Introduction to FEA

Won Hyun Park
FEA Pitfalls

Tony Rizzo (Bell Labs) quotes:

- “Some engineers and managers look upon commercially available FEA programs as automated tools for design. In fact, nothing could be further from reality than that simplistic view of today's powerful programs. The engineer who plunges ahead, thinking that a few clicks of the left mouse button will solve all his problems, is certain to encounter some very nasty surprises.”

- “With the exception of a very few trivial cases, all finite element solutions are wrong, and they are likely to be more wrong than you think. One experienced analysis estimates that 80% of all finite element solutions are gravely wrong, because the engineers doing the analyses make serious modeling mistakes.”

- “Finite element analysis is a very powerful tool with which to design products of superior quality. Like all tools, it can be used properly, or it can be misused. The keys to using this tool successfully are to understand the nature of the calculations that the computer is doing and to pay attention to the physics.”
FEA Theory

- Finite element method – numerical procedure for solving a continuum mechanics problem with acceptable accuracy.
- Subdivide a large problem into small elements connected by nodes.

- FEM by minimizing the total potential energy of the system to obtain primary unknowns - the temperatures, stresses, flows, or other desired...
FEA example for spring

Equilibrium : Minimum of Potential energy

(Assume 1D problem : x axis)

\[ \Pi = \text{strain energy} - \text{work} = \frac{1}{2} \sigma \epsilon - Fx \]

\[ \frac{\partial \Pi}{\partial x} = 0 \]

For a spring,

\[ \Pi = \frac{1}{2} k x^2 - Fx , \quad \frac{\partial \Pi}{\partial x} = kx - F = 0 \]

\[ \Rightarrow F = kx \]
General FEA formula

The total potential energy can be expressed as:

\[ \Pi = \frac{1}{2} \int_{\Omega} \sigma^T \varepsilon dV - \int_{\Omega} d^T \varepsilon dV - \int_{\Gamma} d^T q dS \]

The total potential energy of the discretized individual element:

\[ \Pi_e = \frac{1}{2} \int_{\Omega_e} \left( B^T E B \right)^T u dV - \int_{\Omega_e} u^T N^T p dV - \int_{\Gamma} u^T N^T q dS = 0. \]

\[ \frac{\partial \Pi_e}{\partial u} = 0 \text{ gives: } F = K \, u, \text{ where } K \text{ is stiffness Matrix, } [K]. \]
FEA Solution

Simple Hook’s law

\[
[F] = [K] \cdot [u]
\]

\[
[u] = [K]^{-1} \cdot [F]
\]
System stiffness matrix: 1D example

Global stiffness matrix = 3 x 3

Element stiffness matrix = 2 x 2 = \begin{pmatrix} k & -k \\ -k & k \end{pmatrix}
How to build the stiffness matrix

Global stiffness matrix = 3 x 3

\[
\begin{pmatrix}
  k_1 & -k_1 & 0 \\
  -k_1 & k_1 + k_2 & -k_2 \\
  0 & -k_2 & k_2
\end{pmatrix}
\]
B.C and solve

Boundary condition

\[
\begin{bmatrix}
0 \\
x_2 \\
x_3
\end{bmatrix} = 
\begin{bmatrix}
0 & -k_1 & 0 \\
-k_1 & k_1 + k_2 & -k_2 \\
0 & -k_2 & k_2
\end{bmatrix}^{-1}
\begin{bmatrix}
0 \\
0 \\
P
\end{bmatrix}
\]
Nodes
“What are they”

A node is simply a coordinate location in space where a DOF (degree of freedom) is defined.

Nodes – Properties and Characteristics
• Infinitesimally small
• Defined with reference to a global coordinate system
• Typically nodes are defined on the surface and in the interior of the component you are modeling
• Form a grid work within component as a result of the mesh
• Typically define the corners of elements
• Where we define loads and boundary conditions
• Location of our results (deformation, stress, etc.)
• Nodes are the byproduct of defining elements
An element is a mathematical relation that defines how a DOF of a node relates to the next.

