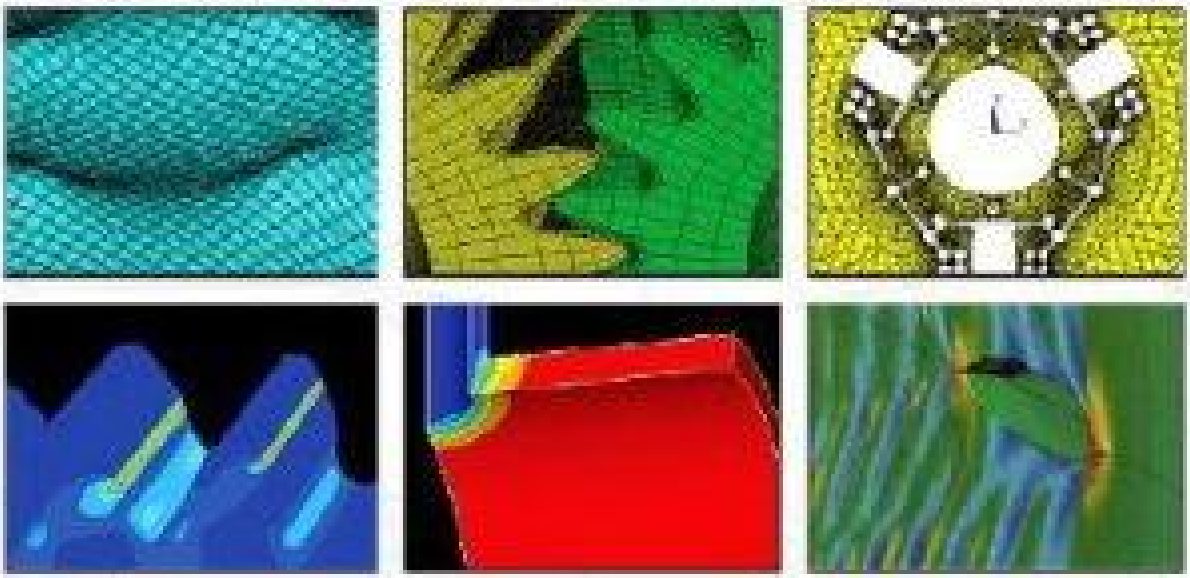
T. Stolarski • Y. Nakasone • S. Yoshimoto

ENGINEERING ANALYSIS WITH ANSYS SOFTWARE



BH

Engineering Analysis With ANSYS Software

This page intentionally left blank

Engineering Analysis With ANSYS Software

Y. Nakasone and S. Yoshimoto

Department of Mechanical Engineering
Tokyo University of Science, Tokyo, Japan

T. A. Stolarski

Department of Mechanical Engineering
School of Engineering and Design
Brunel University, Middlesex, UK

AMSTERDAM • BOSTON • HEIDELBERG • LONDON • NEW YORK • OXFORD
PARIS • SAN DIEGO • SAN FRANCISCO • SINGAPORE • SYDNEY • TOKYO

Butterworth-Heinemann is an imprint of Elsevier

Elsevier Butterworth-Heinemann
Linacre House, Jordan Hill, Oxford OX2 8DP
30 Corporate Drive, Burlington, MA 01803

First published 2006

Copyright © 2006 N. Nakasone, T. A. Stolarski and S. Yoshimoto. All rights reserved

The right of Howard D. Curtis to be identified as the author of this work has been asserted in accordance with the Copyright, Design and Patents Act 1988

No part of this publication may be reproduced in any material form (including photocopying or storing in any medium by electronic means and whether or not transiently or incidentally to some other use of this publication) without the written permission of the copyright holder except in accordance with the provisions of the Copyright, Designs and Patents Act 1988 or under the terms of a licence issued by the Copyright Licensing Agency Ltd, 90 Tottenham Court Road, London, England W1T 4LP. Applications for the copyright holder's written permission to reproduce any part of this publication should be addressed to the publisher

Permissions may be sought directly from Elsevier's Science & Technology Rights Department in Oxford, UK: phone (+44) 1865 843830, fax: (+44) 1865 853333, e-mail: permissions@elsevier.co.uk. You may also complete your request on-line via the Elsevier homepage (http://www.elsevier.com), by selecting 'Customer Support' and then 'Obtaining Permissions'

The copyrighted screen shots of the ANSYS software graphical interface that appear throughout this book are used with permission of ANSYS, Inc.

ANSYS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.

British Library Cataloguing in Publication Data
A catalogue record for this book is available from the British Library

Library of Congress Cataloguing in Publication Data
A catalogue record for this book is available from the Library of Congress

ISBN 0 7506 6875 X

For information on all Elsevier Butterworth-Heinemann publications visit our website at http://books.elsevier.com

Typeset by Charon Tec Ltd (A Macmillan Company), Chennai, India
www.charontec.com
Printed and bound by MPG Books Ltd., Bodmin, Cornwall

Working together to grow libraries in developing countries

www.elsevier.com | www.bookaid.org | www.sabre.org

ELSEVIER BOOK AID International Sabre Foundation

Contents

Preface xiii

The aims and scope of the book xv

Chapter 1

Basics of finite-element method 1

1.1 Method of weighted residuals 2
1.1.1 Sub-domain method (Finite volume method) 2
1.1.2 Galerkin method 4
1.2 Rayleigh–Ritz method 5
1.3 Finite-element method 7
1.3.1 One-element case 10
1.3.2 Three-element case 11
1.4 FEM in two-dimensional elastostatic problems 14
1.4.1 Elements of finite-element procedures in the analysis of plane elastostatic problems 15
1.4.2 Fundamental formulae in plane elastostatic problems 16
1.4.2.1 Equations of equilibrium 16
1.4.2.2 Strain–displacement relations 16
1.4.2.3 Stress–strain relations (constitutive equations) 17
1.4.2.4 Boundary conditions 19
1.4.3 Variational formulae in elastostatic problems: the principle of virtual work 21
1.4.4 Formulation of the fundamental finite-element equations in plane elastostatic problems 21
1.4.4.1 Strain–displacement matrix or [B] matrix 21
1.4.4.2 Stress–strain matrix or [D] matrix 25
1.4.4.3 Element stiffness equations 25
1.4.4.4 Global stiffness equations 27
1.4.4.5 Example: Finite-element calculations for a square plate subjected to uniaxial uniform tension 30

Bibliography 34

CHAPTER 2

OVERVIEW OF ANSYS STRUCTURE AND VISUAL CAPABILITIES 37

2.1 INTRODUCTION 37
2.2 STARTING THE PROGRAM 38
2.2.1 Preliminaries 38
2.2.2 Saving and restoring jobs 40
2.2.3 Organization of files 41
2.2.4 Printing and plotting 42
2.2.5 Exiting the program 43
2.3 PREPROCESSING STAGE 43
2.3.1 Building a model 43
2.3.1.1 Defining element types and real constants 44
2.3.1.2 Defining material properties 46
2.3.2 Construction of the model 47
2.3.2.1 Creating the model geometry 47
2.3.2.2 Applying loads 48
2.4 SOLUTION STAGE 49
2.5 POSTPROCESSING STAGE 50

CHAPTER 3

APPLICATION OF ANSYS TO STRESS ANALYSIS 51

3.1 CANTILEVER BEAM 51
3.1.1 Example problem: A cantilever beam 52
3.1.2 Problem description 53
3.1.2.1 Review of the solutions obtained by the elementary beam theory 53
3.1.3 Analytical procedures 53
3.1.3.1 Creation of an analytical model 53
3.1.3.2 Input of the elastic properties of the beam material 56
3.1.3.3 Finite-element discretization of the beam area 57
3.1.3.4 Input of boundary conditions 62
3.1.3.5 Solution procedures 71
3.1.3.6 Graphical representation of the results 73
3.1.4 Comparison of FEM results with experimental ones 76
3.1.5 Problems to solve 76
APPENDIX: PROCEDURES FOR CREATING STEPPED BEAMS 80
A3.1 Creation of a stepped beam 80
A3.1.1 How to cancel the selection of areas 81
A3.2 Creation of a stepped beam with a rounded fillet 81
A3.2.1 How to display area numbers 84

3.2 THE PRINCIPLE OF ST. VENANT 84
3.2.1 Example problem: An elastic strip subjected to distributed uniaxial tensile stress or negative pressure at one end and clamped at the other end 84
3.2.2 Problem description 85
3.2.3 Analytical procedures 85
3.2.3.1 Creation of an analytical model 85
3.2.3.2 Input of the elastic properties of the strip material 86
3.2.3.3 Finite-element discretization of the strip area 86
3.2.3.4 Input of boundary conditions 88
3.2.3.5 Solution procedures 89
3.2.3.6 Contour plot of stress 92
3.2.4 Discussion 92
3.3 STRESS CONCENTRATION DUE TO ELLIPTIC HOLES 93
3.3.1 Example problem: An elastic plate with an elliptic hole in its center subjected to uniform longitudinal tensile stress σ_o at one end and damped at the other end 93
3.3.2 Problem description 94
3.3.3 Analytical procedures 94
3.3.3.1 Creation of an analytical model 94
3.3.3.2 Input of the elastic properties of the plate material 97
3.3.3.3 Finite-element discretization of the quarter plate area 98
3.3.3.4 Input of boundary conditions 99
3.3.3.5 Solution procedures 100
3.3.3.6 Contour plot of stress 101
3.3.3.7 Observation of the variation of the longitudinal stress distribution in the ligament region 101
3.3.4 Discussion 102
3.3.5 Problems to solve 105
3.4 STRESS SINGULARITY PROBLEM 106
3.4.1 Example problem: An elastic plate with a crack of length $2a$ in its center subjected to uniform longitudinal tensile stress σ_0 at one end and clamped at the other end 106
3.4.2 Problem description 106
3.4.3 Analytical procedures 107
3.4.3.1 Creation of an analytical model 107
3.4.3.2 Input of the elastic properties of the plate material 110
3.4.3.3 Finite-element discretization of the center-cracked tension plate area 110
3.4.3.4 Input of boundary conditions 113
3.4.3.5 Solution procedures 114
3.4.3.6 Contour plot of stress 115
3.4.4 Discussion 116
3.4.5 Problems to solve 118
3.5 TWO-DIMENSIONAL CONTACT STRESS 120

3.5.1 Example problem: An elastic cylinder with a radius of length (a) pressed against a flat surface of a linearly elastic medium by a force′ 120
3.5.2 Problem description 120
3.5.3 Analytical procedures 121
3.5.3.1 Creation of an analytical model 121
3.5.3.2 Input of the elastic properties of the material for the cylinder and the flat plate 123
3.5.3.3 Finite-element discretization of the cylinder and the flat plate areas 123
3.5.3.4 Input of boundary conditions 133
3.5.3.5 Solution procedures 135
3.5.3.6 Contour plot of stress 136
3.5.4 Discussion 136
3.5.5 Problems to solve 138
REFERENCES 141

CHAPTER 4 MODE ANALYSIS 143

4.1 INTRODUCTION 143
4.2 MODE ANALYSIS OF A STRAIGHT BAR 144
4.2.1 Problem description 144
4.2.2 Analytical solution 144
4.2.3 Model for finite-element analysis 145
4.2.3.1 Element type selection 145
4.2.3.2 Real constants for beam element 147
4.2.3.3 Material properties 147
4.2.3.4 Create keypoints 149
4.2.3.5 Create a line for beam element 151
4.2.3.6 Create mesh in a line 152
4.2.3.7 Boundary conditions 154
4.2.4 Execution of the analysis 157
4.2.4.1 Definition of the type of analysis 157
4.2.4.2 Execute calculation 159
4.2.5 Postprocessing 161
4.2.5.1 Read the calculated results of the first mode of vibration 161
4.2.5.2 Plot the calculated results 161
4.2.5.3 Read the calculated results of the second and third modes of vibration 161
4.3 MODE ANALYSIS OF A SUSPENSION FOR HARD-DISC DRIVE 163
4.3.1 Problem description 163
4.3.2 Create a model for analysis 163
4.3.2.1 Element type selection 163
4.3.2.2 Real constants for beam element 165
4.3.2.3 Material properties 168

4.3.2.4 Create keypoints 168
4.3.2.5 Create areas for suspension 171
4.3.2.6 Boolean operation 175
4.3.2.7 Create mesh in areas 177
4.3.2.8 Boundary conditions 179
4.3.3 Analysis 182
4.3.3.1 Define the type of analysis 182
4.3.3.2 Execute calculation 182
4.3.4 Postprocessing 183
4.3.4.1 Read the calculated results of the first mode of vibration 183
4.3.4.2 Plot the calculated results 183
4.3.4.3 Read the calculated results of higher modes of vibration 184
4.4 MODE ANALYSIS OF A ONE-AXIS PRECISION MOVING TABLE USING ELASTIC HINGES 188
4.4.1 Problem description 188
4.4.2 Create a model for analysis 189
4.4.2.1 Select element type 189
4.4.2.2 Material properties 189
4.4.2.3 Create keypoints 192
4.4.2.4 Create areas for the table 193
4.4.2.5 Create mesh in areas 197
4.4.2.6 Boundary conditions 201
4.4.3 Analysis 205
4.4.3.1 Define the type of analysis 205
4.4.3.2 Execute calculation 208
4.4.4 Postprocessing 209
4.4.4.1 Read the calculated results of the first mode of vibration 209
4.4.4.2 Plot the calculated results 209
4.4.4.3 Read the calculated results of the second and third modes of vibration 210
4.4.4.4 Animate the vibration mode shape 211

CHAPTER 5 ANALYSIS FOR FLUID DYNAMICS 215

5.1 INTRODUCTION 215
5.2 ANALYSIS OF FLOW STRUCTURE IN A DIFFUSER 216
5.2.1 Problem description 216
5.2.2 Create a model for analysis 216
5.2.2.1 Select kind of analysis 216
5.2.2.2 Element type selection 217
5.2.2.3 Create keypoints 219
5.2.2.4 Create areas for diffuser 221

5.2.2.5 Create mesh in lines and areas 222
5.2.2.6 Boundary conditions 226
5.2.3 Execution of the analysis 231
5.2.3.1 FLOTRAN set up 231
5.2.4 Execute calculation 233
5.2.5 Postprocessing 234
5.2.5.1 Read the calculated results of the first mode of vibration 234
5.2.5.2 Plot the calculated results 234
5.2.5.3 Plot the calculated results by path operation 237
5.3 ANALYSIS OF FLOW STRUCTURE IN A CHANNEL WITH A BUTTERFLY VALVE 242
5.3.1 Problem description 242
5.3.2 Create a model for analysis 242
5.3.2.1 Select kind of analysis 242
5.3.2.2 Select element type 243
5.3.2.3 Create keypoints 243
5.3.2.4 Create areas for flow channel 245
5.3.2.5 Subtract the valve area from the channel area 245
5.3.2.6 Create mesh in lines and areas 246
5.3.2.7 Boundary conditions 248
5.3.3 Execution of the analysis 251
5.3.3.1 FLOTRAN set up 251
5.3.4 Execute calculation 253
5.3.5 Postprocessing 254
5.3.5.1 Read the calculated results 254
5.3.5.2 Plot the calculated results 255
5.3.5.3 Detailed view of the calculated flow velocity 256
5.3.5.4 Plot the calculated results by path operation 259

CHAPTER 6 APPLICATION OF ANSYS TO THERMO MECHANICS 263

6.1 GENERAL CHARACTERISTIC OF HEAT TRANSFER PROBLEMS 263
6.2 HEAT TRANSFER THROUGH TWO WALLS 265
6.2.1 Problem description 265
6.2.2 Construction of the model 265
6.2.3 Solution 276
6.2.4 Postprocessing 280
6.3 STEADY-STATE THERMAL ANALYSIS OF A PIPE INTERSECTION 285
6.3.1 Description of the problem 285
6.3.2 Preparation for model building 288
6.3.3 Construction of the model 291
6.3.4 Solution 298
6.3.5 Postprocessing stage 306
6.4 HEAT DISSIPATION THROUGH RIBBED SURFACE 312

6.4.1 Problem description 312
6.4.2 Construction of the model 313
6.4.3 Solution 321
6.4.4 Postprocessing 325

CHAPTER 7

APPLICATION OF ANSYS TO CONTACT BETWEEN MACHINE ELEMENTS 331

7.1 GENERAL CHARACTERISTICS OF CONTACT PROBLEMS 331
7.2 EXAMPLE PROBLEMS 332
7.2.1 Pin-in-hole interference fit 332
7.2.1.1 Problem description 332
7.2.1.2 Construction of the model 333
7.2.1.3 Material properties and element type 338
7.2.1.4 Meshing 339
7.2.1.5 Creation of contact pair 342
7.2.1.6 Solution 347
7.2.1.7 Postprocessing 352
7.2.2 Concave contact between cylinder and two blocks 359
7.2.2.1 Problem description 359
7.2.2.2 Model construction 360
7.2.2.3 Material properties 365
7.2.2.4 Meshing 368
7.2.2.5 Creation of contact pair 372
7.2.2.6 Solution 374
7.2.2.7 Postprocessing 379
7.2.3 Wheel-on-rail line contact 382
7.2.3.1 Problem description 382
7.2.3.2 Model construction 385
7.2.3.3 Properties of material 391
7.2.3.4 Meshing 392
7.2.3.5 Creation of contact pair 398
7.2.3.6 Solution 401
7.2.3.7 Postprocessing 404
7.2.4 O-ring assembly 410
7.2.4.1 Problem description 410
7.2.4.2 Model construction 412
7.2.4.3 Selection of materials 413
7.2.4.4 Geometry of the assembly and meshing 423
7.2.4.5 Creating contact interface 427
7.2.4.6 Solution 436
7.2.4.7 Postprocessing (first load step) 442
7.2.4.8 Solution (second load step) 444
7.2.4.9 Postprocessing (second load step) 451

INDEX 453

This page intentionally left blank

Preface

This book is very much the result of a collaboration between the three co-authors: Professors Nakasone and Yoshimoto of Tokyo University of Science, Japan and Professor Stolarski of Brunel University, United Kingdom. This collaboration started some 10 years ago and initially covered only research topics of interest to the authors. Exchange of academic staff and research students have taken place and archive papers have been published. However, being academic does not mean research only. The other important activity of any academic is to teach students on degree courses. Since the authors are involved in teaching students various aspects of finite engineering analyses using ANSYS it is only natural that the need for a textbook to aid students in solving problems with ANSYS has been identified.

The ethos of the book was worked out during a number of discussion meetings and aims to assist in learning the use of ANSYS through examples taken from engineering practice. It is hoped that the book will meet its primary aim and provide practical help to those who embark on the road leading to effective use of ANSYS in solving engineering problems.

Throughout the book, when we state "ANSYS", we are referring to the structural FEA capabilities of the various products available from ANSYS Inc. ANSYS is the original (and commonly used) name for the commercial products: ANSYS Mechanical or ANSYS Multiphysics, both general-purpose finite element analysis (FEA) computer-aided engineering (CAE) software tools developed by ANSYS, Inc. The company actually develops a complete range of CAE products, but is perhaps best known for the commercial products ANSYS Mechanical & ANSYS Multiphysics. For academic users, ANSYS Inc. provides several noncommercial versions of ANSYS Multiphysics such as ANSYS University Advanced and ANSYS University Research.

ANSYS Mechanical, ANSYS Multiphysics and the noncommercial variants commonly used in academia are self-contained analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a unified graphical user interface. These ANSYS Inc. products are general-purpose finite element modeling tools for numerically solving a wide variety of physics, such as static/dynamic structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

For more information on ANSYS products please visit the website www.ansys.com.

J. Nakasone
T. A. Stolarski
S. Yoshimoto

Tokyo, September 2006

This page intentionally left blank

The aims and scope of the book

It is true to say that in many instances the best way to learn complex behavior is by means of imitation. For instance, most of us learned to walk, talk, and throw a ball solely by imitating the actions and speech of those around us. To a considerable extent, the same approach can be adopted to learn to use ANSYS software by imitation, using the examples provided in this book. This is the essence of the philosophy and the innovative approach used in this book. The authors have attempted in this book to provide a reader with a comprehensive cross section of analysis types in a variety of engineering areas, in order to provide a broad choice of examples to be imitated in one's own work. In developing these examples, the authors' intent has been to exercise many program features and refinements. By displaying these features in an assortment of disciplines and analysis types, the authors hope to give readers the confidence to employ these program enhancements in their own ANSYS applications.

The primary aim of this book is to assist in learning the use of ANSYS software through examples taken from various areas of engineering. The content and treatment of the subject matter are considered to be most appropriate for university students studying engineering and practicing engineers who wish to learn the use of ANSYS. This book is exclusively structured around ANSYS, and no other finite-element (FE) software currently available is considered.

This book is divided into seven chapters. Chapter 1 introduces the reader to fundamental concepts of FE method. In Chapter 2 an overview of ANSYS is presented. Chapter 3 deals with sample problems concerning stress analysis. Chapter 4 contains problems from dynamics of machines area. Chapter 5 is devoted to fluid dynamics problems. Chapter 6 shows how to use ANSYS to solve problems typical for thermo-mechanics area. Finally, Chapter 7 outlines the approach, through example problems, to problems related to contact and surface mechanics.

The authors are of the opinion that the planned book is very timely as there is a considerable demand, primarily from university engineering course, for a book which could be used to teach, in a practical way, ANSYS – a premiere FE analysis computer program. Also practising engineers are increasingly using ANSYS for computer-based analyses of various systems, hence the need for a book which they could use in a self-learning mode.

The strategy used in this book, i.e. to enable readers to learn ANSYS by means of imitation, is quite unique and very much different than that in other books where ANSYS is also involved.

This page intentionally left blank

Chapter 1

Basics of finite-element method

Chapter outline

1.1 Method of weighted residuals 2
1.2 Rayleigh–Ritz method 5
1.3 Finite-element method 7
1.4 FEM in two-dimensional elastostatic problems 14
Bibliography 34

Various phenomena treated in science and engineering are often described in terms of differential equations formulated by using their continuum mechanics models. Solving differential equations under various conditions such as boundary or initial conditions leads to the understanding of the phenomena and can predict the future of the phenomena (determinism). Exact solutions for differential equations, however, are generally difficult to obtain. Numerical methods are adopted to obtain approximate solutions for differential equations. Among these numerical methods, those which approximate continua with infinite degree of freedom by a discrete body with finite degree of freedom are called "discrete analysis." Popular discrete analyses are the finite difference method, the method of weighted residuals, and the Rayleigh–Ritz method. Via these methods of discrete analysis, differential equations are reduced to simultaneous linear algebraic equations and thus can be solved numerically.

This chapter will explain first the method of weighted residuals and the Rayleigh–Ritz method which furnish a basis for the finite-element method (FEM) by taking examples of one-dimensional boundary-value problems, and then will compare

the results with those by the one-dimensional FEM in order to acquire a deeper understanding of the basis for the FEM.

1.1 Method of weighted residuals

Differential equations are generally formulated so as to be satisfied at any points which belong to regions of interest. The method of weighted residuals determines the approximate solution $\bar{u}$ to a differential equation such that the integral of the weighted error of the differential equation of the approximate function $\bar{u}$ over the region of interest is zero. In other words, this method determines the approximate solution which satisfies the differential equation of interest on average over the region of interest:

$$\begin{cases} L[u(x)] = f(x) \quad (a \le x \le b) \\ \text{BC (Boundary Conditions): } u(a) = u_a, \;\; u(b) = u_b \end{cases} \tag{1.1}$$

where L is a linear differential operator, $f(x)$ a function of x, and u_a and u_b the values of a function $u(x)$ of interest at the endpoints, or the one-dimensional boundaries of the region D. Now, let us suppose an approximate solution to the function u be

$$\bar{u}(x) = \phi_0(x) + \sum_{i=1}^{n} a_i \phi_i(x) \tag{1.2}$$

where ϕ_i are called trial functions $(i = 1, 2, \ldots, n)$ which are chosen arbitrarily as any function $\phi_0(x)$ and a_i some parameters which are computed so as to obtain a good "fit."

The substitution of $\bar{u}$ into Equation (1.1) makes the right-hand side non-zero but gives some error R:

$$L[\bar{u}(x)] - f(x) = R \tag{1.3}$$

The method of weighted residuals determines $\bar{u}$ such that the integral of the error R over the region of interest weighted by arbitrary functions w_i $(i = 1, 2, \ldots, n)$ is zero, i.e., the coefficients a_i in Equation (1.2) are determined so as to satisfy the following equation:

$$\int_D w_i R \, dv = 0 \tag{1.4}$$

where D is the region considered.

1.1.1 Sub-domain method (finite volume method)

The choice of the following weighting function brings about the sub-domain method or finite-volume method.

$$w_i(x) = \begin{cases} 1 & (\text{for } x \in D) \\ 0 & (\text{for } x \notin D) \end{cases} \tag{1.5}$$

EXAMPLE 1.1

Consider a boundary-value problem described by the following one-dimensional differential equation:

$$\begin{cases} \dfrac{d^2u}{dx^2} - u = 0 \quad (0 \le x \le 1) \\ \text{BC: } u(0) = 0 \quad u(1) = 1 \end{cases} \tag{1.6}$$

The linear operator $L[\cdot]$ and the function $f(x)$ in Equation (1.6) are defined as follows:

$$L[\cdot] \equiv \frac{d^2(\,\cdot\,)}{dx^2} \quad f(x) \equiv u(x) \tag{1.7}$$

For simplicity, let us choose the power series of x as the trial functions ϕ_i, i.e.,

$$\bar{u}(x) = \sum_{i=0}^{n+1} c_i x^i \tag{1.8}$$

For satisfying the required boundary conditions,

$$c_0 = 0, \quad \sum_{i=1}^{n+1} c_i = 1 \tag{1.9}$$

so that

$$\bar{u}(x) = x + \sum_{i=1}^{n} A_i (x^{i+1} - x) \tag{1.10}$$

If the second term of the right-hand side of Equation (1.10) is chosen as a first-order approximate solution

$$\bar{u}_1(x) = x + A_1(x^2 - x) \tag{1.11}$$

the error or residual is obtained as

$$R = \frac{d^2\bar{u}}{dx^2} - \bar{u} = -A_1x^2 + (A_1 - 1)x + 2A_1 \neq 0 \tag{1.12}$$

$$\int_0^1 w_i R\, dx = \int_0^1 1 - \left[-A_1x^2 + (A_1 - 1)x + 2A_1\right] dx = \frac{13}{6}A_1 - \frac{1}{2} = 0 \tag{1.13}$$

Consequently, the first-order approximate solution is obtained as

$$\bar{u}_1(x) = x + \frac{3}{13}x(x - 1) \tag{1.14}$$

(Example 1.1 continued)

which agrees well with the exact solution

$$u(x) = \frac{e^x - e^{-x}}{e - e^{-1}} \tag{1.15}$$

as shown by the dotted and the solid lines in Figure 1.1.

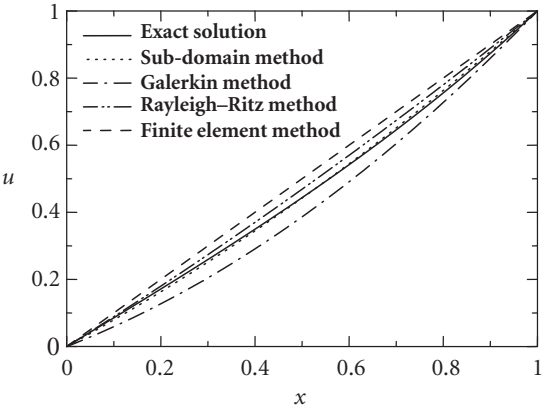


Figure 1.1 Comparison of the results obtained by various kinds of discrete analyses.

1.1.2 GALERKIN METHOD

When the weighting function w_i in Equation (1.4) is chosen equal to the trial function ϕ_i, the method is called the Galerkin method, i.e.,

$$w_i(x) = \phi_i(x) \quad (i = 1, 2, \ldots, n) \tag{1.16}$$

and thus Equation (1.4) is changed to

$$\int_D \phi_i R \, dv = 0 \tag{1.17}$$

This method determines the coefficients a_i by directly using Equation (1.17) or by integrating it by parts.

EXAMPLE 1.2

Let us solve the same boundary-value problem as described by Equation (1.6) in the preceding Section 1.1.1 by the Galerkin method.

The trial function ϕ_i is chosen as the weighting function w_i in order to find the first-order approximate solution:

$$w_1(x) = \phi_1(x) = x(x - 1) \tag{1.18}$$

(Example 1.2 continued)

Integrating Equation (1.4) by parts,

$$\int_0^1 w_i R\,dx = \int_0^1 w_i \left(\frac{d^2\bar{u}}{dx^2} - \bar{u}\right) dx = \left[w_i \frac{d\bar{u}}{dx}\right]_0^1 - \int_0^1 \frac{dw_i}{dx}\frac{d\bar{u}}{dx}dx - \int_0^1 w_i \bar{u}\,dx = 0 \tag{1.19}$$

is obtained. Choosing $\bar{u}_1$ in Equation (1.11) as the approximate solution $\bar{u}$, the substitution of Equation (1.18) into (1.19) gives

$$\int_0^1 \phi_1 R\,dx = -\int_0^1 \frac{d\phi_1}{dx}\frac{d\bar{u}}{dx}dx - \int_0^1 \phi_1 \bar{u}\,dx = -\int_0^1 (2x-1)[1 + A_1(2x-1)]dx$$
$$-\int_0^1 (x^2 - x)[1 + A_1(x^2 - x)]dx = -\frac{A_1}{3} + \frac{1}{12} - \frac{A_1}{30} = 0 \tag{1.20}$$

Thus, the following approximate solution is obtained:

$$\bar{u}_1(x) = x + \frac{5}{22}x(x-1) \tag{1.21}$$

Figure 1.1 shows that the approximate solution obtained by the Galerkin method also agrees well with the exact solution throughout the region of interest.

1.2 Rayleigh–Ritz method

When there exists the functional which is equivalent to a given differential equation, the Rayleigh–Ritz method can be used.

Let us consider the example problem illustrated in Figure 1.2 where a particle having a mass of M slides from point P_0 to lower point P_1 along a curve in a vertical plane under the force of gravity. The time t that the particle needs for sliding from the points P_0 to P_1 varies with the shape of the curve denoted by $y(x)$ which connects the two points. Namely, the time t can be considered as a kind of function $t = F[y]$ which is determined by a function $y(x)$ of an independent variable x. The function

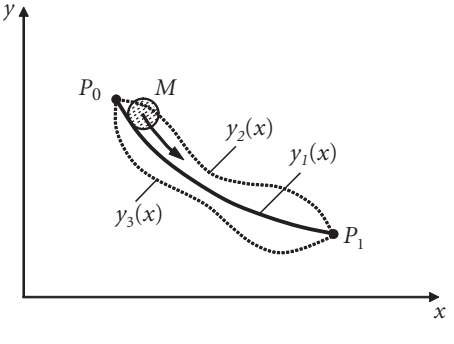


Figure 1.2 Particle M sliding from point P_0 to lower point P_1 under gravitational force.

of a function $F[y]$ is called a functional. The method for determining the maximum or the minimum of a given functional is called the variational method. In the case of Figure 1.2, the method determines the shape of the curve $y(x)$ which gives the possible minimum time $t_{\min}$ in which the particle slides from P_0 to P_1.

The principle of the virtual work or the minimum potential energy in the field of the solid mechanics is one of the variational principles which guarantee the existence of the function which makes the functional minimum or maximum. For unsteady thermal conductivity problems and viscous flow problems, no variational principle can be established; in such a case, the method of weighting residuals can be adopted instead.

Now, let $\Pi[u]$ be the functional which is equivalent to the differential equation in Equation (1.1). The Rayleigh–Ritz method assumes that an approximate solution $\bar{u}(x)$ of $u(x)$ is a linear combination of trial functions ϕ_i as shown in the following equation:

$$\bar{u}(x) = \sum_{i=1}^{n} a_i \phi_i(x) \tag{1.22}$$

where a_i $(i = 1, 2, \ldots, n)$ are arbitrary constants, ϕ_i are C^0-class functions which have continuous first-order derivatives for $a \leq x \leq b$ and are chosen such that the following boundary conditions are satisfied:

$$\sum_{i=1}^{n} a_i \phi_i(a) = u_a \quad \sum_{i=1}^{n} a_i \phi_i(b) = u_b \tag{1.23}$$

The approximate solution $\bar{u}(x)$ in Equation (1.22) is the function which makes the functional $\Pi[u]$ take stationary value and is called the admissible function.

Next, integrating the functional Π after substituting Equation (1.22) into the functional, the constants a_i are determined by the stationary conditions:

$$\frac{\partial \Pi}{\partial a_i} = 0 \quad (i = 1, 2, \ldots, n) \tag{1.24}$$

The Rayleigh–Ritz method determines the approximate solution $\bar{u}(x)$ by substituting the constants a_i into Equation (1.22). It is generally understood to be a method which determines the coefficients a_i so as to make the distance between the approximate solution $\bar{u}(x)$ and the exact one $u(x)$ minimum.

EXAMPLE 1.3

Let us solve again the boundary-value problem described by Equation (1.6) by the Rayleigh–Ritz method. The functional equivalent to the first equation of Equation (1.6) is written as

$$\Pi[u] = \int_0^1 \left[\frac{1}{2} \left(\frac{du}{dx} \right)^2 + \frac{1}{2} u^2 \right] dx \tag{1.25}$$

Equation (1.25) is obtained by intuition, but Equation (1.25) is shown to really give the functional of the first equation of Equation (1.6) as follows: first, let us take the first variation of Equation (1.25) in order to obtain the stationary value of the

equation:

$$\delta\Pi = \int_0^1 \left[\frac{du}{dx}\delta\left(\frac{du}{dx}\right) + u\delta u \right] dx \tag{1.26}$$

Then, integrating the above equation by parts, we have

$$\begin{aligned}\delta\Pi &= \int_0^1 \left(\frac{du}{dx}\frac{d\delta u}{dx} + u\delta u \right) dx = \left[\frac{du}{dx}\delta u \right]_0^1 - \int_0^1 \left[\frac{d}{dx}\left(\frac{du}{dx}\right)\delta u - u\delta u \right] dx \\ &= -\int_0^1 \left(\frac{d^2u}{dx^2} - u \right) \delta u \, dx \end{aligned} \tag{1.27}$$

For satisfying the stationary condition that $\delta\Pi = 0$, the rightmost-hand side of Equation (1.27) should be identically zero over the interval considered ($a \le x \le b$), so that

$$\frac{d^2u}{dx^2} - u = 0 \tag{1.28}$$

This is exactly the same as the first equation of Equation (1.6).

Now, let us consider the following first-order approximate solution $\bar{u}_1$ which satisfies the boundary conditions:

$$\bar{u}_1(x) = x + a_1 x(x-1) \tag{1.29}$$

Substitution of Equation (1.29) into (1.25) and integration of Equation (1.25) lead to

$$\Pi[\bar{u}_1] = \int_0^1 \left\{ \frac{1}{2}[1 + a_1(2x-1)]^2 + \frac{1}{2}\left[x + a_1(x^2 - x)\right]^2 \right\} dx = \frac{2}{3} - \frac{1}{12}a_1 + \frac{1}{3}a_1^2 \tag{1.30}$$

Since the stationary condition for Equation (1.30) is written by

$$\frac{\partial\Pi}{\partial a_1} = -\frac{1}{12} + \frac{2}{3}a_1 = 0 \tag{1.31}$$

the first-order approximate solution can be obtained as follows:

$$\bar{u}_1(x) = x + \frac{1}{8}x(x-1) \tag{1.32}$$

Figure 1.1 shows that the approximate solution obtained by the Rayleigh–Ritz method agrees well with the exact solution throughout the region considered.

1.3 Finite-element method

There are two ways for the formulation of the FEM: one is based on the direct variational method (such as the Rayleigh–Ritz method) and the other on the method of weighted residuals (such as the Galerkin method). In the formulation based on the variational method, the fundamental equations are derived from the stationary conditions of the functional for the boundary-value problems. This formulation has an advantage that the process of deriving functionals is not necessary, so it is

easy to formulate the FEM based on the method of the weighted residuals. In the formulation based on the variational method, however, it is generally difficult to derive the functional except for the case where the variational principles are already established as in the case of the principle of the virtual work or the principle of the minimum potential energy in the field of the solid mechanics.

This section will explain how to derive the fundamental equations for the FEM based on the Galerkin method.

Let us consider again the boundary-value problem stated by Equation (1.1):

$$\begin{cases} L[u(x)] = f(x) \quad (a \le x \le b) \\ \text{BC (Boundary Conditions): } u(a) = u_a \quad u(b) = u_b \end{cases} \tag{1.33}$$

First, divide the region of interest ($a \le x \le b$) into n subregions as illustrated in Figure 1.3. These subregions are called "elements" in the FEM.

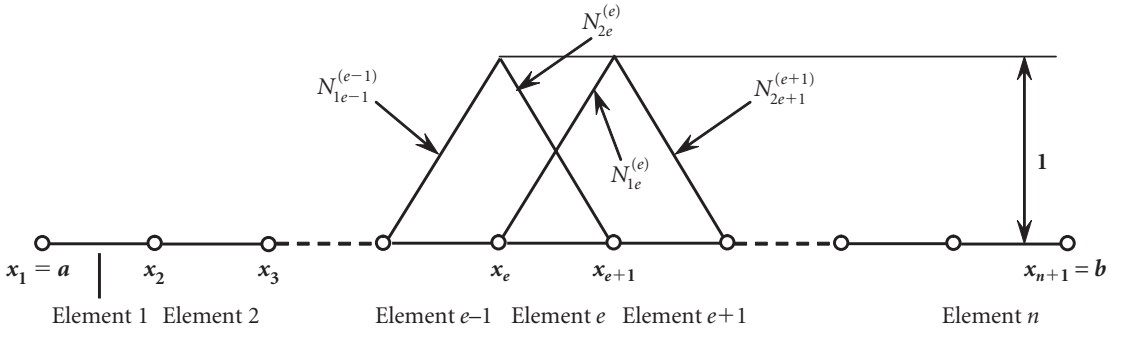


Figure 1.3 Discretization of the domain to analyze by finite elements and their interpolation functions.

Now, let us assume that an approximate solution $\bar{u}$ of u can be expressed by a piecewise linear function which forms a straight line in each subregion, i.e.,

$$\bar{u}(x) = \sum_{i=1}^{n+1} u_i N_i(x) \tag{1.34}$$

where u_i represents the value of u in element "e" at a boundary point, or a nodal point "i" between two one-dimensional elements. Functions $N_i(x)$ are the following piecewise linear functions and are called interpolation or shape functions of the nodal point "i:"

$$\begin{cases} N_{1e}^{(e)} = \dfrac{x_{e+1} - x}{x_{e+1} - x_e} = \dfrac{x_{2e} - x}{x_{2e} - x_{1e}} = \dfrac{h^{(e)} - \xi}{h^{(e)}} \\ N_{2e}^{(e)} = \dfrac{x - x_e}{x_{e+1} - x_e} = \dfrac{x - x_{1e}}{x_{2e} - x_{1e}} = \dfrac{\xi}{h^{(e)}} \end{cases} \tag{1.35}$$

where e ($e = 1, 2, \ldots, n$) denotes the element number, x_i the global coordinate of the nodal point i ($i = 1, \ldots, e-1, e, \ldots n, n+1$), $N_{ie}^{(e)}$ the value of the interpolation function at the nodal point i_e ($i_e = 1_e, 2_e$) which belongs to the eth element, 1_e and 2_e

the number of two nodal points of the eth element. Symbol ξ is the local coordinate of an arbitrary point in the eth element, $\xi = x - x_e = x - x_{1e}$ $(0 \le \xi \le h^{(e)})$, $h^{(e)}$ is the length of the eth element, and $h^{(e)}$ is expressed as $h^{(e)} = x_{e+1} - x_e = x_{2e} - x_{1e}$.

As the interpolation function, the piecewise linear or quadric function is often used. Generally speaking, the quadric interpolation function gives better solutions than the linear one.

The Galerkin method-based FEM adopts the weighting functions $w_i(x)$ equal to the interpolation functions $N_i(x)$, i.e.,

$$w_i(x) = N_i(x) \quad (i = 1, 2, \ldots, n+1) \tag{1.36}$$

Thus, Equation (1.4) becomes

$$\int_D N_i R \, dv = 0 \tag{1.37}$$

In the FEM, a set of simultaneous algebraic equations for unknown variables of $u(x)$ at the ith nodal point u_i and those of its derivatives du/dx, $(du/dx)_i$ are derived by integrating Equation (1.37) by parts and then by taking boundary conditions into consideration. The simultaneous equations can be easily solved by digital computers to determine the unknown variables u_i and $(du/dx)_i$ at all the nodal points in the region considered.

EXAMPLE 1.4

Let us solve the boundary-value problem stated in Equation (1.6) by FEM.
First, the integration of Equation (1.37) by parts gives

$$\int_0^1 w_i R \, dx = \int_0^1 w_i \left(\frac{d^2 \bar{u}}{dx^2} - \bar{u} \right) dx = \left[w_i \frac{d\bar{u}}{dx} \right]_0^1 - \int_0^1 \left(\frac{dw_i}{dx} \frac{d\bar{u}}{dx} + w_i \bar{u} \right) dx = 0$$
$$(i = 1, 2, \ldots, n+1) \tag{1.38}$$

Then, the substitution of Equations (1.34) and (1.36) into Equation (1.38) gives

$$\sum_{j=1}^{n+1} \int_0^1 \left(\frac{dN_i}{dx} \frac{dN_j}{dx} + N_i N_j \right) u_j dx - \left[N_i \frac{d\bar{u}}{dx} \right]_0^1 = 0 \quad (i = 1, 2, \ldots, n+1) \tag{1.39}$$

Equation (1.39) is a set of simultaneous linear algebraic equations composed of $(n+1)$ nodal values u_i of the solution u and also $(n+1)$ nodal values $(du/dx)_i$ of its derivative du/dx. The matrix notation of the simultaneous equations above is written in a simpler form as follows:

$$[K_{ij}]\{u_j\} = \{f_i\} \tag{1.40}$$

where $[K_{ij}]$ is a square matrix of $(n+1)$ by $(n+1)$, $\{f_i\}$ is a column vector of $(n+1)$ by 1, and the components of the matrix and the vector K_{ij} and f_i are expressed as

$$\begin{cases} K_{ij} \equiv \displaystyle\int_0^1 \left(\frac{dN_i}{dx} \frac{dN_j}{dx} + N_i N_j \right) dx & (1 \le i, j \le n+1) \\ f_i \equiv \left[N_i \dfrac{d\bar{u}}{dx} \right]_0^1 & (1 \le i \le n+1) \end{cases} \tag{1.41}$$

1.3.1 One-element case

As the first example, let us compute Equation (1.37) by regarding the whole region as one finite element as shown in Examples 1.1 through 1.3. From Equations (1.34) and (1.35), since $x_1 = 0$ and $x_2 = 1$, the approximate solution $\bar{u}$ and the interpolation functions N_i $(i = 1, 2)$ become

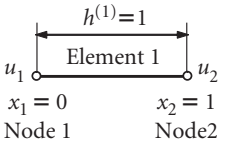


Figure 1.4 One-element model of one-dimensional FEM.

$$\bar{u}(x) = u_1 N_1 + u_2 N_2 \tag{1.42}$$

$$\begin{cases} N_1 = N_{11}^{(1)} = \dfrac{x_2 - x}{x_2 - x_1} = 1 - x \\ N_2 = N_{21}^{(1)} = \dfrac{x - x_1}{x_2 - x_1} = x \end{cases} \tag{1.43}$$

Thus, from Equation (1.41),

$$K_{ij} \equiv K_{ij}^{(1)} \equiv \int_0^1 \left(\frac{dN_i}{dx}\frac{dN_j}{dx} + N_i N_j \right) dx = \begin{cases} 4/3 & (i = j) \\ -5/6 & (i \neq j) \end{cases}$$

$$f_i \equiv \left[N_i \frac{d\bar{u}}{dx} \right]_0^1 = \begin{cases} -\left.\dfrac{d\bar{u}}{dx}\right|_{x=0} \\ \left.\dfrac{d\bar{u}}{dx}\right|_{x=1} \end{cases} \tag{1.44}$$

The global simultaneous equations are obtained as

$$\begin{bmatrix} \frac{4}{3} & -\frac{5}{6} \\ -\frac{5}{6} & \frac{4}{3} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} -\left.\dfrac{d\bar{u}}{dx}\right|_{x=0} \\ \left.\dfrac{d\bar{u}}{dx}\right|_{x=1} \end{Bmatrix} \tag{1.45}$$

According to the boundary conditions, $u_1 = 0$ and $u_2 = 1$ in the left-hand side of the above equations are known variables, whereas $(du/dx)_{x=0}$ and $(du/dx)_{x=1}$ in the left-hand side are unknown variables. The substitution of the boundary conditions into Equation (1.45) directly gives the nodal values of the approximate solution, i.e.,

$$\begin{cases} \left.\dfrac{d\bar{u}}{dx}\right|_{x=0} = 0.8333 \\ \left.\dfrac{d\bar{u}}{dx}\right|_{x=1} = 1.3333 \end{cases} \tag{1.46}$$

which agrees well with those of the exact solution

$$\begin{cases} \left.\dfrac{du}{dx}\right|_{x=0} = \dfrac{2}{e-e^{-1}} = 0.8509 \\ \left.\dfrac{du}{dx}\right|_{x=1} = \dfrac{e+e^{-1}}{e-e^{-1}} = 1.3130 \end{cases} \tag{1.47}$$

The approximate solution in this example is determined as

$$\bar{u}(x) = x \tag{1.48}$$

and agrees well with the exact solution throughout the whole region of interest as depicted in Figure 1.1.

1.3.2 Three-element case

In this section, let us compute the approximate solution $\bar{u}$ by dividing the whole region considered into three subregions having the same length as shown in Figure 1.5. From Equations (1.34) and (1.35), the approximate solution $\bar{u}$ and the interpolation functions N_i $(i = 1, 2)$ are written as

$$\bar{u}(x) = \sum_{i=1}^{4} u_i N_i \tag{1.49}$$

$$\begin{cases} N_{1e} = \dfrac{x_{2e} - x}{x_{2e} - x_{1e}} = \dfrac{h^{(e)} - \xi}{h^{(e)}} \\ N_{2e} = \dfrac{x - x_{1e}}{x_{2e} - x_{1e}} = \dfrac{\xi}{h^{(e)}} \end{cases} \tag{1.50}$$

where $h^{(e)} = 1/3$ and $0 \le \xi \le 1/3$ $(e = 1, 2, 3)$.

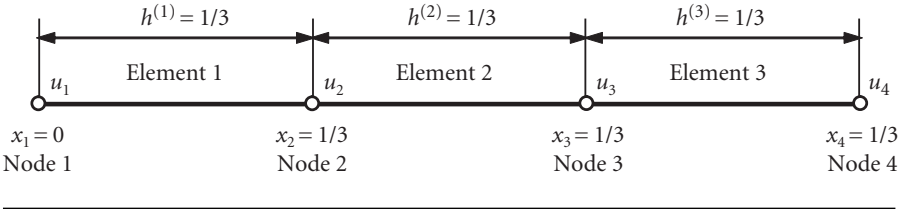


Figure 1.5 Three-element model of a one-dimensional FEM.

By calculating all the components of the K-matrix in Equation (1.41), the following equation is obtained:

$$K_{ij}^{(e)} \equiv \int_0^1 \left(\frac{dN_i^{(e)}}{dx} \frac{dN_j^{(e)}}{dx} + N_i^{(e)} N_j^{(e)} \right) dx$$

$$= \begin{cases} \dfrac{1}{h^{(e)}} + \dfrac{h^{(e)}}{3} = \dfrac{28}{9} & (i = j \text{ and } i, j = 1e, 2e) \\ -\dfrac{1}{h^{(e)}} + \dfrac{h^{(e)}}{6} = -\dfrac{53}{18} & (i \neq j \text{ and } i, j = 1e, 2e) \\ 0 & (i, j \neq 1e, 2e) \end{cases} \tag{1.51a}$$

The components relating to the first derivative of the function u in Equation (1.41) are calculated as follows:

$$f_i \equiv \left[N_i \frac{d\bar{u}}{dx} \right]_0^1 = \begin{cases} -\left. \dfrac{d\bar{u}}{dx} \right|_{x=0} & (i = 1) \\ 0 & (i = 2, 3) \\ \left. \dfrac{d\bar{u}}{dx} \right|_{x=1} & (i = 4) \end{cases} \tag{1.51b}$$

The coefficient matrix in Equation (1.51a) calculated for each element is called "element matrix" and the components of the matrix are obtained as follows:

$$\left[K_{ij}^{(1)} \right] = \begin{bmatrix} \dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{3} & -\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & 0 & 0 \\ -\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & \dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{3} & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \tag{1.52a}$$

$$\left[K_{ij}^{(2)} \right] = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & \dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{3} & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & 0 \\ 0 & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & \dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{3} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \tag{1.52b}$$

$$\left[K_{ij}^{(3)} \right] = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & \dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{3} & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} \\ 0 & 0 & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} & \dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{3} \end{bmatrix} \tag{1.52c}$$

From Equations (1.52a) through (1.52c) above, it is concluded that only the components of the element matrix relating to the nodal points which belong to the corresponding element are non-zero and that the other components are zero. Namely, for example, element 2 is composed of nodal points 2 and 3 and among the components of the element matrix only $K_{22}^{(2)}$, $K_{23}^{(2)}$, $K_{32}^{(2)}$, and $K_{33}^{(2)}$ are non-zero and the other components are zero. The superscript (2) of the element matrix components above indicates that the components are computed in element 2, and the subscripts indicate that the components are calculated for nodal points 2 and 3 in element 2.

A matrix which relates all the known and the unknown variables for the problem concerned is called the global matrix. The global matrix can be obtained simply by summing up Equations (1.52a) through (1.52c) as follows:

$$
\begin{aligned}
\left[K_{ij}\right] &= \left[\sum_{e=1}^{3} K_{ij}^{(e)}\right] \\
&= \begin{bmatrix}
\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{3} & -\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & 0 & 0 \\
-\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & \displaystyle\sum_{e=1}^{2}\left(\frac{1}{h^{(e)}} + \frac{h^{(e)}}{3}\right) & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & 0 \\
0 & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & \displaystyle\sum_{e=2}^{3}\left(\frac{1}{h^{(e)}} + \frac{h^{(e)}}{3}\right) & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} \\
0 & 0 & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} & \dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{3}
\end{bmatrix}
\end{aligned}
\tag{1.53}
$$

Consequently, the global simultaneous equation becomes

$$
\begin{bmatrix}
\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{3} & -\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & 0 & 0 \\
-\dfrac{1}{h^{(1)}} + \dfrac{h^{(1)}}{6} & \displaystyle\sum_{e=1}^{2}\left(\frac{1}{h^{(e)}} + \frac{h^{(e)}}{3}\right) & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & 0 \\
0 & -\dfrac{1}{h^{(2)}} + \dfrac{h^{(2)}}{6} & \displaystyle\sum_{e=2}^{3}\left(\frac{1}{h^{(e)}} + \frac{h^{(e)}}{3}\right) & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} \\
0 & 0 & -\dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{6} & \dfrac{1}{h^{(3)}} + \dfrac{h^{(3)}}{3}
\end{bmatrix}
\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix}
$$

$$
= \begin{Bmatrix}
-\left.\dfrac{d\bar{u}}{dx}\right|_{x=0} \\
0 \\
0 \\
\left.\dfrac{d\bar{u}}{dx}\right|_{x=1}
\end{Bmatrix}
\tag{1.54}
$$

Note that the coefficient matrix $[K_{ij}]$ in the left-hand side of Equation (1.54) is symmetric with respect to the non-diagonal components ($i \neq j$), i.e., $K_{ij} = K_{ji}$. Only the components in the band region around the diagonal of the matrix are non-zero and the others are zero. Due to this nature, the coefficient matrix is called the sparse or band matrix.

From the boundary conditions, the values of u_1 and u_4 in the left-hand side of Equation (1.54) are known, i.e., $u_1 = 0$ and $u_4 = 1$ and, from Equation (1.51b), the values of f_2 and f_3 in the right-hand side are also known, i.e., $f_2 = 0$ and $f_3 = 0$. On the other hand, u_2 and u_3 in the left-hand side and $\left.\frac{d\bar{u}}{dx}\right|_{x=0}$ and $\left.\frac{d\bar{u}}{dx}\right|_{x=1}$ in the right-hand side are unknown variables. By changing unknown variables $\left.\frac{d\bar{u}}{dx}\right|_{x=0}$ and $\left.\frac{d\bar{u}}{dx}\right|_{x=1}$ with the first and the fourth components of the vector in the left-hand side of Equation (1.54) and by substituting $h^{(1)} = h^{(2)} = h^{(3)} = 1/3$ into Equation (1.54), after rearrangement of the equation, the global simultaneous equation is rewritten as follows:

$$\begin{bmatrix} -1 & -\frac{53}{18} & 0 & 0 \\ 0 & \frac{56}{9} & -\frac{53}{18} & 0 \\ 0 & -\frac{53}{18} & \frac{56}{9} & 0 \\ 0 & 0 & -\frac{53}{18} & -1 \end{bmatrix} \begin{Bmatrix} \left.\frac{d\bar{u}}{dx}\right|_{x=0} \\ u_2 \\ u_3 \\ \left.\frac{d\bar{u}}{dx}\right|_{x=1} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ \frac{53}{18} \\ -\frac{28}{9} \end{Bmatrix} \tag{1.55}$$

where the new vector in the left-hand side of the equation is an unknown vector and the one in the right-hand side is a known vector.

After solving Equation (1.55), it is found that $u_2 = 0.2885$, $u_3 = 0.6098$, $\left.\frac{d\bar{u}}{dx}\right|_{x=0} = 0.8496$, and $\left.\frac{d\bar{u}}{dx}\right|_{x=1} = 1.3157$. The exact solutions for u_2 and u_3 can be calculated as $u_2 = 0.2889$ and $u_3 = 0.6102$ from Equation (1.55). The relative errors for u_2 and u_3 are as small as 0.1% and 0.06%, respectively. The calculated values of the derivative $\left.\frac{d\bar{u}}{dx}\right|_{x=0}$ and $\left.\frac{d\bar{u}}{dx}\right|_{x=1}$ are improved when compared to those by the one-element FEM described in Section 1.3.1.

