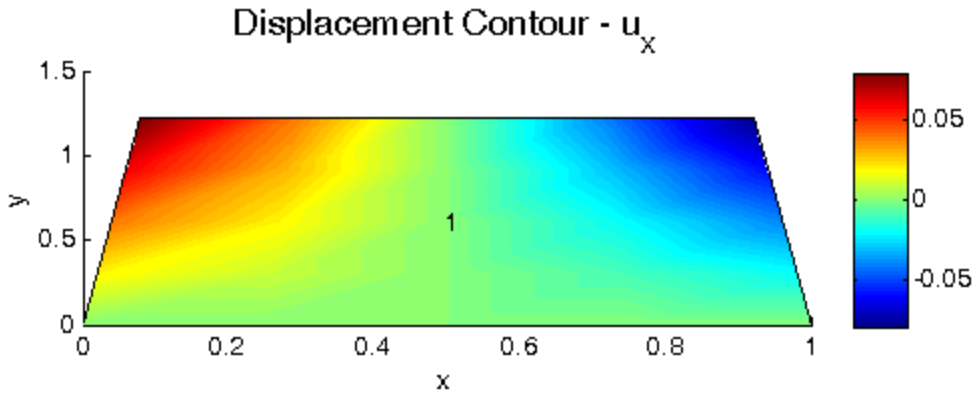
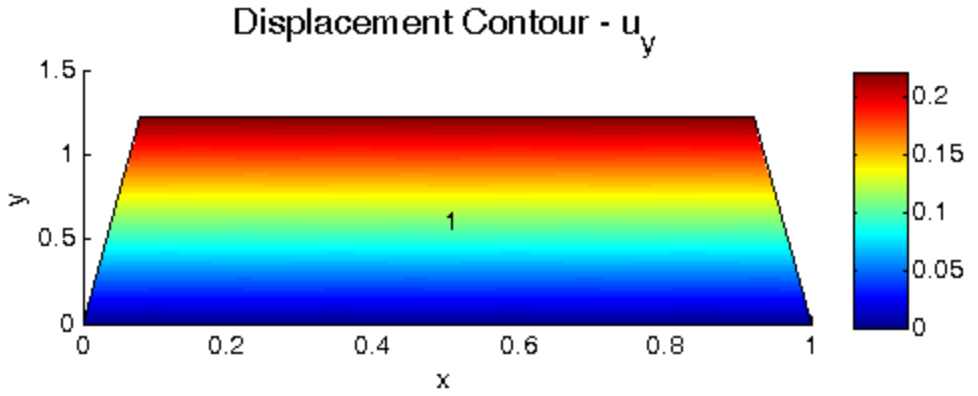
```
% Input File: Two Trianglular Elements Under Axial Load
%
% Copyright (C) Arif Masud and Tim Truster
%
% This input file should be run prior to executing the FEA_Program routine.
%
% Format of required input:
%
%   numnp:           = number of nodes in the mesh (length(NodeTable))
%
%   numel:           = 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 = [e1n1  e1n2  e1n3  e1n4 e1mat
%                          e2        |        e2n1  e2n2  e2n3  e2n4 e2mat
%                          ...       |         ..    ..    ..    ..   ..
%                          enumel    |        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
```