Elements – Properties and Characteristics

• Point, 2D and 3D elements
• Define a line (1D), area (2D) or volume (3D) on or within our model
• Dimensions define an “Aspect Ratio”
• A set of elements is know as the “mesh”
• Mesh shape and density is critical to the analysis
• Typically have many options that may be preset for the user
• Elements are typically what we define
SolidWorks Simulation

Solid Mesh
- TETRA4 and TETRA10 (4 & 10 node tetrahedron solid elements)

Shell Mesh
- SHELL3 and SHELL6 (3 & 6 node thin elements)

Problems, Pitfalls and Tips

Requires planning and element / DOF knowledge
- The element defines the number of active DOFs.

TETRA4 & TETRA10 Elements
- 3 translational DOF per node
- 1 DOF per node for thermal
- TETRA4 (linear) TETRA10 (parabolic)
- Supports adaptive “P” method

SHELL3 & SHELL6 Elements
- 6 DOF per node (3 translational + 3 rotational)
- 1 DOF per node for thermal
- Membrane and bending capabilities
- Uniform thickness element
- SHELL3 (linear) SHELL6 (parabolic)
- Supports adaptive “P” method
4-node tetrahedral mesh

The *unit reference tetrahedron has corners at* \( \{0,0,0\}, \{1,0,0\}, \{0,1,0\}, \{0,0,1\} \)

\[
\begin{bmatrix}
4-6\nu & 1 & 1 & -2\hat{\nu} & -\tilde{\nu} & -\tilde{\nu} & -\tilde{\nu} & -2\nu & 0 & -\tilde{\nu} & 0 & -2\nu \\
1 & 4-6\nu & 1 & -2\nu & -\tilde{\nu} & 0 & -\tilde{\nu} & -2\hat{\nu} & -\tilde{\nu} & 0 & -\tilde{\nu} & -2\nu \\
1 & 1 & 4-6\nu & -2\nu & 0 & -\tilde{\nu} & 0 & -2\nu & -\tilde{\nu} & -\tilde{\nu} & -\tilde{\nu} & -2\nu \\
-2\hat{\nu} & -2\nu & -2\nu & 2\hat{\nu} & 0 & 0 & 0 & 2\nu & 0 & 0 & 0 & 2\nu \\
-\tilde{\nu} & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 & 0 & 0 & 0 & 0 \\
-\tilde{\nu} & 0 & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & 0 & 0 & \tilde{\nu} & 0 & 0 \\
-\tilde{\nu} & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 & 0 & 0 & 0 & 0 \\
-2\nu & -2\hat{\nu} & -2\nu & 2\nu & 0 & 0 & 0 & 2\hat{\nu} & 0 & 0 & 0 & 2\nu \\
0 & -\tilde{\nu} & -\tilde{\nu} & 0 & 0 & 0 & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 \\
-\tilde{\nu} & 0 & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & 0 & 0 & \tilde{\nu} & 0 & 0 \\
0 & -\tilde{\nu} & -\tilde{\nu} & 0 & 0 & 0 & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 \\
-2\nu & -2\nu & -2\hat{\nu} & 2\nu & 0 & 0 & 0 & 2\nu & 0 & 0 & 0 & 2\nu
\end{bmatrix}

\( \mathbf{K}^e = \mathbf{\hat{E}} \)

\[
\mathbf{\hat{E}} = E/(12(1-2\nu)(1+\nu)), \quad \tilde{\nu} = 1 - 2\nu \text{ and } \hat{\nu} = 1 - \nu
\]
<table>
<thead>
<tr>
<th>i</th>
<th>j</th>
<th>k</th>
<th>x</th>
<th>y</th>
<th>z</th>
<th>dx</th>
<th>dy</th>
<th>dz</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
</tr>
</tbody>
</table>

**10-node tetrahedral mesh**

- Node positions: (x, y, z)
- Edge lengths: dx, dy, dz

**Kw**

- Connectivity matrix

**signs of K**:

- Positive values in red
- Negative values in blue

**Notes**:

- This table represents a tetrahedral mesh with 10 nodes.
- The connectivity matrix shows the relationships between nodes.
- The signs of K indicate the orientation of the tetrahedra.
FEA procedure

1. Identify the problem, sketch the structure and loads.
2. Create the geometry with the FE package solid modeler or a CAD system.
3. Apply material properties.
4. Mesh the model.
5. Apply boundary conditions (constraints and loads) on the model.
6. Solve numerical equations.
7. Evaluate the results.
Solidworks Simulation example
Solidworks Simulation example