In this section, only one-dimensional FEM was described. The FEM can be applied to two- and three-dimensional continuum problems of various kinds which are described in terms of ordinary and partial differential equations. There is no essential difference between the formulation for one-dimensional problems and the formulations for higher dimensions except for the intricacy of formulation.

1.4 FEM IN TWO-DIMENSIONAL ELASTOSTATIC PROBLEMS

Generally speaking, elasticity problems are reduced to solving the partial differential equations known as the equilibrium equations together with the stress–strain relations or the constitutive equations, the strain–displacement relations, and the

compatibility equation under given boundary conditions. The exact solutions can be obtained in quite limited cases only and in general cannot be solved in closed forms. In order to overcome these difficulties, the FEM has been developed as one of the powerful numerical methods to obtain approximate solutions for various kinds of elasticity problems. The FEM assumes an object of analysis as an aggregate of elements having arbitrary shapes and finite sizes (called finite element), approximates partial differential equations by simultaneous algebraic equations, and numerically solves various elasticity problems. Finite elements take the form of line segment in one-dimensional problems as shown in the preceding section, triangle or rectangle in two-dimensional problems, and tetrahedron, cuboid, or prism in three-dimensional problems. Since the procedure of the FEM is mathematically based on the variational method, it can be applied not only to elasticity problems of structures but also to various problems related to thermodynamics, fluid dynamics, and vibrations which are described by partial differential equations.

1.4.1 Elements of finite-element procedures in the analysis of plane elastostatic problems

Limited to static (without time variation) elasticity problems, the procedure described in the preceding section is essentially the same as that of the stress analyses by the FEM.

The procedure is summarized as follows:

- Procedure 1: *Discretization* Divide the object of analysis into a finite number of finite elements.
- Procedure 2: *Selection of the interpolation function* Select the element type or the interpolation function which approximates displacements and strains in each finite element.
- Procedure 3: *Derivation of element stiffness matrices* Determine the element stiffness matrix which relates forces and displacements in each element.
- Procedure 4: *Assembly of stiffness matrices into the global stiffness matrix* Assemble the element stiffness matrices into the global stiffness matrix which relates forces and displacements in the whole elastic body to be analyzed.
- Procedure 5: *Rearrangement of the global stiffness matrix* Substitute prescribed applied forces (mechanical boundary conditions) and displacements (geometrical boundary conditions) into the global stiffness matrix, and rearrange the matrix by collecting unknown variables for forces and displacements, say in the left-hand side, and known values of the forces and displacements in the right-hand side in order to set up simultaneous equations.
- Procedure 6: *Derivation of unknown forces and displacements* Solve the simultaneous equations set up in Procedure 5 above to solve the unknown variables for forces and displacements. The solutions for unknown forces are reaction forces and those for unknown displacements are deformations of the elastic body of interest for given geometrical and mechanical boundary conditions, respectively.
- Procedure 7: *Computation of strains and stresses* Compute the strains and stresses from the displacements obtained in Procedure 6 by using the strain–displacement relations and the stress–strain relations explained later.

1.4.2 Fundamental formulae in plane elastostatic problems

1.4.2.1 Equations of equilibrium

Consider the static equilibrium state of an infinitesimally small rectangle with its sides parallel to the coordinate axes in a two-dimensional elastic body as shown in Figure 1.6. If the body forces F_x and F_y act in the directions of the x- and the y-axes, respectively, the equations of equilibrium in the elastic body can be derived as follows:

$$\begin{cases} \dfrac{\partial \sigma_x}{\partial x} + \dfrac{\partial \tau_{xy}}{\partial y} + F_x = 0 \\ \dfrac{\partial \tau_{yx}}{\partial x} + \dfrac{\partial \sigma_y}{\partial y} + F_y = 0 \end{cases} \tag{1.56}$$

where σ_x and σ_y are normal stresses in the x- and the y-axes, respectively, with τ_{xy} and τ_{yx} shear stresses acting in the x–y plane. The shear stresses τ_{xy} and τ_{yx} are generally equal to each other due to the rotational equilibrium of the two-dimensional elastic body around its center of gravity.

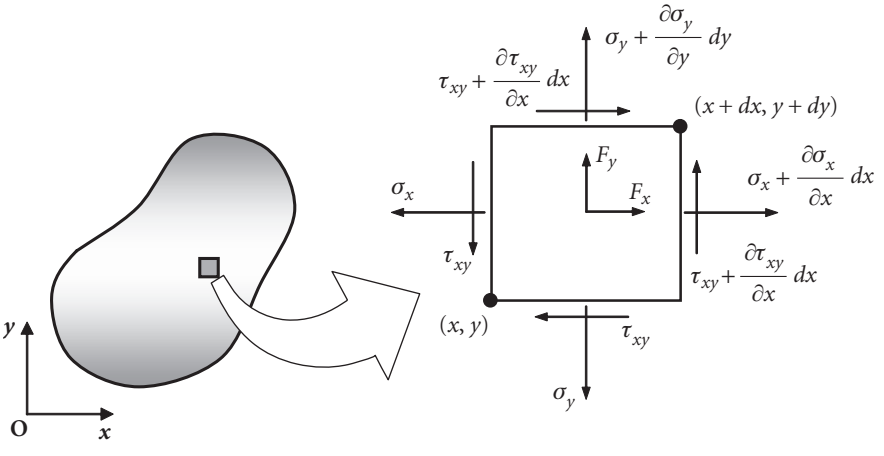


Figure 1.6 Stress states in an infinitesimal element of a two-dimensional elastic body.

1.4.2.2 Strain–displacement relations

If the deformation of a two-dimensional elastic body is infinitesimally small under the applied load, the normal strains ε_x and ε_y in the directions of the x- and the y-axes, respectively, and the engineering shearing strain γ_{xy} in the x–y plane are expressed by

the following equations:

$$\begin{cases} \varepsilon_x = \dfrac{\partial u}{\partial x} \\ \varepsilon_y = \dfrac{\partial v}{\partial y} \\ \gamma_{xy} = \dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y} \end{cases} \tag{1.57}$$

where u and v are infinitesimal displacements in the directions of the x- and the y-axes, respectively.

1.4.2.3 STRESS–STRAIN RELATIONS (CONSTITUTIVE EQUATIONS)

The stress–strain relations describe states of deformation, strains induced by the internal forces, or stresses resisting against applied loads. Unlike the other fundamental equations shown in Equations (1.56) and (1.57) which can be determined mechanistically or geometrically, these relations depend on the properties of the material, and they are determined experimentally and often called constitutive relations or constitutive equations. One of the most popular relations is the generalized Hooke's law which relates six components of the three-dimensional stress tensor with those of strain tensor through the following simple linear expressions:

$$\begin{cases} \sigma_x = \dfrac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_x \\ \sigma_y = \dfrac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_y \\ \sigma_y = \dfrac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_z \\ \tau_{xy} = G\gamma_{xy} = \dfrac{E}{2(1+\nu)}\gamma_{xy} \\ \tau_{yz} = G\gamma_{yz} = \dfrac{E}{2(1+\nu)}\gamma_{yz} \\ \tau_{zx} = G\gamma_{zx} = \dfrac{E}{2(1+\nu)}\gamma_{zx} \end{cases} \tag{1.58a}$$

or inversely

$$\begin{cases} \varepsilon_x = \dfrac{1}{E}\left[\sigma_x - \nu\left(\sigma_y + \sigma_z\right)\right] \\ \varepsilon_y = \dfrac{1}{E}\left[\sigma_y - \nu\left(\sigma_z + \sigma_x\right)\right] \\ \varepsilon_z = \dfrac{1}{E}\left[\sigma_z - \nu\left(\sigma_x + \sigma_y\right)\right] \\ \gamma_{xy} = \dfrac{\tau_{xy}}{G} \\ \gamma_{yz} = \dfrac{\tau_{yz}}{G} \\ \gamma_{zx} = \dfrac{\tau_{zx}}{G} \end{cases} \tag{1.58b}$$

where E is Young's modulus, ν Poisson's ratio, G the shear modulus, and e_v the volumetric strain expressed by the sum of the three normal components of strain, i.e., $e_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$. The volumetric strain e_v can be written in other words as $e_v = \Delta V/V$, where V is the initial volume of the elastic body of interest in an undeformed state and ΔV the change of the volume after deformation.

In the two-dimensional elasticity theory, the three-dimensional Hooke's law is converted into two-dimensional form by using the following two types of approximations:

(1) Plane stress approximation: For thin plates, for example, one can assume the plane stress approximation that all the stress components in the direction perpendicular to the plate surface vanish, i.e., $\sigma_z = \tau_{zx} = \tau_{yz} = 0$. The stress–strain relations in this approximation are written by the following two-dimensional Hooke's law:

$$\begin{cases} \sigma_x = \dfrac{E}{1-\nu^2}\left(\varepsilon_x + \nu\varepsilon_y\right) \\ \sigma_y = \dfrac{E}{1-\nu^2}\left(\varepsilon_y + \nu\varepsilon_x\right) \\ \tau_{xy} = G\gamma_{xy} = \dfrac{E}{2\,(1+\nu)}\gamma_{xy} \end{cases} \tag{1.59a}$$

or

$$\begin{cases} \varepsilon_x = \dfrac{1}{E}\left(\sigma_x - \nu\sigma_y\right) \\ \varepsilon_y = \dfrac{1}{E}\left(\sigma_y - \nu\sigma_x\right) \\ \gamma_{xy} = \dfrac{\tau_{xy}}{G} = \dfrac{2\,(1+\nu)}{E}\tau_{xy} \end{cases} \tag{1.59b}$$

The normal strain component ε_z in the thickness direction, however, is not zero, but $\varepsilon_z = -\nu(\sigma_x + \sigma_y)/E$.

The plane stress approximation satisfies the equations of equilibrium (1.56); nevertheless, the normal strain in the direction of the z-axis ε_z must take a special form, i.e., ε_z must be a linear function of coordinate variables x and y in order to satisfy the compatibility condition which ensures the single-valuedness and continuity conditions of strains. Since this approximation imposes a special requirement for the form of the strain ε_z and thus the forms of the normal stresses σ_x and σ_y, this approximation cannot be considered as a general rule. Strictly speaking, the plane stress state does not exist in reality.

(2) Plane strain approximation: In cases where plate thickness (in the direction of the z-axis) is large, displacement is subjected to large constraints in the direction of the z-axis such that $\varepsilon_z = \gamma_{zx} = \gamma_{yz} = 0$. This case is called the plane stress approximation. The generalized Hooke's law can be written as follows:

$$\begin{cases} \sigma_x = \dfrac{E}{(1+\nu)\,(1-2\nu)}\left[(1-\nu)\,\varepsilon_x + \nu\varepsilon_y\right] \\ \sigma_y = \dfrac{E}{(1+\nu)\,(1-2\nu)}\left[\nu\varepsilon_x + (1-\nu)\,\varepsilon_y\right] \\ \tau_{xy} = G\gamma_{xy} = \dfrac{E}{2\,(1+\nu)}\gamma_{xy} \end{cases} \tag{1.60a}$$

or

$$\begin{cases} \varepsilon_x = \dfrac{1+\nu}{E}\left[(1-\nu)\,\sigma_x - \nu\sigma_y\right] \\ \varepsilon_y = \dfrac{1+\nu}{E}\left[-\nu\sigma_x + (1-\nu)\,\sigma_y\right] \\ \gamma_{xy} = \dfrac{\tau_{xy}}{G} = \dfrac{2\,(1+\nu)}{E}\tau_{xy} \end{cases} \tag{1.60b}$$

The normal stress component σ_z in the thickness direction is not zero, but $\sigma_z = \nu E(\sigma_x + \sigma_y)/[(1+\nu)\,(1-2\nu)]$. Since the plane strain state satisfies the equations of equilibrium (1.56) and the compatibility condition, this state can exist in reality.

If we redefine Young's modulus and Poisson's ratio by the following formulae:

$$E' = \begin{cases} E & \text{(plane stress)} \\ \dfrac{E}{1-\nu} & \text{(plane strain)} \end{cases} \tag{1.61a}$$

$$\nu' = \begin{cases} \nu & \text{(plane stress)} \\ \dfrac{\nu}{1-\nu} & \text{(plane strain)} \end{cases} \tag{1.61b}$$

the two-dimensional Hooke's law can be expressed in a unified form:

$$\begin{cases} \sigma_x = \dfrac{E'}{1-\nu'^2}\left(\varepsilon_x + \nu'\varepsilon_y\right) \\ \sigma_y = \dfrac{E'}{1-\nu'^2}\left(\varepsilon_y + \nu'\varepsilon_x\right) \\ \tau_{xy} = G\gamma_{xy} = \dfrac{E'}{2\,(1+\nu')}\gamma_{xy} \end{cases} \tag{1.62a}$$

or

$$\begin{cases} \varepsilon_x = \dfrac{1}{E'}\left(\sigma_x - \nu'\sigma_y\right) \\ \varepsilon_y = \dfrac{1}{E'}\left(\sigma_y - \nu'\sigma_x\right) \\ \gamma_{xy} = \dfrac{\tau_{xy}}{G} = \dfrac{2\left(1+\nu'\right)}{E'}\tau_{xy} \end{cases} \tag{1.62b}$$

The shear modulus G is invariant under the transformations as shown in Equations (1.61a) and (1.61b), i.e.,

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

1.4.2.4 BOUNDARY CONDITIONS

When solving the partial differential equation (1.56), there remains indefiniteness in the form of integral constants. In order to eliminate this indefiniteness, prescribed

conditions on stress and/or displacements must be imposed on the bounding surface of the elastic body. These conditions are called boundary conditions. There are two types of boundary conditions, i.e. (1) mechanical boundary conditions prescribing stresses or surface tractions and (2) geometrical boundary conditions prescribing displacements.

Let us denote a portion of the surface of the elastic body where stresses are prescribed by S_σ and the remaining surface where displacements are prescribed by S_u. The whole surface of the elastic body is denoted by $S = S_\sigma + S_u$. Note that it is not possible to prescribe both stresses and displacements on a portion of the surface of the elastic body.

The mechanical boundary conditions on S_σ are given by the following equations:

$$\begin{cases} t_x^* = \bar{t}_x^* \\ t_y^* = \bar{t}_y^* \end{cases} \tag{1.63}$$

where t_x^* and t_y^* are the x- and the y-components of the traction force $\mathbf{t}^*$, respectively, while the bar over t_x^* and t_y^* indicates that those quantities are prescribed on that portion of the surface. Taking $\boldsymbol{n} = [\cos\alpha, \sin\alpha]$ as the outward unit normal vector at a point of a small element of the surface portion S_σ, the Cauchy relations which represent the equilibrium conditions for surface traction forces and internal stresses are given by the following equations:

$$\begin{cases} \bar{t}_x^* = \sigma_x \cos\alpha + \tau_{xy} \sin\alpha \\ \bar{t}_y^* = \tau_{xy} \cos\alpha + \sigma_y \sin\alpha \end{cases} \tag{1.64}$$

where α is the angle between the normal vector $\boldsymbol{n}$ and the x-axis. For free surfaces where no forces are applied, $t_x^* = 0$ and $t_y^* = 0$.

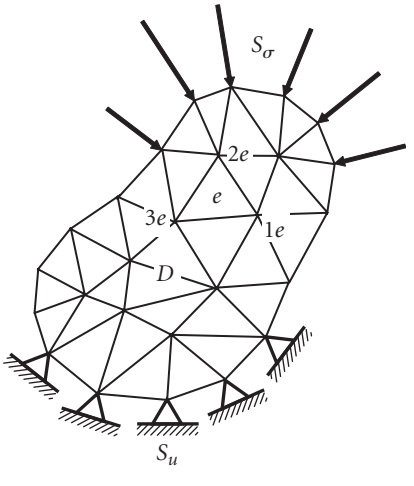


Figure 1.7 Finite-element discretization of a two-dimensional elastic body by triangular elements.

The geometrical boundary conditions on S_u are given by the following equations:

$$\begin{cases} u = \overline{u} \\ v = \overline{v} \end{cases} \tag{1.65}$$

where $\bar{u}$ and $\bar{v}$ are the x- and the y-components of prescribed displacements $\boldsymbol{u}$ on S_u. One of the most popular geometrical boundary conditions, i.e., clamped end condition, is denoted by $u = 0$ and/or $v = 0$ as shown in Figure 1.7.

1.4.3 VARIATIONAL FORMULAE IN ELASTOSTATIC PROBLEMS: THE PRINCIPLE OF VIRTUAL WORK

The variational principle used in two-dimensional elasticity problems is the principle of virtual work which is expressed by the following integral equation:

$$\iint_D \left(\sigma_x \delta\varepsilon_x + \sigma_y \delta\varepsilon_y + \tau_{xy}\delta\gamma_{xy}\right) t\, dx\, dy - \iint_D \left(F_x \delta u + F_y \delta v\right) t\, dx\, dy - \int_{S_\sigma} \left(\overline{t}_x^* \delta u + \overline{t}_y^* \delta v\right) t\, ds = 0 \tag{1.66}$$

where D denotes the whole region of a two-dimensional elastic body of interest, S_σ the whole portion of the surface of the elastic body $S(=S_\sigma \cup S_u)$, where the mechanical boundary conditions are prescribed and t the thickness.

The first term in the left-hand side of Equation (1.66) represents the increment of the strain energy of the elastic body, the second term the increment of the work done by the body forces, and the third term the increment of the work done by the surface traction forces. Therefore, Equation (1.66) claims that the increment of the strain energy of the elastic body is equal to the work done by the forces applied.

The fact that the integrand in each integral in the left-hand side of Equation (1.66) is identically equal to zero brings about the equations of equilibrium (1.56) and the boundary conditions (1.63) and/or (1.65). Therefore, instead of solving the partial differential equations (1.56) under the boundary conditions of Equations (1.63) and/or (1.65), two-dimensional elasticity problems can be solved by using the integral equation (1.66).

1.4.4 FORMULATION OF THE FUNDAMENTAL FINITE-ELEMENT EQUATIONS IN PLANE ELASTOSTATIC PROBLEMS

1.4.4.1 STRAIN–DISPLACEMENT MATRIX OR [B] MATRIX

Let us use the constant-strain triangular element (see Figure 1.8(a)) to derive the fundamental finite-element equations in plane elastostatic problems. The constant-strain triangular element assumes the displacements within the element to be expressed by

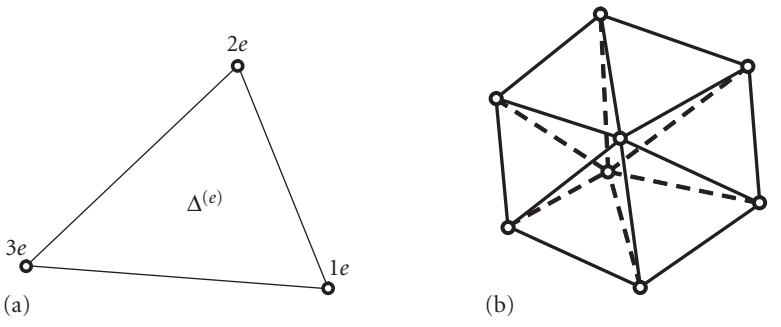


Figure 1.8 (a) Triangular constant strain element and (b) the continuity of displacements.

the following linear functions of the coordinate variables (x, y):

$$\begin{cases} u = \alpha_0 + \alpha_1 x + \alpha_2 y \\ v = \beta_0 + \beta_1 x + \beta_2 y \end{cases} \tag{1.67}$$

The above interpolation functions for displacements convert straight lines joining arbitrarily two points in the element into straight lines after deformation. Since the boundaries between neighboring elements are straight lines joining the apices or nodal points of triangular elements, incompatibility does not occur along the boundaries between adjacent elements and displacements are continuous everywhere in the domain to be analyzed as shown in Figure 1.8(b). For the eth triangular element consisting of three apices or nodal points $(1_e, 2_e, 3_e)$ having the coordinates (x_{1e}, y_{1e}), (x_{2e}, y_{2e}), and (x_{3e}, y_{3e}) and the nodal displacements (u_{1e}, v_{1e}), (u_{2e}, v_{2e}), and (u_{3e}, v_{3e}), the coefficients α_0, α_1, α_2, β_0, β_1, and β_2 in Equations (1.67) are obtained by the following equations:

$$\begin{cases} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \end{Bmatrix} = \begin{pmatrix} a_{1e} & a_{2e} & a_{3e} \\ b_{1e} & b_{2e} & b_{3e} \\ c_{1e} & c_{2e} & c_{3e} \end{pmatrix} \begin{Bmatrix} u_{1e} \\ u_{2e} \\ u_{3e} \end{Bmatrix} \\ \begin{Bmatrix} \beta_0 \\ \beta_1 \\ \beta_2 \end{Bmatrix} = \begin{pmatrix} a_{1e} & a_{2e} & a_{3e} \\ b_{1e} & b_{2e} & b_{3e} \\ c_{1e} & c_{2e} & c_{3e} \end{pmatrix} \begin{Bmatrix} v_{1e} \\ v_{2e} \\ v_{3e} \end{Bmatrix} \end{cases} \tag{1.68}$$

where

$$\begin{cases} a_{1e} = \dfrac{1}{2\Delta^{(e)}} \left(x_{2e} y_{3e} - x_{3e} y_{2e} \right) \\ b_{1e} = \dfrac{1}{2\Delta^{(e)}} \left(y_{2e} - y_{3e} \right) \\ c_{1e} = \dfrac{1}{2\Delta^{(e)}} \left(x_{3e} - x_{2e} \right) \end{cases} \tag{1.69a}$$

$$\begin{cases} a_{2e} = \dfrac{1}{2\Delta^{(e)}}\left(x_{3e}y_{1e} - x_{1e}y_{3e}\right) \\ b_{2e} = \dfrac{1}{2\Delta^{(e)}}\left(y_{3e} - y_{1e}\right) \\ c_{2e} = \dfrac{1}{2\Delta^{(e)}}\left(x_{1e} - x_{3e}\right) \end{cases} \tag{1.69b}$$

$$\begin{cases} a_{3e} = \dfrac{1}{2\Delta^{(e)}}\left(x_{1e}y_{2e} - x_{2e}y_{1e}\right) \\ b_{3e} = \dfrac{1}{2\Delta^{(e)}}\left(y_{1e} - y_{2e}\right) \\ c_{3e} = \dfrac{1}{2\Delta^{(e)}}\left(x_{2e} - x_{1e}\right) \end{cases} \tag{1.69c}$$

The numbers with subscript "e", 1_e, 2_e, and 3_e, in the above equations are called element nodal numbers and denote the numbers of three nodal points of the eth element. Nodal points should be numbered counterclockwise. These three numbers are used only in the eth element. Nodal numbers of the other type called global nodal numbers are also assigned to the three nodal points of the eth element, being numbered throughout the whole model of the elastic body. The symbol $\Delta^{(e)}$ represents the area of the eth element and can be expressed only by the coordinates of the nodal points of the element, i.e.,

$$\Delta^{(e)} = \frac{1}{2}\left[\left(x_{1e} - x_{3e}\right)\left(y_{2e} - y_{3e}\right) - \left(y_{3e} - y_{1e}\right)\left(x_{3e} - x_{2e}\right)\right] = \frac{1}{2}\begin{vmatrix} 1 & x_{1e} & y_{1e} \\ 1 & x_{2e} & y_{2e} \\ 1 & x_{3e} & y_{3e} \end{vmatrix} \tag{1.69d}$$

Consequently, the components of the displacement vector $[u, v]$ can be expressed by the components of the nodal displacement vectors $[u_{1e}, v_{1e}]$, $[u_{2e}, v_{2e}]$, and $[u_{3e}, v_{3e}]$ as follows:

$$\begin{cases} u = \left(a_{1e} + b_{1e}x + c_{1e}y\right)u_{1e} + \left(a_{2e} + b_{2e}x + c_{2e}y\right)u_{2e} + \left(a_{3e} + b_{3e}x + c_{3e}y\right)u_{3e} \\ v = \left(a_{1e} + b_{1e}x + c_{1e}y\right)v_{1e} + \left(a_{2e} + b_{2e}x + c_{2e}y\right)v_{2e} + \left(a_{3e} + b_{3e}x + c_{3e}y\right)v_{3e} \end{cases} \tag{1.70}$$

Matrix notation of Equation (1.70) is

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_{1e}^{(e)} & 0 & N_{2e}^{(e)} & 0 & N_{3e}^{(e)} & 0 \\ 0 & N_{1e}^{(e)} & 0 & N_{2e}^{(e)} & 0 & N_{3e}^{(e)} \end{bmatrix} \begin{Bmatrix} u_{1e} \\ v_{1e} \\ u_{2e} \\ v_{2e} \\ u_{3e} \\ v_{3e} \end{Bmatrix} = [\mathbf{N}]\{\boldsymbol{\delta}\}^{(e)} \tag{1.71}$$

where

$$\begin{cases} N_{1e}^{(e)} = a_{1e} + b_{1e}x + c_{1e}y \\ N_{2e}^{(e)} = a_{2e} + b_{2e}x + c_{2e}y \\ N_{3e}^{(e)} = a_{3e} + b_{3e}x + c_{3e}y \end{cases} \tag{1.72}$$

and the superscript of $\{\boldsymbol{\delta}\}^{(e)}$, (e), indicates that $\{\boldsymbol{\delta}\}^{(e)}$ is the displacement vector determined by the three displacement vectors at the three nodal points of the eth triangular element. Equation (1.72) formulates the definitions of the interpolation functions or shape functions $N_{ie}^{(e)} (i = 1, 2, 3)$ for the triangular constant-strain element.

Now, let us consider strains derived from the displacements given by Equation (1.71). Substitution of Equation (1.71) into (1.57) gives

$$\begin{aligned} \{\boldsymbol{\varepsilon}\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} &= \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial N_{1e}^{(e)}}{\partial x} & 0 & \dfrac{\partial N_{2e}^{(e)}}{\partial x} & 0 & \dfrac{\partial N_{3e}^{(e)}}{\partial x} & 0 \\ 0 & \dfrac{\partial N_{1e}^{(e)}}{\partial y} & 0 & \dfrac{\partial N_{2e}^{(e)}}{\partial y} & 0 & \dfrac{\partial N_{3e}^{(e)}}{\partial y} \\ \dfrac{\partial N_{1e}^{(e)}}{\partial x} & \dfrac{\partial N_{1e}^{(e)}}{\partial y} & \dfrac{\partial N_{2e}^{(e)}}{\partial x} & \dfrac{\partial N_{2e}^{(e)}}{\partial y} & \dfrac{\partial N_{3e}^{(e)}}{\partial x} & \dfrac{\partial N_{3e}^{(e)}}{\partial y} \end{bmatrix} \begin{Bmatrix} u_{1e} \\ v_{1e} \\ u_{2e} \\ v_{2e} \\ u_{3e} \\ v_{3e} \end{Bmatrix} \\ &= \begin{bmatrix} b_{1e} & 0 & b_{2e} & 0 & b_{3e} & 0 \\ 0 & c_{1e} & 0 & c_{2e} & 0 & c_{3e} \\ c_{1e} & b_{1e} & c_{2e} & b_{2e} & c_{3e} & b_{3e} \end{bmatrix} \begin{Bmatrix} u_{1e} \\ v_{1e} \\ u_{2e} \\ v_{2e} \\ u_{3e} \\ v_{3e} \end{Bmatrix} = [\mathbf{B}]\{\boldsymbol{\delta}\}^{(e)} \end{aligned} \tag{1.73}$$

where $[\mathbf{B}]$ establishes the relationship between the nodal displacement vector $\{\boldsymbol{\delta}\}^{(e)}$ and the element strain vector $\{\boldsymbol{\varepsilon}\}$, and is called the strain–displacement matrix or $[\mathbf{B}]$ matrix. All the components of the $[\mathbf{B}]$ matrix are expressed only by the coordinate values of the three nodal points consisting of the element.

From the above discussion, it can be concluded that strains are constant throughout a three-node triangular element, since its interpolation functions are linear functions of the coordinate variables within the element. For this reason, a triangular element with three nodal points is called a "constant-strain" element. Three-node triangular elements cannot satisfy the compatibility condition in the strict sense, since strains are discontinuous among elements. It is demonstrated, however, that the results obtained by elements of this type converge to exact solutions as the size of the elements becomes smaller.

It is known that elements must fulfill the following three criteria for the finite-element solutions to converge to the exact solutions as the subdivision into even-smaller elements is attempted. Namely, the elements must

(1) represent rigid body displacements,

(2) represent constant strains, and

(3) ensure the continuity of displacements among elements.

1.4.4.2 STRESS–STRAIN MATRIX OR [D] MATRIX

Substitution of Equation (1.73) into (1.62a) gives

$$\{\boldsymbol{\sigma}\} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E'}{1-\nu'^2} \begin{bmatrix} 1 & \nu' & 0 \\ \nu' & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu'}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = [\mathbf{D}^e]\{\boldsymbol{\varepsilon}\} = [\mathbf{D}^e][\mathbf{B}]\{\boldsymbol{\delta}\}^{(e)} \tag{1.74}$$

where $[\mathbf{D}^e]$ establishes the relationship between stresses and strains, or the constitutive relations. The matrix $[\mathbf{D}^e]$ is for elastic bodies and thus is called the elastic stress–strain matrix or just $[\mathbf{D}]$ matrix. In the case where initial strains $\{\boldsymbol{\varepsilon}_0\}$ such as plastic strains, thermal strains, and residual strains exist, $\{\boldsymbol{\varepsilon}\} - \{\boldsymbol{\varepsilon}_0\}$ is used instead of $\{\boldsymbol{\varepsilon}\}$.

1.4.4.3 ELEMENT STIFFNESS EQUATIONS

First, let $\{\mathbf{P}\}^{(e)}$ define the equivalent nodal forces which are statically equivalent to the traction forces $\mathbf{t}^* = [t_x^*, t_y^*]$ on the element boundaries and the body forces $\{\mathbf{F}\}^{(e)}$ in the element:

$$\{\mathbf{F}\}^{(e)T} = [F_x, F_y] \tag{1.75}$$

$$\{\mathbf{P}\}^{(e)T} = [X_{1e}, Y_{1e}, X_{2e}, Y_{2e}, X_{3e}, Y_{3e}] \tag{1.76}$$

In the above equations, $\{\mathbf{F}\}$ represents a column vector, $[\mathbf{P}]$ a row vector, and superscript T the transpose of a vector or a matrix.

In order to make differentiations shown in Equation (1.57), displacements assumed by Equation (1.71) must be continuous everywhere in an elastic body of interest. The remaining conditions to be satisfied are the equations of equilibrium (1.56) and the mechanical boundary conditions (1.63); nevertheless these equations generally cannot be satisfied in the strict sense. Hence, the equivalent nodal forces, for instance (X_{1e}, Y_{1e}), (X_{2e}, Y_{2e}), and (X_{3e}, Y_{3e}), are defined on the three nodal points of the eth element via determining these forces by the principle of the virtual work in order to satisfy the equilibrium and boundary conditions element by element. Namely, the principle of the virtual work to be satisfied for arbitrary virtual displacements $\{\boldsymbol{\delta}^*\}^{(e)}$ of the eth element is derived from Equation (1.66) as

$$\{\boldsymbol{\delta}^*\}^{(e)T}\{\mathbf{P}\}^{(e)} = \iint_D (\{\boldsymbol{\varepsilon}^*\}^T\{\boldsymbol{\sigma}\} - \{\mathbf{f}^*\}^T\{\mathbf{F}\}^{(e)})t\,dx\,dy \tag{1.77}$$

where

$$\{\boldsymbol{\varepsilon}^*\} = [\mathbf{B}]\{\boldsymbol{\delta}^*\}^{(e)} \tag{1.78}$$

$$\{\mathbf{f}^*\} = [\mathbf{N}]\{\boldsymbol{\delta}^*\}^{(e)} \tag{1.79}$$

Substitution of Equations (1.78) and (1.79) into (1.77) gives

$$\{\boldsymbol{\delta}^*\}^{(e)T}\{\mathbf{P}\}^{(e)} = \{\boldsymbol{\delta}^*\}^{(e)T}\left(\iint_D [\mathbf{B}]^T\{\boldsymbol{\sigma}\}t\,dx\,dy - \iint_D [\mathbf{N}]^T\{\mathbf{F}\}^{(e)}t\,dx\,dy\right) \tag{1.80}$$

Since Equation (1.80) holds true for any virtual displacements $\{\boldsymbol{\delta}^*\}^{(e)}$, the equivalent nodal forces can be obtained by the following equation:

$$\{\mathbf{P}\}^{(e)} = \iint_D [\mathbf{B}]^T\{\boldsymbol{\sigma}\}t\,dx\,dy - \iint_D [\mathbf{N}]^T\{\mathbf{F}\}^{(e)}t\,dx\,dy \tag{1.81}$$

From Equations (1.73) and (1.74),

$$\{\boldsymbol{\sigma}\} = [\mathbf{D}^e](\{\boldsymbol{\varepsilon}\} - \{\boldsymbol{\varepsilon}_0\}) = [\mathbf{D}^e][\mathbf{B}]\{\boldsymbol{\delta}\}^{(e)} - [\mathbf{D}^e]\{\boldsymbol{\varepsilon}_0\} \tag{1.82}$$

Substitution of Equation (1.82) into (1.81) gives

$$\{\mathbf{P}\}^{(e)} = \left(\iint_D [\mathbf{B}]^T[\mathbf{D}^e][\mathbf{B}]t\,dx\,dy\right)\{\boldsymbol{\delta}\}^{(e)} - \iint_D [\mathbf{B}]^T[\mathbf{D}^e]\{\boldsymbol{\varepsilon}_0\}t\,dx\,dy - \iint_D [\mathbf{N}]^T\{\mathbf{F}\}^{(e)}t\,dx\,dy \tag{1.83}$$

Equation (1.83) is rewritten in the form

$$\{\mathbf{P}\}^{(e)} = [\mathbf{k}^{(e)}]\{\boldsymbol{\delta}\}^{(e)} + \{\mathbf{F}_{\varepsilon_0}\}^{(e)} + \{\mathbf{F}_F\}^{(e)} \tag{1.84}$$

where

$$[\mathbf{k}^{(e)}] \equiv \iint_D [\mathbf{B}]^T[\mathbf{D}^e][\mathbf{B}]t\,dx\,dy = \Delta^{(e)}[\mathbf{B}]^T[\mathbf{D}^e][\mathbf{B}]t \tag{1.85}$$

$$\{\mathbf{F}_{\varepsilon_0}\}^{(e)} \equiv -\iint_D [\mathbf{B}]^T[\mathbf{D}^e]\{\boldsymbol{\varepsilon}_0\}t\,dx\,dy \tag{1.86}$$

$$\{\mathbf{F}_F\}^{(e)} \equiv -\iint_D [\mathbf{N}]^T\{\mathbf{F}\}^{(e)}t\,dx\,dy \tag{1.87}$$

Equation (1.84) is called the element stiffness equation for the eth triangular finite element and $[\mathbf{k}^{(e)}]$ defined by Equation (1.85) the element stiffness matrix. Since the matrices $[\mathbf{B}]$ and $[\mathbf{D}^e]$ are constant throughout the element, they can be taken out of the integral and the integral is simply equal to the area of the element $\Delta^{(e)}$ so that the rightmost side of Equation (1.85) is obtained. The forces $\{\mathbf{F}_{\varepsilon_0}\}^{(e)}$ and $\{\mathbf{F}_F\}^{(e)}$ are the equivalent nodal forces due to initial strains and body forces, respectively. Since the integrand in Equation (1.85) is generally a function of the coordinate variables x and y except for the case of three-node triangular elements, the integrals appearing in Equation (1.85) are often evaluated by a numerical integration scheme such as the Gaussian quadrature.

The element stiffness matrix $[\mathbf{k}^{(e)}]$ in Equation (1.85) is a 6 by 6 square matrix which can be decomposed into nine 2 by 2 submatrices as shown in the following equation:

$$[\mathbf{k}^{(e)}_{ieje}] = \begin{bmatrix} \mathbf{k}^{(e)}_{1e1e} & \mathbf{k}^{(e)}_{1e2e} & \mathbf{k}^{(e)}_{1e3e} \\ \mathbf{k}^{(e)}_{2e1e} & \mathbf{k}^{(e)}_{2e2e} & \mathbf{k}^{(e)}_{2e3e} \\ \mathbf{k}^{(e)}_{3e1e} & \mathbf{k}^{(e)}_{3e2e} & \mathbf{k}^{(e)}_{3e3e} \end{bmatrix} \tag{1.88}$$

$$\mathbf{k}^{(e)}_{ieje} = \mathbf{k}^{(e)T}_{jeie} \tag{1.89}$$

$$\mathbf{k}^{(e)}_{ieje} = \int_{\Delta^{(e)}} [\mathbf{B}_{ie}]^T [\mathbf{D}^e][\mathbf{B}_{je}] t \, dx \, dy \quad \begin{cases} (2 \times 2) \text{ asymmetric matrix} (i_e \neq j_e) \\ (2 \times 2) \text{ symmetric matrix} (i_e = j_e) \end{cases} \tag{1.90}$$

where

$$[\mathbf{B}_{ie}] \equiv \frac{1}{2\Delta^{(e)}} \begin{bmatrix} b_{ie} & 0 \\ 0 & c_{ie} \\ c_{ie} & b_{ie} \end{bmatrix} \quad (i_e = 1, 2, 3) \tag{1.91}$$

and the subscripts i_e and j_e of $\mathbf{k}^{(e)}_{ieje}$ refer to element nodal numbers and

$$\mathbf{k}^{(e)}_{ieje} = [\mathbf{B}_{ie}]^T [\mathbf{D}^e][\mathbf{B}_{je}] t \Delta^{(e)} \tag{1.92}$$

In the above discussion, the formulae have been obtained for just one triangular element, but are available for any elements, if necessary, with some modifications.

1.4.4.4 GLOBAL STIFFNESS EQUATIONS

Element stiffness equations as shown in Equation (1.84) are determined for element by element, and then they are assembled into the global stiffness equations for the whole elastic body of interest. Since nodal points which belong to different elements but have the same coordinates are the same points, the following items during the assembly procedure of the global stiffness equations are to be noted:

(1) The displacement components u and v of the same nodal points which belong to different elements are the same; i.e., there exist no incompatibilities such as cracks between elements.

(2) For nodal points on the bounding surfaces and for those in the interior of the elastic body to which no forces are applied, the sums of the nodal forces are to be zero.

(3) Similarly, for nodal points to which forces are applied, the sums of the nodal forces are equal to the sums of the forces applied to those nodal points.

The same global nodal numbers are to be assigned to the nodal points which have the same coordinates. Taking the items described above into consideration, let us rewrite the element stiffness matrix $[\mathbf{k}^{(e)}]$ in Equation (1.88) by using the global

nodal numbers I, J, and K ($I, J, K = 1, 2, \ldots, 2n$) instead of the element nodal numbers i_e, j_e, and k_e ($i_e, j_e, k_e = 1, 2, 3$); i.e.,

$$\left[\mathbf{k}^{(e)}\right] = \begin{bmatrix} \mathbf{k}_{II}^{(e)} & \mathbf{k}_{IJ}^{(e)} & \mathbf{k}_{IK}^{(e)} \\ \mathbf{k}_{JI}^{(e)} & \mathbf{k}_{JJ}^{(e)} & \mathbf{k}_{JK}^{(e)} \\ \mathbf{k}_{KI}^{(e)} & \mathbf{k}_{KJ}^{(e)} & \mathbf{k}_{KK}^{(e)} \end{bmatrix} \tag{1.93}$$

Then, let us embed the element stiffness matrix in a square matrix having the same size as the global stiffness matrix of $2n$ by $2n$ as shown in Equation (1.94):

$$[\mathbf{K}^{(e)}]\{\boldsymbol{\delta}\} = \begin{array}{c} \\ 1 \\ 2 \\ \vdots \\ 2I-1 \\ 2I \\ \vdots \\ 2J-1 \\ 2J \\ \vdots \\ 2K-1 \\ 2K \\ \vdots \\ 2n-1 \\ 2n \end{array} \begin{array}{c} \begin{array}{cccccccccccc} 1 & 2 & \cdots & 2I-1 & 2I & \cdots & 2J-1 & 2J & \cdots & 2K-1 & 2K & \cdots \end{array} \; 2n-1 \quad 2n \\ \begin{bmatrix} 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 \\ 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 & \cdots & 0 & 0 \\ \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots \\ 0 & 0 & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{II}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{IJ}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{IK}^{(e)}$}} & \cdots & 0 & 0 \\ 0 & 0 & \cdots & & & \cdots & & & \cdots & & & \cdots & 0 & 0 \\ \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots \\ 0 & 0 & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{JI}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{JJ}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{JK}^{(e)}$}} & \cdots & 0 & 0 \\ 0 & 0 & \cdots & & & \cdots & & & \cdots & & & \cdots & 0 & 0 \\ \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots \\ 0 & 0 & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{KI}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{KJ}^{(e)}$}} & \cdots & \multicolumn{2}{c}{\multirow{2}{*}{$\mathbf{k}_{KK}^{(e)}$}} & \cdots & 0 & 0 \\ 0 & 0 & \cdots & & & \cdots & & & \cdots & & & \cdots & 0 & 0 \\ \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots & & \vdots & \vdots \\ 0 & 0 & \cdots & & & \cdots & & & \cdots & & & \cdots & 0 & 0 \\ 0 & 0 & \cdots & & & \cdots & & & \cdots & & & \cdots & 0 & 0 \end{bmatrix} \end{array}$$

$$\times \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_I \\ v_I \\ \vdots \\ u_j \\ v_j \\ \vdots \\ u_k \\ v_k \\ \vdots \\ u_n \\ v_n \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ \vdots \\ X_I^{(e)} \\ Y_I^{(e)} \\ \vdots \\ X_J^{(e)} \\ Y_J^{(e)} \\ \vdots \\ X_K^{(e)} \\ Y_k^{(e)} \\ \vdots \\ 0 \\ 0 \end{Bmatrix} \tag{1.94}$$

where n denotes the number of nodal points. This procedure is called the method of extended matrix. The number of degrees of freedom here means the number of unknown variables. In two-dimensional elasticity problems, since two of displacements and forces in the x- and the y-directions are unknown variables for one nodal point, every nodal point has two degrees of freedom. Hence, the number of degrees of freedom for a finite-element model consisting of n nodal points is $2n$.

By summing up the element stiffness matrices for all the n_e elements in the finite-element model, the global stiffness matrix $[\mathbf{K}]$ is obtained as shown in the following equation:

$$[\mathbf{K}] \equiv [K_{ij}] = \sum_{e=1}^{ne} [\mathbf{K}^{(e)}] \quad (i, j = 1, 2, \ldots, 2n \quad \text{and} \quad e = 1, 2, \ldots, n_e) \tag{1.95}$$

Since the components of the global nodal displacement vector $\{\boldsymbol{\delta}\}$ are common for all the elements, they remain unchanged during the assembly of the global stiffness equations. By rewriting the components of $\{\boldsymbol{\delta}\}$, $u_1, u_2, \ldots, u_n$ as $u_1, u_3, \ldots, u_{2i-1}, \ldots, u_{2n-1}$ and $v_1, v_2, \ldots, v_n$ as $u_2, u_4, \ldots, u_{2i}, \ldots, u_{2n}$, the following expression for the global nodal displacement vector $\{\boldsymbol{\delta}\}$ is obtained:

$$\{\boldsymbol{\delta}\} = \{u_1, u_2, \ldots, u_{2I-1}, u_{2I}, \ldots, u_{2J-1}, u_{2J}, \ldots, u_{2K-1}, u_{2K}, \ldots, u_{2n-1}, u_{2n}\}^T \tag{1.96}$$

The global nodal force of a node is the sum of the nodal forces for all the elements to which the node belongs. Hence, the global nodal force vector $\{\mathbf{P}\}$ can be written as

$$\{\mathbf{P}\} = \{X_1, Y_1, \ldots, X_I, Y_I, \ldots, X_J, V_J, \ldots, X_K, Y_K, \ldots, X_n, Y_n\}^T \tag{1.97}$$

where

$$X_I = \sum X_I^{(e)} \quad Y_I = \sum Y_I^{(e)} \quad (I = 1, 2, \ldots, n) \tag{1.98}$$

By rewriting the global nodal force vector $\{\mathbf{P}\}$ in a similar way to $\{\boldsymbol{\delta}\}$ in Equation (1.96) as

$$\{\mathbf{P}\} = \{X_1, X_2, \cdots, X_{2I-1}, X_{2I}, \ldots, X_{2J-1}, X_{2J}, \ldots, X_{2K-1}, X_{2K}, \ldots, X_{2n-1}, X_{2n}\}^T \tag{1.99}$$

where

$$X_I = \sum X_I^{(e)} \quad (I = 1, 2, \ \ldots, 2n) \tag{1.100}$$

The symbol Σ in Equations (1.98) and (1.100) indicates that the summation is taken over all the elements that possess the node in common. The values of X_I in Equation (1.100), however, are zero for the nodes inside of the elastic body and for those on the bounding surfaces which are subjected to no applied loads.

Consequently, the following formula is obtained as the governing global stiffness equation:

$$[\mathbf{K}]\{\boldsymbol{\delta}\} = \{\mathbf{P}\} \tag{1.101}$$

which is the $2n$th degree simultaneous linear equations for $2n$ unknown variables of nodal displacements and/or forces.

1.4.4.5 EXAMPLE: FINITE-ELEMENT CALCULATIONS FOR A SQUARE PLATE SUBJECTED TO UNIAXIAL UNIFORM TENSION

Procedures 5 through 7 described in Section 1.4.1 will be explained by taking an example of the finite-element calculations for a square plate subjected to uniaxial uniform tension as illustrated in Figure 1.9. The square plate model has a side of unit length 1 and a thickness of unit length 1, and consists of two constant-strain triangular elements, i.e., the model has four nodes and thus eight degrees of freedom.

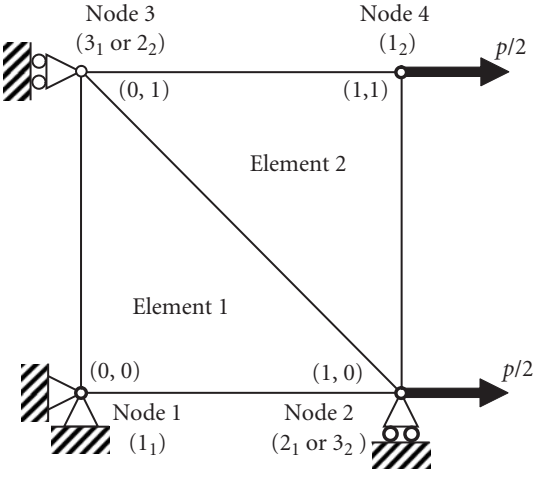


Figure 1.9 Finite element model of a square plate subjected to uniaxial uniform tension.

Let us determine the element stiffness matrix for Element 1. From Equations (1.73) and (1.69a) through (1.69c), the [**B**] matrix of Element 1 is calculated as

$$[\mathbf{B}] = \begin{bmatrix} b_{1_1} & 0 & b_{2_1} & 0 & b_{3_1} & 0 \\ 0 & c_{1_1} & 0 & c_{2_1} & 0 & c_{3_1} \\ c_{1_1} & b_{1_1} & c_{2_1} & b_{2_1} & c_{3_1} & b_{3_1} \end{bmatrix}$$

$$= \frac{1}{2\Delta^{(1)}} \begin{bmatrix} y_{2_1} - y_{3_1} & 0 & y_{3_1} - y_{1_1} & 0 & y_{1_1} - y_{2_1} & 0 \\ 0 & x_{3_1} - x_{2_1} & 0 & x_{1_1} - x_{3_1} & 0 & x_{2_1} - x_{1_1} \\ x_{3_1} - x_{2_1} & y_{2_1} - y_{3_1} & x_{1_1} - x_{3_1} & y_{3_1} - y_{1_1} & x_{2_1} - x_{1_1} & y_{1_1} - y_{2_1} \end{bmatrix} \tag{1.102}$$

Since the area of Element 1 $\Delta^{(e)}$ in the above equation can be easily obtained as 1/2 without using Equation (1.69d),

$$[\mathbf{B}] = \begin{bmatrix} -1 & 0 & 1 & 0 & 0 & 0 \\ 0 & -1 & 0 & 0 & 0 & 1 \\ -1 & -1 & 0 & 1 & 1 & 0 \end{bmatrix} \tag{1.103}$$

Hence, from Equation (1.85), the element stiffness matrix for Element 1 $[\mathbf{k}^{(e)}]$ is calculated as

$$[\mathbf{k}^{(1)}] = \frac{t}{2}\,\frac{E'}{1-\nu'^2}\begin{bmatrix} -1 & 0 & -1 \\ 0 & -1 & -1 \\ 1 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & 0 & 1 \\ 0 & 1 & 0 \end{bmatrix}\begin{bmatrix} 1 & \nu' & 0 \\ \nu' & 1 & 0 \\ 0 & 0 & \alpha \end{bmatrix}\begin{bmatrix} -1 & 0 & 1 & 0 & 0 & 0 \\ 0 & -1 & 0 & 0 & 0 & 1 \\ -1 & -1 & 0 & 1 & 1 & 0 \end{bmatrix} \tag{1.104}$$

where $\alpha = (1-\nu')/2$. After multiplication of the matrices in Equation (1.104), the element stiffness equation is obtained from Equation (1.84) as

$$\begin{Bmatrix} X_{1_1}^{(1)} \\ Y_{1_1}^{(1)} \\ X_{2_1}^{(1)} \\ Y_{2_1}^{(1)} \\ X_{3_1}^{(1)} \\ Y_{3_1}^{(1)} \end{Bmatrix} = \frac{t}{2}\,\frac{E'}{1-\nu'^2}\begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 \\ -1 & -\nu' & 1 & 0 & 0 & \nu' \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\nu' & -1 & \nu' & 0 & 0 & 1 \end{bmatrix}\begin{Bmatrix} u_{1_1} \\ v_{1_1} \\ u_{2_1} \\ v_{2_1} \\ u_{3_1} \\ v_{3_1} \end{Bmatrix} \tag{1.105}$$

since the equivalent nodal forces due to initial strains ε_0 and body forces F_x and F_y, $\{\mathbf{F}_{\varepsilon 0}\}^{(1)}$ and $\{\mathbf{F}_F\}^{(1)}$ are zero. The components of the nodal displacement and force vectors are written by the element nodal numbers. By rewriting these components by the global nodal numbers as shown in Figure 1.9, Equation (1.105) is rewritten as

$$\begin{Bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{Bmatrix} = \frac{t}{2}\,\frac{E'}{1-\nu'^2}\begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 \\ -1 & -\nu' & 1 & 0 & 0 & \nu' \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\nu' & -1 & \nu' & 0 & 0 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \tag{1.106}$$

In a similar way, the element stiffness equation for Element 2 is obtained as

$$\begin{Bmatrix} X_{1_2}^{(2)} \\ Y_{1_2}^{(2)} \\ X_{2_2}^{(2)} \\ Y_{2_2}^{(2)} \\ X_{3_2}^{(2)} \\ Y_{3_2}^{(2)} \end{Bmatrix} = \frac{t}{2}\,\frac{E'}{1-\nu'^2}\begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 \\ -1 & -\nu' & 1 & 0 & 0 & \nu' \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\nu' & -1 & \nu' & 0 & 0 & 1 \end{bmatrix}\begin{Bmatrix} u_{1_2} \\ v_{1_2} \\ u_{2_2} \\ v_{2_2} \\ u_{3_2} \\ v_{3_2} \end{Bmatrix} \tag{1.107}$$

and is rewritten by using the global nodal numbers as

$$\begin{Bmatrix} X_4 \\ Y_4 \\ X_3 \\ Y_3 \\ X_2 \\ Y_2 \end{Bmatrix} = \frac{t}{2}\ \frac{E'}{1-\nu'^2} \begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 \\ -1 & -\nu' & 1 & 0 & 0 & \nu' \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\alpha & -\alpha & 0 & \alpha & \alpha & 0 \\ -\nu' & -1 & \nu' & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_4 \\ v_4 \\ u_3 \\ v_3 \\ u_2 \\ v_2 \end{Bmatrix} \tag{1.108}$$

By assembling the two-element stiffness matrices, the following global stiffness equation for the square plate subjected to uniform tension is obtained (Procedure 4):

$$\frac{t}{2}\ \frac{E'}{1-\nu'^2} \begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' & 0 & 0 \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 & 0 & 0 \\ -1 & -\nu' & 1+\alpha & 0 & 0 & \nu'+\alpha & -\alpha & -\alpha \\ -\alpha & -\alpha & 0 & 1+\alpha & \nu'+\alpha & 0 & -\nu' & -1 \\ -\alpha & -\alpha & 0 & \nu'+\alpha & 1+\alpha & 0 & -1 & -\nu' \\ -\nu' & -1 & \nu'+\alpha & 0 & 0 & 1+\alpha & -\alpha & -\alpha \\ 0 & 0 & -\alpha & -\nu' & -1 & -\alpha & 1+\alpha & \nu'+\alpha \\ 0 & 0 & -\alpha & -1 & -\nu' & -\alpha & \nu'+\alpha & 1+\alpha \end{bmatrix} \times \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \\ X_4 \\ Y_4 \end{Bmatrix} \tag{1.109}$$

where the left- and right-hand sides of the equation are replaced with each other.

