CHAPTER 9

INTRODUCTION TO THE FINITE ELEMENT METHOD

Introduction

So far in this text we have studied the means by which components can be analysed using so-called Mechanics of Materials approaches whereby, subject to making simplifying assumptions, solutions can be obtained by hand calculation. In the analysis of complex situations such an approach may not yield appropriate or adequate results and calls for other methods. In addition to experimental methods, numerical techniques using digital computers now provide a powerful alternative. Numerical techniques for structural analysis divides into three areas; the long established but limited capability *finite difference* method, the finite element method (developed from earlier structural matrix methods), which gained prominence from the 1950s with the advent of digital computers and, emerging over a decade later, the boundary element method. Attention in this chapter will be confined to the most popular finite element method and the coverage is intended to provide

- an insight into some of the basic concepts of the finite element method (fem.), and, hence, some basis of finite element (fe.), practice,
- the theoretical development associated with some relatively simple elements, enabling analysis of applications which can be solved with the aid of a simple calculator, and
- a range of worked examples to show typical applications and solutions.

It is recommended that the reader wishing for further coverage should consult the many excellent specialist texts on the subject.[1–10] This chapter does require some knowledge of matrix algebra, and again, students are directed to suitable texts on the subject.[11]

9.1. Basis of the finite element method

The fem. is a numerical technique in which the governing equations are represented in matrix form and as such are well suited to solution by digital computer. The solution region is represented, (discretised), as an assemblage (mesh), of small sub-regions called *finite elements*. These elements are connected at discrete points (at the extremities (corners), and in some cases also at intermediate points), known as nodes. Implicit with each element is its displacement function which, in terms of parameters to be determined, defines how the displacements of the nodes are interpolated over each element. This can be considered as an extension of the Rayleigh-Ritz process (used in Mechanics of Machines for analysing beam vibrations[6]). Instead of approximating the entire solution region by a single assumed displacement distribution, as with the Rayleigh-Ritz process, displacement distributions are assumed for each element of the assemblage. When applied to the analysis of a continuum (a solid or fluid through which the behavioural properties vary continuously), the discretisation becomes an assemblage of a number of elements each with a limited, i.e. *finite* number of degrees of freedom (dof). The *element* is the basic "building unit", with a predetermined number of dof., and can take various forms, e.g. one-dimensional rod or beam, two-dimensional membrane or plate, shell, and solid elements, see Fig. 9.1.

In stress applications, implicit with each element type is the nodal force/displacement relationship, namely the element stiffness property. With the most popular *displacement formulation* (discussed in §9.3), analysis requires the assembly and solution of a set of

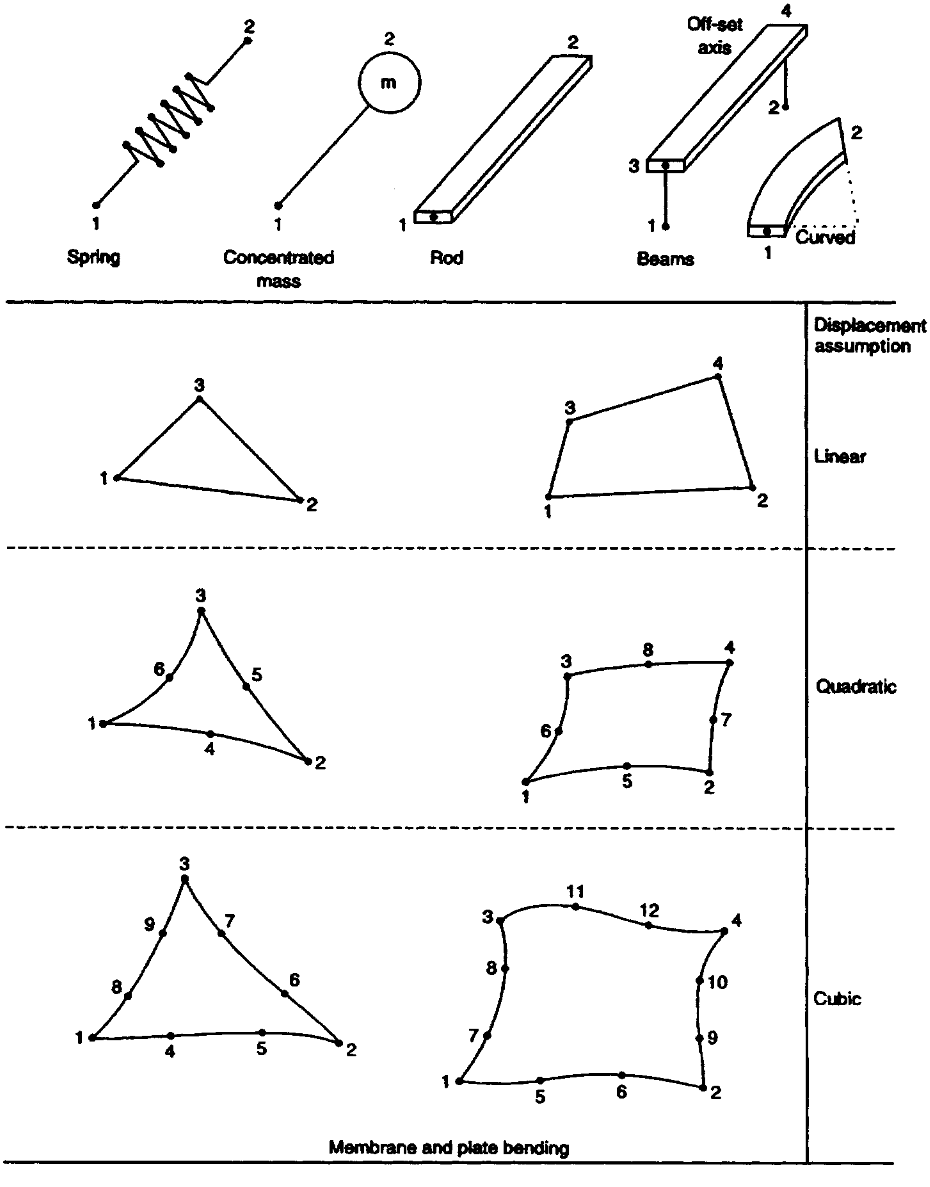

Fig. 9.1(a). Examples of element types with nodal points numbered.

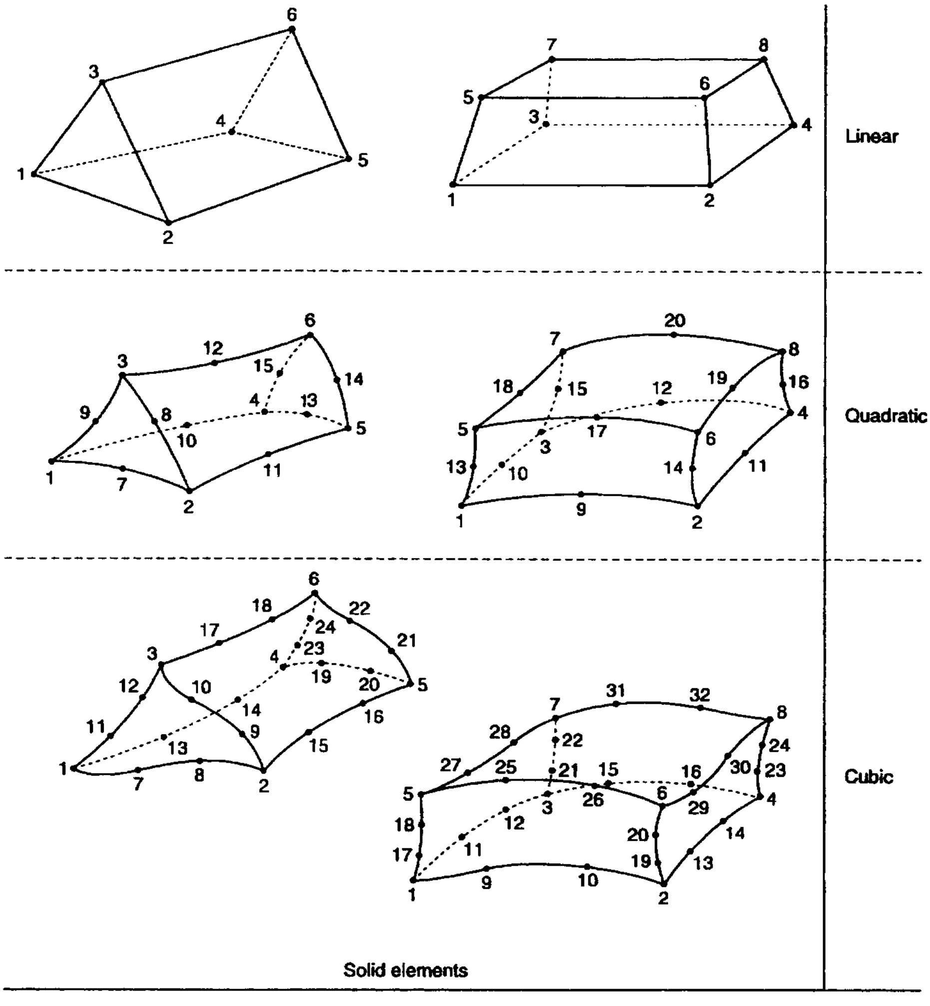

Fig. 9.1(b). Examples of element types with nodal points numbered.

simultaneous equations to provide the displacements for every node in the model. Once the displacement field is determined, the strains and hence the stresses can be derived, using strain-displacement and stress-strain relations, respectively.

9.2. Applicability of the finite element method

The fem. emerged essentially from the aerospace industry where the demand for extensive structural analyses was, arguably, the greatest. The general nature of the theory makes it applicable to a wide variety of boundary value problems (i.e. those in which a solution is required in a region of a body subject to satisfying prescribed boundary conditions, as encountered in equilibrium, eigenvalue and propagation or transient applications). Beyond the basic linear elastic/static stress analysis, finite element analysis (fea.), can provide solutions to non-linear material and/or geometry applications, creep, fracture mechanics, free and forced vibration. Furthermore, the method is not confined to solid mechanics, but is applied successfully to other disciplines such as heat conduction, fluid dynamics, seepage flow and electric and magnetic fields. However, attention in this text will be restricted to linearly elastic static stress applications, for which the assumption is made that the displacements are sufficiently small to allow calculations to be based on the undeformed condition.

9.3. Formulation of the finite element method

Even with restriction to solid mechanics applications, the fem. can be formulated in a variety of ways which broadly divides into 'differential equation', or 'variational' approaches. Of the differential equation approaches, the most important, most widely used and most extensively documented, is the *displacement, or stiffness, based fem.* Due to its simplicity, generality and good numerical properties, almost all major general purpose analysis programmes have been written using this formulation. Hence, only the displacement based fem. will be considered here, but it should be realised that many of the concepts are applicable to other formulations.

In §9.7, 9.8 and 9.9 the theory using the displacement method will be developed for a rod, simple beam and triangular membrane element, respectively. Before this, it is appropriate to consider here, a brief overview of the steps required in a fe. linearly elastic static stress analysis. Whilst it can be expected that there will be detail differences between various packages, the essential procedural steps will be common.

9.4. General procedure of the finite element method

The basic steps involved in a fea. are shown in the flow diagram of Fig. 9.2. Only a simple description of these steps is given below. The reader wishing for a more in-depth treatment is urged to consult some of numerous texts on the subject, referred to in the introduction.

9.4.1. Identification of the appropriateness of analysis by the finite element method

Engineering components, except in the simplest of cases, frequently have non-standard features such as those associated with the geometry, material behaviour, boundary conditions, or excitation (e.g. loading), for which classical solutions are seldom available. The analyst must therefore seek alternative approaches to a solution. One approach which can sometimes be very effective is to simplify the application grossly by making suitable approximations, leading to Mechanics of Materials solutions (the basis of the majority of this text). Allowance for the effects of local disturbances, e.g. rapid changes in geometry, can be achieved through the use of design charts, which provide a means of *local enhancement*. In current practice, many design engineers prefer to take advantage of high speed, large capacity, digital computers and use numerical techniques, in particular the fem. The range of application of the fem. has already been noted in §9.2. The versatility of the fem. combined with the avoidance, or reduction in the need for prototype manufacture and testing offer significant benefits. However, the purchase and maintenance of suitable fe. packages, provision of a computer platform with adequate performance and capacity, application of a suitably

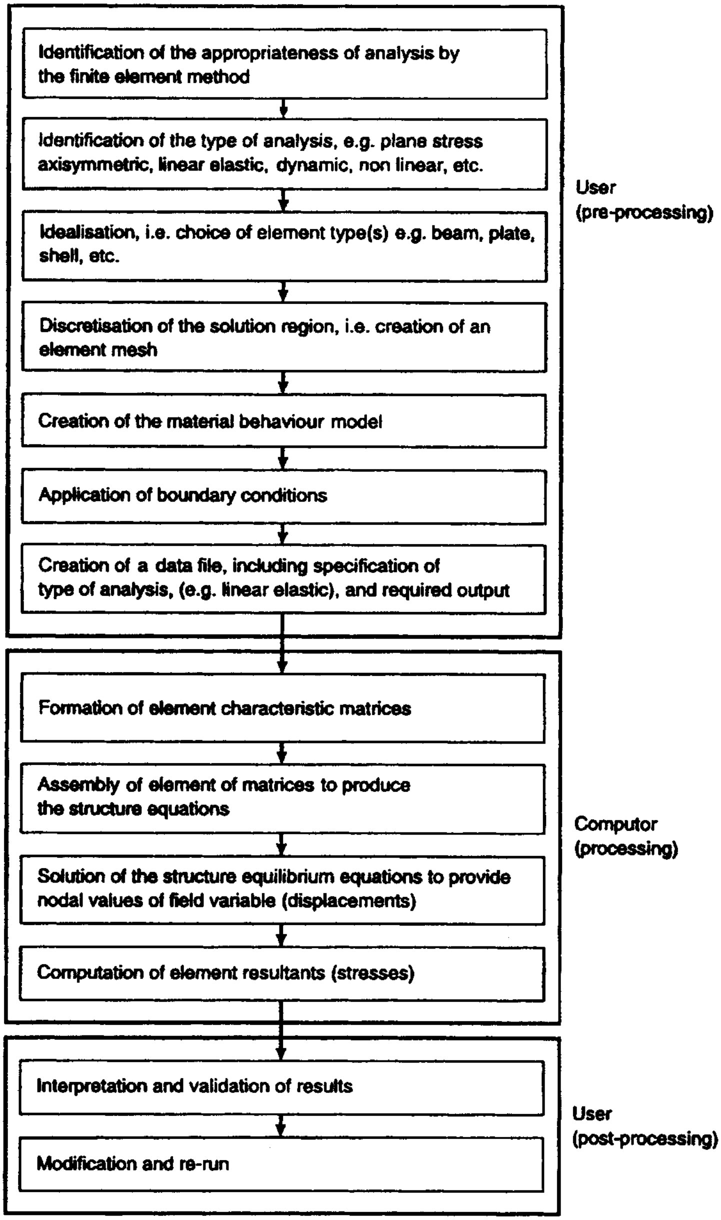

Fig. 9.2. Basic steps in the finite element method.

trained and experienced analyst and time for data preparation and processing should not be underestimated when selecting the most appropriate method. Experimental methods such as those described in Chapter 6 provide an effective alternative approach.

It is desirable that an analyst has access to all methods, i.e. analytical, numerical and experimental, and to not place reliance upon a single approach. This will allow essential validation of one technique by another and provide a degree of confidence in the results.

9.4.2. *Identification of the type of analysis*

The most appropriate type(s) of analysis to be employed needs to be identified in order that the component behaviour can best be represented. The assumption of either plane stress or plane strain is a common example. The high cost of a full three-dimensional analysis can be avoided if the assumption of both geometric and load symmetry can be made. If the application calls for elastic stress analysis, then the system equations will be linear and can be solved by a variety of methods, Gaussian elimination, Choleski factorisation or Gauss-Seidel procedure.[5]

For large displacement or post-yield material behaviour applications the system equations will be non-linear and iterative solution methods are required, such as that of Newton-Raphson.[5]

9.4.3. *Idealisation*

Commercially available finite element packages usually have a number of different elements available in the element library. For example, one such package, HKS ABAQUS[12] has nearly 400 different element variations. Examples of some of the commonly used elements have been given in Fig. 9.1.

Often the type of element to be employed will be evident from the physical application. For example, rod and beam elements can represent the behaviour of frames, whilst shell elements may be most appropriate for modelling a pressure vessel. Some regions which are actually three-dimensional can be described by only one or two independent coordinates, e.g. pistons, valves and nozzles, etc. Such regions can be idealised by using axisymmetric elements. Curved boundaries are best represented by elements having mid-side (or intermediate) nodes in addition to their corner nodes. Such elements are of higher *order* than linear elements (which can only represent straight boundaries) and include quadratic and cubic elements. The most popular elements belong to the so-called *isoparametric* family of elements, where the same parameters are used to define the geometry as define the displacement variation over the element. Therefore, those isoparametric elements of quadratic order, and above, are capable of representing curved sides and surfaces.

In situations where the type of elements to be used may not be apparent, the choice could be based on such considerations as

(a) number of dof.,
(b) accuracy required,
(c) computational effort,
(d) the degree to which the physical structure needs to be modelled.

Use of the elements with a quadratic displacement assumption are generally recommended as the best compromise between the relatively low cost but inferior performance of linear elements and the high cost but superior performance of cubic elements.

9.4.4. *Discretisation of the solution region*

This step is equivalent to replacing the actual structure or continuum having an infinite number of dof. by a system having a finite number of dof. This process, known as *discretisation*, calls for engineering judgement in order to model the region as closely as necessary. Having selected the element type, discretisation requires careful attention to *extent of the model* (i.e. location of model boundaries), *element size and grading, number of elements*, and factors influencing the *quality of the mesh*, to achieve adequately accurate results consistent with avoiding excessive computational effort and expense. These aspects are briefly considered below.

Extent of model

Reference has already been made above to applications which are axisymmetric, or those which can be idealised as such. Generally, advantage should be taken of geometric and loading symmetry wherever it exists, whether it be plane or axial. Appropriate boundary conditions need to be imposed to ensure the reduced portion is representative of the whole. For example, in the analysis of a semi-infinite tension plate with a central circular hole, shown in Fig. 9.3, only a quadrant need be modelled. However, in order that the quadrant is representative of the whole, respective v and u displacements must be prevented along the x and y direction symmetry axes, since there will be no such displacements in the full model/component.

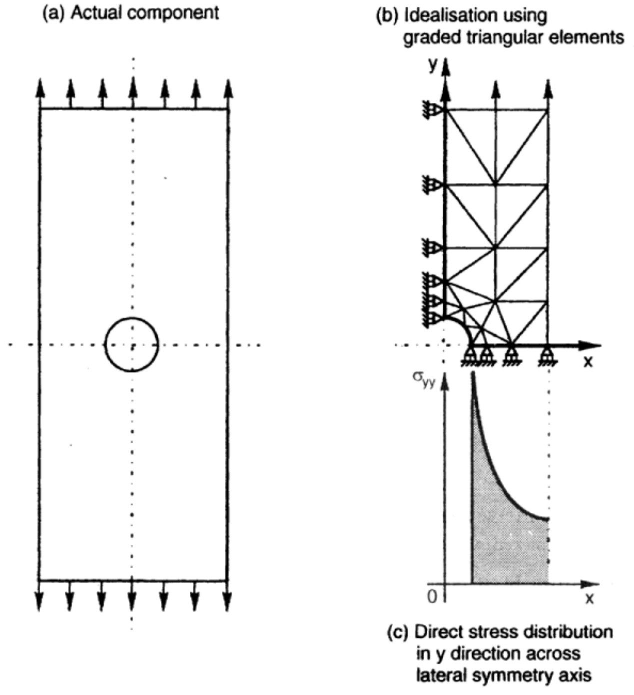

Fig. 9.3. Finite element analysis of a semi-infinite tension plate with a central circular hole, using triangular elements.

Further, it is known that disturbances to stress distributions due to rapid changes in geometry or load concentrations are only local in effect. Saint-Venant's principle states that the effect of stress concentrations essentially disappear within relatively small distances (approximately equal to the larger lateral dimension), from the position of the disturbance. Advantage can therefore be taken of this principle by reducing the necessary extent of a finite element model. A rule-of-thumb is that a model need only extend to one-and-a-half times the larger lateral dimension from a disturbance, see Fig. 9.4.

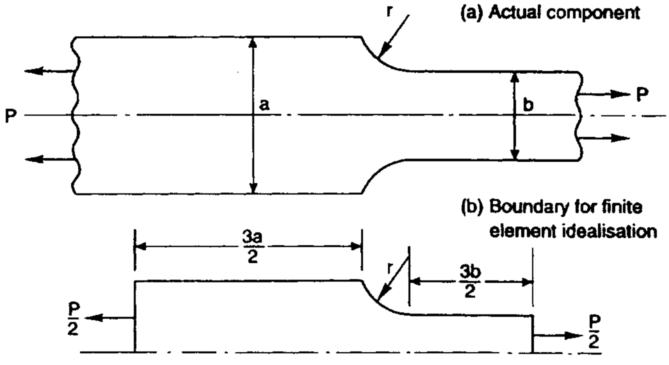

Fig. 9.4. Idealisation of a shouldered tension strip.

Element size and grading

The relative size of elements directly affects the quality of the solution. As the element size is reduced so the accuracy of solution can be expected to increase since there is better representation of the field variable, e.g. displacement, and/or better representation of the geometry. However, as the element size is reduced, so the number of elements increases with the accompanying penalty of increased computational effort. Needlessly small elements in regions with little variation in field variable or geometry will be wasteful. Equally, in regions where the stress variation is not of primary interest then a locally coarse mesh can be employed providing it is sufficiently far away from the region of interest and that it still provides an accurate stiffness representation. Therefore, element sizes should be graded in order to take account of anticipated stress/strain variations and geometry, and the results required. The example of stress analysis of a semi-infinite tension plate with a central circular hole, Fig. 9.3, serves to illustrate how the size of the elements can be graded from small-size elements surrounding the hole (where both the stress/strain and geometry are varying the most), to become coarser with increasing distance from the hole.

Number of elements

The number of elements is related to the previous matter of element size and, for a given element type, the number of elements will determine the total number of dof. of the model, and combined with the relative size determines the *mesh density*. An increase in the number of elements can result in an improvement in the accuracy of the solution, but a limit will be reached beyond which any further increase in the number of elements will not significantly improve the accuracy. This matter of *convergence* of solution is clearly important, and with experience a near optimal mesh may be achievable. As an alternative to increasing the number of elements, improvements in the model can be obtained by increasing the element order. This alternative form of *enrichment* can be performed manually (by substituting elements), or can be performed automatically, e.g. the commercial package RASNA has this capability. Clearly, any increase in the number of elements (or element order), and hence dof., will require greater computational effort, will put greater demands on available computer memory and increase cost.

Quality of the mesh

The quality of the fe. predictions (e.g. of displacements, temperatures, strains or stresses), will clearly be affected by the performance of the model and its constituent elements. The factors which determine quality[13] will now be explored briefly, namely

(a) coincident elements,
(b) free edges,
(c) poorly positioned "midside" nodes,
(d) interior angles which are too extreme,
(e) warping, and
(f) distortion.

(a) Coincident elements

Coincident elements refer to two or more elements which are overlaid and share some of the nodes, see Fig. 9.5. Such coincident elements should be deleted as part of cleaning-up of a mesh.

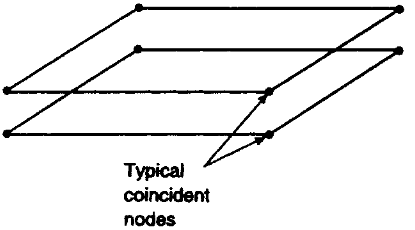

Fig. 9.5. Coincident elements.

(b) Free edges

A free edge should only exist as a model boundary. Neighbouring elements should share nodes along common inter-element boundaries. If they do not, then a free edge exists and will need correction, see Fig. 9.6.

(c) Poorly positioned "midside" nodes

Displacing an element's "midside" node from its mid-position will cause distortion in the mapping process associated with high order elements, and in extreme cases can significantly degrade an element's performance. There are two aspects to "midside" node displacement, namely, the relative position between the corner nodes, and the node's offset from a straight

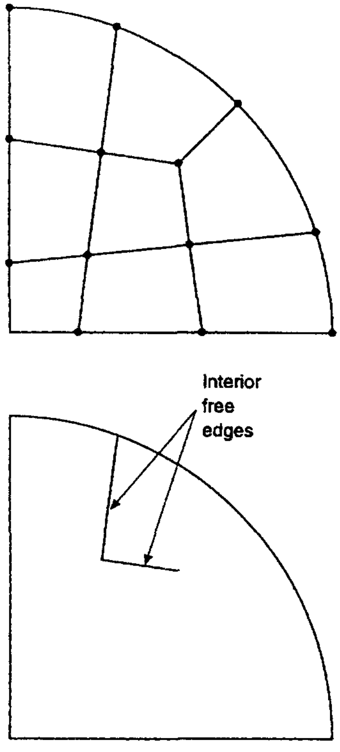

Fig. 9.6. Free edges.

line joining the corner nodes, see Fig. 9.7. The midside node's relative position should ideally be 50% of the side length for a parabolic element and 33.3% for a cubic element. An example of the effect of displacement of the "midside" node to the 25% position, is reported for a parabolic element[14] to result in a 15% error in the major stress prediction.

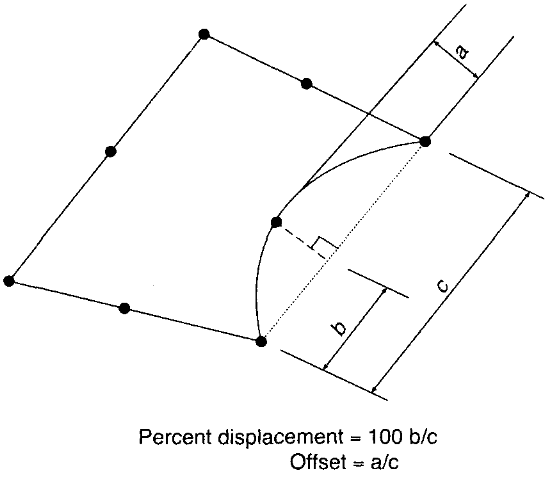

Fig. 9.7. "Midside" node displacement.

(d) Interior angles which are too extreme

Interior angles which are excessively small or large will, like displaced "midside" nodes, cause distortion in the mapping process. A re-entrant corner (i.e. an interior angle greater than 180°), see Fig. 9.8, will cause failure in the mapping as the *Jacobian matrix* (relating the derivatives with respect to curvilinear (r,s), coordinates, to those with respect to cartesian (x,y), coordinates), will not have an inverse (i.e. its determinate will be zero). For quadrilateral elements the ideal interior angle is 90°, and for triangular elements it is 60°.

