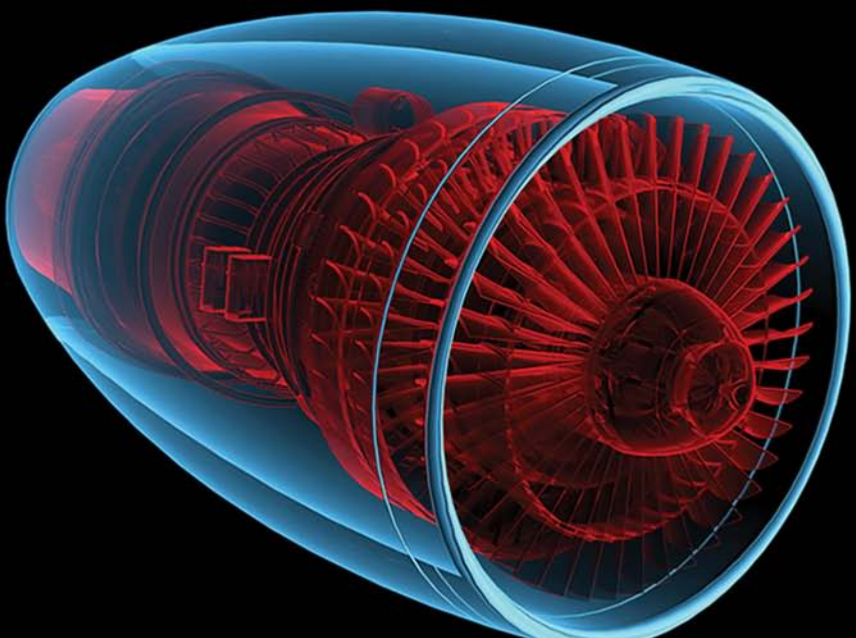
Finite Element Analysis

A Primer

SECOND EDITION



Sarhan M. Musa

FINITE ELEMENT ANALYSIS

Second Edition

Companion files for this title are available by writing to the publisher at info@merclearning.com.

FINITE ELEMENT ANALYSIS

A Primer

Second Edition

SARHAN M. MUSA
(Prairie View A & M University)

MERCURY LEARNING AND INFORMATION
Boston, Massachusetts

Publisher: David Pallai
Mercury Learning and Information
121 High Street, 3rd Floor
Boston, MA 02110
info@merclearning.com
www.degruyter.com
800-232-0223

S. M. Musa. *FINITE ELEMENT ANALYSIS: A Primer. Second Edition.*
ISBN: 978-1-68392-415-9

Library of Congress Control Number: 2023942918
232425321   This book is printed on acid-free paper in the United States of America.

Dedicated to my late father, Mahmoud, my late mother, Fatmeh, and my wife, Lama.

CONTENTS

Preface *xiii*

CHAPTER 1: MATHEMATICAL PRELIMINARIES **1**

1.1 Introduction 1
1.2 Matrix Definition 1
1.3 Types of Matrices 3
1.4 Addition or Subtraction of Matrices 6
1.5 Multiplication of a Matrix by Scalar 7
1.6 Multiplication of a Matrix by Another Matrix 8
1.7 Rules of Matrix Multiplications 9
1.8 Transpose of a Matrix Multiplication 12
1.9 Trace of a Matrix 13
1.10 Differentiation of a Matrix 14
1.11 Integration of a Matrix 14
1.12 Equality of Matrices 15
1.13 Determinant of a Matrix 15
1.14 Direct Methods for Linear Systems 18
1.15 Gaussian Elimination Method 19
1.16 Cramer's Rule 21
1.17 Inverse of a Matrix 24
1.18 Vector Analysis 27
1.19 Eigenvalues and Eigenvectors 36

1.20 Using MATLAB 40
Exercises 49
References 52

CHAPTER 2: INTRODUCTION TO THE FINITE ELEMENT METHOD 53

2.1 Introduction 53
2.2 Methods of Solving Engineering Problems 53
2.2.1 Experimental Method 54
2.2.2 Analytical Method 54
2.2.3 Numerical Method 54
2.3 Procedure of Finite Element Analysis (Related to Structural Problems) 54
2.4 Methods of Prescribing Boundary Conditions 56
2.4.1 Elimination Method 57
2.4.2 Penalty Method 57
2.4.3 Multipoint Constrains Method 57
2.5 Practical Applications of Finite Element Analysis 58
2.6 Finite Element Analysis Software Package 58
2.7 Finite Element Analysis for Structure 58
2.8 Types of Elements 59
2.9 Direct Method for Linear Spring 62
Exercises 63
References 64

CHAPTER 3: FINITE ELEMENT ANALYSIS OF AXIALLY LOADED MEMBERS 65

3.1 Introduction 65
3.1.1 Two-Node Bar Element 68
3.1.2 Three-Node Bar Element 70
3.2 Bars of Constant Cross-Section Area 71
3.3 Bars of Varying Cross-Section Area 104
3.4 Stepped Bar 126
Exercises 152
References 156

CHAPTER 4: FINITE ELEMENT ANALYSIS TRUSSES **157**
4.1 Introduction 157
4.2 Truss 157
Exercises 183
References 186

CHAPTER 5: FINITE ELEMENT ANALYSIS OF BEAMS **189**
5.1 Introduction 189
5.2 Simply Supported Beams 190
5.3 Cantilever Beams 216
Exercises 247
References 250

CHAPTER 6: STRESS ANALYSIS OF A RECTANGULAR PLATE WITH A CIRCULAR HOLE **253**
6.1 Introduction 253
6.2 A Rectangular Plate with a Circular Hole 253
Exercises 268
References 272

CHAPTER 7: THERMAL ANALYSIS **273**
7.1 Introduction 273
7.2 Procedure of Finite Element Analysis (Related to Thermal Problems) 274
7.3 One-Dimensional Heat Conduction 274
7.4 Two-Dimensional Problem with Conduction and with Convection Boundary Conditions 305
Exercises 307
References 310

CHAPTER 8: FLUID FLOW ANALYSIS **311**
8.1 Introduction 311
8.2 Procedure of Finite Element Analysis (Related to Fluid Flow Problems) 312
8.3 Potential Flow Over a Cylinder 313
8.4 Potential Flow Around an Airfoil 316

Exercises 322

References 325

CHAPTER 9: DYNAMIC ANALYSIS 327

9.1 Introduction 327

9.2 Procedure of Finite Element Analysis (Related to Dynamic Problems) 328

9.3 Fixed-Fixed Beam for Natural Frequency Determination 328

9.4 Transverse Vibrations of a Cantilever Beam 340

9.5 Fixed-Fixed Beam Subjected to Forcing Function 348

9.6 Axial Vibrations of a Bar 362

9.7 Bar Subjected to Forcing Function 373

Exercises 375

References 377

CHAPTER 10: ENGINEERING ELECTROMAGNETICS ANALYSIS 379

10.1 Introduction to Electromagnetics 379

10.2 Maxwell's Equations and Continuity Equation 379

10.2.1 Maxwell's Equations and Continuity Equation in Differential Form 380

10.2.2 Maxwell's Equations and Continuity Equation in Integral Form 381

10.2.3 Divergence and Stokes Theorems 381

10.2.4 Maxwell's Equations and Continuity Equation in Quasi-Statics Case 382

10.2.5 Maxwell's Equations and Continuity Equation in Statics Case 382

10.2.6 Maxwell's Equations and Continuity Equation in Source-Free Regions of Space Case 383

10.2.7 Maxwell's Equations and Continuity Equation in Time-Harmonic Fields Case 383

10.3 Lorentz Force Law and Continuity Equation 385

10.4 Constitutive Relations 385

10.5 Potential Equations 390

10.6 Boundary Conditions 391

10.7 Laws for Static Fields in Unbounded Regions 393
10.7.1 Coulomb's Law and Field Intensity 394
10.7.2 Bio-Savart's Law and Field Intensity 395
10.8 Electromagnetic Energy and Power Flow 395
10.9 Loss in Medium 399
10.10 Skin Depth 400
10.11 Poisson's and Laplace's Equations 401
10.12 Wave Equations 402
10.13 Electromagnetic Analysis 408
10.13.1 One-Dimensional Elements 408
10.13.1.1 The Approach to FEM Standard Steps Procedure 408
10.13.1.2 Application to Poisson's Equation in One-Dimension 412
10.13.1.3 Natural Coordinates in One Dimension 417
10.13.2 Two-Dimensional Elements 418
10.13.2.1 Applications of FEM to Electrostatic Problems 418
10.14 Automatic Mesh Generation 439
10.14.1 Rectangular Domains 439
10.14.2 Arbitrary Domains 441
10.15 Higher-Order Elements 443
10.15.1 Pascal Triangle 444
10.15.2 Local Coordinates 445
10.15.3 Shape Functions 447
10.15.4 Fundamental Matrices 450
10.16 Three-Dimensional Element 454
10.17 Finite Element Methods for External Problems 460
10.17.1 Infinite Element Method 461
10.17.2 Boundary Element Method 462
10.17.3 Absorbing Boundary Conditions 462

10.18 Modeling and Simulation of Shielded Microstrip Lines with COMSOL Multiphysics 464
10.18.1 Rectangular Cross-Section Transmission Line 467
10.18.2 Square Cross-Section Transmission Line 468
10.18.3 Rectangular Line with Diamondwise Structure 469
10.18.4 A Single-Strip Shielded Transmission Line 470
10.19 Multistrip Transmission Lines 473
10.19.1 Double-Strip Shielded Transmission Line 475
10.19.2 Three-Strip Line 478
10.19.3 Six-Strip Line 480
10.19.4 Eight-Strip Line 482
10.20 Solenoid Actuator Analysis with ANSYS 484
Exercises 505
References 511

APPENDIX A: ANSYS 525

APPENDIX B: MATLAB 555

APPENDIX C: COMSOL MULTIPHYSICS 575

APPENDIX D: 4-COLOR FIGURES FROM THE TEXT (On the companion files)

INDEX 589

Preface

Today, the finite element method (FEM) has become a common and a very powerful computational tool for solving engineering problems in industries for the obvious reasons of its versatility and affordability. To expose an undergraduate student in engineering to this powerful method, most universities have included this subject in the undergraduate curriculum. This book contains materials applied to mechanical engineering, civil engineering, electrical engineering, and physics. This book is written primarily to help the students and educators as a simple introduction to the practice of FEM analysis in engineering and physics. This book contains many 1D and 2D problems solved by the analytical method, by FEM using hand calculations, and by using ANSYS academic teaching software, COMSOL, and MATLAB. Results of all the methods have been compared. This book compromises 10 chapters and 3 appendices.

Chapter 1 contains mathematical preliminaries needed for understanding the chapters of the book. Chapter 2 provides a brief introduction to FEA, a theoretical background, and its applications. Chapter 3 contains the linear static analysis of bars of constant cross-section, tapered cross-section, and stepped bar. In each section, a different variety of exercise problems are given. Chapter 4 contains the linear static analysis of trusses. Trusses problems are also selected in such a way that each problem has different boundary conditions to apply. Chapter 5 provides the linear static analysis of simply supported and cantilever beams. In Chapters 3 to 5, all the problems are considered as one dimensional in nature. Indeed, stress analysis of a rectangular plate with a circular hole is covered in Chapter 6. In this chapter, emphasis is given on the concept of exploiting symmetric geometry and symmetric loading conditions. Also, stress and deformation plots are given. Chapter 7 introduces the thermal analysis of cylinders and plates. Here both one dimensional and two-dimensional problems are considered. Chapter 8 contains the problems of potential flow distribution over a cylinder and over an airfoil. Chapter 9 provides the dynamic analysis (modal and transient analysis) of bars and beams. Chapter 10 provides the engineering electromagnetics analysis. The chapter gives an overview of electromagnetics theory and provides the finite element method analysis toward electromagnetics; some models are demonstrated using COMSOL multiphysics and ANSYS.

Appendices (available on the companion files)

Appendix A contains the introduction to Classic ANSYS and the ANSYS Workbench. Appendix B contains an overview of a computation in MATLAB. Appendix C contains an overview of COMSOL Multiphysics. Appendix D contains the color figures from the book.

Acknowledgments

It is my pleasure to acknowledge the outstanding help and support of the team at Mercury Learning and Information in preparing this book, especially from David Pallai and Jennifer Blaney.

Sarhan M. Musa
September 2023

CHAPTER 1

MATHEMATICAL PRELIMINARIES

1.1 INTRODUCTION

This chapter introduces matrix and vector algebra which is essential in the formulation and solution of finite element problems. Finite element analysis procedures are most commonly described using matrix and vector notations. These procedures eventually lead to the solutions of a large set of simultaneous equations. This chapter will be a good help in understanding the remaining chapters of the book.

1.2 MATRIX DEFINITION

A *matrix* is an array of numbers or mathematical terms arranged in rows (horizontal lines) and columns (vertical lines). The numbers, or mathematical terms, in the matrix, are called the *elements of the matrix*. We denote the matrix through this book, by a **boldface-letter**, a letter in brackets [], or a letter in braces {}. We sometimes use {} for a column matrix. Otherwise, we define the symbols of the matrices.

EXAMPLE 1.1

The following are matrices.

$$\mathbf{A}=\begin{bmatrix} 0 & 1 \\ 3 & \pi \end{bmatrix}, [B]=\begin{bmatrix} \sin\theta & 0 & 0 \\ 0 & \cos\theta & 0 \\ 0 & \tan\theta & 0 \end{bmatrix}, \{C\}=\begin{bmatrix} \int_0^3 x\,dx \\ \int_4^{11} y\,dy \end{bmatrix}, [D]=\begin{bmatrix} \dfrac{\partial f(x,y)}{\partial x} & \dfrac{\partial f(x,y)}{\partial y} \end{bmatrix}, \ \mathbf{E}=[e]$$

The size (dimension or order) of the matrices varies and is described by the number of rows (m) and the number of columns (n). Therefore, we write the size of a matrix as $m \times n$ (m by n). The sizes of the matrices in Example 1.1 are 2×2, 3×3, 2×1, 1×2, and 1×1, respectively.

We use a_{ij} to denote the element that occurs in row i and column j of matrix **A**. In general, matrix **A** can be written

$$\mathbf{A} = [A] = \begin{bmatrix} a_{11} & a_{12} & . & . & . & a_{1j} & . & . & . & a_{1n} \\ a_{21} & a_{22} & . & . & . & a_{2j} & . & . & . & a_{2n} \\ . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . \\ a_{i1} & a_{i2} & . & . & . & a_{ij} & . & . & . & a_{in} \\ . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . \\ a_{m1} & a_{m2} & . & . & . & a_{mj} & . & . & . & a_{mn} \end{bmatrix} \tag{1.1}$$

EXAMPLE 1.2

Location of an element in a matrix.

$$\text{Let } \mathbf{A} = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix}$$

Find (a) size of the matrix $\mathbf{A}$

(b) location of elements a_{11}, a_{12}, a_{32}, and a_{33}

Solution:

(a) Size of the matrix $\mathbf{A}$ is 3×3

(b) a_{11} is element *a* at row 1 and column 1

a_{12} is element *a* at row 1 and column 2

a_{32} is element *a* at row 3 and column 2

a_{33} is element *a* at row 3 and column 3

Note that, two matrices are equal if they have the same size and their corresponding elements in the two matrices are equal. For example,

let, $[A]=\begin{bmatrix}1 & 3 & 7\end{bmatrix}, [B]=\begin{bmatrix}\pi & 0\\ 1 & e\end{bmatrix}, [C]=\begin{bmatrix}e & 0\\ 1 & \pi\end{bmatrix}$, then $[A]\neq[B]$ since $[A]$ and $[B]$ are not the same size. Also, $[B]\neq[C]$ since the corresponding elements are not all equal.

1.3 TYPES OF MATRICES

The types of matrices are based on the number of rows (m) and the number of columns (n) in addition to the nature of elements and the way the elements are arranged in the matrix.

(a) ***Rectangular matrix*** is a matrix of different number of rows and columns, that is, $m \neq n$. For example, the matrix

$$[X]=\begin{bmatrix}1 & 2\\ -3 & 5\\ 7 & 0\end{bmatrix}, \text{ is rectangular matrix.}$$

(b) ***Square matrix*** is a matrix of equal number of rows and columns, that is, $m = n$. For example, the matrix

$$[K]=\begin{bmatrix}k_1 & k_2\\ k_3 & k_4\end{bmatrix}, \text{ is square matrix.}$$

(c) ***Row matrix*** is a matrix that has one row and has more than one column, that is, $m = 1$ and $n > 1$. For example, the matrix

$$[F]=\begin{bmatrix}x & y & z\end{bmatrix}, \text{ is row matrix.}$$

(d) ***Column matrix*** is a matrix that has one column and has more than one row, that is, $n = 1$ and $m > 1$. For example, the matrix

$$\mathbf{N}=\{N\}=\begin{Bmatrix}0\\ 2\\ 4\end{Bmatrix}, \text{ is column matrix.}$$

(e) ***Scalar matrix*** is a matrix that has the number of columns and the number of rows equal to 1, that is, $m = 1$ and $n = 1$. For example, the matrix

$[M] = [7]$, is a scalar matrix; we can write it as 7 without bracket.

(f) ***Null matrix*** is a matrix whose elements are all zero. For example, the matrix

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}, \text{ is a null matrix.}$$

(g) ***Diagonal matrix*** is a square matrix that has zero elements everywhere except on its main diagonal. That is, for diagonal matrix $a_{ij} = 0$, when $i \neq j$ and not all are zero for a_{ii} when $i = j$. For example, the matrix

$$\begin{bmatrix} a_{11} & 0 & 0 \\ 0 & a_{22} & 0 \\ 0 & 0 & a_{33} \end{bmatrix}, \text{ is a diagonal matrix}$$

Main diagonal

Main diagonal elements have equal row and column subscripts—the main diagonal runs from the upper-left corner to the lower-right corner. The main diagonal of the matrix here is a_{11}, a_{22}, and a_{33}.

(h) ***Identity (unit) matrix*** $[I]$ or $\mathbf{I}$, is a diagonal matrix whose main diagonal elements are equal to unity (1's) for any square matrix. That is, if the elements of an identity matrix are denoted as e_{ij}, then

$$e_{ij} = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases}. \tag{1.2}$$

For example, the matrix

$$[I] = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}, \text{ is an identity matrix.}$$

(i) ***Banded matrix*** is a square matrix that has a band of nonzero elements parallel to its main diagonal. For example, the matrix

$$\begin{bmatrix} a_{11} & a_{12} & 0 & 0 & 0 \\ a_{21} & a_{22} & a_{23} & 0 & 0 \\ 0 & a_{32} & a_{33} & a_{34} & 0 \\ 0 & 0 & a_{43} & a_{44} & a_{45} \\ 0 & 0 & 0 & a_{54} & a_{55} \end{bmatrix}$$, is a banded matrix.

(j) ***Symmetric matrix*** is a square matrix whose elements satisfy the condition $a_{ij} = a_{ji}$ for $i \neq j$. For example, the matrix

$$\begin{bmatrix} a_{11} & 5 & 8 \\ 5 & a_{22} & 2 \\ 8 & 2 & a_{33} \end{bmatrix}$$, is a symmetric matrix.

(k) ***Anti-symmetric (Skew-symmetric) matrix*** is a square matrix whose elements $a_{ij} = -a_{ji}$ for $i \neq j$, and $a_{ii} = 0$. For example, the matrix

$$\begin{bmatrix} 0 & 3 & -7 \\ -3 & 0 & 2 \\ 7 & -2 & 0 \end{bmatrix}$$, is an anti-symmetric matrix.

(l) ***Triangular matrix*** is a square matrix whose elements on one side of the main diagonal are all zero. There are two types of triangular matrices; first, an *upper triangular matrix* whose elements below the main diagonal are zero, that is, $a_{ij} = 0$ for $i > j$; second, a *lower triangular matrix* whose elements above the main diagonal are all zero, that is $a_{ij} = 0$ for $i < j$. For example, the matrix

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ 0 & a_{22} & a_{23} \\ 0 & 0 & a_{33} \end{bmatrix}$$, is an upper triangular matrix.

While the matrix

$$\begin{bmatrix} a_{11} & 0 & 0 \\ a_{21} & a_{22} & 0 \\ a_{31} & a_{32} & a_{33} \end{bmatrix}$$, is a lower triangular matrix.

(m) ***Partitioned matrix*** (***Super-matrix***) is a matrix that can be divided into smaller arrays (*submatrices*) by horizontal and vertical lines; that is, the elements of the partitioned matrix are matrices. For example, the matrix $\left[\begin{array}{cc:c} a_{11} & a_{12} & a_{13} \\ \hdashline a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{array}\right] = \left[\begin{array}{c:c} \mathbf{A} & \mathbf{B} \\ \hdashline \mathbf{C} & \mathbf{D} \end{array}\right]$**,** is partitioned matrix with four smaller matrices, where

$\mathbf{A} = \begin{bmatrix} a_{11} & a_{12} \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} a_{13} \end{bmatrix}$, $\mathbf{C} = \begin{bmatrix} a_{21} & a_{22} \\ a_{31} & a_{32} \end{bmatrix}$, and $\mathbf{D} = \begin{bmatrix} a_{23} \\ a_{33} \end{bmatrix}$. For example, the matrix

$\left[\begin{array}{c:cc} 0 & 1 & 5 \\ 8 & 3 & 4 \\ \hdashline 6 & 2 & 9 \end{array}\right] = \begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{C} & \mathbf{D} \end{bmatrix}$**,** is a partitioned matrix, where $\mathbf{A} = \begin{bmatrix} 0 \\ 8 \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} 1 & 5 \\ 3 & 4 \end{bmatrix}$, $\mathbf{C} = \begin{bmatrix} 6 \end{bmatrix}$, and $\mathbf{D} = \begin{bmatrix} 2 & 9 \end{bmatrix}$.

1.4 ADDITION OR SUBTRACTION OF MATRICES

Addition and subtraction of matrices can only be performed for matrices of the same size; that is, the matrices must have the same number of rows and columns. The addition is accomplished by adding corresponding elements of each matrix. For example, for addition of two matrices **A** and **B**, can give **C** matrix, that is, $\mathbf{C} = \mathbf{A} + \mathbf{B}$ implies that $c_{ij} = a_{ij} + b_{ij}$. Where c_{ij}, a_{ij}, and b_{ij} are typical elements of the $\mathbf{C}$, $\mathbf{A}$, and $\mathbf{B}$ matrices, respectively.

Now, the subtraction of matrices is accomplished by subtracting the corresponding elements of each matrix. For example, the subtraction of two matrices **A** and **B**, can give you **C** matrix, that is, $\mathbf{C} = \mathbf{A} - \mathbf{B}$ implies that $c_{ij} = a_{ij} - b_{ij}$. Note that, both $\mathbf{A}$ and $\mathbf{B}$ matrices are the same size, $m \times n$, then the resulting matrix **C** is also of size $m \times n$.

For example, let $[A] = \begin{bmatrix} 1 & 2 \\ 5 & 7 \end{bmatrix}$ and $[B] = \begin{bmatrix} 0 & 6 \\ 9 & 12 \end{bmatrix}$, then

$$[A] + [B] = \begin{bmatrix} 1 & 2 \\ 5 & 7 \end{bmatrix} + \begin{bmatrix} 0 & 6 \\ 9 & 12 \end{bmatrix} = \begin{bmatrix} 1+0 & 2+6 \\ 5+9 & 7+12 \end{bmatrix} = \begin{bmatrix} 1 & 8 \\ 14 & 19 \end{bmatrix}, \text{ and}$$

$$[A]-[B]=\begin{bmatrix}1 & 2\\5 & 7\end{bmatrix}-\begin{bmatrix}0 & 6\\9 & 12\end{bmatrix}=\begin{bmatrix}1-0 & 2-6\\5-9 & 7-12\end{bmatrix}=\begin{bmatrix}1 & -4\\-4 & -5\end{bmatrix}.$$

Matrices addition and subtraction are associative; that is

$$\begin{aligned}\mathbf{A}+\mathbf{B}+\mathbf{C}&=(\mathbf{A}+\mathbf{B})+\mathbf{C}=\mathbf{A}+(\mathbf{B}+\mathbf{C})\\ \mathbf{A}+\mathbf{B}-\mathbf{C}&=(\mathbf{A}+\mathbf{B})-\mathbf{C}=\mathbf{A}+(\mathbf{B}-\mathbf{C})\end{aligned}\tag{1.3}$$

For example,

let $[A]=\begin{bmatrix}1 & 3\\7 & 8\end{bmatrix}$, $[B]=\begin{bmatrix}2 & 5\\3 & 1\end{bmatrix}$, and $[C]=\begin{bmatrix}9 & 8\\4 & 6\end{bmatrix}$.

Then, $([A]+[B])+[C]=\begin{bmatrix}1+2 & 3+5\\7+3 & 8+1\end{bmatrix}+\begin{bmatrix}9 & 8\\4 & 6\end{bmatrix}=\begin{bmatrix}12 & 16\\14 & 15\end{bmatrix}$

$[A]+([B]+[C])=\begin{bmatrix}1 & 3\\7 & 8\end{bmatrix}+\begin{bmatrix}2+9 & 5+8\\3+4 & 1+6\end{bmatrix}=\begin{bmatrix}12 & 16\\14 & 15\end{bmatrix}$.

Therefore, (A+B) + C = A + (B + C).

Matrices addition and subtraction are commutative; that is

$$\begin{aligned}\mathbf{A}+\mathbf{B}&=\mathbf{B}+\mathbf{A}\\ \mathbf{A}-\mathbf{B}&=-\mathbf{B}+\mathbf{A}\end{aligned}\tag{1.4}$$

For example,

let $[A]=\begin{bmatrix}6 & 5\\2 & 1\end{bmatrix}$ and $[B]=\begin{bmatrix}3 & 2\\1 & 5\end{bmatrix}$, then $[A]+[B]=\begin{bmatrix}6 & 5\\2 & 1\end{bmatrix}+\begin{bmatrix}3 & 2\\1 & 5\end{bmatrix}=\begin{bmatrix}9 & 7\\3 & 6\end{bmatrix}$,

and $[B]+[A]=\begin{bmatrix}3 & 2\\1 & 5\end{bmatrix}+\begin{bmatrix}6 & 5\\2 & 1\end{bmatrix}=\begin{bmatrix}9 & 7\\3 & 6\end{bmatrix}$, therefore, [A] + [B] = [B] + [A].

1.5 MULTIPLICATION OF A MATRIX BY SCALAR

A matrix is multiplied by a scalar, c, by multiplying each element of the matrix by this scalar. That is, the multiplication of a matrix $[A]$ by a scalar c is defined as

$$c[A]=\left[ca_{ij}\right].\tag{1.5}$$

The scalar multiplication is commutative.

For example,

Let $[A]=\begin{bmatrix} -3 & 1 \\ 4 & 2 \end{bmatrix}$, then $5[A]=\begin{bmatrix} -15 & 5 \\ 20 & 10 \end{bmatrix}$.

1.6 MULTIPLICATION OF A MATRIX BY ANOTHER MATRIX

The product of two matrices is $\mathbf{C}=\mathbf{AB}$, if and only if, the number of columns in $\mathbf{A}$ is equal to the number of rows in $\mathbf{B}$. The product of matrix $\mathbf{A}$ of size $m\times n$ and matrix $\mathbf{B}$ of size $n\times r$, the result in matrix $\mathbf{C}$ has size $m\times r$.

$$\text{That is, } [A]_{m\times n}[B]_{n\times r}=[C]_{m\times r}, \tag{1.6}$$

must be equal

$$\text{and } c_{ij}=\sum_{k=1}^{n} a_{ik}b_{kj}, \tag{1.7}$$

where the (ij)th component of matrix $\mathbf{C}$ is obtained by taking the dot product

$$c_{ij}=(i\text{th row of }\mathbf{A})\cdot(j\text{th column of }\mathbf{B}).$$

That is, to find the element in row i and column j of $[A][B]$, you need to single out row i from $[A]$ and column j from $[B]$, then multiply the corresponding elements from the row and column together and add up the resulting products.

For example,

let $[A]_{2\times 2}=\begin{bmatrix} 1 & 3 \\ 7 & 8 \end{bmatrix}$ and $[B]_{2\times 3}=\begin{bmatrix} 3 & 2 & 4 \\ -1 & 0 & 6 \end{bmatrix}$, then

$$[A][B] = \begin{bmatrix} 1 & 3 \\ 7 & 8 \end{bmatrix} \begin{bmatrix} 3 & 2 & 4 \\ -1 & 0 & 6 \end{bmatrix} = \begin{bmatrix} 1\times 3+3\times(-1) & 1\times 2+3\times 0 & 1\times 4+3\times 6 \\ 7\times 3+8\times(-1) & 7\times 2+8\times 0 & 7\times 4+8\times 6 \end{bmatrix}$$

$$= \begin{bmatrix} 0 & 2 & 22 \\ 13 & 14 & 76 \end{bmatrix}$$

Size of $[A][B] = 2\times 3$.

1.7 RULES OF MATRIX MULTIPLICATIONS

Matrix multiplication is associative; that is

$$\mathbf{ABC} = (\mathbf{AB})\mathbf{C} = \mathbf{A}(\mathbf{BC}). \tag{1.8}$$

Matrix multiplication is distributive; that is

$$\mathbf{A}(\mathbf{B}+\mathbf{C}) = \mathbf{AB}+\mathbf{AC} \tag{1.9}$$

or

$$(\mathbf{A}+\mathbf{B})\mathbf{C} = \mathbf{AC}+\mathbf{BC}. \tag{1.10}$$

Matrix multiplication is not commutative; that is

$$\mathbf{AB} \neq \mathbf{BA}. \tag{1.11}$$

A square matrix multiplied by its identity matrix is equal to the same matrix; that is

$$\mathbf{AI} = \mathbf{IA} = \mathbf{A}. \tag{1.12}$$

A square matrix can be raised to an integer power n; that is

$$\mathrm{An} = \overbrace{\mathrm{A\,A\ldots A}}^{n}. \tag{1.13}$$

A same square matrix multiplication with different integer power n and m can be given as

$$\mathbf{A}^n\mathbf{A}^m = \mathbf{A}^{n+m} \text{ and } \left(\mathbf{A}^n\right)^m = \mathbf{A}^{nm}. \tag{1.14}$$

Transpose of product of matrices rule is given as

$$(\mathbf{AB})^T = \left(\mathbf{B}^T\mathbf{A}^T\right), (\mathbf{ABC})^T = \mathbf{C}^T\mathbf{B}^T\mathbf{A}^T. \tag{1.15}$$

EXAMPLE 1.3

Given matrices

$$\{A\}=\begin{Bmatrix}2\\4\\1\\3\end{Bmatrix},\ [B]=\begin{bmatrix}6&1&2&-1\\4&-3&5&9\\8&-2&6&7\\0&7&-8&3\end{bmatrix},\ [C]=\begin{bmatrix}4&0&-3&2\\1&8&4&-4\\5&3&-2&6\\9&-1&0&7\end{bmatrix},\ [D]=\begin{bmatrix}-1&1&0\\2&3&-1\\4&0&5\end{bmatrix}$$

Find the following:

a. $[B]+[C]$

b. $[B]-[C]$

c. $5\{A\}$

d. $[B]\{A\}$

e. $[D]^2$

f. show that $[D][I]=[I][D]=[D]$

Solution:

a. $$[B]+[C]=\begin{bmatrix}6&1&2&-1\\4&-3&5&9\\8&-2&6&7\\0&7&-8&3\end{bmatrix}+\begin{bmatrix}4&0&-3&2\\1&8&4&-4\\5&3&-2&6\\9&-1&0&7\end{bmatrix}$$

$$=\begin{bmatrix}(6+4)&(1+0)&(2-3)&(-1+2)\\(4+1)&(-3+8)&(5+4)&(9-4)\\(8+5)&(-2+3)&(6-2)&(7+6)\\(0+9)&(7-1)&(-8+0)&(3+7)\end{bmatrix}=\begin{bmatrix}10&1&-1&1\\5&5&9&5\\13&1&4&13\\9&6&-8&10\end{bmatrix}$$

b.

$$[B]-[C]=\begin{bmatrix}6&1&2&-1\\4&-3&5&9\\8&-2&6&7\\0&7&-8&3\end{bmatrix}-\begin{bmatrix}4&0&-3&2\\1&8&4&-4\\5&3&-2&6\\9&-1&0&7\end{bmatrix}$$

$$=\begin{bmatrix}(6-4)&(1-0)&(2+3)&(-1-2)\\(4-1)&(-3-8)&(5-4)&(9+4)\\(8-5)&(-2-3)&(6+2)&(7-6)\\(0-9)&(7+1)&(-8-0)&(3-7)\end{bmatrix}=\begin{bmatrix}2&1&5&-3\\3&-11&1&13\\3&-5&8&1\\-9&8&-8&-4\end{bmatrix}$$

c. $5\{A\} = 5\begin{Bmatrix} 2 \\ 4 \\ 1 \\ 3 \end{Bmatrix} = \begin{Bmatrix} 10 \\ 20 \\ 5 \\ 15 \end{Bmatrix}$

d.

$$[B]\{A\} = \begin{bmatrix} 6 & 1 & 2 & -1 \\ 4 & -3 & 5 & 9 \\ 8 & -2 & 6 & 7 \\ 0 & 7 & -8 & 3 \end{bmatrix} \begin{Bmatrix} 2 \\ 4 \\ 1 \\ 3 \end{Bmatrix} = \begin{Bmatrix} (6\times 2)+(1\times 4)+(2\times 1)+(-1\times 3) \\ (4\times 2)+(-3\times 4)+(5\times 1)+(9\times 3) \\ (8\times 2)+(-2\times 4)+(6\times 1)+(7\times 3) \\ (0\times 2)+(7\times 4)+(-8\times 1)+(3\times 3) \end{Bmatrix} = \begin{Bmatrix} 13 \\ 28 \\ 35 \\ 29 \end{Bmatrix}$$

e.

$$[D]^2 = [D][D] = \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix} \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix} =$$

$$= \begin{bmatrix} (-1\times -1)+(1\times 2)+(0\times 4) & (-1\times 1)+(1\times 3)+(0\times 0) & (-1\times 0)+(1\times -1)+(0\times 5) \\ (2\times -1)+(3\times 2)+(-1\times 4) & (2\times 1)+(3\times 3)+(-1\times 0) & (2\times 0)+(3\times -1)+(-1\times 5) \\ (4\times -1)+(0\times 2)+(5\times 4) & (4\times 1)+(0\times 3)+(5\times 0) & (4\times 0)+(0\times -1)+(5\times 5) \end{bmatrix}$$

$$= \begin{bmatrix} 3 & 2 & -1 \\ 0 & 10 & -8 \\ 16 & 4 & 25 \end{bmatrix}$$

f.

$$[D][I] = \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$= \begin{bmatrix} (-1\times 1)+(1\times 0)+(0\times 0) & (-1\times 0)+(1\times 1)+(0\times 0) & (-1\times 0)+(1\times 0)+(0\times 1) \\ (2\times 1)+(3\times 0)+(-1\times 0) & (2\times 0)+(3\times 1)+(-1\times 0) & (2\times 0)+(3\times 0)+(-1\times 1) \\ (4\times 1)+(0\times 0)+(5\times 0) & (4\times 0)+(0\times 1)+(5\times 0) & (4\times 0)+(0\times 0)+(5\times 1) \end{bmatrix}$$

$$= \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix} = [D]$$

and

$$[I][D] == \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix}$$

$$= \begin{bmatrix} (1\times-1)+(0\times2)+(0\times4) & (1\times1)+(0\times3)+(0\times0) & (1\times0)+(0\times-1)+(0\times5) \\ (0\times-1)+(1\times2)+(0\times4) & (0\times1)+(1\times3)+(0\times0) & (0\times0)+(1\times-1)+(0\times5) \\ (0\times-1)+(0\times2)+(1\times4) & (0\times1)+(0\times3)+(1\times0) & (0\times0)+(0\times-1)+(1\times5) \end{bmatrix}$$

$$= \begin{bmatrix} -1 & 1 & 0 \\ 2 & 3 & -1 \\ 4 & 0 & 5 \end{bmatrix} = [D]$$

1.8 TRANSPOSE OF A MATRIX MULTIPLICATION

The *transpose* of a matrix $\mathbf{A} = \left[a_{ij}\right]$ is denoted as $\mathbf{A}^T = \left[a_{ji}\right]$. It is obtained by interchanging the rows and columns in matrix **A**. Thus, if a matrix $\mathbf{A}$ is of order $m \times n$, then $\mathbf{A}^T$ will be of order $n \times m$.

For example,

let $[A]_{2\times3} = \begin{bmatrix} 0 & 1 & 3 \\ -1 & 2 & 5 \end{bmatrix}$, then $[A]^T_{3\times2} = \begin{bmatrix} 0 & -1 \\ 1 & 2 \\ 3 & 5 \end{bmatrix}$.

Note that it is valid that, $(\mathbf{AB})^T = \mathbf{B}^T\mathbf{A}^T$, $(\mathbf{A}+\mathbf{B})^T = \mathbf{A}^T + \mathbf{B}^T$, $(c\mathbf{B})^T = c\mathbf{B}^T$, and $(\mathbf{A}^T)^T = \mathbf{A}$. Also note, if $\mathbf{A}^T = \mathbf{A}$, then $\mathbf{A}$ is a symmetric matrix.

EXAMPLE 1.4

Consider that matrix $[A] = \begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix}$ and $[B] = \begin{bmatrix} -1 & 0 & -3 \\ -4 & -2 & 5 \end{bmatrix}$.

Show that $\left([A][B]\right)^T = [B]^T [A]^T$.

Solution:

$$([A][B]) = \begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix} \begin{bmatrix} -1 & 0 & -3 \\ -4 & -2 & 5 \end{bmatrix}$$

$$= \begin{bmatrix} (1\times -1)+(2\times -4) & (1\times 0)+(2\times -2) & (1\times -3)+(2\times 5) \\ (3\times -1)+(4\times -4) & (3\times 0)+(4\times -2) & (3\times -3)+(4\times 5) \end{bmatrix}$$

$$= \begin{bmatrix} -9 & -4 & 7 \\ -19 & -8 & 11 \end{bmatrix}$$

$$([A][B])^T = \begin{bmatrix} -9 & -19 \\ -4 & -8 \\ 7 & 11 \end{bmatrix}$$

$$[A]^T = \begin{bmatrix} 1 & 3 \\ 2 & 4 \end{bmatrix}, \ [B]^T = \begin{bmatrix} -1 & -4 \\ 0 & -2 \\ -3 & 5 \end{bmatrix}$$

$$[B]^T [A]^T = \begin{bmatrix} -1 & -4 \\ 0 & -2 \\ -3 & 5 \end{bmatrix} \begin{bmatrix} 1 & 3 \\ 2 & 4 \end{bmatrix} = \begin{bmatrix} (-1\times 1)+(-4\times 2) & (-1\times 3)+(-4\times 4) \\ (0\times 1)+(-2\times 2) & (0\times 3)+(-2\times 4) \\ (-3\times 1)+(5\times 2) & (-3\times 3)+(5\times 4) \end{bmatrix}$$

$$= \begin{bmatrix} -9 & -19 \\ -4 & -8 \\ 7 & 11 \end{bmatrix}$$

Therefore, $([A][B])^T = [B]^T [A]^T$.

1.9 TRACE OF A MATRIX

A *trace* of a matrix $\mathbf{A}$, tr($\mathbf{A}$), is a square matrix and is defined to be the sum of the elements on the main diagonal of matrix $\mathbf{A}$.

For example, let, $[A] = \begin{bmatrix} 3 & 5 & 8 \\ 5 & 7 & 2 \\ 8 & 2 & -1 \end{bmatrix}$, then tr($\mathbf{A}$) = 3 + 7 + (-1) = 9.

EXAMPLE 1.5

Consider that matrix

$$[A] = \begin{bmatrix} 5 & 8 \\ 7 & 6 \end{bmatrix}.$$

Find the tr(A).

Solution:

tr(A) = 5 + 6 = 11.

1.10 DIFFERENTIATION OF A MATRIX

Differentiation of a matrix is the differentiation of every element of the matrix separately. For example, if the elements of the matrix **A** are a function of t , then

$$\frac{d\mathbf{A}}{dt} = \left[\frac{da_{ij}}{dt}\right]. \tag{1.16}$$

EXAMPLE 1.6

Consider the matrix $[A] = \begin{bmatrix} 3x^5 & x^2 \\ 7x & 6 \end{bmatrix}$, find the derivative $\dfrac{d[A]}{dx}$.

Solution:

$$\frac{d[A]}{dx} = \begin{bmatrix} 15x^4 & 2x \\ 7 & 0 \end{bmatrix}$$

1.11 INTEGRATION OF A MATRIX

Integration of a matrix is the integration of every element of the matrix separately. For example, if the elements of the matrix **A** are a function of t , then

$$\int \mathbf{A}\, dt = \left[\int a_{ij} dt\right]. \tag{1.17}$$

EXAMPLE 1.7

Consider the matrix $[A]=\begin{bmatrix} 4x^3 & 2 \\ 8x & 1 \end{bmatrix}$, find the derivative $\int[A]dx$.

Solution:

$$\int[A]dx=\begin{bmatrix} x^4 & 2x \\ 4x^2 & x \end{bmatrix}$$

1.12 EQUALITY OF MATRICES

Two matrices are equal if they have the same sizes and their corresponding elements are equal.

EXAMPLE 1.8

Let $\mathbf{A}=\begin{bmatrix} 1 & -4 \\ 5 & 3 \end{bmatrix}$ and $\mathbf{B}=\begin{bmatrix} 2x & w \\ z-2 & k+1 \end{bmatrix}$.

If the matrices **A** and **B** are equal, find the value of x, w, z, and k.

Solution:

$1 = 2x \longrightarrow x = ½$

$w = -4$

$z - 2 = 5 \longrightarrow z = 7$

$3 = k + 1 \longrightarrow k = 2$

1.13 DETERMINANT OF A MATRIX

The determinant of a square matrix **A** is a scalar number denoted by $|\mathbf{A}|$ or det $[A]$. It is the sum of the products $(-1)^{i+j}\,a_{ij}M_{ij}$, where a_{ij} are the elements along any one row or column and M_{ij} are the deleted elements of ith row and jth column from the matrix $[A]$.

For example, the value of the determinant of matrix $[A]$ is $\alpha = det[A] = |\mathbf{A}|$ and can be obtained by expanding along the first row as:

$$\alpha = \begin{vmatrix} a_{11} & a_{12} & a_{13} & \cdots & a_{1n} \\ a_{21} & a_{22} & a_{23} & \cdots & a_{2n} \\ a_{31} & a_{32} & a_{33} & \cdots & a_{3n} \\ \vdots & \vdots & \vdots & \cdots & \vdots \\ a_{n1} & a_{n2} & a_{n3} & \cdots & a_{nn} \end{vmatrix} \tag{1.18}$$

$$= a_{11}M_{11} - a_{12}M_{12} + a_{13}M_{13} + \ldots + (-1)^{n+1} a_{1n}M_{1n}$$

where the minor M_{ij} is a $(n-1)\times(n-1)$ determinant of the matrix formed by removing the i th row and j th column.

Also, the value α can be obtained by expanding along the first column as:

$$\alpha = a_{11}M_{11} - a_{21}M_{21} + a_{31}M_{31} + \ldots + (-1)^{n+1} a_{n1}M_{n1}\,. \tag{1.19}$$

Now, the value of a second-order determinant of (2×2) matrix is calculated by

$$\alpha = \det\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{vmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{vmatrix} = a_{11}a_{22} - a_{12}a_{21}\,. \tag{1.20}$$

The value of a third-order determinate of (3×3) matrix is calculated by

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

$$= a_{11}(-1)^2 \begin{vmatrix} a_{22} & a_{23} \\ a_{32} & a_{33} \end{vmatrix} + a_{12}(-1)^3 \begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} + a_{13}(-1)^4 \begin{vmatrix} a_{21} & a_{22} \\ a_{31} & a_{32} \end{vmatrix}$$

$$= a_{11} \begin{vmatrix} a_{22} & a_{23} \\ a_{32} & a_{33} \end{vmatrix} - a_{12} \begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} + a_{13} \begin{vmatrix} a_{21} & a_{22} \\ a_{31} & a_{32} \end{vmatrix}$$

$$= a_{11}(a_{22}a_{33} - a_{23}a_{32}) - a_{12}(a_{21}a_{33} - a_{23}a_{31}) + a_{13}(a_{21}a_{32} - a_{22}a_{31})\,. \tag{1.21}$$

EXAMPLE 1.9

Evaluate the following determinants:

a. $\begin{vmatrix} 2 & 3 \\ -1 & 4 \end{vmatrix}$

b. $\begin{vmatrix} 1 & 3 & 4 \\ -2 & -1 & 2 \\ 5 & -4 & 6 \end{vmatrix}$

Solution:

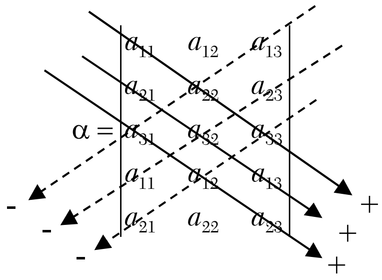
a. $\alpha = \begin{vmatrix} 2 & 3 \\ -1 & 4 \end{vmatrix} = (2 \times 4) - (3 \times -1) = 8 + 3 = 11$

b. $\alpha = \begin{vmatrix} 1 & 3 & 4 \\ -2 & -1 & 2 \\ 5 & -4 & 6 \end{vmatrix} = 1 \times \begin{vmatrix} -1 & 2 \\ -4 & 6 \end{vmatrix} - 3 \times \begin{vmatrix} -2 & 2 \\ 5 & 6 \end{vmatrix} + 4 \times \begin{vmatrix} -2 & -1 \\ 5 & -4 \end{vmatrix} =$

$= 1(-6 + 8) - 3(-12 - 10) + 4(8 + 5) = 2 + 66 + 52 = 120$

An alternative method of obtaining the determinant of a (3×3) matrix is by using the sign rule of each term that is determined by the first row in the diagram as follows:

$\begin{vmatrix} + & - & + \\ - & + & - \\ + & - & + \end{vmatrix}$, or by repeating the first two rows and multiplying the terms diagonally as follows:



$$= a_{11}a_{22}a_{33} + a_{21}a_{32}a_{13} + a_{31}a_{12}a_{23} - a_{13}a_{22}a_{31} - a_{23}a_{32}a_{11} - a_{33}a_{12}a_{21}\ .$$

1.14 DIRECT METHODS FOR LINEAR SYSTEMS

Many engineering problems in finite element analysis will result in a set of simultaneous equations represented by $[A]\{X\}=\{B\}$.

For a set of simultaneous equations having the form

$$\begin{array}{cccccc}
a_{11}x_1 & +a_{12}x_2 & +a_{13}x_3 & +\ldots+ & a_{1n}x_n & =b_1 \\
a_{21}x_1 & +a_{22}x_2 & +a_{23}x_3 & +\ldots+ & a_{2n}x_n & =b_2 \\
a_{31}x_1 & +a_{32}x_2 & +a_{33}x_3 & +\ldots+ & a_{3n}x_n & =b_3 \\
. & . & . & . & . & . \\
. & . & . & . & . & . \\
. & . & . & . & . & . \\
a_{n1}x_1 & +a_{n2}x_2 & +a_{n3}x_3 & +\ldots+ & a_{nn}x_n & =b_n
\end{array} \tag{1.22}$$

where there are n unknown $x_1, x_2, x_3, \ldots, x_n$ to be determined. These equations can be written in matrix form as

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & \ldots & a_{1n} \\ a_{21} & a_{22} & a_{23} & \ldots & a_{2n} \\ a_{31} & a_{32} & a_{33} & \ldots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \ldots & a_{nn} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ . \\ . \\ . \\ x_n \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ . \\ . \\ . \\ b_n \end{bmatrix}.$$

This matrix equation can be written in a compact form as

$$\mathbf{AX}=\mathbf{B}, \tag{1.23}$$

where $\mathbf{A}$ is a square matrix with order $n \times n$, while $\mathbf{X}$ and $\mathbf{B}$ are column matrices defined as

$$\mathbf{A} = \begin{bmatrix} a_{11} & a_{12} & a_{13} & \cdots & a_{1n} \\ a_{21} & a_{22} & a_{23} & \cdots & a_{2n} \\ a_{31} & a_{32} & a_{33} & \cdots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \cdots & a_{nn} \end{bmatrix}, \qquad \mathbf{X} = \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ . \\ . \\ . \\ x_n \end{bmatrix}, \qquad \mathbf{B} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ . \\ . \\ . \\ b_n \end{bmatrix}$$

There are several methods for solving a set of simultaneous equations such as by substitution, Gaussian elimination, Cramer's rule, matrix inversion, and numerical analysis.

1.15 GAUSSIAN ELIMINATION METHOD

In the *argument matrix* of a system, the variables of each equation must be on the left side of the equal sign (vertical line) and the constants on the right side. For example, the *argument matrix* of the system

$$2x_1 - 3x_2 = -5$$
$$x_1 - 4x_2 = 8$$

is $\left[\begin{array}{cc|c} 2 & -3 & -5 \\ 1 & -4 & 8 \end{array}\right]$.

The *argument matrix* is used in the Gaussian elimination method. The Gaussian elimination method is summarized by the following steps:

1. Write the system of equations in the *argument matrix* form.
2. Perform *elementary row operations* to get zeros below the main diagonal.
   - a. interchange any two rows
   - b. replace a row by a nonzero multiply of that row
   - c. replace a row by the sum of that row and a constant nonzero multiple of some other row
3. Use back substitution to find the solution of the system.

We demonstrate the Gaussian elimination method in Example 1.10.

EXAMPLE 1.10

Solve the linear system using the Gaussian elimination method.

$$x_2 + x_3 - 2 = 0$$
$$2x_1 + 3x_3 - 5 = 0$$
$$x_1 + x_2 + x_3 - 3 = 0$$

Solution:

We use R_i to represent the i th row. Write the argument matrix of the system as:

$$\left[\begin{array}{ccc|c} 0 & 1 & 1 & 2 \\ 2 & 0 & 3 & 5 \\ 1 & 1 & 1 & 3 \end{array}\right].$$

Interchange R_1 and R_2, this gives: $\left[\begin{array}{ccc|c} 2 & 0 & 3 & 5 \\ 0 & 1 & 1 & 2 \\ 1 & 1 & 1 & 3 \end{array}\right]$.

$\frac{1}{2}R_1$, this gives: $\left[\begin{array}{ccc|c} 1 & 0 & \frac{3}{2} & \frac{5}{2} \\ 0 & 1 & 1 & 2 \\ 1 & 1 & 1 & 3 \end{array}\right]$.

$-R_1 + R_3$, this gives: $\left[\begin{array}{ccc|c} 1 & 0 & \frac{3}{2} & \frac{5}{2} \\ 0 & 1 & 1 & 2 \\ 0 & 1 & -\frac{1}{2} & \frac{1}{2} \end{array}\right]$.

$-R_2 + R_3$, this gives: $\left[\begin{array}{ccc|c} 1 & 0 & \frac{3}{2} & \frac{5}{2} \\ 0 & 1 & 1 & 2 \\ 0 & 1 & -\frac{3}{2} & -\frac{3}{2} \end{array}\right]$.

$$-\frac{2}{3}R_3\text{, this gives: }\left[\begin{array}{ccc|c} 1 & 0 & \frac{3}{2} & \frac{5}{2} \\ 0 & 1 & 1 & 2 \\ 0 & 1 & 1 & 1 \end{array}\right].$$

R_3 gives $x_3 = 1$, substitute the value of x_3 in R_2 and R_3, this gives $x_2 = 1$, and $x_1 = 1$, respectively.

1.16 CRAMER'S RULE

Cramer's rule can be used to solve the simultaneous equations for $x_1, x_2, x_3, \ldots, x_n$ as

$$x_1 = \frac{\alpha_1}{\alpha}, x_2 = \frac{\alpha_2}{\alpha}, x_3 = \frac{\alpha_3}{\alpha}, \ldots, x_n = \frac{\alpha_n}{\alpha} \tag{1.24}$$

where α's are the determinations expressed as

$$\alpha = \begin{bmatrix} a_{11} & a_{12} & a_{13} & \ldots & a_{1n} \\ a_{21} & a_{22} & a_{23} & \ldots & a_{2n} \\ a_{31} & a_{32} & a_{33} & \ldots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \ldots & a_{nn} \end{bmatrix}, \alpha_1 = \begin{bmatrix} b_1 & a_{12} & a_{13} & \ldots & a_{1n} \\ b_2 & a_{22} & a_{23} & \ldots & a_{2n} \\ b_3 & a_{32} & a_{33} & \ldots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ b_n & a_{n2} & a_{n3} & \ldots & a_{nn} \end{bmatrix},$$

$$\alpha_2 = \begin{bmatrix} a_{11} & b_1 & a_{13} & \ldots & a_{1n} \\ a_{21} & b_2 & a_{23} & \ldots & a_{2n} \\ a_{31} & b_3 & a_{33} & \ldots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & b_n & a_{n3} & \ldots & a_{nn} \end{bmatrix}, \tag{1.25}$$

$$\alpha_3 = \begin{bmatrix} a_{11} & a_{12} & b_1 & \dots & a_{1n} \\ a_{21} & a_{22} & b_2 & \dots & a_{2n} \\ a_{31} & a_{32} & b_3 & \dots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & b_n & \dots & a_{nn} \end{bmatrix}, \dots, \alpha_n = \begin{bmatrix} a_{11} & a_{12} & a_{13} & \dots & b_1 \\ a_{21} & a_{22} & a_{23} & \dots & b_2 \\ a_{31} & a_{32} & a_{33} & \dots & b_3 \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \dots & b_n \end{bmatrix}.$$

It is worth noting that α is the determinant of matrix $\mathbf{A}$ and α_n is the determinant of the matrix formed by replacing the *n*th column of $\mathbf{A}$ by $\mathbf{B}$. Also, Cramer's rule applies only when $\alpha \neq 0$, but when $\alpha = 0$, the set of questions has no unique solution because the equations are linearly dependent.

Summary of Cramer's Rule

1. Form the coefficient matrix of **A** and column matrix **B**.
2. Compute the determinant of matrix of **A**. If det[A] = 0, then the system has no solution; otherwise, go to the next step.
3. Compute the determinant of the new matrix $[A_i]$, by replacing the ith matrix with the column vector **B**.
4. Repeat Step 3 for $i = 1, 2, \dots, n$.
5. Solve for the unknown variable $\mathbf{X}_i$ using

$$\mathbf{X}_i = \frac{|A_i|}{|A|}, \text{ for } i = 1, 2, \dots, n. \tag{1.26}$$

EXAMPLE 1.11

Solve the simultaneous equations

$$2x_1 - 5x_2 = 13, \quad 5x_1 + 3x_2 = -14$$

Solution:

The matrix form of the given equations is

$$\begin{bmatrix} 2 & -5 \\ 5 & 3 \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = \begin{bmatrix} 13 \\ -14 \end{bmatrix}.$$

The determinants are calculated as

$$\alpha = \begin{vmatrix} 2 & -5 \\ 5 & 3 \end{vmatrix} = (2\times 3)-(-5\times 5) = 6+25 = 31$$

$$\alpha_1 = \begin{vmatrix} 13 & -5 \\ -14 & 3 \end{vmatrix} = (13\times 3)-(-5\times -14) = 39-70 = -31$$

$$\alpha_2 = \begin{vmatrix} 2 & 13 \\ 5 & -14 \end{vmatrix} = (2\times -14)-(13\times 5) = -28-65 = -93$$

Thus,

$$x_1 = \frac{\alpha_1}{\alpha} = \frac{-31}{31} = -1, \quad x_2 = \frac{\alpha_2}{\alpha} = \frac{-93}{31} = -3$$

EXAMPLE 1.12

Solve the simultaneous equations

$$10x_1 - 3x_2 - 4x_3 = 15, \quad 2x_1 + 5x_2 - 2x_3 = 0, \quad -2x_1 + x_2 + 6x_3 = 0\,,$$

Solution:

In matrix form, the given set of equations becomes

$$\begin{bmatrix} 10 & -3 & -4 \\ 2 & 5 & -2 \\ -2 & 1 & 6 \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} 15 \\ 0 \\ 0 \end{bmatrix}.$$

The determinants are calculated as

$$\alpha = \begin{vmatrix} 10 & -3 & -4 \\ 2 & 5 & -2 \\ -2 & 1 & 6 \end{vmatrix} = 10\left[(5\times 6)-(-2\times 1)\right]-(-3)\left[(2\times 6)-(-2\times -2)\right]$$

$$+(-4)\left[(2\times 1)-(5\times -2)\right]$$

$$= 320 + 24 - 48 = 296$$

$$\alpha_1 = \begin{vmatrix} 15 & -3 & -4 \\ 0 & 5 & -2 \\ 0 & 1 & 6 \end{vmatrix} = 15\left[(5\times 6)-(-2\times 1)\right]-(-3)\left[(0\times 6)-(-2\times 0)\right]$$

$$+(-4)\left[(0\times 1)-(5\times 0)\right]$$

$$= 480 + 0 - 0 = 480$$

$$\alpha_2 = \begin{vmatrix} 10 & 15 & -4 \\ 2 & 0 & -2 \\ -2 & 0 & 6 \end{vmatrix} = 10\left[(0\times 6)-(-2\times 0)\right]-(15)\left[(2\times 6)-(-2\times -2)\right]$$

$$+(-4)\left[(2\times 0)-(0\times -2)\right]$$

$$= 0 - 120 - 0 = -120$$

$$\alpha_3 = \begin{vmatrix} 10 & -3 & 15 \\ 2 & 5 & 0 \\ -2 & 1 & 0 \end{vmatrix} = 10\left[(5\times 0)-(0\times 1)\right]-(-3)\left[(2\times 0)-(0\times -2)\right]$$

$$+(15)\left[(2\times 1)-(5\times -2)\right]$$

$$= 0 - 0 + 180 = 180$$

Thus,

$$x_1 = \frac{\alpha_1}{\alpha} = \frac{480}{296} = 1.62, \quad x_2 = \frac{\alpha_2}{\alpha} = \frac{-120}{296} = -0.41, x_3 = \frac{\alpha_3}{\alpha} = \frac{180}{296} = 0.61$$

1.17 INVERSE OF A MATRIX

Matrix inversion is used in many applications, including the linear system of equations.

For the matrix equation $\mathbf{AX} = \mathbf{B}$, we can invert $\mathbf{A}$ to obtain $\mathbf{X}$, that is,

$$\mathbf{X} = \mathbf{A}^{-1}\mathbf{B} \tag{1.27}$$

where $\mathbf{A}^{-1}$ is the inverse matrix of **A**. The inverse matrix satisfies

$$\mathbf{A}\mathbf{A}^{-1} = \mathbf{A}^{-1}\mathbf{A} = \mathbf{I} \tag{1.28}$$

where

$$\mathbf{A}^{-1} = \frac{\text{Adj}\,[A]}{|\mathbf{A}|}, \tag{1.29}$$

where $\text{Adj}\,[A]$ is the adjoint of matrix **A.** The $\text{Adj}\,[A]$ is the transpose of the cofactors of matrix **A**. For example, let the $n \times n$ matrix **A** be presented as

$$\mathbf{A} = \begin{bmatrix} a_{11} & a_{12} & \dots & a_{1n} \\ a_{21} & a_{22} & \dots & a_{2n} \\ \vdots & \vdots & \vdots & \vdots \\ a_{n1} & a_{n2} & \dots & a_{nn} \end{bmatrix}.$$

The cofactors of matrix **A** are written in matrix **F** as

$$\mathbf{F} = \text{cof}\,[\text{A}] = \begin{bmatrix} f_{11} & f_{12} & \cdots & f_{1n} \\ f_{21} & f_{22} & \cdots & f_{2n} \\ \vdots & \vdots & \vdots & \vdots \\ f_{n1} & f_{n2} & \cdots & f_{nn} \end{bmatrix} \tag{1.30}$$

where f_{ij} is the product of $(-1)^{i+j}$ and the determinant of the $(n-1)\times(n-1)$ submatrix is obtained by removing the i th row and j th column from matrix **A**. For instance, by removing the first row and the first column of matrix **A**, we find the cofactor f_{11} as

$$(-1)^2 f_{11} = \begin{vmatrix} a_{22} & a_{23} & \dots & a_{2n} \\ a_{32} & a_{33} & \dots & a_{3n} \\ \vdots & \vdots & \vdots & \vdots \\ a_{n2} & a_{n3} & \dots & a_{nn} \end{vmatrix}. \tag{1.31}$$

Now the adjoint of matrix **A** can be obtained as

$$\text{Adj } [A] = [F]^T = \begin{bmatrix} f_{11} & f_{12} & \cdots & f_{1n} \\ f_{21} & f_{22} & \cdots & f_{2n} \\ \vdots & \vdots & \vdots & \vdots \\ f_{n1} & f_{n2} & \cdots & f_{nn} \end{bmatrix}^T . \tag{1.32}$$

So, the inverse of **A** matrix can be written as

$$\mathbf{A}^{-1} = \frac{[F]^T}{|\mathbf{A}|}. \tag{1.33}$$

A matrix that possesses an inverse is called *invertible matrix (nonsingular matrix)*. A matrix without an inverse is called a *noninvertible matrix (singular matrix)*.

Consider a 2×2 matrix, if

$\mathbf{A} = \begin{bmatrix} a & b \\ c & d \end{bmatrix}$, and $ad - bc \neq 0$, then

$$\mathbf{A}^{-1} = \frac{1}{|\mathbf{A}|}\begin{bmatrix} d & -b \\ -c & a \end{bmatrix} = \frac{1}{ad-bc}\begin{bmatrix} d & -b \\ -c & a \end{bmatrix} = \begin{bmatrix} \dfrac{d}{ad-bc} & \dfrac{-b}{ad-bc} \\ \dfrac{-c}{ad-bc} & \dfrac{a}{ad-bc} \end{bmatrix}. \tag{1.34}$$

The inverse of product of matrices rule can be presented as

$$(\mathbf{AB})^{-1} = \left(\mathbf{B}^{-1}\mathbf{A}^{-1}\right), (\mathbf{ABC})^{-1} = \mathbf{C}^{-1}\mathbf{B}^{-1}\mathbf{A}^{-1}. \tag{1.35}$$

EXAMPLE 1.13

Let matrix, $\mathbf{A} = \begin{bmatrix} 3 & 2 \\ 2 & 1 \end{bmatrix}$, find its inverse matrix, **A**-1.

Solution:

$$\mathbf{A}^{-1} = \frac{1}{|\mathbf{A}|}\begin{bmatrix} 1 & -2 \\ -2 & 3 \end{bmatrix} = \frac{1}{3-4}\begin{bmatrix} 1 & -2 \\ -2 & 3 \end{bmatrix} = \begin{bmatrix} -1 & 2 \\ 2 & -3 \end{bmatrix}$$

EXAMPLE 1.14

Let matrix, $\mathbf{A} = \begin{bmatrix} 2 & 1 \\ -1 & 1 \end{bmatrix}$, find its inverse matrix, **A**-1.

Solution:

Using the concept of equation (1.27), we get

$$\mathbf{A}^{-1} = \begin{bmatrix} a & b \\ c & d \end{bmatrix}, \text{then } \mathbf{AA}^{-1} = \begin{bmatrix} 2 & 1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} a & b \\ c & d \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

$2a + c = 1 \longrightarrow 3a = 1 \longrightarrow a = 1/3$

$2b + d = 0 \longrightarrow 3b = -1 \longrightarrow b = -1/3$

$-a + c = 0 \longrightarrow a = c = 1/3$

$-b + d = 1 \longrightarrow d = 1 + b = 2/3$

Therefore,

$$\mathbf{A}^{-1} = \begin{bmatrix} 1/3 & -1/3 \\ 1/3 & 2/3 \end{bmatrix}.$$

1.18 VECTOR ANALYSIS

A *vector* is a special case of a matrix with just one row or one column. A vector is a quantity (mathematical or physical) that has both magnitude and direction. Examples of vectors are force, momentum, acceleration, velocity, electric field intensity, and displacement. A *scalar* is a quantity that has only magnitude. Examples of scalars are mass, time, length, volume, distance, temperature, and electric potential.

A vector **A** has both magnitude and direction. A vector **A** in Cartesian (rectangular) coordinates can be written as $\left(A_x, A_y, A_z\right)$ or $\begin{bmatrix} A_x \\ A_y \\ A_z \end{bmatrix}$ where $A_x, A_y,$ and A_z are components of vector **A** in the $x, y,$ and z directions, respectively. The magnitude of vector **A** is a scalar written as $|\mathbf{A}|$ or A and given as

$$|\mathbf{A}| = \sqrt{A_x^2 + A_y^2 + A_z^2}\ . \tag{1.36}$$

A unit vector $\mathbf{a}_A$ along vector **A** is defined as a vector whose magnitude is unity (i.e., 1) and its direction is along vector **A**, that is,

$$\mathbf{A} = |\mathbf{A}|\mathbf{a}_A\ , \tag{1.37}$$

thus, $\left(A_x, A_y, A_z\right) = A_x\mathbf{a}_x + A_y\mathbf{a}_y + A_z\mathbf{a}_z$ (1.38)

and $$\mathbf{a}_A = \frac{\mathbf{A}}{|\mathbf{A}|} = \frac{A_x\mathbf{a}_x + A_y\mathbf{a}_y + A_z\mathbf{a}_z}{\sqrt{A_x^2 + A_y^2 + A_z^2}}\ . \tag{1.39}$$

Figure 1.1(a) illustrates the components of vector **A,** and Figure 1.1(b) shows the unit vectors.

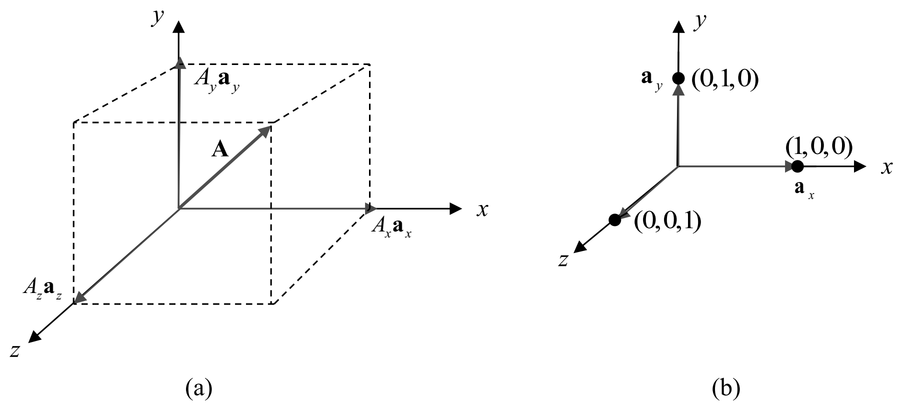


FIGURE 1.1 (a) Components of vector **A**. (b) Unit vectors.

(a) Vectors equality

Two vectors are equal if they are the same type (row or column) and their corresponding elements are equal to each other.

(b) Vector addition and subtraction

Two vectors can be added or subtracted only if they are of the same type (i.e., both row vectors or both column vectors) and they are of the same number of components (elements).

Two vectors $\mathbf{A} = (A_x, A_y, A_z)$ and $\mathbf{B} = (B_x, B_y, B_z)$ can be added together to give another vector **C**, that is,

$$\mathbf{C} = \mathbf{A} + \mathbf{B} \tag{1.40}$$

$$\mathbf{C} = (A_x + B_x)\mathbf{a}_x + (A_y + B_y)\mathbf{a}_y + (A_z + B_z)\mathbf{a}_z . \tag{1.41}$$

Vector subtraction is similarly presented as

$$\mathbf{D} = \mathbf{A} - \mathbf{B} = \mathbf{A} + (-\mathbf{B}) \tag{1.42}$$

$$\mathbf{D} = (A_x - B_x)\mathbf{a}_x + (A_y - B_y)\mathbf{a}_y + (A_z - B_z)\mathbf{a}_z . \tag{1.43}$$

(c) Multiplication of a scalar by a vector

When a vector is multiplied by a scalar, each element is manipulated by the scalar. Let, vector $\mathbf{A} = (A_x, A_y, A_z)$ and scalar k, then

$$k\mathbf{A} = (kA_x, kA_y, kA_z) . \tag{1.44}$$

There are three basic laws of algebra for given vectors **A**, **B**, and **C** when k and l are scalars, summarized in Table 1.1.

TABLE 1.1 Three basic laws of vector algebra.

Law	Addition	Multiplication
Commutative	**A** + **B** = **B** + **A**	k **A** = **A** k
Associative	**A** + (**B** + **C**) = (**A** + **B**) + **C**	k (l **A**) = (k l) **A**
Distributive	K(**A** + **B**) = k**B** + k**A**	

(d) Vector multiplication

There are two types of vector multiplication:

1. Scalar (dot) product, $\mathbf{A} \cdot \mathbf{B}$
2. Vector (cross) product, $\mathbf{A} \times \mathbf{B}$

1. The dot product of two vectors $\mathbf{A} = (A_x, A_y, A_z)$ and $\mathbf{B} = (B_x, B_y, B_z)$ written as $\mathbf{A} \cdot \mathbf{B}$ is defined as

$$\mathbf{A} \cdot \mathbf{B} = |\mathbf{A}||\mathbf{B}|\cos\theta_{AB}, \tag{1.45}$$

where θ_{AB} is the smallest angle between vectors **A** and **B**. Also, the dot product is defined as,

$$\mathbf{A} \cdot \mathbf{B} = A_x B_x + A_y B_y + A_z B_z. \tag{1.46}$$

It is worth it to know that two vectors **A** and **B** are perpendicular (orthogonal) if and only if $\mathbf{A} \cdot \mathbf{B} = 0$. Also, two vectors **A** and **B** are parallel if and only if $\mathbf{B} = k\mathbf{A}$.

For vectors **A**, **B**, **C** and k scalar, the following prosperities dot product hold:

(a) $\mathbf{A} \cdot \mathbf{B} = \mathbf{B} \cdot \mathbf{A}$ (1.47)

(b) $\mathbf{A} \cdot (\mathbf{B} + \mathbf{C}) = \mathbf{A} \cdot \mathbf{B} + \mathbf{A} \cdot \mathbf{C}$ (1.48)

(c) $\mathbf{A} \cdot \mathbf{A} = |\mathbf{A}|^2 = A^2$ (1.49)

(d) $k(\mathbf{A} \cdot \mathbf{B}) = (k\mathbf{A}) \cdot \mathbf{B} = \mathbf{A} \cdot (k\mathbf{B})$ (1.50)

(e)
$$\begin{aligned} \mathbf{a}_x \cdot \mathbf{a}_y &= \mathbf{a}_y \cdot \mathbf{a}_z = \mathbf{a}_z \cdot \mathbf{a}_x = 0 \\ \mathbf{a}_x \cdot \mathbf{a}_x &= \mathbf{a}_y \cdot \mathbf{a}_y = \mathbf{a}_z \cdot \mathbf{a}_z = 1 \end{aligned} \tag{1.51}$$

2. The cross product of two vectors $\mathbf{A} = (A_x, A_y, A_z)$ and $\mathbf{B} = (B_x, B_y, B_z)$ written as $\mathbf{A} \times \mathbf{B}$, is defined as

$$\mathbf{A} \times \mathbf{B} = |\mathbf{A}||\mathbf{B}|\sin\theta_{AB}\mathbf{a}_n, \tag{1.52}$$

where $\mathbf{a}_n$ is a unit vector normal to the plane containing vectors **A** and **B**. The direction of $\mathbf{a}_n$ is taken as the direction of the right thumb when the fingers of the right-hand rotate from vector **A** to vector **B** as shown in Figure 1.2.

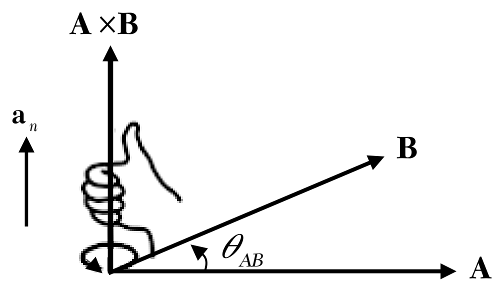


FIGURE 1.2 Right-hand rule for the direction of $\mathbf{A} \times \mathbf{B}$ and $\mathbf{a}_n$.

Also, the cross-product is defined as,

$$\mathbf{A} \times \mathbf{B} = \begin{vmatrix} \mathbf{a}_x & \mathbf{a}_y & \mathbf{a}_z \\ A_x & A_y & A_z \\ B_x & B_y & B_z \end{vmatrix} = \left(A_y B_z - A_z B_y\right)\mathbf{a}_x - \left(A_x B_z - A_z B_x\right)\mathbf{a}_y + \left(A_x B_y - A_y B_x\right)\mathbf{a}_z. \tag{1.53}$$

Because of the direction requirement of the cross-product, the commutative law does not apply to the cross-product. Instead,

$$\mathbf{A} \times \mathbf{B} = -\mathbf{B} \times \mathbf{A}. \tag{1.54}$$

EXAMPLE 1.15

Given $\mathbf{A} = (3,-2,5)$ and $\mathbf{B} = (2,4,-6)$, find:

a. $\mathbf{A} + \mathbf{B}$

b. $\mathbf{A} - \mathbf{B}$

c. $|\mathbf{A}|$

d. $3\mathbf{A} - \mathbf{B}$

e. $|\mathbf{A} + \mathbf{B}|$

f. The component of $\mathbf{B}$ along $\mathbf{a}_y$

Solution:

a. $\mathbf{A}+\mathbf{B}=(3+2)\mathbf{a}_x+(-2+4)\mathbf{a}_y+(5-6)\mathbf{a}_z$

$\mathbf{A}+\mathbf{B}=5\mathbf{a}_x+2\mathbf{a}_y-\mathbf{a}_z=(5,2,-1)$

b. $\mathbf{A}-\mathbf{B}=(3-2)\mathbf{a}_x+(-2-4)\mathbf{a}_y+(5+6)\mathbf{a}_z$

$\mathbf{A}-\mathbf{B}=\mathbf{a}_x-6\mathbf{a}_y+11\mathbf{a}_z=(1,-6,11)$

c. $|\mathbf{A}|=\mathbf{A}-\mathbf{B}=\sqrt{(3-2)^2+(-2-4)^2+(5+6)^2}$

$\mathbf{A}-\mathbf{B}=\sqrt{1+36+121}=\sqrt{158}=12.57$

d. $3\mathbf{A}-\mathbf{B}=(9-2)\mathbf{a}_x+(-6-4)\mathbf{a}_y+(15+6)\mathbf{a}_z$

$3\mathbf{A}-\mathbf{B}=5\mathbf{a}_x-10\mathbf{a}_y+21\mathbf{a}_z=(5,-10,21)$

e. $|\mathbf{A}+\mathbf{B}|=\sqrt{(5)^2+(2)^2+(-1)^2}=\sqrt{25+4+1}=\sqrt{30}$

f. The component of $\mathbf{B}$ along $\mathbf{a}_y$ is $B_y=4$

EXAMPLE 1.16

Given $\mathbf{A}=3\mathbf{a}_x+2\mathbf{a}_y-\mathbf{a}_z$ and $\mathbf{B}=\mathbf{a}_x+\mathbf{a}_y$, find:

a. $\mathbf{A}\cdot\mathbf{B}$

b. $\mathbf{A}\times\mathbf{B}$

c. The angle between $\mathbf{A}$ and $\mathbf{B}$

Solution:

a. $\mathbf{A}\cdot\mathbf{B}=(3)(1)+(2)(1)+(-1)(0)=5$

b. $$\mathbf{A}\times\mathbf{B}=\begin{vmatrix}\mathbf{a}_x & \mathbf{a}_y & \mathbf{a}_z\\ 3 & 2 & -1\\ 1 & 1 & 0\end{vmatrix}=(2\times0-(-1)\times1)\mathbf{a}_x-(3\times0-(-1)\times1)\mathbf{a}_y$$

$$+(3\times1-2\times1)\mathbf{a}_z$$

$\mathbf{A}\times\mathbf{B}=\mathbf{a}_x+\mathbf{a}_y+\mathbf{a}_z$

c. $\mathbf{A}\cdot\mathbf{B} = |\mathbf{A}||\mathbf{B}|\cos\theta_{AB} \Rightarrow \cos\theta_{AB} = \dfrac{\mathbf{A}\cdot\mathbf{B}}{|\mathbf{A}||\mathbf{B}|}$

$$|\mathbf{A}| = \sqrt{(3)^2 + (2)^2 + (-1)^2} = \sqrt{9+4+1} = \sqrt{14}$$

$$|\mathbf{B}| = \sqrt{(1)^2 + (1)^2 + (0)^2} = \sqrt{1+1+0} = \sqrt{2}$$

$$\cos\theta_{AB} = \frac{5}{\sqrt{14}\sqrt{2}} = \frac{5}{\sqrt{28}} = \frac{5}{2\sqrt{7}} = 0.9449$$

$$\theta_{AB} = \cos^{-1}\left(\frac{5}{2\sqrt{7}}\right) = 19.11^\circ$$

(e) The Del (∇) operator

The *Del* (∇) operator is a vector differential operator and is known as a *gradient* operator.

We obtain ∇ in Cartesian coordinates (x, y, z) as,

$$\nabla = \mathbf{a}_x \frac{\partial}{\partial x} + \mathbf{a}_y \frac{\partial}{\partial y} + \mathbf{a}_z \frac{\partial}{\partial z}. \tag{1.55}$$

We obtain ∇ in cylindrical coordinates (ρ, ϕ, z) as,

$$\nabla = \mathbf{a}_\rho \frac{\partial}{\partial \rho} + \mathbf{a}_\phi \frac{1}{\rho}\frac{\partial}{\partial \phi} + \mathbf{a}_z \frac{\partial}{\partial z}. \tag{1.56}$$

We obtain ∇ in spherical coordinates (r, θ, ϕ) as,

$$\nabla = \mathbf{a}_r \frac{\partial}{\partial r} + \mathbf{a}_\theta \frac{1}{r}\frac{\partial}{\partial \theta} + \mathbf{a}_\phi \frac{1}{r\sin\theta}\frac{\partial}{\partial \phi}. \tag{1.57}$$

The Del (∇) operator is useful in defining the following operations on a scalar or a vector:

1. ∇A is the *gradient* of a scalar A (the result of this operation is a vector)

(a) For Cartesian coordinates, $\nabla A = \mathbf{a}_x \dfrac{\partial A}{\partial x} + \mathbf{a}_y \dfrac{\partial A}{\partial y} + \mathbf{a}_z \dfrac{\partial A}{\partial z}$. (1.58)

(b) For cylindrical coordinates, $\nabla A = \mathbf{a}_\rho \dfrac{\partial A}{\partial \rho} + \mathbf{a}_\phi \dfrac{1}{\rho}\dfrac{\partial A}{\partial \phi} + \mathbf{a}_z \dfrac{\partial A}{\partial z}$. (1.59)

(c) For spherical coordinates, $\nabla A = \mathbf{a}_r \dfrac{\partial A}{\partial r} + \mathbf{a}_\theta \dfrac{1}{r}\dfrac{\partial A}{\partial \theta} + \mathbf{a}_\phi \dfrac{1}{r\sin\theta}\dfrac{\partial A}{\partial \phi}$. (1.60)

Considering A and B are scalars and n is an integer, the following formulas are true on a gradient:

- $\nabla(A+B) = \nabla A + \nabla B$ (1.61)
- $\nabla(AB) = A\nabla B + B\nabla A$ (1.62)
- $\nabla\left(\dfrac{A}{B}\right) = \dfrac{B\nabla A - A\nabla B}{B^2}$ (1.63)
- $\nabla A^n = nA^{n-1}\nabla A$ (1.64)

2. $\nabla \cdot \mathbf{A}$ is the *divergence* of a vector $\mathbf{A}$ (the result of this operation is a scalar)

(a) For Cartesian coordinates, $\nabla \cdot \mathbf{A} = \dfrac{\partial A_x}{\partial x} + \dfrac{\partial A_y}{\partial y} + \dfrac{\partial A_z}{\partial z}$. (1.65)

(b) For cylindrical coordinates, $\nabla \cdot \mathbf{A} = \dfrac{1}{\rho}\dfrac{\partial}{\partial \rho}\left(\rho A_\rho\right) + \dfrac{1}{\rho}\dfrac{\partial A_\phi}{\partial \phi} + \dfrac{\partial A_z}{\partial z}$. (1.66)

(c) For spherical coordinates,

$$\nabla \cdot \mathbf{A} = \frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2 A_r\right) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \theta}\left(A_\theta \sin\theta\right) + \frac{1}{r\sin\theta}\frac{\partial A_\phi}{\partial \phi}. \quad (1.67)$$

Considering $\mathbf{A}$ and $\mathbf{B}$ are vectors and k is a scalar, the following formulas are true on divergence of a vector:

- $\nabla \cdot (\mathbf{A} + \mathbf{B}) = \nabla \cdot \mathbf{A} + \nabla \cdot \mathbf{B}$ (1.68)
- $\nabla \cdot (k\mathbf{A}) = k\nabla \cdot \mathbf{A} + \mathbf{A} \cdot \nabla k$ (1.69)

3. $\nabla \times \mathbf{A}$ is the *curl* of a vector $\mathbf{A}$ (the result of this operation is a vector)

(a) For Cartesian coordinates, $\nabla \times \mathbf{A} = \begin{vmatrix} \mathbf{a}_x & \mathbf{a}_y & \mathbf{a}_z \\ \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ A_x & A_y & A_z \end{vmatrix}$ (1.70)

or

$$\nabla \times \mathbf{A} = \left(\frac{\partial A_z}{\partial y} - \frac{\partial A_y}{\partial z}\right)\mathbf{a}_x + \left(\frac{\partial A_x}{\partial z} - \frac{\partial A_z}{\partial x}\right)\mathbf{a}_y + \left(\frac{\partial A_y}{\partial x} - \frac{\partial A_x}{\partial y}\right)\mathbf{a}_z . \tag{1.71}$$

(b) For cylindrical coordinates, $\nabla \times \mathbf{A} = \frac{1}{\rho}\begin{vmatrix} \mathbf{a}_\rho & \rho\mathbf{a}_\phi & \mathbf{a}_z \\ \frac{\partial}{\partial \rho} & \frac{\partial}{\partial \phi} & \frac{\partial}{\partial z} \\ A_\rho & \rho A_\phi & A_z \end{vmatrix}$ (1.72)

or

$$\nabla \times \mathbf{A} = \left(\frac{1}{\rho}\frac{\partial A_z}{\partial \phi} - \frac{\partial A_\phi}{\partial z}\right)\mathbf{a}_\rho + \left(\frac{\partial A_\rho}{\partial z} - \frac{\partial A_z}{\partial \rho}\right)\mathbf{a}_\phi + \frac{1}{\rho}\left(\frac{\partial(\rho A_\phi)}{\partial \rho} - \frac{\partial A_\rho}{\partial \phi}\right)\mathbf{a}_z . \tag{1.73}$$

(c) For spherical coordinates, $\nabla \times \mathbf{A} = \frac{1}{r^2 \sin\theta}\begin{vmatrix} \mathbf{a}_r & r\mathbf{a}_\theta & r\sin\theta\,\mathbf{a}_\phi \\ \frac{\partial}{\partial r} & \frac{\partial}{\partial \theta} & \frac{\partial}{\partial \phi} \\ A_r & rA_\theta & r\sin\theta A_\phi \end{vmatrix}$ (1.74)

or

$$\nabla \times \mathbf{A} = \frac{1}{r\sin\theta}\left(\frac{\partial(A_\phi \sin\theta)}{\partial \theta} - \frac{\partial A_\theta}{\partial \phi}\right)\mathbf{a}_r + \frac{1}{r}\left(\frac{1}{\sin\theta}\frac{\partial A_r}{\partial \phi} - \frac{\partial(rA_\phi)}{\partial r}\right)\mathbf{a}_\theta + \frac{1}{r}\left(\frac{\partial(rA_\theta)}{\partial r} - \frac{\partial A_r}{\partial \theta}\right)\mathbf{a}_\phi . \tag{1.75}$$

Considering **A** and **B** are vectors and k is a scalar, the following formulas are true on curl of a vector:

- $\nabla \times (\mathbf{A} + \mathbf{B}) = \nabla \times \mathbf{A} + \nabla \times \mathbf{B}$ (1.76)
- $\nabla \times (\mathbf{A} \times \mathbf{B}) = \mathbf{A}(\nabla \cdot \mathbf{B}) - \mathbf{B}(\nabla \cdot \mathbf{A}) + (\mathbf{B} \cdot \nabla\mathbf{A}) - (\mathbf{A} \cdot \nabla)\mathbf{B}$ (1.77)
- $\nabla \times (k\mathbf{A}) = k\nabla \times \mathbf{A} + \nabla k \times \mathbf{A}$ (1.78)
- $\nabla \cdot (\nabla \times \mathbf{A}) = 0$ (1.79)
- $\nabla \times \nabla k = 0$ (1.80)

4. $\nabla^2 A$ **-**Laplacian of a scalar A (the result of this operation is a scalar)

(a) For Cartesian coordinates, $\nabla^2 A = \dfrac{\partial^2 A}{\partial x^2} + \dfrac{\partial^2 A}{\partial y^2} + \dfrac{\partial^2 A}{\partial z^2}$. (1.81)

(b) For cylindrical coordinates, $\nabla^2 A = \dfrac{1}{\rho}\dfrac{\partial}{\partial \rho}\left(\rho \dfrac{\partial A}{\partial \rho}\right) + \dfrac{1}{\rho^2}\dfrac{\partial^2 A}{\partial \phi^2} + \dfrac{\partial^2 A}{\partial z^2}$. (1.82)

(c) For spherical coordinates,

$$\nabla^2 A = \frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2 \frac{\partial A}{\partial r}\right) + \frac{1}{r^2 \sin\theta}\frac{\partial}{\partial \theta}\left(\sin\theta \frac{\partial A}{\partial \theta}\right) + \frac{1}{r^2 \sin^2\theta}\frac{\partial^2 A}{\partial \phi^2}. \tag{1.83}$$

The Laplacian of a vector **A**, can be defined as

$$\nabla^2 \mathbf{A} = \nabla(\nabla \cdot \mathbf{A}) - \nabla \times \nabla \times \mathbf{A}. \tag{1.84}$$

EXAMPLE 1.17

Find the gradient of the scalar field $A = e^{-z} \sin 3x \cosh y$.

Solution:

$$\nabla A = 3e^{-z} \cos 3x \cosh y \mathbf{a}_x + e^{-z} \sin 3x \sinh y \mathbf{a}_y - e^{-z} \sin 3x \cosh y \mathbf{a}_z$$

EXAMPLE 1.18

Find the divergence (∇○**A**) of the vector field $\mathbf{A} = xyz^2 \mathbf{a}_x + yz\mathbf{a}_z$.

Solution:

$$\nabla \cdot \mathbf{A} = \frac{\partial A_x}{\partial x} + \frac{\partial A_y}{\partial y} + \frac{\partial A_z}{\partial z}$$

$$\nabla \cdot \mathbf{A} = yz^2 + y = y(z^2 + 1)$$

1.19 EIGENVALUES AND EIGENVECTORS

Eigenvalues problems arise from many branches of engineering, especially in the analysis of the vibration of elastic structures and electrical systems.

The eigenvalue problem is presented in linear equations in the form

$$[A]\cdot\{X\}-\lambda\{X\}=\{0\}. \tag{1.85}$$

where $[A]$ is a square matrix; λ is a scalar and called eigenvalue of matrix $[A]$; $\{X\}$ is eigenvector of matrix $[A]$ corresponding to λ.

To find the eigenvalues of a square matrix [A], we rewrite the equation (1.55) as

$$[A]\{X\}=\lambda[I]\{X\} \tag{1.86}$$

or

$$[\lambda\mathbf{I}-\mathbf{A}]\cdot\{X\}=\{0\}. \tag{1.87}$$

There must be a nonzero solution of equation (1.87) in order for λ to be an eigenvalue. However, equation (1.87) can have a nonzero solution if and only if

$$|\lambda\mathbf{I}-\mathbf{A}|=0. \tag{1.88}$$

Equation (1.88) is called the *characteristic equation* of matrix $[A]$, and the scalars satisfy the equation (1.88) are the eigenvalues of matrix $[A]$. If matrix $[A]$ has the form

$$\mathbf{A}=\begin{bmatrix} a_{11} & a_{12} & a_{13} & \ldots & a_{1n} \\ a_{21} & a_{22} & a_{23} & \ldots & a_{2n} \\ a_{31} & a_{32} & a_{33} & \ldots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \ldots & a_{nn} \end{bmatrix}$$, then equation (1.88) can be written as

$$\mathbf{A} = \begin{vmatrix} a_{11}-\lambda & a_{12} & a_{13} & \dots & a_{1n} \\ a_{21} & a_{22}-\lambda & a_{23} & \dots & a_{2n} \\ a_{31} & a_{32} & a_{33}-\lambda & \dots & a_{3n} \\ . & . & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ a_{n1} & a_{n2} & a_{n3} & \dots & a_{nn}-\lambda \end{vmatrix} = 0 . \tag{1.89}$$

The equation (1.89) can be expanded to a polynomial equation in λ as

$$\lambda^n + c_1\lambda^{n-1} + \dots + c_{n-1}\lambda + c_n = 0 . \tag{1.90}$$

Thus, the *n*th-degree polynomial is

$$\left|\lambda\mathbf{I} - \mathbf{A}\right| = \lambda^n + c_1\lambda^{n-1} + \dots + c_{n-1}\lambda + c_n . \tag{1.91}$$

Equation (1.91) is called a *characteristic polynomial* of $n \times n$ matrix $[A]$. Indeed, the *n*th roots of the polynomial equation are the *n*th eigenvalues of matrix $[A]$. The solutions of equation (1.87) with the eigenvalues substituted on the equation are called *eigenvectors*.

EXAMPLE 1.19

Find the eigenvalues and eigenvectors of the 2×2 matrix $\mathbf{A} = \begin{bmatrix} 6 & -3 \\ -4 & 5 \end{bmatrix}$

Solution:

Since

$$[\lambda\mathbf{I} - \mathbf{A}] = \lambda\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} - \begin{bmatrix} 6 & -3 \\ -4 & 5 \end{bmatrix} = \begin{bmatrix} \lambda-6 & 3 \\ 4 & \lambda-5 \end{bmatrix},$$

the characteristic polynomial of matrix $[A]$ is

$$\left|\lambda\mathbf{I} - \mathbf{A}\right| = \begin{vmatrix} \lambda-6 & 3 \\ 4 & \lambda-5 \end{vmatrix} = (\lambda-6)(\lambda-5) - (3\times 4) = \lambda^2 - 11\lambda + 18 .$$

And the characteristic equation of matrix $[A]$ is

$$\lambda^2 - 11\lambda + 18 = 0 .$$

The solutions of this equation are $\lambda_1 = 2$ and $\lambda_2 = 9$; these values are the eigenvalues of matrix $[A]$.

The eigenvectors for each of the above eigenvalues are calculated using equation (1.87).

For $\lambda_1 = 2$, we obtain

$$\begin{bmatrix} 2-6 & 3 \\ 4 & 2-5 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} .$$

The above equation yields two simultaneous equations for x_1 and x_2, as follows:

$$-4x_1 + 3x_2 = 0 \quad \text{gives } x_1 = \frac{3}{4} x_2$$

$$4x_1 - 3x_2 = 0 \quad \text{gives } x_1 = \frac{3}{4} x_2 .$$

Thus, choosing $x_2 = 4$, we obtain the eigenvector $\mathbf{x}_2 = k \begin{Bmatrix} 3 \\ 4 \end{Bmatrix}$, where k is an arbitrary constant.

For $\lambda_2 = 9$, we obtain

$$\begin{bmatrix} 9-6 & 3 \\ 4 & 9-5 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} .$$

The above equation yields two simultaneous equations for x_1 and x_2, as follows:

$$3x_1 + 3x_2 = 0 \quad \text{gives } x_1 = -x_2$$

$$4x_1 + 4x_2 = 0 \quad \text{gives } x_1 = -x_2 .$$

Thus, choosing $x_1 = -1$, we obtain the eigenvector $\mathbf{x}_1 = k\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}$, where k is an arbitrary constant.

1.20 USING MATLAB

MATLAB is a numerical computation and simulation tool that uses matrices and vectors. Also, MATLAB enables users to solve wide analytical problems. The majority of engineering systems are presented by matrix and vector equations. Therefore, MATLAB becomes essential to reduce the computational workload.

All MATLAB commands or expressions are entered in the command window at the MATLAB prompt "≫". To execute a command or statement, we must press *return* or *enter* at the end. If the command does not fit on one line, we can continue the command on the next line by typing three consecutive periods (…) at the end of the first line. A semicolon (;) at the end of a command suppresses the screen output, and the command is carried out. Typing anything following a % is considered as comment, except when the % appears in a quote-enclosed character string or certain I/O format statements. Comment statements are not executable. To get help on a topic (such as matrix), you can type the command *help matrix*. Here, we introduce basic ideas of matrix and vector operations. For more details, see Appendix B.

Elements of a matrix are enclosed in brackets, and they are row-wise. The consecutive elements of a row are separated by a comma or a space and are entered in rows separated by a space or a comma, and the rows are separated by semicolons (;) or carriage returns (enter).

A vector is entered in the MATLAB environment in the same way as a matrix.

For example, matrix **A**,

$\mathbf{A} = \begin{bmatrix} 1 & 0 \\ 3 & 2 \end{bmatrix}$, is typed in MATLAB as

```
>> A = [1  0; 3  2]

A =

     1     0

     3     2
```

The basic scalar operations are shown in Table 1.2. In addition to operating on mathematical scalar, MATLAB allows us to work easily with vectors and matrices. Arithmetic operations can apply to matrices, and Table 1.3 shows extra common operations that can be implemented to matrices and vectors.

TABLE 1.2 MATLAB common arithmetic operators.

Operators symbols	Descriptions
+	Addition
–	Subtraction
*	Multiplication
/	Right division (means $\frac{a}{b}$)
\	Left division (means $\frac{b}{a}$)
^	Exponentiation (raising to a power)
‘	Converting to complex conjugate transpose
( )	Specify evaluation order

TABLE 1.3 Matrix operations.

Operations	Descriptions
A'	Transpose of matrix A
det(A)	Determinant of matrix A
inv(A)	Inverse of matrix A
eig(A)	Eigenvalues of matrix A
diag(A)	Diagonal elements of matrix A
rank(A)	Rank of matrix A
cond(A)	Condition number of matrix A
eye(n)	The $n \times n$ identity matrix (1's on the main diagonal)
eye(m, n)	The $m \times n$ identity matrix (1's on the main diagonal)
trace(A)	Summation of diagonal elements of matrix A

Operations	Descriptions
zeros(m, n)	The $m \times n$ matrix consisting of all zeros
ones(m, n)	The $m \times n$ matrix consisting of all ones
rand(m, n)	The $m \times n$ matrix consisting of random numbers
randn(m, n)	The $m \times n$ matrix consisting of normally distributed numbers
diag(A)	Extraction of the diagonal matrix A as vector
diag(A,1)	Extracting of first upper off-diagonal vector of matrix A
diag(**u**)	Generating of a diagonal matrix with a vector **u** on the diagonal
expm(A)	Exponential of matrix A
ln(A)	LU decomposition of matrix A
svd(A)	Singular value decomposition of matrix A
qr(A)	QR decomposition of matrix A
min(A)	Minimum of vector A
max(A)	Maximum of vector A
sum(A)	Sum of elements of vector A
std(A)	Standard deviation of the data collection of vector A
sort(A)	Sort the elements of vector A
mean(A)	Means value of vector A
triu(A)	Upper triangular of matrix A
triu(A, I)	Upper triangular with zero diagonals of matrix A
tril(A)	Lower triangular of matrix A
tril(A, I)	Lower triangular with zero diagonals of matrix A

EXAMPLE 1.20

Given the following matrices:

$$[A]=\begin{bmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \\ 7 & 8 & 9 \end{bmatrix}, [B]=\begin{bmatrix} 0 & -1 & 0 \\ 2 & -3 & 1 \\ 4 & -5 & 3 \end{bmatrix}, \text{ and } [C]=\begin{Bmatrix} 2 \\ 0 \\ 4 \end{Bmatrix}$$

Use MATLAB to perform the following operations:

a. $[A]+[B]$

b. $[A]-[B]$

c. $5[B]$

d. $[A][B]$

e. $[A][C]$

f. $[A]^2$

g. $[A]^T$

h. $[B]^{-1}$

i. tr $(\mathbf{A})$

j. $|B|$

Solution:

a. $[A]+[B]$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> B=[0 –1 0;2 –3 1;4 –5 3];
>> A+B

ans =
   1   1    3
   6   2    7
  11   3   12
```

b. $[A]-[B]$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> B=[0 -1 0;2 –3 1;4 –5 3];
>> A–B

ans =
   1    3   3
   2    8   5
   3   13   6
```

c. $5[B]$

```
>> B=[0 -1 0;2 -3 1;4 -5 3];
>> 5*B

ans =
    0   -5     0
   10   -15    5
   20   -25   15
```

d. $[A][B]$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> B=[0 -1 0;2 -3 1;4 -5 3];
>> A*B

ans =
   16   –22   11
   34   –49   23
   52   –76   35
```

e. $[A][C]$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> C=[2;0;4];
>> A*C

ans =
   14
   32
   50
```

f. $[A]^2$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> A^2

ans =
    30    36    42
    66    81    96
   102   126   150
```

g. $[A]^T$

```
>> A=[1 2 3;4 5 6;7 8 9];
>> A'

ans =
   1   4   7
   2   5   8
   3   6   9
```

h. $[B]^{-1}$

```
>> B=[0 -1 0;2 -3 1;4 -5 3];
>> inv(B)

ans =
   –2.0000   1.5000   –0.5000
   –1.0000        0         0
    1.0000  –2.0000    1.0000
```

i. tr ($\mathbf{A}$)

```
>> A=[1 2 3;4 5 6;7 8 9];
>> trace(A)

ans =
   15
```

j. $|B|$

```
>> B=[0 -1 0;2 -3 1;4 -5 3];
>> det(B)

ans =
   2
```

EXAMPLE 1.21

Solve the following system of three equations:

$$5x + y + 2z = 6$$
$$-x + 4y + z = 7$$
$$x - 2y - z = -3$$

using the following methods:

a. The matrix inverse

b. Gaussian elimination

c. Reverse Row Echelon Function

Solution:

a. Since we know $A^{-1}A = 1$, we can find the solution of the system of linear equations $AX = B$ by using $X = A^{-1}B$.

Now, we write the system of equations by using the following matrices:

$$\mathbf{A} = \begin{bmatrix} 5 & 1 & 2 \\ -1 & 4 & 1 \\ 1 & -2 & -1 \end{bmatrix}, \mathbf{X} = \begin{bmatrix} x \\ y \\ z \end{bmatrix}, \mathbf{B} = \begin{bmatrix} 6 \\ 7 \\ -3 \end{bmatrix}$$

```
>> A = [5 1 2; –1 4 1; 1 –2 –1];
>> B = [6; 7; -3];
>> X= inv(A)*B

X =
    0.8571
    2.0000
   –0.1429
```

Generally, using the matrix inverse to solve linear systems of equations should be avoided due to the excessive round-off errors.

b. We use the left division operator in MATLAB X = A\B to solve linear systems of equations using Gaussian elimination.

```
>> A = [5 1 2; -1 4 1; 1 -2 -1];
>> B = [6; 7; -3];
>>  X=A\B

X =
    0.8571
    2.0000
   –0.1429
```

c. The reduced row echelon function use, *rref*, to solve the system of linear equations. The *rref* function requires an expanded matrix as input, representing the coefficients and results. The last column in the output array represents the solution of equations.

```
>> A = [5 1 2; -1 4 1; 1 -2 -1];
>> B = [6; 7; -3];
>> C = [A,B];
>> rref(C)

ans =
   1.0000         0         0    0.8571
        0    1.0000         0    2.0000
        0         0    1.0000   –0.1429
```

EXAMPLE 1.22

Solve the following set of equations using Cramer's rule:

$$5x_1 + x_3 + 2x_4 = 3$$
$$x_1 + x_2 + 3x_3 + x_4 = 5$$
$$x_1 + x_2 + 2x_4 = 1$$
$$x_1 + x_2 + x_3 + x_4 = -1$$

Solution:

```
>> A = [5 0 1 2;1 1 3 1; 1 1 0 2;1 1 1 1];
>> B = [3;5;1;-1];
>> A1 = [B A(:,[2:4])];
>> A2=[A(:,1) B A(:,[3:4])];
>> A3=[A(:,[1:2]) B A(:,4)];
>> A4=[A(:,[1:3]) B];
>> x1=det(A1)/det(A)

x1 =
   –2
>> x2=det(A2)/det(A)

x2 =
   –7
>> x3=det(A3)/det(A)

x3 =
   3
>> x4=det(A4)/det(A)

x4 =
   5
```

EXAMPLE 1.23

Consider the matrix $\mathbf{A} = \begin{bmatrix} 2 & 1 & -5 \\ -1 & 4 & 3 \\ 1 & 2 & -4 \end{bmatrix}$, find the eigenvalue and eigenvector of matrix **A**.

Solution:

```
>> A=[2,1,-5;-1,4,3;1,2,-4];
>> y=eig(A)

y =
   –3.6006
    0.8831
    4.7176
>> [V,D]=eig(A)