Let us now impose boundary conditions on the nodes. Namely, the node 1 is clamped in both the x- and the y-directions, the node 2 is clamped only in the x-direction, and the nodes 2 and 4 are subjected to equal nodal forces $X_2 = X_4 = (p \times 1 \times 1)/2 = p/2$, respectively, in the x-direction. A pair of the equal nodal forces $p/2$ applied to the nodes 2 and 4 in the x-direction is the finite-element model of a uniformly distributed tension force p per unit area exerted on the side $\overline{24}$ in the x-direction as illustrated in Figure 1.9. The geometrical boundary and mechanical conditions for the present case are

$$u_1 = v_1 = v_2 = u_3 = 0 \tag{1.110}$$

and

$$X_2 = X_4 = (pt)/2 = p/2 \quad Y_3 = Y_4 = 0 \tag{1.111}$$

respectively. Substitution of Equations (1.110) and (1.111) into Equation (1.109) gives the following global stiffness equation:

$$\frac{t}{2}\frac{E'}{1-\nu'^2}\begin{bmatrix} 1+\alpha & \nu'+\alpha & -1 & -\alpha & -\alpha & -\nu' & 0 & 0 \\ \nu'+\alpha & 1+\alpha & -\nu' & -\alpha & -\alpha & -1 & 0 & 0 \\ -1 & -\nu' & 1+\alpha & 0 & 0 & \nu'+\alpha & -\alpha & -\alpha \\ -\alpha & -\alpha & 0 & 1+\alpha & \nu'+\alpha & 0 & -\nu' & -1 \\ -\alpha & -\alpha & 0 & \nu'+\alpha & 1+\alpha & 0 & -1 & -\nu' \\ -\nu' & -1 & \nu'+\alpha & 0 & 0 & 1+\alpha & -\alpha & -\alpha \\ 0 & 0 & -\alpha & -\nu' & -1 & -\alpha & 1+\alpha & \nu'+\alpha \\ 0 & 0 & -\alpha & -1 & -\nu' & -\alpha & \nu'+\alpha & 1+\alpha \end{bmatrix} \times \begin{Bmatrix} 0 \\ 0 \\ u_2 \\ 0 \\ 0 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} X_1 \\ Y_1 \\ p/2 \\ Y_2 \\ X_3 \\ 0 \\ p/2 \\ 0 \end{Bmatrix} \tag{1.112}$$

Rearrangement of Equation (1.112) by collecting the unknown variables for forces and displacements in the left-hand side and the known values of the forces and displacements in the right-hand side brings about the following simultaneous equations (Procedure 5):

$$\frac{t}{2}\frac{E'}{1-\nu'^2}\begin{bmatrix} -\frac{2}{t}\frac{E'}{1-\nu'^2} & 0 & -1 & 0 & 0 & -\nu' & 0 & 0 \\ 0 & -\frac{2}{t}\frac{E'}{1-\nu'^2} & -\nu' & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 1+\alpha & 0 & 0 & \nu'+\alpha & -\alpha & -\alpha \\ 0 & 0 & 0 & -\frac{2}{t}\frac{E'}{1-\nu'^2} & 0 & 0 & -\nu' & -1 \\ 0 & 0 & 0 & 0 & -\frac{2}{t}\frac{E'}{1-\nu'^2} & 0 & -1 & -\nu' \\ 0 & 0 & \nu'+\alpha & 0 & 0 & 1+\alpha & -\alpha & -\alpha \\ 0 & 0 & -\alpha & 0 & 0 & -\alpha & 1+\alpha & \nu'+\alpha \\ 0 & 0 & -\alpha & 0 & 0 & -\alpha & \nu'+\alpha & 1+\alpha \end{bmatrix} \times \begin{Bmatrix} X_1 \\ Y_1 \\ u_2 \\ Y_2 \\ X_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ p/2 \\ 0 \\ 0 \\ 0 \\ p/2 \\ 0 \end{Bmatrix} \tag{1.113}$$

Equation (1.113) can be solved numerically by, for instance, the Gauss elimination procedure. The solutions for Equation (1.113) are

$$u_2 = u_4 = p/E', \quad v_3 = v_4 = -\nu' p/E', \quad X_1 = X_3 = -p/2, \quad Y_1 = Y_2 = 0 \tag{1.114}$$

(Procedure 6).

The strains and stresses in the square plate can be calculated by substituting the solutions (1.114) into Equations (1.73) and (1.74), respectively (Procedure 7). The resultant strains and stresses are given by the following equations:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = [\mathbf{B}] \begin{Bmatrix} u_4 \\ v_4 \\ u_3 \\ v_3 \\ u_2 \\ v_2 \end{Bmatrix} = \frac{p}{E'} \begin{bmatrix} 1 & 0 & -1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & -1 \\ 1 & 1 & 0 & 1 & -1 & 0 \end{bmatrix} \begin{Bmatrix} 1 \\ -\nu' \\ 0 \\ \nu' \\ 1 \\ 0 \end{Bmatrix}$$

$$= \frac{p}{E'} \begin{Bmatrix} 1 \\ -\nu' \\ 1 - \nu' + \nu' - 1 \end{Bmatrix} = \frac{p}{E'} \begin{Bmatrix} 1 \\ -\nu' \\ 0 \end{Bmatrix} \tag{1.115}$$

and

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E'}{1 - \nu'^2} \begin{bmatrix} 1 & \nu' & 0 \\ \nu' & 1 & 0 \\ 0 & 0 & \dfrac{1 - \nu'}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{p}{1 - \nu'^2} \begin{bmatrix} 1 & \nu' & 0 \\ \nu' & 1 & 0 \\ 0 & 0 & \dfrac{1 - \nu'}{2} \end{bmatrix} \begin{Bmatrix} 1 \\ -\nu' \\ 0 \end{Bmatrix}$$

$$= \frac{p}{1 - \nu'^2} \begin{Bmatrix} 1 - \nu'^2 \\ 0 \\ 0 \end{Bmatrix} = p \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix} \tag{1.116}$$

The results obtained by the present finite-element calculations imply that a square plate subjected to uniaxial uniform tension in the x-direction is elongated by a uniform strain of p/E' in the loading direction, whereas it is contacted by a uniform strain of $-\nu' p/E'$ in the direction perpendicular to the loading direction and that only a uniform normal stress of $\sigma_x = p$ is induced in the plate. The result that the nodal reaction forces at the nodes 1 and 3 are equal to $-p/2$, i.e., $X_1 = X_3 = -p/2$ implies that a uniform reaction force of $-p$ is produced along the side $\overline{13}$. It is concluded that the above results obtained by the FEM agree well with the physical interpretations of the present problem.

Bibliography

1. O. C. Zienkiewicz and K. Morgan, *Finite Elements and Approximation*, John Wiley & Sons, New York (1983).
2. O. C. Zienkiewicz and R. Taylor, *The Finite Element Method* (4th edn), Vol. 1, McGraw-Hill, London (1989).

3. G. W. Rowe *et al.*, *Finite-Element Plasticity and Metalforming Analysis*, Cambridge University Press, (1991).
4. C. L. Dym and I. H. Shames, *Solid Mechanics*: *A Variational Approach*, McGraw-Hill, New York (1973).
5. K. Washizu, *Variational Methods in Elasticity and Plasticity* (2nd edn), Pergamon, New York (1975).
6. K.-J. Bathe, *Finite Element Procedures*, Printice-Hall Int., New Jersy (1975).
7. B. A. Finlayson, *The Method of Weighted Residuals and Variational Principles*, Academic Press, New York (1972).
8. M. Gotoh, *Engineering Finite Element Method – For Analysis of Large Elastic–Plastic Deformation*, Corona Publishing Co., Ltd., Tokyo (1995) (in Japanese).
9. H. Togawa, *Introduction to the Finite Element Method*, Baifukan Co., Ltd., Tokyo (1984) (in Japanese).
10. G. Yagawa *et al.*, *Computational Mechanics*, Iwanami Shoten Publishers, Tokyo (2000) (in Japanese).
11. K. Washizu *et al.*, *Handbook of the Finite Element Method (Basics)*, Baifukan Co., Ltd., Tokyo (1994) (in Japanese).

This page intentionally left blank

CHAPTER 2

OVERVIEW OF ANSYS STRUCTURE AND VISUAL CAPABILITIES

CHAPTER OUTLINE

2.1 INTRODUCTION 37
2.2 STARTING THE PROGRAM 38
2.3 PREPROCESSING STAGE 43
2.4 SOLUTION STAGE 49
2.5 POSTPROCESSING STAGE 50

2.1 INTRODUCTION

ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

In general, a finite-element solution may be broken into the following three stages.

(1) *Preprocessing: defining the problem*
The major steps in preprocessing are (i) define keypoints/lines/areas/volumes, (ii) define element type and material/geometric properties, and (iii) mesh lines/areas/ volumes as required.

The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

(2) *Solution: assigning loads, constraints, and solving*
Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally solve the resulting set of equations.

(3) *Postprocessing: further processing and viewing of the results*
In this stage one may wish to see (i) lists of nodal displacements, (ii) element forces and moments, (iii) deflection plots, and (iv) stress contour diagrams or temperature maps.

2.2 Starting the program

2.2.1 Preliminaries

There are two methods to use ANSYS. The first is by means of the graphical user interface or GUI. This method follows the conventions of popular Windows and X-Windows based programs. The GUI method is exclusively used in this book.

The second is by means of command files. The command file approach has a steeper learning curve for many, but it has the advantage that the entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements.

The ANSYS environment contains two windows: the Main Window and an Output Window. Within the Main Window there are five divisions (see Figure 2.1):

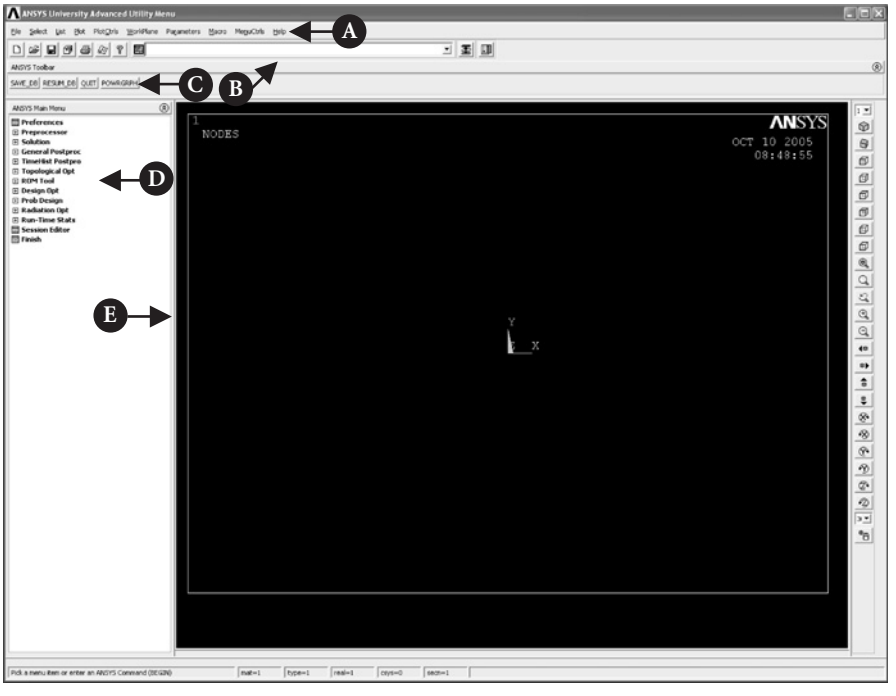


Figure 2.1 Main window of ANSYS.

(1) *Utility Menu*: The Utility Menu [A] contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls, and parameters.

(2) *Input Line*: The Input Line [B] shows program prompt messages and allows to type in commands directly.

(3) *Toolbar*: The Toolbar [C] contains push buttons that execute commonly used ANSYS commands. More push buttons can be made available if desired.

(4) *Main Menu*: The Main Menu [D] contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, and design optimizer. It is from this menu that the vast majority of modeling commands are issued.

(5) *Graphics Windows*: The Graphics Window [E] is where graphics are shown and graphical picking can be made. It is here where the model in its various stages of construction and the ensuing results from the analysis can be viewed.

The Output Window, shown in Figure 2.2, displays text output from the program, such as listing of data, etc. It is usually positioned behind the Graphics Window and can be put to the front if necessary.

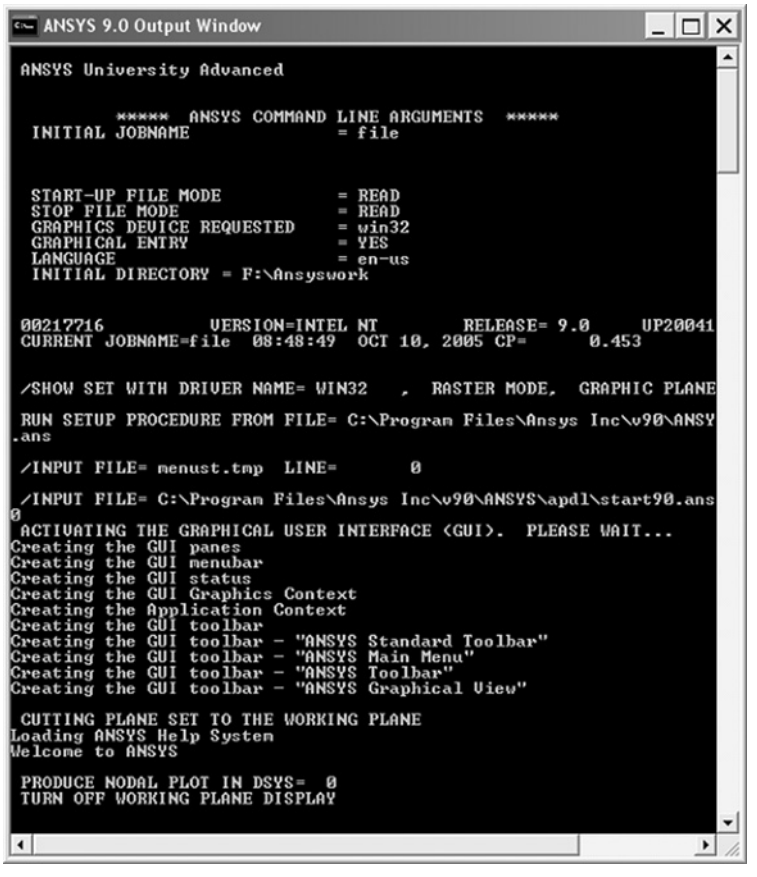


Figure 2.2 Output window of ANSYS.

2.2.2 SAVING AND RESTORING JOBS

It is a good practice to save the model at various stages during its creation. Very often the stage in the modeling is reached where things have gone well and the model ought to be saved at this point. In that way, if mistakes are made later on, it will be possible to come back to this point.

To save the model, from **ANSYS Utility Menu** select, **File → Save as Jobname.db**.

The model will be saved in a file called Jobname.db, where Jobname is the name that was specified in the Launcher when ANSYS was first started. It is a good idea to save the job at different times throughout the building and analysis of the model to backup the work in case of a system crash or other unforeseen problems.

Alternatively, select **File → Save as**. In response to the second option frame, shown in Figure 2.3, is produced.

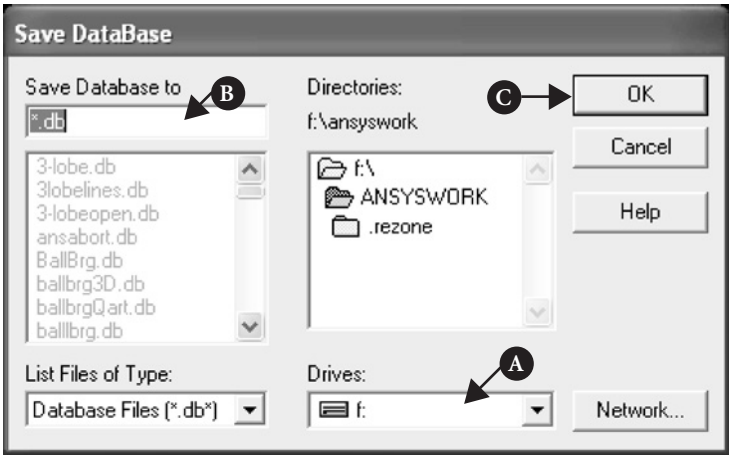


Figure 2.3 Save database of the problem.

Select appropriate drive [A] and give the file a name [B]. Clicking [C] **OK** button saves the model as a database with the name given.

Frequently there is a need to start up ANSYS and recall and continue a previous job. There are two methods to do this:

(1) Using the Launcher. (i) In the ANSYS Launcher, select Interactive and specify the previously defined jobname. (ii) When ANSYS is running, select Utility Menu: File: Resume Jobname.db. This will restore as much of the database (geometry, loads, solution, etc.) as was previously saved.

(2) Start **ANSYS** and select **Utility Menu**: **File → Resume from** and click on the job from the list that appears. Figure 2.4 shows the resulting frame.

Select appropriate file from the list [A] and click [B] **OK** button to resume the analysis.

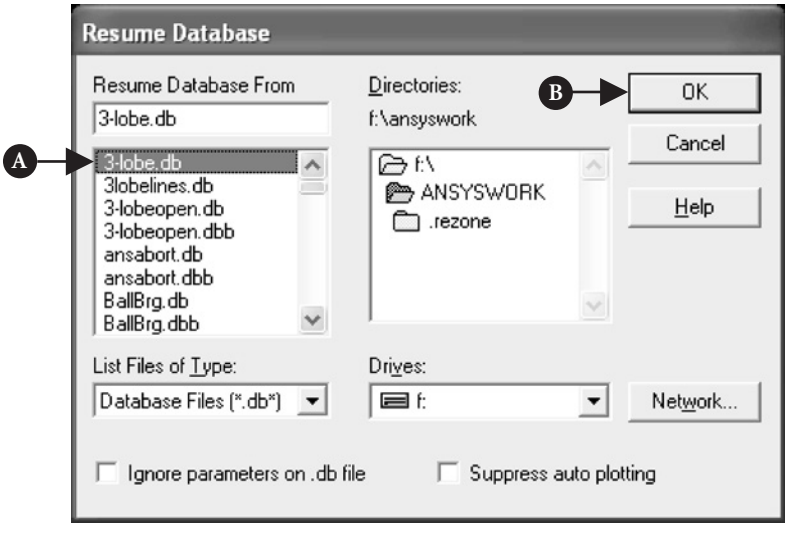


Figure 2.4 Resume problem from saved database.

2.2.3 Organization of files

A large number of files are created when ANSYS is run. If ANSYS is started without specifying a jobname, the name of all files created will be File.*, where the * represents various extensions described below. If a jobname is specified, say Beam, then the created files will all have the file prefix, Beam again with various extensions:

beam.db – database file (binary). This file stores the geometry, boundary conditions, and any solutions.

beam.dbb – backup of the database file (binary).

beam.err – error file (text). Listing of all error and warning messages.

beam.out – output of all ANSYS operations (text). This is what normally scrolls in the output window during ANSYS session.

beam.log – log file or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

Depending on the operations carried out, other files may have been written. These files may contain, for example, results. It is important to know what to save when, for instance, there is a need to clean up a directory or to move things from the /scratch directory. If the GUI is always used, then only the .db file is required. This file stores the geometry, boundary conditions, and any solutions. Once the ANSYS program has started, and the job name has been specified, only the resume command has to be activated to proceed from where the model was last left off. If, however, ANSYS command files are planned to be used, then only command file and/or the log file have to be stored. The log file contains a complete list of the ANSYS commands used to get the model to its current stage. That file may be run as is, or edited and rerun as desired.

2.2.4 PRINTING AND PLOTTING

ANSYS produces lists and tables of many types of results that are normally displayed on the screen. However, it is often desired to save the results to a file to be later analyzed or included in a report.

In case of stresses, instead of using **Plot Results** to plot the stresses, choose **List Results**. In the same way as described above. When the list appears on the screen in its own window, select File: **Save As** and give a file name to store the results. Any other solution can be done in the same way. For example, select **Nodal Solution** from the **List Results** menu to get displacements. Preprocessing and Solution data can be listed and saved from the **List Menu** in the **ANSYS Utility Menu**. Save the resulting list in the same way described above.

When there is a need to quickly save an image of the entire screen or the current Graphics Window, from **ANSYS Utility Menu** select **Plot Ctrls → Hard Copy**. There are two options, namely **To Printer** (see Figure 2.5) and **To File** (see Figure 2.6).

In the frame shown in Figure 2.5, select the printer [A] from the list of printers and press [B] **Print** button.

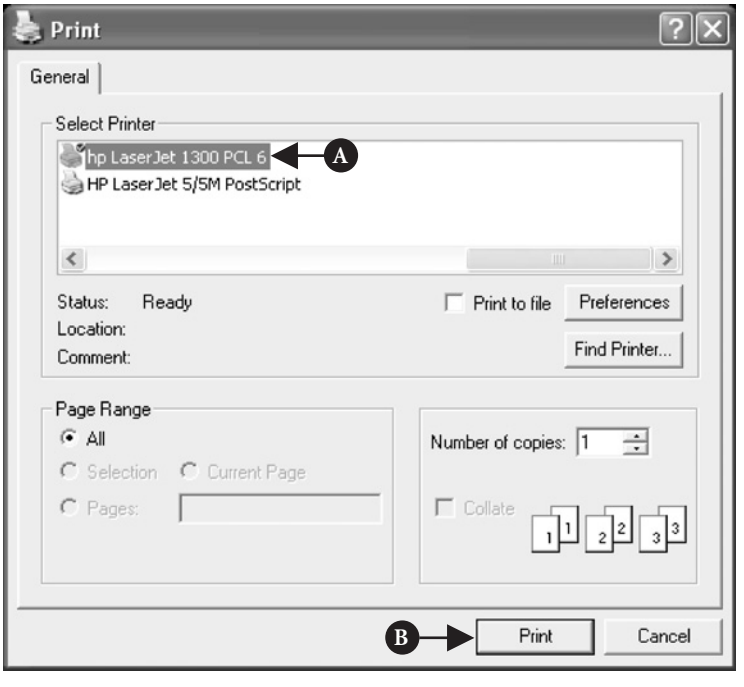


Figure 2.5 Print to printer.

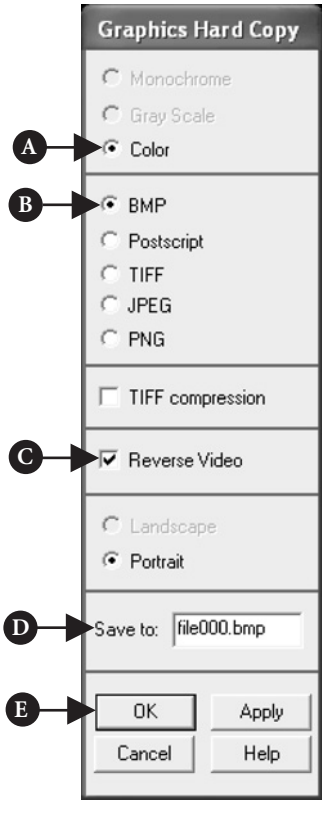


Figure 2.6 Print to file.

In the frame that is shown in Figure 2.6, an example selection could be [A] **Color**, [B] **BMP**, [C] **Reverse Video**, and finally [D] **Save to**. Then enter an appropriate file name and click [E] **OK** button. This image file may now be printed on a PostScript printer or included in a document.

2.2.5 Exiting the program

The ANSYS program can be exited in a number of different ways. When the current analysis is saved from **Utility Menu** select **File → Exit**. A frame shown in Figure 2.7 appears.

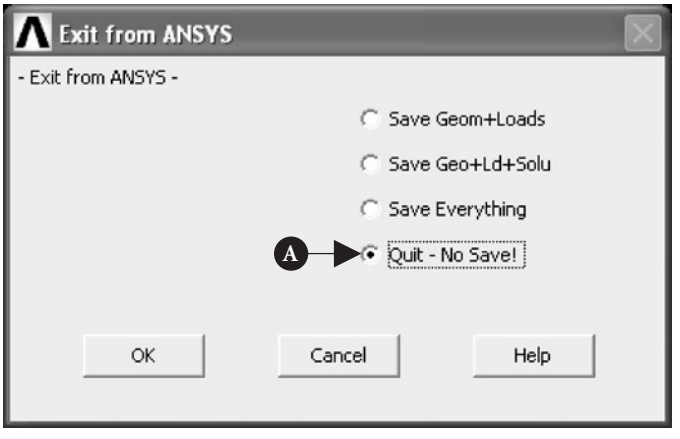


Figure 2.7 Exit from ANSYS.

There are four options, depending on what is important to be saved. If nothing is to be saved then select [A] **Quite – No Save** as indicated.

2.3 Preprocessing stage

2.3.1 Building a model

The ANSYS program has many finite-element analysis capabilities, ranging from a simple linear static analysis to a complex nonlinear transient dynamic analysis. Building a finite-element model requires more time than any other part of the analysis. First, a jobname and analysis title have to be specified. Next, the PREP7 preprocessor is used to define the element types, element real constants, material properties, and the model geometry. It is important to remember that ANSYS does not assume a system of units for intended analysis. Except in magnetic field analyses, any system of units can be used so long as it is ensured that units are consistent for all input data. Units

cannot be set directly from the GUI. In order to set units as the international system of units (SI) from **ANSYS Main Menu**, select **Preprocessor → Material Props → Material Library → Select** Units. Figure 2.8 shows the resulting frame.

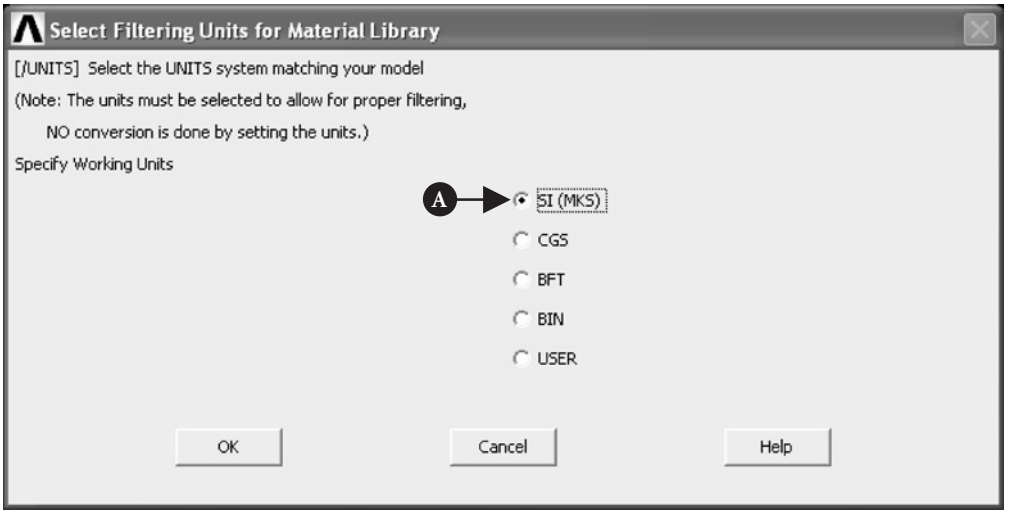


Figure 2.8 Selection of units for the problem.

Activate [A] **SI** (MKS) button to inform the ANSYS program that this system of units is proposed to be used in the analysis.

2.3.1.1 Defining element types and real constants

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category. In order to define element types, one must be in PREP7. From **ANSYS Main Menu**, select **Preprocessor or → Element Type → Add/Edit/Delete**. In response, the frame shown in Figure 2.9 appears.

Click on [A] **Add** button and a new frame, shown in Figure 2.10, appears.

Select an appropriate element type for the analysis performed, e.g., [A] **Solid** and [B] **8node 183** as shown in Figure 2.10.

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. As with element types, each set of real constant has a reference number and the table of reference number versus real constant set is called the real constant table. Not all element types require real constant, and different elements of the same type may have different real constant values. **ANSYS Main Menu** command **Preprocessor → Modeling → Create →**

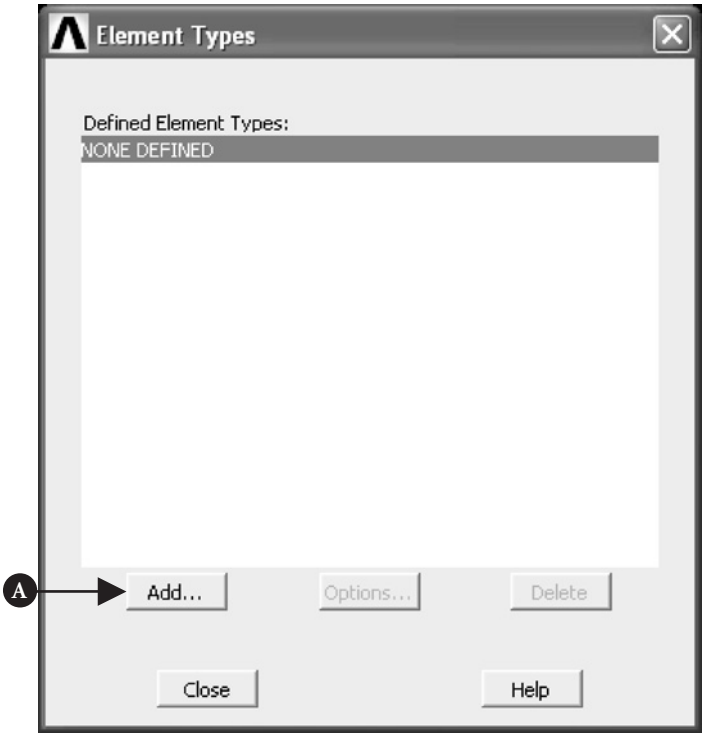


Figure 2.9 Definition of element types to be used.

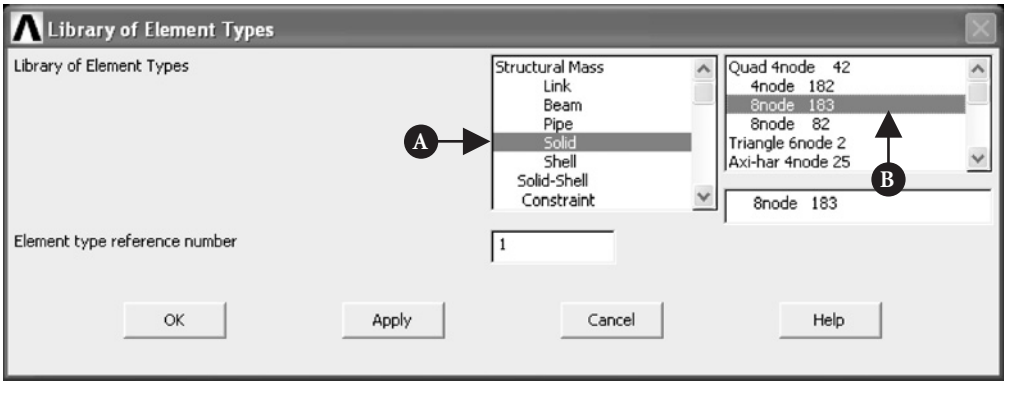


Figure 2.10 Selection of element types from the library.

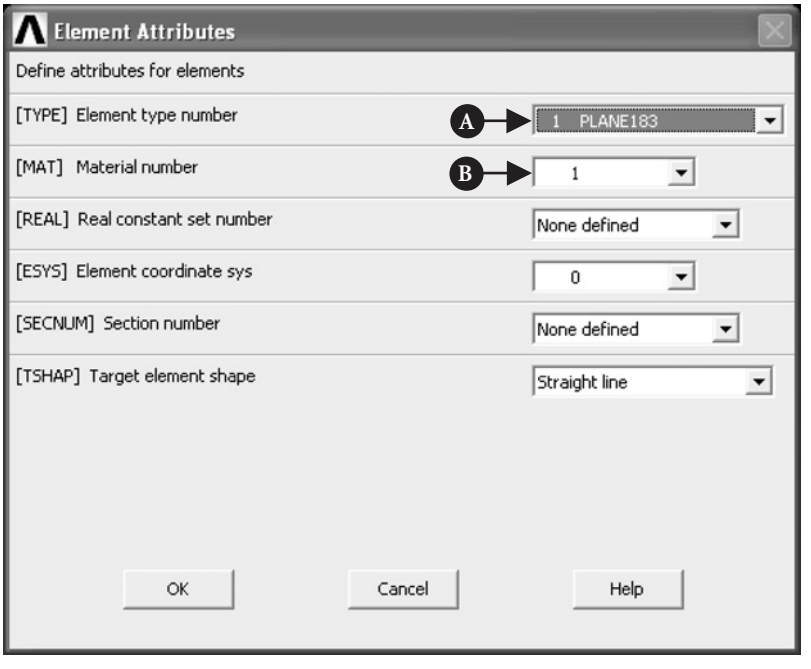


Figure 2.11 Element Attributes.

Elements → Element Attributes can be used to define element real constant. Figure 2.11 shows a frame in which one can select element type. According to Figure 2.11, an element type already selected is [A] **Plane183** for which real constant is being defined. A corresponding [B] **Material number**, allocated by ANSYS when material properties are defined (see Section 2.3.1.2) is also shown in the frame.

Other element attributes can be defined as required by the type of analysis performed. Chapter 7 contains sample problems where elements attributes are defined in accordance with the requirements of the problem.

2.3.1.2 Defining material properties

Material properties are required for most element types. Depending on the application, material properties may be linear or nonlinear, isotropic, orthotropic or anisotropic, constant temperature or temperature dependent. As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. In one analysis there may be multiple material property sets corresponding with multiple materials used in the model. Each set is identified with a unique reference number. Although material properties can be defined separately for each finite-element analysis, the ANSYS program enables storing a material property set in an archival material library file, then retrieving the set and reusing it in multiple

analyses. Each material property set has its own library file. The material library files also make it possible for several users to share commonly used material property data.

In order to create an archival material library file, the following steps should be followed:

(i) Tell the ANSYS program what system of units is going to be used.

(ii) Define properties of, for example, isotropic material. Use **ANSYS Main Menu** and select **Preprocessor → Material Props → Material Models**. A frame shown in Figure 2.12 appears.

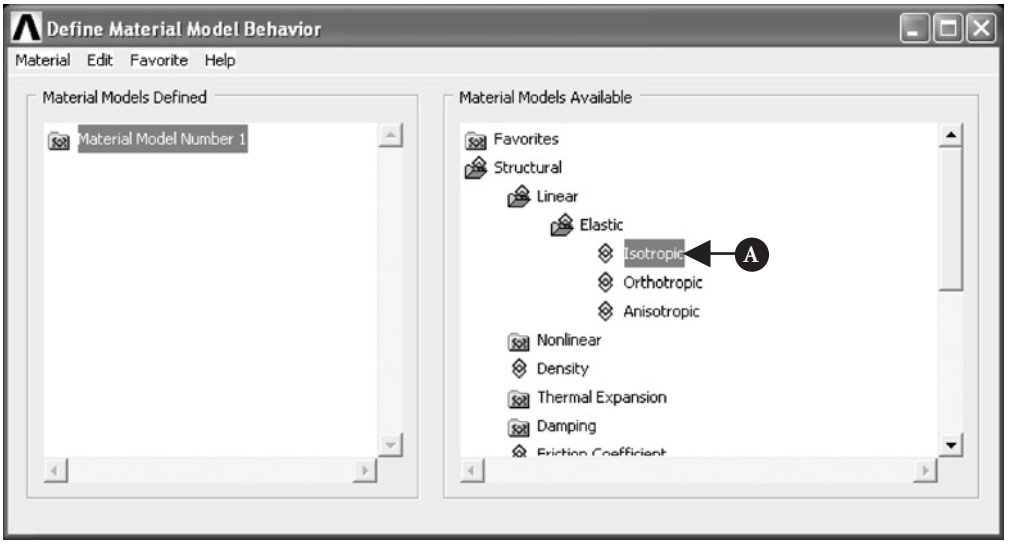


Figure 2.12 Define Material Model Behavior.

As shown in Figure 2.12, [A] **Isotropic** was chosen. Clicking twice on [A] **Isotropic** calls up another frame shown in Figure 2.13.

Enter data characterizing the material to be used in the analysis into appropriate field. For example, [A] **EX** = 2.1E+009 and [B] **PRXY** = 0.33 as shown in Figure 2.13. If the problem requires a number of different materials to be used, then the above procedure should be repeated and another material model created with appropriate material number allocated by the program.

2.3.2 Construction of the model

2.3.2.1 Creating the model geometry

Once material properties are defined, the next step in an analysis is generating a finite-element model – nodes and element adequately describing the model geometry. There

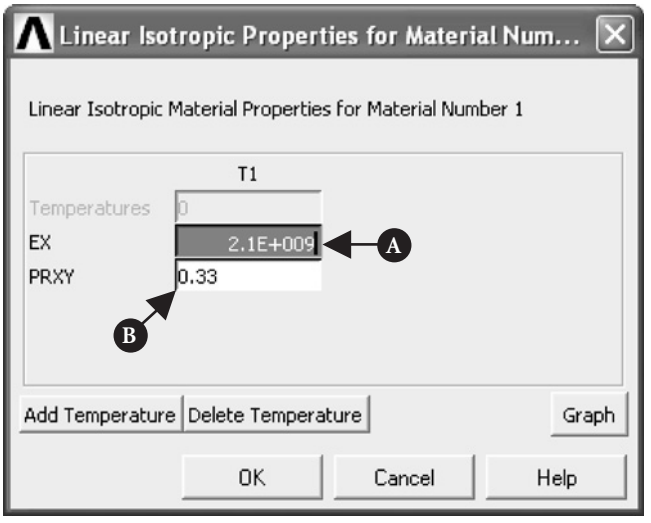


Figure 2.13 Linear isotropic material properties.

are two methods to create the finite-element model: solid modeling and direct generation. With solid modeling, the geometry of shape of the model is described, and then the ANSYS program automatically meshes the geometry with nodes and elements. The size and shape of the elements that the program creates can be controlled. With direct generation, the location of each node and the connectivity of each element is manually defined. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc., are available.

Solved example problems in this book amply illustrate, in a step-by-step manner, how to create the model geometry.

2.3.2.2 Applying loads

Loads can be applied using either **PREP7** preprocessor or the **SOLUTION** processor. Regardless of the chosen strategy, it is necessary to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite-element solution.

The analysis type to be used is based on the loading conditions and the response which is wished to calculate. For example, if natural frequencies and mode shapes are to be calculated, then a modal analysis ought to be chosen. The ANSYS program offers the following analysis types: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring. Not all analysis types are valid for all disciplines. Modal analysis, for instance, is not valid for thermal models. Analysis options allow for customization of analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton–Raphson options. In order to define the analysis type and analysis options, use **ANSYS Main Menu** and select **Main Menu**: **Preprocessor → Loads → Analysis Type → New Analysis**. In response to the selection, the frame shown in Figure 2.14 appears.

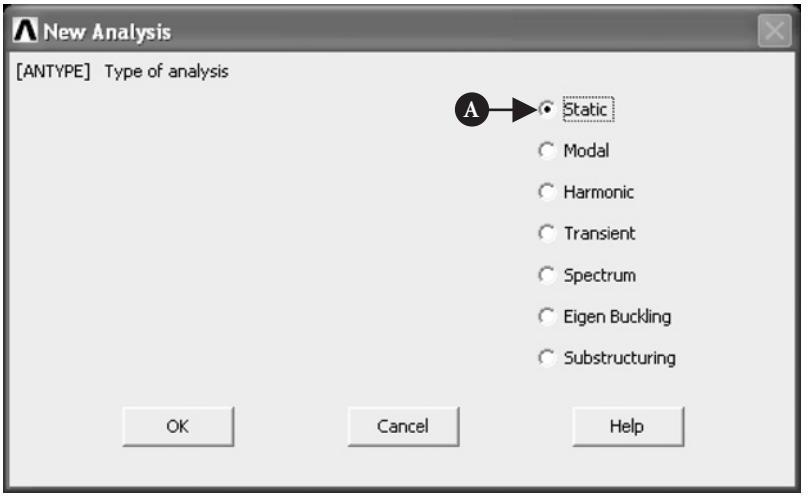


Figure 2.14 Type of analysis definition.

Select the type of analysis that is appropriate for the problem at hand by activating [A] **Static** button for example.

The word loads used here includes boundary conditions, i.e., constraints, supports, or boundary field specifications. It also includes other externally and internally applied loads. Loads in the ANSYS program are divided into six categories: DOF constraints, forces, surface loads, body loads, inertia loads, and coupled field loads. Most of these loads can be applied either on the solid model (keypoints, lines, and areas) or the finite-element model (nodes and elements).

There are two important load-related terms. A load step is simply a configuration of loads for which the solution is obtained. In a structural analysis, for instance, wind loads may be applied in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Substeps are incremental steps taken within a load step. They are mainly used for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as time steps which are taken over a period of time.

Load step options are alternatives that can be changed from load step to load step, such as number of substeps, time at the end of a load step, and output controls. Depending on the type of analysis performed, load step options may or may not be required. Sample problems solved here provide practical guide to appropriate load step options as necessary.

2.4 Solution stage

To initiate solution calculations, use **ANSYS Main Menu** selecting **Solution → Solve → Current LS**. Figure 2.15 shows resulting frame.

Figure 2.15 Start solution of current problem.

After reviewing the summary information about the model, click [A] **OK** button to start the solution. When this command is issued, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file and also to the database. The only difference is that only one set of results can reside in the database at one time, while a number of result sets can be written to the results file.

Once the solution has been calculated, the ANSYS postprocessors can be used to review the results.

2.5 Postprocessing stage

Two postprocessors are available:

(1) *POST1*: The general postprocessor is used to review results at one substep (time step) over the entire model or selected portion of the model. The command to enter POST1 requires selection from **ANSYS Main Menu General Postprocessor**. Using this postprocessor contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis can be obtained. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.

(2) *POST26*: The time history postprocessor is used to review results at specific points in the model over all time steps. The command to enter POST26 is as follows: from **ANSYS Main Menu** select **TimeHist Postprocessor**. Graph plots of results data versus time (or frequency) and tabular listings can be obtained. Other POST26 capabilities include arithmetic calculations and complex algebra.

Chapter 3

Application of ANSYS to Stress Analysis

Chapter outline

3.1	Cantilever beam	51
3.2	The principle of St. Venant	84
3.3	Stress concentration due to elliptic holes	93
3.4	Stress singularity problem	106
3.5	Two-dimensional contact stress	120
References		141

3.1 Cantilever beam

Beams are important fundamental structural and/or machine elements; they are found in buildings and in bridges. Beams are also used as shafts in cars and trains, as wings in aircrafts and bookshelves in bookstores. Arms and femurs of human beings and branches of trees are good examples of portions of living creatures which support their bodies. Beams play important roles not only in inorganic but also in organic structures.

Mechanics of beams is one of the most important subjects in engineering.

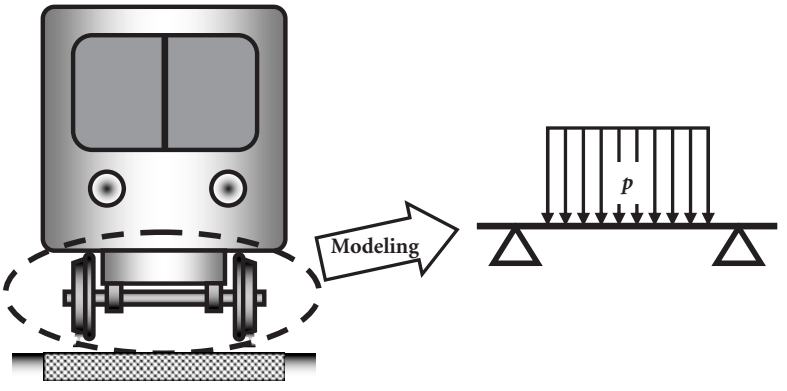


Figure 3.1 Modeling of an axle shaft by a simply supported beam.

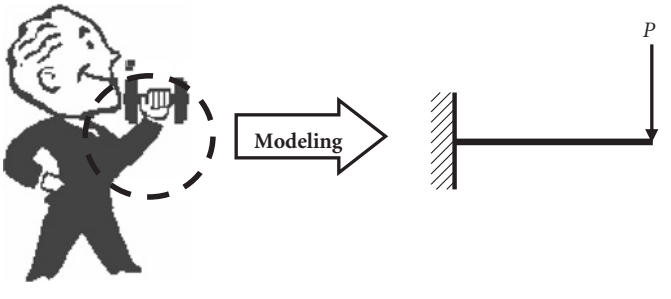


Figure 3.2 Modeling of an arm of a human being by a cantilever beam.

3.1.1 Example problem: a cantilever beam

Perform an finite-element method (FEM) analysis of a 2-D cantilever beam shown in Figure 3.3 below and calculate the deflection of the beam at the loading point and the longitudinal stress distribution in the beam.

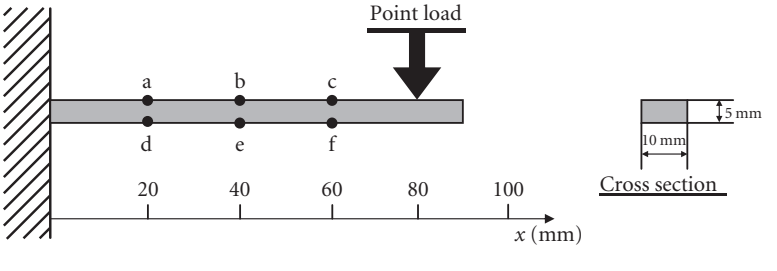


Figure 3.3 Bending of a cantilever beam to solve.

3.1.2 Problem description

Geometry: length $l = 90$ mm, height $h = 5$ mm, thickness $b = 10$ mm
Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.
Boundary conditions: The beam is clamped to a rigid wall at the left end and loaded at $x = 80$ mm by a point load of $P = 100$ N.

3.1.2.1 Review of the solutions obtained by the elementary beam theory

Before proceeding to the FEM analysis of the beam, let us review the solutions to the example problem obtained by the elementary beam theory. The maximum deflection of the beam $\delta_{\max}$ can be calculated by the following equation:

$$\delta_{\max} = \frac{Pl_1^3}{3EI}\left(1 + \frac{3l_1}{l_2}\right) \tag{3.1}$$

where l_1 (=80 mm) is the distance of the application point of the load from the rigid wall and $l_2 = l - l_1$.

The maximum tensile stress $\sigma_{\max}(x)$ at x in the longitudinal direction appears at the upper surface of the beam in a cross section at x from the wall;

$$\sigma_{\max}(x) = \begin{cases} \frac{P(l_1 - x)}{I}\frac{h}{2} & (0 \le x \le l_1) \\ 0 & (0 \le x) \end{cases} \tag{3.2}$$

where l (=90 mm) is the length, h (=5 mm) the height, b (=10 mm) the thickness, E (=210 GPa) Young's modulus and I the area moment of inertia of the cross section of the beam. For a beam having a rectangular cross section of a height h by a thickness b, the value of I can be calculated by the following equation:

$$I = \frac{bh^3}{12} \tag{3.3}$$

3.1.3 Analytical procedures

Figure 3.4 shows how to make structural analyses by using FEM. In this chapter, the analytical procedures will be explained following the flowchart illustrated in Figure 3.4.

3.1.3.1 Creation of an analytical model

[1] Creation of a beam shape to analyze
Here we will analyze a rectangular slender beam of 5 mm (0.005 m) in height by 90 mm (0.09 m) in length by 10 mm (0.01 m) in width as illustrated in Figure 3.3.

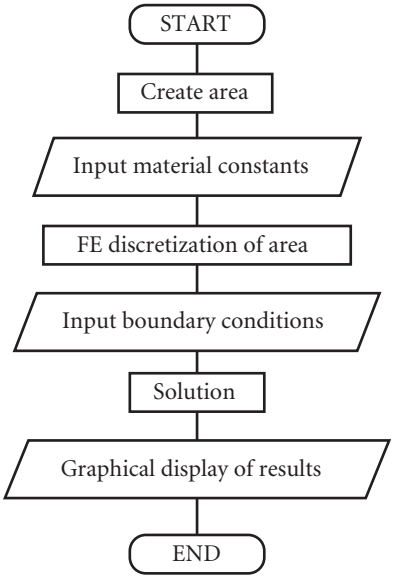


Figure 3.4 Flowchart of the structural analyses by ANSYS.

Figure 3.5 shows the "ANSYS Main Menu" window where we can find layered command options imitating folders and files in the Microsoft® Explorer folder window.

In order to prepare for creating the beam, the following operations should be made:

(1) Click [A] **Preprocessor** to open its sub-menus in **ANSYS Main Menu** window.
(2) Click [B] **Modeling** to open its sub-menus and select **Create** menu.
(3) Click [C] **Areas** to open its sub-menus and select **Rectangle** menu.
(4) Click to select [D] **By 2 corners** menu.

After carrying out the operations above, a window called **Rectangle by 2 corners** as shown in Figure 3.6 appears for the input of the geometry of a 2-D rectangular beam.

[2] Input of the beam geometry to analyze

The **Rectangle by 2 corners** window has four boxes for inputting the coordinates of the lower left corner point of the rectangular beam and the width and height of the beam to create. The following operations complete the creation procedure of the beam:

(1) Input two **0**'s into [A] **WP X** and [B] **WP Y** to determine the lower left corner point of the beam on the Cartesian coordinates of the working plane.
(2) Input **0.09** and **0.005** (m) into [C] **Width** and [D] **Height**, respectively to determine the shape of the beam model.
(3) Click [E] **OK** button to create the rectangular area, or beam on the **ANSYS Graphics** window as shown in Figure 3.7.

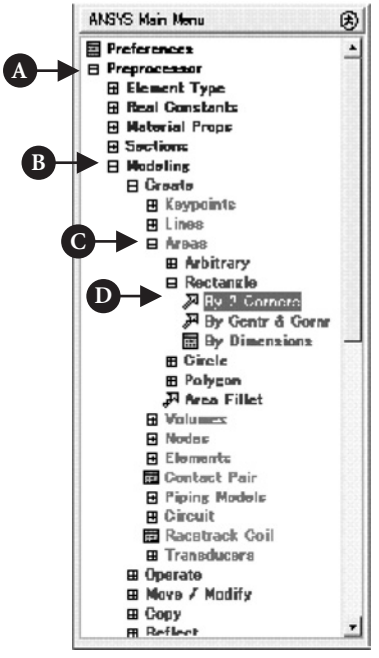


Figure 3.5 "ANSYS Main Menu" window.

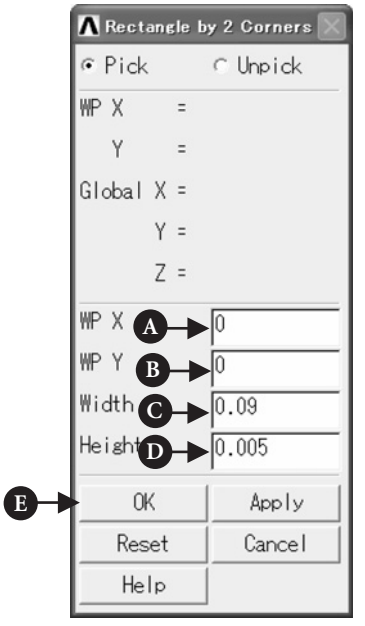


Figure 3.6 "Rectangle By 2 Corners" window.

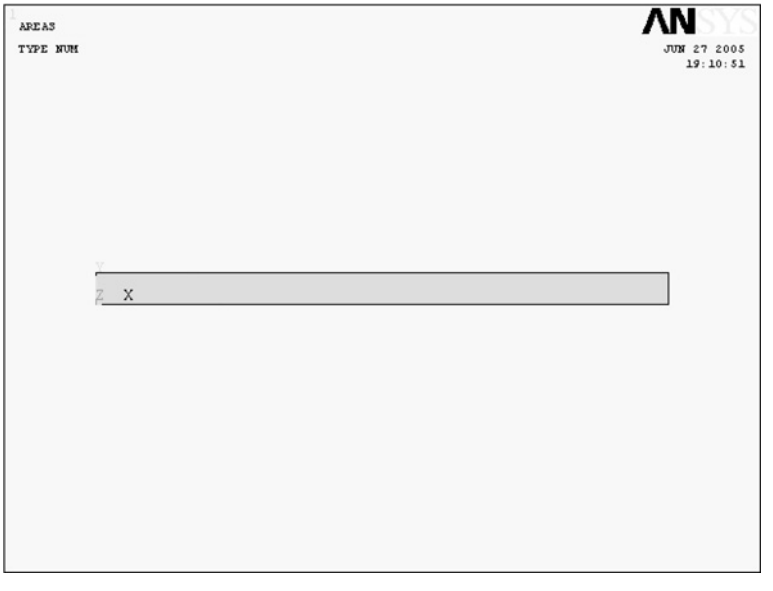


Figure 3.7 2-D beam created and displayed on the "ANSYS Graphics" window.

How to correct the shape of the model
In case of correcting the model, delete the area first, and repeat the procedures [1] and [2] above. In order to delete the area, execute the following commands:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Delete → Area and Below**

Then, the **Delete Area and Below** window opens and an upward arrow (↑) appears on the **ANSYS Graphics** window:

(1) Move the arrow to the area to delete and click the left button of the mouse.
(2) The color of the area turns from light blue into pink.
(3) Click **OK** button and the area is to be deleted.

3.1.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE BEAM MATERIAL

Next, we specify elastic constants of the beam. In the case of isotropic material, the elastic constants are Young's modulus and Poisson's ratio. This procedure can be performed any time before the solution procedure, for instance, after setting boundary conditions. If this procedure is missed, we cannot perform the solution procedure.

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

Then the **Define Material Model Behavior** window opens as shown in Figure 3.8:

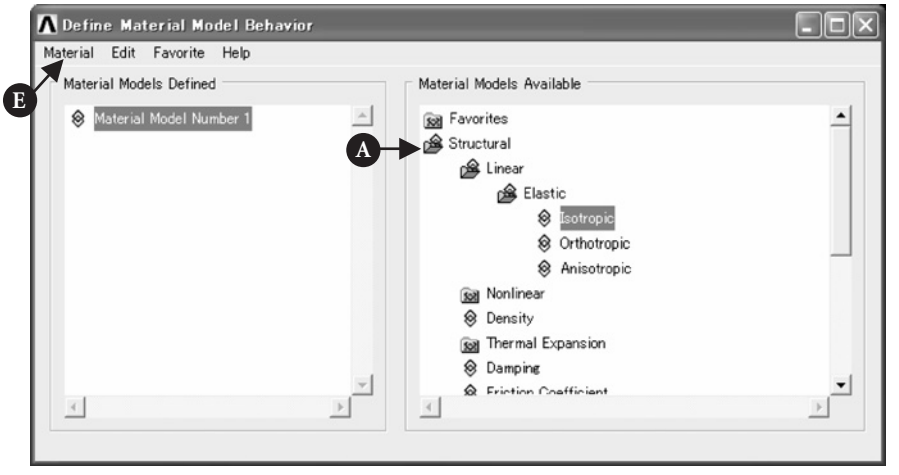


Figure 3.8 "Define Material Model Behavior" window.

(1) Double-click [A] **Structural**, **Linear**, **Elastic**, and **Isotropic** buttons one after another.