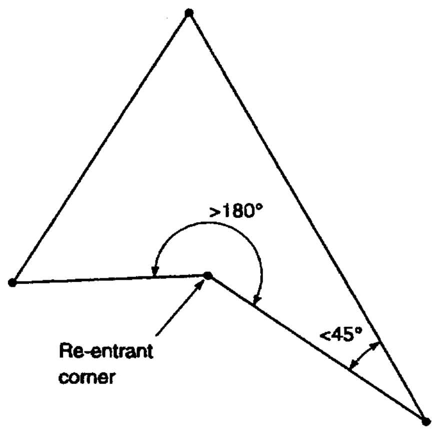

Fig. 9.8. Extreme interior angles.

(e) Warping

Warping refers to the deviation of the face of a "planar" element from being planar, see Fig. 9.9. The analogy of the three-legged milking stool (which is steady no-matter how uneven the surface is on which it is placed), to the triangular element serves to illustrate an advantage of this element over its quadrilateral counterpart.

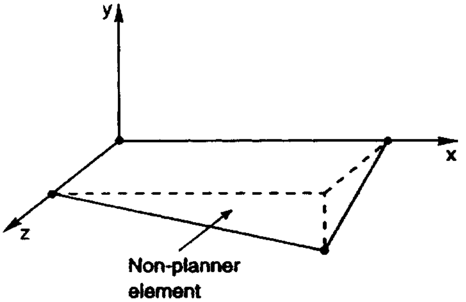

Fig. 9.9. Warping.

(f) Distortion

Distortion is the deviation of an element from its ideal shape, which corresponds to that in curvilinear coordinates. SDRC I-DEAS[13] gives two measures, namely

(1) the departure from the basic element shape which is known as *distortion*, see Fig. 9.10. Ideally, for a quadrilateral element, with regard to distortion, the shape should be a rectangle, and

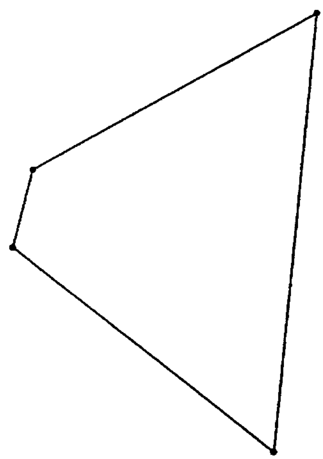

Fig. 9.10. Distortion.

(2) the amount of elongation suffered by an element which is known as *stretch*, or *aspect ratio distortion*, see Fig. 9.11. Ideally, for a quadrilateral element, with regard to stretch, the shape should be square.

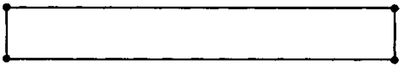

Fig. 9.11. Stretch.

Whilst small amounts of deviation of an element's shape from that of the parent element can, and must, be tolerated, unnecessary and excessive distortions and stretch, etc. must be avoided if degraded results are to be minimised. High order elements in gradually varying strain fields are most tolerant of shape deviation, whilst low order elements in severe strain fields are least tolerant.[5] There are automatic means by which element shape deviation can be measured, using information derived from the Jacobian matrix. Errors in a solution and the rate of convergence can be judged by computing so-called energy norms derived from successive solutions.[7] However, it is left to the judgement of the user to establish the degree of shape deviation which can be tolerated. Most packages offer quality checking facilities, which allows the user to interrogate the shape deviation of all, or a selection of, elements. I-DEAS provides a measure of element quality using a value with a range of -1 to $+1$, (where

+1 is the target value corresponding to zero distortion, and stretch, etc.). Negative values, which arise for example, from re-entrant corners, referred to above, will cause an attempted solution to fail, and hence need to be rectified. A distortion value above 0.7 can be considered acceptable, but errors will be incurred with any value below 1.0. However, circumstances may dictate acceptance of elements with a distortion value below 0.7. Similarly, as a rule-of-thumb, a stretch value above 0.5 can be considered acceptable, but again, errors will be incurred with any value below 1.0. Companies responsible for analyses may issue guidelines for quality, an example of which is shown in Table 9.1.

Table 9.1. Example of element quality guidelines.

Element	Interior angle°	Warpage	Distortion	Stretch
Triangle	30–90	N/A	0.35	0.3
Quadrilateral	45–135	0.2	0.60	0.3
Wedge	30–90	N/A	0.35	0.3
Tetrahedron	30–90	N/A	0.10	0.1
Hexahedron	45–135	0.2	0.5	0.3

9.4.5. Creation of the material model

The least material data required for a stress analysis is the empirical elastic modulus for the component under analysis describing the relevant stress/strain law. For a dynamic analysis, the material density must also be specified. Dependent upon the type of analysis, other properties may be required, including Poisson's ratio for two- and three-dimensional models and the coefficient of thermal expansion for thermal analyses. For analyses involving non-linear material behaviour then, as a minimum, the yield stress and yield criterion, e.g. von Mises, need to be defined. If the material within an element can be assumed to be isotropic and homogeneous, then there will be only one value of each material property. For non-isotropic material, i.e. orthotropic or anisotropic, then the material properties are direction and spacially dependent, respectively. In the extreme case of anisotropy, 21 independent values are required to define the material matrix.[5]

9.4.6. Node and element ordering

Before moving on to consider boundary conditions, it is appropriate to examine node and element ordering and its effect on efficiency of solution by briefly exploring the methods used. The formation of the element characteristic matrices (to be considered in §9.7, 9.8 and 9.9), and the subsequent solution are the two most computationally intensive steps in any fe. analysis. The computational effort and memory requirements of the solution are affected by the method employed, and are considered below.

It will be seen in Section 9.7, and subsequently, that the displacement based method involves the assembly of the *structural*, or *assembled, stiffness matrix* $[K]$, and the load and displacement column matrices, $\{P\}$ and $\{p\}$, respectively, to form the governing equation for stress analysis $\{P\} = [K]\{p\}$. With reference to §9.7, and subsequently, two features of the fem. will be seen to be that the assembled stiffness matrix $[K]$, is sparsely populated and is symmetric. Advantage can be taken of this in reducing the storage requirements of the computer. Two solution methods are used, namely, *banded* or *frontal*, the choice of which is dependent upon the number of dof. in the model.

Banded method of solution

The banded method is appropriate for small to medium size jobs (i.e. up to 10 000 dof.). By carefully ordering the dof. the assembled stiffness matrix $[K]$, can be banded with non-zero terms occurring only on the leading diagonal. Symmetry permits only half of the band to be stored, but storage requirements can still be high. It is advantageous therefore to minimise the *bandwidth*. A comparison of different node numbering schemes is provided by Figs. 9.12 and 9.13 in which a simple model comprising eight triangular linear elements is considered, and for further simplicity the nodal contributions are denoted as shaded squares, the empty squares denoting zeros.

The semi-bandwidth can be seen to depend on the node numbering scheme and the number of dof. per node and has a direct effect on the storage requirements and computational effort.

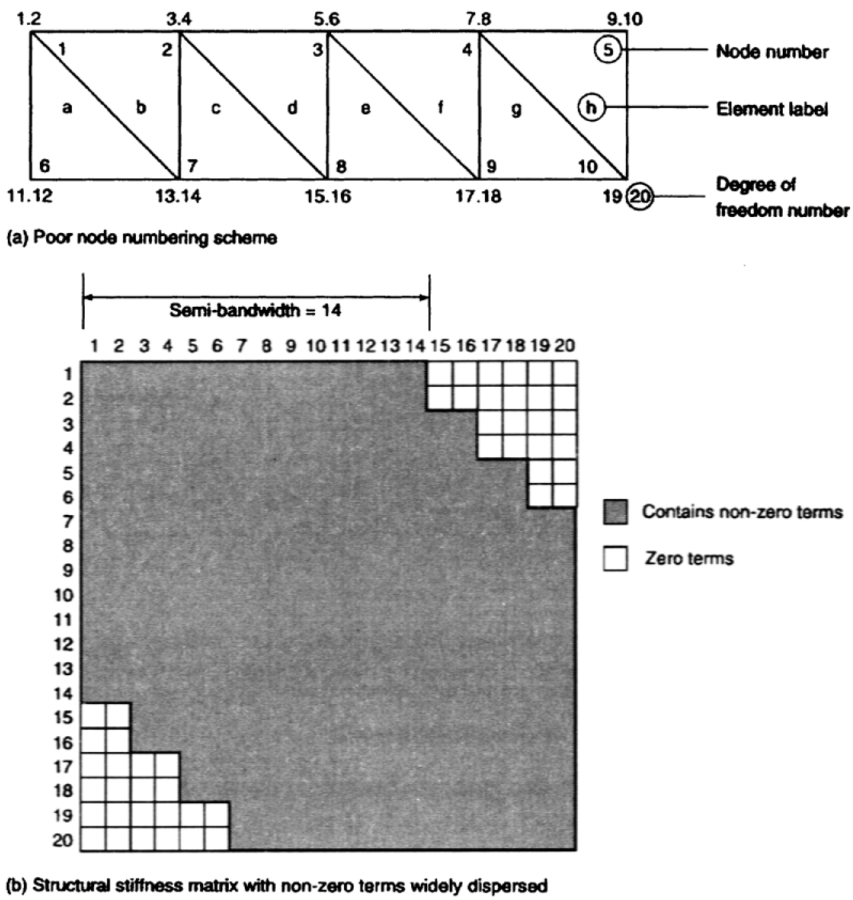

Fig. 9.12. Structural stiffness matrix corresponding to poor node ordering.

For a given number of dof. per node, which is generally fixed for each assemblage, the bandwidth can be minimised by using a proper node numbering scheme.

With reference to Figs. 9.12 and 9.13 there are a total of 20 dof. in the model (i.e. 10 nodes each with an assumed 2 dof.), and if the symmetry and bandedness is not taken advantage of, storage of the entire matrix would require $20^2 = 400$ locations. For the efficiently numbered model with a semi-bandwidth of 8, see Fig. 9.13, taking advantage of the symmetry and bandedness, the storage required for the upper, or lower, half-band is only $8 \times 20 = 160$ locations.

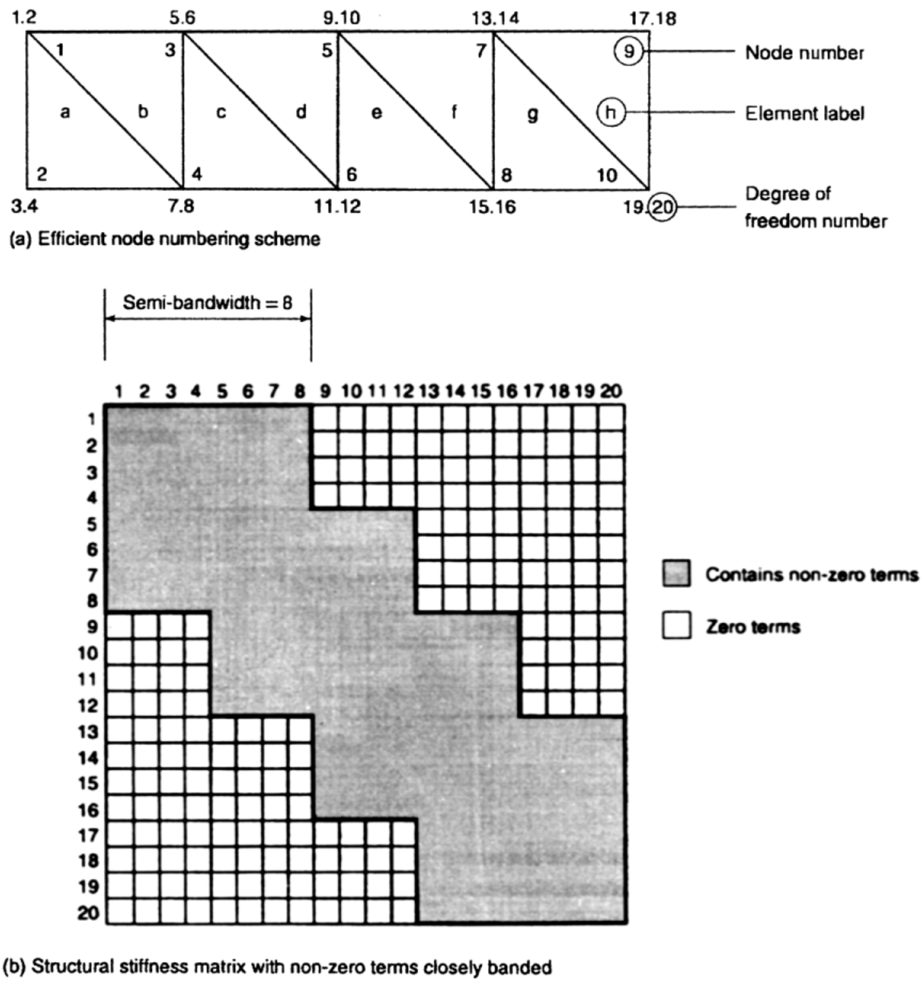

Fig. 9.13. Structural stiffness matrix corresponding to efficient node ordering.

From observation of Figs. 9.12 and 9.13 it can be deduced that the semi-bandwidth can be calculated from

$$\text{semi-bandwidth} = f(d + 1)$$

where f is the number of dof. per node and d is the maximum largest difference in the node numbers for all elements in the assemblage. This expression is applicable to any type of finite element. It follows that to minimise the bandwidth, d must be minimised and this is achieved by simply numbering the nodes across the shortest dimension of the region.

For large jobs the capacity of computer memory can be exceeded using the above banded method, in which case a frontal solution is used.

Frontal method of solution

The frontal method is appropriate for medium to large size jobs (i.e. greater than 10 000 dof.). To illustrate the method, consider the simple two-dimensional mesh shown in Fig. 9.14. Nodal contributions are assembled in element order. With reference to Fig. 9.14, with the assembly of element number 1 terms (i.e. contributions from nodes 1, 2, 6 and 7), all information relating to node number 1 will be complete since this node is not common to any other element. Thus the dofs. for node 1 can be eliminated from the set of system equations. Element number 2 contributions are assembled next, and the system matrix will now contain contributions from nodes 2, 3, 6, 7 and 8. At this stage the dofs. for node number 2 can be eliminated. Further element contributions are merged and at each stage any nodes which do not appear in later elements are *reduced out*. The solution thus proceeds as a front through the system. As, for example, element number 14 is assembled, dofs. for the nodes indicated by line B are required, see Fig. 9.14. After eliminations which follow assembly of element number 14, dofs. associated with line C are needed. The solution front has thus moved from line A to C.

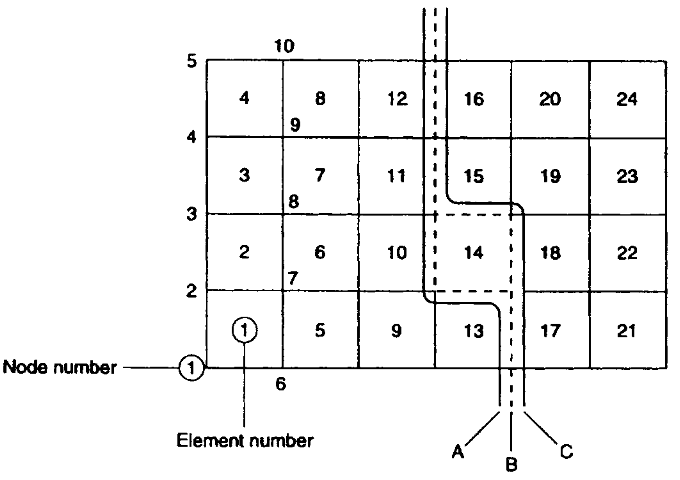

Fig. 9.14. Frontal method of solution.

To minimise memory requirements, which is especially important for jobs with large numbers of dof., the instantaneous width, i.e. *front size*, of the stiffness matrix during merging should be kept as small as possible. This is achieved by ensuring that elements are selected for merging in a specific order. Figures 9.15(a) and (b) serve to illustrate badly and well ordered elements, respectively, for a simple two-dimensional application. Front ordering facilities are available with some fe. packages which will automatically re-order the elements to minimise the front size.

4	9	13	7	2
15	11	5	10	14
1	6	12	8	3

(a) Badly ordered elements

3	6	9	12	15
2	5	8	11	14
1	4	7	10	13

(b) Well ordered elements

Fig. 9.15. Examples of element ordering for frontal method.

9.4.7. Application of boundary conditions

Having created a mesh of finite elements and before the job is submitted for solution, it is necessary to enforce conditions on the boundaries of the model. Dependent upon the application, these can take the form of

- restraints,
- constraints,
- structural loads,
- heat transfer loads, or
- specification of active and inactive dof.

Attention will be restricted to a brief consideration of restraints and structural loads, which are sufficient conditions for a simple stress analysis. The reader wishing for further coverage is again urged to consult the many specialist texts.[1–10]

Restraints

Restraints, which can be applied to individual, or groups of nodes, involve defining the displacements to be applied to the possible six dof., or perhaps defining a temperature. As an example, reference to Fig. 9.3(b) shows the necessary restraints to impose symmetry conditions. It can be assumed that the elements chosen have only 2 dof. per node, namely u and v translations, in the x and y directions, respectively. The appropriate conditions are

along the x-axis, $v = 0$, and
along the y-axis, $u = 0$.

The usual symbol, representing a frictionless roller support, which is appropriate in this case, is shown in Fig. 9.16(a), and corresponds to zero normal displacement, i.e. $\delta_n = 0$, and zero tangential shear stress, i.e. $\tau_t = 0$, see Fig. 9.16(b).

In a static stress analysis, unless sufficient restraints are applied, the system equations (see §9.4.5), cannot be solved, since an inverse will not exist. The physical interpretation of this is that the loaded body is free to undergo unlimited *rigid body motion*. Restraints must be

Fig. 9.16. Boundary node with zero shear traction and zero normal displacement.

chosen to be sufficient, but not to create rigidity which does not exist in the actual component being modelled. This important matter of appropriate restraints can call for considerable engineering judgement, and the choice can significantly affect the behaviour of the model and hence the validity of the results.

Structural loads

Structural loads, which are applied to nodes can, through usual program facilities, be specified for application to groups of nodes, or to an entire model, and can take the form of loads, temperatures, pressures or accelerations. At the program level, only nodal loads are admissible, and hence when any form of distributed load needs to be applied, the nodal equivalent loads need to be computed, either manually or automatically. One approach is to simply define a set of statically equivalent loads, with the same resultant forces and moments as the actual loads. However, the most accurate method is to use kinematically equivalent loads[5] as simple statically equivalent loads do not give a satisfactory solution for other than the simplest element interpolation. Figure 9.17 illustrates the case of an element with a quadratic displacement interpolation. Here the distributed load of total value W, is replaced by three nodal loads which produce the same work done as that done by the actual distributed load.

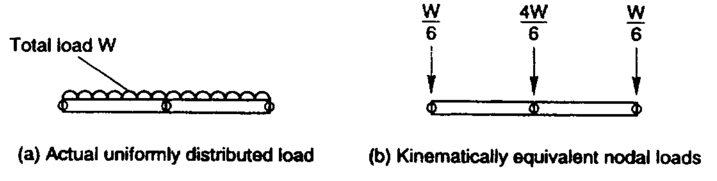

Fig. 9.17. Structural load representation.

9.4.8. Creation of a data file

The data file, or *deck*, will need to be in precisely the format required by the particular program being used; although essentially all programs will require the same basic model data, i.e. nodal coordinates, element type(s) and connectivity, material properties and boundary conditions. The type of solution will need to be specified, e.g. linear elastic, normal modes, etc. The required output will also need to be specified, e.g. deformations, stresses, strains, strain energy, reactions, etc. Much of the tedium of producing a data file is removed if automatic data preparation is available. Such an aid is beneficial with regard to minimising the possibility of introducing data errors. The importance of avoiding errors cannot be over-emphasised, as the validity of the output is clearly dependent upon the correctness of the data. Any capability of a program to detect errors is to be welcomed. However, it should be realised that it is impossible for a program to detect all forms of error. e.g. incorrect but possible coordinates, incorrect physical or material properties, incompatible units, etc., can all go undetected. The user must, therefore, take every possible precaution to guard against errors. Displays of the mesh, including "shrunken" or "exploded" element views to reveal absent elements, restraints and loads should be scrutinised to ensure correctness before the solution stage is entered; the material and physical properties should also be examined.

9.4.9. Computer, processing, steps

The steps performed by the computer can best be followed by means of applications using particular elements, and this will be covered in §9.7 and subsequent sections.

9.4.10. Interpretation and validation of results

The numerical output following solution is often provided to a substantial number of decimal places which gives an aura of precision to the results. The user needs to be mindful that the fem. is numerical and hence is approximate. There are many potential sources of error, and a responsibility of the analyst is to ensure that errors are not significant. In addition to approximations in the model, significant errors can arise from round-off and truncation in the computation.

There are a number of checks that should be routine procedure following solution, and these are given below

- Ensure that any warning messages, given by the program, are pursued to ensure that the results are not affected. Error messages will usually accompany a failure in solution and clearly, will need corrective action.
- An obvious check is to examine the deformed geometry to ensure the model has behaved as expected, e.g. Poisson effect has occurred, slope continuity exists along axes of symmetry, etc.
- Ensure that equilibrium has been satisfied by checking that the applied loads and moments balance the reactions. Excessive out of balance indicates a poor mesh.
- Examine the smoothness of stress contours. Irregular boundaries indicate a poor mesh.
- Check inter-element stress discontinuities (stress jumps), as these give a measure of the quality of model. Large discontinuities indicate that the elements need to be enhanced.
- On traction-free boundaries the principal stress normal to the boundary should be zero. Any departure from this gives an indication of the quality to be expected in the other principal stress predictions for this point.
- Check that the directions of the principal stresses agree with those expected, e.g. normal and tangential to traction-free boundaries and axes of symmetry.

Results should always be assessed in the light of common-sense and engineering judgement. Manual calculations, using appropriate simplifications where necessary, should be carried out for comparison, as a matter of course.

9.4.11. Modification and re-run

Clearly, the need for design modification and subsequent fe. re-runs depends upon the particular circumstances. The computational burden may prohibit many re-runs. Indeed, for large jobs, (which may involve many thousands of dof. or many increments in the case of non-linear analyses), re-runs may not be feasible. The approach in such cases may be to run several exploratory crude models to gain some initial understanding how the component behaves, and hence aid final modelling.

9.5. Fundamental arguments

Regardless of the type of structure to be analysed, irrespective of whether the loading is static or dynamic, and whatever the material behaviour may be, there are only three types of argument to be invoked, namely, *equilibrium, compatibility and stress/strain law*. Whilst these arguments will be found throughout this text it is worthwhile giving them some explicit attention here as a sound understanding will help in following the theory of the fem. in the proceeding sections of this chapter.

9.5.1. Equilibrium

External nodal equilibrium

Static equilibrium requires that, with respect to some orthogonal coordinate system, the reactive forces and moments must balance the externally applied forces and moments. In fea. this argument extends to all nodes in the model. With reference to Fig. 9.18, some nodes may be subjected to applied forces and moments, (node number 4), and others may be support points (node numbers 1 and 6). There may be other nodes which appear to be neither of these (node numbers 2, 3, and 5), but are in fact nodes for which the applied force, or moment, is zero, whilst others provide support in one or two orthogonal directions and are loaded (or have zero load), in the remaining direction(s) (node number 6). Hence, for each node and with respect to appropriate orthogonal directions, satisfaction of external equilibrium requires

external loads or reactions = summation of internal, element, loads

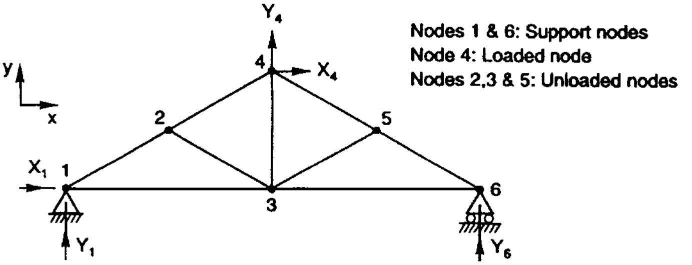

Fig. 9.18. Structural framework.

Then, for the jth node
$$\{P_j\} = \sum_{e=1}^{m} \{S^{(e)}\}$$

where the summation is of the internal loads at node j from all m elements joined at node j.

Use of this relation can be illustrated by considering the simple frame, idealised as planar with pin-joints and discretised as an assemblage of three elements, as shown in Fig. 9.19. The nodal force column matrix for the structure is

$$\{P\} = \{P_1 P_2 P_3\} = \{X_1 Y_1 X_2 Y_2 X_3 Y_3\}$$

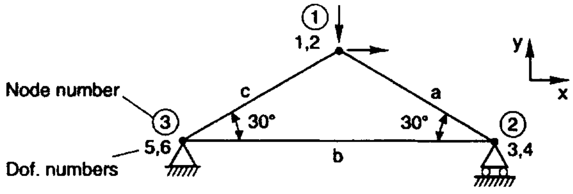

Fig. 9.19. Simple pin-jointed plane frame.

and the element force column matrix for the structure is

$$\{S\} = \{\{S^{(a)}\}\{S^{(b)}\}\{S^{(c)}\}\} = \{U_1 V_1 U_2 V_2,\ U_2 V_2 U_3 V_3,\ U_1 V_1 U_3 V_3\}$$

It follows from the above that external, nodal, equilibrium for the structure is satisfied by forming the relationship between the nodal and element forces as

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 1 \end{bmatrix} \left[\begin{array}{c} U_1 \\ V_1 \\ U_2 \\ V_2 \\ \hline U_2 \\ V_2 \\ U_3 \\ V_3 \\ \hline U_1 \\ V_1 \\ U_3 \\ V_3 \end{array}\right]$$

Or, more concisely,
$$\{P\} = [a]^{\mathrm{T}}\{S\} \tag{9.1}$$

which relates the nodal forces $\{P\}$ to the element forces $\{S\}$ for the whole structure.

Internal element equilibrium

Internal equilibrium can be explained most easily by considering an axial force element. For static equilibrium, the axial forces at each end will be equal in magnitude and opposite in direction. If the element is pin-ended and has a uniform cross-sectional area, A, then for equilibrium within the element

$$A\sigma_x = U, \tag{9.2}$$

in which the axial stress σ_x is taken to be constant over the cross-section.

9.5.2. Compatibility

External nodal compatibility

The physical interpretation of external compatibility is that any displacement pattern is not accompanied with voids or overlaps occurring between previously continuous members. In fea. this requirement is usually only satisfied at the nodes. Often it is only the displacement field which is continuous at the nodes, and not an element's first or higher order displacement derivatives. Figure 9.20 shows quadratically varying displacement fields for two adjoining quadrilateral elements and serves to illustrate these limitations.

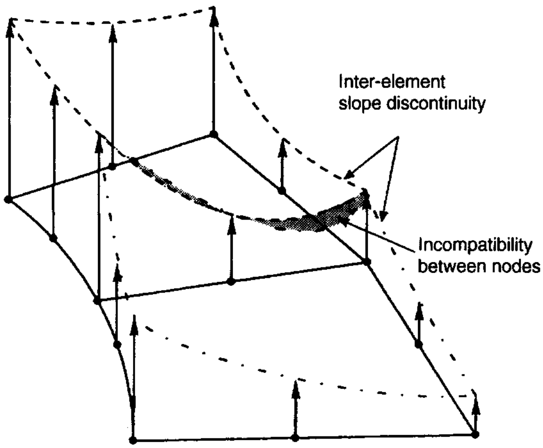

Fig. 9.20. Quadrilateral elements with quadratically varying displacement fields.

