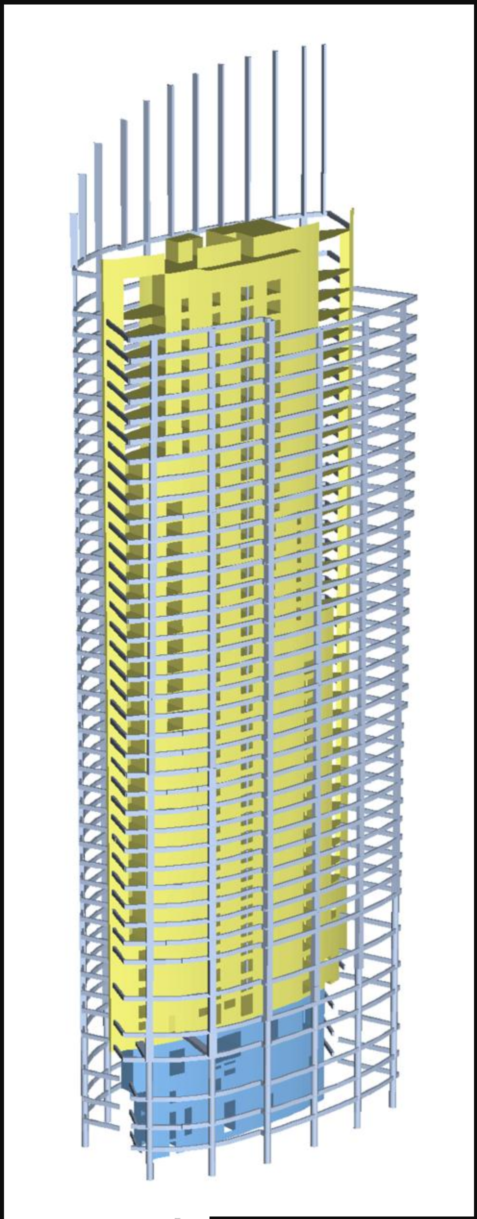
2015

THE UNIVERSITY OF SYDNEY

# Finite Element Modelling

## For Civil Engineering



Strand7


Fernando Alonso-Marroquin
The University of Sydney

2/3/2015

# CONTENTS

Chapter 1

# INTRODUCTION TO FINITE ELEMENT MODELLING

This course aims to provide a modern formulation of finite element analysis for modelling engineering systems. The main idea of *modelling* is to use physical principles and mathematics to arrive at an approximate description of phenomena. These phenomena span a wide range of situations in civil engineering that demand predictive capabilities. A few examples: material behaviour of human-made materials, stability of structures, and transport of heat, water, or contaminants. In structural engineering, one of the responsibilities of the design engineer is to use predictive tools to devise arrangements and establish proportions of members – to withstand, economically and efficiently, the conditions anticipated over the lifetime of a structure. In environmental engineering the description of phenomena is used to improve the natural environment, to provide healthy water, air, and soil for humans and ecosystems, and to remediate pollution produced by human activities.

Mathematical modelling complements methods based on empirical experience. Empiricists base their formulae and design decisions on experimental analysis, and this approach can be very competitive and effective if the analysis is carried out properly. Repeatability, rapidity, and reliable accuracy are among its strengths; but the major disadvantage of the empirical method is that it usually yields only one data point of information in the spectrum of the physics involved. If the system is changed from the originally tested specimen (perhaps in dimensions, materials, or loading conditions), the experiment needs to be repeated on the new structure. The costs can be prohibitive.

Experiments should be used as the starting point of any investigation. Results of experimental tests provide a window of insight, and hence clues to the behaviour of the structure and the phenomenon governing it. The best engineering approach to a problem is to evolve mathematical methods based on mechanical principles and experimental insight, and to use empirical methods for the ultimate verification of any theoretical or numerical solutions obtained through modelling.

Development of mathematical models leads to a set of differential equations called *governing equations*. In just a few cases it is possible to solve these equations *analytically*. With *analytical expressions* we achieve explicit derivation of unknown variables in terms of the known parameters using well-known mathematical functions. These expressions are *closed form solutions*, and often they make strong assumptions – such as perfect elasticity, and extremely simplified geometry. But real engineering problems often require a detailed description of the geometry of systems, like the cross section of a beam or a retaining wall; or they may be insoluble without a complex specification of material behaviour, perhaps with non-linearity or irreversibility. In these cases elegant analytical solutions are not available. We use *numerical analysis* instead, which involves the use of algorithms implemented on computers to arrive at approximate solutions of the governing equations, to the necessary degree of precision.

Thanks to the rapid increase of computer power, numerical analysis is one of the fastest-growing areas in engineering. Finite element modelling is among the most popular methods of numerical analysis for engineering, as it allows modelling of physical processes in domains with complex geometry and a wide range of constraints. The basic idea of finite element modelling is to divide the system into parts and apply the governing equations at each one of them. The analysis for each part leads to a set of algebraical equations. Equations for all of the parts are assembled to create a global matrix equation, which is solved using numerical methods. The beauty of finite element modelling is that it has a strong mathematical basis in variational methods pioneered by mathematicians such as Courant, Ritz, and Galerkin. The people who actually elaborated the method were engineers working toward greater stability for fuselages and wings of aircraft. In 1943 Richard Courant (in the United States, having left Germany early in World War II) came up with the first finite element modelling using nothing more than high-school mathematics. In 1960, John Argyris (University of Stuttgart) established the mathematical basis of the method to allow its application to problems beyond structures, such as seepage analysis, heat transfer, and long-time settlement.

In the sixties, the golden age of finite element modelling, scientists and engineers pushed the boundaries of its application, and developed ever more efficient algorithms. Nowadays, finite element analysis is a well-established method available in several commercial codes. But numerical analysis research has not stopped there! Mesh-free methods have been proposed, which do not require the mesh used in finite elements. Discrete element methods have been developed with the aim of investigating systems of many parts interacting via contact forces. Enthusiasm for these models has spilled beyond the borders of science and engineering. They are now used in several computer games, and have inspired movies and visions of interactive computer simulations such as The Matrix. Such fascinating advances in computer modelling would be impossible without a foundation in finite element methods.

Welcome to the fascinating world of the numerical modelling!

Chapter 2

# MATHEMATICAL FOUNDATION OF FINITE ELEMENT ANALYSIS

In this chapter we introduce the key concepts of finite element analysis by considering few one-dimensional problems. The formulation includes three steps. The first step is the derivation of the governing equations of the problem along with the identification of its *boundary conditions*. The second step involves the conversion of the governing equations into a *weak form* that allows the formulation of the finite element theory. In the third step we subdivide the *domain* of the system into a set of discrete sub-domains that are called *elements*, and we define the *shape functions* in each element. By expressing the seeking solution in terms of those shape functions, the governing equations are converted into a global matrix equation that is solved numerically. This is the essence of all finite element methods.

## 2.1: Governing equations: strong formulation

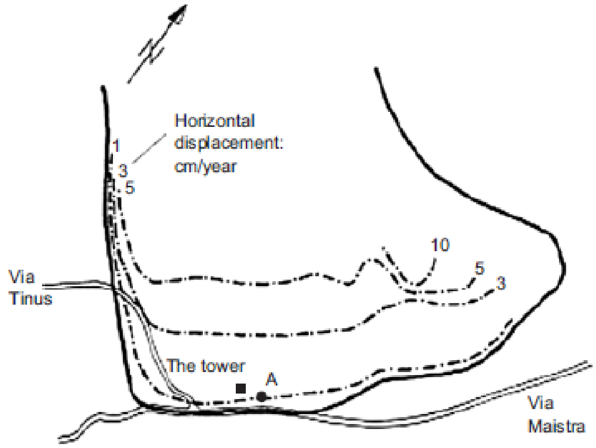
The first step towards the mathematical modelling of any problem in science or engineering is the derivation of the differential equations of the quantity that needs to be solved. This quantity can be the displacement on a building under wind load, the temperature in an electrical circuit, the distribution of pore pressure in a dam, or the electro-magnetic field produced by an antenna. In most of the case these equations can be assembled using three different parts.

1) **Kinematic equations**, describing the gradient (derivative) of the variable we want to solve. For example: the gradient of the displacement is the strain, and the gradient of the head is the hydraulic gradient.

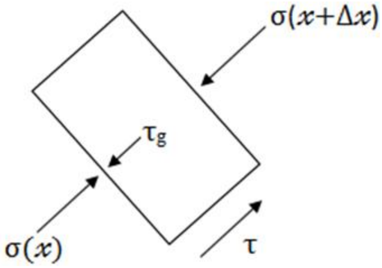
2) **Balance equations**, which are the mathematical expression of the conservation laws in physics. Conserved quantities are usually mass, momentum, and energy. For example, for structures in equilibrium, the conservation of momentum lead to the so-called *static equations*; The Naiver-Stokes equations in fluid mechanics involve conservation of mass and momentum.

3) **Constitutive equations**, which represent the material properties of the system of study. These properties are usually derived from experimental tests. In structural mechanics the constitutive model is the stress-strain relation which is given in terms of a stiffness tensor. In transport of heat, radiation or pollutants the constitutive models consist on transport coefficients, such as permeability in seepage flow, or conductivity in heat transfer.

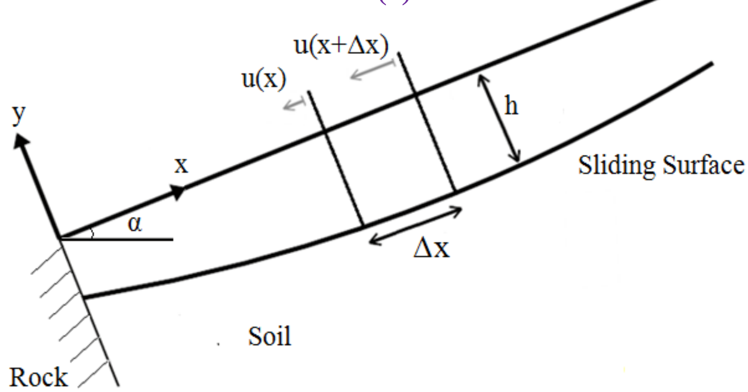
(a)




(b)




(c)




(d)

Figure 2.1. From left to right: (a) the Leaning Tower of St Moritz; (b) the landslide displacements in the lower 200 m, (c) the mathematical model to predicting landslide displacement; and (d) free body diagram of a slide (Puzrin, A. M. & Sterba, I. (2006). Geotechnique 56, No. 7, 483–489)

To illustrate the derivation of the governing equations, let us consider a simple mathematical model for a complex engineering problem, as shown in Figure (2.1). This is related to a landslide displacement in Switzerland, which have led to the leaning of the St Moritz Tower: the displacement of the inclined slope is constrained by a rock outcrop along the Via Maistra, as shown in the Figure (2.1). Geological survey has shown that the deformation occurs above a sliding layer, and it is constrained by a rock outcrop at the bottom. For simplicity, we assume that the deformation u(x) only occurs in the direction of the slope. We want to derive the governing equations of u using the method of infinitesimals. First we divide the slope in slides perpendicular to the slope direction. Let u(x) and u(x+Δx) be the displacement at both side of the slide initially placed at the position x. The width of the slide, Δx, is assumed to be infinitesimally small, which means, very small.

The kinematic equation is nothing more than the definition of strain:

$$\varepsilon \doteq \frac{u(x+\Delta x)-u(x)}{\Delta x},$$

where '.=' means that the expression is valid when Δx is infinitesimally small. Thus the equation can be converted into a differential equation using the definition of derivative

$$\varepsilon=\frac{du}{dx} \tag{2.1}$$

This corresponds to the kinematic equation of the problem.

Now we will construct the balance equation assuming that the system is in equilibrium. Since the problem is one-dimensional, the equation of conservation of momentum corresponds to the balance of forces in the x-direction:

$$\sigma(x)h-\sigma(x+\Delta x)h+\tau\Delta x-\tau_g\Delta x=0 \tag{2.2}$$

where σ is the stress acting on the x-direction, also known as earth pressure, $\tau$ is the shear stress acting on the sliding layer, and $\tau_g=\gamma h \sin\alpha$ is the gravity force, and γ is the unit weight of the soil. Eq. (2.2) can be rearranged as

$$\frac{\sigma(x+\Delta x)-\sigma(x)}{\Delta x}=-\frac{\tau_g-\tau}{h} \tag{2.3}$$

Since Δx is infinitesimal, the equation above can be converted into

$$\frac{d\sigma}{dx}=-f(x) \tag{2.4}$$

where f(x) is the *external loads* apply to the system that in this case consists of a gravitational load minus the shear stress at the bottom of the boundary. This equation corresponds to the static equation of the problem.

We notice that Eq. (2.1) and (2.4) are not sufficient to obtain the displacement profile of the slope. We still need an equation that relates stress and strain that is precisely the constitutive equation of the problem:

$$\sigma=E\varepsilon \tag{2.5}$$

E is the Young's modulus that gives the material property of the soil. It can depend on the position for *non-homogenous* soil, or on the stress for *non-linear* materials. Now we can combine Eq. (2.1), (2.4) and (2.5) to obtain the *governing equation* of our problem:

$$\frac{d}{dx}\left(E\frac{du}{dx}\right)=-f(x) \tag{2.6}$$

If the soil behaviour is not lineal, (i.e. E does not depend on σ) this equation can be directly integrated to obtain the displacement along the landslide. We should not forget that every time we integrate we obtain an integration constant, which lead us to an indeterminate solution of our problem. In order to obtain a single solution we need to complete Eq. (2.6) with the so-called *boundary conditions*. They correspond to the condition of the unknown variable u at the boundary of the domain. Since our slope is constrained by a rock outcrop at the bottom, and free to move at the top, the boundary conditions are

$$u(0)=0 \quad \text{and} \quad \left.\frac{du}{dx}\right|_{x=L}=0 \tag{2.7}$$

where L is the length of the landslide. The first condition is called *essential* or *fixed boundary condition.* It states that the displacement at the bottom of the slope is always zero. The latter one is called *natural*, or *free boundary condition,* and it comes after using Eq. (2.1) and (2.5), and the fact that σ=0 at the top of the landslide. It is left as an exercise for the reader to show that if the soil is homogeneous and linear (E= cte) and the top boundary is at the critical state ( $\tau=\sigma_n \tan(\varphi)$ , where $\sigma_n=\gamma h\cos\alpha$ is the normal stress φ is the angle of friction of the soil), an *analytical solution* exists for Eq. (2.6) with boundary condition given by Eq. (2.7)

$$u(x)=\frac{\gamma\left(\sin(\alpha)-\cos(\alpha)\tan(\varphi)\right)}{2E}x(2L-x) \tag{2.8}$$

Note that to obtain this analytical solution we require several strong assumptions, such as one-dimensional deformation, linear elastic soil, and a sliding layer of zero thickness at the critical state. In practice we cannot always depend on too strong assumptions. If we relax the assumptions the resulting governing equation does not have analytical solution. That is where numerical solutions take place. Eq. (2.6) for non-linear material behaviour could in solved replacing the derivatives by finite differences. This leads to a set of algebraic equations that can be resolved numerically. This is the essence of the finite differences method that is useful for systems with simple domains. Yet several real-world problems involve complex domains and the finite different method become problem dependent. The boundary conditions are much simpler to plug in finite element modelling. Here is where the power of the finite element modelling appears, as it provides a unified framework for solving the governing equation of a wide range of problems for any kind of domains and boundary conditions. In the section 2.3 we present the key concepts of finite element modelling which will allow us to understand the general idea of this method.

## 2.2: Euler Bernoulli Beam Theory

The Euler-Bernoulli beam theory is a simplification of the linear theory of elasticity used to calculate the deflection produced by applied loads. As any theory, it has a certain number of simplifications:

(1) The loads are perpendicular only;
(2) The deflection are small; and
(3) Plane sections of the beam remain plane and perpendicular to the longitudinal axis.

Derivation of the bending equation of the Euler-Bernoulli theory will be presented here.

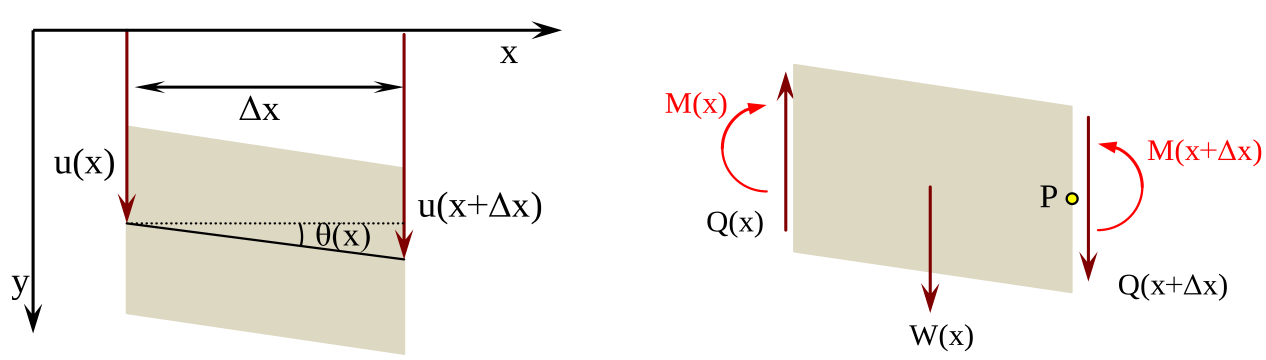


Figure 2.2 Kinematic Variables (left) and free body diagram (right) of the Euler-Bernoulli beam.

***Kinematic equations***

The rotation of the infinitesimal element is related to the deflection at their edges (Figure 2.2) by:

$$\theta.=\frac{u(x+\Delta x)-u(x)}{\Delta x}=\frac{du}{dx}$$

(2.9)

The derivation of this expression used the assumption that the rotations are small enough so that $\theta \approx \sin\theta \approx \tan\theta$ .

Curvature is defined as the inverse of the radius of curvature ρ of the beam (Figure 2.2). The exact calculation of curvature is obtained from differential calculus:

$$\kappa=\frac{1}{\rho}=\frac{\left|d^2u/dx^2\right|}{\left[1+(du/dx)^2\right]^{3/2}} \;,$$

Since θ=du/dx is assumed to be much smaller than one, the curvature can be approximated to:

$$\kappa \approx -\frac{d^2u}{dx^2}=-\frac{d\theta}{dx} \qquad (2.10)$$

Thus we obtain the kinematic relation between deflection and curvature

$$\kappa= -\frac{d^2u}{dx^2} \qquad (2.11)$$

***Balance equation***

The free body diagram of the infinitesimal element is shown in Figure 2.2. Q(x), M(x), W(x) represent the shear force, the moment, and the force per unit of length at point x. For this problem we need to use both balance of forces and balance of moments. By balancing the forces in the y-direction we get

$$Q(x) - Q(x+\Delta x) - W\,\Delta x =0$$

The above equation results into

$$\frac{dQ}{dx}= -W$$

(2.12)

Now we use the balance of angular momentum

$$-Q(x)dx+M(x+\Delta x)-M(x)+W\Delta x\frac{\Delta x}{2}=0$$

The last term vanishes since it is a second order infinitesimal, and the resulting equation is

$$\frac{dM}{dx}=Q \qquad (2.13)$$

Eq. (2.12) and Eq. (2.13) can be combined to obtain the balance equation of the bending problem

$$\frac{d^2M}{dx^2} = -W \tag{2.14}$$

***Constitutive relation***

This is the relationship that connects moments to curvature

$$M=C\kappa$$

Using elasticity theory, it is possible to show that the constant is the product between the Young modulus of the material E and the moment of inertia of the cross section area I. Therefore the constitutive relation can be written as:

$$M=E\,I\,\kappa \tag{2.15}$$

Finally, if we combine the constitutive equation with the kinematics and the balance equations we obtain the governing equation of the problem:

$$\frac{d^2}{dx^2}\left(EI\frac{d^2u}{dx^2}\right)=W$$

## 2.3: Weak formulation

We are about to introduce the weak formulation of the governing equations. In structural mechanics, this formulation is equivalent to the principle of virtual work. This principle plays a very vital role in structural analysis and in the finite element formulation of partial differential equations.
We want to solve the governing equation plus boundary conditions:

$$\frac{d}{dx}\left(E\frac{du}{dx}\right)=-f(x), \qquad u(0)=0, \qquad \left.\frac{du}{dx}\right|_{x=L}=0 \tag{2.16}$$

The solution above requires to have a second derivative, so that it need to be continuous and with no corners. We want to relax this assumption, and find solution that being continuous can have corners, i. e. discontinuities in the derivative. Let us define the *test function* $u^*(x)$, as continuous and piece-wise differentiable, satisfying the <u>essential</u> boundary conditions of the governing equation. The meaning of this test function may appear obscure at this point of the book, but it will be clarified when we arrive to the weak formulation. The equation above can be written as

$$\int_0^L\left[\frac{d}{dx}\left(E\frac{du}{dx}\right)+f(x)\right]u^*(x)dx=0 \tag{2.17}$$

Now we want to get rid of the second derivatives to allow continuous function with 'corners' to satisfy the new equation. With this aim we will "integrate by parts" the first term of the above equation. First we recall the 'product rule' of differential calculus

$$\frac{d}{dx}(vw)=\frac{dv}{dx}w+v\frac{dw}{dx}$$

Using $v=Edu/dx$ and $w=u^*$ we obtain the following identity

$$\frac{d}{dx}\left(E\frac{du}{dx}u^*\right)=\frac{d}{dx}\left(E\frac{du}{dx}\right)u^*+E\frac{du}{dx}\frac{du^*}{dx},$$

this is rewritten as

$$\frac{d}{dx}\left(E\frac{du}{dx}\right)u^*=\frac{d}{dx}\left(E\frac{du}{dx}u^*\right)-E\frac{du}{dx}\frac{du^*}{dx} \tag{2.18}$$

Replacing this equation into Eq. (2.17) we get

$$\int_0^L\left[\frac{d}{dx}\left(E\frac{du}{dx}u^*(x)\right)-E\frac{du}{dx}\frac{du^*}{dx}+f(x)u^*(x)\right]dx=0 \tag{2.19}$$

Integrating the first term

$$E\frac{du}{dx}u^*(x)\Big|_0^L-\int_0^L\left[E\frac{du}{dx}\frac{du^*}{dx}-f(x)u^*(x)\right]dx=0$$

and using the boundary condition given in Eq.(2.16) on u and $u^*$ we obtain the so-called *weak formulation* of the problem

$$\int_0^L\left[E\frac{du}{dx}\frac{du^*}{dx}-f(x)u^*(x)\right]dx=0 \tag{2.20}$$

The reader may ask, what does it mean? Why it is weak? Why is it important? It is called weak form because the conditions of the seeking solution u(x) are weaker than in the Eq. (2.16): In the weak form, our solution does not need to have continuous second derivative. We only require a solution that is continuous and differentiable, so that we can seek piece-wise linear solution. The weak form is also of great importance in structural mechanics because it corresponds to an important principle in mechanics: To show that, using Eq (2.1) and (2.5) we can write Eq(2.20) as

$$\int_0^L\left[\sigma\varepsilon^*-fu^*\right]dx=0 \tag{2.21}$$

The first term looks like the energy done on the system by internal forces after a virtual displacement $u^*(x)$ consistent to the essential boundary condition. The second term is the energy given by external forces due to this virtual displacement. We have found that the weak formulation corresponds to the well-known *principle of virtual work*. This principle states that the equilibrium solution of the system u(x) is such that the internal work equals the external virtual work for any displacement consistent with the boundary conditions.

## 2.4: Finite Difference Method

Until now we have introduced the strong form and the weak formulation of the governing equations. The strong form can be used to solve numerically the equation using the method of finite differences. The weak form is the basis of the finite element formulation.

In the "finite difference" method, a solution of the basic governing differential equations is sought at discrete points within the domain investigated. The domain is divided in rectangular grids and the derivatives are approximated by a finite difference

$$\frac{du}{dx} . = \frac{u(x+\Delta x) - u(x)}{\Delta x} \qquad (2.22)$$

The second derivative can be also approximated by a finite difference expression

$$\frac{d^2u}{dx^2} . = \frac{\left.\frac{du}{dx}\right|_{x} - \left.\frac{du}{dx}\right|_{x-\Delta x}}{\Delta x} \qquad (2.23)$$

Using both equations above, we obtain a finite difference expression for the second derivative

$$\frac{d^2u}{dx^2} . = \frac{u(x+\Delta x) - 2u(x) + u(x - \Delta x)}{\Delta x^2} \qquad (2.24)$$

We can replace the above equation into Eq. (2.6) to obtain

$$-u(x+\Delta x) + 2u(x) - u(x - \Delta x) = \frac{\Delta x^2 f(x)}{E(x)} \qquad (2.25)$$

Then the governing equations are converted into algebraical equations, which are completed using the boundary conditions. Thus a *pointwise* approximation is obtained. The beauty of this method is there is that the derivation of the algebraical equation is straightforward. Unfortunately, this feature often cannot outweigh its main disadvantage, namely that the method is not very tolerant of irregular boundary conditions as shown in Figure 2.3. The other problem is that the conversion of the boundary conditions into algebraical equations is not always easy and it needs special treatment in each case.

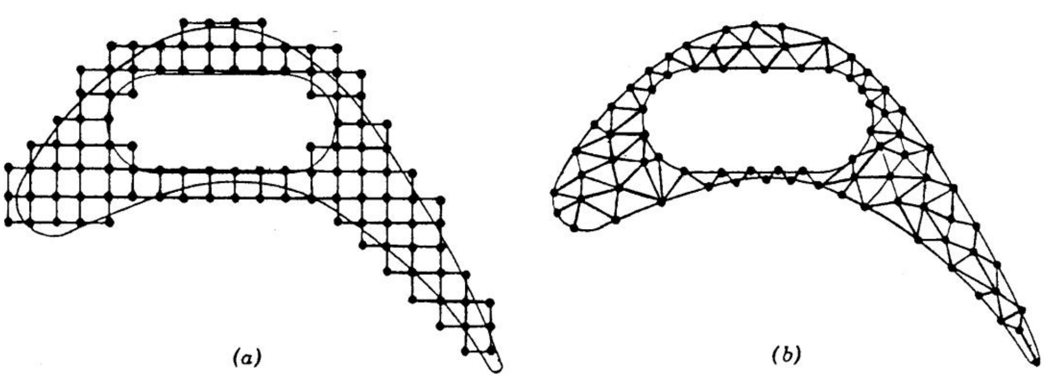



**Figure 2.3 Discretization of a turbine blade using (a) finite difference method and (b) finite element method [after Hubner, 1942]**

### 2.5: Finite Element Method

A more flexible technique in this respect is the "finite element method". Once again, the physical body or continuum is discretised into smaller sub-regions or into what are more commonly known as *finite elements*. A *shape function* is defined in each discrete element. Then we seek a solution as a linear combination of the shape function

$$u(x) = \sum_{i=1}^{n_{dof}} u_i N_i(x)$$

Where the sum goes over all "degrees of freedom" (dof) of the system. In other words, a *piecewise* approximation to the governing equation is arrived at, whose solution is obtained by finding the coefficients $u_i$. Very complex shapes can be modelled with relative ease (Figure 2.1b) using this method. Furthermore, by introducing intermediate points along element lines, it is even possible to ideally model curved boundaries. Next section introduces the basic concept of finite element method in one dimension.

We are ready to introduce the brilliant idea of Galerkin (Russian Mathematician and Engineer) to find an approximate solution of the weak form of the governing equations. We will introduce the Galerkin method by formulating the finite element method in one dimension. The basic procedure is essentially the same for two and three-dimensional problems:

1. Decompose the domain into a set finite elements;
2. Define a set of *shape function*s, each one sitting in what are called *nodes* of the finite elements;
3. Unknown field variable u(x) is expressed as a linear combination of the shape functions; and
4. The governing equation is transformed into a matrix equation that is solved to obtain coefficient of the linear combination.

**Domain Discretisation**

The domain of the slope problem is the interval [0,L]. Let us divide the interval into four *elements* ($e_1$, $e_2$, $e_3$, $e_4$). These elements will be joined by five *nodes* ($x_0$, $x_1$, $x_2$, $x_3$, $x_4$). We seek and approximate solutions at the nodes given by $u_i=u(x_i)$, i=0,1,..4. The natural question is how many element we need to use. The general rule is that as more elements we use more accurate will be the solution but more calculations need to be done. But in the practice we need to use smaller element in those part of the domain where we expect the solution will change more abruptly. In analysis of structures this happens near to the holes or the interfaces where different bodies interact.

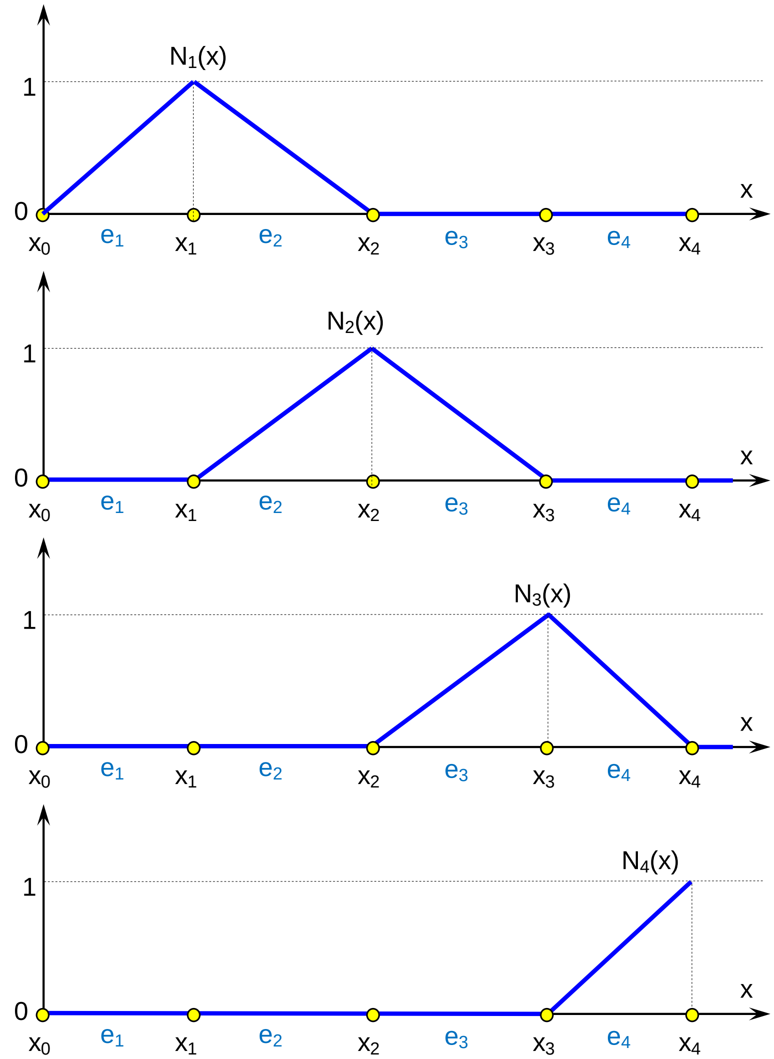



**Figure 2.4 Definition of hat shape function in finite element analysis**

## Global shape function

After discretisation we seek for a solution of the Eq. (2.20) on the domain. The main idea is to sit a *shape function* in each one of the nodes, (Figure 2.4) and then express the virtual displacement as a linear combination. Each shape function will account for deformation at one node, and the total virtual deformation is expressed as a linear combination of the shape functions. In particular, Eq. (2.20) will be valid for $u^*(x)=N_i(x)$ (i=1,2,3,4). Thus Eq. (2.20) is written as:

$$\int_0^L (E\frac{du}{dx}\frac{dN_i}{dx} - fN_i(x))dx=0 \quad \text{where } i=1,2,3,4 \tag{2.26}$$

## Linear combination

The function u(x) is expressed also as a combination of the shape functions:

$$u(x)=u_1N_1(x)+ u_2N_2(x)+u_3N_3(x)+ u_4N_4(x)=\sum_{i=1}^{4}u_iN_i(x) \tag{2.27}$$

If we use shape function as the *hat* shown in Figure 2.4, it is easy to show that $\delta_i$ is the deformation at the $i^{th}$-node.

**Global matrix equation**

Replacing Eq. (2.27) into Eq. (2.26) we obtain a *global matrix equation*

$$\sum_{i=1}^{4} K_{ij} u_j = F_i \tag{2.28}$$

$$K_{ij} = \int_0^L E \frac{dN_i}{dx}\frac{dN_j}{dx} dx \tag{2.29}$$

$$F_i = \int_0^L f(x)\, N_i(x) dx \tag{2.30}$$

The FEM solution consists of calculating the elements of the 'stiffness matrix' Eq. (2.29) and the vector $F_i$. . It is left as an exercise for the reader to show that if $E(x) = E_0$ and $f(x) = f_0$, the global matrix equation is given by

$$\frac{E_0}{\Delta x}\begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = f_0 \Delta x \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix} \quad \therefore\ \mathbf{K}.\mathbf{u}=\mathbf{F} \tag{2.31}$$

We notice that our smart selection of the shape function concentrated at the nodes allow us to obtain a banded matrix with zeros outside of the band. This simplifies the calculation of the inverse. The finite element programs have a *solver* that is in charge of inverting the stiffness matrix to find the solution at the nodes as

$$\mathbf{u}=\mathbf{K}^{-1}.\mathbf{F} \tag{2.32}$$

**Finite element solver**

Most of the computational work involved in Finite Element Analysis lies in the inversion of the stiffness matrix. The part of the program that does this inversion is called *solver*. The first steps that the solver needs to check is whether the determinant of the matrix is different from zero. If it vanished the matrix is singular, which means that it cannot be inverted. In other words, we do not have a unique solution of the problem, or we may not have any. Singular matrix appears when the boundary condition is not 'well posed'. This is the case for example, when the boundary conditions are free at both ends of the domain. Singular matrices appear also when the material properties of the materials such as Young modulus of thickness of the materials are entered with zero values. This is a common mistake of beginners.

The second problem that may be encounter by the solver in problems with large matrices is that the computer needs too much time to invert the matrix. This usually happen when the elements are not properly indexed, leading to sparse matrices. Ideally we want that the elements of the stiffness matrix vanish above a certain distance from the diagonal that is called bandwidth.

A typical finite element program consists of three basic units: *pre-processor*, *processor* and *post-processor*. In the pre-processor the geometry of the problem, the boundary conditions and material parameters are entered into the program. The processor generates the finite element mesh, assembles the stiffness matrix, and inverts it using the solver. The last component of the program is the post-processor that computes the solution and its derivatives and print or plot the results. In this book we will focus on the theoretical aspects of the implementation of the *finite elements* within the processor. We focus not only on structural problems, but also in non-structural cases such as seepage analysis and thermal conduction problems. We will study the so-called *static solvers* that give solutions of static problems. However, the reader shall bear in mind that there are solvers for many situation, such as buckling analysis and dynamics systems.

## 2.6: Finite Element for non-structural problems

The above example was related to a simple example of a one-dimensional structure. We will show that here that similar governing equations can be derived for other kind of situations beyond structural mechanics.
Let us start with the equation for the conduction of heat. Here the unknown variable is the temperature, which is given in terms of the position. The *kinematic equation* corresponds to the temperature gradient, for one-dimensional flow it is given by

$$\nabla T=\frac{dT}{dx} \tag{2.33}$$

The constitutive equation corresponds to the Fourier law, which states that the flow of heat is proportional to the gradient of temperature by a factor k that is the conductivity:

$$q=-k\nabla T \tag{2.34}$$

Then the balance equation correspond to the principle of conservation of mass, that the state that the heat generated in an infinitesimal element $f(x)\Delta xA$ equals to the heat that flow in the boundaries of the element $A(q(x+\Delta x)\ -q(x))$, written in differential form

$$\frac{dq}{dx}\ = f(x) \tag{2.35}$$

Putting all equation together we get the same equation as before

$$\frac{d}{dx}\left(k\frac{dT}{dx}\right)=-f(x) \tag{2.36}$$

Similar equations are derived for seepage flow by changed temperature by hydraulic head, and f(x) by the amount of water generated inside the elementary volume. In both case the fixed boundary condition correspond to a fixed heat/temperature at the extreme, and the free boundary condition correspond to impermeable/isolated boundaries.

## 2.7: Variational principle: minimal form

The principle of virtual work can be derived from a variational formulation. This formulation leads to a wide range of numerical methods to find equilibrium configuration of complex systems, such as the configuration of DNA molecules. One of these is the Finite Element Method that we have derived from the virtual work principle.

Here we present an alternative to derive the weak formulation which is based on energetic principles. This formulation is useful when we are interested in the equilibrium of the system, which is the case of most structural analysis problem. If we want to investigate the transient dynamics we need other methods. The definition define the 'energy' as a 'functional' (it means, a function whose argument is a function, and whose value is a real number, which in this case represents the energy.

$$E(u)=\int_0^L \left(\frac{1}{2}E\left[\frac{du}{dx}\right]^2 + fu\right)dx \tag{2.37}$$

We seek for the function u(x) that minimizes the energy. This can be done by using the 'variational derivative'.

$$E'(u). = \frac{E(u+\varepsilon u^*) - E(u)}{\varepsilon} = 0 \tag{2.38}$$

Where $u^*(x)$ is a 'test function' that satisfies the boundary condition of the problem.
Replacing Eq. (2.37) in Eq. (2.38) we obtain:

$$\int_0^L \left(E\frac{du}{dx}\frac{du^*}{dx} - fu^*\right)dx = 0 \tag{2.39}$$

This corresponds to the weak form. We can also derive the strong formulation for Eq. (2.39). By integrating this equation by parts,

$$0 = \int_0^L \left(E\frac{du}{dx}\frac{du^*}{dx} - fu^*\right)dx = -\int_0^L \left(E\frac{d^2u}{dx}u^* + fu^*\right)dx + u^*(x)\frac{du}{dx}\Big|_0^L$$

Using the boundary condition it leads to

$$\int_0^L \left[\frac{d}{dx}\left(E\frac{du}{dx}\right) + f(x)\right]u^*(x)dx = 0$$

Since this equation is valid for any virtual displacement we can assume that the integrand vanish in all points

$$\frac{d}{dx}\left(E\frac{du}{dx}\right) + f(x) = 0 \tag{2.40}$$

We can conclude that the governing equation of an engineering problem can be written in three different forms: the strong form, which is useful in finite differences method; the weak form which allows the finite element formulation; and the minimal form, which allow numerical solutions using variational methods.

### *Problem 1*

This question is related to the governing equation of the constrained landslide problem

$$E_o \frac{d^2u}{dx^2} = -f_o \quad u(0)=0 \quad \text{and} \quad \left.\frac{du}{dx}\right|_{x=L} = 0$$

Where $E_0$ is the Young modulus of the soils, $f_0$ is the external forces per unit of length, and u(x) is the displacement we want to obtain. Find the analytical solution of this equation. Here we will compare this result with the numerical solutions from the finite difference method and the finite element method. Divide the space domain of the landslide in four equally-spaced intervals with nodes

$$x_0=0,\ x_1=\Delta x,\ x_2=2\Delta x,\ \ldots, x_4=L,$$

Show that the finite different method (FDM) solution of the governing equation is given by

$$K = \frac{1}{\Delta x^2}\begin{bmatrix} -2 & 1 & 0 & 0 \\ 1 & -2 & 1 & 0 \\ 0 & 1 & -2 & 1 \\ 0 & 0 & 1 & -1 \end{bmatrix} \qquad \begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f_0 \Delta x^2}{E_0}\begin{bmatrix} 1 \\ 1 \\ 1 \\ 1 \end{bmatrix}$$

Solve this equation by inverting the matrix, and find the displacement at the nodes.

### *Problem 2*

For the differential equation in Question 2.1, Construct the global matrix equation using the finite element method (FEM). You have to do the following

1) Calculate the integrals for $K_{11}$, $K_{12}$, $K_{13}$, $K_{44}$, $F_1$, and $F_4$.
2) Using these calculations to show that the matrix equation is given by

$$\begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f_0 \Delta x^2}{E_0}\begin{bmatrix} 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix}$$

Invert the matrix to solve the displacement of the nodes.

### *Problem 3*

Compare the numerical solutions of both FDM and FEM with the analytical solution. What is the numerical error of the solution in each case? How does the numerical error change if the number of elements is duplicated?

Hint: to compare the numerical solutions, you can work with dimensionless variables by assuming that

$f_0 L^2/E_0 = 1$ and $L=1$

### *Problem 4*

A soil layer of depth H over a rock bed has a uniform unit weight γ and a Young modulus E(x) that varies linearly with depth as shown in Figure 1 center. A uniform load P is applied on the surface. The soil deforms due to the combined action of its weight and the surface load. The deformation due to the surface load is called settlement.

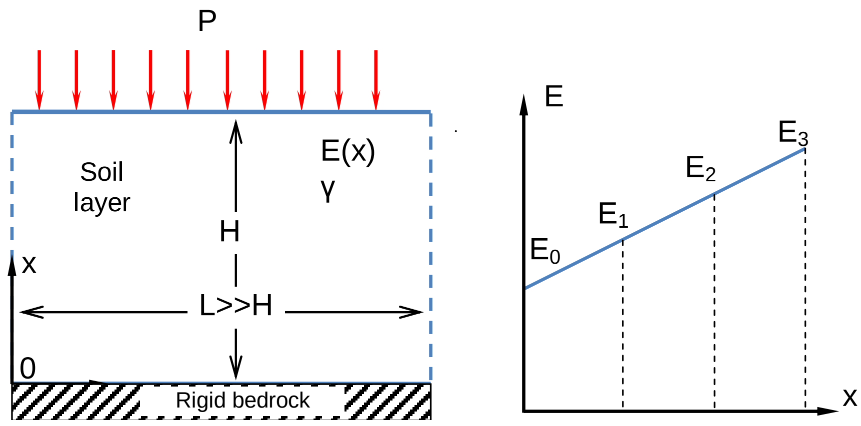



**Figure 1**

1) Derive the strong form and the weak form of the governing equations.

2) Plot suitable global shape function ($\mathbf{N}_i(x)$) for a finite element analysis with three linear elements. Then derive global matrix equation **KU=F** for the soil deformation in terms of H, $\gamma$, P, $E_0$, $E_1$, $E_2$, and $E_3$.

Hint: Check that in the case when , $E_0=E_1= E_2= E_3$ your stiffness matrix becomes

$$\frac{E_0}{\Delta x}\begin{bmatrix} 2 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

3) Derive the analytical solution for the settlement and compare with the numerical solution of step 2.

Hint: write the Young Modulus as

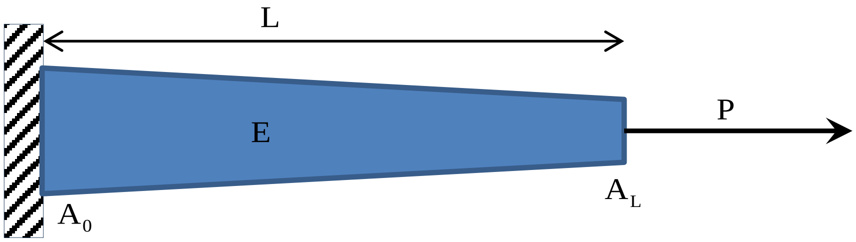
$$E(x)=E_0+ax$$

After deriving the displacement of the soil, you should take $\gamma=0$ to obtain the settlement:

$$S(x)=\frac{P}{a}\ln(1+\frac{ax}{E_0})$$

To compare the numerical and analytical solutions, assume that the Young modulus at the bottom of the soil layer is 30% larger than the one at the top.

### *Problem 5*




A steel bar (E=200 GPa) is fixed to a wall as shown the figure above. The bar is pulled by a horizontal force P=1000 N applied at the right. The area changes from 0.01 $m^2$ to 0.0064 $m^2$. The length of the bar is 1m. This problem is about finding the horizontal displacement along the bar. It is assumed that displacements in the right direction are positive.

1) Derive the governing equations of the deformation of the bar.

2) Derive the weak form of the governing equations.

3) Find the global matrix equation with three linear elements.

4) Derive the analytical solution for the deformation of the bar.

# Chapter 3

# FINITE ELEMENT CONCEPT

In the previous chapter the basic concept of the finite element formulation was introduced, and the stiffness matrix was derived using global shape functions. Although the stiffness matrices of a few more element types may be obtained using similar procedures, for other types of finite elements, such as continuum triangular or rectangular elements, the derivation is not straightforward. Therefore it is necessary to develop a general procedure that can be used for derivation of the stiffness matrices of all element types. The general method consists of constructing the stiffness matrix of individual elements, and then assembly them into a global stiffness matrix of the complete structure.

The aim of this chapter is to introduce this general formulation of the finite element method. The procedure will used to form the stiffness matrix of two different element types, bar element and a flexural beams element.

## 3.1: The Principle of virtual work

We recall the principle of virtual work for a single element (bar or beam) of the structure. The principle of virtual work states that during any virtual displacement imposed on the boundary of an element, the total work done by the external loads $W_{ext}$ must be equal to the total internal work done $W_{int}$ by the internal stresses $\sigma(x)$.

$$W_{int} = W_{ext}$$

$$W_{int} = \int_{V_e} \varepsilon^*(x)\sigma(x)dV$$

$$W_{ext} = \int_{V_e} u^*(x)f(x)dV$$

where f(x) are the external load, and $\boldsymbol{\varepsilon}^*(x)$ is the virtual strains produced by the virtual displacement $u^*(x)$. The integral goes over the volume of the element $V_e = AL$, where A is the cross section area and L the length of the element.

## 3.2: General Procedure in Finite Element Analysis

Most finite element computations in numerical analysis comprise the following steps:

1. Chose a suitable coordinate system. While for many of the geometries a Cartesian coordinate is suitable, a cylindrical coordinate system may be used for problems with axial symmetry.
2. Divide the geometry of the problem into a number of finite elements. Different types of elements may be used to represent differences in physical properties. In structural mechanics, these can be beams, cables, plates, bricks, etc.
3. Use a suitable node numbering system for the elements of the structure.

4. Derive the stiffness equations for all finite elements using the principle of virtual work (or the principle of minimum potential energy). These equations are typically in the form of:

$$\mathbf{k}^e \, . \, \mathbf{u}^e = \mathbf{f}^e$$

where $\mathbf{k^e}$ is the element stiffness matrix, $\mathbf{u^e}$ is the vector of element nodal displacements, and $\mathbf{f^e}$ is the vector of element nodal forces.

