People's Democratic Republic of Algeria

Ministry of Higher Education and Scientific Research

University of Sciences and Technology Mohamed Boudiaf Oran

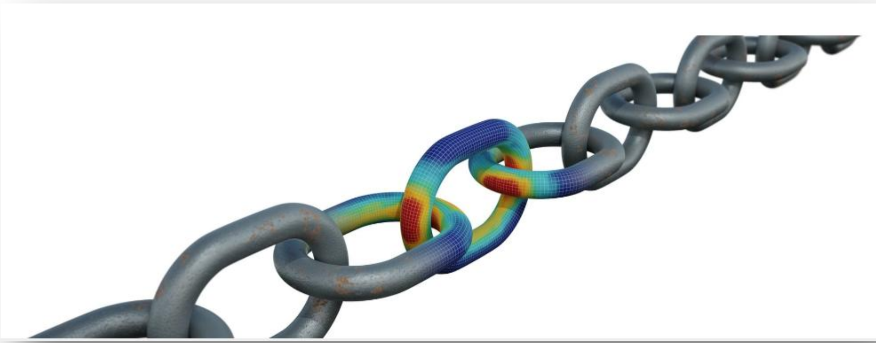
Faculty of Mechanical Engineering

Department of Naval Engineering

Specialty: Naval Engineering

Cours Handou of

Finite Element Method



Presented at the Naval Engineering department

Intended for Naval Engineering Bachelor's Degree Students

Dr. ADJAL YASSINE

Academic year 2024/2025

Course Overview

This course on the Finite Element Method (FEM) is primarily designed for third-year undergraduate students enrolled in the Naval Engineering program, specializing in shipbuilding and naval architecture under the LMD system. The handout aims to facilitate students' understanding the use of FEM to solve problems related to strength of materials filed . The course is organized into three comprehensive chapters: The first chapter offers a broad overview of FEM, covering its objectives, fundamental concepts, advantages, and areas of application. It also introduces the discretization approach used in finite element calculations, explaining how continuous domains are subdivided into finite elements for analysis. The second chapter focuses on deriving the stiffness matrix for a bar element. It extends this study to truss structures, illustrating how these elements combine to form complex frameworks. Several solved problems are included to reinforce the theoretical concepts. Additionally, three classical methods for calculating deflection and rotation are presented: The third chapter develops the application of FEM to beam bending problems. It begins with the construction of the stiffness matrix for a beam element and builds upon the theoretical foundation to analyze bending behavior. This chapter includes key questions, worked examples, and straightforward applications designed to deepen students' comprehension and practical skills. Throughout the document, emphasis is placed on linking theory with practice, enabling students to confidently apply FEM techniques to real-world naval engineering problems.

Keywords: Finite element method, bar element, beam , truss, stiffness matrix

Contents

Introduction

The finite element method FEM is an effective numerical method for solving engineering and physical problems. Its application ranges from stress analysis in aircraft or automobile structures to calculations of such complex systems as nuclear power plants. It is used to consider the movement of liquids through pipes, through dams, in porous media, to study the flow of compressed gas, to solve electrostatic and lubrication problems, and to analyze system vibrations. FEM is a numerical method for solving differential equations encountered in physics and engineering. The origin of this method is connected with the solution of space problems (1950). The scope of application of FEM significantly expanded when it was shown that the equations describing the elements in problems of structural mechanics, heat propagation and hydromechanics are similar. FEM from a numerical procedure for solving problems of structural mechanics turned into a general method for the numerical solution of differential equations.

The use of finite element analysis (FEA) techniques has grown drastically in the last decade. Several structural failures have demonstrated that, if not used properly, the FEA may mislead the designer with erroneous results. The programs have become so user friendly, that engineers with little previous design experience may use them and commit fundamental mistakes, which can result in inadequate strength in the structure.

Finite element analysis (FEA) is the most common structural analysis tool in use today. In marine industries, the use of this technique is becoming more widespread in the design, reliability analysis and performance evaluation of ship structures. Users of FEA have considerable freedom in designing the finite element model, exercising it and interpreting the results. Key components of this process include the selection of the computer program, the determination of the loads and boundary conditions, development of the engineering model, choice of elements and the design of the mesh.

Specific Objectives

- **Understand the Basic Principles of FEM:**
  Students will learn the foundational concepts behind the finite element analysis procedure, including the theory and characteristics of finite elements that represent engineering structures such as bars and beams

- **Develop Computational Skills:**
  The course aims to develop students' abilities to apply matrix algebra and numerical methods for describing and solving mechanical problems, including the formulation and assembly of stiffness matrices and the application of boundary conditions
- **Apply FEM to Structural Problems:**
  Learners will gain practical experience in modeling and analyzing determinate and indeterminate structural problems, including bars, trusses, and beams, both in one and two dimensions
- **Interpret and Evaluate Results:**
  Students will be trained to interpret the results of finite element analyses, understanding the relationship between external loads, displacements, and structural stiffness, and to use this understanding to inform engineering decisions[2].
- **Bridge Theory and Practice:**
  By combining theoretical knowledge with hands-on calculation examples, the course ensures that students can both comprehend and apply FEM to real-world structural engineering tasks, including the calculation of nodal displacements, reaction forces, and stresses

Learning Outcomes

By the end of the course, students should be able to:

- Explain the theory, fundamentals, and applications of FEM for structural engineering problems
- Discretize a structure into finite elements and describe the degrees of freedom for structural problems
- Formulate and assemble stiffness matrices for bar and beam elements, both in local and global coordinate systems
- Apply appropriate boundary conditions and solve the resulting system of equations for displacements and stresses

Chapter 1 An overview of the finite elemnt method

1. What is finite element method (FEM) ?

The finite element method (FEM) is a numerical technique used to model the behavior of physical objects under load conditions. FEM is used in many fields of science and technology, including construction, aviation, automotive, ship building and medicine. enbling to solve various mathematical problems such as differential equations, equations of motion, heat conduction equations, mechanical equations, and many others. The finite element method is particularly effective for problems for which an analytical solution is impossible or very difficult to obtain. FEM analysis requires determining the geometry, materials, and boundary conditions of a structure, then dividing the entire analysis domain into small finite elements, which are then analyzed separately. Each finite element consists of nodes for which the solution values are calculated. Solving this system of equations allows you to determine the displacements, deformations, and stresses throughout the structure.

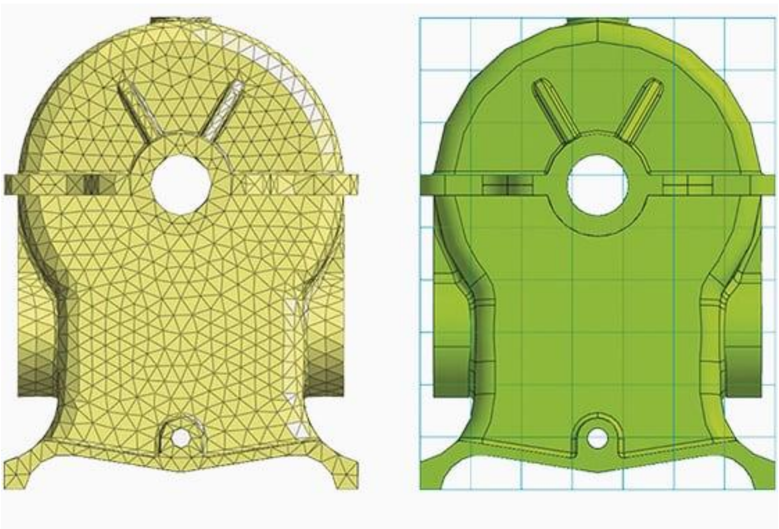
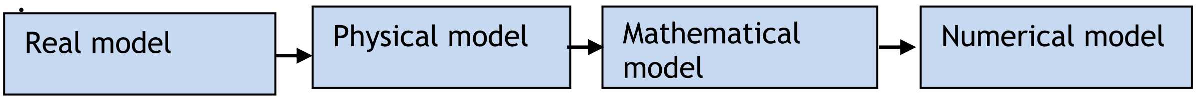


Figure 1.1 Modeling of structure using Finite element method

1.2 Numerical model sequence

When trying to predict the behavior of reality, we resort to a simplification of it, called a model. The model admits a gradation, in the sense of representing better or worse or of exposing one or another aspect of reality. For example, when analyzing a solid body mechanically, one can leave aside thermal, electrical and magnetic phenomena, assuming that they do not interfere in the analysis of the object. Abstracting from these, one can still consider, or not, the deformation of the solid body, leading respectively to a model of a deformable solid, or to a model of a rigid body. By using one or the other model, more or less phenomena are observed. In the case of choosing a rigid body model, it is not possible at all to observe the vibration that it actually undergoes, such is the level of simplification of this model. There is a chain of models until reaching the model of finite elements of a physical phenomenon, as shown in following figure . At the beginning of it, on the left, is the physical model of the phenomenon, which takes into account the geometry, the material constitution and the interaction of the body with the surrounding environment. A central part of this modeling stage is the identification of the physical laws involved in the phenomenon and the relevance of each of them for the intended analysis. This relevance is dictated by the degree of complexity



Sequence of models.

The FEM is applicable to a wide range of problems related to various physical phenomena subject to a wide variety of interactions with the surroundings in which they occur. Furthermore, the stability and accuracy of the method are well studied and solidly supported by mathematical theories, which gives it robustness. Hence its wide use as a tool for analysis in various fields of science and engineering, Fig1.2.

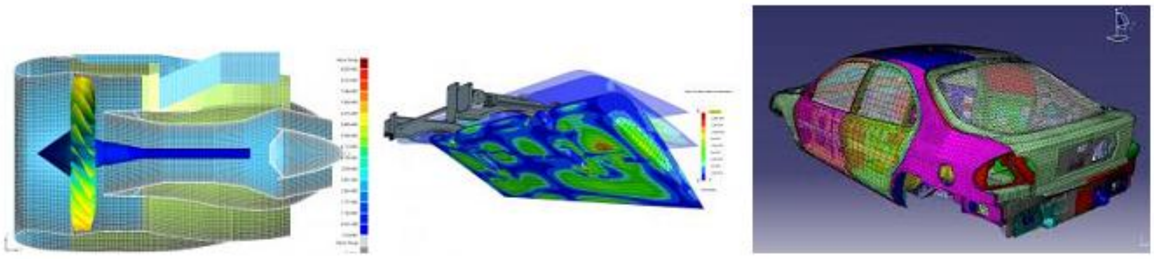


Figure 1.2. Analysis of a turbine , an airplane and an automobile (d) by FEM

1.3 Fundamental of the finite element method

In the professional activities of an engineer, there is often a need to solve practical tasks related to the mathematical modeling of physical objects, processes, or phenomena. Mathematical models for many applied problems are represented by differential equations, combined with a set of appropriate boundary and/or initial conditions derived from fundamental physical laws applicable to the system in question.

Typically, the exact solutions to these equations, which express the balance of mass, force, or energy, cannot be obtained analytically due to the complexity of the equations and the challenges posed by the boundary and initial conditions. This necessitates the use of various numerical methods that provide approximate solutions to the problem. Unlike analytical methods, which accurately describe the behavior of the system at any pont, numerical methods approximate the exact solution only at specific points, known as nodes of the calculation grid.

To simplify the calculation process, the discretization of space is performed using special algorithms. Existing numerical methods are generally divided into two main classes: the finite difference method and the finite element method. The finite difference method involves writing differential equations for each node and replacing derivatives with difference schemes, resulting in a system of simple algebraic equations. While this method is relatively easy to use for straightforward tasks, it becomes challenging for problems involving complex geometries, intricate boundary conditions, and anisotropic materials.

.

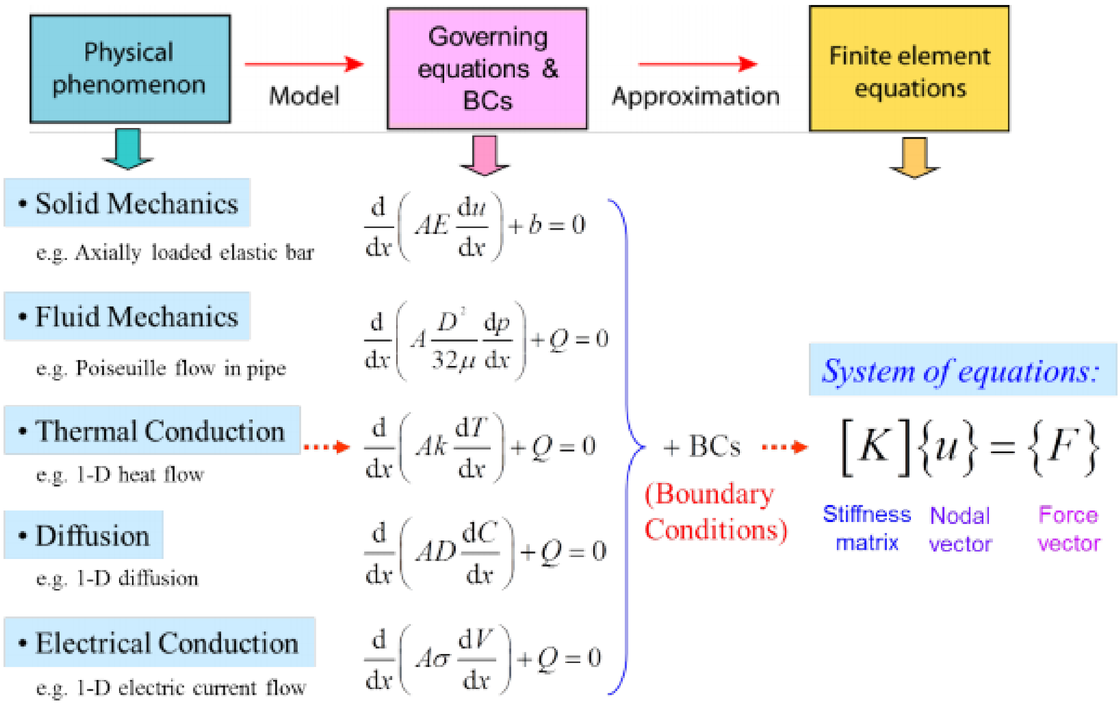


Figure 1.3 Application domain of the finite element method

1.4 Benefits of Using FEM Analysis in structural design

Ensuring Accuracy. FEM analysis provides highly accurate results, which is crucial in many fields such as mechanical, aerospace, and space engineering.

Saving Time and cost . Using FEM analysis allows for faster and more cost-effective study of object behavior by replacing costly and time-consuming experimental studies.

Improving Product Quality. Using FEM analysis, designers can test different design variants and select the most optimal one, leading to better product quality.

Ability to Simulate Different Conditions. FEM analysis allows for the simulation of various conditions, which is difficult, if not impossible, to achieve in experimental studies.

Design Optimization: Using FEM analysis, designers can optimize designs to meet technical and economic requirements.

Ability to analyze non-standard geometries. FEA analysis allows the analysis of non-standard geometries that are otherwise difficult to study.

Allows the analysis of multiple types of loads. FEM analysis can analyze different types of loads, including dynamic, static, thermal, and others.

Effective in solving problems for which an analytical solution is impossible or very difficult to obtain. This method allows the study of complex mathematical models that would be virtually impossible to solve analytically.

The ability to analyze different types of physical phenomena such as heat, electricity, motion, stress, and many others. This allows us to study many different aspects of an object's behavior, allowing us to obtain a more complete picture of its properties.