External, nodal, displacement compatibility will be shown to be automatically satisfied by a system of nodal displacements. For the simple frame shown in Fig. 9.19, the nodal displacement column matrix is

$$\{p\} = \{p_1 p_2 p_3\} = \{u_1 v_1, u_2 v_2, u_3 v_3\}$$

and the element displacement column matrix is

$$\{s\} = \{\{s^{(a)}\}\{s^{(b)}\}\{s^{(c)}\}\} = \{u_1 v_1 u_2 v_2, u_2 v_2 u_3 v_3, u_1 v_1 u_3 v_3\}$$

It follows from the above that external, nodal, compatibility is satisfied by forming the relationship between the element and nodal displacements as

$$
\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \hline u_2 \\ v_2 \\ u_3 \\ v_3 \\ \hline u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}
$$

Or, more concisely, $\{s\} = [a]\{p\}$ (9.3)

which relates all the element displacements $\{s\}$ to the nodal displacements $\{p\}$ for the whole structure.

Internal element compatibility

Again, for simplicity consider an axial force element. For the displacement within such an element not to introduce any voids or overlaps the displacement along the element, u, needs to be a continuous function of position, x. The compatibility condition is satisfied by

$$\partial u/\partial x = \varepsilon_x \tag{9.4}$$

9.5.3. Stress/strain law

Assuming for simplicity the material behaviour to be homogeneous, isotropic and linearly elastic, then Hooke's law applies giving, for a one-dimensional stress system in the absence of thermal strain,

$$\varepsilon_x = \sigma_x/E \tag{9.5}$$

in which E is the empirical modulus of elasticity.

9.5.4. Force/displacement relation

Combining eqns. (9.2), (9.4) and (9.5) and taking u to be a function of x only, gives

$$U/A = \sigma_x = \sigma_x E = E\,du/dx$$

Or,

$$U\,dx = AE\,du$$

Integrating, and taking $u(0) = u_i$ and $u(L) = u_j$, corresponding to displacements at nodes i and j of an axial force element of length L, gives the force/displacement relationship

$$U = AE(u_j - u_i)/L \tag{9.6}$$

in which $(u_j - u_i)$ denotes the deformation of the element. Thus the force/displacement relationship for an axial force element has been derived from equilibrium, compatibility and stress/strain arguments.

9.6. The principle of virtual work

In the previous section the three basic arguments of equilibrium, compatibility and constitutive relations were invoked and, in the subsequent sections, it will be seen how these arguments can be used to derive rod and simple beam element equations. However, some situations, for example, may require elements of non-uniform cross-section or representation of complex geometry, and are not amenable to solution by this approach. In such situations, alternative approaches using energy principles are used, which allow the field variables to be represented by approximating functions whilst still satisfying the three fundamental arguments. Amongst the number of energy principles which can be used, the one known as the *principle of virtual work* will be considered here.

The equation of the principle of virtual work

Virtual work is produced by perturbing a system slightly from an equilibrium state. This can be achieved by allowing small, kinematically possible displacements, which are not necessarily real, and hence are *virtual displacements*. In the following brief treatment the corresponding equation of virtual work is derived by considering the linearly elastic, uniform cross-section, axial force element in Fig. 9.21. For a more rigorous treatment the reader is referred to Ref. 8 (p. 350). In Fig. 9.21 the nodes are shown detached to distinguish between the nodal and element quantities.

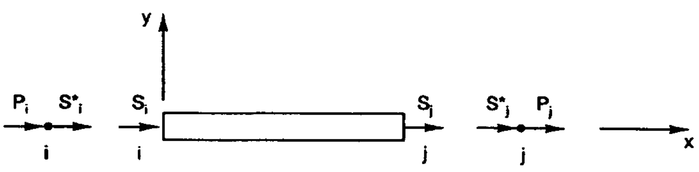

Fig. 9.21. Axial force element, shown with detached nodes.

Giving the nodal points virtual displacements $\overline{u}_i$ and $\overline{u}_j$, the virtual work for the two nodal points is

$$\overline{W} = (P_i + S_i^*)\overline{u}_i + (P_j + S_j^*)\overline{u}_j \tag{9.7}$$

This virtual work must be zero since the two nodal points are rigid bodies. It follows, since the virtual displacements are arbitrary and independent, that

$$P_i + S_i^* = 0 \quad \text{or} \quad P_i = -S_i^*$$

and

$$P_j + S_j^* = 0 \quad \text{or} \quad P_j = -S_j^*$$

which, for the single element, are the equations of external equilibrium.

Applying Newton's third law, the forces between the nodes and element are related as

$$S_i^* = -S_i \quad \text{and} \quad S_j^* = -S_j \tag{9.8}$$

Substituting eqns. (9.8) into eqn. (9.7) gives

$$\begin{aligned} \overline{W} = 0 &= (P_i - S_i)\overline{u}_i + (P_j - S_j)\overline{u}_j \\ &= (P_i\overline{u}_i + P_j\overline{u}_j) - (S_i\overline{u}_i + S_j\overline{u}_j) \end{aligned} \tag{9.9}$$

The first quantity $(P_i\overline{u}_i + P_j\overline{u}_j)$, to first order approximation assuming linearly elastic behaviour represents the virtual work done by the applied external forces, denoted as $\overline{W}_e$. The second quantity, $(S_i\overline{u}_i + S_j\overline{u}_j)$, again to first order approximation represents the virtual work done by element internal forces, denoted as $\overline{W}_i$. Hence, eqn. (9.9) can be abbreviated to

$$0 = \overline{W}_e - \overline{W}_i \tag{9.10}$$

which is the equation of the principle of virtual work for a deformable body.

The external virtual work will be found from the product of external loads and corresponding virtual displacements, recognising that no work is done by reactions since they are associated with suppressed dof. The internal virtual work will be given by the strain energy, expressed using real stress and virtual strain (arising from virtual displacements), as

$$\overline{W}_i = \int_v \overline{\varepsilon}\sigma \, dv \tag{9.11}$$

which, for the case of a prismatic element with constant stress and strain over the volume, becomes

$$\overline{W}_i = \overline{\varepsilon}\sigma AL$$

9.7. A rod element

The formulation of a rod element will be considered using two approaches, namely the use of fundamental equations, based on equilibrium, compatibility and constitutive (i.e. stress/strain law), arguments and use of the principle of virtual work equation.

9.7.1. Formulation of a rod element using fundamental equations

Consider the structure shown in Fig. 9.18, for which the deformations (derived from the displacements), member forces, stresses and reactions are required. Idealising the structure such that all the members and loads are taken to be planar, and all the joints to act as frictionless hinges, i.e. pinned, and hence incapable of transmitting moments, the corresponding behaviour can be represented as an assemblage of rod finite elements. A typical element is shown in Fig. 9.22, for which the physical and material properties are taken to be constant throughout the element. Changes in properties, and load application are only admissible at nodal positions, which occur only at the extremities of the elements. Each node is considered to have two translatory freedoms, i.e. two dof., namely u and v displacements in the element x and y directions, respectively.

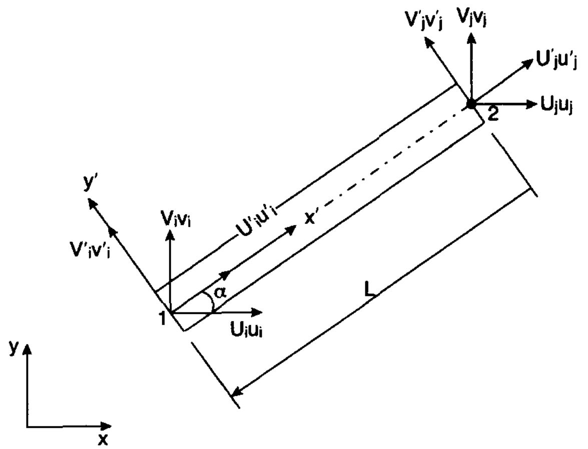

Fig. 9.22. An axial force rod element.

Element stiffness matrix in local coordinates

Quantities in the element coordinate directions are denoted with a prime ($'$), to distinguish from those with respect to the global coordinates. Displacements in the local element x' direction will cause an elongation of the element of $u'_j - u'_i$, with corresponding strain, $(u'_j - u'_i)/L$. Assuming Hookean behaviour, the element loads in the positive, local, x' direction will hence be given as

$$U'_i = AE(u'_i - u'_j)/L \quad \text{and} \quad U'_j = AE(u'_j - u'_i)/L$$

which are force/displacement relations similar to eqn. (9.6), and hence satisfy internal element equilibrium, compatibility and the appropriate stress/strain law.

In isolation, the element will not have any stiffness in the local y' direction. However, stiffness in this direction will arise from assembly with other elements with different inclinations. The element force/displacement relation can now be written in matrix form, as

$$\begin{bmatrix} U'_i \\ V'_i \\ U'_j \\ V'_j \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{bmatrix} \tag{9.12}$$

Or, more concisely, $$\{S'^{(e)}\} = [k'^{(e)}]\{s'^{(e)}\} \tag{9.13}$$

from which the *element stiffness* matrix with respect to local coordinates is:

$$[k'^{(e)}] = \frac{AE}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \tag{9.14}$$

Element stress matrix in local coordinates

For a pin-jointed frame the only significant stress will be axial. Hence, with respect to local coordinates, the axial stress for a rod element will be given as:

$$\sigma^{(e)} = \frac{E}{L}[-1 \quad 0 \quad 1 \quad 0]\begin{bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{bmatrix} \tag{9.15}$$

Or, more concisely, $$\sigma^{(e)} = [H'^{(e)}]\{s'^{(e)}\} \tag{9.16}$$

from which the stress matrix with respect to local coordinates is:

$$[H'^{(e)}] = \frac{E}{L}[-1 \quad 0 \quad 1 \quad 0] \tag{9.17}$$

Transformation of displacements and forces

To enable assembly of contributions from each constituent element meeting at each joint, it is necessary to transform the force/displacement relationships to some global coordinate system, by means of a transformation matrix $[T^{(e)}]$. This matrix is derived by establishing the relationship between the displacements (or forces), in local coordinates x', y', and those in global coordinates x,y. Note that the element inclination α, is taken to be positive when acting clockwise viewed from the origin along the positive z-axis, and is measured from the positive global x-axis. With reference to Fig. 9.23, for node i,

$$u'_i = u_i \cos\alpha + v_i \sin\alpha, \quad \text{and} \quad v'_i = -u_i \sin\alpha + v_i \cos\alpha.$$

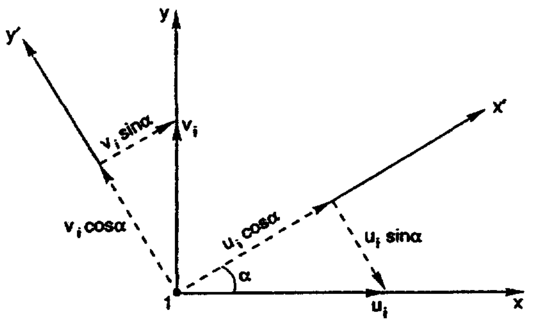

Fig. 9.23. Transformation of displacements.

Similarly, for node j,

$$u'_j = u_j \cos\alpha + v_j \sin\alpha, \quad \text{and} \quad v'_j = -u_j \sin\alpha + v_j \cos\alpha.$$

Writing in matrix form the above becomes:

$$\begin{bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \\ 0 & 0 & -\sin\alpha & \cos\alpha \end{bmatrix}\begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{bmatrix}$$

Or, more concisely, $$\{S'^{(e)}\} = [T^{(e)}]\{s^{(e)}\} \tag{9.18}$$

in which the transformation matrix is:

$$[T^{(e)}] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 \\ 0 & 0 & \cos\alpha & \sin\alpha \\ 0 & 0 & -\sin\alpha & \cos\alpha \end{bmatrix} \tag{9.19}$$

The same transformation enables the relationship between member loads in local and global coordinates to be written as:

$$\{S'^{(e)}\} = [T^{(e)}]\{S^{(e)}\} \tag{9.20}$$

Expressing eqn. (9.20) in terms of the member loads with respect to the required global coordinates, we obtain:

$$\{S^{(e)}\} = [T^{(e)}]^{-1}\{S'^{(e)}\}$$

Substituting from eqn. (9.13) gives:

$$\{S^{(e)}\} = [T^{(e)}]^{-1}[k'^{(e)}]\{s'^{(e)}\}$$

Further, substituting from eqn. (9.18) gives:

$$\{S^{(e)}\} = [T^{(e)}]^{-1}[k'^{(e)}][T^{(e)}]\{s^{(e)}\}$$

It can be shown, by equating work done in the local and global coordinates systems, that

$$[T^{(e)}]^{\mathrm{T}} = [T^{(e)}]^{-1}$$

(This property of the transformation matrix, $[T^{(e)}]$, whereby the inverse equals the transpose is known as orthogonality.) Hence, element loads are given by:

$$\{S^{(e)}\} = [T^{(e)}]^{\mathrm{T}}[k'^{(e)}][T^{(e)}]\{s^{(e)}\}$$

Or, more simply

$$\{S^{(e)}\} = [k'^{(e)}]\{s^{(e)}\} \tag{9.21}$$

in which the element stiffness matrix in global coordinates is

$$[k^{(e)}] = [T^{(e)}]^{\mathrm{T}}[k'^{(e)}][T^{(e)}] \tag{9.22}$$

Element stiffness matrix in global coordinates

Substituting from eqns. (9.14) and (9.19) into eqn. (9.22), transposing the transformation matrix and performing the triple matrix product gives the element stiffness matrix in global coordinates as:

$$[k^{(e)}] = \frac{AE}{L}\begin{bmatrix} \cos^2\alpha & \cos\alpha\sin\alpha & -\cos^2\alpha & -\cos\alpha\sin\alpha \\ -\sin\alpha\cos\alpha & \sin^2\alpha & -\sin\alpha\cos\alpha & -\sin^2\alpha \\ -\cos^2\alpha & -\cos\alpha\sin\alpha & \cos^2\alpha & \cos\alpha\sin\alpha \\ -\sin\alpha\cos\alpha & -\sin^2\alpha & \sin\alpha\cos\alpha & \sin^2\alpha \end{bmatrix} \tag{9.23}$$

Element stress matrix in global coordinates

The element stress matrix found in local coordinates, eqn. (9.17), can be transformed to global coordinates by substituting from eqn. (9.18) into eqn. (9.16) to give

$$\sigma^{(e)} = [H'^{(e)}][T^{(e)}]\{s^{(e)}\} \tag{9.24}$$

in which the element stress matrix in global coordinates is

$$[H^{(e)}] = [H'^{(e)}][T^{(e)}] \tag{9.25}$$

Substituting from eqn. (9.17) and (9.19) into eqn. (9.25) gives the element stress matrix in global coordinates as

$$[H^{(e)}] = E^{(e)}[-\cos\alpha \quad -\sin\alpha \quad \cos\alpha \quad \sin\alpha]^{(e)}/L^{(e)} \tag{9.26}$$

Formation of structural governing equation and assembled stiffness matrix

With reference to § 9.5, external nodal equilibrium is satisfied by relating the nodal loads, $\{P\}$, to the element loads, $\{S\}$, via

$$\{P\} = [a]^{\mathrm{T}}\{S\} \tag{9.1}$$

Similarly, external, nodal, compatibility is satisfied by relating the element displacements, $\{s\}$, to the nodal displacements, $\{p\}$, via

$$\{s\} = [a]\{p\} \tag{9.3}$$

Substituting from eqn. (9.3) into eqn. (9.21) for all elements in the structure, gives:

$$\{S\} = [k][a]\{p\} \tag{9.27}$$

in which $[k]$ is the *unassembled stiffness* matrix. Premultiplying the above by $[a]^{\mathrm{T}}$ and substituting from eqn. (9.1) gives:

$$\{P\} = [a]^{\mathrm{T}}[k][a]\{p\}$$

Or, more simply

$$\{P\} = [K]\{p\} \tag{9.28}$$

which is the *structural governing equation* for static stress analysis, relating the nodal forces $\{P\}$ to the nodal displacements $\{p\}$ for all the nodes in the structure, in which the *structural, or assembled stiffness* matrix

$$[K] = [a]^{\mathrm{T}}[k][a] \tag{9.29}$$

9.7.2. Formulation of a rod element using the principle of virtual work equation

Here, the principle of virtual work approach, described in § 9.6, will be used to formulate the equations for an axial force rod element. As described, the approach permits the displacement field to be represented by approximating functions, known as *interpolation* or *shape functions*, a brief description of which follows.

Shape functions

As the name suggests shape functions describe the way in which the displacements are interpolated throughout the element and often take the form of polynomials, which will be complete to some degree. The terms required to form complete linear, quadratic and cubic, etc., polynomials are given by Pascal's triangle and tetrahedron for two- and three dimensional elements, respectively. As well as completeness, there are other considerations to be made when choosing polynomial terms to ensure the element is well behaved, and the reader is urged to consult detailed texts.[6] One consideration, which will become apparent, is that the total number of terms in an interpolation polynomial should be equal to the number of dof. of the element.

Consider the axial force rod element shown in Fig. 9.24, for which the local and global axes have been taken to coincide. The purpose is to simplify the appearance of the equations by avoiding the need for the prime in denoting local coordinate dependent quantities. This element has only two nodes and each is taken to have only an axial dof. The total of only two dof. for this element limits the displacement interpolation function to a linear polynomial, namely

$$u(x) = \alpha_1 + \alpha_2 x$$

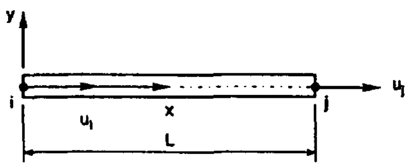

Fig. 9.24. Axial force rod element aligned with global x-axis.

where α_1 and α_2, to be determined, are known as *generalised coefficients*, and are dependent on the nodal displacements and coordinates.
Writing in matrix form

$$u(x) = [1 \quad x] \begin{bmatrix} \alpha_1 \\ \alpha_2 \end{bmatrix}$$

Or, more concisely,

$$u(x) = \lfloor x \rfloor \{\alpha\} \tag{9.30}$$

At the nodal points, $u(0) = u_i$ and $u(L) = u_j$.

Substituting into eqn. (9.30) gives

$$\begin{bmatrix} u_i \\ u_j \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 1 & L \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \end{bmatrix}$$

Or, more concisely

$$\{u\} = [A]\{\alpha\}$$

The column matrix of generalised coefficients, $\{\alpha\}$, is obtained by evaluating

$$\{\alpha\} = [A]^{-1}\{u\} \tag{9.31}$$

for which the required inverse of matrix $[A]$, i.e. $[A]^{-1}$ is obtained using standard matrix inversion whereby

$$[A]^{-1} = \text{adj } [A]/ \det [A]$$

in which $\text{adj } [A] = [C]^{\mathrm{T}}$, where $[C]$ is the cofactor matrix of $[A]$

i.e. $$\text{adj } [A] = \begin{bmatrix} L & -1 \\ 0 & 1 \end{bmatrix}^{\mathrm{T}} = \begin{bmatrix} L & 0 \\ -1 & 1 \end{bmatrix}$$

and $\det[A] = 1 \times L - 0 \times 1 = L$

Hence,
$$[A]^{-1} = \frac{1}{L}\begin{bmatrix} L & 0 \\ -1 & 1 \end{bmatrix}$$

Substituting eqn. (9.31) into eqn. (9.30) and utilising the above result for $[A]^{-1}$, gives

$$\begin{aligned} u(x) &= [1 \quad x]\frac{1}{L}\begin{bmatrix} L & 0 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_i \\ u_j \end{bmatrix} \\ &= \frac{1}{L}[L - x, x]\begin{bmatrix} u_i \\ u_j \end{bmatrix} = [1 - x/L, x/L]\begin{bmatrix} u_i \\ u_j \end{bmatrix} \\ &= [N]\{u\} \end{aligned} \tag{9.32}$$

in which $[N]$ is the matrix of shape functions. In this case, $N_1 = 1 - x/L$ and $N_2 = x/L$, and hence vary linearly over the element, as shown in Fig. 9.25. Note that the shape functions have the value unity at the node corresponding to the nodal displacement being interpolated and zero at all other nodes (in this case at the only other node), and is a property of all shape functions.

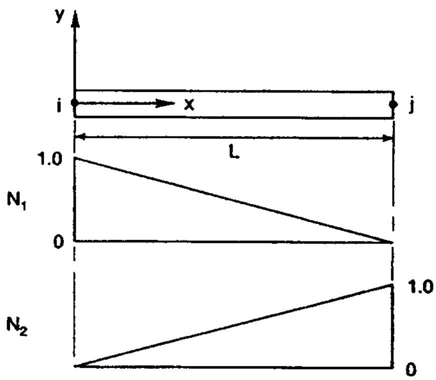

Fig. 9.25. Shape functions for the axial force rod element.

Element stiffness matrix in local coordinates

Consider the axial force element shown in Fig. 9.24. The only strain present will be a direct strain in the axial direction and is given by eqn. (9.4) as

$$\varepsilon_x = \varepsilon = \partial u/\partial x$$

Substituting from eqn. (9.32), gives

$$\varepsilon = \partial[N]\{u\}/\partial x = [B]\{u\} \tag{9.33}$$

and, taking the virtual strain to have a similar form to the real strain, gives

$$\bar{\varepsilon} = [B]\{\bar{u}\} \tag{9.34}$$

where

$$[B] = \partial[N]/\partial x$$

In the present case of the two-node linear rod element, eqn. (9.32) shows $[N] = \frac{1}{L}[L - x, x]$, and hence

$$[B] = \frac{1}{L}[-1 \;\; 1] \tag{9.35}$$

Note that in this case the derivative matrix $[B]$ contains only constants and does not involve functions of x and hence the strain, given by eqn. (9.33), will be constant along the length of the rod element.

Assuming Hookean behaviour and utilising eqn. (9.33)

$$\sigma = E[B]\{u\} \tag{9.36}$$

It follows for the linear rod element that the stress will be constant and is given as

$$\sigma = \frac{E}{L}[-1 \;\; 1]\begin{bmatrix} u_i \\ u_j \end{bmatrix} = \frac{E}{L}(u_j - u_i)$$

in which $(u_j - u_i)/L$ is the strain.

Substituting the expression for virtual strain from eqn. (9.34) and the real stress from eqn. (9.36) into eqn. (9.11) gives the internal virtual work as

$$\overline{W}_i = \int_v \bar{\varepsilon}\sigma \, dv = \int_v \{\bar{u}\}^{\mathrm{T}}[B]^{\mathrm{T}}E[B]\{u\}\, dv$$

Since the real and virtual displacements are constant, they can be taken outside the integral, to give

$$\overline{W}_i = \{\bar{u}\}^{\mathrm{T}} \int_v [B]^{\mathrm{T}}E[B]\, dv\, \{u\} \tag{9.37}$$

The external virtual work will be given by

$$\overline{W}_e = \{\bar{u}\}^{\mathrm{T}}\{P\} \tag{9.38}$$

Substituting from eqns. (9.37) and (9.38) into the equation of the principle of virtual work, eqn. (9.10) gives

$$0 = \{\bar{u}\}^{\mathrm{T}}\{P\} - \{\bar{u}\}^{\mathrm{T}} \int_v [B]^{\mathrm{T}}E[B]\, dv\, \{u\}$$

Or,

$$= \{\bar{u}\}^{\mathrm{T}}(\{P\} - \int_v [B]^{\mathrm{T}}E[B]\, dv\, \{u\})$$

The virtual displacements, $\{\bar{u}\}$, are arbitrary and nonzero, and hence the quantity in parentheses must be zero,

i.e.

$$\{P\} = \int_v [B]^{\mathrm{T}}E[B]\, dv\, \{u\} = [k'^{(e)}]\{u\} \tag{9.39}$$

where $$[k'^{(e)}] = \int_v [B]^{\mathrm{T}} E [B]\, dv \tag{9.40}$$

Evaluating the element stiffness matrix $[k'^{(e)}]$ for the linear rod element by substituting from eqns. (9.35) gives

$$[k'^{(e)}] = \int_v \frac{1}{L}\begin{bmatrix} -1 \\ 1 \end{bmatrix} E \frac{1}{L}[-1 \quad 1]\, dv$$
$$= \frac{E}{L^2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \int_v dv$$

For a prismatic element $\int_v dv = AL$, and the element stiffness matrix becomes

$$[k'^{(e)}] = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{9.41}$$

Expanding the force/displacement eqn. (9.39) to include terms associated with the y- direction, requires the insertion of zeros in the stiffness matrix of eqn. (9.41) and hence becomes identical to eqn. (9.14).

Element stress matrix in local coordinates

For the case of a linear rod element, substituting from eqn. (9.35) into eqn. (9.36) gives the element stress as

$$\sigma^{(e)} = \frac{E}{L}[-1 \ 1]\{u\} \tag{9.42}$$

Again, by inserting zeros in the matrix, to accommodate terms associated with the y- direction, eqn. (9.42) becomes identical to eqn. (9.15).

Transformation of element stiffness and stress matrices to global coordinates

The element stiffness and stress matrices obtained above can be transformed from local to global coordinates using the procedures of § 9.7.1 to give the results previously obtained, namely the stiffness matrix of eqn. (9.23) and stress matrix of eqn. (9.26).

Formation of structural governing equation and assembled stiffness matrix

Section 9.7.1 has covered the combination of individual element stiffness contributions, necessary to analyse an assemblage of elements representing a complete framework. Equilibrium and compatibility arguments were used to form the structural governing eqn. (9.28) and hence the assembled stiffness matrix, eqn. (9.29). Now, the alternative principle of virtual work will be used to derive the same equations.

