Due Tuesday, September 21st, 12:00 midnight

The first problem discusses a plane truss with inclined supports. You will need to modify the MatLab software from homework 1. The next 3 problems consider the analysis of beams as discussed in class. In problem 4, you will need to extend the provided beam MatLab programs by including a tapered beam finite element. A list of MatLab programs for beam analysis is provided together with an example problem. You will need to modify these programs to address problems 2 and 3.

Problem 1 - Analysis of trusses with inclined supports (MatLab)

The truss problems examined in earlier homework account for boundary displacement conditions posed directly in terms of the \( u, v, w \) displacements along the \( x, y, z \) axes. For a change, consider the five-bar truss shown above with an inclined roller support at node 1. All elements are made of the same material \( E = 70 \text{ GPa} \) and \( A = 10^{-3} \text{ m}^2 \). The load is \( P = 20 \text{ kN} \). Modify the finite element formulation examined in class (and programmed in the given MatLab files) to account for inclined supports. The modifications you can do can either be specific to this problem or can be general enough to accommodate many and different nodes with inclined support.

Hint: For those interested to program this for the general case, here is one approach using the penalty method. In some finite element modeling situations, it becomes necessary to introduce constraints between several different degrees of freedom. Such constraints are known as multipoint constraints and in general are expressed as follows:

\[
c_{11}d_1 + c_{12}d_2 + \ldots + c_{1n}d_n = q_1 \\
c_{21}d_1 + c_{22}d_2 + \ldots + c_{2n}d_n = q_2 \\
\ldots
\]

where \( c_{ij}, i, j = 1, 2, \ldots \), and \( q_i, i = 1, 2, \ldots \), are specified constants and \( d_i, i = 1, 2, \ldots \), are the nodal degrees of freedom. In matrix form the constraints equations can be expressed as follows:

\[
Cd = q
\]

where with \( m \) constraints \( C \) is a \( m \times n \) matrix and \( q \) is a \( m \times 1 \) matrix.

We can then modify the FEM problem statement as follows:

Find \( d \) such that
Minimize \( \phi = \frac{1}{2} d^T K d - d^T R + \frac{1}{2} \mu (C d - q)^T (C d - q) \)

Subject to \( C d - q = 0 \)

With \( \mu \) (the penalty parameter) being large, the minimization process forces the constraints to be satisfied. The necessary conditions for the minimum results in the following system of equations:

\[
\frac{\partial \phi}{\partial d} = 0 \Rightarrow K d - R + \mu (C^T C d - C^T q) = 0
\]

Rearranging terms, the system of linear equations can be expressed as follows:

\[
(K + \mu C^T C) d = R + \mu C^T q
\]

The performance of the method depends on the value chosen for the penalty parameter \( \mu \). Large values, say of the order of \( \mu = 10^5 \), give accurate solutions; however, the resulting system of equations may be ill-conditioned. If \( \mu \) values are small as compared to other terms in the global equations, the solution will not satisfy the constraints very accurately. A general rule of thumb is to set \( \mu \) equal to \( 10^5 \) times the largest number in the global \( K \) matrix.

So to program this you need to do two things: Introduce in your problem data the constraints in the matrix form \( C d = q \) and then modify the stiffness and load vectors as above!

This problem looks difficult but the required solution is much shorter than the hint provided here!

**Solution:**

You can introduce the constraints in the `InputData.m` and then modify the stiffness and load vectors in the `NodalSoln.m`. However, it is noted that you need to apply the essential boundary conditions first and augmented the reduced global equations (Kf) with the lagrange multiplier. Therefore, the dimension of C is equal to the number of degrees of freedom without essential boundary conditions.

The multipoint constraint due to inclined support at node 1 is

\[
u_i \sin(\pi/6) + v_j \cos(\pi/6) = 0.
\]

We construct the constraint as follows:

```matlab
% Read information for constraints
C = zeros(1, neq - length(debc)); % The dimension of C is the same as
% neq minus the degrees of freedom
% from essential boundary condition
% condition. And there is only one
% constraint
C(1) = sin(pi/6); C(2) = cos(pi/6);
q = 0;
```

We modify the function `NodalSoln` to take \( C \) and \( q \) as input, and

```matlab
[m n] = size(C); % Extract Number of constraint
```

\( \mu = 10^5 \times \max(\max(\text{abs(Kf)})) \); %To use the penalty function approach,
% we choose the penalty parameter
% \( \mu \) equal to \( 10^5 \) times the largest number
% in the global \( K_f \) matrix

\[
K_f = K_f + \mu C^*C;
\]

% Modify the system of linear equations
\[
R_f = R_f + \mu C^*q;
\]

% as \( (K + \mu C^*C)d = R + \mu C^*q \)

Results of Problem 1:

<table>
<thead>
<tr>
<th>node #</th>
<th>( x )-displacement (m)</th>
<th>( y )-displacement (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.14291E-03</td>
<td>-2.96923E-03</td>
</tr>
<tr>
<td>2</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>1.68630E-02</td>
<td>1.27881E-02</td>
</tr>
<tr>
<td>4</td>
<td>-1.42857E-03</td>
<td>1.17595E-02</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>element #</th>
<th>strain</th>
<th>stress (Pa)</th>
<th>force (Nt)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.33E-04</td>
<td>23323807.58</td>
<td>23323.8076</td>
</tr>
<tr>
<td>2</td>
<td>3.33E-04</td>
<td>23323807.58</td>
<td>23323.8076</td>
</tr>
<tr>
<td>3</td>
<td>9.90E-04</td>
<td>69282032.3</td>
<td>69282.0323</td>
</tr>
<tr>
<td>4</td>
<td>-2.86E-04</td>
<td>-20000000</td>
<td>-20000</td>
</tr>
<tr>
<td>5</td>
<td>-1.71E-04</td>
<td>-12000000</td>
<td>-12000</td>
</tr>
</tbody>
</table>

To check the result, first we check if the constraint is satisfied
\[
Cu = 2.6946 \times 10^{-6} \approx 0
\]
So the constraint is reasonably satisfied.

Then we calculate the reaction force at node 2:

<table>
<thead>
<tr>
<th>node #</th>
<th>x-reaction force (Nt)</th>
<th>y-reaction force (Nt)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>20000.000000</td>
<td>69282.032303</td>
</tr>
</tbody>
</table>

Notice the axial force in element 3 and 4 together with the reaction force at node 2 satisfy the equilibrium condition.

The initial and deformed shape is
Problem 2 - Analysis of a uniformly loaded beam (hand calculation)

Consider a beam AB subjected to uniform transverse loading as shown in the figure. Using a single finite element, calculate the maximum deflection by hand. Assume EI is a constant.

Solution:

The stiffness matrix for the single element is

\[
K = \frac{EI}{L^3} \begin{bmatrix}
  12 & 6L & -12 & 6L \\
  6L & 4L^2 & -6L & 2L^2 \\
 -12 & 6L & 12 & -6L \\
  6L & 2L^2 & -6L & 4L^2 \\
\end{bmatrix}
\]

and the force due to the distributed loading is given by
\[
    f_\Omega = \frac{pL}{2} \begin{bmatrix} 1 \\ L/6 \\ 1 \\ -L/6 \end{bmatrix}
\]

where \( p \) is the distributed load.

The boundaries also have reaction forces and moments. The clamped end has a reaction force and moment and the pinned end has a reaction force. The force vector due to boundary forces and moments is given by

\[
    f_r = \begin{bmatrix} R_{u1} \\ 0 \\ R_{\theta 1} \\ 0 \\ R_{u2} \end{bmatrix} \mathcal{V} + \begin{bmatrix} 0 \\ R_{\theta 1} \\ 0 \\ 0 \end{bmatrix} \mathcal{M} = \begin{bmatrix} R_{u1} \\ R_{\theta 1} \\ R_{u2} \end{bmatrix}
\]

Assembling the force vector from all load contributions gives

\[
    f = \begin{bmatrix} \frac{pL}{2} + R_{u1} \\ \frac{pL^2}{12} + R_{\theta 1} \\ \frac{pL}{2} + R_{u2} \\ -\frac{pL^2}{12} \end{bmatrix}
\]

To solve for displacements and slopes, set up the equation \( Ku = f \)

\[
    \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & 6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} u_{y1} \\ \theta_{y1} \\ u_{y2} \\ \theta_{y2} \end{bmatrix} = \begin{bmatrix} pL/2 + R_{u1} \\ pL^2/12 + R_{\theta 1} \\ pL/2 + R_{u2} \\ -pL^2/12 \end{bmatrix}
\]

Because of the boundary conditions, we know that \( u_{y1}, \theta_{y1}, \) and \( u_{y2} \) are zero so we can partition the matrix.

\[
    \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & 6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} u_{y1} \\ \theta_{y1} \\ \theta_{y2} \end{bmatrix} = \begin{bmatrix} pL/2 + R_{u1} \\ pL^2/12 + R_{\theta 1} \\ pL/2 + R_{u2} \\ -pL^2/12 \end{bmatrix}
\]

From this we extracted the equation
Solving for $\theta_{y2}$ gives $\theta_{y2} = -\frac{pL^3}{48EI}$.

