

Yongtao Lyu

# Finite Element Method

Element Solutions

# Finite Element Method

Yongtao Lyu

# Finite Element Method

## Element Solutions



@seismicisolation

Yongtao Lyu  
Department of Engineering Mechanics  
Dalian University of Technology  
Dalian, Liaoning, China

ISBN 978-981-19-3362-2      ISBN 978-981-19-3363-9 (eBook)  
<https://doi.org/10.1007/978-981-19-3363-9>

© The Editor(s) (if applicable) and The Author(s), under exclusive license to Springer Nature Singapore Pte Ltd. 2022

This work is subject to copyright. All rights are solely and exclusively licensed by the Publisher, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilms or in any other physical way, and transmission or information storage and retrieval, electronic adaptation, computer software, or by similar or dissimilar methodology now known or hereafter developed.

The use of general descriptive names, registered names, trademarks, service marks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

The publisher, the authors and the editors are safe to assume that the advice and information in this book are believed to be true and accurate at the date of publication. Neither the publisher nor the authors or the editors give a warranty, expressed or implied, with respect to the material contained herein or for any errors or omissions that may have been made. The publisher remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

This Springer imprint is published by the registered company Springer Nature Singapore Pte Ltd.  
The registered company address is: 152 Beach Road, #21-01/04 Gateway East, Singapore 189721,  
Singapore

# Preface

The textbook *Finite Element Method: Element Solutions* is intended to be used by the senior engineering undergraduate and the graduate students. Nowadays, the finite element method has become one of the most-widely used techniques in all the engineering fields, including the aerospace engineering, the mechanical engineering, the car engineering, the civil engineering, the biomedical engineering, etc. To unveil the finite element technique, the textbook provides a detailed description of the finite element method, starting from the most important basic theoretical basis, e.g., the Galerkin method, the variational principle, followed by the detailed description of the various types of finite elements including the bar, the beam, the triangular, the rectangular, the 3D elements. The primary aim of the textbook is to provide a comprehensive description of the FE solutions using different types of elements. Therefore, the properties of different elements and the solution discrepancies caused by the application of different elements are highlighted in this textbook. Therefore, the textbook is very helpful for engineers to understand the behaviors of different types of elements. Additionally, the textbook can help the students and engineers write their own FE codes based on the theories presented in the book. Furthermore, the textbook can serve as the basis for some advanced computational mechanics courses, such as the nonlinear finite element method. It should be noted that the type of analysis presented in this textbook is limited to the static mechanical analysis. With regard to other types of mechanical analysis using the finite element method, such as the dynamic and thermal analysis, the readers should refer to other finite element books.

Dalian, China

Yongtao Lyu

**Acknowledgements** The publication of this textbook is made possible thanks to the huge assistances made by the graduate students, Mr. Wenhong Liu, Mr. Jiongyi Wu, Mr. Tingxiang Gong, Miss Yanli Wu, Mr. Yibiao Niu and Mr. Dangqing Wang. In addition, I would like to acknowledge the Department of Engineering Mechanics, Dalian University of Technology for giving me the

chance to teach the finite element method. I have learned a lot in the teaching process. In the end, I would like to acknowledge the Faculty of Vehicle Engineering and Mechanics, Dalian University of Technology, for providing the publication funding.

# Contents

<b>1</b>	<b>Introduction .....</b>	1
1.1	Introduction .....	1
1.2	Introduction to the Finite Element Method .....	1
1.2.1	Brief History of the Finite Element Method .....	1
1.2.2	Introduction to the Commonly Used Finite Element Software .....	3
1.3	Some Basic Knowledge from the Theory of Elasticity .....	4
1.3.1	The Four Basic Assumptions .....	4
1.3.2	Some Preliminary Knowledge on Tensor Operation .....	6
1.3.3	Three Main Equations in the Theory of Elasticity .....	7
1.3.4	Types of Boundary Conditions .....	14
1.3.5	The Plane Stress and Plane Strain Problems .....	15
<b>2</b>	<b>Theoretical Basis of the Finite Element Method .....</b>	19
2.1	Introduction .....	19
2.2	Equivalent Integral Form of the Differential Equation .....	20
2.3	The Weighted Residual Method .....	22
2.4	The Variational Principle .....	35
2.4.1	Establishment of the Variational Principle for Differential Equations .....	35
2.4.2	The Ritz Method .....	37
2.5	The Principle of Virtual Work .....	40
2.5.1	The Virtual Displacement Principle .....	41
2.5.2	The Virtual Stress Principle .....	42
2.5.3	The Minimal Potential Energy Principle .....	43
2.5.4	The Minimal Complementary Energy Principle .....	43
<b>3</b>	<b>Finite Element Analysis Using Bar Element .....</b>	45
3.1	Introduction .....	45
3.2	The Finite Element Calculation Procedure .....	45

3.3	Property of the Shape Function for the bar Element .....	51
3.4	Property of the Stiffness Matrix for the bar Element .....	53
3.5	The Coordinate Transformation for bar Elements .....	56
3.6	An Example of the FE Analysis Using the bar Element .....	59
<b>4</b>	<b>Finite Element Analysis Using Beam Element .....</b>	<b>65</b>
4.1	Introduction .....	65
4.2	The Finite Element Calculation Procedure .....	65
4.2.1	Some Preliminary Knowledge on Beam Element .....	66
4.3	FE Analysis Procedure Using Beam Element .....	69
4.4	Calculation of the Elemental Equivalent Nodal Forces .....	75
4.5	Coordinate Transformation in the Beam Analysis .....	82
4.6	Treatment of the Boundary Conditions .....	88
<b>5</b>	<b>Finite Element Analysis Using Triangular Element .....</b>	<b>93</b>
5.1	Introduction .....	93
5.2	FE Analysis Procedure Using Triangular Element .....	93
5.3	Properties of the Shape Function for Triangular Element .....	98
5.4	The Area Coordinate .....	102
5.5	Properties of the Global Stiffness Matrix .....	106
5.6	Calculation of the Equivalent Nodal Forces .....	110
5.7	An Example of the FE Analysis Using Triangular Element .....	114
<b>6</b>	<b>Finite Element Analysis Using Rectangular Element .....</b>	<b>119</b>
6.1	Introduction .....	119
6.2	FE Analysis Procedure Using Rectangular Element .....	119
6.3	The Shape Function for Rectangular Element .....	127
6.4	Iso-Parametric Element .....	132
6.5	Numerical Integration .....	140
6.5.1	The Newton–Cotes Integration Method .....	141
6.5.2	The Gauss Integration Method .....	144
6.6	An Example of the FE Analysis Using Rectangular Element .....	148
<b>7</b>	<b>Finite Element Analysis Using 3D Elements .....</b>	<b>159</b>
7.1	Introduction .....	159
7.2	FE Analysis Using Tetrahedral Element .....	159
7.2.1	FE Analysis Procedure Using Tetrahedral Element .....	159
7.2.2	The Volume Coordinates and Their Properties .....	164
7.3	FE Analysis Procedure Using Hexahedral Element .....	166
<b>8</b>	<b>High Order Lagrange Element .....</b>	<b>171</b>
8.1	Introduction .....	171
8.2	Definition of Lagrange and Hermite Elements .....	171
8.3	The 1D High Order Lagrange Element .....	172

8.4	The 2D High Order Lagrange Element .....	176
8.4.1	The High Order Triangular Element .....	176
8.4.2	The High Order Rectangular Element .....	181
8.5	The 3D High Order Lagrange Element .....	187
8.5.1	The High Order Tetrahedral Element .....	187
8.5.2	The High Order Hexahedral Element .....	191

# About the Author

**Dr. Yongtao Lyu** is an associate professor of engineering mechanics at the Department of Engineering Mechanics, Dalian University of Technology. Dr. Lyu received his bachelor and master degrees from the Department of Engineering Mechanics, Huazhong University of Technology, Wuhan, China. In November, 2010, he received the Ph.D. degree from Cardiff University, Cardiff, UK. His Ph.D. project is entitled “Soft tissue modelling and facial movement simulation using the finite element method” which is funded by the British Engineering and Physical Sciences Research Council (EPSRC), UK. Afterward, he worked as a postdoctoral researcher from December 2010 to October 2013 at the Institute of Biomechanics, Hamburg University of Technology, Hamburg, Germany, and then as a senior research associate from November 2013 to February 2016 at the Institute for in silico Medicine, the University of Sheffield, Sheffield, UK. At the end of 2015, he started the Faculty position at Dalian University of Technology, China and worked until now.

Dr. Lyu’s research interests lie primarily on the mechanical behaviors of biomaterials, the computational modeling of human body systems, the bionic design of bone implants, the machine learning-based numerical predictions, etc. He has developed the clinical image-based finite element model of bone tissues, which has increased the prediction power of bone fracture risk from the present 50% to approximately 90%. The next stage will be the translation of the technique into clinical usage. Additionally, he has developed the constitutive model for human skeletal muscle, which is able to simulate the active, viscoelastic and fiber-reinforced hyperelastic behaviors of skeletal muscle. He has published over 50 SCI-indexed journal papers. Most of them are published in the top-level journals in the field of biomechanics, including, the *Journal of Biomechanics*, the *Journal of the Mechanical Behavior of Biomedical Materials*, *Annals of Biomedical Engineering*, etc. He has received many awards, including the “Young Talent” prize from the Ministry of Human Resources and Social Security of Dalian, Liaoning Province, China, the “Qian Lingxi Youth Teacher” prize, the Teaching award from Dalian University of Technology, China, etc.

# Abbreviations

1D	One-dimensional
2D	Two-dimensional
3D	Three-dimensional
CAD	Computer-aided design
CFD	Computational fluid dynamics
FDM	Finite difference method
FE	Finite element
FEA	Finite element analysis
FEM	Finite element method
FEPG	Finite element program generator
HBW	Half band width
HFSS	High-frequency simulation software
INTESIM	INTEGRATED Engineering Simulation Platform
LSTC	Livermore Software Technology Corp
MDI	Mechanical Dynamics Inc
SiPESC	Software Integration Platform for Engineering and Scientific Computation

# Symbols

$\sigma_{ij}$	Stress tensor
$\sigma_{ij,i}$	Partial derivative of the stress tensor
$u$	Displacement in the $x$ direction
$v$	Displacement in the $y$ direction
$w$	The displacement in the $z$ direction
$\bar{u}$	The prescribed displacements in the $x$ direction
$\bar{v}$	The prescribed displacements in the $y$ direction
$\bar{p}_x, \bar{p}_y$	External forces
$S_p$	Force boundary
$G$	Shear modulus
$\varepsilon$	Normal strain
$\gamma$	Shear strain
$\sigma$	Normal stress
$\tau$	Shear stress
<b>D</b>	Elasticity matrix
$\Omega$	Solution domain
$\Gamma$	The boundary of the solution domain $\Omega$
$T$	Temperature
$k$	Heat transfer coefficient
$\bar{T}$	Prescribed temperature
$\bar{q}$	Prescribed influx heat
$n$	Outer normal direction of the relevant boundary
$Q$	Density of the heat source
$N_i$	Shape function or interpolation function
$\delta$	Dirac delta function
<b>R</b>	Residual for the differential equation
$\Pi$	Functional
$\sigma$	Stress tensor
$\varepsilon$	Strain tensor
$\bar{\mathbf{f}}$	Body force vector
$\bar{\mathbf{T}}$	Area force vector

$\delta\mathbf{u}$	Variational of the displacement
$\mathbf{u}$	Displacement vector
$\mathbf{S}$	Compliance matrix
$\Pi_c(\boldsymbol{\sigma})$	Complementary energy
$\mathbf{N}(x)$	Shape function matrix
$\mathbf{q}^e$	Nodal displacement vector
$\mathbf{B}(x)$	Geometric function matrix
$\mathbf{S}(x)$	Stress-strain matrix
$U^e$	Elemental internal energy
$W^e$	Elemental energy caused by the external forces
$R_1$	External force at the first node
$k_{ii}$	Component of the stiffness matrix
$\bar{\mathbf{q}}^e$	Nodal displacement vector in the global coordinate system
$\mathbf{T}^e$	Elemental coordinate transformation matrix
$\bar{\mathbf{K}}^e$	Elemental stiffness matrix in the global coordinate system
$\bar{\mathbf{P}}^e$	Nodal force vector in the global coordinate system
$l, h$ and $b$	Length, height and width of the beam
$M$	Moment
$Q$	Shear force
$A$ and $l$	Cross-sectional area and length of the beam
$dS$	The length of the line on the neural layer after the deformation
$R$	The radius of the circle formed by the neutral layer
$E$	Elastic modulus
$u$	Axial displacement of the beam
$v$	Deflection of the beam
$\theta$	Rotation of the beam
$\mathbf{q}^e$	Elemental nodal displacement vector
$\mathbf{P}^e$	Elemental nodal force vector
$P_{v1}, M_1$	Forces and moments at the first node
$\mathbf{N}(\xi)$	Elemental shape function
$\mathbf{N}'(\xi)$	First derivative of the elemental shape function
$\mathbf{B}(\xi)$	Strain-displacement matrix of the element
$\mathbf{S}(\xi)$	Stress-displacement matrix of the element
$W^e$	Elemental external work
$U^e$	Elemental strain (internal) energy
$\mathbf{K}^e$	Elemental stiffness matrix
$I_Z$	Moment of inertia
$u_o$	Axial displacement along the beam
$v_o$	Deflection of the beam at $O$
$\theta_o$	Rotation of the beam at $O$
$\mathbf{N}^T$	Shape function matrix
$R_A$ and $R_B$	Reaction forces at nodes A and B
$M_A$ and $M_B$	Reaction moments at nodes A and B
$\bar{p}(x)$	External loading
$\mathbf{K}$	Global stiffness matrix

$\xi$	Normalized coordinates in the beam
$R_{x3}$	Reaction force along the $x$ direction
$R_{y3}$	Reaction force along the $y$ direction
$R_{\theta 3}$	Reaction moment at the node
$\bar{\mathbf{q}}_a$	Displacement vector at node $a$
$n$	Total number of the degree of freedoms
$\mathbf{q}^e$	Nodal displacement vector
$u$	Displacement in the $x$ directions
$v$	Displacement in the $y$ directions
$A$	Area of the triangular element
$\mathbf{B}$	Strain-displacement matrix
$\nu$	Poisson's ratio
$\mathbf{S}$	Stress-displacement matrix
$\mathbf{K}^e$	Stiffness matrix of the element
$d_i$	HBW for each element
$\omega$	Number of nodal degree of freedom
$n$	Number of triangular elements
$\mathbf{f}$	Body force
$\mathbf{T}$	Surface force
$\mathbf{P}_f^e$	Equivalent nodal forces for the body force
$\mathbf{P}_S^e$	Equivalent nodal forces for the surface force
$\mathbf{P}^e$	Nodal force vector
$a$	Half of the length of the rectangular element
$b$	Half of the height of the rectangular element
$u$	Displacement in the $x$ direction
$v$	Displacement in the $y$ direction
$[\partial]$	Operator matrix of the strain-displacement equation
$\gamma_{xy}$	Shear strain
$t$	Thickness of the triangular element
$a$ and $b$	Dimension of the rectangular element
$m$	Number of elemental nodes
$\mathbf{J}$	Jacobian matrix
$i$	Unit vectors in the $x$ direction
$j$	Unit vectors in the $y$ direction
$n$	Number of integration
$A_k$	Weighted coefficients of the integration
$\xi_i$	Position of the integration points
$l_i^{n-1}(\xi)$	Lagrange interpolation function
$\delta_{ij}$	Kronecker symbol
$h$	Distance between the integration points
$A_i$	Integration weighting factor
$\beta_i$	Coefficient
$P(\xi)$	Polynomial function
$\mathbf{q}$	Nodal displacement vector
$\mathbf{P}$	Nodal force vector

<b>u</b>	Displacement field
<b>ε</b>	Strain field
<b>σ</b>	Stress field
<i>U</i>	Strain energy
<i>W</i>	External work
$\Pi$	Potential energy
<b>R</b>	Reaction force vector
$\Pi_e$	System complemental energy
<i>u</i>	Displacement in the <i>x</i> direction
<i>v</i>	Displacement in the <i>y</i> direction
<i>w</i>	Displacement in the <i>z</i> direction
$N_i$	Shape function
<i>V</i>	Volume of the tetrahedron
<b>B</b>	Strain-displacement vector
<i>E</i>	Young's modulus
<b>b̄</b>	Body force vector in the solution domain
<b>p̄</b>	Surface force vector in the boundary region
$V_i$	Volume of the sub-tetrahedron
<i>l</i>	Length of the element
$\xi$	Dimensionless coordinate
<i>n</i>	Number of elemental nodes
<b>N</b>	Shape function matrix
<i>A</i>	Area of the triangular
$A_i$	Area of the sub-triangular
<i>h</i>	Height of the element
$l_i^{(r)}(\xi)$	Interpolation function defined in the <i>x</i> direction
$l_j^{(p)}(\eta)$	Interpolation function defined in the <i>y</i> direction
$l_k^{(m)}(\zeta)$	Interpolation function defined in the <i>z</i> direction
<i>w</i>	Width of the element

# List of Figures

Fig. 1.1	Finite element discretization of a 2D structure .....	2
Fig. 1.2	The stress strain curve of a ductile material .....	5
Fig. 1.3	A representative 2D unit under the balanced forces .....	8
Fig. 1.4	The diagram for deriving the strain formulation .....	10
Fig. 1.5	Illustration of the force boundary condition .....	14
Fig. 1.6	Illustration of the plane stress problem .....	16
Fig. 1.7	a The 3D problem. b The simplified 2D plain strain problem ...	18
Fig. 2.1	Illustration of the solution domain and the boundary .....	20
Fig. 2.2	Comparison of the solutions obtained from the collocation method and the analytic method .....	28
Fig. 2.3	Comparison of the solutions obtained from the least square method and the analytic method .....	31
Fig. 2.4	Comparison of the solutions obtained from the Galerkin method and the analytic method .....	34
Fig. 3.1	A staircase 2-bar structure .....	46
Fig. 3.2	The two elements used to discretize the structure .....	46
Fig. 3.3	One-dimensional bar element .....	46
Fig. 3.4	Shape functions of bar element .....	52
Fig. 3.5	Physical meaning of the diagonal components in elemental stiffness matrix .....	54
Fig. 3.6	Physical meaning of the non-diagonal components in the elemental stiffness matrix .....	54
Fig. 3.7	Coordinate transformation for the bar element in the 2D plane .....	57
Fig. 3.8	The truss structure formed by four bars .....	59
Fig. 4.1	A long beam in the equilibrium state .....	66
Fig. 4.2	A representative cross-section of the beam element .....	66
Fig. 4.3	Cross section of the long beam under the distributed loading ...	68
Fig. 4.4	A representative general beam element .....	69
Fig. 4.5	The beam element with no axial displacement .....	70
Fig. 4.6	The calculation of the equivalent nodal forces .....	76

Fig. 4.7	Illustration of the calculation of the equivalent nodal forces . . . . .	78
Fig. 4.8	A beam structure under the external pressure . . . . .	80
Fig. 4.9	Calculation of the equivalent nodal forces for the second element . . . . .	80
Fig. 4.10	Illustration of the coordinate transformation in the beam element . . . . .	83
Fig. 4.11	A 3-beam frame structure under the uniform pressure . . . . .	84
Fig. 4.12	The equivalent nodal forces on the first element . . . . .	85
Fig. 5.1	Illustration of a triangular element . . . . .	94
Fig. 5.2	Plotting of the three shape functions for triangular element . . . . .	99
Fig. 5.3	Analysis of the property of shape function on the edge of the triangular . . . . .	100
Fig. 5.4	Analysis of the displacement on the shared edge . . . . .	102
Fig. 5.5	Illustration of the area coordinate in the triangular element . . . . .	103
Fig. 5.6	Assembly of the global stiffness matrix using the elemental matrices . . . . .	106
Fig. 5.7	Assembling the elemental stiffness matrix to the global stiffness matrix . . . . .	107
Fig. 5.8	Reduction of the dimension of the global stiffness matrix . . . . .	109
Fig. 5.9	The half band widths for different numbering of the same structure . . . . .	109
Fig. 5.10	A body force on the triangular element . . . . .	111
Fig. 5.11	An external uniform pressure on one edge of the triangular element . . . . .	112
Fig. 5.12	A horizontal uniform pressure on one edge of the triangular element . . . . .	113
Fig. 5.13	A triangular pressure on one edge of the triangular element . . . . .	114
Fig. 5.14	A 2D plane structure . . . . .	115
Fig. 5.15	The 2D plane structure discretized using two triangular elements . . . . .	115
Fig. 6.1	The plane 4-node rectangular element . . . . .	120
Fig. 6.2	The distribution of the <b>a</b> normal strain $y\varepsilon_y$ and the <b>b</b> shear strain $x\gamma_{xy}$ within the rectangular element . . . . .	124
Fig. 6.3	Distribution of the shape function within the rectangular element . . . . .	129
Fig. 6.4	Illustration of the edge shared by two rectangular elements . . . . .	130
Fig. 6.5	The iso-parametric mapping between the reference and physical coordinates . . . . .	133
Fig. 6.6	The illustration of the differential area . . . . .	133
Fig. 6.7	The transformable and non-transformable iso-parametric elements . . . . .	139
Fig. 6.8	The calculation of the integration using the Newton–Cotes numerical method . . . . .	143
Fig. 6.9	The plane rectangular structure . . . . .	148

Fig. 6.10	Distribution of the displacement, the strain and the stress field formulations in the structure solved using one rectangular element .....	149
Fig. 6.11	Distribution of the displacement, the strain and the stress field formulations in the structure solved using two triangular elements .....	150
Fig. 6.12	The distribution of the displacement, the strain and the stress field formulations in the scenario of the second type of boundary condition solved using one rectangular element .....	151
Fig. 6.13	The distribution of the displacement, the strain and the stress field formulations in the scenario of the second type of boundary condition solved using two triangular elements .....	152
Fig. 7.1	The 4-node tetrahedral element .....	160
Fig. 7.2	The Pascal's pyramid .....	160
Fig. 7.3	Illustration of the definition of the volume coordinate .....	165
Fig. 7.4	The 8-node hexahedral element .....	166
Fig. 8.1	The 3-node second order bar element .....	173
Fig. 8.2	The 4-node third order bar element .....	174
Fig. 8.3	The 6-node second order triangular element .....	176
Fig. 8.4	The Pascal's triangle .....	177
Fig. 8.5	The area coordinates defined in the 6-node second order triangular element .....	178
Fig. 8.6	The 10-node third order triangular element .....	180
Fig. 8.7	Illustration of the high order rectangular element .....	182
Fig. 8.8	The 9-node second order rectangular element .....	183
Fig. 8.9	The 16-node third order rectangular element .....	184
Fig. 8.10	Illustration of the inner nodes using the Pascal's triangle .....	185
Fig. 8.11	The 10-node second order tetrahedral element .....	185
Fig. 8.12	The Pascal's pyramid .....	188
Fig. 8.13	The 20-node third order tetrahedral element .....	189
Fig. 8.14	Illustration of the high order hexahedral element .....	192
Fig. 8.15	The 27-node second order hexahedral element .....	193

# List of Tables

Table 3.1	Nodes and their coordinates .....	59
Table 3.2	Elements and their associated nodes .....	60
Table 3.3	The length of the elements and their cosine values .....	60
Table 4.1	Values of shape functions at elemental nodes .....	72
Table 4.2	The equivalent nodal forces for the beams under different external loadings .....	79
Table 4.3	Numbering for the element and node .....	85
Table 5.1	The coordinates and values of the constants for the first element .....	116

# Chapter 1

## Introduction



### 1.1 Introduction

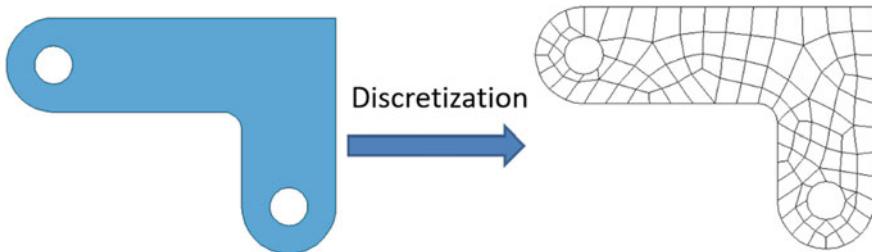
The finite element (FE) method is a widely used method for numerically solving differential equations arising in engineering and mathematical modeling. Typical engineering problems include the structural analysis, the heat transfer, the fluid analysis, etc. In this book, the FE analysis for dealing with the static structural analysis is introduced.

### 1.2 Introduction to the Finite Element Method

#### 1.2.1 Brief History of the Finite Element Method

The finite element method (FEM) is a general numerical approach for solving partial differential equations. To solve a general engineering problem using the FEM, the continuum system is firstly discretized into finite elements, which are connected by nodes, and then a collection of discrete elements is used to represent the original continuum system (Fig. 1.1). In the FE model, the element is defined as the minimum cell/part after discretization and the node is defined as the connection point between elements. It should be noted that in the FE analysis, the forces are passing through the nodes, not the elemental edges, which is different from that in the real engineering structure. Therefore, in the FE calculations, if the forces are applied on the edge, the surface or the body of the elements, the equivalent nodal forces need to be worked out, the details of which will be introduced in this book.

The history of the FEM can be briefly summarized as follows: in 1943, Courant first tried the idea of using several triangular to solve a continuum deformable body. In 1956, in the annual meeting of aerospace in New York, Clough et al. proposed a new idea which is the beginning of the finite element method, i.e., partitioning the structure into triangular and rectangular, forming the global stiffness matrix and then solving



**Fig. 1.1** Finite element discretization of a 2D structure

the system. In 1960, the word “finite element” was first proposed in the book “The finite element method in plane stress analysis”. In 1968, some mathematicians recognized the advantages of the finite element method and further developed the method to a new depth. In 1970s, the first nonlinear FE software—MARC (MSC.Software, Los Angeles, U.S.) was released in the market. In 1975, the nonlinear solver—ADINA (Automatic Dynamic Incremental Nonlinear Analysis) (ADINA R&D Inc., Massachusetts, U.S.) emerged in the market. In 1977, the Mechanical Dynamics Inc. (MDI) released the software ADAMS (MSC.Software, Los Angeles, U.S.), which is mainly used in the mechanical kinematics analysis, the dynamics simulation, etc. In 1978, the major FE software – Abaqus (Dassault SIMULIA, Rhode Island, U.S.) was released in the market, which still serves as one of the main software in the market nowadays. In 1988, Livermore Software Technology Corp. (LSTC) released the software—LS-DYNA (LSTC Inc., California, U.S.) which is mainly used for solving the nonlinear dynamical problem. In the twenty-first century, another major software—ANSYS(ANSYS Inc., Pennsylvania, U.S.), including the ANSYS Mechanical, the ANSYS Computational Fluid Dynamics (CFD), the ANSYS Workbench, etc., was released, which is a major competitor of Abaqus nowadays. From the history of the development of the FEM, it can be seen that the FEM originates and develops in US, which is partially due to the reason that the third industrial revolution occurred in US and consequently the computer programs and relevant software were rapidly developing in U.S. In China, there is also a time period when the national FE software develops very fast, i.e., during the period from 1950 to 1980s. Unfortunately, since the foreign software entered the China market, they start to dominate the market and leave no room/market for the continuous development of the national FE software. Nowadays, due to the barriers among different countries, the national FE software starts to develop again for the national security.

### ***1.2.2 Introduction to the Commonly Used Finite Element Software***

There are many commercial FE software available in the market. Below a brief introduction of some widely used software is given.

The Abaqus is a software suite for finite element analysis and computer-aided engineering, originally released in 1978. The Abaqus product suite consists of five core software products: (1) Abaqus/CAE is a software application used for both the modeling and analysis of mechanical components and visualizing the finite element analysis results; (2) Abaqus/Standard is a general-purpose FE analyzer that employs the implicit integration scheme; (3) Abaqus/Explicit is a special-purpose FE analyzer that employs explicit integration scheme to solve highly nonlinear systems with many complex contacts; (4) Abaqus/CFD is a Computational Fluid Dynamics software application which provides advanced computational fluid dynamics capabilities and (5) Abaqus/Electromagnetic is a computational electromagnetics software application solving advanced computational electromagnetic problems.

ANSYS is another widely-used software which offers a comprehensive software suite that spans the entire range of physics, providing access to virtually any field of engineering simulation that a design process requires. The main products include the Ansys Mechanical/Structural FEA Analysis Software, the Ansys Fluent/Fluid Simulation Software, the Ansys Sherlock/Electronics Reliability Prediction Software, the Ansys HFSS/3D High Frequency Simulation Software, etc. In this book, some element types using the Ansys Mechanical module will be introduced.

LS-DYNA is an advanced general-purpose Multiphysics simulation software package developed by LSTC. Its core competency lies in the highly nonlinear transient dynamics FE analysis using explicit time integration, although more and more possibilities of the analysis of many complex real world problems. LS-DYNA is widely used in the fields of automobile, aerospace, military, manufacturing and bioengineering.

MARC is a powerful, general-purpose, nonlinear finite element analysis solution to accurately simulate the product behavior under static, dynamic and multi-physics loading scenarios. MARC is ideal for the robust nonlinear solutions, having the capabilities to elegantly simulate all kinds of nonlinearities including geometric, material and boundary condition nonlinearity. It also possesses the ability to predict damage, failure and crack propagation.

It should be noted that the FE software listed above is strong at solving the linear/nonlinear FE equations. Regarding the FE pre-processing, i.e., the discretization (meshing) of the deformable body, the professional pre-processing software is required, among which the HyperMesh (Altair Inc., Michigan, U.S.) is widely used. HyperMesh is the market-leading, multi-disciplinary finite element pre-processor managing the generation of the largest and most complex models, starting with the import of Computer-aided Design (CAD) geometry to exporting a ready-to-run solver file. Regarding the post-processing of the result data, it should be noted that sometimes, the FE software listed above cannot meet the need and consequently some

professional post-processing software has been developed. For example, LS-PrePost is an advanced post-processor delivered free with LS-DYNA. In LS-PrePost, functions such as fast rendering, particle visualization, section analysis, etc. can be efficiently achieved. Additionally, the Python and Matlab can be used to post-process the result data obtained from the FE software.

Regarding the national FE software in China, due to the national security, some national FE software was developed and used only in the national defense field, for example, the FEPG (Finite Element Program Generator) and the PANDA developed by the China Academy of Engineering Physics. Regarding the general-purpose software, there are also some available in the market. For example, the SIMRIGHT developed in Shanghai, the INTESIM (Integrated Engineering Simulation Platform) developed by a group in Dalian, the SiPESC (Software Integration Platform for Engineering and Scientific Computation) developed by the department of Engineering Mechanics, Dalian University of Technology.

## 1.3 Some Basic Knowledge from the Theory of Elasticity

### 1.3.1 The Four Basic Assumptions

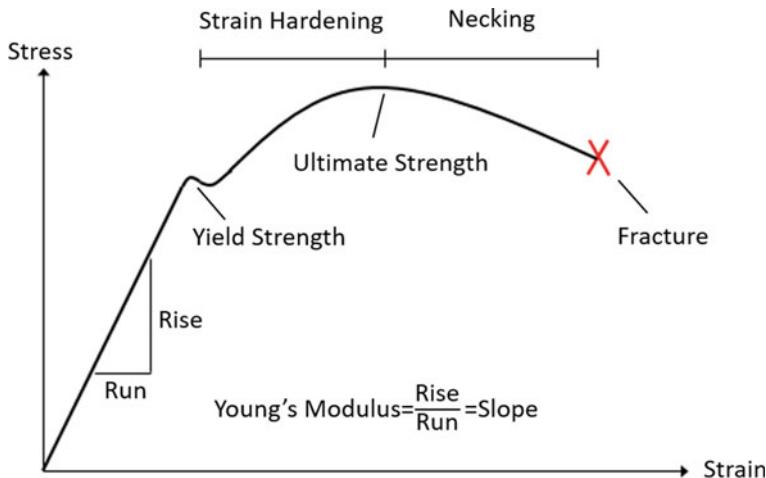
The FE calculation is a complex process, always involving the nonlinear and large deformation problems, which have to be solved using the professional computational tool, such as Matlab, Fortran. In this book, in order to simplify the calculations and make it feasible to be calculated by hand, the following four basic assumptions are made.

#### (1) Continuum

The assumption of *continuum* is made in the material perspective. The continuum assumption means the matter in the body is continuously distributed and fills the entire region of the space it occupies. The advantage of this assumption is that because of this assumption, the physical measures such as the strain, stress, etc. are continuous, and thus the continuous functions can be used to express them. It should be noted that this assumption is always applied in the macro scale. In the micro scale level, the matter is always discrete, in which scenario, the finite element method cannot be used and the molecular dynamics method should be used instead.

#### (2) Linear elasticity

The assumption of *linear elasticity* is made in the perspective of the mechanical behavior of the materials. The mechanical behaviors of the materials can be classified into the linear and nonlinear ones (Fig. 1.2). For the nonlinear materials, the stress is nonlinearly changing with the strain since the start of the stretch/compression. For the linear materials, the stress is linearly changing with the strain at the first stage of the stretch/compression and goes to the nonlinear stage when the material yield point



**Fig. 1.2** The stress–strain curve of a ductile material

is reached. In this lecture, only the simplest case, i.e., the linear stage in the linear material, is dealt with. Therefore, it is assumed that the constitutive relationship of the material is linear elastic. Because of this assumption, the elastic parameters of the materials are constants which enables us to easily characterize the mechanical property of the material using the Young's modulus, the shear modulus, the elasticity matrix, etc.

### (3) Isotropy and homogeneity

The assumption of *isotropy and homogeneity* is also made in the perspective of the mechanical behavior of materials. Depend on the way the deformable body is formed, the material can be classified into the isotropic, the anisotropic, the homogeneous, the heterogeneous, etc. If the deformable body is made of one material and there are the same mechanical behaviors in any part and any direction of the object, the material is isotropic and homogeneous. It should be noted that this is the simplest case and in this lecture, only the simplest case is dealt with. The advantage of this assumption is that any part of the deformable body can be taken as the representative volume of the analysis. However, it should be noted that the mechanical behavior of the material can be much more complex. For example, if the deformable body is made of several different materials, the material exhibits the heterogeneous behavior. If the isotropic base material is reinforced with parallel fibers, the material exhibits the transverse isotropic behavior. However, the formulations, especially the constitutive relationship and the finite element formulations, for these materials are out of the scope of this lecture.

#### (4) Small deformation

The assumption of *small deformation* is made in the perspective of the mechanical behavior of materials. The fourth assumption made in this lecture is the small deformation, i.e., the displacement and deformation in the deformable body are much less than its dimension. The advantage of this assumption is that the dimension change of the deformable body caused by the deformation can be ignored, and consequently the high order parts in the Taylor series equation can be ignored and subsequent equations can be simplified as the linear elastic equations. It should be noted that the finite element formulations, especially the geometric equations (strain–displacement equation) presented in this lecture are based on the assumption of small deformation. It should be noted that in the scenario of large deformation, the strain–displacement formulations for the deformable body are completely different and consequently the FE formulations will be different and this is also out of the scope of the present book.

It should be noted again that the four assumptions are made to simplify the calculations involved in the present book. In the real engineering problem, the four assumptions, especially the second, the third and the fourth assumptions are no longer satisfied, which implies the nonlinear, anisotropic behaviors of the materials and large deformation, etc. should be considered in the FE calculations.

#### 1.3.2 Some Preliminary Knowledge on Tensor Operation

Tensor notation and tensor operation are important preliminary knowledge in this book, because many formulae are expressed in the tensor format and consequently tensor operations are involved. The reason why the tensor notation is used in the formula is that they are concise and much easier for the writing, reading and calculation, i.e., the tensor notation enables the efficient representation of all variables and the governing equations using a single standardized method. It should be noted that the tensor notation is also called the *index notation*, which is defined as a shorthand scheme whereby a whole set of numbers or components can be represented by a single symbol with subscripts.

Using the tensor notation, all the variables can be called a tensor with different degrees of order. Depend on the degree of order in the variables, the tensor can be called the 0th order tensor, the first order tensor, the second order tensor, etc. The 0th order tensor is the variable with no free index, which is also called the *scalar*. Examples for the 0th order tensor include the strain energy, the body weight, height, etc. The first order tensor is the variable with one free index, which is also called the *vector*. Examples for the first order tensor include the displacement vector, the direction vector, etc. The second order tensor is the variable with two free indexes, e.g., the stress and strain tensors. From these definitions, it is seen that the third order tensor is the variable with three free indices, the forth order tensor is the variable with four free indexes, and so on. In the definitions above, the free index is defined as the index appeared in the subscript of the variable, e.g., in the definition of the stress

tensor  $\sigma_{ij}$ ,  $i$  and  $j$  are two free indexes. In addition to the free index, there is another important definition, the dumb index, which is the subscript appearing twice in the same term. For example, in the equation,  $a_{ij}x_j = b_i$ , the index  $j$  appeared twice in the first term and thus  $j$  is the dumb index. It should be noted that the dumb index means the summation over the range of the dumb index, e.g.

$$a_{ii} = \sum_{i=1}^3 a_{ii} = a_{11} + a_{22} + a_{33} (i = 1, 2, 3) \quad (1.1a)$$

$$a_{ij}b_j = \sum_{j=1}^3 a_{ij}b_j = a_{i1}b_1 + a_{i2}b_2 + a_{i3}b_3 (i, j = 1, 2, 3) \quad (1.1b)$$

Using the definition of the free and dumb indexes, the writing of a set of equations can be simplified, e.g.,

$$\begin{aligned} a_{11}x_1 + a_{12}x_2 + a_{13}x_3 &= b_1 \\ a_{21}x_1 + a_{22}x_2 + a_{23}x_3 &= b_2 \\ a_{31}x_1 + a_{32}x_2 + a_{33}x_3 &= b_3 \end{aligned} \quad (1.2)$$

The equations above can be rewritten out as

$$\sum_{j=1}^3 a_{ij}x_j = b_i (i = 1, 2, 3) \quad (1.3)$$

In the equation above, the summation symbol is always ignored. Then the equation can be further written out as

$$a_{ij}x_j = b_i (i = 1, 2, 3) \quad (1.4)$$

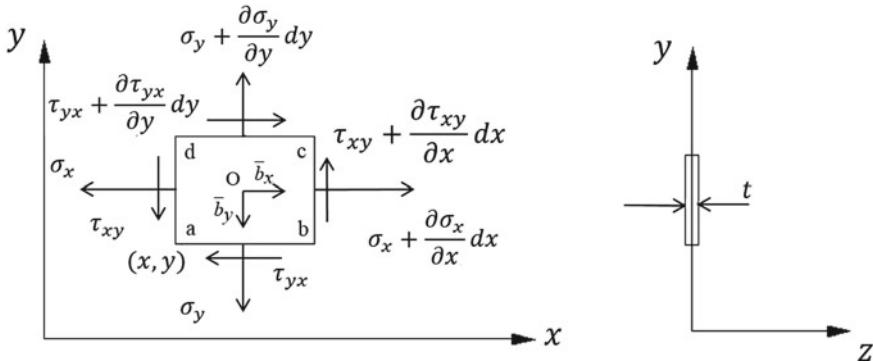
It can be seen the equations have been simplified as a simple equation, which is very concise.

### 1.3.3 Three Main Equations in the Theory of Elasticity

Below a brief review of the three main equations in the theory of elasticity is given, because they are widely used in the FE calculations.

#### (1) The equilibrium equation

The *equilibrium equation* is used to set up the relationship between the internal stress and the external forces. In the book, the equilibrium equations are set for the static problems. The equilibrium equation is based on the assumption that any point in the



**Fig. 1.3** A representative 2D unit under the balanced forces

deformable body needs to fulfill the equilibrium equation in the balanced system. The following two-dimensional (2D) plane problem (Fig. 1.3) is taken as the example to illustrate the equilibrium equation:

The equilibrium conditions of the balanced 2D plane problem shown above are: (a) the summation of all the forces in the  $x$  direction equals to zero, (b) the summation of all the forces in the  $y$  direction equals to zero and (c) the summation of all the moments along any point equals to zero.

First, the summation of all the forces in the  $x$  direction equals to zero, i.e.,  $\sum F_x = 0$ .

The equation can be expanded as:

$$\left(\sigma_x + \frac{\partial\sigma_x}{\partial x}dx\right)dy \cdot t - \sigma_x dy \cdot t + \left(\tau_{yx} + \frac{\partial\tau_{yx}}{\partial y}dy\right)dx \cdot t - \tau_{yx} dx \cdot t + \bar{b}_x dxdy \cdot t = 0 \quad (1.5)$$

After the mathematical operation, the equation can be rewritten out as:

$$\frac{\partial\sigma_x}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \bar{b}_x = 0 \quad (1.6)$$

Similarly, in the  $y$  direction,  $\sum F_y = 0$

$$\frac{\partial\sigma_y}{\partial y} + \frac{\partial\tau_{xy}}{\partial x} + \bar{b}_y = 0 \quad (1.7)$$

Third, the summation of all the moments along any point equals to zero. The center point (O) is chosen as the rotating center and then the sum of all the moments about O should equal to zero, i.e.,  $\sum M_o = 0$

$$\left(\tau_{xy} + \frac{\partial\tau_{xy}}{\partial x}dx\right)dy \cdot t \frac{dx}{2} + \tau_{xy} dy \cdot t \frac{dx}{2} - \left(\tau_{yx} + \frac{\partial\tau_{yx}}{\partial y}dy\right)dx \cdot t \frac{dy}{2} - \tau_{yx} dx \cdot t \frac{dy}{2} = 0 \quad (1.8)$$

After the mathematical operation, the equation above can be simplified as:

$$\tau_{xy} = \tau_{yx} \quad (1.9)$$

In summary, the equilibrium equations for the given 2D problem can be written out as:

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \bar{b}_x &= 0 \\ \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \bar{b}_y &= 0 \\ \tau_{xy} &= \tau_{yx} \end{aligned} \quad (1.10)$$

In case of the three-dimensional (3D) problem, the equilibrium equations can be written out as:

$$\left\{ \begin{array}{l} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + b_x = 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + b_y = 0 \\ \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + b_z = 0 \\ \tau_{xy} = \tau_{yx}, \tau_{xz} = \tau_{zx}, \tau_{zy} = \tau_{yz} \end{array} \right. \quad (1.11)$$

In the notation format, the first three equations can be written concisely as:

$$\sigma_{ij,i} + b_j = 0, (i, j = x, y, z) \quad (1.12)$$

where the dot in  $\sigma_{ij,i}$  indicates the operation of partial derivative.

## (2) The strain–displacement equation

The *strain–displacement equation* is to describe the relation between the strain and the displacement. Because no physical parameters (e.g., the elastic modulus) are involved, the strain–displacement equation is also called the *geometric equation*. Now let's use the 2D case below to illustrate the strain–displacement relation.

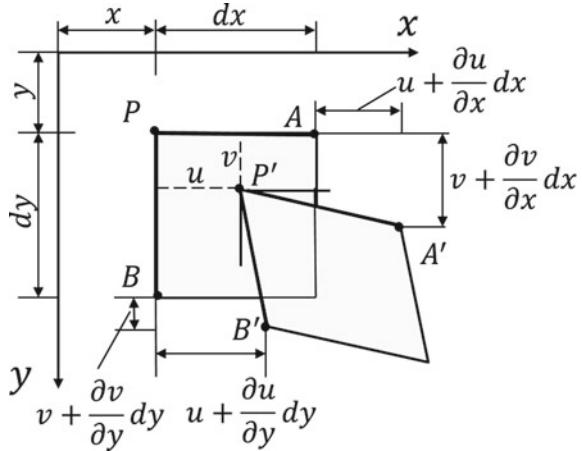
As shown in Fig. 1.4, a deformable body  $PAB$  is deformed to the new position  $P'A'B'$ . It should be noted that the deformation is considered to be small. The coordinate at  $P$  is  $(x, y)$ , the dimension of  $PA$  is  $dx$  and the dimension of  $PB$  is  $dy$ . The symbols  $u$  and  $v$  are used to represent the displacements in the  $x$  and  $y$  directions. Therefore, the displacements in the  $x$  and  $y$  directions at  $P$  can be written as:

$$\left\{ \begin{array}{l} u = u(x, y, z) \\ v = v(x, y, z) \end{array} \right. \quad (1.13)$$

where  $u$  and  $v$  are the displacements in the  $x$  and  $y$  directions, respectively.

It can be seen that the displacements at any point within the deformable body are a function of the coordinate. Therefore, the displacements at points  $A$  and  $B$  can be written as:

**Fig. 1.4** The diagram for deriving the strain formulation



$$A : u = u(x + dx, y, z), v = v(x + dx, y, z) \quad (1.14a)$$

$$B : u = u(x, y + dy, z), v = v(x, y + dy, z) \quad (1.14b)$$

Using the Taylor expansion series, the following equation can be obtained

$$u(x + dx, y, z) = u(x, y, z) + \frac{du}{dx} dx + \frac{d^2u}{dx^2} (dx)^2 + \dots \quad (1.15)$$

Because the deformation from  $PAB$  to  $P' A' B'$  is small, the higher order terms in the equation can be ignored. Therefore, the displacements at points  $A$  and  $B$  can be re-written as:

$$A : u = u(x, y, z) + \frac{du}{dx} dx, v = v(x, y, z) + \frac{dv}{dx} dx \quad (1.16a)$$

$$B : u = u(x, y, z) + \frac{du}{dy} dy, v = v(x, y, z) + \frac{dv}{dy} dy \quad (1.16b)$$

After the representation of the displacements at points  $P$ ,  $A$  and  $B$ , the strain components can be obtained using the definition of strain, i.e., the strain is the value using the change in the dimension divided by the undeformed (initial) dimension. Therefore, the strain along the  $x$  direction can be calculated as the change in the length of  $PA$  divided by the initial length of  $PA$ , i.e.:

$$\varepsilon_x = \frac{(u + \frac{\partial u}{\partial x} dx + dx - u) - dx}{dx} = \frac{\frac{\partial u}{\partial x} dx}{dx} = \frac{\partial u}{\partial x} \quad (1.17)$$

Similarly, the strain along the  $y$  direction can be calculated as the change in the length of  $PB$  divided by the initial length of  $PB$ , i.e.:

$$\varepsilon_y = \frac{\left(v + \frac{\partial v}{\partial y} dy + dy - v\right) - dy}{dy} = \frac{\frac{\partial v}{\partial y} dy}{dy} = \frac{\partial v}{\partial y} \quad (1.18)$$

The shear strain at point  $P$  can be expressed as the angle change between  $PA$  and  $PB$ . The angle between  $PA$  and  $P' A'$  is denoted as alpha ( $\alpha$ ), which can be expressed as:

$$\tan \alpha = \frac{\left(v + \frac{\partial v}{\partial x} dx\right) - v}{dx} = \frac{\partial v}{\partial x} \quad (1.19)$$

Because it is a small deformation, the following relation can be obtained

$$\alpha \approx \partial v / \partial x \quad (1.20)$$

Similarly, the angle between  $PB$  and  $P' B'$  is denoted as beta ( $\beta$ ) and can be calculated as:

$$\tan \beta = \frac{\left(u + \frac{\partial u}{\partial y} dy\right) - u}{dy} = \frac{\partial u}{\partial y} \approx \beta \quad (1.21)$$

Afterwards, the change in the angle between  $PA$  and  $PB$  after the deformation can be expressed as:

$$\gamma_{xy} = \alpha + \beta = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad (1.22)$$

In summary, the strain–displacement relation for the 2D problem can be written as:

$$\begin{aligned} \varepsilon_x &= \frac{\partial u}{\partial x} \\ \varepsilon_y &= \frac{\partial v}{\partial y} \\ \gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{aligned} \quad (1.23)$$

The equations can be concisely written in format of the index notation as below:

$$\varepsilon_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i})(i, j = x, y) \quad (1.24)$$

The strain–displacement relation for the 3D case can be written as:

$$\varepsilon_x = \frac{\partial u}{\partial x}, \varepsilon_y = \frac{\partial v}{\partial y}, \varepsilon_z = \frac{\partial w}{\partial z} \quad (1.25a)$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}, \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}, \gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \quad (1.25b)$$

and in terms of the index notation:

$$\varepsilon_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \quad (i, j = x, y, z) \quad (1.26)$$

### (3) The stress-strain equation

The *stress-strain relation* is also called the constitutive relation or the physical relation, because the physical variables, e.g., the Young's modulus, are involved in the relation. In this book, because of the assumption of the isotropic, linear elastic and homogeneous behavior of the materials, two variables, i.e., the elastic modulus and the Poisson's ratio, are sufficient to describe the mechanical behavior of materials. Based on the theory of elasticity, the stress-strain relation can be written as:

$$\begin{aligned} \varepsilon_x &= \frac{1}{E}[\sigma_x - \mu(\sigma_y + \sigma_z)] \\ \varepsilon_y &= \frac{1}{E}[\sigma_y - \mu(\sigma_x + \sigma_z)] \\ \varepsilon_z &= \frac{1}{E}[\sigma_z - \mu(\sigma_x + \sigma_y)] \end{aligned} \quad (1.27a)$$

$$\begin{aligned} \gamma_{xy} &= \frac{1}{G}\tau_{xy} \\ \gamma_{yz} &= \frac{1}{G}\tau_{yz} \\ \gamma_{zx} &= \frac{1}{G}\tau_{zx} \end{aligned} \quad (1.27b)$$

where  $E$  is the elastic modulus,  $\mu$  is Poisson's ratio and  $G$  is the shear modulus. The stress-strain relation can also be written as below:

$$\begin{aligned} \sigma_x &= \frac{E(1-\mu)}{(1+\mu)(1-2\mu)} \left( \varepsilon_x + \frac{\mu}{1-\mu} \varepsilon_y + \frac{\mu}{1-\mu} \varepsilon_z \right) \\ \sigma_y &= \frac{E(1-\mu)}{(1+\mu)(1-2\mu)} \left( \varepsilon_y + \frac{\mu}{1-\mu} \varepsilon_x + \frac{\mu}{1-\mu} \varepsilon_z \right) \\ \sigma_z &= \frac{E(1-\mu)}{(1+\mu)(1-2\mu)} \left( \varepsilon_z + \frac{\mu}{1-\mu} \varepsilon_y + \frac{\mu}{1-\mu} \varepsilon_x \right) \end{aligned} \quad (1.28a)$$

$$\begin{aligned}\tau_{xy} &= \frac{E}{2(1+\mu)}\gamma_{xy} \\ \tau_{yz} &= \frac{E}{2(1+\mu)}\gamma_{yz} \\ \tau_{zx} &= \frac{E}{2(1+\mu)}\gamma_{zx}\end{aligned}\tag{1.28b}$$

where  $E$  is the elastic modulus and  $\mu$  is Poisson's ratio.

The stress-strain relation can be written in the matrix format

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\epsilon}\tag{1.29}$$

where  $\boldsymbol{\sigma} = [\sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yz} \tau_{zx}]^T$ ,  $\boldsymbol{\epsilon} = [\varepsilon_x \varepsilon_y \varepsilon_z \gamma_{xy} \gamma_{yz} \gamma_{zx}]^T$  and  $\mathbf{D}$  is the elasticity matrix given as below:

$$\mathbf{D} = \frac{E(1-\mu)}{(1+\mu)(1-2\mu)} \begin{bmatrix} 1 & \frac{\mu}{1-\mu} & \frac{\mu}{1-\mu} & 0 & 0 & 0 \\ \frac{\mu}{1-\mu} & 1 & \frac{\mu}{1-\mu} & 0 & 0 & 0 \\ \frac{\mu}{1-\mu} & \frac{\mu}{1-\mu} & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2(1-\mu)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\mu}{2(1-\mu)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\mu}{2(1-\mu)} \end{bmatrix}\tag{1.30a}$$

The elasticity matrix can also be written as:

$$\mathbf{D} = \begin{bmatrix} \lambda + 2G & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda + 2G & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda + 2G & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix}\tag{1.30b}$$

where  $\lambda = \frac{E(1-\mu)}{(1+\mu)(1-2\mu)}$ ,  $G = \frac{E}{2(1+\mu)}$ .

From the calculations above, it can be seen that the equilibrium equation has used the assumption of continuum, the strain-displacement relation has used the assumption of small deformation, and the stress-strain relation has used the assumption of linear elasticity, isotropy, homogeneity and small deformation.

### 1.3.4 Types of Boundary Conditions

The boundary condition is an important part in the FE modeling. Without the boundary conditions, the deformable body cannot be deformed. The boundary condition in the FE models can be classified into two types, i.e., the displacement boundary condition and the force boundary condition, which are explained as below:

#### (1) The displacement boundary condition

The *displacement boundary condition* refers to the one where the displacements in some regions of the deformable body are given. The prescribed displacement boundary condition can be zero or a non-zero value. If the zero displacement is defined, the region is fully fixed or constrained. If the given values are prescribed, the displacement boundary conditions on the boundary (2D problem) can be expressed as:

$$u = \bar{u}, v = \bar{v} \quad (1.31)$$

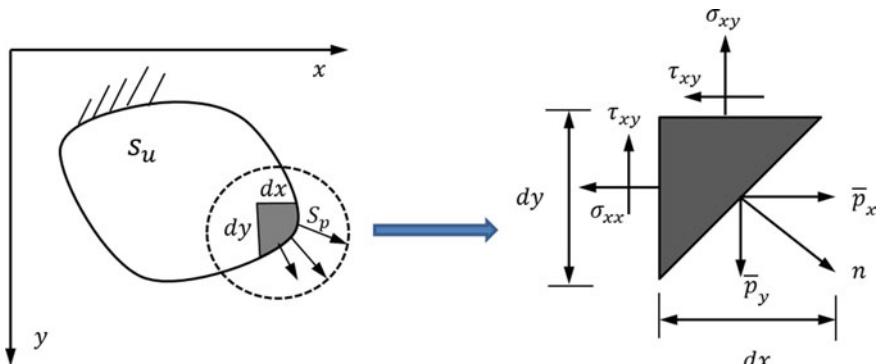
where  $\bar{u}$  and  $\bar{v}$  are the prescribed displacements in the  $x$  and  $y$  directions, respectively.

In the FE calculations, the displacement boundary conditions can be taken into account by introducing the equations above into the FE formulations.

#### (2) The force boundary condition

The *force boundary condition* refers to the one where the forces in some regions of the deformable body are given (Fig. 1.5). In Fig. 1.5, an external force  $(\bar{p}_x, \bar{p}_y)$  is applied on the boundary  $S_p$ . Because the system is static and balanced, according to the equilibrium condition, first, the sum of all the forces in the  $x$  direction equals to zero, i.e.,  $\sum F_x = 0$ , from which the following equation can be obtained

$$-\sigma_x dy \cdot t - \tau_{xy} dx \cdot t + \bar{p}_x ds \cdot t = 0 \quad (1.32)$$



**Fig. 1.5** Illustration of the force boundary condition

After the mathematical operation, the equation can be simplified as

$$\sigma_x n_x + \tau_{xy} n_y = \bar{p}_x \quad (1.33)$$

where  $n_x = dy/ds$ ,  $n_y = dx/ds$ .

Similarly, the sum of all the forces in the y direction equals to zero, i.e.,  $\sum F_y = 0$ , from which the following equation can be obtained

$$\sigma_y n_y + \tau_{yx} n_x = \bar{p}_y \quad (1.34)$$

In summary, for the 2D problem, the force boundary conditions on  $S_p$  can be written as:

$$\sigma_x n_x + \tau_{xy} n_y = \bar{p}_x \quad (1.35a)$$

$$\sigma_y n_y + \tau_{yx} n_x = \bar{p}_y \quad (1.35b)$$

In terms of the index notation, the force boundary condition on  $S_p$  can be written as

$$\sigma_{ij} n_j = \bar{p}_i \quad (i, j = x, y) \quad (1.36)$$

In the FE analysis, to take into account the force boundary conditions, the equations given above should be introduced into the FE formulations.

### 1.3.5 The Plane Stress and Plane Strain Problems

At the end of this chapter, the 2D plane stress and plane strain problems are introduced. The reason they are introduced is that sometimes, a complex 3D problem can be simplified as a 2D problem, and then the subsequent calculations can be largely simplified. Therefore, the types of the plane problems need to be clarified before knowing under which conditions the 3D problems can be simplified to the corresponding 2D problems. In this part, the simplification conditions and also the features of the two types of plane problems are given.

### (1) The plane stress problem

The *plane stress* problem is defined as the scenario when the stress only occurs in a plane and the stress component perpendicular to this plane is zero. Under the following conditions, the 3D problem can be simplified as a plane stress problem:

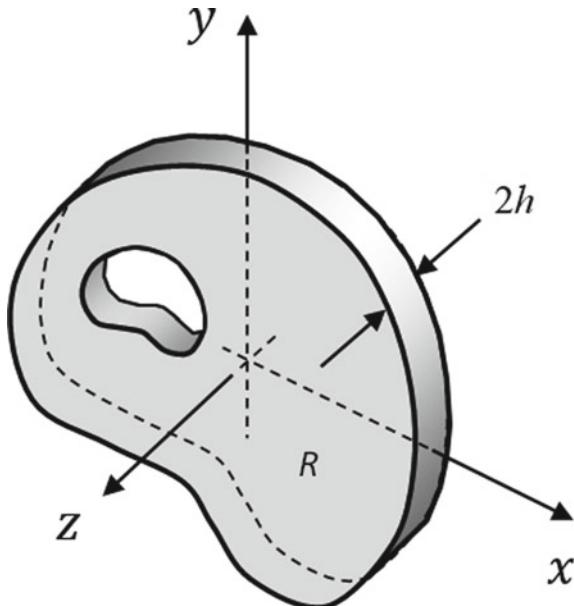
- (a) A very thin deformable body;
- (b) The forces are parallel to the plane of the deformable body;
- (c) The forces are evenly distributed along the thickness direction of the deformable body.