(2) Input the value of Young's modulus, **2.1e11** (Pa), and that of Poisson's ratio, **0.3**, into [B] **EX** and [C] **PRXY** boxes, and click [D] **OK** button of the **Linear Isotropic Properties for Material Number 1** as shown in Figure 3.9.

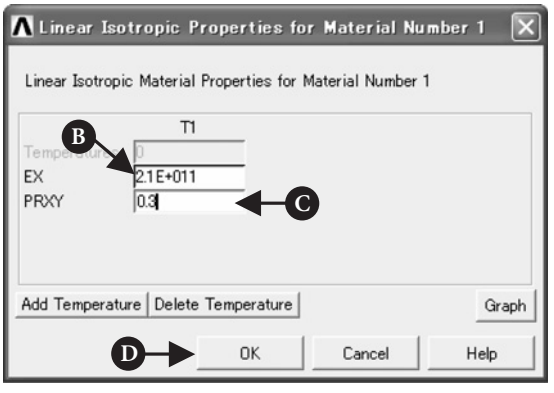


Figure 3.9 Input of elastic constants through the "Linear Isotropic Properties for Material Number 1" window.

(3) Exit from the **Define Material Model Behavior** window by selecting **Exit** in [E] **Material** menu of the window (see Figure 3.8).

3.1.3.3 FINITE-ELEMENT DISCRETIZATION OF THE BEAM AREA

Here we will divide the beam area into finite elements. The procedures for finite-element discretization are firstly to select the element type, secondly to input the element thickness and finally to divide the beam area into elements.

[1] Selection of the element type

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/ Edit/Delete**

Then the **Element Types** window opens:

(1) Click [A] **Add . . .** button to open the **Library of Element Types** window as shown in Figure 3.11 and select the element type to use.

(2) To select the 8-node isoparametric element, select [B] **Structural Mass – Solid**.

(3) Select [C] **Quad 8node 82** and click [D] **OK** button to choose the 8-node isoparametric element.

(4) Click [E] **Options . . .** button in the **Element Types** window as shown in Figure 3.10 to open the **PLANE82 element type options** window as depicted in Figure 3.12. Select [F] **Plane strs w/thk** item in the **Element behavior** box and click [G] **OK** button to return to the **Element Types** window. Click [H] **Close** button to close the window.

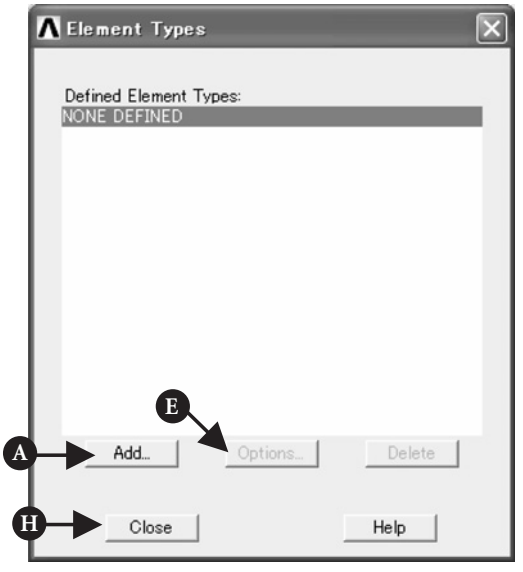


Figure 3.10 "Element Types" window.

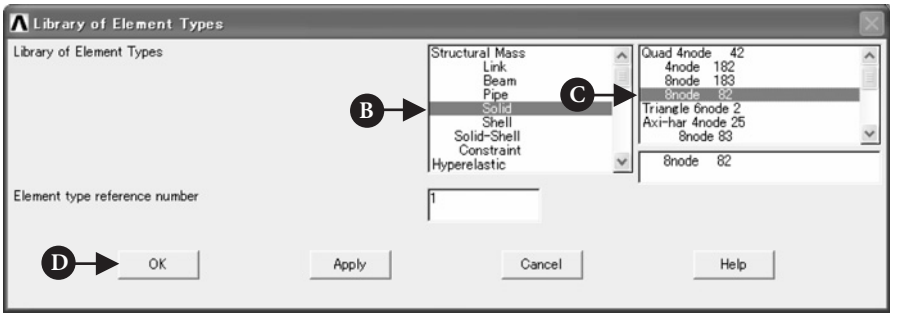


Figure 3.11 "Library of Element Types" window.

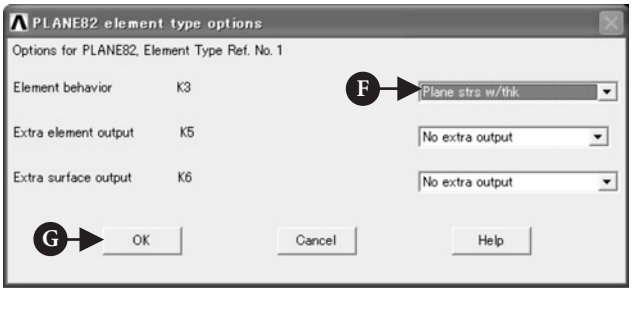


Figure 3.12 "PLANE82 element type options" window.

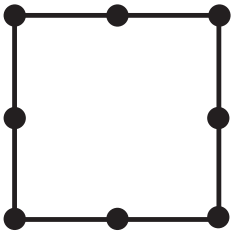


Figure 3.13 8-node isoparametric rectangular element.

The 8-node isoparametric element is a rectangular element which has four corner nodal points and four middle points as shown in Figure 3.13 and can realize the finite-element analysis with higher accuracy than the 4-node linear rectangular element. The beam area is divided into these 8-node rectangular #82 finite elements.

[2] Input of the element thickness

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants**

Select [A] **Real Constants** in the **ANSYS Main Menu** as shown in Figure 3.14.

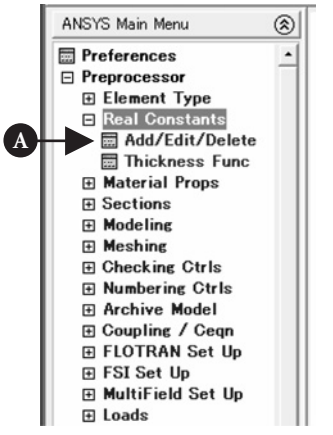


Figure 3.14 Setting of the element thickness from the real constant command.

(1) Click [A] **Add/Edit/Delete** button to open the **Real Constants** window as shown in Figure 3.15 and click [B] **Add …** button.

(2) Then the **Element Type for Real Constants** window opens (see Figure 3.16). Click [C] **OK** button.

(3) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead as shown in Figure 3.17.

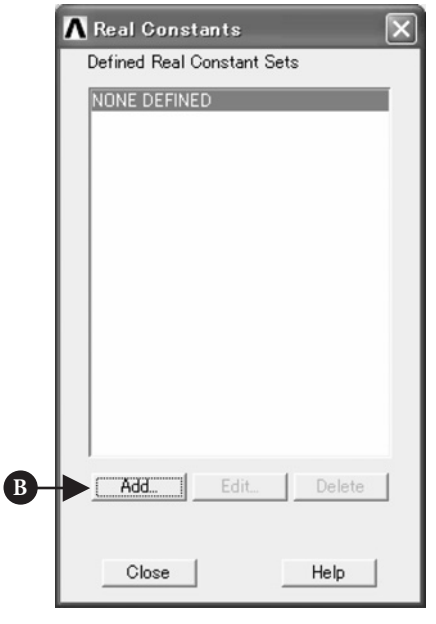


Figure 3.15 "Real Constants" window before setting the element thickness.

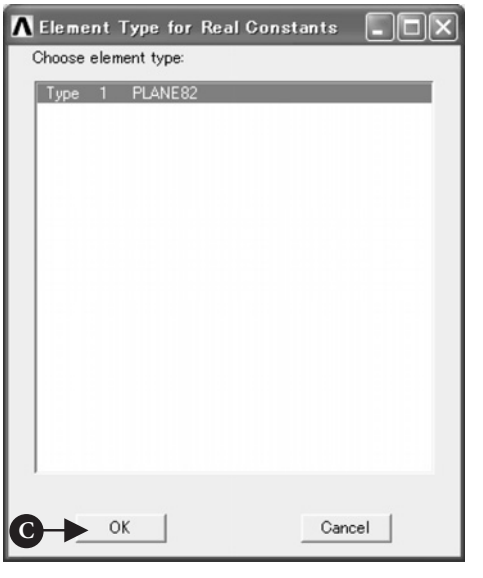


Figure 3.16 "Element Type for Real Constants" window.

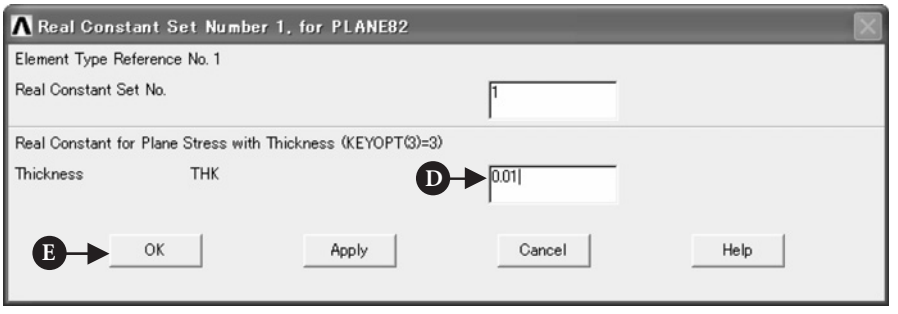


Figure 3.17 "Real Constants Set Number 1. for PLANE82" window.

Input a plate thickness of "**0.01**" (m) in [D] **Thickness** box and click [E] **OK** button.

(4) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1** as shown in Figure 3.18. Click [F] **Close** button, which makes the operation of setting the plate thickness completed.

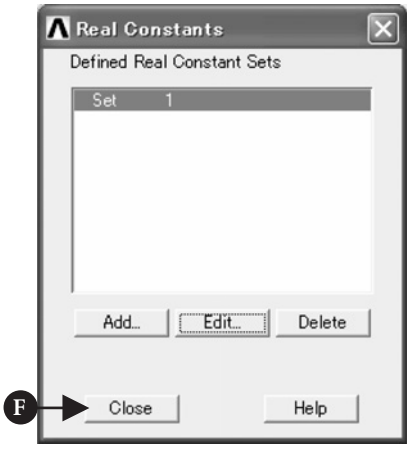


Figure 3.18 "Real Constants" window after setting the element thickness.

[3] Sizing of the elements

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size**

The **Global Element Sizes** window opens as shown in Figure 3.19:

(1) Input **0.002** (m) in [A] **SIZE** box and click [B] **OK** button.
By the operations above the element size of 0.002, i.e., 0.002 m, or 2 mm is specified and the beam of 5 mm by 90 mm is divided into rectangular finite-elements with one side 2 mm and the other side 3 mm in length.

[4] Meshing

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

The **Mesh Areas** window opens as shown in Figure 3.20:

(1) An upward arrow (↑) appears in the **ANSYS Graphics** window. Move this arrow to the beam area and click this area to mesh.

(2) The color of the area turns from light blue into pink. Click [A] **OK** button to see the area meshed by 8-node rectangular isoparametric finite elements as shown in Figure 3.21.

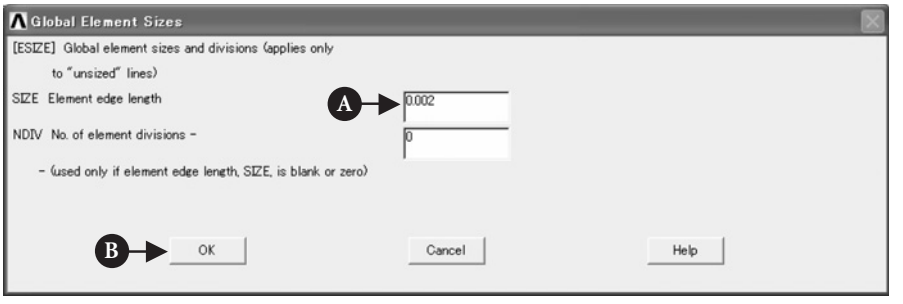


Figure 3.19 "Global Element Sizes" window.

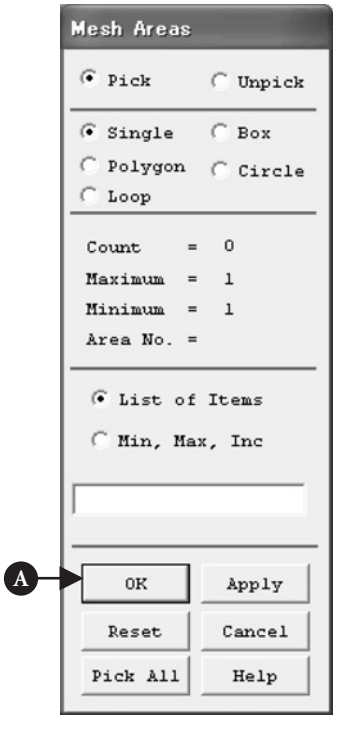


Figure 3.20 "Mesh Areas" window.

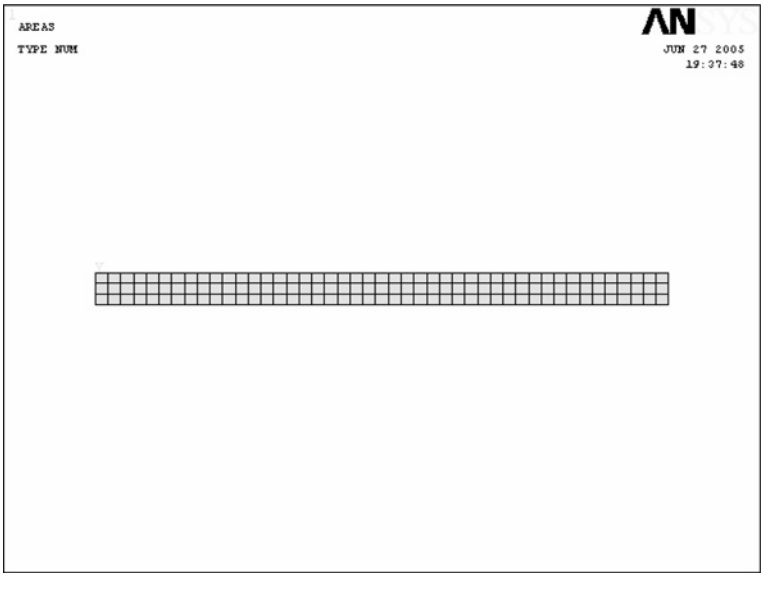


Figure 3.21 Beam area subdivided into 8-node isoparametric rectangular elements.

How to modify meshing

In case of modifying meshing, delete the elements, and repeat the procedures [1] through [4] above. Repeat from the procedures [1], from [2] or from [3] for modifying the element type, for changing the plate thickness without changing the element type, or for changing the element size only.

In order to delete the elements, execute the following commands:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Clear → Areas**

The **Clear Areas** window opens:

(1) An upward arrow (↑) appears in the **ANSYS Graphics** window. Move this arrow to the beam area and click this area.

(2) The color of the area turns from light blue into pink. Click **OK** button to delete the elements from the beam area. After this operation, the area disappears from the display. Execute the following commands to replot the area.

COMMAND | **ANSYS Utility Menu → Plot → Areas**

3.1.3.4 INPUT OF BOUNDARY CONDITIONS

Here we will impose constraint and loading conditions on nodes of the beam model. Display the nodes first to define the constraint and loading conditions.

[1] Nodes display

COMMAND | **ANSYS Utility Menu → Plot → Nodes**

The nodes are plotted in the **ANSYS Graphics** window opens as shown in Figure 3.22.

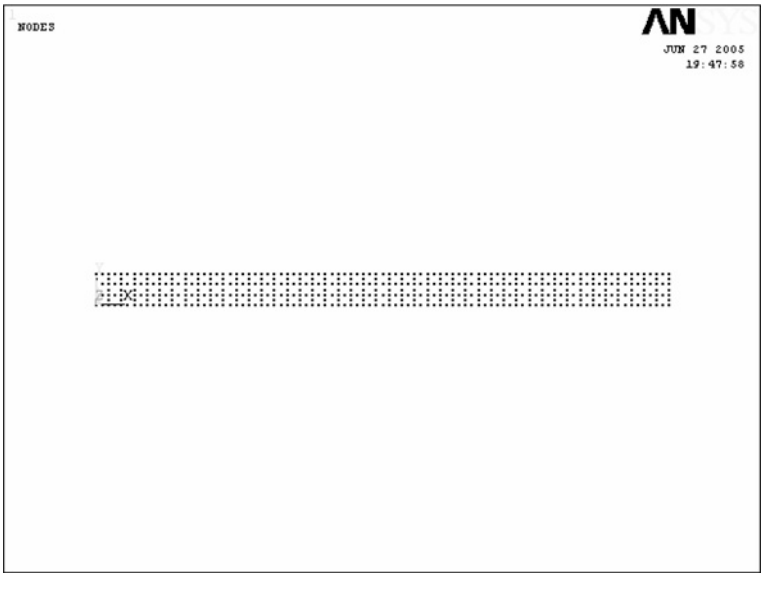


Figure 3.22 Plots of nodes.

[2] Zoom in the nodes display
Enlarged view of the models is often convenient when imposing constraint and loading conditions on the nodes. In order to zoom in the node display, execute the following commands:

COMMAND | **ANSYS Utility Menu → PlotCtrls → Pan Zoom Rotate . . .**

The **Pan-Zoom-Rotate** window opens as shown in Figure 3.23:

(1) Click [A] **Box Zoom** button.
(2) The shape of the mouse cursor turns into a magnifying glass in the **ANSYS Graphics** window. Click the upper left point and then the lower right point which enclose a portion of the beam area to enlarge as shown in Figure 3.24. Zoom in the left end of the beam.
(3) In order to display the whole view of the beam, click [B] **Fit** button in the "**Pan-Zoom-Rotate**" window.

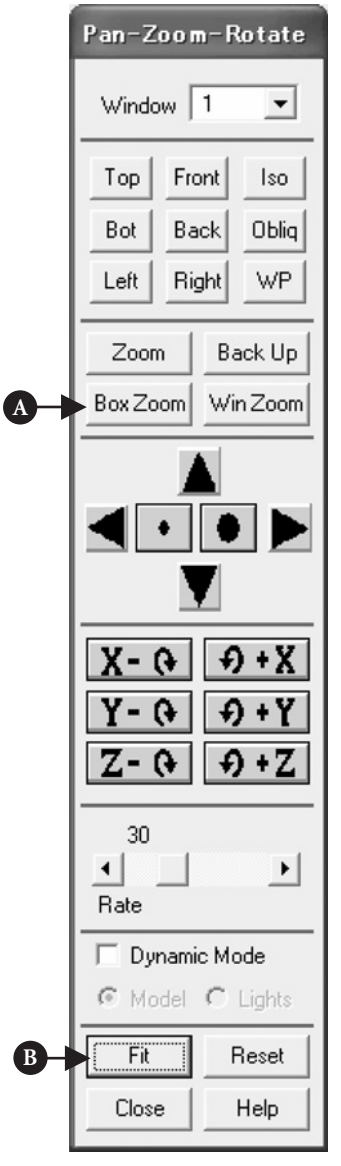


Figure 3.23 "Pan-Zoom-Rotate" window.

[3] Definition of constraint conditions
Selection of nodes

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes**

The **Apply U. ROT on Nodes** window opens as shown in Figure 3.25.

(1) Select [A] **Box** button and drag the mouse in the **ANSYS Graphics** window so as to enclose the nodes on the left edge of the beam area with the yellow rectangular frame as shown in Figure 3.26. The **Box** button is selected to pick multiple nodes at once, whereas [B] the "**Single**" button is chosen to pick a single node.
(2) After confirming that only the nodes to impose constraints on are selected, i.e., the nodes on the left edge of the beam area, click [C] **OK** button.

How to reselect nodes Click [D] **Reset** button to clear the selection of the nodes before clicking [C] **OK** button in the procedure (2) above, and repeat the procedures (1) and (2) above. The selection of

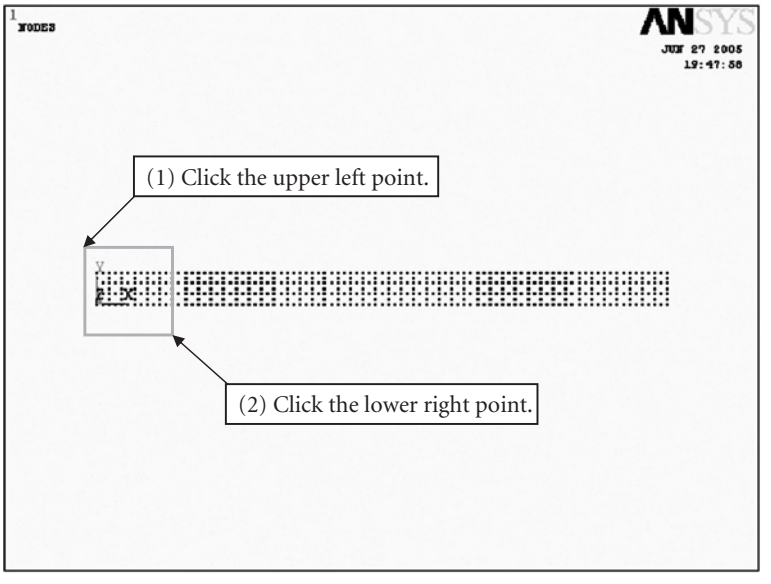


Figure 3.24 Magnification of an observation area.

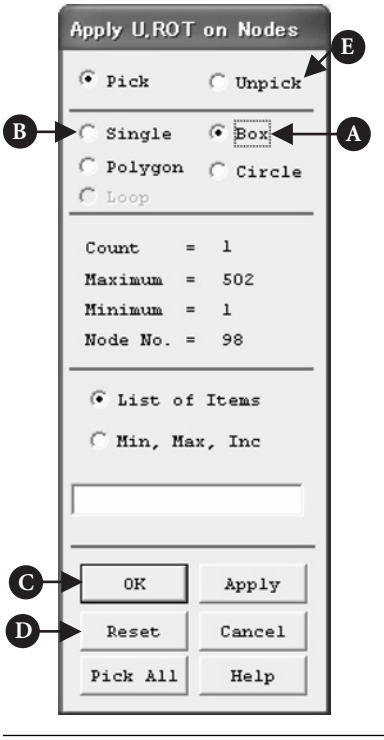


Figure 3.25 "Apply U. ROT on Nodes" window.

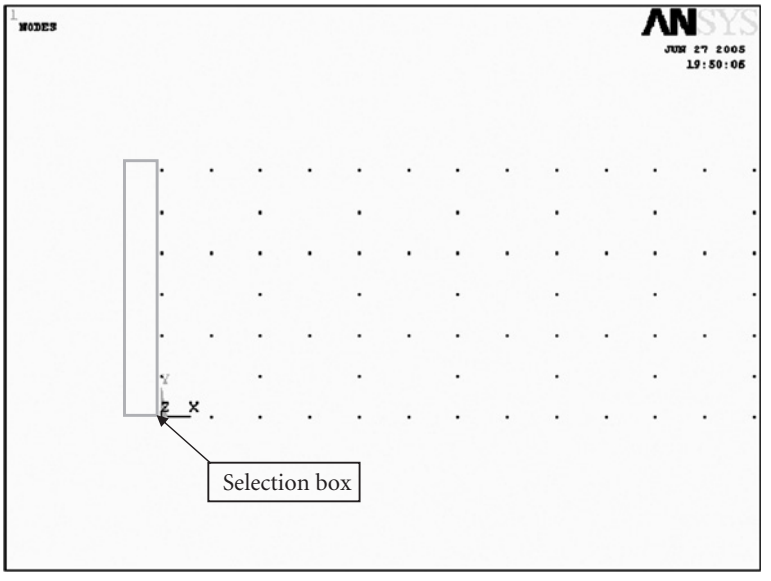


Figure 3.26 Picking multiple nodes by box.

nodes can be reset either by picking selected nodes after choosing [E] **Unpick** button or clicking the right button of the mouse to turn the upward arrow upside down.

Imposing constraint conditions on nodes
The **Apply U. ROT on Nodes** window (see Figure 3.27) opens after clicking [C] **OK** button in the procedure (2) in the subsection "Selection of nodes" above.

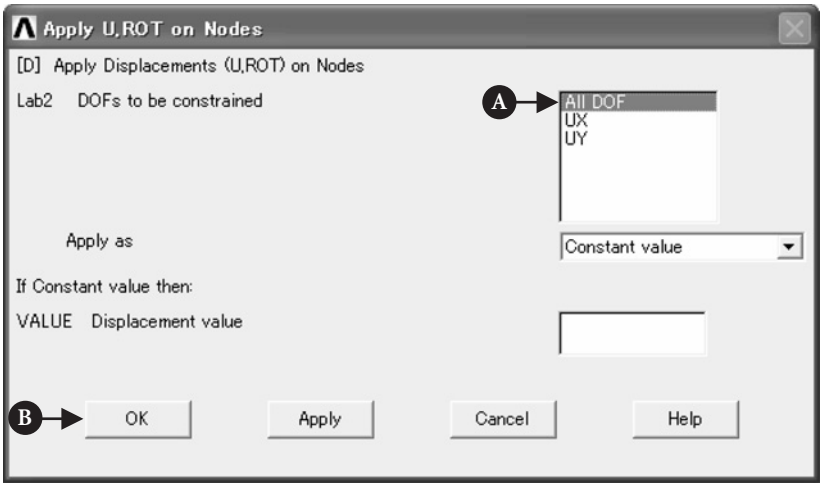


Figure 3.27 "Apply U. ROT on Nodes" window.

(1) In case of selecting [A] **ALL DOF**, the nodes are to be clamped, i.e., the displacements are set to zero in the directions of the x- and y-axes. Similarly, the selection of **UX** makes the displacement in the x-direction equal to zero and the selection of **UY** makes the displacement in the y-direction equal to zero.

(2) Click [B] **OK** button and blue triangular symbols, which denote the clamping conditions, appear in the **ANSYS Graphics** window as shown in Figure 3.28. The upright triangles indicate that each node to which the triangular symbol is attached is fixed in the y-direction, whereas the tilted triangles indicate the fixed condition in the x-direction.

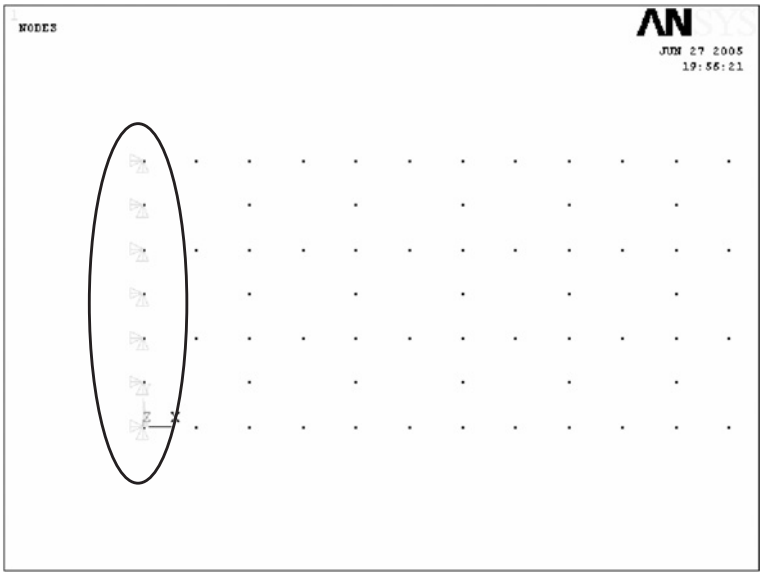


Figure 3.28 Imposing the clamping conditions on nodes.

How to clear constraint conditions

COMMAND **ANSYS Main Menu → Solution → Define Loads → Delete → Structural → Displacement → On Nodes**

The **Delete Node Constrai…** window opens.

(1) Click **Pick All** button to delete the constraint conditions of all the nodes that the constraint conditions are imposed. Select **Single** button and pick a particular node by the upward arrow in the **ANSYS Graphics** window and click **OK** button.

(2) The **Delete Node constraints** window appears. Select **ALL DOF** and click **OK** button to delete the constraint conditions both in the x- and y-directions. Select **UX** and **UY** to delete the constraints in the x- and the y-directions, respectively.

[4] Imposing boundary conditions on nodes

Before imposing load conditions, click **Fit** button in the **Pan-Zoom-Rotate** window (see Figure 3.23) to get the whole view of the area and then zoom in the right end of the beam area for ease of the following operations.

Selection of the nodes

(1) Pick the node at a point where $x = 0.08$ m and $y = 0.005$ m for this purpose, click

COMMAND | **ANSYS Utility Menu → PlotCtrls → Numbering . . .**

consecutively to open the **Plot Numbering Controls** window as shown in Figure 3.29.

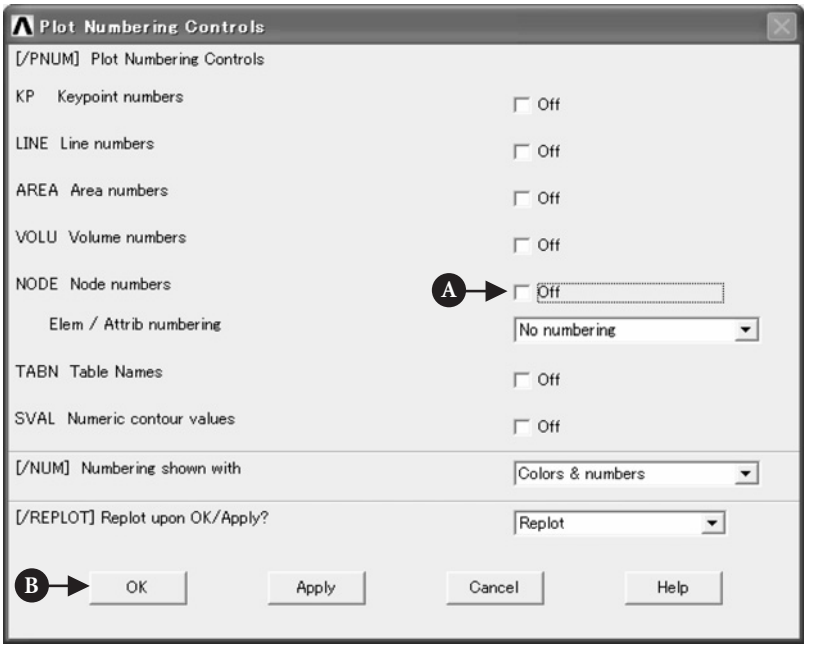


Figure 3.29 "Plot Numbering Controls" window.

(2) Click [A] **NODE** ☐ **Off** box to change it to ☑ **On** box.

(3) Click [B] **OK** button to display node numbers adjacent to corresponding nodes in the **ANSYS Graphics** window as shown in Figure 3.30.

(4) To delete the node numbers, click [A] **NODE** ☑ **On** box again to change it to ☐ **Off** box.

(5) Execute the following commands:

COMMAND | **ANSYS Utility Menu → List → Nodes . . .**

and the **Sort NODE Listing** window opens (see Figure 3.31). Select [A] **Coordinates only** button and then click [B] **OK** button.

(6) The "**NLIST Command**" window opens as shown in Figure 3.32. Find the number of the node having the coordinates $x = 0.08$ m and $y = 0.005$ m, namely node #108 in Figure 3.32.

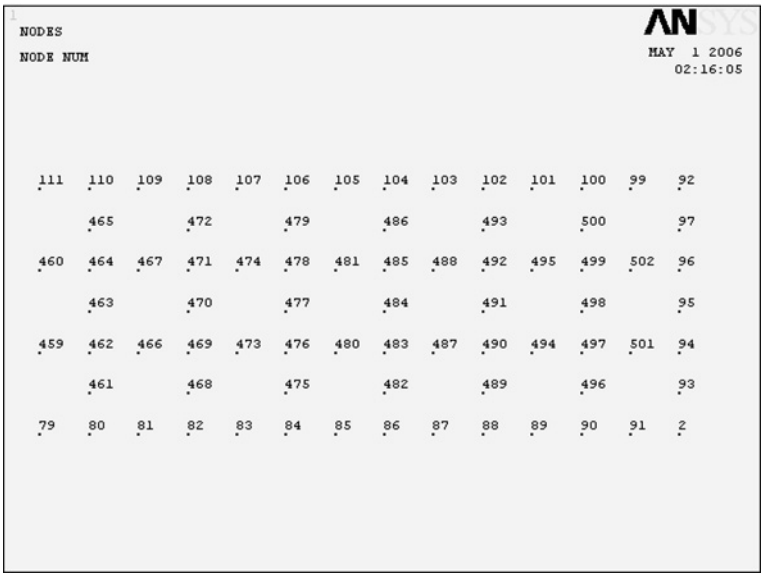


Figure 3.30 Nodes and nodal numbers displayed on the ANSYS Graphics window.

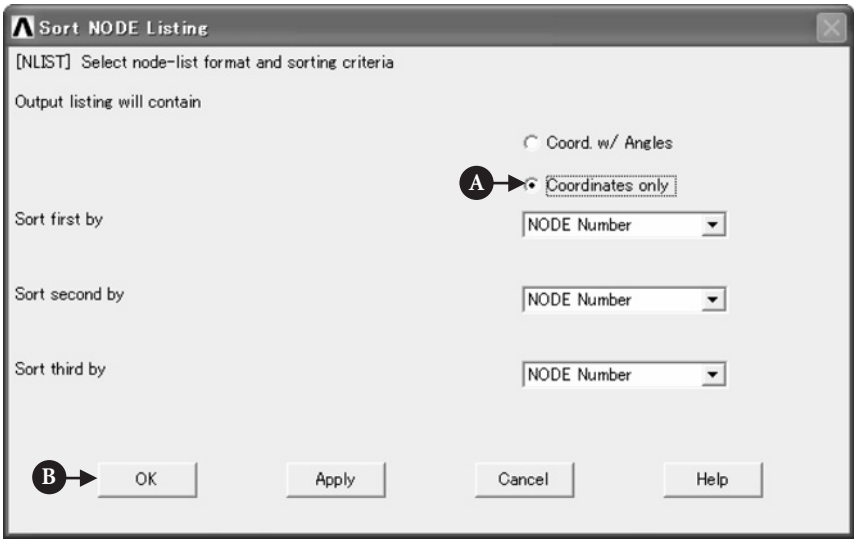


Figure 3.31 "Sort NODE Listing" window.

(7) Execute the following commands:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Force/Moment → On Nodes**

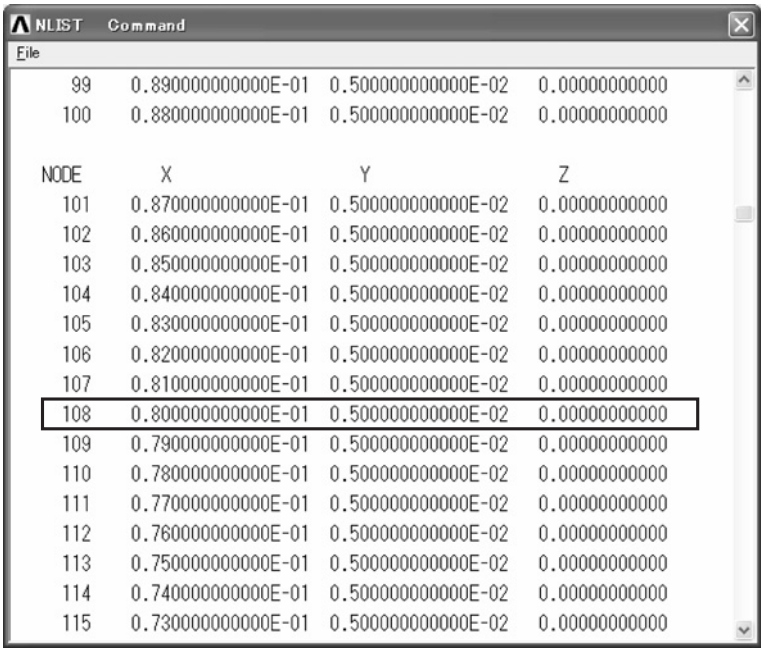


Figure 3.32 "NLIST Command" window showing the coordinates of the nodes; the framed portion indicates the coordinates of the node for load application.

to open the **Apply F/M on Nodes** window (see Figure 3.33)

(8) Pick only the #108 node having the coordinates $x = 0.08$ m and $y = 0.005$ m with the upward arrow as shown in Figure 3.34.

(9) After confirming that only the #108 node is enclosed with the yellow frame, click [A] **OK** button in the **Apply F/M on Nodes** window.

How to cancel the selection of the nodes of load application Click **Reset** button before clicking **OK** button or click the right button of the mouse to change the upward arrow to the downward arrow and click the yellow frame. The yellow frame disappears and the selection of the node(s) of load application is canceled.

Imposing load conditions on nodes

Click [A] **OK** in the **Apply F/M on Nodes** window to open another **Apply F/M on Nodes** window as shown in Figure 3.35.

(1) Choose [A] **FY** in the **Lab Direction of force/mom** box and input [B] **−100** (*N*) in the **VALUE** box. A positive value for load indicates load in the upward or rightward direction, whereas a negative value load in the downward or leftward direction.

(2) Click [C] **OK** button to display the red downward arrow attached to the #108 node indicating the downward load applied to that point as shown in Figure 3.36.

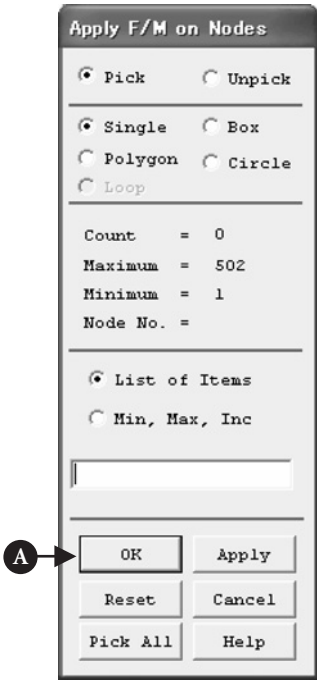


Figure 3.33 "Apply F/M on Nodes" window.

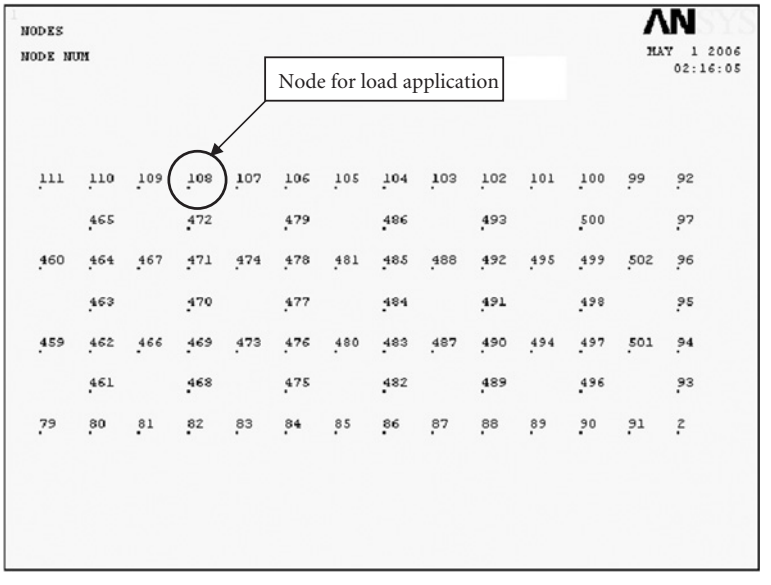


Figure 3.34 Selection of a node for load application.

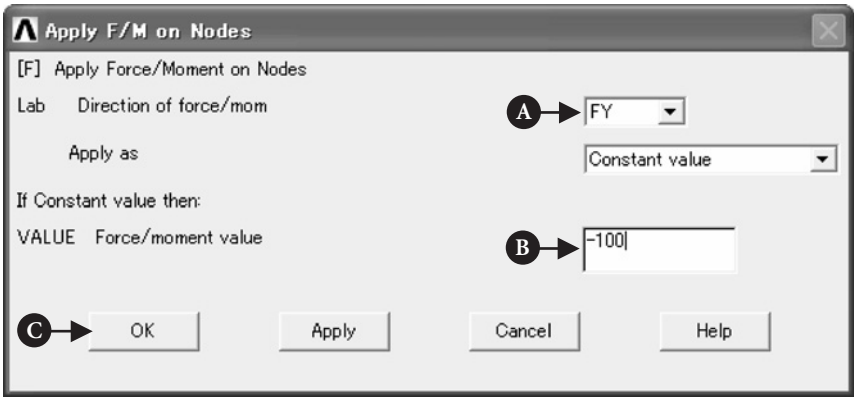


Figure 3.35 "Apply F/M on Nodes" window.

How to delete load conditions Execute the following commands:

COMMAND **ANSYS Main Menu → Solution → Define Loads → Delete → Structural → Force/Moment → On Nodes**

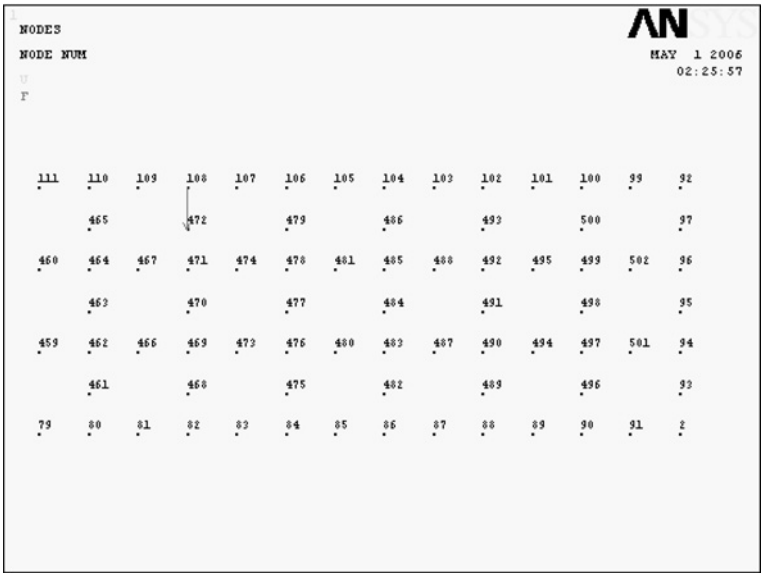


Figure 3.36 Display of the load application on a node by arrow symbol.

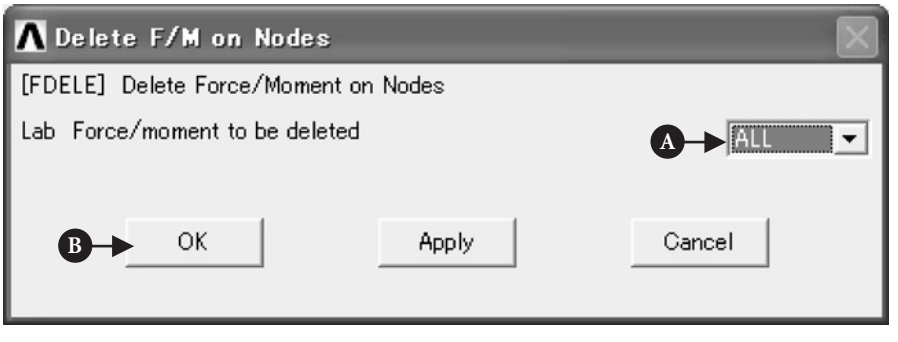


Figure 3.37 "Delete F/M on Nodes" window.

to open the **Delete F/M on Nodes** window (see Figure 3.37). Choose [A] **FY** or **ALL** in the **Lab Force/moment to be deleted** and click **OK** button to delete the downward load applied to the #108 node.

3.1.3.5 SOLUTION PROCEDURES

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

The **Solve Current Load Step** and **/STATUS Command** windows appear as shown in Figures 3.38 and 3.39, respectively.

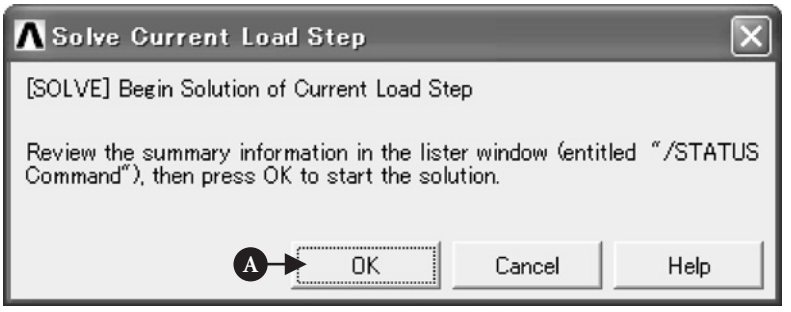


Figure 3.38 "Solve Current Load Step" window.

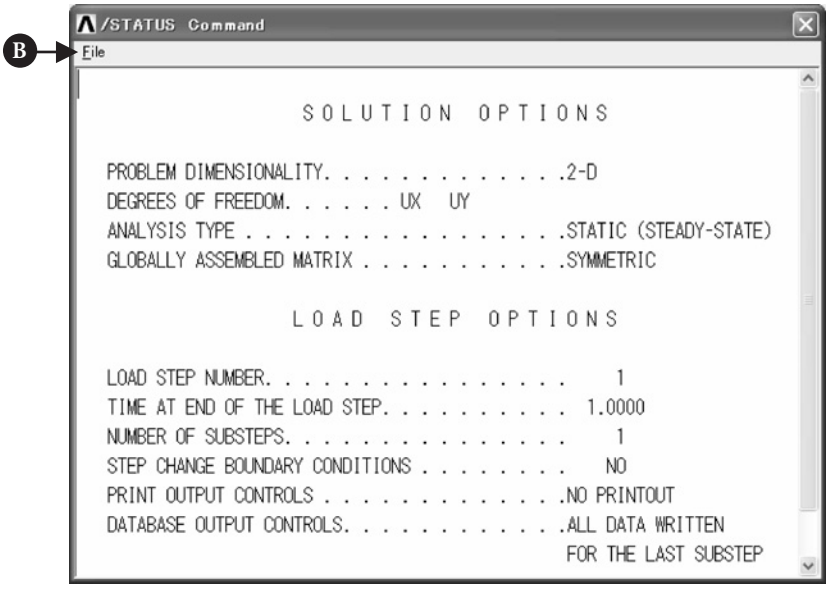


Figure 3.39 "/STATUS Command" window.

(1) Click [A] **OK** button in the **Solve Current Load Step** window as shown in Figure 3.38 to begin the solution of the current load step.

(2) The **/STATUS Command** window displays information on solution and load step options. Select [B] **File** button to open the submenu and select **Close** button to close the **/STATUS Command** window.

(3) When solution is completed, the **Note** window (see Figure 3.40) appears. Click [C] **Close** button to close the window.

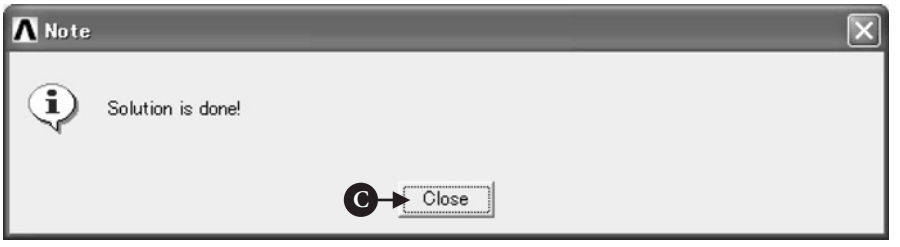


Figure 3.40 "Note" window.

3.1.3.6 GRAPHICAL REPRESENTATION OF THE RESULTS

[1] Contour plot of displacements

COMMAND **ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution**

The **Contour Nodal Solution Data** window opens as shown in Figure 3.41.

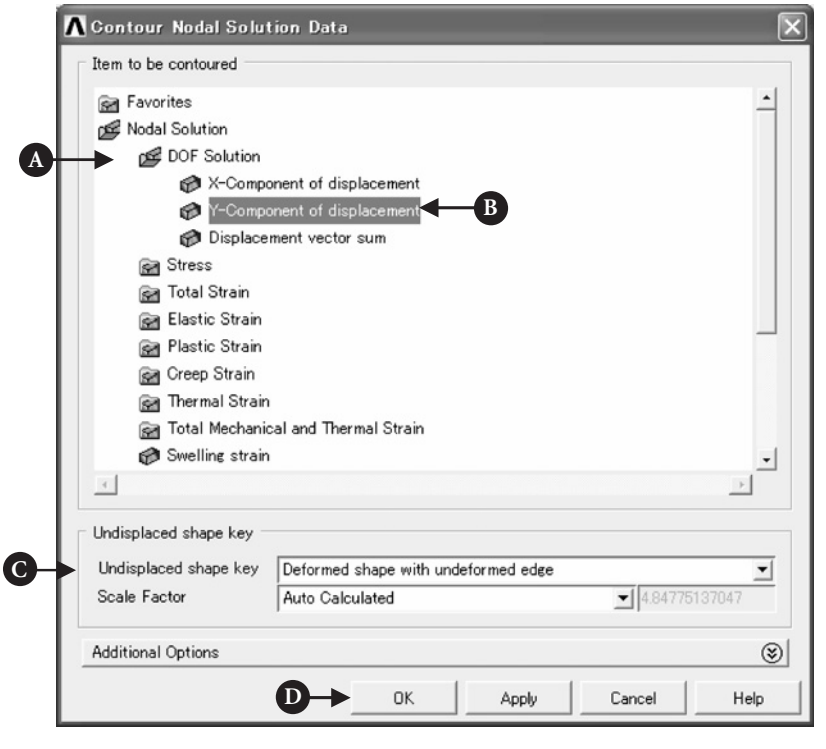


Figure 3.41 "Contour Nodal Solution Data" window.

(1) Select [A] **DOF Solution** and [B] **Y-Component of displacement**.

(2) Select [C] **Deformed shape with undeformed edge** in the **Undisplaced shape key** box to compare the shapes of the beam before and after deformation.

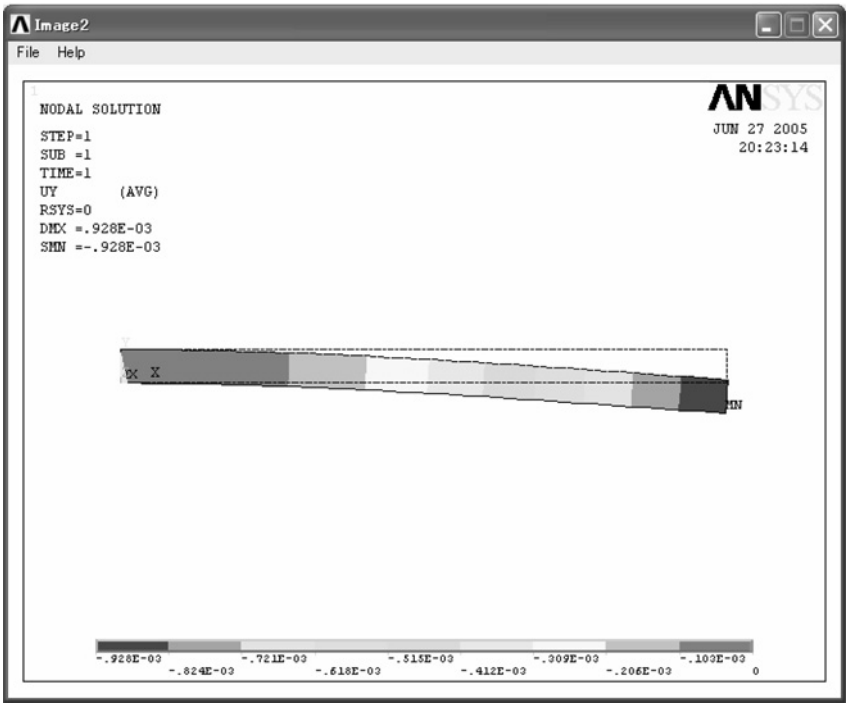


Figure 3.42 Contour map representation of the distribution of displacement in the *y*- or vertical direction.

(3) Click [D] **OK** button to display the contour of the *y*-component of displacement, or the deflection of the beam in the **ANSYS Graphics** window (see Figure 3.42). The **DMX** value shown in the Graphics window indicates the maximum deflection of the beam.

[2] Contour plot of stresses

(1) Select [A] **Stress** and [B] **X-Component of stress** as shown in Figure 3.43.

(2) Click [C] **OK** button to display the contour of the *x*-component of stress, or the bending stress in the beam in the **ANSYS Graphics** window (see Figure 3.44). The **SMX** and **SMN** values shown in the Graphics window indicate the maximum and the minimum stresses in the beam, respectively.

(3) Click [D] **Additional Options** bar to open additional option items to choose. Select [E] **All applicable** in the **Number of facets per element edge** box to calculate stresses and strains at middle points of the elements.

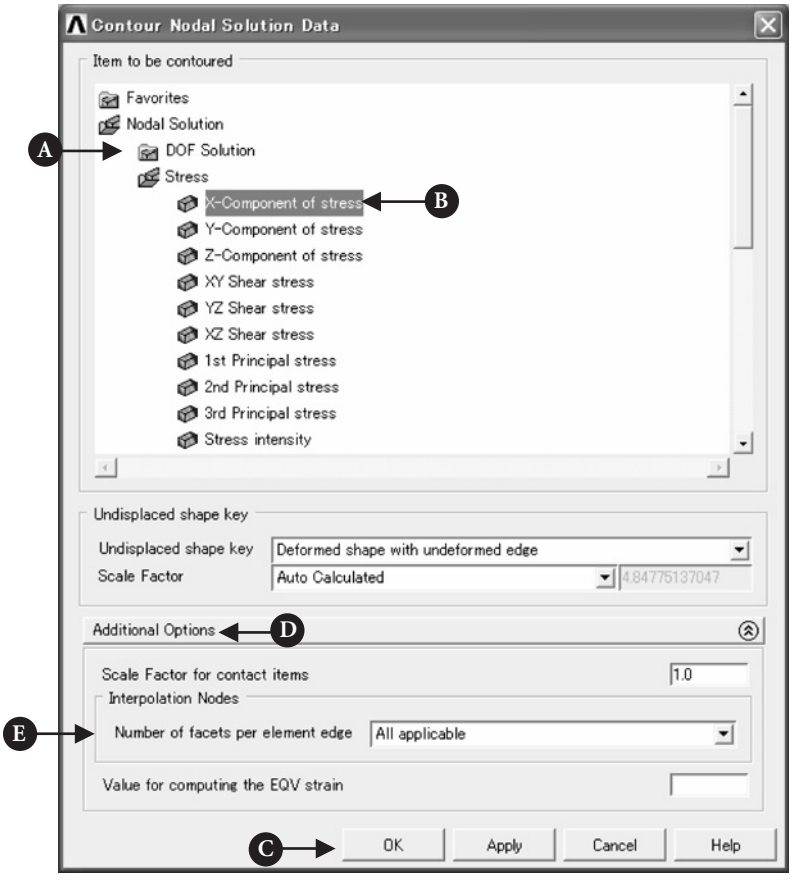


Figure 3.43 "Contour Nodal Solution Data" window.

