cupress

Finite Element Method in Engineering

with MATLAB, Mathematica & FEATool Multiphysics

Pramote Dechaumphai

Finite Element Method in Engineering

Finite Element Method in Engineering

Pramote Dechaumphai

Chulalongkorn University press

2024

375.–

Dechaumphai, Pramote
Finite Element Method in Engineering / Pramote Dechaumphai
1. Finite element method. 2. Finite element method -- Data processing.

620.00151825
ISBN (e-book) 978-974-03-4321-9
CUP. 2692

Knowledge to All
www.cupress.chula.ac.th

Published by **Chulalongkorn University Press**
First edition 2024
www.cupress.chula.ac.th [CUB6701-008K]
Tel. 0-2218-3562-3

Managing editor : Prof. Dr.Aran Hansuebsai Mrs.Orathai Nanthanadisai

Academic Editorial Department : Emeritus Professor Dr.Piyanart Bunnag
Prof.Chitsanu Pancharoen
Assoc. Prof. Dr.Pimpan Dachakupt
Assoc. Prof. Dr.Vimolvan Pimpan

Coordinator : Wasana Sumsen

Proof reader : Tassanee Phewkham

Artwork and Cover : Kaiumporn Phongkhachorn

Contact : **Chulalongkorn University Book Center**
Phyathai Road, Pathumwan District, Bangkok 10330, Thailand
http://www.chulabook.com
Tel: 08-6323-3703-4
customer@cubook.chula.ac.th, info@cubook.chula.ac.th
Apps: CU-eBook Store

Preface

The inception of the book, *Finite Element Method in Engineering*, dates back three decades, originating from the author's lecture notes during teaching at Old Dominion University and George Washington University. Drawing extensively from his tenure as an aerospace engineer at NASA Langley Research Center, the book has undergone multiple revisions to keep pace with the evolving landscape of the finite element method. Over the years, it has achieved recognition through widespread publication and distribution globally.

This textbook is structured into two primary sections, addressing both fundamentals and applications. The foundational section meticulously presents crucial formulations of the finite element method. The derivation of finite element equations is elucidated through various approaches, including the direct approach, variational approach, and the method of weighted residuals. Diverse element types are explored in detail, accompanied by numerous examples to facilitate a comprehensive understanding. To further enhance readers' readiness, the book incorporates the use of software packages like MATLAB, Mathematica, and FEATool Multiphysics, ensuring a solid foundation before engaging with commercial finite element software.

In the application section, details of the finite element method for analyzing structural, heat transfer and fluid flow problems are explained. For the structural problem, both the static and dynamic formulations for the truss, beam, plate, plane stress/strain, axisymmetric and 3D solid elements are presented. Formulations for linear/nonlinear, steady-state/transient conduction, convection and radiation heat transfers are presented with examples. For the fluid flow problems, the formulations of the potential flow, inviscid/viscous and incompressible/compressible flows are explained with academic and practical examples.

The author extends heartfelt gratitude to his esteemed former Professor, Dr. Earl A. Thornton, and his supervisor, Dr. Allan R. Wieting, from the Aerothermal Loads Branch at NASA Langley

Research Center. Appreciation is also extended to the students at NASA Langley Research Center, Old Dominion University, and Chulalongkorn University who actively participated in the finite element method courses he offered.

Special thanks are reserved for Chulalongkorn University Press for their role in publishing the book, contributing to its dissemination and impact. Lastly, the author expresses deep appreciation to his wife, Mrs. Yupa Dechaumphai, for her understanding and unwavering support throughout the writing process, acknowledging the significant role she played in bringing the book to fruition.

Pramote Dechaumphai

Contents

Part I

Fundamentals of Finite Element Method

Chapter 1

Overview of Finite Element Method

1.1 Introduction

Finding solutions to scientific and engineering problems is a crucial aspect of improving our daily lives. The laws of physics, in the form of differential and integral equations, can explain most of the natural phenomena we encounter. For example, the calculation of the temperature distribution in a car engine requires solving differential equations that take into account the conservation of energy in heat transfer. Similarly, determining the stresses in airplane wings under varying pressure during flight requires solving the differential equations that characterize the structure's equilibrium. Moreover, differential equations governing fluid flow conservation can be employed to predict the wind direction and speed in a typhoon.

Deriving differential equations that explain physical phenomena is often a straightforward process. However, obtaining exact solutions to these equations can be challenging, and quite often even impossible. As a result, numerous methods have been developed to find approximate solutions. One of the most widely used methods over the past few decades has been the finite difference method.

The finite difference method involves transforming differential equations into a set of algebraic equations that employ basic arithmetic operations such as addition, subtraction, multiplication, and division. It is highly regarded for its simplicity and ease of understanding. The solutions it provides can be conveniently obtained by developing computer programs for performing numerical calculations. However, one of the disadvantages of the finite difference method is the difficulty of implementing arbitrary boundary conditions, as well as modeling complex geometries.

In Figure 1.1(a), the geometry of an aluminum plate that supports a part within an airplane wing frame is depicted. The plate features straight and curved edges, as well as two circular holes. By utilizing the finite difference method, it is possible to analyze the stress distribution of the aluminum plate when subjected to an applied load. Firstly, the plate is discretized into smaller square grids, as displayed in Figure 1.1(b). These grids are joined together at their corner points, which act as the grid points. The number of these grid points determines the size of the problem, or the number of unknowns involved.

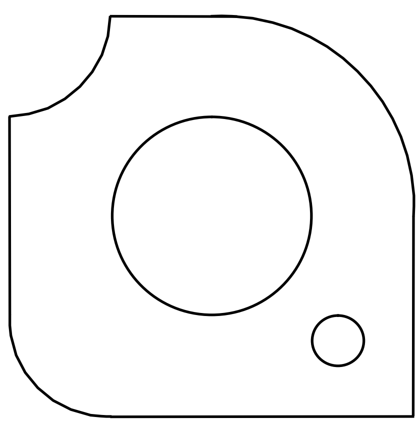


Figure 1.1(a) An aluminum plate with arbitrary geometry.

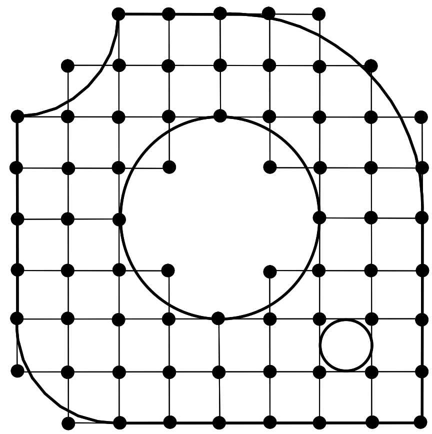


Figure 1.1(b) A finite difference model for the aluminum plate.

As shown in Figure 1.1(b), it is evident that the grids used in the finite difference model fail to accurately depict the original configuration. To enhance the representation of the plate geometry, smaller squares must be employed. Nonetheless, the use of smaller squares results in an increase in the number of grid points, and subsequently, a large number of finite difference equations. With a larger number of equations, more computational time and computer memory are required during the solving process.

The shortcomings and complexities of the finite difference method necessitated the development of another approximate solution technique known as the finite element method (FEM). FEM is particularly well-suited for handling problems with intricate geometry since it can provide a more accurate representation of arbitrary shapes. Like the finite difference method, FEM also employs the discretization of the original geometry into small pieces or elements. In addition, FEM allows for a wider range of element shapes, including triangles, quadrilaterals, and their combinations, as demonstrated in Figure 1.1(c).

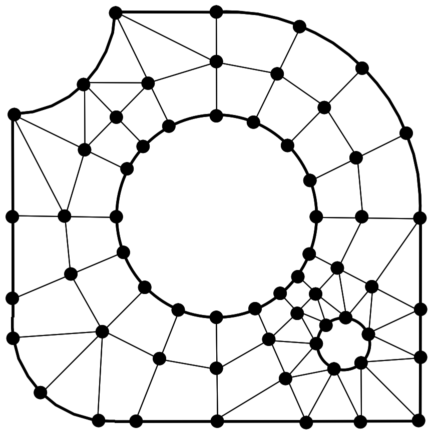


Figure 1.1(c) A finite element model for the aluminum plate.

Comparison of Figs 1.1(b) and 1.1(c) clearly highlights the advantages of the finite element method over the finite difference method. Figure 1.1(c) showcases the finite element method's ability to more closely approximate the original geometry of the aluminum piece. Such enhanced fidelity to the original geometry translates to greater accuracy in the computed solution.

1.2 What is the Finite Element Method ?

Most of engineering problems are normally governed by the differential equations and boundary conditions. The derived exact solution is always valid throughout the domain and provides solutions at an infinite number of locations. As mentioned earlier, exact solutions are not easy to derive, especially for problems with complex geometry. Approximate or numerical solutions that can roughly satisfy the same set of the differential equations and boundary conditions are alternatively considered. Instead of providing the infinite number of solutions as in the exact solutions, the finite element concept is to determine solutions only at some finite locations. This is done by first discretizing the geometry of the model into a number of finite elements as previously shown in Fig. 1.1(c). These elements are connected at grid points or nodes at which the unknowns are to be determined.

The key idea of the finite element method is to transform the differential equations into a set of algebraic equations for each element. The finite element equations from all elements are then assembled together to form a large set of simultaneous equations. The boundary conditions of the problem are applied prior to solving for the unknowns at all nodes.

From this brief explanation, the accuracy of the approximate solution thus depends on the size and number of elements in the analysis. The accuracy of the solution also depends on the element interpolation functions that will be explained in details later. Element interpolation functions could be in linear, quadratic, or cubic forms, etc., depending on the type of the selected elements. For example, if the temperatures at the three corner nodes of a triangular element with linear interpolation functions are 30, 40 and 50°C respectively, the temperature inside this element will distribute as a flat plane with values ranging from 30 to 50°C.

The explanation above is a very broad view of how the finite element method works. Details of this method with examples will be explained, step by step, starting from Chapter 2. Because the finite difference method gives approximate solutions similar to the finite element method and is easy to understand, the method will be first studied to help understanding the finite element method more conveniently.