FEM analysis allows for the consideration of different types of boundary conditions, allowing for a more accurate representation of the real-world conditions under which a given object operates.

1.5 Disadvantages of using FEM analysis in structural design

Computational complexity and cost. FEM analysis is a complex and time-consuming computational process that requires significant computing resources. For complex models, this process can take several hours or days, resulting in high computational costs.

Risk of numerical errors. If the FEM analysis parameters are poorly defined or if the mathematical model is inappropriate, there is a risk of numerical errors that can lead to incorrect results.

Need for calibration and validation. To obtain accurate results, it is necessary to calibrate and validate the FEM model. Calibration involves comparing analysis results with actual data from experiments or simulations. Validation, in turn, involves comparing simulation results with data from other sources or methods. These steps require appropriate knowledge and experience.

Need for adequate model preparation. To obtain accurate results, it is necessary to properly prepare the model, including correctly arranging the finite element mesh and appropriately determining the model parameters. Improper model preparation may lead to incorrect analysis results.

1.6 Step of finite element method

- **Model preparation step**

The first step in the FEM analysis of a beam is to create its geometric model. A beam is described by its dimensions and material properties, such as Young's modulus or Poisson's ratio. At this stage, boundary conditions, such as the applied forces and moments that will act on the beam, are also determined.

- **Discretization step**

The next step is the discretization of the beam, i.e., the division into a finite number of elements. Through this stage, the geometric and material properties of the individual elements can be precisely determined, enabling an accurate FEM analysis. In the case of a beam, these elements will generally be rectangular or triangular, and their number depends on the accuracy of the analysis.

- **Computational Step**

After discretizing the beam, you can move on to the computational stage. This process involves solving equations describing the internal reactions of each element, based on input data such as boundary conditions and material properties. This stage provides information on the stress and strain distribution in the various beam elements, enabling accurate strength analysis.

- **Validation and Optimization Step**

The final stage is the validation and optimization of the FEM model. In this stage, the results obtained from the analysis are compared with the results of laboratory or experimental measurements to ensure the model's reliability. If errors are detected in the model, they must be removed and the analysis repeated. The next step is to optimize the beam shape to reduce stresses and production costs.

FEM analysis can determine exactly where the greatest stresses occur in the beam and at what stage of production modifications should be made to optimize the shape and reduce stresses. FEM analysis also allows for an accurate estimation of the beam's strength, which is crucial in structural design. Furthermore, FEM analysis can take into account various types of loads, such as external forces, moments, thermal stresses, and many others, allowing for an accurate determination of the structure's behavior under various conditions.

In the case of a beam, FEM analysis can also determine the maximum bending moment and maximum stresses, which is crucial for ensuring the structure's safety and preventing damage. Based on the analysis results, it is also possible to optimize the structure's materials and geometry, thereby reducing production costs and increasing its durability.

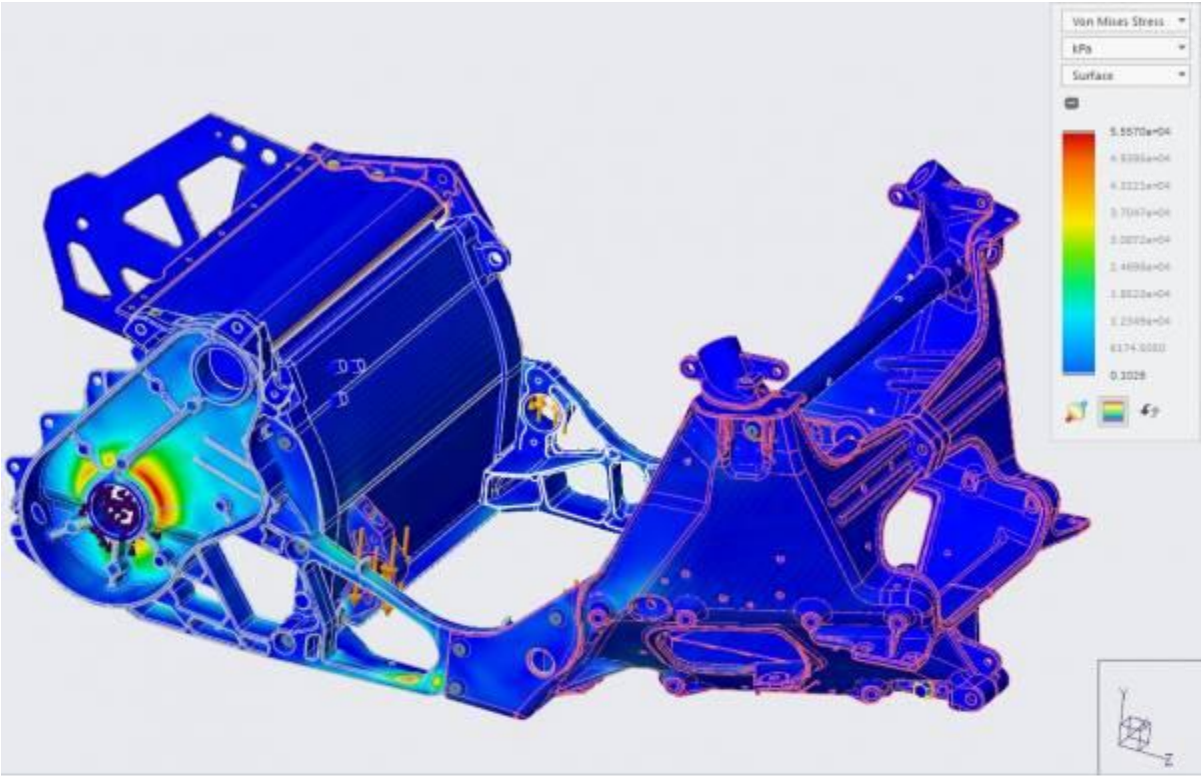


Figure 1.4 stress concentration acorss a critical area

1.7 Basics of the finite element method

The basic idea of FEM is that any continuous quantity, such as temperature, pressure, and displacement, can be approximated by a discrete model, which is built on a set of piecewise continuous functions. In the general case, a continuous quantity is known in advance, and it is necessary to determine the value of this quantity at some internal points of the region. A discrete model is very easy to construct if we first assume that the numerical values of this quantity in each internal region are known. After this, we can move on to the general case. Thus, when constructing a discrete model of a continuous quantity, we proceed as follows:

1.In the region under consideration, a finite number of points are fixed. These points are called nodal points or nodes.

2. The value of a continuous quantity at each point is considered a variable to be determined.

3. The domain of a continuous quantity is divided into a finite number of regions called elements. These elements have common nodal points and together approximate the shape of

the domain. A continuous quantity is approximated at each element by a polynomial that is defined using the nodal values of that quantity. A polynomial is defined for each element, but the polynomials are chosen in such a way that the continuity of the quantity is preserved along the boundaries of the element (it is called the element function). The choice of the element form and functions for specific problems depends on the ingenuity and skill of the engineer, and it is quite clear that this determines the accuracy of the approximate solution.

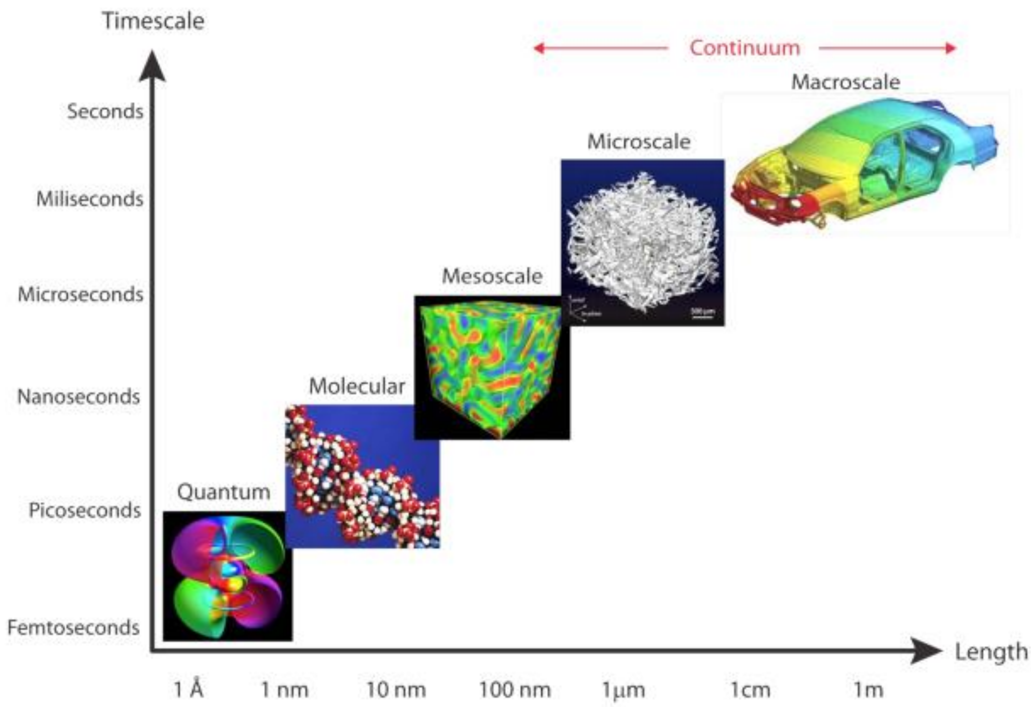


Figure 1.5 Scale level of structure

1.8 Constructing a finite element approximation (one-dimensional case)

Differential and variational equations generally cannot be solved analytically, except for a few simple cases. If the geometry of the domain in which the solution is sought is complex, then finding an analytical solution to the desired function $u(x)$ can be difficult. Often, the analytical solution is in the form of a series or an infinite sum, but even in this case it is difficult to satisfy all the boundary conditions of the problem. In the finite element method, instead of looking for an exact solution to a differential or variational equation, we look for an approximate solution. Our approximate solution $u(x)$ is the sum of several functions, which are called trial functions: As an example, consider a one-dimensional domain in the form of a straight line. When this domain is divided into finite elements, it becomes possible to approximate the solution using continuous piecewise linear polynomials. Within each element, the approximate solution is a linear function. Two adjacent elements have the same solution at a common node. Obviously, as the number of elements increases, our approximate piecewise linear solution will converge to the exact solution. It is also possible to obtain a more accurate approximate solution by increasing the order of the polynomial within each element (for example, using a quadratic function instead of a linear one). After the domain is divided into finite elements, the integration in the variational equation is performed within each element. For example, consider a domain on the interval (0,1), which we divide into 10 elements. Then the integration on the interval (0,1) can be reduced to the sum of 10 integrals on each of the 10 elements:

$$\int_0^1 \square dx = \int_0^{0.1} \square dx + \int_{0.1}^{0.2} \square dx + \cdots + \int_{0.9}^{1} \square dx.$$

After the domain is divided into elements, the solution on each of these elements is sought in the form of simple polynomials. As an example, consider a one-dimensional domain that is divided into n elements (Fig. 1.5).

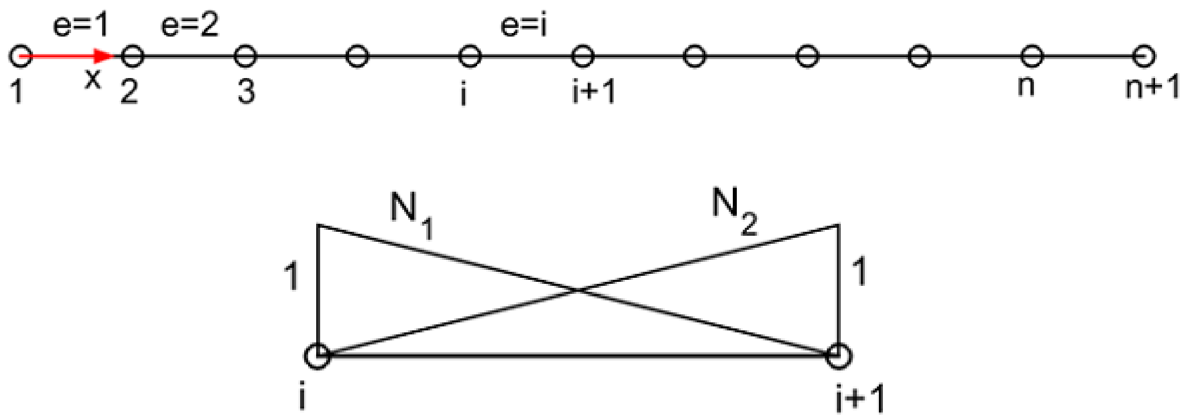


Figure 1.6 Detailed discretization of the state. Linear interpolation functions in each element

Let's number the nodes 1, 2, …, $n + 1$. Each element has 2 nodes at the ends. Let's say that the element with the number i has 2 nodes with the numbers i and $i + 1$. Let's denote the x coordinate for the node i as xi. In order to find the solution inside each element, we will use the solution value at two nodes of the element. If we want to interpolate the solution using only 2 nodes, then we can represent the solution as a linear function, since it has 2 unknown coefficients:

The unknown coefficients a_0 and a_1 can be expressed through the solution at the nodes of the element $u(x_i) = u_i$ and $u(x_i+1) = u_i+1$. We do not know these values yet, but they will be found in the process of solving the problem. Thus, we have 2 equations for finding the coefficients a_0 and a_1:

Expressing the coefficients $a0$ and $a1$ through ui and $ui+1$, we can obtain

Now the solution can be represented as

The functions $N_1(x)$ and $N_2(x)$ are called interpolation functions, or shape functions. Obviously, if $x = xi$, then $N_1(x) = 1$, and $N_2(x) = 0$. At the right node of the element $x = xi+1$, the conditions $N_1(x) = 0$, and $N_2(x) = 1$ are satisfied. Note that the approximate solution (1.3) is identical to the representation (1.1).

1.9 Finite element method in solving problems of strength of materials

The idea of FEM is that any continuous quantity can be approximated by a piecewise continuous function, which is constructed on the values of the quantity under study at a finite number of points of the elements under consideration. When constructing a discrete model of a continuous quantity, the following is done:

– the domain of definition of the quantity under study is divided into a finite number of elements that have common nodal points and, in aggregate, approximate the shape of the domain

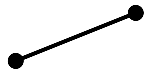
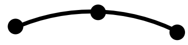
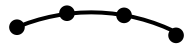
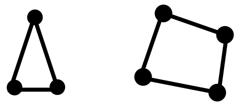
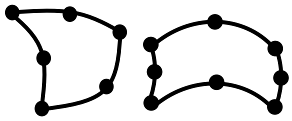
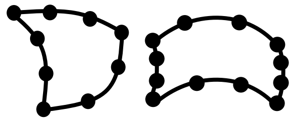
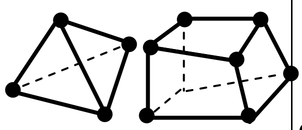
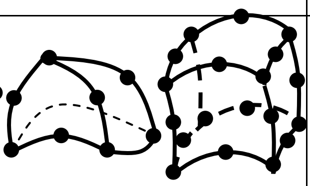
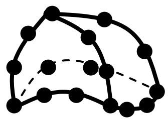
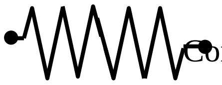
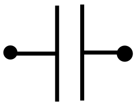
– nodes are fixed in the region under consideration;

– using the values of the continuous quantity under consideration at the nodal points and the approximating function, the values of the quantity inside the region are determined. Approximating functions are most often selected in the form of linear, quadratic or cubic polynomials. A polynomial associated with a given element is called an element function. From this point of view, a structure can be considered as a certain set of structural elements connected at a finite number of nodal points. If the relationships between forces and displacements for each element are known, then the properties can be described and the behavior of the structure as a whole can be investigated. Thus, when using FEM, a solution to a boundary value problem for a given region is sought in the form of a set of functions defined on finite elements.