5. Assemble the global stiffness matrix for the complete structure from the stiffness matrices of the individual finite elements and assemble the global force vector to form the global stiffness equations:

$$\mathbf{K.u = F}$$

where $\mathbf{K}=\Sigma_e\mathbf{K^e}$ is the global stiffness matrix, $\mathbf{u}$ is the vector of global nodal displacements, and $\mathbf{F}=\Sigma_e\mathbf{F^e}$ is the vector of global nodal forces.

6. Apply boundary conditions by eliminating equations related to nodes with zero displacements.
7. Solve the global stiffness equations to obtain the unknown nodal displacements:

$$\mathbf{u} = \mathbf{K}^{-1} . \mathbf{F}$$

8. Compute the relevant physical quantities in all elements: stresses, strains, curvature and moments.

The Calculation of the element stiffness matrix, $\mathbf{k^e}$, is an important step in the finite element computations and therefore is dealt with in detail in the next section.

## 3.3: Calculation of Element Stiffness Matrix

A general procedure is presented here that can be used for derivation of the stiffness matrix of various finite elements. The aim is to relate the *nodal loads* to the *nodal displacements*, and thereby define the element stiffness matrix.

Different types of elements have different numbers of *nodes* and different numbers of *degrees of freedom* per node and therefore the size of the stiffness matrix is generally different for different element types. In most structural analyses the term *degree of freedom* may be regarded as the different modes of displacement at each node. However, in general, the term "degree-of-freedom" is applied to any nodal quantity such as displacement, curvature, temperature, hydraulic head, etc. If the number of nodes in the chosen finite element is $n_{ne}$ and the number of degree of freedom per node is $d_{of}$, then the total degrees of freedom for the element is $n_{dof} = n_{ne} \times d_{of}$. The size of the element displacement vector, $\mathbf{u^e}$, and the element force vector, $\mathbf{f^e}$, is equal to $n_{dof}$ and the size of the element stiffness matrix, $\mathbf{k^e}$, is equal to $n_{dof} \times n_{dof}$. The element stiffness equations are defined by:

$$\mathbf{k}^e . \mathbf{u}^e = \mathbf{f}^e \tag{3.1}$$

The specific case considered here is a two-node bar element shown in Figure (3.1) . Similar to the element of the constrained landslide, we assume that this element can only carry axial loads. The rotation and the deflection normal to the element axis are assumed to be zero. For this element $n_{ne}$=2, $d_{of}$=1, $n_{dof}$=2, and therefore the size of the stiffness matrix is 2×2.

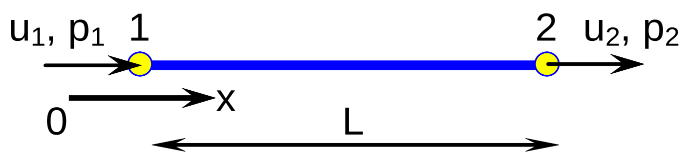



**Figure 3.1 Two-node bar element**

Following are the steps required for calculation of the element stiffness matrix, $\mathbf{k^e}$.

1. Chose local coordinate and node numbering systems that best suit the shape of the element. Define the load and displacement associated with each degree-of-freedom at each node.

For the bar element in Figure (3.1) The local coordinate system is along the axis of the element. Each node has a displacement, u, and an associated force, p, measured along the direction of the bar. Therefore the stiffness equations for the element can be presented as:

$$\begin{bmatrix} p_1 \\ p_2 \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \tag{3.2}$$

2. Select a suitable displacement function that uniquely defines the state of displacements at all points within the element.

The aim is to express the variation of displacements at any point within the element, u(**x**), in terms of the nodal displacements, $\mathbf{u^e}$. In many cases the variation of displacements can be approximated with sufficient accuracy by a polynomial function. The assumed polynomial function must contain one unknown coefficient for each independent degree of freedom that exists at the nodal points:

$$\begin{aligned} u(\mathbf{x}) &= a_1 f_1(\mathbf{x}) + a_2 f_2(\mathbf{x}) \ldots + a_{n_{dof}} f_{n_{dof}}(\mathbf{x}) = \mathbf{f}^T(\mathbf{x}) \,.\, \mathbf{a} \\ \mathbf{f}^T(x) &= \begin{bmatrix} f_1(x) & f_2(x) & \ldots & f_{n_{dof}}(x) \end{bmatrix} \\ \mathbf{a} &= \begin{bmatrix} a_1 & a_2 & \ldots & a_{n_{dof}} \end{bmatrix} \end{aligned} \tag{3.3}$$

where **a** is the vector of unknown coefficients of the polynomial function f(**x**).

For the specific case of the bar element, the total number of degrees of freedom is 2. Therefore the polynomial function representing the variation of displacements must have 2 unknown coefficients and it is a function of x only:

$$u(x) = a_1 + a_2 \,.\, x = \begin{bmatrix} 1 & x \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} = \mathbf{f}^T(x) \,.\, \mathbf{a}$$

where $f(x) = [1 \;\; x]^T$ and $\mathbf{a} = [a_1 \; a_2]^T$.

3. Relate displacements within the element to the nodal displacements.

The function u(**x**) represents the displacements at any point and therefore it is also valid at nodal points. The displacements at the nodes, $\mathbf{u^e}$, can be simply obtained by substituting the nodal coordinates into Eq. (3.3). For example for node 1, with coordinates of $x_1$ and $y_1$, Eq. (3.3) becomes:

$$\mathbf{u}_1^e = u(\mathbf{x}_1) = \mathbf{f}^T(\mathbf{x}_1)\ .\ \mathbf{a}$$

If this procedure is followed for all other nodes attached to an element, Eq. (3.3) becomes:

$$\mathrm{u}^e = \begin{bmatrix} \mathrm{u}_1^e \\ \mathrm{u}_2^e \\ \vdots \\ \mathrm{u}_{n_{dof}}^e \end{bmatrix} = \begin{bmatrix} \mathbf{f}^T(\mathbf{x}_1)\mathbf{a} \\ \mathbf{f}^T(\mathbf{x}_2)\mathbf{a} \\ \vdots \\ \mathbf{f}^T(x_{n_{dof}})\mathbf{a} \end{bmatrix} = \begin{bmatrix} f_1(\mathbf{x}_1) & f_2(\mathbf{x}_1) & \cdots & f_{n_{dof}}(\mathbf{x}_1) \\ f_1(\mathbf{x}_2) & f_2(\mathbf{x}_2) & \cdots & f_{n_{dof}}(\mathbf{x}_2) \\ \vdots & \vdots & \vdots & \vdots \\ f_1(\mathbf{x}_{n_{dof}}) & f_2(\mathbf{x}_{n_{dof}}) & \cdots & f_{n_{dof}}(\mathbf{x}_{n_{dof}}) \end{bmatrix} \begin{bmatrix} \mathrm{a}_1 \\ \mathrm{a}_2 \\ \vdots \\ \mathrm{a}_{n_{dof}} \end{bmatrix} \tag{3.4}$$

or

$$\mathbf{u}^e = \mathbf{C}.\mathbf{a}$$

The unknown coefficients, **a**, can now be determined from Eq.(3.4):

$$\mathbf{a} = \mathbf{C}^{-1}.\mathbf{u}^e \tag{3.5}$$

The polynomial coefficients **a** can now be substituted into Eq. (3.3) to form:

$$u(\mathbf{x}) = \mathbf{N}(\mathbf{x})\ .\ \mathbf{u}^e \text{ with } \mathbf{N}(\mathbf{x}) = \mathbf{f}^T(\mathbf{x})\ .\ \mathbf{C}^{-1} \tag{3.6}$$

Eq. (3.6) relates the displacements at any point within the element, u(**x**), to the nodal displacements, $\mathbf{u^e}$. In general, the terms in the expression $\mathbf{N}(\mathbf{x}) = \mathbf{f}^T(\mathbf{x}).\mathbf{C}^{-1}$, called *local shape functions* in the finite element method, present a means of interpolation within the element through which any quantity within the element can be calculated from its values at nodal points. Details of the derivation of the shape functions for different element types will be given later.

For the specific case of the bar element, $\mathbf{C}^{-1}$ can be calculated and substituted in Eq (3.6).

$$\mathbf{C} = \begin{bmatrix} 1 & x_1 \\ 1 & x_2 \end{bmatrix} \qquad \mathbf{C}^{-1} = \frac{1}{x_2 - x_1} \begin{bmatrix} x_2 & -x_1 \\ -1 & 1 \end{bmatrix}$$

$$N(x)=\mathbf{f}^T(x).\mathbf{C}^{-1}=\begin{bmatrix}1 & x\end{bmatrix}\begin{bmatrix}\frac{x_2}{x_2-x_1} & \frac{-x_1}{x_2-x_1}\\ -\frac{1}{x_2-x_1} & \frac{1}{x_2-x_1}\end{bmatrix}=\begin{bmatrix}\frac{x_2-x}{x_2-x_1} & \frac{x-x_1}{x_2-x_1}\end{bmatrix}=\begin{bmatrix}N_1(x) & N_2(x)\end{bmatrix}$$

where $N_1(x)=\dfrac{x_2-x}{x_2-x_1}$ and $N_2(x)=\dfrac{x-x_1}{x_2-x_1}$ are called the shape functions.

The shape functions depend only on the geometry of the nodal points and the type of the interpolation function used. The shape functions $N_1(x)$ and $N_2(x)$ vary linearly between $x_1$ and $x_2$ as shown in Figure 3.2. Note that the value of the shape function $N_1(x)$ is 1 at point 1 and zero at point 2. Similarly the value of the shape function $N_2(x)$ is 1 at point 2 and zero at point 1.

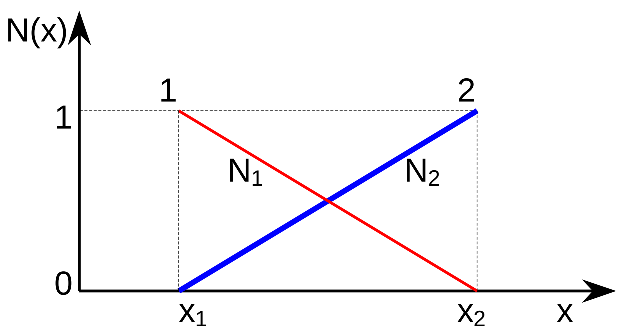



**Figure 3.2 Linear shape functions**

Therefore the displacement at any point within the element at (x) can be found using the following equation:

$$u(x)=N_1(x)\,u_1^e+N_2(x)u_2^e$$

4. Relate the strains within the element to the nodal displacements

The strains $\varepsilon(\mathbf{x})$ at any point within the element can be related to the displacements at the point, $u(\mathbf{x})$, and hence to the nodal displacements, $\mathbf{u}^e$. The strains can be expressed in the form of differentials of the displacements. The exact form of the differentials depends on the type of the element and may be obtained from the theory of elasticity. Details of the exact form of the strains for different types of continuum elements will be given later. In general the strains can be defined as:

$$\boldsymbol{\varepsilon}(\mathbf{x}) = \mathbf{L}\left[u(\mathbf{x})\right] \quad (3.7)$$

Where $\mathbf{L}$ is a differential operator, which depends on the problem we are analyzing. In particular, $\mathbf{L}=d/dx$ for bars and $\mathbf{L}=-d^2/dx^2$ for beams. Substituting Eq. (3.6) into Eq. (3.7) results in:

$$\varepsilon(x) = \mathbf{B}.\mathbf{u}^e \qquad B=\mathbf{L}\left[\mathbf{N}(\mathbf{x})\right], \tag{3.8}$$

Where **B** is generally a function of **x**. Eq. (3.8) represents the relationship between strains at any point within the element to the nodal displacements.

For the specific case of the bar element the relation between the strain and displacement is given by

$$\varepsilon = \frac{du}{dx}$$

Therefore $\mathbf{L} = \frac{d}{dx}$. **B** and **ε** can be calculated as follows.

$$B = \frac{dN}{dx} = \begin{bmatrix} \frac{dN_1}{dx} & \frac{dN_2}{dx} \end{bmatrix} = \begin{bmatrix} -\frac{1}{x_2 - x_1} & \frac{1}{x_2 - x_1} \end{bmatrix}$$

$$\boldsymbol{\varepsilon}(x) = \mathbf{B}.\mathbf{u}^e = \frac{u_2 - u_1}{x_2 - x_1}$$

It can be seen that the axial strain in the bar element is independent of the coordinates, i.e., it is constant all through the element.

5. Relate the stresses within the element to strains and to the nodal displacements

The stresses **σ**(**x**) at any point within the element can be related to the strains at the point, **ε**(**x**), and hence to the nodal displacements, $\mathbf{u}^{\mathbf{e}}$. The relationship between the strains and stresses can be expressed by the elastic properties of the element.

$$\boldsymbol{\sigma}(\mathbf{x}) = \mathbf{D}.\,\boldsymbol{\varepsilon}(\mathbf{x}) \tag{3.9}$$

Where D is the stress-strain matrix and contains the elastic properties of the element, such as Young’s modulus E and Poisson’s ratio ν, assuming that the element deforms elastically under loads. Details of matrix **D** for different types of continuum elements will be given later. Substituting Eq. (3.8) into Eq. (3.9), results in a relationship between the stresses at any point within the element to the nodal displacements.

$$\boldsymbol{\sigma}(\mathbf{x}) = \mathbf{D}.\,\mathbf{B}.\,\mathbf{u}^e \tag{3.10}$$

For the specific case of the bar element the axial normal stress can be related to the axial normal strain as:

$$\sigma(x) = E\,\varepsilon(x) = \frac{E\,(u_2 - u_1)}{x_2 - x_1}$$

The axial stress is therefore constant for the bar element.

6. Relating the internal stresses to the nodal loads.

The internal stresses can now be related to the loads at the nodal points. In this way the nodal displacements will be related to the nodal loads, since the stresses at each point within the element are related to the nodal displacements. The relationship between the nodal loads and the nodal displacement constitutes the stiffness matrix of the element.

Here the principle of virtual work will be used to determine a set of nodal forces that are statically equivalent to the sum of the stresses within the element. If an arbitrary set of virtual nodal displacements, $\mathbf{u*}^{\mathbf{e}}$, is imposed on the element where the actual nodal forces $\mathbf{f}^{\mathbf{e}}$ are applied, it causes virtual strains $\boldsymbol{\varepsilon}*(\mathbf{x})$ at a point within the element where the actual stresses are $\boldsymbol{\sigma}(\mathbf{x})$. The virtual displacement are related to virtual nodes by

$$u*(\mathbf{x}) = \mathbf{N}\,\mathbf{u*}^{\mathbf{e}} = \mathbf{u*}^{\mathbf{eT}}\,\mathbf{N}^{T}$$

The external virtual work done by the nodal loads, $W_{ext}$, is given by:

$$W_{ext} = \int_{V_e} u^*(\mathbf{x}) f(\mathbf{x}) dV = \int_{V_e} \mathbf{u}^{eT*} \mathbf{N}^T(x) f(\mathbf{x}) dV = \mathbf{u}*^{eT} \left( \int_{V_e} \mathbf{N}^T(x) f(\mathbf{x}) dV \right) \tag{3.11}$$

That we can write as:

$$W_{ext} = \mathbf{u}*^{eT} .\, \mathbf{f}^{e} \tag{3.12}$$

Where the nodal forces of the element is given by:

$$\mathbf{f}^{e} = \int_{V_e} \mathbf{N}^T(x) f(\mathbf{x}) dV \tag{3.13}$$

Now we calculate the work done by the internal stresses. The virtual strains can be related to the nodal virtual displacements using Eq. (3.8)

$$\boldsymbol{\varepsilon}*(\mathbf{x}) = \mathbf{B}\,\mathbf{u}*^{e}$$

$$\boldsymbol{\varepsilon}^{*\,T}(\mathbf{x}) = \mathbf{u}^{*eT}\,\mathbf{B}^T \tag{3.14}$$

The total internal work can be as

$$W_{int} = \int_{V_e} \boldsymbol{\varepsilon}^{*T}(\mathbf{x})\boldsymbol{\sigma}(\mathbf{x})dV = \int_{V_e} \mathbf{u}^{*T}\mathbf{B}^T(\mathbf{x})\mathbf{D}\mathbf{B}(\mathbf{x})\mathbf{u}dV = \mathbf{u}^{*T}\left(\int_{V_e} \mathbf{B}^T(\mathbf{x})\mathbf{D}\mathbf{B}(\mathbf{x})dV\right)\mathbf{u} \tag{3.15}$$

That can be written as

$$W_{int} = \mathbf{u}^{*eT}\,\mathbf{k}^e\,\mathbf{u} \tag{3.16}$$

where

$$\mathbf{k}^e = \int_{V_e} \mathbf{B}^T(\mathbf{x})\mathbf{D}\mathbf{B}(\mathbf{x})dV \tag{3.17}$$

Based on the principle of virtual work the total internal virtual work given by Eq (3.15) , must be equal to the total external virtual work, Eq. (3.11), i.e.,

$$\mathbf{u}^{*eT}\,(\mathbf{k}^e\,\mathbf{u} - \mathbf{f}^e) = 0 \tag{3.18}$$

Eq. (3.18) is valid for any set of virtual displacements. Therefore the vector of virtual displacements, $\mathbf{u}^{*e}$, can be removed from both sides of Eq. (3.18). (The arbitrary set of virtual displacements may also be assumed to have unit values so that $\mathbf{u}^{*e}$ can be removed from the equation. Thus Eq. (3.18) gives:

$$\mathbf{k}^e.\mathbf{u}^e = \mathbf{f}^e \tag{3.19}$$

Therefore, to calculate the element stiffness matrix, the strain-displacement matrix, **B**, and the stress-strain matrix, **D**, must be evaluated and then the matrix multiplication and integration must be performed.

For the specific case of the bar element the stiffness matrix can be calculated as follows.

$$B = \begin{bmatrix} -1/L & 1/L \end{bmatrix}$$

$$D = E$$

$$dV = A\,.\,dx$$

$$k^e = \int_V B^T.D.B.dV = A\int_0^L B^T.E.B.dx = A.E.B^T.B\int_0^L dx = A.E.L.B^T.B$$

$$k^e = AEL\begin{bmatrix} -1/L \\ 1/L \end{bmatrix}\begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} = AEL\begin{bmatrix} 1/L^2 & -1/L^2 \\ -1/L^2 & 1/L^2 \end{bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

The above stiffness matrix for a bar element, obtained from the method presented in this chapter, is identical to the one given by other method from structural analysis.

The load vector of Equation 3.13 can be calculated assuming that f(x) is constant. The direct integration leads to

$$\mathbf{F}^e = \begin{bmatrix} \int_{x_1}^{x_2} N_1(x) f(x) A dx \\ \int_{x_1}^{x_2} N_2(x) f(x) A dx \end{bmatrix} = \frac{Af\Delta x}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \tag{3.20}$$

Thus Equation (3.20) becomes

$$\frac{E}{\Delta x}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \frac{f\Delta x}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \tag{3.21}$$

This is the element matrix equation of the one dimensional bar problem.

## 3.4: Calculation of the Stiffness Matrix of a two-dimensional bar element

The aim of this section is to present an approach to the construction of the element stiffness matrices of the elements of two-dimensional structures through transformation of coordinates. A structural frame usually consists of members set at various angles to one another. Therefore it is more convenient to set up the stiffness matrix in terms of the local member coordinates and then transform each of the local coordinate system to the global coordinate system adopted for the complete structure.

A two-dimensional bar element which is inclined at an angle θ to the global system is shown in Figure 3.3. Axes X and Y refer to the local member system and axes x and y to the global coordinate system. In a framed structure each end of the bar could be displaced in both directions. The displacements U and V, u and v, and the forces P and Q, p and q are related to the local and the global systems, as shown in Figure 3.3.

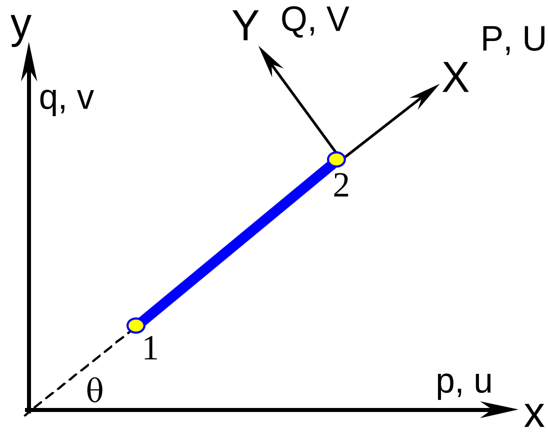



**Figure 3.3 Two-dimensional bar element**

We start with:

$$\frac{AE_a}{L}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}U_1\\ U_2\end{bmatrix}=\begin{bmatrix}P_1\\ P_2\end{bmatrix} \qquad \frac{AE_l}{L}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}V_1\\ V_2\end{bmatrix}=\begin{bmatrix}Q_1\\ Q_2\end{bmatrix}$$

where $E_a$ and $E_l$ are the axial and lateral Young modulus. In the special case of a truss-element, $E_l$=0 that reflects the fact that displacement of the nodes does not lead to shear forces. More precisely, the nodal forces are always parallel to the bar element so that $Q_1$=$Q_2$=0

We expand the matrices

$$\frac{AE_a}{L}\begin{bmatrix}1 & 0 & -1 & 0\\ 0 & 0 & 0 & 0\\ -1 & 0 & 1 & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{bmatrix}U_1\\ V_1\\ U_2\\ V_2\end{bmatrix}=\begin{bmatrix}P_1\\ 0\\ P_2\\ 0\end{bmatrix} \qquad \frac{AE_l}{L}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 1 & 0 & -1\\ 0 & 0 & 0 & 0\\ 0 & -1 & 0 & 1\end{bmatrix}\begin{bmatrix}U_1\\ V_1\\ U_2\\ V_2\end{bmatrix}=\begin{bmatrix}0\\ Q_1\\ 0\\ Q_2\end{bmatrix}$$

And then we sum both equations

$$\frac{A}{L}\begin{bmatrix}E_a & 0 & -E_a & 0\\ 0 & E_l & 0 & -E_l\\ -E_a & 0 & E_a & 0\\ 0 & -E_l & 0 & E_l\end{bmatrix}\begin{bmatrix}U_1\\ V_1\\ U_2\\ V_2\end{bmatrix}=\begin{bmatrix}P_1\\ Q_1\\ P_2\\ Q_2\end{bmatrix}$$

For the special case of a truss element $E_a$=E and $E_l$=0, so that the equation above reduces to

$$\frac{AE}{L}\begin{bmatrix}1 & 0 & -1 & 0\\ 0 & 0 & 0 & 0\\ -1 & 0 & 1 & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{bmatrix}U_1\\ V_1\\ U_2\\ V_2\end{bmatrix}=\begin{bmatrix}P_1\\ Q_1\\ P_2\\ Q_2\end{bmatrix} \qquad (3.22)$$

The local and global systems of forces at each node can be related by Eq. (B.3) in Appendix B:

$$\begin{bmatrix} P \\ Q \end{bmatrix} = \begin{bmatrix} +\cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{bmatrix} p \\ q \end{bmatrix} \tag{3.23}$$

Thus the relationship between the applied forces in the local and global systems is:

$$\begin{bmatrix} P_1 \\ Q_1 \\ P_2 \\ Q_2 \end{bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 & 0 \\ -\sin(\theta) & \cos(\theta) & 0 & 0 \\ 0 & 0 & \cos(\theta) & \sin(\theta) \\ 0 & 0 & -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \end{bmatrix}, \tag{3.24}$$

or simply:

$$\mathbf{F} = \mathbf{T}^{T}.\mathbf{f} \tag{3.25}$$

where F and f are the force vectors in the local and global systems, respectively.
A similar relationship also exists between the two sets of displacements in the local and global systems:

$$\mathbf{\Delta} = \mathbf{T}^{T}.\mathbf{u} \tag{3.26}$$

**Δ** and **u** are the displacement vectors in the local and global systems
The stiffness matrix for a member in the global system can now be established. The basic force-displacement relationship for the bar element, given in Eq.(3.22), states that:

$$\mathbf{F} = \mathbf{K}^{e}\,\mathbf{\Delta} \tag{3.27}$$

$K^e$ refers to the element stiffness matrix in the local coordinate system. Substituting F and **Δ** from Eq. (3.25) and Eq. (3.26) into Eq. (3.27) results in:

$$\mathbf{T}^{T}\,\mathbf{f} = \mathbf{K}^{e}\,\mathbf{T}^{T}\,\mathbf{u} \tag{3.28}$$

Both sides of the above equation are multiplied by T.

$$\mathbf{T}.\mathbf{T}^{T}.\mathbf{f} = \mathbf{T}.\mathbf{K}^{e}.\mathbf{T}^{T}.\mathbf{u} \tag{3.29}$$

One useful property of the T matrix is that its transpose is equal to its inverse, i.e,

$$\mathbf{T}^{T} = \mathbf{T}^{-1}, \quad \mathbf{T}\ \mathbf{T}^{T} = \mathbf{T}\ \mathbf{T}^{-1} = \mathbf{1} \tag{3.30}$$

Therefore;

$$\mathbf{f} = \mathbf{T}.\mathbf{K}^e.\mathbf{T}^T.\mathbf{u} = \mathbf{k}^e.\mathbf{u} \tag{3.31}$$

Whereby $k^e$ is the stiffness matrix of the element in the global system.

$$\mathbf{k}^e = \mathbf{T}.\mathbf{K}^e.\mathbf{T}^T \tag{3.32}$$

It can be seen that the global stiffness matrix for a member, $k^e$, can be obtained from the stiffness matrix of the member in the local member coordinate system. So that the stiffness matrix of the bar elements can be written in the global system as shown below.

$$\mathbf{k}^e = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \qquad c=\cos(\theta) \quad s=\sin(\theta). \tag{3.33}$$

Eq. (3.22) can now be written in the global system:

$$\frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} = \begin{bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \end{bmatrix} \tag{3.34}$$

In the assembly of the global stiffness matrix for a structure, an important point is that the stiffness matrix of any member, established in local coordinates, must be transformed into the global coordinate system before commencing the assembly process.

## 3.5: Calculation of the Stiffness Matrix of Flexural Beam Elements

The procedure explained in Section 4 is employed here to calculate the stiffness matrix of a flexural beam element. Beam elements are the basic members of rigid jointed frames.

The beam element considered here has two nodes, a uniform cross-section A, and is loaded by forces and moments at each node as shown in Figure 3.4. The beam is assumed to be slender so that the effects of shear deformations can be ignored. The effects of axial forces and deformations are also ignored here. The sign conventions for the moments and the shear forces are shown in Figure 3.4.

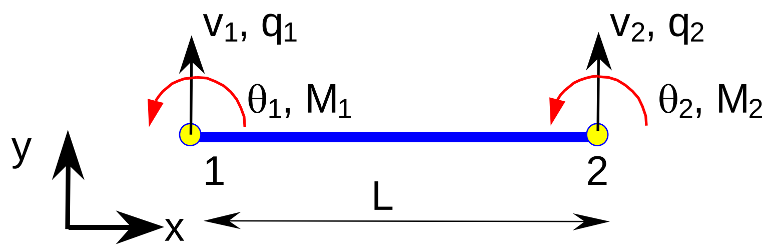



**Figure 3.4 Two-node beam element**

1. Local coordinate and node numbering system

The node numbering and coordinate system shown in Figure 3.4 may be used for the element where the y-axis is normal to the axis of the beam. The number of nodes is $n_{ne} = 2$, the number of degrees of freedom per node is $d_{of} = 2$, that is a deflection normal to the beam axis, v, and a rotation about the z-axis, θ. Therefore the total number of degrees of freedom for the element is $n_{dof}=n_{ne}\times d_{of} = 4$. The nodal forces associated with the rotation and deflection of the beam at each node are a moment about the z-axis, M, and a shear force in the y-direction, q. The size of the displacement vector, $\mathbf{u}^e$, and the element force vector, $\mathbf{f}^e$, is 4 and the size of the element stiffness matrix, $\mathbf{k}^e$, is 4×4.

$$\begin{bmatrix} q_1 \\ M_1 \\ q_2 \\ M_2 \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix}, \tag{3.35}$$

2. Displacement function

The variation of the transverse displacement can be approximated by a polynomial function. The polynomial function, w, must contain one unknown coefficient for each degree of freedom:

$$v(x) = a_1 + a_2 x + a_3 x^2 + a_4 x^3$$

$$v(x) = [1 \;\; x \;\; x^2 \;\; x^3]\,[a_1 \;\; a_2 \;\; a_3 \;\; a_4]^T \tag{3.36}$$

Where $a_1$ to $a_4$ are the unknown coefficients. The rotation at any point can be expressed as θ=dv/dx, thus:

$$\theta(x) = dv/dx = a_2 + 2a_3 x + 3a_4 x^2$$

$$\theta(x) = [0 \;\; 1 \;\; 2x \;\; 3x^2][a_1 \;\; a_2 \;\; a_3 \;\; a_4]^T \tag{3.37}$$

Therefore the "displacements" at any point along the beam can be obtained from Eq. (3.36) and Eq. (3.37) as:

$$\begin{bmatrix} v \\ \theta \end{bmatrix} = \begin{bmatrix} 1 & x & x^2 & x^3 \\ 0 & 1 & 2x & 3x^2 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \end{bmatrix} \tag{3.38}$$

The matrix f(x) and the vector **a** can be defined for the beam element by comparing Eq. (3.38) with Eq. (3.3):

$$\mathbf{f}^T(x) = \begin{bmatrix} \mathbf{f}_v^T(x) \\ \mathbf{f}_\theta^T(x) \end{bmatrix} = \begin{bmatrix} 1 & x & x^2 & x^3 \\ 0 & 1 & 2x & 3x^2 \end{bmatrix}, \; \mathbf{a} = \begin{bmatrix} a_1 & a_2 & a_3 & a_4 \end{bmatrix}^T \tag{3.39}$$

3. Relate displacements within the element to the nodal displacements

   The general displacements within the element can be related to the nodal displacements using Eq. (3.6).

$$v(x) = \mathbf{f}^T(x)\,.\mathbf{C}^{-1}.\mathbf{u}^e$$

$$\mathbf{C} = \begin{bmatrix} \mathbf{f}^T(x_1) \\ \mathbf{f}^T(x_2) \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 1 & L & L^2 & L^3 \\ 0 & 1 & 2L & 3L^2 \end{bmatrix}$$

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix} \tag{3.40}$$

Thus the shape functions can then be calculated by:

$$\mathbf{N}^e(x) = \mathbf{f}^T(x)\,.\mathbf{C}^{-1} = \begin{bmatrix} 1 & x & x^2 & x^3 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ \frac{-3}{L^2} & \frac{-2}{L} & \frac{3}{L^2} & \frac{-1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & \frac{-2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

This results in

$$\mathbf{N}^e(x) = \begin{bmatrix} 1-\frac{3}{L^2}x^2+\frac{2}{L^3}x^3 & x-\frac{2}{L}x^2+\frac{x^3}{L^2} & \frac{3}{L^2}x^2-\frac{2}{L^3}x^3 & -\frac{x^2}{L}+\frac{x^3}{L^2} \end{bmatrix}$$

4. Strain-displacment relationship:

   The "strains" $\varepsilon(x)$ at any point within the element can be related to the nodal displacements, $\mathbf{u}^e$ based on Eq. (3.8)

$$\boldsymbol{\varepsilon}(\mathbf{x}) = \mathbf{B.u}^e \tag{3.41}$$

Where $\varepsilon(x)$ is the "strain" for the beam. The only strain that need to be considered is the curvature about the z-axis. For the beam considered here, all other strains such as shear strain and axial strain are assumed to be zero. The curvature at any point is defined as: $\varepsilon(x)=-d^2v/dx^2$. Therefore, the matrix **B** in Eq.(3.41) is defined as:

$$\mathbf{B} = [\ -d^2\mathbf{f}_\theta^T(x)/dx^2\ ]\,\mathbf{C}^{-1} = [0\ 0\ \ -2\ \ -6x]\,\mathbf{C}^{-1}$$

$$\mathbf{B} = \begin{bmatrix} \frac{6}{L^2}-\frac{12x}{L^3} & \frac{4}{L}-\frac{6x}{L^2} & -\frac{6}{L^2}+\frac{12x}{L^3} & \frac{2}{L}-\frac{6x}{L^2} \end{bmatrix} \tag{3.42}$$

5. Stress-strain relationship

   The "stress" for the beam element, which corresponds to the "strain" or curvature of the beam, is the internal moment. The moment at any point within the beam can be related to the curvature as:

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

   Therefore, the stress-strain relationship is:

$$\boldsymbol{\sigma}(x) = \mathbf{D}.\boldsymbol{\varepsilon}(x) = \mathbf{EI.B.u}^e \tag{3.43}$$

6. Relate the Internal stresses to the nodal loads

   Based on the principle of virtual work the stiffness matrix was obtained as:

$$\mathbf{k}^e = \int \mathbf{B}^T.\mathbf{D}.\mathbf{B}.dV$$

$$\mathbf{k}^e = A\int_0^L \mathbf{B}^T\,\mathbf{EI}.\,\mathbf{B}\,dx = EIA\int_0^L \mathbf{B}^T.\mathbf{B}\ dx$$

$$\mathbf{k}^e = AEI\int_0^L \begin{bmatrix} \frac{36}{L^4}-\frac{144x}{L^5}+\frac{144x^2}{L^6} & \frac{24}{L^3}-\frac{84x}{L^4}+\frac{72x^2}{L^5} & -\frac{36}{L^4}+\frac{144x}{L^5}-\frac{144x^2}{L^6} & \frac{12}{L^3}-\frac{60x}{L^4}+\frac{72x^2}{L^5} \\ \frac{24}{L^3}-\frac{84x}{L^4}+\frac{72x^2}{L^5} & \frac{16}{L^2}-\frac{48x}{L^3}+\frac{36x^2}{L^4} & -\frac{24}{L^3}+\frac{84x}{L^4}-\frac{72x^2}{L^5} & \frac{8}{L^2}-\frac{36x}{L^3}+\frac{36x^2}{L^4} \\ -\frac{36}{L^4}+\frac{144x}{L^5}-\frac{144x^2}{L^6} & -\frac{24}{L^3}+\frac{84x}{L^4}-\frac{72x^2}{L^5} & \frac{36}{L^4}-\frac{144x}{L^5}+\frac{144x^2}{L^6} & -\frac{12}{L^3}+\frac{60x}{L^4}-\frac{72x^2}{L^5} \\ \frac{12}{L^3}-\frac{60x}{L^4}+\frac{72x^2}{L^5} & \frac{8}{L^2}-\frac{36x}{L^3}+\frac{36x^2}{L^4} & -\frac{12}{L^3}+\frac{60x}{L^4}-\frac{72x^2}{L^5} & \frac{4}{L^2}-\frac{24x}{L^3}+\frac{36x^2}{L^4} \end{bmatrix} dx$$

$$\mathbf{k}^e = A\begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}$$

The stiffness matrix of the beam element is symmetric, as expected.

The final step is to calculate the nodal load vector assuming that the distributed load is constant along the beam, f(x)=w The nodal forces for the beam are given by:

$$\mathbf{f}^e = \int_0^L \mathbf{N}^T(x).f(x)dx$$

Thus

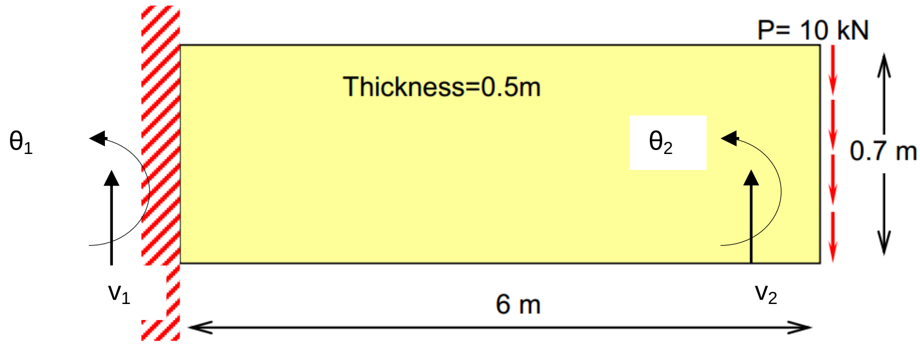
$$\mathbf{f}^e = w\int_0^L \begin{bmatrix} 1-\frac{3}{L^2}x^2+\frac{2}{L^3}x^3 \\ x-\frac{2}{L}x^2+\frac{x^3}{L^2} \\ \frac{3}{L^2}x^2-\frac{2}{L^3}x^3 \\ -\frac{x^2}{L}+\frac{x^3}{L^3} \end{bmatrix} dx = w\begin{bmatrix} L-L+\frac{L}{2} \\ \frac{L^2}{2}-\frac{2L^2}{3}+\frac{L^2}{4} \\ L-\frac{L}{2} \\ -\frac{L^2}{3}+\frac{L^2}{4} \end{bmatrix} = \begin{bmatrix} \frac{wL}{2} \\ \frac{wL^2}{12} \\ \frac{wL}{2} \\ -\frac{wL^2}{12} \end{bmatrix}$$

The element matrix equation of the beam becomes

$$A\begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}\begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} \frac{wL}{2} \\ \frac{wL^2}{12} \\ \frac{wL}{2} \\ -\frac{wL^2}{12} \end{bmatrix}$$

***Problem 1***

Solve the problem of cantilever beam of Assignment 2, question 2 using this finite element solution with one element. Compare the deflection, bending moment, and shear force versus position, to the analytical solution of simple beam.


P= 10 kN
Thickness=0.5m
θ1
θ2
0.7 m
v1
v2
6 m

# Chapter 4

# BAR AND BEAM FRAMES

The behaviour of frames structures consisting of bar and beam elements is considered in this chapter. Simple forms of these structures may be analysed using a variety of manual techniques. However, a complex structure like the frame structure in Figure 4.1 consisting of many thousands of these elements, or a structure combining these elements with continuum elements as shown in Figure 4.2, is best suited to analysis by the finite element method.

The stiffness of the complete structure can be constructed using the stiffness of each individual element. This matrix represents the relationship between the forces applied to any particular node to the displacement of all the nodes in the structure. But since one node may be shared by different elements, the assembly of the global stiffness matrix is not straighforwards. In this chapter we will deal with this important step of the finite element analysis: given the stiffness matrices of all individual elements in a structure. How can these matrices be combined to form the stiffness matrix of the complete structure?

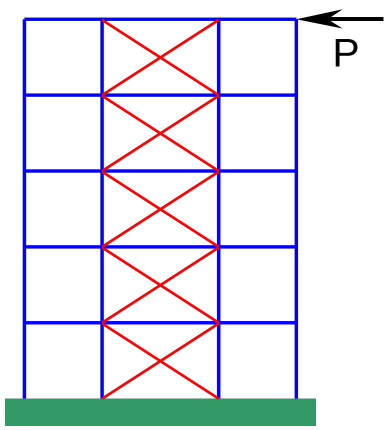



**Figure 4.1 Framed structure**

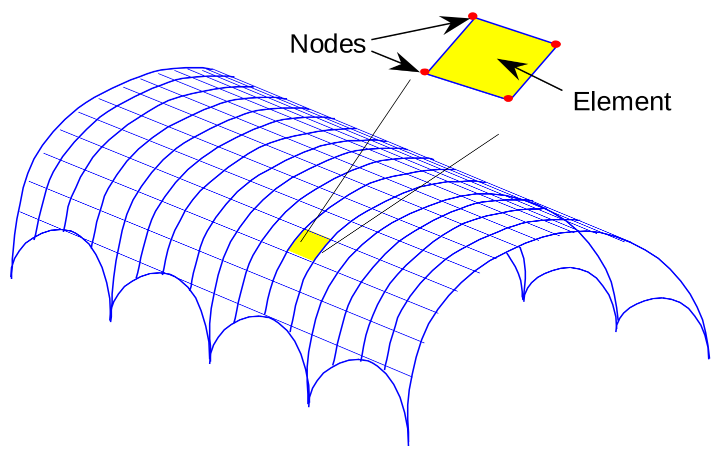



**Figure 4.2 Continuum structure and finite element**

## 4.1: Assembly of global stiffness matrix.

In this section we will learn how to assemble the global matrices from the corresponding element matrices. For a complex structure consisting of beams and columns and braces (Figure 4.1), the global stiffness matrix defines the relationship between the load applied at any point to the deformation of any other point in the structure. (The distinct points in a structure where the loads are applied or where the displacements are required are termed “nodes”). The stiffness matrix of individual element is given by

$$\mathbf{f}^e = \mathbf{k}^e.\mathbf{u}^e, \quad e=1,\ldots, n, \tag{4.1}$$

In the first step of the assembly, the element matrices $\mathbf{f}^e$ and $\mathbf{k}^e$ of size n x n are expanded to $\mathbf{F}^e$ and $\mathbf{K}^e$ so that the equation above results in

$$\mathbf{F}^e = \mathbf{K}^e.\mathbf{u}$$

The expanded stiffness matrices have the dimensions $n_{dof}$ x $n_{dof}$ of the global matrix equations. The column vector **u** contains the degrees of freedom of the whole structure. The column vector $\mathbf{F^e}$ and the matrix $\mathbf{K^e}$ is completed with zeros for all nodes that do not belong to the element.

In the second step of the assembly, the global matrix equation is created by summing all the expanded equations, leading to

$$F = k.u, k = \sum_{e=1}^{n} k^e, F = \sum_{e=1}^{n} F^e \tag{4.2}$$

## 4.2: Stiffness Matrix of a Simple One-dimensional Structure

First we present the procedure for the assembly of the stiffness matrix of a simple structure consisting of two bar elements will be examined in detail. Consider the two-bar-structure in Figure 4.3. The structure has 3 nodes, each of which may deform and to each of which a force may be applied. Therefore, the force vector or displacement vector has 3 components and the stiffness matrix is of order 3×3.

$$\begin{bmatrix} p_1 \\ p_2 \\ p_3 \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}, \tag{4.3}$$

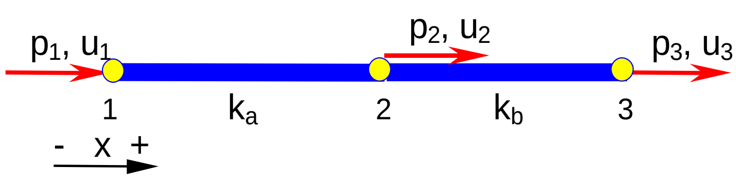



**Figure 4.3 Two-bar-structure**

By examining the stiffness matrix of the structure more closely, it may be visualized that the stiffness matrix of the complete structure can be formed by the stiffness matrices of the individual elements. The stiffness matrices, the load vectors and the displacement vectors of each of the elements can be written as:

Element a: $$\begin{bmatrix} p_1^a \\ p_2^a \end{bmatrix} = \begin{bmatrix} k_a & -k_a \\ -k_a & k_a \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \tag{4.4}$$

Element b: $$\begin{bmatrix} p_2^b \\ p_3^b \end{bmatrix} = \begin{bmatrix} k_b & -k_b \\ -k_b & k_b \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} \tag{4.5}$$

Although the two stiffness matrices are of the same order they may not be added directly since they relate to different sets of nodes. However, by adding rows and columns of zeros, both of the element stiffness matrices may be expanded in such a way that each row and column relates to the three nodes. (A general procedure for the assembly of the stiffness matrices of elements will be given later.)

Element a:
$$\begin{bmatrix} p_1^a \\ p_2^a \\ 0 \end{bmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & k_a & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} \tag{4.6}$$

Element b:
$$\begin{bmatrix} 0 \\ p_2^b \\ p_3^b \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & k_b & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} \tag{4.7}$$

The above matrices can now be added together to assemble the stiffness matrix of the complete structure.

Two-bar structure:
$$\begin{bmatrix} p_1 \\ p_2 \\ p_3 \end{bmatrix} = \begin{bmatrix} p_1^a \\ p_2^a + p_2^b \\ p_3^b \end{bmatrix} = \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & k_a + k_b & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}, \tag{4.8}$$

The simple procedure for the assembly of the global stiffness matrix for the two-bar-element structure can be extended for more complex structures.

## 4.3: Stiffness Matrix of a Simple Two-dimensional Trusses

The value of the stiffness matrix of a one-dimensional bar element has been obtained from the standard stress-strain relationship. The individual components of the stiffness matrix have been determined by permitting the element to adopt each independent mode of deformation in turn and determining the relationship between this deformation and the nodal forces. This method can be generalised to calculate the stiffness matrix of two-dimensional bar elements, e.g. element (a) in Figure 4.4

Structural frames usually consist of members that are connected to each other at various angles. Before useful expressions can be written for the analysis of complete structures, it is necessary to express the nodal forces, nodal displacements and the stiffness matrix of each element in a coordinate system that is common to all members of the structure. A suitable frame of reference, normally a Cartesian coordinate system, is used, as shown in Figure 4.4 for a simple pin-jointed frame and in Figure 4.5 for one member of the frame.