When the three conditions above are met, the 3D problem can be simplified as a 2D problem and consequently the simplified formulations can be used to solve the original 3D problem. It should be noted that the first condition is given from the perspective of the dimension of the deformable body. The second and the third conditions are given from the perspective of the external force, i.e., the second refers to the direction of the force and the third refers to the distribution pattern of the force. The three conditions are the necessary conditions of the simplification, which means if any one of them is not satisfied, the 3D problem cannot be simplified as a 2D problem.

Once the three conditions are satisfied, the thickness in the thin deformable body can be ignored and the problem can be solved using the following simplified equations. First, only three of the six stress components are zeros, i.e.,  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$  in Fig. 1.6.

In summary, for the 2D plane stress problem, the stress can be simplified as:

**Fig. 1.6** Illustration of the plane stress problem



$$\left\{ \begin{array}{l} \sigma_x = \sigma_x(x, y) \\ \sigma_y = \sigma_y(x, y) \\ \tau_{xy} = \tau_{xy}(x, y) \\ \sigma_z = \tau_{xz} = \tau_{yz} = 0 \end{array} \right. \quad (1.37)$$

Now let's have a look whether the strain and displacement variables can be simplified. Unfortunately, the strain and displacement components cannot be simplified and it should be noted that the normal strain and the displacement in the  $z$  direction are not necessarily zeros, i.e.,

$$w \neq 0, \varepsilon_z \neq 0$$

where  $w$  is the displacement in the  $z$  direction.

## (2) The plane strain problem

The *plane strain* problem is defined as the scenario when the strain only occurs in a plane and the strain component perpendicular to this plane is zero. Under the following conditions, the 3D problem can be simplified as a plane strain problem:

- (a) A very long deformable body;
- (b) The forces are parallel to the cross-section of the deformable body;
- (c) The forces are evenly distributed along the length direction of the deformable body.

Similar to the description in the plane stress problem, these three conditions are the necessary conditions of simplifying the 3D problem into the 2D problem. Furthermore, the first condition is given from the perspective of the dimension for the deformable body, the second is given from the perspective of the force direction and the third is given from the perspective of the force distribution. Once the three conditions are satisfied, a cross-section of the 3D deformable body, parallel to the external force, can be taken and used to represent the original 3D problem.

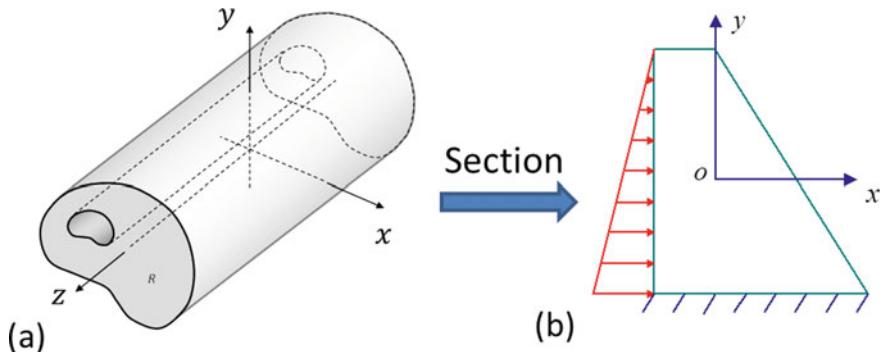
Using the 2D cross-section of the original 3D body, the following simplified formulations can be set up and used to solve the problem. First, the strain components associated with the  $z$  coordinate are zeros (Fig. 1.7), i.e.,

$$\varepsilon_z = 0, \varepsilon_{xz} = 0, \varepsilon_{yz} = 0$$

This is the reason the simplified 2D problem is called the plane strain problem. Second, the displacement in the  $z$  direction is zero, i.e.,

$$w = 0$$

The original 3D problem can be used to understand this condition, i.e., the original 3D body is a very long deformable body and consequently it is impossible to deform the body in the indefinitely long direction. Third, the two shear strain components are zeros, i.e.,  $\varepsilon_{xz} = 0, \varepsilon_{yz} = 0$ . According to the stress-strain relation, i.e.,



**Fig. 1.7** **a** The 3D problem. **b** The simplified 2D plain strain problem

$$\tau_{xz} = 2G\varepsilon_{xz}, \tau_{yz} = 2G\varepsilon_{yz} \quad (1.38)$$

where \$G\$ is the shear modulus, we can obtain that the shear stresses related to the \$z\$ direction are zeros, i.e.,

$$\tau_{xz} = 0, \tau_{yz} = 0$$

It should be noted that the stress-strain relation in Eq. (1.28a, b) is based on the assumptions of small deformation, linear elastic, etc. Therefore, the third formulation, i.e., Eq. (1.38) is also based on these assumptions. At the end of this part, it should be noted that in the plane strain formulations, the normal stress perpendicular to the cross-section, i.e., the stress in the \$z\$ direction, is not zero, i.e., \$\sigma\_z \neq 0\$ in Fig. 1.7b.

## Chapter 2

# Theoretical Basis of the Finite Element Method



### 2.1 Introduction

In the field of engineering and technology science, the mathematical models and equations can be given for many mechanics problems. However, the scenarios where the analytic equations and the exact solutions can be found are occasional and normally only in the problems with simple geometries. For most engineering problems, because of the complicated geometries and the nonlinearity of the equations, only numerical solutions can be obtained. Compared to the analytic method, the computational process in the numerical method is much more complicated and time consuming. However, this is not an issue anymore since the rapid development in modern computers. Consequently, the numerical analysis method has become one of the main methods in scientific computations.

The numerical methods to solve the partial differential equations can be classified into two groups. The finite difference method (FDM) is the representative method in the first group. In this group of methods, the basic differential equations together with the boundary conditions are directly solved. The idea of the FDM is to mesh the solution domain using elements and then use the difference equations to approximate the differential equations. When more elements and nodes are used, the accuracy of the approximated solution can be improved. The FDM is suitable for the fluid mechanics, where the coordinate system is fixed in the spatial space. However, it is not suitable for the solid mechanics, where the coordinate system is fixed on the deformable body. In this case, the finite element method (FEM), which represents the other group of methods, needs to be used.

Different from the FDM, the FEM does not solve the differential equations directly, but starts with the equivalent integral form of the differential equations. The general form of the equivalent integral form is the weighted residual method. Different from the traditional weighted residual method, in which the approximation function is assumed in the entire solution domain, the approximation function is assumed and given in each individual element in the FEM. By doing this, the difficulties of defining the approximation function in the entire solution domain

can be resolved and some important breakthroughs have been made in engineering computations in the past a few years. In this chapter, the theoretical basis of the FEM, including the equivalent integral form of the differential equation, the weighted residual method, the variational principle method, etc. are introduced.

## 2.2 Equivalent Integral Form of the Differential Equation

The general engineering or physical problems are always proposed using the differential equations and boundary conditions as a set of the unknown field functions (Fig. 2.1). The general differential equations for these problems can be expressed using the unknown field functions as below:

$$\mathbf{A}(\mathbf{u}) = \begin{Bmatrix} A_1(u) \\ A_2(u) \\ \vdots \end{Bmatrix} = 0 \text{ in } \Omega \quad (2.1)$$

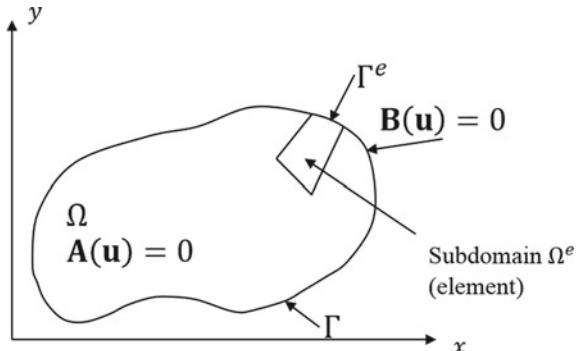
where the solution domain  $\Omega$  can be a volumetric domain, an area domain, etc. Meanwhile, the unknown field function  $\mathbf{u}$  should satisfy the boundary conditions, which are given as below

$$\mathbf{B}(\mathbf{u}) = \begin{Bmatrix} B_1(u) \\ B_2(u) \\ \vdots \end{Bmatrix} = 0 \text{ on } \Gamma \quad (2.2)$$

where  $\Gamma$  is the boundary of the solution domain  $\Omega$ .

The unknown function to be solved, i.e.,  $\mathbf{u}$ , can be a scalar field, e.g., the temperature, or the vector field formed by several variables, e.g., the displacement, the strain, the stress, etc.  $\mathbf{A}$  and  $\mathbf{B}$  represent the differential operators with respect to

**Fig. 2.1** Illustration of the solution domain and the boundary



the independent variables, e.g., the spatial coordinate, the time coordinate, etc. The number of differential equations agrees with the number of unknown field functions. Therefore, the differential equation can be a single equation or a series of equations, and consequently the matrix format is used in Eqs. (2.1) and (2.2).

Because the differential Eqs. (2.1) are satisfied at every point within the solution domain  $\Omega$ , and thus we have the following integration equation.

$$\int_{\Omega} \mathbf{v}^T \mathbf{A}(\mathbf{u}) d\Omega = \int_{\Omega} (v_1 A_1(\mathbf{u}) + v_2 A_2(\mathbf{u}) + \dots) d\Omega = 0 \quad (2.3)$$

where  $\mathbf{v}$  is the vector containing a series of arbitrary functions which have the same number as the number of the differential equations, i.e.,

$$\mathbf{v} = [v_1 v_2 \dots]^T$$

Equation (2.3) is the equivalent integral form of the differential Eqs. (2.1). Equation (2.3) can be interpreted as follows: if the integration Eq. (2.3) is satisfied for any  $\mathbf{v}$ , then the differential Eqs. (2.1) can be satisfied at any point within the field  $\Omega$ . This inference is obvious, because if the differential equation  $\mathbf{A}(\mathbf{u})$  is not satisfied at some points within the field  $\Omega$ , i.e.,  $\mathbf{A}(\mathbf{u}) \neq 0$ , then we can find an appropriate function  $\mathbf{v}$  which makes the integral equation not equal to zero.

Similarly, if the boundary conditions, i.e., Eq. (2.2), are satisfied at all the points on the boundary of the deformable body, then the following equation should be satisfied for a series of arbitrary functions  $\bar{\mathbf{v}}$ :

$$\int_{\Gamma} \bar{\mathbf{v}}^T \mathbf{B}(\mathbf{u}) d\Gamma = \int_{\Gamma} (\bar{v}_1 B_1(\mathbf{u}) + \bar{v}_2 B_2(\mathbf{u}) + \dots) d\Gamma = 0 \quad (2.4)$$

The following equation can be obtained by combining together Eqs. (2.3) and (2.4):

$$\int_{\Omega} \mathbf{v}^T \mathbf{A}(\mathbf{u}) d\Omega + \int_{\Gamma} \bar{\mathbf{v}}^T \mathbf{B}(\mathbf{u}) d\Gamma = 0 \quad (2.5)$$

Equation (2.5) is satisfied for any  $\mathbf{v}$  and  $\bar{\mathbf{v}}$ , implying that the differential Eq. (2.1) and the boundary conditions (Eq. 2.2) are satisfied. Equation (2.5) is called the equivalent integral form of the differential equations.

An example is used below to further demonstrate the calculation process of the equivalent integral form of the differential equations.

**Example 2.1:** The equations for a 2D steady heat transfer problem are given as:

$$\mathbf{A}(T) = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + Q = 0 \text{ (in the solution domain } \Omega\text{)}$$

$$B(T) = \begin{cases} T - \bar{T} = 0 & (\text{on the boundary } \Gamma_T) \\ k \frac{\partial T}{\partial n} - \bar{q} = 0 & (\text{on the boundary } \Gamma_q) \end{cases}$$

where  $T$  is the temperature,  $k$  is the heat transfer coefficient,  $\bar{T}$  is the temperature given on the boundary  $\Gamma_T$ ,  $\bar{q}$  is the influx heat given on the boundary  $\Gamma_q$ ,  $n$  is the outer normal direction of the relevant boundary and  $Q$  is the density of the heat source. The question is to work out the equivalent integral form of the differential equations.

**Solution:** Following the procedure introduced, the equivalent integral form of the problem can be written as below:

$$\int_{\Omega} v^T \left( \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + Q \right) d\Omega + \int_{\Gamma} \bar{v}^T (T - \bar{T}) d\Gamma + \int_{\Gamma} \bar{v}^T \left( k \frac{\partial T}{\partial n} - \bar{q} \right) d\Gamma = 0$$

It should be noted that in the problems above, if  $k$  and  $Q$  are only the functions of the spatial coordinates, then the problem is a linear one. If  $k$  and  $Q$  are also the functions of the temperature  $T$ , then the problem becomes a nonlinear one.

## 2.3 The Weighted Residual Method

In the solution domain  $\Omega$ , if the field function is an accurate solution, then the differential Eq. (2.1) is exactly met at all the points within the field and the boundary conditions (Eq. 2.2) are exactly satisfied at all the points on the boundary. Consequently, the equivalent integral form (Eq. 2.5) will be strictly satisfied. However, in the complex engineering problems, the exact solutions can be hardly reached and instead the approximate solutions with acceptable errors have to be worked out.

The weighted residual method assumes the unknown field function can be represented using the approximation function for the physical problem expressed using the differential equations and the boundary conditions. The approximation function is a function formed by a series of coefficients to be determined, which can be written as

$$\mathbf{u} \approx \bar{\mathbf{u}} = \sum_{i=1}^n N_i a_i = \mathbf{Na} \quad (2.6)$$

where  $a_i$  are the coefficients to be determined,  $N_i$  are the shape functions or the interpolation functions. It should be noted that the equation above is only for the 1D problem. In the 3D problem, in which there are 3 unknown field functions, the approximate solutions can be written out as:

$$\begin{aligned}
 u &= N_1 u_1 + N_2 u_2 + \dots + N_n u_n = \sum_{i=1}^n N_i u_i \\
 v &= N_1 v_1 + N_2 v_2 + \dots + N_n v_n = \sum_{i=1}^n N_i v_i \\
 w &= N_1 w_1 + N_2 w_2 + \dots + N_n w_n = \sum_{i=1}^n N_i w_i
 \end{aligned} \tag{2.7}$$

where  $u$ ,  $v$  and  $w$  are the displacement field functions in the  $x$ ,  $y$  and  $z$  directions, respectively.

In the 3D case, the coefficient matrix can be written out as

$$\mathbf{a}_i = [u_i \ v_i \ w_i]^T \tag{2.8}$$

where  $u_i$ ,  $v_i$  and  $w_i$  are the coefficients to be determined, the numbers of which are  $3 \times n$ .  $\mathbf{N}_i = \mathbf{I}N_i$  are the interpolation functions, where  $\mathbf{I}$  is the  $3 \times 3$  unit matrix.

It is obvious that the differential equations and the boundary conditions cannot be exactly satisfied if the approximate solution is taken as the solution. There exists a residual  $\mathbf{R}$  for the differential equations and a residual  $\bar{\mathbf{R}}$  for the boundary conditions, i.e.,

$$\mathbf{A}(\mathbf{Na}) = \mathbf{R}, \quad \mathbf{B}(\mathbf{Na}) = \bar{\mathbf{R}} \tag{2.9}$$

The finite weighting function is used to replace the arbitrary function, i.e.,

$$\mathbf{v} = \mathbf{W}_j, \quad \bar{\mathbf{v}} = \bar{\mathbf{W}}_j \quad (j = 1, 2, \dots, n) \tag{2.10}$$

Then the corresponding equivalent integral form can be obtained:

$$\int_{\Omega} \mathbf{W}_j^T \mathbf{A}(\mathbf{Na}) d\Omega + \int_{\Gamma} \bar{\mathbf{W}}_j^T \mathbf{B}(\mathbf{Na}) d\Gamma = 0 \tag{2.11}$$

which can also be written in terms of residuals

$$\int_{\Omega} \mathbf{W}_j^T \mathbf{R} d\Omega + \int_{\Gamma} \bar{\mathbf{W}}_j^T \bar{\mathbf{R}} d\Gamma = 0 \tag{2.12}$$

Equation (2.12) implies that the sum of the weighted integrals of the residuals equals to zero, from which a series of equations can be set up. Using these equations, the coefficients can be determined and then the approximate solution for the original differential equations can be obtained. It should be noted that by solving Eq. (2.12) the residual of the differential equations and the residual of the boundary conditions

can be forced to close to zero if appropriate values for the undetermined coefficients  $a_i$  are chosen.

In the calculation procedure above, the method of exploring the approximate solution for the differential equations using the weighted integrals of the residuals is called the *weighted residual method*. The weighted residual method is an efficient method to find the approximate solution for the differential equations.

Below an example is used to demonstrate the calculation of the differential equation using the weighted residual method.

**Example 2.2:** To solve the second order differential equation given as below:

$$\frac{d^2u}{dx^2} + x^2 = 0 \quad (0 \leq x \leq 1)$$

The boundary conditions are given as below:

$$\begin{cases} u(0) = 0 \\ u(1) = 0 \end{cases}$$

**Solution:** To solve this problem using the weighted residual method, first, the approximate solution, denoted also as the trial function, is chosen as below:

$$\tilde{u} = x(1-x)(a_1 + a_2x + \dots)$$

where  $a_1, a_2, \dots$  are the coefficients to be determined,  $N_1 = x(1-x)$ ,  $N_2 = x(1-x)x, \dots$  are the weighting functions. It is obvious that the trial functions satisfy the boundary conditions but do not satisfy the differential equation and a residual will be present in the field. Because the boundary conditions are satisfied, we don't need to deal with them in the subsequent calculations. For this specific example, Eq. (2.12) takes the following form:

$$\int_0^1 \mathbf{W}_i^T \mathbf{R} dx = 0 \quad (i = 1, 2, \dots, n)$$

The meaning of the equation above is that the weighted integral of the residual equals to zero in the physical domain. The number of terms in the trial function can be one, two, three or even more. It should be noted that the more the terms are, the higher the accuracy is, but the calculation complexity is accordingly increased when more terms are involved. In this chapter, for the purpose of demonstrating the weighted residual method, the cases of using one and two terms in the trial function are discussed.

First, only one term is used in the trial function, i.e.,  $n = 1$ :

$$\tilde{u}_1 = a_1 x(1-x)$$

Putting it into the original differential equation, the residual can be obtained as:

$$R_1 = \frac{d^2\tilde{u}_1}{dx^2} + x^2 = -2a_1 + x^2$$

Second, two terms are used in the trial function, i.e.,  $n = 2$ :

$$\tilde{u}_2 = x(1-x)(a_1 + a_2x) = a_1x(1-x) + a_2x(1-x)x$$

Similarly, putting it into the original differential equation, the residual can be obtained as:

$$R_2 = \frac{d^2\tilde{u}_2}{dx^2} + x^2 = -2a_1 + 2a_2 - 6a_2x + x^2$$

In Eq. (2.12), any independent function can be selected as the weighting function. Based on the form of the weighting function, different weighted residual methods can be obtained. The commonly used weighted residual methods are the collocation method, the least squares method and the Galerkin method. As mentioned before, they are differed by the form of the weighting function. Below some examples are used to illustrate these methods.

### (1) The collocation method

In the collocation method, the weighting function is chosen as:

$$\mathbf{W}_i = \delta(\mathbf{x} - \mathbf{x}_i) (i = 1, 2, \dots, n) \quad (2.13)$$

where  $\delta$  is the Dirac delta function and has the following properties:

$$\delta(x - x_i) = \begin{cases} +\infty, & (x - x_i) = 0 \\ 0, & (x - x_i) \neq 0 \end{cases} \quad (2.14)$$

$$\int_{-\infty}^{+\infty} \delta(x - x_i) dx = 1 \quad (2.15)$$

Furthermore,

$$\int_{-\infty}^{+\infty} f(x - x_i) \delta(x - x_i) dx = f(x_i) \quad (2.16)$$

Therefore, the following integral equation can be obtained

$$\int_{\Omega} \mathbf{W}_i^T \mathbf{R} d\Omega = R(x_i) = 0 \quad (i = 1, 2, \dots, n) \quad (2.17)$$

In the collocation method, the residual is forced to be zero at the collocation points. Therefore, the following  $n$  equations to solve the  $n$  unknown coefficients are obtained.

$$\begin{cases} R_n(x_1) = 0 \\ R_n(x_2) = 0 \\ \vdots \\ R_n(x_n) = 0 \end{cases} \quad (2.18)$$

**Example 2.3:** To solve the following differential equation using the collocation method

$$\frac{d^2u}{dx^2} + x^2 = 0 \quad (0 \leq x \leq 1)$$

The boundary conditions are given as below

$$\begin{cases} u(0) = 0 \\ u(1) = 0 \end{cases}$$

**Solution:** Following the calculation procedure for the collocation method, first, the approximate solution is chosen as below

$$\tilde{u} = x(1-x)(a_1 + a_2x + \dots)$$

The integral of the residual equals to zero, i.e.,

$$\int_0^1 \mathbf{W}_i^T \mathbf{R} dx = 0 \quad (i = 1, 2, \dots, n)$$

Below the calculation processes using the approximate solutions with one and two terms are demonstrated.

First, one term in the trial function is used, i.e.,

$$\tilde{u} = a_1 x(1-x)$$

The middle point, i.e.,  $x = 1/2$ , is chosen as the collocation point.

Putting the trial function into the differential equation and the residual can be obtained as:

$$R_1\left(\frac{1}{2}\right) = \frac{1}{4} - 2a_1$$

Forcing the residual to be close to zero, the following equation can be obtained

$$\frac{1}{4} - 2a_1 = 0$$

Then the unknown coefficient can be obtained as  $a_1 = \frac{1}{8}$ .

The approximate solution for the original differential equation can be written as:

$$\tilde{u}_1 = \frac{1}{8}x(1-x)$$

Second, using two terms in the trial function, i.e., two unknown coefficients,

$$\tilde{u} = x(1-x)(a_1 + a_2x)$$

Choosing  $x = 1/3$  and  $x = 2/3$  as the collocation points, the following equations can be obtained:

$$\begin{cases} R_2\left(\frac{1}{3}\right) = \frac{1}{9} - 2a_1 = 0 \\ R_2\left(\frac{2}{3}\right) = \frac{4}{9} - 2a_1 - 2a_2 = 0 \end{cases}$$

Solving the equations above, the answers using two coefficients can be obtained:

$$\begin{cases} a_1 = \frac{1}{18} \\ a_2 = \frac{1}{6} \end{cases}$$

Then the approximate solution for the original differential equation is obtained as

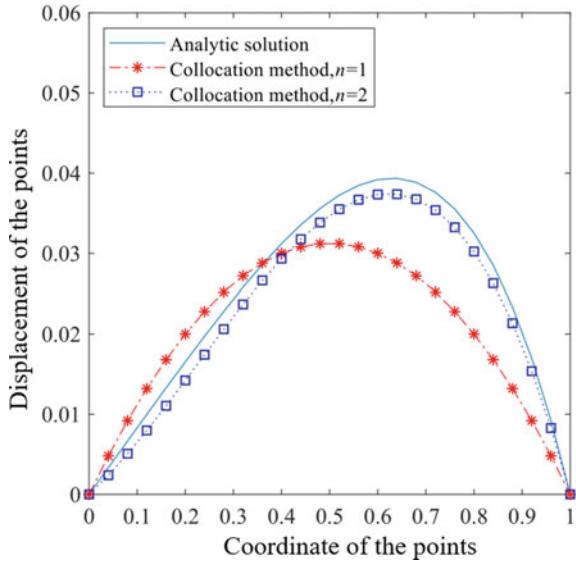
$$\tilde{u}_2 = \frac{1}{18}(-3x^3 + 2x^2 + x)$$

Below the solutions obtained from different methods are drawn in one figure to make the comparisons (Fig. 2.2). It is seen that, the solution using one collocation point is a bit far away from the analytic solution, which is taken as the ground truth. When two collocation points are used, the corresponding solution becomes very close to the analytic answer, implying the solution using two collocation points possesses a much higher accuracy compared to the solution using only one collocation point.

## (2) The least squares method

In the least squares method, the following approximation function is used:

**Fig. 2.2** Comparison of the solutions obtained from the collocation method and the analytic method



$$\tilde{\mathbf{u}} = \sum_{i=1}^n N_i a_i \quad (2.19)$$

The weighting function is selected as:

$$\mathbf{W}_i = \frac{\partial \mathbf{R}}{\partial a_i} = \frac{\partial}{\partial a_i} \mathbf{A} \left( \sum_{i=1}^n N_i a_i \right) \quad (i = 1, 2, \dots, n) \quad (2.20)$$

Then,

$$\begin{aligned} \int_{\Omega} \mathbf{W}_i^T \mathbf{A}(\mathbf{N}\mathbf{a}) d\Omega &= \frac{1}{2} \frac{\partial}{\partial a_i} \int_{\Omega} \mathbf{A}^T \left( \sum_{i=1}^n N_i a_i \right) \mathbf{A} \left( \sum_{i=1}^n N_i a_i \right) d\Omega \\ &= \frac{1}{2} \frac{\partial}{\partial a_i} \int_{\Omega} R^2 d\Omega = \frac{\partial I}{\partial a_i} \quad (i = 1, 2, \dots, n) \end{aligned} \quad (2.21)$$

The least squares method is to determine the coefficients  $a_i$  to make the function  $I(a_i)$  get to the minimal, i.e.,

$$\frac{\partial I}{\partial a_i} = 0 \quad (i = 1, 2, \dots, n) \quad (2.22)$$

**Example 2.4:** To solve the following differential equation using the least squares method

$$\frac{d^2u}{dx^2} + x^2 = 0 \quad (0 \leq x \leq 1)$$

The boundary conditions are given as below:

$$\begin{cases} u(0) = 0 \\ u(1) = 0 \end{cases}$$

**Solution:** To solve this problem, first the following trial function is chosen as the approximate solution:

$$\tilde{u} = x(1-x)(a_1 + a_2x + \dots)$$

where  $a_1, a_2, \dots$  are the coefficients to be determined.

The integral of the residual equals to zero, i.e.,

$$\int_0^1 \mathbf{W}_i^T \mathbf{R} dx = 0 \quad (i = 1, 2, \dots, n)$$

Based on the equations above, the equations can be set up and the solutions for the coefficients can be obtained. Below, the calculation processes using the approximation functions with one and two terms are demonstrated.

First, only one term is used in the approximate solution, i.e., one unknown coefficient:

$$\tilde{u} = a_1 x(1-x)$$

Putting this equation into the differential equation, the residual can be obtained as below:

$$R_1 = -2a_1 + x^2$$

Then, the following relation can be obtained

$$I = \int_0^1 R_1^2 dx = 4a_1^2 - \frac{4}{3}a_1 + \frac{1}{5}$$

Getting the function  $I$  close to the minimal, i.e., the partial derivative of  $I$  with respect to the unknown coefficient  $a_1$  is zero, then we have the following equation:

$$\frac{\partial I}{\partial a_1} = 8a_1 - \frac{4}{3} = 0$$

The unknown coefficient can be obtained by solving the equation above:

$$a_1 = 1/6$$

Therefore, the approximate solution for the original differential equation can be obtained:

$$\tilde{u}_1 = \frac{1}{6}x(1-x)$$

Below the calculation process for the same differential equation using two terms in the trial function, i.e., two coefficients to be determined, is demonstrated. In this case, the trial function can be written as:

$$\tilde{u} = x(1-x)(a_1 + a_2x)$$

Putting this equation into the differential equation, the residual can be obtained as below:

$$R_2 = -2a_1 + 2a_2 - 6a_2x + x^2$$

Then, the following relation can be obtained

$$I = \int_0^1 R_2^2 dx = 4a_1^2 + 4a_1a_2 - \frac{4}{3}a_1 + 4a_2^2 - \frac{5}{3}a_2 + \frac{1}{5}$$

Getting the function  $I$  close to the minimal, i.e., the partial derivatives of  $I$  with respect to the unknown coefficients  $a_1$  and  $a_2$  are zero, then the following equations can be obtained:

$$\begin{cases} \frac{\partial I}{\partial a_1} = 8a_1 + 4a_2 - \frac{4}{3} = 0 \\ \frac{\partial I}{\partial a_2} = 4a_1 + 8a_2 - \frac{5}{3} = 0 \end{cases}$$

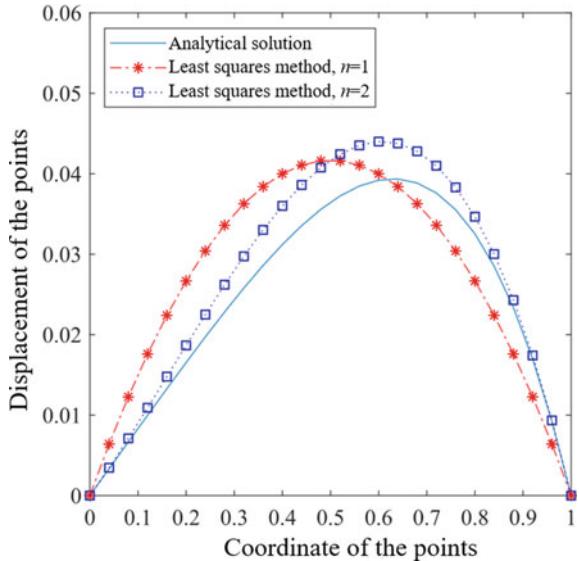
The answers for the unknown coefficients can be obtained by solving the equations above, i.e.,

$$\begin{cases} a_1 = \frac{1}{12} \\ a_2 = \frac{1}{6} \end{cases}$$

Therefore, the approximate solution for the original differential equation is:

$$\tilde{u}_2 = \frac{1}{12}(-2x^3 + x^2 + x)$$

**Fig. 2.3** Comparison of the solutions obtained from the least square method and the analytic method



Similarly, the answers obtained from different methods are plotted in one figure to make the comparisons (Fig. 2.3). It is clearly seen that the answer using two coefficients in the least squares method is closer to the analytic solution, implying a higher accuracy is present in the least squares method when more coefficients are used in the approximation function.

### (3) The Galerkin method

In the Galerkin method, the shape function is used as the weighting function, i.e.,

$$\mathbf{W}_j = \mathbf{N}_j \text{ (in the solution domain)} \quad (2.23)$$

and

$$\bar{\mathbf{W}}_j = -\mathbf{N}_j \text{ (on the boundary)} \quad (2.24)$$

Then the integral form takes the following form:

$$\int_{\Omega} \mathbf{W}_j^T \mathbf{R} d\Omega + \int_{\Gamma} \mathbf{W}_j^T \bar{\mathbf{R}} d\Gamma = 0 \quad (j = 1, 2, \dots, n) \quad (2.25)$$

It should be noted that the equation above is a series of  $n$  equations.

Putting Eq. (2.9) into the equations above, the integral form can be written as:

$$\int_{\Omega} \mathbf{N}_j^T \mathbf{A} \left( \sum_{i=1}^n N_i a_i \right) d\Omega - \int_{\Gamma} \mathbf{N}_j^T \mathbf{B} \left( \sum_{i=1}^n N_i a_i \right) d\Gamma = 0 \quad (j = 1, 2, \dots, n) \quad (2.26)$$

The variation of the approximate solution is as below:

$$\delta \tilde{\mathbf{u}} = \mathbf{N}_1 \delta a_1 + \mathbf{N}_2 \delta a_2 + \dots + \mathbf{N}_n \delta a_n \quad (2.27)$$

where  $\delta a_i$  is arbitrary. Therefore, the equation can be rewritten as:

$$\int_{\Omega} \delta \tilde{\mathbf{u}}^T \mathbf{A}(\tilde{\mathbf{u}}) d\Omega - \int_{\Gamma} \delta \tilde{\mathbf{u}}^T \mathbf{B}(\tilde{\mathbf{u}}) d\Gamma = 0 \quad (2.28)$$

The equation above is the basis for the Galerkin method. It should be noted that the Galerkin method is widely used to build the FE formulations. Below an example is used to demonstrate the calculation of the differential equation using the Galerkin method.

**Example 2.5:** To solve the following differential equation using the Galerkin method

$$\frac{d^2 u}{dx^2} + x^2 = 0 \quad (0 \leq x \leq 1)$$

The boundary conditions are given as below

$$\begin{cases} u(0) = 0 \\ u(1) = 0 \end{cases}$$

**Solution:** To solve the problem, first the following approximation function is used:

$$\tilde{u} = x(1-x)(a_1 + a_2 x + \dots)$$

The equations can be set up by making the integral of the residual equal to zero, i.e.,

$$\int_0^1 \mathbf{W}_i^T \mathbf{R} dx = 0 \quad (i = 1, 2, \dots, n)$$

For the demonstration purpose, only the calculation processes using one and two unknown coefficients are demonstrated below.

First, one term, i.e., one unknown coefficient, is used in the trial function, i.e.,

$$\tilde{u}_1 = a_1 x(1-x)$$

The weighting function is the shape function in the formula above, i.e.,

$$W_1 = N_1 = x(1 - x)$$

Putting the trial function into the differential equation, the residual can be obtained:

$$R_1 = -2a_1 + x^2$$

Using the weighting function and the residual, the following equation can be obtained

$$\int_0^1 N_1 R_1 dx = -\frac{1}{3}a_1 + \frac{1}{20}$$

Solving the equation above, the answer for the unknown coefficient can be obtained as

$$a_1 = \frac{3}{20}$$

Therefore, the solution using one term in the trial function is:

$$\tilde{u}_1 = \frac{3}{20}x(1 - x)$$

Now, let's demonstrate the calculation using two terms in the trial function. In this case, the approximate solution takes the following form:

$$\tilde{u}_2 = a_1x(1 - x) + a_2x(1 - x)x$$

The corresponding weighting functions are as following

$$W_1 = N_1 = x(1 - x)$$

$$W_2 = N_2 = x(1 - x)x$$

Putting the approximate solution into the differential equation, the residual can be obtained:

$$R_2 = -2a_1 + 2a_2 - 6a_2x + x^2$$

Then according to the Galerkin method, the following two equations can be obtained

$$\begin{cases} \int_0^1 N_1 R_2 dx = -\frac{1}{3}a_1 - \frac{1}{6}a_2 + \frac{1}{20} = 0 \\ \int_0^1 N_2 R_2 dx = -\frac{1}{6}a_1 - \frac{2}{15}a_2 + \frac{1}{20} = 0 \end{cases}$$

The answers for the unknown coefficients  $a_1$  and  $a_2$  can be obtained by solving the two equations above:

$$\begin{cases} a_1 = \frac{1}{15} \\ a_2 = \frac{1}{6} \end{cases}$$

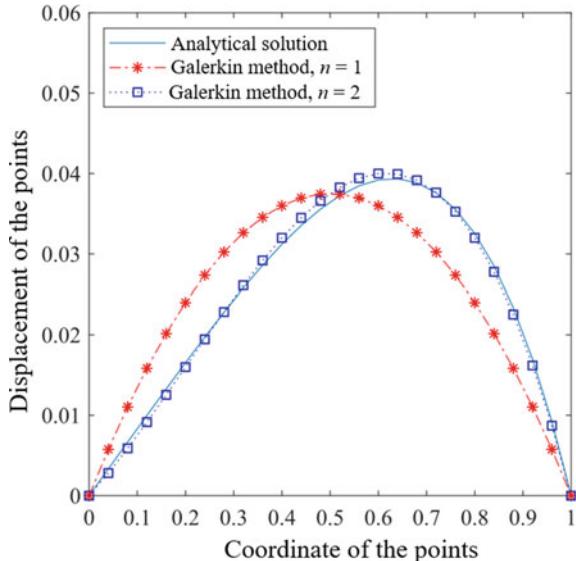
Therefore, the solution using two terms in the trial function is:

$$\tilde{u}_2 = \frac{1}{30}(-5x^3 + 3x^2 + 2)$$

Similarly, the answers obtained from different methods are plotted in one figure to make the comparisons. It is seen in Fig. 2.4 that the solution obtained from the Galerkin method using two terms in the trial function has a very high accuracy, i.e., the curve almost overlaps with that obtained from the analytic solution.

At the end of this part, some notes about the weighted residual method are discussed. First, the weighted residual method is applicable to various types of equations. Second, different weighted residual methods can be generated by selecting different weighting functions. In this chapter, the collocation method, the least squares method and the Galerkin method are presented. Overall, the accuracy of

**Fig. 2.4** Comparison of the solutions obtained from the Galerkin method and the analytic method



the Galerkin method is the best among the three methods. Third, if the approximation function is taken as the complete form, then it can satisfy the continuity requirements. Consequently, the approximate solution will closely approach to the exact solution if the number of terms in the trial function is largely increased.

A couple of shortcomings of the weighted residual method should also be noted. First, the convergence of the solution is not strictly proved theoretically. Secondly, the approximate solution does not have clear upper and lower bounds.

## 2.4 The Variational Principle

In the previous session, the weak form is described as an integral form of the differential equation. For problems in which time derivative terms are not present, an alternative integral expression known as variational principle is present. The variational principle can also be used as the basis for constructing the finite element equations.

In the one-dimensional (1D) problems, the variational principle specifies a scalar quantity  $\Pi$ , called the *functional*, which is defined in an integral format as below

$$\Pi = \int_{\Omega} \mathbf{F}\left(u, \frac{du}{dx}, \dots\right) dx + \mathbf{E}\left(u, \frac{du}{dx}, \dots\right)|_{\Gamma} \quad (2.29)$$

where  $u$  is the unknown function and  $\mathbf{F}$  and  $\mathbf{E}$  are specified differential operators. The solution for the continuous problems is a function  $u$  which makes  $\Pi$  stationary with respect to arbitrary changes  $\delta u$ . Therefore, to obtain the solution for a continuous problem, the variation of the functional can be written as below

$$\delta \Pi(\mathbf{u}) = 0 \quad (2.30)$$

The equation above is satisfied for any  $\delta u$ , which defines the condition of stationarity.

If the variational principle can be found, then the equations can be established to obtain the approximate solutions in standard and integral forms suitable for finite element analysis.

### 2.4.1 Establishment of the Variational Principle for Differential Equations

#### (1) Definition of the linear operator

For the differential equation given below:

$$L(u) + b = 0 \quad \text{in } \Omega \quad (2.31)$$

if the differential operator  $L$  possesses the following property, then it is called *linear operator*:

$$L(\alpha u_1 + \beta u_2) = L(\alpha u_1) + L(\beta u_2) = \alpha L(u_1) + \beta L(u_2) \quad (2.32)$$

where  $\alpha$  and  $\beta$  are two constants. For example, because  $\frac{d}{dx}(\alpha u_1 + \beta u_2) = \alpha \frac{du_1}{dx} + \beta \frac{du_2}{dx}$ ,  $\frac{d}{dx}$  is a linear operator.

### (2) Definition of the self-adjoint operator

Defining the inner product of  $L(u)$  and an arbitrary function  $v$  as  $\int_{\Omega} L(u)v d\Omega$ , and using the integration by parts rule, we can obtain

$$\int_{\Omega} L(u)v d\Omega = \int_{\Omega} u L^*(v) d\Omega + b.t.(u, v) \quad (2.33)$$

where  $b.t.(u, v)$  is the integration term on the boundary ( $\Gamma$ ) of  $\Omega$ .  $L^*$  is called the *adjoint operator* of  $L$ . If  $L^* = L$ ,  $L^*$  is called the *self-adjoint operator* of  $L$ .

By checking whether Eq. (2.33) is satisfied or not, we can judge whether an operator is a self-adjoint operator or not. For example, the operator  $L(u) = -\frac{d^2u}{dx^2}$  can be proved to be a self-adjoint operator as following:

$$\begin{aligned} \int_{x_1}^{x_2} L(u)v dx &= \int_{x_1}^{x_2} \left( -\frac{d^2u}{dx^2} \right) v dx \\ &= \int_{x_1}^{x_2} \frac{du}{dx} \frac{dv}{dx} dx - \left( \frac{du}{dx} v \right) \Big|_{x_1}^{x_2} \\ &= \int_{x_1}^{x_2} \left( -\frac{d^2v}{dx^2} \right) u dx + \left( u \frac{dv}{dx} \right) \Big|_{x_1}^{x_2} - \left( \frac{du}{dx} v \right) \Big|_{x_1}^{x_2} \end{aligned} \quad (2.34)$$

It should be noted that in the calculation process above, the integration by parts rule is used twice.

### (3) Establishment of the functional based on the linear self-adjoint operator

The differential equation and boundary condition of the physical problem are given as below:

$$\mathbf{A}(\mathbf{u}) = \mathbf{L}(\mathbf{u}) + \mathbf{f} = 0 \quad \text{in } \Omega \quad (2.35)$$

$$\mathbf{B}(\mathbf{u}) = 0 \quad \text{on } \Gamma \quad (2.36)$$

The equivalent Galerkin integral form is given as below:

$$\int_{\Omega} \delta \mathbf{u}^T [\mathbf{L}(\mathbf{u}) + \mathbf{f}] d\Omega - \int_{\Gamma} \delta \mathbf{u}^T \mathbf{B}(\mathbf{u}) d\Gamma = 0 \quad (2.37)$$

Because  $\mathbf{L}$  is a linear self-adjoint operator, we have the following operations

$$\begin{aligned} \int_{\Omega} \delta \mathbf{u}^T \mathbf{L}(\mathbf{u}) d\Omega &= \int_{\Omega} \left[ \frac{1}{2} \delta \mathbf{u}^T \mathbf{L}(\mathbf{u}) + \frac{1}{2} \delta \mathbf{u}^T \mathbf{L}(\mathbf{u}) \right] d\Omega \\ &= \int_{\Omega} \left[ \frac{1}{2} \delta \mathbf{u}^T \mathbf{L}(\mathbf{u}) + \frac{1}{2} \mathbf{u}^T \mathbf{L}(\delta \mathbf{u}) \right] d\Omega + \text{b.t.}(\delta \mathbf{u}, \mathbf{u}) \\ &= \int_{\Omega} \left[ \frac{1}{2} \delta \mathbf{u}^T \mathbf{L}(\mathbf{u}) + \frac{1}{2} \mathbf{u}^T \delta \mathbf{L}(\mathbf{u}) \right] d\Omega + \text{b.t.}(\delta \mathbf{u}, \mathbf{u}) \\ &= \delta \int_{\Omega} \frac{1}{2} \mathbf{u}^T \mathbf{L}(\mathbf{u}) d\Omega + \text{b.t.}(\delta \mathbf{u}, \mathbf{u}) \end{aligned} \quad (2.38)$$

Putting this equation into Eq. (2.37), the variational principle form for the original physical problems can be obtained:

$$\delta \Pi(\mathbf{u}) = 0 \quad (2.39)$$

where

$$\Pi(\mathbf{u}) = \int_{\Omega} \left[ \frac{1}{2} \mathbf{u}^T \mathbf{L}(\mathbf{u}) + \mathbf{u}^T \mathbf{f} \right] d\Omega + \text{b.t.}(\mathbf{u})$$

is the functional of the original physical problem.

It can be seen from the processes above that, the equivalent Galerkin integration form of the original problem (i.e., the differential equation and the boundary conditions) is equivalent to the variational principle of the original problem, i.e., the original differential equation and boundary conditions are equivalent to the variational of functional equaling to zero (i.e., find the stationary of the functional).

### 2.4.2 The Ritz Method

After the establishment of the variational principle equations for the original physical problems, the Ritz method can be used to find the approximate solution for the original problems. The Ritz method can be described as follows:

First, using a trial function, which contains the coefficients to be determined, to represent the approximate solution for the unknown function, the following expression can be obtained:

$$\mathbf{u} \approx \tilde{\mathbf{u}} = \sum_{i=1}^n N_i a_i = \mathbf{Na} \quad (2.40)$$

where  $a_i$  are the parameters to be determined,  $N_i$  are known interpolation functions. Putting the trial function into the variational principle, i.e., Eq. (2.39), the following equation can be obtained:

$$\delta \Pi = \frac{\partial \Pi}{\partial a_1} \delta a_1 + \frac{\partial \Pi}{\partial a_2} \delta a_2 + \cdots + \frac{\partial \Pi}{\partial a_n} \delta a_n = 0 \quad (2.41)$$

where  $\delta a_1, \delta a_2, \dots$  are arbitrary. Therefore, to make Eq. (2.41) satisfied, the following equations have to be satisfied, i.e.:

$$\frac{\partial \Pi}{\partial \mathbf{a}} = \left\{ \begin{array}{c} \frac{\partial \Pi}{\partial a_1} \\ \frac{\partial \Pi}{\partial a_2} \\ \vdots \\ \frac{\partial \Pi}{\partial a_n} \end{array} \right\} = 0 \quad (2.42)$$

There are  $n$  equations which are sufficient to solve the  $n$  unknown coefficients, i.e.,  $a_1, a_2, \dots, a_n$ .

Below the calculation of the differential equation using the Ritz method is demonstrated.

**Example 2.6:** Given the following differential equation and the boundary conditions:

$$\frac{d^2u}{dx^2} + u + x = 0 \quad (0 \leq x \leq 1)$$

The boundary conditions are given as below

$$\begin{cases} u(0) = 0 \\ u(1) = 0 \end{cases}$$

The question is to work out the solution for  $u$  using the Ritz method.

**Solution:** The first step in the Ritz method is to work out the functional. According to Eq. (2.30), the functional for this specific problem can be written as:

$$\Pi = \int_0^1 \left[ -\frac{1}{2} \left( \frac{du}{dx} \right)^2 + \frac{1}{2} u^2 + ux \right] dx$$

Below the calculation processes using different forms of trial functions are given.

- (1) The polynomial function, i.e.,  $\tilde{u} = a_1x(1 - x)$  is chosen as the trial function. It should be noted that the boundary conditions are satisfied using this trial function, i.e.,  $u(0) = 0, u(1) = 0$ .

Putting the trial function into the functional, the following relation can be obtained:

$$\Pi = -\frac{1}{2} \left( \frac{3}{10} \right) a_1^2 + \frac{1}{12} a_1$$

where  $a_1$  is the coefficient to be determined. According to the variational principle, the derivative of the functional with respect to  $a_1$  equals to zero, i.e.,

$$\partial \Pi / \partial a_1 = 0$$

The unknown  $a_1$  can be solved using the equation above, which gives the following answer:

$$a_1 = \frac{5}{18}$$

Then, the approximate solution for the original differential equation is:

$$\tilde{u} = \frac{5}{18}x(1 - x)$$

- (2) The polynomial function, i.e.,  $\tilde{u} = a_1 \sin x + a_2 x$  is chosen as the trial function, and then the boundary conditions should be checked. The condition of  $u(0) = 0$  is satisfied. The condition  $u(1) = 0$  should also be satisfied, and then the following equation can be obtained:

$$a_1 \sin x + a_2 x = a_1 \sin 1 + a_2 1 = 0$$

The relation between  $a_1$  and  $a_2$  can be obtained from the equation above, i.e.:

$$a_2 = -a_1 \sin 1$$

Then, the trial function can be rewritten as:

$$\tilde{u} = a_1(\sin x - x \sin 1)$$

The derivative of the trial function with respect to  $x$  can be written as:

$$\frac{d\tilde{u}}{dx} = a_1(\cos x - \sin 1)$$

Putting the equations above into the functional, the following equation can be obtained:

$$\Pi = -\frac{1}{2}a_1^2 \sin 1 \left( \frac{2}{3} \sin 1 - \cos 1 \right) + a_1 \left( \frac{2}{3} \sin 1 - \cos 1 \right)$$

Similarly, according to the variational principle, the derivative of the functional with respect to  $a_1$  equals to zero, i.e.,

$$\frac{\partial \Pi}{\partial a_1} = \left( \frac{2}{3} \sin 1 - \cos 1 \right) (-a_1 \sin 1 + 1) = 0$$

The answer for the unknown coefficient  $a_1$  can be obtained by solving the equation above

$$a_1 = \frac{1}{\sin 1}$$

Therefore, the approximate solution for the original differential equation can be written as:

$$\tilde{u} = \frac{\sin x}{\sin 1} - x$$

It should be noted that only one and two coefficients are used in the trial functions. Generally, when the number of coefficients to be determined is increased, the accuracy of the approximate solution can be increased accordingly, but the complexity of the calculation will also be increased. Therefore, when using the Ritz method, the selection of the number of the coefficient in the trial function should be decided based on the specific application and both the calculation efficiency and accuracy should be considered.

## 2.5 The Principle of Virtual Work

In the deformable solid body, the virtual work of an arbitrary balanced force system in the admissible deformation state, which satisfies the compatibility conditions, equals to zero, i.e., the sum of the internal virtual work and the external virtual work equals to zero. Depending on the given boundary conditions, the principle of virtual work can be divided into the virtual displacement principle and the virtual stress principle. The *virtual displacement principle* is the equivalent integral weak form of the equilibrium equation and the force boundary condition. The *virtual stress principle* is the equivalent integral weak form of the geometric equation and the displacement boundary condition. Below the detailed formulations for the virtual displacement and virtual stress principles are given.

### 2.5.1 The Virtual Displacement Principle

When the force boundary condition is given, the virtual displacement principle can be established using the equilibrium equation:

$$\mathbf{A}\sigma + \bar{\mathbf{f}} = 0 \text{ (in the deformable body)} \quad (2.43)$$

$$\mathbf{n}\sigma - \bar{\mathbf{T}} = 0 \text{ (on the boundary)} \quad (2.44)$$

where the stress tensor is defined as  $\sigma = [\sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yz} \tau_{zx}]^T$ , the strain tensor is defined as  $\varepsilon = [\varepsilon_x \varepsilon_y \varepsilon_z \gamma_{xy} \gamma_{yz} \gamma_{zx}]^T$ , the body force vector is defined as  $\bar{\mathbf{f}} = [\bar{f}_x \bar{f}_y \bar{f}_z]^T$ , the area force vector is defined as  $\bar{\mathbf{T}} = [\bar{T}_x \bar{T}_y \bar{T}_z]^T$ .  $\mathbf{A}$  and  $\mathbf{n}$  are the differential operators, which are defined as

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

$$\mathbf{n} = \begin{bmatrix} n_x & 0 & 0 & n_y & 0 & n_z \\ 0 & n_y & 0 & n_x & n_z & 0 \\ 0 & 0 & n_z & 0 & n_y & n_x \end{bmatrix}$$

The equivalent integral form of the equilibrium equation and boundary condition can be written as below:

$$\int_V \delta \mathbf{u}^T (\mathbf{A}\sigma + \bar{\mathbf{f}}) dV - \int_S \delta \mathbf{u}^T (\mathbf{n}\sigma - \bar{\mathbf{T}}) dS = 0 \quad (2.45)$$

where  $\delta \mathbf{u}$  is the variation of the displacement. The weak form of Eq. (2.45) can be obtained using the integration by parts, which gives the following result:

$$\int_V (\delta \varepsilon^T \sigma - \delta \mathbf{u}^T \bar{\mathbf{f}}) dV - \int_S \delta \mathbf{u}^T \bar{\mathbf{T}} dS = 0 \quad (2.46)$$

It should be noted that if the force system is balanced, the sum of the work done by the virtual displacement or the virtual strain should be zero. On the contrary, if the sum of the work done by the virtual displacement is zero, then the force system is balanced. Therefore, the virtual displacement principle is the necessary and sufficient condition of the equilibrium of the force system. It also should be noted that when deriving the virtual displacement principle, the physical equation is not involved. Therefore, the virtual displacement principle can be suitable for not only the linear elastic problems but also the nonlinear elastic problems.

### 2.5.2 The Virtual Stress Principle

When the displacement boundary condition is given, the virtual stress principle can be established using the geometric equation:

$$\varepsilon = \mathbf{L}\mathbf{u} \text{ (in the deformable body)} \quad (2.47)$$

$$\mathbf{u} - \bar{\mathbf{u}} = 0 \text{ (on the boundary)} \quad (2.48)$$

where the strain tensor is defined as  $\varepsilon = [\varepsilon_x \ \varepsilon_y \ \varepsilon_z \ \gamma_{xy} \ \gamma_{yz} \ \gamma_{zx}]^T$ , the displacement vector is defined as  $\mathbf{u} = [u_x \ u_y \ u_z]^T$ , the prescribed displacement vector is defined as  $\bar{\mathbf{u}} = [\bar{u}_x \ \bar{u}_y \ \bar{u}_z]^T$ ,  $\mathbf{L}$  is the differential operator and defined as

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

The equivalent integral form of the geometric equation and displacement boundary condition can be written as below:

$$\int_V \delta\sigma^T (\varepsilon - \mathbf{L}\mathbf{u}) dV - \int_S \delta\mathbf{T}^T (\mathbf{u} - \bar{\mathbf{u}}) dS = 0 \quad (2.49)$$

where  $\delta\sigma$  is the variation of the stress and satisfies the equilibrium equation, i.e.,  $\delta\sigma = 0$ . The weak form of Eq. (2.49) can be obtained using the integration by parts which gives the following result:

$$\int_V \delta\sigma^T \varepsilon dV - \int_S \delta\mathbf{T}^T \bar{\mathbf{u}} dS = 0 \quad (2.50)$$

It should be noted that if the displacement is compatible, then sum of the work done by the virtual stress and the reaction force on the virtual boundary equals to zero. On the contrary, if the sum of the work done by the virtual force system equals to zero, the displacement is then satisfying the strain-displacement relation. Therefore, the virtual stress principle is the necessary and sufficient condition of the equilibrium for the force system.

Similar to the virtual displacement principle, when deriving the virtual stress principle, the physical equation is not involved. Therefore, the virtual stress principle is suitable for not only the linear elastic problems but also the nonlinear elastic problems.

### 2.5.3 The Minimal Potential Energy Principle

The minimal potential energy principle is established based on the virtual displacement principle:

$$\int_V (\delta \varepsilon^T \sigma - \delta \mathbf{u}^T \bar{\mathbf{f}}) dV - \int_S \delta \mathbf{u}^T \bar{\mathbf{T}} dS = 0 \quad (2.51)$$

Taking use of the physical equation  $\sigma = \mathbf{D}\varepsilon$ , where  $\mathbf{D}$  is the elasticity matrix, the following equation can be obtained

$$\delta \Pi_p(\mathbf{u}) = 0 \quad (2.52)$$

where  $\Pi_p(\mathbf{u}) = \int_V \left( \frac{1}{2} \varepsilon^T \mathbf{D} \varepsilon - \mathbf{u}^T \bar{\mathbf{f}} \right) d\Omega - \int_{S_\sigma} \mathbf{u}^T \bar{\mathbf{T}} dS$  is the potential energy. The term  $\int_V \frac{1}{2} \varepsilon^T \mathbf{D} \varepsilon d\Omega$  is called the *strain energy density* widely used in many mechanical problems and is also called the *elastic deformation potential energy*. The terms  $\int_V \mathbf{u}^T \bar{\mathbf{f}} d\Omega$  and  $\int_{S_\sigma} \mathbf{u}^T \bar{\mathbf{T}} dS$  are the external force potential energy.

### 2.5.4 The Minimal Complementary Energy Principle

The minimal complementary energy principle is established based on the virtual stress principle:

$$\int_V \delta \sigma^T \varepsilon dV - \int_{S_U} \delta \mathbf{T}^T \bar{\mathbf{u}} dS = 0 \quad (2.53)$$

Taking use of the physical equation  $\boldsymbol{\varepsilon} = \mathbf{S}\sigma$  where  $\mathbf{S}$  is the compliance matrix, the following equation can be obtained

$$\delta \Pi_c(\sigma) = 0 \quad (2.54)$$

where  $\Pi_c(\sigma)$  is the complementary energy and can be written as below:

$$\Pi_c(\sigma) = \int_V \frac{1}{2} \sigma^T \mathbf{S} \sigma d\Omega - \int_{S_u} \mathbf{T}^T \bar{\mathbf{u}} dS$$

where the term  $\int_V \frac{1}{2} \sigma^T \mathbf{S} \sigma d\Omega$  is called the *elastic complementary energy*, and the term  $\int_{S_u} \mathbf{T}^T \bar{\mathbf{u}} dS$  is called the *external force complementary energy*.

It should be noted that the solution obtained from the minimum potential energy principle is the lower bound of the actual solution. That's to say the approximate

displacement field is smaller and the analysis model of the structure is more rigid. The solution obtained from the minimum complementary energy principle is the upper bound of the actual solution. That's to say the approximate displacement field is larger and the analysis model of structure is softer. The upper and lower bounds of problems can be obtained using two principles at the same time. In addition, we can estimate the error of the approximate solution more accurately, which is meaningful for the engineering problems.

# Chapter 3

## Finite Element Analysis Using Bar Element



### 3.1 Introduction

In the previous chapter, the basic principle of the finite element analysis (FEA), i.e. the minimal potential energy principle, is introduced. However, the FEA is to find the trial functions for the discretized segments and then assemble them together to obtain the solution for the entire domain. The method is not to find the trial function for the entire domain directly. The processing of the discretized segments requires a large amount of computational power, especially when a large number of discretized elements are used. With the development of computer technology, the computer power is not an issue anymore and consequently the FEM has become more and more widely used in many different fields. In this book, the FE analysis procedures for the 1D, 2D and 3D commonly used elements are given, starting with the simplest 1D element, i.e., the bar element, in this chapter.

