10

Finite element analysis

10.1 Introduction

Finite element methods offer a route to solve many engineering problems. The method was initially developed for the solution of structural mechanics problems[1], but is being applied to many other problems in dynamics, fluids, thermal analysis, and more.

From a general point of view, the finite element method (FEM) can be applied to any mathematical problem for which the *weak formulation*—a variational functional—exists (see variational calculus texts[2] for details, and for those mathematically inclined, consider functional analysis including Sobolev spaces[3]). Traditionally, students are made familiar with the differential representation of the equations—the *strong* formulation—that model the behavior of a domain—the object or system under study—whether composed of fluid, solid, gas, or something else. These are solved across the domain, making use of initial and boundary conditions as needed, but requiring extraordinary effort at times, even for apparently simple geometries. The *weak* formulation of the equations, however, are integrodifferential equations that define scalar quantities to represent the domain as a whole. For example, the work performed or the internal energy present in the domain.

FEM is a numerical technique that discretizes the continuous representation of the domain into individual parcels where the integrodifferential equations are easily solved to generate algebraic equations that relate what is happening at the parcel's boundary with

[1] Robert D Cook et al. *Concepts and applications of finite element analysis*. John Wiley & Sons, 2007; Klaus-Jürgen Bathe. *Finite element procedures*. Klaus-Jurgen Bathe, 2006; Junuthula Narasimha Reddy. *An introduction to the finite element method*. McGraw-Hill New York, 1993; Junuthula Narasimha Reddy. *An Introduction to Nonlinear Finite Element Analysis: with applications to heat transfer, fluid mechanics, and solid mechanics*. OUP Oxford, 2014; Tod A Laursen. *Computational contact and impact mechanics: fundamentals of modeling interfacial phenomena in nonlinear finite element analysis*. Springer Science & Business Media, 2013; and Joel H Ferziger and Milovan Peric. *Computational methods for fluid dynamics*. Springer Science & Business Media, 2012

[2] M. Kot. *A First Course in the Calculus of Variations:*. Student Mathematical Library. American Mathematical Society, 2014; and Kevin W Cassel. *Variational Methods with Applications in Science and Engineering*. Cambridge University Press, 2013

[3] Haim Brezis. *Functional analysis, Sobolev spaces and partial differential equations*. Springer Science & Business Media, 2010

what is going on inside it. Each parcel has its own set of algebraic equations that represent it, and the entire domain can be represented by an assembly of these equation sets, taking into account the common interfaces between each parcel. The arduous process of solving the partial differential equations with the relevant boundary conditions is replaced with the tedious but straightforward process of solving algebraic equations. The accuracy of the FEM solution depends upon convergence criteria: tests on the scalar quantities that represent the domain, and measures of error in these quantities (i.e., the *error norm*).

The structure that is being analyzed is first divided into distinct, non-overlapping regions (*elements*). The elements are connected at discrete points (*nodes* or nodal points). For a nominal element shape, FEM makes use of an exact solution of the partial differential equations at the nodal points and interpolates along a predefined *shape function* to provide an estimate for the solution in between the nodes. Small changes in the element shape, for the purposes of accommodating the shape of a domain, are permitted, but can lead to error if the shape change is too significant: such elements are said to have *aspect ratio* problems: the element no longer looks like the original shape used to represent the partial differential equation solution, and so gives a poor result. Fig. 10.1 shows an arbitrary shape, divided into a finite number of elements that are interconnected at the nodes.

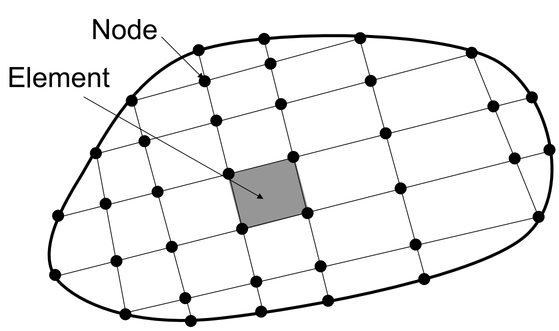

Figure 10.1: A domain—a structure—being represented with nodes and elements.

The general procedure of the finite element method for static structural problems can be outlined as follows:

1. Divide the (complex) structure into a finite number of sub-domains, called elements.

2. Transition of an infinite number of degrees of freedom in the real structure to a finite number of degrees of freedom in the finite element model by introducing shape functions in each element. A shape function is an interpolation function, which expresses the value of a degree of freedom (physical parameter) at any point in an element as a function of the value of that degree of freedom in the nodal points of that element.

3. Define the local equilibrium equations for each individual element. Create the *element stiffness matrix*.

4. Assemble all individual element stiffness matrices to form a *global stiffness matrix* and a global stiffness equation.

5. Apply *boundary conditions*.

6. Solve the *global stiffness equation*.

The advantages of the finite element method lie in the fact that nearly any arbitrary geometry can be modeled, composed out of different materials, and arbitrary support and load conditions, all contingent on being able to define the geometry with a mesh of elements.[4]

[4] Issues that commonly arise include *reentrant corners*, where the mesh has elements too sharply defined because of a corner; aspect ratios that are too great, where an element has one length far greater than the other which invalidates the algebraic solution for the underlying integrodifferential equations; and failure to generate suitable mesh size transitions between large and small features.

The disadvantage, however, is that a specific problem generates a specific solution. Hence, different geometries require complete recalculation of new solutions every time the geometry is changed. Moreover, experience and judgment are indispensable to obtain a good model. The finite element method is very computer intensive (depending on the size of the model), so a fast computer with large memory and storage capabilities is desirable.

10.2 *Discretization*

In any continuum, the actual number of degrees of freedom is infinite, and, generally, exact analysis is impossible. An approximate solution is attempted assuming that the continuum solution can be obtained using a finite number of unknowns. In finite element analysis, the continuum is divided into a series of elements which are connected via a finite number of nodes. Fig. 10.2 shows an example of a FEM mesh for a hard disk drive slider, used to calculate the pressure distribution under the slider, when it is flying over a hard disk. Triangular elements were used.

From Fig. 10.2, one observes that the FEM mesh is non-uniform, i.e., some elements are larger than others, resulting in areas that have more nodes, and producing a denser mesh than in other areas. The reason for using a non-uniform mesh is straightforward: FEM provides an exact solution at the nodal points and interpolates the solution by means of shape functions in between the nodal points. Hence, adding more nodes (producing smaller elements), will increase the accuracy of the solution, since the exact solution at the nodes is calculated at more locations throughout the structure. However, adding nodes increases the calculation time and the roundoff error that arises from the greater number of calculations. Therefore, a tradeoff between accuracy and calculation time exists. A reasonable balance is to have a gradient in the mesh density so that a lot of nodes (and small elements) are present

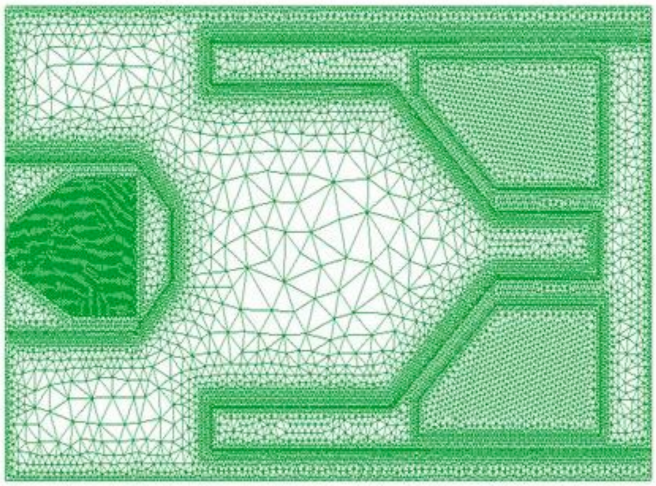

Figure 10.2: An example mesh used in the finite element method.

in areas where the value of the degree of freedom one is calculating (physical parameter) changes quickly, i.e., where steep gradients exist. In other areas, where the calculated values change more gradually, fewer nodes will suffice to provide an accurate solution. Hence, a lot of calculation time can be saved.

10.3 *Shape functions*

A shape function is an *interpolation function* that expresses the value of a degree of freedom (physical parameter) at any point in an element as a function of the value of that degree of freedom at the nodes of that element:

$$\boldsymbol{d} = [N]\,\boldsymbol{d}^{\boldsymbol{e}}, \tag{10.1}$$

where $\boldsymbol{d}$ is the displacement vector of an arbitrary point of an element, $[N]$ is a matrix that contains the shape functions of all nodes of the element and $\boldsymbol{d}^{\boldsymbol{e}}$ is a vector containing all **nodal** displacements. Equation (10.1) is the key equation of the finite element method, because it specifies that a structure with an infinite number of degrees of freedom, requiring partial differential equations to represent it in a model, is reduced to a finite number of degrees of freedom, requiring only algebraic relations. Fig. 10.3 illustrates the concept of shape functions for a one dimensional linear case. For each node i and j, there is only one degree of freedom, the displacement d_i^e and d_j^e, respectively. The *continuous* value of the displacement $d(x)$ at any point x, within the element (between nodes i and j) can be defined through two shape functions N_i and N_j as a function of the *discrete* nodal displacements

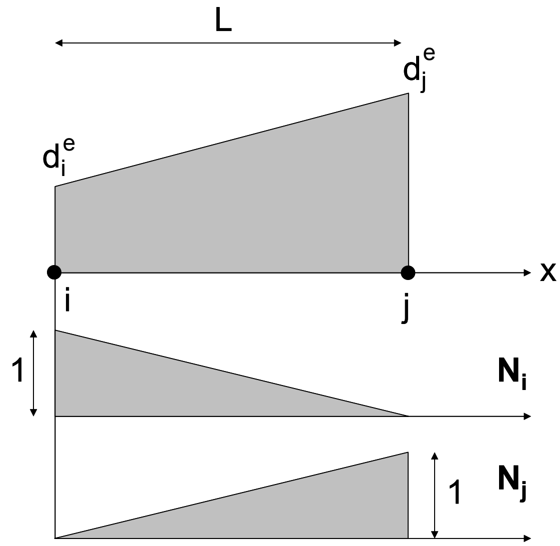

Figure 10.3: One dimensional linear shape function.

d_i^e and d_j^e, i.e.,

$$\boldsymbol{d}(x) = \begin{bmatrix} N_i & N_j \end{bmatrix} \begin{Bmatrix} d_i^e \\ d_j^e \end{Bmatrix} \tag{10.2}$$

with

$$N_i = 1 - \frac{x}{L} \tag{10.3}$$

and

$$N_j = \frac{x}{L}. \tag{10.4}$$

Thus eqn. (10.2) produces

$$\boldsymbol{d}(x) = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} \begin{Bmatrix} d_i^e \\ d_j^e \end{Bmatrix}. \tag{10.5}$$

Shape function $N_i = 1$ at node i and is 0 at node j. Shape function $N_j = 1$ at node j and is 0 at node i. For $x = L/2$, one finds that the displacement will be $d(L/2) = d_i/2 + d_j/2$. The shape functions provide an interpolation to provide the continuous displacement distribution from x_i to x_j based upon the discrete values of the displacement, d_i^e and d_j^e, at these nodal locations.