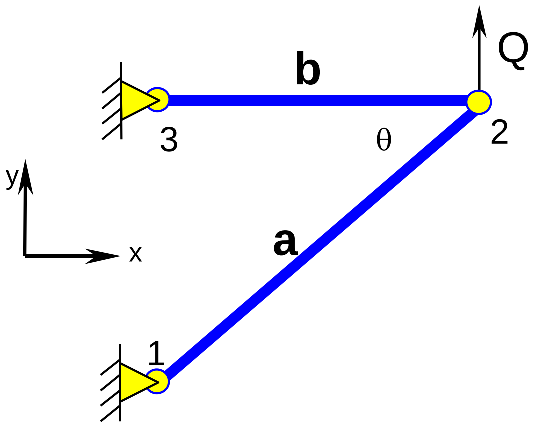



**Figure 4.4 Two-dimensional truss**

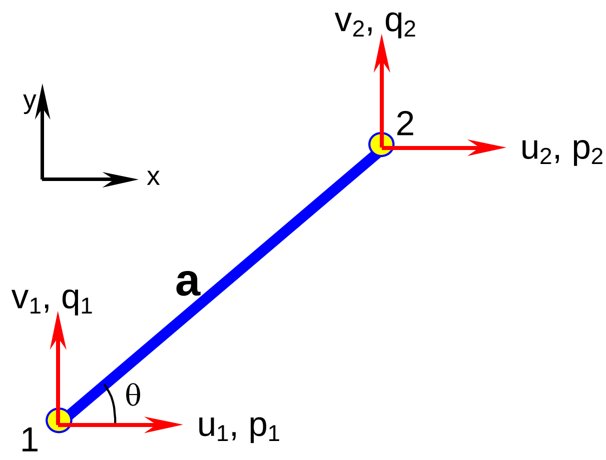
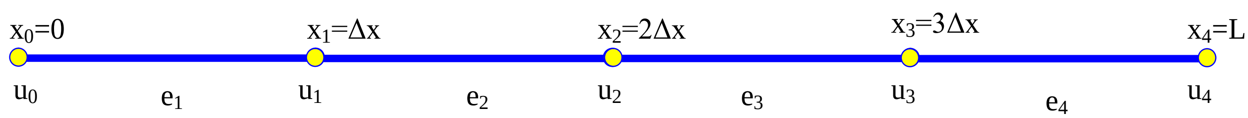



**Figure 4.5 Two-dimensional bar element**

The nodal displacements and the nodal forces of each individual bar element can be expressed as: $\mathbf{q}=(u_1, v_1, u_2, v_2)^T$ and $\mathbf{f}=(p_1, q_1, p_2, q_2)^T$, respectively. These two vectors are related to each other by a 4×4 stiffness matrix, as $\mathbf{kq}=\mathbf{f}$.

***Example 1:***
For example, consider the landslide example from Chapter 2. The domain of the problem was divided as




From expanding the matrix for each of the elements

$$\frac{EA}{\Delta x}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_{i-1} \\ u_i \end{bmatrix} = \frac{f\Delta x A}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \qquad i=1,2,3,4$$

We expand the matrix in each one of the elements

$$\begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{2E}\begin{bmatrix} 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & -1 & 0 & 0 \\ 0 & -1 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{2E}\begin{bmatrix} 0 \\ 1 \\ 1 \\ 0 \\ 0 \end{bmatrix}$$

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & -1 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{2E}\begin{bmatrix} 0 \\ 0 \\ 1 \\ 1 \\ 0 \end{bmatrix} \qquad \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & -1 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{2E}\begin{bmatrix} 0 \\ 0 \\ 0 \\ 1 \\ 1 \end{bmatrix}$$

By summing all of the expanded matrices we obtain the unrestrained global matrix equation

$$\begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ -1 & 2 & -1 & 0 & 0 \\ 0 & -1 & 2 & -1 & 0 \\ 0 & 0 & -1 & 2 & -1 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{E} \begin{bmatrix} 1/2 \\ 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix}$$

The next step is to impose the boundary condition at the first node $u_0 = 0$. First we remove the first row of the above equation (this equation provides information about the reaction force at the restrained node that is not of our interest at this moment). The resulting equation is

$$\begin{bmatrix} -1 & 2 & -1 & 0 & 0 \\ 0 & -1 & 2 & -1 & 0 \\ 0 & 0 & -1 & 2 & -1 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_0 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{E} \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix}$$

Then we separate the first column from the Equation to obtain

$$\begin{bmatrix} -1 \\ 0 \\ 0 \\ 0 \end{bmatrix} u_0 + \begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{E} \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix}$$

Imposing the boundary condition at the first node $u_0 = 0$ we obtain

$$\begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{f\Delta x^2}{E} \begin{bmatrix} 1 \\ 1 \\ 1 \\ 1/2 \end{bmatrix}$$

This is the same result as derived in Chapter 2 but using a different method: In Chapter 2 we obtained the global matrix equation using the global shape function; here we calculate first the element matrix equation and then assembled all matrices and apply boundary conditions. Note that the essential boundary condition (nodes with zero displacement) was applied by eliminating the row and column of the corresponding node.

Example 2:

Construction of the stiffness matrix for a simple pin-jointed structure consists of two bar elements as shown in Figure 4.6 - is considered here as an example. Both elements have the same cross-section area, A, and Young's modulus, E.

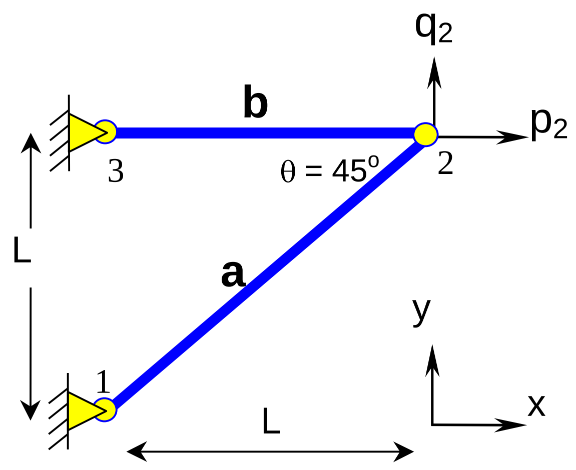



**Figure 4.6 Two-dimensional frame structure**

The stiffness matrices for elements in the global coordinate system and their relevant force vectors and displacement vectors are shown below. Element (a) connects node 1 to node 2. The inclination angle of element (a) is $\theta = 45^o$. Therefore:

Element (a):
$$\frac{A.E}{\sqrt{2}\ L}\begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} = \begin{bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \end{bmatrix}$$

Element (b) connects node 2 to node 3, thus $\theta = 180^o$.

Element (b):
$$\frac{A.E}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} p_2 \\ q_2 \\ p_3 \\ q_3 \end{bmatrix}$$

These matrices can now be combined to assemble the stiffness matrix of the complete structure.

$$\frac{A.E}{L}\begin{bmatrix} 1/2\sqrt{2} & 1/2\sqrt{2} & -1/2\sqrt{2} & -1/2\sqrt{2} & 0 & 0 \\ 1/2\sqrt{2} & 1/2\sqrt{2} & -1/2\sqrt{2} & -1/2\sqrt{2} & 0 & 0 \\ -1/2\sqrt{2} & -1/2\sqrt{2} & 1/2\sqrt{2}+1 & 1/2\sqrt{2} & -1 & 0 \\ -1/2\sqrt{2} & -1/2\sqrt{2} & 1/2\sqrt{2} & 1/2\sqrt{2} & 0 & 0 \\ 0 & 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \\ p_3 \\ q_3 \end{bmatrix} \tag{4.9}$$

Eq. (4.9) can be solved if sufficient boundary restrains are applied to the structure.

## 4.4: Two-Dimensional Trusses

Plane trusses consist of a pin-jointed assembly of bar elements, each of which is in a state of pure tension or compression. A simple truss structure is shown in Figure 4.7, which is the subject of the analysis in this section. The general procedure explained in section 2.2 for finite element analyses is employed here for the analysis of the truss structure. The general procedure for finite element analyses depends very little on the type of the structure and whether the structure is a truss, a frame, or a discretised continuum.

1. Coordinate system

   A cartesian coordinate system is best suited to any type of truss.

2. Discretisation

   The truss structure consists of 10 members. Each of the members is chosen as a pin-jointed finite element. No further discretisation is required for a simple truss structure. The finite elements are numbered from 1 to 10 (in circles) as shown inFigure 4.7. A linear bar element has two nodes, and each node has 2 degrees-of-freedom.

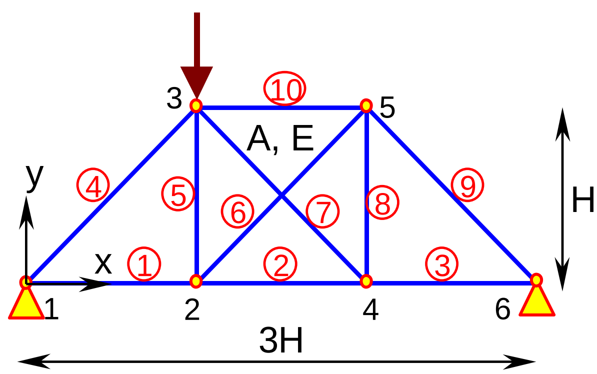



**Figure 4.7 Truss structure**

3. Node numbering system

   The choice of the node numbering system for a structure affects the distribution of the non-zero stiffness components in the global stiffness matrix. It also affects the storage size of the stiffness matrix in many finite element programs. In general, a good node numbering system shall minimise the difference between the end node numbers of any member that is a part of the structure. Such a numbering system for the nodes is shown in Figure 4.7.

4. Element stiffness matrix

   The stiffness matrix of bar elements has been derived in the previous sections as:

$$k^e = \frac{A.E}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{4.10}$$

Where A, E, and L are the cross-section area, the Young's modulus, and the length of the bar element, respectively, and c=cos θ, s=sin θ, where θ is the inclination angle of the element axis with respect to the global x-axis, measured in the anti-clock wise direction. The stiffness matrices of all elements are calculated from Eq. (4.10) and shown in Table 4-1. The displacement vectors for different elements are also shown in the same table.

**Table 4-1 Stiffness matrices and displacement vectors of the bar elements**

| Element No. | Displacement vectors | Stiffness matrices |
|---|---|---|
| 1, 2, 3, 10<br><br>L=H<br>θ=0 $^o$ | $u_1^e = [u_1, v_1, u_2, v_2]$<br>$u_2^e = [u_2, v_2, u_4, v_4]$<br>$u_3^e = [u_4, v_4, u_6, v_6]$<br>$u_{10}^e = [u_3, v_3, u_5, v_5]$ | $k^e_{1,2,3,10} = \frac{A.E}{H}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$ |
| 4, 6<br><br>L= $\sqrt{2}$ H<br>θ =45 $^o$ | $u_4^e = [u_1, v_1, u_3, v_3]$<br>$u_6^e = [u_2, v_2, u_5, v_5]$ | $k^e_{4,6} = \frac{A.E}{2\sqrt{2}\,H}\begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix}$ |
| 5, 8<br><br>L=H,<br>θ =90 $^o$ | $u_5^e = [u_2, v_2, u_3, v_3]$<br>$u_8^e = [u_4, v_4, u_5, v_5]$ | $k^e_{5,8} = \frac{A.E}{H}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$ |
| 7, 9<br><br>L= $\sqrt{2}$ H<br>θ =135 $^o$ | $u_7^e = [u_4, v_4, u_3, v_3]$<br>$u_9^e = [u_6, v_6, u_5, v_5]$ | $k^e_{7,9} = \frac{A.E}{2\sqrt{2}\,H}\begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix}$ |

5. Global stiffness matrix

The element stiffness matrices can be enlarged to full structure size and added together to assemble the global stiffness matrix for the complete structure. An example of this type of assembly has been given in Chapter 1. Since each node has two degrees-of-freedom, the unrestrained global stiffness matrix for the 6-noded structure is of the order 12×12:

$$K=\frac{EA}{H}\begin{bmatrix}
1+\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -1 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & 0 & 0 & 0 \\
\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & 0 & 0 & 0 \\
-1 & 0 & 2+\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 0 & 0 & -1 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
0 & 0 & \frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 & 0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -1 & 0 & 0 & 0 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & -1 & 0 & 1+\frac{1}{\sqrt{2}} & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & 0 \\
0 & 0 & -1 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 2+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & -1 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 & 0 & 0 \\
0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -1 & 0 & 0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\
0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & -1 & 0 & 1+\frac{1}{\sqrt{2}} & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}}
\end{bmatrix}$$

6. Boundary conditions

The boundary conditions shall be applied by eliminating rows and columns of the global stiffness matrix associated with the fixed degrees-of-freedom. Four of the degrees-of-freedom are restrained, i.e., $u_1$, $v_1$, $u_6$, $v_6$. Therefore, columns 1, 2, 11, 12 and rows 1, 2, 11,12 of the global stiffness matrix are eliminated and the size of the restrained stiffness matrix reduces to 8×8:

$$K_R=\frac{EA}{H}\begin{bmatrix}
2+\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 0 & 0 & -1 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 & 0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -1 & 0 \\
0 & -1 & 0 & 1+\frac{1}{\sqrt{2}} & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
-1 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 2+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
0 & 0 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -1 & 0 & 0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & -1 & 0 & 1+\frac{1}{\sqrt{2}}
\end{bmatrix} \qquad (4.11)$$

The restrained degrees-of-freedom shall also be eliminated from the global displacement vector and the global force vector:

$$\Delta_R=[\ u_2\ ,\ v_2\ ,\ u_3\ ,\ v_3\ ,\ u_4\ ,\ v_4\ ,\ u_5\ ,\ v_5\ ]^T$$

$$F_R=[\ p_2\ ,\ q_2\ ,\ p_3\ ,\ q_3\ ,\ p_4\ ,\ q_4\ ,\ p_5\ ,\ q_5\ ]^T$$

7. Solution of the finite element equations

The finite element equations can now be solved for the unknown nodal displacements:

$$\mathbf{K_R^{-1}}\,.\,\mathbf{F_R}=\mathbf{\Delta_R} \qquad (4.12)$$

where:

$$K_R^{-1} = \frac{H}{EA}\begin{bmatrix} 0.6547 & -0.1845 & 0.1488 & -0.1488 & 0.3453 & 0.1488 & 0.1845 & 0.1845 \\ -0.1845 & 2.7294 & -0.0641 & 1.9497 & -0.1488 & 1.5454 & 0.4895 & 1.4323 \\ 0.1488 & -0.0641 & 1.1200 & -0.1772 & 0.1845 & -0.4895 & 0.6736 & -0.2692 \\ -0.1488 & 1.9497 & -0.1772 & 2.0628 & -0.1845 & 1.4323 & 0.2692 & 1.2120 \\ 0.3453 & -0.1488 & 0.1845 & -0.1845 & 0.6547 & 0.1845 & 0.1488 & 0.1488 \\ 0.1488 & 1.5454 & -0.4895 & 1.4323 & 0.1845 & 2.7294 & 0.0641 & 1.9497 \\ 0.1845 & 0.4895 & 0.6736 & 0.2692 & 0.1488 & 0.0641 & 1.1200 & 0.1772 \\ 0.1845 & 1.4323 & -0.2692 & 1.2120 & 0.1488 & 1.9497 & 0.1772 & 2.0628 \end{bmatrix}$$

Assuming that a vertical load of 1000 kN is applied at node 3, as shown in Eq.(4.10), and $E=2\times10^8$ kPa, $A=0.01\,m^2$, H=4 m, a solution to Eq. (4.12) results in:

$$\Delta_R = [\ u_2\ ,\ v_2\ ,\ u_3\ ,\ v_3\ ,\ u_4\ ,\ v_4\ ,\ u_5\ ,\ v_5\ ]^T$$

$$= [\ .0003,\ -.0039,\ .00035,\ -.00413,\ .00037,\ -.00286,\ -.00054,\ -.00242\ ]^T$$

The displacements associated with the restrained degrees-of-freedom, $u_1$, $v_1$, $u_6$ and $v_6$ are all zero. The reactions at node 1 and 6, i.e., $p_1$, $q_1$, $p_6$, $q_6$, can be calculated by multiplying the first, second, eleventh and twelfth rows of the unrestrained stiffness matrix by the displacement vector, $\Delta$ :

$$[\ p_1\ ,\ q_1\ ,\ p_6\ ,\ q_6\ ]^T = [\ 517.88,\ 666.67,\ -517.88,\ 333.33\ ]$$

8. Calculation of stresses and strains for each element

The axial strain, $\varepsilon$, and axial stress, $\sigma$, in any element can be calculated from the element nodal displacements. The nodal displacements should be transformed into the local coordinate system of the element under consideration. The relationship between the element nodal displacements in the local coordinate system, $\Delta^e$, and the element nodal displacements in the global coordinate systems, $u^e$, was given in Eq. (4.1):

$$\mathbf{\Delta}^e = \mathbf{T}^T \, . \, \mathbf{u}^e \qquad (4.13)$$

Where T is the transformation matrix, defined by:

$$T = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 0 & 0 \\ \sin(\theta) & \cos(\theta) & 0 & 0 \\ 0 & 0 & \cos(\theta) & -\sin(\theta) \\ 0 & 0 & \sin(\theta) & \cos(\theta) \end{bmatrix}$$

In which $\theta$ is the inclination angle of the element. The axial strain and stress for element 6, for example, are calculated as follow. The element nodal displacement vector in the global system, $u^e$, is:

$$u^6 = [\, u_2 \,, v_2 \,, u_5 \,, v_5 \,]^T = [\, 0.00030 \,, -0.00390, -0.00054 \,, -0.00242 \,]^T$$

Therefore the element nodal displacements in the local coordinate system is:

$$\Delta^6 = T^T \,.\, u^e = [U_1 \,, V_1 \,, U_2 \,, V_2]^T$$

$$\Delta^6 = \begin{bmatrix} \cos(45) & \sin(45) & 0 & 0 \\ -\sin(45) & \cos(45) & 0 & 0 \\ 0 & 0 & \cos(45) & \sin(45) \\ 0 & 0 & -\sin(45) & \cos(45) \end{bmatrix} \begin{bmatrix} 0.00030 \\ -.00390 \\ -.00054 \\ -.00242 \end{bmatrix} = \begin{bmatrix} -0.00255 \\ -0.00297 \\ -0.00209 \\ -0.00133 \end{bmatrix}$$

The axial strain and stress can be calculated for the element as:

$$\varepsilon = \frac{U_2 - U_1}{L} = \frac{-0.00209 + 0.00255}{4\sqrt{2}} = 0.00008$$

$$\sigma = E \,.\, \varepsilon = 15986 \;\; kPa$$

## 4.5: Two-Dimensional flexural Frames

Flexural frames are structures with rigid jointed members that resist loads primarily by flexural action. The stiffness relation is first derived in a local coordinate system, defined by the member axes, and is then transformed to the global system Figure 4.8. The stress resultants at any point of such members consist of a moment, a transverse shear force, and an axial force. Thus the number of degrees-of-freedom at each node is $d_{of}$=3. The total degrees-of-freedom for the two-noded flexural element shown in Figure 4.8 is therefore $n_{dof}$=6. The size of the element stiffness matrix is 6×6.

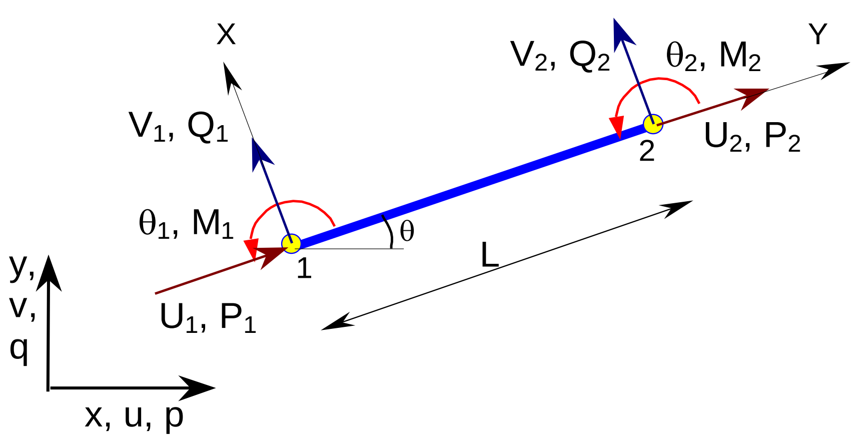



**Figure 4.8 Two-node beam element**

The stiffness equation of a beam element in its local coordinate system was obtained in Chapter 2, ignoring the effects of shear deformations and axial forces, as:

$$\begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{bmatrix} V_1 \\ \theta_1 \\ V_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} Q_1 \\ M_1 \\ Q_2 \\ M_2 \end{bmatrix} \tag{4.14}$$

Equation 4.14 can be expanded to include the effects of axial forces, $p_1$ and $p_2$:

$$\begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{bmatrix} U_1 \\ V_1 \\ \theta_1 \\ U_2 \\ V_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} P_1 \\ Q_1 \\ M_1 \\ P_2 \\ Q_2 \\ M_2 \end{bmatrix} \tag{4.15}$$

For an arbitrarily oriented beam element, inclined at an angle θ, it is necessary to express the stiffness matrix in the global coordinate system. The local and global systems of forces and displacements at each node can be related by:

$$\begin{bmatrix} P \\ Q \\ M \end{bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} p \\ q \\ M \end{bmatrix}$$

$$\begin{bmatrix} U \\ V \\ \theta \end{bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} u \\ v \\ \theta \end{bmatrix}$$

Therefore, local and global nodal forces and displacements are related by:

$$\mathbf{F^e = T^T . f^e} \quad , \quad \mathbf{\Delta^e = T^T . u^e}$$

Where

$$
T = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 0 & 0 & 0 & 0 \\ \sin(\theta) & \cos(\theta) & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos(\theta) & -\sin(\theta) & 0 \\ 0 & 0 & 0 & \sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}
$$

The element stiffness matrix in the global coordinate system can be expressed as:

$$
\mathbf{k^e = T.K^e.T^T}
$$

or:

$$
k^e = \begin{bmatrix}
\left(\frac{EA}{L}c^2+\frac{12EI}{L^3}s^2\right) & \left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & -\frac{6EI}{L^2}s & -\left(\frac{EA}{L}c^2+\frac{12EI}{L^3}s^2\right) & -\left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & -\frac{6EI}{L^2}s \\
\left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & \left(\frac{EA}{L}s^2+\frac{12EI}{L^3}c^2\right) & \frac{6EI}{L^2}c & -\left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & -\left(\frac{EA}{L}s^2+\frac{12EI}{L^3}c^2\right) & \frac{6EI}{L^2}c \\
-\frac{6EI}{L^2}s & \frac{6EI}{L^2}c & \frac{4EI}{L} & \frac{6EI}{L^2}s & -\frac{6EI}{L^2}c & \frac{2EI}{L} \\
-\left(\frac{EA}{L}c^2+\frac{12EI}{L^3}s^2\right) & -\left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & \frac{6EI}{L^2}s & \left(\frac{EA}{L}c^2+\frac{12EI}{L^3}s^2\right) & \left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & \frac{6EI}{L^2}s \\
-\left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & -\left(\frac{EA}{L}s^2+\frac{12EI}{L^3}c^2\right) & -\frac{6EI}{L^2}c & \left(\frac{EA}{L}-\frac{12EI}{L^3}\right)sc & \left(\frac{EA}{L}s^2+\frac{12EI}{L^3}c^2\right) & -\frac{6EI}{L^2}c \\
-\frac{6EI}{L^2}s & \frac{6EI}{L^2}c & \frac{2EI}{L} & \frac{6EI}{L^2}s & -\frac{6EI}{L^2}c & \frac{4EI}{L}
\end{bmatrix} \qquad (4.16)
$$

Note that in this case, the force vector at any point comprises stress resultants at the point consisting of a moment, a transverse force and an axial force. The displacement vector at any point also comprises a curvature, a transverse displacement and an axial displacement. For this reason, these vectors are often called the generalised force vector and generalised displacement vector, respectively.

## 4.6: Suitable Node Numbering System

A suitable node numbering system is required to minimise the computer storage required for storing the global stiffness matrices and also to save on the time required for calculation of the inverse of the stiffness matrices and solving the finite element equations. This section provides simple instructions for a suitable node numbering system.

The method for calculation of the stiffness matrix of a two-bar structure given in section 4.3 can be theoretically applied to other types of structures. It was shown that the global stiffness matrix of a structure can be formed by giving each node in the structure a unit displacement in one direction (while all other nodes being held fixed), and calculating the nodal forces developed. Since all other nodes are held fixed, forces are not developed at nodes beyond the ones linked by a member to the node being displaced. It follows that only a few stiffness terms result from a unit displacement.

If the nodes are suitably numbered so that the maximum difference between nodal numbers in any one member is kept small, the stiffness matrix consists of a narrow band of non-zero numbers clustered about the main diagonal. Figure 4.9(a) shows diagrammatically such a banded stiffness matrix. In this figure $N_{dof}$ is the order of the full square stiffness matrix and B is the “bandwidth”, defined as:

$$B = d_{of} \times \left(1 + \left| (\, Node_i - Node_j \,)_{max} \right| \right) \qquad (4.17)$$

Where $d_{of}$ is the number of degrees-of-freedom at each node and $\left|(\, Node_i - Node_j \,)_{max}\right|$ is the difference between end node numbers in the member that has the maximum difference in end node numbers.

The stiffness matrices are also symmetric. Therefore, for the purpose of efficient storage, the compact storage of Figure 4.9(b) should be adopted, in which only the upper half of the band of the whole stiffness matrix is stored. The diagonal of the whole stiffness matrix becomes the first column of the compact matrix. In large problems B may be only a few percent of $N_{dof}$. Thus very large savings in storage can be made by the compact storage of global stiffness matrix.

A large portion of the computational time in a finite element analysis is spent on solving the stiffness equations, i.e., finding the inverse of the stiffness matrix. The computational time required for solving the stiffness equations is approximately proportional to the square of the bandwidth of the stiffness matrix. Therefore, a suitable node numbering system allows considerable reductions in computational time by reducing the bandwidth.

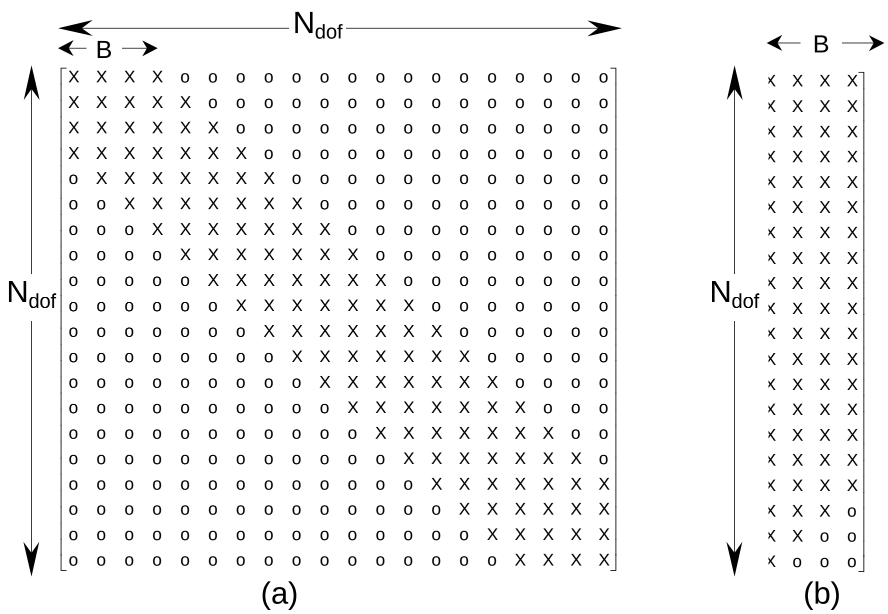



**Figure 4.9 The banded system and compact storage of the stiffness matrix**

To demonstrate the effectiveness of a suitable node numbering system in reducing the bandwidth of a structure, consider a five-story frame structure consisting of 10 nodes, each with three degrees-of-freedom. The restrained structure has 30 degrees-of-freedom, thus $N_{dof}$=30. Three different node-numbering systems are shown in Figure 4.10, together with the stiffness matrices resulting from each of the systems.

Each (x) in the stiffness matrices represents a 3×3 matrix containing the stiffness coefficients associated with a node. For system (a) the bandwidth B is equal to 9, and for systems (b) and (c), B=18 and B=30, respectively. Obviously for this structure the most suitable node numbering system is the one presented in Figure 4.10(a). The worst node numbering system is case (c) in Figure 4.10.

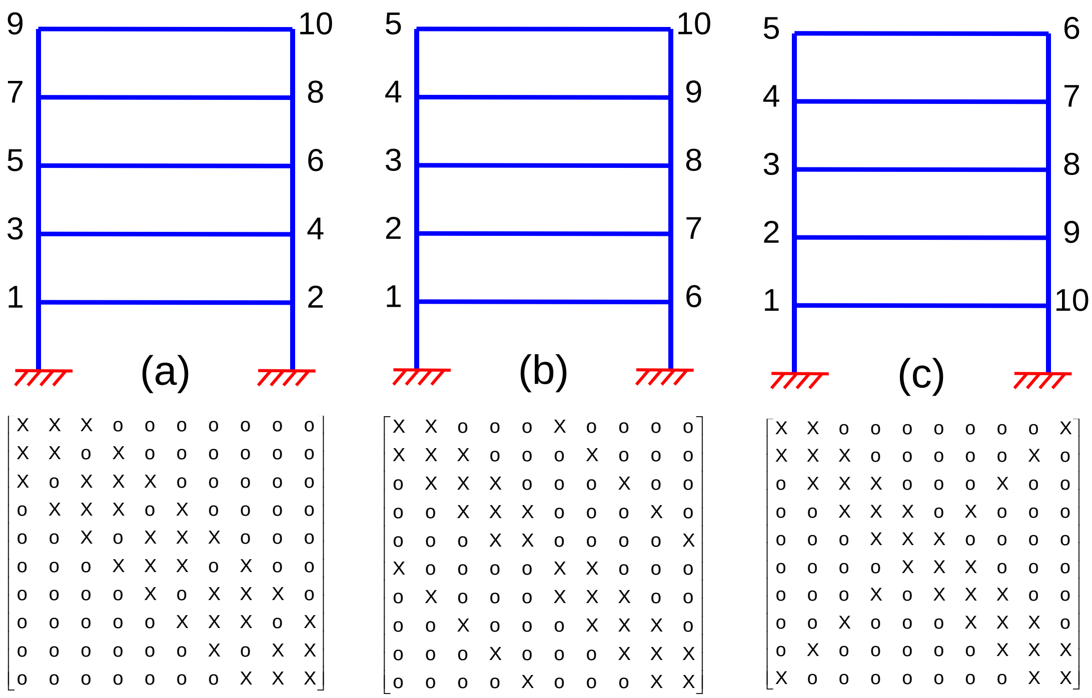
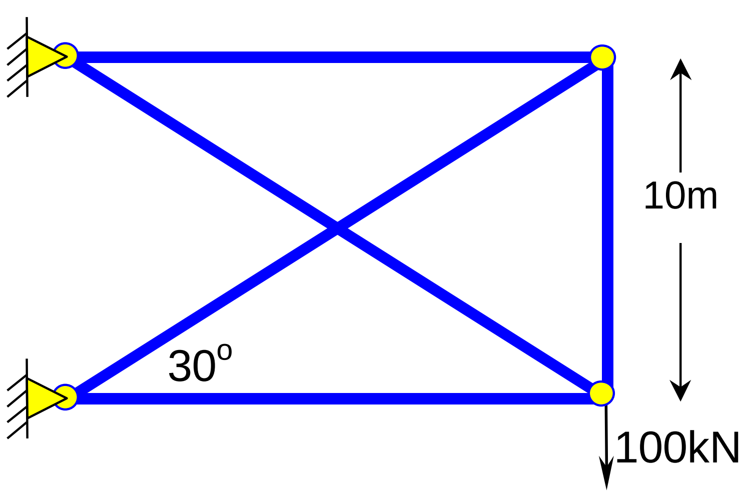



**Figure 4.10 Different node numbering systems**

(After Dawe, D. J., 1984, Matrix and Finite Element Displacement Analysis of Structures)

### *Problem 1*

- Calculate the nodal displacements and reactions for the pin-jointed structure shown below.
- All members of the structure have a cross section area A = 0.001m$^2$ and a Young's modulus E=2×10$^8$ kPa.
- Evaluate the results, are they reasonable?
- If the cross section area of the vertical member is increased by 1000 times, how does this change affect the results?




### *Problem 2*

shows a plane frame consists of 5 elements that are rigidly connected together. The supports are also fully fixed. The properties of the elements are:

Elements 1, 2, 3: $A=0.0025m^2$, $I=0.00005m^4$, $E=2\times10^8$ kPa
Elements 4, 5: $A=0.0010m^2$, $I=0.00025m^4$, $E=2\times10^8$ kPa

If a horizontal load of $p_2$=1000kN is applied at node2, calculate the rotations of node2.

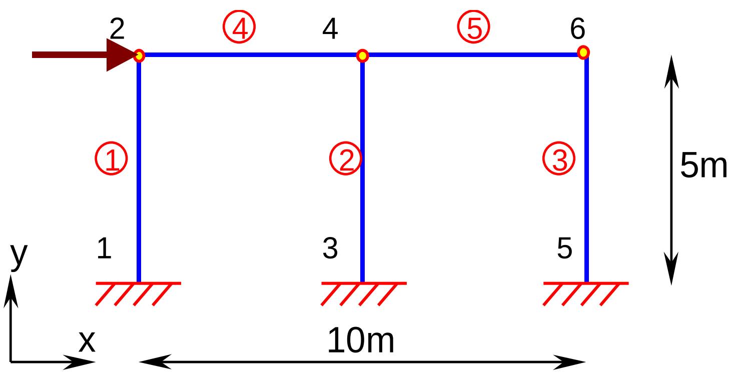



**Figure 4.11 Frame structure**

The general procedure for finite element analyses, explained in section2.2, is employed here for the analysis of the frame.

1–3 The coordinate system, discretisation (element numbering) and node numbering system used for the analysis of the frame is shown in

4. Element stiffness matrix

   The stiffness matrices of all elements are calculated using Eq. (4.16) and shown in , together with the element displacement vectors.

5. Global stiffness matrix

   The global stiffness matrix is assembled using the direct method explained in Chapter 4. The Transformation matrices for different elements are shown in Section 4.3.The restrained global stiffness matrix for the complete structure is given as:

$$K_R = \begin{bmatrix} 40960 & 0 & 2400 & -40000 & 0 & 0 & 0 & 0 & 0 \\ 0 & 104800 & 12000 & 0 & -4800 & 12000 & 0 & 0 & 0 \\ 2400 & 12000 & 48000 & 0 & -12000 & 20000 & 0 & 0 & 0 \\ -40000 & 0 & 0 & 80960 & 0 & 2400 & -40000 & 0 & 0 \\ 0 & -4800 & -12000 & 0 & 109600 & 0 & 0 & -4800 & 12000 \\ 0 & 12000 & 20000 & 2400 & 0 & 88000 & 0 & -12000 & 20000 \\ 0 & 0 & 0 & -40000 & 0 & 0 & 40960 & 0 & 2400 \\ 0 & 0 & 0 & 0 & -4800 & -12000 & 0 & 104800 & -12000 \\ 0 & 0 & 0 & 0 & 12000 & 20000 & 2400 & -12000 & 48000 \end{bmatrix}$$

**Table 4-2 Stiffness matrices of the elements**

| Element No. | Displacement vectors | Stiffness matrices |
|---|---|---|
| 1, 2, 3 | $u_1^e=[u_1, v_1, \theta_1, u_2, v_2, \theta_2]$ <br> $u_2^e=[u_3, v_3, \theta_3, u_4, v_4, \theta_4]$ <br> $u_3^e=[u_5, v_5, \theta_5, u_6, v_6, \theta_6]$ | $k_{1,2,3}^e=\begin{bmatrix} 960 & 0 & -2400 & -960 & 0 & -2400 \\ 0 & 100000 & 0 & 0 & -100000 & 0 \\ -2400 & 0 & 8000 & 2400 & 0 & 4000 \\ -960 & 0 & 2400 & 960 & 0 & 2400 \\ 0 & -100000 & 0 & 0 & 100000 & 0 \\ -2400 & 0 & 4000 & 2400 & 0 & 8000 \end{bmatrix}$ |
| 4, 5 | $u_4^e=[u_2, v_2, \theta_2, u_4, v_4, \theta_4]$ <br> $u_5^e=[u_4, v_4, \theta_4, u_6, v_6, \theta_6]$ | $k_{4,5}^e=\begin{bmatrix} 40000 & 0 & 0 & -40000 & 0 & 0 \\ 0 & 4800 & 12000 & 0 & -4800 & 12000 \\ 0 & 12000 & 40000 & 0 & -12000 & 20000 \\ -40000 & 0 & 0 & 40000 & 0 & 0 \\ 0 & -4800 & -12000 & 0 & 4800 & -12000 \\ 0 & 12000 & 20000 & 0 & -12000 & 40000 \end{bmatrix}$ |

6. Boundary conditions

   The boundary conditions have been applied to stiffness matrix by the direct assembly method. The vectors of the restrained global degree-of-freedom and the global force vector for the structure are:

$$\Delta_R=[u_2, v_2, \theta_2, u_4, v_4, \theta_4, u_6, v_6, \theta_6]^T=[a_1, a_2, a_3, a_4, a_5, a_6, a_7, a_8, a_9]^T$$
$$F_R=[p_2, q_2, M_2, p_4, q_4, M_4, p_6, q_6, M_6]^T=[1000,\ 0,\ 0,\ 0,\ 0,\ 0,\ 0,\ 0,\ 0]^T$$

7. Solution of the finite element equation

   The finite element equations can now be solved which result in the unknown nodal displacements:

$$\Delta_R=[\ 0.394, 0.002, -0.019, 0.377, 0.000, -0.002, 0.370, -0.002, -0.018\ ]^T$$

   Therefore the rotation of node2 is $\theta_2=-0.019$ radians; the negative sign indicates a clockwise rotation.

***Problem 3***

This question is about finding the structure of the finite element analysis using the steps listed below. Most of these steps belong to the three main components of the analysis: pre-processing, processing, and post-processing. Few of the steps are not necessary. Find the steps for each component and sort them in the order they should be executed during the analysis.

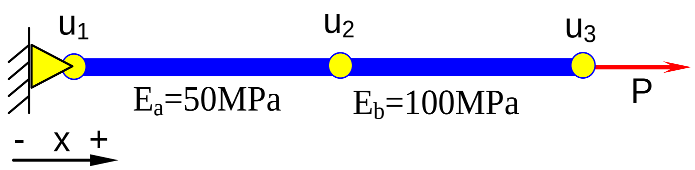
a) calculate displacement at the domain
b) assembly unrestrained global matrix equation
c) input boundary conditions
d) calculate stress at the domain
e) input material properties
f) invert global stiffness matrix
g) apply boundary conditions
h) calculate nodal loads
i) create element matrix equations
j) invert element stiffness matrices
k) input nodes
l) invert unrestrained global matrix equation
m) calculate stress at the nodes
n) input elements
o) calculate nodal displacement

Write your solution in the table below. Note: Not all boxes have to be filled

| Component | include the letters a–o of the steps, in the order they should be executed | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| **Pre-processor** | | | | | | | | | |
| **Solver** | | | | | | | | | |
| **Post-processor** | | | | | | | | | |

### *Problem 4*

This problem is about the construction of the stiffness matrix for a simple structure that consists of two bar elements as shown in the figure below. Both elements have the same cross-section area $A=0.01m^2$ and the same length L=1m. A load P=10N is applied at the right node. Write your solutions in the boxes below.




1) Write down the element matrix equation $\mathbf{f}^e = \mathbf{k}^e.\mathbf{u}^e$ for each bar.

2) Find the expanded element matrix equation $\mathbf{F}^e = \mathbf{K}^e.\mathbf{u}$ for each bar

3) Find the unrestrained global matrix equation $\mathbf{F} = \mathbf{K}.\mathbf{u}$, $\quad \mathbf{K}=\sum_{e=1}^{2}\mathbf{K}^e$, $\quad \mathbf{F}=\sum_{e=1}^{2}\mathbf{F}^e$

4) Find the global matrix equation after applying the boundary conditions.

5) Find the displacement of the unrestrained nodes

# Chapter 5
# STRAIN AND STRESS IN CONTINUA

The general equations for derivation of the finite element relationships have been established in the previous chapters through consideration of simple one-dimensional elements such as bars and beams. The extension of the general equations to two or three-dimensional elements differs from the unidirectional case only in the degree of complexity involved and not in the basic concepts. The remainder of the text will be focussed with two-dimensional elements, but before such elements can be studied in detail, a review of the relevant concepts and the governing relations of continuum mechanics will be presented.

To carry out a stress analysis of a structure using the finite element method, it is first necessary to understand the concepts of stress and strain in matrix form. Furthermore, people who intend using this method with such an aim in mind should have a good comprehension of constitutive modelling. The reason for this is obvious. Human lives will depend on how well the engineer models the structure and interprets the results. Ultimately, it is a stress analysis problem the engineer is investigating – not a computer analysis problem as often depicted in glossy FEM commercial package sales brochures. No matter how sophisticated the computer method may be, experience and knowledgeable engineering judgement should always be the absolute criterion for a correct engineering design decision. In this Chapter, the strains and stresses in continua are presented followed by the stress-strain relationships. Consideration on constitutive modelling is limited to linear isotropic elasticity and only a brief review on the theory of elasticity is provided.

## 5.1: Kinematic Equation: definition of strain

In this section the concept of normal strain and shear strain in a solid continuum will be reviewed. Expressions for transformation of strains from one coordinate system to another are also provided. When a body is subjected to applied loads it will distort. A small element which is subject to in-plane loading may deform in the manner shown schematically in Figure 5.1.

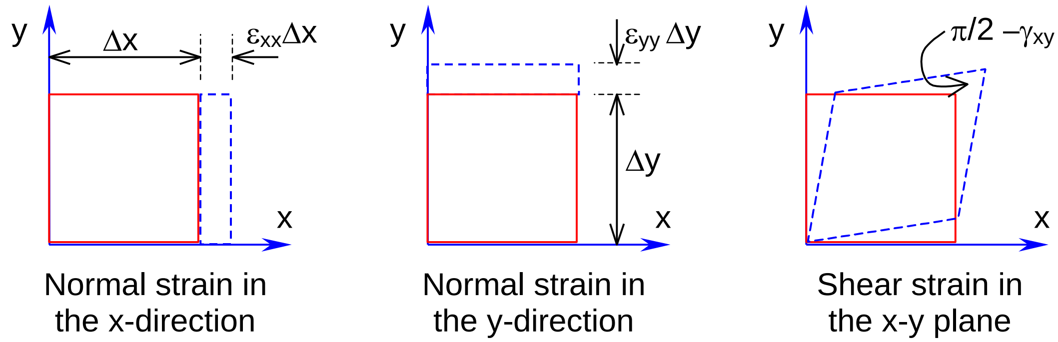



**Figure 5.1 Normal and shear strain in x-y plane**

In general a small planar distortion can be broken up into:

(a) a rigid body translation in the x direction
(b) a rigid body translation in the y direction
(c) a rigid body rotation about the z axis
(d) a normal strain $\varepsilon_{xx}$ in the x direction
(e) a normal strain $\varepsilon_{yy}$ in the y direction
(f) a shear strain $\gamma_{xy}$ in the xy plane.

The rigid body components (a, b, c) involve no change in shape and hence no strain. The axial extensions (d,e) involve a change in area while the shear strain (f) involves no change in area.

### *Relation of strains to displacements*

An examination of the displacements for the element shown in Figure 5.1 shows that for small deformations and changes of shape, the strains can be expressed in terms of the displacement components as follows:

$$\varepsilon_{xx} = \frac{\partial u_x}{\partial x}$$
$$\varepsilon_{yy} = \frac{\partial u_y}{\partial y} \qquad (5.1)a$$
$$\gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x}$$

These equations can be written in matrix form

$$\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix}$$

It is clear that by examining the deformation of elements in the yz and zx planes it is possible to identify similarly the strains in these planes:

$$\varepsilon_{zz} = \frac{\partial u_z}{\partial z}$$
$$\gamma_{yz} = \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial y} \qquad (5.1)\text{b}$$
$$\gamma_{zx} = \frac{\partial u_x}{\partial z} + \frac{\partial u_z}{\partial x}$$

The full three-dimensional kinematic relation can be written in a compact for as

$$\boldsymbol{\varepsilon} = \mathbf{L}\left[\mathbf{u}(x)\right]$$

Where:

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{yy} & \varepsilon_{zz} & \gamma_{xy} & \gamma_{yz} & \gamma_{xz} \end{bmatrix}^T$$

$$\mathbf{u} = \begin{bmatrix} u_x & u_y & u_z \end{bmatrix}^T$$

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix}$$

Eq. ((5.1)a, b) can be used to evaluate expressions for the strain components if the displacements are known. These expressions may be exact as in an analytic solution or approximate as in the case when the displacements are expressed in terms of interpolation functions. Eq.((5.1)a,b) give zero strain whenever the displacements considered correspond to a rigid body movement.

The volumetric strain $\varepsilon_v$ for an element is defined to be the increase in volume divided by the initial volume of the element and for small strains it is related to the normal strains by the following relationship.

$$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} \qquad (5.2)$$

## 5.2: Transformation of strain

It is sometimes convenient to determine the strains in terms of a local coordinate system. It is therefore necessary to find a method for transformation of strains from one coordinate system to another. The transformation of strains is facilitated by introducing the mathematical component of shear strain $\varepsilon_{xy}$. In contrast to the engineering shear strain, $\gamma_{xy}$, this is defined by the relation:

$$\varepsilon_{xy} = \varepsilon_{yx} = \frac{\gamma_{xy}}{2}$$

$$\varepsilon_{yz} = \varepsilon_{zy} = \frac{\gamma_{yz}}{2}$$

$$\varepsilon_{zx} = \varepsilon_{xz} = \frac{\gamma_{zx}}{2}$$

The strain tensor **ε** is then defined as

$$\varepsilon = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{yx} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{bmatrix}$$

Where the components of the strain tensor can be calculated from the displacements using the relationship:

$$\varepsilon_{pq} = \frac{1}{2}\left(\frac{\partial u_p}{\partial q} + \frac{\partial u_q}{\partial p}\right)$$

And p, q can be any of the symbols x, y, z.
In the transformed coordinate system the strain tensor has the form

$$E = \begin{bmatrix} \varepsilon_{XX} & \varepsilon_{XY} & \varepsilon_{XZ} \\ \varepsilon_{YX} & \varepsilon_{YY} & \varepsilon_{YZ} \\ \varepsilon_{ZX} & \varepsilon_{ZY} & \varepsilon_{ZZ} \end{bmatrix}$$

where $E_{PQ} = \frac{1}{2}\left(\frac{\partial U_P}{\partial Q} + \frac{\partial U_Q}{\partial P}\right)$ and P, Q can be any of the symbols X, Y, Z.

