Chapter 10
FEM Essentials

In Chapters 8 and 9 the stiffness method was introduced in the framework of discrete models out of a pre-FEM era. In those models we had to apply different element types to model the complete membrane behaviour: springs for normal forces and panels for shear forces. The same was necessary in modeling plates in bending; there separate rotational springs and torsion panels were used. Compared to this approach the Finite Element Method (FEM) has been a major step forward. One and the same membrane element accounts for normal forces in two directions and shear forces, and one and the same plate bending element accounts for bending in two directions and torsion. The present Chapter is an overview of the main features of FEM codes and comments on its practical use. The aim of this book is not an in-depth presentation of FEM theory. For that purpose we refer to standard text books of Zienkiewicz et al. [16] and Hughes [17].

10.1 Elements and Degrees of Freedom

The Finite Element Method provides an approximation to structural behaviour. The first step is to divide the structure into a large number of *elements*. In this book the elements are plane elements; the dimension in the third direction is small compared to the other two. The elements are joined to each other at element corners; these are called *nodes*. Sometimes also mid-side nodes occur on the edges of elements. In the nodes we choose degrees of freedom. In a membrane plate analysis they are two orthogonal displacements u_x and u_y. In a plate bending element we normally use three degrees

J. Blaauwendraad, *Plates and FEM: Surprises and Pitfalls*, Solid Mechanics and Its Applications 171, DOI 10.1007/978-90-481-3596-7_10,

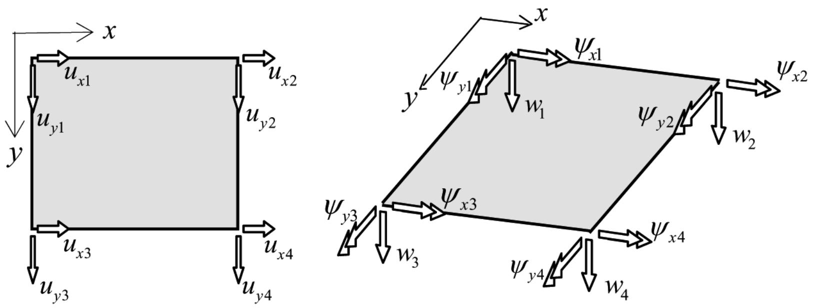

Figure 10.1 Degrees of freedom for membrane and bending analysis.

of freedom per node: the displacement w, the rotation ψ_x about the x-axis and ψ_y about the y-axis. For rectangular examples we refer to Figure 10.1.

Sign definition

We stress that the definition of the rotations in FE codes is different from the definition we used in Chapters 3 and 4 for the derivation of differential equations. To avoid confusion we use another symbol. In Chapters 3 and 4 we had chosen φ, now we use $\boldsymbol{\psi}$. The sign convention and subscripts are also different. The rotations φ_x and φ_y in Chapters 3 and 4 are positive when they lead to positive displacements u_x and u_x, respectively, for positive z-values in the plate. The rotations ψ_x and $\boldsymbol{\psi}_y$ have a different definition. They are rotations about the x- and y-axis, respectively, and are positive according to the right-hand rotation rule.

Plate elements may be spatially assembled. Examples are box-beams and multi-cell bridges. In such structures, elements are needed both for membrane and bending action; we usually need six degrees of freedom per node. Regretfully, structural engineers call such elements *shell elements*. It is true, FE codes include shell elements for curved surfaces, and in the limit case of zero curvature they are used for combined membrane bending action. We recommend calling such flat elements *membrane-bending elements*. In shell structures there is interaction between the membrane and bending state; this is absent in flat plates.