In a complex structure, the value of the degree of freedom varies from node to node according to the laws of physics applicable for the specific problem one is solving. For example, in the case of the pressure distribution under the hard disk drive slider (Fig. 10.2), the pressure is obtained from the Reynolds equation (in the forthcoming lubrication equations of Chapter 12). FEM calculates the exact value of the solution of the Reynolds equation in each node. The shape function tries to approximate this exact solution and interpolates the nodal values for all points in between nodes. Oftentimes higher order polynomials are used as shape functions instead of the linear shape functions described above.

To obtain a more accurate FEM solution, two approaches are thus possible:

1. increase the number of nodes or
2. increase the order of the shape functions.

Both methods will increase the calculation time.

10.4 Local equilibrium equations

The local equilibrium equations can be derived by two methods:

1. Minimum potential energy method
2. Method of virtual work

10.4.1 Minimum potential energy method

Assume an elastic structure with loads acting on it. This system is conservative if the work done in deforming the structure to any deformed shape and restoring it back to its original shape is zero. The potential energy of a system is contained in the elastic deformations and the capacity of the loads to do work.

The principle of minimum potential energy states that among all displacement configurations that satisfy internal compatibility and kinematic boundary conditions, those that also satisfy the equilibrium equations make the potential energy minimum. If the stationary value is a minimum, the equilibrium is stable. The strain vector ε can be related to the displacement vector $\boldsymbol{d}$ as

$$\varepsilon = [L]\,\boldsymbol{d}, \tag{10.6}$$

where $\boldsymbol{L}$ is an operator matrix that calculates the derivative of the displacement vector $\boldsymbol{d}$. The strain ε within an element can be expressed in terms of the element nodal displacements, $\boldsymbol{d}^e$; when substituting eqn. (10.1) in eqn. (10.6),

$$\varepsilon = [L]\,[N]\,\boldsymbol{d}^e \tag{10.7}$$

and thus,

$$\varepsilon = [B]\,\boldsymbol{d}^e, \tag{10.8}$$

where $[B]$= $[L]\,[N]$ is the strain matrix, made up of the derivatives of the shape functions. The strain vector is related to the stress vector by the elasticity matrix $[D]$, i.e.,

$$\sigma = [D]\,\varepsilon. \tag{10.9}$$

Substituting eqn. (10.8) in eqn. (10.9) produces

$$\boldsymbol{\sigma} = [D]\,[B]\,\boldsymbol{d}^{\boldsymbol{e}}. \tag{10.10}$$

An individual element's potential energy π_e can be expressed as

$$\pi_e = \frac{1}{2}\int_{V_e} \boldsymbol{\sigma}^t \boldsymbol{\varepsilon}\, dV - \int_{V_e} \boldsymbol{d}^t \boldsymbol{p}\, dV - \int_{S_e} \boldsymbol{d}^t \boldsymbol{q}\, dS. \tag{10.11}$$

The first term in eqn. (10.11) describes the internal strain energy, the second term describes the work of body forces $\boldsymbol{p}$ and the third term describes the work of surface forces $\boldsymbol{q}$; $\boldsymbol{d}$ is the displacement vector. The first two terms are integrated over the element volume V_e, while the third term is integrated over the element surface S_e. Note that the superscript t indicates the transposed matrix. The total potential energy of the continuum will be the sum of the potential energy contributions of the individual elements

$$\pi = \sum_e \pi_e. \tag{10.12}$$

Substituting eqns. (10.8) and (10.10) in the internal strain energy term of eqn. (10.11) produces the following result (the elasticity matrix $[D]^t$=$[D]$ because of symmetry):

$$\frac{1}{2}\int_{V_e} \boldsymbol{\sigma}^t \boldsymbol{\varepsilon}\, dV = \frac{1}{2}\int_{V_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[B]^t\,[D]\,[B]\,\boldsymbol{d}^{\boldsymbol{e}}\, dV. \tag{10.13}$$

Substituting eqn. (10.1) in the body force term produces

$$\int_{V_e} \boldsymbol{d}^t \boldsymbol{p}\, dV = \int_{V_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[N]^t\,\boldsymbol{p}\, dV, \tag{10.14}$$

and equivalently for the surface forces

$$\int_{S_e} \boldsymbol{d}^t \boldsymbol{q}\, dS = \int_{S_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[N]^t\,\boldsymbol{q}\, dS. \tag{10.15}$$

Substituting eqns. (10.13)–(10.15) into eqn. (10.11) gives

$$\pi_e = \frac{1}{2}\int_{V_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[B]^t\,[D]\,[B]\,\boldsymbol{d}^{\boldsymbol{e}} dV - \int_{V_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[N]^t\,\boldsymbol{p}\, dV - \int_{S_e} \boldsymbol{d}^{\boldsymbol{e}t}\,[N]^t\,\boldsymbol{q}\, dS. \tag{10.16}$$

Minimization of the potential energy can be calculated as the derivative of the potential energy of an element with respect to the displacement in the nodes of that element:

$$\frac{\partial \pi_e}{\partial \boldsymbol{d}^{\boldsymbol{e}}} = 0, \tag{10.17}$$

or,

$$\frac{\partial \pi_e}{\partial \boldsymbol{d}^{\boldsymbol{e}}} = \int_{V_e} [B]^t\,[D]\,[B]\,\boldsymbol{d}^{\boldsymbol{e}}\, dV - \int_{V_e} [N]^t\,\boldsymbol{p}\, dV - \int_{S_e} [N]^t\,\boldsymbol{q}\, dS = 0. \tag{10.18}$$

Thus,

$$\frac{\partial \pi_e}{\partial \boldsymbol{d}^e} = [K^e]\,\boldsymbol{d}^e - \boldsymbol{F}^e = 0, \tag{10.19}$$

where the element stiffness matrix

$$[K^e] = \int_{V_e} [B]^t\,[D]\,[B]\,dV \tag{10.20}$$

and the external forces acting on the element

$$\boldsymbol{F}^e = \int_{V_e} [N]^t\,\boldsymbol{p}\,dV + \int_{S_e} [N]^t\,\boldsymbol{q}\,dS. \tag{10.21}$$

The local equilibrium (stiffness) equation for an element, can thus be written as

$$[K^e]\,\boldsymbol{d}^e = \boldsymbol{F}^e. \tag{10.22}$$

10.4.2 Method of virtual work

A different approach for obtaining the equilibrium equation uses the principle of virtual work. Consider a single element with nodal loads $\boldsymbol{F}_S^e$ and body forces $\boldsymbol{p}$ which result in stresses $\boldsymbol{\sigma}$. Subjecting this element to an arbitrary virtual nodal displacement $\boldsymbol{d}_*^e$ results in compatible internal displacement and strain distributions of $\boldsymbol{d}_*$ and $\boldsymbol{\varepsilon}_*$.

The theorem of virtual work states that the external virtual work done must be equal to the internal virtual work. Or in other words; the work done by the external forces is equal to the increase in strain energy:

$$\boldsymbol{d}_*^{et}\boldsymbol{F}_S^e + \int_{V_e} \boldsymbol{d}_*^t\boldsymbol{p}\,dV = \int_{V_e} \boldsymbol{\varepsilon}_*^t\boldsymbol{\sigma}_*\,dV. \tag{10.23}$$

Adapting eqns. (10.1) and (10.8) for virtual work produces

$$\boldsymbol{d}_* = [N]\,\boldsymbol{d}_*^e \tag{10.24}$$

and

$$\boldsymbol{\varepsilon}_* = [B]\,\boldsymbol{d}_*^e. \tag{10.25}$$

Substituting eqns. (10.24) and 10.25 into 10.23 gives

$$\boldsymbol{d}_*^{et}\boldsymbol{F}_S^e + \int_{V_e} \boldsymbol{d}_*^{et}\,[N]^t\,\boldsymbol{p}\,dV = \int_{V_e} \boldsymbol{d}_*^{et}\,[B]^t\,\boldsymbol{\sigma}_*\,dV \tag{10.26}$$

or, after rearranging

$$\boldsymbol{d}_*^{et}\left[\boldsymbol{F}_S^e + \int_{V_e} [N]^t\,\boldsymbol{p}\,dV\right] = \boldsymbol{d}_*^{et}\int_{V_e} [B]^{\boldsymbol{t}}\,\boldsymbol{\sigma}_*\,dV. \tag{10.27}$$

The above expression must hold for all values of $\boldsymbol{d}_*^e$, since the virtual nodal displacement has been chosen arbitrarily. Thus,

$$\boldsymbol{F}_S^e + \int_{V_e} [N]^t\,\boldsymbol{p}\,dV = \int_{V_e} [B]^{\boldsymbol{t}}\,\boldsymbol{\sigma}_*dV. \tag{10.28}$$

Substituting $\boldsymbol{\sigma}_* = [D]\,[B]\,\boldsymbol{d}_*^{\boldsymbol{e}}$ in eqn. (10.28) gives

$$\boldsymbol{F}_{\boldsymbol{S}}^{\boldsymbol{e}} + \int_{V_e} [N]^t\,\boldsymbol{p}\,dV = \int_{V_e} [B]^{\boldsymbol{t}}\,[D]\,[B]\,\boldsymbol{d}_*^{\boldsymbol{e}}\,dV; \qquad (10.29)$$

eqn. (10.29) can be rewritten as

$$[K^e]\,\boldsymbol{d}_*^{\boldsymbol{e}} = \boldsymbol{F}_*^{\boldsymbol{e}}, \qquad (10.30)$$

where the element stiffness matrix $[K^e] = \int_{V_e} [B]^t\,[D]\,[B]\,dV$ as in eqn. (10.20) and

$$\boldsymbol{F}_*^{\boldsymbol{e}} = \boldsymbol{F}_{\boldsymbol{S}}^{\boldsymbol{e}} + \int_{V_e} [N]^t\,\boldsymbol{p}\,dV. \qquad (10.31)$$

Equation (10.30) is the stiffness equation for one element and is equivalent to eqn. (10.22), obtained with the minimum potential energy method. For more than one element, one needs to sum over all elements. The stiffness equations are assembled for the global structure (global stiffness matrix) and the set of equations is solved for the nodal displacements.