The local and global coordinate systems are related by the relation given in Eq.(B.5), Appendix B:

$$\mathbf{r} = \mathbf{H.R}$$

$$\text{Where } r = \begin{bmatrix} x \\ y \\ z \end{bmatrix}, \quad R = \begin{bmatrix} X \\ Y \\ Z \end{bmatrix}, \quad H^T = \begin{bmatrix} I_1 & m_1 & n_1 \\ I_2 & m_2 & n_2 \\ I_3 & m_3 & n_3 \end{bmatrix}$$

And $I_i$, $m_i$, $n_i$ are the cosine of the anti-clockwise angles between the different axes of the two coordinate systems, as defined by Eq.(B.6) in Appendix B.

The strain tensors in the different coordinate systems can be related by the relations:

$$\boldsymbol{\varepsilon} = \mathbf{H} . \mathbf{E} . \mathbf{H}^T$$

$$\mathbf{E} = \mathbf{H}^T . \boldsymbol{\varepsilon} . \mathbf{H}$$

### *Strains in a cylindrical polar coordinate*

The strain components in cylindrical polar coordinates can be found by determining the strains relative to a set of reference axes X, Y, Z with the X axis parallel to the r direction, the Y axis parallel to the θ direction and the Z axis parallel to the z axis as shown in Figure B.4, Appendix B. Thus:

$$\begin{bmatrix} \varepsilon_{rr} & \varepsilon_{r\theta} & \varepsilon_{rz} \\ \varepsilon_{\theta r} & \varepsilon_{\theta\theta} & \varepsilon_{\theta z} \\ \varepsilon_{zr} & \varepsilon_{z\theta} & \varepsilon_{zz} \end{bmatrix} = \begin{bmatrix} c & s & 0 \\ -s & c & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{yx} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{bmatrix} \begin{bmatrix} c & -s & 0 \\ s & c & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{5.3}$$

Where c=cosθ and s=sinθ.

The expressions for strains in terms of displacement components in polar coordinates are more complex than in cartesian coordinates. It is found:

$$\begin{aligned} \varepsilon_{rr} &= \frac{\partial u_r}{\partial r} \\ \varepsilon_{\theta\theta} &= \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{u_r}{r} \\ \gamma_{z\theta} &= \frac{\partial u_\theta}{\partial z} + \frac{1}{r}\frac{\partial u_z}{\partial \theta} = 2\varepsilon_{z\theta} \\ \gamma_{r\theta} &= \frac{1}{r}\frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r} = 2\varepsilon_{r\theta} \end{aligned} \tag{5.4}$$

## 5.3: Balance Equation: Definition of Stress

The previous section has been concerned with deformation of a continuous body. In this section the forces within the body that cause this deformation will be examined, stress components under three-dimensional conditions will be defined, and the concept of a stress tensor (matrix) will be introduced together with transformation of stresses in different coordinate systems.

Consider a small rectangular box, having sides of length $\Delta x$, $\Delta y$, $\Delta z$ parallel to the x, y, z axes respectively, which surrounds the point P. The material outside the boxes will exert a force on each of the six sides of the box. As the dimensions of the box approach zero the forces on the sides of the box also approach zero. However the force per unit area approaches a limiting value that is called the *traction*. Consider the positive x face (the face having the x axis as its outward normal) and assume the x, y, z components of the force acting on this face are denoted $\Delta F_{xx}$, $\Delta F_{xy}$, $\Delta F_{xz}$ respectively.

The stress components ($\sigma_{xx}$, $\sigma_{xy}$, $\sigma_{xz}$) at point P inside the face are defined by the relationships:

$$\sigma_{xx}. = \frac{\Delta F_{xx}}{\Delta A_x} \quad \sigma_{xy}. = \frac{\Delta F_{xy}}{\Delta A_x} \quad \sigma_{xz}. = \frac{\Delta F_{xz}}{\Delta A_x}$$

Where $\Delta A_x = \Delta y.\Delta z$ is the area of the x face.

It is similarly possible, by considering the force acting on the y, z faces, to define the stress components ($\sigma_{yx}$, $\sigma_{yy}$, $\sigma_{yz}$) acting on the y face and those acting on the z face ($\sigma_{zx}$, $\sigma_{zy}$, $\sigma_{zz}$). In general

$$\sigma_{pq}. = \frac{\Delta F_{pq}}{\Delta A_p}$$

Where $\Delta F_{pq}$ is the force acting on the p-face along the q-direction and $\Delta A_p$ is the area of the p-face. This leads to the conclusion that the forces per unit area are as summarized in Figure 5..

The collection of stress components $\sigma_{pq}$ (where the indices p, q can take any of the values x, y, z) is called the stress tensor at point P, and is defined below:

$$\sigma = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \qquad (5.5)$$

The stress components $\sigma_{xx}$, $\sigma_{yy}$, $\sigma_{zz}$ are called normal or direct stresses. The components $\sigma_{xy}$, $\sigma_{yz}$, $\sigma_{zx}$,, $\sigma_{yx}$, $\sigma_{zy}$, $\sigma_{xz}$ are called shear stresses. In this course a tensile normal stress will be assumed to have a positive value.

### *5.4: Traction acting on a plane*

The stress tensor defined in the previous section can be used to calculate the force per unit area acting on any plane passing through P. Suppose that a plane passing through point P has an outward unit normal **n** as shown in Fig.5.2.

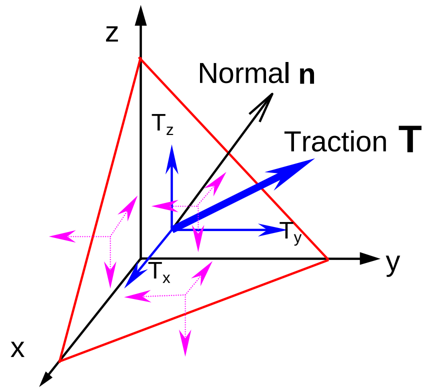



**Fig. 5.2: Traction acting on a plane**

By considering the equilibrium of the tetrahedron shown in Fig.5.2 it can be shown that the traction T (force per unit area) acting on the plane is given by:

$$\mathbf{T} = \boldsymbol{\sigma} \, . \, \mathbf{n} \tag{5.6}$$

$$T_x = \sigma_{xx} n_x + \sigma_{xy} n_y + \sigma_{xz} n_z$$

$$T_y = \sigma_{yx} n_x + \sigma_{yy} n_y + \sigma_{yz} n_z$$

$$T_z = \sigma_{zx} n_x + \sigma_{zy} n_y + \sigma_{zz} n_z$$

A simple demonstration of this is found by considering the x-y plane system of stresses in which there are no shear stresses acting on the z face, so that $\sigma_{xz}$=0 and $\sigma_{yz}$ = 0. The situation is shown schematically in Fig.5.3.

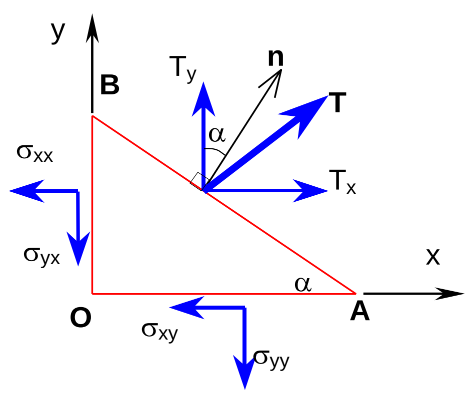



**Fig. 5.3: Relation of stress and traction**

Equilibrium of the forces in x and y directions reveals that:

$$T_x \left|AB\right| = \sigma_{xx} \left|OB\right| + \sigma_{xy} \left|OA\right|$$

$$T_y \left|AB\right| = \sigma_{yx} \left|OB\right| + \sigma_{yy} \left|OA\right|$$

$$\left|OA\right| = \cos\alpha \left|AB\right|$$

$$\left|OB\right| = \sin\alpha \left|AB\right|$$

The normal to AB is given by:

$$\mathbf{n} = [n_x \;\; n_y]^T = [\sin\alpha \;\; \cos\alpha]^T$$

So that:

$$T_x = \sigma_{xx}\, n_x + \sigma_{xy}\, n_y$$

$$T_y = \sigma_{yx}\, n_x + \sigma_{yy}\, n_y$$

## *5.5: Static equations for the stress*

Under most cases the stress distribution will vary from point to point. In most civil engineering analyses it can be assumed that processes are quasi static, i.e., the effects of acceleration can be neglected. In this case consider the equilibrium of rectangular box shown in Figure 5..

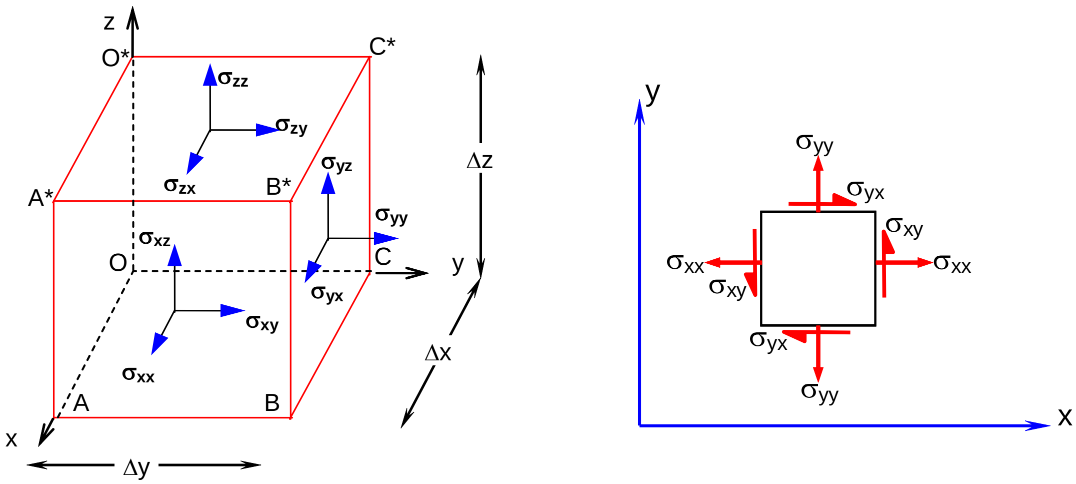



**Figure 5.4 Left: equilibrium in a rectangular box whose center is the point (x,y,z). Right: stress components seen from the top.**

The force in the z direction acting on the face $A^*B^*C^*0^*$ is: $+\sigma_{zz}\left(x,y,z+\Delta z/2\right)\Delta x\Delta y$

The force in the z direction acting on the face A B C O is: $-\sigma_{zz}\left(x,y,z-\Delta z/2\right)\Delta x\Delta y$

The force in the z direction acting on the face A B B*A* is: $+\sigma_{xz}\left(x+\Delta x/2,y,z\right)\Delta y\Delta z$

The force in the z direction acting on the face O C C*O* is: $-\sigma_{xz}\left(x-\Delta x/2,y,z\right)\Delta y\Delta z$

The force in the z direction acting on the face B C C*B* is: $+\sigma_{yz}\left(x,y+\Delta y/2,z\right)\Delta z\Delta x$

The force in the z direction acting on the face A O O*A* is: $-\sigma_{yz}\left(x,y-\Delta y/2,z\right)\Delta z\Delta x$

The force in the z direction due to the self-weight of the material is: $w_Z \Delta x \Delta y \Delta z$

In the above relations, the quantities in brackets "()" indicate the coordinates of the point at which the stress is taken.

The sum of these 7 force components must vanish. By dividing the resulting equation by the volume of the box and letting $\Delta x, \Delta y, \Delta z \rightarrow 0$ it is found that:

$$\frac{\sigma_{xz}(x+\Delta x/2,y,z)-\sigma_{xz}(x-\Delta x/2,y,z)}{\Delta x}+\frac{\sigma_{yz}(x,y+\Delta y/2,z)-\sigma_{yz}(x,y-\Delta y/2,z)}{\Delta y}+\frac{\sigma_{zz}(x,y,z+\Delta z/2)-\sigma_{zz}(x,y,z-\Delta z/2)}{\Delta z}+w_z = 0$$

Now we use the concept of the partial derivative to obtain

$$\frac{\partial\sigma_{xz}}{\partial x}+\frac{\partial\sigma_{yz}}{\partial y}+\frac{\partial\sigma_{zz}}{\partial z}+w_z=0$$

The complete set of equilibrium equations can be derived in similar fashion and it is found that:

$$\begin{aligned}
&\frac{\partial\sigma_{xx}}{\partial x}+\frac{\partial\sigma_{yx}}{\partial y}+\frac{\partial\sigma_{xz}}{\partial z}+w_x=0\\
&\frac{\partial\sigma_{xy}}{\partial x}+\frac{\partial\sigma_{yy}}{\partial y}+\frac{\partial\sigma_{yz}}{\partial z}+w_y=0\\
&\frac{\partial\sigma_{xz}}{\partial x}+\frac{\partial\sigma_{zy}}{\partial y}+\frac{\partial\sigma_{zz}}{\partial z}+w_z=0
\end{aligned} \tag{5.7}$$

Where $w_x$, $w_y$, $w_z$ are the components of the unit weight of the material in the x, y z directions respectively. It can be written in a compact form

$$\mathbf{L}^T\boldsymbol{\sigma}+\mathbf{w}=\mathbf{0}$$

Where L is the differential operator defined above, and

$$\boldsymbol{\sigma}=\begin{bmatrix}\sigma_{xx} & \sigma_{yy} & \sigma_{zz} & \sigma_{xy} & \sigma_{yz} & \sigma_{xz}\end{bmatrix}^T$$
$$\mathbf{w}=\begin{bmatrix}w_x & w_y & w_z\end{bmatrix}^T$$

### *Stress components in different coordinate systems*

The stress components defined by Eq. (5.5) were based on the x, y, z coordinate system. The coordinate system X, Y, Z could also have been used to define the stress tensor $\Sigma$ and in that case it would have been found that:

$$\Sigma = \begin{bmatrix} \sigma_{XX} & \sigma_{XY} & \sigma_{XZ} \\ \sigma_{YX} & \sigma_{YY} & \sigma_{YZ} \\ \sigma_{ZX} & \sigma_{ZY} & \sigma_{ZZ} \end{bmatrix}$$

If Eq. (5.9) is applied to the three planes having the X, Y, Z directions as outward normals respectively, the stress tensors are related by the following equations:

$$\boldsymbol{\sigma} = \mathbf{H}\boldsymbol{\Sigma}\mathbf{H}^{T}$$

$$\boldsymbol{\Sigma} = \mathbf{H}^{T}\boldsymbol{\sigma}\mathbf{H}$$

Where **H** is the transformation matrix which relates two coordinate systems and defined by Eq.(B.6).

### *Example 5.1*

In example 5.1 the stress state was given relative to the x, y, z coordinate system. However, when examining the stress state in the silt seam it is more appropriate to use a local (X, Y, Z) axes in which the Y axis is normal to the seam and the X, Z axes are in the plane of the seam. Thus

$$\boldsymbol{\Sigma} = \begin{bmatrix} \sigma_{XX} & \sigma_{XY} & \sigma_{XZ} \\ \sigma_{YX} & \sigma_{YY} & \sigma_{YZ} \\ \sigma_{ZX} & \sigma_{ZY} & \sigma_{ZZ} \end{bmatrix} = \begin{bmatrix} +0.9397 & -0.3420 & 0 \\ +0.3420 & +0.9397 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} -250 & 0 & 0 \\ 0 & -300 & 0 \\ 0 & 0 & -250 \end{bmatrix} \begin{bmatrix} +0.9397 & +0.3420 & 0 \\ -0.3420 & +0.9397 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$\boldsymbol{\Sigma} = \begin{bmatrix} -255.85 & 16.07 & 0 \\ 16.07 & -294.15 & 0 \\ 0 & 0 & -250 \end{bmatrix} \text{(kPa)}$$

### *Symmetry of the stress tensor*

The convention adopted in defining the stress components is that $\sigma_{pq}$ defines the "p" component of traction (force per unit area) acting on the plane having the "q" axis as the outward normal. By considering the moment equilibrium of the rectangular box shown in Figure 5., it can be shown that:

$$\sigma_{pq} = \sigma_{qp} \qquad (5.8)$$

*Stress components in cylindrical polar coordinates*

The stress components for a set of cylindrical polar coordinates correspond to those for a set of Cartesian axes having an X axis parallel to the r direction, a Y axis parallel to the θ direction and a Z axis parallel to the z direction.

$$\begin{bmatrix} \sigma_{rr} & \sigma_{r\theta} & \sigma_{rz} \\ \sigma_{\theta r} & \sigma_{\theta\theta} & \sigma_{\theta z} \\ \sigma_{zr} & \sigma_{z\theta} & \sigma_{zz} \end{bmatrix} = \begin{bmatrix} c & s & 0 \\ -s & c & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \begin{bmatrix} c & -s & 0 \\ s & c & 0 \\ 0 & 0 & 1 \end{bmatrix} \qquad (5.9)$$

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

The conditions of equilibrium expressed in terms of polar coordinates are:

$$\begin{aligned} &\frac{\partial\sigma_{rr}}{\partial r} + \frac{1}{r}\frac{\partial\sigma_{r\theta}}{\partial\theta} + \frac{\partial\sigma_{zr}}{\partial z} + \frac{\sigma_{rr} - \sigma_{\theta\theta}}{r} + w_r = 0 \\ &\frac{\partial\sigma_{r\theta}}{\partial r} + \frac{1}{r}\frac{\partial\sigma_{\theta\theta}}{\partial\theta} + \frac{\partial\sigma_{\theta z}}{\partial z} + \frac{2\sigma_{r\theta}}{r} + w_\theta = 0 \\ &\frac{\partial\sigma_{zr}}{\partial r} + \frac{1}{r}\frac{\partial\sigma_{\theta z}}{\partial\theta} + \frac{\partial\sigma_{zz}}{\partial z} + \frac{\sigma_{zr}}{r} + w_z = 0 \end{aligned} \qquad (5.10)$$

Where $w_r$, $w_\theta$, $w_z$ denote the components of body force acting in the r, θ, z directions respectively so that:

$$\begin{bmatrix} w_r \\ w_\theta \\ w_z \end{bmatrix} = \begin{bmatrix} w_x\cos\theta + w_y\sin\theta \\ -w_x\sin\theta + w_y\cos\theta \\ w_z \end{bmatrix} \qquad (5.11)$$

## 5.5: Stress-Strain Relations

The concepts and relationships developed in the previous sections are applicable to any material. Different materials respond to application of forces in different ways and are said to have different constitutive behaviours. In this section the Hooke law's relationship between strains and stresses under three-dimensional conditions will be introduced. We assume that the material is isotropic and it behaves elastically. The relationships for the special cases of plane strain, plane stress and axi-symmetric conditions will be derived from the general relationship.

Consider a simple element in a structure. In general the element will not be in a state of zero stress. It will almost certainly be subjected to atmospheric pressure, however it may also be subjected to additional stresses. For example an element of concrete in a gravity dam, shown in Figure 5., will be subjected to stresses due to the self-weight of the material, or an element in a steel section may be stressed because of the rolling process or heat treatment used in its production.

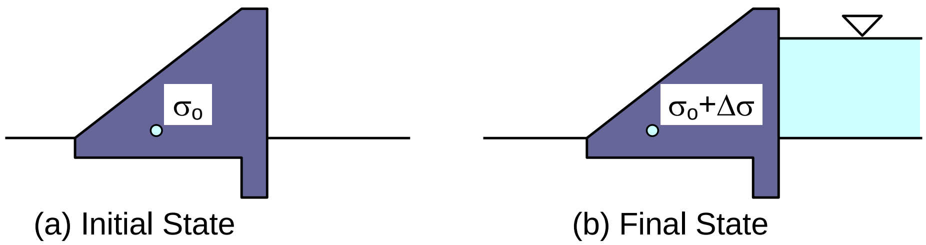



**Figure 5.5 Dam subjected to water loading**

If the element is subjected to an increase in stress it will respond by undergoing an increase in strain. Many materials, to sufficient accuracy, respond in the following simple manner:

i) The increment of strain is directly proportional to the increase in stress, i.e., if the increment in stress is doubled/halved the increment of strain is doubled/halved.

ii) The increment of strain due to the combined action of two sets of stress, e.g., a normal stress together with a shear stress, is the sum of the strains due to each of the sets of stress applied individually.

Such materials are said to be elastic solids and are said to respond elastically.

### *Isotropic elasticity*

An isotropic body is one in which the behaviour on an element within the body does not depend on the orientation of the element. Suppose an element of an isotropic elastic material shown in Figure 5. is subjected to increases in both normal stress and shear stress. From the previous discussion it can be seen that the response to this loading can be found by summing the responses of the six components of the loading as shown in Figure 5..

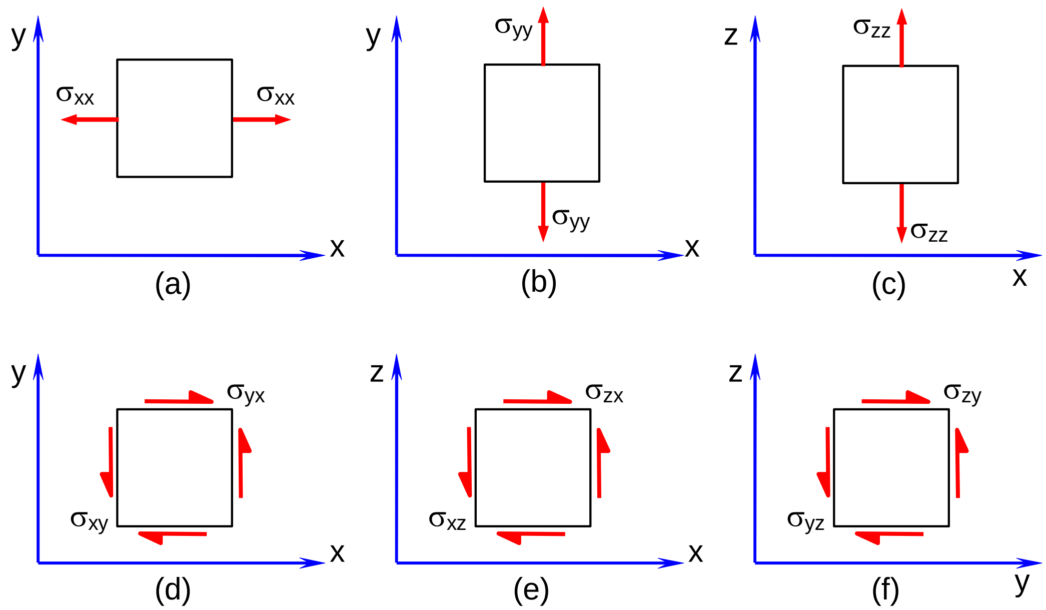



**Figure 5.6 Stress components**

Consider component (a) in Figure 5., it is clear from symmetry that the components of shear strain $\gamma_{yz}$, $\gamma_{zx}$, $\gamma_{xy}$ are all zero and also that $\varepsilon_{yy} = \varepsilon_{zz}$. Hooke's law for uniaxial behaviour states that:

$$\begin{aligned} \varepsilon_{xx} &= \frac{\sigma_{xx}}{E} \\ \varepsilon_{yy} &= -\nu\frac{\sigma_{xx}}{E} \\ \varepsilon_{zz} &= -\nu\frac{\sigma_{xx}}{E} \end{aligned} \tag{5.12}$$

where E and ν are material constants called Young's modulus and Poisson's ratio, respectively. A consideration of the component (b) leads to the conclusion that the only non-zero strain components are:

$$\begin{aligned} \varepsilon_{xx} &= -\nu\frac{\sigma_{yy}}{E} \\ \varepsilon_{yy} &= \frac{\sigma_{yy}}{E} \\ \varepsilon_{zz} &= -\nu\frac{\sigma_{yy}}{E} \end{aligned} \tag{5.13}$$

Similarly it is found that the response to the component (c) leads to the non-zero strains:

$$\varepsilon_{xx} = -\nu\frac{\sigma_{zz}}{E}$$

$$\varepsilon_{yy} = -\nu \frac{\sigma_{zz}}{E} \tag{5.14}$$

$$\varepsilon_{zz} = \frac{\sigma_{zz}}{E}$$

The response to the combined normal stresses is thus:

$$\varepsilon_{xx} = \frac{\sigma_{xx} - \nu(\sigma_{yy} + \sigma_{zz})}{E}$$

$$\varepsilon_{yy} = \frac{\sigma_{yy} - \nu(\sigma_{xx} + \sigma_{zz})}{E} \tag{5.15}$$

$$\varepsilon_{zz} = \frac{\sigma_{zz} - \nu(\sigma_{xx} + \sigma_{yy})}{E}$$

The shear strain increment, $\gamma_{xy}$ occurs due to an increment of shear stress $\sigma_{xy}$, as shown in Figure 5.(d), can be calculated by the following relation:

$$\gamma_{xy} = \frac{\sigma_{xy}}{G} \tag{5.16(a)}$$

where G is a material property called the shear modulus. Similarly, the responses to the stress changes (e) and (f) are:

$$\gamma_{yz} = \frac{\sigma_{yz}}{G} \tag{5.16(b)}$$

$$\gamma_{zx} = \frac{\sigma_{zx}}{G} \tag{5.16(c)}$$

The complete set of stress strain equations is given by Eq.(5.15) and Eq.(5.16).

Because of the isotropy of the material the stress-strain relations expressed in terms of another set of coordinate axes (X, Y, Z) should have precisely the same form as Eq.(5.15) and Eq.(5.16). This implies that the shear modulus must be related to Young's modulus and Poisson's ratio. The relationship between the shear modulus, Young's modulus and Poisson's ratio for an isotropic elastic material is:

$$G = \frac{E}{2(1+\nu)} \tag{5.17}$$

The complete expression for strain in terms of stress can be presented in a matrix format as:

$$\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & 1/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & -\nu/E & 1/E & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G \end{bmatrix} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{bmatrix} \tag{5.18}$$

It is often useful to be able to determine the volumetric strain and it is found that:

$$\varepsilon_v = \frac{\sigma_m}{K} \tag{5.19}$$

- Where $\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}$ is the volumetric strain
- $\sigma_m = \left(\sigma_{xx} + \sigma_{yy} + \sigma_{zz}\right)/3$ is called the mean stress
- $K = \dfrac{E}{3(1-2\nu)}$ is the bulk modulus.

***Expression for stress in terms of strain***

In many cases it is necessary to calculate the stresses resulting from application of a set of strains to an element. Clearly in such cases it is much more convenient to have an expression for stress in terms of strain. There is no difficulty in developing an expression for the increment of shear stress in terms of increment of shear strain from Eq.(5.16).

$$\sigma_{xy} = G\,.\,\gamma_{xy}$$

$$\sigma_{yz} = G\,.\,\gamma_{yz}$$

$$\sigma_{zx} = G\,.\,\gamma_{zx}$$

An expression for the increase in normal stress caused by the increase in normal strain may be found by writing the first of the relations in Eq. (5.12) to Eq. (5.14) in the form:

$$\varepsilon_{xx} = \left(\frac{1+\nu}{E}\right)\sigma_{xx} - \frac{\nu}{E}\left(\sigma_{xx} + \sigma_{yy} + \sigma_{zz}\right) \tag{5.20}$$

and then using Eq.(5.19) to show that:

$$\sigma_{xx} = \lambda\,\varepsilon_v + 2G\,\varepsilon_{xx}$$

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}$$

The quantity $\lambda$ is called the Lamé modulus. Similar expressions can be found for $\sigma_{yy}$ and $\sigma_{zz}$. Thus the complete expression for an increment of stress in terms of an increment of strain is:

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{bmatrix} = \begin{bmatrix} \lambda+2G & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2G & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2G & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} \tag{5.21}$$

or in a familiar matrix notation:

$$\boldsymbol{\sigma} = \mathbf{D}.\,\boldsymbol{\varepsilon} \tag{5.22}$$

where

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{bmatrix}, \ \boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix}, \ \mathbf{D} = \begin{bmatrix} \lambda+2G & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2G & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2G & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \tag{5.23}$$

**D** is called the matrix of elastic moduli.

It is perhaps worth observing at this stage that the matrix **D** in Eq.(5.22) is symmetric and positive definite. This is a general characteristic of elastic material and leads to the reciprocal theorem.

As stated before, in an isotropic material the form of the stress-strain relation is independent of the particular choice of coordinate system. Therefore, the relationships given in Eq.(5.18) and Eq.(5.21) can be written for cylindrical polar coordinates as:

$$\begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \gamma_{r\theta} \\ \gamma_{\theta z} \\ \gamma_{zr} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & 1/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & -\nu/E & 1/E & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G \end{bmatrix} \begin{bmatrix} \sigma_{rr} \\ \sigma_{\theta\theta} \\ \sigma_{zz} \\ \sigma_{r\theta} \\ \sigma_{\theta z} \\ \sigma_{zr} \end{bmatrix} \tag{5.24}$$

$$\begin{bmatrix} \sigma_{rr} \\ \sigma_{\theta\theta} \\ \sigma_{zz} \\ \sigma_{r\theta} \\ \sigma_{\theta z} \\ \sigma_{zr} \end{bmatrix} , = \begin{bmatrix} \lambda+2G & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2G & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2G & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \gamma_{r\theta} \\ \gamma_{\theta z} \\ \gamma_{zr} \end{bmatrix} \tag{5.25}$$

### *Plane stress, plane strain and axial symmetry*

There are certain circumstances in which it is not necessary to carry out a full three-dimensional analysis. One of these is shown schematically in Figure 5., where a uniform thin plate is subjected to edge loads parallel to the plane of the plate. Clearly the increments of stresses $\sigma_{zz}$, $\sigma_{yz}$, $\sigma_{xz}$ are all zero on both faces of the plate. It is found that to sufficient accuracy these are zero throughout the entire thickness of the plate. It thus follows that the increases in stresses within the body are completely specified by $\sigma_{xx}$, $\sigma_{yy}$, $\sigma_{xy}$. It can also be shown that to sufficient accuracy these stresses do not vary throughout the thickness of the plate and hence depend only on x, y but not on z. The stress strain relationship can then be written in the form:

$$\begin{aligned} \varepsilon_{xx} &= \frac{\left(\sigma_{xx} - \nu\sigma_{yy}\right)}{E} \\ \varepsilon_{yy} &= \frac{\left(\sigma_{yy} - \nu\sigma_{xx}\right)}{E} \\ \varepsilon_{zz} &= -\frac{\nu}{1-\nu}\left(\varepsilon_{xx} + \varepsilon_{yy}\right) \\ \gamma_{xy} &= \frac{\sigma_{xy}}{G} \end{aligned} \tag{5.26}$$

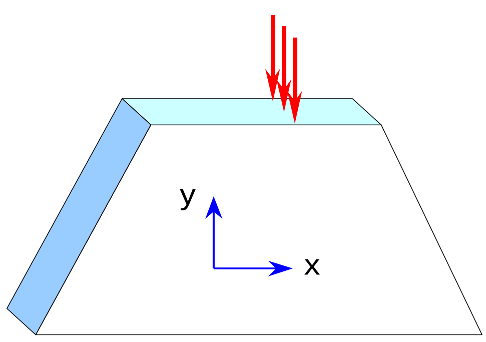



**Figure 5.7 Plane stress of a thin plate**

The increments in stresses can be expressed in terms of the increments in strains as:

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} E/\left(1-\nu^2\right) & E\nu/\left(1-\nu^2\right) & 0 \\ E\nu/\left(1-\nu^2\right) & E/\left(1-\nu^2\right) & 0 \\ 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} \tag{5.27}$$

The situation illustrated in Figure 5. and described mathematically by Eq.(5.26) and Eq.(5.27) is known as "plane stress".

The second case in which a similar simplification is possible is when a long prismatic body, such as the one shown schematically in Figure 5. is subjected to loads which are uniform along the length of the body and are in the plane perpendicular to the axis of the body.

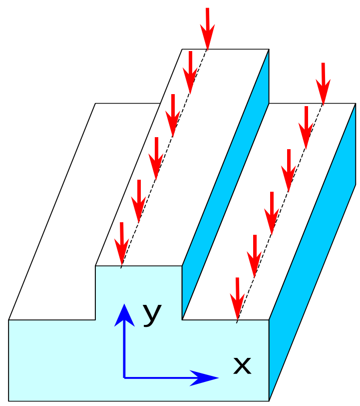



**Figure 5.8 Plane strain of a long prismatic body**

For these conditions it is found that the axial displacement $u_z$ is zero in the central portion of the body, that is the region remote from the ends, and the remaining two components of displacement are independent of z. This leads to the relations:

$$\begin{bmatrix} \gamma_{yz} \\ \gamma_{xz} \\ \varepsilon_{zz} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}$$

In terms of the remaining components of strain, it follows from Eq. (5.21)that

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} \lambda+2G & \lambda & 0 \\ \lambda & \lambda+2G & 0 \\ 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix}. \tag{5.28}$$

The only remaining non-zero component of stress can be found from Eq. (5.15) is:

$$\sigma_{zz} = \nu\left(\sigma_{xx} + \sigma_{yy}\right) \tag{5.29}$$

The situation illustrated in Figure 5. and described mathematically by Eq. (5.28)and Eq. (5.29) is known as "plane strain".

The third case for which another simplified form of stress-strain relationship can be presented includes bodies of revolution which are subjected to axi-symmetric loading. These bodies constitute another important category of structures which are essentially two dimensional in nature. Such structures are called axi-symmetric continua.
A typical axi-symmetric body is shown in Figure 5.. The z-axis is the vertical axis about which the geometry and loading is symmetric, the r axis is radially outwards and θ is the polar angle.

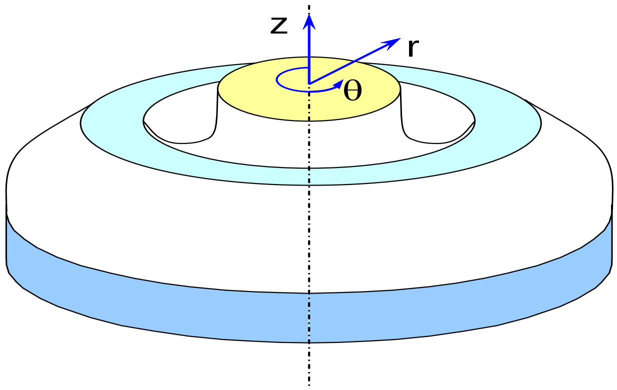



**Figure 5.9 Axis-symmetric body**

The non-zero displacement components are in z and r directions only and do not vary with θ, since the prescription of symmetry indicates that the tangential component of displacement is zero everywhere. Therefore, the vector of strain components for axi-symmetric continua can be derived from Eq. (5.4) as:

$$\varepsilon = \begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \gamma_{rz} \end{bmatrix} = \begin{bmatrix} \partial u_r/\partial r \\ u_r/r \\ \partial u_z/\partial z \\ \partial u_z/\partial r + \partial u_r/\partial z \end{bmatrix}$$

The corresponding vector of stresses is:

$$\sigma = \left[\ \sigma_{rr},\ \sigma_{\theta\theta},\ \sigma_{zz}, \sigma_{rz}\right]^T$$

The stress-strain relationship for axi-symmetric continua consisting of isotropic materials can be found from Equation (5.25) as:

$$\begin{bmatrix} \sigma_{rr} \\ \sigma_{\theta\theta} \\ \sigma_{zz} \\ \sigma_{rz} \end{bmatrix}, = \begin{bmatrix} \lambda+2G & \lambda & \lambda & 0 \\ \lambda & \lambda+2G & \lambda & 0 \\ \lambda & \lambda & \lambda+2G & 0 \\ 0 & 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \gamma_{rz} \end{bmatrix} \tag{5.30}$$

The situation illustrated in Figure 5. and described by Eq. (5.30) is known as "axial symmetry".

### *Problem 1*

In a geological site a layer of silt was found which is inclined at 20 to the horizontal. The global and local coordinate systems were set up as shown in Figure 5.6. At a point on the silt layer the vertical stress is 300 kPa and the horizontal stress is 250 kPa. Recalling that tensile normal stresses are considered to be positive, the stress tensor (in the global system of coordinates) is;

$$\sigma = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} = \begin{bmatrix} -250 & 0 & 0 \\ 0 & -300 & 0 \\ 0 & 0 & -250 \end{bmatrix} \quad (5.31)$$

The unit vector normal to the surface is:

$$n = \begin{pmatrix} \sin(20^o) \\ \cos(20^o) \\ 0 \end{pmatrix}$$

Hence the traction acting on the seam is given by:

$$\begin{bmatrix} T_x \\ T_y \\ T_z \end{bmatrix} = \begin{bmatrix} -250 & 0 & 0 \\ 0 & -300 & 0 \\ 0 & 0 & -250 \end{bmatrix} \begin{bmatrix} 0.3420 \\ 0.9397 \\ 0 \end{bmatrix} = \begin{bmatrix} -85.505 \\ -281.908 \\ 0 \end{bmatrix} (\text{kPa}) \quad (5.32)$$

The components of traction normal and tangential to the seam are given by:

$$T_n = -0.3420 \times 85.505 - 0.9397 \times 281.908 = -294.15 \text{ kPa}$$

$$T_t = -0.9397 \times 85.505 + 0.3420 \times 281.908 = 16.07 \text{ kPa}$$

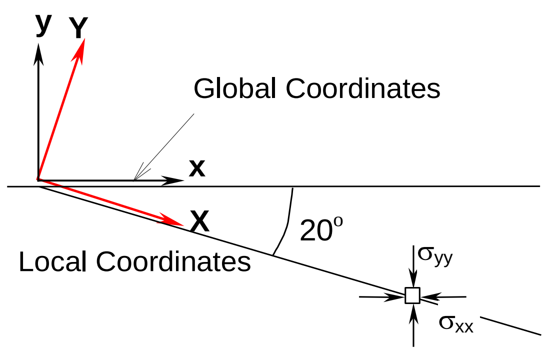



**Figure 5.10: Local and global coordinates**

### *Problem 2*

In a plane system the stress in global coordinates is:

$$\boldsymbol{\sigma} = \begin{bmatrix} 80.0000 & 34.6410 \\ 34.6410 & 40.000 \end{bmatrix} (\text{MPa}) \quad (5.33)$$

Calculate the traction on a plane making an angle of 120° with the x-axis.

*Answer:* $T_x = 86.6025\ MPa$ $T_y = 50\ MPa$

### *Problem 3*

A local set of coordinates with the X axis inclined at 30° to the axis and the Y axis inclined at 120° to the x axis. If the stresses in the global (x, y) system are given by Eq. , show that the stress components in the local (X, Y) coordinate system can be given by

$$\mathbf{\Sigma} = \begin{bmatrix} 100 & 0 \\ 0 & 20 \end{bmatrix} \ \textit{(Mpa)}$$

# Chapter 6
# INTERPOLATION

In the finite element method the body to be analysed is broken up into a number of elements that join with each other at a discrete number of points or nodes. The method is an approximate one and so it is not usual to determine the displacement of every point in every element. In fact the displacement is only evaluated at a number of nodes and the displacement at any other point is inferred from these nodal values by interpolation.

In the previous section a general procedure for calculation of the stiffness matrix of a finite element was explained. One of the major steps in the procedure was the establishment of the relationship between the strains or displacements within the element and the nodal displacements. It was shown that the value of a quantity at any point within an element can be related to its nodal values using the shape functions. The aim of this section is to present a general method for derivation of the shape functions for various finite elements.

## 6.1: One-Dimensional Interpolation

A polynomial interpolation is used in derivation of the stiffness matrix for most of the finite elements. The use of polynomial functions allows high order elements to be formulated. In this section linear and quadratic interpolation functions are discussed.

### *Linear interpolation*

Consider that a continuous function w(x) is to be approximated over the interval $x_1 \leq x \leq x_2$ using a linear function (Figure 6.1). The values of the function at point 1 and 2 are $W_1$ and $W_2$, respectively. Assume that the function w(x) can be approximated by a linear function such as:

$$w(x) = a_1 + a_2 x \tag{6.1}$$

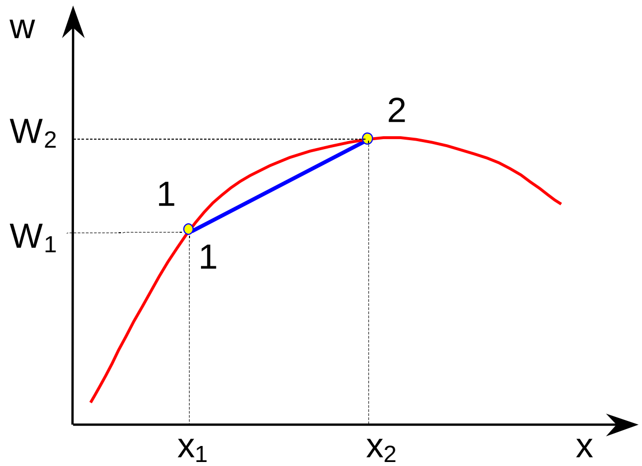



**Figure 6.1 Linear interpolation**

where $a_1$ and $a_2$ are unknown coefficients of the function. The coefficients can be determined from the known values at points 1 and 2.

$$W_1 = w(x_1) = a_1 + a_2\, x_1$$
$$W_2 = w(x_2) = a_1 + a_2\, x_2$$

This set of equations can be solved for the unknown coefficients:

$$a_1 = \frac{W_1 x_2 - W_2 x_1}{x_2 - x_1} \quad , \; a_2 = \frac{W_2 - W_1}{x_2 - x_1}$$

Therefore the value of the function w at any point x within the interval $x_1 \le x \le x_2$ can be expressed as:

$$w(x) = \frac{W_1 x_2 - W_2 x_1}{x_2 - x_1} + \frac{W_2 - W_1}{x_2 - x_1}\, x$$

Rearranging the above equation results in:

$$w(x) = \frac{x_2 - x}{x_2 - x_1}\, W_1 + \frac{x - x_1}{x_2 - x_1}\, W_2$$

or:

$$w(x) = N_1(x)\; W_1 + N_2(x)\; W_2 \qquad (6.2)$$

where $N_1(x) = \frac{x_2 - x}{x_2 - x_1}$ and $N_2(x) = \frac{x - x_1}{x_2 - x_1}$ are called the shape functions.

The shape functions depend only on the geometry of the nodal points and the type of the interpolation function used. The shape functions $N_1(x)$ and $N_2(x)$ vary linearly between $x_1$ and $x_2$ as shown in Figure 6.2. Note that the value of the shape function $N_1(x)$ is 1 at point 1 and zero at point 2. Similarly the value of the shape function $N_2(x)$ is 1 at point 2 and zero at point 1.

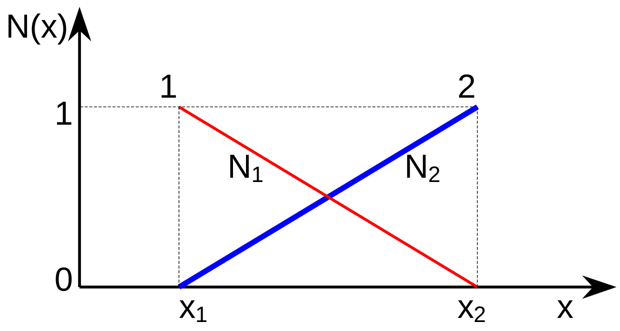



**Figure 6.2 Linear shape functions**

*Quadratic interpolation*

Consider that the value of a continuous function w(x) is to be approximated over the interval $x_1 \le x \le x_3$ using a quadratic function (Figure 6.3). The values of the function at point 1, 2 and 3 are $W_1$, $W_2$ and $W_3$, respectively.

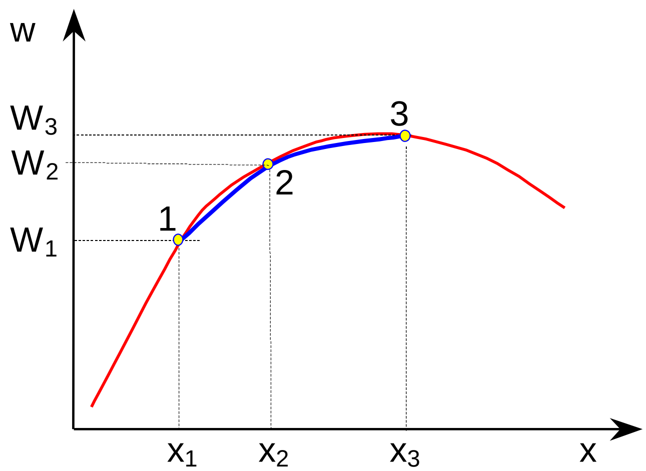



**Figure 6.3 Quadratic interpolation**

The function w(x) can be approximated by a polynomial quadratic function such as:

$$w(x) = a_1 + a_2\, x + a_3\, x^2 \tag{6.3}$$

Where $a_1$ to $a_3$ are unknown coefficients of the function. The coefficients can be determined from the known values at points 1, 2 and 3.

$$W_1 = w(x_1) = a_1 + a_2\, x_1 + a_3\, x_1^2$$

$$W_2 = w(x_2) = a_1 + a_2\, x_2 + a_3\, x_2^2$$

$$W_3 = w(x_3) = a_1 + a_2\, x_3 + a_3\, x_3^2$$

This set of equations can be solved for the unknown coefficients:

$$a_1 = -\frac{(x_2 - x_3)x_2x_3W_1 + (x_3 - x_1)x_3x_1W_2 + (x_1 - x_2)x_1x_2W_3}{(x_1 - x_2)(x_2 - x_3)(x_3 - x_1)}$$

$$a_2 = \frac{(x_2^2 - x_3^2)W_1 + (x_3^2 - x_1^2)W_2 + (x_1^2 - x_2^2)W_3}{(x_1 - x_2)(x_2 - x_3)(x_3 - x_1)}$$

$$a_3 = -\frac{(x_2 - x_3)W_1 + (x_3 - x_1)W_2 + (x_1 - x_2)W_3}{(x_1 - x_2)(x_2 - x_3)(x_3 - x_1)}$$

Substituting $a_1$, $a_2$ and $a_3$ into Eq. (6.3) results in a quadratic interpolation as a function of nodal values:

$$w(x) = N_1(x)\, W_1 + N_2(x)\, W_2 + N_3(x)\, W_3 \qquad (6.4)$$

Where $N_1(x) = \frac{(x - x_2)(x - x_3)}{(x_1 - x_2)(x_1 - x_3)}$, $N_2(x) = \frac{(x - x_1)(x - x_3)}{(x_2 - x_1)(x_2 - x_3)}$, $N_3(x) = \frac{(x - x_1)(x - x_2)}{(x_3 - x_1)(x_3 - x_2)}$ are the quadratic shape functions.

