FINITE ELEMENT METHODS

CHAPTER 1 – INTRODUCTION [1]

Course: Eindige Element Metodes (EEM) 414, Department of Mechanical Engineering, Stellenbosch University

Textbook: Finite Element Modelling for Stress Analysis, 1995, Robert D. Cook, John Wiley & Sons, Inc.

Compiled by: Corné Coetzee

Date: 2006

[1] Finite Element Modelling for Stress Analysis, 1995, Robert D. Cook, John Wiley & Sons, Inc.
Chapter and section numbering follows that of Cook, 1995. Additional information is given using numbers starting with "A".

1.1 The Finite Element Method

The Mathematical Modelling of Physical Systems and specifically the field of Mechanics, can be subdivided into three major areas: *Theoretical, Applied, and Computational.*

Theoretical Mechanics deals with fundamental laws and principles of mechanics studied for their intrinsic scientific value. *Applied Mechanics* transfers this theoretical knowledge to scientific and engineering applications, especially through the construction of mathematical models of physical phenomena. *Computational Mechanics* solves specific problems by simulation through ***numerical methods*** (such as FEM) implemented on digital computers.[2] Figure A.1-1 shows the different approaches of modelling a physical system.

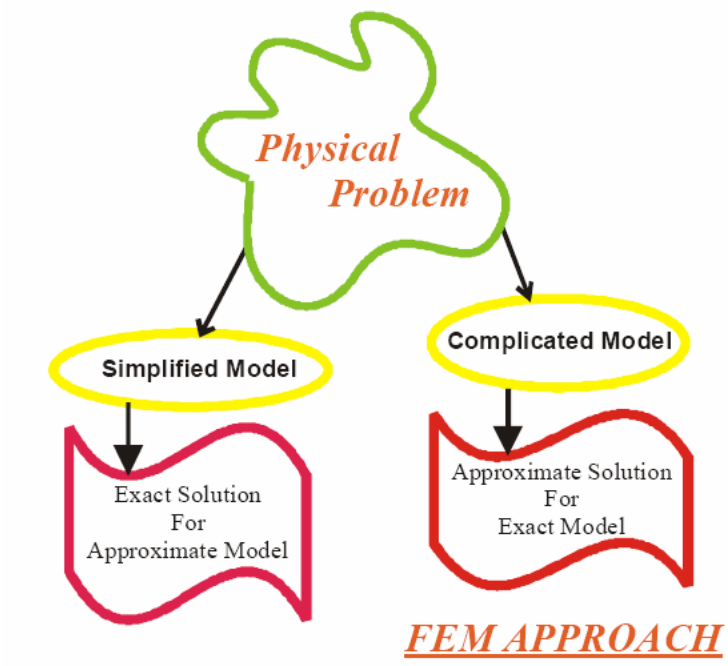

Figure A.1-1 – Modelling approaches.[2]

[2] Finite Element Methods Lecture Notes: 1, Dr Majid Mirzaei, Dept. of Mechanical Eng., TMU

The finite element method is a numerical method used to solve problems in structural mechanics, thermodynamics, fluid flow, and much more. The solution is approximate unless the problem to be solved is so simple that a convenient exact formula is available.

Unsophisticated description: It involves cutting a structure into several elements, describing the behaviour of each element in a simple way, then reconnecting elements at *nodes* (Figure 1.1-1). This process results in a set of equations that needs to be solved simultaneously. The equations can be either linear or non-linear.

A large number of such elements are needed for two reasons: (1) To accurately model the geometry of the structure for example, and (2) to approach the exact solution (called mesh refinement). The large number of elements (equations) makes it impractical to solve by hand and computers are utilised.