### 3.2 The Finite Element Calculation Procedure

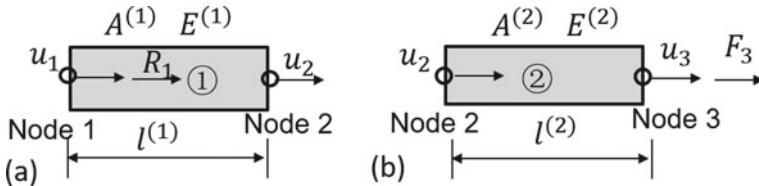
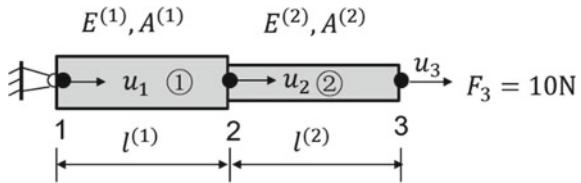
A staircase 2-bar structure (Fig. 3.1) is used to illustrate the FE procedure for solving the bar structure. The mechanical properties and dimensions of the two bars are given as below:  $E^{(1)} = E^{(2)} = 2.0 \times 10^7 \text{ Pa}$ ,  $A^{(1)} = A^{(2)} = 2.0 \text{ cm}^2$ ,  $l^{(1)} = l^{(2)} = 10.0 \text{ cm}$ . An external force 10.0 N is applied on the right side of the structure (Fig. 3.1). The question is to find the reaction force in the system.

The FE procedure for solving this problem is given as below:

#### (1) Discretization of the structure

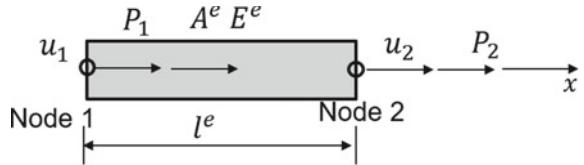
In this step, the structure is meshed using sufficient finite elements. In general, the number of elements used to discretize the structure should be determined from the mesh convergence study. To simplify the problem and for the aim to illustrate the FE procedure, the structure is discretized using only two bar elements (Fig. 3.2).

**Fig. 3.1** A staircase 2-bar structure



**Fig. 3.2** The two elements used to discretize the structure

**Fig. 3.3** One-dimensional bar element



As shown in Fig. 3.2, in each bar element, there are two nodes. In the two elements, there is a reaction force at the first node, an external force at the third node and no external force at the second node.

## (2) Analysis of individual element

In this step, the stiffness matrix for each element is worked out. The first element is used as an example to illustrate the procedure (Fig. 3.3).

Writing out the trial function for the element is the first step to solve the element. For a general one-dimensional (1D) bar element, the displacement field function  $u(x)$  can be written as below:

$$u(x) = a_0 + a_1x + a_2x^2 + \dots \quad (3.1)$$

where  $a_0, a_1, a_2, \dots$  are the coefficients to be determined. Here the polynomial function in terms of the Taylor series is used and this kind of function is widely used for the displacement field functions for different types of finite elements, because the mathematical operations (e.g., the differential, the integration, etc.) on these functions are much easier.

Regarding the 1D bar element shown in Fig. 3.3, the displacement values at the two nodes, i.e.,  $u_1$  and  $u_2$ , are taken as the known variables and used for

the interpolation to obtain the displacements at other locations of the element. Because only two equations can be set up using  $u_1$  and  $u_2$ , only two unknown coefficients can be solved, i.e.,  $a_0$  and  $a_1$ . Therefore, the following displacement field function is set up for the 1D linear bar element:

$$u(x) = a_0 + a_1 x \quad (3.2)$$

where  $a_0$  and  $a_1$  are the coefficients to be determined. Below, the procedure for solving the two coefficients is illustrated. Taking the displacement value at the two nodes as the knowns, the following equations can be obtained:

$$u(x)|_{x=0} = a_0 = u_1 \quad (3.3a)$$

$$u(x)|_{x=l^e} = a_0 + a_1 l^e = u_2 \quad (3.3b)$$

Solving Eq. (3.3), the two coefficients can be obtained:

$$a_0 = u_1 \quad (3.4a)$$

$$a_1 = (u_2 - u_1)/l^e \quad (3.4b)$$

Then the displacement field function can be rewritten as:

$$u(x) = u_1 + \left( \frac{u_2 - u_1}{l^e} \right) x = \left( 1 - \frac{x}{l^e} \right) u_1 + \left( \frac{x}{l^e} \right) u_2 = \mathbf{N}(x) \mathbf{q}^e \quad (3.5)$$

where  $\mathbf{N}(x)$  is the shape function vector which can be given as below:

$$\mathbf{N}(x) = \left[ \left( 1 - \frac{x}{l^e} \right) \quad \frac{x}{l^e} \right] \quad (3.6)$$

In Eq. (3.5),  $\mathbf{q}^e$  is the *nodal displacement vector* which is given as below:

$$\mathbf{q}^e = [u_1 \ u_2]^T \quad (3.7)$$

After the displacement field is solved, the next step is to work out the strain field within the element. The strain field can be solved using the strain–displacement relation, which takes the following form for the 1D problem:

$$\boldsymbol{\varepsilon}(x) = \frac{du(x)}{dx} = \left[ -\frac{1}{l^e} \quad \frac{1}{l^e} \right] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \mathbf{B}(x) \mathbf{q}^e \quad (3.8)$$

where  $\mathbf{B}(x)$  is the *geometric function vector* and has the following form:

$$\mathbf{B}(x) = \frac{d}{dx}(x) = \begin{bmatrix} -\frac{1}{l^e} & \frac{1}{l^e} \end{bmatrix}$$

Afterwards, the stress field function within the 1D bar element can be set up using the physical equation, i.e., the stress–strain relation:

$$\sigma(x) = E^e \varepsilon(x) = E^e \mathbf{B}(x) q^e = \mathbf{S}(x) q^e \quad (3.9)$$

where  $\mathbf{S}(x)$  is the *stress–strain vector* and has the following form:

$$\mathbf{S}(x) = E^e \mathbf{B}(x) = \begin{bmatrix} -\frac{E^e}{l^e} & \frac{E^e}{l^e} \end{bmatrix}$$

From the elemental strain and stress formulations, the elemental potential energy can be written as below

$$\begin{aligned} \Pi^e &= U^e - W^e \\ &= \frac{1}{2} \int \sigma(x) \varepsilon(x) d\Omega - (P_1 u_1 + P_2 u_2) \\ &= \frac{1}{2} \int_0^{l^e} \mathbf{q}^{eT} \mathbf{S}^T(x) \mathbf{B}(x) \mathbf{q}^e A^e dx - (P_1 u_1 + P_2 u_2) \\ &= \frac{1}{2} [u_1 \ u_2] \begin{bmatrix} \frac{E^e A^e}{l^e} & -\frac{E^e A^e}{l^e} \\ -\frac{E^e A^e}{l^e} & \frac{E^e A^e}{l^e} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} - [P_1 \ P_2] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \\ &= \frac{1}{2} \mathbf{q}^{eT} \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^{eT} \mathbf{q}^e \end{aligned} \quad (3.10)$$

where  $U^e$  is the internal energy,  $W^e$  is the energy caused by the external forces,  $\mathbf{K}^e$  is the elemental stiffness matrix, which is given as below:

$$\mathbf{K}^e = \frac{E^e A^e}{l^e} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$\mathbf{P}^e$  is the elemental nodal force vector and takes the following form:

$$\mathbf{P}^e = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

### (3) Assembly of the discrete elements

After the potential energies for each element are obtained, the next step is to assemble them together and obtain the total potential energy for the entire structure. For the 2-bar structure given in Fig. 3.1, the total potential energy is the summation of the two elemental potentials, i.e.,

$$\begin{aligned}
\Pi &= \Pi^{(1)} + \Pi^{(2)} \\
&= \left[ \frac{1}{2} \mathbf{q}^{(1)\top} \mathbf{K}^{(1)} \mathbf{q}^{(1)} - \mathbf{P}^{(1)\top} \mathbf{q}^{(1)} \right] + \left[ \frac{1}{2} \mathbf{q}^{(2)\top} \mathbf{K}^{(2)} \mathbf{q}^{(2)} - \mathbf{P}^{(2)\top} \mathbf{q}^{(2)} \right] \\
&= \frac{1}{2} [u_1 \ u_2] \begin{bmatrix} \frac{E^{(1)}A^{(1)}}{l^{(1)}} & -\frac{E^{(1)}A^{(1)}}{l^{(1)}} \\ -\frac{E^{(1)}A^{(1)}}{l^{(1)}} & \frac{E^{(1)}A^{(1)}}{l^{(1)}} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} - [R_1 \ 0] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \\
&\quad + \frac{1}{2} [u_2 \ u_3] \begin{bmatrix} \frac{E^{(2)}A^{(2)}}{l^{(2)}} & -\frac{E^{(2)}A^{(2)}}{l^{(2)}} \\ -\frac{E^{(2)}A^{(2)}}{l^{(2)}} & \frac{E^{(2)}A^{(2)}}{l^{(2)}} \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} - [0 \ F_3] \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} \\
&= \frac{1}{2} [u_1 \ u_2 \ u_3] \begin{bmatrix} \frac{E^{(1)}A^{(1)}}{l^{(1)}} & -\frac{E^{(1)}A^{(1)}}{l^{(1)}} & 0 \\ -\frac{E^{(1)}A^{(1)}}{l^{(1)}} & \frac{E^{(1)}A^{(1)}}{l^{(1)}} + \frac{E^{(2)}A^{(2)}}{l^{(2)}} & -\frac{E^{(2)}A^{(2)}}{l^{(2)}} \\ 0 & -\frac{E^{(2)}A^{(2)}}{l^{(2)}} & \frac{E^{(2)}A^{(2)}}{l^{(2)}} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \\
&\quad - [R_1 \ 0 \ F_3] \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}
\end{aligned} \tag{3.11}$$

#### (4) Treatment of the boundary condition

Now the boundary condition, particularly the displacement boundary condition, can be introduced into the equations. For the given structure in Fig. 3.1, the displacement boundary condition is  $u_1 = 0$ . Putting this displacement boundary condition into Eq. (3.11), the following reduced form of the total potential energy can be obtained:

$$\Pi = \frac{1}{2} [u_2 \ u_3] \begin{bmatrix} \frac{E^{(1)}A^{(1)}}{l^{(1)}} + \frac{E^{(2)}A^{(2)}}{l^{(2)}} & -\frac{E^{(2)}A^{(2)}}{l^{(2)}} \\ -\frac{E^{(2)}A^{(2)}}{l^{(2)}} & \frac{E^{(2)}A^{(2)}}{l^{(2)}} \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} - [0 \ F_3] \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} \tag{3.12}$$

#### (5) Establishment of stiffness equation

Establishment of the stiffness equation is the key step in FE analysis. In the stiffness equations, the nodal displacements are taken as the unknowns and once they are solved, the displacements at other locations within the element, the strain and stress fields, etc. can all be solved. In the FE calculation, the structural stiffness equation can be obtained using the minimal potential principle, i.e.,

$$\min_{u_2, u_3} \Pi \Rightarrow \frac{\partial \Pi}{\partial u_2} = 0, \frac{\partial \Pi}{\partial u_3} = 0 \tag{3.13}$$

In terms of the matrix format:

$$\begin{bmatrix} \frac{E^{(1)}A^{(1)}}{l^{(1)}} + \frac{E^{(2)}A^{(2)}}{l^{(2)}} & -\frac{E^{(2)}A^{(2)}}{l^{(2)}} \\ -\frac{E^{(2)}A^{(2)}}{l^{(2)}} & \frac{E^{(2)}A^{(2)}}{l^{(2)}} \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_3 \end{Bmatrix} \tag{3.14}$$

where  $u_2$  and  $u_3$  are the displacements at the second and third nodes and are the unknowns,  $F_3$  is the external force at the third node.

### (6) Calculation of the nodal displacement

Putting the values of  $E^{(1)}$ ,  $E^{(2)}$ ,  $A^{(1)}$ ,  $A^{(2)}$ ,  $l^{(1)}$ ,  $l^{(2)}$  and  $F_3$  into Eq. (3.14), the stiffness equation can be rewritten as:

$$2 \times 10^4 \begin{bmatrix} 3 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 10 \end{Bmatrix} \quad (3.15)$$

Then the nodal displacements can be obtained by solving the equation above, which are given as below:

$$u_2 = 2.5 \times 10^{-4} \text{ m}, u_3 = 7.5 \times 10^{-4} \text{ m} \quad (3.16)$$

In summary, the nodal displacement vector for the first element is:

$$\mathbf{q}^{(1)} = [u_1 \ u_2]^T = [0 \ 2.5 \times 10^{-4}]^T \quad (3.17)$$

The nodal displacement vector for the second element is:

$$\mathbf{q}^{(2)} = [u_2 \ u_3]^T = [2.5 \times 10^{-4} \ 7.5 \times 10^{-4}]^T \quad (3.18)$$

The nodal displacement vector for the 2-bar structure is:

$$\mathbf{q} = [u_1 \ u_2 \ u_3]^T = [0 \ 2.5 \times 10^{-4} \ 7.5 \times 10^{-4}]^T \quad (3.19)$$

### (7) Calculation of the strain field within the element

Based on Eq. (3.8), the strain within each element can be obtained as:

$$\varepsilon^{(1)}(x) = \mathbf{B}^{(1)}(x)\mathbf{q}^{(1)} = \left[ -\frac{1}{l^{(1)}} \ \frac{1}{l^{(1)}} \right] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = 2.5 \times 10^{-3} \quad (3.20a)$$

$$\varepsilon^{(2)}(x) = \mathbf{B}^{(2)}(x)\mathbf{q}^{(2)} = \left[ -\frac{1}{l^{(2)}} \ \frac{1}{l^{(2)}} \right] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = 5 \times 10^{-3} \quad (3.20b)$$

### (8) Calculation of the reaction force

Several ways can be used to work out the structural reaction force. Here, the elemental stiffness equation is used to obtain the reaction force. Because the reaction force occurs at the first node, the first element is considered and taken as an

independent structure. The stiffness equation for the first element can be obtained by  $\frac{\partial \Pi^e}{\partial q^e} = 0$ . Then, the following elemental stiffness equation can be obtained:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \quad (3.21)$$

In terms of the matrix format:

$$\frac{E^{(1)} A^{(1)}}{l^{(1)}} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} \quad (3.22)$$

where  $P_1$  is the external force at the first node, i.e., the reaction force  $P_2$  is the force at the second node induced by the second element. In Eq. (3.22), the displacement variables  $u_1$  and  $u_2$  are already solved using the global stiffness equation. Therefore, the two unknowns, i.e.,  $P_1$  and  $P_2$ , can be obtained by solving the Eq. (3.22):

$$P_1 = -10.0 \text{ N}$$

$$P_2 = 10.0 \text{ N}$$

### 3.3 Property of the Shape Function for the bar Element

Shape function is an important part in the finite element analysis. Meantime, the shape function possesses clear physical meanings and consequently it plays an important role in understanding the FE method. In this chapter, the 1D bar element is used to illustrate the properties of the shape function, but the properties can be extended to other types of elements.

From the previous parts, it is known that the displacement field function for the 1D bar element can be written as:

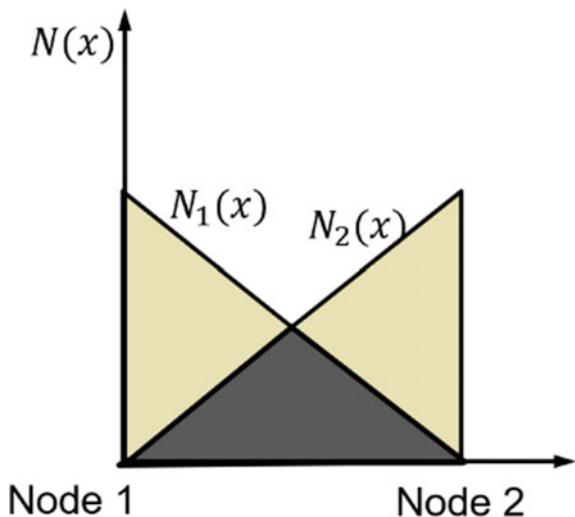
$$u(x) = N_1(x)u_1 + N_2(x)u_2 = \mathbf{N}(x)\mathbf{q}^e \quad (3.23)$$

where  $u_1$  and  $u_2$  are the nodal displacements,  $N_1(x)$  and  $N_2(x)$  are the shape functions associated with the first and second nodes, respectively,  $\mathbf{N}(x)$  is the shape function vector. The two basic properties of the shape function are as following:

**Property 1:** The value of the shape function  $N_i(x)$  at node  $i$  is one, and its values at other nodes are zeros (Fig. 3.4).

Two special cases of the 2-node bar element are used to understand and prove this property. In the first case, the displacement at the left end of the bar element (the first node) is made equal to one and the right end of the element (the second node) is fully constrained, i.e.,  $u_1 = 1, u_2 = 0$ . Then based on Eq. (3.5), the following equation can be obtained:

**Fig. 3.4** Shape functions of bar element



$$u(x) = N_1(x) \quad (3.24a)$$

The equation above gives the physical meaning of the shape function  $N_1(x)$ , i.e.,  $N_1(x)$  represents the elemental displacement field function which makes the displacement at the first node equal to one and the displacement at the other node equal to zero (Fig. 3.4).

In the second case, the displacement at the left end of the bar element (the first node) is fully constrained and the displacement at the right end of the element (the second node) is made equal to one, i.e.,  $u_1 = 0, u_2 = 1$ . Then based on Eq. (3.5), the following equation can be obtained:

$$u(x) = N_2(x) \quad (3.24b)$$

The equation above gives the physical meaning of the shape function  $N_2(x)$ , i.e.,  $N_2(x)$  represents the elemental displacement field function which makes the displacement at the second node equal to one and the displacement at the other node equal to zero (similar to  $N_1(x)$ ) (Fig. 3.4).

**Property 2:** At any location within the element, the sum of the values of all the shape functions equals to one, i.e.,

$$\sum_{i=1}^n N_i(x) = 1 \quad (3.25)$$

where  $n$  is the number of element node. This property can be proved using the scenario of rigid displacement, in which case every point within the element has the same displacement, and thus the following equations can be obtained:

$$u(x) = \bar{u}_0 \quad (3.26)$$

$$u_1 = u_2 = \bar{u}_0 \quad (3.27)$$

Putting these equations into the displacement field function, i.e., Eq. (3.5), the following equation can be obtained:

$$\bar{u}_0 = N_1(x)\bar{u}_0 + N_2(x)\bar{u}_0 \quad (3.28)$$

Eliminating the variable  $\bar{u}_0$  in the equation above, the following equation can be obtained:

$$N_1(x) + N_2(x) = 1 \quad (3.29)$$

### 3.4 Property of the Stiffness Matrix for the bar Element

The stiffness matrix is an important part of the FE method. Understanding the properties of the stiffness matrix can help understand the FE analysis. In this chapter, we use the 2-node bar element to introduce the properties of the stiffness matrix. There are basically 5 properties of the stiffness matrix for the bar element. To introduce them, the stiffness equation for the 2-node bar element is revisited:

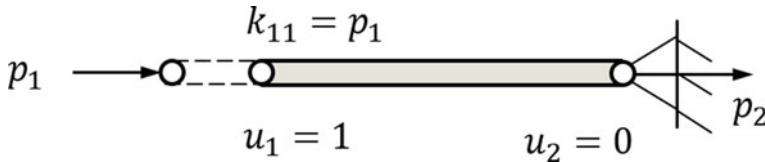
$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} \quad (3.30)$$

**Property 1** The diagonal component  $k_{ii}$  means that the force on node  $i$ , which makes the displacement at node  $i$  equal to one and the displacements at other nodes equal to zeros!

This property gives the physical meaning of the diagonal components in the stiffness matrix. To prove this property, let's consider the scenario when the displacement at the left end of the bar element (the first node) is one and the displacement at the right end of the bar element (the second node) is fully constrained, i.e.,  $u_1 = 1, u_2 = 0$ . Putting these equations into the elemental stiffness equation, i.e., Eq. (3.30), the following equation can be obtained:

$$k_{11} = p_1 \quad (3.31a)$$

The equation above gives the physical meaning of the diagonal component— $k_{11}$ , i.e.,  $k_{11}$  means the force needed on the first node which makes the displacement at the first node equal to one and the displacement at the second node equal to zero (Fig. 3.5).



**Fig. 3.5** Physical meaning of the diagonal components in elemental stiffness matrix

**Property 2** The non-diagonal component  $k_{ij}$  ( $i \neq j$ ) means that the force on node  $i$ , which makes the displacement at node  $j$  equal to one and displacements at other nodes equal to zeros!

This property gives the physical meaning of the non-diagonal elements in the stiffness matrix. Similarly to the proof of the first property, let's consider the scenario when the displacement at the left end of the bar element (the first node) is zero and the displacement at the right end of the bar element (the second node) is one, i.e.,  $u_1 = 0, u_2 = 1$ . Putting these equations into the elemental stiffness equation, i.e., Eq. (3.30), then the following equation can be obtained

$$k_{12} = p_1 \quad (3.31b)$$

The equation above gives the physical meaning of the non-diagonal element— $k_{12}$ , i.e.,  $k_{12}$  means the force needed on the first node which makes the displacement at the first node equal to zero and the displacement at the second node equal to one (Fig. 3.6).

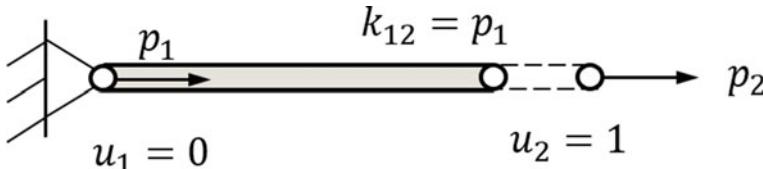
**Property 3** The stiffness matrix is symmetric, i.e.  $k_{ij} = k_{ji}$ .

This property can be proved using the formulation of the stiffness matrix, i.e.,

$$[\mathbf{K}^e]^T = \left[ \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \right]^T = \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega = \mathbf{K}^e \quad (3.32)$$

**Property 4** The stiffness matrix is positive semi-definite.

To prove this property, the elemental potential energy in terms of the nodal displacement is revisited:



**Fig. 3.6** Physical meaning of the non-diagonal components in the elemental stiffness matrix

$$\begin{aligned}
U &= \frac{1}{2} \mathbf{q}^T \mathbf{K}^e \mathbf{q}^e \\
&= \frac{1}{2} (k_{11}u_1^2 + \cdots + k_{1i}u_1u_i + \cdots + k_{1n}u_1u_n + \cdots + k_{j1}u_ju_1 + \cdots + k_{ji}u_ju_i \\
&\quad + \cdots + k_{jn}u_ju_n + \cdots + k_{n1}u_nu_1 + \cdots + k_{ni}u_nu_i + \cdots + k_{nn}u_nu_n) \\
&= \frac{1}{2} \sum_{i=1}^n \sum_{j=1}^n k_{ij}u_iu_j
\end{aligned} \tag{3.33}$$

where  $\mathbf{q}^e$  is the nodal displacement vector. Equation (3.33) implies that the element potential energy  $U$  is a quadratic homogeneous polynomial function for the displacement vector  $\mathbf{q}^e$ . Therefore, the matrix  $\mathbf{K}^e$  is the quadratic matrix. It is obvious that the potential energy  $U$  is always positive for all the  $\mathbf{q}^e$  except for  $\mathbf{q}^e = 0$ . However, the scenario of the rigid displacement should be excluded. From the mathematical point view, the matrix with this property is called the *positive-definite matrix*. However, in the scenario of rigid displacement, the nodal displacements are not zeros, i.e.,  $\mathbf{q}^e \neq 0$ , but the potential energy is zero, i.e.,  $U = 0$ . Therefore, the determinant of the stiffness matrix has to be zero, i.e.,  $|\mathbf{K}^e| = 0$ , from which we can obtain that, the elemental stiffness matrix is positive semi-definite.

**Property 5** The stiffness matrix is singular, i.e.,  $|\mathbf{K}^e| = 0$ .

The stiffness matrix is singular implying that there is no inverse matrix for the stiffness matrix. To prove this property, let's consider the scenario of rigid displacement using the 1D 2-node bar element:

Supposing the elemental nodal displacements under the nodal forces  $p_1$  and  $p_2$  are:

$$u_1 = c_1^{(1)} \tag{3.34a}$$

$$u_2 = c_2^{(2)} \tag{3.34b}$$

Additionally, supposing that a rigid displacement occurs in the element under the balanced forces, then, the nodal displacements can be written as:

$$u_1 = c_1^{(1)} + u_0 = c_1^{(2)} \tag{3.35a}$$

$$u_2 = c_2^{(1)} + u_0 = c_2^{(2)} \tag{3.35b}$$

where  $u_0$  is the magnitude of the rigid displacement. The corresponding elemental stiffness equations are:

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} c_1^{(1)} \\ c_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} \tag{3.36a}$$

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} c_1^{(2)} \\ c_2^{(2)} \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} \quad (3.36b)$$

Using Eq. (3.36b) to subtract Eq. (3.36a), the following equations can be obtained

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} c_1^{(2)} - c_1^{(1)} \\ c_2^{(2)} - c_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \quad (3.37)$$

Because  $c_1^{(2)} \neq c_1^{(1)}$ ,  $c_2^{(2)} \neq c_2^{(1)}$ , the condition of the non-zero solution for the equation above is

$$\begin{vmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{vmatrix} = 0, \text{ i.e., } |\mathbf{K}^e| = 0 \quad (3.38)$$

Equation (3.38) has proved that the stiffness matrix is a singular matrix.

On the other hand, using the equation for rigid displacement, i.e.,  $u_1 = u_2 = u_0$ , Eq. (3.30) can be rewritten as

$$k_{11}u_0 + k_{12}u_0 = 0 \quad (3.39a)$$

$$k_{21}u_0 + k_{22}u_0 = 0 \quad (3.39b)$$

Eliminating the variable  $u_0$  in the equations, the following equations can be obtained

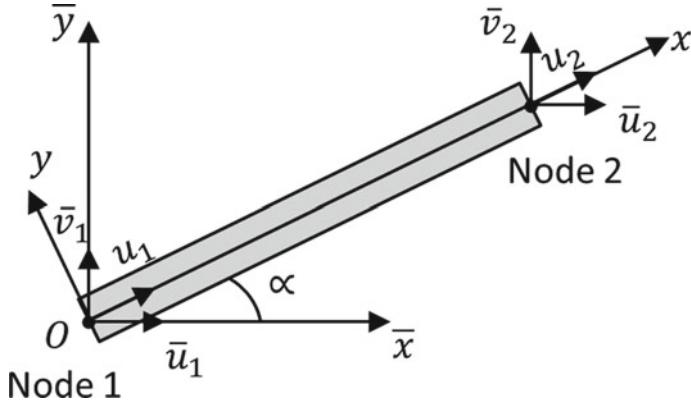
$$k_{11} + k_{12} = 0 \quad (3.40a)$$

$$k_{21} + k_{22} = 0 \quad (3.40b)$$

The equations above imply that  $k_{11}$  is linearly correlated with  $k_{12}$ , and  $k_{21}$  is linearly correlated with  $k_{22}$ .

## 3.5 The Coordinate Transformation for bar Elements

In the previous parts, the FE analysis procedure using the bar element is given, where the coordinate system is on the bar element, i.e., the  $x$ -axis is along the length direction of the bar. However, in the real engineering problem, several bars form a structure system, in which, some bars are aligned with the coordinate axis, and some are not. In order to solve the bar system, all the equations have to be transformed into one system, i.e., the global coordinate system. Otherwise, it is not possible to assemble the stiffness matrix of different elements to set up the global stiffness



**Fig. 3.7** Coordinate transformation for the bar element in the 2D plane

equation. Therefore, in this part, the transformation of the FE formulations from the local bar elemental system to the global bar coordinate system is introduced.

The following example is used to illustrate the process of coordinate transformation (Fig. 3.7). In the figure, the local element coordinate system is ( $O_{\bar{x}\bar{y}}$ ), which is along the length direction of the bar. The global coordinate system is ( $O_{xy}$ ).

The nodal displacement vector in the local coordinate system is:

$$\mathbf{q}^e = [u_1 \ u_2]^T \quad (3.41)$$

The nodal displacement vector in the global coordinate system is:

$$\bar{\mathbf{q}}^e = [\bar{u}_1 \ \bar{v}_1 \ \bar{u}_2 \ \bar{v}_2]^T \quad (3.42)$$

In Fig. 3.7, it can be seen the displacements at the first node in the global coordinate system are  $\bar{u}_1$  and  $\bar{v}_1$  along the  $\bar{x}$  and  $\bar{y}$  axes, respectively. The resultant of the  $\bar{u}_1$  and  $\bar{v}_1$  should equal to the corresponding displacement in the local coordinate system, i.e.,  $u_1$ . Similarly, the resultant of the displacements  $\bar{u}_2$  and  $\bar{v}_2$  in the global coordinate system should equal to the corresponding displacement in the local system, i.e.,  $u_2$ . Therefore, the following equations can be obtained:

$$u_1 = \bar{u}_1 \cos \alpha + \bar{v}_1 \sin \alpha \quad (3.43a)$$

$$u_2 = \bar{u}_2 \cos \alpha + \bar{v}_2 \sin \alpha \quad (3.43b)$$

The equations can be rewritten in terms of the matrix format:

$$\mathbf{q}^e = \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 \\ 0 & 0 & \cos \alpha & \sin \alpha \end{bmatrix} \begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \\ \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix} = \mathbf{T}^e \bar{\mathbf{q}}^e \quad (3.44)$$

where  $\mathbf{T}^e$  is the coordinate transformation matrix given in the following form.

$$\mathbf{T}^e = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 \\ 0 & 0 & \cos \alpha & \sin \alpha \end{bmatrix} \quad (3.45)$$

Using the transformation matrix  $\mathbf{T}^e$ , the elemental stiffness equation in the global coordinate system can be derived. It should be noted that the elemental potential energy is a scalar and its value is independent of the coordinate system. Therefore, to obtain the stiffness equation in the global coordinate system, the nodal displacement vector needs to be transformed from the local coordinate system to the global coordinate system, i.e.:

$$\begin{aligned} \Pi^e &= \frac{1}{2} \mathbf{q}^{eT} \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^{eT} \mathbf{q}^e \\ &= \frac{1}{2} \bar{\mathbf{q}}^{eT} (\mathbf{T}^{eT} \mathbf{K}^e \mathbf{T}^e) \bar{\mathbf{q}}^e - (\mathbf{T}^{eT} \mathbf{P}^e)^T \bar{\mathbf{q}}^e \\ &= \frac{1}{2} \bar{\mathbf{q}}^{eT} \bar{\mathbf{K}}^e \bar{\mathbf{q}}^e - \bar{\mathbf{P}}^{eT} \bar{\mathbf{q}}^e \end{aligned} \quad (3.46)$$

where  $\bar{\mathbf{K}}^e$  is the elemental stiffness matrix in the global coordinate system,  $\bar{\mathbf{P}}^e$  is the nodal force vector in the global coordinate system, i.e.:

$$\bar{\mathbf{K}}^e = \mathbf{T}^{eT} \mathbf{K}^e \mathbf{T}^e$$

$$\bar{\mathbf{P}}^e = \mathbf{T}^{eT} \mathbf{P}^e$$

Then based on the minimal potential energy principle, the stiffness equation in the global coordinate system can be obtained by working out the derivative of the potential energy with respect to the nodal displacement vector  $\bar{\mathbf{q}}^e$ :

$$\bar{\mathbf{K}}^e \bar{\mathbf{q}}^e = \bar{\mathbf{P}}^e \quad (3.47)$$

## 3.6 An Example of the FE Analysis Using the bar Element

After the introduction of the coordinate transformation for the bar element, we should now be able to analyze any bar structure in the 2D problems. Let's use an example to demonstrate the process.

**Example 3.1:** Given the truss structure formed by four bars (Fig. 3.8), the elastic Young's modulus and cross-sectional area are  $E = 29.5 \times 10^4 \text{ N/mm}^2, A = 100 \text{ mm}^2$  for all four bars. The question is to find the nodal displacement, the elemental stress and the reaction force in the truss structure.

**Solution:** Below the FE analysis procedure is used to solve this problem.

### (1) Discretization of the structure

The first step in the FE analysis is to discretize the structure using sufficient elements. In this example, to simplify the calculation, each bar is discretized using only one element. However, in the real engineering problems, a mesh convergence study should be performed to determine the number of elements used for the structure. After the discretization, the numberings of the elements and nodes are defined (Fig. 3.8). Based on the numbering information, the coordinate of every node, the element information and the coordinate transformation information are given in Tables 3.1, 3.2 and 3.3.

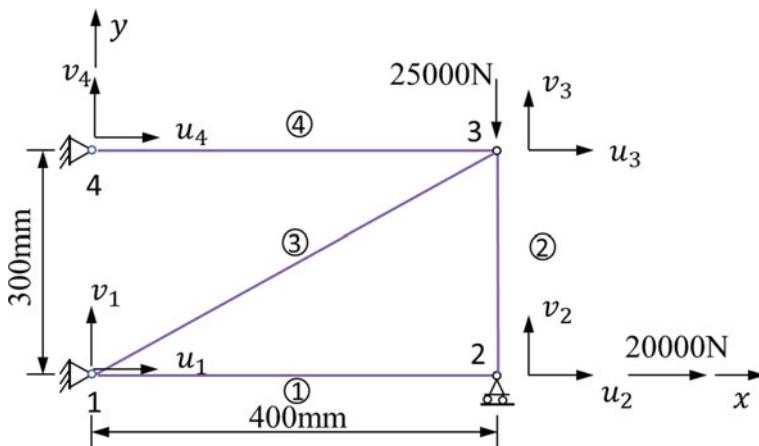


Fig. 3.8 The truss structure formed by four bars

**Table 3.1** Nodes and their coordinates

Node	x(mm)	y(mm)
1	0	0
2	400	0
3	400	300
4	0	300

**Table 3.2** Elements and their associated nodes

Element	Node index	
①	1	2
②	3	2
③	1	3
④	4	3

**Table 3.3** The length of the elements and their cosine values

Element	$l$	$n_x(\cos \alpha)$	$n_y(\sin \alpha)$
①	400	1	0
②	300	0	-1
③	500	0.8	0.6
④	400	1	0

## (2) Analysis on individual element

The key in the analysis of each individual element is to work out the stiffness matrix for each element. Based on Eq. (3.10), the stiffness matrix for each element in the local (elemental) coordinate system can be worked out, and then the transformation matrix for each element can be used to transform the stiffness matrix from the elemental coordinate system to the global coordinate system based on Eq. (3.46)

$$\mathbf{K}^{(1)} = \frac{29.5 \times 10^4 \times 100}{400} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \quad (3.48)$$

$$\mathbf{K}^{(2)} = \frac{29.5 \times 10^4 \times 100}{300} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \quad (3.49)$$

$$\mathbf{K}^{(3)} = \frac{29.5 \times 10^4 \times 100}{500} \begin{bmatrix} 0.64 & 0.48 & -0.64 & -0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix} \quad (3.50)$$

$$\mathbf{K}^{(4)} = \frac{29.5 \times 10^4 \times 100}{400} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \quad (3.51)$$

where  $\mathbf{K}^{(1)}$ ,  $\mathbf{K}^{(2)}$ ,  $\mathbf{K}^{(3)}$  and  $\mathbf{K}^{(4)}$  are the stiffness matrices for the first, the second, the third and the fourth elements, respectively. The number in superscript represents the element number.

### (3) Establishment of the global stiffness equation

To establish the global stiffness equation, the nodal displacement vector, the nodal force vector and the stiffness matrix in the global coordinate system need to be worked out. The nodal displacement vector depends on the number of nodes and the degree of freedom at each node. For the structure given in Fig. 3.8, the nodal displacement vector is given as:

$$\mathbf{q} = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T \quad (3.52)$$

The nodal force vector is the summation of the external forces and the reaction forces at each node, and for the structure given in Fig. 3.8, the nodal force vector is given as:

$$\mathbf{P} = \mathbf{R} + \mathbf{F} = [R_{x1} \ R_{y1} \ 2 \times 10^4 \ R_{y2} \ 0 \ -2.5 \times 10^4 \ R_{x4} \ R_{y4}]^T \quad (3.53)$$

where  $R_{x1}$  and  $R_{y1}$  are the reaction forces at the first node along the  $x$  and  $y$  axes,  $R_{y2}$  is the reaction force at the second node along the  $y$  axis,  $R_{x4}$  and  $R_{y4}$  are the reaction forces at the fourth node along the  $x$  and  $y$  axes.

The global stiffness matrix is formed by assembling the elemental stiffness matrix defined in the global coordinate system, i.e.,

$$\mathbf{K} = \mathbf{K}^{(1)} + \mathbf{K}^{(2)} + \mathbf{K}^{(3)} + \mathbf{K}^{(4)} \quad (3.54)$$

Afterwards, the global stiffness equation can be given as below:

$$\begin{aligned} & \frac{29.5 \times 10^4 \times 100}{6000} \begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 22.68 & 5.76 & -15.0 & 0 \\ -7.68 & -5.76 & 0 & 0 & -20.0 & 5.76 & 24.32 & 0 \\ -5.76 & -4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \\ &= \begin{Bmatrix} R_{x1} \\ R_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ R_{x4} \\ R_{y4} \end{Bmatrix} = \begin{Bmatrix} R_{x1} \\ R_{y1} \\ 2 \times 10^4 \\ R_{y2} \\ 0 \\ -2.5 \times 10^4 \\ R_{x4} \\ R_{y4} \end{Bmatrix} \end{aligned} \quad (3.55)$$

#### (4) Treatment of the boundary condition

In the truss structure given in Fig. 3.8, the displacement boundary conditions are:

$$u_1 = v_1 = v_2 = u_4 = v_4 = 0 \quad (3.56)$$

Putting these conditions back to Eq. (3.55), the global stiffness equation can be simplified as the following form:

$$\frac{29.5 \times 10^4 \times 100}{6000} \begin{bmatrix} 15 & 0 & 0 \\ 0 & 22.68 & 5.76 \\ 0 & 5.76 & 24.32 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} 2 \times 10^4 \\ 0 \\ -2.5 \times 10^4 \end{Bmatrix} \quad (3.57)$$

The equations above can be easily solved and then the displacements at the second and third nodes can be solved:

$$\begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} 0.2712 \\ 0.0565 \\ -0.225 \end{Bmatrix} \quad (3.58)$$

Therefore, the nodal displacement vector for the truss structure is given as below

$$\mathbf{q} = [0 \ 0 \ 0.2712 \ 0 \ 0.0565 \ -0.2225 \ 0 \ 0]^T \quad (3.59)$$

#### (5) Calculation of the stress in each element

The stress in the first element can be solved using the physical equation, i.e.,

$$\sigma^{(1)} = E \mathbf{B} \mathbf{T} \mathbf{q} = \frac{E}{l} [-1 \ 1] \mathbf{T} \mathbf{q} = \frac{29.5 \times 10^4}{400} [-1 \ 0 \ 1 \ 0] \begin{Bmatrix} 0 \\ 0 \\ 0.2712 \\ 0 \end{Bmatrix} = 200 \quad (3.60a)$$

where  $\mathbf{T}$  is the coordinate transformation matrix. Similarly, the stresses in other elements can be solved.

$$\begin{aligned} \sigma^{(2)} &= -218.8 \\ \sigma^{(3)} &= 52.08 \\ \sigma^{(4)} &= 41.67 \end{aligned} \quad (3.60b)$$

### (6) Calculation of the reaction force

Putting the nodal displacement solution back to the global stiffness equation, the reaction force vector can be solved using the equation below:

$$\begin{aligned} \begin{Bmatrix} R_{x1} \\ R_{y1} \\ R_{y2} \\ R_{x4} \\ R_{y4} \end{Bmatrix} &= \frac{29.5 \times 10^4 \times 100}{6000} \begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \\ &\times \begin{Bmatrix} 0 \\ 0 \\ 0.2712 \\ 0 \\ 0.0565 \\ -0.2225 \\ 0 \\ 0 \end{Bmatrix} = \begin{Bmatrix} -15833.0 \\ 3126.0 \\ 21879.0 \\ -4167.0 \\ 0 \end{Bmatrix} \end{aligned}$$

# Chapter 4

## Finite Element Analysis Using Beam Element



### 4.1 Introduction

In the previous chapter, the finite element analysis procedure using the bar element is introduced. However, it should be noted that only the axial loading is allowed in the bar element, which limits its applications in the complex engineering problems. To deal with the complex engineering scenarios, the beam element has to be used. Different from the bar element, the complex loadings, i.e., not only the axial loading but also the bending, are allowed in the beam element. It should be noted that in terms of geometry, there is no difference between the bar and beam elements. Therefore, whether to use the bar or the beam element somehow depends on the external loading. If only the axial loading is applied on the frame structure, then the bar element should be used in the FE analysis to simplify the calculation. Otherwise, the beam element should also be used to consider the deflection in the structure. However, it should be noted that in the real engineering problems, the frame structure under only the axial loading is very rare. Therefore, when the loadings rather than the axial loading can be ignored or relatively small compared to the axial loading, then the frame structure should be analyzed using the bar element. The obvious advantage of using the bar element is that computational efficiency is largely increased.

### 4.2 The Finite Element Calculation Procedure

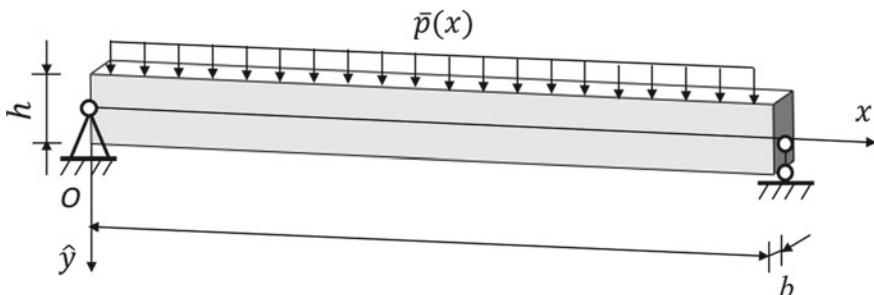
In the previous chapter, the equilibrium equation, the geometric equation, the constitutive equation, etc. of the bar element are used to formulate the FE formulations. Because some equations used to analyze the beam element are very different from those for the bar element, some important preliminary knowledge on the beam element is first reviewed and introduced.

### 4.2.1 Some Preliminary Knowledge on Beam Element

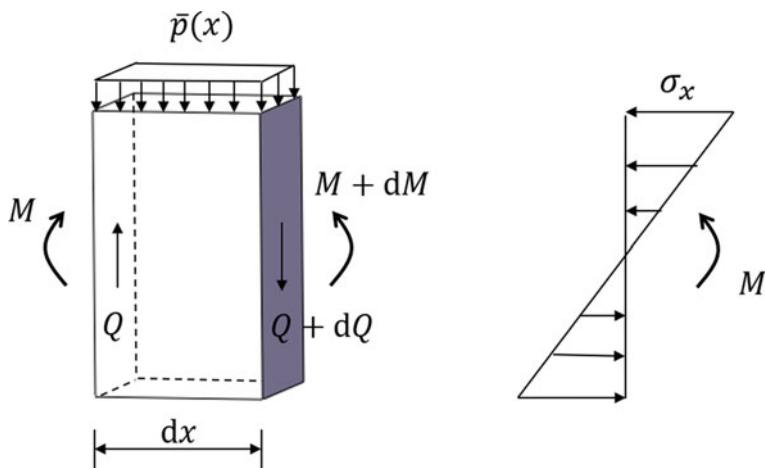
A long beam subject to the distributed loading (Fig. 4.1) is used to introduce the equilibrium equation, the geometric equation and the constitutive equation for the beam. In the beam structure, one end of the beam is fully constrained while the other end is free to move only in the axial direction. The length, the height and the width of the beam are  $l$ ,  $h$  and  $b$ , respectively. A uniformly distributed pressure is applied on the top surface of the beam.

#### (1) Equilibrium equation for the beam element

To work out the equilibrium equation for the beam element, a cross-section of the beam is used (Fig. 4.2). The length of this segment is  $dx$ . The moment and shear force on the left end of the segment are defined as  $M$  and  $Q$ , respectively. Then the moment and shear force on the right end of the segment are  $(M + dM)$  and  $(Q + dQ)$ ,



**Fig. 4.1** A long beam in the equilibrium state



**Fig. 4.2** A representative cross-section of the beam element

respectively. The deflection on the neutral axis, i.e.,  $\hat{y} = 0$  of the beam element is denoted as  $v(x, \hat{y} = 0)$ . Then the equilibrium equations can be obtained as follows: First, the summation of all the forces in the  $x$  direction in the section where moment  $M$  is applied, is zero, i.e.,  $\sum \mathbf{F}_x = 0$ , which can be expanded as:

$$M - \int_A \sigma_x \hat{y} dA = 0 \quad (4.1)$$

where  $dA$  is the cross-sectional area of the representative volume shown in Fig. 4.2.

From the equation above, the following relation can be obtained,

$$M = \int_A \sigma_x \hat{y} dA \quad (4.2)$$

Second, the summation of all the forces in the  $\hat{y}$  direction is zero, i.e.,  $\sum \mathbf{F}_{\hat{y}} = 0$ , which can be written out as:

$$dQ + \bar{p}(x)dx = 0 \quad (4.3)$$

The equation can be rewritten as:

$$\frac{dQ}{dx} + \bar{p} = 0 \quad (4.4)$$

Third, the summation of the moments about any point within the segment is zero, i.e.,  $\sum M_o = 0$ , which can be expanded as:

$$dM - Qdx = 0 \quad (4.5)$$

From which the following relation can be obtained:

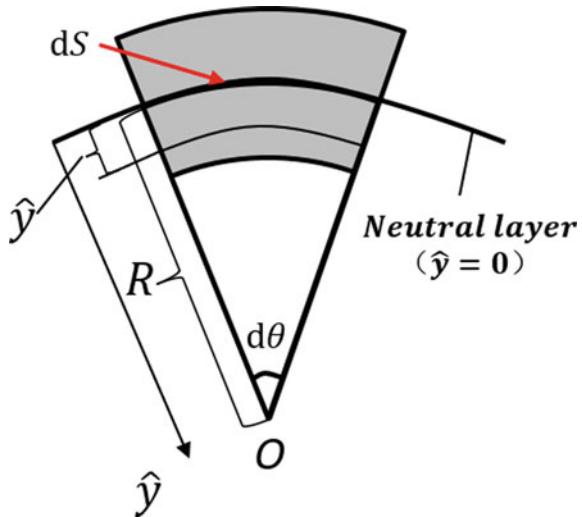
$$Q = \frac{dM}{dx} \quad (4.6)$$

## (2) Geometric equation for the beam element

The geometric equation, i.e., the strain-displacement equation, can be obtained using the cross section of the long beam (Fig. 4.3). Figure 4.3 shows the deformed cross section of the long beam. The distance from the neutral axis to the central point is denoted as  $R$ . The length of the line on the neural layer after the deformation is denoted as  $dS$ . Then based on the definition, the strain on the  $\hat{y}$  layer can be written as below.

$$\varepsilon_x(\hat{y}) = \frac{(R - \hat{y})d\theta - R d\theta}{R d\theta} = -\frac{\hat{y}}{R} \quad (4.7)$$

**Fig. 4.3** Cross section of the long beam under the distributed loading



where  $R$  is the radius of the circle formed by the neutral layer and its relation with the curvature is:

$$\kappa = \frac{d\theta}{dS} = \frac{d\theta}{R d\theta} = \frac{1}{R} = \frac{d^2 v}{dx^2} \quad (4.8)$$

Then, the strain-displacement equation can be obtained as:

$$\varepsilon_x(x, \hat{y}) = -\hat{y} \frac{d^2 v}{dx^2} \quad (4.9)$$

It should be noted that Eq. (4.9) is based on the assumption of small deformation. In the scenario of large deformation, i.e.,  $dS \neq R d\theta$ , and then the formulation for the strain will be different.

### (3) Constitutive equation for the beam element.

The constitutive equation for the beam element can be obtained according to the Hooke's law:

$$\sigma_x = E \varepsilon_x = -E \hat{y} \frac{d^2 v}{dx^2} \quad (4.10)$$

It should be noted that the assumption of isotropic, homogeneous properties of the materials with the elastic and small deformation are assumed in the constitutive equation. Therefore, mechanical properties of the material can be described using the elastic modulus  $E$  and Poisson's ratio. To describe the anisotropic materials or the nonlinear non-elastic mechanical behaviors, other constitutive relations should be applied.

In summary, the following equations are obtained in preparation for the FE formulations of the beam element.

$$M(x) = \int_A \sigma_x \hat{y} dA = \int_A -\hat{y}^2 E \frac{d^2 v}{dx^2} dA = -EI \frac{d^2 v}{dx^2} \quad (4.11)$$

$$\frac{dQ}{dx} + \bar{p} = 0 \quad (4.12)$$

The equations above can also be written as:

$$-EI \frac{d^4 v}{dx^4} + \bar{p}(x) = 0 \quad (4.13a)$$

$$\begin{aligned} \varepsilon_x(x, \hat{y}) &= -\hat{y} \frac{d^2 v}{dx^2} \\ \sigma_x(x, \hat{y}) &= -E \hat{y} \frac{d^2 v}{dx^2} \end{aligned} \quad (4.13b)$$

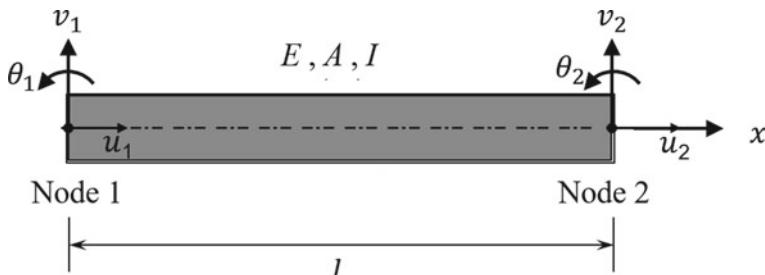
### 4.3 FE Analysis Procedure Using Beam Element

A general beam under the complex loadings (Fig. 4.4) is used to illustrate the FE analysis procedure. The general beam has two nodes (nodes 1 and 2) and each node has three degrees of freedom, i.e., the axial displacement  $u$ , the deflection  $v$  and the rotation  $\theta$ . Therefore, the nodal displacement vector can be written as:

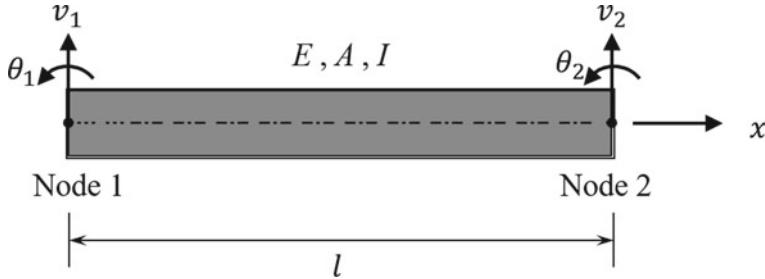
$$\mathbf{q}^e = [v_1 \ \theta_1 \ u_1 \ v_2 \ \theta_2 \ u_2]^T \quad (4.14)$$

where  $v_1, v_2, u_1, u_2, \theta_1$  and  $\theta_2$  are the axial displacements, the deflections and the rotations at the first and second nodes, respectively.

In the analysis of beam element, the axial displacement is assumed to be independent of the deflection and rotation of the beam. Therefore, the nodal displacement vector can be decoupled as



**Fig. 4.4** A representative general beam element



**Fig. 4.5** The beam element with no axial displacement

$$\mathbf{q}^e = [u_1 \ u_2]^T \quad (4.15)$$

and

$$\mathbf{q}^e = [v_1 \ \theta_1 \ v_2 \ \theta_2]^T \quad (4.16)$$

The first displacement vector  $\mathbf{q}^e = [u_1 \ u_2]^T$ , i.e., the scenario of the axial displacement, can be regarded as a problem of bar element and can be solved using the FE formulations for the bar element introduced in Chap. 3. Therefore, only the second displacement vector, i.e.,  $\mathbf{q}^e = [v_1 \ \theta_1 \ v_2 \ \theta_2]^T$  needs to be dealt with in this Chapter. The second displacement vector corresponds to the deflected beam, i.e., the beam with no axial displacement (Fig. 4.5).

In Fig. 4.5, the length of the beam is  $l$ , the cross-sectional area is  $A$ , the Young's modulus is  $E$  and the inertial moment of the cross-section is  $I$ . Corresponding to the nodal displacements, the nodal force vector can be written as:

$$\mathbf{P}^e = [P_{v1} \ M_1 \ P_{v2} \ M_2]^T \quad (4.17)$$

where  $P_{v1}$ ,  $P_{v2}$ ,  $M_1$  and  $M_2$  are the forces and moments at the first and second nodes, respectively. Below, the procedure for deriving the FE formulations for the deflected beam with no axial displacement is introduced.

### (1) The displacement field formulation

Regarding the displacement field formulations for beam element, it should be noted that for the problem given in Fig. 4.5, the deflection and the rotation at the point within the element are not independent. The relation between the deflection and the rotation of the beam is:

$$\theta = \partial v(x)/\partial x \quad (4.18)$$

where  $v(x)$  is the *deflection*,  $\theta$  is the *rotation*. Because of Eq. (4.18), only the displacement field formulation for one variable needs to be worked out and the other variable can be derived from it using Eq. (4.18).

In this chapter, the deflection of the beam is taken as the unknown variable in the displacement field formulation and the rotation of the beam is worked out from the deflection. Therefore, the displacement field formulation for the deflection needs to be worked out. Because when working out the displacement field formulations, the deflections and rotations at the two nodes are taken as the known values and consequently four equations can be set up. Therefore, four unknown coefficients can be set up in the formulation and the displacement field formulation can be written out using the polynomial form as:

$$v(x) = a_0 + a_1x + a_2x^2 + a_3x^3 \quad (4.19)$$

where  $x$  is the coordinate in the  $x$  direction,  $a_0, a_1, a_2, a_3$  are the coefficients to be determined. The displacement boundary conditions are:

$$\begin{aligned} v(0) &= v_1, v'(0) = \theta_1 \\ v(l) &= v_2, v'(l) = \theta_2 \end{aligned} \quad (4.20)$$

Using these equations, the four undetermined coefficients can be solved and written as:

$$\begin{aligned} a_0 &= v_1 \\ a_1 &= \theta_1 \\ a_2 &= \frac{1}{l^2}(-3v_1 - 2\theta_1 l + 3v_2 - \theta_2 l) \\ a_3 &= \frac{1}{l^3}(2v_1 + \theta_1 l - 2v_2 + \theta_2 l) \end{aligned} \quad (4.21)$$

Putting Eq. (4.21) into Eq. (4.19), the displacement field function can be rewritten as:

$$\begin{aligned} v(\xi) &= (1 - 3\xi^2 + 2\xi^3)v_1 + l(\xi - 2\xi^2 + \xi^3)\theta_1 \\ &\quad + (3\xi^2 - 2\xi^3)v_2 + l(\xi^3 - \xi^2)\theta_2 = \mathbf{N}(\xi)\mathbf{q}^e \end{aligned} \quad (4.22)$$

where  $\xi = x/l$  is the *dimensionless coordinate*,  $\mathbf{N}(\xi)$  is the *elemental shape function*. Then based on the relationship between the deflection and rotation, the elemental rotation function can be written as:

$$\theta(x) = \frac{dv(x)}{dx} = \frac{dv}{d\xi} \frac{d\xi}{dx} = \frac{1}{l}\mathbf{N}'(\xi)\mathbf{q}^e \quad (4.23)$$

where  $\mathbf{N}'(\xi)$  is the derivative of the elemental shape function with respective to  $\xi$ .  $\mathbf{N}(\xi)$  and  $\mathbf{N}'(\xi)$  are given as below:

$$\mathbf{N}(\xi) = [(1 - 3\xi^2 + 2\xi^3) \quad l(\xi - 2\xi^2 + \xi^3) \quad (3\xi^2 - 2\xi^3) \quad l(\xi^3 - \xi^2)] \quad (4.24a)$$

$$\mathbf{N}'(\xi) = [(6\xi^2 - 6\xi) \ l(1 - 4\xi + 3\xi^2) \ (6\xi - 6\xi^2) \ l(3\xi^2 - 2\xi)] \quad (4.24b)$$

It should be noted that the beam element is a type of element different from bar element, i.e., the beam is the Hermit element and the bar is the Lagrange element. Therefore, the two properties of the shape function discussed in the previous chapter are not satisfied in the beam element, which can be proved as below:

- (a) The summation of all the shape functions can be written as below:

$$\begin{aligned} \sum_{i=1}^4 N_i &= (1 - 3\xi^2 + 2\xi^3) + l(\xi - 2\xi^2 + \xi^3) \\ &\quad + (3\xi^2 - 2\xi^3) + l(\xi^3 - \xi^2) = 1 + l(\xi - 3\xi^2 + 2\xi^3) \end{aligned} \quad (4.25)$$

It can be seen the sum of the four shape functions not equals to one.