The quadratic shape functions vary quadratically between $x_1$ and $x_3$ as shown in Figure 6.4. The value of the shape function $N_1(x)$ is1 at point 1 and zero at points 2 and 3. Similarly the value of the shape function $N_2(x)$ is 1 at point 2 and zero at points 1 and 3, and the value of the shape function $N_3(x)$ is 1 at point 3 and zero at points 1 and 2.

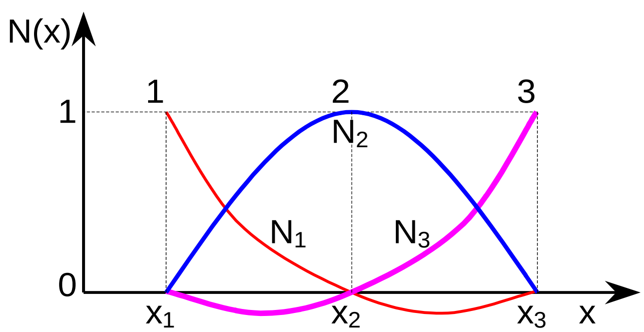



**Figure 6.4 Quadratic shape functions**

The method used above for calculation of the linear of quadratic shape functions can be applied to calculate higher order interpolation functions. However, for higher order polynomials it is difficult to find the unknown coefficients. An alternative method is presented in the next section that is applicable to all types of one or two-dimensional interpolation functions.

## 6.2: General Procedure for Derivation of Shape Functions

Suppose an element has m nodes and the values of some quantity of interest (w), such as displacement, head, temperature, are known at each of the nodes. It is assumed that within the element the variation of w at position x can be approximated by a polynomial expression:

$$w(x) = a_1 f_1(x) + a_2 f_2(x) + \ldots + a_k f_k(x) + \ldots + a_m f_m(x) \tag{6.5}$$

Where $a_k$ are polynomial coefficients and $f_k$ are known functions of the position x. Eq. (6.5) can be written in matrix format as:

$$w(x) = a^T . f(x) = f^T(x) . a \tag{6.6}$$

Where $a=[a_1, a_2, \ldots, a_k, .. a_m]^T$ and $\mathbf{f}(x) = [f_1(x), f_2(x), \ldots, f_k(x), \ldots, f_m(x)]^T$.
Suppose that the element nodes are located at the points $x_1, x_2, \ldots, x_m$. At the '$k^{th}$' node the value of the quantity w is:

$$W_k = a_1 f_1(x_k) + a_2 f_2(x_k) + \ldots a_k f_k(x_k) + \ldots + a_m f_m(x_k) \tag{6.7}$$

Eq. (6.7) holds at each of the m nodes. These equations may be written in matrix form as follows:

$$W = C . a \tag{6.8}$$

where

$$a = \begin{bmatrix} a_1 \\ a_2 \\ \vdots \\ a_k \\ \vdots \\ a_m \end{bmatrix}, \quad W = \begin{bmatrix} W_1 \\ W_2 \\ \vdots \\ W_k \\ \vdots \\ W_m \end{bmatrix}, \quad C = \begin{bmatrix} f_1(x_1) & f_2(x_1) & \ldots & f_K(x_1) & \ldots & f_m(x_1) \\ f_1(x_2) & f_2(x_2) & \ldots & f_k(x_2) & \ldots & f_m(x_2) \\ \ldots & \ldots & \ldots & \ldots & \ldots & \ldots \\ f_1(x_k) & f_2(x_k) & \ldots & f_k(x_k) & \ldots & f_m(x_k) \\ \ldots & \ldots & \ldots & \ldots & \ldots & \ldots \\ f_1(x_m) & f_2(x_m) & \ldots & f_k(x_m) & \ldots & f_m(x_m) \end{bmatrix}.$$

The solution of Eq. (6.8) is:

$$a = C^{-1} W \tag{6.9}$$

When **a** from Eq. (6.9) is substituted into Eq. (6.6) it is found that the quantity of interest can be expressed in the form of:

$$w(x) = f^T(x).C^{-1}.W = N^T(x)\, W \tag{6.10}$$

where

$$\mathbf{N}^T(x)=\mathbf{f}^T(x).\mathbf{C}^{-1} = [\ N_1(x),\ N_2(x),\ \ldots.,\ N_k(x),\ \ldots.\ N_m(x)\ ]$$

is the vector of shape functions.

If Eq. (6.10) for w(x) is written out in full it takes the form of:

$$w(x) = W_1N_1(x) + W_2N_2(x) + \ldots. + W_kN_k(x), \ldots. + W_mN_m(x)$$

The above equation expresses the value of the quantity w at any position x in terms of the m nodal values $W_1$ to $W_m$ and the shape functions $N_1(x)$ to $N_m(x)$ which can be determined from Eq. (6.10) Assume that the inverse of the matrix **C** is:

$$C^{-1} = \begin{bmatrix} \gamma_{11} & \cdots & \gamma_{1m} \\ \vdots & \cdots & \vdots \\ \gamma_{m1} & \cdots & \gamma_{mm} \end{bmatrix}$$

Where the coefficients $\gamma_{ij}$ are known values. Thus the vector of shape functions is:

$$N^T(x) = f^T(x)C^{-1} = \begin{bmatrix} f_1(x)\gamma_{11} + \cdots + f_m(x)\gamma_{m1} \\ \vdots \\ f_1(x)\gamma_{1m} + \cdots + f_m(x)\gamma_{mm} \end{bmatrix}^T$$

Therefore each of the shape functions $N_k$ is given by:

$$N_k = f_1(x)\gamma_{1k} + f_2(x)\gamma_{2k} + \ldots + f_k(x)\gamma_{kk} + \ldots. + f_m(x)\gamma_{mk}$$

***Example 6.1***

The linear shape functions for a one-dimensional two-noded element can be found using the generalized method. The function used for approximation of w at position x is:

$$w(x) = a_1 + a_2\,x = [1, x]\,.\,[a_1, a_2]^T = f^T(x)\,.\,a$$

Where $\mathbf{f}^T(x) = [f_1(x), f_2(x)] = [1, x]$ and $\mathbf{a}=[a_1, a_2]^T$. Then matrix **C** can be written as:

$$C = \begin{bmatrix} 1 & x_1 \\ 1 & x_2 \end{bmatrix}$$

$$C^{-1} = \begin{bmatrix} \frac{x_2}{x_2 - x_1} & -\frac{x_1}{x_2 - x_1} \\ -\frac{1}{x_2 - x_1} & \frac{1}{x_2 - x_1} \end{bmatrix}$$

Therefore the vector of the shape functions is calculated as:

$$N^T(x) = f^T(x) . C^{-1} = [1,\ x] \begin{bmatrix} \frac{x_2}{x_2 - x_1} & -\frac{x_1}{x_2 - x_1} \\ -\frac{1}{x_2 - x_1} & \frac{1}{x_2 - x_1} \end{bmatrix} = \left[ \left( \frac{x_2}{x_2 - x_1} - \frac{x}{x_2 - x_1} \right), \left( -\frac{x_1}{x_2 - x_1} + \frac{x}{x_2 - x_1} \right) \right]$$

And the shape functions for linear one-dimensional elements are:

$$N_1(x) = \frac{x_2 - x}{x_2 - x_1} \text{ and } N_1(x) = \frac{x - x_1}{x_2 - x_1}.$$

***Example 6.2***

The quadratic shape functions for a one-dimensional three-noded element can be found as follows.

$$w(x) = a_1 + a_2 x + a_3 x^2 = [1, x, x^2] . [a_1, a_2, a_3]^T = f^T(x) . a$$

Where $\mathbf{f}^{\mathbf{T}}(x) = [f_1(x), f_2(x), f_3(x)] = [1, x, x^2]$ and $\mathbf{a}=[a_1, a_2, a_3]^T$. Then matrix $\mathbf{C}$ is:

$$C = \begin{bmatrix} 1 & x_1 & x_1^2 \\ 1 & x_2 & x_2^2 \\ 1 & x_3 & x_3^2 \end{bmatrix}$$

$$C^{-1} = \begin{bmatrix} \frac{x_2 x_3}{(x_1 - x_2)(x_1 - x_3)} & \frac{x_1 x_3}{(x_2 - x_1)(x_2 - x_3)} & \frac{x_1 x_2}{(x_3 - x_1)(x_3 - x_2)} \\ -\frac{x_2 + x_3}{(x_1 - x_2)(x_1 - x_3)} & -\frac{x_1 + x_3}{(x_2 - x_1)(x_2 - x_3)} & -\frac{x_1 + x_2}{(x_3 - x_1)(x_3 - x_2)} \\ \frac{1}{(x_1 - x_2)(x_1 - x_3)} & \frac{1}{(x_2 - x_1)(x_2 - x_3)} & \frac{1}{(x_3 - x_1)(x_3 - x_2)} \end{bmatrix}$$

Therefore the vector of the shape functions for quadratic one-dimensional elements is calculated as:

$$N^T(x) = f^T(x)C^{-1} = \left[ \left( \frac{(x - x_2)(x - x_3)}{(x_1 - x_2)(x_1 - x_3)} \right), \left( \frac{(x - x_1)(x - x_3)}{(x_2 - x_1)(x_2 - x_3)} \right), \left( \frac{(x - x_1)(x - x_2)}{(x_3 - x_1)(x_3 - x_2)} \right) \right]$$

## 6.3: Two-Dimensional Interpolation

The general method explained in the previous section can be used to derive the shape functions for two-dimensional elements. The shape functions for linear triangles and rectangles are calculated here. The quadratic shape functions for a triangular element are also derived for a specific case.

### *Linear triangles*

Consider the quantity w is known at 3 nodes of a triangular element having its vertices at nodes 1, 2 and 3 as shown in Figure 6.5 The coordinates of nodes 1 to 3 are $(x_1,y_1)$, $(x_2,y_2)$, and $(x_3,y_3)$ respectively and the values of w at nodes are $W_1$, $W_2$ and $W_3$. If it is assumed that within the element the variation of w is linear with respect to x and y, then the value of w at position (x, y) can be approximated by a simple polynomial expression such as:

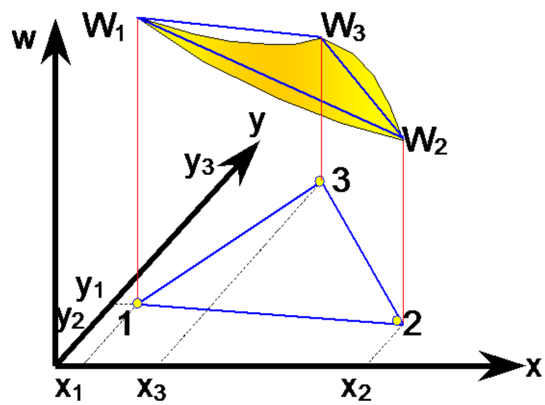



**Figure 6.5 Linear triangular element**

$$w(x, y) = a_1 + a_2\,x + a_3\,y \tag{6.11}$$

or:

$$w(x, y) = [1, x, y]\,.\,[a_1, a_2, a_3]^T = f^T(x, y)\,.\,a$$

The values of w are known at the nodes. Therefore, Eq.(6.11) can be written for all the nodes by substituting the coordinates of the nodes into Eq. (6.11):

$$W_1 = a_1 + a_2\,x_1 + a_3\,y_1$$

$$W_2 = a_1 + a_2\,x_2 + a_3\,y_2$$

$$W_3 = a_1 + a_2\,x_3 + a_3\,y_3$$

or:

$$\begin{bmatrix} W_1 \\ W_2 \\ W_3 \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix} \quad \text{or} \quad W = C\,.\,a$$

The quantity w(x, y) can now be expressed in the form of:

$$w(x, y) = f^T(x, y).C^{1}.W = N^T(x, y)W$$

$$C^{-1} = \frac{1}{2\Delta}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

where $2\Delta = \det[C] = (x_2y_3 - x_3y_2) - (x_1y_3 - x_3y_1) + (x_1y_2 - x_2y_1) = 2\times$area of triangle.

The shape functions can be found as:

$$N^T(x,y) = f^T(x,y)C^{-1} = \frac{1}{2\Delta}[1,\ x,\ y]\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

$$N(x,y) = \begin{bmatrix} N_1(x, y) \\ N_2(x, y) \\ N_3(x, y) \end{bmatrix} = \begin{bmatrix} \dfrac{(x_2y_3 - x_3y_2) + x(y_2 - y_3) + y(x_3 - x_2)}{2\Delta} \\ \dfrac{(x_3y_1 - x_1y_3) + x(y_3 - y_1) + y(x_1 - x_3)}{2\Delta} \\ \dfrac{(x_1y_2 - x_2y_1) + x(y_1 - y_2) + y(x_2 - x_1)}{2\Delta} \end{bmatrix} \tag{6.12}$$

$$w(x, y) = N^T(x, y).W = N_1(x, y).W_1 + N_2(x, y).W_2 + N_3(x, y).W_3$$

The correctness of the shape functions may be verified by checking the following conditions:

1) $\sum N_i(x, y) = 1$ at every point within the element

2) $N_i(x, y) = 1$ at node i where $x = x_i$ and $y = y_i$
$N_i(x, y) = 0$ at all nodes k where $k \neq i$

It can be shown that the sum of all the shape functions is equal to 1, so that condition 1 is satisfied. Condition 2 is also true for all the shape functions. For example, at node 1, $x = x_1$ and $y = y_1$:

$$N_1(x_1, y_1) = \frac{(x_2y_3 - x_3y_2) + x_1(y_2 - y_3) + y_1(x_3 - x_2)}{(x_2y_3 - x_3y_2) - (x_1y_3 - x_3y_1) + (x_1y_2 - x_2y_1)} = 1$$

At node 2, $x = x_2$ and $y = y_2$:

$$N_1(x_2, y_2) = \frac{(x_2y_3 - x_3y_2) + x_2(y_2 - y_3) + y_2(x_3 - x_2)}{2\Delta} = 0$$

At node 3, $x = x_3$ and $y = y_3$:

$$N_1(x_3, y_3) = \frac{(x_2 y_3 - x_3 y_2) + x_3(y_2 - y_3) + y_3(x_3 - x_2)}{2\Delta} = 0$$

### *Example 6.3*

Consider a seepage analysis and suppose that the head has been determined at 3 vertices (nodes) of a triangular element. The coordinates (x, y) of the nodes and the value of the head (h) are shown in the table below:

| **Node** | **x (m)** | **y (m)** | **H (m)** |
|---|---|---|---|
| 1 | 0.4 | 0.6 | 1.832 |
| 2 | 4.0 | 1.4 | 66.76 |
| 3 | 1.4 | 3.0 | - 8.968 |

If it is assumed that the head may be approximated linearly throughout the element by the simple expression:

$$h(x, y) = a_1 + a_2\, x + a_3\, y$$

Determine the head at point $x_o = 2.0$, $y_o = 1.5$.

The variation of h can be approximated as:

$$h(x, y) = N^T(x, y).H = N_1(x, y).H_1 + N_2(x, y).H_2 + N_3(x, y).H_3$$

Where H=[1.832, 66.76, -8.968]$^T$ is the vector of the known nodal head values.
The shape functions for the triangular element can be calculated from Eq. (6.12)

$$N(x,y) = \begin{bmatrix} N_1(x, y) \\ N_2(x, y) \\ N_3(x, y) \end{bmatrix} = \begin{bmatrix} \dfrac{(x_2 y_3 - x_3 y_2) + x(y_2 - y_3) + y(x_3 - x_2)}{2\Delta} \\ \dfrac{(x_3 y_1 - x_1 y_3) + x(y_3 - y_1) + y(x_1 - x_3)}{2\Delta} \\ \dfrac{(x_1 y_2 - x_2 y_1) + x(y_1 - y_2) + y(x_2 - x_1)}{2\Delta} \end{bmatrix} = \begin{bmatrix} +1.281 - 0.204x - 0.332y \\ -0.046 + 0.306x - 0.128y \\ -0.235 - 0.102x + 0.459y \end{bmatrix}$$

The values of the shape functions at point $x_o = 2.0$, $y_o = 1.5$ are:

$$N(x_o, y_o) - N(2.0,\ 1.5) - \begin{bmatrix} N_1(x_o, y_o) \\ N_2(x_o, y_o) \\ N_3(x_o, y_o) \end{bmatrix} - \begin{bmatrix} +1.281 - 0.204 \times 2.0 - 0.332 \times 1.5 \\ -0.046 + 0.306 \times 2.0 - 0.128 \times 1.5 \\ -0.235 - 0.102 \times 2.0 + 0.459 \times 1.5 \end{bmatrix} - \begin{bmatrix} 0.375 \\ 0.375 \\ 0.25 \end{bmatrix}$$

$$h(x_o, y_o) = N^T(x_o, y_o).H = N_1(x_o, y_o).H_1 + N_2(x_o, y_o).H_2 + N_3(x_o, y_o).H_3$$

$$h(2.0, 1.5) = \{0.375,\ 0.375,\ 0.25\}.\{1.832,\ 66.76,\ -8.968\}^T = 23.48\,m$$

The shape functions can be used to obtain other quantities of interest, for example the hydraulic gradients:

$$i_x = \frac{\partial h(x, y)}{\partial x} = \frac{\partial\left(N^T(x, y).H\right)}{\partial x} = \frac{\partial N^T(x, y)}{\partial x}.H = \frac{\partial N_1(x, y)}{\partial x}H_1 + \frac{\partial N_2(x, y)}{\partial x}H_2 + \frac{\partial N_3(x, y)}{\partial x}H_3$$

The derivatives of the shape functions with respect to x and y are:

$$\frac{\partial N(x,y)}{\partial x} = \begin{bmatrix} \partial N_1(x, y)/\partial x \\ \partial N_2(x, y)/\partial x \\ \partial N_3(x, y)/\partial x \end{bmatrix} = \begin{bmatrix} -0.204 \\ +0.306 \\ -0.102 \end{bmatrix}, \text{ and } \frac{\partial N(x,y)}{\partial y} = \begin{bmatrix} \partial N_1(x, y)/\partial y \\ \partial N_2(x, y)/\partial y \\ \partial N_3(x, y)/\partial y \end{bmatrix} = \begin{bmatrix} -0.332 \\ -0.128 \\ +0.459 \end{bmatrix}$$

It can be seen that the derivatives of the shape functions, which were derived according to a linear interpolation function, have constant values over the entire area of the triangular element. Therefore, at point $x_o = 2.0$, $y_o = 1.5$, or at any other point within the triangle, the hydraulic gradients with respect to x and y are:

$$i_x = -0.204 \times 1.832 + 0.306 \times 66.76 + 0.102 \times 8.968 = 20.978 \text{ m/m}$$
$$i_y = -0.332 \times 1.832 - 0.128 \times 66.76 - 0.459 \times 8.968 = -13.241 \text{ m/m}$$

### *Linear rectangles*

The shape functions for a rectangular element are derived in this section by a direct method as well as by the general procedure explained in the previous section.

A rectangular element which lies in the x, y plane and has sides of length A and B has its nodes at $P_1(0, 0)$, $P_2(A, 0)$, $P_3(A, B)$, $P_4(0, B)$. Suppose that throughout the element the variation of w can be approximated as follows:

$$w(x, y) = a_1 + a_2\frac{x}{A} + a_3\frac{y}{B} + a_4\frac{xy}{AB} \tag{6.13}$$

If w is evaluated at the 4 nodes of the rectangular element it follows that:

$$W_1 = a_1$$
$$W_2 = a_1 + a_2$$
$$W_3 = a_1 + a_2 + a_3 + a_4$$
$$W_4 = a_1 + a_3 \tag{6.14}$$

Solving Eq. (6.14) for the coefficients $a_1, \cdots, a_4$ results in:

$$a_1 = W_1$$

$$a_2 = W_2 - W_1$$
$$a_3 = W_4 - W_1 \qquad (6.15)$$
$$a_4 = W_1 + W_3 - W_2 - W_4$$

If Eq. (6.15) are substituted in Eq. (6.13) it is found that:

$$w(x,y) = W_1 + (W_2 - W_1)\frac{x}{A} + (W_4 - W_1)\frac{y}{B} + (W_1 + W_3 - W_2 - W_4)\frac{xy}{AB}$$

Or upon collecting terms:

$$w(x,y) = W_1(1 - \frac{x}{A} - \frac{y}{B} + \frac{xy}{AB}) + W_2(\frac{x}{A} - \frac{xy}{AB}) + W_3(\frac{xy}{AB}) + W_4(\frac{y}{B} - \frac{xy}{AB})$$

This may be rewritten:

$$w(x,y) = W_1 N_1(x,y) + W_2 N_2(x,y) + W_3 N_3(x,y) + W_4 N_4(x,y)$$

Where $N_k$ are the shape functions and in this case have the explicit expressions:

$$\begin{aligned}
N_1(x,y) &= 1 - \frac{x}{A} - \frac{y}{B} + \frac{xy}{AB} = (1-\frac{x}{A})(1-\frac{y}{B}) \\
N_2(x,y) &= \frac{x}{A} - \frac{xy}{AB} = \frac{x}{A}(1-\frac{y}{B}) \\
N_3(x,y) &= \frac{xy}{AB} \\
N_4(x,y) &= \frac{y}{B} - \frac{xy}{AB} = \frac{y}{B}(1-\frac{x}{A})
\end{aligned} \qquad (6.16)$$

The correctness of the shape functions can be checked; each of the shape functions $N_i$ takes the value 1 at the node i but zero at all other nodes. This is a general property of shape functions and ensures that $w = W_i$ at each of the nodes i. The sum of all the shape functions at any arbitrary point (x,y) is equal to 1.

The general procedure explained in the previous section can also be followed to derive the shape functions for the rectangular element.

$$w(x,y) = a_1 + a_2\frac{x}{A} + a_3\frac{y}{B} + a_4\frac{xy}{AB} = \left(1,\ \frac{x}{A},\ \frac{y}{B},\ \frac{xy}{AB}\right).(a_1,\ a_2,\ a_3,\ a_4)^T = f^T(x,y).a$$

Therefore:

$$C = \begin{bmatrix} 1 & x_1/A & y_1/B & x_1y_1/AB \\ 1 & x_2/A & y_2/B & x_2y_2/AB \\ 1 & x_3/A & y_3/B & x_3y_3/AB \\ 1 & x_4/A & y_4/B & x_4y_4/AB \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 1 & 1 & 0 & 0 \\ 1 & 1 & 1 & 1 \\ 1 & 0 & 1 & 0 \end{bmatrix}$$

$$C^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ -1 & 0 & 0 & 1 \\ 1 & -1 & 1 & -1 \end{bmatrix}$$

Then the shape functions can be derived from:

$$N^T(x, y) = f^T(x, y)C^{-1} = \left(1, \ \frac{x}{A}, \ \frac{y}{B}, \ \frac{xy}{AB}\right) \cdot \begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ -1 & 0 & 0 & 1 \\ 1 & -1 & 1 & -1 \end{bmatrix}$$

As:

$$N_1(x,y) = 1 - \frac{x}{A} - \frac{y}{B} + \frac{xy}{AB} = (1 - \frac{x}{A})(1 - \frac{y}{B})$$

$$N_2(x,y) = \frac{x}{A} - \frac{xy}{AB} = \frac{x}{A}(1 - \frac{y}{B})$$

$$N_3(x,y) = \frac{xy}{AB}$$

$$N_4(x,y) = \frac{y}{B} - \frac{xy}{AB} = \frac{y}{B}(1 - \frac{x}{A})$$

The above expressions for the shape functions are identical to those obtained previously, Equation (6.16)

### *Quadratic triangle*

The shape functions for a 6-noded triangular element are derived here for a specific case using the general procedure explained in section 3.2.

Suppose that in a seepage analysis the head has been determined at 6 nodes of a triangular element having its vertices at nodes 1, 3, 5. Nodes 2, 4 and 6 are located at the mid-side of the triangle. The coordinates (x, y) of the nodes and the value of the head (h) are shown in the table below:

| **Node** | **x (m)** | **y (m)** | **H (m)** |
|---|---|---|---|
| 1 | 0.4 | 0.6 | 1.832 |
| 2 | 2.2 | 1.0 | 22.968 |
| 3 | 4.0 | 1.4 | 66.760 |
| 4 | 2.7 | 2.2 | 27.488 |
| 5 | 1.4 | 3.0 | - 8.968 |
| 6 | 0.9 | 1.8 | - 1.248 |

Assume that the head may be approximated throughout the element by a polynomial expression:

$$h(x, y) = a_1 + a_2\, x + a_3\, y + a_4\, x^2 + a_5\, x\, y + a_6\, y^2 = f^T(x, y).a$$

$$h(x,y) = \left[1, x, y, x^2, xy, y^2\right].\left[a_1, a_2, a_3, a_4, a_5, a_6\right]^T = f^T(x,y).\, a$$

Therefore matrix **C** is calculated as:

$$C = \begin{bmatrix} 1.00 & 0.40 & 0.60 & 0.16 & 0.24 & 0.36 \\ 1.00 & 2.20 & 1.00 & 4.84 & 2.20 & 1.00 \\ 1.00 & 4.00 & 1.40 & 16.00 & 5.60 & 1.96 \\ 1.00 & 2.70 & 2.20 & 7.29 & 5.94 & 4.84 \\ 1.00 & 1.40 & 3.00 & 1.96 & 4.20 & 9.00 \\ 1.00 & 0.90 & 1.80 & 0.81 & 1.62 & 3.24 \end{bmatrix}$$

$$C^{-1} = \begin{bmatrix} 2.00 & -0.24 & 0.05 & 0.04 & 0.34 & -1.20 \\ -0.84 & 1.61 & -0.36 & -0.27 & 0.20 & -0.33 \\ -1.37 & -0.59 & 0.15 & 0.04 & -0.89 & 2.66 \\ 0.08 & -0.25 & 0.19 & -0.12 & 0.02 & 0.08 \\ 0.27 & -0.30 & -0.16 & 0.61 & -0.19 & -0.24 \\ 0.22 & 0.17 & 0.03 & -0.23 & 0.42 & -0.61 \end{bmatrix}.$$

Thus the shape functions for the 6-noded triangular element are:

$$N^T(x,y) = f^T(x,y)C^{-1} = \begin{bmatrix} 1 \\ x \\ y \\ x^2 \\ xy \\ y^2 \end{bmatrix}^T . \begin{bmatrix} 2.00 & -0.24 & 0.05 & 0.04 & 0.34 & -1.20 \\ -0.84 & 1.61 & -0.36 & -0.27 & 0.20 & -0.33 \\ -1.37 & -0.59 & 0.15 & 0.04 & -0.89 & 2.66 \\ 0.08 & -0.25 & 0.19 & -0.12 & 0.02 & 0.08 \\ 0.27 & -0.30 & -0.16 & 0.61 & -0.19 & -0.24 \\ 0.22 & 0.17 & 0.03 & -0.23 & 0.42 & -0.61 \end{bmatrix}$$

$$N(x,y) = \begin{bmatrix} +2.00 - 0.84x - 1.37y + 0.08x^2 + 0.27xy + 0.22y^2 \\ -0.24 + 1.61x - 0.59y - 0.25x^2 - 0.30xy + 0.17y^2 \\ +0.05 - 0.36x + 0.15y + 0.19x^2 - 0.16xy + 0.03y^2 \\ +0.04 - 0.27x + 0.04y - 0.12x^2 + 0.61xy - 0.23y^2 \\ +0.34 + 0.20x - 0.89y + 0.02x^2 - 0.19xy + 0.42y^2 \\ -1.20 - 0.33x + 2.66y + 0.08x^2 - 0.24xy - 0.61y^2 \end{bmatrix}$$

The head at point $x_o = 2.0$, $y_o = 1.5$ can be calculated as:

$$h(x_o, y_o) = N^T(x_o, y_o).H = f^T(x_o, y_o).\, C^{-1}.\, H$$

$$f^T(x_o, y_o)^T = \left[1, 2, 1.5, 4.0, 3.0, 2.25\right]$$

$$N^T(2,1.5) = f^T(2,1.5).C^{-1} = \begin{bmatrix} 1.00 \\ 2.00 \\ 1.50 \\ 4.00 \\ 3.00 \\ 2.25 \end{bmatrix}^T . \begin{bmatrix} 2.00 & -0.24 & 0.05 & 0.04 & 0.34 & -1.20 \\ -0.84 & 1.61 & -0.36 & -0.27 & 0.20 & -0.33 \\ -1.37 & -0.59 & 0.15 & 0.04 & -0.89 & 2.66 \\ 0.08 & -0.25 & 0.19 & -0.12 & 0.02 & 0.08 \\ 0.27 & -0.30 & -0.16 & 0.61 & -0.19 & -0.24 \\ 0.22 & 0.17 & 0.03 & -0.23 & 0.42 & -0.61 \end{bmatrix}$$

$$N^T(2,\ 1.5) = [-0.094\ ,\ 0.563\ ,\ -0.094\ ,\ 0.375\ ,\ -0.125\ ,\ 0.375]$$

$$h(2,1.5) = N^T(2,1.5).H = 17.45\ \text{m}$$

The hydraulic gradient with respect to x can be calculated as follows:

$$i_x = \frac{\partial h(x,\ y)}{\partial x} = \frac{\partial\left(N^T(x,\ y).H\right)}{\partial x} = \frac{\partial N^T(x,\ y)}{\partial x}.H = \frac{\partial\left(f^T(x,\ y).C^{-1}\right)}{\partial x}H = \frac{\partial f^T(x,\ y)}{\partial x}C^{-1}H$$

Where $\partial \mathbf{f}^{\mathbf{T}}(x,y)/\ \partial x = [0\ ,\ 1\ ,\ 0\ ,\ 2x\ ,\ y\ ,\ 0]$. At point: $x_o$=2.0, $y_o$=1.5

$\partial \mathbf{f}^{\mathbf{T}}(x_o,\ y_o)/\ \partial x = [0\ ,\ 1\ ,\ 0\ ,\ 4\ ,\ 1.5\ ,\ 0]$.

The hydraulic gradient at $x_o$=2.0, $y_o$=1.5 is calculated as $i_x$=18.2m/m. The hydraulic gradient with respect to y can also be calculated in the same way as $i_y$= — 5.4m/m. The hydraulic gradient is a function of x and y since the variation of the head is no longer linear but quadratic throughout the element. For example the hydraulic gradients at point $x_o$=2.0, $y_o$=2.0 are $i_x$=19.2m/m and $i_y$= — 8.4m/m.

***Problem 1***

It is observed that a beam, which lies in the interval 0< x <2m, undergoing flexural distortion has deflections $v_1$ = 10mm when x = 0 and $v_2$ = 12mm when x = 2m and rotations $\theta_1$ = 0.01 when x = 0 and $\theta_2$ = — 0.02 when x = 2m where $\theta = \partial v/\partial x$. Assuming that $v = a_1 + a_2\, x + a_3\, x^2 + a_4\, x^3$ calculate the deflection, rotation and curvature ($\partial^2 v/\partial x^2$) at x = 1.5m.

***Problem 2***

For the 6-noded triangular element considered in section 3.3.3, assume that the vector of nodal head is:

$$H\ =\ [1.832\ ,\ 34.296\ ,\ 66.76\ ,\ 28.896\ ,\ -8.968\ ,\ -3.568]^T$$

Calculate the hydraulic gradients, $i_x$ and $i_y$, at points $x_o$=2.0, $y_o$=1.5 and $x_o$=2.0, $y_o$=2.0. Compare the results with those obtained in Example 3.3. Explain a reason for similarity between the results obtained here with those obtained from a 3-noded element in Example3.3.

### *Problem 3*

A rectangular element bounded by the lines x = 0, x = 2a, y = 0, y = 2b has nodes at its vertices and the midpoints of its sides. Assume an appropriate polynomial function for variation of quantities within the element and show that the shape functions for node 4 (x = 2a, y = b) and node 5 (x = 2a, y = 2b) are:

$$N_4 = \frac{xy(2b - y)}{2ab^2}$$

$$N_5 = \frac{3xy(x/3a + y/3b - 1)}{4ab}$$

### *Problem 4*

A triangular plane element has 6 nodes at the points ($x_i$, $y_i$ ); i = 1, ..., 6. Assuming the temperature T can be approximated in the form:

$$T = a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 xy + a_6 y^2$$

Determine the shape functions for the element in the global coordinate system and in a local coordinate system which has its origin at the centroid and the X axis parallel to the side joining nodes 1-3 (vector 13 should run in the positive X-direction).

Use these to calculate the temperature at the centroid and the temperature gradients $\partial T/\partial x$, $\partial T/\partial y$ and $\partial T/\partial X$, $\partial T/\partial Y$

Your particular values of ($x_i$, $y_i$ $T_i$ ) i = 1, ..., 6 will be given to you. Temperatures are in $^o$c and coordinates in meters. Make sure you show the units of your answers.
Your solution should have:

(b) Data ($x_i$, $y_i$ $T_i$ )
(c) Shape function $N_4$ in global coordinates (x,y)
(d) Shape function $N_4$ in local coordinates (X,Y)
(e) The temperature at the centroid of the element
(f) The temperature gradients $\partial T/\partial x$, $\partial T/\partial X$ at the centroid

Chapter 7

# FINITE ELEMENT FORMULATION OF ELASTIC CONTINUA

In the finite element analysis of a problem, the medium is idealised as a number of finite elements interconnected only at their nodes. In the analysis of structures consisting of bars and beams, the elements making up the complete structure usually correspond to discrete well-defined parts of the structure. However when a two- or three-dimensional continuum problem is to be analysed, there may not be clear discrete parts, to be named finite elements, rather the continuum can be separated into finite elements by making imaginary cuts. There is generally no unique way of idealising a continuum structure with finite elements because such elements provide only approximate mathematical relationships for the continuum rather than discrete parts.

The accuracy of the finite element solution of a continuum problem is dependent upon the accuracy of the individual element relationships and the number, type, and arrangement of the elements from which the structure is assembled. Considerable choices are available for the basic shape of the elements, the function used to approximate the displacement field for the elements, and the arrangement of the elements. The remainder of the text will cover some of the most popular two-dimensional elements and the relevant topics such as their accuracy and efficiency.

Plane elasticity encompasses continuum problems of plane stress, plane strain and axial symmetry. The formulation of each type of problem is almost the same, and the computer code for solving plane stress problems can be adopted with only minor modifications to plane strain and axial symmetry. In these types of problems, the forces normal to the plane and, therefore, the “out-of-plane” displacements are zero. Problems of this kind can basically be treated as two-dimensional problems.

In this section of this chapter, we convert the strong formulation of the continuum mechanics of the previous chapter in the weak form, which is better known as principle of virtual work. Then we introduce the general method to use this weak form to formulate the equations of the finite element analysis. We illustrate some simplest formulations by introducing one of the simplest yet most versatile family of finite elements, the triangular elements. Then we introduce the linear rectangular element, which is the basis of more sophisticated, high order, rectangular elements used in most commercial codes .

As it was explained in the previous chapter, in planar elasticity problems the forces normal to the plane are zero and therefore the “out-of-plane” displacements are also zero.

## 7.1 Derivation of the Weak Form

The equation above is also called the strong form of the equilibrium equation. For the Finite Element modelling, it better to formulate equilibrium using the weak form, as we did in Chapter 1. Here we present the derivation of the weak form, or principle of virtual work, for a general tree-dimensional deformation.
We assume that in the domain V the continuum structure satisfies the differential equation:

$$\mathbf{L}^{T}\left[\boldsymbol{\sigma}(\mathbf{x})\right]+\mathbf{w}(\mathbf{x})=\mathbf{0} \tag{7.1}$$

$$\boldsymbol{\sigma}(\mathbf{x})=\mathbf{D}\boldsymbol{\varepsilon}(\mathbf{x}) \tag{7.2}$$

$$\boldsymbol{\varepsilon}(\mathbf{x})=\mathbf{L}\left[\mathbf{u}(\mathbf{x})\right] \tag{7.3}$$

Where

$$\mathbf{L}=\begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \tag{7.4}$$

We divide the boundary of the domain into two disjoint sets:

$$S=S_1\cup S_2 \qquad S_1\cap S_2=\varnothing \tag{7.5}$$

Each sub-domain satisfies the boundary condition

$$\mathbf{u}(\mathbf{x})=\mathbf{0} \qquad x\in S_1 \tag{7.6}$$

$$\mathbf{n}.\boldsymbol{\sigma}(\mathbf{x})=\boldsymbol{\tau}(\mathbf{x}) \tag{7.7}$$

Where $\boldsymbol{\tau}(\mathbf{x})$ is the traction applied at the surface $S_2$ and **n** is a unit vector perpendicular to the surface at point **x**. It is left to exercise of the reader to show that the traction (force per unit of area) related to the stress tensor given in Eq. (5.5) via Eq. (7.7). We introduce here a virtual displacement satisfying

$$\mathbf{u}^{*}(\mathbf{x})=0 \qquad x\in S_1 \tag{7.8}$$

Eq. (7.1) is multiply by a virtual displacement and integrated over the domain

$$\int_V \mathbf{u}^{*T}\left(\mathbf{L}^{T}\boldsymbol{\sigma}(\mathbf{x})+\mathbf{w}(\mathbf{x})\right)dV=\mathbf{0} \tag{7.9}$$

Using the chain rule $\mathbf{L}^T(\mathbf{u}^{*T}\boldsymbol{\sigma})=\mathbf{u}^{*T}\mathbf{L}^T\boldsymbol{\sigma}+(\mathbf{L}^T\mathbf{u}^{*T})\boldsymbol{\sigma}$ The first terms is transformed

$$\int_V \mathbf{L}^T(\mathbf{u}^{*T}\boldsymbol{\sigma})dV - \int_V (\mathbf{L}^T\mathbf{u}^*)\boldsymbol{\sigma}dV + \int_V \mathbf{u}^{*T}\mathbf{w}dV = \mathbf{0} \tag{7.10}$$

The first term vanishes due to the boundary condition. The term $\boldsymbol{\varepsilon}^{*T} = \mathbf{L}^T\mathbf{u}^*$ is the transpose of the strain produced by the virtual displacements. Then Eq. (7.10) becomes

$$\int_V \boldsymbol{\varepsilon}^{*T}\boldsymbol{\sigma}dV = \int_{S_2} \mathbf{u}^{*T}\boldsymbol{\tau}dA + \int_V \mathbf{u}^{*T}\mathbf{w}dV \tag{7.11}$$

This is the general form of the principle of virtual work.

## 7.2 Derivation of the stiffness matrix

Using the concept of interpolation function of the previous Chapter, the finite element formulation of the continuum mechanical boundary value problem above can be simplified in the following steps:

1) Displacement function is connected to displacement at the nodes using the shape function

$$\mathbf{u} = \mathbf{N}_e.\mathbf{u}^e \tag{7.12}$$

2) Strains $\boldsymbol{\varepsilon}=\mathbf{L}[\mathbf{u}]$ are connected to nodal displacement

$$\boldsymbol{\varepsilon} = \mathbf{B}_e.\mathbf{u}^e \qquad \mathbf{B}_e=\mathbf{L}[\mathbf{N}_e] \tag{7.13}$$

3) Using constitutive equation $\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon}$, the stress is related to displacement

$$\boldsymbol{\sigma} = \mathbf{D}.\mathbf{B}_e.\mathbf{u}_e \tag{7.14}$$

4) Using the principle of virtual work Eq. (7.11), with $\mathbf{u}^* = \mathbf{N}\mathbf{u}^{*e}$ as the virtual displacement and $\boldsymbol{\varepsilon}^* = \mathbf{B}\mathbf{u}^{*e}$ satisfying the corresponding virtual strain

$$\int_{V_e} \mathbf{u}^{*T}\mathbf{B}_e^T.\mathbf{D}.\mathbf{B}_e\mathbf{u}^e dV = \int_{V_e} \mathbf{u}^{*T}\mathbf{N}_e^T\mathbf{w}dV + \int_{S_e} \mathbf{u}^{*T}\mathbf{N}_e^T\boldsymbol{\tau}da \tag{7.15}$$

Where $V_e$ is the volume of the element and $S_e$ is the surface of the element where the traction is applied. Since the equation is valid for any virtual displacement satisfying the boundary condition, the equation above becomes:

$$\mathbf{k}^e\mathbf{u}^e = \mathbf{f}^e \tag{7.16}$$

$$\mathbf{k}^e = \int_{V^e} \mathbf{B}_e^T.\mathbf{D}.\mathbf{B}_e dV \tag{7.17}$$

$$\mathbf{f}^e = \int_{V_e} \mathbf{N}_e^T.\mathbf{w}dV + \int_{S_e} \mathbf{N}_e^T.\boldsymbol{\tau}da \tag{7.18}$$

Here we conclude that the calculation of the element stiffness matrix requires three steps:

1) Calculate $\mathbf{N_e}$
2) Calculate **B** by applying the derivative operator to $\mathbf{N_e}$ as $\mathbf{B_e=L[N_e]}$
3) Use Eq. (7.17)to get the stiffness matrix

Next chapter we will illustrate the method using linear triangular elements and linear rectangular elements.

## 7.3: Triangular Elements in plane elasticity (T3)

A general procedure is given to calculate the stiffness matrix of a simple 3-noded triangular element. The simple triangular element has nodal points at its vertices only, but a range of higher order triangular elements exists having additional nodes and consequently a more refined representation of displacement and stress fields. Some of the higher order triangular elements will also be introduced in this chapter.

The 3-noded triangular element shown in Figure 7.1 is the simplest possible planar element and one of the earliest finite elements. It has nodes at the vertices of the triangle only. For a plane elasticity problem, where all displacements are in the plane, the element has two degrees-of-freedom at each node, u and v, corresponding to the displacements in x and y directions respectively. Thus the element has a total of 6 degrees-of-freedom. The displacement vector and the force vector are:

$$\mathbf{u}^e = \left[u_1 , \ v_1 , u_2 , \ v_2 , u_3 , \ v_3 \right]^T$$

$$\mathbf{f}^e = \left[p_1 , \ q_1 , \ p_2 , q_2 , \ p_3 , \ q_3 \right]^T$$

Since each of these vectors contains 6 components, the size of the element stiffness matrix, $\mathbf{k^e}$, is 6×6.

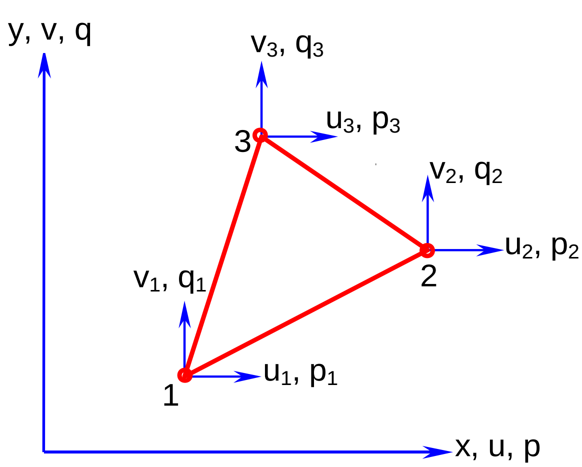



**Figure 7.1 Three-noded triangular element**

### *Stiffness matrix of linear triangular finite element*

The general procedure explained in Eq. (7.20) is employed here to calculate the stiffness matrix of the 3-noded triangular element. The node numbering and the Cartesian coordinate system shown in Figure 7.1 may be used for the element. The nodes are numbered in increasing order anti-clockwise. The coordinates of the nodes are $(x_1, y_1)$, $(x_2, y_2)$ and $(x_3, y_3)$. It is noted that the orientation of the element with respect to the xy coordinate system is completely arbitrary. Therefore the element stiffness matrix will be directly expressed in the xy global coordinate system. Here we perform the three steps mentioned in Section 6.2 to calculate the stiffness matrix:

**1. Derivation of the shape function N**

The variation of the displacement components, u and v, within the element can be expressed as complete linear polynomials of x and y:

$$\begin{aligned} u(x,y) &= a_1 + a_2 x + a_3 y = \mathbf{f}(x,y)\,.\,\mathbf{a} \\ v(x,y) &= b_1 + b_2 x + b_3 y = \mathbf{f}(x,y)\,.\,\mathbf{b} \end{aligned} \tag{7.19}$$

Where $f(x,y) = [1,\ x,\ y]$, $a = [a_1,\ a_2,\ a_3]^T$ and $b = [b_1,\ b_2,\ b_3]^T$.The values of u and v are known at the nodes. Therefore, Eq. (7.19) can be written for all the nodes by substituting the coordinates of the nodes into these equations:

$$\begin{aligned} u_1 &= a_1 + a_2 x_1 + a_3 y_1 \\ u_2 &= a_1 + a_2 x_2 + a_3 y_2 \\ u_3 &= a_1 + a_2 x_3 + a_3 y_3 \end{aligned} \quad \text{or} \quad \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

$$\begin{aligned} v_1 &= b_1 + b_2 x_1 + b_3 y_1 \\ v_2 &= b_1 + b_2 x_2 + b_3 y_2 \\ v_3 &= b_1 + b_2 x_3 + b_3 y_3 \end{aligned} \quad \text{or} \quad \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix}$$

We rewrite these equations as

$$\mathbf{u}^e = \mathbf{C}\ \mathbf{a} \quad \text{and} \quad \mathbf{v}^e = \mathbf{C}\ \mathbf{b}$$

where

$$\mathbf{C} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \tag{7.20}$$

Using Eqs. (7.19) and (7.20) the displacement fields u(x, y) and v(x,y) can now be expressed in the form of:

$$\begin{aligned} u(x, y) &= \mathbf{f}^T(x, y)\mathbf{C}^{-1}\mathbf{u}^e = \mathbf{N}^T(x, y)\,\mathbf{u}^e \\ v(x, y) &= \mathbf{f}^T(x, y)\mathbf{C}^{-1}\mathbf{v}^e = \mathbf{N}^T(x, y)\,\mathbf{v}^e \end{aligned} \tag{7.21}$$

with

$$C^{-1} = \frac{1}{2A}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix} \tag{7.22}$$

where 2A=det [C]= $(x_2\, y_3 - x_3\, y_2)$- $(x_1\, y_3 - x_3\, y_1)$+ $(x_1\, y_2 - x_2\, y_1)$ =2 × area of triangle.
The shape functions can be found as:

$$\mathbf{N}^T(x,y) = \mathbf{f}^T(x,y)\mathbf{C}^{-1} = \frac{1}{2A}[1,\ x,\ y]\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

$$\mathbf{N}(x,y) = \begin{bmatrix} N_1(x, y) \\ N_2(x, y) \\ N_3(x, y) \end{bmatrix} = \begin{bmatrix} \dfrac{(x_2y_3 - x_3y_2) + x(y_2 - y_3) + y(x_3 - x_2)}{2A} \\ \dfrac{(x_3y_1 - x_1y_3) + x(y_3 - y_1) + y(x_1 - x_3)}{2A} \\ \dfrac{(x_1y_2 - x_2y_1) + x(y_1 - y_2) + y(x_2 - x_1)}{2A} \end{bmatrix} \tag{7.23}$$

The general displacements within the element can be related to the nodal displacements using shape functions:

$$\begin{aligned} u &= N_1u_1 + N_2u_2 + N_3u_3 = \mathbf{N}^T\mathbf{u}^e \\ v &= N_1v_1 + N_2v_2 + N_3v_3 = \mathbf{N}^T\mathbf{v}^e \end{aligned} \tag{7.24}$$

Eq. (7.24) can now be written in matrix format as:

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \quad \text{or} \quad u(x,y) = \mathbf{N}^e.\mathbf{u}^e \tag{7.25}$$

## 2. Derivation of the matrix B

The matrix $\mathbf{B_e}$ has been defined for a general case in Eq. (7.13) and contains derivatives of the shape functions.

$$\mathbf{B}_e = \mathbf{L}[\mathbf{N}_e] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}$$

$$\mathbf{B}_e = \begin{bmatrix} N_{1x} & 0 & N_{2x} & 0 & N_{3x} & 0 \\ 0 & N_{1y} & 0 & N_{2y} & 0 & N_{3y} \\ N_{1y} & N_{1x} & N_{2y} & N_{2x} & N_{3y} & N_{3x} \end{bmatrix} \tag{7.26}$$

The derivatives of the shape functions for the triangular element can be obtained as:

$$\begin{bmatrix} N_{1x} \\ N_{1y} \\ N_{2x} \\ N_{2y} \\ N_{3x} \\ N_{3y} \end{bmatrix} = \frac{1}{2A} \begin{bmatrix} (y_2 - y_3) \\ (x_3 - x_2) \\ (y_3 - y_1) \\ (x_1 - x_3) \\ (y_1 - y_2) \\ (x_2 - x_1) \end{bmatrix} \tag{7.27}$$

Therefore the matrix $\mathbf{B_e}$ is obtained for the linear triangular element as:

$$\mathbf{B}_e = \frac{1}{2A} \begin{bmatrix} (y_2 - y_3) & 0 & (y_3 - y_1) & 0 & (y_1 - y_2) & 0 \\ 0 & (x_3 - x_2) & 0 & (x_1 - x_3) & 0 & (x_2 - x_1) \\ (x_3 - x_2) & (y_2 - y_3) & (x_1 - x_3) & (y_3 - y_1) & (x_2 - x_1) & (y_1 - y_2) \end{bmatrix} \tag{7.28}$$

It can be seen that $\mathbf{B_e}$ and therefore strains within the linear triangular element are independent of x and y. For this reason, this element is often called the “constant strain triangle”.