10.5 *Assembling the global stiffness equation*

Once the individual stiffness equation for each element of the mesh is known, the global stiffness equation can be assembled by means of the compatibility conditions. These conditions simply state that a certain degree of freedom in a node, common to different elements, has the same value with respect to each element. Indeed, if one node, belonging to element X would have a different value for the displacement than the same node belonging to an adjacent element Y, discontinuities would appear in the displacement field at the element boundaries. The best way to explain the assembly of the individual stiffness equations to the global stiffness equation is by means of a simple example. Fig. 10.4 shows a simple structure. In Fig. 10.4 the elements are indicated by

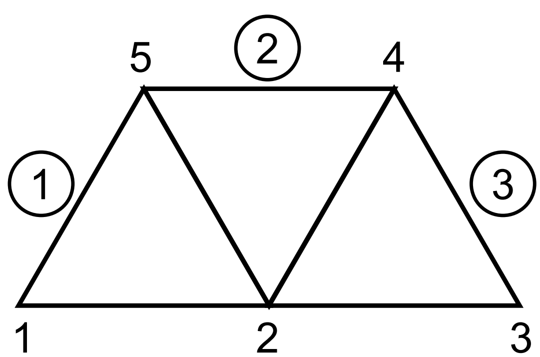

Figure 10.4: A truss for illustrating the assembly of the stiffness equations.

circled numbers, while the nodes are denoted by just numbers. Each node i has one degree of freedom, d_i. External forces F_i^j are applied to the structure, with the index i referring to the node number, and the index j referring to the element number. The element stiffness equation for element 1 can then be written as

$$\begin{bmatrix} K_{11}^1 & K_{12}^1 & K_{15}^1 \\ K_{21}^1 & K_{22}^1 & K_{25}^1 \\ K_{51}^1 & K_{52}^1 & K_{55}^1 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_1^1 \\ F_2^1 \\ F_5^1 \end{Bmatrix}. \tag{10.32}$$

In expanded form, it becomes

$$\begin{bmatrix} K_{11}^1 & K_{12}^1 & 0 & 0 & K_{15}^1 \\ K_{21}^1 & K_{22}^1 & 0 & 0 & K_{25}^1 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ K_{51}^1 & K_{52}^1 & 0 & 0 & K_{55}^1 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ 0 \\ 0 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_1^1 \\ F_2^1 \\ 0 \\ 0 \\ F_5^1 \end{Bmatrix}. \tag{10.33}$$

For element 2,

$$\begin{bmatrix} K_{22}^2 & K_{24}^2 & K_{25}^2 \\ K_{42}^2 & K_{44}^2 & K_{45}^2 \\ K_{52}^2 & K_{54}^2 & K_{55}^2 \end{bmatrix} \begin{Bmatrix} d_2 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_2^2 \\ F_4^2 \\ F_5^2 \end{Bmatrix}, \tag{10.34}$$

and in expanded form

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & K_{22}^2 & 0 & K_{24}^2 & K_{25}^2 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & K_{42}^2 & 0 & K_{44}^2 & K_{45}^2 \\ 0 & K_{52}^2 & 0 & K_{54}^2 & K_{55}^2 \end{bmatrix} \begin{Bmatrix} 0 \\ d_2 \\ 0 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2^2 \\ 0 \\ F_4^2 \\ F_5^2 \end{Bmatrix}. \tag{10.35}$$

For element 3,

$$\begin{bmatrix} K_{22}^3 & K_{23}^3 & K_{24}^3 \\ K_{32}^3 & K_{33}^3 & K_{34}^3 \\ K_{42}^3 & K_{43}^3 & K_{44}^3 \end{bmatrix} \begin{Bmatrix} d_2 \\ d_3 \\ d_4 \end{Bmatrix} = \begin{Bmatrix} F_2^3 \\ F_3^3 \\ F_4^3 \end{Bmatrix}, \tag{10.36}$$

and in expanded form

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & K_{22}^3 & K_{23}^3 & K_{24}^3 & 0 \\ 0 & K_{32}^3 & K_{33}^3 & K_{34}^3 & 0 \\ 0 & K_{42}^3 & K_{43}^3 & K_{44}^3 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ d_2 \\ d_3 \\ d_4 \\ 0 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2^3 \\ F_3^3 \\ F_4^3 \\ 0 \end{Bmatrix}. \tag{10.37}$$

The global stiffness equation can now be calculated as the sum of the contributions of the individual stiffness equations. The forces appearing in the right hand side vector, however, each need only to be counted once, since they are the same force that appear for different elements.

$$\begin{bmatrix} K_{11}^1 & K_{12}^1 & 0 & 0 & K_{15}^1 \\ K_{21}^1 & K_{22}^1+K_{22}^2+K_{22}^3 & K_{23}^3 & K_{24}^2+K_{24}^3 & K_{15}^1+K_{25}^2 \\ 0 & K_{32}^3 & K_{33}^3 & K_{34}^3 & 0 \\ 0 & K_{42}^2+K_{42}^3 & K_{43}^3 & K_{44}^2+K_{44}^3 & K_{45}^2 \\ K_{51}^1 & K_{51}^1+K_{52}^2 & 0 & K_{54}^2 & K_{55}^1+K_{55}^2 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{Bmatrix}. \tag{10.38}$$

Equation (10.38) is the global stiffness equation. Similar to the individual stiffness matrices, the global stiffness matrix is also symmetric.

The numbering of the nodes will have great impact on the global stiffness matrix. From a numerical point of view, it is most favorable to have a banded matrix structure, where all non-zero elements of a matrix are clustered around the main diagonal. In this way, the least computer memory is needed and the faster the calculation will be done. A sparse matrix, where the non-zero elements are not clustered around the main diagonal, requires more computation time. Commercial FEM packages have algorithms to number elements and nodes automatically in an optimized way. For small FEM problems (i.e., your assignments) it might be wise to sit back for a minute and think about the optimum numbering of the nodes and elements to reduce the calculation effort.

10.6 *Boundary conditions*

Referring back to Fig. 10.4, we assume that the structure is clamped in nodes 1 and 3, i.e., the displacements d_1 and d_3 are zero and consequently the external forces must be replaced by unknown reaction forces F_1 and F_3. This can easily be implemented in the global stiffness equation

$$\begin{bmatrix} K_{11}^1 & K_{12}^1 & 0 & K_{14}^1 & 0 \\ K_{21}^1 & K_{22}^1+K_{22}^2+K_{22}^3 & K_{23}^3 & K_{24}^1+K_{24}^2 & K_{15}^1+K_{25}^3 \\ 0 & K_{32}^3 & K_{33}^3 & K_{34}^3 & K_{35}^3 \\ K_{41}^1 & K_{42}^1+K_{42}^2 & K_{43}^3 & K_{44}^1+K_{44}^2 & K_{45}^2 \\ 0 & K_{51}^1+K_{52}^3 & 0 & K_{54}^2 & K_{55}^2+K_{55}^3 \end{bmatrix} \begin{Bmatrix} 0 \\ d_2 \\ 0 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{Bmatrix}. \tag{10.39}$$

Equation (10.39) reduces to

$$\begin{bmatrix} K_{22}^1+K_{22}^2+K_{22}^3 & K_{24}^1+K_{24}^2 & K_{15}^1+K_{25}^3 \\ K_{42}^1+K_{42}^2 & K_{44}^1+K_{44}^2 & K_{45}^2 \\ K_{51}^1+K_{52}^3 & K_{54}^2 & K_{55}^2+K_{55}^3 \end{bmatrix} \begin{Bmatrix} d_2 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_2 \\ F_4 \\ F_5 \end{Bmatrix}. \tag{10.40}$$

10.7 Solving the global stiffness equation

Once the global stiffness equation is known and the boundary conditions have been implemented, eqn. (10.40) can be solved for the unknown displacements, using numerical techniques:

$$\boldsymbol{d^e} = [K^e]^{-1} \boldsymbol{F^e}. \tag{10.41}$$

To find a solution for eqn. (10.40), the inverse of the global stiffness matrix must be calculated. When the boundary conditions have been applied correctly, the global stiffness matrix will be non-singular, which is a necessary condition for a matrix to have an inverse. Calculating the inverse of a matrix is time consuming. Once the unknown displacements are calculated, the unknown reaction forces can be calculated by using the following two equations

$$\begin{bmatrix} K^1_{12} & K^1_{14} & 0 \\ K^3_{32} & 0 & K^3_{35} \end{bmatrix} \begin{Bmatrix} d_2 \\ d_4 \\ d_5 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_3 \end{Bmatrix}. \tag{10.42}$$

After all displacements are calculated, one can in turn calculate the strains with eqn. (10.7) and the stresses with eqn. (10.9).

10.8 Finite element model of truss structure

Fig. 10.5 shows an inclined bar or truss element. The truss element is characterized by a certain inclination β, a cross-sectional area A, a length L and a Young's modulus E. The length of the truss can be

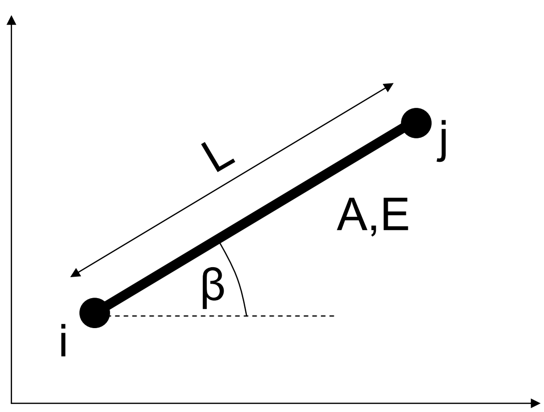

Figure 10.5: A truss element.

calculated as

$$L = \sqrt{\left(x_j - x_i\right)^2 + \left(y_j - y_i\right)^2}. \tag{10.43}$$

The individual stiffness matrix of a truss element is given by

$$[K^e] = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{10.44}$$

where $c = \cos\beta$ and $s = \sin\beta$.

Example 10.8.1. Figure 10.6 shows a truss structure, consisting of three elements. All angles are 45 or 90 degrees and P represents a load of 50 kg. The length of truss 2 and 3 is 1 m, the length of truss 1 is 1.41 m ($\sqrt{2}$ m). All three trusses have a diameter of 20 mm and are made of cast iron ($E = 180$ GPa). Verify the displacement of point 3, where the load is attached. The individual stiffness matrix for element

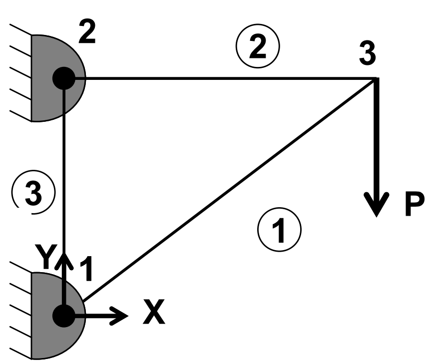

Figure 10.6: Truss structure.

