FINITE ELEMENT METHODS IN BIOMEDICAL ENGINEERING

Lecture 3

Presented by
Dr Paul Wong
AMME4981/9981
Semester 1, 2016
Workflow for Biomedical Problems

1. Data acquisition
   • Scan region of interest
   • Obtain material properties for tissues and implants
   • Estimate expected loads

2. Solid modelling
   • Convert image stacks into a virtual replica
   • Combine with CAD model of prosthesis

3. Finite element analysis
   • Generate appropriate mesh
   • Characterise interaction between anatomy and prosthesis
   • Verify simulation results and prosthesis design
(RE)-INTRODUCTION TO FINITE ELEMENT ANALYSIS
Rationale and Concept
Continuum Mechanics

- Analytical methods are only suitable for simple scenarios
Limitations of Analytical Methods

- Biomedical problems are complicated:
  - Sophisticated geometry
  - Non-linear, anisotropic materials
  - Complex loading and boundary conditions
  - Coupled physics
Limitations of Analytical Methods — Example

\[ R = 1730 \text{ N}, \]
\[ R_x = -300 \text{ N}, \]
\[ R_y = -310 \text{ N}, \]
\[ R_z = -1670 \text{ N}, \]
\[ M = 1270 \text{ N}, \]
\[ M_x = 300 \text{ N}, \]
\[ M_y = 310 \text{ N}, \]
\[ M_z = 1190 \text{ N}. \]
Limitations of Analytical Methods – Example

– Is the approximation good enough?
– How much detail is required/feasible?
– Are the assumptions reasonable?
– How would you validate it?

![Diagram of a bone](image)

![Diagram of a beam under force](image)
The Finite Element Method (FEM)

1. Approximate the complex shape (geometry) by breaking it down (discretising) into simpler, independent blocks (elements)
2. Recast the physics problem as a relation within each element
   — i.e. connect the nodes of each element using a shape function
   — Depends on material properties
3. Link all the elements together
   — i.e. form the global equilibrium equation
4. Impose constraints (loads and boundary conditions)
5. Solve all equations simultaneously
A Simple Example

伟:  \[ \pi = \frac{C}{2r} \]

http://www.xkcd.com/1184/
A Simple Example

\[ p_n = n \sin(\pi/n) \]

<table>
<thead>
<tr>
<th>n</th>
<th>( p_n = n \sin(\pi/n) )</th>
<th>Exact ( \pi ) (to 16 s.f.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2.828427124746190</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>3.121445152258052</td>
<td></td>
</tr>
<tr>
<td>64</td>
<td>3.140331156954753</td>
<td></td>
</tr>
<tr>
<td>256</td>
<td>3.141513801144301</td>
<td>3.141592653589793</td>
</tr>
</tbody>
</table>
Discretisation

Continuum model

Discretised model

Discontinuous between elements, but connected through interface (e.g. same displacement at shared nodes)

Continuous distribution within each element (via shape function)
Discretising the Hip Implant
Element Types

- Tetrahedron
- Pyramid
- Triangle
- Line
- Quadrilateral
- Brick
- Prism
Effect of Element Order on Model Size

Discretised model (lower order)  
Elements: 2  
Nodes per element: 8  
Total nodes: 12  
Total DOFs: 36

Discretised model (higher order)  
Elements: 2  
Nodes per element: 20  
Total nodes: 32  
Total DOFs: 96

Total Degrees of Freedom = Total number of nodes × DOFs at each node
Mixing Element Types

- T3 and Q4 are usually used together in a mesh with linear elements
- T6 and Q8 are usually used together when quadratic elements are desired
- Similar for volume elements
Suggestions

- Focus on **understanding the concept** of finite elements and **performing analyses** in the software (ANSYS Workbench)
- Try using the program yourself
  - Step-by-step guides for running a basic analysis are available on the [UoS website](https://uoswebsite.com) (see Assessments tab)
- Take advantage of group learning
(RE)-INTRODUCTION TO FINITE ELEMENT ANALYSIS