- (b) The values of the shape function at the elemental nodes can be worked out as below (Table 4.1):

It can be seen the value of shape function  $N_1$  at the first node is one and its value at the second node is zero; the value of shape function  $N_3$  at the second node is one and its value at the first node is zero. Rather than these two shape functions, other shape functions do not have these properties.

## (2) The strain formulation

The strain within the beam element can be worked out using the geometric equation, i.e.

$$\begin{aligned} \varepsilon(x, \hat{y}) &= -\hat{y} \frac{d^2v}{dx^2} \\ &= -\hat{y} \left[ \frac{1}{l^2}(12\xi - 6), \frac{1}{l}(6\xi - 4), -\frac{1}{l^2}(12\xi - 6), \frac{1}{l}(6\xi - 2) \right] \mathbf{q}^e \end{aligned}$$

**Table 4.1** Values of shape functions at elemental nodes

Shape function	Node 1 ( $\xi = 0$ )	Node 2 ( $\xi = 1$ )
$N_1 = (1 - 3\xi^2 + 2\xi^3)$	1	0
$N_2 = l(\xi - 2\xi^2 + \xi^3)$	0	0
$N_3 = (3\xi^2 - 2\xi^3)$	0	1
$N_4 = l(\xi^3 - \xi^2)$	0	0
$N'_1 = (6\xi^2 - 6\xi)$	0	0
$N'_2 = l(1 - 4\xi + 3\xi^2)$	$l$	0
$N'_3 = (6\xi - 6\xi^2)$	0	0
$N'_4 = l(3\xi^2 - 2\xi)$	0	$l$

$$= \mathbf{B}(\xi) \mathbf{q}^e \quad (4.26)$$

where  $\mathbf{B}(\xi)$  is the *strain–displacement matrix* of the element and can be rewritten as

$$\mathbf{B}(\xi) = -\hat{y}[B_1 \ B_2 \ B_3 \ B_4] \quad (4.27)$$

where

$$B_1 = \frac{1}{l^2}(12\xi - 6)$$

$$B_2 = \frac{1}{l}(6\xi - 4)$$

$$B_3 = -\frac{1}{l^2}(12\xi - 6)$$

$$B_4 = \frac{1}{l}(6\xi - 2)$$

It can be seen from Eq. (4.26) that the strain within the beam element is a linear function of both  $\xi$  and  $\hat{y}$ .

### (3) The stress formulation

The stress formulation within the beam element can be derived using the constitutive relation, i.e.

$$\sigma(x, \hat{y}) = E\varepsilon(x, \hat{y}) = E\mathbf{B}(x, \hat{y})\mathbf{q}^e = \mathbf{S}(x, \hat{y})\mathbf{q}^e \quad (4.28)$$

where  $\mathbf{S}(\xi)$  is the *stress–displacement matrix* for the element and can be rewritten as

$$\mathbf{S}(\xi) = -\hat{y}[S_1 \ S_2 \ S_3 \ S_4]$$

where

$$S_1 = E\frac{1}{l^2}(12\xi - 6), \quad S_2 = E\frac{1}{l}(6\xi - 4)$$

$$S_3 = -E\frac{1}{l^2}(12\xi - 6), \quad S_4 = E\frac{1}{l}(6\xi - 2)$$

It can be seen from Eq. (4.28) that the stress within the beam element is a function of both  $x$  and  $\hat{y}$ , and it has the similar distribution pattern as that of the strain for the linear elastic, isotropic solid.

#### (4) The potential energy

The potential energy within the beam element can be written as

$$\Pi^e = U^e - W^e \quad (4.29)$$

where  $W^e$  is the *external work*,  $U^e$  is the *strain (internal) energy* and can be written as

$$\begin{aligned} U^e &= \frac{1}{2} \int_0^l \int_A \sigma(x, \hat{y}) \varepsilon(x, \hat{y}) dA dx \\ &= \frac{1}{2} \mathbf{q}^{eT} \left[ \int_0^l \int_A \mathbf{B}^T E \mathbf{B} dA dx \right] \mathbf{q}^e \\ &= \frac{1}{2} \mathbf{q}^{eT} \mathbf{K}^e \mathbf{q}^e \end{aligned}$$

where  $\mathbf{K}^e$  is the *elemental stiffness matrix* and can be written as:

$$\begin{aligned} \mathbf{K}^e &= \int_0^l \int_A (-\hat{y}) \begin{bmatrix} B_1 \\ B_2 \\ B_3 \\ B_4 \end{bmatrix} E \begin{bmatrix} B_1 & B_2 & B_3 & B_4 \end{bmatrix} (-\hat{y}) dA dx \\ &= E \int_A (-\hat{y})^2 dA \int_0^l \begin{bmatrix} B_1^2 & B_1 B_2 & B_1 B_3 & B_1 B_4 \\ B_1 B_2 & B_2^2 & B_2 B_3 & B_2 B_4 \\ B_1 B_3 & B_2 B_3 & B_3^2 & B_3 B_4 \\ B_1 B_4 & B_2 B_4 & B_3 B_4 & B_4^2 \end{bmatrix} dx \\ &= E \frac{I_Z}{l^3} \begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -6l & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix} \end{aligned}$$

where  $I_Z$  is the moment of inertia and  $I_Z = \int_A \hat{y}^2 dA$ .  
In Eq. (4.29),  $W^e$  can be written as:

$$W^e = P_{v1}v_1 + M_1\theta_1 + P_{v2}v_2 + M_2\theta_2 = \mathbf{P}^{eT} \mathbf{q}^e \quad (4.30)$$

where  $\mathbf{P}^{eT}$  is the *elemental nodal force vector* and  $\mathbf{P}^{eT} = [P_{v1} \ M_1 \ P_{v2} \ M_2]^T$ .

### (5) The elemental stiffness equation

The elemental stiffness equation for the deflected beam with no axial displacement can be written as:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \quad (4.31)$$

where  $\mathbf{K}^e$  is the elemental stiffness matrix,  $\mathbf{q}^e$  is the elemental nodal displacement vector and  $\mathbf{P}^e$  is the elemental nodal force vector.

After the deflected beam with no axial displacement is solved, we can now move forward to the FE formulations for a general beam. For a general beam, the elemental stiffness equation should be expanded to also include the components associated with the axial displacement. In this case, the elemental stiffness equation can be written as:

$$\mathbf{K}'^e \mathbf{q}'^e = \mathbf{P}'^e \quad (4.32)$$

where  $\mathbf{q}'^e = [u_1 \ v_1 \ \theta_1 \ u_2 \ v_2 \ \theta_2]^T$  and  $\mathbf{P}'^e = [P_{u1} \ P_{v1} \ M_1 \ P_{u2} \ P_{v2} \ M_2]^T$  and  $\mathbf{K}'^e$  is a  $6 \times 6$  matrix given as below

$$\mathbf{K}'^e = \begin{bmatrix} \frac{EA}{l} & 0 & 0 & -\frac{EA}{l} & 0 & 0 \\ 0 & \frac{12EI}{l^3} & \frac{6EI}{l^2} & 0 & -\frac{12EI}{l^3} & \frac{6EI}{l^2} \\ 0 & \frac{6EI}{l^2} & \frac{4EI}{l} & 0 & -\frac{6EI}{l^2} & \frac{2EI}{l} \\ -\frac{EA}{l} & 0 & 0 & \frac{EA}{l} & 0 & 0 \\ 0 & -\frac{12EI}{l^3} & -\frac{6EI}{l^2} & 0 & \frac{12EI}{l^3} & -\frac{6EI}{l^2} \\ 0 & \frac{6EI}{l^2} & \frac{2EI}{l} & 0 & -\frac{6EI}{l^2} & \frac{4EI}{l} \end{bmatrix}$$

where  $I$  is the *moment of inertia*,  $E$  is the elastic modulus,  $A$  and  $l$  are the cross-sectional area and length of the beam, respectively.

It should be noted that when deriving the formulations for the elemental solutions, e.g., the elemental stiffness matrix, the local (elemental) coordinate system is used and consequently the stiffness matrix derived is independent of the global coordinates. Therefore, when setting up the elemental stiffness equation, the stiffness matrix can be directly used and thus the procedure for deriving the stiffness matrix given in this part can be skipped. However, it is still important to fully understand the FE calculation procedures in case some new FE elements need to be solved in the future.

## 4.4 Calculation of the Elemental Equivalent Nodal Forces

Different from the bar element, in which there is only the axial displacement and only the axial force is allowed, the external forces on the beam element can be in different forms, e.g., the pressure over the entire beam, the concentrated force at the center of the beam, etc. When the external forces are not at the elemental nodes, the

corresponding equivalent nodal forces have to be worked out to integrate them into the stiffness equation. In this part, the calculation of the elemental equivalent nodal forces is introduced and an example is used to illustrate the calculation process using the virtual work principle.

Considering a 2-node beam element (Fig. 4.6), the external force vector  $(P_x \ P_y \ m)$  is applied at the middle point of the element. Because the external force is not at the nodes, the equivalent nodal forces need to be worked out.

Because each node of the beam has three force components, the equivalent nodal force vector can be written out as below:

$$\mathbf{F}_d^e = [F_{xi} \ F_{yi} \ m_i \ F_{xj} \ F_{yj} \ m_j]^T \quad (4.33)$$

Virtual displacements are then prescribed on all the degrees of freedom, i.e.,

$$\mathbf{q}^{*e} = [u_i^* \ v_i^* \ \theta_i^* \ u_j^* \ v_j^* \ \theta_j^*]^T \quad (4.34)$$

The virtual displacement at the point where the external force is applied can be given as below:

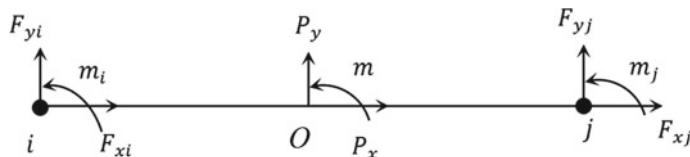
$$\mathbf{U}^* = [u_o \ v_o \ \theta_o]^T \quad (4.35)$$

where  $u_o$  is the axial displacement along the beam,  $v_o$  is the deflection of the beam at  $O$  and  $\theta_o$  is the rotation at the point.

Based on the formulations given in Chap. 3, the relationship between the axial displacement at the two nodes and the axial displacement at the point  $O$  can be written out as below:

$$u_o = N_1^u u_i^* + N_2^u u_j^* = \left[ 1 - \frac{x}{l} \ \frac{x}{l} \right] \begin{Bmatrix} u_i^* \\ u_j^* \end{Bmatrix} \quad (4.36)$$

The relationship between the deflections at the two nodes and that at the middle point of the beam can be worked out as:



**Fig. 4.6** The calculation of the equivalent nodal forces

$$v_o = N_1 v_i^* + N_2 \theta_i^* + N_3 v_j^* + N_4 \theta_j^* = [N_1 \ N_2 \ N_3 \ N_4] \begin{Bmatrix} v_i^* \\ \theta_i^* \\ v_j^* \\ \theta_j^* \end{Bmatrix} \quad (4.37)$$

Similarly, the relationship between the rotations at the two nodes and that at the middle point of the beam can be worked out as:

$$\theta_o = \frac{1}{l} (N'_1 v_i^* + N'_2 \theta_i^* + N'_3 v_j^* + N'_4 \theta_j^*) = \frac{1}{l} [N'_1 \ N'_2 \ N'_3 \ N'_4] \begin{Bmatrix} v_i^* \\ \theta_i^* \\ v_j^* \\ \theta_j^* \end{Bmatrix} \quad (4.38)$$

Equations (4.36), (4.37) and (4.38) can be re-written in the matrix format as below:

$$\mathbf{U}^* = \begin{Bmatrix} u_o \\ v_o \\ \theta_o \end{Bmatrix} = \begin{bmatrix} N_1^u & 0 & 0 & N_2^u & 0 & 0 \\ 0 & N_1 & N_2 & 0 & N_3 & N_4 \\ 0 & \frac{1}{l} N'_1 & \frac{1}{l} N'_2 & 0 & \frac{1}{l} N'_3 & \frac{1}{l} N'_4 \end{bmatrix} \begin{Bmatrix} \theta_o \\ v_i^* \\ \theta_i^* \\ u_j^* \\ v_j^* \\ \theta_j^* \end{Bmatrix} \quad (4.39)$$

where the shape functions are given as below:

$$N_1^u = 1 - \xi, N_2^u = \xi$$

$$N_1 = 1 - 3\xi^2 + 2\xi^3, N_2 = l(\xi - 2\xi^2 + \xi^3)$$

$$N_3 = 3\xi^2 - 2\xi^3, N_4 = l(\xi^3 - \xi^2)$$

The virtual work at the nodes can be written out as:

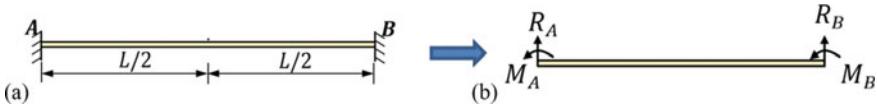
$$W_d = (\mathbf{q}^{*e})^T \mathbf{F}_d^e \quad (4.40)$$

The virtual work at the non-nodes is given as below:

$$W_p = \mathbf{U}^{*T} \mathbf{P} = (\mathbf{N} \mathbf{q}^{*e})^T \mathbf{P} \quad (4.41)$$

Based on the virtual work principle, i.e., the virtual work done at the nodes equal to the virtual work done at the non-nodes, the following equation can be obtained:

$$(\mathbf{q}^{*e})^T \mathbf{F}_d^e = (\mathbf{q}^{*e})^T \mathbf{N}^T \mathbf{P} \quad (4.42)$$



**Fig. 4.7** Illustration of the calculation of the equivalent nodal forces

where  $\mathbf{N}^T$  is the shape function matrix. Equation (4.42) can be further written as:

$$\mathbf{F}_d^e = \mathbf{N}^T \mathbf{P} \quad (4.43)$$

The equation above can be used to transfer the non-nodal concentrated forces into the nodal forces. However, Eq. (4.43) is only applicable to the scenario of concentrated non-nodal forces. In cases of other types of non-nodal forces, e.g., the pressure load, the equivalent nodal forces can be obtained using the virtual work principle following the same procedure. Here, the formulation for the equivalent nodal forces of the pressure loading in the beam is directly given as below:

$$\mathbf{F}_d^e = \int_l \mathbf{N}^T \mathbf{p} dx \quad (4.44)$$

In addition to the two loading scenarios given above, many different types of external non-nodal forces may be present in the structure, e.g., an off-center concentrated force, a non-uniformly distributed pressure. No matter what form the external non-nodal forces are, the same principle, i.e., the virtual displacement, is used to work out the equivalent nodal forces. Below in Fig. 4.7 and Table 4.2, the equivalent nodal forces for the beam under various different external loadings are illustrated. The two ends of the beam are fully constrained (Fig. 4.7a). Four different external loadings in the vertical direction (Table 4.2), i.e., the concentrated loading, the uniformly distributed and the triangularly distributed loadings, are applied in the beam individually. Before the calculation, we can know the equivalent nodal force vector in the beam can be written as:

$$\mathbf{F}^e = [R_A \ M_A \ R_B \ M_B] \quad (4.45)$$

where  $R_A$  and  $R_B$  are the reaction forces at nodes A and B, respectively, in which the force from the bottom to the top is defined as the positive value;  $M_A$  and  $M_B$  are the reaction moments at nodes A and B, respectively, in which the moment in the counter-clockwise direction is defined as the positive value.

Based on Eq. (4.44), the equivalent nodal forces can be worked out as:

$$\mathbf{F}^e = [R_A \ M_A \ R_B \ M_B] = \int_l \bar{p}(x) \mathbf{N}(x) dx \quad (4.46)$$

**Table 4.2** The equivalent nodal forces for the beams under different external loadings

Supports and external forces	The equivalent nodal forces
	$R_A = -P/2$ $R_B = -P/2$ $M_A = -PL/8$ $M_B = PL/8$
	$R_A = -(Pb^2/L^3)(3a + b)$ $R_B = -(Pa^2/L^3)(a + 3b)$ $M_A = -Pab^2/L^2$ $M_B = Pa^2b/L^2$
	$R_A = -\bar{p}_0L/2$ $R_B = -\bar{p}_0L/2$ $M_A = -\bar{p}_0L^2/12$ $M_B = \bar{p}_0L^2/12$
	$R_A = -3\bar{p}_0L/20$ $R_B = -7\bar{p}_0L/20$ $M_A = -\bar{p}_0L^2/30$ $M_B = \bar{p}_0L^2/20$

where  $\bar{p}(x)$  is the external loading and  $\mathbf{N}(x)$  is the shape function for the beam element with no axial displacement, i.e., the one given in Eq. (4.24).

Now we should be able to use the FE analysis to solve the beam under various external loadings. Below an example is used to demonstrate the FE analysis procedure using beam element.

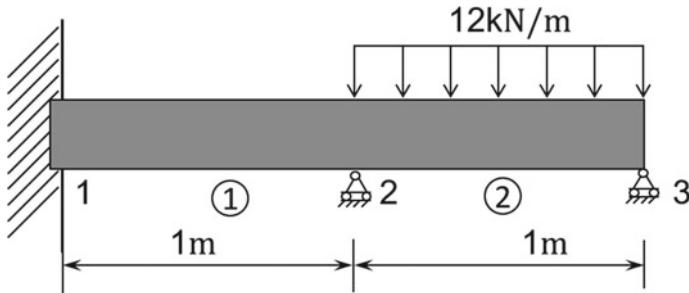
#### Example 4.1: FE calculation using beam element.

A beam structure is shown in Fig. 4.8. The left end of the structure is fully constrained and a uniformly distributed pressure (12.0 kN/m) is applied on the right half of the beam structure. The length of the beam structure is 2.0 m. The Young's modulus and the moment of inertial of the beam are  $E = 200$  GPa and  $I = 4 \times 10^{-6}$  m<sup>4</sup>, respectively. The question is to work out the nodal displacement in the beam structure.

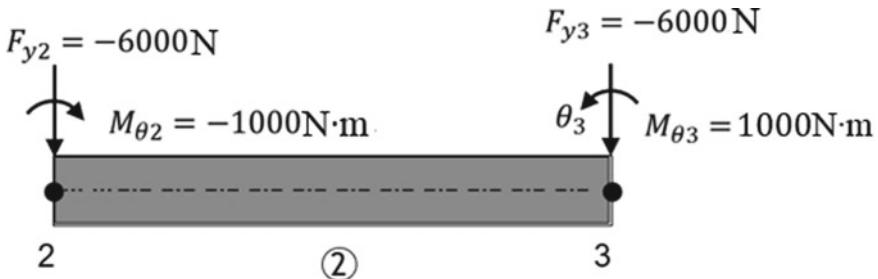
#### Solution:

##### (1) Discretization of the structure and numbering

For the demonstration purpose, two elements are used to mesh the structure. However, it should be noted that in the real engineering analysis, the mesh convergence should be performed to determine the number of the element used in the analysis. The element and node numbering are shown in Fig. 4.8. It is obvious that there is no axial displacement. Therefore, there are two degrees



**Fig. 4.8** A beam structure under the external pressure



**Fig. 4.9** Calculation of the equivalent nodal forces for the second element

of freedom of each node and the nodal displacement vector can be written as:

$$\mathbf{q} = [v_1 \ \theta_1 \ v_2 \ \theta_2 \ v_3 \ \theta_3]^T$$

where  $v_1, \theta_1, v_2, \theta_2, v_3$  and  $\theta_3$  are the deflections and rotations at the first, the second and the third nodes, respectively.

Regarding the nodal force vector, the equivalent nodal forces at the second element can be calculated using Eq. (4.44) (Fig. 4.9), then the nodal force vector for the structure can be worked out as below:

$$\begin{aligned} \mathbf{P} &= [R_{y1} \ M_{\theta 1} \ R_{y2} + F_{y2} \ M_{\theta 2} \ R_{y3} + F_{y3} \ M_{\theta 3}]^T \\ &= [R_{y1} \ M_{\theta 1} \ R_{y2} - 6000 \ -1000 \ R_{y3} - 6000 \ 1000]^T \end{aligned}$$

## (2) The elemental stiffness matrix

The elemental stiffness matrix can be worked out based on Eq. (4.32). Here, the answers for each element are directly given.

For the first element:

$$\mathbf{K}^{(1)} = 8 \times 10^5 \begin{bmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 2 & -6 & 4 \end{bmatrix}$$

For the second element:

$$\mathbf{K}^{(2)} = 8 \times 10^5 \begin{bmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 2 & -6 & 4 \end{bmatrix}$$

### (3) Establishment of the global stiffness equation

The global stiffness equation can be written as:

$$\mathbf{K}\mathbf{q} = \mathbf{P}$$

where  $\mathbf{K}$  is the *global stiffness matrix* and given as below:

$$\mathbf{K} = \mathbf{K}^{(1)} + \mathbf{K}^{(2)}$$

It should be noted that the equation above means the assembly of the elemental stiffness matrix, i.e., the summation between two matrices, which should be performed based on the corresponding of the relevant displacement components. The global stiffness equation can be written as below:

$$8 \times 10^5 \times \begin{bmatrix} 12 & 6 & -12 & 6 & 0 & 0 \\ 6 & 4 & -6 & 2 & 0 & 0 \\ -12 & -6 & 24 & 0 & -12 & 6 \\ 6 & 2 & 0 & 8 & -6 & 2 \\ 0 & 0 & -12 & -6 & 12 & -6 \\ 0 & 0 & 6 & 2 & -6 & 4 \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} R_{y1} \\ M_{\theta 1} \\ R_{y2} - 6000 \\ -1000 \\ R_{y3} - 6000 \\ 1000 \end{Bmatrix}$$

### (4) Treatment of the boundary condition and calculation of the stiffness equation

To enable to calculate the stiffness equation by hand calculation, the boundary conditions should be introduced into the stiffness equations to simplify the calculations. The displacement boundary conditions of this problem are:

$$v_1 = 0, \theta_1 = 0, v_2 = 0, v_3 = 0$$

Putting these conditions into the global stiffness equation, the reduced dimension of the global stiffness equation can be obtained as below:

$$8 \times 10^5 \times \begin{bmatrix} 8 & 2 \\ 2 & 4 \end{bmatrix} \begin{Bmatrix} \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -1000 \\ 1000 \end{Bmatrix}$$

Solving the stiffness equation above, the answers for the unknown displacements can be obtained, i.e.,

$$\theta_2 = -2.68 \times 10^{-4}, \quad \theta_3 = 4.46 \times 10^{-4}$$

### (5) Calculation of the displacements within the element

Based on Eq. (4.22), the deflection field function in the second element can be written as:

$$\begin{aligned} v^{(2)}(x) = \mathbf{N}(x)\mathbf{q}^{(2)} &= N_1(x)v_2 + N_2(x)\theta_2 + N_3(x)v_3 + N_4(x)\theta_3 \\ &= (1 - 3\xi^2 + 2\xi^3)v_2 + l(\xi - 2\xi^2 + \xi^3)\theta_2 \\ &\quad + (3\xi^2 - 2\xi^3)v_3 + l(\xi^3 - \xi^2)\theta_3 \end{aligned}$$

where  $\xi$  is the normalized coordinate in the second element. Therefore, the deflection at the middle point of the element can be worked out as below:

$$v^{(2)}\left(\xi = \frac{1}{2}\right) = -8.93 \times 10^{-5} \text{ m}$$

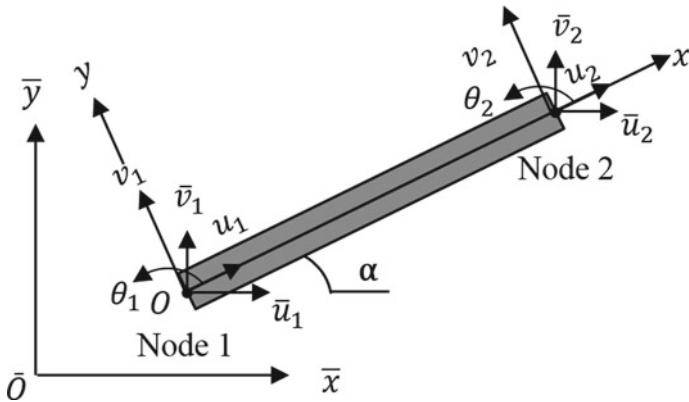
It should be noted after the calculation of the global stiffness equation, i.e., the nodal displacement vector, not only the deflection in the second element has been solved, but also other variables, e.g., the strain and stress within the element have been worked out. Therefore, the assembly of the global stiffness equation and the calculation of the stiffness equation are the key steps in the FE analysis.

## 4.5 Coordinate Transformation in the Beam Analysis

Similar to the bar element, when analyzing the beam element in the real engineering problems, the beam element can be placed in any orientation in the global coordinate. In this case, the stiffness equation needs to be set up in the global coordinate system.

Below, an example is used to illustrate the transformation of the variables from the local coordinate system to the global coordinate system. As shown in Fig. 4.10, a 2-node beam element is placed in the plane where its long axis is aligned to the  $x$  axis with a degree of  $\alpha$ . The length of the beam is  $l$ , Young's modulus is  $E$ , cross-sectional area is  $A$  and the initial moment is  $I$ .

The local coordinate system ( $Oxy$ ) is aligned along the long axis of the beam element (Fig. 4.10), i.e.,  $x$  is along the length direction and  $y$  is perpendicular to the



**Fig. 4.10** Illustration of the coordinate transformation in the beam element

beam. In the local coordinate system, there are three degrees of freedom of each node and thus the nodal displacement vector can be written as:

$$\mathbf{q}^e = [u_1 \ v_1 \ \theta_1 \ u_2 \ v_2 \ \theta_2]^T$$

where  $u_1, v_1, \theta_1, u_2, v_2$  and  $\theta_2$  are the axial displacements, the deflections and the rotations at the first and second nodes, respectively. In the global coordinate system  $\bar{O}xy$ , there are three degrees of freedom of each node. It should be noted that the elemental rotation is independent of the coordinate system and thus the rotation maintains the same in the two systems. Consequently, the nodal displacement vector in the global coordinate system can be written out as:

$$\bar{\mathbf{q}}^e = [\bar{u}_1 \ \bar{v}_1 \ \theta_1 \ \bar{u}_2 \ \bar{v}_2 \ \theta_2]^T$$

where  $\bar{u}_1, \bar{v}_1, \theta_1, \bar{u}_2, \bar{v}_2$  and  $\theta_2$  are the axial displacements, the deflections and rotations in the global coordinate system at the first and the second nodes, respectively. The variables with a bar on top of them represent the variables in the global coordinate system.

Based on the principle that the displacement vector for the same node should be equivalent in the two coordinate systems (Fig. 4.10), the relation between the displacement vectors, i.e., the axial displacement and the deflection, in the local coordinate system and those in the global coordinate system can be written as,

$$\begin{cases} u_1 = \bar{u}_1 \cos \alpha + \bar{v}_1 \sin \alpha \\ v_1 = -\bar{u}_1 \sin \alpha + \bar{v}_1 \cos \alpha \\ u_2 = \bar{u}_2 \cos \alpha + \bar{v}_2 \sin \alpha \\ v_2 = -\bar{u}_2 \sin \alpha + \bar{v}_2 \cos \alpha \end{cases}$$

In the matrix format, it can be written as:

$$\mathbf{q}^e = \mathbf{T}^e \bar{\mathbf{q}}^e$$

where  $\mathbf{T}^e$  is the elemental coordinate transformation matrix and given as:

$$\mathbf{T}^e = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 & 0 & 0 \\ -\sin \alpha & \cos \alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos \alpha & \sin \alpha & 0 \\ 0 & 0 & 0 & -\sin \alpha & \cos \alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Using the coordinate transformation matrix  $\mathbf{T}^e$ , the elemental stiffness equation in the global coordinate system can be written as:

$$\bar{\mathbf{K}}^e \bar{\mathbf{q}}^e = \bar{\mathbf{P}}^e$$

where  $\bar{\mathbf{K}}^e = \mathbf{T}^{eT} \mathbf{K}^e \mathbf{T}^e$ , and  $\bar{\mathbf{P}}^e = \mathbf{T}^{eT} \mathbf{P}^e$ . The superscript ‘e’ means the variables are defined in the element.

After the introduction of the coordinate transformation and the nodal equivalent forces, we now should be able to solve any frame structure using the beam element. Below an example is used to demonstrate the calculation process.

**Example 4.2:** Finite element calculations using beam element.

A plane frame structure formed by 3 beams is shown in Fig. 4.11. A uniformly distributed force (4167.0 N/m) is applied at the top beam and a concentrated force (3000 N) is applied at the first node of the top beam along the  $x$  direction. The

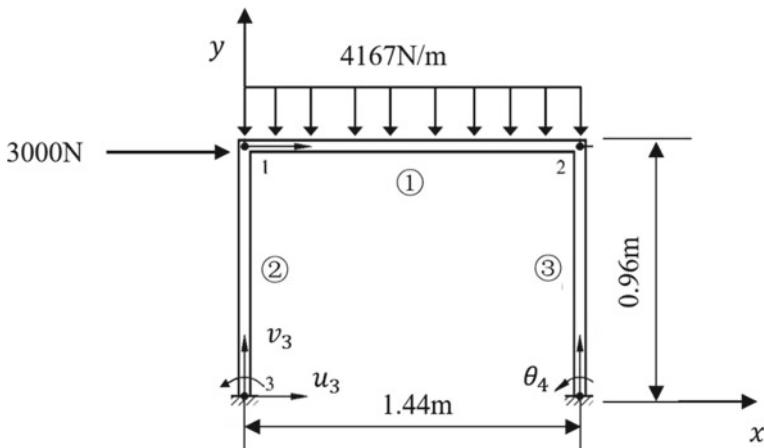
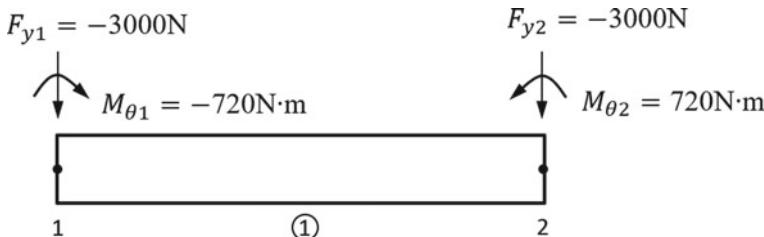


Fig. 4.11 A 3-beam frame structure under the uniform pressure

**Table 4.3** Numbering for the element and node

Element	Node number	
①	1	2
②	3	1
③	4	2

**Fig. 4.12** The equivalent nodal forces on the first element

dimension of the structure is shown in the figure and other properties of the frame structure are:  $E = 3.0 \times 10^{11}\text{Pa}$ ,  $I = 6.5 \times 10^{-7}\text{m}^4$ ,  $A = 6.8 \times 10^{-4}\text{m}^2$ . The question is to find the displacements in the structure.

**Solution:**

(1) **Structural discretization and numbering**

To simplify the calculation, the frame structure is discretized using only three beam elements, and the numbering of the element and node is shown in Fig. 4.11. The association between the elements and nodes is shown in Table 4.3.

Because the pressure is distributed over the top surface of the first element, to establish the global stiffness matrix, the non-nodal forces have to be transformed to the nodal forces. Following the procedure given in Sect. 4.2, the equivalent nodal forces can be worked out as below (Fig. 4.12), i.e.,  $F = -3000\text{ N}$  and  $M = -720\text{ N}\cdot\text{m}$  at the first node and  $F = -3000\text{ N}$  and  $M = -720\text{ N}\cdot\text{m}$  at the second node.

There are four nodes in the structure and each node has three degrees of freedom. The nodal displacement vector in the global coordinate system can be written as:

$$\mathbf{q}^e = [u_1 \ v_1 \ \theta_1 \ u_2 \ v_2 \ \theta_2 \ u_3 \ v_3 \ \theta_3 \ u_4 \ v_4 \ \theta_4]^T$$

In the structure, there are external forces only at the first and second nodes. Therefore, the external nodal force vector can be written as:

$$\mathbf{F} = [F_{x1} \ F_{y1} \ M_{\theta 1} \ 0 \ F_{y2} \ M_{\theta 2} \ 0 \ 0 \ 0 \ 0 \ 0 \ 0]^T$$

In the structure, the third and fourth nodes are constrained and the reaction forces will be generated at these two nodes. Therefore, the nodal reaction force vector can be written as:

$$\mathbf{R} = [0 \ 0 \ 0 \ 0 \ 0 \ R_{x3} \ R_{y3} \ R_{\theta3} \ R_{x4} \ R_{y4} \ R_{\theta4}]^T$$

where  $R_{x3}$ ,  $R_{y3}$  and  $R_{\theta3}$  are the reaction force along the  $x$  direction, the reaction force along the  $y$  direction, and the reaction moment at the first node, respectively.  $R_{x4}$ ,  $R_{y4}$ ,  $R_{\theta4}$  are the reaction force along the  $x$  direction, the reaction force along the  $y$  direction, and the reaction moment at the fourth node, respectively.

The total nodal force vector is the summation of the external force vector and the reaction force vector. Therefore, the nodal force vector in the global coordinate system can be written as:

$$\mathbf{P} = \mathbf{F} + \mathbf{R} = [3000 \ -3000 \ -720 \ 0 \ -3000 \ 720 \ R_{x3} \ R_{y3} \ R_{\theta3} \ R_{x4} \ R_{y4} \ R_{\theta4}]$$

## (2) Finite element analysis on individual element

Now the stiffness matrix for individual element in the global coordinate system needs to be worked out, which will then be assembled to form the global stiffness matrix.

Regarding the first element, the local coordinate system agrees with the global coordinate system, and thus the elemental stiffness matrix can be worked out as:

$$\mathbf{K}^{(1)} = 10^6 \times \begin{bmatrix} 141.7 & 0 & 0 & -141.7 & 0 & 0 \\ 0 & 0.78 & 0.56 & 0 & -0.78 & 0.56 \\ 0 & 0.56 & 0.54 & 0 & -0.56 & 0.27 \\ -141.7 & 0 & 0 & 141.7 & 0 & 0 \\ 0 & -0.78 & -0.56 & 0 & 0.78 & -0.56 \\ 0 & 0.56 & 0.27 & 0 & -0.56 & 0.54 \end{bmatrix}$$

Regarding the second and third elements, they have the same orientation with respect to the global coordinate system, but different numbering. The elemental stiffness matrix for the second and third elements in the local coordinate system can be worked out as:

$$\hat{\mathbf{K}}^{(2)} = 10^6 \times \begin{bmatrix} 212.5 & 0 & 0 & -212.5 & 0 & 0 \\ 0 & 2.65 & 1.27 & 0 & -2.65 & 1.27 \\ 0 & 1.27 & 0.81 & 0 & -1.27 & 0.41 \\ -212.5 & 0 & 0 & 212.5 & 0 & 0 \\ 0 & -2.65 & -1.27 & 0 & 2.65 & -1.27 \\ 0 & 1.27 & 0.41 & 0 & -1.27 & 0.81 \end{bmatrix}$$

Based on Fig. 4.10, the coordinate transformation matrix can be worked out as:

$$\mathbf{T} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Therefore, the elemental stiffness matrix for the second and third elements in the global coordinate system can be worked out as below:

$$\mathbf{K}^{(2)} = \mathbf{T}^T \hat{\mathbf{K}}^{(2)} \mathbf{T} = 10^6 \times \begin{bmatrix} 2.65 & 0 & -1.27 & -2.65 & 0 & -1.27 \\ 0 & 212.5 & 0 & 0 & -212.5 & 0 \\ -1.27 & 0 & 0.81 & 1.27 & 0 & 0.41 \\ -2.65 & 0 & 1.27 & 2.65 & 0 & 1.27 \\ 0 & -212.5 & 0 & 0 & 212.5 & 0 \\ -1.27 & 0 & 0.41 & 1.27 & 0 & 0.81 \end{bmatrix}$$

The nodal displacement vectors for the second and third elements can be given as below:

$$\mathbf{q}^{(2)} = [u_3 \ v_3 \ \theta_3 \ u_1 \ v_1 \ \theta_1]^T$$

$$\mathbf{q}^{(3)} = [u_4 \ v_4 \ \theta_4 \ u_2 \ v_2 \ \theta_2]^T$$

### (3) Establishment of the global stiffness equation

After the completion of previous steps, the elemental stiffness matrix in the global coordinate system can now be assembled and the global stiffness equation can be set up as below:

$$\mathbf{Kq} = \mathbf{P}$$

where  $\mathbf{K}$  is the global stiffness matrix and assembled from the elemental stiffness matrices using the following equation:

$$\mathbf{K} = \mathbf{K}^{(1)} + \mathbf{K}^{(2)} + \mathbf{K}^{(3)}$$

where  $\mathbf{K}^{(1)}, \mathbf{K}^{(2)}$  and  $\mathbf{K}^{(3)}$  are the stiffness matrices for the first, second and third elements in the global coordinate system, respectively. They are assembled based on the corresponding of the nodal displacement vector .

#### (4) Treatment of the boundary condition and the solution

To make the calculation easier, the boundary conditions are introduced into the global stiffness equation to simplify the equation. It should be noted that in the commercial FE software and the computer programming, a different approach may be followed to introduce the boundary conditions into the stiffness equation, which will be described in more details in Sect. 4.4.

For the problems given in Fig. 4.10, the displacement boundary conditions are:

$$u_3 = v_3 = \theta_3 = u_4 = v_4 = \theta_4 = 0$$

Putting these conditions into the global stiffness equation, the reduced form of the stiffness equation can be obtained:

$$10^6 \times \begin{bmatrix} 144.3 & 0 & 1.27 & -141.7 & 0 & 0 \\ 0 & 213.3 & 0.56 & 0 & -0.78 & 0.56 \\ 1.27 & 0.56 & 1.35 & 0 & -0.56 & 0.27 \\ -141.7 & 0 & 0 & 144.3 & 0 & 1.27 \\ 0 & -0.78 & -0.56 & 0 & 213.3 & -0.56 \\ 0 & 0.56 & 0.27 & 1.27 & -0.56 & 1.35 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} 3000 \\ -3000 \\ -720 \\ 0 \\ -3000 \\ 720 \end{Bmatrix}$$

Solving the stiffness equation above, the nodal displacement vector for the structure can be obtained:

$$u_1 = 0.92 \text{ mm}, v_1 = -0.10 \text{ mm}, \theta_1 = -1.39 \times 10^{-5} \text{ rad}$$

$$u_2 = 0.90 \text{ mm}, v_2 = -0.02 \text{ mm}, \theta_2 = 3.88 \times 10^{-5} \text{ rad}$$

## 4.6 Treatment of the Boundary Conditions

As mentioned before, in the FE calculation, the boundary conditions have to be introduced into the stiffness equation to be dealt with. In the previous parts, because the hand calculation is used, the boundary conditions of zero displacements are introduced into the stiffness equation to reduce the stiffness matrix. However, this method is generally not used in the FE programming. Furthermore, this method cannot be used to deal with other types of boundary conditions, e.g., the prescribed displacement on the boundary. In this part, three methods are introduced to deal with the general displacement boundary conditions.

Before introducing the three methods, first the two types of displacement boundary conditions are reviewed:

The first type is the zero displacement boundary condition, i.e., the displacement degrees of freedom are fully constrained on certain boundaries. This boundary condition can be written using the equation below:

$$\bar{\mathbf{q}}_a = 0 \quad (4.47)$$

where  $\bar{\mathbf{q}}_a$  is the displacement vector at node  $a$ , which can be given as below for a beam element,

$$\mathbf{q}_a = [u_1 \ v_1 \ \theta_1]^T \quad (4.48)$$

The second type is the prescribed displacements on certain boundaries, i.e., the displacements on certain boundaries are prescribed with certain values. In terms of equations, this boundary condition can be written as:

$$\bar{\mathbf{q}}_a = \bar{\mathbf{u}} \quad (4.49)$$

The global stiffness equation considering the boundary conditions can be written as:

$$\begin{bmatrix} \mathbf{K}_{aa} & \mathbf{K}_{ab} \\ \mathbf{K}_{ba} & \mathbf{K}_{bb} \end{bmatrix} \begin{Bmatrix} \bar{\mathbf{q}}_a \\ \mathbf{q}_b \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_a \\ \bar{\mathbf{P}}_b \end{Bmatrix} \quad (4.50)$$

where  $\mathbf{K}_{ij}$ ,  $\bar{\mathbf{q}}_a$  and  $\bar{\mathbf{P}}_b$  are the known parameters,  $\mathbf{q}_b$  is the unknown nodal displacement and  $\mathbf{P}_a$  is the nodal force to be solved. Now the three methods to deal with different boundary conditions are introduced.

#### (1) The direct method

The direct method can be used to deal with the first type of boundary condition, i.e.,  $\bar{\mathbf{q}}_a = 0$ . Using this method, the corresponding lines and columns in Eq. (4.50) are first eliminated, and then the following reduced form of equation can be obtained:

$$\mathbf{K}_{bb}\mathbf{q}_b = \bar{\mathbf{P}}_b \quad (4.51)$$

Then the unknown nodal displacement vector can be solved as below:

$$\mathbf{q}_b = \mathbf{K}_{bb}^{-1}\bar{\mathbf{P}}_b \quad (4.52)$$

The direct method can also be used to deal with the second type of boundary condition, i.e.,  $\bar{\mathbf{q}}_a = \bar{\mathbf{u}}$ . Using this method, Eq. (4.50) can be rewritten as:

$$\mathbf{K}_{aa}\bar{\mathbf{q}}_a + \mathbf{K}_{ab}\mathbf{q}_b = \mathbf{P}_a \quad (4.53)$$

$$\mathbf{K}_{ba}\bar{\mathbf{q}}_a + \mathbf{K}_{bb}\mathbf{q}_b = \bar{\mathbf{P}}_b \quad (4.54)$$

Putting the condition  $\bar{\mathbf{q}}_a = \bar{\mathbf{u}}$  into Eq. (4.54), the following equation can be obtained

$$\mathbf{K}_{bb}\mathbf{q}_b = \bar{\mathbf{P}}_b - \mathbf{K}_{ba}\bar{\mathbf{u}} \quad (4.55)$$

Then the unknown nodal displacement can be solved as:

$$\mathbf{q}_b = \mathbf{K}_{bb}^{-1}(\bar{\mathbf{P}}_b - \mathbf{K}_{ba}\bar{\mathbf{u}}) \quad (4.56)$$

Several advantages and disadvantages regarding the direct method should be noted. First, the direct method can be used for both types of displacement boundary conditions. Second, the process of using the direct method is very obvious and clear. Third, the direct method can reduce the dimension of the stiffness matrix and consequently it is more suitable for the hand calculation. Fourth, when using the direct method, the numbering and ordering of the node and element in the stiffness matrix are changed, and thus this method is not suitable for the computer programming.

## (2) Set ‘1’ method

It should be noted that the set ‘1’ method is only applicable for the first type of displacement boundary condition, i.e.,  $\bar{q}_r = 0$ , where  $\bar{q}_r$  is the  $r$ -th degree of freedom in the system. Supposing the  $r$ -th displacement is zero, i.e.,  $\bar{q}_r = 0$ , to use the set ‘1’ method, the corresponding element in the stiffness matrix, i.e.,  $k_{rr}$  is set to 1, i.e.,  $k_{rr} = 1$ , and other components in the  $r$ -th row and the  $r$ -th column are set to zeros, i.e.,  $k_{rs} = k_{sr} = 0(r \neq s)$ . Meanwhile, the corresponding element in the force vector is set to zero, i.e.,  $p_r = 0$ . Then the following equation can be obtained

$$\begin{bmatrix} k_{11} & \cdots & 0 & \cdots & k_{1n} \\ \vdots & & \vdots & & \vdots \\ \ddots & & \ddots & & \vdots \\ \vdots & & \vdots & & \vdots \\ 0 & \cdots & k_{rr} = 1 & \cdots & 0 \\ \vdots & & \vdots & & \vdots \\ \ddots & & \ddots & & \vdots \\ \vdots & & \vdots & & \vdots \\ k_{n1} & \cdots & 0 & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} q_1 \\ \vdots \\ \bar{q}_r \\ \vdots \\ q_n \end{Bmatrix} = \begin{Bmatrix} p_1 \\ \vdots \\ p_r = 0 \\ \vdots \\ p_n \end{Bmatrix} \quad (4.57)$$

where  $n$  is the total number of the degrees of freedom in the system.

The equation above is equivalent to the original equation with the application of the boundary condition  $\bar{q}_r = 0$ . In Eq. (4.57), if only considering the elements in the  $r$ -th row, the corresponding equation can be written as:

$$k_{rr}\bar{q}_r = p_r \quad (4.58)$$

Because  $k_{rr} = 1$ ,  $p_r = 0$ , the equation above is equivalent to the following equation:

$$\bar{q}_r = 0$$

In other rows rather than the  $r$ -th row, because the elements in the  $r$ -th column (except for  $k_{rr}$ ) are set to zeros, the  $\bar{q}_r = 0$  has already been taken into account into the equations. From the calculation process above, it can be seen the boundary condition  $\bar{q}_r = 0$  has been well dealt with using Eq. (4.57).

The advantage of the set ‘1’ method is that the dimension of the original matrix maintains the same during the calculation, no re-ordering is required, and thus the method is good for the computer programming.

### (3) Multiply a big number method

In this method, the  $r$ -th displacement is prescribed with a given value, i.e.,  $\bar{q}_a = \bar{u}$ . To consider this condition into the global stiffness equation, the corresponding element in the global stiffness matrix, i.e.,  $k_{rr}$ , is first multiplied by a big value,  $\alpha$ . Additionally, the corresponding element in the load vector, i.e.,  $p_r$ , is set to  $k_{rr}\bar{u}$ . The global stiffness equation can then be written as below:

$$\begin{bmatrix} k_{11} & \cdots & 0 & \cdots & k_{1n} \\ \vdots & & \vdots & & \vdots \\ \ddots & & \ddots & & \vdots \\ \vdots & & \vdots & & \vdots \\ k_{r1} & \cdots & \alpha k_{rr} & \cdots & k_{rn} \\ \vdots & & \vdots & & \vdots \\ \ddots & & \ddots & & \vdots \\ \vdots & & \vdots & & \vdots \\ k_{n1} & \cdots & 0 & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} q_1 \\ \vdots \\ \bar{q}_r \\ \vdots \\ q_n \end{Bmatrix} = \begin{Bmatrix} p_1 \\ \vdots \\ \alpha k_{rr}\bar{u} \\ \vdots \\ p_n \end{Bmatrix} \quad (4.59)$$

After these settings, Eq. (4.59) is equivalent to the original equation with the application of the boundary condition,  $\bar{q}_a = \bar{u}$ . Now let’s prove the equivalence. The equation from the  $r$ -th row can be written as:

$$k_{r1}q_1 + k_{r2}q_2 + \cdots + \alpha k_{rr}\bar{q}_r + \cdots + k_nq_n = \alpha k_{rr}\bar{u} \quad (4.60)$$

Because  $\alpha k_{rr} \gg k_{ri}$  ( $i = 1, 2, 3, \dots, r - 1, r, r + 1, \dots, n$ ), the equation above can be re-written as:

$$\alpha k_{rr}\bar{q}_r \approx \alpha k_{rr}\bar{u}$$

i.e.,

$$\bar{q}_r \approx \bar{u} \quad (4.61)$$

which means the displacement condition given in Eq. (4.49) has been taken into account. In the rows except for the  $r$ -th row, because  $\bar{q}_r \approx \bar{u}$  is set, thus this displacement condition has been taken into account in the corresponding equations. Therefore, the condition  $\bar{q}_r \approx \bar{u}$  has been well taken into account in Eq. (4.59).

It should be noted that the multiply a big number method can be used to deal with both types of the displacement boundary conditions. Second, the dimension of the matrix to be solved maintains the same and no re-ordering is required, and thus this method is suitable for computer programming. Third, the symmetric property of the global stiffness matrix is maintained, which makes the calculations easier.

At the end of this part, it should be noted that the three methods introduced above are for dealing with the displacement boundary conditions. Regarding the force boundary conditions, the prescribed forces can be transferred to the equivalent nodal forces and then directly added into the nodal force vector in the stiffness equation.

# Chapter 5

## Finite Element Analysis Using Triangular Element



### 5.1 Introduction

In the previous two chapters, the FE analysis procedures using the bar and beam elements are presented. These two types of elements are used to analyze the structure formed by the bars and beams. In the real complex engineering problems, most structures are formed not only by the bars and beams, but also by the plates, etc. When the structures are formed by the 2D surfaces and the 2D elements, such as the triangular and rectangular, have to be used. In the present and the next chapters, the FE analysis procedures using the triangular and rectangular elements are presented.

### 5.2 FE Analysis Procedure Using Triangular Element

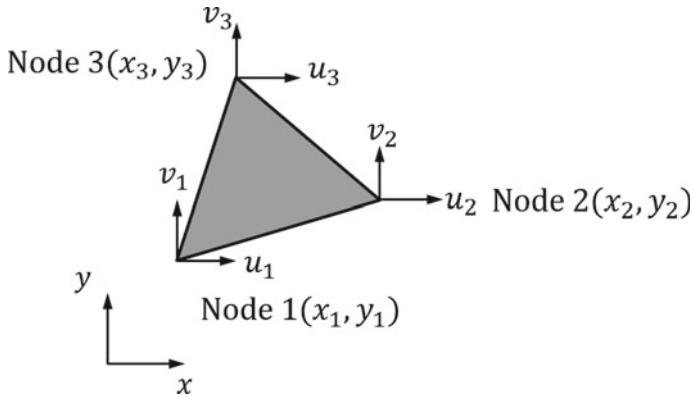
A 3-node triangular element with the thickness of  $t$  is used to illustrate the finite element analysis procedure using the triangular element. It should be noted that the 3-node triangular element is the simplest type of triangular element. In Chap. 8, the FE procedure using more complicated triangular elements is presented. It should be noted that no matter what type of element it is, the FE analysis procedure is the same. Below the FE procedure presented in the previous chapters is followed to solve the triangular element (Fig. 5.1).

For the 3-node triangular element, there are two degrees of freedom at each node, i.e., the displacements in the  $x$  and  $y$  directions, respectively. Therefore, the displacement nodal vector can be written as:

$$\mathbf{q}^e = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3]^T \quad (5.1)$$

The corresponding nodal force vector can be written as:

$$\mathbf{P}^e = [P_{x1} \ P_{y1} \ P_{x2} \ P_{y2} \ P_{x3} \ P_{y3}]^T \quad (5.2)$$



**Fig. 5.1** Illustration of a triangular element

As mentioned in the previous chapters, when setting up the displacement field formulation, the variables in the displacement nodal vector are taken as the known values. In the 3-node triangular element, there are 6 values in the displacement nodal vector. Therefore, 6 unknown coefficients in the displacement field formulations can be solved. In the triangular element, every node within the triangular element has the displacements in the  $x$  and  $y$  directions. Therefore, the displacement field formulations in the  $x$  and  $y$  directions can be given as below:

$$u(x, y) = \bar{a}_0 + \bar{a}_1x + \bar{a}_2y \quad (5.3a)$$

$$v(x, y) = \bar{b}_0 + \bar{b}_1x + \bar{b}_2y \quad (5.3b)$$

where  $u$  and  $v$  represent the displacements in the  $x$  and  $y$  directions, respectively,  $\bar{a}_0, \bar{a}_1, \bar{a}_2$  and  $\bar{b}_0, \bar{b}_1, \bar{b}_2$  are the coefficients to be solved.

In Eq. (5.3), the values at the elemental nodes are taken as the knowns to solve the 6 coefficients.

$$\left. \begin{array}{l} u(x_i, y_i) = u_i \\ v(x_i, y_i) = v_i \end{array} \right\} \quad (i = 1, 2, 3) \quad (5.4)$$

The answers for the coefficients  $\bar{a}_0, \bar{a}_1$  and  $\bar{a}_2$  can be obtained by solving the equations above:

$$\begin{aligned}
 \bar{a}_0 &= \frac{1}{2A} \begin{vmatrix} u_1 & x_1 & y_1 \\ u_2 & x_2 & y_2 \\ u_3 & x_3 & y_3 \end{vmatrix} = \frac{1}{2A}(a_1u_1 + a_2u_2 + a_3u_3) \\
 \bar{a}_1 &= \frac{1}{2A} \begin{vmatrix} 1 & u_1 & y_1 \\ 1 & u_2 & y_2 \\ 1 & u_3 & y_3 \end{vmatrix} = \frac{1}{2A}(b_1u_1 + b_2u_2 + b_3u_3) \\
 \bar{a}_2 &= \frac{1}{2A} \begin{vmatrix} 1 & x_1 & u_1 \\ 1 & x_2 & u_2 \\ 1 & x_3 & u_3 \end{vmatrix} = \frac{1}{2A}(c_1u_1 + c_2u_2 + c_3u_3)
 \end{aligned} \tag{5.5}$$

Similarly

$$\begin{aligned}
 \bar{b}_0 &= \frac{1}{2A}(a_1v_1 + a_2v_2 + a_3v_3) \\
 \bar{b}_1 &= \frac{1}{2A}(b_1v_1 + b_2v_2 + b_3v_3) \\
 \bar{b}_2 &= \frac{1}{2A}(c_1v_1 + c_2v_2 + c_3v_3)
 \end{aligned} \tag{5.6}$$

where

$$A = \frac{1}{2} \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix} = \frac{1}{2}(a_1 + a_2 + a_3) = \frac{1}{2}(b_1c_2 - b_2c_1) \tag{5.7a}$$

$$\begin{aligned}
 a_1 &= \begin{vmatrix} x_2 & y_2 \\ x_3 & y_3 \end{vmatrix} = x_2y_3 - x_3y_2 \\
 b_1 &= -\begin{vmatrix} 1 & y_2 \\ 1 & y_3 \end{vmatrix} = y_2 - y_3 \\
 c_1 &= \begin{vmatrix} 1 & x_2 \\ 1 & x_3 \end{vmatrix} = -x_2 + x_3
 \end{aligned} \tag{5.7b}$$

where  $A$  is the area of the triangular element. Other parameters, i.e.,  $a_2$ ,  $b_2$ ,  $c_2$ ,  $a_3$ ,  $b_3$  and  $c_3$  can be obtained by  $1 \rightarrow 2$ ,  $2 \rightarrow 3$ ,  $3 \rightarrow 1$ .

Then the nodal displacement formulations can be rewritten as

$$u(x, y) = N_1(x, y)u_1 + N_2(x, y)u_2 + N_3(x, y)u_3 \tag{5.8a}$$

$$v(x, y) = N_1(x, y)v_1 + N_2(x, y)v_2 + N_3(x, y)v_3 \tag{5.8b}$$

In terms of the matrix format

$$\mathbf{u}(x, y) = \begin{Bmatrix} u(x, y) \\ v(x, y) \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \mathbf{N}(x, y)\mathbf{q}^e \quad (5.9)$$

where  $N$  is the matrix of the shape function given as below

$$\mathbf{N}(x, y) = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \quad (5.10a)$$

$$N_i = \frac{1}{2A}(a_i + b_i x + c_i y) \quad (i = 1, 2, 3) \quad (5.10b)$$

The strain formulation for the triangular element can be worked out using the strain-displacement relation, i.e.,

$$\boldsymbol{\varepsilon}(x, y) = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u(x, y) \\ v(x, y) \end{Bmatrix} = [\partial]\mathbf{u} \quad (5.11)$$

where  $[\partial]$  is the operator matrix of the strain-displacement equation.

Putting the displacement formulation into Eq. (5.11), the following equation can be obtained

$$\boldsymbol{\varepsilon}(x, y) = [\partial]\mathbf{N}(x, y)\mathbf{q}^e = \mathbf{B}(x, y)\mathbf{q}^e \quad (5.12)$$

where  $\mathbf{B}(x, y)$  is the strain-displacement matrix and is defined as below:

$$\mathbf{B}(x, y) = [\partial]\mathbf{N} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \quad (5.13)$$

Putting the formulation for the shape function into Eq. (5.13), the following relation can be obtained

$$\mathbf{B}(x, y) = \frac{1}{2A} \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix} = [\mathbf{B}_1 \ \mathbf{B}_2 \ \mathbf{B}_3] \quad (5.14)$$

where

$$\mathbf{B}_i = \frac{1}{2A} \begin{bmatrix} b_i & 0 \\ 0 & c_i \\ c_i & b_i \end{bmatrix} \quad (i = 1, 2, 3) \quad (5.15)$$

The stress formulation for the triangular element can be obtained using the stress-strain relation:

$$\sigma(x, y, z) = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} = \mathbf{D}\varepsilon \quad (5.16)$$

where  $\mathbf{D}$  is the elasticity matrix. For the plane stress problem, it can be written as

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad (5.17)$$

where  $E$  is the Young's modulus and  $\nu$  is the Poisson's ratio of the material.

For the plane strain problem,  $\mathbf{D}$  can be written as:

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad (5.18)$$

with

$$\sigma_z = \nu(\sigma_x + \sigma_y)$$

Putting the strain formulation into the Eq. (5.16), the following relation can be obtained

$$\sigma = \mathbf{DBq}^e = \mathbf{Sq}^e \quad (5.19)$$

where  $\mathbf{S}$  is the stress-displacement matrix.

