```
% An analysis of nodal reaction forces, nodal deformations, element
% internal forces, and element internal stresses for a truss of a
 certain
% configuration is required. This analysis must use the direct
 stiffness
% method. A plot of maximum stress vs loaded mass is also desired, as
 is a
% deformed shape plot for Case 5.

% For this particular truss (UBID 50084114), height and width are both
% 4 meters. It is composed of magnesium alloy, and the masses loaded
% are 150 kg. Material properties are assumed to be those of Magnesium
% AM60B Cast Alloy (the properties of this alloy can be found at the
% following link: http://www.azom.com/article.aspx?ArticleID=9237#4).
 Case
% 1 has a mass at joint 2. Case 2 has a mass at joint 2 and 3. Case 3
 has a
% mass at joint 2, 3, and 4. Case 4 has a mass at joint 2, 3, 4, and
 5.
% Case 5 has a mass at joint 2, 3, 4, 5, and 6.

% The effects of member weight and member buckling are ignored in this
% analysis, and all joints are assumed to be pin joints.

% The truss was analyzed using a script and function. The script
 stores
% user inputs (load vector, E, A, nodes, etc) for the parameters of
 the
% truss, and passes them along to the function TrussDirectStiffness.
 It is
% also responsible for plotting maximum stress vs mass loaded, and
% creating a fit function along with calculating the goodness of fit
 for
% that function.

% The TrussDirectStiffness function takes user inputs for the b
 vector,
% along with the node coordinates, members, E, A, and other
 information,
% and uses this to create localized stiffness matrices and concatenate
% them. Finally, it solves the resulting system of equations (using
% boundary conditions to eliminate rows and columns of the K matrix
 and b
% and u vectors that would otherwise render the system unsolvable) and
% returns a variety of outputs.

% For each case, the reactions, nodal deformations, internal forces,
 and
% internal stresses are outputted to the user. The highest magnitude
 axial
% stresses and nodal displacements are also shown. A yielding check is
% performed (if the user so desires) as a preliminary check for member
```

```
% failure. The MATLAB function TrussDirectStiffness also outputs a
 shape
% plot if prompted to do so, which shows a representation of the truss
 in its
% deformed state.

% For the analysis results of each case, along with a test case
 showing the
% deformed shape plot in greater detail, see the code output. Design
% details are included in the extensive comments in the code itself.

clear all; close all; clc; % Clear everything, close everything.
% TRUSS PARAMETERS ENTERED HERE
nodes = [0 0;4 0;8 0;12 0;16 0;20 0;24 0;4 4;20 4;8 8;12 8;16 8]; %
 Enter the coordinates of our joints (meters) [x y].
elements = [1 2;1 8;2 3;2 8;3 4;3 10;4 5;4 12;4 11;5 6;5 9;5 12;6 7;6
 9;8 3;8 10;9 7;10 4;10 11;11 12;12 9]; % Enter the joints that our
 members connect [startjoint endjoint].
E = 45000000000; % Enter the Modulus of Elasticity here (Pa). For
 magnesium alloy, this is 45 GPa.
A = 0.00755; % Enter our area here (m^2).
sigmay = 130000000; % Enter our yield strength here (for magnesium
 alloy, this is 130 MPa for both compressive and tensile stresses).
unconstrained = [3 4 5 6 7 8 9 10 11 12 15 16 17 18 19 20 21 22 23
 24]; % Enter boundary conditions: the unconstrained joint directions
 (3 4 means joint 2 is free in x and y).
constrained = [1 2 13 14]; % Enter boundary conditions: the
 constrained joint directions (1 2 means joint 1 cannot move in x and
 y).
% END OF TRUSS PARAMETERS
% C1
load1 = [0;0;0;150;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0]; % Enter
 the load vector (kg).
b1=-9.81*load1; % The load vector is converted into N, and comprises
 the initial b vector fed into the function.
[maxstress(1,1)] =
 TrussDirectStiffness(nodes,elements,b1,E,A,unconstrained,constrained,1,sigmay); %
 Call TrussDirectStiffness function with defined parameters, and
 obtain the max stress (which is stored in a column vector).
% C2
load2 = [0;0;0;150;0;150;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0];
b2=-9.81*load2;
[maxstress(2,1)] =
 TrussDirectStiffness(nodes,elements,b2,E,A,unconstrained,constrained,1,sigmay);
% C3
load3 = [0;0;0;150;0;150;0;150;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0];
b3=-9.81*load3;
[maxstress(3,1)] =
 TrussDirectStiffness(nodes,elements,b3,E,A,unconstrained,constrained,1,sigmay);
% C4
load4 = [0;0;0;150;0;150;0;150;0;150;0;0;0;0;0;0;0;0;0;0;0;0;0;0];
b4=-9.81*load4;
[maxstress(4,1)] =
 TrussDirectStiffness(nodes,elements,b4,E,A,unconstrained,constrained,1,sigmay);
```

```
% C5
load5 = [0;0;0;150;0;150;0;150;0;150;0;150;0;0;0;0;0;0;0;0;0;0;0;0];
b5=-9.81*load5;

figure1=figure('Position', [100, 100, 800, 600]); % Plot options
 including vertical/horizontal size.
[maxstress(5,1)] =
 TrussDirectStiffness(nodes,elements,b5,E,A,unconstrained,constrained,1,sigmay,1);
 Note that the deformed shape plot has been enabled.