The nitty gritty stuff (a.k.a. mathematics)
The Finite Element Method (FEM)

1. Approximate the complex shape (geometry) by breaking it down (discretising) into simpler, independent blocks (elements)
2. Recast the physics problem as a relation within each element
   - i.e. connect the nodes of each element using a shape function
   - Depends on material properties
3. Link all the elements together
   - i.e. form the global equilibrium equation
4. Impose constraints (loads and boundary conditions)
5. Solve all equations simultaneously
Intra-element Interpolation
The Linear Triangular Element

- Only has 3 nodes
- **Linear displacement** interpolation within element
  - **Uniform strain** inside element bounds
  - Known as constant strain triangle (CST)
- Use where strain gradient is small
  - Best to avoid in critical areas of structure and areas with stress concentration
- Recommended for preliminary FEA (quick but low accuracy)
The Linear Triangular Element

Displacements within each element are a weighted function of the displacements at the nodes

\[ U^h(x, y) = N(x, y)d_e \]

\[ N = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \]

\[ d_e = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \]

Displacements at node 1
Displacements at node 2
Displacements at node 3
The Linear Triangular Element

Shape Function

- For linear interpolation:

\[
N_1 = a_1 + b_1 x + c_1 y
\]
\[
N_2 = a_2 + b_2 x + c_2 y
\]
\[
N_3 = a_3 + b_3 x + c_3 y
\]
\[
\begin{cases}
1 & \text{for } i = j \\
0 & \text{for } i \neq j
\end{cases}
\]

- Consider the Kronecker Delta function:

\[
N_i(x_j, y_j) = \begin{cases}
1 & \text{for } i = j \\
0 & \text{for } i \neq j
\end{cases}
\]

\[
N_1(x_1, y_1) = 1
\]
\[
N_1(x_2, y_2) = 0
\]
\[
N_1(x_3, y_3) = 0
\]

- Let’s focus on \( N_1 \) (for now)…
The Linear Triangular Element

Shape Function

- We know that the weighting at node 1 is 100% (i.e. 0% contribution from the other nodes. Therefore, we have:

\[
N_1(x_1, y_1) = a_1 + b_1x_1 + c_1y_1 = 1
\]

\[
N_1(x_2, y_2) = a_1 + b_1x_2 + c_1y_2 = 0
\]

\[
N_1(x_3, y_3) = a_1 + b_1x_3 + c_1y_3 = 0
\]

- Solve for constants:

\[
a_1 = \frac{x_2y_3 - x_3y_2}{2A_e}, \quad b_1 = \frac{y_2 - y_3}{2A_e}, \quad c_1 = \frac{x_3 - x_2}{2A_e}
\]

- Substitute back into \( N_1 \):

\[
N_1 = \frac{1}{2A_e}[(y_2 - y_3)(x - x_2) + (x_3 - x_2)(y - y_2)]
\]
The Linear Triangular Element

Shape Function

- Similarly:

\[ N_2(x_1, y_1) = 0 \]
\[ N_2(x_2, y_2) = 1 \]
\[ N_2(x_3, y_3) = 0 \]

\[ N_2 = \frac{1}{2A_e} [(x_3 y_1 - x_1 y_3) + (y_3 - y_1)x + (x_1 - x_3)y] \]

\[ N_3(x_1, y_1) = 0 \]
\[ N_3(x_2, y_2) = 0 \]
\[ N_3(x_3, y_3) = 1 \]

\[ N_3 = \frac{1}{2A_e} [(x_1 y_2 - x_2 y_1) + (y_1 - y_2)x + (x_2 - x_1)y] \]
The Linear Triangular Element

Strain Matrix

- Displacement function:
  
  \[ u = N_1 u_1 + N_2 u_2 + N_3 u_3 \]
  
  \[ v = N_1 v_1 + N_2 v_2 + N_3 v_3 \]