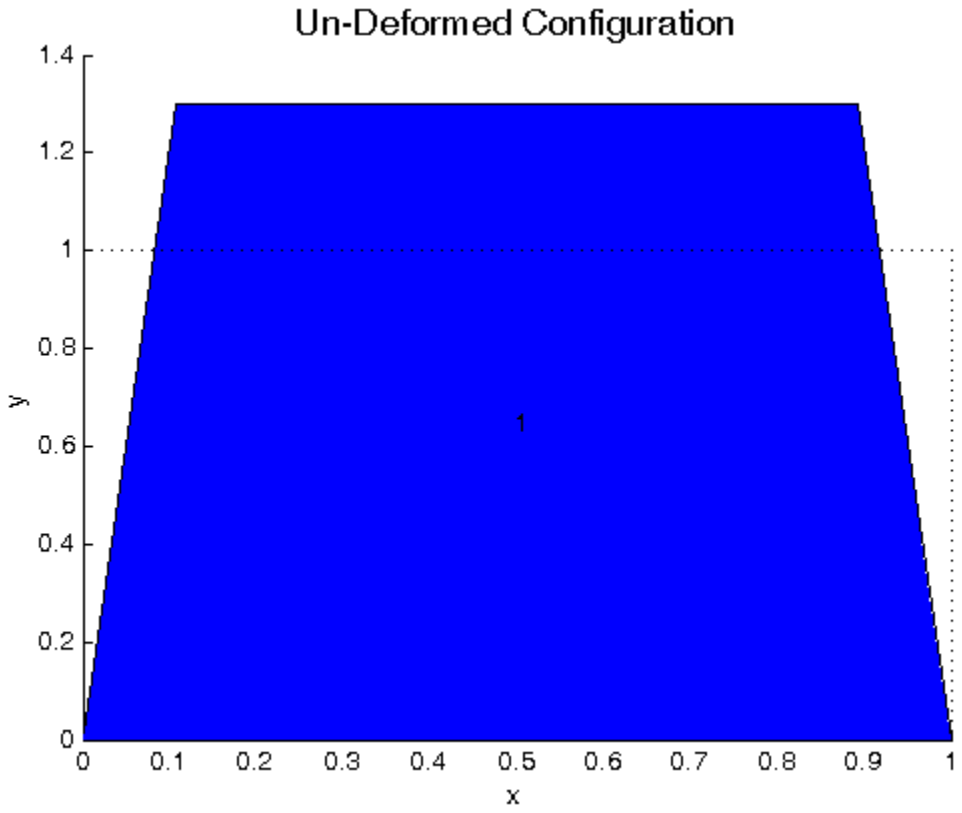
```
%                       the number of nodal forces
%
%   NodeBC:           = table of prescribed nodal displacement boundary
%                       conditions; it contains lists of nodes, the
%                       direction of the displacement prescribed (x=1, y=2),
%                       and the value of the displacement (set 0 for fixed
%                       boundary); the length of the table must match the
%                       entry in BCLIndex(1), otherwise an error will result
%                       if too few conditions are given or extra BCs will be
%                       ignored in the model input module;  format of the
%                       table is as follows:
%                           BCs  |            nodes direction value
%                           bc1  |  NodeBC = [bc1n   bc1dir   bc1u
%                           bc2  |            bc2n   bc2dir   bc2u
%                           ...  |             ..     ..       .. ];
%
%   NodeLoad:         = table of prescribed nodal forces; it contains lists
%                       of nodes, the direction of the force prescribed
%                       (x=1, y=2), and the value of the force; the length
%                       of the table must match the entry in BCLIndex(2),
%                       otherwise an error will result if too few conditions
%                       are given or extra loads will be ignored in the
%                       model input module; format of the table is as
%                       follows:
%                           Loads  |              nodes direction value
%                           P1     |  NodeLoad = [ P1n    P1dir    P1P
%                           P2     |               P2n    P2dir    P2P
%                           ...    |               ..     ..       .. ];
%
%
%           4 -------------- 3       2
%           |                |       | \
%           |                |       |  \
%           |                |       |   \
%           |                |       |    \
%           |                |       |     \
%           |                |       |      \
%           1 -------------- 2       3-------1
%
```