V =
   –0.6724   0.9685   –0.0436
    0.1935   0.0859    0.9748
   –0.7145   0.2335    0.2186

D =
   –3.6006      0         0
       0      0.8831      0
       0        0       4.7176
)
```

EXERCISES

1. Identify the size and the type of the given matrices. Identify if the matrix is a square, column, diagonal, row, identity, banded, symmetric, or triangular.

a. $\begin{Bmatrix} x_1 \\ y_1 \\ z_1 \\ t \end{Bmatrix}$ **b.** $\begin{bmatrix} 7 & 5 & 3 & 1 \end{bmatrix}$ **c.** $\begin{bmatrix} -1 & 0 & 1 \\ 2 & 6 & 4 \\ 7 & 5 & 2 \end{bmatrix}$ **d.** $\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$ **e.** $\begin{bmatrix} 2 & 0 \\ 0 & 4 \end{bmatrix}$

f. $\begin{bmatrix} 1 & 3 & 0 \\ 5 & 6 & 4 \\ 2 & 0 & 7 \end{bmatrix}$ **g.** $\begin{bmatrix} 1 & b & c & d \\ 0 & 1 & e & f \\ 0 & 0 & 1 & a \\ 0 & 0 & 0 & 1 \end{bmatrix}$ **h.** $\begin{bmatrix} 2 & 4 & 0 & 0 & 0 \\ 3 & 9 & -1 & 0 & 0 \\ 0 & 4 & 8 & 2 & 0 \\ 0 & 0 & 6 & 7 & 3 \\ 0 & 0 & 0 & 1 & 5 \end{bmatrix}$

i. $\begin{bmatrix} a & 0 & 0 & 0 \\ 0 & b & 0 & 0 \\ 0 & 0 & c & 0 \\ 0 & 0 & 0 & d \end{bmatrix}$

2. Given the matrices $\mathbf{A} = \begin{bmatrix} 2 & 1 & 6 \\ 0 & 3 & 5 \\ 1 & -7 & 4 \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} 5 & 2 & 4 \\ 3 & 1 & 6 \\ 0 & -2 & 1 \end{bmatrix}$, and $\mathbf{C} = \begin{Bmatrix} 3 \\ 2 \\ 1 \end{Bmatrix}$ find

a. **A+B**

b. **A-B**

c. 4**A**

d. **AB**

e. **A{C}**

f. **A**2

g. **IA**

h. **AI**

3. Given the matrices $\mathbf{A} = \begin{bmatrix} 1 & 8 & 3 \\ 5 & 3 & 1 \\ 0 & -3 & 4 \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} 2 & 3 & 0 \\ 1 & 5 & -6 \\ 0 & 4 & 7 \end{bmatrix}$, find the following:

a. $\mathbf{A}^T$

b. $\mathbf{B}^T$

c. $|A|$

d. $[A]^{-1}$

e. $[B]^{-1}$

4. What are the 3×3 null matrix and the 5×5 identity matrix?

5. Express the following systems of equations in matrix form **AX** = **B**.

a. $3x_1 + 2x_2 = 10, \quad 3x_1 + 4x_2 = -8$

b. $2x_1 + 3x_2 + 5x_3 = 20, \quad x_1 + 3x_2 - 5x_3 = 0, \quad 2x_1 - 3x_2 - 4x_3 = 0$,

6. Solve the system using the Gaussian elimination method.

$$x_1 + x_2 + 2x_3 = 8$$
$$-x_1 - 2x_2 + 3x_3 = 1$$
$$3x_1 - 7x_2 + 4x_3 = 10$$

7. Solve the simultaneous equations using Cramer's rule.

$$2x_1 + 3x_2 = 8$$
$$3x_1 + 4x_2 - 5x_3 = 2$$
$$x_1 - x_2 + 2x_3 = 1$$

8. Show that vector $\mathbf{A} \cdot \mathbf{B} = A_x B_x + A_y B_y + A_z B_z$, know that $\mathbf{a}_x \cdot \mathbf{a}_x = \mathbf{a}_y \cdot \mathbf{a}_y = \mathbf{a}_z \cdot \mathbf{a}_z = 1$.

9. Given vectors $\mathbf{A} = 2\mathbf{a}_x + 4\mathbf{a}_y - 5\mathbf{a}_z$ and $\mathbf{B} = 3\mathbf{a}_x - \mathbf{a}_y + \mathbf{a}_z$, find:

a. $\mathbf{A} \cdot \mathbf{B}$

b. $\mathbf{A} \times \mathbf{B}$

c. $3\mathbf{A} - 5\mathbf{B}$

10. Given vectors $\mathbf{A} = \mathbf{a}_x + 5\mathbf{a}_y$ and $\mathbf{B} = 4\mathbf{a}_x + 3\mathbf{a}_y - 2\mathbf{a}_z$, find θ_{AB}.

11. Show that if vector $\mathbf{A} = 5\mathbf{a}_x - 4\mathbf{a}_y - \mathbf{a}_z$ and vector $\mathbf{B} = \mathbf{a}_x + 2\mathbf{a}_y + 2\mathbf{a}_z$, then they are perpendicular or not.

12. Given vectors $\mathbf{A} = 6\mathbf{a}_x + \mathbf{a}_y - 2\mathbf{a}_z$ and $\mathbf{B} = 3\mathbf{a}_x + \lambda\mathbf{a}_y + \mu\mathbf{a}_z$

a. If **A** and **B** are parallel, find λ and μ

b. If **A** and **B** are perpendicular, find λ and μ

13. Determine the gradient of the scalar fields $A = x^3 y + xyz$.

14. Determine the gradient of the scalar fields $A = \rho y \sin\phi + x^2 \cos\phi + 3\rho$.

15. Find the divergence $(\nabla \bullet \mathbf{A})$ of the vector field $\mathbf{A} = 2\rho \sin \mathbf{a}_\rho + \rho^3 y^2 z \mathbf{a}_\phi + 4z \cos\phi \mathbf{a}_z$.

16. Given $A = \rho^2 yz \cos 3\phi$, find the Laplacian $\nabla^2 A$.

17. Find the eigenvalues and eigenvectors of the 2×2 matrix $\mathbf{A} = \begin{bmatrix} 3 & 4 \\ 2 & 7 \end{bmatrix}$.

18. Solve problem 2 using MATLAB.

19. Solve problem 3 using MATLAB.

20. Consider the matrix $\mathbf{A} = \begin{bmatrix} -3 & 2 & 6 \\ 8 & 5 & 7 \\ -1 & 3 & 2 \end{bmatrix}$, find the eigenvalue and eigenvector of matrix **A**.

REFERENCES

1. R. Butt, "Applied Linear Algebra and Optimization using MATLAB," Mercury Learning and Information, 2011.
2. H. Anton, "Elementary Linear Algebra," 6th Edition, John Wiley and Sons, INC., 1991.
3. S. Nakamura, "Applied Numerical Methods with Software," Prentice-Hall, 1991.
4. B. Kolman, "Introductory Linear Algebra with Applications," 6th Edition, John, Prentice Hall, 1997.
5. H. Schneider and G. P. Barker, "Matrix and Linear Algebra with Applications," 2nd Edition, Holt, Rinehart and Winston, Inc., 1973.

CHAPTER 2

Introduction to the Finite Element Method

2.1 INTRODUCTION

The finite element method (FEM) is a computational method to divide the structure, body, or region being analyzed into a large number of finite elements. These elements may be one, two, or three dimensions. The finite element analysis (FEA) method is a numerical procedure that applies to many areas in real-world engineering problems, including structural/stress analysis, fluid flow analysis, heat transfer analysis, and electromagnetics analysis. Indeed, finite element has several advantages and features such as the capability of solving complicated and complex geometries, flexibility, strong mathematical foundation, and high-order approximation. Therefore, FEA has become an important method in the design and modeling of a physical event in many engineering disciplines. The actual component in the FEA method is placed by a simplified model that is identified by a finite number of *elements* connected at common points called *nodes*, with an assumed response of each element to applied loads, and then evaluating the unknown field variable (displacement) at these nodes.

2.2 METHODS OF SOLVING ENGINEERING PROBLEMS

There are three common methods to solve any engineering problem:

1. Experimental method
2. Analytical method
3. Numerical method

2.2.1 Experimental Method

This method involves actual measurement of the system response. This method is time-consuming and needs expensive setup. This method is applicable only if the physical prototype is available. The results obtained by this method cannot be believed blindly, and a minimum of three to five prototypes must be tested. Examples of this method are strain photo elasticity, heat transfer for a gas turbine engine, static and dynamic response for aircraft and spacecraft, amount of water that is lost for groundwater seepage, etc.

2.2.2 Analytical Method

This is a classic approach. This method gives closed-form solutions. The results obtained with this method are accurate within the assumptions made. This method is applicable only for solving problems of simple geometry and loading, like cantilevers and simply supported beams, etc. Analytical methods produce exact solutions to the problem. Examples of this method are integral solutions (such as Laplace and Fourier transform), conformal mapping, perturbation methods, separation of variables, and series expansion.

2.2.3 Numerical Method

This approximate method is resorted to when the analytical method fails. This method is applicable to real-life problems of a complex nature. Results obtained by this method cannot be believed blindly and must be carefully assessed against experience and the judgment of the analyst. Examples of this method are FEM, finite difference method, moment method, etc.

2.3 PROCEDURE OF FINITE ELEMENT ANALYSIS (RELATED TO STRUCTURAL PROBLEMS)

Step (i). Discretization of the structure

This first step involves dividing the structure or domain of the problem into small divisions or elements. The analyst has to decide about the type, size, and arrangement of the elements.

Step (ii). Selection of a proper interpolation (or displacement) model

A simple polynomial equation (linear/quadratic/cubic) describing the variation of state variable (e.g., displacement) within an element is assumed. This model generally is the interpolation/shape function type. Certain conditions are to be satisfied by this model so that the results are meaningful and converging.

Step (iii). Derivation of element stiffness matrices and load vectors

Response of an element to the loads can be represented by the element equation of the form

$$[k]\{q\}=\{Q\} \tag{2.1}$$

where $[k]=$ Element stiffness matrix,

$\{q\}=$ Element response matrix or element nodal displacement vector, or nodal degree of freedom,

$\{Q\}=$ Element load matrix or element nodal load vector.

From the assumed displacement model, the element properties, namely stiffness matrix and the load vector are derived. Element stiffness matrix $[k]$ is a characteristic property of the element and depends on geometry as well as material. There are three approaches for deriving element equations. They are

(a) Direct approach,

(b) Variational approach,

(c) Weighted residual approach.

(a) **Direct approach:** In this method, direct physical reasoning is used to establish the element properties (stiffness matrices and load vectors) in terms of pertinent variables. Although this approach is limited to simple types of elements, it helps to understand the physical interpretation of the FEM.

(b) **Variational approach:** This approach can be adopted when the variational theorem (extremum principle) that governs the physics of the problem is available. This method involves minimizing a scalar quantity known as functional that is typical of the problem at hand (e.g., potential energy in stress analysis problems).

(c) **Weighted residual approach:** This approach is more general in that it applies to all situations where the governing differential equation of the problem is available. This method involves minimizing error resulting from substituting trial solution into the differential equation.

Step (iv). Assembling of element equations to obtain the global equations

Element equations obtained in *Step (iii)* are assembled to form global equations in the form of

$$[K]\{r\} = \{R\} \tag{2.2}$$

where [K] is the global stiffness matrix,

$\{r\}$ is the vector of global nodal displacements, and

{R} is the global load vector of nodal forces for the complete structure.

Equation (2.2) describes the behavior of the entire structure.

Step (v). Solution for the unknown nodal displacements

The global equations are to be modified to account for the boundary conditions of the problem. After specifying the boundary conditions, the equilibrium equations can be expressed as

$$[K_1]\left\{r_1\right\} = \left\{R_1\right\}. \tag{2.3}$$

For linear problems, the vector $\left\{r_1\right\}$ can be solved very easily.

Step (vi). Computation of element strains and stresses

From the known nodal displacements $\left\{r_1\right\}$, the element's strains and stresses can be computed using predefined structural equations.

The terminology used in the previous six steps must be modified to extend the concept to other fields. For example, put the field variable in place of displacement, the characteristic matrix in place of the stiffness matrix, and the element resultants in place of element strains.

2.4 METHODS OF PRESCRIBING BOUNDARY CONDITIONS

There are three methods of prescribing boundary conditions.

2.4.1 Elimination Method

This method is useful when performing hand calculations. It poses difficulties in implementing software. This method has been used in this book for solving the problems by FEM using hand calculations and results in reduced sizes of matrices, thus making it suitable for hand calculations. The method is explained below in brief. Consider the following set of global equations,

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \\ P_4 \end{Bmatrix}. \tag{2.4}$$

Let u_3 be prescribed, i.e., $u_3 = s$.

This condition is imposed as follows:

i. Eliminate the row corresponding to u_3 (third row).

ii. Transfer the column corresponding to u_3 (third column) to the right-hand side after multiplying it by "s." These steps result in the following set of modified equations,

$$\begin{bmatrix} k_{11} & k_{12} & k_{14} \\ k_{21} & k_{22} & k_{24} \\ k_{41} & k_{42} & k_{44} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ P_4 \end{Bmatrix} - s \begin{Bmatrix} k_{13} \\ k_{23} \\ k_{43} \end{Bmatrix}. \tag{2.5}$$

This set of equations may now be solved for nontrivial solution.

2.4.2 Penalty Method

This is the method used in most commercial software because this method facilitates prescribing boundary conditions without changing the sizes of the matrices involved. This makes implementation easier.

2.4.3 Multipoint Constrains Method

This method is commonly used in functional analysis between nodes. For example, there are many applications in trusses where the end supports are on an inclined plane and do not coincide with the coordinate system used to describe the truss. Another application of the method is the functional relationship between the temperature at one node and the temperature at one or more other nodes.

2.5 PRACTICAL APPLICATIONS OF FINITE ELEMENT ANALYSIS

There are three practical applications of FEA:

- Analysis of new design
- Optimization projects
- Failure analysis

2.6 FINITE ELEMENT ANALYSIS SOFTWARE PACKAGE

There are three main steps involved in solving an engineering problem using any commercial software:

Step (i). Preprocessing

In this step, a CAD model of the system (component) is prepared and meshed (discretized). Boundary conditions (support conditions and loads) are applied to the meshed model.

Step (ii). Processing

In this step, the software internally calculates the elements stiffness matrices, element load vectors, global stiffness matrix, global load vector, and solves after applying boundary conditions for primary unknowns (e.g., displacements/temperatures, etc.) and secondary unknowns (e.g., stress/strain/heat flux, etc.).

Step (iii). Postprocessing

Postprocessing involves sorting and plotting the output to make the interpretation of results easier.

2.7 FINITE ELEMENT ANALYSIS FOR STRUCTURE

Several common methods in FEA are used for evaluating displacements, stresses, and strains in any structure under different boundary conditions and loads. They are summarized below:

1. *Displacement Method:*

This method is the most commonly used method. The structure is subjected to applied loads or/and specific displacements. The primary unknowns are displacements found by using an inversion of the stiffness matrix, and the derived unknowns are stresses and strains. Indeed, the stiffness matrix for any element can be calculated by the variational principle.

2. *Force Method:*

The structure is subjected to applied loads or/and specific displacements. The primary unknowns are member forces, found by using an inversion of the flexibility matrix, and the derived unknowns are stresses and strains. Indeed, the calculation of the flexibility matrix is possible only for discrete structural elements (e.g., piping, beams, and trusses).

3. *Mixed Method:*

The structure is subjected to applied loads or/and specific displacements. This method uses very large stiffness coefficients and very small flexibility coefficients in the same matrix.

4. *Hybrid Method*:

The structure is subjected to applied loads and stress boundary conditions. This hybrid method has the merit of the FEA method, i.e., the flexibility and sparse matrix of FEM for complicated inhomogeneous scatterers.

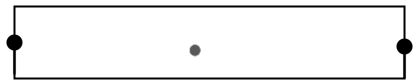
2.8 TYPES OF ELEMENTS

In general, the region in space is considered nonregular geometric. However, the FEA method divides the nonregular geometric region into small regular geometric regions. There are three types of elements in finite elements.

1. One-Dimensional Elements: The objects are subdivided into short-line segments. A one-dimensional finite element expresses the object as a function of one independent variable such as one coordinate x. Finite elements use one-dimensional elements to solve systems that are governed by ordinary differential equations in terms of an independent variable. The number of node points in an element can vary from two up to any value needed. Indeed, increasing the number of nodes for an element increases the accuracy of the solution, but it also increases the complexity of calculations. When the elements have a polynomial approximation

higher than first order, we call them *higher order elements*. Figure 2.1 shows one-dimensional elements. For example, the one-dimensional element is sufficient in dealing with heat dissipation in cooling fins.

(a) Two-node first order element (simplest)

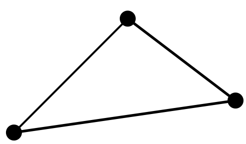


(b) Three-node higher order element

(c) Four-node higher order element

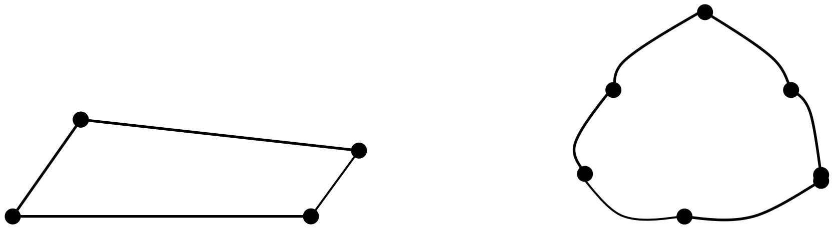
FIGURE 2.1 One-dimensional elements.

2. Two-Dimensional Elements: The objects can be divided into triangles, rectangles, quadrilaterals, or other suitable subregions. A two-dimensional finite element expresses the object as a function of two variables such as the two coordinates x and y. A finite element uses two-dimensional elements to solve systems that are governed by partial differential equations. The simplest two-dimensional element is the triangular element. Figure 2.2 shows two-dimensional elements. For example, a two-dimensional element is sufficient in plane stress or plane strain.



(a) Three-node triangular element (simplest)

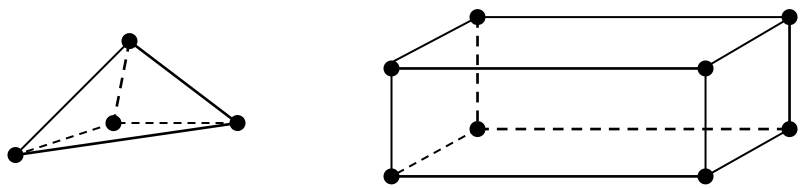
(b) Four-node rectangular element



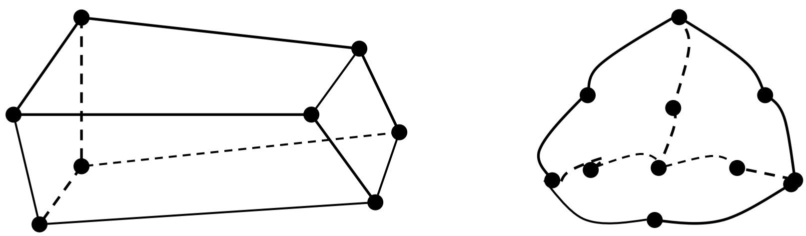
(c) Four-node quadrilateral element (d) Six-node curved triangular isoparametric element

FIGURE 2.2 Two-dimensional elements.

3. Three-Dimensional Elements: The objects can be divided into tetrahedral elements, rectangular prismatic elements, pie-shaped elements, or other suitable shapes of elements. A three-dimensional finite element expresses the object as a function of three variables such as the three coordinates x, y, and z. A finite element uses three-dimensional elements to solve systems that are governed by differential equations. The simplest three-dimensional element is the tetrahedral element. Figure 2.3 shows three-dimensional elements.



(a) Four-node tetrahedral element (simplest) (b) Eight-node rectangular solid element



(c) Eight-node hexahedral solid element (d) Ten-node curved tetrahedral solid isoparametric element

FIGURE 2.3 Three-dimensional elements.

These three types of elements are applied and discussed in the electromagnetics analysis chapter of this book.

2.9 DIRECT METHOD FOR LINEAR SPRING

Here, we will use the direct method in a one-dimension domain to derive the stiffness matrix for the linear spring element shown in Figure 2.4. Reference points 1 and 2, located at the ends of the linear spring element, are the nodes. The symbols f_1 and f_2 are local nodal forces (or axial loads) associated with the local axis x. The symbols u_1 and u_2 are local nodal displacements (or degree of freedom at each node) for the spring element. u_i is the displacement of the spring due to a load f_i. The symbol k is the stiffness of the spring (or spring constant). k is load required to give the spring a unit displacement. The symbol L is the bar length. The local axis x acts in the same direction of the spring which can lead to direct measurement of forces and displacements along the spring.

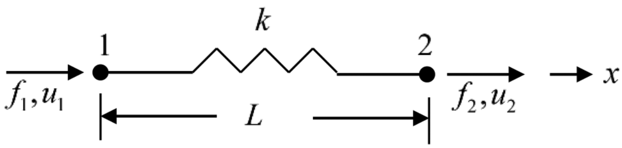


FIGURE 2.4 Linear spring element.

The displacements can be defined as related to forces as

$$u = u_1 - u_2 \tag{2.6}$$

$$f_1 = ku = k(u_1 - u_2). \tag{2.7}$$

The equilibrium of forces gives

$$f_2 = -f_1. \tag{2.8}$$

Based on equation (2.7), the above equation becomes

$$f_2 = k(u_2 - u_1). \tag{2.9}$$

By combining equations (2.7) and (2.9) and writing the resulting equations in matrix form, we get

$$\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{2.10}$$

or

$$\{f_i\} = [k]\{u_i\} \tag{2.11}$$

where

$\{f_i\}$ = a vector of internal nodal forces = $\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}$

$[k]$ = the elemental stiffness matrix = $\begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$

$\{u_i\}$ = a vector of nodal displacements = $\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$.

For many interconnected spring elements, we can use the following:

$$\{Q_i\} = [K]\{u_i\} \tag{2.12}$$

where

$\{Q_i\}$ = a vector of external nodal forces = $\sum\{f_i\}$

$[K]$ = the structural stiffness matrix = $\sum[k]$

$\{u_i\}$ = a vector of nodal displacements of the structure.

EXERCISES

1. Define the finite element method.
2. Define finite element analysis.
3. What are the advantages and features of finite element analysis?
4. What are the three common methods to solve any engineering problem?
5. What is the procedure of finite element analysis (related to structural problems)?

6. What are the two methods for prescribing boundary conditions?
7. Give the three practical applications of finite element analysis.
8. What are the three main steps involved in solving an engineering problem using any commercial software?
9. What are the four common methods in finite element analysis used for evaluating displacements, stresses, and strains in any structure under different boundary conditions and loads?
10. What is the primary variable in finite element method structural analysis?
11. Calculate the structural stiffness matrix of the system as shown in Figure 2.5.

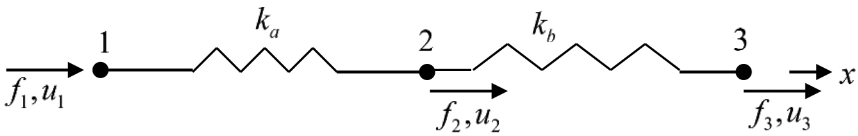


FIGURE 2.5 Two springs in series structure.

REFERENCES

1. P. P. Silvester and R. L. Ferrari, "Finite Elements for Electrical Engineering," Cambridge Press, 1983.
2. B. Szabo and I. Babuska, "Finite Element Analysis," John Wiley & Sons, 1991.
3. R. D. Cook and D. S. Malkus, and M. E. Plesha, "Concepts and Applications of Finite Element Analysis," John Wiley and Sons, 1989.
4. O. C. Zienkiewicz, "The Finite Element Method," 3rd ed., McGraw-Hill, 1979.
5. W. B. Bickford, "A First Course in the Finite Element Method," Richard D. Irwin Publisher, 1989.
6. G. L. Narasaiah, "Finite Element Analysis," CRC Press, 2009.
7. P. E. Allaire, "Basics of the Finite Element Method: Solid Mechanics, Heat Transfer, and Fluid Mechanics," Wm. C. Brown Publishers, 1985.
8. CTF Ross, "Finite Element Methods in Structural Mechanics," Ellis Horwood Limited Publishers, 1985.

CHAPTER 3

Finite Element Analysis of Axially Loaded Members

3.1 INTRODUCTION

In this chapter, we will use the bar element in the analysis of rod-like axially loaded members. We start with the two popular bar elements using a two-node element and a three-node element as well as bars of constant cross-section area, bars of varying cross-section area, and the stepped bar.

Stress is an internal force that has been distributed over the area of the rod's cross-section, and it is defined as

$$\sigma = \frac{F}{A}, \tag{3.1}$$

where σ is the stress, F is the force, and A is the cross-sectional area.

Thus, stress is a measure of force per unit area. When the stress tends to lengthen the rod, the stress is called tension, and $\sigma > 0$. When the stress tends to shortened the rod, the stress is called compression, and $\sigma < 0$. The orientations of forces in tension and compression are shown in Figure 3.1.

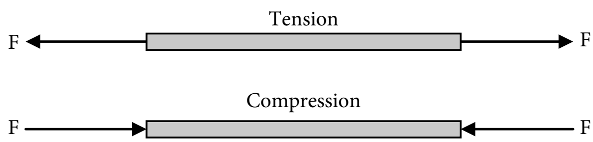


FIGURE 3.1 Directions of tensile and compressive forces.

The derived unit of stress is the pascal (Pa), where pascal is equal to newtons per square meter (N/m^2), 1 Pa = 1 N/m^2, pascal is used in the SI units. The derived unit of stress is the dimension pound-per-square-inch (psi), where 1 psi = 1 lb/in^2. Psi is used in the USCS (U.S. Customary) units. In stresses, calculations are generally very large; therefore, they often use the prefixes kilo- (k), mega- (M), and giga- (G) for factors of 10^3, 10^6, and 10^9, respectively. Thus,

$$1 \text{ kPa} = 10^3 \text{ Pa}, \qquad 1 \text{ MPa} = 10^6 \text{ Pa}, \qquad 1 \text{ GPa} = 10^9 \text{ Pa}.$$

The numerical values for stresses unit conversion between the USCS and SI can be presented as

$$1 \text{ psi} = 6.895 \times 10^{-3} \text{ MPa}.$$

Strain (ε) is the amount of elongation that occurs per unit of the rod's original length and is calculated as

$$\varepsilon = \frac{\Delta L}{L}, \tag{3.2}$$

where ΔL is the change in length of the rod (elongation).

Strain is a dimensionless quantity and is generally very small.

For each individual rod, the applied force and elongation are proportional to each other based on the following expression.

$$F = k\Delta L, \tag{3.3}$$

where k is the stiffness.

Figure 3.2 shows force and elongation behaviors of rods at various cross-sectional areas and lengths.

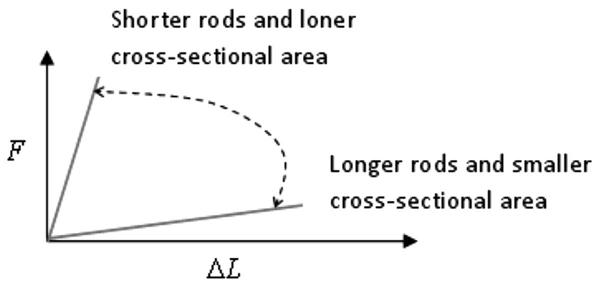


FIGURE 3.2 Force and elongation behaviors of rods at various cross-sectional areas and lengths.

Stress and strain are useful in mechanical engineering because they are scaled with respect to the rod's size.

Stress and strain are proportional to each other and presented as

$$\sigma = E\varepsilon, \tag{3.4}$$

where E is the elastic modulus (or Young's modulus).

The elastic modulus has the dimensions of force per unit area. The elastic modulus is a physical material property and is the slope of the stress-strain curve for low strain.

By combining equations (3.1) and (3.2), we get

$$\Delta L = \frac{FL}{EA}. \tag{3.5}$$

With the stiffness in equation (3.3), it can be written as

$$k = \frac{EA}{L}. \tag{3.6}$$

Each rod formed of the same material has similar stress-strain behavior as presented in Figure 3.3.

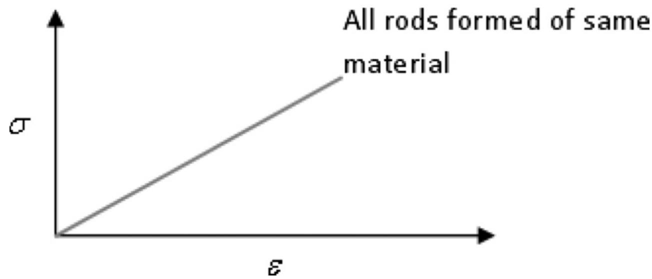


FIGURE 3.3 All rods formed of the same material have similar stress-strain behavior.

When a system is motionless or has constant velocity, then the system has zero acceleration, and the system is to be in *equilibrium*. The *static equilibrium* is used for a system at rest. For equilibrium, the resultant of all forces and all moments acting on the system is balanced to zero resultant. That is, the sum of all force vectors (**F**) acting upon a system is zero, and the sum of all moment vectors (**M**) acting upon a system is zero, and they can be written as

$$\sum \mathbf{F} = 0 \tag{3.7}$$

$$\sum \mathbf{M} = 0. \tag{3.8}$$

The total extension (or contraction) of a uniform bar in pure tension or compression is defined as

$$\delta = \frac{FL}{AE}. \tag{3.9}$$

The equation (3.9) does not apply to a long bar loaded in compression if there is a possibility of bucking.

3.1.1 Two-Node Bar Element

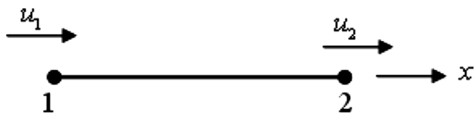


FIGURE 3.4 Two-node bar for rod-like axially loaded members.

This element has two end nodes, and each node has 1 degree of freedom, namely translation along its length. Its formulation is based on linear interpolation. It gives accurate results only if loads are applied at nodes, and the area is constant over the element. However, required accuracy for practical purposes in other cases can be obtained by taking a larger number of smaller elements. The interpolation equation, element stiffness matrix, strain-displacement matrix, element strain, and element stress for a 2-node (linear) bar element are given by

$$\{u\} = \begin{bmatrix} N_1 & N_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{3.10}$$

$$\{u\} = \begin{bmatrix} \frac{(x_2 - x)}{L} & \frac{(x - x_1)}{L} \end{bmatrix} \tag{3.11}$$

$$[k] = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{3.12}$$

$$[B]=\frac{1}{L}\begin{bmatrix}-1 & 1\end{bmatrix} \tag{3.13}$$

$$\{\varepsilon\}=[B]\{q\} \tag{3.14}$$

$$\{\sigma\}=E[B]\{q\} \tag{3.15}$$

where

u_1 and u_2 = nodal (unknown) displacements (degree of freedom) at nodes 1 and 2, respectively

$\{u\}$ = displacement matrix at the nodes

A = cross-section of the area of the bar

$L=x_2-x_1$ = length of the bar

E = Young's modulus (modulus of elasticity)

$\frac{AE}{L}$ = bar constant

$[k]$ = stiffness matrix of the element

$[B]$ = strain-displacement matrix

$\{\varepsilon\}$ = strain matrix

$\{\sigma\}$ = stress matrix.

Uniformly distributed load per unit length w acting on the element can be converted into equivalent loads using,

$$\{W\}=wL\begin{Bmatrix}\frac{1}{2}\\ \frac{1}{2}\end{Bmatrix}, \tag{3.16}$$

$\{W\}$ = the potential energy of load system.

Thermal loads due to a change in temperature ΔT can be converted into equivalent nodal loads using

$$\{Q\} = EA\alpha\,(\Delta T)\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}, \tag{3.17}$$

where

α is the coefficient of thermal expansion.

3.1.2 Three-Node Bar Element

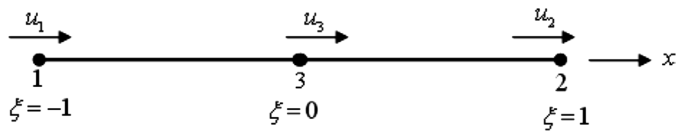


FIGURE 3.5 Three-node bar for rod-like axially loaded members.

This element has a midside node, in addition to 2 end nodes. Each node has 1 degree of freedom, namely translation along its length. Its formulation is based on quadratic interpolation, and this element gives accurate results even with distributed loads and a linearly varying cross-sectional area. A coarse mesh with fewer of these elements can give the desired accuracy as compared to a fine mesh of a 2-node bar element. The interpolation equation, element stiffness matrix, strain-displacement matrix, element strain, and element stress for the quadratic bar element are given by,

$$[u] = [N_1 \quad N_2 \quad N_3]\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} -\dfrac{\xi}{2}+\dfrac{\xi^2}{2} & \dfrac{\xi}{2}+\dfrac{\xi^2}{2} & 1-\xi^2 \end{bmatrix} \tag{3.18}$$

$$[k] = \frac{AE}{3L}\begin{bmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{bmatrix} \tag{3.19}$$

$$[B] = \frac{2}{L}\begin{bmatrix} -\dfrac{1-2\xi}{2}, & \dfrac{1+2\xi}{2}, & -2\xi \end{bmatrix} \tag{3.20}$$

$$\{\varepsilon\} = [B]\{q\} \tag{3.21}$$

$$\{\sigma\} = E[B]\{q\}. \tag{3.22}$$

Uniformly distributed load per unit length w, acting on the element, can be converted into equivalent loads using,

$$\{W\} = w\frac{L}{6}\begin{Bmatrix}1\\1\\4\end{Bmatrix}. \tag{3.23}$$

3.2 BARS OF CONSTANT CROSS-SECTION AREA

This section will demonstrate examples on bars of constant cross-sectional area using FEA.

EXAMPLE 3.1

Consider a 2 m long steel bar of 50 mm^2 cross-sectional areas, as shown in Figure 3.6. Use a two-element mesh to model this problem. Find nodal displacements, element stresses, and reactions.

Take Young's modulus, $E = 2\times\frac{10^5 N}{mm^2}$, P = 100 N.

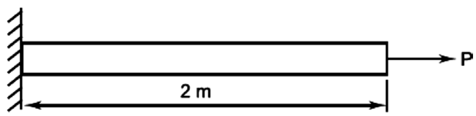


FIGURE 3.6 Bar with tip load for Example 3.1.

Solution

(I) Analytical method. [Refer to Figure 3.6(a).]

FIGURE 3.6(a) Analytical method for bar with tip load for Example 3.1.

Displacement calculation

Displacement at section *a-a*,

$$\delta = \frac{Px}{AE} = \frac{100x}{50 * 2 * 10^5} = 1 * 10^{-5} x.$$

Displacement at node 2,

$$\delta_{x\,=\,1000} = 1 \times 10^{-5} \times 1000 = 0.01 \text{ mm}.$$

Displacement at node 3,

$$\delta_{x\,=\,2000} = 1 \times 10^{-5} \times 2000 = 0.02 \text{ mm}.$$

Stress calculation

Maximum stress in the bar = $\dfrac{P}{A} = \dfrac{100}{50} = 2 \text{ N/mm}^2$ (Constant).

Reaction calculation

For reaction calculation, $\sum F_x = 0$

$R_1 + 100 = 0$

$R_1 = -100 \text{ N}$ (Direction is leftwards).

(II) FEM by hand calculations.

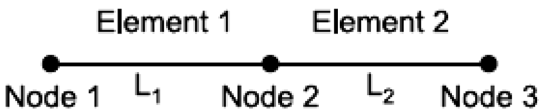


FIGURE 3.6(B) Finite element model for Example 3.1.

$$L_1 = L_2 = 1000 \text{ mm}$$

$$A = A_1 = A_2 = 50 \text{ mm}^2$$

$$E = E_1 = E_2 = 2 \times 10^5 \text{ N/mm}^2$$

Stiffness matrix for element 1 is,

$$[k_1] = \frac{A_1 E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50 \times 2 \times 10^5}{1000}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.1 \times 10^5 \begin{array}{c} \begin{array}{cc} 1 & \;\;2 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 2 \end{array}$$

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50\times 2\times 10^5}{1000}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.1\times 10^5 \begin{array}{c} \begin{array}{cc} 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 2 \\ 3 \end{array}$$

Global equation is,

$$[\mathrm{K}]\,\{r\} = \{\mathrm{R}\} \tag{3.24}$$

$$0.1\times 10^5 \begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 0 \\ 100 \end{Bmatrix}. \tag{3.25}$$

Boundary conditions are, at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$0.1\times 10^5 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 100 \end{Bmatrix}.$$

By matrix multiplication, we get

$$0.1\times 10^5 \left(2\times u_2 - u_3\right) = 0 \tag{3.26}$$

$$0.1\times 10^5 \left(-u_2 + u_3\right) = 100. \tag{3.27}$$

By solving equations (3.26) and (3.27), we get

$$u_2 = 0.01 \text{ mm}$$

$$u_3 = 0.02 \text{ mm}.$$

Stress (σ) calculation

Stress for element 1 is,

$$\{\sigma_1\} = \frac{E}{L_1}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \frac{2\times 10^5}{1000}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 0.01 \end{Bmatrix} = 2 \text{ N/mm}^2.$$

Stress for element 2 is,

$$\{\sigma_2\} = \frac{E}{L_2}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{2\times 10^5}{1000}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0.01 \\ 0.02 \end{Bmatrix} = 2 \text{ N/mm}^2.$$

Reaction calculation
From equation (3.24)

$$0.1\times10^5\left(u_1-u_2\right)=R_1$$

$$0.1\times10^5\left(0-0.01\right)=R_1$$

$$R_1=-100\text{ N.}$$

(III) Software results.

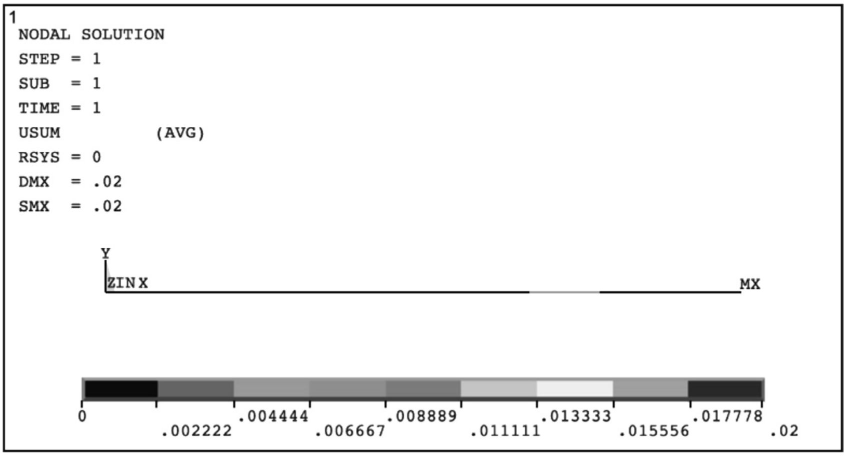


FIGURE 3.6(c) Deflection pattern for a bar (refer to Appendix D for color figures).

Deflection values as node *(Computer generated output)*
The following degree of freedom results is in global coordinates:

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.10000E−01	0.0000	0.0000	0.10000E−01
3	0.20000E−01	0.0000	0.0000	0.20000E−01

Maximum absolute values

NODE	3	0	0	3
VALUE	0.20000E−01	0.0000	0.0000	0.20000E−01

FIGURE 3.6(d) Stress pattern for a bar (refer to Appendix D for color figures).

Stress values at elements *(Computer generated output)*

STAT	CURRENT
ELEM	LS1
1	2.0000
2	2.0000

Reaction value *(Computer generated output)*
The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY
1	−100.00	0.0000

ANSWERS FOR EXAMPLE 3.1

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	0.01 mm	0.01 mm	0.01 mm
Displacement at node 3 (Maximum displacement)	0.02 mm	0.02 mm	0.02 mm
Maximum stress in element 1	2 N/mm^2	2 N/mm^2	2 N/mm^2
Maximum stress in element 2	2 N/mm^2	2 N/mm^2	2 N/mm^2
Reaction at fixed end	−100 N	−100 N	−100 N

EXAMPLE 3.2

Bar under distributed and concentrated forces. Consider the bar shown in Figure 3.7 subjected to loading as shown below. Use four-element mesh models and find nodal displacements, element stresses, and reactions at the fixed end. Take $E = 2\times10^5$ N/mm^2, $A = 50$ mm^2, $P = 100$ N.

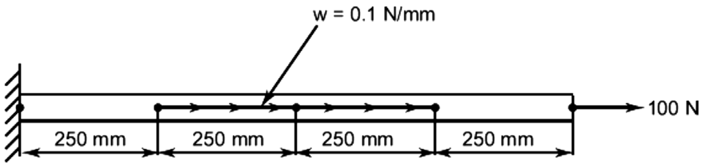


FIGURE 3.7 Bar under distributed and concentrated forces for Example 3.2.

Solution

(I) Analytical method [Refer to Figure 3.7(a)].

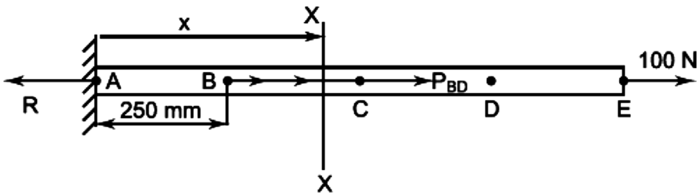


FIGURE 3.7(a) Analytical method for the bar under distributed and concentrated forces for Example 3.2.

Reaction calculation

$$-R + w\left[L_2 + L_3\right] + P = 0$$

$$-R + (0.1)\times\left[250 + 250\right] + 100 = 0$$

$$R = 150 \text{ N}$$

Stress calculation

$$\sigma_{AB} = \frac{P}{A} = \frac{150}{50} = 3 \text{ N/mm}^2$$

$$\sigma_{DE} = \frac{P}{A} = \frac{100}{50} = 2 \text{ N/mm}^2$$

To find σ_{BD}, consider section XX

$$\sigma_{XX} = \frac{P_{BD}}{A_{BD}} = \frac{150-(x-250)\times 0.1}{50} = \frac{150-0.1x+25}{50} = \frac{175-0.1x}{50}$$

$$\sigma_{XX}\big|_{\text{at B}} = \sigma_{XX}\big|_{x=250} = \frac{175-0.1\times 250}{50} = 3\ \text{N/mm}^2$$

$$\sigma_{XX}\big|_{\text{at C}} = \sigma_{XX}\big|_{x=500} = \frac{175-0.1\times 500}{50} = 2.5\ \text{N/mm}^2$$

$$\sigma_{XX}\big|_{\text{at D}} = \sigma_{XX}\big|_{x=750} = \frac{175-0.1\times 750}{50} = 2\ \text{N/mm}^2$$

Displacement calculation:

Displacement at E,

$$\delta_E = \Delta_{AB} + \Delta_{BD} + \Delta_{DE}$$

$$\delta_E = \frac{\sigma_{AB}L_{AB}}{E} + \int_{x=250}^{750} \frac{(175-0.1x)\,dx}{AE} + \left(\frac{\sigma_{DE}L_{DE}}{E}\right)$$

$$\delta_E = \frac{3\times 250}{2\times 10^5} + \frac{1}{50\times 2\times 10^5}\left(175x - 0.1\frac{x^2}{2}\right)_{250}^{750} + \frac{2\times 250}{2\times 10^5}$$

$$\delta_E = 0.00375 + 0.00625 + 0.0025 = 0.0125\ \text{mm}$$

Displacement at B,

$$\delta_B = \Delta_{AB} = \frac{3\times 250}{2\times 10^5} = 0.00375\ \text{mm}$$

Displacement at D,

$$\delta_D = \Delta_{AB} + \Delta_{BD} = \frac{3\times 250}{2\times 10^5} = 0.00375 + 0.00625 = 0.01\ \text{mm}$$

Displacement at C,

$$\delta_C = \Delta_{AB} + \int_{x=250}^{500} \frac{(175-0.1x)\,dx}{AE} = 0.00375 + \frac{1}{50\times2\times10^5}\left(175 - \frac{0.1x^2}{2}\right)_{250}^{500} = 0.0072 \text{ mm}$$

(II) FEM by hand calculations.

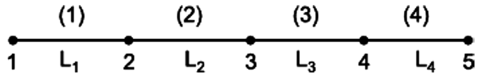


FIGURE 3.7(b) Finite element model for Example 3.2.

$$L_1 = L_2 = L_3 = L_4 = 250 \text{ mm}$$

$$A = A_1 = A_2 = A_3 = A_4 = 50 \text{ mm}^2$$

$$E = E_1 = E_2 = E_3 = E_4 = 2\times10^5 \text{ N/mm}^2$$

Stiffness matrix for elements is,

$$[k_1] = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50\times2\times10^5}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.4\times10^5 \begin{array}{c} \begin{array}{cc} 1 & 2 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{l} \\ 1 \\ 2 \end{array}$$

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50\times2\times10^5}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.4\times10^5 \begin{array}{c} \begin{array}{cc} 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{l} \\ 2 \\ 3 \end{array}$$

$$[k_3] = \frac{A_3E_3}{L_3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50\times2\times10^5}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.4\times10^5 \begin{array}{c} \begin{array}{cc} 3 & 4 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{l} \\ 3 \\ 4 \end{array}$$

$$[k_4] = \frac{A_4E_4}{L_4}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{50\times2\times10^5}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.4\times10^5 \begin{array}{c} \begin{array}{cc} 4 & 5 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{l} \\ 4 \\ 5 \end{array}$$

Nodal load calculation for elements 2 and 3,

$$W_1 = \begin{Bmatrix} \dfrac{wL_2}{2} \\ \dfrac{wL_2}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{0.1\times 250}{2} \\ \dfrac{0.1\times 250}{2} \end{Bmatrix} = \begin{Bmatrix} 12.5 \\ 12.5 \end{Bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$$

$$W_2 = \begin{Bmatrix} \dfrac{wL_3}{2} \\ \dfrac{wL_3}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{0.1\times 250}{2} \\ \dfrac{0.1\times 250}{2} \end{Bmatrix} = \begin{Bmatrix} 12.5 \\ 12.5 \end{Bmatrix} \begin{matrix} 3 \\ 4 \end{matrix}$$

Global equation is,

$$[\mathrm{K}]\,\{r\} = \{\mathrm{R}\} \tag{3.28}$$

$$0.4\times 10^5 \begin{array}{c} \begin{matrix} 1 & \quad 2 & \quad 3 & \quad 4 & \quad 5 \end{matrix} \\ \begin{bmatrix} 1 & 1 & 0 & 0 & 0 \\ -1 & 1+1 & -1 & 0 & 0 \\ 0 & -1 & 1+1 & -1 & 0 \\ 0 & 0 & -1 & 1+1 & -1 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix} \end{array} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \end{matrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 12.5 \\ 12.5+12.5 \\ 12.5 \\ 100 \end{Bmatrix} \tag{3.29}$$

Boundary conditions are at node 1, $u_1 = 0$

By using the elimination method the above matrix reduces to,

$$0.4\times 10^5 \begin{bmatrix} 2 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} 12.5 \\ 25 \\ 12.5 \\ 100 \end{Bmatrix}.$$

By solving the above matrix and equations, we get

$$u_2 = 0.0038 \text{ mm}$$

$$u_3 = 0.0072 \text{ mm}$$

$$u_4 = 0.01 \text{ mm}$$

$$u_5 = 0.0125 \text{ mm}.$$

Stress (σ) calculation

$$\{\sigma_1\} = \frac{E}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\end{Bmatrix} = \frac{2\times10^5}{250}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0\\0.0038\end{Bmatrix} = 3.04\ \text{N/mm}^2$$

$$\{\sigma_2\} = \frac{E}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_2\\u_3\end{Bmatrix} = \frac{2\times10^5}{250}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0.0038\\0.0072\end{Bmatrix} = 2.72\ \text{N/mm}^2$$

$$\{\sigma_3\} = \frac{E}{L_3}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_3\\u_4\end{Bmatrix} = \frac{2\times10^5}{250}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0.0072\\0.01\end{Bmatrix} = 2.24\ \text{N/mm}^2$$

$$\{\sigma_4\} = \frac{E}{L_4}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_4\\u_5\end{Bmatrix} = \frac{2\times10^5}{250}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0.01\\0.0125\end{Bmatrix} = 2\ \text{N/mm}^2$$

Reaction calculation: from equation (3.29)

$$0.4\times10^5\left(u_1 - u_2\right) = R_1$$

$$0.4\times10^5\left(0 - 0.0038\right) = R_1$$

$$R_1 = -152\ \text{N}$$

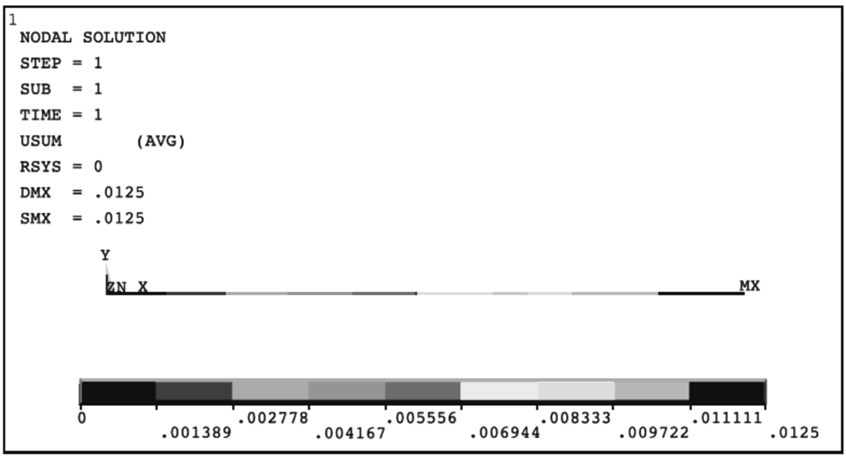


FIGURE 3.7(c) Deflection pattern for a bar (refer to Appendix D for color figures).

Deflection values at nodes *(Computer generated output)*
The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.37500E−02	0.0000	0.0000	0.37500E−02
3	0.71875E−02	0.0000	0.0000	0.71875E−02
4	0.10000E−01	0.0000	0.0000	0.10000E−01
5	0.12500E−01	0.0000	0.0000	0.12500E−01

Maximum absolute value

NODE	5	0	0	5
VALUE	0.12500E−01	0.0000	0.0000	0.12500E−01

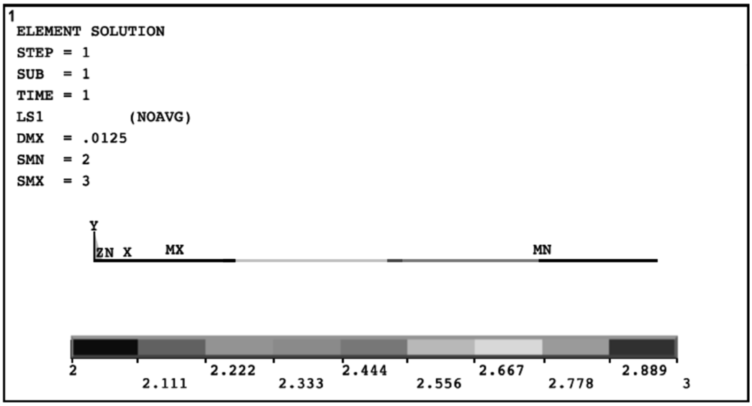


FIGURE 3.7(d) Stress pattern for a bar (refer to Appendix D for color figures).

Stress values at elements *(Computer generated output)*

STAT	CURRENT
ELEM	LS1
1	3.0000
2	2.7500
3	2.2500
4	2.0000

Reaction value *(Computer generated output)*

The following *X*, *Y*, *and* Z SOLUTIONS are in global coordinates

NODE	FX	FY
1	−150.00	0.0000

ANSWERS TO EXAMPLE 3.2

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	0.00375 mm	0.0038 mm	0.00375 mm
Displacement at node 3	0.0072 mm	0.0072 mm	0.00719 mm
Displacement at node 4	0.01 mm	0.01 mm	0.01 mm
Displacement at node 5	0.0125 mm	0.0125 mm	0.0125 mm
Stress in element 1	3 N/mm^2	3.04 N/mm^2	3 N/mm^2
Stress in element 2	3 N/mm^2 to 2.5 N/mm^2	2.72 N/mm^2	2.75 N/mm^2
Stress in element 3	2.5 N/mm^2 to 2 N/mm^2	2.24 N/mm^2	2.25 N/mm^2
Stress in element 4	2 N/mm^2	2 N/mm^2	2 N/mm^2
Reaction at fixed end	−1.50 N	−152 N	−150 N

EXAMPLE 3.3

A and P = 80 kN is applied, as shown in Figure 3.8. Determine the nodal displacements, element stresses, and support reactions in the bar. Take $E = 20 \times 10^3\ \text{N/mm}^2$.

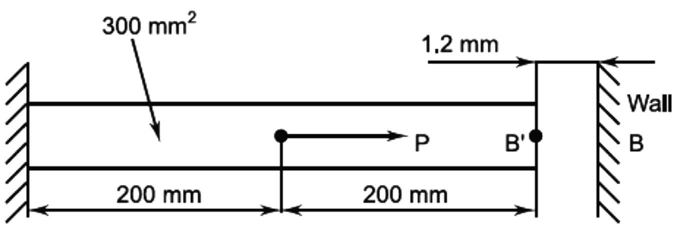


FIGURE 3.8 Example 3.3.

Solution

(I) Analytical method [Refer to Figure 3.8(*a*)].

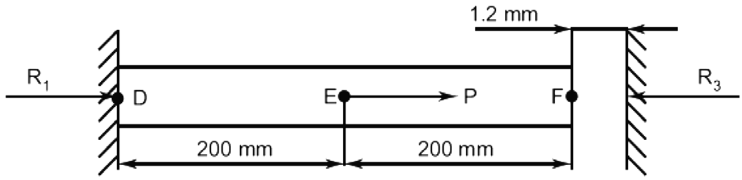


FIGURE 3.8(a) Analytical method for Example 3.3.

Let R_3 be the reaction developed at the wall after contact.

$$\frac{P_{DE}L}{AE}+\frac{P_{EF}L}{AE}=1.2.$$

$$\frac{R_1\times 200}{300\times 20\times 10^3}+\frac{(-R_3)\times 200}{300\times 20\times 10^3}=1.2. \tag{3.30}$$

$$\sum F_x=0 \quad\Rightarrow\quad R_1+R_3=P=80\times 10^3. \tag{3.31}$$

Solving equations (3.30) and (3.31)

$$R_1=58018\ \text{N}$$

$$R_3=21982\ \text{N}$$

Stresses are, $\sigma_{DE}=\dfrac{R_1}{A}=\dfrac{58018}{300}=193.39\ \text{N/mm}^2$

$$\sigma_{EF}=-\frac{R_3}{A}=-\frac{21982}{300}=-73.27\ \text{N/mm}^2.$$

Deflections are, $\delta_2=\delta_E=\Delta_{DE}=\dfrac{\sigma_{DE}L}{E}=\dfrac{193.39\times 200}{20\times 10^3}=1.934\ \text{mm}$

$$\delta_3=1.2\ \text{mm}.$$

(II) FEM by hand calculations

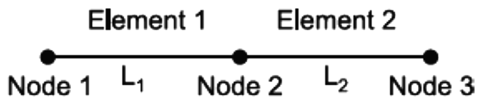


FIGURE 3.8(b) Finite element model for Example 3.3.

$$L_1 = 200 \text{ mm}, \; L_2 = 200\text{mm}$$

First, we should check whether contact occurs between the bar and the wall. For this, assume that the wall does not exist. The solution to the problem is as below. (Consider the two element model.)

Stiffness matrices are,

$$[k_1] = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{300\times 20\times 10^3}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 30\times 10^3 \begin{array}{cc} 1 & 2 \\ \left[\begin{matrix} 1 & -1 \\ -1 & 1 \end{matrix}\right. & \left.\right] \end{array}\begin{array}{c} 1 \\ 2 \end{array}$$

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{300\times 20\times 10^3}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 30\times 10^3 \begin{array}{cc} 2 & 3 \\ \left[\begin{matrix} 1 & -1 \\ -1 & 1 \end{matrix}\right] & \end{array}\begin{array}{c} 2 \\ 3 \end{array}.$$

Global equation is,

$$[\text{K}]\,\{r\} = \{\text{R}\} \tag{3.32}$$

$$30\times 10^3 \begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 80\times 10^3 \\ 0 \end{Bmatrix}.$$

Boundary conditions are at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$30\times 10^3 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 80\times 10^3 \\ 0 \end{Bmatrix}.$$

By matrix multiplication, we get

$$30\times10^3\left(2\times u_2-1\times u_3\right)=80\times10^3 \tag{3.33}$$

$$30\times10^3\left(-1\times u_2+1\times u_3\right)=0 \tag{3.34}$$

By solving equations (3.33) and (3.34), we get, $u_2=2.67$ mm and $u_3=2.67$ mm.

Since displacement at node 3 is 2.67 mm (greater than 1.2 mm), we can say that contact does occur. The problem has to be resolved since the boundary conditions are now different. The displacement at B' is specified to be 1.2 mm, as shown in Figure 3.8.

Global element equation is,

$$[\mathrm{K}]\ \{r\} = \{\mathrm{R}\} \tag{3.35}$$

$$30\times10^3\begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array}\begin{array}{c} \\ 1 \\ 2 \\ 3 \end{array}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}=\begin{Bmatrix} R_1 \\ 80\times10^3 \\ 0 \end{Bmatrix} \tag{3.36}$$

Boundary conditions are at node 1, $u_1 = 0$ and at node 3, $u_2 = 1.2$.

By using the elimination method, the above matrix reduces to,

$$30\times10^3\,[2]\{u_2\}=\left\{80\times10^3\right\}-1.2\left[30\times10^3\times-1\right]$$

$$30\times10^3\times2\times u_2=80\times10^3+36\times10^3$$

$$u_2=1.933 \text{ mm.}$$

Stress (σ) calculation: stress for element 1 is,

$$\{\sigma_1\}=\frac{E}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\end{Bmatrix}=\frac{20\times10^3}{200}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0\\1.933\end{Bmatrix}=193.3 \text{ N/mm}^2.$$

Stress for element 2 is,

$$\{\sigma_2\}=\frac{E}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_2\\u_3\end{Bmatrix}=\frac{20\times10^3}{200}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}1.933\\1.2\end{Bmatrix}=-73.3 \text{ N/mm}^2.$$

Reaction calculation: from equation (3.36)

$$30\times10^3\left(u_1-u_2\right)=R_1$$

$$30\times10^3\left(0-1.933\right)=R_1$$

$$R_1=-57990\text{ N}\left(\text{Direction is leftwards}\right).$$

We know that,

$$R_1+P+R_3=0$$

$$-57990+80\times10^3+R_3=0$$

$$R_3=-22010\text{ N }\left(\text{Direction is leftwards}\right).$$

(III) Software results

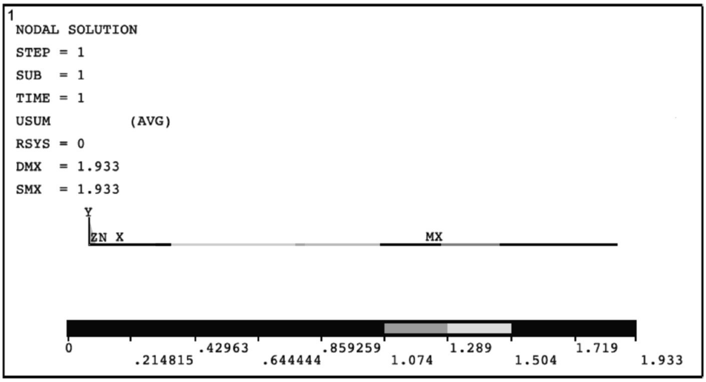


FIGURE 3.8(c) Deflection pattern for a bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	1.9333	0.0000	0.0000	1.9333
3	1.2000	0.0000	0.0000	1.2000

Maximum absolute values

NODE	2	0	0	2
VALUE	1.9333	0.0000	0.0000	1.9333

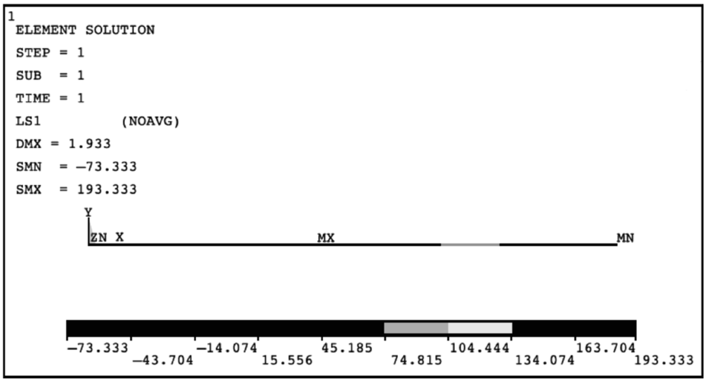


FIGURE 3.8(d) Stress pattern for a bar (refer to Appendix D for color figures).

Stress Values at Elements

STAT ELEM	CURRENT LS1
1	193.33
2	−73.333

Reaction value

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY
1	−58000	0.0000
3	−22000	

ANSWERS TO EXAMPLE 3.3

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	1.934 mm	1.933 mm	1.933 mm
Displacement at node 3	1.2 mm	1.2 mm	1.2 mm
Stress in element 1	193.39 N/mm^2	193.3 N/mm^2	193.33 N/mm^2
Stress in element 2	−73.27 N/mm^2	−73.3 N/mm^2	−73.333 N/mm^2
Reaction at fixed end	58.02 kN	57.94 KN	−58 KN
Reaction at wall	−21.98 kN	−22.01 kN	−22 kN

EXAMPLE 3.4

A bar is subjected to self-weight. Determine the nodal displacement for the bar hanging under its own weight as shown in Figure 3.9. Use two equal length elements. Let $E = 2\times10^{11}$ N/mm^2, mass density $\rho = 7800$ kg/m^3, Area $A = 1000$ mm^2. Consider length of rod L = 2 m.

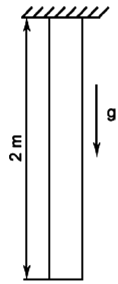


FIGURE 3.9 Bar under self-weight for Example 3.4.

Solution

(I) Analytical method [Refer to Figure 3.9(a)].

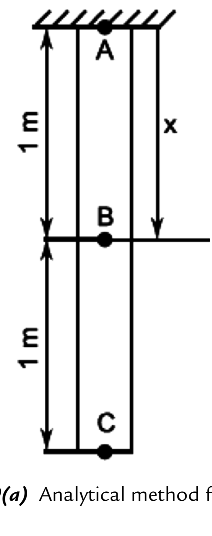


FIGURE 3.9(a) Analytical method for Example 3.4.

$$\delta_3 = \delta_C = \frac{\rho g L^2}{2E} = \frac{7800 \times 9.81 \times (2)^2}{2 \times 2 \times 10^{11}} = 7.6518 \times 10^{-7}\, m$$

$$\delta_2 = \delta_B = \int_1^2 \frac{xA\rho g}{AE} dx = \frac{\rho g}{E}\left(\frac{x^2}{2}\right)_1^2 = \frac{7800 \times 9.81}{2 \times 10^{11}}\left(\frac{(2)^2}{2} - \frac{(1)^2}{2}\right) = 5.7389 \times 10^{-7}\, m$$

(II) FEM by hand calculations

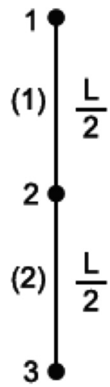


FIGURE 3.9(b) Finite element model for Example 3.4.

$$L_1 = L_2 = \frac{L}{2} = \frac{2}{2} = 1 \text{ m}$$

The element stiffness matrices are,

For element 1,

$$[k_1] = \frac{AE}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{1 \times 10^{-3} \times 2 \times 10^{11}}{1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2 \times 10^8 \begin{array}{c} \begin{array}{cc} 1 & 2 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{c} \\ 1 \\ 2 \end{array}.$$

For element 2,

$$[k_2] = \frac{AE}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{1 \times 10^{-3} \times 2 \times 10^{11}}{1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2 \times 10^8 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{array}{c} 2 \\ 3 \end{array}.$$

Nodal load vector due to weight is,

$$F_1 = \begin{Bmatrix} \dfrac{\rho A g L_1}{2} \\ \dfrac{\rho A g L_1}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{7800 \times 1 \times 10^{-3} \times 9.81 \times 1}{2} \\ \dfrac{7800 \times 1 \times 10^{-3} \times 9.81 \times 1}{2} \end{Bmatrix} = \begin{array}{c} \begin{Bmatrix} 38.26 \\ 38.26 \end{Bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \end{array}$$

$$F_2 = \begin{Bmatrix} \dfrac{\rho A g L_2}{2} \\ \dfrac{\rho A g L_2}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{7800 \times 1 \times 10^{-3} \times 9.81 \times 1}{2} \\ \dfrac{7800 \times 1 \times 10^{-3} \times 9.81 \times 1}{2} \end{Bmatrix} = \begin{Bmatrix} 38.26 \\ 38.26 \end{Bmatrix} \begin{array}{c} 2 \\ 3 \end{array}.$$

Global equation is,

$$[K]\,\{r\} = \{R\} \tag{3.37}$$

$$2 \times 10^8 \begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3 \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 38.26 \\ 76.52 \\ 38.26 \end{Bmatrix} \tag{3.38}$$

Boundary conditions are at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$2 \times 10^8 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 76.52 \\ 38.26 \end{Bmatrix}.$$

By matrix multiplication, we get

$$2 \times 10^8 \left(2 \times u_2 - 1 \times u_3\right) = 76.52 \tag{3.39}$$

$$2 \times 10^8 \left(-1 \times u_2 + 1 \times u_3\right) = 38.26. \tag{3.40}$$

By solving equations (3.39) and (3.40),

we get

$$u_2 = 5.739 \times 10^{-7} \text{ m}$$

$$u_3 = 7.652 \times 10^{-7} \text{ m}.$$

(III) Software results

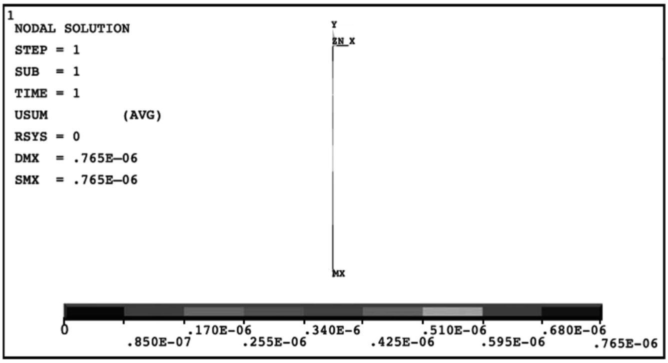


FIGURE 3.9(c) Deflection pattern for a bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.57389E–06	0.0000	0.57389E–06
3	0.0000	0.76518E–06	0.0000	0.76518E–06

NODE	0	3	0	3
VALUE	0.0000	0.76518E–06	0.0000	0.76518E–06

Maximum absolute values

ANSWERS TO EXAMPLE 3.4

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	5.7389×10^{-7} m	5.7389×10^{-7} m	5.7389×10^{-7} m
Displacement at node 3	7.6518×10^{-7} m	7.6518×10^{-7} m	7.6518×10^{-7} m

EXAMPLE 3.5

A rod rotating at a constant angular velocity ω = 45 rad/sec is shown in Figure 3.10. Determine the nodal displacements and stresses in the rod. Consider only the centrifugal force. Ignore the bending of the rod. Use two quadratic elements. Take A = 350 mm^2, E = 70GPa, Mass density $\rho = 7850$ kg/m^3, Length of the rod L = 1 m.

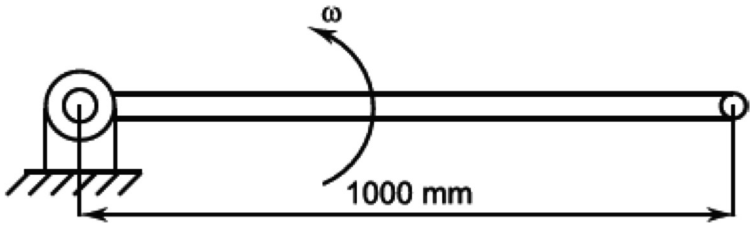


FIGURE 3.10 Rod rotation at a constant angular velocity for Example 3.5.

Solution

(I) Analytical method [Refer to Figure 3.10 (a)**].**

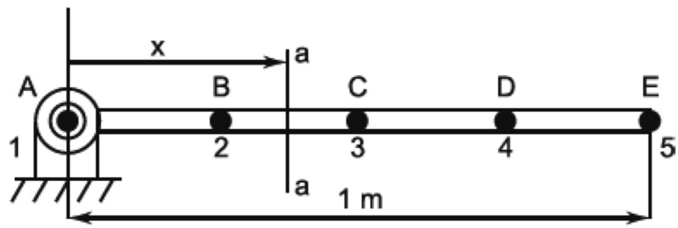


FIGURE 3.10(a) Analytical method for rod rotation at a constant angular velocity for Example 3.5.

$$L = 1 \text{ m}$$

Stress calculation: Stress at section a-a,

$$\sigma = \frac{mr\omega^2}{Area} = \frac{\rho \times A \times (L-x)\left(x + \dfrac{(L-x)}{2}\right)\omega^2}{A} = \frac{\rho\omega^2}{2}\left(L^2 - x^2\right) \qquad (3.41)$$

$$\sigma_2 = \sigma_B = \sigma_{x=\frac{L}{4}} = \frac{\rho\omega^2}{2}\left(L^2 - \frac{L^2}{16}\right) = \frac{7850 \times (45)^2}{2}\left((1)^2 - \frac{(1)^2}{16}\right) = 7.45 \text{ MPa}$$

$$\sigma_3 = \sigma_C = \sigma_{x=\frac{L}{2}} = 5.96 \text{ MPa}$$

$$\sigma_4 = \sigma_D = \sigma_{x=\frac{3L}{4}} = 3.48 \text{ MPa}$$

$$\sigma_1 = \sigma_A = \sigma_{x=0} = 7.95 \text{ MPa.}$$

Displacement at section *a-a* = change in length of *x*,

$$\Delta x = \int_0^x \frac{P_x}{AE} dx = \int_0^x \frac{A\rho\omega^2(L^2 - x^2)}{2AE}\, dx = \frac{\rho\omega^2}{2E}\left(L^2 x - \frac{x^3}{3} \right)_0^x$$

$$\delta_1 = \delta_A = \Delta x_{x=0} = 0$$

$$\delta_2 = \delta_B = \Delta x\Big|_{x=\frac{L}{4}} = \frac{\rho\omega^2}{2E}\left(L^2 x - \frac{x^3}{3} \right)_0^{L/4} = \frac{7850\times(45)^2}{2\times70\times10^9}\left(L^2 \times \frac{L}{4} - \frac{L^3}{64\times3} \right)$$

$$\delta_2 = \frac{7850\times(45)^2}{2\times70\times10^9}\left(\frac{(1)^3}{4} - \frac{(1)^3}{192} \right) = 2.78\times10^{-5} \text{ m} = 0.0278 \text{ mm}$$

$$\delta_3 = \delta_C = \Delta x\Big|_{x=\frac{L}{2}} = 0.052 \text{ mm}$$

$$\delta_4 = \delta_D = \Delta x\Big|_{x=\frac{3L}{4}} = 0.069 \text{ mm}$$

$$\delta_5 = \delta_E = \Delta x\Big|_{x=L} = 0.076 \text{ mm.}$$

Reaction calculation

$$\sum F_x = 0$$

$$R_1 + AL\rho \frac{L}{2}\omega^2 = 0$$

$$\therefore \qquad R_1 = -\frac{AL^2\rho\omega^2}{2} = -\frac{3.5\times10^{-4}\times(1)^2\times7850\times(45)^2}{2} - 2781.84 \text{ N.}$$

(II) FEM by hand calculations.

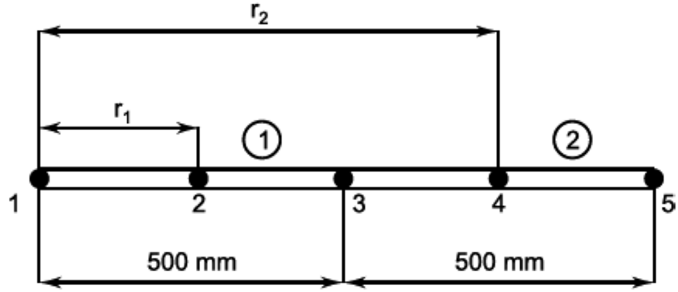


FIGURE 3.10(b) Finite element model for Example 3.5 (with two quadratic elements).

A finite element model of the rod, with two quadratic elements, is shown in Figure 3.10(b). The element stiffness matrices are,

$$L = L_1 = L_2 = 0.5 \text{ m}$$

$$[k_1] = \frac{A_1 E_1}{3L_1}\begin{bmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{bmatrix} = \frac{35\times10^{-4}\times70\times10^{9}}{3\times0.5}\begin{bmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{bmatrix}$$

$$[k_1] = 163.33\times10^{5} \begin{array}{c} \begin{array}{ccc} 1 & 3 & 2 \end{array} \\ \begin{bmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 3 \\ 2 \end{array}$$

$$[k_2] = 163.33\times10^{5} \begin{array}{c} \begin{array}{ccc} 3 & 5 & 4 \end{array} \\ \begin{bmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{bmatrix} \end{array} \begin{array}{c} \\ 3 \\ 5. \\ 4 \end{array}$$

Thus, the global stiffness matrix is,

$$[K] = 163.33\times10^{5} \begin{array}{c} \begin{array}{ccccc} 1 & 2 & 3 & 4 & 5 \end{array} \\ \begin{bmatrix} 7 & -8 & 1 & 0 & 0 \\ -8 & 16 & -8 & 0 & 0 \\ 1 & -8 & 14 & -8 & 1 \\ 0 & 0 & -8 & 16 & -8 \\ 0 & 0 & 1 & -8 & 7 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3. \\ 4 \\ 5 \end{array}$$

The centrifugal force or body force $F_c\left(\text{kg/m}^3\right)$ is given by,

$$F_c = \frac{\rho r \omega^2}{g}. \tag{3.42}$$

Note that F is a function of the distance r from the pin. Taking the average values of F over each element, we have,

$$F_1 = \frac{\rho r_1 \omega^2}{g} = \frac{7850 \times 0.25 \times (45)^2}{9.81} = 405103.2 \text{ kg/m}^3$$

$$F_2 = \frac{\rho r_2 \omega^2}{g} = \frac{7850 \times 0.75 \times (45)^2}{9.81} = 1215309.6 \text{ kg/m}^3.$$

Thus, the element body force vectors are,

$$f_1 = A \times L_1 \times F_1 \begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix} = 3.5 \times 10^{-4} \times 0.5 \times 405103.2 \begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix}$$

$$= 70.89 \begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix} = \begin{array}{c} \begin{Bmatrix} 11.815 \\ 11.815 \\ 47.26 \end{Bmatrix} \end{array} \begin{array}{l} 1 \\ 2 \\ 3 \end{array} \quad \text{Global dof}$$

$$f_2 = A \times L_2 \times F_2 \begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix} = 3.5 \times 10^{-4} \times 0.5 \times 1215309.6 \begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix}$$

$$= 212.68\begin{Bmatrix} \frac{1}{6} \\ \frac{1}{6} \\ \frac{2}{3} \end{Bmatrix} = \begin{array}{c} \begin{Bmatrix} 35.45 \\ 35.45 \\ 141.79 \end{Bmatrix} \end{array} \begin{array}{l} 3 \\ 5 \\ 4 \end{array} \quad \text{Global dof}$$

Assembling f_1 and f_2, we obtain,

$$F = \begin{Bmatrix} 11.815 \\ 47.26 \\ 47.26 \\ 141.79 \\ 35.45 \end{Bmatrix} \times 9.81 = \begin{Bmatrix} 115.91 \\ 463.62 \\ 463.62 \\ 1390.96 \\ 347.76 \end{Bmatrix} \text{N}.$$

The global equation is,

$$[\mathrm{K}]\,\{\mathrm{r}\} = \{\mathrm{R}\} \tag{3.43}$$

$$163.33\times10^5 \begin{bmatrix} 7 & -8 & 1 & 0 & 0 \\ -8 & 16 & -8 & 0 & 0 \\ 1 & -8 & 14 & -8 & 1 \\ 0 & 0 & -8 & 16 & -8 \\ 0 & 0 & 1 & -8 & 7 \end{bmatrix} \begin{array}{l} 1 \\ 2 \\ 3 \\ 4 \\ 5 \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 115.91 \\ 463.62 \\ 463.62 \\ 1390.96 \\ 347.76 \end{Bmatrix} \tag{3.44}$$

Boundary conditions are at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$163.33\times10^5 \begin{bmatrix} 16 & -8 & 0 & 0 \\ -8 & 14 & -8 & 1 \\ 0 & -8 & 16 & -8 \\ 0 & 1 & -8 & 7 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} 463.62 \\ 463.62 \\ 1390.96 \\ 347.76 \end{Bmatrix}.$$

By solving the above matrix and equations, we get

$$u_2 = 2.661\times10^{-5}\ \text{mm} = 0.0266\ \text{mm}$$

$$u_3 = 0.0497\ \text{mm}$$

$$u_4 = 0.0657\ \text{mm}$$

$$u_5 = 0.0709\ \text{mm}.$$

The stress at node 1 in element 1 is given by,

$$\sigma_{11} = \frac{2E}{L_1}\begin{bmatrix}-1.5 & -0.5 & 2\end{bmatrix}\begin{Bmatrix}u_1\\u_3\\u_2\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}-1.5 & -0.5 & 2\end{bmatrix}\begin{Bmatrix}0\\0.0497\\0.0266\end{Bmatrix} = 7.924\ \text{MPa}.$$

The stress at node 2 in element 1 is given by,

$$\sigma_{12} = \frac{2E}{L_1}\begin{bmatrix}-0.5 & 0.5 & 0\end{bmatrix}\begin{Bmatrix}u_1\\u_3\\u_2\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}-0.5 & 0.5 & 0\end{bmatrix}\begin{Bmatrix}0\\0.0497\\0.0266\end{Bmatrix} = 6.972\ \text{MPa}.$$

The stress at node 3 in element 1 is given by,

$$\sigma_{13} = \frac{2E}{L_1}\begin{bmatrix}0.5 & 1.5 & -2\end{bmatrix}\begin{Bmatrix}u_1\\u_3\\u_2\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}0.5 & 1.5 & -2\end{bmatrix}\begin{Bmatrix}0\\0.0497\\0.0266\end{Bmatrix} = 5.992\ \text{MPa}.$$

The stress at node 1 in element 2 is given by,

$$\sigma_{21} = \frac{2E}{L_2}\begin{bmatrix}-1.5 & -0.5 & 2\end{bmatrix}\begin{Bmatrix}u_3\\u_5\\u_4\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}-1.5 & -0.5 & 2\end{bmatrix}\begin{Bmatrix}0.0497\\0.0709\\0.0657\end{Bmatrix} = 5.992\ \text{MPa}.$$

The stress at node 2 in element 2 is given by,

$$\sigma_{22} = \frac{2E}{L_2}\begin{bmatrix}-0.5 & 0.5 & 0\end{bmatrix}\begin{Bmatrix}u_3\\u_5\\u_4\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}-0.5 & 0.5 & 0\end{bmatrix}\begin{Bmatrix}0.0497\\0.0709\\0.0657\end{Bmatrix} = 2.968 \text{ MPa.}$$

The stress at node 3 in element 2 is given by,

$$\sigma_{23} = \frac{2E}{L_2}\begin{bmatrix}0.5 & 1.5 & -2\end{bmatrix}\begin{Bmatrix}u_3\\u_5\\u_4\end{Bmatrix} = \frac{2\times70\times10^3}{500}\begin{bmatrix}0.5 & 1.5 & -2\end{bmatrix}\begin{Bmatrix}0.0497\\0.0709\\0.0657\end{Bmatrix} = -0.056 \text{ MPa.}$$

(III) Software results.

While solving the problem using software, 4 linear bar elements are taken instead of 2 quadratic elements.

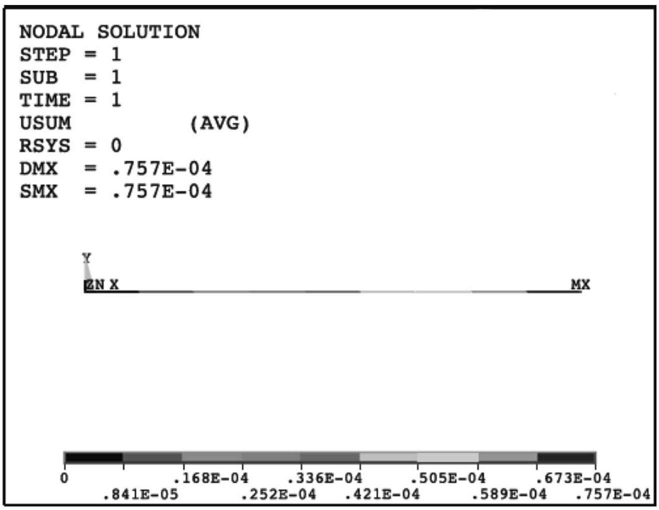


FIGURE 3.10(c) Deflection pattern for a rod (refer to Appendix D for color figures).

Deflection values at nodes (in m)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.27795E–04	0.0000	0.0000	0.27795E–04
3	0.52041E–04	0.0000	0.0000	0.52041E–04
4	0.69191E–04	0.0000	0.0000	0.69191E–04
5	0.75696E–04	0.0000	0.0000	0.75696E–04

Maximum absolute values

NODE	5	0	0	5
VALUE	0.75696E–04	0.0000	0.0000	0.75696E–04

Reaction value

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY
1	–2781.8	–3.3691

ANSWERS OF EXAMPLE 3.5

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	0.0278 mm	0.0266 mm	0.0278 mm
Displacement at node 3	0.052 mm	0.0497 mm	0.052 mm
Displacement at node 4	0.069 mm	0.0657 mm	0.069 mm
Displacement at node 5	0.076 mm	0.0709 mm	0.076 mm
Stress in node 1 of element 1	7.95 MPa	7.924 MPa	–
Stress in node 2 of element 1	7.45 MPa	6.972 MPa	–
Stress in node 3 of element 1	5.96 MPa	5.992 MPa	–
Stress in node 1 of element 2	5.96 MPa	5.992 MPa	–
Stress in node 2 of element 2	3.48 MPa	2.968 MPa	–
Stress in node 3 of element 2	0 MPa	–0.056 MPa	–
Reaction at fixed end	–2781.84 N	–	–2781.8 N

Procedure for solving the problem using ANSYS® 11.0 academic teaching software

Each problem given in this book uses a different procedure for solving using software. For familiarizing, procedure for one problem is given from each chapter using software. Other problems are left to the user to explore the software for solving the problems.

For Example 3.3

Preprocessing

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Structural Link > 2D spar 1 > OK > Close**

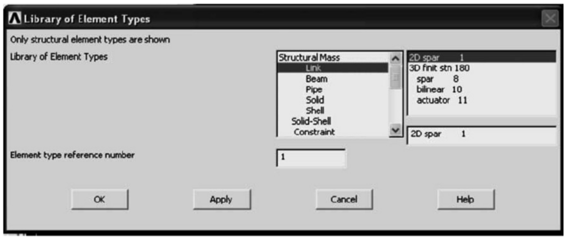


FIGURE 3.11 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

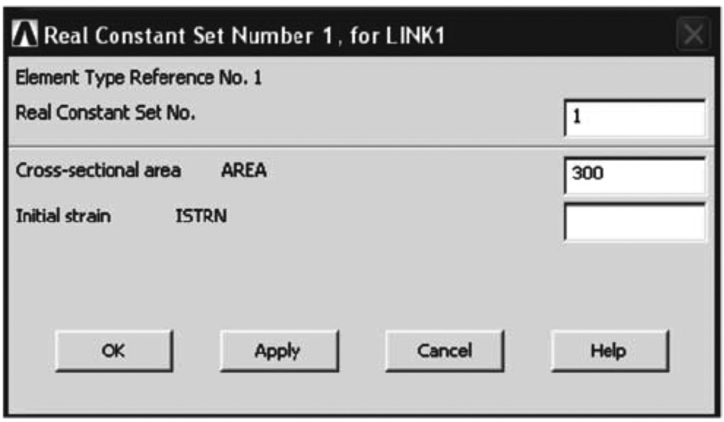


FIGURE 3.12 Enter the cross-sectional area.

Cross-sectional area AREA > **Enter 300 > OK > Close**

Enter the material properties

3. **Main Menu > Preprocessor > Material Props > Material Models** Material Model Number 1, click **Structural > Linear > Elastic > Isotropic** Enter **EX= 2E4 and PRXY=0.3 > OK**
(Close the Define Material Model Behavior window.)
Create the nodes and elements. Create 3 nodes 2 elements.

4. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS**

   Enter the coordinates of node 1 > **Apply**

   Enter the coordinate of node 2 > **Apply**

   Enter the coordinates of node 3 > **OK**

Node locations		
Node number	**X coordinate**	**Y coordinate**
1	**0**	**0**
2	**200**	**0**
3	**400**	**0**

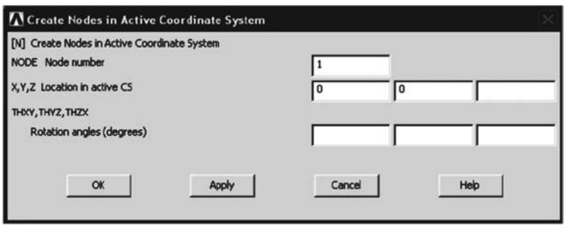


FIGURE 3.13 Enter the node coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru node** Pick the 1st and 2nd node > **Apply** Pick 2nd and 3rd node > **OK**

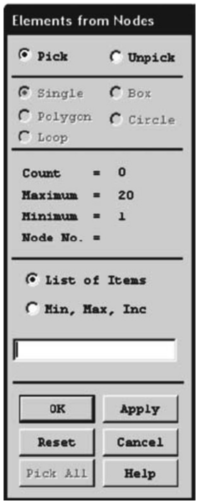


FIGURE 3.14 Pick the nodes to create elements.

Apply the displacement boundary conditions and loads.

6. **Main Menu > Preprocessor > Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st node > **Apply > All DOF=0 > OK**
7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 3rd node > **Apply > Select UX and enter displacement value = 1.2 > OK**
8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes** Pick the 2nd > **OK > Force. Moment value=80e3 > OK**

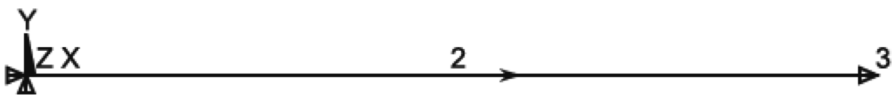


FIGURE 3.15 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First to be safe, save the model.

Solution

The interactive solution proceeds.

9. Main Menu > Solution > Solve > Current LS > OK

The **STATUS Command** window displays the problem parameters and the **Solve Current Load Step** window is shown. Check the solution options in the / **STATUS** window and if all is OK, select **File > Close**

In the **Solve Current Load Step** WINDOW, Select **OK,** and the solution is complete, **close** the **'Solution is Done!'** window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK

The result is shown in Figure 3.8(c).

To find the axial stress, the following procedure is followed.

11. Main Menu > General Postproc > Element Table > Define Table > Add

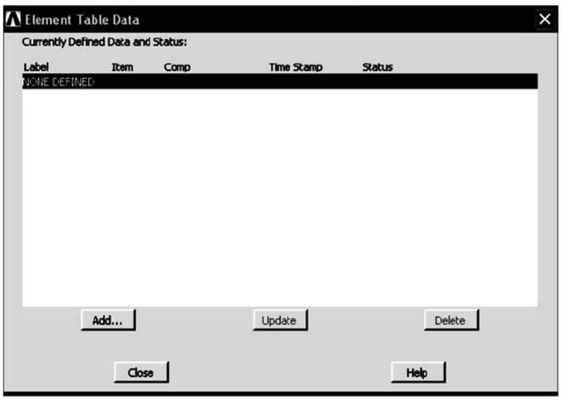


FIGURE 3.16 Define the element table.

Select **By sequence num and LS** and type **1 after LS** as shown in Figure 3.17.

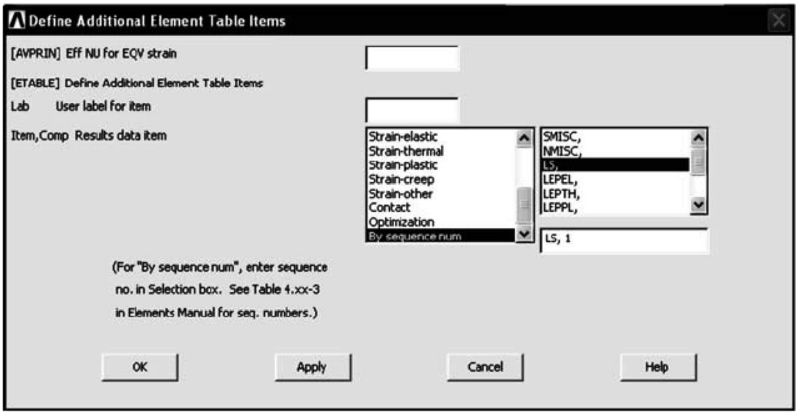


FIGURE 3.17 Selecting options in element table.

OK >Close

12. Main Menu > General Postproc > Plot Results > Contour Plot > Elem Table > Select > LS1 > OK

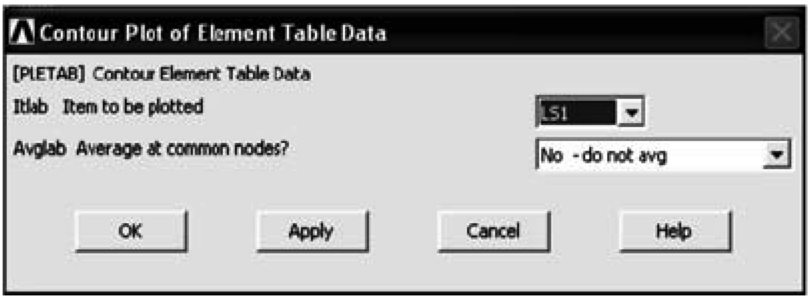


FIGURE 3.18 Selecting options for finding out axial stress.

The result is shown in Figure 3.8(d).

3.3 BARS OF VARYING CROSS-SECTION AREA

This section will demonstrate thorough examples explaining FEA on bars of varying cross-section area.

EXAMPLE 3.6

Solve for displacement and stress given in Figure 3.19 using 2 finite elements model. Take Young's modulus E = 200 GPa.

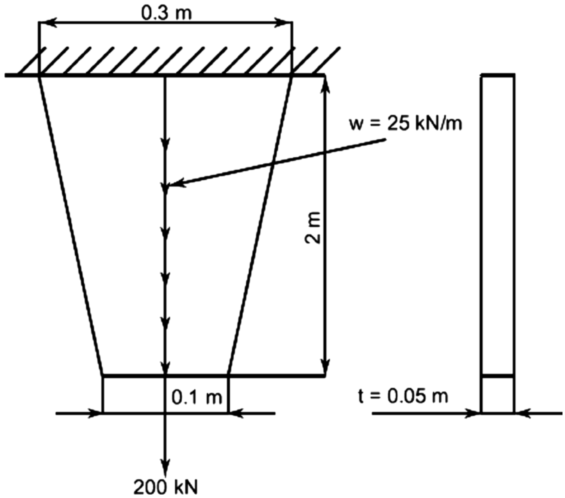


FIGURE 3.19 Example 3.6.

Solution

(I) Analytical method [Refer to Figure 3.19(a)].

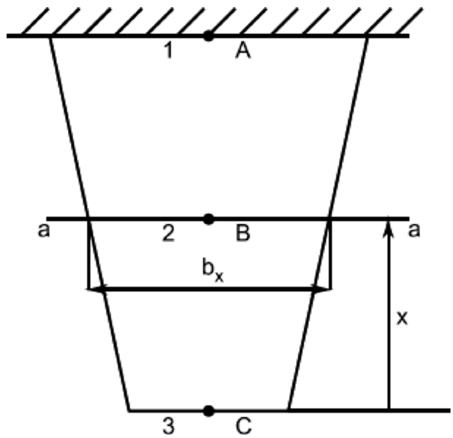


FIGURE 3.19(a) Analytical method for Example 3.6.

$$L = 2\text{ m}.$$

Stress calculation

$$b_x = 0.1 + (0.3 - 0.1)\frac{x}{2} = 0.1 + 0.1x$$

$$A_x = b_x \times t = (0.1 + 0.1x)0.05$$

$$\sigma_x = \frac{P_x}{A_x} = \frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05}$$

$$\sigma_1 = \sigma_A = \sigma_x\big|_{x=2} = \frac{200\times10^3 + 25\times10^3 \times 2}{(0.1+0.1\times2)0.05} = 16.67 \text{ MPa}$$

$$\sigma_2 = \sigma_B = \sigma_x\big|_{x=1} = \frac{200\times10^3 + 25\times10^3 \times 1}{(0.1+0.1\times1)0.05} = 22.5 \text{ MPa}$$

$$\sigma_3 = \sigma_C = \sigma_x\big|_{x=0} = \frac{200\times10^3 + 25\times10^3 \times 0}{(0.1+0.1\times0)0.05} = 40 \text{ MPa.}$$

Displacement Calculation

Displacement at section *a-a* = change in length of (L-x)

$$\delta = \int \frac{\sigma_x}{E} = \int_{L-x}^{L} \frac{P_x}{A_x E}\,dx = \int_{L-x}^{L} \left(\frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05\times200\times10^9} \right) dx$$

$$\delta_3 = \delta_C = \int_{L-2}^{2} \left(\frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05\times200\times10^9} \right) dx = \int_{0}^{2} \left(\frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05\times200\times10^9} \right)$$
$$dx = 0.2423 \text{ mm}$$

$$\delta_2 = \delta_B = \int_{L-1}^{2} \left(\frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05\times200\times10^9} \right) dx = \int_{1}^{2} \left(\frac{200\times10^3 + 25\times10^3 \times x}{(0.1+0.1x)0.05\times200\times10^9} \right)$$
$$dx = 0.096 \text{ mm.}$$

(II) FEM by hand calculations.

Using 2 elements each of 1 m length, we obtain the finite element model as shown in Figure 3.19(c). We can write the equivalent model as shown in Figure 3.19(b). At the middle of the bar width is,

$$\frac{(0.3+0.1)}{2}=0.2 \text{ m}.$$

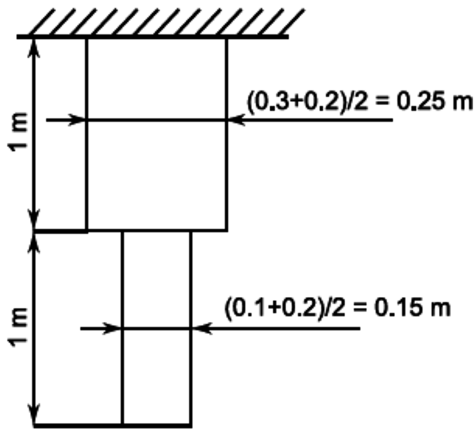


FIGURE 3.19(b) Equivalent model of Finite element model for Example 3.6.

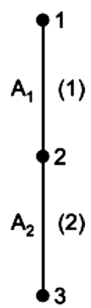


FIGURE 3.19(c) Finite element model for Example 3.6.

$$A_1 = 0.25\times0.05 = 0.0125 \text{ m}^2$$

$$A_2 = 0.15\times0.05 = 0.0075 \text{ m}^2$$

$$E_1 = E_2 = 200\times10^9 \text{ N/m}^2$$

$$L_1 = L_2 = 1 \text{ m}.$$

Stiffness matrix for element 1 is,

$$[k_1] = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{0.0125\times 200\times 10^9}{1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2.5\times 10^9 \begin{array}{c c} 1 & 2 \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & \begin{array}{c} 1 \\ 2 \end{array} \end{array}.$$

Stiffness matrix for element 2 is,

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{0.0075\times 200\times 10^9}{1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 1.5\times 10^9 \begin{array}{c c} 2 & 3 \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & \begin{array}{c} 2 \\ 3 \end{array} \end{array}.$$

Distributes load calculation for elements 1 and 2,

$$W_1 = \begin{Bmatrix} \dfrac{w\times L_1}{2} \\ \dfrac{w\times L_1}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{25\times 1}{2} \\ \dfrac{25\times 1}{2} \end{Bmatrix}\times 10^3 = \begin{Bmatrix} 12.5 \\ 12.5 \end{Bmatrix}\begin{array}{c} 1 \\ 2 \end{array} \times 10^3$$

$$W_2 = \begin{Bmatrix} \dfrac{w\times L_2}{2} \\ \dfrac{w\times L_2}{2} \end{Bmatrix} = \begin{Bmatrix} \dfrac{25\times 1}{2} \\ \dfrac{25\times 1}{2} \end{Bmatrix}\times 10^3 = \begin{Bmatrix} 12.5 \\ 12.5 \end{Bmatrix}\begin{array}{c} 2 \\ 3 \end{array} \times 10^3.$$

Global equation is,

$$[K]\{r\} = \{R\}. \tag{3.45}$$

$$10^9 \begin{array}{c c} \begin{array}{ccc} 1 & 2 & 3 \end{array} & \\ \begin{bmatrix} 2.5 & -2.5 & 0 \\ -2.5 & 2.5+1.5 & -1.5 \\ 0 & -1.5 & 1.5 \end{bmatrix} & \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 12.5+R_1 \\ 25 \\ 12.5+200 \end{Bmatrix}\times 10^3. \tag{3.46}$$

Boundary conditions are at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$10^9 \begin{bmatrix} 4 & -1.5 \\ -1.5 & 1.5 \end{bmatrix}\begin{Bmatrix} 25 \\ 212.5 \end{Bmatrix}\times 10^3.$$

By matrix multiplication, we get

$$10^9\left(4\times u_2-1.5\times u_3\right)=25\times10^3 \tag{3.47}$$

$$10^9\left(-1.5\times u_2+1.5\times u_3\right)=212.5\times10^3. \tag{3.48}$$

By solving equations (3.47) and (3.48), we get

$$u_2=9.5\times10^{-5}\ \text{m}=0.095\ \text{mm}$$

$$u_3=2.37\times10^{-4}\ \text{m}=0.237\ \text{mm}.$$

Stress (σ) calculation
Stress in element 1,

$$\{\sigma_1\}=\frac{E_1}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\end{Bmatrix}=\frac{2\times10^5}{1000}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0\\0.095\end{Bmatrix}=19\ \text{MPa}.$$

Stress in element 2,

$$\{\sigma_2\}=\frac{E_2}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_2\\u_3\end{Bmatrix}=\frac{2\times10^5}{1000}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0.095\\0.237\end{Bmatrix}=28.4\ \text{MPa}.$$

(III) Software results.

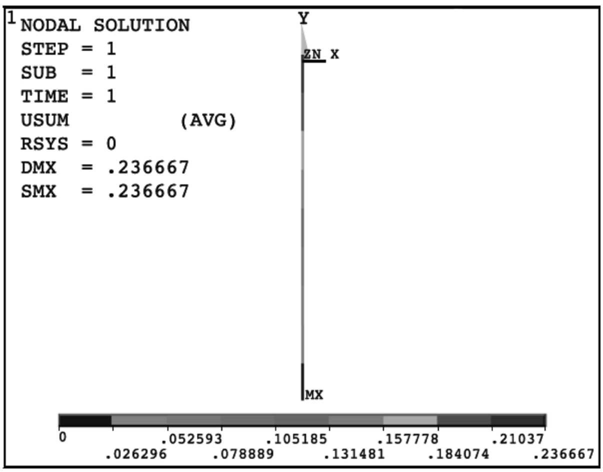


FIGURE 3.19(d) Deflection pattern for a tapered bar (refer to Appendix D for color figures).

Deflection values at node

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	−0.95000E−01	0.0000	0.95000E−01
3	0.0000	−0.23667	0.0000	0.23667

Maximum absolute values

NODE	0	3	0	3
VALUE	0.0000	−0.23667	0.0000	0.23667

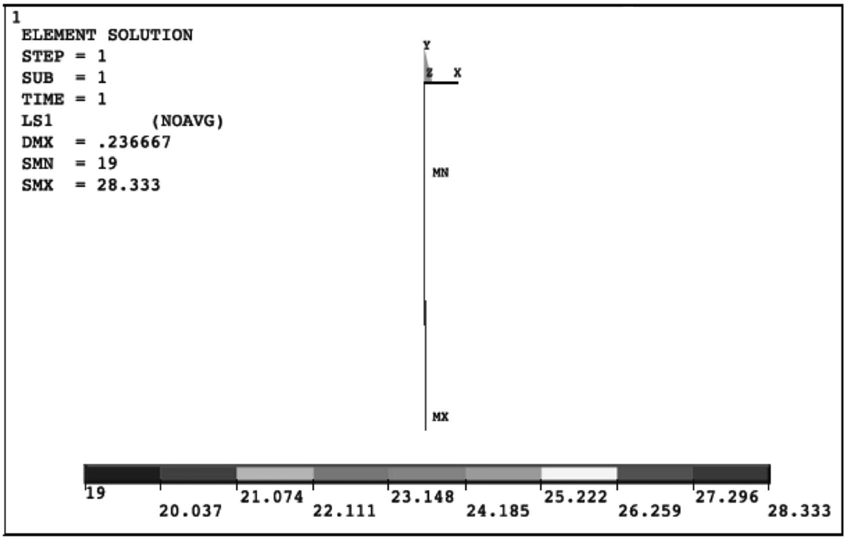


FIGURE 3.19(e) Stress pattern for a tapered bar (refer to Appendix D for color figures).

Stress values at elements

STAT	CURRENT
ELEM	LS1
3	19.000
4	28.333

Reaction value

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY
1	0.0000	0.25000E+06

ANSWERS OF EXAMPLE 3.6

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	0.096 mm	0.095 mm	0.095 mm
Displacement at node 3	0.2423 mm	0.237 mm	0.23667 mm
Stress in element 1	16.67 MPa to 22.5 MPa	19 MPa	19 MPa
Stress in element 2	22.5 MPa to 40 MPa	28.4 MPa	28.33 MPa

In the above example, 2 elements are used for solving the problem by hand calculation and by software. To get the convergence of the solution with the analytical method a higher number of elements are to be used.

EXAMPLE 3.7

Find the displacement and stress distribution in the tapered bar shown in Figure 3.20 using 2 finite elements under an axial load of P =100 N.

Cross-sectional area at fixed end = 22 mm^2

Cross-sectional are at free end =100 mm^2

Young's modulus E = 200 GPa

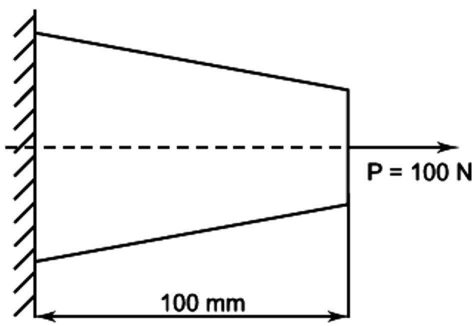


FIGURE 3.20 Example 3.7.

Solution

(I) Analytical method [Refer to Figure 3.20(a)]**.**

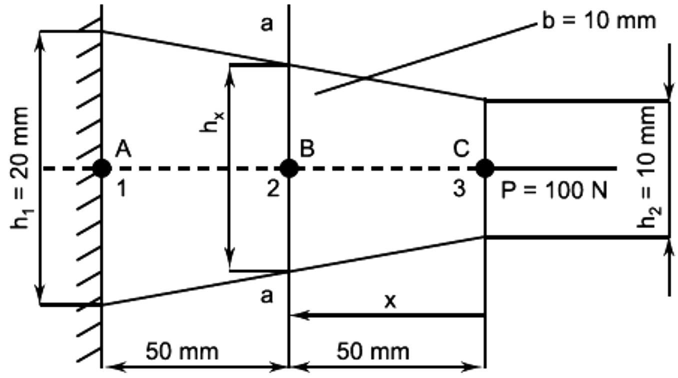


FIGURE 3.20(a) Analytical method for Example 3.7.

Assume, b= thickness = 10 mm

Area at section a-a = $b \times h_x$

$$h_x = \left(h_2 + \left(\frac{h_1 - h_2}{L} \right) x \right)$$

$$A_x = b \times h_x = 10\left(10 + \left(\frac{20-10}{100} \right) x \right) = 10(10 + 0.1x).$$

Stress calculation

$$\sigma_x = \frac{P}{A_x} = \frac{100}{10(10+0.1x)}$$

$$\sigma_1 = \sigma_{x=100} = \frac{100}{10(10+0.1\times 100)} = 0.5 \text{ MPa}$$

$$\sigma_2 = \sigma_{x=50} = \frac{100}{10(10+0.1\times 50)} = 0.667 \text{ MPa}$$

$$\sigma_3 = \sigma_{x=0} = \frac{100}{10(10+0.1\times 0)} = 1 \text{ MPa.}$$

Displacement calculation

$$\delta_3 = \delta_C = \frac{PL}{Eb(h_1 - h_2)} \ln\frac{h_1}{h_2} = \frac{100 \times 100}{2 \times 10^5 \times 10(20-10)} \ln\frac{20}{10} = 3.47 \times 10^{-4} \text{ mm}$$

$$\delta_2 = \delta_B = \frac{100 \times 50}{2 \times 10^5 \times 10(20-15)} \ln\frac{20}{15} = 1.44 \times 10^{-4} \text{ mm.}$$

(II) FEM by hand calculations.

Using 2 elements, each 50 mm in length, we obtain the finite element model as shown in Figure 3.20(c). We can write the equivalent model as shown in Figure 3.20(b) in the middle, area of cross-section of bar is $\frac{(200+100)}{2} = 150 \text{ mm}^2$.

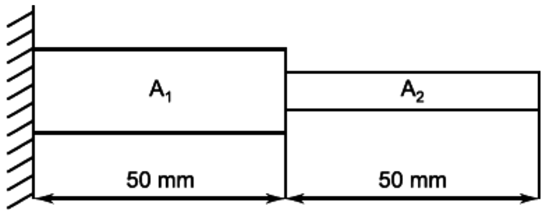


FIGURE 3.20(b) Equivalent model of finite element model for Example 3.7.

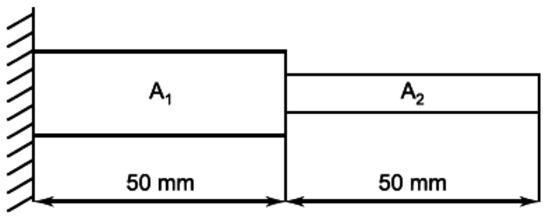


FIGURE 3.20(c) Finite element model for Example 3.7.

$$A_1 = \frac{(200+150)}{2} = 175 \text{ mm}^2$$

$$A_2 = \frac{(150+100)}{2} = 125 \text{ mm}^2$$

$$L_1 = L_2 = 50 \text{ mm}$$

$$E_1 = E_2 = 2 \times 10^5 \text{ N/mm}^2.$$

Stiffness matrix for element 1 is,

$$[k_1]=\frac{A_1E_1}{L_1}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{0.175\times 2\times 10^5}{50}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=7\times 10^5\begin{array}{c}\begin{array}{cc}1 & 2\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\end{array}.$$

Stiffness matrix for element 2 is,

$$[k_2]=\frac{A_2E_2}{L_2}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{0.125\times 2\times 10^5}{50}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=5\times 10^5\begin{array}{c}\begin{array}{cc}2 & 3\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 2\\ 3\end{array}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{3.49}$$

$$10^5\begin{array}{c}\begin{array}{ccc}1 & 2 & 3\end{array}\\ \begin{bmatrix}7 & -7 & 0\\ -7 & 7+5 & -5\\ 0 & -5 & 5\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\\ 3\end{array}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}R_1\\ 0\\ 100\end{Bmatrix}. \tag{3.50}$$

Boundary conditions are at node 1, $u_1 = 0$.
By using the elimination method, the above matrix reduces to,

$$10^5\begin{bmatrix}12 & -5\\ -5 & 5\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\end{Bmatrix}=\begin{Bmatrix}0\\ 100\end{Bmatrix}.$$

By matrix multiplication, we get

$$10^5\left(12\times u_2-5\times u_3\right)=0 \tag{3.51}$$

$$10^5\left(-5\times u_2+5\times u_3\right)=100. \tag{3.52}$$

By solving equations (3.51) and (3.52), we get

$$u_2=1.429\times 10^{-4}\ \text{mm}$$

$$u_3=3.429\times 10^{-4}\ \text{mm}.$$

Stress calculation

Stress in element 1,

$$\{\sigma_1\} = \frac{E_1}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\end{Bmatrix} = \frac{2\times10^5}{50}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}0\\1.429\times10^{-4}\end{Bmatrix} = 0.5716 \text{ MPa.}$$

Stress in element 2,

$$\{\sigma_2\} = \frac{E_2}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_2\\u_3\end{Bmatrix} = \frac{2\times10^5}{50}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}1.429\times10^{-4}\\3.429\times10^{-4}\end{Bmatrix} = 0.8 \text{ MPa.}$$

(III) Software results.

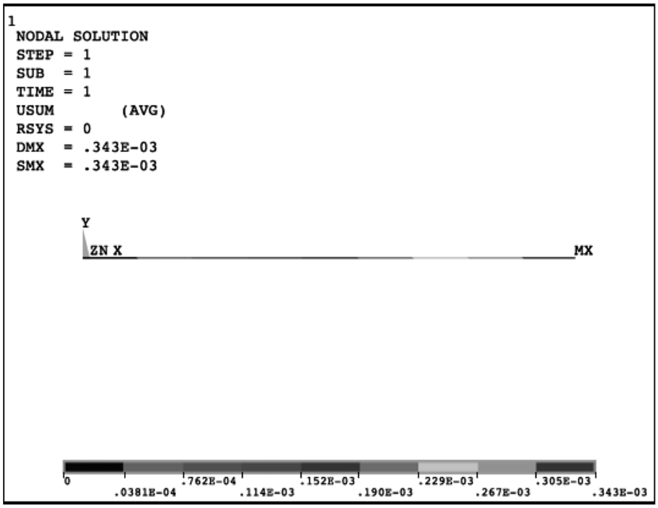


FIGURE 3.20(d) Deflection pattern for a tapered bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.14286E−03	0.0000	0.0000	0.14286E−03
3	0.34286E−03	0.0000	0.0000	0.34286E−03

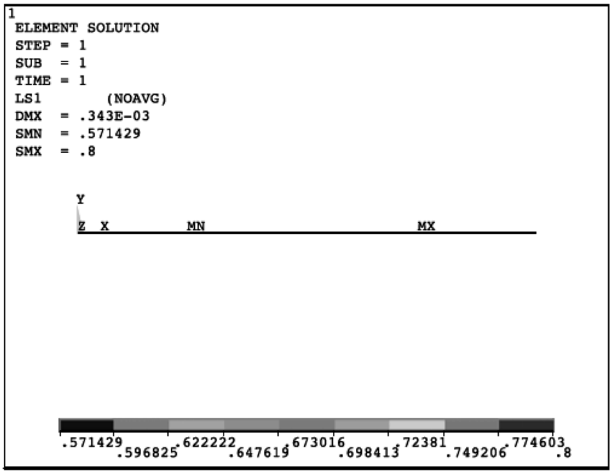


FIGURE 3.20(e) Stress pattern for a tapered bar (refer to Appendix D for color figures).

Stress values at elements

STAT	CURRENT
ELEM	LS1
1	0.57143
2	0.80000

ANSWER FOR EXAMPLE 3.7

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	1.44×10^{-4} mm	1.429×10^{-4} mm	1.4286×10^{-4} mm
Displacement at node 3	3.47×10^{-4} mm	3.429×10^{-4} mm	3.4286×10^{-4} mm
Stress in element 1	0.5 MPa to 0.667 MPa	0.5716 MPa	0.57143 MPa
Stress in element 2	0.667 MPa to 1 MPa	0.8 MPa	0.8 MPa

EXAMPLE 3.8

Find the nodal displacements, element stresses, and reaction in the tapered bar subjected to a load of 6000 N as shown in Figure 3.21. Further the member experiences a temperature increase of 30°C. Use three equal length elements for finite element model. Take E = 200 GPa, υ = 0.3, and $\alpha = 7\times10^{-6} / {}^{o}C$.

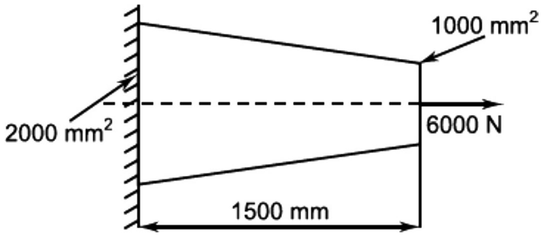


FIGURE 3.21 Example 3.8.

Solution

(I) FEM by hand calculations.

We obtain the finite element model as shown in Figure 3.21(b). We can write the equivalent model as shown in Figure 3.21(a)

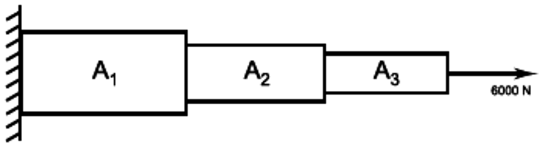


FIGURE 3.21(a) Equivalent model of the finite element model for Example 3.8.

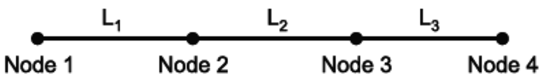


FIGURE 3.21(b) Finite element model for Example 3.8.

$$L_1 = L_2 = L_3 = 500 \text{ mm}$$

$$\Delta T = 30^o C$$

$$A_1 = 2000 \text{ mm}^2$$

$$A_2 = \frac{2000 + 1000}{2} = 1500 \text{ mm}^2$$

$$A_3 = 1000 \text{ mm}^2.$$

Element stiffness matrices are,

$$[k_1]=\frac{A_1E}{L_1}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{2000\times 2\times 10^5}{500}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=8\times 10^5\begin{array}{c}\begin{array}{cc}1 & \;\;2\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\end{array}$$

$$[k_2]=\frac{A_2E}{L_2}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{1500\times 2\times 10^5}{500}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=6\times 10^5\begin{array}{c}\begin{array}{cc}2 & \;\;3\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 2\\ 3\end{array}$$

$$[k_3]=\frac{A_3E}{L_3}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{1000\times 2\times 10^5}{500}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=4\times 10^5\begin{array}{c}\begin{array}{cc}3 & \;\;34\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 3\\ 4\end{array}.$$

Nodal loads due to thermal effect are,

$$\{Q_{1(th)}\}=EA_1\alpha\,(\Delta T)\begin{Bmatrix}-1\\ 1\end{Bmatrix}=2\times 10^5\times 2000\times 7\times 10^{-6}\times 30\begin{Bmatrix}-1\\ 1\end{Bmatrix}=84\times 10^3\begin{Bmatrix}-1\\ 1\end{Bmatrix}\begin{array}{c}1\\ 2\end{array}$$

$$\{Q_{2(th)}\}=EA_2\alpha\,(\Delta T)\begin{Bmatrix}-1\\ 1\end{Bmatrix}=2\times 10^5\times 1500\times 7\times 10^{-6}\times 30\begin{Bmatrix}-1\\ 1\end{Bmatrix}=63\times 10^3\begin{Bmatrix}-1\\ 1\end{Bmatrix}\begin{array}{c}2\\ 3\end{array}$$

$$\{Q_{3(th)}\}=EA_3\alpha\,(\Delta T)\begin{Bmatrix}-1\\ 1\end{Bmatrix}=2\times 10^5\times 1000\times 7\times 10^{-6}\times 30\begin{Bmatrix}-1\\ 1\end{Bmatrix}=42\times 10^3\begin{Bmatrix}-1\\ 1\end{Bmatrix}\begin{array}{c}3\\ 4\end{array}.$$

Global forced vector

$$\{R\}=\begin{Bmatrix}-84\times 10^3\\ 84\times 10^3-63\times 10^3\\ 63\times 10^3-42\times 10^3\\ 42\times 10^3\end{Bmatrix}\begin{array}{c}1\\ 2\\ 3\\ 4\end{array}=\begin{Bmatrix}-84\times 10^3\\ 21\times 10^3\\ 21\times 10^3\\ 42\times 10^3\end{Bmatrix}\begin{array}{c}1\\ 2\\ 3\\ 4\end{array}.$$

Global equation is,

$$10^5\begin{array}{c}\begin{array}{cccc}1 & \quad 2 & \quad 3 & \quad 4\end{array}\\ \begin{bmatrix}8 & -8 & 0 & 0\\ -8 & 8+6 & -6 & 0\\ 0 & -6 & 6+4 & -4\\ 0 & 0 & -4 & 4\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\\ 3\\ 4\end{array}\begin{Bmatrix}u_1\\ u_2\\ u_3\\ u_4\end{Bmatrix}=\begin{Bmatrix}-84\times 10^3+R_1\\ 21\times 10^3\\ 21\times 10^3\\ 42\times 10^3+6000\end{Bmatrix}\begin{array}{c}1\\ 2\\ 3\\ 4\end{array}. \qquad (3.53)$$

Using the elimination method of applying boundary conditions, i.e., $u_1 = 0$. The equation (3.53) reduces to,

$$10^5 \begin{bmatrix} 14 & -6 & 0 \\ -6 & 10 & -4 \\ 0 & -4 & 4 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 21\times10^3 \\ 21\times10^3 \\ 48\times10^3 \end{Bmatrix}.$$

Solving the above matrix and equations, we get

$$u_2 = 0.1125 \text{ mm}$$

$$u_3 = 0.2275 \text{ mm}$$

$$u_4 = 0.3475 \text{ mm}.$$

Stress calculation

$$\sigma_1 = \frac{E}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} - E\alpha\,(\Delta T) = \frac{2\times10^5}{500}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} 0 \\ 0.1125 \end{Bmatrix}$$
$$-2\times10^5\times7\times10^{-6}\times30 = 3 \text{ MPa}$$

$$\sigma_2 = \frac{E}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} - E\alpha\,(\Delta T) = \frac{2\times10^5}{500}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} 0.1125 \\ 0.2275 \end{Bmatrix}$$
$$-2\times10^5\times7\times10^{-6}\times30 = 4 \text{ MPa}$$

$$\sigma_3 = \frac{E}{L_3}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} u_3 \\ u_4 \end{Bmatrix} - E\alpha\,(\Delta T) = \frac{2\times10^5}{500}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} 0.2275 \\ 0.3475 \end{Bmatrix}$$
$$-2\times10^5\times7\times10^{-6}\times30 = 6 \text{ MPa}.$$

Reaction calculation: from equation (3.53),

$$8\times10^5 u_1 - 8\times10^5 u_2 = -84\times10^3 + R_1$$

$$R_1 = -6000 \text{ N}.$$

(III) Software results.

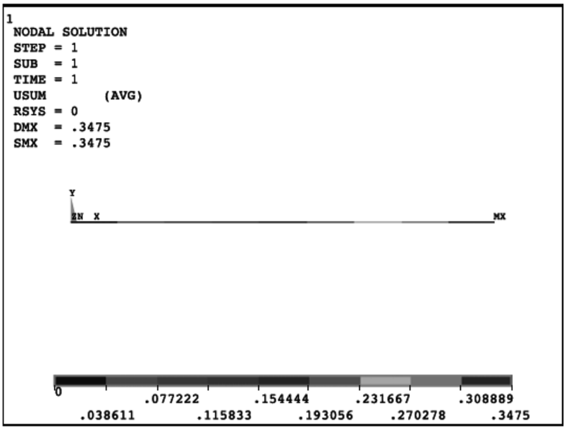


FIGURE 3.21(c) Deflection pattern for a tapered bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.11250	0.0000	0.0000	0.11250
3	0.22750	0.0000	0.0000	0.22750
4	0.34750	0.0000	0.0000	0.34750

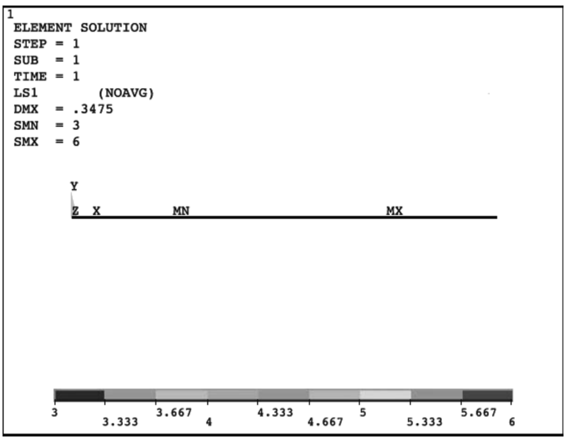


FIGURE 3.21(d) Stress pattern for a tapered bar (refer to Appendix D for color figures).

Stress values at elements

STAT	CURRENT
ELEM	LS1
1	3.0000
2	4.0000
3	6.0000

Reaction value

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY
1	−6000.0	0.0000

ANSWERS FOR EXAMPLE 3.8

Parameter	FEM-hand calculations	Software results
Displacement at node 2	0.1125 mm	0.1125 mm
Displacement at node 3	0.2275 mm	0.2275 mm
Displacement at node 4	0.3475 mm	0.3475 mm
Stress in node 1 of element 1	3 MPa	3 MPa
Stress in node 2 of element 1	4 MPa	4 MPa
Stress in node 3 of element 1	6 MPa	6 MPa
Reaction at fixed end	−6000 N	−6000 N

Procedure for solving the problem using ANSYS® 11.0 academic teaching software

For Example 3.6

PROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit > Delete > Add > Structural Link > 2D spar 1 > OK > Close**

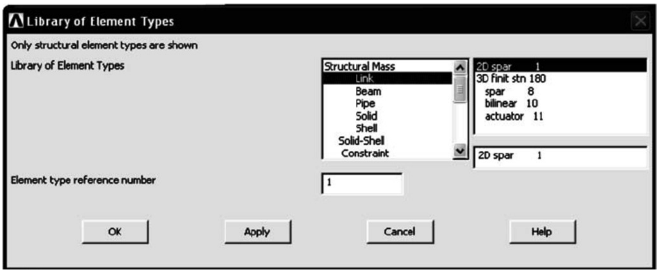


FIGURE 3.22 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

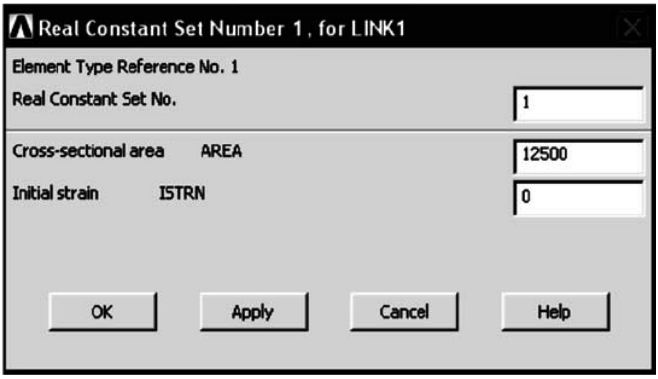


FIGURE 3.23 Enter the cross-sectional area of 1st element.

Cross-sectional area > Enter 12500 > OK > Add > OK

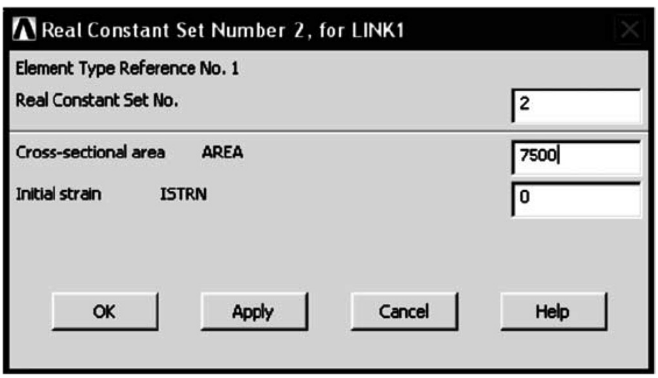


FIGURE 3.24 Enter the cross-sectional area of 2nd element.

Cross-sectional area AREA > Enter 7500 > OK > Close

Enter the material properties.

3. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

   Enter **EX=200E3 and PRXY=0.3 > OK**

   (**Close** the Define Material Model Behavior window.)

   Create the nodes and element. As stated in the example, use 2 element model.

   Hence create 3 nodes and 2 elements.

4. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply** Enter the coordinates of node 2 > **Apply** Enter the coordinates of node 3 > **OK.**

Node locations		
Node number	**X coordinate**	**Y coordinate**
1	0	0
2	0	−1000
3	0	−2000

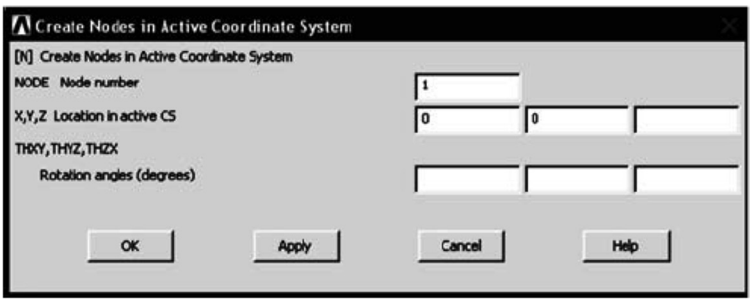


FIGURE 3.25 Enter the node coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes > OK > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **OK**

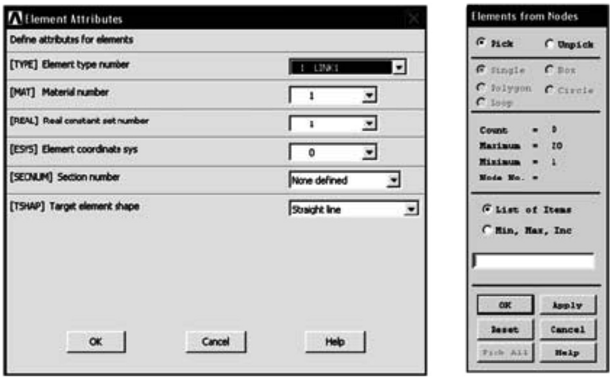


FIGURE 3.26 Assigning element attributes to element 1 and creating element 1.

Elem Attributes > change the Real constant set number to 2 > OK > Auto Numbered > Thru nodes Pick the 2nd and 3rd node > **OK**

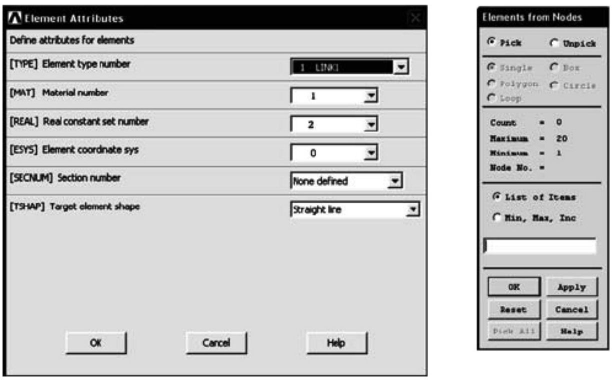


FIGURE 3.27 Assigning element attributes to element 2 and creating element 2.

Apply the displacement boundary conditions and loads.

6. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st node > **Apply > All DOF=0 > OK**

7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes** Pick the 2nd node > **OK > Force/Moment value=-25e3 in FY direction > OK > Force/Moment > On Nodes** Pick the 1st node > **OK > Force/Moment value=-12.5e3 in FY direction > OK**

FIGURE 3.28 Model with loading and displacement boundary conditions.

The mode-building step is now complete, and we can proceed to the solution. First to be safe, save the model.

Solution. The interactive solution proceeds.

8. **Main Menu > Solution > Solve > Current LS > OK** The/ **STATUS Command** window displays the problem parameters and the **Solve Current Load Step** window. Select **OK,** and when the solution is complete, close the **'Solution is DONE!'** window.

   POSTPROCESSING

   We can now plot the results of this analysis and also list the computed values.

9. **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK**

   This result is shown in Figure 3.16(d).

   To find the axial stress, the following procedure is followed.

   10. **Main Menu > General Postproc > Element Table > Define Table > Add**

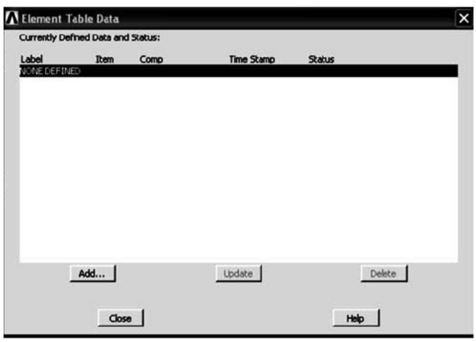


FIGURE 3.29 Defining the element table.

Select **By sequence num LS** and type **1 after LS** as shown in Figure 3.27.

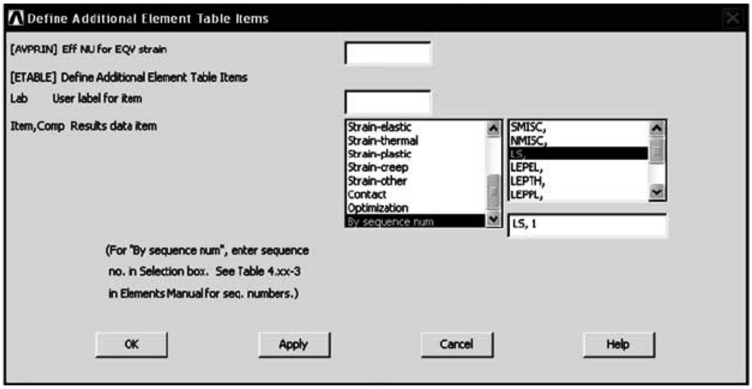


FIGURE 3.30 Selecting options in the element table.

OK

11. Main Menu > General PostProc > Plot Results > Contour Plot > Elem Table > Select **LS1 > OK**

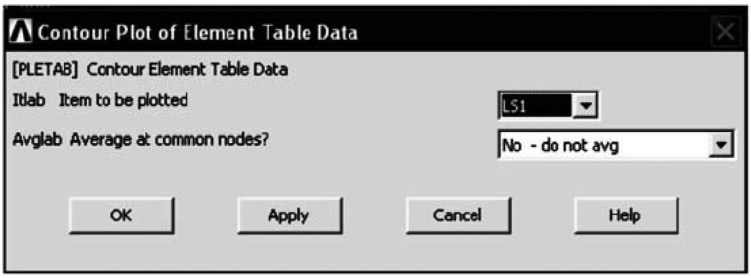


FIGURE 3.31 Selecting options for finding out axial stress.

This result is shown in Figure 3.19(e).

3.4 STEPPED BAR

This section will demonstrate examples on stepped bar using FEA.

EXAMPLE 3.9

Find the nodal displacements, stresses in each element, and reaction at the fixed end for the Figure 3.32 shown below. Take, A_2 = 100 mm^2, and E_1 = E_2 = 200 GPa.

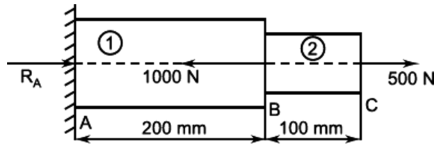


FIGURE 3.32 Example 3.9.

Solution

(I) Analytical method [Refer to Figure 3.32].

Displacement calculation

$$d_C = D_{AC} = D_{AB} + D_{BC} = \frac{-P_{AB}L_{AB}}{A_{AB}E} + \frac{P_{BC}L_{BC}}{A_{BC}E} = \frac{-500\times 200}{200\times 2\times 10^5} + \frac{500\times 100}{200\times 2\times 10^5}$$

$$d_C = -2.5\times 10^{-3} + 2.5\times 10^{-3} = 0$$

$$d_B = D_{AB} = \frac{-P_{AB}L_{AB}}{A_{AB}E} = \frac{-500\times 200}{200\times 2\times 10^5} = -2.5\times 10^{-3}\ \text{mm.}$$

Stress calculation

$$\sigma_{AB} = \frac{P_{AB}}{A_{AB}} = \frac{-500}{200} = -2.5\ \text{MPa (Compressive)}$$

$$\sigma_{BC} = \frac{P_{BC}}{A_{BC}} = \frac{500}{100} = 5\ \text{MPa (Tensile).}$$

Reaction calculation

$$\sum F_x = 0$$

$$R_A - 1000 + 500 = 0$$

$$R_A = 500\ \text{N.}$$

(II) FEM by hand calculations.

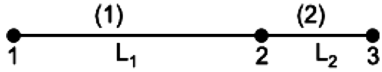


FIGURE 3.32(a) Finite element model for Example 3.9.

$$L_1 = 200 \text{ mm}$$

$$L_2 = 100 \text{ mm.}$$

Displacement calculation

Stiffness matrices for elements 1 and 2 are,

$$[k_1] = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{200\times 2\times 10^5}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2\times 10^5 \begin{array}{c} \begin{matrix} 1 & \;\;2\;\; \end{matrix} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{matrix} \\ 1 \\ 2 \end{matrix}$$

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{100\times 2\times 10^5}{100}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2\times 10^5 \begin{array}{c} \begin{matrix} 2 & \;\;3\;\; \end{matrix} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{matrix} \\ 2 \\ 3 \end{matrix}.$$

Global equation is,

$$2\times 10^5 \begin{array}{c} \begin{matrix} 1 & \quad 2 & \quad 3 \end{matrix} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array}\begin{matrix} \\ 1 \\ 2 \\ 3 \end{matrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ -1000 \\ 500 \end{Bmatrix}. \tag{3.54}$$

Using the elimination method and applying boundary conditions at node 1, $u_1 = 0$.

The equation (3.54) reduces to

$$2\times 10^5 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} -1000 \\ 500 \end{Bmatrix}.$$

By solving the above matrix and equations,

we get,

$$u_2 = 2.5 \times 10^{-3} \text{ mm}$$
$$u_3 = 0.$$

Stress calculations

$$\sigma_1 = \frac{E_1}{L_1}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \frac{2\times 10^5}{200}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ -2.5\times 10^{-3} \end{Bmatrix} = -2.5 \text{ MPa (Compressive)}$$