The equation for the deflection is given by

\[ u_y(\xi) = [N_{u1} \quad N_{\theta1} \quad N_{u2} \quad N_{\theta2}] \begin{pmatrix} 0 \\ 0 \\ 0 \\ -\frac{pL^3}{48EI} \end{pmatrix} = -\frac{pL^4}{384EI}(1 + \xi)^2(\xi - 1) \]

To find the location of the maximum deflection, we took the derivative of the deflection function (the slope) and set it equal to zero.

\[ \frac{du_y}{d\xi} = -\frac{pL^4}{384EI}(1 + \xi)^2 - \frac{pL^4}{192EI}(\xi - 1)(1 + \xi) = 0 \]

The maximum deflection thus occurs at $\xi = 1/3$. To find the deflection, substitute $\xi$ back into $u_y(\xi)$.

\[ u_y = -\frac{pL^4}{384EI}\left(1 + \frac{1}{3}\right)^2\left(\frac{1}{3} - 1\right) = \frac{pL^4}{324EI} \]

**Problem 3 – Analysis of a two-span beam** (MatLab)

Consider a two-span beam shown in the Figure. The beam is subjected to uniformly distributed loading, point force at $x = 2 \text{ m}$ and point moment at $x = 6 \text{ m}$ as shown in above Figure. The beam bending stiffness is $EI = 2 \times 10^7 \text{ N m}^2$.

Using the finite element program provided, plot the deflection, bending moment, and shear force distribution of the beam. If you have four elements, what is the optimal mesh? Repeat the solution with the eight-element mesh, four for each span. Comment on the results. Is your solution right? How can you improve the finite element solution?

**Solution:**
If we have four elements, we need to place one element node on the point force and point moment. Modify the input file for the mesh and data. The boundary conditions are that there are no vertical deflections at node A, B and C.

The plotting for deflection, bending moments and shear force are as follows:

![Displacement Plot](image1)

![Moment Plot](image2)
To get accurate results, we would use more elements in the segment with the distributed load. Now we use 8 elements in the distributed load region:
The 4 and 8 element solution produced similar diagrams for deflection, slightly different diagrams for bending moment, and very different diagrams for shear force. Hence, we can conclude that the solution is not exactly accurate, but provides a good approximate solution for deflection, and a rough trend for bending moment. The solution for shear force is poor with few elements.

This poor solution for shear force is obtained because the assumed shape factor function for displacement is cubic. Since

\[ V = -\frac{d}{dx}\left(EI\frac{d^2u_x}{dx^2}\right) = \text{constant for cubic } u_y \]

The shear force calculated will be a constant over the length of the element. However, we know that with a uniform distributed load, the actual theoretical shear force should be linear.

Thus, the finite element solution can be improved by increasing the number of elements in the solution, in order to better approximate the bending moment and shear force solution diagrams.

We next try to use 256 elements in the distributed load region.
Now in the distributed load region, it shows a nearly linear shear diagram.
**Problem 4 – Analysis of a two-span beam (Ansys)**

Repeat problem 3 using Ansys. Compare your answers with those computed via the MatLab programs. Provide a complete list of the command sequence used to solve this problem.

**ANSYS Steps:**

1. Element type: BEAM 3 -2d
2. Set Real Constants:
   - Area = 1e-2, IZZ= 1e-4, height = 0.1
3. Material Models: Structural Linear Elastic Isotropic:
   - EX= 200e9, PRXY= 0.3
   - K,1,0,0
   - K,2,2,0
   - K,3,4,0
   - K,4,6,0
   - K,5,8,0
5. Set Lines:
   - L,1,2
   - L,2,3
   - L,3,4
   - L,4,5
6. Meshing >> Mesh tool
7. Apply displacements on keypoints (apply constraints on geometry whenever possible)
   - 0 Y displacement on keypoints 1 and 5, 0 x and y displacement on keypoint 3.
8. Apply loads on keypoints
   - Keypoint 2: FY= -10000, Keypoint 4: ROTZ= 5000
9. Apply pressure on all beams:
   - positive 2000: sign convention is positive downwards for pressure on beams.
10. Solve >> current LS
11. Plot deformed shape
12. List MFOR and MMOMZ
    - Define tables SMISC, 2, 8, 6, 12.
    - List tables.
13. Remesh: meshing tool >> Set lines>> 2 elements per line
14. Reapply pressure on beams.
15. Solve >> Current LS (for the 8 element solution)
Results with 4 elements:

```
1

DISPLACEMENT

STEP=1
SUB =1
TIME=1
DMX =.597E-03

Y

X

3

4

1

2

3

4

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

<table>
<thead>
<tr>
<th>STAT</th>
<th>CURRENT M_I</th>
<th>CURRENT M_J</th>
<th>CURRENT V_I</th>
<th>CURRENT V_J</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEM</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.25011E-11</td>
<td>9968.8</td>
<td>-6984.4</td>
<td>-2984.4</td>
</tr>
<tr>
<td>2</td>
<td>9968.8</td>
<td>-8062.5</td>
<td>7015.6</td>
<td>11016.</td>
</tr>
<tr>
<td>3</td>
<td>-8062.5</td>
<td>2468.8</td>
<td>-7265.6</td>
<td>-3265.6</td>
</tr>
<tr>
<td>4</td>
<td>-2531.2</td>
<td>-0.22737E-12</td>
<td>-3265.6</td>
<td>734.38</td>
</tr>
</tbody>
</table>

MINIMUM VALUES

| ELEM | 3 | 2 | 3 | 3 |
| VALUE | -8062.5 | -8062.5 | -7265.6 | -3265.6 |

MAXIMUM VALUES


Finite Element Analysis for Mechanical & Aerospace Design

Page 12 of 29
Results with 8 elements:

```
1
DISPLACEMENT
STEP = 1
SUB = 1
TIME = 1
DIMX = .597E-03

STAT CURRENT CURRENT CURRENT CURRENT
ELEM M_I M_J V_I V_J
1 -0.62528E-12 5984.4 -6984.4 -4984.4
2 5984.4 9968.7 -4984.4 -2984.4
3 9968.7 1953.1 7015.6 9015.6
4 1953.1 -8062.5 9015.6 11016.
5 -8062.5 -1796.9 -7265.6 -5265.6
6 -1796.9 2468.8 -5265.6 -3265.6
7 -2531.2 -265.62 -3265.6 -1285.6
8 -265.62 -0.17053E-12 -1265.6 734.38
```
MINIMUM VALUES
ELEM  5  4  5  5
VALUE  -8062.5  -8062.5  -7265.6  -5265.6

MAXIMUM VALUES
ELEM  3  2  4  4
VALUE  9968.7  9968.7  9015.6  11016.

Results with 40 elements:
PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

<table>
<thead>
<tr>
<th>STAT</th>
<th>CURRENT</th>
<th>CURRENT</th>
<th>CURRENT</th>
<th>CURRENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEM</td>
<td>M_I</td>
<td>M_J</td>
<td>V_I</td>
<td>V_J</td>
</tr>
<tr>
<td>1</td>
<td>-0.97025E-11</td>
<td>1356.9</td>
<td>-6984.4</td>
<td>-6584.4</td>
</tr>
<tr>
<td>2</td>
<td>1356.9</td>
<td>2633.7</td>
<td>-6584.4</td>
<td>-6184.4</td>
</tr>
<tr>
<td>3</td>
<td>2633.7</td>
<td>3830.6</td>
<td>-6184.4</td>
<td>-5784.4</td>
</tr>
<tr>
<td>4</td>
<td>3830.6</td>
<td>4947.5</td>
<td>-5784.4</td>
<td>-5384.4</td>
</tr>
<tr>
<td>5</td>
<td>4947.5</td>
<td>5984.4</td>
<td>-5384.4</td>
<td>-4984.4</td>
</tr>
<tr>
<td>6</td>
<td>5984.4</td>
<td>6941.2</td>
<td>-4984.4</td>
<td>-4584.4</td>
</tr>
<tr>
<td>7</td>
<td>6941.2</td>
<td>7818.1</td>
<td>-4584.4</td>
<td>-4184.4</td>
</tr>
<tr>
<td>8</td>
<td>7818.1</td>
<td>8615.0</td>
<td>-4184.4</td>
<td>-3784.4</td>
</tr>
<tr>
<td>9</td>
<td>8615.0</td>
<td>9331.9</td>
<td>-3784.4</td>
<td>-3384.4</td>
</tr>
<tr>
<td>10</td>
<td>9331.9</td>
<td>9968.7</td>
<td>-3384.4</td>
<td>-2984.4</td>
</tr>
<tr>
<td>11</td>
<td>9968.7</td>
<td>8525.6</td>
<td>7015.6</td>
<td>7415.6</td>
</tr>
<tr>
<td>12</td>
<td>8525.6</td>
<td>7002.5</td>
<td>7415.6</td>
<td>7815.6</td>
</tr>
<tr>
<td>13</td>
<td>7002.5</td>
<td>5399.4</td>
<td>7815.6</td>
<td>8215.6</td>
</tr>
<tr>
<td>14</td>
<td>5399.4</td>
<td>3716.3</td>
<td>8215.6</td>
<td>8615.6</td>
</tr>
<tr>
<td>15</td>
<td>3716.2</td>
<td>1953.1</td>
<td>8615.6</td>
<td>9015.6</td>
</tr>
<tr>
<td>16</td>
<td>1953.1</td>
<td>110.00</td>
<td>9015.6</td>
<td>9415.6</td>
</tr>
<tr>
<td>17</td>
<td>110.00</td>
<td>-1813.1</td>
<td>9415.6</td>
<td>9815.6</td>
</tr>
<tr>
<td>18</td>
<td>-1813.1</td>
<td>-3816.2</td>
<td>9815.6</td>
<td>10216.0</td>
</tr>
<tr>
<td>19</td>
<td>-3816.2</td>
<td>-5899.4</td>
<td>10216.0</td>
<td>10616.0</td>
</tr>
<tr>
<td>20</td>
<td>-5899.4</td>
<td>-8062.5</td>
<td>10616.0</td>
<td>11016.0</td>
</tr>
<tr>
<td>21</td>
<td>-8062.5</td>
<td>-6494.9</td>
<td>-7265.6</td>
<td>-6865.6</td>
</tr>
<tr>
<td>22</td>
<td>-6494.9</td>
<td>-5316.2</td>
<td>-6865.6</td>
<td>-6465.6</td>
</tr>
<tr>
<td>23</td>
<td>-5316.2</td>
<td>-4063.1</td>
<td>-6465.6</td>
<td>-6065.6</td>
</tr>
<tr>
<td>24</td>
<td>-4063.1</td>
<td>-2890.0</td>
<td>-6065.6</td>
<td>-5665.6</td>
</tr>
<tr>
<td>25</td>
<td>-2890.0</td>
<td>-1796.9</td>
<td>-5665.6</td>
<td>-5265.6</td>
</tr>
<tr>
<td>26</td>
<td>-1796.9</td>
<td>-783.75</td>
<td>-5265.6</td>
<td>-4865.6</td>
</tr>
<tr>
<td>27</td>
<td>-783.75</td>