Eqn. (9.37) gives the element internal virtual work in local coordinates as

$$W_i^{(e)} = \{u'^{(e)}\}^{\mathrm{T}} \int_v [B^{(e)}]^{\mathrm{T}} E^{(e)} [B^{(e)}] dv \{u'^{(e)}\}$$

Summing all such contributions for the entire structure of m elements, gives

$$\overline{W}_i = \sum_{e=1}^{m}(\{\overline{u}'^{(e)}\}^{\mathrm{T}} \int_v [B^{(e)}]^{\mathrm{T}} E^{(e)} [B^{(e)}]\, dv\, \{u'^{(e)}\}) \tag{9.43}$$

Summing the contributions over all n nodes, the external virtual work will be given by

$$\overline{W}_e = \sum_{i=1}^{n} \overline{p}_i P_i = \{\overline{p}\}^{\mathrm{T}}\{P\} \tag{9.44}$$

where $\{\overline{p}\}$ is the column matrix of all nodal virtual displacements for the structure and $\{P\}$ is the column matrix of all nodal forces. Substituting from eqns. (9.43) and (9.44) into the equation of the principle of virtual work, eqn. (9.10) gives

$$0 = \{\overline{p}\}^{\mathrm{T}}\{P\} - \sum_{e=1}^{m}(\{\overline{u}'^{(e)}\}^{\mathrm{T}} \int_v [B^{(e)}]^{\mathrm{T}} E^{(e)} [B^{(e)}]\, dv\, \{u'^{(e)}\}) \tag{9.45}$$

Relating the virtual displacements in local and global coordinates via the transformation matrix $[T^{(e)}]$, gives

$$\{\overline{u}^{(e)}\} = [T^{(e)}]\{\overline{p}^{(e)}\} \quad \text{and} \quad \{\overline{u}'^{(e)}\}^{\mathrm{T}} = \{\overline{p}^{(e)}\}^{\mathrm{T}}[T^{(e)}]^{\mathrm{T}}$$

Summing the contributions and recalling $\{\overline{p}\}$ denotes the nodal displacements for the entire structure, gives

$$\sum_{e=1}^{m}\{\overline{u}'^{(e)}\}^{\mathrm{T}} = \{\overline{p}\}^{\mathrm{T}}\sum_{e=1}^{m}[T^{(e)}]^{\mathrm{T}} \quad \text{and} \quad \sum_{e=1}^{m}\{\overline{u}'^{(e)}\} = \sum_{e=1}^{m}([T^{(e)}])\{p\}$$

Hence, eqn. (9.45) can be re-written as

$$\begin{aligned}\{\overline{p}\}^{\mathrm{T}}\{P\} &= \{\overline{p}\}^{\mathrm{T}}(\sum_{e=1}^{m}[T^{(e)}]^{\mathrm{T}} \int_v [B^{(e)}]^{\mathrm{T}} E^{(e)} [B^{(e)}]\, dv\, [T^{(e)}])\{p\} \\ &= \{\overline{p}\}^{\mathrm{T}}\sum_{e=1}^{m}[k^{(e)}]\{p\} = \{\overline{p}\}^{\mathrm{T}}[K]\{p\}\end{aligned} \tag{9.46}$$

where
$$[k^{(e)}] = [T^{(e)}]^{\mathrm{T}} \int_v [B^{(e)}]^{\mathrm{T}} E^{(e)} [B^{(e)}]\, dv\, [T^{(e)}]$$

and the assembled stiffness matrix

$$[K] = \sum_{e=1}^{m}[k^{(e)}] \tag{9.47}$$

It follows from eqn. (9.46) since $\{\overline{p}\}$ is arbitrary and non-zero, that

$$\{P\} = [K]\{p\}$$

which is the structural governing equation and the same as eqn. (9.28), and implies nodal force equilibrium.

9.8. A simple beam element

As with the previous treatment of the rod element, the two approaches using fundamental equations and the principle of virtual work will be employed to formulate the necessary equations for a simple beam element.

9.8.1. Formulation of a simple beam element using fundamental equations

Consider the case, similar to §9.7.1, in which the deformations, member stresses and reactions are required for planar frames, excepting that the member joints are now taken to be rigid and hence capable of transmitting moments. The behaviour of such frames can be represented as an assemblage of beam finite elements. A typical simple beam element is shown in Fig. 9.26, the physical and material properties of which are taken to be constant throughout the element. As with the previous rod element, changes in properties and load application are only admissible at nodal positions. In addition to u and v translatory freedoms, each node has a rotational freedom, θ, about the z axis, giving three dof. per node. Hence, axial, shear and flexural deformations will be represented, whilst torsional deformations which are inappropriate for planar frames will be ignored.

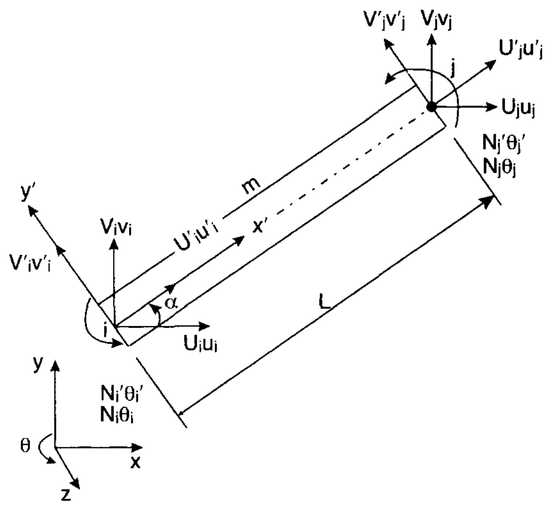

Fig. 9.26. A simple beam element.

Element stiffness matrix in local coordinates

The differential equation of flexure appropriate to a beam element can be written as

$$d^2v'/dx'^2 = N'/EI \tag{9.48}$$

in which v' denotes the displacement in the local y' direction, N' is the element moment, E is the modulus of elasticity and I is the relevant second moment of area of the beam. The first derivative of the moment with respect to distance x' along a beam is known to give the shear force, V',

i.e. $$dN'/dx' = V' \tag{9.49}$$

Similarly, the first derivative of the shear force will give the loading intensity, ω',

i.e. $$dV'/dx' = \omega' \tag{9.50}$$

Differentiating eqn. (9.48) and utilising eqn. (9.49) gives

$$d^3v'/dx'^3 = V'/EI \tag{9.51}$$

Differentiating again and utilising eqn. (9.50) gives

$$d^4v'/dx'^4 = \omega'/EI \tag{9.52}$$

Integrating eqn. (9.52), recalling that loads can only be applied at the nodes, and hence $\omega' = 0$, gives

$$d^3v'/dx'^3 = C_1 = V'/EI, \quad \text{(from eqn. 9.51)} \tag{9.53}$$

Further integration gives

$$d^2v'/dx'^2 = C_1x' + C_2 = N'/EI, \quad \text{(from eqn. 9.48)} \tag{9.54}$$

$$dv'/dx' = C_1x'^2/2 + C_2x' + C_3 = \theta' \tag{9.55}$$

$$\text{and} \quad v' = C_1x'^3/6 + C_2x'^2/2 + C_3x' + C_4 \tag{9.56}$$

With reference to Fig. 9.26, it can be seen that

$$v'(0) = v'_i, \quad v'(L) = v'_j, \quad dv'/dx'(0) = \theta'_i, \quad dv'/dx'(L) = \theta'_j$$

It follows from eqn. (9.56) that $$v'_i = C_4 \tag{9.57}$$

from eqn. (9.55) $$\theta'_i = C_3 \tag{9.58}$$

from eqn. (9.56)

$$v'_j = C_1L^3/6 + C_2L^2/2 + C_3L + C_4 = C_1L^3/6 + C_2L^2/2 + \theta'_iL + v'_i \tag{9.59}$$

and from eqn. (9.55)

$$\theta'_j = C_1L^2/2 + C_2L + C_3 \quad = C_1L^2/2 + C_2L + \theta'_i \tag{9.60}$$

An expression for C_2 can now be obtained by multiplying eqn. (9.60) throughout by $L/3$ and subtracting the result from eqn. (9.59), (to eliminate C_1), to give

$$v'_j - \theta'_jL/3 = C_2(L^2/2 - L^2/3) + \theta'_i(L - L/3) + v'_i = C_2L^2/6 + \theta'_i2L/3 + v'_i$$

Rearranging, C_2 $$= 6(v'_j - v'_i)/L^2 - 6(\theta'_jL/3 + \theta'_i2L/3)/L$$

$$= 6(-v'_i + v'_j)/L^2 - 2(2\theta'_i + \theta'_j)/L \tag{9.61}$$

Rearranging eqn. (9.60) and substituting from eqn. (9.61) gives

$$C_1 = (2/L^2)[(\theta'_j - \theta'_i) - 6(-v'_i + v'_j)/L + 2(2\theta'_i + \theta'_j)]$$

$$= 12(v'_i - v'_j)/L^3 + 6(\theta'_i + \theta'_j)/L^2 \tag{9.62}$$

Substituting for constant C_1 from eqn. (9.62) into eqn. (9.53) gives shear force

$$V' = EIC_1 = 12EI(v_i' - v_j')/L^3 + 6EI(\theta_i' + \theta_j')/L^2 \tag{9.63}$$

Substituting for constants C_1 and C_2 from eqns. (9.62) and (9.63) into eqn. (9.54) gives the moment

$$\begin{aligned} N' &= EI(C_1x' + C_2) \\ &= 6EI(2x' - L)(v_i' - v_j')/L^3 + 6EIx'(\theta_i' + \theta_j')/L^2 - 2EI(2\theta_i' + \theta_j')/L \end{aligned} \tag{9.64}$$

Note that the shear force, eqn. (9.63) is independent of distance x' along the beam, i.e. constant, whilst the moment, eqn. (9.64) is linearly dependent on distance x', consistent with a beam subjected to concentrated forces.

It follows that the nodal shear force and moments are given as

$$V'(0) = V'(L) = 12EI(v_i' - v_j')/L^3 + 6EI(\theta_i' + \theta_j')/L^2 \tag{9.65}$$

$$N'(0) = 6EI(-v_i' + v_j')/L^2 - 2EI(2\theta_i' + \theta_j')/L \tag{9.66}$$

$$N'(L) = 6EI(v_i' - v_j')/L^2 + 2EI(\theta_i' + 2\theta_j')/L \tag{9.67}$$

The shear force and moments given by eqns. (9.65)–(9.67) use a Mechanics of Materials sign convention, namely, a positive shear force produces a clockwise couple and a positive moment produces sagging. To conform with the sign convention shown in Fig. 9.26, the following changes are required:

$$V_i' = -V_j' = V'(0)$$

$$N_i' = -N'(0)$$

$$N_j' = N'(L)$$

Writing in matrix form, eqns. (9.65)–(9.67) become

$$\begin{bmatrix} V_i' \\ N_i' \\ V_j' \\ N_j' \end{bmatrix} = \frac{EI}{L} \begin{bmatrix} 12/L^2 & 6/L & -12/L^2 & 6/L \\ 6/L & 4 & -6/L & 2 \\ -12/L^2 & -6/L & 12/L^2 & -6/L \\ 6/L & 2 & -6/L & 4 \end{bmatrix} \begin{bmatrix} v_i' \\ \theta_i' \\ v_j' \\ \theta_j' \end{bmatrix} \tag{9.68}$$

Combining eqn. (9.68) with eqn. (9.12) gives the matrix equation relating element axial and shear forces and moments to the element displacements as

$$\begin{bmatrix} U_i' \\ V_i' \\ N_i' \\ U_j' \\ V_j' \\ N_j' \end{bmatrix} = \begin{bmatrix} AE/L & 0 & 0 & -AE/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 4EI/L & 0 & -6EI/L^2 & 2EI/L \\ -AE/L & 0 & 0 & AE/L & 0 & 0 \\ 0 & -12EI/L^3 & -6EI/L^2 & 0 & 12EI/L^3 & -6EI/L^2 \\ 0 & 6EI/L^2 & 2EI/L & 0 & -6EI/L^2 & 4EI/L \end{bmatrix} \begin{bmatrix} u_1' \\ v_i' \\ \theta_i' \\ u_j' \\ v_j' \\ \theta_j' \end{bmatrix} \tag{9.69}$$

from which the element stiffness matrix with respect to local coordinates is:

$$[k^{(e)}] = \begin{bmatrix} AE/L & 0 & 0 & -AE/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 4EI/L & 0 & -6EI/L^2 & 2EI/L \\ -AE/L & 0 & 0 & AE/L & 0 & 0 \\ 0 & -12EI/L^3 & -6EI/L^2 & 0 & 12EI/L^3 & -6EI/L^2 \\ 0 & 6EI/L^2 & 2EI/L & 0 & -6EI/L^2 & 4EI/L \end{bmatrix} \quad (9.70)$$

Element stress matrix in local coordinates

Only bending and axial stresses will be considered, shear stresses being taken as insignificant. The points for stress calculation will be the extreme top and bottom fibres at each end of the element, which will always include the maximum stress point. With reference to Fig. 9.27, the beam element stress matrix will be

$$\{\sigma^{(e)}\} = \{\sigma_i^{\text{top}} \sigma_i^{\text{btm}} \sigma_j^{\text{top}} \sigma_j^{\text{btm}}\}$$

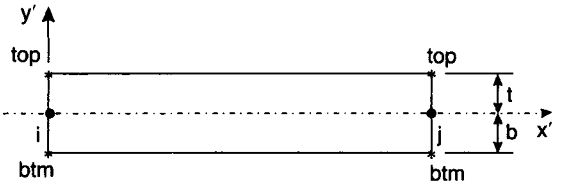

Fig. 9.27. Locations for beam element stress calculation.

Relating these stresses to the internal loads gives

$$\begin{bmatrix} \sigma_i^{\text{top}} \\ \sigma_i^{\text{btm}} \\ \sigma_j^{\text{top}} \\ \sigma_j^{\text{btm}} \end{bmatrix} = \begin{bmatrix} -1/A & 0 & t/I & 0 & 0 & 0 \\ -1/A & 0 & -b/I & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/A & 0 & -t/I \\ 0 & 0 & 0 & 1/A & 0 & b/I \end{bmatrix} \begin{bmatrix} U'_i \\ V'_i \\ N'_i \\ U'_j \\ V'_j \\ N'_j \end{bmatrix} \quad (9.71)$$

Or, more concisely, $\{\sigma^{(e)}\} = [h^{(e)}]\{S'^{(e)}\}$ (9.72)

Substituting for the element loads column matrix using a relation of the form of eqn. (9.13) gives

$$\{\sigma^{(e)}\} = [h^{(e)}][k'^{(e)}]\{s'^{(e)}\} = [H'^{(e)}]\{s'^{(e)}\}$$

which is the same form as eqn. (9.16) and $[H'^{(e)}]$ is the stress matrix with respect to local coordinates. Evaluating $[h^{(e)}][k'^{(e)}]$ gives the stress matrix as

$$[H'^{(e)}] = \frac{E}{L}\begin{bmatrix} -1 & 6t/L & 4t & 1 & 6t/L & 2t \\ -1 & -6b/L & -4b & 1 & -6b/L & -2b \\ -1 & -6t/L & -2t & 1 & -6t/L & -4t \\ -1 & 6b/L & 2b & 1 & 6b/L & 4b \end{bmatrix} \quad (9.73)$$

Transformation of displacements and loads

Relations of similar form to those of eqns. (9.18)–(9.23) but with additional rotational dof. terms, previously not included in the rod element transformation, will enable the above element stiffness and stress matrices to be transformed from local to global coordinates. The expanded form of eqn. (9.18) for the beam element will be given as

$$\begin{bmatrix} u_i \\ v_i' \\ \theta_i' \\ u_j' \\ v_j' \\ \theta_j' \end{bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\alpha & \sin\alpha & 0 \\ 0 & 0 & 0 & -\sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ \theta_i \\ u_j \\ v_j \\ \theta_j \end{bmatrix}$$

in which the transformation matrix is:

$$[T^{(e)}] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\alpha & \sin\alpha & 0 \\ 0 & 0 & 0 & -\sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{9.74}$$

Element stiffness matrix in global coordinates

$$[k^{(e)}] = \frac{E}{L}\begin{bmatrix} A\cos^2\alpha + (12I\sin^2\alpha/L^2), & & & & & \\ (A - 12I/L^2)\cos\alpha\sin\alpha, & A\sin^2\alpha + (12I\cos^2\alpha)/L^2, & & & \text{symmetric} & \\ -(6I\sin\alpha)/L, & (6I\cos\alpha)/L, & 4I, & & & \\ -A\cos^2\alpha - (12I\sin^2\alpha)/L^2, & (-A + 12I/L^2)\cos\alpha\sin\alpha, & (6I\sin\alpha)/L, & A\cos^2\alpha + (12I\sin^2\alpha)/L^2, & & \\ (-A + 12I/L^2)\cos\alpha\sin\alpha, & -A\sin^2\alpha - (12I\cos^2\alpha)/L^2, & -(6I\cos\alpha)/L, & (A - 12I/L^2)\cos\alpha\sin\alpha, & A\sin^2\alpha + (12I\cos^2\alpha)/L^2, & \\ -(6I\sin\alpha)/L, & (6I\cos\alpha)/L, & 2I, & (6I\sin\alpha)/L, & -(6I\cos\alpha)/L, & 4I \end{bmatrix} \tag{9.75}$$

Element stress matrix in global coordinates

Substituting from eqns. (9.73) and (9.74) into eqn. (9.25) gives the element stress matrix in global coordinates as:

$$[H^{(e)}] = \frac{E}{L}\begin{bmatrix} -\cos\alpha - 6t\sin(\alpha)/L & -\sin\alpha + 6t\cos(\alpha)/L & 4t & \cos\alpha + 6t\sin(\alpha)/L & \sin\alpha - 6t\cos(\alpha)/L & 2t \\ -\cos\alpha + 6b\sin(\alpha)/L & -\sin\alpha - 6b\cos(\alpha)/L & -4b & \cos\alpha - 6b\sin(\alpha)/L & \sin\alpha + 6b\cos(\alpha)/L & -2b \\ -\cos\alpha + 6t\sin(\alpha)/L & -\sin\alpha - 6t\cos(\alpha)/L & -2t & \cos\alpha - 6t\sin(\alpha)/L & \sin\alpha + 6t\cos(\alpha)/L & -4t \\ -\cos\alpha - 6b\sin(\alpha)/L & -\sin\alpha + 6b\cos(\alpha)/L & 2b & \cos\alpha + 6b\sin(\alpha)/L & \sin\alpha - 6b\cos(\alpha)/L & 4b \end{bmatrix} \tag{9.76}$$

Formation of structural governing equation and assembled stiffness matrix

Whilst the beam element matrices include rotational dof. terms, not present in the rod element matrices, the procedures of Section 9.7.1 still apply, and lead to the structural governing equation

$$\{P\} = [K]\{p\} \tag{9.28}$$

and the assembled stiffness matrix

$$[K] = [a]^{\mathrm{T}}[k][a] \tag{9.29}$$

9.8.2. *Formulation of a simple beam element using the principle of virtual work equation*

As Section 9.7.2 the principle of virtual work equation will be invoked, this time to formulate the equations for a simple beam element.

Consider the simple beam element shown in Fig. 9.28, for which the local and global axes have again been taken to coincide to avoid need for the prime and hence to simplify the appearance of the equations. The two nodes are each taken to have only normal and rotational dof. The terms associated with the omitted axial dof. have already been derived for the linear rod element in §9.7 and will be incorporated once the other terms have been derived. The total of four dof. for this beam element permits the displacement to be interpolated by a fourth order polynomial, namely

$$v(x) = \alpha_1 + \alpha_2 x/L + \alpha_3 x^2/L^2 + \alpha_4 x^3/L^3$$

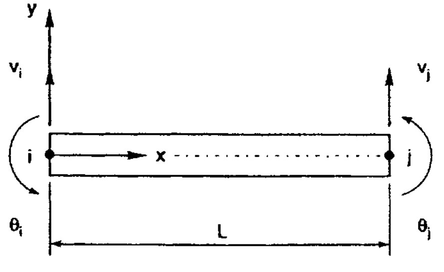

Fig. 9.28. Simple beam element, aligned with global x-axis.

where α_1 to α_4 are generalised coefficients to be determined. Utilisation of eqns. (9.48) and (9.49) shows this polynomial will provide for a linearly varying moment and constant shear force and hence will enable an exact solution for beams subjected to concentrated loads.

Writing in matrix form,

$$v(x) = [1, x/L, x^2/L^2, x^3/L^3] \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix}$$

or, more concisely

$$v(x) = [x]\{\alpha\} \tag{9.77}$$

At the nodal points, $v(0) = v_i$; $v(L) = v_j$;
and $dv/dx(0) = \theta_i$; $dv/dx(L) = \theta_j$

Substituting into eqn. (9.77) gives

$$\begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1/L & 0 & 0 \\ 1 & 1 & 1 & 1 \\ 0 & 1/L & 2/L & 3/L \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix}$$

Or, more concisely,

$$\{v\} = [A]\{\alpha\}$$

Expressing in terms of the column matrix of generalised coefficients, $\{\alpha\}$, gives

$$\{\alpha\} = [A]^{-1}\{v\} \tag{9.78}$$

Evaluation of eqn. (9.78) requires the inverse of matrix $[A]$ obtained from

$$\text{adj }[A] = [C]^{\mathrm{T}} = \begin{bmatrix} 1/L^2 & 0 & -3/L^2 & 2/L^2 \\ 0 & 1/L & -2/L & 1/L \\ 0 & 0 & 3/L^2 & -2/L^2 \\ 0 & 0 & -1/L & 1/L \end{bmatrix}^{\mathrm{T}}$$

$$= \begin{bmatrix} 1/L^2 & 0 & 0 & 0 \\ 0 & 1/L & 0 & 0 \\ -3/L^2 & -2/L & 3/L^2 & -1/L \\ 2/L^2 & 1/L & -2/L^2 & 1/L \end{bmatrix}$$

and $\det [A] = 1(1/L)(1 \times 3/L - 1 \times 2/L) \quad = 1/L^2$

Hence, $$[A]^{-1} = L^2 \begin{bmatrix} 1/L^2 & 0 & 0 & 0 \\ 0 & 1/L & 0 & 0 \\ -3/L^2 & 2/L & 3/L^2 & -1/L \\ 2/L^2 & 1/L & -2/L^2 & 1/L \end{bmatrix}$$

$$= \begin{bmatrix} 1 & 0 & 0 & 0 \\ 3 & L & 0 & 0 \\ -3 & -2L & 3 & -L \\ 2 & L & -2 & L \end{bmatrix}$$

Substituting eqn. (9.78) into eqn. (9.77) and utilising the above result for $[A]^{-1}$, gives

$$v(x) = [1, x/L, x^2/L^2, x^3/L^3] \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & L & 0 & 0 \\ -3 & -2L & 3 & -L \\ 2 & L & -2 & L \end{bmatrix} \begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix}$$

$$= [1 - 3x^2/L^2 + 2x^3/L^3, x - 2x^2/L + x^3/L^2, 3x^2/L^2 - 2x^3/L^3, -x^2/L + x^3/L^2] \begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix} \tag{9.79}$$

which has the same form as eqn. (9.32), in this case with shape functions

$$N_1 = 1 - 3x^2/L^2 + 2x^3/L^3$$
$$N_2 = x - 2x^2/L + x^3/L^2$$
$$N_3 = 3x^2/L^2 - 2x^3/L^3$$
$$N_4 = -x^2/L + x^3/L^2$$

the variation of which is shown in Fig. 9.29.

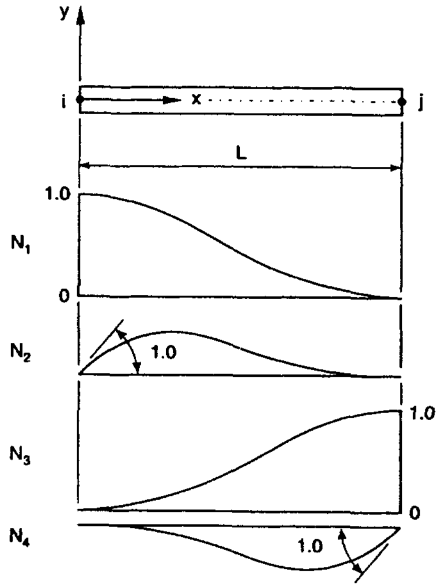

Fig. 9.29. Shape functions for a simple beam element.

Element stiffness matrix in local coordinates

Longitudinal bending stress is given by simple bending theory as

$$\sigma = My/I$$

in which, from the differential equation of flexure, eqn. (9.48),

$$M = EId^2v/dx^2$$

to give

$$\sigma = Ey\,d^2v/dx^2 \tag{9.80}$$

Substituting eqn. (9.79) into eqn. (9.80) gives

$$\sigma = Ey[(12x/L^3 - 6/L^2)(6x/L^2 - 4/L)(-12x/L^3 + 6/L^2)(6x/L^2 - 2/L)] \begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix}$$

$$= E[B]\{u\} \tag{9.81}$$

which has the same form as eqn. (9.36), except in this case

$$[B] = yd^2[N]/dx^2 \tag{9.82}$$

Assuming Hookean behaviour, the longitudinal bending strain is obtained using eqn. (9.81), as

$$\varepsilon = \sigma/E = [B]\{u\}$$

which has the same form as eqn. (9.33).

Taking the virtual longitudinal strain to be given in a form similar to the real strain, i.e.

$$\varepsilon = [B]\{u\} \tag{9.83}$$

Substituting the real stress from eqn. (9.81) and the virtual strain from eqn. (9.83) into the equation of the principle of virtual work, (9.10), gives

$$0 = \{u\}^T\{P\} - \int_v \{u\}^T[B]^T E[B]\{u\}\,dv$$

The real and virtual displacements, being constant, can be taken outside the integral, to give

$$= \{u\}^T(\{P\} - \int_v [B]^T E[B]\,dv\,\{u\})$$

The virtual displacements, $\{u\}$, are arbitrary and nonzero, hence

$$\{P\} = \int_v [B]^T E[B]\,dv\,\{u\} = [k'^{(e)}]\{u\}$$

where $[k'^{(e)}] = \int_v [B]^T E[B]\,dv$, which is an identical result to eqn. (9.40).
Substituting from eqn. (9.82) gives

$$[k'(e)] = \int_v y(d^2[N]/dx^2)^T E y(d^2[N]/dx^2)\,dv$$

$$= EI\int_0^L (d^2[N]/dx^2)^T(d^2[N]/dx^2)\,dx$$

$$= EI\int_0^L \begin{bmatrix} \frac{6}{L^2}\left(2\frac{x}{L}-1\right) \\ \frac{2}{L}\left(3\frac{x}{L}-2\right) \\ \frac{6}{L^2}\left(-2\frac{x}{L}+1\right) \\ \frac{2}{L}\left(3\frac{x}{L}-1\right) \end{bmatrix} \times \left[\frac{6}{L^2}\left(2\frac{x}{L}-1\right)\ \frac{2}{L}\left(3\frac{x}{L}-2\right)\ \frac{6}{L^2}\left(-2\frac{x}{L}+1\right)\ \frac{2}{L}\left(3\frac{x}{L}-1\right)\right] dx \tag{9.84}$$

The following gives examples of evaluating the integrals of eqn. (9.84) for two elements of the stiffness matrix, the rest are obtained by the same procedure.

$$k_{11} = EI\int_0^L \left[\frac{6}{L^2}\left(2\frac{x}{L}-1\right)\frac{6}{L^2}\left(2\frac{x}{L}-1\right)\right]dx = \frac{36EI}{L^4}\int_0^L\left(4\frac{x^2}{L^2}-4\frac{x}{L}+1\right)dx$$

$$= \frac{36EI}{L^4}\left[\frac{4x^3}{3L^2} - 2\frac{x^2}{L} + x\right] = \frac{36EI}{L^3}\left[\frac{4}{3} - 2 + 1\right] = \frac{12EI}{L^3}$$

and $k_{12} = k_{21} = EI\int_0^L \left[\frac{6}{L^2}\left(2\frac{x}{L}-1\right)\frac{2}{L}\left(3\frac{x}{L}-2\right)\right]dx = \frac{12EI}{L^3}\int_0^L \left(6\frac{x^2}{L^2}-7\frac{x}{L}+2\right)dx$

$$= \frac{12EI}{L^3}\left[\frac{2x^3}{L^2}-\frac{7x^2}{2L}+2x\right] = \frac{12EI}{L^2}\left[2-\frac{7}{2}+2\right] = \frac{6EI}{L^2}$$

Evaluation of all the integrals of eqn. (9.84) leads to the beam element flexural stiffness matrix

$$[k'^{(e)}] = \frac{EI}{L}\begin{bmatrix} 12/L^2 & 6/L & -12/L^2 & 6/L \\ 6/L & 4 & -6/L & 2 \\ -12/L^2 & -6/L & 12/L^2 & -6/L \\ 6/L & 2 & -6/L & 4 \end{bmatrix}$$

which is identical to the stiffness matrix of eqn. (9.68) derived using fundamental equations. The same arguments made in Section 9.8.1 apply with regard to including axial terms to give the force/displacement relation, eqn. (9.69), and corresponding element stiffness matrix, eqn. (9.70).