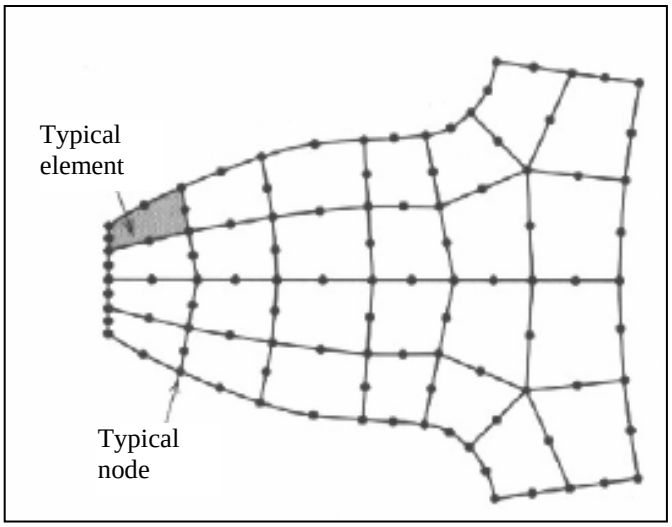

Figure 1.1-1 – A coarse-mesh, two dimensional model of a gear tooth. All nodes and elements lie in the plane of the paper.

Sophisticated description: FEM is a piecewise polynomial interpolation. That is, over an element, a field quantity such as displacement is interpolated from values of the field quantity at the nodes. By connecting elements together, the field quantity becomes interpolated over the entire structure in piecewise fashion. The "best" values of the field quantities at the nodes are those that minimize some function such as *total energy*. The minimization process generates a set of simultaneous algebraic equations for values of the field quantity at the nodes, Figure A.1-2.

This set of equations can be written in matrix form as $\underline{\overline{K}}\,\overline{D} = \overline{R}$, where $\overline{D}$ is a vector of unknown nodal field quantities (such as displacements or temperature) to be calculated, $\overline{R}$ is a vector of known loads and $\underline{\overline{K}}$ is a matrix of known material constants. In stress analysis $\underline{\overline{K}}$ is known as the *stiffness matrix*. The set of equations $\underline{\overline{K}}\,\overline{D} = \overline{R}$, is similar to the single equation of a linear spring $F = k\,x$, with F the force acting on the spring, k the spring stiffness and x the displacement.

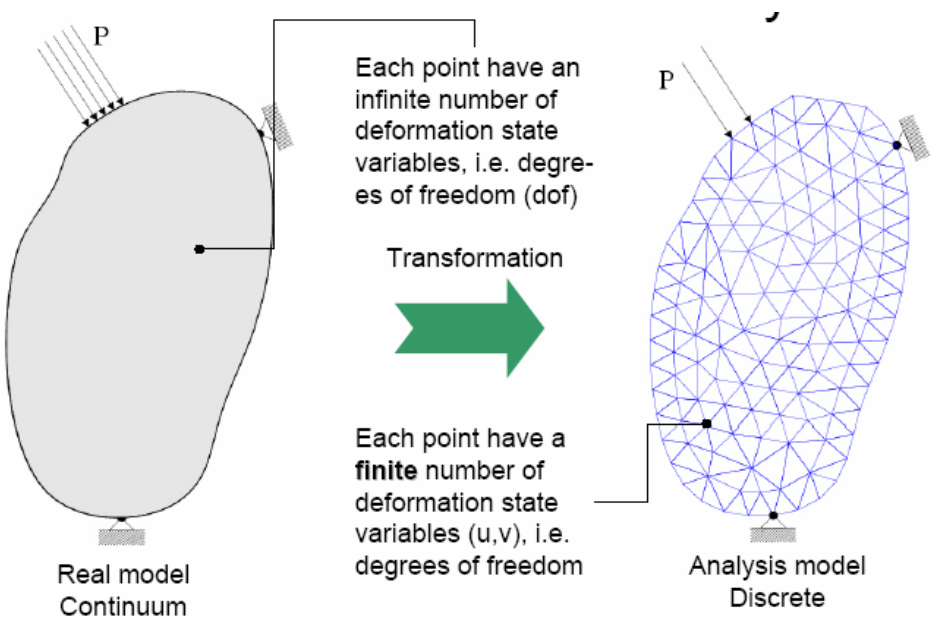

Figure A.1-2 – Continuum vs discrete model.

