

---

```

% Input File: Two Triangular 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:
%
%   numnp:           = number of nodes in the mesh (length(NodeTable))
%
%   numel:           = number of elements in the mesh
%
%   nen:             = maximum number of nodes per element (4)
%
%   PSPS:            = flag for plane stress ('s') or plane strain ('n')
%
%   NodeTable:       = table of mesh nodal coordinates defining the
%                     geometry of the mesh; format of the table is as
%                     follows:
%
%                     Nodes |           x-coord  y-coord
%                     n1   |   NodeTable = [x1    y1
%                     n2   |                   x2    y2
%                     ...   |                   ..    ..
%                     nnumnp |                   xnumnp ynumnp];
%
%   ix:              = table of mesh connectivity information, specifying
%                     how nodes are attached to elements and how materials
%                     are assigned to elements; entries in the first nen
%                     columns correspond to the rows of NodeTable
%                     representing the nodes attached to element e;
%                     entries in the last nen+1 column are rows from MateT
%                     signifying the material properties assigned to
%                     element e; format of the table is as follows:
%
%                     Elements |           n1    n2    n3    n4    mat
%                     e1      |   ix = [eln1  eln2  eln3  eln4  elmat
%                     e2      |           e2n1  e2n2  e2n3  e2n4  e2mat
%                     ...      |           ..    ..    ..    ..    ..
%                     numel    |   values for element numel  ];
%
%   MateT:           = table of mesh material properties for each distinct
%                     set of material properties; these sets are
%                     referenced by element e by setting the value of
%                     ix(e,nen+1) to the row number of the desired
%                     material set; format of the table is as follows:
%
%                     Materials |           E    v    t
%                     mat1     |   MateT = [E1  v1  t1
%                     mat2     |           E2  v2  t2
%                     ...       |           ..  ..  ..];
%
%   BCLIndex:        = list of the number of boundary conditions and loads
%                     applied to the mesh; first entry is the number of
%                     prescribed displacements at nodes; second entry is