- Strain-displacement relationship:
  
  \[ \varepsilon_{xx} = \frac{\partial u}{\partial x}, \quad \varepsilon_{yy} = \frac{\partial v}{\partial y}, \quad \varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \]

- Combining, we get:
  
  \[ \{\varepsilon_e\} = [B_e] \{d_e\} \]

where

\[ [B_e] = \begin{bmatrix}
 a_1 & 0 & a_2 & 0 & a_3 & 0 \\
 0 & b_1 & 0 & b_2 & 0 & b_3 \\
 b_1 & a_1 & b_2 & a_2 & b_3 & a_3
\end{bmatrix} \]

\[ \{d_e\} = \{u_1, v_1, u_2, v_2, u_3, v_3\}^T \]
The Linear Quadrilateral Element

- **Displacement interpolation**

\[ U^h(x, y) = N(x, y)d_e \]

- **Shape function**

\[
N = \begin{bmatrix}
N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\
0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4
\end{bmatrix}
\]

\[
N_1 = \frac{1}{4} (1 - \xi)(1 - \eta) \quad N_3 = \frac{1}{4} (1 + \xi)(1 + \eta)
\]
\[
N_2 = \frac{1}{4} (1 + \xi)(1 - \eta) \quad N_4 = \frac{1}{4} (1 - \xi)(1 + \eta)
\]

- **Natural coordinates**

\[
\xi = x/a, \quad \eta = y/b
\]
The Linear Quadrilateral Element

- Strain inside element is NOT constant due to bilinear interpolation

\[
[B_e] = \begin{bmatrix}
-\frac{1-\eta}{a} & 0 & \frac{1-\eta}{a} & 0 & \frac{1+\eta}{a} & 0 & -\frac{1+\eta}{a} & 0 \\
0 & -\frac{1-\xi}{b} & 0 & -\frac{1+\xi}{b} & 0 & \frac{1+\xi}{b} & 0 & \frac{1-\xi}{b} \\
-\frac{1-\xi}{b} & -\frac{1-\eta}{a} & -\frac{1+\xi}{b} & \frac{1-\eta}{a} & \frac{1+\xi}{b} & \frac{1+\eta}{b} & \frac{1-\xi}{b} & -\frac{1+\eta}{a}
\end{bmatrix}
\]

- What does this imply?
Element Types

Tetrahedron

Pyramid

Prism

Brick

Quadrilateral

Triangle

Line
Linear vs Quadratic Interpolation

Displacement

Location within element
The Quadratic Triangular Element

- Additional node at each midpoint
  - 6 nodes per element (12 DOFs)
- 2nd order interpolation function
  \[ u = b_1 + b_2 x + b_3 y + b_4 x^2 + b_5 xy + b_6 y^2 \]
  \[ v = b_7 + b_8 x + b_9 y + b_{10} x^2 + b_{11} xy + b_{12} y^2 \]
- Linear strain
  \[ \varepsilon_x = b_2 + 2b_4 x + b_5 y \]
  \[ \varepsilon_y = b_9 + b_{11} x + 2b_{12} y \]
  \[ \gamma_{xy} = (b_3 + b_8 ) + (b_5 + 2b_{10} )x + (2b_6 + b_{11} )y \]
- Better at capturing high strain gradients and curved boundaries
The Quadratic Quadrilateral Element

- Additional node at each midpoint
  - 8 nodes per element (16 DOFs)
- Quadratic shape function

\[
\begin{align*}
N_1 &= \frac{1}{4}(1 - \xi)(\eta - 1)(\xi + \eta + 1) & N_5 &= \frac{1}{2}(1 - \eta)(1 - \xi^2) \\
N_2 &= \frac{1}{4}(1 + \xi)(\eta - 1)(\eta - \xi + 1) & N_6 &= \frac{1}{2}(1 + \xi)(1 - \eta^2) \\
N_3 &= \frac{1}{4}(1 + \xi)(1 + \eta)(\xi + \eta - 1) & N_7 &= \frac{1}{2}(1 + \eta)(1 - \xi^2) \\
N_4 &= \frac{1}{4}(\xi - 1)(\eta + 1)(\xi - \eta + 1) & N_8 &= \frac{1}{2}(1 - \xi)(1 - \eta^2)
\end{align*}
\]