1.3 Similarities between Finite Difference Method and Finite Element Method

There are several engineering problems that the governing differential equations are in the form of the Laplace's equation. Such problems include the equilibrium equations in solid, the conservation of energy in heat conduction, and the conservation of mass in fluid flow. The two-dimensional Laplace's equation on *x*-*y* plane can be written in the form of the differential equation as $\nabla^2\phi = 0$; where $\nabla^2 = \partial^2/\partial x^2 + \partial^2/\partial y^2$ denotes the differential operator and $\phi = \phi(x,y)$ is the unknown variable in the domain of Ω. For example, ϕ may represent the temperature distribution at

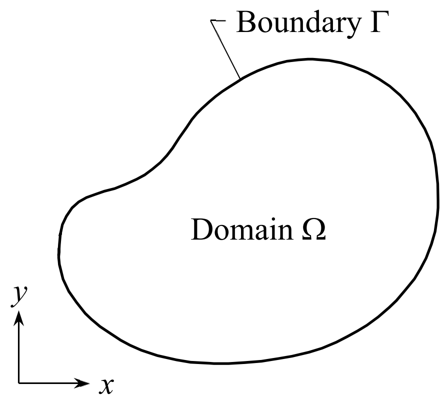


Figure 1.2 Domain and boundary of a two-dimensional geometry.

any x, y location in a thin plate. The $\phi = f(x, y)$ is a known function specified along the outer boundary Γ. For example, ϕ along the boundary can be the temperature of 30°C.

The following steps explain the procedure to determine the approximate solution of $\phi(x, y)$ within the domain Ω by using the finite difference method. There are four basic steps in the procedure of determining approximate solution as follows.

Step 1 *Create a rectangular mesh to represent the entire domain*. Small rectangles are generated for the entire rectangular domain that lies in x-y plane as shown in Fig. 1.3. These small rectangles have side lengths of Δx and Δy in the x- and y-direction respectively. The rectangles are connected at grid points. The location i, j denotes the grid point at $x = i$ and $y = j$. These grid points are the locations that the unknowns of the problem will be determined. For example, in heat transfer problem, the unknowns at the grid points are the temperatures.

Step 2 *Transform the differential equation into the equations that contain unknown variables at grid points*. In this case, the Laplace's equation above is used as an example of the differential equation.

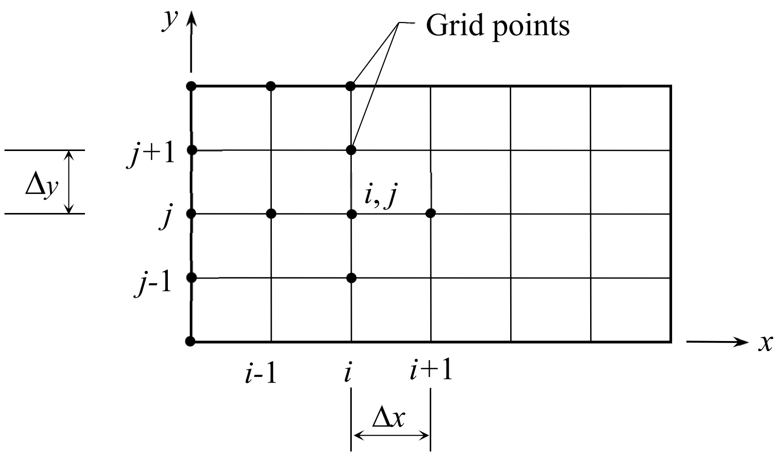


Figure 1.3 A finite difference model of a rectangular region.

$$\frac{\partial^2\phi}{\partial x^2}+\frac{\partial^2\phi}{\partial y^2} = 0 \tag{1.1}$$

The unknown ϕ in this differential equation can be expressed by employing the Taylor series expansion. For instance, the temperature at grid point i+1 can be written in terms of the temperature and its derivatives at the grid point i as

$$\phi_{i+1} = \phi_i+\left.\frac{\partial\phi}{\partial x}\right|_i \Delta x+\frac{1}{2!}\left.\frac{\partial^2\phi}{\partial x^2}\right|_i (\Delta x)^2+\frac{1}{3!}\left.\frac{\partial^3\phi}{\partial x^3}\right|_i (\Delta x)^3+\frac{1}{4!}\left.\frac{\partial^4\phi}{\partial x^4}\right|_i (\Delta x)^4+\dots\dots\dots \tag{1.2}$$

Similarly, the temperature at grid point $i-1$ can also be written in terms of the temperature and its derivatives at the grid point i as

$$\phi_{i-1} = \phi_i-\left.\frac{\partial\phi}{\partial x}\right|_i \Delta x+\frac{1}{2!}\left.\frac{\partial^2\phi}{\partial x^2}\right|_i (\Delta x)^2-\frac{1}{3!}\left.\frac{\partial^3\phi}{\partial x^3}\right|_i (\Delta x)^3+\frac{1}{4!}\left.\frac{\partial^4\phi}{\partial x^4}\right|_i (\Delta x)^4-\dots\dots\dots \tag{1.3}$$

By combining Eqs. (1.2) and (1.3) together,

$$\phi_{i+1} + \phi_{i-1} \;=\; 2\phi_i + \frac{2}{2!}\frac{\partial^2 \phi}{\partial x^2}\bigg|_i (\Delta x)^2 + \frac{2}{4!}\frac{\partial^4 \phi}{\partial x^4}\bigg|_i (\Delta x)^4 + \ldots\ldots. \tag{1.4}$$

Because the second-order term, $\partial^2\phi/\partial x^2$ is needed for the Laplace's equation, the higher-order terms of Eq. (1.4) is neglected. Thus the second-order term in the x-direction is approximated as

$$\frac{\partial^2 \phi}{\partial x^2} \;\cong\; \frac{\phi_{i+1} - 2\phi_i + \phi_{i-1}}{(\Delta x)^2} \tag{1.5}$$

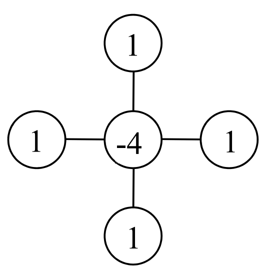
Similarly, the second-order term in the y-direction is approximated as

$$\frac{\partial^2 \phi}{\partial y^2} \;\cong\; \frac{\phi_{j+1} - 2\phi_j + \phi_{j-1}}{(\Delta y)^2} \tag{1.6}$$

By substituting the approximated second-order terms from Eqs. (1.5) and (1.6) into the Laplace's equation (1.1) and if Δx is equal to Δy,

$$\phi_{i+1} + \phi_{i-1} + \phi_{j+1} + \phi_{j-1} - 4\phi_i \;=\; 0 \tag{1.7}$$

The above Eq. (1.7) can be written in a stencil form as



$= \; 0$ (1.8)

To solve a given problem for an approximate solution, the stencil form (1.8) is applied to every grid point within the domain of the problem as shown in Fig. 1.4.

Step 3 *Apply the stencil form (1.8) to all the grid points within the domain of the problem.* Such application leads to a set of algebraic simultaneous equations, consisting of the unknowns ϕ_i at grid points, which are solved for solutions of the problem.

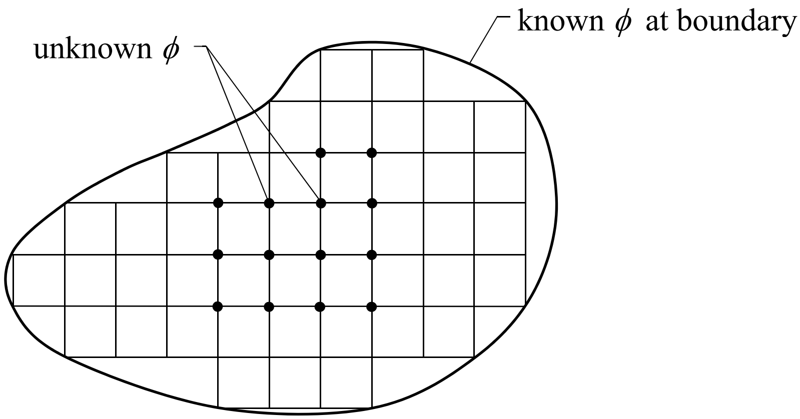


Figure 1.4 Application of the stencil form of Eq. (1.8) to all the grid points inside the domain.

__Step 4__ *Solve the set of the algebraic simultaneous equations for the solutions of the unknowns at grid points.*

To get a better understanding of how to apply the finite difference method through these four steps above, a heat transfer problem is examined in which temperatures at different locations within the plate are unknowns. From this example, some characteristics of the equations evolved from the finite difference method are observed. Such characteristics are useful and help us to understand the finite element method when studied in details later.

Example 1.1 A metal plate with 40 cm long and 20 cm wide, as shown in Fig. 1.5, is under steady-state conduction heat transfer. The temperature distribution can be described by the Laplace's equation as,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = 0 \tag{1.9}$$

If the temperature along the four edges of the plate is at 30°C, determine the temperature inside the plate by the finite difference method.

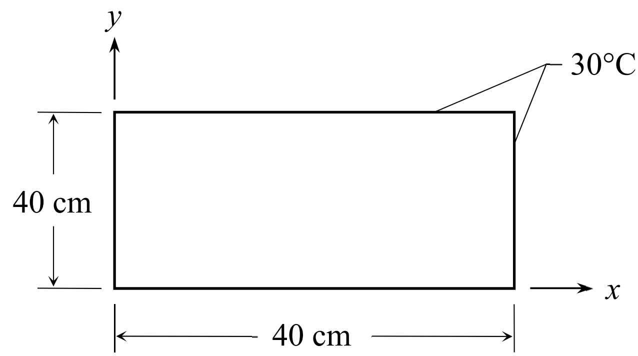


Figure 1.5 Determination of temperature distribution in a plate under steady-state heat conduction by the finite difference method.

Solution Procedure: Following the four-step procedure for the finite difference method that was explained earlier.

<u>Step 1</u> A rectangular mesh is created to represent the entire domain. For this example, we will use the square mesh as shown in Fig. 1.6, where each square is of area 10×10 cm^2 ($\Delta x = \Delta y = 10$ cm).