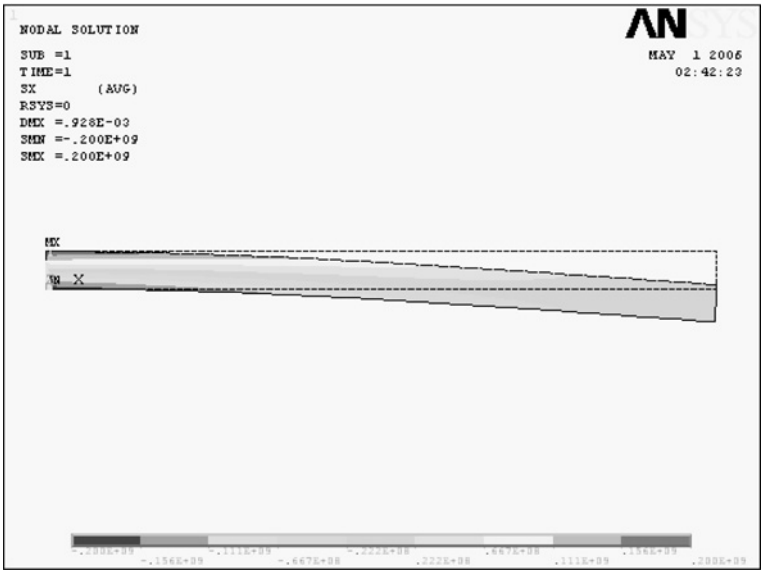


Figure 3.44 Contour map representation of the distribution of normal stress in the *x*- or horizontal direction.

3.1.4 Comparison of FEM results with experimental ones

Figure 3.45 compares longitudinal stress distributions obtained by ANSYS with those by experiments and by the elementary beam theory. The results obtained by three different methods agree well with one another. As the applied load increases, however, errors among the three groups of the results become larger, especially at the clamped end. This tendency arises from the fact that the clamped condition can be hardly realized in the strict sense.

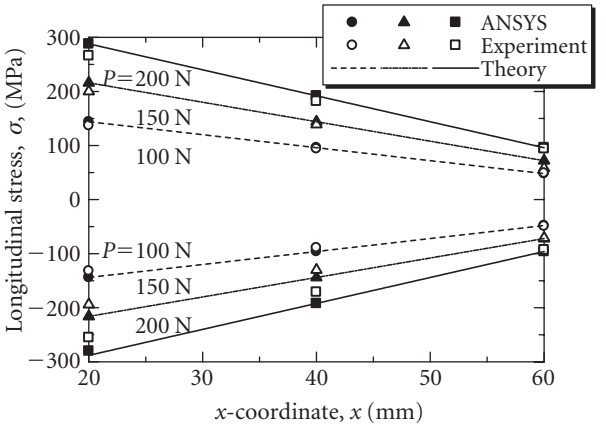


Figure 3.45 Comparison of longitudinal stress distributions obtained by ANSYS with those by experiments and by the elementary beam theory.

3.1.5 Problems to solve

Problem 3.1

Change the point of load application and the intensity of the applied load in the cantilever beam model shown in Figure 3.3 and calculate the maximum deflection.

Problem 3.2

Calculate the maximum deflection in a beam clamped at the both ends as shown in Figure P3.2 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm.

(Answer: 0.00337 mm)

Problem 3.3

Calculate the maximum deflection in a beam simply supported at the both ends as shown in Figure P3.3 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm.

(Answer: 0.00645 mm)

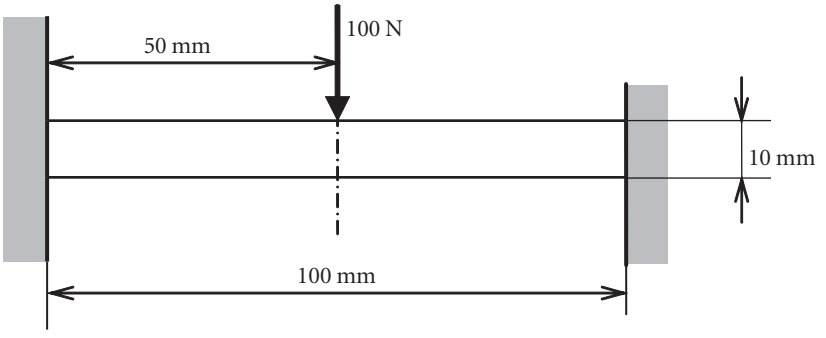


Figure P3.2 A beam clamped at the both ends and subjected to a concentrated force of 100 N at the center of the span.

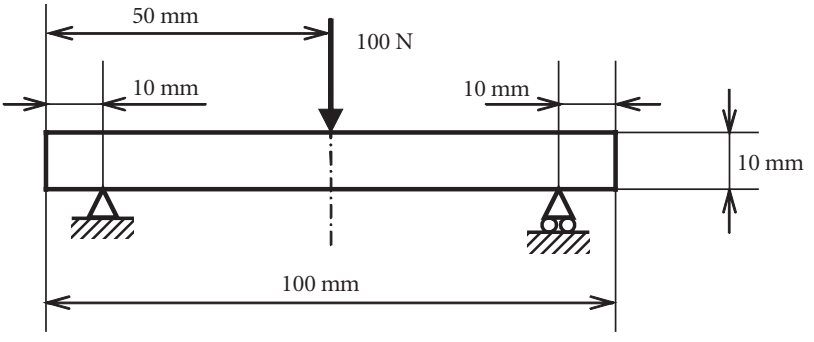


Figure P3.3 A beam simply supported at the both ends and subjected to a concentrated force of 100 N at the center of the span.

PROBLEM 3.4

Calculate the maximum deflection in a beam shown in Figure P3.4 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm. Choose an element size of 1 mm.

(Answer: 0.00337 mm)

Note that the beam shown in Figure P3.2 is bilaterally symmetric so that the x-component of the displacement (DOF X) is zero at the center of the beam span. If the beam shown in Figure P3.2 is cut at the center of the span and the finite-element calculation is made for only the left half of the beam by applying a half load of 50 N to its right end which is fixed in the x-direction but is deformed freely in the y-direction as shown in Figure P3.4, the solution obtained is the same as that for the left half of the beam in Problem 3.2. Problem 3.2 can be solved by its half model as shown in Figure P3.4. A half model can achieve the efficiency of finite-element calculations.

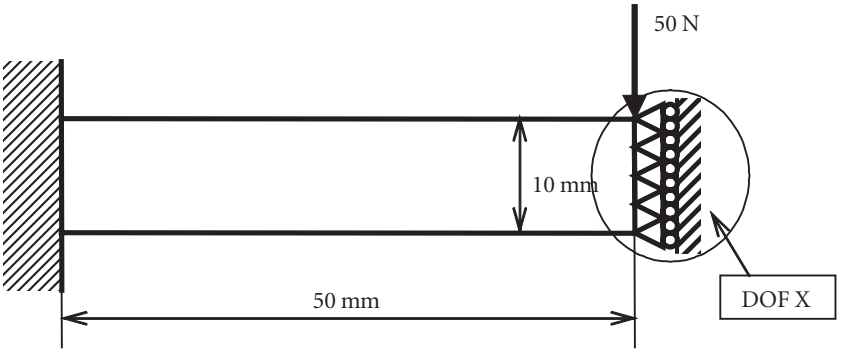


Figure P3.4 A half model of the beam in Problem 3.2.

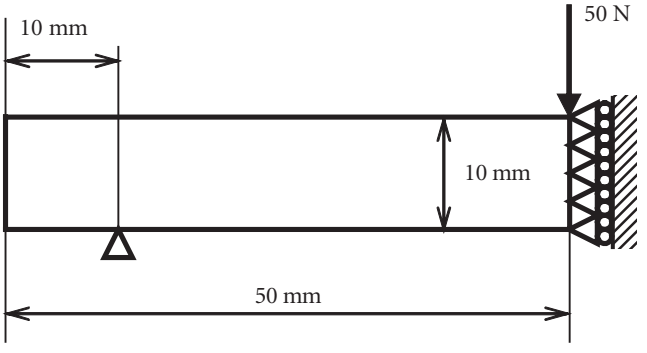


Figure P3.5 A half model of the beam in Problem 3.3.

PROBLEM 3.5

Calculate the maximum deflection in a beam shown in Figure P3.5 where the thickness of the beam in the direction perpendicular to the page surface is 10 mm. This beam is the half model of the beam of Problem 3.3.

(Answer: 0.00645 mm)

PROBLEM 3.6

Calculate the maximum value of the von Mises stress in the stepped beam as shown in Figure P3.6 where Young's modulus $E = 210$ GPa, Poisson's ratio $\nu = 0.3$, the element size is 2 mm and the beam thickness is 10 mm. Refer to the Appendix to create the stepped beam. The von Mises stress σ_{eq} is sometimes called the equivalent stress or

the effective stress and is expressed by the following formula:

$$\sigma_{eq} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} \quad \text{(P3.6)}$$

in three-dimensional elasticity problems. It is often considered that a material yields when the value of the von Mises stress reaches the yield stress of the material σ_Y which is determined by the uniaxial tensile tests of the material.

(Answer: 40.8 MPa)

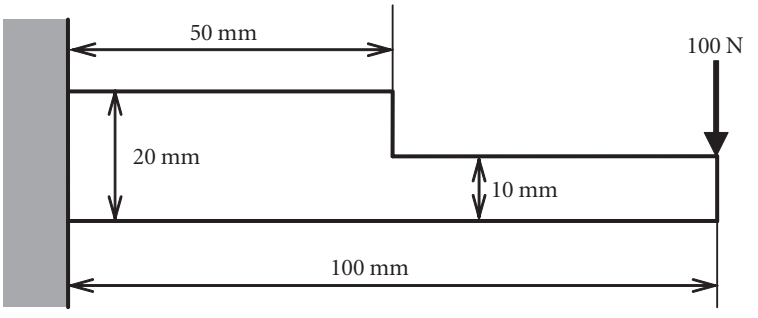


Figure P3.6 A stepped cantilever beam subjected to a concentrated force of 100 N at the free end.

PROBLEM 3.7

Calculate the maximum value of the von Mises stress in the stepped beam with a rounded fillet as shown in Figure P3.7 where Young's modulus $E = 210$ GPa, Poisson's ratio $\nu = 0.3$, the element size is 2 mm and the beam thickness is 10 mm. Refer to the Appendix to create the stepped beam with a rounded fillet.

(Answer: 30.2 MPa)

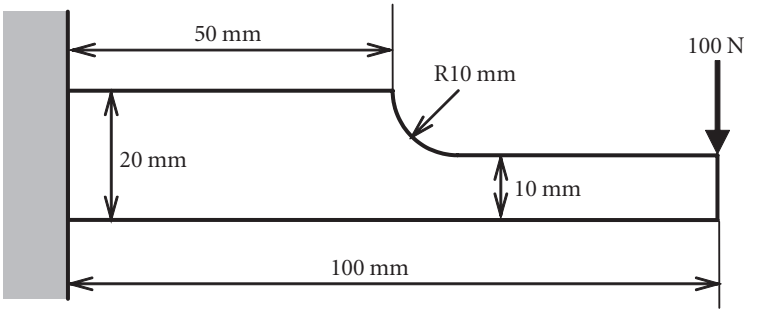


Figure P3.7 A stepped cantilever beam with a rounded fillet subjected to a concentrated force of 100 N at the free end.

Appendix: Procedures for Creating Stepped Beams

A3.1 Creation of a Stepped Beam

A stepped beam as shown in Figure P3.6 can be created by adding two rectangular areas of different sizes:

(1) Create two rectangular areas of different sizes, say 50 mm by 20 mm with WP X = −50 mm and WP Y = −10 mm, and 60 mm by 10 mm with WP X = −10 mm and WP Y = −10 mm, following operations described in 3.1.3.1.

(2) Select the Boolean operation of adding areas as follows to open the **Add Areas** window (see Figure 3.46)

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas**

(3) Pick all the areas to add by the upward arrow.

(4) The color of the areas picked turns from light blue into pink (see Figure 3.47). Click [A] **OK** button to add the two rectangular areas to create a stepped beam area as shown in Figure 3.48.

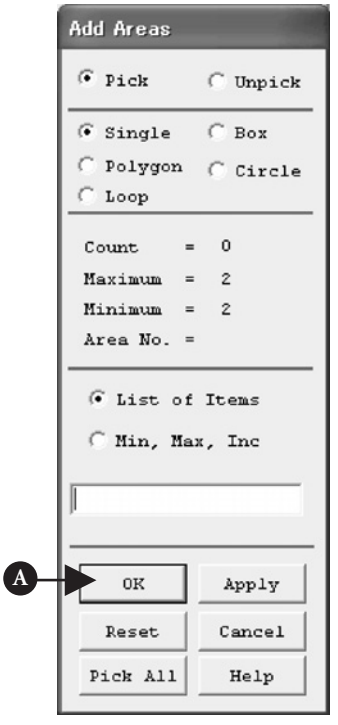


Figure 3.46 "Add Areas" window.

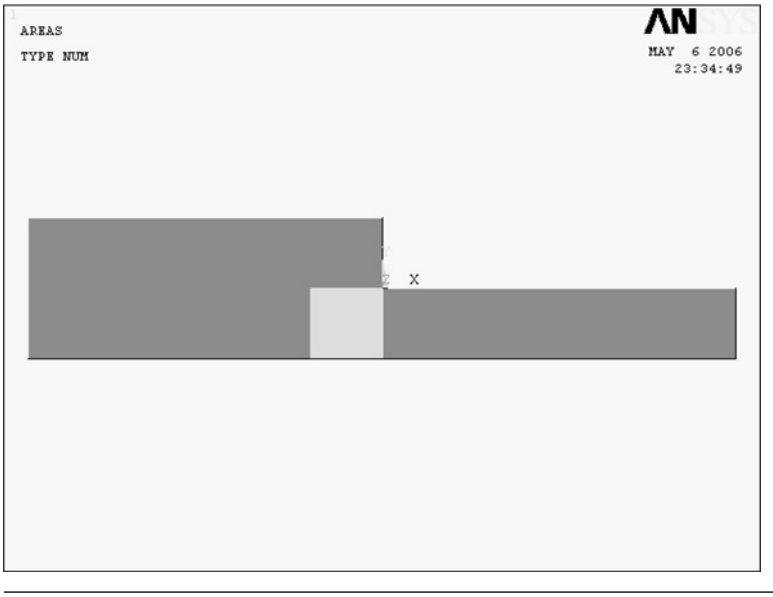


Figure 3.47 Two rectangular areas of different sizes to be added to create a stepped beam area.

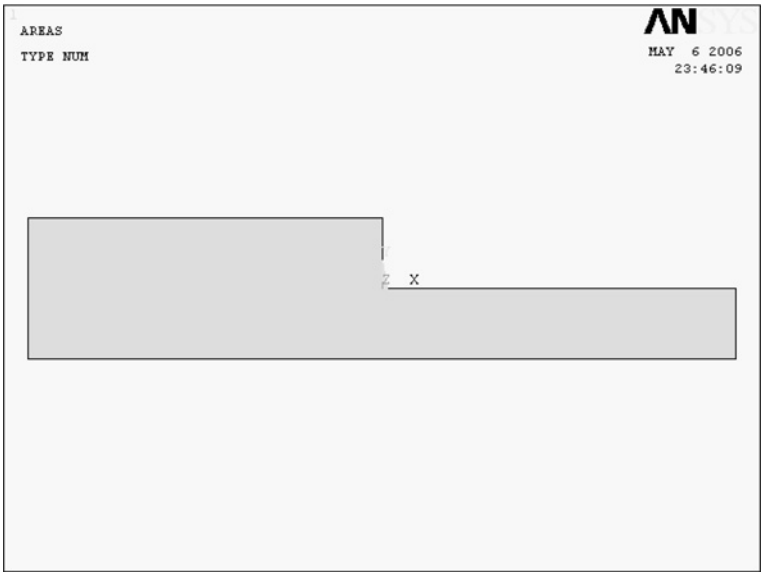


Figure 3.48 A stepped beam area created by adding two rectangle areas.

A3.1.1 How to cancel the selection of areas

Click **Reset** button or click the right button of the mouse to change the upward arrow to the downward arrow and click the area(s) to pick. The color of the unpicked area(s) turns pink into light blue and the selection of the area(s) is canceled.

A3.2 Creation of a stepped beam with a rounded fillet

A stepped beam with a rounded fillet as shown in Figure P3.7 can be created by subtracting a smaller rectangular area and a solid circle from a larger rectangular area:

(1) Create a larger rectangular area of 100 mm by 20 mm with WP X = −100 mm and WP Y = −10 mm, a smaller rectangular area of, say 50 mm by 15 mm with WP X = 10 mm and WP Y = 0 mm, and a solid circular area having a diameter of 10 mm with WP X = 10 mm and WP Y = 10 mm as shown in Figure 3.49. The solid circular area can be created by executing the following operation:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

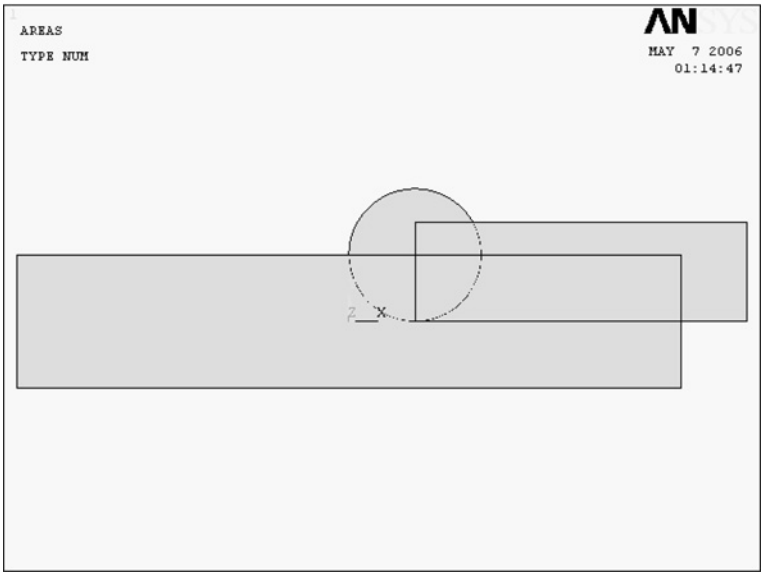


Figure 3.49 A larger rectangular area, a smaller rectangular area and a solid circular area to create a stepped beam area with a rounded fillet.

to open the **Solid Circular Area** window. Input the coordinate of the center ([A] **WP X**, [B] **WP Y**) and [C] **Radius** of the solid circle and click [D] **OK** button as shown in Figure 3.50.

(2) Select the Boolean operation of subtracting areas as follows to open the **Subtract Areas** window (see Figure 3.51):

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

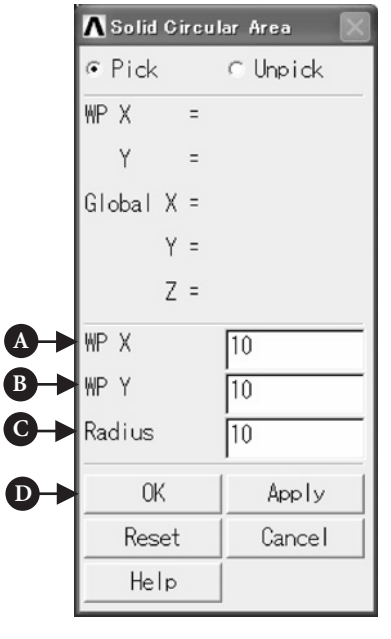


Figure 3.50 "Solid Circular Area" window.

(3) Pick the larger rectangular area by the upward arrow as shown in Figure 3.52 and the color of the area picked turns from light blue into pink. Click [A] **OK** button.

(4) Pick the smaller rectangular area and the solid circular area by the upward arrow as shown in Figure 3.53 and the color of the two areas picked turns from light blue into pink. Click [A] **OK** button to subtract the smaller rectangular and solid

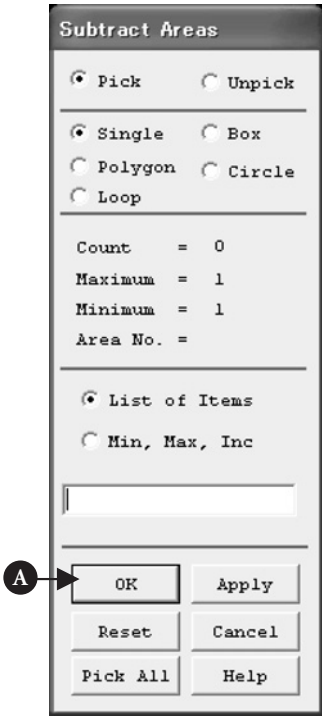


Figure 3.51 "Subtract Areas" window.

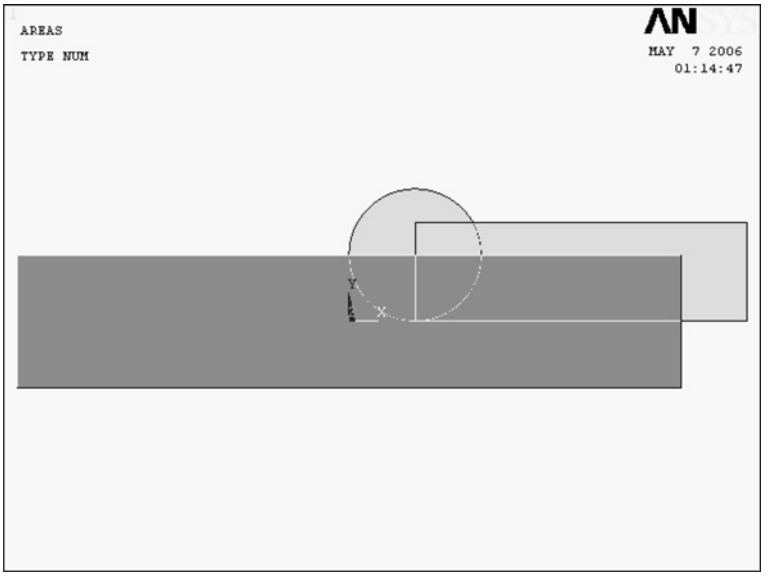


Figure 3.52 The larger rectangular area picked.

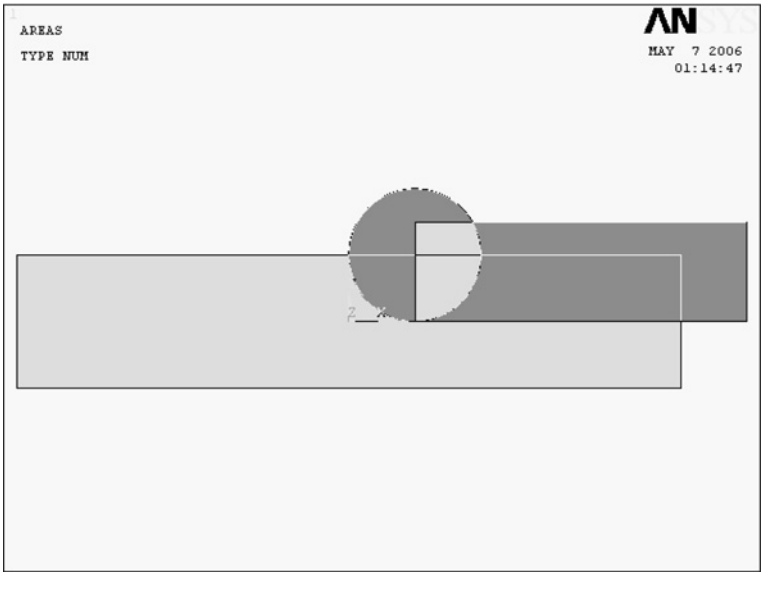


Figure 3.53 The smaller rectangular area and the solid circular area picked to be subtracted from the larger rectangular area to create a stepped beam area with a rounded fillet.

circular areas from the larger rectangular area to create a stepped beam area with a rounded fillet as shown in Figure 3.54.

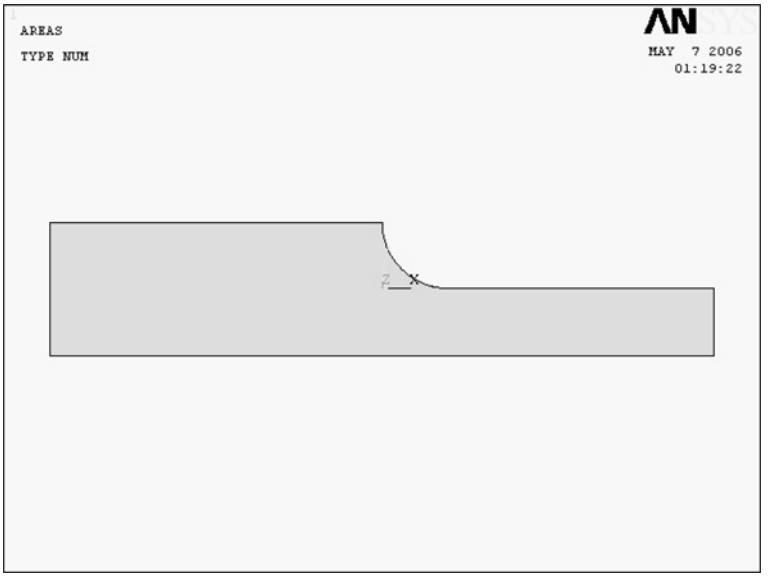


Figure 3.54 A stepped cantilever beam area with a rounded fillet.

A3.2.1 How to display area numbers

Area numbers can be displayed in the "**ANSYS Graphics**" window by the following procedure.

Command | **ANSYS Utility Menu → PlotCtrls → Numbering . . .**

(1) The **Plot Numbering Controls** window opens as shown in Figure 3.29.
(2) Click **AREA** ☐ **Off** box to change it to ☑ **On** box.
(3) Click **OK** button to display area numbers in the corresponding areas in the **ANSYS Graphics** window.
(4) To delete the area numbers, click **AREA** ☑ **On** box again to change it to ☐ **Off** box.

3.2 The principle of St. Venant

3.2.1 Example problem

An elastic strip subjected to distributed uniaxial tensile stress or negative pressure at one end and clamped at the other end.

Perform an FEM analysis of a 2-D elastic strip subjected to a distributed stress in the longitudinal direction at one end and clamped at the other end (shown in Figure 3.55 below) and calculate the stress distributions along the cross sections at different distances from the loaded end in the strip.

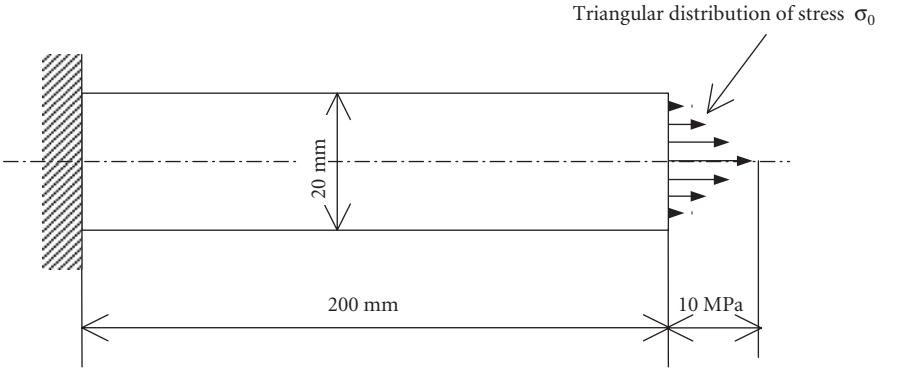


Figure 3.55 A 2-D elastic strip subjected to a distributed force in the longitudinal direction at one end and clamped at the other end.

3.2.2 PROBLEM DESCRIPTION

Geometry: length $l = 200$ mm, height $h = 20$ mm, thickness $b = 10$ mm.
Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.
Boundary conditions: The elastic strip is subjected to a triangular distribution of stress in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.2.3 ANALYTICAL PROCEDURES

3.2.3.1 CREATION OF AN ANALYTICAL MODEL

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners**

(1) Input two **0**'s into the "**WP X**" and "**WP Y**" boxes in the "**Rectangle by 2 Corners**" window to determine the lower left corner point of the elastic strip on the Cartesian coordinates of the working plane.

(2) Input **200** and **20** (mm) into the **Width** and **Height** boxes, respectively, to determine the shape of the elastic strip model.

(3) Click the **OK** button to create the rectangular area, or beam on the **ANSYS Graphics** window.

In the procedures above, the geometry of the strip is input in millimeters. You must decide what kind of units to use in finite-element analyses. When you input the geometry of a model to analyze in millimeters, for example, you must input applied loads in N (Newton) and Young's modulus in MPa, since 1 MPa is equivalent to 1 N/mm^2. When you use meters and N as the units of length and load, respectively, you must input Young's modulus in Pa, since 1 Pa is equivalent to 1 N/m^2. You can choose any system of unit you would like to, but your unit system must be consistent throughout the analyses.

3.2.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE STRIP MATERIAL

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) The **Define Material Model Behavior** window opens.
(2) Double-click **Structural, Linear, Elastic**, and **Isotropic** buttons one after another.
(3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
(4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.2.3.3 FINITE-ELEMENT DISCRETIZATION OF THE STRIP AREA

[1] Selection of the element type

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

(1) The **Element Types** window opens.
(2) Click the **Add …** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.
(3) Select **Structural Mass – Solid** and **Quad 8node 82.**
(4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.
(5) Click the **Options …** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strs w/thk** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Input of the element thickness

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/Delete**

(1) The **Real Constants** window opens.
(2) Click [A] **Add/Edit/Delete** button to open the **Real Constants** window and click the **Add …** button.

(3) The **Element Type for Real Constants** window opens. Click the **OK** button.

(4) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead. Input a strip thickness of 10 (mm) in the **Thickness** box and click the **OK** button.

(5) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1.** Click the **Close** button.

[3] Sizing of the elements

COMMAND **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size**

(1) The **Global Element Sizes** window opens.

(2) Input **2** in the **SIZE** box and click the **OK** button.

[4] Dividing the right-end side of the strip area into two lines

Before proceeding to meshing, the right-end side of the strip area must be divided into two lines for imposing the triangular distribution of the applied stress or pressure by executing the following commands.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Divide → Lines w/Options**

(1) The **Divide Multiple Lines …** window opens as shown in Figure 3.56.

(2) When the mouse cursor is moved to the **ANSYS Graphics** window, an upward arrow (↑) appears.

(3) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the strip area and click the left button of the mouse.

(4) Click the **OK** button in the **Divide Multiple Lines . . .** window to display the **Divide Multiple Lines with Options** window as shown in Figure 3.57.

(5) Input **2** in [A] **NDIV** box and **0.5** in [B] **RATIO** box, and select **Be modified** in [C] **KEEP** box.

(6) Click [D] **OK** button.

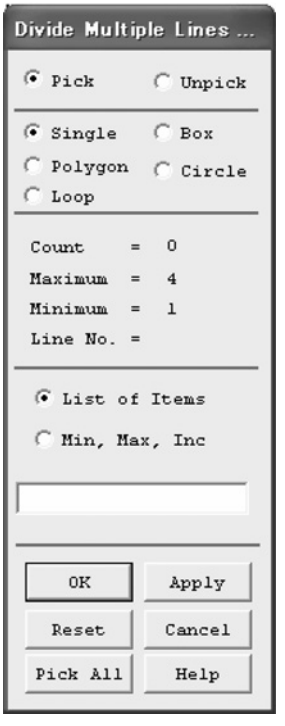


Figure 3.56 "Divide Multiple Lines . . ." window.

[5] Meshing

COMMAND **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

(1) The **Mesh Areas** window opens.

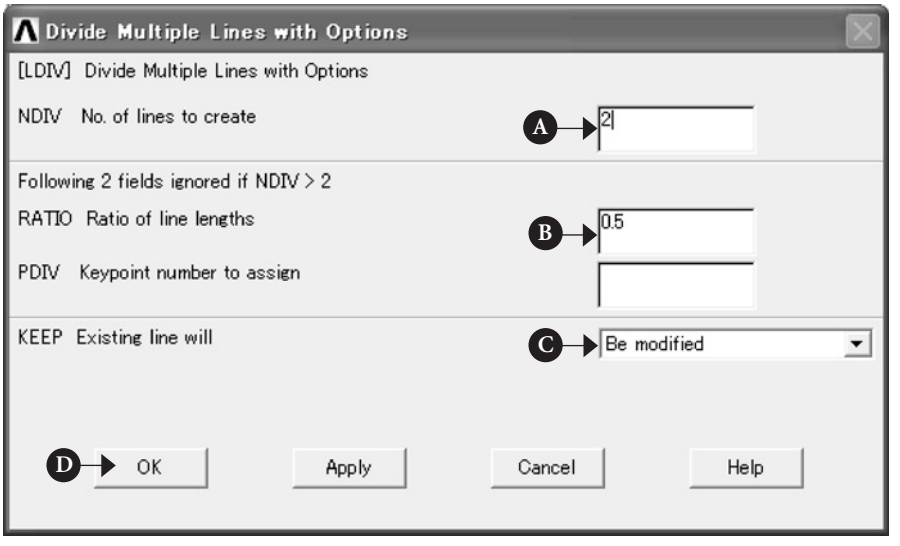


Figure 3.57 "Divide Multiple Lines with Options" window.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the elastic strip area and click this area.

(3) The color of the area turns from light blue into pink. Click the **OK** button to see the area meshed by 8-node rectangular isoparametric finite elements.

3.2.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the left end of the strip

COMMAND **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

(1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left-end side of the strip area and click the left button of the mouse.

(3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.

(4) Select **ALL DOF** in the **Lab2** box and click **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a triangular distribution of applied stress on the right end of the strip
Distributed load or stress can be defined by pressure on lines and the triangular distribution of applied load can be defined as the composite of two linear distributions

of pressure which are symmetric to each other with respect to the center line of the strip area.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

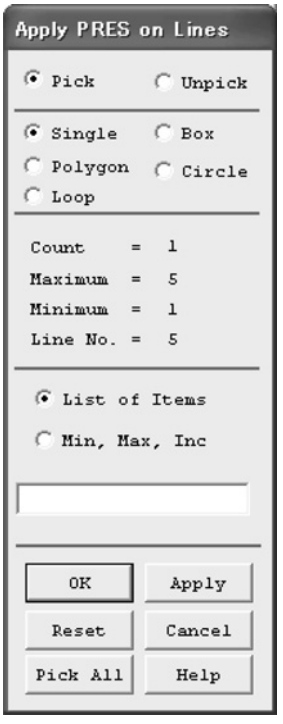


Figure 3.58 "Apply PRES on Lines" window for picking the lines to which pressure is applied.

(1) The **Apply PRES on Lines** window opens (see Figure 3.58) and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the upper line of the right-end side of the strip area and click the left button of the mouse. Then, click the **OK** button. Remember that the right-end side of the strip area was divided into two lines in Procedure [4] in the preceding Section 3.2.3.3.

(3) Another **Apply PRES on Lines** window opens (see Figure 3.59). Select **Constant value** in [A] **[SFL] Apply PRES on lines as a** box and input [B] **−10** (MPa) in **VALUE Load PRES value** box and [C] **0** (MPa) in **Value** box.

(4) Click [D] **OK** button in the window to define a linear distribution of pressure on the upper line which is zero at the upper right corner and −10 (MPa) at the center of the right-end side of the strip area (see Figure 3.60).

(5) For the lower line of the right-end side of the strip area, repeat the commands above and Procedures (2) through (4).

(6) Select **Constant value** in [A] **[SFL] Apply PRES on lines as a** box and input [B] **0** (MPa) in **VALUE Load PRES value** box and [C] **−10** (MPa) in **Value** box as shown in Figure 3.61. Note that the values to input in the lower two boxes in the **Apply PRES on Lines** window is interchanged, since the distributed pressure on the lower line of the right-end side of the strip area is symmetric to that on the upper line with respect to the center line of the strip area.

(7) Click [D] **OK** button in the window shown in Figure 3.60 to define a linear distribution of pressure on the lower line which is −10 MPa at the center and zero at the lower right corner of the right-end side of the strip area as shown in Figure 3.62.

3.2.3.5 SOLUTION PROCEDURES

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

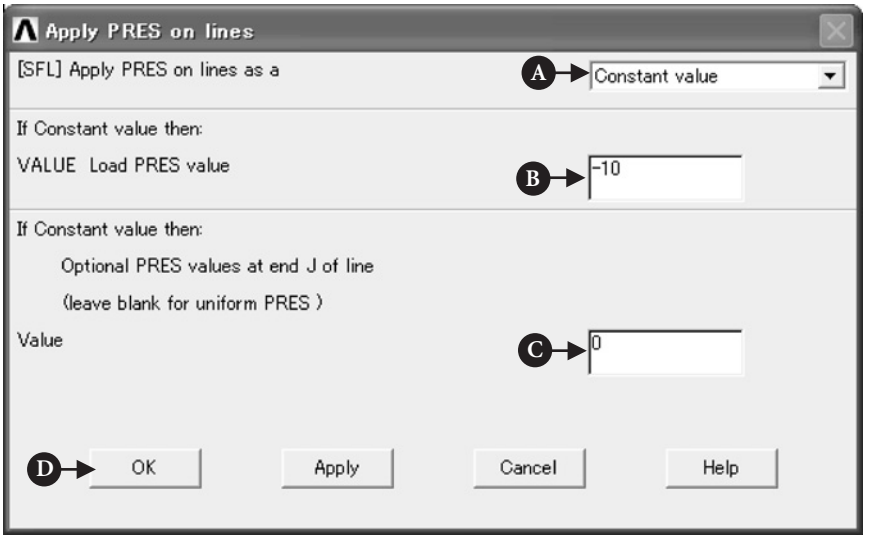


Figure 3.59 "Apply PRES on Lines" window for applying linearly distributed pressure to the upper half of the right end of the elastic strip.

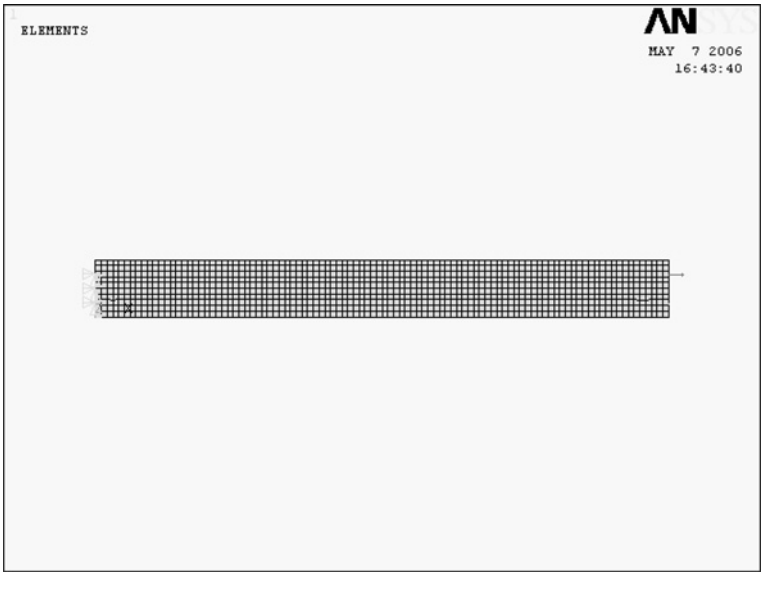


Figure 3.60 Linearly distributed negative pressure applied to the upper half of the right-end side of the elastic strip.

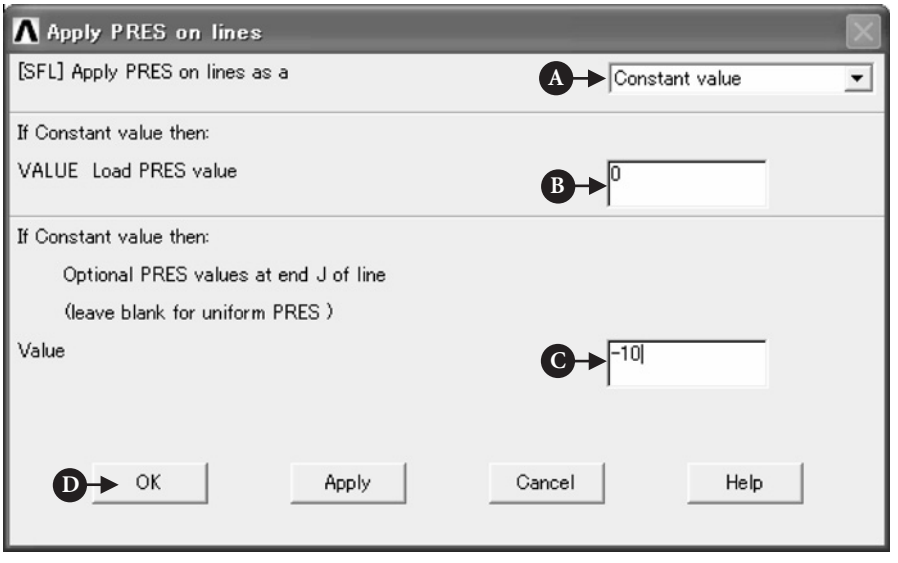


Figure 3.61 "Apply PRES on Lines" window for applying linearly distributed pressure to the lower half of the right end of the elastic strip.

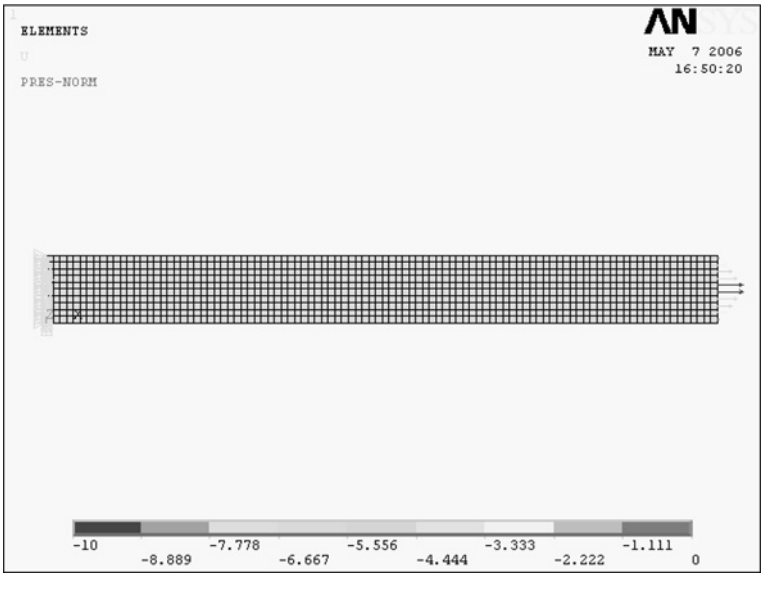


Figure 3.62 Triangular distribution of pressure applied to the right end of the elastic strip.

(1) The **Solve Current Load Step** and **/STATUS Command** windows appear.

(2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.

(3) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.

(4) When solution is completed, the **Note** window appears. Click the **Close** button to close the "**Note**" window.

3.2.3.6 CONTOUR PLOT OF STRESS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution**

(1) The **Contour Nodal Solution Data** window opens.

(2) Select **Stress** and **X-Component of stress**.

(3) Click the **OK** button to display the contour of the x-component of stress in the elastic strip in the **ANSYS Graphics** window as shown in Figure 3.63.

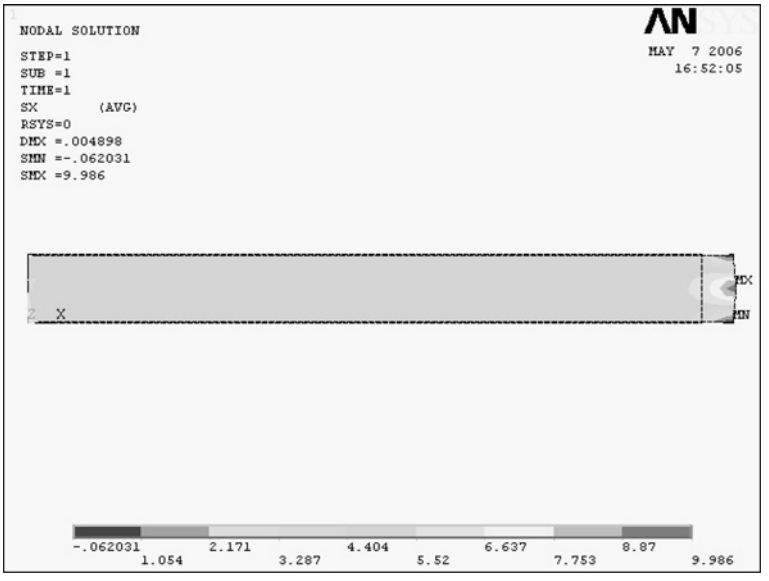


Figure 3.63 Contour of the x-component of stress in the elastic strip showing uniform stress distribution at one width or larger distance from the right end of the elastic strip to which triangular distribution of pressure is applied.

3.2.4 DISCUSSION

Figure 3.64 shows the variations of the longitudinal stress distribution in the cross section with the x-position of the elastic strip. At the right end of the strip, or at

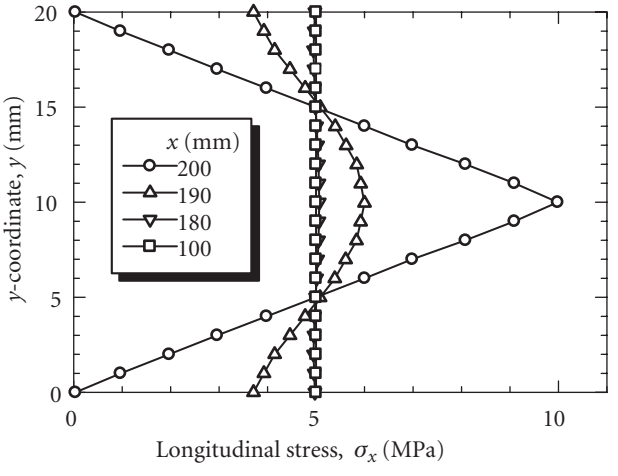


Figure 3.64 Variations of the longitudinal stress distribution in the cross section with the x-position of the elastic strip.

$x = 200$ mm, the distribution of the applied longitudinal stress takes the triangular shape which is zero at the upper and lower corners and 10 MPa at the center of the strip. The longitudinal stress distribution varies as the distance of the cross section from the right end of the strip increases, and the distribution becomes almost uniform at $x = 180$ mm, i.e., at one width distance from the end of the stress application. The total amount of stress in any cross section is the same, i.e., 1 kN in the strip and, stress is uniformly distributed and the magnitude of stress becomes 5 MPa at any cross section at one width or larger distance from the end of the stress application.

The above result is known as the principle of St. Venant and is very useful in practice, or in the design of structural components. Namely, even if the stress distribution is very complicated at the loading points due to the complicated shape of load transfer equipment, one can assume a uniform stress distribution in the main parts of structural components or machine elements at some distance from the load transfer equipment.

3.3 Stress concentration due to elliptic holes

3.3.1 Example problem

An elastic plate with an elliptic hole in its center is subjected to uniform longitudinal tensile stress σ_0 at one end and clamped at the other end in Figure 3.65. Perform the FEM stress analysis of the 2-D elastic plate and calculate the maximum longitudinal stress σ_{max} in the plate to obtain the stress concentration factor $\alpha = \sigma_{max}/\sigma_0$. Observe the variation of the longitudinal stress distribution in the ligament between the foot of the hole and the edge of the plate.

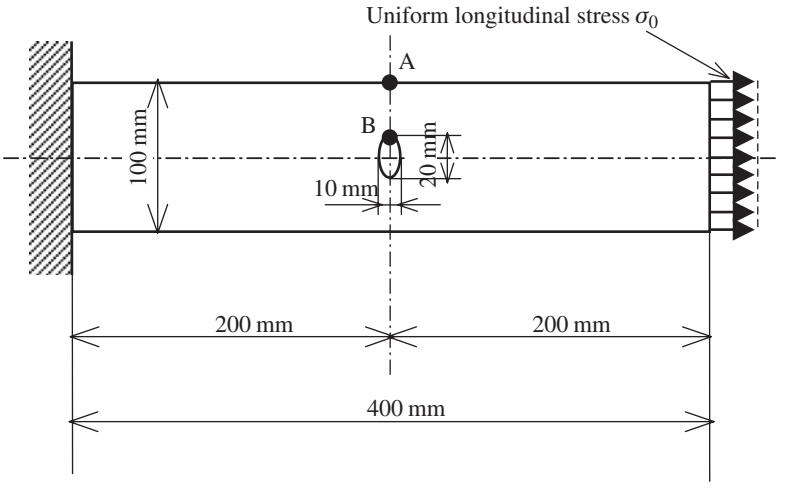


Figure 3.65 A 2-D elastic plate with an elliptic hole in its center subjected to a uniform longitudinal stress at one end and clamped at the other end.

3.3.2 PROBLEM DESCRIPTION

Plate geometry: $l = 400$ mm, height $h = 100$ mm, thickness $b = 10$ mm.
Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.
Elliptic hole: An elliptic hole has a minor radius of 5 mm in the longitudinal direction and a major radius of 10 mm in the transversal direction.
Boundary conditions: The elastic plate is subjected to a uniform tensile stress of $\sigma_0 = 10$ Mpa in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.3.3 ANALYTICAL PROCEDURES

3.3.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a quarter model of the elastic plate with an elliptic hole as illustrated later in Figure 3.70, since the plate is symmetric about the horizontal and vertical centerlines. The quarter model can be created by a slender rectangular area from which an elliptic area is subtracted by using the Boolean operation described in Section A3.1.

First, create the rectangular area by the following operation:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners**

(1) Input two **0**'s into the **WP X** and **WP Y** boxes in the **Rectangle by 2 Corners** window to determine the lower left corner point of the elastic plate on the Cartesian coordinates of the working plane.

(2) Input **200** and **50** (mm) into the **Width** and **Height** boxes, respectively, to determine the shape of the quarter elastic plate model.

(3) Click the **OK** button to create the quarter elastic plate on the **ANSYS Graphics** window.

Then, create a circular area having a diameter of 10 mm and then reduce its diameter in the longitudinal direction to a half of the original value to get the elliptic area. The following commands create a circular area by designating the coordinates (**UX**, **UY**) of the center and the radius of the circular area:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

(1) The **Solid Circular Area** window opens as shown in Figure 3.66.

(2) Input two **0**'s into [A] **WP X** and [B] **WP Y** boxes to determine the center position of the circular area.

(3) Input [C] **10** (mm) in **Radius** box to determine the radius of the circular area.

(4) Click [D] **OK** button to create the circular area superimposed on the rectangular area in the **ANSYS Graphics** window as shown in Figure 3.67.

In order to reduce the diameter of the circular area in the longitudinal direction to a half of the original value, use the following **Scale → Areas** operation:

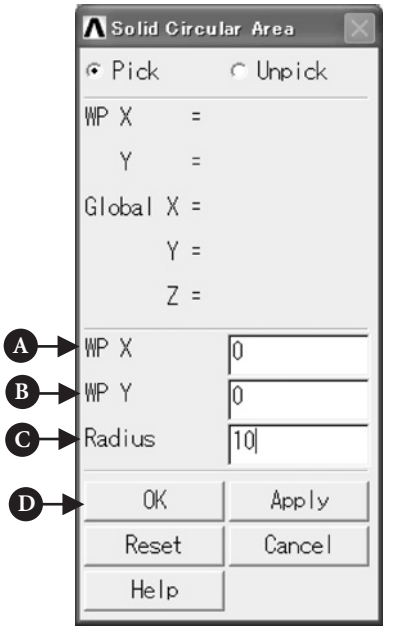


Figure 3.66 "Solid Circular Area" window.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Scale → Areas**

(1) The **Scale Areas** window opens as shown in Figure 3.68.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move the arrow to the circular area and pick it by clicking the left button of the mouse. The color of the circular area turns from light blue into pink and click [A] **OK** button.

(3) The color of the circular area turns into light blue and another **Scale Areas** window opens as shown in Figure 3.69.

(4) Input [A] **0.5** in **RX** box, select [B] **Areas only** in **NOELEM** box and [C] **Moved** in **IMOVE** box.

(5) Click [D] **OK** button. An elliptic area appears and the circular area still remains. The circular area is an afterimage and does not exist in reality. To erase this

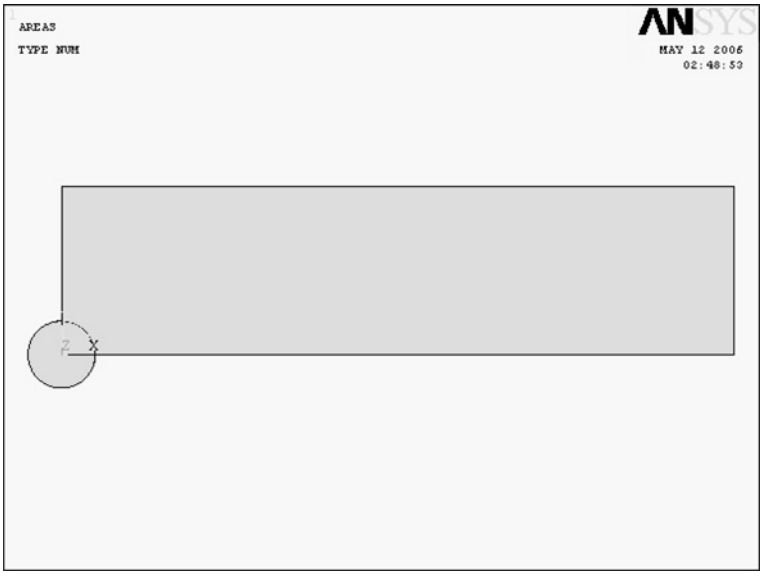


Figure 3.67 Circular area superimposed on the rectangular area.

