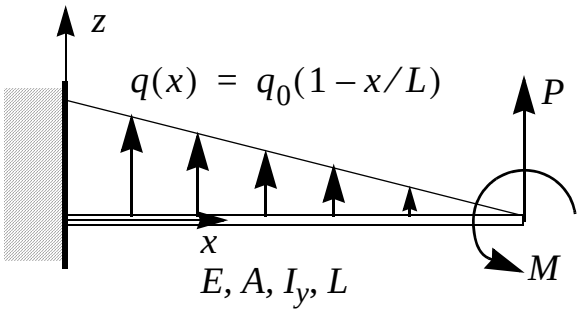
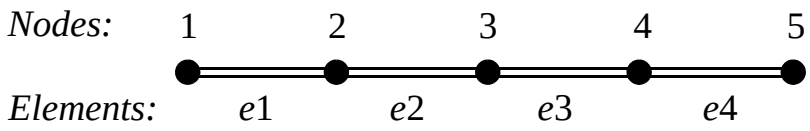
Matlab program: beam1D

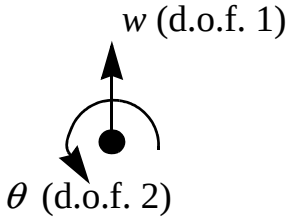
Exemple:



External load can be applied in terms of point forces, moments and distributed loads. Only linearly varying distributed loads can be applied, defined by the load intensity q [force/unit length]. Thus, in the example to the right, q_0 has unit [force/unit length]. q is written to the input file as the value at the local nodes in each element subjected to a distributed load.

FEM analysis using 4 elements:





Input data to the Matlab program is written to the file: ***beam1D.inp*** (see below), and results from an analysis is stored in the file ***beam1D.out***.

```
*NODE [node number, x-coordinate]
1, 0.0
2, 0.25
3, 0.50
4, 0.75
5, 1.0
*ELEMENT  [element number, node 1, node 2, E-modul, Iy]
1, 1,  2,  200E9,  2.0833E-7
2, 2,  3,  200E9,  2.0833E-7
3, 3,  4,  200E9,  2.0833E-7
4, 4,  5,  200E9,  2.0833E-7
*ZeroDOF [node number, dof]
1, 1
1, 2
*PointForceMoment [node number, dof, value]
5, 1, P
5, 2, M
*DistributedForce [element number, qI, qJ]
1, q0, 0.75q0
2, 0.75q0, 0.5q0
3, 0.5q0, 0.25q0
4, 0.25q0, 0
*END
```

Längd L = 1 m

Elastic modulus
E = 200GPa

h = 0.05 m

b = 0.02 m

$$I_y = \frac{bh^3}{12}$$

Zero displacement Boundary condition