ASEE Introduction to Abaqus Workshop

What to do to get started

Open Abaqus 6.14

You can close the 3D mouse window

Choose this option
What we will cover today

• What is Finite Element Analysis
• What can we use Finite Element Analysis for
• Overview of how to navigate Abaqus
• Overview of the basic procedure for performing Finite Element Analysis
• Specific instructions for performing static structural analysis
What is the Finite Element Method?

The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

Definitions

Elements: smaller simpler parts in with equations are solved over
Nodes: corners of the elements
Mesh: pattern in which you subdivide your problem into
What can you do with FEA?

- Modal Analysis
- Thermal – Fluid
- Impact
- Thermal – Mechanical
- Composites - Dynamic
- Piezoelectric - Acoustic
Element Types

1D
- Beams
  - 2-noded
  - 3-noded

2D
- Triangles
  - 3-noded
  - 6-noded
- Quadrilaterals
  - 4-noded
  - 8-noded

3D
- Tetrahedrons
  - 4-noded
  - 10-noded
- Hexahedrons
  - 8-noded
  - 20-noded
- Pentahedrons
  - 6-noded
  - 15-noded
Simplifying Models

[Diagram showing the simplification of a structural model]
Simplifying Models
Meshes

- Coarse mesh (14 elements)
- Normal mesh (112 elements)
- Fine mesh (448 elements)
- Very fine mesh (1792 elements)

Graph:

- Normalized Result
- Stress at attachment
- Stress on hole
- Displacement of hole

Relative Mesh Density
Meshes
Module

Model Tree

Prompt Area

Message Area
Detailed Procedure

• Part: Create the Part
  • Property
    • Create the Materials
    • Create the Section
    • Assign the Section to the Part
  • Assembly: Create the Assembly

• Step: Define the Analysis Type

• Interaction: Define how different parts interact within an assembly (if there is only one part this will be skipped)

• Load
  • Define Loads
  • Define Boundary Conditions

• Mesh: Create the Mesh

• Job: Run the analysis

• Visualization: View the results
# Units in Abaqus

Abaqus does not have units, therefore you must use consistent units

<table>
<thead>
<tr>
<th>Quantity</th>
<th>SI</th>
<th>SI (mm)</th>
<th>US Unit (ft)</th>
<th>US Unit (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>m</td>
<td>mm</td>
<td>ft</td>
<td>in</td>
</tr>
<tr>
<td>Force</td>
<td>N</td>
<td>N</td>
<td>lbf</td>
<td>lbf</td>
</tr>
<tr>
<td>Mass</td>
<td>kg</td>
<td>tonne (10^3 kg)</td>
<td>slug</td>
<td>lbf s^2/in</td>
</tr>
<tr>
<td>Time</td>
<td>s</td>
<td>s</td>
<td>s</td>
<td>s</td>
</tr>
<tr>
<td>Stress</td>
<td>Pa (N/m^2)</td>
<td>MPa (N/mm^2)</td>
<td>lbf/ft^2</td>
<td>psi (lbf/in^2)</td>
</tr>
<tr>
<td>Energy</td>
<td>J</td>
<td>mJ (10^-3 J)</td>
<td>ft lbf</td>
<td>in lbf</td>
</tr>
<tr>
<td>Density</td>
<td>kg/m^3</td>
<td>tonne/mm^3</td>
<td>slug/ft^3</td>
<td>lbf s^2/in^4</td>
</tr>
</tbody>
</table>
Other Abaqus Tips

• Files
  • .CAE is a file that contains everything needed to run the analysis
  • .ODB is the file created while the analysis is being run and contains the results of the simulation

• View the part
  • Ctrl + Alt + Left Button: Rotate Part
  • Ctrl + Alt + Center/Scroll Button: Translate Part
Example #1

\[ p = 10 \text{ N/m} \]

\[ l = 2m \]

\[ h = 1m \]

\[ R = 0.25m \]
Example #2

\[ L = 1 \text{ m} \]

\[ P = 10 \text{ N} \]

\[ t = 0.01 \text{ m} \]

\[ h = 0.07 \text{ m} \]

\[ w = 0.1 \text{ m} \]
References

Slide 3
https://www.comsol.com/multiphysics/fea-software

Slide 4
http://www.creocommunity.com/print_article.php?cpfeatureid=8763
http://www.cerom.lsu.edu/serhan.htm
http://www.mdpi.com/1424-8220/14/9/17174/htm
https://www.simuleon.com/simulia-abaqus/abacus-explicit/
http://www.iitrpr.ac.in/sites/default/files/navin_web/gallery.html

Slide 5
http://illustrations.marin.ntnu.no/structures/analysis/FEM/theory/index.html

Slide 6
http://article.sapub.org/10.5923.j.mechanics.20130305.02.html

Slide 7
http://solidmechanics.org/Text/Chapter7_1/Chapter7_1.php
http://plmsource.industrysoftware.automation.siemens.com/blog/2013/03/plysim-automates-composite-fe-modeling-reduces-design-time/

Slide 8
http://www.egr.msu.edu/software/abaqus/Documentation/docs/v6.7/books/gsa/default.htm?startat=ch04s04.html

Slide 9
http://www.egr.msu.edu/software/abaqus/Documentation/docs/v6.7/books/gsa/default.htm?startat=ch04s04.html
http://confluence.diamond.ac.uk/display/EDG/Design+of+flexural+hinges

Slide 12
https://www.researchgate.net/post/can_someone_help_me_with_on_Abaqus_CAE_data_and_measurement_units