$$\sigma_2 = \frac{E_2}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_2\\u_3\end{Bmatrix} = \frac{2\times10^5}{100}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}-2.5\times10^{-3}\\0\end{Bmatrix} = 5 \text{ MPa (Tensile)}.$$

Reaction calculation

From equation (i)

$$2\times10^5\left(u_1 - u_2\right) = R_1$$

$$2\times10^5\left(0-\left(-2.5\times10^{-3}\right)\right) = R_1$$

$$R_1 = 500 \text{ N}.$$

(III) Software results.

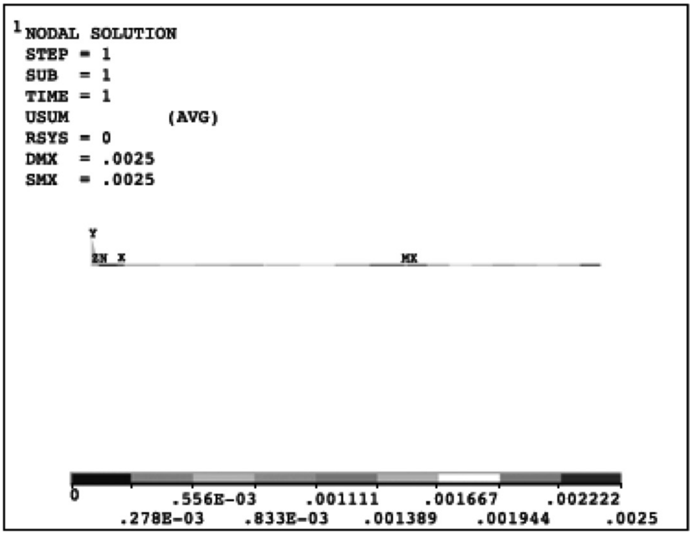


FIGURE 3.32(b) Deflection pattern for a stepped bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	−0.25000E−02	0.0000	0.0000	0.25000E−02
3	0.0000	0.0000	0.0000	0.0000

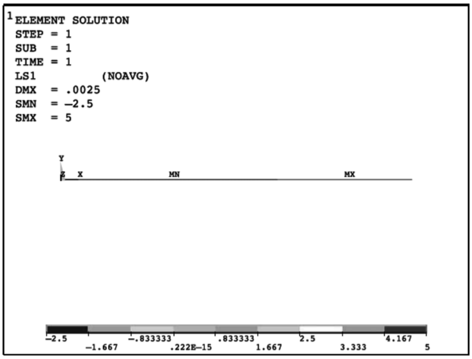


FIGURE 3.32(c) Stress for a stepped bar (refer to Appendix D for color figures).

Stress value at elements

STAT	CURRENT
ELEM	**LS1**
1	−2.5000
3	5.0000

Reaction value

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY
1	500.00	0.0000

ANSWERS FOR EXAMPLE 3.9

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	-2.5×10^{-3} mm	-2.5×10^{-3} mm	-2.5×10^{-2} mm
Displacement at node 3	0	0	0
Stress in element 1	−2.5 MPa	−2.5 MPa	−2.5 MPa
Stress in element 2	5 MPa	−5 MPa	5 MPa
Reaction at fixed end	500 N	500 N	500 N

EXAMPLE 3.10

Find the nodal displacements, stress in each element, and reaction of the fixed end for Figure 3.33 shown below. Take $E_1 = 2 \times 10^5$ N/mm^2 and $E_2 = 1 \times 10^5$ N/mm^2.

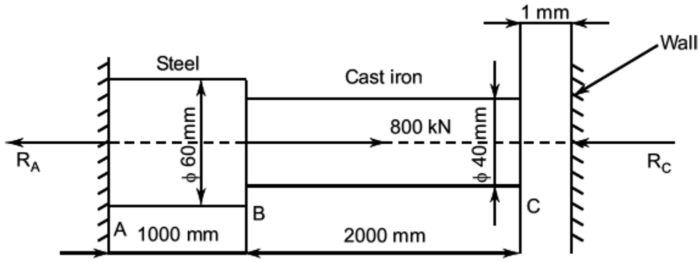


FIGURE 3.33 Example 3.10.

Solution

(I) Analytical method [Refer to Figure 3.33].

$$A_{AB} = \frac{\pi}{4}d_1^2 = \frac{\pi}{4}(60)^2 = 2827.43 \text{ mm}^2$$

$$A_{BC} = \frac{\pi}{4}d_1^2 = \frac{\pi}{4}(40)^2 = 1256.64 \text{ mm}^2.$$

In the absence of the right wall,

$$\Delta_L = \Delta_{AB} = \frac{P_{AB} \times L_{AB}}{A_{AB} \times E_{AB}} = \frac{800 \times 10^3 \times 1000}{2827.43 \times 2 \times 10^5} = 1.415 \text{ mm}.$$

Hence, the contact does occur with the right will since $u_3 = 1.415\ mm$.

Let R_A and R_c be the reactions developed due to constraint.

$$R_A + R_C = 800 \times 10^3 \tag{3.55}$$

$$\frac{R_A \times L_{AB}}{A_{AB} \times E_{AB}} + \frac{(-R_C) \times L_{BC}}{A_{BC} \times E_{BC}} = 1$$

$$\frac{R_A \times (1000)}{2827.43 \times 2 \times 10^5} + \frac{(-R_C) \times (2000)}{1256.64 \times 1 \times 10^5} = 1$$

$$1.7684 \times 10^{-6} \times R_A - 1.5915 \times 10^{-5} \times R_C = 1. \tag{3.56}$$

By solving equations (3.55) and (3.56),

we get

$$R_A = 776547.49 \text{ N}$$

$$R_C = 23452.51 \text{ N}.$$

Displacement calculation

$$\delta_B = \Delta_{AB} = \frac{R_A \times L_{AB}}{A_{AB} \times E_{AB}} = \frac{776547.49 \times 1000}{2827.43 \times 2 \times 10^5} = 1.373 \text{ mm}$$

$$\delta_C = 1 \text{ mm}.$$

Stress calculation

$$\sigma_{AB} = \frac{R_A}{A_{AB}} = \frac{776547.49}{2827.43} = 274.65 \text{ MPa}$$

$$\sigma_{BC} = \frac{(-R_C)}{A_{BC}} = \frac{-23452.51}{1256.64} = -18.66 \text{ MPa}.$$

(II) FEM by hand calculations.

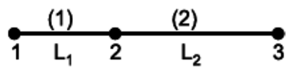


FIGURE 3.33(a) Finite element model for Example 3.10.

$$L_1 = 1000 \text{ mm}$$

$$L_2 = 2000 \text{ mm}.$$

In this example, first determine whether contact occurs between the bar and the wall. To do this, assume that the wall does not exist. Then the solution to the problem is (consider the 2 element model),

Stiffness matrix for element 1 is,

$$[k_1]=\frac{A_1E_1}{L_1}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{2827.43\times 2\times 10^5}{1000}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=5.655\times 10^5\begin{array}{c} \begin{array}{cc}1 & \;\;2\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\end{array}.$$

Stiffness matrix for element 2 is,

$$[k_2]=\frac{A_2E_2}{L_2}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{1256.64\times 1\times 10^5}{2000}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=0.628\times 10^5\begin{array}{c} \begin{array}{cc}2 & \;\;3\end{array}\\ \begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\end{array}\begin{array}{c}\\ 2\\ 3\end{array}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{3.57}$$

$$10^5\begin{array}{c}\begin{array}{ccc}1 & \qquad 2 & \qquad\qquad 3\end{array}\\ \begin{bmatrix}5.655 & -5.655 & 0\\ -5.655 & 5.655+0.628 & -0.628\\ 0 & -0.628 & 0.628\end{bmatrix}\end{array}\begin{array}{c}\\ 1\\ 2\\ 3\end{array}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}R_1\\ 800\times 10^3\\ 0\end{Bmatrix}. \tag{3.58}$$

Boundary conditions are at node 1, $u_1 = 0$.

By using the elimination method, the above matrix reduces to,

$$10^5\begin{bmatrix}6.283 & -0.628\\ -0.628 & 0.628\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}800\times 10^3\\ 0\end{Bmatrix}.$$

By matrix multiplication, we get

$$10^5\left(6.283\times u_2-0.628\times u_3\right)=800\times 10^3 \tag{3.59}$$

$$10^5\left(0.628\times u_2+0.628\times u_3\right)=0 \tag{3.60}$$

By solving equations (3.59) and (3.60)

we get.

$$u_2 = 1.415 \text{ mm and } u_3 = 1.415 \text{ mm}.$$

Since the displacement of node 3 is 1.415 mm, we can say that contact does occur. The problem has to be resolved since the boundary conditions are now different. The displacement at node 3 is given as 1 mm.

Global equation is,

$$\begin{array}{cc} & \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ 10^5 & \begin{bmatrix} 5.655 & -5.655 & 0 \\ -5.655 & 5.655+0.628 & -0.628 \\ 0 & -0.628 & 0.628 \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 800\times10^3 \\ 0 \end{Bmatrix}. \tag{3.61}$$

Boundary conditions at node 1, $u_1 = 0$ and at node 3, $u_2 = 1$ mm.

By using the elimination method, the above matrix reduces to,

$$10^5 [6.283]\{u_2\} = \left[800\times10^3\right] - 1\left[10^5 \times (-0.628)\right]$$

$$10^5 [6.283]\{u_2\} = 800\times10^3 + 0.628\times10^5$$

$$u_2 = 1.373 \text{ mm.}$$

Stress calculation

$$\sigma_1 = \frac{E_1}{L_1}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \frac{2\times10^5}{1000}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} 0 \\ 1.373 \end{Bmatrix} = 274.6 \text{ MPa}$$

$$\sigma_2 = \frac{E_2}{L_2}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{1\times10^5}{2000}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix} 1.373 \\ 1 \end{Bmatrix} = 18.65 \text{ MPa.}$$

Reaction calculation

From equation (3.60)

$$5.655\times10^5 \times u_1 - 5.655\times10^5 \times u_2 = R_1$$

$$0 - 5.655\times10^5 \times 1.373 = R_1$$

$$R_1 = -776431.5 \text{ N} (\text{Direction is leftwards}).$$

We know that,

$$R_1 + P + R_3 = 0$$

$$-776431.5 + 800\times10^3 + R_3 = 0$$

$$R_3 = -23568.5 \text{ N (Direction is leftwards).}$$

(III) Software results.

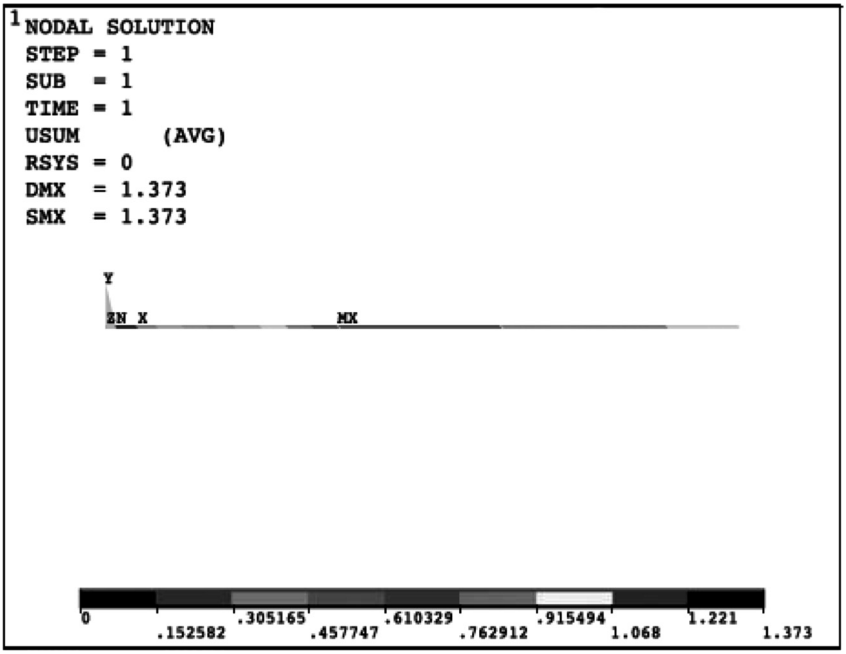


FIGURE 3.33(b) Deflection pattern for a stepped bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	1.3732	0.0000	0.0000	1.3732
3	1.0000	0.0000	0.0000	1.0000

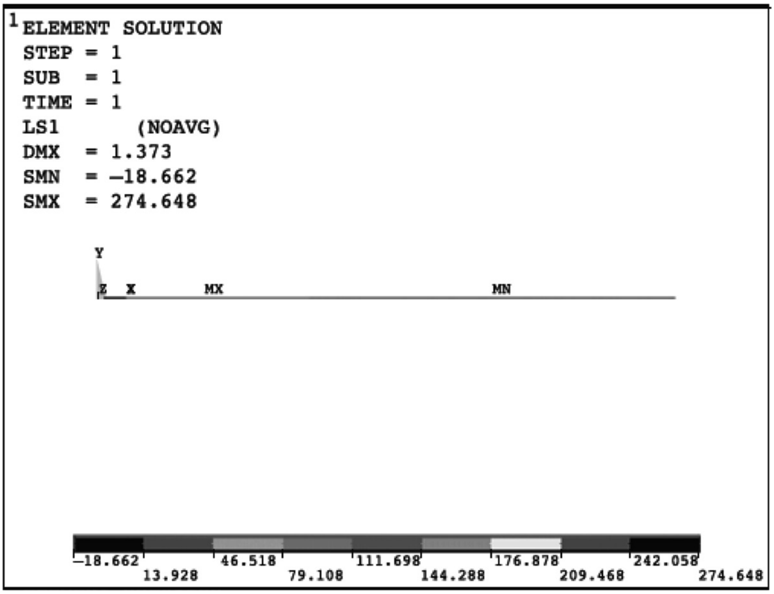


FIGURE 3.33(c) Stress pattern for a stepped bar (refer to Appendix D for color figures).

Stress values at elements

STAT	CURRENT
ELEM	LS1
1	247.65
2	−18.662

Reaction value

The following X, Y, Z solutions are in global coordinates

NODE	FX	FY
1	−0.77655E +06	0.0000
3	−23451.	

Answers for Example 3.10

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	1.373 mm	1.373 mm	1.3732 mm
Displacement at node 3	1 mm	1 mm	1 mm
Stress in element 1	274.65 MPa	274.65 MPa	–274.65 MPa
Stress in element 2	–18.66 MPa	–5 MPa	5 MPa
Reaction at fixed end	–776.5 kN	–776.4 kN	–776.55 kN
Reaction at wall	–23.45 kN	–23.57 kN	–23.451 kN

EXAMPLE 3.11

Find the nodal displacement, stress in each element, and reaction at fixed ends for Figure 3.34 as shown below. If the structure is subjected to an increase in temperature, $\Delta T = 75°\text{C}$, $P_1 = 50$ kN, $P_2 = 75$ kN .

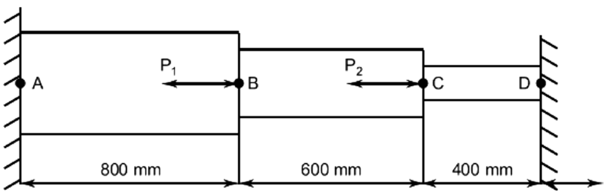


FIGURE 3.34 Example 3.11.

Bronze	Aluminum	Steel
A = 2400 mm^2	1200 mm^2	600 mm^2
E = 83 GPa	70 GPa	200 GPa
$\alpha = 18.9\times10^{-6} / {}^oC$	$23\times10^{-6} / {}^oC$	$11.7\times10^{-6} / {}^oC$

Solution

(I) Analytical method [Refer to Figure 3.34].

Problem can be solved by method of superposition by considering load and temperature separately.

Step 1: Consider only the loads, P_1, P_2 and neglect rise in temperature.

$$R_1' + R_2' = 125\times10^3 \quad \left(R_1' \text{ and } R_2' \text{ are reactions due to } P_1 \text{ and } P_2\right) \tag{3.62}$$

$$\frac{(-P_{AB})\times800}{2400\times83\times10^3}+\frac{(-P_{BC})\times600}{1200\times70\times10^3}+\frac{(P_{CD})\times400}{600\times200\times10^3}=0.$$

But $\quad P_{AB} = R_1'$, $P_{CD} = R_2'$ and $P_{BC} = R_1' - 50\times10^3$

$$\frac{(-R_1')\times800}{2400\times83\times10^3}-\frac{(R_1'-50\times10^3)\times600}{1200\times70\times10^3}+\frac{(P_{CD})\times400}{600\times200\times10^3}=0. \tag{3.63}$$

Solving equations (3.61) and (3.62)

$$R_1' = 53.39 \text{ kN}$$

$$R_2' = 71.61 \text{ kN}$$

$$\therefore \quad \sigma'_{AB} = \frac{\left(-53.39\times10^3\right)}{2400} = -22.25 \text{ MPa}$$

$$\therefore \quad \sigma'_{BC} = \frac{-\left(53.39\times10^3 - 50\times10^3\right)}{1200} = -2.825 \text{ MPa}$$

$$\therefore \quad \sigma'_{CD} = \frac{\left(71.61\times10^3\right)}{600} = 119.35 \text{ MPa}$$

$$\Delta'_{AB} = \frac{-22.25\times800}{83\times10^3} = -0.2144 \text{ mm}$$

$$\Delta'_{BC} = \frac{-2.825\times600}{70\times10^3} = -0.0242 \text{ mm}$$

$$\Delta'_{CD} = \frac{119.35 \times 400}{200 \times 10^3} = 0.2395 \text{ mm.}$$

Step 2. Consider only the rise in temperature and neglect P_1, and P_2.

Free expansions due to ΔT = 75°C are

$$(\Delta L_{AB})_T = \alpha \times L \times (\Delta T) = 18.9 \times 10^{-6} \times 800 \times 75 = 1.134 \text{ mm}$$

$$(\Delta L_{BC})_T = \alpha \times L \times (\Delta T) = 23 \times 10^{-6} \times 600 \times 75 = 1.035 \text{ mm}$$

$$(\Delta L_{CD})_T = \alpha \times L \times (\Delta T) = 11.7 \times 10^{-6} \times 400 \times 75 = 0.351 \text{ mm}$$

$$\text{Total } (\Delta L)_T = 1.134 + 1.035 + 0.351 = 2.52 \text{ mm.}$$

For equilibrium

$$\frac{(-R_1'') \times 800}{2400 \times 83 \times 10^3} + \frac{(-R_1'') \times 600}{1200 \times 70 \times 10^3} + \frac{(-R_1'') \times 400}{600 \times 200 \times 10^3} = -\text{ Total } (\Delta L)_T$$

$$\frac{(-R_1'') \times 800}{2400 \times 83 \times 10^3} + \frac{(-R_1'') \times 600}{1200 \times 70 \times 10^3} + \frac{(-R_1'') \times 400}{600 \times 200 \times 10^3} = -\ 2.52.$$

Solving,

$$R_1'' = 173.89 \text{ kN}$$

$$\sigma''_{AB} = \frac{-173890}{2400} = -72.45 \text{ MPa}$$

$$\sigma''_{BC} = \frac{-173890}{1200} = -144.91 \text{ MPa}$$

$$\sigma''_{CD} = \frac{-173890}{600} = -289.82 \text{ MPa}$$

$$(\Delta L_{AB})_{Load} = \frac{-72.45 \times 800}{83 \times 10^3} = -0.698 \text{ mm}$$

$$(\Delta L_{BC})_{Load} = \frac{-144.91 \times 600}{70 \times 10^3} = -1.242 \text{ mm}$$

$$(\Delta L_{CD})_{Load} = \frac{-289.82 \times 400}{200 \times 10^3} = -0.5796 \text{ mm}$$

$$\Delta''_{AB} = 1.134 - 0.698 = 0.436 \text{ mm}$$

$$\Delta''_{BC} = 1.035 - 1.242 = -0.207 \text{ mm}$$

$$\Delta''_{CD} = 0.351 - 0.5796 = -0.2286 \text{ mm}.$$

Step 3. Use method of superposition and combine steps (1) and (2).

Stresses are, $\sigma_{AB} = \sigma'_{AB} + \sigma''_{AB} = -22.45 - 72.45 = -94.7 \text{ MPa}$.

Similarity, $\sigma_{BC} = \sigma'_{BC} + \sigma''_{BC} = -147.74 \text{ MPa}$

$$\sigma_{CD} = \sigma'_{CD} + \sigma''_{CD} = -170.47 \text{ MPa}.$$

Change in lengths are,

$$\Delta_{AB} = \Delta'_{AB} + \Delta''_{AB} = -0.2144 + 0.436 = 0.2216 \text{ mm}$$

$$\Delta_{BC} = \Delta'_{BC} + \Delta''_{BC} = -0.0242 - 0.207 = -0.2312 \text{ mm}$$

$$\Delta_{CD} = \Delta'_{CD} + \Delta''_{CD} = 0.2395 - 0.2286 = 0.0109 \text{ mm}$$

$$u_2 = \Delta_{AB} = 0.2216 \text{ mm}$$

$$u_3 = \Delta_{CD} = 0.0109 \text{ mm}.$$

Reactions are,

$$R_1 = R'_1 + R''_1 = 53.39 + 173.89 = 227.28 \text{ kN}$$

$$R_2 = R'_2 + R''_2 = 71.61 - 173.89 = -102.28 \text{ kN}.$$

(II) FEM by hand calculations.

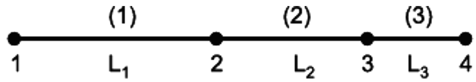


FIGURE 3.34(a) Finite element model for Example 3.11.

$L_1 = 800 \text{ mm}$, $L_2 = 600 \text{ mm}$, $L_3 = 400 \text{ mm}$

$A_1 = 2400 \text{ mm}^2$, $A_2 = 1200 \text{ mm}^2$, $A_3 = 600 \text{ mm}^2$

$E_1 = 83\times10^3 \text{ N/mm}^2$, $E_2 = 70\times10^3 \text{ N/mm}^2$, $E_3 = 200\times10^3 \text{ N/mm}^2$

$a_1 = 18.9\times10^{-6} /^{\circ}C$, $a_2 = 23\times10^{-6} /^{\circ}C$, $a_3 = 11.7\times10^{-6} /^{\circ}C.$

Element stiffness matrices are,

$$[k_1] = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2400\times83\times10^3}{800}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 249\times10^3 \begin{array}{c} \begin{array}{cc} 1 & 2 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \end{array}$$

$$[k_2] = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{1200\times70\times10^3}{600}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 140\times10^3 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{array}{c} 2 \\ 3 \end{array}$$

$$[k_3] = \frac{A_3E_3}{L_3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{600\times200\times10^3}{400}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 300\times10^3 \begin{array}{c} \begin{array}{cc} 3 & 4 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 3 \\ 4 \end{array}.$$

Effect of temperature and thermal loads are,

$$\{Q_{1(th)}\} = E_1A_1\alpha_1(\Delta T)\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 83\times10^3\times2400\times18.9\times10^{-6}\times75\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 282.37\times10^3 \begin{array}{c} \begin{array}{cc} 3 & 4 \end{array} \\ \begin{Bmatrix} -1 \\ 1 \end{Bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \end{array}$$

$$\{Q_{2(th)}\} = E_2A_2\alpha_2(\Delta T)\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 70\times10^3\times1200\times23\times10^{-6}\times75\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 144.9\times10^3 \begin{Bmatrix} -1 \\ 1 \end{Bmatrix} \begin{array}{c} 2 \\ 3 \end{array}$$

$$\{Q_{3(th)}\} = E_3A_3\alpha_3(\Delta T)\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 200\times10^3\times600\times11.7\times10^{-6}\times75\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} = 105.3\times10^3 \begin{Bmatrix} -1 \\ 1 \end{Bmatrix} \begin{array}{c} 3 \\ 4 \end{array}.$$

Global Force vector

$$\{R\} = \begin{Bmatrix} -282.37\times10^3 \\ 282.37\times10^3 - 144.9\times10^3 \\ 144.9\times10^3 - 105.3\times10^3 \\ 105.3\times10^3 \end{Bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} = \begin{Bmatrix} -282.37\times10^3 \\ 137.47\times10^3 \\ 39.6\times10^3 \\ 105.3\times10^3 \end{Bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix}$$

$$10^3 \begin{bmatrix} 249 & -249 & 0 & 0 \\ -249 & 249+140 & -140 & 0 \\ 0 & -140 & 140+300 & -300 \\ 0 & 0 & -300 & 300 \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = 10^3 \begin{Bmatrix} -282.37 + R_1 \\ 137.47 - 50 \\ 39.6 - 75 \\ 105.3 + R_4 \end{Bmatrix}. \quad (3.64)$$

Using the elimination method and applying boundary conditions,

i.e., $u_1 = u_4 = 0.$

The equation (3.63) reduces to,

$$\cancel{10^3} \begin{bmatrix} 389 & -140 \\ -140 & 440 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \cancel{10^3} \begin{Bmatrix} 87.47 \\ -35.4 \end{Bmatrix}.$$

By solving the above matrix and equation,

we get $u_2 = 0.2212$ mm and $u_3 = -0.0101$ mm.

Stress calculation

$$\sigma_1 = \frac{E_1}{L_1}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} - E_1\alpha_1(\Delta T)$$

$$= \frac{83\times10^3}{800}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 0.2212 \end{Bmatrix} - 83\times10^3\times18.9\times10^{-6}\times75 = -94.7 \text{ MPa}$$

$$\sigma_2 = \frac{E_2}{L_2}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} - E_2\alpha_2(\Delta T)$$

$$= \frac{70\times10^3}{600}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0.2212 \\ -0.0101 \end{Bmatrix} - 70\times10^3\times23\times10^{-6}\times75 = -145.38 \text{ MPa}$$

$$\sigma_3 = \frac{E_3}{L_3}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}u_3\\u_4\end{Bmatrix} - E_3 a_3(\Delta T)$$

$$= \frac{200\times10^3}{400}\begin{bmatrix}-1 & 1\end{bmatrix}\begin{Bmatrix}-0.0101\\0\end{Bmatrix} - 200\times10^3\times11.7\times10^{-6}\times75 = -170.45\text{ MPa.}$$

Reaction calculation

$$249\times10^3\times u_1 - 249\times10^3\times u_2 = -282.37\times10^3 + R_1$$

$$0 - 249\times10^3\times0.2212 = -282.37\times10^3 + R_1$$

$$R_1 = 227.29\text{ kN}$$

$$-300\times10^3\times u_3 + 300\times10^3\times u_4 = 105.3\times10^3 + R_4$$

$$-300\times10^3\times(-0.0101) + 0 = 105.3\times10^3 + R_4$$

$$R_4 = -102.27\text{ kN.}$$

(III) Software results.

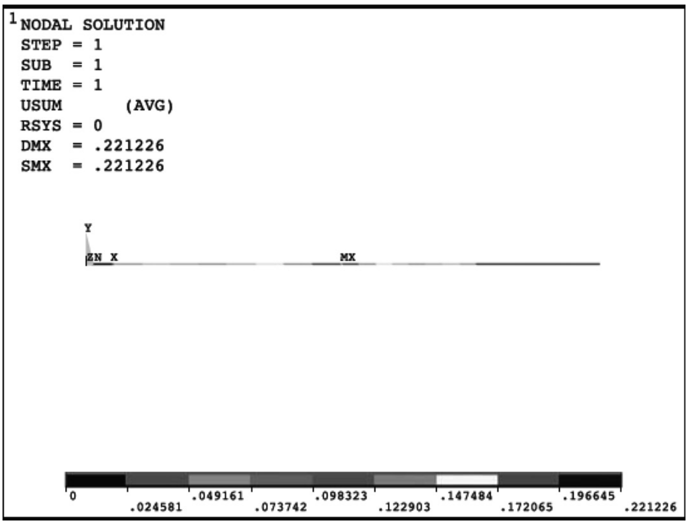


FIGURE 3.34(b) Deflection pattern for a stepped bar (refer to Appendix D for color figures).

Deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.22123	0.0000	0.0000	0.22123
3	−0.10064E−01	0.0000	0.0000	0.10064E−01
4	0.0000	0.0000	0.0000	0.0000

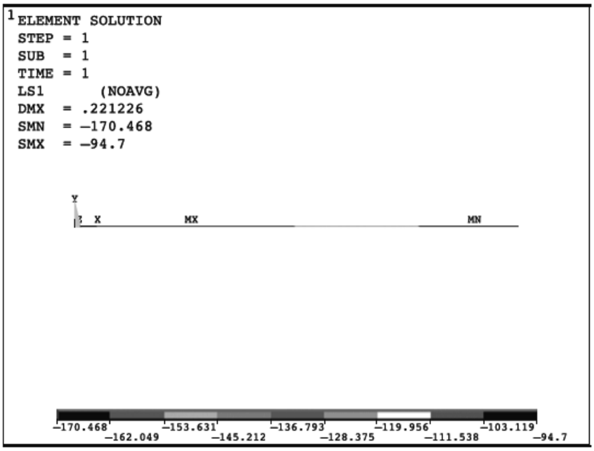


FIGURE 3.34(c) Stress pattern for a stepped bar (refer to Appendix D for color figures).

Stress values at elements

STAT	CURRENT
ELEM	**LS1**
1	−94.700
2	−147.73
3	−170.47

Reaction values

The following X, Y, Z solutions are in global coordinates

NODE	FX	FY
1	0.22728E +06	0.0000
4	−0.10228E +06	0.0000

Answers for Example 3.11

Parameter	Analytical method	FEM-hand calculations	Software results
Displacement at node 2	0.2216 mm	0.2212 mm	0.22123 mm
Displacement at node 3	−0.0109 mm	−0.0101 mm	−0.010064 mm
Stress in element 1	−94.7 MPa	−94.7 MPa	−94.7 MPa
Stress in element 2	−147.74 MPa	−145.38 MPa	−147.73 MPa
Stress in element 3	−170.47 MPa	−170.45 MPa	−170.47 MPa
Reaction at fixed end	227.28 kN	227.2912 kN	227.28 kN
Reaction at wall	−102.28 kN	−102.27 kN	−102.28 kN

Procedure for solving the example using ANSYS® 11.0 academic teaching software

For Example 3.11

Preprocessing

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Structural Link > 2D spar 1 > Ok > Close**

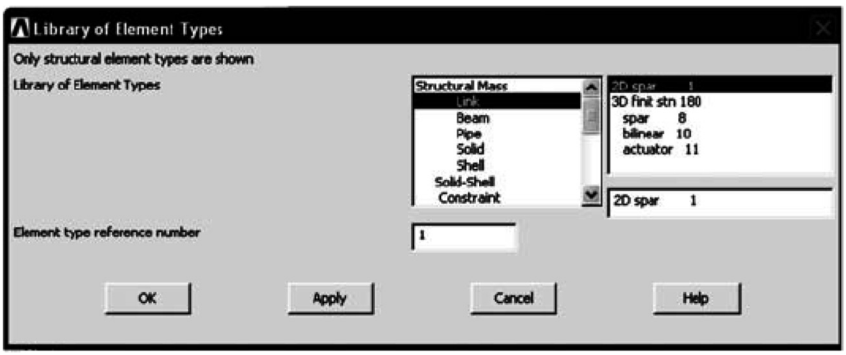


FIGURE 3.35 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

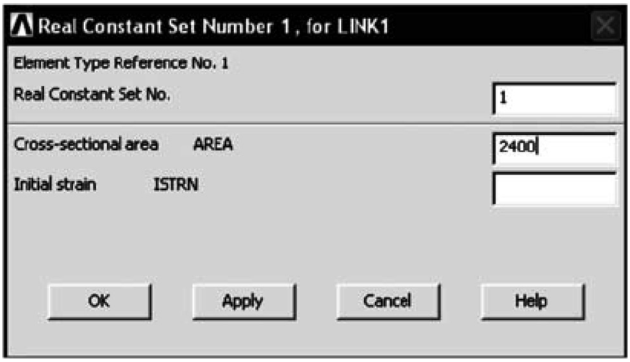


FIGURE 3.36 Enter the cross-sectional area of 1st element.

Cross-sectional area AREA >**Enter 2400 > OK > Add > OK**

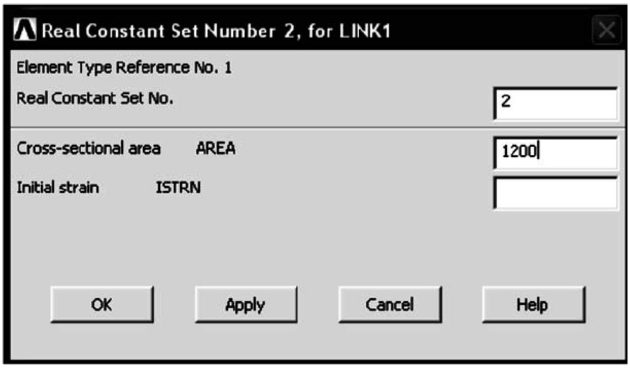


FIGURE 3.37 Enter the cross-sectional area of 2nd element.

Cross-sectional area AREA > **Enter 1200 > OK > Add > OK**

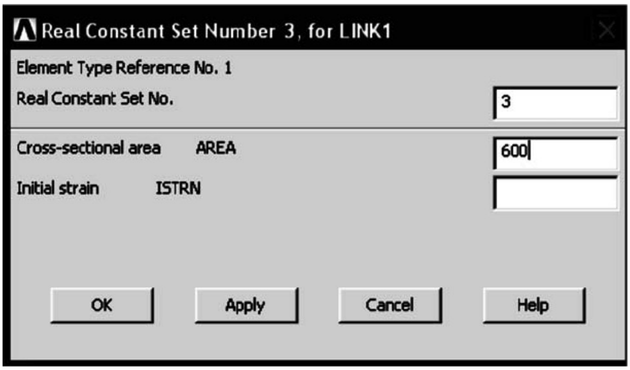


FIGURE 3.38 Enter the cross-sectional area of 3rd element.

Cross-sectional area AREA > **Enter 600 > OK > Add > OK > Close**

Enter the material properties.

3. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1,
   Click **Structural > Linear > Elastic > Isotropic**
   Enter **EX = 0.83E5 and PRXY = 0.34 > OK**
   Enter the coefficient of thermal expansion α
   Click **Structural > Thermal Expansion > Secant coefficient > Isotropic**
   Enter **ALPX – 18.9E-6 > OK**
   Then in the material model window click on **Material menu > New Model > OK**
   Material Model Number 2,
   Click **Structural > Linear > Elastic > Isotropic**
   Enter **EX = 0.7E5 AND PRXY -0.35 > OK**
   Enter the coefficient of thermal expansion α
   Click **Structural > Thermal Expansion > Secant coefficient > Isotropic**
   Enter **ALPX = 23E-6 > OK**
   Then in the material model window click on **Material menu > New Model > OK**
   Material Model Number 3,
   Click **Structural > Linear > Elastic > Isotropic**
   Enter **EX =2E5 and PRXY = 0.3 > OK**
   Enter the coefficient of thermal expansion α
   Click **Structural > Thermal Expansion > Secant coefficient > Isotropic**
   Enter **ALPX = 11.7E-6** > **OK**
   (**Close** the Define Material Model Behavior window.)
   Create the nodes and elements. Use 3 element models. Hence create 4 nodes and 3 elements.

4. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply** Enter the coordinates of node > **Apply** Enter the coordinates of node 3 > **Apply >** Enter the coordinates of node 4 > **OK.**

Node locations		
Node number	**X COORDINATE**	**Y COORDINATE**
1	0	0
2	800	0
3	1400	0
4	1800	0

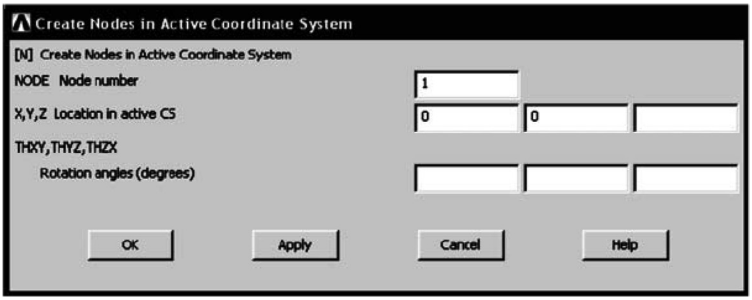


FIGURE 3.39 Enter the node coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes > OK > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **OK**

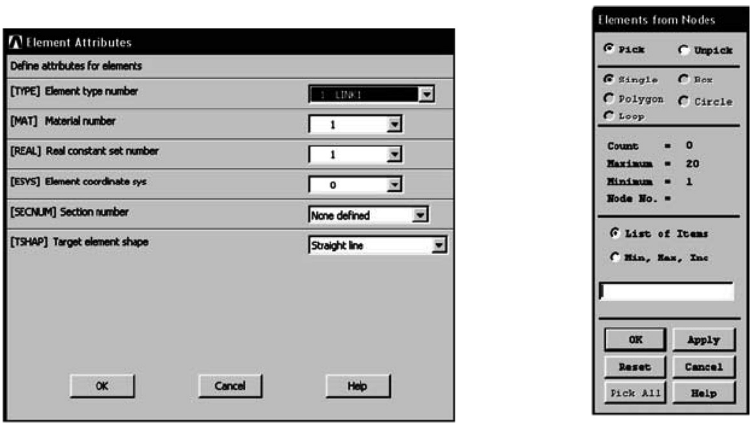


FIGURE 3.40 Assigning element attributes to element 1 and creating element 1.

Elem Attributes > change the material number to 2 > change the Real constant set number to 2 > OK > Auto Numbered > Thru nodes Pick the 2nd and 4th node > **OK**

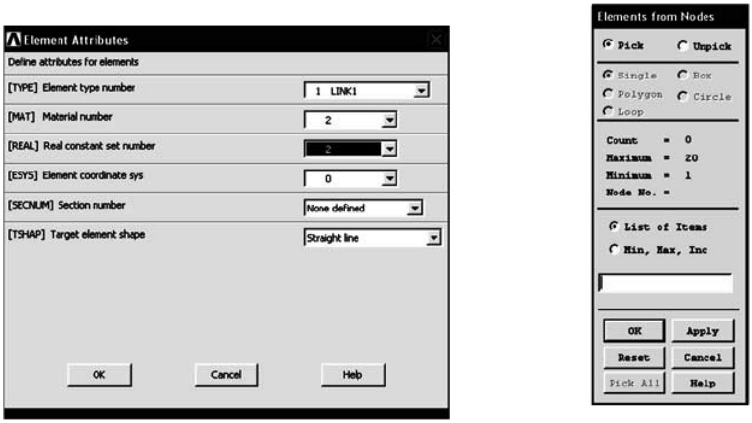


FIGURE 3.41 Assigning element attributes to element 2 and creating element 2.

Elem Attributes > change the material number to 3 > change the Real constant set number to 3 > OK > Auto Numbered > Thru nodes Pick the 3rd and 4th node > **OK**

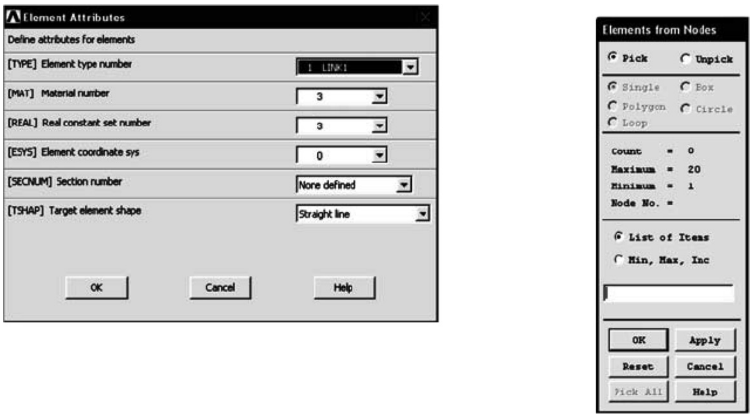


FIGURE 3.42 Assigning element attributes to element 3 and creating element 3.

Apply the displacement boundary conditions, load, and temperature.

6. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st and 4th node > **Apply > All DOF = 0. > OK**
7. **Main Menu > preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes** Pick the 2nd node > **OK**

> Force/Moment value = -50e3 in FX direction > OK > Force/ Moment > On Nodes Pick the 3rd node > **OK > Force/Moment value =-75e3 in FX direction > OK**

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Temperature > On Elements** Pick the element, 2nd element and 3rd element > **OK**

   Enter **Temperature at location N = 75** as shown in Figure 3.38.

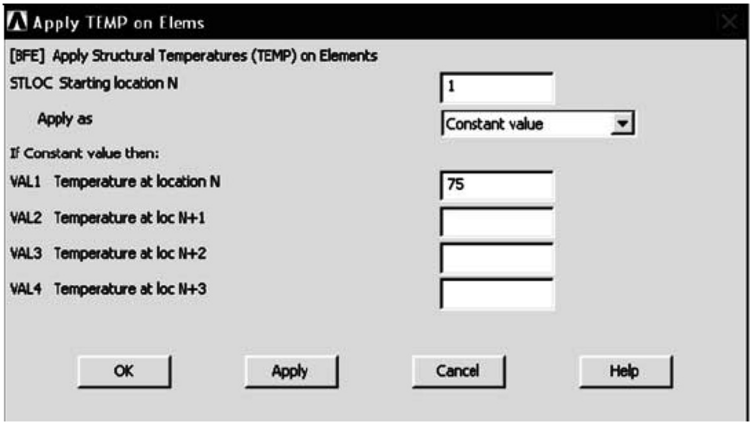


FIGURE 3.43 Enter the rise in temperature on elements.

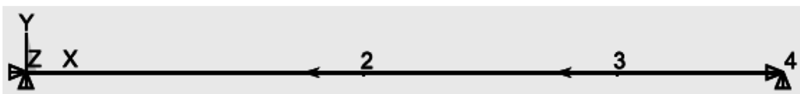


FIGURE 3.44 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First to be safe, save the model.

Solution. The interactive solution proceeds.

9. **Main Meni > Solution > Solve > Current LS > OK**

   The /**STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window and if all is OK, select **FILE > CLOSE**

In the Solve **Current Load Step** window, Select **OK,** and when the solution is complete, **close** the **'Solution is Done!'** window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK**

    This result is shown in Figure 3.34(b).

    To find the axial stress, the following procedure is followed.

11. **Main Menu > General Postproc > Element Table > Define Table > Add**

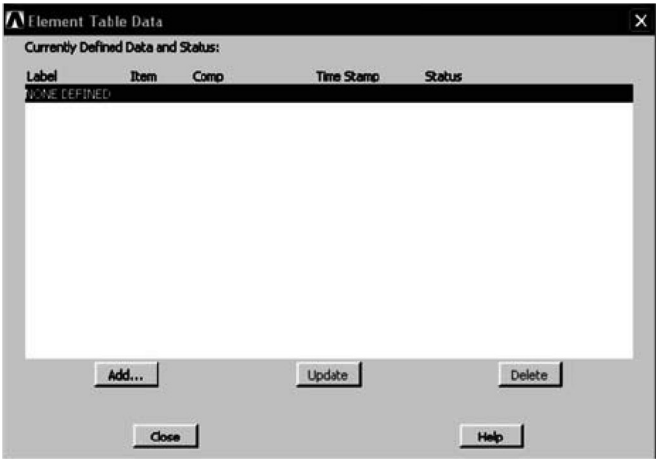


FIGURE 3.45 Defining the element table.

Select **By sequence num and LS** and type **1 after LS** as shown in Figure 3.43.

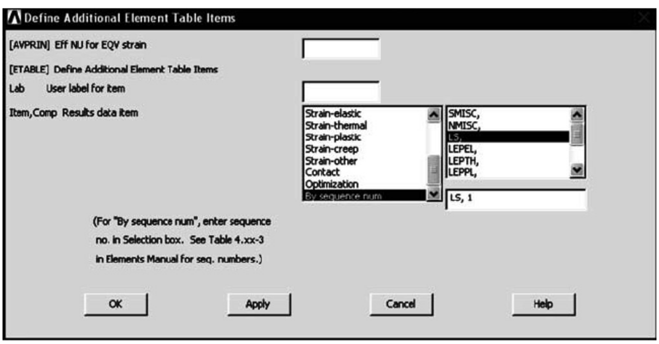


FIGURE 3.46 Selecting options in element table.

>OK

12. Main Menu > General Postproc > Plot Results > Contour Plot > Elem Table > Select **LS1 > OK**

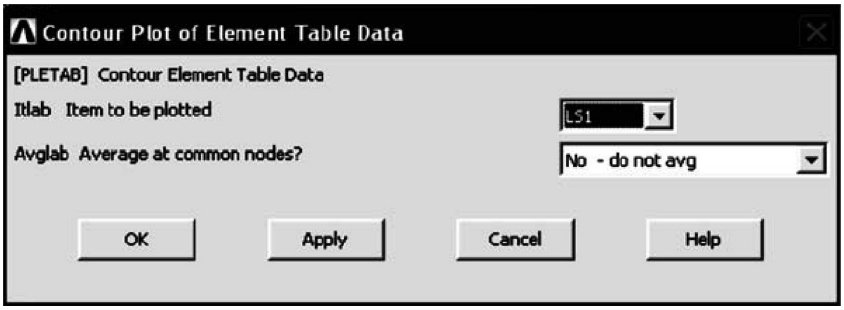


FIGURE 3.47 Selecting options for finding out axial stress.

This result is shown in Figure 3.34(c).

EXERCISES

1. Determine the nodal displacement and element stress for the bar shown in Figure 3.48. Take 3 elements finite element model. Take E = 70 GPa.

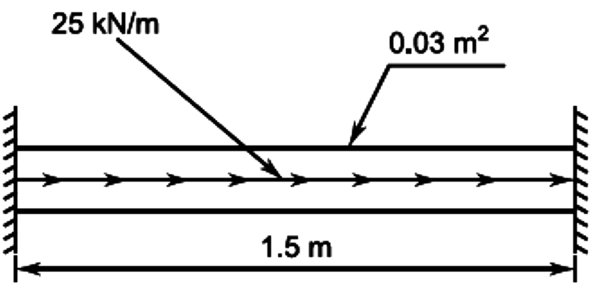


FIGURE 3.48 Exercise 1.

2. Determine the nodal displacements and stresses in the element for the axial distributed loading shown in Figure 3.49. Take one element model. Take E = 200 GPa, $A = 5 \times 10^{-4}$ m^2.

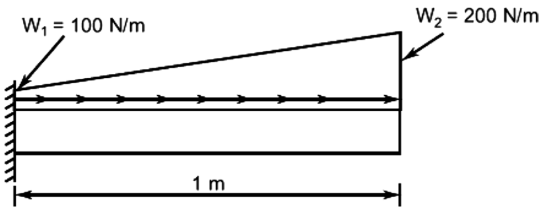


FIGURE 3.49 Exercise 2.

3. For the bar assembly shown in Figure 3.50, determine the nodal displacements, stresses in each element, and reactions. Take E = 210 GPa, $A = 5 \times 10^{-4}\ \text{m}^2$.

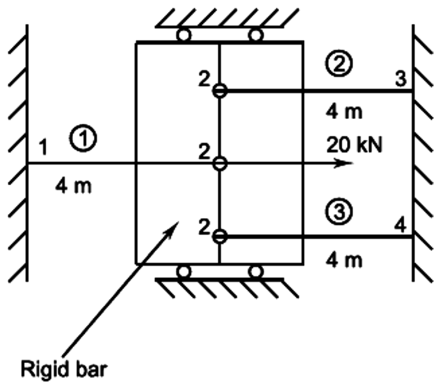


FIGURE 3.50 Exercise 3.

4. Find the deflection at the free end under its own weight for a tapered bar shown in Figure 3.51. Use 2 element models. Take E = 200 GPa, weight density $\rho = 7800\ \text{kg/m}^3$.

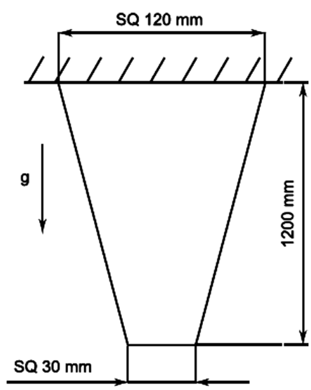


FIGURE 3.51 Exercise 4.

5. Determine the displacement, element stresses, and reactions for the tapered bar shown in Figure 3.52. Use 2 elements finite element models. Take E = 200 GPa, A_1 = 2000 mm^2, A_2 = 4000 mm^2.

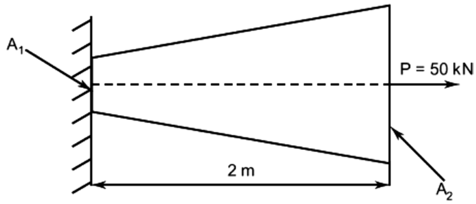


FIGURE 3.52 Exercise 5.

6. Consider the bar shown in Figure 3.53. An axial load P = 500 kN is applied as shown. Determine the

(a) Nodal displacement (b) Stresses in each material (c) Reaction forces.

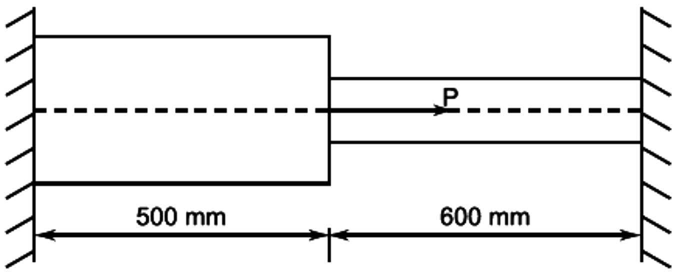


FIGURE 3.53 Exercise 6.

Aluminum	Steel
A_1 = 3000 mm^2	A_2 = 1000 mm^2
E_1 = 70 GPa	E_2 = 200 GPa

7. In Figure 3.54, determine displacements at 2 and 3 stresses in the members and reactions if the temperature is increased by 60°.

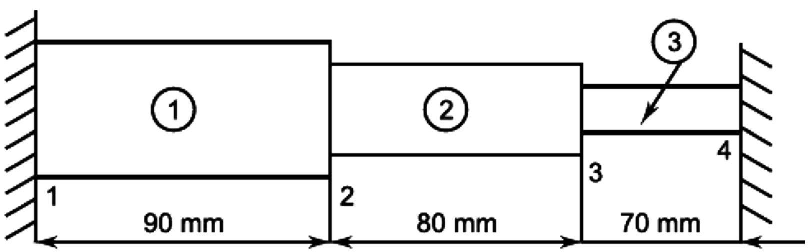


FIGURE 3.54 Exercise 7.

Member	Area A (mm^2)	Youngs modulus E (GPa)	Thermal expansion coefficient α (/°C)
1	1000	70	23×10^{-6}
2	500	100	19×10^{-6}
3	300	200	12×10^{-6}

8. For the vertical bar shown in Figure 3.55, for the deflection at 2 and 3 and stress distribution. Take E = 25 GPa and density, $\rho = 2100$ kg/m^3. Take self-weight of the bar into consideration and solve the problem using 2 elements.

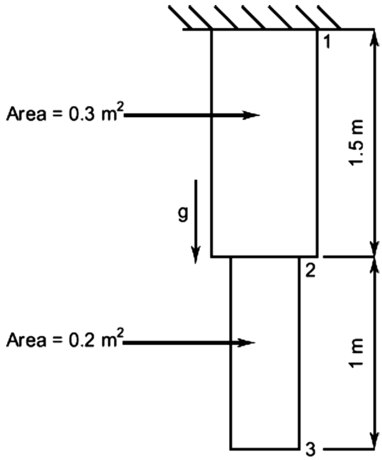


FIGURE 3.55 Exercise 8.

9. Find displacement and stresses shown in Figure 3.56. Take E = 200 GPa.

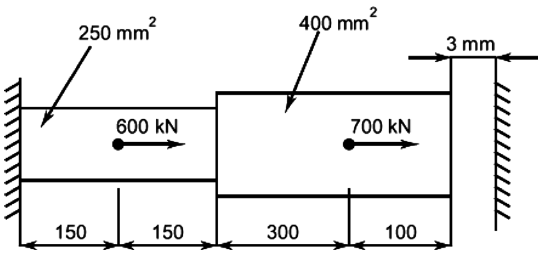


FIGURE 3.56 Exercise 9 (all dimensions are in mm).

REFERENCES

1. J. Wickert, "An Introduction to Mechanical Engineering," Second Edition, Thomson Publisher, 2006.
2. J. M. Gere and S. P. Timoshenko, "Mechanics of Materials," Fourth Edition, PWS Publishing, 1997.
3. R. G. Budynas and J. K. Nisbett, "Shigley's Mechanical Engineering Design," eighth edition, McGraw-Hill Higher Education, 2008.
4. H. C. Martin and G. F. Carey, "Introduction to Finite Element Analysis: Theory and Applications," McGraw-Hill Book Company, 1973.
5. W. B. Bickford, "A First Course in the Finite Element Method," Second Edition, Richard D. Irwin, Inc., 1990.

CHAPTER 4

FINITE ELEMENT ANALYSIS TRUSSES

4.1 INTRODUCTION

This chapter introduces the basic concepts in finite element formulation of trusses and provides the illustration of its ANSYS program.

4.2 TRUSS

Truss, by definition, is a load bearing structure formed by connecting members using pin joints. Truss element is used in the analysis of 2D trusses.

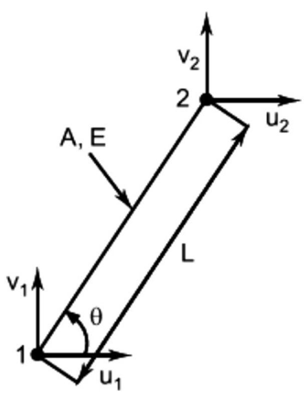


FIGURE 4.1 A 2-D Truss.

The element has two nodes, each having two degrees of freedom namely translations along the x- and y-axes.

The element stiffness matrix and element stress for a truss element are given by,

$$[k]=\frac{AE}{L}\begin{bmatrix}\cos^2\theta & \cos^2\theta\times\sin\theta & -\cos^2\theta & -\cos\theta\times\sin\theta\\ \cos\theta\times\sin\theta & \sin^2\theta & -\cos\theta\times\sin\theta & -\sin^2\theta\\ -\cos^2\theta & -\cos\theta\times\sin\theta & \cos^2\theta & \cos\theta\times\sin\theta\\ -\cos\theta\times\sin\theta & -\sin^2\theta & \cos\theta\times\sin\theta & \sin^2\theta\end{bmatrix} \quad (4.1)$$

$$=\frac{AE}{L}\begin{bmatrix}c^2 & cs & -c^2 & -cs\\ cs & s^2 & -cs & -s^2\\ -c^2 & -cs & c^2 & cs\\ -cs & -s^2 & cs & s^2\end{bmatrix}$$

$$\{\sigma\}=\frac{E}{L}\begin{bmatrix}-\cos\theta & -\sin\theta & \cos\theta & \sin\theta\end{bmatrix}\{q\},\text{ where }\{q\}=\begin{Bmatrix}u_1\\ v_1\\ u_2\\ v_2\end{Bmatrix}. \quad (4.2)$$

θ = angle of truss element at node 1 with positive x-axis (in degrees)

EXAMPLE 4.1

Determine the nodal displacements, element stresses, and support reactions for the three member truss shown in Figure 4.2. Take $A = 800\text{ mm}^2$ and E = 200 GPa for all members.

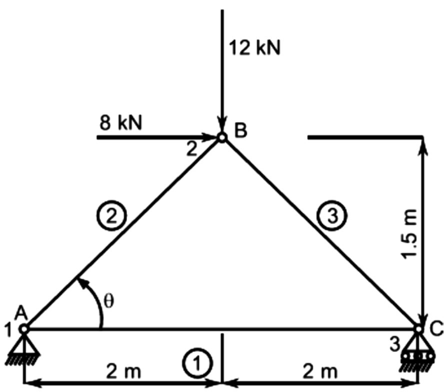


FIGURE 4.2 Example 4.1.

Solution

(I) Analytical method [Refer to Figure 4.2].

$$AB = AC = \sqrt{(2)^2 + (1.5)^2} = 2.5 \text{ m}$$

$$\sin\theta = \frac{3}{2},\ \cos\theta = \frac{4}{5}.$$

Consider equilibrium of joint B,

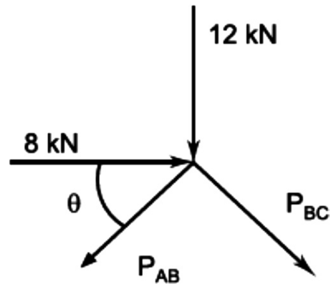


FIGURE 4.2(a) Analytical method for joint B in Example 4.1.

$$\sum F_x = 0 \text{ and } \sum F_y = 0$$

$$8 - P_{AB}\cos\theta + P_{BC}\cos\theta = 0 \tag{4.3}$$

$$-12 - P_{AB}\sin\theta - P_{BC}\sin\theta = 0 \cdot \tag{4.4}$$

Solving equations (4.3) and (4.4)

$P_{AB} = -5$ kN and $P_{BC} = -15$ kN $\quad$ (P_{AB} and P_{BC} are compressive).

Consider equilibrium of joint A,

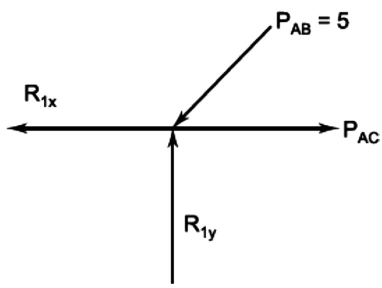


FIGURE 4.2(b) Analytical method for joint A in Example 4.1.

$$\sum F_x = 0 \text{ and } \sum F_y = 0$$

$$-R_{1x} + P_{AC} - P_{AB}\cos\theta = 0 \tag{4.5}$$

$$R_{1y} - P_{AB}\sin\theta = 0.$$

Consider equilibrium of joint C,

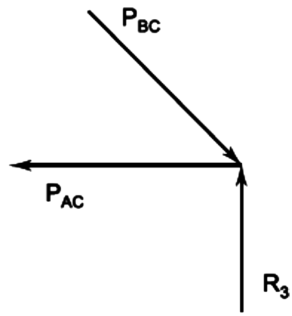


FIGURE 4.2(c) Analytical method for joint C in Example 4.1.

$$\sum F_x = 0 \text{ and } \sum F_y = 0$$

$$P_{AC} = P_{BC}\cos\theta = 15\times\frac{4}{5} = 12 \text{ kN}$$

$$R_3 = P_{BC}\sin\theta = 15\times\frac{3}{5} = 9 \text{ kN}.$$

For equation (4.5)

$$R_{1x} = P_{AC} - P_{AB}\cos\theta = 12 - 5\times\frac{4}{5} = 8 \text{ kN}$$

$$\sigma_{AB} = \sigma_2 = \frac{P_{AB}}{A_{AB}} = \frac{-5\times10^3}{800} = -6.25 \text{ MPa} \;\; (\text{Compressive})$$

$$\sigma_{BC} = \sigma_3 = \frac{P_{BC}}{A_{BC}} = \frac{-15\times10^3}{800} = -18.75 \text{ MPa} \;\; (\text{Compressive})$$

$$\sigma_{AC} = \sigma_1 = \frac{P_{AC}}{A_{AC}} = \frac{12\times10^3}{800} = 15 \text{ MPa} \;\; (\text{Tensile})$$

$$\Delta_{AB} = \frac{P_{AB}L_{AB}}{A_{AB}E_{AB}} = \frac{-5\times10^3\times2500}{800\times2\times10^5} = -0.078125 \text{ mm}$$

$$\Delta_{BC} = \frac{P_{BC}L_{BC}}{A_{BC}E_{BC}} = \frac{-15\times10^3\times2500}{800\times2\times10^5} = -0.234375 \text{ mm}$$

$$\Delta_{AC} = \frac{P_{AC}L_{AC}}{A_{AC}E_{AC}} = \frac{12\times10^3\times4000}{800\times2\times10^5} = 0.3 \text{ mm.}$$

Calculation of nodal displacements u_2, v_2, and u_3

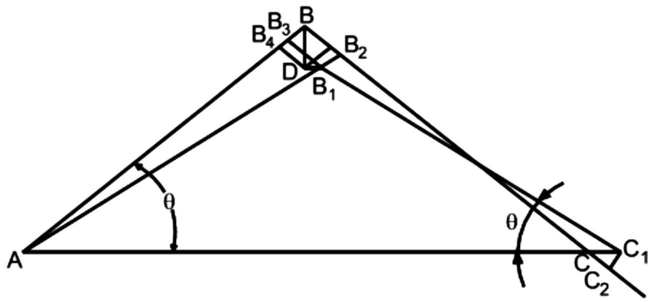


FIGURE 4.2(d) Analytical method for Calculation of nodal displacements u_2, v_2, and u_3 in Example 4.1.

$$DB_1 = u_2,\ BD = v_2 \text{ and } CC_1 = u_3$$
$$BB_3 = \Delta_{AB}$$
$$BB_2 = \Delta_{BC}$$
$$CC_1 = u_3 = \Delta_{AC} = 0.3 \text{ mm}$$
$$\text{CC}_2 = CC_1 \cos\theta = \Delta_{AC}\cos\theta = 0.3\times\frac{4}{3} = 0.24 \text{ mm.}$$

From geometry [Refer to Figure 4.2(d)].

$$BB_3 = \Delta_{AB} = BD\sin\theta - DB_1\cos\theta = v_2\sin\theta - u_2\cos\theta \tag{4.6}$$

$$BB_2 = BC - B_2C = BC - (B_2C_2 - CC_2) = (BC - B_2C_2) + CC_2$$

$$BB_2 = \Delta_{BC} + CC_2 = BD\sin\theta + DB_1\cos\theta$$

$$\Delta_{BC} + CC_2 = v_2\sin\theta + u_2\cos\theta\,\cdot \tag{4.7}$$

Substituting in equations (4.6) and (4.7)

$$0.078125 = v_2\times\frac{3}{5} - u_2\times\frac{4}{5} \tag{4.8}$$

$$0.234375 + 0.24 = v_2 \times \frac{3}{5} + u_2 \times \frac{4}{5}$$

$$0.474375 = v_2 \times \frac{3}{5} + u_2 \times \frac{4}{5}. \tag{4.9}$$

Solving equations (4.8) and (4.9), we get,

$v_2 = 0.4604$ mm (since point B moves downwards). Hence, $v_2 = -0.6404$ mm.

(**III**) **FEM by hand calculation** [Refer to Figure 4.2].

Elements	Node numbers		θ	cos θ	sin θ	L (mm)
	Local 1	Local 2				
1	1	3	0	1	0	4000
2	1	2	36.87	0.8	0.6	2500
3	2	3	–36.87	0.8	–0.6	2500

Angle calculation

For element 2

$$\sin\theta = \frac{1.5}{2.5} \Rightarrow \theta = 36.87^\circ.$$

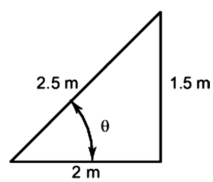


FIGURE 4.2(e) Angle calculation for element 2 in Example 4.1.

For element 3

$$\sin\theta = \frac{1.5}{2.5} \Rightarrow \theta = -36.87^\circ.$$

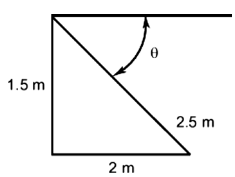


FIGURE 4.2(f) Angle calculation for element 3 in Example 4.1.

Element stiffness matrix for element 1 is,

$$[k_1]=\frac{AE}{L}\begin{bmatrix} \cos^2\theta & \cos^2\theta\times\sin\theta & -\cos^2\theta & -\cos\theta\times\sin\theta \\ \cos\theta\times\sin\theta & \sin^2\theta & -\cos\theta\times\sin\theta & -\sin^2\theta \\ -\cos^2\theta & -\cos\theta\times\sin\theta & \cos^2\theta & \cos\theta\times\sin\theta \\ -\cos\theta\times\sin\theta & -\sin^2\theta & \cos\theta\times\sin\theta & \sin^2\theta \end{bmatrix}$$

$$[k_1]=\frac{800\times200\times10^3}{4000}\begin{bmatrix} \cos^2 0 & \cos^2 0\times\sin 0 & -\cos^2 0 & -\cos 0\times\sin 0 \\ \cos 0\times\sin 0 & \sin^2 0 & -\cos 0\times\sin 0 & -\sin^2 0 \\ -\cos^2 0 & -\cos 0\times\sin 0 & \cos^2 0 & \cos 0\times\sin 0 \\ -\cos 0\times\sin 0 & -\sin^2 0 & \cos 0\times\sin 0 & \sin^2 0 \end{bmatrix}$$

$$[k_1]=40\times10^3\begin{array}{c} \begin{matrix} u_1 & v_1 & u_3 & v_3 \end{matrix} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array}\begin{matrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{matrix}.$$

Element stiffness matrix for element 2 is,

$$[k_2]=\frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

$$=\frac{800\times200\times10^3}{2500}\begin{bmatrix} (0.8)^2 & 0.8\times0.6 & -(0.8)^2 & -0.8\times0.6 \\ 0.8\times0.6 & (0.6)^2 & -0.8\times0.6 & -(0.6)^2 \\ -(0.8)^2 & -0.8\times0.6 & (0.8)^2 & 0.8\times0.6 \\ -0.8\times0.6 & -(0.6)^2 & 0.8\times0.6 & (0.6)^2 \end{bmatrix}$$

$$[k_2]=64\times10^3\begin{array}{c} \begin{matrix} u_1 & v_1 & u_2 & v_2 \end{matrix} \\ \begin{bmatrix} 0.64 & 0.48 & -0.64 & -0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix} \end{array}\begin{matrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{matrix}.$$

Element stiffness matrix for element 3 is,

$$[k_3] = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} = 64\times10^3 \begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 0.64 & -0.48 & -0.64 & 0.48 \\ -0.48 & 0.36 & 0.48 & -0.36 \\ -0.64 & 0.48 & 0.64 & -0.48 \\ 0.48 & -0.36 & -0.48 & 0.36 \end{bmatrix} \end{array} \begin{array}{c} \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array}.$$

Global stiffness matrix is,

$$[K] = 10^3 \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_1 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 40+40.96 & 30.72 & -40.96 & -30.72 & -40 & 0 \\ 30.72 & 23.04 & -30.72 & -23.04 & 0 & 0 \\ -40.96 & -30.72 & 40.96+40.96 & 30.72-30.72 & -40.96 & 30.72 \\ -30.72 & -23.04 & 30.72-30.72 & 23.04+23.04 & 30.72 & -23.04 \\ -40 & 0 & -40.96 & 30.72 & 40+40.96 & -30.72 \\ 0 & 0 & 30.72 & -23.04 & -30.72 & 23.04 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array}$$

$$[K] = 10^3 \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_1 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 80.96 & 30.72 & -40.96 & -30.72 & -40 & 0 \\ 30.72 & 23.04 & -30.72 & -23.04 & 0 & 0 \\ -40.96 & -30.72 & 81.92 & 0 & -40.96 & 30.72 \\ -30.72 & -23.04 & 0 & 46.08 & 30.72 & -23.04 \\ -40 & 0 & -40.96 & 30.72 & 80.96 & -30.72 \\ 0 & 0 & 30.72 & -23.04 & -30.72 & 23.04 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array}.$$

Global equation is,

$$[K] = 10^3 \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_1 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 80.96 & 30.72 & -40.96 & -30.72 & -40 & 0 \\ 30.72 & 23.04 & -30.72 & -23.04 & 0 & 0 \\ -40.96 & -30.72 & 81.92 & 0 & -40.96 & 30.72 \\ -30.72 & -23.04 & 0 & 46.08 & 30.72 & -23.04 \\ -40 & 0 & -40.96 & 30.72 & 80.96 & -30.72 \\ 0 & 0 & 30.72 & -23.04 & -30.72 & 23.04 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} R_{1x} \\ R_{1y} \\ 8 \\ -12 \\ 0 \\ R_{3y} \end{Bmatrix} \times 10^3.$$

Using the elimination method for applying boundary conditions,

$$u_1 = v_1 = v_3 = 0 \cdot$$

Then the above matrix reduces to,

$$\cancel{10^3}\begin{bmatrix} 81.92 & 0 & -40.96 \\ 0 & 46.08 & 30.72 \\ -40.96 & 30.72 & 80.96 \end{bmatrix}\begin{bmatrix} u_2 \\ v_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 8 \\ -12 \\ 0 \end{bmatrix} \times \cancel{10^3} \cdot$$

Solving the above matrix and equations,

we get $u_2 = 0.2477$ mm, $v_2 = -0.4604$ mm, and $v_3 = 0.3$ mm.

Stress calculation

Stress in element 1 is,

$$\sigma_1 = \frac{E}{L_1}\begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix} = \frac{200 \times 10^3}{4000}\begin{bmatrix} -c & -s & c & s \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix}$$

$$\sigma_1 = \frac{200 \times 10^3}{4000}\begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ 0.3 \\ 0 \end{bmatrix} = 15 \text{ MPa.}$$

Stress in element 2 is,

$$\sigma_2 = \frac{E}{L_2}\begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} = \frac{200 \times 10^3}{2500}\begin{bmatrix} -c & -s & c & s \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}$$

$$\sigma_2 = \frac{200 \times 10^3}{2500}\begin{bmatrix} -0.8 & -0.6 & 0.8 & 0.6 \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ 0.2477 \\ -0.4604 \end{bmatrix} = -6.249 \text{ MPa.}$$

Stress in element 3 is,

$$\sigma_3 = \frac{E}{L_3}\begin{bmatrix}-\cos\theta & -\sin\theta & \cos\theta & \sin\theta\end{bmatrix}\begin{bmatrix}u_2\\v_2\\u_3\\v_3\end{bmatrix} = \frac{200\times10^3}{2500}\begin{bmatrix}-c & -s & c & s\end{bmatrix}\begin{bmatrix}u_2\\v_2\\u_3\\v_3\end{bmatrix}$$

$$\sigma_3 = \frac{200\times10^3}{2500}\begin{bmatrix}-0.8 & 0.6 & 0.8 & -0.6\end{bmatrix}\begin{bmatrix}0.2477\\-0.4604\\0.3\\0\end{bmatrix} = -18.752 \text{ MPa.}$$

Reaction Calculation

From global equation,

$$-40.96\times u_2 - 30.72\times v_2 - 40\times u_3 = R_{1x}$$
$$-40.96\times 0.2477 - 30.72\times(-0.4604) - 40\times 0.3 = R_{1x}$$
$$R_{1x} = -8 \text{ kN}$$

$$-30.72\times u_2 - 23.04\times v_2 = R_{1y}$$
$$-30.72\times 0.2477 - 23.04\times(-0.4604) = R_{1y}$$
$$R_{1y} = 3 \text{ kN}$$

$$-30.72\times u_2 - 23.04\times v_2 - 30.72\times u_3 = R_{3y}$$
$$-30.72\times 0.2477 - 23.04\times(-0.4604) - 30.72\times(0.3) = R_{3y}$$
$$R_{3y} = 9 \text{ kN}.$$

(III) Software results.

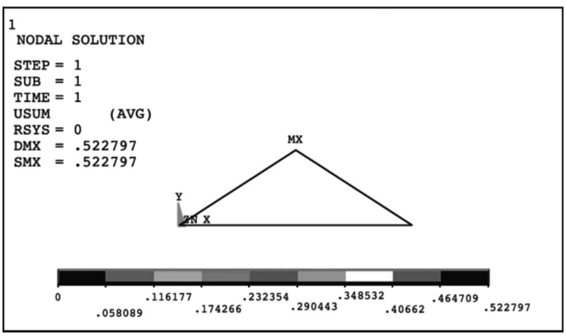


FIGURE 4.2(g) Deflection pattern for a truss for Example 4.1 (refer to Appendix D for color figures).

Deflection value at nodes

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.24766	–0.46042	0.0000	0.52280
3	0.30000	0.0000	0.0000	0.30000

Maximum absolute values

NODE	3	2	0	2
VALUE	0.30000	–0.46042	0.0000	0.52280

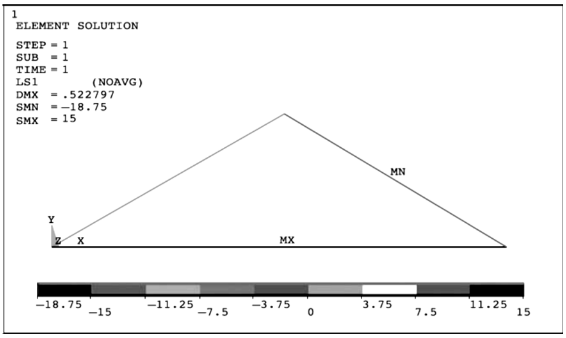


FIGURE 4.2(h) Stress pattern for a truss for Example 4.1 (refer to Appendix D for color figures).

Stress values of elements

STAT	CURRENT
ELEM	LS1
1	15.000
2	–6.2500
3	–18.750

Reaction values

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY
1	–8000.0	3000.0
3		9000.0

ANSWERS FOR EXAMPLE 4.1

Parameter	Analytical method	FEM-Hand calculations	Software results
Displacement of node 2 in			
x-direction	0.2477 mm	0.2477 mm	0.24766 mm
y-direction	–0.4604 mm	–0.4604 mm	–0.46042 mm
Displacement of node 3 in			
x-direction	0.3 mm	0.3 mm	0.3 mm
Stress in			
Element 1	15 MPa	15 MPa	15 MPa
Element 2	–6.25 MPa	–6.248 MPa	–6.25 MPa
Element 3	–18.75 MPa	–18.752 MPa	–18.75 MPa
Reaction			
At 1 in x-direction	–8 kN	–8 kN	–8 kN
At 1 in y-direction	3 kN	3 kN	3 kN
At 3 in y-direction	9 kN	9 kN	9 kN

EXAMPLE 4.2

For the truss shown in Figure 4.3, determine nodal displacements and stresses in each member. All elements have E = 200 GPa and A = 500 mm^2.

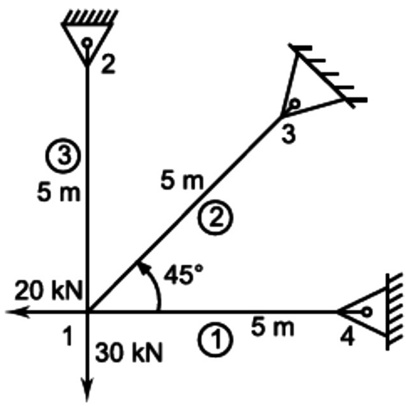


FIGURE 4.3 Example 4.2.

Solution

$$E = 2\times10^5 \text{ N/mm}^2$$

$$A = 500 \text{ mm}^2.$$

(I) FEM by hand calculation

Elements	Node numbers		θ	cos θ	sin θ	L (mm)
	Local 1	Local 2				
1	1	4	0	1	0	5000
2	1	3	45	0.707	0.707	5000
3	1	2	90	0	1	5000

Stiffness matrices for elements 1, 2, and 3 are,

$$[k_1] = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} = \frac{500\times2\times10^5}{5000}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$[k_1] = 20\times10^3 \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_4 & v_4 \end{array} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_4 \\ v_4 \end{array}$$

$$[k_2] = 20\times10^3 \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_3 \\ v_3 \end{array}$$

$$[k_3] = 20\times10^3 \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \end{array}.$$

Global stiffness matrix is,

$$[K] = 20\times10^3 \begin{array}{c} \begin{array}{cccccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & u_4 & v_4 \end{array} \\ \begin{bmatrix} 1+0.5 & 0.5 & 0 & 0 & -0.5 & -0.5 & -1 & 0 \\ 0.5 & 0.5+1 & 0 & -1 & -0.5 & -0.5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{array}$$

$$[K] = 20\times10^3 \begin{array}{c} \begin{array}{cccccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & u_4 & v_4 \end{array} \\ \begin{bmatrix} 1.5 & 0.5 & 0 & 0 & -0.5 & -0.5 & -1 & 0 \\ 0.5 & 1.5 & 0 & -1 & -0.5 & -0.5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{array}.$$

Global equation is,

$$20\times\cancel{10^3} \begin{array}{c} \begin{array}{cccccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & u_4 & v_4 \end{array} \\ \begin{bmatrix} 1.5 & 0.5 & 0 & 0 & -0.5 & -0.5 & -1 & 0 \\ 0.5 & 1.5 & 0 & -1 & -0.5 & -0.5 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -0.5 & -0.5 & 0 & 0 & 0.5 & 0.5 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{array} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} -20 \\ -30 \\ R_{2x} \\ R_{2y} \\ R_{3x} \\ R_{3y} \\ R_{4x} \\ R_{4y} \end{Bmatrix} \times \cancel{10^3}.$$

Using the elimination method for applying boundary conditions,

$$u_2 = v_2 = u_3 = v_3 = u_4 = v_4 = 0\cdot$$

Then the above matrix reduces to,

$$20\begin{bmatrix} 1.5 & 0.5 \\ 0.5 & 1.5 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \end{bmatrix} = \begin{bmatrix} -20 \\ -30 \end{bmatrix}.$$

Solving the above matrix and equations,

we get,

$$u_1 = -0.375 \text{ mm}$$

$$v_1 = -0.875 \text{ mm}.$$

Stress calculation

Stress in element 1 is,

$$\sigma_1 = \frac{E}{L_1}\begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_4 \\ v_4 \end{bmatrix} = \frac{2\times 10^5}{5000}\begin{bmatrix} -c & -s & c & s \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_4 \\ v_4 \end{bmatrix}$$

$$\sigma_1 = \frac{2\times 10^5}{5000}\begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix}\begin{bmatrix} -0.375 \\ -0.875 \\ 0 \\ 0 \end{bmatrix} = 15 \text{ MPa}.$$

Stress in element 2 is,

$$\sigma_2 = \frac{E}{L_2}\begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix} = \frac{2\times 10^5}{5000}\begin{bmatrix} -c & -s & c & s \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix}$$

$$\sigma_2 = \frac{2\times 10^5}{5000}\begin{bmatrix} -0.707 & -0.707 & 0.707 & 0.707 \end{bmatrix}\begin{bmatrix} -0.375 \\ -0.875 \\ 0 \\ 0 \end{bmatrix} = 35.352 \text{ MPa}.$$

Stress in element 3 is,

$$\sigma_3 = \frac{E}{L_3}\begin{bmatrix}-\cos\theta & -\sin\theta & \cos\theta & \sin\theta\end{bmatrix}\begin{bmatrix}u_1\\v_1\\u_2\\v_2\end{bmatrix} = \frac{2\times10^5}{5000}\begin{bmatrix}-c & -s & c & s\end{bmatrix}\begin{bmatrix}u_1\\v_1\\u_2\\v_2\end{bmatrix}$$

$$\sigma_3 = \frac{2\times10^5}{5000}\begin{bmatrix}0 & -1 & 0 & 1\end{bmatrix}\begin{bmatrix}-0.375\\-0.875\\0\\0\end{bmatrix} = 35 \text{ MPa.}$$

(II) Software results.

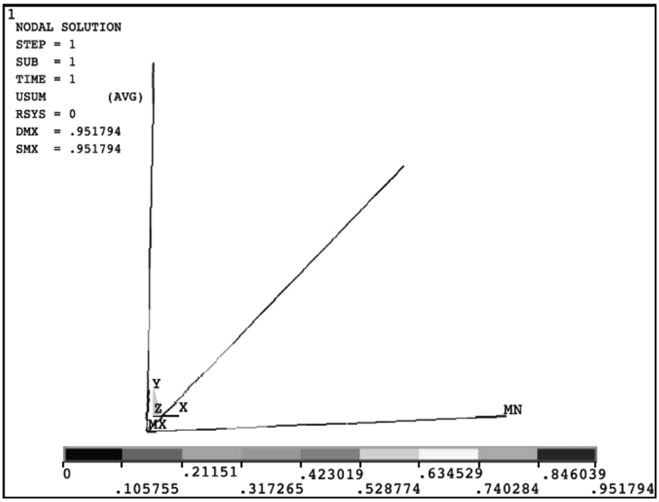


FIGURE 4.3(a) Deflection pattern for a truss for Example 4.2 (refer to Appendix D for color figures).

Deflection value at nodes

The following degree of freedom results are in global coordinates system

NODE	UX	UY	UZ	USUM
1	−0.37486	−0.87486	0.0000	0.95179
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000	0.0000

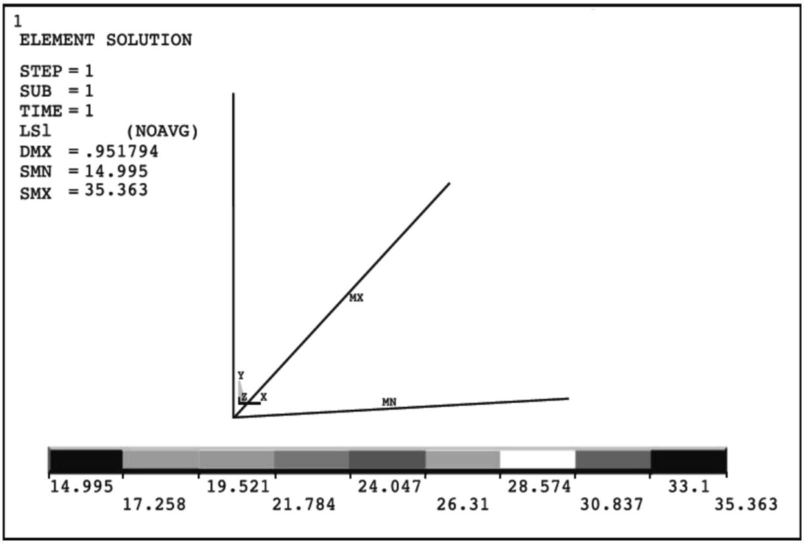


FIGURE 4.3(b) Stress pattern for a truss for Example 4.2 (refer to Appendix D for color figures).

Stress values of elements

STAT	CURRENT
ELEM	LS1
1	14.995
2	35.363
3	34.995

ANSWERS FOR EXAMPLE 4.2

Parameter	FEM- Hand calculations	Software results
Displacement of node 1 in		
x-direction	–0.375 mm	–0.37486 mm
y-direction	–0.875 mm	–0.87486 mm
Stress in element 1	15 MPa	14.995 MPa
Stress in element 2	35.352 MPa	35.363 MPa
Stress in element 3	35 MPa	34.995 MPa

EXAMPLE 4.3

The bar truss shown in Figure 4.4, determine the displacement of node 1 and the axial stress in each member. Take E =210 GPa and $A = 600\ \text{mm}^2$. Solve the problem if node 1 settles an amount of $\delta = 25\ \text{mm}$ in the negative y-direction.

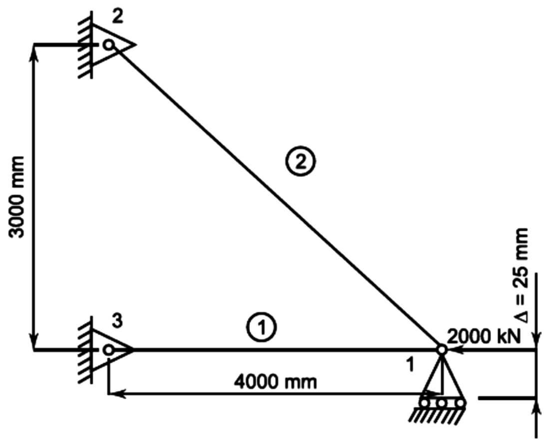


FIGURE 4.4 Example 4.3.

Solution

(I) FEM by hand calculation.

Elements	Node numbers		θ	cos θ	sin θ	L (mm)
	Local 1	Local 2				
1	3	1	0	1	0	4000
2	2	1	–36.87	0.8	–0.6	5000

Angle calculation

For 2nd element,

$$\sin\theta = \frac{3}{5} = 0.6 \Rightarrow \theta = -36.87^\circ\cdot$$

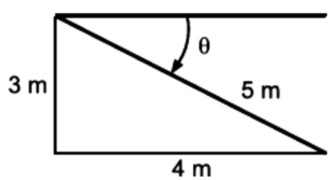


FIGURE 4.4(a) Angle calculation for 2nd element for Example 4.3.

Stiffness matrices for elements 1 and 2 are,

$$[k_1]=\frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}=\frac{600\times 210\times 10^3}{4000}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$[k_1]=31.5\times 10^3 \begin{array}{c} \begin{array}{cccc} u_3 & v_3 & u_1 & v_1 \end{array} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_3 \\ v_3 \\ u_1 \\ v_1 \end{array}$$

$$[k_2]=\frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}=\frac{600\times 210\times 10^3}{5000}\begin{bmatrix} 0.64 & -0.48 & -0.64 & 0.48 \\ -0.48 & 0.36 & 0.48 & -0.36 \\ -0.64 & 0.48 & 0.64 & -0.48 \\ 0.48 & -0.36 & -0.48 & 0.36 \end{bmatrix}$$

$$[k_2]=25.2\times 10^3 \begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_1 & v_1 \end{array} \\ \begin{bmatrix} 0.64 & -0.48 & -0.64 & 0.48 \\ -0.48 & 0.36 & 0.48 & -0.36 \\ -0.64 & 0.48 & 0.64 & -0.48 \\ 0.48 & -0.36 & -0.48 & 0.36 \end{bmatrix} \end{array} \begin{array}{c} \\ u_2 \\ v_2 \\ u_1 \\ v_1 \end{array}.$$

Global stiffness matrix is,

$$[K]=10^3 \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 31.5+16.13 & -12.1 & -16.13 & 12.1 & -31.5 & 0 \\ -12.1 & 9.1 & 12.1 & -9.1 & 0 & 0 \\ -16.13 & 12.1 & 16.13 & -12.1 & 0 & 0 \\ 12.1 & -9.1 & -12.1 & 9.1 & 0 & 0 \\ -31.5 & 0 & 0 & 0 & 31.5 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array}$$

$$[K] = 10^3 \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 47.63 & -12.1 & -16.13 & 12.1 & -31.5 & 0 \\ -12.1 & 9.1 & 12.1 & -9.1 & 0 & 0 \\ -16.13 & 12.1 & 16.13 & -12.1 & 0 & 0 \\ 12.1 & -9.1 & -12.1 & 9.1 & 0 & 0 \\ -31.5 & 0 & 0 & 0 & 31.5 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array}.$$

Global equation is,

$$[K] = \cancel{10^3} \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 47.63 & -12.1 & -16.13 & 12.1 & -31.5 & 0 \\ -12.1 & 9.1 & 12.1 & -9.1 & 0 & 0 \\ -16.13 & 12.1 & 16.13 & -12.1 & 0 & 0 \\ 12.1 & -9.1 & -12.1 & 9.1 & 0 & 0 \\ -31.5 & 0 & 0 & 0 & 31.5 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{array} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} -2000 \\ R_{1y} \\ R_{2x} \\ R_{2y} \\ R_{3x} \\ R_{3y} \end{Bmatrix} \cancel{10^3}.$$

Using the elimination method for applying boundary conditions,

i.e., $$u_2 = v_2 = u_3 = v_3 = 0\cdot$$

Then the above matrix reduces to,

$$\begin{bmatrix} 47.63 & -12.1 \\ -12.1 & 9.1 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} = \begin{bmatrix} -2000 \\ R_{1y} \end{bmatrix}.$$

We know that $v_1 = -25$ mm, substitute this in the above matrix, then,

$$\begin{bmatrix} 47.63 & -12.1 \\ -12.1 & 9.1 \end{bmatrix} \begin{bmatrix} u_1 \\ -25 \end{bmatrix} = \begin{bmatrix} -2000 \\ R_{1y} \end{bmatrix}.$$

Solving the above matrix and equations we get,

$$u_1 = -48.34 \text{ mm}.$$

Stress calculation

Stress in element 1 is,

$$\sigma_1 = \frac{E}{L_1}\begin{bmatrix}-\cos\theta & -\sin\theta & \cos\theta & \sin\theta\end{bmatrix}\begin{bmatrix}u_3\\ v_3\\ u_1\\ v_1\end{bmatrix} = \frac{210\times10^3}{4000}\begin{bmatrix}-c & -s & c & s\end{bmatrix}\begin{bmatrix}u_3\\ v_3\\ u_1\\ v_1\end{bmatrix}$$

$$\sigma_1 = \frac{210\times10^3}{4000}\begin{bmatrix}-1 & 0 & 1 & 0\end{bmatrix}\begin{bmatrix}0\\ 0\\ -48.34\\ -25\end{bmatrix} = -2537.85 \text{ MPa.}$$

Stress in element 2 is,

$$\sigma_2 = \frac{E}{L_2}\begin{bmatrix}-\cos\theta & -\sin\theta & \cos\theta & \sin\theta\end{bmatrix}\begin{bmatrix}u_2\\ v_2\\ u_1\\ v_1\end{bmatrix} = \frac{210\times10^3}{5000}\begin{bmatrix}-c & -s & c & s\end{bmatrix}\begin{bmatrix}u_2\\ v_2\\ u_1\\ v_1\end{bmatrix}$$

$$\sigma_1 = \frac{210\times10^3}{5000}\begin{bmatrix}-0.8 & 0.6 & 0.8 & -0.6\end{bmatrix}\begin{bmatrix}0\\ 0\\ -48.34\\ -25\end{bmatrix} = -994.22 \text{ MPa.}$$

(II) Software results.

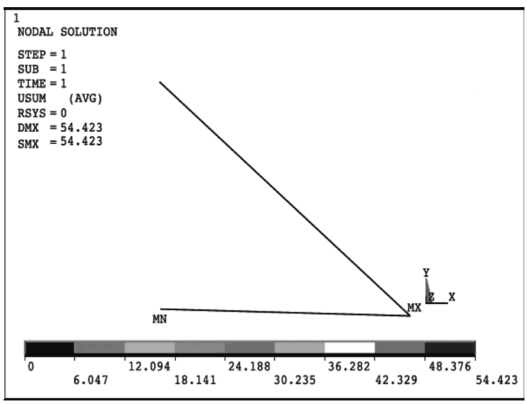


FIGURE 4.4(b) Deflection pattern for a truss for Example 4.3 (refer to Appendix D for color figures).

Deflection value at nodes

The following degree of freedom results are in global coordinates system

NODE	UX	UY	UZ	USUM
1	−48.341	−25.000	0.0000	54.423
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000	0.0000

Maximum absolute values

NODE	1	1	0	1
VALUE	−48.341	−25.000	0.0000	54.423

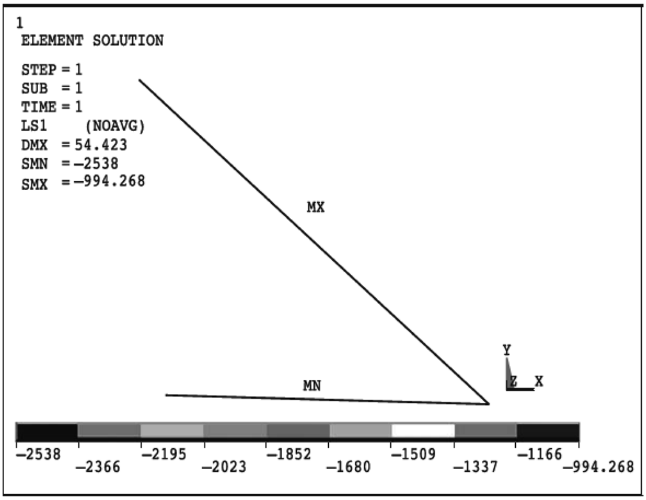


FIGURE 4.4(c) Stress pattern for a truss for Example 4.3 (refer to Appendix D for color figures).

Stress values of elements

STAT	CURRENT
ELEM	LS1
1	−2537.9
2	−994.27

ANSWERS FOR EXAMPLE 4.2

Parameter	FEM-Hand calculations	Software results
Displacement of node 1 in		
x-direction	−48.34 mm	−48.341 mm
y-direction	−25 mm	−25 mm
Stress in element 1	−2537.85 MPa	−2537.9 MPa
Stress in element 2	−994.22 MPa	−994.27 MPa

Procedure for solving the problems using ANSYS ® 12.0 academic teaching software
For Example 4.3

PREPROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Structural Link > 2D spar 1 > OK > Close**

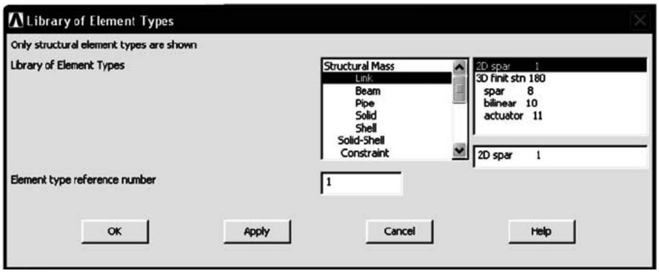


FIGURE 4.5 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

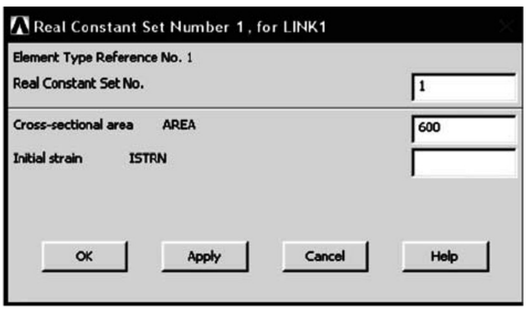


FIGURE 4.6 Enter the cross-sectional area.

Cross-sectional area AREA > **Enter 600 > OK > Close**

Enter the material properties.

3. **Main Menu > Preprocessor > Material Props > Material Models** Material Model Number 1, Click **Structural > Linear > Elastic > Isotropic** Enter **EX=2.1E5 and PRXY =0.3 > OK**
(Close the Define Material Model Behavior window.)
Create the nodes and elements as shown in the figure.
4. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply** Enter the coordinates of node 2 > **Apply** Enter the coordinates of node 3 > **OK**

	Node locations	
Node number	**X-coordinate**	**Y-coordinate**
1	0	0
2	–4000	3000
3	–4000	0

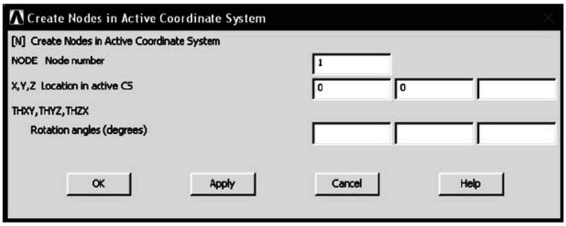


FIGURE 4.7 Enter the node coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **Apply** Pick the 1st and 3rd node > **OK**

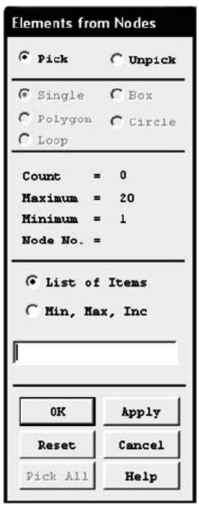


FIGURE 4.8 Pick the nodes to create elements.

Apply the displacement boundary conditions and loads.

6. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 2nd and 3rd node > **Apply > All DOF=0 > OK**
7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st node > **Apply > UY=–25 > OK**
8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes** Pick the 1st node > **OK > Force/Moment value=–2000e3 > OK**

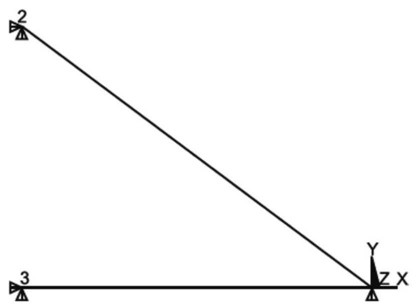


FIGURE 4.9 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First to be safe, save the model.

Solution

The interactive solution proceeds.

9. **Main Menu > Solution > Solve > Current LS > OK**
   The **/STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window and if all is OK, select **File > Close.**

   In the **Solve Current Load Step** window, select **OK,** and the solution is complete, **close** the '**Solution is Done!'** window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK**
    This result is shown in Figure 4.4(b).

To find the axial stress, the following procedure is followed.

11. MAIN Menu > General Postproc > Element Table > Define Table > Add

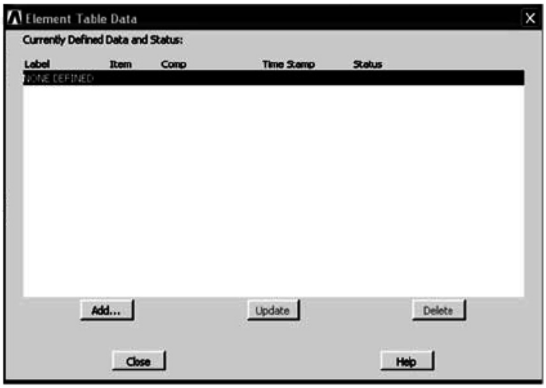


FIGURE 4.10 Defining the element table.

Select **By sequence num and LS** and type 1 **after LS** as shown in Figure 4.11.**>OK**

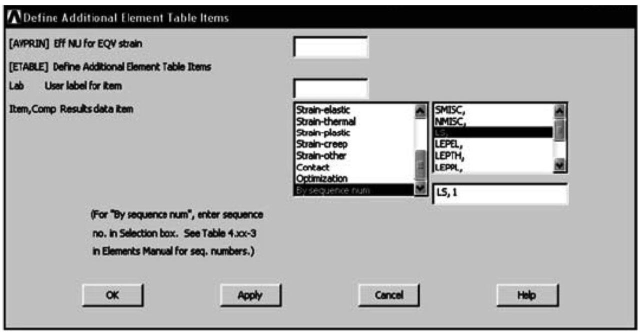


FIGURE 4.11 Selecting options in element table.

12. Main Menu > General Postproc > Plot Results > Contour Plot > Elem Table > Select LS1 > OK

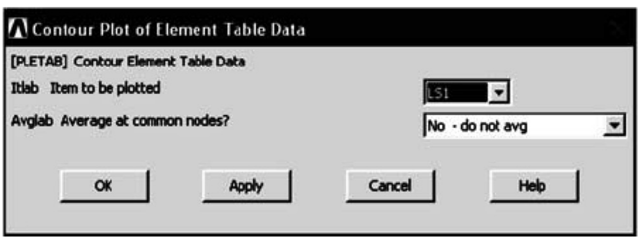


FIGURE 4.12 Selecting options for finding out axial stress.

This result is shown in Figure 4.4(c).

EXERCISES

1. For a 5 bar truss shown in Figure 4.13, determine the following:
   a. nodal displacements
   b. stresses in each element
   c. reaction forces.

   Take E = 200 GPa and Area A = 750 mm^2 for all elements.

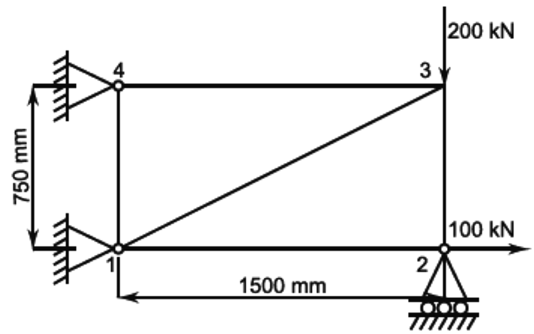


FIGURE 4.13 Exercise 1.

2. For the 3 bar truss shown in Figure 4.14, determine the displacement of node 1 and the stresses in elements. Take A= 300 mm^2 and E = 210 GPa.

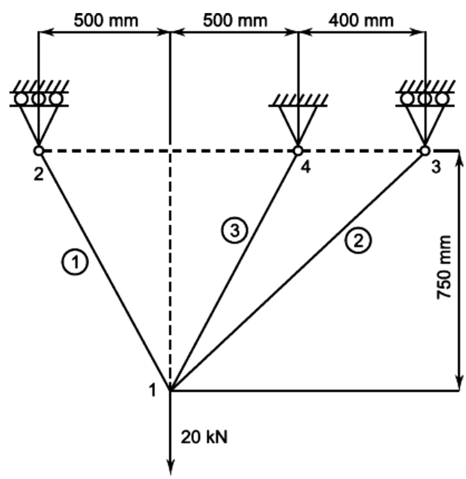


FIGURE 4.14 Exercise 2.

3. Consider the truss shown in Figure 4.15, determine the nodal displacements, element stresses, and reactions. Take E = 200 GPa. $A_1 = A_2 = A_3 = 500\ \text{mm}^2,\ P_1 = 300\ \text{kN},\ P_2 = 200\ \text{kN}\cdot$

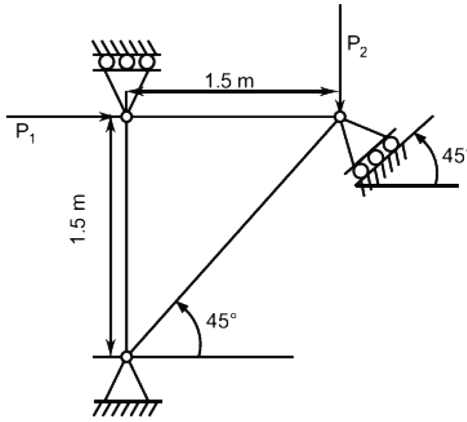


FIGURE 4.15 Exercise 3.

4. Consider the truss structure shown in Figure 4.16, determine the stresses of the truss structure. Take all members have elastic modulus (E) of 210 GPa and cross-sectional area (A) of 250 mm^2.

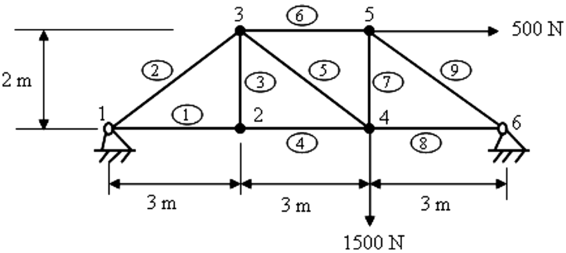


FIGURE 4.16 Exercise 4.

5. Consider the truss structure shown in Figure 4.17, derive the finite element matrix equations using 2 elements. Determine the displacements and the stresses in the member. Assume all members have elastic modulus (E) of 200 GPa and cross-sectional area (A) of 300 mm^2.

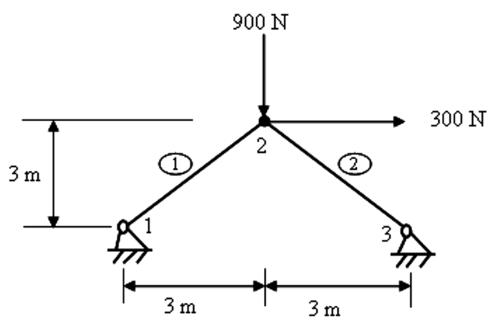


FIGURE 4.17 Exercise 5.

6. Consider the truss structure shown in Figure 4.18, determine the nodal displacement and the element forces assuming that all elements have the same AE.

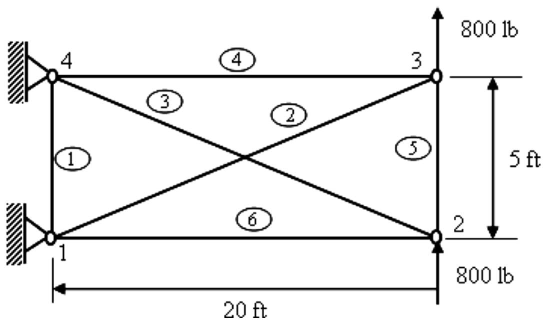


FIGURE 4.18 Exercise 6.

7. Determine the nodal displacements, element stresses, and support reactions for the 3 member truss shown in Figure 4.19. Take $A_1 = 10$ in^2, $A_2 = 15$ in^2, $A_3 = 10$ in^2 and E = 20 msi for all members.

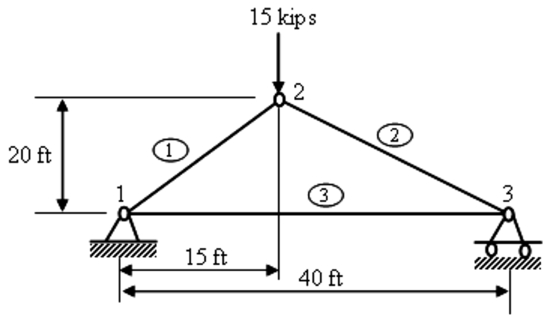


FIGURE 4.19 Exercise 7.

8. Determine the nodal displacements, element stresses and support reactions for the three member truss shown in Figure 4.20. Take $A_1 = 1$ in^2, $A_2 = 2$ in^2, $A_3 = 3$ in^2, and $E = 30$ Mlb/in^2 for all members.

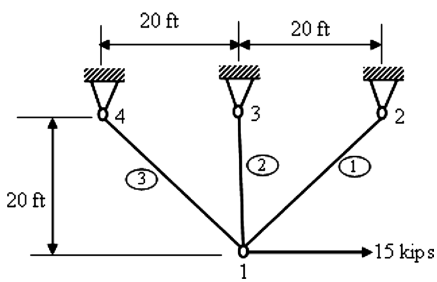


FIGURE 4.20 Exercise 8.

9. Determine the nodal displacements, element stresses, and support reactions for the 3 member truss shown in Figure 4.21. Take $A_1 = 6\ \text{cm}^2$, $A_2 = 8\ \text{cm}^2$, $A_2 = 8\ \text{cm}^2$, and $E = 20\ \text{MN/cm}^2$ for all members.

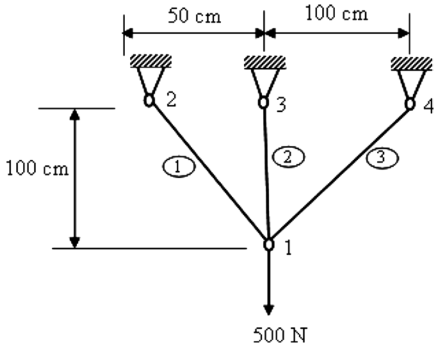


FIGURE 4.21 Exercise 9.

10. For the bar shown in Figure 4.22, determine the axial stress. Let $A = 6 \times 10^{-5}\ \text{m}^2$, E = 220 GPa, and L = 4 m, and let the angle between x and x' be 45°. Assume the global displacements have been previously determined to be u_1 = 0.46 mm, v_1 = 0.0, u_2 = 0.70 mm, and v_2 = 0.90 mm.

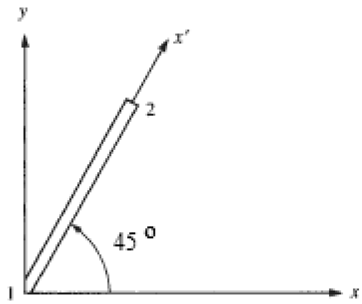


FIGURE 4.22 Exercise 10.

REFERENCES

1. Y. W. Hwon and H. Bang, "The Finite Element Method Using MATLAB," Second Edition, CRC Press, 2000.

2. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, Cengage Learning, 2012.

3. S. Moaveni, "Finite Element Analysis: Theory and Application with ANSYS," Third Edition, Prentice Hall, 2008.
4. J. N. Reddy, "An Introduction to the Finite Element Method," Third Edition, McGraw Hill Higher Education, 2004.
5. C. T. F. Ross, "Finite Element Method in Structural Mechanics," Ellis Horwood Limited Publishers, 1985.
6. F. L. Stasa, "Applied Finite Element Analysis for Engineering," Holt, Rinehart and Winston, 1985.
7. L. J. Segerlind, "Applied Finite Element Analysis," Second Edition, John Wiley and Sons, 1984.

CHAPTER 5

FINITE ELEMENT ANALYSIS OF BEAMS

5.1 INTRODUCTION

Beam is a very common structure in many engineering applications because of its efficient load-carrying capability. Beam, by definition, is a transversely loaded structural member. Beam element is used in the analysis of beams.

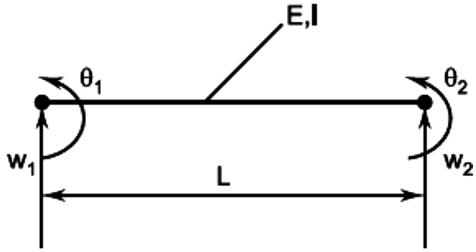


FIGURE 5.1 Beam element.

This element has 2 end nodes each having 2 degrees of freedom, namely transverse displacement and slope. Beam element gives accurate results if acted upon by nodal forces and moments. A greater number of small elements will be necessary in the case of a beam acted upon by distributed loads in order to get good results. The interpolation equation and element stiffness matrix for beam element are given by

$$w = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{Bmatrix} \tag{5.1}$$

$$[K]=\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}. \tag{5.2}$$

5.2 SIMPLY SUPPORTED BEAMS

EXAMPLE 5.1

For the beam shown in Figure 5.2, determine the nodal displacements, slope, and reactions. Take $E = 210$ GPa and $I = 4\times10^{-4}$ m^4.

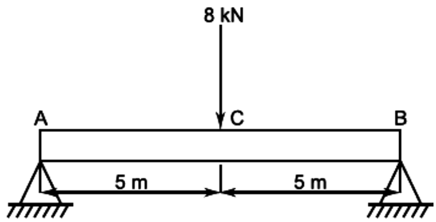


FIGURE 5.2 The beam for Example 5.1.

Solution

(I) Analytical method [Refer to Figure 5.2].

$$\begin{aligned} L &= 10 \text{ m} \\ P &= 8 \text{ kN} \end{aligned}.$$

Deflection,

$$\delta_C = -\frac{PL^3}{48EI} = -\frac{8\times10^3\times(10)^3}{48\times210\times10^9\times4\times10^{-4}} = -1.98\times10^{-3} \text{ m} = -1.98 \text{ mm}$$

$$|\theta_C| = |\theta_B| = \frac{PL^2}{16EI} = \frac{8\times10^3\times(10)^2}{16\times210\times10^9\times4\times10^{-4}} = 5.95\times10^{-4} \text{ rad}$$

$\theta_C = 0$, by symmetry.

Reaction,

$$R_A = R_B = \frac{8}{2} = 4 \text{ kN}.$$

(II) FEM by hand calculations [Refer to Figure 5.2(a)].

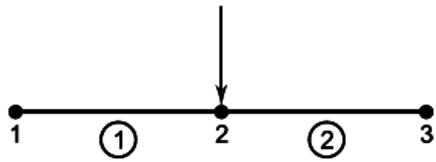


FIGURE 5.2(a) Finite element model for Example 5.1.

Element stiffness matrices are,

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

$$[k_1]=\frac{210\times10^9\times4\times10^{-4}}{(5)^3}\begin{bmatrix}12 & 6(5) & -12 & 6(5)\\ 6(5) & 4(5)^2 & -6(5) & 2(5)^2\\ -12 & -6(5) & 12 & -6(5)\\ 6(5) & 2(5)^2 & -6(5) & 4(5)^2\end{bmatrix}$$

$$[k_1]=672\times10^3\begin{array}{c}\begin{array}{cccc}w_1 & \theta_1 & w_2 & \theta_2\end{array}\\ \begin{bmatrix}12 & 30 & -12 & 30\\ 30 & 100 & -30 & 50\\ -12 & -30 & 12 & -30\\ 30 & 50 & -30 & 100\end{bmatrix}\end{array}\begin{array}{c}\\ w_1\\ \theta_1\\ w_2\\ \theta_2\end{array}.$$

Due to symmetry,

$$[k_1]=[k_2]$$

$$[k_2]=672\times10^3\begin{array}{c}\begin{array}{cccc}w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 30 & -12 & 30\\ 30 & 100 & -30 & 50\\ -12 & -30 & 12 & -30\\ 30 & 50 & -30 & 100\end{bmatrix}\end{array}\begin{array}{c}\\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{5.3}$$

$$672\times10^{3}\begin{array}{cc} \begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{array} & \\ \begin{bmatrix} 12 & 30 & -12 & 30 & 0 & 0 \\ 30 & 100 & -30 & 50 & 0 & 0 \\ -12 & -30 & 12+12 & -30+30 & -12 & 30 \\ 30 & 50 & -30+30 & 100+100 & -30 & 50 \\ 0 & 0 & -12 & -30 & 12 & -30 \\ 0 & 0 & 30 & 50 & -30 & 100 \end{bmatrix} & \begin{array}{c} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array} \end{array} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 0 \\ -8\times10^3 \\ 0 \\ R_3 \\ 0 \end{Bmatrix}.$$

Using the elimination method for applying boundary conditions,

$$w_1 = w_3 = 0.$$

The above matrix reduces to

$$672\times10^{3}\begin{array}{c} \begin{array}{cccc} \theta_1 & \mathrm{w}_2 & \theta_2 & \theta_3 \end{array} \\ \begin{bmatrix} 100 & -30 & 50 & 0 \\ -30 & 24 & 0 & 30 \\ 50 & 0 & 200 & 50 \\ 0 & 30 & 50 & 100 \end{bmatrix} \end{array} \begin{Bmatrix} \theta_1 \\ w_2 \\ \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -8\times10^3 \\ 0 \\ 0 \end{Bmatrix}.$$

By solving the above equations, we get,

$$w_2 = -0.002 \text{ m} = -2 \text{ mm},$$

$$\theta_1 = -0.0006 \text{ rad}, \theta_2 = 0 \text{ rad, and } \theta_3 = 0.0006 \text{ rad}.$$

Reaction calculation

$$672\times10^3\left(30\times\theta_1 - 12\times w_2\right) = R_1$$

$$672\times10^3\left(30\times(-0.0006) - 12\times(-0.002)\right) = R_1$$

$$R_1 = 4.032 \text{ kN}$$

$$672\times10^3\left(-12\times w_2 - 30\times\theta_3\right) = R_3$$

$$672\times10^3\left(-12\times(-0.002) - 30\times(0.0006)\right) = R_3$$

$$R_3 = 4.032 \text{ kN}.$$

(III) Software results.

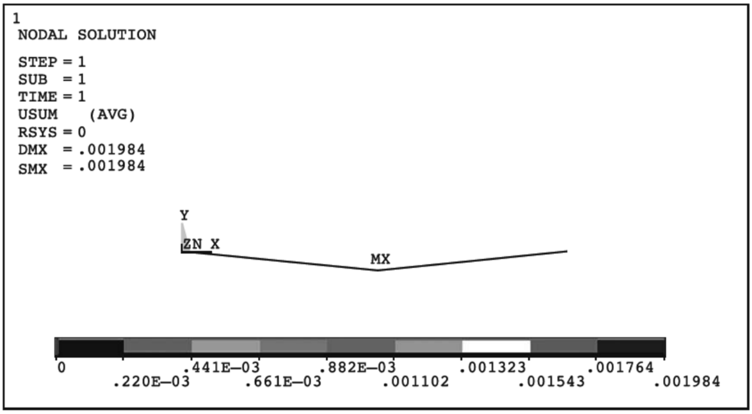


FIGURE 5.2(b) Deflection pattern for a simply supported beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	−0.19841E−02	0.0000	−0.19841E−02
3	0.0000	0.0000	0.0000	0.0000

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	−0.59524E−03
2	0.0000
3	0.59524E−03

Reaction values

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	4000.0	
3	0.0000	4000.0	

ANSWERS FOR EXAMPLE 5.1

Parameter	Analytical method	FEM-Hand calculations	Software results
Displacement at node 2	–1.98 mm	–2 mm	–1.9841 mm
Slope at node			
1	-5.95×10^{-4} rad	–0.0006 rad	-0.59524×10^{-3} rad
2	0	0	0
3	-5.95×10^{-4} rad	–0.0006 rad	0.59524×10^{-3} rad
Reaction at node			
1	4 kN	4.032 kN	4 kN
3	4 kN	4.032 kN	4 kN

EXAMPLE 5.2

For the beam shown in Figure 5.3, determine displacements, slopes, reactions, maximum bending moment, shear force, and maximum bending stress. Take $E = 210$ GPa and $I = 2\times10^{-4}$ m^4. The beam has rectangular cross-section of depth $h = 1$ m.

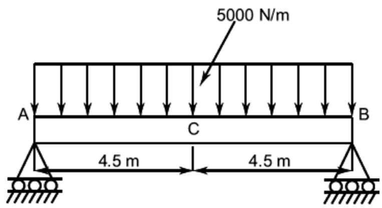


FIGURE 5.3 The beam for Example 5.2.

Solution

(I) Analytical method [Refer to Figure 5.3].

Reaction,

$$R_A = R_B = \frac{5000\times9}{2} = 22500 \text{ N} = 22.5 \text{ kN}$$

$$\delta_C = -\frac{5PL^4}{388EI} = -\frac{5\times5000\times(9)^4}{384\times210\times10^9\times2\times10^{-4}} = -0.0102 \text{ m} = -10.2 \text{ mm}$$

$$|\theta_A| = |\theta_B| = \frac{PL^3}{24EI} = \frac{5000\times(9)^3}{24\times210\times10^9\times2\times10^{-4}} = 3.62\times10^{-3} \text{ rad}$$

$\theta_C = 0$, by symmetry.

Maximum bending moment,

$$M_{\max} = \frac{PL^2}{8} = \frac{5000 \times (9)^2}{8} = 50625 \text{ N-m.}$$

Shear force,

$$SF = \frac{PL}{2} = \frac{5000 \times 9}{2} = 22500 \text{ N.}$$

Maximum bending stress,

$$f_{\max} = \frac{M_{\max}}{I} \times y_{\max} \tag{5.4}$$

$$y_{\max} = \frac{h}{2} = \frac{1}{2} = 0.5 \text{ m}$$

$$f_{\max} = \frac{50625}{2 \times 10^{-4}} \times 0.5 = 126.56 \text{ MPa}$$

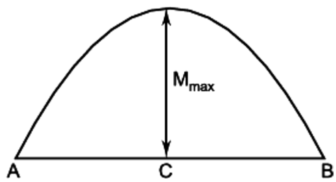


FIGURE 5.3(a) Bending moment diagram.

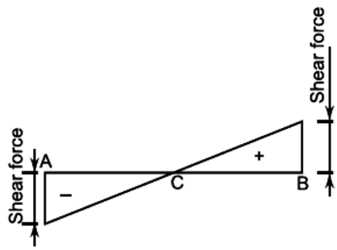


FIGURE 5.3(b) Shear force diagram.

(II) FEM by hand calculations [Refer to Figure 5.3(c)].

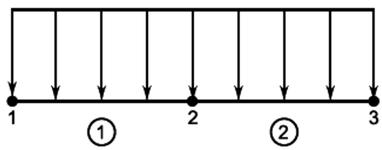


FIGURE 5.3(c) Finite element model for Example 5.2.

Stiffness matrices are,

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

$$[k_1]=\frac{210\times10^9\times2\times10^{-4}}{(4.5)^3}\begin{bmatrix} 12 & 6(4.5) & -12 & 6(4.5) \\ 6(4.5) & 4(4.5)^2 & -6(4.5) & 2(4.5)^2 \\ -12 & -6(4.5) & 12 & -6(4.5) \\ 6(4.5) & 2(4.5)^2 & -6(4.5) & 4(4.5)^2 \end{bmatrix}$$

$$[k_1]=460905.35\begin{array}{c} \begin{matrix} w_1 & \theta_1 & w_2 & \theta_2 \end{matrix} \\ \begin{bmatrix} 12 & 27 & -12 & 27 \\ 27 & 81 & -27 & 40.5 \\ -12 & -27 & 12 & -27 \\ 27 & 40.5 & -27 & 81 \end{bmatrix} \end{array}\begin{matrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{matrix}.$$

Due to symmetry,

$$[k_1]=[k_2]$$

$$[k_2]=460905.35\begin{array}{c} \begin{matrix} w_2 & \theta_2 & w_3 & \theta_3 \end{matrix} \\ \begin{bmatrix} 12 & 27 & -12 & 27 \\ 27 & 81 & -27 & 40.5 \\ -12 & -27 & 12 & -27 \\ 27 & 40.5 & -27 & 81 \end{bmatrix} \end{array}\begin{matrix} w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{matrix}.$$

Nodal force calculation

For element 1,

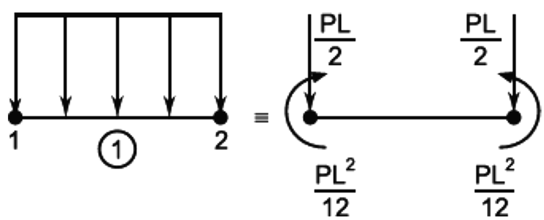


FIGURE 5.3(d) Nodal force calculation for element 1 in Example 5.2.

Nodal forces and moments for element 1 is,

$$\{F_1\}=\begin{Bmatrix}-\dfrac{PL}{2}\\-\dfrac{PL^2}{12}\\-\dfrac{PL}{2}\\\dfrac{PL^2}{12}\end{Bmatrix}=\begin{Bmatrix}-\dfrac{5000\times4.5}{2}\\-\dfrac{5000\times(4.5)^2}{12}\\-\dfrac{5000\times4.5}{2}\\\dfrac{5000\times(4.5)^2}{12}\end{Bmatrix}=\begin{Bmatrix}-11250\\-8437.5\\-11250\\8437.5\end{Bmatrix}\begin{matrix}f_1\\m_1\\f_2\\m_2\end{matrix}.$$

For element 2,

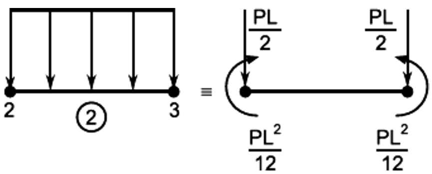


FIGURE 5.3(e) Nodal force calculation for element 2 in Example 5.2.

Due to symmetry,

$$\{F_1\}=\{F_2\}$$

$$\{F_2\}=\begin{Bmatrix}-11250\\-8437.5\\-11250\\8437.5\end{Bmatrix}\begin{matrix}f_2\\m_2\\f_3\\m_3\end{matrix}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{5.5}$$

$$460905.35\begin{array}{c}\begin{matrix}w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3\end{matrix}\\ \begin{bmatrix}12 & 27 & -12 & 27 & 0 & 0\\27 & 81 & -27 & 40.5 & 0 & 0\\-12 & -27 & 12+12 & -27+27 & -12 & 27\\27 & 40.5 & -27+27 & 81+81 & -27 & 40.5\\0 & 0 & -12 & -27 & 12 & -27\\0 & 0 & 27 & 40.5 & -27 & 81\end{bmatrix}\end{array}\begin{matrix}w_1\\\theta_1\\w_2\\\theta_2\\w_3\\\theta_3\end{matrix}\begin{Bmatrix}w_1\\\theta_1\\w_2\\\theta_2\\w_3\\\theta_3\end{Bmatrix}=\begin{Bmatrix}-11250+R_{1y}\\-8437.5\\-11250-11250\\8437.5-8437.5\\-11250+R_{3y}\\8437.5\end{Bmatrix}.$$

Using the elimination method for applying boundary conditions,

$$w_1 = w_3 = 0.$$

The above matrix reduces to

$$\begin{array}{c} \begin{array}{cccc} \theta_1 & \mathrm{w}_2 & \theta_2 & \theta_3 \end{array} \\ 460905.35\begin{bmatrix} 81 & -27 & 40.5 & 0 \\ -27 & 24 & 0 & 27 \\ 40.5 & 0 & 162 & 40.5 \\ 0 & 27 & 40.5 & 81 \end{bmatrix}\begin{Bmatrix} \theta_1 \\ w_2 \\ \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -8437.5 \\ -22500 \\ 0 \\ 8437.5 \end{Bmatrix}. \end{array}$$

By solving the above equations, we get,

$$w_2 = -0.0102 \text{ m},$$

$$\theta_1 = -0.0036 \text{ rad}, \ \theta_2 = 0 \text{ rad, and } \theta_3 = 0.0036 \text{ rad}.$$

Reactions are calculated from 1st and 5th rows of global matrix.

$$460905.35\begin{bmatrix} 12 & 27 & -12 & 27 & 0 & 0 \end{bmatrix}\begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = -11250 + R_{1y}$$

$$\therefore \qquad 11615 = -11250 + R_{1y}$$

$$\therefore \qquad R_{1y} = 22865 \text{ N} = 22.865 \text{ kN}.$$

Similarly from 5th row

$$R_{3y} = 22.865 \text{ kN}.$$

(III) Software results.

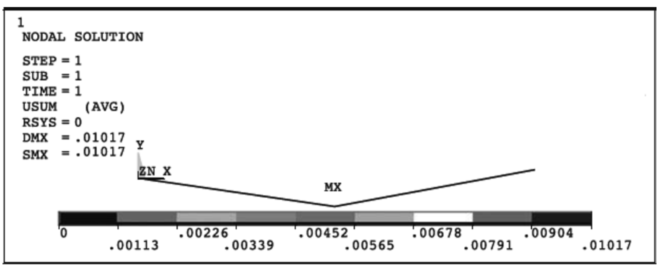


FIGURE 5.3(f) Deflection pattern for a simply supported beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	−0.10170E−01	0.0000	−0.10170E−01
3	0.0000	0.0000	0.0000	0.0000

Slope values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	−0.36161E−02
2	0.0000
3	0.36161E−02

Reaction values

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	22500	
3	0.0000	22500	

Total values

VALUE	0.0000	45000	0.0000

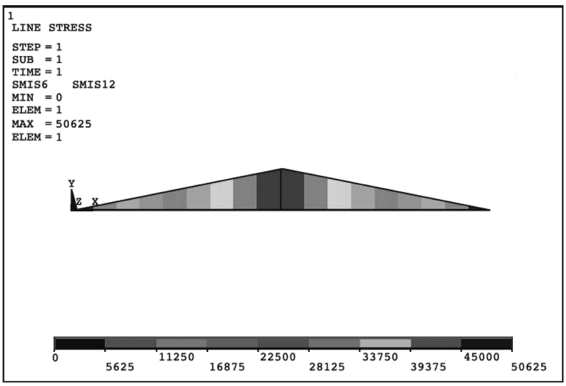


FIGURE 5.3(g) Bending moment diagram for a simply supported beam (refer to Appendix D for color figures).

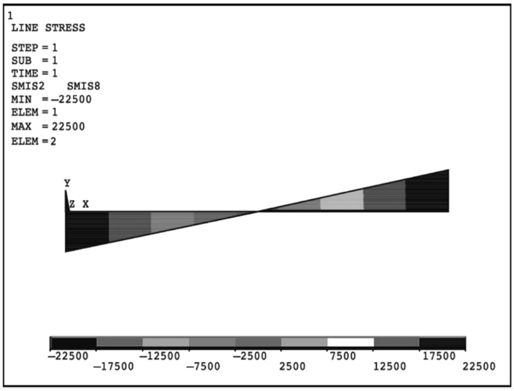


FIGURE 5.3(h) Shear force diagram for a simply supported beam (refer to Appendix D for color figures).

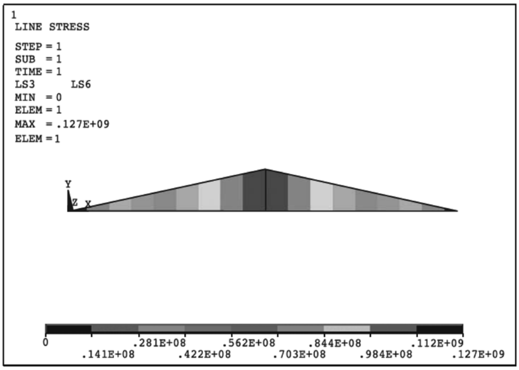


FIGURE 5.3(i) Bending stress for a simply supported beam (refer to Appendix D for color figures).

ANSWERS FOR EXAMPLE 5.2

Parameter	Analytical method	FEM-Hand calculations	Software results
Displacement at node 2	–0.0102 m	–0.0102 m	–0.01017 m
Slope at node			
1	-3.62×10^{-3} rad	0.0036 rad	-0.36161×10^{-2} rad
2	0	0	0
3	3.62×10^{-3} rad	0.0036 rad	0.36161×10^{-2} rad
Reaction at node			
1	22500 N	22865 N	22500 N
3	22500 N	22865 N	22500 N
Maximum bending moment	50625 N-m	……	50625 N-m
Shear force	22500 N	…….	22500 N
Maximum bending stress	126.56 MPa	…….	127 MPa

EXAMPLE 5.3

For the beam shown in Figure 5.4, determine displacements, slopes, and reactions. Take $E = 200$ GPa and $I = 6.25 \times 10^{-4}$ m^4.

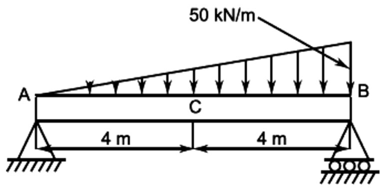


FIGURE 5.4 The beam for Example 5.3.

Solution

(I) Analytical method [Refer to Figure 5.4].

Reaction,

$$R_A = \frac{PL}{6} = \frac{50 \times 10^3 \times 8}{6} = 66666.67 \text{ N} = 66.67 \text{ kN}$$

$$R_B = \frac{PL}{3} = \frac{50 \times 10^3 \times 8}{3} = 133333.33 \text{ N} = 133.33 \text{ kN}$$

$$\theta_A = -\frac{7PL^3}{360EI} = \frac{7 \times 50 \times 10^3 \times (8)^3}{360 \times 200 \times 10^9 \times 6.25 \times 10^{-4}} = -0.00398 \text{ rad}$$

$$\theta_B = -\frac{PL^3}{45EI} = \frac{50 \times 10^3 \times (8)^3}{45 \times 200 \times 10^9 \times 6.25 \times 10^{-4}} = 0.00455 \text{ rad}$$

$$\delta_C = \frac{1}{EI}\left(\frac{PL}{36}x^3 - \frac{P}{120 \times L}x^5 - \frac{7PL^3}{360}x\right)_{x=\frac{L}{2}}$$

$$\delta_C = \frac{1}{EI}\left(\frac{PL}{36} \times \left(\frac{L}{2}\right)^3 - \frac{P}{120 \times L} \times \left(\frac{L}{2}\right)^5 - \frac{7PL^3}{360} \times \left(\frac{L}{2}\right)\right) = \frac{1}{EI}\left(\frac{PL^4}{288} - \frac{PL^4}{3840} - \frac{7PL^4}{720}\right)$$

$$\delta_C = \frac{PL^4}{EI}\left(\frac{40 - 3 - 112}{11520}\right)$$

$$\therefore \quad \delta_C = -\frac{75PL^4}{11520EI} = -\frac{75 \times 50 \times 10^3 \times (8)^4}{11520 \times 200 \times 10^9 \times 6.25 \times 10^{-4}} = -0.01067 \text{ m}$$

$$\theta_C = \frac{1}{EI}\left(\frac{PL}{12}x^2 - \frac{P}{24 \times L}x^4 - \frac{7PL^3}{360}x\right)_{x=\frac{L}{2}}$$

$$\theta_C = \frac{1}{EI}\left(\frac{PL}{12}\times\left(\frac{L}{2}\right)^2 - \frac{P}{24\times L}\times\left(\frac{L}{2}\right)^4 - \frac{7PL^3}{360}\times\left(\frac{L}{2}\right)\right) = \frac{PL^3}{EI}\left(\frac{1}{48} - \frac{1}{384} - \frac{7}{360}\right)$$

$$\therefore \qquad \theta_C = -\frac{1.2153\times10^{-3}\times PL^3}{EI} = -\frac{1.2153\times10^{-3}\times50\times(8)^3}{200\times10^9\times6.25\times10^{-4}} = 2.4889\times10^{-4} \text{ rad.}$$

(II) FEM by hand calculations [Refer to Figure 5.4(a)].

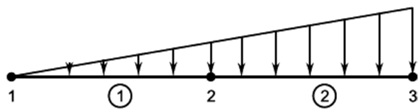


FIGURE 5.4(a) Finite element model for Example 5.3.

Stiffness matrices are,

$$[k_1] = \frac{EI}{L_1^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \frac{200\times10^9\times6.25\times10^{-4}}{(4)^3}\begin{bmatrix} 12 & 6(4) & -12 & 6(4) \\ 6(4) & 4(4)^2 & -6(4) & 2(4)^2 \\ -12 & -6(4) & 12 & -6(4) \\ 6(4) & 2(4)^2 & -6(4) & 4(4)^2 \end{bmatrix}$$

$$[k_1] = 195.3125\times10^4 \begin{array}{c} \begin{array}{cccc} w_1 & \theta_1 & w_2 & \theta_2 \end{array} \\ \begin{bmatrix} 12 & 24 & -12 & 24 \\ 24 & 64 & -24 & 32 \\ -12 & -24 & 12 & -24 \\ 24 & 32 & -24 & 64 \end{bmatrix} \end{array} \begin{array}{c} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{array}.$$

Due to symmetry,

$$[k_1] = [k_2]$$

$$[k_2] = \frac{EI}{L_2^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \frac{200\times10^9\times6.25\times10^{-4}}{(4)^3}\begin{bmatrix} 12 & 6(4) & -12 & 6(4) \\ 6(4) & 4(4)^2 & -6(4) & 2(4)^2 \\ -12 & -6(4) & 12 & -6(4) \\ 6(4) & 2(4)^2 & -6(4) & 4(4)^2 \end{bmatrix}$$

$$[k_2] = 195.3125\times10^4 \begin{array}{c} \begin{array}{cccc} w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\ \begin{bmatrix} 12 & 24 & -12 & 24 \\ 24 & 64 & -24 & 32 \\ -12 & -24 & 12 & -24 \\ 24 & 32 & -24 & 64 \end{bmatrix} \end{array} \begin{array}{c} w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array}.$$

Global stiffness matrix is,

$$[K]=195.3125\times10^{4}\begin{array}{cc} \begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{array} & \\ \begin{bmatrix} 12 & 24 & -12 & 24 & 0 & 0\\ 24 & 64 & -24 & 32 & 0 & 0\\ -12 & -24 & 12+12 & -24+24 & -12 & 24\\ 24 & 32 & -24+24 & 64+64 & -24 & 32\\ 0 & 0 & -12 & -24 & 12 & -24\\ 0 & 0 & 24 & 32 & -24 & 64 \end{bmatrix} & \begin{array}{c} w_1\\ \theta_1\\ w_2\\ \theta_2\\ w_3\\ \theta_3 \end{array} \end{array}.$$

Load vector,

$$\{F\}=\frac{L}{20}\begin{Bmatrix} 7P_1+3P_2\\ \frac{L}{3}(3P_1+2P_2)\\ 3P_1+7P_2\\ -\frac{L}{3}(2P_1+3P_2) \end{Bmatrix}.$$

For element 1,

$$P_1=0,\ P_2=-25\ \text{kN/m},\ \text{L}=4\ \text{m}$$

$$\{F_1\}=\frac{4}{20}\begin{Bmatrix} -75\\ \frac{4}{3}(-50)\\ -175\\ -\frac{4}{3}(-75) \end{Bmatrix}=\begin{Bmatrix} -15\ \text{kN}\\ -13.33\ \text{kN-m}\\ -35\ \text{kN}\\ 20\ \text{kN-m} \end{Bmatrix}=\begin{Bmatrix} -15000\ \text{N}\\ -1333\ \text{N-m}\\ -35000\ \text{N}\\ 2000\ \text{N-m} \end{Bmatrix}.$$

For element 2,

$$P_1=-25\ \text{kN/m},\ P_2=-50\ \text{kN/m},\ \text{L}=4\ \text{m}$$

$$\{F_2\}=\frac{4}{20}\begin{Bmatrix} -175-150\\ \frac{4}{3}(-75-100)\\ -75-350\\ -\frac{4}{3}(-50-150) \end{Bmatrix}=\begin{Bmatrix} -65000\ \text{N}\\ -46667\ \text{N-m}\\ -85000\ \text{N}\\ 53333\ \text{N-m} \end{Bmatrix}.$$

Global load vector is,

$$\{F\}=\begin{Bmatrix}-15000\\-13333\\-100000\\-26667\\-85000\\53333\end{Bmatrix}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{5.6}$$

$$195.3125\times10^4\begin{array}{c}\begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 24 & -12 & 24 & 0 & 0\\ 24 & 64 & -24 & 32 & 0 & 0\\ -12 & -24 & 12+12 & -24+24 & -12 & 24\\ 24 & 32 & -24+24 & 64+64 & -24 & 32\\ 0 & 0 & -12 & -24 & 12 & -24\\ 0 & 0 & 24 & 32 & -24 & 64\end{bmatrix}\end{array}\begin{array}{c} w_1\\ \theta_1\\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}\begin{Bmatrix}w_1\\ \theta_1\\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{Bmatrix}=\begin{Bmatrix}-15000+R_1\\-13333\\-100000\\-26667\\-85000+R_3\\53333\end{Bmatrix}.$$

Using the elimination method for applying boundary conditions,

$$w_1=w_3=0.$$

The above matrix reduces to

$$195.3125\times10^4\begin{array}{c}\begin{array}{cccc}\theta_1 & \mathrm{w}_2 & \theta_2 & \theta_3\end{array}\\ \begin{bmatrix}64 & -24 & 32 & 0\\ -24 & 24 & 0 & 24\\ 32 & 0 & 128 & 32\\ 0 & 24 & 32 & 64\end{bmatrix}\end{array}\begin{Bmatrix}\theta_1\\ w_2\\ \theta_2\\ \theta_3\end{Bmatrix}=\begin{Bmatrix}-13333\\-100000\\-26667\\53333\end{Bmatrix}.$$

By solving the above equations, we get,

$$\begin{Bmatrix}\theta_1\\ w_2\\ \theta_2\\ \theta_3\end{Bmatrix}=\begin{Bmatrix}-0.00398\text{ rad}\\-0.01067\text{ m}\\-0.00025\text{ rad}\\0.00455\text{ rad}\end{Bmatrix}.$$

Reaction calculation

$$1953125(12\times w_1 + 24\times\theta_1 - 12\times w_2 + 24\times\theta_2) = -15000 + R_1$$
$$R_1 = 66796.875 \text{ N} = 66.79 \text{ kN}$$

$$1953125(-12(-0.01067) - 24\times(-0.00025) - 24\times(0.00455)) = -85000 + R_3$$
$$R_3 = 133515.63 \text{ N} = 133.52 \text{ kN}.$$

(III) Software results.

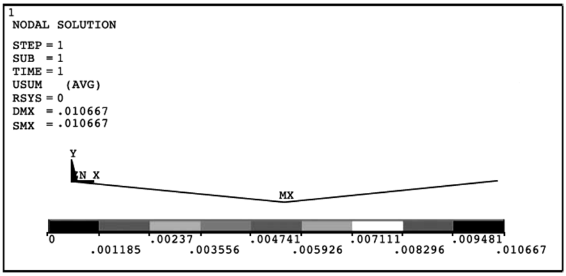


FIGURE 5.4(b) Deflection pattern for a simply supported beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	−0.10667E−01	0.0000	0.10667E−01
3	0.0000	0.0000	0.0000	0.0000

Maximum absolute values

NODE	0	2	0	2
VALUE	0.0000	−0.10667E−01	0.0000	0.10667E−01

Slope values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	−0.39822E−02
2	−0.24889E−03
3	0.45511E−02

Reaction values

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	66667	
3	0.0000	0.13333E +06	

Total values

VALUE	0.0000	0.20000E + 06	0.0000

ANSWERS FOR EXAMPLE 5.3

Parameter	Analytical method	FEM-Hand calculations	Software results
Displacement at node 2	–0.01067 m	–0.01067 m	–0.010667 m
Slope at node			
1	–0.00398 rad	–0.00398 rad	–rad
2	-2.4889×10^{-4} rad	–0.00025 rad	–0.00024889 rad
3	0.00455 rad	0.00455 rad	0.0045511 rad
Reaction at node			
1	66.67 kN	66.79 kN	66.667 kN
3	133.33 kN	133.52 kN	133.33 kN

EXAMPLE 5.4

Calculate the maximum deflection in the beam shown in Figure 5.5. Take E = 200 GPa.

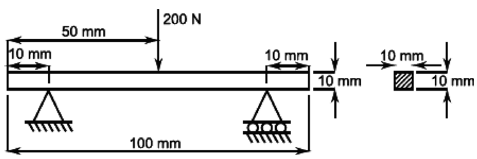


FIGURE 5.5 The beam for Example 5.4.

Solution

(I) Analytical method [Refer to Figure 5.5(a)].

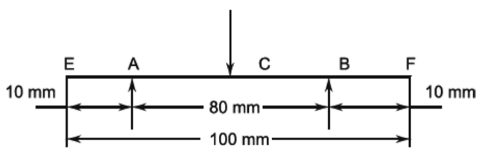


FIGURE 5.5(a) Analytical method for Example 5.4.

$$I = \frac{bh^3}{12}$$

$$I = \frac{0.01 \times (0.01)^3}{12} = 8.333 \times 10^{-10} \text{ m}^4$$

$$\delta_C = \frac{PL^2}{48EI} = -\frac{200 \times (0.08)^3}{48EI} = -\frac{2.133 \times 10^{-3}}{EI} = -\frac{2.133 \times 10^{-3}}{200 \times 10^9 \times 8.33 \times 10^{-10}}$$

$$\delta_C = -1.2803 \times 10^{-5} \text{ m} = -0.0128 \text{ mm}$$

$$\theta_C = 0$$

$$|\theta_A| = |\theta_B| = \frac{PL^2}{16EI} = -\frac{200 \times (0.08)^2}{16EI} = \frac{0.08}{EI} = \frac{0.08}{200 \times 10^9 \times 8.33 \times 10^{-10}} = 4.802 \times 10^{-4} \text{ rad}$$

$$\delta_E = \delta_F = \theta_B \times BF$$

$$\delta_E = \delta_F = 4.802 \times 10^{-4} \times 10 = 4.802 \times 10^{-3} \text{ mm.}$$

(II) FEM by hand calculations [Refer to Figure 5.5(b)].

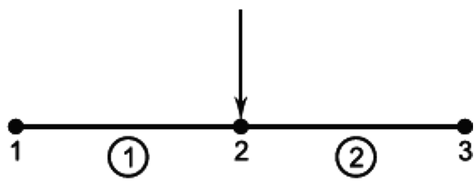


FIGURE 5.5(b) Finite element model for Example 5.4.

For beam,

$$I = \frac{bh^3}{12}$$

$$I = \frac{10 \times (10)^3}{12} = 833.34 \text{ mm}^4.$$

For element 1 and 2, L = 40 mm

$$[k_1] = \frac{EI}{L}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \frac{200 \times 10^3 \times 833.34}{(40)^3}\begin{bmatrix} 12 & 6(40) & -12 & 6(40) \\ 6(40) & 4(40)^2 & -6(4) & 2(40)^2 \\ -12 & -6(40) & 12 & -6(40) \\ 6(40) & 2(40)^2 & -6(40) & 4(40)^2 \end{bmatrix}$$

$$[k_1] = 2604.1875 \begin{array}{c} \begin{array}{cccc} w_1 & \theta_1 & w_2 & \theta_2 \end{array} \\ \begin{bmatrix} 12 & 240 & -12 & 240 \\ 240 & 6400 & -240 & 3200 \\ -12 & -240 & 12 & -240 \\ 240 & 3200 & -240 & 6400 \end{bmatrix} \end{array} \begin{array}{c} \\ w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{array}.$$

Due to symmetry,

$$[k_1] = [k_2]$$

$$[k_2] = \frac{EI}{L} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \frac{200 \times 10^3 \times 833.34}{(40)^3} \begin{bmatrix} 12 & 6(40) & -12 & 6(40) \\ 6(40) & 4(40)^2 & -6(4) & 2(40)^2 \\ -12 & -6(40) & 12 & -6(40) \\ 6(40) & 2(40)^2 & -6(40) & 4(40)^2 \end{bmatrix}$$

$$[k_2] = 2604.1875 \begin{array}{c} \begin{array}{cccc} w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\ \begin{bmatrix} 12 & 240 & -12 & 240 \\ 240 & 6400 & -240 & 3200 \\ -12 & -240 & 12 & -240 \\ 240 & 3200 & -240 & 6400 \end{bmatrix} \end{array} \begin{array}{c} \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array}.$$

Global stiffness matrix is,

$$[K] = 2604.1875 \begin{array}{c} \begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\ \begin{bmatrix} 12 & 240 & -12 & 240 & 0 & 0 \\ 240 & 6400 & -24 & 3200 & 0 & 0 \\ -12 & -240 & 12+12 & -240+240 & -12 & 240 \\ 240 & 3200 & -240+240 & 6400+6400 & -240 & 3200 \\ 0 & 0 & -12 & -240 & 12 & -240 \\ 0 & 0 & 240 & 3200 & -240 & 6400 \end{bmatrix} \end{array} \begin{array}{c} \\ w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array}.$$

Global load vector is,

$$\{F\} = \begin{Bmatrix} R_1 \\ 0 \\ -200 \\ 0 \\ R_3 \\ 0 \end{Bmatrix}.$$

Global equation is,

$$[K]\{r\}=\{R\} \tag{5.7}$$

$$2604.1875\begin{array}{c} \begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\ \left[\begin{array}{cccccc} 12 & 240 & -12 & 240 & 0 & 0 \\ 240 & 6400 & -240 & 3200 & 0 & 0 \\ -12 & -240 & 24 & 0 & -12 & 240 \\ 240 & 3200 & 0 & 12800 & -240 & 3200 \\ 0 & 0 & -12 & -240 & 12 & -240 \\ 0 & 0 & 240 & 3200 & -240 & 6400 \end{array}\right] \end{array} \begin{array}{c} \\ w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 0 \\ -200 \\ 0 \\ R_3 \\ 0 \end{Bmatrix}.$$

Using the elimination method for applying boundary conditions,

$$w_1 = w_3 = 0.$$

The above matrix reduces to

$$2604.1875\begin{array}{c} \begin{array}{cccc} \theta_1 & \mathrm{w}_2 & \theta_2 & \theta_3 \end{array} \\ \begin{bmatrix} 6400 & -240 & 3200 & 0 \\ -240 & 24 & 0 & 240 \\ 3200 & 0 & 12800 & 3200 \\ 0 & 240 & 3200 & 6400 \end{bmatrix} \end{array} \begin{Bmatrix} \theta_1 \\ w_2 \\ \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -200 \\ 0 \\ 0 \end{Bmatrix}.$$

By solving the above equations, we get,

$$\begin{Bmatrix} \theta_1 \\ w_2 \\ \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -4.8\times10^{-4}\ \text{rad} \\ -0.0128\ \text{mm} \\ 0 \\ 4.8\times10^{-4}\ \text{rad} \end{Bmatrix}.$$

At $\theta_2 = 0$, max deflection between supports is 0.0128 mm.
Deflection at ends (overhang) = $4.8\times10^{-4}\times10 = 4.8\times10^{-3}$ mm.

(III) Software results.

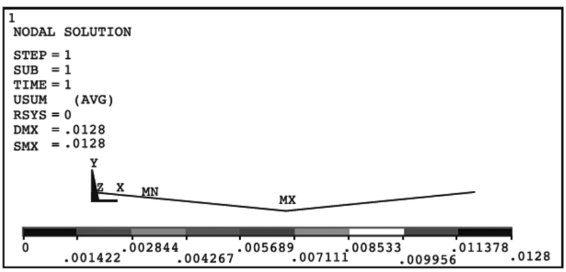


FIGURE 5.5(c) Deflection pattern for a simply supported beam (refer to Appendix D for color figures).

Deflection values at nodes (in mm)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UX	USUM
1	0.0000	0.48000E−02	0.0000	0.48000E−02
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	−0.12800E−01	0.0000	0.12800E−01
4	0.0000	0.0000	0.0000	0.0000
5	0.0000	0.48000E−02	0.0000	0.48000E−02

Maximum absolute values

NODE	0	3	0	3
VALUE	0.0000	−0.12800E−01	0.0000	0.12800E−01

Slope values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	−0.48000E−03
2	−0.48000E−03
3	0.0000
4	0.48000E−03
5	0.48000E−03

ANSWERS FOR EXAMPLE 5.4

Parameter	Analytical method	FEM-Hand calculations	Software results
Deflection at applied load	−0.0128 mm	−0.0128 mm	−0.0128 mm
Deflection at ends (overhang)	4.802×10^{-3} mm	4.8×10^{-3} mm	4.8×10^{-3} mm
Slope at hinged support	-4.802×10^{-4} rad	-4.8×10^{-4} rad	-4.8×10^{-4} rad
Slope at roller support	4.802×10^{-4} rad	4.8×10^{-4} rad	4.8×10^{-4} rad

Procedure for solving the problems using ANSYS ® 11.0 academic teaching software.

FOR EXAMPLE 5.2

PREPROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Beam> 2D elastic 3 > OK > Close**

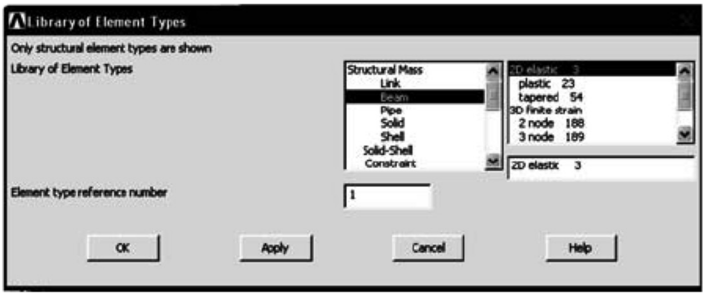


FIGURE 5.6 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

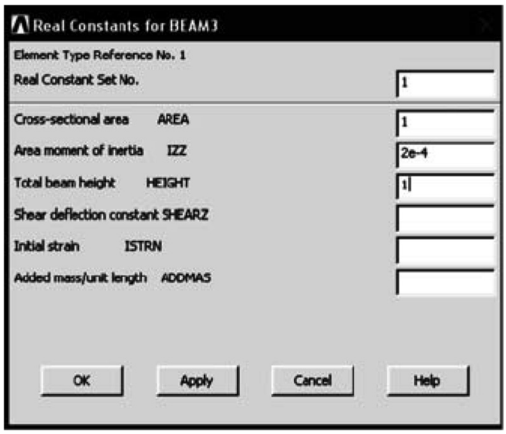


FIGURE 5.7 Enter the area, moment of inertia, and height of beam.

Cross-sectional area AREA > **Enter 1**
Area moment of inertia IZZ > **Enter 2e-4**
Total beam height HEIGHT > **Enter 1 > OK > Close**
Enter the material properties.

3. **Main Menu > Preprocessor > MATERIAL Props > Material Models**
Material Model Number 1, Click **Structural > Linear > Elastic > Isotropic** Enter **EX = 210E9 and PRXY = 0.3 > OK**
(**Close** the define material model behavior window.)
Create the nodes and elements as shown in the table below and Figure 5.8.

4. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply** Enter the coordinates of node 2 > **Apply** Enter the coordinates of node 3 > **OK**.

Node locations		
Node number	**X-coordinate**	**Y-coordinate**
1	0	0
2	4.5	0
3	9	0

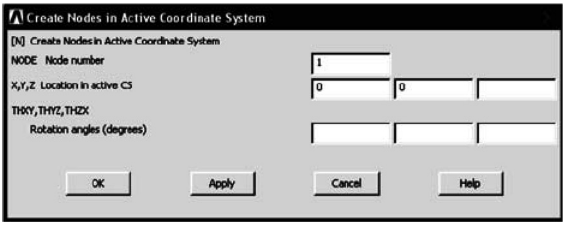


FIGURE 5.8 Enter the node coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **Apply** Pick the 2nd and 3rd node > **OK**

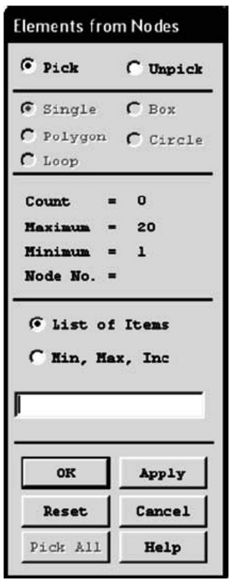


FIGURE 5.9 Pick the nodes to create elements.

Apply the displacement boundary conditions and loads.

6. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st node and 3rd node > **Apply > Select UX and UY and Enter displacement value = 0 > OK**

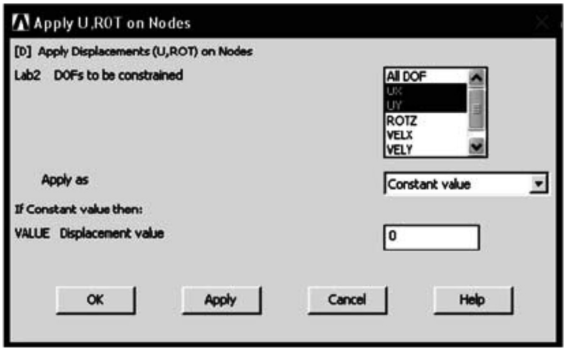


FIGURE 5.10 Apply the displacement constraint.

7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Beams** Pick the 1st element > **OK > Enter Pressure values at node I= 5000 > OK**

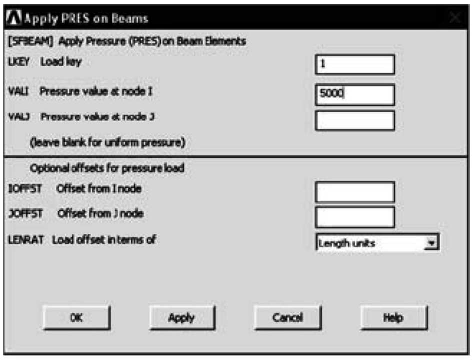


FIGURE 5.11 Applying loads on element 1.

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Beams** Pick the 2nd element > **OK > Enter Pressure value at node I=5000 > OK**

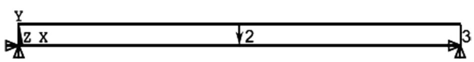


FIGURE 5.12 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First, to be safe, save the model.

Solution

The interactive solution proceeds.

9. **Main Menu > Solution > Solve > Current LS > OK**

   The /**STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window and if all is OK, select **File > Close**.

   In the **Solve Current Load Step** window, select **OK**, and when the solution is complete, **close** the '**Solution is Done!'** window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK**

This result is shown in Figure 5.3(*f*).

11. **Main Menu > General Postproc > List Results > Nodal Solution > Select Rolation vector sum > OK**
12. **Main Menu > General Postproc > List Results > Reaction Solu > OK**

To find the **bending moment diagram,** the following procedure is followed.

13. **Main Menu > General Postproc > Element Table > Define Table > Add** as shown in Figure 5.13.

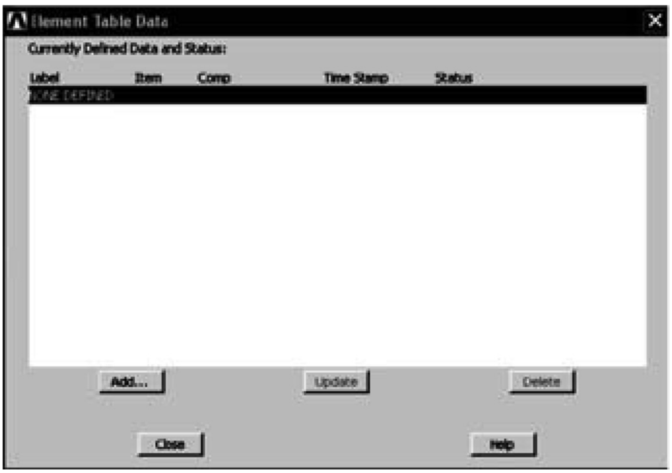


FIGURE 5.13 Define the element table.

Select **By sequence num and SMISC** and type **6 after SMISC** as shown in Figure 5.14. **>APPLY**

FIGURE 5.14 Selecting options in element table.

Then again select **By sequence num and SMISC** and type **12 after SMISC > OK**

14. Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res > Select **SMIS 6 and SMIS 12 in the rows of LabI and LabJ** respectively as shown in Figure 5.15 > **OK**

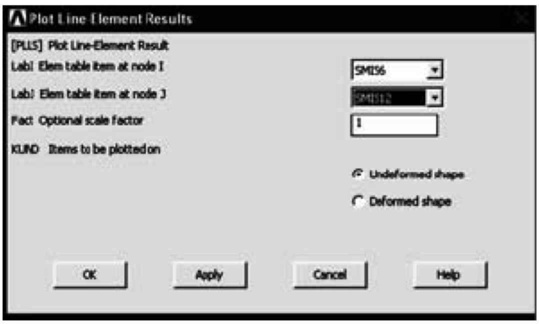


FIGURE 5.15 Selecting options for finding out bending moment.

This result is shown in Figure 5.3(g).

To find the **shear force diagram** the following procedure is followed.

15. Main Menu > General Postproc > Element Table > Define Table > Add

Select **By sequence num and SMISC** and type **2 after SMISC > APPLY**

Then again select **By sequence num and SMISC** and type **8 after SMISC > OK**

16. **Main Menu > General Postproc > Plot Results > Contour Plot > Lone Elem Res >** Select **SMIS 2 and SMIS 8 > OK**

    This result is shown in Figure 5.3(h).

    To find the **bending stress,** the following procedure is followed.

17. **Main Menu > General Postproc > Element Table > Define Table > Add**

    Select **By sequence num** and **LS and** type **3 after LS > APPLY**

    Then again select **By sequence num and LS** and type **6 after LS > OK**

18. **Main Menu > General Postpro c> Plot Results > Contour Plot > Line Elem Res >** Select **LS 3 and LS 6 > OK**

    This result is shown is Figure 5.3(i).

5.3 CANTILEVER BEAMS

EXAMPLE 5.5

Beam subjected to concentrated load. For the beam shown in Figure 5.16, determine the deflections and reactions. Let E = 210 GPa and $I = 2 \times 10^{-4}\ \text{m}^4$. Take 2 elements.

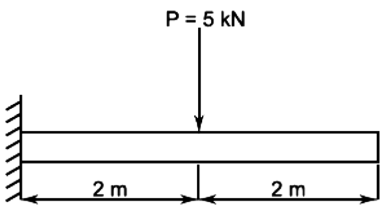


FIGURE 5.16 Beam subjected to concentrated load for Example 5.5.

Solution

(I) Analytical method [Refer to Figure 5.16(a)].

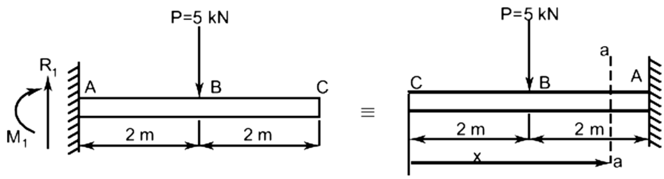


FIGURE 5.16(a) Analytical method for Example 5.5.

The solution is obtained by Macaulay's method. The number within the brackets < > is to be neglected whenever it is less than zero.

At section a-a

$$EIy'' = -P < x-2 >$$

$$EIy' = \frac{-P < x-2 >^2}{2} + C_1$$

$$EIy = \frac{-P < x-2 >^3}{6} + C_1 x + C_2$$

$$\text{At } x = 4 \text{ , } y' = 0 \quad \Rightarrow C_1 = 2P$$

$$\text{At } x = 4 \text{ , } y = 0 \quad \Rightarrow C_2 = \frac{-20P}{3}$$

$$y' = \frac{1}{EI}\left(\frac{-P < x-2 >^2}{2} + 2P\right)$$

$$y = \frac{1}{EI}\left(\frac{-P < x-2 >^3}{6} + 2Px - \frac{20P}{3}\right)$$

$$y'_B = y'_{x=2} = \frac{1}{EI}(0+2P) = \frac{2\times5000}{210\times10^9\times2\times10^{-4}} = 2.381\times10^{-4} \text{ rad}$$

$$y'_C = y'_{x=0} = \frac{1}{EI}(0+2P) = \frac{2\times5000}{210\times10^9\times2\times10^{-4}} = 2.381\times10^{-4} \text{ rad.}$$

Similarly,

$$y_B = y_{x=2} = \frac{1}{EI}\left(4P - \frac{20P}{3}\right) = \frac{2\times5000}{210\times10^9\times2\times10^{-4}}\left(4\times5000 - \frac{20\times5000}{3}\right)$$

$$y_B = y_{x=2} = -3.1746\times10^{-4} \text{ m}$$

$$y_C = y_{x=0} = \frac{1}{EI}\left(-\frac{20P}{3}\right) = \frac{2\times5000}{210\times10^9\times2\times10^{-4}}\left(-\frac{20\times5000}{3}\right)$$

$$y_C = y_{x=0} = -7.9365\times10^{-4} \text{ m}$$

$$\sum F_y = 0 \Rightarrow R_1 = 5\ kN$$

$$\sum M = 0 \Rightarrow M_1 = 10\ kN - m.$$

(II) FEM by hand calculations.

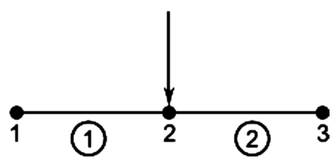


FIGURE 5.16(b) Finite element model.

Element stiffness matrix for element 1 is,

$$[k_1]=\frac{EI}{L^3}\begin{bmatrix}12 & 6L & -12 & 6L\\ 6L & 4L^2 & -6L & 2L^2\\ -12 & -6L & 12 & -6L\\ 6L & 2L^2 & -6L & 4L^2\end{bmatrix}=\frac{210\times10^9\times2\times10^{-4}}{(2)^3}\begin{bmatrix}12 & 6(2) & -12 & 6(2)\\ 6(2) & 4(2)^2 & -6(2) & 2(2)^2\\ -12 & -6(2) & 12 & -6(2)\\ 6(2) & 2(2)^2 & -6(2) & 4(2)^2\end{bmatrix}$$

$$[k_1]=5.25\times10^6\begin{array}{c}\begin{array}{cccc}w_1 & \theta_1 & w_2 & \theta_2\end{array}\\ \begin{bmatrix}12 & 12 & -12 & 12\\ 12 & 16 & -12 & 8\\ -12 & -12 & 12 & -12\\ 12 & 8 & -12 & 16\end{bmatrix}\end{array}\begin{array}{c}\\ w_1\\ \theta_1\\ w_2\\ \theta_2\end{array}.$$

Element stiffness matrix for element 2 is,

$$[k_2]=\frac{EI}{L^3}\begin{bmatrix}12 & 6L & -12 & 6L\\ 6L & 4L^2 & -6L & 2L^2\\ -12 & -6L & 12 & -6L\\ 6L & 2L^2 & -6L & 4L^2\end{bmatrix}=\frac{210\times10^9\times2\times10^{-4}}{(2)^3}\begin{bmatrix}12 & 6(2) & -12 & 6(2)\\ 6(2) & 4(2)^2 & -6(2) & 2(2)^2\\ -12 & -6(2) & 12 & -6(2)\\ 6(2) & 2(2)^2 & -6(2) & 4(2)^2\end{bmatrix}$$

