FEA Simulation

266720[1]

Date: Monday 29^{th} April 2024

Abstract

This report is an answer to two questions laid out by H7137 Numerical Modelling and Engineering Simulations by Mark Puttock Brown and the University of Sussex. The two questions are: static equilibrium of a planar truss structure and FEA analysis of a Belleville spring under loading conditions.

Analytical methods, Matlab simulations, and Finite Element Analysis (FEA) in ANSYS were used. The truss problem shows close agreement between analytical and simulation results, but slight differences are likely due to numerical approximations and boundary conditions. However, simulating the Belleville spring, we find that it closely matches the force data provided by the manufacturer. However, there are notable differences in the spring's stiffness and the highest stresses it can handle.

Nomenclature

δ	Deflection (mm)
ϵ	Strain
σ	Stress (MPa)
A	CrossSectional Area (mm^2)
d_i	Inner Diameter (mm)
d_o	Outer Diameter (mm)
E	Young's Modulus (GPa)
F	Force (N)
K	Spring Constant
L	Length (m)
P	Load (N)
t	Thickness (mm)
X_i	Nodal Displacement (mm)

Contents

1. DECLARATION

This report, titled: "FEA Simulation", is entirely my own work, unless otherwise acknowledged, and performed under guidance of and for the module of Dr. Mark Puttock-Brown, and the University of Sussex.

2. PROBLEM 1 - EQUILIBRIUM TRUSS

2.1. Introduction

In this experiment, we analyze the static equilibrium of a planar truss structure subjected to a loading condition.

A truss is a structural framework composed of straight members connected at joints, commonly used to support loads over a span and static equilibrium is the state where all forces and torques acting on an object or system balance out, resulting in no net force or torque and thus no linear or rotational motion. The truss structure being planar means that it sticks to the 2D plane.

2.2. Setup

The trusses consist of eight circular tubes, numbered from 1 to 8, with outer diameter d_o and inner diameter d_i. It is loaded at node 4 with a force denoted as P and fixed at the wall in two places, shown in Fig. 1.

2.2.1. Matlab

The Matlab code was split into a main body, and a Gauss function provided. The main body is split into three different sections: input variation, defining the matrices, and calling the Gauss Function.

For the variation input, a simple if is used to see if the user wants to change the inputs of the question. If the user does not want to change the inputs then the original numbers provided in the question are used.

In the Matrix A, the force equations that were calculated by hand are inserted into the matrix, using simplification from the two equation in the previous lines. The by Matlab and the hand calculations are shown at the end to make the report easier to read. The following listing is the main MATLAB code:

2.2.2. FEA Parameters and Boundary conditions

For this experiment the structure doesn't move in the x direction, it is a purely 2D structure. We have fixed the two 'wall' nodes in place. The beams were defined as trusses and then defined as beams. The geometry inputted is much the same as the question and is shown in Fig. 2. Structural steel properties were used in this, with a Young's modulus of 200GPa. The force included was the same as the question's.

A reaction force probe was placed at the support nodes to verify your simulation.

2.3. Analysis

In table 1, we show the numbers obtained from our Matlab results, the reference numbers and then the numbers obtained from our FEA Analysis. We have two separate columns for the FEA using trusses and FEA using beams.

Table 1: Table of results (1)

Matlab mm	Reference mm	FEA Truss mm	FEA Beam mm
-9.1617	-9.1617	-9.1622	-9.1585
1.5719	1.5719	1.572	1.5737
-7.5898	-7.5898	-7.5902	-7.5876
-3.1438	3.1438	-3.144	-3.1445
-19.8953	-19.8953	-19.896	-19.897
-4.7157	-4.7157	-4.7159	-4.717
-21.4672	-21.4672	-21.468	-21.47
-17.0212	-17.0212	-17.022	-17.027

2.4. Discussion

2.4.1. Matlab and hand calculations

The Matlab and Hand calculations matched exactly with the reference answers. There is nothing to comment, except that the answers can now be assumed to be correct.

2.4.2. FEA Trusses and Beams

For the truss simulations all magnitudes of the results are $\tilde{0}.005\%$ - $\tilde{0}.006\%$ higher than the analytical answers. The beam simulations have a maximum magnitude of 0.0276% greater than the analytical answers, however there is a single result which has a magnitude of 0.1145% lower, which will be discussed later.