Element stress matrix in local coordinates

Bending and axial stresses are obtained using the same relations as those in §9.8.1.

Transformation of element stiffness and stress matrices to global coordinates

The element stiffness and stress matrices are transformed from local to global coordinates using the procedures of §2.4.8.1 to give the stiffness matrix of eqn. (9.75) and stress matrix of eqn. (9.76).

Formation of structural governing equation and assembled stiffness matrix

The theorem of virtual work used in §9.7.2 to formulate rod element assemblages applies to the present beam elements. It follows, therefore, that the assembled stiffness matrix will be given by eqn. (9.47). The displacement column matrices will, for beams, include rotational dof., not present for rod elements. Further, at the nodes, moment equilibrium, as well as force equilibrium, is now implied by eqn. (9.28).

9.9. A simple triangular plane membrane element

The common occurrence of thin-walled structures merits devoting attention here to their analysis. Many applications are designed on the basis of in-plane loads only with resistance arising from membrane action rather than bending. Whilst thin plates can be curved to resist normal loads by membrane action, for simplicity only planar applications will be considered here. Membrane elements can have three or four edges, which can be straight or curvilinear, however, attention will be restricted here to the simplest, triangular, membrane element.

Unlike the previous rod and beam element formulations, with which displacement fields can be represented exactly and derived from fundamental arguments, the displacement fields represented by two-dimensional elements can only be approximate, and need to be derived using an energy principle. Here, the principle of virtual work will be invoked to derive the membrane element equations.

9.9.1. Formulation of a simple triangular plane membrane element using the principle of virtual work equation

With reference to Fig. 9.30, each node of the triangular membrane element has two dof., namely u and v displacements in the global x and y directions, respectively. The total of six dof. for the element limits the u and v displacement to linear interpolation. Hence

$$u(x, y) = \alpha_1 + \alpha_2 x + \alpha_3 y$$

and

$$v(x, y) = \alpha_4 + \alpha_5 x + \alpha_6 y$$

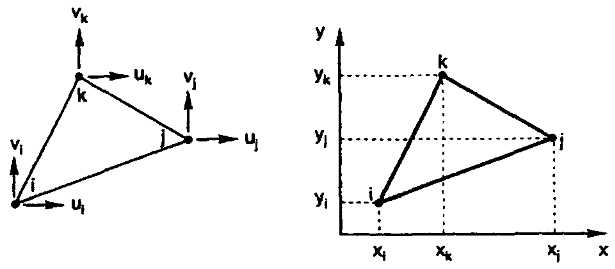

Fig. 9.30. Triangular plane membrane element.

or, in matrix form

$$\begin{bmatrix} u(x, y) \\ v(x, y) \end{bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{bmatrix} \tag{9.85}$$

At the nodal points,

$$u(x_i, y_i) = u_i, \qquad v(x_i, y_i) = v_i,$$
$$u(x_j, y_j) = u_j, \qquad v(x_j, y_j) = v_j,$$

and

$$u(x_k, y_k) = u_k, \qquad v(x_k, y_k) = v_k$$

Substituting into eqn. (9.85) gives

$$\begin{bmatrix} u_i \\ u_j \\ u_k \\ v_i \\ v_j \\ v_k \end{bmatrix} = \left[\begin{array}{ccc|ccc} 1 & x_i & y_i & 0 & 0 & 0 \\ 1 & x_j & y_j & 0 & 0 & 0 \\ 1 & x_k & y_k & 0 & 0 & 0 \\ \hline 0 & 0 & 0 & 1 & x_i & y_i \\ 0 & 0 & 0 & 1 & x_j & y_j \\ 0 & 0 & 0 & 1 & x_k & y_k \end{array}\right] \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{bmatrix} \tag{9.86}$$

Or, more concisely, $\{p\} = [A]\{\alpha\}$

Similar to the previous sections, the column matrix of generalised coefficients, $\{\alpha\}$, is obtained by evaluating

$$\{\alpha\} = [A]^{-1}\{p\} \tag{9.87}$$

where, by arranging the dof. in the above sequence enables suitable partitioning of $[A]$ and minimises the effort required to obtain the inverse. Unlike the previous treatment of the rod and beam element, the evaluation of $[A]^{-1}$ is delayed until Example 9.5. The result, however, is given by eqn. (9.88). It is hoped this departure will enable the element formulation to be more easily assimilated.

$$[A]^{-1} = \frac{1}{2a}\begin{bmatrix} x_2y_3 - x_2y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 & 0 & 0 & 0 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 & 0 & 0 & 0 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ 0 & 0 & 0 & y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ 0 & 0 & 0 & x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix} \tag{9.88}$$

where a is equal to the area of the element.

Substituting from eqn. (9.87) into eqn. (9.85), utilising the result for $[A]^{-1}$, i.e. eqn. (9.88), and writing concisely the result of the matrix multiplication, gives

$$\begin{bmatrix} u(x, y) \\ v(x, y) \end{bmatrix} = \begin{bmatrix} N_i & N_j & N_k & 0 & 0 & 0 \\ 0 & 0 & 0 & N_i & N_j & N_k \end{bmatrix} \begin{bmatrix} u_i \\ u_j \\ u_k \\ v_k \\ v_j \\ v_k \end{bmatrix}$$

where the shape functions are given as

$$\begin{aligned} N_i &= \frac{1}{2a}[x_jy_k - x_ky_j + (y_j - y_k)x + (x_k - x_j)y] \\ N_j &= \frac{1}{2a}[x_ky_i - x_iy_k + (y_k - y_i)x + (x_i - x_k)y] \\ N_k &= \frac{1}{2a}[x_iy_j - x_jy_i + (y_i - y_j)x + (x_j - x_i)y] \end{aligned} \tag{9.89}$$

Note that the shape functions of eqns. (9.89) are linear in x and y. Further, evaluation of eqns. (9.89) shows that shape function $N_i(x_i, y_i) = 1$ and $N_i(x, y) = 0$ at nodes j and k, and at all points on the line joining these nodes. Similarly, $N_j(x_j, y_j) = 1$ and $N_k(x_k, y_k) = 1$, and equal zero at, and on the line between, the other nodes.

Formulation of element stiffness matrix

For plane stress analysis, the strain/displacement relations are

$$\varepsilon_{xx} = \partial u/\partial x, \quad \varepsilon_{yy} = \partial v/\partial y, \quad \varepsilon_{xy} = \partial u/\partial y + \partial v/\partial x$$

where ε_{xx} and ε_{yy} are the direct strains parallel to the x and y axes, respectively, and ε_{xy} is the shear strain in the xy plane. Writing in matrix form gives

$$\{\varepsilon\} = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}$$

Substituting from eqn. (9.85) and performing the partial differentiation, the above becomes

$$\{\varepsilon\} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{bmatrix}$$

Substituting from eqn. (9.87) gives

$$\{\varepsilon\} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} [A]^{-1} \begin{bmatrix} u_i \\ u_j \\ u_k \\ v_i \\ v_j \\ v_k \end{bmatrix}$$

Or, more concisely, $\{\varepsilon\} = [B]\{u\}$ (9.90)

where $$[B] = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} [A]^{-1} \tag{9.91}$$

Note that matrix $[B]$ is independent of position within the element with the consequence that the strain, and hence the stress, will be constant throughout the element.

For plane stress analysis ($\sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0$) with isotropic material behaviour, the stress/strain relations in matrix form are

$$\{\sigma\} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{E}{(1 - v^2)} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \dfrac{1 - v}{2} \end{bmatrix} \{\varepsilon\}$$

Or, more concisely, $\{\sigma\} = [D]\{\varepsilon\}$ (9.92)

where σ_{xx} and σ_{yy} are the direct stresses parallel to x and y axes, respectively, σ_{xy} is the shear stress in the xy plane, and $[D]$ is known as the *elasticity matrix*.

Following the same arguments used in the rod and beam formulations, namely, taking the expression for virtual strain to have a similar form to the real strain, eqn. (9.90), and substituting this and the expression for real stress, eqn. (9.92), into the equation of the principle of virtual work, (9.10), gives the element stiffness matrix as

$$[k^{(e)}] = \int_v [B]^{\mathrm{T}}[D][B]\, dv \tag{9.93}$$

The only departure of eqn. (9.93) from the previous expressions is the replacement of the modulus of elasticity, E, by the elasticity matrix $[D]$, due to the change from a one- to a two-dimensional stress system.

Recalling, for the present case that the displacement fields are linearly varying, then matrix $[B]$ is independent of the x and y coordinates. The assumption of isotropic homogeneous material means that matrix $[D]$ is also independent of coordinates. It follows, assuming a constant thickness, t, throughout the element, of area, a, eqn. (9.93) can be integrated to give

$$[k^{(e)}] = at\ [B]^{\mathrm{T}}[D][B] \tag{9.94}$$

Element stress matrix

The expression for the element direct and shear stresses is obtained by substituting from eqn. (9.90) into eqn. (9.92), to give

$$\{\sigma^{(e)}\} = [D][B]\{u\}$$

or, more fully,

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = [D][B] \begin{bmatrix} u_i \\ u_j \\ u_k \\ v_i \\ v_j \\ v_k \end{bmatrix} \tag{9.95}$$

These stresses are with respect to the global coordinate axes and are taken to act at the element centroid.

Formation of structural governing equation and assembled stiffness matrix

As Sections 9.7.2 and 9.8.2, the structural governing equation is given by eqn. (9.28) and the assembled stiffness matrix by eqn. (9.47).

9.10. Formation of assembled stiffness matrix by use of a dof. correspondence table

Element stiffness matrices given, for example, by eqn. (9.23), are formed for each element in the structure being analysed, and are combined to form the assembled stiffness matrix $[K]$. Where nodes are common to more than one element, the assembly process requires that appropriate stiffness contributions from all such elements are summed for each node. Execution of finite element programs will enable assembly of the element stiffness contributions by utilising, for example, eqn. (9.29) deriving matrix $[a]$, and hence $[a]^{\mathrm{T}}$, from the connectivity information provided by the element mesh. Alternatively, eqn. (9.47) can be used, the matrix summation requiring that all element stiffness matrices, $[k^{(e)}]$, are of the same order as the assembled stiffness matrix $[K]$. However, by efficient "housekeeping" only those rows and columns containing the non-zero terms need be stored.

For the purpose of performing hand calculations, the tedium of evaluating the triple matrix product of eqn. (9.29) can be avoided by summing the element stiffness contributions according to eqn. (9.47). The procedure to be adopted follows, and uses a so-called *dof. correspondence table*. Consider assembly of the element stiffness contributions for the simple pin-jointed plane frame idealised as three rod elements, shown in Fig. 9.19. The element stiffness matrices in global coordinates can be illustrated as:

$$[k^{(a)}] = \begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ a_{21} & a_{22} & a_{23} & a_{24} \\ a_{31} & a_{32} & a_{33} & a_{34} \\ a_{41} & a_{42} & a_{43} & a_{44} \end{bmatrix}$$

$$[k^{(b)}] = \begin{bmatrix} b_{11} & b_{12} & b_{13} & b_{14} \\ b_{21} & b_{22} & b_{23} & b_{24} \\ b_{31} & b_{32} & b_{33} & b_{34} \\ b_{41} & b_{42} & b_{43} & b_{44} \end{bmatrix}$$

$$[k^{(c)}] = \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} \\ c_{21} & c_{22} & c_{23} & c_{24} \\ c_{31} & c_{32} & c_{33} & c_{34} \\ c_{41} & c_{42} & c_{43} & c_{44} \end{bmatrix}$$

The procedure is as follows:

- Label a diagram of the frame with dof. numbers in node number sequence.
- Construct a dof. correspondence table, entering a set of dof. numbers for each node of every element. For the rod element there will be two dof. in each set, namely, u and v displacements, and two sets per element, one for each node. The sequence of the sets must correspond to progression along the local axis direction, i.e. along each positive x' direction. This is essential to maintain consistency with the element matrices, above, the terms of which have been shown in eqn. (9.23) to involve angle α, the value of which will correspond to the inclination of the element at the end chosen as the origin of its local axis. The sequence shown in Table 9.2 corresponds to $\alpha_a = 330°, \alpha_b = 180°$ and $\alpha_c = 210°$. The u and v dof. sequence within each set must be maintained.
- Choose an element for which the stiffness contributions are to be assembled.
- Assemble by either rows or columns according to the dof. correspondence table.
- Repeat for the remaining elements until all are assembled.

Table 9.2. Dof. correspondence table for assembly of structural stiffness matrix, $[K]$.

Row and/or column in element stiffness matrix, $[k^{(e)}]$	Row and/or column in assembled stiffness matrix, [K]		
	element a	element b	element c
1	1	3	1
2	2	4	2
3	3	5	5
4	4	6	6

For example, choosing to assemble element b contributions by rows, then the first and the "element b" columns of the dof. table, Table 9.2, are used. Start by inserting in row 3, columns 3, 4, 5, 6 of structural stiffness matrix $[K]$, the stiffness contributions respectively from row 1, columns 1, 2, 3, 4 of element stiffness matrix, $[k^{(b)}]$. Repeat for the remaining rows 4, 5, 6, inserting in columns 3, 4, 5, 6 of $[K]$, the respective contributions from rows

2, 3, 4, columns 1, 2, 3, 4 of $[k^{(b)}]$. Repeat for remaining elements a and c, to give finally:

$$[K] = \begin{bmatrix} a_{11}+c_{11} & a_{12}+c_{12} & a_{13} & a_{14} & c_{13} & c_{14} \\ a_{21}+c_{21} & a_{22}+c_{22} & a_{23} & a_{24} & c_{23} & c_{24} \\ a_{31} & a_{32} & a_{33}+b_{11} & a_{34}+b_{12} & b_{13} & b_{14} \\ a_{41} & a_{42} & a_{43}+b_{21} & a_{44}+b_{22} & b_{23} & b_{24} \\ c_{31} & c_{32} & b_{31} & b_{32} & b_{33}+c_{33} & b_{34}+c_{34} \\ c_{41} & c_{42} & b_{41} & b_{42} & b_{43}+c_{43} & b_{44}+c_{44} \end{bmatrix}$$

The above assembly procedure is generally applicable to any element, albeit with detail changes. In the case of the simple beam element, with its rotational, as well as translational dof., reference to § 9.8 shows that the element stiffness matrix is of order 6 × 6, and hence there will be two additional rows in the dof. correspondence table. A similar argument holds for the triangular membrane element, with its three nodes each having 2 dof. The Examples at the end of this chapter illustrate the assembly for rod, beam and membrane elements.

9.11. Application of boundary conditions and partitioning

With reference to §9.4.7, before the governing eqn. (9.28) can be solved to yield the unknown displacements, appropriate restraints need to be imposed. At some nodes the displacements will be prescribed, for example, at a fixed node the nodal displacements will be zero. Hence, some of the nodal displacements will be unknown, $\{p_\alpha\}$, and some will be prescribed, $\{p_\beta\}$. Following any necessary rearrangement to collect together equations relating to unknown, and those relating to prescribed, displacements, eqn. (9.28) can be partitioned into

$$\begin{bmatrix} \{P_\alpha\} \\ \{P_\beta\} \end{bmatrix} = \begin{bmatrix} [K_{\alpha\alpha}] & [K_{\alpha\beta}] \\ [K_{\beta\alpha}] & [K_{\beta\beta}] \end{bmatrix} \begin{bmatrix} \{p_\alpha\} \\ \{p_\beta\} \end{bmatrix} \qquad (9.96)$$

It will be found that where the loads are known, $\{P_\alpha\}$, [i.e. prescribed nodal forces (and moments, in beam applications)], the corresponding displacements will be unknown, $\{p_\alpha\}$, and where the displacements are known, $\{p_\beta\}$, (i.e. prescribed nodal displacements), the forces, $\{P_\beta\}$, (and moments, in beam applications), usually the reactions, will be unknown.

9.12. Solution for displacements and reactions

A solution for the unknown nodal displacements, $\{p_\alpha\}$, is obtained from the upper partition of eqn. (9.96)

$$\{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\} + [K_{\alpha\beta}]\{p_\beta\}$$

Rearranging

$$[K_{\alpha\alpha}]\{p_\alpha\} = \{P_\alpha\} - [K_{\alpha\beta}]\{p_\beta\}$$

To obtain a solution for the unknown nodal displacements, $\{p_\alpha\}$, it is only necessary to invert the submatrix $[K_{\alpha\alpha}]$. Pre-multiplying the above equation by $[K_{\alpha\alpha}]^{-1}$ (and using the matrix relation, $[K_{\alpha\alpha}]^{-1}[K_{\alpha\alpha}] = [I]$, the unit matrix), will yield the values of the unknown nodal displacements as

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\} - [K_{\alpha\alpha}]^{-1}[K_{\alpha\beta}]\{p_\beta\} \qquad (9.97)$$

If all the prescribed displacements are zero, i.e. $\{p_\beta\} = \{0\}$, the above reduces to

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\} \tag{9.98}$$

The unknown reactions, $\{P_\beta\}$, can be found from the lower partition of eqn. (9.96)

$$\{P_\beta\} = [K_{\beta\alpha}]\{p_\alpha\} + [K_{\beta\beta}]\{p_\beta\} \tag{9.99}$$

Again, if all the prescribed displacements are zero, the above reduces to

$$\{P_\beta\} = [K_{\beta\alpha}]\{p_\alpha\} \tag{9.100}$$

Bibliography

1. R.T. Fenner, *Finite Element Methods for Engineers,* Macmillan Press, London, 1996.
2. E. Hinton and D.R.J. Owen, *An Introduction to Finite Element Computations,* Pineridge Press, Swansea, 1979.
3. R.K. Liversley, *Finite Elements; An Introduction for Engineers,* Cambridge University Press, 1983.
4. NAFEMS, *Guidelines to Finite Element Practice,* DTI, NEL, Glasgow, 1992.
5. NAFEMS, *A Finite Element Primer,* DTI, NEL, Glasgow, 1992.
6. S.S. Rao, *The Finite Element Method in Engineering,* Pergamon Press, Oxford, 1989.
7. J.N. Reddy, *An Introduction to the Finite Element Method,* McGraw-Hill, 1993.
8. K.C. Rockey, *et al., The Finite Element Method; A Basic Introduction,* Crosby Lockwood Staples, London, 1975.
9. R.L. Sack, *Matrix Structural Analysis*, PWS-Kent, 1989.
10. O.C. Zienkiewicz, *The Finite Element Method,* McGraw-Hill, London, 1988.
11. C.A. Brebbia and A.J. Ferrante, *Computational Methods for the Solution of Engineering Problems,* Pentech Press, London, 1986.
12. HKS ABAQUS, User's Manual, 1995.
13. SDRC I-DEAS Finite Element Modelling, User's Guide, 1991.
14. K. Thomas, Effects of geometric distortion on the accuracy of plane quadratic isoparametric finite elements, Guidelines for finite element idealisation, Meeting Preprint 2504, ASCE, pp. 161–204, 1975.

Examples

Example 9.1

Figure 9.31 shows a planar steel support structure, all three members of which have the same axial stiffness, such that $AE/L = 20$ MN/m throughout. Using the displacement based finite element method and treating each member as a rod:

(a) assemble the necessary terms in the structural stiffness matrix;

(b) hence, determine, with respect to the global coordinates (i) the nodal displacements, and (ii) the reactions, showing the latter on a sketch of the structure and demonstrating that equilibrium is satisfied.

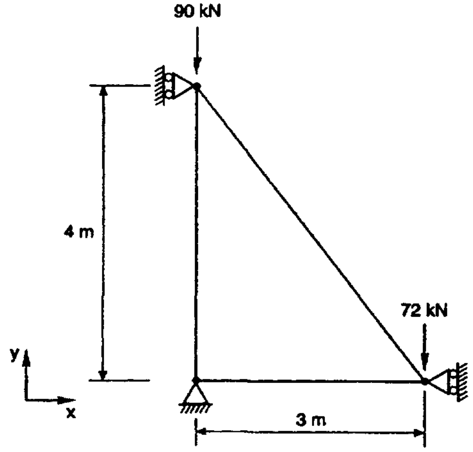

Fig. 9.31.

Solution

(a) Figure 9.32 shows suitable node, dof. and element labelling. Lack of symmetry prevents any advantage being taken to reduce the calculations. None of the members are redundant and hence the stiffness contributions of all three members need to be included.

All three elements will have the same stiffness matrix scalar,

i.e. $$(AE/L)^{(a)} = (AE/L)^{(b)} = (AE/L)^{(c)} = AE/L$$

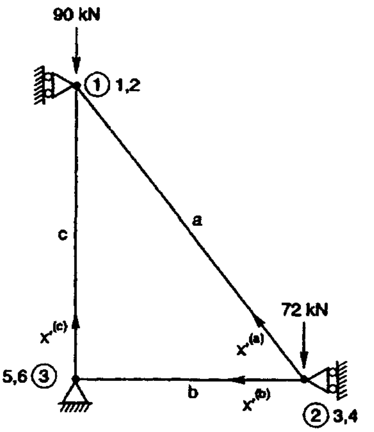

Fig. 9.32.

With reference to §9.7, the element stiffness matrix with respect to global coordinates is given by

$$[k^{(e)}] = \left(\frac{AE}{L}\right)^{(e)} \begin{bmatrix} \cos^2\alpha & & & \\ \sin\alpha\cos\alpha & \sin^2\alpha & \text{symmetric} & \\ -\cos^2\alpha & -\sin\alpha\cos\alpha & \cos^2\alpha & \\ -\sin\alpha\cos\alpha & -\sin^2\alpha & \sin\alpha\cos\alpha & \sin^2\alpha \end{bmatrix}^{(e)}$$

Evaluating the stiffness matrix for each element:

Element a

$$\alpha^{(a)} = -\tan^{-1}(4/3), \quad \cos\alpha^{(a)} = -0.6, \quad \sin\alpha^{(a)} = 0.8$$

$$[k^{(a)}] = \frac{AE}{L}\begin{bmatrix} 0.36 & -0.48 & -0.36 & 0.48 \\ -0.48 & 0.64 & 0.48 & -0.64 \\ -0.36 & 0.48 & 0.36 & -0.48 \\ 0.48 & -0.64 & -0.48 & 0.64 \end{bmatrix}$$

Element b

$$\alpha^{(b)} = 180^\circ, \quad \cos\alpha^{(b)} = -1, \quad \sin\alpha^{(b)} = 0$$

$$[k^{(b)}] = \frac{AE}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Element c

$$\alpha^{(c)} = 90, \quad \cos\alpha^{(c)} = 0, \quad \sin\alpha^{(c)} = 1$$

$$[k^{(c)}] = \frac{AE}{L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

The structural stiffness matrix can now be assembled using a dof. correspondence table, (ref. §9.10). Observation of the highest dof. number, i.e. 6, gives the order (size), of the structural stiffness matrix, i.e. 6 × 6. The structural governing equations and hence the required structural stiffness matrix are therefore given as

Row/column in $[k^{(e)}]$	Row/column in $[K]$		
	a	*b*	*c*
1	3	3	5
2	4	4	6
3	1	5	1
4	2	6	2

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = \frac{AE}{L}\left[\begin{array}{c|c|c|c|c|c} 0.36 & -0.48 & -0.36 & 0.48 & & \\ 0 & 0 & & & 0 & 0 \\ \hline -0.48 & 0.64 & 0.48 & -0.64 & & \\ 0 & 1 & & & 0 & -1 \\ \hline -0.36 & 0.48 & 0.36 & -0.48 & & \\ & & 1 & 0 & -1 & 0 \\ \hline 0.48 & -0.64 & -0.48 & 0.64 & & \\ & & 0 & 0 & 0 & 0 \\ \hline & & -1 & 0 & 1 & 0 \\ 0 & 0 & & & 0 & 0 \\ \hline & & 0 & 0 & 0 & 0 \\ 0 & -1 & & & 0 & 1 \end{array}\right]\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}$$

(b) (i) Rearranging and partitioning, with $u_1 = u_2 = u_3 = v_3 = 0$, (i.e. $\{p_\beta\} = 0$)

$$\begin{bmatrix} Y_1 \\ Y_2 \end{bmatrix} = \begin{bmatrix} -90 \times 10^3 \\ -72 \times 10^3 \end{bmatrix} = \frac{AE}{L}\begin{bmatrix} 1.64 & -0.64 \\ -0.64 & 0.64 \end{bmatrix}\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} \; i.e. \{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\}$$

Inverting $[K_{\alpha\alpha}]$ to enable a solution for the displacements using $\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$

$$\text{adj } [K_{\alpha\alpha}] = \frac{AE}{L}\begin{bmatrix} 0.64 & 0.64 \\ 0.64 & 1.64 \end{bmatrix} \quad \text{and} \quad \det[K_{\alpha\alpha}] = 0.64(AE/L)^2$$

$$\text{Then } [K_{\alpha\alpha}]^{-1} = \frac{L}{AE}\begin{bmatrix} 1 & 1 \\ 1 & 2.5625 \end{bmatrix} \; \text{Check: } \frac{L}{AE}\begin{bmatrix} 1 & 1 \\ 1 & 2.5625 \end{bmatrix}\frac{AE}{L}\begin{bmatrix} 1.64 & -0.64 \\ -0.64 & 0.64 \end{bmatrix} = [I]$$

The required displacements are found from

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$$

$$\text{Substituting, } \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{L}{AE}\begin{bmatrix} 1 & 1 \\ 1 & 2.5625 \end{bmatrix}\begin{bmatrix} Y_1 \\ Y_2 \end{bmatrix} = -5 \times 10^{-8}\begin{bmatrix} 1 & 1 \\ 1 & 2.5625 \end{bmatrix}\begin{bmatrix} 90.10^3 \\ 72.10^3 \end{bmatrix}$$

$$= \begin{bmatrix} -8.10 \\ -13.73 \end{bmatrix}_{\text{mm}}$$

The required nodal displacements are therefore $v_1 = -8.10$ mm and $v_2 = -13.73$ mm.

