\abaqus\pipe_flange.cae [Viewport: 1] ls Plug-ins Heb <b>\?</b>	
CSYS: (Glo	obal) Edit) Create Uniform	Create	Step: Step-1	
□ U1:	0		× T	
U3:	0	radians		
UR2:	0 (Ramp)	radians Create		
Note: The di mainta	isplacement value will be ained in subsequent steps.			
	К Са Хеззее Ух]	ncel		7
T: Brown_ The model "T: Norown_ The model "T: Norown_ The model	Stuff teaching en174 Lecture Stuff teaching en174 Lecture to Stuff teaching en174 Lecture to Stuff teaching en174 Lecture Stuff teaching en174 teaching en17	s for the boundary condition inclass_examples inclass_examples inclass_examples	n Done Sets proe\inclass\abaqus\pipe_flange.cae". .proe\inclass\abaqus\pipe_flange.cae". .proe\inclass\abaqus\pipe_flange.cae".	PSIMUL ^



#### Load Module

- Select the flange face
- Click Done
- Specify the following BC's:
  - Displacement u1=0
  - Rotations about u2, u3=0
- Click OK

🗖 Edit Boı	undary Condition			
Name: BC-:	3		174\Lecture\inclass_examples\proe\inclass\abaqus\pipe_flange.cae [Viewport: 1]	
Type: Disp	lacement/Rotation		] = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 =	
Step: Step	p-1 (Static, General)		Step: Step-1	
Region: (Pic	ked)		Y	
CSYS: (G1	obal) Edit Create			
Distribution:	Uniform 💌	Create	z	
🗹 U1:	0			
U2:				
🔲 U3:				
UR1:		radians		
🔽 UR2:	0	radians		
UR3:	0	radians		
Amplitude:	(Ramp) 🔽	Create		
Note: The d maint	lisplacement value will be ained in subsequent steps.	ncel	y x	
<u> </u>				
T: Bro The moo "T: Sho "T: Sho "T: Sho "T: Sho	wn_Stuff\teaching\en174\Lectur del database has been saved to swn_Stuff\teaching\en174\Lectur del database has been saved to swn_Stuff\teaching\en174\Lectur	e\inclass_example e\inclass_example e\inclass_example	es\proe\inclass\abaqus\pipe_flange.cae". es\proe\inclass\abaqus\pipe_flange.cae". es\proe\inclass\abaqus\pipe_flange.cae".	<



#### Load Module

- Create a pressure load on the inside of the pipe
- Load > Create
- Select Pressure for Types..
- Click Continue...





Load Module

- Select the face shown
- Click Done
- Use 500 for Magnitude
- Click OK
- SAVE!



EN1740, S2012

BCB – Apr. 26, 2012



Mesh Module

- Switch to Mesh module
- Toggle to operate on the Part rather than Instance





#### Mesh Module

- Prior to creating mesh, we need to tell ABAQUS approximately how big the elements should be
- Seed > Part
- Enter .2 as Approximate global size
- Click OK



BCB – Apr. 26, 2012



#### Mesh Module

- We need to select the element type
- Mesh > Element Type
- Select >
  - Standard
  - 3D Stress
  - Quadratic
  - Tet
    - Uncheck formulation options
- Click OK

	+ Abaqus/CAE Version 6.8-4 - Model Database: T:\Brown_Stuff\teaching\en174\Lectur
	Ele Model Viewport View Seed Mesh Adaptivity Feature Iools Plug-ins Help M
Element Type	Element Type
Element Library Family	Part
Standard C Explicit 3D Stress	Delete Part Mesh
Acoustic	Delete Region Mesh
Cohesive	Agsociate Mesh with Geometry
	Edit
x Wedge Tet	Verify
Element Controls	
Hybrid formulation	
Modified formulation	
Distortion control:   Use default  Yes  No	s (1) R <sub>3</sub> C
Length ratio: 0.1	straints
ear bulk viscosity scaling factor: 1	(XY2) ‡
uadratic bulk viscosity scaling factor: 1	「「「「「」」「「」」
010: A 10-node quadratic tetrahedron.	1 m.
ote: To select an element shape for meshing,	
select "Mesh->Controls" from the main menu bar.	
OK Defaults	Cancel