1, 2 and 3 is given by K_1, K_2, and K_3:

$$K_1 = \begin{bmatrix} \frac{k_1}{2} & \frac{k_1}{2} & -\frac{k_1}{2} & -\frac{k_1}{2} \\ \frac{k_1}{2} & \frac{k_1}{2} & -\frac{k_1}{2} & -\frac{k_1}{2} \\ -\frac{k_1}{2} & -\frac{k_1}{2} & \frac{k_1}{2} & \frac{k_1}{2} \\ -\frac{k_1}{2} & -\frac{k_1}{2} & \frac{k_1}{2} & \frac{k_1}{2} \end{bmatrix}, \tag{10.45}$$

$$K_2 = \begin{bmatrix} k_2 & 0 & -k_2 & 0 \\ 0 & 0 & 0 & 0 \\ -k_2 & 0 & k_2 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}, \tag{10.46}$$

and

$$K_3 = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & k_3 & 0 & -k_3 \\ 0 & 0 & 0 & 0 \\ 0 & -k_3 & 0 & k_3 \end{bmatrix}, \tag{10.47}$$

where $k_1 = \frac{EA_1}{L_1}$, $k_2 = \frac{EA_2}{L_2}$ and $k_3 = \frac{EA_3}{L_3}$. Hence, the individual stiffness equations for element 1 can be written as

$$\begin{bmatrix} \frac{k_1}{2} & \frac{k_1}{2} & 0 & 0 & -\frac{k_1}{2} & -\frac{k_1}{2} \\ \frac{k_1}{2} & \frac{k_1}{2} & 0 & 0 & -\frac{k_1}{2} & -\frac{k_1}{2} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{k_1}{2} & -\frac{k_1}{2} & 0 & 0 & \frac{k_1}{2} & \frac{k_1}{2} \\ -\frac{k_1}{2} & -\frac{k_1}{2} & 0 & 0 & \frac{k_1}{2} & \frac{k_1}{2} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ 0 \\ 0 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ 0 \\ 0 \\ 0 \\ -P \end{Bmatrix}, \tag{10.48}$$

since $F_{x3} = 0$ and $F_{y3} = -P$ from the loading on node 3. For element 2,

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & k_2 & 0 & -k_2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -k_2 & 0 & k_2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F_{x2} \\ F_{y2} \\ 0 \\ 0 \end{Bmatrix}, \tag{10.49}$$

and, for element 3,

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & k_3 & 0 & -k_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -k_3 & 0 & k_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ 0 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ 0 \\ 0 \end{Bmatrix}, \tag{10.50}$$

where u_i is a displacement in the x-direction and v_i is a displacement in the y-direction, respectively. Notice that because P is present for element 1, we do not include it a second time for element 3.The global stiffness equation can be found by summing the contributions of the individual stiffness equations. Thus,

$$\begin{bmatrix} \frac{k_1}{2} & \frac{k_1}{2} & 0 & 0 & -\frac{k_1}{2} & -\frac{k_1}{2} \\ \frac{k_1}{2} & \frac{k_1}{2}+k_3 & 0 & -k_3 & -\frac{k_1}{2} & -\frac{k_1}{2} \\ 0 & 0 & k_2 & 0 & -k_2 & 0 \\ 0 & -k_3 & 0 & k_3 & 0 & 0 \\ -\frac{k_1}{2} & -\frac{k_1}{2} & -k_2 & 0 & k_2+\frac{k_1}{2} & \frac{k_1}{2} \\ -\frac{k_1}{2} & -\frac{k_1}{2} & 0 & 0 & \frac{k_1}{2} & \frac{k_1}{2} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ 0 \\ -P \end{Bmatrix}. \tag{10.51}$$

The trusses are clamped at nodes 1 and 2 (Fig. 10.6). Hence, $u_1 = u_2 = v_1 = v_2 = 0$. Notice that if you know the values of the displacements, you will have to find the associated forces to enforce those displacements. Likewise, if you know the forces applied, you will have to find the displacements those forces cause. Implementing the boundary conditions in the matrix produces

$$\begin{bmatrix} k_2 + \frac{k_1}{2} & \frac{k_1}{2} \\ \frac{k_1}{2} & \frac{k_1}{2} \end{bmatrix} \begin{Bmatrix} u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -P \end{Bmatrix}. \tag{10.52}$$

Before solving this set of two equations and two unknowns we will determine the numerical values of k_1, k_2 and k_3

$$A_1 = A_2 = A_3 = \frac{\pi D^2}{4} = \frac{\pi\ 0.02^2}{4} = 3.14 \times 10^{-4} \left[m^2\right], \tag{10.53}$$

so

$$k_1 = \frac{EA_1}{L_1} = \frac{180E9 \times 3.14 \times 10^{-4}}{\sqrt{2}} = 3.996 \times 10^7 \left[\frac{N}{m}\right], \tag{10.54}$$

$$k_2 = \frac{EA_2}{L_2} = \frac{180E9 \times 3.14 \times 10^{-4}}{1} = 5.652 \times 10^7 \left[\frac{N}{m}\right], \tag{10.55}$$

and

$$k_3 = \frac{EA_3}{L_3} = \frac{180E9 \times 3.14 \times 10^{-4}}{1} = 5.652 \times 10^7 \left[\frac{N}{m}\right]. \tag{10.56}$$

Substituting,

$$10^7 \times \begin{bmatrix} 7.65 & 1.998 \\ 1.998 & 1.998 \end{bmatrix} \begin{Bmatrix} u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -500 \end{Bmatrix} \tag{10.57}$$

produces

$$u_3 = 8.84 \times 10^{-6}\,[m] \tag{10.58}$$

and

$$v_3 = -33.8 \times 10^{-6}\,[m]. \tag{10.59}$$

10.9 *Finite element model of a truss structure with internal forces*

Previously, truss problems with only external forces have been studied. However, forces due to truss weight and due to thermal expansion are also important and could be taken into account. Figure 10.7 shows the same truss as shown in Fig. 10.5, but now the weight w of the truss has been included. The weight acts in the centroidal point of the truss and needs to be distributed over the nodes, as indicated in the figure. A load of $w/2$ has been added in each node, to account for the weight of the truss (presuming a uniform weight distribution).

In matrix notation this produces

$$\boldsymbol{w}_w^e = \frac{w}{2}\begin{Bmatrix} 0 & -1 & 0 & -1 \end{Bmatrix}^t. \tag{10.60}$$

Forces due to thermal heating, acting on a fully restrained truss can be expressed as

$$\boldsymbol{w}_T^e = \alpha EAT \begin{Bmatrix} -c & -s & c & s \end{Bmatrix}^t \tag{10.61}$$

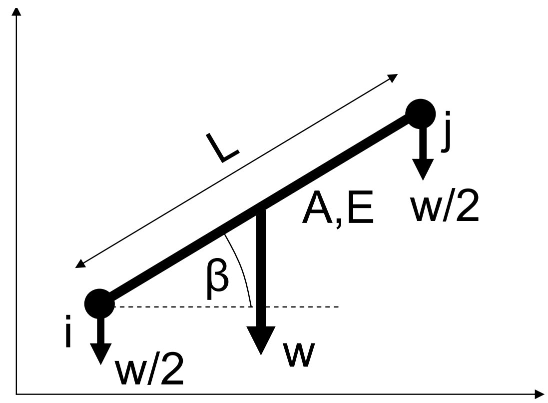

Figure 10.7: Truss element with internal forces.

where α is the thermal expansion coefficient, E is the Young's modulus of the material of the truss, A is the cross-sectional area of the truss and T is the temperature. The forces $[K]\,\boldsymbol{d} = \boldsymbol{F}$, are produced by nodal displacements and are defined as forces applied by nodes to an element. Equal and opposite forces are applied by the element to the nodes. Hence,

$$\boldsymbol{w}^{\boldsymbol{e}}_{\boldsymbol{net}} = -[K^e]\,\boldsymbol{d}^{\boldsymbol{e}} + \boldsymbol{w}^{\boldsymbol{e}}_{w} + \boldsymbol{w}^{\boldsymbol{e}}_{\boldsymbol{T}}. \tag{10.62}$$

Structural equilibrium conditions are written for each node, i.e.,

$$\sum \boldsymbol{F}^i = \boldsymbol{0} \tag{10.63}$$

which can be expressed in the x and y directions. Loads consist of externally applied forces $\boldsymbol{P}$ and forces $\boldsymbol{w}_{\boldsymbol{net},i}$ from each truss. Thus,

$$\boldsymbol{P} + \sum_{\boldsymbol{i}=\boldsymbol{1}}^{N} \boldsymbol{w}^{\boldsymbol{e}}_{\boldsymbol{net},i} = 0 \tag{10.64}$$

with N defined as the number of elements. Substituting eqn. (10.62) into eqn. (10.64) produces

$$\boldsymbol{P} + \sum_{\boldsymbol{i}=\boldsymbol{1}}^{N} (-[K^e]\,\boldsymbol{d}^{\boldsymbol{e}} + \boldsymbol{w}^{\boldsymbol{e}}_{w} + \boldsymbol{w}^{\boldsymbol{e}}_{\boldsymbol{T}})_i = 0 \tag{10.65}$$

Since

$$\sum_{\boldsymbol{i}=\boldsymbol{1}}^{N} ([K^e]\,\boldsymbol{d}^{\boldsymbol{e}})_i = \boldsymbol{F}, \tag{10.66}$$

eqn. (10.65) gives

$$\boldsymbol{P} + \sum_{\boldsymbol{i}=\boldsymbol{1}}^{N} (\boldsymbol{w}^{\boldsymbol{e}}_{w} + \boldsymbol{w}^{\boldsymbol{e}}_{\boldsymbol{T}})_i = \boldsymbol{F}. \tag{10.67}$$

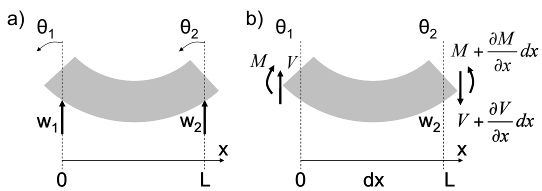

Figure 10.8: Beam of length L.

