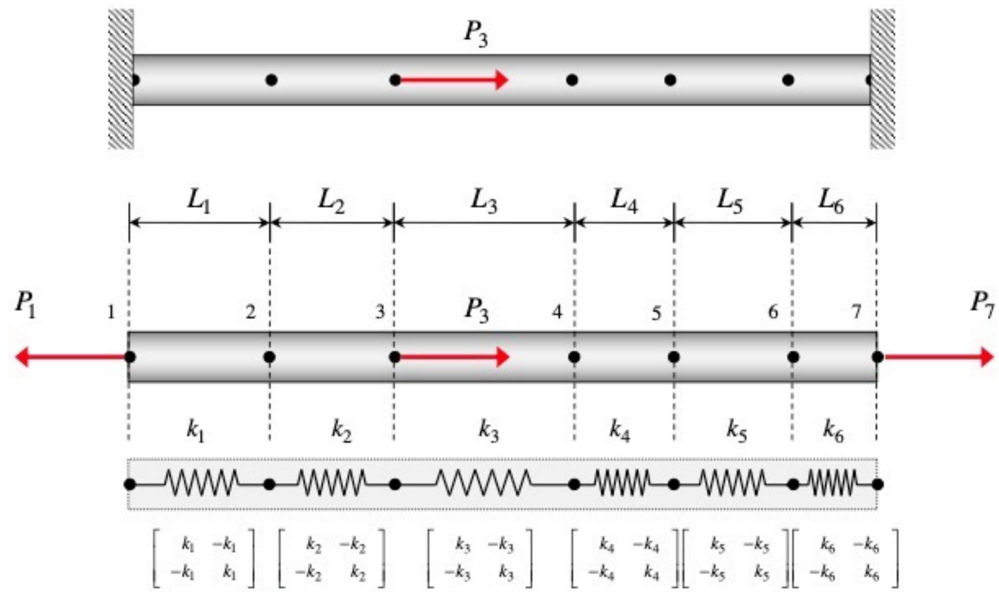
Building and solving a finite element model for a rod

METHOD

- *Defining the nodes and elements for the problem*. Choose a set of $N+1$ *nodes* along the length of the rod at locations $x_1(=0), x_2, x_3, ..., x_N, x_{N+1}(=L)$. The subdomain of $x_i < x < x_{i+1}$ is known as the ith *element* of length $L_i = x_{i+1} - x_i$, for $i = 1,2,...,N$. The value of $k_i = (EA)_i / L_i$ is determined through the average value of EA over the i^{th} element and the element length L_i.
- *Constructing the global stiffness matrix*. Construct the stiffness matrix [K]. The resulting matrix will be tri-diagonal and of size $(N+1)\times(N+1)$.
- *Constructing the force vector*. Construct the force vector $\{F\}$ as being made up on the resultant *external* force acting on each node. The resulting vector will be of length $N+1$.
- *Enforcing fixed-displacement boundary conditions*. The fixed-displacement boundary conditions are enforced through the elimination of appropriate terms in the resulting stiffness matrix $[K]$ and forcing vector $\{F\}$. For example, if the ith node has a fixed (zero) displacement, we eliminate the ith row and ith column of $[K]$ and the ith row of $\{F\}$. If the problem has "n" fixed nodal displacements, then the stiffness matrix and force vector will be of sizes $(N-n+1)\times(N-n+1)$ and $N-n+1$, respectively[1].
- *Solving*. The nodal displacements u_k ; $k = 1,2,...,N-n+1$ are found from the solution of the algebraic equilibrium equations:

$$[K]\{u\} = \{F\}$$

through a linear equation solver in an application such as Matlab or Mathematica.



$$k_j = \frac{E_j A_j}{L_j}$$

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 & & & & \\ -k_1 & k_2 & -k_2 & 0 & & & \\ 0 & -k_2 & k_2+k_3 & -k_3 & 0 & & \\ & 0 & -k_3 & k_3+k_4 & -k_4 & 0 & \\ & & 0 & -k_4 & k_4+k_5 & -k_5 & 0 \\ & & & 0 & -k_5 & k_5+k_6 & -k_6 \\ & & & & 0 & -k_6 & k_6 \end{bmatrix} \qquad \{F\} = \begin{pmatrix} -P_1 \\ 0 \\ P_3 \\ 0 \\ 0 \\ 0 \\ P_7 \end{pmatrix}$$