mass(1,1) = sum(load1); % Calculate the total weight loaded on the
 truss for each loading scenario.
mass(2,1) = sum(load2);
mass(3,1) = sum(load3);
mass(4,1) = sum(load4);
mass(5,1) = sum(load5);

figure2=figure('Position', [100, 100, 800, 600]); % Plot options
 including vertical/horizontal size.
scatter(mass,abs(maxstress),'filled') % A scatter plot is most
 appropriate for these discrete points, this one plots the max stress
 vs total mass loaded.
xlabel('Total Mass (kg)','fontsize',16);
ylabel('Maximum Stress Magnitude (Pa)','fontsize',16);
title('Maximum Stress Magnitude vs Total Mass','fontsize',16); %
 Title, x label, y label, x interval settings, etc.
set(gca,'XTick',0:150:mass(end,1));
hold on

p = polyfit(mass(:,1),abs(maxstress(:,1)),3); % Fit the max stress vs
 mass scatterplot with a polynomial, order 3.
yfit = @(x) p(1)*x.^3 + p(2)*x.^2 + p(3)*x +p(4); % Using the four
 coefficients outputted by the built in polyfit function, we define
 fit function.
fplot(yfit,[mass(1,1) mass(end,1)]); % Plot our fit function.

resid = abs(maxstress(:,1)) - feval(yfit,mass(:,1)); % This series of
 calculations evaluates goodness of fit.
SSresid = sum(resid.^2);
SStotal = (length(maxstress(:,1))-1)*var(maxstress(:,1));
r2 = 1-SSresid/SStotal; % r2, the coefficient of determination, is
 calculated.

disp(['The coefficient of determination r^2 is ' num2str(r2) ' for
 this fitting function.']); % Text output for user.
```

```
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

            1       613.12       1226.2
            2            0      -1471.5
            3            0            0
            4            0            0
```

```
             5           0           0
             6           0           0
             7     -613.12      245.25
             8           0           0
             9           0           0
            10           0           0
            11           0           0
            12           0           0

Nodal Displacements (m) ----- [Node     X     Y]

displacement =

             1           0           0
             2  7.2185e-06 -9.4672e-05
             3  1.4437e-05 -7.4924e-05
             4  1.2993e-05 -4.3248e-05
             5  8.6623e-06 -2.4565e-05
             6  4.3311e-06 -1.2035e-05
             7           0           0
             8  3.6514e-05 -7.7348e-05
             9 -3.8678e-06 -1.2035e-05
            10  4.3122e-07 -5.7599e-05
            11 -3.8999e-06 -4.3248e-05
            12  -8.231e-06 -2.4565e-05

Element Internal Forces (N) ----- [Element     Internal Force]

force =

             1      613.12
             2     -1734.2
             3      613.12
             4      1471.5
             5     -122.62
             6      735.75
             7     -367.88
             8       274.2
             9  2.8778e-13
            10     -367.87
            11 -2.0349e-13
            12 -1.4389e-13
            13     -367.87
            14           0
            15     -1040.5
            16     -693.67
            17     -346.84
            18      -274.2
            19     -367.87
            20     -367.87
            21     -346.84

Element Axial Stresses (Pa) ----- [Element     Axial Stress]
```

```
stress =

           1        81209
           2  -2.2969e+05
           3        81209
           4    1.949e+05
           5       -16242
           6        97450
           7       -48725
           8        36318
           9   3.8116e-11
          10       -48725
          11  -2.6952e-11
          12  -1.9058e-11
          13       -48725
          14            0
          15  -1.3782e+05
          16       -91877
          17       -45939
          18       -36318
          19       -48725
          20       -48725
          21       -45939

The largest magnitude axial stress is -229692.6332 (Pa).
The largest magnitude displacement is -9.4672e-05 (m).
Yielding due to axial stress does not occur.
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

           1       1348.9       2207.3
           2            0      -1471.5
           3            0      -1471.5
           4            0            0
           5            0            0
           6            0            0
           7      -1348.9       735.75
           8            0            0
           9            0            0
          10            0            0
          11            0            0
          12            0            0

Nodal Displacements (m) ----- [Node     X     Y]

displacement =

           1            0            0
           2   1.0106e-05   -0.0001696
           3   2.0212e-05  -0.00018633
           4   2.1656e-05  -0.00010524
           5   1.4437e-05  -6.3135e-05
           6   7.2185e-06  -3.6599e-05
```

```
        7            0            0
        8    7.877e-05  -0.00015227
        9  -1.2099e-05  -3.6599e-05
       10   1.1853e-05  -0.00013436
       11  -1.1409e-06  -0.00010524
       12  -1.4134e-05  -6.3135e-05

Element Internal Forces (N) ----- [Element     Internal Force]

force =

        1       858.37
        2      -3121.5
        3       858.37
        4       1471.5
        5       122.63
        6       2207.3
        7      -613.13
        8       822.59
        9   1.1511e-12
       10      -613.13
       11  -6.1047e-13
       12   5.7556e-13
       13      -613.12
       14            0
       15      -1040.5
       16        -2081
       17      -1040.5
       18      -822.59
       19      -1103.6
       20      -1103.6
       21      -1040.5

Element Axial Stresses (Pa) ----- [Element     Axial Stress]

stress =

        1   1.1369e+05
        2  -4.1345e+05
        3   1.1369e+05
        4    1.949e+05
        5        16242
        6   2.9235e+05
        7       -81209
        8   1.0895e+05
        9   1.5247e-10
       10       -81209
       11  -8.0857e-11
       12   7.6233e-11
       13       -81209
       14            0
       15  -1.3782e+05
       16  -2.7563e+05
       17  -1.3782e+05
```

```
      18  -1.0895e+05
      19  -1.4618e+05
      20  -1.4618e+05
      21  -1.3782e+05

The largest magnitude axial stress is -413446.7398 (Pa).