To clarify, trusses mean that all the bodies can only undergo tension and compression, whereas the beam simulations can undergo different forces as well. So trusses would make greater magnitudes of force, because all of the force keeping the system in static equilibrium, goes into tension and compression, whereas for beams it might go into torsion, bending or others.

One of the reasons the simulation may be different could be due to the numerical approximations.

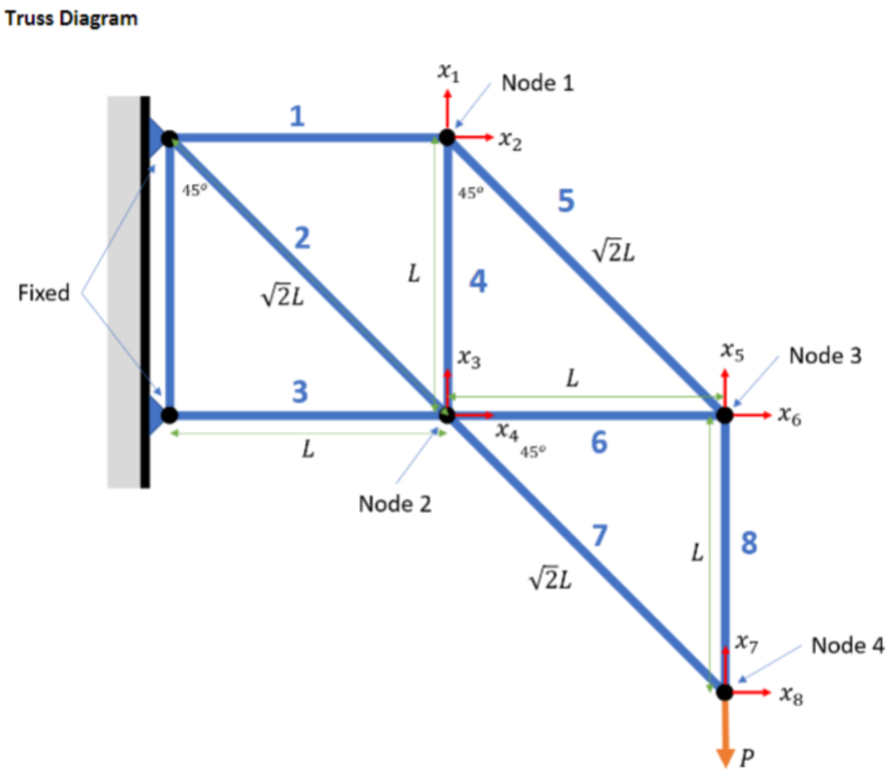


Fig. 1: Geometry of Problem 1

Simulation methods use discretising the problem and solving it numerically, which cause errors because of finite element discretisation, numerical integration, and round-off errors.

Differences in how boundary conditions are applied in the simulation compared to the analytical model could also cause discrepancies in results. For example, constraints or loads might not be perfectly replicated in the simulation.

The level of mesh refinement in the simulation could also affect the accuracy of the results. Higher mesh density generally leads to more accurate results.

3. PROBLEM 2 - BELLEVILLE SPRING

3.1. Introduction

This question analyses Belleville spring washers. Analytical equations and FEA in ANSYS was used to determine the spring constant and study deformation and stress under different loading conditions. By comparing the findings with manufacturer data, the accuracy of our results can be evaluated.

3.2. Setup

3.2.1. 2D Simulations

Only half was modelled because the system is symmetrical.

Large displacement is on, because the spring needs to be able to move in order to register a displacement.

Boundary conditions were set up. The 2D setup has no z axis, as should be expected. The bottom corner of the spring was fixed on the y, to simulate a 'floor' and the inside edge was set to only have a y displacement of -3.72mm simulating the lowest the spring can go. All the boundary conditions are shown in Fig. 3

3.2.2. 3D Simulations

The 2D model was revolved around the y axis to create a circular 3D model.

Large displacement is still on. Boundary conditions were set up again. The bottom edge of the spring was fixed on the y, to simulate the 'floor' again and the inside face this time was set to only have a y displacement of -3.72mm simulating the lowest the spring can go.

