

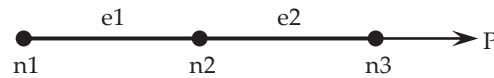
Chapter 2

Solution to Problems in Chapter 2 of FEACM Using ANSYS—2nd Edition

© Ever J. Barbero (2014) all rights reserved

Problem 2.1

Model



Properties:

$$L_{e1} = L_{e2} = \frac{1}{2} L = 375 \text{ mm}$$

$$E_{e1} = E_{e2} = E = 200000 \text{ MPa}$$

$$A_{e1} = A_{e2} = A = \frac{1}{4} \pi d^2 = 63.617 \text{ mm}^2$$

Conditions:

$$P = 100000 \text{ MPa}$$

$$u_{n1} = 0 \text{ (clamped point)}$$

FE Modeling

Element 1 (corresponding to nodes 1 and 2)

$$\text{Stiffness matrix: } [K]^{(1)} = \frac{E_{e1} A_{e1}}{L_{e1}} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 33929 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\text{Applied force vector: } \{f\}^{(1)} = \begin{Bmatrix} R \\ 0 \end{Bmatrix}$$

Element 2 (corresponding to nodes 2 and 3)

$$\text{Stiffness matrix: } [K]^{(2)} = \frac{E_{e2} A_{e2}}{L_{e2}} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 33929 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\text{Applied force vector: } \{f\}^{(2)} = \begin{Bmatrix} 0 \\ P \end{Bmatrix} = \begin{Bmatrix} 0 \\ 100000 \end{Bmatrix}$$

Global arrays:

$$\text{Stiffness matrix: } [K] = 33929 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

$$\text{Applied force: } \{f\} = \begin{Bmatrix} R \\ 0 \\ 100000 \end{Bmatrix}$$

$$\text{Displacement: } \{u\} = \begin{Bmatrix} 0 \\ u_{n2} \\ u_{n3} \end{Bmatrix}$$

The global system is expressed in the form: $[K]\{u\} = \{f\}$

The first column and the first row of the stiffness matrix can be removed together with the first row of both the force and displacement vectors because the displacement u_{n1} is already known. Solving for the two remaining displacements it's gotten that $u_{n2} = 2.9473$ mm and $u_{n3} = 5.8947$ mm.

$$\text{Displacement: } \{u\} = \begin{Bmatrix} 0 \\ 2.9473 \\ 5.8947 \end{Bmatrix}$$

The strain in each element is calculated based on the displacement by:

$$\varepsilon_x = \frac{du}{dx} = \frac{d}{dx}([N]^e \{u\}^e) = \begin{bmatrix} \frac{dN_1}{dx} & \frac{dN_2}{dx} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \frac{1}{L^e} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

$$\text{Element 1: } \varepsilon_x = \frac{1}{375} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{Bmatrix} 0 \\ 2.9473 \end{Bmatrix} = 0.0079 \text{ mm/mm}$$

$$\text{Element 2: } \varepsilon_x = \frac{1}{375} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{Bmatrix} 2.9473 \\ 5.8947 \end{Bmatrix} = 0.0079 \text{ mm/mm}$$

The stress in each element is calculated based on the strain by $\sigma_x = E\varepsilon_x$, then $\sigma_x = 1571.9$ MPa

Problem 2.2

The interpolation functions allow us to calculate the displacements in any point of the domain as long as we have found the nodal values by:

$$u(x) = N_1 u_1 + N_2 u_2 = \left(\frac{L^e - x}{L^e} \right) u_1 + \left(\frac{x}{L^e} \right) u_2$$

(a) Displacement at the point $x = 700$ mm

This point is inside of the element 2 of the discretization, then:

$$L^e = 375 \text{ mm}$$

$$u_1 = 2.9473 \text{ mm}$$

$$u_2 = 5.8947 \text{ mm}$$

$$x \text{ in local coordinates is: } 700 - 375 = 325 \text{ mm}$$

$$u(x = 700) = \left(\frac{375 - 325}{375} \right) * 2.9473 + \left(\frac{325}{375} \right) * 5.8947 = 5.5012 \text{ mm}$$