The largest magnitude displacement is -0.00018633 (m).
Yielding due to axial stress does not occur.
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

       1       2084.6         2943
       2            0      -1471.5
       3            0      -1471.5
       4            0      -1471.5
       5            0            0
       6            0            0
       7      -2084.6       1471.5
       8            0            0
       9            0            0
      10            0            0
      11            0            0
      12            0            0

Nodal Displacements (m) ----- [Node     X     Y]

displacement =

       1            0            0
       2   1.0106e-05  -0.00021284
       3   2.0212e-05  -0.00024832
       4   2.1656e-05  -0.00019794
       5   1.4437e-05  -0.00012513
       6   7.2185e-06  -7.9847e-05
       7            0            0
       8   9.7517e-05  -0.00019552
       9  -3.0846e-05  -7.9847e-05
      10   2.4846e-05  -0.00019635
      11  -1.1409e-06  -0.00019794
      12  -2.7128e-05  -0.00012513

Element Internal Forces (N) ----- [Element     Internal Force]

force =

       1       858.37
       2        -4162
       3       858.37
       4       1471.5
       5       122.63
       6       2207.3
       7      -613.13
       8       1645.2
```

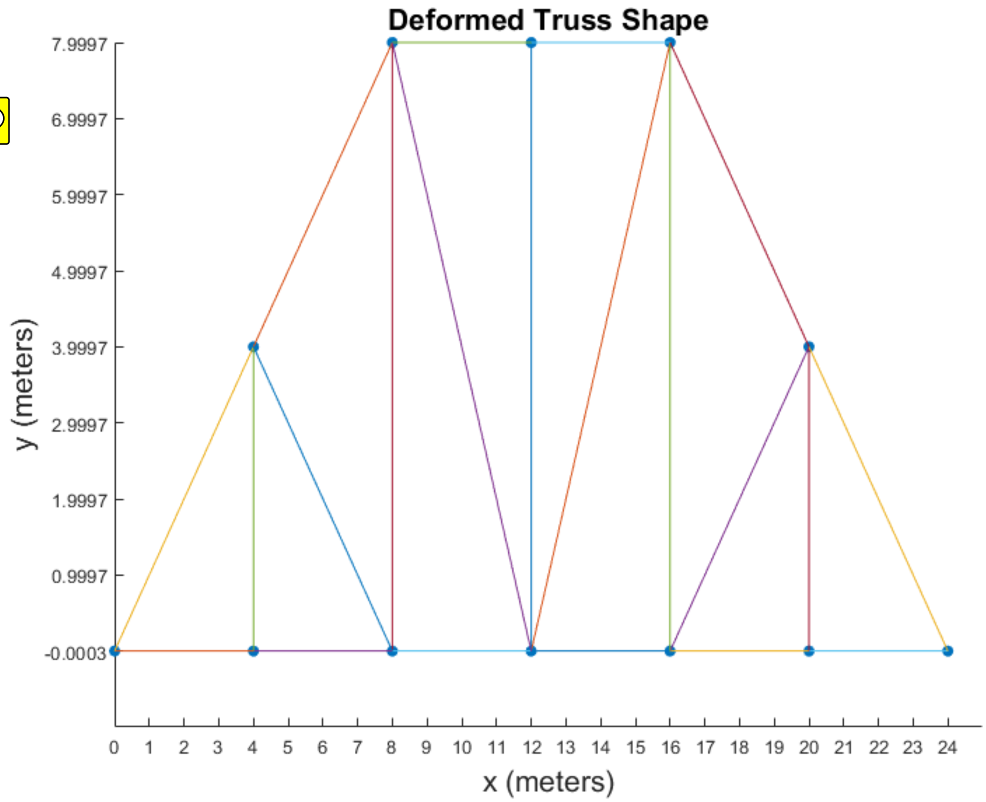
```
             9   1.1511e-12
            10      -613.12
            11  -1.2209e-12
            12   1.1511e-12
            13      -613.12
            14            0
            15      -1040.5
            16      -3121.5
            17        -2081
            18   1.3835e-12
            19      -2207.3
            20      -2207.3
            21        -2081

Element Axial Stresses (Pa) ----- [Element     Axial Stress]

stress =

             1   1.1369e+05
             2  -5.5126e+05
             3   1.1369e+05
             4    1.949e+05
             5        16242
             6   2.9235e+05
             7       -81209
             8   2.1791e+05
             9   1.5247e-10
            10       -81209
            11  -1.6171e-10
            12   1.5247e-10
            13       -81209
            14            0
            15  -1.3782e+05
            16  -4.1345e+05
            17  -2.7563e+05
            18   1.8325e-10
            19  -2.9235e+05
            20  -2.9235e+05
            21  -2.7563e+05

The largest magnitude axial stress is -551262.3197 (Pa).
