An Introduction to Finite Element Analysis
With Emphasis on Altair HyperWorks Suite Applications for FSAE

December 3\textsuperscript{rd}, 2012

Billy Wight
President
Luxon Engineering
What is FEA?

Finite Element Analysis is not a black box!

• Too often FEA is regarded as quick to do, or simple, but in reality it is quite complex
• Today’s software has made the FEA simple to use which makes it easy to make mistakes without knowing!
  • SolidWorks is a great example…

• The analysis results are only as good as the engineer performing the analysis
What is FEA? (Cont.)

Finite Element Analysis (FEA) is a numerical technique of obtaining solutions to the differential equations that describe or approximate a physical problem.

- FEA uses the finite element method (FEM) to discretize a region (CAD model) into many smaller regions (elements).
- Each element is joined to adjacent elements at points (nodes). Loads and boundary conditions are applied to the nodes to represent the problem to be solved.
- Differential equations are created at each element and approximately solved. The assembly of all the equations solutions describes the behavior of the entire region.
What is FEA? (Cont.)

- The Finite Element Model is discretized onto smaller pieces – Elements, who's vertices become the unknowns in the equations to be solved
- Displacements are calculated at the nodes
- Stresses and Strains are calculated within the elements
  - Shape functions – The nodal displacements are integrated with respect to the shape functions within the elements to determine stresses/strains
  - Element quality is critical to correct results, particularly stresses/strains
Types of Analysis

Many applications of FEA/FEM:

• Structural Linear Static
• Structural Quasi-Dynamic and Dynamic
• Modal (Frequency) and Buckling
• Fatigue
• Computational Fluid Dynamics (CFD)
• Thermal and Heat Transfer
• Electromagnetic (EMS)
• Rigid-Body Dynamics
  • Dynamic motion solvers, not really FEM, but a fairly typical engineering analysis
• Combinations of the above
  • Fluid-Structure interactions (CFD+FEA)
  • Flexible-Body Dynamics (Structural Analysis + Rigid-Body Dynamics)
  • Others…
• Optimization Studies of any of the above
What is Linear Static?

The most common type of analysis you will perform

- **Linear:**
  - Material linearity - linear elastic, properties do not change with respect to strain, strain rate, etc.
    - Most metals, some plastics
  - Deformations must be linear
    - Must have small deformations and small rotations
    - Most not have snap-through response (buckling)
  - Boundary conditions must be constant
    - No contact
    - No sliding
    - No friction

- **Static:**
  - Boundary conditions do not change in time
    - Loading and constraints are always constant
  - Loading must be assumed to be applied slowly
    - No inertial effects
Math (Linear Static)

Equations representing the problem are dependent on the problem…

- A linear static analysis (majority of structural analysis) is of the form:

\[ \{ F \} = [K]\{X\} \]

where \( \{ F \} \) is a vector representation of loads on the model (known)

- Solution involves inverting the stiffness matrix to solve for \( \{X\} \), this takes up the majority of the computational time

- An infinite number of geometrically possible displacement solutions exist, How does it calculate the correct one?
  - Minimization of potential energy:
    - The displacement solution that minimizes the difference between the internal strain energy of the model and the external work done on the model by the applied loads is the correct solution.

Elemental stiffness matrix equation corresponding to the minimum potential energy of the element:

\[ [K_e] = \iiint_{V_e} [N](x, y, z)^T \cdot [B]^T \cdot [C] \cdot [B] \cdot [N](x, y, z) \, dV_e \]
What is Modal Analysis?

Really should be done on all linear static models

- Determines the natural frequencies of the system
  - If your component operates near a natural frequency, linear static assumption does not apply!
    - Example: A brake rotor requires structural analysis, but a modal analysis shows a natural frequency within the operating range at 18 Hz (18 Hz ≈ 64 mph for a FSAE wheel)
      - Linear static does not apply and dynamic analysis must be preformed!