(b) Displacement at the point $x = 500$ mm

This point is inside of the element 2 of the discretization, then:

$$L^e = 375 \text{ mm}$$

$$u_1 = 2.9473 \text{ mm}$$

$$u_2 = 5.8947 \text{ mm}$$

$$x \text{ in local coordinates is: } 500 - 375 = 125 \text{ mm}$$

$$u(x = 500) = \left(\frac{375 - 125}{375} \right) * 2.9473 + \left(\frac{125}{375} \right) * 5.8947 = 3.9274 \text{ mm}$$

Problem 2.3

Let $\tilde{u} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$, it's possible to define $u(x) = [N]_{1 \times 3} \tilde{u}$; where $[N]$ is a matrix containing the shape

functions for the element.

$$[N]^T = \begin{bmatrix} \frac{x - x_2}{x_1 - x_2} \times \frac{x - x_3}{x_1 - x_3} \\ \frac{x - x_1}{x_2 - x_1} \times \frac{x - x_3}{x_2 - x_3} \\ \frac{x - x_1}{x_3 - x_1} \times \frac{x - x_2}{x_3 - x_2} \end{bmatrix} = \begin{bmatrix} \frac{x - h/2}{-h/2} \times \frac{x - h}{-h} \\ \frac{x}{h/2} \times \frac{x - h}{-h/2} \\ \frac{x}{h} \times \frac{x - h/2}{h/2} \end{bmatrix} = \frac{1}{h^2} \begin{bmatrix} (2x - h)(x - h) \\ -4x(x - h) \\ x(2x - h) \end{bmatrix}$$

The strain-displacement matrix is given by:

$$[B]^T = \partial_x [N]^T = \partial_x \left(\frac{1}{h^2} \begin{bmatrix} (2x - h)(x - h) \\ -4x(x - h) \\ x(2x - h) \end{bmatrix} \right) = \frac{1}{h^2} \begin{bmatrix} 4x - 3h \\ -8x + 4h \\ 4x - h \end{bmatrix}$$

The element stiffness matrix is given by: $[K] = \int_V [B]^T [C] [B] dV$

By Hooke's law in the unidimensional case, $[C] = E$; also $dV = A dx$. Then

$$[K] = \int_0^h [B]^T E [B] A dx = \frac{EA}{h^4} \int_0^h \begin{bmatrix} 4x-3h \\ -8x+4h \\ 4x-h \end{bmatrix} \begin{bmatrix} 4x-3h & -8x+4h & 4x-h \end{bmatrix} dx$$

$$[K] = \frac{EA}{h^4} \int_0^h \begin{bmatrix} (4x-3h)^2 & (4x-3h)(-8x+4h) & (4x-3h)(4x-h) \\ (4x-3h)(-8x+4h) & (-8x+4h)^2 & (-8x+4h)(4x-h) \\ (4x-3h)(4x-h) & (-8x+4h)(4x-h) & (4x-h)^2 \end{bmatrix} dx$$

$$[K] = \frac{EA}{3h} \begin{bmatrix} 7 & -8 & 1 \\ -8 & 16 & -8 \\ 1 & -8 & 7 \end{bmatrix}$$

The force vector is given by:

$$F = \int_h f(x) [N]^T dx + \int_S [N]^T t dS$$

Since $f(x) = f = cte$ and $\int_S [N]^T t dS = \text{applied forces in nodal points}$. Then:

$$F = \frac{f}{h^2} \int_0^h \begin{bmatrix} (2x-h)(x-h) \\ -4x(x-h) \\ x(2x-h) \end{bmatrix} dx + \begin{Bmatrix} P_1 \\ 0 \\ P_3 \end{Bmatrix}$$

$$F = \frac{1}{6} fh \begin{Bmatrix} 1 \\ 4 \\ 1 \end{Bmatrix} + \begin{Bmatrix} P_1 \\ 0 \\ P_3 \end{Bmatrix}$$