The largest magnitude displacement is -0.00024832 (m).
Yielding due to axial stress does not occur.
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

             1       2820.4       3433.5
             2            0      -1471.5
             3            0      -1471.5
             4            0      -1471.5
             5            0      -1471.5
             6            0            0
             7      -2820.4       2452.5
```

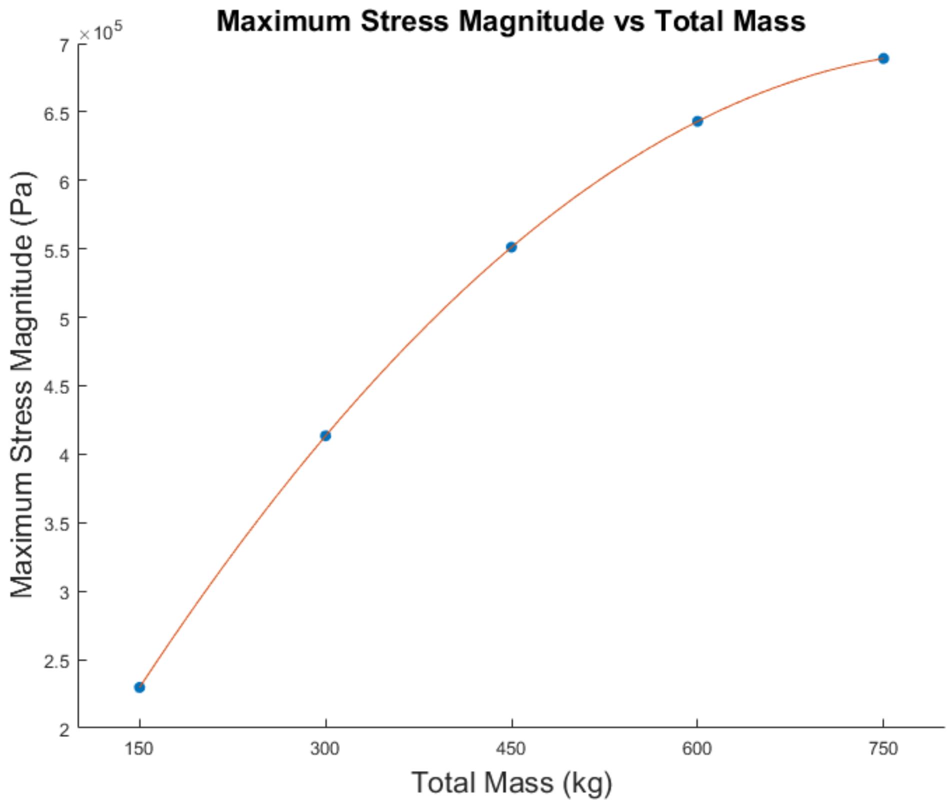
```
            8            0            0
            9            0            0
           10            0            0
           11            0            0
           12            0            0

Nodal Displacements (m) ----- [Node     X     Y]

displacement =

            1            0            0
            2   7.2185e-06  -0.00023741
            3   1.4437e-05  -0.00028689
            4   1.2993e-05  -0.00025994
            5   8.6623e-06  -0.00023653
            6   4.3311e-06  -0.00015477
            7            0            0
            8   0.00010575  -0.00022008
            9  -7.3102e-05  -0.00015477
           10   3.0749e-05  -0.00023492
           11  -3.8999e-06  -0.00025994
           12  -3.8549e-05  -0.00020189

Element Internal Forces (N) ----- [Element     Internal Force]

force =

            1       613.12
            2      -4855.7
            3       613.12
            4       1471.5
            5      -122.63
            6       2207.2
            7      -367.88
            8       1096.8
            9   2.3022e-12
           10      -367.87
           11  -2.8489e-12
           12       1471.5
           13      -367.87
           14            0
           15      -1040.5
           16      -3815.2
           17      -3468.4
           18        548.4
           19        -2943
           20        -2943
           21      -3468.4

Element Axial Stresses (Pa) ----- [Element     Axial Stress]

stress =

            1        81209
```

```
          2  -6.4314e+05
          3        81209
          4    1.949e+05
          5       -16242
          6   2.9235e+05
          7       -48725
          8   1.4527e+05
          9   3.0493e-10
         10       -48725
         11  -3.7733e-10
         12    1.949e+05
         13       -48725
         14            0
         15  -1.3782e+05
         16  -5.0532e+05
         17  -4.5939e+05
         18        72635
         19   -3.898e+05
         20   -3.898e+05
         21  -4.5939e+05

The largest magnitude axial stress is -643139.373 (Pa).
The largest magnitude displacement is -0.00028689 (m).
Yielding due to axial stress does not occur.