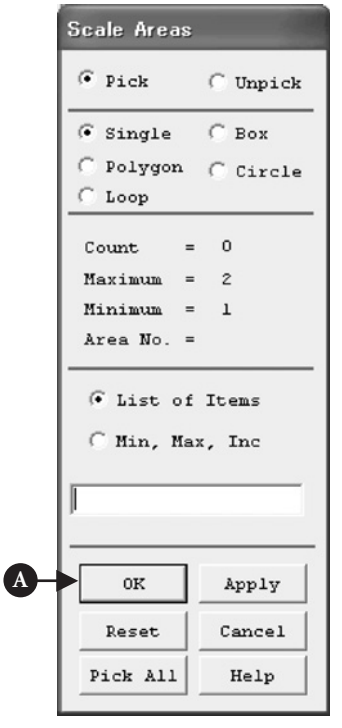


Figure 3.68 "Scale Areas" window.

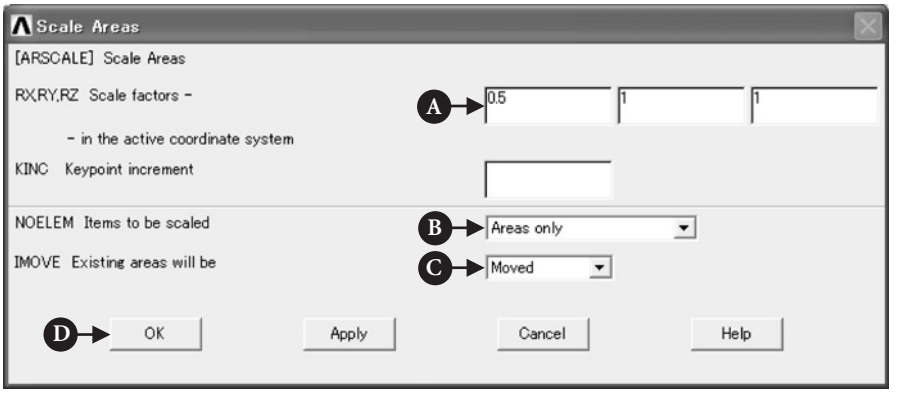


Figure 3.69 "Scale Areas".

after-image, perform the following commands:

COMMAND | **ANSYS Utility Menu → Plot → Replot**

The circular area vanishes. Subtract the elliptic area from the rectangular area in a similar manner as described in Section A3.2, i.e.,

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) Pick the rectangular area by the upward arrow and confirm that the color of the area picked turns from light blue into pink. Click the **OK** button.

(2) Pick the elliptic area by the upward arrow and confirm that the color of the elliptic area picked turns from light blue into pink. Click the **OK** button to subtract the elliptic area from the rectangular area to get a quarter model of a plate with an elliptic hole in its center as shown in Figure 3.70.

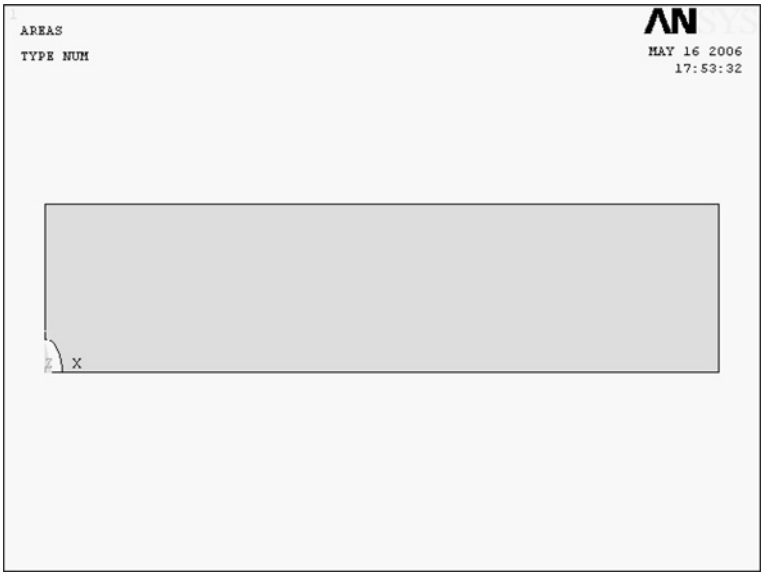


Figure 3.70 Quarter model of a plate with an elliptic hole in its center created by subtracting an elliptic area from a rectangular area.

3.3.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE PLATE MATERIAL

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) The **Define Material Model Behavior** window opens.

(2) Double-click **Structural, Linear, Elastic**, and **Isotropic** buttons one after another.

(3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.

(4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.3.3.3 FINITE-ELEMENT DISCRETIZATION OF THE QUARTER PLATE AREA

[1] Selection of the element type

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

(1) The **Element Types** window opens.

(2) Click the **Add . . .** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.

(3) Select **Structural Mass – Solid** and **Quad 8node 82.**

(4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.

(5) Click the **Options . . .** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strs w/thk** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Input of the element thickness

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/Delete**

(1) The **Real Constants** window opens.

(2) Click [A] **Add/Edit/Delete** button to open the **Real Constants** window and click the **Add . . .** button.

(3) The **Element Type for Real Constants** window opens. Click the **OK** button.

(4) The **Element Type for Real Constants** window vanishes and the **Real Constants Set Number 1. for PLANE82** window appears instead. Input a strip thickness of **10** (mm) in the **Thickness** box and click the **OK** button.

(5) The **Real Constants** window returns with the display of the **Defined Real Constants Sets** box changed to **Set 1.** Click the **Close** button.

[3] Sizing of the elements

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size**

(1) The **Global Element Sizes** window opens.

(2) Input **1.5** in the **SIZE** box and click the **OK** button.

[4] Meshing

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

(1) The **Mesh Areas** window opens.
(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter plate area and click this area.
(3) The color of the area turns from light blue into pink. Click the **OK** button to see the area meshed by 8-node isoparametric finite elements as shown in Figure 3.71.

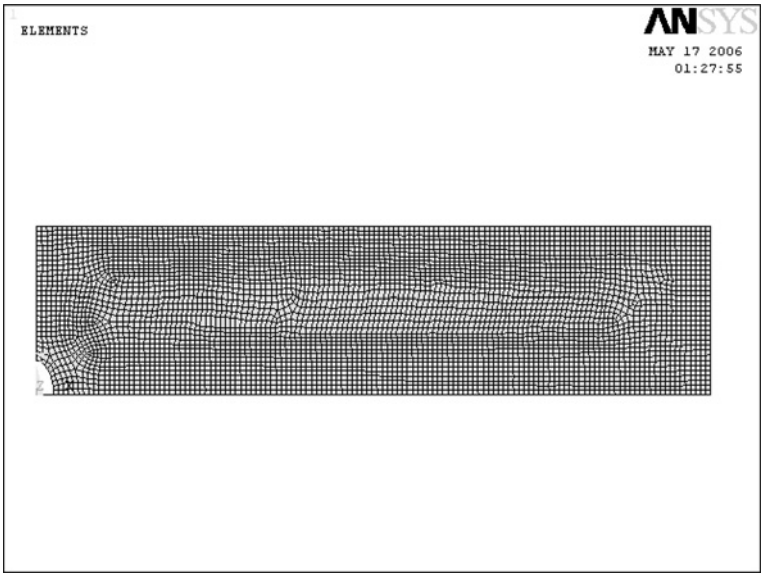


Figure 3.71 Quarter model of a plate with an elliptic hole meshed by 8-node isoparametric finite elements.

3.3.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the left end and the bottom side of the quarter plate model

Due to the symmetry, the constraint conditions of the quarter plate model are **UX**-fixed condition on the left end and **UY**-fixed condition on the bottom side of the quarter plate model. Apply the constraint conditions onto the corresponding lines by the following commands:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

(1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left-end side of the quarter plate area and click the left button of the mouse.

(3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.

(4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform longitudinal stress on the right end of the quarter plate model

A uniform longitudinal stress can be defined by pressure on the right-end side of the plate model as described below:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

(1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the quarter plate area and click the left button of the mouse.

(3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the [**SFL**] **Apply PRES on lines as a** box and input **−10** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.

(4) Click the **OK** button in the window to define a uniform tensile stress of 10 MPa applied to the right end of the quarter plate model.

Figure 3.72 illustrates the boundary conditions applied to the center notched plate model by the above operations.

3.3.3.5 SOLUTION PROCEDURES

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

(1) The **Solve Current Load Step** and **/STATUS Command** windows appear.

(2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.

(3) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.

(4) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

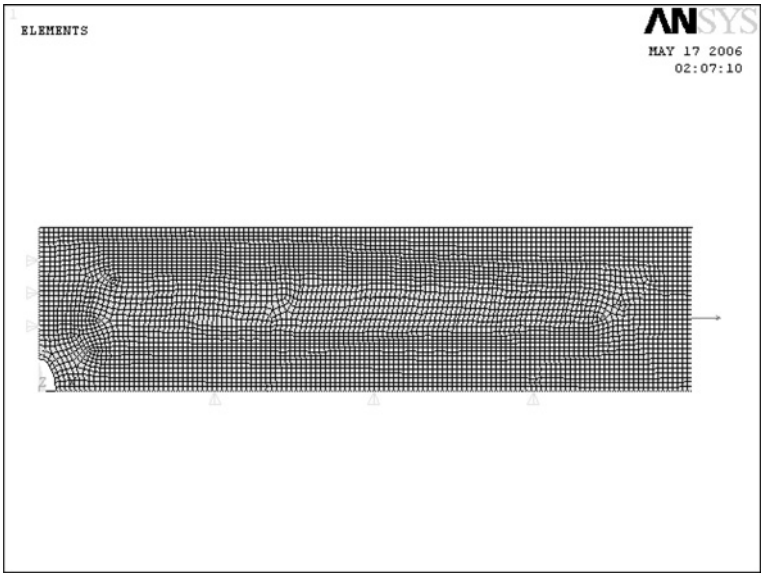


Figure 3.72 Boundary conditions applied to the quarter model of the center notched plate.

3.3.3.6 CONTOUR PLOT OF STRESS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution**

(1) The **Contour Nodal Solution Data** window opens.
(2) Select **Stress** and **X-Component of stress**.
(3) Select **Deformed shape with undeformed edge** in the **Undisplaced shape key** box to compare the shapes of the notched plate before and after deformation.
(4) Click the **OK** button to display the contour of the x-component of stress in the quarter model of the center notched plate in the **ANSYS Graphics** window as shown in Figure 3.73.

3.3.3.7 OBSERVATION OF THE VARIATION OF THE LONGITUDINAL STRESS DISTRIBUTION IN THE LIGAMENT REGION

In order to observe the variation of the longitudinal stress distribution in the ligament between the foot of the hole and the edge of the plate, carry out the following commands:

COMMAND | **ANSYS Utility Menu → PlotCtrls → Symbols …**

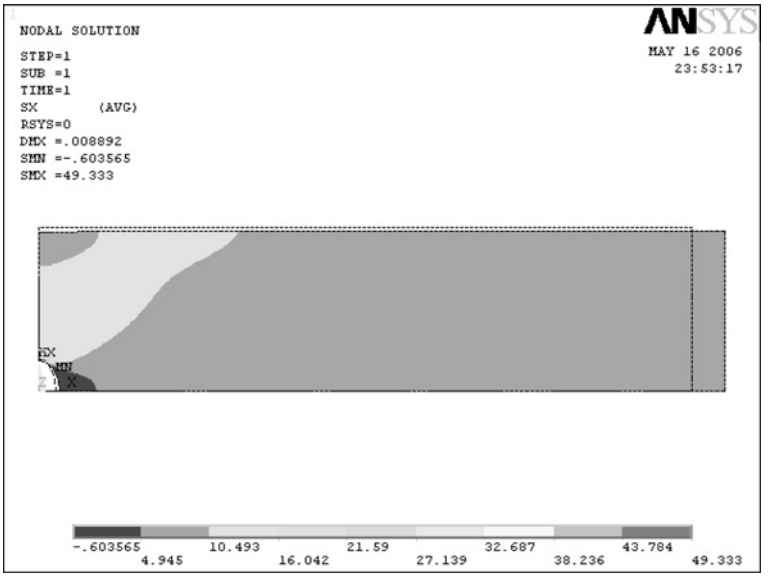


Figure 3.73 Contour of the x-component of stress in the quarter model of the center notched plate.

The **Symbols** window opens as shown in Figure 3.74:

(1) Select [A] **All Reactions** in the **[/PBC] Boundary condition symbol** buttons, [B] **Pressures** in the **[/PSF] Surface Load Symbols** box and [C] **Arrows** in the **[/PSF] Show pres and convect as** box.

(2) The distributions of the reaction force and the longitudinal stress in the ligament region (see [A] in Figure 3.75) as well as that of the applied stress on the right end of the plate area are superimposed on the contour x-component of stress in the plate in the **ANSYS Graphics** window. The reaction force is indicated by the leftward red arrows, whereas the longitudinal stress by the rightward red arrows in the ligament region.

The longitudinal stress reaches its maximum value at the foot of the hole and is decreased approaching to a constant value almost equal to the applied stress σ_0 at some distance, say, about one major diameter distance, from the foot of the hole. This tendency can be explained by the principle of St. Venant as discussed in the previous section.

3.3.4 Discussion

If an elliptical hole is placed in an infinite body and subjected to a remote uniform stress of σ_0, the maximum stress $\sigma_{\max}$ occurs at the foot of the hole, i.e., Point B in

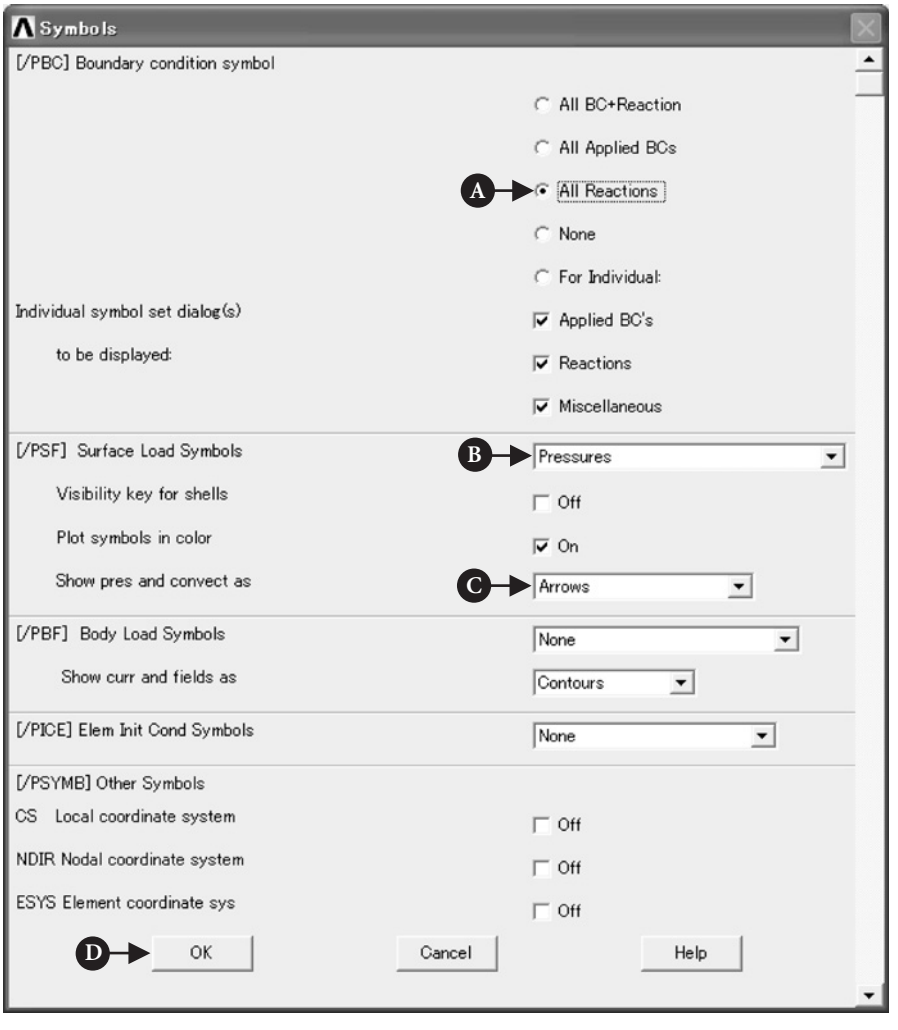


Figure 3.74 "Symbols" window.

Figure 3.65, and is expressed by the following formula:

$$\sigma_{\max} = \left(1 + 2\frac{a}{b}\right)\sigma_0 = \left(1 + \sqrt{\frac{a}{\rho}}\right)\sigma_0 = \alpha_0\sigma_0 \tag{3.4}$$

where a is the major radius, b the minor radius, ρ the local radius of curvature at the foot of the elliptic hole and α_0 the stress concentration factor which is

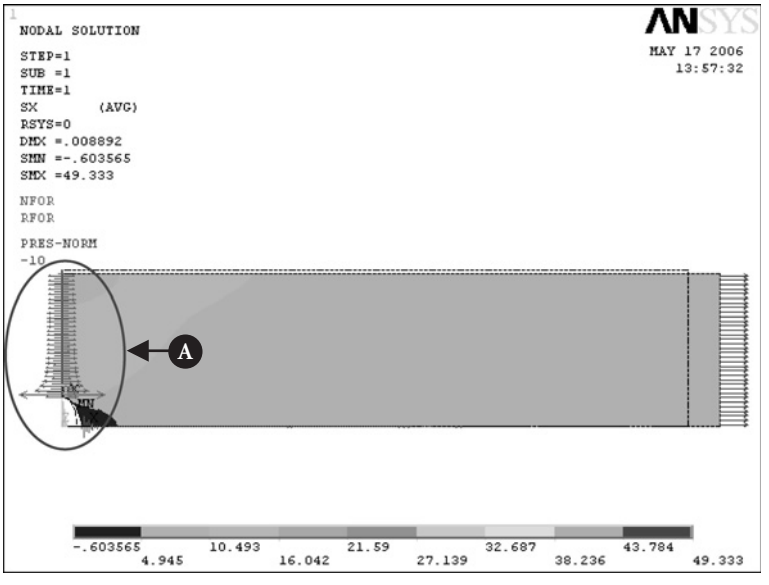


Figure 3.75 Applied stress on the right end side of the plate and resultant reaction force and longitudinal stress in its ligament region.

defined as,

$$\alpha_0 \equiv \frac{\sigma_{\max}}{\sigma_0} = \left(1 + 2\frac{a}{b}\right) = \left(1 + \sqrt{\frac{a}{\rho}}\right) \tag{3.5}$$

The stress concentration factor α_0 varies inversely proportional to the aspect ratio of elliptic hole b/a, namely the smaller the value of the aspect ratio b/a or the radius of curvature ρ becomes, the larger the value of the stress concentration factor α_0 becomes.

In a finite plate, the maximum stress at the foot of the hole is increased due to the finite boundary of the plate. Figure 3.76 shows the variation of the stress concentration factor α for elliptic holes having different aspect ratios with normalized major radius $2a/h$, indicating that the stress concentration factor in a plate with finite width h is increased dramatically as the ligament between the foot of the hole and the plate edge becomes smaller.

From Figure 3.76, the value of the stress intensity factor for the present elliptic hole is approximately 5.16, whereas Figure 3.75 shows that the maximum value of the longitudinal stress obtained by the present FEM calculation is approximately 49.3 MPa, i.e., the value of the stress concentration factor is approximately $49.3/10 = 4.93$. Hence, the relative error of the present calculation is approximately, $(4.93 - 5.16)/5.16 \approx -0.0446 = 4.46\%$ which may be reasonably small and so be acceptable.

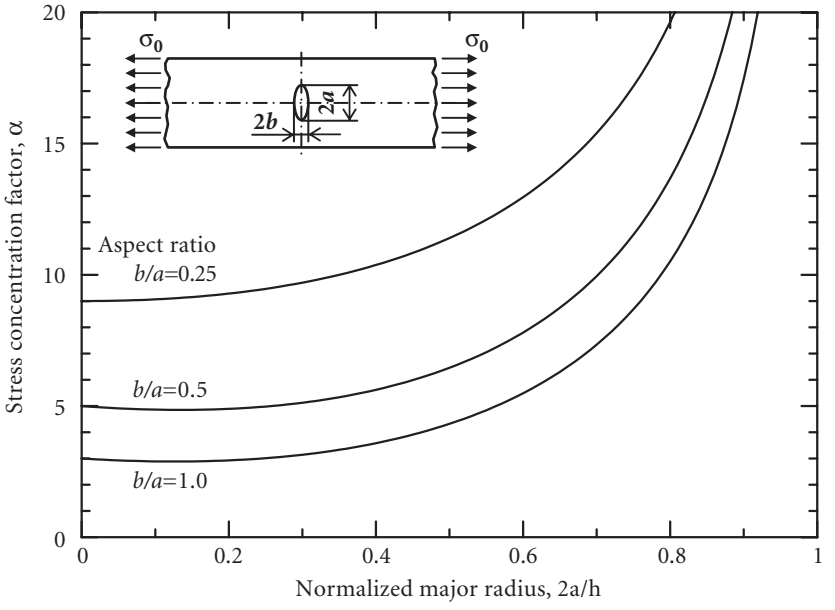


Figure 3.76 Variation of the stress concentration factor α for elliptic holes having different aspect ratios with normalized major radius $2a/h$.

3.3.5 Problems to solve

Problem 3.8

Calculate the value of stress concentration factor for the elliptic hole shown in Figure 3.65 by using the whole model of the plate and compare the result with that obtained and discussed in 3.3.4.

Problem 3.9

Calculate the values of stress concentration factor α for circular holes for different values of the normalized major radius $2a/h$ and plot the results as the α versus $2a/h$ diagram as shown in Figure 3.76.

Problem 3.10

Calculate the values of stress concentration factor α for elliptic holes having different aspect ratios b/a for different values of the normalized major radius $2a/h$ and plot the results as the α versus $2a/h$ diagram.

PROBLEM 3.11

For smaller values of $2a/h$, the disturbance of stress in the ligament between the foot of a hole and the plate edge due to the existence of the hole is decreased and stress in the ligament approaches to a constant value equal to the remote stress σ_0 at some distance from the foot of the hole (remember the principle of St. Venant in the previous section). Find how much distance from the foot of the hole the stress in the ligament region can be considered to be almost equal to the value of the remote stress σ_0.

3.4 STRESS SINGULARITY PROBLEM

3.4.1 EXAMPLE PROBLEM

An elastic plate with a crack of length $2a$ in its center subjected to uniform longitudinal tensile stress σ_0 at one end and clamped at the other end as illustrated in Figure 3.77. Perform an FEM analysis of the 2-D elastic center-cracked tension plate illustrated in Figure 3.77 and calculate the value of the mode I (crack-opening mode) stress intensity factor for the center-cracked plate.

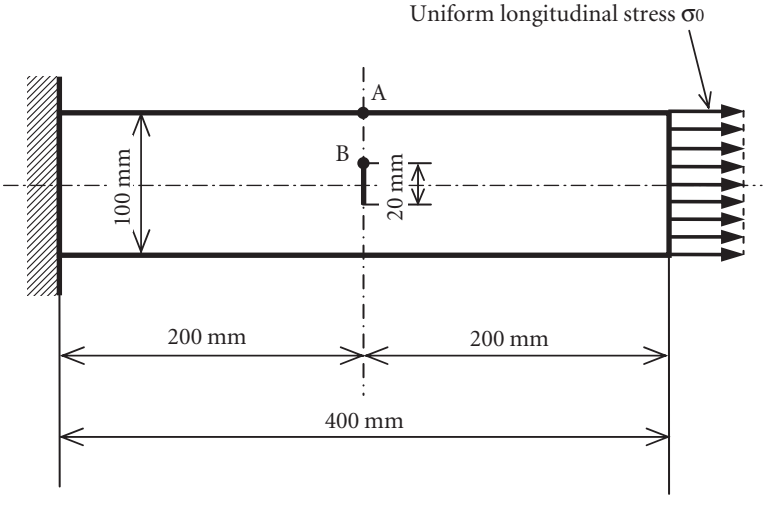


Figure 3.77 Two dimensional elastic plate with a crack of length $2a$ in its center subjected to a uniform tensile stress σ_0 in the longitudinal direction at one end and clamped at the other end.

3.4.2 PROBLEM DESCRIPTION

Specimen geometry: length $l = 400$ mm, height $h = 100$ mm.
Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.

Crack: A crack is placed perpendicular to the loading direction in the center of the plate and has a length of 20 mm. The center-cracked tension plate is assumed to be in the plane strain condition in the present analysis.
Boundary conditions: The elastic plate is subjected to a uniform tensile stress in the longitudinal direction at the right end and clamped to a rigid wall at the left end.

3.4.3 ANALYTICAL PROCEDURES

3.4.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a quarter model of the center-cracked tension plate as illustrated in Figure 3.77, since the plate is symmetric about the horizontal and vertical center lines.

Here we use the singular element or the quarter point element which can interpolate the stress distribution in the vicinity of the crack tip at which stress has the $1/\sqrt{r}$ singularity where r is the distance from the crack tip ($r/a \ll 1$). An ordinary isoparametric element which is familiar to you as "**Quad 8node 82**" has nodes at the corners and also at the midpoint on each side of the element. A singular element, however, has the midpoint moved one-quarter side distance from the original midpoint position to the node which is placed at the crack tip position. This is the reason why the singular element is often called the quarter point element instead. ANSYS software is equipped only with a 2-D triangular singular element, but neither with 2-D rectangular nor with 3-D singular elements. Around the node at the crack tip, a circular area is created and is divided into a designated number of triangular singular elements. Each triangular singular element has its vertex placed at the crack tip position and has the quarter points on the two sides joining the vertex and the other two nodes.

In order to create the singular elements, the plate area must be created via keypoints set at the four corner points and at the crack tip position on the left-end side of the quarter plate area.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

(1) The **Create Keypoints in Active Coordinate System** window opens as shown in Figure 3.78.

(2) Input [A] each **keypoint number** in the **NPT** box and [B] ***x***-, ***y***-, and ***z*-coordinates** of each keypoint in the three **X, Y, Z** boxes, respectively. Figure 3.78 shows the case of Key point #5, which is placed at the crack tip having the coordinates (0, 10, 0). In the present model, let us create Key points #1 to #5 at the coordinates (0, 0, 0), (200, 0, 0), (200, 50, 0), (0, 50, 0), and (0, 10, 0), respectively. Note that the z-coordinate is always 0 in 2-D elasticity problems.

(3) Click [C] **Apply** button four times and create Key points #1 to #4 without exiting from the window and finally click [D] **OK** button to create key point #5 at the crack tip position and exit from the window (see Figure 3.79).

Then create the plate area via the five key points created in the procedures above by the following commands:

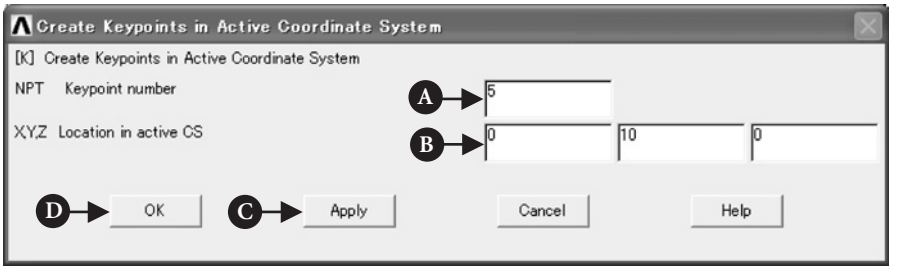


Figure 3.78 "Create Keypoints in Active Coordinate System" window.

Figure 3.79 Five key points created in the "**ANSYS Graphics**" window.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → Through KPs**

(1) The **Create Area thru KPs** window opens.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to Key point #1 and click this point. Click Key points #1 through #5 one by one counterclockwise (see Figure 3.80).

(3) Click the **OK** button to create the plate area as shown in Figure 3.81.

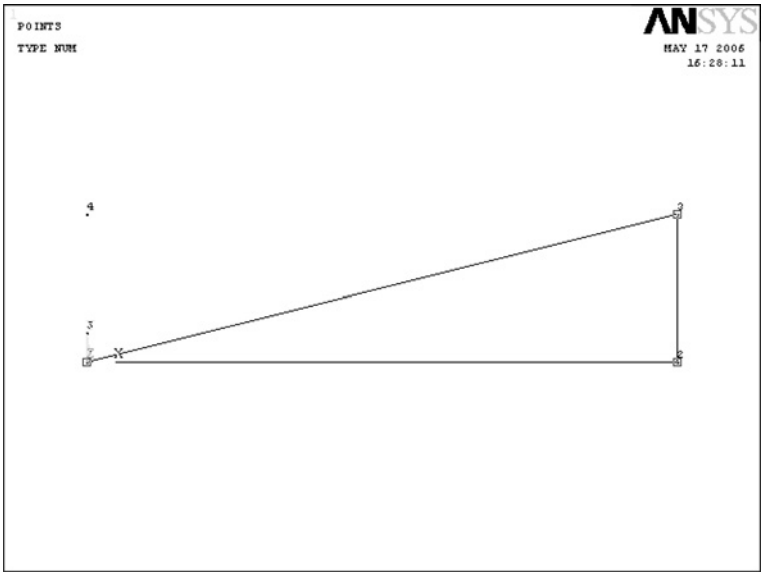


Figure 3.80 Clicking Key points #1 through #5 one by one counterclockwise to create the plate area.

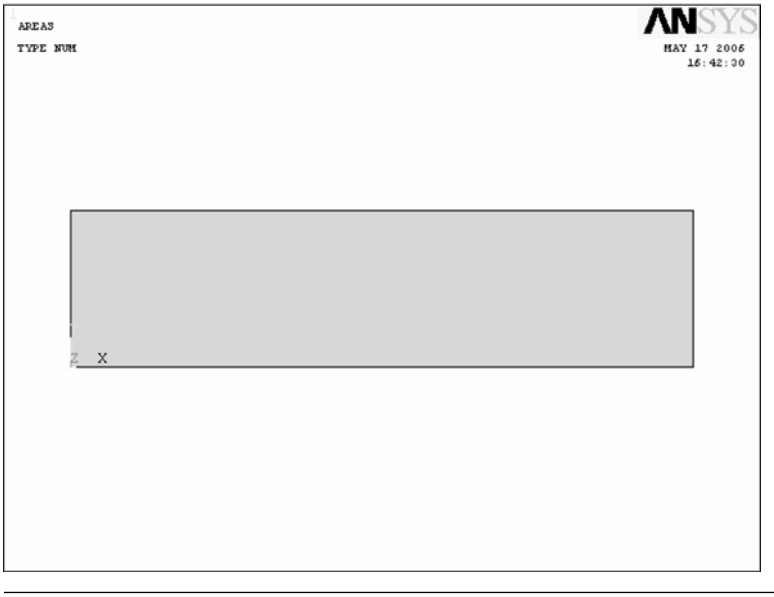


Figure 3.81 Quarter model of the center cracked tension plate.

3.4.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE PLATE MATERIAL

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) The **Define Material Model Behavior** window opens.
(2) Double-click **Structural, Linear, Elastic**, and **Isotropic** buttons one after another.
(3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.
(4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.4.3.3 FINITE-ELEMENT DISCRETIZATION OF THE CENTER-CRACKED TENSION PLATE AREA

[1] Selection of the element type

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

(1) The **Element Types** window opens.
(2) Click the **Add …** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.
(3) Select **Structural Mass – Solid** and **Quad 8 node 82.**
(4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element.
(5) Click the **Options …** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strain** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

[2] Sizing of the elements
Before meshing, the crack tip point around which the triangular singular elements will be created must be specified by the following commands:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Concentrate KPs → Create**

(1) The **Concentration Keypoint** window opens.
(2) Display the key points in the **ANSYS Graphics** window by

COMMAND | **ANSYS Utility Menu → Plot → Keypoints → Keypoints.**

(3) Pick Key point #5 by placing the upward arrow onto Key point #5 and by clicking the left button of the mouse. Then click the **OK** button in the **Concentration Keypoint** window.

(4) Another **Concentration Keypoint** window opens as shown in Figure 3.82.

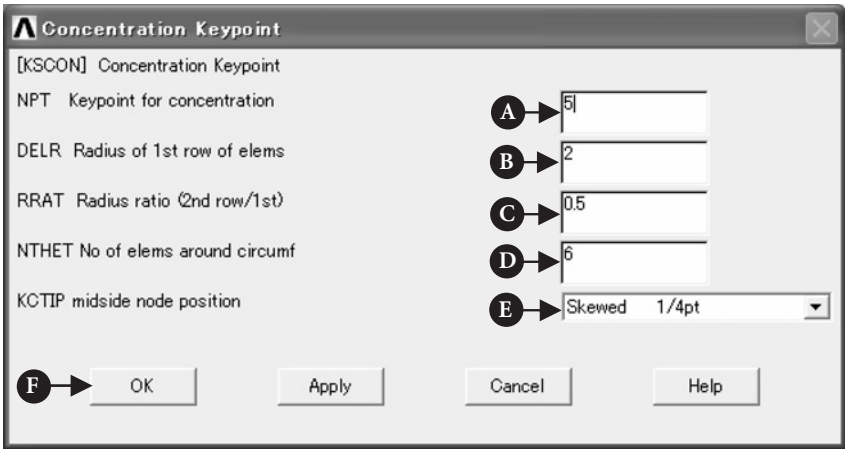


Figure 3.82 "Concentration Keypoint" window.

(5) Confirming that [A] **5,** i.e., the key point number of the crack tip position is input in the **NPT** box, input [B] **2** in the **DELR** box, [C] **0.5** in the **RRAT** box and [D] **6** in the **NTHET** box and select [E] **Skewed 1/4pt** in the **KCTIP** box in the window. Refer to the explanations of the numerical data described after the names of the respective boxes on the window. **Skewed 1/4pt** in the last box means that the mid nodes of the sides of the elements which contain Key point #5 are the quarter points of the elements.

(6) Click [F] **OK** button in the **Concentration Keypoint** window.

The size of the meshes other than the singular elements and the elements adjacent to them can be controlled by the same procedures as have been described in the previous sections of the present Chapter 3.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Global → Size**

(1) The **Global Element Sizes** window opens.

(2) Input **1.5** in the **SIZE** box and click the **OK** button.

[3] Meshing

The meshing procedures are also the same as before.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

(1) The **Mesh Areas** window opens.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter plate area and click this area.

(3) The color of the area turns from light blue into pink. Click the **OK** button.

(4) The **Warning** window appears as shown in Figure 3.83 due to the existence of six singular elements. Click [A] **Close** button and proceed to the next operation below.

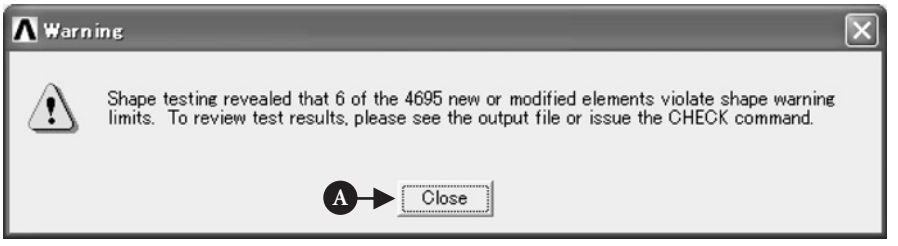


Figure 3.83 "Warning" window.

(5) Figure 3.84 shows the plate area meshed by ordinary 8-node isoparametric finite elements except for the vicinity of the crack tip where we have six singular elements.

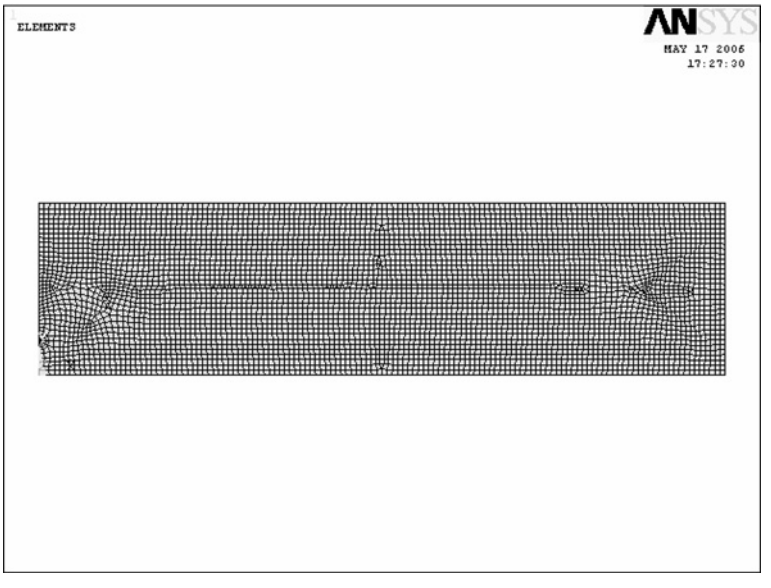


Figure 3.84 Plate area meshed by ordinary 8-node isoparametric finite elements and by singular elements.

Figure 3.85 is an enlarged view of the singular elements around [A] the crack tip showing that six triangular elements are placed in a radial manner and that the size of the second row of elements is half the radius of the first row of elements, i.e., triangular singular elements.

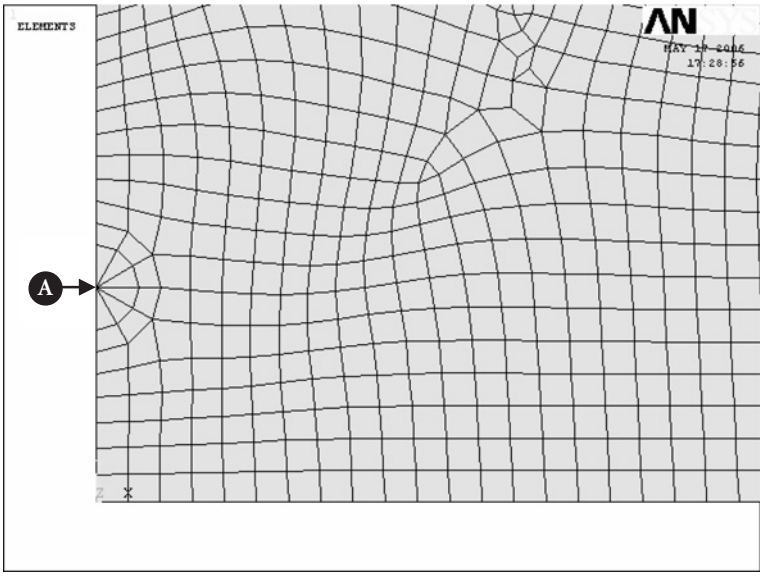


Figure 3.85 Enlarged view of the singular elements around the crack tip.

3.4.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the ligament region of the left end and the bottom side of the quarter plate model

Due to the symmetry, the constraint conditions of the quarter plate model are **UX**-fixed condition on the ligament region of the left end, that is, the line between Key points #4 and #5, and **UY**-fixed condition on the bottom side of the quarter plate model. Apply these constraint conditions onto the corresponding lines by the following commands:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

(1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the line between Key points #4 and #5 and click the left button of the mouse.

(3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.

(4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform longitudinal stress on the right end of the quarter plate model

A uniform longitudinal stress can be defined by pressure on the right-end side of the plate model as described below:

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

(1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the right-end side of the quarter plate area and click the left button of the mouse.

(3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the [**SFL**] **Apply PRES on lines as a** box and input **−10** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.

(4) Click the **OK** button in the window to define a uniform tensile stress of 10 MPa applied to the right end of the quarter plate model.

Figure 3.86 illustrates the boundary conditions applied to the center-cracked tension plate model by the above operations.

3.4.3.5 SOLUTION PROCEDURES

The solution procedures are also the same as usual.

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

(1) The **Solve Current Load Step** and **/STATUS Command** windows appear.

(2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.

(3) The **Verify** window opens as shown in Figure 3.87. Proceed to the next operation below by clicking [A] **Yes** button in the window.

(4) Select the **File** button in **/STATUS Command** window to open the submenu and select the **Close** button to close the **/STATUS Command** window.

(5) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

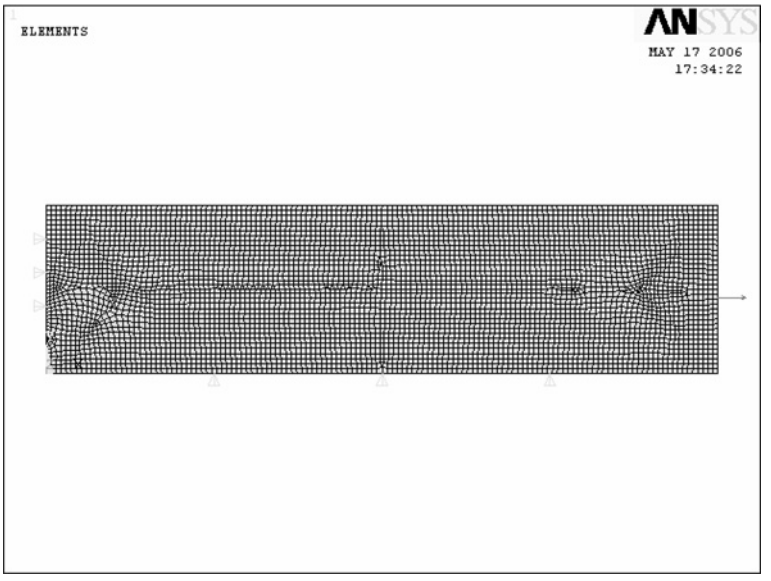


Figure 3.86 Boundary conditions applied to the center-cracked tension plate model.

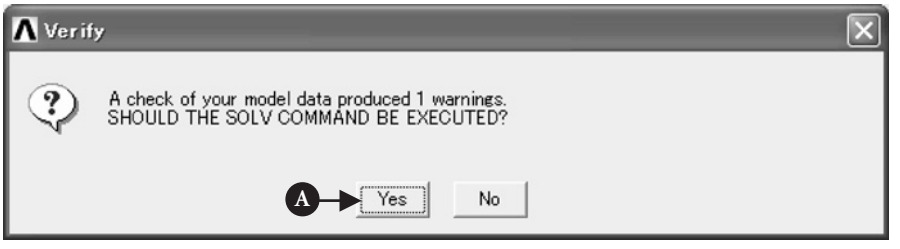


Figure 3.87 "Verify" window.

3.4.3.6 CONTOUR PLOT OF STRESS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution**

(1) The **Contour Nodal Solution Data** window opens.

(2) Select **Stress** and **X-Component of stress**.

(3) Select **Deformed shape with deformed edge** in the **Undisplaced shape key** box to compare the shapes of the cracked plate before and after deformation.

(4) Click the **OK** button to display the contour of the x-component of stress, or longitudinal stress in the center-cracked tension plate in the **ANSYS Graphics** window as shown in Figure 3.88.

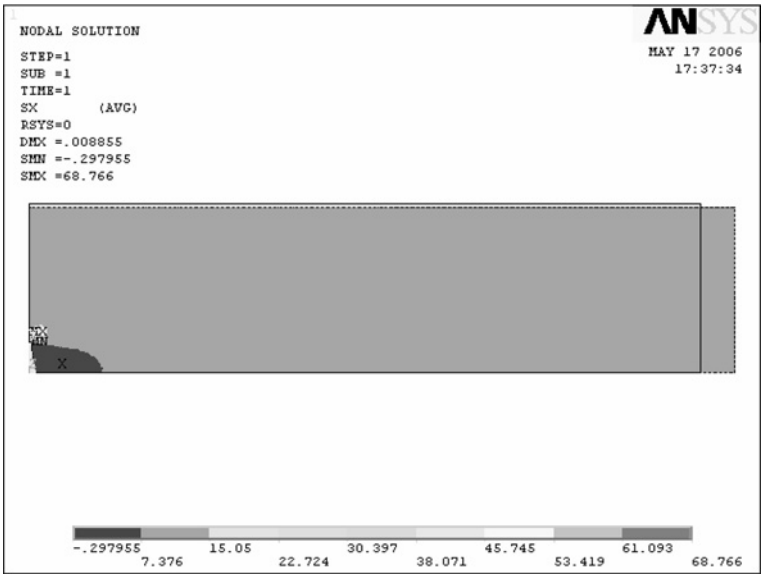


Figure 3.88 Contour of the x-component of stress in the center-cracked tension plate.

Figure 3.89 is an enlarged view of the longitudinal stress distribution around the crack tip showing that very high tensile stress is induced at the crack tip whereas almost zero stress around the crack surface and that the crack shape is parabolic.

3.4.4 Discussion

Figure 3.90 shows extrapolation of the values of the correction factor, or the non-dimensional mode I (the crack-opening mode) stress intensity factor $F_I = K_I/(\sigma\sqrt{\pi a})$ to the point where $r = 0$, that is, the crack tip position by the hybrid extrapolation method [1]. The plots in the right region of the figure are obtained by the formula:

$$K_{\mathrm{I}} = \lim_{r\to 0} \sqrt{2\pi r}\sigma_x(\theta = 0) \tag{3.6}$$

or

$$F_{\mathrm{I}}(\lambda) = \frac{K_{\mathrm{I}}}{\sigma\sqrt{\pi a}} = \lim_{r\to 0} \frac{\sqrt{\pi r}\sigma_x(\theta = 0)}{\sqrt{h}\sigma\sqrt{\pi\lambda}} \quad \textit{where } \lambda = 2a/h \tag{3.6'}$$

whereas those in the left region

$$K_{\mathrm{I}} = \lim_{r\to 0} \sqrt{\frac{2\pi}{r}} \frac{E}{(1+\nu)(\kappa+1)} u_x(\theta = \pi) \tag{3.7}$$

or

$$F_{\mathrm{I}}(\lambda) = \frac{K_{\mathrm{I}}}{\sigma\sqrt{\pi a}} = \lim_{r\to 0} \sqrt{\frac{\pi}{hr}} \frac{E}{(1+\nu)(\kappa+1)\sigma\sqrt{\pi\lambda}} u_x(\theta = \pi) \tag{3.7'}$$

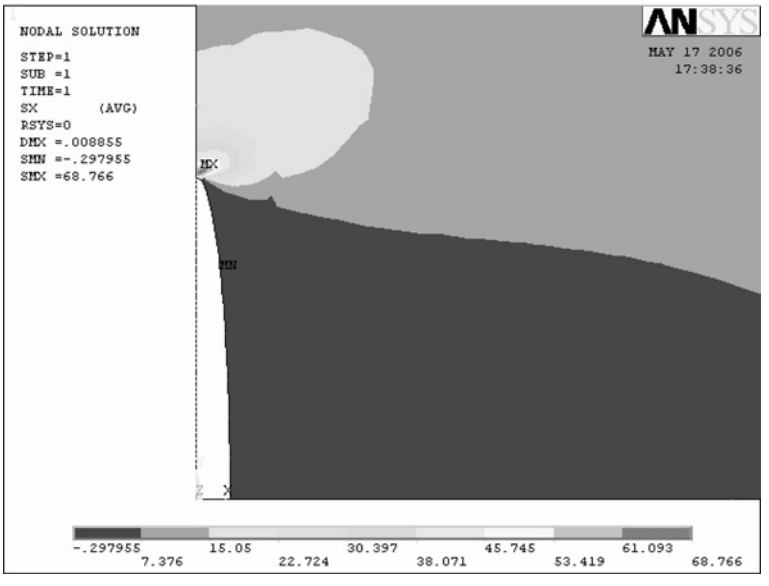


Figure 3.89 Enlarged view of the longitudinal stress distribution around the crack tip.

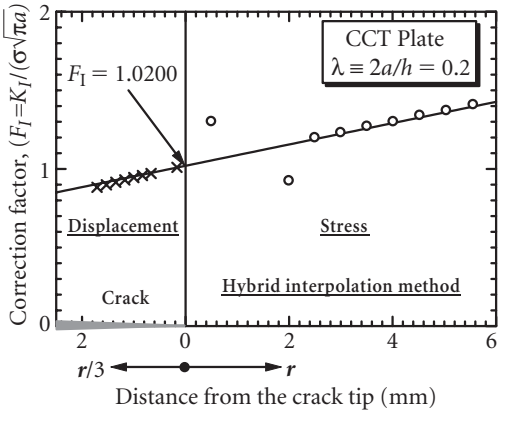


Figure 3.90 Extrapolation of the values of the correction factor $F_{\mathrm{I}} = K_I/(\sigma\sqrt{\pi a})$ to the crack tip point where $r = 0$.

where r is the distance from the crack tip, σ_x $(\theta = 0)$ the x-component of stress at $\theta = 0$, i.e., along the ligament, σ the applied uniform stress in the direction of the x-axis, a the half crack length, E Young's modulus, $u_x(\theta = \pi)$ the x-component of strain at $\theta = \pi$, i.e., along the crack surface, $\kappa = 3-4\,\nu$ for plane strain condition and $\kappa = (3-\nu)/(1+\nu)$ for plane stress condition and θ the angle around the crack tip. The correction factor F_{I} accounts for the effect of the finite boundary, or the edge effect of the plate. The value of F_{I} can be evaluated either by the stress extrapolation method

(the right region) or by the displacement extrapolation method (the left region). The hybrid extrapolation method [1] is a blend of the two types of the extrapolation methods above and can bring about better solutions. As illustrated in Figure 3.90, the extrapolation lines obtained by the two methods agree well with each other when the gradient of the extrapolation line for the displacement method is rearranged by putting $r' = r/3$. The value of F_{I} obtained for the present CCT plate by the hybrid method is 1.0200.

Table 3.1 Correction factor $F_{\mathrm{I}}(\lambda)$ as a function of non-dimensional crack length $\lambda = 2a/h$

$\lambda = 2a/h$	Isida Equation (3.8)	Feddersen Equation (3.9)	Koiter Equation (3.10)	Tada Equation (3.11)
0.1	1.0060	1.0062	1.0048	1.0060
0.2	1.0246	1.0254	1.0208	1.0245
0.3	1.0577	1.0594	1.0510	1.0574
0.4	1.1094	1.1118	1.1000	1.1090
0.5	1.1867	1.1892	1.1757	1.1862
0.6	1.3033	1.3043	1.2921	1.3027
0.7	1.4882	1.4841	1.4779	1.4873
0.8	1.8160	1.7989	1.8075	1.8143
0.9	2.5776	–	2.5730	2.5767

Table 3.1 lists the values of F_{I} calculated by the following four equations [2–6] for various values of non-dimensional crack length $\lambda = 2a/h$:

$$F_{\mathrm{I}}(\lambda) = 1 + \sum_{k=1}^{35} A_k \lambda^{2k} \tag{3.8}$$

$$F_{\mathrm{I}}(\lambda) = \sqrt{\sec(\pi\lambda/2)} \quad (\lambda \leq 0.8) \tag{3.9}$$

$$F_{\mathrm{I}}(\lambda) = \frac{1 - 0.5\lambda + 0.326\lambda^2}{\sqrt{1-\lambda}} \tag{3.10}$$

$$F_I(\lambda) = (1 - 0.025\lambda^2 + 0.06\lambda^4)\sqrt{\sec(\pi\lambda/2)} \tag{3.11}$$

Among these, Isida's solution is considered to be the best. The value of $F_{\mathrm{I}}(0.2)$ by Isida is 1.0246. The relative error of the present result is $(1.0200 - 1.0246)/1.0246 = -0.45\%$, which is a reasonably good result.

3.4.5 Problems to solve

Problem 3.12

Calculate the values of the correction factor for the mode I stress intensity factor for the center-cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$

to 0.9. Compare the results with the values of the correction factor $F(\lambda)$ listed in Table 3.1

PROBLEM 3.13

Calculate the values of the correction factor for the mode I stress intensity factor for a double edge cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$ to 0.9 (see Figure P3.13). Compare the results with the values of the correction factor $F(\lambda)$ calculated by Equation (P3.13) [7, 8]. Note that the quarter model described in the present section can be used for the FEM calculations of this problem simply by changing the boundary conditions.

$$F_I(\lambda) = \frac{1.122 - 0.561\lambda - 0.205\lambda^2 + 0.471\lambda^3 - 0.190\lambda^4}{\sqrt{1-\lambda}} \quad \text{where } \lambda = 2a/h \tag{P3.13}$$

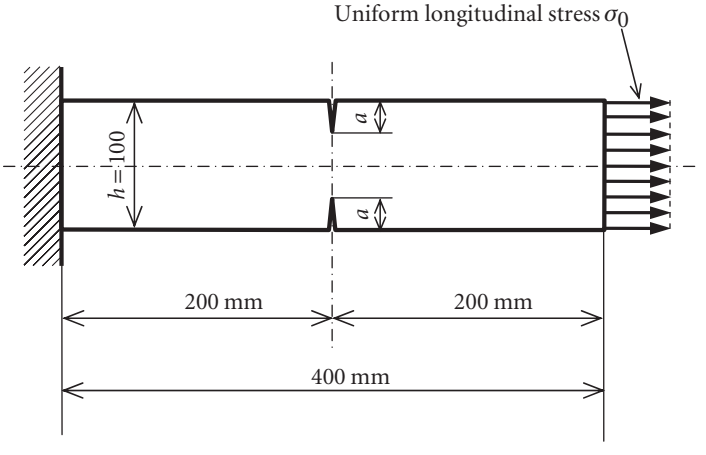


Figure P3.13 Double edge cracked tension plate subjected to a uniform longitudinal stress at one end and clamped at the other end.