$$[k_2]=5.25\times10^6\begin{array}{c}\begin{array}{cccc}w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 12 & -12 & 12\\ 12 & 16 & -12 & 8\\ -12 & -12 & 12 & -12\\ 12 & 8 & -12 & 16\end{bmatrix}\end{array}\begin{array}{c}\\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}.$$

Global stiffness matrix is,

$$[K]=5.25\times10^6\begin{array}{c}\begin{array}{cccccc}w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 12 & -12 & 12 & 0 & 0\\ 12 & 16 & -12 & 8 & 0 & 0\\ -12 & -12 & 12+12 & -12+12 & -12 & 12\\ 12 & 8 & -12+12 & 16+16 & -12 & 8\\ 0 & 0 & -12 & -12 & 12 & -12\\ 0 & 0 & 12 & 8 & -12 & 16\end{bmatrix}\end{array}\begin{array}{c}\\ w_1\\ \theta_1\\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}.$$

The global equations are,

$$[K]\{r\}=\{R\} \tag{5.8}$$

$$5.25\times10^{6}\begin{array}{cc} \begin{matrix} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{matrix} & \\ \begin{bmatrix} 12 & 12 & -12 & 12 & 0 & 0 \\ 12 & 16 & -12 & 8 & 0 & 0 \\ -12 & -12 & 24 & 0 & -12 & 12 \\ 12 & 8 & 0 & 32 & -12 & 8 \\ 0 & 0 & -12 & -12 & 12 & -12 \\ 0 & 0 & 12 & 8 & -12 & 16 \end{bmatrix} & \begin{matrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{matrix} \end{array} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} 0+R_1 \\ 0+M_1 \\ -5\times10^{3} \\ 0 \\ 0 \\ 0 \end{Bmatrix}.$$

By using the elimination method for applying boundary conditions,

$$w_1=\theta_1=0.$$

The above matrix reduces to,

$$5.25\times10^{6}\begin{array}{c} \begin{matrix} w_2 & \theta_2 & \mathrm{w}_3 & \theta_3 \end{matrix} \\ \begin{bmatrix} 24 & 0 & -12 & 12 \\ 0 & 32 & -12 & 8 \\ -12 & -12 & 12 & -12 \\ 12 & 8 & -12 & 16 \end{bmatrix} \end{array} \begin{Bmatrix} w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -5\times10^{3} \\ 0 \\ 0 \\ 0 \end{Bmatrix}.$$

By solving the above matrix and equations, we get,

Deflections and slopes as

$$\begin{aligned} w_2 &= -0.3175\times10^{-3}\ \text{m} \\ \theta_2 &= -0.2381\times10^{-3}\ \text{rad} \\ w_3 &= -0.7937\times10^{-3}\ \text{m} \\ \theta_3 &= -0.2381\times10^{-3}\ \text{rad} \end{aligned}.$$

Reaction calculation

$$5.25\times10^{6}\left(12w_1+12\theta_1-12w_2+12\theta_2\right)=R_1$$

$$5.25\times10^{6}\left(12\times0+12\times0-12\left(-0.3175\times10^{-3}\right)+12\left(-0.2381\times10^{-3}\right)\right)=R_1.$$

$$\therefore\quad R_1=5002.2\ \text{N}\approx5\ \text{kN}$$

$$5.25\times10^{6}\left(-12w_2+8\theta_2\right)=M_1$$

$$5.25\times10^{6}\left(-12\left(-0.3175\times10^{-3}\right)+8\left(-0.238\times10^{-3}\right)\right)=M_1 \cdot$$

$$\therefore \quad \mathrm{M}_1=10002.3 \text{ N-m}\approx 10 \text{ kN-m}$$

(III) Software results.

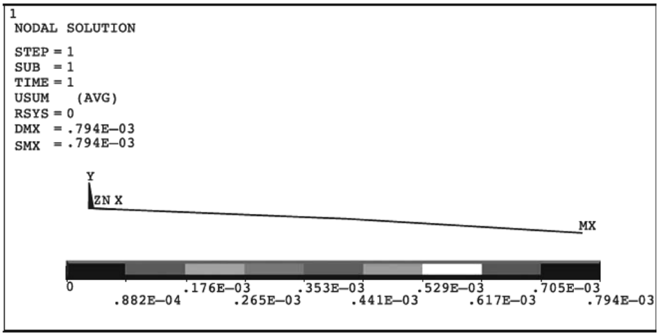


FIGURE 5.16(c) Deflection pattern for a cantilever beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	−0.31746E−03	0.0000	0.31746E−03
3	0.0000	−0.79365E−03	0.0000	0.79365E−03

Maximum absolute values

NODE	0	3	0	3
VALUE	0.0000	−0.79365E−03	0.0000	0.79365E−03

Rotational deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	0.0000
2	−0.23810E−03
3	−0.23810E−03

Reaction values

The following X, Y, and Z solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	5000.0	10000.

ANSWERS FOR EXAMPLE 5.5

Parameter	Analytical method	FEM-Hand calculations	Software results
Deflection at node			
2	-3.1746×10^{-4} m	-0.3175×10^{-3} m	-0.31746×10^{-3} m
3	-7.9365×10^{-4} m	-0.7937×10^{-3} m	-0.79365×10^{-3} m
Rotational deflection at node			
2	-2.381×10^{-4} rad	-0.2381×10^{-3} rad	-0.2381×10^{-3} rad
3	-2.381×10^{-4} rad	-0.2381×10^{-3} rad	-0.2381×10^{-3} rad
Reaction force at node 1	5 kN	5 kN	5 kN
Reaction moment at node 1	10 kN-m	10 kN-m	10 kN-m

EXAMPLE 5.6

Propped cantilever beam with distributed load. Find nodal displacements and support reactions for the beam shown in Figure 5.17. Let E = 70 GPa and $I = 6 \times 10^{-4}\ \text{m}^4$.

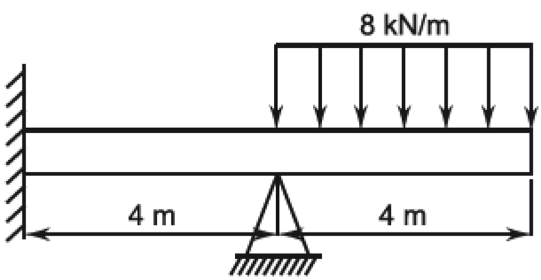


FIGURE 5.17 Propped cantilever beam with distributed load for Example 5.6.

Solution

(I) Analytical method [Refer to Figure 5.17(a)].

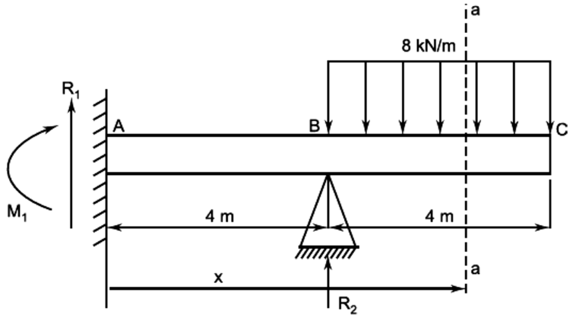


FIGURE 5.17(a) Analytical method for Example 5.6.

The solution is obtained by Macaulay's method. The number within the brackets < > is to be neglected whenever it is less than zero.

$$\sum F_y = 0 \Rightarrow R_1 = -R_2 + 8\times10^3\times4 = 32000 - R_2$$
$$\sum M = 0 \Rightarrow M_1 = 4R_2 - \left(8\times10^3\times4\right)\times6 = \left(4R_2 - 192000\right) \text{ N-m.}$$

At section a-a

$$M_x = M_1 + R_1 x + R_2 < x-4 > -\frac{8\times10^3}{2} < x-4 >^2$$

$$EIy'' = \left(4R_2 - 192000\right) + \left(32000 - R_2\right)x + R_2 < x-4 > -4\times10^3 < x-4 >^2$$

$$\begin{aligned} EIy' &= 4R_2 x - 192000x + \left(32000\times\frac{x^2}{2}\right) - \left(R_2\times\frac{x^2}{2}\right) \\ &+ \frac{R_2}{2} < x-4 >^2 - \frac{4\times10^3 < x-4 >^3}{3} + C_1 \end{aligned} \tag{5.9}$$

$$\begin{aligned} EIy &= \left(4R_2\times\frac{x^2}{2}\right) - \left(192000\times\frac{x^2}{2}\right) + \left(32000\times\frac{x^3}{6}\right) - \left(R_2\times\frac{x^3}{6}\right) \\ &+ \frac{R_2}{6} < x-4 >^3 - \frac{4\times10^3 < x-4 >^4}{12} + C_1 x + C_2 \quad . \end{aligned} \tag{5.10}$$

Boundary conditions are,

$$\text{At } x = 0, y = 0 \Rightarrow C_2 = 0$$

$$\text{At } x = 0, y' = 0 \Rightarrow C_1 = 0$$

$$\text{At } x = 4, y = 0 \Rightarrow R_2 = 56008 \text{ N}$$

$$\therefore \quad R_1 = 32000 - R_2 = -24008 \text{ N}$$

$$M_1 = 4R_2 - 192000 = 32032 \text{ N-m (Clockwise), (negative).}$$

Substituting in equations (5.9) and (5.10)

$$EIy' = 4\times 56008x - 192000x + \left(32000\times\frac{x^2}{2}\right) - \left(56008\times\frac{x^2}{2}\right)$$
$$+\frac{56008}{2} < x-4 >^2 - \frac{4\times 10^3 < x-4 >^3}{3}$$

$$y'\Big|_{x=4} = -0.00152 \text{ rad}$$
$$y'\Big|_{x=8} = -0.00355 \text{ rad}$$

and

$$EIy = \left(4\times 56008\times\frac{x^2}{2}\right) - \left(192000\times\frac{x^2}{2}\right) + \left(32000\times\frac{x^3}{6}\right) - \left(56008\times\frac{x^3}{6}\right)$$
$$+\frac{56008}{6} < x-4 >^3 - \frac{4\times 10^3 < x-4 >^4}{12}$$

$$y\Big|_{x=8} = y_C = y_3 = -0.0122 \text{ m}$$

$$y_A = y_1 = 0 \text{ and } y_B = y_2 = 0 \text{ (Given boundary conditions).}$$

(II) FEM by hand calculations.

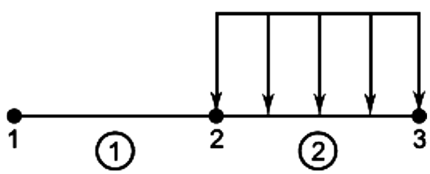


FIGURE 5.17(b) Finite element model for Example 5.6.

$$E = 70\times 10^3 \text{ N/mm}^2 \quad \text{and} \quad I = 6\times 10^8 \text{ mm}^4.$$

Stiffness matrix for element 1 is,

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

$$[k_1]=\frac{70\times10^3\times6\times10^8}{(4000)^3}\begin{bmatrix}12 & 6(4000) & -12 & 6(4000)\\ 6(4000) & 4(4000)^2 & -6(4000) & 2(4000)^2\\ -12 & -6(4000) & 12 & -6(4000)\\ 6(4000) & 2(4000)^2 & -6(4000) & 4(4000)^2\end{bmatrix}$$

$$[k_1]=656.25\begin{array}{c}\begin{array}{cccc} w_1 & \theta_1 & w_2 & \theta_2\end{array}\\ \begin{bmatrix}12 & 24000 & -12 & 24000\\ 24000 & 64\times10^6 & -24000 & 32\times10^6\\ -12 & -24000 & 12 & -24000\\ 24000 & 32\times10^6 & -24000 & 64\times10^6\end{bmatrix}\end{array}\begin{array}{c} \\ w_1\\ \theta_1\\ w_2\\ \theta_2\end{array}.$$

Due to symmetry,

$$[k_1]=[k_2]$$

$$[k_2]=656.25\begin{array}{c}\begin{array}{cccc} w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 24000 & -12 & 24000\\ 24000 & 64\times10^6 & -24000 & 32\times10^6\\ -12 & -24000 & 12 & -24000\\ 24000 & 32\times10^6 & -24000 & 64\times10^6\end{bmatrix}\end{array}\begin{array}{c} \\ w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}.$$

Nodal force calculation

For element 2,

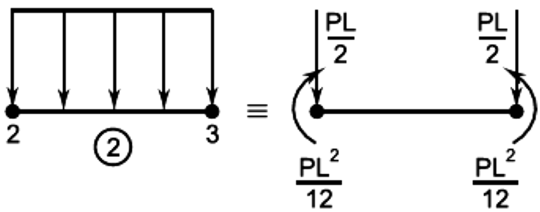


FIGURE 5.17(c) Nodal force calculation for element 2 in Example 5.6.

$$P = 8 \text{ N/mm}$$
$$\text{L} = 4000 \text{ mm}$$
$$\frac{\text{PL}}{2} = \frac{8 \times 4000}{2} = 16000 \text{ N}$$
$$\frac{\text{PL}^2}{12} = \frac{8 \times (4000)^2}{12} = 10.667 \times 10^6 \text{ N-mm}.$$

The nodal forces and moments for element 2 is,

$$[F_2] = \begin{Bmatrix} -\dfrac{PL}{2} \\ -\dfrac{PL^2}{12} \\ -\dfrac{PL}{2} \\ -\dfrac{PL^2}{12} \end{Bmatrix} = \begin{Bmatrix} -1600 \\ -10.667 \times 10^6 \\ -16000 \\ -10.667 \times 10^6 \end{Bmatrix} \begin{matrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{matrix}.$$

The global equations are,

$$[K]\{r\} = \{R\} \tag{5.11}$$

$$\begin{array}{cc}
 & \begin{array}{cccccc} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\
656.25 & \left[\begin{array}{cccccc}
12 & 24000 & -12 & 24000 & 0 & 0 \\
24000 & 64 \times 10^6 & -24000 & 32 \times 10^6 & 0 & 0 \\
-12 & -24000 & 12 + 12 & -24000 + 24000 & -12 & 24000 \\
24000 & 32 \times 10^6 & -24000 + 24000 & 64 \times 10^6 + 64 \times 10^6 & -24000 & 32 \times 10^6 \\
0 & 0 & -12 & -24000 & 12 & -24000 \\
0 & 0 & 24000 & 32 \times 10^6 & -24000 & 64 \times 10^6
\end{array}\right] \begin{array}{c} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array}
\end{array}$$

$$\times \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} 0 + R_1 \\ 0 + M_1 \\ 16000 + R_2 \\ -10.667 \times 10^6 \\ -16000 \\ 10.667 \times 10^6 \end{Bmatrix}$$

By using the elimination method for applying boundary conditions,

$$w_1 = \theta_1 = w_2 = 0.$$

The above matrix reduces to

$$656.25\begin{bmatrix} 128\times10^6 & -24000 & 32\times10^6 \\ -24000 & 12 & -24000 \\ 32\times10^6 & -24000 & 64\times10^6 \end{bmatrix}\begin{Bmatrix} \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -10.667\times10^6 \\ -16000 \\ 10.667\times10^6 \end{Bmatrix}.$$

Solving the above matrix and equations, we get,

$$w_2 = -12.19 \text{ mm} = -0.01219 \text{ m}$$

$$\theta_3 = -0.00355 \text{ rad}$$

$$\theta_2 = -0.00152 \text{ rad.}$$

Reaction calculation

$$656.25(12w_1 + 24000\times\theta_1 - 12w_2 + 24000\times\theta_2) = R_1$$

$$656.25(12\times0 + 24000\times0 - 12\times0 + 24000\times(-0.00152)) = R_1$$

$$\therefore \quad R_1 = -23.94 \text{ kN} \approx -24 \text{ kN}$$

$$656.25(24000w_1 + 64\times10^6\times\theta_1 - 24000w_2 + 32\times10^6\times\theta_2) = M_1$$

$$656.25(24000\times0 + 64\times10^6\times0 - 24000\times0 + 32\times10^6\times(-0.00152)) = M_1$$

$$\therefore \quad \text{M}_1 = -31.92 \text{ kN-m} \approx -32 \text{ kN-m}$$

$$656.25(-12w_1 - 24000\times\theta_1 + 24w_2 + 0\times\theta_2 - 12w_3 + 24000\times\theta_3) = R_2 - 16000$$

$$656.25(-12\times0 - 24000\times0 + 24\times0 + 0\times\theta_2 - 12(-12.19) + 24000\times(-0.00355))$$

$$= R_2 - 16000.$$

$$\therefore \quad R_2 = 56.08 \text{ kN} \approx 56 \text{ kN}$$

(III) Software results.

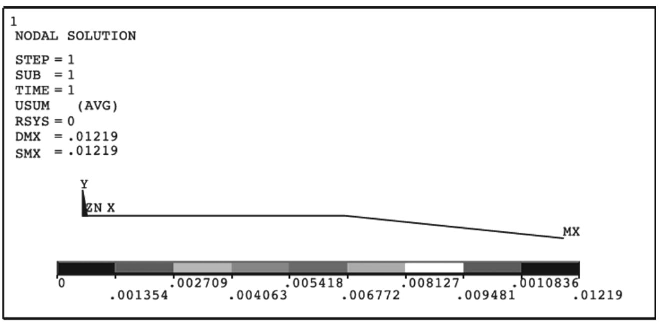


FIGURE 5.17(d) Deflection pattern for a cantilever beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.31746E−03
3	0.0000	−0.12190E−01	0.0000	0.12190E−01

Rotational deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	0.0000
2	−0.15238E−02
3	−0.35556E−02

Reaction values

The following X, Y, Z solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	−24000	−32000
2	0.0000	56000	

ANSWER FOR EXAMPLE 5.6

Parameter	Analytical method	FEM-Hand calculations	Software results
Deflection at node 3	−0.0122 m	−0.01219 m	−0.01219 m
Rotational deflection at node			
2	−0.00152 rad	−0.00152 rad	−0.001524 rad
3	−0.00355 rad	−0.00355 rad	−0.00355 rad
Reaction force at			
1	−24 kN	−24 kN	−24 kN
2	56 kN	56 kN	56 kN
Reaction moment at node 1	−32 kN-m	−32 kN-m	−32 kN-m

EXAMPLE 5.7

Propped cantilever beam with varying load. For the beam shown in Figure 5.18, determine the nodal displacements, slopes, reactions, maximum bending moment, shear force, and maximum bending stress. Take E = 200GPa.

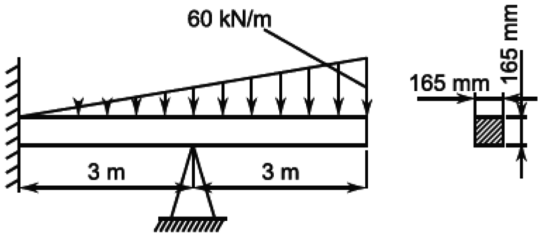


FIGURE 5.18 Propped cantilever beam with varying load for Example 5.7.

Solution

(I) FEM-hand calculations.

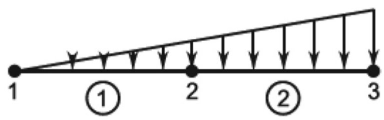


FIGURE 5.18(a) Finite element model for Example 5.7.

$$I = \frac{bh^3}{12} = \frac{165 \times (165)^3}{12} = 61766718.75 \text{ mm}^4 = 6.18 \times 10^{-5} \text{ m}^4.$$

Stiffness matrices for element 1 and 2 are,

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

$$[k_1] = \frac{200 \times 10^9 \times 6.18 \times 10^{-5}}{(3)^3} \begin{bmatrix} 12 & 6(3) & -12 & 6(3) \\ 6(3) & 4(3)^2 & -6(3) & 2(3)^2 \\ -12 & -6(3) & 12 & -6(3) \\ 6(3) & 2(3)^2 & -6(3) & 4(3)^2 \end{bmatrix}$$

$$[k_1] = \frac{200\times10^9\times6.18\times10^{-5}}{(3)^3}\begin{bmatrix} 12 & 18 & -12 & 18 \\ 18 & 36 & -18 & 18 \\ -12 & -18 & 12 & -18 \\ 18 & 18 & -18 & 36 \end{bmatrix}$$

$$[k_1] = 457.78\times10^3 \begin{array}{c} \begin{array}{cccc} w_1 & \theta_1 & w_2 & \theta_2 \end{array} \\ \begin{bmatrix} 12 & 18 & -12 & 18 \\ 18 & 36 & -18 & 18 \\ -12 & -18 & 12 & -18 \\ 18 & 18 & -18 & 36 \end{bmatrix} \end{array} \begin{array}{c} \\ w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{array}.$$

Due to symmetry,

$$[k_1] = [k_2]$$

$$[k_1] = 457.78\times10^3 \begin{array}{c} \begin{array}{cccc} w_2 & \theta_2 & w_3 & \theta_3 \end{array} \\ \begin{bmatrix} 12 & 18 & -12 & 18 \\ 18 & 36 & -18 & 18 \\ -12 & -18 & 12 & -18 \\ 18 & 18 & -18 & 36 \end{bmatrix} \end{array} \begin{array}{c} \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{array}.$$

Nodal force calculation

For element 1,

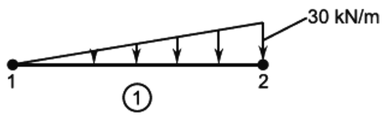


FIGURE 5.18(b) Nodal force calculation for element 1 for Example 5.7.

$$P_1 = 0 \text{ and } P_2 = 30 \text{ kN/m}.$$
$$\text{L} = 3 \text{ m}$$

The nodal forces and moments for element 1 is,

$$[F_1] = \frac{L}{20}\begin{Bmatrix} -(7P_1+3P_2) \\ -\frac{L}{3}(3P_1+2P_2) \\ -(3P_1+7P_2) \\ \frac{L}{3}(2P_1+3P_2) \end{Bmatrix} = \frac{3}{20}\begin{Bmatrix} -(7\times0+3\times30\times10^3) \\ -\frac{3}{3}(3\times0+2\times30\times10^3) \\ -(3\times0+7\times30\times10^3) \\ \frac{3}{3}(2\times0+3\times30\times10^3) \end{Bmatrix} = \begin{Bmatrix} -13.5\times10^3 \\ -9\times10^3 \\ -31.5\times10^3 \\ 13.5\times10^3 \end{Bmatrix} \begin{array}{c} f_1 \\ m_1 \\ f_2 \\ m_2 \end{array}.$$

For element 2,

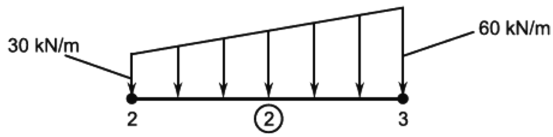


FIGURE 5.18(c) Nodal force calculation for element 2 for Example 5.7.

$$P_1 = 30\times10^3 \text{ N/m and } P_2 = 60\times10^3 \text{ N/m}.$$
$$\text{L} = 3 \text{ m}$$

The nodal forces and moments for element 2 is,

$$[F_2] = \frac{L}{20}\begin{Bmatrix} -(7P_1+3P_2) \\ -\frac{L}{3}(3P_1+2P_2) \\ -(3P_1+7P_2) \\ -\frac{L}{3}(2P_1+3P_2) \end{Bmatrix} = \frac{3}{20}\begin{Bmatrix} -(7\times30\times10^3+3\times60\times10^3) \\ -\frac{3}{3}(3\times30\times10^3+2\times60\times10^3) \\ -(3\times30\times10^3+7\times60\times10^3) \\ \frac{3}{3}(2\times30\times10^3+3\times60\times10^3) \end{Bmatrix} = \begin{Bmatrix} -58.5\times10^3 \\ -31.5\times10^3 \\ -76.5\times10^3 \\ 36\times10^3 \end{Bmatrix}\begin{matrix} f_2 \\ m_2 \\ f_3 \\ m_3 \end{matrix}.$$

The combined nodal forces and moments matrix is,

$$[F] = \begin{Bmatrix} -13.5\times10^3 \\ -9\times10^3 \\ (-31.5-58.5)\times10^3 \\ (13.5-31.5)\times10^3 \\ -76.5\times10^3 \\ 36\times10^3 \end{Bmatrix} = \begin{Bmatrix} -13.5\times10^3 \\ -9\times10^3 \\ -90\times10^3 \\ -18\times10^3 \\ -76.5\times10^3 \\ 36\times10^3 \end{Bmatrix}\begin{matrix} f_1 \\ m_1 \\ f_2 \\ m_2 \\ f_3 \\ m_3 \end{matrix}.$$

The global equations are,

$$457.78\times\cancel{10^3}\begin{array}{c} \begin{matrix} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{matrix} \\ \begin{bmatrix} 12 & 18 & -12 & 18 & 0 & 0 \\ 18 & 36 & -18 & 18 & 0 & 0 \\ -12 & -18 & 12+12 & -18+18 & -12 & 18 \\ 18 & 18 & -18+18 & 36+36 & -18 & 18 \\ 0 & 0 & -12 & -18 & 12 & -18 \\ 0 & 0 & 18 & 18 & -18 & 36 \end{bmatrix} \end{array}\begin{matrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{matrix}\begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \cancel{10^3}\times\begin{Bmatrix} -13.5+R_1 \\ -9+M_1 \\ -90+R_2 \\ -18 \\ -76.5 \\ 36 \end{Bmatrix}.$$

By using the elimination method for applying boundary conditions, $w_1 = \theta_1 = w_2 = 0$. The above matrix reduces to,

$$457.78\begin{bmatrix} 72 & -18 & 18 \\ -18 & 12 & -18 \\ 18 & -18 & 36 \end{bmatrix}\begin{Bmatrix} \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -18 \\ -76.5 \\ 36 \end{Bmatrix}.$$

By solving the above matrix and equations, we get

$$\theta_2 = -0.0128 \text{ rad}, w_3 = -0.0811 \text{ m, and } \theta_3 = -0.0319 \text{ rad}.$$

Reaction calculation

$$457.78\times10^3\left(12w_1 + 18\theta_1 - 12w_2 + 18\theta_2\right) = -13.5\times10^3 + R_1$$
$$457.78\times10^3\left(12\times0 + 18\times0 - 12\times0 + 18\times(-0.0128)\right) = -13.5\times10^3 + R_1$$
$$\therefore \quad R_1 = -91972.5 \text{ N}$$

$$457.78\times10^3\left(18w_1 + 36\theta_1 - 18w_2 + 18\theta_2\right) = -9\times10^3 + M_1$$
$$457.78\times10^3\left(18\times0 + 36\times0 - 18\times0 + 18\times(-0.0128)\right) = -9\times10^3 + M_1$$
$$\therefore \quad M_1 = -96472.5 \text{ N-m}$$

$$457.78\times10^3\left(-12w_1 - 18\theta_1 + 24w_2 + 0\times\theta_2 - 12w_3 + 18\theta_3\right) = -90\times10^3 + R_2$$
$$457.78\times10^3\left(-12\times0 - 18\times0 + 24\times0 + 0\times\theta_2 - 12\times(-0.0811) + 18\times(-0.0319)\right)$$
$$= -90\times10^3 + R_2.$$
$$\therefore \quad R_2 = 272654.22 \text{ N}$$

(II) Software results.

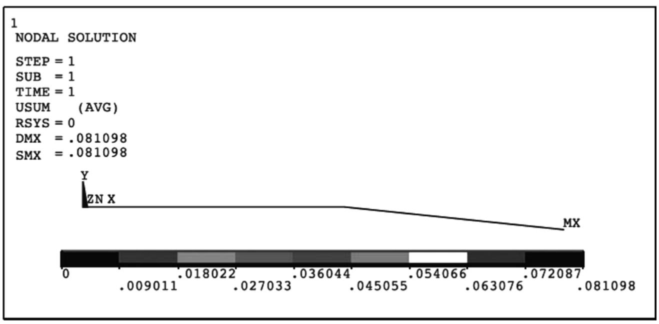


FIGURE 5.18(d) Deflection pattern for a cantilever beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	−0.81098E−01	0.0000	0.810898E−01

Maximum absolute values

NODE	0	3	0	3
VALUE	0.0000	−0081098E−01	0.0000	0.81098E−01

Rotational deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	0.0000
2	−0.12834E−01
3	−0.31948E−01

Reaction values

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	−92250	−96750
2	0.0000	0.27225E + 06	

Total values

VALUE	0.0000	0.18000E + 06	−96750

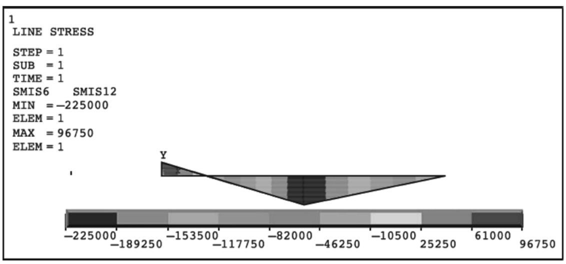


FIGURE 5.18(e) Bending moment diagram for a propped cantilever beam (refer to Appendix D for color figures).

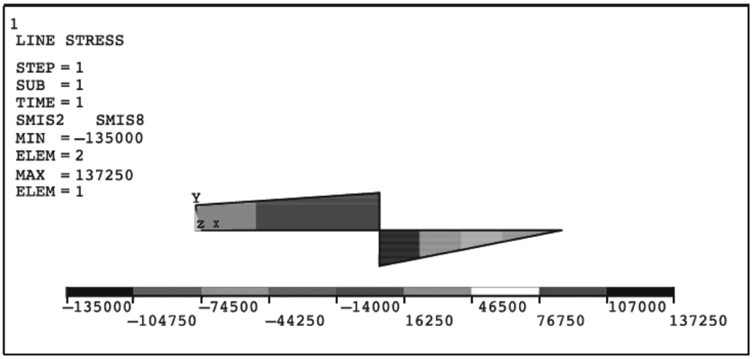


FIGURE 5.18(f) Shear force diagram for a propped cantilever beam (refer to Appendix D for color figures).

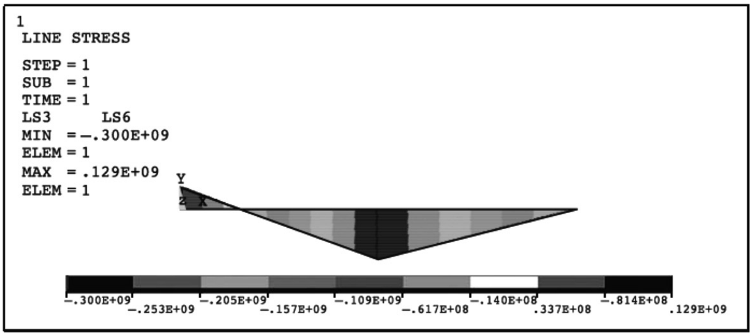


FIGURE 5.18(g) Bending stress diagram for a propped cantilever beam (refer to Appendix D for color figures).

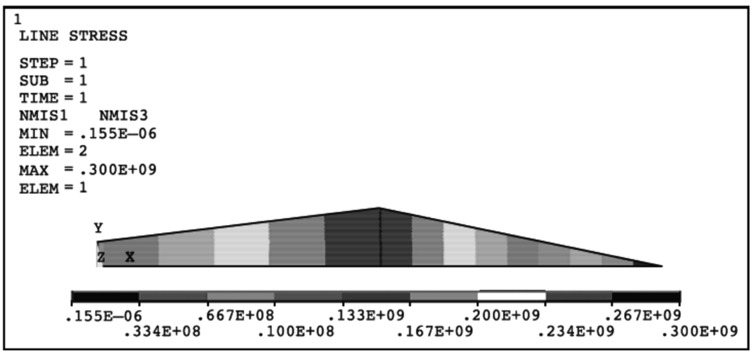


FIGURE 5.18(h) Maximum stress diagram for a propped cantilever beam (refer to Appendix D for color figures).

ANSWERS FOR EXAMPLE 5.7

Parameter	FEM-Hand calculations	Software results
Deflection at node 3	−0.0811 m	−0.081098 m
Rotational deflection at node		
2	−0.0128 rad	−0.012834 rad
3	−0.0319 rad	−0.031948 rad
Reaction force at		
1	–91.97 kN	–92.25 kN
2	272.65 kN	272.25 kN
Reaction moment at node 1	−96.47 kN-m	−96.75 kN-m
Maximum bending moment	…..	96750 N-m
Shear force	…..	137250 N
Maximum bending stress	…..	129 MPa
Maximum stress (bending stress + direct stress)	…..	300 MPa

EXAMPLE 5.8

Propped cantilever beam with stepped loading. Analyze the beam in Figure 5.19 by finite element method and determine the reactions. Also, determine the deflections.

Given $$E = 200 \text{ GPa and } I = 5\times10^{-4} \quad \text{m}^4.$$

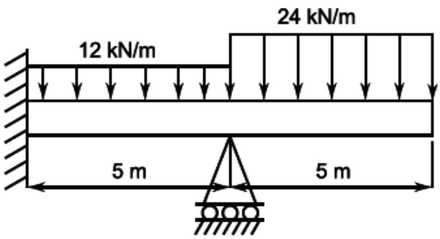


FIGURE 5.19 Propped cantilever beam with stepped loading for Example 5.8.

Solution

(I) FEM by hand calculations.

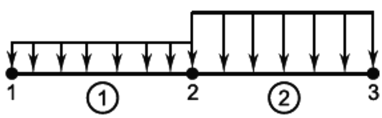


FIGURE 5.19(a) Finite element model for Example 5.8.

Stiffness matrix for element 1 and 2 are,

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

$$[k_1]=\frac{200\times10^9\times5\times10^{-4}}{(5)^3}\begin{bmatrix}12 & 6(5) & -12 & 6(5)\\ 6(5) & 4(5)^2 & -6(5) & 2(5)^2\\ -12 & -6(5) & 12 & -6(5)\\ 6(5) & 2(5)^2 & -6(5) & 4(5)^2\end{bmatrix}$$

$$[k_1]=800\times10^3\begin{array}{c}\begin{array}{cccc}w_1 & \theta_1 & w_2 & \theta_2\end{array}\\ \begin{bmatrix}12 & 30 & -12 & 30\\ 30 & 100 & -30 & 50\\ -12 & -30 & 12 & -30\\ 30 & 50 & -30 & 100\end{bmatrix}\end{array}\begin{array}{c}w_1\\ \theta_1\\ w_2\\ \theta_2\end{array}$$

$$[k_1]=800\times10^3\begin{array}{c}\begin{array}{cccc}w_2 & \theta_2 & w_3 & \theta_3\end{array}\\ \begin{bmatrix}12 & 30 & -12 & 30\\ 30 & 100 & -30 & 50\\ -12 & -30 & 12 & -30\\ 30 & 50 & -30 & 100\end{bmatrix}\end{array}\begin{array}{c}w_2\\ \theta_2\\ w_3\\ \theta_3\end{array}.$$

Nodal force calculation

For element 1,

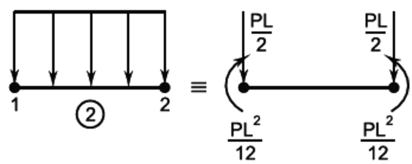


FIGURE 5.19(b) Nodal force calculation for element 1 in Example 5.8

$$P=12\ \text{kN/m}=12\times10^3\ \text{N/m}$$

$$\text{L}=5\ \text{m}$$

$$\frac{\text{PL}}{2}=\frac{12\times10^3\times5}{2}=30\times10^3\ \text{N}$$

$$\frac{\text{PL}^2}{12}=\frac{12\times10^3\times(5)^2}{2}=25\times10^3\ \text{N-m}$$

The nodal forces and moments for element 1 is,

$$[F_1]=\begin{Bmatrix} -\dfrac{PL}{2} \\ -\dfrac{PL^2}{12} \\ -\dfrac{PL}{2} \\ \dfrac{PL^2}{12} \end{Bmatrix}=\begin{Bmatrix} -30\times10^3 \\ -25\times10^3 \\ -30\times10^3 \\ 25\times10^3 \end{Bmatrix}\begin{matrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{matrix}.$$

For element 2,

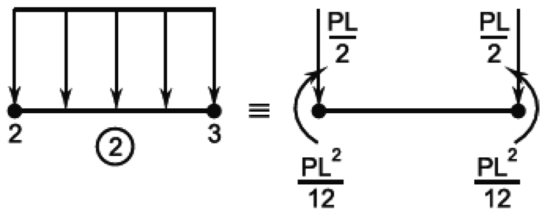


FIGURE 5.19(c) Nodal force calculation for element 2 in Example 5.8.

$$P = 24 \text{ kN/m}=24\times10^3 \text{ N/m}$$
$$\text{L}= 5 \text{ m}$$
$$\frac{\text{PL}}{2}=\frac{24\times10^3\times5}{2}=60\times10^3 \text{ N}$$
$$\frac{\text{PL}^2}{12}=\frac{24\times10^3\times(5)^2}{2}=50\times10^3 \text{ N-m}$$

The nodal forces and moments for element 2 is,

$$[F_2]=\begin{Bmatrix} -\dfrac{PL}{2} \\ -\dfrac{PL^2}{12} \\ -\dfrac{PL}{2} \\ \dfrac{PL^2}{12} \end{Bmatrix}=\begin{Bmatrix} -60\times10^3 \\ -50\times10^3 \\ -60\times10^3 \\ 50\times10^3 \end{Bmatrix}\begin{matrix} f_2 \\ m_2 \\ f_3 \\ m_3 \end{matrix}.$$

The combined nodal forces and moments is,

$$[F]=\begin{Bmatrix} -30\times10^3 \\ -25\times10^3 \\ -30\times10^3-60\times10^3 \\ 25\times10^3-50\times10^3 \\ -60\times10^3 \\ 50\times10^3 \end{Bmatrix}=\begin{Bmatrix} -30\times10^3 \\ -25\times10^3 \\ -90\times10^3 \\ -25\times10^3 \\ -60\times10^3 \\ 50\times10^3 \end{Bmatrix}\begin{matrix} f_1 \\ m_1 \\ f_2 \\ m_2 \\ f_3 \\ m_3 \end{matrix}.$$

The global equations are,

$$[K]\{r\}=\{R\} \tag{5.12}$$

$$800\times10^3\begin{array}{c} \begin{matrix} w_1 & \theta_1 & w_2 & \theta_2 & w_3 & \theta_3 \end{matrix} \\ \begin{bmatrix} 12 & 30 & -12 & 30 & 0 & 0 \\ 30 & 100 & -30 & 50 & 0 & 0 \\ -12 & -30 & 12+12 & -30+30 & -12 & 30 \\ 30 & 50 & -30+30 & 100+100 & -30 & 50 \\ 0 & 0 & -12 & -30 & 12 & -30 \\ 0 & 0 & 30 & 50 & -30 & 100 \end{bmatrix} \end{array}\begin{matrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{matrix}\begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix}=\begin{Bmatrix} -30\times10^3+R_1 \\ -25\times10^3+M_1 \\ -90\times10^3+R_2 \\ -25\times10^3 \\ -60\times10^3 \\ 50\times10^3 \end{Bmatrix}.$$

By using the elimination method for applying boundary conditions,

$$w_1=\theta_1=w_2=0,$$

the above matrix reduces to

$$800\times10^3\begin{bmatrix} 200 & -30 & 50 \\ -30 & 12 & -30 \\ 50 & -30 & 100 \end{bmatrix}\begin{Bmatrix} \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix}=\begin{Bmatrix} -25\times10^3 \\ -60\times10^3 \\ 50\times10^3 \end{Bmatrix}.$$

By solving the above matrix and equations, we get

Reaction Calculation

$$800\times10^3\left(12w_1+30\theta_1-12w_2+30\theta_2\right)=-30\times10^3+R_1$$

$$800\times10^3\left(12\times0+30\times0-12\times0+30\times(-0.003438)\right)=-30\times10^3+R_1$$

$$\therefore\quad R_1=-52512\ \text{N}$$

$$800\times10^3\left(30w_1+100\theta_1-30w_2+50\theta_2\right)=-25\times10^3+M_1$$
$$800\times10^3\left(30\times0+100\times0-30\times0+50\times(-0.003438)\right)=-25\times10^3+M_1$$
$$\therefore\quad M_1=-112520\text{ N-m}$$

$$800\times10^3\left(-12w_1-30\theta_1+24w_2+0\times\theta_2-12w_3+30\theta_3\right)=-90\times10^3+R_2$$
$$457.78\times10^3\left(-12\times0-30\times0+24\times0+0\times\theta_2-12\times(-0.0035938)+30\times(-0.008438)\right)$$
$$=-90\times10^3+R_2.$$
$$\therefore\quad R_2=232492.8\text{ N}$$

(II) Software results.

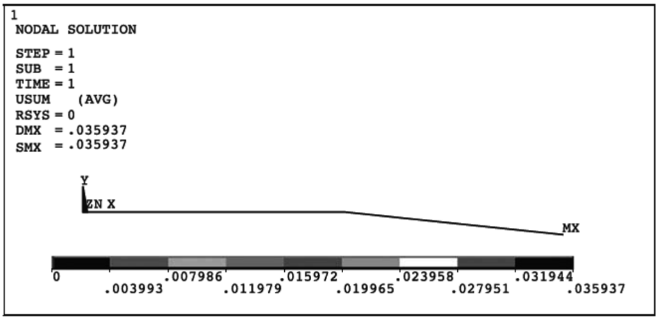


FIGURE 5.19(d) Deflection pattern for a cantilever beam (refer to Appendix D for color figures).

Deflection values at nodes (in meters)

The following degree of freedom results are in global coordinates

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000
3	0.0000	−0.35938E−01	0.0000	0.35938E−01

Maximum absolute values

NODE	0	3	0	3
VALUE	0.0000	−0.35938E−01	0.0000	0.35938E−01

Rotational deflection values at nodes

The following degree of freedom results are in global coordinates

NODE	ROTZ
1	0.0000
2	−0.34375E−02
3	−0.84375E−02

Reaction values

The following *X*, *Y*, *Z* solutions are in global coordinates

NODE	FX	FY	MZ
1	0.0000	−52500	−0.11250E +06
2	0.0000	0.23250E + 06	

ANSWERS FOR EXAMPLE 5.8

Parameter	FEM-Hand calculations	Software results
Deflection at node 3	−0.035938 m	−0.035938 m
Rotational deflection at node		
2	−0.003438 rad	−0.0034375 rad
3	−0.003438 rad	−0.0034375 rad
Reaction force at		
1	−52.512 kN	−52.5 kN
2	232.493 kN	232.25 kN
Reaction moment at node 1	−112.52 kN-m	−112.5 kN-m

Procedure for solving the problems using ANSYS ® 11.0 academic teaching software.

FOR EXAMPLE 5.7

PREPROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Beam > 2D elastic 3 > OK > Close**

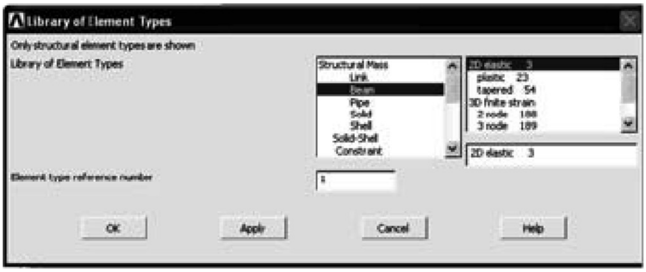


FIGURE 5.20 Element selection.

2. **Main Menu > Preprocessor > Sections > Beam > Common sections,** following dialog box appears

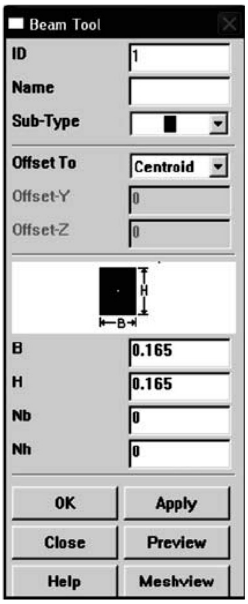


FIGURE 5.21 Choose cross-section of the beam.

In that dialog box, select **Sub-Type,** choose **Square Cross-Section,** then Enter value of B = 0.165 and H = 0.165 as shown in Figure 5.21.
Click on **Preview > OK**
The following figure appears on the screen.

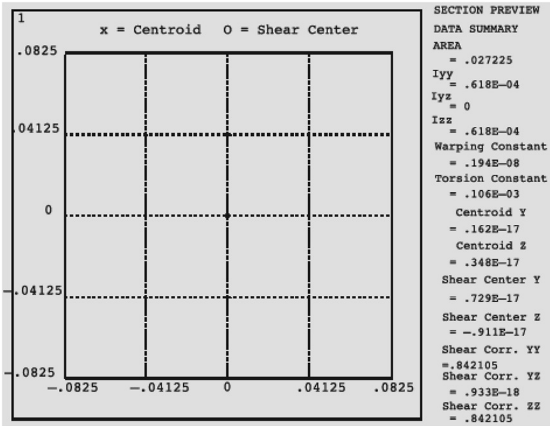


FIGURE 5.22 Details of geometrical properties of the beam.

From Figure 5.22, note down the values of Area A = 0.027225 m^2 and moment of inertia $I_{zz} = 0.618 \times 10^{-4}$ m^4.

3. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

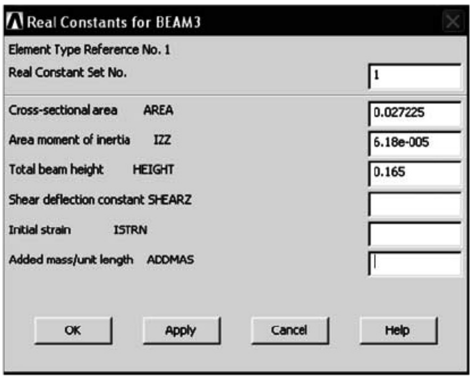


FIGURE 5.23 Enter the area moment of inertia.

Cross-sectional area AREA > **Enter 0.027225**
Area moment of inertia I_{ZZ}> **Enter 0.618e-4**
Total beam height HEIGHT > **Enter 0.165 > OK > Close**
Enter the material properties.

4. **Main Menu > Preprocessor > Material Props > Material Models**
Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**
Enter **EX = 200E9 and RRXY = 0.3 > OK**
(**Close** the Define Material Model Behavior window.)
Create the nodes and elements as shown in the figure.

5. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinated of node 1 > **Apply** Enter the coordinates of node 2 > **Apply** Enter the coordinate of node 3 > **OK**.

Node locations		
Node number	**X-coordinate**	**Y-coordinate**
1	0	0
2	3	0
3	6	0

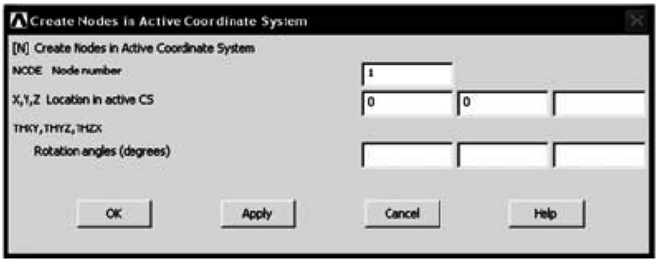


FIGURE 5.24 Enter the node coordinate.

6. **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **Apply** Pick the 2nd and 3rd node > **OK**

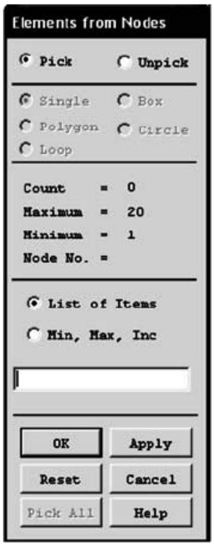


FIGURE 5.25 Pick the nodes to create elements.

Apply the displacement boundary conditions and loads.

7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 1st node > **Apply > All DOF= 0 > OK**

8. **Main Menu > Preprocessor > loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the 2nd node > **Apply > Select UX and UY = 0 > OK**

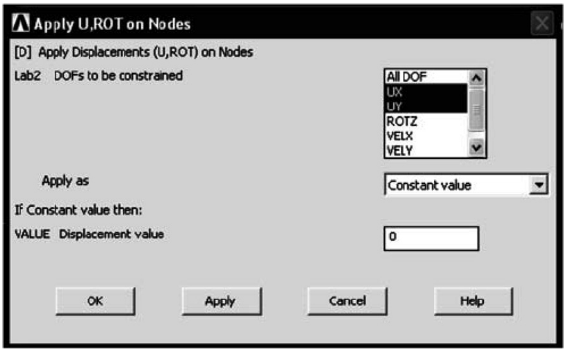


FIGURE 5.26 Applying boundary conditions on node 2.

9. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Beams** Pick the 1st element > **OK > Enter Pressure value at node I = 0 and Pressure value at node J = 30e3 > OK**

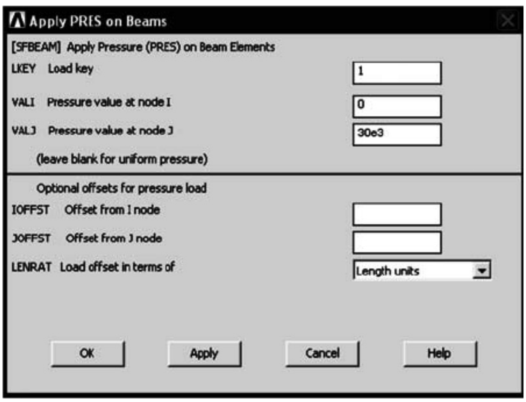


FIGURE 5.27 Applying loads on element 1.

10. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Beams Pick the 2nd element > **OK > Enter Pressure value at node I = 30e3 and Pressure value at node J = 60e3 > OK**

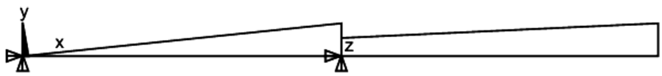


FIGURE 5.28 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First to be safe, save the model.

Solution

The interactive solution proceeds.

11. Main Menu > Solution > Solve> Current LS > OK

The **/STATUS Command** window displays the problem parameters and **the Solve Control Load Step** window is shown. Click the solution options in the **/STATUS** window and if all is OK, select **File > Close**.

In the **Solve Current Load Step** window, select **OK,** and when the solution is complete, **close** the **"Solution is Done!"** window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

12. Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK

This result is shown in Figure 5.18(d).

13. Main Menu > General Postproc > List Results > Nodal Solu > Select Roatation vector sum > OK

14. Main Menu > General Postproc > List Results > Reaction Solu > PL

To find the **bending moment diagram** following procedure is followed.

15. Main Menu > General Postproc > Element Table > Define Table > Add

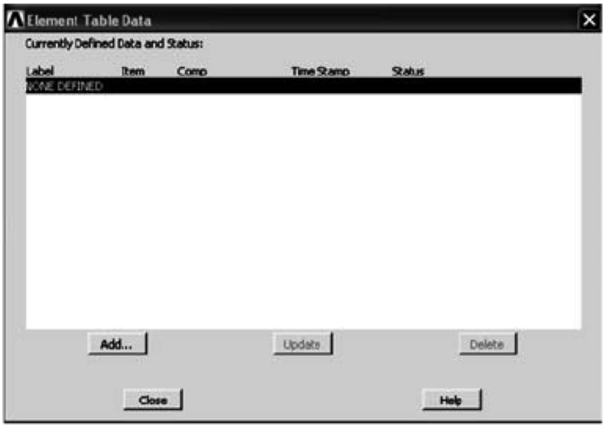


FIGURE 5.29 Define the element table.

Select **By sequence num and SMISC** and type **6 after SMISC** as shown in Figure 5.30. >**APPLY**

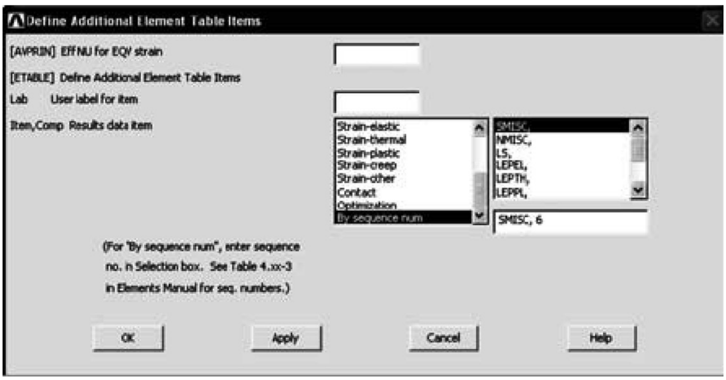


FIGURE 5.30 Selecting options in element table.

Then again select **By sequence num and SMISC** and type **12 after SMISC > OK**

16. **Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res >** Select **SMIS 6 and SMIS 12 in the rows of LabI and LabJ,** respectively as shown in Figure 5.31 > **OK**

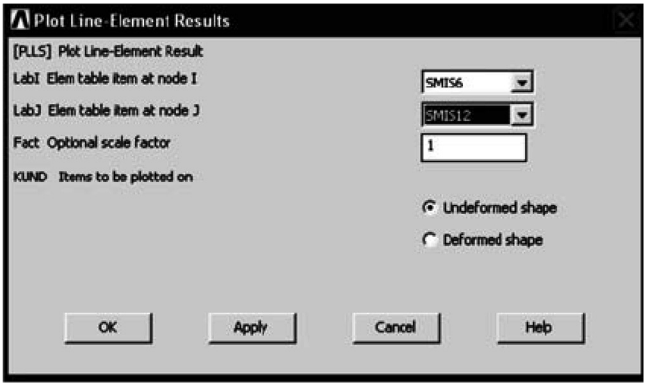


FIGURE 5.31 Selecting options for finding out bending moment.

This result is shown in Figure 5.18(e).

To find the **shear force diagram** following procedure is followed.

17. **Main Menu > General Postproc > Element Table > Define Table > Add**

    Select **By sequence num and SMISC** and type **2 after SMISC > APPLY**

    Then again select **By sequence num and SMISC** and type **8 after SMISC > OK > Close**

18. **Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res >** Select **SMIS 2 and SMIS 8 > OK**

    This result is shown in Figure 5.18(f).

    To find the **bending stress** the following procedure is followed.

19. **Main Menu > General Postproc > Element Table > Define Table > Add**

    Select **By sequence num and LS** and type **3 after LS > APPLY**

    Then again select **By sequence num and LS** and type **6 after LS > OK**

20. **Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res >**

    Select **LS 3 and LS 6 > OK**

    This result is shown in Figure 5.18(g).

    To find the **maximum stress (direct stress + bending stress)** following procedure is followed.

21. Main Menu > General Postproc > Element Table > Define Table > Add

Select **By sequence num and NMISC** and **type 1 after NMISC > APPLY**

Then again select **By sequence num and NMISC and type 3 after NMISC > OK**

22. Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res > Select **NMISC 1 and NIMS 3 > OK**

EXERCISES

1. For the bean shown in Figure 5.32, determine the deflection, slopes, reactions, maximum bending moment, shear force, and maximum bending stress.
Take E = 210 GPa and $I = 7 \times 10^{-4}\ \text{m}^4$.

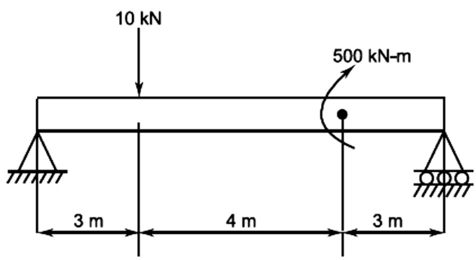


FIGURE 5.32 Exercise 1.

2. Find the deflection, slopes, reactions, maximum bending moment, shear force, and maximum bending stress for the aluminum beam shown in Figure 5.33.
Take E = 200 GPa and $I = 3 \times 10^{-4}\ \text{m}^4$.

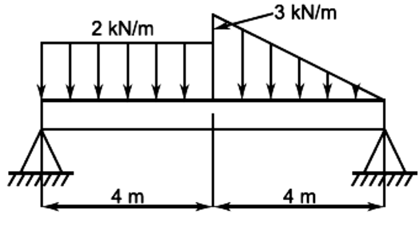


FIGURE 5.33 Exercise 2.

3. Find the deflection at the load and the slopes at the end for the shaft shown in Figure 5.34. Also find the maximum bending moment, maximum bending stress, and reactions developed in the bearings. Consider the shaft to be simply supported at bearings A and B. Take E = 200 GPa.

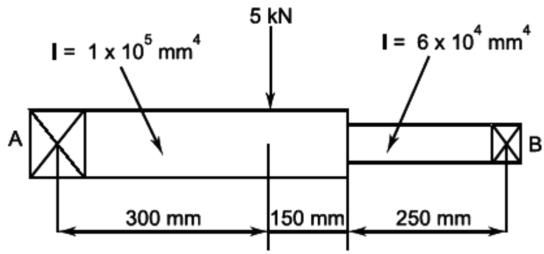


FIGURE 5.34 Exercise 3.

4. Find the deflection of the bean shown in Figure 5.35 under self-weight. Take E = 200 GPa and mass density $\rho = 7800$ kg/m^3.

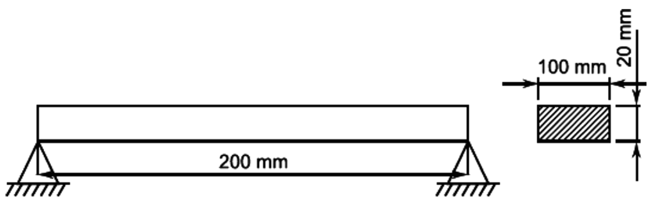


FIGURE 5.35 Exercise 4.

5. Find the deflection and bending stress distribution for the cantilever beam shown in Figure 5.36 under combined loading. Take E = 200 GPa.

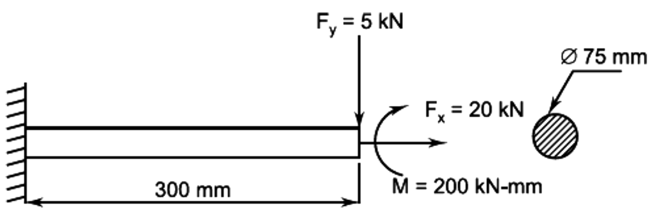


FIGURE 5.36 Exercise 5.

HINT $I = \dfrac{\pi d^4}{64}$

6. For the beam shown in Figure 5.37, determine the deflection at nodes and reaction. Also, plot the bending moment diagram, shear force diagram and find the bending stress. Take E = 200 GPa and $I = 8\times10^{-4}\ \text{m}^4$.

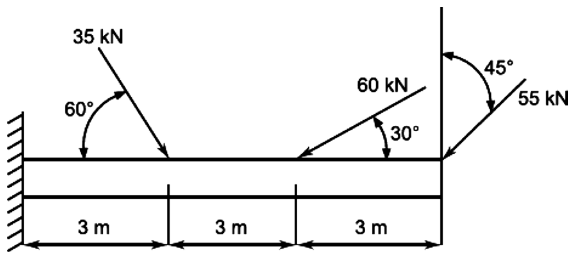


FIGURE 5.37 Exercise 6.

7. A cantilever beam is shown in Figure 5.38. Using 2 beam elements determine the nodal deflection and reaction. Take $E = 0.25\times10^5\ \text{N/mm}^2$ and $I = 8\times10^{-4}\ \text{m}^4$.

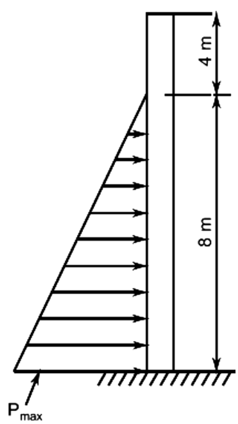


FIGURE 5.38 Exercise 7.

HINT $P_{\max} = \rho \times g \times h$

8. Determine the deflection, reaction, and bending stress for the beam shown in Figure 5.39. Also, plot the bending moment and shear force diagram. Take E = 207 GPa, W = 150 N/mm, h = 800 mm, b = 400 mm, $t_1 = 40$ mm, $t_2 = 40$ mm, and $t_3 = 50$ mm.

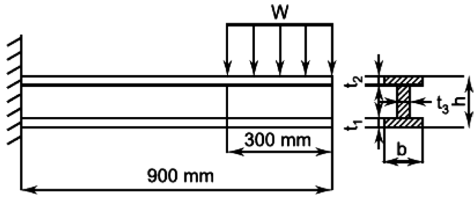


FIGURE 5.39 Exercise 8.

9. Figure 5.40 presents a beam fixed at one end, supported by a cable at the other end, subjected to a uniformly distributed load of 70 lb/in. Take $E = 30 \times 10^6$ psi, Beam cross-section = 4 in $\times$ 4 in, and cable cross-section = 1 in^2. Determine the finite element equilibrium equations of the system by using one finite element for the beam and one finite element for the cable, the displacement of nodes 1 and 2, and the stress distribution in the beam and in the cable.

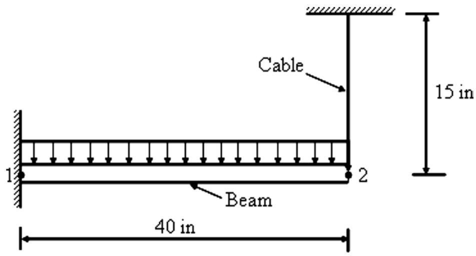


FIGURE 5.40 Exercise 9.

10. What are the differences between truss and beam elements?

REFERENCES

1. Y. W. Hwon and H. Bang, "The Finite Element Method Using MATLAB," Second Edition, CRC Press, 2000.
2. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, Cengage Learning, 2012.
3. S. Moaveni, "Finite Element Analysis: Theory and Application with ANSYS," Third Edition, Prentice Hall, 2008.
4. J. N. Reddy, "An Introduction to the Finite Element Method," Third Edition, McGraw Hill Higher Education, 2004.

5. C. T. F. Ross, "Finite Element Method in Structural Mechanics," Ellis Horwood Limited Publishers, 1985.

6. F. L. Stasa, "Applied Finite Element Analysis for Engineering," Holt, Rinehart and Winston, 1985.

7. L. J. Segerlind, "Applied Finite Element Analysis," Second Edition, John Wiley and Sons, 1984.

8. S. S. Rao, "The Finite Element Method in Engineering," Fifth Edition, Butterworth-Heinemann, 2011.

9. N. Willems and W. M. Lucas, Jr., "Structural Analysis for Engineering," McGraw-Hill, 1978.

10. R. C. Hibbleer, "Mechanics of Materials," Second Edition, Macmillan, 1994.

CHAPTER 6

Stress Analysis of a Rectangular Plate with a Circular Hole

6.1 INTRODUCTION

Two-dimensional problems in structural analysis are dealt with in this chapter. Hand calculations, even with two elements, become too long and hence are not given for these problems: only analytical method solutions and software solutions using ANSYS have been provided.

Two-dimensional problems can either be plane stress or plane strain problems. The method of analysis is the same for both, except that stress–strain matrix is different in two cases.

Plane bodies that are flat and of constant thickness that are subjected to in-plane loading fall under the category of plane stress problems. Stress components σ_z, τ_{xz}, and τ_{yz} assume zero values in these problems.

Some of the elements used in the analysis of two-dimensional problems are constant strain triangles (CST), linear strain triangle (LST), linear quadrilateral, isoparametric quadrilateral, etc. Each of these elements has 2 degrees of freedom per node, namely the translation in the x and y directions.

Stress within the element may be calculated using the equation,

$$\{\sigma\} = [D][B]\{q\}. \tag{6.1}$$

6.2 A RECTANGULAR PLATE WITH A CIRCULAR HOLE

The stress analysis of a rectangular plate with a circular hole problem is assumed as a two-dimensional plane stress problem. Plane stress is defined as a state of stress in which the normal stress and the shear stress directed perpendicular to the plane are assumed to be negligible.

The above problem can be categorized into three subcases.

Subcase 1

A rectangular plate with a very small circular hole at the center with one vertical edge fixed and the other vertical edge is acted upon by a horizontal tensile load in the form of pressure.

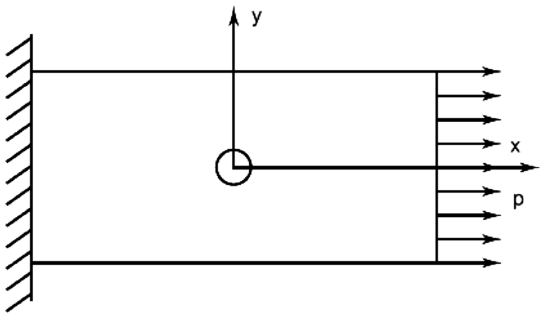


FIGURE 6.1 Rectangular plate with a very small circular hole subjected to tensile load at one edge.

Subcase 2

A rectangular plate with a small circular hole at the center and a horizontal tensile force in the form of pressure is acting on both the vertical edges of the plate.

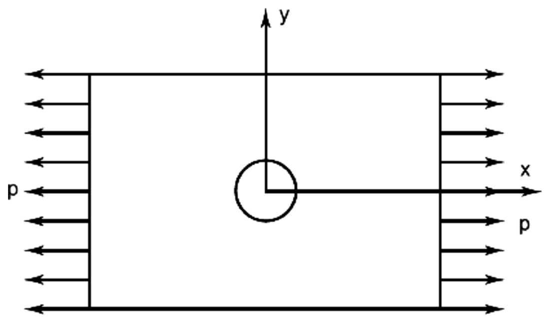


FIGURE 6.2 Rectangular plate with a hole subjected to tensile load at both the edges.

The above problem is solved by exploiting the symmetric geometry and symmetric loading boundary conditions. Now we can draw the above Figure 6.2 for the analysis purpose (Refer to Figure 6.3).

Place the origin of x–y coordinates at the center of the hole and pull on both ends of the plate. Then points on the centerlines will not move perpendicular to them but move along the centerlines. This indicated the appropriate displacement conditions to use, as shown in Figure 6.3.

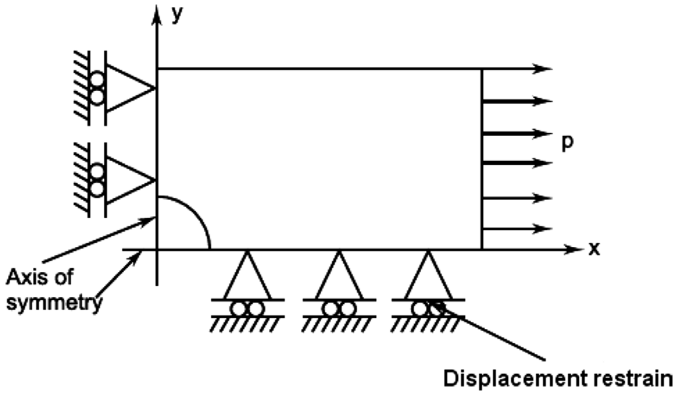


FIGURE 6.3 Finite element model of one-quarter of the plate.

Subcase 3

A rectangular plate with a large circular hole at the center and uniform pressure acts on the boundary of the hole.

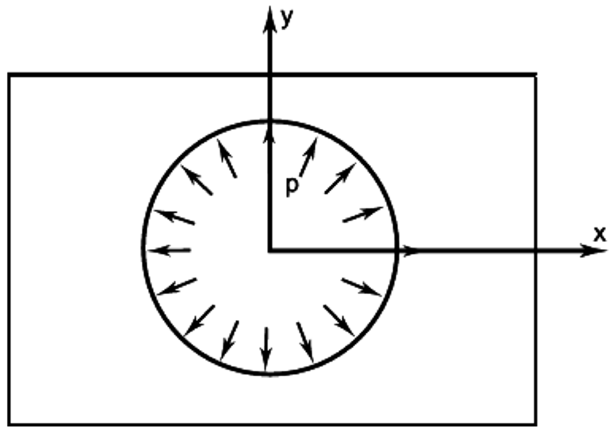


FIGURE 6.4 Rectangular plate with a hole subjected to uniform pressure at the boundary of the hole.

The problem above can be solved by considering one-quarter of the plate and by exploiting the symmetric geometry and loading conditions. The finite element model is shown below.

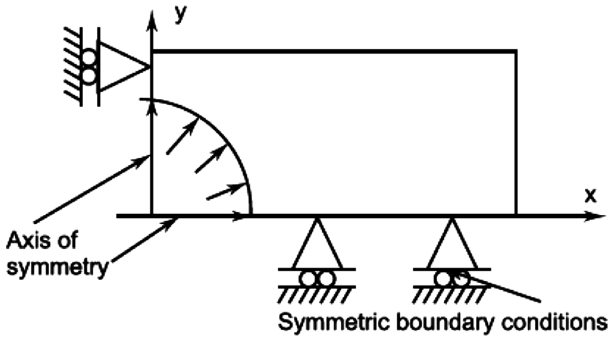


FIGURE 6.5 Finite element model of one-quarter of the plate.

EXAMPLE 6.1

A rectangular plate of size 1000 mm × 500 mm is subjected to uniform pressure, as shown in Figure 6.6. The plate has a thickness of 10 mm and a central hole of 50 mm in diameter. The material of the plate is steel with Young's modulus E = 210 GPa and Poisson's ratio $v = 0.3$. Assume a case of plane stress. Plot the Von Mises stress distribution and compare the result with the analytical method.

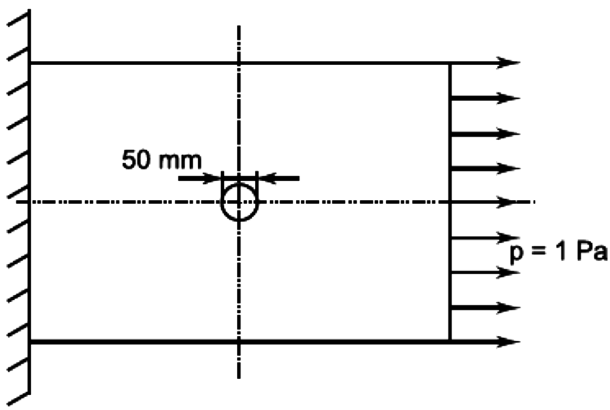


FIGURE 6.6 Rectangular plate with a very small circular hole at the center of the plate.

Solution

(I) Analytical method.

Comparing the above case with the infinite plate with a very small circular hole, for this, the stress concentration factor is (SCF) = 3:

$$\text{SCF} = \frac{\text{Maximum stress}}{\text{Nominal stress}}. \tag{6.2}$$

Hence,

Tensile force = Pressure × cross-sectional area (6.3)

Tensile force = 1 × 0.5 × 0.01 = 0.005 N

$$\text{Nominal stress} = \frac{\text{Tensile force}}{\text{Cross-sectional area}} \tag{6.4}$$

$$\text{Nominal stress} = \frac{0.005}{0.5 \times 0.01} = 1\ \text{N/m}^2$$

Maximum stress = SCF × Nominal stress = 3×1 = 3 Pa.

(II) Software results.

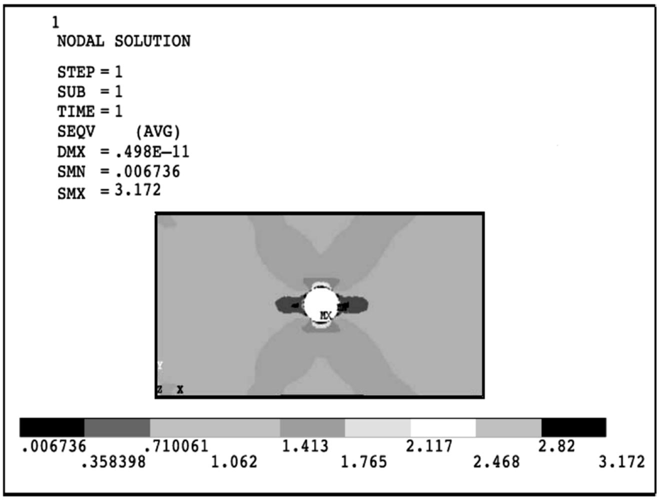


FIGURE 6.6(a) Von Mises stress distribution pattern (refer to Appendix D for color figures).

From the software, we got, Maximum stress (Von Mises stress) = 3.172 Pa.

ANSWERS FOR EXAMPLE 6.1

Parameter	Analytical method	Software results	Percentage of error
Maximum stress	3 Pa	3.172 Pa	5.42

EXAMPLE 6.2

A rectangular plate with a hole at the center is subjected to uniform pressure, as shown in Figure 6.7. The plate is under plane stress. Find the maximum deflection and maximum stress distribution. Also, find the deformed shape of the hole. Assume plate thickness, t = 25 mm, E = 207 GPa, and $v = 0.3$.

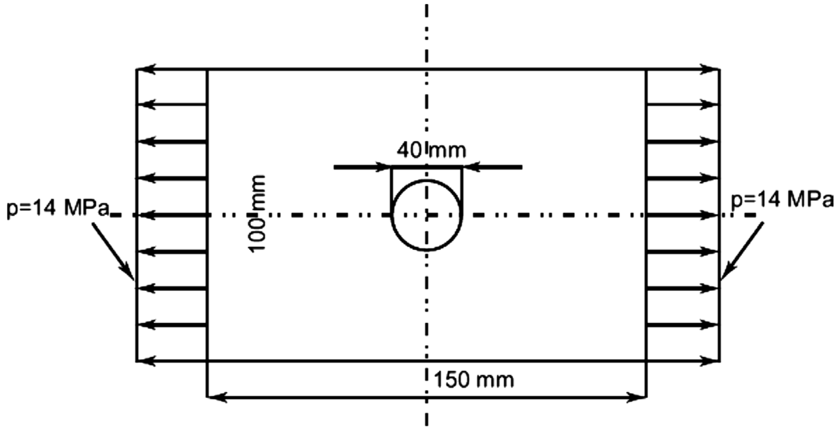


FIGURE 6.7 Rectangular plate with a hole with symmetrical loading.

Solution

(I) Analytical method.

$$\text{Geometric factor} = \frac{\text{Diameter of hole}}{\text{Width of plate}} = \frac{d}{w} \tag{6.5}$$

$$\text{Geometric factor} = \frac{40}{100} = 0.4 .$$

From the design data handbook,

for $\frac{d}{w}$ of 0.4, the stress concentration factor (SCF) = 2.25

$$\text{SCF} = \frac{\text{Maximum stress}}{\text{Nominal stress}} .$$

Hence,

Tensile force = Pressure × Cross-Sectional Area= 14 × 100 × 25 = 35000 N

$$\text{Nominal stress} = \frac{\text{Tensile force}}{\text{Cross-sectional area}} = \frac{\text{Tensile force}}{(w-d)t} = \frac{3500}{(100-40)25} = 23.33 \text{ MPa}$$

Maximum stress = SCF × Nominal stress = 2.25 × 23.333 = 52.5 MPa.

(II) Software results.

For the analysis using software, one-quarter of the plate is modeled and analyzed.

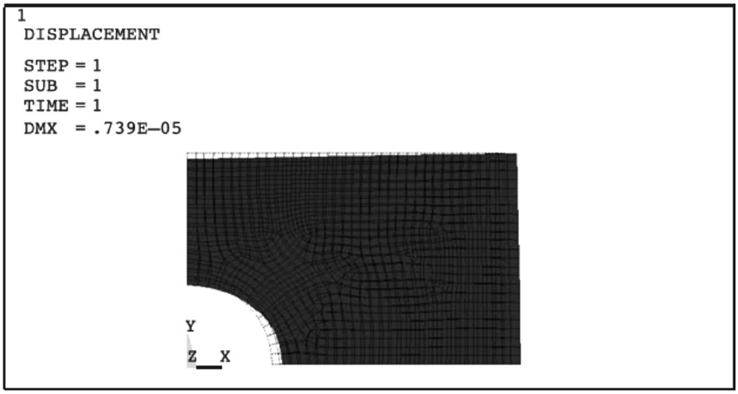


FIGURE 6.7(a) Deformed shape of the hole (refer to Appendix D for color figures).

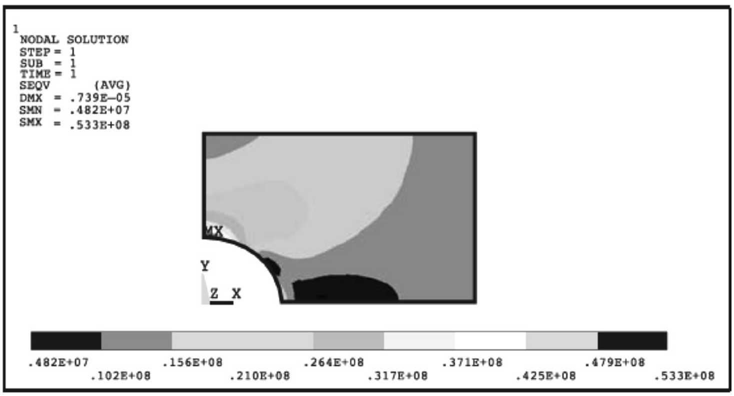


FIGURE 6.7(b) Von Mises stress distribution pattern (refer to Appendix D for color figures).

From the software, we got, maximum stress = 53.3 MPa.

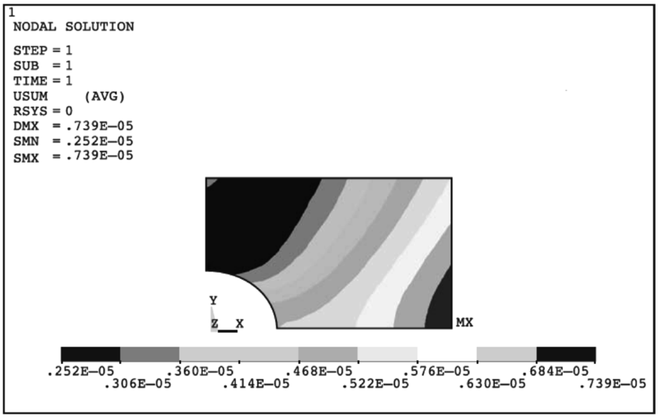


FIGURE 6.7(c) Deflection pattern (refer to Appendix D for color figures).

ANSWERS FOR EXAMPLE 6.2

Parameter	Analytical method	Software results	Percentage of error
Maximum stress	52.5 MPa	53.3 MPa	1.5
Maximum deflection	……	7.39×10^{3} mm	…..

EXAMPLE 6.3

Determine the stress distribution and displacement for a rectangular plate with a hole at the center of the plate with a uniform thickness of 10 mm. A uniform pressure of p = 10 MPa acts on the boundary of the hole, as shown in Figure 6.8. Assume Young's modulus E = 120 GPa and the Poisson's ratio is 0.28. Assume plane stress condition.

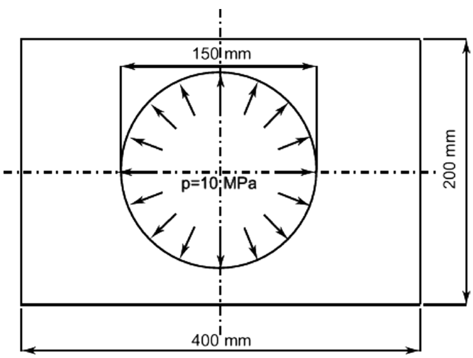


FIGURE 6.8 Rectangular plate with a hole subjected to uniform pressure at the boundary of the hole.

Solution

(I) Software results.

For the analysis using software, one-quarter of the plate is modeled and analyzed.

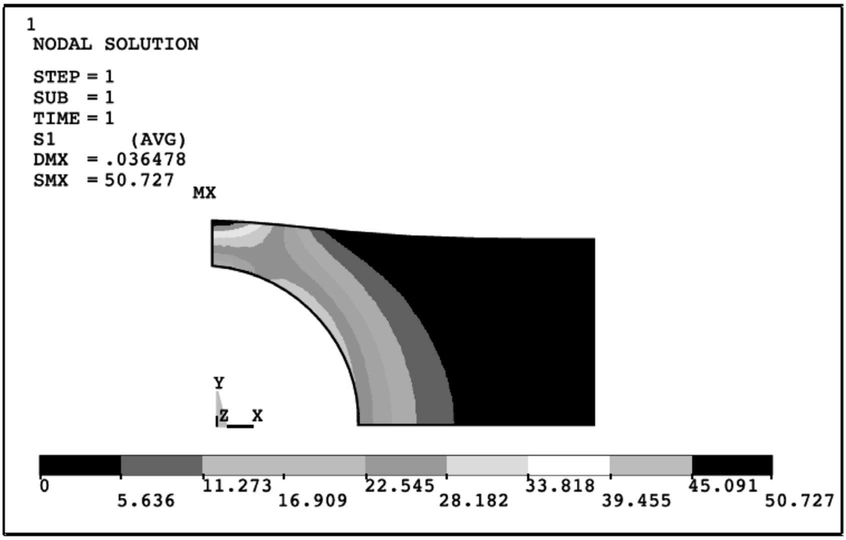


FIGURE 6.8(a) First principal stress distribution pattern (refer to Appendix D for color figures).

From the software, we got maximum stress = 50.727 MPa.

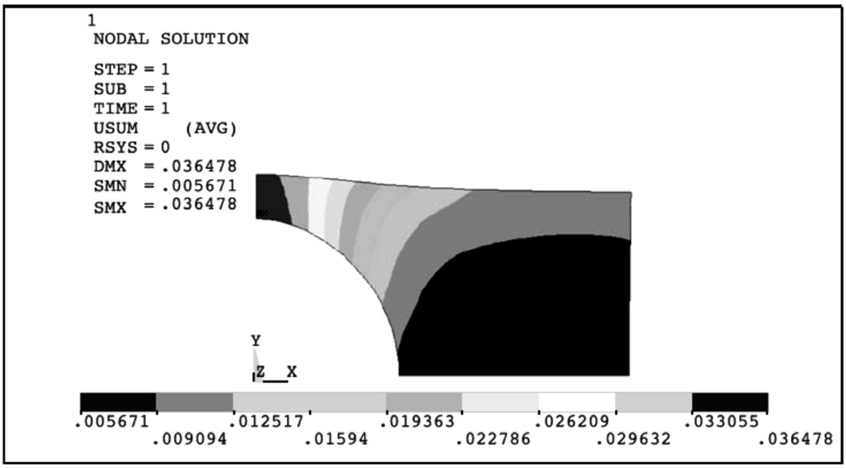


FIGURE 6.8(b) Deflection pattern (refer to Appendix D for color figures).

Validation of the results

The reactions at the supports must balance the applied forces. Therefore, from the software, the total reaction force in the x-direction is – 7500 N.

Applied force = (pressure) × (projected distance in the x-direction of the line along which the constant pressure acts) × (thickness) = $p \times r \times t$ (6.6)

Applied force = 10 × 75 × 10 = 7500 N in the positive x-direction.

So the reaction cancels out the applied force in the x-direction.

ANSWERS FOR EXAMPLE 6.3

Parameter	Software results
Maximum stress	50.727 MPa
Maximum deflection	0.036478 mm

Procedure for solving the problem using ANSYS ® 11.0 academic teaching software.

For Example 6.2

PREPROCESSING:

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Structural Solid > Quad 4 node 42 > OK**

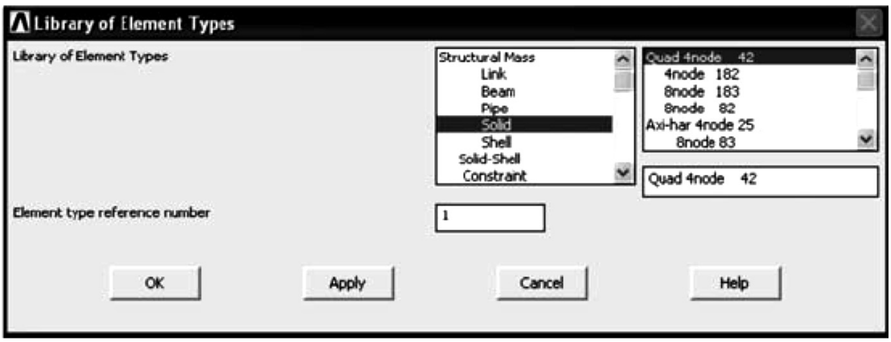


FIGURE 6.9 Element selection.

Select the option where you define the plate thickness.

2. Options (Element behavior K3) > Plane stress w/ thk > OK > Close

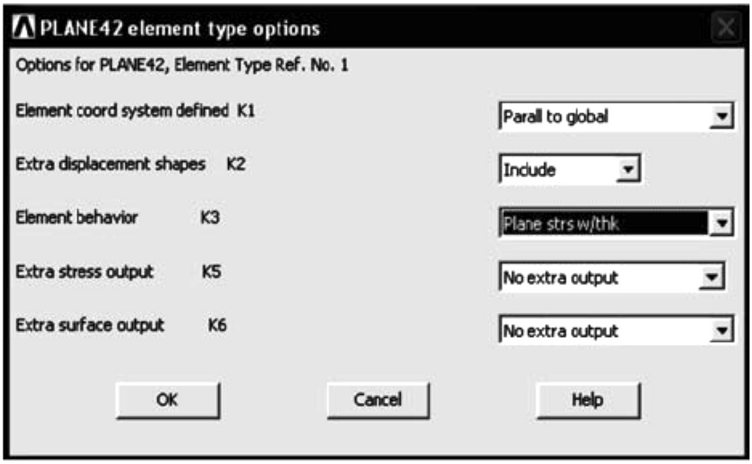


FIGURE 6.10 Element options.

3. Maun Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK

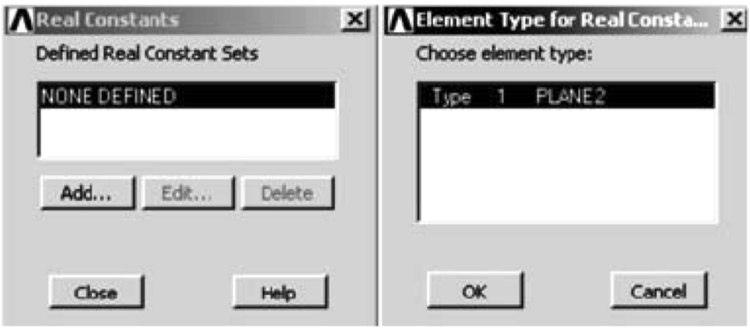


FIGURE 6.11 Real constants.

(Enter the plate thickness of 0.025 m) > **Enter 0.025 > OK > Close**

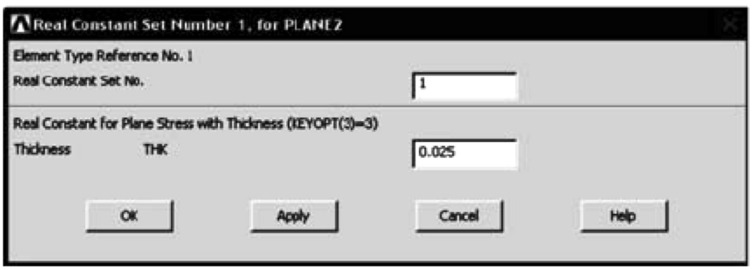


FIGURE 6.12 Enter the plate thickness.

Enter the material properties.

4. **Main Menu > Preprocessor > Material Props > Material Models**

Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

Enter **EX = 2.07E11** and **PRXY = 0.3 > OK** (**Close** the Define Material Model Behavior window.)

Create the geometry for the upper-right quadrant of the plate by subtracting a 0.04 m diameter circle from a 0.075 × 0.05 m rectangle. Generate the rectangle first.

5. **Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners**

Enter (lower left corner) **WP X =0.0, WP Y = 0.0 and Width = 0.075, Height = 0.05 > OK**

6. **Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle** Enter **WP X = 0.0, WP Y =0.0 and Radius = 0.02 > OK**

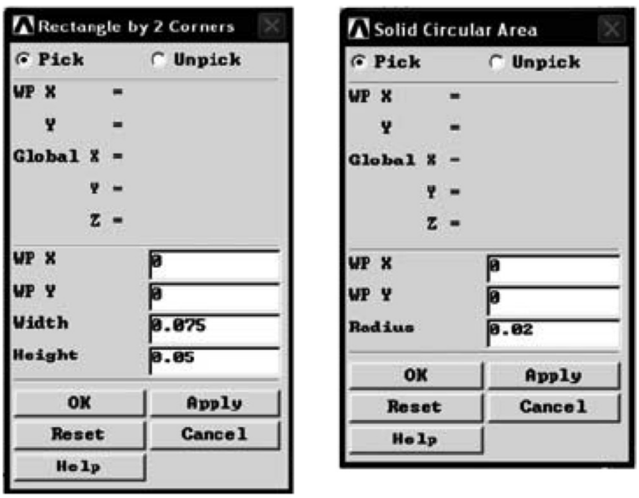


FIGURE 6.13 Create areas.

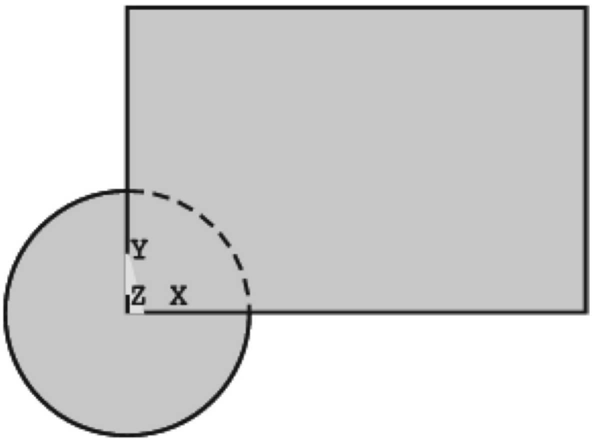


FIGURE 6.14 Rectangle and circle.

Now **subtract** the circle form the rectangle. (Read the messages in the window at the bottom of the screen as necessary.)

7. **Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Area**s

Pick the rectangle > **OK,** then pick the circle **> OK**

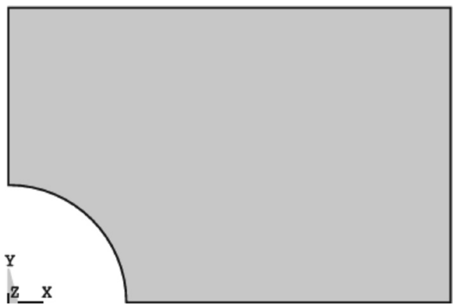


FIGURE 6.15 Geometry for quadrant of plate.

Create a mesh triangular element over the quadrant area.

8. **Main Menu > Preprocessor > Meshing > Mesh Tool**

The **Mesh Tool** dialog box appears. In that dialog box, click on the **Smart Size** and move the slider available below the **Smart Size to 2** (i.e., toward the **Fine** side**).** Then close the Mesh Toolbox.

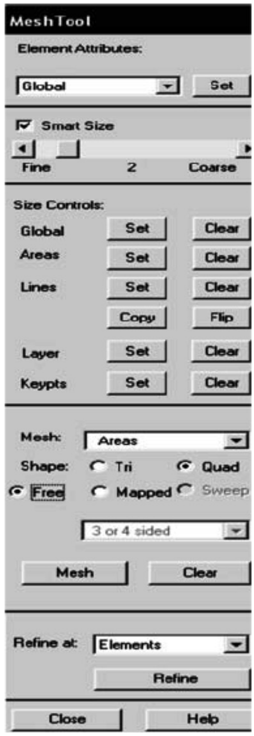


FIGURE 6.16 Mesh toolbox.

9. **Main Menu > Preprocessor > Meshing > Mesh > Areas > Free** Pick the quadrant **> OK**

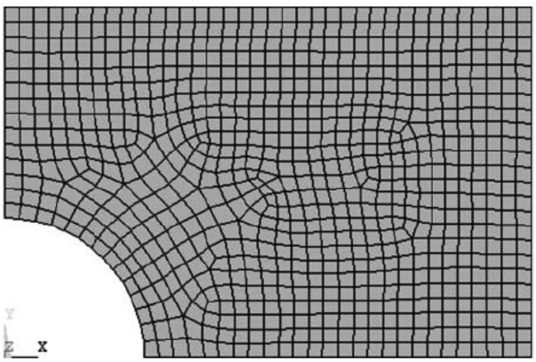


FIGURE 6.17 Quad element mesh.

Apply the displacement boundary conditions and loads.

10. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines** Pick the left edge of the quadrant > **OK > UX = 0 > OK**
11. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines** pick the bottom edge of the quadrant > **OK > UY = 0 > OK**
12. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines.** Pick the right edge of the quadrant > **OK > Pressure = -14E6 > OK**

(A positive pressure would be a compressive load, so we use a negative pressure. The pressure is shown as a single arrow.)

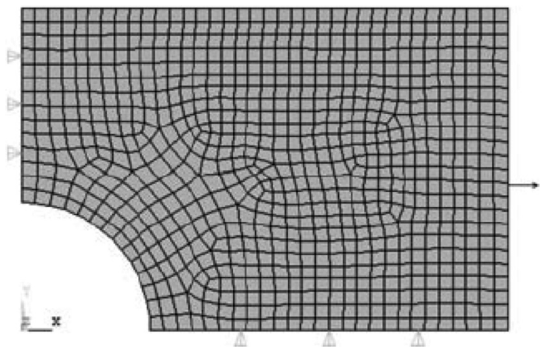


FIGURE 6.18 Model with loading and displacement boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First, to be safe, save the model.

Solution

The interactive solution proceeds

13. **Main Menu > Solution > Solve > Current LS > OK**

The **/STATUS Command** window displays the problem parameters, and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window, and if all is OK, select **File > Close.**

In the **Solve Current Load Step** window, select **OK,** and when the solution is complete close the "**Solution is Done!**" window

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values. First, examine the deformed shape.

14. Main Menu > General Posrproc > Plot Results > Deformed Shape > Def. + Undeformed > OK

This result is shown in Figure 6.7(a).

15. Main Menu > General Posrproc > Plot Results > Contour Plot > Nodal Solu > Stress > Von Mises stress > OK

This result is shown in Figure 6.7(b).

16. Main Menu > General Posrproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement vector sum > OK

This result is shown in Figure 6.7(c).

EXERCISES

1. Find the maximum stress in the aluminum plate shown in Figure 6.19. Consider an aluminum plate 10 mm thick with a hole at the center of the plate. Assume plane stress condition. Take E = 70 GPa and υ = 0.35. Also, calculate the maximum stress by analytical method and compare the results.

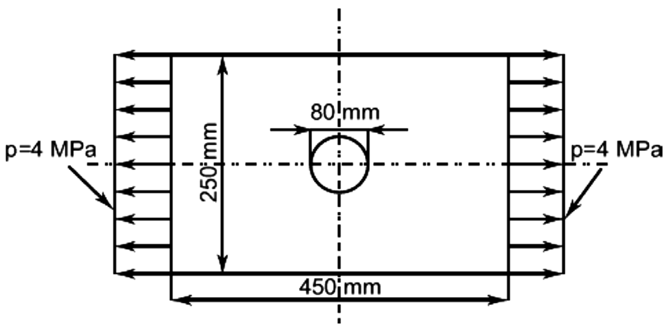


FIGURE 6.19 Exercise 1.

2. Find the maximum stress for the plate shown in Figure 6.20 if the hole is located halfway between the center line and the top edge, as shown. Take E = 70 GPa and υ = 0.35. Assume plane stress condition.

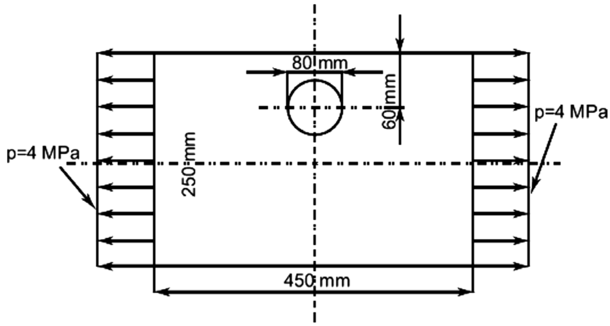


FIGURE 6.20 Exercise 2.

[Hint: Model half of the plate by taking symmetry about the y-axis.]

3. For the plate shown in Figure 6.21, find the maximum stress. Take Young's modulus E= 210 GPa, Poisson's ratio = 0.3. Assume plane stress condition. The thickness of the plate = 10 mm with a hole at the center of the plate.

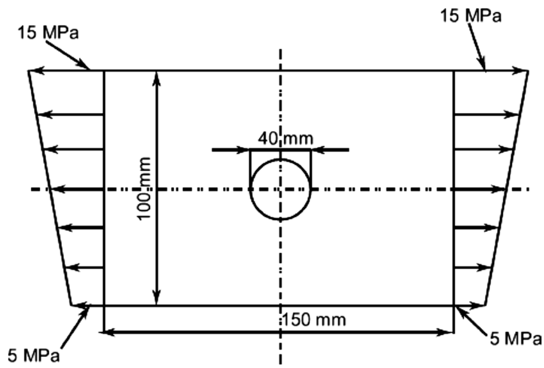


FIGURE 6.21 Exercise 3.

4. For the plate shown in Figure 6.22, find the maximum stress. The plate is made up of two materials.

For Material 1, E = 210 GPa and v = 0.3.

For Material 2, E = 70 GPa and v = 0.35.

Assume plane stress condition.

The thickness of the plate = 10 mm with a hole at the center of the plate.

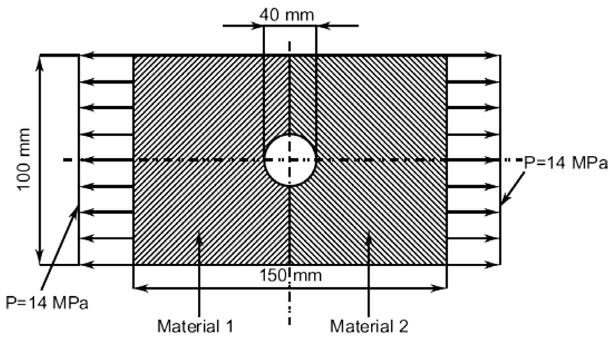


FIGURE 6.22 Exercise 4.

5. For the plate with a hole at the center shown in Figure 6.23, find the maximum stress. Take E = 210 GPa and υ = 0.3, the thickness of plate t = 10 mm. Assume plane stress condition.

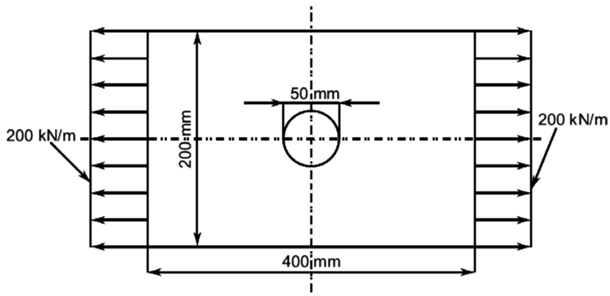


FIGURE 6.23 Exercise 5.

[Hint: To find the pressure, divide the distributed load by the thickness of the plate.]

6. Determine the stresses in the plate with the round hole subjected to the tensile stresses in Figure 6.24. Find the maximum stress. Take E = 210 GPa and υ = 0.25, the thickness of plate t = 10 mm. Assume plane stress condition.

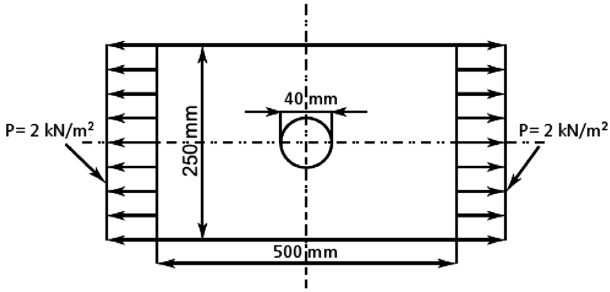


FIGURE 6.24 Exercise 6.

7. For the plate with a hole at the center shown in Figure 6.25, find the maximum stress. Take E = 210 GPa and υ = 0.3, the thickness of plate t = 0.375 in. Assume plane stress condition.

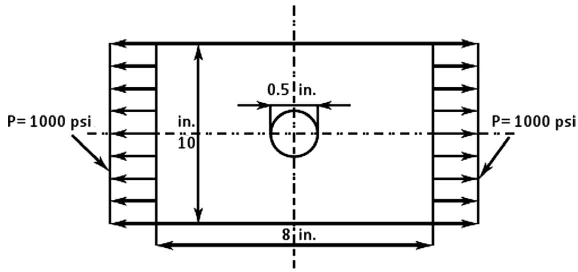


FIGURE 6.25 Exercise 7.

8. For the plate with a hole at the center shown in Figure 6.26, find the maximum stress. Take E = 30×10^6 psi and ν = 0.25, the thickness of plate t = 0.1 in. Assume plane stress condition.

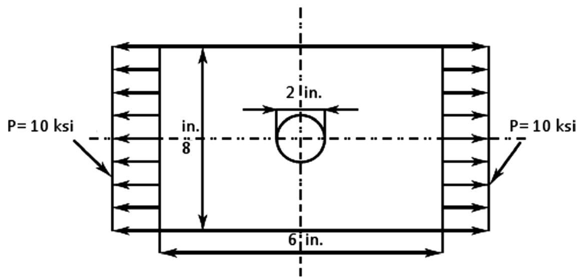


FIGURE 6.26 Exercise 8.

9. Find the maximum stress for the plate shown in Figure 6.27 if the hole is located halfway between the center line and the top edge, as shown. Take E = 20×10^6 N/cm^2 and ν = 0.25. Assume plane stress condition.

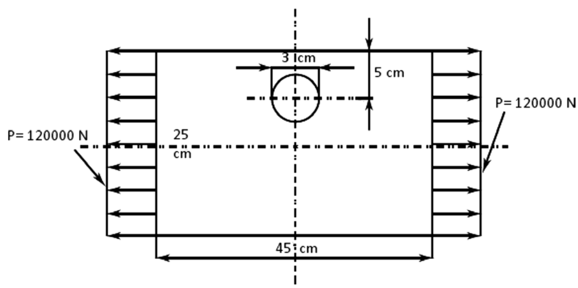


FIGURE 6.27 Exercise 9.

[Hint: Model half of the plate by taking symmetry about the y-axis.]

REFERENCES

1. C. S. Desai and J. F. Abel, "Introduction to the Finite Element Method," Van Nostrad Reinhold, New York, 1972.
2. L. J. Segerlind, "Applied Finite Element Analysis," Second Edition, John Wiley and Sons, New York, 1984.
3. M. Asghar Bhatti, "Fundamental Finite Element Analysis and Applications with Mathematica and MATLAB Computations," John Wiley and Sons, New York, 2005.
4. H. Grandin, Jr., "Fundamentals of the Finite Element Method," Macmillan Publishing Company, New York, 1986.
5. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, CL Engineering, California, 2012.
6. F. L. Stasa, "Applied Finite Element Analysis for Engineering," Holt, Rinehart, and Winston, Texas, 1985.
7. N. Troyani, C. Gomes, and G. Sterlacci, "Theoretical Stress Concentration Factors for Short Rectangular Plates with Central Circular Holes," Journal of Mechanical Design, ASME, Vol. 124, pp. 126–128, 2002.
8. T. Hayashi, "Stress Analysis of a Rectangular Plate with a Circular Hole under Uniaxial Loading," Journal of Thermoplastic Composite Materials, Vol. 2, pp. 143-151, 1989.

CHAPTER 7

THERMAL ANALYSIS

7.1 INTRODUCTION

The computation of temperature distribution within a body will be used in this chapter due to its importance in many engineering applications. Conduction (q) is the transfer of heat through materials without any net motion of the mass of the material. The rate of heat flow in the x-direction by conduction (q) is given by

$$q = kA\frac{\partial T}{\partial x} \tag{7.1}$$

where

k is the thermal conductivity of the material, A is the area normal to the x-direction through which heat flows, T is the temperature, and x is the length parameter.

Convection is the process by which thermal energy is transferred between a solid and a fluid surrounding it. The rate of heat flow by convection (q) is given by

$$q = hA(T - T_\infty) \tag{7.2}$$

where

h is the heat transfer coefficient, A is the surface area of the body through which heat flows, T is the temperature of the surface of the body, and T_∞ is the temperature of the surrounding medium.

Thermal analysis is one of the scalar field problems. These problems have only 1 degree of freedom per node, namely temperature. In this chapter, one-dimensional and two-dimensional heat conduction problems are dealt with. In these problems, a bar element with 2 end nodes, each having temperature (T) as a sole degree of freedom, is useful. Nodal heat flow rates (Q) or heat fluxes are analogous quantities to nodal forces, in structural bar element.

The governing equation for this element is given by,

$$\frac{Ak}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = \frac{q}{2}\begin{Bmatrix} L \\ L \end{Bmatrix} + \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} \tag{7.3}$$

where,

q = heat generation rate per unit length

$\frac{Ak}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$ = element heat conductivity matrix.

7.2 PROCEDURE OF FINITE ELEMENT ANALYSIS (RELATED TO THERMAL PROBLEMS)

Step 1. Select element type.

Step 2. Select temperature distribution function.

Step 3. Define the temperature gradient/temperature and heat flux/temperature gradient relationships.

Step 4. Derive the element conduction matrix and heat flux matrix.

Step 5. Assemble the element equations to obtain the global equations and introduce boundary conditions.

Step 6. Solve for the nodal temperatures.

Step 7. Solve for the element temperature gradients and heat fluxes.

7.3 ONE-DIMENSIONAL HEAT CONDUCTION

EXAMPLE 7.1

A composite wall consists of three materials. The outer temperature is T_0 = 20°C. Convection heat transfer takes place on the inner surface of the wall with T_∞ = 800°C and h = 25 W/m2°C. Determine the temperature distribution in the wall. Take k_1 = 30 W/m°C, k_2 = 50 W/m°C, k_3 = 20 W/m°C.

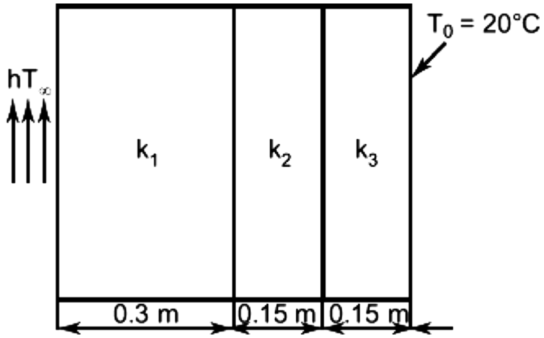


FIGURE 7.1 A composite wall consists of three materials for Example 7.1.

Solution

(I) Analytical method.

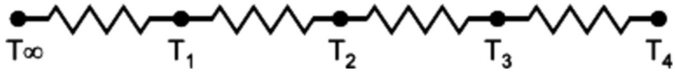


FIGURE 7.1(a) Analytical method for Example 7.1.

Heat flow rate per unit area,

$$Q = \frac{T_\infty - T_4}{\frac{1}{h} + \frac{L_1}{k_1} + \frac{L_2}{k_2} + \frac{L_3}{k_3}} = \frac{800 - 20}{\frac{1}{25} + \frac{0.3}{30} + \frac{0.15}{50} + \frac{0.15}{20}} = 12892.6 \text{ W/m}^2.$$

Now,

$$Q = h(T_\infty - T_1) = \frac{k_1(T_1 - T_2)}{L_1} = \frac{k_2(T_2 - T_3)}{L_2} = \frac{k_3(T_3 - T_4)}{L_3}$$

$$12892.6 = 25(800 - T_1) = \frac{30(T_1 - T_2)}{0.3} = \frac{50(T_2 - T_3)}{0.15} = \frac{20(T_3 - 20)}{0.15}.$$

By solving the above, we get

$$T_1 = 284.3\,°\text{C}$$
$$T_2 = 155.37\,°\text{C}$$
$$T_3 = 116.7\,°\text{C}.$$

(II) FEM by calculations [Refer to Figure 7.1(b)],

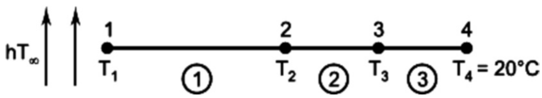


FIGURE 7.1(b) Finite element model for Example 7.1.

Governing equation is,

$$\frac{k}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = q\begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}.$$

Since there is no heat generation specified, $q = 0$.

For element 1,

$$\frac{k_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = q\begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}$$

$$\frac{30}{0.3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = 0 + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}.$$

For element 2,

$$\frac{k_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} = q\begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_2 \\ +Q_3 \end{Bmatrix}$$

$$\frac{50}{0.15}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} = 0 + \begin{Bmatrix} -Q_2 \\ +Q_3 \end{Bmatrix}.$$

For element 3,

$$\frac{k_3}{L_3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_3 \\ T_4 \end{Bmatrix} = q\begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_3 \\ +Q_4 \end{Bmatrix}$$

$$\frac{20}{0.15}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_3 \\ T_4 \end{Bmatrix} = 0 + \begin{Bmatrix} -Q_3 \\ +Q_4 \end{Bmatrix}.$$

Global equation after assembly,

$$\left[\begin{array}{cccc} 100 & -100 & 0 & 0 \\ -100 & 100+333.33 & -333.33 & 0 \\ 0 & -333.33 & 333.33+133.33 & -133.33 \\ \hline 0 & 0 & -133.33 & 133.33 \end{array}\right]\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} -Q_1 \\ 0 \\ 0 \\ +Q_4 \end{Bmatrix}.$$

Boundary conditions are T_4 = 20°C and

$$Q_1 = -h(T_\infty - T_1)$$
$$-Q_1 = 25(800 - T_1) = 20000 - 25T_1.$$

So modified equation,

$$\begin{bmatrix} 100+25 & -100 & 0 \\ -100 & 433.33 & -333.33 \\ 0 & -333.33 & 466.66 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 20000 \\ 0 \\ 0 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ 20\times 133.33 \end{Bmatrix}$$

$$\begin{bmatrix} 125 & -100 & 0 \\ -100 & 433.33 & -333.33 \\ 0 & -333.33 & 466.66 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 20000 \\ 0 \\ 0 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ 2666.6 \end{Bmatrix}.$$

After solving the matrix and simultaneous equations, we get,

$$T_1 = 284.3\,°\mathrm{C}$$
$$T_2 = 155.37\,°\mathrm{C}$$
$$T_3 = 116.7\,°\mathrm{C}.$$

(III) Software results.

Temperature values

NODE	TEMP
1	284.30
2	155.37
3	116.69
4	20.000
5	800.00

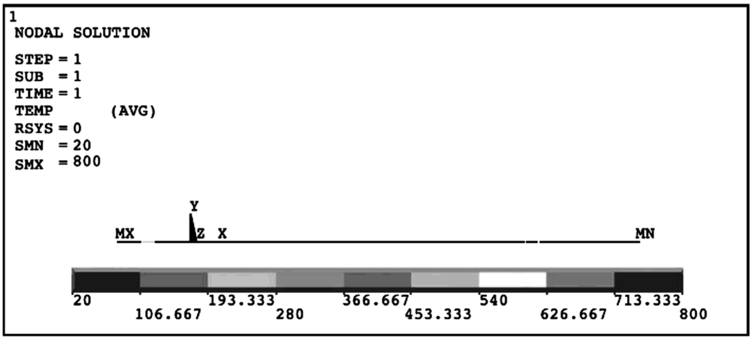


FIGURE 7.1(c) Temperature distribution in a composite wall (refer to Appendix D for color figures).

ANSWERS FOR EXAMPLE 7.1

Parameter	Analytical method	FEM-hand calculation	Software results
Temperature			
at node 1	284.3°C	284.3°C	284.3°C
at node 2	155.37°C	155.37°C	155.37°C
at node 3	116.7°C	116.7°C	116.69°C

Procedure for solving the problem using ANSYS ®11.0 academic teaching software.

FOR EXAMPLE 7.1

PREPROCESSING

1. **Main Menu > Preferences, then select Thermal > OK**

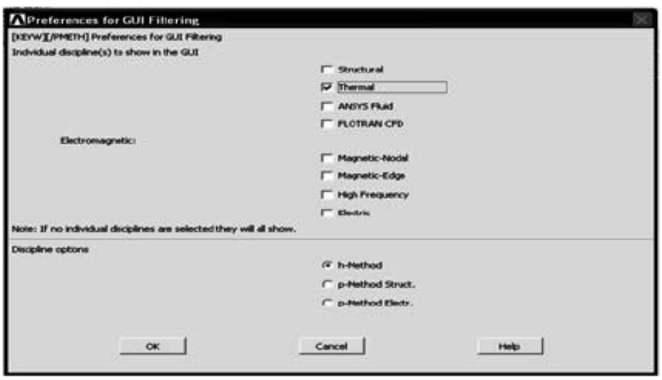


FIGURE 7.2 Selecting the preferences.

2. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add> Click on Link > then on 2d conduction 32 > OK > Add > Click on Link > then on 3D convection 34 > OK > Close**

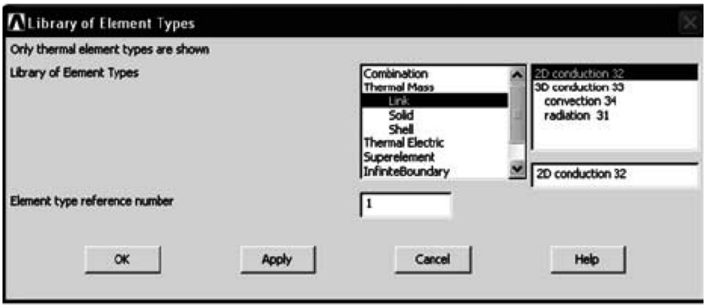


FIGURE 7.3 Selecting the element for conduction.

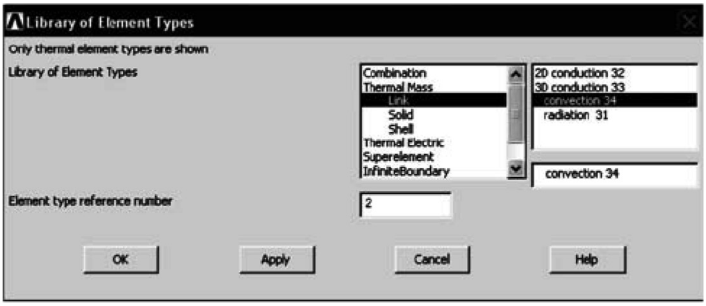


FIGURE 7.4 Selecting the element for convection.

3. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > Click on Link 32 > OK**

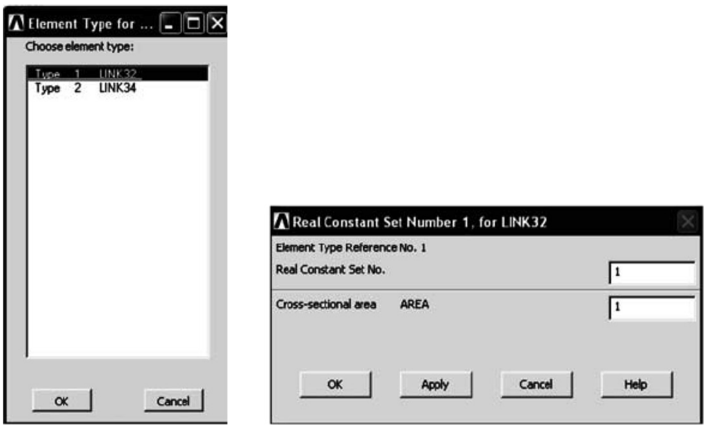


FIGURE 7.5 Enter the cross-sectional area for Link 32.

Enter cross-sectional area AREA > **Enter 1 > OK**

Add > Click on Link 34 > OK

Enter cross-sectional area AREA > **Enter 1 > OK > Close**

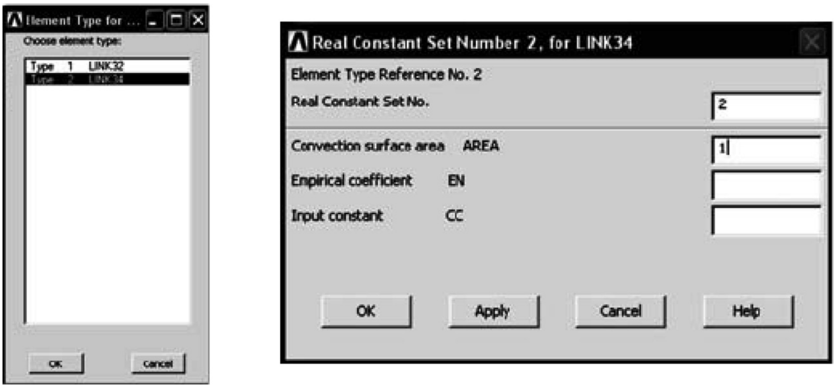


FIGURE 7.6 Enter the cross-sectional area for Link 34.

Enter the material properties.

4. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1,

   click **Thermal > Conductivity > Isotropic**

   Enter **KXX = 30 > OK**

   Then in the material model window, click on **Material menu > New Model > OK**

   Material Model Number 2,

   click **Thermal > Conductivity > Isotropic**

   Enter **KXX = 50 > OK**

   Then in the material model window, click on **Material menu > New Model > OK**

   Material Model Number 3,

click **Thermal > Conductivity > Isotropic**

Enter **KXX = 20 > OK**

Then in the material model window, click on **Material menu > New Model > OK**

Material Model Number 4,

click **Thermal > Convection or Film Coef.**

Enter **HF = 25 > OK**

(**Close** the Define Material Model Behavior window.)

Create the nodes and elements.

5. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply** Enter the coordinates of node 2 > **Apply** Enter the coordinates of node 3 > **Apply** Enter the coordinates of node 4 > **Apply** Enter the coordinates of node 5 > **OK**

Node Locations		
Node number	X coordinates	Y coordinates
1	0	0
2	0.3	0
3	0.45	0
4	0.6	0
5	–0.1	0

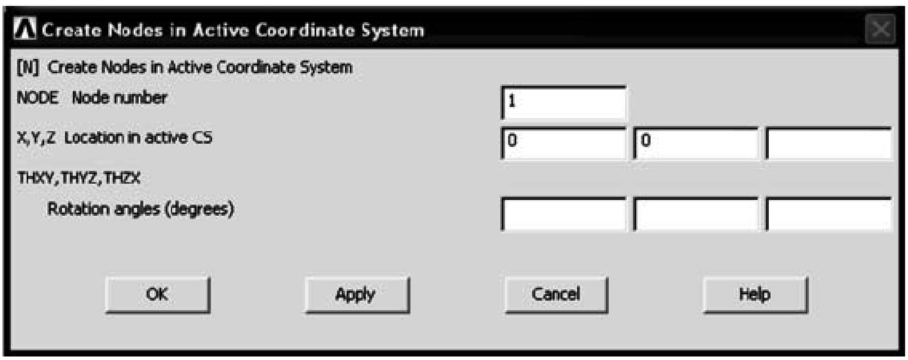
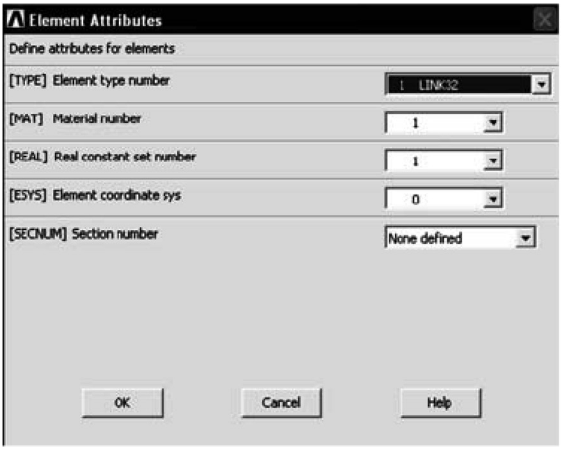


FIGURE 7.7 Enter the node coordinates.

6. **Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes > OK > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **OK**



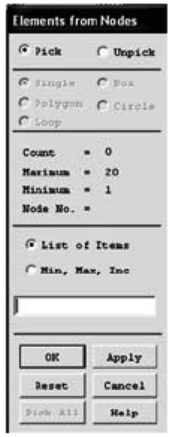
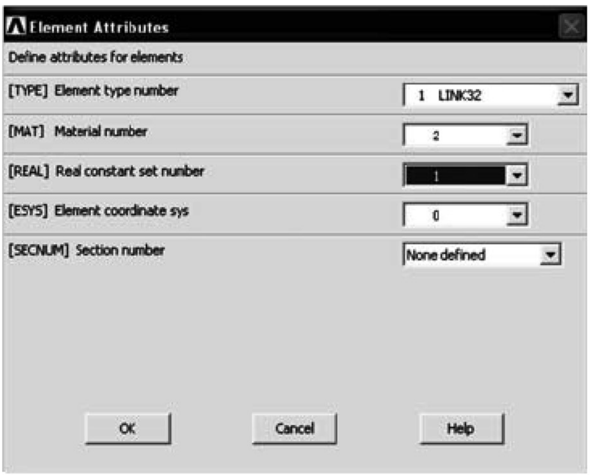


FIGURE 7.8 Assigning element attributes to element 1 and creating element 1.

Elem Attributes > change the material number to 2 > OK > Auto Numbered > Thru nodes Pick the 2nd and 3rd node > **OK**



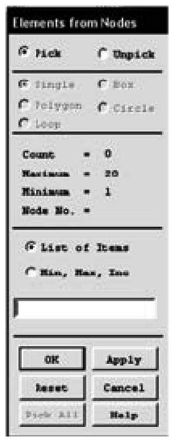


FIGURE 7.9 Assigning element attributes to element 2 and creating element 2.

Elem Attributes > change the material number to 3 > OK > Auto Numbered > Thru nodes Pick the 3rd and 4th node > **OK**

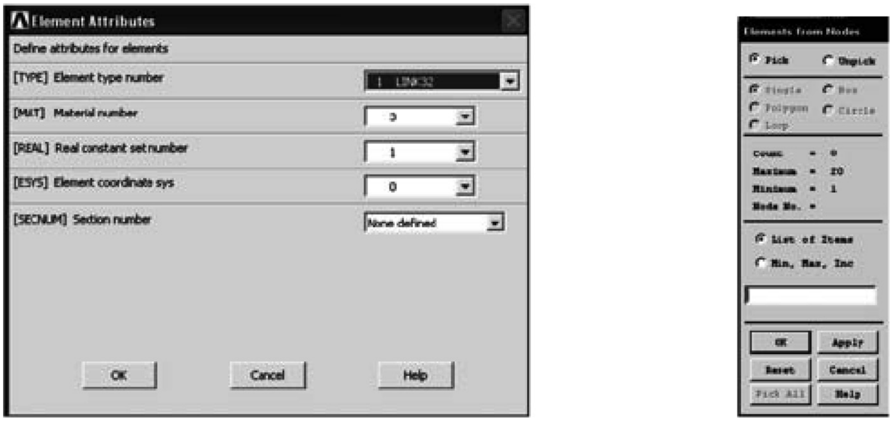


FIGURE 7.10 Assigning element attributes to element 3 and creating element 3.

Elem Attributed > change the element type to Link 34 > change the material number to 4 > change the Real constant set number to 2 > OK > Auto Numbered > Thru nodes Pick the 1st and 5th node > **OK**

FIGURE 7.11 Assigning element attributes to element 4 and creating element 4.

Apply the boundary conditions and temperature.

7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes** Pick the 4th node > **Apply > Click on TEMP and Enter Value = 20 > OK**

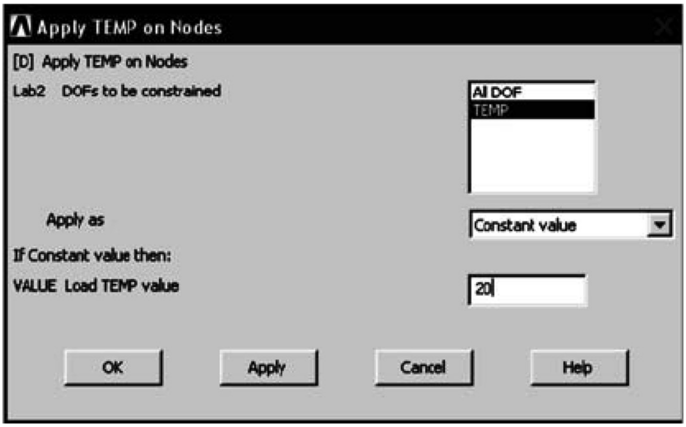


FIGURE 7.12 Applying temperature on node 4.

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes** Pick the 5th node > **Apply > Click on TEMP and Enter Value = 800 > OK**

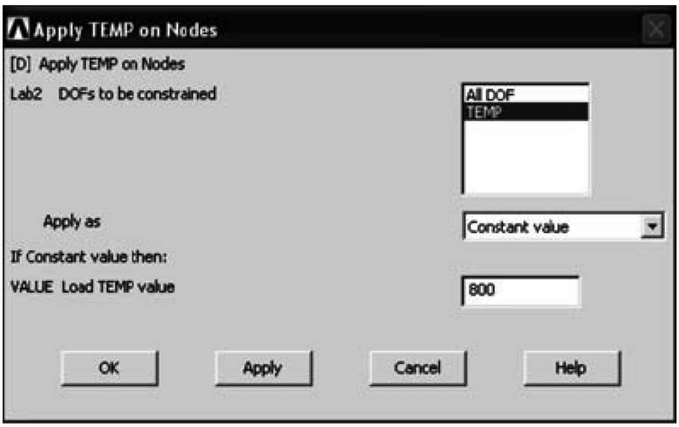


FIGURE 7.13 Applying temperature on node 5.

Solution

The interactive solution proceeds.

9. Main Menu > Solution > Solve > Current LS > OK

The **/STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window and if all is OK, select **File> Close.**

In the **Solve Current Load Step** window, select **OK**, and when the solution is complete, **close the** "**Solution is Done!**" window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. Main Menu > General Posrproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Temperature > OK

This result is shown in Figure 7.1(c).

11. Main Menu > General Postproc > List Results > Nodal Solu > Select Temperature > OK

EXAMPLE 7.2

Heat is generated in a large plate with k = 0.75 W/m°C at the rate 6000 W/m^3. The plate is 40 cm thick. The outside surfaces of the plate are exposed to fluid at 35°C with a convective heat transfer coefficient of 15W/m^2°C. Determine temperature distribution in wall. The two element model to be used for solution.

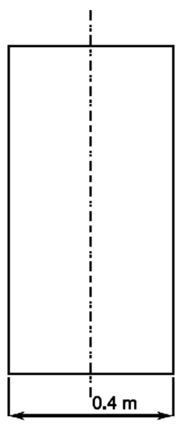


FIGURE 7.14 Example 7.2.

Solution

(I) Analytical method.

Governing equation is,

$$\frac{d^2T}{dx^2} = -\frac{q}{k}$$

$$\frac{dT}{dx} = -\frac{q \times x}{k} + c_1$$

$$\text{At } x = 0, \quad \frac{dT}{dx} = 0 \Rightarrow c_1 = 0$$

$$\frac{dT}{dx} = \frac{-600 \times x}{0.75} + 0 = -8000x \tag{7.4}$$

$$T = -\frac{q}{k}\frac{x^2}{2} + c_2 = -4000x^2 + c_2 . \tag{7.5}$$

Boundary conditions are,

$$\begin{aligned}
&\text{At } x = L, \\
&-k\frac{dT}{dx} = h(T_3 - T_\infty) \\
&-k(8000 \times x) = h(T_3 - T_\infty) \\
&-0.75(8000 \times 0.2) = 15(T_3 - 35) \Rightarrow T_3 = 115\,°\text{C.}
\end{aligned}$$

We know at $x = 0.2,\ T_3 = 115\,°C$.

Substituting this in equation (7.5),

$$115 = -4000(0.2)^2 + c_2 \Rightarrow c_2 = 275 .$$

Substituting c_2 in equation (7.5),

$$T = -4000 \times x^2 + 275$$

$$T_2 = T\big|_{x=0.1} = -4000(0.1)^2 + 275 = 235\,°\text{C}$$

$$T_1 = T\big|_{x=0} = 275\,°\text{C}.$$

(II) FEM by hand calculations [Refer to Figure 7.14(a)].

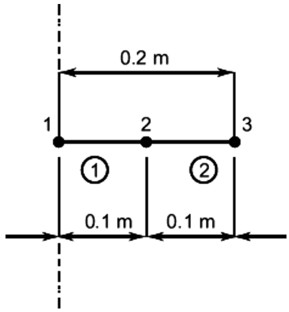


FIGURE 7.14(a) Symmetric finite element model for Example 7.2.

Given: $t = 40$ cm $= 0.4$ m, $T_\infty = 35\,°\text{C}$, $h = 15$ W/m^2 °C, $k = 0.75$ W/m°C

Governing equation is,

$$\frac{k}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = q\begin{Bmatrix} \dfrac{L}{2} \\ \dfrac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}.$$

For element 1,

$$\frac{k_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = q\begin{Bmatrix} \dfrac{L}{2} \\ \dfrac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}$$

$$\frac{0.75}{0.10}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}$$

$$7.5\begin{array}{cc} \begin{array}{cc} 1 & \;\;2 \end{array} & \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & \begin{array}{c} 1 \\ 2 \end{array} \end{array}\begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ +Q_2 \end{Bmatrix}.$$

For element 2,

$$\frac{k_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} = q\begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} -Q_2 \\ +Q_3 \end{Bmatrix}$$

$$\frac{0.75}{0.10}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} -Q_2 \\ +Q_3 \end{Bmatrix}$$

$$7.5\begin{array}{c} \begin{array}{cc} 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}\begin{array}{c} \\ 2 \\ 3 \end{array}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} -Q_2 \\ +Q_3 \end{Bmatrix}.$$

$$\text{Assembling} \Rightarrow 7.5\begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array}\begin{array}{c} \\ 1 \\ 2 \\ 3 \end{array}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.05+0.05 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} -Q_1 \\ 0 \\ +Q_3 \end{Bmatrix}.$$

Boundary conditions are, $Q_1 = 0$ and $Q_3 = -h(T_3 - T_\infty) \Rightarrow Q_3 = -15(T_3 - 35) = -15T_3 + 525$

$$7.5\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = 6000\begin{Bmatrix} 0.05 \\ 0.1 \\ 0.05 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ -15T_3 + 525 \end{Bmatrix}$$

$$\begin{bmatrix} 7.5 & -7.5 & 0 \\ -7.5 & 15 & -7.5 \\ 0 & -7.5 & 7.5 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 300+0 \\ 600+0 \\ 300-15T_3+525 \end{Bmatrix}.$$

Now,

$$\begin{bmatrix} 7.5 & -7.5 & 0 \\ -7.5 & 15 & -7.5 \\ 0 & -7.5 & 7.5+15 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 300 \\ 600 \\ 825 \end{Bmatrix}.$$

By solving the above matrix and simultaneous equations, we have temperature distribution as,

$$\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 275 \\ 235 \\ 115 \end{Bmatrix}.$$

Therefore,

$$\begin{aligned} T_1 &= 275\,°\mathrm{C} \\ T_2 &= 235\,°\mathrm{C} \\ T_3 &= 115\,°\mathrm{C}. \end{aligned}$$

(**III**) **Software results.**

Due to symmetry of the geometry, only half of the finite element model is created for software analysis.

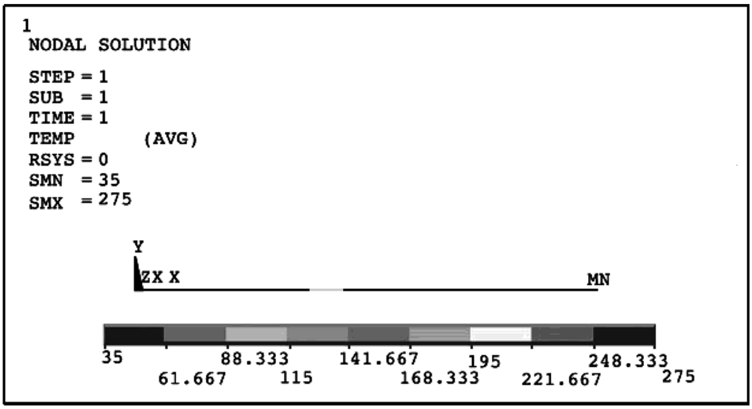


FIGURE 7.14(b) Temperature distribution in a large plate (refer to Appendix D for color figures).

Temperature values

NODE	TEMP
1	275.00
2	235.00
3	115.00
4	35.000

ANSWERS FOR EXAMPLE 7.2

Parameter	Analytical method	FEM-hand calculation	Software results
Temperature			
At node 1	275°C	275°C	275°C
At node 2	235°C	235°C	235°C
At node 3	115°C	115°C	115°C

Procedure for solving the problem using ANSYS ®11.0 academic teaching software.

FOR EXAMPLE 7.2

PREPROCESSING

1. **Main Menu > Preferences, then select Thermal > OK**

FIGURE 7.15 Selecting the preferences.

2. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Click on Link > then on 2D conduction 32 > OK > Add > Click on Link > then on 3D convection 34 > OK > Close**

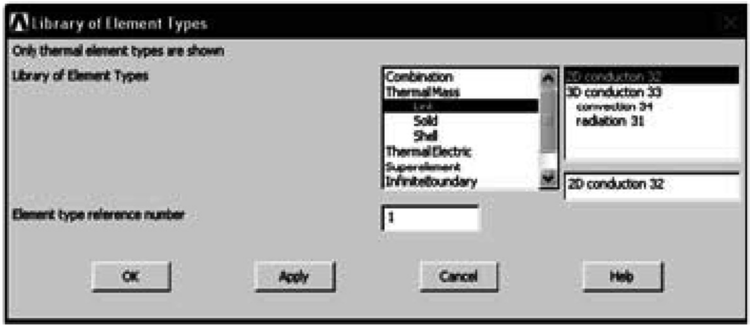


FIGURE 7.16 Selecting the element for conduction.

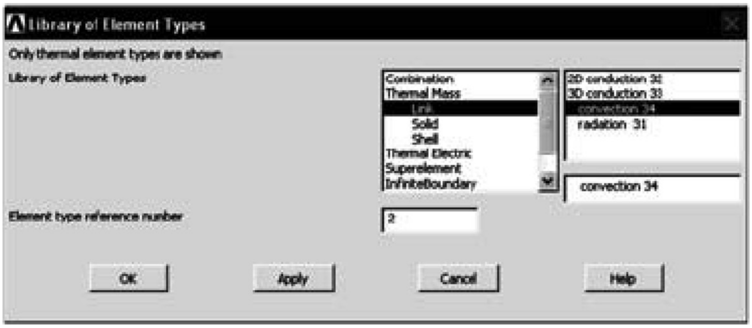


FIGURE 7.17 Selecting the element for convection.

3. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > Click on Link 32 > OK**

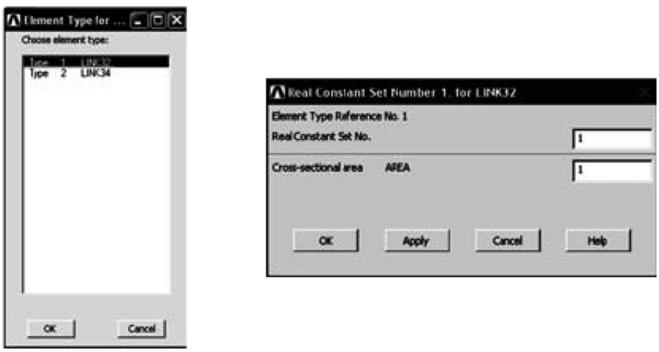


FIGURE 7.18 Enter the cross-sectional area for Link 32.

Enter cross-sectional area AREA > **Enter 1 > OK**

Add > Click on Link 34 > OK

Enter cross-sectional area AREA > **Enter 1 > OK**

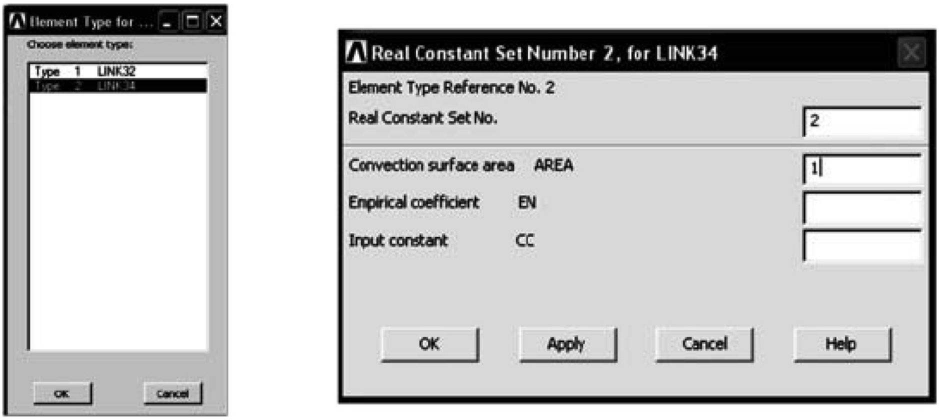


FIGURE 7.19 Enter the cross-sectional area for Link 34.

Enter the material properties

4. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1,

   click Thermal > Conductivity > Isotropic

   Enter **KXX = 0.75 > OK**

   Then in the material model window, click on **Material menu > New Model > OK**

   Material Model Number 2,

   Click **Thermal > Convection or Film Coef.**

   Enter **HF = 15 > OK**

   (**Close** the Define Material Model Behavior window.)

   Create the nodes and elements. Due to geometric symmetry, only half of the model is created.

5. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinate of node 1 > **Apply** Enter the coordinates

of node 2 > **Apply** Enter the coordinates of node 3 > **Apply** Enter the coordinates of node 4 > **OK**

Node locations		
Node number	**X coordinates**	**Y coordinates**
1	0	0
2	0.1	0
3	0.0	0
4	0.3	0

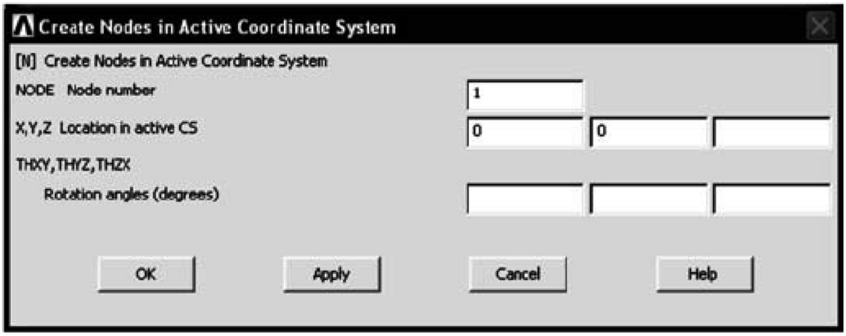


FIGURE 7.20 Enter the node coordinates.

6. **Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes > OK > Auto Numbered > Thru nodes** Pick the 1st and 2nd node > **Apply** > then Pick the 2nd and 3rd node **OK**

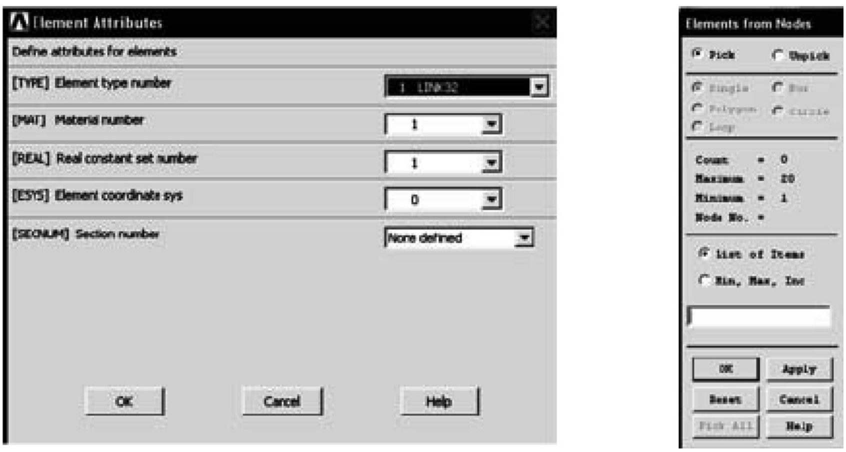


FIGURE 7.21 Assigning element attributes to elements 1 and 2 and creating elements 1 and 2.

Elem Attributes > change the element type to Link 34 > change the material number to 2 > change the Real constant set number to 2 > OK > Auto Numbered > Thru nodes Pick the 3rd and 4th node > **OK**

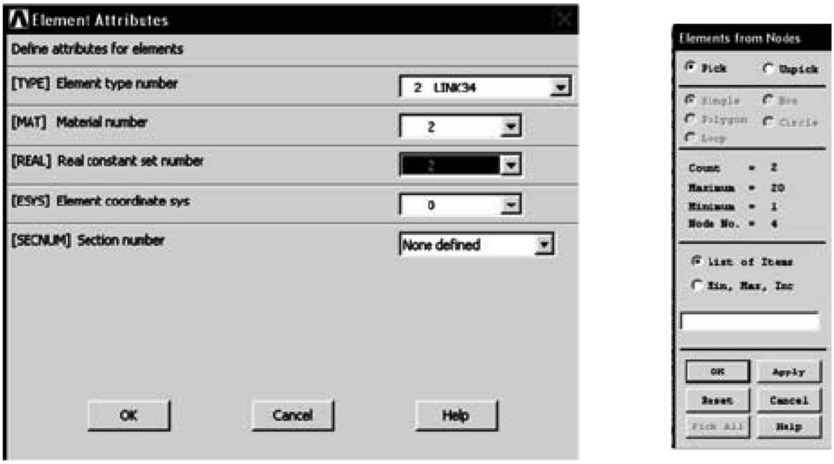


FIGURE 7.22 Assigning elements attributes to element 3 and creating element 3.

Apply the boundary conditions and temperature.

7. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes** Pick the 4th node **> Apply > Click on TEMP and Enter Value = 35 > OK**

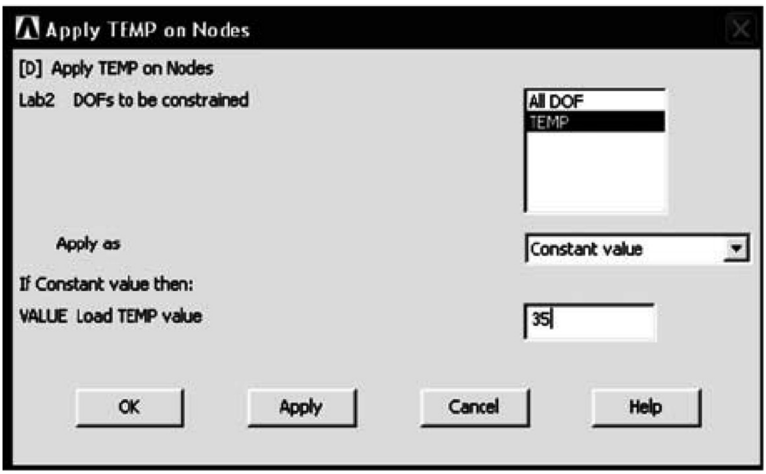


FIGURE 7.23 Applying temperature on node 4.

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Heat Generat > On Nodes** Pick the 1st, 2nd, and 3rd nodes > **Apply > Enter HGEN Value = 6000 > OK**

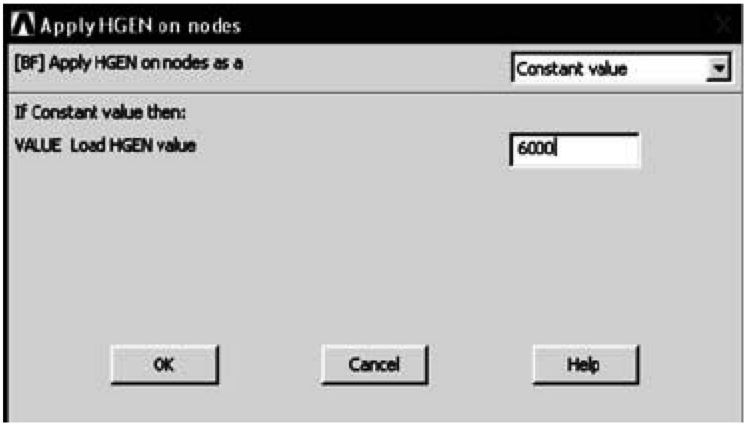


FIGURE 7.24 Assigning heat generation on nodes 1, 2, and 3.

Solution

The interactive solution proceeds.

9. **Main Menu > Solution > Solve > Current LS > OK**

The **/STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window and if all is OK, select **File > Close.**

In the **Solve Current Load Step** window, select **OK**, and when the solution is complete, **close the** "**Solution is Done!**" window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

10. **Main Menu > General Posrproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Temperature > OK**

    This result is shown in Figure 7.14(b).

11. **Main Menu > General Postproc > List Results > Nodal Solu > Select Temperature > OK**

EXAMPLE 7.3

Compute the temperature distribution in a long steel cylinder with an inner radius of 125 mm and an outer radius of 250 mm. The interior of the cylinder is kept at 300°K and heat is lost on the exterior by convection to a fluid whose temperature is 280°K. The convection heat transfer coefficient h is 0.994 W/m^2°K and the thermal conductivity for steel k is 0.031 W/m°K.

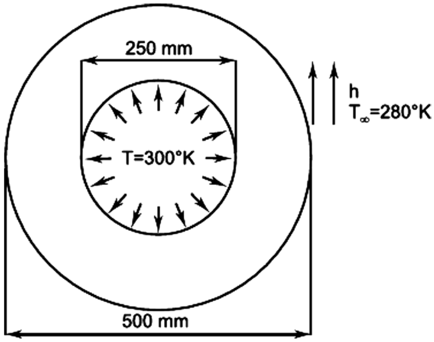


FIGURE 7.25 Example 7.3.

Solution

(I) Analytical method.

Here the problem is solved considering heat flow in radial direction.

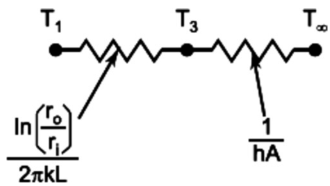


FIGURE 7.25(a) Analytical method for Example 7.3.

$r_0 = 250$ mm = 0.25 m and $r_1 = 125$ mm = 0.125 m

Assume unit length of the cylinder

$$Q = \frac{(T_1 - T_\infty)}{\left(\frac{\ln\left(\frac{r_0}{r_1}\right)}{2\pi kL} + \frac{1}{hA}\right)} = \frac{(300-280)}{\left(\frac{\ln\left(\frac{250}{125}\right)}{2\pi\times 0.031\times 1} + \frac{1}{0.994(2\pi\times 0.25)}\right)} = 4.762934 \text{ W}.$$

Now,

$$Q = \frac{(T_1 - T_3)}{\left(\frac{\ln\left(\frac{r_0}{r_1}\right)}{2\pi kL}\right)} \Rightarrow \frac{(300 - T_3)}{\left(\frac{\ln\left(\frac{250}{125}\right)}{2\pi\times 0.031\times 1}\right)} = 4.762934 \ \Rightarrow T_3 = 280.51^\circ\text{C}.$$

Let

$T = T_2$ at $r = 187.5$ mm, then

$$\frac{(300 - T_2)}{\left(\frac{\ln\left(\frac{250}{187.5}\right)}{2\pi\times 0.031\times 1}\right)} = 4.762934 \ \Rightarrow T_2 = 283.1^\circ\text{C}.$$

(II) FEM by hand calculations.

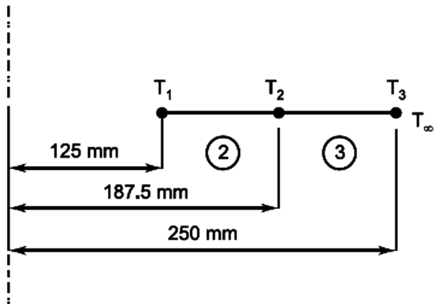


FIGURE 7.25(b) Finite element model for Example 7.3.

$r_1 = 125$ mm, $r_2 = 187.5$ mm, and $r_3 = 250$ mm

Element matrices are,

$$k_{1C} = \frac{2\pi kL}{\ln\left(\frac{r_2}{r_1}\right)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2\pi\times 0.031\times 1}{\ln\left(\frac{187.5}{125}\right)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.48 \begin{array}{c} \\ \\ \end{array}\begin{array}{cc|c} T_1 & T_2 & \\ \end{array}$$

$$0.48\begin{array}{cc} & \\ \end{array}\begin{array}{c}\begin{array}{cc} T_1 & T_2 \end{array} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\end{array}\begin{array}{c} \\ T_1 \\ T_2 \end{array}$$

$$k_{2C} = \frac{2\pi kL}{\ln\left(\dfrac{r_3}{r_2}\right)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2\pi \times 0.031 \times 1}{\ln\left(\dfrac{250}{187.5}\right)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 0.68 \begin{array}{c c} \quad T_2 & T_3 \\ \left[\begin{array}{cc} 1 & -1 \\ -1 & 1 \end{array}\right] & \begin{array}{c} T_2 \\ T_3 \end{array} \end{array}.$$

Global conduction matrix is,

$$[K_C] = \begin{bmatrix} 0.48 & -0.48 & 0 \\ -0.48 & 0.48 + 0.68 & -0.68 \\ 0 & -0.68 & 0.68 \end{bmatrix} = \begin{bmatrix} 0.48 & -0.48 & 0 \\ -0.48 & 1.16 & -0.68 \\ 0 & -0.68 & 0.68 \end{bmatrix}.$$

Global equation is,

$$\begin{bmatrix} 0.48 & -0.48 & 0 \\ -0.48 & 1.16 & -0.68 \\ 0 & -0.68 & 0.68 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix}.$$

Applying boundary conditions, $T_1 = 300°K$ and $Q_3 = -hA_0(T_3 - T_\infty)$

$$A_0 = 2\pi r_3 = 2\pi \times 0.25 = 1.57 \text{ m}^2.$$

Therefore, $Q_3 = -hA_0(T_3 - T_\infty) = -0.994 \times 1.57(T_3 - 280) = -(1.56T_3 - 437)$

$$\begin{bmatrix} 0.48 & -0.48 & 0 \\ -0.48 & 1.16 & -0.68 \\ 0 & -0.68 & 0.68 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -(1.56T_3 - 437) \end{Bmatrix}$$

$$\begin{bmatrix} \cancel{0.48} & \cancel{-0.48} & \cancel{0} \\ -0.48 & 1.16 & -0.68 \\ 0 & -0.68 & 0.68 + 1.56 \end{bmatrix}\begin{Bmatrix} \cancel{T_1} \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} \cancel{0} \\ 0 \\ 437 \end{Bmatrix}$$

$$\begin{bmatrix} 1.16 & -0.68 \\ -0.68 & 2.24 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 437 \end{Bmatrix} - \begin{Bmatrix} -0.48 \times 300 \\ 0 \end{Bmatrix} = \begin{Bmatrix} 144 \\ 437 \end{Bmatrix}.$$

Solving the above equation, we get $T_3 = 283.16°K$ and $T_2 = 290.13°K$.

(III) Software results.

Due to the symmetry of the cylinder geometry, only a quarter of the geometry is drawn for finite element analysis.

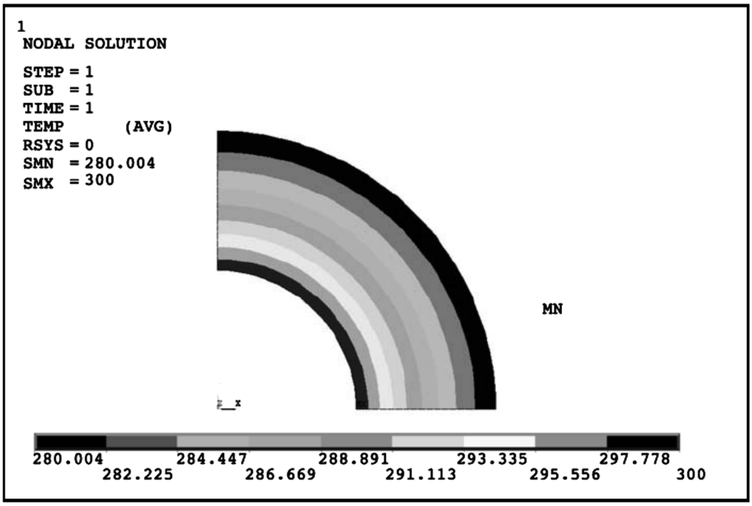


FIGURE 7.25(c) Temperature distribution in a long cylinder (refer to Appendix D for color figures).

The temperature in the interior is 300°K and on the outside wall, it is found to be 280.004°K.

ANSWERS OF EXAMPLE 7.3

Parameter	Analytical method	FEM-hand calculation	Software results
Temperature on the interior surface	300°K	300°K	300°K
Temperature at radius 187.5 mm	238.1°K	290.13°K	288.891°K
Temperature on the outside wall	280.51°K	283.16°K	280.004°K

Procedure for solving the problem using ANSYS ®11.0 academic teaching software.

FOR EXAMPLE 7.3

PREPROCESSING

1. **Main Menu > Preferences, then select Thermal > OK**

FIGURE 7.26 Selecting the preferences.

2. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Click on Solid > then on Quad 8 node 77 > OK > Close**

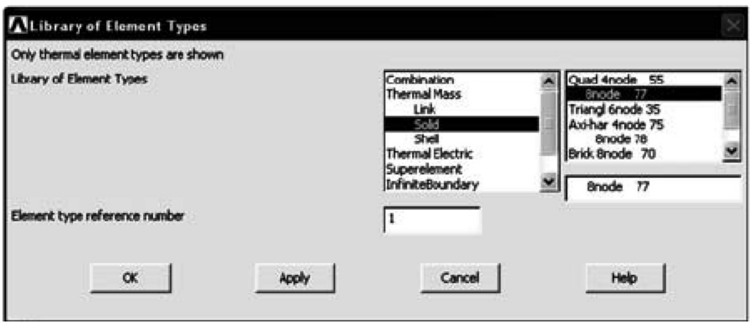


FIGURE 7.27 Selecting the element.

3. **PLANE 77 does not require any real constant**

   Enter the material properties.

4. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1,

Click **Thermal > Conductivity > Isotropic**

Enter **KXX = 0.031 > OK**

(**Close** the Define Material Model Behavior window.)

Recognize symmetry of the problem, and a quadrant of a section through the cylinder is created.

5. **Main Menu > Preprocessor > Modeling > Create > Areas > Circles > Partial Annulus**

Enter the data as shown below.

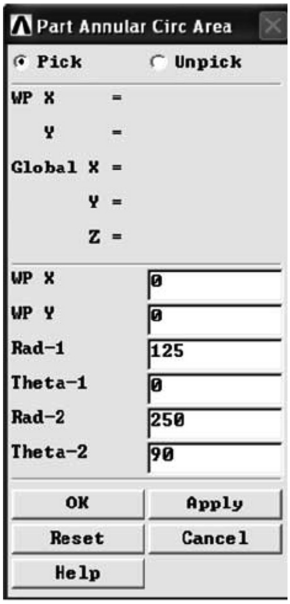


FIGURE 7.28 Create partial annular area.

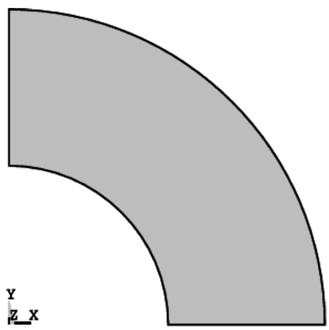


FIGURE 7.29 Quadrant of a cylinder.

6. **Main Menu > Preprocessor > Meshing > Mesh Tool**

   The **Mesh Tool** dialog box appears. In that dialog box, click on the **Smart Size** and move the slider available below the **Smart Size to 2** (i.e., toward **Fine** side). Then close the Mesh Tool box.

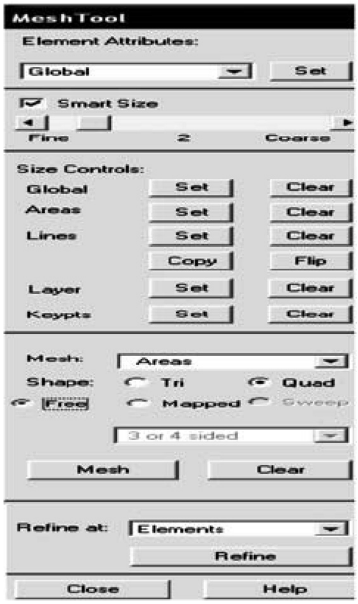


FIGURE 7.30 Mesh tool box.

7. **Main Menu > Preprocessor > Meshing > Mesh > Areas > Free.** Pick the quadrant **> OK**

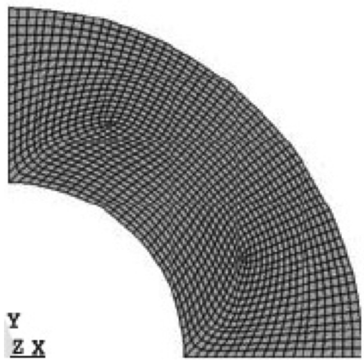


FIGURE 7.31 Quad element mesh.

8. Main Menu > Preprocessor > Loads > Define loads > Apply > Thermal > Temperatures > On Lines

Select the line on the interior and set the temperature to 300.

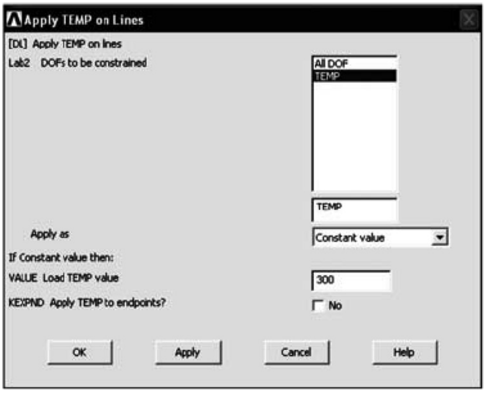


FIGURE 7.32 Setting the temperature on the interior of the cylinder.

9. Main Menu > Preprocessor > Loads > Apply > Convection > On Lines

Select the lines defining the outer surface and set the convection coefficient to 0.994 and the fluid temp to 280.

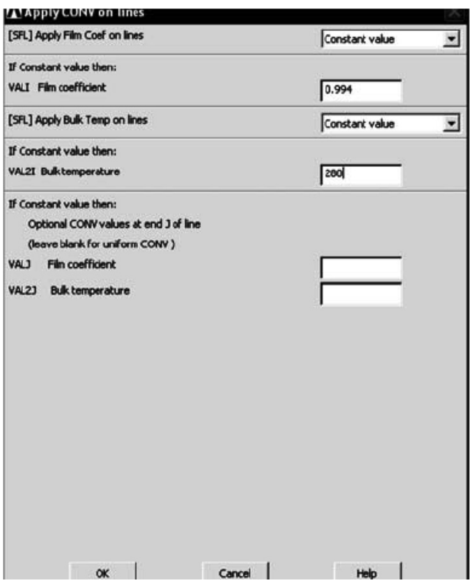


FIGURE 7.33 Setting the convection coefficient on outer surface.

10. Main Menu > Preprocessor > Loads > Apply > Heat Flux > On Lines

To account for symmetry, select the vertical and horizontal lines of symmetry and set the heat flux to zero.

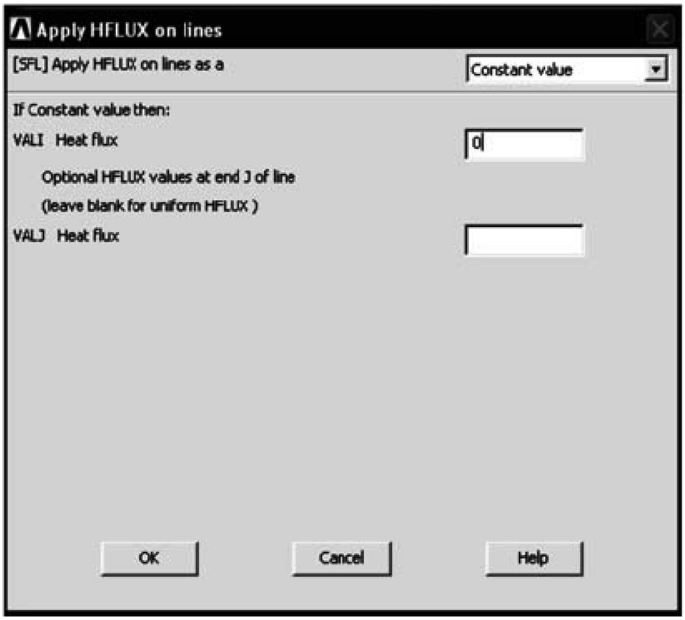


FIGURE 7.34 Setting the heat flux.

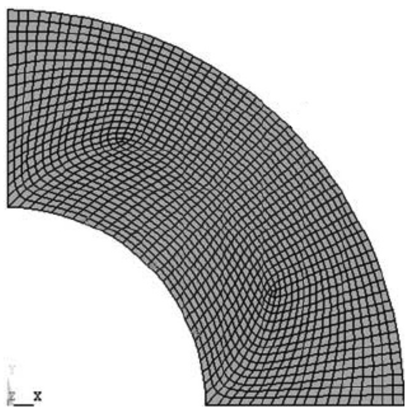


FIGURE 7.35 Model with boundary conditions.

Solution

The interactive solution proceeds.

11. Main Menu > Solution > Solve > Current LS > OK

The **/STATUS Command** window displays the problem parameters and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window and if all is OK, select **File > Close.**

In the **Solve Current Load Step** window, select **OK,** and when the solution is complete, **close the** "**Solution is Done!**" window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

12. Main Menu > General Posrproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Temperature > OK

This result is shown in Figure 7.25(b).

13. Main Menu > General Postproc > List Results > Nodal Solu > Select Temperature > OK

7.4 TWO-DIMENSIONAL PROBLEM WITH CONDUCTION AND WITH CONVECTION BOUNDARY CONDITIONS

EXAMPLE 7.4

A body having rectangular cross-section is subjected to boundary conditions as shown in Figure 7.36. The thermal conductivity of the body is 1.5 W/m°. On one side of the body, it is insulated and on the other side, convection takes place with h = 50 W/m^2°C and T_∞ = 35 °C. The top and bottom sides are maintained at a uniform temperature of 180°C. Determine the temperature distribution in the body.

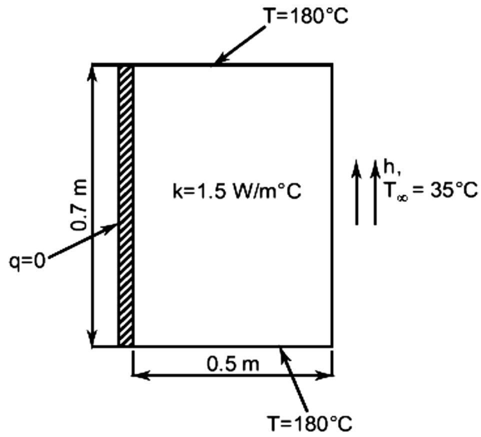


FIGURE 7.36 Example 7.4.

Solution

(I) Software results.

The temperature at the top and bottom edges is found to be 180°C and at the right edge the temperature is found to be 46.802°C.

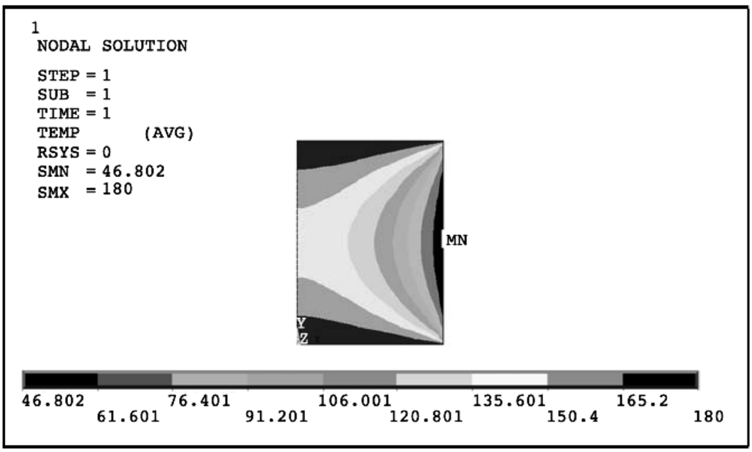


FIGURE 7.36(a) Temperature distribution in a body of rectangular cross-section (refer to Appendix D for color figures).

EXERCISES

1. Define conduction and convection.
2. Write the formulas for the rate of heat flow in x-direction by conduction and the rate of heat flow by convection.
3. Determine the temperature distribution for the two-dimensional body shown in Figure 7.37, subjected to boundary conditions as shown in the figure. The top and bottom edges are insulated. The left side of the body is maintained at a temperature of 45°C. On the right side, the convection process takes place with heat transfer coefficient h = 100 W/m^2°C and T_∞ = 20°C. The thermal conductivity of the body is k = 45 W/m°C.

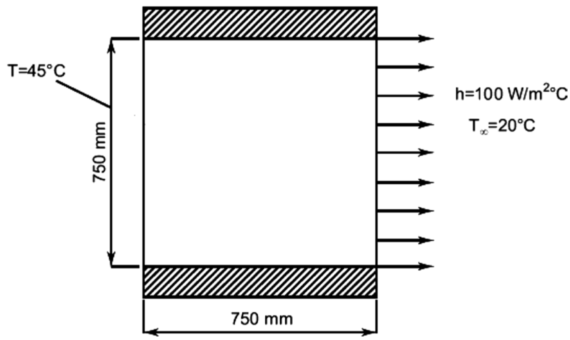


FIGURE 7.37 Exercise 3.

4. Determine the temperature distribution for the two-dimensional body shown in Figure 7.38. The temperature of 200°C is maintained at the top and bottom edges. The left and right edges are insulated. Heat is generated at the rate of q = 2000 W/m^3 in a body as shown in the figure. Let k= 35 W/ m°C.

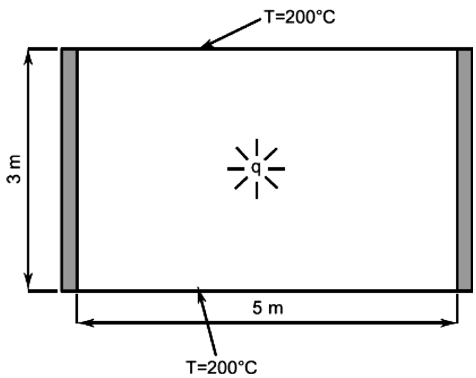


FIGURE 7.38 Exercise 4.

5. Determine the temperature distribution for the two-dimensional body shown in Figure 7.39, subjected to boundary conditions as shown in the figure. The top and bottom edges are insulated. The left side of the body is maintained at a temperature of 50°C. On the right side, the convection process takes place with heat transfer coefficient h = 150 W/m^2°C and T_∞ = 25°C. The thermal conductivity of the body is k = 50 W/m°C.

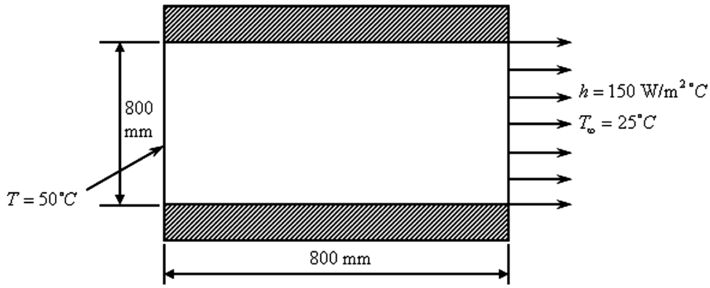


FIGURE 7.39 Exercise 5.

6. Determine the temperature distribution for the two-dimensional body shown in Figure 7.40. The temperature of 200°C is maintained at the top and bottom edges. The left and right edges are insulated. Heat is generated at the rate of q = 2100 W/m^3 in a body as shown in figure. Let k = 45W/ m°C.

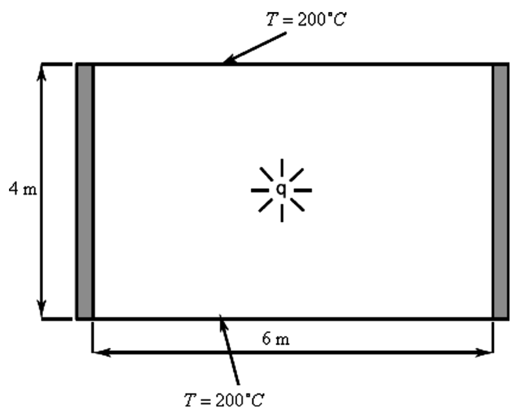


FIGURE 7.40 Exercise 6.

7. Determine the load matrix and the global load matrix for Figure 7.41. The top and bottom edges are insulated.

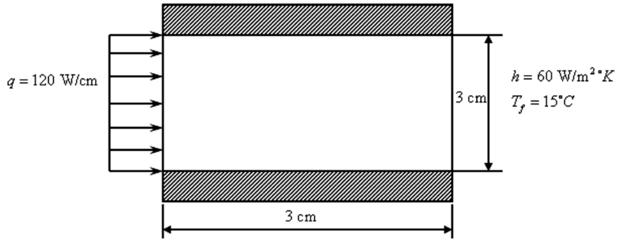


FIGURE 7.41 Exercise 7.

8. Consider the rectangular plate shown in Figure 7.42. The outer temperature is T_0 = 30°C. Convection heat transfer takes place on the inner surface of the wall with T_∞ = 80°C and h = 50 W/m^2°K. Determine the temperature distribution in the wall. Take the thermal conductivity value k = 160 W/m°K.

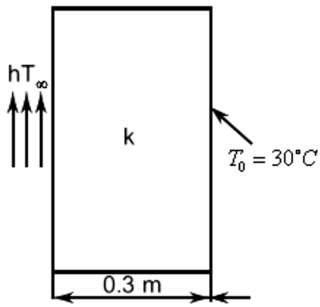


FIGURE 7.42 Exercise 8.

9. Consider a composite wall consisting of two materials shown in Figure 7.43. The outer temperature is T_0 = 30°C. Convection heat transfer takes place on the inner surface of the wall with T_∞ = 80°C and h = 50 W/m^2°K. Determine the temperature distribution in the wall. Take the thermal conductivity value k_1 = 40 W/m°C and k_2 = 60 W/m°C.

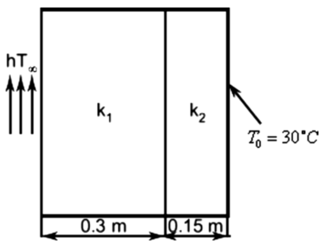


FIGURE 7.43 Exercise 9.

10. For the one-dimensional (1D) bar fixed at both ends and subjected to a uniform temperature rise T = 40°C, as shown in Figure 7.44, determine the reactions at the fixed ends and the axial stress in the bar. Let E = 250 GPa, A = 28 cm^2, L= 1.4 m, and α = 1.30 × 10^{-6} (mm/mm)/°C.

FIGURE 7.44 Bar subjected to a uniform temperature rise.

REFERENCES

1. L. Segrlind, "Applied Finite Element Analysis," Second Edition, Jon Wiley and Sons, New York, 1984.
2. S. Moaveni, "Finite Element Analysis: Theory and Application with ANSYS," Third Edition, Prentice Hall, New Jersey, 2008.
3. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, Cengage Learning, Boston, Massachusetts, 2012.
4. F. P. Incropera and D. P. DeWitt, "Fundamentals of Heat and Mass Transfer," Fourth Edition, Wiley, New Jersey, 1996.
5. F. P. Incropera, D. P. DeWitt, T. L. Bergman, and A. S. Lavine, "Introduction to Heat Transfer," Fifth Edition, Wiley, New Jersey, 2007.
6. S. S. Rao, "The Finite Element Method in Engineering," Fifth Edition, Butterworth-Heinemann, Oxford, 2011.

CHAPTER 8

FLUID FLOW ANALYSIS

8.1 INTRODUCTION

A substance (liquid or gas) that will deform continuously by applied surface (shearing) stresses is called a *fluid*. The magnitude of shear stress depends on the magnitude of angular deformation. Indeed, different fluids have different relations between stress and the rate of deformation. Also, fluids are classified as *compressible* (usually gas) and *incompressible* (usually liquid).

The terms of velocities and accelerations of fluid particles at different times and different points throughout the fluid-filled space are used to describe the flow field. The fluid is called ideal when the fluid has zero viscosity and is incompressible. A fluid is said to be *incompressible* if the volume change is zero (i.e., ρ = constant)

$$\nabla \cdot \mathbf{v} = 0 ,$$

where $\mathbf{v}$ is the velocity vector.

Depending on the importance of the viscosity of the fluid in the analysis, a flow can be termed as *inviscid* or *viscous*. An inviscid flow is a frictionless flow characterized by zero viscosity, that is, there is no real fluid. In other words, a fluid is called *inviscid* if the viscosity is zero (i.e., $\mu = 0$).

A viscous flow is a flow in which the fluid is assumed to have nonzero viscosity. An *irrotational flow* is a flow in which the particles of the fluid are not rotating, and the rotation is zero. In other words, an *irrotational flow* is a flow with negligible angular velocity, if

$$\nabla \times \mathbf{v} = 0 .$$

On the other hand, a potential *flow* is an irrotational flow of an ideal fluid (i.e., ρ = constant and $\mu = 0$).

A line that connects a series of points in space at a given instant where all particles falling on the line at that instant have velocities whose vectors are tangent to the line is called a *streamline*.

The flow is *steady*, which means that the flow pattern or streamlines do not change over time and the streamlines represent the trajectory of the fluid's particles. But, when the flow is *ideal* that means that the fluid has zero velocity.

This chapter covers the finite element solution of ideal or potential flow (inviscid, incompressible flow) problems. Typical examples of potential flow are flow over a cylinder, flow around an airfoil, and flow out of an orifice.

The two-dimensional potential flow (irrotational flow) problems can be formulated in terms of a velocity potential function (φ) or a stream function (Ψ). The selection between velocity and stream function formulations in the finite element analysis depends on the ease of applying boundary conditions. If the geometry is simple, any one function can be used.

Fluid elements (e.g., FLUID141) are used in the steady-state or transient analysis of fluid systems. Pressure, velocity, and temperature distributions can be obtained using these elements.

Two-dimensional fluid elements are defined using 3 (triangular element) or 4 (quadrilateral element) nodes added by isotropic properties. Inputs to these elements are nodal coordinates, real constants, material properties, surface and body loads, etc. Outputs of interest are nodal values of pressure and velocity.

8.2 PROCEDURE OF FINITE ELEMENT ANALYSIS (RELATED TO FLUID FLOW PROBLEMS)

Step 1. Select element type—the basic three-node triangular element can be used.

Step 2. Choose a potential function.

Step 3. Define the gradient/potential and velocity/gradient relationships.

Step 4. Derive the element stiffness matrix and equations.

Step 5. Assemble the element equations to obtain the global equations and introduce boundary conditions.

Step 6. Solve for the nodal potentials.

Step 7. Solve for the element velocities and volumetric rates.

The finite element solution using software for potential flow problems is illustrated below. Only potential function formulation is considered. Two cases are considered in this chapter.

8.3 POTENTIAL FLOW OVER A CYLINDER

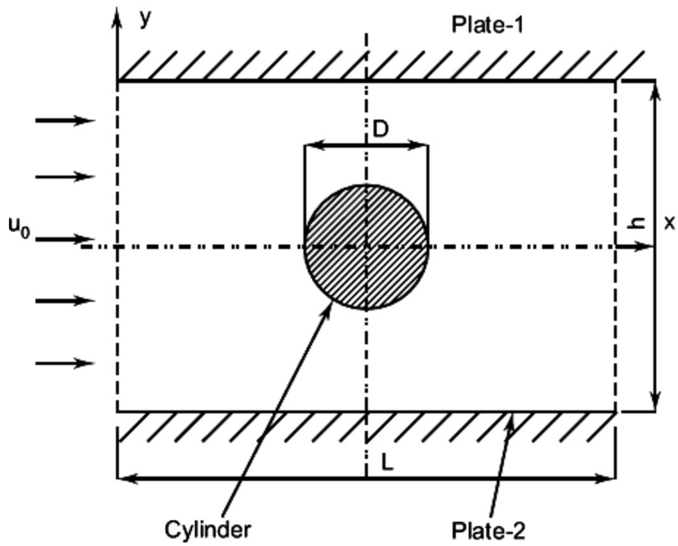


FIGURE 8.1 Potential flow over a cylinder.

The previous figure depicts the steady-state irrotational flow of an ideal fluid over a cylinder, confined between two parallel plates. We assume that, at the inlet, velocity is uniform, say u_0. Here, we have to determine the flow velocities near the cylinder.

Flow past a fixed circular cylinder can be obtained by combining uniform flow with a doublet.

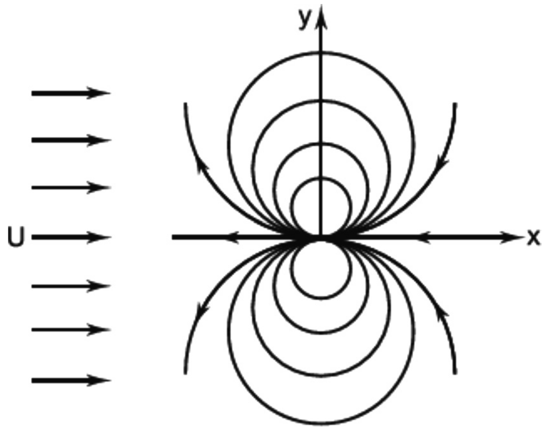


FIGURE 8.2 Superposition of a uniform flow and a doublet.

The superimposed stream function and velocity potential are given by,

$$\Psi = \Psi_{\text{uniform flow}} + \Psi_{\text{doublet}} = U \times r \times \sin\theta - K \times \frac{\sin\theta}{r} \tag{8.1}$$

and

$$\Phi = \Phi_{\text{uniform flow}} + \Phi_{\text{doublet}} = U \times r \times \cos\theta - K \times \frac{\cos\theta}{r}\text{, respectively,} \tag{8.2}$$

where U is velocity.

Because the streamline that passes through the stagnation point has a value of zero, the stream function on the surface of the cylinder of radius a is then given by,

$$\Psi = U \times a \times \sin\theta - K \times \frac{\sin\theta}{a} = 0 \tag{8.3}$$