The elemental potential energy of the triangular element can be written as

$$\begin{aligned} \prod^e &= \frac{1}{2} \int_{\Omega^e} \sigma^T \varepsilon d\Omega - \left[ \int_{\Omega^e} \bar{\mathbf{b}}^T \mathbf{u} d\Omega + \int_{S_p^e} \bar{\mathbf{p}}^T \mathbf{u} dA \right] \\ &= \frac{1}{2} \mathbf{q}^{eT} \left( \int_{\Omega^e} \mathbf{B}^T \mathbf{DB} d\Omega \right) \mathbf{q}^e - \left( \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA \right)^T \mathbf{q}^e \\ &= \frac{1}{2} \mathbf{q}^{eT} \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^{eT} \mathbf{q}^e \end{aligned} \quad (5.20)$$

where  $\mathbf{K}^e$  is the stiffness matrix for the element, i.e.,

$$\mathbf{K}^e = \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega = \int_{A^e} \mathbf{B}^T \mathbf{D} \mathbf{B} dA \cdot t \quad (5.21)$$

where  $t$  is the thickness of the triangular element and  $\mathbf{B}$  is the constant coefficient matrix.

The elemental stiffness equation can be obtained by taking the derivative of the potential energy with respect to the nodal displacement vector, i.e.

$$\partial \prod / \partial \mathbf{q}^e = 0 \quad (5.22)$$

from which, the stiffness equation can be obtained as below:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \quad (5.23)$$

where  $\mathbf{K}^e$  is the stiffness matrix given in Eq. (5.21),  $\mathbf{q}^e$  is the nodal displacement vector,  $\mathbf{P}^e$  is the elemental equivalent nodal force vector, i.e.

$$\begin{aligned} \mathbf{P}^e &= \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA \\ &= \int_{A^e} \mathbf{N}^T \bar{\mathbf{b}} t dA + \int_{t_p^e} \mathbf{N}^T \bar{\mathbf{p}} t dl \end{aligned} \quad (5.24)$$

### 5.3 Properties of the Shape Function for Triangular Element

Shape function is an important part for understanding the finite element. Regarding the shape function for the triangular element, there are three important properties, the first two of which are the same as those of the bar elements.

**Property 1:** At node  $i$ , the corresponding shape function equals to one and the values of other shape functions are zero. i.e.,  $N_i = 1$ ,  $N_j = 0$ ,  $N_m = 0$

This property can be proved as follows. The shape function at the first node can be written as:

$$\begin{aligned} N_1(x_1, y_1) &= \frac{1}{2A}(a_1 + b_1x_1 + c_1y_1) \\ &= \frac{1}{2A}(a_1 + b_1x_1 + c_1y_1) \end{aligned}$$

$$\begin{aligned}
 &= \frac{1}{2A} [a_1 + (y_2 - y_3)x_1 + (x_3 - x_2)y_1] \\
 &= \frac{1}{2A} [a_1 + a_2 + a_3] = 1
 \end{aligned} \tag{5.25}$$

Similarly, the values for the same shape function ( $N_1$ ) at the second and third nodes can be obtained as below:

$$N_1(x_2, y_2) = 0 \tag{5.26a}$$

$$N_1(x_3, y_3) = 0 \tag{5.26b}$$

Similarly, the shape functions  $N_2$  and  $N_3$  at the first, second and third nodes can be worked out as:

$$N_2(x_1, y_1) = 0, N_2(x_2, y_2) = 1, N_2(x_3, y_3) = 0 \tag{5.27a}$$

$$N_3(x_1, y_1) = 0, N_3(x_2, y_2) = 0, N_3(x_3, y_3) = 1 \tag{5.27b}$$

The shape functions  $N_1$ ,  $N_2$ . and  $N_3$  can be plotted within the triangular region as below (Fig. 5.2):

**Property 2:** At any location within the triangular element, the summation of the three shape functions equals to one, i.e.,  $N_1 + N_2 + N_3 = 1$

This property can be proved as below:

$$\begin{aligned}
 &N_1(x, y) + N_2(x, y) + N_3(x, y) \\
 &= \frac{1}{2A} (a_1 + b_1x + c_1y + a_2 + b_2x + c_2y + a_3 + b_3x + c_3y) \\
 &= \frac{1}{2A} [a_1 + a_2 + a_3 + (b_1 + b_2 + b_3)x + (c_1 + c_2 + c_3)y] \\
 &= \frac{1}{2A} (2A + 0x + 0y) = 1
 \end{aligned} \tag{5.28}$$

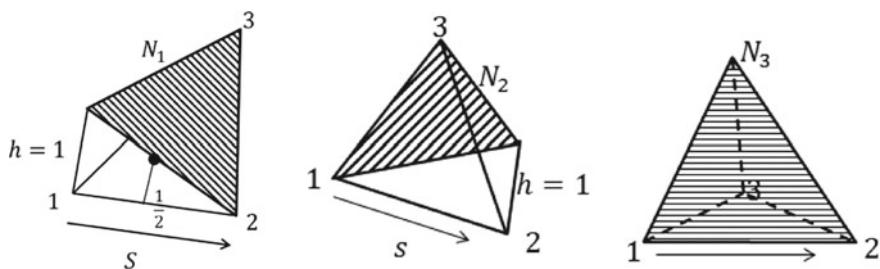


Fig. 5.2 Plotting of the three shape functions for triangular element

The property implies that only two of the three shape functions are independent. Second, this property reflects the rigid displacement of the triangular element. For example, when the rigid displacement occurs, i.e.,  $u_1 = u_2 = u_3 = u^*$ , the displacement at any point within the element can be written as  $u = N_1u_1 + N_2u_2 + N_3u_3 = u^*$ , which reflects the second property.

**Property 3:** On any edge of the triangular element, the shape function is only determined by the nodes of this edge. For example, on the edge  $\vec{12}$ , the shape functions  $N_1$  and  $N_2$  are only associated with the first and second nodes and are independent of the third node. After the calculation, the shape functions  $N_1$  and  $N_2$  can be expressed using the coordinates of the first and second nodes as below:

$$\begin{aligned}N_1(x, y) &= 1 - \frac{x - x_1}{x_2 - x_1}, \\N_2(x, y) &= \frac{x - x_1}{x_2 - x_1}, \\N_3(x, y) &= 0\end{aligned}\tag{5.29}$$

The formulation for  $N_2$  can be proved using Fig. 5.3

Based on the line equation for the line  $\vec{12}$ , the following equation can be obtained

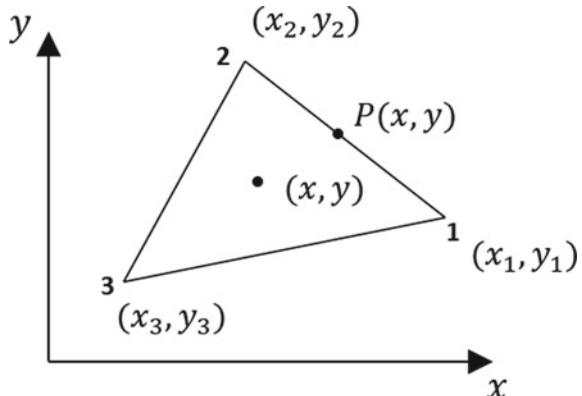
$$\frac{x - x_1}{x_2 - x_1} = \frac{y - y_1}{y_2 - y_1}\tag{5.30}$$

From the equation above, the following relation can be obtained

$$y = y_1 + \frac{y_2 - y_1}{x_2 - x_1}(x - x_1) = y_1 - \frac{b_3}{c_3}(x - x_1)\tag{5.31}$$

On the other hand, the shape function  $N_2$  can be written as

**Fig. 5.3** Analysis of the property of shape function on the edge of the triangular



$$\begin{aligned}
N_2(x, y) &= \frac{1}{2A}(a_2 + b_2x + c_2y) \\
&= \frac{1}{2A} \left[ a_2 + b_2x + c_2(y_1 + \frac{-b_3}{c_3}(x - x_1)) \right] \\
&= \frac{1}{2A} [a_2 + b_2x_1 - b_2x_1 + b_2x + c_2 \left( y_1 - \frac{b_3}{c_3}(x - x_1) \right)] \\
&= \frac{1}{2A} [a_2 + b_2x_1 + c_2y_1 + b_2(x - x_1) - \frac{c_2b_3}{c_3}(x - x_1)] \\
&= \frac{1}{2A} \left( \frac{b_2c_3 - c_2b_3}{c_3} \right) (x - x_1)
\end{aligned} \tag{5.32}$$

$$\begin{aligned}
b_2c_3 - c_2b_3 &= b_2(x_2 - x_1) + b_3(x_3 - x_1) \\
&= b_2x_2 + b_3x_3 + b_1x_1 - (b_1 + b_2 + b_3)x_1 \\
&= b_1x_1 + b_2x_2 + b_3x_3 \\
&= x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2) \\
&= x_2y_3 - x_3y_2 + x_3y_1 - x_1y_3 + x_1y_2 - x_2y_1 \\
&= a_1 + a_2 + a_3 \\
&= 2A
\end{aligned} \tag{5.33}$$

Then the formulation for  $N_2$  can be simplified as

$$N_2(x, y) = \frac{1}{2A} \frac{2A}{c_3} (x - x_1) = \frac{1}{c_3} (x - x_1) = \frac{x - x_1}{x_2 - x_1} \tag{5.34}$$

Similarly

$$N_1(x, y) = 1 - N_2(x, y) - N_3(x, y) = 1 - \frac{x - x_1}{x_2 - x_1} \tag{5.35}$$

Furthermore, the formulation for  $N_3$  can be worked out as

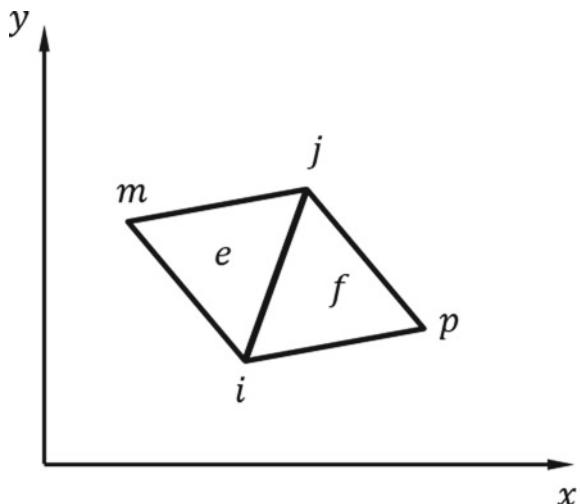
$$N_3(x, y) = 0 \tag{5.36}$$

The third property of the triangular element implies that the displacement on the common edge is continuous. This can be proved using Fig. 5.4 as follows. At any point on the shared edge  $\vec{ij}$ , the displacements in the  $x$  and  $y$  directions can be written using the shape functions and the nodal displacement values as below:

$$u(x, y) = N_i(x, y)u_i + N_j(x, y)u_j \tag{5.37a}$$

$$v(x, y) = N_i(x, y)v_i + N_j(x, y)v_j \tag{5.37b}$$

**Fig. 5.4** Analysis of the displacement on the shared edge



Based on the third property of the triangular element, the following equations can be obtained:

$$N_m(x, y) = N_p(x, y) = 0 \quad (5.38)$$

$N_i(x, y)$  and  $N_j(x, y)$  are the functions of the coordinates for the nodes  $i$  and  $j$ . Therefore,  $u$  and  $v$  on the shared edge only depend on the nodes  $i$  and  $j$ , i.e., the displacement on the common edge of triangular elements is continuous.

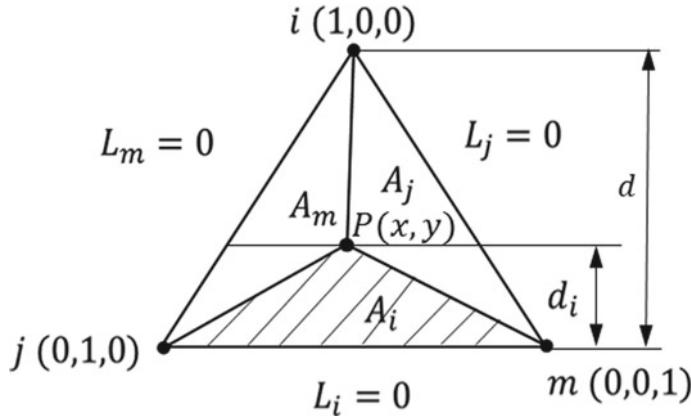
## 5.4 The Area Coordinate

In the previous parts, the Cartesian coordinate system is used, in which the calculation of the shape function is complicated and it is even more complicated when the order of the element is increased. To simplify the calculation of the shape function for the triangular element, the area coordinate is introduced in this part. The triangular shown in Fig. 5.5 is used to illustrate the area coordinate

The area of the triangular can be written as:

$$A = \frac{1}{2} \begin{vmatrix} 1 & x_i & y_i \\ 1 & x_j & y_j \\ 1 & x_m & y_m \end{vmatrix} \quad (5.39)$$

A point  $P(x, y)$  within the triangular partitions the area into three parts,  $A_i$ ,  $A_j$  and  $A_m$ , the areas of which can be worked out as:



**Fig. 5.5** Illustration of the area coordinate in the triangular element

$$\begin{aligned}
 A_i &= \frac{1}{2} \begin{vmatrix} 1 & x & y \\ 1 & x_j & y_j \\ 1 & x_m & y_m \end{vmatrix} = \frac{1}{2}(a_i + b_i x + c_i y) \\
 A_j &= \frac{1}{2}(a_j + b_j x + c_j y) \\
 A_m &= \frac{1}{2}(a_m + b_m x + c_m y)
 \end{aligned} \tag{5.40}$$

Below these areas are used to define the area coordinates  $L_i$ ,  $L_j$  and  $L_m$ :

$$\begin{aligned}
 L_i &= \frac{A_i}{A} = \frac{1}{2A}(a_i + b_i x + c_i y) \\
 L_j &= \frac{A_j}{A} = \frac{1}{2A}(a_j + b_j x + c_j y) \\
 L_m &= \frac{A_m}{A} = \frac{1}{2A}(a_m + b_m x + c_m y)
 \end{aligned} \tag{5.41}$$

It can be seen that the area coordinates are defined using the subareas and the total area of the triangular. Therefore, the area coordinates are independent of the Cartesian coordinates. When the area coordinates are written in terms of the Cartesian coordinates, coincidentally they have the same formulations as those of the three shape functions, i.e.,  $N_i$ ,  $N_j$  and  $N_m$ . Below some special properties of the area coordinate are presented. First, from the formulations of the area coordinates, it is clearly seen that the summation of the three area coordinates equals to one, i.e.,

$$L_i + L_j + L_m = 1 \tag{5.42}$$

Second, from the definition of the area coordinates, their values at the three corner nodes can be easily obtained. To work out the area coordinates at node  $i$ , the point  $P$  is moved to node  $i$ , then  $A_i = 1$ ,  $A_j = 0$ ,  $A_m = 0$  are obtained. Therefore, the area coordinate of node  $i$  is  $(1 \ 0 \ 0)$ . Similarly, the area coordinates for nodes  $j$  and  $m$  are  $(0 \ 1 \ 0)$  and  $(0 \ 0 \ 1)$ , respectively.

Third, the behavior of the area coordinate on the edges of the triangular is investigated. To work out the behavior of the area coordinate on edge  $\bar{ij}$ , the point  $P$  is moved to the edge  $\bar{ij}$ , then  $A_m = 0$  is obtained, i.e., the area coordinate of  $L_m$  is zero. Similarly,  $L_i = 0$  on edge  $\bar{jm}$  and  $L_j = 0$  on edge  $\bar{mi}$  are obtained.

Last, the behavior of the area coordinate on the lines parallel to the edge of the triangular is investigated. The edge  $\bar{jm}$  is used as the example for the demonstration. To work out the area coordinate on the lines parallel to the edge  $\bar{jm}$ , the point  $P$  is moved on the parallel line and it is clearly seen that the area  $A_i$  maintains the same when moving  $P$ , i.e., all the  $L_i$  equal on the lines parallel to the edge  $\bar{jm}$ .

A point in the triangular can be expressed using the area coordinate and the Cartesian coordinate. These two coordinate systems can be switched between each other using the formulations given below:

First, the Cartesian coordinate can be changed to the area coordinate using the following relation:

$$\begin{Bmatrix} L_i \\ L_j \\ L_m \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} a_i & b_i & c_i \\ a_j & b_j & c_j \\ a_m & b_m & c_m \end{bmatrix} \begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} \quad (5.43)$$

Second, the area coordinate can be changed to the Cartesian coordinate using the following relation:

$$\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_i & x_j & x_m \\ y_i & y_j & y_m \end{bmatrix} \begin{Bmatrix} L_i \\ L_j \\ L_m \end{Bmatrix} \quad (5.44)$$

The differential operation with respect to the area coordinate can also be changed to that with respect to the Cartesian coordinate using the following relations:

$$\left. \begin{aligned} \frac{\partial}{\partial x} &= \frac{\partial}{\partial L_i} \frac{\partial L_i}{\partial x} + \frac{\partial}{\partial L_j} \frac{\partial L_j}{\partial x} + \frac{\partial}{\partial L_m} \frac{\partial L_m}{\partial x} \\ &= \frac{1}{2A} \left( b_i \frac{\partial}{\partial L_i} + b_j \frac{\partial}{\partial L_j} + b_m \frac{\partial}{\partial L_m} \right) \\ \frac{\partial}{\partial y} &= \frac{1}{2A} \left( c_i \frac{\partial}{\partial L_i} + c_j \frac{\partial}{\partial L_j} + c_m \frac{\partial}{\partial L_m} \right) \end{aligned} \right\} \quad (5.45)$$

where

$$\frac{\partial L_i}{\partial x} = \frac{\partial N_i}{\partial x} = \frac{1}{2A} b_i \quad (5.46a)$$

$$\frac{\partial L_i}{\partial y} = \frac{\partial N_i}{\partial y} = \frac{1}{2A} c_i \quad (5.46b)$$

The corresponding relations with respect to  $L_j$  and  $L_m$  can be obtained by changing the subscripts from  $i$  to  $j$  and from  $i$  to  $m$  in the two formulations above, respectively.

At the end of this part, the formula for the integral operations on the area coordinate is given as below:

$$\int_A L_i^a L_j^b L_m^c dx dy = \frac{a!b!c!}{(a+b+c+2)!} 2A \quad (5.47)$$

The formula above is the integration over the area of the triangular. If it is integrated along the edge of the triangular, the following formula can be used:

$$\int_l L_i^a L_j^b dl = \frac{a!b!}{(a+b+1)!} l \quad (5.48)$$

Examples using the formulae are demonstrated as below:

when  $a = 1, b = 0$  and  $c = 0$ , the integration over the triangular can be calculated as:

$$\int_A L_i dx dy = \frac{1!0!0!}{(1+0+0+2)!} 2A = \frac{A}{3} \quad (5.49a)$$

when  $a = 2, b = 0$  and  $c = 0$ , the integration over the triangular can be calculated as:

$$\int_A L_i^2 dx dy = \frac{2!0!0!}{(2+0+0+2)!} 2A = \frac{A}{6} \quad (5.49b)$$

when  $a = 1, b = 1$  and  $c = 0$ , the integration over the triangular can be calculated as:

$$\int_A L_i L_j dx dy = \frac{1!1!0!}{(1+1+0+2)!} 2A = \frac{A}{12} \quad (5.49c)$$

Equations (5.47) and (5.48) demonstrate one advantage using the area coordinate, i.e., the integration calculation can be quickly obtained using the given equations.

## 5.5 Properties of the Global Stiffness Matrix

In Chap. 3, the elemental stiffness matrix is introduced. In this part, the global stiffness matrix is presented. The difference between the elemental and the global stiffness matrix is that the global stiffness matrix is formed by several elemental stiffness matrices. As shown in Fig. 5.6, there are four 3-node triangular elements in the structure. Therefore, the dimension of the elemental stiffness matrix is  $6 \times 6$ . The dimension of the global stiffness matrix depends on the total number of the nodes in the structure. There are 6 nodes and each node has two degrees of freedom. Therefore, the dimension of the global stiffness matrix is  $12 \times 12$ . The assembly of the elemental stiffness matrices to form the global stiffness matrix is based on the nodal correspondence between the elemental nodes and the structural nodes. For example, in the second element, the indices for the three nodes in the local element are  $i_2, j_2$  and  $m_2$ , which correspond to nodes 5, 2 and 4 in the structure. Therefore, the components in the stiffness matrix of the second element should be placed in the global stiffness matrix corresponding to nodes 2, 4 and 5. For example,  $K_{jj}^{(2)}$  should be placed in the second row of the second column in the global stiffness matrix.

Similar to the elemental stiffness matrix, there are several properties associated with the global stiffness matrix. It should be noted that some properties are exclusively for the global stiffness matrix and not for the elemental stiffness matrix.

**Property 1:** The global stiffness matrix is a sparse matrix, i.e., most components in the global stiffness matrix are zeros.

As shown in Fig. 5.7, after the elemental stiffness matrix is assembled into the global stiffness matrix, the non-zero components are marked using the squares. The number of non-zero components is 43 and the number of zero components is 57. It should be noted that the number of non-zero components in the row of the global stiffness matrix equals to the number of nodes connected to the node (corresponding to that row) plus one. For example, 6 nodes are connected to node 5 and thus 7

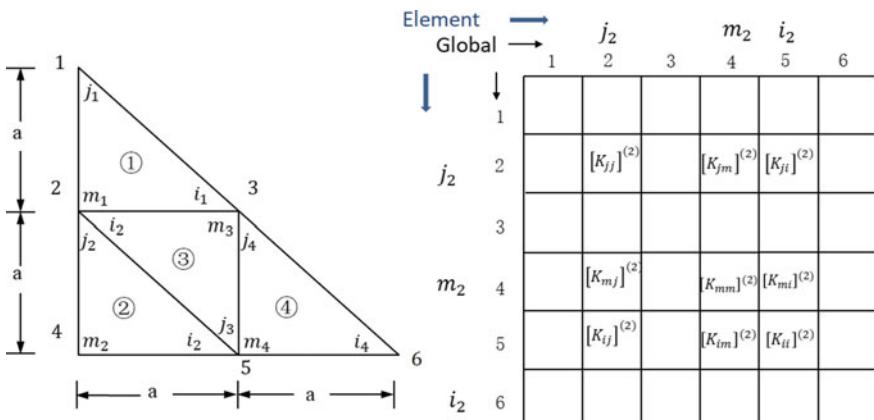
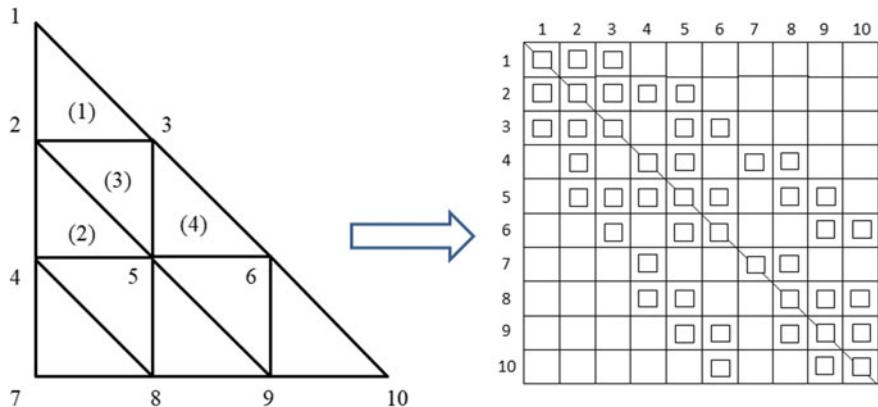


Fig. 5.6 Assembly of the global stiffness matrix using the elemental matrices



**Fig. 5.7** Assembling the elemental stiffness matrix to the global stiffness matrix

non-zero components are in the fifth row of the global stiffness matrix. Additionally, it can be seen that the more the nodes are in the structure, the sparser the global stiffness matrix is.

**Property 2:** The global stiffness matrix is symmetric.

Because the elemental stiffness matrix is symmetric and thus it can be seen from the assembling process that the global stiffness matrix is also symmetric.

**Property 3:** The global stiffness matrix is singular, i.e.,  $|\mathbf{K}| = 0$

This property can be proved using the scenario of rigid displacement. When the structure has a rigid displacement of  $\mathbf{u}^*$ , the external forces are balanced in the structure because it is rigid displacement. Therefore, the stiffness equation can be written as:

$$\mathbf{K}\mathbf{q} = \mathbf{K}\mathbf{u}^* = 0 \quad (5.50)$$

In the equation above, because  $\mathbf{u}^* \neq 0$ , to obtain the solution for the equation, the determinant of the stiffness matrix has to be zero, i.e.,  $|\mathbf{K}| = 0$ , which implies that the nodal displacements cannot be determined under the loading scenario of balanced forces.

**Property 4:** The column  $K_{11}$  in the global stiffness matrix means the forces needed on all the nodes when  $u_1 = 1$ ,  $u_i = 0$  ( $i \neq 1$ ) and  $v_i = 0$  ( $i = 1, \dots, n$ ).

This property can be proved as below:

The global stiffness equation for the system is:

$$\mathbf{K}\mathbf{q} = \mathbf{P} \quad (5.51)$$

i.e.

$$\begin{bmatrix} k_{11} & \cdots & k_{1,2n} \\ \vdots & \ddots & \vdots \\ k_{2n,1} & \cdots & k_{2n,2n} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ \vdots \\ v_n \end{Bmatrix} = \begin{Bmatrix} P_{1x} \\ P_{1y} \\ \vdots \\ P_{nx} \\ P_{ny} \end{Bmatrix} \quad (5.52)$$

In the equation above,  $u_1 = 1$ ,  $u_i = 0(i \neq 1)$  and  $v_i = 0$  are set and then the following equations can be obtained

$$\begin{bmatrix} k_{11} & \cdots & k_{1,2n} \\ \vdots & \ddots & \vdots \\ k_{2n,1} & \cdots & k_{2n,2n} \end{bmatrix} \begin{Bmatrix} 1 \\ 0 \\ \vdots \\ 0 \end{Bmatrix} = \begin{Bmatrix} P_{1x} \\ P_{1y} \\ \vdots \\ P_{nx} \\ P_{ny} \end{Bmatrix} \quad (5.53)$$

The equations above are equivalent to the following equation

$$[k_{11} \ \cdots \ k_{2n,1}]^T = [P_{1x} \ P_{1y} \ \cdots \ P_{nx} \ P_{ny}]^T \quad (5.54)$$

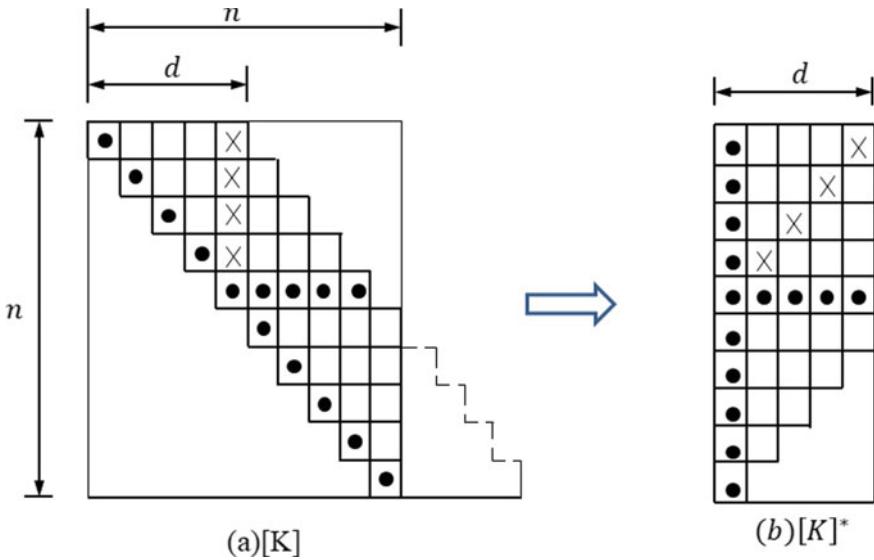
The physical meaning of the equations above is that column  $k_{i1}$  (the left side of Eq. (5.54)) means the forces needed on all the nodes (the right side of Eq. (5.54)) when  $u_1 = 1$ ,  $u_i = 0(i \neq 1)$  and  $v_i = 0$  (condition for obtaining Eq. (5.53)).

At the end of this part, the definition of the half band width (HBW) is introduced. Because the global stiffness matrix is a sparse matrix, when storing the global stiffness matrix using Fortran or Matlab, a large amount of storage spaces will be wasted and the relevant calculations are not efficient. To solve these problems, the HBW is introduced. Basically, HBW is used to reduce the dimension of the global stiffness matrix from  $n \times n$  to  $d \times n$ , where  $d$  is called the *half band width*.

The reduction procedure from  $n \times n$  to  $d \times n$  is illustrated in Fig. 5.8. First, the diagonal elements in the original matrix are placed in the first column in the reduced matrix (Fig. 5.8b). Then the non-zero elements in the upper part of the original matrix (Fig. 5.8a) are placed in the corresponding places in the reduced matrix. Therefore, the width of the dimension of the reduced matrix depends on the distance from the diagonal element to the farthest non-zero element in each row of the original matrix. It is complicated and time consuming to count the distance in each row of the original global stiffness matrix when the global stiffness matrix is very large. Therefore, the following equation is summarized and used to work out the HBW:

$$d_i = (\text{the maximum nodal number difference in the element} + 1) \times \omega$$

where  $d_i$  is the HBW in each element,  $\omega$  is the number of the degrees of freedom in each node. Afterwards, the HBW for the structure can be worked as below:



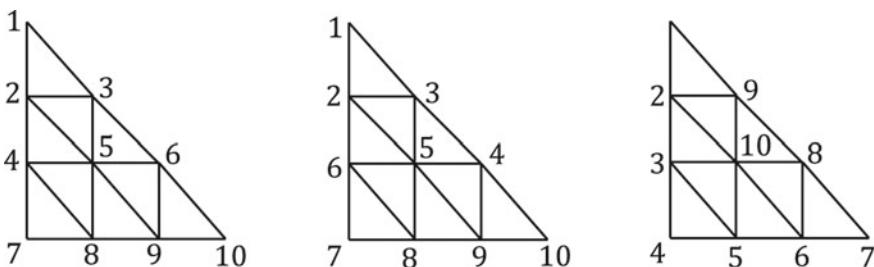
**Fig. 5.8** Reduction of the dimension of the global stiffness matrix

$$d = \max\{d_i\}, (i = 1, 2, 3, \dots, n) \quad (5.55)$$

where  $n$  is the number of triangular elements in the structure.

Using Eq. (5.55), the HBW for the structure with the following nodal number ordering can be worked out.

From the left to the right in Fig. 5.9, the HBWs are 5, 7 and 9 and are 10, 14 and 18 when the nodal degrees of freedom are considered. It is also demonstrated in the example that for the same structure, if the ordering of the nodes is changed, the HBW will be different, i.e., different forms of the reduced global stiffness matrices will be obtained.



**Fig. 5.9** The half band widths for different numbering of the same structure

## 5.6 Calculation of the Equivalent Nodal Forces

In Chap. 4, the calculation of the equivalent nodal force is introduced. Now let's first review some of the basic knowledge. The reason why the equivalent nodal force has to be worked out is that for the beam and triangular elements, the external force can be applied on the non-nodal positions. To introduce these non-nodal external forces into the stiffness equation, the equivalent nodal force has to be worked out. The virtual displacement principle is used to work out the equivalent nodal force, i.e., the work done by the non-nodal forces should equal to the work done by the equivalent nodal forces. For the triangular element, the same principle and the same method are used to work out the equivalent nodal force and the same formulae are obtained, i.e.,

$$\mathbf{P}_f^e = \int_{\Omega^e} \mathbf{N}^T \mathbf{f} \cdot t \cdot dxdy \quad (5.56a)$$

$$\mathbf{P}_S^e = \int_{S^e} \mathbf{N}^T \mathbf{T} \cdot t \cdot dS \quad (5.56b)$$

where  $\mathbf{f}$  is the body force,  $\mathbf{T}$  is the surface force,  $t$  is the element thickness,  $\mathbf{P}_f^e$  is the equivalent nodal force for the body force and  $\mathbf{P}_S^e$  is the equivalent nodal force for the surface force.

Below several examples are used to demonstrate the calculation of the equivalent nodal forces using Eqs. (5.56a) and (5.56b)

**Example 5.1:** A triangular element is under the self-weight. The density of the element is  $\rho$  and the acceleration is  $g$ . The question is to work out the equivalent nodal forces for the loading of the self-weight (Fig. 5.10).

**Solution:** The external force for the self-weight can be written as:

$$\mathbf{f} = \begin{Bmatrix} 0 \\ -\rho g \end{Bmatrix}$$

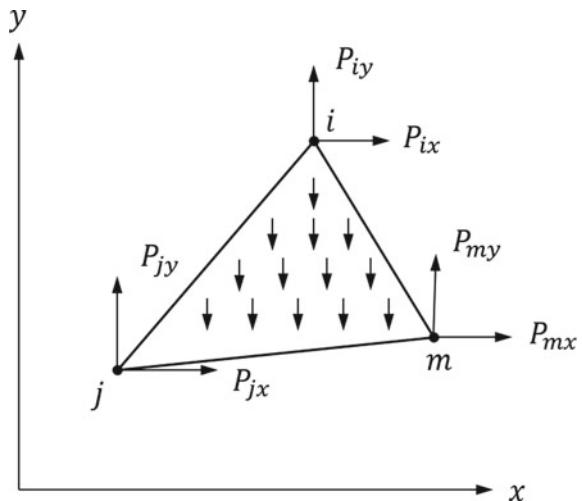
Based on Eq. (5.56a), the equivalent nodal forces at the nodes  $i, j$  and  $m$  can be written as:

$$\mathbf{P}_{\rho}^e = \begin{Bmatrix} \mathbf{P}_i \\ \mathbf{P}_j \\ \mathbf{P}_m \end{Bmatrix}_{\rho} = \int_{\Omega^e} \begin{Bmatrix} \mathbf{N}_i \\ \mathbf{N}_j \\ \mathbf{N}_m \end{Bmatrix} \begin{Bmatrix} 0 \\ -\rho g \end{Bmatrix} t dxdy$$

where  $\mathbf{P}_i$  is the equivalent force at node  $i$  which can be worked out as:

$$\mathbf{P}_{i\rho} = \begin{bmatrix} P_{ix} \\ P_{iy} \end{bmatrix}_{\rho} = \int_{\Omega^e} \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} \begin{Bmatrix} 0 \\ -\rho g \end{Bmatrix} t dxdy$$

**Fig. 5.10** A body force on the triangular element



$$= \left\{ \begin{array}{c} 0 \\ - \int_{\Omega^e} N_i \rho g t dxdy \end{array} \right\} = \left\{ \begin{array}{c} 0 \\ - \frac{1}{3} \rho g t A \end{array} \right\}$$

Similarly, the equivalent forces at nodes  $j$  and  $m$  can be worked out. Therefore, the equivalent nodal forces at the three nodes are:

$$\mathbf{P}_\rho = -\frac{1}{3} \rho g t A [0 \ 1 \ 0 \ 1 \ 0 \ 1]^T$$

From the result, it can be seen that the self-weight has been evenly transferred to the forces at the three elemental nodes and the directions of the equivalent forces are in agreement with that of the self-weight.

**Example 5.2:** A uniform pressure is applied on the edge  $\overrightarrow{ij}$  of a triangular element. The question is to work out the equivalent nodal forces at the triangular (Fig. 5.11).

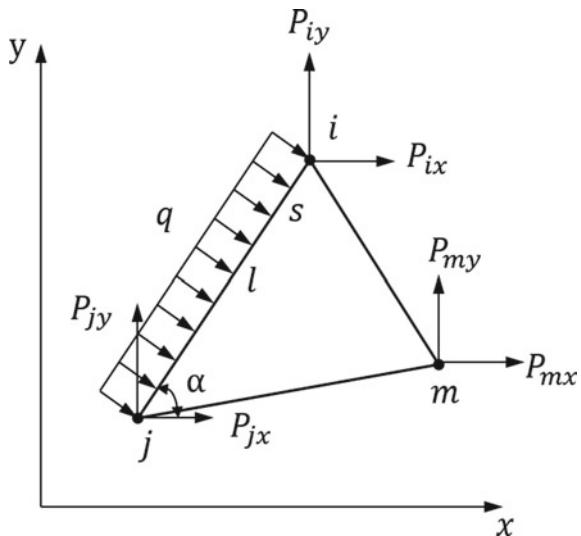
**Solution:** Before the calculation, first it can be concluded that the equivalent forces will be at nodes  $i$  and  $j$  and not at node  $m$ . The calculation of the equivalent nodal forces can be divided into three steps. First, the uniform pressure can be decoupled to the force component in the  $x$ -direction and that in the  $y$ -direction, i.e.:

$$q_x = q \sin \alpha = \frac{q}{l} (y_i - y_j)$$

$$q_y = -q \cos \alpha = \frac{q}{l} (x_j - x_i)$$

Therefore, the external force vector can be written as:

**Fig. 5.11** An external uniform pressure on one edge of the triangular element



$$\mathbf{T} = \begin{Bmatrix} q_x \\ q_y \end{Bmatrix} = \frac{q}{l} \begin{Bmatrix} y_i - y_j \\ x_j - x_i \end{Bmatrix}$$

Second, because the external force is applied on the edge  $\overline{ij}$ , the shape functions on this edge should be determined, which are given as below:

$$N_i = 1 - \frac{s}{l}, N_j = \frac{s}{l}, N_m = 0 \quad (5.57)$$

Afterwards, the equivalent nodal forces can be worked out using Eq. (5.57):

$$P_{ix} = \int_l N_i q_x t ds = \int_l \left(1 - \frac{s}{l}\right) q_x t ds = \frac{t}{2} q (y_i - y_j)$$

$$P_{jx} = \int_l N_j q_x t ds = \int_l \frac{s}{l} q_x t ds = \frac{t}{2} q (y_i - y_j)$$

Similarly

$$P_{iy} = \frac{t}{2} q (x_j - x_i)$$

$$P_{jy} = \frac{t}{2} q (x_j - x_i)$$

Furthermore,

$$P_{mx} = P_{my} = 0$$

Therefore, the equivalent nodal force vector can be written as:

$$\mathbf{P}_q = \frac{1}{2}qt[y_i - y_j \ x_j - x_i \ y_i - y_j \ x_j - x_i \ 0 \ 0]^T$$

**Example 5.3:** A uniform pressure along the  $x$ -axis is applied on edge  $\vec{ij}$  of a triangular element. The question is to work out the equivalent nodal forces at the triangular (Fig. 5.12).

**Solution:** The external pressure can be written in terms of the force vector as:

$$\mathbf{T} = \begin{Bmatrix} q \\ 0 \end{Bmatrix} \quad (5.58)$$

Then using Eq. (5.58), the equivalent nodal force vector can be worked out as:

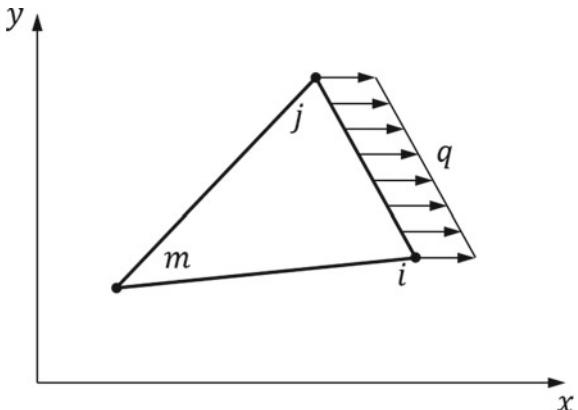
$$\mathbf{P}^e = \frac{1}{2}qlt[1 \ 0 \ 1 \ 0 \ 0 \ 0]^T \quad (5.59)$$

In Eq. (5.59) the answer for the equivalent nodal forces has been given directly, but the readers should follow the procedure described in the previous section to work out the details.

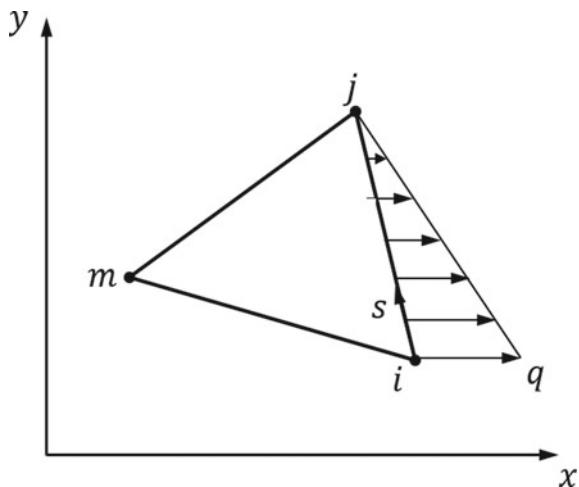
**Example 5.4:** A pressure with the triangular shape is applied along the  $x$ -direction on edge  $\vec{ij}$  of a triangular element. The question is to work out the equivalent nodal forces at the triangular (Fig. 5.13).

**Solution:** The external pressure can be written in terms of the force vector as:

**Fig. 5.12** A horizontal uniform pressure on one edge of the triangular element



**Fig. 5.13** A triangular pressure on one edge of the triangular element



$$\mathbf{T} = \begin{Bmatrix} (1 - \frac{s}{l})q \\ 0 \end{Bmatrix} \quad (5.60)$$

Then using Eq. (5.60), the equivalent nodal force vector can be worked out as:

$$\mathbf{P}^e = \frac{1}{2}qlt \begin{bmatrix} \frac{2}{3} & 0 & \frac{1}{3} & 0 & 0 & 0 \end{bmatrix}^T$$

Again, the readers should follow the procedure described in the previous section to work out the details.

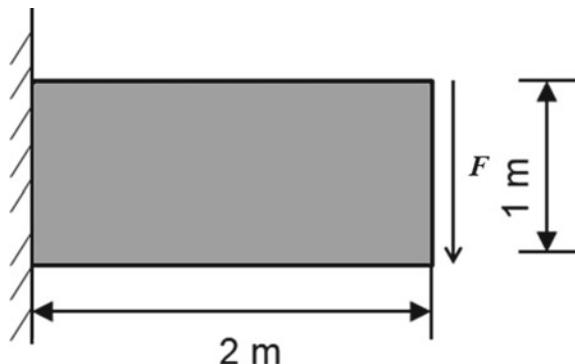
## 5.7 An Example of the FE Analysis Using Triangular Element

At the end of this chapter, an example is used to demonstrate the calculation of the 2D plane problem using the triangular element.

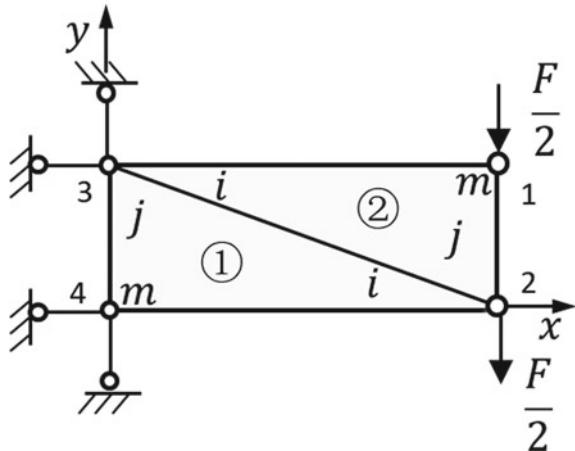
**Example 5.5:** A 2D plane structure is shown in Fig. 5.14. The length, the width and the thickness of the structure are 2.0 m, 1.0 m and 1.0, respectively. The elastic modulus  $E = 1.0$  and Poisson's ratio  $\nu = 0.33$ . An external force  $F = 1.0$  is applied on the right end of the structure. The tasks are to work out the nodal displacements and the reaction forces in the structure. It should be noted that to simplify the calculation, 'meaningless' values are used for the elastic modulus and the thickness and thus no unit is presented for them.

**Solution:** Below the FE procedure is followed to solve the problem. First, the structure is discretized into two 3-node triangular elements (Fig. 5.15). Therefore,

**Fig. 5.14** A 2D plane structure



**Fig. 5.15** The 2D plane structure discretized using two triangular elements



the nodal displacement vector can be written as:

$$\mathbf{q} = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T$$

The external force vector can be written as:

$$\bar{\mathbf{F}} = \left[ 0 \ -\frac{F}{2} \ 0 \ -\frac{F}{2} \ 0 \ 0 \ 0 \ 0 \right]^T$$

The reaction force vector in the structure can be written as:

$$\mathbf{R} = [0 \ 0 \ 0 \ 0 \ R_{x3} \ R_{y3} \ R_{x4} \ R_{y4}]^T$$

Therefore, the total nodal force vector is:

**Table 5.1** The coordinates and values of the constants for the first element

	$i(1)$	$j(2)$	$m(3)$
$x_k$	2	0	0
$y_k$	0	1	0
$a_i = x_j y_m - x_m y_j$	0	0	2
$b_i = y_j - y_m$	1	0	-1
$c_i = -x_j + x_m$	0	2	-2

$$\mathbf{P} = \bar{\mathbf{F}} + \mathbf{R} = \left[ 0 \quad -\frac{F}{2} \quad 0 \quad -\frac{F}{2} \quad R_{x3} \quad R_{y3} \quad R_{x4} \quad R_{y4} \right]^T$$

Now let's work out the shape functions for the given structure. Regarding the first element (Table 5.1):

The area of the first element is 1.0. Therefore, based on Eq. (5.10), the three shape functions for the first element can be worked out as:

$$N_i = \frac{1}{2}(0 + 1x + 0y) = \frac{x}{2}$$

$$N_j = \frac{1}{2}(0 + 0x + 2y) = y$$

$$N_m = \frac{1}{2}(2 - 1x - 2y) = 1 - \frac{1}{2}x - y$$

The strain-displacement matrix for the first element can be worked out based on Eq. (5.14):

$$\mathbf{B}(x, y) = \frac{1}{2A} \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix} = [\mathbf{B}_1 \quad \mathbf{B}_2 \quad \mathbf{B}_3]$$

Putting the values of  $b_1, \dots, c_3$  into Eq. (5.14), the answer for the strain-displacement matrix can be obtained:

$$\mathbf{B} = \frac{1}{2} \begin{bmatrix} 1 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 2 & 0 & -2 \\ 0 & 1 & 2 & 0 & -2 & -1 \end{bmatrix}$$

A plane stress problem is assumed and thus the elasticity matrix can be written as:

$$\mathbf{D} = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix}$$

Therefore, the stress-displacement matrix for the first element can be worked out as:

$$\mathbf{S} = \mathbf{DB}$$

Based on Eq. (5.21), the elemental stiffness matrix for the two elements can be worked out as:

$$\mathbf{K}^{(1),(2)} = \begin{bmatrix} k_{ii} & k_{ij} & k_{im} \\ k_{ji} & k_{jj} & k_{jm} \\ k_{mi} & k_{mj} & k_{mm} \end{bmatrix} = \frac{9Et}{32} \begin{bmatrix} 1 & 0 & 0 & \frac{2}{3} & -1 & -\frac{2}{3} \\ 0 & \frac{1}{3} & \frac{2}{3} & 0 & -\frac{2}{3} & -\frac{1}{3} \\ 0 & \frac{2}{3} & \frac{4}{3} & 0 & -\frac{4}{3} & -\frac{2}{3} \\ \frac{2}{3} & 0 & 0 & 4 & -\frac{2}{3} & -4 \\ -1 & -\frac{2}{3} & -\frac{4}{3} & -\frac{2}{3} & \frac{7}{3} & \frac{4}{3} \\ -\frac{2}{3} & -\frac{1}{3} & -\frac{2}{3} & -4 & \frac{4}{3} & \frac{13}{3} \end{bmatrix}$$

After the elemental stiffness matrices are worked out, they can be assembled to form the global stiffness matrix, i.e.:

$$\mathbf{K} = \mathbf{K}^{(1)} + \mathbf{K}^{(2)}$$

$$\mathbf{K} = \begin{bmatrix} k_{mm}^{(2)} & k_{mj}^{(2)} & k_{mi}^{(2)} \\ k_{jm}^{(2)} & k_{jj}^{(2)} + k_{ii}^{(1)} & k_{ji}^{(2)} + k_{ij}^{(1)} & k_{im}^{(1)} \\ k_{im}^{(2)} & k_{ij}^{(2)} + k_{ji}^{(1)} & k_{ii}^{(2)} + k_{jj}^{(1)} & k_{jm}^{(1)} \\ k_{mi}^{(1)} & k_{mj}^{(1)} & k_{mj}^{(1)} & k_{mm}^{(1)} \end{bmatrix}$$

Substituting the global stiffness matrix to the structural stiffness equation, i.e.:

$$\mathbf{K}\mathbf{q} = \mathbf{P}$$

The stiffness equation for the given structure can be obtained as below:

$$\begin{bmatrix} \frac{7}{3} & \frac{4}{3} & -\frac{4}{3} & -\frac{2}{3} & -1 & -\frac{2}{3} & 0 & 0 \\ \frac{4}{3} & \frac{13}{3} & -\frac{2}{3} & -4 & -\frac{2}{3} & -\frac{1}{3} & 0 & 0 \\ -\frac{4}{3} & -\frac{2}{3} & \frac{7}{3} & 0 & 0 & \frac{4}{3} & -1 & -\frac{2}{3} \\ -\frac{2}{3} & -4 & 0 & \frac{13}{3} & \frac{4}{3} & 0 & -\frac{2}{3} & -\frac{1}{3} \\ -1 & -\frac{2}{3} & 0 & \frac{4}{3} & \frac{7}{3} & 0 & -\frac{4}{3} & -\frac{2}{3} \\ -\frac{2}{3} & -\frac{1}{3} & \frac{4}{3} & 0 & 0 & \frac{13}{3} & -\frac{2}{3} & -4 \\ 0 & 0 & -1 & -\frac{2}{3} & -\frac{4}{3} & -\frac{2}{3} & \frac{7}{3} & \frac{4}{3} \\ 0 & 0 & -\frac{2}{3} & -\frac{1}{3} & -\frac{2}{3} & -4 & \frac{4}{3} & \frac{13}{3} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -\frac{F}{2} \\ 0 \\ -\frac{F}{2} \\ R_{x3} \\ R_{y3} \\ R_{x4} \\ R_{y4} \end{Bmatrix}$$

When the hand calculation is used, the boundary condition is introduced to the stiffness equation to reduce the complexity of the stiffness equation. The boundary condition in the structure given in Fig. 5.15 is:

$$u_3 = 0, v_3 = 0, u_4 = 0, v_4 = 0$$

Therefore, the stiffness equation (Eq. 5.61) can be reduced to the following form:

$$\begin{bmatrix} \frac{7}{3} & \frac{4}{3} & -\frac{4}{3} & -\frac{2}{3} \\ \frac{4}{3} & \frac{13}{3} & -\frac{2}{3} & -4 \\ -\frac{4}{3} & -\frac{2}{3} & \frac{7}{3} & 0 \\ -\frac{2}{3} & -4 & 0 & \frac{13}{3} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -\frac{F}{2} \\ 0 \\ -\frac{F}{2} \end{Bmatrix}.$$

Solving the equations above, the answers for the nodal displacements can be obtained:

$$[u_1 \ v_1 \ u_2 \ v_2]^T = \frac{F}{Et} [1.88 \ -8.99 \ -1.50 \ -8.42]^T$$

Afterwards, the reaction forces can be worked out by putting the nodal displacements back to the global stiffness equation and then the reaction forces can be obtained as:

$$R_{x3} = \frac{9Et}{32} \left( -u_1 - \frac{2}{3}v_1 + \frac{4}{3}v_2 \right) = -2F$$

$$R_{y3} = \frac{9Et}{32} \left( -\frac{2}{3}u_1 - \frac{1}{3}v_1 + \frac{4}{3}u_2 \right) = -0.07F$$

$$R_{x4} = \frac{9Et}{32} \left( -u_2 - \frac{2}{3}v_2 \right) = 2F$$

$$R_{y4} = \frac{9Et}{32} \left( -\frac{2}{3}u_2 - \frac{1}{3}v_2 \right) = 1.07F$$

# Chapter 6

## Finite Element Analysis Using Rectangular Element



### 6.1 Introduction

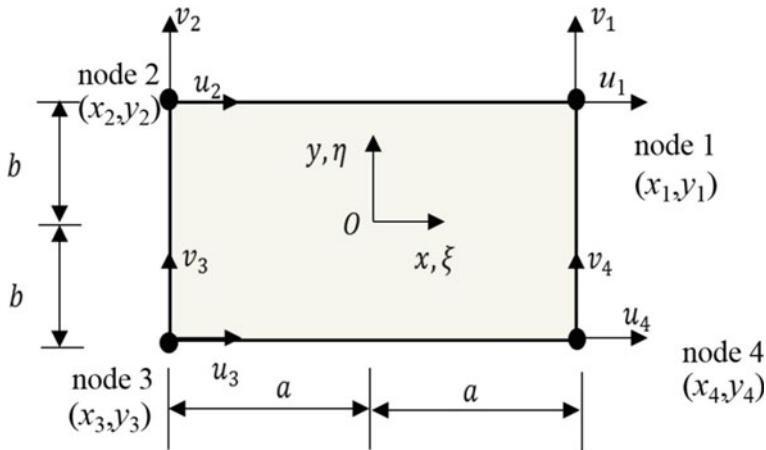
In the previous chapter, the 3-node triangular element is introduced. It is known that the 3-node triangular element is a linear element, in which the displacement within the element is a linear function of the coordinate and the strain within the element is a constant. To increase the order of the elemental displacement functions and to enable the efficient calculations, the 4-node rectangular element is introduced in this chapter. First, the shape functions for the 4-node rectangular element are given, and then the iso-parameter elements are introduced followed by the introduction of the numerical integrations.

### 6.2 FE Analysis Procedure Using Rectangular Element

In this part, the 4-node regular rectangular is taken as the reference and used to demonstrate the FE calculation procedure. However, it should be noted that in the complex engineering problems, the quadrilateral elements with any shape may be required to mesh the structure. The answers for these quadrilateral elements can be mapped from that of the reference element, which will also be introduced in this chapter.

#### (1) The geometry and node information of rectangular element

First, the geometry and node information of the rectangular element is introduced using a plane 4-node rectangular element (Fig. 6.1). In the element, there are two degrees of freedom of each node, i.e., the displacement along the  $x$  direction is denoted as  $u$  and the displacement along the  $y$  direction is denoted as  $v$ . Therefore, there are 8 degrees of freedom in the 4-node rectangular element. It should be noted



**Fig. 6.1** The plane 4-node rectangular element

that the displacements in the  $x$  and  $y$  directions are independent of each other. Therefore, two independent field functions are needed to describe the displacements in the  $x$  and  $y$  directions, respectively.

The nodal displacement vector for the 4-node rectangular element can be written as:

$$\mathbf{q}^e = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T \quad (6.1)$$

The corresponding force vector can be written as:

$$\mathbf{P}^e = [P_{x1} \ P_{y1} \ P_{x2} \ P_{y2} \ P_{x3} \ P_{y3} \ P_{x4} \ P_{y4}]^T \quad (6.2)$$

It should be noted that for the rectangular element, there are the scenarios when the external forces are not at the nodes, e.g., the surface pressure. In these scenarios, the procedure presented in Chap. 4 (Beam element) should be followed to work out the equivalent nodal forces and then the subsequent FE analysis can be completed.

To unify the formulations for different rectangular elements with different dimensions, the dimensionless coordinates are used. When the origins of the Cartesian and the dimensionless coordinate systems are defined in the center of the rectangular element (Fig. 6.1), the relation between the Cartesian and the dimensionless coordinates can be written as below:

$$\xi = \frac{x}{a}, \eta = \frac{y}{b} \quad (6.3)$$

where  $a$  is the half of length of the rectangular element and  $b$  is the half of height of the element,  $(x, y)$  and  $(\xi, \eta)$  are the Cartesian and dimensionless coordinates, respectively.

Using the dimensionless coordinates defined in Eq. (6.3), the coordinates for the four nodes are:

$$\xi_1 = 1, \xi_2 = -1, \xi_3 = -1, \xi_4 = 1$$

$$\eta_1 = 1, \eta_2 = 1, \eta_3 = -1, \eta_4 = -1$$

It should be noted that it is not required to define the origins of the Cartesian and dimensionless coordinate systems in the center of the rectangular element. In some scenarios, the origins can be at one corner of the rectangular element. In this chapter, the origin in the center is used, because the corresponding calculations are the simplest.

## (2) The elemental displacement field formulation

When calculating the elemental displacement field formulation, the displacements at the four nodes of the rectangular element are assumed to be known. Then 8 boundary conditions can be obtained as below:

$$\begin{cases} u(x_i, y_i) = u_i & (i = 1, 2, 3, 4) \\ v(x_i, y_i) = v_i \end{cases} \quad (6.4)$$

Using the 8 equations, 8 unknown coefficients can be solved. Previously, it is mentioned that two independent displacement field formulations are required to describe the displacements in the  $x$  and  $y$  directions, respectively. Therefore, there are 4 unknown coefficients in each of the elemental displacement field formulations:

$$\begin{cases} u(x, y) = a_0 + a_1x + a_2y + a_3xy \\ v(x, y) = b_0 + b_1x + b_2y + b_3xy \end{cases} \quad (6.5)$$

where  $a_0, \dots, a_3$  and  $b_0, \dots, b_3$  are the coefficients to be solved. It can be seen, compared to the displacement field formulations for the triangular element, a higher order term, i.e.,  $xy$ , is present in the formulation, which can make the displacement distribution within the element nonlinear. The reason why the terms 1,  $x$ ,  $y$  and  $xy$  are used in the displacement field formulation can be explained using the Pascal's triangle, which will be introduced in Chap. 8. The displacement field formulations, i.e., Eq. (6.5), show that the displacements within the rectangular element are linearly distributed along the  $x$  and  $y$  directions, and they are nonlinearly distributed along the diagonal direction, which is an improvement from the linear 3-node triangular element.