PROBLEM 3.14

Calculate the values of the correction factor for the mode I stress intensity factor for a single edge cracked tension plate having normalized crack lengths of $\lambda = 2a/h = 0.1$ to 0.9 (see Figure P3.14). Compare the results with the values of the correction factor $F(\lambda)$ calculated by Equation (P3.14) [7, 9]. In this case, a half plate model which is clamped along the ligament and is loaded at the right end of the plate. Note that the whole plate model with an edge crack needs a somewhat complicated procedure. An

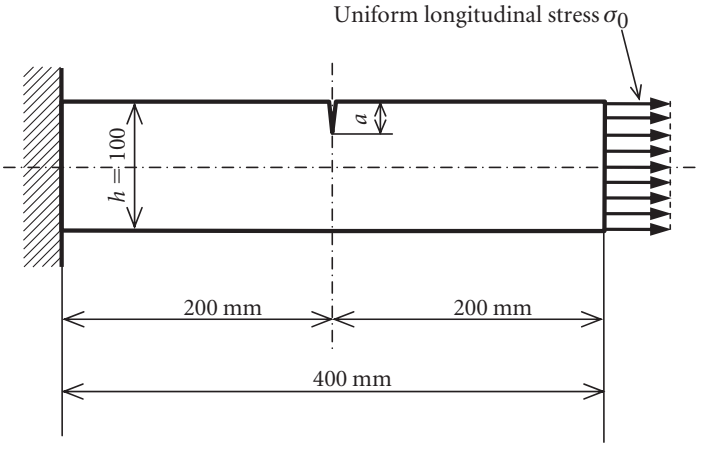


Figure P3.14 Single edge cracked tension plate subjected to a uniform longitudinal stress at one end and clamped at the other end.

approximate model as described above gives good solutions with an error of a few percent or less and is sometimes effective in practice.

$$F_{\mathrm{I}}(\lambda) = \frac{0.752 + 2.02\lambda + 0.37[1 - \sin(\pi\lambda/2)]^3}{\cos(\pi\lambda/2)} \sqrt{\frac{2}{\pi\lambda} \tan\left(\frac{\pi\lambda}{2}\right)} \quad \text{where } \lambda = a/h \tag{P3.14}$$

3.5 Two-dimensional contact stress

3.5.1 Example problem

An elastic cylinder of mild steel with a radius of R pressed against a flat surface of a linearly elastic medium of the same material by a force P'. Perform an FEM analysis of the elastic cylinder pressed against the elastic flat plate illustrated in Figure 3.91 and calculate the resulting contact stress in the vicinity of the contact point in the flat plate.

3.5.2 Problem description

Geometry of the cylinder: radius $R = 1000$ mm.
Geometry of the flat plate: width $W = 1000$ mm, height $h = 500$ mm.
Material: mild steel having Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$.
Boundary conditions: The elastic cylinder is pressed against the elastic flat plate by a force of $P' = 1$ kN/mm and the plate is clamped at the bottom. Note that the cylinder and the plate is considered to have an unit thickness in the case of plane stress condition.

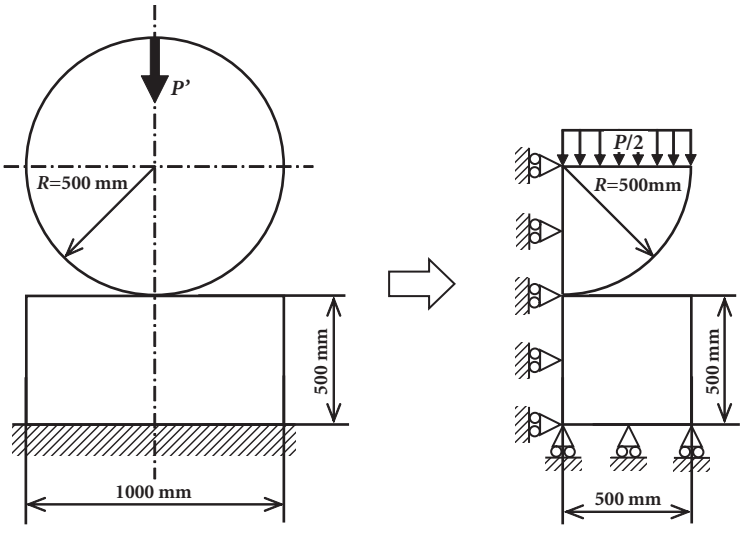


Figure 3.91 Elastic cylinder of mild steel with a radius of R pressed against an elastic flat plate of the same steel by a force P' and the FEM model of the cylinder-plate system.

3.5.3 ANALYTICAL PROCEDURES

3.5.3.1 CREATION OF AN ANALYTICAL MODEL

Let us use a half model of the indentation problem illustrated in Figure 3.91, since the problem is symmetric about the vertical center line.

[1] Creation of an elastic flat plate

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By 2 Corners**

(1) Input two **0**'s into the **WP X** and **WP Y** boxes in the **Rectangle by 2 Corners** window to determine the upper left corner point of the elastic flat plate on the Cartesian coordinates of the working plane.
(2) Input **500** and **−500** (mm) into the **Width** and **Height** boxes, respectively, to determine the shape of the elastic flat plate model.
(3) Click the **OK** button to create the plate on the **ANSYS Graphics** window.

[2] Creation of an elastic cylinder

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Partial Annulus**

(1) The **Part Annular Circ Area** window opens as shown in Figure 3.92.

(2) Input **0** and **500** into [A] **WP X** and [B] **WP Y** boxes, respectively, in the **Part Annular Circ Area** window to determine the center of the elastic cylinder on the Cartesian coordinates of the working plane.

(3) Input two **500**'s (degree) into [C] **Rad-1** and [E] **Rad-2** boxes to designate the radius of the elastic cylinder model.

(4) Input **−90** and **0** (mm) into [D] **Theta-1** and [F] **Theta-2** boxes, respectively, to designate the starting and ending angles of the quarter elastic cylinder model.

(5) Click [G] **OK** button to create the quarter cylinder model on the **ANSYS Graphics** window as shown in Figure 3.93.

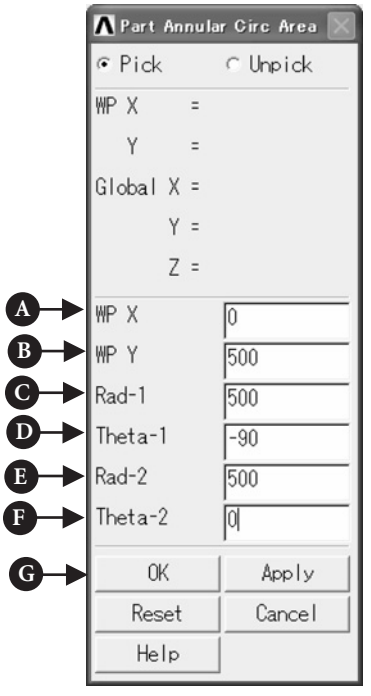


Figure 3.92 "Part Annular Circ Area" window.

[3] Combining the elastic cylinder and the flat plate models

The elastic cylinder and the flat plate models must be combined into one model so as not to allow the rigid-body motions of the two bodies.

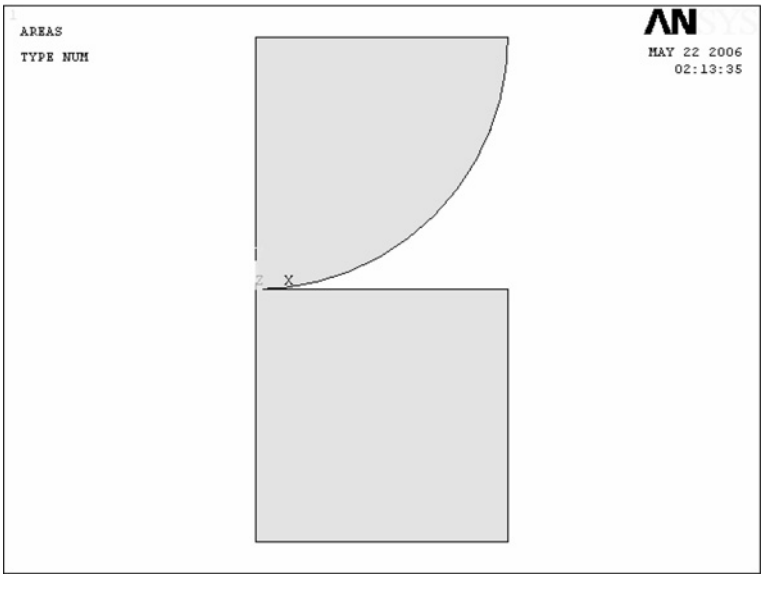


Figure 3.93 Areas of the FEM model for an elastic cylinder pressed against an elastic flat plate.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Glue → Areas**

(1) The **Glue Areas** window opens as shown in Figure 3.94.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter cylinder area and click this area, and then move the arrow on to the half plate area and click this area to combine these two areas.

(3) Click [A] **OK** button to create the contact model on the **ANSYS Graphics** window as shown in Figure 3.93.

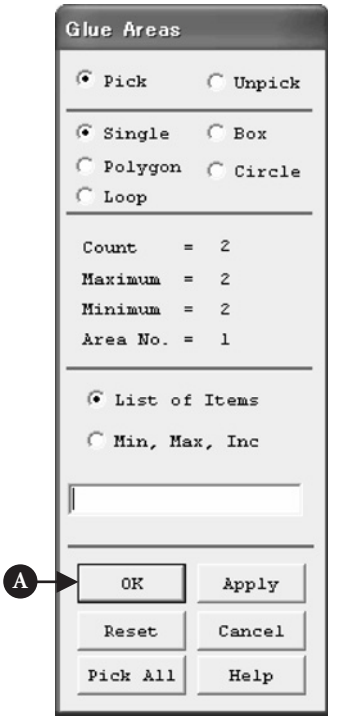


Figure 3.94 "Glue Areas" window.

3.5.3.2 INPUT OF THE ELASTIC PROPERTIES OF THE MATERIAL FOR THE CYLINDER AND THE FLAT PLATE

COMMAND **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) The **Define Material Model Behavior** window opens.

(2) Double-click **Structural, Linear, Elastic**, and **Isotropic** buttons one after another.

(3) Input the value of Young's modulus, **2.1e5** (MPa), and that of Poisson's ratio, **0.3**, into **EX** and **PRXY** boxes, and click the **OK** button of the **Linear Isotropic Properties for Materials Number 1** window.

(4) Exit from the **Define Material Model Behavior** window by selecting **Exit** in the **Material** menu of the window.

3.5.3.3 FINITE-ELEMENT DISCRETIZATION OF THE CYLINDER AND THE FLAT PLATE AREAS

[1] Selection of the element types

Select the 8-node isoparametric elements as usual.

COMMAND **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

(1) The **Element Types** window opens.

(2) Click the **Add …** button in the **Element Types** window to open the **Library of Element Types** window and select the element type to use.

(3) Select **Structural Mass – Solid** and **Quad 8 node 82.**

(4) Click the **OK** button in the **Library of Element Types** window to use the 8-node isoparametric element. Note that this element is defined as Type 1 element as indicated in the **Element type reference number** box.

(5) Click the **Options …** button in the **Element Types** window to open the **PLANE82 element type options** window. Select the **Plane strain** item in the **Element behavior** box and click the **OK** button to return to the **Element Types** window. Click the **Close** button in the **Element Types** window to close the window.

In contact problems, two mating surfaces come into contact with each other exerting great force on each other. Contact elements must be used in contact problems for preventing penetration of one object into the other:

(6) Repeat clicking the **Add …** button in the **Element Types** window to open the **Library of Element Types** window and select **Contact** and **2D target 169**.

(7) Click the **OK** button in the **Library of Element Types** window to select the target element. Note that this target element is defined as Type 2 element as indicated in the **Element type reference number** box.

(8) Clicking again the **Add …** button in the **Element Types** window to open the **Library of Element Types** window and select **Contact** and **3 nd surf 172**.

(9) Click the **OK** button in the **Library of Element Types** window to select the surface effect element. Note that this target element is defined as Type 3 element as indicated in the **Element type reference number** box.

[2] Sizing of the elements

Here let us designate the element size by specifying number of divisions of lines picked. For this purpose, carry out the following commands:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Lines → Picked Lines**

(1) The **Element Size on Pick …** window opens as shown in Figure 3.95.

(2) Move the upward arrow to the upper and the right sides of the flat plate area and click [A] **OK** button and the **Element Sizes on Picked Lines** window opens as shown in Figure 3.96.

(3) Input [A] **60** in **NDIV** box and [B] **1/10** in **SPACE** box to divide the upper and right side of the plate area into 60 subdivisions. The size of the subdivisions decreases in geometric progression as approaching to the left side of the

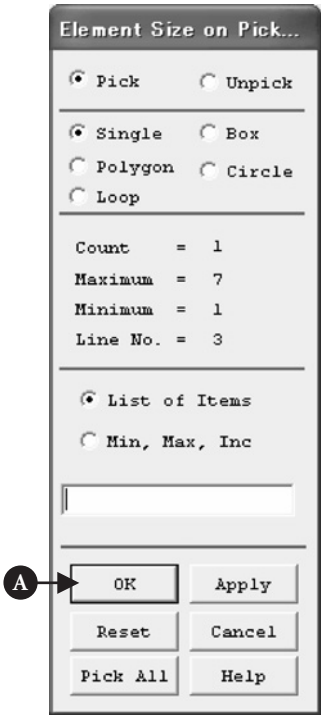


Figure 3.95 "Element Size on Pick . . ." window.

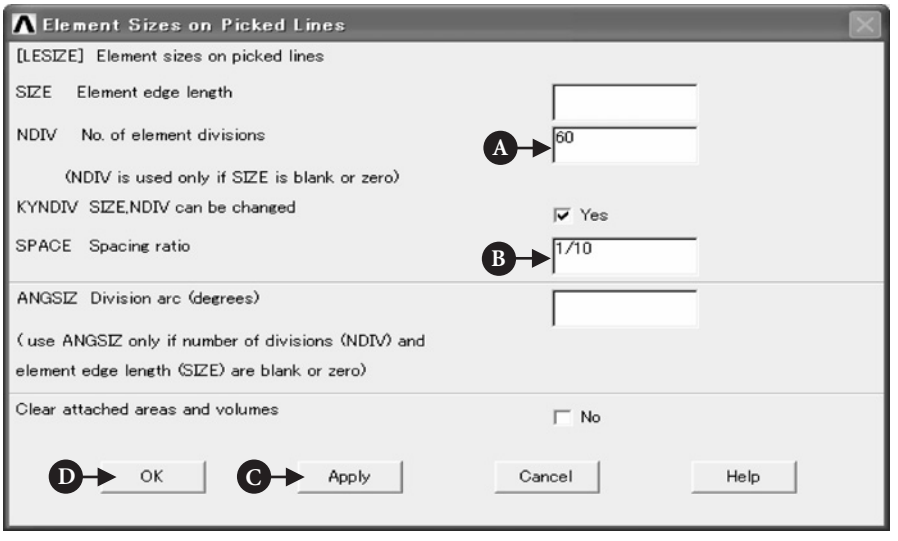


Figure 3.96 "Element Sizes on Picked Lines" window.

plate area. It would be preferable that we may have smaller elements around the contact point where high stress concentration occurs. Click [C] **Apply** button to continue to input more **NDIV** and **SPACE** data. Input **60** in the **NDIV** box and **10** in the **SPACE** box for the left and the bottom sides of the plate, whereas **40** and **10** for the left side and the circumference of the quarter circular area and **40** and **1/10** for the upper side of the quarter circular area in the respective boxes. Finally, click [D] **OK** button to close the window.

[3] Meshing

The meshing procedures are the same as usual:

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

(1) The **Mesh Areas** window opens.

(2) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter cylinder and the half plate areas and click these areas.

(3) The color of the areas turns from light blue into pink. Click the **OK** button to see the areas meshed by 8-node isoparametric finite elements as shown in Figure 3.97.

(4) Figure 3.98 is an enlarged view of the finer meshes around the contact point.

[4] Creation of target and contact elements

First select the lower surface of the quarter cylinder to which the contact elements are attached:

COMMAND | **ANSYS Utility Menu → Select → Entities …**

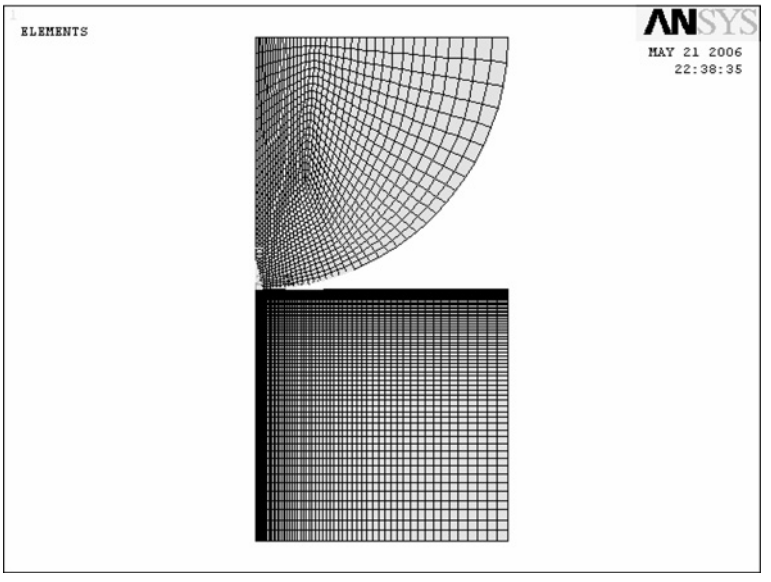


Figure 3.97 Cylinder and flat plate areas meshed by 8-node isoparametric finite elements.

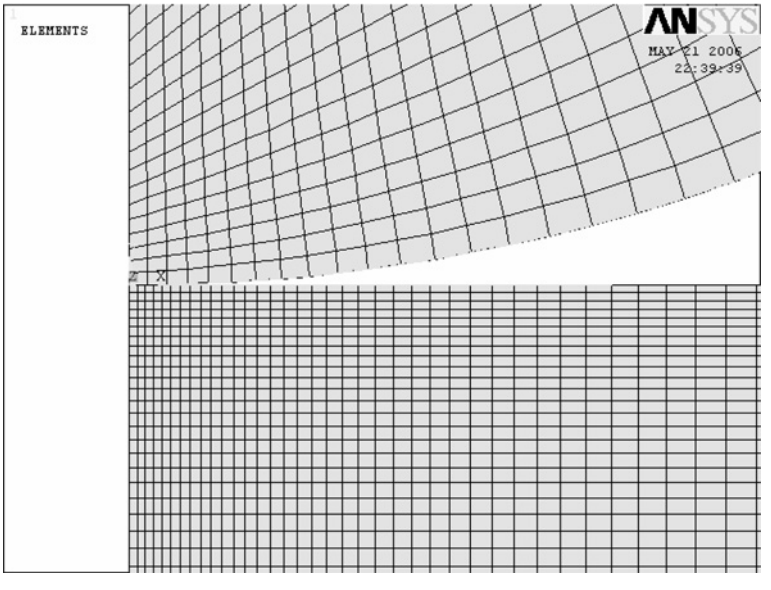


Figure 3.98 Enlarged view of the finer meshes around the contact point.

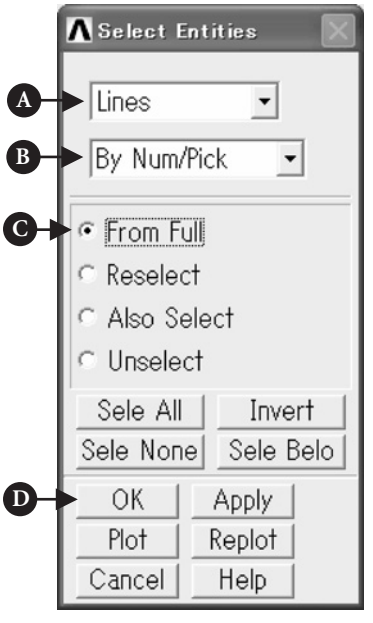


Figure 3.99 "Select Entities" window.

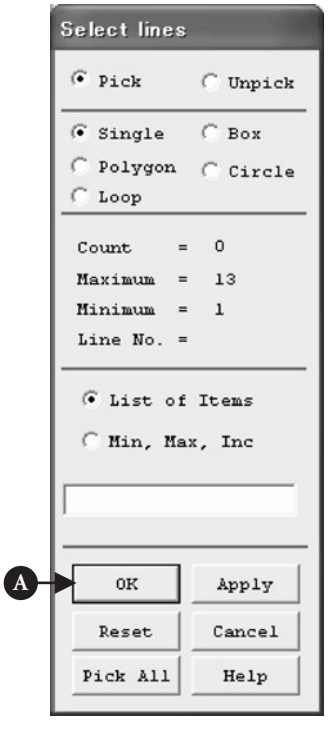


Figure 3.100 "Select lines" window.

(1) The **Select Entities** window opens as shown in Figure 3.99.

(2) Select [A] **Lines**, [B] **By Num/Pick** and [C] **From Full** in the **Select Entities** window.

(3) Click [D] **OK** button in the **Select Entities** window and the **Select lines** window opens as shown in Figure 3.100.

(4) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter circumference of the cylinder and click this line. The lower surface of the cylinder is selected as shown in Figure 3.101.

Repeat the **Select** → **Entities ...** commands to select nodes on the lower surface line of the cylinder:

COMMAND | **ANSYS Utility Menu → Select → Entities …**

(1) The **Select Entities** window opens as shown in Figure 3.102.

(2) Select [A] **Nodes**, [B] **Attached to**, [C] **Lines, all** and [D] **Reselect** in the **Select Entities** window.

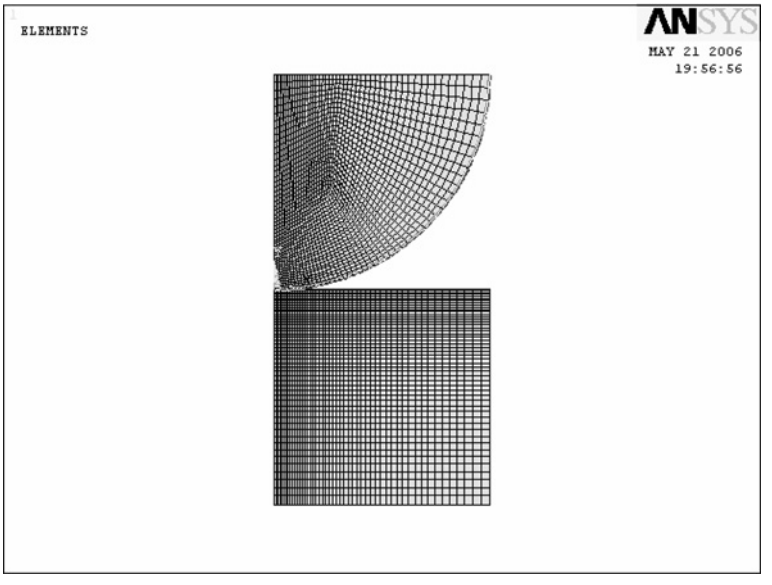


Figure 3.101 Lower surface of the cylinder selected.

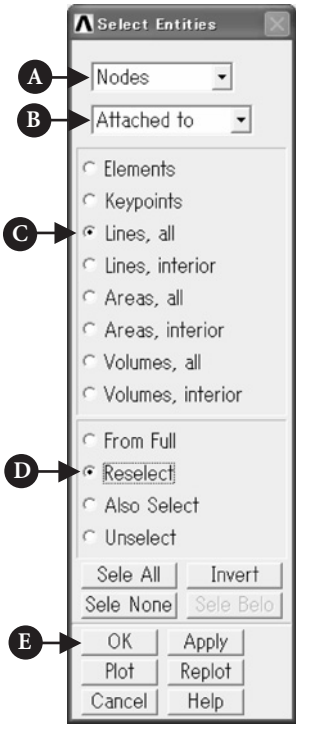


Figure 3.102 "Select Entities" window.

(3) Click [E] **OK** button and perform the following commands:

COMMAND | **ANSYS Utility Menu → Plot → Nodes**

Only nodes on the lower surface of the cylinder are plotted in the **ANSYS Graphics** window as shown in Figure 3.103.

Repeat the **Select → Entities ...** commands again to select a fewer number of the nodes on the portion of the lower surface of the cylinder in the vicinity of the contact point:

COMMAND | **ANSYS Utility Menu → Select → Entities …**

(1) The **Select Entities** window opens as shown in Figure 3.104.

(2) Select [A] **Nodes**, [B] **By Num/Pick** and [C] **Reselect** in the **Select Entities** window.

(3) Click [D] **OK** button and the **Select nodes** window opens as shown in Figure 3.105.

(4) The upward arrow appears in the **ANSYS Graphics** window. Select [A] **Box** instead of **Single**, then move the upward arrow to the leftmost end of the array of nodes and select nodes by enclosing them with a rectangle formed by dragging

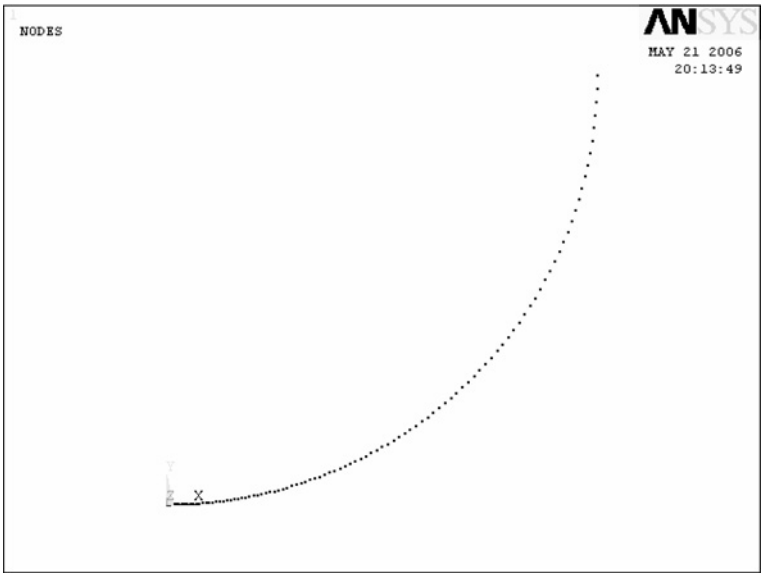


Figure 3.103 Nodes on the lower surface of the cylinder plotted in the "ANSYS Graphics" window.

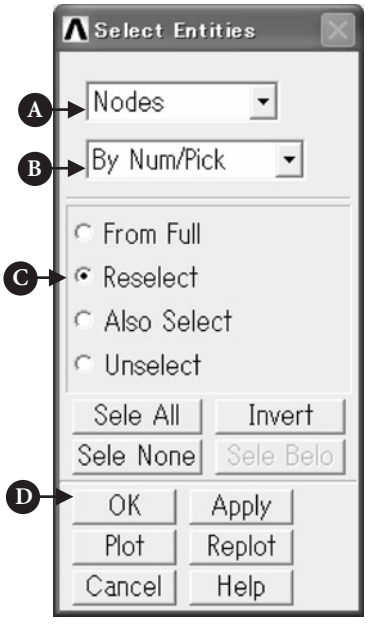


Figure 3.104 "Select Entities" window.

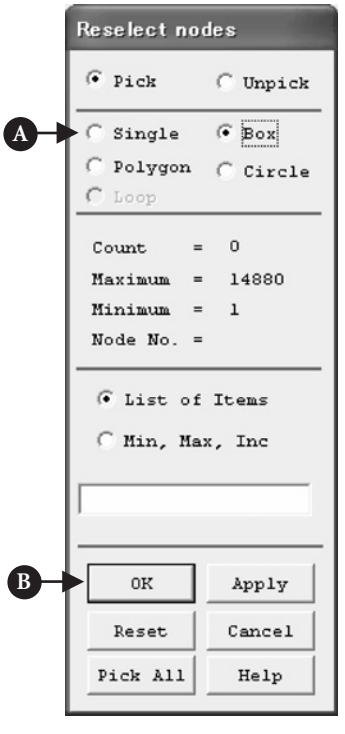


Figure 3.105 "Reselect nodes" window.

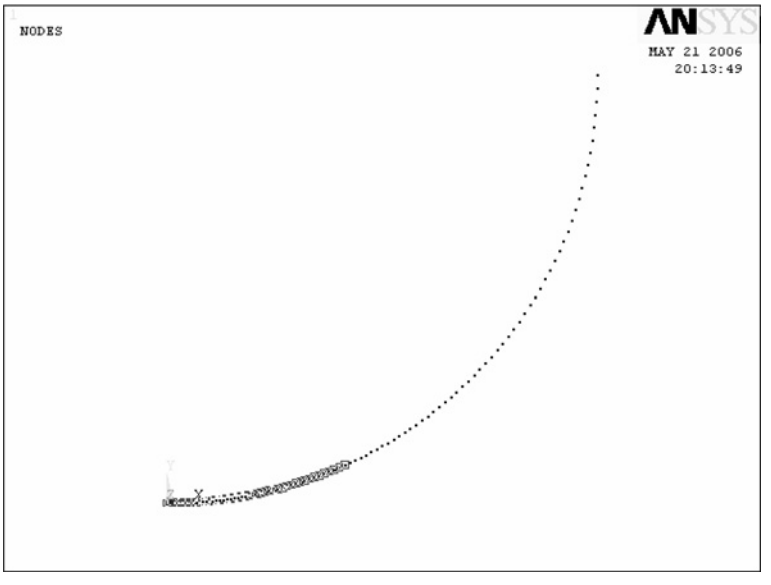


Figure 3.106 Selection of nodes on the portion of the bottom surface of the cylinder.

the mouse as shown in Figure 3.106. Click [B] **OK** button to select the nodes on the bottom surface of the cylinder.

Next, let us define contact elements to be attached to the lower surface of the cylinder by the commands as follows:

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Elem Attributes**

(1) The **Element Attributes** window opens as shown in Figure 3.107.

(2) Select [A] **3 CONTA172** in the **TYPE** box and click [B] **OK** button in the **Element Attributes** window.

Then, perform the following commands to attach the contact elements to the lower surface of the cylinder.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Surf/Contact → Surf to Surf**

(1) The **Mesh Free Surfaces** window opens as shown in Figure 3.108.

(2) Select [A] **Bottom surface** in the **Tlab** box and click [B] **OK** button in the window.

(3) Another **Mesh Free Surfaces** like the **Reselect notes** windows as shown in Figure 3.105 window opens. The upward arrow appears in the **ANSYS Graphics** window. Select the **Box** radio button and move the upward arrow to the left end of the array of nodes. Enclose all the nodes as shown in Figure 3.16 with a rectangle formed

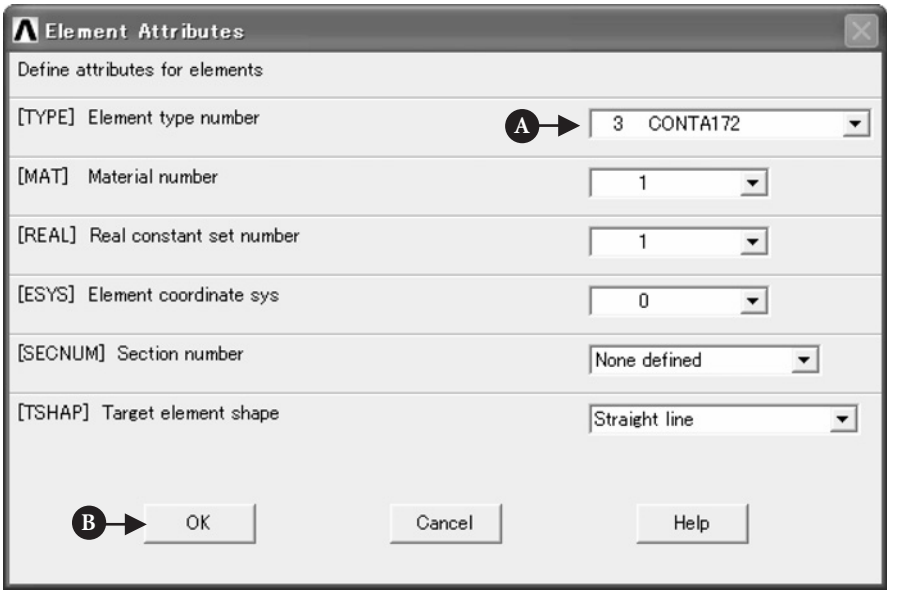


Figure 3.107 "Element Attributes" window.

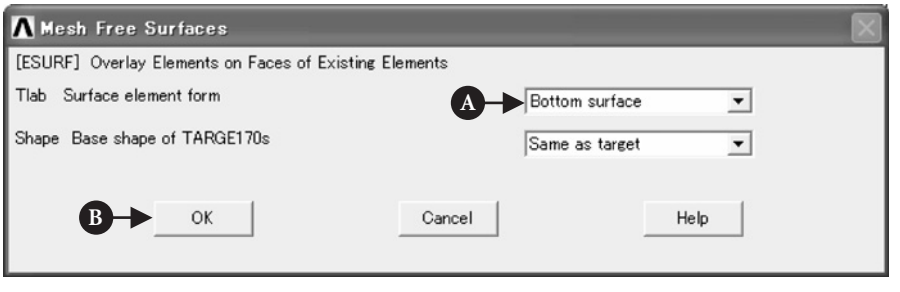


Figure 3.108 "Mesh Free Surfaces" window.

by dragging mouse to the right end of the node array. Click the **OK** button to select the nodes to which **CONTA172** element are to be attached.

(4) **CONTA172** elements are created on the lower surface of the cylinder as shown in Figure 3.109.

Before proceeding to the next step, **Select → Everything** command in the **ANSYS Utility Menu** must be carried out in order to select nodes other than those selected by the previous procedures.

Repeat operations similar to the above in order to create **TARGET169** elements on the top surface of the flat plate: Namely, perform the **Select → Entities …** commands to select only nodes on the portion of the top surface of the flat plate in the vicinity of

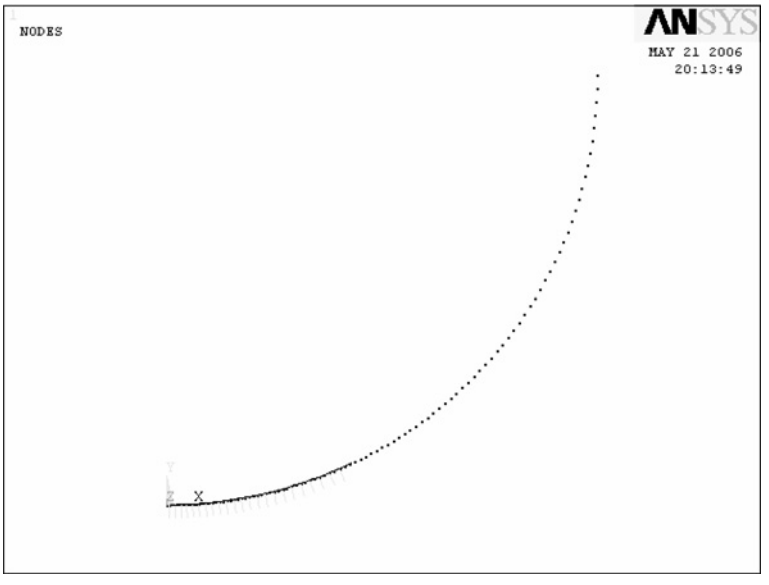


Figure 3.109 "CONTA172" elements created on the lower surface of the cylinder.

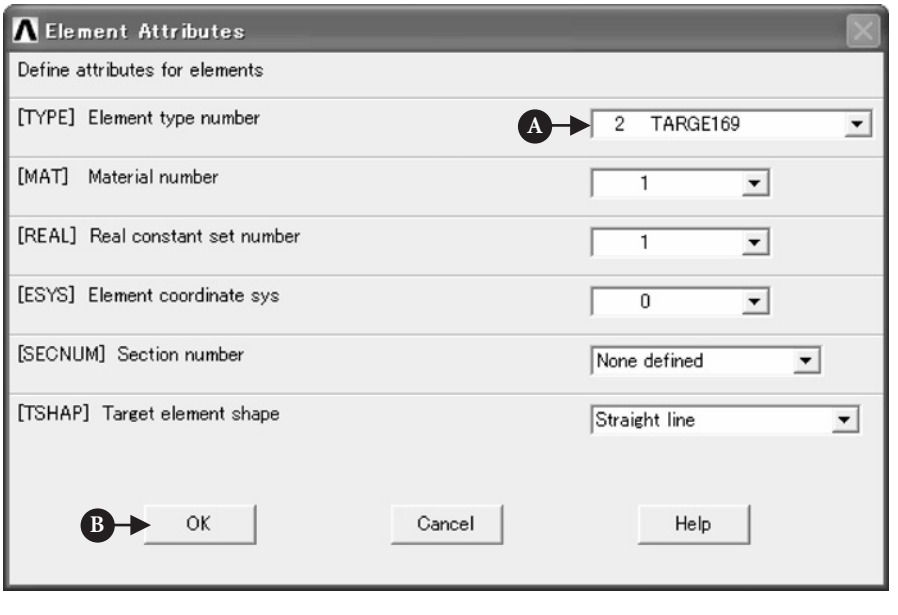


Figure 3.110 "Element Attributes" window.

the contact point. Then, define the target elements to be attached to the top surface of the flat plate by the commands as follows:

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Elem Attributes**

(1) The **Element Attributes** window opens as shown in Figure 3.110.
(2) Select [A] **2 TARGET169** in the **TYPE** box and click [B] **OK** button in the **Element Attributes** window.

Then, perform the following commands to attach the target elements to the top surface of the flat plate:

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Elements → Surf/Contact → Surf to Surf**

(3) The **Mesh Free Surfaces** window opens as shown in Figure 3.111.

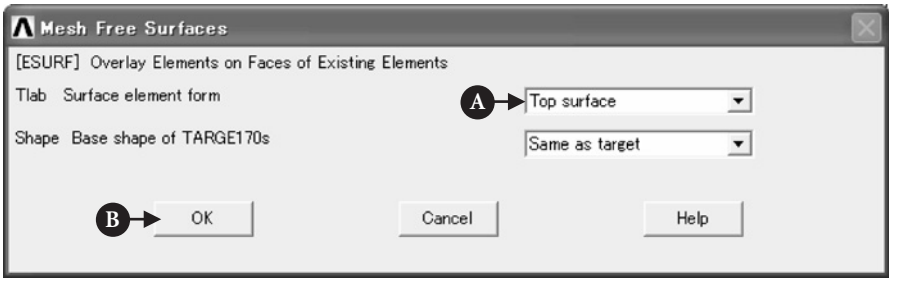


Figure 3.111 "Mesh Free Surfaces" window.

(4) Select [A] **Top surface** in the **Tlab** box and click [B] **OK** button in the window.
(5) **TARGET169** elements are created on the top surface of the flat plate as shown in Figure 3.112.
(6) Perform the **Select → Everything** command in the **ANSYS Utility Menu** in order to impose boundary conditions on nodes other than those selected by the previous procedures.

3.5.3.4 INPUT OF BOUNDARY CONDITIONS

[1] Imposing constraint conditions on the left sides of the quarter cylinder and the half flat plate models

Due to the symmetry, the constraint conditions of the present models are **UX**-fixed condition on the left sides of the quarter cylinder and the half flat plate models, and **UY**-fixed condition on the bottom side of the half flat plate model. Apply these constraint conditions onto the corresponding lines by the following commands:

COMMAND **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

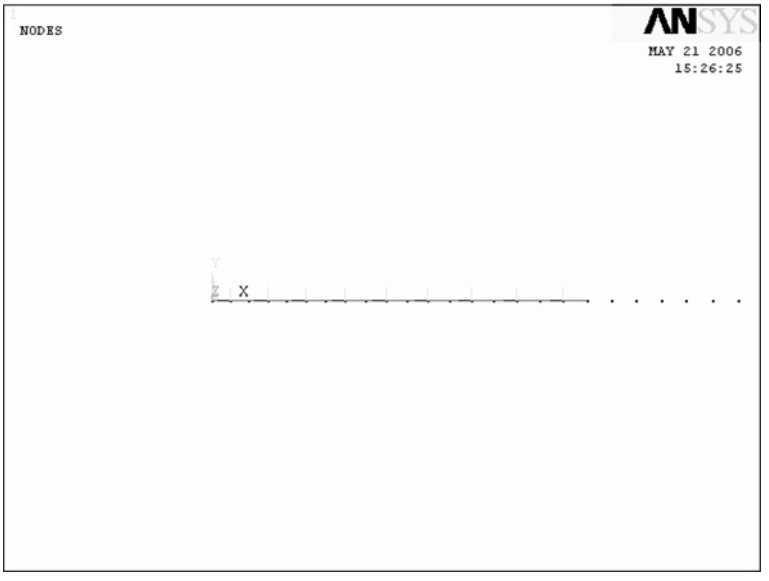


Figure 3.112 "TARGET169" elements created on the top surface of the flat plate.

(1) The **Apply U. ROT on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.
(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the left sides of the quarter cylinder and the half flat plate models and click the left button of the mouse on each model.
(3) Click the **OK** button in the **Apply U. ROT on Lines** window to display another **Apply U. ROT on Lines** window.
(4) Select **UX** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

Repeat the commands and operations (1) through (3) above for the bottom side of the flat plate model. Then, select **UY** in the **Lab2** box and click the **OK** button in the **Apply U. ROT on Lines** window.

[2] Imposing a uniform pressure on the upper side of the quarter cylinder model

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines**

(1) The **Apply PRES on Lines** window opens and the upward arrow appears when the mouse cursor is moved to the **ANSYS Graphics** window.

(2) Confirming that the **Pick** and **Single** buttons are selected, move the upward arrow onto the upper side of the quarter cylinder model and click the left button of the mouse.

(3) Another **Apply PRES on Lines** window opens. Select **Constant value** in the [**SFL**] **Apply PRES on lines as a** box and input **1** (MPa) in the **VALUE Load PRES value** box and leave a blank in the **Value** box.

(4) Click the **OK** button in the window to define a downward force of $P' = 1$ kN/mm applied to the elastic cylinder model.

Figure 3.113 illustrates the boundary conditions applied to the finite-element model of the present contact problem by the above operations.

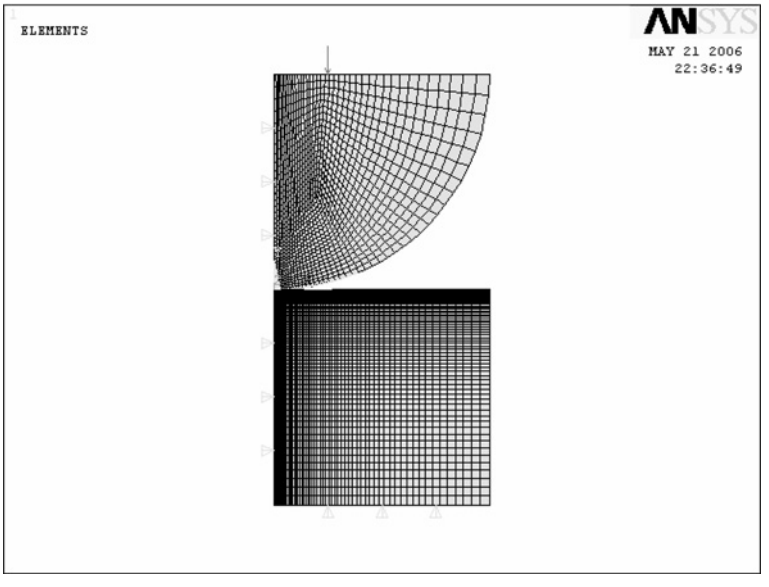


Figure 3.113 Boundary conditions applied to the finite-element model of the present contact problem.

3.5.3.5 SOLUTION PROCEDURES

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

(1) The **Solve Current Load Step** and **/STATUS Commands** windows appear.

(2) Click the **OK** button in the **Solve Current Load Step** window to begin the solution of the current load step.

(3) Select the **File** button in **/STATUS Commands** window to open the submenu and select the **Close** button to close the **/STATUS Commands** window.

(4) When the solution is completed, the **Note** window appears. Click the **Close** button to close the **Note** window.

3.5.3.6 CONTOUR PLOT OF STRESS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution**

(1) The **Contour Nodal Solution Data** window opens.

(2) Select **Stress** and ***Y*-Component of stress**.

(3) Click the **OK** button to display the contour of the y-component of stress in the present model of the contact problem in the **ANSYS Graphics** window as shown in Figure 3.114.

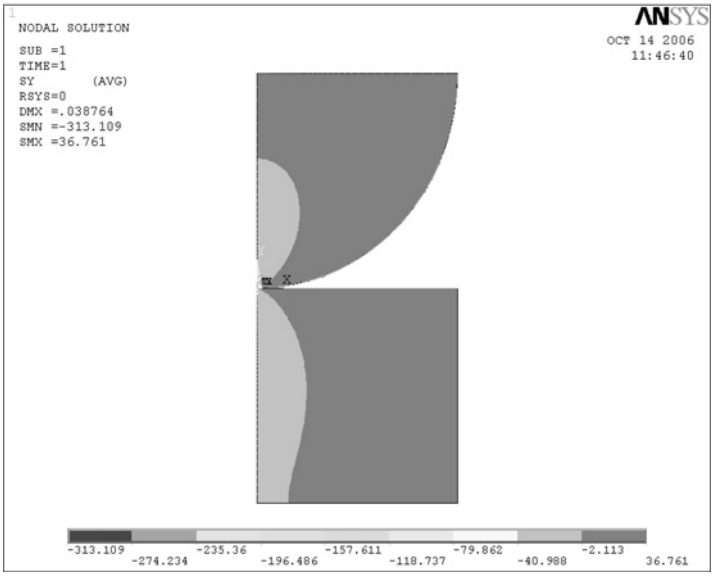


Figure 3.114 Contour of the y-component of stress in the present model of the contact problem.

Figure 3.115 is an enlarged contour of the y-component of stress around the contact point showing that very high compressive stress is induced in the vicinity of the contact point known as "Hertz contact stress."

3.5.4 DISCUSSION

Open circular symbols in Figure 3.116 show the plots of the y-component stress, or vertical stress along the upper surface of the flat plate obtained by the present finite-element calculation. The solid line in the figure indicates the theoretical curve $p(x)$ expressed by a parabola [10, 11], i.e.,

$$p(x) = k\sqrt{b^2 - x^2} = 115.4\sqrt{2.349^2 - x^2} \tag{3.12}$$

where x is the distance from the center of contact, $p(x)$ the contact stress at a point x, k the coefficient given by Equation (3.13) below, and b half the width of contact

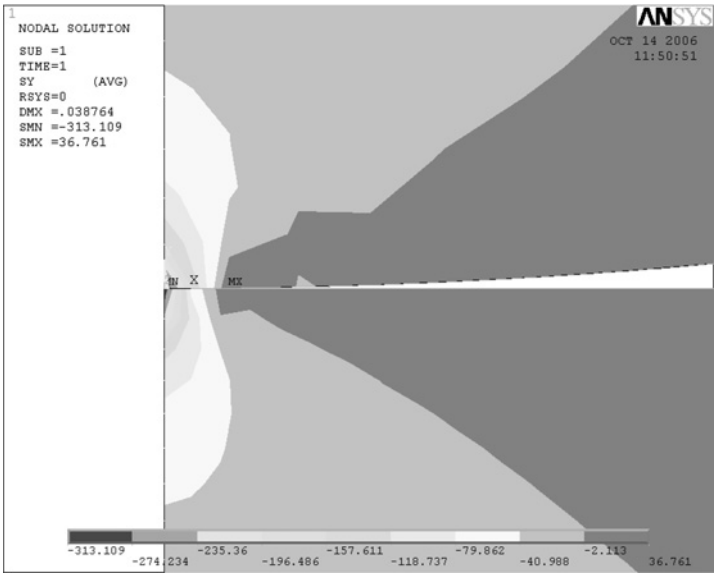


Figure 3.115 Enlarged contour of the y-component of stress around the contact point.

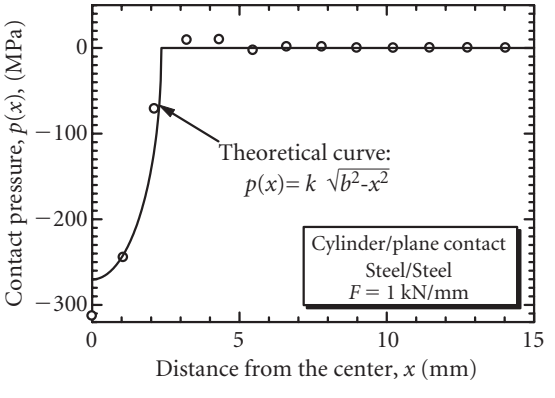


Figure 3.116 Plots of the *y*-component stress along the upper surface of the flat plate obtained by the present finite-element calculation.

given by Equation (3.14) below.

$$k = \frac{2P'}{\pi b^2} = 115.4\,\mathrm{N/mm^2} \tag{3.13}$$

$$b = 1.522\sqrt{\frac{P'R}{E}} = 2.348\,\mathrm{mm} \tag{3.14}$$

The maximum stress p_0 is obtained at the center of contact and expressed as,

$$p_0 = 0.418\sqrt{\frac{P'E}{R}} = 270.9\,\text{MPa} \tag{3.15}$$

The length of contact is as small as 4.7 mm, so that the maximum value of the contact stress is as high as 270.9 MPa. The number of nodes which fall within the contact region is only 3, although the total number of nodes is approximately 15,000 and meshes are finer in the vicinity of the contact point. The three plots of the present results agree reasonably well with the theoretical curve as shown in Figure 3.116.

3.5.5 Problems to solve

Problem 3.15

Calculate the contact stress distribution between a cylinder having a radius of curvature $R_1 = 500$ mm and another cylinder having $R_2 = 1000$ mm against which the smaller cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.15. The two cylinders are made of the same steel having Young's modulus = 210 GPa and Poisson's ratio $\nu = 0.3$. The theoretical expression for the contact stress distribution $p(x)$ is given by Equation (P3.15a) with the parameters given by Equations (P3.15b)–(P3.15d).

$$p(x) = k\sqrt{b^2 - x^2} \tag{P3.15a}$$

$$k = \frac{2P'}{\pi b^2} \tag{P3.15b}$$

$$b = 1.522\sqrt{\frac{P'}{E} \cdot \frac{R_1 R_2}{R_1 + R_2}} \tag{P3.15c}$$

$$p_0 = 0.418\sqrt{\frac{P'E(R_1 + R_2)}{R_1 R_2}} \tag{P3.15d}$$

Problem 3.16

Calculate the contact stress distribution between a cylinder having a radius of curvature $R = 500$ mm and a flat plate of 1000 mm width by 500 mm height against which the cylinder is pressed by a force of $P' = 1$ kN/mm as shown in Figure P3.16. The cylinder is made of fine ceramics having Young's modulus $E_1 = 320$ GPa and Poisson's ratio $\nu_1 = 0.3$, and the flat plate stainless steel having $E_2 = 192$ GPa and $\nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.16a) with the parameters expressed by Equations (P3.16b) to (P3.16d) on page 140. Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.16a)–(P3.16d).

$$p(x) = k\sqrt{b^2 - x^2} \tag{P3.16a}$$

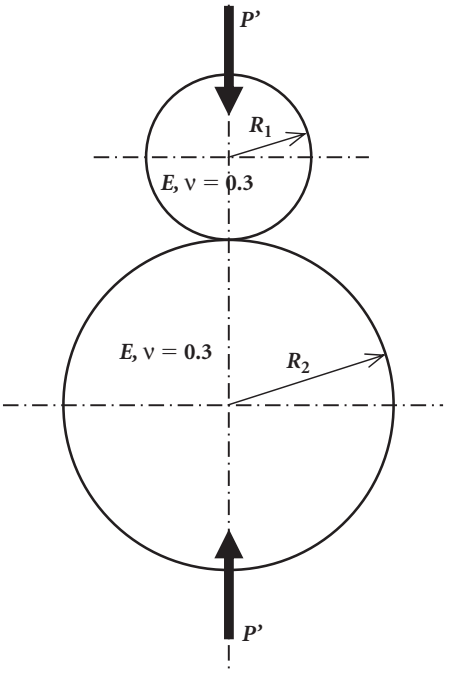


Figure P3.15 A cylinder having a radius of curvature $R_1 = 500$ mm pressed against another cylinder having $R_2 = 1000$ mm by a force $P' = 1$ kN/mm.

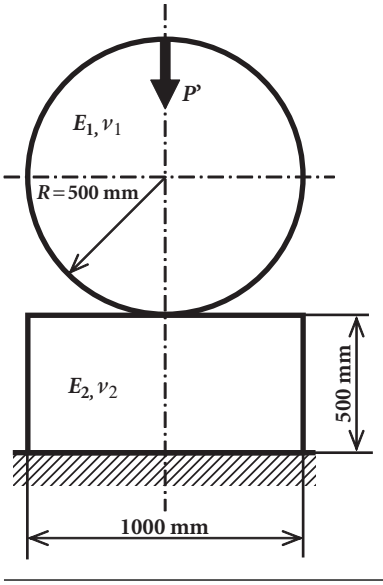


Figure P3.16 A cylinder having a radius of curvature $R = 500$ mm pressed against a flat plate.

$$k = \frac{2P'}{\pi b^2} \tag{P3.16b}$$

$$b = \frac{2}{\sqrt{\pi}}\sqrt{P'R_1\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)} \tag{P3.16c}$$

$$p_0 = \frac{1}{\sqrt{\pi}}\sqrt{\frac{P'}{R_1} \cdot \frac{1}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)}} \tag{P3.16d}$$

PROBLEM 3.17

Calculate the contact stress distribution between a cylinder having a radius of curvature $R_1 = 500$ mm and another cylinder having $R_2 = 1000$ mm against which the smaller cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.15. The upper cylinder is made of fine ceramics having Young's modulus $E_1 = 320$ GPa and Poisson's ratio $\nu_1 = 0.3$, and the lower cylinder stainless steel having $E_2 = 192$ GPa and $\nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.17a) with the parameters expressed by Equations (P3.17b) to (P3.17d) below. Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.17a)–(P3.17d).

$$p(x) = k\sqrt{b^2 - x^2} \tag{P3.17a}$$

$$k = \frac{2P'}{\pi b^2} \tag{P3.17b}$$

$$b = \frac{2}{\sqrt{\pi}}\sqrt{\frac{P'R_1R_2}{R_1 + R_2}\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)} \tag{P3.17c}$$

$$p_0 = \frac{1}{\sqrt{\pi}}\sqrt{\frac{P'(R_1 + R_2)}{R_1R_2} \cdot \frac{1}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)}} \tag{P3.17d}$$

PROBLEM 3.18

Calculate the contact stress distribution between a cylinder of fine ceramics having a radius of curvature $R_1 = 500$ mm and a cylindrical seat of stainless steel having $R_2 = 1000$ mm against which the cylinder is pressed by a force $P' = 1$ kN/mm as shown in Figure P3.18. Young's module of fine ceramics and stainless steel are $E_1 = 320$ GPa and $E_2 = 192$ GPa, respectively and Poisson's ratio of each material is the same, i.e., $\nu_1 = \nu_2 = 0.3$. The theoretical curve of the contact stress $p(x)$ is given by Equation (P3.18a) with the parameters expressed by Equations (P3.18b) to (P3.18d) below.