10.10 *Finite element model of a beam*

A typical beam of length L is shown in Fig. 10.8; Fig. 10.8(a) shows the boundary conditions, while Fig. 10.8(b) shows the free body diagram. The differential equation for beam deflection w is given by

$$\frac{d^2}{dx^2}\left(EI\frac{d^2w(x)}{dx^2}\right) = 0 \tag{10.68}$$

for $0 < x < L$. After integration with respect to x this gives

$$w(x) = C_1x^3 + C_2x^2 + C_3x + C_4. \tag{10.69}$$

The integration constants can be found from the boundary conditions, indicated in Fig. 10.8, i.e.,

$$w(0) = w_1, \tag{10.70}$$

$$w(L) = w_2, \tag{10.71}$$

$$\left.\frac{dw}{dx}\right|_{x=0} = \theta_1, \tag{10.72}$$

and

$$\left.\frac{dw}{dx}\right|_{x=L} = \theta_2. \tag{10.73}$$

Hence,

$$w(0) = w_1 = C_4, \tag{10.74}$$

$$\left.\frac{dw}{dx}\right|_{x=0} = \theta_1 = C_3, \tag{10.75}$$

$$w(L) = C_1L^3 + C_2L^2 + C_3L + C_4 = w_2, \tag{10.76}$$

and

$$\left.\frac{dw}{dx}\right|_{x=L} = 3C_1L^2 + 2C_2L + C_3 = \theta_2. \tag{10.77}$$

The integration constants can be written as

$$C_1 = \frac{1}{L^3}\left(2w_1 + L\theta_1 - 2w_2 + L\theta_2\right), \tag{10.78}$$

$$C_2 = \frac{1}{L^2}\left(-3w_1 - 2L\theta_1 + 3w_2 - L\theta_2\right), \tag{10.79}$$

$$C_3 = \theta_1, \tag{10.80}$$

and

$$C_4 = w_1. \tag{10.81}$$

One can now write the deflection of a beam in terms of the nodal deflections as

$$w(x) = \left[\frac{1}{L^3}\left(2w_1 + L\theta_1 - 2w_2 + L\theta_2\right)\right]x^3 + \left[\frac{1}{L^2}\left(-3w_1 - 2L\theta_1 + 3w_2 - L\theta_2\right)\right]x^2 + \theta_1 x + w_1. \tag{10.82}$$

After rearranging this gives

$$\begin{aligned} w(x) = &\left[1 - 3\left(\tfrac{x}{L}\right)^2 + 2\left(\tfrac{x}{L}\right)^3\right]w_1 + \left[\tfrac{x}{L} - 2\left(\tfrac{x}{L}\right)^2 + \left(\tfrac{x}{L}\right)^3\right]L\theta_1 \\ &+ \left[3\left(\tfrac{x}{L}\right)^2 - 2\left(\tfrac{x}{L}\right)^3\right]w_2 + \left[-\left(\tfrac{x}{L}\right)^2 + \left(\tfrac{x}{L}\right)^3\right]L\theta_2 \end{aligned} \tag{10.83}$$

or

$$w(x) = N_1(x)\,w_1 + N_2(x)\,L\theta_1 + N_3(x)\,w_2 + N_4(x)\,L\theta_2, \tag{10.84}$$

where $N_i(x)$ are the shape functions that describe how to interpolate for displacements within the elements, if the displacements at the nodes are known. In matrix notation, eqn. (10.84) can be written as

$$w(x) = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix}\left\{ w_1 \quad L\theta_1 \quad w_2 \quad L\theta_2 \right\}^t \tag{10.85}$$

where

$$\boldsymbol{d^{et}} = \left\{ w_1 \quad L\theta_1 \quad w_2 \quad L\theta_2 \right\}^t \tag{10.86}$$

is the nodal displacement vector. The nodal forces are

$$\boldsymbol{f^t} = \left\{ f_1 \quad f_2 \quad f_3 \quad f_4 \right\}^t, \tag{10.87}$$

and

$$\boldsymbol{f^t} = \left\{ V|_{x=0} \quad M|_{x=0} \quad V|_{x=L} \quad M|_{x=L} \right\}^t, \tag{10.88}$$

where M is the bending moment and $V = -\partial M/\partial x$ is the shear force. The bending displacements are related to the nodal forces f_1, f_2, f_3, and f_4 by

$$f_1 = EI\left.\frac{d^3w}{dx^3}\right|_{x=0}, \tag{10.89}$$

$$f_2 = -EI\left.\frac{d^2w}{dx^2}\right|_{x=0}, \tag{10.90}$$

$$f_3 = -EI\left.\frac{d^3w}{dx^3}\right|_{x=L}, \tag{10.91}$$

and

$$f_4 = EI \left. \frac{d^2 w}{dx^2} \right|_{x=L}. \tag{10.92}$$

Thus, using eqn. (10.83) in eqn. (10.89) through eqn. (10.92) one finds the nodal forces in terms of nodal displacements as

$$f_1 = \frac{EI}{L^3} \left(12 w_1 + 6L\theta_1 - 12 w_2 + 6L\theta_2 \right), \tag{10.93}$$

$$f_2 = \frac{EI}{L^2} \left(6 w_1 + 4L\theta_1 - 6 w_2 + 2L\theta_2 \right), \tag{10.94}$$

$$f_3 = \frac{EI}{L^3} \left(-12 w_1 - 6L\theta_1 + 12 w_2 - 6L\theta_2 \right), \tag{10.95}$$

and

$$f_4 = \frac{EI}{L^2} \left(6 w_1 + 2L\theta_1 - 6 w_2 + 4L\theta_2 \right). \tag{10.96}$$

Rewriting eqns. (10.93) through (10.96) in matrix notation, one finds

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix} = \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & \frac{-12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & \frac{-6EI}{L^2} & \frac{2EI}{L} \\ \frac{-12EI}{L^3} & \frac{-6EI}{L^2} & \frac{12EI}{L^3} & \frac{-6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & \frac{-6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{Bmatrix}, \tag{10.97}$$

which may be written as

$$\boldsymbol{f} = [K^e] \, \boldsymbol{d}^e. \tag{10.98}$$

10.11 *Transferring the load to the nodes*

In the finite element method, all loads need to be defined at the nodes. Fig. 10.9 shows three different cases where the load is not defined at the nodes and hence, needs to be transferred to the nodes. Fig. 10.9(a) shows the case of a distributed load q, Fig. 10.9(b) shows the case of a point load P acting in the middle of the beam and Fig. 10.9(c) shows the case of a moment M acting in the middle of the beam.

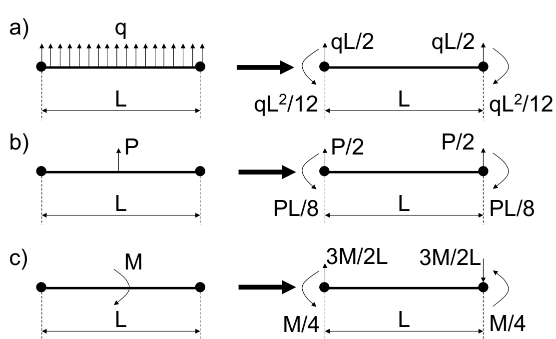

Figure 10.9: Transferring load to nodes.

10.12 *Finite element model of a hard disk drive suspension with beam elements*

The suspension spring in a magnetic hard disk drive is an excellent example for the use of finite element analysis. The suspension spring is a flat spring, tapered from the base, with flanges protruding on either side, as shown in Fig. 10.10. The suspension beam can be

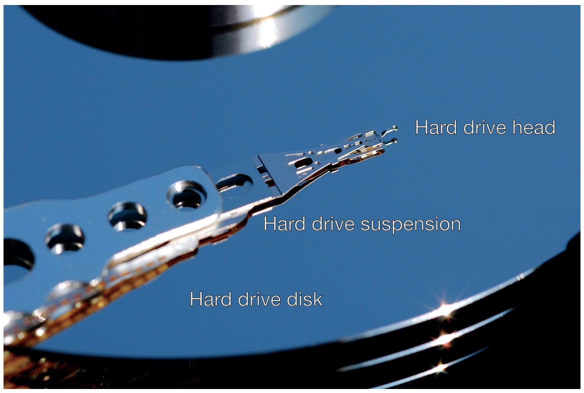

Figure 10.10: Hard disk drive read/write head suspension, indicating the head and one of the several disks in the platter.

approximated by two beam elements of constant cross section; A_1 and A_2, respectively, as shown in Fig. 10.11. To find the displacement at

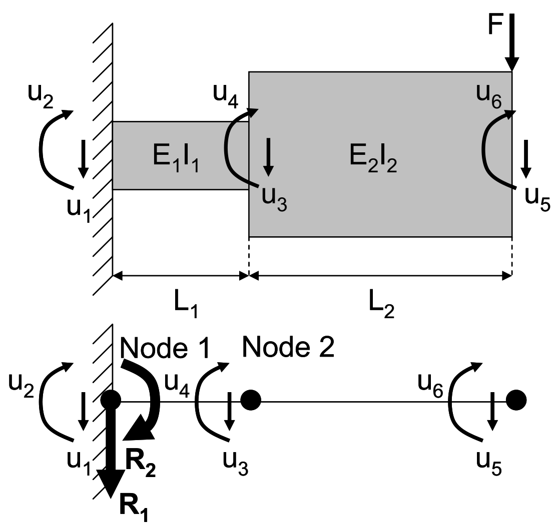

Figure 10.11: FEM model of hard disk drive suspension. Here we show the case where $E_1 \neq E_2$, but we assume in the text that $E_1 = E_2 = E$.

each node using finite element analysis, one needs to

1. assemble the individual stiffness matrices $[K_1]$ and $[K_2]$ for elements 1 and 2
2. assemble the overall stiffness matrix,
3. write down the displacement vector,

4. write down the load vector, and

5. apply boundary conditions and invert the matrix equation $[K]\boldsymbol{u} = \boldsymbol{f}$, to solve for the displacements.

10.12.1 *Assemble the individual stiffness matrices $[K_1]$ and $[K_2]$*

First, assemble the individual stiffness matrices k_1 and k_2 for element 1 and 2, respectively:

$$[K_1] = \begin{bmatrix} \frac{12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} & \frac{-12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} \\ \frac{6EI_1}{L_1^2} & \frac{4EI_1}{L_1} & \frac{-6EI_1}{L_1^2} & \frac{2EI_1}{L_1} \\ \frac{-12EI_1}{L_1^3} & \frac{-6EI_1}{L_1^2} & \frac{12EI_1}{L_1^3} & \frac{-6EI_1}{L_1^2} \\ \frac{6EI_1}{L_1^2} & \frac{2EI_1}{L_1} & \frac{-6EI_1}{L_1^2} & \frac{4EI_1}{L_1} \end{bmatrix} \tag{10.99}$$

and

$$[K_2] = \begin{bmatrix} \frac{12EI_2}{L_2^3} & \frac{6EI_2}{L_2^2} & \frac{-12EI_2}{L_2^3} & \frac{6EI_2}{L_2^2} \\ \frac{6EI_2}{L_2^2} & \frac{4EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{2EI_2}{L_2} \\ \frac{-12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} & \frac{12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} \\ \frac{6EI_2}{L_2^2} & \frac{2EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{4EI_2}{L_2} \end{bmatrix}. \tag{10.100}$$

Matrix $[K_1]$ is associated with the degrees of freedom u_1, u_2, u_3, u_4, while matrix $[K_2]$ is associated with the degrees of freedom u_3, u_4, u_5, u_6.

10.12.2 *Assemble the overall stiffness matrix $[K]$*

This is done by "expanding to size" the individual stiffness matrices $[K_1]$ and $[K_2]$, similar to the process of "expanding to size" performed previously with the truss elements of the truss structure:

$$[K] = \begin{bmatrix} \frac{12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} & \frac{-12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} & 0 & 0 \\ \frac{6EI_1}{L_1^2} & \frac{4EI_1}{L_1} & \frac{-6EI_1}{L_1^2} & \frac{2EI_1}{L_1} & 0 & 0 \\ \frac{-12EI_1}{L_1^3} & \frac{-6EI_1}{L_1^2} & \frac{12EI_1}{L_1^3} + \frac{12EI_2}{L_2^3} & \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{-12EI_2}{L_2^3} & \frac{6EI_2}{L_2^2} \\ \frac{6EI_1}{L_1^2} & \frac{2EI_1}{L_1} & \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{4EI_1}{L_1} + \frac{4EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{2EI_2}{L_2} \\ 0 & 0 & \frac{-12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} & \frac{12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} \\ 0 & 0 & \frac{6EI_2}{L_2^2} & \frac{2EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{4EI_2}{L_2} \end{bmatrix}. \tag{10.101}$$

10.12.3 *Write down the displacement vector*

The displacement vector can be expressed as a column vector

$$\left\{ u_1 \quad u_2 \quad u_3 \quad u_4 \quad u_5 \quad u_6 \right\}^t. \tag{10.102}$$

10.12.4 *Write down the load vector*

Similarly, the load vector can be written as a column vector

$$\left\{ R_1 \quad R_2 \quad R_3 \quad R_4 \quad R_5 \quad R_6 \right\}^t = \left\{ R_1 \quad R_2 \quad 0 \quad 0 \quad F \quad 0 \right\}^t, \tag{10.103}$$

where R_1 is an unknown reaction force while R_2 is an unknown reaction moment, and F is a known load.