FEM and the typical user: A typical FEM user asks what kinds of elements should be used, how many elements? Where should the mesh be fine and where may it be coarse? Can the model be simplified (symmetry)? Is the important behaviour static or dynamic? How accurate will the answers be and how can they be checked? One needs to understand the mathematics behind FEM to answer these questions. A competent user must understand how the different elements behave in order to choose the type, size and shape. FEM is based on mathematical theory and several assumptions and limitations exist. The user must be aware of these limitations when creating a FEM model.

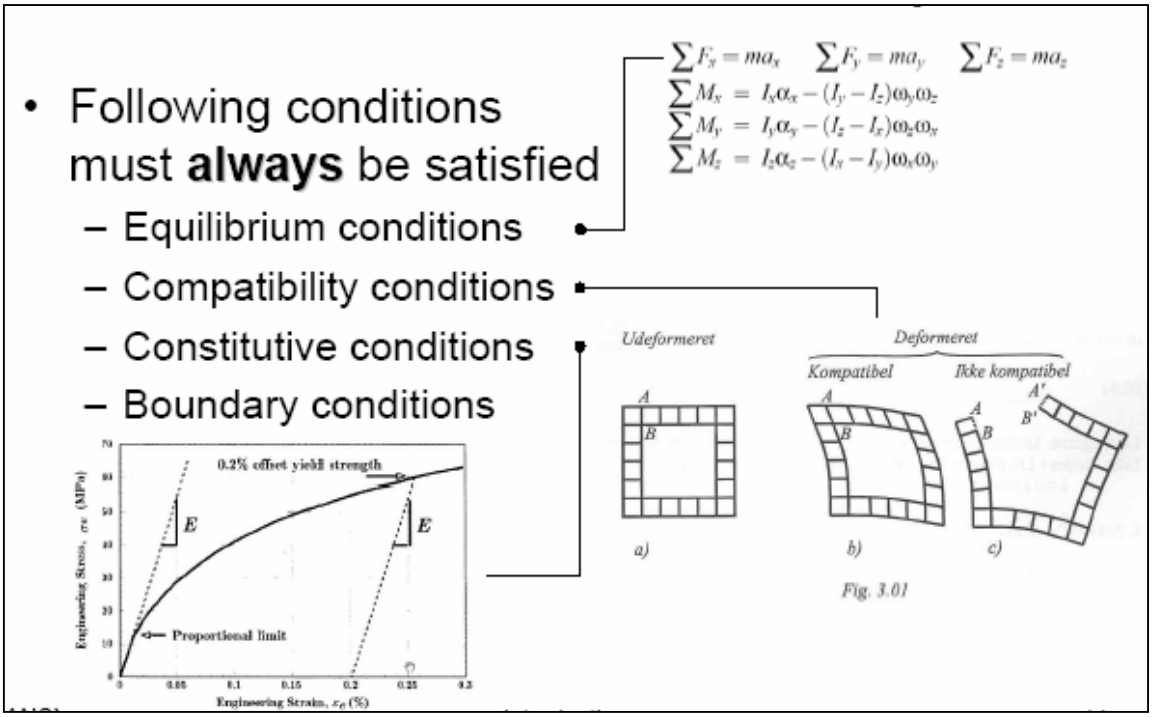

Figure A.1-3 – Conditions that must be satisfied.[3]

Figure A.1-3 shows the four conditions that must always be satisfied in a FEM model[3]. Equilibrium ensures that either Hook's law (Quasi-static analysis) or Newton's law (Dynamic analysis) is satisfied. Compatibility ensures that no gaps can form within the material. The constitutive model is also called the *material model*. Examples of material models are: Linear Elastic, non-linear elastic, and plastic models such as von Mises, Mohr-Coulomb and

[3] Ansys, Computational Mechanics, AAU, Esbjerg.