which gives the strength of the doublet as,

$$K = U \times a^2. \tag{8.4}$$

The stream function and velocity potential for flow past a fixed circular cylinder becomes

$$\Psi = U \times r\left(1 - \left(\frac{a}{r}\right)^2\right)\sin\theta \tag{8.5}$$

and

$$\Phi = U \times r\left(1-\left(\frac{a}{r}\right)^2\right)\cos\theta \text{ , respectively.} \tag{8.6}$$

The plot of the streamlines is shown in Figure 8.3.

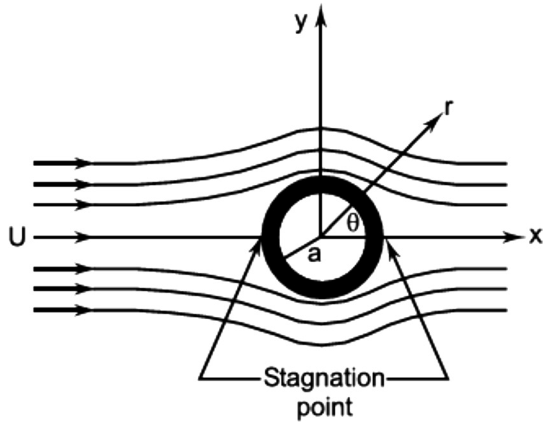


FIGURE 8.3 Streamlines for flow past a fixed cylinder.

The velocity components can be determined by,

$$v_r = \frac{1}{r}\frac{\partial\Psi}{\partial\theta} = U\left(1-\left(\frac{a}{r}\right)^2\right)\cos\theta \tag{8.7}$$

$$v_\theta = \frac{\partial\Psi}{\partial r} = -U\left(1-\left(\frac{a}{r}\right)^2\right)\sin\theta \text{ .} \tag{8.8}$$

Along the cylinder ($r = a$), the velocity components reduce to $v_r = 0$ and $v_\theta = -2U\sin\theta$.

The radial velocity component is always zero along the cylinder, while the tangential velocity component varies from 0 at the stagnation point ($\theta = \pi$) to a maximum velocity of $2U$ at the top and bottom of the cylinder ($\theta = \frac{\pi}{2}$ or $\theta = -\frac{\pi}{2}$).

8.4 POTENTIAL FLOW AROUND AN AIRFOIL

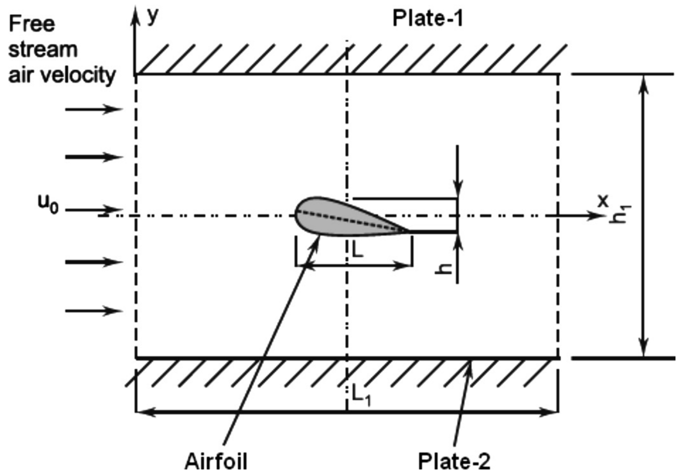


FIGURE 8.4 Potential flow around an airfoil.

The x- and y-components of fluid's velocity, respectively, can be expressed in a stream function $\Psi(x,y)$ as

$$v_x = \frac{\partial \Psi}{\partial y} \text{ and } v_y = -\frac{\partial \Psi}{\partial x}. \tag{8.9}$$

The x- and y-components of fluid's velocity with irrotational flows, respectively, can be expressed in a potential function $\phi(x,y)$ as

$$v_x = \frac{\partial \phi}{\partial x} \text{ and } v_y = \frac{\partial \phi}{\partial y}. \tag{8.10}$$

EXAMPLE 8.1

Flow over a circular cylinder between two parallel plates is shown in Figure 8.5. Assume unit thickness. Find the velocity distribution for the flow over a circular cylinder. Consider the flow of a liquid over a circular cylinder. Take liquid as water. Water density = 1000 kg/m^3 and viscosity = 0.001 N-s/m^2.

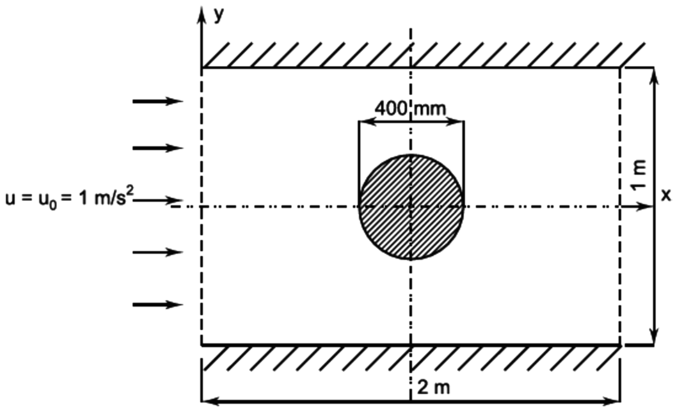


FIGURE 8.5 Flow over a cylinder.

Solution

(I) Software results.

Procedure for solving the problem using ANSYS® 11.0 academic teaching software.

PREPROCESSING

1. **Main Menu > Preferences, then select FLOTRAN CFD > OK**

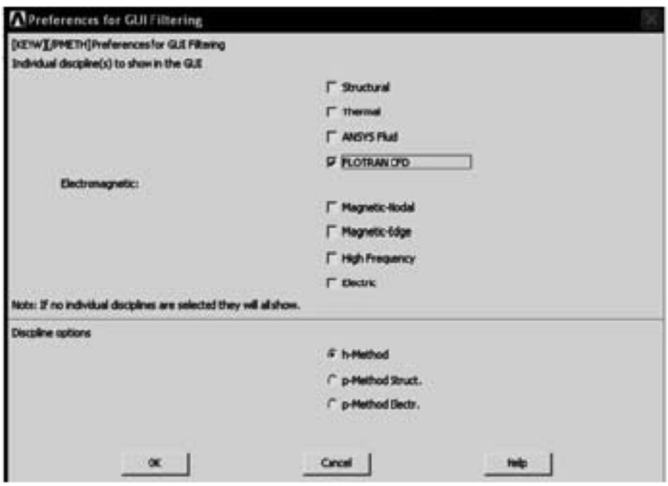


FIGURE 8.6 Selecting the preferences.

2. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > FLOTRAN CFD > 2D FLOTRAN 141 > OK**

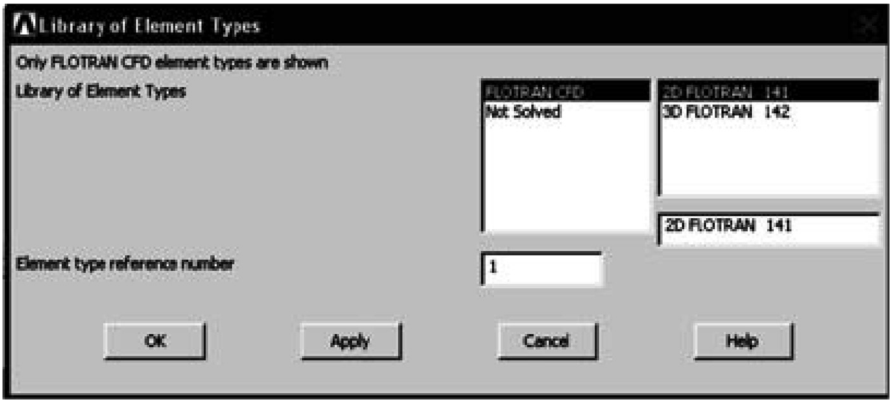


FIGURE 8.7 Element selection.

3. **Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners** Enter (lower left corner) **WP X = 0.0, WP Y = 0.0 and Width = 2, Height = 1 > OK**

4. **Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle.** Enter **WP X = 1, WP Y = 0.5 and Radius = 200e-3 > OK**

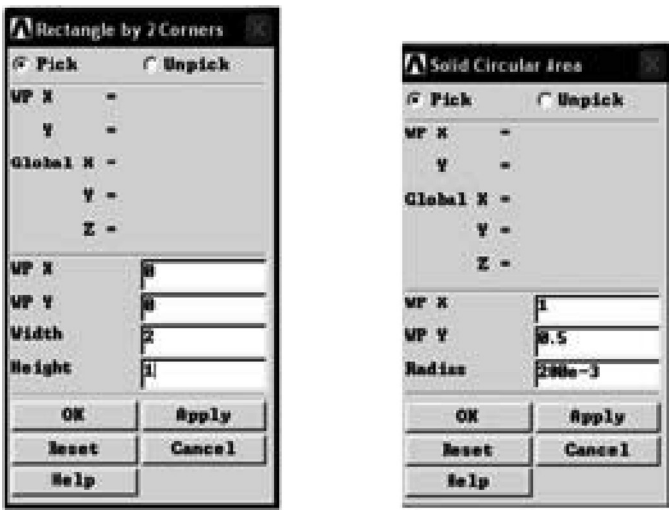


FIGURE 8.8 Create areas.

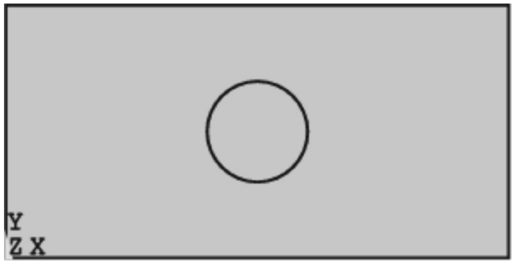


FIGURE 8.9 Rectangle and circle.

Now **subtract** the circle from the rectangle. (Read the messages in the window at the bottom of the screen as necessary.)

5. **Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas >** Pick the rectangle **> OK,** then pick the circle **> OK**

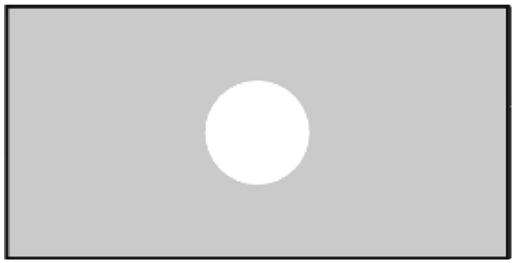


FIGURE 8.10 Geometry for the flow over a cylinder.

Create a mesh of quadrilateral elements over the area.

6. **Main Menu > Preprocessor > Meshing > Mesh Tool**
   The **Mesh Tool** dialog box appears. Close the **Mesh Tool** box.
7. **Main Menu > Preprocessor > Meshing > Mesh > Areas > Free** Pick the area **> OK**

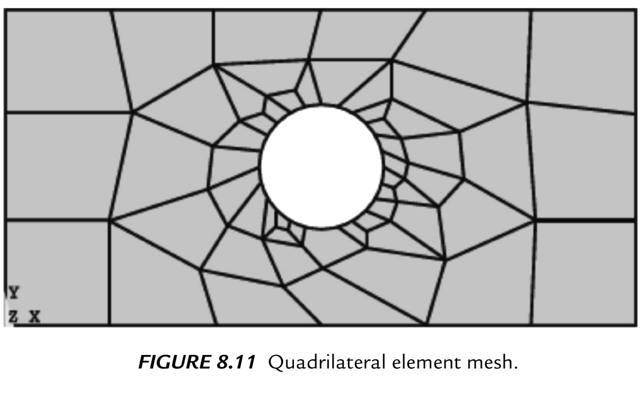


FIGURE 8.11 Quadrilateral element mesh.

Apply the velocity boundary conditions and pressure.

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Fluid/CFD > Velocity > On Lines** Pick the left edge of the plate **> OK > Enter VX = 1 > OK**
   (VX = 1 means an initial velocity of 1 m/s^2)
9. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Fluid/CFD > Velocity > On Lines** Pick the edges around the cylinder **> OK > Enter VX = 0 and VY = 0 > OK**
10. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Fluid/CFD > Pressure DOF > On Lines** Pick the top, bottom, and right edges of the plate **> OK > OK**
    Once all the boundary conditions are applied, the cylinder with the plate will look like Figure 8.12.

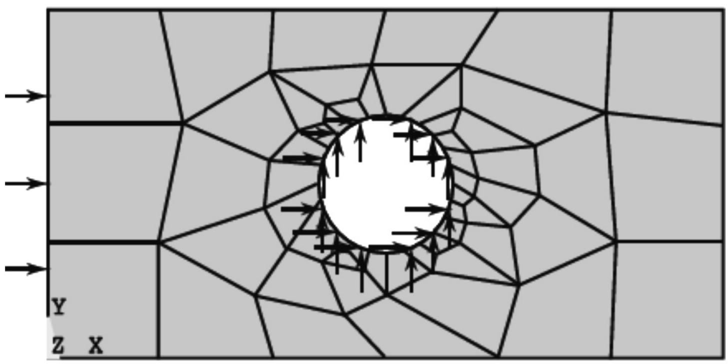


FIGURE 8.12 Model with boundary conditions.

The model-building step is now complete, and we can proceed to the solution. First, save the model.

Solution

The interactive solution proceeds.

11. **Main Menu > Solution > FLOTRAN Set Up > Fluid Properties** > A dialog in that select against density as liquid and against viscosity as liquid > **OK**
    Then another dialog box appears and, in that, enter the value of density = 1000 value = 0.001 > **OK**

12. **Main Menu > Solution > FLOTRAN Set Up > Execution Ctrl** > a dialog in that Enter in the first row “Global iterations EXEC” = 200

13. **Main Menu > Solution > Run FLOTRAN**
When the solution is complete, close the “**Solution is Done!**” window.

POSTPROCESSING

We can now plot the results of this analysis and also list the computed values.

14. **Main Menu > General Postproc > Read Results > Last Set**

15. **General Postproc > Plot Results > Contour Plot > Nodal Solu**
Select **DOF Solution** and **Fluid Velocity** and click **OK**
This is what the solution should look like:

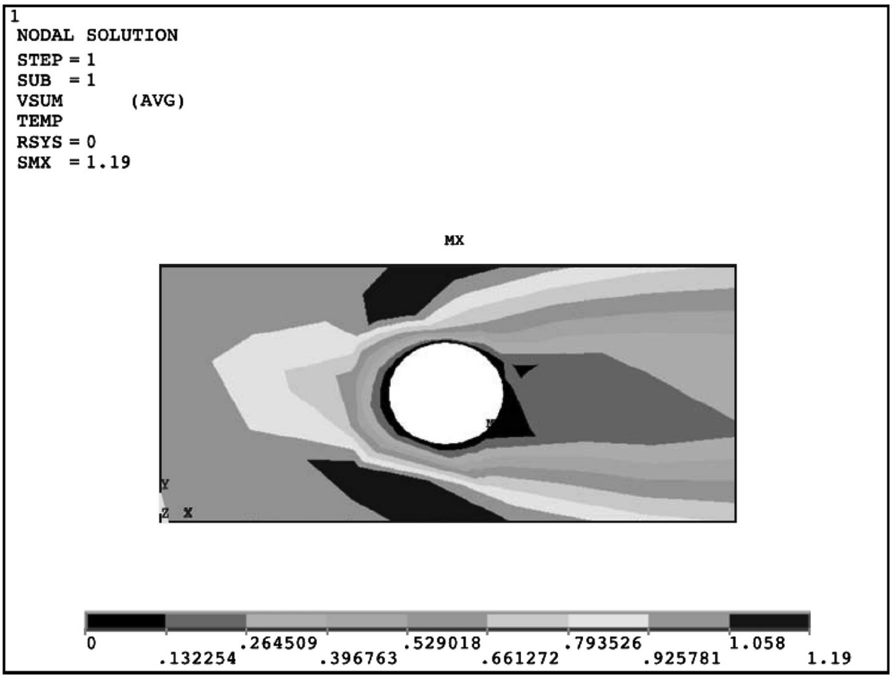


FIGURE 8.13 Velocity distribution over a cylinder (refer to Appendix D for color figures).

16. Next, go to **Main Menu > General Postproc > Plot Results > Vector Plot > Predefined.** One window will appear, then click **OK**

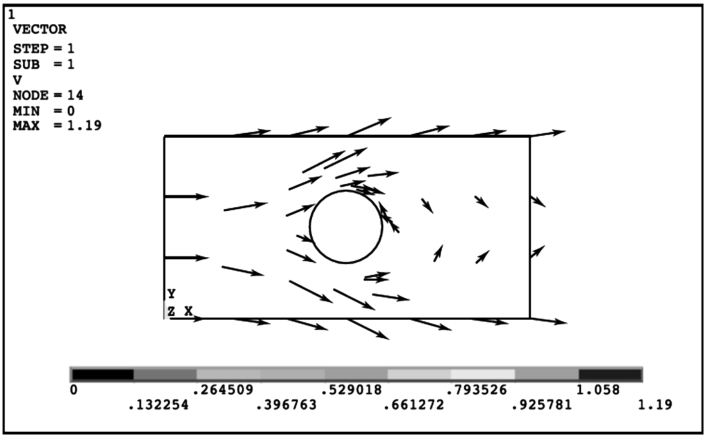


FIGURE 8.14 Vector plot of the fluid velocity (refer to Appendix D for color figures).

17. General Postproc > Plot Results > Contour Plot > Nodal Solu
Select **DOF Solution** and **Pressure** and Click **OK**

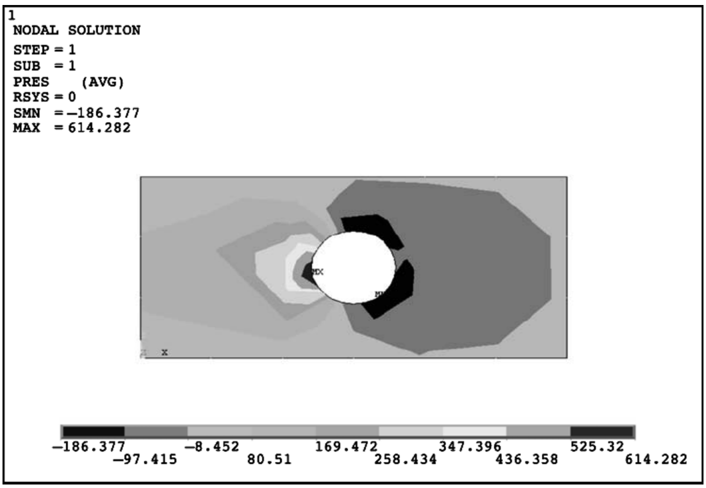


FIGURE 8.15 Pressure distribution over a cylinder (refer to Appendix D for color figures).

EXERCISES

1. Define fluid, inviscid flow, viscous flow, and irrotational flow.
2. What are the two fluid classifications?
3. Define streamline in a graphic of fluid motion?
4. What do we mean when we say the flow is steady and ideal?

5. Define irrotational *flow and potential flow*?

6. Compute and plot velocity distribution over the airfoil, as shown in Figure 8.16. Assume unit thickness. Take density of air = 1.23 kg/m^3 and viscosity = 1.79×10^{-5} N-s/m^2.

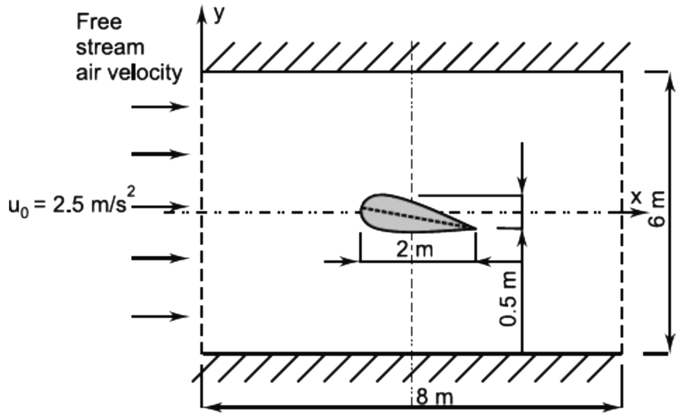


FIGURE 8.16 Flow over an airfoil.

7. Flow over a circular cylinder between two parallel plates is as shown in Figure 8.17. Assume unit thickness. Find the velocity distribution for the flow over a circular cylinder. Consider the flow of a liquid over a circular cylinder. Take liquid as water. Water density = 1000 kg/m^3 and viscosity = 0.001 N-s/m^2, $u = u_0 = 2$ m/s^2, h = 2 m, and L = 4 m.

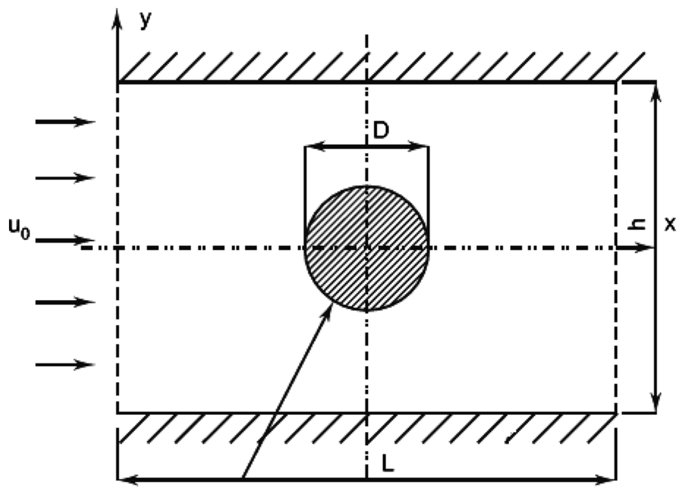


FIGURE 8.17 Flow over a circular cylinder between two parallel plates.

8. Compute and plot velocity distribution over the airfoil, as shown in Figure 8.18. Assume unit thickness. Take density of air = 1.23 kg/m^3 and viscosity = 1.79×10^{-5} N-s/m^2, L = 4 m, $L_1 = 20$ m, $h_1 = 18$ m, and $u = u_0 = 3$ m/s^2.

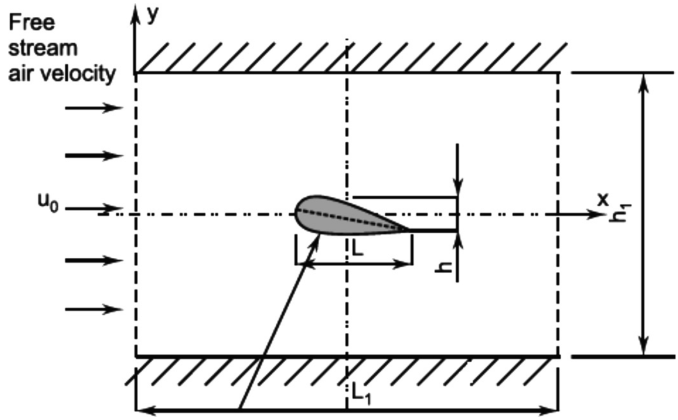


FIGURE 8.18 Flow over an airfoil.

9. Flow over an elliptical cylinder between two parallel plates is shown in Figure 8.19. Assume unit thickness. Find the velocity distribution for the flow over a circular cylinder. Consider the flow of a liquid over a circular cylinder. Take liquid as water. Water density = 1000 kg/m^3 and viscosity = 0.001 N-s/m^2, $u = u_0 = 1$ m/s^2, $D = 2$ m, $b = 1$ m, h = 4 m, and L = 8 m.

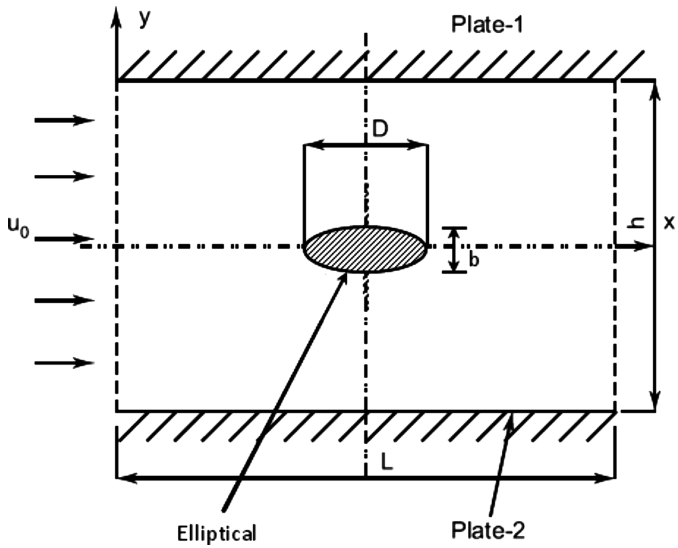


FIGURE 8.19 Flow over an elliptical cylinder between two parallel plates.

10. For the smooth pipe discretized in Figure 8.20 with a uniform cross-section of 6 cm^2, find the flow velocities at the center and right end, where the velocity at the left end is $v_x = 6$ cm/s.

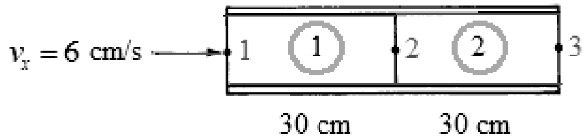


FIGURE 8.20 Discretized pipe for fluid flow.

11. A busbar is a rectangular conductor used in the distribution of electric power in a distribution box. The ground and busbar are considered perfect insulators. Assume the potential of the busbar is 220 V. For the system shown in Figure 8 .21, find the voltage distribution in the air (ε = 1) around the busbar and the maximum electric field intensity.

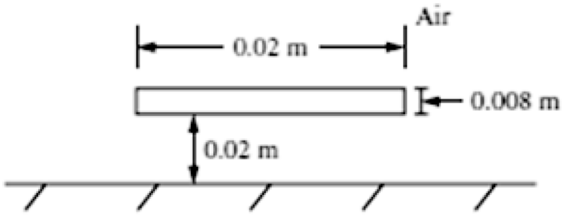


FIGURE 8.21 Exercise 8.11.

REFERENCES

1. J. G. Knudsen and D. L. Katz, "Fluid Dynamics and Heat Transfer," McGraw-Hill, 1958.
2. J. W. Daily and D. R. F. Harleman, "Fluid Dynamics," Addison-Wesley, Reading, 1966.
3. S. S. Rao, "The Finite Element Method in Engineering," Fifth Edition, Butterworth-Heinemann, 2011.

4. T. J. Chung, "Finite Element Analysis in Fluid Dynamics," McGraw-Hill, 1978.
5. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, Cengage Learning, 2012.
6. J. N. Reddy, "An Introduction to the Finite Element Method," Third Edition, McGraw-Hill, 2004.

CHAPTER 9

DYNAMIC ANALYSIS

9.1 INTRODUCTION

A dynamic system is a system that has mass and components, or parts, that are capable of relative motion. Structural dynamics encompass modal analysis, harmonic response analysis, and transient response analysis. The modal analysis consists of finding natural frequencies and corresponding modal shapes of structures. Finding the amplitude of vibration when the loads vary sinusoidal with time is known as *harmonic response analysis*. Finding the structural response to arbitrary time-dependent loading is referred to as transient response analysis.

In this chapter, one-dimensional problems relating to these topics are covered. In vibration analysis, mass matrix and damping matrix will also be discussed in addition to the stiffness matrix.

Governing equation of undamped free vibration assumes the form,

$$\left([k]-\omega^2[m]\right)\{q\}=0. \tag{9.1}$$

The nontrivial solution of equation (9.1) is determinate,

$$\left|\left([k]-\omega^2[m]\right)\right|=0 \tag{9.2}$$

where ω = radian (or natural) frequency.

The solution of equation (9.2) gives natural frequencies (ω). Substituting the value of ω back into the governing equation gives modal shapes (or amplitudes of the displacements) defined by $\{q\}$.

The governing equation for the complete structure in global coordinates is

$$\left([K]-\omega^2[M]\right)\{q\}=0.$$

Mass matrices for bar elements and beam elements are given by,

$$[m]_{Bar}=\rho AL\begin{bmatrix}\frac{1}{3} & \frac{1}{6}\\ \frac{1}{6} & \frac{1}{3}\end{bmatrix} \tag{9.3}$$

$$[m]_{Beam}=\frac{\rho AL}{420}\begin{bmatrix}156 & 22L & 54 & -13L\\ 22L & 4L^2 & 13L & -3L^2\\ 54 & 13L & 156 & -22L\\ -13L & -3L^2 & -22L & 4L^2\end{bmatrix}, \tag{9.4}$$

where ρ = density of the element material; A = cross-sectional area; L = length.

9.2 PROCEDURE OF FINITE ELEMENT ANALYSIS (RELATED TO DYNAMIC PROBLEMS)

Step 1. Select element type.

Step 2. Select a displacement function.

Step 3. Define the strain/displacement and stress/strain relationships.

Step 4. Derive the element stiffness and mass matrices and equations.

Step 5. Assemble the element equations to obtain the global equations and introduce boundary conditions.

Step 6. Solve for the natural frequencies and mode shapes.

9.3 FIXED-FIXED BEAM FOR NATURAL FREQUENCY DETERMINATION

EXAMPLE 9.1

Determine the first two natural frequencies for the fixed-fixed beam shown in Figure 9.1. The beam is made of steel with modulus of elasticity E = 209 GPa, Poisson's ratio = 0.3, length L = 0.75 m, cross-section area A = 625 mm^2, mass density ρ = 7800 kg/m^3, moment of inertia I = 34,700 mm^4.

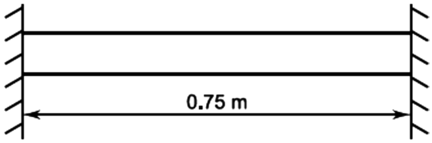


FIGURE 9.1 Fixed-fixed beam for Example 9.1.

Solution

(I) Analytical method.

$$\therefore \quad \omega_1 = \frac{22.4}{L^2}\sqrt{\frac{EI}{\rho A}} \tag{9.5}$$

$$\omega_1 = \frac{22.4}{(0.75)^2}\sqrt{\frac{209\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 1535.95 \text{ rad/s.}$$

Frequency,

$$f_1 = \frac{\omega_1}{2\pi} \tag{9.6}$$

$$f_1 = \frac{1535.95}{2\pi} = 244.45 \text{ Hz}$$

$$\therefore \quad \omega_2 = \frac{61.7}{L^2}\sqrt{\frac{EI}{\rho A}} \tag{9.7}$$

$$\omega_2 = \frac{61.7}{(0.75)^2}\sqrt{\frac{209\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 1535.95 \text{ rad/s.}$$

Frequency,

$$f_2 = \frac{\omega_2}{2\pi} \tag{9.8}$$

$$f_2 = \frac{4230.71}{2\pi} = 673.34 \text{ Hz.}$$

(II) FEM by hand calculations.

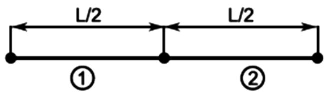


FIGURE 9.1(a) Finite element model.

Mass matrices are,

$$[M_1]=[M_2]=\frac{\rho\times A\times\frac{L}{2}}{420}\begin{bmatrix} 156 & 22\left(\frac{L}{2}\right) & 54 & -13\left(\frac{L}{2}\right) \\ 22\left(\frac{L}{2}\right) & 4\left(\frac{L}{2}\right)^2 & 13\left(\frac{L}{2}\right) & -3\left(\frac{L}{2}\right)^2 \\ 54 & 13\left(\frac{L}{2}\right) & 156 & -22\left(\frac{L}{2}\right) \\ -13\left(\frac{L}{2}\right) & -3\left(\frac{L}{2}\right)^2 & -22\left(\frac{L}{2}\right) & 4\left(\frac{L}{2}\right)^2 \end{bmatrix}$$

$$[M_1]=[M_2]=\frac{\rho\times A\times L}{840}\begin{bmatrix} 156 & 11L & 54 & -6.5L \\ 11L & L^2 & 6.5L & -3\left(\frac{L^2}{4}\right) \\ 54 & 13\left(\frac{L}{2}\right) & 156 & -11L \\ -6.5L & -3\left(\frac{L^4}{4}\right) & -11L & L^2 \end{bmatrix}.$$

Stiffness matrices are,

$$[k_1]=[k_2]=\frac{EI}{\left(\frac{L}{2}\right)^3}\begin{bmatrix} 12 & 6\left(\frac{L}{2}\right) & -12 & 6\left(\frac{L}{2}\right) \\ & 4\left(\frac{L}{2}\right)^2 & -6\left(\frac{L}{2}\right) & 2\left(\frac{L}{2}\right)^2 \\ & & 12 & -6\left(\frac{L}{2}\right) \\ Symmetric & & & 4\left(\frac{L}{2}\right)^2 \end{bmatrix}$$

$$= \frac{8EI}{L^3}\begin{bmatrix} 12 & 3L & -12 & 3L \\ & L^2 & -3L & \frac{L^2}{2} \\ & & 12 & -3L \\ Symmetric & & & L^2 \end{bmatrix}$$

Global mass matrix,

$$[M] = \frac{\rho AL}{840}\begin{bmatrix} 156+156 & -11L+11L \\ -11L+11L & L^2+L^2 \end{bmatrix} = \frac{\rho AL}{840}\begin{bmatrix} 312 & 0 \\ 0 & 2L^2 \end{bmatrix}.$$

Global stiffness matrix,

$$[K] = \frac{8EI}{L^3}\begin{bmatrix} 12+12 & 3L+-3L \\ 3L+-3L & L^2+L^2 \end{bmatrix} = \frac{8EI}{L^3}\begin{bmatrix} 24 & 0 \\ 0 & 2L^2 \end{bmatrix}.$$

Governing equation is,

$$\left([K]-\omega^2[M]\right)\{q\}=0,$$

$$\left(\frac{8EI}{L^3}\begin{bmatrix} 24 & 0 \\ 0 & 2L^2 \end{bmatrix} - \omega^2\frac{\rho AL}{840}\begin{bmatrix} 312 & 0 \\ 0 & 2L^2 \end{bmatrix}\right)\{q\}=0$$

$$\frac{L^3}{8EI}\times\left(\left(\frac{8EI}{L^3}\begin{bmatrix} 24 & 0 \\ 0 & 2L^2 \end{bmatrix} - \omega^2\frac{\rho AL}{840}\begin{bmatrix} 312 & 0 \\ 0 & 2L^2 \end{bmatrix}\right)\{q\}=0\right)$$

$$\left(\begin{bmatrix} 24 & 0 \\ 0 & 2L^2 \end{bmatrix} - \frac{\omega^2\rho ALL^3}{840\times 8EI}\begin{bmatrix} 312 & 0 \\ 0 & 2L^2 \end{bmatrix}\right)\{q\}=0$$

$$\left(\begin{bmatrix} 24-312a & 0 \\ 0 & 2L^2-2L^2a \end{bmatrix}\right)\{q\}=0$$

where $a = \frac{\omega^2\rho AL^4}{6720EI}$.

For nontrivial solution,

$$\left|\left([K]-\omega^2[M]\right)\right|=0$$

$$\left| \begin{bmatrix} 24-312a & 0 \\ 0 & 2L^2-2L^2a \end{bmatrix} \right| = 0.$$

Solving, we get

$$a = 1 \quad \text{or} \quad a = 0.076923$$

$$1 = \frac{\omega^2 \rho A L^4}{6720EI} \quad \text{or} \quad 0.076923 = \frac{\omega^2 \rho A L^4}{6720EI}$$

$$\therefore \quad \omega_1 = \frac{22.74}{L^2}\sqrt{\frac{EI}{\rho A}} \qquad \therefore \quad \omega_2 = \frac{81.98}{L^2}\sqrt{\frac{EI}{\rho A}}.$$

Given

$$E = 209 \text{ GPa}$$

$$\text{A}=625\times 10^{-6} \text{ m}^2$$

$$I = 34700 \times 10^{-12} \text{ m}^4$$

$$\rho = 7800 \text{ kg/m}^3$$

$$L = 0.75 \text{ m.}$$

Substituting, we get

$$\therefore \quad \omega_1 = \frac{22.74}{(0.75)^2}\sqrt{\frac{209\times 10^9 \times 34700 \times 10^{-12}}{7800 \times 625 \times 10^{-6}}} = 1559.26 \text{ rad/s.}$$

Frequency,

$$f_1 = \frac{\omega_1}{2\pi} = \frac{1559.26}{2\pi} = 248.164 \text{ Hz}$$

$$\therefore \quad \omega_2 = \frac{81.98}{(0.75)^2}\sqrt{\frac{209\times 10^9 \times 34700 \times 10^{-12}}{7800 \times 625 \times 10^{-6}}} = 5621.29 \text{ rad/s.}$$

Frequency,

$$f_2 = \frac{\omega_2}{2\pi} = \frac{5621.29}{2\pi} = 894166 \text{ Hz.}$$

(III) Software results.

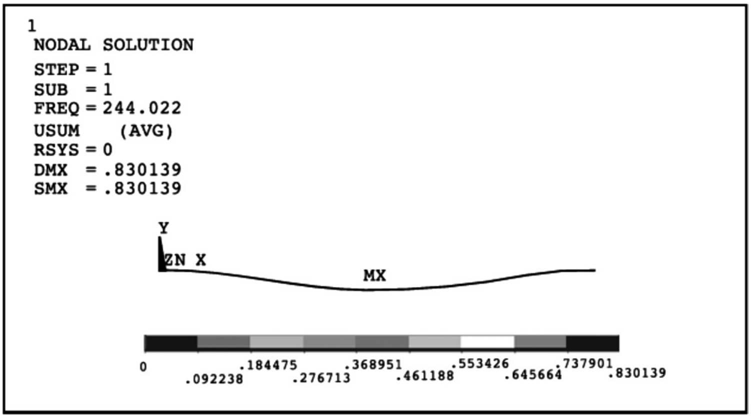


FIGURE 9.1(b) Deflection pattern for a fixed-fixed beam for mode 1 (refer to Appendix D for color figures).

Frequency values (in Hz)

SET	TIME/FREQ	LOAD STEP	SUB STEP	CUMULATIVE
1	244.02	1	1	1
2	671.69	1	2	2

The following are the mode shapes:

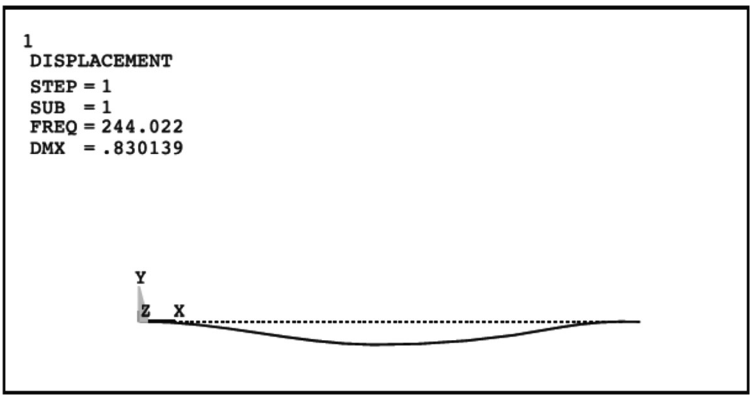


FIGURE 9.1(c) Mode 1 for fixed-fixed beam (refer to Appendix D for color figures).

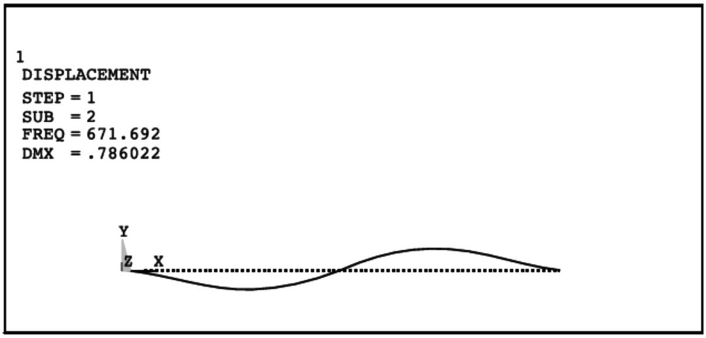


FIGURE 9.1(d) Mode 2 for fixed-fixed beam (refer to Appendix D for color figures).

ANSWERS FOR EXAMPLE 9.1

Parameter	Analytical method	FEM–hand calculation (with 2 elements)	Software results (with 10 elements)
Natural frequency			
f_1	244.45 Hz	248.16 Hz	244.02 Hz
f_2	673.34 Hz	894.66 Hz	671.69 Hz

Procedure for solving the problem using ANSYS® 11.0 academic teaching software

FOR EXAMPLE 9.1

PREPROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Beam > 2D**

   elastic 3 > OK > Close

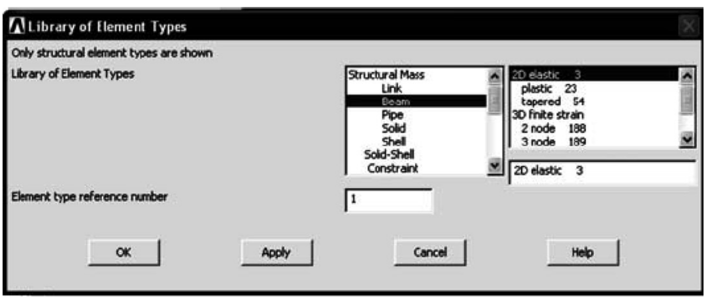


FIGURE 9.2 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

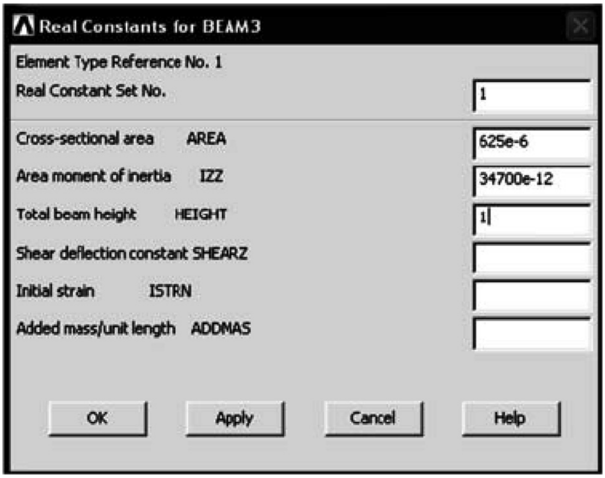


FIGURE 9.3 Enter the area and moment of inertia.

Cross-sectional area AREA > **Enter 625e-6**

Area moment of inertia IZZ > **Enter 34700e-12**

Total beam height HEIGHT > **Enter 1 > OK > Close**

Enter the material properties.

3. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1, **click Structural > Linear > Elastic > Isotropic**

   Enter **EX = 209E9 and PRXY = 0.3 > OK**

   click **Structural > Linear > Density**

   Enter DENS = 7800 > OK

   (**Close** the Define Material Model Behavior window.)

   Create the keypoints and lines as shown in the figure.

4. **Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS,** Enter the coordinates of keypoint 1 > **Apply** Enter the coordinates of keypoint 2 > **OK**

Keypoint locations		
Keypoint number	**X coordinate**	**Y coordinate**
1	0	0
2	0.75	0

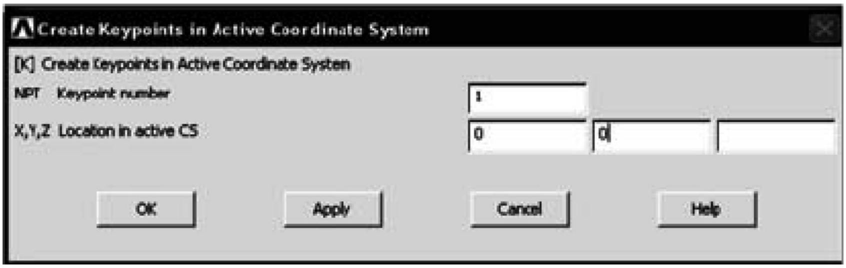


FIGURE 9.4 Enter the keypoint coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line,** Pick

   the 1st and 2nd keypoint > **OK**

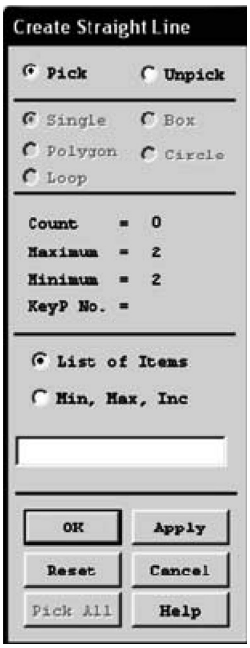


FIGURE 9.5 Pick the keypoints to create lines.

6. **Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All**

   Lines > Enter **NDIV No. of element divisions = 10**

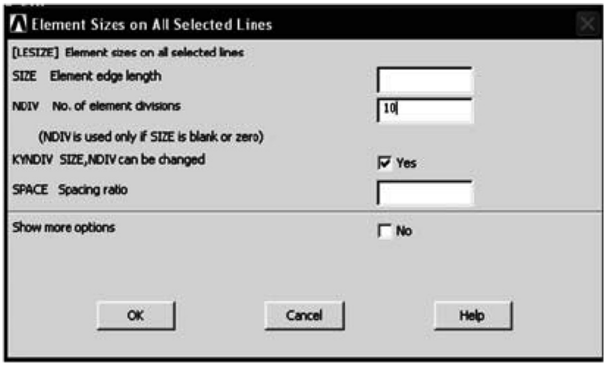


FIGURE 9.6 Specify element length.

7. **Main Menu > Preprocessor > Meshing > Mesh > Lines > Click Pick All**

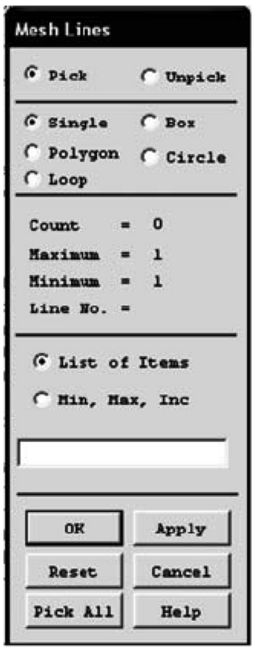


FIGURE 9.7 Create elements by meshing.

8. Main Menu > Solution > Analysis Type > New Analysis > Select Modal > OK

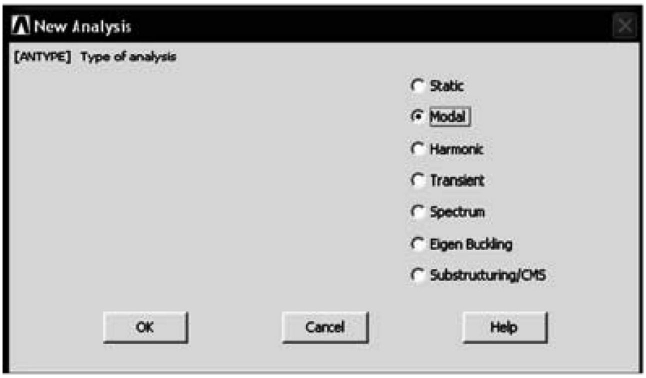


FIGURE 9.8 Define the type of analysis.

9. Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >

Displacement > On Nodes Pick the left most node and right most node > **Apply >**

Select All DOF > OK

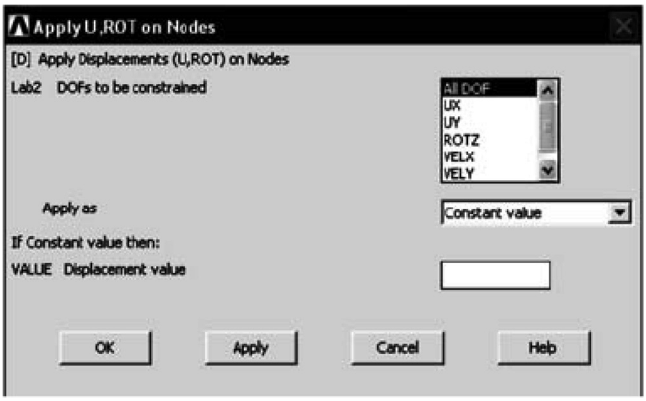


FIGURE 9.9 Apply the displacement constraint.

10. Main Menu > Solution > Analysis Type > Analysis Options > Select PCG Lanczos option

Enter No. of modes to extract = 2

NMODE No. of modes to expand = 2 > OK

After OK one more window will appear, for that also click OK

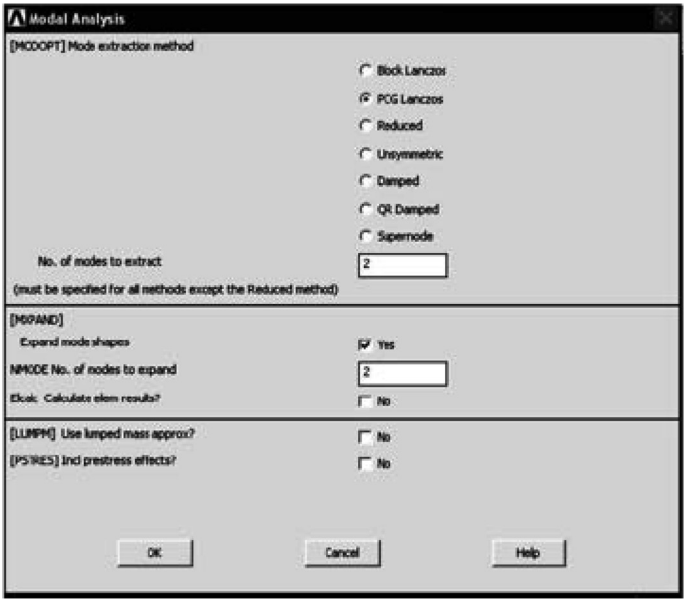


FIGURE 9.10 Select the number of modes to extract.

Solution

The interactive solution proceeds.

11. Main Menu > Solution > Solve > Current LS > OK

The **/STATUS Command** window displays the problem parameters, and **the Solve Current Load Step** window is shown. Check the solution options in the **/STATUS** window, and if all is OK, select **File > Close.**

In the **Solve Current Load Step** window, select **OK**, and when the solution is complete, **close** the **"Solution is Done!"** window.

POSTPROCESSING

12. Main Menu > General Postproc > Results Summary

This result is shown as frequency values in Hz.

13. Main Menu > General Postproc > Read Results > First Set

14. Main Menu > General Postproc > Plot Results > Deformed Shape > Click Def + undeformed > OK

This result is the first mode shown in Figure 9.1(c).

15. Main Menu > General Postproc > Read Results > Next Set

16. Main Menu > General Postproc > Plot Results > Deformed Shape > Click Def + undeformed > OK

This result is the second mode as shown in Figure 9.1(d).

9.4 TRANSVERSE VIBRATIONS OF A CANTILEVER BEAM

EXAMPLE 9.2

Determine the first four natural frequencies for the cantilever beam shown in Figure 9.11. The beam is made of steel with modulus of elasticity, E = 207 GPa, Poisson's ratio = 0.3, length L = 0.75 m, cross-section area A = 625 mm^2, mass density ρ = 7800 kg/m^3, moment of inertia I = 34,700 mm^4**.**

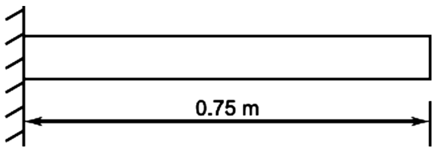


FIGURE 9.11 Cantilever beam for Example 9.2.

Solution

(I) Analytical solution.

$$\omega_1 = \frac{3.52}{L^2}\sqrt{\frac{EI}{\rho A}}$$

$$\omega_1 = \frac{3.52}{(0.75)^2}\sqrt{\frac{207\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 240 \text{ rad/s.}$$

Frequency,

$$f_1 = \frac{\omega_1}{2\pi}$$

$$f_1 = \frac{240}{2\pi} = 38.197 \text{ Hz}$$

$$\omega_2 = \frac{22}{L^2}\sqrt{\frac{EI}{\rho A}}$$

$$\omega_2 = \frac{22}{(0.75)^2}\sqrt{\frac{207\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 1501 \text{ rad/s.}$$

Frequency,

$$f_2 = \frac{\omega_2}{2\pi}$$

$$f_2 = \frac{1501}{2\pi} = 238.89 \text{ Hz}$$

$$\omega_3 = \frac{61.7}{L^2}\sqrt{\frac{EI}{\rho A}}$$

$$\omega_2 = \frac{61.7}{(0.75)^2}\sqrt{\frac{207\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 4210 \text{ rad/s.}$$

Frequency,

$$f_3 = \frac{\omega_3}{2\pi}$$

$$f_3 = \frac{4210}{2\pi} = 670.04 \text{ Hz}$$

$$\omega_4 = \frac{121}{L^2}\sqrt{\frac{EI}{\rho A}}$$

$$\omega_2 = \frac{121}{(0.75)^2}\sqrt{\frac{207\times10^9\times34700\times10^{-12}}{7800\times625\times10^{-6}}} = 8257 \text{ rad/s.}$$

Frequency,

$$f_4 = \frac{\omega_4}{2\pi}$$

$$f_4 = \frac{8257}{2\pi} = 1314.14 \text{ Hz.}$$

(II) FEM by hand calculations.

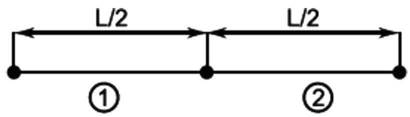


FIGURE 9.11(a) Finite element model.

Stiffness matrices are,

$$[k_1] = [k_2] = \frac{EI}{\left(\frac{L}{2}\right)^3}\begin{bmatrix} 12 & 6\left(\frac{L}{2}\right) & -12 & 6\left(\frac{L}{2}\right) \\ 6\left(\frac{L}{2}\right) & 4\left(\frac{L}{2}\right)^2 & -6\left(\frac{L}{2}\right) & 2\left(\frac{L}{2}\right)^2 \\ -12 & -6\left(\frac{L}{2}\right) & 12 & -6\left(\frac{L}{2}\right) \\ 6\left(\frac{L}{2}\right) & 2\left(\frac{L}{2}\right)^2 & -6\left(\frac{L}{2}\right) & 4\left(\frac{L}{2}\right)^2 \end{bmatrix}.$$

Global stiffness matrix,

$$[K]=\frac{8EI}{L^3}\begin{bmatrix} 24 & 0 & -12 & 3L \\ 0 & 2L^2 & -3L & \dfrac{L^2}{2} \\ -12 & -3L & 12 & -3L \\ 3L & \dfrac{L^2}{2} & -3L & L^2 \end{bmatrix}$$

$$=136209.07\begin{bmatrix} 24 & 0 & -12 & 2.25 \\ 0 & 1.125 & -2.25 & 0.28125 \\ -12 & -2.25 & 12 & -2.25 \\ 2.25 & 0.28125 & -2.25 & 0.5625 \end{bmatrix}.$$

Mass matrices are,

$$[M_1]=[M_2]=\frac{\rho\times A\times\dfrac{L}{2}}{420}\begin{bmatrix} 156 & 22\left(\dfrac{L}{2}\right) & 54 & -13\left(\dfrac{L}{2}\right) \\ 22\left(\dfrac{L}{2}\right) & 4\left(\dfrac{L}{2}\right)^2 & 13\left(\dfrac{L}{2}\right) & -3\left(\dfrac{L}{2}\right)^2 \\ 54 & 13\left(\dfrac{L}{2}\right) & 156 & -22\left(\dfrac{L}{2}\right) \\ -13\left(\dfrac{L}{2}\right) & -3\left(\dfrac{L}{2}\right)^2 & -22\left(\dfrac{L}{2}\right) & 4\left(\dfrac{L}{2}\right)^2 \end{bmatrix}$$

$$[M_1]=[M_2]=\frac{\rho\times A\times L}{840}\begin{bmatrix} 156 & 11L & 54 & -6.5L \\ 11L & L^2 & 6.5L & -3\left(\dfrac{L^2}{4}\right) \\ 54 & 13\left(\dfrac{L}{2}\right) & 156 & -11L \\ -6.5L & -3\left(\dfrac{L^4}{4}\right) & -11L & L^2 \end{bmatrix}.$$

Global mass matrix is,

$$[M]=\frac{\rho\times A\times L}{840}\begin{bmatrix} 312 & 0 & 54 & -6.5L \\ 0 & L^2 & 6.5L & -3\left(\frac{L^2}{4}\right) \\ 54 & 13\left(\frac{L}{2}\right) & 156 & -11L \\ -6.5L & -3\left(\frac{L^4}{4}\right) & -11L & L^2 \end{bmatrix}$$

$$[M]=4.352676\times10^{-3}\begin{bmatrix} 312 & 0 & 54 & -4.875 \\ 0 & 1.125 & 4.875 & -0.421875 \\ 54 & 4.875 & 156 & -8.25 \\ -4.875 & -0.421875 & -8.25 & 0.5625 \end{bmatrix}.$$

Governing equation is,

$$\left([K]-\omega^2[M]\right)\{q\}=\begin{Bmatrix}\omega_2\\ \theta_2\\ \omega_3\\ \theta_3\end{Bmatrix}=0.$$

For a nontrivial solution

$$\text{Det }\left([K]-\omega^2[M]\right)=0\Rightarrow\left|\left([K]-\omega^2[M]\right)\right|=0.$$

Substituting and solving, we get

$$\omega^2=\begin{Bmatrix}\omega_1^2\\ \omega_2^2\\ \omega_3^2\\ \omega_4^2\end{Bmatrix}=10^8\begin{Bmatrix}0.0006\\ 0.0230\\ 0.2630\\ 2.2159\end{Bmatrix}$$

$$\omega_1=245\text{ rad/s}\Rightarrow f_1=\frac{\omega_1}{2\pi}=\frac{245}{2\pi}=38.993\text{ Hz}$$

$$\omega_2=1517\text{ rad/s}\Rightarrow f_2=\frac{\omega_2}{2\pi}=\frac{1517}{2\pi}=241.44\text{ Hz}$$

$$\omega_3 = 5128 \text{ rad/s} \Rightarrow f_3 = \frac{\omega_3}{2\pi} = \frac{5128}{2\pi} = 816.147 \text{ Hz}$$

$$\omega_3 = 14885.9 \text{ rad/s} \Rightarrow f_4 = \frac{\omega_4}{2\pi} = \frac{14885.9}{2\pi} = 2369.16 \text{ Hz.}$$

(III) Software results.

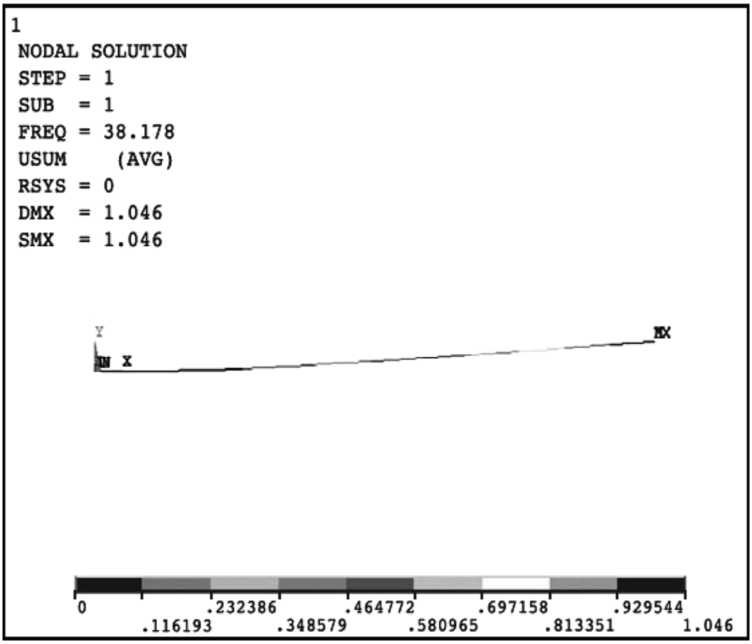


FIGURE 9.11(b) Deflection pattern for a fixed-fixed beam for mode 1 (refer to Appendix D for color figures).

Frequency values (in Hz)

SET	TIME/FREQ	LOAD STEP	SUB STEP	CUMULATIVE
1	38.178	1	1	1
2	238.94	1	2	2
3	667.71	1	3	3
4	1305.2	1	4	4

The following are the mode shapes:

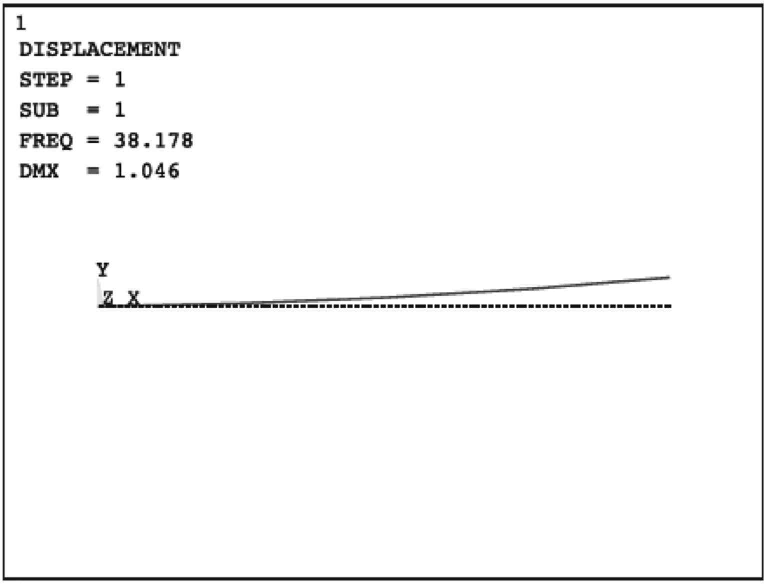


FIGURE 9.11(c) Mode 1 for cantilever beam (refer to Appendix D for color figures).

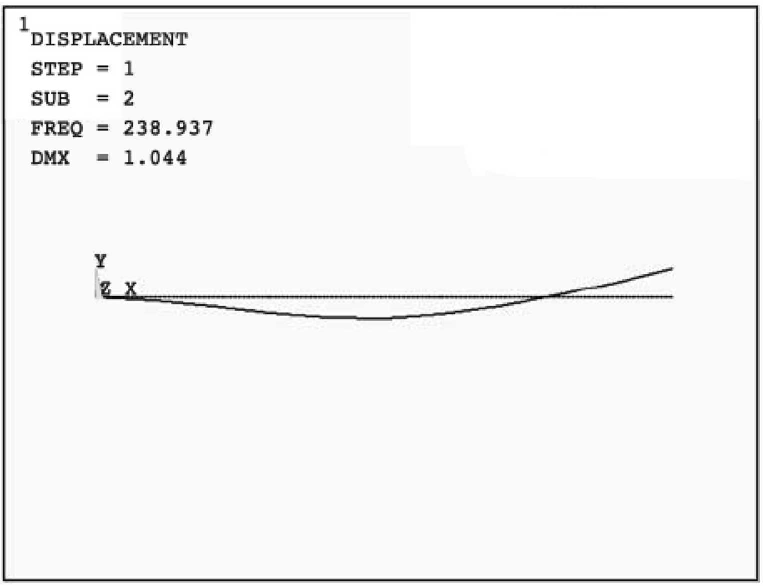


FIGURE 9.11(d) Mode 2 for cantilever beam (refer to Appendix D for color figures).

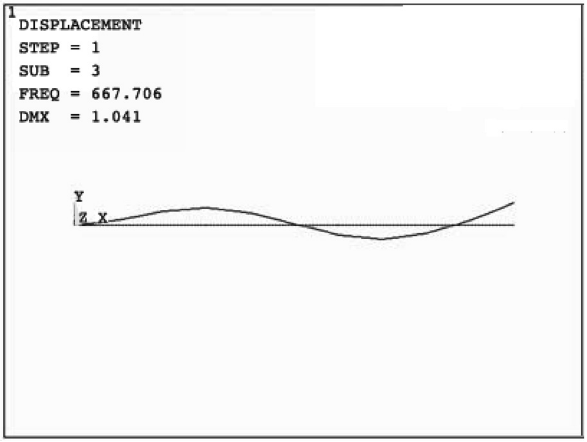


FIGURE 9.11(e) Mode 3 for cantilever beam (refer to Appendix D for color figures).

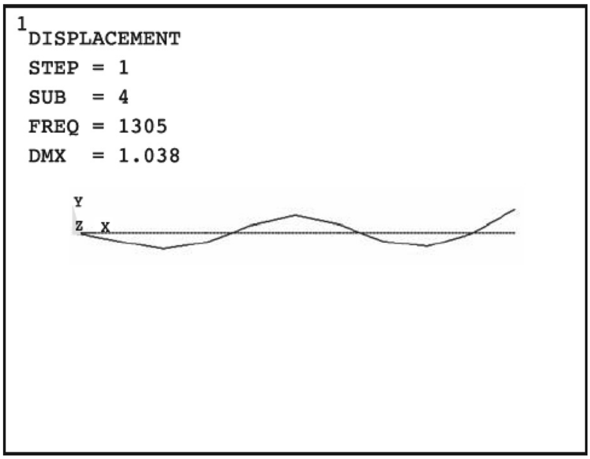


FIGURE 9.11(f) Mode 4 for cantilever beam (refer to Appendix D for color figures).

Answers for Example 9.2

Parameter	Analytical method	FEM–hand calculation (with 2 elements)	Software results (with 10 elements)
Natural frequency			
f_1	38.197 Hz	38.993 Hz	38.178 Hz
f_2	238.89 Hz	241.44 Hz	238.94 Hz
f_3	670.04 Hz	816.147 Hz	667.71 Hz
f_4	1314.14 Hz	2369.16 Hz	1305.2 Hz

9.5 FIXED-FIXED BEAM SUBJECTED TO FORCING FUNCTION

EXAMPLE 9.3

For the fixed-fixed beam subjected to the time-dependent forcing function shown in Figure 9.12, determine the displacement response for 0.2 seconds. Use time step integration of 0.01 sec. Let E = 46 GPa, Poisson's ratio = 0.35, length of beam L = 5 m, cross-section area A= 1 m^2, mass density, ρ = 1750 kg/m^3, moment of inertia I = 4.2×10^{-5} m^4.

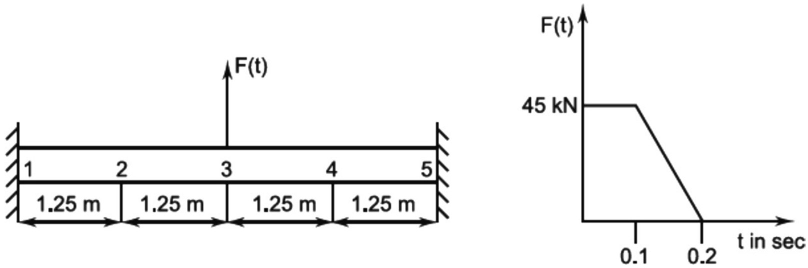


FIGURE 9.12 Fixed-fixed beam subjected to the time-dependent forcing function for Example 9.3.

Solution

(I) Software results.

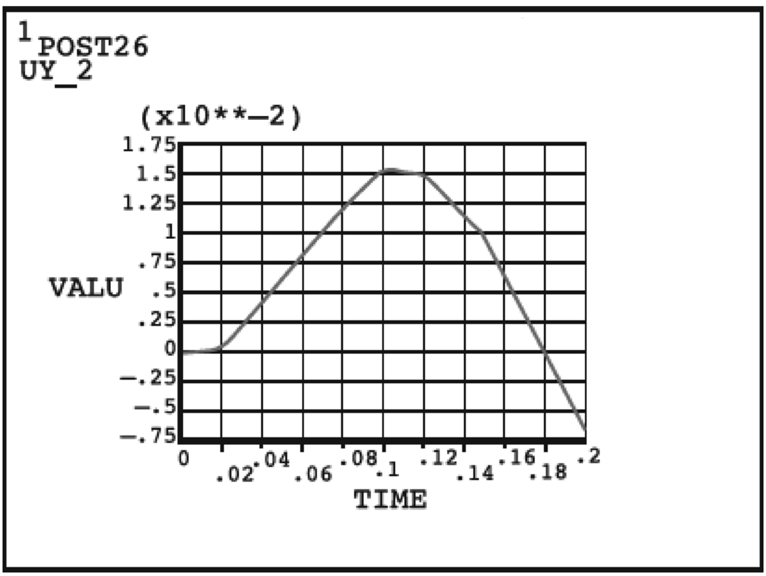


FIGURE 9.12(a) Displacement response for 0.2 sec for node 2 (refer to Appendix D for color figures).

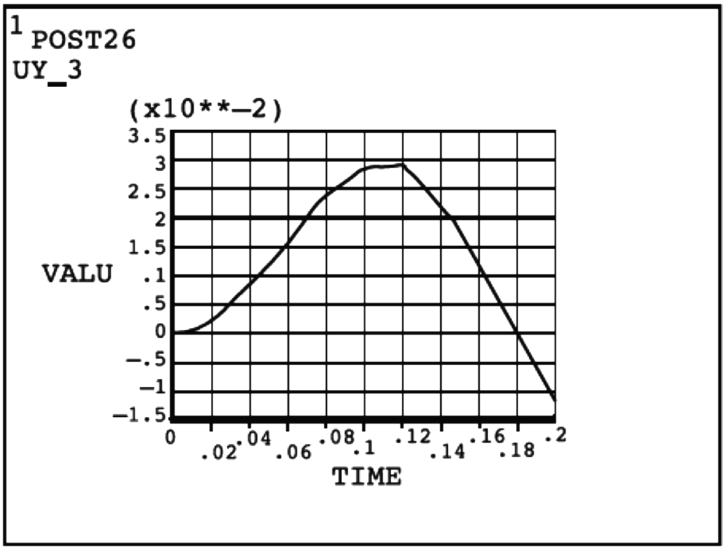


FIGURE 9.12(b) Displacement response for 0.2 sec for node 3 (refer to Appendix D for color figures).

Displacement values (in meters) for node 2

TIME	2 UY
	UY_2
0.0000	0.00000
0.10000E–01	0.421220E–05
0.20000E–01	0.284618E–03
0.50000E–01	0.602161E–02
0.80000E–01	0.121677E–01
0.10000	0.153042E–01
0.12000	0.148820E–01
0.15000	0.979873E–02
0.18000	0.868368E–04
0.20000	–0.649350E–02

Displacement values (in meters) for node 3

TIME	3 UY
	UY_3
0.0000	0.00000
0.10000E–01	0.505126E–03
0.20000E–01	0.218959E–02
0.50000E–01	0.113766E–01
0.80000E–01	0.241211E–01
0.10000	0.286233E–01
0.12000	0.292504E–01
0.15000	0.183799E–01
0.18000	–0.205644E–03
0.20000	–0.117477E–01

Procedure for solving the problem using ANSYS® 11.0 academic teaching software.

FOR EXAMPLE 9.3

PREPROCESSING

1. **Main Menu > Preferences > Select Structural > OK**

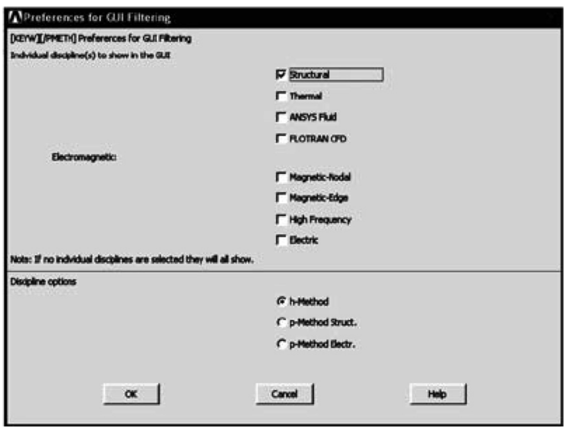


FIGURE 9.13 Selecting the preferences.

2. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Beam > 2D**

   elastic 3 > OK > Close

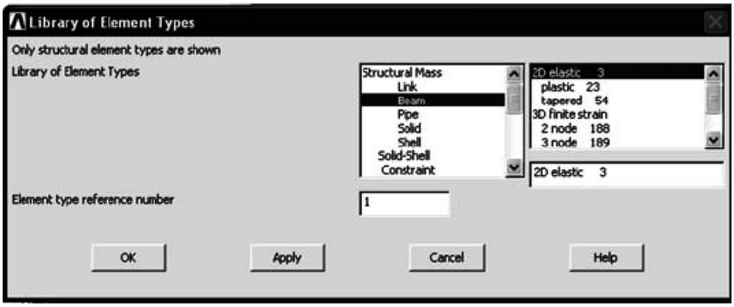


FIGURE 9.14 Element selection.

3. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

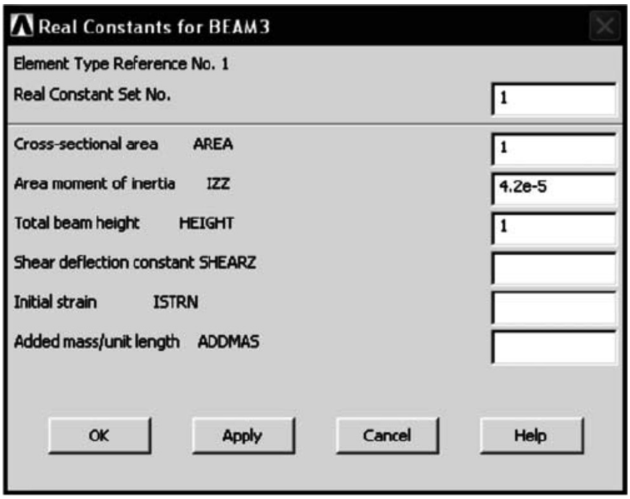


FIGURE 9.15 Enter the area and moment of inertia.

Cross-sectional area AREA > **Enter 1**

Area moment of inertia IZZ > **Enter 4.2e–5**

Total beam height HEIGHT > **Enter 1 > OK > Close**

Enter the material properties.

4. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

   Enter **EX = 46E9 and PRXY = 0.35 > OK**

   Click **Structural > Linear > Density**

   Enter DENS = 1750 > OK

   (**Close** the Define Material Model Behavior window.)

   Create the nodes and elements as shown in the figure.

5. **Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS** Enter the coordinates of node 1 > **Apply >** Enter the coordinates of node 2 **> Apply > Enter** the coordinates of node 3 **> Apply >** Enter the coordinates of node 4 **> Apply** Enter the coordinates of node 5 **> OK**

Node locations		
Node number	**X coordinate**	**Y coordinate**
1	0	0
2	1.25	0
3	2.5	0
4	3.75	0
5	5	0

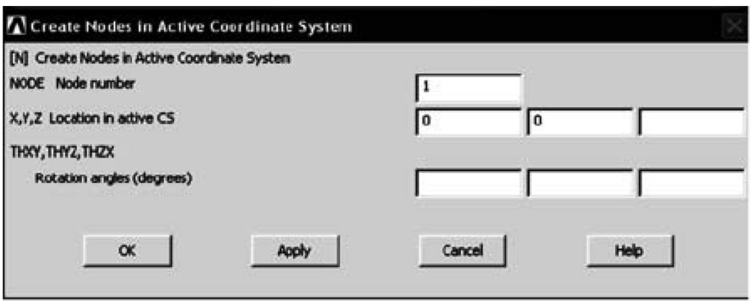


FIGURE 9.16 Enter the node coordinates.

6. **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru**

   Nodes Pick the 1st and 2nd node > Apply > Pick the 2nd and 3rd node > Apply > Pick the 3rd and 4th node > Apply > Pick the 4th and 5th node > **OK**

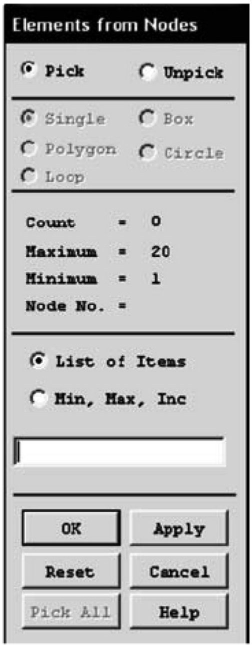


FIGURE 9.17 Pick the nodes to create elements.

7. **Main Menu > Solution > Analysis Type > New Analysis > Select Transient > OK**

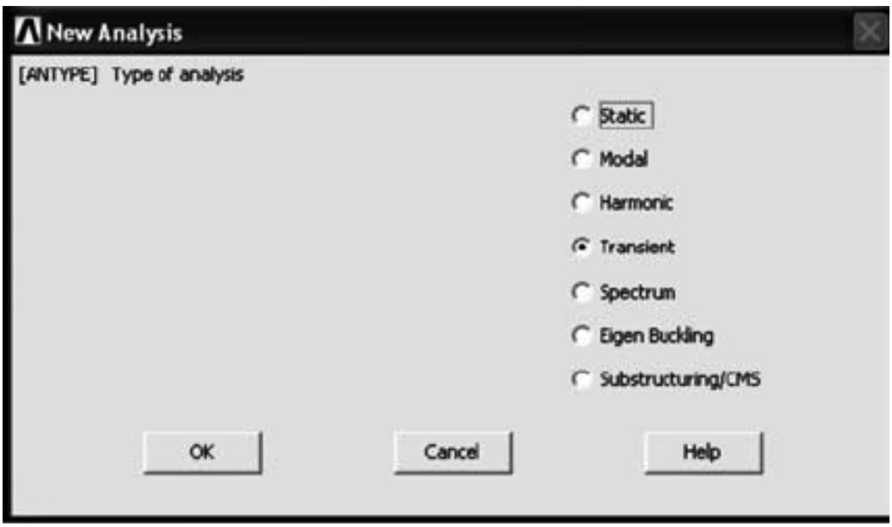


FIGURE 9.18 Define the type of analysis.

then select > **Reduced > OK**

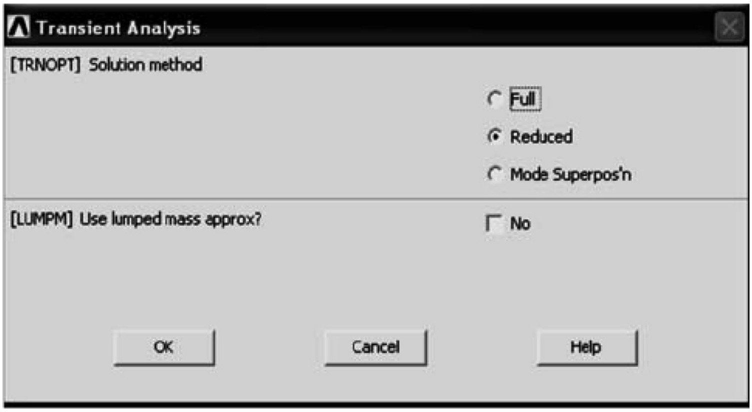


FIGURE 9.19 Define the type of transient analysis.

8. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes >** Pick the left most node and right most node **> Apply > Select All DOF > OK**

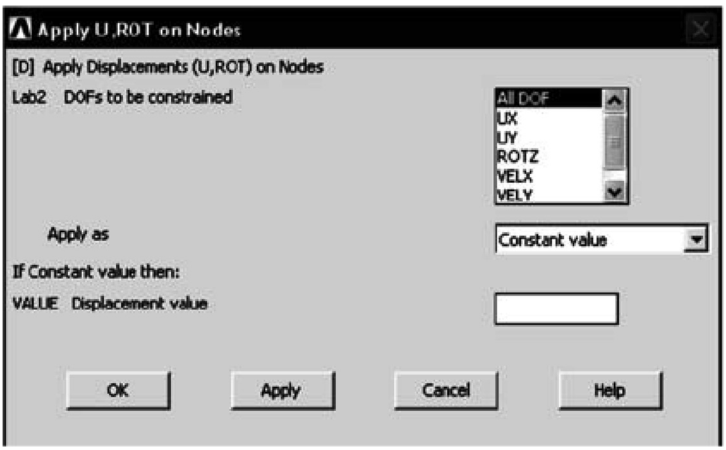


FIGURE 9.20 Apply the displacement constraint.

9. **Main Menu > Solution > Master DOFs > User Selected > Define > Pick 2nd, 3rd, and 4th node > Apply > Select UY from Lab 1 1st degree of freedom > OK**

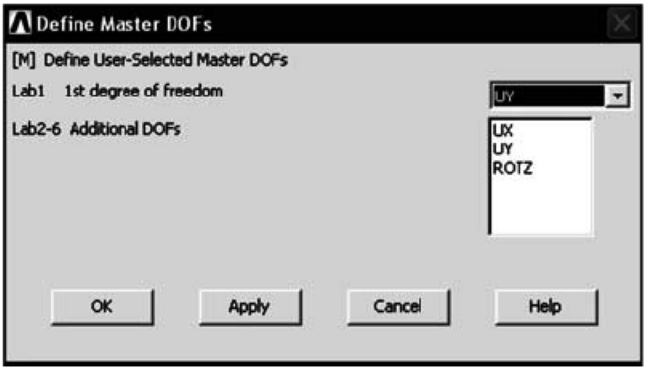


FIGURE 9.21 Defining master DOF.

10. Main Menu > Solution > Load Step Opts > Time/Frequency > Time-Time Step

Enter [TIME] Time at the end of load step – 0

Enter [DELTIM] Time step size – 0.01 **< OK**

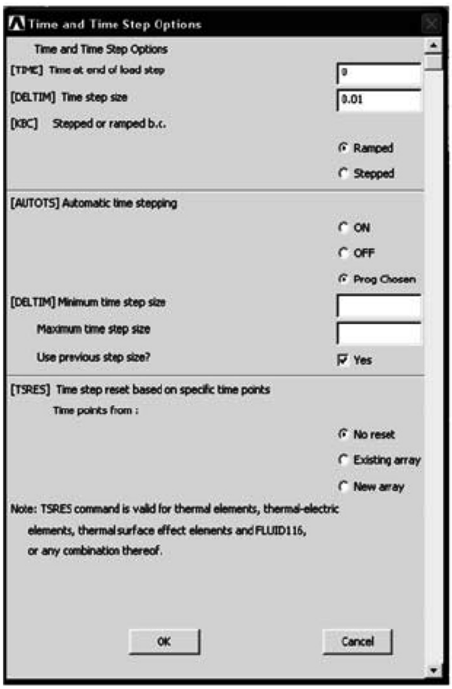


FIGURE 9.22 Defining time step size.

11. **Main Menu > Solution > Load Step Opts > Write LS File**

    Enter LSNUM Load step fine number n = 1 > **OK**

FIGURE 9.23 Creating LS file.

12. **Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes** > Pick the middle or 3rd node **Apply > Enter FY = 45e3 > OK**

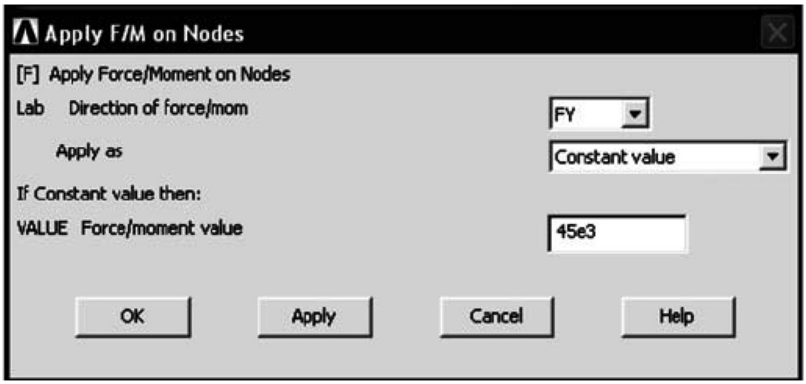


FIGURE 9.24 Applying force on node.

13. **Main Menu > Solution > Load Step Opts > Time/Frequency > Time-Time Step**

    Enter [TIME] Time at the end of load step – 0.01 **> OK**

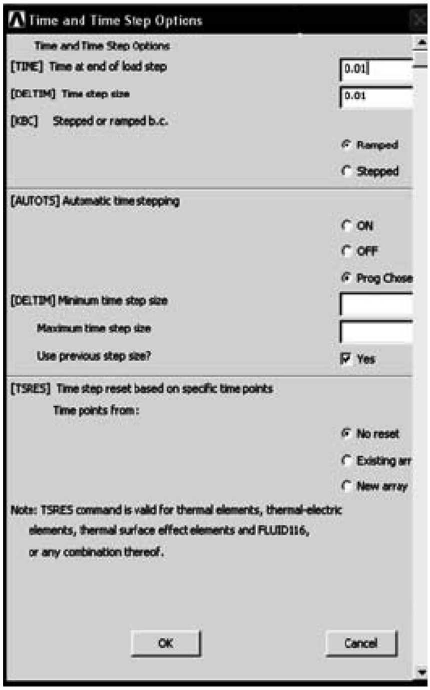


FIGURE 9.25 Defining time at the end of 1st load step.

14. Main Menu > Solution > Load Step Opts > Write LS File

Enter LSNUM Load step fine number n = 2 > **OK**

FIGURE 9.26 Creating LS file for 1st load step.

Similarly, repeat Steps 13 and 14 for Time at the end of load step of 0.02, 0.05, 0.08, and 0.1, and each time create an LS file with the next numbers (n), i.e., 3, 4, 5, and 6.

15. **Main Menu > Solution > Define Loads > Delete > Structural > Force/Moment > On Nodes >** Pick the middle or 3rd node **Apply > OK**

16. **Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes >** Pick the middle or 3rd node **Apply > Enter FY = 36e3 > OK**

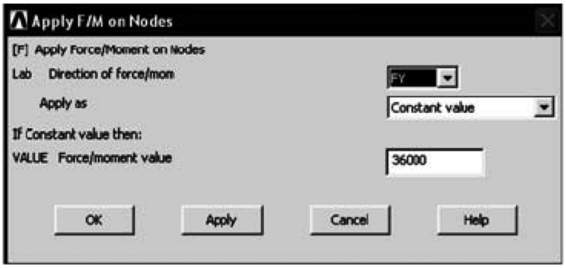


FIGURE 9.27 Applying force on node.

17. **Main Menu > Solution > Load Step Opts > Time/Frequency > Time-Time Step**

    Enter [TIME] Time at the end of load step – 0.12 **> OK**

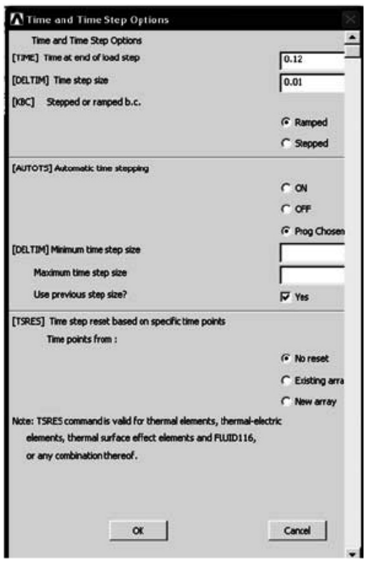


FIGURE 9.28 Defining time at the end of 6th load step.

18. **Main Menu > Solution > Load Step Opts > Write LS File**

    Enter LSNUM Load step fine number n = 7 > **OK**

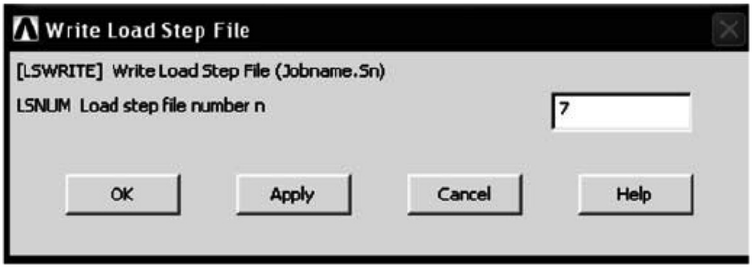


FIGURE 9.29 Creating LS file for 6th load step.

Repeat Step 15 and delete the force.

Then apply the force of 22.5 kN (i.e., 22.5e3) and define the Time at the end of the load step of 0.15 and create an LS file with number (n) = 8.

Again, repeat Step 15 and delete the force.

Then apply the force of 9 kN (i.e., 9e3) and define the Time at the end of the load step of 0.18 and create an LS file with number (n) = 9.

Again, repeat Step 15 and delete the force.

Define the Time at the end of the load step of 0.2 and create an LS file with the number (n) = 10.

19. **Main Menu > Solution > Solve > From LS Files**

    Enter LSMIN Starting LS file number = 1

    Enter LSMAX Ending LS file number = 10

    LSINC File number increment = 1

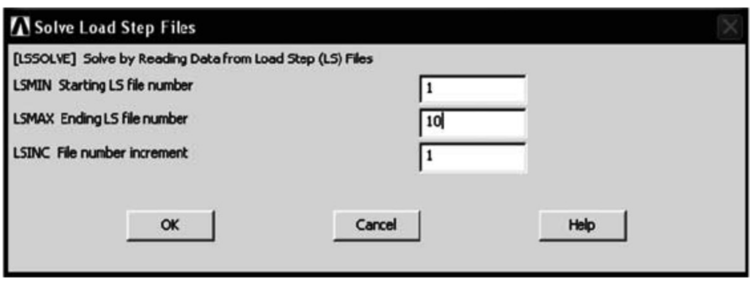


FIGURE 9.30 Solving from L S files.

20. Main Menu > TimeHist Postpro

The following dialog box will appear.

FIGURE 9.31 Timehist dialog box.

In that dialog box click on the first icon, i.e., on Add data, one more dialog box will appear as shown below. **Then click on DOF Solution > y-Component of displacement > OK.**

FIGURE 9.32 Selecting the displacement in *y*-direction.

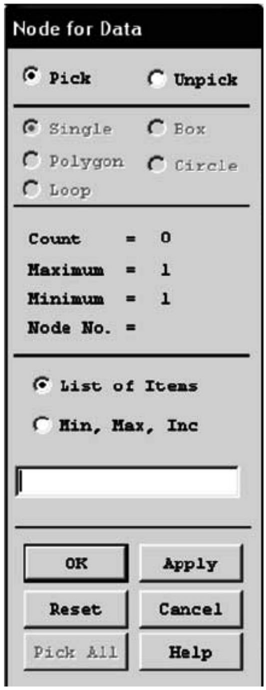


FIGURE 9.33 Selecting the node.

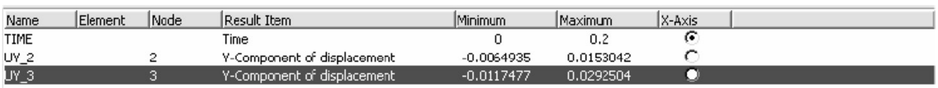
It asks for the node to pick, so pick node 3 or the middle node < OK.

Then, in the Timehist dialog box, click on 4th icon, i.e., List Data (refer to Figure 9.31).

This result is shown as displacement values for node 3 in the software results of the problem. Then, in the Timehist dialog box, click on 3rd icon, i.e., Graph Data (refer to Figure 9.31).

The result is shown in Figure 9.12(b) for node 3 in the software results of the problem.

Maximum displacement values (in meters)



Name	Element	Node	Result Item	Minimum	Maximum	X-Axis
TIME			Time	0	0.2	
UY_2		2	Y-Component of displacement	-0.0064935	0.0153042	
UY_3		3	Y-Component of displacement	-0.0117477	0.0292504	

FIGURE 9.34 Values of displacement.

9.6 AXIAL VIBRATIONS OF A BAR

EXAMPLE 9.4

For the bar shown in Figure 9.35, determine the first two natural frequencies. Let E = 207 GPa, Poisson's ratio = 0.3, length L = 2.5 m, cross-section area A = 1 m^2, mass density ρ = 7800 kg/m^3.

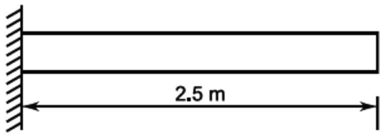


FIGURE 9.35 **The Bar** for Example 9.4.

Solution

(I) Analytical method.

$$\therefore \quad \omega_1 = \frac{1.57}{L}\sqrt{\frac{E}{\rho}}$$

$$\omega_1 = \frac{1.57}{2.5}\sqrt{\frac{207\times10^9}{7800}} = 3235.17 \text{ rad/s.}$$

Frequency,

$$f_1 = \frac{\omega_1}{2\pi}$$

$$f_1 = \frac{3235.17}{2\pi} = 514.89 \text{ Hz}$$

$$\therefore \quad \omega_2 = \frac{4.71}{L}\sqrt{\frac{E}{\rho}}$$

$$\omega_2 = \frac{4.71}{2.5}\sqrt{\frac{207\times10^9}{7800}} = 9705.52 \text{ rad/s.}$$

Frequency,

$$f_2 = \frac{\omega_2}{2\pi}$$

$$f_2 = \frac{9705.52}{2\pi} = 1544.68 \text{ Hz.}$$

(II) FEM by hand calculations.

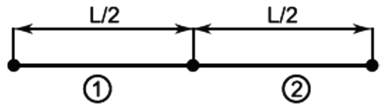


FIGURE 9.35(a) Finite element model.

Mass matrices are,

$$[M_1] = [M_2] = \frac{\rho A\left(\frac{L}{2}\right)}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} = \frac{\rho AL}{12}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}.$$

Stiffness matrices are,

$$[k_1] = [k_2] = \frac{EA}{\left(\frac{L}{2}\right)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}.$$

Global mass matrix is,

$$[M] = \frac{\rho AL}{12}\begin{bmatrix} 2 & 1 & 0 \\ 1 & 4 & 1 \\ 0 & -1 & 2 \end{bmatrix}.$$

Global stiffness matrix is,

$$[K] = \frac{2EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}.$$

Governing equation,

$$\left([K]-\omega^2[M]\right)\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}=[0]$$

$$\left(\frac{2EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}-\omega^2\times\frac{\rho AL}{12}\begin{bmatrix} 2 & 1 & 0 \\ 1 & 4 & 1 \\ 0 & -1 & 2 \end{bmatrix}\right)\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}=0.$$

Boundary conditions are, $u_1 = 0$.

Applying boundary conditions and for a nontrivial solution,

$$\left\|\begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix}-\frac{\omega^2\rho L^2}{24E}\begin{bmatrix} 4 & -1 \\ -1 & 2 \end{bmatrix}\right\|=0,$$

i.e.,

$$\left\|\begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix}-\begin{bmatrix} 4a & a \\ a & 2a \end{bmatrix}\right\|=0,$$

where

$$a=\frac{\omega^2\rho L^2}{24E}\cdot$$

By solving, $\left\|\begin{bmatrix} 2-4a & -1-a \\ -1-a & 1-2a \end{bmatrix}\right\|=0$, we get

$$a=0.1081941=\frac{\omega^2\rho L^2}{24E} \qquad \text{or} \qquad a=1.3203772=\frac{\omega^2\rho L^2}{24E}$$

$$\therefore \quad \omega_1=\frac{1.61}{L}\sqrt{\frac{E}{\rho}}$$

$$\omega_1=\frac{1.61}{2.5}\sqrt{\frac{207\times 10^9}{7800}}=3317.6 \text{ rad/s.}$$

Frequency,

$$f_1 = \frac{\omega_1}{2\pi}$$

$$f_1 = \frac{3317.6}{2\pi} = 528.01 \text{ Hz}$$

$$\therefore \quad \omega_2 = \frac{5.63}{L}\sqrt{\frac{E}{\rho}}$$

$$\omega_2 = \frac{5.63}{2.5}\sqrt{\frac{207\times10^9}{7800}} = 11601.3 \text{ rad/s.}$$

Frequency,

$$f_2 = \frac{\omega_2}{2\pi}$$

$$f_2 = \frac{11601.3}{2\pi} = 1846.4 \text{ Hz.}$$

(III) Software results.

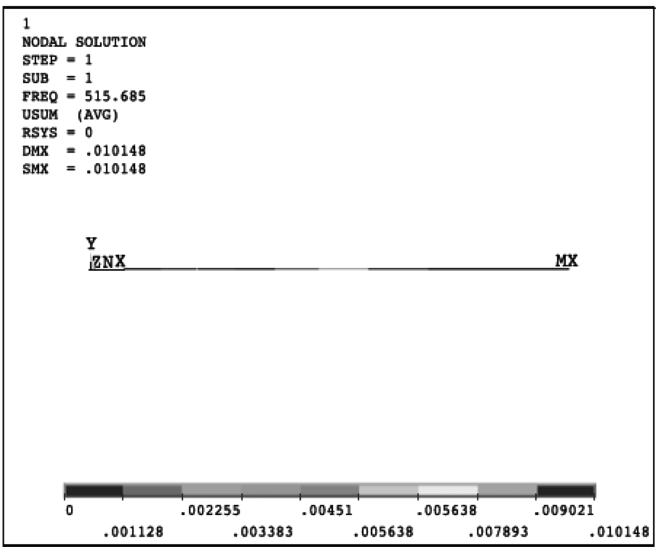


FIGURE 9.35(b) Deflection pattern for a bar (refer to Appendix D for color figures).

Frequency values (in Hz)

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	515.68	1	1	1
2	1559.8	1	2	2

Answers for Example 9.4

Parameter	Analytical method	FEM-hand calculation (with 2 elements)	Software results (with 10 elements)
Natural frequency			
f_1	514.89 Hz	528.01 Hz	515.68 Hz
f_2	1544.68 Hz	1846.4 Hz	1559.8 Hz

Procedure for solving the problem using ANSYS® 11.0 academic teaching software.

For Problem 9.4

PREPROCESSING

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add > Link > 2D spar 1 > OK > Close**

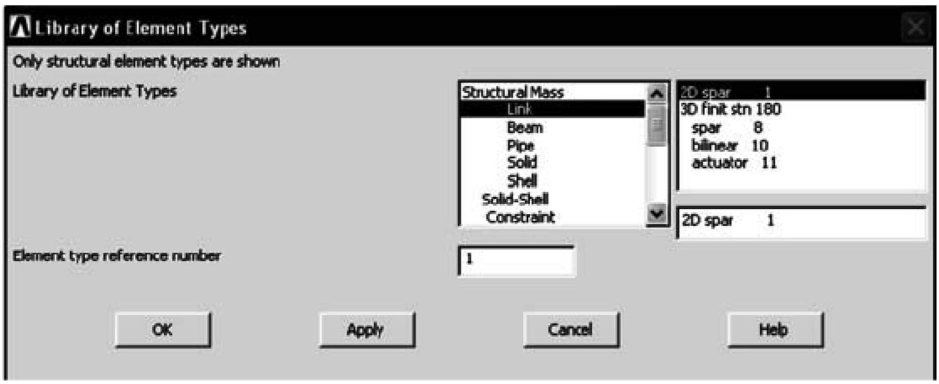


FIGURE 9.36 Element selection.

2. **Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add > OK**

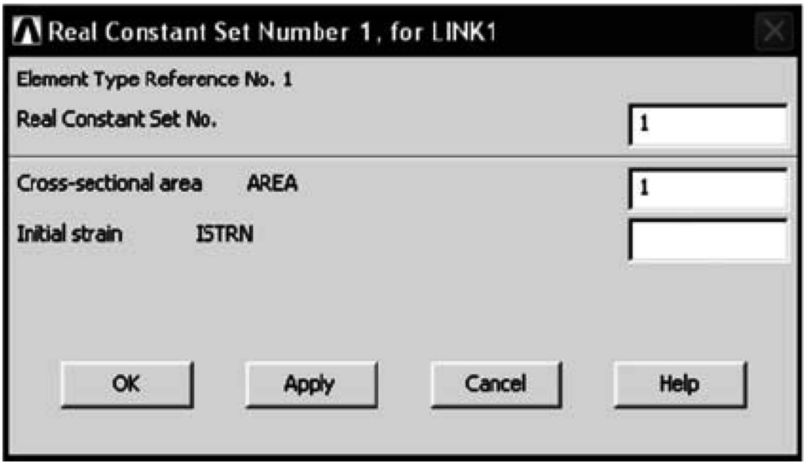


FIGURE 9.37 Enter the cross-sectional area.

Cross-sectional area AREA **> Enter 1 > OK > Close**

Enter the material properties.

3. **Main Menu > Preprocessor > Material Props > Material Models**

   Material Model Number 1, click **Structural > Linear > Elastic > Isotropic**

   Enter **EX = 207E9 and PRXY = 0.3 > OK**

   Click **Structural > Linear > Density**

   Enter DENS = 7800

   (**Close** the Define Material Model Behavior window.)

   Create the keypoints and lines as shown in the figure.

4. **Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS** Enter the coordinates of keypoint 1 **> Apply** Enter the coordinates of keypoint 2 **> OK**

Keypoint locations		
Keypoint number	**X coordinate**	**Y coordinate**
1	0	0
2	2.5	0

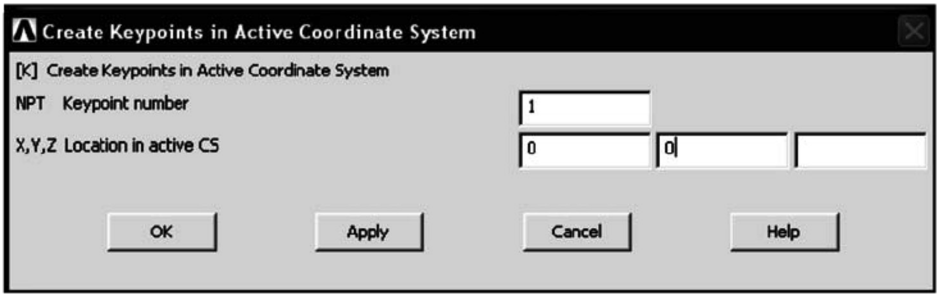


FIGURE 9.38 Enter the keypoint coordinates.

5. **Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line** Pick the 1st and 2nd keypoint **> OK**

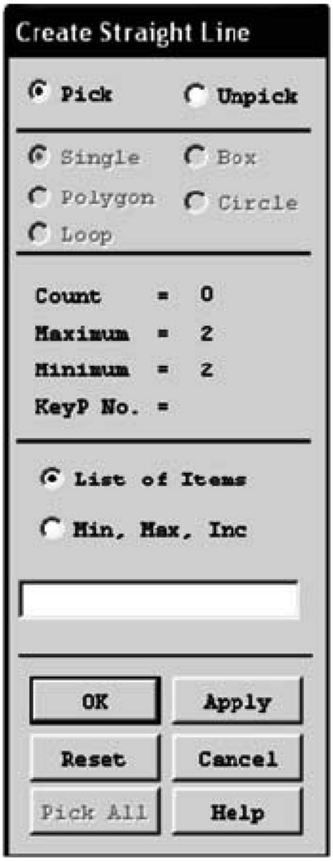


FIGURE 9.39 Pick the keypoints to create lines.

6. Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines > Enter NDIV No. of element divisions = 10

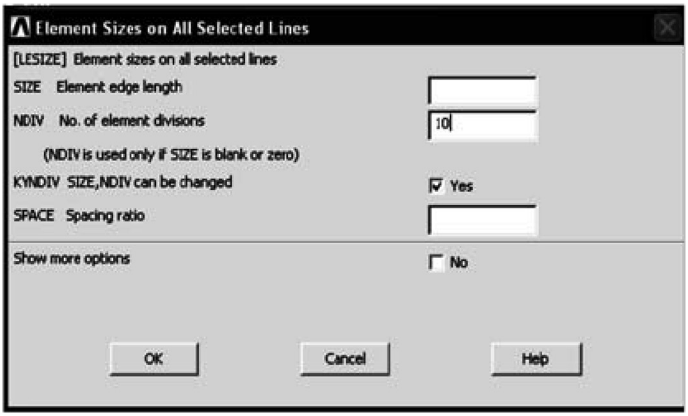


FIGURE 9.40 Specify element length.

7. Main Menu > Preprocessor > Meshing > Mesh > Lines > Click Pick All

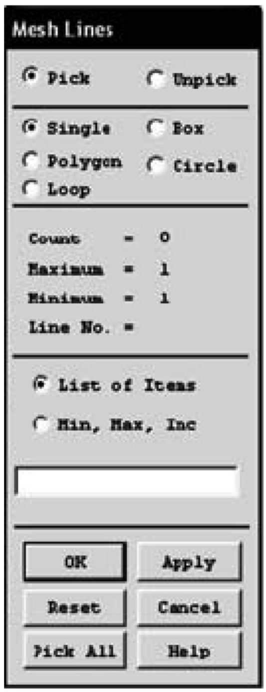


FIGURE 9.41 Create elements by meshing.

8. **Main Menu > Solution > Analysis Type > New Analysis > Select Modal > OK**

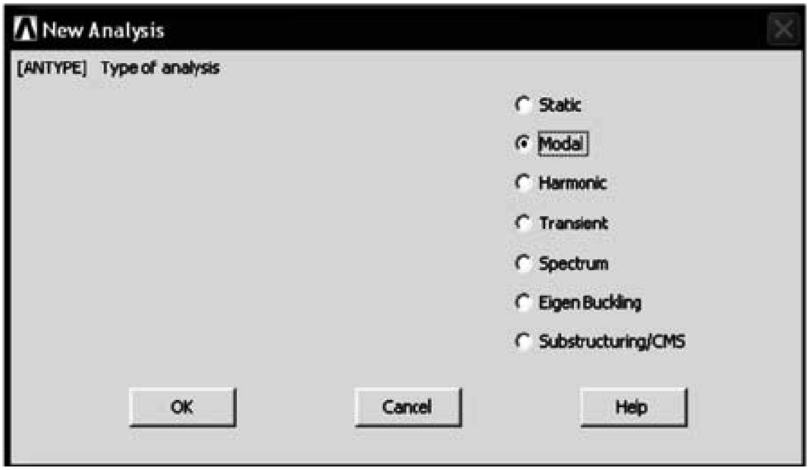


FIGURE 9.42 Define the type of analysis.

9. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes** Pick the left most node **> Apply > Select All DOF > OK**

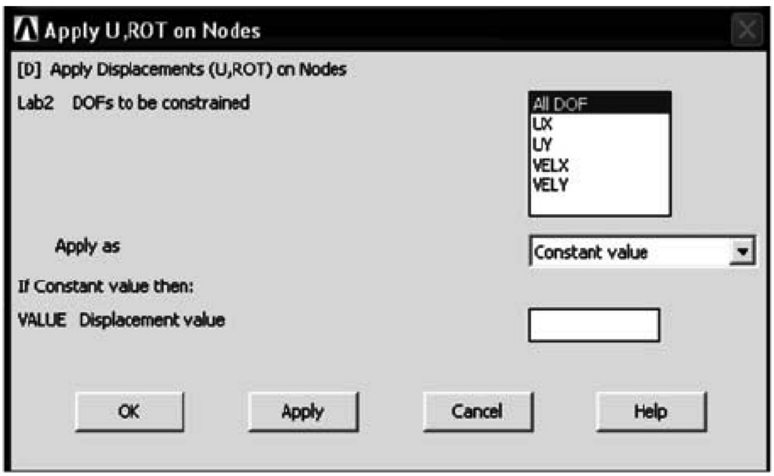


FIGURE 9.43 Apply the displacement constraint.

10. **Main Menu > Solution > Analysis Type > Analysis Options > Select Reduced option**

    Enter No. of modes to extract = 2

    NMODE No. of modes to expand = 2 > OK

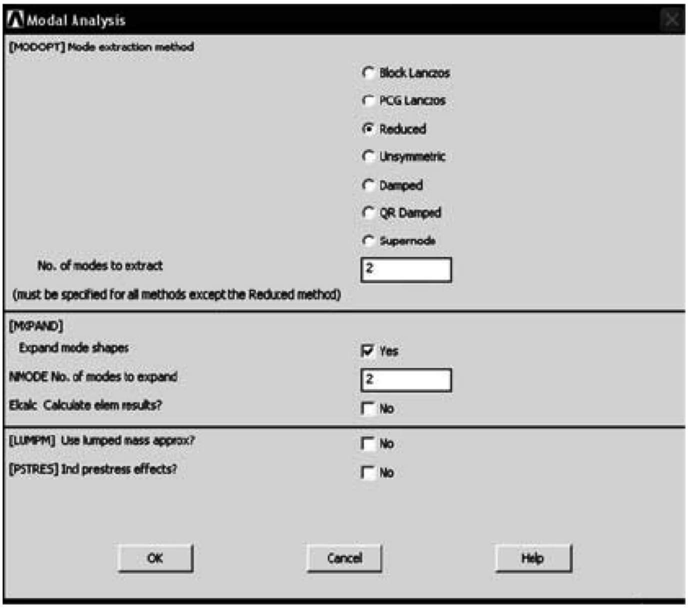


FIGURE 9.44 Select the number of modes to extract.

Enter FREQE Frequency range 0 2500 > OK

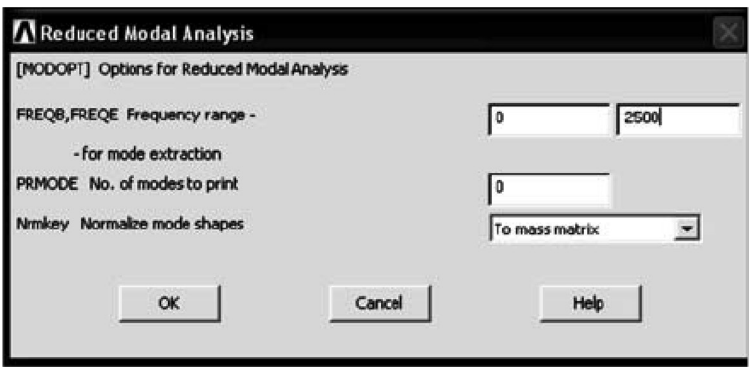


FIGURE 9.45 Enter the frequency range.

11. **Main Menu > Solution > Master DOFs > User Selected > Define > Pick all nodes except left most node > OK > Select UX from Lab 1 1st degree of freedom > OK**

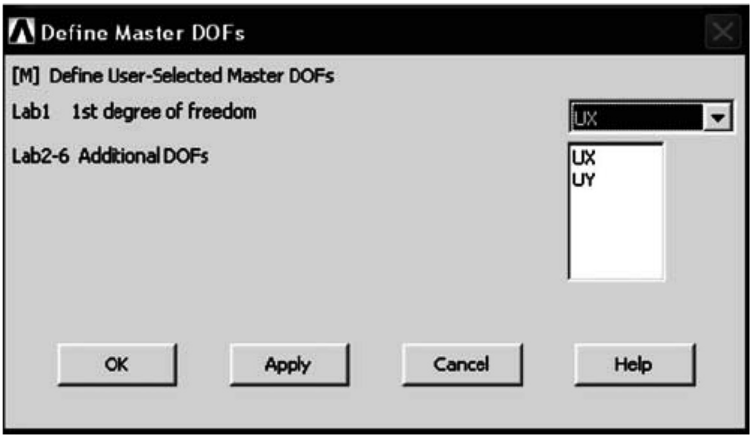


FIGURE 9.46 Defining the master degree of freedom.

FIGURE 9.47 Model with master DOF applied.

Solution

The interactive solution proceeds.

12. **Main Menu > Solution > Solve > Current LS > OK**

The **/STATUS Command** window displays the problem parameters and the **Solve Current Load Step** window is shown. Check the solution options in the /STATUS window and if all is OK, select **File > Close.**

In the **Solve Current Load Step** window, select **OK,** and when the solution is complete, **close** the "**Solution is Done!**" window.

POSTPROCESSING

13. **Main Menu > General Postproc > Results Summary**

This result is shown as frequency values in Hz.

14. Main Menu > General Postproc > Read Results > First Set

15. Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution, click on Displacement vector sum > OK

This result is shown in Figure 9.35(b).

9.7 BAR SUBJECTED TO FORCING FUNCTION

EXAMPLE 9.5

The bar shown in Figure 9.48 is subjected to a time-dependent forcing function, as shown, to determine the nodal displacements for five time steps using two finite elements. Let E = 207 GPa, Poisson's ratio = 0.3, length of beam L = 5 m, cross-section area $A = 625 \times 10^{-6}$ m^2, mass density ρ = 7800 kg/m^3. Use the time step of integration 0.00025 seconds.

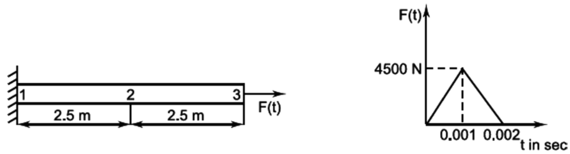


FIGURE 9.48 The bar for Example 9.5.

Solution

(I) Software results.

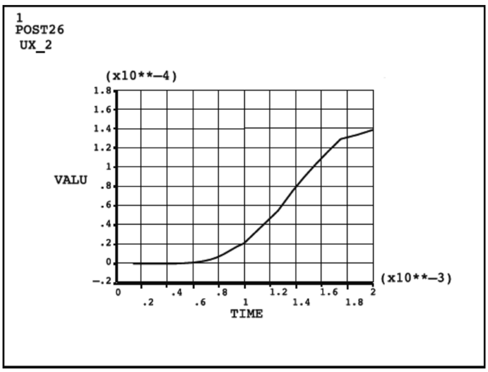


FIGURE 9.48(a) Displacement response for 0.00025 sec for node 2 (refer to Appendix D for color figures).

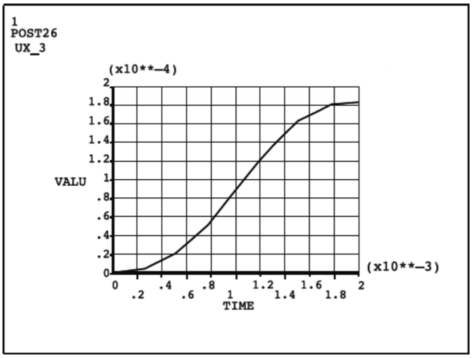


FIGURE 9.48(b) Displacement response for 0.00025 sec for node 3 (refer to Appendix D for color figures).

Displacement values (in meters) for node 2

TIME	2 UX
	UX_2
0.0000	0.00000
0.25000E–03	–0.467370E–06
0.50000E–03	–0.821457E–06
0.75000E–03	0.396081E–05
0.10000E–02	0.210563E–04
0.12500E–02	0.535055E–04
0.15000E–02	0.950064E–04
0.17500E–02	0.128841E–03
0.20000E–02	0.138387E–03

Displacement values (in meters) for node 3

TIME	2 UX
	UX_2
0.0000	0.00000
0.25000E–03	0.375512E–05
0.50000E–03	0.191517E–04
0.75000E–03	0.488709E–04
0.10000E–02	0.889759E–04

0.12500E–02	0.130597E–03
0.15000E–02	0.161991E–03
0.17500E–02	0.179673E–03
0.20000E–02	0.184097E–03

Maximum displacement values (in meters)

Name	Element	Node	Result Item	Minimum	Maximum	X-Axis
TIME			Time	0	0.002	
UX_2		2	X-Component of displacement	-6.21457e-007	0.000138387	
UX_3		3	X-Component of displacement	0	0.000184097	

FIGURE 9.48(c) Values of displacement.

EXERCISES

1. What is the governing equation of undamped free vibration and its non-trivial solution?
2. What are the mass matrices for bar elements and beam elements?
3. Determine the first five natural frequencies for the fixed-fixed beam shown in Figure 9.49. The beam is made of steel with E = 200 GPa, Poisson's ratio = 0.3, length = 2 m, cross-section area = 60 cm^2, mass density ρ = 7800 kg/m^3, moment of inertia I = 200 mm^4.

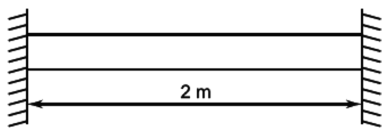


FIGURE 9.49 Fixed-fixed beam for Exercise 3.

4. For the bar shown in Figure 9.50, determine nodal displacements for the five time finite elements. Let E = 70 GPa, ρ = 2700 kg/m^3, A = 645 mm^2, and L = 2.5 m.

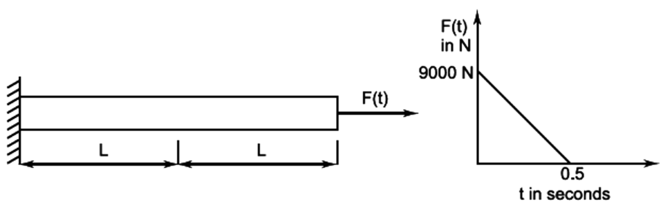


FIGURE 9.50 The bar for Exercise 4.

5. The beam shown in Figure 9.51 is subjected to the forcing functions shown, to determine the maximum deflections. Let E = 207 GPa, ρ = 7800 kg/m^3, A = 0.0194 m^2, I = 8.2 × 10^{-5} m^4, L = 6 m. Take time step of 0.05 seconds.

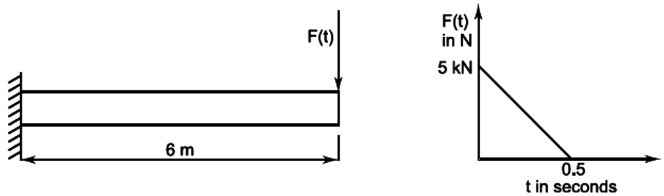


FIGURE 9.51 The beam for Exercise 5.

6. Determine the natural frequencies of vibrations for the cantilever beam shown in Figure 9.52.

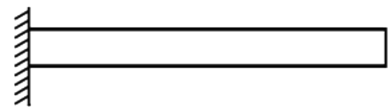


FIGURE 9.52 Cantilever beam for Exercise 6.

Hint: $[K]=\dfrac{EI}{L^3}\begin{bmatrix} 16 & -6L \\ -6L & 4L^2 \end{bmatrix}$, $[M]=\dfrac{\rho AL}{420}\begin{bmatrix} 156 & -22L \\ -22L & 4L^2 \end{bmatrix}$

7. For the bar shown in Figure 9.53, determine nodal displacements for the five time finite elements. Let E = 210 GPa, ρ = 2800 kg/m^3, A = 825 mm^2, and L = 3 m.

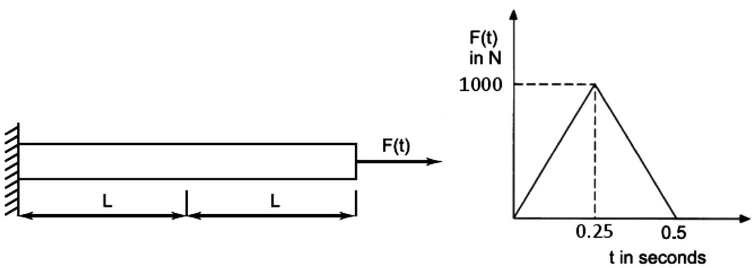


FIGURE 9.53 The bar for Exercise 7.

8. For the beam shown in Figure 9.54, determine the mode shapes. Let E = 310 × 10^6 psi, ρ = 0.283 lbf/in^3, A = 1 in^2, $\upsilon = 0.3$, and L = 30 in.

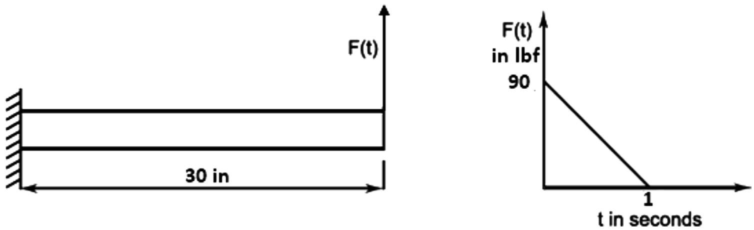


FIGURE 9.54 The beam for Exercise 8.

9. For the bar shown in Figure 9.55, subjected to the forcing functions shown, determine the nodal displacement, velocities, acceleration, and maximum deflections for five time steps using two finite elements. Let $E = 2 \times 10^6$ psi, $\rho = 2$ lb-s^2/in^4, A = 2 in^2, I = 322.83 in^4, L = 10 in.

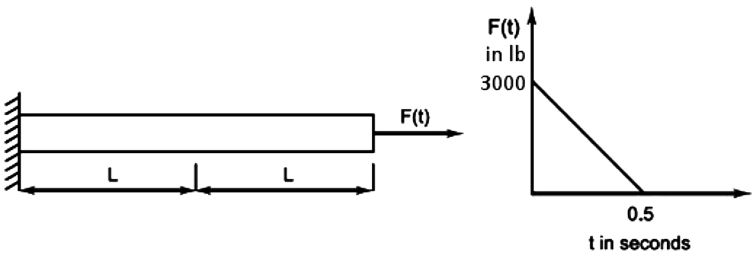


FIGURE 9.55 The bar for Exercise 9.

REFERENCES

1. D. L. Logan, "A First Course in the Finite Element Method," Fifth Edition, Cengage Learning, Boston, Massachusetts, 2012.
2. S. Moaveni, "Finite Element Analysis: Theory and Application with ANSYS," Third Edition, Prentice-Hall, New Jersey, 2008.
3. C. T. F. Ross, "Finite Element Method in Structural Mechanics," Ellis Horwood Limited Publishers, 1985.
4. S. S. Rao, "The Finite Element Method in Engineering," Fifth Edition, Butterworth-Heinemann, Oxford, 2011.
5. G. L. Narasaiah, "Finite Element Analysis," CRC Press, Florida, 2009.
6. W. T. Thompson and M. D. Dahleh, "Theory of Vibrations with Applications," Fifth Edition, Prentice-Hall, New Jersey, 1998.

CHAPTER 10

ENGINEERING ELECTROMAGNETICS ANALYSIS

10.1 INTRODUCTION TO ELECTROMAGNETICS

Electromagnetics (EM) governs many applications in engineering such as the transmission lines system. Therefore, it is essential to understand the fundamental concepts of EM in order to properly design and model electrical systems and devices using the finite element method (FEM). Furthermore, EM has become more useful in designing engineering systems with recent technologies, especially due to the increasing speeds of digital devices and the increased use of modern electronics circuits such as printed-circuit-board and communications systems such as cellular phones. The most important equations in EM theory are Maxwell's equations, which are known as the foundation of EM theory.

10.2 MAXWELL'S EQUATIONS AND CONTINUITY EQUATION

In electromagnetic analysis on a macroscopic level, it is based on solving the *Maxwell's equations* issue on certain boundary conditions. Also, there is another fundamental equation that can specify the conservation (indestructibility) of electric charge known as the *equation of continuity.* Maxwell's equations and continuity equation can be written in both differential and integral forms. We choose to start here with the differential form because it leads to differential equations that the FEM can handle.

10.2.1 Maxwell's Equations and Continuity Equation in Differential Form

Now, we can present the four Maxwell's equations in differential form in time-varying EM fields as:

$$\nabla \times \mathbf{H} = \mathbf{J}_e + \frac{\partial \mathbf{D}}{\partial t} \text{ (Ampere's law)} \tag{10.1}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} - \mathbf{J}_m \text{ (Faraday's law of induction)} \tag{10.2}$$

$$\nabla \cdot \mathbf{D} = \rho_v \text{ (Gauss's law-for electric field)} \tag{10.3}$$

$$\nabla \cdot \mathbf{B} = 0 \text{ (Gauss's law-for magnetic field)} \tag{10.4}$$

where

$\mathbf{E}$ = Electric field intensity, (in volt/meter) –V/m^2

$\mathbf{D}$ = Electric flux density (or electric displacement), (in coulomb/meter^2) –C/m^2

$\mathbf{H}$ = Magnetic field intensity, (in ampere/meter) –A/m^2

$\mathbf{B}$ = Magnetic flux density, (in tesla or weber/meter^2)–T or Wb/m^2

$\mathbf{J}_e$ = Electric Current density or charge flux (surface), (in ampere/meter^2) –A/m^2

$\mathbf{J}_m$ = The magnetic conductive current density, (in volt/meter^2) –V/m^2, where

$$\mathbf{J}_m = \sigma_m \mathbf{H}$$

σ_m = The magnetic conductive resistivity (in ohm/meter) –Ω/m

ρ_v = Electric charge density (volume), (in coulomb/meter^2) –C/m^2.

Now, the *equation of continuity* can be written in differential form as

$$\nabla \cdot \mathbf{J}_e = -\frac{\partial \rho_v}{\partial t} \text{ (Continuity equation).} \tag{10.5}$$

There are three *independent* equations from the above five equations. They are either equations 1, 2, and 3, or equations 1, 2, and 5. The other two equations 4 and 5, or equations 3 and 4 can be derived from the independent equations, and therefore are called *dependent equations.* Additionally, equation 5 can be derived the divergence of equation 1 and using equation 3.

10.2.2 Maxwell's Equations and Continuity Equation in Integral Form

Furthermore, let us now look to the four Maxwell's equations and the continuity equation in integral form in time-varying EM fields. The integrals are taken over in an open surface S or its boundary contour L as shown in Figure 10.1, where I is the electric current that flows through the path L.

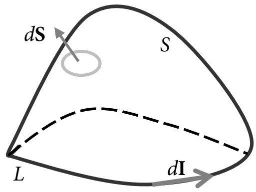


FIGURE 10.1 The surface S and contour L for the integral form of Maxwell's equations.

$$\oint_L \mathbf{H} \cdot d\mathbf{l} = \int_S \left(\mathbf{J}_e + \frac{\partial \mathbf{D}}{\partial t} \right) \cdot d\mathbf{S} \text{ (Ampere's law)} \tag{10.6}$$

$$\oint_L \mathbf{E} \cdot d\mathbf{l} = -\int_S \left(\frac{\partial \mathbf{B}}{\partial t} + \mathbf{J}_m \right) \cdot d\mathbf{S} \text{ (Faraday's law of induction)} \tag{10.7}$$

$$\oint_S \mathbf{D} \cdot d\mathbf{S} = \oint_v \rho_v dv \text{ (Gauss's law-for electric field)} \tag{10.8}$$

$$\oint_S \mathbf{B} \cdot d\mathbf{S} = 0 \text{ (Gauss's law-for magnetic field)} \tag{10.9}$$

$$-\int_S \mathbf{J}_e \cdot d\mathbf{S} = \frac{\partial}{\partial t} \int_v \rho_v dv \text{ (Continuity equation)} \tag{10.10}$$

where the surface S encloses the volume v, while the contour L encloses the surface$S$. $\mathbf{l}$ is the line vector over the contour L and $\mathbf{S}$ is the surface vector. Note that, the direction of $d\mathbf{l}$ must be consistent with the direction of the $d\mathbf{S}$ in agreement with the right-hand rule.

10.2.3 Divergence and Stokes Theorems

Indeed, equations 6 through 10, the integral forms can be derived from the differential forms or vice versa. This can be done by using either divergence (Gauss's) theorem or Stokes' theorem,

$$\oint_S \mathbf{F} \cdot d\mathbf{S} = \int_v \nabla \cdot \mathbf{F} dv \text{ (Divergence theorem)} \tag{10.11}$$

$$\oint_L \mathbf{F} \cdot d\mathbf{l} = \int_v \nabla \times \mathbf{F} \cdot d\mathbf{S} \text{ (Stokes theorem),} \tag{10.12}$$

where $\mathbf{F}$ is any arbitrary vector field.

10.2.4 Maxwell's Equations and Continuity Equation in Quasi-Statics Case

So far, we have done Maxwell's equations in a fully dynamic case. Now, we can express Maxwell's equations in quasi-statics case in which the displacement current ($\mathbf{D}$) is neglected. That is,

$$\nabla \times \mathbf{H} = \mathbf{J}_e. \tag{10.13}$$

Whereas equations (10.2), (10.3), and (10.4) remain the same. Also, we can write the continuity equation (10.5) in the quasi-statics case as

$$\nabla \cdot \mathbf{J}_e = 0. \tag{10.14}$$

Indeed, the quasi-static approximation is mainly used for time-varying fields in various conducting media. This is due to the fact that, for good conductors, the conduction current greatly exceeds the displacement current, $\mathbf{D}$, for the frequencies.

10.2.5 Maxwell's Equations and Continuity Equation in Statics Case

In the statics field case, the current displacement term ($\frac{\partial \mathbf{D}}{\partial t}$) and the time-varying magnetic flux density term ($\frac{\partial \mathbf{B}}{\partial t}$) are neglected (the field quantities do not vary with time). Therefore, Maxwell's equations in static form are expressed as

$$\nabla \times \mathbf{E} = 0. \tag{10.15}$$

Whereas equations (10.3), (10.4), and (10.13) still hold. Also, the continuity equation (10.14) remains the same.

To emphasize, there is no interaction between the electric and the magnetic fields. Thus, the static case can be divided into two separate cases, *electrostatic case* and *magnetostatic case*.

In electrostatic case, it can be described by equations (10.3) and (10.15), while for magnetostatic case, it can be described by equations (10.4) and (10.13).

10.2.6 Maxwell's Equations and Continuity Equation in Source-Free Regions of Space Case

The sources of the electromagnetic fields can be the volume charge density (ρ_v) and the electric current density ($\mathbf{J}_e$). In fact, these densities are localized in space. Also, these sources can make the generated electric and magnetic fields to radiate away from them, and they can make the generated electric and magnetic fields to propagate to larger distances to the receiving destination. Therefore, Maxwell's equations can be written in *source-free* regions of space (away from the source) as:

$$\nabla \times \mathbf{H} = \frac{\partial \mathbf{D}}{\partial t} \tag{10.16}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \tag{10.17}$$

$$\nabla \cdot \mathbf{D} = 0 \tag{10.18}$$

whereas equation (10.4) remains the same. With this in mind, the continuity equation (10.14) also remains the same.

10.2.7 Maxwell's Equations and Continuity Equation in Time-Harmonic Fields Case

So far, we considered the arbitrary time variation of electromagnetic fields. Here, we consider only the steady-state (equilibrium) solution of electromagnetic fields when produced by sinusoidal currents. The time-harmonic (sinusoidal steady-state) field for Maxwell's equations exists when the field quantities in the equations are harmonically oscillating functions with a single sinusoidal frequency ω. The time-harmonic fields case is the most regularly used in electrical engineering. Now, an arbitrary time-dependent field $\mathbf{F}(x,y,z,t)$ or $\mathbf{F}(\mathbf{r},t)$ can be written as

$$\mathbf{F}(\mathbf{r},t) = \mathrm{Re}\left(\mathbf{F}_s(\mathbf{r})e^{jwt}\right) \tag{10.19}$$

where e^{jwt} is the time convention, ω is the angular frequency (rad/s) of the sinusoidal excitation, $\mathbf{F}_s(\mathbf{r}) = \mathbf{F}_s(x,y,z)$ is the phasor form of $\mathbf{F}(\mathbf{r},t)$, and it is in general complex, and Re( ) indicates taking the real part of quantity in the parenthesis. Furthermore, the electromagnetic field quantities can be expressed in phasor notation as

$$\begin{bmatrix} \mathbf{H}(\mathbf{r},t) \\ \mathbf{E}(\mathbf{r},t) \\ \mathbf{D}(\mathbf{r},t) \\ \mathbf{B}(\mathbf{r},t) \end{bmatrix} = \begin{bmatrix} \mathbf{H}(\mathbf{r}) \\ \mathbf{E}(\mathbf{r}) \\ \mathbf{D}(\mathbf{r}) \\ \mathbf{B}(\mathbf{r}) \end{bmatrix} e^{jwt}. \tag{10.20}$$

For example, the fields can be expresses in time-dependent e^{jwt}, as in equation (10.20), $\mathbf{H}(\mathbf{r},t) = \mathbf{H}(\mathbf{r})e^{jwt}$ and $\mathbf{E}(\mathbf{r},t) = \mathbf{E}(\mathbf{r})e^{jwt}$, etc.

As a result, using the phasor representation can allow us to replace the time derivations $\frac{\partial}{\partial t}$ by $j\omega$ because

$$\frac{\partial e^{j\omega t}}{\partial t} = j\omega e^{j\omega t}. \tag{10.21}$$

Therefore, Maxwell's equations can be expressed in time-harmonic as

$$\nabla \times \mathbf{H}_s = \mathbf{J}_{es} + j\omega \mathbf{D}_s \tag{10.22}$$

$$\nabla \times \mathbf{E}_s = -\frac{\partial \mathbf{B}_s}{\partial t} - \mathbf{J}_{ms} \tag{10.23}$$

$$\nabla \cdot \mathbf{D}_s = \rho_{vs} \tag{10.24}$$

$$\nabla \cdot \mathbf{B}_s = 0. \tag{10.25}$$

Now, the continuity equation can be presented as

$$\nabla \cdot \mathbf{J}_{es} = -j\omega \rho_{vs}. \tag{10.26}$$

On the other hand, a non-sinusoidal field can be presented as

$$\mathbf{F}(\mathbf{r},t) = \mathrm{Re}\left(\int_{-\infty}^{\infty} \mathbf{F}_s(\mathbf{r},t) e^{j\omega t} d\omega \right). \tag{10.27}$$

Therefore, the solutions to Maxwell's equations for a non-sinusoidal field can be found by assuming that all the Fourier components $\mathbf{F}_s(\mathbf{r},\omega)$ over ω.

10.3 LORENTZ FORCE LAW AND CONTINUITY EQUATION

The Lorentz Force **F** is the force on a charge q with a vector velocity **u** in the present electric filed **E** and magnetic field **B** and can be obtained as

$$\mathbf{F} = q(\mathbf{E} + \mathbf{u} \times \mathbf{B}). \tag{10.28}$$

In addition, the volume charge ρ_v and the current distribution **J** can be subjected to the forces in the presence of fields. Thus, Lorentz Force **F** per unit volume acting on the volume charge and the current distribution can be expressed as

$$\mathbf{F} = \rho_v \mathbf{E} + \mathbf{J} \times \mathbf{B}. \tag{10.29}$$

However, if the current distribution **J** occurs from the motion of the charges q within the volume charge ρ_v, then the current distribution **J** can be formed as $\mathbf{J} = \rho_v \mathbf{v}$. This can make the Lorentz Force **F** as

$$\mathbf{F} = \rho_v (\mathbf{E} + \mathbf{v} \times \mathbf{B}). \tag{10.30}$$

Moreover, the Lorentz Force law is essential to understand the interaction between EM fields and matter. Indeed, the law is used in the design of many electrical devices.

Furthermore, the continuity equation which expresses the conservation of electric charge can be written as

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho_v}{\partial t}. \tag{10.31}$$

Equation (10.31) is implicit in Maxell's equations.

10.4 CONSTITUTIVE RELATIONS

In addition to Maxwell's equations and the continuity equation, there are constitutive relations that describe the macroscopic properties of the medium in which the fields exist. In other words, constitutive relations describe the relationship between the EM fields through the properties of the medium.

Indeed, Maxwell's equations and constitutive relations are used to obtain the solutions of EM fields that exist in any microwave structure. The constitutive relations can be presented in vacuum (free space) as

$$\mathbf{D} = \varepsilon_0 \mathbf{E} \tag{10.32}$$

$$\mathbf{B} = \mu_0 \mathbf{H} \tag{10.33}$$

$$\mathbf{J}_e = \sigma_e \mathbf{E} \tag{10.34}$$

$$\mathbf{J}_m = \sigma_m \mathbf{M} \tag{10.35}$$

where

ε_0 = the permittivity of vacuum

μ_0 = the permeability of vacuum

σ_e = the electrical conductivity

$\mathbf{M}$ = magnetization field.

The numerical values of ε_0 and μ_0 are written as

$$\varepsilon_0 = 8.854 \times 10^{-12}\, Farad/m \cong \frac{1}{36\pi} \times 10^{-9}\, F/m,$$

$$\mu_0 = 12.6 \times 10^{-7}\, Henry/m = 4\pi \times 10^{-7}\, H/m. \tag{10.36}$$

We can use these two quantities to define the *speed of light* (c_0) and the *characteristic impedances in vacuum* (η_0) as:

$$c_0 = \frac{1}{\sqrt{\varepsilon_0 \mu_0}} = 3 \times 10^8\, m/\text{sec.}, \quad \eta_0 = \sqrt{\frac{\mu_0}{\varepsilon_0}} = 377\Omega. \tag{10.37}$$

To emphasis, the constitutive relations are needed to solve for EM field quantities using Maxwell's equations.

For simple homogenous isotropic dielectric and for magnetic material (linear and isotropic media), the constitutive relations are given as

$$\mathbf{D} = \varepsilon \mathbf{E} \tag{10.38}$$

$$\mathbf{B} = \mu \mathbf{E} \tag{10.39}$$

Whereas equations (10.34) and (10.35) remain the same.

where ε is the permittivity of the material, and μ is the permeability of the material.

For inhomogeneous media, the constitutive relations are functions of the position.

The permittivity of the material ε and the permeability of the material μ can be presented as

$$\varepsilon = \varepsilon_0(1+\chi_e)$$

$$\mu = \mu_0(1+\chi_m) \tag{10.40}$$

where χ_e is the electric susceptibility of the material, which is the measure of the electric polarization property of the material (dimensionless scalar), and χ_m is the magnetic susceptibility of the material, which is the measure of the magnetic polarization property of the material (dimensionless scalar).

Moreover, the speed of light in the material c and the characteristic impedance of the material η is expressed as

$$c = \frac{1}{\sqrt{\varepsilon\mu}},\ \eta = \sqrt{\frac{\mu}{\varepsilon}}. \tag{10.41}$$

The relative permittivity ε_r of a material, the relative permeability μ_r of a material, and the refractive index n of a material are formed as

$$\varepsilon_r = \frac{\varepsilon}{\varepsilon_0} = 1+\chi_e,\ \mu_r = \frac{\mu}{\mu_0} = 1+\chi_m,\ n = \sqrt{\varepsilon_r\mu_r} \rightarrow n^2 = \varepsilon_r\mu_r. \tag{10.42}$$

By using equations (10.41) and (10.42), we get

$$c = \frac{c_0}{n} \text{ and } \eta = \frac{\eta_0 n}{\varepsilon_r}. \tag{10.43}$$

It is good to know that for nonmagnetic material $\mu_r = 1$ or $\mu_r = \mu_0$, and $\eta = \frac{\eta_0}{n}$.

Now, the constitutive relations for time-harmonic fields in simple media are:

$$\mathbf{D} = \varepsilon_0 \varepsilon_r(\omega)\mathbf{E} = \varepsilon(\omega)\mathbf{E} \tag{10.44}$$

$$\mathbf{B} = \mu_0 \mu_r(\omega)\mathbf{H} = \mu(\omega)\mathbf{H} \tag{10.45}$$

$$\mathbf{J}_e = \sigma_e(\omega)\mathbf{E}. \tag{10.46}$$

Furthermore, both the electric polarization **P** (Coulomb/m^2), which describes how the material is polarized when an electric field **E** is present, and the magnetization **M** (Ampere/m), which describes how the material is magnetized when a magnetic field **H** can be included in the constitutive relations in any material as

$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P} \tag{10.47}$$

$$\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M}) \tag{10.48}$$

$$\mathbf{J}_e = \sigma_e \mathbf{E} \tag{10.49}$$

$$\mathbf{J}_m = \sigma_m \mathbf{M} \tag{10.50}$$

where $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$ and $\mathbf{M} = \chi_m \mathbf{H}$.

Next, for nonlinear material, the constitutive relationships can be presented as

$$\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_{re} \tag{10.51}$$

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_{re} \tag{10.52}$$

$$\mathbf{J}_e = \sigma_e \mathbf{E} + \mathbf{J}_{ex} \tag{10.53}$$

where $\mathbf{D}_{re}$ is the remanent displacement that is the displacement when the electric field is not present, $\mathbf{B}_{re}$ is the remanent magnetic flux density that is the magnetic flux density when the magnetic field is not present, and $\mathbf{J}_{ex}$ is an externally generated current.

It is beneficial to know that Maxwell's equations can be expressed in an approach that ensures the contribution of the medium in terms of the fields **E** and **B** as

$$\nabla \times \mathbf{B} = \varepsilon_0 \mu_0 \frac{\partial \mathbf{E}}{\partial t} + \mu_0 \left(\mathbf{J} + \frac{\partial \mathbf{P}}{\partial t} + \nabla \times \mathbf{M} \right) \tag{10.54}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \tag{10.55}$$

$$\nabla \cdot \mathbf{E} = \frac{1}{\varepsilon_0} \left(\rho_v - \nabla \cdot \mathbf{P} \right) \tag{10.56}$$

$$\nabla \cdot \mathbf{B} = 0. \tag{10.57}$$

EXAMPLE 10.1

Given $\mathbf{H} = He^{j(\omega t + \beta z)}\mathbf{a}_x$ in free space, calculate **E**.

Solution

We know $\mathbf{D} = \varepsilon \mathbf{E}$ and $\nabla \times \mathbf{H} = \frac{\partial \mathbf{D}}{\partial t}$, therefore

$$\frac{\partial \mathbf{D}}{\partial t} = \frac{\partial}{\partial z} He^{j(\omega t + \beta z)}\mathbf{a}_y$$

$$\frac{\partial \mathbf{D}}{\partial t} = j\beta He^{j(\omega t + \beta z)}\mathbf{a}_y$$

$$\mathbf{D} = \frac{\beta H}{\omega} e^{j(\omega t + \beta z)}\mathbf{a}_y$$

$$\mathbf{E} = \frac{\beta H}{\varepsilon \omega} e^{j(\omega t + \beta z)}\mathbf{a}_y.$$

10.5 POTENTIAL EQUATIONS

Often under certain circumstances, it can be essential to formulate EM problems in terms of potential functions, that is, the scalar electric potential V_e and vector magnetic potential **A**. These potential functions are arbitrary, and they are required to satisfy Maxwell's equations. They are described by

$$\mathbf{B} = \nabla \times \mathbf{A} \tag{10.58}$$

$$\mathbf{E} = -\nabla V_e - \frac{\partial \mathbf{A}}{\partial t}. \tag{10.59}$$

In fact, equation (10.55) is a direct consequence of the magnetic Gauss' law, and equation (10.55) results from Faraday's law. In the magnetostatic case (there are no currents present), Ampere's law reduces to

$$\nabla \times \mathbf{H} = 0. \tag{10.60}$$

Indeed, when equation (10.57) holds, we can present the scalar magnetic potential V_m by

$$\mathbf{H} = -\nabla V_m. \tag{10.61}$$

It is clear that equations (10.58) and (10.59) satisfy Maxwell's equations (1.2) and (1.4). Now, to relate the potential functions to the other two Maxwell's equations (1.1) and (1.3), by assuming the Lorentz condition holds, that is,

$$\nabla \cdot \mathbf{A} = -\varepsilon\mu \frac{\partial V_e}{\partial t}. \tag{10.62}$$

These equations can be written in the case of linear and homogenous medium as

$$\nabla^2 \cdot V_e - \varepsilon\mu \frac{\partial^2 V_e}{\partial t^2} = -\frac{\rho_v}{\varepsilon} \tag{10.63}$$

$$\nabla^2 \cdot \mathbf{A} - \varepsilon\mu \frac{\partial^2 \mathbf{A}}{\partial t^2} = -\mu \mathbf{J}. \tag{10.64}$$

Equations (10.63) and (10.64) as wave equations, and the integral solutions to these equations are known as the *retarded* potential solutions, i.e.,

$$V_e = \int \frac{[\rho_v]dv}{4\pi\varepsilon R} \tag{10.65}$$

$$\mathbf{A} = \int \frac{\mu[\mathbf{J}]dv}{4\pi R} \tag{10.66}$$

where R is the distance from the source point to the field point at which the potential is required, and the square brackets [ ] denote that ρ_v and $\mathbf{J}$ are specified at a time $R\sqrt{\varepsilon\mu}$ earlier than for which V_e or $\mathbf{A}$ is being formed.

10.6 BOUNDARY CONDITIONS

The material medium in which an electromagnetic field exists is usually characterized by its constitutive parameters σ, ε, and μ. If σ, ε, and μ are independent of **E** and **H**, the medium is *linear*. Also, if σ, ε, and μ are dependent of **E** and **H**, the medium is *nonlinear*. Now, if σ, ε, and μ are functions of space variables, the medium is *inhomogeneous*. But, if σ, ε, and μ are not functions of space variables, the medium is *homogeneous*. Additionally, if σ, ε, and μ are independent of direction (scalars), the medium is *isotropic*. If σ, ε, and μ are dependent on direction (vectors), the medium is *anisotropic*. Indeed, most of substrates used in electronic circuits are homogenous, isotropic, and linear.

The boundary conditions at the interface separating two different media 1 and 2, with parameters $(\varepsilon_1, \mu_1, \sigma_1)$ and $(\varepsilon_2, \mu_2, \sigma_2)$, respectively, as shown in Figure 10.2.

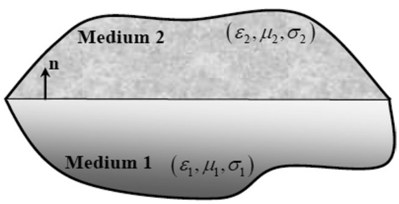


FIGURE 10.2 Interface between two media.

The boundary conditions for the EM fields across material boundaries are derived from the integral form of Maxwell's equations. They are given by

$$\mathbf{n}\times(\mathbf{E}_1-\mathbf{E}_2)=0 \text{ or } E_{1t}-E_{2t}=0 \tag{10.67}$$

$$\mathbf{n}\cdot(\mathbf{D}_1-\mathbf{D}_2)=\rho_s \text{ or } D_{1n}-D_{2n}=\rho_s \tag{10.68}$$

$$\mathbf{n}\times(\mathbf{H}_1-\mathbf{H}_2)=\mathbf{J}_s \text{ or } H_{1t}-H_{2t}=J_s \tag{10.69}$$

$$\mathbf{n}\cdot(\mathbf{B}_1-\mathbf{B}_2)=0 \text{ or } B_{1n}-B_{2n}=0 \tag{10.70}$$

where $\mathbf{n}$ is a unit normal vector directed from medium 1 to medium 2, subscript t and n denote tangent and normal components of the fields, respectively, ρ_s and $\mathbf{J}_s$ are surface electric charge density (coulomb/m^2) and surface current density (ampere/m), respectively. Furthermore, equations (10.67) and (10.70) state that the tangential components of **E** and the normal components of **B** are continuous across the boundary. But, equation (10.68) states that the discontinuity in the normal component **D** is the same as the surface electric charge density ρ_s on the boundary. However, equation (1.69) states that the tangential component of **H** is discontinuous by the surface current density $\mathbf{J}_s$ on the boundary. In many interface problems, only two of Maxwell's equations are used, equations (10.68) and (10.70), when a medium is source-free ($\mathbf{J}=0, \rho_v=0$), since the other two boundary conditions are implied. In such a case, the boundaries conditions may be written as

$$E_{1t}=-E_{2t} \tag{10.71}$$

$$D_{1n}=D_{2n} \tag{10.72}$$

$$H_{1t}=H_{2t} \tag{10.73}$$

$$B_{1n}=B_{2n}\,. \tag{10.74}$$

Moreover, Maxwell's equations under the source-free condition are applicable to passive microwave structures such as transmissions lines.

However, when one of the media is a perfect conductor, boundary conditions are different. A perfect conductor has infinite electrical conductivity and thus no internal electric field (full of free charges). Or else, it would produce an infinite current density according to the third constitutive relations. When an EM field is applied to a perfect conductor medium, the free charges which are pushed to the applied EM field, move themselves in such a way that they produce an opposite EM field that completely cancels the applied EM field. Indeed, this causes the creation of the surface charges and currents on the boundary of the perfect conductor. At an interface between a dielectric and a perfect conductor, the boundary conditions for **E** and **D** fields are simplified. Now, assume that medium 1 is a perfect conductor, then $\mathbf{E}_1 = 0$ and $\mathbf{D}_1 = 0$. Also, if it is a time-varying case, then $\mathbf{H}_1 = 0$ and $\mathbf{B}_1 = 0$, and, in addition, as a correspondence of Maxwell's equations. Therefore, the boundary conditions for the fields in the dielectric medium for the time-varying at the surface are

$$-\mathbf{n}\times\mathbf{E}_2 = 0 \tag{10.75}$$

$$-\mathbf{n}\cdot\mathbf{D}_2 = \rho_s \tag{10.76}$$

$$-\mathbf{n}\times\mathbf{H}_2 = \mathbf{J}_s \tag{10.77}$$

$$-\mathbf{n}\cdot\mathbf{B}_2 = 0. \tag{10.78}$$

Furthermore, we can apply the integral form of the continuity equation (10.10) to the surface at the interface between lossy media (i.e., $\sigma_1 \neq 0, \sigma_2 \neq 0$) or lossy dielectric (i.e., $\sigma_1 \neq \sigma_2$ and $\varepsilon_1 \neq \varepsilon_2$), or perfect conductor (i.e., no fields inside the media). Therefore, the interface condition for current density **J** can be obtained as

$$\mathbf{n}\cdot(\mathbf{J}_1 - \mathbf{J}_2) = -\frac{\partial \rho_s}{\partial t} \text{ or } (J_{1n} - J_{2n}) = -\frac{\partial \rho_s}{\partial t}. \tag{10.79}$$

Equation (10.79) states that the normal component **J** is continuous, except where the time-varying surface electric charge density ρ_s on the boundary may exist.

10.7 LAWS FOR STATIC FIELDS IN UNBOUNDED REGIONS

Coulomb's law and Biot-Savart's law are the two fundamental laws governing the static fields in unbounded regions.

10.7.1 Coulomb's Law and Field Intensity

Coulomb's law is an experimental law that deals with the force a point charge exerts on another point charge. In other words, Coulomb's law states that the force F (in newtons) between two points charges Q_1 (in coulombs) and Q_2 is

$$F = \frac{Q_1 Q_2}{4\pi\varepsilon_0 R^2} \tag{10.80}$$

where R (in meter) is the distance between the two charges. We can define the electrostatic field intensity $\mathbf{E}$ as the force F applied by 1 charge Q on a unit positive point charge as

$$\mathbf{E} = \frac{Q\mathbf{a}_R}{4\pi\varepsilon_0 R^2}. \tag{10.81}$$

Knowing that, the point at which the charge Q is located is called the source point, and the point at which the electrostatic field intensity $\mathbf{E}$ is taken is called the field point. Thus, here $\mathbf{a}_R$ is the unite vector in the direction from the source point toward the field point, and R is the distance between the source point and the field point.

Now, it is possible to obtain a continuous charge along a line, on a surface, or in a volume, respectively as

$$\mathbf{E} = \int_L \frac{\rho_l \mathbf{a}_R}{4\pi\varepsilon_0 R^2}\, dl \tag{10.82}$$

$$\mathbf{E} = \int_S \frac{\rho_s \mathbf{a}_R}{4\pi\varepsilon_0 R^2}\, dS \tag{10.83}$$

$$\mathbf{E} = \int_v \frac{\rho_v \mathbf{a}_R}{4\pi\varepsilon_0 R^2}\, dv, \tag{10.84}$$

where L is the line along which the charge is distributed, S is the surface on which the charge is distributed, v is the volume enclosed by a surface S. $\rho_l, \rho_s,$ and ρ_v, are the line, surface, and volume charge density, respectively.

10.7.2 Bio-Savart's Law and Field Intensity

Bio-Savart's law is a magnetostatic law used to express the static magnetic field as a summation over elementary current sources. Now, we can obtain the Bio-Savart law for the line current, surface current, and volume current, respectively, in terms of the distributed current sources as

$$\mathbf{H} = \int_L \frac{Id\mathbf{I}\times\mathbf{a}_R}{4\pi R^2} \tag{10.85}$$

$$\mathbf{H} = \int_S \frac{\mathbf{J}_s dS\times\mathbf{a}_R}{4\pi R^2} \tag{10.86}$$

$$\mathbf{H} = \int_v \frac{\mathbf{J}_v dv\times\mathbf{a}_R}{4\pi R^2},\quad \mathbf{J}_s ds \tag{10.87}$$

where $\mathbf{I}$ is the line current density, $\mathbf{J}_s$ is the surface charge density, $\mathbf{J}_v$ is the volume charge density, and $\mathbf{a}_R$ is a unit vector pointing from the differential elements of current to the point of interest. Indeed, the source elements are related as

$$Id\mathbf{I} \equiv \mathbf{J}_s ds \equiv \mathbf{J}_v dv. \tag{10.88}$$

10.8 ELECTROMAGNETIC ENERGY AND POWER FLOW

The electric energy W_e is defined as

$$W_e = \int_v \left(\int_0^D \mathbf{E}\cdot d\mathbf{D} \right) dv = \int_v \left(\int_0^T \mathbf{E}\cdot \frac{\partial \mathbf{D}}{\partial t} dt \right) dv \tag{10.89}$$

where D is the magnitude of electric displacement, and T is the period.

The electrostatic energy present in an assembly of charges can be written as

$$W_e = \frac{1}{2}\sum_{k=1}^{n} Q_k V_k \tag{10.90}$$

where V is the potential, and Q is the point charge. Now, instead of point charges, the region has a continuous charge distribution, and the summation equation (10.90) becomes integrations for line charge, surface charge, and volume charge, respectively, as

$$W_e = \frac{1}{2}\int_L \rho_l V dl \tag{10.91}$$

$$W_e = \frac{1}{2}\int_S \rho_s V dS \tag{10.92}$$

$$W_e = \frac{1}{2}\int_v \rho_v V dv. \tag{10.93}$$

In the meantime, $\rho_v = \nabla \cdot \mathbf{D}$, $\mathbf{E} = -\nabla V$, and $\mathbf{D} = \varepsilon_0 \mathbf{E}$, and by using the identity for vector and scalar rules and applying the divergence theorem, and knowing that in a simple medium whose constitutive parameters (μ, ε, and σ) do not change with time, we have

$$\mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} = \mathbf{E} \cdot \frac{\partial (\varepsilon \mathbf{E})}{\partial t} = \frac{1}{2}\frac{\partial (\varepsilon \mathbf{E} \cdot \mathbf{E})}{\partial t} = \frac{\partial}{\partial t}\left(\frac{1}{2}\varepsilon E^2\right). \tag{10.94}$$

We can obtain electrostatic energy as

$$W_e = \frac{1}{2}\int_v \mathbf{D} \cdot \mathbf{E}\, dv = \frac{1}{2}\int_v \varepsilon_0 \mathrm{E}^2 \, dv. \tag{10.95}$$

Also, the electrostatic energy density w_e (in J/m^2) can be obtained as

$$w_e = \frac{1}{2}\mathbf{D} \cdot \mathbf{E} = \frac{1}{2}\varepsilon_0 E^2 = \frac{D^2}{2\varepsilon_0}. \tag{10.96}$$

When a wave propagates in a medium, it carries the electric field and power. However, the time derivative of equation (10.89) is the electric power which is written as

$$P_e = \int_v \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dv. \tag{10.97}$$

Furthermore, the magnetic energy can be defined as

$$W_m = \int_v \left(\int_0^B \mathbf{H} \cdot d\mathbf{B} \right) dv = \int_v \left(\int_0^T \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dt \right) dv, \tag{10.98}$$

where B is the magnitude of magnetic flux density, and T is the period.

The magnetostatic energy present in an assembly of currents k can be written as

$$W_m = \frac{1}{2} \sum_{k=1}^{n} I_k \Phi_k \tag{10.99}$$

where I_k is the kth current, and $\mathbf{\Phi}_k$ is kth magnetic flux.

Note, knowing that in a simple medium, whose constitutive parameters (μ, ε, and σ) do not change with time, we have

$$\mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} = \mathbf{H} \cdot \frac{\partial (\mu \mathbf{H})}{\partial t} = \frac{1}{2} \frac{\partial (\mu \mathbf{H} \cdot \mathbf{H})}{\partial t} = \frac{\partial}{\partial t} \left(\frac{1}{2} \mu H^2 \right). \tag{10.100}$$

We can obtain magnetostatic energy as

$$W_e = \frac{1}{2} \int_v \mathbf{B} \cdot \mathbf{H} \, dv = \frac{1}{2} \int_v \mu H^2 \, dv. \tag{10.101}$$

Also, the magnetostatic energy density w_m (in J/m^2) can be obtained as

$$w_m = \frac{1}{2} \mathbf{B} \cdot \mathbf{H} = \frac{1}{2} \mu H^2 = \frac{B^2}{2\mu}. \tag{10.102}$$

When a wave propagates in a medium, it carries the magnetic field and power. However, the time derivatives of equation (10.98) are the magnetic power that is written as

$$P_m = \int_v \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dv. \tag{10.103}$$

The instantaneous power density vector associated with the electromagnetic field at a given point is known as the *Poynting vector* $\mathbf{P}_{ov}$ (in W/m^2), which is written as

$$\mathbf{P}_{ov} = \mathbf{E}\times\mathbf{H}. \tag{10.104}$$

For more practical value than $\mathbf{P}_{ov}$, we determine the time-average instantaneous Poynting vector (or power average density) (in W/m^2) over the period $T = \frac{2\pi}{\omega}$ as

$$\mathbf{P}_{ov\text{-}ave}(z) = \frac{1}{T}\int_0^T P_{ov}(z,t)\,dt. \tag{10.105}$$

In addition, for time-harmonic fields, we can define a phasor *Poynting vector* as

$$\mathbf{P}_{ovs} = \mathbf{E}_s \times \mathbf{H}_s^* \tag{10.106}$$

where $\mathbf{H}_s^*$ is the complex conjugate of $\mathbf{H}_s$. Now, for a phasor *Poynting vector*, we can define the time-average power, which is equivalent to equation (10.106) as

$$\mathbf{P}_{ov\text{-}ave}(z) = \frac{1}{2}\mathrm{Re}\left(\mathbf{E}_s \times \mathbf{H}_s^*\right) \tag{10.107}$$

where Re( ) stands for the real part of a complex quantity. Furthermore, the total time-average power crossing a given surface S is given by

$$\mathbf{P}_{tave} = \frac{1}{2}\mathrm{Re}\int_S (\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S} = \int_S \mathbf{P}_{ov-ave}\cdot d\mathbf{S}. \tag{10.108}$$

The electric and magnetic power quantities are related through *Poynting's theorem* as

$$-\int_v\left(\mathbf{E}\cdot\frac{\partial\mathbf{D}}{\partial t} + \mathbf{H}\cdot\frac{\partial\mathbf{B}}{\partial t}\right)dv = \int_v \mathbf{J}\cdot\mathbf{E}dv + \oint_S(\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S} \quad \text{(10.109)-105}$$

where

$\int_v \mathbf{J}\cdot\mathbf{E}dv$ is called *resistive losses,* which result in heat dissipation in the material.

$\oint_S(\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S}$ is called the *radiative losses*.

However, *Poynting's theorem* as presented in equation (10.109), can be written as

$$\oint_S (\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S} = -\frac{\partial}{\partial t}\int_v \left(\frac{1}{2}\varepsilon E^2 + \frac{1}{2}\mu H^2\right)dv - \int_v \sigma E^2 dv \tag{10.110}$$

where

$\oint_S (\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S}$ is the total power leaving the volume.

$-\frac{\partial}{\partial t}\int_v \left(\frac{1}{2}\varepsilon E^2 + \frac{1}{2}\mu H^2\right)dv$ is the rate of decrease in energy stored in electric and magnetic fields.

$-\int_v \sigma E^2 dv$ is the decrease in ohmic power density (dissipated).

Indeed, under the material is linear and isotropic, it holds that.

$$\mathbf{E}\cdot\frac{\partial \mathbf{D}}{\partial t} = \mathbf{E}\cdot\frac{\partial(\varepsilon\mathbf{E})}{\partial t} = \frac{1}{2}\frac{\partial(\varepsilon\mathbf{E}\cdot\mathbf{E})}{\partial t} \tag{10.111}$$

$$\mathbf{H}\cdot\frac{\partial \mathbf{B}}{\partial t} = \frac{1}{\mu}\mathbf{B}\cdot\frac{\partial \mathbf{B}}{\partial t} = \frac{\partial}{\partial t}\left(\frac{1}{2\mu}\mathbf{B}\cdot\mathbf{B}\right). \tag{10.112}$$

Therefore, based on equations (10.111) and (10.112), equation (10.109) can be written as

$$-\frac{\partial}{\partial t}\int_v \left(\frac{1}{2}\varepsilon\mathbf{E}\cdot\mathbf{E} + \frac{1}{2\mu}\mathbf{B}\cdot\mathbf{B}\right)dv = \int_v \mathbf{J}\cdot\mathbf{E}dv + \oint_S (\mathbf{E}\times\mathbf{H})\cdot d\mathbf{S}. \tag{10.113}$$

Now, by integrating the left-hand side of equation (10.113) is the total electromagnetic energy density w_t

$$w_t = w_e + w_m = \frac{1}{2}\left(\varepsilon\mathbf{E}\cdot\mathbf{E} + \frac{1}{\mu}\mathbf{B}\cdot\mathbf{B}\right). \tag{10.114}$$

10.9 LOSS IN MEDIUM

The electronic circuits have dielectrics that are not always perfect. Thus, there is always loss in any practical nonmagnetic dielectrics, known as *dielectric loss*. This dielectric loss is due to a nonzero conductibility of the medium. Now, we can write the time-harmonic Maxwell's equation (10.22), making use of the time-harmonic constitutive relations (10.44) and (10.46), as

$$\nabla \times \mathbf{H}_s = j\omega\varepsilon \left(1 - j\frac{\sigma}{\omega\varepsilon}\right)\mathbf{E} \tag{10.115}$$

or

$$\nabla \times \mathbf{H}_s = j\omega\varepsilon \left(1 - j\tan\delta\right)\mathbf{E} \tag{10.116}$$

where

$$\tan\delta = \frac{\sigma}{\omega\varepsilon}. \tag{10.117}$$

Equation (10.117) is called the *loss tangent* of the medium, which is usually used to characterize the medium's loss. In addition, now we can define a complex dielectric constant of a lossy medium $\hat{\varepsilon}$ as

$$\hat{\varepsilon} = \varepsilon' - j\varepsilon'' \tag{10.118}$$

where the real part ε' of the complex dielectric constant is the dielectric property that contributes to the stored electric energy in the medium, and it is defined as

$$\varepsilon' = \varepsilon = \varepsilon_0 \varepsilon_r \tag{10.119}$$

and the imaginary part ε'' contains the finite conductivity and results in loss in the medium, which is defined as

$$\varepsilon'' = \frac{\sigma}{\omega} = \varepsilon \tan\delta. \tag{10.120}$$

For example, the loss tangent for GaAs material is 0.006 at frequency 10 Ghz, relative dielectric constant equal to 12.9, and temperature 25° C. Also, the loss tangent for silicon material is 0.004 at frequency 10Ghz, relative dielectric constant equal to 11.9, and temperature 25° C.

10.10 SKIN DEPTH

The measure of the depth to which the electromagnetic wave can penetrate the medium is known as *skin depth* (or *depth of penetration*). Skin depth is one of the most important parameters of a medium because it presents the distance from the medium surface over which the magnitude of the fields of a wave traveling in the medium is reduced to e^{-1} (or 0.368) of those at the medium's surface. The skin depth δ of a good conductor is approximately written as

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}} = \frac{1}{\sqrt{\pi f \mu\sigma}} \tag{10.121}$$

where $\omega = 2\pi f$.

It is essential to know that the skin depth of good conductors is very small, especially at high frequencies. Thus, it results in a low conduction loss.

EXAMPLE 10.2

Calculate the skin depth, δ, for aluminum in a 1.6 $\times 10^6$ Hz field ($\sigma = 38.2\times10^6 S/m$ and $\mu = 1$).

Solution

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}} = \frac{1}{\sqrt{\pi f \mu\sigma}} = \frac{1}{\sqrt{\pi\times1.6\times10^6\times1\times38.2\times10^6}} = 64.4\mu m.$$

10.11 POISSON'S AND LAPLACE'S EQUATIONS

Poisson's and Laplace's equations are derived from Gauss's law (for a linear, isotropic material medium)

$$\nabla\cdot\mathbf{D} = \nabla\cdot\varepsilon\,\mathbf{E} = \rho_v \tag{10.122}$$

and

$$\mathbf{E} = -\nabla V. \tag{10.123}$$

By substituting equation (10.123) into equation (10.122), we get

$$\nabla\cdot(-\varepsilon\nabla V) = \rho_v \tag{10.124}$$

for an inhomogeneous medium. Equation (10.124) can be obtained for a homogeneous medium as

$$\nabla^2 V = -\frac{\rho_v}{\varepsilon}. \tag{10.125}$$

Equation (10.125) is known as *Poisson's equation*.

Now, *Laplace's equation* is a special case of Poisson's equation when $\rho_v = 0$ (i.e., for a charge-free region), and it can be described as

$$\nabla^2 V = 0. \tag{10.126}$$

Laplace's equation is used to determine the static or quasi-static characteristic impedance and effective relative dielectric constant of a transmission line.

10.12 WAVE EQUATIONS

We have used so far Maxwell's equations and constitutive relations directly to determine the EM fields. However, it can be very convenient to obtain the EM fields by solving wave equations.

When the electromagnetic wave is in a simple (linear, isotropic, and homogenous) nonconducting medium (ε , μ , and $\sigma = 0$), the homogenous vector wave equations can be presented as

$$\nabla^2 \mathbf{E} - \frac{1}{c^2}\frac{\partial^2 \mathbf{E}}{\partial t^2} = 0 \tag{10.127}$$

and

$$\nabla^2 \mathbf{H} - \frac{1}{c^2}\frac{\partial^2 \mathbf{H}}{\partial t^2} = 0. \tag{10.128}$$

On the other hand, the relation between scalar potential V and vector potential $\mathbf{A}$ is called the Lorentz condition (or Lorentz gauge) for potentials that is expressed as

$$\nabla \cdot \mathbf{A} + \mu\varepsilon \frac{\partial V}{\partial t} = 0. \tag{10.129}$$

The non-homogenous wave equation for vector potential **A** is given by

$$\nabla^2 \mathbf{A} - \mu\varepsilon \frac{\partial^2 \mathbf{A}}{\partial t^2} = -\mu \mathbf{J}. \tag{10.130}$$

But, the non-homogenous wave equation for scalar potential V is given by

$$\nabla^2 V - \mu\varepsilon \frac{\partial^2 V}{\partial t^2} = -\frac{\rho}{\varepsilon}. \tag{10.131}$$

The time-harmonic wave equations for vector potential **A** and scalar potential V equations can be obtained, respectively as

$$\nabla^2 \mathbf{A} + k^2 \mathbf{A} = -\mu \mathbf{J} \tag{10.132}$$

and

$$\nabla^2 V + k^2 V = -\frac{\rho}{\varepsilon} \tag{10.133}$$

where

$$k = \omega\sqrt{\mu\varepsilon} = \frac{\omega}{c}. \tag{10.134}$$

Equation (10.134) is called *wave number*, and equations (10.132) and (10.133) are known as *non-homogenous Helmholtz's equations*.

However, when the EM wave in a simple, nonconducting source-free medium (characterized by $\rho = 0, \mathbf{J} = 0, \sigma = 0$) and the time-harmonic wave equations can be obtained as

$$\nabla^2 \mathbf{E} + k^2 \mathbf{E} = 0 \tag{10.135}$$

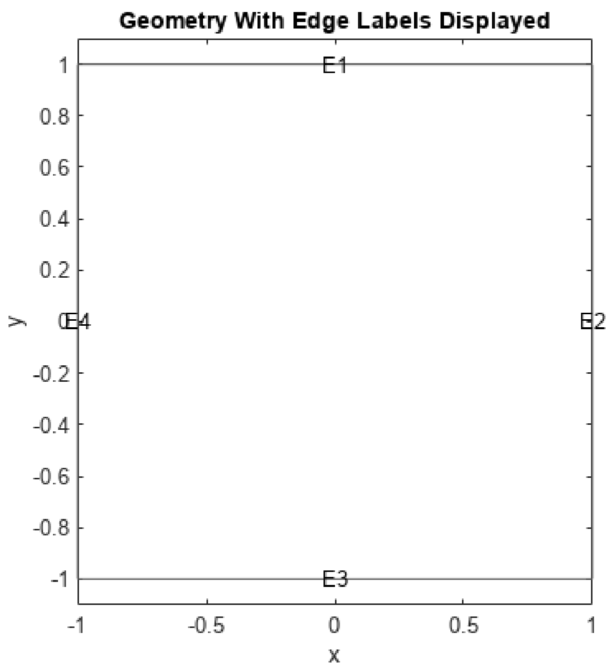
and

$$\nabla^2 \mathbf{H} + k^2 \mathbf{H} = 0. \tag{10.136}$$

Equations (10.135) and (10.136) are known as the homogenous vector *Helmholtz's equations*.

For example, we can solve standard wave equation on the square domain with MATLAB as:

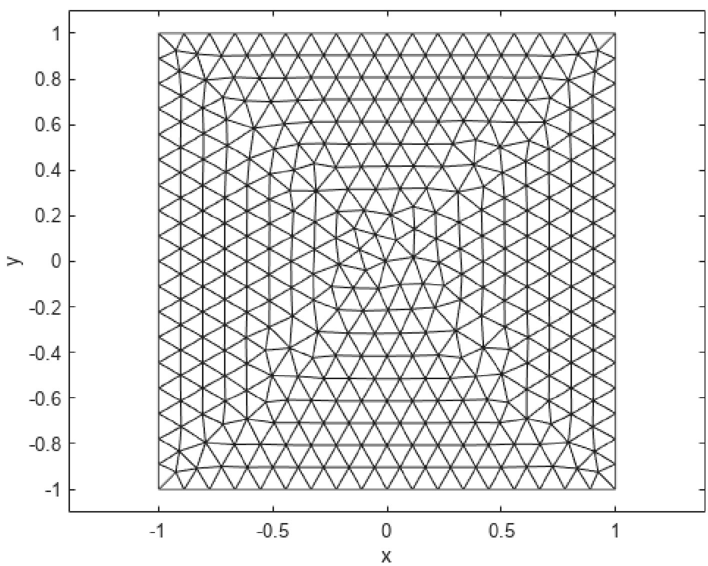
```
numberOfPDE = 1;