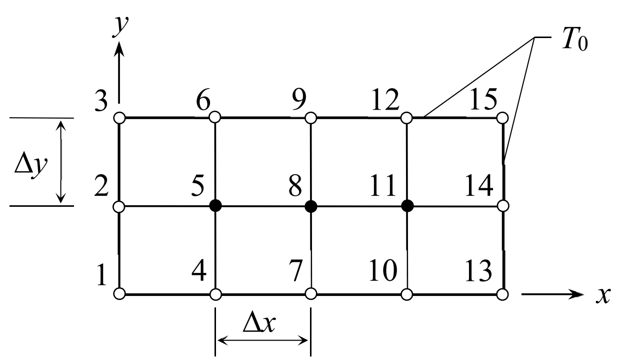
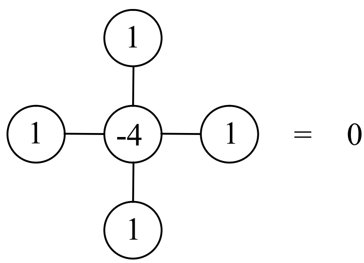


Figure 1.6 The plate that was discretized into squares by the finite difference method.

These squares are connected at the grid points where the temperatures will be determined. With this mesh, there are 15 grid points. Since the temperatures along the four edges are specified as 30°C, the temperatures at the grid point number 5, 8, and 11 are to be determined. These grid point temperatures are denoted by T_5, T_8 and T_{11}.

<u>Step 2</u> Apply the stencil form of Eq. (1.8) at the grid points with unknowns.



(1.8)

For example, if the stencil form is applied at grid point number 5, the resulting equation is,

$$T_2 + T_6 + T_8 + T_4 - 4T_5 = 0$$

Because $T_2 = T_6 = T_4$ is equal to T_0, the above equation is reduced to,

$$4T_5 - T_8 = 3T_0 \tag{1.10}$$

Similarly, for grid point number 8,

$$T_5 + T_9 + T_{11} + T_7 - 4T_8 = 0$$

Because $T_9 = T_7 = T_0$, the above equation is reduced to,

$$-T_5 + 4T_8 - T_{11} = 2T_0 \tag{1.11}$$

Finally, by applying the stencil form to grid point number 11 again,

$$T_8 + T_{12} + T_{14} + T_{10} - 4T_{11} = 0$$

Because $T_{12} = T_{14} = T_{10} = T_0$, the above equation is reduced to,

$$-T_8 + 4T_{11} = 3T_0 \tag{1.12}$$

Step 3 By writing Eqs. (1.10), (1.11), and (1.12) together,

$$\begin{aligned} 4T_5 - T_8 \qquad &= 3T_0 \\ -T_5 + 4T_8 - T_{11} &= 2T_0 \\ - T_8 + 4T_{11} &= 3T_0 \end{aligned}$$

which can be rewritten in matrix form as,

$$\begin{bmatrix} 4 & -1 & 0 \\ -1 & 4 & -1 \\ 0 & -1 & 4 \end{bmatrix} \begin{Bmatrix} T_5 \\ T_8 \\ T_{11} \end{Bmatrix} = \begin{Bmatrix} 3T_0 \\ 2T_0 \\ 3T_0 \end{Bmatrix} \tag{1.13}$$

Or, in a more general form of,

$$\underset{(3\times 3)}{[K]}\ \underset{(3\times 1)}{\{T\}} = \underset{(3\times 1)}{\{Q\}} \tag{1.14}$$

Because this example involves conduction heat transfer, $[K]$ is called the conduction matrix, $\{T\}$ the vector of nodal unknowns, and $\{Q\}$ the load vector.

Step 4 The set of simultaneous algebraic equations, Eqs. (1.13), is then solved for the temperatures at grid points 5, 8, and 11, yielding

$$T_5 = T_8 = T_{11} = T_0 = 30°\text{C}$$

That is, the temperatures inside the plate are 30°C, which are equal to the specified temperature along the edges. The temperature distribution of the plate in the steady-state condition is shown in Fig. 1.7.

From Fig. 1.7, the temperature which is obtained from the finite difference method is the exact solution of the problem. If Eq. (1.5) is re-examined, it may be wonder why the exact solution is obtained after eliminating the higher-order terms in the approximation of the equation that represents the Laplace's equation. This is because those high-order terms are zero for this particular example

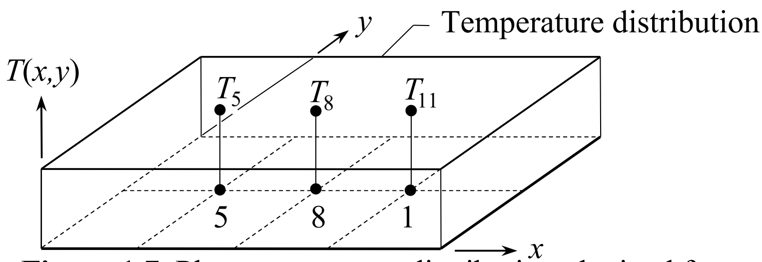


Figure 1.7 Plate temperature distribution obtained from using the finite difference method.

due to the uniform specified temperatures along the edges. If arbitrary temperatures are specified along the edges as shown in Fig. 1.8, the temperatures at grid point numbers 5, 8, and 11 are not exact. The magnitude of the error depends on the size of the squares and the total number of the grid points. Nevertheless, all four steps of the finite difference method can still be applied to solve for solution in the same manner.

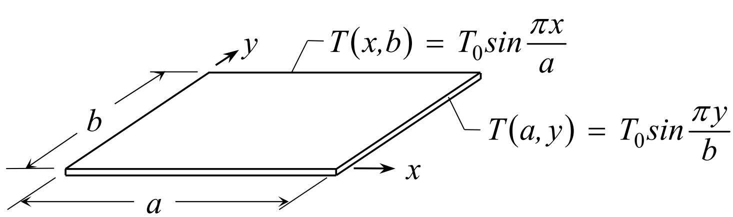


Figure 1.8 The plate with arbitrary temperature distributions along the edges.

An interesting point from this example is the properties of the conduction matrix $[K]$ as shown in Eqs. (1.13)-(1.14). The matrix $[K]$ has the distribution of non-zero terms along and around its diagonal line as highlighted in Fig. 1.9.

The properties of the matrix $[K]$ are as follows

(a) Diagonal terms are dominated

(b) Diagonal terms are positive

(c) It is symmetric, i.e., $[K] = [K]^T$

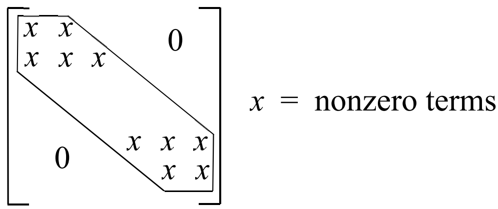


Figure 1.9 The conduction matrix with non-zero terms along and near its diagonal line.

(d) The non-zero terms are clustered and forming a band along the diagonal line as shown in the diagonal frame of Fig. 1.9.

It should be noted that the properties (c) and (d) provide an advantage in computer programming, especially for practical problems that normally contain a large number of unknowns. If the matrix $[K]$ is symmetric and banded, fewer numbers of coefficients of the matrix $[K]$ are needed to be stored in the computer memory, resulting in less computational time and effort.

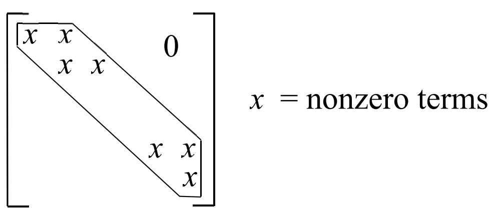


Figure 1.10 Fewer coefficients in matrix $[K]$ needed to be stored in computer memory resulting in computational efficiency.

1.4 The Differences between Finite Difference Method and Finite Element Method

From the example of the finite difference method in the preceding section, it can be seen that the **finite difference method** is a numerical technique that finds an approximate solution of a given problem. The concept of this method is replacing the derivatives that appear in the differential equation by an algebraic approximation. The unknowns of the approximate algebraic equations are the dependent variables at the grid points.

The **finite element method** is also a numerical technique that is used to find an approximate solution of a given problem. The geometry is discretized into small segments called elements. These elements are connected at the nodes where the unknowns of the problem are to be determined.

If the finite element method is used to analyze the heat transfer in the plate as previously shown in Fig. 1.5, the plate first discretized into small elements. The elements may be in quadrilateral or triangular shape as shown in Fig. 1.11.

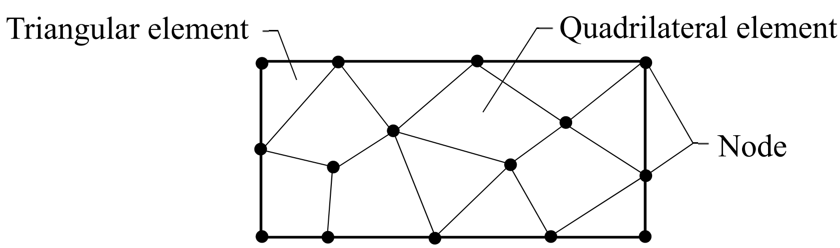


Figure 1.11 Discretization of the plate into quadrilateral and triangular element shapes.

1.5 Finite Element Procedures

Analyzing procedure of a given problem by the finite element method normally consists of six main steps as follows.

Step 1 *Discretization of the computational domain into a number of elements*. The domain could represent the elasticity, heat transfer, or fluid flow problems, etc., as shown in Fig. 1.12.