Drucker-Prager. Boundary conditions are specified at the boundary of the model and can be, for example, fixed constraints or moving boundaries.

An analyst unable to do even a crude pencil-and-paper analysis of the problem probably does not know enough about it to attempt a solution by FEM!

Advantages of FEM

- Irregular Boundaries
- General Loads
- Different Materials
- Boundary Conditions
- Variable Element Size
- Easy Modification
- Dynamics
- Nonlinear Problems (Geometric and/or Material)

Disadvantages of FEM

- An approximate solution
- An element dependent solution (element size and shape)
- Error in input data such as geometry and material properties

The power of FEM is its versatility. The structure analysed may have arbitrary shape, arbitrary supports (constraints) and arbitrary loads. On the other hand, analytical solutions exist only for a couple of simple problems.

1.2 Elements and Nodes

Finite elements resemble fragments of the structure. Nodes appear on element boundaries and serve as connectors that fasten elements together. In Figure 1.2-1, elements are triangular or quadrilateral areas and nodes are indicated by dots.

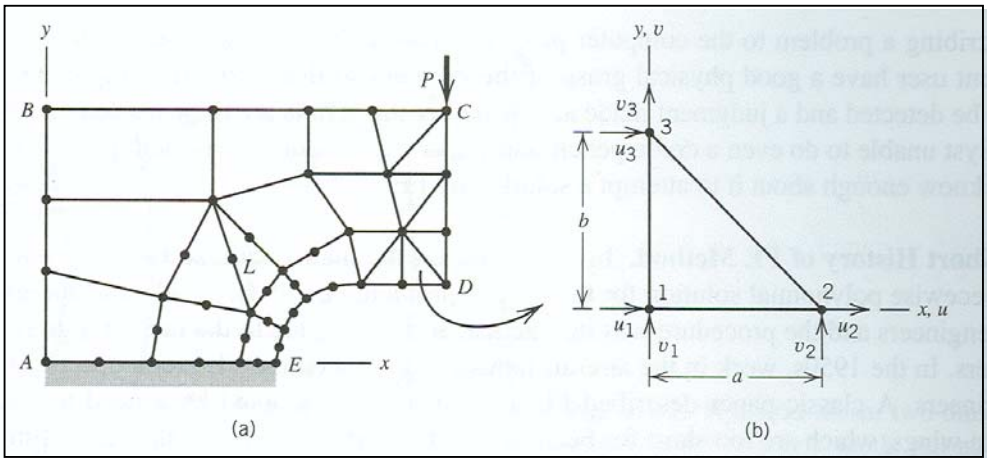

Figure 1.2-1 – (a) A flat bracket modelled by several element types. (b) One of the elements, a "constant strain triangle". All nodes and elements lie in the plane of the paper.

All elements that share a node have the same displacement components at that node. Normally element types are *not* mixed (triangular and quadrilateral), but this is only for demonstration purposes. Each element is restricted in its mode of deformation. This prevents that gaps would from between elements.

Consider the plane triangular element in Figure 1.2-1b. Let u and v be the x and y displacements of *any* point *within* the element or on the element boundaries. Using linear polynomials in x and y:

$$u = \beta_1 + \beta_2 x + \beta_3 y \qquad \text{1.2-1a}$$

$$v = \beta_4 + \beta_5 x + \beta_6 y \qquad \text{1.2-1b}$$

where the β's are called "generalized coordinates", or displacement amplitudes. β_1 and β_4, for example, are rigid body displacements of the element in the *x* and *y* directions respectively. The other β's are amplitudes of linearly varying displacements in the *x* and *y* directions respectively. The β's are however still undefined, but it can be written in terms of nodal displacements.

Why do we need the displacements in terms of the nodal displacements?