model = createpde(numberOfPDE);

geometryFromEdges(model,@squareg);

pdegplot(model,"EdgeLabels","on");

ylim([-1.1 1.1]);

axis equal

title("Geometry With Edge Labels Displayed")

xlabel("x")

ylabel("y")
```



%Specify PDE coefficients.

```
specifyCoefficients(model,"m",m,"d",0,"c",c,"a",a,"f",f);
```

```
%Set zero Dirichlet boundary conditions on the left (edge 4)
and right (edge 2) and zero Neumann boundary conditions on the
top (edge 1) and bottom (edge 3).

applyBoundaryCondition(model,"dirichlet","Edge",[2,4],"u",0);
applyBoundaryCondition(model,"neumann","Edge",([1 3]),"g",0);

%Create and view a finite element mesh for the problem.

generateMesh(model);
figure
pdemesh(model);
ylim([-1.1 1.1]);
axis equal
xlabel x
ylabel y
```



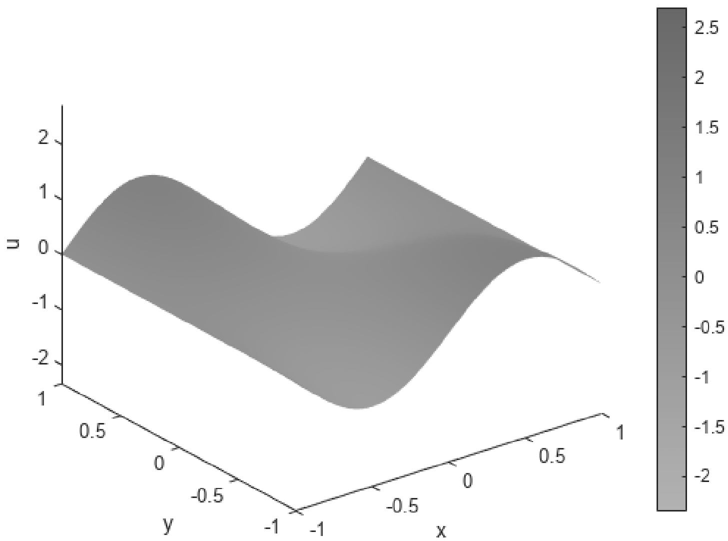
Set the following initial conditions:

(1). $u(x,0)=\tan^{-1}\left(\cos\left(\frac{\pi x}{2}\right)\right)$

(2). $\frac{\partial u}{\partial t}\Big|_{t=0}=3\sin(\pi x)\exp\left(\sin\left(\frac{\pi y}{2}\right)\right)$

```
u0 = @(location) atan(cos(pi/2*location.x));
ut0 = @(location) 3*sin(pi*location.x).*exp(sin(pi/2*location.y));
setInitialConditions(model,u0,ut0);
%This choice avoids putting energy into the higher vibration modes and permits a reasonable time step size. Specify the solution times as 31 equally-spaced points in time from 0 to 5.

n = 31;
tlist = linspace(0,5,n);

%Set the SolverOptions.ReportStatistics of model to 'on'.

model.SolverOptions.ReportStatistics ='on';
result = solvepde(model,tlist);

441 successful steps
34 failed attempts
952 function evaluations
1 partial derivatives
115 LU decompositions
951 solutions of linear systems

u = result.NodalSolution;
```

```
%Create an animation to visualize the solution for all time
steps. Keep a fixed vertical scale by first calculating the maxi-
mum and minimum values of u over all times, and scale all plots
to use those z-axis limits.