Commercially available packages usually offer a number of element shapes to be used. Figure 10.2 shows triangles, rectangles and quadrilaterals that can be inserted in the model. If a mid-side node is applied, the edge

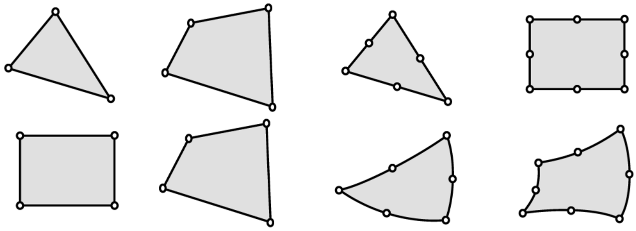

Figure 10.2 Commercially available packages offer various element shapes.

can be curved. For reasons beyond the scope of this chapter, such elements are called *isoparametric* elements. Sometimes a commercial package offers a quadrilateral element which, unknown to the user, is in fact an assemblage of triangles.

In order to make an analysis the structure is divided into elements. Figure 10.3 shows two examples. The left part comes close to the Brazilian test to determine the splitting tensile strength of a concrete cylinder. The right-hand part of the figure shows a shear wall example with a vertical row of openings as may occur in a tall building. In both examples a coarse mesh is drawn. In reality finer meshes will be applied. It is common practise that the software itself makes an appropriate mesh on the basis of the available element types. Usually the user needs to specify just the average size of the elements; the program does the rest. Figure 10.4 shows two examples of spatial structures which are assembled from flat membrane-bending plate elements: a multi-cell bridge and a part of a train carriage.

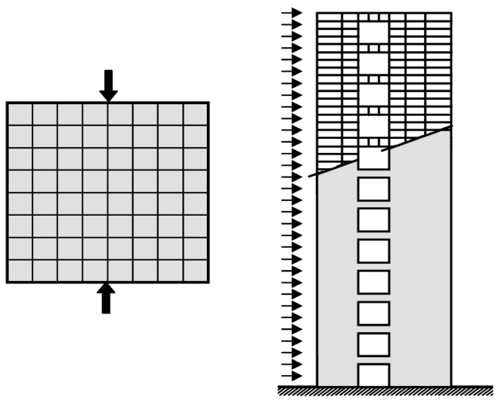

Figure 10.3 Examples of membrane plate structures with element mesh.

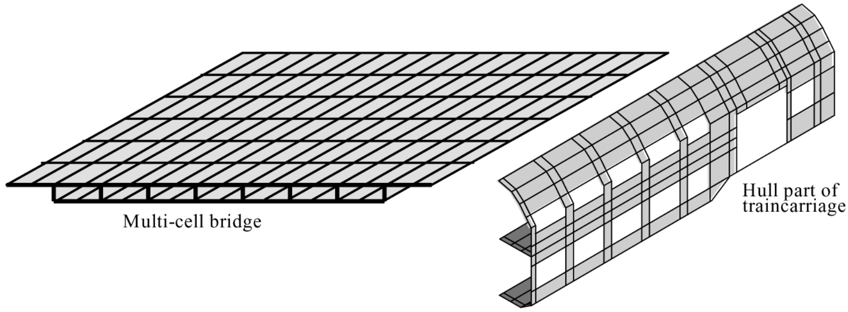

Figure 10.4 Examples of spatial structures composed of plate elements.

10.2 Stiffness Matrix and Constraints

The degrees of freedom at a node are common to all elements that meet at that node. An individual element, in turn, shares the degrees of freedom of different nodes. These degrees of freedom together form the *displacement vector* of the element (this may also contain rotations). A generalized force (which may be also a moment) is associated with each degree of freedom; these forces together form the *force vector* of the element. The element *stiffness matrix* relates the element displacement vector to the element force vector. For a triangular element with corner nodes i, j and k the matrix relation appears as

$$\begin{bmatrix} * & * & * \\ * & * & * \\ * & * & * \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \\ u_k \end{Bmatrix} = \begin{Bmatrix} F_{e,i} \\ F_{e,j} \\ F_{e,k} \end{Bmatrix} \tag{10.1}$$

Herein the vector u_i represents the degrees of freedom in node i and $F_{e,i}$ the vector of generalized element forces. The stiffness matrix governs the structural behaviour of the element. Its derivation is based on the approximation of the displacement field within the element. The higher the degree of polynomials in the field, the more accurate the element will perform. As a rule, the performance of an element is better if it has more degrees of freedom, but it is not a guarantee. The quality of an element is dependent on a number of items: the chosen displacement field, the way numerical integrations are done, the extent to which displacements in adjacent elements are compatible, etc. An element with mid-side nodes is expected to perform better for the same mesh compared to elements without, but sometimes they may do not.