Step 2 *Selecting element types and their interpolation functions*. For example, a typical triangular element in Fig. 1.12 contains three nodes. This typical element is shown again in Fig. 1.13 with the node numbers 1, 2 and 3. At these three nodes, the nodal unknowns are denoted by ϕ_1, ϕ_2 and ϕ_3 respectively. The nodal unknowns can be the nodal displacements in an elasticity problem, the nodal temperatures in a heat transfer problem, or the nodal flow velocities in a fluid problem. The distribution of unknowns over the element can be written in the form of the element interpolation functions and the nodal unknowns as follows

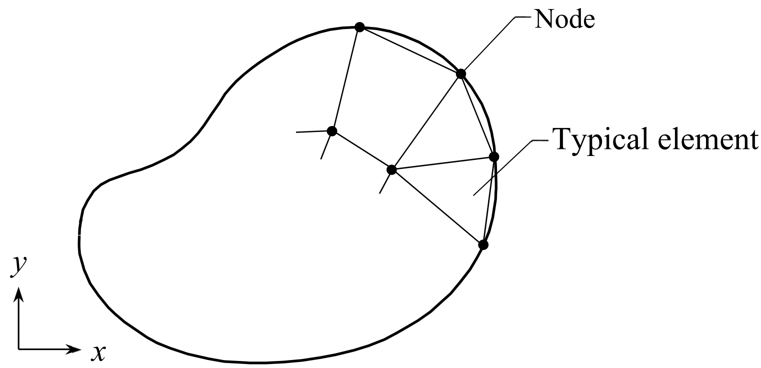


Figure 1.12 Discretization of the computational domain into a number of elements.

$$\phi(x,y) \;=\; N_1(x,y)\phi_1 + N_2(x,y)\phi_2 + N_3(x,y)\phi_3 \qquad (1.15)$$

where $N_i(x,y)$, $i = 1, 2, 3$ are the element interpolation functions

Equation (1.15) can be written in matrix form as

$$\phi(x,y) \;=\; \lfloor N_1 \quad N_2 \quad N_3 \rfloor \begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \end{Bmatrix} \;=\; \underset{(1\times 3)}{\lfloor N \rfloor} \; \underset{(3\times 1)}{\{\phi\}} \qquad (1.16)$$

where $\lfloor N \rfloor$ is the element interpolation function matrix and $\{\phi\}$ is the vector of the element nodal unknowns. In this textbook, the symbol $\lfloor\;\rfloor$ denotes a row matrix and $\{\;\}$ denotes a column matrix which is normally called a vector. More information about the matrix, its properties and manipulation can be found in Appendix A.

Step 3 *Deriving the finite element equations*. For example, the finite element equations for the triangular element as shown in Fig. 1.13 are in the form

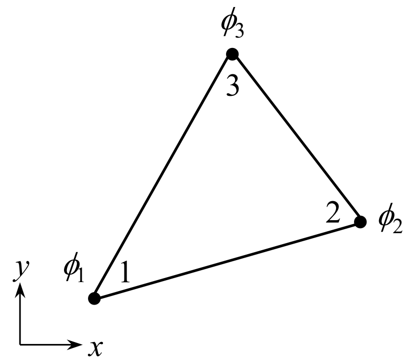


Figure 1.13 Typical triangular element with unknowns at nodes.

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix}_e \begin{Bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \end{Bmatrix}_e = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}_e \tag{1.17}$$

Or, in short

$$[K]_e \{\phi\}_e = \{F\}_e \tag{1.18}$$

This third step is considered the most important step in the finite element analysis. The finite element equations, in the form of Eq. (1.17), may be derived from

(a) Direct approach, which is presented in Chapter 2

(b) Variational approach, which is presented in Chapter 3

(c) Method of weighted residuals, which is presented in Chapter 4

Step 4 *Assembling the element equations to form a system of equations*. The system of equations is in the form of simultaneous algebraic equations

$$\sum(\textit{element equations}) \Rightarrow [K]_{sys}\{\phi\}_{sys} = \{F\}_{sys} \tag{1.19}$$

Step 5 *Applying the boundary conditions and solving for the nodal solutions*. Boundary conditions of the problem are then applied to the system of equations, Eq. (1.19), before solving for the nodal unknown of $\{\phi\}_{sys}$. The nodal unknowns could be displacements, temperatures, or fluid velocities for solid mechanics, heat transfer, or fluid flow problems, respectively.

Step 6 *Computing other quantities if needed.* After the nodal unknowns are determined, other quantities of interest can be further computed. For examples, the stresses, heat fluxes, and the flow rates can be computed from the now-known displacements, temperatures, and the flow velocities, respectively.

From the 6 steps described above, it is clear that there is a routine procedure in the finite element method. The most important step is the derivation of the finite element equations in step 3. The derivation of the finite element equations will be explained in details in Chapter 2, 3 and 4. Several examples will be presented in these chapters to ensure clear understanding of the method.

1.6 Applications of the Finite Element Method

The finite element method is widely used nowadays for analysis and design of new products as the method is suitable for problems with complex geometry. The method has been applied to analyze problems in different fields, such as in solid and structures, fluid flows, electromagnetics, etc. Some applications and the benefits of the method for solving engineering problems will be presented in this section.

In the past, the finite element method was mainly applied to solve problems in solid mechanics. The method was developed to analyze structural problems with irregular geometry. For example, the method was used to determine the deformation and stress that occur in an aircraft component as shown in Fig. 1.14. The figure shows the geometry of the aircraft component, the finite element mesh, and the computed stress distribution plotted on its deformed body. With such solution, designer can alter the geometry of the component to reduce the excessive stress that may occur.

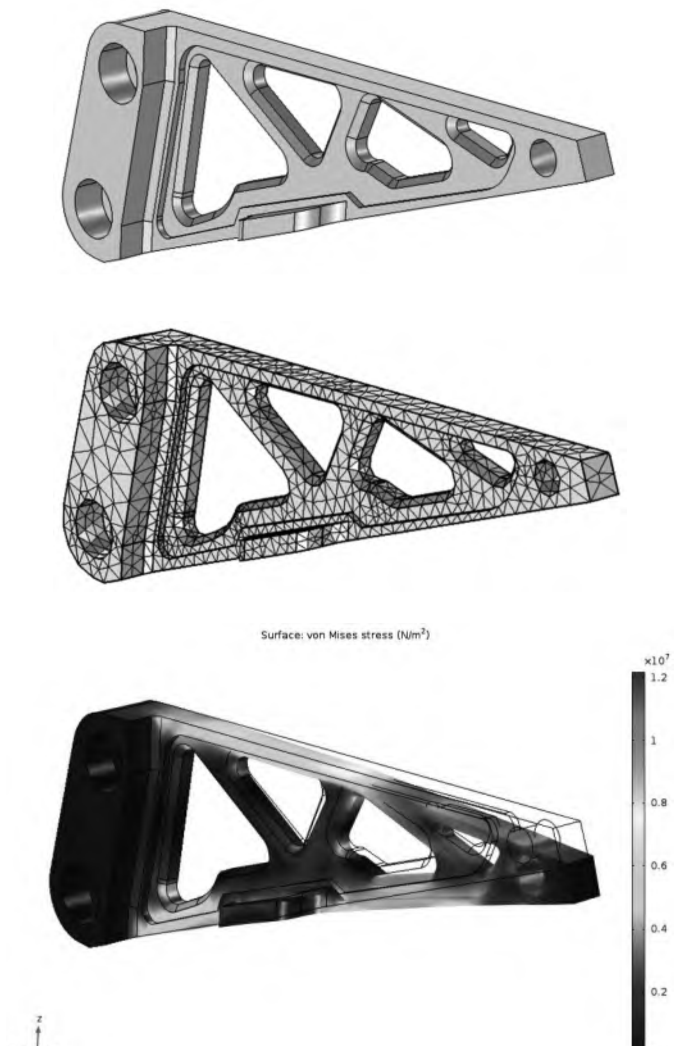


Figure 1.14 Stress in an aircraft structural component.

The finite element method can also be used to analyze structural dynamic problems. Figure 1.15 shows a finite element mesh of a passenger car frame. The modal analysis is performed to determine the natural frequencies and mode shapes of the car frame. Such analysis helps avoiding an uncontrollable vibration caused by the external force.

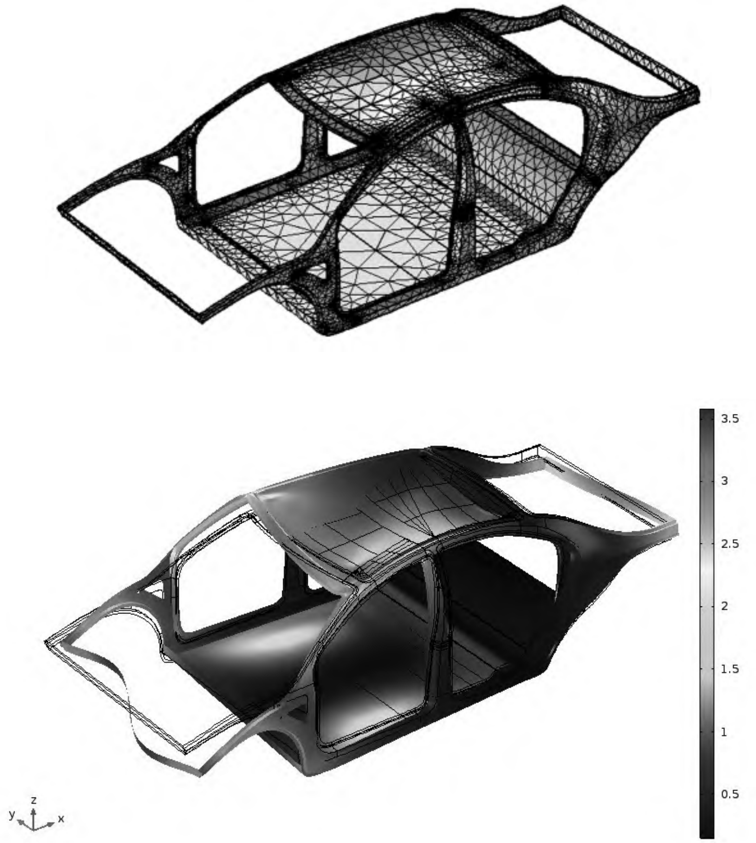


Figure 1.15 Modal analysis of a passenger car frame.

Practical problems may involve analyzing several engineering disciplines at the same time. Figure 1.16 shows the finite element mesh and the computed thermal stress on the deformed shape of a combustion engine cylinder. The increased cylinder temperature together with the combustion pressure cause the cylinder to deform with the thermal stress occurs. Note that, it is not possible to obtain such solution by using the analytical method normally learned in typical undergraduate courses.