figure
umax = max(max(u));
umin = min(min(u));
for i = 1:n
    pdeplot(model,"XYData",u(:,i),"ZData",u(:,i), ...
                   "ZStyle","continuous","Mesh","off");
    axis([-1 1 -1 1 umin umax]);
    caxis([umin umax]);
    xlabel x
    ylabel y
    zlabel u
    M(i) = getframe;
end
```



10.13 ELECTROMAGNETIC ANALYSIS

Due to the cost-effectiveness of experiments and testing, the development of transmission lines in integrated circuit systems is time-consuming. Today, researchers, designers, and engineers used several numerical and analytical methods to study and investigate the parameter variations and properties of designing high-speed integrated circuits (microwave circuits) and electromagnetic (EM) problems. The most common analytical methods used for exact solutions in electromagnetic are conformal mapping, integral solutions, separation of variables, and series expansion. Also, the most popular numerical methods used for approximate solutions are methods called moment methods, methods of line, finite difference methods, and FEM.

FEM has great success in electromagnetic analysis compared to other methods. In contrast to other numerical methods, it is very useful for solving problems in complex geometries and inhomogeneous media. In this chapter, we show an overview of the finite element method. FEM requires that any problem involved in the geometrical region to be subdivided into finite number of smaller regions or elements. An approximate solution for the partial differential equation can be developed for each of these elements. In addition, the total solution is generated by assembling together the individual solutions taking care in order to ensure continuity at the interelement boundaries. Basically, there are four steps used in FEM: *first*, creating and discretizing the solution region (domain) into a finite number of subregions or elements; that is, divide the problem into nodes and elements and assume a shape function to represent the physical behavior of an element; *second*, developing equations for an element; *third*, assembling all the elements to represent in solution region, constructing the global coefficient matrix and applying boundary conditions and initial conditions; *fourth*, solving the system of equations to obtain the important information of the problem.

10.13.1 One-Dimensional Elements

10.13.1.1 The Approach to FEM Standard Steps Procedure

The *first* step is the discretization step, that is, the solution domain is divided into finite elements. Figure 10.3 provides an example of elements employed in one dimension. It shows the points of intersection of the lines that make up the sides of the elements called *nodes,* and the sides themselves are known as *nodal lines*.

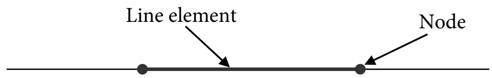


FIGURE 10.3 Example of elements in one-dimensional (1D).

The *second* step is the development of equations to approximate the solution for each element. It can be done by choosing an approximate function with unknown coefficients that will be used to approximate the solution. We use a first-order polynomial (straight line) as a linear variation of potential between the nodes over element m, i.e.,

$$V^{(m)}(x) = a + bx. \tag{10.137}$$

where $V(x)$ is the dependent variable (potential function); a and $b$are constants; x is the independent variable.

We can find the two constants a and b by using the two nodes to satisfy the equation at the location of the two nodes as:

$$V_1^{(m)} = a + bx_1 \tag{10.138}$$

and

$$V_2^{(m)} = a + bx_2 \tag{10.139}$$

where $V_1^{(m)} = V^{(m)}(x_1)$ and $V_2^{(m)} = V^{(m)}(x_2)$. By using Cramer's rule, we can solve equations (10.138) and (10.139), i.e.,

$$a = \frac{V_1^{(m)} x_2 - V_2(m) x_1}{x_2 - x_1} \tag{10.140}$$

$$b = \frac{V_2^{(m)} - V_1^{(m)}}{x_2 - x_1}. \tag{10.141}$$

Equations (10.140) and (10.141) can be substituted into equation (10.137) to give the approximate (or shape) function $V(x)$ in terms of the interpolation functions, H_1 and H_2 over element m, that is,

$$V^{(m)}(x) = a_1^{(m)}(x) V_1^{(m)} + a_2^{(m)}(x) V_2^{(m)} \tag{10.142}$$

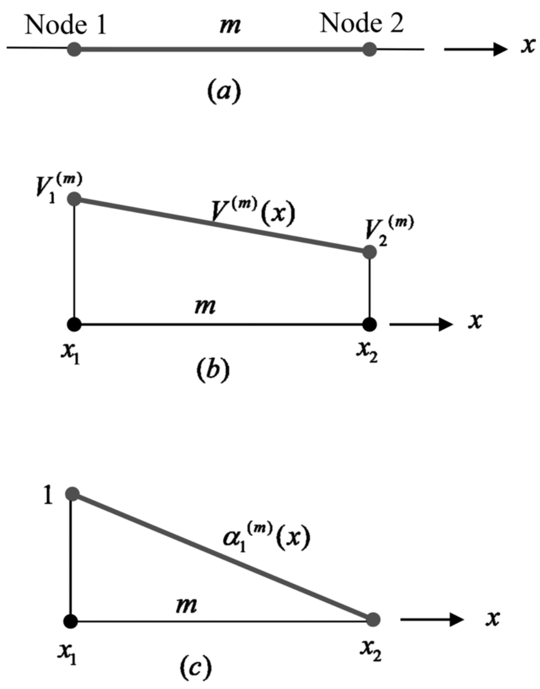
where

$$a_1^{(m)}(x) = \frac{x_2 - x}{x_2 - x_1} \tag{10.143}$$

$$a_2^{(m)}(x) = \frac{x - x_1}{x_2 - x_1}. \tag{10.144}$$

Indeed, equation (10.142) is a first-order interpolating polynomial. In addition, it provides a means to calculate intermediate values between the given values V_1 and V_2 at the nodes.

The shape function, along with the corresponding interpolation functions, is presented in Figure 10.4. Moreover, the sum of the interpolation functions, a_1 and a_2, that is, $\sum_{i=1}^{2} a_i = 1$.



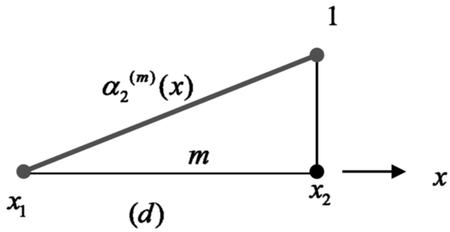


FIGURE 10.4 (a) a line element, (b) a linear approximation (or shape) function, (c) the corresponding interpolation function $\alpha_1^{(m)}(x)$ for $V^{(m)}(x)$, and (d) the corresponding interpolation function $\alpha_2^{(m)}(x)$ for $V^{(m)}(x)$.

Furthermore, it follows that,

$$\frac{dV^{(m)}}{dx} = \frac{d\alpha_1^{(m)}}{dx} V_1^{(m)} + \frac{d\alpha_2^{(m)}}{dx} V_2^{(m)} \tag{10.145}$$

$$\frac{d\alpha_1^{(m)}}{dx} = \frac{-1}{x_2 - x_1} \tag{10.146}$$

and

$$\frac{d\alpha_2^{(m)}}{dx} = \frac{1}{x_2 - x_1}. \tag{10.147}$$

Thus,

$$\frac{dV^{(m)}}{dx} = \frac{\left(-V^{(m)}{}_1 + V_2^{(m)}\right)}{x_2 - x_1}. \tag{10.148}$$

Now, the integral of $V^{(m)}$ is:

$$\int_{x_1}^{x_2} V^{(m)} dx = \int_{x_1}^{x_2} \left(\alpha_1^{(m)} V_1^{(m)} + \alpha_2^{(m)} V_2^{(m)}\right) dx = \frac{\left(V^{(m)}{}_1 + V_2^{(m)}\right)\left(x_2 - x_1\right)}{2}. \tag{10.149}$$

Now, we evaluate the coefficients so that the function approximates the solution in a best approach. The most common methods used for this proposal are the variational approaches, the weighted residuals, and the direct approaches. These methods can specify the relationships between the unknowns in equation (10.142) that satisfy the partial differential equation in an optimal approach. The resulting element equations can be expressed in a set of linear equations in matrix form, i.e.,

$$\left[C^{(m)}\right]\left\{V^{(m)}\right\}_c = \left\{\Psi_c^{\ (m)}\right\} \tag{10.150}$$

where

$\left[C^{(m)}\right]$ is element property (stiffness) matrix; $\left\{V_c^{(m)}\right\}$ is a column vector of unknowns at the nodes over element m; and $\left\{\Psi_c^{\ (m)}\right\}$ is a column vector reflecting the effect of any external influences applied at the node over element m.

Third, we assemble all the elements to represent the solution region. The solutions for the closest elements are matched so that the unknown values at their common nodes are equivalent. Therefore, the total sum will be continuous. Then, the assembled system needs to be modified for its boundary condition. The system can be expressed as:

$$\left[C_a^{\ (m)}\right]\left\{V_{ac}^{\ (m)}\right\} = \left\{\Psi_{ac}^{\ (m)}\right\} \tag{10.151}$$

where

$\left[C_a^{\ (m)}\right]$ is the assemblage element property (stiffness) matrix; $\left\{V_{ac}^{\ (m)}\right\}$ is the assemblage column vector of unknowns at the nodes over element m; and $\left\{\Psi_{ac}^{\ (m)}\right\}$ is the assemblage column vector reflecting the effect of any external influences applied at the node over element m.

Fourth, solving the system of equations (10.151) to obtain important information on the problem can be obtained by the LU decomposition technique.

10.13.1.2 Application to Poisson's Equation in One-Dimension

In this section, we solve the one-dimensional (1D) Poisson's equation for the potential distribution $V(x)$

$$\frac{d^2}{dx^2}V = -\frac{\rho_v}{\varepsilon} \tag{10.152}$$

with boundary conditions (BCs) $V(a) = v_1$, $V(b) = v_2$.

Using the same essential four steps as in the previous section with FEM, we focus here on the source term and only the major differences.

We will use the variational principle and the weighted residuals method to obtain the solution of the one-dimensional (1D) Poisson's equation.

(1) Variational Approach

The deriving element governing equations step. We look for the potential distribution $V(x)$ that can minimize an energy function $F(V)$ as

$$F(V)=\int_a^b\left(\frac{1}{2}\left(\frac{dV}{dx}\right)^2-\frac{\rho_m}{\varepsilon}V(x)\right)dx. \tag{10.153}$$

Two nodal values of $V(x)$are required to define uniquely a line variation of $V^{(m)}(x)$ over an element (m). Hence, the linear variation of $V^{(m)}(x)$ can be presented as

$$V^{(m)}(x)=\alpha_1(x)V_1+\alpha_2(x)V_2 \tag{10.154}$$

where the interpolation functions $\alpha_1(x)$and $\alpha_2(x)$ are presented as

$$\alpha_1(x)=\frac{x_2-x}{x_2-x_1},\ \ \alpha_2(x)=\frac{x-x_1}{x_2-x_1}. \tag{10.155}$$

The resulting element equation (10.154) can be expressed in a set of linear equations in matrix form:

$$V^{(m)}=[\alpha_1,\alpha_2]\begin{bmatrix}V_1\\V_2\end{bmatrix}=[\alpha]\{V_c^{(m)}\}. \tag{10.156}$$

The energy function can be written as

$$F(V)=\sum_{m=1}^{N}F^{(m)}\left(V^{(m)}\right) \tag{10.157}$$

where N is the number of elements with the domain $a\le x\le b$.

Now, substituting equation (10.156) into (10.153) can give

$$F^{(m)}(V^{(m)})=\int_{x_1}^{x_2}\left(\frac{1}{2}\left(\left[\frac{d\alpha_1}{dx}\frac{d\alpha_2}{dx}\right]\begin{bmatrix}V_1\\V_2\end{bmatrix}\right)^2-\frac{\rho_v}{\varepsilon}\left([\alpha_1\alpha_2]\begin{bmatrix}V_1\\V_2\end{bmatrix}\right)^2\right)dx. \tag{10.158}$$

By minimizing the $F^{(m)}(V^{(m)})$ with respect to the nodal values of V, we obtain the following equations for an element (m)

$$\frac{\partial F^{(m)}}{\partial V_1} = \int_{x_1}^{x_2} \left(\frac{da_1}{dx} \begin{bmatrix} \frac{da_1}{dx} & \frac{da_2}{dx} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} - \frac{\rho_v}{\varepsilon} a_1 \right) dx = 0 \tag{10.159}$$

and

$$\frac{\partial F^{(m)}}{\partial V_2} = \int_{x_1}^{x_2} \left(\frac{da_2}{dx} \begin{bmatrix} \frac{da_1}{dx} & \frac{da_2}{dx} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} - \frac{\rho_v}{\varepsilon} a_2 \right) dx = 0. \tag{10.160}$$

These equations can be expressed in matrix form as

$$\left[C^{(m)} \right] \left\{ V_c^{(m)} \right\} = \left\{ \Psi_c^{(m)} \right\} \tag{10.161}$$

where

$$\left[C^{(m)} \right] = \int_{x_1}^{x_2} \begin{bmatrix} \frac{da_1^{(m)}}{dx} \frac{da_1^{(m)}}{dx} & \frac{da_1^{(m)}}{dx} \frac{da_2^{(m)}}{dx} \\ \frac{da_2^{(m)}}{dx} \frac{da_1^{(m)}}{dx} & \frac{da_2^{(m)}}{dx} \frac{da_2^{(m)}}{dx} \end{bmatrix} dx \tag{10.162}$$

$$\left\{ V_c^{(m)} \right\} = \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \tag{10.163}$$

$$\left\{ \Psi_c^{(m)} \right\} = \int_{x_1}^{x_2} \begin{bmatrix} -\frac{\rho_v}{\varepsilon} a_1 \\ -\frac{\rho_v}{\varepsilon} a_2 \end{bmatrix} dx \tag{10.164}$$

where the elements of $\left\{ \Psi_c^{(m)} \right\}$ are the nodal forcing functions. The equations in (10.161) can give the characteristics of Poisson's equation in 1D. Indeed, in spite of the type of element we choose to formulate Poisson's equation in 1D, the element equations will have the form of equation (10.161). For the solution of Poisson's equation in 1D, it is essential to derive the equations for all the elements in the assemblage and then assemble these algebraic equations.

(2) Weighted Residuals Method

In the variational approach for 1D Poisson equation with boundary condition, we derive the element matrices $\left[C^{(m)}\right]$ and $\left\{\Psi_c^{(m)}\right\}$ for a linear variation of potential $V(x)$ over element (m) with two nodes. Now, we will use Galerkin's method with weighting functions $W_i = \alpha_i$ to derive the element matrices. We approximate the exact unknown solution $V_m(x)$ by

$$V_m(x) = \sum_{i=1}^{N} \alpha_i(x) V_{mi} \tag{10.165}$$

where

N is the number of nodes (here $N = 2$), V_{mi} is the unknown nodal values, $i = 1,2$.

Note that we do not consider the fixed boundary conditions at the element level, but these are included after the assembly process as in the previous method.

Now, by applying Galerkin's method, we get:

$$\int_{x_1}^{x_2} \left(\frac{d^2 V_m}{dx^2} + \frac{\rho_v}{\varepsilon} \right) \alpha_i(x)\, dx = 0, \qquad i = 1,2 \tag{10.166}$$

where

x_1 and x_2 are the coordinates of the end nodes of the line element.

By using integration by parts to the term with the derivatives of $V_m(x)$, that is,

$$\alpha_i \frac{dV_m}{dx}\bigg|_{x_1}^{x_2} - \int_{x_1}^{x_2} \frac{dV_m}{dx} \frac{d\alpha_i}{dx}\, dx + \int_{x_1}^{x_2} \frac{\rho_v}{\varepsilon}\, \alpha_i(\mathrm{x})\, dx = 0, \;\; i = 1,2. \tag{10.167}$$

Taking the derivative of equation (2.29) as

$$\frac{dV_m}{dx} = \sum_{i=1}^{N} \frac{d\alpha_i}{dx}\, V_{mi} = \left[\frac{d\alpha_i}{dx} \right] \left\{ V_c^{(m)} \right\} \tag{10.168}$$

where

$\{V_c^{(m)}\}$ is the column vector of nodal unknowns for the element m.

Thus, equation (2.30) becomes

$$\int_{x_1}^{x_2} \left[\frac{d\alpha_i}{dx}\right] \frac{d\alpha_i}{dx} dx \{V_c^{(m)}\} = \alpha_i \frac{dV_m}{dx}\bigg|_{x_1}^{x_2} + \int_{x_1}^{x_2} \frac{\rho_v}{\varepsilon}\, \alpha_i(x)\, dx\ , \qquad i = 1,2. \quad (10.169)$$

Furthermore, the first term on the right-hand side of equation (10.169) represents natural boundary conditions for the element m. We obtain these as

$$i = 1,\ \ \alpha_i \frac{dV_m}{dx}\bigg|_{x_1}^{x_2} = \alpha_1(x_2)\frac{dV_m}{dx}(x_2) - \alpha_1(x_1)\frac{dV_m}{dx}(x_1) = -\frac{dV_m}{dx}(x_1) \quad (10.170)$$

because $\alpha_1(x_2) = 0,\ \alpha_1(x_1) = 1$,

and

$$i = 2,\ \ \alpha_i \frac{dV_m}{dx}\bigg|_{x_1}^{x_2} = \alpha_2(x_2)\frac{dV_m}{dx}(x_2) - \alpha_2(x_1)\frac{dV_m}{dx}(x_1) = \frac{dV_m}{dx}(x_2) \quad (10.171)$$

because $\alpha_2(x_2) = 1,\ \alpha_2(x_1) = 0$.

We use the end-point values of α_i shown in Figure 10.4. Thus, the element equations are presented as

$$\begin{bmatrix} C_{11}^{(m)} & C_{12}^{(m)} \\ C_{21}^{(m)} & C_{22}^{(m)} \end{bmatrix} \begin{bmatrix} V_1^{(m)} \\ V_2^{(m)} \end{bmatrix} = \begin{bmatrix} -\dfrac{dV^{(m)}}{dx}(x_1) \\ \dfrac{dV^{(m)}}{dx}(x_2) \end{bmatrix} + \begin{bmatrix} \Psi_1^{(m)} \\ \Psi_2^{(m)} \end{bmatrix} \quad (10.172)$$

where

$$\left[C^{(m)}\right] = \int_{x_1}^{x_2} \begin{bmatrix} \dfrac{d\alpha_1}{dx}\dfrac{d\alpha_1}{dx} & \dfrac{d\alpha_1}{dx}\dfrac{d\alpha_2}{dx} \\ \dfrac{d\alpha_2}{dx}\dfrac{d\alpha_1}{dx} & \dfrac{d\alpha_2}{dx}\dfrac{d\alpha_2}{dx} \end{bmatrix} dx \ \text{ and } \left\{\Psi_c^{(m)}\right\} = \int_{x_1}^{x_2} \begin{bmatrix} \dfrac{\rho_v}{\varepsilon}\alpha_1 \\ \dfrac{\rho_v}{\varepsilon}\alpha_2 \end{bmatrix} dx. \quad (10.173)$$

The extension to an element with N nodes follows the same steps but with $i = 1,2,\ldots,N$. In addition, the matrices for an element with N nodes contain terms similar to the equation (10.172) but with additional rows and columns to account for N element equations.

10.13.1.3 Natural Coordinates in One Dimension

We use natural (length) coordinates in deriving interpolation functions that can be used to evaluate the integrals in the element equations. In addition, we use the natural coordinate system in describing the location of a point inside an element in terms of the coordinates associated with the nodes of the element. Let η_i be the natural coordinates, where $i = 1,2,\ldots,N$; N is the number of external nodes of the element. Knowing that, natural coordinates are functions of the global Cartesian coordinate system in which the element is defined, the one coordinate is associated with node i and has a unit value there.

Figure 10.5 shows a line element with natural coordinates η_1, η_2 and location point x_l.

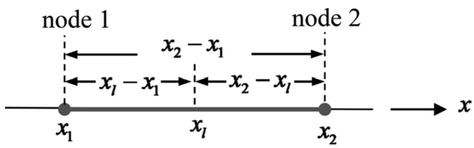


FIGURE 10.5 Example of two-node line element in one-dimensional (1D) with global coordinate x_l.

The global coordinate x_l can be expressed as

$$x_l = \eta_1 x_1 + \eta_2 x_2. \tag{10.174}$$

We can interpret natural (length) coordinates η_1 and η_2 as weighting functions relating the coordinates of the end modes to the coordinate of any interior point. As we know that,

$$\eta_1 + \eta_2 = 1 \tag{10.175}$$

although, the weighting functions are not independent. Let us consider $x_l = x$ and solving for η_1 and η_2 from equations (10.174) and (10.175), we get

$$\eta_1(x) = \frac{x_2 - x}{x_2 - x_1},\ \eta_2(x) = \frac{x - x_1}{x_2 - x_1}. \tag{10.176}$$

The linear interpolation used for the potential distribution variable $V(x)$ in the previous section, which can be written as

$$V(x) = V_1\eta_1 + V_2\eta_2. \tag{10.178}$$

By differential of $V(x)$ using the chain rule, we get

$$\frac{dV}{dx} = \frac{\partial V}{\partial \eta_1}\frac{\partial \eta_1}{\partial x} + \frac{\partial V}{\partial \eta_2}\frac{\partial \eta_2}{\partial x} \tag{10.179}$$

where

$$\frac{\partial \eta_1}{\partial x} = \frac{-1}{x_2 - x_1}, \quad \frac{\partial \eta_2}{\partial x} = \frac{1}{x_2 - x_1}. \tag{10.180}$$

Now, taking the integration of length coordinates over the length of an element, that is,

$$\int_{x_1}^{x_2} \eta_1^i \eta_2^j dx = \frac{i!\,j!(x_2 - x_1)}{(1 + j + 1)!} \tag{10.181}$$

where i and j are integer exponents.

10.13.2 Two-Dimensional Elements

10.13.2.1 Applications of FEM to Electrostatic Problems

It is often known that FEM is a numerical method used to find the approximate solutions either for partial differential equations or integral equations. These equations are most involved in electromagnetic problems. We illustrate the four steps above used to find the solution in FEM through three different types of differential equations, Laplace's equation, Poisson's equation, and wave equation.

10.13.2.1.1 Solution of Laplace's Equation $\nabla^2 V = 0$ with FEM

To find the potential distribution, $V(x,y)$, for the two-dimensional (2-D) solution region, as shown in Figure 10.6. We illustrate the following steps to get the solution of Laplace equation, $\nabla^2 V = 0$.

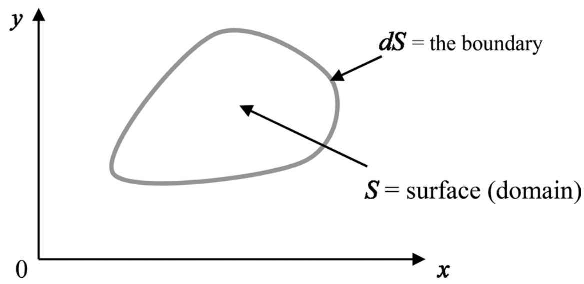


FIGURE 10.6 The solution region of the problem showing domain for the 2-D boundary value.

First step, using finite element discretization to find the potential distribution for the two-dimensional solution, $V(x,y)$ as shown in Figure 10.7, where the solution region is subdivided into seven nonoverlapping finite elements of triangles. It is always preferable in computation to have the same type of elements through the solution region which in our case is the triangle.

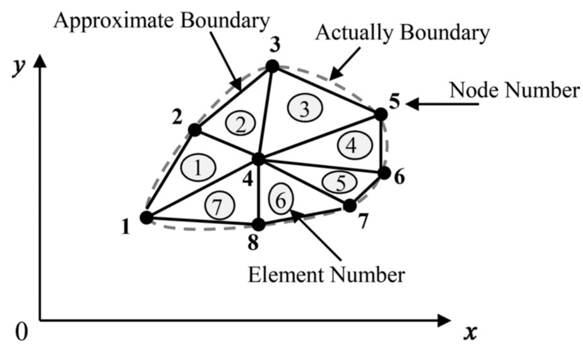


FIGURE 10.7 The finite element discretization of the solution.

We look for an approximation solution for the potential $V_m(x,y)$ within an element m and then interrelate the potential distribution in various elements such that the potential is continuous across interelement boundaries. We can express the approximation solution for the whole region as

$$V(x,y) \approx \sum_{m=1}^{N} V_m(x,y), \tag{10.182}$$

where N is the number of triangle elements into which the solution region is divided.

The most common form of approximation for $V_m(x,y)$ within an element is polynomial approximation for a triangle element, that is,

$$V_m(x,y) = a + bx + cy, \tag{10.183}$$

where the constants a, b, and c are to be determined. The potential $V_m(x,y)$ in general is nonzero within element m, but zero outside m. Furthermore, our assumption of linear variation of potential within the triangle element as in equation (2.46) is the same as assuming that the electric field is uniform within the element, that is to say,

$$\mathbf{E}_m = -\nabla V_m = -(b\mathbf{a}_x + c\mathbf{a}_y). \tag{10.184}$$

Second step, developing equations for the element. Let us choose a typical triangle element shown in Figure 10.8.

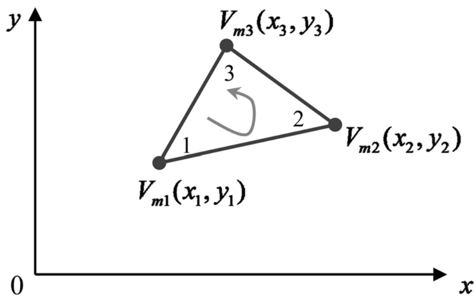


FIGURE 10.8 Typical triangle element; local node numbering 1-2-3 must proceed counterclockwise as indicated by the arrow.

The potential $V_{m1}(x_1,y_1)$, $V_{m2}(x_2,y_2)$, and $V_{m3}(x_3,y_3)$ at nodes 1, 2, and 3, respectively, are obtained using equation (10.183), namely

$$\begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix}. \tag{10.185}$$

The coefficients a, b, and c are determined from equation (10.185) as

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}^{-1} \begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \end{bmatrix}. \tag{10.186}$$

Therefore, equation (10.183) can be rewritten by substituting for *a, b*, and *c*, i.e.,

$$V_m = \begin{bmatrix} 1 & x & y \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} 1 & x & y \end{bmatrix} \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}^{-1} \begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \end{bmatrix}. \tag{10.187}$$

Equation (10.187) can be written as

$$V_m(x,y) = \sum_{i=1}^{3} \alpha_i(x,y) V_{mi}, \tag{10.188}$$

where $\alpha_i(x,y)$ is given by

$$\alpha_i(x,y) = \frac{1}{2A}(a_i + b_i x + c_i y), \text{ where } i = 1,2,3 \tag{10.189}$$

and a_i, b_i, and c_i are given by

$$a_i = x_j y_k - x_k y_j \tag{10.190}$$

$$b_i = y_j - y_k \tag{10.191}$$

$$c_i = x_k - x_j \tag{10.192}$$

where *i, J and k* are cyclical, that is, $(i = 1, j = 2, k = 3)$, $(i = 2, j = 3, k = 1)$, and $(i = 3, j = 1, k = 2)$.

Note that by substituting equations (10.190), (10.191), and (10.192) into equation (10.187) gives

$$V_m(x,y)=\begin{bmatrix}1 & x & y\end{bmatrix}\frac{1}{2A}\begin{bmatrix}(x_2y_3-x_3y_2) & (x_3y_1-x_1y_3) & (x_1y_2-x_2y_1)\\ (y_2-y_3) & (y_3-y_1) & (y_1-y_2)\\ (x_3-x_2) & (x_1-x_3) & (x_2-x_1)\end{bmatrix}\begin{bmatrix}V_{m1}\\ V_{m2}\\ V_{m3}\end{bmatrix}. \tag{10.193}$$

Also, using equations (10.190), (10.191), and (10.192) in equation (10.188) gives

$$\alpha_1(x,y)=\frac{1}{2A}\left[(x_2y_3-x_3y_2)+(y_2-y_3)x+(x_3-x_2)y\right], \tag{10.194a}$$

$$\alpha_2(x,y)=\frac{1}{2A}\left[(x_3y_1-x_1y_3)+(y_3-y_1)x+(x_1-x_3)y\right], \tag{10.194b}$$

$$\alpha_3(x,y)=\frac{1}{2A}\left[(x_1y_2-x_2y_1)+(y_1-y_2)x+(x_2-x_1)y\right], \tag{10.194c}$$

and A is given by

$$A=\frac{1}{2}\begin{vmatrix}1 & x_1 & y_1\\ 1 & x_2 & y_2\\ 1 & x_3 & y_3\end{vmatrix}=\frac{1}{2}\left[(x_1y_2-x_2y_1)+(x_3y_1-x_1y_3)+(x_2y_3-x_3y_2)\right]$$

or

$$A=\frac{1}{2}\left[(x_2-x_1)(y_3-y_1)-(x_3-x_1)(y_2-y_1)\right] \tag{10.195}$$

where A is the area of the element m.

The value of A is positive if the nodes are numbered counterclockwise, starting from any nodes, as shown by the arrow in Figure 10.5.

Furthermore, equation (10.188) gives the potential at any point (x,y) within the element provided that the potentials at the vertices are known. In addition, $\alpha_i(x,y)$ are linear interpolation functions. They are called the element shape functions, and they have the following properties:

$$a_i(x,y) = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases}, \tag{10.196a}$$

$$\sum_{i=1}^{3} a_i(x,y) = 1. \tag{10.196b}$$

The shape of functions $a_1(x,y), a_2(x,y)$, and $a_3(x,y)$, for example, is illustrated in Figure 10.9.

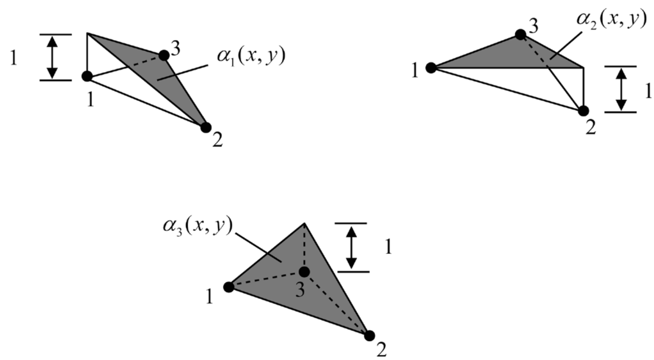


FIGURE 10.9 Shape functions $a_1(x,y)$, $a_2(x,y)$, and $a_3(x,y)$ for a triangle element.

The functional, W_m, corresponding to Laplace's equation, which physically is the energy per unit length associated with element m, is given by

$$W_m = \frac{1}{2}\int \varepsilon \left|\mathbf{E}_m\right|^2 dS = \frac{1}{2}\int \varepsilon \left|\nabla V_m\right|^2 dS. \tag{10.197}$$

But from equation (10.188),

$$\nabla V_m = \sum_{i=1}^{3} V_{mi} \nabla a_i. \tag{10.198}$$

By substituting equation (10.198) into equation (10.197), gives

$$W_m = \frac{1}{2}\sum_{i=1}^{3}\sum_{j=1}^{3} \varepsilon V_{mi}\left[\int \nabla a_i \cdot \nabla a_j dS\right] V_{mj}. \tag{10.199}$$

If we define the term in brackets as

$$C_{ij}^{(m)} = \int \nabla a_i \cdot \nabla a_j dS, \tag{10.200}$$

now, we can write equation (10.199) in matrix form as

$$W_m = \frac{1}{2}\varepsilon \left[V_m\right]^t \left[C^{(m)}\right]\left\{V_m\right\}, \tag{10.201}$$

where the superscript t denotes the transpose of the matrix,

$$\left[V_m\right]^t = \left[V_{m1} \quad V_{m2} \quad V_{m3}\right], \tag{10.202a}$$

$$\left\{V_m\right\} = \begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \end{bmatrix}, \tag{10.202b}$$

and

$$\left[C^{(m)}\right] = \begin{bmatrix} C_{11}^{(m)} & C_{12}^{(m)} & C_{13}^{(m)} \\ C_{21}^{(m)} & C_{22}^{(m)} & C_{23}^{(m)} \\ C_{31}^{(m)} & C_{32}^{(m)} & C_{33}^{(m)} \end{bmatrix}. \tag{10.202c}$$

The matrix $\left[C^{(m)}\right]$ is usually called the *element coefficient matrix* (or *stiffness matrix in structural analysis*). The element $C_{ij}^{(m)}$ of the coefficient matrix may be regarded as the coupling between nodes i and j; its value is obtained from equations (10.194) and (10.200). For instance,

$$\begin{aligned} C_{12}^{(m)} &= \int \nabla a_1 \cdot \nabla a_2 \\ &= \frac{1}{4A^2}\left[(y_2 - y_3)(y_3 - y_1) + (x_3 - x_2)(x_1 - x_3)\right]\int dS \\ &= \frac{1}{4A}\left[(y_2 - y_3)(y_3 - y_1) + (x_3 - x_2)(x_1 - x_3)\right]. \end{aligned} \tag{10.203a}$$

Similarly,

$$C_{13}^{(m)} = \frac{1}{4A}\left[(y_2 - y_3)(y_1 - y_2) + (x_3 - x_2)(x_2 - x_1)\right], \tag{10.203b}$$

$$C_{23}^{(m)} = \frac{1}{4A}\left[(y_3 - y_1)(y_1 - y_2) + (x_1 - x_3)(x_2 - x_1)\right], \tag{10.203c}$$

$$C_{11}^{(m)} = \frac{1}{4A}\left[(y_2 - y_3)^2 + (x_3 - x_2)^2\right], \tag{10.203d}$$

$$C_{22}^{(m)} = \frac{1}{4A}\left[(y_3 - y_1)^2 + (x_1 - x_3)^2\right], \tag{10.203e}$$

$$C_{33}^{(m)} = \frac{1}{4A}\left[(y_1 - y_2)^2 + (x_2 - x_1)^2\right]. \tag{10.203f}$$

Additionally,

$$C_{23}^{(m)} = C_{32}^{(m)},\ C_{13}^{(m)} = C_{31}^{(m)},\ C_{12}^{(m)} = C_{21}^{(m)}. \tag{10.204}$$

Now, for the third step, after having considered a typical element, the next step is to assemble all such elements in the solution region. The energy associated with the assemblage of elements assuming that the whole solution region is homogeneous so that ε is constant, i.e.,

$$W = \sum_{m=1}^{N} W_m = \frac{1}{2}\varepsilon\,[V]^t\,[C]\{V\}, \tag{10.205}$$

where

$$\{V\} = \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ \cdot \\ \cdot \\ \cdot \\ V_n \end{bmatrix}, \tag{10.206}$$

where

n is the number of nodes, N is the number of elements, and $[C]$ is called the overall or *global coefficient matrix*, which is the assemblage of individual element coefficient matrices.

For an inhomogeneous solution region such as that shown in Figure 10.10, the region is discretized with triangle elements such that each finite element is homogeneous. In this case, equation (10.197) still holds; however, equation (10.205) does not apply since ε ($\varepsilon = \varepsilon_r \varepsilon_0$) or simply ε_r varies from element to element. To apply equation (10.205), we need to replace ε by ε_0 and multiply the integrand in equation (10.200) by ε_r.

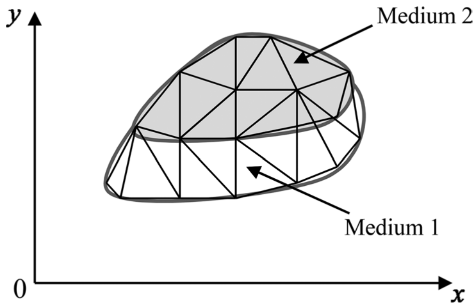


FIGURE 10.10 Discretization of an inhomogeneous solution region with triangle elements.

We use an example to illustrate the process by which individual element coefficient matrices are assembled to obtain the global coefficient matrix. In this example, we consider the finite element mesh consisting of three finite elements, as shown in Figure 10.11. Observe the numberings of the mesh.

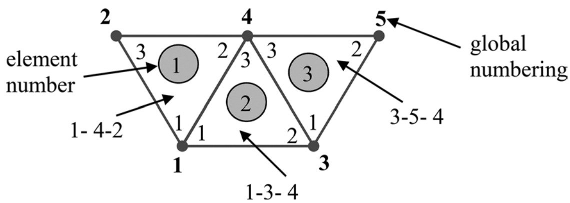


FIGURE 10.11 Assembly of three elements; $i-j-k$ corresponding to local numbering (1-2-3) of the element in Figure 10.5.

The numbering of nodes 1, 2, 3, 4, and 5 is called *global* numbering. The numbering $i-j-k$ is called *local* numbering, and it corresponds with 1-2-3 of the element in Figure 10.5; the local numbering must be in a counterclockwise sequence starting from any node of the element. For element 1, we could choose 2-1-4 instead of 1-4-2 to correspond with 1–2–3 of the element in Figure 10.8. Thus, the numbering in Figure 10.8 is not unique. But whichever numbering is used, the global coefficient matrix remains the same.

Assuming the particular numbering in Figure 10.11, the global coefficient matrix is expected to have the form

$$[C]=\begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} \end{bmatrix}, \tag{10.207}$$

which is a 5×5 matrix since five nodes ($n=5$) are involved. As we know, C_{ij} is the coupling between nodes i and j. The C_{ij} can be obtained by using the fact that the potential distribution must be continuous across interelement boundaries. The contribution to the i, j position in $[C]$ comes from all elements containing nodes i and j. For instance, in Figure 10.11, elements 1 and 2 have node 1 in common; therefore

$$C_{11}=C_{11}^{(1)}+C_{11}^{(2)}. \tag{10.208a}$$

Node 2 belongs to element 1 only; therefore

$$C_{22}=C_{33}^{(1)}.. \tag{10.0208b}$$

Node 4 belongs to elements 1, 2, and 3; accordingly

$$C_{44}=C_{22}^{(1)}+C_{33}^{(2)}+C_{33}^{(3)}. \tag{10.208c}$$

Nodes 1 and 4 belong simultaneously to elements 1 and 2; as a result

$$C_{14}=C_{41}=C_{12}^{(1)}+C_{13}^{(2)}. \tag{10.208d}$$

Since there is no coupling (or direct link) between nodes 2 and 3; hence

$$C_{23}=C_{32}=0. \tag{10.208e}$$

By continuing with this approach, we can obtain all the terms in the global coefficient matrix by inspection of Figure 10.11 as

$$[C]=\begin{bmatrix} C_{11}^{(1)}+C_{11}^{(2)} & C_{13}^{(1)} & C_{12}^{(2)} & C_{12}^{(1)}+C_{13}^{(2)} & 0 \\ C_{31}^{(1)} & C_{33}^{(1)} & 0 & C_{32}^{(1)} & 0 \\ C_{21}^{(2)} & 0 & C_{22}^{(2)}+C_{11}^{(3)} & C_{23}^{(2)}+C_{13}^{(3)} & C_{12}^{(3)} \\ C_{21}^{(1)}+C_{23}^{(2)} & C_{23}^{(1)} & C_{32}^{(2)}+C_{31}^{(3)} & C_{22}^{(1)}+C_{33}^{(2)}+C_{33}^{(3)} & C_{32}^{(3)} \\ 0 & 0 & C_{21}^{(3)} & C_{23}^{(3)} & C_{22}^{(3)} \end{bmatrix}. \tag{10.209}$$

Note that element coefficient matrices overlap at nodes shared by elements, and that are 27 terms (9 for each of the 3 elements) in the global coefficient matrix $[C]$. Also note the following properties of the matrix $[C]$:

1. It is symmetric ($C_{ij}=C_{ji}$) just as the element coefficient matrix.
2. Since $C_{ij}=0$ if no coupling exists between nodes i and j, it is expected that for a large number of elements $[C]$ becomes sparse. Matrix $[C]$ is also banded if the nodes are carefully numbered. It can be shown using equation (10.203), i.e.,

$$\sum_{i=1}^{3} C_{ij}^{(m)}=0=\sum_{j=1}^{3} C_{ij}^{(m)}. \tag{10.210}$$

3. It is singular. Although this is not obvious, it can be shown using the finite element coefficient matrix of equation (10.202c).

Finally, *fourth step*, by solving the resulting equations. It can be shown that, from variational calculus, it is known that Laplace's (or Poisson's) equation is satisfied when the total energy in the solution region is minimum. Therefore, we require that the partial derivatives of W with respect to each nodal value of the potential be zero; that is,

$$\frac{\partial W}{\partial V_1}=\frac{\partial W}{\partial V_2}=\cdots=\frac{\partial W}{\partial V_n}=0,$$

or

$$\frac{\partial W}{\partial V_k}=0, \qquad k=1,2,\ldots,n. \tag{10.211}$$

For instance, to get $\frac{\partial W}{\partial V_1} = 0$ for the finite element mesh of Figure 2.9, we substitute equation (10.207) into equation (10.205) and take the partial derivative of W with respect to V_1. We obtain

$$0 = \frac{\partial W}{\partial V_1} = 2V_1C_{11} + V_2C_{12} + V_3C_{13} + V_4C_{14} + V_5C_{15} + V_2C_{21} + V_3C_{31} + V_4C_{41} + V_5C_{51},$$

or

$$0 = V_1C_{11} + V_2C_{12} + V_3C_{13} + V_4C_{14} + V_5C_{15}. \tag{10.212}$$

In general case, $\frac{\partial W}{\partial V_k} = 0$ leads to

$$0 = \sum_{i=1}^{n} V_i C_{ik}, \tag{10.213}$$

where n is the number of nodes in the mesh. By writing equation (10.213) for all nodes $k = 1,2,3,..,n$, we obtain a set of simultaneous equations from which the solution of the transpose matrix for the potential distribution, $[V]^t = [V_1 \quad V_2 \quad . \quad . \quad . \quad V_n]$, can be found. This can be done in two ways:

(1) Iteration Method

Suppose node 1 in Figure 10.11, for example, is a free node. The potential at node 1 can be obtained from equation (10.212) as

$$V_1 = -\frac{1}{C_{11}} \sum_{i=2}^{5} V_i C_{1i}. \tag{10.214}$$

Thus, in general case, the potential at a free node k in a mesh with n nodes is obtained from equation (10.213) as

$$V_k = -\frac{1}{C_{kk}} \sum_{i=1, j \neq k}^{n} V_i C_{ik}. \tag{10.215}$$

Since $C_{ki} = 0$ is not directly connected to node i, only nodes that are directly linked to node k contribute to V_k in equation (10.215). Note equation (10.215) can be applied iteratively to all the free nodes. The iteration process begins by setting the potentials of fixed nodes (where the potentials are prescribed or known) to their prescribed values and the potentials at the free nodes (where the potentials are known) equal to zero or to the average potential

$$V_{ave} = \frac{1}{2}\left(V_{\min} + V_{\max}\right), \tag{10.216}$$

where $V_{\min}$ and $V_{\max}$ are the minimum and maximum values of the prescribed potentials at the fixed nodes, V, respectively. With these initial values, the potentials at the free nodes are calculated using equation (10.215). At the end of the first iteration, when the new values have been calculated for all the free nodes, they become the old values for the second iteration. Indeed, the procedure is repeated until the change between subsequent iteration is negligible enough.

(2) Band Matrix Method

If all free nodes are numbered first and the fixed nodes last, equation (10.205) can be written such that

$$W = \frac{1}{2}\varepsilon \begin{bmatrix} V_f & V_p \end{bmatrix} \begin{bmatrix} C_{ff} & C_{fp} \\ C_{pf} & C_{pp} \end{bmatrix} \begin{bmatrix} V_f \\ V_p \end{bmatrix}, \tag{10.217}$$

where subscripts f and p, refer to nodes with free and fixed (or prescribed) potentials, respectively. Since V_p is constant (it consists of known, fixed values), we only differentiate with respect to V_f so that applying equations (10.211) to (10.217) which yields to

$$\begin{bmatrix} C_{ff} & C_{fp} \end{bmatrix} \begin{bmatrix} V_f \\ V_p \end{bmatrix} = 0$$

or

$$\begin{bmatrix} C_{ff} \end{bmatrix}\begin{bmatrix} V_f \end{bmatrix} = -\begin{bmatrix} C_{fp} \end{bmatrix}\begin{bmatrix} V_p \end{bmatrix}. \tag{10.218}$$

This equation can be written as

$$[A][V] = [B] \tag{10.219a}$$

or

$$[V]=[A]^{-1}[B], \tag{10.219b}$$

where $[V]=\left[V_f\right]$, $[\mathrm{A}]=\left[C_{ff}\right]$, $[B]=-\left[C_{fp}\right]\left[V_p\right]$. Since, in general, nonsingular, the potential at the free nodes can be found using equation (10.219). Note we can solve for [V] in equation (10.219a) using Gaussian elimination technique. Also, we can solve for [V] in equation (10.219b) using matrix inversion if the size of the matrix to be inverted is not large.

In fact, it is sometimes necessary to impose the Neumann condition ($\frac{\partial V}{\partial n}=0$) as a boundary condition or at the line of symmetry when we take advantage of the symmetry of the problem. Indeed, suppose that for concreteness, a solution region is symmetric along the y-axis as in Figure 10.12. We impose condition ($\frac{\partial V}{\partial x}=0$) along the y-axis by making

$$V_1=V_2,\ \ V_4=V_5,\ \ V_7=V_8\,. \tag{10.220}$$

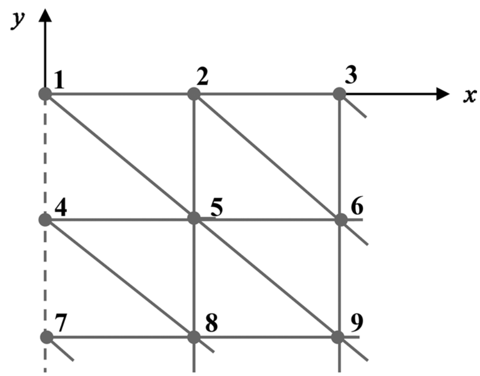


FIGURE 10.12 A solution region that is symmetric along the y-axis.

With this in mind, from equation (10.197) onward, the solution has been restricted to a two-dimensional problem involving Laplace's equation $\nabla^2 V=0$.

10.13.2.1.2 Solution of Passion's Equation $\nabla^2 V=-\frac{\rho_v}{\varepsilon}$ with FEM

In this section, we solve the two-dimensional (2D) Poisson's equations

$$\nabla^2 V=-\frac{\rho_v}{\varepsilon}, \tag{10.221}$$

using the same essential four steps as in the previous section with FEM, we focus here on the source term and only the major differences.

The deriving element governing equations step. We divide the solution region into triangles, and then we approximate the potential distribution $V_m(x,y)$ and the source term ρ_{vm} over each triangle element by linear combinations of the local interpolation polynomial α_i, namely,

$$V_m = \sum_{i=1}^{3} V_{mi}\alpha_i(x,y), \tag{10.222}$$

and

$$\rho_{vm} = \sum_{i=1}^{3} \rho_{mi}\alpha_i(x,y) \tag{10.223}$$

where

V_{mi} is the values of V at vertex i of element m; ρ_{mi} is the values of ρ_v at vertex i of element m. The values of ρ_{mi} are known since $\rho_v(x,y)$ is prescribed, while the values of V_{mi} are to be determined.

An energy functional which associates Euler equation with equation (10.221) is

$$F(V_m) = \frac{1}{2}\int_S \left[\varepsilon\left|\nabla V_m\right|^2 - 2\rho_{vm}V_m\right] dS, \tag{10.224}$$

where

$F(V_m)$ is the total energy per length within element m; $\frac{1}{2}\varepsilon\left|\nabla V_m\right|^2$ is the energy density in the electrostatic system and it is equal to $\frac{1}{2}\mathbf{D}\cdot\mathbf{E}$; $\rho_{vm}V_m dS$ is the work done in moving the charge $\rho_{vm}dS$ to its location at potential V_m.

Now, by substituting equations (10.222) and (10.223) into equation (10.224), we get

$$F(V_m) = \frac{1}{2}\sum_{i=1}^{3}\sum_{j=1}^{3}\varepsilon V_{mi}\left[\int \nabla\alpha_i \cdot \nabla\alpha_j dS\right]V_{mj} - \sum_{i=1}^{3}\sum_{j=1}^{3} V_{mi}\left[\int \alpha_i\alpha_j dS\right]\rho_{mj}. \tag{10.225}$$

Equation (10.225) can be applied to every element in the solution region.

Also, it can be written in matrix form as

$$F(V_m) = \frac{1}{2}\varepsilon\,[V_m]^t\left[C^{(m)}\right][V_m] - [V_m]^t\left[T^{(m)}\right][\rho_m] \tag{10.226}$$

where

$$C_{ij}^{(m)} = \int \nabla\alpha_i \cdot \nabla\alpha_j dS \tag{10.227}$$

we know that equation (10.226) is already defined in equation (10.203) and

$$T_{ij}^{(m)} = \int \alpha_i \alpha_j dS. \tag{10.228}$$

Also, $T_{ij}^{(m)}$ can be written as

$$T_{ij}^{(m)} = \begin{cases} A/6, & i = j \\ A/12, & i \neq j \end{cases} \tag{10.229}$$

where A is the area of the triangle element.

We can obtain the discredited functional for the whole solution region, with N elements and n nodes, as the sum of the functional for the individual elements, that is, from equation (10.229),

$$F(V) = \sum_{m=1}^{N} F(V_m) = \frac{1}{2}\varepsilon\,[V]^t[C][V] - [V]^t[T][\rho] \tag{10.230}$$

where t is the transposition symbol. In equation (10.230), the column matrix $[V]$ consists of the values of V_{mi}, while the column matrix $[\rho]$ contains n values of the source function, ρ_v, at the nodes. The functional in equation (10.230) is now minimized by differentiating with respect to V_{mi} and setting the result equal to zero.

Now, we work on the *solving the resulting equations step*. We can solve the resulting equations by using either the iteration method or the band matrix method.

(1) Iteration Methods

By considering the solution region in Figure 10.8 which has five nodes, $n = 5$ and from the equation (10.230), we can get the energy functional as

$$F = \frac{1}{2}\varepsilon\begin{bmatrix} V_1 & V_2 & \cdot & \cdot & \cdot & V_5 \end{bmatrix}\begin{bmatrix} C_{11} & C_{12} & \cdot & \cdot & \cdot & C_{15} \\ C_{21} & C_{22} & \cdot & \cdot & \cdot & C_{25} \\ \cdot & & & & & \cdot \\ \cdot & & & & & \cdot \\ \cdot & & & & & \cdot \\ C_{51} & C_{52} & \cdot & \cdot & \cdot & C_{55} \end{bmatrix}\begin{bmatrix} V_1 \\ V_2 \\ \cdot \\ \cdot \\ \cdot \\ V_5 \end{bmatrix}$$

$$-\begin{bmatrix} V_1 & V_2 & \cdot & \cdot & \cdot & V_5 \end{bmatrix}\begin{bmatrix} T_{11} & T_{12} & \cdot & \cdot & \cdot & T_{15} \\ T_{21} & T_{22} & \cdot & \cdot & \cdot & T_{25} \\ \cdot & & & & & \cdot \\ \cdot & & & & & \cdot \\ \cdot & & & & & \cdot \\ T_{51} & T_{52} & \cdot & \cdot & \cdot & T_{55} \end{bmatrix}\begin{bmatrix} \rho_1 \\ \rho_2 \\ \cdot \\ \cdot \\ \cdot \\ \rho_5 \end{bmatrix} . \quad (10.231)$$

The energy can be minimized by applying

$$\frac{\partial F}{\partial V_k} = 0, \qquad k = 1, 2, \cdot \cdot \cdot n. \quad (10.232)$$

For example, from equation (10.231), we get $\frac{\partial F}{\partial V_1} = 0$, as

$$\frac{\partial F}{\partial V_1} = \varepsilon\left[V_1 C_{11} + V_2 C_{21} + \cdots + V_5 C_{51}\right] - \left[T_{11}\rho_1 + T_{21}\rho_2 + \cdots + T_{51}\rho_5\right] = 0$$

or

$$V_1 = -\frac{1}{C_{11}}\sum_{i=2}^{5} V_i C_{i1} + \frac{1}{\varepsilon C_{11}}\sum_{i=1}^{5} T_{i1}\rho_i \,. \quad (10.233)$$

Therefore, in general, for a mesh with n nodes

$$V_k = -\frac{1}{C_{kk}}\sum_{i=1, i\neq k}^{n} V_i C_{ki} + \frac{1}{\varepsilon C_{kk}}\sum_{i=1}^{n} T_{ki}\rho_i \quad (10.234)$$

where

node k is assumed to be a free node.

By fixing the potential at the prescribed nodes and setting the potential at the free nodes initially equal to zero, we apply equation (10.234) iteratively to all free nodes until convergence is reached.

(2) Band Matrix Method

In this method, we let the free nodes be numbered first and the prescribed nodes last. In doing this, equation (10.230) can be written as

$$F(V)=\frac{1}{2}\varepsilon\begin{bmatrix}V_f & V_p\end{bmatrix}\begin{bmatrix}C_{ff} & C_{fp}\\ C_{pf} & C_{pp}\end{bmatrix}\begin{bmatrix}V_f\\ V_p\end{bmatrix}-\begin{bmatrix}V_f & V_p\end{bmatrix}\begin{bmatrix}T_{ff} & T_{fp}\\ T_{pf} & T_{pp}\end{bmatrix}\begin{bmatrix}\rho_f\\ \rho_p\end{bmatrix} \quad (10.235)$$

where

subscript f is the free node; subscript p is the prescribed node;

ρ_f is the submatrix containing the values of ρ at free node; ρ_p is the submatrix containing the values of ρ at the prescribed node.

Minimizing $F(V)$ with respect to V_f, namely,

$$\frac{\partial F}{\partial V_f}=0$$

gives

$$\varepsilon\left(C_{ff}V_f+C_{pf}V_p\right)-\left(T_{ff}\rho_f+T_{fp}\rho_p\right)=0$$

or

$$\left[C_{ff}\right]\left[V_f\right]=-\left[C_{fp}\right]\left[V_p\right]+\frac{1}{\varepsilon}\left[T_{ff}\right]\left[\rho_f\right]+\frac{1}{\varepsilon}\left[T_{fp}\right]\left[\rho_p\right]. \quad (10.236)$$

Indeed, equation (10.236) can be written.

$$[A][V]=[B] \quad (10.237)$$

where

$[A]=\left[C_{ff}\right]$, $[V]=\left[V_f\right]$, and $[B]$ is the right-hand side of equation (10.236). Equation (10.237) can be solved to determine $[V]$ either by matrix inversion or Gaussian elimination technique.

10.13.2.1.3 Solution of Wave's Equation $\nabla^2\Phi + k^2\Phi = g$ with FEM

A typical wave equation is the inhomogeneous scalar Helmholtz's equation

$$\nabla^2\Phi + k^2\Phi = g \tag{10.238}$$

where

Φ is the potential (for waveguide problem, $\Phi = H_z$ for *TE* mode or E_z for *TM* mode) to be determined, g is the source function, and $k = \omega\sqrt{\mu\varepsilon}$ is the wave number of the medium. The following three distinct special cases of equation (10.238) should be noted:

1. $k = 0$ and $g = 0$; Laplace's equation;
2. $k = 0$; Poisson's equation; and
3. k is an unknown, $g = 0$: homogeneous, scalar Helmholtz's equation.

It is known that the variational solution to the operator equation

$$L\Phi = g \tag{10.239}$$

is obtained by examining the functional

$$I(\Phi) = < L, \Phi > -2 < \Phi, g > \tag{10.240}$$

where L is an operator (differential, integral, or integrodifferential), g is the unknown excitation or source, and Φ is the unknown function to be determined (here is the potential).

Therefore, the solution of equation (10.238) is equivalent to satisfying the boundary conditions and minimizing the functional.

$$I(\Phi) = \frac{1}{2}\iint\left[\left|\nabla\Phi\right|^2 - k^2\Phi^2 + 2\Phi g\right] dS. \tag{10.241}$$

Note that if other than the natural boundary conditions (i.e., Dirichlet of homogenous Neumann conditions) must be satisfied, appropriate terms must be added to the functional. The potential Φ and the source function g can be expressed now in terms of the shape functions α_i over a triangle element as

$$\Phi_m(x, y) = \sum_{i=1}^{3} \alpha_i \Phi_{mi} \tag{10.242}$$

where

ϕ_{mi} is the value of Φ at the nodal point i of element m.

And

$$g_m(x,y) = \sum_{i=1}^{3} \alpha_i g_{mi} \tag{10.243}$$

g_{mi} is the value of g at the nodal point i of element m.

Substituting equations (10.242) and (10.243) into equation (10.241) gives

$$\begin{aligned} I(\Phi_m) &= \frac{1}{2}\sum_{i=1}^{3}\sum_{j=1}^{3}\Phi_{mi}\Phi_{mj}\iint \nabla\alpha_i \cdot \nabla\alpha_j \, dS - \frac{k^2}{2}\sum_{i=1}^{3}\sum_{j=1}^{3}\Phi_{mi}\Phi_{mj}\iint \alpha_i\alpha_j \, dS \\ &\quad + \sum_{i=1}^{3}\sum_{j=1}^{3}\Phi_{mi} g_{mj}\iint \alpha_i\alpha_j \, dS \\ &= \frac{1}{2}[\Phi_m]^t\left[C^{(m)}\right][\Phi_m] - \frac{k^2}{2}[\Phi_m]^t\left[T^{(m)}\right][\Phi_m] + [\Phi_m]^t\left[T^{(m)}\right][G_m] \end{aligned} \tag{10.244}$$

where

$[\Phi_m] = \left[\Phi_{m1,}\Phi_{m2,}\Phi_{m3}\right]^t, [G_m] = \left[g_{m1,}g_{m2,}g_{m3}\right]^t,$ and $\left[C^{(m)}\right]$ and $\left[T^{(m)}\right]$ are defined in equations (10.158) and (10.185), respectively.

The equation (10.244) is for a single element, but it can be applied for all N elements in the solution region. Therefore,

$$I(\Phi) = \sum_{m=1}^{N} I(\Phi_m). \tag{2.108}$$

From equations (10.244) and (10.245), $I(\Phi)$ can be expressed in matrix form as

$$I(\Phi) = \frac{1}{2}[\Phi]^t[C][\Phi] - \frac{k^2}{2}[\Phi]^t[T][\Phi] + [\Phi]^t[T][G] \tag{10.246}$$

where

$$[\Phi] = \left[\Phi_1, \quad \Phi_2, \quad . \quad . \quad . \quad ,\Phi_N\right]^t, \tag{10.247a}$$

$$[G] = \left[g_1, \quad g_2, \quad . \quad . \quad . \quad ,g_N\right]^t \tag{10.247b}$$

$[C]$, and $[T]$ are global matrices consisting of local matrices $\left[C^{(m)}\right]$ and $\left[T^{(m)}\right]$, respectively.

Now, if free nodes are numbered first and the prescribed nodes last, and considering the source function $g=0$, we can write equation (2.109) as

$$I=\frac{1}{2}\begin{bmatrix}\Phi_f & \Phi_p\end{bmatrix}\begin{bmatrix}C_{ff} & C_{fp}\\ C_{pf} & C_{pp}\end{bmatrix}\begin{bmatrix}\Phi_f\\ \Phi_p\end{bmatrix}-\frac{k^2}{2}\begin{bmatrix}\Phi_f & \Phi_p\end{bmatrix}\begin{bmatrix}T_{ff} & T_{fp}\\ T_{pf} & T_{pp}\end{bmatrix}\begin{bmatrix}\Phi_f\\ \Phi_p\end{bmatrix}. \quad (10.248)$$

By setting $\dfrac{\partial I}{\partial \phi_f}=0$, gives

$$\begin{bmatrix}C_{ff} & C_{fp}\end{bmatrix}\begin{bmatrix}\Phi_f\\ \Phi_p\end{bmatrix}-k^2\begin{bmatrix}T_{ff} & T_{fp}\end{bmatrix}\begin{bmatrix}\Phi_f\\ \Phi_p\end{bmatrix}=0\,. \quad (10.249)$$

For TM modes, $\Phi_p=0$ and hence

$$\left[C_{ff}-k^2T_{ff}\right]\Phi_f=0\,. \quad (10.250)$$

Premultiplying equation (10.250) by T_{ff}^{-1}, gives

$$\left[T_{ff}^{-1}C_{ff}-k^2I\right]\Phi_f=0\,. \quad (10.251)$$

By letting

$$T_{ff}^{-1}C_{ff}=A\,,\qquad k^2=\beta\,,\ \Phi_f=X\,, \quad (10.252a)$$

and U is a unit matrix,

we can obtain the standard eigenvalue problem

$$(A-\beta U)X=0\,. \quad (10.252b)$$

Any standard procedure may be used to obtain some or all of the eigenvalues $\beta_1,\beta_2,\ldots,\beta_{nf}$ and eigenvectors $X_1,X_2,\ldots,X_{nf}$, where nf is the number of free nodes. The eigenvalues are always real since C and T are symmetric.

The solution of the algebraic eigenvalue problems in equation (10.252) furnishes eigenvalues and eigenvectors, which form good approximations to the eigenvalues and eigenfunctions of the Helmholtz problem, i.e., the cutoff wavelengths and field distribution patterns of the various modes possible in a given waveguide.

The solution of the problem of equation (10.238) is summarized in equation (10.251), and can be viewed as the finite element solution of homogeneous waveguides. The idea can be extended to handle inhomogeneous waveguide problems. However, in applying FEM to inhomogeneous problems, a serious difficulty is the appearance of spurious, nonphysical solutions. There are several techniques that have been proposed to overcome the difficulty.

10.14 AUTOMATIC MESH GENERATION

It is a fact that one of the major difficulties encountered in the finite element analysis of continuum problems is the tedious and time-consuming effort required in data preparation. Indeed, efficient finite element programs must have node and element generating schemes, referred to collectively as *mesh generators*. Automatic mesh generation minimizes the input data required to specify a problem. In fact, it not only reduces the time involved in data preparation, it eliminates human errors that are introduced when data preparation is preformed manually. Furthermore, combining the automatic mesh generation program with computer graphics is particularly valuable since the output can be monitored visually.

A number of mesh generation algorithms of varying degrees of automation have been proposed. In this section, we focus on two types of domains, rectangular domains and arbitrary domains.

10.14.1 Rectangular Domains

Since some applications of FEM to EM problems involve simple rectangular domains, we consider the generation of simple meshes. Now, let us consider a rectangular solution region of size $a \times b$, as shown in Figure 10.13. The goal here is to divide the region into rectangular elements, each of which is later divided into two triangular elements.

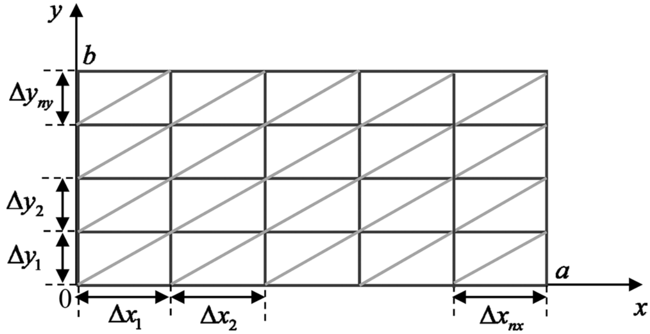


FIGURE 10.13 Discretization of a rectangular region into a nonuniform mesh.

Suppose n_x and n_y are the number of divisions in x and y directions, the total number of elements and nodes are, respectively, given by

$$n_m = 2n_x n_y$$
$$n_d = (n_x + 1)(n_y + 1). \tag{10.253}$$

As a result, it is easy to figure out from Figure 10.13 a systematic way of numbering the elements and nodes. Indeed, to obtain the global coordinates (x, y) for each node, we need an array containing Δx_i, $i = 1,2,\ldots,n_x$ and Δy_j, $j = 1,2,\ldots,n_y$, which are, respectively, the distances between nodes in the x and y directions. If the order of node numbering is from left to right along horizontal rows and from bottom to top along the vertical rows, then the first node is the origin $(0,0)$. The next node is obtained as $x \to x + \Delta x_1$ while $y = 0$ remains unchanged. The following node $x \to x + \Delta x_2$, $y = 0$, and so on until Δx_i are exhausted. We start the second next horizontal row by starting with $x = 0$, $y \to y + \Delta y_1$ and increasing x until Δx_i are exhausted. We repeat the process until the last node $(n_x + 1)(n_y + 1)$ is reached, i.e., when Δx_i and Δy_i are exhausted simultaneously.

The procedure presented here allows for generating uniform and nonuniform meshes. A mesh is uniform if all Δx_i are equal and all Δy_i are equal; it is nonuniform otherwise. A nonuniform mesh is preferred if it is known in advance that the parameter of interest varies rapidly in some parts of the solution domain. This allows a concentration of relatively small elements in the regions where the parameter changes rapidly, particularly since these regions are often of greatest interest in the solution. Additionally, without the preknowledge of the rapid change in the unknown parameter, a uniform mesh can be used. In that case, we set

$$\Delta x_1 = \Delta x_2 = . \quad . \quad . = h_x$$
$$\Delta y_1 = \Delta y_2 = . \quad . \quad . = h_y \tag{10.254}$$

where

$h_x = a / n_x$ and $h_y = a / n_y$.

In some cases, we also need a list of prescribed nodes. If we assume that all boundary points have prescribed potentials, the number n_p of prescribed nodes is given by

$$n_p = 2(n_x + n_y). \tag{10.255}$$

A simple way to obtain the list of boundary points is to enumerate points on the bottom, right, top, and left of the rectangular region in that order.

10.14.2 Arbitrary Domains

The basic steps involved in mesh generation are as follows:

(A) subdivide the solution region into few quadrilateral blocks,

(B) separately subdivide each block into elements,

(C) connect individual blocks.

A. *Definition of Blocks*

The solution region is subdivided into quadrilateral blocks. Subdomains with different constitutive parameters $(\sigma, \mu, \varepsilon)$ must be represented by separate blocks. As input data, we specify block topologies and the coordinates at eight points describing each block. Each block is represented by an eight-node quadratic isoparametric element. With natural coordinate system (ζ, η), the x and y coordinates are represented as

$$x(\zeta, \eta) = \sum_{i=1}^{8} \alpha_i(\zeta, \eta) x_i \tag{10.256}$$

$$y(\zeta, \eta) = \sum_{i=1}^{8} \alpha_i(\zeta, \eta) y_i \tag{10.257}$$

where $\alpha_i(\zeta, \eta)$ is a shape function associated with node i, and (x_i, y_i) are the coordinates of node i defining the boundary of the quadrilateral block as shown in Figure 10.14.

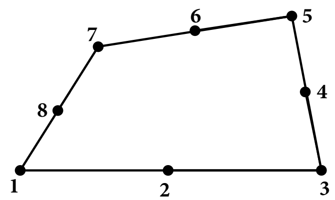


FIGURE 10.14 Typical quadrilateral block.

The shape functions are expressed in terms of the quadratic or parabolic isoparametric elements shown in Figure 10.15.

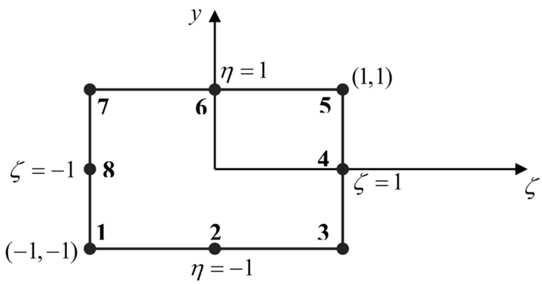


FIGURE 10.15 Eight-node serendipity element.

They are given by:

$$a_i = \frac{1}{4}(1+\zeta\zeta_i)(1+\eta\eta_i)(\zeta\zeta_i+\eta\eta_i+1), \quad i=1,3,5,7. \qquad (10.258)$$

For corner nodes,

$$a_i = \frac{1}{2}\zeta_i^2(1+\zeta\zeta_i)(1-\eta^2)+\frac{1}{2}\eta_i^2(1+\eta\eta_i+1)(1-\zeta^2), \quad i=2,4,6,8. \quad (10.259)$$

For midside nodes, note the following properties of the shape functions:

(1) They satisfy the conditions.

$$\sum_{i=1}^{n} a_i(\zeta,\eta) = 1 \qquad (10.260a)$$

$$a_i(\zeta_j,\eta_j) = \begin{cases} 1, & i=j \\ 0, & i\neq j \end{cases} \qquad (10.260b)$$

(2) They become quadratic along element edges $(\zeta = \pm 1, \eta = \pm 1)$.

B. *Subdivision of Each Block*

Furthermore, for each block, we specify $N\ DIV\ X$ and $N\ DIV\ \mathrm{Y}$, the number of element subdivisions to be made in the ζ and η directions, respectively. In addition, we specify the weighting factors $(W_\zeta)_i$ and $(W_\eta)_i$ allow for graded mesh within a block. It is essential to know that, in specifying $N\ DIV\ X$, $N\ DIV\ Y$, $\left(W_\zeta\right)_i$, and $\left(W_\eta\right)_i$ care must be taken to ensure that the subdivision along block interfaces (for adjacent blocks) are compatible. We initialize ζ and η to a value of -1 so that the natural coordinates are incremented according to

$$\zeta_i = \zeta_i + \frac{2(W_\zeta)_i}{W_\zeta^T \times F} \tag{10.261}$$

$$\eta_i = \eta_i + \frac{2(W_\eta)_i}{W_\eta^T \times F} \tag{10.262}$$

where

$$W_\zeta^T = \sum_{j=1}^{N\ DIV\ X} \left(W_\zeta\right)_j \tag{10.263a}$$

$$W_\eta^T = \sum_{j=1}^{N\ DIV\ X} \left(W_\eta\right)_j \tag{10.263b}$$

and

$$F = \begin{cases} 1, & \text{for linear elements} \\ 2, & \text{for quadratic elements} \end{cases}.$$

Now, there are three element types permitted: (1) linear four-node quadrilateral elements, (2) linear three-node triangle elements, and (3) quadratic eight-node isoparametric elements.

C. *Connection of Individual Blocks*

After subdividing each block and numbering its nodal points separately, it is necessary to connect the blocks and have each node numbered uniquely. This is accomplished by comparing the coordinates of all nodal points and assigning the same number to all nodes having identical coordinates. In other words, we compare the coordinates of node 1 with all other nodes, and then node 2 with other nodes, etc., until all repeated nodes are eliminated.

10.15 HIGHER-ORDER ELEMENTS

Finite elements use higher-order elements. The shape function or interpolation polynomial of the order two or more is called a *higher-order element*.

To emphasize, the accuracy of a finite element solution can be improved by using finer mesh or using higher-order elements or both. Desai and Abel studied mesh refinement versus higher-order elements in [44]. Generally, fewer higher-order elements are needed to achieve the same degree of accuracy in the final results. Moreover, the higher-order elements are particularly useful when the gradient of the field variable is expected to vary rapidly.

10.15.1 Pascal Triangle

High-order triangular elements can be systematically developed with the aid of the so-called Pascal triangle given in Figure 10.16. The family of finite elements generated in this matter with the distribution of nodes is illustrated in Figure 10.17. Note that in higher-order elements, some secondary (side and/or interior) nodes are introduced in addition to the primary (corner) nodes so as to produce exactly the right number of nodes required to define the shape function of that order.

a_1 Constant term, $n = 0$

a_2x a_3y Linear term, $n = 1$

a_4x^2 a_5xy a_6y^2 Quadratic term, $n = 2$

a_7x^3 a_8x^2y a_9xy^2 $a_{10}y^3$ Cubic term, $n = 3$

FIGURE 10.16 The Pascal triangle (2D).

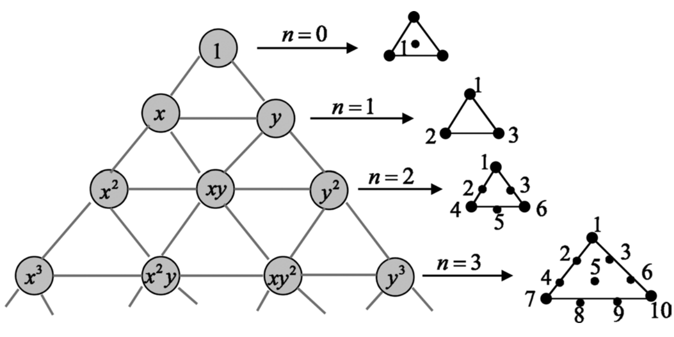


FIGURE 10.17 Pascal triangle (2D) and the associated polynomial basis functions degree $n = 1$ to 3.

Indeed, the Pascal triangle contains terms of the basic functions of various degrees in variable x and y. An arbitrary function $\Phi_i(x, y)$ can be approximated in an element in terms of a complete nth-order polynomial as

$$\Phi_i(x,y) = \sum_{i=1}^{r} a_i \Phi_i \tag{10.264}$$

where

$$r = \frac{1}{2}(n+1)(n+2)\,. \tag{10.265}$$

r is the number of terms in complete polynomials (also the number of nodes in the triangle). For example, for the third-order ($n = 3$) or cubic (ten-node) triangular elements,

$$\Phi_m(x,y) = a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 xy + a_6 y^2 + a_7 x^3 + a_8 x^2 y + a_9 xy^2 + a_{10} y^3\,. \tag{10.266}$$

Equation (10.266) has ten coefficients, and hence the element must have ten nodes. It is also complete through the third-order terms. A systematic derivation of the interpolation function a for the higher-order elements involves the use of the local coordinates.

10.15.2 Local Coordinates

Now, the triangular local coordinates (η_1, η_2, η_3) are related to Cartesian coordinates (x, y) as

$$x = \eta_1 x_1 + \eta_2 x_2 + \eta_3 x_3 \tag{10.267a}$$

$$y = \eta_1 y_1 + \eta_2 y_2 + \eta_3 y_3\,. \tag{10.267b}$$

The local coordinates are dimensionless, with values ranging from 0 to 1. Furthermore, by definition, η_i at any point within the triangle is the ratio of the perpendicular distance from the point to the side opposite to vertex i to the length of the altitude drawn from vertex i. Therefore, from Figure 10.18, the value of η_i at P, for example, is given by the ratio of the perpendicular distance d from the side opposite vertex 1 to the altitude h of that side, namely,

$$\eta_1 = \frac{d}{h}\,. \tag{10.268}$$

Alternatively, from Figure 10.15, η_i at P can be defined as

$$\eta_i = \frac{A_i}{A} \tag{10.269}$$

so that

$$\eta_1 + \eta_2 + \eta_3 = 1 \tag{10.270}$$

Since $A_1 + A_2 + A_3 = A$. The local coordinates η_i in equation (10.269) are also called *area coordinates*. The variation of (η_1, η_2, η_3) inside an element is shown in Figure 10.19.

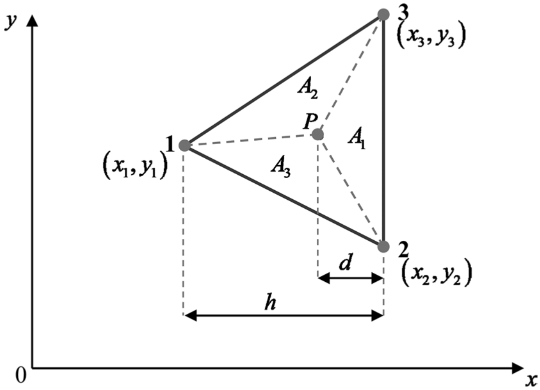


FIGURE 10.18 Definition of local coordinates.

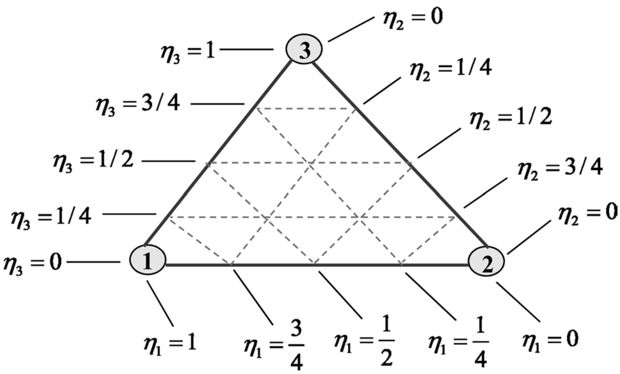


FIGURE 10.19 Variation of local coordinates.

Although the coordinates η_1, η_2, and η_3 are used to define a point P, only two are independent since they must satisfy equation (10.270). The inverted form of equations (10.267) and (10.268) is

$$\eta_i = \frac{1}{2A}\left(c_i + b_i x + a_i y\right) \tag{10.271}$$

where

$$a_i = x_k - x_j ,$$

$$b_i = y_j - y_k$$

$$c_i = x_j y_k - x_k y_j$$

$$A = \text{area of the triangle } = \frac{1}{2}(b_1 a_2 - b_2 a_1), \tag{10.272}$$

and (i, j, k) is an even permutation of $(1,2,3)$. The differentiation and integration in local coordinates are carried out using [47]:

$$\frac{\partial f}{\partial \eta_1} = a_2 \frac{\partial f}{\partial x} - b_2 \frac{\partial f}{\partial y} \tag{10.273a}$$

$$\frac{\partial f}{\partial \eta_2} = -a_1 \frac{\partial f}{\partial x} + b_1 \frac{\partial f}{\partial y} \tag{10.273b}$$

$$\frac{\partial f}{\partial x} = \frac{1}{2A}\left(b_1 \frac{\partial f}{\partial \eta_1} + b_2 \frac{\partial f}{\partial \eta_2} \right) \tag{10.273c}$$

$$\frac{\partial f}{\partial y} = \frac{1}{2A}\left(a_1 \frac{\partial f}{\partial \eta_1} + a_2 \frac{\partial f}{\partial \eta_2} \right) \tag{10.273d}$$

$$\iint f \; dS = 2A \int_0^1 \left(\int_0^{1-\eta_2} f(\eta_1, \eta_2)\, d\eta_1 \right) d\eta_2 \tag{10.273e}$$

$$\iint \eta_1^i \eta_2^j \eta_3^k \; dS = \left(\frac{i!\, j!\, k!}{(i+j+k+2)!} \right) \times 2A \tag{10.273f}$$

$$dS = 2A\, d\eta_1 d\eta_2 \tag{10.273g}$$

10.15.3 Shape Functions

Now, we may express the shape function for higher-order elements in terms of local coordinates. Indeed, sometimes, it is convenient to label each point in the finite elements in Figure 10.17 with three integers, $i, j,$ and k from which its local coordinates (η_1, η_2, η_3) can be found or vice versa. For instance, at each point P_{ijk}

$$(\eta_1,\eta_2,\eta_3)=\left(\frac{i}{n},\frac{j}{n},\frac{k}{n}\right). \tag{10.274}$$

Thus, if a value of Φ, say Φ_{ijk}, is prescribed at each point P_{ijk}, equation (10.264) can be expressed as

$$\Phi(\eta_1,\eta_2,\eta_3)=\sum_{i=1}^{r}\sum_{j=1}^{r-i}a_{ijk}(\eta_1,\eta_2,\eta_3)\Phi_{ijk} \tag{10.275}$$

where

$$a_l=a_{ijk}=p_i(\eta_1)p_j(\eta_2)p_k(\eta_3), \quad l=1,2,... \tag{10.276}$$

$$p_e(\eta)=\begin{cases}\dfrac{1}{e!}\displaystyle\prod_{t=0}^{e-1}(n\eta-t), & e>0\\ 1, & e=0\end{cases} \tag{10.277}$$

and $e\in(i,j,k)$. Further, $p_e(\eta)$ may also be written as

$$p_e(\eta)=\frac{(n\eta-e+1)}{e}\times p_{e-1}(\eta), \quad \text{e>0} \tag{10.278}$$

where $p_0(\eta)=1$.

The relationships between the subscript $q\in\{1,2,3\}$ on η_q, $l\in\{1,2,..,r\}$ on a_l, and $e\in(i,j,k)$ on p_e and P_{ijk} in equations (10.276) to (10.278) are illustrated in Figure 10.20 for n ranging from 1 to 3. Furthermore, point P_{ijk} will be written as P_n for conciseness.

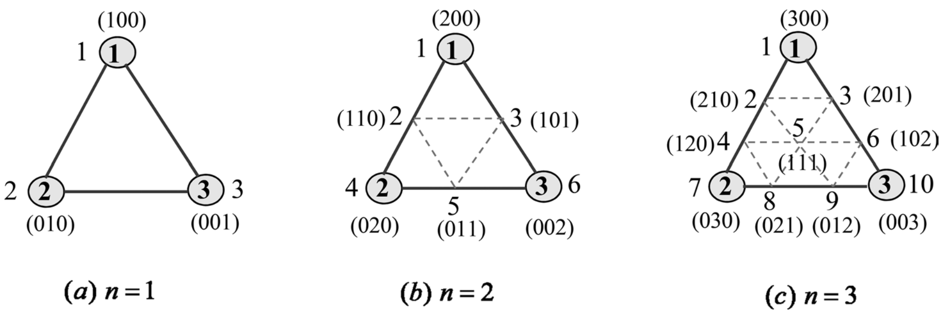


FIGURE 10.20 Distribution of nodes over triangles for n = 1 to 3. The triangles are in standard position.

Notice from equations (10.277) or (10.278) that

$$
\begin{aligned}
p_0(\eta) &= 1 \\
p_1(\eta) &= n\eta \\
p_2(\eta) &= \frac{1}{2}(n\eta - 1)n\eta \\
p_3(\eta) &= \frac{1}{6}(n\eta - 2)(n\eta - 1)n\eta \quad \text{, etc.}
\end{aligned}
\tag{10.279}
$$

Indeed, by substituting equation (10.279) into equation (10.276) gives the shape functions α_l for nodes $l = 1, 2, .., r$, as shown in Table 10.1 for n = 1 to 3. In addition, observe that each α_l takes the value of 1 at node l and value 0 at all other nodes in the triangle. It can be verified by using equation (10.274) in conjunction with Figure 10.20.

TABLE 10.1 Polynomial Basic Functions $\alpha_l(\eta_1, \eta_2, \eta_3)$ for First, Second, and Third.

***n* = 1**	***n* = 2**	***n* = 3**
$\alpha_1 = \eta_1$	$\alpha_1 = \eta_1(2\eta_1 - 1)$	$\alpha_1 = \frac{1}{2}\eta_1(3\eta_1 - 2)(3\eta_1 - 1)$
$\alpha_2 = \eta_2$	$\alpha_2 = 4\eta_1\eta_2$	$\alpha_2 = \frac{9}{2}\eta_1(3\eta_1 - 1)\eta_2$
$\alpha_3 = \eta_3$	$\alpha_3 = 4\eta_1\eta_3$	$\alpha_3 = \frac{9}{2}\eta_1(3\eta_1 - 1)\eta_3$
	$\alpha_4 = \eta_2(2\eta_2 - 1)$	$\alpha_4 = \frac{9}{2}\eta_1(3\eta_2 - 1)\eta_2$
	$\alpha_5 = 4\eta_2\eta_3$	$\alpha_5 = 27\eta_1\eta_2\eta_3$
	$\alpha_6 = \eta_3(2\eta_3 - 1)$	$\alpha_6 = \frac{9}{2}\eta_1(3\eta_3 - 1)\eta_3$
		$\alpha_7 = \frac{1}{2}\eta_2(3\eta_2 - 2)(3\eta_2 - 1)$
		$\alpha_8 = \frac{9}{2}\eta_2(3\eta_2 - 1)\eta_3$
		$\alpha_9 = \frac{9}{2}\eta_2(3\eta_3 - 1)\eta_3$
		$\alpha_{10} = \frac{1}{2}\eta_3(3\eta_3 - 2)(3\eta_3 - 1)$

10.15.4 Fundamental Matrices

The fundamental matrices [T] and [Q] for triangle elements can be derived using the shape functions in Table 10.1. The matrix [T] is defined as

$$T_{ij} = \iint \alpha_i \alpha_j dS . \tag{10.280}$$

From Table 10.1, we substitute α_l in equation (10.280) and apply equations (10.273f) and (10.273g) to obtain elements of T. For example, for $n = 1$,

$$T_{ij} = 2A \int_0^1 \int_0^{1-\eta_2} \eta_i \eta_j \, d\eta_1 d\eta_2 . \tag{10.281}$$

Furthermore, when $i \neq j$, T_{ij} can be written as

$$T_{ij} = \frac{2A(1!)(1!)(0!)}{4!} = \frac{A}{12}, \tag{10.282a}$$

but, when $i = j$,

$$T_{ij} = \frac{2A(2!)}{4!} = \frac{A}{6} . \tag{10.282b}$$

Thus,

$$T = \frac{A}{12} \begin{bmatrix} 2 & 1 & 1 \\ 1 & 2 & 1 \\ 1 & 1 & 2 \end{bmatrix} . \tag{10.283}$$

Now, by following the same procedure, higher-order T matrices can be obtained. The T matrices for orders up to $n = 3$ are tabulated in Table 10.2 where the factor A, the area of the element, has been expressed. The actual matrix elements are obtained from Table 2.2 by multiplying the tabulated numbers by A and dividing by the indicated common denominator. Indeed, the following properties of the T matrix are worth knowing:

(1) T is symmetric with positive elements;

(2) elements of T all add up to the area of the triangle, that is, $\sum_i^r \sum_j^r T_{ij} = A$, since by definition $\sum_{l=1}^r \alpha_l = 1$ at any point within the element;

(3) elements for which the two triple subscripts from similar permutations are equal, that is, $T_{ijk,peq} = T_{ikj,peq} = T_{kij,epq} = T_{kji,eqp} = T_{jki,qep} = T_{jik,qpe}$; this should be obvious from equations (2.280) and (10.276).

As a result, the above properties are not only useful in checking the matrix; they have proved useful in saving computer time and storage.

TABLE 10.2 Table of T matrices for n = 1 to 3.

$n = 1$	Common denominator = 12 $T = \begin{bmatrix} 2 & 1 & 1 \\ 1 & 2 & 1 \\ 1 & 1 & 2 \end{bmatrix}$
$n = 2$	Common denominator = 180 $T = \begin{bmatrix} 6 & 0 & 0 & -1 & -4 & -1 \\ 0 & 32 & 16 & 0 & 16 & -4 \\ 0 & 16 & 32 & -4 & 16 & 0 \\ -1 & 0 & -4 & 6 & 0 & -1 \\ -4 & 16 & 16 & 0 & 32 & 0 \\ -1 & -4 & 0 & -1 & 0 & 6 \end{bmatrix}$
$n = 3$	Common denominator = 6720 $T = \begin{bmatrix} 76 & 18 & 18 & 0 & 36 & 0 & 11 & 27 & 27 & 11 \\ 18 & 540 & 270 & -189 & 162 & -135 & 0 & -135 & -54 & 27 \\ 18 & 270 & 540 & -135 & 162 & -189 & 27 & -54 & -135 & 0 \\ 0 & -189 & -135 & 540 & 162 & -54 & 18 & 270 & -135 & 27 \\ 36 & 162 & 162 & 162 & 1944 & 162 & 36 & 162 & 162 & 36 \\ 0 & -135 & -189 & -54 & 162 & 540 & 27 & -135 & 270 & 18 \\ 11 & 0 & 27 & 18 & 36 & 27 & 76 & 18 & 0 & 11 \\ 27 & -135 & -54 & 270 & 162 & -135 & 18 & 540 & -189 & 0 \\ 27 & -54 & -135 & -135 & 162 & 270 & 0 & -189 & 540 & 18 \\ 11 & 27 & 0 & 27 & 36 & 18 & 11 & 0 & 18 & 76 \end{bmatrix}$

In equation (10.227), elements of $[C]$ matrix are defined by

$$C_{ij} = \iint \left(\frac{\partial \alpha_i}{\partial x} \frac{\partial \alpha_j}{\partial x} + \frac{\partial \alpha_i}{\partial y} \frac{\partial \alpha_j}{\partial y} \right) dS \,. \tag{10.284}$$

By applying equations (10.273a) to (10.273d) to equation (2.147), it can be shown that:

$$C_{ij} = \frac{1}{2A}\sum_{q=1}^{3}\cot\theta_q \iint\left(\frac{\partial \alpha_i}{\partial \eta_{q+1}} - \frac{\partial \alpha_i}{\partial \eta_{q-1}}\right)\left(\frac{\partial \alpha_j}{\partial \eta_{q+1}} - \frac{\partial \alpha_j}{\partial \eta_{q-1}}\right) dS$$

or

$$C_{ij} = \sum_{q=1}^{3} Q_{ij}^{(q)} \cot\theta_q \tag{10.285}$$

where θ_q is the include angle of vertex $q \in \{1,2,3\}$ of the triangle and

$$Q_{ij}^{(q)} = \iint\left(\frac{\partial \alpha_i}{\partial \eta_{q+1}} - \frac{\partial \alpha_i}{\partial \eta_{q-1}}\right)\left(\frac{\partial \alpha_j}{\partial \eta_{q+1}} - \frac{\partial \alpha_j}{\partial \eta_{q-1}}\right) d\eta_1 d\eta_2 . \tag{10.286}$$

It is clear that matrix C depends on the triangle shape, whereas the matrices $Q^{(q)}$ do not. The $Q^{(1)}$ matrices for n = 1 to 3 are tabulated in Table 10.3.

The following properties of Q matrices should be noted as:

(1) they are symmetric;

(2) the row and column sums of any Q matrix are zero, that is, $\sum_{i=1}^{r} Q_{ij}^{(q)} = 0 = \sum_{j=1}^{r} Q_{ij}^{(q)}$ so that the C matrix is singular.

$Q^{(2)}$ and $Q^{(3)}$ are easily obtained from $Q^{(1)}$ by row and column permutations so that the matrix C for any triangular element is constructed easily if $Q^{(1)}$ is known.

TABLE 10.3 Table of Q matrices for n = 1 to 3.

n = 1	Common denominator = 2 $Q = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 1 & -1 \\ 0 & -1 & 1 \end{bmatrix}$

$n = 2$	Common denominator = 6 $$Q=\begin{bmatrix}0&0&0&0&0&0\\0&8&-8&0&0&0\\0&-8&8&0&0&0\\0&0&0&3&-4&1\\0&0&0&-4&8&-4\\0&0&0&1&-4&3\end{bmatrix}$$
$n = 3$	Common denominator = 80 $$Q=\begin{bmatrix}0&0&0&0&0&0&0&0&0&0\\0&135&-135&-27&0&27&3&0&0&-3\\0&-135&135&27&0&-27&-3&0&0&3\\0&-27&27&135&-162&27&3&0&0&-3\\0&0&0&-162&324&-162&0&0&0&0\\0&27&-27&27&-162&135&-3&0&0&3\\0&3&-3&3&0&-3&34&-54&27&-7\\0&0&0&0&0&0&-54&135&-108&27\\0&0&0&0&0&0&27&-108&135&-54\\0&-3&3&-3&0&3&-7&27&-54&34\end{bmatrix}$$

For example, for $n=1$, the rotation matrix is basically derived from Figure 10.21 as

$$R=\begin{bmatrix}0&0&1\\1&0&0\\0&1&0\end{bmatrix} \tag{10.287}$$

where $R_{ij}=1$ node i is replaced by node j after one counterclockwise rotation, or $R_{ij}=0$ otherwise.

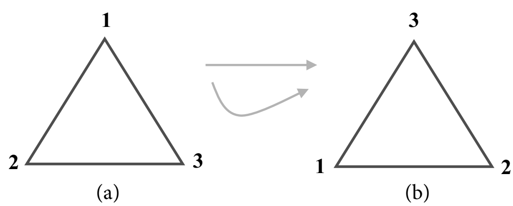


FIGURE 10.21 One counterclockwise rotation of the triangle in (a) gives the triangle in (b).

Moreover, Table 10.4 presents the R matrices for $n = 1$ to 3. Note that each row or column of R has only one nonzero element since R is essentially a unit matrix with rearranged elements.

TABLE 10.4 Table of R matrices for n = 1 to 3.

$n = 1,$	$R = \begin{bmatrix} 0 & 0 & 1 \\ 1 & 0 & 0 \\ 0 & 1 & 0 \end{bmatrix}$
$n = 2,$	$R = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \end{bmatrix}$
$n = 3,$	$R = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \end{bmatrix}$

Now, once the R is known, we can obtain $Q^{(2)}$ and $Q^{(3)}$ as

$$Q^{(2)} = RQ^{(1)}R^t \tag{10.288a}$$

$$Q^{(3)} = RQ^{(2)}R^t \tag{10.288b}$$

where R^t is the transpose of R.

10.16 THREE-DIMENSIONAL ELEMENT

In this section, we will discuss the finite element analysis of Helmholtz's equation in three dimensions, i.e.,

$$\nabla^2\Phi + k^2\Phi = g\,. \tag{10.289}$$

First, we divide the solution region into tetrahedral or hexahedral (rectangular prism) elements as in Figure 10.22.

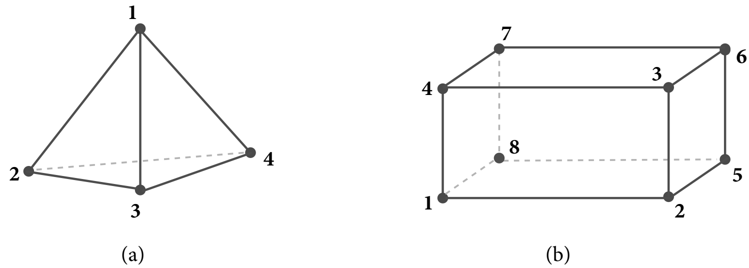


FIGURE 10.22 Three-dimensional elements: (a) Four-node or linear-order tetrahedral, (b) eight-node or linear-order hexahedral.

Now, assuming a four-node tetrahedral element, the function Φ is represented within element by

$$\Phi_m = a + bx + cy + dz\,. \tag{10.290}$$

The same applies to the function g. Since equation (10.290) must be satisfied at the four nodes of the tetrahedral elements,

$$\Phi_{mi} = a + bx_i + cy_i + dz_i, \qquad i = 1,2,3,4\,. \tag{10.291}$$

Therefore, we have four simultaneous equations with the potentials $V_{m1}, V_{m2}, V_{m3},$ and V_{m4} at nodes 1, 2, 3, and 4, respectively, i.e.,

$$\begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \\ V_{m4} \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{bmatrix} \begin{bmatrix} a \\ b \\ c \\ d \end{bmatrix}. \tag{10.292}$$

The coefficients $a, b, c,$ and d are determined from equation (10.291)

$$\begin{bmatrix} a \\ b \\ c \\ d \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{bmatrix}^{-1} \begin{bmatrix} V_{m1} \\ V_{m2} \\ V_{m3} \\ V_{m4} \end{bmatrix}. \tag{10.293}$$

The determinant of the system of equations is

$$\det = \begin{vmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{vmatrix} = 6v, \tag{10.294}$$

where v is the volume of the tetrahedron. By finding $a, b, c,$ and d, we can express $\boldsymbol{\Phi}_m$ as,

$$\boldsymbol{\Phi}_m = \sum_{i=1}^{4} a_i(x, y)\boldsymbol{\Phi}_{mi} \tag{10.295}$$

where

$$a_1 = \frac{1}{6v}\begin{vmatrix} 1 & x & y & z \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{vmatrix} \tag{10.296a}$$

$$a_2 = \frac{1}{6v}\begin{vmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x & y & z \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{vmatrix}, \tag{10.296b}$$

$$a_3 = \frac{1}{6v}\begin{vmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x & y & z \\ 1 & x_4 & y_4 & z_4 \end{vmatrix}, \tag{10.296c}$$

$$a_4 = \frac{1}{6v}\begin{vmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x & y & z \end{vmatrix}. \tag{10.296d}$$

Indeed, for higher-order approximation, the matrices for $a\,s$ become large in size and we resort to local coordinates. The existence of integration equations for local coordinates can simplify the evaluation of the fundamental matrices T and Q.

Now, for the tetrahedral element, the local coordinates are η_1, η_2, η_3, and η_4, each perpendicular to a side. They are defined at a given point as the ratio of the distance from that point to the appropriate apex to the perpendicular distance from the side to the opposite apex. In addition, they can be interpreted as volume ratios, that is, at a point *P*

$$\eta_i = \frac{v_i}{v} \tag{10.297}$$

where v_i is the volume bound by *P* and face *i*. It is evident that

$$\sum_{i=1}^{4} \eta_i = 1 . \tag{10.298}$$

Note that, the following properties are useful in evaluating integration involving local coordinates:

$$dv = 6v d\eta_1 d\eta_2 d\eta_3 , \tag{10.299a}$$

$$\iiint f dv = 6v \int_0^1 \left(\int_0^{1-\eta_3} \left(\int_0^{1-\eta_2-\eta_3} f \; d\eta_1 \right) d\eta_2 \right) d\eta_3 , \tag{10.299b}$$

$$\iiint \eta_1^i \eta_2^j \eta_3^k \eta_4^l \; dv = \frac{i!\,j!\,k!\,l!}{(i+j+k+l+3)!} \times 6v . \tag{10.299c}$$

In terms of the local coordinates, an arbitrary function $\Phi(x,y)$ can be approximated within an element in terms of a complete *n*th order polynomial as

$$\Phi_m(x,y) = \sum_{i=1}^{r} a_i(x,y)\Phi_{mi} \tag{10.300}$$

where $r = \frac{1}{6}(n+1)(n+2)(n+3)$ is the number of nodes in the tetrahedron or number of terms in the polynomial. The terms in a complete three-dimensional polynomial may be arrayed for polynomial basic functions degree $n = 1$ to 3, as shown in Figure 10.23.

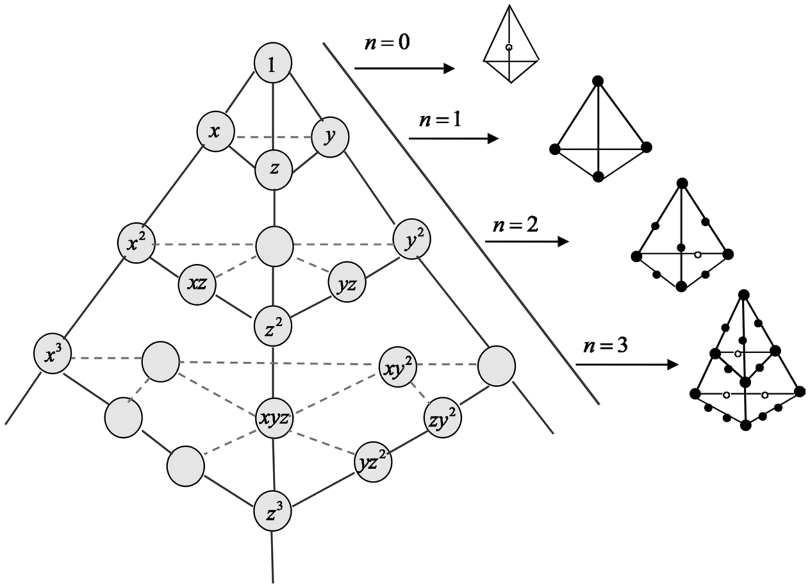


FIGURE 10.23 Pascal tetrahedral (3D) and the associated polynomial basic functions degree n = 1 to 3.

Each point in the tetrahedral element is represented by four integers, $i, j, k,$ and l which can be used to determine the local coordinates $(\eta_1, \eta_2, \eta_3, \eta_4)$. That is at point P_{ijkl},

$$(\eta_1, \eta_2, \eta_3, \eta_4) = \left(\frac{i}{n}, \frac{j}{n}, \frac{k}{n}, \frac{l}{n}\right). \tag{10.301}$$

Thus, at each node,

$$a_q = a_{ijkl} = p_i(\eta_1) p_j(\eta_2) p_k(\eta_3) p_l(\eta_4) \tag{10.302}$$

where $q = 1, 2, \ldots, r$ and p_e is defined in equation (10.277) or (10.278). Figure 10.22 illustrates the relationship between the node numbers q and $ijkl$ for the second-order tetrahedron (n = 2). The shape functions obtained by substituting equation (10.277) into (10.293) are presented in Table 2.5 for n = 3.

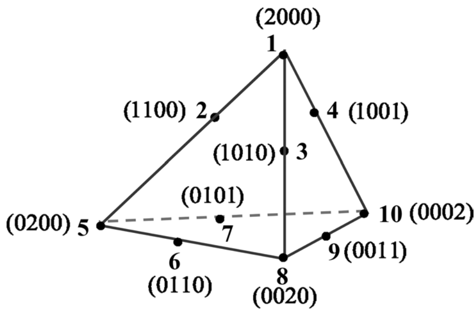


FIGURE 10.24 Numbering scheme for second-order tetrahedral.

TABLE 10.5 Polynomial Basic Functions $\alpha_q(\eta_1, \eta_2, \eta_3)$ for n = 1 to 3.

n = 1	n = 2	n = 3
$\alpha_1 = \eta_1$	$\alpha_1 = \eta_1(2\eta_1 - 1)$	$\alpha_1 = \frac{1}{2}\eta_1(3\eta_1 - 2)(3\eta_1 - 1)$
$\alpha_2 = \eta_2$	$\alpha_2 = 4\eta_1\eta_2$	$\alpha_2 = \frac{9}{2}\eta_1(3\eta_1 - 1)\eta_2$
$\alpha_3 = \eta_3$	$\alpha_3 = 4\eta_1\eta_3$	$\alpha_3 = \frac{9}{2}\eta_1(3\eta_1 - 1)\eta_3$
$\alpha_4 = \eta_4$	$\alpha_4 = 4\eta_1\eta_4$	$\alpha_4 = \frac{9}{2}\eta_1(3\eta_1 - 1)\eta_4$
	$\alpha_5 = \eta_2(2\eta_2 - 1)$	$\alpha_5 = \frac{9}{2}\eta_1(3\eta_3 - 1)\eta_2$
	$\alpha_6 = 4\eta_2\eta_3$	$\alpha_6 = 27\eta_1\eta_2\eta_3$
	$\alpha_7 = 4\eta_2\eta_4$	$\alpha_7 = 27\eta_1\eta_2\eta_4$
	$\alpha_8 = \eta_2(2\eta_3 - 1)$	$\alpha_8 = \frac{9}{2}\eta_1(3\eta_3 - 1)\eta_3$
	$\alpha_9 = 4\eta_3\eta_4$	$\alpha_9 = 27\eta_1\eta_3\eta_4$
	$\alpha_{10} = \eta_4(2\eta_4 - 1)$	$\alpha_{10} = \frac{9}{2}\eta_1(3\eta_4 - 1)\eta_4$
		$\alpha_{11} = \frac{1}{2}\eta_2(3\eta_2 - 1)(3\eta_2 - 2)$
		$\alpha_{12} = \frac{9}{2}\eta_2(3\eta_2 - 1)\eta_3$
		$\alpha_{13} = \frac{9}{2}\eta_2(3\eta_2 - 1)\eta_4$

n = 1	*n* = 2	*n* = 3
		$\alpha_{14} = \frac{9}{2}\eta_2(3\eta_3 - 1)\eta_3$
		$\alpha_{15} = 27\eta_2\eta_3\eta_4$
		$\alpha_{16} = \frac{9}{2}\eta_2(3\eta_3 - 1)\eta_3$
		$\alpha_{17} = \frac{1}{2}\eta_3(3\eta_3 - 1)(3\eta_3 - 2)$
		$\alpha_{18} = \frac{9}{2}\eta_3(3\eta_3 - 1)\eta_4$
		$\alpha_{19} = \frac{9}{2}\eta_3(3\eta_4 - 1)\eta_4$
		$\alpha_{20} = \frac{1}{2}\eta_4(3\eta_4 - 1)(3\eta_4 - 2)$

The fundamental matrices [*T*] and [*Q*] are involved triple integration. For Helmholtz equation, for example, equation (10.250) applies, namely,

$$\left[C_{ff} - k^2 T_{ff}\right]\Phi_f = 0 \tag{10.303}$$

except that

$$C_{ij}^{(m)} = \int_v \nabla\alpha_i \cdot \nabla\alpha_j \, dv = \int_v \left(\frac{\partial\alpha_i}{\partial x}\frac{\partial\alpha_j}{\partial x} + \frac{\partial\alpha_i}{\partial y}\frac{\partial\alpha_j}{\partial y} + \frac{\partial\alpha_i}{\partial z}\frac{\partial\alpha_j}{\partial z}\right) dv, \tag{10.304}$$

$$T_{ij}^{(m)} = \int_v \alpha_i \alpha_j \, dv = 6v \iiint \alpha_i \alpha_j \, d\eta_1 d\eta_2 d\eta_3. \tag{10.305}$$

10.17 FINITE ELEMENT METHODS FOR EXTERNAL PROBLEMS

We can apply the finite element to exterior or unbounded problems such as open-type transmission lines (e.g., microstrip). They pose certain difficulties. In this section, we will consider three common approaches: first, the infinite element method; second, the boundary element method; and third, the absorbing boundary condition.

10.17.1 Infinite Element Method

Let us consider the solution region shown in Figure 10.25. We can divide the entire domain into a near-field, bounded region, and a far-field, unbounded region. As usual, the near-field region is divided into finite triangular elements, while the far-field region is divided into infinite elements. Knowing that, each infinite element shares two nodes with a finite element. We will mainly be focusing on the infinite elements.

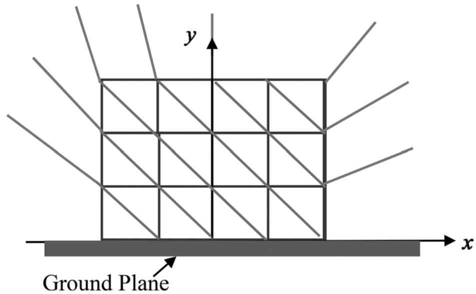


FIGURE 10.25 Division of solution region into finite and infinite elements.

Now, consider the infinite element in Figure 10.26 with nodes 1 and 2 and radial sides intersecting at point (x_0, y_0).

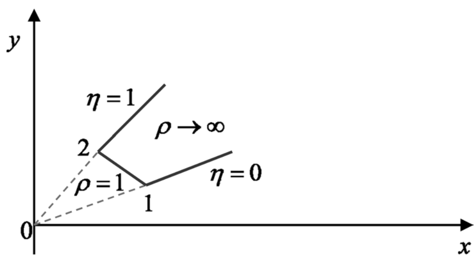


FIGURE 10.26 Typical infinite element.

We can relate triangular polar coordinates (ρ, η) to the global Cartesian coordinates (x, y) as:

$$
\begin{aligned}
x &= x_0 + \rho\left(\left(x_1 - x_0\right) + \eta\left(x_2 - x_1\right)\right) \\
y &= y_0 + \rho\left(\left(y_1 - y_0\right) + \eta\left(y_2 - y_1\right)\right)
\end{aligned}
\tag{10.306}
$$

where $1 \le \rho < \infty$, $0 \le \eta \le 1$. The potential distribution within the element is approximated by a linear variation as

$$V = \sum_{i=1}^{2} \alpha_i V_i \tag{10.307}$$

or

$$V = \frac{1}{\rho}\left(V_1(1-\eta) + V_2\eta\right)$$

where V_1 and V_2 are potentials at nodes 1 and 2 of the infinite elements, α_1 and α_2 are the interpolation or shape functions, that is,

$$\alpha_1 = \frac{1-\eta}{\rho}, \quad \alpha_2 = \frac{\eta}{\rho}. \tag{10.308}$$

Moreover, the infinite element is compatible with the ordinary first-order finite element and satisfies the boundary condition at infinity. Indeed, with the shape functions in equation (10.308), we can obtain the $\left[C^{(m)}\right]$ and $\left[T^{(m)}\right]$ matrices. We obtain solution for the exterior problem by using a standard finite element program with the $\left[C^{(m)}\right]$ and $\left[T^{(m)}\right]$ matrices of the infinite elements added to the $[C]$ and $[T]$ matrices of the near field region.

10.17.2 Boundary Element Method

The boundary element method is a finite element approach for handling exterior problems. It involves obtaining the integral equation formulation of the boundary value problem, and solving this by a discretization procedure similar to that used in regular finite element analysis. But, since the boundary element method is based on the boundary integral equivalent to the governing differential equation, only the surface of the problem domain needs to be modeled. Moreover, for the dimension of 2D problems, the boundary elements are taken to be straight-line segments, whereas for 3D problems, they are taken as triangular elements.

10.17.3 Absorbing Boundary Conditions

To apply the finite element approach to open region problems, an artificial boundary is introduced in order to bound the region and limit the number of unknowns to a manageable size. It can be expected that, as the boundary approaches infinity, the approximate solution tends to be the exact one. However, the closer the boundary to the modeled object, the less computer memory is required. To avoid the error caused by this truncation, an *absorbing boundary condition* (ABC) can be imposed on the artificial boundary S, as typically portrayed in Figure 10.27.

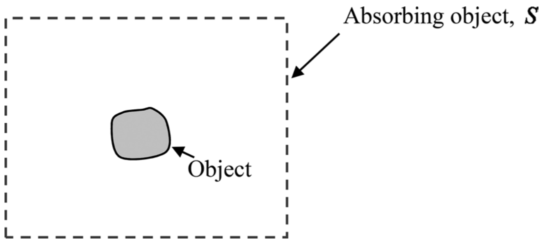


FIGURE 10.27 An object surrounded by an absorbing boundary.

Indeed, the ABC minimizes the nonphysical reflections from the boundary. The major challenge of these ABCs is to bring the truncation boundary as close as possible to the object without sacrificing accuracy and to absorb the outgoing waves with little or no reflection:

$$\Phi(r,\theta,\phi) = \frac{e^{-jkr}}{kr}\sum_{i=0}^{\infty}\frac{F_i(\theta,\phi)}{(kr)^i}. \tag{10.309}$$

Furthermore, the sequence of BGT operators can be obtained by the recursion relation, i.e.,

$$\begin{aligned} B_1 &= \left(\frac{\partial}{\partial r} + jk + \frac{1}{r}\right) \\ B_m &= \left(\frac{\partial}{\partial r} + jk + \frac{2m-1}{r}\right)B_{m-1}, \quad \mathrm{m} = 2,3,.... \end{aligned} \tag{10.310}$$

Now, since Φ satisfies the higher-order radiation condition

$$B_m\Phi = \mathrm{O}\left(\frac{1}{(r)^{2m+1}}\right). \tag{10.311}$$

By imposing the mth-order boundary condition

$$B_m \Phi = 0, \qquad \text{on } S \tag{10.312}$$

will compel the solution Φ to match the first $2m$ terms of the expansion in equation (10.309). Equation (10.312), along with other appropriate equations is solved for Φ using the FEM.

10.18 MODELING AND SIMULATION OF SHIELDED MICROSTRIP LINES WITH COMSOL MULTIPHYSICS

In today's fast-paced research and development culture, simulation power gives you the competitive edge. COMSOL Multiphysics delivers the ideal tool to build simulations that accurately replicate the important characteristics of your designs. Its unparalleled ability to include all relevant physical effects that exist in the real world is known as multiphysics. This approach delivers results—tangible results that save precious development time and spark innovation. COMSOL Multiphysics brings you this remarkable technology in an easy-to-use, intuitive interface, making it accessible to all engineers including designers, analysts, and researchers.

Today, electromagnetic propagation on multiple parallel transmission lines has been a very attractive area in computational electromagnetics. Multiple parallel transmission lines have been successfully applied and used by designers in compact packaging, semiconductor device, high-speed interconnecting buses, monolithic integrated circuits, and other applications. Microstrip lines are the most commonly used in all planar circuits despite the frequency ranges of the applied signals. Microstrip lines are the most commonly used transmission lines at high frequencies. Quasi-static analysis of microstrip lines involves evaluating them as parallel plate transmission lines, supporting a pure "TEM" mode. Development in microwave circuits using rectangular coaxial lines as a transmission medium has been improving over the past decades. Reid and Webster used rectangular coaxial transmission lines to fabricate a 60 GHz branch-line coupler. The finite difference time domain method has been used for analyzing a satellite beamforming network consisting of rectangular coaxial lines.

Advances in microwave solid-state devices have stimulated interest in the integration of microwave circuits. Today, microstrip transmission lines have attracted great attention and interest in microwave-integrated circuit applications. This creates the need for accurate modeling and simulation of microstrip transmission lines. Due to the difficulties associated with analytical methods for calculating the capacitance of shielded microstrip transmission lines, other methods have been applied. Such methods include the finite difference technique, extrapolation, point-matching method, boundary element method, spectral-space domain method, finite element method, conformal mapping method, transverse modal analysis, and mode-matching method.

In this book, we consider systems of rectangular coaxial lines as well as single-strip, double-strip, three-strip, six-strip, and eight-strip (multiconductor) shielded microstrip lines. Using COMSOL, a finite element package, we performed the simulation of these systems of microstrip lines. We compared the results with other methods and found them to be in good agreement.

The rectangular coaxial line consists of a two-conductor transmission system along which the TEM wave propagates. The characteristic impedance of such a lossless line is given by

$$Z = \sqrt{\frac{L}{C}} = \frac{1}{cC} \tag{10.313}$$

where

Z = characteristic impedance of the line

L = inductance per unit length of the line

C = capacitance per unit length of the line

c = 3×10^8 m/s (the speed of light in vacuum).

As shown in Figure 10.28, a rectangular coaxial line consists of inner and outer rectangular conductors with a dielectric material separating them.

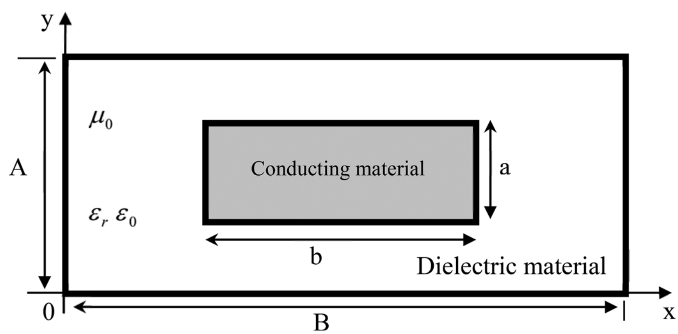


FIGURE 10.28 Cross-section of the rectangular coaxial line.

Using COMSOL for each type of the rectangular line involves taking the following steps:

1. Develop the geometry of the inner and outer conductors, such as shown in Figure 10.28.

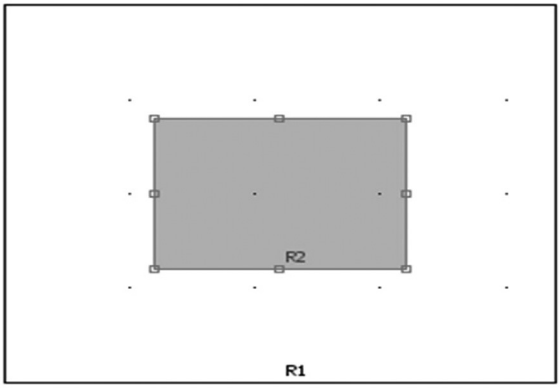


FIGURE 10.28 Geometry of the rectangular coaxial line model.

2. Select both conductors/rectangle and take the difference.
3. We select the relative permittivity as 1 for the difference in Step 2. For the boundary, we select the outer conductor as ground and inner conductor as port.
4. We generate the finite element mesh as in Figure 10.29.

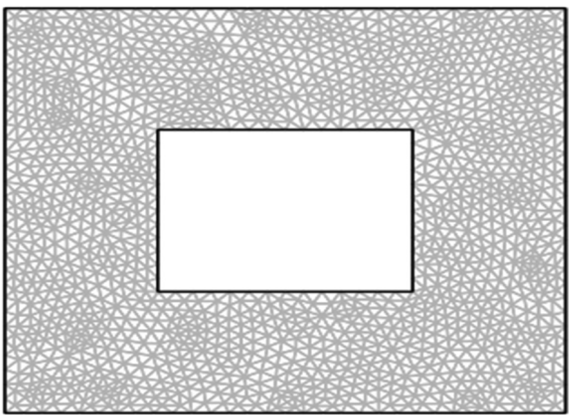


FIGURE 10.29 Mesh of the rectangular coaxial line.

5. We solve the model and obtain the potential shown in Figure 10.30.

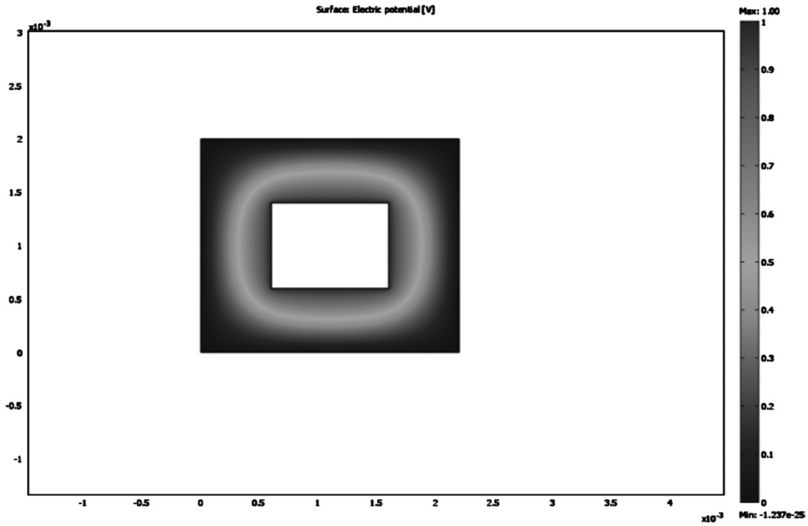


FIGURE 10.30 2D for the potential distribution of the rectangular coaxial line.

6. As postprocessing, we select Point Evaluation and choose capacitance element 11 to find the capacitance per unit length of the line.

We now consider the following three models.

10.18.1 Rectangular Cross-Section Transmission Line

For COMSOL, we use the following values.

Dielectric material:

$\varepsilon_r = 1$, $\mu_r = 1$, $\sigma = 0$ S/m (air)

Conducting material:

$\varepsilon_r = 1$, $\mu_r = 1$, $\sigma = 5.8 \times 10^7$ S/m (copper)

where

ε_0 = permittivity of free space = $\dfrac{1}{36\pi} \times 10^{-9} = 8.854 \times 10^{-12}$ F/m

ε_r = dielectric constant

μ_r = relative permeability

μ_0 = permeability of free space = $4\pi \times 10^{-7} = 1.257 \times 10^{-6}$ H/m

σ = conductivity of the conductor a = width of the inner conductor = 1 mm

b = height of the inner conductor = 0.8 mm

A= width of the outer conductor = 2.2 mm

B = height of the outer conductor = 2 mm

From the COMSOL model, we obtained the capacitance per unit length (based on the dimensions given above) as 72.94 pF/m. Using the finite difference (FD) method, we obtained the capacitance per unit length of the line as 71.51 pF/m. Table 10.6 shows the comparison of the characteristic impedance using equation (10.313) of several models. It is evident from the table that the results are very close.

TABLE 10.6 Comparison of Characteristic Impedance Values of Rectangular Coaxial Line.

Name	Z_0
Zheng	45.789
Chen	45.759
Costamagna and Fanni	45.767
Lau	45.778
Finite difference (FD)	46.612
COMSOL	45.70

10.18.2 Square Cross-Section Transmission Line

This is only a special case of the rectangular line. We used the same values for the dielectric and conducting materials. We used the following dimensions for the line.

a = width of the inner conductor = 2 mm

b = height of the inner conductor = 2 mm

A= width of the outer conductor = 4 mm

B = height of the outer conductor = 4 mm

From the COMSOL model, we obtained the capacitance per unit length as 90.696 pF/m. Using the FD method, we obtained the capacitance per unit length of the line as 90.714 pF/m. Table 10.7 presents the comparison of the characteristic impedance of several models. It is evident from the table that the results are in good agreement.

TABLE 10.7 Comparison of Characteristic Impedance Values of Square Coaxial Line

Name	Z_0
Zheng	36.79
Lau	36.81
Cockcroft	36.80
Bowan	36.81
Green	36.58
Ivanov and Djankov	36.97
Costamagna and Fanni	36.81
Riblet	36.80
Finite difference (FD)	36.75
COMSOL	36.75

10.18.3 Rectangular Line with Diamondwise Structure

The geometry of the cross-section of this line is shown in Figure 10.31. The same dielectric and conducting materials used for the rectangular line are used for this line.

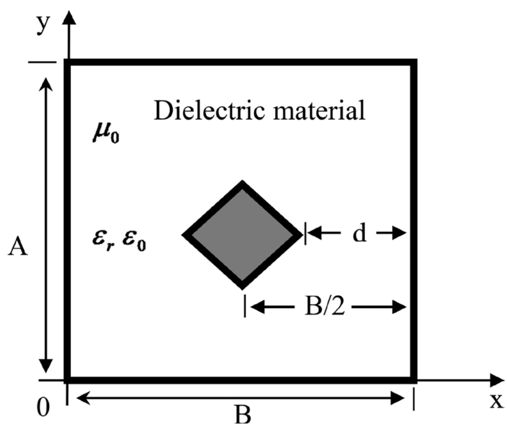


FIGURE 10.31 Cross-Section of the Diamondwise (or Rhombus) Structure with 45° Offset Angle.

The following values are used for the COMSOL model of the line.

d = 1 mm

A= width of the outer conductor = 4 mm

B = height of the outer conductor = 4 mm

For the COMSOL model, we obtained the capacitance per unit line as 57.393 pF/m.

Table 10.8 displays the comparison of the characteristic impedance of several models. It is evident from the table that the results are in good agreement.

TABLE 10.8 Comparison of Characteristic Impedance Values of Diamondwise Structure

Name	Z_0
Zheng et al.	56.742
Bowan	56.745
Riblet	56.745
COMSOL	58.079

10.18.4 A Single-Strip Shielded Transmission Line

Figure 10.32 presents the cross-section of a single-strip shielded transmission line.

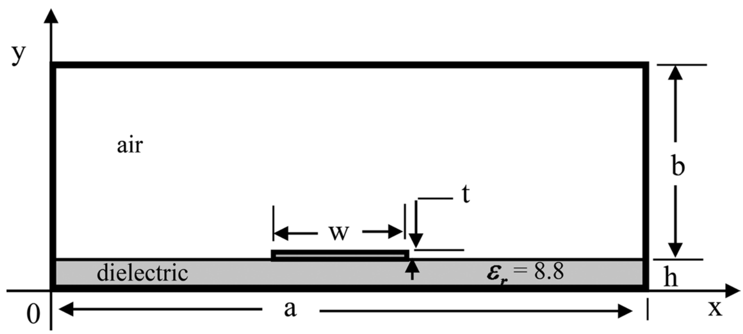


FIGURE 10.32 Cross-Section of the Single-Strip Shielded Transmission Line.

The following parameters are used in modeling the line. The characteristic impedance of such a lossless line is given by

$$Z = \frac{1}{c\sqrt{CC_o}} \tag{10.314}$$

where

Z = characteristic impedance of the line

C_o = capacitance per unit length of the line when the substrate is replaced with air

C = capacitance per unit length of the line when the substrate is in place

c = 3×10^8 m/s (the speed of light in vacuum).

For COMSOL, the simulation was done twice in Figure 10.32 (to find C_o and C) using the following values.

Air:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 0$ S/m

Dielectric material:

$\varepsilon_r = 8.8,\ \mu_r = 1,\ \sigma = 0$ S/m

Conducting material:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 5.8 \times 10^7$ S/m (copper)

w = width of the inner conductor = 1 mm

t = height of the inner conductor = 0.1×10^{-4} m

h = height of dielectric material = 1 mm

a = width of the outer conductor = 19 mm

b = height of the air-filled region = 9 mm

Using COMSOL for modeling and simulation of the lines involves taking the following steps:

1. Develop the geometry of the line, such as shown in Figure 10.33.

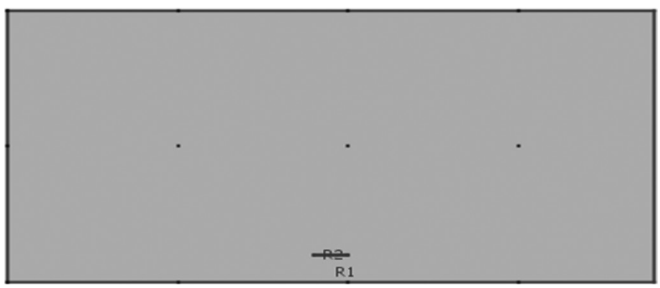


FIGURE 10.33 Geometry of a Single-Strip Shielded Transmission Line at Air.

2. We take the difference between the conductor and dielectric material.
3. We select the relative permittivity as 1 for the difference in Step 2.
4. For the boundary, we select the outer conductor as ground and inner conductor as port.
5. We generate the finite element mesh, and then we solve the model and obtain the potential.
6. As postprocessing, we select Point Evaluation and choose capacitance element 11 to find the capacitance per unit length of the line.
7. We add a dielectric region under the inner conductor with relative permittivity as 8.8, as in Figure 10.33. Then we take the same steps from 3 to 6 to generate the mesh as in Figure 10.34 and the potential distribution as in Figure 10.35.

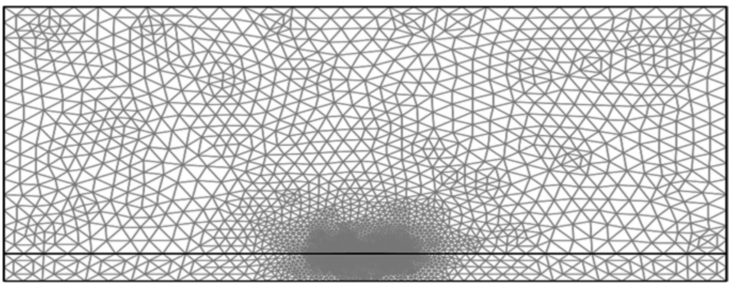


FIGURE 10.34 Mesh of a Single-Strip Shielded Transmission Line.

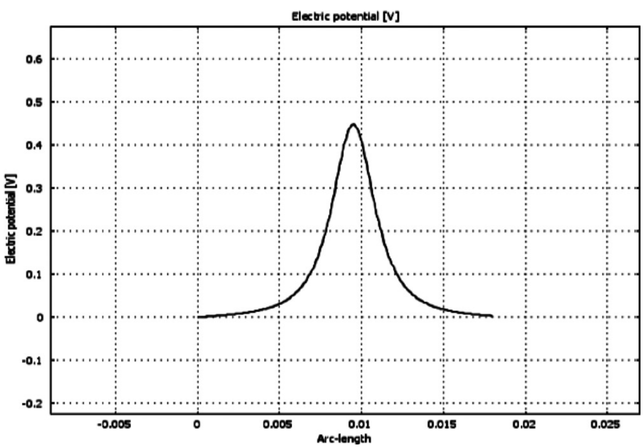


FIGURE 10.35 The Potential Distribution along y = 0.002.

Table 10.9 shows the comparison between our method using COMSOL and other methods. It is evident that the results are very close.

TABLE 10.9 Comparison of capacitance values for a single-strip shielded transmission line

Methods	C_o(pF/m)	C(pF/m)
Finite difference method	26.79	1405.2
Extrapolation	26.88	1393.6
Analytical derivation	27.00	1400.9
COMSOL	26.87	1574.0

10.19 MULTISTRIP TRANSMISSION LINES

Recently, with the advent of integrated circuit technology, the coupled microstrip transmission lines consisting of multiple conductors embedded in a multilayer dielectric medium have led to a new class of microwave networks. Multiconductor transmission lines have been utilized as filters in the microwave region, which make it interesting in various circuit components. For coupled multiconductor microstrip lines, it is convenient to write:

$$Q_i = \sum_{j=1}^{m} C_{sij} V_j \qquad (i = 1, 2, \ldots.., m) \tag{10.315}$$

where Q_i is the charge per unit length, V_j is the voltage of jth conductor with reference to the ground plane, C_{sij} is the short circuit capacitance between ith conductor and jth conductor. The short circuit capacitances can be obtained either from measurement or from numerical computation. From the short circuit capacitances, we obtain

$$C_{ii} = \sum_{j=1}^{m} C_{sij} \tag{10.316}$$

where C_{ii} is the capacitance per unit length between the ith conductor and the ground plane. Also,

$$C_{ij} = -C_{sij}, \qquad j \neq i \tag{10.317}$$

where C_{ij} is the coupling capacitance per unit length between the ith conductor and jth conductor. The coupling capacitances are illustrated in Figure 10.36.

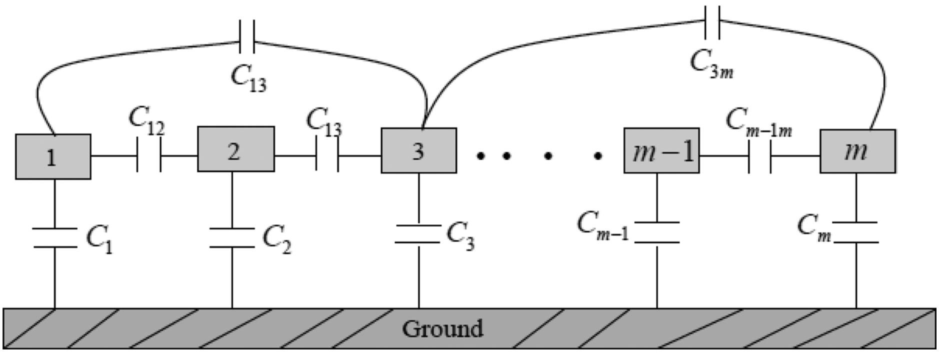


FIGURE 10.36 The Per unit Length Capacitances of a General m-conductor Transmission Line.

For m-strip line, the per unit length capacitance matrix is given by

$$C = \begin{bmatrix} C_{11} & -C_{12} & \cdots & -C_{1m} \\ -C_{21} & C_{22} & \cdots & -C_{2m} \\ \vdots & \vdots & & \vdots \\ -C_{m1} & -C_{m2} & \cdots & C_{mm} \end{bmatrix}. \tag{10.318}$$

Also, we can determine the characteristic impedance matrix for m-strip line by using

$$Z_o = \begin{bmatrix} Z_{11} & Z_{12} & \cdots & Z_{1m} \\ Z_{21} & Z_{22} & \cdots & Z_{2m} \\ \vdots & \vdots & & \vdots \\ Z_{m1} & Z_{m2} & \cdots & Z_{mm} \end{bmatrix} \tag{10.319}$$

where Z_o is the characteristic impedance per unit length.

Using COMSOL for modeling and simulation of the lines involves taking the following steps:

1. Develop the geometry of the line.
2. We take the difference between the conductor and dielectric material.
3. We select the relative permittivity as 1 for the difference in Step 2.

4. We add a dielectric region under the inner conductors with specified relative permittivity.
5. For the boundary, we select the outer conductor as ground and the inner conductors as ports.
6. We generate the finite element mesh, and then we solve the model.
7. As postprocessing, we select Point Evaluation and choose capacitance elements to find the coupling capacitance per unit length of the line.

These steps were taken for the following four cases.

10.19.1 Double-Strip Shielded Transmission Line

Figure 10.37 presents the cross-section of double-strip shielded transmission line, which consists of two inner conductors.

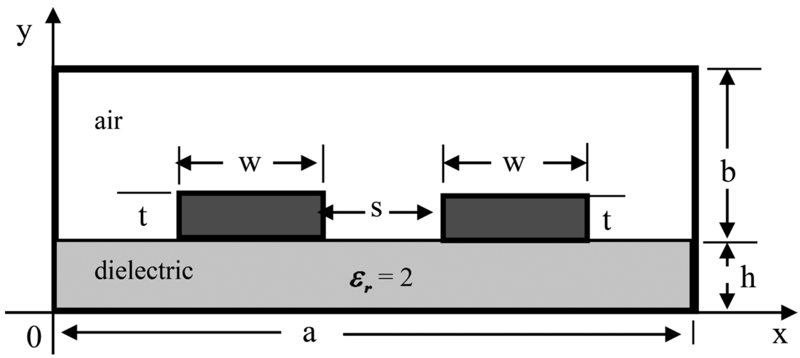


FIGURE 10.37 Cross-section of the Double-strip Shielded Transmission Line.

For COMSOL, the simulation was done twice in Figure 10.36 (one for C_o and the other for C) using the following values.

Air:

$\varepsilon_r = 1$, $\mu_r = 1$, $\sigma = 0$ S/m

Dielectric material:

$\varepsilon_r = 2$, $\mu_r = 1$, $\sigma = 0$ S/m

Conducting material:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 5.8 \times 10^7$ S/m (copper)

For the geometry (see Figure 10.37), we followed the following values:

w = width of each of the inner conductors = 3 mm

t = height (or thickness) of the inner conductors = 1 mm

s = distance between the inner conductors = 2 mm

h = height of dielectric material = 1 mm

a = width of the outer conductor = 11 mm

b = height of the air-filled region = 2.7 mm

From the COMSOL model, the simulation was done twice, one for the case in which the line is air-filled (the dielectric was replaced by air) and the other case in which the dielectric is in place, as shown in Figure 10.37. Figure 10.38 shows the finite element mesh, while Figure 10.39 depicts the potential distribution for the dielectric case. The potential distribution for y = 1 mm is portrayed in Figure 10.40.

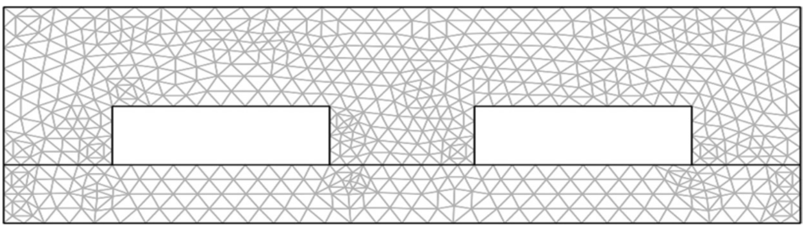


FIGURE 10.38 Mesh the of Double-strip Shielded Transmission Line.

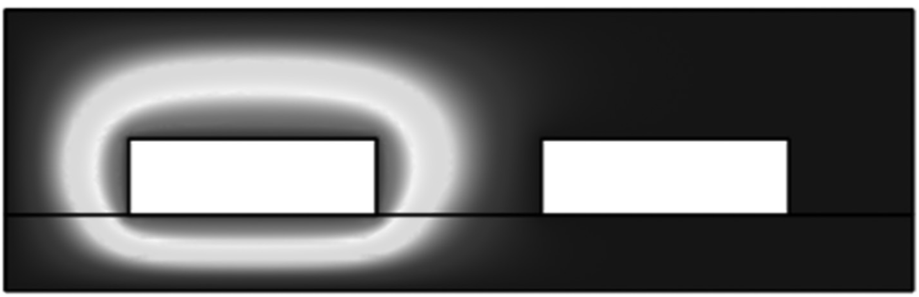


FIGURE 10.39 Potential Distribution.

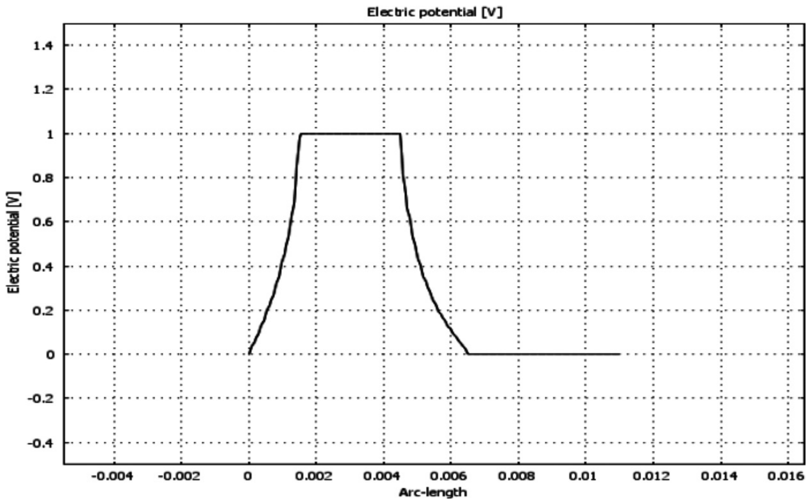


FIGURE 10.40 Potential distribution at y =1 mm.

We obtained the capacitances per unit length (C_o and C) by taking the steps enumerated above for the single-strip transmission line. The results are shown in Table 10.9. Table 10.10 is for the case in which the line is air-filled, i.e., the dielectric in Figure 10.37 is replaced by air. Table 10.11 is for the case in which the dielectric is in place. The results in Table 10.11 are compared with other methods and found to be close.

TABLE 10.10 Capacitance Values for Double-strip Air-filled Shielded Transmission Line

Methods	$C_{11} = C_{22}$ (pF/m)	$C_{12} = C_{21}$ (pF/m)
COMSOL	72.9	−4.591

TABLE 10.11 Comparison of Capacitance Values for Double-strip Shielded Transmission Line Shown in Figure 10.36

Methods	$C_{11} = C_{22}$ (pF/m)	$C_{12} = C_{21}$ (pF/m)
Spectral-space domain method	108.1	−4.571
Finite element method	109.1	−4.712
Point-matching method	108.8	−4.683
COMSOL	108.5	−4.618

10.19.2 Three-Strip Line

Figure 10.40(a) shows the cross-section for three-strip transmission line. For COMSOL, the simulation was done twice in Figure 10.40 (one for C_o and the other for C) using the following values:

Air:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 0$ S/m

Dielectric material:

$\varepsilon_r = 8.6,\ \mu_r = 1,\ \sigma = 0$ S/m

Conducting material:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 5.8 \times 10^7$ S/m (copper)

For the geometry (see Figure 10.40(a)), we used the following values:

a = width of the outer conductor = 13 mm

b = height of the free space region (air) = 4 mm

h = height of the dielectric region = 2 mm

w = width of each inner strip = 2 mm

t = thickness of each inner strip = 0.01 mm

D = distance between the outer conductor and the first strip = 2.5 mm

s = distance between two consecutive strips = 1 mm

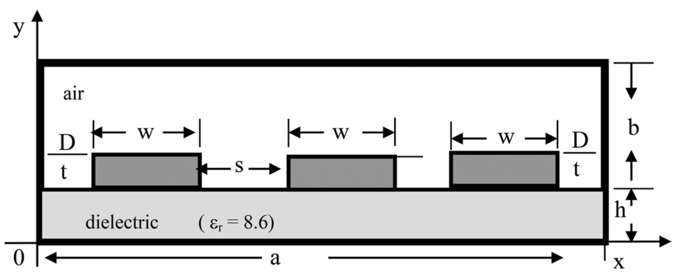


FIGURE 10.40(a) Cross-section of the Three-strip Transmission Line.

Figure 10.41 shows the finite element mesh, while Figure 10.42 illustrates the potential distribution along the line $y = h$.

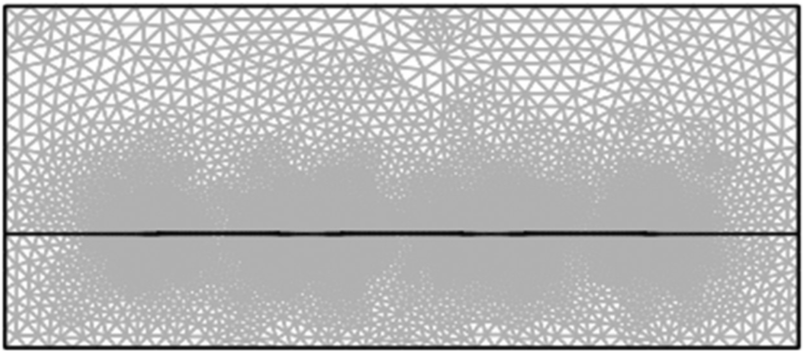


FIGURE 10.41 Mesh for the Three-strip Transmission Line.

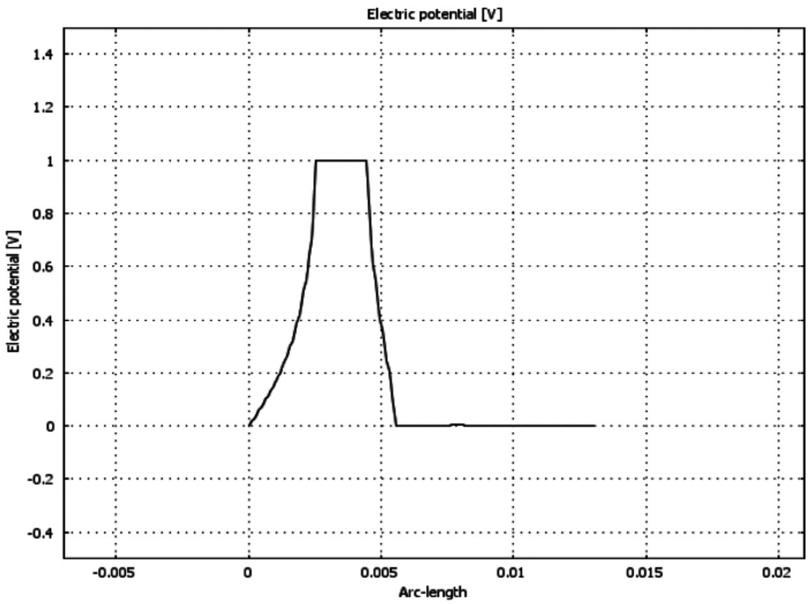


FIGURE 10.42 Potential Distribution along the Air-dielectric Interface ($y = h$) for the Three-strip Transmission Line.

Table 10.12 shows the finite element results for the three-strip line. Unfortunately, we could not find any work in the literature to compare our results.

TABLE 10.12 Capacitance Values (in pF/m) for Three-strip Shielded Microstrip Line

Methods	C_{11}	C_{21}	C_{31}
COMSOL	163.956	−27.505	−0.4301

10.19.3 Six-Strip Line

Figure 10.43 shows the cross-section for six-strip transmission line. For COMSOL, the simulation was done twice in Figure 10.42 (one for C_o and the other for C) using the following values:

Air:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 0$ S/m

Dielectric material:

$\varepsilon_r = 6,\ \mu_r = 1,\ \sigma = 0$ S/m

Conducting material:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 5.8 \times 10^7$ S/m (copper)

For the geometry (see Figure 10.43), we used the following values:

a = width of the outer conductor = 15 mm

b = height of the free space region (air) = 2 mm

h = height of the dielectric region = 8 mm

w = width of each inner strip = 1 mm

t = thickness of each inner strip = 0.01 mm

D = distance between the outer conductor and the first strip = 2 mm

s = distance between two consecutive strips = 1 mm

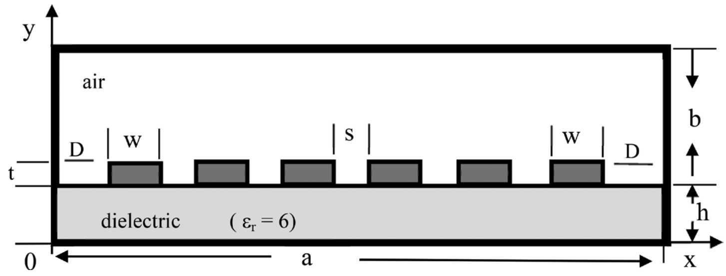


FIGURE 10.43 Cross-section of the Six-strip Transmission Line.

Figure 10.44 shows the finite element mesh, while Figure 10.45 depicts the potential distribution along line $y = h$.

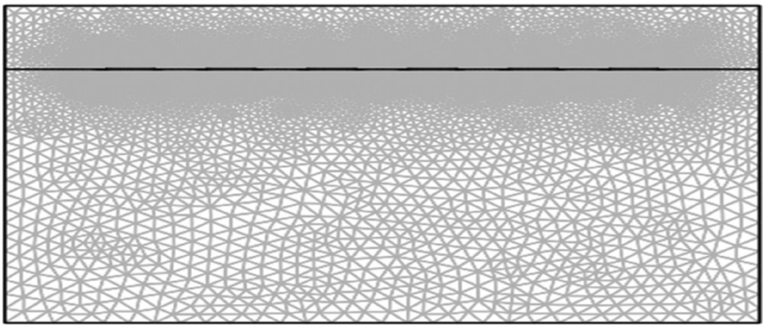


FIGURE 10.44 Mesh for the Six-strip Transmission Line.

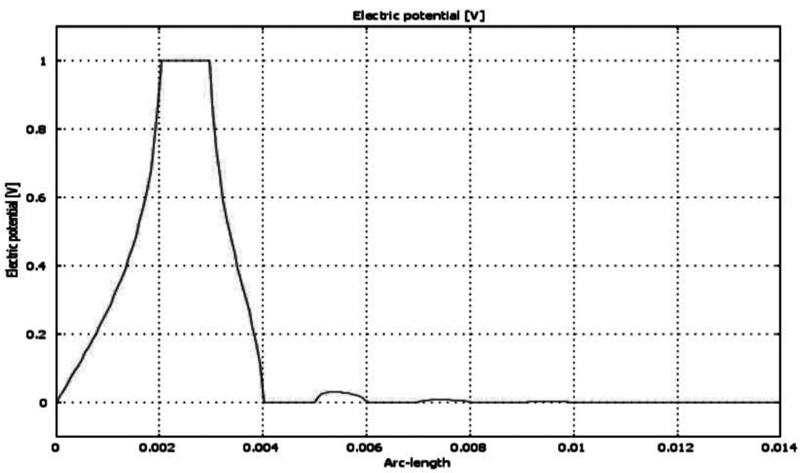


FIGURE 10.45 Potential Distribution along the Air-dielectric Interface ($y = h$) for the Six-strip Transmission Line.

The capacitance values for six-strip shielded microstrip line are compared with other methods, as shown in Table 10.13, where "iterative" refers to an iterative method and ABC refers to the asymptotic boundary condition. It is evident from the table that the finite element methods based closely agree. The finite element methods seem to be more accurate than the iterative and ABC techniques. (The negative capacitances are expected from equation (10.318).)

TABLE 10.13 Capacitance Values (in pF/m) for Six-strip Shielded Microstrip Line

Methods	C_{11}	C_{21}	C_{31}	C_{41}	C_{51}	C_{61}
Iterative	66.8	−27.9	−5.49	−2.08	−0.999	−0.704
Finite Element	84.8	−26.4	−3.71	−1.17	−0.456	−0.812
ABC	68.6	−31.5	−6.00	−2.25	−0.792	−0.602
COMSOL	80.4	−23.9	−3.61	−1.15	−0.451	−0.180

10.19.4 Eight-Strip Line

Figure 10.46 shows the cross-section for eight-strip transmission line. For COMSOL, the simulation was done twice in Figure 10.45 (one for C_o and the other for C) using the following values:

Air:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 0$ S/m

Dielectric material:

$\varepsilon_r = 12.9,\ \mu_r = 1,\ \sigma = 0$ S/m

Conducting material:

$\varepsilon_r = 1,\ \mu_r = 1,\ \sigma = 5.8 \times 10^7$ S/m (copper)

For the geometry (see Figure 10.46), we used the following values:

a = width of the outer conductor = 175 mm

b = height of the free space region (air) = 100 mm

h = height of the dielectric region = 16 mm

sw = width of each inner strip = 1 mm

t = thickness of each inner strip = 0.01 mm

D = distance between the outer conductor and the first strip = 80 mm

s = distance between two consecutive strips = 1 mm

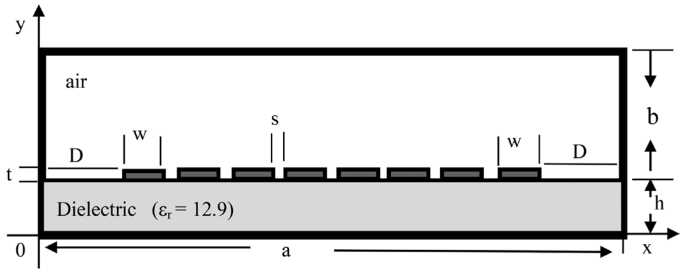


FIGURE 10.46 Cross-section of the Eight-strip Transmission Line.

Figure 10.47 shows the finite element mesh, while Figure 10.48 depicts the potential distribution along line y = 20 mm.

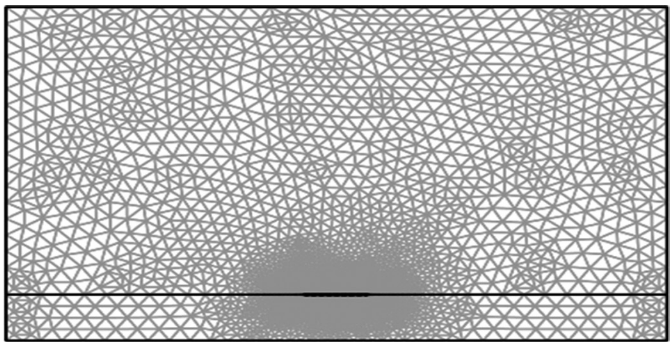


FIGURE 10.47 Mesh for the Eight-strip Transmission Line.

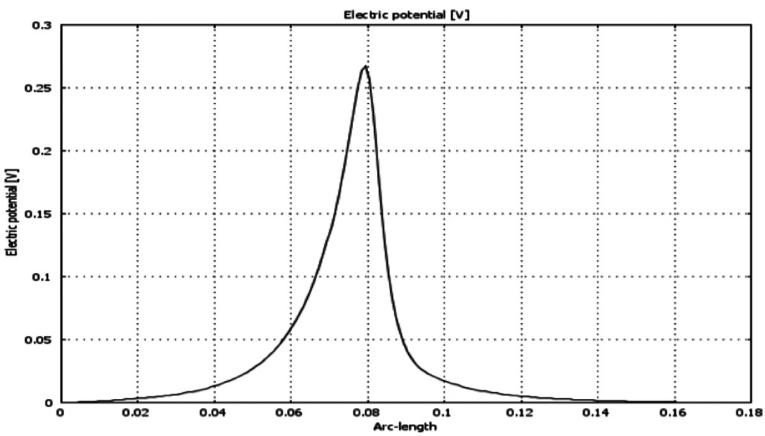


FIGURE 10.48 Potential Distribution along the Air-dielectric Interface (y = 20 mm) for the Eight-strip Transmission Line.

The capacitance values for eight-strip shielded microstrip line are compared with other methods, as shown in Table 10.14, where other authors used the analytic approach and Fourier series expansion. It is evident from the table that the results from the finite element method (COMSOL) closely agree with the analytic approach.

TABLE 10.14 Capacitance Values (in pF/m) for Eight-strip Shielded Microstrip Line

Method	C_{11}	C_{21}	C_{31}	C_{41}	C_{51}	C_{61}	C_{71}	C_{81}
Analytic	127.776	–58.446	–13.024	–5.721	–3.104	–1.892	–1.282	–1.211
Fourier series	126.149	–57.066	–12.927	–5.684	–3.086	–1.875	–1.264	–1.185
COMSOL	128.204	–58.759	–13.064	–5.739	–3.1206	–1.902	–1.290	–1.226

10.20 SOLENOID ACTUATOR ANALYSIS WITH ANSYS

We use ANSYS to do magnetic analysis (linear static) of a solenoid actuator. A solenoid actuator is to be analyzed as a 2D axisymmetric model as shown in Figure 10.49. For the given current, we determine the force on the armature.

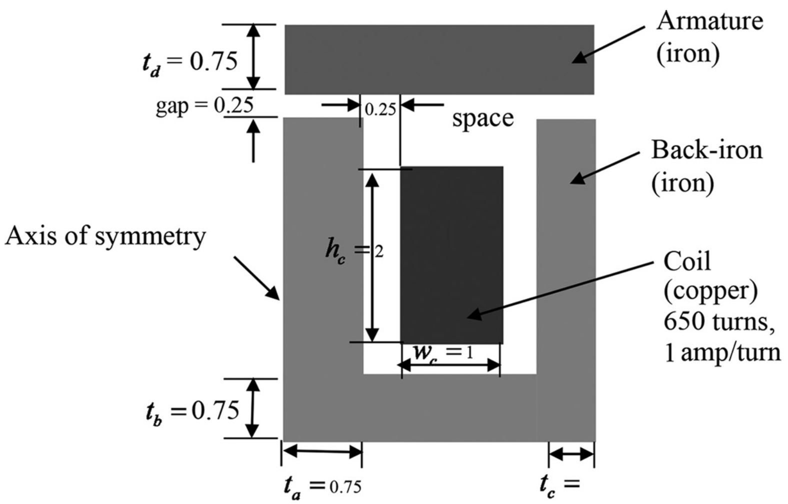


FIGURE 10.49 Cross-Section of the Solenoid Actuator.

The dimensions of the solenoid actuator are in centimeters. The *armature* is the moving component of the actuator. The *back-iron* is the stationary iron component of the actuator that completes the magnetic circuit around the coil. The stranded, wound *coil* of 650 windings with 1 amp/turn supplies the predefined current. The current per winding is 1 amp. The *air gap* is the thin rectangular region of air between the armature and the pole faces of the back-iron.

The magnetic flux produced by the coil current is assumed to be so small that no saturation of the iron occurs. This allows a single-iteration linear analysis. The flux leakage out of the iron at the perimeter of the model is assumed to be negligible. This assumption is made simple to keep the model small. The model would normally be created with a layer of air surrounding the iron equal to or greater than the maximum radius of the iron.

The air gap is modeled so that a quadrilateral mesh is possible. A quadrilateral mesh allows for an uniform thickness of the air elements adjacent to the armature where the virtual work force calculation is performed. This is desirable for an accurate force calculation. The program requires the current to be input in the form of current density (current over the area of the coil). The assumption of no leakage at the perimeter of the model means that the flux will be acting parallel to this surface. This assumption is enforced by the "flux parallel" boundary condition placed around the model. This boundary condition is used for models in which the flux is contained in an iron circuit. Forces for the virtual work calculation are stored in an element table and then summed. The force is also calculated by the Maxwell Stress Tensor method, and the two values are found to be relatively close. Table 10.15 summarizes the parameters of the model for the actuator geometry.

TABLE 10.15 Parameters of the model for the actuator geometry.

Parameter	Value
Number of turns in the coil; used in postprocessing	$n = 650$
Current per turn	$I = 1.0$
Thickness of inner leg of magnetic circuit	$t_a = 0.75$
Thickness of lower leg of magnetic circuit	$t_b = 0.75$
Thickness of outer leg of magnetic circuit	$t_c = 0.50$
Armature thickness	$t_d = 0.75$
Width of coil	$w_c = 1$
Height of coil	$h_c = 2$
Air Gap	gap = 0.25
Space around coil	space =0.25
w_s	$w_s = w_c + 2 *$ space
h_s	$h_s = h_c + 0.75$

Parameter	Value
Total width of model	$w = t_a + w_s + t_c$
h_b	$h_b = t_b + h_s$
Total height of model	$h = h_b + gap + t_d$
Coil area	$acoil = w_c * h_c$
Current density of coil	$idens = n*i/acoil$

The below steps are a guideline for solving the above model.

1. Input the geometry of the model

We use the information in the problem description to make Figure 10.50.

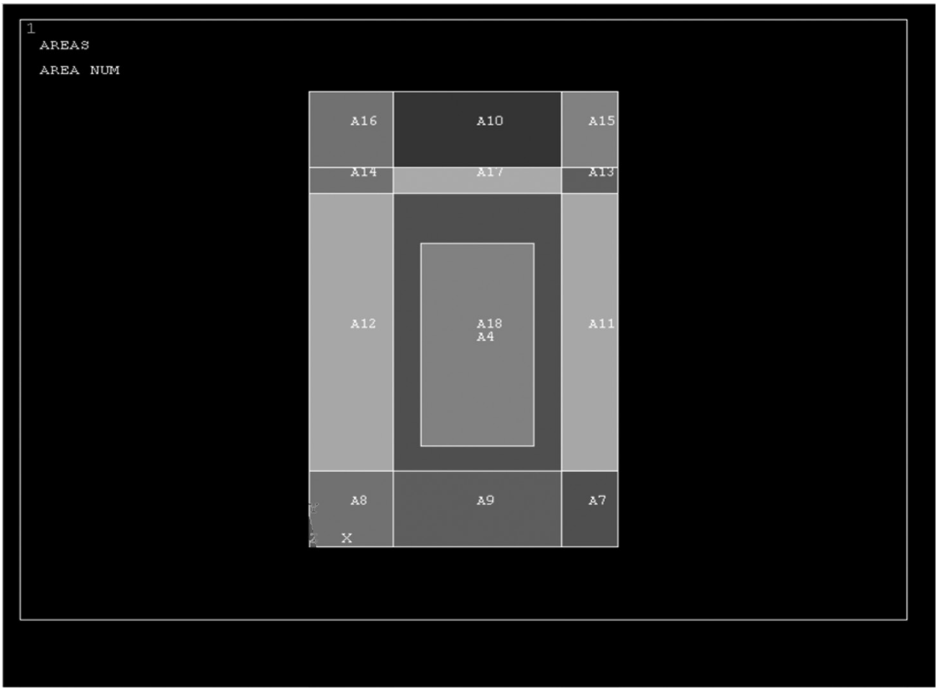


FIGURE 10.50 The 2D Geometry of the Solenoid Actuator Model.

2. Define the materials

(a) Set preferences

You will now set preferences in order to filter quantities that pertain to this discipline only.

1. **Main Menu> Preferences**
2. **Check** "Magnetic-Nodal" as in Figure 10.50a
3. **OK**

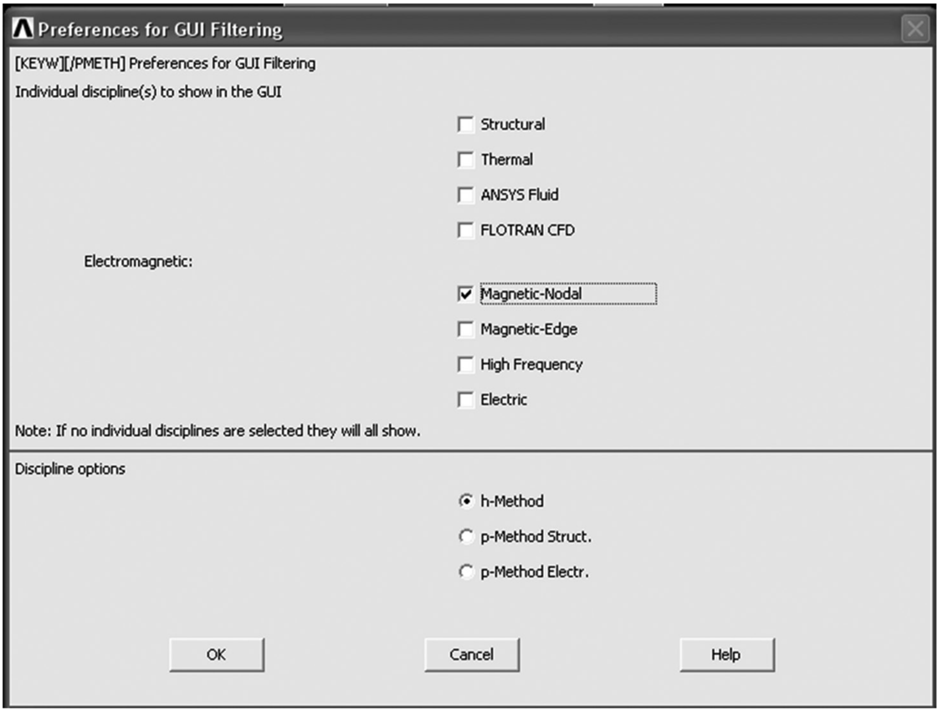


FIGURE 10.50(a) Preferences for GUI filtering.

(b) Specify material properties

Now specify the material properties for the magnetic permeability of air, back-iron, coil, and armature. For simplicity, all material properties are assumed to be linear. (Typically, iron is input as a nonlinear B-H curve.) Material 1 will be used for the air elements. Material 2 will be used for the back-iron elements. Material 3 will be used for the coil elements. Material 4 will be used for the armature elements.

1. **Main Menu > Preprocessor > Material Props > Material Models**
2. **Double-click** "Electromagnetics", then "Relative Permeability", then "Constant"

3. "MURX" = 1
4. **OK**
5. **Edit > Copy**
6. **OK** to copy Material Model Number 1 to become Material Model Number 2.
7. **Double-click** "Material Model Number 2", then "Permeability (Constant)"
8. "MURX" = 1000 as shown in Figure 10.51
9. **OK**
10. **Edit > Copy**
11. "from Material Number" = 1
12. "to Material Number" = 3
13. OK
14. **Edit > Copy**
15. "from Material Number" = 2
16. "to Material Number" = 4
17. **OK**
18. **Double-click** "Material Model Number 4", then "Permeability (Constant)"
19. "MURX" = 2000 as shown in Figure 10.52
20. **OK**
21. **Material > Exit**
22. **Utility Menu > List > Properties > All Materials**
23. Review the list of materials, then: as shown in Figure 10.53

    File > Close (Windows)

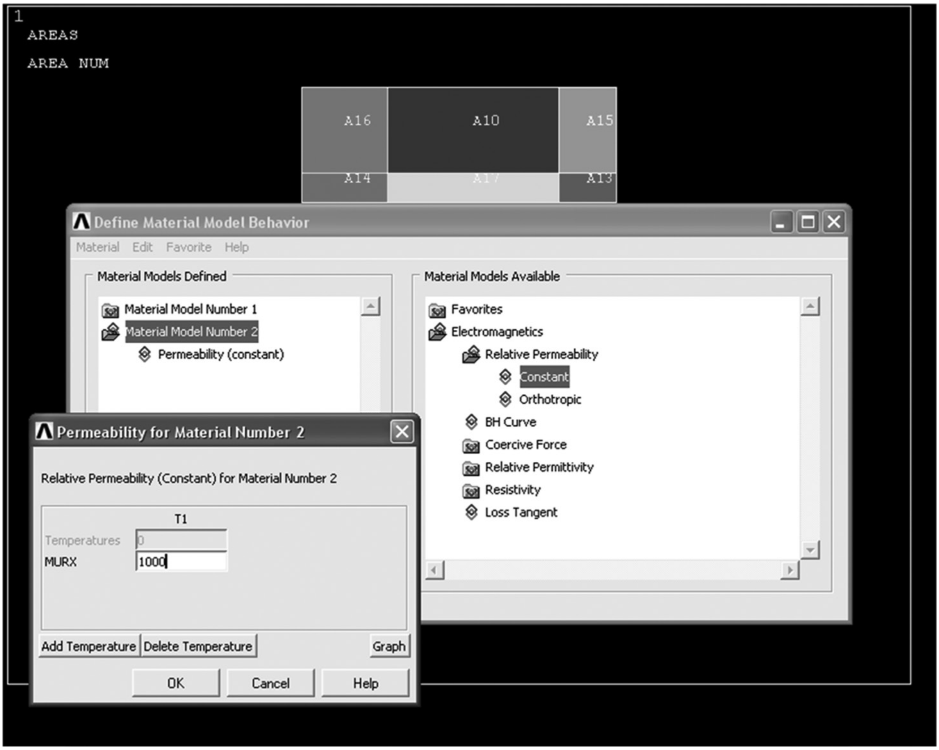


FIGURE 10.51 Definition of Material Model Behavior for Numbers 1 and 2.

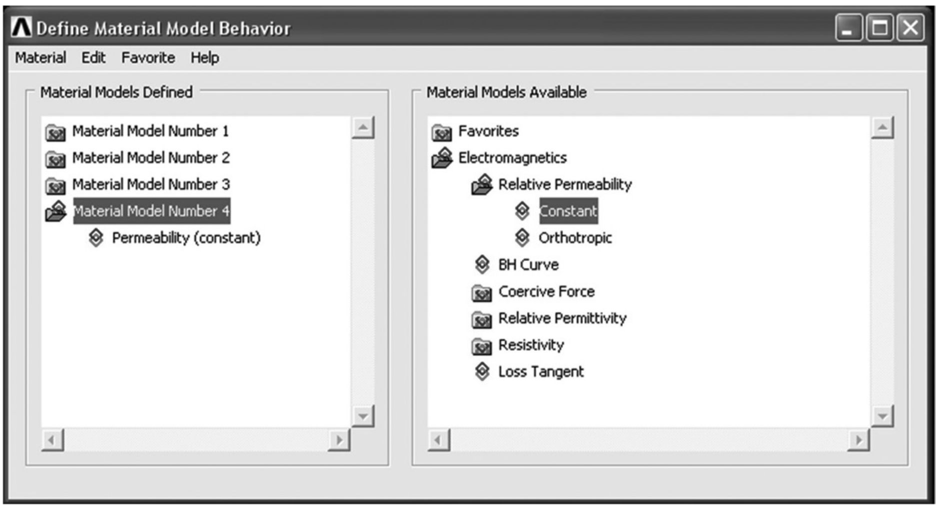


FIGURE 10.52 Definition of Material Model Behavior for Numbers 1, 2, 3, and 4.

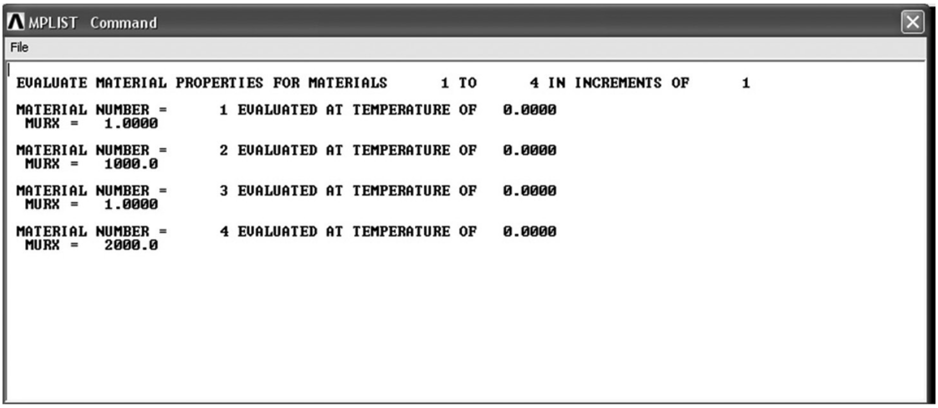


FIGURE 10.53 Review the List of Materials of the Model.

3. Generating the Mesh

(a) Define element types and options

In this step, you will define element types and specify options associated with these element types.

The higher-order element PLANE53 is normally preferred, but to keep the model size small, use the lower-order element PLANE13.

1. **Main Menu > Preprocessor > Element Type > Add/Edit/Delete**
2. **Add...**
3. "Magnetic Vector" (left column)
4. "Vect Quad 4nod13 (PLANE13" (right column)
5. **OK**
6. **Options...**
7. (drop down) "Element behavior" = Axisymmetric, as shown in Figure 10.54
8. **OK**
9. **Close**

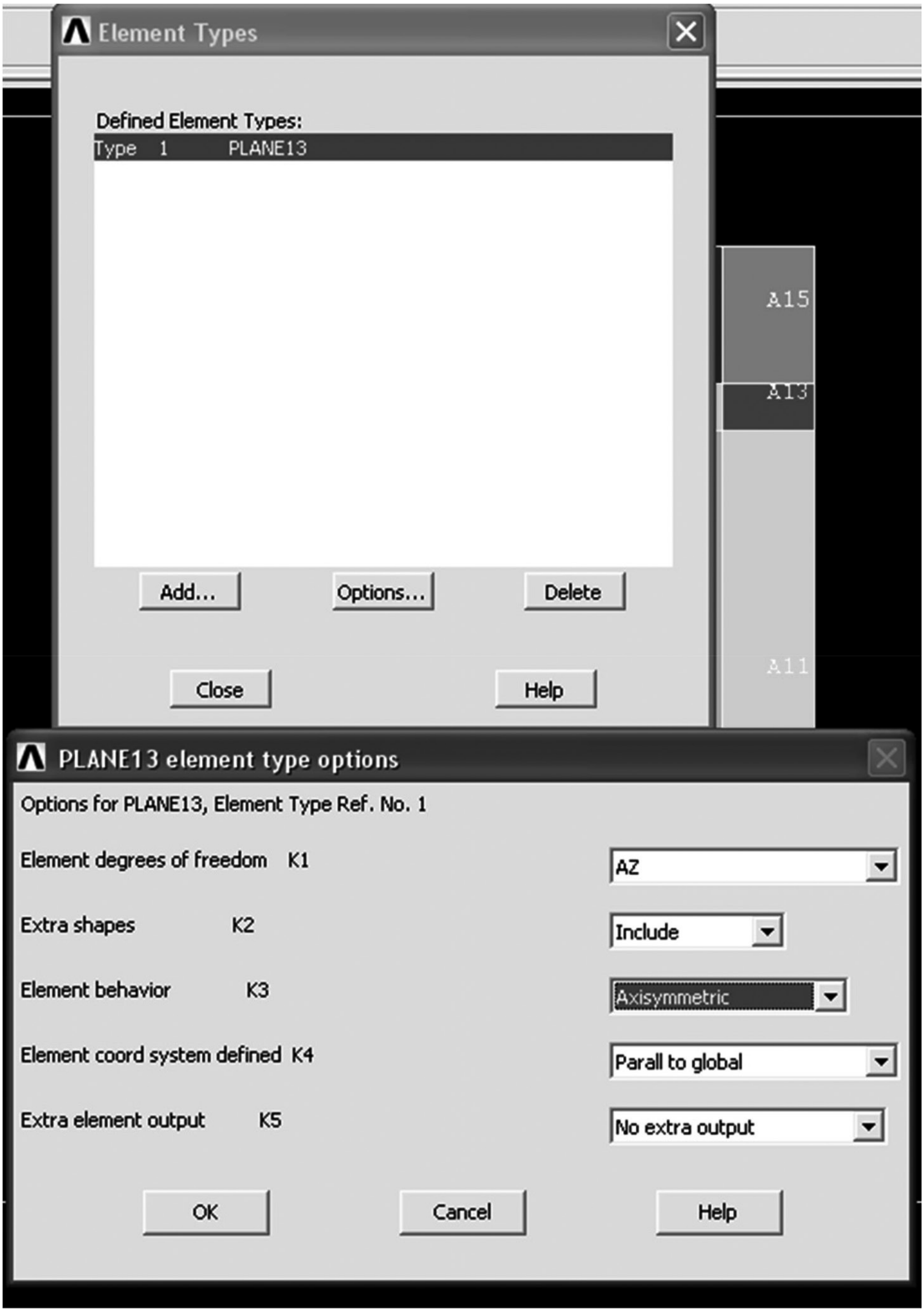


FIGURE 10.54 Element Type PLANE13.

(b) Assign material properties.

Now assign material properties to air gaps, iron, coil, and armature areas.

1. **Main Menu > Preprocessor > Meshing > MeshTool**
2. (drop down) "Element Attributes" = Areas; then [Set] as in Figure 10.55
3. Pick four areas of air gaps, A13, A14, A17, and A18 (the picking "hot spot" is at the area number label).

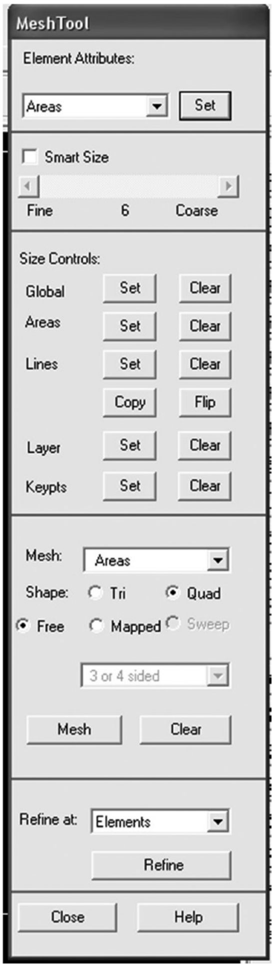


FIGURE 10.55 Element Attribute for Mesh Tool.

4. **OK**
5. (drop down) "Material number" = 1
6. **Apply**
7. Pick the five back-iron areas, A7, A8, A9, A11, A12. as in Figure 10.56

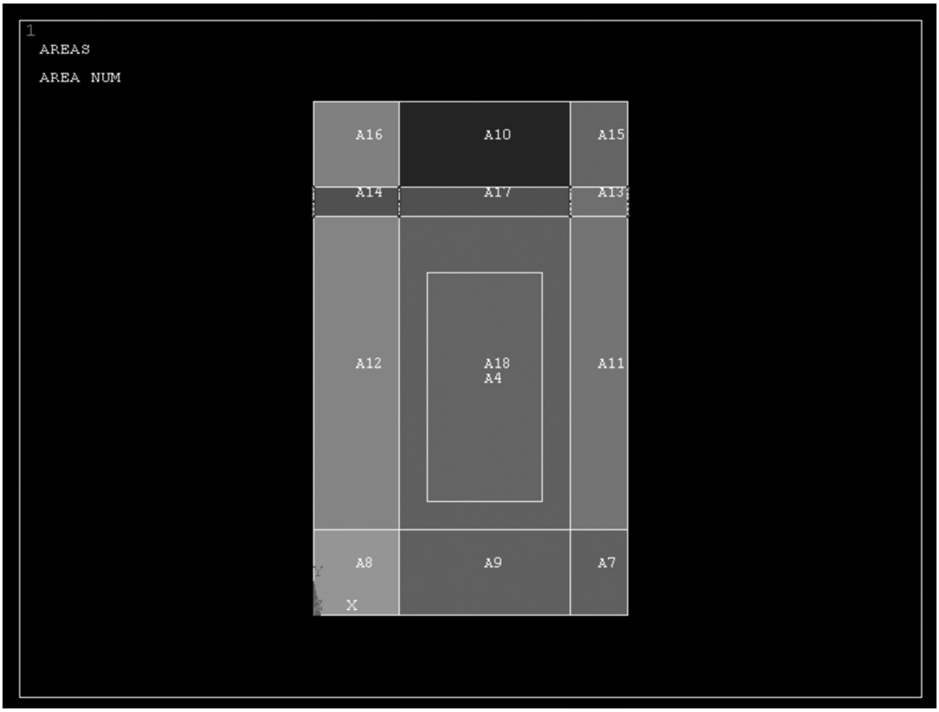


FIGURE 10.56 Five Back-iron Areas, A7, A8, A9, A11, A12.

8. **OK**
9. (drop down) "Material number" = 2
10. **Apply**
11. Pick coil area, A4
12. **OK**
13. (drop down) "Material number" = 3
14. **Apply**

15. Pick armature area, A10, A15, A16
16. **OK**
17. (drop down) "Material number" = 4
18. **OK**
19. Toolbar: **SAVE_DB**

(c) Specify meshing-size controls on air gap

Adjust meshing-size controls to get two element divisions through the air gap.

1. Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines
2. Pick four vertical lines through air gap
3. **OK**
4. "No. of element divisions" = 2
5. **OK**

(d) Mesh the model using the MeshTool

1. "Size control global" = [Set]
2. "Element edge length" = 0.25
3. **OK** as in Figure 10.57
4. (drop down) "Mesh" = Areas
5. **Mesh**
6. **Pick All**
7. **Close**

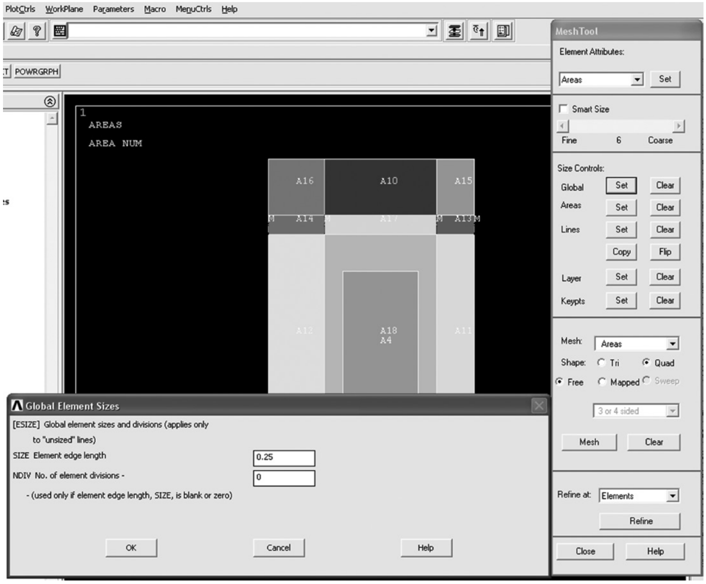


FIGURE 10.57 Global Element Sizes.

8. **Utility Menu > PlotCtrls > Numbering** as in Figure 10.58
9. (drop down) "Elem / attrib numbering" = Material numbers as in Figure 10.59
10. **OK** as in Figure 10.60

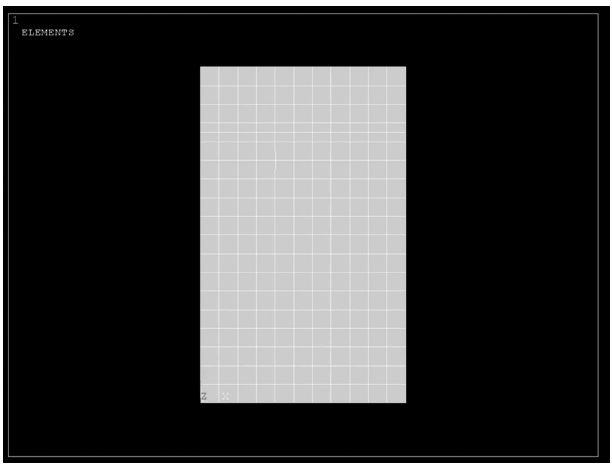


FIGURE 10.58 Numbering after PlotCtrls.

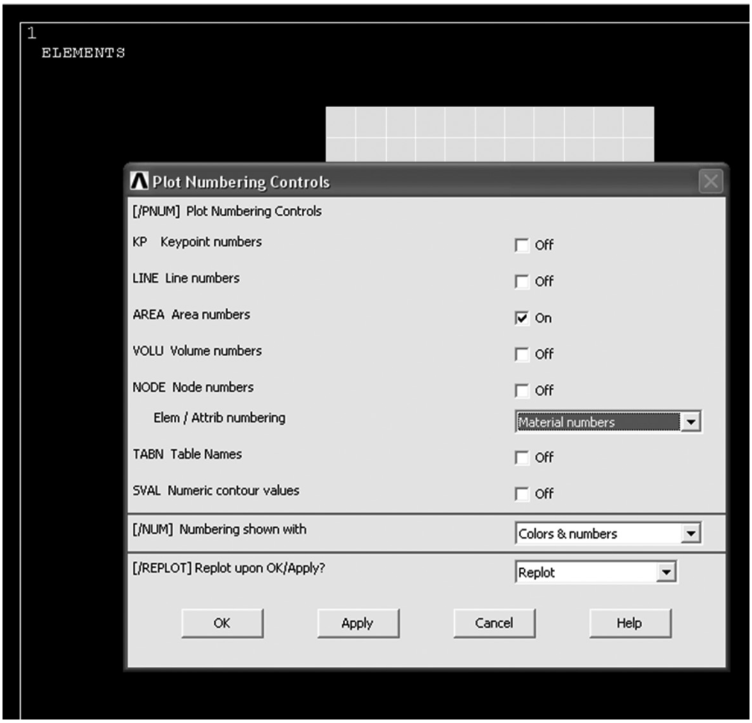


FIGURE 10.59 Plot Numbering Control.

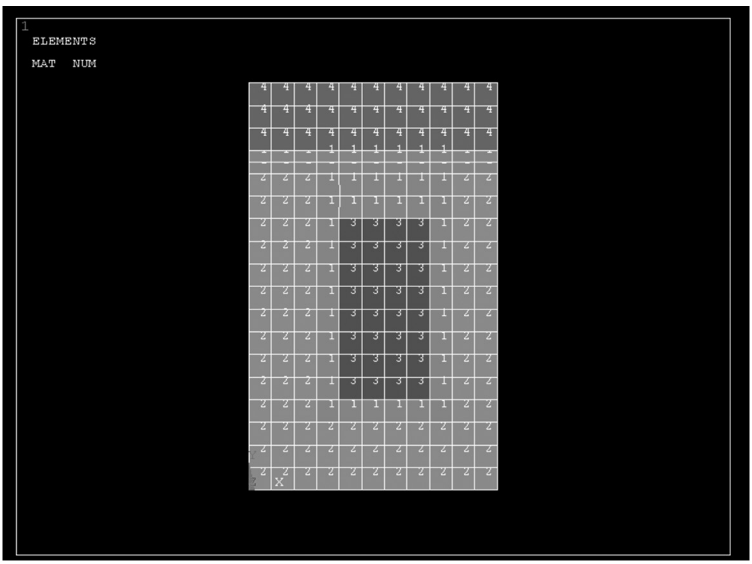


FIGURE 10.60 Numbering of the Model

(e) Scale model to meters for solution

For a magnetic analysis, a consistent set of units must be used. In this tutorial, MKS units are used, so you must scale the model from centimeters to meters.

1. **Main Menu > Preprocessor > Modeling > Operate > Scale > Areas**
2. **Pick All**
3. "RX,RY,RZ Scale Factors" = 0.01, 0.01, 1
4. (drop down) "Existing areas will be" = Moved
5. **OK** as in Figure 10.61
6. Toolbar: **SAVE_DB**

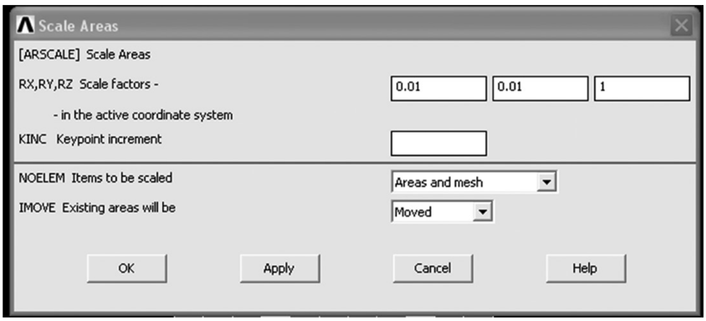


FIGURE 10.61 Scale area of the model.

4. **Apply Loads**

(a) Define the armature as a component

The armature can conveniently be defined as a component by selecting its elements.

1. **Utility Menu > Select > Entities**
2. (first drop down) "Elements"
3. (second drop down) "By Attributes"
4. "Min, Max, Inc" = 4

5. **OK** as in Figure 10.62

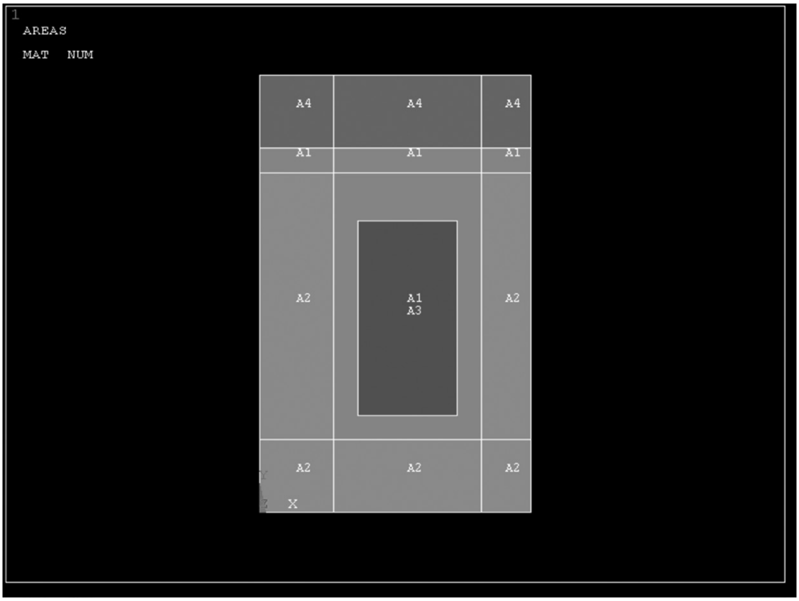


FIGURE 10.62 The Entities of the Model.

6. **Utility Menu > Plot > Elements** as in Figure 10.63

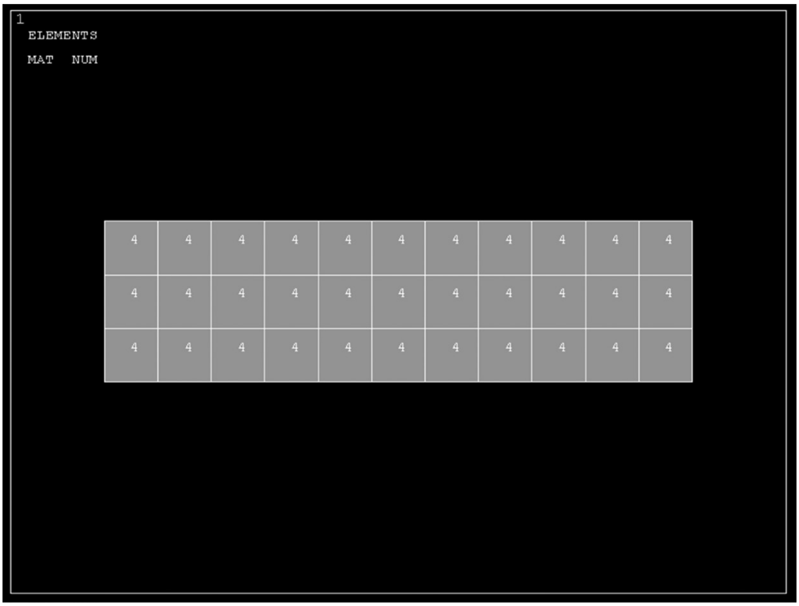


FIGURE 10.63 The Armature as a Component of the Model.

7. **Utility Menu > Select > Comp/Assembly > Create Component**
8. "Component name" = ARM
9. (drop down) "Component is made of" = Elements
10. **OK**

(b) Apply force boundary conditions to armature

1. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Magnetic > Flag > Comp. Force/Torq**
2. (highlight) "Component name" = ARM
3. **OK**
4. **Utility Menu > Select > Everything**
5. **Utility Menu > Plot > Elements** as in Figure 10.64

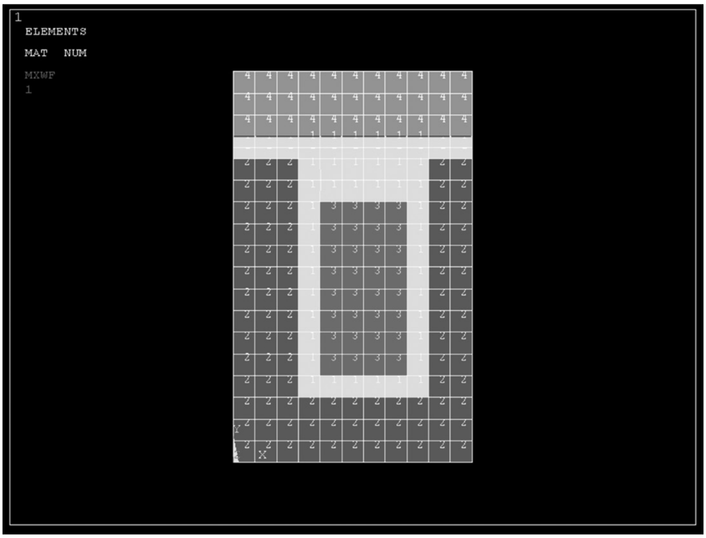


FIGURE 10.64 Plot of Apply Force Boundary Conditions to Armature.

(c) Apply the current density

The current density is defined as the number of coil windings times the current, divided by the coil area. This equals (650)(1)/2, or 325. To account for scaling from centimeters to meters, the calculated value needs to be divided by .01**2.

1. **Utility Menu > Plot > Areas**
2. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Magnetic > Excitation > Curr Density > On Areas**
3. Pick the coil area, which is the area in the center
4. **OK**
5. "Curr density value" = 325/.01**2
6. **OK**

   Close any warning messages that appear.

(d) Obtain a flux parallel field solution

Apply a perimeter boundary condition to obtain a "flux parallel" field solution. This boundary condition assumes that the flux does not leak out of the iron at the perimeter of the model. Of course, at the centerline, this is true due to axisymmetry.

1. **Utility Menu > Plot > Lines**
2. **Main Menu > Preprocessor > Loads > Define Loads > Apply > Magnetic > Boundary > Vector Poten > Flux Par'l > On Lines**
3. Pick all lines around perimeter of model (14 lines) as in Figure 10.65

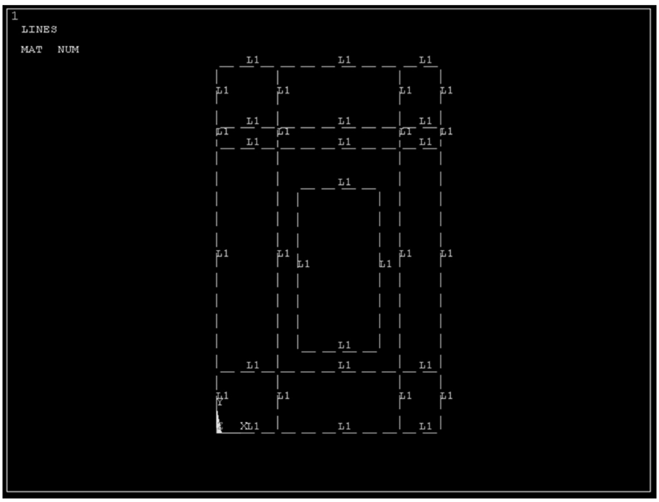


FIGURE 10.65 Plot of Lines for Flux Parallel Field of the Model.

4. **OK**
5. Toolbar: **SAVE_DB**

5. **Obtain solution**

(a) **Solve**

1. **Main Menu > Solution > Solve > Electromagnet > Static Analysis > Opt & Solve**
2. **OK** to initiate the solution
3. **Close** the information window when solution is done

6. **Review results**

(a) **Plot the flux lines in the model**

Note that a certain amount of undesirable flux leakage occurs out of the back-iron.

1. **Main Menu > General Postproc > Plot Results > Contour Plot > 2D Flux Lines**
2. **OK,** as in Figure 10.66

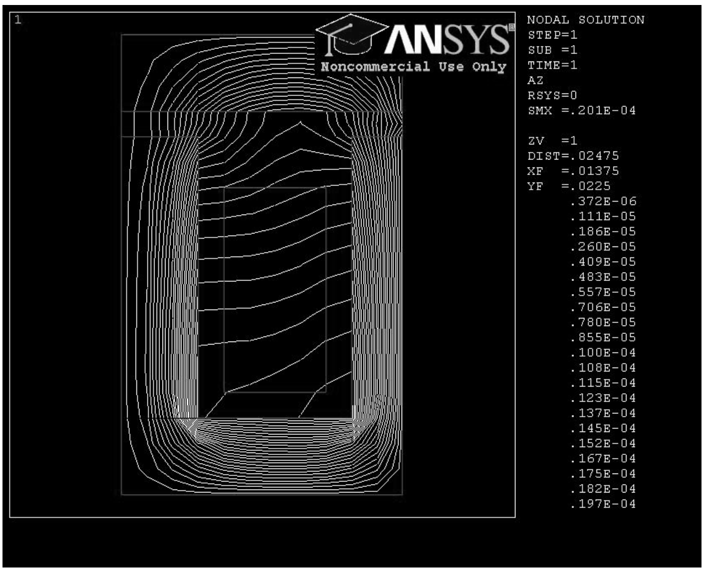


FIGURE 10.66 Contour Plot for 2D Flux Lines of the Model.

Your results may vary slightly from what is shown here due to variations in the mesh.

(b) Summarize magnetic forces

1. **Main Menu > General Postproc > Elec & Mag Calc > Component Based > Force**
2. (highlight) "Component name(s)" = ARM
3. **OK**
4. Review the information, then choose:

   File > Close (Windows),

   or

   Close (X11/Motif) to close the window.

(c) Plot the flux density as vectors

1. **Main Menu > General Postproc > Plot Results > Vector Plot > Predefined**
2. "Flux & gradient" (left column)
3. "Mag flux dens B" (right column)
4. **OK** as in Figure 10.67

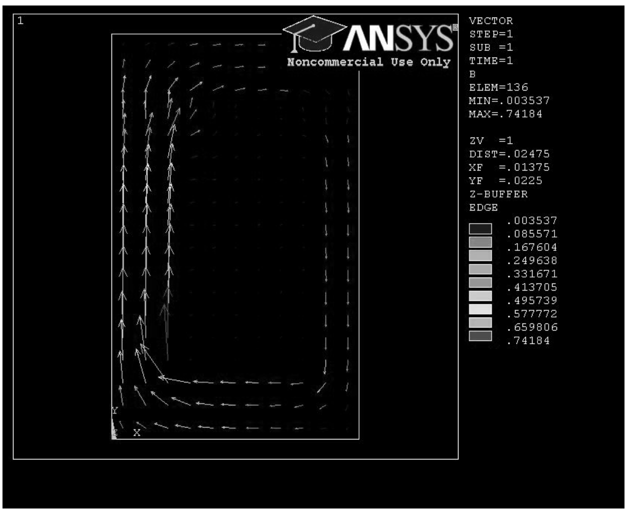


FIGURE 10.67 Plot the Flux Density as Vectors of the Model.

(d) Plot the magnitude of the flux density

Plot the magnitude of the flux density without averaging the results across material discontinuities.

1. **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu**
2. Choose "Magnetic Flux Density," then "Magnetic flux density vector sum"
3. **OK** as in Figure 10.68

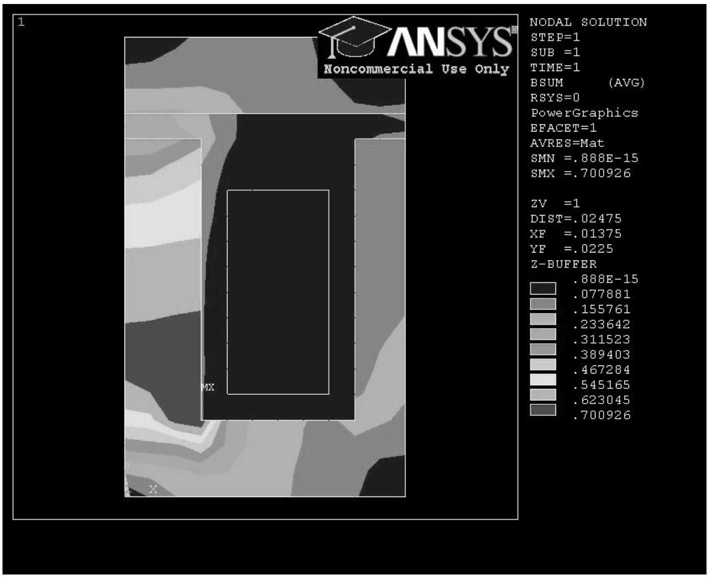


FIGURE 10.68 Contour Plot of the Model.

Next, you will see how the flux density is distributed throughout the entire actuator. Up to this point, the analysis and all associated plots have used the 2D axisymmetric model, with the axis of symmetry aligned with the left vertical portion of the device. ANSYS will continue the analysis on the 2D finite element model, but will allow you to produce a three-quarter expanded plot representation of the flux density throughout the device, based on the defined axisymmetry. This function is purely graphical. No changes to the database will be made when you produce this expanded plot.

4. **Utility Menu > PlotCtrls > Style > Symmetry Expansion > 2D Axi-Symmetric**

5. (check) "3/4 expansion" as in Figure 10.69
6. **OK** as in Figure 10.70

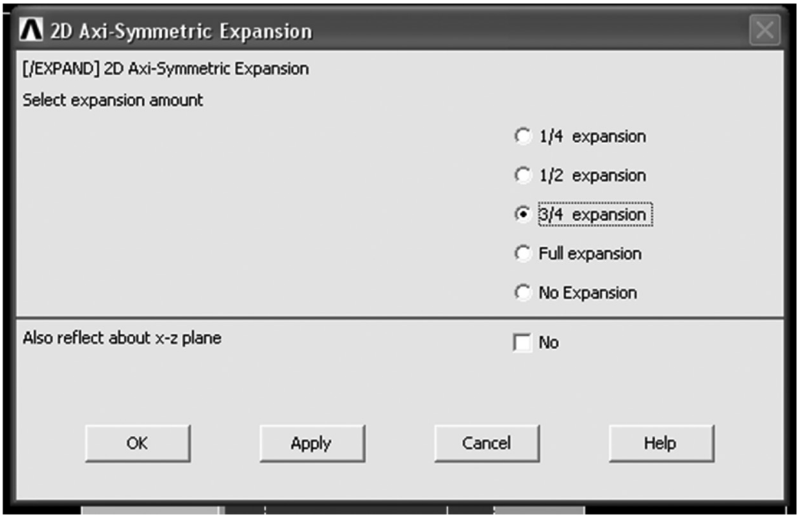


FIGURE 10.69 2D Axi-symmetric Expansion with Amount ¾.

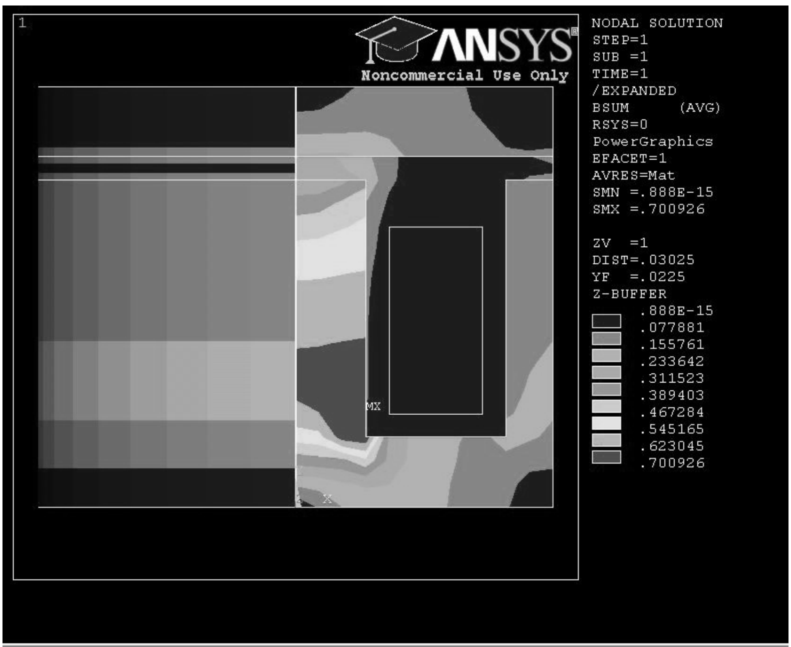


FIGURE 10.70 2D Axi-Symmetric Plot of the Model.

7. **Utility Menu > PlotCtrls > Pan,Zoom,Rotate, as in Figure 10.71**
8. **Iso**
9. **Close**

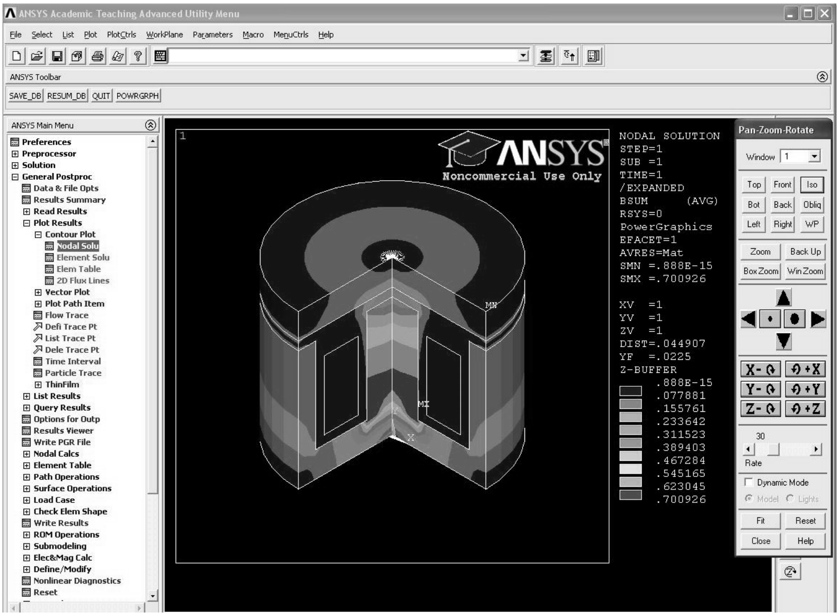


FIGURE 10.71 Rotation of the Model.

(e) Exit the ANSYS program

1. Toolbar: **QUIT**
2. (check) "Quit - No Save!"
3. **OK**

EXERCISES

1. Given $\mathbf{H} = He^{j(\omega t + 2\beta z)}\mathbf{a}_x$ in free space, known that, $\nabla \times \mathbf{H} = \dfrac{\partial \mathbf{D}}{\partial t}$, find **E**.
2. Calculate the skin depth, δ, for a copper conductor in a 50 Hz field ($\sigma = 56 \times 10^6\, S/m$).

3. EM exercises and examples.
4. For the axisymmetric coaxial cable illustrated in Figure 10.71. Determine a one-dimension finite element general solution based on the following:
   - **a.** Obtain and solve the governing differential solution for the coaxial cable, hint: $\frac{\varepsilon}{r}\frac{d}{dr}\left(r\frac{d\phi}{dr}\right) = -\rho$.
   - **b.** Obtain the boundary conditions and continuity conditions, hint: $\phi_1(r=a)=\phi_a$, $\phi_2(r=c)=0$, and the electric potential and the electric displacement are continuous at $r=b$.
   - **c.** Formulate the equations of part (b) as a matrix equation that can be solved for the constants of integrations.
   - **d.** Determine the shape functions for a general three-node quadratic element in terms of $x_1, x_2,$ and x_3 .
   - **e.** Determine the shape functions for a general three-node quadratic element when $x_1 = -L, x_2 = 0,$ and $x_3 = L$.
   - **f.** Find the local stiffness matrix for an element of length 2L with coordinates (–L, 0, L).

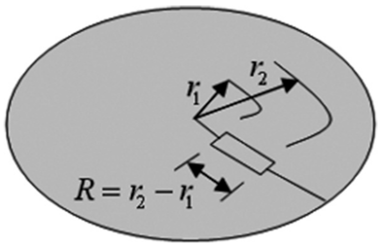


FIGURE 10. 71(a) axisymmetric radial element.

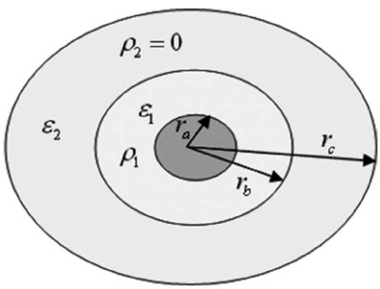


FIGURE 10.71(b) Coaxial cable.

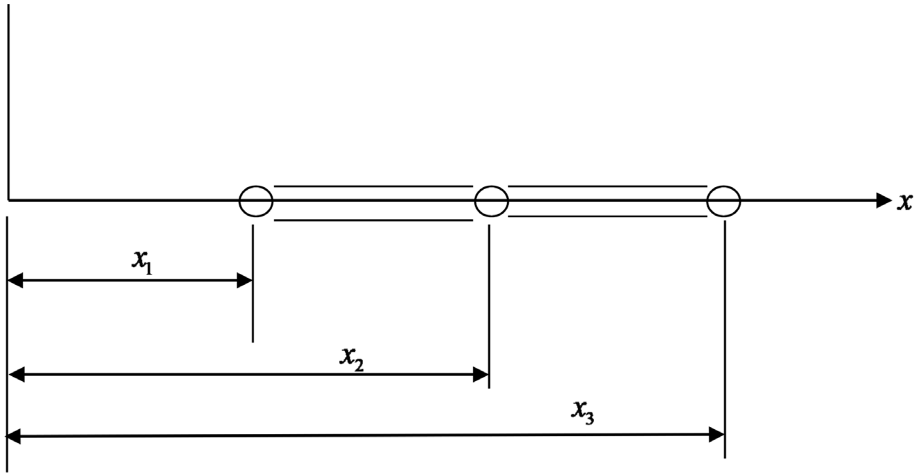


FIGURE 10.71(c) Three-node element.

5. Determine the variational function for two-dimensional axisymmetric heat conduction in r, z coordinate and formulate the corresponding local finite stiffness matrix using three-node triangular elements.

6. Use COMSOL in modeling the four-conductor transmission lines with the following parameters as in Figure 10.72:

   ε_{r1} = dielectric constant of the dielectric material = 4.2

   ε_{r2} = dielectric constant of the free space = 1.0

   W = width of the dielectric material = 10 mm

   w = width of a single conductor line = 1 mm

   H_1 = distance of conductors 1 and 2 from the ground plane = 3 mm

   H_2 = distance of conductor 4 from the ground plane = 1 mm

   H_3 = distance of conductor 3 from the ground plane = 2 mm

   s = distance between the two coupled conductors = 1 mm

   t = thickness of the strips = 0.01 mm

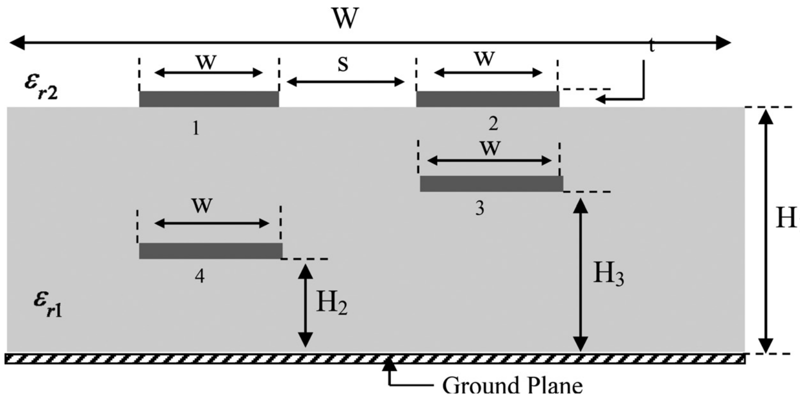


FIGURE 10.72 Cross-section of the four-conductor transmission lines.</FT<

The geometry is enclosed by a 10 × 10 mm shield. Find the capacitances per unit length, C_{11}, C_{12}, C_{13}, C_{14}, C_{22}, C_{23}, C_{24}, C_{33}, C_{34}, and C_{44}.

7. Use COMSOL in modeling of the shielded two vertically coupled striplines geometry is enclosed by a 3.4 × 1 mm shield with the following parameters as in Figure 10.73:

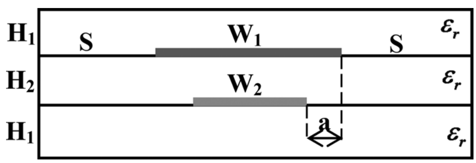


FIGURE 10.73 Cross-section of the two shielded vertically coupled striplines embedded in dielectric material.

ε_r = dielectric constant = 1 and 7.5

W_1 = width of the stripline 1 = 1.4 mm

W_2 = width of the stripline 2 = 1 mm

H_1 = height from stripline 1 and stripline 2 to the upper side and lower side of the shield, respectively = 0.4 mm

H_2 = distance between the two striplines = 0.2 mm

S = distance between the stripline 1 and right/left side of the shield = 1mm

a = $(W_1 - W_2)/2 = 0.2$ mm

t = thickness of the striplines = 0.01 mm

Find the capacitances per unit length, C_{11} , C_{21} , and C_{22} .

8. Use ANSYS Modeling of harmonic high-frequency electromagnetic of a coaxial waveguide as shown in Figure 10.74. The properties of the model are summarized as

Material property:

$\mu_r = 1.0, \varepsilon_r = 1.0$,

Geometric property:

$r_i = 0.025m$, $r_o = 0.075m$, $I = 0.375m$,

Load used

Port voltage = 1.0

$\Omega = 0.8\ GHz$

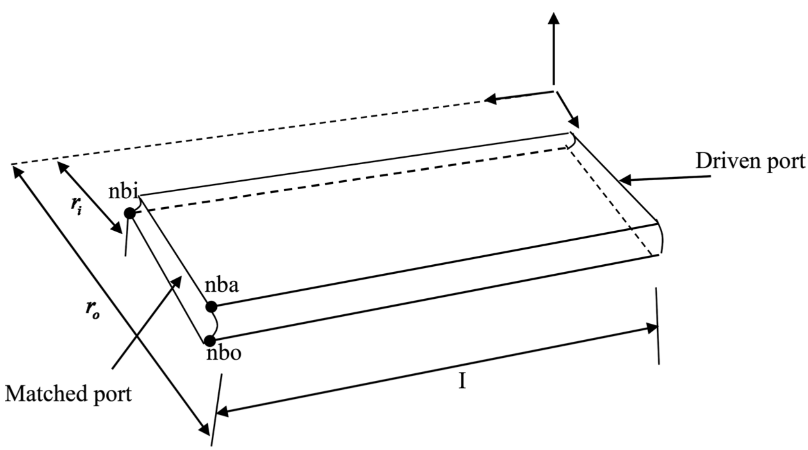


FIGURE 10.74 Cross-section of a coaxial waveguide.

Find $S11, S12, Z_{\mathrm{Re}}, Z_{im}, RL$

9. Use ANSYS Modeling of electrostatic of a shielded microstrip transmission line consisting of a substrate, microstrip, and a shield. The strip is at potential V_1, and the shield is at a potential V_0. Find the capacitance of the transmission line as shown in Figure 10.75.

The properties of the model is summarized as

Material property:

Air: $\varepsilon_r = 1$

Substrate: $\varepsilon_r = 12$

Geometric property:

a = 10 cm

b = 1 cm

w = 2 cm

Loading property:

V_0 = 1 V

V_1 = 10 V

Knowing that the electrostatic energy, W_e is defined as

$$W_e = \frac{1}{2} C \left(V_1 - V_0 \right)^2 .$$

Also, you need to type the following values in scalar parameters as:

C = (w*2)/((V_1–V_0)**2) and C = ((C*2)*1e12.

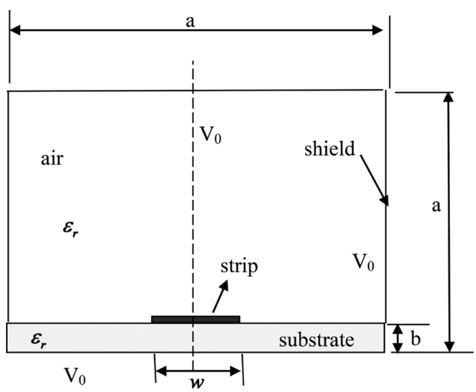


FIGURE 10.75 Cross-section of shielded microstrip line.

REFERENCES

1. M. N. O. Sadiku, "A Simple Introduction to Finite Element Analysis of Electromagnetic Problem," *IEEE Trans. Educ.*, vol. 32, no. 2, 1989, pp. 85–93.
2. P. Tong, "Exact Solution of Certain Problems by the Finite Element Method," *AIAA*, vol. 7, no. 1, 1969, pp. 179–180.
3. E. A. Thornton, P. Dechaumphai, and K. K. Tamma, "Exact Finite Elements for Conduction and Convection," *Proceedings of Second International Conference on Numerical Methods in Thermal Problems*, Venice, Italy, July 7–10, 1981, Swansea, Wales, Pineridge Press, 1981, pp. 1133–1144.
4. B. A. Finlayson and L. E. Scriven, "The method of weighted residuals-a review," *Appl. Mech. Rev.*, vol. 19, no. 9, 1966, pp. 735–748.
5. R. W. Klopfenstein and C. P. Wu, "Computer Solution of One-Dimensional Poisson's Equation," *IEEE Transactions on Electron Devices*, vol. 22, no. 6, 1976, pp. 329–333.
6. P. P. Silverster and R. L. Ferrari, *Finite Elements for Electrical Engineers.* Cambridge: Cambridge University Press, 3rd ed., 1996.
7. C. R. Paul and S. A. Nasar, *Introduction to Electromagnetic Fields.* New York: McGraw-Hill, 1982, pp. 465–472.
8. O. W. Anderson, "Laplacian Electrostatic Field Calculations by Finite Elements with Automatic Grid Generation," *IEEE Trans. Power App. Syst.*, vol. PAS-92, no. 5, 1973, pp. 1485–1492.
9. M. L. James et al., *Applied Numerical Methods for Digital Computation.* New York: Harper and Row, 1985, pp. 146–274.
10. R. E. Collin, *Field Theory of Guide Waves.* New York: McGraw-Hill, 1960, p. 128.
11. B. S. Garhow, *Matrix Eigensystem Routine-EISPACK Guide Extension.* Berlin: Springer-Verlag, 1977.
12. S. Ahmed and P. Daly, "Finite-Element Methods for Inhomogeneous Waveguides," *Proc. IEEE*, vol. 116, no. 10, Oct. 1969, pp. 1661–1664.

13. Z. J. Csendes and P. Silvester, "Numerical Solution of Dielectric Loaded Waveguides: I -Finite-Element Analysis," *IEEE Trans. Micro. Tech.*, vol. MTT-18, no. 12, Dec. 1970, pp. 1124–1131.

14. Z. J. Csendes and P. Silvester, "Numerical Solution of Dielectric Loaded Waveguides: II-Modal Approximation Technique," *IEEE Trans. Micro. Theo. Tech.*, vol. MTT-19, no. 6, 1971, pp. 504–509.

15. M. Hano, "Finite-Element Analysis of Dielectric-Loaded Waveguides," *IEEE Trans. Micro.Theo.Tech.*, vol. MTT-32, no. 10, 1984, pp. 1275–1279.

16. A. Konrad, "Vector Variational Formulation of Electromagnetic Fields in Anisotropic Media," *IEEE Trans. Micro. Theo. Tech.*, vol. MTT-24, Sept. 1976, pp. 553–559.

17. M. Koshiba et al., "Improve Finite-Element Formulation in Terms of the Magnetic Field Vector for Dielectric Waveguides," *IEEE Trans. Micro. Theo. Tech.*, vol. MTT-33, no. 3, 1985, pp. 227–233.

18. M. Koshiba et al., "Finite-Element Formulation in Terms of the Electric-Field Vector for Electromagnetic Waveguide Problems," *IEEE Trans. Micro. Theo. Tech.*, vol. MTT-33, no. 10, 1985, pp. 900–905.

19. K. Hayata et al., "Vectorial Finite-Element Method Without Any Spurious Solutions for Dielectric Waveguiding Problems Using Transverse Magnetic-Field Component," *IEEE Trans. Micro. Theo. Tech.*, vol. MTT-34, no. 11, 1986, pp. 1120–1124.

20. K. Hayata et al., "Novel Finite-Element Formulation Without Any Spurious Solutions for Dielectric Waveguides," *Elect. Lett.*, vol. 22, no. 6, 1986, pp. 295–296.

21. S. Dervain, "Finite Element Analysis of Inhomogeneous Waveguides," Master thesis, Department of Electrical and Computer Engineering, Florida Atlantic University, Boca Raton, 1987.

22. J. R. Winkler and J. B. Davies, "Elimination of Spurious Modes in Finite Element Analysis," *J. Comp. Phys.*, vol. 56, no. 1, 1984, pp. 1–14.

23. W. R. Buell and B. A. Bush, "Mesh Generation-A Survey," *J. Eng. Ind.*, Feb. 1973, pp. 332–338.

24. W. C. Thacker, "A Brief Review of Techniques for Generating Irregular Computational Grids," *Int. J. Num. Meth. Eng*., vol. 15, 1980, pp. 1335–1341.

25. E. Hinton and D. R. J. Owen, *An Introduction to Finite Element Computations.* Swansea, U. K.: Pineridge, 1980, pp. 247, 260, 328–346.

26. J. N. Reddy, *An Introduction to the Finite Element Method.* New York: McGraw-Hill, 1984, pp. 340–345, 436.

27. M. N. O. Sadiku et al., "A Further Introduction to Finite Element Analysis of Electromagnetic Problems," *IEEE Trans. Educ*., vol. 34, no. 4, 1991, pp. 322–329.

28. M. Kono, "A Generalized Automatic Mesh Generation Scheme for Finite Element Method," *Inter. J. Num. Meth. Engr.*, vol. 15, 1980, pp. 713–731.

29. J. C. Cavendish, "Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method," *Inter. J. Num. Meth. Engr*., vol. 8, 1974, pp. 676–696.

30. A. O. Moscardini et al., "AGTHOM-Automatic Generation of Triangular and Higher Order Meshes," *Inter. J. Num. Meth. Engr*., vol. 19, 1983, pp. 1331–1353.

31. C. O. Frederick et al., "Two-Dimensional Automatic Mesh Generation for Structured Analysis," *Inter. J. Num. Meth. Engr*., vol. 2, no. 1, 1970, pp. 133–144.

32. E. A. Heighway, "A Mesh Generation for Automatically Subdividing Irregular Polygon into Quadrilaterals," *IEEE Trans. Mag*., vol. MAG-19, no. 6, 1983, pp. 2535–2538.

33. C. Kleinstreuer and J. T. Holdeman, "A Triangular Finite Element Mesh Generator for Fluid Dynamic Systems of Arbitrary Geometry," *Inter. J. Num. Meth. Engr.*, vol. 15, 1980, pp. 1325–1334.

34. A. Bykat, "Automatic Generation of Triangular Grid I-Subdivision of a General Polygon into Convex Subregions. II-Triangulation of Convex Polygons," *Inter. J. Num. Meth. Engr*., vol. 10, 1976, pp. 1329–1342.

35. N. V. Phai, "Automatic Mesh Generator with Tetrahedron Elements," *Inter. J. Num. Meth. Engr*., vol. 18, 1982, pp. 273–289.

36. F. A. Akyuz, "Natural Coordinates Systems-an Automatic Input Data Generation Scheme for a Finite Element Method," *Nuclear Engr. Design*, vol. 11, 1970, pp. 195–207.

37. P. Girdinio et al., "New Developments of Grid Optimization by the Grid Iteration Method," in Z. J. Csendes (ed.), *Computational Electromagnetism.* New York: North-Holland, 1986, pp. 3–12.

38. M. Yokoyama, "Automated Computer Simulation of Two-Dimensional Electrostatic Problems by Finite Element Method," *Inter. J. Num. Meth. Engr.*, vol. 21, 1985, pp. 2273–2287.

39. G. F. Carey, "A Mesh-Refinement Scheme for Finite Element Computations," *Comp. Meth. Appl. Mech. Engr.*, vol. 7, 1976, pp. 93–105.

40. K. Preiss, "Checking the Topological Consistency of a Finite Element Mesh," *Inter. J. Meth. Engr.*, vol. 14, 1979, pp. 1805–1812.

41. H. Kardestuncer (ed.), *Finite Element Handbook.* New York: McGraw-Hill, 1987, pp. 4.191–4.207.

42. W. C. Thacker, "A Brief Review of Techniques for Generating Irregular Computational Grids," *Inter. J. Num. Meth. Engr.*, vol. 15, 1980, pp. 1335–1341.

43. E. Hinton and D. R. J. Owen, "*An Introduction to finite Element Computations.* Swansea, UK: Pineridge Press, 1979, pp. 247, 328–346.

44. C. S. Desai and J. F. Abel, *Introduction to the Finite Element Method: A Numerical Approach for Engineering Analysis.* New York: Van Nostrand Reinhold, 1972.

45. M. V. K. Chari and P. P. Silvester (eds.), *Finite Elements for Electrical and Magnetic Field Problems.* Chichester: John Wiley, 1980, pp. 125–143.

46. P. Silvester, "Construction of Triangular Finite Element for Universal Matrices," *Inter. J. Num. Meth. Engr.*, vol. 12, 1978, pp. 237–244.

47. P. Silvester, "Higher-Order Polynomial Triangular Finite Elements for Potential Problems," *Inter. J. Engr. Sci.*, vol. 7, 1969, pp. 849–861.

48. G. O. Stone, "High-Order Finite Elements for Inhomogeneous Acoustic Guiding Structures," *IEEE Trans. Micro. Theory Tech.*, vol. MTT-21, no. 8, 1973, pp. 538–542.

49. A. Konrad, "High-Order Triangular Finite Elements for Electromagnetic Waves in Anisotropic Media," *IEEE Trans. Micro. Theory Tech.*, vol. MTT-25, no. 5, 1977, pp. 353–360.

50. P. Daly, "Finite Elements for Field Problems in Cylindrical Coordinates," *Inter. J. Num. Meth. Engr.*, vol. 6, 1073, pp. 169–178.

51. C. A. Brebbia and J. J. Connor, *Fundamentals of Finite Element Technique.* London: Butterworth, 1973, pp. 114–118, 150–163, 191.

52. M. N. O. Sadiku and L. Agba, "New Rules for Generating Finite Elements Fundamental Matrices," *Proc. IEEE Southeastcon*, 1989, pp. 797–801.

53. R. L. Ferrari and G. L. Maile, "Three-Dimensional Finite Element Method for Solving Electromagnetic Problems," *Elect. Lett.*, vol. 14, no. 15, 1978, p. 467.

54. M. dePourcq, "Field and Power-Density Calculation by Three-Dimensional Finite Elements," *IEEE Proc.*, vol. 130, no. 6, 1983, pp. 377–384.

55. M V. K. Chari et al., "Finite Element Computation of Three-Dimensional Electrostatic and Magnetostatic Filed Problems," *IEEE Trans. Mag.*, vol. 19, no. 16, 1983, pp. 2321–2324.

56. O. A. Mohammed et al., "Validity of Finite Element Formulation and Solution of Three Dimensional Magnetostatic Problems in Electrical Devices with Applications to Transformers and Reactors," *IEEE Trans. Pow. App. Syst.*, vol. 103, no. 7, 1984, pp. 1846–1853.

57. J. S. Savage and A. F. Peterson, "Higher-Order Vector Finite Elements for Tetrahedral Cells," *IEEE Trans. Micro. Theo. Theo. Tech.*, vol. 44, no. 6, 1996, pp. 874–879.

58. J. F. Lee and Z. J. Cendes, "Transfinite Elements: A Highly Efficient Procedure for Modeling Open Field Problems," *Jour. Appl. Phys.*, vol. 61, no. 8, 1987, pp. 3913–3915.

59. B. H. McDonald and A. Wexler, "Finite-Element Solution of Unbounded Field Problems," *IEEE Trans. Micro. Theo. Tech.* vol. 20, no. 12, 1977, pp. 1267–1270.

60. P. P. Silvester et al., "Exterior Finite Elements for 2-Dimentional Field Problems with Open Boundaries," *Proc. IEEE*, vol. 124, no. 12, 1972, pp. 841–847.

61. S. Washisu et al., "Extension of Finite-Element Method to Unbounded Field Problems," *Elect. Lett.*, vol. 15, no. 24, 1979, pp. 772–774.

62. P. P. Silvester and M. S. Hsieh, "Finite-Element Solution of 2-Dimentional Exterior-Field Problems," *Proc. IEEE*, vol. 118, no. 12, 1971, pp. 1743–1747.

63. Z. J. Csendes, "A Note on the Finite-Element Solution of Exterior-Field Problems," *IEEE Trans. Micro. Theo. Tech.*, vol. 24, no. 7, 1976, pp. 468–473.

64. T. Corzani et al., "Numerical Analysis of Surface Wave Propagation Using Finite and Infinite Elements," *Alta Frequenza*, vol. 51, no. 3, 1982, pp. 127–133.

65. O. C. Zienkiewicz et al., "Mapped Infinite Elements for Exterior Wave Problems," *Iner. J. Num. Meth. Engr.*, vol. 21, 1985.

66. F. Medina, "An Axisymmetric Infinite Element," *Int. J. Num. Meth Engr.*, vol. 17, 1981, pp. 1177–1185.

67. S. Pissanetzky, "A Simple Infinite Element," *Int. J. Comp. Math Elect. Engr.*, vol. 3, no. 2, 1984, pp. 107–114.

68. Z. Pantic and R. Mittra, "Quasi-TEM Analysis of Microwave Transmission Lines by the Finite-Element Method," *IEEE Trans. Micro. Theo. Tech.*, vol. 34, no. 11, 1986, pp. 1096–1103.

69. K. Hayata et al., "Self-Consistent Finite/Infinite Element Scheme for Unbounded Guided Wave Problems," *IEEE Trans. Micro. Theo. Tech.*, vol. 36, no. 3, 1988, pp. 614–616.

70. P. Petre and L. Zombory, "Infinite Elements and Base Functions for Rotationally Symmetric Electromagnetic Waves," *IEEE Trans. Ant. Prog.*, vol. 36, no. 10, 1988, pp. 1490–1491.

71. Z. J. Csenes and J. F. Lee, "The Transfinite Element Method for Modeling MMIC Device," *IEEE Trans. Micro. Theo. Tech.*, vol. 36, no. 12, 1988, pp. 1639–1649.

72. K. H. Lee et al., "A Hybrid Three-Dimensional Electromagnetic Modeling Scheme," *Geophys.*, vol. 46, no. 5, 1981, pp. 779–805.

73. S. J. Salon and J. M. Schneider, "A Hybrid Finite Element-Boundary Integral Formulation of Poisson's Equation," *IEEE Trans. Mag.*, vol. 17, no. 6, 1981, pp. 2574–2576.

74. S. J. Salon and J. Peng, "Hybrid Finite-Element Boundary-Element Solutions to Axisymmetric Scalar Potential Problems," in Z. J. Csendes (ed.), *Computational Electromagnetics.* New York: North-Holland/ Elsevier, 1986, pp. 251–261.

75. J. M. Lin and V. V. Liepa, "Application of Hybrid Finite Element Method for Electromagnetic Scattering from Coated Cylinders," *IEEE Trans. Ant. Prop.*, vol. 36, no. 1, 1988, pp. 50–54.

76. J. M. Lin and V. V. Liepa, "A Note on Hybrid Finite Element Method for Solving Scattering Problems," *IEEE Trans. Ant. Prop.*, vol. 36, no. 10, 1988, pp. 1486–1490.

77. M. H. Lean and A. Wexler, "Accurate Field Computation with Boundary Element Method," *IEEE Trans. Mag.*, vol. 18, no. 2, 1982, pp. 331–335.

78. R. F. Harrington and T. K. Sarkar, "Boundary Elements and Method of Moments," in C. A. Brebbia et al. (eds), *Boundary Elements.* Southampton: CML Publ., 1983, pp. 31–40.

79. M. A. Morgan et al., "Finite Element-Boundary Integral Formulation for Electromagnetic Scattering," *Wave Motion*, vol. 6, no. 1, 1984, pp. 91–103.

80. S. Kagami and I. Fukai, "Application of Boundary-Element Method to Electromagnetic Field Problems," *IEEE Trans. Micro. Theo. Tech.*, vol. 32, no. 4, 1984, pp. 455–461.

81. Y. Tanaka et al., "A Boundary-Element Analysis of TEM Cells in Three Dimensions," *IEEE Trans. Elect. Comp.*, vol. 28, no. 4, 1986, pp. 179–184.

82. N. Kishi and T. Okoshi, "Proposal for a Boundary-Integral Method Without Using Green's Function," *IEEE Trans. Micro. Theo. Tech.*, vol. 35, no. 10, 1987, pp. 887–892.

83. D. B. Ingham et al., "Boundary Integral Equation Analysis of Transmission-Line Singularities," *IEEE Trans. Micro. Theo. Tech.*, vol. 29, no. 11, 1981, pp. 1240–1243.

84. S. Washiru et al., "An Analysis of Unbounded Field Problems by Finite Element Method," *Electr. Comm. Japan*, vol. 64-B, no. 1, 1981, pp. 60–66.

85. T. Yamabuchi and Y. Kagawa, "Finite Element Approach to Unbounded Poisson and Helmholtz Problems Using-Type Infinite Element," *Electr. Comm. Japan*, Pt. I, vol. 68, no. 3, 1986, pp. 65–74.

86. K. L. Wu and J. Litva, "Boundary Element Method for Modeling MIC Devices," *Elect. Lett.*, vol. 26, no. 8, 1990, pp. 518–520.

87. P. K. Kythe, *An Introduction to Boundary Element Methods.* Boca Raton, FL: CRC Press, 1995, p. 2.

88. J. M. Jin et al., "Fictitious Absorber for Truncating Finite Element Meshes in Scattering," *IEEE Proc. H*, vol. 139, 1992, pp. 472–476.

89. R. Mittra and O. Ramahi, "Absorbing Bounding Conditions for Direct Solution of Partial Differential Equations Arising in Electromagnetic Scattering Problems," in M. A. Morgan (ed.), *Finite Element and Finite Difference Methods in Electromagnetics.* New York: Elsevier, 1990, pp. 133–173.

90. U. Pekel and R. Mittra, "Absorbing Boundary Conditions for Finite Element Mesh Truncation," in T. Itoh se al. (eds.), *Finite Element Software for Microwave Engineering.* New York: John Wiley & Sons, 1996, pp. 267–312.

91. U. Pekel and R. Mittra, "A Finite Element Method Frequency Domain Application of the Perfectly Matched Layer (PML) Concept," *Micro. Opt. Technol. Lett.*, vol. 9, pp. 117–122.

92. A. Boag and R. Mittra, "A numerical absorbing boundary condition for finite difference and finite element analysis of open periodic structures," *IEEE Trans. Micro. Theo. Tech.*, vol. 43, no. 1, 1995, pp. 150–154.

93. P. P. Silvester and G. Pelosi (eds.), *Finite Elements for Waves Electromagnetics: Methods and Techniques.* New York: IEEE Press, 1994, pp. 351–490.

94. A. M. Bayliss, M. Gunzburger, and E. Turkel, "Boundary conditions for the numerical solution of elliptic equation in exterior regions," *SIAM Jour. Appl. Math.*, vol. 42, 1982, pp. 430–451.

95. M. N. O. Sadiku, *Elements of Electromagnetics*. Fifth Edition. New York: Oxford University Press, 2010.

96. J. A. Kong, *Electromagnetic Wave Theory.* New York: John Wiley and Sons, Inc., 1986.

97. *www.comsol.com*

98. J. Jin, *The Finite Element Method in Electromagnetics*, Second Edition, New York: John Wiley & Sons Inc., 2002.

99. S. R. H. Hoole, *Computer-Aided and Design of Electromagnetic Devices.* New York: Elsevier, 1989.

100. U. S. Inan and R. A. Marshall, *Numerical Electromagnetics: The FDTD Method,* Cambridge, New York: Cambridge University Press, 2011.

101. C. W. Steele, *Numerical Computation of Electric and Magnetic Fields*, New York: Van Nostrand Reinhold Company, 1987.

102. J. Jin, *Theory and Computation of Electromagnetic Fields*, New York: John Wiley & Sons Inc., 2010.

103. P. P. Silvester and R. L. Ferrari, *Finite Elements for Electrical Engineering.* Second Edition, Cambridge, New York: Cambridge University Press, 1990.

104. D. K. Cheng, *Field and Wave Electromagnetic.* Second Edition. New York: John Wiley and Sons, Inc., 1992.

105. R. F. Harrington, *Time-Harmonic Electromagnetic Fields*, New York: McGraw-Hill, 1961.

106. M. L. Crawford, "Generation of Standard EM Fields Using TEM Transmission Cells," *IEEE Transactions on. Electromagnetic Compatibility*, vol. EMC-16, pp. 189–195, Nov. 1974.

107. J. R. Reid and R. T. Webster, "A 60 GHz Branch Line Coupler Fabricated Using Integrated Rectangular Coaxial Lines," *Microwave Symposium Digest, 2004 IEEE MTT-S International,* Vol. 2, pp. 441–444, 6–11 June 2004.

108. S. Xu and P. Zhou, "FDTD Analysis for Satellite BFN Consisting of Rectangular Coaxial Lines," *Asia Pacific Microwave Conference*, pp.877–880, 1997.

109. J. G. Fikioris, J. L. Tsalamengas, and G. J. Fikioris, "Exact Solutions for Shielded Printed Microstrip Lines by the Carleman-Vekua Method," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 37, No. 1, pp. 21–33, Jan. 1989.

110. S. Khoulji and M. Essaaidi, "Quasi-Static Analysis of Microstrip Lines with Variable-Thickness Substrates Considering Finite Metallization Thickness," *Microwave and Optic Technology Letters*, Vol. 33, No. 1, pp. 19–22, April 2002.

111. T. K. Seshadri, S. Mahapatra, and K. Rajaiah, "Corner Function Analysis of Microstrip Transmission Lines," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 28, No. 4, pp. 376–380, April 1980.

112. S. V. Judd, I. Whiteley, R. J. Clowes, and D. C. Rickard, "An Analytical Method for Calculating Microstrip Transmission Line Parameters," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 18, No. 2, pp. 78–87, Feb. 1970.

113. N. H. Zhu, W. Qiu, E. Y. B. Pun, and P. S. Chung, "Quasi-Static Analysis of Shielded Microstrip Transmission Lines with Thick Electrodes," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 45, No. 2, pp. 288–290, Feb. 1997.

114. T. Chang and C. Tan, "Analysis of a Shielded Microstrip Line with Finite Metallization Thickness by the Boundary Element Method," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 38, No. 8, pp. 1130–1132, Aug. 1990.

115. G. G. Gentili and G. Macchiarella, "Quasi-Static Analysis of Shielded Planar Transmission Lines with Finite Metallization Thickness by a Mixed Spectral-Space Domain Method," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 42, No. 2, pp. 249–255, Feb. 1994.

116. A. Khebir, A. B. Kouki, and R. M. Mittra, "Higher Order Asymptotic Boundary Condition for Finite Element Modeling of Two-Dimensional Transmission Line Structures," *IEEE Transactions on Microwave Theory and Techniques,* Vol. 38, No. 10, pp. 1433–1438, Oct. 1990.

117. G. W. Slade and K. J. Webb, "Computation of Characteristic Impedance for Multiple Microstrip Transmission Lines Using a Vector Finite Element Method," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 40, No. 1, pp. 34–40, Jan. 1992.

118. M. S. Alam, K. Hirayama, Y. Hayashi, and M. Koshiba, "Analysis of Shielded Microstrip Lines with Arbitrary Metallization Cross Section Using a Vector Finite Element Method," *IEEE Transactions on*

Microwave Theory and Techniques, Vol. 42, No. 11, pp. 2112–2117, Nov. 1994.

119. J. Svacina, "A New Method for Analysis of Shielded Microstrips," *Proceedings of Electrical Performance of Electronic Packaging*, pp. 111–114, 1993.

120. H. Y. Yee, and K. Wu, "Printed Circuit Transmission-Line Characteristic Impedance by Transverse Modal Analysis," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 34, No. 11, pp. 1157–1163, Nov. 1986.

121. I. P. Hong, N. Yoon, S. K. Park, and H. K. Park, "Investigation of Metal-Penetrating Depth in Shielded Microstrip Line," *Microwave and Optic Technology Letters*, Vol. 19, No. 6, pp. 396–398, Dec. 1998.

122. http://www.comsol.com/

123. Q. Zheng, W. Lin, F. Xie, and M. Li, "Multipole Theory Analysis of a Rectangular Transmission Line Family," *Microwave and Optical Technology Letters*, Vol. 18, No. 6, pp. 382–384, Aug. 1998.

124. T. S. Chen, "Determination of the Capacitance, Inductance, and Characteristic Impedance of Rectangular Lines," *IEEE Transactions on Microwave Theory and Techniques*, Volume 8, Issue 5, pp.510–519, Sep. 1960.

125. E. Costamagna and A. Fanni, "Analysis of Rectangular Coaxial Structures by Numerical Inversion of the Schwarz-Christoffel Transformation," *IEEE Transactions on Magnets*, Vol. 28, pp. 1454–1457, Mar. 1992.

126. K. H. Lau, "Loss Calculation for Rectangular Coaxial Lines," *IEE Proceedings*, Vol. 135, Pt. H. No. 3, pp. 207–209, June 1988.

127. Personal computer program based on finite difference method.

128. J. D. Cockcroft, "The Effect of Curved Boundaries on the Distribution of Electrical Stress Round Conductors," *J. IEE*, Vol. 66, pp. 385–409, Apr. 1926.

129. F. Bowan, "Notes on Two Dimensional Electric Field Problems," *Proc. London Mathematical Society.*, Vol. 39, No. 211, pp. 205–215, 1935.

130. H. E. Green, "The Characteristic Impedance of Square Coaxial Line," *IEEE Transactions Microwave Theory and Techniques*, Vol. MTT-11, pp. 554–555, Nov. 1963.

131. S. A. Ivanov and G. L. Djankov, "Determination of the Characteristic Impedance by a Step Current Density Approximation," *IEEE Transactions on Microwave Theory and Techniques.*, Vol. MTT-32, pp. 450–452, Apr. 1984.

132. H. J. Riblet, "Expansion for the Capacitance of a Square in a Square with a Comparison," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 44, pp. 338–340, Feb. 1996.

133. M. S. Lin, "Measured Capacitance Coefficients of Multiconductor Microstrip Lines with Small Dimensions," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 13, No. 4, pp. 1050–1054, Dec. 1990.

134. F. Y. Chang, "Transient Analysis of Lossless Coupled Transmission Lines in a Nonhomogeneous Dielectric Media," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 18, No. 9, pp. 616–626, Aug. 1970.

135. P. N. Harms, C. H. Chan, and R. Mittra, "Modeling of Planar Transmission Line Structures for Digital Circuit Applications," *Arch. Eleck. Ubertragung.*, Vol. 43, pp. 245–250, 1989.

136. A. Kherbir, A. B. Kouki, and R. Mittra, "Absorbing Boundary Condition for Quasi-TEM Analysis of Microwave Transmission Lines via the Finite Element Method," *J. Electromagnetic Waves and Appl*ications, Vol. 4, No. 2, 1990.

137. A. Khebir, A. B. Kouki, and R. Mittra, "High Order Asymptotic Boundary Condition for the Finite Element Modeling of Two-Dimensional Transmission Line Structures," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 38, No. 10, pp. 1433–1438, Oct. 1990.

138. D. Homentcovschi, G. Ghione, C. Naldi, and R. Oprea, "Analytic Determination of the Capacitance Matrix of Planar or Cylindrical Multiconductor Lines," *IEEE Transactions on. Microwave Theory and Techniques*, pp. 363–373, Feb. 1995.

139. M. K. Amirhosseini, "Determination of Capacitance and Conductance Matrices of Lossy Shielded Coupled Microstrip Transmission Lines," *Progress In Electromagnetics Research*, PIER 50, pp. 267–278, 2005.

140. *http://www.ansys.com/*

141. S. M. Musa and M. N. O. Sadiku, "Modeling and Simulation of Shielded Microstrip Lines," *The Technology Interface*, Fall 2007.

142. J. A. Edminister, "Theory and Problems of Electromagnetics," Schaum's Outline Series, McGraw-Hill, 1979.

143. G. R. Buchanan, "Finite Element Analysis," Schaum's outline, McGraw-Hill, 1995.

APPENDIX A

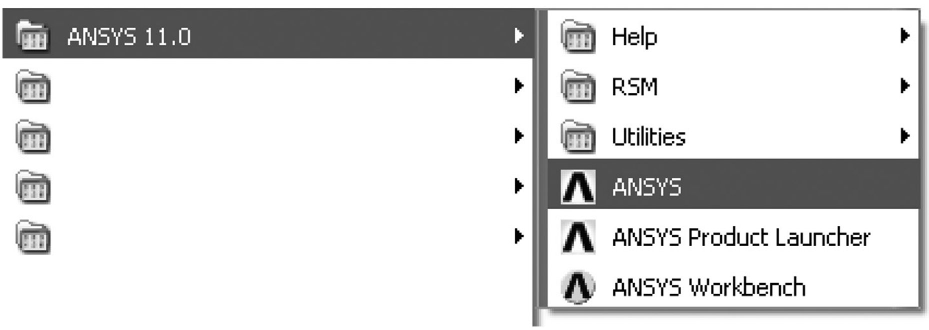
ANSYS

A.1 ANSYS (CLASSIC OR TRADITIONAL)

ANSYS is a finite element modeling package for numerically solving variety of engineering problems such as structural, thermal, fluid, static and dynamic, linear and nonlinear, acoustic, electromagnetics, multiphysics, etc.

ANSYS has two methods. The graphical user interface (GUI) and the command files. The focus in this section, as in the book only on the ANSYS GUI environment (traditional or classic ANSYS), and in the next section, we focus on the ANSYS workbench environment.

To start ANSYS, double-click on the **ANSYS** icon, or you can use start > Programs > **ANSYS 11.0 > ANSYS**.



Then the ANSYS main window will be shown with the ANSYS output window. The ANSYS GUI environment has two windows, the main window and the output window.

The ANSYS main window has the following components:

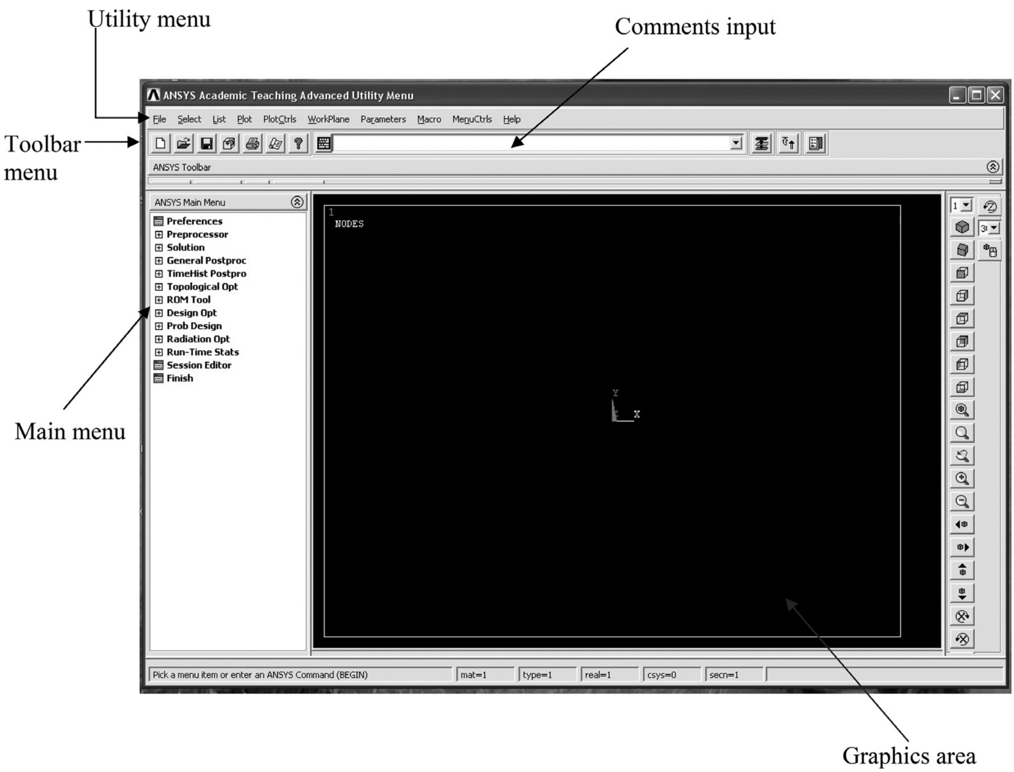
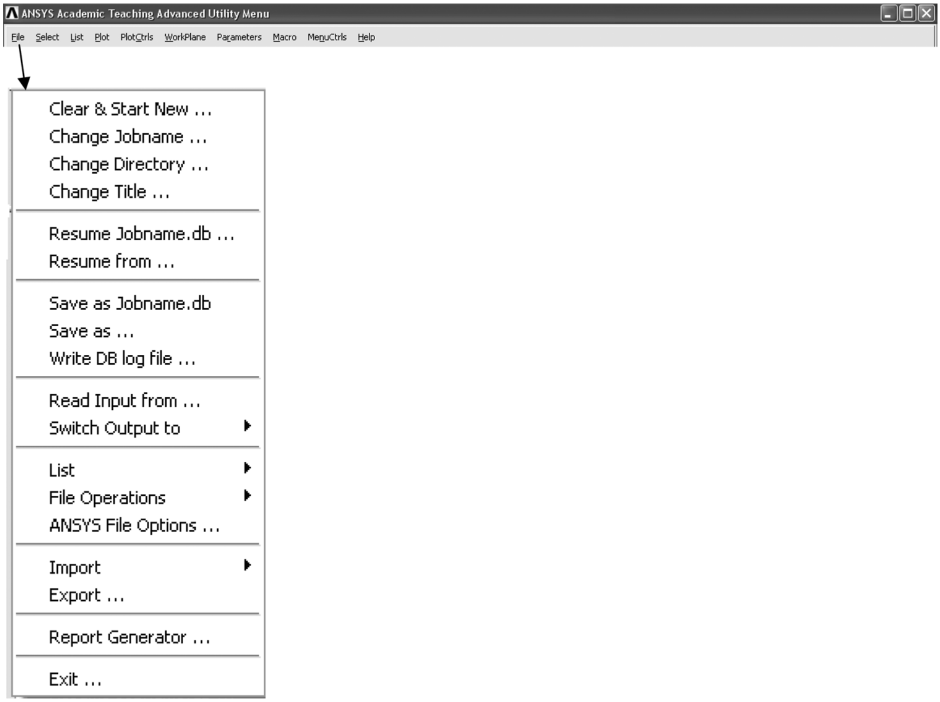


FIGURE A.1 ANSYS GUI main window.

1. *Utility menu*: contains functions that are available through out the ANSYS sessions.

(a) File menu:

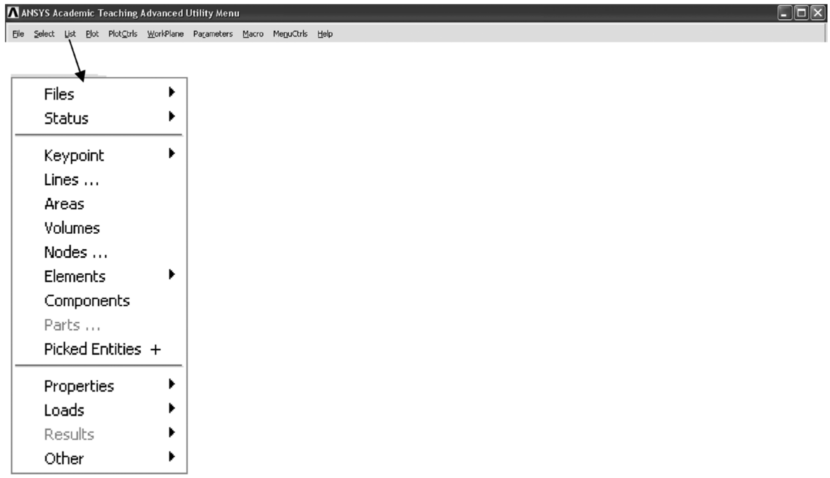


File menu contains file and database-related functions such as options to clear the database, change, resume, and save the current model.

(b) Select menu

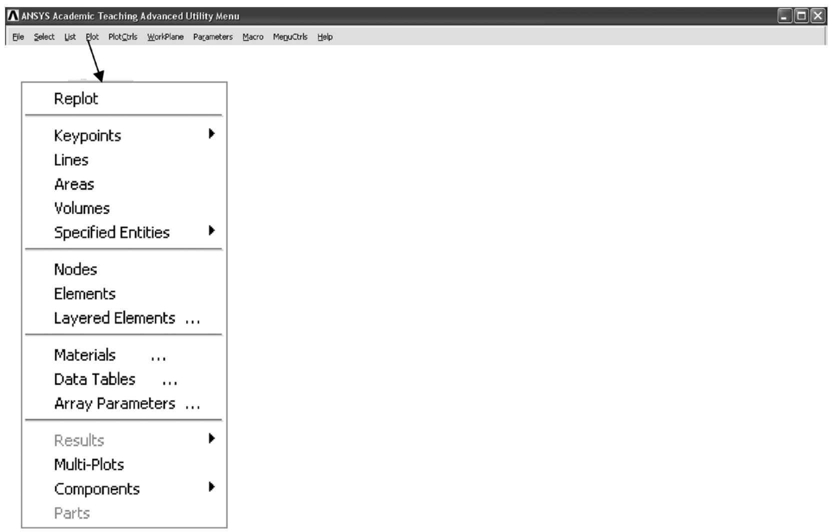
Select menu provides functions that allow the user to select subsets of entities and create components.

(c) List menu



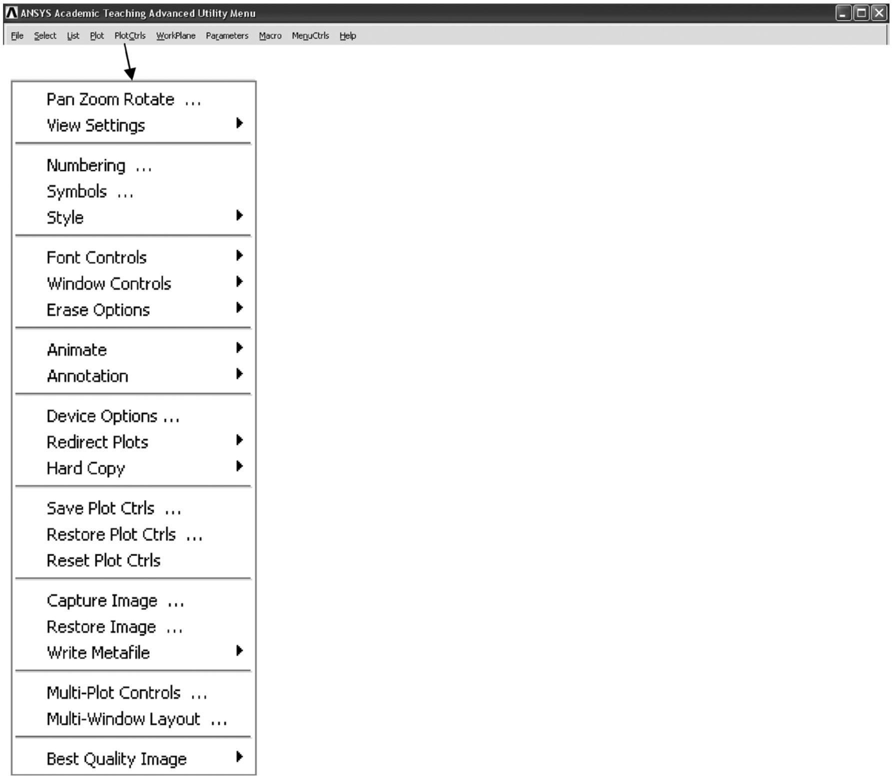
List menu allows the user to view any data item stored in the ANSYS database, view log and error files, obtain listing of geometries entities and their components, elements and their properties, nodes, and boundary conditions and loads. You can obtain information about different areas of the program and list the contents of files residing in the system.

(d) Plot menu



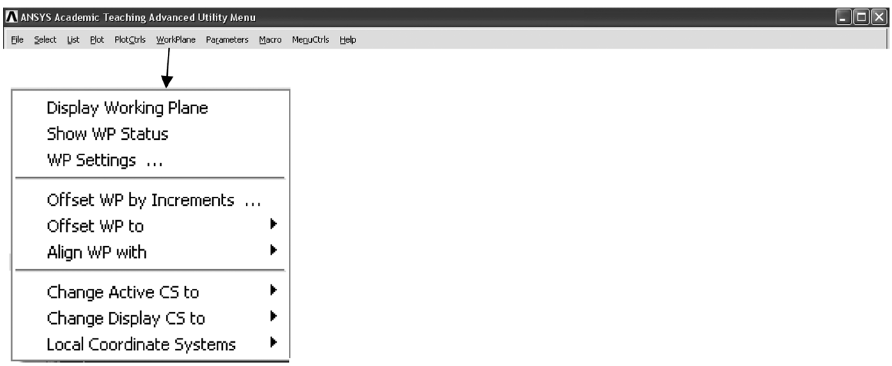
Plot menu allows you to plot various components of the model such as keypoints, lines, areas, volumes, nodes, elements, and other data can be plotted.

(e) **PlotCtrls** menu



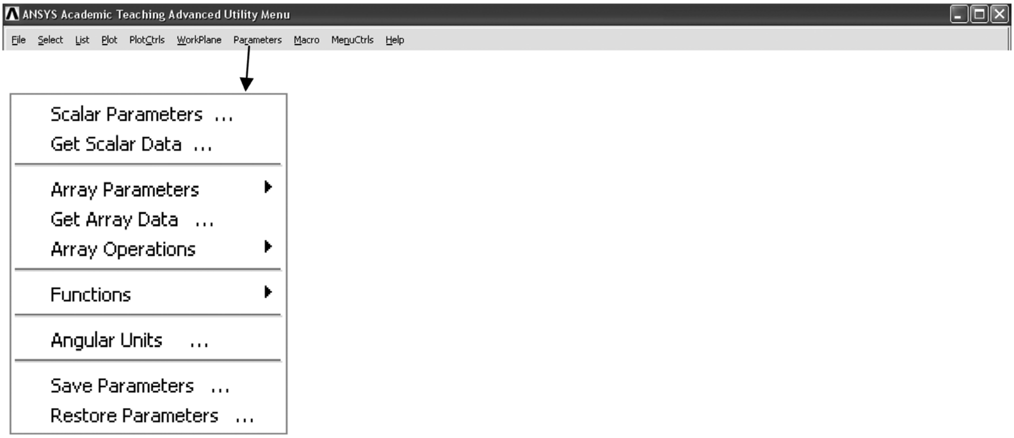
PlotCtrls menu includes functions to control the view and pan/zoom/rotate the model, select numbering options, change styles, and make hard copies of the plots.

(f) **WorkPlane** menu



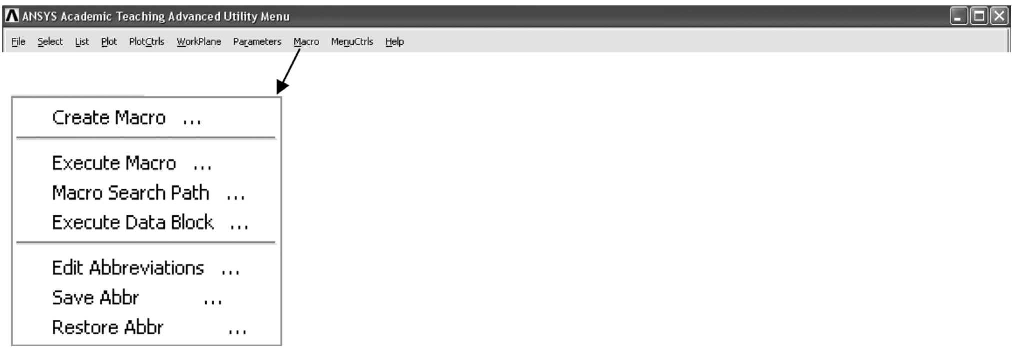
WorkPlane menu allows the user to toggle the working plane on or off and to move and rotate, and you can create, delete, and switch coordinate systems.

(g) Parameters menu



Parameters menu allows the user to define, edit, and delete scalar and array parameters.

(h) Macro menu



Macro menu allows the user to create and execute macros and execute data blocks. Also, the user can create, edit, and delete abbreviations.

(i) **MenuCtrls** menu

MenuCtrls menu allows the user to create, edit, and delete abbreviations on the ANSYS toolbar and modify the colors and fonts in the GUI display.

(j) **Help** menu

Help menu allows the user to find all the manuals and tutorials available on ANSYS.

2. *Comments input*: shows program prompt messages and allows the user to type in commands directly.
3. *Toolbar menu*: contains graphics buttons that execute frequently used ANSYS commands, that is, collects commands that are frequently used.
4. *Main menu*: contains primary ANSYS functions.
5. *Graphics area*: displays geometrics, finite elements, and simulations.

The ANSYS output window dynamically provides important information during the preprocessor, solution, and postprocessor. The warnings in the output window should be considered to prevent future errors. The output window is usually positioned behind the main window and can be put to the front, if necessary.

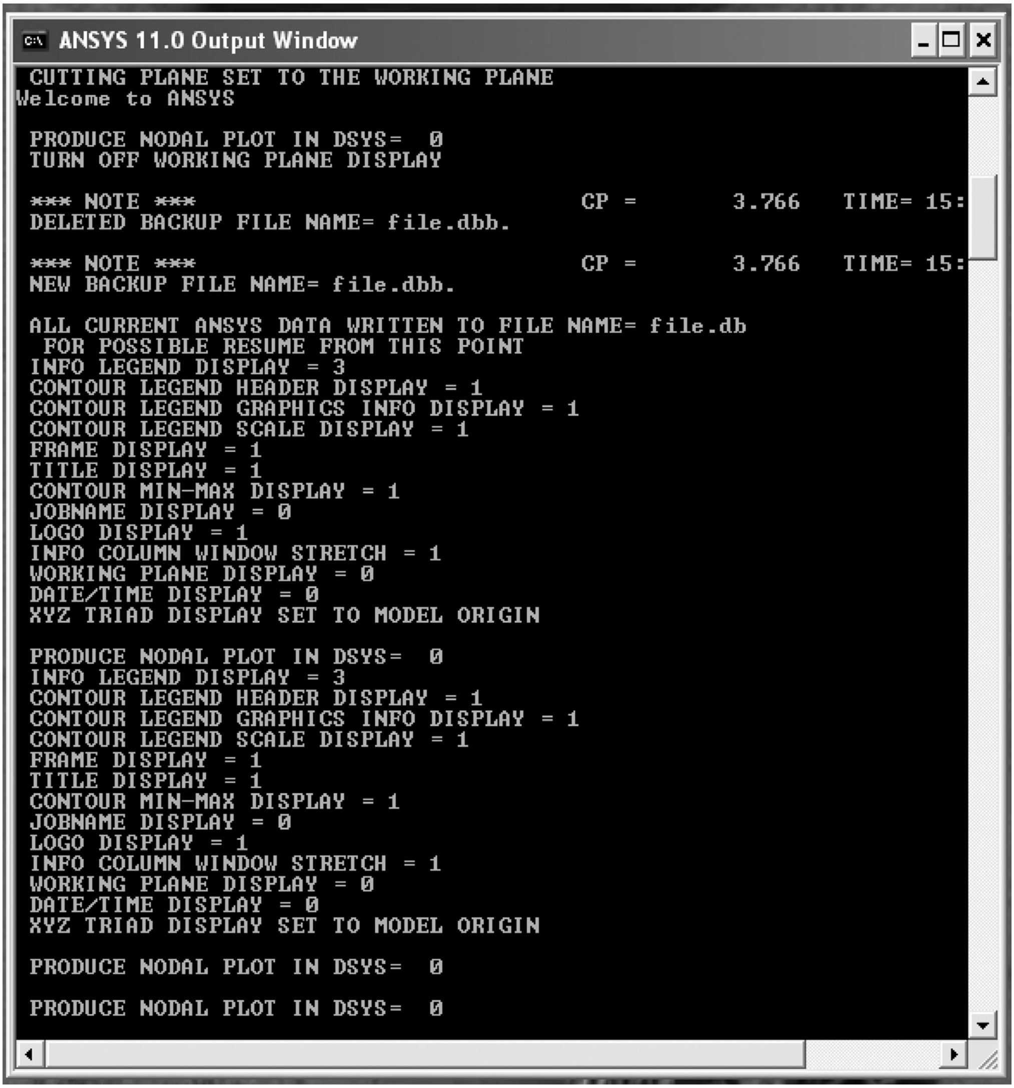


FIGURE A.2 ANSYS GUI output window.

To save the model, select from **ANSYS Utility Menu**, **File** > **Save as Jobname.db**.

Or you can save your model by selecting **File > Save as**, as shown below.

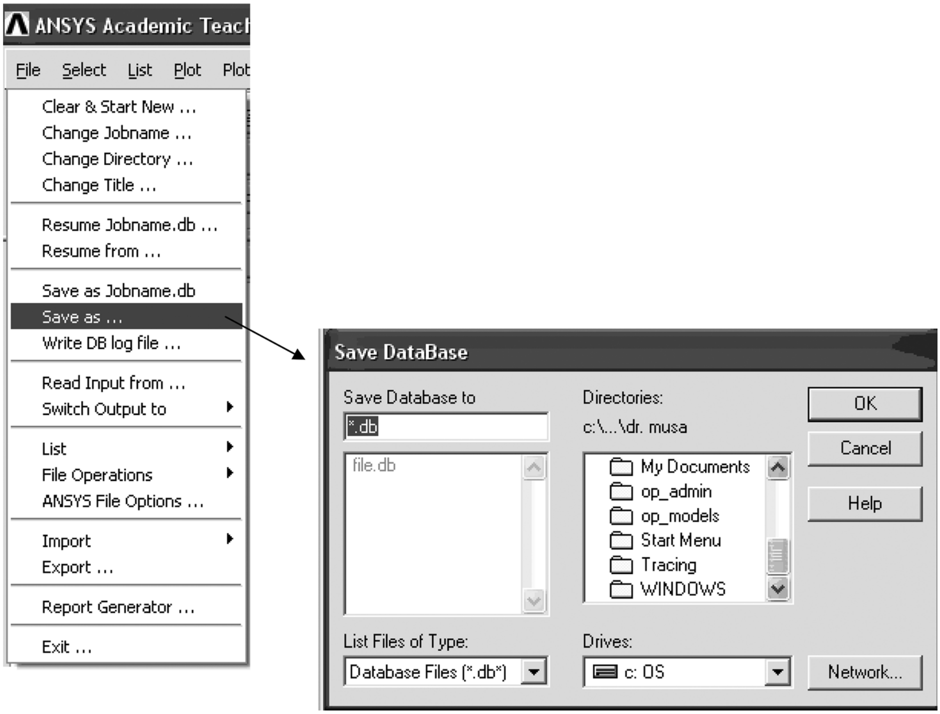


FIGURE A.3 Save database of the model.

To start up ANSYS, recall and continue previous job by starting **ANSYS** and then select **Utility Menu: File > Resume from** and click on the model from the list that appears.

To quickly save an image of the entire screen or the current Graphics window, go to **ANSYS Utility Menu** > **PlotCtrls** > **Hard copy.** There are two options to choose: **To Printer** and **To File**.

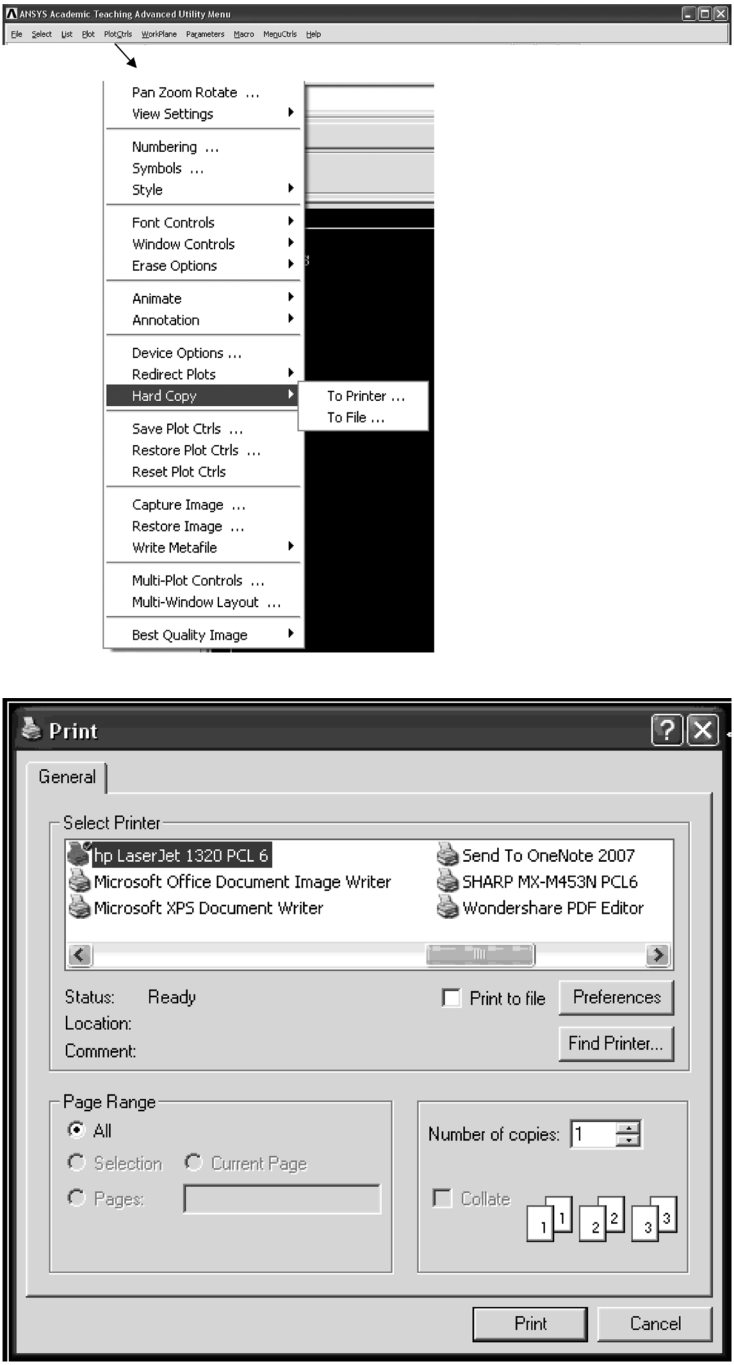


FIGURE A.4 Print to printer.

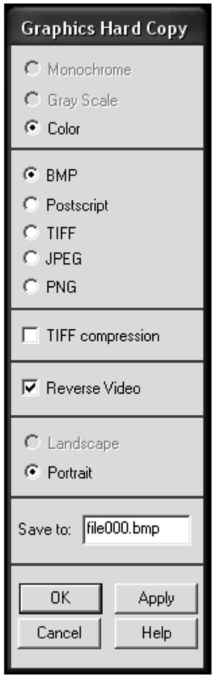


FIGURE A.5 Print to file.

To finish ANSYS, choose **File** > **Exit** command. Then **Exit from ANSYS** window will show up as shown in Figure A.6. Choose a proper option and click **OK** to finish the ANSYS program.

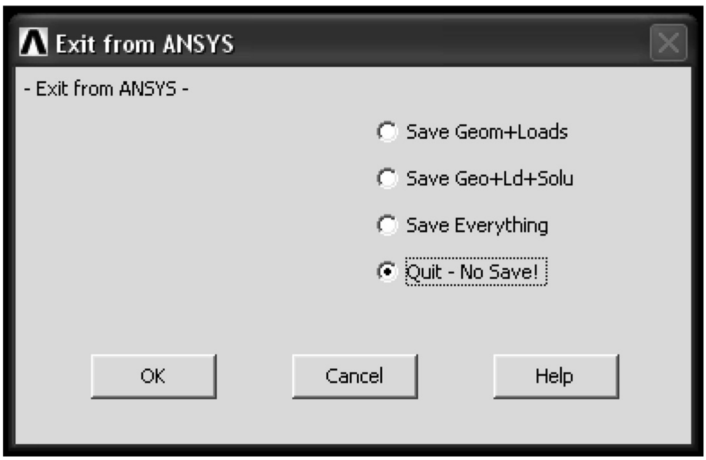


FIGURE A.6 Exit from ANSYS.

A.2 ANSYS WORKBENCH

The ANSYS workbench provides unified data sharing and project file management across the range of ANSYS products. Workbench is not an application on its own. It is a group of technologies for developing simulating tools. This has been created by ANSYS incorporation to help developers develop robust tools to meet the needs of designers and analysts. It provides a common interface for the user to access the wealth of technology that ANSYS has with it.

Simulation data is displayed in a hierarchical tree that separates out the various groups of inputs and outputs. Workbench makes use of a project file storage method to assist with the organization of data, and each can contain all types of information. Workbench stores the model information at the highest level, and this contains:

1. geometry that your mesh is built on,
2. contacts (which define how parts interact),
3. mesh and
4. environment.

The mesh is the core element-based representation of the parts, on which all FEA/CFD tasks are based. Workbench allows you to intelligently refine the mesh.

The Workbench user interface: Tabs-based user interface, that is familiar to most users, is used in Workbench. We start with the Project Page tab, and this provides us access to the tools we need to read geometry in, set up the boundary conditions, create results visualization assets, and report on results.

Geometry definition: Data pertaining to geometry can be read from the majority of leading software, including Pro/Engineer, Solid Works, NX/ Unigraphics, Catia, etc. The system reads the data into the Workbench user interface and preserves part names and material definitions, if any.

Design Modeler: Design Modeler provides us with many tools that allow us to take data from a variety of sources, integrate it into a single pack, and use it.

Finite Element Modeler: It allows you to read meshed, element-based data from a wide variety of sources. It performs checks for integrity, extracts the information, and places it into the appropriate folders of the simulation tree.

Materials definition: ANSYS Workbench is supplied with a range of standard materials. Materials (if present) can be extracted from the CAD file or may be adapted or enter new materials.

Reporting: Reports can be created based on a wizard-style workflow. Various inputs (title, description, etc.) may be defined, and the system handles the rest of the work. Workbench allows you to create diagrams at any point, so the graphics window and a caption is stored and then added to the report.

Design optimization-Design Xplorer: Workbench has inherent links with the CAD system, and bi-directionality can be used to our advantage to carry out optimization to the maximum extent.

Mechanisms: Workbench also makes it possible to carry out in-depth studies of mechanisms intelligently. It allows us to model both rigid and flexible bodies in one environment.

Lastly, it can be said that ANSYS provides a single unified environment where the user has access to good functionality within the simulation arena. It helps us to build high-quality products that cost less and serve better their intended purpose.

The method of solving problems using workbench is illustrated using a simple example.

EXAMPLE A.1

For the cantilever beam shown in Figure A.7, determine the total deflection, maximum stress, reaction force, and reaction moment. Take Young's modulus E = 210 GPa and Poisson ratio v = 0.3. Force P = 5000 N. Assume plane stress condition.

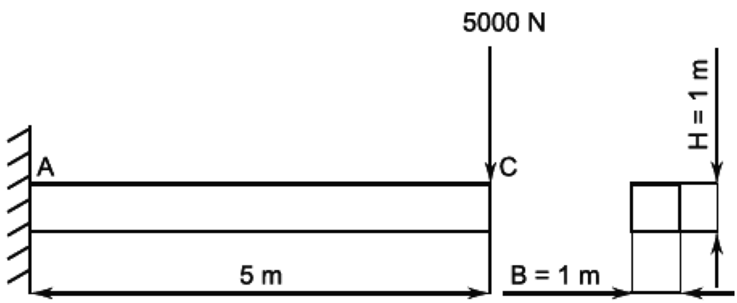


FIGURE A.7 The cantilever beam for Example A.1

Solution:

(I) Software method

Procedure for solving the problems using ANSYS® 11.0 academic teaching software using ANSYS Workbench.

Step 1. **Selecting Analysis System and Specifying Material Properties**

Start ANSYS Workbench

Start > Programs > ANSYS 11.0 > Workbench

In **Toolbox Customization, click Static Structural (ANSYS)**

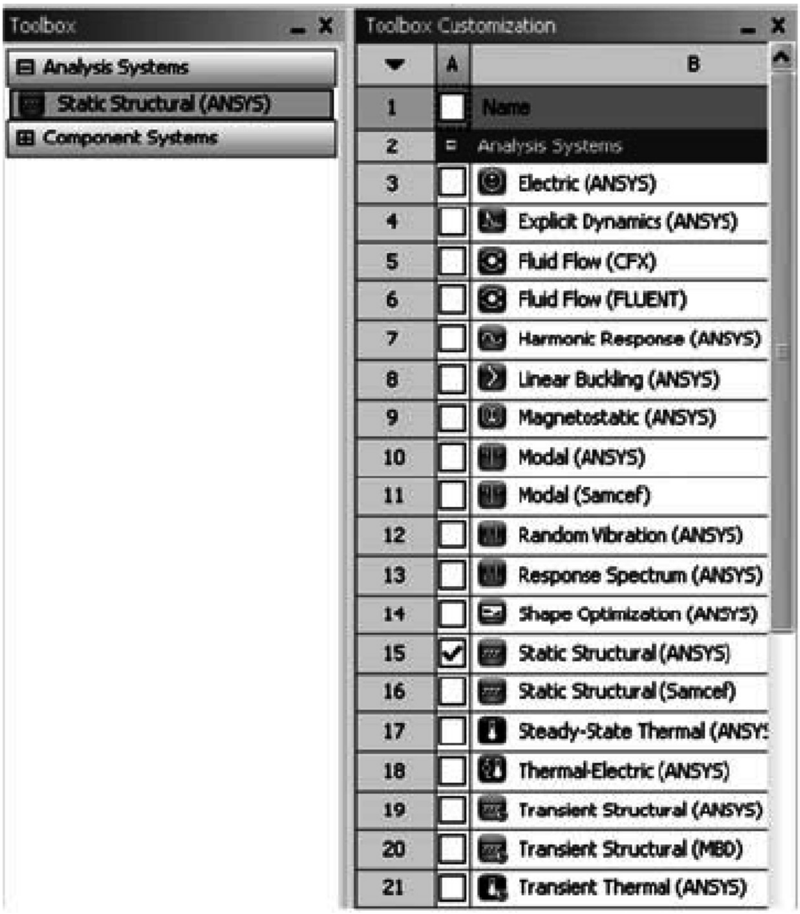


FIGURE A.8 Selecting type of analysis.

In **Toolbox**, left-click (and hold) on **Static Structural (ANSYS)** and drag the icon to the empty space in the **Project Schematic**. In **Project Schematic** space, right-click on **Static Structural (ANSYS)** and **Rename** the project to **Cantilever_beam**.

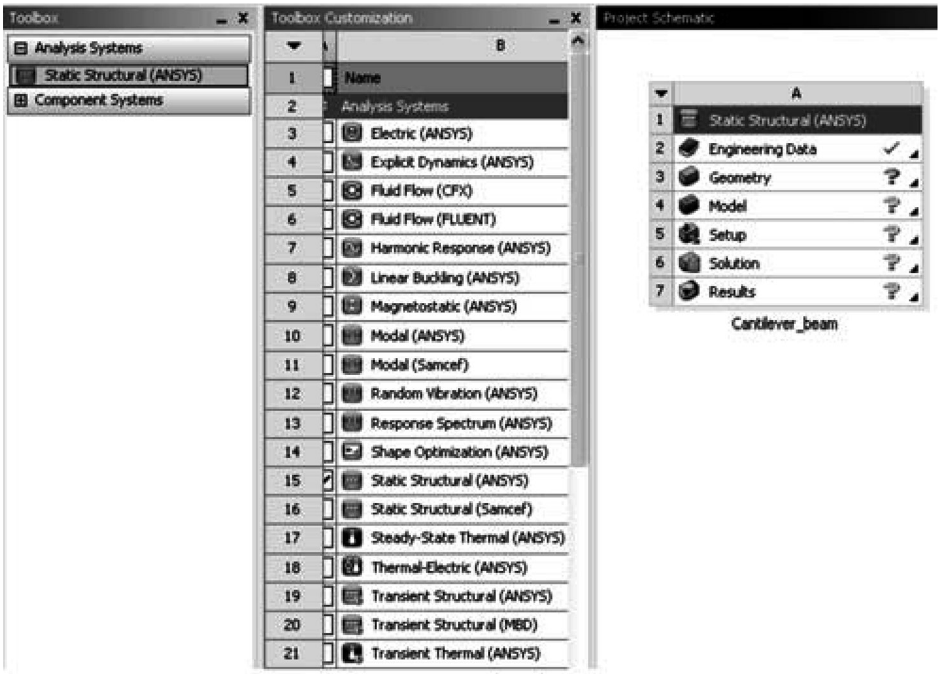


FIGURE A.9 Creating cantilever beam cell.

Specify Material Properties

In the cantilever beam cell, double-click on **Engineering Data**. A new page will be opened. The default material given is **Structural Steel**. This will be seen in the **Outline of Schematic**.

EXAMPLE A2: Engineering Data.

In the **Properties of Outline Row 3: Structural Steel** window, double click on **Isotropic Elasticity** and change the **Young's Modulus** value. To change the value, double click on the numerical value and enter E = 2.1e11 Pa and **Poisson ratio** = 0.3.

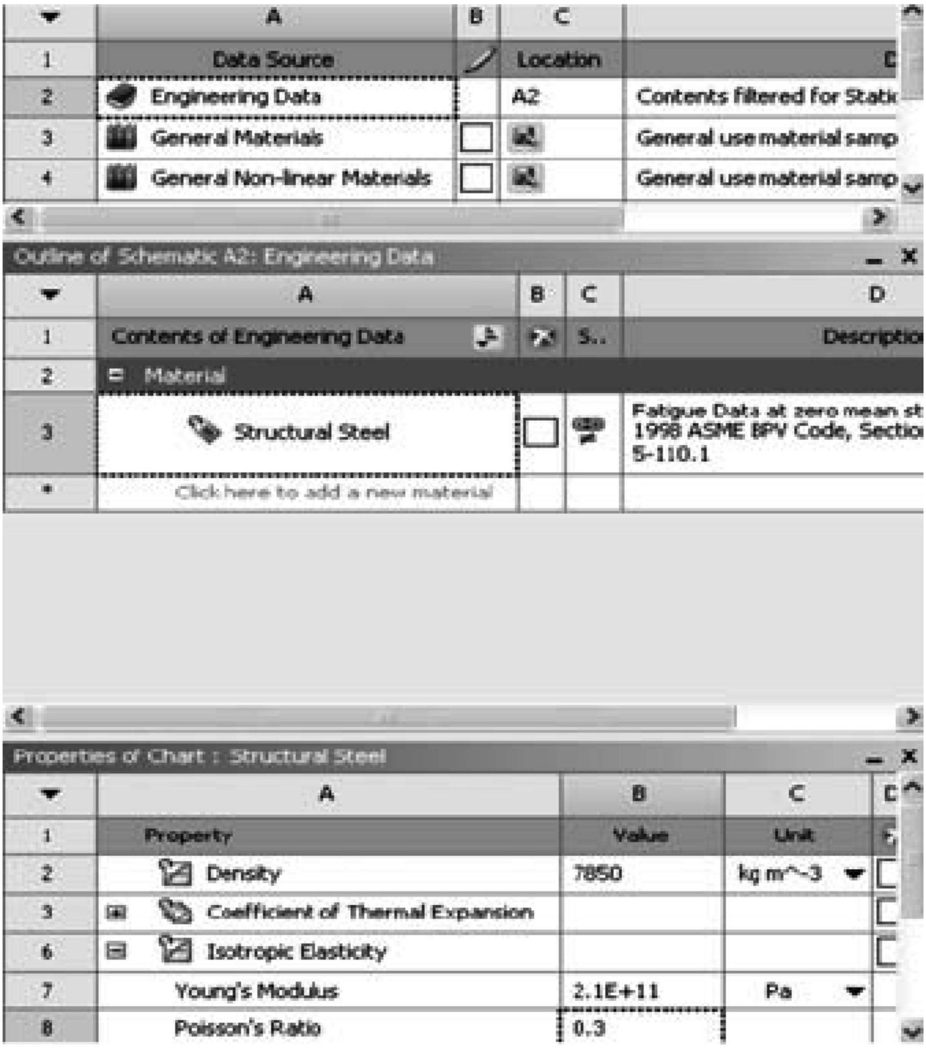


FIGURE A.10 Specifying material properties.

Next, click on Return to Project to return to Workbench Project Schematic window.

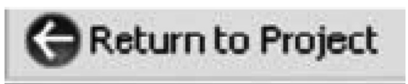


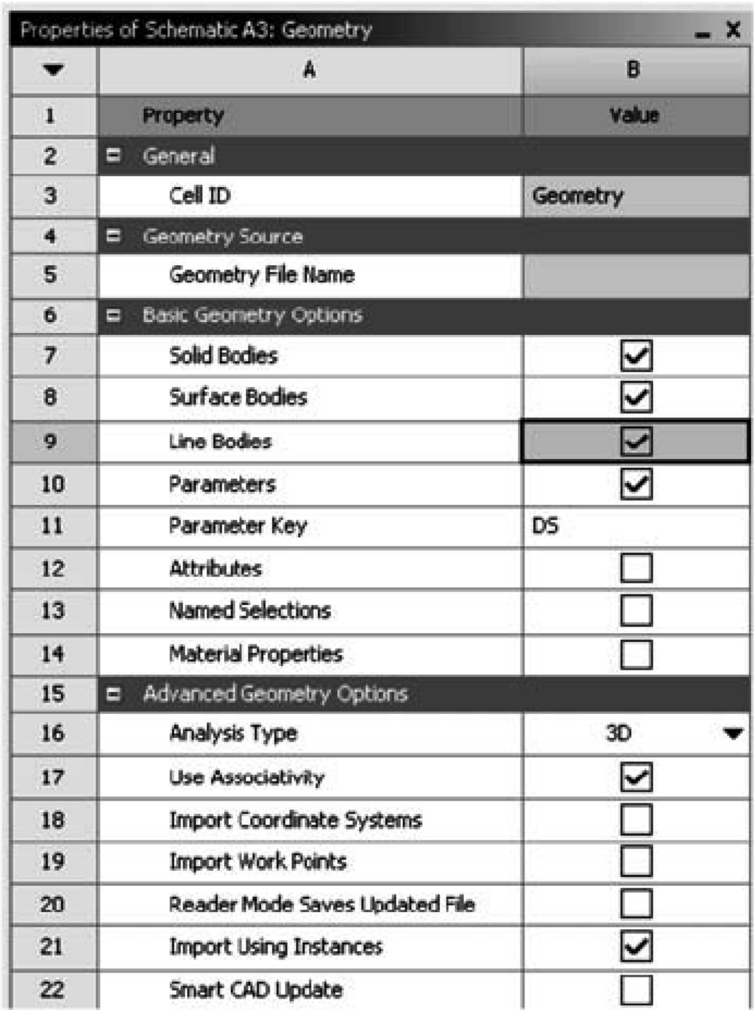
FIGURE A.11 Return to Workbench.

Step 2. **Geometry Creation**

In the **Project Schematic contilever_beam cell,** right-click on **Geometry** and select **Properties**.

A window named **Properties of Schematic A3: Geometry** will be seen on the right-hand side of the Workbench window.

In that window, under **Basic Geometry Options**, select **Line bodies**.


Properties of Schematic A3: Geometry

	A	B
1	Property	Value
2	General	
3	Cell ID	Geometry
4	Geometry Source	
5	Geometry File Name	
6	Basic Geometry Options	
7	Solid Bodies	☑
8	Surface Bodies	☑
9	Line Bodies	☑
10	Parameters	☑
11	Parameter Key	DS
12	Attributes	☐
13	Named Selections	☐
14	Material Properties	☐
15	Advanced Geometry Options	
16	Analysis Type	3D
17	Use Associativity	☑
18	Import Coordinate Systems	☐
19	Import Work Points	☐
20	Reader Mode Saves Updated File	☐
21	Import Using Instances	☑
22	Smart CAD Update	☐

FIGURE A.12 Selecting line bodies.

Then in the **Project Schematic** window, under the cell, double left click on **Geometry** to start preparing geometry. At this point, a new window **ANSYS Design Modeler** will be opened. Select **meter** unit as desired length unit and click **OK**.

Creating a Sketch

Use **XY Plane** for creation of sketch. On the left-hand side in **Tree Outline,** click on **XY Plane** and then click on **Sketching**, next to **Modeling** tab.

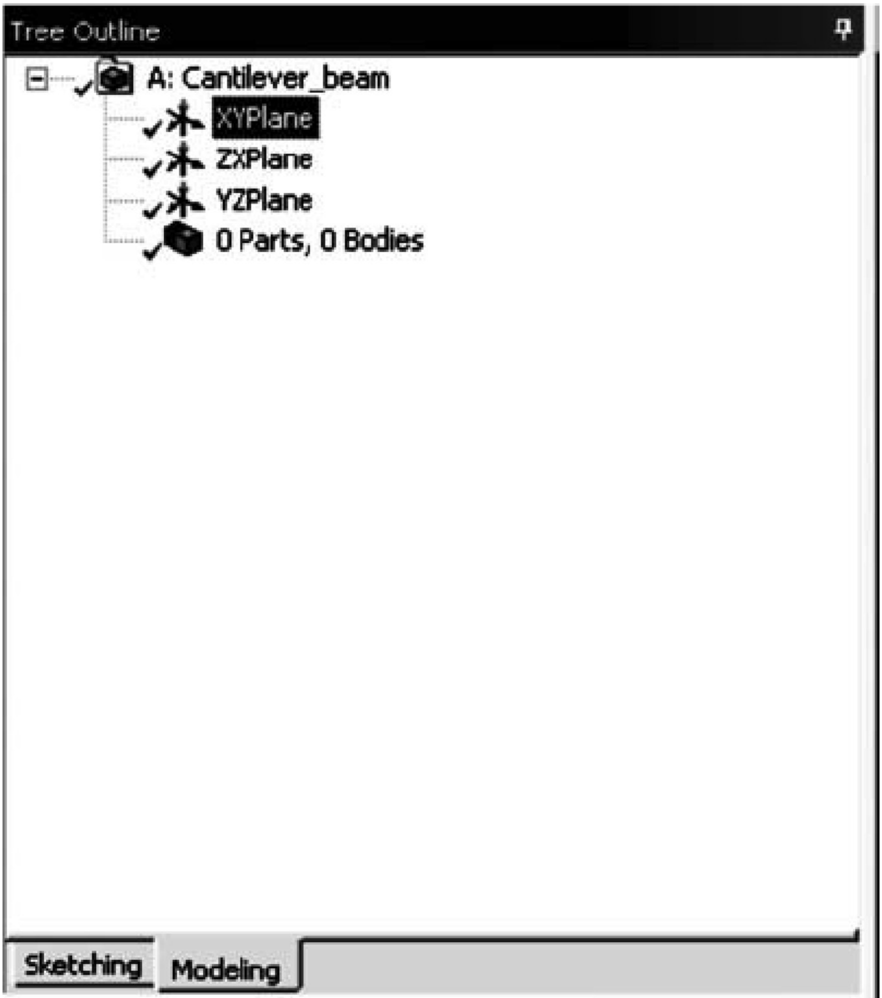


FIGURE A.13 Selecting XY plane for sketching.

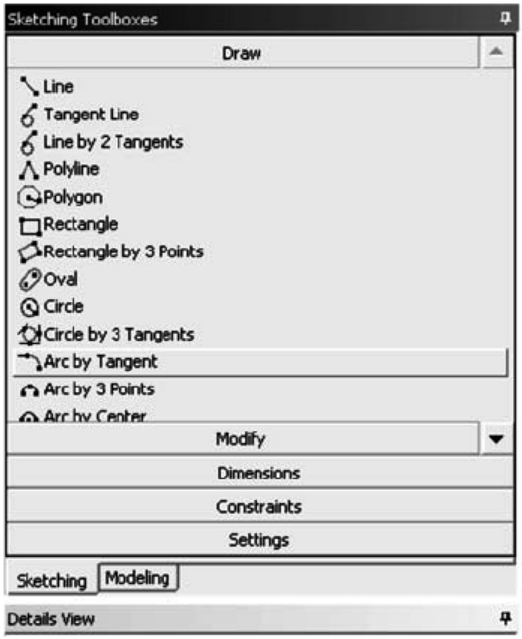


FIGURE A.14 Sketching toolbox.

Note: In sketching, **undo** features can be used.

On the right-hand side, there is a **Graphics** window. At the lower right-hand corner of the **Graphics** window, click on **+Z** axis. Then **Graphics** window will appear as shown below.

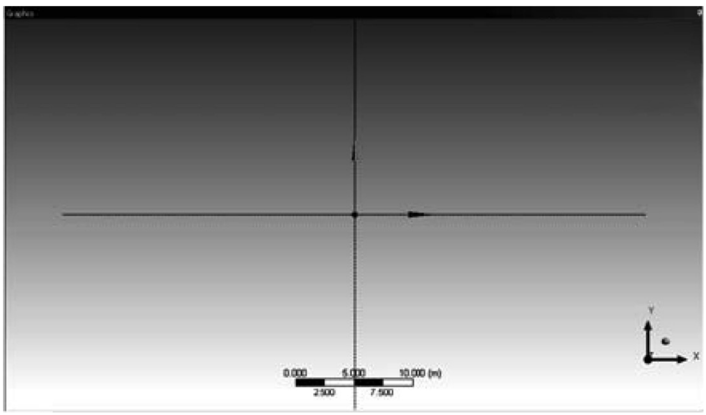


FIGURE A.15 Graphics window.

In the **Sketching toolbox,** select **Line** and, in the **Graphics** window, create one rough line starting from the origin in the positive XY-direction. (The letter P should be seen at the origin before creating the Line. The letter P indicates geometry is constrained at the origin.)

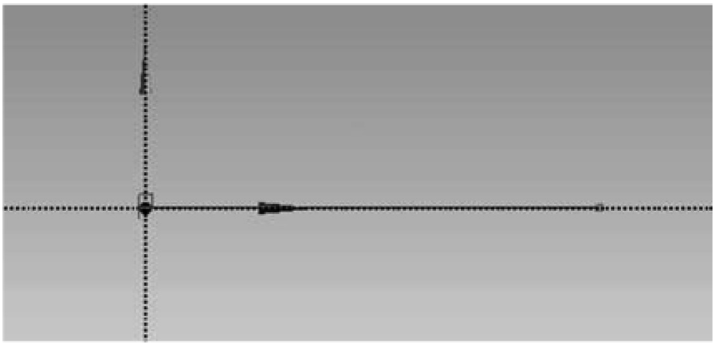


FIGURE A.16 Creating a line.

Dimensions

Under the **Sketching Toolbox**, select the **Dimensions** tab. (Use the default dimensioning tools.)

Then left click on the line in the Graphics window and drag the dimension line. The dimension line shows **H1.**

On the lower left-hand corner, under **Details View,** change **H1** to **5**. This **5** is the length of the line drawn. Now sketching is done.

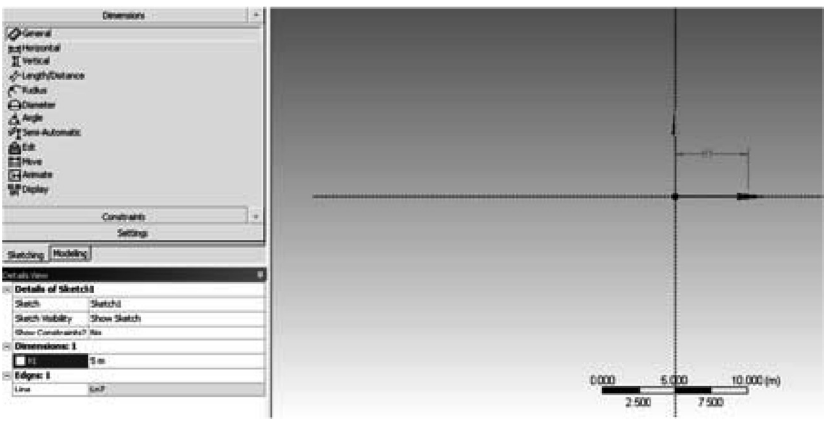


FIGURE A.17 Dimensioning the line and entering the length value.

Surface Creation

On top menu of the **Design Modeler** window, click on

Concept > Lines from Sketches.

This will create a new line, **Line1**. In **Tree Outline,** double-click on **XY Plane,** then select **Sketch 1**, and under **Details View**, in **Base Objects,** click **Apply.** Finally, on the top, click **Generate** to generate the surface.

Create Cross-Section

Concept > Cross-Section > Rectangular

Under **Details View**, input B = 1 m and H = 1.

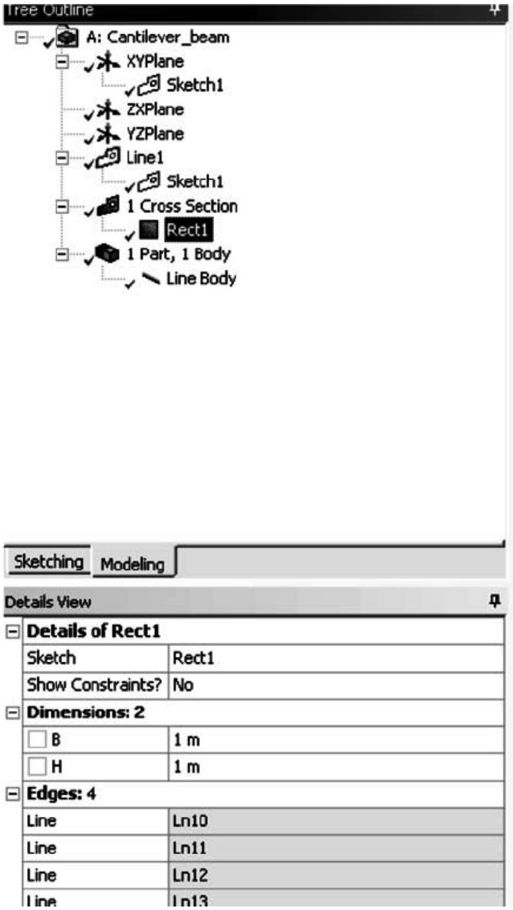


FIGURE A.18 Entering the breadth and height values of the beam.

Now under **Tree Outline > 1 Part, 1 Body > Line Body,** attach **Rect1** to **Cross-Section** under **Details View** as shown below.

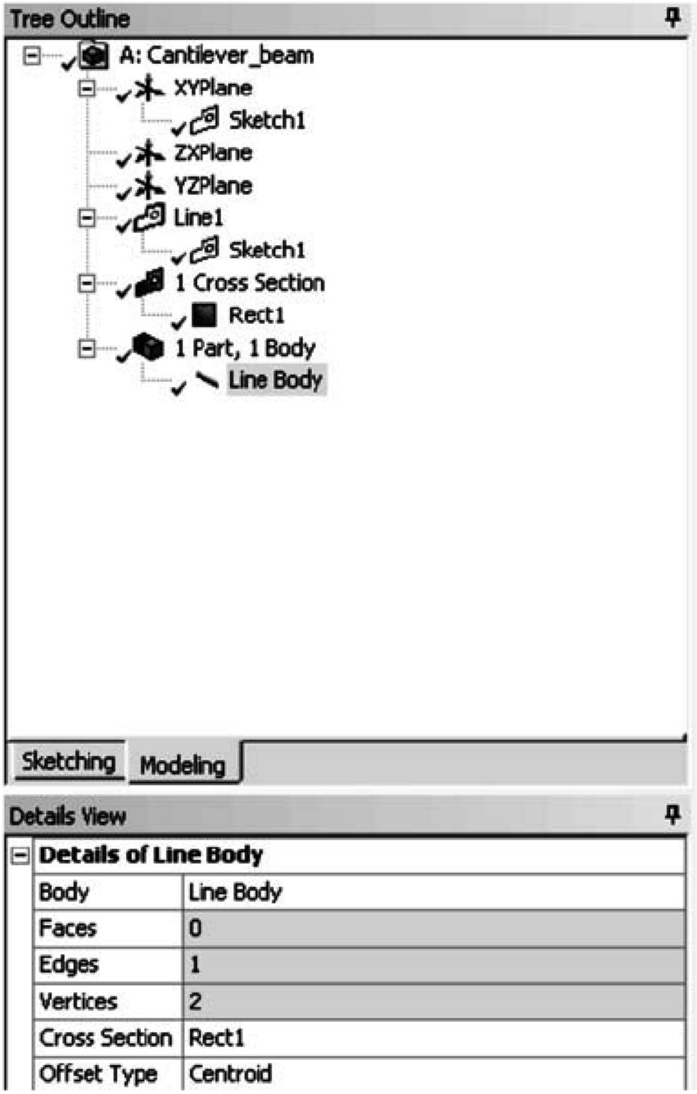


FIGURE A.19 Attaching a rectangular cross-section.

Now geometry is done. Close the **Design Modeler** and go back to **Workbench**.

Step 3. **Meshing**

Save the work in the **Workbench** window.

In the **Cantilever_beam cell**, right click on **Model** and click **Edit**. A new **ANSYS Model** window will open.

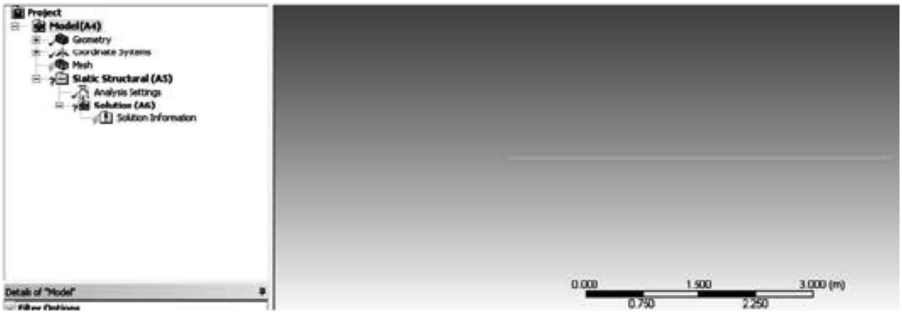


FIGURE A.20 ANSYS model window for meshing, for applying boundary conditions and for the solution.

In **Outline** heading, right click on **Mesh** and click **Generate Mesh**. Then Meshed model will appear in Graphics window.

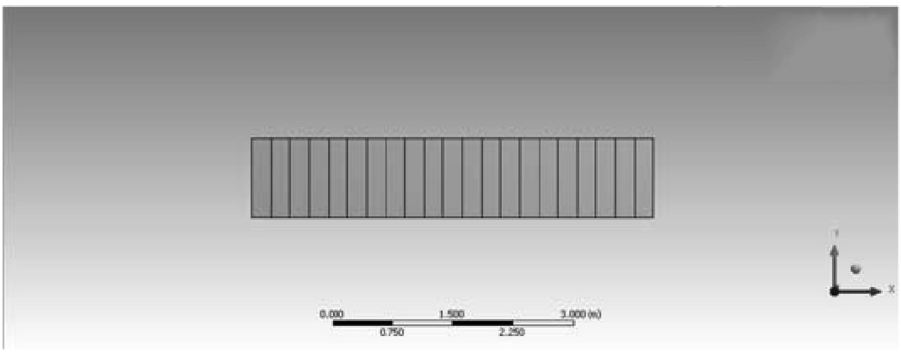


FIGURE A.21 Meshed model.

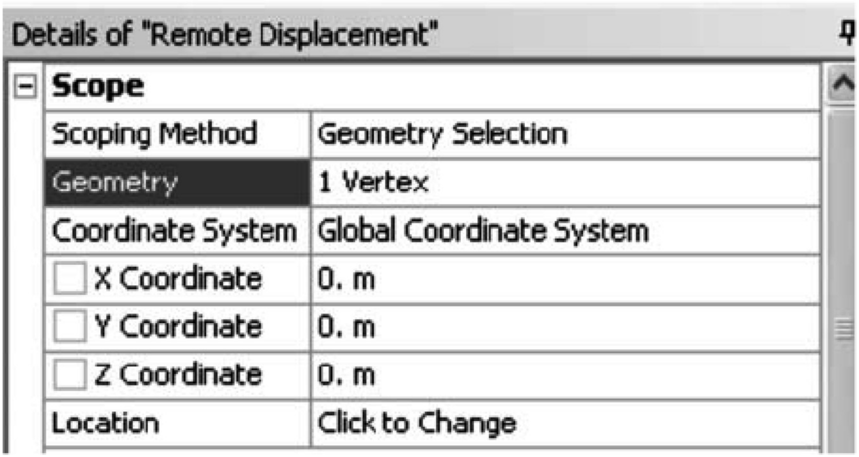
Step 4. **Boundary Conditions**

Start the setting up of boundary condition at A.

In **Outline,** right click **Static Structural (A5) > Insert** then select **Remote Displacement**.

In **Graphics** window right click and change **Cursor Mode** to **Vertex**.

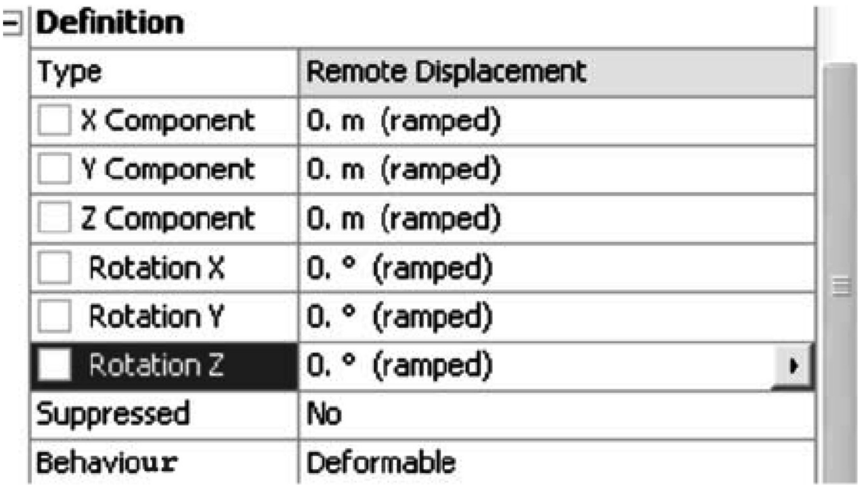
Select point A in the **Graphics** window and click **Apply** next to **Geometry** under Details of **"Remote Displacement"**


Details of "Remote Displacement"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change

FIGURE A.22 Selecting point A.

Now under the heading of **Details of "Remote Displacement"** and under **Definition** heading change **UX, UY, UZ, ROTX, ROTY, and ROTZ to zero**.



Definition	
Type	Remote Displacement
X Component	0. m (ramped)
Y Component	0. m (ramped)
Z Component	0. m (ramped)
Rotation X	0. ° (ramped)
Rotation Y	0. ° (ramped)
Rotation Z	0. ° (ramped)
Suppressed	No
Behaviour	Deformable

FIGURE A.23 Applying displacement boundary conditions.

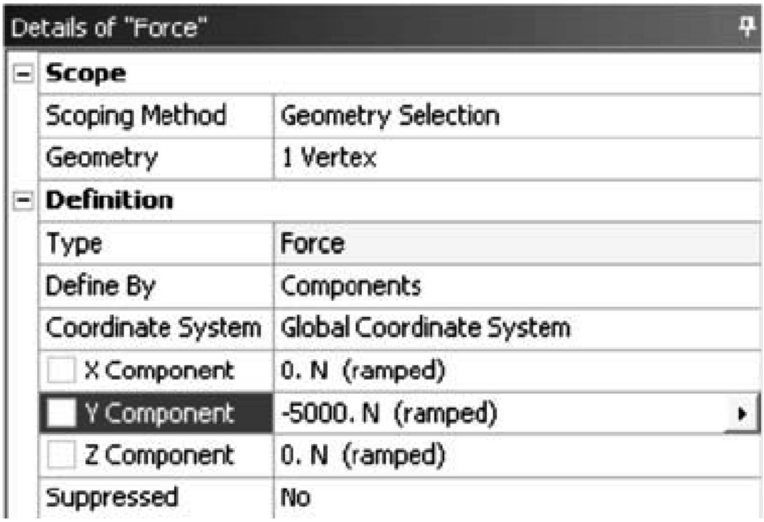
At point C, force component needs to be applied.

In **Graphics** window, select point **C**.

Then **Outline > Static Structural (A5) > Insert > Force.**

Under **Details of "Force",** in **Definition** heading, next to **Define By,** change **Vector** to **Components**.

Enter–**5000** for **Y component**.



Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. N (ramped)
Y Component	-5000. N (ramped)
Z Component	0. N (ramped)
Suppressed	No

FIGURE A.24 Applying force.

Click on **Static Structural (A5)** to view the model with boundary conditions in **Graphics** window shown below.

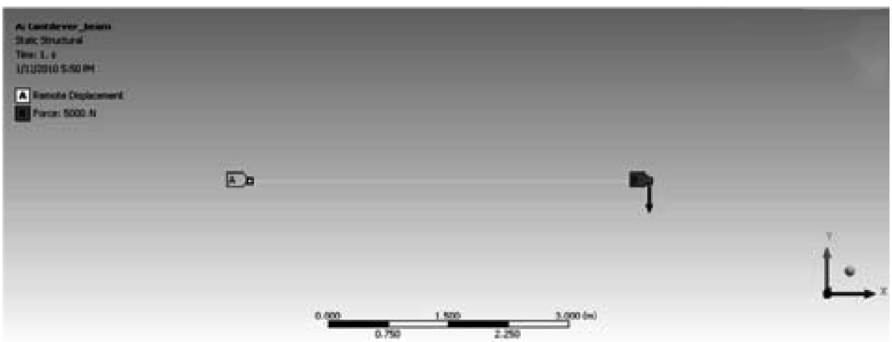


FIGURE A.25 Model with boundary conditions.

Step 5. **Solution**

For finding deflection, under **Outline** heading, right-click **Solution (A6), Insert > Deformation > Total.**

For finding Stresses in beam, **Outline > Solution (A6) > Insert > Beam Tool > Beam Tool.**

For finding reaction force, **Outline > Solution (A6) > Insert > Probe > Force Reaction.**

Select point A and Under **Details of "Force Reaction"**, in Definition heading next to **Boundary Condition,** select **Remote Displacement**.

For finding reaction moment, **Outline > Solution (A6) > Insert > Probe > Moment Reaction.**

Select point A and Under **Details of "Moment Reaction"**, in **Definition** heading next to **Boundary Condition,** select **Remote Displacement**.

Then on top, click Solve button.

Step 6. **Results**

Under **Outline** and under **Solution (A6),** click on **Total Deformation**. The result is shown below.

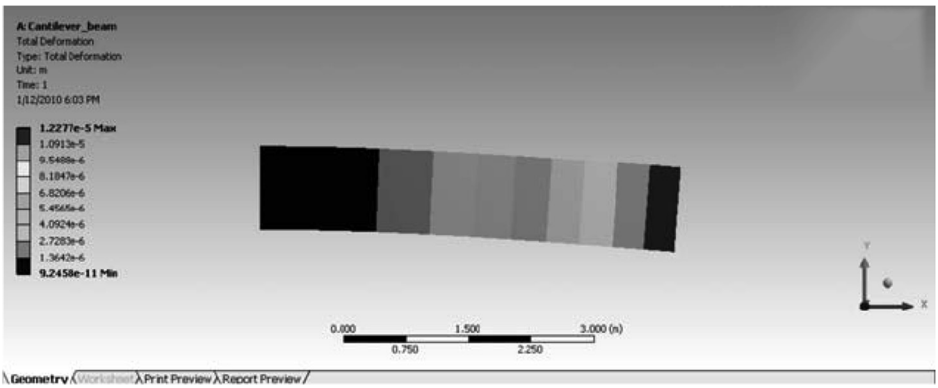


FIGURE A.26 Deflection pattern for a cantilever beam.

Details of "Total Deformation"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
Results	
Minimum	9.2458e-011 m
Maximum	1.2277e-005 m
Information	

FIGURE A.27 Deflection values for a cantilever beam.

Then double-click on **Beam Tool** and click on **Maximum combined stress**. This result is shown below.

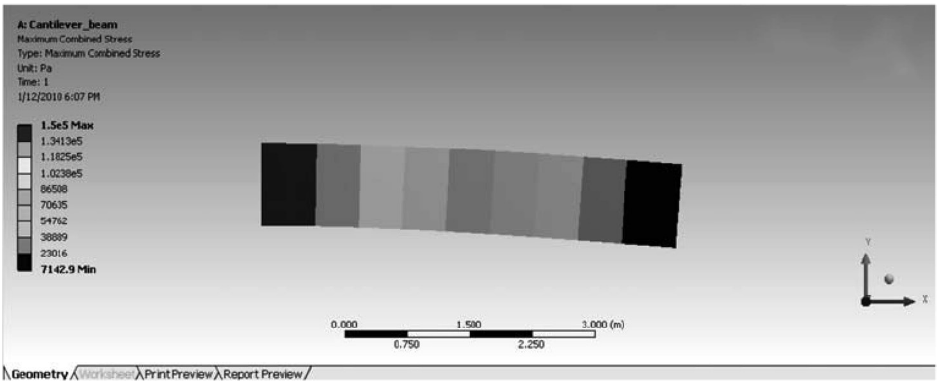


FIGURE A.28 Maximum combined stress on a cantilever beam.

Details of "Maximum Combined Stress"

Definition	
Type	Maximum Combined Stress
By	Time
Display Time	Last
Calculate Time History	Yes
Use Average	Yes
Identifier	
Results	
Minimum	7142.9 Pa
Maximum	1.5e+005 Pa

FIGURE A.29 Maximum combined stress values.

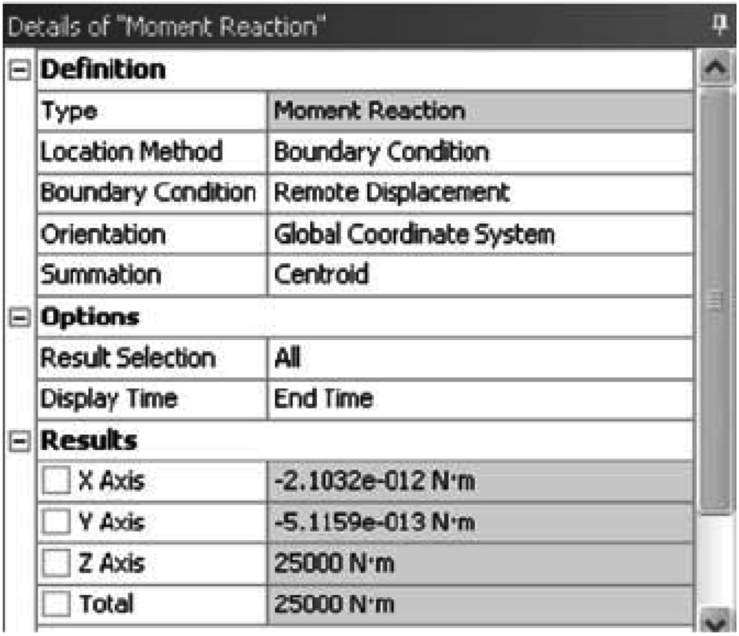
Then click on **Force Reaction**, and this result is as shown below.

Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Remote Displacement
Orientation	Global Coordinate System
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	8.5265e-014 N
Y Axis	5000. N
Z Axis	-7.9581e-013 N
Total	5000. N

FIGURE A.30 Values of reaction force.

Then click on **Moment Reaction**; this result is shown below.



Details of "Moment Reaction"

Definition	
Type	Moment Reaction
Location Method	Boundary Condition
Boundary Condition	Remote Displacement
Orientation	Global Coordinate System
Summation	Centroid
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	-2.1032e-012 N·m
Y Axis	-5.1159e-013 N·m
Z Axis	25000 N·m
Total	25000 N·m

FIGURE A.31 Values of reaction moment.

(II) Analytical method (Refer Figure A.1).

Maximum bending stress occurs at the support.

Bending moment at support $M = P \times L = 5 \times 10^3 \times 5 = 25000$ N-m

Section modulus $= s = \dfrac{B \times H^2}{6} = \dfrac{1 \times 1^2}{6} = 0.1667\ \text{m}^3$

Maximum stress $= \sigma_{\max} = \dfrac{M}{s} = \dfrac{25000}{0.1667} = 150000\ \text{Pa} = 0.15\ \text{MPa}$

Moment of inertia $= I = \dfrac{B \times H^3}{12} = \dfrac{1 \times 1^3}{12} = 0.08333\ \text{m}^4$

$$\text{Maximum deflection} = \delta_{\max} = \frac{PL^3}{3EI} = \frac{5 \times 10^3 \times (5)^3}{3 \times 210 \times 10^9 \times 0.08333} = 1.2 \times 10^{-5}\ \text{m}.$$
$$= 0.012\ \text{mm}$$

Answers of Example A.1

Parameter	Software results	Analytical method
Maximum deflection	0.012277 mm	0.012 mm
Maximum stress	0.15 MPa	0.15 MPa
Reaction force	5000 N	5000 N
Reaction moment	25000 N-m	25000 N-m

REFERENCES

1. ANSYS Release 11.0 Tutorials.
2. ANSYS Tutorials of University of Alberta, *http://www.mece.ualberta.ca*
3. ANSYS Tutorials, *http://mae.uta.edu/~lawrence*
4. ANSYS Tutorials, *http://www.andrew.cmu.edu*
5. ANSYS Tutorials, *https://confluence.cornell.edu/display/SIMULATION*
6. E. M. Alawadhi, "Finite Element Simulations Using ANSYS," CRC Press, 2010.
7. Y. Nakasone, S. Yoshimoto, and T. A. Stolarski, "Engineering Analysis with ANSYS Software," Butterworth-Heinemann, 2006.

APPENDIX B

MATLAB

MATLAB (short for Matrix Laboratory) has become a useful and dominant tool for technical professionals around the world. MATLAB is a sophisticated numerical computation and simulation tool that uses matrices and vectors. Also, MATLAB enables users to solve a wide variety of analytical problems.

A copy of MATLAB software can be obtained from

The Mathworks, Inc.
3 Apple Hill Drive
Natick, MA 01760-2098
Phone: 508-647-7000
Web site: *http://www.mathworks.com*

This brief overview of MATLAB is presented here to give a general idea about the computation of the software. MATLAB computational applications to engineering systems are used to solve practical problems.

B.1 GETTING STARTED AND WINDOWS OF MATLAB

When you double-click on the MATLAB icon on the desktop or use the start menu to find the program, it opens, as shown in Figure B.1. The command window, where the special **>>** prompt appears, is the main area in which the user interacts with MATLAB. To make the Command Window active, you need to click anywhere inside its border. The MATLAB prompt **>>** tells the user that MATLAB is ready to enter a command. To quit MATLAB, you can select **EXIT MATLAB** from the **File** menu, or by enter *quit* or *exit* at the Command Window prompt. Note, do not click on the X (close box) in the top right corner of the MATLAB window, because it may cause problems with the operating software. Figure B.1 contains four default windows: Command Window, Workplace Window, Command History Window, and Current Folder Window. Table B.1 shows a list of the various windows and their purpose in MATLAB.

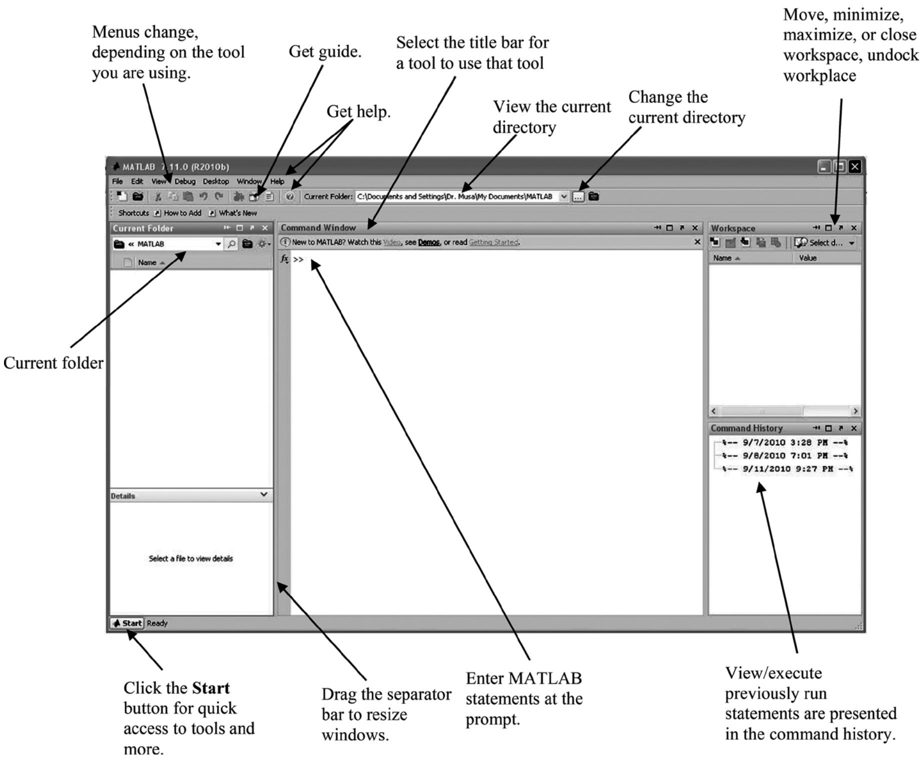


FIGURE B.1 MATALB default environment.

TABLE B.1 MATLAB Windows.

MATLAB Window	Description
Command window	Main window, enter variables, runs programs
Workplace window	Gives information about the variables used
Command History window	Records commands entered in the Command window
Current Folder window	Shows the files in the current directory with details
Editor window	Makes and debugs script and function files
Help window	Gives help information
Figure window	Contains output from the graphic commands
Launch Pad window	Provides access to tools, demos, and documentation

B.2 USING MATLAB IN CALCULATIONS

Table B.2 shows the MATLAB common arithmetic operators. The order of operations is as follows: first, parentheses ( ), the innermost is executed first for nested parentheses; second, exponentiation ^; third, Multiplication * and division / (they are equal precedence); fourth, addition + and subtraction −.

TABLE B.2 MATLAB Common Arithmetic Operators.

MATLAB Operators Symbols	Descriptions
+	Addition
−	Subtraction
*	Multiplication
/	Right division (means $\frac{a}{b}$)
\	Left division (means $\frac{b}{a}$)
^	Exponentiation (raising to a power)
‘	Converting to complex conjugate transpose
( )	Specify evaluation order

For example,