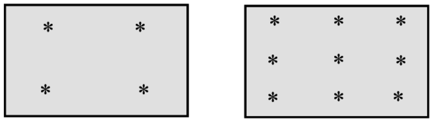

Figure 10.5 Two schemes for Gaussian points.

Programmers use *Gaussian points* in the mathematical integration procedure to construct element stiffness matrices. Gaussian points relate to integrals of polynomials over the area of an element. Gauss showed that one can write such integrals as weighted sums of values of the polynomial at a number of discrete points. The points do not coincide with nodes, but are situated inside elements at some distance from the edges. In rectangles sometimes a two-by-two scheme is used, sometimes three-by-three, see Figure 10.5. Normally the user need not know at all about such integration points, but occasionally programs may refer to the Gaussian points, therefore they are mentioned here. We refer for more details to [11] or [12].

The global stiffness matrix equation of a total structure with N degrees of freedom is similar to that for a discrete model as shown in Eq. (8.2)

$$\begin{bmatrix} \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot \end{bmatrix} \begin{Bmatrix} u_1 \\ \cdot \\ \cdot \\ u_N \end{Bmatrix} = \begin{Bmatrix} F_1 \\ \cdot \\ \cdot \\ F_N \end{Bmatrix} \tag{10.2}$$

The global stiffness matrix is an assemblage of the individual element matrices. Simple examples of the procedure appeared in Chapters 8 and 9. The right-hand vector holds the load to which the structure is subjected. This load consists of point loads at the nodes. When the user inputs a distributed load, the program will replace it by statically equivalent point loads. With the high speed of computers this is no problem in practice, and the user can apply fine meshes. The assembling procedure has a physical meaning. If M elements join together in node i then the M generalized element force vectors $F_{e,i}$ together balance the applied load F_i at that node.

The set of equations in (10.2) cannot be solved as long as rigid body displacements can occur, for then the global stiffness matrix will be singular. We must specify displacement constraints to prevent such singularity. Therefore, all commercial packages offer features to specify rigid or flexible supports. If a support is rigid, the corresponding displacement in that direction must be zero. So, it is no longer a degree of freedom. Therefore, the row and column in the global matrix equation which correspond with that degree of freedom must be omitted from the set. If a spring support in a node is specified, the program will add the spring stiffness to the main diagonal term which corresponds with the degree of freedom to be supported. This option can also be used to specify a rigid support. Then, a very large spring stiffness must be introduced. Some programs offer the choice for a support type in which only a compressive force can occur. In fact, this makes the analysis nonlinear. The analysis can be made by a linear-elastic analysis in an iterative way. We start including all supports. After the analysis we release the supports with a tensile reaction and restart the analysis. The iterative procedure is stopped when all support reactions are compressive or zero. Some programs do the iterative procedure itself.

10.3 Model Input

The preparation of input for a FE analysis covers all data for assembling a solvable set of algebraic equations. At least the following data must be specified:

- the shape of the structure,
- fineness of the mesh,
- element type and/or section profile,
- supports,
- material properties,
- load cases,
- applicable code of practice,
- combinations and weight factors.

Finite Element packages will always offer the feature of several load cases and the possibility of making various combinations. Codes of practice define which combinations must be considered, and which load factor must be assigned to each load case in a combination. The user can make a choice of point loads, line loads and distributed loads. Usually the load case for the self-weight of the structure is generated automatically on basis of the inputted data for geometry and material properties.

Often the element mesh is automatically generated by the program. However, the user may be asked to indicate the order of magnitude of the average element size. The choice of the element type is a very important decision by the user. For instance, in case of a slab analysis one must decide whether to use Kirchhoff elements or Mindlin ones. In Chapter 15 we will explain the importance of this choice. Many a user is not aware, however, that commercial packages have default options. For truss or beam elements, to be inserted in plate models, the user must make a choice from a section library.