- Can be used to debug an under-constrained model:
  - First 6 modes are rigid body motion
  - Modal analysis can resolve issues with missing constraints
Math (Modal)

An undamped modal (frequency) analysis is of the form:

\[
[M]\{\ddot{X}\} + [K]\{X\} = 0
\]

where \([M]\) is a matrix representation of the mass and loads on the model (known)

\([K]\) is a square matrix of nodal stiffnesses (known)

\(\{X\}\) is a vector of displacements (unknown)

\(\{\ddot{X}\}\) is a vector of accelerations (unknown)

The Solution assumes:

\(\{X\} = \{X_o\} \sin(\omega \cdot t)\)

so that

\(\{\ddot{X}\} = -\omega^2 \{X_o\} \sin(\omega \cdot t)\)

The solution for the \(i^{th}\) natural frequency can be written as:

\([K]\{X_i\} - \omega_i^2[M]\{X_i\} = 0\)

where \(\{X_i\}\) is the \(i^{th}\) mode shape and \(\omega_i\) is the corresponding natural frequency.
CFD, Rigid Body Dynamics, Optimization

- Beyond the scope of this presentation, but worth mentioning
- (Flexbody and Optimizations Examples)
General Analysis Structure

Pre-Processing
  • Geometry, Materials, Mesh, Boundary Conditions

Solving
  • Solve the driving equations

Post-Processing
  • View and interpret the results
Pre-Processing: Geometry

Geometry Definition
- Used to create the mesh (nodes and elements)
- Typically imported from a CAD program
  - SolidWorks, Pro/Engineer, Catia, Unigraphics, etc.
- Can be created within the analysis software
  - Usually a very tedious process

Geometry Cleanup
- Unnecessary model details are removed to simplify meshing
  - Small holes, fillets and chamfers, shared edge removal, sliver surface removal
- Can be done within the FE program or before CAD import
Pre-Processing: Material

Material Definition

- Various material properties need to be defined
  - Analysis Dependent…
    - Linear Static
      - Young’s Modulus (E)
      - Poisson’s Ratio (ν)
      - Density (ρ)
  - Material properties available online:
    - www.matweb.com
  - Physical testing
    - Most accurate
    - Unavailable material data
Pre-Processing: Meshing

Element Types

- **0D**
  - Point Masses, Joints

- **1D**
  - Bars (Truss), Beams, Rigids, Springs, etc.

- **2D (Plate and Shell)**
  - Tria, Quad

- **3D (Solid)**
  - Tetra, Penta, Hexa
Processing

- Solves the driving equations
  - Equations are iteratively solved until convergence
    - Convergence criteria can be specified
  - Fully automated (for the most part)
  - Inputs are typically computational related
    - RAM allocation
    - Number of processors
    - Scratch disk location
Post-Processing

• View and interpret the results
  • Many plotting options are available
    • Deformation (nodal displacement), Stress, Strain, etc.
    • Contour plots, tensor plots, etc.
    • Iso-clipping, Planar clipping, etc.
  • Lots of analysis type specific options
    • Damage plot (fatigue)
    • Pathlines (CFD)
    • Mode Shape (modal)
Pre-Processing: Meshing – Cantilever Example

- Cantilever arm model
  - Left end fixed
  - Right end 10N force downward

  - Can model using 1D (Beam), 2D (Shell), or 3D (Solid) elements
  - Results can vary significantly based on element type, configuration, and number
Pre-Processing: Meshing – Cantilever Example (Cont.)

- Expected Results (Shown using 3D elements)
  - Tensile stress on top
  - Compressive stresses on the bottom
  - Neutral plane at the centre of the model
  - Highest stresses at fixed end decreasing linearly to end with applied force
Pre-Processing: Meshing – Cantilever Example (Cont.)

- Beam theory hand calculation predicts 0.500 mm deflection
- Results for 3 element types show good correlation to expected result

![Beam deflection diagrams](image-url)
Pre-Processing: Meshing – Cantilever Example (Cont.)

