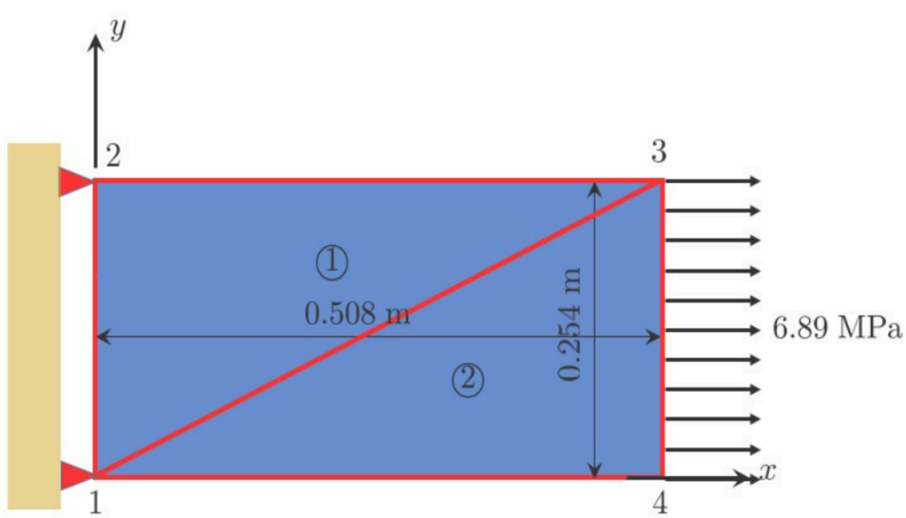
Thin plate under tension

2 solutions submitted (max: 3) | View my solutions

The thin plate shown in the figure has an out of plane thickness of 0.0254 m and is loaded with a traction as shown. It is dscretised into two T3 elements. The node and element numbers are as indicated. Do the following

1. Determine the equivalent nodal load vector for the structure
2. Determine the local stiffness matrices for each element and store them in `Ke1` and `Ke2`
3. Assemble the global stiffness into `Kg` and apply the specified boundary conditions.
4. Solve for the nodal displacements `U`
5. Determine the stress vector $\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}$ for each element and store them in `stress1`, `stress2` for elements 1 and 2 respectively.

Make suitable assumptions about reducing the 3-d problem into 2-d and use the isotropic linear elastic properties shown.



$E = 206.8$ GPa, $\nu = 0.3$

```
th=0.0254;      % Thickness
Ym = 206.8*10^9;
   %Ym: Young's modulus
Pr = 0.3;       % Pr: Poisson's ratio

type=1;         % 1 for plane stress and 2 for plane strain

nodal_coords=[0 0;0 0.2540;0.5080 0.2540;0.5080 0];
connectivity=[1 3 2;1 4 3];

coor = [ 0   0;
        0.5 0;
        1   0;
        1   0.5;
        1   1;
        0.5 1;
        0   1;
        0   0.5;
        0.6 0.5 ];                % Nodal coordinates (9 nodes)

conn = [1 2 9 8;
       2 3 4 9;
       9 4 5 6;
       8 9 6 7];                  % Element-node connectivity (4
elements)

%% ---------------------- BOUNDARY CONDITIONS ----------------------
force       = [];                 % External force DOFs
force_vals  = [];

fixed       = [1 2 4 6 10 12 13 14 15];         % Prescribed DOFs
fixed_vals  = [0 0 0 0 1e-4 1e-4 0 1e-4 0];     % Prescribed values

%% ----------------------- MATERIAL PROPERTIES ----------------------
E  = 1000;         % Young's modulus
nu = 0.3;          % Poisson ratio
D  = (E/(1-nu^2))*[1 nu 0;
                  nu 1 0;
                  0  0 (1-nu)/2];     % Plane stress constitutive matrix

%% ----------------------- INITIAL SETUP ---------------------------
dim        = 2;
```

```
nnode      = size(coor,1);
nelem      = size(conn,1);
nperelem   = size(conn,2);
ngauss     = 4;
nstress    = 3;

% Global matrices
Kglobal = zeros(2*nnode);
F       = zeros(2*nnode,1);
U       = zeros(2*nnode,1);

% Insert forces
F(force) = force_vals;

% Stress/strain storage at Gauss points
stress = zeros(ngauss, nstress*nelem);
strain = zeros(ngauss, nstress*nelem);

%% ---------------------- ASSEMBLY MAPPING -----------------------
dest = zeros(nelem, 2*nperelem);   % To map element DOFs → global DOFs

for ie = 1:nelem
   for j = 1:nperelem
       node = conn(ie,j);
       dest(ie, 2*j-1) = 2*node - 1;    % x DOF
       dest(ie, 2*j  ) = 2*node;        % y DOF
   end
end

%% ==================== ELEMENT ASSEMBLY LOOP ====================
for ie = 1:nelem

   nodes  = conn(ie,:);
   xlocal = coor(nodes,1);
   ylocal = coor(nodes,2);

   Ulocal      = U(dest(ie,:));
   stress_loc  = stress(:, (ie-1)*nstress + (1:nstress));
   strain_loc  = strain(:, (ie-1)*nstress + (1:nstress));

   [Klocal, ~, ~] = elemstiffQ4(xlocal, ylocal, Ulocal, stress_loc,
strain_loc, D);

   % Global assembly
```

```
   Kglobal(dest(ie,:), dest(ie,:)) = Kglobal(dest(ie,:), dest(ie,:)) +
Klocal;
end

%% ==================== APPLY BOUNDARY CONDITIONS =================
Kp = Kglobal;
Fp = F;

% Modify RHS (for non-zero prescribed values)
for k = 1:length(fixed)
   Fp = Fp - fixed_vals(k) * Kglobal(:,fixed(k));
end

% Impose constraints
for k = 1:length(fixed)
   dof = fixed(k);
   Kp(:,dof) = 0;
   Kp(dof,:) = 0;
   Kp(dof,dof) = 1;
   Fp(dof) = fixed_vals(k);
end

%% ===================== SOLVE FOR DISPLACEMENTS ==================
U = Kp \ Fp;

%% =================== COMPUTE STRESS & STRAIN ====================
for ie = 1:nelem

   nodes  = conn(ie,:);
   elemDOF = dest(ie,:);

   xlocal = coor(nodes,1);
   ylocal = coor(nodes,2);
   Ulocal = U(elemDOF);

   stress_loc  = stress(:, (ie-1)*nstress + (1:nstress));
   strain_loc  = strain(:, (ie-1)*nstress + (1:nstress));

   [~, stress_new, strain_new] =
elemstiffQ4(xlocal,ylocal,Ulocal,stress_loc,strain_loc,D);

   stress(:, (ie-1)*nstress + (1:nstress)) = stress_new;
   strain(:, (ie-1)*nstress + (1:nstress)) = strain_new;
end
```