Putting Eq. (6.4) into Eq. (6.5), rearranging the equation and then the following equations can be obtained:

$$\begin{cases} u(x, y) = N_1(x, y)u_1 + N_2(x, y)u_2 + N_3(x, y)u_3 + N_4(x, y)u_4 \\ v(x, y) = N_1(x, y)v_1 + N_2(x, y)v_2 + N_3(x, y)v_3 + N_4(x, y)v_4 \end{cases} \quad (6.6)$$

In the matrix format, Eq. (6.6) can be rewritten as:

$$\mathbf{u}(x, y) = \begin{Bmatrix} u(x, y) \\ v(x, y) \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \mathbf{N}\mathbf{q}^e \quad (6.7)$$

where  $\mathbf{N}(x, y)$  is the matrix for the elemental shape function,  $N_1, N_2, N_3$  and  $N_4$  are given as below:

$$\begin{cases} N_1(x, y) = \frac{1}{4}(1 + \frac{x}{a})(1 + \frac{y}{b}) \\ N_2(x, y) = \frac{1}{4}(1 - \frac{x}{a})(1 + \frac{y}{b}) \\ N_3(x, y) = \frac{1}{4}(1 - \frac{x}{a})(1 - \frac{y}{b}) \\ N_4(x, y) = \frac{1}{4}(1 + \frac{x}{a})(1 - \frac{y}{b}) \end{cases} \quad (6.8)$$

Using the dimensionless coordinates defined in Eqs. (6.3), (6.8) can be rewritten in terms of the following unified form:

$$N_i = \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta) \quad (i = 1, 2, 3, 4) \quad (6.9)$$

where  $(\xi_i, \eta_i)$  ( $i = 1, 2, 3, 4$ ) are the coordinates for the four nodes in the dimensionless coordinate system.

### (3) The elemental strain field formulation

Based on the 2D geometrical equation, the elemental strain field formulation can be written as:

$$\boldsymbol{\varepsilon}(x, y) = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = [\partial] \mathbf{u} = [\partial] \mathbf{N} \mathbf{q}^e = \mathbf{B} \mathbf{q}^e \quad (6.10)$$

where  $\mathbf{B}(x, y)$  is the strain-displacement matrix and can be written as:

$$\mathbf{B}(x, y) = [\partial] \mathbf{N} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} = [\mathbf{B}_1 \ \mathbf{B}_2 \ \mathbf{B}_3 \ \mathbf{B}_4]$$

where  $\mathbf{B}_1, \mathbf{B}_2, \mathbf{B}_3$  and  $\mathbf{B}_4$  can be written in the following unified form:

$$\mathbf{B}_i = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} \end{bmatrix} (i = 1, 2, 3, 4)$$

Below the behavior of the strain distribution within the rectangular element is analyzed.

First, let's look at the normal strain in the  $x$  direction, i.e.,  $\varepsilon_x$ . According to Eq. (6.10), the formulation for  $\varepsilon_x$  can be written as:

$$\varepsilon_x = \frac{\partial N_1}{\partial x} u_1 + \frac{\partial N_2}{\partial x} u_2 + \frac{\partial N_3}{\partial x} u_3 + \frac{\partial N_4}{\partial x} u_4$$

where the first order derivative of the shape function with respect to  $x$  can be worked out as:

$$\frac{\partial N_1}{\partial x} = \frac{1}{4a} \left(1 + \frac{y}{b}\right)$$

$$\frac{\partial N_2}{\partial x} = -\frac{1}{4a} \left(1 + \frac{y}{b}\right)$$

$$\frac{\partial N_3}{\partial x} = -\frac{1}{4a} \left(1 - \frac{y}{b}\right)$$

$$\frac{\partial N_4}{\partial x} = \frac{1}{4a} \left(1 - \frac{y}{b}\right)$$

Then we have:

$$\varepsilon_x = a'y + b' \quad (6.11)$$

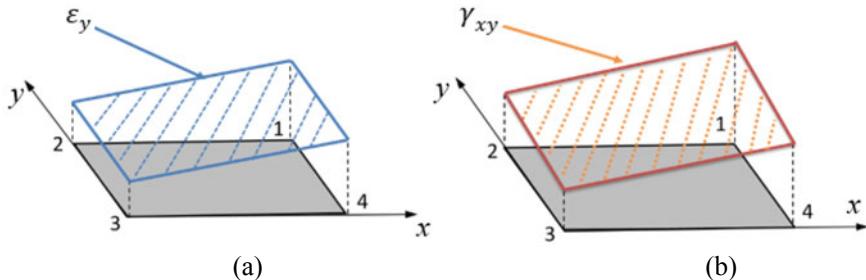
where  $a'$  and  $b'$  are known constants.

Following the same procedure, the formulations for the normal strain in the  $y$  direction, i.e.,  $\varepsilon_y$  and the shear strain  $\gamma_{xy}$  can be worked out as below:

$$\varepsilon_y = c'x + d' \quad (6.12)$$

$$\gamma_{xy} = a''x + b''y + c'' \quad (6.13)$$

It is seen from Fig. 6.2 that, first, on the lines parallel to the  $y$  axis,  $\varepsilon_y$  is constantly distributed. Second, on the lines parallel to the  $x$  axis,  $\varepsilon_y$  is linearly changing with the  $x$  coordinate, which is reflected in Eq. (6.12), i.e.,  $\varepsilon_y$  is only a linear function of the  $x$  coordinate. Third,  $\gamma_{xy}$  is a linear function of both the  $x$  and  $y$  coordinates. Therefore, in Fig. 6.2, on the lines parallel to the  $y$  axis,  $\gamma_{xy}$  is linearly changing with the  $y$  coordinate, and on the lines parallel to the  $x$  axis,  $\gamma_{xy}$  is linearly changing with



**Fig. 6.2** The distribution of the normal strain  $\varepsilon_y$  (a) and the shear strain  $\gamma_{xy}$  (b) within the rectangular element

the  $x$  coordinate. It is clearly seen that, compared to the 3-node triangular element (constant strain), the order of the strain formulations has been increased and the strain is linearly distributed within the rectangular element.

#### (4) The elemental stress field formulation

Using the physical equations, the stress field within the element can be written as:

$$\sigma = \mathbf{D}\boldsymbol{\epsilon} = \mathbf{D}\mathbf{B}\mathbf{q}^e = \mathbf{S}\mathbf{q}^e \quad (6.14)$$

where  $\mathbf{D}$  is the elasticity matrix and  $\mathbf{S}$  is the stress displacement matrix. Similar to the triangular element, the exact formulation for  $\mathbf{D}$  depends on the type of the 2D plane problems. For the 2D plane stress problem,  $\mathbf{D}$  is given as below:

$$\mathbf{D} = \frac{E}{(1-v^2)} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1-v}{2} \end{bmatrix}$$

where  $E$  is the Young's modulus and  $v$  is the Poisson's ratio of the material.

The assumptions in Eq. (6.14) should be noted again, i.e., the material should be isotropic and homogeneous and the small elastic deformation is assumed.

#### (5) The elemental potential energy

Similar to the elements introduced previously, the elemental potential energy consists of the internal energy induced by the internal stress and strain, and the external energy induced by the nodal external forces. Using the displacement, the strain and the stress field formulations of the rectangular element, the elemental potential energy can be worked out as:

$$\Pi^e = \frac{1}{2} \mathbf{q}^{eT} \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^{eT} \mathbf{q}^e \quad (6.15)$$

where  $\mathbf{K}^e$  is the elemental stiffness matrix

$$\mathbf{K}^e = \int_{A^e} \mathbf{B}^T \mathbf{D} \mathbf{B} dA \cdot t = \begin{bmatrix} k_{11} & & & \\ k_{21} & k_{22} & \text{sym} & \\ k_{31} & k_{32} & k_{33} & \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \quad (6.16)$$

where  $t$  is the thickness of the rectangular element. The components in Eq. (6.21) are:

$$\mathbf{K}_{rs} = \int_{A^e} \mathbf{B}^T \mathbf{D} \mathbf{B} \cdot t \cdot dx dy \quad (r, s = 1, 2, 3, 4) \quad (6.17)$$

Putting the values of  $\mathbf{B}$  and  $\mathbf{D}$  into Eq. (6.17), we can have:

$$\mathbf{K}_{rs} = \frac{Et}{4(1 - \mu^2)ab} \begin{bmatrix} k_1 & k_3 \\ k_2 & k_4 \end{bmatrix} \quad (6.18)$$

where

$$\begin{aligned} k_1 &= b^2 \xi_r \xi_s \left( 1 + \frac{1}{3} \eta_r \eta_s \right) + \frac{1 - \mu}{2} a^2 \eta_r \eta_s \left( 1 + \frac{1}{3} \xi_r \xi_s \right) \\ k_2 &= ab \left( \mu \eta_r \xi_s + \frac{1 - \mu}{2} \xi_r \eta_s \right) \\ k_3 &= ab \left( \mu \xi_r \eta_s + \frac{1 - \mu}{2} \eta_r \xi_s \right) \\ k_4 &= a^2 \eta_r \eta_s \left( 1 + \frac{1}{3} \xi_r \xi_s \right) + \frac{1 - \mu}{2} b^2 \xi_r \xi_s \left( 1 + \frac{1}{3} \eta_r \eta_s \right) \quad (r, s = 1, 2, 3, 4) \end{aligned}$$

where  $a$  and  $b$  are the dimensions of the rectangular element defined in Fig. 6.1. Therefore, the elemental stiffness matrix for the rectangular element can be written explicitly as:

$$\begin{aligned}
& \left[ \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{ab}{8}(1+\mu) \quad \frac{ab}{8}(1-\mu)a^2 \quad -\frac{1}{3}(b^2 - \frac{1-\mu}{4}a^2) \quad -\frac{ab}{8}(1-3\mu) \quad -\frac{ab}{8}(1-\mu)a^2 \right] \\
& \left[ \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{ab}{8}(1-3\mu) \quad \frac{ab}{8}(1-\mu)b^2 \quad \frac{1}{3}(b^2 - \frac{1-\mu}{4}a^2) \quad \frac{1}{6}[a^2 - (1-\mu)b^2] \quad -\frac{ab}{8}(1+\mu) \right] \\
& \left[ \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{ab}{8}(1+\mu) \quad -\frac{ab}{8}(1-\mu)a^2 \quad -\frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad -\frac{ab}{8}[b^2 - (1-\mu)a^2] \quad \frac{1}{6}[b^2 - (1-\mu)b^2] \right] \\
& \left[ \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}a^2) \right] \\
& \left[ \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(b^2 + \frac{1-\mu}{2}a^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}b^2) \quad \frac{1}{3}(a^2 + \frac{1-\mu}{2}a^2) \right]
\end{aligned}$$

*sym*

$$K^* = \frac{Et}{ab(1-\mu^2)} \left[ \frac{\frac{ab}{8}(1+\mu)}{\frac{ab}{8}(1-3\mu)} \quad \frac{\frac{ab}{8}(1-\mu)}{\frac{ab}{8}(1-3\mu)} \quad \frac{\frac{ab}{8}(1+\mu)}{\frac{ab}{8}(1-3\mu)} \quad \frac{\frac{ab}{8}(1-\mu)}{\frac{ab}{8}(1-3\mu)} \quad \frac{\frac{ab}{8}(1+\mu)}{\frac{ab}{8}(1-3\mu)} \quad \frac{\frac{ab}{8}(1-\mu)}{\frac{ab}{8}(1-3\mu)} \right]$$

It can be seen that the calculation of the elemental stiffness matrix is a complicated process, which is not an issue for the computer programming. In the hand calculations, the complicated process can be avoided by directly using Eq. (6.17) to work out the elemental stiffness matrix.

### (6) The elemental stiffness equation

The elemental stiffness equation can be set up by working out the first order derivative of the potential energy with respect to the nodal displacement:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \quad (6.19)$$

where  $\mathbf{K}^e$  is the elemental stiffness matrix,  $\mathbf{q}^e$  is the nodal displacement vector and  $\mathbf{P}^e$  is the external nodal force vector.

## 6.3 The Shape Function for Rectangular Element

The shape functions are an important part of the element and reflect many properties of the element. Regarding the shape function for the rectangular element, there are four properties.

**Property 1** The value of the shape function  $N_i(x)$  at node  $i$  is one and its values at other nodes are zeros.

Taking  $N_1$  as the example to prove this property, according to Eq. (6.9), the formulation for  $N_1$  can be written as:

$$N_1(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta)$$

where  $\xi$  and  $\eta$  are the dimensionless coordinates for the point within the rectangular element.

The coordinates for the four nodes in the dimensionless coordinate system are:

$$\xi_1 = 1, \xi_2 = 1, \xi_3 = -1, \xi_4 = 1$$

$$\eta_1 = 1, \eta_2 = 1, \eta_3 = -1, \eta_4 = -1$$

Therefore, the value of  $N_1$  at node 1 can be obtained, i.e.,  $N_1(1, 1) = 1$ . Its values at other nodes can also be obtained as  $N_1(1, -1) = 0$ ,  $N_1(-1, 1) = 0$ ,  $N_1(-1, -1) = 0$ . Similarly, the first property can be proved at nodes 2, 3 and 4.

**Property 2** At any location within the rectangular element, the sum of the values of all the shape functions equals to one, i.e.,

$$\sum_{i=1}^n N_i(i) = 1$$

where  $n$  is the total number of the elemental nodes.

This property can be proved using the formulations of the shape function, i.e.:

$$\sum_{i=1}^4 N_i = \sum_{i=1}^4 \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta) = \frac{1}{4} \left[ 1 + \xi \sum \xi_i + \eta \sum \eta_i + \eta \xi \sum \xi_i \eta_i \right] = 1 \quad (6.20)$$

It should be noted that the first two properties are the same as those for the shape function for the bar element and these two properties are applicable for all the Lagrange type of finite elements.

**Property 3**  $N_i$  is linearly distributed on the lines parallel to the boundary edges of the rectangular element, and quadratically distributed on the lines non-parallel to the edges!

Again  $N_1$  is used as the example to prove this property. The formulation for  $N_1$  is given as below:

$$N_1(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta)$$

On the edge  $\vec{14}$ , the shape function can be worked out as:

$$N_1 = \frac{1}{4}(1 + 1)(1 + \eta) = \frac{1}{2}(1 + \eta)$$

On the edge  $\vec{23}$ :

$$N_1 = 0$$

On the edge  $\vec{12}$ :

$$N_1 = \frac{1}{4}(1 + \xi)(1 + 1) = \frac{1}{2}(1 + \xi)$$

On the edge  $\vec{34}$ :

$$N_1 = 0$$

On the diagonal line  $\vec{13}$ ,  $\xi = \eta = k$  can be set, then, we can have:

$$N_1 = \frac{1}{4}(1+k)(1+k)$$

On the diagonal line  $\vec{24}$ ,  $\xi = -\eta = k$  can be set, then, we can have:

$$N_1 = \frac{1}{4}(1+k)(1-k)$$

Using the equations for these lines, the shape function for  $N_1(\xi, \eta)$  within the rectangular element can be plotted in Fig. 6.3, in which the shape function is linearly distributed on the lines parallel to the edges and quadratically distributed on the diagonal lines.

**Property 4** On any boundary of the rectangular element, the shape function only depends on the nodes of this edge and is independent of other nodes.

This property can be proved using the formulation of the shape function:

$$N_i(\xi, \eta) = \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta) \quad (i=1, 2, 3, 4)$$

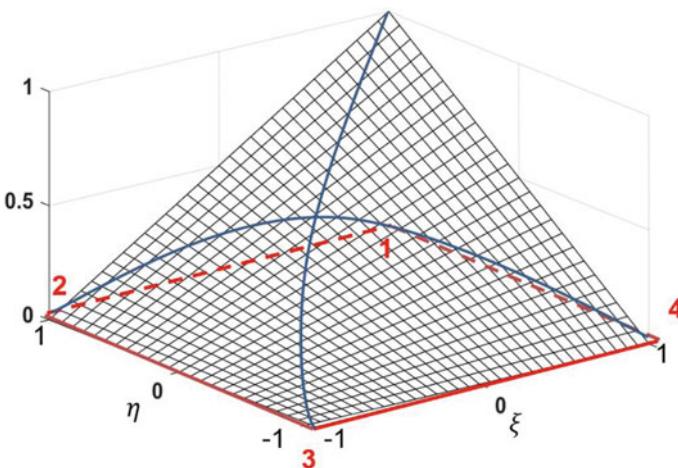
The values of the shape functions on each edge can be worked out.

On the edge  $\vec{14}$ :

$$N_1 = \frac{1}{4}(1+1)(1+\eta) \quad 0$$

$$N_2 = 0$$

$$N_3 = 0$$



**Fig. 6.3** Distribution of the shape function within the rectangular element

$$N_4 = \frac{1}{4}(1+1)(1-\eta)$$

It is clearly seen from the equations above that on the edge  $\vec{14}$ , the shape function only depends on the first and fourth nodes, i.e.,  $N_1$  and  $N_4$  are non-zeros. The shape function on the edge  $\vec{14}$  is independent of the second and third nodes, i.e.,  $N_2$  and  $N_3$  are zeros.

Similarly, on the edge  $\vec{23}$ , we have:

$$N_1 = 0$$

$$N_2 = \frac{1}{4}(1+1)(1+\eta)$$

$$N_3 = \frac{1}{4}(1+1)(1-\eta)$$

$$N_4 = 0$$

Using the fourth property, the continuous properties on the edge shared by two rectangular elements can be obtained, e.g., edge  $\vec{25}$  in Fig. 6.4. The shared edge has the following properties:

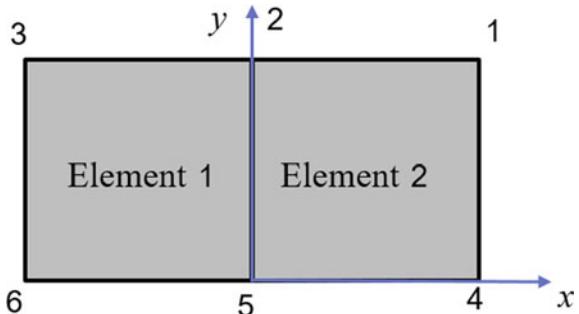
1. The displacement on the shared edge is continuous.
2. The strain component  $\varepsilon_y$  on the shared edge is continuous.
3. All the strain components except for  $\varepsilon_y$  on the shared edge are not continuous.
4. All the stress components on the shared edge are not necessarily continuous.

The first property can be proved as below:

Regarding the first element in Fig. 6.4, the displacement field function can be written as:

$$\begin{aligned} u^{(1)}(x, y) &= a_0^{(1)} + a_1^{(1)}x + a_2^{(1)}y + a_3^{(1)}xy \\ v^{(1)}(x, y) &= b_0^{(1)} + b_1^{(1)}x + b_2^{(1)}y + b_3^{(1)}xy \end{aligned}$$

**Fig. 6.4** Illustration of the edge shared by two rectangular elements



On the common edge  $\overrightarrow{25}$ ,  $x = 0$ , and thus the displacement on the common edge  $\overrightarrow{25}$  can be written as:

$$\begin{aligned} u_{(x=0,y)}^{(1)} &= a_0^{(1)} + a_2^{(1)}y \\ v_{(x=0,y)}^{(1)} &= b_0^{(1)} + b_2^{(1)}y \end{aligned}$$

where  $a_0^{(1)}$ ,  $a_2^{(1)}$ ,  $b_0^{(1)}$  and  $b_2^{(1)}$  are the constants to be determined using the nodal displacement values of nodes 2 and 5, i.e.:

$$\begin{cases} u_{(x=0,y=0)}^{(1)} = a_0^{(1)} = u_5 \\ u_{(x=0,y=h)}^{(1)} = a_0^{(1)} + a_2^{(1)}h = u_2 \\ v_{(x=0,y=0)}^{(1)} = b_0^{(1)} = v_5 \\ v_{(x=0,y=h)}^{(1)} = b_0^{(1)} + b_2^{(1)}h = v_2 \end{cases} \quad (6.21a)$$

It can be seen that  $a_0^{(1)}$ ,  $a_2^{(1)}$ ,  $b_0^{(1)}$  and  $b_2^{(1)}$  can be determined using the nodal displacement values of  $u_2$ ,  $v_2$ ,  $u_5$  and  $v_5$ .

Similarly, regarding the second element, we can have:

$$\begin{cases} u_{(x=0,y=0)}^{(2)} = a_0^{(2)} = u_5 \\ u_{(x=0,y=h)}^{(2)} = a_0^{(2)} + a_2^{(2)}h = u_2 \\ v_{(x=0,y=0)}^{(2)} = b_0^{(2)} = v_5 \\ v_{(x=0,y=h)}^{(2)} = b_0^{(2)} + b_2^{(2)}h = v_2 \end{cases} \quad (6.21b)$$

Comparing Eq. (6.21a) with Eq. (6.21b), we can have:

$$\begin{aligned} a_0^{(1)} &= a_0^{(2)}, a_2^{(1)} = a_2^{(2)} \\ b_0^{(1)} &= b_0^{(2)}, b_2^{(1)} = b_2^{(2)} \end{aligned} \quad (6.22)$$

From Eq. (6.22), we can know that the displacements of the two elements on the common edge  $\overrightarrow{25}$  are equal, i.e.:

$$\begin{aligned} u_{(x=0,y)}^{(1)} &= u_{(x=0,y)}^{(2)} \\ v_{(x=0,y)}^{(1)} &= v_{(x=0,y)}^{(2)} \end{aligned}$$

The second property can be proved using the similar procedure. Regarding the first element, the strain field function on the common edge  $\overrightarrow{25}$  can be written as:

$$\begin{aligned} \varepsilon_{x(x=0,y)}^{(1)} &= a_1^{(1)} + a_3^{(1)}y \\ \varepsilon_{y(x=0,y)}^{(1)} &= b_2^{(1)} \\ \gamma_{xy(x=0,y)}^{(1)} &= a_2^{(1)} + b_1^{(1)} + b_3^{(1)}y \end{aligned} \quad (6.23a)$$

Similarly, the strain field function on the edge  $\vec{25}$  of the second element can be written as:

$$\begin{aligned}\varepsilon_{x(x=0,y)}^{(2)} &= a_1^{(2)} + a_3^{(2)}y \\ \varepsilon_{y(x=0,y)}^{(2)} &= b_2^{(2)} \\ \gamma_{xy(x=0,y)}^{(2)} &= a_2^{(2)} + b_1^{(2)} + b_3^{(2)}y\end{aligned}\quad (6.23b)$$

Comparing the second equations in Eq. (6.23a) and (b), we can have:

$$\varepsilon_{y(x=0,y)}^{(1)} = \varepsilon_{y(x=0,y)}^{(2)}$$

The equation above shows that, on the shared edge  $\vec{25}$  of the two elements, the strain components along the edges, i.e.,  $\varepsilon_y$ , are equal, that is to say, they are continuous.

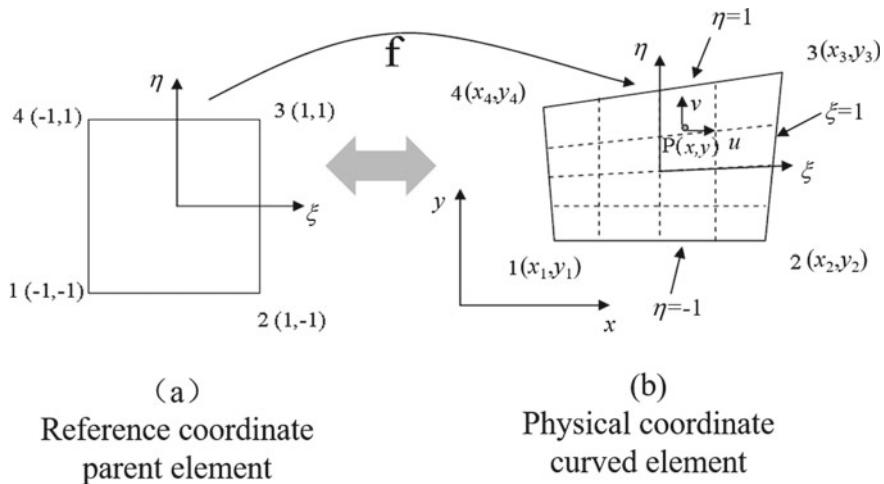
Comparing the first and third equations in Eq. (6.23a) and (b), because  $a_1^{(1)}$  is not necessarily equal to  $a_1^{(2)}$  and  $a_3^{(1)}$  is not necessarily equal to  $a_3^{(2)}$ , and thus  $\varepsilon_x$  is not necessarily continuous. Similarly, because  $a_2^{(1)}$  is not necessarily equal to  $a_2^{(2)}$ ,  $b_1^{(1)}$  is not necessarily equal to  $b_1^{(2)}$  and  $b_3^{(1)}$  is not necessarily equal to  $b_3^{(2)}$ , and thus  $\gamma_{xy}$  is not necessarily continuous.

Regarding the stress distribution on the common edge, all the stress components are not necessarily continuous. From the theory of elasticity, we know the distribution of stress depends on the strain and the elastic constants of the material. The two elements can cover two different materials. Therefore, even when the strain distribution is continuous on the shared edge, the stress may not be continuous on the shared edge, not to say the scenario when the strain is not continuous on the shared edge.

## 6.4 Iso-Parametric Element

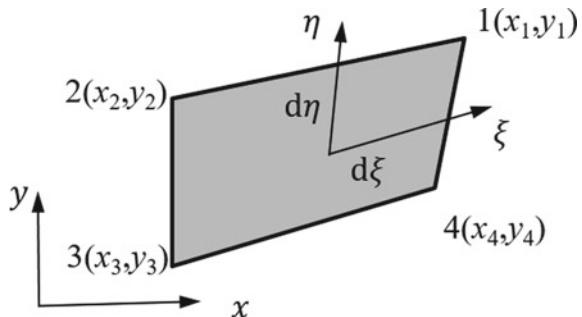
From the previous parts, it can be seen that the accuracy of the 4-node rectangular element is indeed one order higher than that of the 3-node triangular element. Despite of this, the regular well-shape rectangular element (Fig. 6.5a) can be hardly used in the complex engineering problems, because the deformable bodies in the real world are often with highly complicated geometries, e.g., the curved surfaces. To solve this issue, the irregular quadrilateral element (Fig. 6.5b) should be used. Then, the FE formulations for the irregular quadrilateral element should be established. Fortunately, this can be easily done using the FE formulations for the regular rectangular element. In this part, the FE solution for the irregular quadrilateral element is introduced using the iso-parametric transformation.

### (1) Concept of the iso-parametric transformation



**Fig. 6.5** The iso-parametric mapping between the reference and physical coordinates

**Fig. 6.6** The illustration of the differential area



The idea of the iso-parametric transformation is to establish a transformation relation between the regular reference rectangular element (Fig. 6.5a) and the curved irregular quadrilateral element (Fig. 6.5b), and then use the formulations for the regular elements to solve the irregular element. As shown in Fig. 6.5, the reference coordinate system, originating at the center of the element, is placed on the regular element, and the Cartesian coordinate system is placed on the irregular quadrilateral element. It is assumed there is one-to-one mapping for all the points within the element and thus the coordinate transformation between the two systems can be written as (Fig. 6.6):

$$\begin{Bmatrix} x_i \\ y_i \end{Bmatrix} = [f] \begin{Bmatrix} \xi_i \\ \eta_i \end{Bmatrix} \quad (i = 1, 2, 3, 4) \quad (6.24)$$

The point coordinates in the physical coordinate system can be written in terms of the interpolation format, i.e.:

$$x = \sum_{i=1}^m N'_i x_i, \quad y = \sum_{i=1}^m N'_i y_i \quad (6.25)$$

where  $m$  is the number of elemental nodes,  $x_i$  and  $y_i$  are the coordinates of the elemental nodes in the physical coordinate system,  $N'_i$  is the interpolation function.

The coordinate transformation between the two coordinate systems, i.e., the reference and the physical coordinate systems, has been established in Eq. (6.24). Using this transformation, the regular rectangular element in the reference system can be transformed to the distorted irregular quadrilateral element in the physical coordinate system. The rectangular in the reference coordinate system is also called the *mother element* and the corresponding distorted element in the physical coordinate system is called the *sub-element*.

It can be seen that the coordinate transformation in Eq. (6.25) has the same form as the interpolated function, i.e.,  $\Phi = \sum_{i=1}^m N'_i \phi_i$ . When the nodes used for the coordinate transformation and the interpolated function are the same and the same interpolation functions are used, i.e.,  $m = n$ ,  $N'_i = N_i$ , then the transformation is called the *iso-parametric transformation*. When the nodes for the coordinate transformation are more than those for the interpolated function, i.e.,  $m > n$ , then the transformation is called the *super-parametric transformation*. When the nodes for the coordinate transformation are less than those for the interpolated function, i.e.,  $m < n$ , the transformation is called the *sub-parametric transformation*.

## (2) Transformation of the elemental matrix

The displacement field formulations for the 4-node rectangular element are:

$$\begin{aligned} u(x, y) &= a_0 + a_1x + a_2y + a_3xy \\ v(x, y) &= b_0 + b_1x + b_2y + b_3xy \end{aligned}$$

Transforming the coordinates using the interpolation functions, we can have

$$\begin{cases} x = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi\eta \\ y = \alpha_5 + \alpha_6\xi + \alpha_7\eta + \alpha_8\xi\eta \end{cases} \quad (6.26)$$

where  $\alpha_1, \dots, \alpha_8$  are the unknown coefficients, which can be determined using the coordinates of the four nodes, i.e.,

$$\begin{Bmatrix} x_1 \\ y_1 \end{Bmatrix} = [f] \begin{Bmatrix} -1 \\ -1 \end{Bmatrix}, \quad \begin{Bmatrix} x_2 \\ y_2 \end{Bmatrix} = [f] \begin{Bmatrix} 1 \\ -1 \end{Bmatrix}, \quad \begin{Bmatrix} x_3 \\ y_3 \end{Bmatrix} = [f] \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}, \quad \begin{Bmatrix} x_4 \\ y_4 \end{Bmatrix} = [f] \begin{Bmatrix} -1 \\ 1 \end{Bmatrix}$$

Equation (6.26) can be rewritten as

$$\begin{cases} x = N_1x_1 + N_2x_2 + N_3x_3 + N_4x_4 \\ y = N_1y_1 + N_2y_2 + N_3y_3 + N_4y_4 \end{cases} \quad (6.27)$$

where

$$\begin{cases} N_1(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 - \eta) \\ N_2(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 - \eta) \\ N_3(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta) \\ N_4(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 + \eta) \end{cases}$$

In the unified term:

$$N_i(\xi, \eta) = \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta) \quad (i = 1, 2, 3, 4)$$

It can be seen that the interpolation functions used in the coordinate transformation, i.e., Eq. (6.27), are the same as the shape functions used in the displacement field formulations for the rectangular element.

In the finite element analysis, to establish the stiffness equation, the integrations over the volume and areas of the element are often required. Because it is standardized and much easier to perform these integrations in the reference coordinate system, the integrations in the physical coordinate system need to be transformed to those in the reference coordinate system. Therefore, the transformations of the partial derivative and differential area between the two coordinate systems are required to be established.

### (a) Transformation of the partial derivative

In this part, the transformation of the partial derivative between the coordinates, i.e.,  $\left( \frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right) \Rightarrow \left( \frac{\partial}{\partial \xi}, \frac{\partial}{\partial \eta} \right)$  is intended to be established. For an arbitrary function  $\Phi(x, y)$  in the physical coordinate system, its partial derivatives with respect to the reference coordinates  $\xi$  and  $\eta$  are:

$$\begin{cases} \frac{\partial \Phi}{\partial \xi} = \frac{\partial \Phi}{\partial x} \frac{\partial x}{\partial \xi} + \frac{\partial \Phi}{\partial y} \frac{\partial y}{\partial \xi} \\ \frac{\partial \Phi}{\partial \eta} = \frac{\partial \Phi}{\partial x} \frac{\partial x}{\partial \eta} + \frac{\partial \Phi}{\partial y} \frac{\partial y}{\partial \eta} \end{cases} \quad (6.28)$$

Therefore, the transformation relation between the partial derivatives can be obtained:

$$\begin{cases} \frac{\partial}{\partial \xi} = \frac{\partial x}{\partial \xi} \frac{\partial}{\partial x} + \frac{\partial y}{\partial \xi} \frac{\partial}{\partial y} \\ \frac{\partial}{\partial \eta} = \frac{\partial x}{\partial \eta} \frac{\partial}{\partial x} + \frac{\partial y}{\partial \eta} \frac{\partial}{\partial y} \end{cases} \quad (6.29)$$

In terms of the matrix format:

$$\begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = \mathbf{J} \begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{Bmatrix}$$

where

$$\mathbf{J} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}$$

where  $\mathbf{J}$  is called the *Jacobian matrix*. The inverse of the transformation, i.e., Eq. (6.29), can be written as:

$$\begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{Bmatrix} = \mathbf{J}^{-1} \begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = \frac{1}{|\mathbf{J}|} \begin{bmatrix} \frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \end{bmatrix} \begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} \quad (6.30)$$

where  $|\mathbf{J}|$  is the determinant of the Jacobian matrix, i.e.,

$$|\mathbf{J}| = \frac{\partial x}{\partial \xi} \frac{\partial y}{\partial \eta} - \frac{\partial y}{\partial \xi} \frac{\partial x}{\partial \eta}$$

The matrix format of the transformation, i.e., Eq. (6.30), can also be written as the following explicit format:

$$\begin{cases} \frac{\partial}{\partial x} = \frac{1}{|\mathbf{J}|} \left( \frac{\partial y}{\partial \eta} \frac{\partial}{\partial \xi} - \frac{\partial y}{\partial \xi} \frac{\partial}{\partial \eta} \right) \\ \frac{\partial}{\partial y} = \frac{1}{|\mathbf{J}|} \left( -\frac{\partial x}{\partial \eta} \frac{\partial}{\partial \xi} + \frac{\partial x}{\partial \xi} \frac{\partial}{\partial \eta} \right) \end{cases}$$

### (b) Transformation of the differential area

Below the transformation of the differential area between the two coordinate systems is introduced (Fig. 6.6). In the physical coordinate system, the area of the differential parallelogram formed by  $d\xi$  and  $d\eta$  is:

$$dA = |d\xi \times d\eta| \quad (6.31)$$

In the physical coordinate system, the vectors  $d\xi$  and  $d\eta$  can be written in terms of their components as below:

$$\begin{cases} d\xi = \frac{\partial x}{\partial \xi} d\xi \mathbf{i} + \frac{\partial y}{\partial \xi} d\xi \mathbf{j} \\ d\eta = \frac{\partial x}{\partial \eta} d\eta \mathbf{i} + \frac{\partial y}{\partial \eta} d\eta \mathbf{j} \end{cases} \quad (6.32)$$

where  $\mathbf{i}$  and  $\mathbf{j}$  are the unit vectors in the  $x$  and  $y$  directions, respectively. Putting Eq. (6.32) into the area equation, i.e., Eq. (6.31), we have:

$$dA = \begin{vmatrix} \frac{\partial x}{\partial \xi} d\xi & \frac{\partial y}{\partial \xi} d\xi \\ \frac{\partial x}{\partial \eta} d\eta & \frac{\partial y}{\partial \eta} d\eta \end{vmatrix} = |\mathbf{J}| d\xi d\eta \quad (6.33)$$

Equation (6.33) gives the transformation equation for the differential areas between the two coordinate systems. Similarly, in the 3D problem, in the physical coordinate system, i.e.,  $(x, y, z)$ , the volume of the differential hexahedron formed

by  $d\xi$ ,  $d\eta$  and  $d\zeta$  can be written as:

$$d\Omega = d\xi \cdot (d\eta \times d\zeta) \quad (6.34)$$

Similarly,  $d\xi$ ,  $d\eta$  and  $d\zeta$  can be written in terms of their components as below:

$$\begin{cases} d\xi = \frac{\partial x}{\partial \xi} d\xi \mathbf{i} + \frac{\partial y}{\partial \xi} d\xi \mathbf{j} + \frac{\partial z}{\partial \xi} d\xi \mathbf{k} \\ d\eta = \frac{\partial x}{\partial \eta} d\eta \mathbf{i} + \frac{\partial y}{\partial \eta} d\eta \mathbf{j} + \frac{\partial z}{\partial \eta} d\eta \mathbf{k} \\ d\zeta = \frac{\partial x}{\partial \zeta} d\zeta \mathbf{i} + \frac{\partial y}{\partial \zeta} d\zeta \mathbf{j} + \frac{\partial z}{\partial \zeta} d\zeta \mathbf{k} \end{cases} \quad (6.35)$$

Then, the transformation equation for the differential volumes between the two coordinate systems is given as below:

$$d\Omega = \begin{vmatrix} \frac{\partial x}{\partial \xi} d\xi & \frac{\partial y}{\partial \xi} d\xi & \frac{\partial z}{\partial \xi} d\xi \\ \frac{\partial x}{\partial \eta} d\eta & \frac{\partial y}{\partial \eta} d\eta & \frac{\partial z}{\partial \eta} d\eta \\ \frac{\partial x}{\partial \zeta} d\zeta & \frac{\partial y}{\partial \zeta} d\zeta & \frac{\partial z}{\partial \zeta} d\zeta \end{vmatrix} = |\mathbf{J}| d\xi d\eta d\zeta \quad (6.36)$$

### (c) Transformation of the element

Below the transformation of the FE formulations for the elements between the two coordinate systems is introduced. In the reference coordinate system  $(\xi, \eta)$ , the displacement field function for the 4-node rectangular element can be written as

$$\begin{Bmatrix} u(\xi, \eta) \\ v(\xi, \eta) \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \mathbf{q}^e = \mathbf{N}(\xi, \eta) \mathbf{q}^e$$

where  $\mathbf{q}^e = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T$ .

Based on Eq. (6.27), the coordinate transformation between the two coordinate systems is:

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = \tilde{\mathbf{N}}(\xi, \eta) \tilde{\mathbf{q}}$$

To transform the elemental stiffness matrix from the physical coordinate system  $(x, y)$  to the reference coordinate system  $(\xi, \eta)$ , the transformation for the elemental strain-displacement matrix needs to be established, i.e.,

$$\mathbf{B}\left(x, y, \frac{\partial}{\partial x}, \frac{\partial}{\partial y}\right) = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \mathbf{N}(x, y) = \mathbf{B}^*\left(\xi, \eta, \frac{\partial}{\partial \xi}, \frac{\partial}{\partial \eta}\right) \quad (6.37)$$

Then the transformation of the elemental stiffness matrix from the physical coordinate system to the reference coordinate system can be written as:

$$\begin{aligned}\mathbf{K}^e &= \int_{A^e} \mathbf{B}^T \left( x, y, \frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right) \mathbf{D} \mathbf{B} \left( x, y, \frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right) dA \cdot t \\ &= \int_{-1}^1 \int_{-1}^1 \mathbf{B}^{*T} \left( \xi, \eta, \frac{\partial}{\partial \xi}, \frac{\partial}{\partial \eta} \right) \mathbf{D} \mathbf{B}^* \left( \xi, \eta, \frac{\partial}{\partial \xi}, \frac{\partial}{\partial \eta} \right) |\mathbf{J}| d\xi d\eta \cdot t \quad (6.38)\end{aligned}$$

where  $t$  is the thickness of the rectangular element.

### (3) Conditions of the iso-parametric transformation

In the previous parts, the iso-parametric transformation of the rectangular elements is introduced. However, it should be noted that not all rectangular elements can be transformed. From Eq. (6.30), it can be known that the condition of the one-to-one transformation between the coordinates is that the determinant of the Jacobian, i.e.,  $|\mathbf{J}|$ , should not equal to zero. The iso-parametric transformation is one kind of coordinate transformation and thus should also obey this transformation condition. Based on Eq. (6.33), if  $|\mathbf{J}| = 0$ , the area of the differential area in the global coordinate is zero, i.e., the differential area formed by  $d\xi$  and  $d\eta$  in the reference coordinate corresponds to one point in the physical coordinate system. This kind of transformation (mapping) is obvious not one-to-one mapping. On the other hand, if  $|\mathbf{J}| = 0$ ,  $\mathbf{J}^{-1}$  will not exist, and then the partial derivatives between the two coordinate systems, i.e., Eq. (6.30) cannot be performed. Now let's have a look at how to avoid the occurrence of  $|\mathbf{J}| = 0$ , i.e., to guarantee the one-to-one iso-parametric transformation.

Based on Eq. (6.31), the differential area can be written as:

$$dA = |d\xi \times d\eta| = |d\xi||d\eta| \sin(d\xi, d\eta)$$

and

$$dA = \begin{vmatrix} \frac{\partial x}{\partial \xi} d\xi & \frac{\partial y}{\partial \xi} d\xi \\ \frac{\partial x}{\partial \eta} d\eta & \frac{\partial y}{\partial \eta} d\eta \end{vmatrix} = |\mathbf{J}| d\xi d\eta$$

Then, we have:

$$|\mathbf{J}| = \frac{|d\xi||d\eta| \sin(d\xi, d\eta)}{d\xi d\eta}$$

Therefore, to avoid  $|\mathbf{J}| = 0$ , the following three conditions should be avoided

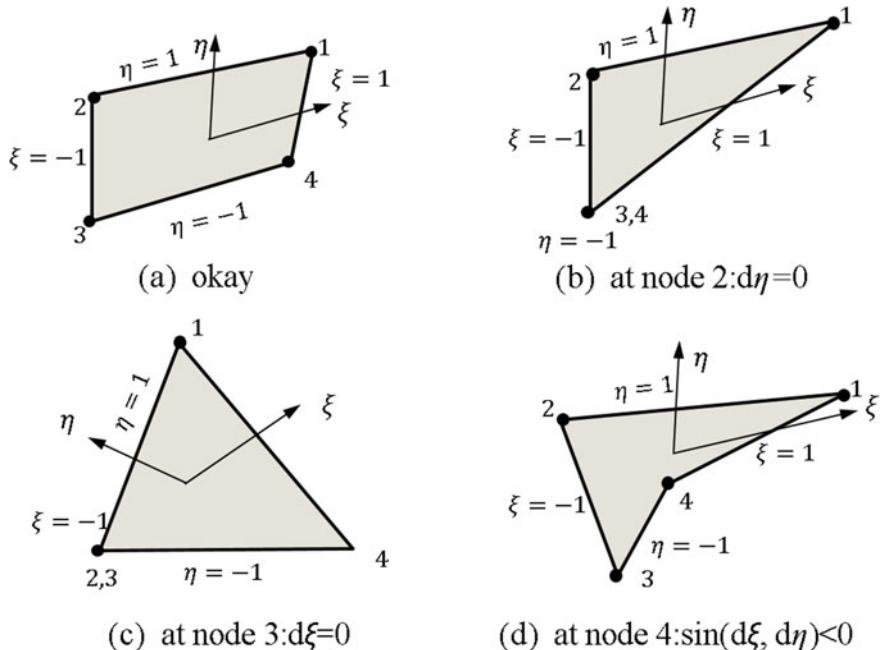
$$|d\xi| = 0$$

$$|d\eta| = 0$$

$$\sin(d\xi, d\eta) = 0$$

Figure 6.7a shows the situation when the one-to-one iso-parametric transformation can be performed. Figure 6.7b-d shows the situations when the one-to-one iso-parametric transformation cannot be performed. In Fig. 6.7b, the third and fourth nodes are merged into one node, where  $|d\xi| = 0$ . In Fig. 6.7c, the second and third nodes are merged into one node, where  $|d\eta| = 0$ . In Fig. 6.7d, the angle from edge  $\bar{1}2$  to edge  $\bar{2}3$  is acute, and thus  $\sin(d\xi, d\eta) > 0$ . However, the angle from edge  $\bar{1}4$  to edge  $\bar{4}3$  is obtuse, and thus  $\sin(d\xi, d\eta) < 0$ . Because  $\sin(d\xi, d\eta)$  is continuously changed within the element,  $\sin(d\xi, d\eta) = 0$  would exist within the element, i.e., the situation that  $d\xi$  and  $d\eta$  on the same line will occur

In summary, to guarantee the one-to-one iso-parametric transformation between the two coordinate systems, in the rectangular element, the merging of two nodes into one should be avoided. Additionally, the inner angle between the edges of the rectangular larger than 180 degrees should be avoided.



**Fig. 6.7** The transformable and non-transformable iso-parametric elements

## 6.5 Numerical Integration

Generally, it is challenging to perform the numerical integration, especially when the function to be integrated is complex. However, from the previous chapters, we know the integration formulations are involved in the FE calculations, e.g., in the calculation of the elemental stiffness matrix, i.e.,

$$\mathbf{K}^e = \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \quad (6.39)$$

To avoid performing the numerical integration, some methods, e.g., the Newton–Cotes and Gauss integration methods, are developed. The idea of these methods is to select some points, i.e., the integration points, work out the values of these functions at these points and then use the weighted summation of these functions to approximate the value of the original integration. The process can be expressed using the following equation:

$$\int_{-1}^1 f(\xi) d\xi \approx \sum_{k=1}^n A_k f(\xi_k) \quad (6.40)$$

where  $f(\xi)$  is the function to be integrated,  $n$  is the number of integration,  $A_k$  is the weighted coefficients of the integration,  $\xi_i$  is the position of the integration points. When  $n$  is determined,  $A_k$  and  $\xi_k$  can also be determined using the methods below.

The following integration is used to illustrate the idea of the numerical integration.

$$I = \int_{-1}^1 f(\xi) d\xi \quad (6.41)$$

The first step is to construct a polynomial function  $\varphi(\xi_i)$ , and make their values equivalent to those of  $f(\xi)$  at the points  $(\xi_i, i = 1, 2, \dots, n)$ , i.e.,

$$\varphi(\xi_i) = f(\xi_i) (i = 1, 2, \dots, n)$$

then the polynomial function  $\varphi(\xi)$  is used to approximately replace the original function  $f(\xi)$ , and the integration, i.e., Eq. (6.41), can be rewritten as:

$$I = \int_{-1}^1 f(\xi) d\xi \approx \int_{-1}^1 \varphi(\xi) d\xi$$

Below the method for constructing the polynomial function  $\varphi(\xi)$  which can approximate  $f(\xi)$  as well as possible is discussed. The following two approximation methods are introduced. It should be noted that  $\varphi(\xi)$  is a polynomial function where the Taylor series terms, i.e.,  $1, x, x^2, \dots, x^n$  are used. The reason why the polynomial function is constructed instead of other forms of functions is that it is much easier to work out the values of the polynomial function at certain points than those of other functions. Additionally, the mathematical operations, e.g., the differential and integration operations, are also easier when the polynomial function is used.

### 6.5.1 The Newton–Cotes Integration Method

In the Newton–Cotes integration method, the Lagrange interpolation polynomial function is used for the approximation function  $\varphi(\xi)$ , i.e.:

$$\varphi(\xi) = a_0 + a_1\xi + \dots + a_{n-1}\xi^{n-1} = \sum_{i=1}^n l_i^{n-1}(\xi) f(\xi_i) \quad (6.42)$$

where,  $a_0, a_1, \dots, a_{n-1}$  are the constants,  $l_i^{n-1}(\xi)$  is the  $(n - 1)$ th order Lagrange interpolation function, i.e.,

$$l_i^{n-1}(\xi) = \frac{(\xi - \xi_1)(\xi - \xi_2) \cdots (\xi - \xi_{i-1})(\xi - \xi_{i+1}) \cdots (\xi - \xi_n)}{(\xi_i - \xi_1)(\xi_i - \xi_2) \cdots (\xi_i - \xi_{i-1})(\xi_i - \xi_{i+1}) \cdots (\xi_i - \xi_n)} = \prod_{\substack{j=1 \\ j \neq i}}^n \frac{(\xi - \xi_j)}{(\xi_i - \xi_j)} \quad (6.43)$$

Based on Eq. (6.43), we have

$$\begin{aligned} l_i^{n-1}(\xi_j) &= \delta_{ij} \\ \varphi(\xi_i) &= f(\xi_i) \end{aligned}$$

where  $\xi_j$  ( $j = 1, \dots, n$ ) are the locations of the integration points,  $\delta_{ij}$  is the Kronecker symbol, which has the following property:

$$\delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases}$$

It should be noted that in the Newton–Cotes numerical method, the integration points are equally distributed in the integration region, i.e.,

$$\xi_i = a + ih \quad (i = 0, 1, 2, \dots, n - 1)$$

where  $h$  is the distance between the integration points. For the 1D integration region  $[a b]$ ,  $h = (b - a)/(n - 1)$ .

Using Eq. (6.43), the integration of the original function  $f(\xi)$  can be transformed to the weighted summation of the values of the polynomial function  $\varphi(\xi)$  at the integration points, i.e.,

$$\begin{aligned} I &= \int_{-1}^1 f(\xi) d\xi \approx \int_{-1}^1 \varphi(\xi) d\xi \\ &= \int_{-1}^1 \sum_{i=1}^n l_i^{n-1}(\xi) f(\xi_i) d\xi \\ &= \sum_{i=1}^n \left[ \left( \int_{-1}^1 l_i^{n-1}(\xi) d\xi \right) f(\xi_i) \right] \\ &= \sum_{i=1}^n A_i f(\xi_i) = \sum_{i=1}^n A_i \varphi(\xi_i) \end{aligned} \quad (6.44)$$

where  $A_i$  is the integration weighting factor determined by the location and the number of the integration points, and is given as below

$$A_i = \int_{-1}^1 l_i^{n-1}(\xi) d\xi \quad (6.45)$$

**Example 6.1** Below an example is used to illustrate the process of approximating the numerical integration using the Newton–Cotes numerical method. As shown in Fig. 6.8, the question is to work out the integration of  $f(\xi)$  at the 1D region  $[a b]$  using two integration points, i.e., to work out the coefficients in the following equation.

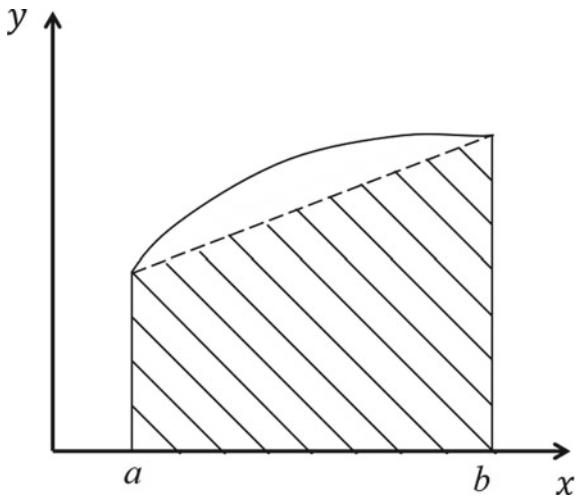
$$I = \int_a^b f(\xi) d\xi \approx A_1 f(\xi_1) + A_2 f(\xi_2)$$

**Solution** First, because the integration points are equally distributed in the Newton–Cotes numerical method, the location of the integration points can be easily worked out, i.e.

$$\xi_1 = a, \xi_2 = b.$$

Second, using Eq. (6.45), the integration weighting factors can be worked out:

**Fig. 6.8** The calculation of the integration using the Newton–Cotes numerical method



$$A_1 = \int_a^b l_i^{n-1}(\xi) d\xi = \int_a^b l_1^1(\xi) d\xi = \int_a^b \frac{\xi - \xi_2}{\xi_1 - \xi_2} d\xi = \frac{b - a}{2}$$

$$A_2 = \int_a^b l_i^{n-1}(\xi) d\xi = \int_a^b l_2^1(\xi) d\xi = \int_a^b \frac{\xi - \xi_1}{\xi_2 - \xi_1} d\xi = \frac{b - a}{2}$$

Therefore,

$$I = \int_a^b f(\xi) d\xi \approx \frac{b - a}{2} (f(a) + f(b)) \quad (6.46)$$

It can be seen that Eq. (6.46) is the classical Trapezoidal area formula. In Fig. 6.8, it is seen that Eq. (6.46) represents the shaded area, while the original integration represents the area under the curved line. It can be seen that there are some errors between the two areas. Therefore, the errors in the Newton–Cotes numerical method should be noted when this method is used. The errors should be controlled within the allowed accuracy. If the errors are too high, some strategies should be used to reduce the errors. One strategy is to increase the number of the integration points, but the computation complexity will be increased accordingly. Another strategy is to change the numerical method, e.g., using the Gauss integration method, which has a higher accuracy.

### 6.5.2 The Gauss Integration Method

In the Gauss integration method, the integration points are not evenly distributed within the given domain. The accuracy of the interpolation function is increased by increasing the order of the interpolation function from  $n$  to  $(2n-1)$ , which is realized by adjusting the locations of the integration points. The interpolation function in the Gauss integration method can be written as:

$$\varphi(\xi) = \sum_{i=1}^n l_i^{n-1}(\xi) f(\xi_i) + \sum_{i=0}^{n-1} \beta_i \xi^i P(\xi)$$

where  $l_i^{n-1}(\xi)$  is the  $(n-1)$ th order Lagrange interpolation function,  $\beta_i$  is the coefficient and  $P(\xi)$  is a  $n$ th order polynomial function, which is given as below:

$$P(\xi) = (\xi - \xi_1)(\xi - \xi_2) \cdots (\xi - \xi_n) = \prod_{i=1}^n (\xi - \xi_i), \quad (i = 1, 2, \dots, n) \quad (6.47)$$

The coordinates for the integration points in the Gauss integration method can be determined using the following equation:

$$\int_{-1}^1 \xi^i P(\xi) d\xi = 0 \quad (i = 0, 1, 2, \dots, n-1) \quad (6.48)$$

Therefore, the integration of the original function can be transformed as below:

$$\begin{aligned} I &= \int_{-1}^1 f(\xi) d\xi \approx \int_{-1}^1 \varphi(\xi) d\xi \\ &= \sum_{i=1}^n \left[ \left( \int_{-1}^1 l_i^{n-1}(\xi) d\xi \right) f(\xi_i) \right] + \sum_{i=0}^{n-1} \int_{-1}^1 \beta_i \xi^i P(\xi) d\xi + R \\ &= \sum_{i=1}^n A_i f(\xi_i) + R \end{aligned} \quad (6.49)$$

where

$$A_i = \int_{-1}^1 l_i^{n-1}(\xi) d\xi \quad (6.50)$$

and the formulation for  $P(\xi)$  is given in Eq. (6.47).  $P(\xi)$  has the following two obvious properties:

1. At the integration points  $\xi_i$ ,  $P(\xi_i) = 0$
2. The polynomial points  $P(\xi)$  and  $\xi^0, \xi^1, \xi^2, \dots, \xi^{(n-1)}$  are orthogonal

The first property can be easily proved by putting the  $\xi_i$  into the formulation of  $P(\xi)$ , i.e., Eq. (6.47). The second property can be obviously proved using Eq. (6.48).

Below the Gauss integration formulae for the one and two integration points are given:

When one Gauss integration point, i.e.,  $n = 1$ , is used, the only weighting factor  $A_1$  can be worked out as below:

$$A_1 = \int_{-1}^1 l_i^{n-1}(\xi) d\xi = \int_{-1}^1 1 d\xi = 2$$

The only Gauss point can be worked out using Eq. (6.48):

$$\int_{-1}^1 \xi^0 P(\xi_1) d\xi = \int_{-1}^1 \xi^0 (\xi - \xi_1) d\xi = 0$$

Solving the equations above, we have:

$$\xi_1 = 0$$

Then, Eq. (6.49) can be rewritten as:

$$I = \int_{-1}^1 f(\xi) d\xi \approx 2f(0) \quad (6.51)$$

When two Gauss integration points,  $n = 2$ , are used, Eq. (6.49) can be rewritten as

$$I = \int_{-1}^1 f(\xi) d\xi \approx A_1 f(\xi_1) + A_2 f(\xi_2)$$

The processes for calculating the integration points  $\xi_1$  and  $\xi_2$ , and the weighting factors  $A_1$  and  $A_2$  are demonstrated in the following example.

**Example 6.2** Below the same example is used to illustrate the calculation process using the Gauss integration method. Two integration points are chosen in Eq. (6.49), which can then be written as following:

$$I = \int_{-1}^1 f(\xi) d\xi \approx A_1 f(\xi_1) + A_2 f(\xi_2) \quad (6.52)$$

**Solution** When two integration points are used,  $P(\xi)$  can be written as:

$$P(\xi) = (\xi - \xi_1)(\xi - \xi_2)$$

Below Eq. (6.48) is used to work out the locations for the two integration points. We set  $i = 0$  and  $1$ , respectively, then the following equations are obtained:

$$\int_{-1}^1 \xi(\xi - \xi_1)(\xi - \xi_2) d\xi = \frac{2}{3} + 2\xi_1\xi_2 = 0 \quad (6.53a)$$

$$\int_{-1}^1 \xi(\xi - \xi_1)(\xi - \xi_2) d\xi = -\frac{2}{3}(\xi_1 + \xi_2) = 0 \quad (6.53b)$$

From the two equations above,  $\xi_1$  and  $\xi_2$  can be solved and their values are:

$$\xi_1 = -\frac{1}{\sqrt{3}}, \xi_2 = \frac{1}{\sqrt{3}}$$

Then Eq. (6.50) is used to work out the weighted coefficients  $A_1$  and  $A_2$

$$A_1 = \int_{-1}^1 l_i^{n-1}(\xi) d\xi = \int_{-1}^1 l_1^1(\xi) d\xi = \int_{-1}^1 \frac{\xi - \xi_2}{\xi_1 - \xi_2} d\xi = 1 \quad (6.54)$$

Similarly

$$A_2 = 1$$

Therefore, the original integration can be approximated using the following equation:

$$I = \int_{-1}^1 f(\xi) d\xi \approx f(-\frac{1}{\sqrt{3}}) + f(\frac{1}{\sqrt{3}}) \quad (6.55)$$

From the equation above, it is seen that the problem of calculating the numerical integration has been transferred to work out the values of the polynomial function at the integration points, which is much easier for both computer programming and hand calculations.