10.12.5 *Assemble the equations and apply the boundary conditions*

Assembling the equilibrium equation for the beam produces

$$\begin{bmatrix} \frac{12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} & \frac{-12EI_1}{L_1^3} & \frac{6EI_1}{L_1^2} & 0 & 0 \\ \frac{6EI_1}{L_1^2} & \frac{4EI_1}{L_1} & \frac{-6EI_1}{L_1^2} & \frac{2EI_1}{L_1} & 0 & 0 \\ \frac{-12EI_1}{L_1^3} & \frac{-6EI_1}{L_1^2} & \frac{12EI_1}{L_1^3} + \frac{12EI_2}{L_2^3} & \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{-12EI_2}{L_2^3} & \frac{6EI_2}{L_2^2} \\ \frac{6EI_1}{L_1^2} & \frac{2EI_1}{L_1} & \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{4EI_1}{L_1} + \frac{4EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{2EI_2}{L_2} \\ 0 & 0 & \frac{-12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} & \frac{12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} \\ 0 & 0 & \frac{6EI_2}{L_2^2} & \frac{2EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{4EI_2}{L_2} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ R_2 \\ 0 \\ 0 \\ F \\ 0 \end{Bmatrix}. \tag{10.104}$$

The next step is to apply the boundary conditions. We know that the suspension is clamped at one end. Hence, $u_1 = u_2 = 0$, since no displacement and rotation can occur at node 1. These boundary conditions are equivalent to deleting rows 1 and 2 and columns 1 and 2 from the equilibrium equations. This results in

$$\begin{bmatrix} \frac{12EI_1}{L_1^3} + \frac{12EI_2}{L_2^3} & \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{-12EI_2}{L_2^3} & \frac{6EI_2}{L_2^2} \\ \frac{-6EI_1}{L_1^2} + \frac{6EI_2}{L_2^2} & \frac{4EI_1}{L_1} + \frac{4EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{2EI_2}{L_2} \\ \frac{-12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} & \frac{12EI_2}{L_2^3} & \frac{-6EI_2}{L_2^2} \\ \frac{6EI_2}{L_2^2} & \frac{2EI_2}{L_2} & \frac{-6EI_2}{L_2^2} & \frac{4EI_2}{L_2} \end{bmatrix} \begin{Bmatrix} u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F \\ 0 \end{Bmatrix}. \tag{10.105}$$

Equation (10.105) can be rewritten in matrix form as

$$[K]\, \boldsymbol{u} = \boldsymbol{f}. \tag{10.106}$$

Left multiplying both sides of eqn. (10.7) with $[K]^{-1}$ produces the displacements at the nodes

$$\boldsymbol{u} = [K]^{\mathbf{-1}} \boldsymbol{f}. \tag{10.107}$$

10.13 *Modal analysis*

In many engineering problems it is important to determine the displacements of a structure as a result of vibrations. Therefore, it is necessary to determine the eigenfrequencies of a structure, including the shapes of the eigenmodes associated with those eigenfrequencies.

To calculate the eigenmodes of a structure, we have to solve

$$[M]\ddot{\boldsymbol{x}} + [K]\boldsymbol{x} = 0 \tag{10.108}$$

where

$$x = A \sin \omega t. \tag{10.109}$$

Then

$$\dot{x} = A\omega \cos \omega t \tag{10.110}$$

and

$$\ddot{x} = -A\omega^2 \sin \omega t = -\omega^2 x. \tag{10.111}$$

Equation (10.108) can be rewritten as

$$\left(-\omega^2[M] + [K]\right)\boldsymbol{x} = 0 \tag{10.112}$$

or, after multiplying eqn. (10.112) with $[M]^{-1}$,

$$\left\{[M]^{-1}[K] - \omega^2[I]\right\}\boldsymbol{x} = 0. \tag{10.113}$$

Equation (10.113) is an eigenvalue problem. The solution can be found by calculating the determinant $\left|[M]^{-1}[K] - \omega^2[I]\right| = 0$.

Example 10.13.1. Solve the eigenvalue problem given by

$$\left|A - \omega^2 I\right| = 0, \tag{10.114}$$

where $A = \begin{bmatrix} -2 & 2 & 3 \\ 2 & 1 & -6 \\ -1 & -2 & 0 \end{bmatrix}$ and

$$\omega^2 I = \begin{bmatrix} \omega^2 & 0 & 0 \\ 0 & \omega^2 & 0 \\ 0 & 0 & \omega^2 \end{bmatrix}. \tag{10.115}$$

Calculating the determinant gives

$$\left|A - \omega^2 I\right| = \begin{vmatrix} -2-\omega^2 & 2 & 3 \\ 2 & 1-\omega^2 & -6 \\ -1 & -2 & -\omega^2 \end{vmatrix} = -\left(\omega^2\right)^3 - \left(\omega^2\right)^2 + 15\omega^2 + 27 = 0. \tag{10.116}$$

From eqn. (10.116), one finds that ω_1^2 = -2.16, ω_2^2 = 4.16, ω_3^2 = -3. In eqn. (10.113), $[K]$ is known. The question now is how to determine the mass matrix $[M]$. Several approaches exist to determine the mass matrix of an element, and subsequently, the overall mass matrix.

The mass matrix formulation

$$[M]_{el} = \frac{m}{2}\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \tag{10.117}$$

neglects the rotary inertia of the mass at the nodes. Other formulations for the mass matrix include

$$[M]_{el} = \frac{m}{2}\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & L^2/39 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & L^2/39 \end{bmatrix} \tag{10.118}$$

and

$$[M]_{el} = \frac{m}{2}\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & L^2/12 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & L^2/12 \end{bmatrix}. \tag{10.119}$$

The mass matrices eqn. (10.118) and eqn. (10.119) account for rotary inertia of a mass and, thus, are more accurate than the formulation without rotary inertia. Using eqn. (10.117) for the overall mass matrix of the hard disk drive suspension, we obtain the so-called "lumped mass matrix"

$$[M] = \begin{bmatrix} \frac{m_1}{2} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \frac{m_1+m_2}{2} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{m_2}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}. \tag{10.120}$$

10.14 *Derivation of finite element analysis equations using assumed modes or shape functions*

So how does one perform the derivations from first principles, and determine quantities like the mass matrix, $[M]$ as provided by eqns. (10.117) through (10.119)? We explore this in more detail here.

We make an assumption for the displacement

$$d(x,t) = \sum_{i=1}^{N} \psi_i(x) D(t), \tag{10.121}$$

an assumption of the ability to separate the dependence of d on x and t into two functions $\psi_i(x)$ and $D(t)$, each independently a function of the position x and time t.

Each $\psi_i(x)$ represents a deflected shape of the structure, the i^{th} one of N in total. With FEA, we assume our structure above to be a small part of the overall structure, an *element* in the system.

10.14.1 Axial motion

We could treat axial motion, for example, in a truss element:

$$u(x,t) = \psi_1(x)u_1(t) + \psi_2(x)u_2(t). \tag{10.122}$$

Since $u(0,t) = u_1(t)$ and $u(L,t) = u_2(t)$, the *shape functions* (or *assumed modes*) $\psi_1(x)$ and $\psi_2(x)$ must satisfy the boundary conditions

$$\psi_1(0) = 1, \psi_1(L) = 0 \quad \text{and} \quad \psi_2(0) = 1, \psi_2(L) = 1. \tag{10.123}$$

We can determine what the shape functions ψ_i are if we determine what the deformation would be for the physical structure that connects 1 and 2. The vibration/deformation equation for a bar in axial loading is

$$\frac{\partial}{\partial x}\left(AE\frac{\partial u}{\partial x}\right) + p(x,t) = \rho A \frac{\partial^2 u}{\partial t^2}, \tag{10.124}$$

where A, E, $p(x,t)$, ρ, and u are the cross-sectional area of the bar, its Young's modulus, the axial load upon it per unit length, its density, and its axial displacement, respectively. This equation can be derived from Newton's second law on an infinitesimal slice of the bar or by Hamilton's principle (usually taught in a simple fashion as Lagrange's equations).

If we assume no applied load and no dynamics, then eqn. (10.124) becomes

$$\frac{\partial}{\partial x}\left(AE\frac{\partial u}{\partial x}\right) = 0. \tag{10.125}$$

For a uniform element, EA is a constant. So $u(x,t) = C_1 + C_2(x/L)$ via integration and division by L so that C_1 and C_2 have the same dimension $[L]$. Now $u(0,t) = u_1(t)$ and $u(L,t) = u_2(t)$, so $C_1 = u_1(t)$ while $C_2 = u_2(t) - u_1(t)$, giving

$$u(x,t) = u_1(t) + [u_2 - u_1](x/L) = \underbrace{\left(1 - \frac{x}{L}\right)}_{\psi_1(t)} u_1(t) + \underbrace{\left(\frac{x}{L}\right)}_{\psi_2(t)} u_2(t). \tag{10.126}$$

Then $\psi_1(x) = \left(1 - \frac{x}{L}\right)$ and $\psi_2(x) = \left(\frac{x}{L}\right)$ for our shape functions.