1.10 Types of finite element

There are a large number of different types of finite elements (FE) .The Finite Element Method has developed a series of finite element types that can be initially classified as:

- one-dimensional finite elements (usually bars);
- two-dimensional finite elements (plates and the same volumes);
- three-dimensional finite elements (solid blocks).

Elements	linear	parabolic (quadratic)	cubic
one-dimensional			
two-dimensional			
three-dimensional			
other types	 Mass Spring	 Contact	

Chapter 2 Tension-Compression in a beam

2. Derivation of the Stiffness Matrix

The main characteristics of a finite element are expressed by its stiffness matrix. For a structural finite element, the stiffness matrix contains information about the geometry and properties of the material, which specifies the element's resistance to deformation under the influence of a load. Such deformation may include tension-compression, bending, shear.

2.1 Spring element

A linear elastic spring is a mechanical device designed to support only axial loads. Its extension or compression is directly proportional to the magnitude of the applied axial force, following Hooke's Law. The constant of proportionality between the applied load and the resulting deformation is known as the spring constant, spring stiffness, or simply k . This spring constant has the units of force per unit length (for example, newtons per meter), reflecting the ratio of force required to produce a unit displacement in the spring.

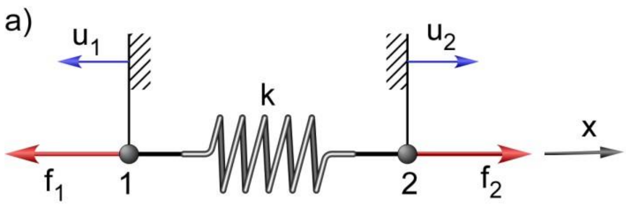


Figure 2.1 –a linear spring element

In the figure under consideration, these forces are designated by the vectors f_1 and f_2, also directed in opposite directions. Assuming that the displacements of both nodes are zero when the spring is not deformed, the resulting deformation of the spring can be expressed by the formula:

$$\delta = u_2 - u_1 \tag{2.1}$$

Then, the resulting axial force in the spring will be determined by the following expression:

$$F = k\delta = k\left(u_2 - u_1\right) \tag{2.2}$$

Therfore, the equilibrium of the spring will be achieved when the forces at its nodes are determined by the following equations:

$$F_1 = -k(u_2 - u_1) \tag{2.3}$$

$$F_2 = k(u_2 - u_1) \tag{2.4}$$

Equations(2.3)And(2.5 5)can be written in matrix form as:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \tag{2.5}$$

which in simplified form

$$[K_e]\{u\} = \{F\} \tag{2.6}$$

where $[K_e]$is the element stiffness matrix in the local coordinate system (element coordinate system);

$\{u\}$ - vector of nodal displacements;

$\{F\}$ – vector of nodal forces applied to the element.

Equation(2.5)shows that the stiffness matrix of a linear spring element is a 2×2 matrix, this is explained by the fact that the element is characterized by two nodal displacements (or two degrees of freedom), which are dependent, since the spring is continuous and elastic. In addition, the matrix is symmetrical. Solution of the equation (2.5) for given nodal loads is reduced to determining unknown nodal displacements, and can formally be written as:

$$\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [K_e]^{-1} \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \tag{2.7}$$

where $[K_e]^{-1}$is the inverse matrix of the spring element stiffness.

The described procedure for deriving the stiffness matrix of a spring element is based on determining the conditions of its equilibrium. The same procedure, carried out by using the equilibrium equation for each node, can also be used for systems of interconnected spring elements. However, instead of compiling calculation schemes for each node and deriving equilibrium equations on their basis, these equations can be obtained more efficiently by considering the contribution made by the force acting in each element to the equations for

of each node. This process is called "assembly" because it uses the combination of individual rigid components to derive a system of equations. The assembly of the characteristics of individual elements into a system of equations can be shown using the simplest example of a system of two linear springs connected as shown in the figure 2.2 .

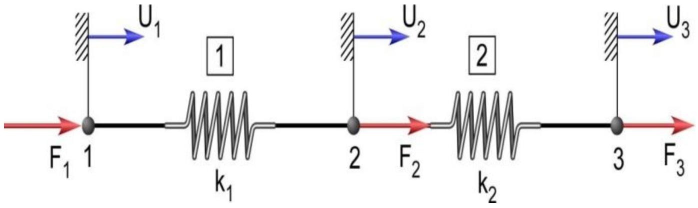


Figure 2.2 Loading diagram of a linear spring element

For generalization, assume two springs with different spring constants, k_1 and k_2, connected at nodes 1, 2, and 3. Node 2 is common to both springs and represents their physical connection point. The displacements of the nodes in the global coordinate system are denoted as U_1, U_2, and U_3, where uppercase letters indicate global (system-level) displacements, distinguishing them from local displacements of individual elements. Similarly, the applied nodal loads are represented as F_1, F_2, and F_3.

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} F_1^{(1)} \\ F_2^{(1)} \end{Bmatrix} \tag{2.8}$$

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = \begin{Bmatrix} F_2^{(2)} \\ F_3^{(2)} \end{Bmatrix} \tag{2.9}$$

where the superscript denotes the element number.

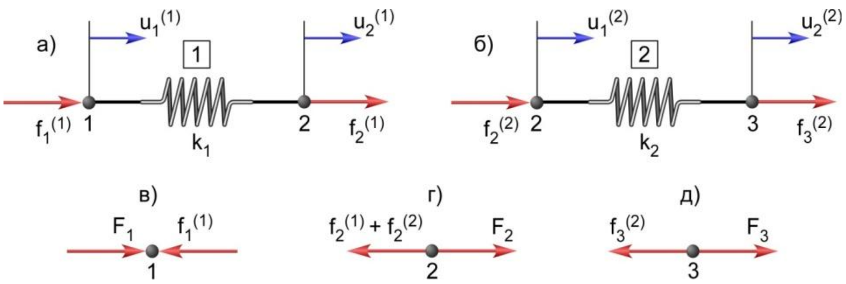


Figure 2.3 Loading diagram of a linear spring element

To begin assembling the equilibrium equations that describe the behavior of a system of two springs, it is necessary to formulate the conditions of compatibility of the displacements that relate the displacements of the elements to the displacements of the system. In this case, these conditions have the form:

$$u_1^{(1)} = U_1 \quad u_2^{(1)} = U_2 \quad u_1^{(2)} = U_2 \quad u_2^{(2)} = U_3 \tag{2.10}$$

The compatibility conditions reflect the physical fact that the springs, being connected at node 2, remain connected at this node after deformation, from which it follows that the displacements of both springs at this node are the same. Thus, continuity of displacements is ensured at the nodes when moving from element to element.

Substituting the conditions from (11) into equations (2.9) and (2.10) yields equations of the following form:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} F_1^{(1)} \\ F_2^{(1)} \end{Bmatrix} \tag{2.11}$$

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = \begin{Bmatrix} F_2^{(2)} \\ F_3^{(2)} \end{Bmatrix} \tag{2.12}$$

This notation clearly indicates that the elements are physically connected at node 2. To proceed, the matrices involved must first be expanded, resulting in equations of the following form:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1^{(1)} \\ F_2^{(1)} \\ 0 \end{Bmatrix} \tag{2.13}$$

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2^{(2)} \\ F_3^{(2)} \end{Bmatrix} \tag{2.14}$$

Addition of equations (2.13) And (2.15) leads to the equation:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1^{(1)} \\ F_2^{(1)} + F_2^{(2)} \\ F_3^{(2)} \end{Bmatrix} \tag{2.15}$$

Equilibrium conditions are of the following form:

$$F_1^{(1)} = F_1 \quad F_2^{(1)} + F_2^{(1)} = F_2 \quad F_3^{(2)} = F_3 \tag{2.16}$$

Substituting these conditions into the equation (2.15) leads to the final equation describing the behavior of the spring system:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \tag{2.17}$$

The first matrix in this equation is the system stiffness matrix **[K]**, which:

- Is linear, as is typical for all linear systems analyzed within an orthogonal coordinate system;
- Is singular (degenerate) due to the lack of constraints preventing rigid body motion within the system;
- Represents a straightforward superposition of the stiffness matrices of the individual elements.

2.2 Bar element

2.2.1 Displacement function

To facilitate understanding of the general relationships, let us examine the characteristics of a finite element using the example of a stepped rod subjected to tension and compression (see Fig. 2.4).

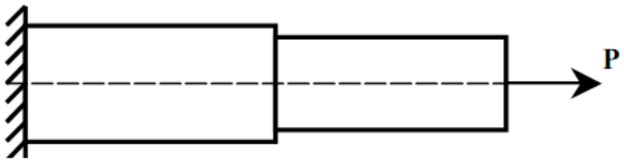


Figure 2.4 - Stepped beam

In this case, a bar is taken as the final element

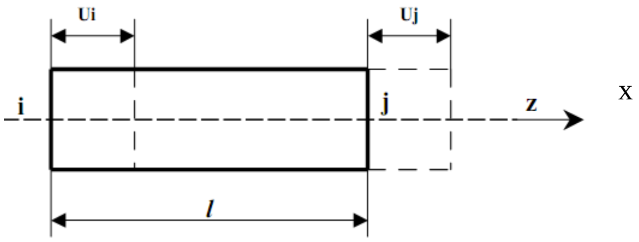


Figure 2.5 Bar element

The displacement function $\{N\}$ for this case has the form

$$\{N\} = \left|\frac{l-x}{l}, \frac{x}{l}\right| \cdot \begin{Bmatrix} U_i \\ U_j \end{Bmatrix} \tag{2.18}$$

where U_i and U_j are the displacements of nodes i and j ;

$\frac{l-x}{l} = N_1$ and $\frac{x}{l} = N_2$ shape functions; ranging from 1 to 0. The displacement function $\{\delta\}$ depends on the shape of the finite element.

2.2.2 Deformation function

The deformation function, or deformation vector, is expressed in terms of the displacement function. When the bar is stretched, its relative elongation is described by this relationship.

$$\{\varepsilon\} = \left\{\frac{\partial N}{\partial x}\right\} = \frac{1}{l}\left|-1 \quad 1\right| \cdot \begin{Bmatrix} U_i \\ U_j \end{Bmatrix} \tag{2.19}$$

The expression $\frac{1}{l}\left|-1; \quad 1\right|$ is denoted by the matrix $|B|$, then

$$\{\varepsilon\} = |B|\{\delta\}^e \,, \tag{2.20}$$

$\{\delta\}^e = \begin{Bmatrix} U_i \\ U_j \end{Bmatrix}$ represents the vector of displacements of elemen's nodes.

2.2.3 Stress function

the stress vector is expressed through the strain vector

$$\{\sigma\} = |D|\{\varepsilon\} = |D||B|\{\delta\}^e \,, \tag{2.21}$$

where $|D|$ is the elasticity matrix (connects stresses and deformations).

2.2.4 System of FEM equations for the enitre structure

To determine the work of external force (virtual) movements $d\{\delta\}^e$. The work of internal forces per unit volume on virtual displacement is equal to $d\{\varepsilon\}^T\{\sigma\}$

where $\{\varepsilon\}^T$ is the deformation vector transposed to the vector $\{\varepsilon\}$

Work of internal forces on the entire finite element

$$\int_V d\{\varepsilon\}^T\{\sigma\}dV = \int_V d\left(\{\delta\}^e\right)^T |B|^T .|D||B|dV.$$

The work of external nodal forces $\{F\}^e$ on virtual displacements of the element is equal to

$$d\left(\{\delta\}^e\right)^T\{F\}^e$$

By equating the work of external and internal forces on possible movements of the element, we obtain

$$d\left(\{\delta\}^e\right)^T \int_V |B|^T|D||B|dV\{\delta\}^e = \left(d\{\delta\}^e\right)^T\{F\}^e \,,$$

reducing $d\left(\{\delta\}^e\right)^T$, we get

$$\int_V |B|^T|D||B|dV\{\delta\}^e = \{F\}^e$$

or

$$|K|^e\{\delta\}^e = \{F\}^e \,, \tag{2.22}$$

where

$$|K|^e = \int_V |B|^T |D||B| dV \qquad (2.23)$$

The stiffness matrix of the finite element. For the entire structure, the finite element method equation can be expressed as:

$$|K|\{\delta\} = \{R\}, \qquad (2.24)$$

$|K|$ the stiffness matrix of the structure as a whole, it is the sum of the stiffness matrices of the finite elements that make up the structure;

$\{\delta\}$ displacement vector of all nodes;

$\{R\}$ vector of nodal loads.

Any FEM problem is ultimately reduced to the system of equations (2.24). Its order is equal to the product of the number of nodes and the number of degrees of freedom of the node. Solving the FEM problem, we see that we obtain a system of algebraic equations instead of differential equations In conclusion, it should be noted that when obtaining the stiffness matrix of the finite element, the initial displacements of the nodes, the effect of temperature and the initial stresses are not taken into account.

$$|K|^e = \int_V |B|^T |D||B| dV,$$

2.2. 5 Variable distributed loading

Consider a bar element of constant cross-section, of length L, subjected to tension-compression. The element subjeted a linearly distributed load q:

$$q(x) = \left[N_1(x) \;\; N_2(x) \right] \begin{Bmatrix} q_i \\ q_j \end{Bmatrix} \tag{2.25}$$

$$q(x) = \left[\left(1 - \frac{x}{L}\right) q_i + \left(\frac{x}{L}\right) q_j \right] \tag{2.26}$$

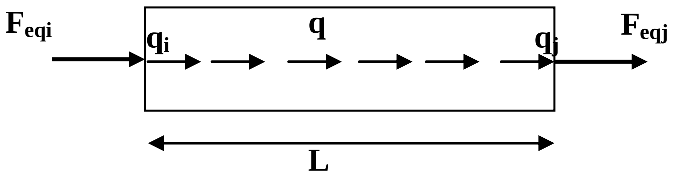


Figure 2.6 Bar uner distributed load

Concentrated force are applied only at the nodes of the tension bar elements. Regarding the presence of distributed loads, the approach used is to replace the distributed load with nodal forces and moments so that the mechanical work performed by the nodal load system will be equivalent. The mechanical work performed by distributed load can be expressed by the following equation:

$$W = \int_0^L q(x) u(x) dx \tag{2.27}$$

The objective is to determine the equivalent nodal loads so that the work expressed in the previous equation is identical to:

$$W = \int_0^L q(x) u(x) dx = F_{eqi} u_i + F_{eqj} u_j \tag{2.28}$$

or F_{eqi} and F_{eqj} are the equivalent forces at nodes i and j, respectively. Substituting the discretized displacement function, the work integral becomes.

$$W = \int_0^L q(x)\left[\left(1-\frac{x}{L}\right)u_i + \left(\frac{x}{L}\right)u_j\right]dx \tag{2.29}$$

$$W = \int_0^L q(x)\left[\left(1-\frac{x}{L}\right)u_i dx + \int_0^L q(x)\left(\frac{x}{L}\right)u_j\right]dx \tag{2.30}$$