<td>149.38</td>
<td>-4865.6</td>
<td>-4465.6</td>
</tr>
<tr>
<td>28</td>
<td>149.38</td>
<td>1002.5</td>
<td>-4465.6</td>
<td>-4065.6</td>
</tr>
<tr>
<td>29</td>
<td>1002.5</td>
<td>1775.6</td>
<td>-4065.6</td>
<td>-3665.6</td>
</tr>
<tr>
<td>30</td>
<td>1775.6</td>
<td>2468.8</td>
<td>-3665.6</td>
<td>-3265.6</td>
</tr>
<tr>
<td>31</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>2468.8</td>
<td>-3265.6</td>
<td>-2865.6</td>
<td></td>
</tr>
</tbody>
</table>
The ANSYS solution matches up well with the MATLAB solution. With 40 elements, the deformation plot looks similar to the MATLAB deformation plot. The values for bending moment and shear force for each of the 3 ANSYS outputs match at the same keypoints, thus are consistent. A comparison of the listed values of bending moment and shear force to the 80 element MATLAB diagrams shows that they are similar. However, the values vary slightly from the 4 and 8 element MATLAB diagrams. This is consistent with the idea that adding more elements will increase the accuracy of the results obtained.

**Problem 5 – Beams with non-uniform loading** (MatLab programming required)

Consider a beam finite element with trapezoidal loading as shown below. Derive the equivalent nodal forces for this element.

With this nodal force, modify the MATLAB codes to solve the following problem for a uniform beam subjected to a linearly increasing load. Plot the deflection, bending moment, and shear force distribution of the beam. The modifications should be general enough to accommodate many elements and put one element node at the support.
Assuming the following numerical data:
\[ a = 5 \text{kN/m}, \quad L = 5 \text{m}, \quad E = 200 \text{GPa}, \quad I = 10^6 \text{mm}^4 \]

**Solution:**

We define the distributed load as a function of \( \xi \):

\[
q(\xi) = \left( \frac{q_2 - q_1}{2} \right) \xi + \left( \frac{q_1 + q_2}{2} \right)
\]

or \( q(\xi) = A \xi + B, \quad -1 < \xi < 1 \)

And we use the following interpolation functions for a 2-noded beam element:

\[
N_{u1} = \frac{1}{4} (2 - 3 \xi + \xi^3)
\]

\[
N_{01} = \frac{L^e}{8} (1 - \xi - \xi^2 + \xi^3)
\]

\[
N_{u2} = \frac{1}{4} (2 + 3 \xi - \xi^3)
\]

\[
N_{02} = \frac{L^e}{8} (-1 - \xi + \xi^2 + \xi^3)
\]

Applying the following formula:

\[
f_{e} = \frac{L^e}{2} \int_{-1}^{1} q(\xi) \left[ N(\xi) \right] d\xi
\]

(for the first term of the load vector):

\[
f_e(1) = \frac{L^e}{2} \int_{-1}^{1} (A \xi + B) \left[ \frac{1}{4} (2 - 3 \xi + \xi^3) \right] d\xi
\]

\[
f_e(1) = \frac{L^e}{8} \int_{-1}^{1} 2A \xi - 3A \xi^2 + A \xi^4 + 2B - 3B \xi + B \xi^3 d\xi
\]

\[
f_e(1) = \frac{L^e}{8} \int_{-1}^{1} 2B + (2A - 3B) \xi - 3A \xi^2 + B \xi^3 + A \xi^4 d\xi
\]

and further simplifying
\[ f_e(l) = \frac{L^2}{8} \left[ 4B - 2A + \frac{2A}{5} \right] \]
\[ f_e(l) = \frac{L^2}{20} (7q_i + 3q_2) \]

Proceeding in a similar fashion for the other components of the force vector we obtain

\[
f_e^\alpha = \frac{L^2}{2} \int_{-1}^{1} q(\xi) [N(\xi)]^T d\xi =
\begin{bmatrix}
\frac{L^2}{20} (7q_1 + 3q_2) \\
\frac{(L^3)^2}{60} (3q_1 + 2q_2) \\
\frac{L^2}{20} (3q_1 + 7q_2) \\
\frac{-(L^3)^2}{60} (2q_1 + 3q_2)
\end{bmatrix}
\]

Modify the InputGrid.m:

```matlab
nsd = 2; % number of spatial dimensions
n = 128; % number of total elements in each span
nel = 2*n; % number of total elements
nno = nel+1; % number of total nodes
nen = 2; % number of nodes on each element
L = 5;
Nodes = [linspace(0,L/3,n+1) linspace(L/3,L,n+1)]; % Define the node position.
Nodes = unique(Nodes); % There is a redundant

Elems = [1:(nno-1); 2: nno ]; % Define the global node number (connectivity) in each beam element

Modify the InputData.m:

```matlab
n = nel/2; % Extract number of element in each span
EI = ones(nel,1)*2e5;
a = 5e3;
L = 5;
q = a*Nodes/L; % Extract q at each node, it is not the distributed load as in original program
debc = [2*n+1, 2*nno-1, 2*nno]; % Define the list of DOF which have essential boundary conditions
```

Modify the BeamElement.m:

```matlab
q1 = q(glb1);
q2 = q(glb2);
```
be = [1/20*L*(7*q1+3*q2); 1/60*L^2*(3*q1+2*q2); 1/20*L*(3*q1+7*q2); -1/60*L^2*(2*q1+3*q2)];

We use 128 elements in each span and have the following plots:
Check:
The displacement obtained from the MATLAB solution matches reasonably well with the
ANSYS solution of the same beam under the same loading conditions:

```
PRINT U  NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP=  1  SUBSTEP=  1
TIME=  1.0000  LOAD CASE=  0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

    NODE  UX    UY    UZ    USUM
     1  0.0000 -0.11788E-01  0.0000  0.11788E-01
     2  0.0000  0.0000  0.0000  0.0000
     3  0.0000  0.88413E-02  0.0000  0.88413E-02
     4  0.0000  0.0000  0.0000  0.0000

MAXIMUM ABSOLUTE VALUES
   NODE  VALUE
     0  0.0000
     1  1
     0  0
     1  1

Problem 6 – Analysis of plane frames (MatLab)

This problem develops elements that combine (in an uncoupled way) the axial
deformation of the truss 2-node elements from HW1 and the 2-node beam elements
discussed in class. Members in a plane frame are designed to resist axial and bending
deforations.

a) Using the known results from 2-node truss (axial-deformation) and beam
elements, derive the response of a plane frame element \([K^e]\{u^e\} = \{F^e\}\) in local and
global coordinates (see Figures below).

