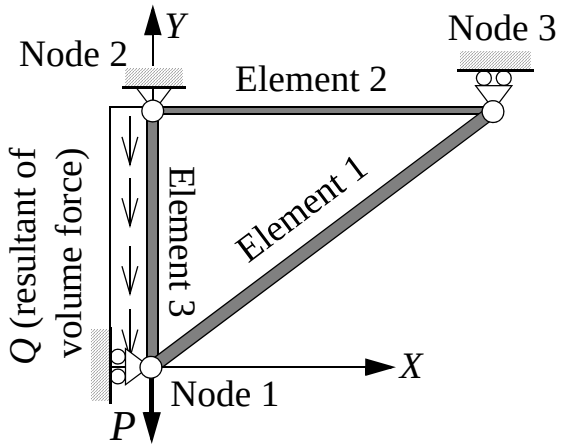
Matlab program: truss2D

The input to the program must be written in a file named ***truss2D.inp***. The structure of the input data is illustrated by the example below. Only prescribed point forces and axially distributed element loads can be applied as external load. Results from an analysis is stored in the file: ***truss2D.out***, which is generated during an analysis.

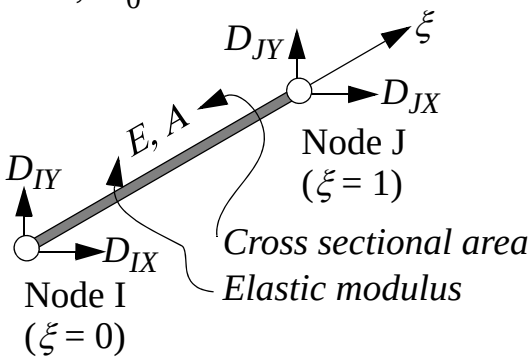


Node coordinates		
Node	X / L	Y / L
1	0	0
2	4	3
3	0	3

Element properties		
Elem.	E / GPa	$Area / A_0$
1	200	0.0025
2	200	0.0004
3	200	0.0006

$L = 1$ m, $A_0 = 0.0001$ m^2

2 node truss element with
2 D.O.F. (degrees of freedom) /node



"Zero" displacement boundary conditions: $D_{1X} = D_{2Y} = D_{3X} = D_{3Y} = 0$

Applied point force: $F_{1Y} = -1000$ N

The force per unit volume is expressed as: $K = q_0 + q_1\xi + q_2\xi^2$
K is acting in the axial direction of the element and positive in the positive ξ-dir.
In the current case $q_1 = q_2 = 0$ and $q_0 = -Q/(A_0 3L)$. For Q = 18 kN = > q0 = –1.0e7

Input data to the Matlab program is written to the file: ***truss2D.inp***, as shown below!

```
*NODE [Node number, X-coordinate, Y-coordinate]
1, 0.0, 0.0
2, 4.0, 3.0
3, 0.0, 3.0
*ELEMENT [Element no., Node 1, Node 2, E, A]
1, 1, 2, 200e9, 25.0e-4
2, 2, 3, 200e9,  4.0e-4
3, 1, 3, 200e9,  6.0e-4
*ZeroDiplacement [Node number, D.O.F.]
1,1
2,1
2,2
3,2
*PointForce [Node number, D.O.F., force value]
1,2, -5.0e4
*VolumeForce [Element number, q0, q1, q2]
3, -1.0e7, 0,  0
*END
```