Comparing the equations and we obtain

$$F_{eqi} = \int_0^L q(x)\left(1-\frac{x}{L}\right)dx \tag{2.31}$$

$$F_{eqj} = \int_0^L q(x)\left(\frac{x}{L}\right)dx \tag{2.32}$$

For example for a constant uniform load $q(x) = q$, the integration of these equations is as follows

$$F_{eqi} = \int_0^L q\left(1-\frac{x}{L}\right)dx = \frac{qL}{2} \tag{2.33}$$

$$F_{eqj} = \int_0^L q\left(\frac{x}{L}\right)dx = \frac{qL}{2} \tag{2.34}$$

the equivalent forces is written $\begin{Bmatrix} F_{eqi} \\ F_{eqj} \end{Bmatrix} = \begin{Bmatrix} \frac{qL}{2} \\ \frac{qL}{2} \end{Bmatrix}$

Example 1

Determine the dispalcements, internal forces and stress in each element (Figure 2.6)

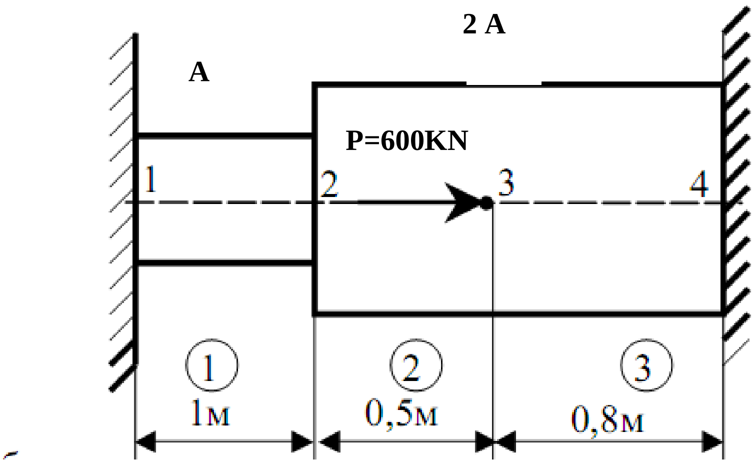


Figure 2.7 - Stepped beam

$$|K|_I^e = \frac{EA}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = EA\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix};$$

$$|K|_{II}^e = \frac{EA}{0.5}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = EA\begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix};$$

$$|K|_{III}^e = \frac{EA}{0.8}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = EA\begin{bmatrix} 2.5 & -2.5 \\ -2.5 & 2.5 \end{bmatrix}.$$

Now, we assemble the global stiffness matrix of the structure. Since there are four nodes, each with one degree of freedom, the stiffness matrix has dimensions of 4×4. The coefficients corresponding to the same position in the matrix are summed algebraically. The right-hand column contains the values of the external loads applied at the respective nodes. In this case, a force is applied to the third node to the right.

$$|K|_g = EA\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 1+4 & -4 & 0 \\ 0 & -4 & 4+2.5 & -2.5 \\ 0 & 0 & -2.5 & 2.5 \end{bmatrix}\begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 600 \\ 0 \end{Bmatrix}.$$

We take into account the boundary conditions: since the bar is clamped at the ends, then $U_1 = U_4 = 0$. Based on this, we delete the first and fourth columns and rows. After this, we obtain

$$EA\begin{bmatrix} 5 & -4 \\ -4 & 6.5 \end{bmatrix}\begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix}\begin{Bmatrix} 0 \\ 600 \end{Bmatrix},$$

Solving these equation's system, we obtain

$$U_3 = \frac{182}{EA}, \quad U_2 = \frac{145.6}{EA}.$$

We determine the values of longitudinal forces (axial forces):

$$\{N\}_I = \frac{EA}{l_1}|-1;1|\begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \frac{EA}{1}|-1;1|\begin{Bmatrix} 0 \\ \dfrac{145.6}{EA} \end{Bmatrix} = 145.6KN$$

$$\{N\}_{II} = \frac{E2A}{0.5}|-1;1|\begin{Bmatrix} \dfrac{145.6}{EA} \\ \dfrac{182}{EA} \end{Bmatrix} = 145.6KN$$

$$\{N\}_{II} = \frac{E2A}{0.8}|-1;1|\begin{Bmatrix} \dfrac{182}{EA} \\ 0 \end{Bmatrix} = -455KN$$

Dividing the longitudinal force by the area on each section, we obtain the stress value:

taken A= 20 cm^2.

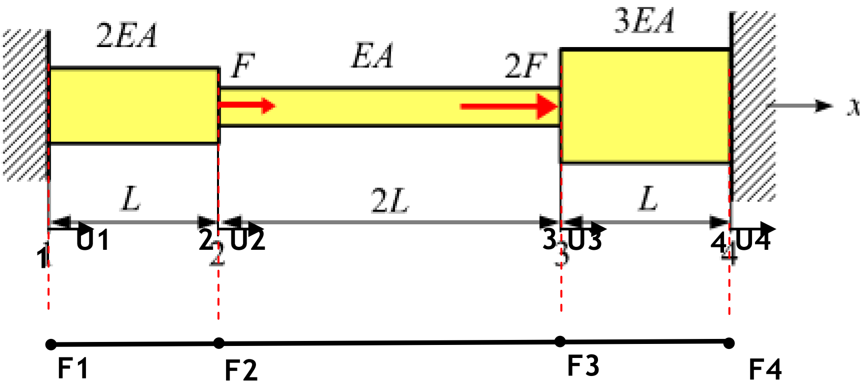
$$\sigma_I = \frac{N_I}{A} = \frac{145.6\ 10^{-3}}{20\ 10^{-4}} = 72.8MPa,$$

$$\sigma_I = \frac{N_{II}}{A} = \frac{145.6\ 10^{-3}}{2\ 20\ 10^{-4}} = 36.4MPa.$$

$$\sigma_I = \frac{N_I}{A} = \frac{-455\ 10^{-3}}{2\ 20\ 10^{-4}} = 113.7MPa,$$

Example 2

Determine the displacements and verify the equilibrium condition.



$U_{1.4}$ nodal displacements

$F_{1.4}$ nodal forces

Table represents the geometric and physical characteristics of the elements

Element	Nodes		E_eA_e	Length
	i	I		
1	1	2	2EA	L
2	2	3	EA	2L
3	3	4	2EA	L

1-The elementary stiffness matrices are:

Element 1

$$\begin{matrix} & & |U_1 & U_2| & \\ [K_{1-2}] = & \dfrac{2EA}{L} & \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & & \begin{vmatrix} U_1 \\ U_2 \end{vmatrix} \end{matrix}$$

Element 2

$$\begin{array}{c} \quad |U_2 \quad U_3| \\ [K_{2-3}] = \dfrac{EA}{2L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{vmatrix} U_2 \\ U_3 \end{vmatrix} \end{array}$$

Element 3

$$\begin{array}{c} \quad |U_3 \quad U_4| \\ [K_{3-4}] = \dfrac{2EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{vmatrix} U_3 \\ U_4 \end{vmatrix} \end{array}$$

2- Global stiffness matrix (Assembly)

$$\begin{array}{c} \quad |U_1 \quad \mathrm{U}_2 \quad U_3 \quad U_4| \\ [K_g] = \dfrac{EA}{L}\begin{bmatrix} 2 & -2 & 0 & 0 \\ -2 & 2+\dfrac{1}{2} & -\dfrac{1}{2} & 0 \\ 0 & -1/2 & \dfrac{1}{2}+3 & -3 \\ 0 & 0 & -3 & 3 \end{bmatrix}\begin{vmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{vmatrix} \end{array}$$

3- Application of boundary conditions

$$[K_g] \qquad \{u_i\} \quad = \quad \{F_i\}$$

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 & 0 \\ -2 & 2+\dfrac{1}{2} & -\dfrac{1}{2} & 0 \\ 0 & -\dfrac{1}{2} & \dfrac{1}{2}+3 & -3 \\ 0 & 0 & -3 & 3 \end{bmatrix}\begin{Bmatrix} U_1 = 0 \\ U_2 = ? \\ U_3 = ? \\ U_4 = 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 = F \\ F_3 = 2F \\ F_4 \end{Bmatrix}$$

4- Reduced matrix after elimination (Boundary conditions)

The unknown displacements are the solutions to the following equation

$$[K_{réduite}] = \frac{EA}{L}\begin{bmatrix} \dfrac{5}{2} & -\dfrac{1}{2} \\ -\dfrac{1}{2} & \dfrac{7}{2} \end{bmatrix}\begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F \\ 2F \end{Bmatrix}$$

from where

$$U_2 = \frac{9FL}{17EA} \qquad \text{And}\, U_3 = \frac{11FL}{17EA}$$

The reaction forces are determined

$$\frac{EA}{L}\begin{bmatrix} -2 & 0 \\ 0 & -3 \end{bmatrix}\begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{x4} \end{Bmatrix}$$

where

$$F_{x1} = -\frac{18}{17}F \ , \quad F_{x4} = -\frac{33}{17}F$$

The Equilibrium condition is verified

$$\mathrm{F}_{x1} + F_{x2} + F_{x3} + F_{x4} = -\frac{18}{17}F + F + 2F + -\frac{33}{17}F = 0$$

Example 3

Determine the equivalent nodal loads and displacements.

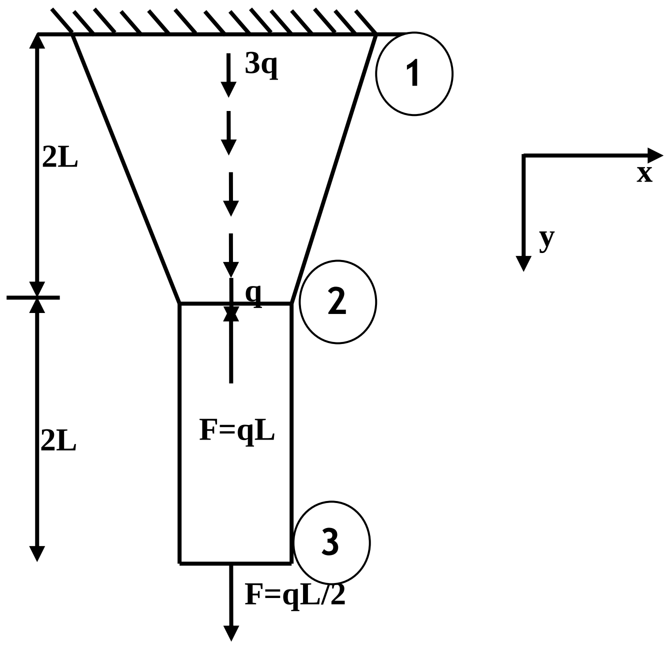


Figure 2.8 Variable cross-sectional structure subjected to a variable distributed load

Solution

$$q(y) = ay + b$$

$$y = 2L \Rightarrow q(2L) = a.2L + 3q = q \Rightarrow a = -\frac{q}{L}$$

$$q(y) = ay + b$$

$$y = 0 \Rightarrow q(0) = b = 3q$$

$$y = 2L \Rightarrow q(2L) = a.2L + 3q = q \Rightarrow a = -\frac{q}{L}$$

$$q(y) = q\left(3 - \frac{y}{L}\right)$$

Equation of the distributed varialbe load

$$A(y) = ay + b$$

$$y(0) = b = 2A$$

$$y(2L) \Rightarrow 2La + 2A = \Rightarrow a = -\frac{A}{2L}$$

$$A(y) = A\left(2 - \frac{y}{2L}\right)$$

by modeling bars 1-2 and 2-3 of the structure as simple elements Element 1-2

$$\{F\} = \{F_{eq}\} = [K]\{\delta\}$$

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} + \begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \int_0^{2L} [B]^T E [B] dV = \int_0^{2L} \begin{bmatrix} \frac{-1}{2L} \\ \frac{1}{2L} \end{bmatrix} E \begin{bmatrix} \frac{-1}{2L} & \frac{1}{2L} \end{bmatrix} y dy$$

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} + \begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \frac{3EA}{4L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} v_1 \\ v_2 \end{Bmatrix}$$

The load q(x) varies linearly, so

$$\begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \frac{L}{6} \begin{bmatrix} 2q_1 + q_2 \\ q_1 + 2q_2 \end{bmatrix}$$

$$\begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \frac{L}{6} \begin{bmatrix} 2q_1 + q_2 \\ q_1 + 2q_2 \end{bmatrix}$$

$$\begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \frac{L}{3} \begin{bmatrix} 6q + q \\ 3q + 2q \end{bmatrix} = \begin{Bmatrix} \frac{7qL}{3} \\ \frac{5qL}{3} \end{Bmatrix}$$

$$\begin{Bmatrix} F_{eq1} \\ F_{eq2} \end{Bmatrix} = \frac{L}{3}\begin{bmatrix} 6q+q \\ 3q+2q \end{bmatrix} = \begin{Bmatrix} \frac{7qL}{3} \\ \frac{5qL}{3} \end{Bmatrix}$$

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} + \begin{Bmatrix} \frac{7qL}{3} \\ \frac{5qL}{3} \end{Bmatrix} = \frac{3EA}{4L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} v_1 \\ v_2 \end{Bmatrix}$$

Element 2-3

$$\begin{Bmatrix} F_2 \\ F_3 \end{Bmatrix} + = \frac{EA}{2L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} v_2 \\ v_3 \end{Bmatrix}$$

Assembly

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} + \begin{Bmatrix} 7qL/3 \\ 5qL/3 \\ 0 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 3/4 & -3/4 & 0 \\ -3/4 & 3/4 & -1/2 \\ 0 & -1/2 & 1/2 \end{bmatrix}\begin{Bmatrix} v_1 \\ v_2 \\ v_3 \end{Bmatrix}$$

Boundary conditions

$$\begin{Bmatrix} 0 \\ -qL \\ qL/2 \end{Bmatrix} + \begin{Bmatrix} 7qL/3 \\ 5qL/3 \\ 0 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 3/4 & -3/4 & 0 \\ -3/4 & 3/4 & -1/2 \\ 0 & -1/2 & 1/2 \end{bmatrix}\begin{Bmatrix} v_1 = 0 \\ v_2 \\ v_3 \end{Bmatrix}$$

$$v_2 = \frac{14}{9}\frac{qL^2}{EA}$$

$$v_3 = \frac{23}{9}\frac{qL^2}{EA}$$

2.3. Modeling of Truss structure using bar element

2.3.1 Transformation of coordinates from the local system to the global system

Figure 2.9 Photo of a Bridg structure

In a truss or lattice structure, Fig.2.8 , the parts can assume different directions. Since the stiffness matrix and the displacement and load vectors are defined for a coordinate system oriented locally in relation to the element, it is necessary to represent them in a common coordinate system, called global, so that there is physical coherence when assembling the stiffness matrix and the global load vectors. This coherence concerns the components of the displacement and nodal force vectors of the mesh elements, which can only be correctly added if they are all referred to the same coordinate system, the global coordinate system. The requirement of a common coordinate system leads to representations of local matrices in this common system, obtained through transformations, shown below.