The result of the system of equations, $\underline{\overline{K}}\,\overline{D} = \overline{R}$, is $\overline{D}$, the *nodal* displacements. With the nodal displacements known, the displacements can be calculated at any position within the element using equations such as 1.2-1. With the displacements known, the gradients of displacement is used to define the strains, and the strains are used to calculate the stresses.

Let the nodal displacements for node 1 be u_1 and v_1 in the *x*- and *y*-directions respectively; and similarly for the other nodes u_2, v_2, u_3, v_3. Looking at Figure 1.2-1, the following conditions can be used:

$$\begin{aligned} &u = u_1 \quad and \quad v = v_1 \quad at \quad x = 0 \quad and \quad y = 0 \\ &u = u_2 \quad and \quad v = v_2 \quad at \quad x = a \quad and \quad y = 0 \\ &u = u_3 \quad and \quad v = v_3 \quad at \quad x = 0 \quad and \quad y = b \end{aligned} \qquad 1.2\text{-}2$$

Substituting the above conditions into Equation 1.2-1, yields six equations in six unknowns. Solving for the β's yield

$$u = \left(1 - \frac{x}{a} - \frac{y}{b}\right)u_1 + \frac{x}{a}u_2 + \frac{y}{b}u_3 \qquad 1.2\text{-}3a$$

$$v = \left(1 - \frac{x}{a} - \frac{y}{b}\right)v_1 + \frac{x}{a}v_2 + \frac{y}{b}v_3 \qquad 1.2\text{-}3b$$

This can also be written in terms of the *shape functions* N_i.

$$\begin{aligned} u &= N_1 u_1 + N_2 u_2 + N_3 u_3 \\ v &= N_1 v_1 + N_2 v_2 + N_3 v_3 \end{aligned}$$

or in matrix notation

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} = \underline{\overline{N}}\,\overline{d}$$

To calculate the displacement of any point within the element, six quantities should be known; the nodal displacement components – two components at each of the three nodes. This element is said to have six *degrees-of-freedom* (dof). The displacement field $u = u(x)$ and $v = v(x)$ can now be used to calculate the strain components within the element.

What is the definition of strain in a continuum?

Under the assumption of small deformation gradients, the linear strain tensor is given by

$$\varepsilon_{ij} = e_{ij} = \frac{1}{2}\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] \qquad \text{where } u_i \text{ is the displacement vector.} \qquad \text{A.2-1}$$

$$\varepsilon_{ij} = \begin{bmatrix} \frac{\partial u_x}{\partial x_x} & \frac{1}{2}\left[\frac{\partial u_x}{\partial x_y} + \frac{\partial u_y}{\partial x_x}\right] & \frac{1}{2}\left[\frac{\partial u_x}{\partial x_z} + \frac{\partial u_z}{\partial x_x}\right] \\ \frac{1}{2}\left[\frac{\partial u_y}{\partial x_x} + \frac{\partial u_x}{\partial x_y}\right] & \frac{\partial u_y}{\partial x_y} & \frac{1}{2}\left[\frac{\partial u_y}{\partial x_z} + \frac{\partial u_z}{\partial x_y}\right] \\ \frac{1}{2}\left[\frac{\partial u_z}{\partial x_x} + \frac{\partial u_x}{\partial x_z}\right] & \frac{1}{2}\left[\frac{\partial u_z}{\partial x_y} + \frac{\partial u_y}{\partial x_z}\right] & \frac{\partial u_z}{\partial x_z} \end{bmatrix} \qquad \text{A.2-2}$$

Assuming two-dimensional conditions, the strain tensor can be written as

$$\varepsilon_{ij} = \begin{bmatrix} \frac{\partial u_x}{\partial x_x} & \frac{1}{2}\left[\frac{\partial u_x}{\partial x_y} + \frac{\partial u_y}{\partial x_x}\right] \\ \frac{1}{2}\left[\frac{\partial u_y}{\partial x_x} + \frac{\partial u_x}{\partial x_y}\right] & \frac{\partial u_y}{\partial x_y} \end{bmatrix} \qquad \text{A.2-3}$$