- Preferred for stress analysis due to high accuracy and capacity for modelling complex geometries
Elemental Stiffness Matrix

- Strain energy measures the amount of work performed on an elastic structure during deformation.
- Expressing this in terms of strain energy density for a differential volume:

\[
U_e = \frac{1}{2} \int_{V} \{\sigma_e\}^T \{\varepsilon_e\} dV = \frac{1}{2} \int_{V} \left( \sigma_x \varepsilon_x + \sigma_y \varepsilon_y + \sigma_{xy} \varepsilon_{xy} \right) dV
\]

\[
= \frac{1}{2} \int_{V} \left( [E_e] \{\varepsilon_e\} \right)^T \{\varepsilon_e\} dV = \frac{1}{2} \int_{V} \{\varepsilon_e\}^T [E_e] \{\varepsilon_e\} dV
\]

\[
= \frac{1}{2} \{d_e\}^T \left( \int_{V} [B_e]^T [E_e] B_e \right) \{d_e\}
\]

\[
= \frac{1}{2} \{d_e\}^T [k_e] \{d_e\}
\]

- The stiffness is therefore defined as:

\[
[k_e]_{6 \times 6} = \int_{V} [B_e]^T [E_e]_{3 \times 3} [B_e]_{3 \times 6} dV
\]
The Finite Element Method (FEM)

1. Approximate the complex shape (geometry) by breaking it down (discretising) into simpler, independent blocks (elements)
2. Recast the physics problem as a relation within each element
   - i.e. connect the nodes of each element using a shape function
   - Depends on material properties
3. Link all the elements together
   - i.e. form the global equilibrium equation
4. Impose constraints (loads and boundary conditions)
5. Solve all equations simultaneously
Global Stiffness Matrix

For an entire structure composed of finite elements

\[
U = U_1 + U_2 + \cdots + U_N = \sum_{e=1}^{N} U_e
\]

\[
= \sum_{e=1}^{N} \frac{1}{2} \{d_e\}^T [k_e] \{d_e\} = \frac{1}{2} \{u\}^T [K] \{u\}
\]

where the global stiffness matrix is given by:

\[
[K] = [\hat{k}_1] + [\hat{k}_2] + \cdots + [\hat{k}_N]
\]

- Matrix size depends on total DOFs
- Larger matrices require more computation
Global Equilibrium Equation

- In equilibrium, strain energy equals work done by external force
- Consider virtual energy method:

\[
\frac{1}{2} \{u\}^T [K]\{u\} = {u}^T \{p\}
\]

\[
\frac{\partial}{\partial \{u\}} \left( \frac{1}{2} \{u\}^T [K]\{u\} \right) = \frac{\partial}{\partial \{u\}} \left( {u}^T \{p\} \right)
\]

- Therefore, equilibrium equation is:

\[
[K]\{u\} = \{p\}
\]

- Impose loads and boundary conditions
- Solve for the nodal displacement vector \{u\}
- Then determine stress, strain, etc.
Imposing Constraints

\[ U_1 = U_0 + 1 \]

\[ U_2 = U_1 + 0.5 \]

\[ U_3 = U_2 + 2 \]
The Finite Element Method (FEM)

1. Approximate the complex shape (geometry) by breaking it down (discretising) into simpler, independent blocks (elements)
2. Recast the physics problem as a relation within each element
   - i.e. connect the nodes of each element using a shape function
   - Depends on material properties
3. Link all the elements together
   - i.e. form the global equilibrium equation
4. Impose constraints (loads and boundary conditions)
5. Solve all equations simultaneously
PRACTICAL CONSIDERATIONS
Exporting from ScanIP