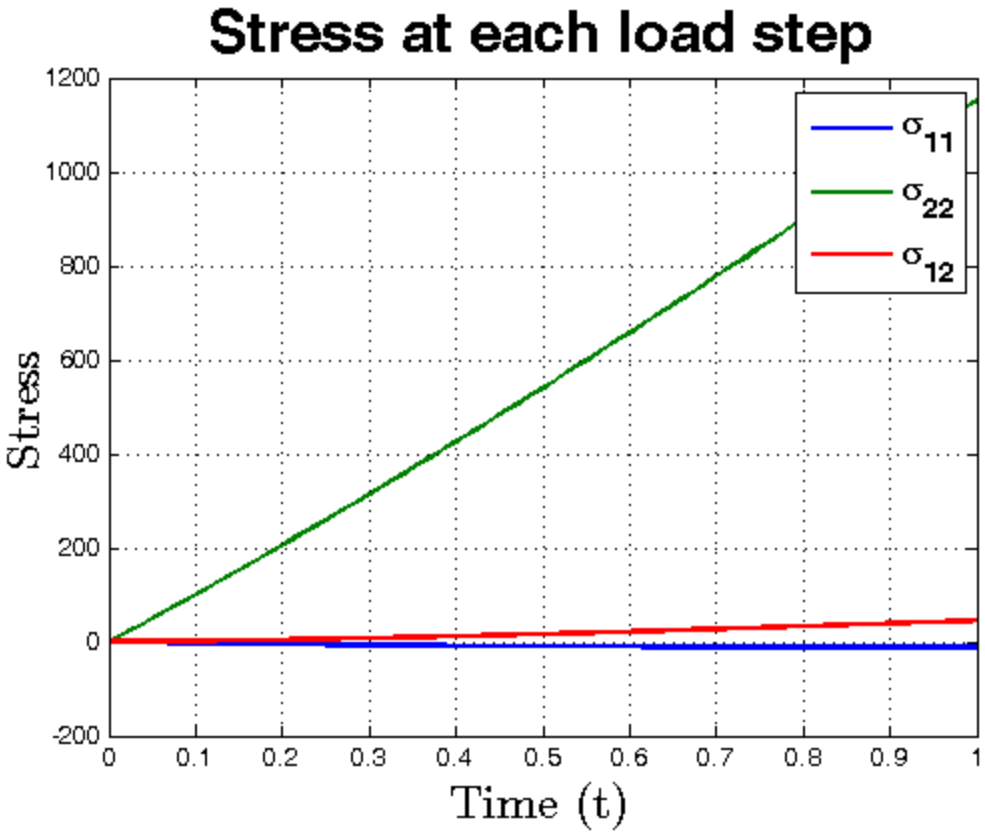
```
clear all;close all; clc

% Arbitrary data for assistance in defining the mesh

L = 1;
H = 1;

%Parameters for strain-energy function (Units consistent throughout)

mu1=100;
mu2=1000;
kappa=1000;
```