3.2.3. Matlab

The Matlab is shown at the end.

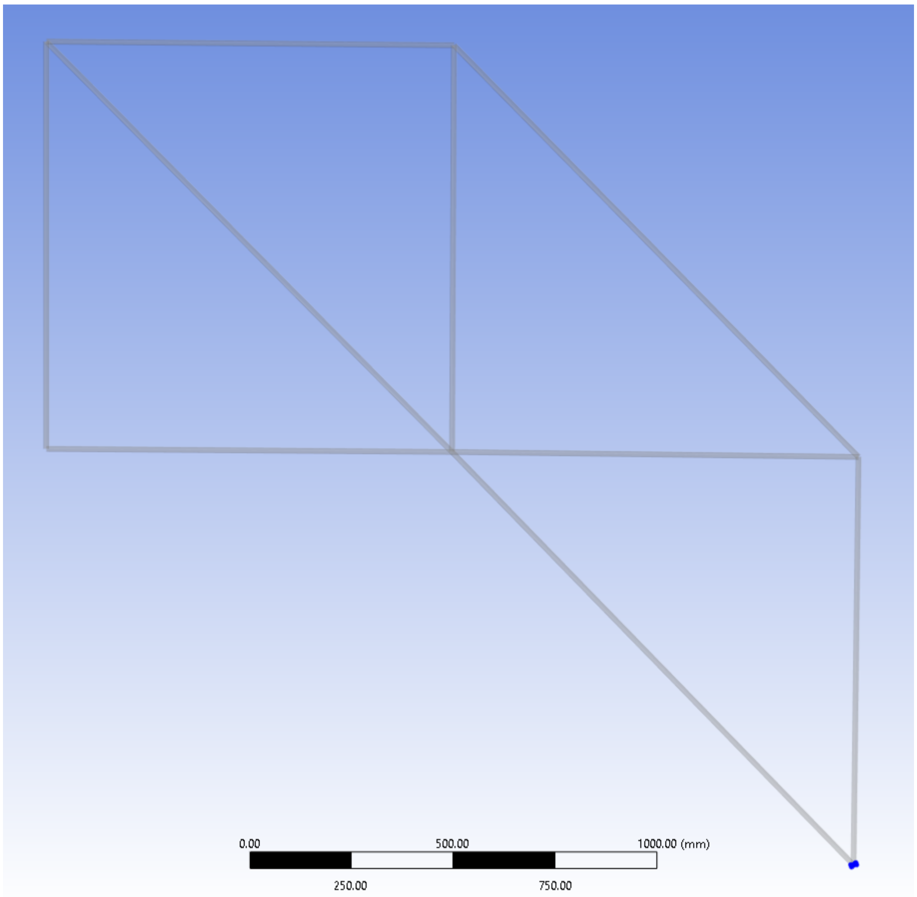


Fig. 2: Beam structure in Ansys

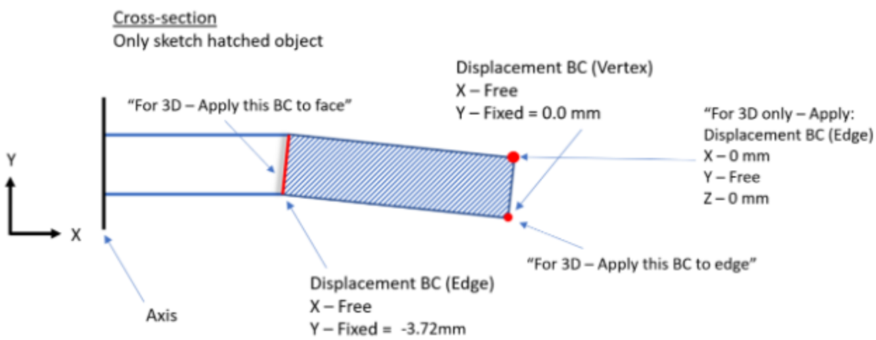


Fig. 3: Belleville spring boundary conditions(3)

3.3. Analysis

Table 2 shows the numbers obtained from our Matlab results, the reference numbers and then the numbers obtained from our FEA Analysis. We have two separate columns for the FEA using trusses and FEA using beams.