In Cook's notation we have $u_x = u$, $u_y = v$, $x_x = x$ and $x_y = y$ with the strain tensor given by

$$\varepsilon_{ij} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{1}{2}\left[\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right] \\ \frac{1}{2}\left[\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right] & \frac{\partial v}{\partial y} \end{bmatrix} \qquad \text{A.2-4}$$

Comparing this tensor with the definition of *Engineering Strain*,

$$\varepsilon_{ij} = \begin{bmatrix} \varepsilon_x & \frac{\gamma_{xy}}{2} \\ \frac{\gamma_{yx}}{2} & \varepsilon_y \end{bmatrix} \qquad \text{A.2-5}$$

See Appendix A – The Strain Tensor, for more details and the physical meaning of each component.

With the displacement field given by Equation 1.2-1 and 1.2-3, the strain can be calculated within the element,

$$\varepsilon_x = \frac{\partial u}{\partial x} = \beta_2 = \frac{u_2 - u_1}{a}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} = \beta_6 = \frac{v_3 - v_1}{b} \qquad \text{1.2-4}$$

$$\gamma_{xy} = \gamma_{yx} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \beta_3 + \beta_5 = \frac{u_3 - u_1}{b} + \frac{v_2 - v_1}{a}$$

From these results it can be seen that all the strain components are constant within the element, i.e. they are not dependent on x or y. This type of element is therefore called a *constant strain triangle*.

Let's analyse the deformation of the element and specifically the deformation of the sides. Along side 1-3, the x-coordinate is zero. Setting $x = 0$ in Equation 1.2-3a and b,

$$u = \left(1 - \frac{y}{b}\right)u_1 + \frac{y}{b}u_3 \qquad \text{A.2-6a}$$

$$v = \left(1 - \frac{y}{b}\right)v_1 + \frac{y}{b}v_3 \qquad \text{A.2-6b}$$

From this it is clear that u and v is only a function in y. Furthermore, the displacement components depend only on the nodal displacements of node 1 and node 3, the two nodes connecting side 1-3. This indicates that side 1-3 will remain straight, and the same can be demonstrated for the other sides of this element. Further, the same can be shown for neighbouring elements. This shows that, for two neighbouring elements, the sides will remain straight and no gaps can form between the elements – fulfils the *compatibility requirements*.

The same can be shown for all other types and shapes of elements – although the algebra can be more complicated and tedious☺.

Element *L* in Figure 1.2-1 is a six noded triangular element, i.e. each of the sides have 3 nodes available. With three nodes per side, it means that the deformation of the side can be quadratic (second order). Further, each of the six nodes will have two degrees-of-freedom (translation in the *x*- and *y*-direction respectively), and thus the element will have a total of 12 degrees-of-freedom. The displacement field can be written in terms of these 12 degrees-of-freedom,

$$u = \beta_1 + \beta_2 x + \beta_3 y + \beta_4 x^2 + \beta_5 xy + \beta_6 y^2 \qquad \text{1.2-5a}$$

$$v = \beta_7 + \beta_7 x + \beta_9 y + \beta_{10} x^2 + \beta_{11} xy + \beta_{12} y^2 \qquad \text{1.2-5b}$$

Making use of Equation 1.2-4, it is clear that the strain components will have both constant and linear terms. This element is more competent and complex compared to the *constant strain* element. Another question arises: Is it better to use many simple elements or a few complex elements? The answer to this question is not simple and will be dealt with later in the course.

1.3 Modelling the Problem and Checking Results

1. Modelling requires that the physical action of the problem be understood well enough to choose suitable kinds of elements, and enough of them, to represent the physical action adequately.
2. Elements too large would not give great detail variations in the field functions.
3. An *over-refined* mesh will take long to solve, consuming computer resources.
4. Usually you will have a finer mesh where the field gradients are higher, for example at stress concentrations – the user should identify such cases when setting up the model.
5. Computer results **must** be checked to see if they are reasonable.
6. Un-averaged stresses are usually discontinuous over element boundaries – the finer the mesh, the less the discontinuities.