- Results can vary significantly for different element configurations and densities
- It is important to have the correct setup for the analysis you are preforming as it is easy to get bad results

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Order</th>
<th>Size Across Thickness</th>
<th>Total Elements</th>
<th>Total Nodes</th>
<th>Maximum Deformation (mm)</th>
<th>Deformation Error</th>
<th>Stress (MPa), Elements, 58mm In</th>
<th>Elemental Stress Error</th>
<th>Descriptions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.500</td>
<td>0.0%</td>
<td>13.61</td>
<td>0.0%</td>
<td>Beam Theory</td>
</tr>
<tr>
<td>1D BEAM 1st</td>
<td>3</td>
<td></td>
<td>67</td>
<td>158</td>
<td>0.500</td>
<td>0.1%</td>
<td>-</td>
<td>-</td>
<td>Altair 1D Demo (Tensile/Comp Stresses Only)</td>
</tr>
<tr>
<td>2D QUAD 1st</td>
<td>3</td>
<td></td>
<td>356</td>
<td>686</td>
<td>0.495</td>
<td>-0.9%</td>
<td>13.90</td>
<td>2.1%</td>
<td>Altair 2D Demo</td>
</tr>
<tr>
<td>3D HEXA 1st</td>
<td>3</td>
<td>6</td>
<td>3,216</td>
<td>4,304</td>
<td>0.494</td>
<td>-1.1%</td>
<td>11.59</td>
<td>-10.7%</td>
<td>Altair 3D Demo</td>
</tr>
<tr>
<td>2D QUAD 2nd</td>
<td>3</td>
<td>N/A</td>
<td>356</td>
<td>1,893</td>
<td>0.514</td>
<td>2.9%</td>
<td>13.90</td>
<td>2.1%</td>
<td>Altair 2D Demo (2nd Order)</td>
</tr>
<tr>
<td>3D HEXA 2nd</td>
<td>3</td>
<td>6</td>
<td>3,216</td>
<td>16,005</td>
<td>0.505</td>
<td>1.9%</td>
<td>11.59</td>
<td>-10.7%</td>
<td>Altair 3D Demo (2nd Order)</td>
</tr>
<tr>
<td>3D HEXA 1st</td>
<td>1.667</td>
<td>3</td>
<td>5,760</td>
<td>8,228</td>
<td>0.495</td>
<td>-0.9%</td>
<td>11.59</td>
<td>2.1%</td>
<td>Altair 3D Demo</td>
</tr>
<tr>
<td>3D HEXA 2nd</td>
<td>1.667</td>
<td>3</td>
<td>5,760</td>
<td>30,303</td>
<td>0.508</td>
<td>1.7%</td>
<td>9.28</td>
<td>18.8%</td>
<td>Altair 3D Demo (2nd Order)</td>
</tr>
<tr>
<td>3D HEXA 1st</td>
<td>0.833</td>
<td>6</td>
<td>43,200</td>
<td>52,297</td>
<td>0.495</td>
<td>-0.9%</td>
<td>11.37</td>
<td>-16.5%</td>
<td>Altair 3D Demo Perfect Elements</td>
</tr>
<tr>
<td>3D HEXA 2nd</td>
<td>0.833</td>
<td>6</td>
<td>43,200</td>
<td>199,813</td>
<td>0.502</td>
<td>0.5%</td>
<td>11.37</td>
<td>-16.5%</td>
<td>Altair 3D Demo Perfect Elements (2nd Order)</td>
</tr>
<tr>
<td>3D HEXA 1st</td>
<td>3</td>
<td>1</td>
<td>536</td>
<td>1,224</td>
<td>0.495</td>
<td>-0.9%</td>
<td>9.29</td>
<td>-18.8%</td>
<td>Altair 3D Demo - Bad</td>
</tr>
<tr>
<td>3D HEXA 2nd</td>
<td>3</td>
<td>1</td>
<td>536</td>
<td>4,130</td>
<td>0.508</td>
<td>1.7%</td>
<td>9.28</td>
<td>-18.8%</td>
<td>Altair 3D Demo - Bad (2nd Order)</td>
</tr>
<tr>
<td>3D TETRA 1st</td>
<td>3</td>
<td>1</td>
<td>3,216</td>
<td>1,224</td>
<td>0.160</td>
<td>-88.0%</td>
<td>4.05</td>
<td>-88.0%</td>
<td>Altair 3D Demo - Bad</td>
</tr>
<tr>
<td>3D TETRA 2nd</td>
<td>3</td>
<td>1</td>
<td>3,216</td>
<td>6,885</td>
<td>0.500</td>
<td>0.1%</td>
<td>6.97</td>
<td>-88.0%</td>
<td>Altair 3D Demo - Bad (2nd Order)</td>
</tr>
<tr>
<td>3D TETRA 1st</td>
<td>1.667</td>
<td>3</td>
<td>34,360</td>
<td>8,228</td>
<td>0.355</td>
<td>-13.6%</td>
<td>10.22</td>
<td>-13.6%</td>
<td>Altair 3D Demo - Good</td>
</tr>
<tr>
<td>3D TETRA 2nd</td>
<td>1.667</td>
<td>3</td>
<td>34,560</td>
<td>5,671</td>
<td>0.506</td>
<td>1.3%</td>
<td>11.67</td>
<td>-14.7%</td>
<td>Altair 3D Demo - Good (2nd Order)</td>
</tr>
<tr>
<td>3D TETRA 2nd</td>
<td>5.8501</td>
<td>1</td>
<td>1,327</td>
<td>2,572</td>
<td>0.493</td>
<td>-13.9%</td>
<td>7.00</td>
<td>-18.6%</td>
<td>SolidWorks Coarse Settings</td>
</tr>
<tr>
<td>3D TETRA 2nd</td>
<td>2.9251</td>
<td>2</td>
<td>8,190</td>
<td>14,147</td>
<td>0.495</td>
<td>-0.9%</td>
<td>10.30</td>
<td>-2.6%</td>
<td>SolidWorks Default Settings</td>
</tr>
<tr>
<td>3D TETRA 2nd</td>
<td>1.4625</td>
<td>3</td>
<td>58,004</td>
<td>88,969</td>
<td>0.495</td>
<td>-0.9%</td>
<td>12.50</td>
<td>-8.2%</td>
<td>SolidWorks Fine Settings</td>
</tr>
<tr>
<td>3D TETRA 1st</td>
<td>5.8501</td>
<td>1</td>
<td>1,327</td>
<td>4,443</td>
<td>0.169</td>
<td>-88.2%</td>
<td>5.10</td>
<td>-88.2%</td>
<td>SolidWorks Coarse Settings - Draft Quality</td>
</tr>
<tr>
<td>3D TETRA 1st</td>
<td>2.9251</td>
<td>2</td>
<td>8,190</td>
<td>2,212</td>
<td>0.307</td>
<td>-19.5%</td>
<td>7.50</td>
<td>-19.5%</td>
<td>SolidWorks Default Settings - Draft Quality</td>
</tr>
<tr>
<td>3D TETRA 1st</td>
<td>1.4625</td>
<td>3</td>
<td>58,004</td>
<td>12,598</td>
<td>0.416</td>
<td>-16.7%</td>
<td>12.90</td>
<td>-5.2%</td>
<td>SolidWorks Fine Settings - Draft Quality</td>
</tr>
</tbody>
</table>
**Sources of Error**