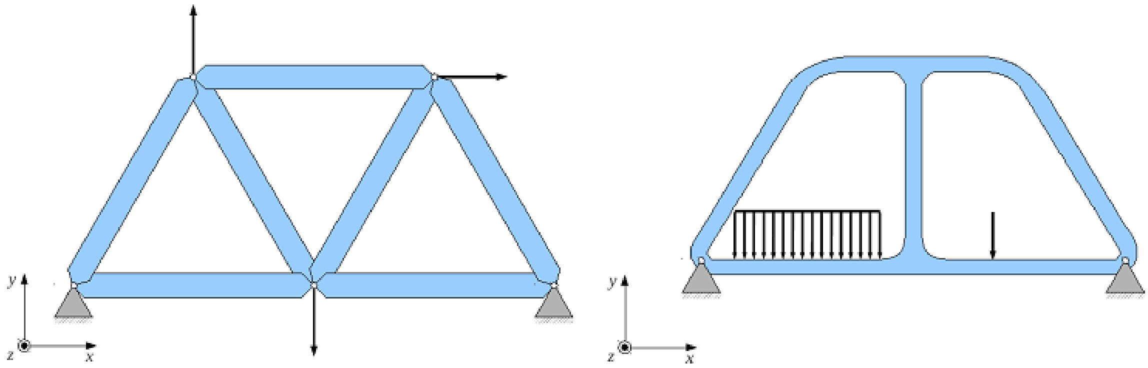


Figure 2.10   Truss structure

2.3.1.1 Formulation of stiffness matrix

The displacements in the local xOy system can be expressed as a function of the global displacements. . The displacements in the local system are u_1 and u_2 and in the global system U_1, W_1 and U_2, W_2. The angle (θ) is considered the angle between the X axis and the positive direction of the articulated bar 1-2.

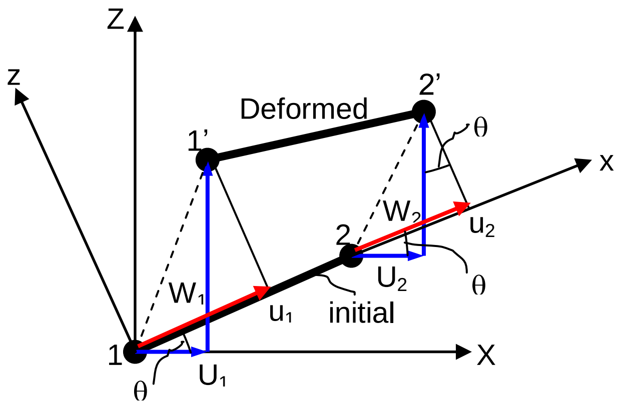


Figure 2.11 Local and global coordinate systems

$$
\begin{aligned}
u_1 &= U_1 \cos\theta + W_1 \sin\theta \\
u_2 &= U_2 \cos\theta + W_2 \sin\theta
\end{aligned}
\tag{2.35}
$$

or in matrix form:

$$
\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \cdot \begin{Bmatrix} U_1 \\ W_1 \\ U_2 \\ W_2 \end{Bmatrix} \Rightarrow \{u\} = [T] \cdot [U] \tag{2.36}
$$

or [T] is called the transformation matrix of the local system into the global one and:

$$
\cos\theta = \frac{X_2 - X_1}{L} \;,\; \sin\theta = \frac{Z_2 - Z_1}{L} \;\; et \;\; L = \sqrt{(X_2 - X_1)^2 + (Z_2 - Z_1)^2} \,,
$$

L represents the length of the articulated bar element.

2.3.1.2 Transformation of forces from the local system into the global system

We consider the articulated bar of Figure 3.2 subjected in the local system by the forces f1 and f2 applied in nodes 1 and 2 in the local coordinate axis system. The components of these forces in the global system will be:

$$F_{x1} = f_1 \cos\theta \;\; ; \;\; F_{z1} = f_2 \cos\theta \qquad (2.37)$$
$$F_{x2} = f_1 \sin\theta \;\; ; \;\; F_{z2} = f_2 \sin\theta$$

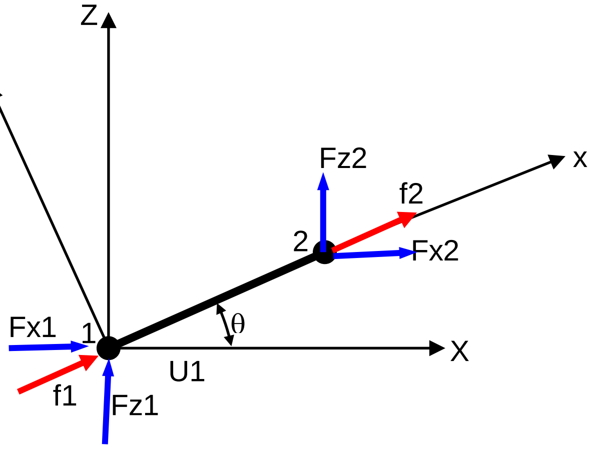


Figure 2.12 Nodal forces in the local and global coordinate axis system

In matrix form, relations 3.3 can be written as:

$$\begin{Bmatrix} F_{x1} \\ F_{z1} \\ F_{x2} \\ F_{z2} \end{Bmatrix}_e = \begin{bmatrix} \cos\theta & 0 \\ \sin\theta & 0 \\ 0 & \cos\theta \\ 0 & \sin\theta \end{bmatrix} \cdot \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}_e \qquad (2.38)$$

or

$$\{F^e\} = [T^e]^T \cdot \{f^e\} \qquad (2.39)$$

for the global system of axes, we have :

$$\{F^e\} = [T^e]^T \cdot \{f^e\} = [T^e]^T \cdot [k^e] \cdot \{u^e\} = \underbrace{[T^e]^T \cdot [k^e] \cdot [T]}_{[K^e]} \cdot \{U^e\} = [K^e] \cdot \{U^e\} \qquad (2.40)$$

or

- {Fe} = vector of nodal forces in the global system of axes;
- {fe} = vector of nodal forces in the local system of axes;
- [Ke] = stiffness matrix in the global axis system;
- [ke] = stiffness matrix in the local axis system;
- {Ue} = vector of nodal displacements in the global system of axes;
- {ue} = vector of nodal displacements in the local system of axes.

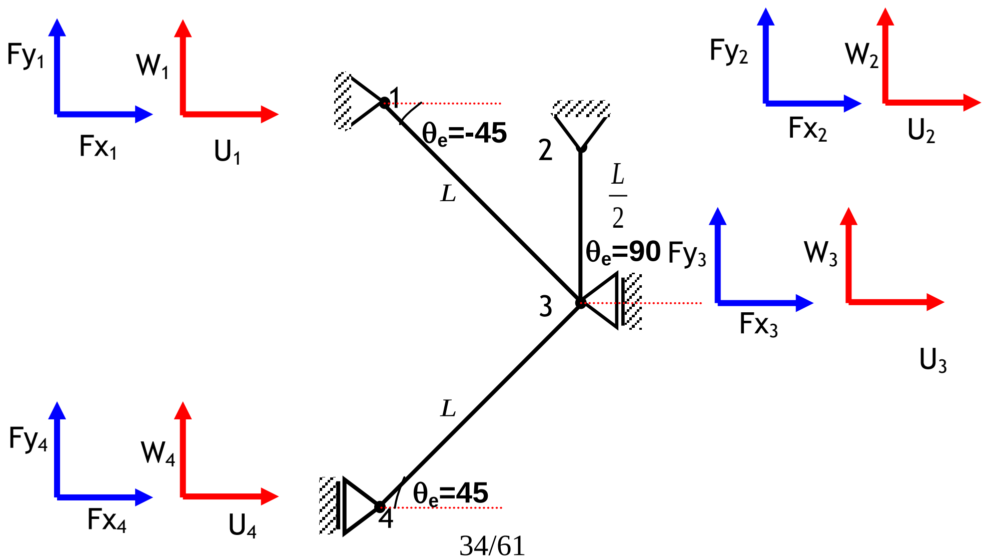
$$\left[K^e\right]=\begin{bmatrix}\cos\theta & 0\\ \sin\theta & 0\\ 0 & \cos\theta\\ 0 & \sin\theta\end{bmatrix}\cdot\frac{\mathrm{EA}}{\mathrm{L}}\cdot\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\cdot\begin{bmatrix}\cos\theta & \sin\theta & 0 & 0\\ 0 & 0 & \cos\theta & \sin\theta\end{bmatrix}=\frac{\mathrm{EA}}{\mathrm{L}}\cdot\begin{bmatrix}c^2 & cs & -c^2 & -cs\\ cs & s^2 & -cs & -s^2\\ -c^2 & -cs & c^2 & cs\\ -cs & -s^2 & cs & s^2\end{bmatrix} \tag{2.41}$$

where c = cosθ   and s = sinθ.

NB. The stiffness matrix in the global system of axes is symmetric, singular and, as can be easily noticed, the elements of the main diagonal are positive.

Example 1

A plane truss structure consists of three truss elements connected to four nodes, as shown to the right. All trusses have cross-sectional area A and elastic modulus E. The length of each truss element is evident by the figure. A point force, P, is acting on node4. Calculate the displacements at the nodes and the reaction forces at nodes 1 and 2, respectively. Show also that global equilibrium is satisfied in the vertical direction.



Element	Nodes		θ_e [°]	$\cos\theta_e$	$\sin\theta_e$	c^2	s^2	cs
	i	j						
1	1	3	-45	$\frac{\sqrt{2}}{2}$	$-\frac{\sqrt{2}}{2}$	$\frac{1}{2}$	$\frac{1}{2}$	$-\frac{1}{2}$
2	3	2	90	0	1	0	1	0
3	4	3	45	$\frac{\sqrt{2}}{2}$	$\frac{\sqrt{2}}{2}$	$\frac{1}{2}$	$\frac{1}{2}$	$\frac{1}{2}$

For element 1 nodes (1,3) ,elements length (L)

$$\begin{array}{c} \quad U_1 \quad W_1 \quad U_3 \quad W_3 \end{array}$$

$$\left[K^1\right] = \frac{EA}{L} \cdot \begin{bmatrix} \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{vmatrix} U_1 \\ W_1 \\ U_3 \\ W_3 \end{vmatrix}$$

For element 2 nodes (3,2) ,elements length (L/2)

$$\begin{vmatrix} U_3 & W_3 & U_2 & W_2 \end{vmatrix}$$

$$\left[K^2\right] = \frac{2EA}{L} \cdot \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} = \frac{EA}{L} \cdot \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 2 & 0 & -2 \\ 0 & 0 & 0 & 0 \\ 0 & -2 & 0 & 2 \end{bmatrix} \begin{vmatrix} U_3 \\ W_3 \\ U_2 \\ W_2 \end{vmatrix}$$

For element 3 nodes (4,3) ,elements length (L)

$$\begin{vmatrix} U_4 & W_4 & U_3 & W_3 \end{vmatrix}$$

$$\left[K^3\right] = \frac{EA}{L} \cdot \begin{bmatrix} \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} \\ \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} \\ -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} \\ -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{vmatrix} U_4 \\ W_4 \\ U_3 \\ W_3 \end{vmatrix}$$

Now we write the global stiffnes matrix

$$
\begin{array}{cccccccc} U_1 & W_1 & U_2 & W_2 & U_3 & W_3 & U_4 & W_4 \end{array}
$$

$$
[K]=\frac{EA}{L}\begin{bmatrix}
\frac{1}{2} & -\frac{1}{2} & 0 & 0 & -\frac{1}{2} & \frac{1}{2} & 0 & 0 \\
-\frac{1}{2} & \frac{1}{2} & 0 & 0 & \frac{1}{2} & -\frac{1}{2} & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 2 & 0 & -2 & 0 & 0 \\
\frac{1}{2} & \frac{1}{2} & 0 & 0 & \frac{1}{2}+\frac{1}{2}+0 & \frac{1}{2}-\frac{1}{2}+0 & -\frac{1}{2} & -\frac{1}{2} \\
\frac{1}{2} & -\frac{1}{2} & 0 & -2 & \frac{1}{2}-\frac{1}{2}+0 & \frac{1}{2}+\frac{1}{2}+2 & -\frac{1}{2} & -\frac{1}{2} \\
0 & 0 & 0 & 0 & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} \\
0 & 0 & 0 & 0 & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2}
\end{bmatrix}
\begin{vmatrix} U_1=0 \\ W_1=0 \\ U_2=0 \\ W_2=0 \\ U_3=0 \\ W_3\neq 0 \\ U_4=0 \\ W_4\neq 0 \end{vmatrix}
$$

Application of Boundary Conditions

$$
[K]=\frac{EA}{L}\begin{bmatrix}
\frac{1}{2} & -\frac{1}{2} & 0 & 0 & -\frac{1}{2} & \frac{1}{2} & 0 & 0 \\
-\frac{1}{2} & \frac{1}{2} & 0 & 0 & \frac{1}{2} & -\frac{1}{2} & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 2 & 0 & -2 & 0 & 0 \\
\frac{1}{2} & \frac{1}{2} & 0 & 0 & \frac{1}{2}+\frac{1}{2}+0 & \frac{1}{2}-\frac{1}{2}+0 & -\frac{1}{2} & -\frac{1}{2} \\
\frac{1}{2} & -\frac{1}{2} & 0 & -2 & \frac{1}{2}-\frac{1}{2}+0 & \frac{1}{2}+\frac{1}{2}+2 & -\frac{1}{2} & -\frac{1}{2} \\
0 & 0 & 0 & 0 & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} \\
0 & 0 & 0 & 0 & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2}
\end{bmatrix}
$$

Reduced Matrix after elimanting

$$\frac{EA}{L}\begin{bmatrix} 3 & -\frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} \end{bmatrix}\begin{Bmatrix} W_3 \\ W_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -p \end{Bmatrix}$$

$$W_3 = -\frac{2PL}{5EA} \text{ and } W_4 = -\frac{12PL}{5EA}$$

Reaction forces and the global equilibrium verfication

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{x4} \end{Bmatrix} = \frac{EA}{L}\cdot\begin{bmatrix} \frac{1}{2} & 0 \\ -\frac{1}{2} & 0 \\ 0 & 0 \\ -2 & 0 \\ 0 & -\frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} \end{bmatrix}\cdot\begin{Bmatrix} W_3 \\ W_4 \end{Bmatrix} = \begin{Bmatrix} -\frac{P}{5} \\ \frac{P}{5} \\ 0 \\ \frac{4P}{5} \\ \frac{6P}{5} \\ P \end{Bmatrix}.$$

Global equilibrium of the vertical direction

$$F_{y1} + F_{y2} - P = \frac{P}{5} + \frac{4P}{5} - p = p - p = 0 \quad \text{Ok!}$$

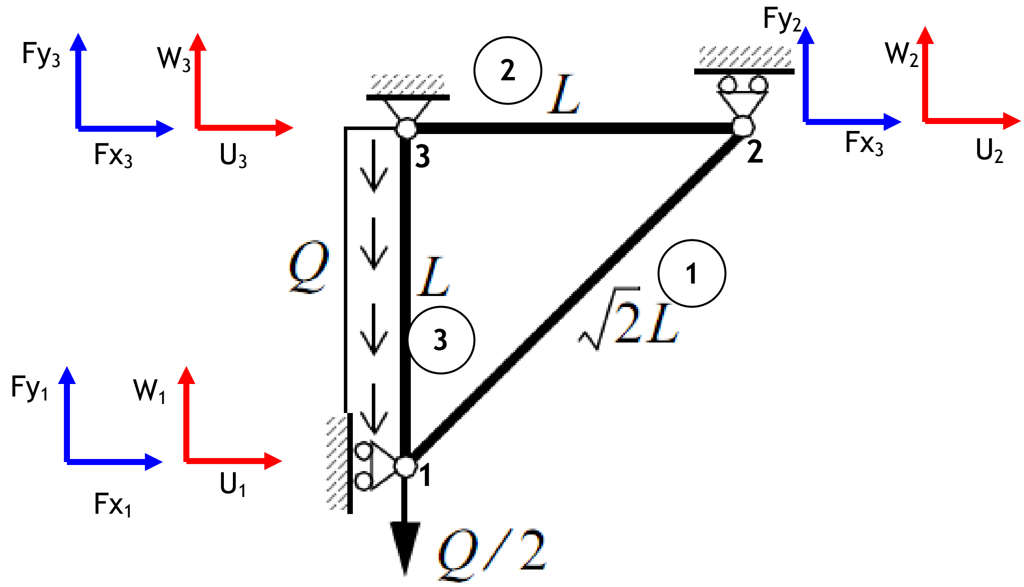
Global equilibrium of the horizontal direction

$$F_{x1} + F_{x2} + F_{x3} + F_{x4} = -\frac{P}{5} + 0 + \frac{6P}{5} - p = 0 \quad \text{Ok!}$$

Example 2

A truss structure is consisted of three bar elements , having an elastic modulus E and a transversal cross-sectional area (**A)**. The structure is loaded by a concentrated force Q/2, and with distributed load Q acting at the vertical element.