## 3. Calculating the element stiffness matrix

The internal stress can be related to the external loads using the principle of virtual work for the element. This leads to the equation for calculation of the element stiffness matrix.

$$\mathbf{k}^e = \int_{V_e} \mathbf{B}_e^T . \mathbf{D} . \mathbf{B}_e \, dV = \mathbf{B}_e^T . \mathbf{D} . \mathbf{B}_e \, A . t \tag{7.29}$$

Where A and t are the area and the thickness of the element, respectively. Note that because $\mathbf{B_e}$ and $\mathbf{D}$ are independent of coordinate location (x, y), the integration over this element can be performed easily and exactly.

## 7.4: Linear Rectangular Element (Q4)

The linear rectangular element is the simplest rectangular element for planar analysis. The interpolation function used to approximate variation of displacements within the element is linear with respect to x and y. For simplicity, a Cartesian coordinate system is adopted where the axes x and y run along two of the element edges, as shown in Figure 7.2. Therefore this coordinate system is the local one. The origin of the x-y axes is chosen for convenience to be at a corner of the rectangle, but could be located at some other points without affecting the procedure for calculation of the element properties. The element has nodes at the four corner points, each node has 2 degrees-of-freedom u and v, corresponding to the displacements in x and y directions respectively Figure 7.2. Thus the element has a total of 8 degrees-of-freedom. The displacement vector and the force vector are:

$$\mathbf{u}^e = \left[u_1\,,\ v_1\,,\ u_2\,,\ v_2\,,\ u_3\,,\ v_3\,,\ u_4\,,\ v_4\right]^T$$

$$\mathbf{f}^e = \left[p_1\,,\ q_1\,,\ p_2\,,\ q_2\,,\ p_3\,,\ q_3\,,\ p_4\,,\ q_4\right]^T$$

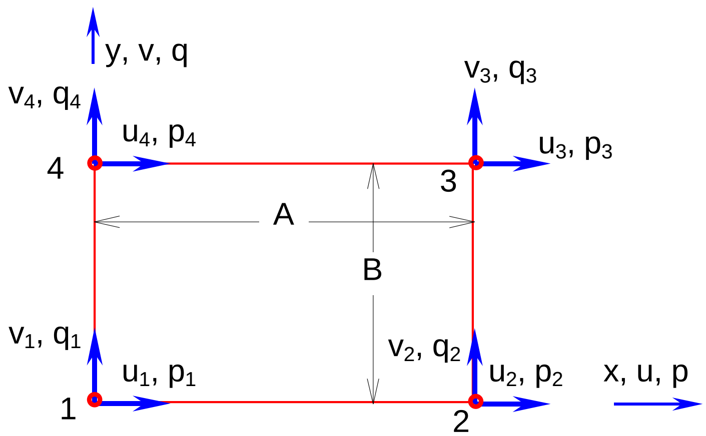



**Figure 7.2 Four-noded linear rectangular element**

Since each of these vectors contains 8 components, the size of the element stiffness matrix, $\mathbf{K}^e$, is 8×8. The general procedure explained in Eq. (7.20) is employed here to calculate the stiffness matrix of the 4-noded rectangular element. The node numbering and the Cartesian coordinate system shown in Figure 7.2 are used here. The nodes are numbered in increasing order anti-clockwise. The coordinates of the nodes 1 to 4 are (0, 0), (A, 0), (A, B) and (0, B). The coordinate system shown in Figure 7.2 is a local one so that the element stiffness matrix should be transformed to the global coordinate system before it can be assembled into the global stiffness matrix.

### 1. Calculation of the shape function

The variation of the displacement components, u and v, within the element can be expressed as complete linear polynomials of x and y:

$$u(x,y) = a_1 + a_2 x + a_3 y + a_4 xy = \mathbf{f}(x,y)\,.\,\mathbf{a}$$
$$v(x,y) = b_1 + b_2 x + b_3 y + b_4 xy = \mathbf{f}(x,y)\,.\,\mathbf{b}$$

where

$$f(x,y) = \left[1,\ x,\ y,\ xy\right]^T\ ,\ a = \left[a_1,\ a_2,\ a_3,\ a_4\right]^T$$

and

$$b = \left[b_1, b_2, b_3, b_4\right]^T .$$

The general displacements within the element are related to the nodal displacements using shape functions:

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3 + N_4 u_4 = N^T . u^e$$

$$v = N_1 v_1 + N_2 v_2 + N_3 v_3 + N_4 v_4 = N^T . v^e$$

$$N^T = f^T(x, y).C^{-1}$$

$$C = \begin{bmatrix} 1 & x_1 & y_1 & x_1 y_1 \\ 1 & x_2 & y_2 & x_2 y_2 \\ 1 & x_3 & y_3 & x_3 y_3 \\ 1 & x_4 & y_4 & x_4 y_4 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 1 & A & 0 & 0 \\ 1 & A & B & AB \\ 1 & 0 & B & 0 \end{bmatrix}$$

$$C^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ -1/A & 1/A & 0 & 0 \\ -1/B & 0 & 0 & 1/B \\ 1/AB & -1/AB & 1/AB & -1/AB \end{bmatrix}$$

Therefore the shape functions are:

$$N = \begin{bmatrix} N_1 \\ N_2 \\ N_3 \\ N_4 \end{bmatrix} = f^T(x,y)C^{-1} = \begin{bmatrix} 1 - x/A - y/B + xy/AB \\ x/A - xy/AB \\ xy/AB \\ y/B - xy/AB \end{bmatrix}$$

The equation above can now be written in matrix format as:

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} \quad \text{or} \quad u(x,y) = \mathbf{N}_e . \mathbf{u}^e$$

## 2. Calculation of the B matrix

The matrix $\mathbf{B_e}$ can be obtained from $N_e$ (equation above) as:

$$\mathbf{B}_e = \begin{bmatrix} N_{1x} & 0 & N_{2x} & 0 & N_{3x} & 0 & N_{4x} & 0 \\ 0 & N_{1y} & 0 & N_{2y} & 0 & N_{3y} & 0 & N_{4y} \\ N_{1y} & N_{1x} & N_{2y} & N_{2x} & N_{3y} & N_{3x} & N_{4y} & N_{4x} \end{bmatrix} \tag{7.30}$$

where $N_{ix}$, $N_{iy}$ are the derivatives of the shape functions with respect to x, and y, respectively. The derivatives of the shape functions for the 4-noded rectangular element can be obtained as:

$$\begin{bmatrix} N_{1x} \\ N_{1y} \\ N_{2x} \\ N_{2y} \\ N_{3x} \\ N_{3y} \\ N_{4x} \\ N_{4y} \end{bmatrix} = \begin{bmatrix} -1/A + y/AB \\ -1/B + x/AB \\ 1/A - y/AB \\ -x/AB \\ y/AB \\ x/AB \\ -y/AB \\ 1/B - x/AB \end{bmatrix} \tag{7.31}$$

Therefore the matrix $\mathbf{B_e}$ is obtained for the linear rectangular element as:

$$\mathbf{B}_e = \begin{bmatrix} \frac{y}{AB}-\frac{1}{A} & 0 & \frac{1}{A}-\frac{y}{AB} & 0 & \frac{y}{AB} & 0 & -\frac{y}{AB} & 0 \\ 0 & \frac{x}{AB}-\frac{1}{B} & 0 & -\frac{x}{AB} & 0 & \frac{x}{AB} & 0 & \frac{1}{B}-\frac{x}{AB} \\ \frac{x}{AB}-\frac{1}{B} & \frac{y}{AB}-\frac{1}{A} & -\frac{x}{AB} & \frac{1}{A}-\frac{y}{AB} & \frac{x}{AB} & \frac{y}{AB} & \frac{1}{B}-\frac{x}{AB} & -\frac{y}{AB} \end{bmatrix} \tag{7.32}$$

It can be seen that $\mathbf{B_e}$ and therefore the strains within the element are a function of x and y. The normal strain $\varepsilon_{xx}$ varies linearly with y but not with x, while $\varepsilon_{yy}$ varies linearly with x but not with y. The shear strain varies linearly with x and y throughout the element, as can be seen from the form of the strain-displacement matrix, $\mathbf{B}_e$, in Eq. (7.32)

## 3. Calculation of the stiffness matrix

In general, for plane stress or plane strain problems, the matrix **D** can be written in the form of:

$$\mathbf{D} = \begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{12} & d_{22} & 0 \\ 0 & 0 & d_{33} \end{bmatrix} \tag{7.33}$$

This leads to the equation for calculation of the element stiffness matrix in the local coordinate system.

$$\mathbf{K}^e = \int_{V_e} \mathbf{B}_e^T \, \mathbf{D} \, \mathbf{B}_e \, dV = t \iint \mathbf{B}_e^T \, \mathbf{D} \, \mathbf{B}_e \, dx \, dy \tag{7.34}$$

whereby (t) is the thickness of the element.

The product of $\mathbf{B}_e^T . \mathbf{D} . \mathbf{B}_e$ has to be evaluated first and the components of the resulting matrix have to be integrated over the area of the element. The final value of the stiffness matrix obtained from these calculations is given by:

$$K^e = \frac{t}{12}\begin{bmatrix}
4d_{11}\frac{B}{A}+4d_{33}\frac{A}{B} & 3d_{12}+3d_{33} & -4d_{11}\frac{B}{A}+2d_{33}\frac{A}{B} & 3d_{12}-3d_{33} & -2d_{11}\frac{B}{A}-2d_{33}\frac{A}{B} & -3d_{12}-3d_{33} & 2d_{11}\frac{B}{A}-4d_{33}\frac{A}{B} & -3d_{12}+3d_{33} \\
 & 4d_{11}\frac{A}{B}+4d_{33}\frac{B}{A} & -3d_{12}+3d_{33} & 2d_{11}\frac{A}{B}-4d_{33}\frac{B}{A} & -3d_{12}-3d_{33} & -2d_{11}\frac{A}{B}-2d_{33}\frac{B}{A} & 3d_{12}-3d_{33} & -4d_{11}\frac{A}{B}+2d_{33}\frac{B}{A} \\
 & & 4d_{11}\frac{B}{A}+4d_{33}\frac{A}{B} & -3d_{12}-3d_{33} & 2d_{11}\frac{B}{A}-4d_{33}\frac{A}{B} & 3d_{12}-3d_{33} & -2d_{11}\frac{B}{A}-2d_{33}\frac{A}{B} & 3d_{12}+3d_{33} \\
 & & & 4d_{11}\frac{A}{B}+4d_{33}\frac{B}{A} & -3d_{12}+3d_{33} & -4d_{11}\frac{A}{B}+2d_{33}\frac{B}{A} & 3d_{12}+3d_{33} & -2d_{11}\frac{A}{B}-2d_{33}\frac{B}{A} \\
 & & & & 4d_{11}\frac{B}{A}+4d_{33}\frac{A}{B} & 3d_{12}+3d_{33} & -4d_{11}\frac{B}{A}+2d_{33}\frac{A}{B} & 3d_{12}-3d_{33} \\
 & \text{Symmetric} & & & & 4d_{11}\frac{A}{B}+4d_{33}\frac{B}{A} & -3d_{12}+3d_{33} & 2d_{11}\frac{A}{B}-4d_{33}\frac{B}{A} \\
 & & & & & & 4d_{11}\frac{B}{A}+4d_{33}\frac{A}{B} & -3d_{12}-3d_{33} \\
 & & & & & & & 4d_{11}\frac{A}{B}+4d_{33}\frac{B}{A}
\end{bmatrix}.$$

Now the stiffness matrix needs to be converted into the global coordinate system. We note here that this step was not required for triangular element, where we used the global coordinate system to calculate the matrix. Here we used a local coordinate system oriented with the rectangle.

The transformation of the components of the stiffness matrix from the local coordinate system to the global system is identical to that derived in Chapter 1, i.e.,

$$\mathbf{k}^e = \mathbf{T}.\mathbf{K}^e.\mathbf{T}^T$$

Where $\mathbf{k}^e$ is the stiffness matrix in the global coordinate system and $\mathbf{T}^T$ is the transformation matrix defined as:

$$T^T = \begin{bmatrix} H^T & 0 & 0 & 0 \\ 0 & H^T & 0 & 0 \\ 0 & 0 & H^T & 0 \\ 0 & 0 & 0 & H^T \end{bmatrix}$$

where

$$H^T = \begin{bmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix}$$

And $\theta$ is the angle between the local coordinates and the global coordinates, as defined in Appendix B

### *Problem 1*

The data for a finite element analysis of a structure under plane strain conditions will be given to you. Use the information to calculate:

(a) The global stiffness matrix.
(b) The vector of applied nodal forces.
(c) The vector of nodal displacements.
(d) Displacements at the centroid of element 6
(e) Strains at the centroid of element 6
(f) Stresses at the centroid of element 6

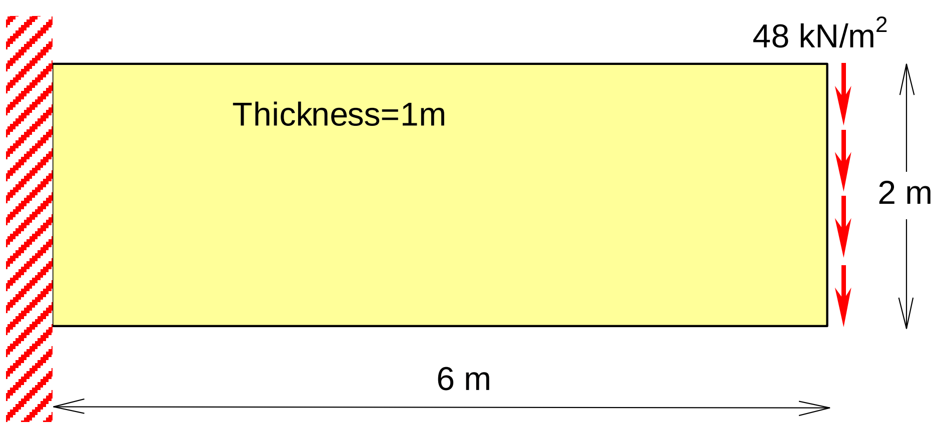
### *Problem 2*

A deep cantilever beam is subjected to a uniformly distributed traction at its free end. The dimension of the beam is shown in the figure below, The Young's modulus, E (MN/m2), is equal to the sum of the numerals of your SID and the Poisson's ratio, is equal to the last two numerals in your SID divided by 200.

1. Calculate the maximum vertical deflection of the beam using the constant strain triangular finite elements. Use 2x6, 4x12 and 8x24 subdivisions to generate different finite element meshes to approximate the deflection of the beam.

2. Compare the performances of the constant strain and the linear strain triangular finite elements on the basis of the number of nodes (or degrees-of-freedom) used in a finite element mesh. For example, use 1x3, 2x6, 4x12 and 8x24 subdivisions to generate finite element meshes of linear strain triangular elements and compare the results with those obtained in section 1.

3. Comment on the distribution of stresses (in particular, normal stress, $\sigma_{xx}$, close to the beam support) predicted by the finite element analyses using different element types and different number of nodes.



# Chapter 8

# ACCURACY AND EFFICIENCY IN FINITE ELEMENT MODELLING

## 8.1: Accuracy and Efficiency of Linear Triangular Elements

The linear element is the basic planar element and one of the first elements developed and used in practice. As noted previously, the strains and stresses are constant over the entire area of one element. Therefore a high degree of mesh refinement is required where significant strain gradients exist.

Consider two constant strain triangular finite elements shown in Figure 8.1. Assume that only one node of element b is displaced while other nodes are fixed. Then element b is subjected to non-zero strains and stresses, which are constant over the area of the element, while strains and stresses within element a are all zero. An infinitesimal element at the boundary of the two finite elements, the shaded area in Figure 8.1, is not in equilibrium. There are obviously discontinuities in strains and stresses at the boundary of the two elements. In view of this fact, it is necessary to use a fine mesh of these elements where high stress or strain gradients are expected.

The linear triangular element has the advantages of simplicity in its formulation. The strain-displacement matrix **B** is independent of the coordinates. Therefore, the integration of the stiffness matrix $\int B^T . D . B \; dv$ imposes no difficulty.

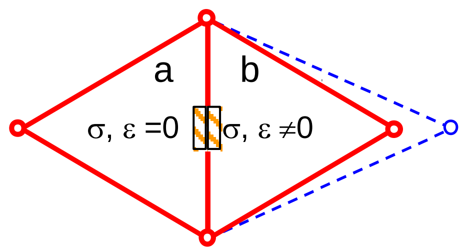



**Figure 8.1 Discontinuity of stress and strain**

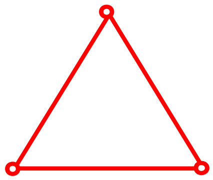
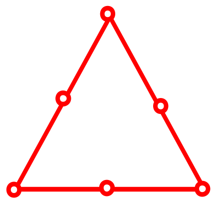
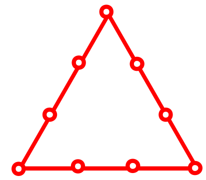
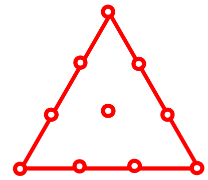
## 8.2: Higher Order Triangular Elements

Higher order elements have more degrees-of-freedom and usually give a more accurate representation of the actual behaviour. Application of these elements ensures more accurate solutions to be achieved with fewer elements.

It was shown in the previous section that the constant strain triangular element has a certain disadvantage, particularly in regions of high stress gradients. One method of dealing with this problem is to use a very fine mesh while still using the basic linear elements. However, an alternative approach is to use higher order elements, elements for which higher order polynomials are used to approximate their displacement functions. This can be done by specifying additional nodes for the elements, thus giving it more degrees-of-freedom. The resulting elements have the advantage that fewer of them are required to achieve certain accuracy. However, this is at the expense of greater computational complexity.

Some of the higher-order triangular elements are shown in Table 8-1 together with the interpolation functions used to derive the stiffness matrices of the elements.

**Table 8-1 Planar triangular element types**

| | | | | |
|---|---|---|---|---|
| **Shape** |  |  |  |  |
| **No. DOF** | **6** | **12** | **18** | **20** |
| **Displ. function** | **Linear** | **Parabolic** | **Cubic (Non-standard)** | **Cubic (Standard)** |

## 8.3: Accuracy and efficiency of Linear Rectangular Elements

The linear 4-noded element is the simplest rectangular element for planar analysis. The normal stress and strain in the x direction, $\sigma_{xx}$ and $\varepsilon_{xx}$, vary linearly with y within the element and the normal stress and strain in the y direction, $\sigma_{yy}$ and $\varepsilon_{yy}$, vary linearly with x. Because the variation of strains and stresses are not restricted to a uniform value over the whole element, the linear rectangular element is generally more efficient and slightly more accurate than the basic linear 3-noded triangular element, although it is less adaptable to bodies with a complex geometry. The triangular element has the advantage that it can be used for bodies with irregular boundary shapes and its formulation is simpler than the 4-noded rectangular element. Both the 4-noded rectangular element and the 3-noded triangular element were developed based on the assumption that the displacements vary linearly within the elements and thus at element edges. It follows that these two types of element can be connected to one another without any loss of compatibility and can be combined together to model a finite element mesh with a complex geometry in a planar analysis.

The 4-noded rectangular element has shown some deficiencies in finite element analyses. For example, it is unable to represent accurately one of the most commonly occurring stress states, i.e., the state of bending stress. This can be illustrated by subjecting a simple rectangular planar element to a pure bending stress as shown in Figure 8.2. The top and bottom edges of the finite element remain straight under pure bending moment. The approximation of the state of pure bending by the finite element results in a fictitious prediction of relatively large shear strains.
The unwanted shear strain causes the behaviour of the finite element to be too stiff. The effect of the unwanted shear strain becomes significant for elements with large aspect ratio(a/b).

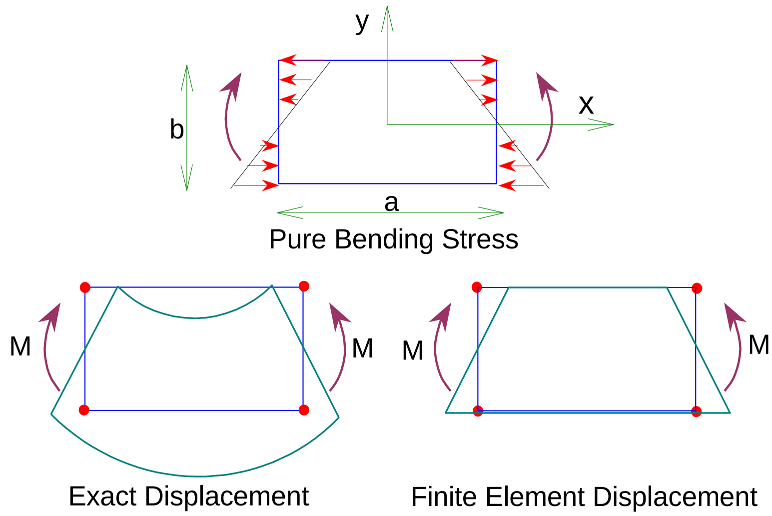



**Figure 8.2 Deformation under pure bending**

A large number of these elements have to be used in order to achieve an acceptable accuracy in problems where bending action is important or where a high stress gradient is expected. The use of a greater number of elements in a finite element analysis usually means a longer computation time. Therefore, it is desirable to use higher order elements in many practical analyses.

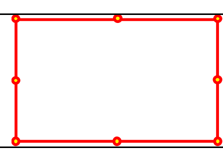
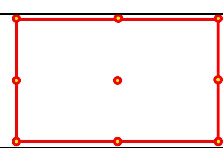
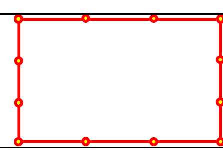
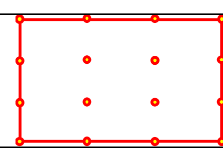
## 8.4: High Order Rectangular Elements

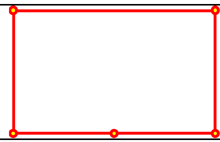
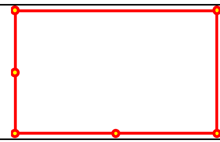
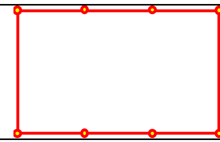
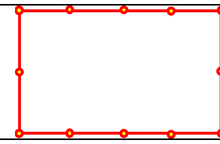
Higher order elements have more degrees-of-freedom and usually represent a more accurate displacement field, and therefore stress and strain field. Application of fewer elements of this kind in a finite element analysis usually results in a more accurate solution compared with the results of an analysis obtained using lower order elements.

To develop higher order elements, higher order polynomials are required in order to approximate displacement functions for the elements. This requires additional nodes which results in more degrees-of-freedom for the elements. Some of the higher order rectangular elements are shown in Table 8.2. The interpolation functions used to interpolate the displacements of these elements are usually of the same order in the x and y directions. However, this is not a strict requirement in developing the higher order elements, since it is trivial to generate shape functions for rectangular elements using a different order of interpolation in the x direction to that used in the y direction. This will result in a series of elements which have different numbers of nodal points in the x direction to those in the y direction. Some of the elements of this kind are also shown in Table 8-2.

It should be noted that as the number of nodes in a finite element increases, calculation of the stiffness matrix of the element becomes more complex. Some of the complexities arise from the parametric multiplication of large matrices and the high number of integration operations.

**Table 8-2 Higher order rectangular elements**

| | | | | |
|---|---|---|---|---|
| Shape |  |  |  |  |

| No. DOF | 16 | 18 | 24 | 32 |
|---|---|---|---|---|
| Displ. function in x/y direction | Parabolic (Non-standard) | Parabolic (Standard) | Cubic (Non-standard) | Cubic (Standard) |
| | | | | |
| Shape |  |  |  |  |
| No. DOF | 10 | 12 | 16 | 24 |
| Displ. function<br>In x direction<br>In y direction | Semi-linear<br>Linear | Semi-linear<br>Semi-linear | Cubic<br>Linear | Quadratic<br>Parabolic |

## 8.5: Coordinate Transformation and Numerical Integration

The derivation of the stiffness matrix for any high order element requires integrations of a high number of functions over the area of the element. As the order of the interpolation function for the element increases, integration operations become more complex if they are to be performed analytically. However, analytical integrations may be avoided using coordinate transformation and numerical integration. One of the advantages of numerical integration, as opposed to analytical integration, is that it can all be carried out by the computer. Elements with curved boundaries, or non-rectangular quadrilateral elements can also be easily developed and their stiffness matrices can be integrated without additional difficulty.

### *Summary of numerical integration*

"Quadrature" is the name applied to evaluating an integral numerically rather than analytically. There are a number of methods available for numerical integration. However, the Gauss quadrature rule is the one most often used in finite element analyses and is therefore dealt with here.

Gauss quadrature rules are written for a finite integral over the interval $[-1, 1]$ in each coordinate direction. Integration of a function, $f(\eta)$ in one dimension is expressed as:

$$\int_{-1}^{1} f(\eta)\,d\eta = \sum_{i=1}^{n} f(\eta_i)\,w_i \tag{8.1}$$

where n is the number of integration points (or Gauss points) selected for the integration, $\eta_i$ is the coordinate for Gauss point i, and $w_i$ is the weight for Gauss pointi. The coordinates of the Gauss points and their weights are well known and some are given in Table 8-3 for various orders of numerical integration.

**Table 8-3 Gauss points and weights for one-dimensional integration**

| Number of Gauss points, n | Coordinate $\eta_i$ | Weight $w_i$ |
|---|---|---|
| 1 | 0 | 2 |
| 2 | $+1/\sqrt{3}$<br>$-1/\sqrt{3}$ | 1<br>1 |
| 3 | $\sqrt{0.6}$<br>0<br>$-\sqrt{0.6}$ | 5/9<br>8/9<br>5/9 |

The Gauss quadrature rules are designed for cases where f(η) is a polynomial. A rule with n Gauss points is exact for a one-dimensional polynomial integrand of degree up to 2n-1. For example, an integral with 2 Gauss points gives no error for linear, parabolic and cubic polynomials. Gauss quadrature rules may also be used for cases where the integrand is not a polynomial, but the result will only be an approximate one.

Integration of a function, f(η,μ) in two dimensions over the area of a quadrilateral, i.e. over the interval [ −1,1] for both η and μ, can be expressed as:

$$\int_{-1}^{1}\int_{-1}^{1} f(\eta,\mu)\ d\eta\ d\mu = \sum_{i=1}^{n} f(\eta_i,\mu_i)\,w_i \qquad (8.2)$$

where $\eta_i$ and $\mu_i$ are the coordinates for Gauss point i. The coordinates of the Gauss points and their weights for two dimensions are given in Table 8-4. The two-dimensional quadrature rules are a simple generalization of the one-dimensional rules where f(η, μ) is a polynomial. A rule with n Gauss points is exact for a two-dimensional polynomial integrand of degree up to $2\sqrt{n}-1$. For example, a one-Gauss point rule is valid for a constant or linear function, a 4-Gauss point rule gives no integration error for a polynomial up to and including a cubic.

**Table 8-4 Gauss points and weights for two-dimensional integration**

| Number of Gauss points, n | Coordinate $\eta_i$ | Coordinate $\mu_i$ | Weight $w_i$ |
|---|---|---|---|
| 1 | 0 | 0 | 4 |
| 4 | $-1/\sqrt{3}$<br>$+1/\sqrt{3}$<br>$-1/\sqrt{3}$<br>$+1/\sqrt{3}$ | $-1/\sqrt{3}$<br>$-1/\sqrt{3}$<br>$+1/\sqrt{3}$<br>$+1/\sqrt{3}$ | 1<br>1<br>1<br>1 |
| 9 | $-\sqrt{0.6}$<br>0<br>$+\sqrt{0.6}$<br>$-\sqrt{0.6}$<br>+0<br>$+\sqrt{0.6}$<br>$-\sqrt{0.6}$<br>0<br>$+\sqrt{0.6}$ | $-\sqrt{0.6}$<br>$-\sqrt{0.6}$<br>$-\sqrt{0.6}$<br>0<br>0<br>0<br>$+\sqrt{0.6}$<br>$+\sqrt{0.6}$<br>$+\sqrt{0.6}$ | 25/81<br>40/81<br>25/81<br>40/81<br>64/81<br>40/81<br>25/81<br>40/81<br>25/81 |

Integration schemes for two-dimensional triangular elements can be found in most of the finite element textbooks. The number of Gauss points that can be used in numerical integration over triangular elements can be 1, 3, 6, …
It is recommended that an integration rule with 3 Gauss points is used for all triangular elements and an integration rule with at least 4 Gauss points is used for quadrilateral elements.

### *Natural coordinates*

In order to evaluate an integral over the area or volume of an arbitrary-oriented element, it is necessary to transform the coordinates. In this procedure it is convenient to introduce a system of natural coordinates. A model element is chosen in the interval of $[-1,1]$ in each direction, so that all of the integrations required to form the element stiffness matrix can be performed using a quadrature rule. Then the real finite element, in the coordinate system x and y, can be mapped onto the model element, in the natural coordinate system, using a standard transformation.

### *One-Dimensional Elements*

Consider a linear 2-noded bar element shown in Figure 8.3 -a. A model element of this kind can be defined in the natural coordinate $\eta$, as shown in Figure 8.3-b. The coordinate of the model element in the natural coordinate is chosen to range from —1 to +1. Therefore numerical integration rules can be easily applied.

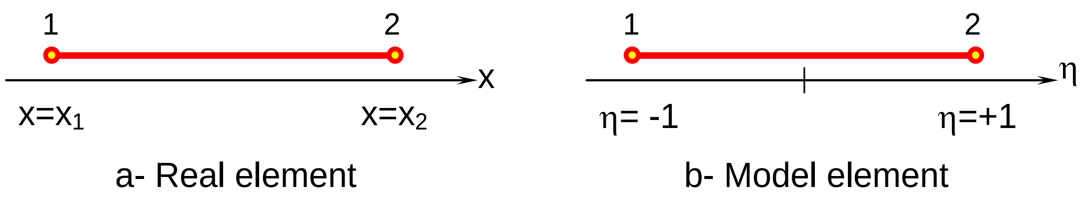



**Figure 8.3 Real and model linear one-dimensional elements**

The shape functions associated with the nodes of the model element can be defined as:

$$N_1 = (1-\eta)/2 \qquad (8.3)$$

$$N_2 = (1+\eta)/2$$

The displacement at any point within the element can be obtained using the shape functions and the nodal displacements at nodes 1 and 2, $u_1$ and $u_2$ respectively.

$$u(\eta) = N_1(\eta)U_1 + N_2(\eta)U_2$$

The shape functions can also be used to find the x-coordinate of a point within the element, if the element is iso-parametric. The x-coordinate associated with a point within the model element can be obtained in a similar form to the displacements:

$$x(\eta) = N_1(\eta)x_1 + N_2(\eta)x_2 \;= N^T x \qquad (8.4)$$

Where $x = [x_1, x_2]^T$. For example, if $x_1$ = 11m and $x_2$ = 17.5m, the centre of the real bar element, which corresponds to the centre of the model element at η=0, can be calculated as:

$$N_{1(\eta=0)} = 1/2$$

$$N_{2(\eta=0)} = 1/2$$

$$x_{(\eta=0)} = N_{1(\eta=0)}\,x_1 + N_{2(\eta=0)}\,x_2 \;= 1/2 \times 11 \;+\; 1/2 \times 17.5 = 14.25\text{m}$$

In calculation of the stiffness matrix of the element, it is necessary to find the strain-displacement matrix, $\mathbf{B}_e$, and hence the derivatives of the shape functions with respect to real coordinate x, i.e

$$B_e = \left[\frac{\partial N_1}{\partial x} \;,\; \frac{\partial N_2}{\partial x}\right] \qquad (8.5)$$

Since the shape functions are defined in terms of the model coordinate, η, the derivatives of the shape functions can be found using the chain rule:

$$\frac{\partial N_1}{\partial x} = \frac{\partial N_1}{\partial \eta}\frac{\partial \eta}{\partial x} = \frac{\partial}{\partial \eta}\big((1-\eta)/2\big)\frac{\partial \eta}{\partial x} = -\frac{1}{2}\frac{1}{\partial x/\partial \eta} \tag{8.6}$$

$$\frac{\partial N_2}{\partial x} = \frac{\partial N_2}{\partial \eta}\frac{\partial \eta}{\partial x} = \frac{\partial}{\partial \eta}\big((1+\eta)/2\big)\frac{\partial \eta}{\partial x} = \frac{1}{2}\frac{1}{\partial x/\partial \eta} \tag{8.7}$$

The quantity $\partial x/\partial \eta$ is known as the Jacobian matrix, **J**, and relates the derivatives of the shape functions with respect to the two coordinate systems, i.e.,

$$\frac{\partial N_i}{\partial \eta} = J\frac{\partial N_i}{\partial x} \quad \text{or} \quad \frac{\partial N_i}{\partial x} = J^{-1}\frac{\partial N_i}{\partial \eta}$$

The Jacobian can be found by differentiating Eq. (8.4):

$$J = \frac{\partial x}{\partial \eta} = \frac{\partial N_1}{\partial \eta}x_1 + \frac{\partial N_2}{\partial \eta}x_2 = \begin{bmatrix} \frac{\partial N_1}{\partial \eta}, & \frac{\partial N_2}{\partial \eta} \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = \begin{bmatrix} -\frac{1}{2}, & \frac{1}{2} \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = L/2 \tag{8.8}$$

Where L is the length of the bar element. Substituting the value of $\partial x/\partial \eta$ from Eq. (8.8) into Eq. (8.6) and Eq. (8.7) results in the derivatives of the shape functions with respect to the real coordinate x and can be used to form the strain-displacement matrix $\mathbf{B}_e$:

$$B_e = \begin{bmatrix} \frac{\partial N_1}{\partial x}, & \frac{\partial N_2}{\partial x} \end{bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial \eta}, & \frac{\partial N_2}{\partial \eta} \end{bmatrix} J^{-1} = \begin{bmatrix} -\frac{1}{2}, & \frac{1}{2} \end{bmatrix}\frac{2}{L} = \begin{bmatrix} -\frac{1}{L}, & \frac{1}{L} \end{bmatrix} \tag{8.9}$$

To transform the variables and the region with respect to which the integration is made a standard process will be used which involves the determinant of the Jacobian matrix, det**J**. For example a volume element becomes:

$$dx\, dy\, dz = \det J\, d\eta\, d\mu\, d\zeta$$

Where $\eta$, $\mu$ and $\zeta$ are the natural coordinates corresponding to the actual coordinates of x, y and z.
The equation for calculation of the stiffness matrix can now be written in terms of the natural coordinate $\eta$.

$$k^e = A\int_{x_1}^{x_2} B_e^T\, D\, B_e\, dx = A\int_{-1}^{+1} B_e^T\, D\, B_e \det J\, d\eta = A\sum_{i=1}^{n} B_{ei}^T\, D\, B_{ei} \det J\, w_i \tag{8.10}$$

Where $\mathbf{B}_{ei}$ is the matrix $\mathbf{B}_e$ evaluated at Gauss point i. The determinant of a 1×1 Jacobian matrix is equal to the Jacobian matrix itself, so that det**J**=L/2. A one Gauss point integration rule, n=1, can be selected for the numerical integration. The weight for the Gauss point is obtained from Table8-3 as w=2. Then the stiffness matrix is calculated as:

$$k^e = A\sum_{i=1}^{1}\begin{bmatrix}-1/L\\1/L\end{bmatrix}.E.\begin{bmatrix}-1/L & 1/L\end{bmatrix}\frac{L}{2}.2 = \frac{EA}{L}\begin{bmatrix}1 & -1\\-1 & 1\end{bmatrix} \qquad (8.11)$$

***Two-Dimensional Elements***

A linear quadrilateral 4-noded planar element is shown in Figure 8.4-a. A model element of this kind can be defined in the natural coordinate system (η,μ), as shown in Figure 8.4-b. The coordinates of the model element in the natural system are chosen to range from -1 to +1 in both directions.

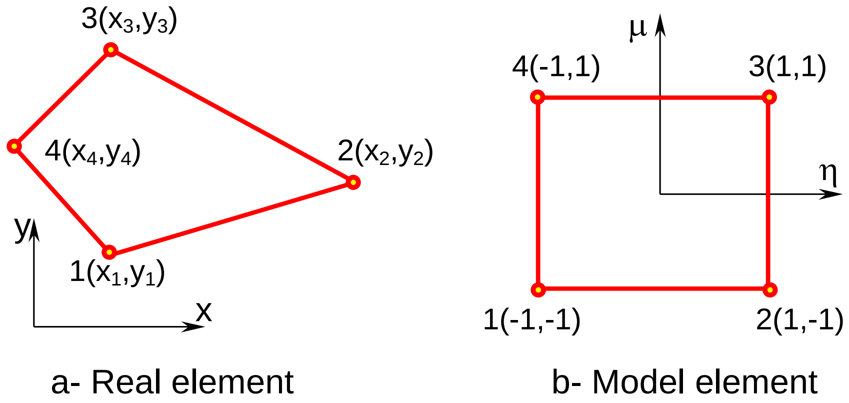



**Figure 8.4 Real and model linear quadrilateral elements**

The shape functions associated with the nodes of the model element can be defined as:

$$\begin{aligned}N_1 &= (1-\eta)(1-\mu)/4\\ N_2 &= (1+\eta)(1-\mu)/4\\ N_3 &= (1+\eta)(1+\mu)/4\\ N_4 &= (1-\eta)(1+\mu)/4\end{aligned} \qquad (8.12)$$

A point in the real element, with real coordinates x and y, can be associated with a point in the model element, with natural coordinates η and μ. The real coordinates of the point can then be obtained using the shape functions and the natural coordinates of the point:

$$x(\eta,\mu) = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4 = \mathbf{N}^T\mathbf{x} \qquad (8.13)$$

$$y(\eta,\mu) = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4 = \mathbf{N}^T\mathbf{y} \qquad (8.14)$$

Where $x = [x_1, x_2, x_3, x_4]^T$ and $y = [y_1, y_2, y_3, y_4]^T$ are vectors of the nodal coordinates in the x and y directions, respectively.

The formation of the strain-displacement matrix, $\mathbf{B}_e$, requires the derivatives of the shape functions with respect to the real coordinates x and y:

$$\frac{\partial N_i}{\partial x} = J^{-1}\frac{\partial N_i}{\partial \eta} \text{ and } \frac{\partial N_i}{\partial y} = J^{-1}\frac{\partial N_i}{\partial \mu}$$

It is therefore necessary to define the Jacobian matrix for two-dimensional cases:

$$\mathbf{J} = \begin{bmatrix} \partial x/\partial\eta & \partial y/\partial\eta \\ \partial x/\partial\mu & \partial y/\partial\mu \end{bmatrix} \tag{8.15}$$

The determinant of the Jacobian matrix is:

$$\det\mathbf{J} = \frac{\partial x}{\partial \eta}\frac{\partial y}{\partial \mu} - \frac{\partial x}{\partial \mu}\frac{\partial y}{\partial \eta} \tag{8.16}$$

Where the components of the Jacobian matrix can be calculated by differentiating Eq. (8.13) and Eq. (8.14):

$$\begin{aligned}
\frac{\partial x}{\partial \eta} &= \frac{\partial N_1}{\partial \eta}x_1 + \frac{\partial N_2}{\partial \eta}x_2 + \frac{\partial N_3}{\partial \eta}x_3 + \frac{\partial N_4}{\partial \eta}x_4 = \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}x_i \\
\frac{\partial x}{\partial \mu} &= \frac{\partial N_1}{\partial \mu}x_1 + \frac{\partial N_2}{\partial \mu}x_2 + \frac{\partial N_3}{\partial \mu}x_3 + \frac{\partial N_4}{\partial \mu}x_4 = \sum_{i=1}^{4}\frac{\partial N_i}{\partial \mu}x_i \\
\frac{\partial y}{\partial \eta} &= \frac{\partial N_1}{\partial \eta}y_1 + \frac{\partial N_2}{\partial \eta}y_2 + \frac{\partial N_3}{\partial \eta}y_3 + \frac{\partial N_4}{\partial \eta}y_4 = \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}y_i \\
\frac{\partial y}{\partial \mu} &= \frac{\partial N_1}{\partial \mu}y_1 + \frac{\partial N_2}{\partial \mu}y_2 + \frac{\partial N_3}{\partial \mu}y_3 + \frac{\partial N_4}{\partial \mu}y_4 = \sum_{i=1}^{4}\frac{\partial N_i}{\partial \mu}y_i
\end{aligned} \tag{8.17}$$

By substituting the derivatives of the shape functions with respect to $\eta$ and $\mu$ into the relations given by Eq.(8.17), the components of the Jacobian matrix and hence its determinant can be obtained.

$$J = \begin{bmatrix} \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}x_i & \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}y_i \\ \sum_{i=1}^{4}\frac{\partial N_i}{\partial \mu}x_i & \sum_{i=1}^{4}\frac{\partial N_i}{\partial \mu}y_i \end{bmatrix}$$

For the general case of an arbitrary-oriented quadrilateral, det**J** is a polynomial function of $\eta$ and $\mu$. Therefore, the value of det**J** varies within the quadrilateral element and needs to be determined at individual Gauss points.

By substituting the derivatives of the shape functions into the matrix $\mathbf{B}_e$, Eq. (7.30), the strain-displacement matrix is obtained as functions of $\eta$ and $\mu$. It also needs to be evaluated at individual Gauss points.

The procedure of integration to form the element stiffness matrix $\mathbf{k}^e$, can now be carried out with respect to the natural coordinates, $\eta$ and $\mu$. The differential of the area in the x, y system and $\eta$, $\mu$ system can be related by:

$$dx\,dy = \det\mathbf{J}\,d\eta\,d\mu$$

Therefore the required integral for calculation of the stiffness matrix can be written in the natural coordinate system as:

$$\mathbf{k}^e = t\int_y\int_x \mathbf{B}^T.\mathbf{D}.\,\mathbf{B}\,dx\,dy = t\int_{-1}^{+1}\int_{-1}^{+1} \mathbf{B}^T.\mathbf{D}.\,\mathbf{B}\,\det\mathbf{J}\,d\eta\,d\mu$$

The integration can then be carried out using a quadrature rule.

$$t\int_{-1}^{+1}\int_{-1}^{+1} \mathbf{B}^T.\mathbf{D}.\mathbf{B}\,\det\mathbf{J}\,d\eta\,d\mu = t\sum_{i=1}^{n} \mathbf{B}_i^T.\mathbf{D}.\mathbf{B}_i\,\det\mathbf{J}\,w_i$$

### *Problem 1*

A part of a pre-stressed concrete beam is shown in the figure below. The beam has a cross section area of 1m × 2m and a total length of 25 m. The pre-stressing cables apply a total end force of 24000 kN over an area of 0.8m×0.4m. The Young's modulus and Poisson's ratio for the concrete material is 30 GPa and 0.25, respectively.