Problem 2.4

MATLAB® code:

```
% INPUT (PREPROCESSING)
clear; clc;
```

SOLUTIONS MANUAL for
Finite Element Analysis of Composite Materials Using ANSYS® 2nd Edition

```
% Material properties
E = 200000; % Young's Modulus
% Geometry
L = 750; % bar's length
A = 63.617; % bar cross section's area
% number of elements
nel = 20;
% element's length
le = L/nel;
% number of nodes
nnod = nel + 1;
% nodal coordinates
for i = 1:nnod
    coord(i) = (i-1)*le;
end
% connectivity
for i = 1:nel
    conn(i,:) = [i,(i+1)];
end
% Boundary conditions [node desp-value]
bc = [1 0];
% Applied forces [node force-vale]
nf = [nnod 100000];

% PROCESSING
% Global arrays
K = zeros(nnod,nnod);
F = zeros(nnod,1);
U = zeros(nnod,1);
% Assign the applied load
F(nf(1,1)) = nf(1,2);
% Assign the known displacement
U(bc(1,1)) = bc(1,2);
% Loop over the elements
for i = 1:nel
    % connectivity of the element
    nod1 = conn(i,1);
    nod2 = conn(i,2);
    % stiffness matrix of the element
    ke = E*A/le*[1 -1; -1 1];
    % Assembling of the global matrix
    K(nod1:nod2,nod1:nod2) = K(nod1:nod2,nod1:nod2) + ke;
end
```

```
% Removing the data for the known nodes
K(bc(1,1),:) = [];
K(:,bc(1,1)) = [];
% Solving for the unknown nodes
U(2:nnod,1) = inv(K)*F(2:nnod,1);
```

```
% Strain and Stress
for i = 1:nel
    % connectivity of the element
    nod1 = conn(i,1);
    nod2 = conn(i,2);
    % Strain, ex = dN/dx*u
    ex(i) = 1/le*[-1 1]*[U(nod1) U(nod2)]';
    % Stress, sx = E*ex
    sx = E*ex;
end
```

Problem 2.5

MATLAB® code:

```
% INPUT (PREPROCESSING)
clear; clc;
% Material properties
E = 200000; % Young's Modulus
% Geometry
L = 750; % bar's length
A = 63.617; % bar cross section's area
% number of elements
nel = 2;
% element's length
le = L/nel;
% number of nodes
nnod = 2*nel + 1;
% nodal coordinates
for i = 1:nnod
    coord(i) = (i-1)*0.5*le;
end
% connectivity
for i = 1:nel
    conn(i,:) = [(2*i-1),(2*i),(2*i+1)];
end
% Boundary conditions [node desp-value]
bc = [1 0];
```

```
% Applied forces [node force-value]
nf = [nnod 100000];

% PROCESSING
% Global arrays
K = zeros(nnod,nnod);
F = zeros(nnod,1);
U = zeros(nnod,1);
% Assign the applied load
F(nf(1,1)) = nf(1,2);
% Assign the known displacement
U(bc(1,1)) = bc(1,2);
% Loop over the elements
for i = 1:nel
    % connectivity of the element
    nod1 = conn(i,1);
    nod2 = conn(i,2);
    nod3 = conn(i,3);
    % stiffness matrix of the element
    ke = E*A/(3*le)*[7 -8 1; -8 16 -8; 1 -8 7];
    % Assembling of the global matrix
    K(nod1:nod3,nod1:nod3) = K(nod1:nod3,nod1:nod3) + ke;
end
% Removing the data for the known nodes
K(bc(1,1),:) = [];
K(:,bc(1,1)) = [];
% Solving for the unknown nodes
U(2:nnod,1) = inv(K)*F(2:nnod,1);

% Strain and Stress (at the mid point of the element)
for i = 1:nel
    % connectivity of the element
    nod1 = conn(i,1);
    nod2 = conn(i,2);
    nod3 = conn(i,3);
    x = coord(nod2);
    % Strain, ex = dN/dx*u
    ex(i) = 1/le^2*[(4*x - 3*le) (-8*x + 4*le) (4*x - le)]*[U(nod1) U(nod2) U(nod3)]';
    % Stress, sx = E*ex
    sx = E*ex;
end
```