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

          1       3433.5       3678.7
          2            0      -1471.5
          3            0      -1471.5
          4            0      -1471.5
          5            0      -1471.5
          6            0      -1471.5
          7      -3433.5       3678.8
          8            0            0
          9            0            0
         10            0            0
         11            0            0
         12            0            0

Nodal Displacements (m) ----- [Node     X     Y]

displacement =

          1            0            0
          2   2.8874e-06  -0.00024944
          3   5.7748e-06  -0.00031146
          4  -7.5757e-20  -0.00030319
          5  -5.7748e-06  -0.00031146
          6  -2.8874e-06  -0.00024944
          7            0            0
          8   0.00010962  -0.00023212
          9  -0.00010962  -0.00023212
```

```
        10    3.898e-05  -0.00025949
        11  -2.2522e-20  -0.00030319
        12   -3.898e-05  -0.00025949

Element Internal Forces (N) ----- [Element     Internal Force]

force =

         1       245.25
         2      -5202.5
         3       245.25
         4       1471.5
         5       -490.5
         6       2207.3
         7       -490.5
         8       822.59
         9   2.3022e-12
        10       245.25
        11      -1040.5
        12       2207.3
        13       245.25
        14       1471.5
        15      -1040.5
        16        -4162
        17      -5202.5
        18       822.59
        19      -3310.9
        20      -3310.9
        21        -4162

Element Axial Stresses (Pa) ----- [Element     Axial Stress]

stress =

         1        32483
         2  -6.8908e+05
         3        32483
         4    1.949e+05
         5       -64967
         6   2.9235e+05
         7       -64967
         8   1.0895e+05
         9   3.0493e-10
        10        32483
        11  -1.3782e+05
        12   2.9235e+05
        13        32483
        14    1.949e+05
        15  -1.3782e+05
        16  -5.5126e+05
        17  -6.8908e+05
        18   1.0895e+05
        19  -4.3853e+05
        20  -4.3853e+05
```

```
        21   -5.5126e+05

The largest magnitude axial stress is -689077.8997 (Pa).
The largest magnitude displacement is -0.00031146 (m).
Yielding due to axial stress does not occur.
The coefficient of determination r^2 is 1 for this fitting function.
```





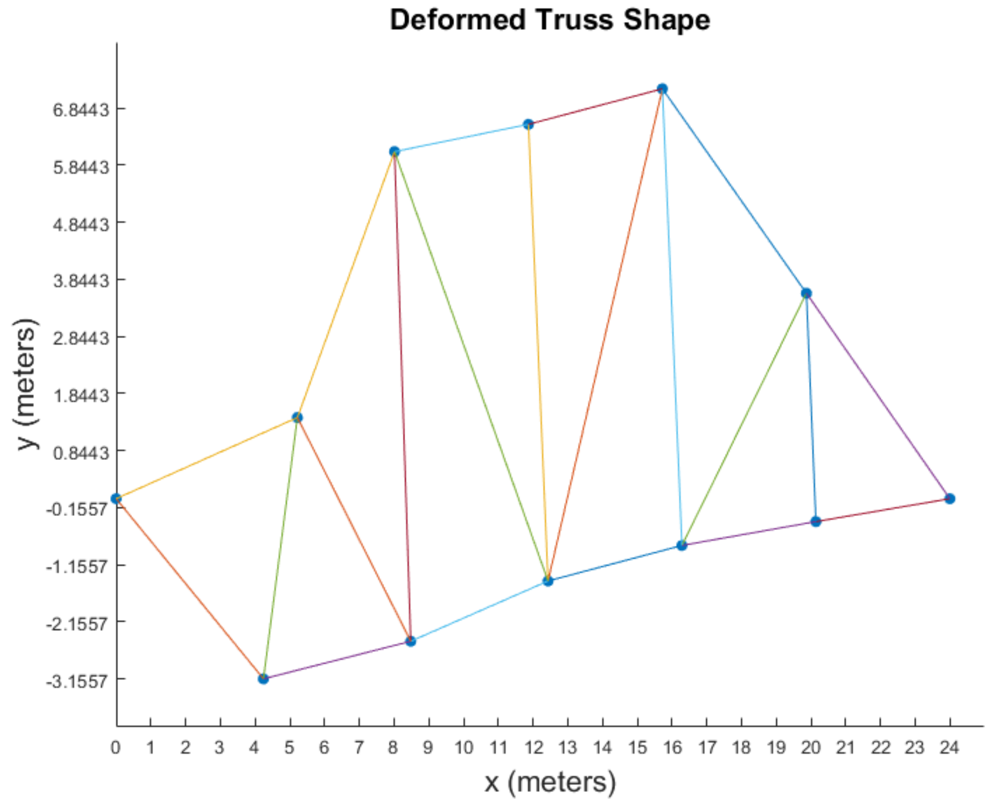
Published with MATLAB® R2016a

```
% This is a test case that I used to troubleshoot this code. It also
% shows what a large deformation would look like (hard to tell if the
% deformed shape plot is working with the small deformations that
 occur
% in the 5 provided cases).

clear all; close all; clc; % Clear everything, close everything.
% TRUSS PARAMETERS ENTERED HERE
nodes = [0 0;4 0;8 0;12 0;16 0;20 0;24 0;4 4;20 4;8 8;12 8;16 8]; %
 Enter the coordinates of our joints (meters) [x y].
elements = [1 2;1 8;2 3;2 8;3 4;3 10;4 5;4 12;4 11;5 6;5 9;5 12;6 7;6
 9;8 3;8 10;9 7;10 4;10 11;11 12;12 9]; % Enter the joints that our
 members connect [startjoint endjoint].
E = 45000000000; % Enter the Modulus of Elasticity here (Pa). For
 magnesium alloy, this is 45 GPa.