We can write the strain energy within the bar due to its deformation, $V_{\text{total}} = \sum_i V_i$;

$$V_i = \frac{1}{2}\int_0^L EA(u_i')^2\,dx \tag{10.127}$$

from

$$V_i \equiv \frac{1}{2}\int_V \varepsilon_i\sigma_i\,dV = \frac{1}{2}\int_V E\varepsilon_i^2\,dV = \frac{1}{2}\int_V E\left(\frac{\partial u_i}{\partial x}\right)^2 dV = \frac{1}{2}\int_0^L EA\left(\frac{\partial u_i}{\partial x}\right)^2 dx. \tag{10.128}$$

Likewise, the kinetic energy within the bar is $T_{\text{total}} = \sum_i T_i$;

$$T_i \equiv \frac{1}{2}\int_V \rho\,(\dot{u}_i)^2\,dV = \frac{1}{2}\int_0^L \rho A\,(\dot{u}_i)^2\,dx. \tag{10.129}$$

Since $u(x,t) = \psi_1(x)u_1(t) + \psi_2(x)u_2(t)$, $u'(x,t) = \psi_1'(x)u_1(t) + \psi_2'(x)u_2(t)$ and $\dot{u}(x,t) = \psi_1(x)\dot{u}_1(t) + \psi_2(x)\dot{u}_2(t)$. Substituting into the strain and kinetic energy expressions, eqns. (10.127) and (10.129),

$$\begin{aligned} V =& \frac{1}{2}\int_0^L EA\left(\sum_{i=1}^{2}\psi_i'(x)u_i(t)\right)^2 dx = \\ & \frac{1}{2}\int_0^L EA\left[\sum_{i=1}^{2}\sum_{j=1}^{2}\psi_i'(x)\psi_j'(x)u_i(t)u_j(t)\right]dx = \\ & \frac{1}{2}\sum_{i=1}^{2}\sum_{j=1}^{2}K_{ij}u_i(t)u_j(t) \end{aligned} \tag{10.130}$$

where $K_{ij} = \int_0^L EA\psi_i'(x)\psi_j'(x)\,dx$: the stiffness matrix terms. We can write this in matrix form:

$$V = \frac{1}{2}[u]^t[K][u] = \frac{1}{2}\begin{bmatrix} u_1 & u_2 \end{bmatrix}\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix}, \tag{10.131}$$

a quadratic form.

In a similar fashion,

$$T = \frac{1}{2}\int_0^L \rho A\,(\dot{u})^2\,dx = \frac{1}{2}\int_0^L \rho A\left(\sum_{i=1}^{2}\psi_i(x)\dot{u}_i(t)\right)^2 dx = \tag{10.132}$$

$$\frac{1}{2}\int_0^L \rho A\left[\sum_{i=1}^{2}\sum_{j=1}^{2}\psi_i(x)\psi_j(x)\dot{u}_i(t)\dot{u}_j(t)\right]dx = \tag{10.133}$$

$$\sum_{i=1}^{2}\sum_{j=1}^{2}M_{ij}\dot{u}_i(t)\dot{u}_j(t), \tag{10.134}$$

where $M_{ij} = \int_0^L \rho A\psi_i(x)\psi_j(x)\,dx$ represent terms of the *mass matrix*. Then $T = \frac{1}{2}[\dot{u}]^t[M][\dot{u}]$ as another quadratic form. *This* is how one determines the appropriate values for the mass matrix.

To handle applied forces on the bar, we use virtual work,

$$W = \int_0^L p(x,t)u(x,t)\,dx \tag{10.135}$$

as the work done by applied forces $p(x,t)$ on the bar. Taking the first variation of the work,

$$\delta W = \int_0^L p(x,t)\delta(u(x,t))\,dx = \int_0^L p(x,t)\sum_{i=1}^{2}\psi_i(x)\delta u_i(t)\,dx. \quad (10.136)$$

Then

$$\delta W = \sum_{i=1}^{2}\int_0^L p(x,t)\psi_i(x)\delta u_i(t)\,dx = \sum_{i=1}^{2}\mathbb{P}_i\delta u_i(t) \quad (10.137)$$

where $\mathbb{P}_i = \int_0^L p(x,t)\psi_i(x)\,dx$.

10.14.2 *Finding the equations of motion for the axial motion of a bar*

So how did we find the equations of motion (10.124) for the axial motion of a bar in the first place? We can find it using Lagrange's equations:

$$\frac{d}{dt}\left(\frac{\partial\mathcal{L}}{\partial\dot{q}_i}\right) - \frac{\partial\mathcal{L}}{\partial q_i} = Q_i \quad \text{where} \quad \mathcal{L} = T - V. \quad (10.138)$$

Substituting for T and V, and noting that $Q_i = \mathbb{P}_i$ according to the derivation of Lagrange's equations,

$$\sum_{j=1}^{2} m_{ij}\ddot{u}_j + \sum_{j=1}^{2} k_{ij}u_j = \mathbb{P}_i \quad (10.139)$$

for $i \in \{1, 2, \ldots, N\}$. This, in matrix form, becomes

$$[M]\,[\ddot{u}] + [K]\,[u] = [\mathbb{P}]. \quad (10.140)$$

Now $K_{ij} = \int_0^L EA\psi_i'(x)\psi_j'(x)\,dx$ produces $K_{11} = \int_0^L EA\psi_1'(x)\psi_1'(x)\,dx = EA/L$. In this way, K_{21}, K_{12}, and K_{22} can be found, giving

$$[K] = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}. \quad (10.141)$$

Similarly,

$$[M] = \frac{\rho AL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}. \quad (10.142)$$

If we were to use the static equation of motion, $\frac{\partial}{\partial x}\left(AE\frac{\partial u}{\partial x}\right) = 0$, we could write $[K][u] = [\mathbb{P}]$.

10.14.3 *Flexural motion of a beam*

To find the shape functions for the bending motion of a Bernoulli-Euler beam, we define the transverse displacement of the beam as

$$w(x,t) = \sum_{i=1}^{4}\psi_i(x)w_i(t) \quad (10.143)$$

to eventually find four shape functions for the four degrees of freedom the beam element has: displacement and slope at each end. The boundary conditions are

$$\psi_1(0) = 1; \psi_1'(0) = \psi_1(L) = \psi_1'(L) = 0 \tag{10.144}$$

$$\psi_2'(0) = 1; \psi_2(0) = \psi_2(L) = \psi_2'(L) = 0 \tag{10.145}$$

$$\psi_3(L) = 1; \psi_3(0) = \psi_3'(0) = \psi_3'(L) = 0 \tag{10.146}$$

$$\psi_4'(L) = 1; \psi_4(0) = \psi_4'(0) = \psi_4(L) = 0. \tag{10.147}$$

The equation of motion is

$$\frac{\partial^2}{\partial x^2}\left(EI\frac{\partial^2 w}{\partial x^2}\right) = \rho A\frac{\partial^2 w}{\partial t^2} = 0 \tag{10.148}$$

if we assume the beam deformation is static. This has a general solution of $w(x) = C_1 + C_2(x/L) + C_3(x/L)^2 + C_4(x/L)^3$. Solving as before using the boundary conditions,

$$\psi_1(x) = 1 - 3\left(\frac{x}{L}\right)^2 + 2\left(\frac{x}{L}\right)^3 \tag{10.149}$$

$$\psi_2(x) = x - 2L\left(\frac{x}{L}\right)^2 + L\left(\frac{x}{L}\right)^3 \tag{10.150}$$

$$\psi_3(x) = 3\left(\frac{x}{L}\right)^2 - 2\left(\frac{x}{L}\right)^3 \tag{10.151}$$

$$\psi_4(x) = -L\left(\frac{x}{L}\right)^2 + L\left(\frac{x}{L}\right)^3. \tag{10.152}$$

Same as before,

$$K_{ij} = \int_0^L EI\psi_i''(x)\psi_j''(x)\,dx, \tag{10.153}$$

$$M_{ij} = \int_0^L \rho A\psi_i(x)\psi_j(x)\,dx, \text{and} \tag{10.154}$$

$$\mathbb{P}_i = \int_0^L p(x,t)\psi_i(x)\,dx. \tag{10.155}$$

We use the static equation $\frac{\partial^2}{\partial x^2}\left(EI\frac{\partial^2 w}{\partial x^2}\right) = 0$ to help us find the shape functions. Then

$$[K] = \left(\frac{EI}{L^3}\right)\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \tag{10.156}$$

and

$$[M] = \left(\frac{\rho AL}{420}\right)\begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix} \tag{10.157}$$

which can be used as before for static problems, $[K][u] = [\mathbb{P}]$, modal analysis $\left([M]^{-1}[K] - \omega^2[I]\right)[u] = 0$, harmonic analysis $[M]^{-1}[K] - \omega^2[I] = [M]^{-1}[\hat{\mathbb{P}}]$ where $\hat{\mathbb{P}}$ is harmonic in time: $\hat{\mathbb{P}} = \mathbb{P}e^{\imath\omega t}$.

10.14.4 *Torsion*

For torsion, we define the strain energy of a beam as

$$V = \frac{1}{2}\int_0^L GJ\,(\theta')^2\,dx, \tag{10.158}$$

where $\theta = \theta(x,t)$ as the torsion along the beam from $x = 0$ to $x = L$, and GJ represents the torsional stiffness; $J = I_p$, the latter being the polar moment of inertia only when the beam's cross-section is circular. For non-circular cross sections, the beam's centroid and the axis of twist do not coincide, and so the analysis requires the inclusion of bending with the torsion. For example, flutter of an airplane wing. Furthermore, with non-circular cross-sectioned beams, torsion gives rise to axial deformation as well, and this may be significant.[5]

[5] Daniel Kuang-Chen Liu, James Friend, and Leslie Yeo. The axial-torsional vibration of pretwisted beams. *Journal of Sound and Vibration*, 321(1-2):115–136, 2008

The kinetic energy of torsion of a beam is

$$T = \frac{1}{2}\int_0^L \rho I_p \left(\dot{\theta}\right)^2 dx. \tag{10.159}$$

We assume separation of variables is possible such that $\theta(x,t) = \psi_1(x)\theta_1(t) + \psi_2(x)\theta_2(t)$.

If we assume a static torsion problem, then $(GJ\theta')' = 0$ (notice the second prime as a derivative with respect to x for the expression $GJ\theta'$). The method is similar to the axial beam deformation problem, and in fact the equations are analogous, producing

$$K_{ij} = \int_0^L GJ\psi_i'\psi_j'\,dx \tag{10.160}$$

$$M_{ij} = \int_0^L \rho I_p \psi_i \psi_j\,dx \tag{10.161}$$

$$\mathbb{P}_i = \int_0^L T(x,t)\psi_i dx, \tag{10.162}$$

where $T(x,t)$ is a distributed torque per unit length on the beam. Substituting the shape functions from the axial beam deformation problem,

$$[K] = \left(\frac{GJ}{L}\right)\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \qquad [M] = \left(\frac{\rho I_p L}{6}\right)\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}. \tag{10.163}$$

The rest of the analysis proceeds in the same manner as with the axial beam deformation problem.