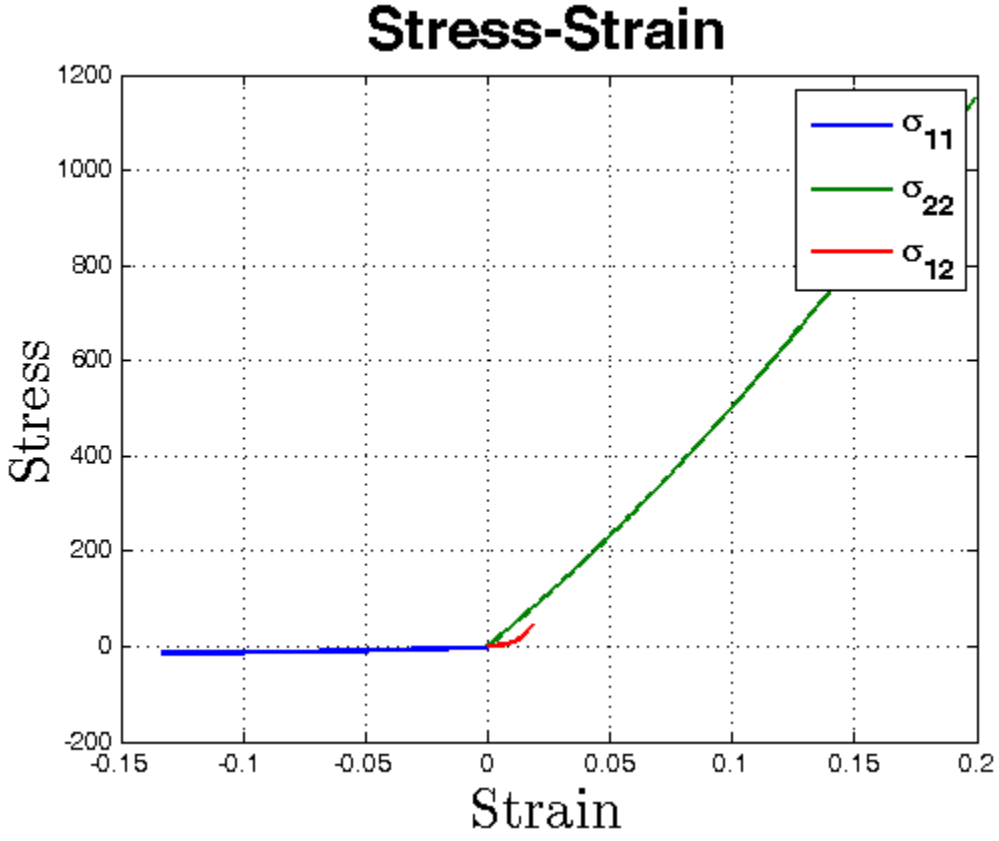
```
% Mesh Nodal Coordinates
NodeTable = [0     0
             L     0
             L     H
             0     H];

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  2   0
            4  2   0];

LoadDist=[0.5;0.5];   % Distribution of the Load on the two points.
tol=10^-12;           %tolerance used in N-R Method
Fext_app=1060;        %increments in which external load are applied

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

% Calling the FEA_Program to run the file:
FEA_Program
```