To scale deformation shown in Fig. 4 and stress is shown in Fig. 5.

3.4. Discussion

3.4.1. Force

The analytical and FEA simulations have forces

Analysis	Deflection	Force	k	Peak Von-Mises Stress
-	mm	N	N/mm	MPa
Manufacturer Data	3.72	68204	-	1000
Analytical	3.72	68791	1.296×10^7	-
FEA 2D	3.72	76738	1.163×10^7	3946
FEA 3D	3.72	79479	2.040×10^7	3287.5

Table 2: Force, Stress and Deflection Simulation vs Manufacturer Data(1)

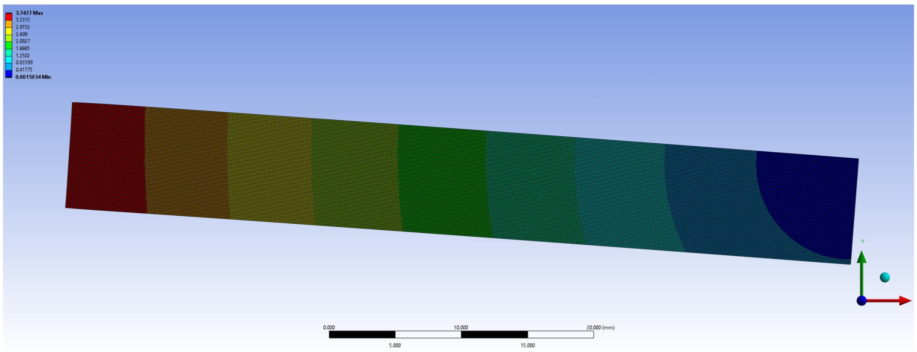


Fig. 4: Matlab showing variation of property inputs

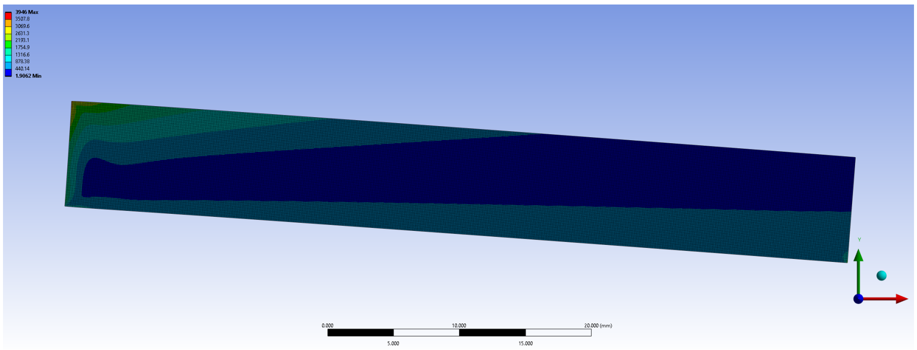


Fig. 5: Matlab showing variation of property inputs

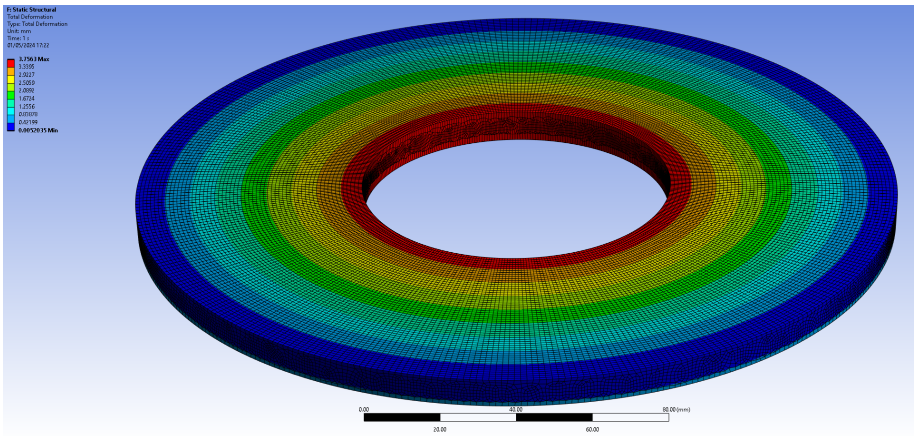


Fig. 6: Matlab showing variation of property inputs

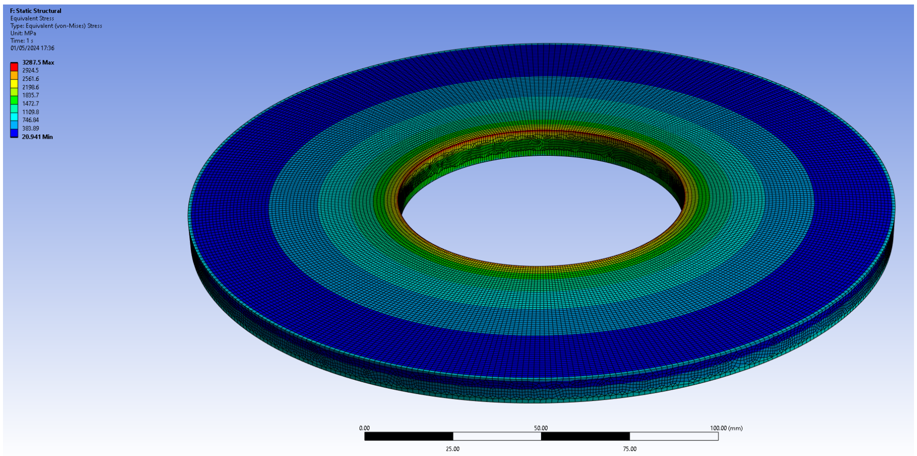


Fig. 7: Matlab showing variation of property inputs

close to the manufacturer data, with slight variations. The analytical result is slightly higher, (68791 N) than the manufacturer data (68204 N), an increase of 0.86%, while the FEA 2D and 3D simulations provide higher forces (76738 N and 79479 N, respectively) which is an increase of 12.51% and 16.53% respectively.

These variations could be due to differences in modeling assumptions and numerical methods between analytical and FEA approaches. However, the percentage differences are within a reasonable range, suggesting that the simulations provide a reasonably accurate estimation of the force.

However,

3.4.2. Spring constant

The spring constants calculated from the analytical and FEA simulations show significant variations compared to each other. The analytical and FEA 2D simulations have similar spring constants, while the FEA 3D simulation results in a significantly higher value.