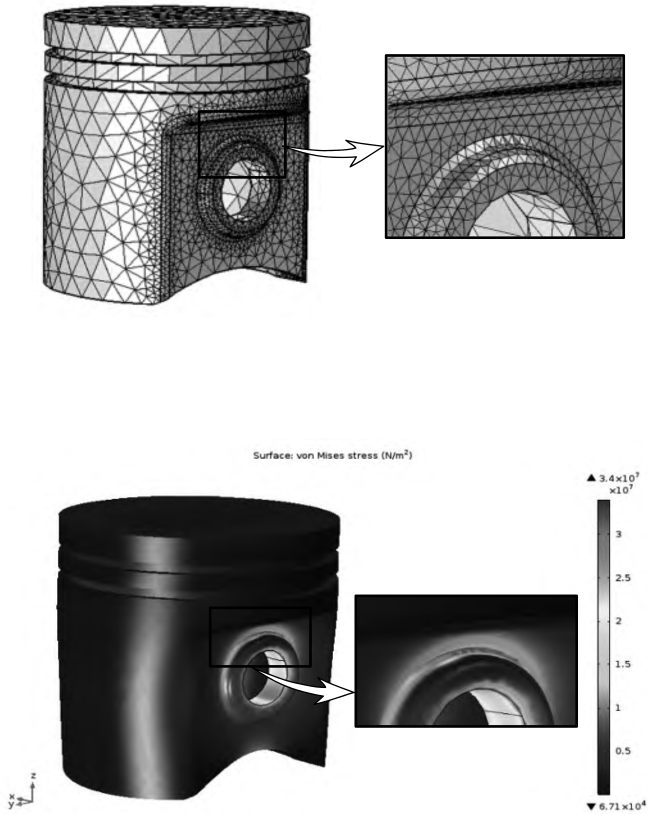


Figure 1.16 Thermal stress in combustion engine cylinder.

The finite element method has been extended to computational fluid dynamics for analyzing flow behaviors over different configurations. Figure 1.17 shows a flow in a piping system. Analysts today can use the finite element method to find the detailed flow behavior. The figure also shows the predicted flow velocity and pressure distributions inside the piping system. Such solutions helps them to fully understand the flow phenomena in the piping system.

Velocity Pressure

Figure 1.17 Flow in piping system.

The finite element method has been further extended to solve more complex problems, such as the compressible flow behavior over shuttle nose and cockpit at Mach 3. Figure 1.18 show the flow domain, the finite element meshes, and the predicted velocity, pressure and temperature distributions. Because the shape of shock waves is not known a priori, an adaptive meshing technique has been employed. The technique generates meshes that adapt automatically to the computed flow solutions. The combined finite element method and the adaptive meshing technique can provide high solution accuracy at a reduced computational effort.

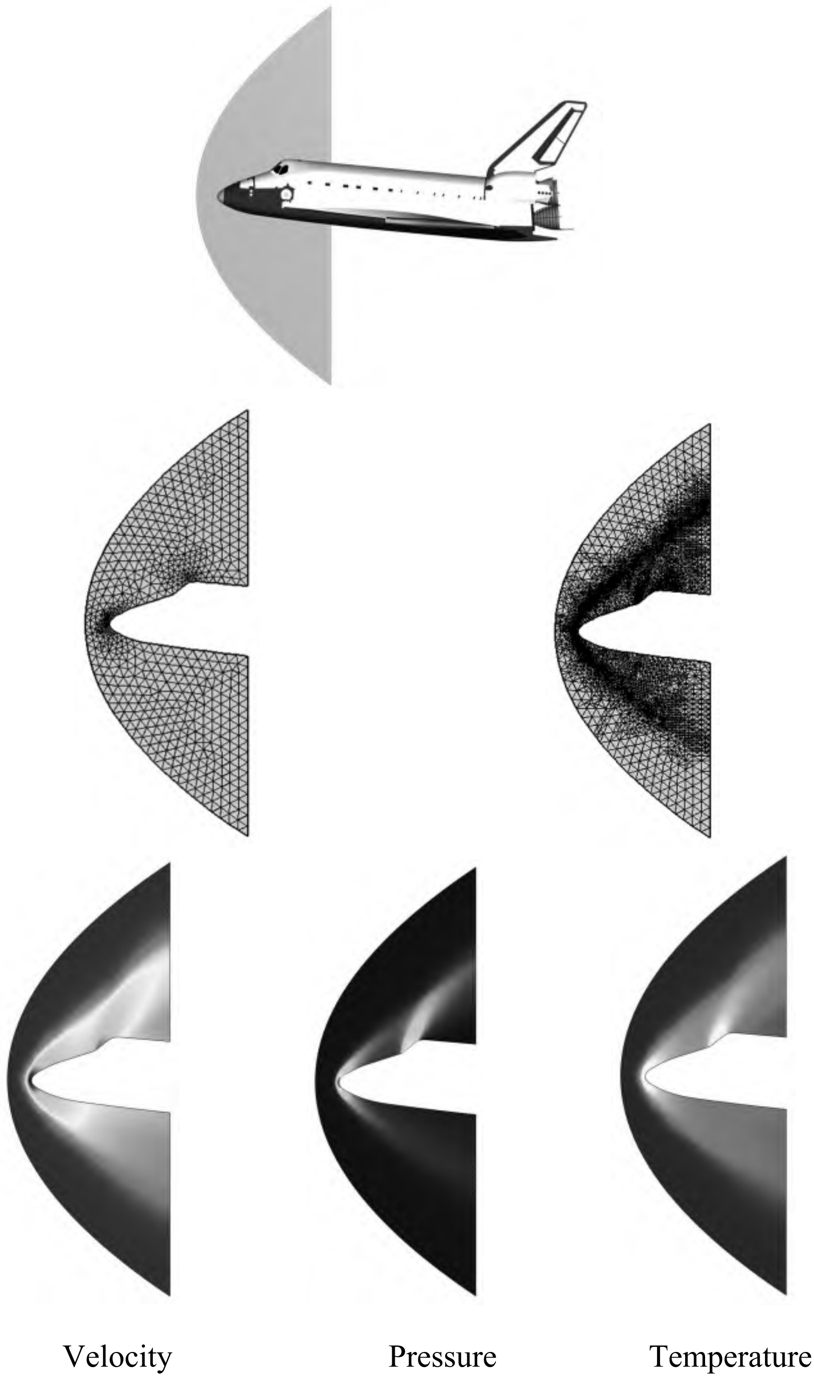


Figure 1.18 Flow over shuttle nose and cockpit.

Currently, such combined finite element method and adaptive meshing technique has been applied to analyze other interdisciplinary problems. One such example is the flow-structure interaction of a future transportation vehicle that flies at Mach 25. At that speed, complex flow behaviors including shock waves and expansions occur in the flow around the vehicle. Very high aerodynamic heating rate also occurs on the vehicle exterior. Such high heating rate causes the structure temperature to rise. The vehicle structure then deforms causing thermal stresses. The deformed structure, in turn, alters the flow field. The problem thus involves coupling between the fluid flow, the heat transfer in the structure, and the structural deformation. This coupled problem can be analyzed by the finite element method conveniently because the method is applicable to all disciplines.

1.7 Closure

In this chapter, overview of the finite element method and its applications were presented. The chapter started with the finite difference method because of it properties are similar to the finite element method. The finite difference method transforms differential equations into approximate algebraic equations. The first step of the finite difference method is to divide the domain into small rectangles. From the studied example, the method is simple and easy to understand. However, rectangles cannot accurately represent the arbitrary geometry with curves and circles. The method cannot provide accurate solutions for practical problems of which geometries are usually complex.

General procedure of the finite element method that consists of six steps was then explained. The first step is to discretize the computational domain into a number of elements. Element shapes could be triangle or quadrilateral which can model arbitrary two-dimensional geometry more precisely. These elements are connected by nodes at which the unknowns are to be determined. The most important step of the finite element method is the derivation of the finite element equations that correspond to the given problem which will be studied later in Chapter 2, 3 and 4. The element equations from each element are then assembled, leading to a large set of algebraic equations. Boundary conditions are applied prior to solving for solutions at all nodes.

At the end of this chapter, some applications analyzed by using the finite element method were presented. There are several areas in engineering that the finite element method can be applied to obtain the solutions. These areas include solid mechanics, heat transfer, fluid dynamics, and electromagnetics, etc. As the finite element method is a numerical technique for solving differential equations, the method has been nowadays extended to analyze problems in other fields, such as medical and environmental sciences. At the same time, a large number of the finite element software has been developed and are now used widely in many industries around the world in design of new products. It is thus important for engineers to understand the finite element theory, as well as to be familiar with the use of finite element software.

Chapter 2

Direct Method

2.1 Introduction

The first step of the finite element method is to discretize the computational domain of the problem into a number of small elements of different shapes and sizes. The finite element equations, corresponding to the problem type, are then derived. The type of problem can be solid mechanics, heat transfer, fluid mechanics, etc. The finite element equations are assembled to form a large set of simultaneous equations. Appropriate boundary conditions are then applied prior to solving the set of equations for solutions at nodes.

From the above procedure, the most important step of the finite element method is the derivation of the finite element equations. The finite element equations can be derived by several approaches. The simplest approach, the direct method, is presented in this chapter.

2.2 Derivation of Finite Element Equations in One Dimension

2.2.1 Rod Element

The truss problem can assist the understanding on the derivation of finite element equations. A simple configuration of a truss problem is shown in Fig. 2.1.

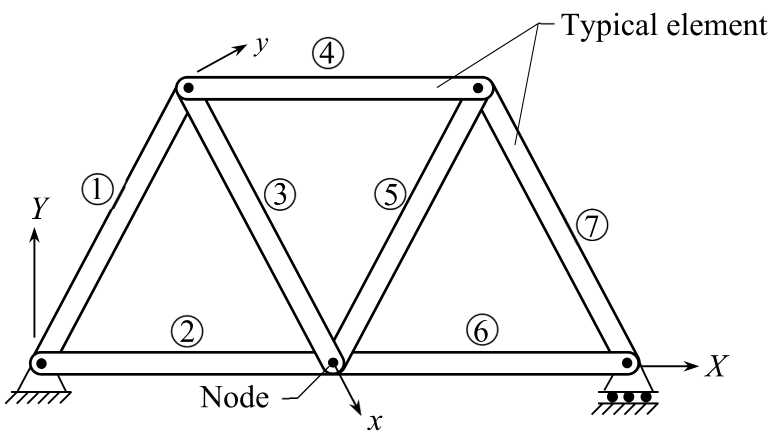


Figure 2.1 Simple configuration of a truss problem.

The above truss problem consists of 7 rods that are modeled by 7 rod elements in the global *X*-*Y* coordinate system. The rod element, sometimes called truss element, can elongate or shorten only in its axial direction. When the truss is loaded, each element deforms. Figure 2.2 shows a typical element containing two nodes 1 and 2 that deform with displacements u_1 and u_2, respectively. The rod has the cross-sectional area *A*, the length *L*, and the modulus of elasticity *E*. Under the equilibrium condition, the axial forces at the node numbers 1 and 2 are F_1 and F_2, respectively.