It should be noted that the four coefficients,  $A_1$ ,  $A_2$ ,  $\xi_1$  and  $\xi_2$ , can also be worked out using the direct method as follows. We set  $f(\xi)$  to  $1$ ,  $\xi$ ,  $\xi^2$ ,  $\xi^3$  and put them to Eq. (6.51), respectively. Then the following four equations can be obtained:

$$2 = A_1 + A_2$$

$$0 = A_1 \xi_1 + A_2 \xi_2$$

$$\frac{2}{3} = A_1 \xi_1^2 + A_2 \xi_2^2$$

$$0 = A_1 \xi_1^3 + A_2 \xi_2^3$$

Solving the equations above, we can obtain  $\xi_1 = -\frac{1}{\sqrt{3}}$ ,  $\xi_2 = \frac{1}{\sqrt{3}}$ ,  $A_1 = A_2 = 1$ . Therefore, we have the following 2-point Gauss integration formula:

$$I = \int_{-1}^1 f(\xi) d\xi \approx f\left(-\frac{1}{\sqrt{3}}\right) + f\left(\frac{1}{\sqrt{3}}\right) \quad (6.56)$$

It can be seen although different methods are used, the same 2-point Gauss integration formula is obtained.

The equation given in Eq. (6.49) is the Gauss integration formulation for the 1D case, but it can be extended to the 2D and 3D cases.

In the 2D case:

$$\begin{aligned} I &= \int_{-1}^1 \int_{-1}^1 f(\xi, \eta) d\xi d\eta = \int_{-1}^1 \sum_{j=1}^n A_j f(\xi_j, \eta) d\eta \\ &= \sum_{i=1}^n \left[ A_i \sum_{j=1}^n A_j f(\xi_j, \eta_i) \right] = \sum_{i=1}^n \sum_{j=1}^n A_i A_j f(\xi_j, \eta_i) \\ &= \sum_{i,j=1}^n A_{ij} f(\xi_j, \eta_i) \end{aligned} \quad (6.57)$$

where  $A_{ij} = A_i A_j$ ,  $\xi_j$  and  $\eta_i$  are the integration points,  $A_i$  and  $A_j$  are the weighted coefficients, which can be obtained from the 1D Gauss integration formulations.

In the 3D case:

$$\begin{aligned}
 I &= \int_{-1}^1 \int_{-1}^1 \int_{-1}^1 f(\xi, \eta, \zeta) d\xi d\eta d\zeta \\
 &= \sum_{m=1}^n \sum_{j=1}^n \sum_{i=1}^n A_i A_j A_m f(\xi_i, \eta_j, \zeta_m) \\
 &= \sum_{m,j,i=1}^n A_{ijm} f(\xi_i, \eta_j, \zeta_m)
 \end{aligned} \tag{6.58}$$

where  $A_{ijm} = A_i A_j A_m$ ,  $\xi_i$ ,  $\eta_j$  and  $\zeta_m$  are the integration points,  $A_i$ ,  $A_j$  and  $A_m$  are the weighted coefficients, which can be obtained from the 1D Gauss integration formulations.

At the end of this chapter, an example is used to demonstrate the finite element analysis procedure using the rectangular element.

## 6.6 An Example of the FE Analysis Using Rectangular Element

**Example 6.3** A plane structure is shown in Fig. 6.9. The height, the width and the thickness of the plane structure are all equal to 1.0. The mechanical properties of the plane structure are  $E = 1.0$ ,  $v = 0.25$ . The question is to find the displacement, the strain and the stress field functions, the reaction force, the strain energy and the total potential energy in the structure.

Two different types of the boundary conditions are defined in the plane rectangular structure. The first is given in Fig. 6.9a, where the displacement boundary conditions

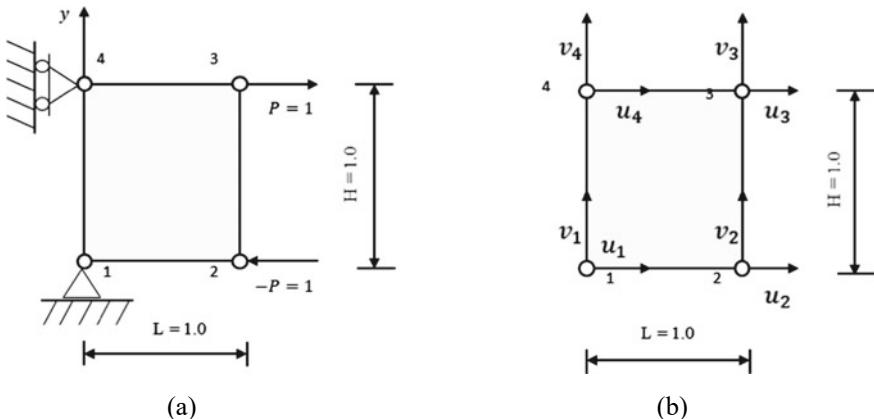
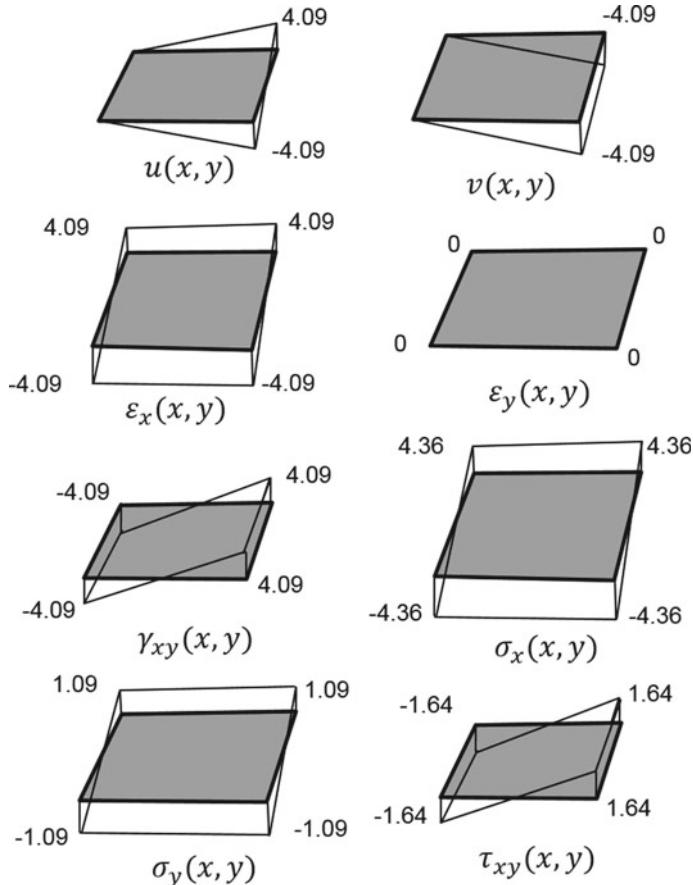


Fig. 6.9 The plane rectangular structure



**Fig. 6.10** Distribution of the displacement, the strain and the stress field formulations in the structure solved using one rectangular element

and the traction boundary conditions are given, i.e.

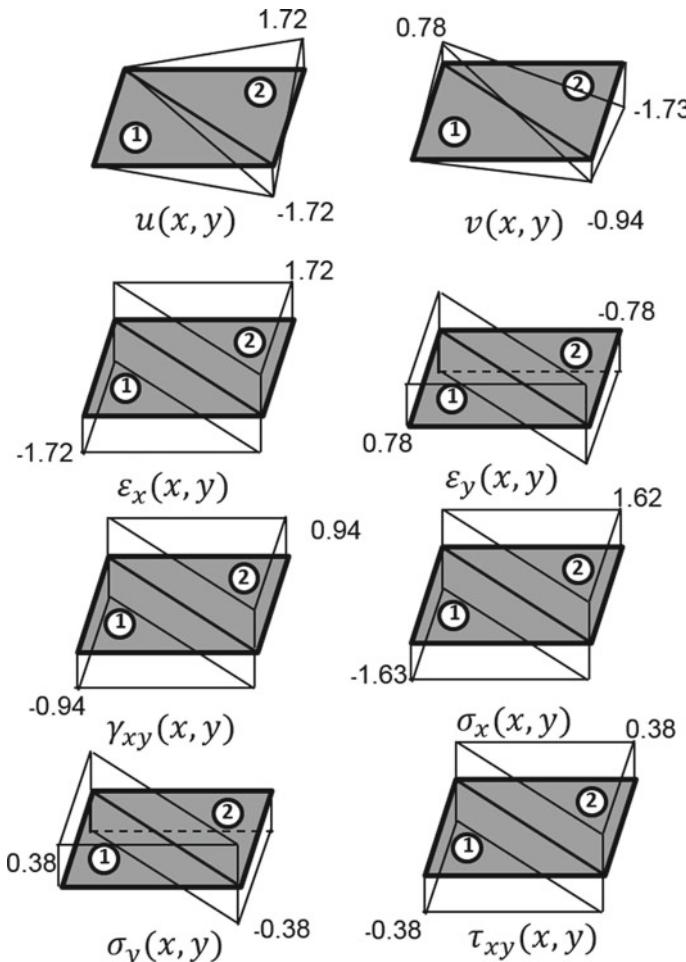
$$u_1 = 0, v_1 = 0, u_4 = 0$$

$$P_{x2} = -1, P_{y2} = 0, P_{x3} = 1, P_{y3} = 0, P_{y4} = 0$$

In the second scenario (Fig. 6.9b), the nodal displacement boundary conditions are given:

$$u_1 = 0.1, u_2 = 0.2, u_3 = 0.4, u_4 = 0.3$$

$$v_1 = 0.1, v_2 = 0.1, v_3 = 0.2, v_4 = 0.3$$

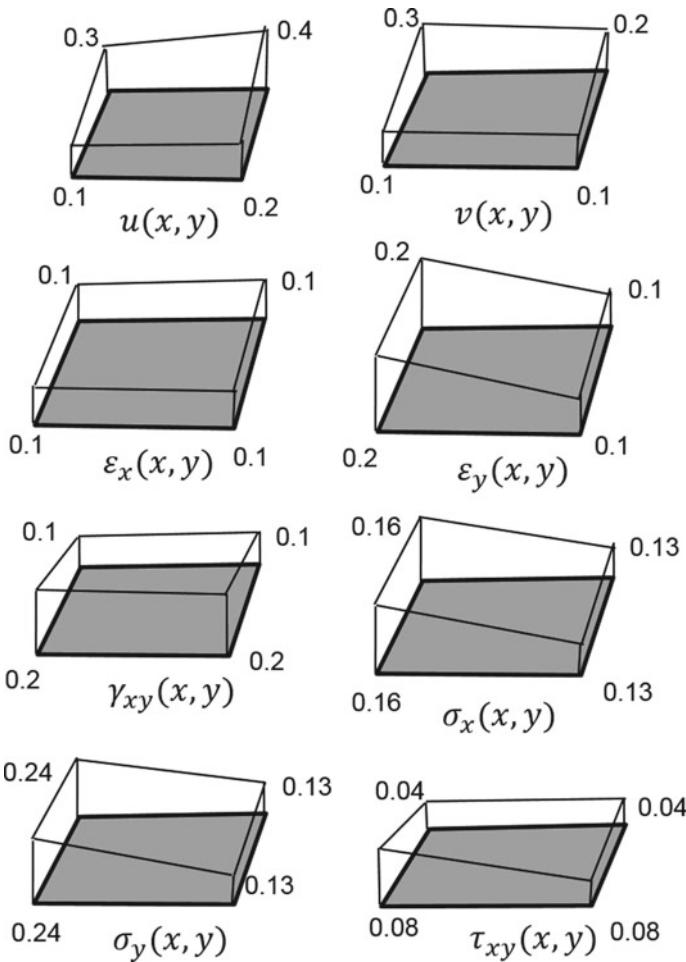


**Fig. 6.11** Distribution of the displacement, the strain and the stress field formulations in the structure solved using two triangular elements

**Solution** Below the FE calculation process using two strategies is demonstrated. The first is to use two 3-node triangular elements to solve the problems and the second is to use one 4-node rectangular element to solve the problems. No matter which strategy is used, the nodal displacement vector for the plane structure is given as below:

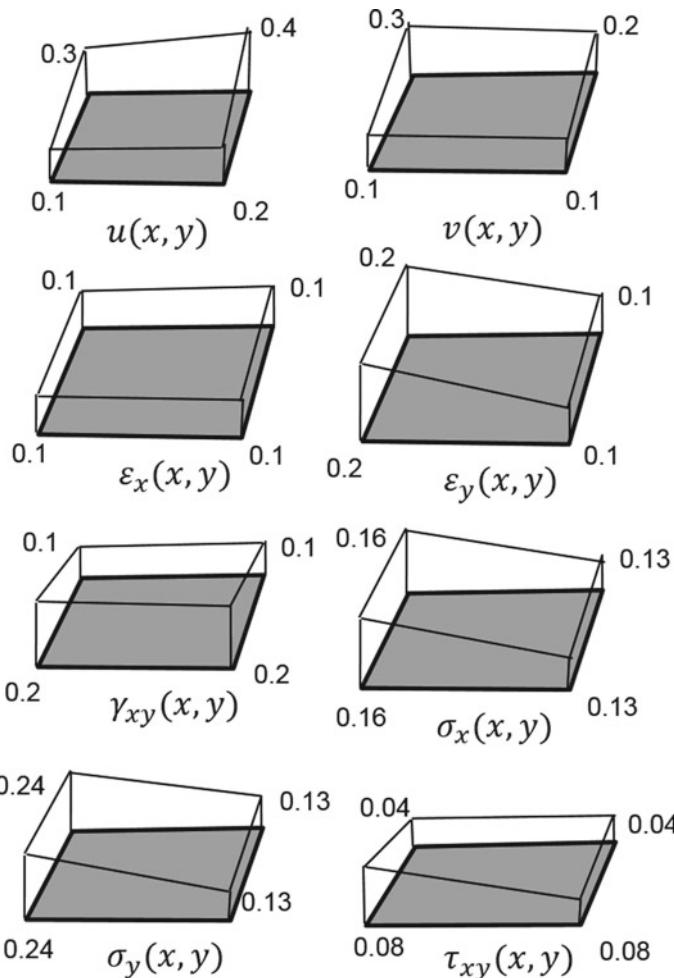
$$\mathbf{q} = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T \quad (6.59)$$

When the 4-node rectangular element is used to mesh the structure, the elemental stiffness matrix can be worked out using Eq. (6.16), which gives the following answer



**Fig. 6.12** The distribution of the displacement, the strain and the stress field formulations in the scenario of the second type of boundary condition solved using one rectangular element

$$\mathbf{K} = \begin{bmatrix} 0.49 & 0.17 & -0.29 & -0.03 & -0.24 & -0.17 & 0.04 & 0.03 \\ 0.17 & 0.49 & 0.03 & 0.04 & -0.17 & -0.24 & -0.03 & -0.29 \\ -0.29 & 0.03 & 0.49 & -0.17 & 0.04 & -0.03 & -0.24 & 0.17 \\ -0.03 & 0.04 & -0.17 & 0.49 & 0.03 & -0.29 & 0.17 & -0.24 \\ -0.24 & -0.17 & 0.04 & 0.03 & 0.49 & 0.17 & -0.29 & -0.03 \\ -0.17 & -0.24 & -0.03 & -0.29 & 0.17 & 0.489 & 0.03 & 0.04 \\ 0.04 & -0.03 & -0.24 & 0.17 & -0.29 & 0.03 & 0.49 & -0.17 \\ 0.03 & -0.29 & 0.17 & -0.24 & -0.03 & 0.04 & -0.17 & 0.49 \end{bmatrix} \quad (6.60)$$



**Fig. 6.13** The distribution of the displacement, the strain and the stress field formulations in the scenario of the second type of boundary condition solved using two triangular elements

Because only one element is used, the elemental stiffness matrix is the global stiffness matrix, and then the global stiffness equation is:

$$\mathbf{K}\mathbf{q} = \mathbf{P} \quad (6.61)$$

where  $\mathbf{q}$  is the nodal displacement vector and  $\mathbf{P}$  is the nodal force vector.

The elemental displacement, the strain and the stress field formulations can be worked out using the following equations:

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \mathbf{N}(x, y)\mathbf{q} \quad (6.62)$$

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \mathbf{B}(x, y)\mathbf{q}$$

$$\boldsymbol{\sigma} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \mathbf{DB}(x, y)\mathbf{q}$$

(a) **In case of the first type of boundary condition**

In case of the first type of boundary conditions, putting the boundary conditions into the stiffness Eq. (6.61), and then the nodal displacement and the reaction force vectors can be worked out

$$v_2 = -4.09, v_3 = -4.09, v_4 = 0$$

$$u_2 = -4.09, u_3 = 4.09$$

$$R_{x1} = 1, R_{y1} = 0, R_{x4} = -1$$

The nodal displacement vector for the plane structure can be written as:

$$\begin{aligned} \mathbf{q} &= [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T \\ &= [0 \ 0 \ -4.09 \ -4.09 \ 4.09 \ -4.09 \ 0 \ 0]^T \end{aligned}$$

Then using Eq. (6.62), the elemental displacement field formulation is

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{Bmatrix} -4.09(x - 2xy) \\ -4.09x \end{Bmatrix}$$

The strain field formulation is

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} -4.09(1 - 2y) \\ 0 \\ -4.09(1 - 2x) \end{Bmatrix}$$

The stress field formulation is

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} -4.36(1-2y) \\ -1.09(1-2y) \\ -1.63(1-2x) \end{Bmatrix}$$

The distributions of the displacement, the strain and the stress field formulations are plotted in Fig. 6.10:

The system strain energy is

$$U = \frac{1}{2} \mathbf{q}^T \mathbf{K} \mathbf{q} = 4.09$$

The external work is

$$W = \mathbf{P}^T \mathbf{q} = 8.18$$

Then the system potential is

$$\Pi = U - W = -4.09$$

The problem can also be solved using two triangular elements using the formulae given in previous chapter. Below the answers are directly given.

The displacement field in the first element is:

$$\mathbf{u}^{(1)} = \begin{Bmatrix} u^{(1)} \\ v^{(1)} \end{Bmatrix} = \begin{Bmatrix} -1.72x \\ -0.94x + 0.78y \end{Bmatrix}$$

The displacement field in the second element is:

$$\mathbf{u}^{(2)} = \begin{Bmatrix} u^{(2)} \\ v^{(2)} \end{Bmatrix} = \begin{Bmatrix} -1.72(x+2y-2) \\ 1.56 - 2.5x - 0.78y \end{Bmatrix}$$

The strain field in the first element is:

$$\boldsymbol{\epsilon}^{(1)} = [\varepsilon_x \ \varepsilon_y \ \gamma_{xy}]^T = [-1.72 \ 0.78 \ -0.94]^T$$

The strain field in the second element is:

$$\boldsymbol{\epsilon}^{(2)} = [\varepsilon_x \ \varepsilon_y \ \gamma_{xy}]^T = [1.72 \ -0.78 \ 0.94]^T$$

The stress field in the first element is:

$$\boldsymbol{\sigma}^{(1)} = [\sigma_x \ \sigma_y \ \tau_{xy}]^T = [-1.6 \ 0.38 \ -0.38]^T$$

The stress field in the second element is:

$$\boldsymbol{\sigma}^{(2)} = [\sigma_x \ \sigma_y \ \tau_{xy}]^T = [1.62 \ -0.38 \ 0.38]^T$$

The distributions of the displacement, the strain and the stress field formulations in the two triangular elements are plotted in Fig. 6.11.

The system strain energy is

$$U = \frac{1}{2} \mathbf{q}^T \mathbf{K} \mathbf{q} = 1.72$$

The external work is

$$W = \mathbf{P}^T \mathbf{q} = 3.44$$

Then the system potential is

$$\Pi = U - W = -1.72$$

It can be seen that the displacement, the strain, the stress and the energies calculated using one rectangular element are different from those calculated using two triangular elements. The results from both methods are correct and the differences are reasonable. Because the FE method is a numerical method and the corresponding answers are the approximation solutions. Therefore, different approximation solutions can be obtained using different FE calculation strategies. However, in the real engineering problems, the errors between the FE solution and the exact solution should be controlled. Otherwise, the FE solutions will become meaningless. The strategies to control the accuracy of the FE solution include the increase of the element number, the selection of the high order element, etc.

### (b) In case of the second type of boundary condition

In the second type of boundary conditions, the nodal displacements are given. Therefore, the nodal force vector only contains the nodal reaction forces. In this scenario, the minimal complemental energy principle should be used to establish the FE equations. Putting the nodal displacements and the nodal reaction forces into the stiffness equation, i.e., Eq. (6.61), and then the nodal displacement and reaction force vectors can be worked out

$$\begin{aligned} \mathbf{q} &= [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T \\ &= [0.1 \ 0.1 \ 0.2 \ 0.1 \ 0.4 \ 0.2 \ 0.3 \ 0.3]^T \end{aligned}$$

$$\begin{aligned} \mathbf{R} &= [R_{x1} \ R_{y1} \ R_{x2} \ R_{y2} \ R_{x3} \ R_{y3} \ R_{x4} \ R_{y4}]^T \\ &= [-0.10 \ -0.14 \ 0.04 \ -0.05 \ 0.10 \ 0.11 \ -0.04 \ 0.08]^T \end{aligned}$$

The element displacement field formulation is

$$\mathbf{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{Bmatrix} 0.1 + 0.1x + 0.2y \\ 0.1 - 0.1xy + 0.2y \end{Bmatrix}$$

The strain field formulation is

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} 0.1 \\ 0.2 - 0.1x \\ 0.2 - 0.1y \end{Bmatrix}$$

The stress field formulation is

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} 0.16 - 0.03x \\ 0.24 - 0.10x \\ 0.08 - 0.04y \end{Bmatrix}$$

The distributions of the displacement, the strain and the stress field formulations are plotted in Fig. 6.12.

The system strain energy is

$$U = \frac{1}{2} \mathbf{q}^T \mathbf{K} \mathbf{q} = 0.03$$

The external work is

$$W = \mathbf{P}^T \mathbf{q} = \mathbf{R}^T \mathbf{q} = 0.05$$

Then the system potential is

$$\Pi = U - W = -0.03$$

The system complementary energy is

$$\Pi_e = -\Pi = 0.03$$

The same plane structure under the second type of boundary conditions can also be solved using two 3-node triangular elements. Below the answers are directly given.

The displacement field in the first element is:

$$\mathbf{u}^{(1)} = \begin{Bmatrix} u^{(1)} \\ v^{(1)} \end{Bmatrix} = \begin{Bmatrix} 0.1 + 0.1x + 0.2y \\ 0.1 + 0.2y \end{Bmatrix}$$

The displacement field in the second element is:

$$\mathbf{u}^{(2)} = \begin{Bmatrix} u^{(2)} \\ v^{(2)} \end{Bmatrix} = \begin{Bmatrix} 0.1 + 0.1x + 0.2y \\ 0.2 - 0.1x + 0.1y \end{Bmatrix}$$

The strain field in the first element is:

$$\boldsymbol{\varepsilon}^{(1)} = [\varepsilon_x \ \varepsilon_y \ \gamma_{xy}]^T = [0.1 \ 0.2 \ 0.2]^T$$

The strain field in the second element is:

$$\boldsymbol{\varepsilon}^{(2)} = [\varepsilon_x \ \varepsilon_y \ \gamma_{xy}]^T = [0.1 \ 0.2 \ 0.1]^T$$

The stress field in the first element is:

$$\boldsymbol{\sigma}^{(1)} = [\sigma_x \ \sigma_y \ \tau_{xy}]^T = [0.16 \ 0.24 \ 0.08]^T$$

The stress field in the second element is:

$$\boldsymbol{\sigma}^{(2)} = [\sigma_x \ \sigma_y \ \tau_{xy}]^T = [0.13 \ 0.13 \ 0.04]^T$$

The distributions of the displacement, the strain and the stress field formulations in the two triangular elements are plotted in Fig. 6.13.

The system strain energy is

$$U = \frac{1}{2} \mathbf{q}^T \mathbf{K} \mathbf{q} = 0.03$$

The external work is

$$W = \mathbf{P}^T \mathbf{q} = 0.05$$

Then the system potential is

$$\Pi = U - W = -0.03$$

The system complemental energy is

$$\Pi_e = -\Pi = 0.03$$

# Chapter 7

## Finite Element Analysis Using 3D Elements



### 7.1 Introduction

In previous chapters, the finite element analysis procedures using the 1D elements (bar and beam) and the 2D elements (triangular and rectangular) are introduced. In this chapter, the FE analysis procedure using the 3D elements (tetrahedron and hexahedron) is introduced. In fact, the FE calculation procedure is the same, except for the formulations of the shape functions, the displacement field formulations, etc. Therefore, the readers should pay attention to the similarities and differences between the contents presented in this chapter and those in the previous chapters.

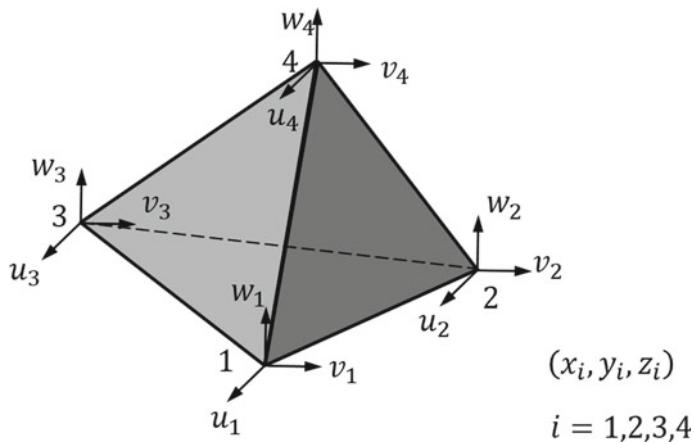
### 7.2 FE Analysis Using Tetrahedral Element

#### 7.2.1 *FE Analysis Procedure Using Tetrahedral Element*

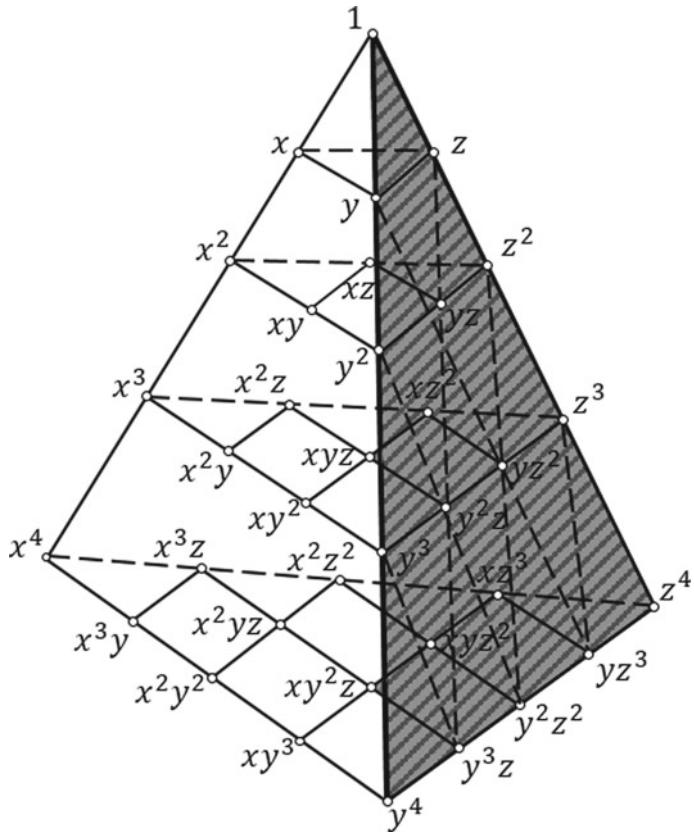
The simplest tetrahedral element, i.e. the 4-node tetrahedron (Fig. 7.1), is used to illustrate the FE analysis procedure using the tetrahedral element. In the 3D space, there are three degrees of freedom at each node in the tetrahedron and there are 4 nodes in the tetrahedron. Therefore, the nodal displacement vector can be written out as:

$$\mathbf{q}^e = [u_1 \ v_1 \ w_1 \ u_2 \ v_2 \ w_2 \ u_3 \ v_3 \ w_3 \ u_4 \ v_4 \ w_4]^T \quad (7.1)$$

where  $u_1$ ,  $v_1$  and  $w_1$  are the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively. It should be noted that, similar to the analysis using the 2D elements, it is assumed that the displacements in the  $x$ ,  $y$  and  $z$  directions are independent of each other. For example, the displacement in the  $x$  direction, i.e.,  $u$ , will not influence the displacements in the  $y$  and  $z$  directions.



**Fig. 7.1** The 4-node tetrahedral element



**Fig. 7.2** The Pascal's pyramid

Corresponding to the nodal displacement vector, the nodal force vector of the tetrahedron is:

$$\mathbf{P}^e = [P_{x1} \ P_{y1} \ P_{z1} \ P_{x2} \ P_{y2} \ P_{z2} \ P_{x3} \ P_{y3} \ P_{z3} \ P_{x4} \ P_{y4} \ P_{z4}]^T \quad (7.2)$$

Regarding the elemental displacement field formulation, because the displacements in the  $x$ ,  $y$  and  $z$  directions are independent, three independent functions are needed to describe the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively. When working out the elemental displacement field formulation, the nodal displacement values, i.e., Eq. (7.1), are assumed to be known. Therefore, 12 equations can be set up and four coefficients in each displacement field formulation can be solved. The elemental displacement field formulations can be written out as:

$$\begin{aligned} u(x, y, z) &= a_0 + a_1x + a_2y + a_3z \\ v(x, y, z) &= b_0 + b_1x + b_2y + b_3z \\ w(x, y, z) &= c_0 + c_1x + c_2y + c_3z \end{aligned} \quad (7.3)$$

where  $a_0, \dots, a_3, b_0, \dots, b_3$  and  $c_0, \dots, c_3$  are the coefficients to be determined. The terms 1,  $x$ ,  $y$  and  $z$  are used in the formulations and based on the Pascal's pyramid (Fig. 7.2), which will be introduced in the next chapter. In short, because the displacement field formulation for the tetrahedron takes the complete form, the terms until the first order, corresponding to the second plane in the Pascal's pyramid, should be included in the displacement field formulation.

Below the procedure for solving the coefficients is given.

The 12 boundary conditions are given as below:

$$\begin{aligned} u(x_i, y_i, z_i) &= u_i \\ v(x_i, y_i, z_i) &= v_i \\ w(x_i, y_i, z_i) &= w_i \end{aligned} \quad (7.4)$$

where  $(x_i, y_i, z_i)$  are the coordinates of the four nodes,  $i = 1, 2, 3, 4$ .

The elemental displacement field formulations can be reformulated in terms of the interpolation:

$$\begin{aligned} u(x, y, z) &= N_1(x, y, z)u_1 + N_2(x, y, z)u_2 + N_3(x, y, z)u_3 + N_4(x, y, z)u_4 \\ v(x, y, z) &= N_1(x, y, z)v_1 + N_2(x, y, z)v_2 + N_3(x, y, z)v_3 + N_4(x, y, z)v_4 \\ w(x, y, z) &= N_1(x, y, z)w_1 + N_2(x, y, z)w_2 + N_3(x, y, z)w_3 + N_4(x, y, z)w_4 \end{aligned} \quad (7.5)$$

where  $N_i$  is called the *interpolation function* or the shape function,  $u$ ,  $v$  and  $w$  are the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively.

Equation (7.5) can be rewritten in terms of the matrix format:

$$\mathbf{u}(x, y, z) = \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 \end{bmatrix} \mathbf{q}^e = \mathbf{N} \mathbf{q}^e \quad (7.6)$$

where  $\mathbf{q}^e$  is the nodal displacement vector,  $N_i = \frac{1}{6V}(a_i + b_i x + c_i y + d_i z)(i = 1, 2, 3, 4)$ ,  $V$  is the volume of the tetrahedron. The variables  $a_i$ ,  $b_i$ ,  $c_i$  and  $d_i$  can be obtained by solving the boundary condition equations, i.e., Eq. (7.4), which give the following answers:

$$\begin{aligned} a_1 &= \det \begin{bmatrix} x_2 & y_2 & z_2 \\ x_3 & y_3 & z_3 \\ x_4 & y_4 & z_4 \end{bmatrix} \\ b_1 &= \det \begin{bmatrix} y_2 & z_2 & 1 \\ y_3 & z_3 & 1 \\ y_4 & z_4 & 1 \end{bmatrix} \\ c_1 &= \det \begin{bmatrix} z_2 & 1 & x_2 \\ z_3 & 1 & x_3 \\ z_4 & 1 & x_4 \end{bmatrix} \\ d_1 &= \det \begin{bmatrix} 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix} \end{aligned} \quad (7.7)$$

Up to now, the elemental displacement field formulations have been solved. Now, let's move to the calculation of the strain and stress field formulations.

The strain formulation for the tetrahedral element can be worked out using the strain-displacement relation in the 3D form, i.e.,

$$\boldsymbol{\varepsilon}(x, y, z) = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = [\partial] \mathbf{u} = [\partial] \mathbf{N} \mathbf{q}^e = \mathbf{B} \mathbf{q}^e \quad (7.8)$$

where  $[\partial]$  is the operator matrix of the strain-displacement equation,  $\mathbf{B}$  is the strain-displacement vector and can be written as below:

$$\mathbf{B} = [\partial] \mathbf{N} = [\mathbf{B}_1 \ \mathbf{B}_2 \ \mathbf{B}_3 \ \mathbf{B}_4] \quad (7.9)$$

Putting the formulation for the shape function into Eq. (7.9), the components of the strain-displacement vector can be worked out as:

$$\mathbf{B}_i = [\partial] \begin{bmatrix} N_i & 0 & 0 \\ 0 & N_i & 0 \\ 0 & 0 & N_i \end{bmatrix} = \frac{1}{6V} \begin{bmatrix} b_i & 0 & 0 \\ 0 & c_i & 0 \\ 0 & 0 & d_i \\ c_i & b_i & 0 \\ 0 & d_i & c_i \\ d_i & 0 & b_i \end{bmatrix} \quad (i = 1, 2, 3, 4) \quad (7.10)$$

The stress formulation for the tetrahedral element can be obtained using the 3D stress-strain relation:

$$\boldsymbol{\sigma}(x, y, z) = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \mathbf{D}\boldsymbol{\epsilon} \quad (7.11)$$

where  $\mathbf{D}$  is the elasticity matrix and can be written out as:

$$\mathbf{D} = \frac{E}{(1-2\nu)(1+\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2}(1-2\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{2}(1-2\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{2}(1-2\nu) \end{bmatrix} \quad (7.12)$$

where  $E$  is the Young's modulus and  $\nu$  is the Poisson's ratio of the material.

Putting the strain formulation into Eq. (7.11), the stress can be solved

$$\boldsymbol{\sigma} = \mathbf{D}\mathbf{B}\mathbf{q}^e = \mathbf{S}\mathbf{q}^e \quad (7.13)$$

where  $\mathbf{S}$  is the stress-displacement matrix.

The elemental potential energy for the tetrahedral element can be written out as:

$$\begin{aligned} \prod^e &= \frac{1}{2} \int_{\Omega^e} \boldsymbol{\sigma}^T \boldsymbol{\epsilon} d\Omega - \left[ \int_{\Omega^e} \bar{\mathbf{b}}^T \mathbf{u} d\Omega + \int_{S_p^e} \bar{\mathbf{p}}^T \mathbf{u} dA \right] \\ &= \frac{1}{2} \mathbf{q}^{eT} \left( \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \right) \mathbf{q}^e - \left( \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA \right)^T \mathbf{q}^e \end{aligned}$$

$$= \frac{1}{2} \mathbf{q}^e^T \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^e^T \mathbf{q}^e \quad (7.14)$$

where  $\mathbf{K}^e$  is the stiffness matrix of the element, i.e.,

$$\mathbf{K}^e = \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \quad (7.15)$$

The elemental stiffness equation can be obtained by taking the derivative of the potential energy with respect to the nodal displacement vector, i.e.

$$\partial \prod / \partial \mathbf{q}^e = 0 \quad (7.16)$$

From the equations above, the stiffness equation can be written as below:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \quad (7.17)$$

where  $\mathbf{K}^e$  is the stiffness matrix given in Eq. (7.15),  $\mathbf{q}^e$  is the nodal displacement vector,  $\mathbf{P}^e$  is the elemental equivalent nodal force, i.e.

$$\mathbf{P}^e = \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA = \int_{A^e} \mathbf{N}^T \bar{\mathbf{b}}_t dA + \int_{t_p^e} \mathbf{N}^T \bar{\mathbf{p}}_t dl \quad (7.18)$$

where  $\bar{\mathbf{b}}$  is the body force vector in the solution domain and  $\bar{\mathbf{p}}$  is the surface force vector in the boundary region.

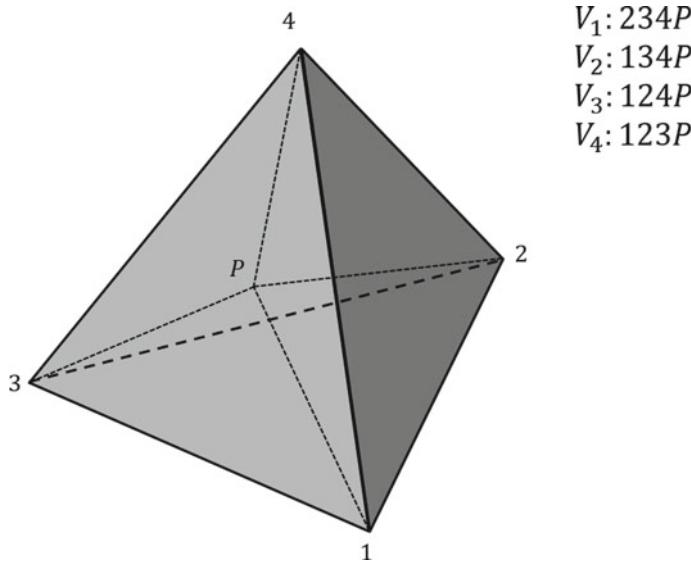
### 7.2.2 The Volume Coordinates and Their Properties

Similar to the area coordinates defined in the triangular element, the volume coordinates can be defined in the tetrahedron and used for the calculation of the shape function. Figure 7.3 is used to illustrate the definition of the volume coordinate.

The volume coordinates for the 4-node tetrahedron (Fig. 7.3) are defined as below:

$$L_i = V_i / V (i = 1, 2, 3, 4) \quad (7.19)$$

where  $V$  is the volume of the tetrahedron,  $V_i$  is the volume of the sub-tetrahedron within the tetrahedron. There are four faces in the tetrahedron and thus four sub-tetrahedrons can be formed, which are denoted as  $V_1, V_2, V_3$  and  $V_4$ . The corresponding coordinates are denoted as  $L_1, L_2, L_3$  and  $L_4$ . It can be seen that the volume coordinates are independent of the Cartesian coordinates. Their relationship with the shape functions is as follows:



**Fig. 7.3** Illustration of the definition of the volume coordinate

$$L_i = N_i \quad (i = 1, 2, 3, 4) \quad (7.20)$$

The two basic properties of the shape functions for the tetrahedral element can be easily proved using Eq. (7.20). First, the sum of the four shape functions should equal to 1, which can be proved as below:

$$N_1 + N_2 + N_3 + N_4 = L_1 + L_2 + L_3 + L_4 = (V_1 + V_2 + V_3 + V_4)/V = 1 \quad (7.21)$$

Second, from the definition of the volume coordinates, the values of the shape functions (volume coordinates) at the four corner nodes can be easily obtained. To work out the volume coordinates at node  $i$ , the point  $P$  is moved to node  $i$ , then  $V_i = 1$ ,  $V_j = 0$ ,  $V_m = 0$ ,  $V_k = 0$  are obtained. Therefore, the volume coordinate at node  $i$  is  $(1\ 0\ 0\ 0)$ . Similarly, the volume coordinates at nodes  $j$ ,  $m$  and  $k$  are  $(0\ 1\ 0\ 0)$ ,  $(0\ 0\ 1\ 0)$  and  $(0\ 0\ 0\ 1)$ , respectively.

Furthermore, the behavior of the volume coordinates in the surface of the tetrahedron is investigated. To work out the behavior of the volume coordinate in the plane  $jm$ , the point  $P$  is moved to the plane  $jm$ , then  $V_i = 0$  is obtained, i.e., the volume coordinate of  $L_i$  is zero. Similarly,  $V_j = 0$  in the plane  $im$ ,  $V_m = 0$  in the plane  $ik$  and  $V_k = 0$  in the plane  $ijm$  are obtained.

At the end of this part, the behavior of the volume coordinates in the planes parallel to the surface of the volume is investigated. The surface  $ijm$  is used as the example for demonstration. To work out the volume coordinates in the planes parallel to the surface  $ijm$ , the point  $P$  is moved in the parallel plane and it is obviously seen that

the volume  $V_i$  maintains the same when moving the point  $P$ , i.e., all the  $L_i$  equal in the planes parallel to the surface  $ijm$ . Similarly, it can be seen that, all the  $L_i$  equal in the planes parallel to the surface  $ilm$ , etc.

### 7.3 FE Analysis Procedure Using Hexahedral Element

In this part, the FE analysis procedure using the simplest hexahedral element, the 8-node first order hexahedral element (Fig. 7.4) is introduced. In the 3D element, there are three degrees of freedom at each node, i.e.,  $u$ ,  $v$  and  $w$ . Therefore, the nodal displacement vector can be written out as:

$$\mathbf{q}^e = [u_1 \ v_1 \ w_1 \ u_2 \ v_2 \ w_2 \ \dots \ u_8 \ v_8 \ w_8]^T \quad (7.22)$$

where  $u_1$ ,  $v_1$  and  $w_1$  are the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively. In corresponding to the nodal displacement vector, the nodal force vector of the tetrahedron is:

$$\mathbf{P}^e = [P_{x1} \ P_{y1} \ P_{z1} \ P_{x2} \ P_{y2} \ P_{z2} \ \dots \ P_{x8} \ P_{y8} \ P_{z8}]^T \quad (7.23)$$

Again, it is assumed that the displacements in the  $x$ ,  $y$  and  $z$  directions are independent of each other. Therefore, three independent functions are required to describe the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively. When calculating the elemental displacement field formulation, the nodal displacement values, i.e., Eq. (7.22), are assumed to be known. Therefore, 24 equations can be set up and 8 coefficients in each displacement field formulation can be solved. The elemental

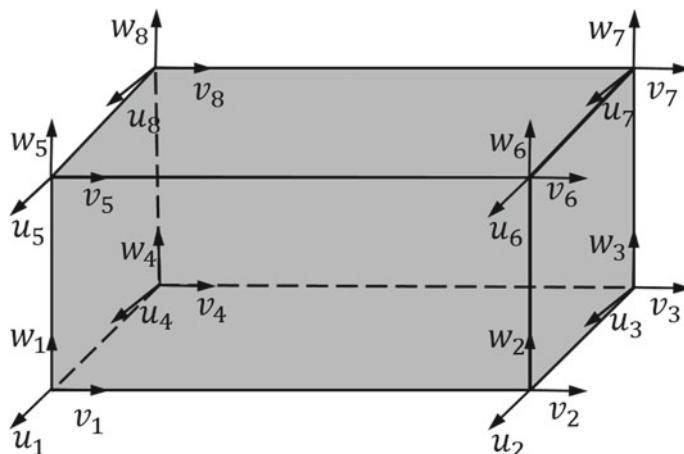


Fig. 7.4 The 8-node hexahedral element

displacement field formulations can be written out as:

$$\begin{aligned} u(x, y, z) &= a_0 + a_1x + a_2y + a_3z + a_4xy + a_5yz + a_6zx + a_7xyz \\ v(x, y, z) &= b_0 + b_1x + b_2y + b_3z + b_4xy + b_5yz + b_6zx + b_7xyz \\ w(x, y, z) &= c_0 + c_1x + c_2y + c_3z + c_4xy + c_5yz + c_6zx + c_7xyz \end{aligned} \quad (7.24)$$

where  $a_0, \dots, a_7, b_0, \dots, b_7$  and  $c_0, \dots, c_7$  are the coefficients to be determined. The terms  $1, x, y, z, xy, yz, zx$  and  $xyz$  are used in the formulations and based on the Pascal's pyramid, which will be introduced in the next chapter. In short, because the displacement field formulation for the hexahedral element takes the incomplete form, the terms until the first order, corresponding to the second plane in the Pascal's pyramid, should be included in the displacement field formulation. Additionally, a diamond shape which is symmetric about the second plane should be formed and all the terms covered by the diamond should be included in the displacement field formulation. Therefore, the terms  $1, x, y, z, xy, yz, zx$  and  $xyz$  are present in Eq. (7.24)

Below the procedure for solving the coefficients to be determined is shown.

The 24 boundary conditions are as below:

$$\begin{aligned} u(x_i, y_i, z_i) &= u_i \\ v(x_i, y_i, z_i) &= v_i \\ w(x_i, y_i, z_i) &= w_i \end{aligned} \quad (7.25)$$

where  $(x_i, y_i, z_i)$  are the coordinates of the 8 nodes,  $i = 1, 2, 3, 4, \dots, 8$ . The elemental displacement field formulations can be reformulated in terms of the interpolation:

$$\begin{aligned} u(x, y, z) &= N_1(x, y, z)u_1 + N_2(x, y, z)u_2 + N_3(x, y, z)u_3 + \dots + N_8(x, y, z)u_8 \\ v(x, y, z) &= N_1(x, y, z)v_1 + N_2(x, y, z)v_2 + N_3(x, y, z)v_3 + \dots + N_8(x, y, z)v_8 \\ w(x, y, z) &= N_1(x, y, z)w_1 + N_2(x, y, z)w_2 + N_3(x, y, z)w_3 + \dots + N_8(x, y, z)w_8 \end{aligned} \quad (7.26)$$

where  $N_i$  are called the interpolation functions or the shape functions,  $u_i, v_i$  and  $w_i$  are the displacements in the  $x, y$  and  $z$  directions, respectively.

Equation (7.26) can be rewritten in terms of the matrix format:

$$\mathbf{u}(x, y, z) = \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \dots & N_8 & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \dots & 0 & N_8 & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \dots & 0 & 0 & N_8 \end{bmatrix} \mathbf{q}^e = \mathbf{Nq}^e \quad (7.27)$$

The strain formulation for the hexahedral element can be worked out using the strain-displacement relation in the 3D form, i.e.,

$$\boldsymbol{\epsilon}(x, y, z) = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = [\partial]\mathbf{u} = [\partial]\mathbf{N}\boldsymbol{\epsilon}^e = \mathbf{B}\boldsymbol{\epsilon}^e \quad (7.28)$$

where  $[\partial]$  is the operator matrix of the strain–displacement equation,  $\mathbf{B}$  is the strain–displacement vector and can be written as below:

$$\mathbf{B} = [\partial]\mathbf{N} = [\mathbf{B}_1 \ \mathbf{B}_2 \ \mathbf{B}_3 \ \cdots \ \mathbf{B}_8] \quad (7.29)$$

Putting the formulation for the shape function into Eq. (7.29), the components of the strain–displacement vector can be worked out as:

$$\mathbf{B}_i = [\partial] \begin{bmatrix} N_i & 0 & 0 \\ 0 & N_i & 0 \\ 0 & 0 & N_i \end{bmatrix} = \frac{1}{6V} \begin{bmatrix} b_i & 0 & 0 \\ 0 & c_i & 0 \\ 0 & 0 & d_i \\ c_i & b_i & 0 \\ 0 & d_i & c_i \\ d_i & 0 & b_i \end{bmatrix} \quad (i = 1, 2, 3, 4) \quad (7.30)$$

The stress formulation for the tetrahedral element can be obtained using the 3D stress–strain relation:

$$\boldsymbol{\sigma}(x, y, z) = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \mathbf{D}\boldsymbol{\epsilon} \quad (7.31)$$

where  $\mathbf{D}$  is the elasticity matrix given in Eq. (7.12):

Putting the strain formulation into Eq. (7.31), the stress within the element can be solved

$$\boldsymbol{\sigma} = \mathbf{D}\mathbf{B}\boldsymbol{\epsilon}^e = \mathbf{S}\mathbf{q}^e \quad (7.32)$$

where  $\mathbf{S}$  is the stress-displacement matrix.

The elemental potential energy for the tetrahedral element can be written out as:

$$\begin{aligned}
\prod^e &= \frac{1}{2} \int_{\Omega^e} \boldsymbol{\sigma}^T \boldsymbol{\epsilon} d\Omega - \left[ \int_{\Omega^e} \bar{\mathbf{b}}^T \mathbf{u} d\Omega + \int_{S_p^e} \bar{\mathbf{p}}^T \mathbf{u} dA \right] \\
&= \frac{1}{2} \mathbf{q}^{e^T} \left( \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \right) \mathbf{q}^e - \left( \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA \right)^T \mathbf{q}^e \\
&= \frac{1}{2} \mathbf{q}^{e^T} \mathbf{K}^e \mathbf{q}^e - \mathbf{P}^{e^T} \mathbf{q}^e
\end{aligned} \tag{7.33}$$

where  $\mathbf{K}^e$  is the stiffness matrix of the element, i.e.,

$$\mathbf{K}^e = \int_{\Omega^e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \tag{7.34}$$

The elemental stiffness equation can be obtained by taking the derivative of the potential energy with respect to the nodal displacement vector, i.e.

$$\partial \prod^e / \partial \mathbf{q}^e = 0 \tag{7.35}$$

From the equations above, the stiffness equation can be written as below:

$$\mathbf{K}^e \mathbf{q}^e = \mathbf{P}^e \tag{7.36}$$

where  $\mathbf{K}^e$  is the stiffness matrix given in Eq. (7.35),  $\mathbf{q}^e$  is the nodal displacement vector and  $\mathbf{P}^e$  is the elemental equivalent nodal force, i.e.

$$\mathbf{P}^e = \int_{\Omega^e} \mathbf{N}^T \bar{\mathbf{b}} d\Omega + \int_{S_p^e} \mathbf{N}^T \bar{\mathbf{p}} dA = \int_{A^e} \mathbf{N}^T \bar{\mathbf{b}} t dA + \int_{I_p^e} \mathbf{N}^T \bar{\mathbf{p}} t dl \tag{7.37}$$

where  $\bar{\mathbf{b}}$  is the body force in the solution domain and  $\bar{\mathbf{p}}$  is the surface force in the boundary region.

# Chapter 8

## High Order Lagrange Element



### 8.1 Introduction

The elements previously discussed, i.e., the bar, the triangular and the rectangular elements, are the first order (linear) elements. The accuracy of these elements is limited. For example, the strain is constantly distributed within the linear triangular element, which limits its application in the complex nonlinear engineering problems. To solve the issue, in this chapter, the high order elements, which possess the higher accuracy, are introduced.

### 8.2 Definition of Lagrange and Hermite Elements

When the order of the element is increased, the displacement field formulations, the shape functions, etc. are all changed. Previously, the nodal displacements are used as the boundary conditions to work out these formulations. However, this method is time consuming and a large amount of calculations are involved. To avoid these complex calculations, new and yet efficient approaches are needed to obtain the formulations for the high order elements without solving the equations established using the nodal displacements, which is feasible for the Lagrange type of element. To fully illustrate the calculation process, we first give the definition of the Lagrange element, which is also called the  $C_0$  element. This type of element uses the Lagrange polynomial function as the interpolation function. Furthermore, it only uses the nodal values for the interpolation and only requires the continuity of the displacement at the shared nodes and on the shared edges. Most of the elements discussed in the previous chapters, e.g., the bar element, the triangular and rectangular elements, belong to the type of the Lagrange element. The advantage of the Lagrange element is that the shape functions can be directly worked out using the interpolation function, the details on which will be given in this chapter. However, some shortcomings of the Lagrange element should also be noted. First, when the element order is increased, the number

of the inner nodes is increased accordingly, i.e., the number of the non-completeness polynomial terms is increased, such as the high order rectangular element. Therefore, too many inner nodes will be present if the order of the element is increased. The addition of the non-completeness terms will not increase the element accuracy but the computation complexity.

In addition to the Lagrange element, the Hermite element is another type of finite elements. The *Hermite element* is also called the *C<sub>1</sub> element* and uses the Hermite polynomial function as the interpolation function. In the Hermite element, not only the nodal field formulations but also the first order derivatives of the nodal field formulations are used for the interpolation. Therefore, not only the displacement is continuous but also the first order derivative of the displacement is continuous on the boundaries shared by elements. One example of the Hermite element is the beam element discussed in Chap. 4. In this chapter, only the high order Lagrange elements are introduced. The readers can refer to other FE books for the high order Hermite elements.

From the previous chapters, it is known that the key to solve the finite elements is to work out their field displacement formulations and the shape functions, because based on the shape functions, the strain–displacement matrix and the stiffness matrix can be worked out and consequently the stiffness equation can be set up. Therefore, in this chapter, our focuses are on the elemental displacement formulations and the shape functions for the high order Lagrange elements, i.e. the high order bar element (1D), the triangular and rectangular elements (2D), the tetrahedral and hexahedral elements (3D).

### 8.3 The 1D High Order Lagrange Element

The bar element previously introduced is the 1D Lagrange element. Previously in Chap. 3, it is known that the first order bar element is a linear element, which implies that the displacement within the element is a linear function of the coordinates. Furthermore, the strain within the bar element is constantly distributed, because the strain is calculated from the first order derivative of the displacement field formulation. Below, the displacement field formulations and the shape functions for the high order bar elements are introduced.

First, let's review the displacement field function for the bar element:

$$\mathbf{u}(x) = \mathbf{N}(x)\mathbf{q}^e = \sum_{i=1}^n N_i u_i \quad (8.1)$$

where  $N_i$  are the shape functions and can be expressed using the Lagrange polynomial functions:

$$N_i = l_i^{(n-1)}(x) = \prod_{(j=1, j \neq i)}^n \frac{x - x_j}{x_i - x_j} \quad (8.2)$$

where  $(n - 1)$  is the order of the polynomial function,  $\prod$  represents the multiplication operation on polynomial functions.

It should be noted that Eqs. (8.1) and (8.2) are formulated in the Cartesian coordinate system and the shape function is independent of the length of the bar element. Therefore, the dimensionless (natural) coordinate can be used to re-formulate the shape functions to simplify the relevant calculations.

To convert the coordinates from the Cartesian coordinate system to the natural coordinate system, the following equation is used:

$$\xi = \frac{x - x_1}{x_n - x_1} = \frac{x - x_1}{l} \quad (8.3)$$

where  $l$  is the length of the bar element,  $\xi$  is the dimensionless coordinate defined in the natural coordinate system and  $x_1$  is the coordinate of the first node in the Cartesian coordinate system.

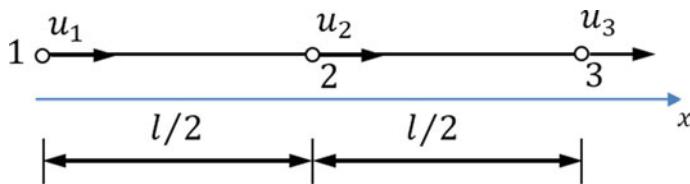
Using the dimensionless coordinates, the shape function can be re-formulated as below:

$$N_i = l_i^{(n-1)}(\xi) = \prod_{(j=1, j \neq i)}^n \frac{\xi - \xi_j}{\xi_i - \xi_j} \quad (8.4)$$

where  $\xi_i$  and  $\xi_j$  are the dimensionless coordinates at nodes  $i$  and  $j$ , respectively,  $n$  is the number of elemental nodes. It can be seen from Eq. (8.4) that there are  $(n - 1)$  terms doing the multiplication operations. Therefore, the corresponding displacement field function is a  $(n - 1)$  th order polynomial function and the order of the corresponding element is  $(n - 1)$ .

Using Eqs. (8.3) and (8.4), the displacement field function and the shape function for the Lagrange bar element with any high order can be worked out. Below a 3-node bar element is used to demonstrate the calculation process.

There are three nodes in the element, and thus we have  $n = 3$  and the order of the bar element is 2. From this analysis, it is known that the displacement field and the shape formulations for this element are the second order polynomial functions,



**Fig. 8.1** The 3-node second order bar element

which can help justify the calculation of these functions. It should be noted that in the Lagrange bar element, the order of the element increases with the increase of the number of the elemental node. The higher the order of the element, the more accurate the solution.