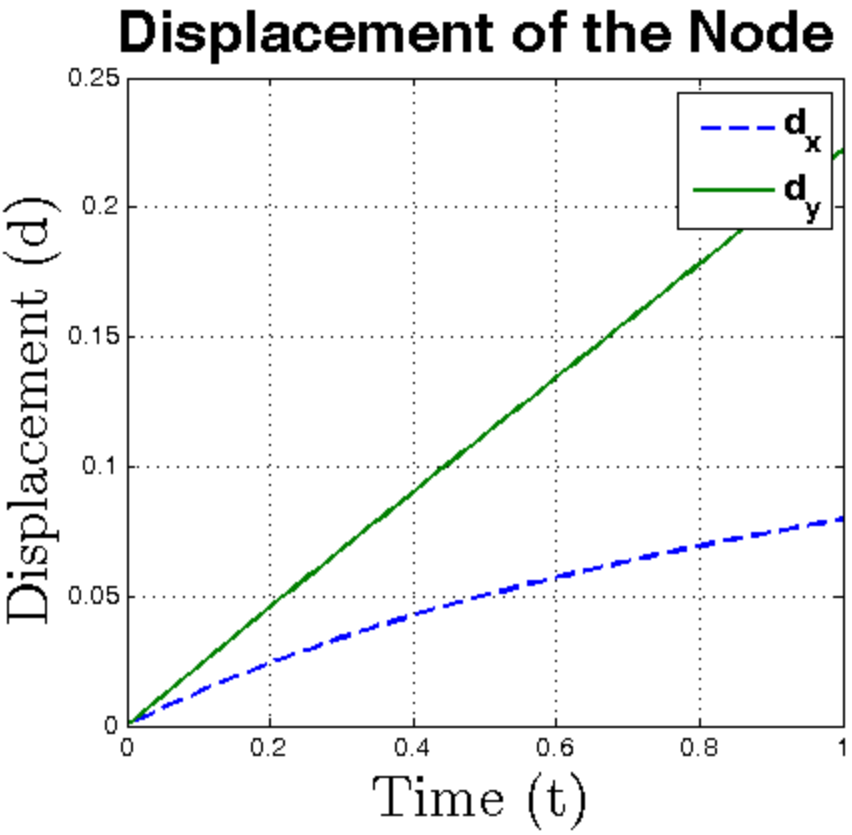

Displacement Contour - u_x
1.5
1
0.5
0
y
0
0.2
0.4
0.6
0.8
1
x
1
0.05
0
-0.05

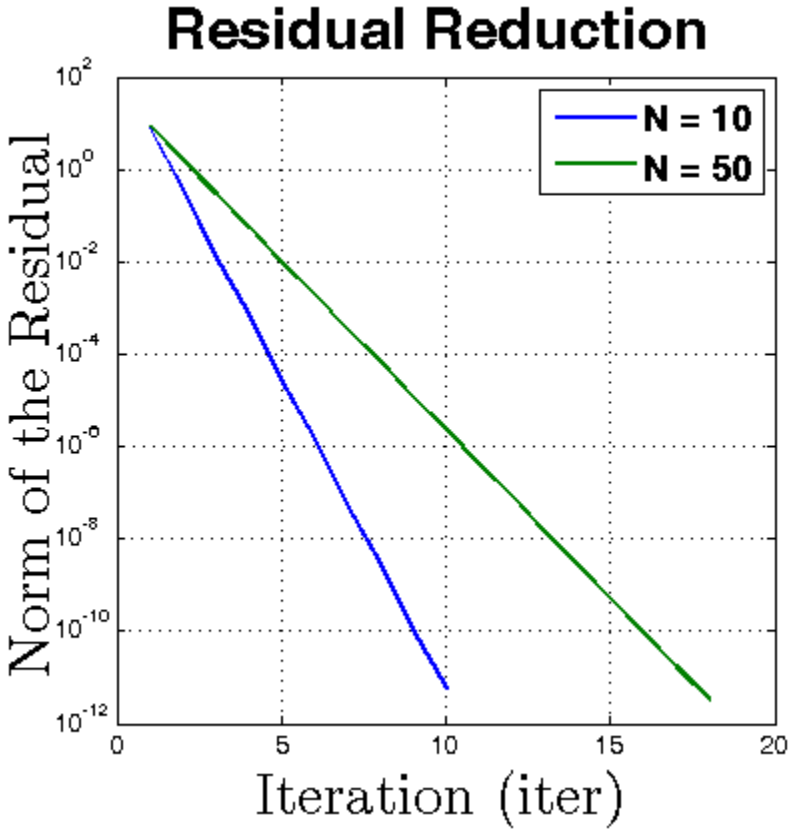

Displacement Contour - u_y
1.5
1
0.5
0
y
0
0.2
0.4
0.6
0.8
1
x
1
0.2
0.15
0.1
0.05
0


Un-Deformed Configuration
1.4
1.2
1
0.8
0.6
0.4
0.2
0
y
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1
x
1


Stress at each load step
Stress
Time (t)
σ11
σ22
σ12
1200
1000
800
600
400
200
0
-200
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1


Stress-Strain
Stress
Strain
σ11
σ22
σ12
1200
1000
800
600
400
200
0
-200
-0.15
-0.1
-0.05
0
0.05
0.1
0.15
0.2


Displacement of the Node
Displacement (d)
Time (t)
d_x
d_y
0.25
0.2
0.15
0.1
0.05
0
0
0.2
0.4
0.6
0.8
1


Residual Reduction
Norm of the Residual
Iteration (iter)
N = 10
N = 50
10^2
10^0
10^-2
10^-4
10^-6
10^-8
10^-10
10^-12
0
5
10
15
20

Published with MATLAB® R2013a