The discrepancy could be attributed to differences in modeling complexity and boundary conditions between the simulations. The percentage variations are substantial, indicating potential limitations in the accuracy of the simulations in capturing the spring's mechanical behavior.

3.4.3. Peak Stress

The peak Von-Mises stresses from the FEA simulations are a lot higher than the manufacturer data (3946 MPa for FEA 2D and 3287.5 MPa for FEA 3D compared to 1000 MPa). This discrepancy suggests that the FEA simulations may be overestimating the stress levels in the spring. Possible reasons for this could be simplifications or inaccuracies in material properties, boundary conditions, or mesh resolution in the simulations.

4. CONCLUSIONS

This report studied an equilibrium truss and Belleville spring which showed some disparities between analytical and simulation results. While the truss problem showed close agreement between Matlab and FEA simulations, slight variations existed, likely due to numerical approximations and boundary conditions.

Regarding the Belleville spring, while simulations closely matched manufacturer force data, significant differences arose in spring constants and peak stresses. These variations could be attributed to modeling complexities and material properties.

REFERENCES

[1] Puttock-Brown, M., *Numerical Modelling and Engineering Simulations* Coursework - 2, Reference Answers, University of Sussex, 2023

[2] Puttock-Brown, M., *Numerical Modelling and Engineering Simulations* Gauss Function, University of Sussex, 2023

[3] Puttock-Brown, M., *Numerical Modelling and Engineering Simulations* BellevilleMark, University of Sussex, 2023

o

.

