EN1740 Computer Aided Visualization and Design

Spring 2012

4/26/2012

Brian C. P. Burke
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

Moldflow

ABAQUS

MSC Software

Adams
Multibody Dynamics

MSC Nastran
Accurate, Efficient & Affordable Finite Element Analysis

ANSYS

LS-DYNA
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
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
**EXERCISE - Structural Analysis**

*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
EXERCISE – Structural Analysis

• Download pipe_flange.prt from the Supporting Materials page of the web site
• Open the component in Pro/Engineer
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
EXERCISE – Structural Analysis

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!
**EXERCISE – Structural Analysis**

**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
EXERCISE – Structural Analysis

Symmetry

• This should be all that’s left

• If you’re here, you’re ready to export the model
**EXERCISE – Structural Analysis**

*Export Geometry*

- File > Save a Copy…
- Select STEP as the file type
- Keep the same file name
**EXERCISE – Structural Analysis**

*Export Geometry*

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

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
EXERCISE – Structural Analysis

Import our geometry

- File > Import > Part
- Navigate over to our STEP file
- Click OK
EXERCISE – Structural Analysis

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
EXERCISE – Structural Analysis

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
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
EXERCISE – Structural Analysis

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
EXERCISE – Structural Analysis

Property Module

• Define material properties
  • Material > Create
  • Rename the material 1018-steel
  • For Material Behaviors select Mechanical > Elasticity > Elastic
EXERCISE – Structural Analysis

Property Module

- Define parameters
  - Leave Type as Isotopic
  - 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
EXERCISE – Structural Analysis

Property Module

• 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
• Click Continue…
EXERCISE – Structural Analysis

Property Module

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

Property Module

• Assign the Section
  • The Section properties must be applied to the geometry
• Assign > Section
**EXERCISE – Structural Analysis**

*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
EXERCISE – Structural Analysis

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
**EXERCISE – Structural Analysis**

**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
**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…
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
EXERCISE – Structural Analysis

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…
EXERCISE – Structural Analysis

Load Module

• Select the highlighted regions
• Click Done
• Specify the following BC’s:
  • Displacement $u_3=0$
  • Rotations about $u_1, u_2=0$
• Click OK
EXERCISE – Structural Analysis

Load Module

• Select the opposite side
• Click Done
• Specify the following BC’s:
  • Displacement $u_2=0$
  • Rotations about $u_1$, $u_3=0$
• Click OK
EXERCISE – Structural Analysis

Load Module

• Select the flange face
• Click Done
• Specify the following BC’s:
  • Displacement $u_1 = 0$
  • Rotations about $u_2, u_3 = 0$
• Click OK
EXERCISE – Structural Analysis

Load Module

• Create a pressure load on the inside of the pipe
  • Load > Create
  • Select Pressure for Types..
  • Click Continue…
EXERCISE – Structural Analysis

Load Module

- Select the face shown
- Click Done
- Use 500 for Magnitude
- Click OK
- SAVE!
**EXERCISE – Structural Analysis**

*Mesh Module*

- Switch to Mesh module
- Toggle to operate on the Part rather than Instance
EXERCISE – Structural Analysis

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
**EXERCISE – Structural Analysis**

**Mesh Module**

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

**Mesh Module**

- We need to tell ABAQUS to use this element type
- Mesh > Controls...
- Click Tet
- Click OK
**EXERCISE – Structural Analysis**

*Mesh Module*

- Create the mesh
- Mesh > Part
- Click Yes
EXERCISE – Structural Analysis

Mesh Module

• Take a look at the mesh we just created
• Save
EXERCISE – Structural Analysis

Job Module

• This module will create an input file and submit it to a solver

• Job > Create

• Click Continue

• Accept defaults and click OK

[Software Interface Image]
**EXERCISE** – Structural Analysis

**Job Module**

- Solve the model
- Job > Submit > Job-1
EXERCISE – Structural Analysis

Job Module

• Hopefully you see this in the message frame
EXERCISE – Structural Analysis

Visualization Module

• In this module you can visualize the results just computed
  • File > Open
  • Select Job-1.odb
EXERCISE – Structural Analysis

Visualization Module

• Click the contour button to see the results