You are required to find regions of high transverse tensile stress in the concrete beam due to the external compressive force of the cables. Prepare a suitable mesh to model the beam for a finite element analysis. The maximum number of nodes that you may use is limited to 2000. Solve the problem and answer the following questions:

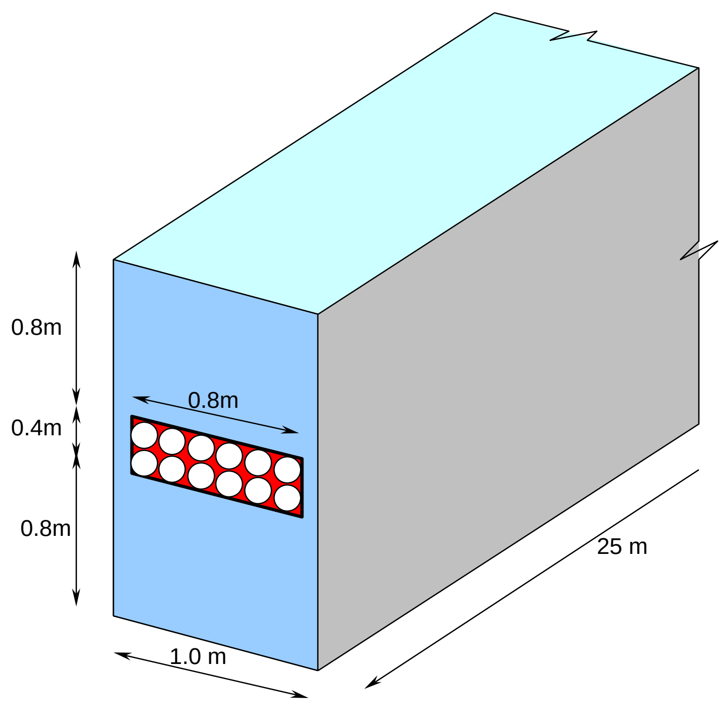
1- Is a plane stress or plane strain analysis more appropriate? Give reasons.

2- Discuss the degree of mesh refinement that is desirable in the several zones of the beam. Do you need to model the whole beam in order to find the tensile stresses?

3- Discuss the advantages of the finite element type that you have used in the analysis. Give the total number of nodes and elements used in your final analysis.

4- At what distance from the ends may the compressive stresses be expected to become approximately uniform?

5- Draw (or plot) the region where the transverse tensile stress is greater than 1000 kPa. Give the extent of the region from one end.



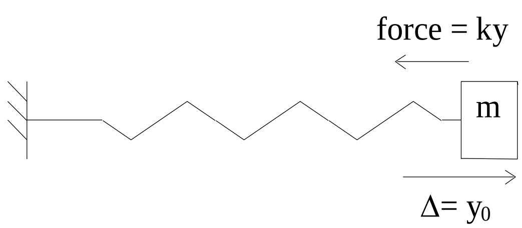
# Chapter 9
# Vibration of Structures

## 9.1 The Natural Frequency of a Structure

If a force is suddenly applied to a structure and then released (a transient excitation), the structure will vibrate at a unique frequency determined by its stiffness, called the ***natural frequency***.

If a structure is subject to sustained excitation, the vibration response of the structure will vary depending on the frequency of the sustained excitation. As the exciting frequency approaches the natural frequency of the structure, the movement of the structure will become magnified, because each application of the exciting force will add to the existing vibration of the structure. This is the phenomenon of ***resonance***.

1. Un-damped Oscillator

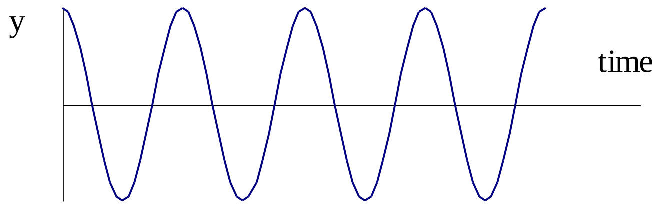



The equation of motion for an *un-damped* oscillator is

$$m\ddot{y} + ky = 0 \tag{9.1}$$

So for a system **initially at rest**,

$$y = y_0 \cos\left(\sqrt{\frac{k}{m}} \times t\right) \tag{9.2}$$

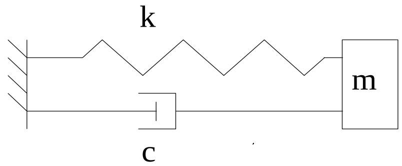



## 9.2 Damping

Damping dissipates energy and then reduces the duration and amplitude of vibrations. As the mass moves the force due to damping is proportional to the velocity, and acts in the opposite direction

$$F_{damping} = -c\dot{x} \quad \text{where c is the damping coefficient} \tag{9.3}$$

Representing the damping effect by a piston, the model of a mass and spring becomes,




The equation of motion for a viscously damped is,

$$m\ddot{x} + c\dot{x} + kx = 0 \tag{9.4}$$

The solution of this equation is of the form $x = x_0 e^{st}$. Substituting,

$$x_0 e^{st}(s^2 + \frac{c}{m}s + \frac{k}{m}) = 0$$

which is satisfied if $s^2 + \frac{c}{m}s + \frac{k}{m} = 0$. Therefore there are 2 roots,

$$s_1 = -\frac{c}{2m} + \sqrt{\left(\frac{c}{2m}\right)^2 - \frac{k}{m}}$$

$$s_2 = -\frac{c}{2m} - \sqrt{\left(\frac{c}{2m}\right)^2 - \frac{k}{m}}$$

The general solution will combine both of these,

$$x = Ae^{s_1 t} + Be^{s_2 t}$$

We can consider three cases for the solution, $\left(\frac{c}{2m}\right)^2 = \frac{k}{m}$, $\left(\frac{c}{2m}\right)^2 > \frac{k}{m}$, or $\left(\frac{c}{2m}\right)^2 < \frac{k}{m}$.

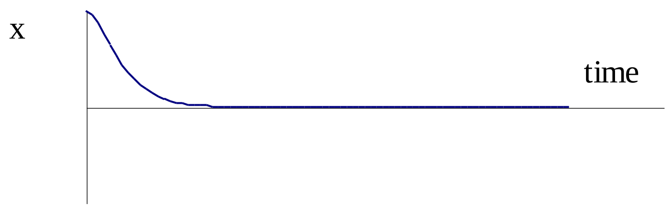
**Case 1: Critically damped**

This is a *special case* with a unique level of damping such that, $\left(\frac{c}{2m}\right)^2 = \frac{k}{m}$

That is $c = 2m\sqrt{\frac{k}{m}} = 2m\omega$. The roots are $s_1 = s_2 = -\frac{c}{2m}$ and in this case the solution is,

$$x = (A + Bt)e^{-\left(\frac{c}{2m}\right)t}$$

This response has the displacement, x decaying to zero in *exactly* one cycle without oscillation,

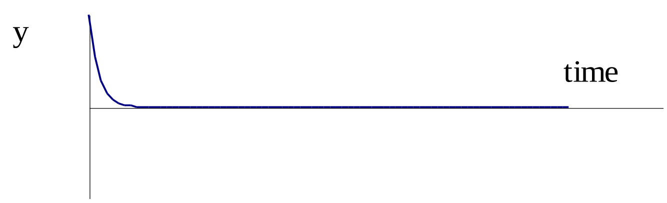



Structures will have less damping than this, but this is an important reference for specifying the level of damping, and is called the critical damping level.

$$\textbf{critical damping, } c_c = 2m\sqrt{\frac{k}{m}} \tag{9.5}$$

**Case 2: Overdamped**

The level of damping is greater than critical, so the displacement will die away even faster than the critically damped case. This is known as **overdamping**. Structures will always have less damping than this, so it is of no real interest.




**Case 3: Underdamped**

The level of damping is less than critical, so the displacement dies away slowly, $\left(\frac{c}{2m}\right)^2 < \frac{k}{m}$. This is known as **underdamped**, and is the situation that occurs with structures. The solution involves complex numbers,

$$s = -\frac{c}{2m} \pm i\sqrt{\frac{k}{m} - \left(\frac{c}{2m}\right)^2}$$

If we define

$$\textbf{damping ratio, } \xi \textbf{ = (actual damping)/(critical damping) = } \frac{c}{c_c} \tag{9.6}$$

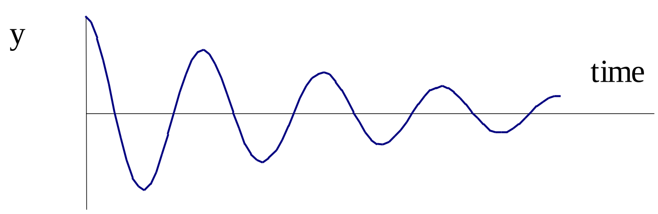
then it can be shown that the circular frequency of the *damped* system, $\omega_D = \sqrt{\frac{k}{m} - \left(\frac{c}{2m}\right)^2} = \omega\sqrt{1-\xi^2}$

(since $\omega = \sqrt{\frac{k}{m}}$)

and the solution for the underdamped case can then be written,

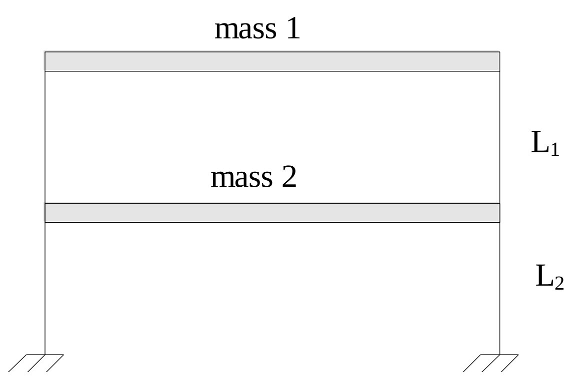
$$x = e^{-\xi\omega t}\left(x_0 \cos\omega_D t + \frac{\dot{x}_0 + x_0\xi\omega}{\omega_D}\sin\omega_D t\right)$$

A typical displacement trace looks like,




## 9.3 - Vibration of multiple degrees of freedom structures

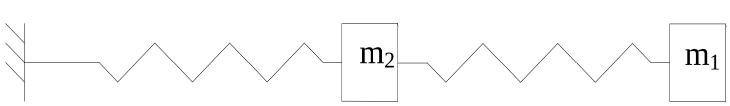
Consider, for example, the two storey building frame drawn below.




It is assumed that the beams are rigid, so the beam/column connection cannot rotate. The shape of the structure can therefore be defined by the horizontal translation of the upper storey and the horizontal translation of the lower storey (degrees of freedom 1 and 2 respectively).

Consistent with this assumption, the mass of each beam can be lumped at its centroid (a **lumped mass model**).

The model of a multi degree of freedom structure is often more generally represented by springs and masses as below,



and so there is an acceleration of the masses in accordance with Newton's second law,

$$\sum F_{m1} = -k_1 x_1 + k_1 x_2 = m\ddot{x}_1$$

$$\sum {}_{m2} = k_1 x_1 - \left(k_1 + k_2\right) x_2 = m\ddot{x}_2$$

which can be written in matrix form as,

$$\begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix}\begin{bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \end{bmatrix} + \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 + k_2 \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = 0$$

Recognising that $\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 + k_2 \end{bmatrix}$ is the stiffness matrix for this structure for these two degrees of freedom, it is possible to express the ***equation of motion for a multi degree of freedom structure*** in the more general form,

$$\mathbf{m}.\ddot{\mathbf{x}} + \mathbf{k}.\mathbf{x} = 0 \qquad (9.7)$$

## Determining the Natural Frequencies of the Structure

From our experience with single degree of freedom structures we can guess that each mass $m_i$ will exhibit simple harmonic motion, so that

$$x_i = A_i \sin\left(\sqrt{\frac{k}{m}} \times t - \alpha\right) = A_i \sin(\omega\, t - \alpha)$$

where $\omega = 2\pi\, f$ is the circular frequency and α is the phase shift. Differentiating,

$$\ddot{x}_i = -\omega^2 A_i \sin(\omega\, t - \alpha)$$

Hence the equation of motion for one mass in a multi degree of freedom structure can be written,

$$-\omega^2 m_i A_i \sin(\omega\, t - \alpha) + kA_i \sin(\omega\, t - \alpha) = 0$$

hence

$$-\omega^2 m_i A_i + kA_i = 0$$

Writing the equations for all the masses, and expressing it in matrix form gives,

$$-\omega^2 \mathbf{m}.\mathbf{A} + \mathbf{k}.\mathbf{A} = 0$$

or

$$\left[\mathbf{k} - \omega^2 \mathbf{m}\right].\mathbf{A} = 0$$

This has a trivial solution if $[A] = 0$. Another solution is obtained if the set of equations is *not* linearly independent, that is if the determinant of the matrix equals zero,

$$\left|\mathbf{k} - \omega^2 \mathbf{m}\right| = 0 \qquad (9.8)$$

It is possible to evaluate the determinant in terms of $\omega^2$ leading to a polynomial. Solving this polynomial gives the values of ω, and hence of the natural frequency, f.

The order of the polynomial, n, is the same as the number of degrees of freedom, and hence we obtain n independent values of the circular frequency (or of the natural frequency).

**A structure has the same number of natural frequencies as there are degrees of freedom to describe the displacements of the masses**

As an alternative to writing the polynomial equation and solving it, notice that the equation $\left|[\mathrm{k}]-\omega^2[\mathrm{m}]\right|=0$ can be rewritten as $\left|[\mathrm{m}]\right|\times\left|[\mathrm{m}]^{-1}[\mathrm{k}]-\omega^2[\mathrm{I}]\right|=0$ from which $\left|[\mathrm{m}]^{-1}[\mathrm{k}]-\omega^2[\mathrm{I}]\right|=0$. This is a statement of the eigenvalue problem - that is *each value of $\omega^2$ must be an eigenvalue of* $\left[[m]^{-1}[k]\right]$.

## Finding the Mode Shapes

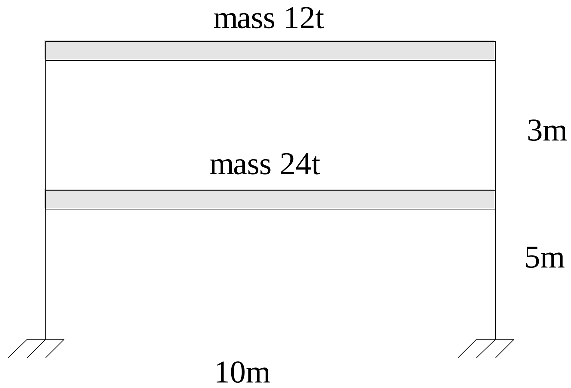
If a structure requires n degrees of freedom to describe the displacement of the masses, then it will have n natural frequencies. Each of these natural frequencies corresponds to a different "mode" of vibration. What are the shapes of the different modes?

To answer this question we need to find the coefficients $A_i$ in $[\mathrm{A}]$ because $x_i = A_i \sin(\omega t - \alpha)$. For each value of ω there is a corresponding matrix $[\mathrm{A}]$, and the values can be found by substituting for ω in $\left[[k]-\omega^2[m]\right][A]=0$ and solving for $[\mathrm{A}]$.

An alternative method for finding $[\mathrm{A}]$ is to remember that the values of $\omega^2$ were the ***eigenvalues*** of $\left[[m]^{-1}[k]\right]$. The values in the $[\mathrm{A}]$ matrix, $A_1$, $A_2$,.....$A_n$ are the ***eigenvectors*** of $\left[[m]^{-1}[k]\right]$. Thus a matrix manipulation software package can be used to find both the eigenvectors and the eigenvalues - that is the natural frequencies and the mode shapes.

### *Problem 1*

A two storey frame building with rigid beams has the dimensions and mass described below. The upper columns have a moment of inertia I = 44 x $10^6$ mm$^4$ and the lower columns 103 x $10^6$ mm$^4$. Determine the natural frequencies and the mode shapes.




There are two degrees of freedom, so there will be two natural frequencies and mode shapes. Dof 1 is the horizontal translation of the upper beam. Dof 2 is the horizontal translation of the lower beam.

# APPENDIX A

## A SUMMARY OF MATRIX ALGEBRA

A matrix is a set of numbers arranged in rows and columns. An r by c matrix A(rxc) is a matrix which has a total of r rows and c columns.
The following rules and definitions apply:

- A square matrix A(nxn) is a matrix which has as many rows as columns.
- An identity matrix I of dimensions nxn is a square matrix.
- A matrix A(rxc) can be multiplied by a scalar number p, by multiplying each of its elements by that number. The result is a matrix B=pA of the same dimensions, rxc, as A.
- Two matrices A ($r_a x c_a$) and B($r_b x c_b$) can be added together or subtracted from each other if and only if $r_a=r_b$ and $c_a=c_b$. The result is a matrix C with the same dimension as A and B.
- Two matrices A ($r_a x c_a$) and B($r_b x c_b$) can be multiplied by each other (AxB) if and only if $c_a=r_b$. The result is a matrix C with dimensions $r_a x c_b$.

## A SUMMARY OF MATRIX OPERATIONS BY MICROSOFT EXCEL

To multiply matrix M1 (mxn) by M2 (nxp) using Microsoft-Excel, do the following:

1. Enter values of matrices M1 (mxn). Select the matrix with the mouse (the area becomes black as you select it). Give the matrix a name (say, **mat1**) in the “name box” in the top left-hand corner of the sheet. Do the same for M2 (**mat2**)
2. Select a blank area the size of the resulting multiplication matrix (mxp) with the mouse
3. Type in: **=MMULT(mat1,mat2).**
   (If, for any reason, you haven’t given the matrices names, you can always select them as you are typing the function)
4. Press **Ctrl-Shift-Enter,** Results are then displayed in the selected area.

A similar procedure applies to other matrix functions.
Other useful MICROSOFT-EXCEL functions are:

**TRANSPOSE**(mat1), transpose of a matrix

**MDETERM**(mat1), determinant of a matrix (result is a scalar, no need to select area of resulting matrix and no need to type Ctrl-Shift-Enter, only Enter)

**MINVERSE**(mat1): inverse of a matrix

**MMULT**(mat1,mat2): multiplication of 2 matrices

*Exercise A1*

Given the following matrices:

$$A=\begin{bmatrix}1&2&-1\\2&2&3\\1&1&1\end{bmatrix}\quad B=\begin{bmatrix}2&0.5&1\\-7&6&2\\1&3&8\end{bmatrix}\quad C=\begin{bmatrix}1&1&-1\\2&-1&2\\2&5&1\end{bmatrix}\quad D=\begin{bmatrix}-1&2&1&-1\\1&9&3&-2\\2&1&-2&2\end{bmatrix}$$

$$E=\begin{bmatrix}1&2&2\\0&-1&3\end{bmatrix}\quad F=\begin{bmatrix}2&1&0\\-2&1&4\end{bmatrix}\quad G=\begin{bmatrix}-1&-3&8\\1&2&-5\\0&1&-2\end{bmatrix}$$

$$H=\begin{bmatrix}1&2&11&-1\\1&2&1&-1\\1&1&1&-2\\2&2&1&-2\\-1&9&2&2\end{bmatrix}\quad I=\begin{bmatrix}1&2&3&3&2&1\\2&1&2&2&1&1\\2&5&5&5&1&-1\end{bmatrix}\quad J=\begin{bmatrix}1&2\\2&2\\1&1\end{bmatrix}\quad b=\begin{pmatrix}2\\1\\1\end{pmatrix}$$

**A**: For each of the following operations indicate:

i: Whether the operation can be performed

ii: If it can be, the size of the resulting matrix? (do not perform any calculations).

1: [A]+[B] 2: [A]-[B]
3: [A]+[D] 4: [A]×C]
5: [A]×[H] 6: [H]×[A]
7: [A×[I] 8: $[D]^{-1}$
9: $[A]^{-1}$ 10: $[D]^{T}$
11: $[A]^{T}$

**B**: Perform manually and then verify with MS Excel the following operations:

1: [A]+[B] 2: [A] — [B] 3:[A]×[D]

**C**: Perform with MS Excel the following operations

1: [A]×[C] 2: [C]×[A] 3: [J]×([E]+[F])
4: [J]×[E]+[J]×[F] 5: [A]×[G]

**D**: Answer the following questions

1: From the previous operations, deduce the inverse $[A]^{-1}$ of matrix [A] without performing any calculations
2: Calculate the inverse $[A]^{-1}$ of matrix [A] with MS Excel
3: Calculate [x] in [A][x]=[b], where [x] is a 3x1 column vector

***Exercise A2:***

Given the following 6x6 matrix [M]:

$$M = \begin{bmatrix} 10 & 1 & 2 & 2 & 1 & 2 \\ 1 & 20 & 4 & 3 & 2 & 3 \\ 2 & 3 & 30 & 3 & 4 & 3 \\ 3 & 4 & 2 & 40 & 4 & 4 \\ 4 & 5 & 2 & 3 & 50 & 3 \\ 4 & 4 & 6 & 6 & 6 & 60 \end{bmatrix}, \qquad b = \begin{pmatrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{pmatrix}$$

You are required to perform the following operations using MS-Excel:

1. $[A]=[M]^T$
2. v = determinant ([A])
3. $[C]=[A]^{-1}$
4. Verify that the matrix inversion is correct by multiplying [A] by [C]
5. Calculate [x] such as [A][x]=[b] by multiplying $[A]^{-1}[b]$
6. Verify that operations are correct by multiplying [A][x] to get [b]

# APPENDIX B

### *B.1: Local and Global Coordinate Systems*

The position of a point in space is usually defined by nominating its coordinates (x, y, z) relative to a fixed set of reference axes. These coordinates are sometimes called the global coordinates.

In the course of analysis it may be more convenient to introduce a local set of coordinates. For example, it may be decided to measure x in the east direction, y in the north direction and z vertically. However in examining the behaviour of a particular piece of bracing it may be much more convenient to adopt a local set of axes with one of the coordinate axes directed along the centroid of bracing.

If a local set of coordinate axes is introduced it is desirable to be able to express the local coordinates in terms of the global coordinates and vice versa. For simplicity of presentation only the two dimensional situation will be considered here. A set of local axes derived from translation is shown in Figure .B.1

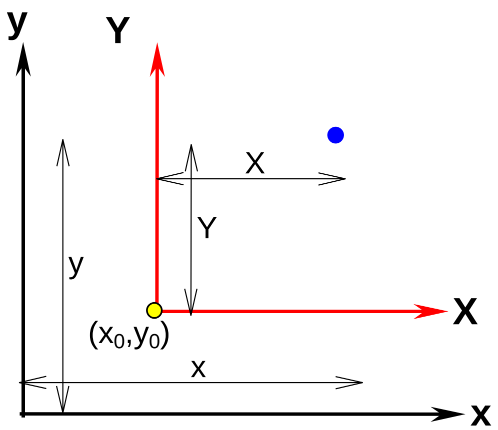



**Figure B.1: Translation of axes**

It can be seen that

$$X = x - x_0 \qquad \text{(B.1)}$$
$$Y = y - y_0$$

A set of local axes generated by anti-clockwise rotation through the angle θ is shown in Figure B.2.

It follows that for this case:

$$\begin{aligned} X &= +x\cos(\theta) + y\sin(\theta) \\ Y &= -x\sin(\theta) + y\cos(\theta) \end{aligned} \qquad \text{(B.2)}$$

or in matrix notation:

$$\begin{bmatrix} X \\ Y \end{bmatrix} = \begin{bmatrix} +\cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{bmatrix} x \\ y \end{bmatrix} \quad \text{(B. 3)}$$

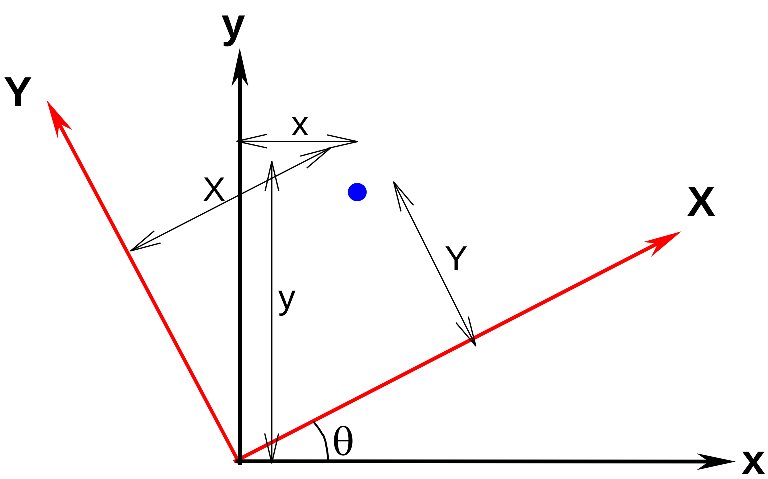



**Figure B.2: Rotation of axes**

The relationship between global and local coordinates is:

$$\begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) \\ \sin(\theta) & +\cos(\theta) \end{bmatrix} \begin{bmatrix} X \\ Y \end{bmatrix} \quad \text{(B.4)}$$

Or alternatively

$$\mathbf{r} = \mathbf{HR} \quad \text{(B.5)}$$

In Equation (B.5)

$$\mathbf{r} = \begin{bmatrix} x \\ y \end{bmatrix}, \quad \mathbf{R} = \begin{bmatrix} X \\ Y \end{bmatrix} \quad \text{and} \quad \mathbf{H} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) \\ \sin(\theta) & +\cos(\theta) \end{bmatrix}$$

Comparison of Equations (B. 3) and (B.4) show that the matrix **H** is orthogonal and so:

$$\mathbf{H}^{-1} = \mathbf{H}^{T}$$

In 3D:

$$\mathbf{H}^{T} = \begin{bmatrix} l_1 & m_1 & n_1 \\ l_2 & m_2 & n_2 \\ l_3 & m_3 & n_3 \end{bmatrix} \quad \text{(B.6)}$$

Where $l_1$, $l_2$ and $l_3$ are the cosines of the counter-clockwise angles between the x-axis and the X, Y and Z axes, respectively. $m_1$, $m_2$ and $m_3$ are the cosines of the counter-clockwise angles between the y-axis and the X, Y and Z axes, respectively, and so on.
In a right-angled Cartesian coordinate system, the following relationships must be satisfied:

$$l_1 l_2 + m_1 m_2 + n_1 n_2 = 0$$

$$l_2 l_3 + m_2 m_3 + n_2 n_3 = 0$$

$$l_3 l_1 + m_3 m_1 + n_3 n_1 = 0$$

***Example B.1***

To illustrate the introduction of local axes, suppose that on a particular site a set of global axes has been set up with the x-axis being in the horizontal plane and the y-axis vertically up. Bore hole data from the site shows the presence of a narrow layer of silt inclined at 20° to the horizontal. In examining the behaviour of the seam it is decided to introduce a local set of axes with its origin 10 m below the surface and with the X-axis directed along the seam (in a downwards direction) and the Y-axis perpendicular to the seam. The global coordinates have their origin at the point of intersection of the seam with the surface.

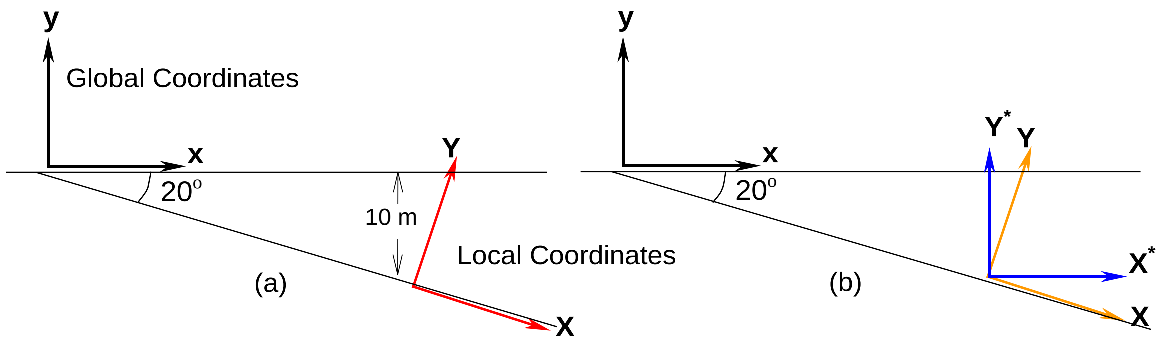



**Figure B.3: Rotation and Translation of axes**

The calculation is best performed in two stages. First consider the intermediate local coordinates (X*,Y*) shown in Figure B.(b). These coordinates are located at $x_0$=27.4748, $y_0$=-10.
Thus

$$X^* = x - 27.4748$$

$$Y^* = y + 10$$

The relationship between (X,Y) and (X*,Y*) can be found by rotation of axes (notice if you use Equation (B.2) that θ = -20°) and it is found that:

$$X = 0.9397X^* - 0.3402Y^*$$

$$Y = 0.3420X^* + 0.9397Y^*$$

This finally leads to the expression for local coordinate in terms of global coordinates:

$$X = 0.9397x - 0.3402y - 29.2380$$

$$Y = 0.3420x + 0.9397y$$

and to the expression of global coordinate in terms of local coordinates:

$$x = 0.9397X + 0.3402Y + 27.4748$$

$$y = -0.3420X + 0.9397Y - 10$$

## *B.2: Cylindrical Polar Coordinates*

The treatment given in the previous sections has been expressed in terms of cartesian coordinates. In many applications it is more convenient to employ curvilinear coordinates. Typical of these are the cylindrical polar coordinates r, θ, z, which are related to cartesian coordinates by the relation

$$x = r\cos\theta, \quad y = r\sin\theta$$

The coordinates are illustrated in Figure B.4

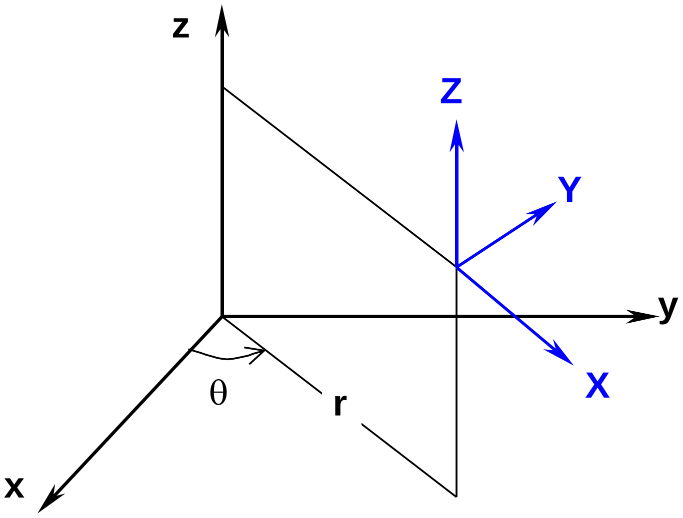



**Figure B.4: Polar coordinates**

### *Exercise B.1*

A rectangular element ABCD has vertices with coordinates as given below.

| | x(m) | y(m) |
|---|---|---|
| A | 2.0000 | 0.5000 |
| B | 6.3879 | 2.8971 |
| C | 1.5937 | 11.6730 |
| | -2.7943 | 9.2758 |

The coordinates of the point P is:

x(P)=1.2768 m          y(P)=8.0814m

If the origin of local coordinates is taken at A and thex-axis is directed along AB and the y-axis is directed along AD find the local coordinates of the point P.

*Answer: X(P) = 3m Y(P) = 7m*

## B.5: Transformation of Displacement

If forces are applied to a body it will deform as shown in Figure B. and thus a point originally at positionP will move to an adjacent position Q.

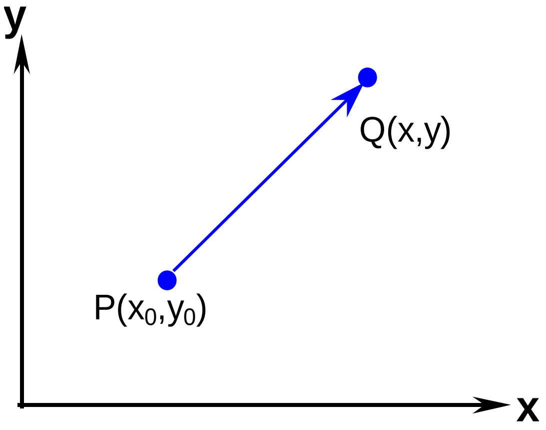



**Figure B.5: Displacement of a point**

The point is said to be displaced and the displacement is defined by:

$$u = r(Q) - r(P) \tag{B.7}$$

Thus the displacement components are given by:

$$\begin{bmatrix} u_x \\ u_y \end{bmatrix} = \begin{bmatrix} x(Q) - x(P) \\ y(Q) - y(P) \end{bmatrix} \tag{B.8}$$

These displacements are expressed in terms of the global coordinate system. It is often convenient to determine what the displacements are in a local coordinate system. Clearly a translation of axes does not change the displacement components, however a rotation of axes does induce a change. Therefore, it can be seen that:

$$\begin{aligned} u_X &= \ \ u_x \cos(\theta) + u_y \sin(\theta) \\ u_Y &= -u_x \sin(\theta) + u_y \cos(\theta) \end{aligned} \tag{B.9}$$

and conversely

$$\begin{aligned} u_x &= u_X \cos(\theta) - u_Y \sin(\theta) \\ u_y &= u_X \sin(\theta) + u_Y \cos(\theta) \end{aligned} \tag{B.10}$$

These equations may be written in matrix form as follows:

$$\begin{aligned} \mathbf{u} &= \mathbf{H}\,\mathbf{U} \\ \mathbf{U} &= \mathbf{H}^T\mathbf{u} \end{aligned} \tag{B.11}$$

Where

$\mathbf{u} = \begin{bmatrix} u_x \\ u_y \end{bmatrix}$ is the displacement vector in the global coordinates

$$\mathbf{U} = \begin{bmatrix} u_X \\ u_Y \end{bmatrix}$$ is the displacement vector in the local coordinates

$$\mathbf{H} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) \\ \sin(\theta) & +\cos(\theta) \end{bmatrix}$$

***Example B.2***

In the situation described in example 1.1 the following movements are recorded:

$$u_x = 18 \text{ mm}$$
$$u_y = -5 \text{ mm}$$

The local coordinate system can be used to assess if this movement corresponds to movement along the seam.
It is found that:

$$\begin{bmatrix} u_X \\ u_Y \end{bmatrix} = \begin{bmatrix} 0.9397 & -0.3420 \\ 0.3420 & +0.9397 \end{bmatrix} \begin{bmatrix} 18 \\ -5 \end{bmatrix} = \begin{bmatrix} 18.62 \; mm \\ 1.46 \; mm \end{bmatrix}$$

and thus there is a substantial movement along the seam.

***B.6: Rigid Body Displacement***

A body may undergo modes of movement in which there is no change of shape. Figure B. illustrates such a movement in the xy plane.

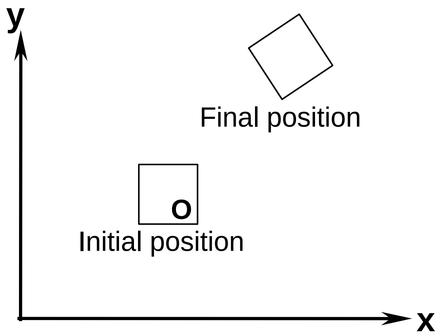



**Figure B.6: Rigid body movement**

It can be seen that the rigid body can be broken up into three distinct movements, a translation in the x direction, a translation in the y direction and a rotation about a line parallel to the z-axis through the point **O**. It can be shown that for small rotations

$$u_x = x_f - x_i = u_{xo} - (y_i - y_o)\, w_z$$
$$u_y = y_f - y_i = u_{yo} + (x_i - x_o)\, w_z$$

Where

$u_{xo}$ is the rigid body translation in the x direction

$u_{yo}$ is the rigid body translation in the y direction

$w_z$ is the rotation about the z axis

and $x_o$, $y_o$ are the coordinates of the reference point **O**.

### *Exercise B. 2*

The centre of a 5m radius silo (a *cylindrical* container) is located at a pointx=25m, y=10m, z=0m. The following deflections:

$$u_x = 10mm$$

$$u_y = 5mm$$

$$u_z = 2mm$$

are detected at the point x=29m, y=13m, z=10m.
Calculate the radial component of deflection.

*Answer: 11mm*

# APPENDIX C

## C.1: Direct Assembly of the Global Stiffness Matrix

It is not necessary to assemble the global stiffness matrix of the unrestrained structure. Instead, a system containing only the unrestrained degrees-of-freedom can be assembled to form the restrained stiffness matrix. There are several different strategies that can be adopted to incorporate boundary conditions while assembling the global stiffness matrix. Here an approach based on transformation of the local degrees-of-freedom to the global degrees-of-freedom is discussed in detail.

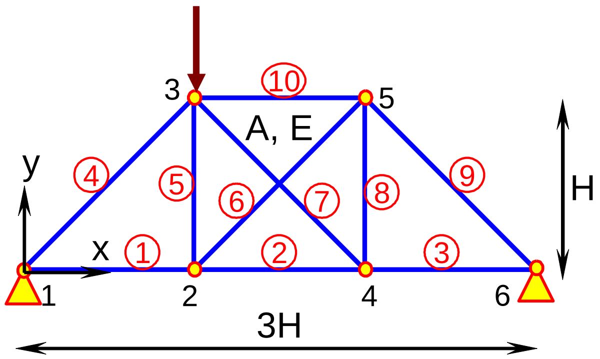



**Fig.C.1: Truss structure**

Each bar element has $n_{dof}$=4 local degrees-of-freedom, which is equal to the number of nodes in the element times the number of degrees-of-freedom per node. The structure has also $N_{dof}$ global degrees-of-freedom, which is equal to the number of nodes in the structure times the number of degrees-of-freedom per node less the number of restrained degrees-of-freedom. The number of the global degrees-of-freedom for the truss structure shown in Fig C.1 is $N_{dof} = 6 \times 2 - 4 = 8$. Assume that the unrestrained degrees-of-freedom can be rearranged as:

$$\Delta_R = \{a_1, a_2, a_3, a_4, a_5, a_6, a_7, a_8\}$$

By convention, the rearrangement of the global degrees-of-freedom is formed by going through all the nodes in ascending sequence and allocating an index number i to each degree of freedom that is unrestrained, $a_i$. The restrained degrees-of-freedom have a value of zero and do not contribute to the vector of the global degrees-of-freedom. For example the restrained and unrestrained degrees-of-freedom for the truss are:

| Unrestrained DOF | $u_1$ | $v_1$ | $u_2$ | $v_2$ | $u_3$ | $v_3$ | $u_4$ | $v_4$ | $u_5$ | $v_5$ | $u_6$ | $v_6$ |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Global DOF | 0 | 0 | $a_1$ | $a_2$ | $a_3$ | $a_4$ | $a_5$ | $a_6$ | $a_7$ | $a_8$ | 0 | 0 |

where $a_1$, for example, is the global label for $u_2$ and $a_2$ is the global label for $v_2$ etc. Therefore, the local degrees-of-freedom for each element can be related to the global degrees-of-freedom. For example for element 6:

| Local DOF for element 6 | $u_2$ | $v_2$ | $u_5$ | $v_5$ |
|---|---|---|---|---|
| Global DOF | $a_1$ | $a_2$ | $a_7$ | $a_8$ |

The vector of the local degrees-of-freedom can be related to the vector of the global degrees-of-freedom for the restrained structure by a transformation matrix, **Q**.

$$\delta^e = Q_e \,.\, \Delta_R \tag{C.1}$$

The size of the transformation matrix is ($n_{dof} \times N_{dof}$). The component $q_{ij}$ of matrix $\mathbf{Q_e}$ is1 if the $i^{th}$ degree-of-freedom of the element e (the local degrees-of-freedom) is equal to $j^{th}$ global degree-of-freedom, otherwise $q_{ij}$ is zero. For element 6, for example, $\mathbf{Q_6}$ is:

$$
\begin{array}{cccccccccc}
a_1 & a_2 & a_3 & a_4 & a_5 & a_6 & a_7 & a_8 & & \\
u_2 & v_2 & u_3 & v_3 & u_4 & v_4 & u_5 & v_6 & & \\
\end{array}
$$

$$
Q_6 = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}
\begin{array}{cc} u_2 & u_1^6 \\ v_2 & v_1^6 \\ u_5 & u_2^6 \\ v_5 & v_2^6 \end{array}
$$

The transformation matrices Q for all elements are given in Table C.1.

Substituting Equation(C.1) into Equations(2.8) and (2.10) results in:

$$\varepsilon = B.\delta^e = B.Q_e \,.\, \Delta_R \tag{C.2}$$

$$\sigma = D.B.\delta^e = D.B.Q_e \,.\, \Delta_R \tag{C.3}$$

where □ and □ are the stress and strain vectors, **B** is the matrix of strain-displacement relationship. Thus the equation of internal virtual work at the element level becomes:

$$\int \varepsilon^{*T} . \sigma \, dV = \int \Delta_R^{*T} . Q_e^T . B^T . D . B . Q_e . \Delta_R \, dV \tag{C.4}$$

**Table C.1: Transformation matrices for the elements**

| | |
|---|---|
| $Q_1 = \begin{bmatrix} 0&0&0&0&0&0&0&0 \\ 0&0&0&0&0&0&0&0 \\ 1&0&0&0&0&0&0&0 \\ 0&1&0&0&0&0&0&0 \end{bmatrix}$ | $Q_6 = \begin{bmatrix} 1&0&0&0&0&0&0&0 \\ 0&1&0&0&0&0&0&0 \\ 0&0&0&0&0&0&1&0 \\ 0&0&0&0&0&0&0&1 \end{bmatrix}$ |
| $Q_2 = \begin{bmatrix} 1&0&0&0&0&0&0&0 \\ 0&1&0&0&0&0&0&0 \\ 0&0&0&0&1&0&0&0 \\ 0&0&0&0&0&1&0&0 \end{bmatrix}$ | $Q_7 = \begin{bmatrix} 0&0&0&0&1&0&0&0 \\ 0&0&0&0&0&1&0&0 \\ 0&0&1&0&0&0&0&0 \\ 0&0&0&1&0&0&0&0 \end{bmatrix}$ |
| $Q_3 = \begin{bmatrix} 0&0&0&0&1&0&0&0 \\ 0&0&0&0&0&1&0&0 \\ 0&0&0&0&0&0&0&0 \\ 0&0&0&0&0&0&0&0 \end{bmatrix}$ | $Q_8 = \begin{bmatrix} 0&0&0&0&1&0&0&0 \\ 0&0&0&0&0&1&0&0 \\ 0&0&0&0&0&0&1&0 \\ 0&0&0&0&0&0&0&1 \end{bmatrix}$ |
| $Q_4 = \begin{bmatrix} 0&0&0&0&0&0&0&0 \\ 0&0&0&0&0&0&0&0 \\ 0&0&1&0&0&0&0&0 \\ 0&0&0&1&0&0&0&0 \end{bmatrix}$ | $Q_9 = \begin{bmatrix} 0&0&0&0&0&0&0&0 \\ 0&0&0&0&0&0&0&0 \\ 0&0&0&0&0&0&1&0 \\ 0&0&0&0&0&0&0&1 \end{bmatrix}$ |
| $Q_5 = \begin{bmatrix} 1&0&0&0&0&0&0&0 \\ 0&1&0&0&0&0&0&0 \\ 0&0&1&0&0&0&0&0 \\ 0&0&0&1&0&0&0&0 \end{bmatrix}$ | $Q_{10} = \begin{bmatrix} 0&0&1&0&0&0&0&0 \\ 0&0&0&1&0&0&0&0 \\ 0&0&0&0&0&0&1&0 \\ 0&0&0&0&0&0&0&1 \end{bmatrix}$ |

Application of the principle of virtual work eliminates the virtual displacement vector from Equation (C.4) and the element stiffness matrix is obtained in terms of the global degrees-of-freedom which is suitable to be used in the global stiffness matrix for the complete structure directly:

$$K_R^e = \int Q_e^T . B^T . D . B . Q_e \, dV \tag{C.5}$$

The unrestrained element stiffness matrix in terms of the local degrees-of-freedom was given in Equation (2.18) as:

$$k^e = \int B^T . D . B \, dV \tag{C.6}$$

Therefore, the relationship between the unrestrained element stiffness matrix, $\mathbf{k^e}$, and the restrained element stiffness matrix in terms of the global degrees-of-freedom, $\mathbf{K}_R^e$ □ can be obtained by comparing Equation (C.5) with Equation (C.6):

$$K_R^e = Q_e^T . k^e . Q_e \tag{C.7}$$

To see how the operation in Equation (C.7) forms the restrained stiffness matrix in terms of the global degrees-of-freedom from a local stiffness matrix, consider, for example, the local stiffness matrix of element 6:

$$
\begin{array}{c} \\ \\ k^6 = \end{array}
\begin{array}{c}
\begin{array}{cccc} a_1 & a_2 & a_7 & a_8 \\ u_2 & v_2 & u_5 & v_5 \end{array} \\
\begin{bmatrix}
k_{11}^6 & k_{12}^6 & k_{13}^6 & k_{14}^6 \\
k_{21}^6 & k_{22}^6 & k_{23}^6 & k_{24}^6 \\
k_{31}^6 & k_{32}^6 & k_{33}^6 & k_{34}^6 \\
k_{41}^6 & k_{42}^6 & k_{43}^6 & k_{44}^6
\end{bmatrix}
\end{array}
\begin{array}{cc} \\ \\ u_2 & a_1 \\ v_2 & a_2 \\ u_5 & a_7 \\ v_5 & a_8 \end{array}
$$

Each component of the stiffness matrix, $k_{ij}^6$, is tagged with one $a_r$ (a global degree-of-freedom associated with row i of the stiffness matrix) and one $a_c$ (a global degree-of-freedom associated with column j of the local stiffness matrix). The tags $a_r$ and $a_c$ show that $k_{ij}$ shall be assembled in row r and column c of the global stiffness matrix. For example, the operation in Equation (C.7) transforms component $k_{23}^6$ into a position at the second row (due to $a_2$) and the seventh column (due to $a_7$) of the global stiffness matrix. The restrained stiffness matrix for element 6 in the global system is:

$$
K_R^6 = \begin{bmatrix}
k_{11}^6 & k_{12}^6 & 0 & 0 & 0 & 0 & k_{13}^6 & k_{14}^6 \\
k_{21}^6 & k_{22}^6 & 0 & 0 & 0 & 0 & k_{23}^6 & k_{24}^6 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
k_{31}^6 & k_{32}^6 & 0 & 0 & 0 & 0 & k_{33}^6 & k_{34}^6 \\
k_{41}^6 & k_{42}^6 & 0 & 0 & 0 & 0 & k_{43}^6 & k_{44}^6
\end{bmatrix}
$$

The stiffness matrices of all the elements in the global system are presented in table C2. The sum of all the element stiffness matrices forms the global stiffness matrix for the restrained structure:

$$
K_R = \frac{EA}{H}\begin{bmatrix}
2+\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 0 & 0 & -1 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 & 0 & 0 & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\
0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -1 & 0 \\
0 & -1 & 0 & 1+\frac{1}{\sqrt{2}} & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
-1 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & 2+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 \\
0 & 0 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & -1 & 0 & 0 & 0 & 1+\frac{1}{\sqrt{2}} & 0 \\
-\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & 0 & -1 & 0 & 1+\frac{1}{\sqrt{2}}
\end{bmatrix}
$$

**Table C2: Element stiffness matrices in the global system**

| | |
|---|---|
| $K_R^1 = \frac{EA}{H}\begin{bmatrix} +1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ | $K_R^6 = \frac{EA}{2\sqrt{2}\,H}\begin{bmatrix} +1 & +1 & 0 & 0 & 0 & 0 & -1 & -1 \\ +1 & +1 & 0 & 0 & 0 & 0 & -1 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & -1 & 0 & 0 & 0 & 0 & +1 & +1 \\ -1 & -1 & 0 & 0 & 0 & 0 & +1 & +1 \end{bmatrix}$ |
| $K_R^2 = \frac{EA}{H}\begin{bmatrix} +1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 0 & +1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ | $K_R^7 = \frac{EA}{2\sqrt{2}\,H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & +1 & -1 & -1 & +1 & 0 & 0 \\ 0 & 0 & -1 & +1 & +1 & -1 & 0 & 0 \\ 0 & 0 & -1 & +1 & +1 & -1 & 0 & 0 \\ 0 & 0 & +1 & -1 & -1 & +1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ |
| $K_R^3 = \frac{EA}{H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & +1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ | $K_R^8 = \frac{EA}{H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & +1 & 0 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 & 0 & +1 \end{bmatrix}$ |
| $K_R^4 = \frac{EA}{2\sqrt{2}\,H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & +1 & +1 & 0 & 0 & 0 & 0 \\ 0 & 0 & +1 & +1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ | $K_R^9 = \frac{EA}{2\sqrt{2}\,H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & +1 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & -1 & +1 \end{bmatrix}$ |
| $K_R^5 = \frac{EA}{H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & +1 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & +1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ | $K_R^{10} = \frac{EA}{H}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & +1 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 0 & 0 & +1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$ |

# APPENDIX D

## D.1: Linear Triangular Elements

From the previous section 6.1, the 3-noded triangular element shown in Fig.B.1 is the simplest possible planar element and one of the earliest finite elements. It has nodes at the vertices of the triangle only. For a plane elasticity problem, where all displacements are in the plane, the element has two degrees-of-freedom at each node, u and v, corresponding to the displacements in x and y directions respectively. Thus the element --has a total of 6 degrees-of-freedom. The displacement vector and the force vector are:

$$\delta^e = \{u_1,\ v_1, u_2,\ v_2, u_3,\ v_3\}^T$$

$$f^e = \{p_1,\ q_1,\ p_2, q_2,\ p_3,\ q_3\}^T$$

Since each of these vectors contains 6 components, the size of the element stiffness matrix, $\mathbf{k^e}$, is 6×6.

