```
% Input File: Two Trianglular Elements Under Axial Load
% Copyright (C) Arif Masud and Tim Truster
% This input file should be run prior to executing the FEA_Program routine.
% Format of required input:
응
응
                     = number of nodes in the mesh (length(NodeTable))
    numnp:
응
                     = number of elements in the mesh
응
    numel:
응
응
    nen:
                     = maximum number of nodes per element (4)
2
응
    PSPS:
                     = flag for plane stress ('s') or plane strain ('n')
응
응
                     = table of mesh nodal coordinates defining the
    NodeTable:
                       geometry of the mesh; format of the table is as
응
                       follows:
응
응
                           Nodes
                                                x-coord y-coord
응
                                    NodeTable = [x1]
                           n1
                                                          у1
%
                           n2
                                                   x2
                                                          у2
응
                            . . .
응
                           nnumnp
                                                   xnumnp ynumnp];
2
응
                     = table of mesh connectivity information, specifying
    ix:
                       how nodes are attached to elements and how materials
응
                       are assigned to elements; entries in the first nen
읒
                       columns correspond to the rows of NodeTable
응
2
                       representing the nodes attached to element e;
응
                       entries in the last nen+1 column are rows from MateT
응
                       signifying the material properties assigned to
                       element e; format of the table is as follows:
응
응
                           Elements
                                               n1
                                                    n2
                                                           n3
                                                                  n4
응
                                         ix = [eln1 eln2 eln3 eln4 elmat]
2
                                               e2n1 e2n2 e2n3 e2n4 e2mat
                           e2.
응
                                                     . .
                            . . .
                                                . .
                                                            . .
                                                                  . .
2
                                              values for element numel ];
                           enumel
읒
%
                     = table of mesh material properties for each distinct
    MateT:
응
                       set of material properties; these sets are
응
                       referenced by element e by setting the value of
응
                       ix(e,nen+1) to the row number of the desired
                       material set; format of the table is as follows:
%
                                                  E v t
응
                           Materials
응
                           mat1
                                         MateT = [E1 v1 t1]
2
                                                  E2 v2 t2
                           mat2
응
                                                   .. .. ..];
                            . . .
2
응
    BCLIndex:
                     = list of the number of boundary conditions and loads
응
                       applied to the mesh; first entry is the number of
응
                       prescribed displacements at nodes; second entry is
```

```
응
                       the number of nodal forces
응
응
    NodeBC:
                     = table of prescribed nodal displacement boundary
                       conditions; it contains lists of nodes, the
응
응
                       direction of the displacement prescribed (x=1, y=2),
응
                       and the value of the displacement (set 0 for fixed
                       boundary); the length of the table must match the
2
                       entry in BCLIndex(1), otherwise an error will result
                       if too few conditions are given or extra BCs will be
읒
                       ignored in the model input module; format of the
9
                       table is as follows:
응
                            BCs
                                              nodes direction value
                                    NodeBC = [bcln bcldir bclu
                            bc1
응
                            bc2
                                              bc2n
                                                    bc2dir bc2u
응
                            . . .
2
응
    NodeLoad:
                     = table of prescribed nodal forces; it contains lists
응
                       of nodes, the direction of the force prescribed
응
                       (x=1, y=2), and the value of the force; the length
                       of the table must match the entry in BCLIndex(2),
응
                       otherwise an error will result if too few conditions
응
응
                       are given or extra loads will be ignored in the
응
                       model input module; format of the table is as
읒
                       follows:
응
                            Loads
                                                  nodes direction value
응
                            P1
                                      NodeLoad = [ Pln
                                                          Pldir
                                                                   P1P
2
                            P2
                                                   P2n
                                                          P2dir
                                                                    P2P
                                                                    ..];
                                                           . .
                            . . .
2
읒
응
응
응
응
2
응
clear all; close all; clc
% Arbitrary data for assistance in defining the mesh
L = 1;
H = 1;
%Parameters for strain-energy function
mu1=100;
mu2=1000;
kappa=1000;
% Mesh Nodal Coordinates
NodeTable = [0]
                   0
             L
             L
                   Η
```

```
0
                 H];
numnp = length(NodeTable);
% Mesh Element Connectivities
ix = [1 \ 2 \ 3 \ 4 \ 1];
nen = 4;
numel = 1;
% Mesh Boundary Conditions and Loads
BCLIndex = [4 2]';
NodeBC = [1 1 0]
         1 2 0
          2 1 0
          2 2 0];
NodeLoad = [3 1 0]
          4 1 0];
LoadDist=[0.5;0.5];
tol=10^-10;
                      %tolerance used in N-R Method
                     %increments in which external load are applied
Fext_app=70;
Steps=100;
thick = 1;
PSPS = 'n';
FEA_Program
```