Option 1: Export Surface Meshes

- STL format
  - RP surface requires masks only
  - Piecewise surface geometry composed of triangles
  - Adjacent components are conformal
- IGES format
  - Requires NURBS module
  - Surface divided into patches defined by splines
  - NOT conformal
- Use as geometry input for CAD or volume mesher
Exporting from ScanIP

Option 2: Export Volume Meshes

- Require FE model from masks
- Export directly to ANSYS, COMSOL, etc. or as NASTRAN
- Two algorithms available:
  - +FE Grid (more robust)
  - +FE Free (more refined)
- Advanced mesh parameters can be used to customise the mesh
Fixing the FE Modeller Licencing Problem

- Start menu > ANSYS > ANSYS Client Licencing > User Licence Preferences
- “PrepPost” tab
- Move “ANSYS Academic Meshing Tools” to the bottom of the list
The Phillip (NAS)Tran Method

- Export surfaces as STL (RP)
  - Binarise, use smart smoothing, and allow part change
  - Disable triangle smoothing and decimation
- Import STLs into ICEM CFD
- Set mesh parameters
  - Global: Curvature refinement
  - Volume meshing: Edge criterion, number of smoothing iterations
  - Part mesh setup: Max/min edge lengths
- Compute using Octree algorithm
- Export volume mesh as NASTRAN
Trade-offs in Computational Modelling

- For a given hardware setup, there is a trade-off between model accuracy and computational cost

<table>
<thead>
<tr>
<th>Model accuracy</th>
<th>Computational cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>High mesh density</td>
<td>Long solution time</td>
</tr>
</tbody>
</table>
| Quadratic elements

For a given hardware setup, there is a trade-off between model accuracy and computational cost.
Trade-offs in Computational Modelling

- Try to find a balance between the two

Model accuracy
- Low density or refined mesh
- Linear elements

Computational cost
- Faster CPU
- More RAM
- Shorter solution time
Simplifying Assumptions

- Look for symmetry:
  - About a plane
  - Around an axis
- Only used in very simplified biomedical problems

- Reduce to a 2D problem using:
  - Plane stress (i.e. thickness is small relative to other two directions)
  - Plane strain (i.e. thickness significantly greater than the other two directions)
Simplifying Assumptions

FE mesh

Bone density distribution
Simplifying Assumptions

pelvis

cement

acetabular cup

Cement

Cup

Load
Mesh Refinement

- A single geometry can be meshed in many different ways
- Depends on the meshing algorithm, and the parameters passed to it
- Different programs use different algorithms

<table>
<thead>
<tr>
<th>TYPE</th>
<th>METHOD</th>
</tr>
</thead>
<tbody>
<tr>
<td>H-refinement</td>
<td>Reduce size of elements</td>
</tr>
<tr>
<td>P-refinement</td>
<td>Increase order of polynomials on element (e.g. linear to quadratic)</td>
</tr>
<tr>
<td>HP-refinement</td>
<td>Use both h- and p-refinement</td>
</tr>
<tr>
<td>R-refinement</td>
<td>Rearrange the nodes in the mesh</td>
</tr>
</tbody>
</table>
Mesh Convergence

- FE models are numerical approximations
- There are substantially fewer nodes in any FE model than there are particles in the actual structure
- Limited number of nodes implies a smaller number of DOFs
- Hence, FE models are generally stiffer than the real structure
- Conversely, displacement results are usually smaller than the true values
- Using more DOFs will approach the exact solution
Contact Conditions

Cemented Interface

- Seamless bonding
- Shared interfacial nodes

Cementless/Frictional Interface

- In contact with friction
- Requires two sets of interfacial nodes
Interpreting Results

http://xkcd.com/1339/
Interpreting Results

- Models are (by definition) simplifications of nature
- Be mindful of the assumptions used to construct the model
  - Geometry simplifications
  - Material properties
  - Boundary conditions
- Be wary of errors
  - Imperfect segmentation
  - Discretisation errors from meshing process
  - Numerical error when solving FE equations
  - Software bugs
- Make sure your calculated value is relevant to your research question
Model Validation