First Problem: Main Body

```
clc
clear
close all

%% Variation of Input

VarInp = input('Do you want to change the original questions input? If yes, input
1. If nah, input 0\n');
%allowing user to change the input of the original question

if VarInp == 1
    D_o = input('What is the outer diameter of your beam in mm?\n'); %change outer
diameter
    D_i = input('what is the inner diameter of your beam in mm?\n'); %change inner
diameter
    L = input('what is length of the beam in M?\n'); %change length
    E = input('what is the Young Modulus of the material used in GPa?\n'); %change
youngs modulus
    P = input('what is the load in N at Node 4?\n'); %change load
else
    D_o = 15; %Defining the questions original variables
    D_i = 12;
    L = 1;
    E = 200;
    P = 20000;
end

%% Defining Matrices
x = zeros(8,1);%Defining matrix x for the displacements

a = (pi*((D_o/1000)/2)^2 - pi*((D_i/1000)/2)^2);
B = 1/(2*sqrt(2));%Creating Hand Equations in the matlab code

Ba = a*B;
aaB = Ba + a;%Simpifying Matrix variables

A = [Ba -aaB 0 0 -Ba Ba 0 0;
    -aaB Ba a 0 Ba -Ba 0 0;
    a 0 -(2*Ba+a) 2*Ba 0 0 Ba -Ba;
    0 0 2*Ba -2*aaB 0 a -Ba Ba;
    Ba -Ba 0 0 -aaB Ba a 0;
    Ba -Ba 0 -a -Ba aaB 0 0;
    0 0 Ba -Ba a 0 -aaB Ba;
    0 0 Ba -Ba 0 0 -Ba Ba];
%Defining Matrix from Hand Equations

b = [0;
    0;
    0;
    0;
    0;
    0;
    (L/(E*10^9))*P;
    0];
%Defining b

%% Calling Gauss Function
```

```
x = gauss(A,b)*1000;
%Calling the Gauss Function

disp(x);
```

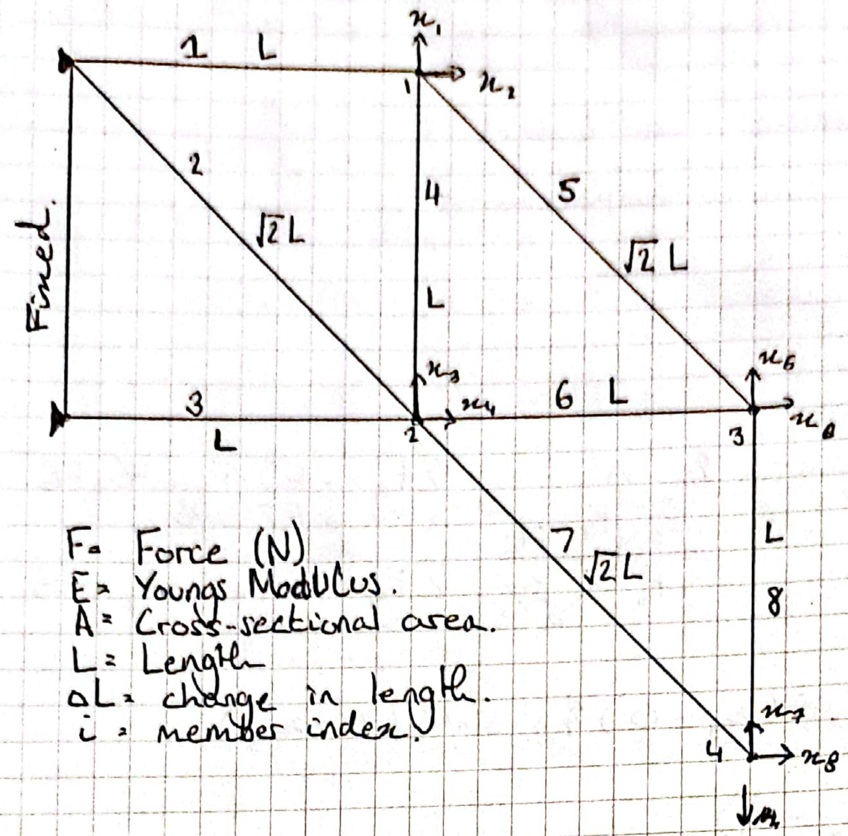
First Problem: Gauss Function

```
function [x,det] = gauss(A,b)

% Solves A*x = b by Gauss elimination and computes det(A).

% USAGE: [x,det] = gauss(A,b)

if size(b,2) > 1
    b = b';
end % b must be column vector

n = length(b);

for k = 1:n-1 % Elimination phase
    for i= k+1:n
        if A(i,k) ~= 0
            lambda = A(i,k)/A(k,k);
            A(i,k+1:n) = A(i,k+1:n) - lambda*A(k,k+1:n);
            b(i)= b(i) - lambda*b(k);
        end
    end
end

if nargout == 2
    det = prod(diag(A));
end
for k = n:-1:1 % Back substitution phase
    b(k) = (b(k) - A(k,k+1:n)*b(k+1:n))/A(k,k);
end

x = b;
```