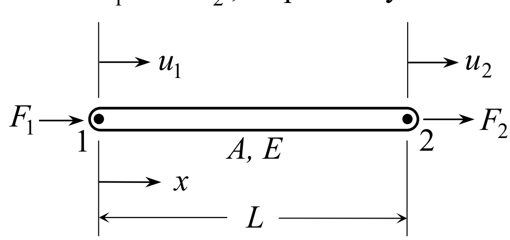


Figure 2.2 A typical rod or truss element.

The derivation for the finite element equations starts with the Hooke's Law which describes the relationship between the stress and strain as

$$\sigma = E\,\varepsilon \tag{2.1}$$

where σ is the stress and ε is the strain. If N denotes the axial force in the positive x-direction, then

$$N = \sigma A = (E\varepsilon) A$$

Or,

$$N = E\frac{u_2 - u_1}{L}A \tag{2.2}$$

Thus, from Fig. 2.2, we have

$$F_2 = -F_1 = N = \frac{AE}{L}(u_2 - u_1) \tag{2.3}$$

leading to two equations as

$$\frac{AE}{L}(u_1 - u_2) = F_1$$

$$-\frac{AE}{L}(u_1 - u_2) = F_2$$

These two equations can be re-written in a matrix form as

$$\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \tag{2.4}$$

Or, in a short form as

$$[K]\{u\} = \{F\} \tag{2.5}$$

where $[K]$ is the element stiffness matrix, $\{u\}$ is the vector of nodal displacements, and $\{F\}$ is the vector of nodal forces.

The format for the element equations in Eq. (2.4) can also be directly applied for the spring element as shown in Fig. 2.3,

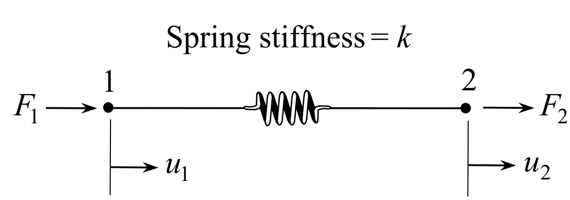


Figure 2.3 A typical spring element.

where k is the spring stiffness with a unit of force over the extension, such as N/cm. Thus, the corresponding finite element equations for the spring in Fig. 2.3 are

$$k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \tag{2.6}$$

2.2.2 Heat Conduction Element

Conduction heat transfer occurs due to different temperatures in a material. It is another simple example that can be used to demonstrate the derivation of finite element equations. Figure 2.4 shows a heat conduction problem in three materials. Heat is transferred across the three materials from a high temperature at the left wall to a lower temperature at the right wall.

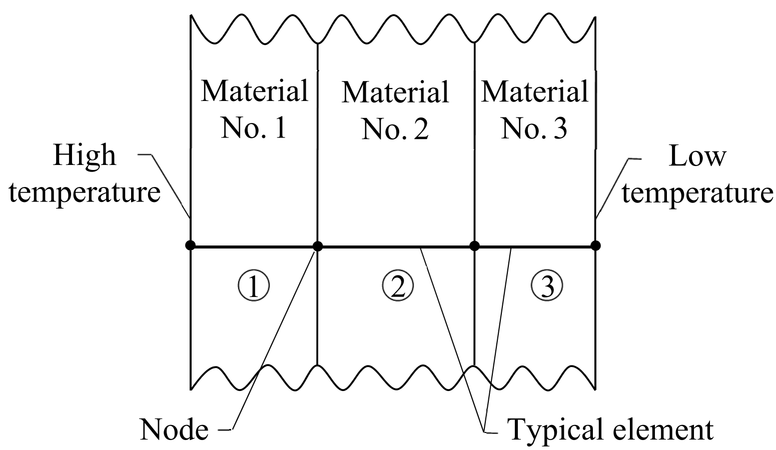


Figure 2.4 Conduction heat transfer in different materials.

Three finite elements may be used to model the problem as shown in the figure. A typical element consists of two nodes, numbering 1 and 2, as shown in Fig. 2.5. The element has the thermal conductivity coefficient k, the cross-sectional area for conduction A and the length L in its local x-direction. The temperatures are T_1 and T_2, which cause the heat fluxes Q_1 and Q_2 at the nodes 1 and 2, respectively.

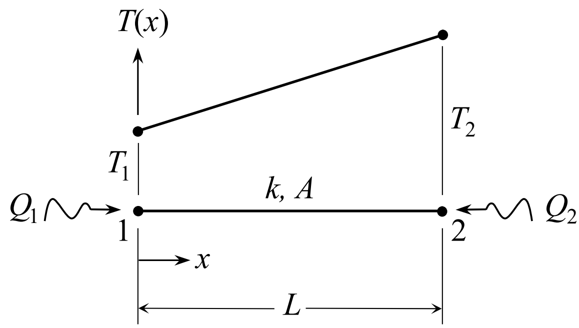


Figure 2.5 One-dimensional heat conduction element.

According to Fourier's Law, the amount of heat flux Q depends on the thermal conductivity coefficient k, the cross-sectional area A, and the temperature gradient $\partial T/\partial x$ in the form

$$Q = -kA\frac{\partial T}{\partial x} \tag{2.7}$$

For a positive temperature gradient, the negative sign is used to indicate that the heat flux flows from left to right. Thus, from Fig. 2.5, we have

$$Q_1 = -Q_2 = Q = -k\,A\frac{T_2 - T_1}{L}$$

which leads to two equations as

$$Q_1 = \frac{k\,A}{L}\,(T_1 - T_2) \tag{2.8}$$

and

$$Q_2 = \frac{kA}{L}\,(-T_1 + T_2) \tag{2.9}$$

Equations (2.8) and (2.9) can be written together as

$$\frac{kA}{L}\,(T_1 - T_2) = Q_1$$

$$\frac{kA}{L}\,(-T_1 + T_2) = Q_2$$

which can be written in matrix form as follow

$$\frac{kA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} \tag{2.10}$$

Or, in short

$$[K]\{T\} = \{Q\} \tag{2.11}$$

where $[K]$ is the element conduction matrix, $\{T\}$ is the vector of nodal temperatures, and $\{Q\}$ is the vector of nodal heat fluxes.

2.2.3 Pipe Element

The finite element method can be used to determine the flow rate in a piping system. Figure 2.6 illustrates an example of a piping system that consists of 5 elements.

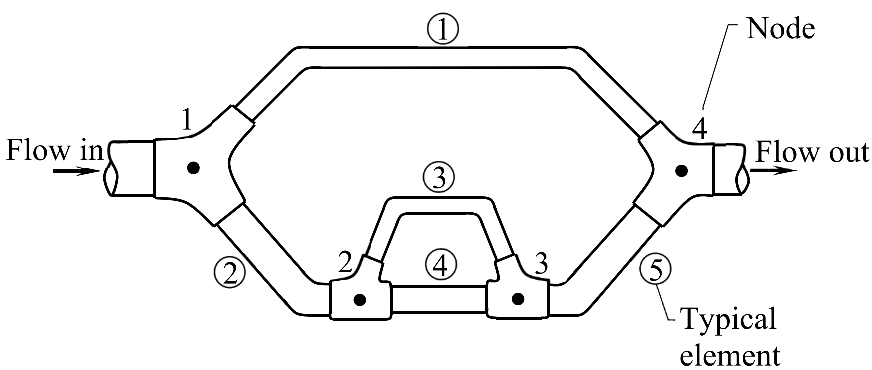


Figure 2.6 Flow in a piping system.

For steady-state laminar incompressible flow in a circular pipe, the flow rate depends on the pressure gradient in its local x-direction as

$$Q = -\frac{\pi D^4}{128\mu}\frac{\partial P}{\partial x} \tag{2.12}$$

where Q is the flow rate, D is the diameter of the pipe, μ is the viscosity of the fluid, and P is the pressure.

In order to derive the finite element equations, a typical pipe element as shown in Fig. 2.7 is used. This element has a diameter D, the length L and is in the local x-coordinate. The element has two nodes at its ends with the numbers 1 and 2. The pressures at these two nodes are P_1 and P_2 that induce the flow rates of Q_1 and Q_2, respectively.

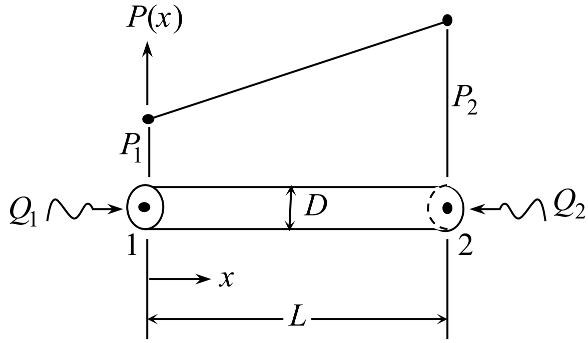


Figure 2.7 A typical pipe element.

From Fig. 2.7, if Q denotes the flow rate in the positive x-direction, then

$$Q_1 = -Q_2 = Q = -\frac{\pi D^4}{128\mu}\frac{P_2 - P_1}{L}$$

Or, $$Q_1 = \frac{\pi D^4}{128\mu L}(P_1 - P_2)$$

and $$Q_2 = \frac{\pi D^4}{128\mu L}(-P_1 + P_2)$$

These two equations can be written together as

$$\frac{\pi D^4}{128\mu L}(P_1 - P_2) = Q_1 \tag{2.13}$$

$$\frac{\pi D^4}{128\mu L}(-P_1 + P_2) = Q_2 \tag{2.14}$$

which can also be written in the matrix form as

$$\frac{\pi D^4}{128\mu L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} \tag{2.15}$$

Or, in short

$$[K]\{P\} = \{Q\} \tag{2.16}$$

where, $[K]$ is the element fluidity matrix, $\{P\}$ is the vector of nodal pressures, and $\{Q\}$ is the vector of nodal flow rates.

From the three simple examples of truss, heat conduction, and fluid flow problems, it can be seen that the finite element equations can be derived from fundamental engineering relationships. These derived finite element equations are in the same form as shown in Eqs. (2.5), (2.11), and (2.16) as

$$[K]\{U\} = \{F\} \tag{2.17}$$

where $[K]$ is the element matrix that depends on problem type, $\{U\}$ is the vector of nodal unknowns, and $\{F\}$ is the vector of nodal loads.