- Write the stiffness matrix of each elements .
- Write the global stiffness matrix
- Apply the boundary condition and determine the displacements. Show also that global equilibrium is satisfied in the vertical direction**.**



Element	Nodes		θ_e [°]	cos θ_e	sin θ_e	c^2	s^2	cs
	i	j						
1	1	2	45	$\frac{\sqrt{2}}{2}$	$-\frac{\sqrt{2}}{2}$	$\frac{1}{2}$	$\frac{1}{2}$	$-\frac{1}{2}$
2	2	3	0	1	0	1	0	0
3	1	3	90	0	1	0	1	0

<u>For element 1</u> nodes (1,2) ,elements length ($\sqrt{2}L$)

$$\begin{matrix} |U_1 & W_1 & U_3 & W_3| \end{matrix}$$

$$\left[K^2\right] = \frac{EA}{L}\cdot\begin{bmatrix} \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\ \frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \end{bmatrix}\begin{vmatrix} U_1 \\ W_1 \\ U_3 \\ W_3 \end{vmatrix}$$

<u>For element 2</u> nodes (2,3) ,element's length (L)

$$\begin{matrix} |U_2 & W_2 & U_3 & W_3| \end{matrix}$$

$$\left[K^2\right]=\frac{EA}{L}\cdot\begin{bmatrix}1 & 0 & -1 & 0\\ 0 & 0 & 0 & 0\\ -1 & 0 & 1 & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{vmatrix}U_2\\ W_2\\ U_3\\ W_3\end{vmatrix}$$

<u>For element 3</u> nodes (1,3) ,element's length (L)

$$\begin{vmatrix}U_1 & W_1 & U_3 & W_3\end{vmatrix}$$

$$\left[K^2\right]=\frac{EA}{L}\cdot\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 1 & 0 & -1\\ 0 & 0 & 0 & 0\\ 0 & -1 & 0 & 1\end{bmatrix}=\begin{vmatrix}U_1\\ W_1\\ U_3\\ W_3\end{vmatrix}$$

Now we write the global stiffnes matrix

$$\begin{vmatrix}U_1 & W_1 & U_2 & W_2 & U_3 & W_3\end{vmatrix}$$

$$[K]=\frac{EA}{L}\begin{bmatrix}\frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0\\ \frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+1 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & -1\\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}+1 & \frac{1}{2\sqrt{2}}+0 & -1 & 0\\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+0 & 0 & 0\\ 0 & 0 & -1 & 0 & 1+0 & 0+0\\ 0 & -1 & 0 & 0 & 0+0 & 0+1\end{bmatrix}\begin{vmatrix}U_1=0\\ W_1\neq 0\\ U_2\neq 0\\ W_2=0\\ U_3=0\\ W_3=0\end{vmatrix}$$

Application of Boundary Conditions

$$[K]=\frac{EA}{L}\begin{array}{c} \begin{matrix} U_1 & W_1 & U_2 & W_2 & U_3 & W_3 \end{matrix} \\ \begin{bmatrix} \frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\ \frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+1 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & -1 \\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}+1 & \frac{1}{2\sqrt{2}}+0 & -1 & 0 \\ -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}+0 & \frac{1}{2\sqrt{2}}+0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 1+0 & 0+0 \\ 0 & -1 & 0 & 0 & 0+0 & 0+1 \end{bmatrix} \end{array} \begin{vmatrix} U_1=0 \\ W_1\neq 0 \\ U_2\neq 0 \\ W_2=0 \\ U_3=0 \\ W_3=0 \end{vmatrix}$$

Reduced Matrix after elimanting

$$[K_r]=\begin{bmatrix} \frac{1}{2\sqrt{2}}+1 & -\frac{1}{2\sqrt{2}} \\ -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}+1 \end{bmatrix}\begin{Bmatrix} W_1 \\ U_2 \end{Bmatrix}=\begin{Bmatrix} \overline{Q} \\ 0 \end{Bmatrix}$$

$$W_1=\frac{1}{2}\frac{\left(-3+\sqrt{2}\right)\overline{Q}L}{EA}$$

$$U_2=-\frac{1}{2}\frac{\left(\sqrt{2}-1\right)\overline{Q}L}{EA}$$

Where

$$\overline{Q}=Q/2(1+L)$$

Nodal Displacements

$$W_1=\frac{1}{2}\frac{\left(-3+\sqrt{2}\right)\overline{Q}L}{EA}$$

$$U_2=-\frac{1}{2}\frac{\left(\sqrt{2}-1\right)\overline{Q}L}{EA}$$

Where

$$\overline{Q}=Q/2(1+L)$$

Example 3

In the system illustrated below, all the bars, of identical lengths L, are hinged at both ends. The bar cross-sections are denoted A and the modulus of elasticity is denoted E. A vertical load P is applied at point B.

- Write the stiffness matrix of elements AB, BC, and CD?
- Write the overall stiffness matrix?
- Solve the system to obtain the displacements at nodes B and C?

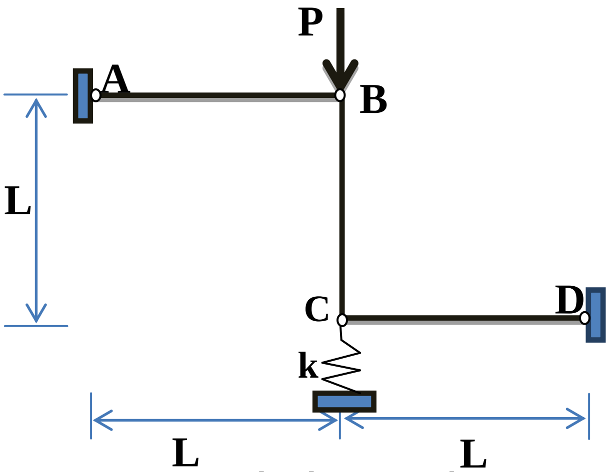


Figure 2.13 Articulated system with spring

Solution

1. Element stiffness matrices

Bar AB and CD $\theta = 0^0$

$$[K_e] = \frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Barre BC $\theta = 90^0$

$$[K_e] = \frac{EA}{L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

Assembly of the structural stiffness matrix

the assembled global stiffness matrix must take into account the stiffness of the spring in the vertical direction in C according to the degree of freedom v_C

$$[K_e] = \frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 & & & & \\ 0 & 0 & 0 & 0 & & & & \\ -1 & 0 & 1+0 & 0+0 & 0 & 0 & & \\ & & 0+0 & 0+1 & 0 & -1 & & \\ & & 0 & 0 & 0+1 & 0+0 & -1 & 0 \\ & & 0 & -1 & 0+0 & 1+0+\dfrac{kL}{EA} & 0 & 0 \\ & & & & -1 & 0 & 1 & 0 \\ & & & & 0 & 0 & 0 & 0 \end{bmatrix}$$

After introducing boundary conditions $(u_A = v_A = u_D = v_D = 0)$, the equations corresponding to these degrees of freedom are deleted and the system is thus reduced to

$$[K_e] = \frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 & & & & \\ 0 & 0 & 0 & 0 & & & & \\ -1 & 0 & 1+0 & 0+0 & 0 & 0 & & \\ & & 0+0 & 0+1 & 0 & -1 & & \\ & & 0 & 0 & 0+1 & 0+0 & -1 & 0 \\ & & 0 & -1 & 0+0 & 1+0+kL/EA & 0 & 0 \\ & & & & -1 & 0 & 1 & 0 \\ & & & & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_A \\ v_A \\ u_B \\ v_B \\ u_C \\ v_C \\ u_D \\ v_D \end{bmatrix}$$

Reduced matrix after elimination

$$[K_e] = \frac{EA}{L}\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 1 & 0 \\ 0 & -1 & 0 & 1+kL/EA \end{bmatrix}\begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix} = \begin{Bmatrix} 0 \\ -P \\ 0 \\ 0 \end{Bmatrix}$$

Solving the system of equations

$$\begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix} = \frac{PL}{EA}\begin{Bmatrix} 0 \\ -1-EA/kL \\ 0 \\ -EA/kL \end{Bmatrix}$$

Example 4

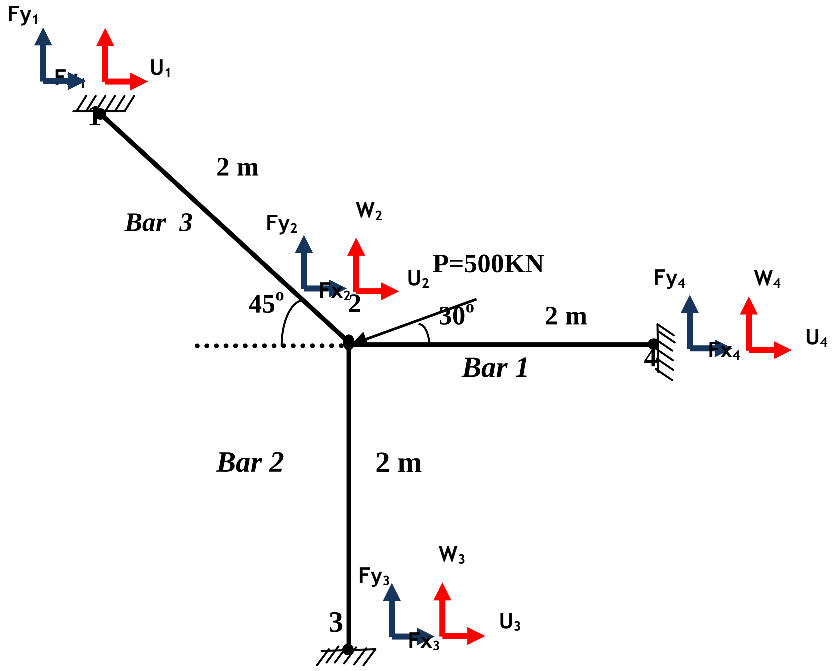


Table represents the geometric characteristics of the elements

Elément	Nœuds		θ_e [°]	cos θ_e	sin θ_e	c^2	s^2	cs
	i	j						
1	2	4	0	1	0	1	0	0
2	2	3	-90	1	0	1	0	0
3	2	1	135	$\frac{\sqrt{2}}{2}$	$-\frac{\sqrt{2}}{2}$	$\frac{1}{2}$	$\frac{1}{2}$	$-\frac{1}{2}$

Element 1

$$\begin{vmatrix} U_2 & W_2 & U_4 & W_4 \end{vmatrix}$$

$$\left[K^1\right] = \frac{EA}{L} \cdot \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{vmatrix} U_2 \\ W_2 \\ U_4 \\ W_4 \end{vmatrix}$$

Element 2

$$\begin{vmatrix} U_2 & W_2 & U_3 & W_3 \end{vmatrix}$$

$$\left[K^2\right] = \frac{EA}{L} \cdot \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{vmatrix} U_2 \\ W_2 \\ U_3 \\ W_3 \end{vmatrix}$$

Element 3

$$\left[K^3\right] = \frac{EA}{L} \cdot \begin{bmatrix} \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{vmatrix} U_2 \\ W_2 \\ U_1 \\ W_1 \end{vmatrix}$$

Global stiffness matrix

$$\begin{vmatrix} U_1 & W_1 & U_2 & W_2 & U_3 & W_3 & U_4 & W_4 \end{vmatrix}$$

$$[K] = \frac{EA}{L} \begin{bmatrix} 1/2 & -1/2 & -1/2 & 1/2 & 0 & 0 & 0 & 0 \\ -1/2 & 1/2 & 1/2 & -1/2 & 0 & 0 & 0 & 0 \\ -1/2 & 1/2 & 0+1+1/2 & 0+0-1/2 & 0 & 0 & -1 & 0 \\ 1/2 & -1/2 & 0-1/2 & 0+1+1/2 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 \\ 0 & 0 & -1 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{vmatrix} U_1 \\ W_1 \\ U_2 \\ W_2 \\ U_3 \\ W_3 \\ U_4 \\ W_4 \end{vmatrix}$$

Application of Boundary conditions

$$\begin{vmatrix} U_1 & W_1 & U_2 & W_2 & U_3 & W_3 & U_4 & W_4 \end{vmatrix}$$

$$[K]=\frac{EA}{L}\begin{bmatrix} 1/2 & -1/2 & -1/2 & 1/2 & 0 & 0 & 0 & 0 \\ -1/2 & 1/2 & 1/2 & -1/2 & 0 & 0 & 0 & 0 \\ -1/2 & 1/2 & 0+1+1/2 & 0+0-1/2 & 0 & 0 & -1 & 0 \\ 1/2 & -1/2 & 0-1/2 & 0+1+1/2 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 \\ 0 & 0 & -1 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{vmatrix} U_1=0 \\ W_1=0 \\ U_2\neq 0 \\ W_2 \neq 0 \\ U_3=0 \\ W_3=0 \\ U_4=0 \\ W_4=0 \end{vmatrix}$$

Reduced Matrice

$$\frac{EA}{L}\begin{bmatrix} \frac{3}{2} & -\frac{1}{2} \\ -\frac{1}{2} & \frac{3}{2} \end{bmatrix}\begin{Bmatrix} U_2 \\ W_2 \end{Bmatrix}=\begin{Bmatrix} F_{x2} \\ F_{y2} \end{Bmatrix} \qquad F_{x2}=-P\cos(30^o)\ ,\quad F_{y2}=-P\sin(30^o)$$

where

$$U_2=\frac{L}{4EA}\left(3F_{x2}+F_{y2}\right) \text{ et } W_2=\frac{L}{4EA}\left(3F_{x2}+3F_{y2}\right)$$

$$P=500KN$$

$$L=2m$$

$$U_2=-\frac{775}{EA} \text{ et } W_2=\frac{591.5}{EA}$$

Reaction forces and equiibrume verification

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{Bmatrix}=\frac{EA}{L}\cdot\begin{bmatrix} \frac{1}{2} & \frac{1}{2} \\ \frac{1}{2} & \frac{-1}{2} \\ 0 & 0 \\ 0 & -1 \\ -1 & 0 \\ 0 & 0 \end{bmatrix}\cdot\begin{Bmatrix} U_2 \\ W_2 \end{Bmatrix}=\begin{Bmatrix} \frac{1}{4}\left(F_{y2}-F_{x2}\right) \\ \frac{1}{4}\left(F_{x2}-F_{y2}\right) \\ 0 \\ \frac{1}{4}\left(-F_{x2}-3F_{y2}\right) \\ \frac{1}{4}\left(-3F_{x2}-F_{y2}\right) \\ 0 \end{Bmatrix}.$$