A = 0.00755; % Enter our area here (m^2).
sigmay = 130000000; % Enter our yield strength here (for magnesium
 alloy, this is 130 MPa for both compressive and tensile stresses).
unconstrained = [3 4 5 6 7 8 9 10 11 12 15 16 17 18 19 20 21 22 23
 24]; % Enter boundary conditions: the unconstrained joint directions
 (3 4 means joint 2 is free in x and y).
constrained = [1 2 13 14]; % Enter boundary conditions: the
 constrained joint directions (1 2 means joint 1 cannot move in x and
 y).
% CTest
load1 = [0;0;0;5000000;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0;0]; %
 That's a nice truss you've got there, shame if someone put 5 million
 kg on joint 2.
b1=-9.81*load1;

figure1=figure('Position', [100, 100, 800, 600]); % Plot options
 including vertical/horizontal size.
[maxstress(1,1)] =
 TrussDirectStiffness(nodes,elements,b1,E,A,unconstrained,constrained,1,sigmay,1);
```

```
Nodal Reactions (N) ----- [Node     X     Y]

reaction =

            1   2.0437e+07   4.0875e+07
            2            0   -4.905e+07
            3            0            0
            4            0            0
            5            0            0
            6            0            0
            7  -2.0437e+07    8.175e+06
            8            0            0
            9            0            0
           10            0            0
           11            0            0
           12            0            0
```

```
Nodal Displacements (m) ----- [Node     X      Y]

displacement =

           1            0            0
           2      0.24062      -3.1557
           3      0.48124      -2.4975
           4      0.43311      -1.4416
           5      0.28874     -0.81882
           6      0.14437     -0.40116
           7            0            0
           8       1.2171      -2.5783
           9     -0.12893     -0.40116
          10     0.014374        -1.92
          11        -0.13      -1.4416
          12     -0.27437     -0.81882

Element Internal Forces (N) ----- [Element     Internal Force]

force =

           1   2.0438e+07
           2  -5.7806e+07
           3   2.0438e+07
           4    4.905e+07
           5  -4.0875e+06
           6   2.4525e+07
           7  -1.2263e+07
           8   9.1399e+06
           9     9.43e-09
          10  -1.2262e+07
          11  -1.3336e-08
          12            0
          13  -1.2262e+07
          14            0
          15  -3.4684e+07
          16  -2.3122e+07
          17  -1.1561e+07
          18  -9.1399e+06
          19  -1.2263e+07
          20  -1.2262e+07
          21  -1.1561e+07

Element Axial Stresses (Pa) ----- [Element     Axial Stress]

stress =

           1    2.707e+09
           2  -7.6564e+09
           3    2.707e+09
           4   6.4967e+09
           5  -5.4139e+08
           6   3.2483e+09
           7  -1.6242e+09
```

```
   8   1.2106e+09
   9    1.249e-06
  10  -1.6242e+09
  11  -1.7664e-06
  12            0
  13  -1.6242e+09
  14            0
  15  -4.5939e+09
  16  -3.0626e+09
  17  -1.5313e+09
  18  -1.2106e+09
  19  -1.6242e+09
  20  -1.6242e+09
  21  -1.5313e+09

The largest magnitude axial stress is -7656421107.5497 (Pa).
The largest magnitude displacement is -3.1557 (m).
Yielding due to axial stress occurs in at least one member.
```



Published with MATLAB® R2016a

```
function [maxstress maxdisplacement reaction
 displacement force stress b u f st K] =
 TrussDirectStiffness(nodes,elements,b,E,A,unconstrained,constrained,yieldcheck,si
 Define our inputs, outputs.

% Function outputs max stress, max displacement, reaction matrix,
% displacement matrix, internal force matrix, axial stress matrix, as
% well as raw form force vector, nodal displacement vector, internal
% force vector, and axial stress vector. The global stiffness matrix K
 is
% also outputted for troubleshooting purposes. The function also
 performs a
% preliminary yield check for all members using inputted yield
 strength
% value (if one is inputted), and outputs a deformed shape plot if
 prompted
% to do so.

% Function requires nodes matrix, elements matrix, b vector, E, A,
% unconstrained and constrained matrix (boundary conditions for the
 direct
% stiffness method.

if nargin<7,error('The following input arguments are required:
 nodes,elements,b,E,A,unconstrained,constrained'),end % Check for
 sufficient input arguments.
if nargin==8 && yieldcheck ~= 0,error('To enable yield check, enter
 the yield strength of the material sigmay. Otherwise, yieldcheck =
 0 or simply leaving the yieldcheck input blank will disable yield
 check.'),end % Enabling yield check without inputting sigma y causes
 an error.
if nargin<8,yieldcheck = 0; end % If there is no yieldcheck input
 provided, that loop is disabled.
if nargin<10,shape = 0; end % If the user does not request a deformed
 shape plot, disable loop.
if yieldcheck ~= 0 && yieldcheck ~= 1, error('Input for yieldcheck
 must be 0 (off) or 1 (on)'),end
if sigmay <= 0, error('Yield strength must be positive.'),end
if shape ~= 0 && shape ~= 1, error('Input for shape must be 0 (off) or
 1 (on)'),end
if size(nodes,2) ~= 2, error('Node coordinate matrix must be of size n
 by 2.'),end % Next lines check for dimensional mismatches.