To simplify the calculations, the natural coordinate system is used. Based on Eq. (8.3) and Fig. (8.1), the dimensionless coordinates of the three nodes are:

$$\xi_1 = 0, \xi_2 = 1/2, \xi_3 = 1. \quad (8.5)$$

Then using Eq. (8.4), the shape function at the first node can be calculated as below:

$$N_1 = l_1^{(2)} = \frac{(\xi - \xi_2)(\xi - \xi_3)}{(\xi_1 - \xi_2)(\xi_1 - \xi_3)} = \frac{(\xi - 1/2)(\xi - 1)}{(0 - 1/2)(0 - 1)} = 2\left(\xi - \frac{1}{2}\right)(\xi - 1) \quad (8.6a)$$

Similarly, we can directly work out the shape functions at the second and third nodes, i.e.,  $N_2$  and  $N_3$ .

$$N_2 = l_2^{(2)} = \frac{(\xi - \xi_1)(\xi - \xi_3)}{(\xi_2 - \xi_1)(\xi_2 - \xi_3)} = \frac{(\xi - 0)(\xi - 1)}{(1/2 - 0)(1/2 - 1)} = 4\xi(1 - \xi) \quad (8.6b)$$

$$N_3 = l_3^{(2)} = \frac{(\xi - \xi_1)(\xi - \xi_2)}{(\xi_3 - \xi_1)(\xi_3 - \xi_2)} = \frac{(\xi - 0)(\xi - 1/2)}{(1 - 0)(1 - 1/2)} = 2\xi\left(\xi - \frac{1}{2}\right) \quad (8.6c)$$

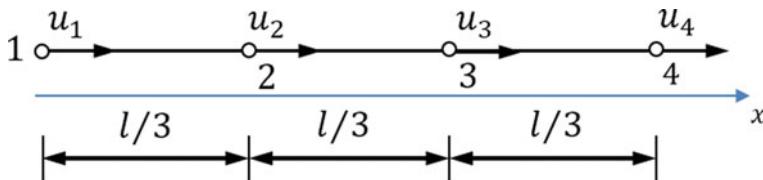
The nodal displacement vector of the 3-node bar element is:

$$\mathbf{q}^e = [u_1 \ u_2 \ u_3]^T \quad (8.7)$$

Then based on Eq. (8.1), the displacement field function for the 3-node bar element can also be quickly worked out. From the calculations above, again it can be seen that the calculation of the shape functions for the element is the first and key step for solving the element.

Below, a 4-node third order bar element (Fig. 8.2) is used to further demonstrate the calculation of the high order Lagrange bar element.

The displacement field function within the 4-node bar element can be written out as:



**Fig. 8.2** The 4-node third order bar element

$$\mathbf{u}(x) = a_1 + a_2 + a_3 x^2 + a_4 x^3 = \mathbf{N} \mathbf{q}^e \quad (8.8)$$

where  $\mathbf{q}^e = [u_1 \ u_2 \ u_3 \ u_4]^T$  is the nodal displacement vector of the 4-node bar element,  $\mathbf{N} = [N_1 \ N_2 \ N_3 \ N_4]$  is the shape function vector and can be calculated using Eq. (8.4), which is given as below:

$$\begin{aligned} N_1 &= \left(1 - \frac{3x}{l}\right) \left(1 - \frac{3x}{2l}\right) \left(1 - \frac{x}{l}\right) \\ N_2 &= 9 \frac{x}{l} \left(1 - \frac{3x}{2l}\right) \left(1 - \frac{x}{l}\right) \\ N_3 &= -\frac{9}{2} \frac{x}{l} \left(1 - \frac{3x}{l}\right) \left(1 - \frac{x}{l}\right) \\ N_4 &= \frac{x}{l} \left(1 - \frac{3x}{l}\right) \left(1 - \frac{3x}{2l}\right) \end{aligned} \quad (8.9)$$

From the calculations above, it can be seen that using the Lagrange interpolation function and the natural coordinate system, the shape functions for the high order bar element can be quickly worked out without solving the boundary condition equations. Summarizing the calculations of the 3-node and 4-node bar elements, the general form of the shape functions for the Lagrange bar element in the natural coordinate system can be written as below:

$$N_i = l_i^{(n-1)}(\xi) = \prod_{(j=1, j \neq i)}^n \frac{f_j(\xi)}{f_j(\xi_i)} \quad (8.10)$$

where  $\xi_j$  is the dimensionless coordinate of the elemental node  $j$ ,  $f_j(\xi) = \xi - \xi_j$  represents the distance between  $\xi$  and  $\xi_j$  and  $f_j(\xi_i)$  is the value of  $f_j(\xi)$  at node  $i$ .

It is clearly seen from Eq. (8.10) that the first property of the shape function is satisfied, i.e., the value of shape function  $N_i$  at node  $i$  is one and its values at other nodes are zeros. This property can be proved as following.

First, let's have a look at the value of shape function  $N_i$  at node  $i$ . We put  $\xi = \xi_i$  into Eq. (8.10), and then the numerator becomes  $f_j(\xi)(\xi = \xi_i) = \xi_i - \xi_j$  which equals to the denominator. Therefore, we have  $N_i(\xi_i) = 1$  at node  $i$ . Second, let's have a look at the value of the shape function  $N_i$  at the elemental nodes rather than its own node (i.e., node  $i$ ), because the value of polynomial in the numerator equals to zero, i.e.,  $f_j(\xi)(\xi = \xi_j) = \xi_j - \xi_j = 0$ , and thus we have  $N_i(\xi_j) = 0$  at other nodes.

## 8.4 The 2D High Order Lagrange Element

### 8.4.1 The High Order Triangular Element

The 3-node triangular element introduced in Chap. 5 is the first order linear element, which implies that the displacement within the 3-node triangular is a linear function of the coordinates and the strain within the element is constantly distributed. Because of these behaviors, the application of the linear triangular element in the complex engineering problems, which always involve nonlinear deformations, is limited. Therefore, the high order triangular elements are necessary for the engineering applications. Below, the efficient calculations of the displacement field formulations and the shape functions for the triangular element with any high order are introduced. The 6-node second order triangular element (Fig. 8.3) is used for the demonstration.

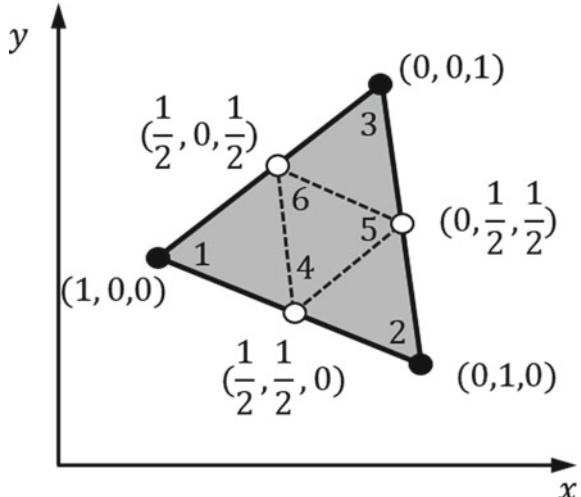
There are 6 nodes in the second order triangular element and each node has two degrees of freedom, i.e., the displacements in the  $x$  and  $y$  directions, respectively. Therefore, the nodal displacement vector can be written out as:

$$\mathbf{q}^e = [u_1 \ v_1 \ \dots \ u_6 \ v_6]^T \quad (8.11)$$

Regarding the elemental displacement field function, because the nodal displacements in Eq. (8.11) are assumed to be known values, 6 undetermined coefficients in each elemental displacement field formulation can be solved and thus the displacement field functions for the second order triangular element can be written out as:

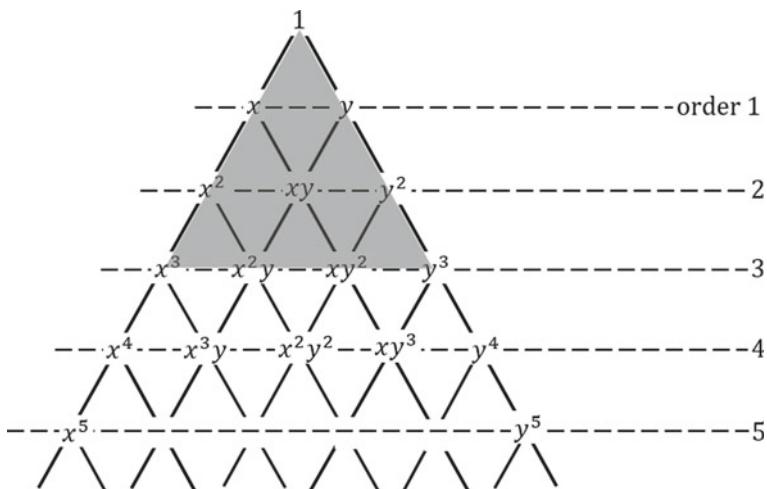
$$u(x, y) = a_1 + a_2x + a_3y + a_4x^2 + a_5xy + a_6y^2 \quad (8.12a)$$

**Fig. 8.3** The 6-node second order triangular element



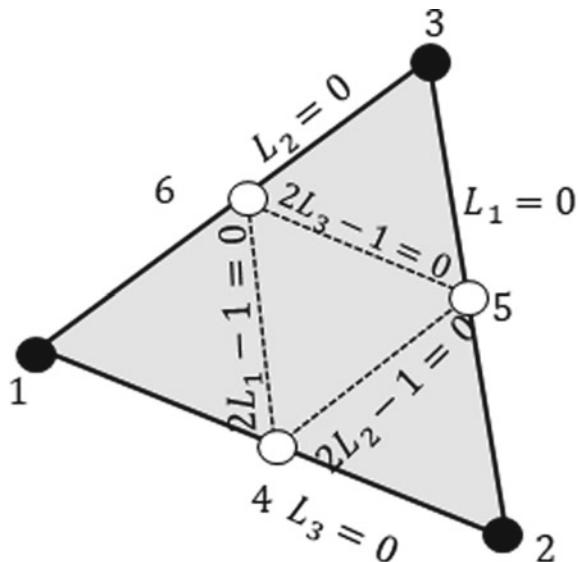
$$v(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 \quad (8.12b)$$

where  $a_1, \dots, a_6$  and  $b_1, \dots, b_6$  are the coefficients to be determined. Now we have a couple of relevant questions. First, regarding the variables appeared in displacement field functions, e.g.,  $x, y, xy$ , etc. in Eq. (8.12a, b), what form should be used? Second, how many nodes should be used in the high order triangular element, e.g., the third order triangular element? Both questions can be answered using the Pascal's triangle (Fig. 8.4). In the Pascal's triangle, the first, the second and the third rows correspond to the constant, the first order and the second order terms in the displacement field functions. Additionally, the displacement field function for the triangular element is the complete form, i.e., the displacement function for the  $n$ th order triangular element should include the terms in the Pascal's triangle until the  $(n + 1)$ th row (including this row). For example, the displacement field formulation for the second order triangular element should contain all the elements in the first, the second and the third lines in the Pascal's triangle. Based on this rule, in the 6-node second order triangular element, the displacement field formulation should include 6 terms, i.e.,  $1, x, y, x^2, xy$  and  $y^2$  and accordingly the displacement field formulation for the triangular element with any order can be worked out. Now let's come to the second question, which can also be answered using the Pascal's triangle, i.e., the number of node for a  $n$ th order triangular element is the total number of terms in the Pascal's triangle until the  $(n + 1)$ th row. For example, the total number of nodes for the second order triangular element should be 6, because the total number of terms until the third row, i.e.,  $1, x, y, x^2, xy$  and  $y^2$ , is 6.



**Fig. 8.4** The Pascal's triangle

**Fig. 8.5** The area coordinates defined in the 6-node second order triangular element



It should be noted that the Pascal's triangle can be applied not only to the Lagrange triangular element but also to the Lagrange rectangular element, which will be introduced in the next part.

Using the Pascal's triangle, the displacement field function can be easily worked out. However, we have another issue to be solved, i.e., the calculation of the shape function for the high order triangular elements. Similar to the bar element, the dimensionless coordinate system is used to work out the shape function for the high order triangular element. The dimensionless coordinate system for the triangular element is the area coordinate system, which has been introduced in Chap. 5. Below the 6-node second order triangular element is used to demonstrate the calculations of the relevant shape functions.

The area coordinates for the 6-node second order triangular element (Fig. 8.5) can be defined as below:

$$L_i = A_i / A \quad (8.13)$$

where  $i$  is the number index of the corner node,  $A$  is the area of the triangular and  $A_i$  is the area of the sub-triangular. The triangular has three corner nodes and thus  $i$  is 1, 2 and 3 for all the triangular elements. Consequently, three area coordinates, i.e.,  $L_1$ ,  $L_2$  and  $L_3$ , are associated with all the triangular elements no matter which orders they are. Using the area coordinates, the shape functions for the high order triangular elements can be written out as:

$$N_i = \prod_{j=1}^n \frac{f_j^{(i)}(L_1, L_2, L_3)}{f_j^{(i)}(L_{1i}, L_{2i}, L_{3i})} \quad (8.14)$$

where  $n$  is the order of the triangular element,  $f_j^{(i)}(L_1, L_2, L_3)$  are the left items of the line equations going through all the nodes except for node  $i$ ,  $L_{1i}$ ,  $L_{2i}$  and  $L_{3i}$  are the area coordinates at node  $i$ .

It should be noted that the number of items in the numerator in Eq. (8.14) should be in agreement with the order of the triangular element. For example, in a second order triangular element, in the numerator of the shape function, there should be two terms doing the multiplication. Otherwise, the second order polynomial function cannot be obtained in the displacement field function. The two terms in the numerator can be determined by finding two lines going through all the nodes of the 6-node triangular except for its own node. Now Eq. (8.14) is used to demonstrate the calculation of the shape function at the first node in the second order triangular element. In this case, in Eq. (8.14), the followings are set:  $i = 1$ ,  $n = 2$ . Therefore, two lines going through all the nodes except for the first node should be determined and it is obvious from Fig. 8.5 that the two line equations are:

$$f_1^{(1)}(L_1, L_2, L_3) = L_1 - 1/2 = 0 \quad (8.15)$$

$$f_2^{(1)}(L_1, L_2, L_3) = L_1 = 0 \quad (8.16)$$

Additionally, the area coordinates at the first node can be worked out as below:

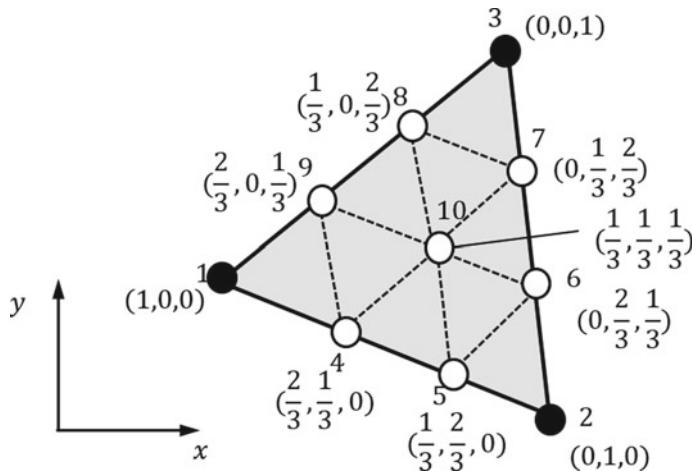
$$L_1(i) = 1, \ L_2(i) = 0, \ L_3(i) = 0 \ (i = 1) \quad (8.17)$$

Therefore, the shape function at the first node can be worked out as:

$$N_1 = \frac{f_1^{(1)}(L_1, L_2, L_3)}{f_1^{(1)}(1, 0, 0)} \frac{f_2^{(1)}(L_1, L_2, L_3)}{f_2^{(1)}(1, 0, 0)} = \frac{L_1 - 1/2}{1 - 1/2} \frac{L_1}{1} = (2L_1 - 1)L_1 \quad (8.18a)$$

Similarly, the shape functions at other nodes can be worked out as:

$$\begin{aligned} N_2 &= \frac{L_2 - \frac{1}{2}}{\frac{1}{2}} \frac{L_2}{1} = (2L_2 - 1)L_2 \\ N_3 &= \frac{L_3 - 1/2}{1/2} \frac{L_3}{1} = (2L_3 - 1)L_3 \\ N_4 &= \frac{L_1}{\frac{1}{2}} \frac{L_2}{1/2} = 4L_1L_2 \\ N_5 &= \frac{L_2}{1/2} \frac{L_3}{1/2} = 4L_2L_3 \\ N_6 &= \frac{L_3}{1/2} \frac{L_1}{1/2} = 4L_3L_1 \end{aligned} \quad (8.18)$$



**Fig. 8.6** The 10-node third order triangular element

Below, a third order triangular element (Fig. 8.6) is used to further demonstrate the calculation of the displacement field formulations and the shape functions. First, based on the Pascal's triangle, the total number of nodes in the third order triangular element is 10. Then in the 10-node third order triangular element shown in Fig. 8.6, there are 2 degrees of freedom at each node and thus the nodal displacement vector can be written out as:

$$\mathbf{q}^e = [u_1 \ v_1 \ u_2 \ v_2 \ \dots \ u_{10} \ v_{10}]^T \quad (8.19)$$

where  $u_1, \dots, u_{10}$  and  $v_1, \dots, v_{10}$  are the displacements in the  $x$  and  $y$  directions, respectively. Regarding the displacement field formulation in the third order triangular element, based on the Pascal's triangle, each displacement field formulation should include the complete terms until the fourth row. Therefore, the displacement field functions can be written as:

$$u(x, y) = a_1 + a_2x + a_3y + a_4x^2 + a_5xy + a_6y^2 + a_7x^3 + a_8x^2y + a_9xy^2 + a_{10}y^3 \quad (8.20a)$$

$$v(x, y) = b_1 + b_2x + b_3y + b_4x^2 + b_5xy + b_6y^2 + b_7x^3 + b_8x^2y + b_9xy^2 + b_{10}y^3 \quad (8.20b)$$

where  $a_1, \dots, a_{10}$  and  $b_1, \dots, b_{10}$  are the coefficients to be determined.

Now let's move to the calculation of the shape functions. Based on Eq. (8.14), all the 10 shape functions can be worked out. Below the shape functions for one corner node (e.g., node 1), one boundary middle node (e.g., node 4) and the center node (e.g., node 10) are directly given:

$$\begin{aligned}
 N_1 &= \frac{1}{2}L_1(3L_1 - 1)(3L_1 - 2) \\
 N_4 &= \frac{3}{2}L_13L_23\left(L_1 - \frac{1}{3}\right) = \frac{9}{2}L_1L_2(3L_1 - 1) \\
 N_{10} &= 3L_13L_23L_3 = 27L_1L_2L_3
 \end{aligned} \tag{8.21}$$

Until now, the procedure and formulations for solving the high order Lagrange triangular elements are given. The readers should be able to work out the total number of nodes, the displacement field formulations and the shape functions for the triangular element with any order.

### 8.4.2 The High Order Rectangular Element

Previously in Chap. 6, the FE calculations using the 4-node first order rectangular element have been given. Different from the first order triangular element, the displacement field in the first order rectangular element is a quadratic function of the coordinates and the strain field within the element is a linear function of the coordinates. Compared to the triangular element, the application of the rectangular element in the engineering problem is always preferred because of many advantageous reasons. For example, the occurrence rate of element distortion in the rectangular element is much lower than that in the triangular element. To meet the high demands from the nonlinear engineering problems associated with large deformations, the high order rectangular elements are demanded. In this part, the calculation of the displacement field formulations and the shape functions are introduced. It should be noted that the Lagrange rectangular element is the extension of the bar element from 1 to 2D. Therefore, the knowledge introduced in the bar element, e.g., the Lagrange interpolation function, can be used in this part.

First, let's have a look at the calculation of the shape functions for the high order rectangular elements. A high order rectangular element shown in Fig. 8.7 is used for the demonstration. There are  $(r+1) \times (p+1)$  nodes in the rectangular element and the dimensionless coordinates  $\xi$  and  $\eta$  are used, which are defined as below:

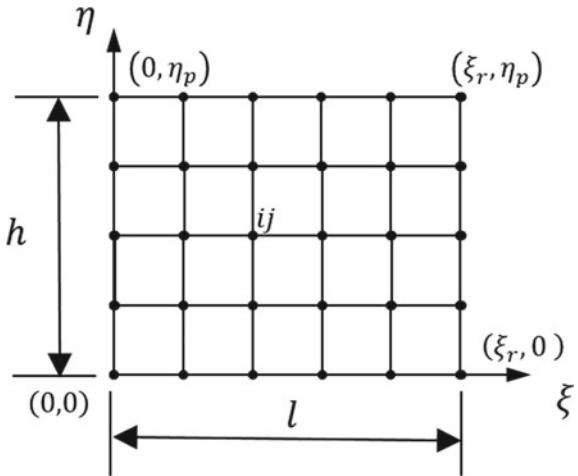
$$\xi = \frac{x - x_1}{x_n - x_1} = \frac{x - x_1}{l} \tag{8.22}$$

$$\eta = \frac{y - y_1}{y_n - y_1} = \frac{y - y_1}{h} \tag{8.23}$$

where  $l$  is the length of the rectangular element in the  $x$  direction and  $h$  is the height of the rectangular element in the  $y$  direction.

The Lagrange interpolation functions in the  $\xi$  and  $\eta$  directions can be defined as:

**Fig. 8.7** Illustration of the high order rectangular element



$$l_i^{(r)}(\xi) = \frac{(\xi - \xi_0)(\xi - \xi_1) \dots (\xi - \xi_{i-1})(\xi - \xi_{i+1}) \dots (\xi - \xi_r)}{(\xi_i - \xi_0)(\xi_i - \xi_1) \dots (\xi_i - \xi_{i-1})(\xi_i - \xi_{i+1}) \dots (\xi_i - \xi_r)} \quad (i = 0, 1, \dots, r) \quad (8.24)$$

$$l_j^{(p)}(\eta) = \frac{(\eta - \eta_0)(\eta - \eta_1) \dots (\eta - \eta_{j-1})(\eta - \eta_{j+1}) \dots (\eta - \eta_p)}{(\eta_j - \eta_0)(\eta_j - \eta_1) \dots (\eta_j - \eta_{j-1})(\eta_j - \eta_{j+1}) \dots (\eta_j - \eta_p)} \quad (j = 0, 1, \dots, p) \quad (8.25)$$

where  $r$  and  $p$  are the orders of the interpolation functions in the  $\xi$  and  $\eta$  directions, respectively,  $i$  and  $j$  are the number indices of the nodes in the  $\xi$  and  $\eta$  directions, respectively. It should be noted that both  $i$  and  $j$  start from zero. Based on the Lagrange interpolation functions defined in Eqs. (8.24) and (8.25), the shape function at the node  $ij$  of the rectangular element can be written as:

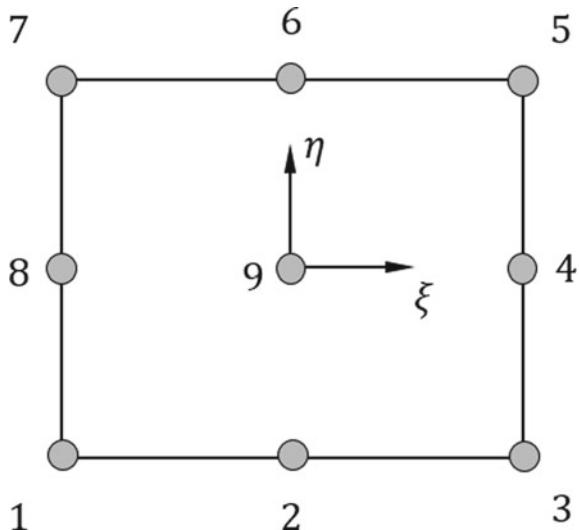
$$N_{ij} = l_i^{(r)}(\xi)l_j^{(p)}(\eta) \quad (8.26)$$

where  $l_i^{(r)}(\xi)$  and  $l_j^{(p)}(\eta)$  are the interpolation functions given in the  $x$  and  $y$  directions, respectively. Equation (8.26) is the generalized equation for the shape function in the high order rectangular element. Examples of high order rectangular elements are used below to demonstrate the calculation of the shape functions and the corresponding displacement field formulations.

First, let's have a look at the 9-node second order rectangular element (Fig. 8.8). According to Eq. (8.26), the shape functions at the corner nodes can be written as:

$$N_1 = \frac{1}{4}\xi\eta(\xi - 1)(\eta - 1) \quad (8.27a)$$

**Fig. 8.8** The 9-node second order rectangular element



The shape functions at the boundary middle nodes, e.g., nodes 2, 4, 6 and 8, can be written as:

$$N_2 = \frac{1}{2}\eta(1 - \xi^2)(\eta - 1) \quad (8.27b)$$

The shape functions at the center node can be written as:

$$N_9 = (1 - \xi^2)(1 - \eta^2) \quad (8.27c)$$

where  $-1 \leq \xi \leq 1$  and  $-1 \leq \eta \leq 1$ .

Now let's have a look at an even higher order rectangular element, i.e., the 16-node third order rectangular element (Fig. 8.9).

According to Eq. (8.26), the shape functions at the corner nodes, e.g., nodes 1 and 4, can be written out as:

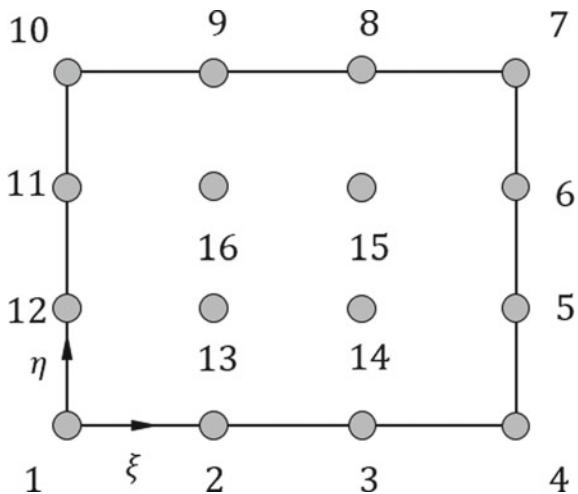
$$N_1 = \frac{81}{4} \left( \xi - \frac{1}{3} \right) \left( \xi - \frac{2}{3} \right) (\xi - 1) \left( \eta - \frac{1}{3} \right) \left( \eta - \frac{2}{3} \right) (\eta - 1) \quad (8.28a)$$

The shape functions at the boundary middle nodes, e.g., nodes 2 and 3, can be written out as:

$$N_2 = -\frac{243}{4} \xi \left( \xi - \frac{2}{3} \right) (\xi - 1) \left( \eta - \frac{1}{3} \right) \left( \eta - \frac{2}{3} \right) (\eta - 1) \quad (8.28b)$$

The shape functions at the elemental inner nodes, e.g., nodes 13 and 14, can be written out as:

**Fig. 8.9** The 16-node third order rectangular element



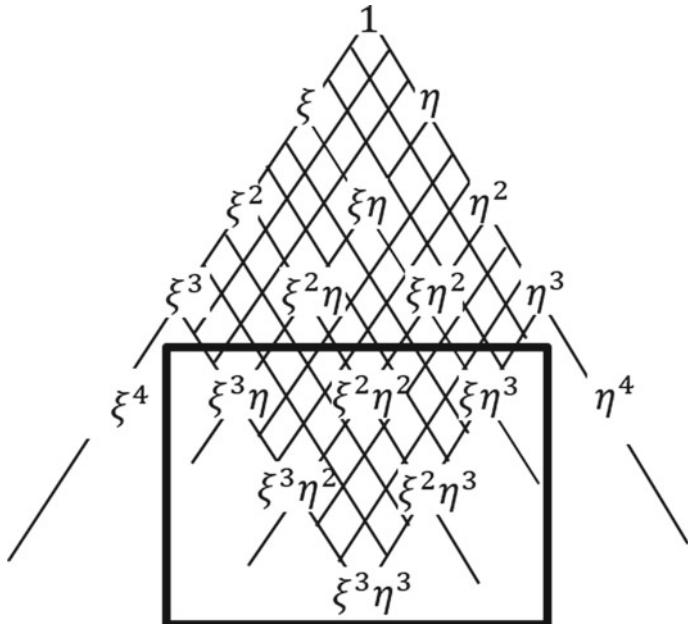
$$N_{13} = \frac{729}{4} \xi \left( \xi - \frac{2}{3} \right) (\xi - 1) \eta \left( \eta - \frac{2}{3} \right) (\eta - 1) \quad (8.28c)$$

where  $-1 \leq \xi \leq 1$  and  $-1 \leq \eta \leq 1$ .

It can be seen that using Eq. (8.26), the shape functions for the Lagrange rectangular element with any order can be worked out. Now we need to answer two more questions: first, how to determine the total number of nodes for a high order rectangular element. Second, how to write out the displacement field functions for a high order rectangular element. To answer these two questions, we still need to refer to the Pascal's triangle (Fig. 8.10). It should be noted that the displacement field formulations for the rectangular element are not the complete polynomial functions and consequently the rhombus shapes in the Pascal's triangle are formed and used to work out the displacement field formulations. For example, for a second order rectangular element, the complete terms until the third row in the Pascal's triangle should be included. Additionally, a rhombus which is symmetric about the third row should be formed and the items covered by this rhombus are the terms used for this second order rectangular element. The number of the order for the second order rectangular element is the number of corresponding terms in the Pascal's triangle. Therefore, there are 9 nodes in the second order rectangular element. The terms covered by the rhombus are used to work out the corresponding displacement field formulations (Fig. 8.11):

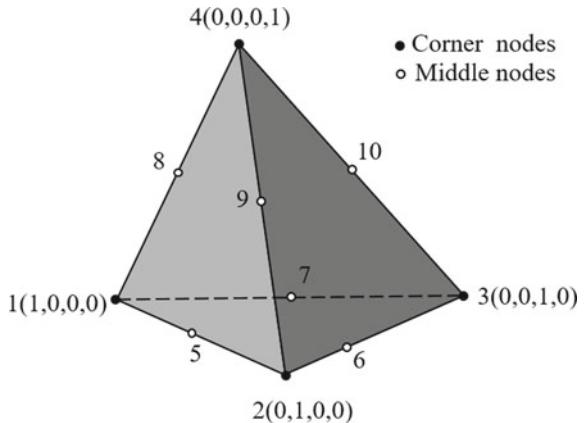
$$u(x, y) = a_1 + a_2\xi + a_3\eta + a_4\xi^2 + a_5\xi\eta + a_6\eta^2 + a_7\xi^2\eta + a_8\xi\eta^2 + a_9\xi^2\eta^2 \quad (8.29a)$$

$$v(x, y) = b_1 + b_2\xi + b_3\eta + b_4\xi^2 + b_5\xi\eta + b_6\eta^2 + b_7\xi^2\eta + b_8\xi\eta^2 + b_9\xi^2\eta^2 \quad (8.29b)$$



**Fig. 8.10** Illustration of the inner nodes using the Pascal's triangle

**Fig. 8.11** The 10-node second order tetrahedral element



where  $a_1, \dots, a_9$  and  $b_1, \dots, b_9$  are the coefficients to be determined.

Now a third order Lagrange rectangular element is used to further demonstrate the calculation process. First, because it is a third order element, the terms until the fourth row in the Pascal's triangle are complete and a rhombus which is symmetric about the fourth row should be formed. Then, the terms which should be included in the displacement field functions can be worked out, i.e., the terms forming the rhombus shape in Fig. 8.10. There are 16 terms (i.e., the constant term from the 0th

row, the linear terms  $\xi$  and  $\eta$  from the 1st row, the quadratic terms  $\eta^2$ ,  $\xi\eta$  and  $\xi^2$  from the 2nd row, the cubic terms  $\xi^3$ ,  $\xi^2\eta$ ,  $\xi\eta^2$  and  $\eta^3$  from the 3rd row, the fourth-order terms  $\xi^3\eta$ ,  $\xi^2\eta^2$  and  $\xi\eta^3$  from the 4th row, the fifth-order terms  $\xi^3\eta^2$  and  $\xi^2\eta^3$  from the 5th row and the sixth-order term  $\xi^3\eta^3$  from the 6th row) and thus the number of nodes for the third order rectangular element is 16. The corresponding displacement field formulations can be written as:

$$u(x, y) = a_1 + a_2\xi + a_3\eta + a_4\xi^2 + a_5\xi\eta + a_6\eta^2 + a_7\xi^3 + a_8\xi^2\eta + a_9\xi\eta^2 + a_{10}\eta^3 + a_{11}\xi^3\eta + a_{12}\xi^2\eta^2 + a_{13}\xi\eta^3 + a_{14}\xi^3\eta^2 + a_{15}\xi^2\eta^3 + a_{16}\xi^3\eta^3 \quad (8.30a)$$

$$v(x, y) = b_1 + b_2\xi + b_3\eta + b_4\xi^2 + b_5\xi\eta + b_6\eta^2 + b_7\xi^3 + b_8\xi^2\eta + b_9\xi\eta^2 + b_{10}\eta^3 + b_{11}\xi^3\eta + b_{12}\xi^2\eta^2 + b_{13}\xi\eta^3 + b_{14}\xi^3\eta^2 + b_{15}\xi^2\eta^3 + b_{16}\xi^3\eta^3 \quad (8.30b)$$

where  $a_1, \dots, a_{16}$  and  $b_1, \dots, b_{16}$  are the coefficients to be determined.

The corresponding nodal displacement vectors of the second order and the third order rectangular elements are given as below:

$$\mathbf{q}^e = [u_1 \ v_1 \ u_2 \ v_2 \ \dots \ u_9 \ v_9]^T \quad (8.31a)$$

$$\mathbf{q}'^e = [u_1 \ v_1 \ u_2 \ v_2 \ \dots \ u_{16} \ v_{16}]^T \quad (8.31b)$$

It can be seen from the calculations above that the increase of the order of the function in the rectangular will make the rhombus bigger and the number of incomplete form will be increased. Correspondingly, the number of inner nodes in the rectangular element is increased, which does not necessarily contribute to the accuracy of the rectangular element. To overcome this obvious shortcoming, some improved elements have been proposed, e.g., the serendipity rectangular elements. The readers can refer to other FE books for more details on the serendipity rectangular element.

## 8.5 The 3D High Order Lagrange Element

Previously in Chap. 7, we have introduced the 4-node linear tetrahedral element and the 8-node hexahedral element. The 4-node tetrahedral element is a linear element, which implies that the displacement within the element is a linear function of the coordinate and the strain within the tetrahedron is constantly distributed. The 8-node hexahedral element is also a first order element, which implies that the first order complete form is used in the displacement field formulation. As discussed before, the accuracy of the first order tetrahedral and hexahedral elements is limited in many engineering problems and consequently the high order 3D elements are required. In this chapter, the 3D high order Lagrange elements, i.e., the high order tetrahedral

element and the high order hexahedral element are introduced. It should be noted that these high order 3D elements are the extensions from the corresponding 2D ones. Therefore, the knowledge of the previous part can be used to solve the high order 3D Lagrange elements.

### 8.5.1 The High Order Tetrahedral Element

The high order Lagrange tetrahedral element is an extension from the triangular element. Similar to the area coordinates used in the triangular element, the volume coordinates are defined and used in the tetrahedral element.

Below the 10-node second order tetrahedral element (Fig. 8.11) is used to illustrate the calculations of the high order Lagrange tetrahedral elements, i.e., the calculation of the shape functions and the displacement field functions.

The volume coordinates for the 10-node tetrahedron are defined as below:

$$L_i = V_i / V \quad (i = 1, 2, 3, 4) \quad (8.32)$$

where  $V$  is the volume of the tetrahedron,  $V_i$  is the volume of the sub-tetrahedron within the tetrahedron. There are four faces in the tetrahedron and thus four sub-tetrahedrons can be formed, which are denoted as  $V_1, V_2, V_3$  and  $V_4$ . The corresponding coordinates are denoted as  $L_1, L_2, L_3$  and  $L_4$ . Similar to the shape functions used for the triangular, the shape functions for the 10-node tetrahedron can be worked out using the volume coordinates as below:

$$N_i = \prod_{j=1}^n \frac{f_j^{(i)}(L_1, L_2, L_3, L_4)}{f_j^{(i)}(L_{1i}, L_{2i}, L_{3i}, L_{4i})} \quad (8.33)$$

where  $n$  is the order of the element,  $f_j^{(i)}(L_1, L_2, L_3, L_4)$  are the left items of the surface equations going through all the nodes except for node  $i$ ,  $L_{1i}, L_{2i}, L_{3i}$  and  $L_{4i}$  are the volume coordinates of node  $i$ .

Similar to the processing in the triangular element, the number of the items in the denominator should match with the order of the element. For example, in the 10-node second order tetrahedral element, two items should appear in the denominator of the shape function and thus we should find two planes going through all the nodes except for its own node. Now let's use Eq. (8.33) to demonstrate the calculation of the shape function at corner and boundary middle nodes of the second order tetrahedral element:

At the corner nodes, the shape functions can be worked out as:

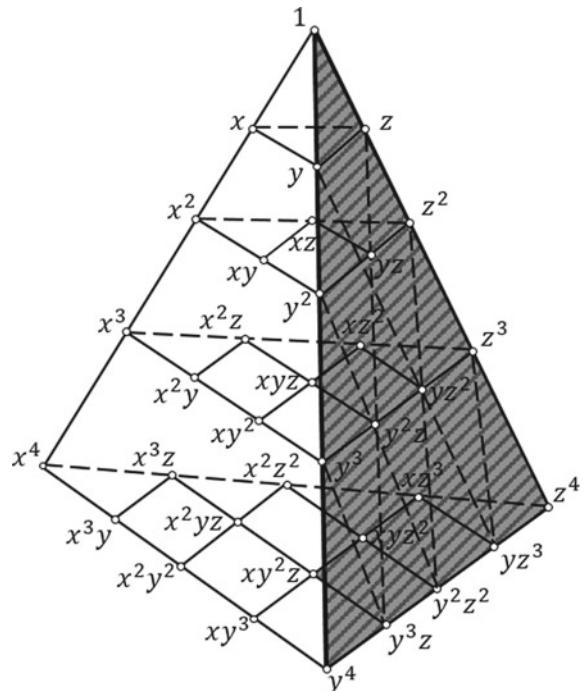
$$N_i = (2L_i - 1)L_i \quad (i = 1, 2, 3, 4) \quad (8.34)$$

At the boundary middle nodes, the shape functions can be worked out as:

$$\begin{aligned}
 N_5 &= 4L_1L_2 \\
 N_6 &= 4L_2L_3 \\
 N_7 &= 4L_3L_1 \\
 N_8 &= 4L_1L_4 \\
 N_9 &= 4L_2L_4 \\
 N_{10} &= 4L_3L_4
 \end{aligned} \tag{8.35}$$

Now let's determine the total number of nodes used for the high order tetrahedral element and in the corresponding displacement field formulations. Previously, we have used the Pascal's triangle to solve these issues of high order triangular element. Here, we can use the Pascal's pyramid (Fig. 8.12) to work out the displacement field formulations. The displacement field formulations for the high order tetrahedral element are the complete polynomial functions. Therefore, the complete items in the Pascal's pyramid should be included. In Fig. 8.12, the items in the first, second, third planes correspond to the 0th, first and second orders, respectively. Therefore, all the items until (including) the second plane should be included into the second order tetrahedral element. We can work out that there are 10 items. Therefore, there are 10 nodes in the second order tetrahedral element. Furthermore, the 10 items, i.e., 1,  $x$ ,  $y$ ,  $z$ ,  $xy$ ,  $yz$ ,  $xz$ ,  $x^2$ ,  $y^2$  and  $z^2$ , should be included in the displacement field function. There are 3 degrees of freedom at each node of the tetrahedron, i.e.,  $u$ ,  $v$  and  $w$ .

**Fig. 8.12** The Pascal's pyramid



Therefore, there are 30 components in the elemental nodal displacement vector, which can be written out as below:

$$\mathbf{q}^e = [u_1 \ v_1 \ w_1 \dots u_{10} \ v_{10} \ w_{10}]^T \quad (8.36)$$

where  $u_1, \dots, u_{10}, v_1, \dots, v_{10}$  and  $w, \dots, w_{10}$  are the displacements in the  $x$ ,  $y$  and  $z$  directions, respectively.

Regarding the displacement field formulations for the tetrahedron, three displacement field functions are used to describe the displacements in the  $x$ ,  $y$  and  $z$  directions, independently. Each displacement field function should include the 10 terms, i.e.,  $1, x, y, z, xy, yz, xz, x^2, y^2$  and  $z^2$ . Therefore, the three displacement field functions can be written as below:

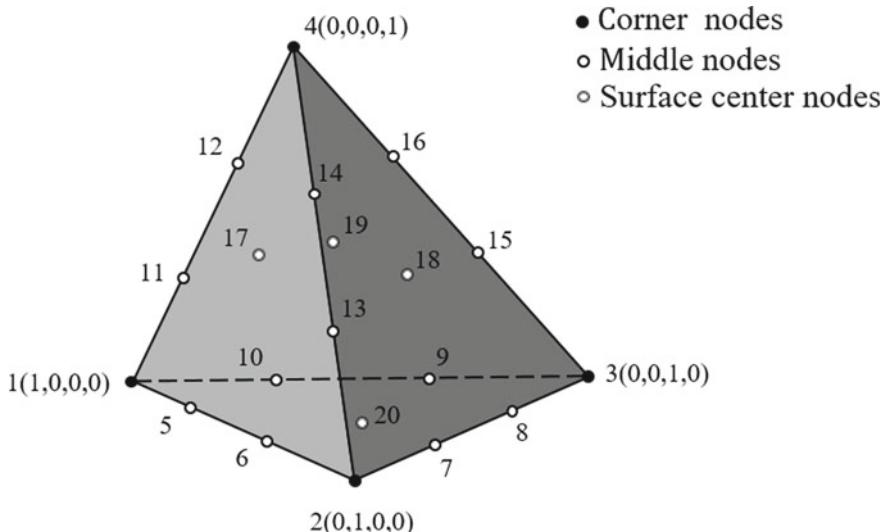
$$u(x, y, z) = a_1 + a_2x + a_3y + a_4z + a_5xy + a_6yz + a_7xz + a_8x^2 + a_9y^2 + a_{10}z^2 \quad (8.37a)$$

$$v(x, y, z) = b_1 + b_2x + b_3y + b_4z + b_5xy + b_6yz + b_7xz + b_8x^2 + b_9y^2 + b_{10}z^2 \quad (8.37b)$$

$$w(x, y, z) = c_1 + c_2x + c_3y + c_4z + c_5xy + c_6yz + c_7xz + c_8x^2 + c_9y^2 + c_{10}z^2 \quad (8.37c)$$

where  $a_1, a_2, \dots, a_{10}, b_1, \dots, b_{10}$  and  $c_1, \dots, c_{10}$  are the coefficients to be determined.

Below a 20-node third order tetrahedral element (Fig. 8.13) is used to further



**Fig. 8.13** The 20-node third order tetrahedral element

demonstrate the calculation of the displacement field functions and the shape functions for the high order tetrahedral element.

First, according to Eq. (8.33), the shape functions at one representative corner node can be written out as:

$$N_1 = \frac{1}{2}L_1(3L_1 - 1)(3L_1 - 2) \quad (8.38a)$$

The shape functions at one representative boundary node can be written out as:

$$N_5 = \frac{9}{2}L_1L_2(3L_1 - 1) \quad (8.38b)$$

The shape functions at one representative face node can be written out as:

$$N_{20} = 27L_1L_2L_3 \quad (8.38c)$$

The displacement function for the third order tetrahedral element is a complete polynomial function. Therefore, the items until (including) the fourth plane should be included. It is seen that there are 20 items and thus there are 20 nodes in the third order tetrahedral element. There are 3 degrees of freedom at each node and thus there are 60 items in the elemental nodal displacement vector, which can be written out as below:

$$\mathbf{q}^e = [u_1 \ v_1 \ w_1 \cdots u_{20} \ v_{20} \ w_{20}]^T \quad (8.39)$$

The 20 items should appear in the displacement field function. Therefore, the elemental displacement field functions can be written as below:

$$\begin{aligned} u(x, y, z) = & a_1 + a_2x + a_3y + a_4z + a_5xy + a_6yz + a_7xz + a_8x^2 + a_9y^2 + a_{10}z^2 \\ & + a_{11}x^3 + a_{12}x^2y + a_{13}xy^2 + a_{14}y^3 + a_{15}y^2z + a_{16}yz^2 + a_{17}z^3 + a_{18}z^2x \\ & + a_{19}x^2z + a_{20}xyz \end{aligned} \quad (8.40)$$

$$\begin{aligned} v(x, y, z) = & b_1 + b_2x + b_3y + b_4z + b_5xy + b_6yz + b_7xz + b_8x^2 + b_9y^2 + b_{10}z^2 \\ & + b_{11}x^3 + b_{12}x^2y + b_{13}xy^2 + b_{14}y^3 + b_{15}x^2z + b_{16}xz^2 + b_{17}z^3 + b_{18}y^2z \\ & + b_{19}yz^2 + b_{20}xyz \end{aligned}$$

$$\begin{aligned} w(x, y, z) = & c_1 + c_2x + c_3y + c_4z + c_5xy + c_6yz + c_7xz + c_8x^2 + c_9y^2 + c_{10}z^2 \\ & + c_{11}x^3 + c_{12}x^2y + c_{13}xy^2 + c_{14}y^3 + c_{15}x^2z + c_{16}xz^2 + c_{17}z^3 + c_{18}y^2z \\ & + c_{19}yz^2 + c_{20}xyz \end{aligned}$$

where  $a_1, a_2, \dots, a_{20}$ ,  $b_1, b_2, \dots, b_{20}$  and  $c_1, c_2, \dots, c_{20}$  are the coefficients to be determined.

It should be noted that although only the calculation of the displacement field functions and the shape functions for the second order and the third order tetrahedral

elements are demonstrated, we should now have the skills to work out the displacement field functions and the shape functions for any high order Lagrange tetrahedral element.

### 8.5.2 The High Order Hexahedral Element

The high order Lagrange hexahedral element is the extension from the Lagrange rectangular element. In this part, a second order Lagrange hexahedral element is used to illustrate the calculation of the shape functions and the displacement field functions. First, a high order hexahedral element shown in Fig. 8.14 is used to demonstrate the calculation of the shape functions for the high order hexahedral element. There are  $(r + 1) \times (p + 1) \times (m + 1)$  nodes in the hexahedral element and the dimensionless coordinates  $\xi$ ,  $\eta$  and  $\zeta$ , originated at the center of the hexahedron, are used, which are defined as below:

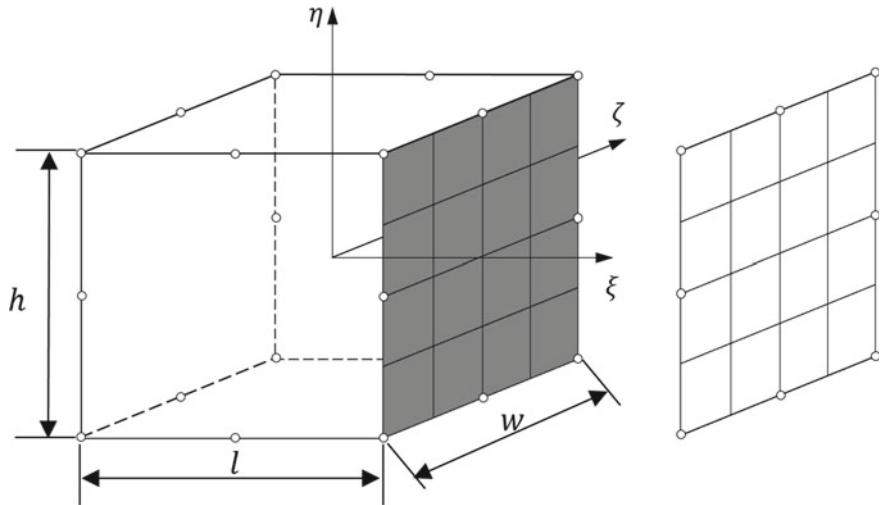
$$\begin{aligned}\xi &= \frac{x - x_1}{x_n - x_1} = \frac{x - x_1}{l} \\ \eta &= \frac{y - y_1}{y_n - y_1} = \frac{y - y_1}{h} \\ \zeta &= \frac{z - z_1}{z_n - z_1} = \frac{z - z_1}{w}\end{aligned}\quad (8.41)$$

where  $l$  is the length of the hexahedral element,  $h$  is the height of the hexahedral element and  $w$  is the width of the hexahedral element.

Then the Lagrange interpolation function in the  $\xi$ ,  $\eta$  and  $\zeta$  directions can be defined as:

$$\begin{aligned}l_i^{(r)}(\xi) &= \frac{(\xi - \xi_0)(\xi - \xi_1) \dots (\xi - \xi_{i-1})(\xi - \xi_{i+1}) \dots (\xi - \xi_r)}{(\xi_i - \xi_0)(\xi_i - \xi_1) \dots (\xi_i - \xi_{i-1})(\xi_i - \xi_{i+1}) \dots (\xi_i - \xi_r)} \\ (i &= 0, 1, \dots, r) \\ l_j^{(p)}(\eta) &= \frac{(\eta - \eta_0)(\eta - \eta_1) \dots (\eta - \eta_{j-1})(\eta - \eta_{j+1}) \dots (\eta - \eta_p)}{(\eta_j - \eta_0)(\eta_j - \eta_1) \dots (\eta_j - \eta_{j-1})(\eta_j - \eta_{j+1}) \dots (\eta_j - \eta_p)} \\ (j &= 0, 1, \dots, p) \\ l_k^{(m)}(\zeta) &= \frac{(\zeta - \zeta_0)(\zeta - \zeta_1) \dots (\zeta - \zeta_{k-1})(\zeta - \zeta_{k+1}) \dots (\zeta - \zeta_m)}{(\zeta_k - \zeta_0)(\zeta_k - \zeta_1) \dots (\zeta_k - \zeta_{k-1})(\zeta_k - \zeta_{k+1}) \dots (\zeta_k - \zeta_m)} \\ (k &= 0, 1, \dots, m)\end{aligned}\quad (8.42)$$

where  $r$ ,  $p$  and  $m$  are the orders of the functions in the  $\xi$ ,  $\eta$  and  $\zeta$  directions, respectively,  $i$ ,  $j$  and  $k$  are the number indices of the nodes in the  $\xi$ ,  $\eta$  and  $\zeta$  directions, respectively. It should be noted that  $i$ ,  $j$  and  $k$  start from zero.



**Fig. 8.14** Illustration of the high order hexahedral element

Based on the Lagrange interpolation functions defined above, the shape function at the node  $ijk$  can be given as:

$$N_{ijk} = l_i^{(r)}(\xi)l_j^{(p)}(\eta)l_k^{(m)}(\zeta) \quad (8.43)$$

where  $l_i^{(r)}(\xi)$ ,  $l_j^{(p)}(\eta)$  and  $l_k^{(m)}(\zeta)$  are the interpolation functions given in the  $x$ ,  $y$  and  $z$  directions, respectively.

Below a 27-node second order hexahedral element (Fig. 8.15) is used to demonstrate the calculation of the shape functions and the displacement field functions.

According to Eq. (8.43), the shape functions at the corner nodes (e.g., node 27) can be written out as:

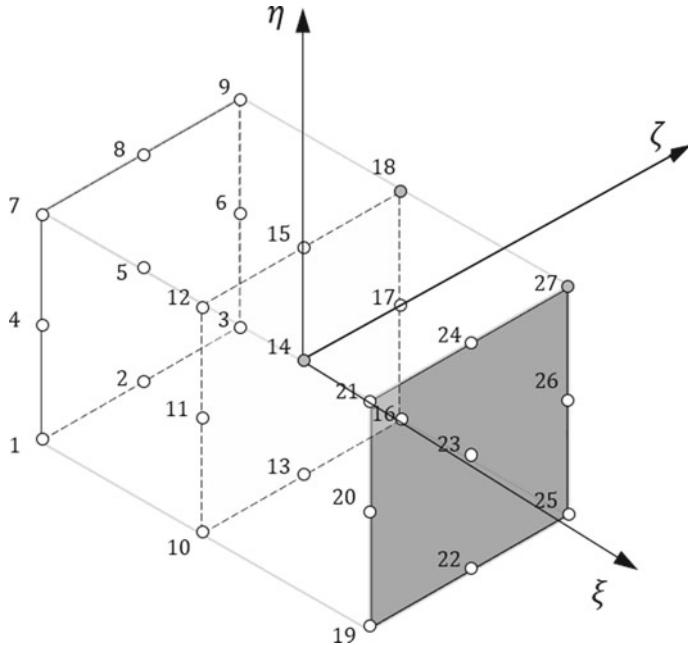
$$N_{27} = \frac{1}{8}\xi\eta\zeta(1+\xi)(1+\eta)(1+\zeta) \quad (8.44a)$$

The shape functions at the boundary middle nodes (e.g., node 18) can be written out as:

$$N_{18} = \frac{1}{4}\eta\zeta(1-\xi^2)(1+\eta)(1+\zeta) \quad (8.44b)$$

The shape functions at the inner center nodes (e.g., node 14) can be written out as:

$$N_{14} = (1-\xi^2)(1-\eta^2)(1-\zeta^2) \quad (8.44c)$$



**Fig. 8.15** The 27-node second order hexahedral element

where  $-1 \leq \xi \leq 1$ ,  $-1 \leq \eta \leq 1$  and  $-1 \leq \zeta \leq 1$ .

Now let's determine the total number of nodes in the second order hexahedral element and the corresponding displacement field formulations. The Pascal's pyramid (Fig. 8.12) is used to work out the displacement field formulations. Similar to the rectangular element, a symmetric diamond is formed and all the items covered by this diamond should be used. Therefore, in the second order hexahedral element, all the items until (including) the second plane should be included. We can work out that there are 27 items. Therefore, there are 27 nodes in the second order hexahedral element. Furthermore, 27 items (i.e., the constant term from the 0th plane, the linear terms  $x$ ,  $y$  and  $z$  from the 1st plane, the quadratic terms  $x^2$ ,  $xy$ ,  $y^2$ ,  $yz$ ,  $z^2$  and  $xz$  from the 2nd plane, the cubic terms  $x^2y$ ,  $xy^2$ ,  $y^2z$ ,  $yz^2$ ,  $xz^2$ ,  $x^2z$  and  $xyz$  from the 3rd plane, the fourth-order terms  $x^2y^2$ ,  $xy^2z$ ,  $y^2z^2$ ,  $xyz^2$ ,  $x^2z^2$  and  $x^2yz$  from the 4th plane, the fifth-order term  $xy^2z^2$ ,  $x^2y^2z$  and  $x^2yz^2$  from the 5th plane and the sixth-order term  $x^2y^2z^2$  from the 6th plane) should be included in the displacement field function. There are 3 degrees of freedom at each node of the tetrahedron, i.e.,  $u$ ,  $v$  and  $w$ . Therefore, there are  $27 \times 3 = 81$  components in the elemental nodal displacement vector, which can be written as below:

$$\mathbf{q}^e = [u_1 \ v_1 \ w_1 \cdots u_{27} \ v_{27} \ w_{27}]^T \quad (8.45)$$

Three displacement field functions are used to describe the displacements in the  $x$ ,  $y$  and  $z$  directions, independently. Each displacement field function should include 27 terms. Therefore, the three displacement field functions can be written as below:

$$\begin{aligned} u(x, y, z) = & a_1 + a_2x + a_3y + a_4z + a_5xy + a_6yz + a_7xz + a_8x^2 + a_9y^2 \\ & + a_{10}z^2 + a_{11}x^2y + a_{12}xy^2 + a_{13}y^2z + a_{14}yz^2 + a_{15}xz^2 + a_{16}x^2z \\ & + a_{17}xyz + a_{18}x^2y^2 + a_{19}xy^2z + a_{20}y^2z^2 + a_{21}xyz^2 + a_{22}x^2z^2 \\ & + a_{23}x^2yz + a_{24}x^2y^2z + a_{25}xy^2z^2 + a_{26}x^2yz^2 + a_{27}x^2y^2z^2 \end{aligned}$$

$$\begin{aligned} v(x, y, z) = & b_1 + b_2x + b_3y + b_4z + b_5xy + b_6yz + b_7xz + b_8x^2 + b_9y^2 \\ & + b_{10}z^2 + b_{11}x^2y + b_{12}xy^2 + b_{13}y^2z + b_{14}yz^2 + b_{15}xz^2 \\ & + b_{16}x^2z + b_{17}xyz + b_{18}x^2y^2 + b_{19}xy^2z + b_{20}y^2z^2 + b_{21}xyz^2 \\ & + b_{22}x^2z^2 + b_{23}x^2yz + b_{24}x^2y^2z + b_{25}xy^2z^2 + b_{26}x^2yz^2 + b_{27}x^2y^2z^2 \end{aligned}$$

$$\begin{aligned} w(x, y, z) = & c_1 + c_2x + c_3y + c_4z + c_5xy + c_6yz + c_7xz + c_8x^2 + c_9y^2 \\ & + c_{10}z^2 + c_{11}x^2y + c_{12}xy^2 + c_{13}y^2z + c_{14}yz^2 + c_{15}xz^2 + c_{16}x^2z \\ & + c_{17}xyz + c_{18}x^2y^2 + c_{19}xy^2z + c_{20}y^2z^2 + c_{21}xyz^2 + c_{22}x^2z^2 \\ & + c_{23}x^2yz + c_{24}x^2y^2z + c_{25}xy^2z^2 + c_{26}x^2yz^2 + c_{27}x^2y^2z^2 \end{aligned}$$

where  $a_1, a_2, \dots, a_{27}, b_1, \dots, b_{27}$  and  $c_1, \dots, c_{27}$  are the coefficients to be determined.