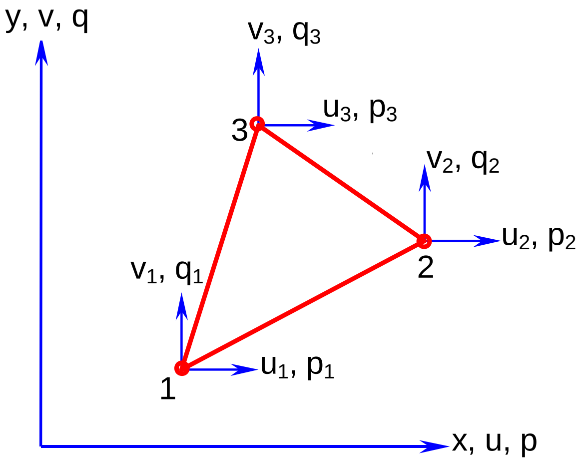



**Fig. B.1: 3-noded triangular element**

### *Stiffness matrix of linear triangular finite element*

The general procedure explained in section 2.3 is employed here to calculate the stiffness matrix of the 3-noded triangular element.

**4.** Local coordinate and node numbering system.

The node numbering and the Cartesian coordinate system shown in Fig. B.1 may be used for the element. The nodes are numbered in increasing order anti-clockwise. The coordinates of the nodes are $(x_1, y_1)$, $(x_2, y_2)$ and $(x_3, y_3)$. It is noted that the orientation of the element with respect to the xy coordinate system is completely arbitrary. Therefore the element stiffness matrix will be directly expressed in the xy global coordinate system.

**5.** Displacement function

The variation of the displacement components, u and v, within the element can be expressed as complete linear polynomials of x and y:

$$u = a_1 + a_2 x + a_3 y \ = \ f(x, y).a \qquad \textbf{(B.12)}$$

$$v = b_1 + b_2 x + b_3 y = f(x, y).b$$

where $f(x, y) = \{1, x, y\}$, $a = \{a_1, a_2, a_3\}^T$ and $b = \{b_1, b_2, b_3\}^T$.

**6.** Relating displacements within the element to the nodal displacements

The general displacements within the element can be related to the nodal displacements using shape functions:

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3 = N^T . u^e \quad (\mathbf{B}.13)$$

$$v = N_1 v_1 + N_2 v_2 + N_3 v_3 = N^T . v^e$$

where $u_i$ and $v_i$ are the nodal displacements in x and y directions, respectively, and $N_i$ are the linear shape functions for the element, as obtained in Chapter 3:

$$N^T = f(x, y).C^{-1}$$

$$C = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \text{ and } C^{-1} = \frac{1}{2A}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

where A is the area of the triangular element, $x_{1,2,3}$ and $y_{1,2,3}$ are the x and y coordinates of the first, the second and the third node of the element.

Therefore the shape functions are:

$$N^T = f^T(x, y)C^{-1} = \frac{1}{2A}\{1, \ x, \ y\}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}$$

$$N = \begin{Bmatrix} N_1 \\ N_2 \\ N_3 \end{Bmatrix} = \begin{Bmatrix} \dfrac{(x_2y_3 - x_3y_2) + x(y_2 - y_3) + y(x_3 - x_2)}{2A} \\ \dfrac{(x_3y_1 - x_1y_3) + x(y_3 - y_1) + y(x_1 - x_3)}{2A} \\ \dfrac{(x_1y_2 - x_2y_1) + x(y_1 - y_2) + y(x_2 - x_1)}{2A} \end{Bmatrix}$$

Equation (C.1) can now be written in matrix format as:

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \quad \text{or} \quad \delta(x, y) = N . \delta^e$$

**7.** Strain-displacement relationship

The strains at any point within the element, □(x,y), can be related to the nodal displacements, □$^e$, by the strain-displacement matrix, $\mathbf{B_e}$.

$$\varepsilon(x, y) = B_e . \delta^e \quad (\mathbf{B}.14)$$

The matrix $\mathbf{B_e}$ has been defined for a general case in section 5.1.2 and contains derivatives of the shape functions. For the general case of a three-dimensional element with m nodes, the strain vector has 6 components and the matrix $\mathbf{B_e}$ can be defined as:

$$B_e = \begin{bmatrix} N_{1x} & 0 & 0 & N_{2x} & 0 & 0 & \cdots & N_{mx} & 0 & 0 \\ 0 & N_{1y} & 0 & 0 & N_{2y} & 0 & \cdots & 0 & N_{my} & 0 \\ 0 & 0 & N_{1z} & 0 & 0 & N_{2z} & \cdots & 0 & 0 & N_{mz} \\ N_{1y} & N_{1x} & 0 & N_{2y} & N_{2x} & 0 & \cdots & N_{my} & N_{mx} & 0 \\ 0 & N_{1z} & N_{1y} & 0 & N_{2z} & N_{2y} & \cdots & 0 & N_{mz} & N_{my} \\ N_{1z} & 0 & N_{1x} & N_{2z} & 0 & N_{2x} & \cdots & N_{mz} & 0 & N_{mx} \end{bmatrix} \qquad (\mathbf{B}.15)$$

where $N_{ix}$, $N_{iy}$ and $N_{iz}$ are derivatives of shape function i with respect to x, y and z, respectively. Each row of the matrix $\mathbf{B_e}$ refers to one component of the strain vector. For planar problems, where some components of the strain vector are zero, the size of the matrix $\mathbf{B_e}$ can be reduced. For example, the strain vector under plane stress and plane strain conditions can be written as:

$$\varepsilon^e = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \partial u/\partial x \\ \partial v/\partial y \\ \partial v/\partial x + \partial u/\partial y \end{Bmatrix} \qquad (\mathbf{B}.16)$$

Therefore, the matrix $\mathbf{B_e}$ for these conditions can be obtained as:

$$B_e = \begin{bmatrix} N_{1x} & 0 & N_{2x} & 0 & \cdots & N_{mx} & 0 \\ 0 & N_{1y} & 0 & N_{2y} & \cdots & 0 & N_{my} \\ N_{1y} & N_{1x} & N_{2y} & N_{2x} & \cdots & N_{my} & N_{mx} \end{bmatrix} \qquad (\mathbf{B}.17)$$

For the triangular element with three nodes, the matrix $\mathbf{B_e}$ is:

$$B_e = \begin{bmatrix} N_{1x} & 0 & N_{2x} & 0 & N_{3x} & 0 \\ 0 & N_{1y} & 0 & N_{2y} & 0 & N_{3y} \\ N_{1y} & N_{1x} & N_{2y} & N_{2x} & N_{3y} & N_{3x} \end{bmatrix} \qquad (\mathbf{B}.18)$$

The derivatives of the shape functions for the triangular element can be obtained as:

$$\begin{Bmatrix} N_{1x} \\ N_{1y} \\ N_{2x} \\ N_{2y} \\ N_{3x} \\ N_{3y} \end{Bmatrix} = \frac{1}{2A} \begin{Bmatrix} (y_2 - y_3) \\ (x_3 - x_2) \\ (y_3 - y_1) \\ (x_1 - x_3) \\ (y_1 - y_2) \\ (x_2 - x_1) \end{Bmatrix} \qquad (\mathbf{B}.19)$$

Therefore the matrix $\mathbf{B_e}$ is obtained for the linear triangular element as:

$$B_e = \frac{1}{2A} \begin{bmatrix} (y_2 - y_3) & 0 & (y_3 - y_1) & 0 & (y_1 - y_2) & 0 \\ 0 & (x_3 - x_2) & 0 & (x_1 - x_3) & 0 & (x_2 - x_1) \\ (x_3 - x_2) & (y_2 - y_3) & (x_1 - x_3) & (y_3 - y_1) & (x_2 - x_1) & (y_1 - y_2) \end{bmatrix} \qquad (\mathbf{B}.20)$$

It can be seen that $\mathbf{B_e}$ and therefore strains within the linear triangular element are independent of x and y. For this reason, this element is often called the “constant strain triangle”.

**8.** Stress-strain relationship

The stress-strain relationships for continuum problems have been defined in section 5.3 as:

$$\sigma = D.\varepsilon \qquad (\textbf{B.}21)$$

where **D** is the matrix of elastic moduli. Expressions for **D** have been given for cases of general three-dimensional problems as well as plane strain, plane stress and axial symmetry problems. For example, **D** for plane strain problems is:

$$-\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} \lambda + 2G & \lambda & 0 \\ \lambda & \lambda + 2G & 0 \\ 0 & 0 & G \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \qquad (\textbf{B.}22)$$

where □ and G are Lamé modulus and shear modulus, respectively.

**9.** Relating the internal stress to the external loads

The internal stress can be related to the external loads using the principle of virtual work for the element. This leads to the equation for calculation of the element stiffness matrix.

$$k^e = \int B_e^T . D . B_e \, dv = B_e^T . D . B_e \; A . t \qquad (\textbf{B.}23)$$

where A and t are the area and the thickness of the element, respectively. Note that because $\mathbf{B_e}$ and **D** are independent of coordinate location (x, y), the integration over this element can be performed easily and exactly.

### *Application of linear triangular elements in analysis of a continuum*

A simple example is given here to demonstrate application of the linear triangular finite elements in the analysis of a continuum problem. The elastic body to be analysed is a simple homogeneous long block, a cross section of which is shown in Fig. B.2(a). It is constrained by a smooth rigid horizontal boundary and a smooth rigid vertical boundary along its two sides. The block is subjected to normal stresses applied to the other two sides. The Young's modulus, E, and the Poisson's ratio, □, for the block are 56 MPa and 0.4, respectively. The general procedure in finite element analysis, explained in section 2.2, will be used here for the analysis of the problem.

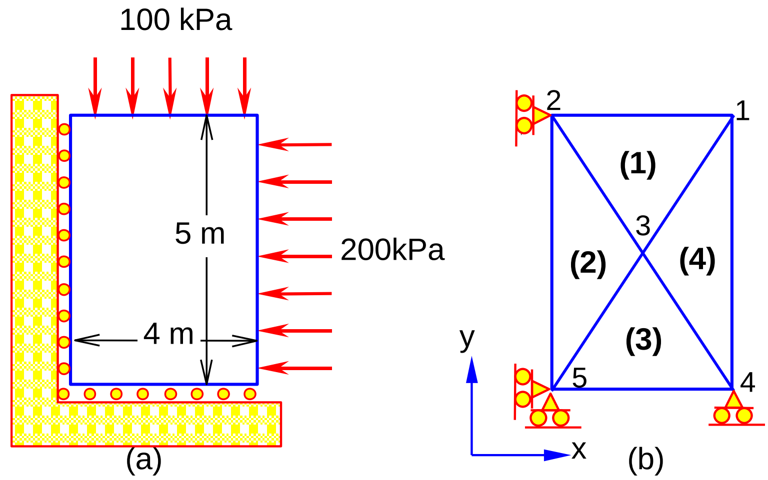



**Fig. B.2: Elastic block subjected to uniform loads**

1. Chose a suitable coordinate system

   The Cartesian coordinate system shown in Fig.B(b) is suitable for the problem.

2. Divide the geometry of the problem into a number of finite elements.

   The geometry is divided into 4 triangular elements as shown in Fig.B.2(b).

3. Use a suitable node numbering system.

   The node numbering system shown in Fig. B.2(b) is chosen. As explained in section 4.5, a good node numbering system should minimise the difference between the node numbers of any member that is a part of the structure.
   The nodal co-ordinates are shown in Table B.5. The data defining each of the elements is given in Table B.6.

**Table B.5: Nodal coordinates**

| Node | x (m) | y (m) |
|---|---|---|
| 1 | 4.0 | 5.0 |
| 2 | 0.0 | 5.0 |
| 3 | 2.0 | 2.5 |
| 4 | 4.0 | 0.0 |
| 5 | 0.0- | 0.0 |

**Table B.6: Element data**

| Element | Node 1 | Node 2 | Node 3 |
|---|---|---|---|
| 1 | 1 | 2 | 3 |
| 2 | 2 | 5 | 3 |
| 3 | 5 | 4 | 3 |
| 4 | 4 | 1 | 3 |

4. Calculate the stiffness matrices of all elements

The stiffness matrix of each element can be calculated using Equation (B.23), assuming a unit thickness for the element.

$$k^e = \int B_e^T . D . B_e \, dv = B_e^T . D . B_e \, A \qquad \text{(B.24)}$$

where **B**$_e$ and **D** can be calculated using Equations  REF Eq9  * MERGEFORMAT   and (B.22). Note that in Equation  REF Eq9  * MERGEFORMAT   $x_{1,2,3}$ and $y_{1,2,3}$ are the x and y coordinates of the first node, the second node and the third node of the element. For example, the matrix **B$_e$** for element 2 is calculated as follows.

$$B_2 = \frac{1}{2A}\begin{bmatrix} (y_5 - y_3) & 0 & (y_3 - y_2) & 0 & (y_2 - y_5) & 0 \\ 0 & (x_3 - x_5) & 0 & (x_2 - x_3) & 0 & (x_5 - x_2) \\ (x_3 - x_5) & (y_5 - y_3) & (x_2 - x_3) & (y_3 - y_2) & (x_5 - x_2) & (y_2 - y_5) \end{bmatrix}$$

and $2A = (x_5\, y_3 - x_3\, y_5) - (x_2\, y_3 - x_3\, y_2) + (x_2\, y_5 - x_5\, y_2)$

Substituting -the x and y coordinates of the three nodes in the above relations results in:

$2A = (0\times2.5 - 2\times0) - (0\times2.5 - 2\times5) + (0\times0 - 0\times5) = 10 \text{ m}^2$

$$B_2 = \frac{1}{10}\begin{bmatrix} (0-2.5) & 0 & (2.5-5) & 0 & (5-0) & 0 \\ 0 & (2-0) & 0 & (0-2) & 0 & (0-0) \\ (2-0) & (0-2.5) & (0-2) & (2.5-5) & (0-0) & (5-0) \end{bmatrix}$$

$$B_2 = \begin{bmatrix} -0.25 & 0 & -0.25 & 0 & 0.50 & 0 \\ 0 & 0.20 & 0 & -0.20 & 0 & 0 \\ 0.20 & -0.25 & -0.20 & -0.25 & 0 & 0.50 \end{bmatrix}$$

The matrices **B$_e$** for all elements are given in Table B.7.

**Table B.7: Strain-displacement matrices**

| | |
|---|---|
| | |

| | |
|---|---|
| $B_1 = \begin{bmatrix} 0.25 & 0 & -0.25 & 0 & 0 & 0 \\ 0 & 0.20 & 0 & 0.20 & 0 & -0.40 \\ 0.20 & 0.25 & 0.20 & -0.25 & -0.40 & 0 \end{bmatrix}$ | $B_2 = \begin{bmatrix} -0.25 & 0 & -0.25 & 0 & 0.50 & 0 \\ 0 & 0.20 & 0 & -0.20 & 0 & 0 \\ 0.20 & -0.25 & -0.20 & -0.25 & 0 & 0.50 \end{bmatrix}$ |
| $B_3 = \begin{bmatrix} -0.25 & 0 & 0.25 & 0 & 0 & 0 \\ 0 & -0.20 & 0 & -0.20 & 0 & 0.40 \\ -0.20 & -0.25 & -0.20 & 0.25 & 0.40 & 0 \end{bmatrix}$ | $B_4 = \begin{bmatrix} 0.25 & 0 & 0.25 & 0 & -0.50 & 0 \\ 0 & -0.20 & 0 & 0.20 & 0 & 0 \\ -0.20 & 0.25 & 0.20 & 0.25 & 0 & -0.50 \end{bmatrix}$ |

The matrix of elastic moduli, **D**, for plane strain analysis is:

$$D = \begin{bmatrix} \lambda + 2G & \lambda & 0 \\ \lambda & \lambda + 2G & 0 \\ 0 & 0 & G \end{bmatrix}$$

where $G = \dfrac{E}{2(1+\nu)}$ =20000 kPa and $\lambda = \dfrac{2G\nu}{(1-2\nu)}$ =80000 kPa. Therefore:

$$D = \begin{bmatrix} 120000 & 80000 & 0 \\ 80000 & 120000 & 0 \\ 0 & 0 & 20000 \end{bmatrix}$$

In this problem the material is assumed to be homogeneous and therefore the matrix of elastic moduli, **D**, is the same for all elements.

The stiffness matrices for all elements are calculated based on Equation (B**.**24) and presented in TableB.8.

**Table B.8: Element stiffness matrices**

$$k^1 = \begin{bmatrix} 41500 & 25000 & -33500 & 15000 & -8000 & -40000 \\ & 30250 & -15000 & 17750 & -10000 & -48000 \\ & & 41500 & -25000 & -8000 & 40000 \\ & & & 30250 & 10000 & -48000 \\ & \text{Sym.} & & & 16000 & 0 \\ & & & & & 96000 \end{bmatrix}$$

$$k^2 = \begin{bmatrix} 41500 & -25000 & 33500 & 15000 & -75000 & 10000 \\ & 20250 & -15000 & -17750 & 40000 & -12500 \\ & & 41500 & 25000 & -75000 & -10000 \\ & & & 30250 & -40000 & -12500 \\ & \text{Sym.} & & & 15000 & 0 \\ & & & & & 25000 \end{bmatrix}$$

$$k^3 = \begin{bmatrix} 41500 & 25000 & -33500 & 15000 & -8000 & -40000 \\ & 30250 & -15000 & 17750 & -10000 & -48000 \\ & & 41500 & -25000 & -8000 & 40000 \\ & & & 30250 & 10000 & -48000 \\ & \text{Sym.} & & & 16000 & 0 \\ & & & & & 96000 \end{bmatrix}$$

$$k^4 = \begin{bmatrix} 41500 & -25000 & 33500 & 15000 & -75000 & 10000 \\ & 30250 & -15000 & -17750 & 40000 & -12500 \\ & & 41500 & 25000 & -75000 & -10000 \\ & & & 30250 & -40000 & -12500 \\ & Sym. & & & 15000 & 0 \\ & & & & & 25000 \end{bmatrix}$$

5. Assemble the global stiffness matrix

The global stiffness matrix is assembled using the direct method of assembly explained in section 4.2. The unrestrained degrees-of-freedom and the global degrees-of-freedom for the whole structure are:

| Unrestrained DOF | $u_1$ | $v_1$ | $u_2$ | $v_2$ | $u_3$ | $v_3$ | $u_4$ | $v_4$ | $u_5$ | $v_5$ |
|---|---|---|---|---|---|---|---|---|---|---|
| Global DOF | $a_1$ | $a_2$ | 0 | $a_3$ | $a_4$ | $a_5$ | $a_6$ | 0 | 0 | 0 |

Therefore the unknown displacements, or the global variables, have six components:

$$\Delta_R = \{a_1, a_2, a_3, a_4, a_5, a_6\}^T$$

where $a_4$, for example, is the global label for $u_3$.
The local degrees-of-freedom for each element can be related to the global degrees-of-freedom. For example, for element 2:

| Local DOF for element 2 | $u_2$ | $v_2$ | $u_5$ | $v_5$ | $u_3$ | $v_3$ |
|---|---|---|---|---|---|---|
| Global DOF | 0 | $a_3$ | 0 | 0 | $a_4$ | $a_5$ |

The vector of the local DOF can be related to the vector of the global DOF for the restrained structure by a transformation matrix, $\mathbf{Q_e}$.

$$\delta^e = Q_e \, . \, \Delta_R$$

For example, $\mathbf{Q_e}$ for element 2 is:

$$\begin{array}{cccccccc} & a_1 & a_2 & a_3 & a_4 & a_5 & a_6 & \\ & u_1 & v_1 & v_2 & u_3 & v_3 & u_4 & \\ & 0 & 0 & 0 & 0 & 0 & 0 & u_2 \\ & 0 & 0 & 1 & 0 & 0 & 0 & v_2 \\ Q_2 = & 0 & 0 & 0 & 0 & 0 & 0 & u_5 \\ & 0 & 0 & 0 & 0 & 0 & 0 & v_5 \\ & 0 & 0 & 0 & 1 & 0 & 0 & u_3 \\ & 0 & 0 & 0 & 0 & 1 & 0 & v_3 \end{array}$$

The transformation matrices, $\mathbf{Q_e}$, for all elements are given in Table B.9.

**Table B.9: Transformation matrices for all elements**

| Element 1: | Element 2: |
|---|---|

| | |
|---|---|
| $\begin{bmatrix} u_1 = a_1 \\ v_1 = a_2 \\ u_2 = 0 \\ v_2 = a_3 \\ u_3 = a_4 \\ v_3 = a_5 \end{bmatrix} \quad Q_1 = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix}$ | $\begin{bmatrix} u_2 = 0 \\ v_2 = a_3 \\ u_5 = 0 \\ v_5 = 0 \\ u_3 = a_4 \\ v_3 = a_5 \end{bmatrix} \quad Q_2 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix}$ |
| Element 3: $\begin{bmatrix} u_5 = 0 \\ v_5 = 0 \\ u_4 = a_6 \\ v_4 = 0 \\ u_3 = a_4 \\ v_3 = a_5 \end{bmatrix} \quad Q_3 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix}$ | Element 4: $\begin{bmatrix} u_4 = a_6 \\ v_4 = 0 \\ u_1 = a_1 \\ v_1 = a_2 \\ u_3 = a_4 \\ v_3 = a_5 \end{bmatrix} \quad Q_4 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix}$ |

The restrained stiffness matrix for an element, expressed in terms of global variables, $\mathbf{K}_R^e$, can be obtained from the element transformation matrix and the element stiffness matrix, expressed in terms of local variables, using the relationship given in Equation(4.10):

$$K_R^e = Q_e^T \, . \, k^e \, . \, Q_e$$

The restrained element stiffness matrices for all elements are given in Table B.10.

**Table B.10: Restrained element stiffness matrices**

| |
|---|
| $k_R^1 = \begin{bmatrix} 41500 & 25000 & 15000 & -8000 & -40000 & 0 \\ & 30250 & 17750 & -10000 & -48000 & 0 \\ & & 30250 & 10000 & -48000 & 0 \\ & & & 16000 & 0 & 0 \\ & \text{sym.} & & & 96000 & 0 \\ & & & & & 0 \end{bmatrix}$ |
| $k_R^2 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ & 0 & 0 & 0 & 0 & 0 \\ & & 30250 & 40000 & -12500 & 0 \\ & & & 150000 & 0 & 0 \\ & \text{sym.} & & & 25000 & 0 \\ & & & & & 0 \end{bmatrix}$ |
| $k_R^3 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ & 0 & 0 & 0 & 0 & 0 \\ & & 0 & 0 & 0 & 0 \\ & & & 16000 & 0 & -8000 \\ & \text{sym.} & & & 96000 & 40000 \\ & & & & & 41500 \end{bmatrix}$ |
| |

$$k_R^4 = \begin{bmatrix} 41500 & 25000 & 0 & -75000 & -10000 & 33500 \\ & 30250 & 0 & -40000 & -12500 & 15000 \\ & & 0 & 0 & 0 & 0 \\ & & & 15000 & 0 & -75000 \\ & sym. & & & 25000 & 10000 \\ & & & & & 41500 \end{bmatrix}$$

The Global stiffness matrix for the whole structure, expressed in terms of the global variables, is obtained by summing the element stiffness matrices.

$$K_R = \begin{bmatrix} 83000 & 50000 & 15000 & -83000 & -50000 & 33500 \\ 50000 & 60500 & 17750 & -50000 & -60500 & 15000 \\ 15000 & 17750 & 60500 & 50000 & -60500 & 0 \\ -83000 & -50000 & 50000 & 332000 & 0 & -83000 \\ -50000 & -60500 & -60500 & 0 & 242000 & 50000 \\ 33500 & 15000 & 0 & -83000 & 50000 & 83000 \end{bmatrix}$$

6. Assemble the force vector

The finite element equation was obtained in Chapter 2 using the principle of virtual work for the case where the external forces were applied at nodal points only. The finite element equation can be expanded to include the effects of any traction, T, applied on the surface of an element or any body force, □, acting within the body of an element.

To make the nodal forces statically equivalent to the actual boundary tractions or body forces, the principle of virtual work is employed. An arbitrary virtual nodal displacement is imposed to the body and the external work done by the various forces and tractions during that displacement are calculated and equated to the internal virtual work.

Lets assume that the virtual displacement □$*^e$ is applied at the nodes of an element. This results in virtual displacements, □*, and virtual strains, □*, within the element:

$$\delta^*(x, y) = N \,.\, \delta^{*e} \quad \text{and} \quad \varepsilon^*(x, y) = B \,.\, \delta^{*e}$$

The work done by the nodal forces is equal to the sum of the products of the individual force at each node and the corresponding displacement.

$$(W_{ext})_1 = \delta^{*eT} . f^e$$

where $f^e$ is the vector of nodal forces. The external virtual work done by tractions per unit area and the external virtual work done by distributed body forces per unit volume are:

$$(W_{ext})_2 = \delta^{*T}\, T = \delta^{*eT}\, N^T . T$$

$$(W_{ext})_3 = \varepsilon^{*T} . \gamma = \delta^{*eT}\, B^T . \gamma$$

Equating the total external work with the total internal work obtained by integrating over the volume of the element results in a more general finite element equation:

$$k^e . \delta^e = f^e + \int N^T . T \, ds + \int B^T . \gamma \, dv \qquad (\mathbf{B.}25)$$

The expression on the right-hand-side of Equation (B.25) may be used to calculate the “consistent nodal forces” for the element.

For the problem of the long block, the external tractions are applied at elements 1 and 4. The equivalent nodal forces due to the tractions are calculated for each of the elements and then included into the global force vector.
Element 1 is subjected to an external uniform traction in the y-direction, $T_y$=-100kPa.

$$T = \begin{Bmatrix} T_x \\ T_y \end{Bmatrix} = \begin{Bmatrix} 0 \\ -100 \end{Bmatrix} \text{kPa}$$

The shape functions for element 1 are:

$$N = \begin{Bmatrix} N_1 \\ N_2 \\ N_3 \end{Bmatrix} = \begin{Bmatrix} \frac{(x_2y_3 - x_3y_2) + x(y_2 - y_3) + y(x_3 - x_2)}{2A} \\ \frac{(x_3y_1 - x_1y_3) + x(y_3 - y_1) + y(x_1 - x_3)}{2A} \\ \frac{(x_1y_2 - x_2y_1) + x(y_1 - y_2) + y(x_2 - x_1)}{2A} \end{Bmatrix} = \begin{Bmatrix} 0.25x + 0.2y - 1 \\ -0.25x + 0.2y \\ -0.4y + 2 \end{Bmatrix}$$

Therefore

$$f^1 = \begin{Bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \\ p_3 \\ q_3 \end{Bmatrix} = \int N^T.T.dx = \int \begin{bmatrix} N_1 & 0 \\ 0 & N_1 \\ N_2 & 0 \\ 0 & N_2 \\ N_3 & 0 \\ 0 & N_3 \end{bmatrix} \begin{Bmatrix} T_x \\ T_y \end{Bmatrix} dx$$

The traction is applied on the top surface of element 1 which has a constant y coordinate, y = 5m. For unit thickness of the element, the surface can be described as:

$$ds = dx, \quad x = 0 \rightarrow 4\text{m} \quad \text{and} \quad y = 5\text{m}$$

Therefore the consistent nodal forces are calculated for a unit thickness of the element by the following equation:

$$\int N^T.T\,ds = \int N^T.T\,dx$$

The shape functions shall be expressed for the surface, a cross section of which connects node 1 to node 2, where y = 5 m:

$$N = \begin{Bmatrix} N_1 \\ N_2 \\ N_3 \end{Bmatrix} = \begin{Bmatrix} 0.25x \\ 1 - 0.25x \\ 0 \end{Bmatrix}$$

Therefore the consistent nodal forces for element 1 can be calculated as:

$$f^1 = \begin{Bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \\ p_3 \\ q_3 \end{Bmatrix} = \int N^T.T.dx = \int \begin{bmatrix} N_1 & 0 \\ 0 & N_1 \\ N_2 & 0 \\ 0 & N_2 \\ N_3 & 0 \\ 0 & N_3 \end{bmatrix} \begin{Bmatrix} T_x \\ T_y \end{Bmatrix} dx = \int \begin{bmatrix} 0.25x & 0 \\ 0 & 0.25x \\ 1 - 0.25x & 0 \\ 0 & 1 - 0.25x \\ 0 & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ -100 \end{Bmatrix} dx$$

$$f^1 = \begin{Bmatrix} p_1 \\ q_1 \\ p_2 \\ q_2 \\ p_3 \\ q_3 \end{Bmatrix} = \begin{bmatrix} 0.125x^2 & 0 \\ 0 & 0.125x^2 \\ x - 0.125x^2 & 0 \\ 0 & x - 0.125x^2 \\ 0 & 0 \\ 0 & 0 \end{bmatrix}_{x=0\to 4} \times \begin{Bmatrix} 0 \\ -100 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -200 \\ 0 \\ -200 \\ 0 \\ 0 \end{Bmatrix} \text{kN/m}$$

The nodal forces applied on element 1 can be written in terms of the global degrees-of-freedom as:

$$f_R^1 = Q_1^T.f^1 = \begin{Bmatrix} f_{a1}^1 \\ f_{a2}^1 \\ f_{a3}^1 \\ f_{a4}^1 \\ f_{a5}^1 \\ f_{a6}^1 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -200 \\ -200 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \text{kN/m}$$

Similarly, the consistent nodal forces for element 4 can also be obtained:

$$f^4 = \begin{Bmatrix} p_4 \\ q_4 \\ p_1 \\ q_1 \\ p_3 \\ q_3 \end{Bmatrix} = \begin{Bmatrix} -500 \\ 0 \\ -500 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \text{kN/m} \quad \text{and} \quad f_R^4 = Q_4^T f^4 = \begin{Bmatrix} f_{a1}^4 \\ f_{a2}^4 \\ f_{a3}^4 \\ f_{a4}^4 \\ f_{a5}^4 \\ f_{a6}^4 \end{Bmatrix} = \begin{Bmatrix} -500 \\ 0 \\ 0 \\ 0 \\ 0 \\ -500 \end{Bmatrix} \text{kN/m}$$

The nodal forces applied on elements and expressed in terms of global degrees-of-freedom can now be added together directly to form the global force vector.

$$F_R = \begin{Bmatrix} f_{a1} \\ f_{a2} \\ f_{a3} \\ f_{a4} \\ f_{a5} \\ f_{a6} \end{Bmatrix} = \begin{Bmatrix} -500 \\ -200 \\ -200 \\ 0 \\ 0 \\ -500 \end{Bmatrix}$$

7. Solve the global equations to obtain the unknown nodal displacements

The finite element equations can now be solved for the unknown nodal displacements

$$K_R \,.\, \Delta_R = F_R$$

or

$$\begin{bmatrix} 83000 & 50000 & 15000 & -83000 & -50000 & 33500 \\ & 60500 & 17750 & -50000 & -60500 & 15000 \\ & & 60500 & 50000 & -60500 & 0 \\ & & & 332000 & 0 & -83000 \\ & \text{sym.} & & & 242000 & 50000 \\ & & & & & 83000 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{Bmatrix} = \begin{Bmatrix} -500 \\ -200 \\ -200 \\ 0 \\ 0 \\ -500 \end{Bmatrix}$$

The unknown displacements can be obtained as:

$$\Delta_R = K_R^{-1} . F_R$$

$$\begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{Bmatrix} = \frac{1}{1000} \begin{bmatrix} 0.3182 & -0.1716 & -0.0284 & 0.0384 & 0.0319 & -0.0782 \\ & 0.3939 & -0.0189 & 0.0122 & 0.0642 & -0.0284 \\ & & 0.3939 & -0.1122 & 0.1233 & -0.1716 \\ & & & 0.0788 & -0.0339 & 0.0816 \\ & sym. & & & 0.1220 & -0.1319 \\ & & & & & 0.3182 \end{bmatrix} \begin{Bmatrix} -500 \\ -200 \\ -200 \\ 0 \\ 0 \\ -500 \end{Bmatrix} = \begin{Bmatrix} -0.008 \\ 0.0025 \\ 0.0025 \\ -0.004 \\ 0.00125 \\ -0.008 \end{Bmatrix}$$

Then the nodal displacements are:

| Unrestrained DOF | $u_1$ | $v_1$ | $u_2$ | $v_2$ | $u_3$ | $v_3$ | $u_4$ | $v_4$ | $u_5$ | $v_5$ |
|---|---|---|---|---|---|---|---|---|---|---|
| Global DOF | -0.008 | 0.0025 | 0 | 0.0025 | -0.004 | 0.00125 | -0.008 | 0 | 0 | 0 |

8. Calculate strains and stresses for each element

The nodal displacements can be used to find the strains and the stresses within each element. For example consider element 2. The nodal displacements for element 2 are:

| Local DOF for element 2 | $u_2$ | $v_2$ | $u_5$ | $v_5$ | $u_3$ | $v_3$ |
|---|---|---|---|---|---|---|
| Global DOF | 0 | 0.0025 | 0 | 0 | -0.004 | 0.00125 |

So that the vector of nodal displacements for element 2 is:

$$\delta^2 = Q_2 . \Delta_R = \{0, \ 0.0025, \ 0, \ 0, \ -0.004, \ 0.00125\}^T$$

The strains for element 2 can be calculated as:

$$\varepsilon(x, y) = B_2 . \delta^2$$

$$B_2 = \begin{bmatrix} -0.25 & 0 & -0.25 & 0 & 0.50 & 0 \\ 0 & 0.20 & 0 & -0.20 & 0 & 0 \\ 0.20 & -0.25 & -0.20 & -0.25 & 0 & 0.50 \end{bmatrix}$$

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} -0.25 & 0 & -0.25 & 0 & 0.50 & 0 \\ 0 & 0.20 & 0 & -0.20 & 0 & 0 \\ 0.20 & -0.25 & -0.20 & -0.25 & 0 & 0.50 \end{bmatrix} \begin{Bmatrix} 0 \\ 0.0025 \\ 0 \\ 0 \\ -0.004 \\ 0.00125 \end{Bmatrix} = \begin{Bmatrix} -0.002 \\ 0.0005 \\ 0 \end{Bmatrix}$$

Note that the strains are independent of the coordinates, i.e., the strains are constant within the element.

Once the strains are known the stresses can be found as:

$$\sigma(x, y) = D . \varepsilon(x, y)$$

$$D = \begin{bmatrix} 120000 & 80000 & 0 \\ 80000 & 120000 & 0 \\ 0 & 0 & 20000 \end{bmatrix}$$

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} 120000 & 80000 & 0 \\ 80000 & 120000 & 0 \\ 0 & 0 & 20000 \end{bmatrix} . \begin{Bmatrix} -0.002 \\ 0.0005 \\ 0 \end{Bmatrix} = \begin{Bmatrix} -200 \\ -100 \\ 0 \end{Bmatrix} \text{kPa}$$

Similar calculations can be made for all elements.

# SOLUTIONS TO SELECTED PROBLEMS

### Problem 2.1.

$$\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{F_0 \Delta x^2}{E_0} \begin{bmatrix} 4 \\ 7 \\ 9 \\ 10 \end{bmatrix}$$

### Problem 2.2.

$$\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \frac{F_0 \Delta x^2}{E} \begin{bmatrix} 3.5 \\ 6 \\ 7.5 \\ 8 \end{bmatrix}$$

### Problem 2.3.

- FEM solution is exact in the nodes.
- FDM has numerical error between nodes.
- Error decreases as the number of elements increases.

### Problem 2.4,

1)

Strong form: $\frac{d}{dx}\left[E(x)\frac{du}{dx}\right] = -\gamma \qquad u(0)=0 \qquad \sigma(H)=P$

Weak form: $Pu^*(H) - \int_0^H E(x)\frac{du}{dx}\frac{du^*}{dx}dx + \int_0^H \gamma u^*(x)dx$

2)

$$\frac{1}{\Delta x}\begin{bmatrix} 2E_1 & -\frac{E_1+E_2}{2} & 0 \\ -\frac{E_1+E_2}{2} & 2E_2 & -\frac{E_2+E_3}{2} \\ 0 & -\frac{E_2+E_3}{2} & \frac{E_2+E_3}{2} \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \gamma\Delta x\begin{bmatrix} 1 \\ 1 \\ \frac{1}{2} \end{bmatrix} + P\begin{bmatrix} 0 \\ 0 \\ 1 \end{bmatrix}$$

3) Displacement versus position

$$u(x) = \frac{\alpha H\gamma + E\gamma + \alpha P}{\alpha^2}\ln\left[\frac{E_0+\alpha x}{E_0}\right] - \frac{\gamma x}{\alpha}$$

Settlement versus position

$$S(x) = \frac{P}{\alpha}\ln\left[\frac{E_0+\alpha x}{E_0}\right] \qquad \alpha = \frac{E_3 - E_0}{H}$$

### Problem 2.5

1) $\frac{d}{dx}\left[AE\frac{du}{dx}\right] = 0 \qquad \sigma(0)=0 \qquad \sigma(L) = \frac{P}{A_L}$

$$2)\int_0^L AE\frac{du}{dx}\frac{du^*}{dx}dx = Pu^*(L)$$

$$3)\frac{E}{\Delta x^2}\begin{bmatrix}\int_0^{2\Delta x}Adx & -\int_{\Delta x}^{2\Delta x}Adx & 0\\ -\int_{\Delta x}^{2\Delta x}Adx & \int_{\Delta x}^{3\Delta x}Adx & -\int_{2\Delta x}^{3\Delta x}Adx\\ 0 & -\int_{2\Delta x}^{3\Delta x}Adx & \int_{2\Delta x}^{3\Delta x}Adx\end{bmatrix}\begin{bmatrix}u_1\\u_2\\u_3\end{bmatrix}=\begin{bmatrix}0\\0\\\frac{P}{A_L}\end{bmatrix}$$

Assuming linear variation of the area

$$\frac{E}{\Delta x}\begin{bmatrix}2A_1 & -\frac{A_{1+}A_2}{2} & 0\\ -\frac{A_{1+}A_2}{2} & 2A_2 & -\frac{A_{2+}A_L}{2}\\ 0 & -\frac{A_{2+}A_L}{2} & \frac{A_{2+}A_L}{2}\end{bmatrix}\begin{bmatrix}u_1\\u_2\\u_3\end{bmatrix}=\begin{bmatrix}0\\0\\\frac{P}{A_L}\end{bmatrix}$$

$$4)\,u(x)=\int_0^L\frac{p}{AE}dx=\frac{p}{E\alpha}\ln\left(\frac{A_0+\alpha x}{A_0}\right)\qquad \alpha=\frac{A_L-A_0}{L}$$

### Problem 3.1

The analytical deflection is given by

$$\therefore v(x)=\frac{1}{EI}\left(\frac{w}{24}x^4-\frac{wL}{6}x^3+\frac{wL^2}{4}x^2\right)$$

The finite element solution of the deflection is

$$\therefore v(x)=v(x)=v_2N_3(x)+\theta_2N_4(x)=-0.00311(\frac{3}{L^2}x^2-\frac{2}{L^3}x^3)-0.00069(-\frac{x^2}{L}+\frac{x^3}{L^2})$$

Where

$$\begin{bmatrix}-24.7\\24.7\end{bmatrix}=EI\begin{bmatrix}\frac{1}{18} & -\frac{1}{6}\\ -\frac{1}{6} & \frac{2}{3}\end{bmatrix}\begin{bmatrix}v_2\\\theta_2\end{bmatrix}$$

### Problem 4.1 (red corresponds to unnecessary steps)

| PREPROCESSOR | PROCESSOR | POST-PROCESSOR |
|---|---|---|
| p) input nodes<br>q) input elements<br>r) input material properties<br>s) input boundary conditions | t) create element matrix equations<br>u) invert element matrix equation<br>v) assembly un-restrained global matrix equation<br>w) invert unrestrained global matrix equation<br>x) apply boundary conditions<br>y) invert global stiffness matrix | z) calculate nodal loads<br>aa) calculate nodal displacement<br>bb) calculate displacement at the domain<br>cc) calculate stress at the domain<br>dd) calculate stress at the nodes |

## Problem 4.2

$$1)\quad \frac{AE_a}{L}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}u_1\\ u_2\end{bmatrix}=\begin{bmatrix}p_1^a\\ p_2^a\end{bmatrix}\qquad \frac{AE_b}{L}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}u_2\\ u_3\end{bmatrix}=\begin{bmatrix}p_2^b\\ p_3^b\end{bmatrix}$$

$$2)\quad \frac{AE_a}{L}\begin{bmatrix}1 & -1 & 0\\ -1 & 1 & 0\\ 0 & 0 & 0\end{bmatrix}\begin{bmatrix}u_1\\ u_2\\ u_3\end{bmatrix}=\begin{bmatrix}p_1^a\\ p_2^a\\ 0\end{bmatrix}\qquad \frac{AE_b}{L}\begin{bmatrix}0 & 0 & 0\\ 0 & 1 & -1\\ 0 & -1 & 1\end{bmatrix}\begin{bmatrix}u_1\\ u_2\\ u_3\end{bmatrix}=\begin{bmatrix}0\\ p_2^b\\ p_3^b\end{bmatrix}$$

$$3)\quad \frac{A}{L}\begin{bmatrix}E_a & -E_a & 0\\ -E_a & E_a+E_b & -E_b\\ 0 & -E_b & E_b\end{bmatrix}\begin{bmatrix}u_1\\ u_2\\ u_3\end{bmatrix}=\begin{bmatrix}p_1^a\\ p_2^a+p_2^b\\ p_3^b\end{bmatrix}$$

$$4)\quad \frac{A}{L}\begin{bmatrix}E_a+E_b & -E_b\\ -E_b & E_b\end{bmatrix}\begin{bmatrix}u_2\\ u_3\end{bmatrix}=\begin{bmatrix}0\\ P\end{bmatrix}$$

$$5)\quad E_B=cE_a \Rightarrow \frac{AE_a}{L}\begin{bmatrix}1+c & -c\\ -c & c\end{bmatrix}\begin{bmatrix}u_2\\ u_3\end{bmatrix}=\begin{bmatrix}0\\ P\end{bmatrix}$$

$$\begin{bmatrix}u_2\\ u_3\end{bmatrix}=\frac{PL}{AE_a}\begin{bmatrix}1+c & -c\\ -c & c\end{bmatrix}^{-1}\begin{bmatrix}0\\ 1\end{bmatrix}=\frac{PL}{AE_a}\begin{bmatrix}1 & 1\\ 1 & \dfrac{c+1}{c}\end{bmatrix}\begin{bmatrix}0\\ 1\end{bmatrix}=\frac{PL}{AE_a}\begin{bmatrix}-1\\ \dfrac{c+1}{c}\end{bmatrix}$$