if size(elements,2) ~= 2, error('Element matrix must be of size n by
 2.'),end
if size(b,2)~= 1, error('b must be a column vector.'),end
if size(b,1)~= 2*size(nodes,1), error('b must be a column vector of
 size 2n'),end
if size(unconstrained,1)~= 1, error('unconstrained must be a row
 vector.'),end
if size(constrained,1)~= 1, error('constrained must be a row
 vector.'),end
```

```
if size(unconstrained,2)+size(constrained,2) ~= 2*size(nodes,1),
 error('Not enough boundary conditions specified.'),end
if size(E,1)~= 1, error('E must be a scalar value.'),end
if size(E,2)~= 1, error('E must be a scalar value.'),end
if size(A,1)~= 1, error('A must be a scalar value.'),end
if size(A,2)~= 1, error('A must be a scalar value.'),end

Nelements = size(elements,1); % The number of "elements" for the
 direct stiffness method (an element is a truss member linking two
 joints).
Nnodes = size(nodes,1); % The number of "nodes" for the direct
 stiffness method (a node is a truss joint).

K = zeros(2*Nnodes); % Initializing K matrix, the assembled global
 stiffness matrix (must be square matrix of size = 2*number of nodes,
 because each node has vertical and horizontal degree of freedom).
 Values are unknown, therefore initialize as zeros.
u = zeros(2*Nnodes,1); % Initializing u displacement matrix (column
 vector of size = 2*number of nodes, 1). Values are unknown, therefore
 initialize as zeros.

for i=1:Nelements % Iterate from 1 to the number of elements.
    elementnodes = elements(i,1:2); % This selects the start and end
 nodes (start and end joints) of each member.
    nodecoordinates = nodes(elementnodes,:); % This selects the
 corresponding node coordinates from the joints matrix.

    x1 = nodecoordinates(1,1); % Retrieves the x1, x2, y1, y2 values
 from the supplied nodecoordinates matrix.
    x2 = nodecoordinates(2,1);
    y1 = nodecoordinates(1,2);
    y2 = nodecoordinates(2,2);

    L = sqrt((x2-x1)^2+(y2-y1)^2);  % Length is calculated in terms of
 node coordinates.
    cos = (x2-x1)/L; % Calculation of cos and sin for usage in Klocal
 matrix.
    sin = (y2-y1)/L;

    Klocal = (E*A/L)*[cos^2 cos*sin -cos^2 -cos*sin;cos*sin sin^2 -
cos*sin -sin^2;-cos^2 -cos*sin cos^2 cos*sin;-cos*sin -sin^2 cos*sin
 sin^2]; % The Klocal matrix is the stiffness matrix for one bar type
 element in global coordinates.

    idBeg = 2*(elementnodes(1)-1)+1:2*(elementnodes(1)-1)+2; % The
 displacement corresponding to the beginning of the element (what
 position it occupies in the u column vector).
    idEnd = 2*(elementnodes(2)-1)+1:2*(elementnodes(2)-1)+2; % The
 displacement corresponding to the end of the element.
    id = [idBeg idEnd]; % This identifies which values of the
 displacement vector u this local stiffness matrix corresponds to, and
 will be used in assembling the matrix below.
```

```
    K(id,id) = K(id,id) + Klocal; % The elementdisplacement vector
 is used to assemble the local stiffness matrices into one global
 stiffness matrix.
end

u(unconstrained) = K(unconstrained,unconstrained)\(b(unconstrained)-
K(unconstrained,constrained)*u(constrained)); % The unconstrained
 node elements of the u vector are found (these are our unknown
 deformations).
b(constrained) = K(constrained,:)*u; % The elements of the force
 vector corresponding to constrained nodes are found (these are our
 unknown reaction forces).

for i=1:Nelements
    elementnodes = elements(i,1:2); % This selects the start and end
 nodes (start and end joints) of each member.
    nodecoordinates = nodes(elementnodes,:); % This selects the
 corresponding node coordinates from the joints matrix.

    x1 = nodecoordinates(1,1); % Retrieves the x1, x2, y1, y2 values
 from the supplied nodecoordinates matrix
    x2 = nodecoordinates(2,1);
    y1 = nodecoordinates(1,2);
    y2 = nodecoordinates(2,2);

    L = sqrt((x2-x1)^2+(y2-y1)^2);  % Length is calculated in terms of
 node coordinates.
    cos = (x2-x1)/L; % Calculation of cos and sin for usage in Klocal
 matrix.
    sin = (y2-y1)/L;

    idBeg = 2*(elementnodes(1)-1)+1:2*(elementnodes(1)-1)+2; % The
 displacement corresponding to the beginning of the element (what
 position it occupies in the u column vector).
    idEnd = 2*(elementnodes(2)-1)+1:2*(elementnodes(2)-1)+2; % The
 displacement corresponding to the end of the element.