(ii) With reference to §9.12, nodal reactions are obtained from

$$\{P_\beta\} = [K_{\beta\alpha}]\{p_\alpha\}$$

Substituting gives

$$\begin{bmatrix} X_1 \\ X_2 \\ X_3 \\ Y_3 \end{bmatrix} = \frac{AE}{L}\begin{bmatrix} -0.48 & 0.48 \\ 0.48 & -0.48 \\ 0 & 0 \\ -1 & 0 \end{bmatrix}\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = 2 \times 10^7\begin{bmatrix} -0.48 & 0.48 \\ 0.48 & -0.48 \\ 0 & 0 \\ -1 & 0 \end{bmatrix}\begin{bmatrix} -8.10 \times 10^{-3} \\ -13.725 \times 10^{-3} \end{bmatrix}$$

$$= \begin{bmatrix} -54 \\ 54 \\ 0 \\ 162 \end{bmatrix}_{\text{kN}}$$

The required nodal reactions are therefore $X_1 = -54$ kN, $X_2 = 54$ kN, $X_3 = 0$ and $Y_3 = 162$ kN.

Representing these reactions together with the applied forces on a sketch of the structure, Fig. 9.33, and considering force and moment equilibrium, gives

$$\sum F_x = (54 - 54) \text{ kN} \qquad = 0$$

$$\sum F_y = (162 - 90 - 72) \text{ kN} \qquad = 0$$

$$\sum M_3 = (54 \times 4 - 73 \times 3) \text{ kNm} = 0$$

Hence, equilibrium is satisfied by the system of forces.

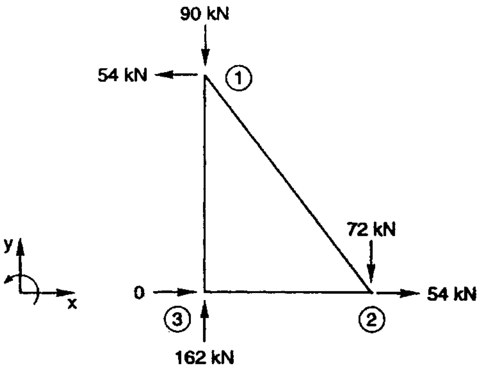

Fig. 9.33.

Example 9.2

Figure 9.34 shows the members and idealised support conditions for a roof truss. All three members of which are steel and have the same cross-sectional area such that $AE = 12$ MN throughout. Using the displacement based finite element method, treating the truss as a pin-jointed plane frame and each member as a rod:

(a) assemble the necessary terms in the structural stiffness matrix;

(b) hence, determine the nodal displacements with respect to the global coordinates, for the condition shown in Fig. 9.34.

(c) If, under load, the left support sinks by 5 mm, determine the resulting new nodal displacements, with respect to the global coordinates.

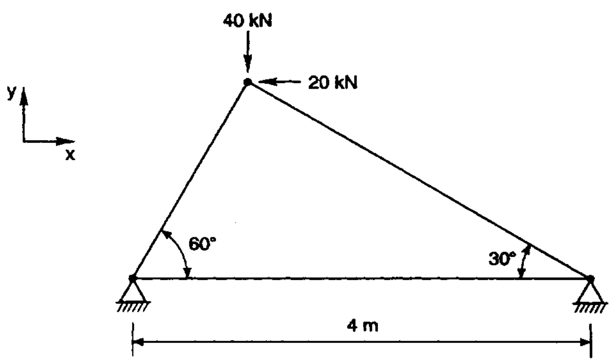

Fig. 9.34.

Solution

(a) Figure 9.35 shows suitable node, dof. and element labelling. Lack of symmetry prevents any advantage being taken to reduce the calculations. However, since both ends of the horizontal member are fixed it is redundant therefore and does not need to be considered further.

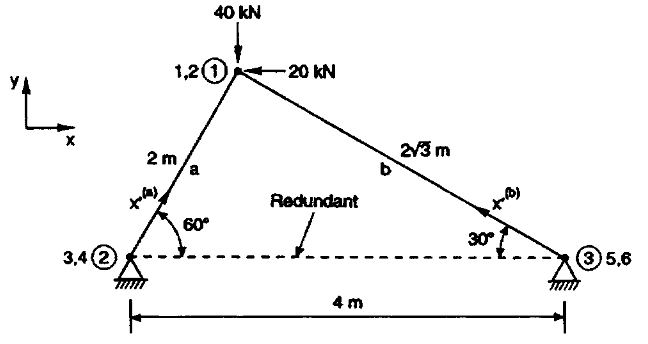

Fig. 9.35.

All three members have the same AE, hence

$$(AE)^{(a)} = (AE)^{(b)} = AE$$

With reference to §9.7, the element stiffness matrix with respect to global coordinates is given by

$$[k^{(e)}] = \left(\frac{AE}{L}\right)^{(e)} \begin{bmatrix} \cos^2\alpha & & \text{symmetric} & \\ \sin\alpha\cos\alpha & \sin^2\alpha & & \\ -\cos^2\alpha & -\sin\alpha\cos\alpha & \cos^2\alpha & \\ -\sin\alpha\cos\alpha & -\sin^2\alpha & \sin\alpha\cos\alpha & \sin^2\alpha \end{bmatrix}^{(e)}$$

Evaluating the stiffness matrix for both elements:

Element a

$$L^{(a)} = 2\text{m}, \quad \alpha^{(a)} = 60°, \quad \cos\alpha^{(a)} = 1/2, \quad \sin\alpha^{(a)} = \sqrt{(3)}/2$$

$$[k^{(a)}] = \frac{AE}{8} \begin{bmatrix} 1 & \sqrt{3} & -1 & -\sqrt{3} \\ \sqrt{3} & 3 & -\sqrt{3} & -3 \\ -1 & -\sqrt{3} & 1 & \sqrt{3} \\ -\sqrt{3} & -3 & \sqrt{3} & 3 \end{bmatrix}$$

Element b

$$L^{(b)} = 2\sqrt{3}m, \quad \alpha^{(b)} = 150°, \quad \cos\alpha^{(b)} = -\sqrt{(3)}/2, \quad \sin\alpha^{(a)} = 1/2$$

$$[k^{(b)}] = \frac{AE}{8} \begin{bmatrix} \sqrt{3} & -1 & -\sqrt{3} & 1 \\ -1 & 1/\sqrt{3} & 1 & -1/\sqrt{3} \\ -\sqrt{3} & 1 & \sqrt{3} & -1 \\ 1 & -1/\sqrt{3} & -1 & 1/\sqrt{3} \end{bmatrix}$$

The structural stiffness matrix can now be multi-assembled using a dof. correspondence table, (ref. §9.10), and will be of order 6 × 6. Only the upper sub-matrices need to be completed, i.e. $[K_{\alpha\alpha}]$ and $[K_{\alpha\beta}]$, since the reactions are not required in this example. The necessary structural governing equations and hence the required structural stiffness matrix are therefore given as

Row/ column in $[k^{(e)}]$	Row/column in $[K]$	
	a	b
1	3	5
2	4	6
3	1	1
4	2	2

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = \frac{AE}{8} \left[\begin{array}{cc:cc:cc:cc:cc:cc} 1 & & \sqrt{3} & & -1 & & -\sqrt{3} & & & & & \\ & \sqrt{3} & & -1 & & & & & & -\sqrt{3} & & 1 \\ \sqrt{3} & & 3 & & -\sqrt{3} & & -3 & & & & & \\ & -1 & & 1/\sqrt{3} & & & & & & 1 & & -1/\sqrt{3} \\ \hdashline & & & & & & & & & & & \\ & & & & & & & & & & & \\ & & & & & & & & & & & \\ & & & & & & & & & & & \end{array}\right] \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}$$

(b) Corresponding to $u_2 = v_2 = u_3 = v_3 = 0$, the partitioned equations reduce to:

$$\begin{bmatrix} X_1 \\ Y_1 \end{bmatrix} = \begin{bmatrix} -20 \times 10^3 \\ -40 \times 10^3 \end{bmatrix} = \frac{AE}{8} \begin{bmatrix} 2.7321 & 0.7321 \\ 0.7321 & 3.5774 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} \quad \text{i.e. } \{P_\alpha\} = [k_{\alpha\alpha}]\{p_\alpha\}$$

Inverting $[K_{\alpha\alpha}]$ to enable a solution for the displacements from $\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$

$$\text{adj } [K_{\alpha\alpha}] = \left(\frac{AE}{8}\right)^2 \begin{bmatrix} 3.5774 & -0.7321 \\ -0.7321 & 2.7321 \end{bmatrix} \text{ and det } [K_{\alpha\alpha}] = \left(\frac{AE}{8}\right)^2 9.2378$$

Then $[K_{\alpha\alpha}]^{-1}$

$$= \frac{8}{AE} \begin{bmatrix} 0.3873 & -0.07925 \\ -0.07925 & 0.2958 \end{bmatrix} \text{Check:} \frac{8}{AE} \begin{bmatrix} 0.3873 & -0.07925 \\ -0.07925 & 0.2958 \end{bmatrix} \frac{8}{AE} \begin{bmatrix} 2.7321 & 0.7321 \\ 0.7321 & 3.5774 \end{bmatrix} = [I]$$

Hence, the required displacements are given by

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$$

$$\text{Substituting, } \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} = \frac{8}{12 \times 10^6} \begin{bmatrix} 0.3873 & -0.07925 \\ -0.07925 & 0.2958 \end{bmatrix} \begin{bmatrix} -20.10^3 \\ -40.10^3 \end{bmatrix} = \begin{bmatrix} -3.05 \\ -6.83 \end{bmatrix}_{\text{mm}}$$

The required nodal displacements are therefore $u_1 = -3.05$ mm and $v_1 = -6.83$ mm.

(c) With reference to §9.12, for non-zero prescribed displacements, i.e. $\{p_\beta\} \neq \{0\}$, the full partition of the governing equation is required, namely,

$$\{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\} + [K_{\alpha\beta}]\{p_\beta\}$$

Rearranging for the unknown displacements

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\} - [K_{\alpha\alpha}]^{-1}[K_{\alpha\beta}]\{p_\beta\}$$

Evaluating

$$[K_{\alpha\alpha}]^{-1}[K_{\alpha\beta}] = \begin{bmatrix} 0.3873 & -0.07925 \\ -0.07925 & 0.2958 \end{bmatrix} \begin{bmatrix} -1 & -1.7321 & -1.7321 & 1 \\ -1.7321 & -3 & 1 & -0.5773 \end{bmatrix}$$

$$= \begin{bmatrix} -0.25 & -0.4331 & -0.75 & 0.4331 \\ -0.4331 & -0.75 & 0.4331 & -0.25 \end{bmatrix}$$

$$\text{and } [K_{\alpha\alpha}]^{-1}[K_{\alpha\beta}]\{p_\beta\} = \begin{bmatrix} -0.25 & -0.4331 & -0.75 & 0.4331 \\ -0.4331 & -0.75 & 0.4331 & -0.25 \end{bmatrix} \begin{bmatrix} 0 \\ -5 \times 10^{-3} \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 2.1655 \\ 3.75 \end{bmatrix}_{\text{mm}}$$

Recalling from part (b) that

$$[K_{\alpha\alpha}]^{-1}\{P_\alpha\} = \begin{bmatrix} -3.05 \\ -6.83 \end{bmatrix}_{\text{mm}}$$

and substituting into the above rearranged governing equation,

$$\text{i.e. } \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} = \begin{bmatrix} -3.05 \\ -6.83 \end{bmatrix} - \begin{bmatrix} 2.1655 \\ 3.75 \end{bmatrix} = \begin{bmatrix} -5.22 \\ -10.58 \end{bmatrix}_{\text{mm}}$$

yields the required new nodal displacements, namely, $u_1 = -5.22$ mm and $v_1 = -10.58$ mm.

Example 9.3

A steel beam is supported and loaded as shown in Fig. 9.36. The relevant second moments of area are such that $I^{(a)} = 2I^{(b)} = 2 \times 10^{-5}$ m^4 and Young's modulus E for the beam material = 200 GN/m^2. Using the displacement based finite element method and representing each member by a simple beam element:

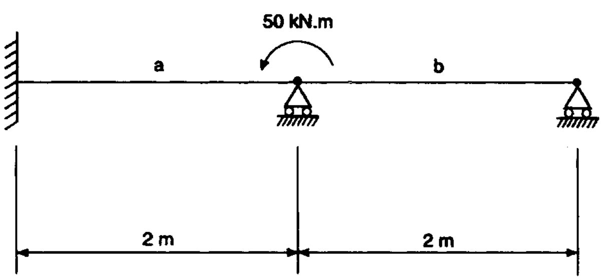

Fig. 9.36.

(a) determine the nodal displacements;
(b) hence, determine the nodal reactions, representing these on a sketch of the deformed geometry. Show that both force and moment equilibrium is satisfied.

Solution

(a) Figure 9.37 shows suitable node, dof. and element labelling. Lack of symmetry prevents any advantage being taken to reduce the calculations. There are no redundant members.

Employing two beam finite elements, (which is the least number in this case), both elements will have the same E/L, i.e.

$$(E/L)^{(a)} = (E/L)^{(b)} = E/L$$

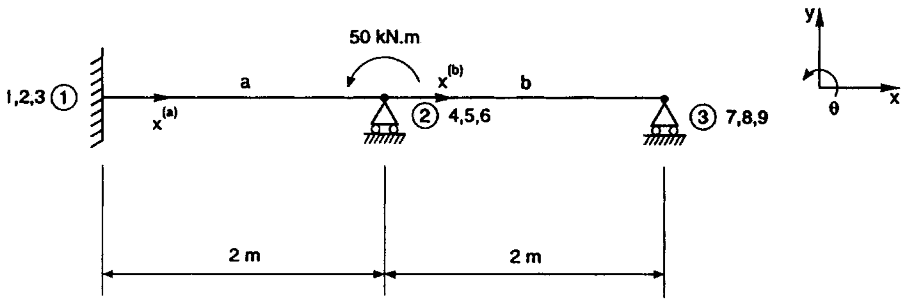

Fig. 9.37.

However, the second moments of area will be different, such that

$$I^{(a)} = 2I \text{ and } I^{(b)} = I$$

and will be the only difference between the two element stiffness matrices.

With reference to §9.8 and in the absence of axial forces, each element stiffness matrix with respect to local coordinates is given as

$$[k^{(e)}] = \left(\frac{EI}{L}\right)^{(e)} \begin{bmatrix} 12/L^2 & & & \\ 6/L & 4 & \text{symmetric} & \\ -12/L^2 & -6/L & 12/L^2 & \\ 6/L & 2 & -6/L & 4 \end{bmatrix}$$

The above local coordinate element stiffness matrix will, in this case, be identical to that with respect to global coordinates since the local and global axes coincide.

Substituting for both elements:

Element a

Recalling $I^{(a)} = 2I$

$$[k^{(a)}] = \left(\frac{EI}{L}\right) \begin{bmatrix} 24/L^2 & & & \\ 12/L & 8 & \text{symmetric} & \\ -24/L^2 & -12/L & 24/L^2 & \\ 12/L & 4 & -12/L & 8 \end{bmatrix}$$

Element b

Recalling $I^{(b)} = I$

$$[k^{(b)}] = \frac{EI}{L}\begin{bmatrix} 12/L^2 & & & \\ 6/L & 4 & \text{symmetric} & \\ -12/L^2 & -6/L & 12/L^2 & \\ 6/L & 2 & -6/L & 4 \end{bmatrix}$$

The structural stiffness matrix can now be assembled. A dof. correspondence table can be used as an aid to assembly. However, observation of the relatively simple element connectivity, shows that the stiffness contributions for element a will occupy the upper left 4×4 locations, whilst those for element b will occupy the lower right 4×4 locations of the 6×6 structural stiffness matrix. The reduced structural stiffness matrix is due to the omission of axial terms, otherwise the matrix would have been of order 9×9. Hence, completing only those columns needed for the solution, gives

$$\begin{bmatrix} Y_1 \\ M_1 \\ Y_2 \\ M_2 \\ Y_3 \\ M_3 \end{bmatrix} = \frac{EI}{L}\left[\begin{array}{c|c|c|c|c|c} & & & 12/L & & \\ \hline & & & 4 & & \\ \hline & & & -12/L \quad 6/L & & 6/L \\ \hline & & & 8 \quad 4 & & 2 \\ \hline & & & -6/L & & -6/L \\ \hline & & & 2 & & 4 \end{array}\right]\begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix}$$

No need to complete these columns

Corresponding to $v_1 = \theta_1 = v_2 = v_3 = 0$ (by omitting axial terms it has already been taken that $u_1 = u_2 = u_3 = 0$), the partitioned equations reduce to

$$\begin{bmatrix} M_2 \\ M_3 \end{bmatrix} = \frac{EI}{L}\begin{bmatrix} 12 & 2 \\ 2 & 4 \end{bmatrix}\begin{bmatrix} \theta_2 \\ \theta_3 \end{bmatrix} \quad \text{i.e. } \{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\}$$

Inverting $[K_{\alpha\alpha}]$ to enable a solution for the displacements from $\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$

$$\text{where adj } [K_{\alpha\alpha}] = \frac{EI}{L}\begin{bmatrix} 4 & -2 \\ -2 & 12 \end{bmatrix} \text{ and det } [K_{\alpha\alpha}] = 44(EI/L)^2$$

$$\text{Then, } [K_{\alpha\alpha}]^{-1} = \frac{L}{44EI}\begin{bmatrix} 4 & -2 \\ -2 & 12 \end{bmatrix} \text{ Check } \frac{L}{44EI}\begin{bmatrix} 4 & -2 \\ -2 & 12 \end{bmatrix}\frac{EI}{L}\begin{bmatrix} 12 & 2 \\ 2 & 4 \end{bmatrix} = [I]$$

The required displacements are found from

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$$

Substituting $\begin{bmatrix}\theta_2\\ \theta_3\end{bmatrix} = \dfrac{L}{44EI}\begin{bmatrix}4 & -2\\ -2 & 12\end{bmatrix}\begin{bmatrix}M_2\\ M_3\end{bmatrix}$

$$= \frac{2}{44\times 200\times 10^9\times 1\times 10^{-5}}\begin{bmatrix}4 & -2\\ -2 & 12\end{bmatrix}\begin{bmatrix}5\times 10^4\\ 0\end{bmatrix}$$

$$= 2.2727\times 10^{-4}\begin{bmatrix}20\\ -10\end{bmatrix} = \begin{bmatrix}4.545.10^{-3}\\ -2.273.10^{-3}\end{bmatrix}_{\text{rad}} = \begin{bmatrix}0.260\\ -0.130\end{bmatrix}_{\text{deg}}$$

The required nodal displacements are therefore $\theta_2 = 0.26^\circ$ and $\theta_3 = -0.13^\circ$.

(b) With reference to §9.12, nodal reactions are obtained from

$$\{P_\alpha\} = [K_{\alpha\beta}]\{p_\alpha\}$$

Substituting gives $\begin{bmatrix}Y_1\\ M_1\\ Y_2\\ Y_3\end{bmatrix} = \dfrac{EI}{L}\begin{bmatrix}12/L & 0\\ 4 & 0\\ -6/L & 6/L\\ -6/L & -6/L\end{bmatrix}\begin{bmatrix}\theta_2\\ \theta_3\end{bmatrix}$

$$= \frac{200\times 10^9\times 1\times 10^{-5}}{2}\begin{bmatrix}6 & 0\\ 4 & 0\\ -3 & 3\\ -3 & -3\end{bmatrix}\begin{bmatrix}4.545\times 10^{-3}\\ -2.273\times 10^{-3}\end{bmatrix}$$

$$= \begin{bmatrix}27.27\text{ kN}\\ 18.18\text{ kNm}\\ -20.45\text{ kN}\\ -6.82\text{ kN}\end{bmatrix}$$

The required nodal reactions are therefore $Y_1 = 27.27$ kN, $M_1 = 18.18$ kNm, $Y_2 = -20.45$ kN and $Y_3 = -6.82$ kN.

Representing these reactions together with the applied moment on a sketch of the deformed beam, Fig. 9.38, and considering force and moment equilibrium, gives

$$\Sigma F_y = (27.27 - 20.45 - 6.82)\text{ kN} = 0$$

$$\Sigma M_1 = (18.18 + 50 - 20.45\times 2 - 6.82\times 4)\text{ kNm} = 0$$

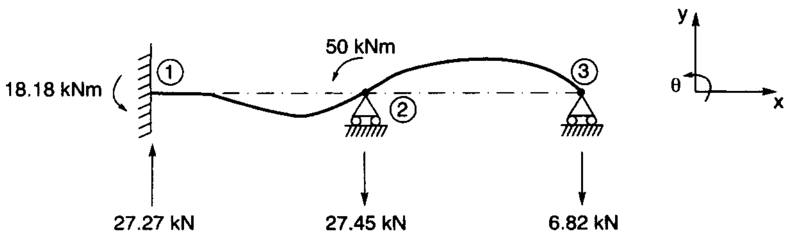

Fig. 9.38.

Example 9.4

The vehicle engine mounting bracket shown in Fig. 9.39 is made from uniform steel channel section for which Young's modulus, $E = 200$ GN/m^2. It can be assumed for both

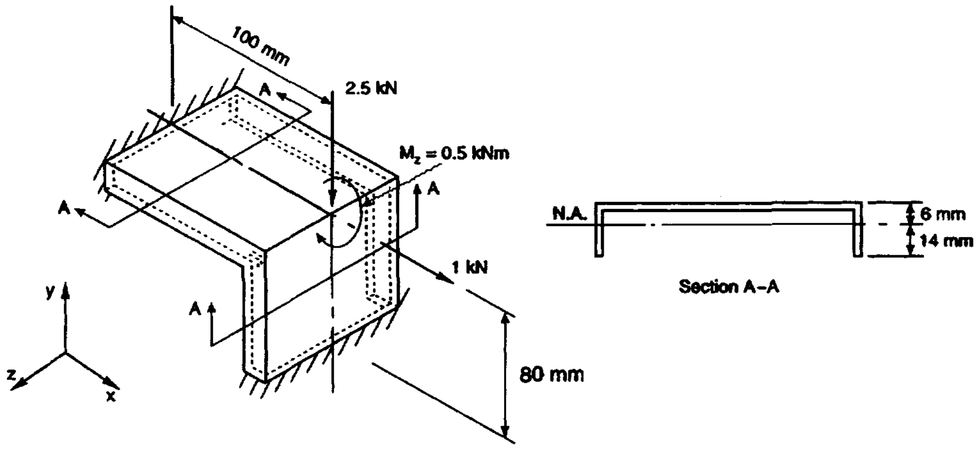

Fig. 9.39.

channels that the relevant second moment of area, $I = 2 \times 10^{-8}$ m^4 and cross-sectional area, $A = 4 \times 10^{-4}$ m^2. The bracket can be idealised as two beams, the common junction of which can be assumed to be infinitely stiff and the other ends to be fully restrained. Using the displacement based finite element method, and representing the constituent members as simple beam elements:

(a) assemble the necessary terms in the structural stiffness matrix;

(b) hence, determine for the condition shown in Fig. 9.39 (i) the nodal displacements with respect to the global coordinates, and (ii) the combined axial and bending extreme fibre stresses at the built-in ends and at the common junction.

Solution

(a) Figure 9.40 shows suitable node, dof. and element labelling. The structure does not have symmetry or redundant members. The least number of beam elements will be used to

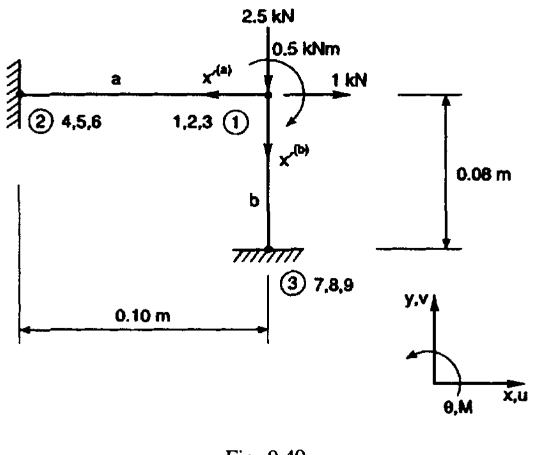

Fig. 9.40.

minimise the hand calculations which, in this example, is two.
Both elements will have the same A, E and I,

i.e. $$(A, E, I)^{(a)} = (A, E, I)^{(b)} = A, E, I,$$

but will have different lengths, i.e. $L^{(a)}$ and $L^{(b)}$.

With reference to §9.8, the element stiffness matrix inclusive of axial terms and in global coordinates is appropriate, namely:

$$[k^{(e)}] = \left(\frac{E}{L}\right)^{(e)} \begin{bmatrix} A\cos^2\alpha + (12I\sin^2\alpha)/L^2, & & & & & \\ (A - 12I/L^2)\cos\alpha\sin\alpha, & A\sin^2\alpha + (12I\cos^2\alpha)/L^2, & & & & \\ -(6I\sin\alpha)L, & (6I\cos\alpha)/L, & 4I, & & & \\ -A\cos^2\alpha - (12I\sin^2\alpha)/L^2, & -(A - 12I/L^2)\cos\alpha\sin\alpha, & (6I\sin\alpha)/L, & A\cos^2\alpha + (12I\sin^2\alpha)/L^2, & \text{symmetric} & \\ -(A - 12I/L^2)\cos\alpha\sin\alpha, & -A\sin^2\alpha - (12I\cos^2\alpha)/L^2, & -(6I\cos\alpha)/L, & (A - 12I/L^2)\cos\alpha\sin\alpha, & A\sin^2\alpha + (12I\cos^2\alpha)/L^2, & \\ -(6I\sin\alpha)/L, & (6I\cos\alpha)/L, & 2I & (6I\sin\alpha)/L, & -(6I\cos\alpha)/L, & 4I \end{bmatrix}$$

Evaluating, for both elements, only those stiffness terms essential for the analysis:

Element a

$$L^{(a)} = 0.1m, \alpha^{(a)} = 180^\circ, \cos\alpha^{(a)} = -1, \sin\alpha^{(a)} = 0$$

$$[k^{(a)}] = \frac{E}{L^{(a)}} \begin{bmatrix} A & 0 & 0 & \cdot & \cdot & \cdot \\ 0 & 12I/L^2 & -6I/L & \cdot & \cdot & \cdot \\ 0 & -6I/L & 4I & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix}^{(a)} = E \times 10^{-4} \begin{bmatrix} 40 & 0 & 0 & \cdot & \cdot & \cdot \\ 0 & 2.4 & -0.12 & \cdot & \cdot & \cdot \\ 0 & -0.12 & 8\times10^{-3} & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix}$$

No need to complete these rows and columns, for these examples

Element b

$$L^{(b)} = 0.08m, \quad \alpha^{(b)} = 270^\circ, \quad \cos\alpha^{(b)} = 0, \quad \sin\alpha^{(b)} = -1$$

$$[k^{(b)}] = \frac{E}{L^{(b)}} \begin{bmatrix} 12I/L^2 & 0 & 6I/L & \cdot & \cdot & \cdot \\ 0 & A & 0 & \cdot & \cdot & \cdot \\ 6I/L & 0 & 4I & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix}^{(b)} = E \times 10^{-4} \begin{bmatrix} 4.6875 & 0 & 0.1875 & \cdot & \cdot & \cdot \\ 0 & 50 & 0 & \cdot & \cdot & \cdot \\ 0.1875 & 0 & 10.10^{-3} & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix}$$

The structural stiffness matrix can now be assembled. Whilst the structure has a total of 9 dof., only 3 are active, the remaining 6 dof. are suppressed corresponding to the statement in the question regarding the ends being fully restrained. The node numbering adopted in Fig. 9.40 simplifies the stiffness assembly, whereby the first 3 × 3 submatrix terms for both elements are assembled in the first 3 × 3 locations of the structural stiffness matrix; these being the only terms associated with the active dofs. It follows that rearrangement is unnecessary, prior to partitioning. The necessary structural governing equations and hence the required structural stiffness matrix are therefore given as

$$\begin{bmatrix} X_1 \\ Y_1 \\ M_1 \\ \hline X_2 \\ Y_2 \\ M_2 \\ X_3 \\ Y_3 \\ M_3 \end{bmatrix} = E \times 10^{-4} \left[\begin{array}{ccc|c} 40 & 0 & 0 & \\ 4.6875 & 0 & 0.1875 & \\ 0 & 2.4 & -0.12 & \\ 0 & 50 & 0 & \\ 0 & -0.12 & 8\times 10^{-3} & \\ 0.1875 & 0 & 10\times 10^{-3} & \\ \hline & & & \\ & & & \end{array}\right] \begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ \hline u_2 \\ v_2 \\ \theta_2 \\ u_3 \\ v_3 \\ \theta_3 \end{bmatrix}$$

These submatrices are not required, for this example.