```
>> a = 8; b = -3; c = 2;
>> x = 9*a + c^3 - 49

x =

   31

>> y = sqrt(x)/8

y =

   0.6960
```

MATLAB has different displays in several formats. Table B.3 provides common numeric display formats. You can obtain more by typing *help* in the Command window (>> format).

TABLE B.3 MATLAB Common Numeric Display Formats.

MATLAB Command	Descriptions
format	Default display, same as short
format short	Display 4 decimal digits
format long	Display 14 decimal digits
format short e	Display 4 decimal digits in scientific notation
format long e	Display 14 decimal digits in scientific notation
format short g	Display the best short format selected by MATLAB
format long g	Display the best long format selected by MATLAB
format rat	Display fractional form

For example,

```
>> pi

ans =

   3.1416

>> format short
>> pi

ans =

   3.1416

>> format long
>> pi

ans =

   3.14159265358979
```

```
>> format short e
>> pi

ans =

   3.1416e+000

>> format long e
>> pi

ans =

   3.141592653589793e+000

>> format short g
>> pi

ans =

   3.1416
```

```
>> format long g
>> pi

ans =

      3.14159265358979
>> format rat
>> pi

ans =

   355/113
```

Table B.4 provides common samples of MATLAB functions. You can obtain more by typing *help* in the Command window (>> help).

TABLE B.4 MATLAB Typical Elementary Math Functions.

MATLAB Function	Description
abs (x)	Absolute value or complex magnitude of x
acos (x), acosh (x)	Inverse cosine and inverse hyperbolic cosine of x (in radians)
angle (x)	Phase angle (in radians) of a complex number x
asin (x), asinh (x)	Inverse sine and inverse hyperbolic sine of x (in radians)
atan (x), atanh (x)	Inverse tangent and inverse hyperbolic tangent of x (in radians)
conj (x)	Complex conjugate of x (in radians)
cos (x), cosh (x)	Cosine and inverse hyperbolic cosine of x (in radians)
cot (x), coth (x)	Inverse cotangent and inverse hyperbolic cotangent of x (in radians)
exp (x)	Exponential of x
fix	Round toward zero
imag (x)	Imaginary part of a complex number x
log (x)	Natural logarithm of x
log2 (x)	Natural logarithm of x to base 2
log10 (x)	Common logarithms (base 10) of x
real (x)	Real part of a complex number of x
sin (x), sinh (x)	Sine and inverse hyperbolic sine of x (in radians)
sqrt (x)	Square root of x
tan (x), tanh (x)	Tangent and inverse hyperbolic tangent of x (in radians)

For example,

```
>> 25-3^(log10(3.25))+11

ans =

   34.2452

>> y=6*sin(pi/6)+tan(pi/4)

y =

   4.0000

>> z = exp(y+3)-2

z =

   1.0946e+00
```

In addition to operating on mathematical functions, MATLAB allows us to work easily with vectors and matrices. A vector (or one-dimensional array) is a special matrix (or two-dimensional array) with one row or one column. Arithmetic operations can apply to matrices and Table B.5 has extra common operations that can be implemented to matrices.

TABLE B.5 MATLAB Matrix Operations.

MATLAB Operations	Descriptions
A'	Transpose of matrix A
det (A)	Determinant of matrix A
inv (A)	Inverse of matrix A
eig (A)	Eigenvalues of matrix A
diag (A)	Diagonal elements of matrix A
rank (A)	Rank of matrix A
cond (A)	Condition number of matrix A
eye(n)	The $n \times n$ identity matrix (1's on the main diagonal)
eye(m, n)	The $m \times n$ identity matrix (1's on the main diagonal)
trace (A)	Summation of diagonal elements of matrix A
zeros (m, n)	The $m \times n$ matrix consisting of all zeros
ones (m, n)	The $m \times n$ matrix consisting of all ones
rand (m, n)	The $m \times n$ matrix consisting of random numbers
randn (m, n)	The $m \times n$ matrix consisting of normally distributed numbers
diag (A)	Extraction of the diagonal matrix A as vector
diag (A,1)	Extracting of first upper off-diagonal vector of matrix A
diag (**u**)	Generating of a diagonal matrix with a vector **u** on the diagonal
expm (A)	Exponential of matrix A
ln (A)	LU decomposition of matrix A
svd (A)	Singular value decomposition of matrix A
qr (A)	QR decomposition of matrix A
min (A)	Minimum of vector A
max (A)	Maximum of vector A
sum (A)	Sum of elements of vector A
std (A)	Standard deviation of the data collection of vector A
sort (A)	Sort the elements of vector A
mean (A)	Means value of vector A
triu (A)	Upper-triangular of matrix A
triu (A, I)	Upper-triangular with zero diagonals of matrix A
tril (A)	Lower-triangular of matrix A
tril (A, I)	Lower-triangular with zero diagonals of matrix A

A vector can be created by typing the elements inside brackets [ ] from a known list of numbers.

Examples are shown in Table B.5.

```
>> A = [1 -2 0 6 4 9]

A =

   1   –2   0   6   4   9

>> B = [3 2 7; -4 6 15; 8 5 16]

B =

    3   2    7
   –4   6   15
    8   5   16
```

```
>> diag (A)

ans =

   1    0   0   0   0   0
   0   –2   0   0   0   0
   0    0   0   0   0   0
   0    0   0   6   0   0
   0    0   0   0   4   0
   0    0   0   0   0   9

>> det(B)

ans =

   –45

>> trace(B)

ans =

   25
```

Also, a vector can be created with constant spacing by using the command *variable-name = [a: n: b]*, where *a* is the first term of the vector; *n* is spacing; *b* is the last term.

For example,

```
>> t = [1:0.5:4]

t =

   1.0000   1.5000   2.0000   2.5000   3.0000   3.5000   4.0000
```

In addition, a vector can be created with constant spacing by using the command *variable-name = linespace (a, b, m)*, where *a* is the first element of the vector; *b* is the last element; *m* is number of elements.

For example,

```
>> x=linspace (0,3*pi,6)

x =

      0   1.8850   3.7699   5.6549   7.5398   9.4248
```

Special constants can be used in MATLAB. Table B.6 provides special constants used in MATLAB.

TABLE B.6 MATLAB Named Constants.

Name	**Content**
pi	$\pi = 3.14159...$
i or *j*	Imaginary unit, $\sqrt{-1}$
eps	Floating-point relative precision, 2^{-52}
realmin	Smallest floating-point number, 2^{-1022}
realmax	Largest floating-point number $(2\text{-}eps).2^{1023}$
bimax	Largest positive integer, $2^{53}-1$
Inf or *Inf*	Infinity
nan or *NaN*	Not a number
rand	Random element
eye	Identity matrix
ones	An array of 1's
zeros	An array of 0's

For example,

```
>> eye(3)

ans =

   1   0   0
   0   1   0
   0   0   1

>> 1/0
Warning: Divide by zero.
(Type "warning off MATLAB:divideByZero" to suppress this warning.)

ans =

   Inf

>> 0/0
Warning: Divide by zero.
(Type "warning off MATLAB:divideByZero" to suppress this warning.)

ans =

   NaN
```

Arithmetic operations on arrays are done element by element. Table B.7 provides MATLAB common arithmetic operations on arrays.

TABLE B.7 MATLAB Common Arithmetic Operations on Arrays.

MATLAB Operators Symbols on Arrays	Descriptions
+	Addition same as matrices
−	Subtraction same as matrices
.*	Element-by-element multiplication
./	Element-by-element right division
.\	Element-by-element left division
.^	Element-by-element power
.'	Unconjugated array transpose

For example,

```
>> A=[0 2 4; 1 3 5; 9 6 3]

A =

   0   2   4
   1   3   5
   9   6   3

>> A.*A

ans =

   0   4    16
   1   9    25
    81    36   9

>> A.^2

ans =

   0   4    16
   1   9    25
    81    36   9
```

B.3 SYMBOLIC COMPUTATION

In this section, MATLAB can manipulate and solve symbolic expressions that make the user compute with math symbols rather than numbers. This process is called *symbolic math*. Table B.8 shows some common symbolic commands.

TABLE B.8 Common Symbolic Commands.

MATLAB Command	Description
diff	Differentiate symbolic expression
int	Integrate symbolic expression
jacobian	Compute Jacobian matrix
limit	Compute limit of symbolic expression
symsum	Evaluate symbolic sum of series
*tayl*or	Taylor series expansion
colspace	Return basis for column space of matrix
det	Compute determinant of symbolic matrix
diag	Create or extract diagonals of symbolic matrices
eig	Compute symbolic eigenvalues and eigenvectors
expm	Compute symbolic matrix exponential
inv	Compute symbolic matrix inverse
jordan	Compute Jordan canonical form of matrix
null	Form basis for null space of matrix
poly	Compute characteristic polynomial of matrix
rank	Compute rank of symbolic matrix
rref	Compute reduced row echelon form of matrix
svd	Compute singular value decomposition of symbolic matrix
tril	Return lower-triangular part of symbolic matrix
triu	Return upper-triangular part of symbolic matrix
coeffs	List coefficients of multivariate polynomial
collect	Collect coefficients
expand	Symbolic expansion of polynomials and elementary functions
factor	Factorization
horner	Horner nested polynomial representation
numden	Numerator and denominator
simple	Search for simplest form of symbolic expression
simplify	Symbolic simplification
subexpr	Rewrite symbolic expression in terms of common subexpressions
subs	Symbolic substitution in symbolic expression or matrix
compose	Functional composition

MATLAB Command	Description
dsolve	Symbolic solution of ordinary differential equations
finverse	Functional inverse
solve	Symbolic solution of algebraic equations
cosint	Cosine integral
sinint	Sine integral
zeta	Compute Riemann zeta function
ceil	Round symbolic matrix toward positive infinity
conj	Symbolic complex conjugate
eq	Perform symbolic equality test
fix	Round toward zero
floor	Round symbolic matrix toward negative infinity
frac	Symbolic matrix elementwise fractional parts
imag	Imaginary part of complex number
log10	Logarithm base 10 of entries of symbolic matrix
log2	Logarithm base 2 of entries of symbolic matrix
mod	Symbolic matrix elementwise modulus
pretty	Pretty-print symbolic expressions
quorem	Symbolic matrix elementwise quotient and remainder
real	Real part of complex symbolic number
round	Symbolic matrix elementwise round
size	Symbolic matrix dimensions
sort	Sort symbolic vectors, matrices, or polynomials
sym	Define symbolic objects
syms	Shortcut for constructing symbolic objects
*symv*ar	Find symbolic variables in symbolic expression or matrix
fourier	Fourier integral transform
ifourier	Inverse Fourier integral transform
ilaplace	Inverse Laplace transform
iztrans	Inverse *z*-transform
laplace	Laplace transform
*ztra*ns	*z*-transform

You can practice some of the symbolic expressions as shown below:

B.3.1 Simplifying Symbolic Expressions

Symbolic simplification is not always straightforward; there is no universal simplification function because the meaning of the simplest representation of a symbolic expression cannot be defined clearly. MATLAB uses the *sym* or *syms* command to declare variables as a symbolic variable. Then, the symbolic can be used in expressions and as arguments to many functions.

For example, to rewrite a polynomial in a standard form, use the *expand* function:

```
>> syms x y; % creating a symbolic variables x and
>> x = sym('x'); y = sym('y'); y = sym('y'); % or equivalently
>> expand (sin(x-y))

ans =

sin(x)*cos(y)-cos(x)*sin(y)
```

You can use the *subs* command to substitute a numeric value for a symbolic variable or replace one with another.

For example,

```
>> syms x;
>> f=5*x^3+2*x-5;
>> subs(f,3)

ans =

   136


>> simplify (3*sin(x)^2 - cos(x)^2)

ans =

-4*cos(x)^2+3
```

B.3.2 Differentiating Symbolic Expressions

Use *diff* ( ) command for differentiation.

For example,

```
>> syms x;
>> f =sin(3*x)+5;
>> diff(f)

ans =

3*cos(3*x)

>> y=4*sin(x)*exp(x);
>> diff(y)

ans =

4*cos(x)*exp(x)+4*sin(x)*exp(x)


>> diff(diff(y))% second derivative of y

ans =

8*cos(x)*exp(x)
```

```
>> syms v u;
>> f = sin(v*u);
>> diff(f,u) % create partial derivative ∂f/∂u

>> diff(f,v) % create partial derivative ∂f/∂v
 ans =

cos(v*u)*u

>> diff(f,u,2) % create second partial derivative ∂²f/∂u²

ans =

-sin(v*u)*v^2
```

B.3.3 Integrating Symbolic Expressions

The *int(f)* function is used to integrate a symbolic expression *f*.

For example,

```
>> syms x;
>> f=2+3*tan(x)^2;
>> int(f)

ans =

-x+3*tan(x)

>> int(1/(1-x^2))

ans =

atanh(x)
```

B.3.4 Limits Symbolic Expressions

The *limit(f)* command is used to calculate the limits of function *f*.

For example,

```
>> syms x y z;
>> limit((cos(x)/x), x, 0)

ans =

NaN

>> limit((tan(x)/x), x, 0)

ans =

1

>> limit(1/x, x, 0, 'right') % lim_{x→0+} 1/x = ∞

ans =

Inf

>> limit(1/x, x, 0, 'left') % lim_{x→0-} 1/x = -∞

ans =

-Inf
```

B.3.5 Taylor Series Symbolic Expressions

Use *taylor( )* function to find the Taylor series of a function with respect to the variable given.

For example,

```
>> syms x; N =5;
>> taylor(exp(-x),N+1) %  f(x) ≅ Σ_{n=0}^{N} (1/n!) f^n(0)

ans =

1-x+1/2*x^2-1/6*x^3+1/24*x^4-1/120*x^5


>> f=exp(2*x);
>> taylor(f,6)

ans =

1+2*x+2*x^2+4/3*x^3+2/3*x^4+4/15*x^5
```

B.3.6 Sums Symbolic Expressions

Use *symsum ( )* function to obtain the sum of a series.

For example,

```
>> syms k n;
>> symsum(k,0,n-1) %  Σ_{k=0}^{n-1} k = 0+1+2+...+n-1 = (1/2)n^2 - (1/2)n

ans =

1/2*n^2-1/2*n


>> syms n N;
>> symsum(1/n^2,1,inf) %  Σ_{n=0}^{N} 1/n^2 = π^2/6

ans =


        1/6 *pi^2
```

B.3.7 Solving Equations as Symbolic Expressions

Many of MATLAB's commands and functions are used to manipulate the vectors or matrices consisting of symbolic expressions.

For example,

```
>> syms a b c d;
>> M=[a b;c d];
>> det(M)

ans =

    a *d - b *c
```

```
>> syms x y;
>> f=solve('3*x+5*y=15','x-8*y=3');
>> x=f.x

x =

135/29

>> y=f.y

y =

6/29
```

```
>> syms x;
>> solve (x^3-6*x^2+11*x-6)

ans =

   1
   2
   3
```

Use *dsolve ( )* function to solve symbolic differential equations.

For example,

```
>> syms x y t;
>> dsolve('Dy+2*y=12') % solve y' + 2y = 12
ans =
6+exp(-2*t)*C1 % C1 is undetermined constant
```

APPENDIX C

COMSOL MULTIPHYSICS

COMSOL is a finite-element-based modeling tool that has a well-developed graphic user interface and several modules for modeling common and advanced types of physics involved in engineering and applied science practices.

COMSOL INTERFACE

You can set up a model guided by the *Model Wizard* (it will guide the users through the steps necessary for building a model) or start from a *Blank Model,* as shown in Figure C.1.

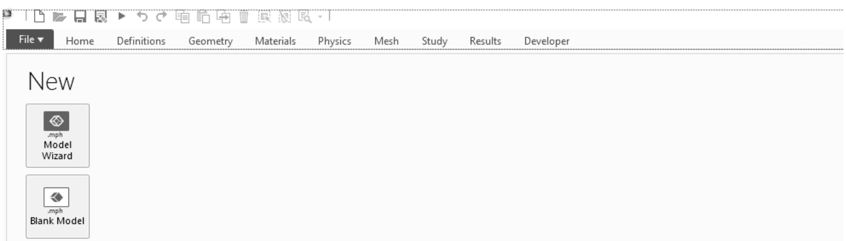


FIGURE C.1 New window opens when launching COMSOL.

After clicking on *Model Wizard* icon, a new window, Select Space Dimension, will open, as shown in Figure C.2. Users can choose the physical dimension of the model by clicking on the relevant icon, which includes 0D, 1D, and 2D axisymmetric cases, as well.

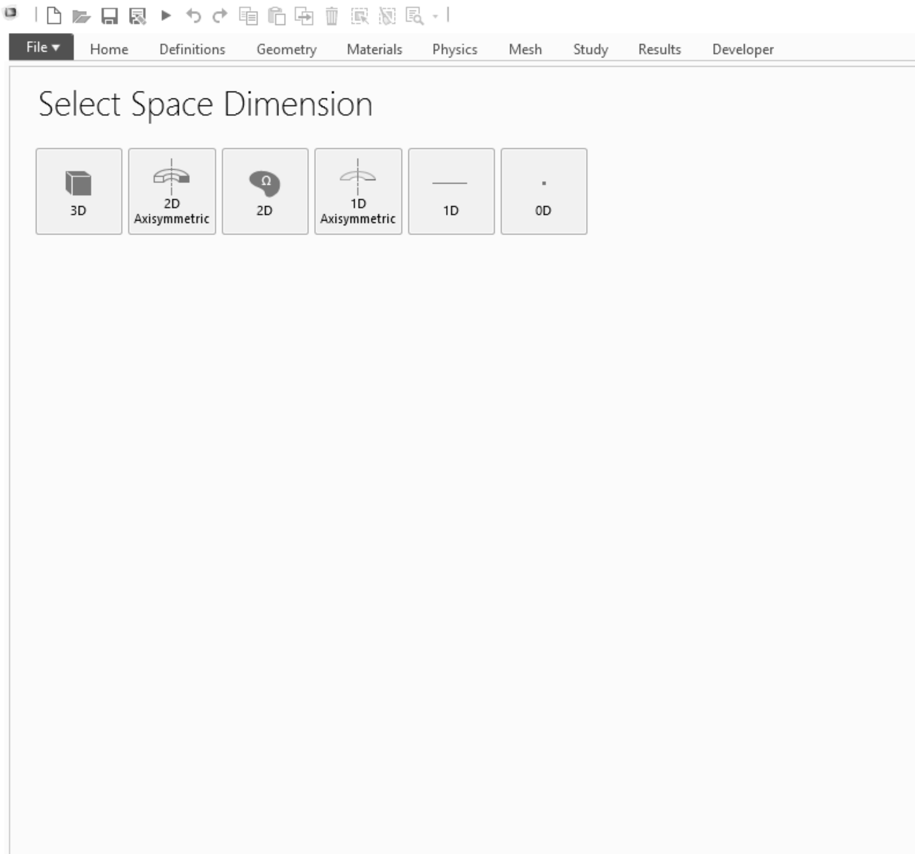


FIGURE C.2 Select Space Dimension window.

CREATING A MODEL GUIDED BY THE MODEL WIZARD

The *Model Wizard* will guide you in setting up the space dimension, physics, and study type in a few steps:

1. Select the space dimension for your model component: 3D, 2D Axisymmetric, 2D, 1D Axisymmetric, or 0D.
2. Add one or more physics interfaces, as shown in Figure C.3. These are organized in a number of physics branches in order to make them easy to locate. These branches do not directly correspond to products. When products are added to your COMSOL Multiphysics installation, one or more branches will be populated with additional physics interfaces.

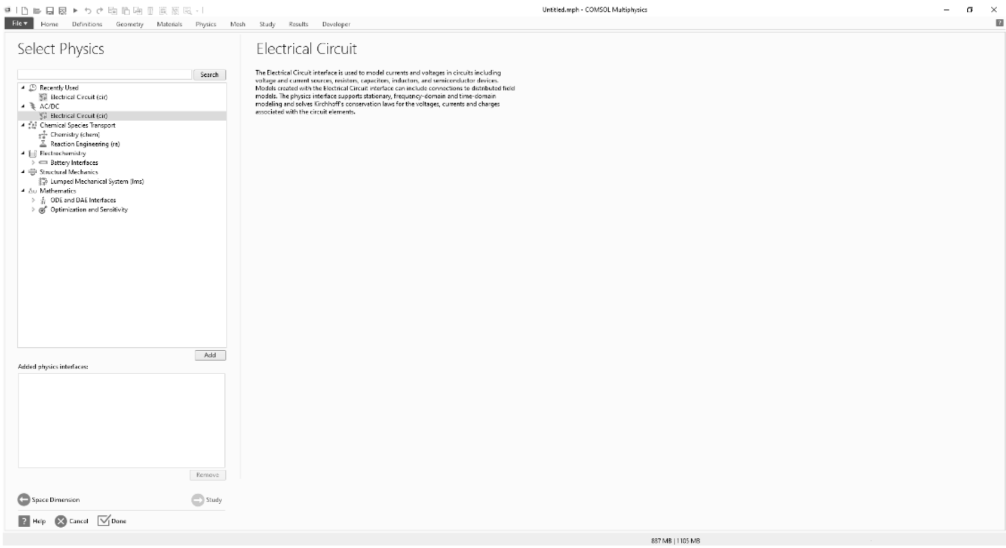


FIGURE C.3 Select Physics interfaces window.

3. Select the Study type that represents the solver or set of solvers that will be used for the computation, as shown in Figure C.4.

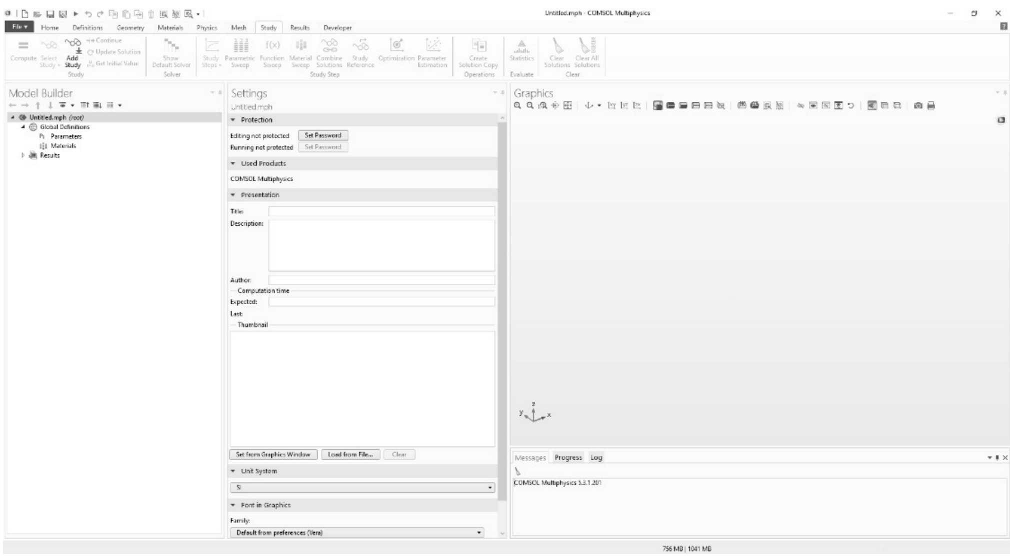


FIGURE C.4 Select study type window.

4. Finally, click **Done**. The desktop is now displayed with the model tree configured according to the choices you made in the *Model Wizard*.

CREATING A BLANK MODEL

The *Blank Model* option will open the COMSOL interface without any Component or Study. You can right-click the model tree to add a Component of a certain space dimension, physics interface, or Study.

The Ribbon and Quick Access Toolbar

The *ribbon tabs* in the COMSOL environment reflect the modeling workflow and give an overview of the functionality available for each modeling step, including building simulation applications from your models. The main toolbar items are listed according to the usual sequence used for building a model; Model, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results, as shown in Figure C.5. The ribbon bar under the Model tab lists the modeling sequence actions required, as well.

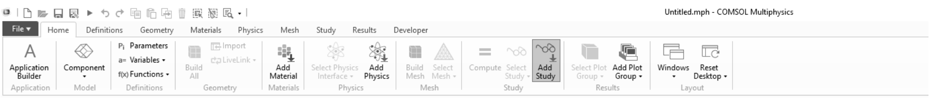


FIGURE C.5 Toolbar and Ribbon for selected Model tab.

The **Home** tab contains buttons for the most common operations for making changes to a model, running simulations, and building and testing applications, as shown in Figure C.6. Examples include changing model parameters for a parameterized geometry, reviewing material properties and physics, building the mesh, running a study, and visualizing the simulation results.

There are standard tabs for each of the main steps in the modeling process. These are ordered from left to right according to the workflow: **Definitions**, **Geometry**, **Materials**, **Physics**, **Mesh**, **Study**, **Results**, and **Developer**.

Contextual tabs are shown only if and when they are needed, such as the **3D Plot Group** tab, which is shown when the corresponding plot group is added or when the node is selected in the model tree.

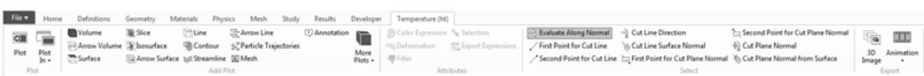


FIGURE C.6 The home tab.

Modal tabs are used for very specific operations when other operations in the ribbon may become temporarily irrelevant. An example is the **Work Plane** modal tab, as shown in Figure C.7. When working with work planes, other tabs are not shown since they do not present relevant operations.

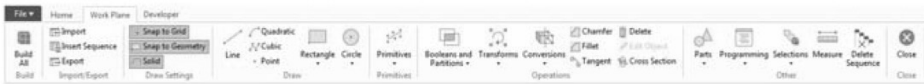


FIGURE C.7 The Work Plane modal tab.

THE RIBBON VS. THE MODEL BUILDER

The ribbon gives quick access to available commands and complements the model tree in the **Model Builder** window, as shown in Figure C.8. Most of the functionality accessed from the ribbon is also accessible from contextual menus by right-clicking nodes in the model tree. Certain operations are only available from the ribbon, such as selecting which desktop window to display. There are also operations that are only available from the model tree, such as reordering and disabling nodes.

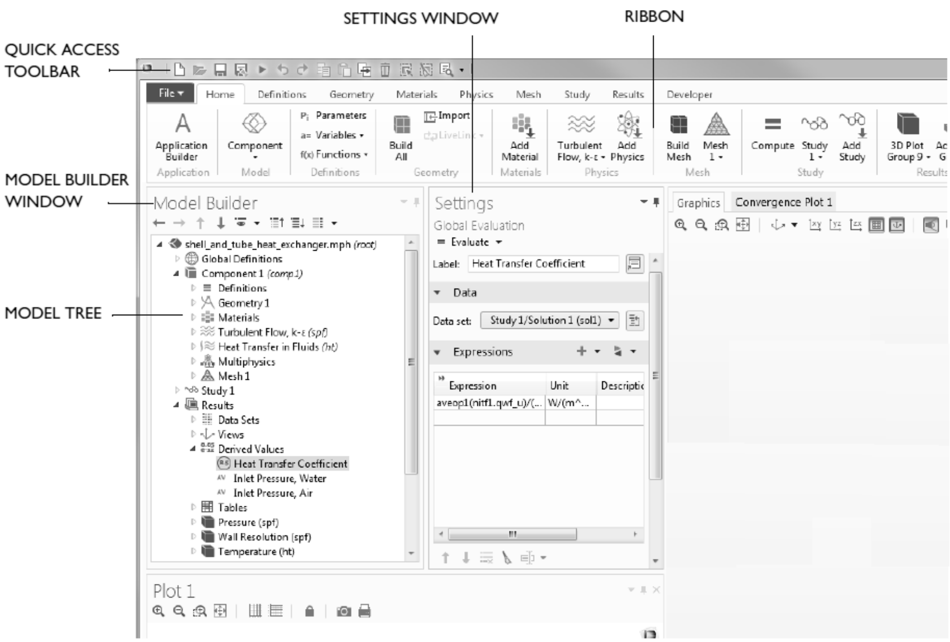


FIGURE C.8 The Windows, toolbar, ribbon, and model tree.

The Quick Access Toolbar

The **Quick Access Toolbar** contains a set of commands that are independent of the ribbon tab that is currently displayed. You can customize the **Quick Access Toolbar** and add most commands available in the **File** menu, including commands for undoing and redoing recent actions, as well as for copying, pasting, duplicating, and deleting nodes in the model tree. You can also choose to position the **Quick Access Toolbar** above or below the ribbon.

The Model Builder and the Model Tree

Using the Model Builder, you build a model by starting with the default model tree, adding nodes, and editing the node settings, as shown in Figure C.9.

All the nodes in the default model tree are top-level parent nodes. You can right-click on them to see a list of child nodes, or subnodes, that you can add beneath them. This is the means by which nodes are added to the tree.

When you click on a child node, you will see its node settings in the Settings window. It is here that you can edit node settings.

It is worth noting that if you have the **Help** window open, which is achieved either by selecting help from the **File** menu or by pressing the function key F1, then you will also get dynamic help when you click on a node.

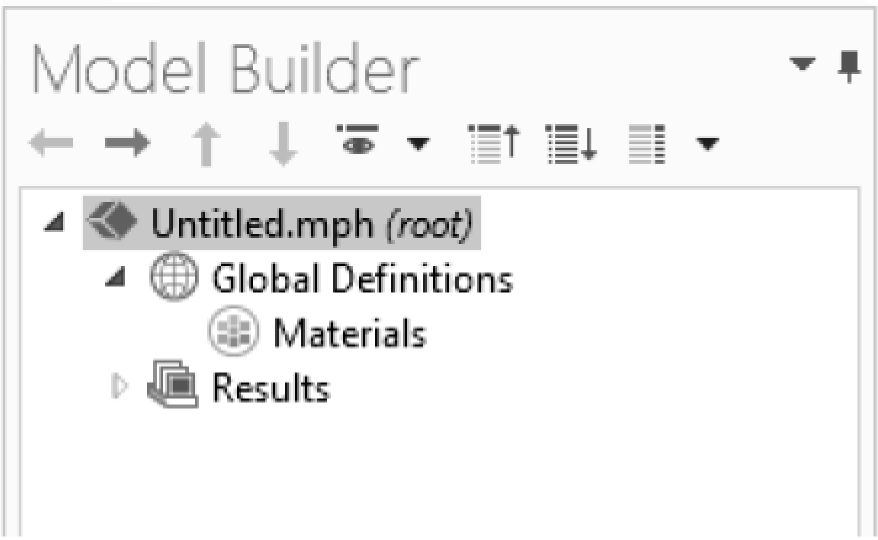


FIGURE C.9 The Model Builder.

THE ROOT, GLOBAL DEFINITIONS, AND RESULTS NODES

A model tree always has a root node (initially labeled Untitled.mph), a **Global Definitions** node, and a **Results** node. The label on the root node is the name of the multiphysics model file, or MPH file, to which the model is saved. The root node has settings for author name, default unit system, and more.

The **Global Definitions** node has a **Materials** subnode by default. The **Global Definitions** node is where you define parameters, variables, functions, and couplings that can be used throughout the model tree. They can be used, for example, to define the values and functional dependencies of material properties, forces, geometry, and other relevant features. The **Global Definitions** node itself has no settings, but its child nodes have plenty of them. The **Materials** subnode stores material properties that can be referenced in the Component nodes of a model.

The **Results** node is where you access the solution after performing a simulation and where you find tools for processing the data. The **Results** node initially has five subnodes, as shown in Figure C.10:

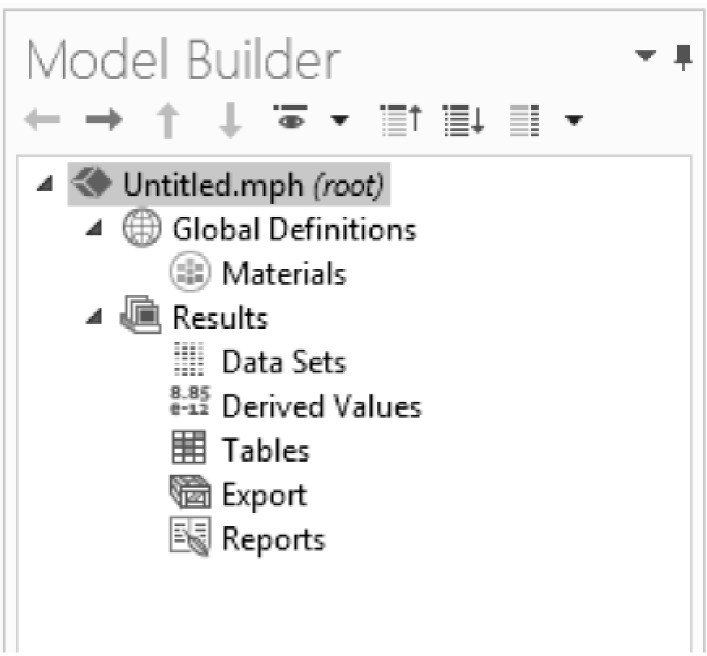


FIGURE C.10 The Results node.

- **Data Sets,** which contains a list of solutions you can work with.
- **Derived Values**, which defines values to be derived from the solution using a number of postprocessing tools.
- **Tables**, which is a convenient destination for the **Derived Values** or for **Results** generated by probes that monitor the solution in real-time while the simulation is running.
- **Export**, which defines numerical data, images, and animations to be exported to files.
- **Reports**, which contains automatically generated or custom reports about the model in HTML or Microsoft® Word format.

The Component and Study Nodes

In addition to the three nodes just described, there are two additional top-level node types: **Component** nodes and **Study** nodes, as shown in Figure C.11. These are usually created by the Model Wizard when you create a new model. After using the Model Wizard to specify what type of physics you are modeling and what type of Study (e.g., steady-state, time-dependent, frequency-domain, or eigenfrequency analysis) you will carry out, the Model Wizard automatically creates one node of each type and shows you their contents.

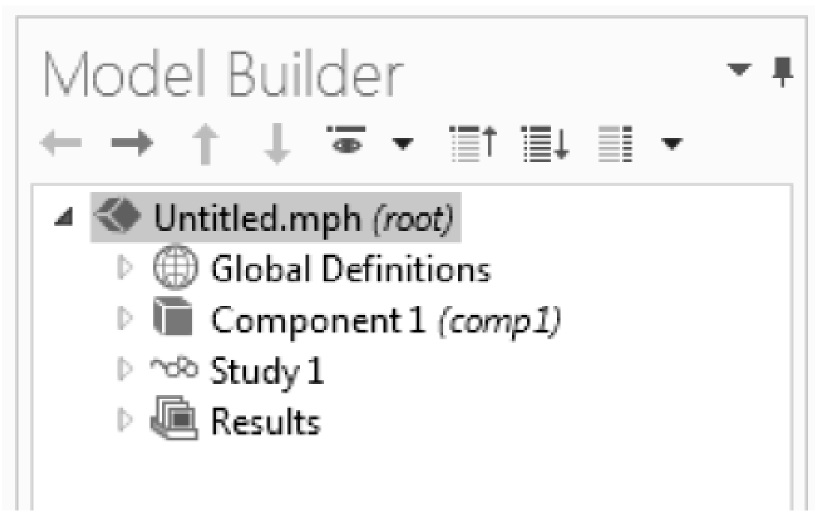


FIGURE C.11 The component and study nodes.

It is also possible to add more **Component** and **Study** nodes as you develop the model, as shown in Figure C.12. A model can contain multiple **Componen**t and **Study** nodes, and it would be confusing if they all had the same name. Therefore, these types of nodes can be renamed to be descriptive of their individual purposes.

If a model has multiple **Component** nodes, they can be coupled to form a more sophisticated sequence of simulation steps.

Note that each Study node may carry out a different type of computation, so each one has a separate **Compute** button.

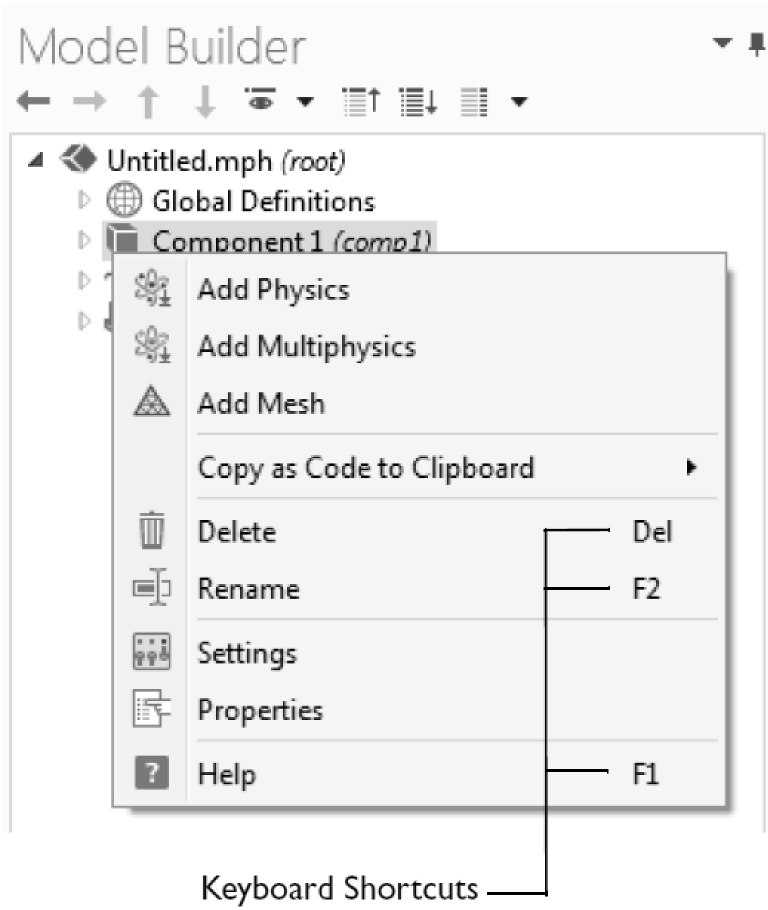


FIGURE C.12 The Component and Study nodes as developing the model.

To be more specific, suppose that you build a model that simulates a coil assembly that is made up of two parts, a coil and a coil housing, as shown in Figure C.12. You can create two Component nodes, one that models the coil and the other the coil housing. You can then rename each of the nodes with the name of the object. Similarly, you can also create two Study nodes, the first simulating the stationary or steady-state behavior of the assembly and the second simulating the frequency response. You can rename these two nodes to be Stationary and Frequency Domain. When the model is complete, save it to a file named Coil Assembly.mph. At that point, the model tree in the Model Builder looks like Figure C.13.

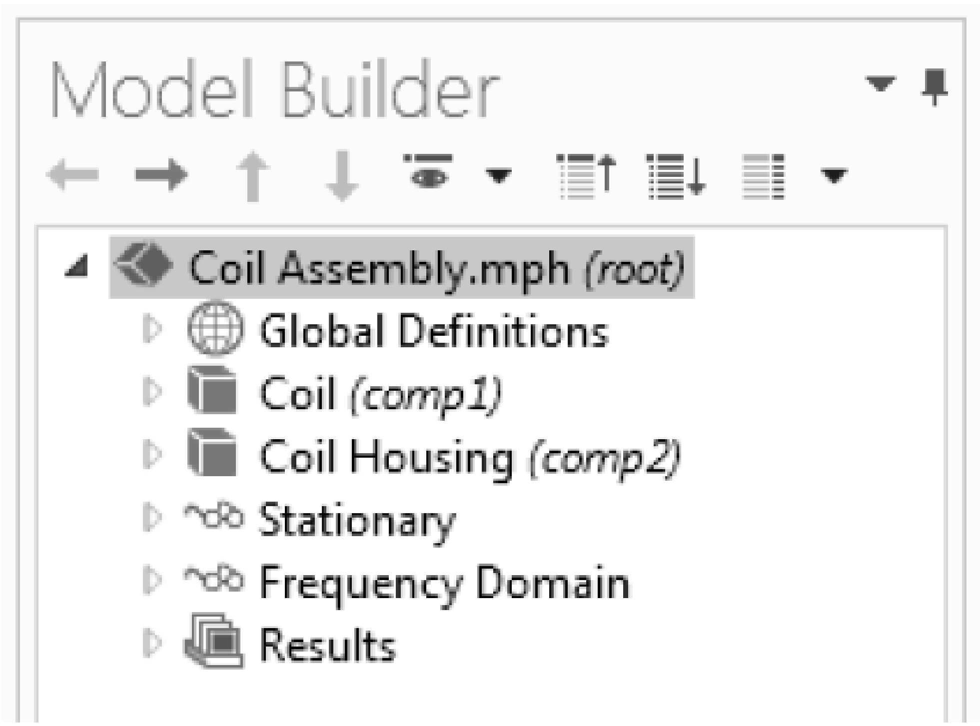


FIGURE C.13 A model that simulates a coil of two parts, a coil and a coil housing.

In Figure C.13, the root node is named Coil Assembly.mph, indicating the file in which the model is saved. The Global Definitions node and the Results node each have their default name. Additionally, there are two Component nodes and two Study nodes with the names chosen in the previous paragraph.

Parameters, Variables, and Scope

Global Parameters:

Global parameters are user-defined constant scalars that are usable throughout the model. That is to say, and they are "global" in nature. Important uses are:

- Parameterizing geometric dimensions.
- Specifying mesh element sizes.
- Defining parametric sweeps (simulations that are repeated for a variety of different values of a parameter such as a frequency or load).

A global parameter expression can contain numbers, global parameters, built-in constants, built-in functions with global parameter expressions as arguments, and unary and binary operators. It is important to know that the names of parameters are case-sensitive. You define global parameters in the **Parameters** node in the model tree under **Global Definitions**.

Results Parameters:

For greater flexibility, it is possible to define parameters that are only used in the Results node. Using these parameters does not require resolving the model.

Result parameters may depend on other result parameters but not on global parameters.

Variables:

Variables have associated Variables nodes in the model tree and can be defined either in the Global Definitions node or in the Definitions subnode of any Component node. Naturally, the choice of where to define the variable depends on whether you want it to be global (that is, usable throughout the model tree) or locally defined within a single **Component** node. Like a parameter expression, a variable expression may contain numbers, parameters, built-in constants, and unary and binary operators. However, it may also contain variables like t, x, y, or z; functions with variable expressions as arguments; and dependent variables that you are solving for in addition to their space and time derivatives.

Variables Used in Applications:

Model parameters and variables can be used in applications. For example, you can let the user of an application change the value of a parameter. In addition, variables to be used in applications can be defined in the Application Builder, in the application tree under the **Declarations** node. Such variables can also be used in model methods.

Scope:

The "scope" of a parameter or variable is a statement about where it may be used in an expression. All global parameters are defined in the **Global Definitions** node of the model tree as a **Parameters** subnode. This means that they are global in scope and can be used throughout the model tree.

A variable may also be defined in the Global Definitions node, as a **Variables** subnode, and have global scope, but they are subject to other limitations. For example, variables may not be used in **Geometry**, **Mesh**, or **Study** nodes (with the one exception that a variable may be used in an expression that determines when the simulation should stop).

A variable that is instead defined under the **Definitions** subnode of a **Component** node has local scope and is intended for use in that particular **Component** (but, again, not in the **Geometry** or **Mesh** nodes). They may be used, for example, to specify material properties in the **Materials** subnode of a **Component** or to specify boundary conditions or interactions. It is sometimes valuable to limit the scope of the variable to only a certain part of the geometry, such as certain boundaries. For that purpose, provisions are available in the settings for a variable to select whether to apply the definition either to the entire geometry of the**Component** or only to a **Domain**, **Boundary**, **Edge**, or **Point**.

Workflow and Sequence of Operations

In the Model Builder window, every step of the modeling process, from defining global variables to the final report of results, is displayed in the model tree.

From top to bottom, the model tree defines an orderly sequence of operations. In the following branches of the model tree, the node order makes a difference, and you can change the sequence of operations by moving the subnodes up or down the model tree:

- Geometry
- Materials
- Physics

- Mesh
- Study
- Plot Groups

In the **Component** > **Definitions** branch of the tree, the ordering of the following node types also makes a difference:

- Perfectly Matched Layer
- Infinite Elements

Nodes may be reordered by these methods:

- Drag-and-drop
- Right-clicking the node and selecting Move Up or Move Down
- Pressing Ctrl + Up arrow or Ctrl + Down arrow

In other branches, the ordering of nodes is not significant with respect to the sequence of operations, but some nodes can be reordered for readability. Child nodes to Global Definitions is one such example.

You can view the sequence of operations presented as program code statements by saving the model as a Model File for MATLAB® or as a Model File for Java® after having selected Compact History in the File menu. Note that the model history keeps a complete record of the changes you make to a model as you build it. As such, it includes all of your corrections, including changes to parameters and boundary conditions and modifications of solver methods. Compacting this history removes all of the overridden changes and leaves a clean copy of the most recent form of the model steps. In the Application Builder, you can use the Record Method option to view and edit program code statements in the Method editor.

As you work with the COMSOL interface and the Model Builder, you will grow to appreciate the organized and streamlined approach. However, any description of a user interface is inadequate until you try it for yourself. In the next chapters, you are invited to work through two examples to familiarize yourself with the software.

INDEX

A

ABC. *See* Absorbing boundary conditions
Absorbing boundary conditions (ABC), 462–464
Airfoil, potential flow
  post-processing, 321–322
  preprocessing, 317–321
  software results, 317
Analytical method, 54
  axial vibrations, 362
  cantilever beams, 216–217, 222–223
  constant cross-section area, 71–72, 76–78, 83, 88–89, 92–93
  engineering problem, 54
  natural frequency determination, 329
  one-dimensional heat conduction problems, 275
  simply supported beams, 190, 194–195, 201–202, 206–207
  stepped bar, 127, 131–132, 138–140
  stress analysis, rectangular plate with circular hole, 257–259
  truss, 159–162
  varying cross-section area, 105–106, 112–113
ANSYS
  classic/traditional, 525–535
  design modeler, 536
  design optimization-design, 537
  engineering data, 539–554
  finite element modeler, 536
  geometry definition, 536
  graphical user interface (GUI), 525
  materials definition, 537
  mechanisms, 537
  reporting, 537
  software method, 538–539
  Solenoid actuator analysis, 484–505
  workbench, 536–554
  workbench user interface, 536
ANSYS program, 157
Anti-symmetric (Skew-symmetric) matrix, 5
Argument matrix, 19
Automatic mesh generation
  arbitrary domains, 441–443
  rectangular domains, 439–441
Axially loaded members
  constant cross-section area, 71–104
  stepped bar, 126–155
  three-node bar element, 70–71

two-node bar element, 68–70
varying cross-section area, 104–1126
Axial vibrations
analytical method, 362
FEM by hand calculations, 363–365
frequency values, 366
post-processing, 372–373
preprocessing, 366–372
software results, 365
Axisymmetric radial element, 506

B

Banded matrix, 5
Beam
beam element, 189
cantilever beams, 216–237
definition, 189
simply supported beams, 190–216
Bio-Savart's law and field intensity, 395
Boundary conditions, 391–393
Boundary element method, 462

C

CAD model, 58
Cantilever beams, 376
analytical method, 216–217, 222–223
bending moment diagram, 232
bending stress diagram, 233
deflection values at nodes, 220, 227, 232, 238
FEM by hand calculations, 217–220, 223–226, 228–231, 234–238
maximum stress diagram, 233
nodal force calculation, 224–226, 229–231, 235–237
post-processing, 244–247
preprocessing, 240–244
reaction calculation, 226, 231, 237–238
reaction values, 221, 227, 232, 239
rotational deflection values at nodes, 220, 227, 232, 239
shear force diagram, 233
software results, 220–221, 226–227, 231–233
total values, 232
Cantilever beam, transverse vibrations
analytical solution, 340–342
FEM by hand calculations, 342–345
of software results, 345–347
Coaxial cable, 506
Coaxial waveguide, 509
Column matrix, 4
Compressible fluid, 311
Compressive forces, 65
COMSOL, 464–478, 480, 482, 484, 507, 508, 575–587
Conduction boundary conditions, two-dimensional problem, 305–306
Constant cross-section area
analytical method, 71–72, 76–78, 83, 88–89, 92–93
deflection values as node, 74–75, 81, 86, 91, 99
displacement calculation, 72, 77–78
FEM by hand calculations, 72–74, 78–82, 84–86, 89–90, 94–98
post-processing, 103–104
preprocessing, 100–103
reaction calculation, 72, 76, 86, 93
reaction value, computer generated output, 75, 82
software results, 74–76, 86–87, 91, 98–99
stress calculation, 72, 73, 76–77, 80, 85, 92–93
stress values, computer generated output, 75

Young's modulus, 71
Constitutive relations, 385–389
Convection boundary conditions, two-dimensional problem, 305–306
Coulomb's law and field intensity, 394
Cramer's rule, 21–24

D

Diagonal matrix, 4
Dielectric loss, 399
Direct approach, 55
Displacement method, 59
Double-strip shielded transmission line, 475–477
Dynamic analysis
bar, axial vibrations of, 362–373
cantilever beam, transverse vibrations of, 340–347
forcing function, bar subjected, 373–375
forcing function, fixed-fixed beam subjected to, 348–361
natural frequency determination, fixed-fixed beam for, 328–340
procedure of finite element analysis, 328

E

Eigenvalues, 36–40
Eigenvectors, 36–40
Eight-strip Line, 482–484
Electromagnetic analysis, 408–439
Electromagnetic energy and power flow, 395–399
Elementary row operations, 19
Element stiffness matrix, 158, 163
Element strains and stresses, 56
Elimination method, 57
Engineering electromagnetics analysis
automatic mesh generation, 439–443
Bio-Savart's law and field intensity, 395
boundary conditions, 391–393
constitutive relations, 385–389
Coulomb's law and field intensity, 394
electromagnetic analysis, 408–439
electromagnetic energy and power flow, 395–399
external problems, 460–464
FEM by hand calculations (*See* Hand calculations, FEM by)
higher order elements, 443–454
Lorentz force law and continuity equation, 385
loss in medium, 399–400
Maxwell's equations and continuity equation, 379–385
multistrip transmision lines, 473–484
Poison's and Laplace's equations, 401–402
potential equations, 390–391
shielded microstrip lines, modeling and simulation of, 464–473
skin depth, 400–401
solenoid actuator analysis, with ANSYS, 484–505
three-dimensional element, 454–460
wave equations, 402–407
Engineering problem
analytical method, 54
experimental method, 54
numerical method, 54
post-processing, 58
preprocessing, 58
processing, 58
Equation of continuity, 379

Equations
  of continuity, 379
  Laplace's equation, 401, 418–431
  Lorentz force law and continuity equation, 385
  Maxwell's equations and continuity equation, 379–385
  Poison's and Laplace's equations, 401–402
  Poisson's equation, 412–417
  potential equations, 390–391
  symbolic expressions, solving equations, 572–573
  wave equations, 402–407
Equilibrium, 67
Experimental method, 54

F

FEA. *See* Finite element method
Finite element method (FEA)
  dynamic problems, procedure of, 328
  fluid flow problems, procedure of, 312–313
  linear spring, direct method for, 62–63
  practical applications of, 58
  prescribing boundary conditions, 56–58
  procedure of, 54–56
  software package, 58
  solving engineering problems, 53–54
  structural problems, procedure of, 54–56
  for structure, 58–59
  thermal problems, procedure of, 274
  types of, 59–62
Fixed-fixed beam, 375
  forcing function, 348–361
  natural frequency determination, 328–340
Fluid flow analysis
  airfoil, potential flow, 316–325
  cylinder, potential flow, 313–315
  procedure of finite element analysis, 312–313
Force and elongation behaviors, 66
Force method, 59
Forcing function, bar
  displacement values, 374
  maximum displacement values, 375
  software results, 373–374
Forcing function, fixed-fixed beam
  maximum displacement values, 361
  preprocessing, 350–361
  software results, 345–347
Four-conductor transmission lines, 508

G

Gaussian elimination method, 19–21
Gauss's law, 401
Global stiffness matrix, 164
Gradient operator, 33

H

Hand calculations, FEM by
  axial vibrations, 363–365
  cantilever beams, 217–220, 223–226, 228–231, 234–238
  cantilever beam, transverse vibrations of, 342–345
  constant cross-section area, 72–74, 78–82, 84–86, 89–90, 94–98
  natural frequency determination, 330–332

simply supported beams, 191–192, 195–198, 202–205, 207–209
stepped bar, 127–130, 132–135, 141–143
truss, 162–172, 174–177
varying cross-section area, 107–111, 113–115, 117–119
Higher order elements
fundamental matrices, 450–454
local coordinates, 445–447
pascal triangle, 444–445
shape functions, 447–449
Homogeneous, 391
Hybrid method, 59

I

Identity (unit) matrix, 4–5
Incompressible fluid, 311
Infinite element method, 461–462
Inhomogeneous, 391
Integration of matrix, 14–15
Inverse of matrix, 24–27
Invertible matrix (nonsingular matrix), 26
Irrotational flow, 311

L

Laplace's equations, 401–402
Linear spring element, 62–63
Linear systems, direct methods for, 18–19
Lorentz force law and continuity equation, 385
Loss tangent, 400

M

Magnetostatic energy, 397
MATLAB, 40–49
MATLAB (Matrix Laboratory)
calculations, 557–565
differentiating symbolic expressions, 569
integrating symbolic expressions, 570
limits symbolic expressions, 570
simplifying symbolic expressions, 568
sums symbolic expressions, 571
symbolic computation, 565–568
symbolic expressions, solving equations, 572–573
Taylor series symbolic expressions, 571
windows, 555–556
Matrix
addition of, 6–7
Cramer's rule, 21–24
definition, 1–3
determinant of, 15–17
differentiation of, 14
eigenvalues and eigenvectors, 36–40
equality of, 15
Gaussian elimination method, 19–21
integration of, 14–15
inverse of, 24–27
linear systems, direct methods for, 18–19
MATLAB, 40–49
multiplication of, 8–9
multiplied by scalar, 7–8
operations, 38
rules of multiplications, 9–12
subtraction of, 6–7
trace of, 13–14
transpose of, 12–13
types of, 3–6
vector analysis, 27–36
Matrix Laboratory. *See* MATLAB
Matrix multiplications
by another matrix, 8–9
rules of, 9–12
by scalar, 7–8

Maximum bending moment, 247–248
Maxwell's equations and continuity equation
  differential form, 380
  divergence and Stokes theorems, 381–382
  integral form, 381
  quasi-statics case, 382
  space case, source-free regions of, 383
  statics case, 382–383
  time-harmonic fields case, 383–385
Mixed method, 59
Multipoint constrains method, 57–58
Multistrip transmission lines, 473–484

N

Natural frequency determination
  analytical method, 329
  FEM by hand calculations, 330–332
  post-processing, 339–340
  preprocessing, 334–339
  software results, 333–334
Nodal displacements, 56, 152, 153, 161, 168
Nodal lines, 408
Nodes, 53
Non-invertible matrix (singular matrix), 26
Nonsingular matrix, 26
Null matrix, 4
Numerical method, 54

O

One-dimensional elements, 59
  FEM standard steps procedure, 408–412
  natural coordinates, 417–418
  Poisson's equation, 412–417
  variational approach, 413–414
  weighted residuals method, 415–417
One-dimensional heat conduction problems, 273–278
  analytical method, 275
  FEM by calculations, 276–277
  post-processing, 285–290, 295–299, 305
  preprocessing, 278–285, 290–295, 300–305
  software results, 277–278, 299

P

Partitioned matrix (Super-matrix), 6
Penalty method, 57
Plot velocity distribution, 324
Poison's and Laplace's equations, 401–402
Poisson's equation, 412–417
Post-processing
  axial vibrations, 372–373
  cantilever beams, 244–247
  constant cross-section area, 103–104
  engineering problem, solving, 58
  natural frequency determination, 339–340
  simply supported beams, 214–216
  stepped bar, 151–152
  stress analysis, rectangular plate with circular hole, 268
  truss, 181–182
  varying cross-section area, 125–126
Potential equations, 390–391
Poynting's theorem, 398
Poynting vector, 397
Practical applications, 58
Preprocessing
  axial vibrations, 366–372
  cantilever beams, 240–244

constant cross-section area, 100–103
engineering problem, solving, 58
natural frequency determination, 334–339
simply supported beams, 211–214
stepped bar, 145–150
stress analysis, rectangular plate with circular hole, 262–267
truss, 179–181
Prescribing boundary conditions
elimination method, 57
multipoint constrains method, 57–58
penalty method, 57
Processing
engineering problem, solving, 58
varying cross-section area, 121–125

R

Rectangular cross-section transmission line, 467–468
Rectangular line, with diamondwise structure, 469–470
Rectangular matrix, 3
Rectangular plate with circular hole. *See* Stress analysis
Row matrix, 3

S

Scalar matrix, 4
Simply supported beams
analytical method, 190, 194–195, 201–202, 206–207
deflection values at nodes, 193, 199, 205, 210
FEM by hand calculations, 191–192, 195–198, 202–205, 207–209
nodal force calculation, 196–198
post-processing, 214–216
preprocessing, 211–214
reaction calculation, 192, 205
reaction values, 193, 199, 206
slope values at nodes, 199, 205, 210
software results, 193, 198–200, 205–206, 209–210
total values, 199, 206
Single-strip shielded transmission line, 470–473
Singular matrix, 26
Six-strip line, 480–482
Skew-symmetric matrix, 5
Skin depth, 400–401
Software package, finite element analysis, 58
Software results
airfoil, potential flow, 317
axial vibrations, 365
cantilever beams, 220–221, 226–227, 231–233
cantilever beam, transverse vibrations of, 345–347
constant cross-section area, 74–76, 86–87, 91, 98–99
forcing function, bar, 373–374
forcing function, fixed-fixed beam, 345–347
natural frequency determination, 333–334
one-dimensional heat conduction problems, 277–278, 299
simply supported beams, 193, 198–200, 205–206, 209–210
stepped bar, 135–137, 143–144
stress analysis, rectangular plate with circular hole, 257–261
truss, 172–174, 177–179
varying cross-section area, 115–116, 120–121

Solenoid actuator analysis, with ANSYS, 484–505
Square cross-section transmission line, 468–469
Square matrix, 3
Static equilibrium, 67
Stepped bar
 analytical method, 127, 131–132, 138–140
 deflection values at nodes, 129, 135–136, 143–144
 displacement calculation, 127, 128, 132
 FEM by hand calculations, 127–130, 132–135, 141–143
 nodal displacements, 126
 post-processing, 151–152
 preprocessing, 145–150
 reaction calculation, 127, 129, 134, 143
 reaction value, 130, 136
 reaction values, 144
 software results, 135–137, 143–144
 stress calculation, 127, 128, 132, 134, 142–143
 stress value at elements, 130, 136, 144
Stiffness matrix, 78, 108
Stokes theorems, 381
Strain, 66
Streamline, 312
Stress, 65
Stress analysis, rectangular plate with circular hole
 analytical method, 257–259
 boundary of hole, 255
 center of plate, 256
 deflection pattern, 261
 element options, 263
 first principal stress distribution, 261
 one-quarter of plate, 255, 256
 plate thickness, 263
 post-processing, 268
 preprocessing, 262–267
 real constants, 263
 software results, 257–261
 sub cases, 254–256
 tensile load at both edges, 254
 tensile load at one edge, 254
 validation of results, 262
 Von Mises stress distribution, 259
Stress calculation
 constant cross-section area, 72, 73, 76–77, 80, 85, 92–93
 stepped bar, 127, 128, 132, 134, 142–143
 truss, 165–166, 171–172, 177
 varying cross-section area, 106, 109, 112, 115, 119
Stress-strain behavior, 67
Structural problems
 direct approach, 55
 discretization of, 54
 element stiffness matrices and load vectors, 55–56
 element strains and stresses, 56
 nodal displacements, 56
 proper interpolation, 54–55
 variational approach, 55
 weighted residual approach, 55–56
Super-matrix, 6
Symbolic expressions, MATLAB
 differentiating symbolic expressions, 569
 integrating symbolic expressions, 570
 limits symbolic expressions, 570
 simplifying symbolic expressions, 568
 sums symbolic expressions, 571

symbolic expressions, solving equations, 572–573
taylor series symbolic expressions, 571
Symmetric matrix, 5

T

Temperature distribution, 306–310
Tensile forces, 65
Tension, 65
Thermal analysis
one-dimensional heat conduction problems, 274–278
procedure of FEM, 274
two-dimensional heat conduction problems, 274, 305–306
Three-dimensional element, 61, 454–460
Three-node bar element, 70–71
Three-node element, 507
Three-strip line, 478–480
Triangular matrix, 5–6
Truss
analytical method, 159–162
angle calculation, 162–165, 174–176
definition, 157
deflection value at nodes, 167, 172–173, 178
2-D trusses, 157
element stiffness matrix, 158
element stress, 158
FEM by hand calculation, 162–172, 174–177
nodal displacements, 158
post-processing, 181–182
preprocessing, 179–181
reaction calculation, 166
software results, 172–174, 177–179
stress calculation, 165–166, 171–172, 177
stress values of elements, 167, 173, 178–179
Two-dimensional elements, 60–61
band matrix method, 430–433, 435–439
FEM to electrostatic problems, 418–431
iteration method, 429, 433–435
Laplace's equation, 418–430
Two-dimensional problem, 305–306
Two-node bar element, 68–70

V

Variational approach, 55
Varying cross-section area
analytical method, 105–106, 112–113
deflection values at node, 110, 115, 120–121
displacement and stress, 104
displacement calculation, 106, 113
equivalent model, 107
FEM by hand calculations, 107–111, 113–115, 117–119
post-processing, 125–126
processing, 121–125
reaction calculation, 119
reaction value, 111, 121
software results, 115–116, 120–121
stress values, 110, 116, 121
Young's modulus, 104
Vector
addition and subtraction, 28–29
algebra, 29
components of, 28
Del (∇) operator, 33–36
equality, 28
multiplication, 29–31
multiplication of scalar, 29

right-hand rule, 31
unit, 28
Vector analysis, 27–36
Von Mises stress distribution, 259

W

Wave equations, 402–403
Weighted residual approach, 55

Y

Young's modulus
constant cross-section
area, 71
varying cross-section
area, 104