- There are many!
  - Discretization error (mesh does not match shape)
  - Uncertain material properties
  - Bad element quality
  - Incorrect elements for the analysis performed and results required
  - Incorrect boundary conditions
  - Results singularity (Example spreadsheet)
  - Incorrect failure criteria
  - Un-converged solution
  - Un-converged mesh size
  - Incorrect results interpretation
  - Results averaging/smoothing
  - Many more

**FACTOR OF SAFETY**

- Accounts for errors beyond our control and other uncertainties (Example Spreadsheet)
Tips and Recommendations

- **Have a plan!**
  - What are the goals of the analysis?
    - Determine strength?
    - Determine deflection?
  - What are the failure criteria?
    - Yield strength?
    - Buckling?
    - Fatigue?
  - What analysis type must be done to satisfy the above two questions?
    - Linear static?
    - Non-linearity?
    - Dynamic effects?
  - What compromises can you make to achieve your goals?
    - Coarse mesh for a global response or fine mesh for local details?
    - 2nd order Tetra elements or 1st order Hexa?
    - Solid or 1D/Shell models?
HyperWorks Student Guide

- It’s Free!
  - Very good reference guide
  - Essentially what was just presented, but in much more detail
  - 453 pages
  - Contact Altair for a copy
Linear Static Example

- Motocross Suspension Linkage Component