```

---



---

```

        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];

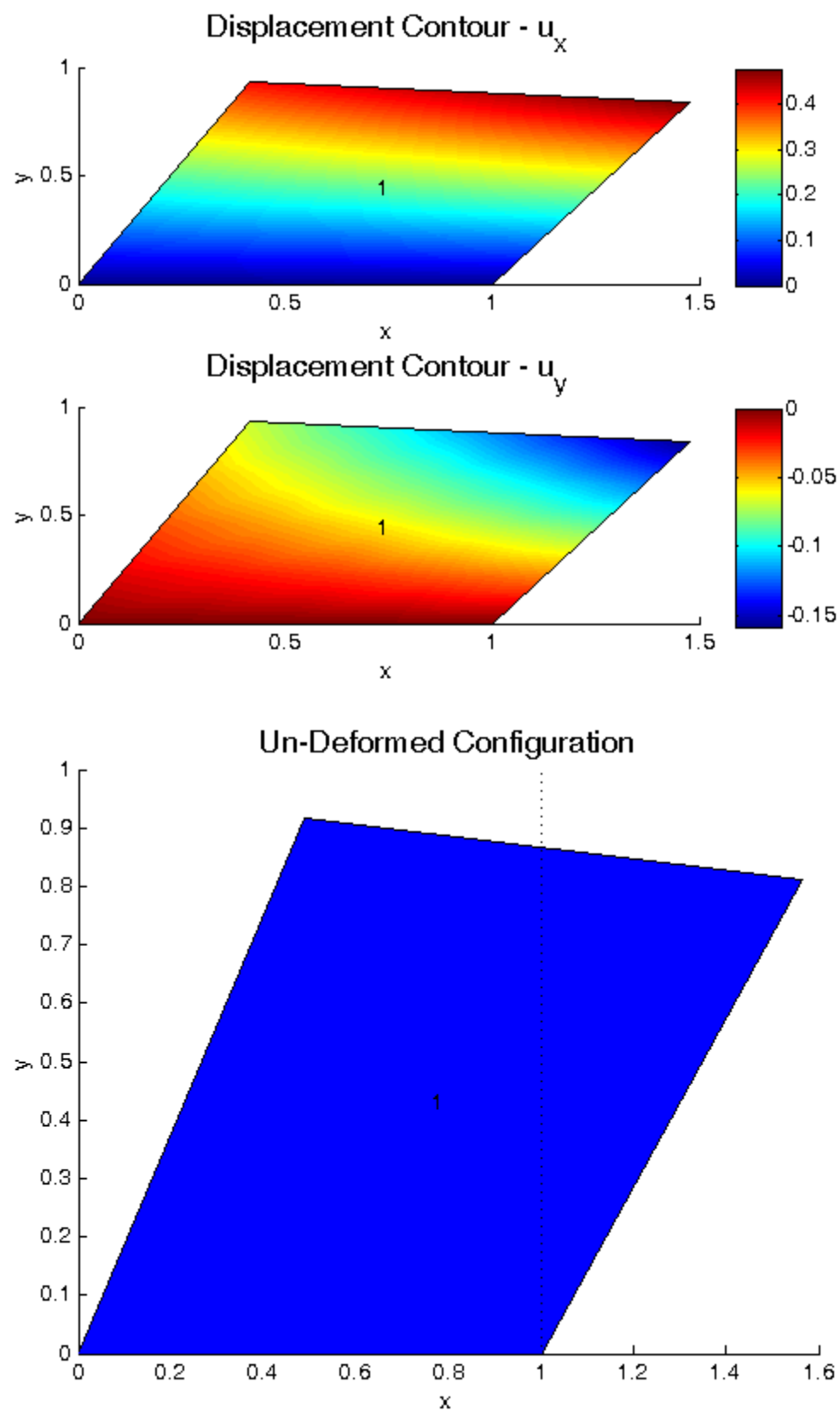
LoadDist=[0.5;0.5];
tol=10^-10;           %tolerance used in N-R Method
Fext_app=70;          %increments in which external load are applied

Steps=100;
thick = 1;
PSPS = 'n';

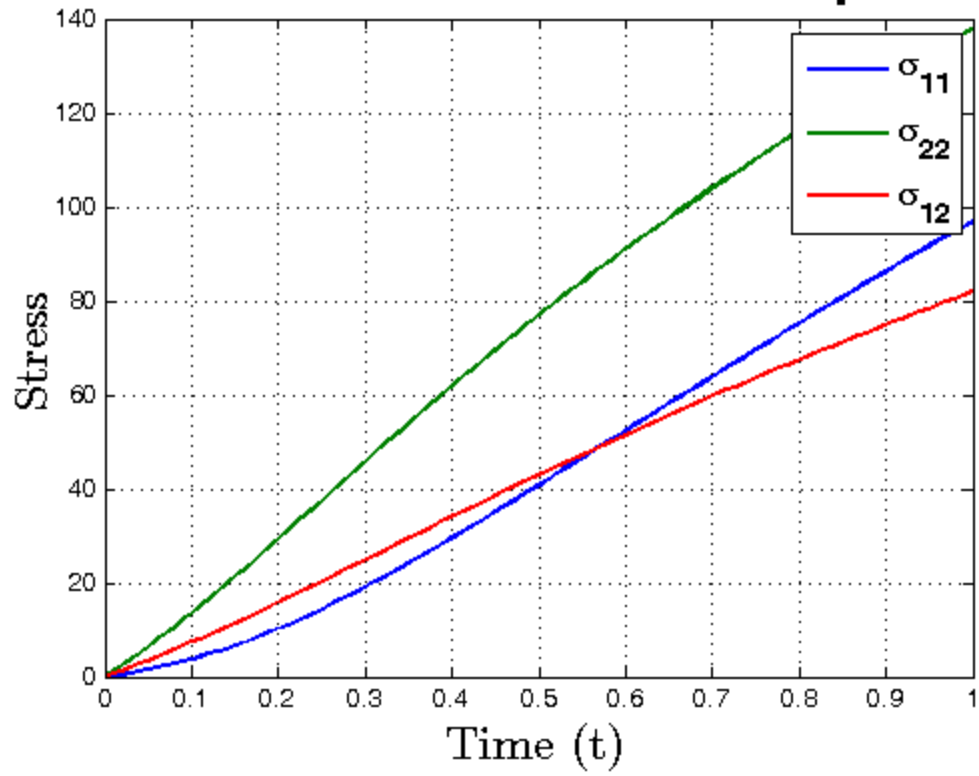
FEA_Program