```

<table>
<thead>
<tr>
<th>Local coordinates</th>
<th>Global coordinates</th>
</tr>
</thead>
<tbody>
<tr>
<td>(UX)</td>
<td>(U_YX)</td>
</tr>
<tr>
<td>(UY)</td>
<td>(U_YY)</td>
</tr>
<tr>
<td>(UZ)</td>
<td>(U_Z)</td>
</tr>
</tbody>
</table>
```
Hint: There are three degrees of freedom per node, u, v and θ. The rotation matrix from global to local coordinate system is given by:

\[
\begin{pmatrix}
    d_1 & \cos \alpha & \sin \alpha & 0 & 0 & 0 & 0 \\
    d_2 & -\sin \alpha & \cos \alpha & 0 & 0 & 0 & 0 \\
    d_3 & 0 & 0 & 1 & 0 & 0 & 0 \\
    d_4 & 0 & 0 & 0 & \cos \alpha & \sin \alpha & 0 \\
    d_5 & 0 & 0 & 0 & -\sin \alpha & \cos \alpha & 0 \\
    d_6 & 0 & 0 & 0 & 0 & 0 & 1
\end{pmatrix}
\begin{pmatrix}
    u_1 \\
    v_1 \\
    \theta_1 \\
    u_2 \\
    v_2 \\
    \theta_2
\end{pmatrix}
\]

Modify the MatLab software provided for the analysis of beams to analyze plane frames. For the two-story frame shown below, use six 2-noded elements to compute the moments, shear forces, horizontal and vertical deflections. The bending stiffness for all beams and columns is \( EI = 2.5 \times 10^7 \text{ N m}^2 \). \( EA = 2.5 \times 10^9 \text{ N} \). For each element, report the computed axial forces, moments and shear forces at element ends. Choose the most left bottom node as the origin.
This problem develops elements that combine (in an uncoupled way) the axial deformation of the truss 2-node elements from HW1 and the 2-node beam elements discussed in class. Members in a plane frame are designed to resist axial and bending deformations.

b) Using the known results from 2-node truss (axial-deformation) and beam elements, derive the response of a plane frame element \([K^e]\{u^e\} = \{F^e\}\) in local and global coordinates (see Figures below).

**Local coordinates**

**Global coordinates**

**Hint:** There are three degrees of freedom per node, \(u, v\) and \(\theta\). The rotation matrix from global to local coordinate system is given by:
Modify the MatLab software provided for the analysis of beams to analyze plane frames. For the two-story frame shown below, use six 2-noded elements to compute the moments, shear forces, horizontal and vertical deflections. The bending stiffness for all beams and columns is $EI = 2.5 \times 10^7 \text{ N m}^2$. $EA = 2.5 \times 10^9 \text{ N}$. For each element, report the computed axial forces, moments and shear forces at element ends. Choose the most left bottom node as the origin.

**Solution:**
With the local order of degrees of freedom as shown in the above figure, the local element equations are therefore as follows:
The transformation between the global and the local degrees of freedom can be written as follows:

\[
\begin{pmatrix}
\frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\
0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\
0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\
-\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\
0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^3} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^3} \\
0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L}
\end{pmatrix}
\begin{pmatrix}
d_1 \\
d_2 \\
d_3 \\
d_4 \\
d_5 \\
d_6
\end{pmatrix}
= 
\begin{pmatrix}
\frac{1}{2} q_L \\
\frac{1}{2} q_L \\
\frac{1}{12} q_L^2 \\
\frac{1}{2} q_L \\
\frac{1}{2} q_L \\
-\frac{1}{12} q_L^2
\end{pmatrix}
\Rightarrow k_d = f_t
\]

Thus, we have the following element equations in terms of global degrees of freedom and applied nodal loads in the global equations:

\[
k_d = f
\]

where \(k = T^T k_t T\) and \(f = T^T f_t\).

In Matlab, it is much easier to write the element equations in local coordinates first and then carry out the matrix multiplications using Matlab.

<table>
<thead>
<tr>
<th>node #</th>
<th>x-displacement (m)</th>
<th>y-displacement (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>6.37058E-04</td>
<td>-3.32040E-06</td>
</tr>
<tr>
<td>3</td>
<td>1.24841E-03</td>
<td>-5.91198E-06</td>
</tr>
<tr>
<td>4</td>
<td>1.24327E-03</td>
<td>-2.28880E-05</td>
</tr>
<tr>
<td>5</td>
<td>6.35184E-04</td>
<td>-1.58796E-05</td>
</tr>
</tbody>
</table>
MATLAB results are generally close to the corresponding ANSYS values (see Problem 7) except for the moment and shear values at elements 3 and 4. We note that these are the elements at which the distributed loads are applied. MATLAB results are likely to be inaccurate for these, since the load is essentially simulated by two point loads at each element’s end, and with a small number of elements this is a poor model, hence the discrepancies.

The initial and deformed shapes are in the following figure:
Problem 7 – Analysis of plane frames (Ansys)

Repeat the solution of problem 6 using Ansys and provide a comparison with the MatLab results. Your solution write up should include all Ansys commands, element type, etc.