(b) (i) Corresponding to $u_2 = v_2 = \theta_2 = u_3 = v_3 = \theta_3 = 0$, the partitioned equations reduce to

$$\begin{bmatrix} X_1 \\ Y_1 \\ M_1 \end{bmatrix} = E \times 10^{-4} \begin{bmatrix} 44.6875 & 0 & 0.1875 \\ 0 & 52.4 & -0.12 \\ 0.1875 & -0.12 & 0.018 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \end{bmatrix}$$

$$= 10^7 \begin{bmatrix} 89.375 & 0 & 0.375 \\ 0 & 104.8 & -0.24 \\ 0.375 & -0.24 & 0.036 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \end{bmatrix}$$

i.e. $\{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\}$

Inverting $[K_{\alpha\alpha}]$ to enable a solution for the displacements from $\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$

where $$\text{adj}\,[K_{\alpha\alpha}] = 10^{14} \begin{bmatrix} 3.7152 & -0.09 & -39.3 \\ -0.09 & 3.0769 & 21.45 \\ -39.3 & 21.45 & 9366.5 \end{bmatrix}$$

and $$\det[K_{\alpha\alpha}] = 10^{21}\{89.375[104.8 \times 0.036 - (-0.24)(-0.24)] - 0 + 0.375(0 - 0.375 \times 104.8)\}$$
$$= 317.3085 \times 10^{21}$$

Then $$[K_{\alpha\alpha}]^{-1} = 10^{-10}\begin{bmatrix} 11.7085 & -0.2836 & -123.8542 \\ -0.2836 & 9.6969 & 67.5998 \\ -123.8542 & 67.5998 & 29518.59 \end{bmatrix}$$

The required displacements are found from

$$p_\alpha = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$$

Substituting $$\begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \end{bmatrix} = 10^{-10}\begin{bmatrix} 11.7085 & -0.2836 & -123.8542 \\ -0.2836 & 9.6969 & 67.5998 \\ -123.8542 & 67.5998 & 29\,518.59 \end{bmatrix} 10^3 \begin{bmatrix} 1 \\ -2.5 \\ -0.5 \end{bmatrix}$$
$$= \begin{bmatrix} 7.434 \times 10^{-6}\ \text{m} \\ -5.833 \times 10^{-6}\ \text{m} \\ -1.505 \times 10^{-3}\ \text{rad} \end{bmatrix}$$

The required nodal displacements are therefore $u_1 = 7.434 \times 10^{-6}$ m, $v_1 = -5.833 \times 10^{-6}$ m and $\theta_1 = -1.505 \times 10^{-3}$ rad.

(b) (ii) With reference to §9.8, the element stress matrix in global coordinates is given as

$$[H^{(e)}] = \frac{E}{L}\begin{bmatrix} -\cos\alpha - 6t\sin(\alpha)/L & -\sin\alpha + 6t\cos(\alpha)/L & 4t & \cos\alpha + 6t\sin(\alpha)/L & \sin\alpha - 6t\cos(\alpha)/L & 2t \\ -\cos\alpha + 6b\sin(\alpha)/L & -\sin\alpha - 6b\cos(\alpha)/L & -4b & \cos\alpha - 6b\sin(\alpha)/L & \sin\alpha + 6b\cos(\alpha)/L & -2b \\ -\cos\alpha + 6t\sin(\alpha)/L & -\sin\alpha - 6t\cos(\alpha)/L & -2t & \cos\alpha - 6t\sin(\alpha)/L & \sin\alpha + 6t\cos(\alpha)/L & -4t \\ -\cos\alpha - 6b\sin(\alpha)/L & -\sin\alpha + 6b\cos(\alpha)/L & 2b & \cos\alpha + 6b\sin(\alpha)/L & \sin\alpha - 6b\cos(\alpha)/L & 4b \end{bmatrix}$$

Evaluating, for both elements, only those terms essential for the analysis:

Element a

$t^{(a)} = 14 \times 10^{-3}$ m, $b^{(a)} = 6 \times 10^{-3}$ m, and recalling from part (a) $L^{(a)} = 0.1$ m, $\alpha^{(a)} = 180°$, $\cos\alpha^{(a)} = -1$, $\sin\alpha^{(a)} = 0$

$$[H^{(a)}] = \frac{200 \times 10^9}{0.1}\begin{bmatrix} 1 & -0.84 & 56 \times 10^{-3} & & & \\ 1 & 0.36 & -24 \times 10^{-3} & & & \\ 1 & 0.84 & -28 \times 10^{-3} & & & \\ 1 & -0.36 & 12 \times 10^{-3} & & & \end{bmatrix}$$

No need to complete these columns for this example

With reference to §9.7, the element stresses are obtained from

With reference to §9.7, the element stresses are obtained from

$$\{\sigma^{(e)}\} = [H^{(e)}]\{s^{(e)}\}$$

where, for element a, the displacement column matrix is

$$\{s^{(a)}\} = \{u_1\ v_1\ \theta_1\ u_2\ v_2\ \theta_2\} \text{ in which } u_2 = v_2 = \theta_2 = 0 \text{ in this example.}$$

Substituting for element a and letting superscript i denote extreme inner fibres and superscript o denote extreme outer fibres, gives

$$\begin{bmatrix}\sigma_1^i\\ \sigma_1^o\\ \sigma_2^i\\ \sigma_2^o\end{bmatrix} = 2\times 10^{12}\left[\begin{array}{c|c|c|c|c|c} 1 & -0.84 & 56\times10^{-3} & & & \\ 1 & 0.36 & -24\times10^{-3} & & & \\ 1 & 0.84 & -28\times10^{-3} & & & \\ 1 & -0.36 & 12\times10^{-3} & & & \end{array}\right]\begin{bmatrix}7.434\times10^{-6}\\ -5.833\times10^{-6}\\ 1.505\times10^{-3}\\ 0\\ 0\\ 0\end{bmatrix} = \begin{bmatrix}-143.89\times10^{6}\\ 82.91\times10^{6}\\ 89.35\times10^{6}\\ -17.05\times10^{6}\end{bmatrix}$$

The required element stresses are therefore $\sigma_1^i = 143.89$ MN/m^2 (C), $\sigma_1^o = 82.91$ MN/m^2(T), $\sigma_2^i = 89.35$ MN/m2(T) and $\sigma_2^o = 17.05$ MN/m^2 (C).

Element b

$t^{(b)} = 6\times10^{-3}$ m, $b^{(b)} = 14\times10^{-3}$ m, and recalling from part (a) $L^{(b)} = 0.08$ m, $\alpha^{(b)} = 270°$, $\cos\alpha^{(b)} = 0$, $\sin\alpha^{(b)} = -1$,

$$[H^{(b)}] = \frac{200\times10^9}{0.08}\left[\begin{array}{c|c|c|c|c|c} 0.45 & 1 & 24\times10^{-3} & & & \\ -1.05 & 1 & -56\times10^{-3} & & & \\ -0.45 & 1 & -12\times10^{-3} & & & \\ 1.05 & 1 & 28\times10^{-3} & & & \end{array}\right]$$

Again, the element stresses are obtained from

$$\{\sigma^{(e)}\} = [H^{(e)}]\{s^{(e)}\}$$

where, for element b, the displacement column matrix is

$$\{s^{(b)}\} = \{u_1\ v_1\ \theta_1\ u_3\ v_3\ \theta_3\} \text{ in which } u_3 = v_3 = \theta_3 = 0 \text{ in this example.}$$

Substituting for element b gives

$$\begin{bmatrix}\sigma_1^o\\ \sigma_1^i\\ \sigma_3^o\\ \sigma_3^i\end{bmatrix} = 2.5\times10^{12}\left[\begin{array}{c|c|c|c|c|c} 0.45 & 1 & 24\times10^{-3} & & & \\ -1.05 & 1 & -56\times10^{-3} & & & \\ -0.45 & 1 & -12\times10^{-3} & & & \\ 1.05 & 1 & 28\times10^{-3} & & & \end{array}\right]\begin{bmatrix}7.434\times10^{-6}\\ -5.833\times10^{-6}\\ -1.505\times10^{-3}\\ 0\\ 0\\ 0\end{bmatrix} = \begin{bmatrix}-96.52\times10^{6}\\ 176.60\times10^{6}\\ 22.20\times10^{6}\\ -100.42\times10^{6}\end{bmatrix}\text{N/m}^2$$

The required element stresses are therefore $\sigma_1^i = 176.60$ MN/m^2 (T), $\sigma_1^o = 96.52$ MN/m^2 (C), $\sigma_3^i = 100.42$ MN/m^2(C) and $\sigma_3^o = 22.20$ MN/m^2 (T).

Example 9.5

Derive the stiffness matrix in global coordinates for a three-node triangular membrane element for plane stress analysis. Assume that the elastic modulus, E, and thickness, t, are constant throughout, and that the displacement functions are

$$u(x, y) = \alpha_1 + \alpha_2 x + \alpha_3 y$$

$$v(x, y) = \alpha_4 + \alpha_5 x + \alpha_6 y$$

Solution

With reference to §9.9 and with respect to the node labelling shown in Fig. 9.41, matrix $[A]$ will be given as:

$$[A] = \left[\begin{array}{ccc|ccc} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ \hline 0 & 0 & 0 & 1 & x_1 & y_1 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{array}\right] = \left[\begin{array}{c|c} A_{\alpha\alpha} & A_{\alpha\beta} \\ \hline A_{\beta\alpha} & A_{\beta\alpha} \end{array}\right]$$

Then $[A]^{-1} = \begin{bmatrix} [A_{\alpha\alpha}]^{-1} & 0 \\ 0 & [A_{\beta\beta}]^{-1} \end{bmatrix}$ where $[A_{\alpha\alpha}]^{-1} = [A_{\beta\beta}]^{-1}$, in this case

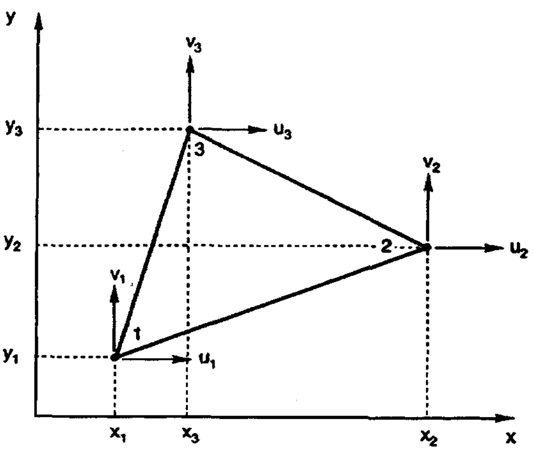

Fig. 9.41.

Obtaining the inverse of the partition

$$\text{adj}\,[A_{\alpha\alpha}] = [C_{\alpha\alpha}]^{\mathrm{T}} = \begin{bmatrix} x_2y_3 - x_3y_2 & y_2 - y_3 & x_3 - x_2 \\ x_3y_1 - x_1y_3 & y_3 - y_1 & x_1 - x_3 \\ x_1y_2 - x_2y_1 & y_1 - y_2 & x_2 - x_1 \end{bmatrix}^{\mathrm{T}}$$

$$= \begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

$$\text{and } \det[A_{\alpha\alpha}] = (x_2y_3 - x_3y_2) - (x_1y_3 - x_3y_1) + (x_1y_2 - x_2y_1)$$
$$= x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$= 2 \times \text{area of element} = 2a, \text{ (see following derivation)}$$

Then
$$[A_{\alpha\alpha}]^{-1} = \frac{\text{adj}[A_{\alpha\alpha}]}{\det[A_{\alpha\alpha}]} = \frac{1}{2a}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

Hence,
$$[A]^{-1} = \frac{1}{2a}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 & 0 & 0 & 0 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 & 0 & 0 & 0 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ 0 & 0 & 0 & y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ 0 & 0 & 0 & x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

Area of element

With reference to Fig. 9.42, area of triangular element,

$$a = \text{area of enclosing rectangle} - (\text{area of triangles b, c and d})$$
$$= (x_2 - x_1)(y_3 - y_1) - (1/2)(x_2 - x_1)(y_2 - y_1) - (1/2)(x_2 - x_3)(y_3 - y_2)$$
$$- (1/2)(x_3 - x_1)(y_3 - y_1)$$
$$= x_2y_3 - x_2y_1 - x_1y_3 + x_1y_1 - (1/2)[x_2y_2 - x_2y_1 - x_1y_2 + x_1y_1)$$
$$+ (x_2y_3 - x_2y_2 - x_3y_3 + x_3y_2) + (x_3y_3 - x_3y_1 - x_1y_3 + x_1y_1)]$$
$$= (1/2)(x_2y_3 - x_2y_1 - x_1y_3 + x_1y_2 - x_3y_2 + x_3y_1)$$
$$= (1/2)[x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)]$$

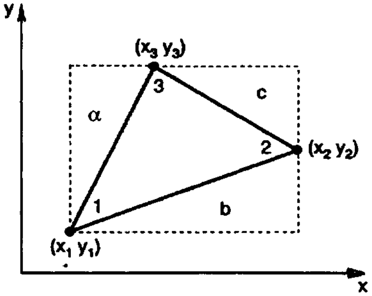

Fig. 9.42.

§9.9 gives matrix $[B]$ as

$$[B] = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix}[A]^{-1}$$

Substituting for $[A]^{-1}$ from above and evaluating the product gives

$$[B] = \frac{1}{2a}\begin{bmatrix} y_2 - y_3 & y_3 - y_1 & y_1 - y_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 & y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \end{bmatrix}$$

The required element stiffness matrix can now be found by substituting into the relation

$$[k] = at\ [B]^T[D][B]$$

$$= \frac{at}{2a}\begin{bmatrix} y_{23} & 0 & x_{32} \\ y_{31} & 0 & x_{13} \\ y_{12} & 0 & x_{21} \\ 0 & x_{32} & y_{23} \\ 0 & x_{13} & y_{31} \\ 0 & x_{21} & y_{12} \end{bmatrix} \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \frac{1}{2a}\begin{bmatrix} y_{23} & y_{31} & y_{12} & 0 & 0 & 0 \\ 0 & 0 & 0 & x_{32} & x_{13} & x_{21} \\ x_{32} & x_{13} & x_{21} & y_{23} & y_{31} & y_{12} \end{bmatrix}$$

where the abbreviation y_{23} denotes $y_2 - y_3$, etc.

Choosing to evaluate the product $[D][B]$ first, gives

$$[k] = \frac{Et}{4a(1-\nu^2)}\begin{bmatrix} y_{23} & 0 & x_{32} \\ y_{31} & 0 & x_{13} \\ y_{12} & 0 & x_{21} \\ 0 & x_{32} & y_{23} \\ 0 & x_{13} & y_{31} \\ 0 & x_{21} & y_{12} \end{bmatrix}\begin{bmatrix} y_{23} & y_{31} & y_{12} & \nu x_{32} & \nu x_{13} & \nu x_{21} \\ \nu y_{23} & \nu y_{31} & \nu y_{12} & x_{32} & x_{13} & x_{21} \\ \frac{(1-\nu)}{2}x_{32} & \frac{(1-\nu)}{2}x_{13} & \frac{(1-\nu)}{2}x_{21} & \frac{(1-\nu)}{2}y_{23} & \frac{(1-\nu)}{2}y_{31} & \frac{(1-\nu)}{2}y_{12} \end{bmatrix}$$

Completing the matrix multiplication, reversing the sequence of some of the coordinates so that all subscripts are in descending order, gives the required element stiffness matrix as

$$[k] = \frac{Et}{4a(1-\nu^2)}\begin{bmatrix} y_{32}^2 + x_{32}^2(1-\nu)/2, & & & & & \\ -y_{32}y_{31} - x_{31}x_{32}(1-\nu)/2, & y_{31}^2 + x_{31}^2(1-\nu)/2, & & & & \\ y_{21}y_{32} + x_{21}x_{32}(1-\nu)/2, & -y_{21}y_{31} - x_{21}x_{31}(1-\nu)/2, & y_{21}^2 + x_{21}^2(1-\nu)/2 & & \text{Symmetric} & \\ -\nu x_{32}y_{32} - y_{32}x_{32}(1-\nu)/2, & \nu x_{32}y_{31} + y_{32}x_{31}(1-\nu)/2, & -\nu x_{32}y_{21} - y_{32}x_{21}(1-\nu)/2, & x_{32}^2 + y_{32}^2(1-\nu)/2, & & \\ \nu x_{31}y_{32} + y_{31}x_{32}(1-\nu)/2, & -\nu x_{31}y_{31} - y_{31}x_{31}(1-\nu)/2, & \nu x_{31}y_{21} + y_{31}x_{21}(1-\nu)/2, & -x_{31}x_{32} - y_{31}y_{32}(1-\nu)/2, & x_{31}^2 + y_{31}^2(1-\nu)/2, & \\ -\nu x_{21}y_{32} - y_{21}x_{32}(1-\nu)/2, & \nu x_{21}y_{31} + y_{21}x_{31}(1-\nu)/2, & -\nu x_{21}y_{21} - y_{21}x_{21}(1-\nu)/2, & x_{21}x_{32} + y_{21}y_{32}(1-\nu)/2, & -x_{21}x_{31} - y_{21}y_{31}(1-\nu)/2, & x_{21}^2 + y_{21}^2(1-\nu)/2 \end{bmatrix}$$

Example 9.6

(a) Evaluate the element stiffness matrix, in global coordinates, for the three-node triangular membrane element, labelled a in Fig. 9.43. Assume plane stress conditions, Young's modulus, $E = 200\ \text{GN/m}^2$, Poisson's ratio, $\nu = 0.3$, thickness, $t = 1$ mm, and the same displacement functions as Example 9.5.

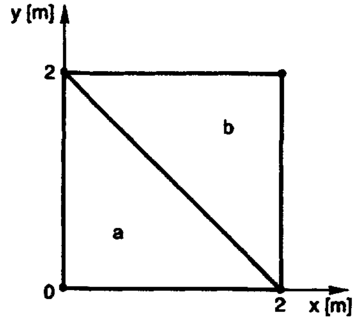

Fig. 9.43.

(b) Evaluate the element stiffness matrix for element b, assuming the same material properties and thickness as element a. Hence, evaluate the assembled stiffness matrix for the continuum.

Solution

(a) Figure 9.44 shows suitable node labelling for a single triangular membrane element. The resulting element stiffness matrix from the previous Example, 9.5, can be utilised. A specimen evaluation of an element stiffness term is given below for k_{11}. The rest are obtained by following the same procedure.

$$k_{11} = \frac{Et}{4a(1-\nu^2)}[y_{32}^2 + x_{32}^2(1-\nu)/2]$$

$$= \frac{Et}{4a(1-\nu^2)}[(y_3 - y_2)^2 + (x_3 - x_2)^2(1-\nu)/2]$$

Substituting

$$= \frac{200 \times 10^9 \times 1 \times 10^{-3}}{4 \times 2(1-0.3^2)}[(2-0)^2 + (0-2)^2(1-0.3)/2]$$

$$= 14.835 \times 10^7 \text{ N/m}$$

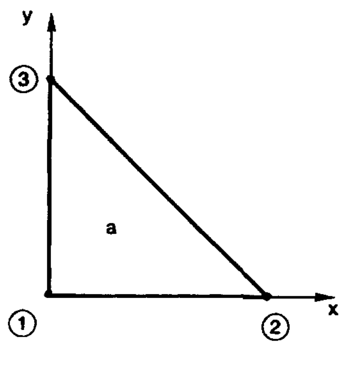

Fig. 9.44.

Evaluation of all the terms leads to the required triangular membrane element stiffness matrix for element a, namely

$$[k^{(a)}] = 10^7[\text{N/m}]\begin{bmatrix} 14.835 & & & & & \\ -10.989 & 10.989 & & \text{symmetric} & & \\ -3.846 & 0 & 3.846 & & & \\ 7.143 & -3.297 & -3.846 & 14.835 & & \\ -3.846 & 0 & 3.846 & -3.846 & 3.846 & \\ -3.297 & 3.297 & 0 & -10.989 & 0 & 10.989 \end{bmatrix}$$

(b) Element b can temporarily also be labelled with node numbers 1, 2 and 3, as element a. To avoid confusion, this is best done with the elements shown "exploded", as in Fig. 9.45. The alternative is to re-number the subscripts in the element stiffness matrix result from Example 9.5.

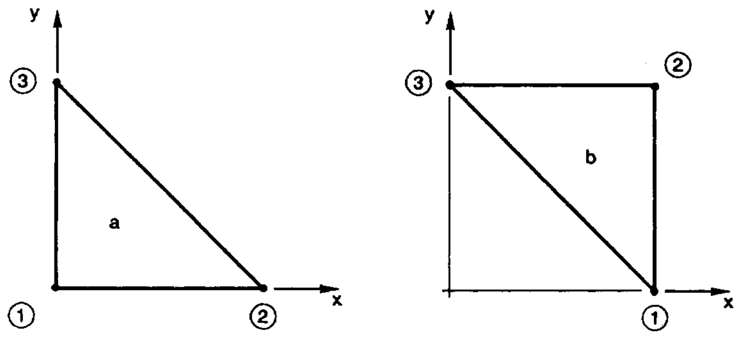

Fig. 9.45.

Performing the evaluations similar to part (a) leads to the required stiffness matrix for element b, namely

$$[k^{(b)}] = 10^7\ [\text{N/m}]\begin{bmatrix} 3.846 & & & & & \\ -3.846 & 14.835 & & \text{symmetric} & & \\ 0 & -10.989 & 10.989 & & & \\ 0 & -3.297 & 3.297 & 10.989 & & \\ -3.846 & 7.143 & -3.297 & -10.989 & 14.835 & \\ 3.846 & -3.846 & 0 & 0 & -3.846 & 3.846 \end{bmatrix}$$

With reference to §9.10, the structural stiffness matrix can now be assembled using a dof. correspondence table. The order of the structural stiffness matrix will be 8 × 8, corresponding to four nodes, each having 2 dof. The dof. sequence, u_1, u_2, u_3, v_1, v_2, v_3, adopted for the convenience of inverting matrix $[A]$, covered in §9.9, can be converted to the more usual sequence, i.e. u_1, v_1, u_2, v_2, u_3, v_3, with the aid a dof. correspondence table. Whilst this re-sequencing is optional, the converted sequence is likely to result in less rearrangement of rows and columns, prior to partitioning the assembled stiffness matrix, than would otherwise be needed.

If row and column interchanges are to be avoided in solving the following Example, 9.7, and therefore save some effort, then the dof. labelling of Fig. 9.46 is recommended. This implies the final node numbering, also shown.

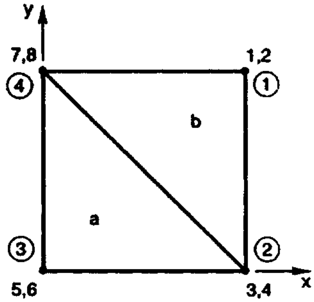

Fig. 9.46.

The dof. correspondence table will be as follows:

Row/column in $[k^{(e)}]$		1	2	3	4	5	6
Row/column in [K]	*a*	5	3	7	6	4	8
	b	3	1	7	4	2	8

Assembling the structural stiffness matrix, gives

$$[K] = 10^7[\mathrm{N/m}]\left[\begin{array}{c|c|c|c|c|c|c|c}
14.835 & 7.143 & -3.846 & -3.297 & & & -10.989 & -3.846 \\ \hline
7.143 & 14.835 & -3.846 & -10.989 & & & -3.297 & -3.846 \\ \hline
\begin{matrix} \\ -3.846\end{matrix} & \begin{matrix} \\ -3.846\end{matrix} & \begin{matrix}10.989\\ 3.846\end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}-10.989\\ \end{matrix} & \begin{matrix}-3.297\\ \end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}3.297\\ 3.846\end{matrix} \\ \hline
\begin{matrix} \\ -3.297\end{matrix} & \begin{matrix} \\ -10.989\end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}3.846\\ 10.989\end{matrix} & \begin{matrix}-3.846\\ \end{matrix} & \begin{matrix}-3.846\\ \end{matrix} & \begin{matrix}3.846\\ 3.297\end{matrix} & \begin{matrix}0\\ 0\end{matrix} \\ \hline
 & & -10.989 & -3.846 & 14.835 & 7.143 & -3.846 & -3.297 \\ \hline
 & & -3.297 & -3.846 & 7.143 & 14.835 & -3.846 & -10.989 \\ \hline
\begin{matrix} \\ -10.989\end{matrix} & \begin{matrix} \\ -3.297\end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}3.846\\ 3.297\end{matrix} & \begin{matrix}-3.846\\ \end{matrix} & \begin{matrix}-3.846\\ \end{matrix} & \begin{matrix}3.846\\ 10.989\end{matrix} & \begin{matrix}0\\ 0\end{matrix} \\ \hline
\begin{matrix} \\ -3.846\end{matrix} & \begin{matrix} \\ -3.846\end{matrix} & \begin{matrix}3.297\\ 3.846\end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}-3.297\\ \end{matrix} & \begin{matrix}-10.989\\ \end{matrix} & \begin{matrix}0\\ 0\end{matrix} & \begin{matrix}10.989\\ 3.846\end{matrix}
\end{array}\right]$$

Summing the element stiffness contributions, and writing the structural governing equations, gives the result as

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \\ X_4 \\ Y_4 \end{bmatrix} = 10^7\ [\mathrm{N/m}] \begin{bmatrix}
14.835 & 7.143 & -3.846 & -3.297 & 0 & 0 & -10.989 & -3.846 \\
7.143 & 14.835 & -3.846 & -10.989 & 0 & 0 & -3.297 & -3.846 \\
-3.846 & -3.846 & 14.835 & 0 & -10.989 & -3.297 & 0 & 7.143 \\
-3.297 & -10.989 & 0 & 14.835 & -3.846 & -3.846 & 7.143 & 0 \\
0 & 0 & -10.989 & -3.846 & 14.835 & 7.143 & -3.846 & -3.297 \\
0 & 0 & -3.297 & -3.846 & 7.143 & 14.835 & -3.846 & -10.989 \\
-10.989 & -3.297 & 0 & 7.143 & -3.846 & -3.846 & 14.835 & 0 \\
-3.846 & -3.846 & 7.143 & 0 & -3.297 & -10.989 & 0 & 14.835
\end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix}$$

i.e. $\{P\} = [K]\{p\}$

where $[K]$ is the required assembled stiffness matrix.