- A model on its own does not represent the truth
- Best practice is to compare *in silico* results to *in vivo* or *in vitro* experimental measurements
  - Each method is subject to error
  - Can be first-hand or second-hand
  - Need to explain any differences
- The best models are still more valuable qualitatively than quantitatively… but this is starting to change
EXAMPLE APPLICATIONS
**Hip Fractures**

- Femoral neck impacted into head using surgical screws
- Questions from orthopaedic surgeons include:
  - Why is there bone loss?
  - Is bone loss predictable?
  - How can we avoid bone loss and ensure greater clinical success?
- Wolff’s law
Spinal Prostheses

- **Spinal fusion**
  - Weight borne by cage
  - Idea is to prevent further loads from compressing damaged spinal disk
  - Usually causes bone loss in adjacent vertebrae

- **Disk prostheses**
  - Completely replaces damaged disk
  - Allows adjacent vertebrae to move
  - Need to maintain DOFs
  - Wear particle formation
Spinal Prostheses

- High fidelity models provide accurate results
- Understanding the mechanics allows for further improvements to prosthesis designs
Dental Implants

Hans Yoo (2007)

Localized refinement

Mitchell Farrar (2010)

Chaiy Rungsiyakull (2011)
Multiphysics Finite Element Analysis

- Steady-state field problem

\[
\frac{\partial}{\partial x} \left( K_x \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( K_y \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left( K_z \frac{\partial \phi}{\partial z} \right) + Q = 0
\]

<table>
<thead>
<tr>
<th>PHYSICS TYPE</th>
<th>K_x, K_y, K_z</th>
<th>( \phi )</th>
<th>Q</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat Transfer</td>
<td>Thermal conductivities</td>
<td>Temperature</td>
<td>Internal heat generation</td>
</tr>
<tr>
<td>Incompressible fluid flow</td>
<td>Unity amount</td>
<td>Stream function or potential function</td>
<td>Q=0</td>
</tr>
<tr>
<td>Electrostatics</td>
<td>Permittivity</td>
<td>Electric potential</td>
<td>Charge density</td>
</tr>
<tr>
<td>Magnetostatics</td>
<td>Reluctivity</td>
<td>Magnetic potential</td>
<td>Charge density</td>
</tr>
</tbody>
</table>
Cochlear Implants

- CIs work by injecting electric current into the inner ear
- This causes auditory nerves to send action potentials to the brain
- Questions:
  - Which path does it take through the volume?
  - Will this evoke the desired/optimal neural response?
  - What could be changed to improve sound perception?
Summary

- Limitations of analytical methods provides rationale for FEM
- FEM solution process
  - Discretise the system into appropriate elements (2D/3D, linear/quadratic)
  - Combine elements into a global stiffness matrix and solve for equilibrium
  - Can be used for many different problem types, not just structural/mechanical
- Practical considerations
  - Trade-offs
  - Simplifying assumptions
  - Mesh refinement
  - Convergence
  - Contact conditions
  - Interpretation
  - Validation
- Example research and clinical applications
GROUP PROJECT UPDATES
No pressure...seriously
TO THE LABS!

START

FIND A MENU ITEM OR BUTTON WHICH LOOKS RELATED TO WHAT YOU WANT TO DO.

I CAN'T FIND ONE

I'VE TRIED THEM ALL

OK

CLICK IT.

Google the name of the program plus a few words related to what you want to do. Follow any instructions.

HAVE YOU BEEN TRYING THIS FOR OVER HALF AN HOUR?

NO

YES

ASK SOMEONE FOR HELP OR GIVE UP.

DID IT WORK?

NO

YES

YOU'RE DONE!