ANSYS Steps:
1. Element type: BEAM 3 -2d
2. Set Real Constants:
   - Area = 1e-2, IZZ= 1e-4, height = 0.1
3. Material Models: Structural Linear Elastic Isotropic:
   - EX= 250e9, PRXY= 0.3
   - N,1,0,0
   - N,2,0,3
   - N,3,0,6
   - N,4,4,6
   - N,5,4,3
   - N,6,4,0
5. Set Elements:
   - E,1,2
   - E,2,3
   - E,3,4
   - E,2,5
   - E,4,5
   - E,5,6
6. Apply displacements on nodes
   0 displacement in X, Y, ROTZ on Nodes 1 and 6
7. Apply loads on nodes
   Nodes 2 and 3: FX=4000
8. Apply pressure on beams:
   positive 2000 in FY direction on beams 3 and 4
9. Solve >> current LS
10. Plot deformed shape
11. List SDIR, MFOR and MMOMZ (axial forces, shear forces and moments)
   Define tables LS, 1, SMISC, 2, 8, 6, 12.
   List tables.

PRINT U  NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP=  1 SUBSTEP=  1
TIME=  1.0000   LOAD CASE=  0
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

<table>
<thead>
<tr>
<th>NODE</th>
<th>UX</th>
<th>UY</th>
<th>UZ</th>
<th>USUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.63706E-03-0.33204E-05</td>
<td>0.0000</td>
<td>0.63707E-03</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.12484E-02-0.59120E-05</td>
<td>0.0000</td>
<td>0.12484E-02</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.12433E-02-0.22888E-04</td>
<td>0.0000</td>
<td>0.12435E-02</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.63518E-03-0.15880E-04</td>
<td>0.0000</td>
<td>0.63538E-03</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

MAXIMUM ABSOLUTE VALUES

<table>
<thead>
<tr>
<th>NODE</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.12484E-02-0.22888E-04</td>
</tr>
<tr>
<td>4</td>
<td>0.12484E-02</td>
</tr>
</tbody>
</table>

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

<table>
<thead>
<tr>
<th>STAT</th>
<th>CURRENT AXIALF</th>
<th>CURRENT M_I</th>
<th>CURRENT M_J</th>
<th>CURRENT V_I</th>
<th>CURRENT V_J</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ELEM</th>
<th>CURRENT AXIALF</th>
<th>CURRENT M_I</th>
<th>CURRENT M_J</th>
<th>CURRENT V_I</th>
<th>CURRENT V_J</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-2767.0</td>
<td>-7155.2</td>
<td>3692.8</td>
<td>-3616.0</td>
<td>-3616.0</td>
</tr>
<tr>
<td>2</td>
<td>-2159.6</td>
<td>-721.56</td>
<td>1641.2</td>
<td>-787.58</td>
<td>-787.58</td>
</tr>
<tr>
<td>3</td>
<td>-3212.4</td>
<td>1641.2</td>
<td>-5720.2</td>
<td>-2159.6</td>
<td>5840.4</td>
</tr>
<tr>
<td>4</td>
<td>-1171.5</td>
<td>4414.4</td>
<td>-9156.2</td>
<td>-607.36</td>
<td>7392.6</td>
</tr>
<tr>
<td>5</td>
<td>-5840.4</td>
<td>-5720.2</td>
<td>3917.0</td>
<td>-3212.4</td>
<td>-3212.4</td>
</tr>
<tr>
<td>6</td>
<td>-13233.</td>
<td>-5239.1</td>
<td>7912.8</td>
<td>-4384.0</td>
<td>-4384.0</td>
</tr>
</tbody>
</table>

MINIMUM VALUES

<table>
<thead>
<tr>
<th>ELEM</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>-13233.</td>
</tr>
</tbody>
</table>

MAXIMUM VALUES

<table>
<thead>
<tr>
<th>ELEM</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>-1171.5</td>
</tr>
</tbody>
</table>

The displacement values and axial forces from the ANSYS solution matched the MATLAB solution, however, the moment and shear forces differ at elements 3 and 4. This difference is due to the way the finite element code handles distributed forces in the MATLAB solution. Uniform distributed forces are converted to point forces and moments at the ends of the element, where:
For each element, the additional applied moment at the end is:

\[
moment = \frac{L^2}{12} q_t
\]

When the element length is large, \( L^2/12 \) will be large, thus the moment will be significant.

Since the MATLAB code uses 1 element for each beam, \( L=4 \) for the horizontal beams, and the moment = 2666.6. This moment is added to the actual moment obtained at the nodes of beams 3 and 4, thus the MATLAB moments at beams 3 and 4 are larger than the ANSYS moments by 2666.6.

This problem can be resolved by refining the mesh to have more elements per beam. By making \( L \) smaller, the \( L^2 \) term will become even smaller, and the additional moment term will approach zero. The result is that with a refined mesh the MATLAB solution will match the ANSYS solution.

For the shear forces, the MATLAB program gives a constant value of \( V \), as the assumed shape factor function for displacement is cubic. Since

\[
V = -\frac{d}{dx} \left( EI \frac{d^2 u_y}{dx^2} \right) = \text{constant for cubic } u_y
\]

The shear force calculated from the MATLAB code will be a constant over the length of the element. This is contrary to theory that the shear force increases linearly with a uniform distributed load, which the ANSYS solution obtains its values from. Hence, the MATLAB shear stresses obtained from beams 3 and 4 are simply the average of the ANSYS values.