1.4 Discretisation and Other Approximations

1. There will also be modelling errors as the physical problem is analyzed using a simplified mathematical model.
2. The type of elements may influence the solution and the accuracy thereof.
3. The physical model has an infinite number of degrees-of-freedom (every point within the continuum body can displace), and the FEM model has a finite (limited) number of degrees-of-freedom.
4. Good practise is to demonstrate convergence of the results with mesh refinement. For example, apply a coarse mesh and obtain the results, then refine the mesh somewhat and compare the results with that of the coarse mesh. There will be a difference in results, depending on how sensitive the problem is to mesh size. Continue to decrease the mesh size until a further reduction has little to no influence on the results.

1.5 Responsibility of the User

1. It is always possible to produce an answer.
2. Some results may appear reasonable on casual inspection.
3. With a poor model, mesh refinement may not have a positive influence on the results.
4. A responsible user *must* understand the physical nature of the problem and the behaviour of FEM well enough to prepare a suitable model and evaluate the quality of the results.

See the publication: Kurowski, P., 2001, *Easily made errors mar FEA results*, Machine Design, September 13, 2001, www.machinedesign.com

1.6 Elementary Matrix Algebra

Finite Element theory can be very simplified when matrix algebra is used. Loads are assembled *load vectors* and the stiffness of elements in a *stiffness matrix*. Some elementary matrix algebra is needed for the manupilation of equations and obtaining solutions.

In FEM, the following equation is solved for the nodal displacements $\overline{D}$

$$\underline{\overline{K}}\,\overline{D} = \overline{R} \qquad \text{A.6-1}$$

where $\underline{\overline{K}}$ is the stiffness matrix and $\overline{R}$ is the load vector. The nodal displacement $\overline{D}$ can be solved by inverting the matric $\underline{\overline{K}}$

$$\overline{D} = \underline{\overline{K}}^{-1}\overline{R} \qquad \text{A.6-2}$$

Since the stiffness matrix is usually large and sparse, it is not efficient to store the whole matrix or to calculate its invert. Other direct methods such as Gauss elimination or indirect methods can be used more efficiently.

A square matrix is singular if it determinant is zero, and the inverse not defined. In such a case a solution to Equation A.6-1 is not possible, even with direct or indirect methods. Examples of the determinant is given by,

$$\det\begin{bmatrix} a & b \\ c & d \end{bmatrix} = ad - bc \qquad \text{A.6-3}$$

$$\det\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} =$$

$$(-1)^{1+1} a_{11}(a_{22}a_{33} - a_{32}a_{23}) + (-1)^{1+2} a_{12}(a_{21}a_{33} - a_{31}a_{23}) + (-1)^{1+3} a_{13}(a_{21}a_{32} - a_{31}a_{22}) \qquad \text{A.6-4}$$

The inverse of matrix $\underline{\overline{A}}$ is constructed using the *cofactor matrix* $\underline{\overline{C}}$,

$$\underline{\overline{A}}^{-1} = \frac{1}{\det \underline{\overline{A}}} \underline{\overline{C}}^{T} \qquad \text{A.6-5}$$

where

$$C_{ij} = (-1)^{i+j} M_{ij} \qquad \text{A.6-6}$$

where M_{ij} is the determinant of the smaller matrix obtained by eliminating the ith row and j^{th} column of $\underline{\overline{A}}$. Examples,

$$\begin{bmatrix} a & b \\ c & d \end{bmatrix}^{-1} = \frac{1}{ad - bc} \begin{bmatrix} d & -b \\ -c & a \end{bmatrix} \qquad \text{A.6-7}$$

$$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 2 \end{bmatrix} = \frac{1}{4-1-2} \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix}^{T} = \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix} \qquad \text{A.6-8}$$