2.3 Assembling of the Finite Element Equations

The derived finite element equations from all elements are assembled together to represent the entire problem. To understand the process for assembling of the finite element equations, the spring system as shown in Fig. 2.8 is studied.

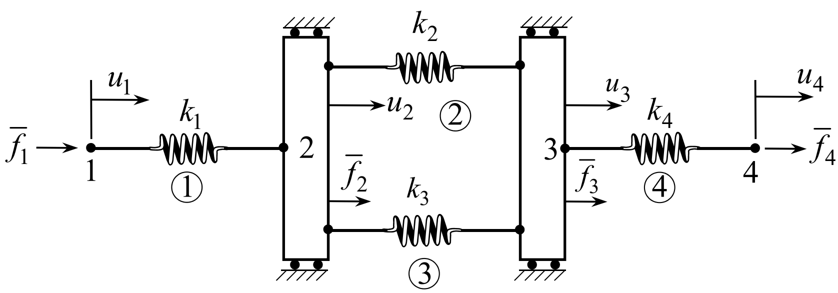


Figure 2.8 Example of a spring system in one dimension.

In Fig. 2.8, a spring system consists of four different spring elements that can deform in one dimension. The four elements have the spring stiffness of k_1, k_2, k_3 and k_4, respectively. There are four nodes in this spring system with the nodal displacements of u_1, u_2, u_3 and u_4. Thus, the system of equations contains four equations in the form

$$\underset{(4\times4)}{\begin{bmatrix} \bar{k}_{11} & \bar{k}_{12} & \bar{k}_{13} & \bar{k}_{14} \\ \bar{k}_{21} & \bar{k}_{22} & \bar{k}_{23} & \bar{k}_{24} \\ \bar{k}_{31} & \bar{k}_{32} & \bar{k}_{33} & \bar{k}_{34} \\ \bar{k}_{41} & \bar{k}_{42} & \bar{k}_{43} & \bar{k}_{44} \end{bmatrix}} \underset{(4\times1)}{\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix}} = \underset{(4\times1)}{\begin{Bmatrix} \bar{f}_1 \\ \bar{f}_2 \\ \bar{f}_3 \\ \bar{f}_4 \end{Bmatrix}} \tag{2.18}$$

Or, in short

$$[K]_{sys}\,\{u\}_{sys} \;=\; \{F\}_{sys} \tag{2.19}$$

where the subscript *sys* represents the system of equations.

2.3.1 Assembling of Element Matrices

Equation (2.19) implies that the coefficients in the system stiffness matrix, $[K]_{sys}$, are needed before computing nodal displacements. The coefficients in the system stiffness matrix are contributed from the stiffness matrices of all elements. For example, if the element number ① in Fig. 2.8 is isolated as shown in Fig. 2.9, the finite element equations for this element, according to Eq. (2.6), are

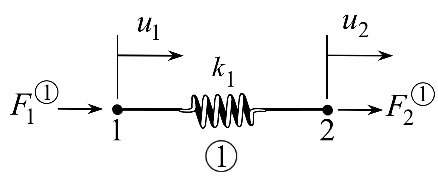


Figure 2.9 Spring element number ① from Fig. 2.8.

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1^{①} \\ F_2^{①} \end{Bmatrix} \tag{2.20}$$

The above two equations in Eq. (2.20) can be put into Eq. (2.18) as

$$\begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1^{①} \\ F_2^{①} \\ 0 \\ 0 \end{Bmatrix} \tag{2.21}$$

Similarly, the spring element number ② in Fig. 2.8 can be isolated as shown as Fig. 2.10. This spring element yields the element equations as

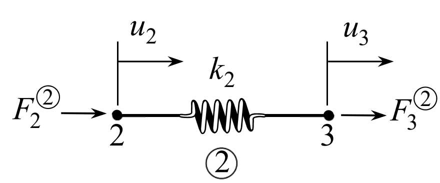


Figure 2.10 Spring element number ② from Fig. 2.8.

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_2^{②} \\ F_3^{②} \end{Bmatrix} \tag{2.22}$$

which can be put into Eq. (2.18) as

$$\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & k_2 & -k_2 & 0 \\ 0 & -k_2 & k_2 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2^{②} \\ F_3^{②} \\ 0 \end{Bmatrix} \tag{2.23}$$

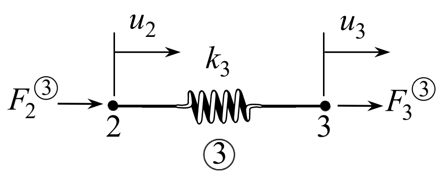


Figure 2.11 Spring element number ③ from Fig. 2.8.

Similarly, the spring element number ③ in Fig. 2.8 can be isolated as shown in Fig. 2.11. The element equations for this spring element are

$$\begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_2^{③} \\ F_3^{③} \end{Bmatrix} \tag{2.24}$$

which can be put into Eq. (2.18) as

$$\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & k_3 & -k_3 & 0 \\ 0 & -k_3 & k_3 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2^{③} \\ F_3^{③} \\ 0 \end{Bmatrix} \tag{2.25}$$

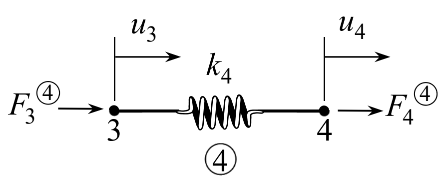


Figure 2.12 Spring element number ④ from Fig. 2.8.

Finally, the element number ④ in Fig. 2.8 can be isolateed as shown in Fig. 2.12. The element equations for this spring element are

$$\begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_3^{④} \\ F_4^{④} \end{Bmatrix} \tag{2.26}$$

which can also be put into Eq. (2.18) as

$$\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & k_4 & -k_4 \\ 0 & 0 & -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F_3^{④} \\ F_4^{④} \end{Bmatrix} \tag{2.27}$$

If we add Eqs. (2.21), (2.23), (2.25) and (2.27) from the elements ①, ②, ③ and ④, this is equivalent to assembling the four springs together leading to the spring system as shown in Fig. 2.8, that is

$$\begin{bmatrix} k_1 & -k_1 & & \\ -k_1 & k_1+k_2+k_3 & -k_2-k_3 & \\ & -k_2-k_3 & k_2+k_3+k_4 & -k_4 \\ & & -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1^{\textcircled{1}} \\ F_2^{\textcircled{1}} + F_2^{\textcircled{2}} + F_2^{\textcircled{3}} \\ F_3^{\textcircled{2}} + F_3^{\textcircled{3}} + F_3^{\textcircled{4}} \\ F_4^{\textcircled{4}} \end{Bmatrix} \tag{2.28}$$

The summation of the element forces for the second and third equations on the right-hand-side of the Eq. (2.28) above must be equal to the resultant forces $\bar{f}_2$ and $\bar{f}_3$, respectively, i.e.,

$$F_2^{\textcircled{1}} + F_2^{\textcircled{2}} + F_2^{\textcircled{3}} = \bar{f}_2$$

and

$$F_3^{\textcircled{2}} + F_3^{\textcircled{3}} + F_3^{\textcircled{4}} = \bar{f}_3$$

while

$$F_1^{\textcircled{1}} = \bar{f}_1 \quad \text{and} \quad F_4^{\textcircled{4}} = \bar{f}_4$$

The system stiffness matrix on the left hand side of Eq. (2.28) is

$$[K]_{sys} = \sum [K]_e$$

$$= \begin{array}{c} \begin{array}{cccc} (1) & (2) & (3) & (4) \end{array} \\ \begin{bmatrix} k_1 & -k_1 & & \\ -k_1 & k_1+k_2+k_3 & -k_2-k_3 & \\ & -k_2-k_3 & k_2+k_3+k_4 & -k_4 \\ & & -k_4 & k_4 \end{bmatrix} \end{array} \begin{array}{c} \\ (1) \\ (2) \\ (3) \\ (4) \end{array} \tag{2.29}$$

where the numbers 1, 2, 3, and 4 in the parenthesis represent the rows and columns of the system matrix corresponding to the equation numbers 1, 2, 3 and 4, respectively.

Assembling the system equations from the element equations, Eq. (2.28) can be easily done without writing their element equations as shown by Eqs. (2.21), (2.23), (2.25) and (2.27) in details. A simpler approach involves the proper designation of numbers for rows and columns. For example, the spring element number ① is connected by the nodes 1 and 2, therefore the rows and columns of its element stiffness matrix is designated with the nodal numbers as

$$[K]_{element①} = \begin{array}{cc} \begin{array}{cc} (1) & (2) \end{array} & \\ \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} & \begin{array}{c} (1) \\ (2) \end{array} \end{array} \tag{2.30}$$

Similarly, element number ② is connected by the nodes 2 and 3, thus

$$[K]_{element②} = \begin{array}{cc} \begin{array}{cc} (2) & (3) \end{array} & \\ \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} & \begin{array}{c} (2) \\ (3) \end{array} \end{array} \tag{2.31}$$

The element number ③ is also connected by the nodes 2 and 3 as the element number ②, therefore

$$[K]_{element③} = \begin{array}{cc} \begin{array}{cc} (2) & (3) \end{array} & \\ \begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix} & \begin{array}{c} (2) \\ (3) \end{array} \end{array} \tag{2.32}$$

Finally, for element number ④, which is connected by the nodes 3 and 4,

$$[K]_{element④} = \begin{array}{cc} \begin{array}{cc} (3) & (4) \end{array} & \\ \begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix} & \begin{array}{c} (3) \\ (4) \end{array} \end{array} \tag{2.33}$$

Once both the rows and columns of all element matrices are numbered, the system stiffness matrix can be formed by putting the coefficients of element matrices into the appropriate rows and columns as

$$\underset{(4\times4)}{[K]_{sys}} = \begin{array}{cc} \begin{array}{cccc} (1) & (2) & (3) & (4) \end{array} & \\ \left[\begin{array}{c|c|c|c} k_1 & -k_1 & & \\ \hline -k_1 & k_1+k_2+k_3 & -k_2-k_3 & \\ \hline & -k_2-k_3 & k_2+k_3+k_4 & -k_4 \\ \hline & & -k_4 & k_4 \end{array}\right] & \begin{array}{c} (1) \\ (2) \\ (3) \\ (4) \end{array} \end{array} \tag{2.34}$$

For example, the coefficient $-k_1$ which is in the row 1 and column 2 of the matrix for element number ① is placed in the row 1 and column 2 of the system matrix as shown in Eq. (2.34).