Second Problem

```
clc
close all
clear

%% Variation of Input

VarInp = input('Do you want to change the original questions input? If yes, input
1. If nah, input 0\n');
%allowing user to change the input of the original question

if VarInp == 1
    D_o = input('what is the outer diameter of the bellville spring in mm?\n');
    D_i = input('what is the inner diameter of the bellville spring in mm?\n');
    T = input('what is the thickness of the bellville spring?\n');
    E = input('what is the Youngs Modulus of the material used in GPa?\n');
    H = input('what is the cone height?\n');
    u = input('what is the poissons ratio of the material?\n');
    d = input('what is the displacement of the bellville spring in mm?\n');
else
    D_o = 200;
    D_i = 82;
    T = 8;
    E = 200;
    H = 6.7;
    u = 0.3;
    d = 3.72;
end

%% Correcting Variables
D_o = D_o/1000; %Correcting to meters
D_i = D_i/1000; %Correcting to meters
T = T/1000; %Correcting to meters
H = H/1000; %Correcting to meters
d = d/1000; %Correcting to meters
E = E*10^9; %Correcting to Pascals
R = D_o/D_i;
S = d/10;
K = (6/(pi*log(R)))*((R-1)^2/R^2); %Equation to figure out k
F = (4*E*d/(K*D_o^2*(1-u^2)))*((H-d)*(H-d/2)*T+T^3); %Figure out force
F1 = (4*E*(d-S)/(K*D_o^2*(1-u^2)))*((H-(d-S))*(H-(d-S)/2)*T+T^3);
F2 = (4*E*(d+S)/(K*D_o^2*(1-u^2)))*((H-(d+S))*(H-(d+S)/2)*T+T^3);
spring_constant = (F2-F1)/(2*S); %Equation to figure out spring constant

%% Displaying

disp('Force = ');
disp(F);

disp('Spring Constant = ');
disp(spring_constant);
```

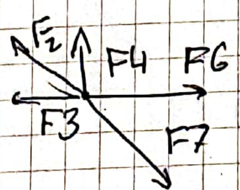
Numerical Modelling - Coursework 2 - Problem 1



F = Force (N)
E = Youngs Modulus.
A = Cross-sectional area.
L = Length
ΔL = change in length.
i = member index.

Solve for Node 2:

Free body diagram.



Relating force to nodal displacements.

$$E_i = \frac{\sigma_i}{\varepsilon_i} = \frac{F_i / A_i}{\Delta L_i / L_i}$$

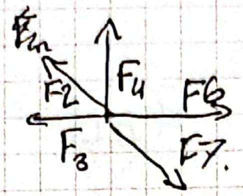
$$F_i = \frac{E_i A_i}{L_i} \quad = \frac{200\,GPa \times (\pi 15^2 mm - \pi 12^2 mm)}{\Delta L_i \,/\, L_i}$$

Defining Change in length ΔL_i

- Nodal displacements are vectors.
- ΔL is the sum of displacements.
- Be consistent with nodes.

