Finite Element Analysis (FEA)

An introduction
Finite Element Analysis -- “a technique that reconstructs stress, strain, and deformation in a digital structure”.
(Rayfield, 2007)

- Used to simulate the consequences of forces that are applied to an object
- Applied to structures or materials that are too complex to estimate responses based on ordinary differential equations
- Used to study the relationship between form and function in morphological structures
- Simulates the distribution of stress, strain, and deformation in a solid object when forces and constraints are applied to it

Graphic representation of stress in a digital model of a skull of *Dinocrocuta*, an extinct hyena from the Miocene, as it bites down on the left third premolar (from Tseng, 2009)
Origin: engineering

Simulated deformation of a bridge under load

Illustration from LUSAS engineering analysis software, IMDplus.
http://www.lusas.com/products/options/imdplus.html
Definitions

**Strain (\(\varepsilon\)) - change in length / length**

Strain is a type of deformation, the physical change in the size or shape of an object when force is applied. Strain is essentially the proportional change in the size (length) of the material at a particular point on the object. For example, compression decreases the size (negative strain) and tension (stretching) increases it (positive strain).

This particular strain metric is also known as Cauchy strain or engineering strain. Other strain measures include stretch ratio, true strain, Green strain, Almansi strain, etc.

**Stress (\(\sigma\)) - force per unit area, F/A.**

Stress is the amount of internal force that arises from strain, or deformation of the object. Stress can arise from external forces like gravity or loads place on an object, from friction, from internal elastic stress that propagates through a deformed object, etc.

Patterns of strain and stress are strongly affected by the material properties of the object, which affect whether it is compressible, brittle, stretchable, etc.
Strain vs. Stress

Strain ($\varepsilon$) - change in length / length

Stress ($\sigma$) - force per unit area, $F/A$

Simulated with FE-Bio
Finite element strategy: Cantilever beam example

Exact method

Finite element method

Exact equations can be used to estimate deformation, stress, strain and other mechanics of simple structures.

Result can be approximated by dividing beam into small sections and calculating how deformation, stress, strain propagate.
Finite elements in morphology

Biological structures often have difficult shapes and unusual loadings. But they can be divided into small blocks (finite elements) where exact equations can be applied. The stresses and strains can be estimated by simultaneously solving the equations for all blocks.
What happens in a single element?

Kinds of elements

2D

- simple
- quadratic

3D

E = elastic properties of material
A = area
L = length

Fig. 6. Diagrammatic representation of the element equations in a simple element. Here, a 1D two-noded bar element $e$ with forces $f_1$ and $f_2$ act on its two nodes with nodal displacements $u_1$ and $u_2$. The stiffness of this element is given by $E^e A^e L^e$, which includes information about the element's elastic material properties, area, and length. Note that the positive x-direction nodal force $f_1$ is a compressive force. Its value is given by the compressive deformation of the bar, $u_1 - u_2$, times the bar's stiffness, $E^e A^e L^e$. Similarly, the nodal force $f_2$ is a tensile force equal to the tensile deformation of the bar, $u_2 - u_1$, times the bar's uniaxial stiffness $E^e A^e L^e$.
Finite element method

1. Model creation = finite element mesh of object + material properties + boundary constraints + boundary loads or displacements

2. Model solution = solve equations for all finite elements in the mesh based on material and boundary conditions

3. Model output = numerical and visual output of distribution of stresses and strains throughout the object
Finite element meshes
What is a mesh?

Digital object composed of elements with vertices, edges, and faces. Elements can be 2D or 3D, triangular, quadrilateral, or polygonal. Mesh can be a surface (no interior elements) or a solid (with interior elements). For FEA, the mesh elements are the finite elements.

How do I get a mesh?

- Create by hand by designing mesh object in CAD (computer assisted drafting) or other software.
- Digitize a real object and create mesh (surface or solid)

Digitization

- Objects can be photographed to create 2D meshes.
- Points on 3D object can be digitized and then meshed (Microscribe arm, Reflex microscope)
- Surface of an object can be scanned with laser or optical scanner and meshed. Surface mesh can be converted to solid mesh with certain software.
- Volume of an object can be scanned with CT, MRI, or similar. Data must be processed by segmenting (isolating bones or structures of interest from the background) and meshing (either as surface or solid mesh)
Example of laser scanned objects
CT scanned objects
Material properties

Young’s modulus - stress / strain (force per area relative to deformation)

Also known as elastic modulus or tensile modulus, Young’s modulus is a measure of stiffness or elasticity. Substances that are very elastic, like rubber bands, have low Young’s modulus; stiffer substances like diamond, have high Young’s modulus.

Bulk modulus -

Measure of uniform resistance to compression.

Density -

mass per volume.

Isotropic -

response to load is the same in all directions (i.e., “strength” is the same in all directions)

Anisotropic -

response to load differs depending on direction (“strength” is not the same in all directions)
Bone materials and histology (anisotropic)

Collagen - elastic protein fibers, low Young’s modulus
Hydroxyapatite crystals - phosphate mineral, high Young’s modulus
Examples of differences in biological materials

<table>
<thead>
<tr>
<th>Tissue</th>
<th>Cement</th>
<th>Dentine</th>
<th>Enamel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hardness</td>
<td>Softest</td>
<td>Intermediate</td>
<td>Hardest</td>
</tr>
<tr>
<td>Mineral Weight</td>
<td>65%</td>
<td>70%</td>
<td>95-97%</td>
</tr>
<tr>
<td>Embryonic origin</td>
<td>Mesodermal</td>
<td>Mesodermal</td>
<td>Ectodermal</td>
</tr>
<tr>
<td>Cellular?</td>
<td>Sometimes</td>
<td>Yes (odt. pr.)</td>
<td>No</td>
</tr>
<tr>
<td>Deposited throughout life</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
</tbody>
</table>
Measuring failure of a material with Von Mises stress

Von Mises Yield criterion

Objects can deform in two ways: elastically, which is where they bounce back to their resting shape when force is relieved, and plastically, which is where they don’t return to their original shape. Breaking and cracking are extreme examples of plastic yield, permanent bending of a coin is a less extreme example.

*Von Mises yield* is a measure of when a particular material will start to yield plastically instead of elastically, in other words it is a property related to when the material starts to fail.

**Von Mises Stress**

*Von Mises stress* is a specific measure of stress in multiple planes (e.g., longitudinal, horizontal). When Von Mises stress reaches the *yield strength* of a particular material, then that material will start to yield plastically.
Constraints and Loads?

Class 1 Lever?

Relationship between the TMJ and tooth rows
Bibliography


Things to look for in the papers for next week:

**Meshes:** how did they obtain them? Are meshes 2D, 3D, solid or hollow? What kinds of elements, how many elements, what shape elements?

**Materials:** what was the real material? what properties did they use to model the material?

**Constraints:** where did they place the constraining boundary conditions? how do those constraints related to the real biology of the problem?

**Loads:** ditto.

**Outputs:** What output do they focus on? stress, strain? what measure of?

**Software:** what software for the data collection? FEA models? processing?