Stress distribution

Displacement plot
Theoretical background for modal analysis

- **Equation of motion (assume zero damping)**

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

- **Solving the equation**

\[
\{u\} = \{\phi\} \sin \omega t
\]

\[
-\omega^2 [M]\{\phi\} \sin \omega t + [K]\{\phi\} \sin \omega t = 0
\]

\[
([K] - \omega^2 [M])\{\phi\} = 0 \iff [A - \lambda I]x = 0
\]

Eigen value problem

\[
\det ( [K] - \lambda [M] ) = 0
\]
Simple analytic model

Two mass block connected with a spring

\[
\begin{align*}
\mathbf{K} &= \begin{pmatrix} k & -k \\ -k & k \end{pmatrix}, \\
\mathbf{M} &= \begin{pmatrix} m_1 & 0 \\ 0 & m_2 \end{pmatrix}, \\
\mathbf{K} - \omega^2 \mathbf{M} &= \begin{pmatrix} k - \lambda m_1 & -k \\ -k & k - \lambda m_2 \end{pmatrix}.
\end{align*}
\]

\[
\det(\mathbf{K} - \lambda \mathbf{M}) = \lambda^2 m_1 m_2 - \lambda k m_2 - \lambda k m_1 \quad \text{where} \quad \lambda = \omega^2
\]

Eigenvalues

\[
\begin{cases}
0, & k m_1 + k m_2 \\
\frac{k m_1 + k m_2}{m_1 m_2} & \text{if } m_1 \neq 0 \land m_2 \neq 0
\end{cases}
\]

Resonant frequency

Eigenfunctions

\[
\begin{pmatrix} 1 \\ 1 \end{pmatrix} \quad \text{and} \quad \begin{pmatrix} -\frac{m_2}{m_1} \\ 1 \end{pmatrix}
\]

Rigid body motion

Mode shape
Simple analytic model

Two mass block connected with a spring
Target model – Modal frequency analysis
Target model – Modal frequency analysis

4th order mode shape for example
Modeling Tips

- **Tip #1** Understand the physics of the problem and always start with a sketch. Before you model have a plan and know:
  - What you are going to model and what results do you need?
  - How you are going to develop your model?
  - How you are going to support the model and apply loads?

- **Tip #2** Start simple and increase complexity as required.
  - When modeling systems – start with a single optical element.

- **Tip #3** Build simple test models for understanding.
  - Check for load and boundary condition accuracy
    - Above depends on the type of analysis you are running
  - Check for mesh accuracy – do convergence studies

- **Tip #4** Always request reaction forces in the output.
  - For models with both structural and gravity loads, turn off gravity and check your reaction forces. Do they match the applied load?
  - Turn on gravity. Is the increase in reaction force consistent with the gravity load? Is the direction correct?
Modeling Tips

- **Tip #5** Understand your constraints and use care not to over constrain the model.
  - Are all six (6) rigid body translations and rotations accounted for?
  - A model with too few constraints causes a singular stiffness matrix.
  - An over constrained model creates alternate load paths.
  - When in doubt, release constraints and add soft springs.

- **Tip #6** Study the deformed shape. Does it look correct?
  - Properly modeled, symmetric loads and constraints will produce symmetric results.
  - Always generate a symmetric mesh for optical surfaces. Automatic mesh generators rarely produce a symmetric mesh.

- **Tip #7** As a starting point, there must be enough elements to accurately predict the deformed shape.
  - Use a minimum of 4 elements through the height or thickness or sections subjected to bending.
  - Do simple convergence studies to determine an acceptable mesh density.

- **Tip #8** An accurate stress analysis requires more elements than an accurate displacement analysis.
  - Increase the mesh density for accurate stress analysis.
  - Check nodal stress in surrounding elements sharing a common node.
Modeling Tips

• **Tip #9** Check, check and recheck your model and results.
  – Do hand calculations and back of the envelope calculations to verify results.
  – Assume the results are wrong until proven correct.

• **Tip #10** When in trouble – get help.
  – Consult a senior analyst for help and tips