Solving for Node 2,

Free body diagram:



$$\Sigma F_x = -F3 - F2\sin 45 + F6 + F7\sin 45$$

$$\Sigma F_y = -F7\sin 45 + F_2 \sin 46 + F4$$

$$F_2 = \frac{EA_2}{L_2}\left(\sin 45 (x_4 - 0) + \sin 45 (\cancel{x_4} - x_3) \right)$$

$$F_3 = \frac{EA_3}{L_3}\left(x_4 - 0 \right)$$

$$F_4 = \frac{EA_4}{L_4}\left(x_1 - x_3 \right)$$

$$F_6 = \frac{EA_6}{L_6}\left(x_6 - x_4 \right)$$

$$F_7 = \frac{EA_7}{L_7}\left(\sin 45 (x_8 - x_4) + \sin 45 (x_7 - x_3) \right)$$

$$F_x : -F3 - F2\sin 45 + F6 + F7\sin 45$$

$$= -\frac{EA_3}{L_3}(x_4 - 0) - \frac{EA_2}{L_2}\left(\sin 45 (x_4 - 0) + \sin 45 (0 - x_3)\right)$$

$$+ \frac{EA_6}{L_6}(x_6 - x_4) + \frac{EA_7}{L_7}\left(\sin 45 (x_8 - x_4) + \sin 45 (\cancel{x} - x_3)\right)$$

$$0 = -\frac{EA_3}{L}(x_4 - 0) - \frac{EA_2}{\sqrt{2}L}\left(\sin 45 (x_4 - x_3)\right) + \frac{EA_6}{L}(x_6 - x_4)$$

$$\frac{EA_7}{\sqrt{2}L}\left(\sin 45 (x_8 - x_4) + \sin 45 (x_7 - x_3)\right)$$

$$0 = -A_3 x_4 - \frac{A_2}{\sqrt{2}}\left(\frac{2}{\sqrt{2}}(x_4 - x_8)\right) + A_6(x_6 - x_4)$$

$$+ \frac{A_7}{\sqrt{2}}\left(\frac{2}{\sqrt{2}}(P + x_8 + x_3 - x_4 - x_7)\right)$$

$$= -A_3 x_4 - \beta A_2(x_4 - x_8) + A_6(x_6 - x_4)$$

$$+ \beta A_7(x_8 + x_3 - x_4 - x_7)$$

F_y: $-F7 \sin 45 + F_2 \sin 45 + F_4$

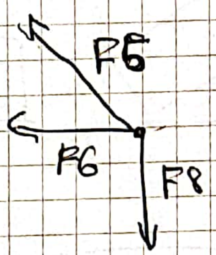
$$0 = \frac{A_7}{\sqrt{2}}\left(\frac{2}{\sqrt{2}}(x_8 - x_4 + P - x_7 - x_3)\right) + \frac{A_2}{\sqrt{2}}\left(\frac{2}{\sqrt{2}}(x_4 - x_3)\right)$$

$$+ A_4(x_1 - x_3)$$

$$= A_7 \beta(P + x_8 - x_3 - x_4 - x_7) + A_2 \beta(x_4 - x_3)$$

$$+ A_4(x_1 - x_3)$$

Solving for node 3:



$\Sigma F_x = -F_6 - F_5 \sin(45)$

$\Sigma F_y = F_5 \sin 45 - F8$

$$F_5 = \frac{EA_5}{L_5}\left(\sin 45 (x_1 - x_5) + \sin 45 (x_6 - x_2)\right)$$

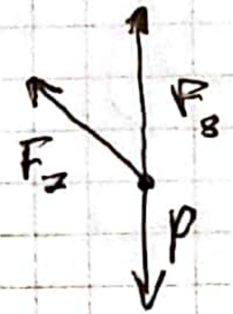
$$F_6 = \frac{EA_6}{L_6}(x_6 - x_4)$$

$$F_8 = \frac{EA_8}{L_8}(x_5 + P - x_7)$$

F_x: $-A_6(x_6 - x_4) - A_6 \beta(x_1 - x_5 + x_6 - x_2)$

F_y: $A_6 \beta(x_1 - x_5 + x_6 - x_2) - A_8(x_5 + P - x_7)$

Node 4: Free body ~~workshop.~~ diagram.



$$\sum F_x = F_7 \sin 45$$

$$\sum F_x = F_8 + P + F_7 \sin 45$$

$$F8 = P$$

$$F_7 = \frac{EA_7}{L_7}\left(\sin 45(x_8 - x_4) + \sin 45(x_3 - x_7)\right)$$

$$F_x = A_7 \beta(x_8 - x_4 + x_3 - x_7)$$

$$F_y = A_7 \beta(x_8 - x_4 + x_3 - x_7)$$