Support data are easy to understand. As we noted earlier, a degree of freedom is either completely restrained, or a spring is introduced. Occasionally a special compression-only support is offered. Spring supports may be point springs, springs distributed along edges or over the area of an element.

10.4 Output Selection

Finite Element packages always offer a wide choice of output options. Standards are

- contour plots,
- trajectories,
- section graphs,
- lists,
- unity check,
- dimensioning/detailing.

The option of coloured contour plots has become very popular. These plots can be selected for displacements and stresses (or stress resultants like membrane forces, plate moments and transverse shear forces). Contour plots for principal stresses, forces and moments and their trajectories are more or less standard. Trajectories for shear forces are an exception, unfortunately.

Plea for graph output in sections

The great popularity of contour plots is to be regretted, the more so when they are used in combination with options for automatic detailing, results of which are also shown as contour plots. It prevents the structural engineer from fully grasping in which way the structure transfers load to supports. A far more valuable facility is the graph option for displacements or forces in sections over the structure. We will return to this in Chapter 14. Software providers should take pride in offering this option. It is very helpful if the structural engineer can output the integral of forces and moments in sections or dedicated part of sections.

Contour plots should at most be used to decide in which sections graphs will be shown. A good alternative to contour plots are 3D pictures of displacements, forces and moments. They are very instructive, because they appeal to the engineer's sense for structural behaviour.

Blameworthy use of envelopes of load cases

A really blameworthy practice is to consider an envelope of load cases and combination results, rather than judging the result of each load case and combination individually. In that way the structural engineer cannot pick up how the structure behaves, and will have no idea about actual safety levels. Apart of this, the approach is not economic. Structural engineers should investigate results of load cases and combinations individually, at least for the most important ones.

When specifying stresses or stress resultants that the engineer wants to be presented, there are several possibilities, such as stress (resultants) at nodes or in the element centres. Stresses at a node differ from element to element. The software may offer the choice of computing the average at a node or the values at the centre of elements. Averaging is a pleasant feature, but is misleading for elements which join in a node and have different thickness. Exceptionally, software may present output at Gaussian points. In such cases the software most probably deals with a specialized subject.

All programs will calculate support reactions. Usually an equilibrium check is done in so far that a check is made to compare the total load to the sum of the support reactions. Strictly speaking it is not proving that the set of equations has been solved correctly, because an ill-conditioned set of equations may lead to a wrong solution even though the test is satisfied. However, in practice it is valuable to test the correctness of the applied load, because ill-conditioned sets of equations are rare in practice.

Finally, the user can usually ask to see at which positions in the structure the permissible stress level is surpassed. This is of great use for steel structures. The stresses can be combined to yield the Von Mises stress which is then compared to the yield stress. In a unity check it is tested whether the ratio of the Von Mises stress and the yield stress is smaller than 1. If not, the design must be changed. For reinforced concrete slabs a FEM package often offers the feature of calculating the wanted reinforcement ratios. Because of its importance, Chapter 16 is fully devoted to this subject.

10.5 Message of the Chapter

- The reliability of a FE analysis highly depends on the competence of the structural engineer. Garbage in is garbage out.
- Seemingly, higher-order elements need not necessarily perform better than simpler elements.
- Programmers sometimes make choices which are hidden from the user. The user should be aware of set defaults.
- FE codes may offer post-processing options for unity-checks and detailing of reinforcement. The user should be aware of the assumptions underlying such design options.
- Contour plots are a popular way to present results, but they prevent the structural engineer from grasping how a structure will transfer load to supports.
- 3D plots and section graphs may appeal much more to the engineer's sense for structural behaviour. Software providers should take pride in offering such options. It must be possible to output the integral of forces and moments in sections and dedicated part of sections.
- The practice of making envelopes for results of load cases and combinations is blameworthy and fundamentally wrong. Structural engineers should investigate results of load cases and combinations individually, at least for the most important ones.