    f(i,1)=(E*A/L)*(cos*(u(idEnd(1),1)-
u(idBeg(1),1))+sin*(u(idEnd(2),1)-u(idBeg(2),1))); % The internal
 force in each member is calculated based on the displacements of the
 beginning and end nodes, and arranged in a column vector.
    st(i,1) = (1/A)*f(i,1); % The stress is easily calculated,
 simply multiply the force vector by the scalar 1/A (equivalent to
 elementwise division by area, sigma = F/A).
end

[~,maxstresselement] = max(abs(st));
maxstress = st(sub2ind(size(st),maxstresselement,1:size(st,2))); % The
 greatest magnitude value in the matrix is returned, with +/- symbol

[~,maxdisplacementnode] = max(abs(u));
maxdisplacement =
 u(sub2ind(size(u),maxdisplacementnode,1:size(u,2))); % This process
 is repeated for maximum displacement.
```

```
reaction = zeros(Nnodes,3); % The u and b column vectors are broken
 down into matrix of size = (Nnodes,2) for readability, so that [x1
 y1;x2 y2;...] corresponds to the x and y reactions/displacements of
 node 1, node 2, etc.
reaction(:,1) = 1:Nnodes;
reaction(:,2) = b(1:2:end);
reaction(:,3) = b(2:2:end);
displacement = zeros(Nnodes,3);
displacement(:,1) = 1:Nnodes;
displacement(:,2) = u(1:2:end);
displacement(:,3) = u(2:2:end);
force = zeros(Nelements,2); % The internal force and stress vectors
 are also placed into a more readable, indexed format.
force(:,1) = 1:Nelements;
force(:,2) = f(:,1);
stress = zeros(Nelements,2);
stress(:,1) = 1:Nelements;
stress(:,2) = st(:,1);

format short g % The reformatted reaction, displacement, force, and
 stress matrices are presented to the user.
disp(['Nodal Reactions (N) ----- [Node     X     Y]']); reaction
disp(['Nodal Displacements (m) ----- [Node     X     Y]']);
 displacement
disp(['Element Internal Forces (N) ----- [Element     Internal
 Force]']); force
disp(['Element Axial Stresses (Pa) ----- [Element     Axial
 Stress]']); stress
disp(['The largest magnitude axial stress is '
 num2str(maxstress) ' (Pa).']); % Maximum stress (greatest magnitude)
 is displayed. Note that this may be compressive or tensile, and
 includes +/- symbol.
disp(['The largest magnitude displacement is '
 num2str(maxdisplacement) ' (m).']); % Maximum displacement (greatest
 magnitude) is displayed.

if yieldcheck == 1 % If the user has decided to enable yieldchecking,
 this loop proceeds.
if abs(maxstress) > sigmay % This if loop checks maximum internal
 stress against the yield strength inputted into our function.
    disp(['Yielding due to axial stress occurs in at least one
 member.']); % If yield strength is exceeded, the user is warned.
else
    disp(['Yielding due to axial stress does not occur.']); % If yield
 strength is not exceeded, no warning.
end
elseif yieldcheck == 0 % If the user has disabled yieldchecking or
 left yieldcheck input blank, the loop is disabled.
    return
end

if shape == 1 % If the "shape" input is 1, this triggers the loop that
 draws the deformed truss shape.
```

```
    deformedshape(:,1:2) = nodes(:,1:2)+ displacement(:,2:3); % The
 new positions of the nodes are calculated by adding the initial node
 positions and displacements.

    scatter(deformedshape(:,1),deformedshape(:,2),'filled') %
 A scatter plot is made of the nodes in their post deformation
 positions.
    xlabel('x (meters)','fontsize',16);
    ylabel('y (meters)','fontsize',16);
    title('Deformed Truss Shape','fontsize',16); % Plot settings and
 labels.
    hold on

for i=1:Nelements % Iterate from 1 to number of elements.
    elementnodes = elements(i,1:2); % Obtain the start and end
 nodes of each element (similar to the step that creates the global
 stiffness matrix).
    nodecoordinates = deformedshape(elementnodes,:); % This selects
 the corresponding node coordinates from the joints matrix.

    x1 = nodecoordinates(1,1); % Retrieves the x1, x2, y1, y2 values
 from the supplied nodecoordinates matrix
    x2 = nodecoordinates(2,1);
    y1 = nodecoordinates(1,2);
    y2 = nodecoordinates(2,2);

    m = (y2-y1)/(x2-x1); % This calculates the slope of the line
 formed by these nodes.
    y = @(x) m*x-m*x1+y1; % The equation of the line passing through
 both nodes is defined.
    if x2 > x1
        fplot(y,[x1 x2]); % If x2>x1, the function is plotted on this
 range. Alternatively, the x2 and x1 positions are switched.
    elseif x2 < x1
        fplot(y,[x2 x1]);
    elseif x2 == x1
        if y2 > y1
        line([x1 x2],[y1 y2]); % If x1 and x2 are equal, a vertical
 line must be drawn. Note the y1 and y2 positions switch much like the
 x1 and x2 positions above.
        else
        line([x1 x2],[y2 y1]);
        end
    end

 set(gca,'XTick',min(deformedshape(:,1)):max(deformedshape(:,1))); % X
 and Y intervals are set.
    set(gca,'YTick',min(deformedshape(:,2)):max(deformedshape(:,2)));
    hold on
end
elseif shape == 0 % If the shape input is 0, the loop above is not
 triggered and a graph is not produced.
    return
end
```

```
end
```

```
Error using TrussDirectStiffness (line 16)
The following input arguments are required:
 nodes,elements,b,E,A,unconstrained,constrained
```

Published with MATLAB® R2016a