10.15 *Two dimensional FEM analysis*

10.15.1 *Triangular elements*

Figure 10.12 shows a plane constant-strain triangle. The triangle has six degrees of freedom. The shape functions (interpolation functions)

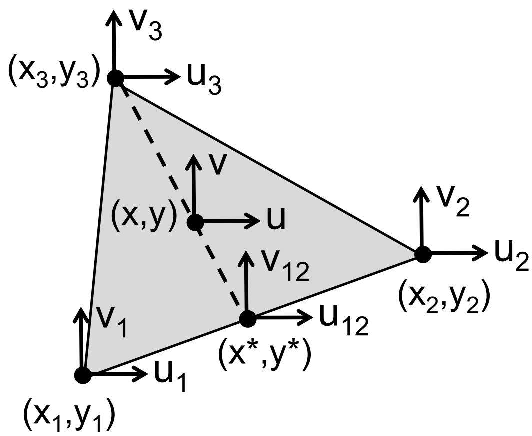

Figure 10.12: Plane, constant strain triangle.

defined at the nodes allows calculating the displacement at each point within the element:

$$u_{12} = \frac{1}{2}\left(\frac{x^* - x_2}{x_1 - x_2} + \frac{y^* - y_2}{y_1 - y_2}\right) u_1 + \frac{1}{2}\left(\frac{x^* - x_1}{x_2 - x_1} + \frac{y^* - y_1}{y_2 - y_1}\right) u_2. \quad (10.164)$$

Along the line (x^*, y^*) to (x_3, y_3) one can define

$$u = \frac{1}{2}\left(\frac{x - x_3}{x^* - x_3} + \frac{y - y_3}{y^* - y_3}\right) u_{12} + \frac{1}{2}\left(\frac{x - x^*}{x_3 - x^*} + \frac{y - y^*}{y_3 - y^*}\right) u_3. \quad (10.165)$$

Slope condition:

$$\frac{y^* - y_1}{x^* - x_1} = \frac{y_2 - y_1}{x_2 - x_1} \quad (10.166)$$

and

$$\frac{y - y_3}{x - x_3} = \frac{y^* - y_3}{x^* - x_3}. \quad (10.167)$$

Solving eqns. (10.165), (10.166), and (10.167), one obtains

$$u = \frac{p_1(x,y)\,u_1 + p_2(x,y)\,u_2 + p_3(x,y)\,u_3}{\Delta}, \quad (10.168)$$

where

$$p_1(x,y) = x_2 y_3 - x_3 y_2 + x(y_2 - y_3) + y(x_3 - x_2), \quad (10.169)$$

$$p_2(x,y) = x_3 y_1 - x_1 y_3 + x(y_3 - y_1) + y(x_1 - x_3), \quad (10.170)$$

$$p_3(x,y) = x_1y_2 - x_2y_1 + x(y_1 - y_2) + y(x_2 - x_1), \tag{10.171}$$

and

$$\Delta = x_2y_3 - x_1y_3 - x_3y_2 + x_1y_2 + x_3y_1 - x_2y_1. \tag{10.172}$$

A similar expression can be derived for v in terms of v_1, v_2 and v_3. Thus the displacement of an arbitrary point within an element can be written in terms of the displacement at the nodes as

$$\boldsymbol{d} = \left\{ \begin{array}{c} u \\ v \end{array} \right\} = \frac{1}{\Delta} \begin{bmatrix} p_1(x,y) & p_2(x,y) & p_3(x,y) & 0 & 0 & 0 \\ 0 & 0 & 0 & p_1(x,y) & p_2(x,y) & p_3(x,y) \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{bmatrix}, \tag{10.173}$$

or

$$\boldsymbol{d} = [N]\,\boldsymbol{d^e}. \tag{10.174}$$

Expressing the strains $\boldsymbol{\varepsilon}$ as a function of the nodal displacements $\boldsymbol{d^e}$, one finds

$$\boldsymbol{\varepsilon} = \left\{ \begin{array}{c} \varepsilon_{\boldsymbol{x}} \\ \varepsilon_{\boldsymbol{y}} \\ \varepsilon_{\boldsymbol{xy}} \end{array} \right\} = [B]\,\boldsymbol{d^e}. \tag{10.175}$$

Since $\varepsilon_{\boldsymbol{x}} = \frac{\partial u}{\partial x}$, $\varepsilon_{\boldsymbol{y}} = \frac{\partial v}{\partial y}$ and $\varepsilon_{\boldsymbol{xy}} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$,

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \frac{\partial}{\partial \boldsymbol{x}} & 0 \\ 0 & \frac{\partial}{\partial \boldsymbol{y}} \\ \frac{\partial}{\partial \boldsymbol{y}} & \frac{\partial}{\partial \boldsymbol{x}} \end{bmatrix} \left\{ \begin{array}{c} u \\ v \end{array} \right\} = \begin{bmatrix} \frac{\partial}{\partial \boldsymbol{x}} & 0 \\ 0 & \frac{\partial}{\partial \boldsymbol{y}} \\ \frac{\partial}{\partial \boldsymbol{y}} & \frac{\partial}{\partial \boldsymbol{x}} \end{bmatrix} [N]\,\boldsymbol{d^e}. \tag{10.176}$$

Thus,

$$[B] = \begin{bmatrix} \frac{\partial}{\partial \boldsymbol{x}} & 0 \\ 0 & \frac{\partial}{\partial \boldsymbol{y}} \\ \frac{\partial}{\partial \boldsymbol{y}} & \frac{\partial}{\partial \boldsymbol{x}} \end{bmatrix} [N] \tag{10.177}$$

or

$$[B] = \frac{1}{\Delta} \begin{bmatrix} y_2 - y_3 & y_3 - y_1 & y_1 - y_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 & y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \end{bmatrix}. \tag{10.178}$$

Note that $[B]$ is not a function of x or y. The stiffness matrix

$$[\boldsymbol{K}] = \int_V [B]^t\,[D]\,[B]\,dV \tag{10.179}$$

where $[D]$ is a matrix that defines the material properties. Assuming plane stress, the matrix is

$$[D] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}, \tag{10.180}$$

but assuming plane strain provides

$$[D] = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix}. \tag{10.181}$$

Equation (10.179) produces the element stiffness matrix. Assembling all individual element stiffness matrices, using the method described in section 10.5, produces the global stiffness matrix.

10.15.2 *Rectangular elements*

Figure 10.13 shows a four node quadrilateral element with eight degrees of freedom.

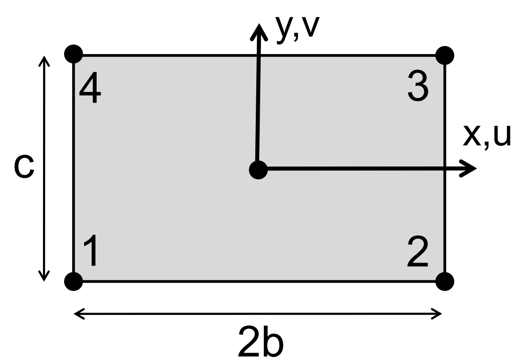

Figure 10.13: Four node quadrilateral element.

One can show that

$$u = \frac{1}{4bc}\left[(b-x)(c-y)u_1 + (b+x)(c-y)u_2 + (b+x)(c+y)u_3 + (b-x)(c+y)u_4\right]. \tag{10.182}$$

A similar result can be derived for v. After u and v have been determined, one can re-write the displacements in matrix notation as

$$\left\{ \begin{array}{c} u \\ v \end{array} \right\} = \frac{1}{4bc} \begin{bmatrix} (b-x)(c-y) & 0 \\ (b+x)(c-y) & 0 \\ (b+x)(c+y) & 0 \\ (b-x)(c+y) & 0 \\ 0 & (b-x)(c-y) \\ 0 & (b+x)(c-y) \\ 0 & (b+x)(c+y) \\ 0 & (b-x)(c+y) \end{bmatrix}^T = [N]\, \boldsymbol{d^e}. \quad (10.183)$$

Note that for convenience of notation $[N]$ has been presented in transposed form. Using

$$[B] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} [N] \quad (10.184)$$

one finds $[B]$ and then $[K]$.

10.16 *Three dimensional FEM analysis*

Three-dimensional elements are a straightforward extension of two-dimensional elements. Since the algebra gets more complicated, no element will be developed in this text. However, textbooks on FEM analysis can be consulted to gain an in-depth understanding of this topic. The 3-D elements can be understood by comparison to the 2-D elements. The equivalent of the 2-D constant strain triangle is the 3-D constant strain tetrahedron, shown in Fig. 10.14.

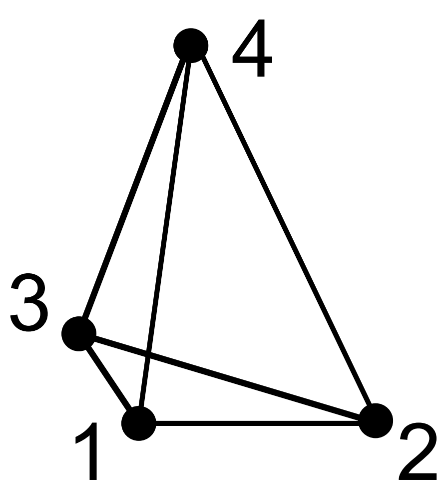

Figure 10.14: Four-node constant strain tetrahedron.

The equivalent of the 2-D four-node rectangle is the 3-D eight-node brick, shown in Fig. 10.15. Two dimensional elements are usually

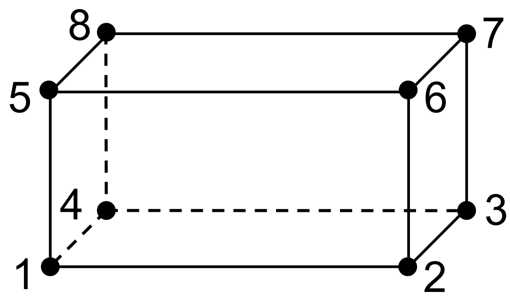

Figure 10.15: Eight-node brick element.

either triangular or quadrilateral. Quadrilateral elements are usually preferred, since these elements allow a structured and organized mesh. However, if the structure has complex contours, using quadrilateral elements makes it more difficult to conform to these contours. In addition, quadrilateral elements show difficulty in maintaining mesh consistency between regions with different mesh density. Hence, in the aforementioned situations it might prove more beneficial to use triangular elements. Elements with mid-side nodes also exist. These elements implement a higher order shape function (interpolation function), which allows higher accuracy, equivalent to using a finer mesh (smaller elements). Both higher order shape functions as finer mesh sizes will increase the calculation time. Figure 10.16 shows an example of a 3-D FEM model of a transmission housing with tetrahedral elements.

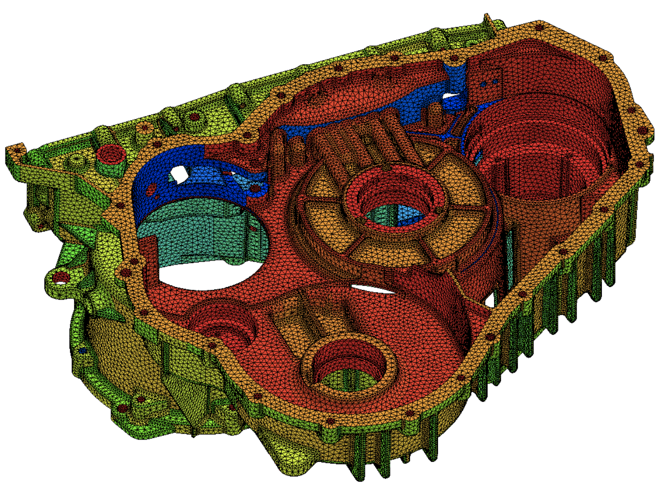

Figure 10.16: Example FEA mesh: a gearbox housing component, complete with colors showing individual volumes that divide the total volume in order to obtain good automatic meshing results as shown.