Example 9.7

Figure 9.47 shows a 1 mm thick sheet of steel, one edge of which is fully restrained whilst the opposite edge is subjected to a uniformly distributed tension of total value 40 kN. For the material Young's modulus, $E = 200$ GN/m^2 and Poisson's ratio, $\nu = 0.3$, and plane stress condition can be assumed.

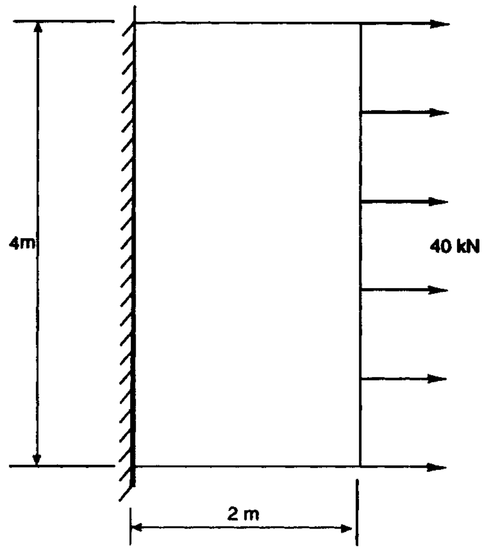

Fig. 9.47.

(a) Taking advantage of any symmetry, using two triangular membrane elements and hence the assembled stiffness matrix derived for the previous Example, 9.6, determine the nodal displacements in global coordinates.
(b) Determine the corresponding element principal stresses and their directions and illustrate these on a sketch of the continuum.

Solution

(a) Advantage can be taken of the single symmetry by modelling only half of the continuum. Figure 9.48 shows suitable node and dof. labelling, and division of the upper half of the continuum into two triangular membrane elements. Reference to the previous Example, 9.6, will reveal that the assembled stiffness matrix derived in answering this question can, conveniently, be utilised in solving the current example.

To simulate the clamped edge, dofs. 5 to 8 need to be suppressed, i.e. $u_3 = v_3 = u_4 = v_4 = 0$. Additionally, whilst node number 2 should be unrestrained in the x-direction, freedom in the y-direction needs to be suppressed to simulate the symmetry condition, i.e. $v_2 = 0$. Applying these boundary conditions and hence partitioning the structural stiffness matrix result from Example 9.6, gives the reduced equations as

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \end{bmatrix} = 10^7[\text{N/m}] \begin{bmatrix} 14.835 & 7.143 & -3.846 \\ 7.143 & 14.835 & -3.846 \\ -3.846 & -3.846 & 14.835 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \end{bmatrix} \quad \text{i.e. } \{P_\alpha\} = [K_{\alpha\alpha}]\{p_\alpha\}$$

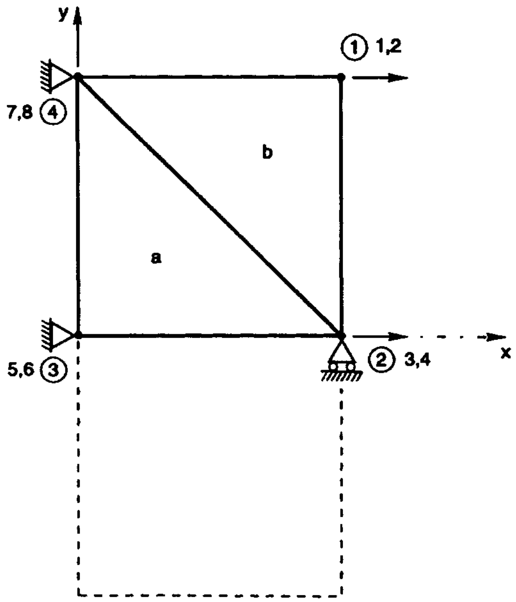

Fig. 9.48.

Inverting $[K_{\alpha\alpha}]$ to enable a solution for the displacements from $\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$

$$\text{where adj } [K_{\alpha\alpha}] = 10^{14}\begin{bmatrix} 205.286 & -91.175 & 29.583 \\ -91.175 & 205.286 & 29.583 \\ 29.583 & 29.583 & 169.055 \end{bmatrix}$$

and det $[K_{\alpha\alpha}] = 10^{21}[14.835(205.286) - 7.143(91.175) - 3.846(29.583)] = 2\,280.4 \times 10^{21}$

$$\text{Then } [K_{\alpha\alpha}]^{-1} = 10^{-10}\begin{bmatrix} 90.03 & -39.98 & 12.97 \\ -39.98 & 90.03 & 12.97 \\ 12.97 & 12.97 & 74.13 \end{bmatrix}$$

With reference to §9.4.7, the nodal load column matrix corresponding to a uniformly distributed load of 10 kN/m, will be given by

$$\{P_\alpha\} = \begin{bmatrix} X_1 \\ Y_1 \\ X_2 \end{bmatrix} = 10^3 \begin{bmatrix} 10 \\ 0 \\ 10 \end{bmatrix}_{[N]}$$

Hence, the nodal displacements are found from

$$\{p_\alpha\} = [K_{\alpha\alpha}]^{-1}\{P_\alpha\}$$

Substituting,

$$\begin{bmatrix} u_1 \\ v_1 \\ u_2 \end{bmatrix} = 10^{-10}\begin{bmatrix} 90.03 & -39.98 & 12.97 \\ -39.98 & 90.03 & 12.97 \\ 12.97 & 12.97 & 74.13 \end{bmatrix} 10^3 \begin{bmatrix} 10 \\ 0 \\ 10 \end{bmatrix} = 10^{-6}\begin{bmatrix} 103 \\ -27 \\ 87 \end{bmatrix}_{m} = \begin{bmatrix} 0.103 \\ -0.027 \\ 0.087 \end{bmatrix}_{mm}$$

The required nodal displacements are therefore $u_1 = 0.103$ mm, $v_1 = -0.027$ mm and $u_2 = 0.087$ mm.

(b) With reference to §9.9, element direct and shearing stresses are found from

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = [D][B] \begin{bmatrix} u_i \\ u_j \\ u_k \\ v_i \\ v_j \\ v_k \end{bmatrix}$$

where, from Example 9.5,

$$[D][B] = \frac{E}{2a(1-v^2)}$$

$$\begin{bmatrix} y_{23} & y_{31} & y_{12} & vx_{32} & vx_{13} & vx_{21} \\ vy_{23} & vy_{31} & vy_{12} & x_{32} & x_{13} & x_{21} \\ \frac{(l-v)}{2}x_{32} & \frac{(l-v)}{2}x_{13} & \frac{(l-v)}{2}x_{21} & \frac{(l-v)}{2}y_{23} & \frac{(l-v)}{2}y_{31} & \frac{(l-v)}{2}y_{12} \end{bmatrix}$$

Evaluating the stresses for each element:

Element a

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{200 \times 10^9}{2 \times 2(1-0.3^2)} \begin{bmatrix} -2 & 2 & 0 & -0.6 & 0 & 0.6 \\ -0.6 & 0.6 & 0 & -2 & 0 & 2 \\ -0.7 & 0 & 0.7 & -0.7 & 0.7 & 0 \end{bmatrix} \begin{bmatrix} 0 \\ 87 \times 10^{-6} \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} 9.56 \times 10^6 \\ 2.87 \times 10^6 \\ 0 \end{bmatrix}_{N/m^2}$$

The required principal stresses for element a are therefore $\sigma_1 = 9.56$ MN/m^2 (T) and $\sigma_2 = 2.87$ MN/m^2 (T), and are illustrated in Fig. 9.49.

Element b

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{200 \times 10^9}{2 \times 2(1-0.3^2)} \begin{bmatrix} 0 & 2 & -2 & -0.6 & 0.6 & 0 \\ 0 & 0.6 & -0.6 & -2 & 2 & 0 \\ -0.7 & 0.7 & 0 & 0 & 0.7 & -0.7 \end{bmatrix} \begin{bmatrix} 87 \times 10^{-6} \\ 103 \times 10^{-6} \\ 0 \\ 0 \\ -27 \times 10^{-6} \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} 10.43 \times 10^6 \\ 0.43 \times 10^6 \\ 2.92 \times 10^6 \end{bmatrix}_{N/m^2}$$

The principal stresses are found from

$$\sigma_1, \sigma_2 = \tfrac{1}{2}(\sigma_{xx} + \sigma_{yy}) \pm \tfrac{1}{2}\sqrt{[(\sigma_{xx} - \sigma_{yy})^2 + 4\sigma_{xy}^2]}$$

Substituting gives

$$\sigma_1, \sigma_2 = \{\tfrac{1}{2}(10.43 + 0.43) \pm \tfrac{1}{2}\sqrt{[(10.43 - 0.43)^2 + 4 \times 2.92^2]}\}10^6 \text{ N/m}^2$$
$$= (5.43 \pm 5.79)10^6 \text{ N/m}^2$$

giving $\sigma_1 = 11.22$ MN/m^2 (T) and $\sigma_2 = 0.36$ MN/m^2 (C)

The directions are found from

$$\theta = \tfrac{1}{2}\tan^{-1}[2\sigma_{xy}/(\sigma_{xx} - \sigma_{yy})]$$

substituting gives

$$\theta = \tfrac{1}{2}\tan^{-1}[2 \times 2.92/(10.43 - 0.43)] = 15.14^\circ$$

The required principal stresses for element b are therefore $\sigma_1 = 11.22$ MN/m^2 (T) and $\sigma_2 = 0.36$ MN/m^2 (C) and are illustrated in Fig. 9.49.

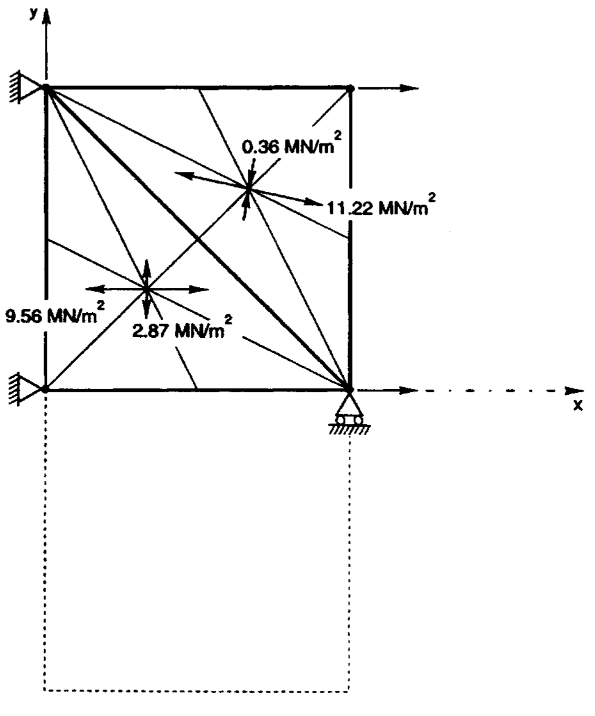

Fig. 9.49.

Problems

9.1 Figure 9.50 shows a support structure in the form of a pin-jointed plane frame, all three members of which are steel, of the same uniform cross-sectional area and length, such that $AE/L = 200$ kN/m, throughout.

(a) Using the displacement based finite element method and treating each member as a rod, determine the nodal displacements with respect to global coordinates for the frame shown in Fig. 9.50.

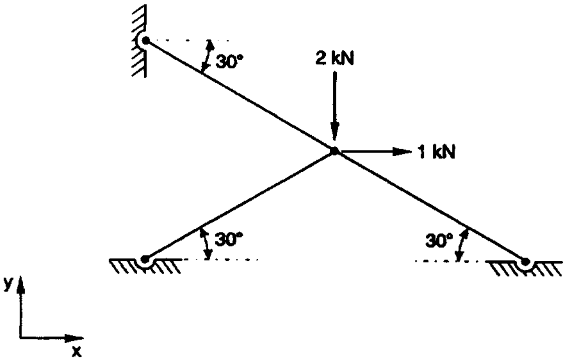

Fig. 9.50.

(b) Hence, determine the nodal reactions.

[−0.387, −13.557 mm, −1.116, 0.644, 1.233, 0.712, −1.116, 0.644 kN]

9.2 Figure 9.51 shows a roof truss, all members of which are made from steel, and have the same cross-sectional area, such that $AE = 10$ MN, throughout. For the purpose of analysis the truss can be treated as a pin-jointed plane frame. Using the displacement based finite element method, taking advantage of any symmetry and redundancies and treating each member as a rod element, determine the nodal displacements with respect to global coordinates.

[0.516, −2.280, −2.313 mm]

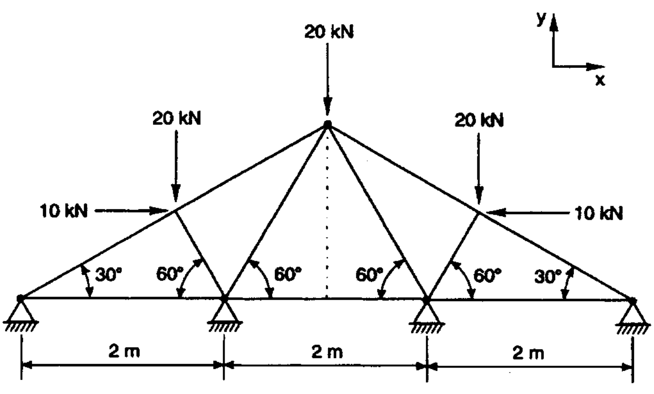

Fig. 9.51.

9.3 A recovery vehicle towing jib is shown in Fig. 9.52. It can be assumed that the jib can be idealised as a pin-jointed plane frame, with all three members made from steel, of the same uniform cross-sectional area, such that $AE = 40$ MN, throughout. Using the displacement based finite element method and treating each member as a rod, determine the maximum load P which can be exerted whilst limiting the resultant maximum deflection to 10 mm.

[36.5 kN]

9.4 A hoist frame, arranged as shown in Fig. 9.53, comprises uniform steel members, each 1m long for which $AE = 200$ MN, throughout.

(a) Using the displacement based finite element method and assuming the frame members to be planar and pin-jointed, determine the nodal displacements with respect to global coordinates for the frame loaded as shown.

(b) Hence, determine the corresponding nodal reactions.

[0.5, 0.75, −0.722 mm, 0, 86.6, −25.0, −43.3 kN]

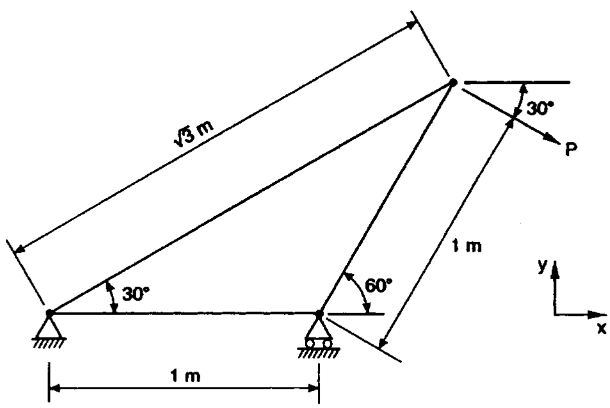

Fig. 9.52.

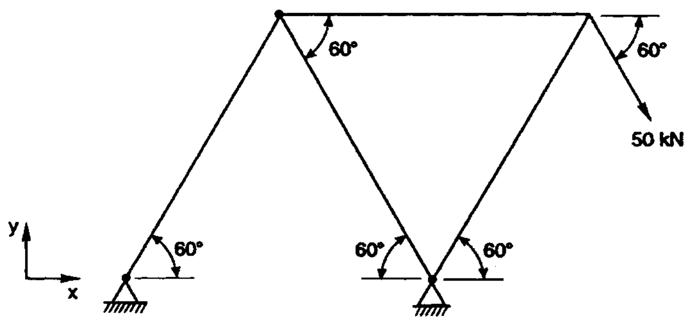

Fig. 9.53.

9.5 Figure 9.54 shows a towing "bracket" for a motor vehicle. Both members are made from uniform cylindrical section steel tubing, of outside diameter 40 mm, cross-sectional area 2.4×10^{-4} m^2, relevant second moment of area 4.3×10^{-8} m^4 and Young's modulus 200 GN/m^2. Using the displacement based finite element method, a simple beam element representation and assuming both members and load are coplanar:

(a) assemble the necessary terms in the structural stiffness matrix;

(b) hence, for the idealisation shown in Fig. 9.54, determine (i) the nodal displacements with respect to global coordinates, and (ii) the resultant maximum stresses at the built-in ends and at the common junction.

[0.421 mm, 0.103°, 131.34, 106.77, 127.20, 110.91 MN/m^2]

9.6 A stepped steel shaft supports a pulley, as shown in Fig. 9.55, is rigidly built-in at one end and is supported in a bearing at the position of the step. The bearing provides translational but not rotational restraint. Young's modulus for the material is 200 GN/m^2.

(a) Using the displacement based finite element method obtain expressions for the nodal displacements in global coordinates, using a two-beam model.

(b) Given that, because of a design requirement, the angular misalignment of the bearing cannot exceed 0.5°, determine the maximum load, P, that can be exerted on the pulley.

(c) Sketch the deformed geometry of the beam.

[$4.985 \times 10^{-8}P$, $-2.905 \times 10^{-8}P$, $8.475 \times 10^{-7}P$, 175 kN]

9.7 Figure 9.56 shows a chassis out-rigger which acts as a body support for an all-terrain vehicle. The out-rigger is constructed from steel channel section rigidly welded at the out-board edge and similarly welded to the vehicle chassis. For the channel material, Young's modulus, $E = 200$ GN/m^2, relevant second moment of area, $I = 2 \times 10^{-9}$ m^4 and cross-sectional area, $A = 4 \times 10^{-5}$ m^2. Using the displacement based finite element method, and representing the constituent members as simple beam elements

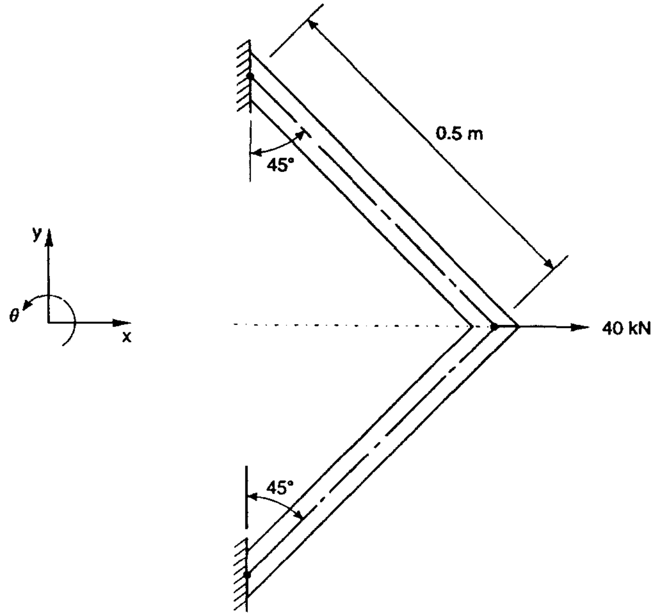

Fig. 9.54.

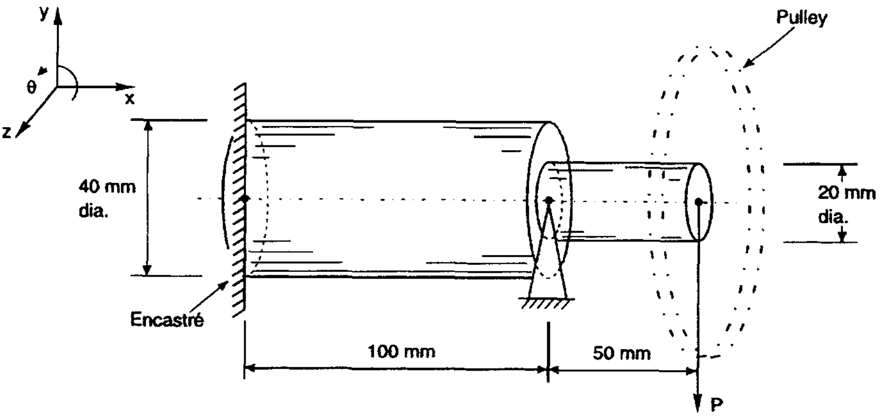

Fig. 9.55.

(a) determine the nodal displacements with respect to global coordinates.

(b) A modal analysis reveals that, to avoid resonance, the vertical stiffness of the out-rigger needs to be increased. Assuming only one of the members is to be stiffened, state which member and whether it should be the cross-sectional area or the second moment of area which should be increased, for most effect.

[0.1155, −0.4418 mm, −0.322°, inclined member's csa.]

9.8 The plane frame shown in Fig. 9.57 forms part of a steel support structure. The three members are rigidly connected at the common junction and are built-in at their opposite ends. All three members can be assumed to be axially rigid, and of constant cross-sectional area, A, relevant second moment of area, I, and Young's modulus, E. Using the displacement based finite element method and representing each member as a simple beam:

(a) show that the angular displacement of the common node, due to application of the moment, M, is given by $ML/8EI$;

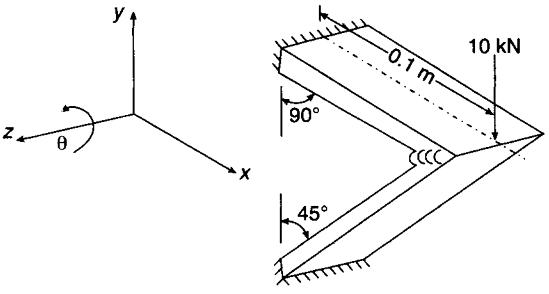

Fig. 9.56.

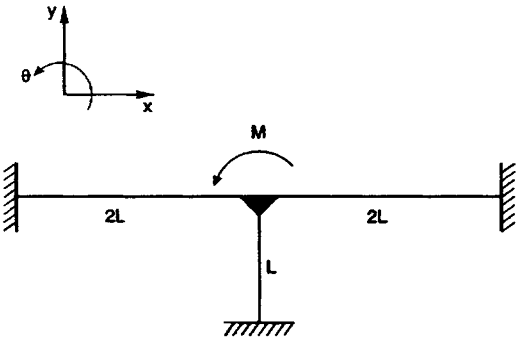

Fig. 9.57.

(b) determine all nodal reaction forces and moments due to this moment, and represent these reactions on a sketch of the frame. Show that both force and moment equilibrium is satisfied.

(c) If, due to a manufacturing defect, the joint at the lower end of the vertical member undergoes an angular displacement of $ML/4EI$, whilst all other properties remain unchanged, obtain a new expression for the angular displacement at the common junction.

[0, −3M/16L, M/8, −3M/4L, 0, 0, 3M/16L, M/8, 3M/4L, 0, M/4, ML/16EI]

9.9 Using the displacement based finite element method and a three-node triangular membrane element representation, determine the nodal displacements in global coordinates for the continuum shown in Fig. 9.58. Take advantage of any symmetry, assume plane stress conditions and use only two elements in the discretisation. For the material assume Young's modulus, $E = 200$ GN/m^2 and Poisson's ratio, $\nu = 0.3$.

[-3.00×10^{-6}, 10.01×10^{-6}, -3.00×10^{-6}, 10.01×10^{-6} m]

9.10 A crude lifting device is fabricated from a triangular sheet of steel, 6 mm thick, as shown in Fig. 9.59. Assume for the material Young's modulus, $E = 200$ GN/m^2 and Poisson's ratio, $\nu = 0.3$, and that plane stress conditions are appropriate.

(a) Taking advantage of any symmetry, ignoring any instability and using only a single three-node triangular membrane element representation, use the displacement based finite element method to predict the nodal displacements in global coordinates.

(b) Determine the corresponding element principal stresses and their directions, and show these on a sketch of the element. [−0.05, −0.17, −0.60 mm, 134.85 MN/m^2 (T) at 31.7° from x-direction, 51.50 MN/m^2 (C)]

9.11 The web of a support structure, fabricated from steel sheet 1 mm thick, is shown in Fig. 9.60. Assume for the material Young's modulus, $E = 207$ GN/m^2 and Poisson's ratio, $\nu = 0.3$, and that plane stress conditions are appropriate.

(a) Neglecting any stiffening effects of adjoining members and any instability and using only a single three-node triangular membrane element representation, use the displacement based finite element method to predict the nodal displacements with respect to global coordinates.

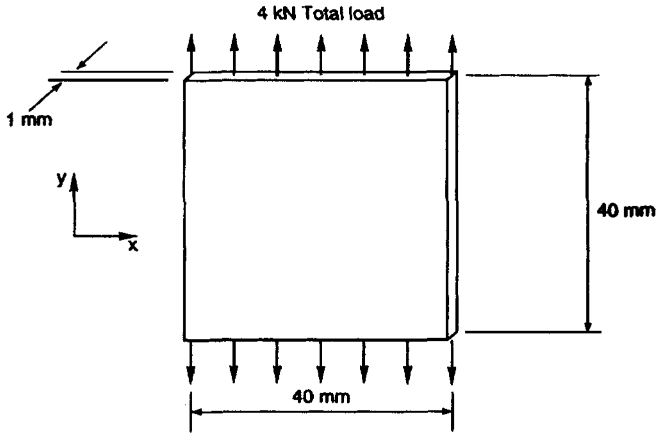

Fig. 9.58.

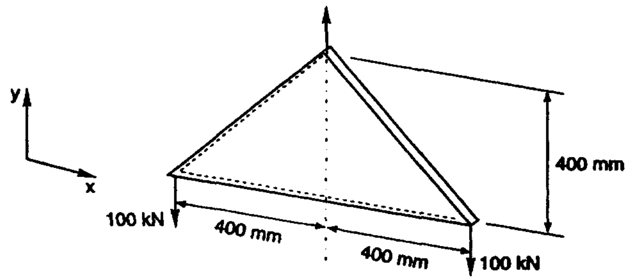

Fig. 9.59.

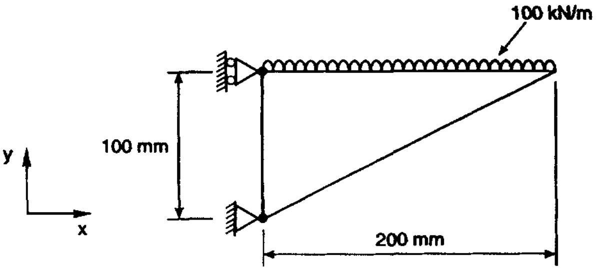

Fig. 9.60.

(b) Determine the corresponding element principal stresses and their directions, and show these on a sketch of the web.

[0.058, −0.60, −0.10 mm, 123.6 MN/m^2 (T) at −31.7° from x-direction, 323.6 MN/m^2 (C)]

9.12 Derive the stiffness matrix in global coordinates for a three-node triangular membrane element for plane *strain* conditions. Assume the displacement functions are the same as those of Example 9.5.