Computational Structural Analysis
2D Direct Stiffness Code

Enis BAHADIR

10545700

June 18, 2017

1 Benchmarks

The accuracy and robustness of the code is tested by solving some sample problems, the exact solutions of which were computed apriori via the classical beam theory. The results are presented in this section using the following scheme:

- Demonstration of the problem
- Input file that addresses the problem
- Output file generated by the program after analysis
- Diagrams obtaned by postprocessing
- Comparison of the results of numerical and analytical solutions

Each problem considered in the first part comprise a single or continous beam subjected to a certain external loading and support condition. The second part deals with a one-bay one-storey frame under different loads and restraints, and the third and final part is devoted to problems involving a prestressed beam, with each problem having different tendon layouts and/or support conditions.

1.1 Single Beam Under Various Loading and Support Conditions

1.1.1 Problem 1: Simple Beam - Uniformly Distributed Load

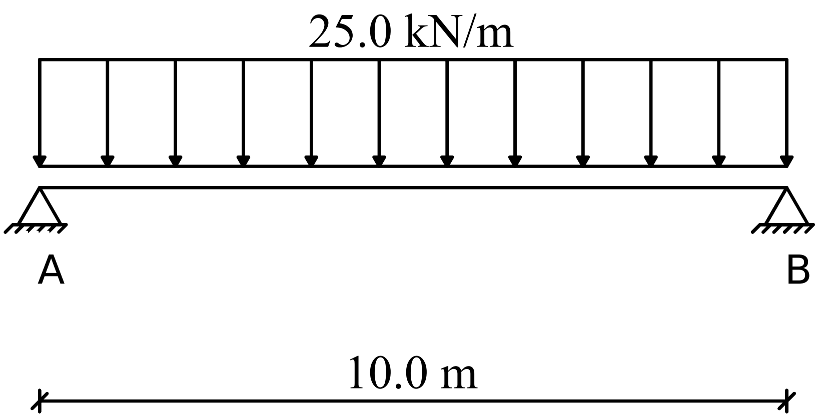

Figure 1: Problem 1: Loading, geometry and supports

```
# Default units - kN/m/C
# ------------------------------------------------------------
# NODES <number of nodes> <x coord, y coord>
2
0.0 0.0
10.0 0.0
# ELEMENTS <number of elements> <start node, end node>
1
1 2
# SECTIONS <number of sections> <A, I, h, X>
1
0.32 0.0170666666666667 0.8 0.0
# SECTION INCIDENCES
1
# MATERIALS <number of materials> <E, v, alpha, gamma>
1
30.e6 0.2 0.0 0.0
# MATERIAL INCIDENCES
1
# RESTRAINTS <number of restraints> <node number, direction>
4
1 1
1 2
2 1
2 2
# LINKS <number of links> <master node, slave node, direction>
0
# ELASTIC RESTRAINTS <number of el. restraints> <node number, direction, k>
0
# ELASTIC LINKS <number of links> <node1, node2, direction, k>
0
# NODAL LOADS <number of nodal loads> <node, direction, magnitude>
0
# ELEMENT LOADS <number> <Element, Px, Py, Py2, DTtop, DTbottom>
1
1 0.0 -25.0 -25.0 0.0 0.0
# PRESTRESSING <number> <Element, e1, em, e2, P>
0

==============================================================================
INPUT DATA: [kN/m/C]
==============================================================================

# NODE COORDINATES
Node          x-coord     y-coord
  1             0.00        0.00
  2            10.00        0.00

# SECTIONS
```

```
Sec.No              A                    I                  h
X
  1           .3200D+00          .1707D-01             0.8000
0.0000

# MATERIALS
Mat.No              E                    v                alpha
gamma
  1           .3000D+08              0.20             .0000D+00
0.0000

# ELEMENTS
El.No      Start Node      End Node       Section       Material
  1            1              2              1              1

# RESTRAINTS
Restraint No         Node             Direction
  1                   1                  1
  2                   1                  2
  3                   2                  1
  4                   2                  2

# LINKS
Link No          Master Node          Slave Node          Direction

# ELASTIC RESTRAINTS
E. Restr. No        Node              Direction               K

# ELASTIC LINKS
E. Link. No        Node 1               Node 2            Direction
K

# NODAL LOADS
Node         Direction          Magnitude

# ELEMENT LOADS
Element       Px         Py1          Py2        DTtop       DTbottom
  1          0.00      -25.00       -25.00        0.00         0.00

# PRESTRESSING
Element       e1          em           e2          P

================================================================================
ANALYSIS RESULTS
================================================================================

>> NODAL DISPLACEMENTS

Node            DX                     DY
```

```
  1      0.0000000000D+00      0.0000000000D+00
  2      0.0000000000D+00      0.0000000000D+00


>> MEMBER END FORCES

Member     N1          T1          M1          N2          T2
M2
  1        0.00       125.00       -0.00       0.00       125.00
0.00
```