The method for assembling element equations into the system of equations in most of the finite element software uses the procedure described above because the procedure is easy and convenient, especially for a complex structure where the node and element numbering is arbitrary. For instance, when the element number 76 is connected by the nodes 1024 and 582, the above procedure is still applicable and works well.

2.3.2 Characteristics of the System Matrix

When the system stiffness matrix in Eq. (2.34) is considered, it is found that there are non-zero coefficients near the matrix diagonal as shown in Fig. 2.13. In addition, the system matrix is symmetric for the spring problem. As an example, if a problem consists of 1,000 nodes, the system matrix contains the total amount of 1,000×1,000 = 1,000,000 coefficients. But with symmetric matrix, just more than 500,000 coefficients may be stored in the computer memory.

The system matrix in Fig. 2.13 also shows that the non-zero values are presented within its boundary of the bandwidth *BW*, while the rest of the coefficients outside the boundary are all zero. These zero coefficients are not needed to be stored in the computer memory. Because of the symmetric system matrix, only coefficients in the half-bandwidth (*HBW*) boundary are needed to be stored. As an example, if there are 1,000 nodes in the model and the half-bandwidth is 2, only 1,000×2 = 2,000 coefficients are stored in the computer memory. This procedure thus requires minimum computer memory and is adequate for the problem solving.

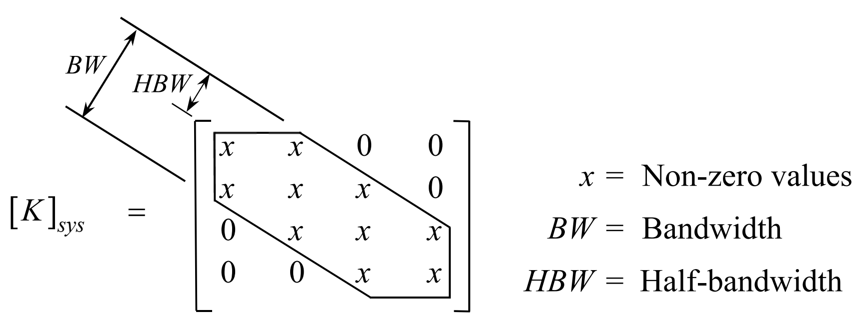


Figure 2.13 Characteristic of the system stiffness matrix with the bandwidth.

From Fig 2.13 and the above explanation, the concept of the half-bandwidth is important for determining the computer memory requirement prior to the analysis of a given problem. The half-bandwidth depends on the number of grid points and can be calculated from

$$HBW = \left(1 + NDIF\right) \times NDOF \tag{2.34}$$

where HBW is the half-bandwidth, $NDIF$ is the largest difference between the element node numbers, and $NDOF$ represents the degree of freedom of the nodes. For one-dimensional problem, the value of $NDOF$ is equal to one. From Fig. 2.8, the HBW can be calculated as

$$HBW = (1+1)\times 1 = 2$$

which corresponds to Eq. (2.34) and Fig. 2.13.

If the node numbers 2 and 4 in Fig 2.8 are switched, $NDIF = 4 - 1 = 3$ occurs at element number 1. Therefore

$$HBW = (1+3)\times 1 = 4$$

Then system matrix in Fig. 2.13 changes to

$$[K]_{sys} = \begin{bmatrix} \boxed{\begin{matrix} x & 0 & 0 & x \\ 0 & x & x & 0 \\ 0 & x & x & 0 \\ x & 0 & 0 & x \end{matrix}} \end{bmatrix}$$

In this case, the boundary of the bandwidth is larger indicating that more coefficients are needed to be stored in the computer memory.

This example illustrates that although the node number assignment in a finite element model does not alter the solution, it affects the analysis efficiency. It may require more computer memory and, therefore, take a longer computational time to find the solution. One thing to keep in mind at all times is that the difference between the node numbers of each element (*NDIF*) should be as small as possible. In reality, however, the finite element model may be very complicate and involve a large number of elements and nodes, resulting in a difficulty in keeping the value of *NDIF* low. Today, there are many small computer programs that help numbering nodes such that the bandwidths are minimized. These bandwidth optimization programs reduce the computer memory requirement as well as the computational time for analyzing the problem.

2.3.3 One-dimensional Example

After studying the characteristics of the element equations, such as for the rod element as explained in section 2.2.1, and the method for assembling element equations to form the system of equations as presented in section 2.3.1, these concepts are then applied to analyze a one-dimensional example as follows.

Example 2.1 Two bars have the modulus of elasticity of 5×10^7 and 10×10^7 N/m^2, lengths of 0.5 m and 1 m, and cross-sectional areas of 20 cm^2 and 10 cm^2, respectively. They are connected at the node number 2 as shown in Fig. 2.14. The left end of the left bar is fixed on a wall at node 1, while the right end of the right bar is subjected to an applied force in the right direction of 500 N at node 3. In addition, the node 2 is also subjected to the applied force of 300 N pointing to the right. Use the finite element method to determine the displacements at nodes 2 and 3, as well as the stresses and the forces within both bars.

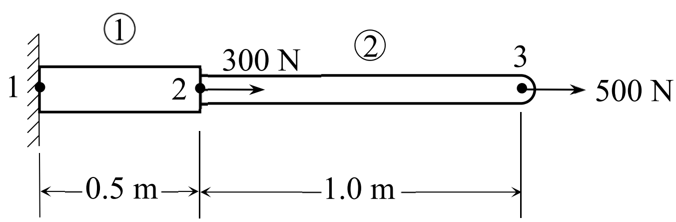


Figure 2.14 Two-bars model with two elements and three nodes.

Solution From Fig. 2.14, a finite element model which consists of 2 elements and 3 nodes is re-drawn for clarity as shown in Fig. 2.15. The nodes 1, 2, and 3 have the displacements u_1, u_2 and u_3, respectively. These nodes may be subjected to the applied external forces (or reaction forces) $\bar{f}_1$, $\bar{f}_2$ and $\bar{f}_3$, respectively.

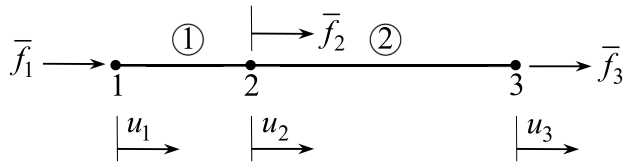


Figure 2.15 Finite element model of the two bars.

From the finite element equations, Eq. (2.4) in section 2.2.1, the derived stiffness matrix for the rod element with cross-sectional area A, modulus of elasticity E, and length L is

$$[K]_e = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{2.4}$$

Therefore, the stiffness matrix for the element ① is

$$[K]_{element\,①} = \begin{array}{c} \begin{array}{cc} \quad(1) & \qquad(2) \end{array} \\ \begin{bmatrix} \dfrac{A_1E_1}{L_1} & -\dfrac{A_1E_1}{L_1} \\ -\dfrac{A_1E_1}{L_1} & \dfrac{A_1E_1}{L_1} \end{bmatrix} \end{array} \begin{array}{c} (1) \\ \\ (2) \end{array} \tag{2.35}$$

while the stiffness matrix for the element ② is

$$[K]_{element\,②} = \begin{array}{c} \begin{array}{cc} \quad(2) & \qquad(3) \end{array} \\ \begin{bmatrix} \dfrac{A_2E_2}{L_2} & -\dfrac{A_2E_2}{L_2} \\ -\dfrac{A_2E_2}{L_2} & \dfrac{A_2E_2}{L_2} \end{bmatrix} \end{array} \begin{array}{c} (2) \\ \\ (3) \end{array} \tag{2.36}$$

Thus, the element equations for both elements can be assembled to yield the system equations as

$$\left[\begin{array}{c:c:c} \dfrac{A_1E_1}{L_1} & -\dfrac{A_1E_1}{L_1} & \\ \hdashline -\dfrac{A_1E_1}{L_1} & \dfrac{A_1E_1}{L_1}+\dfrac{A_2E_2}{L_2} & -\dfrac{A_2E_2}{L_2} \\ \hdashline & -\dfrac{A_2E_2}{L_2} & \dfrac{A_2E_2}{L_2} \end{array}\right]\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} \bar{f}_1 \\ \bar{f}_2 \\ \bar{f}_3 \end{Bmatrix} \tag{2.37}$$

where the vector on the right-hand side of Eq. (2.37) consists of the external or reaction forces at the three nodes.

For the element ①,

$$\frac{A_1 E_1}{L_1} = \frac{(.002)\left(5\times10^7\right)}{0.5} = 2\times10^5 \ \text{N/m}$$

and for the element ②,

$$\frac{A_2 E_2}{L_2} = \frac{(.001)\left(10\times10^7\right)}{1.0} = 1\times10^5 \ \text{N/m}$$

Then, the system equations in Eq. (2.37) become

$$10^5\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} \bar{f}_1 \\ \bar{f}_2 \\ \bar{f}_3 \end{Bmatrix} \tag{2.38}$$

Before solving the system equations, the boundary conditions must be specified. A simple rule for applying the boundary conditions to the system equations is as follows. If the force on the right-hand side force vector is known, then the displacement on the left-hand side is unknown, and vice versa. For this example, the known and unknown displacements and forces are as follows.

Node No.	**Displacement**	**Force**
1	$u_1 = 0$	$\bar{f}_1 = ?$
2	$u_2 = ?$	$\bar{f}_2 = 300$
3	$u_3 = ?$	$\bar{f}_3 = 500$

Therefore, Eq. (2.38) becomes

$$10^5\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} \bar{f}_1 \\ 300 \\ 500 \end{Bmatrix} \tag{2.39}$$

It is noted that if there is no external force acting on node 2, the value of $\bar{f}_2$ on the right-hand side must be zero. Or, if the external force at node 2 points to the left, then the value of $\bar{f}_2$ is -300.

From the system equations, Eq. (2.39), the nodal displacements can be determined from the simultaneous equations 2 and 3, such that