Vertical direction

$$\frac{1}{4}\left(F_{y2}-F_{x2}\right)+F_{x2}+\frac{1}{4}\left(-F_{x2}-3F_{y2}\right)=F_{x2}-F_{x2}=0 \quad \text{Ok!}$$

Horizontal direction

$$\frac{1}{4}\left(F_{x2}-F_{y2}\right)+F_{y2}+\frac{1}{4}\left(-F_{x2}-3F_{y2}\right)=F_{y2}-F_{y2}=0 \quad \text{Ok!}$$

where

$$F_{x1}=45.75KN$$
$$F_{y1}=-45.75KN$$
$$F_{x3}=0$$
$$F_{y3}=295.75KN$$
$$F_{x4}=387.25KN$$
$$F_{y4}=0$$

Chaptre 3 Bending of a beam

3. Formulation of beam finite element

The limitation of using bar elements arises from their inability to transmit bending effects, which restricts their application in modeling of structures with fixed connections, such as welded, riveted, or screwed joints—common features in real-world constructions. Consequently, another type of finite element is utilized for these applications: the beam finite element. This linear (one-dimensional) element is adept at handling both tension-compression and bending with torsion, making it more suitable for complex structural analyses. The characteristics of this element's formulation can be effectively illustrated in the example presented in Figure 3.1

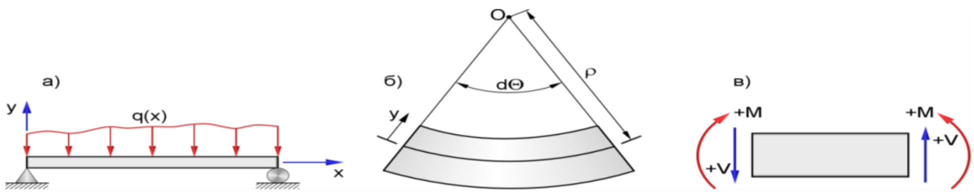


Figure 3.1 – Beam Finite Element: a) Simple beam on two supports, loaded with a distributed load; b) Curved beam element; c) Sign convention for shear forces and bending moments.

The formulation of the beam element is based on the elementary theory of beams according to the following assumptions :

- the beam is loaded only in the direction of the y-axis (in a two-dimensional formulation of the problem);
- beam deformations are small compared to its characteristic dimensions;
- the beam material is linearly elastic, isotropic and homogeneous;
- the beam has a prismatic cross-section, the axis of symmetry of which is located in the plane of beam bending.
- the element has a finite length and is defined by two nodes, one at each end;
- the connection of an element with other elements is carried out only at its nodes;
- -the element is loaded only at its nodes.

3.1 Derivation of the Element Stiffness Matrix

3.1.1. Direct Approach

To model structural elements subjected to bending loads, a beam element is used (Figure 3.2).

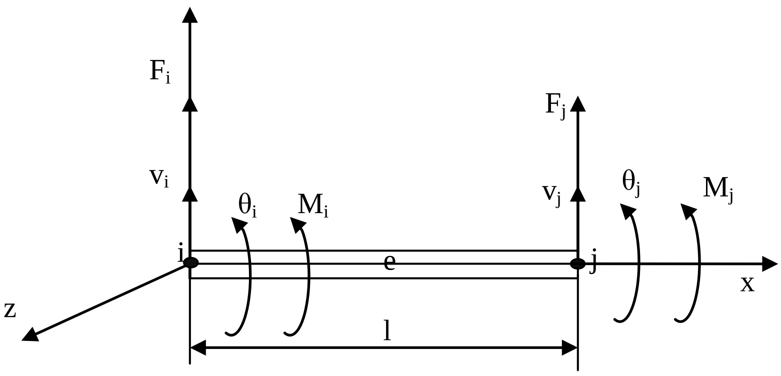


Figure 3.2 Beam element configuration

The main characteristics of a beam element are its length (l), the moment of inertia of the cross-section (I), and the modulus of elasticity of the material (E). A linear beam element (Figure 3.1) is defined by two nodes, (i) and (j), each possessing two degrees of freedom: deflection (v) and the angle of rotation of the section (θ) relative to the z-axis. Due to the small deformations, the following relationship is used :

$$\theta = \frac{dv}{dx} \tag{3.1}$$

The differential equation of the curved axis of the beam is

$$EI\,\frac{d^2v(x)}{dx^2} = M(x). \tag{3.2}$$

which can be written as

$$EI\,\frac{d^2v}{dx^2} = F_i x - M(x). \tag{3.3}$$

Integrating twice with respect to x, we obtain

$$EIv(x) = F_i \frac{x^3}{6} - M_i \frac{x^2}{2} + C_1 x + C_2. \quad (3.4)$$

where C1, C2 are integration constants determined from the boundary conditions

$$v(0) = v_i; \quad (3.5)$$

$$\theta(0) = \theta_j$$

Thus

$$\begin{cases} C_3 = -EI\theta_j; \\ C_4 = EIv_j \end{cases} \quad (3.6)$$

hence

$$EIv(x_i) = F_j \frac{x_i^3}{6} + M_j \frac{x_i^2}{2} - EI\theta_j x_i + EIv_j; \quad (3.7)$$

$$EI\theta(x_i) = F_j \frac{x_i^2}{2} + M_j x_i - EI\theta_j. \quad (3.8)$$

On the other hand, equation (3.2) can be written as

$$EI \frac{d^2 v}{dx_1^2} = F_j x_i + M_j, \quad (3.8)$$

where the new coordinate is entered

$$x_1 = l - x$$

Integrating equation (3.8) twice with respect to x, we obtain

$$EIv(x_1) = F_j \frac{x_i^3}{6} + M_j \frac{x_i^2}{2} + C_3 x_1 + C_4, \quad (3.9)$$

$$\begin{cases} v(0) = v_j; \\ \theta(0) = -\theta_j. \end{cases} \quad (3.10)$$

thus

$$\begin{cases} C_3 = -EI\theta_j; \\ C_4 = -EIv_j. \end{cases} \quad (3.11)$$

hence

$$EIv(x_i) = F_j \frac{x_i^3}{6} + M_j \frac{x_i^2}{2} + EI\theta_j x_i + EIv_j, \tag{3.12}$$

$$EI\theta(x_i) = F_j \frac{x_i^2}{2} + M_j x_i^2 - EI\theta_j. \tag{3.13}$$

v_j and θ_j can be determined from equations (3.7), (3.8):

$$EIv_j = F_i \frac{l^3}{6} - M_i \frac{l^2}{2} + EI\theta_i l + EIv_i; \tag{3.14}$$

$$EI\theta_j = F_i \frac{l^2}{2} - M_i l + EI\theta_i. \tag{3.15}$$

v_i and θ_i can be determined from equations (3.12), (3.13):

$$EIv_j = F_j \frac{l^3}{6} + M_j \frac{l^2}{2} - EI\theta_j l + EIv_j; \tag{3.16}$$

$$-EI\theta_i = F_j \frac{l^2}{2} + M_j l - EI\theta_j. \tag{3.17}$$

Solving the system of four equations (3.14) – (3.17) for Fi, Mi, Fj, Mj, we obtain

$$\begin{cases} F_i = \frac{12EI}{l^3} v_i + \frac{6EI}{l^2}\theta_i - \frac{12EI}{l^3} v_j + \frac{6EI}{l^2}\theta_j; \\ M_i = \frac{6EI}{l^2} v_i + \frac{4EI}{l}\theta_i - \frac{6EI}{l^2} v_j + \frac{2EI}{l}\theta_j; \\ F_i = -\frac{12EI}{l^3} v_i - \frac{6EI}{l^2}\theta_i + \frac{12EI}{l^3} v_j - \frac{6EI}{l^2}\theta_j; \\ M_i = \frac{6EI}{l^2} v_i + \frac{2EI}{l}\theta_i - \frac{6EI}{l^2} v_j + \frac{4EI}{l}\theta_j, \end{cases} \tag{3.18}$$

or in matrix form

$$Ku = F, \tag{3.19}$$

where and is the column vector of nodal displacements and rotation angles

$$u = \begin{Bmatrix} u_i \\ \theta_i \\ u_j \\ \theta_j \end{Bmatrix} \tag{3.20}$$

F – column vector of forces and bending moments

$$F = \begin{Bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{Bmatrix} \tag{3.21}$$

K – stiffness matrix of the beam element

$$[K] = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} = \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{l^2} & \frac{4EI}{l} & -\frac{6EI}{l^2} & \frac{2EI}{l} \\ -\frac{12EI}{l^3} & -\frac{6EI}{l^2} & \frac{12EI}{l^3} & -\frac{6EI}{l^2} \\ \frac{6EI}{l^2} & \frac{2EI}{l} & -\frac{6EI}{l^2} & \frac{4EI}{l} \end{bmatrix} \tag{3.22}$$

3.1.2 Indirect Approach (Variational)

The element stiffness matrix for a beam element can be derived using the variational approach, which involves minimizing the potential energy of the system. In this context, the nodal variables of a beam element must include not only the displacements of its nodes but also their rotations, as illustrated in Figure 3.2. This figure depicts nodes i and j of the element, located at its ends, along with the associated nodal: transverse displacements v1 and v2 and rotations θ1 and θ2.

Then the element displacement function can be represented as

$$v(x) = f(v_i, v_j, \theta_i, \theta_j) \tag{3.23}$$

its boundary conditions will be as follows:

$$v(x_1) = v_i$$
$$v(x_2) = v_j \tag{3.24}$$

$$\left.\frac{dv}{dx}\right|_{x_i} = \theta_i$$
$$\left.\frac{dv}{dx}\right|_{x_j} = \theta_j \tag{3.25}$$

Further derivation of the element shape function involves selecting a coordinate system such that x_1=0 =and x_2= l. While this choice is not mandatory, it simplifies the algebraic representation of the function. Given the boundary conditions and the one-dimensional nature of the problem (with respect to the independent variable), we can assume the existence of a displacement function in the following polynomial form:

$$v(x) = a_0 + a_1 x + a_2 x^2 + a_3 x^3 \tag{3.26}$$

Substitution of boundary conditions (3.24) and (3.25) into the equation (3.26) gives four equations that allow us to determine the coefficients of the polynomial:

$$v(0) = v_i = a_0$$
$$v(L) = v_j = a_0 + a_1 l + a_2 l^2 + a_3 l^3$$
$$\left.\frac{dv}{dx}\right|_{0} = \theta_i = a_1 \tag{3.27}$$
$$\left.\frac{dv}{dx}\right|_{l} = \theta_j = a_1 l + a_2 l^2 + a_3 l^2$$

The solution of these equations gives the following expressions for the coefficients of the polynomial:

$$a_0 = v_i$$
$$a_1 = \theta_j$$
$$a_2 = \frac{3}{l^2}(v_j - v_i) - \frac{1}{l}(2\theta_i + \theta_j) \tag{3.28}$$
$$a_3 = \frac{2}{l^2}(v_i - v_j) + \frac{1}{l^2}(\theta_i + \theta_j)$$

Substitution equations (3.28) into the equation (3.26) gives the final entry of the displacement function in the form:

$$v(x) = \left(1 - \frac{3x^2}{l^2} + \frac{2x^3}{l^3}\right)v_i + \left(x - \frac{2x^2}{l} + \frac{x^3}{l^2}\right)\theta_i + \left(\frac{3x^2}{l^2} - \frac{2x^3}{l^3}\right)v_j + \left(\frac{x^3}{l^2} - \frac{x^2}{l}\right)\theta_j \tag{3.29}$$

A more convenient form , this function is given by using the dimensionless coordinate ξ, such that:

$$\xi = \frac{x}{l} \tag{3.30}$$

the function takes the form:

$$v(x) = \left(1 - 3\xi^2 + 2\xi^3\right)v_i + l\left(\xi - 2\xi^2 + \xi^3\right)\theta_i + \left(3\xi^2 - 2\xi^3\right)v_j + l\xi^2\left(\xi - 1\right)\theta_j \tag{3.31}$$

This form of notation is better suited for integration when deriving the element's stiffness matrix

$$k_e = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \tag{3.32}$$

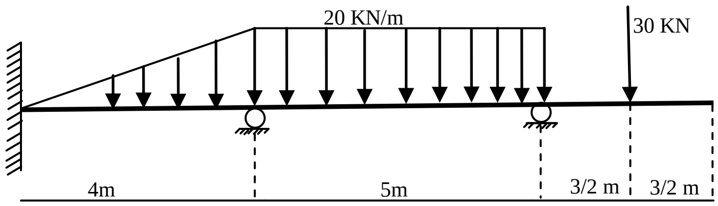
The stiffness matrix of a beam element is derived on the basis of Castigliano's theorem and in a two-dimensional formulation :

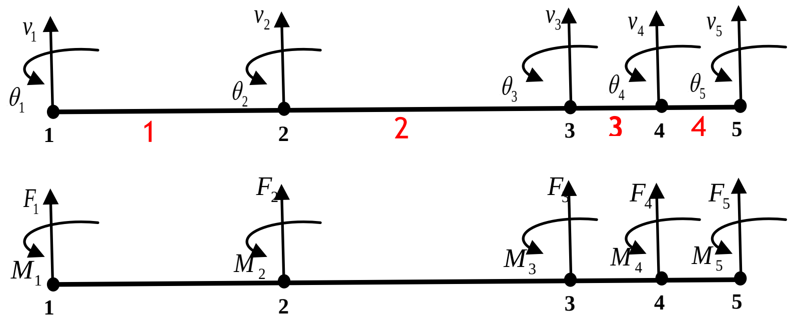
$$k_{mn} = k_{nm} = \frac{EI_z}{L^3}\int_0^1 \frac{d^2N_m}{d\xi^2}\frac{d^2N_n}{d\xi^2}d\xi \quad \text{m,n=1...4,} \tag{3.33}$$

Finally, the beam stiffness matrix is given as follows:

$$[k_e] = \begin{bmatrix} \frac{12EI}{l^3} & \frac{6EI}{l^2} & -\frac{12EI}{l^3} & \frac{6EI}{l^2} \\ \frac{6EI}{l^2} & \frac{4EI}{l} & -\frac{6EI}{l^2} & \frac{2EI}{l} \\ -\frac{12EI}{l^3} & -\frac{6EI}{l^2} & \frac{12EI}{l^3} & -\frac{6EI}{l^2} \\ \frac{6EI}{l^2} & \frac{2EI}{l} & -\frac{6EI}{l^2} & \frac{4EI}{l} \end{bmatrix} \tag{3.34}$$

Example 1





Given : EI = 3000 KNm², q= 20KN/m , F=30KN

Find : Deflections, rotations and reaction forces.

Solution

Step 1 – Enumeration of nodes and elements

Before listing the nodes and elements, it is important to determine where to place the nodes. Nodes should be set at all points where there is:

- A support
- A change in the type of load
- A point load
- A moment of force
- Sisplacement

Next, we will list each degree of freedom, each node, and each element. Note that when assigning degrees of freedom to the nodes, only two are assigned per node: one for vertical displacements and another for rotations. In this case, horizontal displacements are not necessary, as beam elements, unlike frames, can be assumed to not deform axially.