# 🛱 Brown University

### **EXERCISE** – Structural Analysis

#### Mesh Module

- We need to tell ABAQUS to use this element type
- Mesh > Controls...
- Click Tet
- Click OK





#### Mesh Module

- Create the mesh
- Mesh > Part
- Click Yes



BCB – Apr. 26, 2012



#### Mesh Module

• Take a look at the mesh we just created





EN1740, S2012

BCB – Apr. 26, 2012



#### Job Module

- This module will create an input file and submit it to a solver
- Job > Create
- Click Continue
- Accept defaults and click OK

+ Abaqus/CAE Version 6.8-4 - Model Database: T:\Brow	vn_Stuff\teaching\en174\Lecture\inclass_examples\proe\inclass\abaqus\pipe_flange.cae [Viewport: 1]										
Ele Model Viewport View Job Adaptivity Tools F	Nug-ins Help <b>K</b> ?	- 8 ×									
Model Results	Edit Job										
Copy     ►       Copy     ►       Rename     ►       Bail     Models (1)	Name: Job-1										
	Model: Model-1										
Create Job	Description: [										
Name: Job-1	Submission General Memory Parallelization Precision										
Source: Model 🔽	Job Type										
	• Full analysis										
Model-1											
	Restart										
	C Run Mode										
	Background      Queue: Host name:     Type:										
	⊂ Submit Time										
	<ul> <li>Immediately</li> </ul>										
Continue Cancel	🔿 Wait: 🚺 hrs. 🚺 min.										
12635 elements have been generated on	O At:										
Warning: Some of the mid-edge nodes we geometric boundary, in order The model database has been saved to "T:\Brown_Stuff\teaching\en174\Lecture	OK Cancel										

BCB – Apr. 26, 2012



#### Job Module

- Solve the model
- Job > Submit > Job-1





#### Job Module

• Hopefully you see this in the message frame



# 🗿 Brown University

### **EXERCISE** – Structural Analysis

#### Visualization Module

- In this module you can visualize the results just computed
- File > Open
- Select Job-1.odb

÷	baqus/CAE \	/ersion 6	. 8-4 - Mo	del Datab	ase: T:\B	rown_S	tuff\teac	hing\en	174\Lec	ture\ind	lass_ex	amples\	proe\in	c lass \ab	aqus\pi	pe_flang	e.cae [Vie	wport: 1]	
	<u>File M</u> odel	Vie <u>w</u> port	<u>V</u> iew <u>R</u> e	sult <u>P</u> lot	<u>A</u> nimate	Report	Options	<u>T</u> ools	Plug-ins	Help	1?								_ @ X
I C	New		Chillion		:目∄	5	All	~			0	ØG	000		00	DOB			
: 20	Network OE	B Connecto	or F			100													
	⊆lose ODB.	i		Modula	. Vieuslina	tion w	ODR	_											
Mc	Set <u>W</u> ork D	rectory	100000000000		· visualiza		008:	_	×	_	_	_	_	_	_	_	_	_	_
Ses	<u>S</u> ave		Ctrl+S	123	23														
đ	Save As		Jacob	Datab															
(±	Import		open	Datab	ase														
	Export			-							_	,							
[	<u>R</u> un Script.	Dire	ectory:	🧰 Te	mp			1	/ 🖻	] 🖪	· プ	- 🔶	eř	0-0- 0-0-	<u>ا</u> و	<b>i</b> n			
	Print	-						_											
I	Abagus PDB																		
10	<u>1</u> T://aba			-															
	E⊻it		abaqu	>															
			vic_pla	iyer															
			Job-1.	odb															
		-															014		
		Elle	Name:	Job-	l.odb												<u>o</u> k		
		ci-	Eilberr	Cours-	uh Dah		/* - JI	L#\							40-1		Consol		
		File	Filter:	Outp	uc Dat	abase	(*.odi	0")						Rea		۷ 🕒			7
																			DS
	The job	"Job-1	" has b	een cre	ated.														~
1	The job	input	file "J	ob-1.in	o" has Le Proc	been s	ubmitte	d for	analys	is.									
>>	Job Job	-1: Aba	qus/Sta	ndard ç	mplete	d succ	essfull	.y.											
-	-liop lop	-r comp	neted s	uccessi	illy.														<b>M</b>

BCB – Apr. 26, 2012



#### **Visualization Module**

• Click the contour button to see the results



BCB – Apr. 26, 2012