```

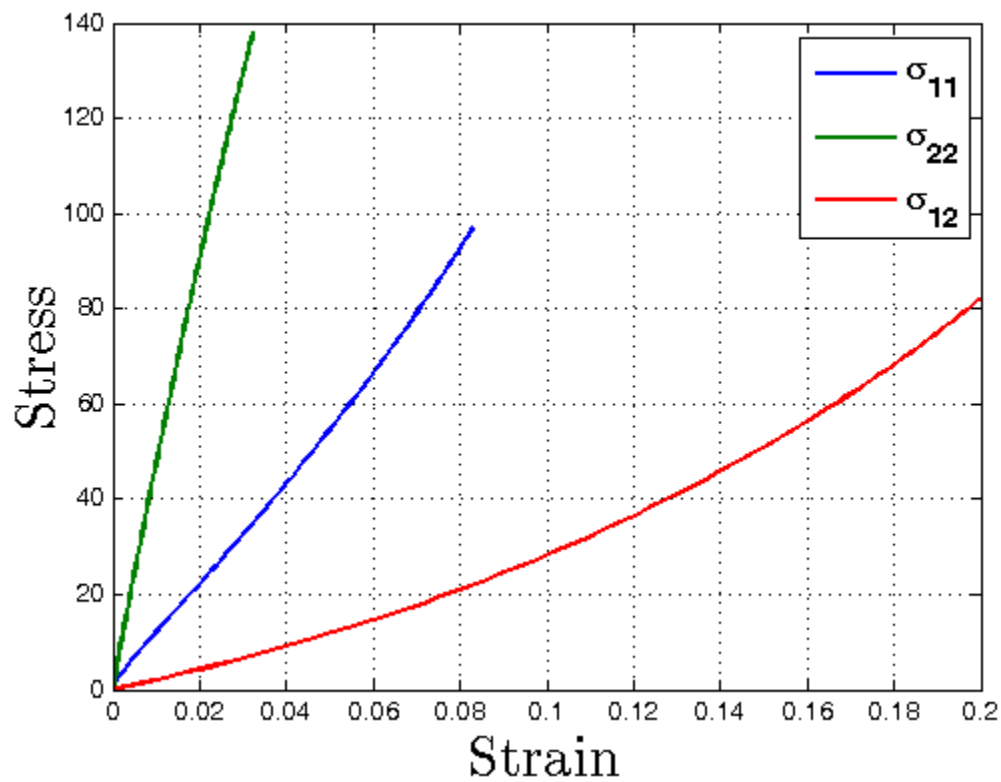
---

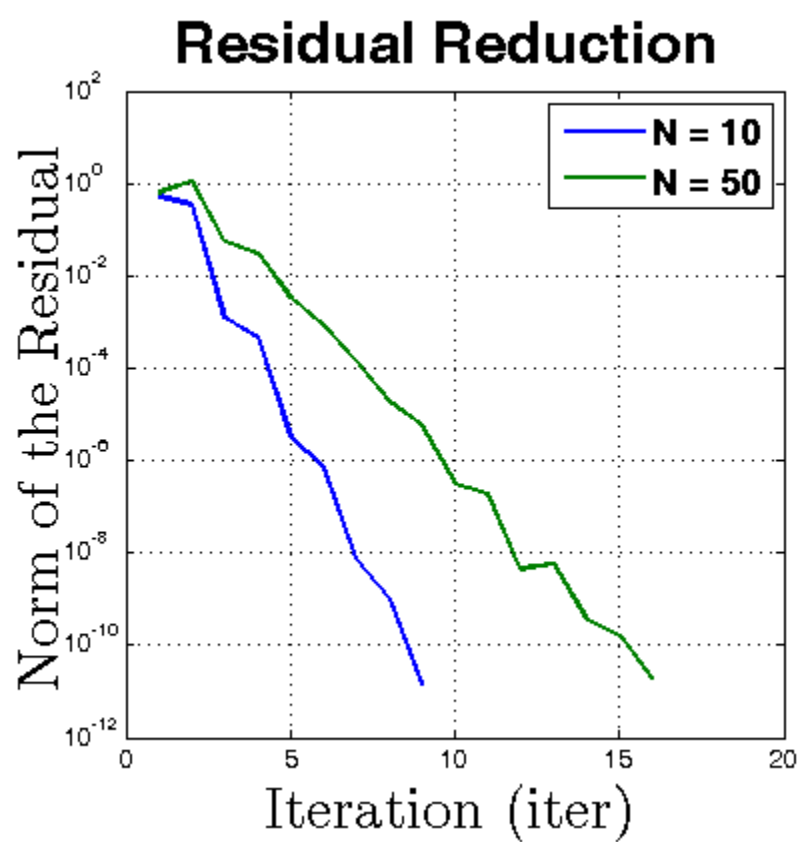