Step 2 – The stiffness matrices for each element are created

The matrix method involves dividing the beam into smaller segments. Each of these segments contains properties that can be expressed mathematically in matrix notation as follows:

$$\left[K^I\right] = \frac{EI}{L^3} \cdot \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}_I$$

Stiffness matrix of each element

element 1

$$\left[K^1\right] = \begin{bmatrix} 562.5 & 1125 & -562.5 & 1125 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1125 & 3000 & -1125 & 1500 & 0 & 0 & 0 & 0 & 0 & 0 \\ -562.5 & -1125 & 562.5 & -1125 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1125 & 1500 & -1125 & 3000 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

element 2

$$\left[K^2\right] = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 288 & 720 & -288 & 720 & 0 & 0 & 0 & 0 \\ 0 & 0 & 720 & 2400 & -120 & 1200 & 0 & 0 & 0 & 0 \\ 0 & 0 & -288 & -720 & 288 & -720 & 0 & 0 & 0 & 0 \\ 0 & 0 & 720 & 1200 & -720 & 2400 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

element 3

$$[K^3] = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 10666.7 & 8000 & -10666.7 & 8000 & 0 & 0 \\ 0 & 0 & 0 & 0 & 8000 & 8000 & -8000 & 4000 & 0 & 0 \\ 0 & 0 & 0 & 0 & -10666.7 & -8000 & -10666.7 & -8000 & 0 & 0 \\ 0 & 0 & 0 & 0 & 8000 & 4000 & -8000 & 8000 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

element 4

$$[K^4] = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 10666.7 & 8000 & -10666.7 & 8000 \\ 0 & 0 & 0 & 0 & 0 & 0 & 8000 & 8000 & -8000 & 4000 \\ 0 & 0 & 0 & 0 & 0 & 0 & -10666.7 & -8000 & 10666.7 & -8000 \\ 0 & 0 & 0 & 0 & 0 & 0 & 8000 & 4000 & -8000 & 8000 \end{bmatrix}$$

This is known as the stiffness matrix of an element. The stiffness matrix relates the forces and moments applied at the nodes to the displacements and rotations of those same nodes using the equation: With the data corresponding to each beam span (each element), the corresponding stiffness matrices must be determined. There are 4 spans, resulting in 4 elements and 4 elementary stiffness matrices:

Note: When generating an elementary stiffness matrix, it is good practice to indicate the degrees of freedom to which this matrix corresponds. This will facilitate the assembly of the overall stiffness matrix later. For example, the second stiffness matrix relates to nodes 2 and 3, with degrees of freedom 3, 4, 5, and 6, respectively. These degrees of freedom should be clearly noted above and alongside the elementary stiffness matrix, as illustrated by the green numbers in the image above.

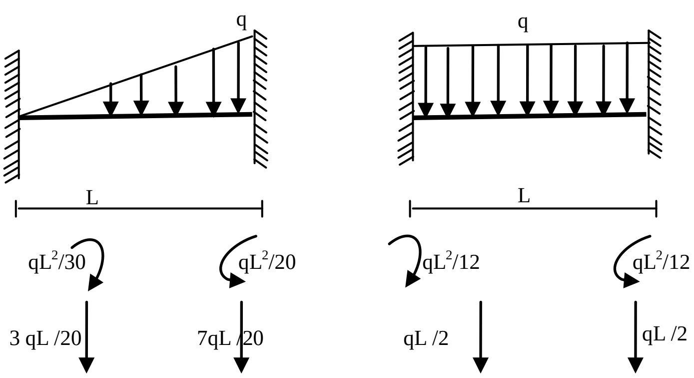
Step 3 – Assembling the Stiffness Matrix

For this step, imagine there is an empty matrix of dimension 10x10 (the same number of degrees of freedom as the entire beam) where we will place the elementary matrices one by one, according to their corresponding position. Repeat the procedure for the other two elements. The generated equation [K global]{U global}={F global} with all the elements added together

is:

$$
\left[K^G\right] = \begin{bmatrix}
562.5 & 1125 & -562.5 & -1125 & 0 & 0 & 0 & 0 & 0 & 0 \\
1125 & 3000 & 1125 & 1500 & 0 & 0 & 0 & 0 & 0 & 0 \\
-562.5 & 1125 & 850 & -405 & -288 & 720 & 0 & 0 & 0 & 0 \\
1125 & 1500 & -405 & 5400 & -720 & 1200 & 0 & 0 & 0 & 0 \\
0 & 0 & -288 & -720 & 10255 & 7280 & -10666.7 & 8000 & 0 & 0 \\
0 & 0 & 720 & 1200 & 7200 & 10400 & -8000 & 4000 & 0 & 0 \\
0 & 0 & 0 & 0 & -10666.7 & -8000 & 21333 & 0 & -10666.7 & 8000 \\
0 & 0 & 0 & 0 & 8000 & 4000 & 0 & 16000 & -8000 & 4000 \\
0 & 0 & 0 & 0 & 0 & 0 & -10666.7 & -8000 & 10666.7 & -8000 \\
0 & 0 & 0 & 0 & 0 & 0 & 8000 & 4000 & -8000 & 8000
\end{bmatrix}
$$

Step 4 – Imposing boundary conditions and loads



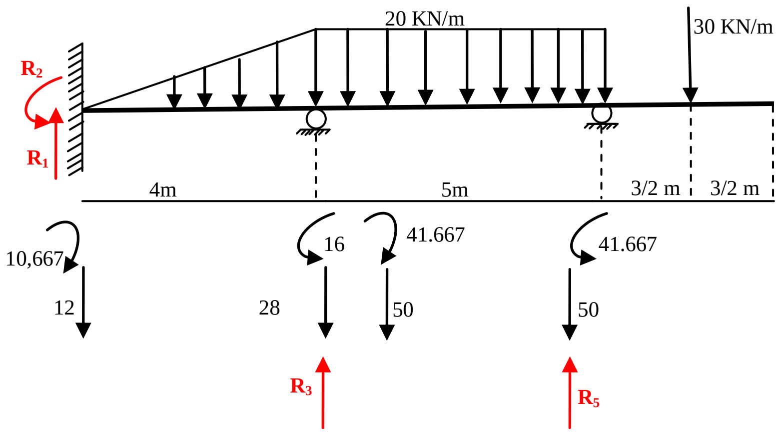
$$
\left[K^{G}\right]=\begin{bmatrix}
562.5 & 1125 & -562.5 & -1125 & 0 & 0 & 0 & 0 & 0 & 0\\
1125 & 3000 & 1125 & 1500 & 0 & 0 & 0 & 0 & 0 & 0\\
-562.5 & 1125 & 850 & -405 & -288 & 720 & 0 & 0 & 0 & 0\\
1125 & 1500 & -405 & 5400 & -720 & 1200 & 0 & 0 & 0 & 0\\
0 & 0 & -288 & -720 & 10255 & 7280 & -10666.7 & 8000 & 0 & 0\\
0 & 0 & 720 & 1200 & 7200 & 10400 & -8000 & 4000 & 0 & 0\\
0 & 0 & 0 & 0 & -10666.7 & -8000 & 21333 & 0 & -10666.7 & 8000\\
0 & 0 & 0 & 0 & 8000 & 4000 & 0 & 16000 & -8000 & 4000\\
0 & 0 & 0 & 0 & 0 & 0 & -10666.7 & -8000 & 10666.7 & -8000\\
0 & 0 & 0 & 0 & 0 & 0 & 8000 & 4000 & -8000 & 8000
\end{bmatrix}
\begin{Bmatrix} v_1=0\\ \theta_1=0\\ v_2=0\\ \theta_2\\ v_3=0\\ \theta_3\\ v_4\\ \theta_4\\ v_5\\ \theta_5 \end{Bmatrix}
=\begin{Bmatrix} F_1\\ M_1\\ F_2\\ M_2\\ F_3\\ M_3\\ F_4\\ M_4\\ F_5\\ M_5 \end{Bmatrix}
$$

In this figure, the restricted degrees of freedom are those corresponding to the supports that prevent the beam from rotating or shifting in that sector. For example, the embedding prevents degrees of freedom 1 and 2 from shifting or rotating, so U_1 and U_2 are zero. The same applies to degrees of freedom 3 and 5. They cannot shift vertically, so they are zero. Therefore, where the displacements are zero, the reactions are unknown. Therefore, each of these equivalent loads must be entered into the force and moment vector in the linear system of equations in the corresponding degree of freedom. Point loads, such as the 30 kN load, do not need to be transformed into an equivalent load, as this is already a load applied directly to a node. Thus, for example, for degree of freedom 3, the concurrent loads in this degree of freedom are:

- The equivalent load of 28 kN resulting from the triangular load
- The equivalent load of 50 kN resulting from the distributed load
- The reaction R3 resulting from the support.

These three forces are entered into the force vector in degree of freedom 3.

The same is repeated for both moments and forces in the corresponding degrees of freedom, thus:



Step 5 Solving the System of Equations

To solve the system, the rows and columns corresponding to the restricted degrees of freedom must first be eliminated from the system. In this case, rows and columns 1, 2, 3, and 5.

In this way, we are left with only the unknown displacements as unknowns.

$$\left[K^{reduced}\right] = \cdot \begin{bmatrix} 5400 & 1200 & 0 & 0 & 0 & 0 \\ 1200 & 10400 & -8000 & 4000 & 0 & 0 \\ 0 & -8000 & 21333.3 & 0 & -10666.7 & 8000 \\ 0 & 4000 & 0 & 16000 & -8000 & 4000 \\ 0 & 0 & -10666.7 & -8000 & 10666.7 & -8000 \\ 0 & 0 & 8000 & 4000 & -8000 & 8000 \end{bmatrix} = \begin{Bmatrix} \theta_2 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{Bmatrix} = \begin{Bmatrix} -25.667 \\ 41.667 \\ -30 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

This elimination of rows and columns might seem arbitrary, but it is not. Columns 1, 2, 3, and 5 are eliminated since, when multiplying [K] by {u}, the coefficients that accompany displacements equal to zero are obviously eliminated in the multiplication. The reduced system of equations shown now is solved classically, by inversion of the reduced stiffness matrix, or by methods such as Gauss-Jordan or any similar method.

$$\begin{Bmatrix} \theta_2 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{Bmatrix} = \begin{Bmatrix} -0.005 \text{ rad} \\ -0.00111 \text{ rad} \\ -0.0095 \text{ m} \\ -0.01013 \text{ rad} \\ -0.02479 \text{ m} \\ -0.01013 \text{ rad} \end{Bmatrix}$$

Reactions forces

To find the reactions R_1, R_2, R_3, and R_5, simply replace the displacements found above and insert them into the overall 10x10 system of equations. This gives:

$$\left[K^G\right] = \begin{bmatrix} 562.5 & 1125 & -562.5 & -1125 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1125 & 3000 & 1125 & 1500 & 0 & 0 & 0 & 0 & 0 & 0 \\ -562.5 & 1125 & 850 & -405 & -288 & 720 & 0 & 0 & 0 & 0 \\ 1125 & 1500 & -405 & 5400 & -720 & 1200 & 0 & 0 & 0 & 0 \\ 0 & 0 & -288 & -720 & 10255 & 7280 & -10666.7 & 8000 & 0 & 0 \\ 0 & 0 & 720 & 1200 & 7200 & 10400 & -8000 & 4000 & 0 & 0 \\ 0 & 0 & 0 & 0 & -10666.7 & -8000 & 21333 & 0 & -10666.7 & 8000 \\ 0 & 0 & 0 & 0 & 8000 & 4000 & 0 & 16000 & -8000 & 4000 \\ 0 & 0 & 0 & 0 & 0 & 0 & -10666.7 & -8000 & 10666.7 & -8000 \\ 0 & 0 & 0 & 0 & 0 & 0 & 8000 & 4000 & -8000 & 8000 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -0.005 \\ 0 \\ 0.0011 \\ -0.0055 \\ 0.01013 \\ -0.02473 \\ -0.01013 \end{Bmatrix} = \begin{Bmatrix} F_1 = R_1 - 12 \\ M_1 = R_2 + 10.667 \\ F_2 = R_3 - 28.50 \\ M_2 = 16 - 41.667 \\ F_3 = R_5 - 50 \\ M_3 = 41.667 \\ F_4 = -30 \\ M_4 = 0 \\ F_5 = 0 \\ M_5 = 0 \end{Bmatrix}$$

Multiplying the two sides on the left, we obtain a 10×1 vector that is equal to the 10×1 vector on the right: Each term on the left is equal to each term on the right. In the rows where there are unknowns, the respective reactions are solved. In the rows where there are no unknowns, the identity is simply verified.

$$R_1 = 6.3749 KN$$
$$R_2 = 3.1665 KN$$
$$R_3 = 80.8252 KN$$
$$R_5 = 82.7999 KN$$

References

Akin, J.E. (2022) *Finite Element Analysis: With Numeric and Symbolic Matlab*. New Jersey: World Scientific Publishing Company.

Alaa Chateauneuf (2010) *Comprendre les éléments finis*. Librairie Eyrolles.

Bagrianski, S. and Prevost, J.H. (2017) *Introduction To Matrix Structural Analysis And Finite Element Methods, An*. World Scientific.

Bécache, E. et al. (2010) *La méthode des éléments finis - Tome II. De la théorie à la pratique. Compléments*. 1st edition. Paris: ENSTA.

Debard, Y. (2011) *Résistance des matériaux : élasticité, méthodes énergétiques, méthode des éléments finis. Rappels de cours et exercices avec solutions*. Institut Universitaire de Technologie du Mans.

Kamenev, S.V. (2019) *Fundamentals of the Finite Element Method in Engineering Applications*. : Orenburg State University. (Original text in Russian)

Larson, M.G. and Bengzon, F. (2013) *The Finite Element Method: Theory, Implementation, and Applications*. 2013th edition. Heidelberg; New York: Springer.

Li, G. (2020) *Introduction to the Finite Element Method and Implementation with MATLAB®*. 1st edition. New York: Cambridge University Press.

Liu, Y. (1997-2013) *Lecture Notes: An Introduction to the Finite Element Method*. Mechanical Engineering Department, University of Cincinnati.

Logan, D.L. (2011) *A First Course in the Finite Element Method*. 5th edition. Stamford, CT: CL Engineering.

Mohamed AMIRAT (2012) , Méthode des éléments finis, Note de cours, ECAM de Rennes.**Ovcharenko, V.A.** (2004) *Calculation of Mechanical Engineering Problems by the Finite Element Method: A Textbook*. Kramatorsk: Donbass State Engineering Academy. ISBN 966-7851-28-1. (Original text in Russian)

Pavlenko, I.V. (2006) *Finite Element Method in Problems of Strength of Materials and Linear Theory of Elasticity: Textbook*. Sumy: Sumy State University Publishing House. ISBN 966-657-087-4. (Original text in Russian)

Smith, I.M., Griffiths, D.V. and Margetts, L. (2013) *Programming the Finite Element Method*. 5th edition. Chichester, West Sussex, United Kingdom: Wiley.

OUINAS, Djamel (2014) *Application de la méthode des éléments finis*. Office des publications universitaires. Algérie.

Mohamed AMIRAT (2012) , Méthode des éléments finis, Note de cours, ECAM de Rennes.