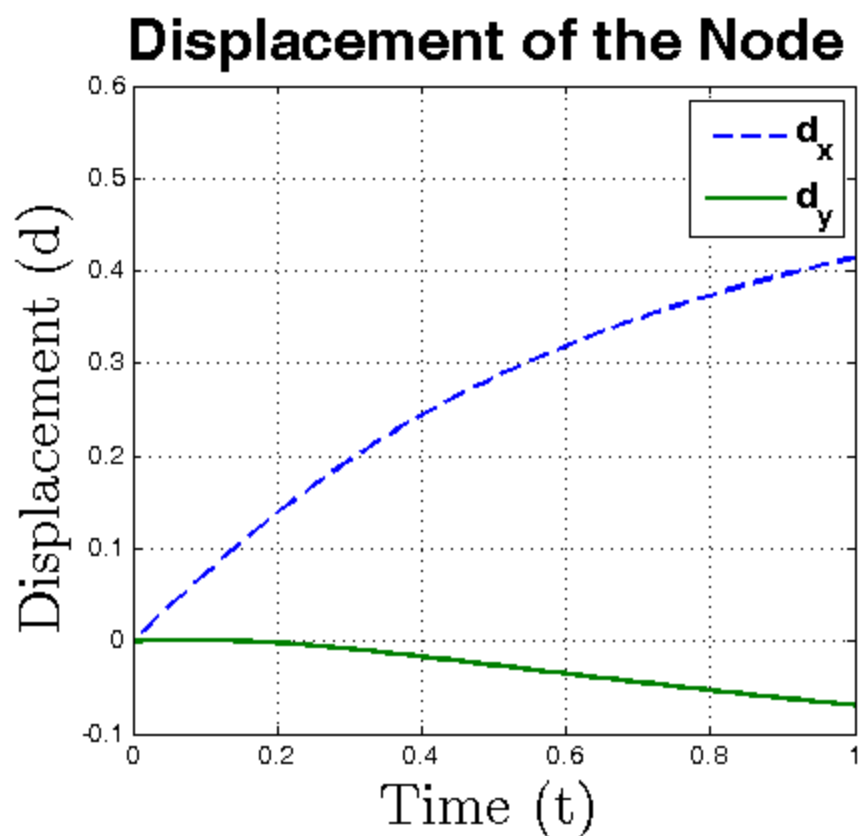


## Stress at each load step



## Stress-Strain





---

*Published with MATLAB® R2013a*