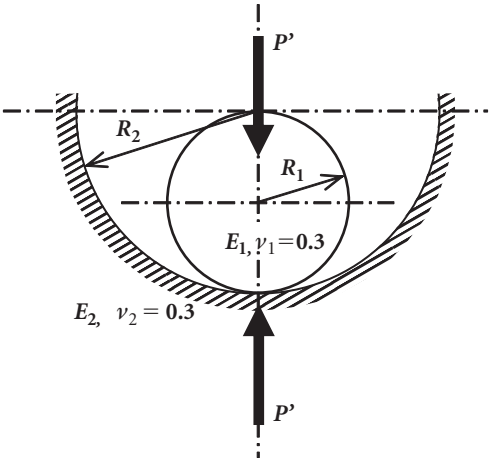


Figure P3.18 A cylinder having a radius of curvature $R_1 = 500$ mm pressed against a cylindrical seat having $R_2 = 1000$ mm by a force $P' = 1$ kN/mm.

Compare the result obtained by the FEM calculation with the theoretical distribution given by Equations (P3.18a)–(P3.18d).

$$p(x) = k\sqrt{b^2 - x^2} \tag{P3.18a}$$

$$k = \frac{2P'}{\pi b^2} \tag{P3.18b}$$

$$b = \frac{2}{\sqrt{\pi}}\sqrt{\frac{P'R_1R_2}{R_2 - R_1}\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)} \tag{P3.18c}$$

$$p_0 = \frac{1}{\sqrt{\pi}}\sqrt{\frac{P'(R_2 - R_1)}{R_1R_2} \cdot \frac{1}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)}} \tag{P3.18d}$$

Note that Equations (P3.18a) to (P3.18d) cover Equations (P3.15a)–(P3.15d), (P3.16a)–(P3.16d) and (P3.17a)–(P3.17d).

References

1. R. Yuuki and H. Kisu, *Elastic Analysis with the Boundary Element Method*, Baifukan Co., Ltd., Tokyo (1987) (in Japanese).
2. M. Isida, *Elastic Analysis of Cracks and their Stress Intensity Factors*, Baifukan Co., Ltd., Tokyo (1976) (in Japanese).

3. M. Isida, *Engineering Fracture Mechanics*, Vol. 5, No. 3, pp. 647–665 (1973).
4. R. E. Feddersen, Discussion, *ASTM STP 410*, pp. 77–79 (1967).
5. W. T. Koiter, *Delft Technological University, Department of Mechanical Engineering*, Report No. 314 (1965).
6. H. Tada, *Engineering Fracture Mechanics*, Vol. 3, No. 2, pp. 345–347 (1971).
7. H. Okamura, *Introduction to the Linear Elastic Fracture Mechanics*, Baifukan Co., Ltd., Tokyo (1976) (in Japanese).
8. H. L. Ewalds and R. J. H. Wanhill, *Fracture Mechanics*, Edward Arnold, Ltd., London (1984).
9. A. Saxena, *Nonlinear Fracture Mechanics for Engineers*, CRC Press, Florida (1997).
10. I. Nakahara, *Strength of Materials*, Vol. 2, Yokendo Co., Ltd., Tokyo (1966) (in Japanese).
11. S. P. Timoshenko and J. N. Goodier, *Theory of Elasticity* (3rd edn), McGraw-Hill Kogakusha, Ltd., Tokyo (1971).

Chapter 4

Mode Analysis

Chapter outline

4.1	Introduction	143
4.2	Mode analysis of a straight bar	144
4.3	Mode analysis of a suspension for hard-disc drive	163
4.4	Mode analysis of a one-axis precision moving table using elastic hinges	188

4.1 Introduction

When a steel bar is hit by a hammer, a clear sound can be heard because the steel bar vibrates at its resonant frequency. If the bar is oscillated at this resonant frequency, it will be found that the vibration amplitude of the bar becomes very large. Therefore, when a machine is designed, it is important to know the resonant frequency of the machine. The analysis to obtain the resonant frequency and the vibration mode of an elastic body is called "mode analysis."

It is said that there are two methods for mode analysis. One is the theoretical analysis and the other is the finite-element method (FEM). Theoretical analysis is usually used for a simple shape of an elastic body, such as a flat plate and a straight bar, but theoretical analysis cannot give us the vibration mode for the complex shape of an elastic body. FEM analysis can obtain the vibration mode for it.

In this chapter, three examples are presented, which use beam elements, shell elements, and solid elements, respectively, as shown in Figure 4.1.

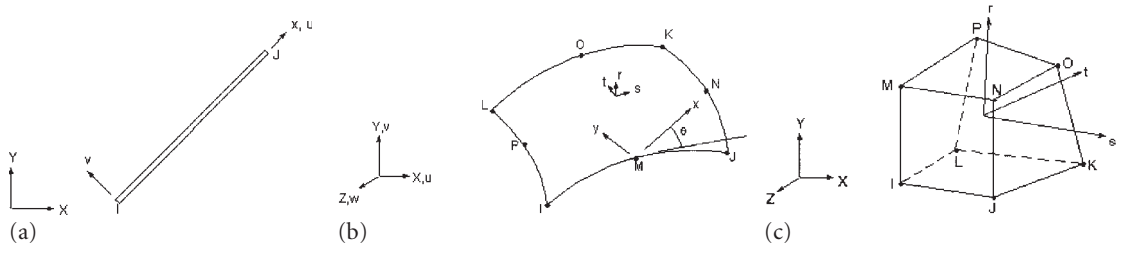


Figure 4.1 FEM element types: (a) beam element; (b) shell element; and (c) solid element.

4.2 Mode analysis of a straight bar

4.2.1 Problem description

Obtain the lowest three vibration modes and resonant frequencies in the y direction of the straight steel bar shown in Figure 4.2.

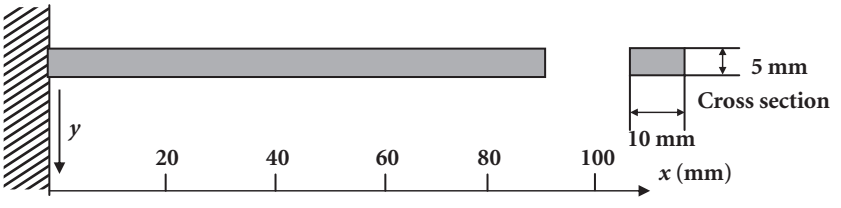


Figure 4.2 Cantilever beam for mode analysis.

Thickness of the bar is 0.005 m, width is 0.01 m, and the length is 0.09 m. Material of the bar is steel with Young's modulus, $E = 206$ GPa, and Poisson's ratio $\nu = 0.3$. Density $\rho = 7.8 \times 10^3$ kg/m^3.

Boundary condition: All freedoms are constrained at the left end.

4.2.2 Analytical solution

Before mode analysis is attempted using ANSYS program, an analytical solution for resonant frequencies will be obtained to confirm the validity of ANSYS solution. The analytical solution of resonant frequencies for a cantilever beam in y direction is given by:

$$f_i = \frac{\lambda_i^2}{2\pi L^2}\sqrt{\frac{EI}{M}} \quad (i = 1, 2, 3, \dots) \tag{4.1}$$

where length of the cantilever beam, $L = 0.09$ m, cross-section area of the cantilever beam, $A = 5 \times 10^{-5}$ m^2, and Young's modulus, $E = 206$ GPa.

The area moment of inertia of the cross-section of the beam is:

$$I = bt^3/12 = (0.01 \times 0.005^3)/12 = 1.042 \times 10^{-10}\,\mathrm{m}^4$$

Mass per unit width $M = \rho AL/L = \rho A = 7.8 \times 10^3\,\mathrm{kg/m}^3 \times 5 \times 10^{-5}\,\mathrm{m}^2 = 0.39\,\mathrm{kg/m}$

$$\lambda_1 = 1.875 \quad \lambda_2 = 4.694 \quad \lambda_3 = 7.855.$$

For that set of data the following solutions are obtained: $f_1 = 512.5$ Hz, $f_2 = 3212$ Hz, and $f_3 = 8994$ Hz.

Figure 4.3 shows the vibration modes and the positions of nodes obtained by Equation (4.1).

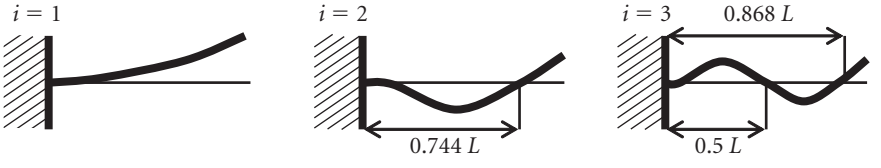


Figure 4.3 Analytical vibration mode and the node position.

4.2.3 MODEL FOR FINITE-ELEMENT ANALYSIS

4.2.3.1 ELEMENT TYPE SELECTION

In FEM analysis, it is very important to select a proper element type which influences the accuracy of solution, working time for model construction, and CPU time. In this example, the two-dimensional elastic beam, as shown in Figure 4.4, is selected for the following reasons:

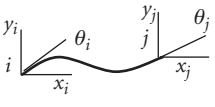


Figure 4.4 Two-dimensional beam element.

(a) Vibration mode is constrained in the two-dimensional plane.

(b) Number of elements can be reduced; the time for model construction and CPU time are both shortened.

Two-dimensional elastic beam has three degrees of freedom at each node (i, j), which are translatory deformations in the x and y directions and rotational deformation around the z-axis. This beam can be subjected to extension or compression bending due to its length and the magnitude of the area moment of inertia of its cross section.

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types**, as shown in Figure 4.5, is opened.

(1) Click [A] **Add**. Then the window **Library of Element Types** as shown in Figure 4.6 opens.

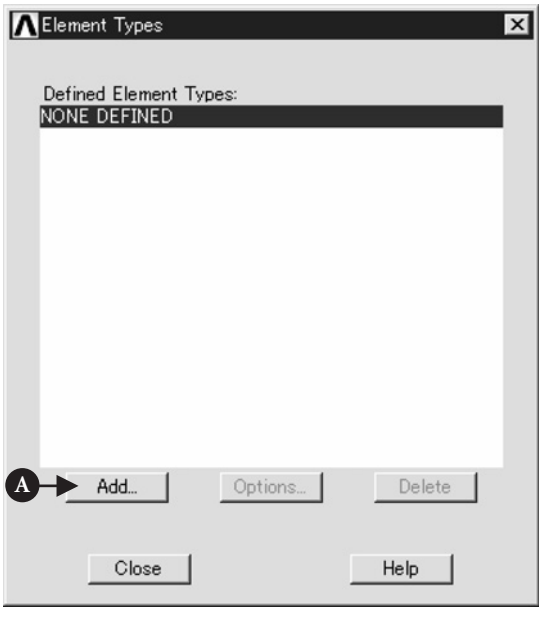


Figure 4.5 Window of **Element Types**.

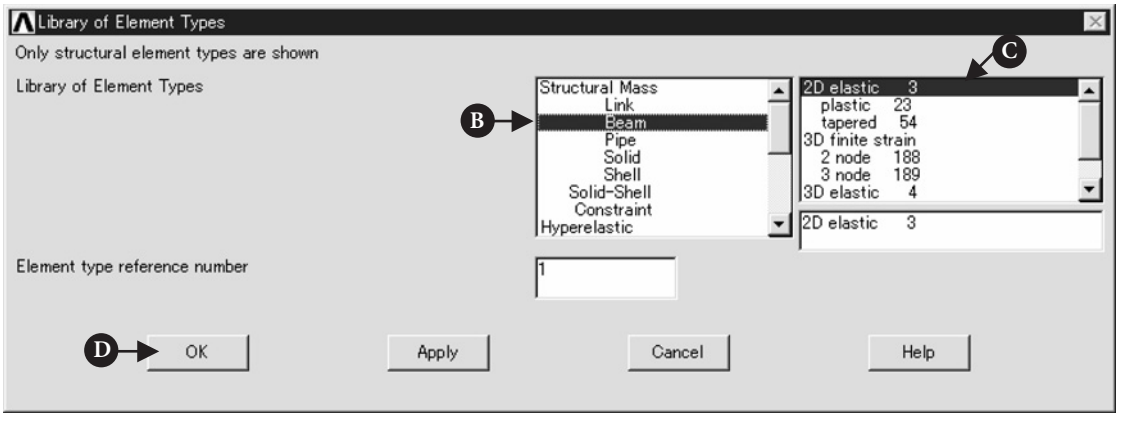


Figure 4.6 Window of **Library of Element Types**.

(2) Select [B] **Beam** in the table **Library of Element Types** and, then, select [C] **2D elastic 3**.

(3) **Element type reference number** is set to **1** and click D **OK** button. Then the window **Library of Element Types** is closed.

(4) Click [E] **Close** button in the window of Figure 4.7.

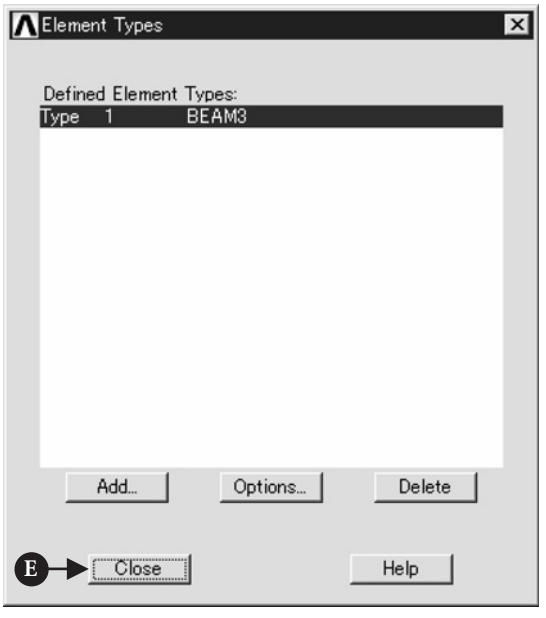


Figure 4.7 Window of **Library of Element Types**.

4.2.3.2 REAL CONSTANTS FOR BEAM ELEMENT

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/Delete → Add**

(1) The window **Real Constants** opens. Click [A] **add** button, and the window **Element Type for Real Constants** appears in which the name of element type selected is listed as shown in Figure 4.8.

(2) Click [B] **OK** button to input the values of real constants and the window **Real Constant** for BEAM3 is opened (Figure 4.9).

(3) Input the following values in Figure 4.10. [C] Cross-sectional area = 5e−5; [D] Area moment of inertia = 1.042e−10; [E] Total beam height = 0.005. After inputting these values, click [F] **OK** button to close the window.

(4) Click [G] **Close** button in the window **Real Constants** (Figure 4.11).

4.2.3.3 MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of the beam element.

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) Click the above buttons in the specified order and the window **Define Material Model Behavior** opens (Figure 4.12).

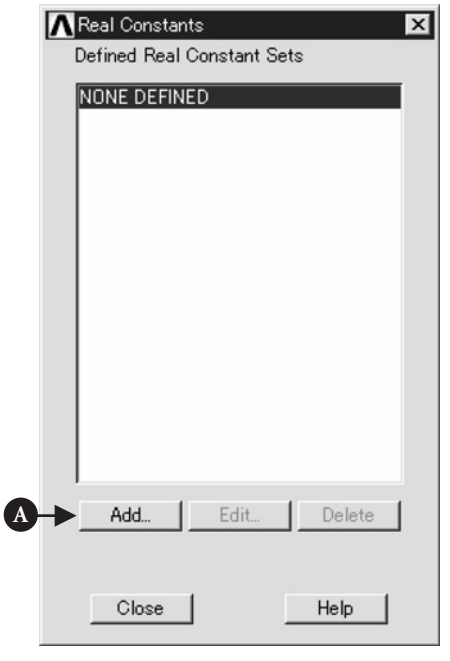


Figure 4.8 Window of **Real Constants**.

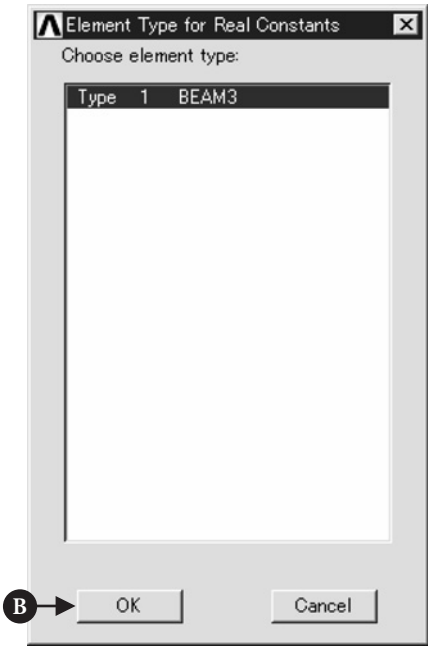


Figure 4.9 Window of **Element Type for Real Constants**.

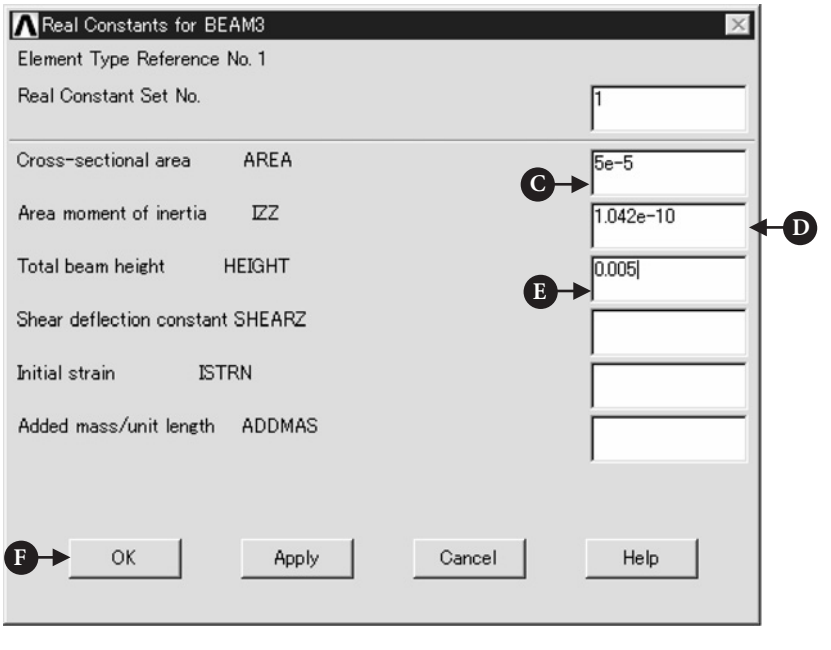


Figure 4.10 Window of **Real Constants for BEAM3**.

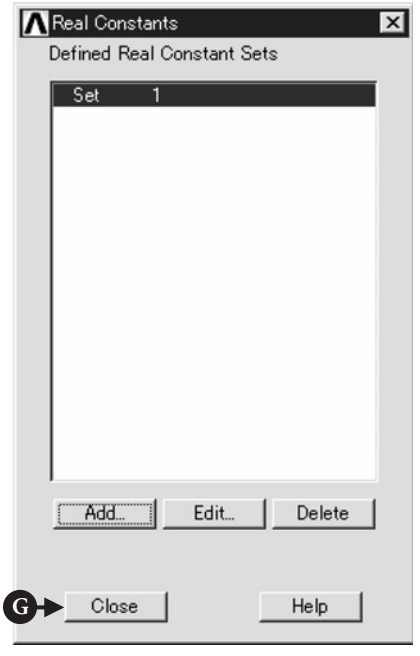


Figure 4.11 Window of **Real Constants**.

(2) Double click the following terms in the window.

[A] **Structural → Linear → Elastic → Isotropic.**

As a result the window **Linear Isotropic Properties for Material Number 1** opens (Figure 4.13).

(3) Input Young's modulus of **206e9** to [B] **EX** box and Poisson ratio of **0.3** to [C] **PRXY** box. Then click [D] **OK** button.

Next, define the value of density of material.

(1) Double click the term of **Density** in Figure 4.12 and the window **Density for Material Number 1** opens (Figure 4.14).

(2) Input the value of density, **7800** to [F] **DENS** box and click [G] **OK** button. Finally, close the window **Define Material Model Behavior** by clicking [H] **X** mark at the upper right corner (Figures 4.12 and 4.14).

4.2.3.4 CREATE KEYPOINTS

To draw a cantilever beam for analysis, the method of using keypoints is described.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create Keypoints → In Active CS**

The window **Create Keypoints in Active Coordinate System** opens.

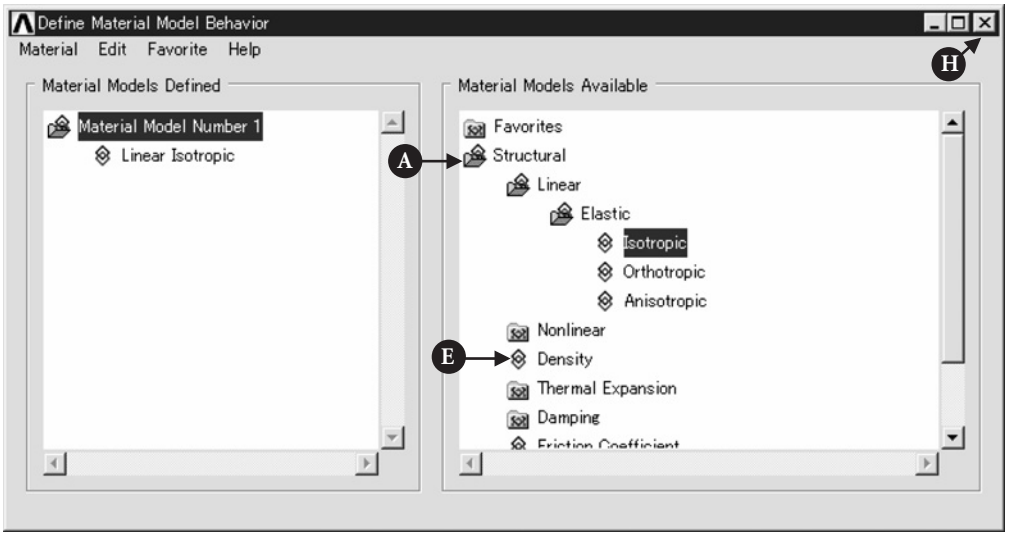


Figure 4.12 Window of **Define Material Model Behavior**.

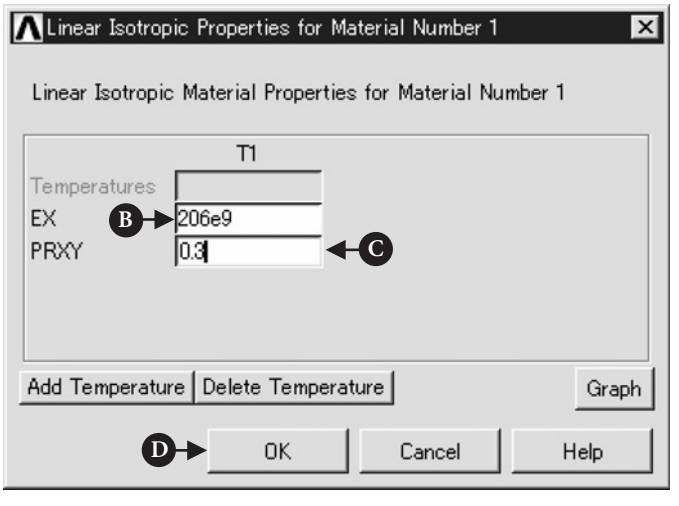


Figure 4.13 Window of **Linear Isotropic Properties for Material Number 1**.

(1) Input **1** to [A] **NPT KeyPoint number** box, **0,0,0** to [B] **X, Y, Z Location in active CS** box, and then click [C] **Apply** button. Do not click **OK** button at this stage. If you click **OK** button, the window will be closed. In this case, open the window **Create Keypoints in Active Coordinate System** and then proceed to step 2 (Figure 4.15).

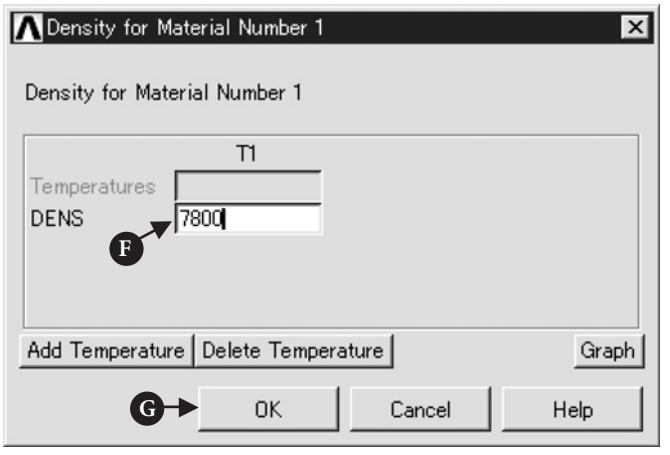


Figure 4.14 Window of **Density for Material Number 1**.

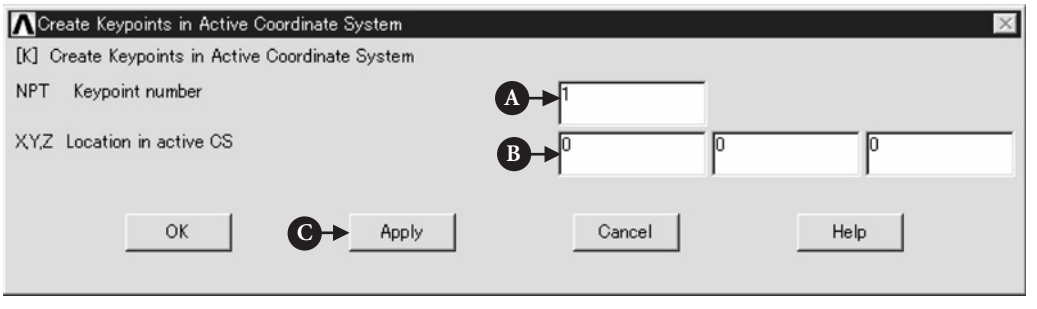


Figure 4.15 Window of **Create Keypoint in Active Coordinate System**.

(2) In the same window, input **2** to [D] **NPT Keypoint number** box, **0.09, 0,0** to [E] **X, Y, Z Location in active CS** box, and then click [F] **OK** button (Figure 4.16).

(3) After finishing the above steps, two keypoints appear in the window (Figure 4.17).

4.2.3.5 CREATE A LINE FOR BEAM ELEMENT

By implementing the following steps, a line between two keypoints is created.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Lines→ Straight Line**

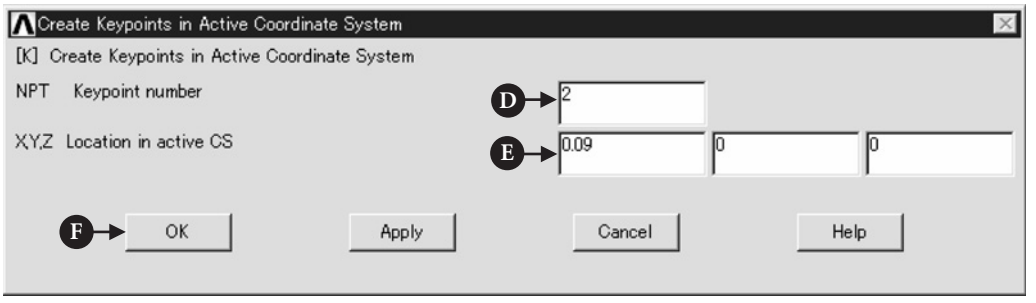


Figure 4.16 Window of **Create Keypoint in Active Coordinate System**.

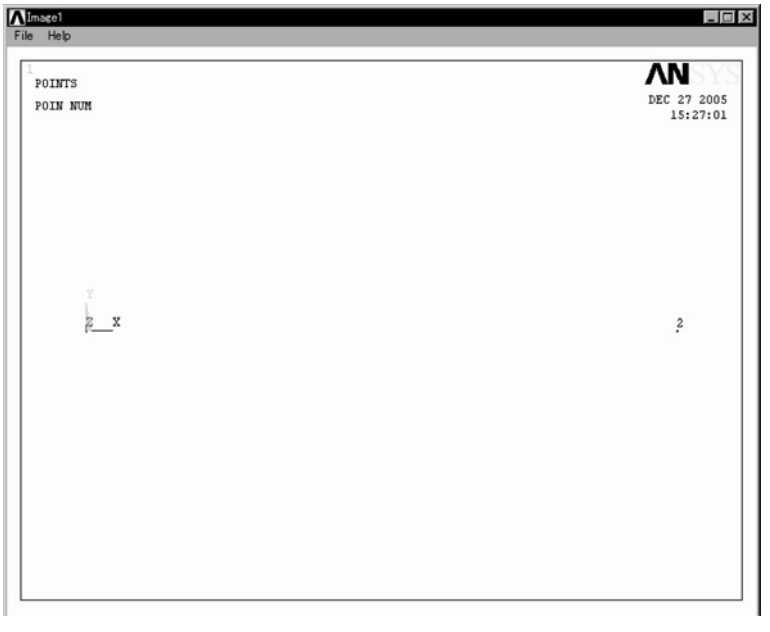


Figure 4.17 **ANSYS Graphics** window.

The window **Create Straight Line**, as shown in Figure 4.18, is opened.

(1) Pick the keypoints [A] **1** and [B] **2** (as shown in Figure 4.19) and click [C] **OK** button in the window **Create Straight Line** (as shown in Figure 4.18). A line is created.

4.2.3.6 CREATE MESH IN A LINE

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Lines → All Lines**

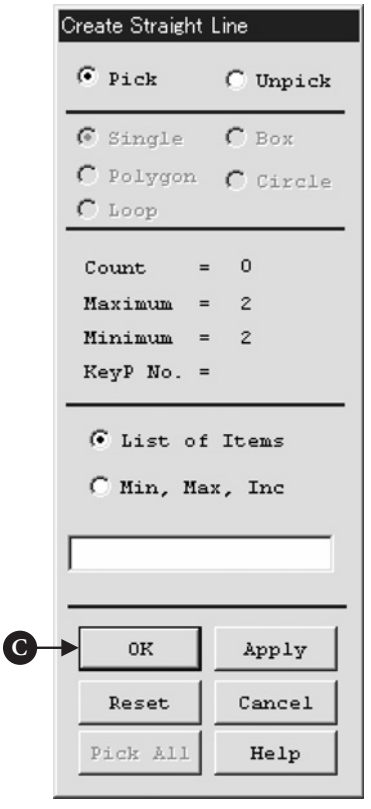


Figure 4.18 Window of **Create Straight Line**.

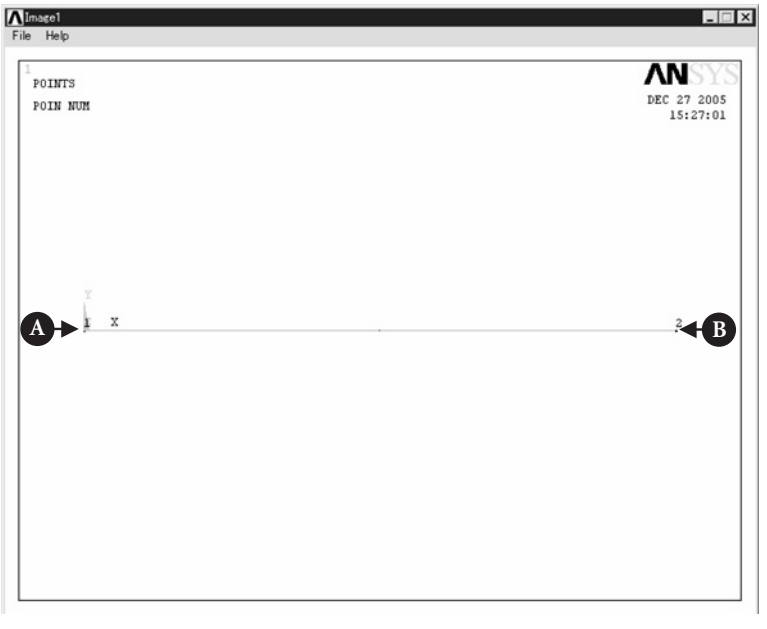


Figure 4.19 **ANSYS Graphics** window.

The window **Element Sizes on All Selected Lines**, as shown in Figure 4.20, is opened.

(1) Input [A] the number of **20** to **NDIV** box. This means that a line is divided into 20 elements.

(2) Click [B] **OK** button and close the window.

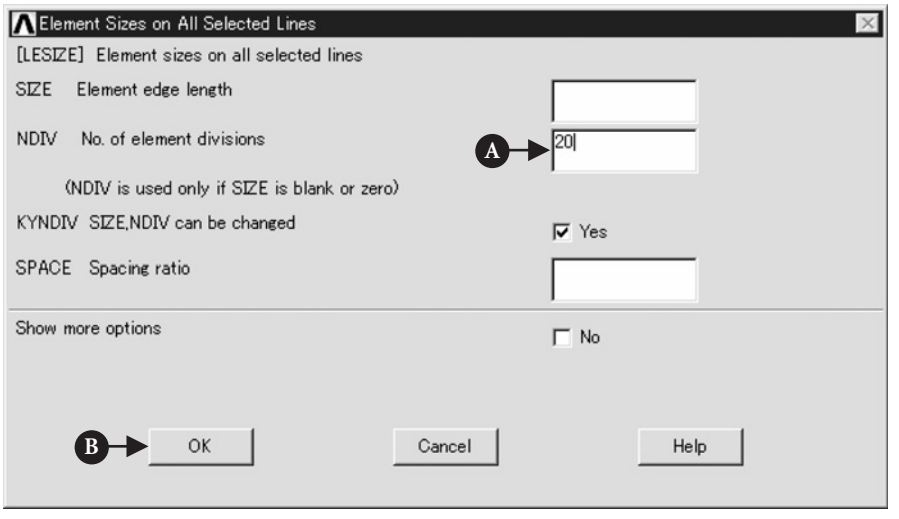


Figure 4.20 Window of **Element Sizes on All Selected Lines**.

From **ANSYS Graphics** window, the preview of the divided line is available, as shown in Figure 4.21, but the line is not really divided at this stage.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Lines**

The window **Mesh Lines**, as shown in Figure 4.22, opens.

(1) Click [C] the line shown in **ANSYS Graphics** window and, then, [D] **OK** button to finish dividing the line.

4.2.3.7 BOUNDARY CONDITIONS

The left end of nodes is fixed in order to constrain the left end of the cantilever beam.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes.**

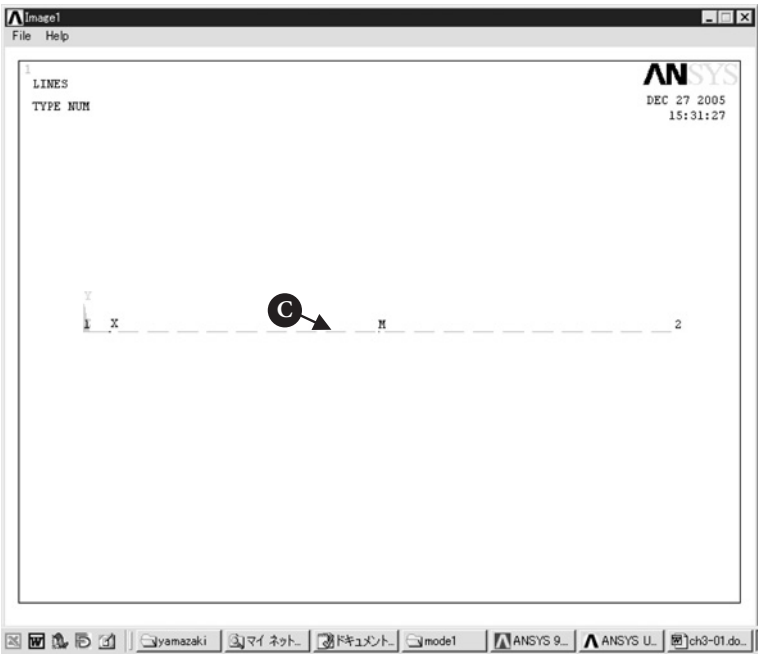


Figure 4.21 Preview of the divided line.

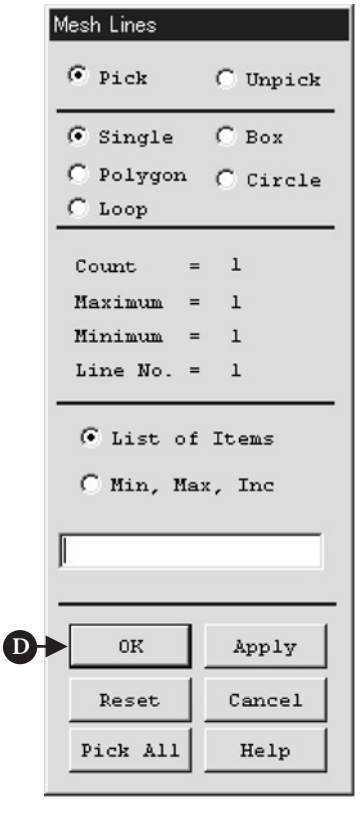


Figure 4.22 Window of **Mesh Lines**.

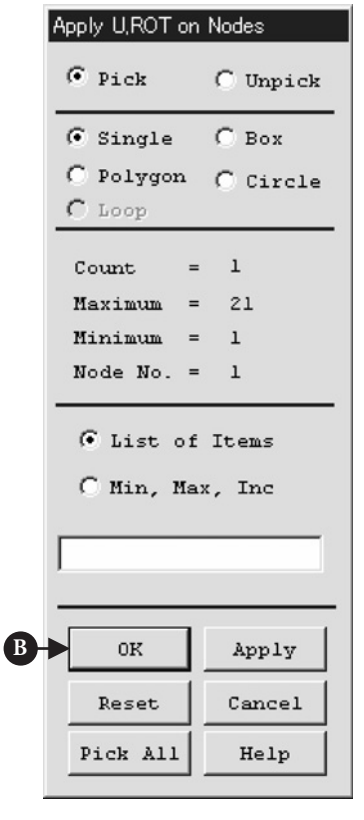


Figure 4.23 Window of **Apply U,ROT on Nodes**.

The window **Apply U,ROT on Nodes**, as shown in Figure 4.23, opens.

(1) Pick [A] the node at the left end in Figure 4.24 and click [B] **OK** button. Then the window **Apply U,ROT on Nodes** as shown in Figure 4.25 opens.
(2) In order to set the boundary condition, select [C] **All DOF** in the box **Lab2**. In the box **VALUE** [D] input **0** and, then, click [E] **OK** button. After these steps, **ANSYS Graphics** window is changed as shown in Figure 4.26.

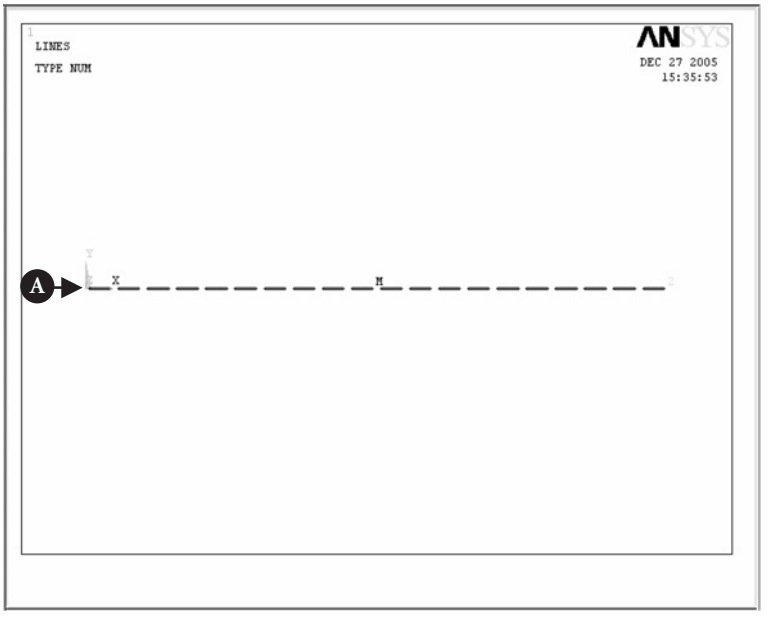


Figure 4.24 **ANSYS Graphics** window.

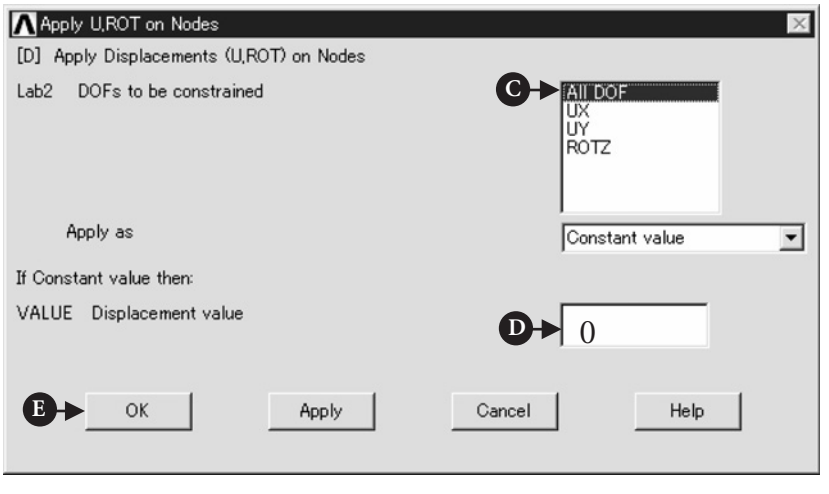


Figure 4.25 Window of **Apply U,ROT on Nodes**.

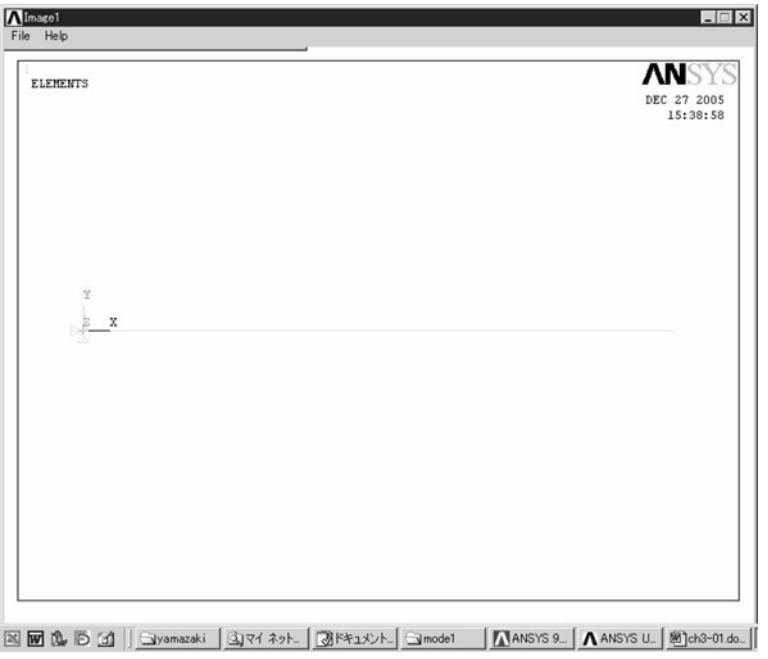


Figure 4.26 Window after the boundary condition was set.

4.2.4 EXECUTION OF THE ANALYSIS

4.2.4.1 DEFINITION OF THE TYPE OF ANALYSIS

The following steps are used to define the type of analysis.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → New Analysis**

The window **New Analysis**, as shown in Figure 4.27, opens.

(1) Check [A] **Modal** and, then, click [B] **OK** button.

In order to define the number of modes to extract, the following procedure is followed.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → Analysis Options**

The window **Modal Analysis**, as shown in Figure 4.28, opens.

(1) Check [C] **Subspace** of **MODOPT** and input [D] **3** in the box of **No. of modes to extract** and click [E] **OK** button.

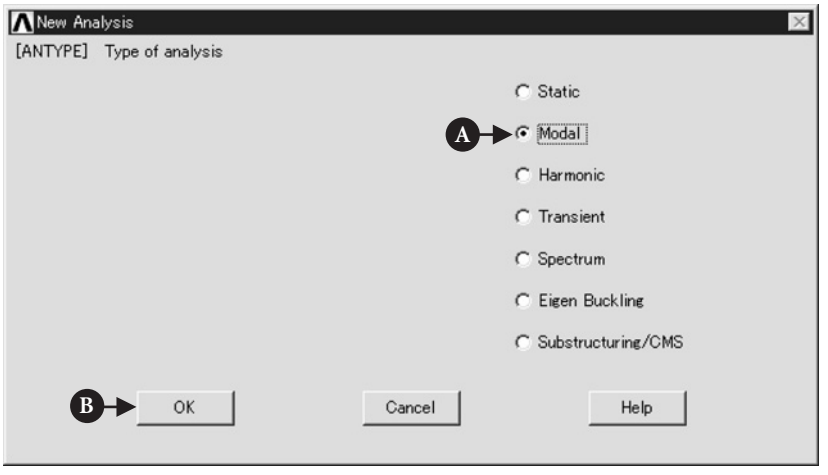


Figure 4.27 Window of **New Analysis**.

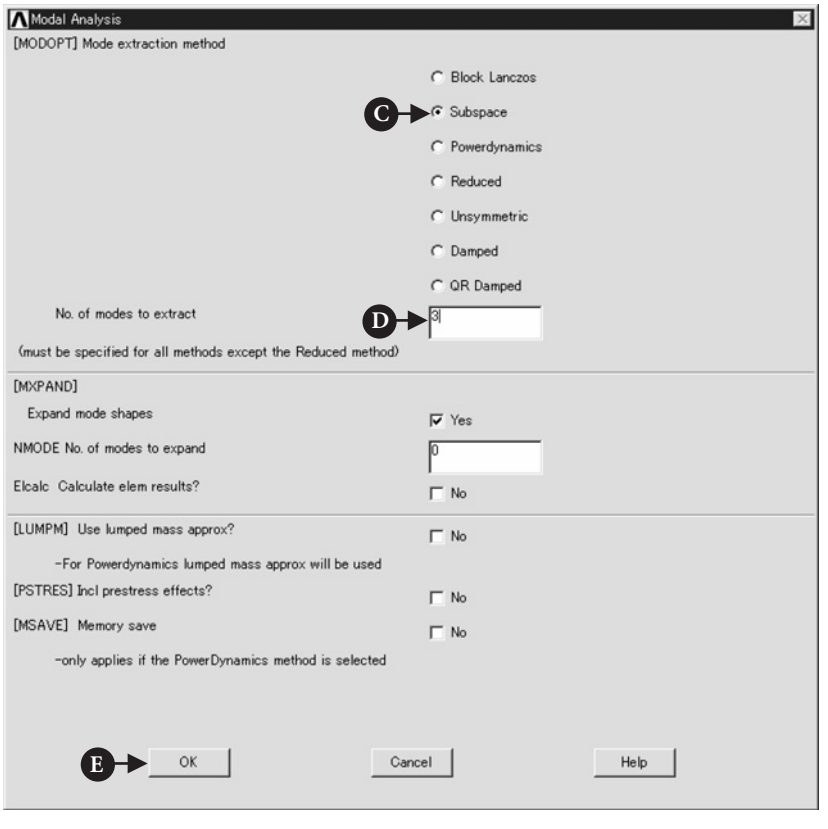


Figure 4.28 Window of **Modal Analysis**.

(2) Then, the window **Subspace Modal Analysis**, as shown in Figure 4.29, opens. Input [F] **10000** in the box of **FREQE** and click [G] **OK** button.

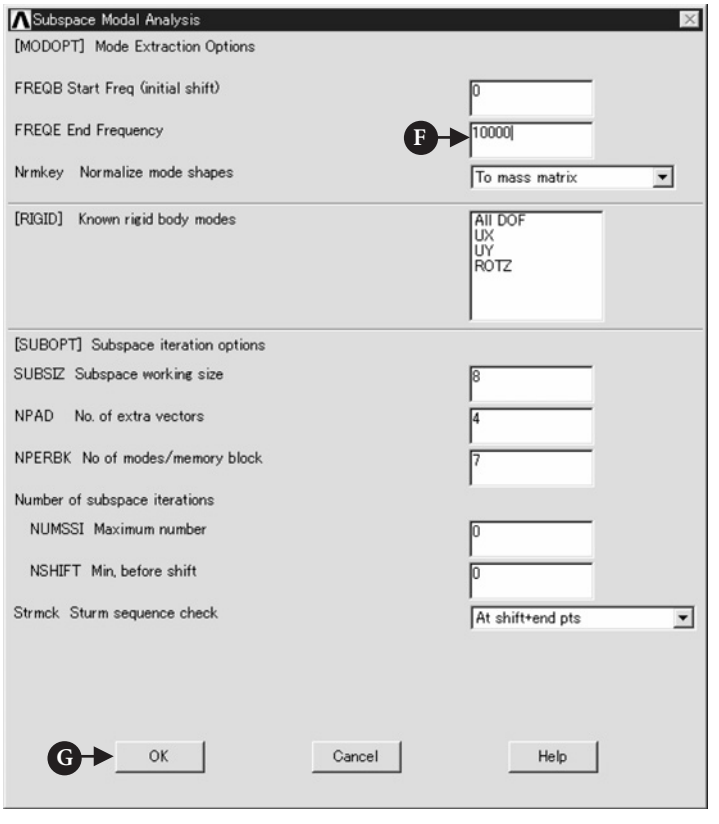


Figure 4.29 Window of **Subspace Modal Analysis**.

4.2.4.2 EXECUTE CALCULATION

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

The window **Solve Current Load Step**, as shown in Figure 4.30, opens.

(1) Click [A] **OK** button to initiate calculation. When the window **Note**, as shown in Figure 4.31 appears, the calculation is finished.

(2) Click [B] **Close** button and to close the window. The window **/STATUS Command**, as shown in Figure 4.32, also opens but this window can be closed by clicking [C] the mark **X** at the upper right-hand corner of the window.

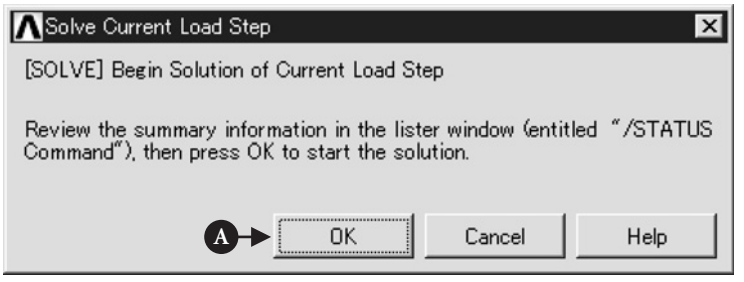


Figure 4.30 Window of **Solve Current Load Step**.

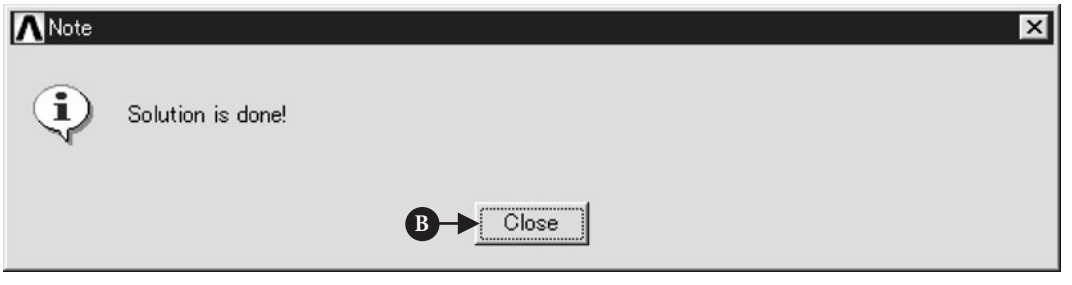


Figure 4.31 Window of **Note**.

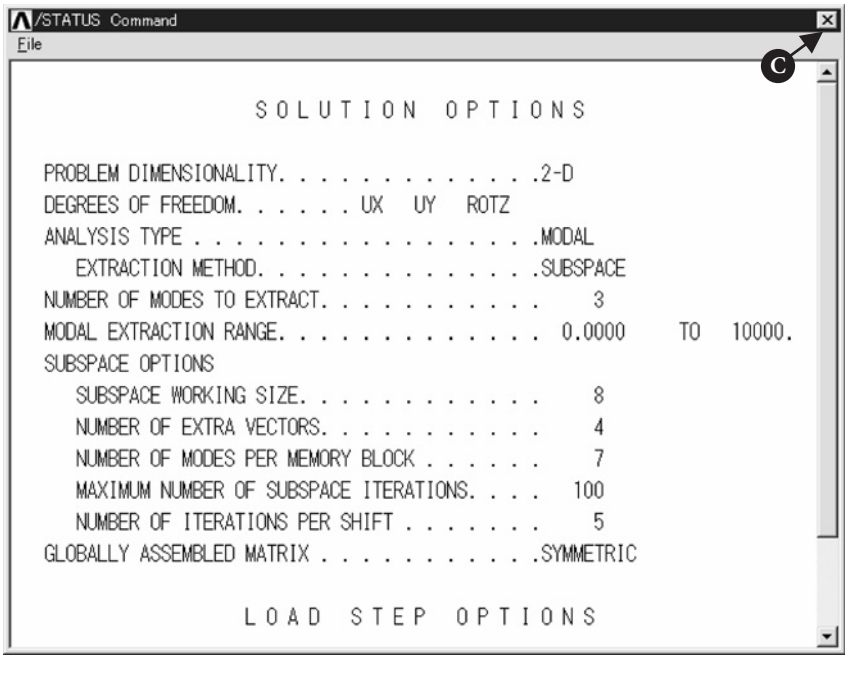


Figure 4.32 Window of **/STATUS Command**.

4.2.5 POSTPROCESSING

4.2.5.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → First Set**

4.2.5.2 PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape**

The window **Plot Deformed Shape**, as shown in Figure 4.33, opens.

(1) Select [A] **Def+Undeformed** and click [B] **OK**.

(2) Calculated result for the first mode of vibration is displayed in the **ANSYS Graphics** window as shown in Figure 4.34. The resonant frequency is shown as FRQE at the upper left-hand side on the window.

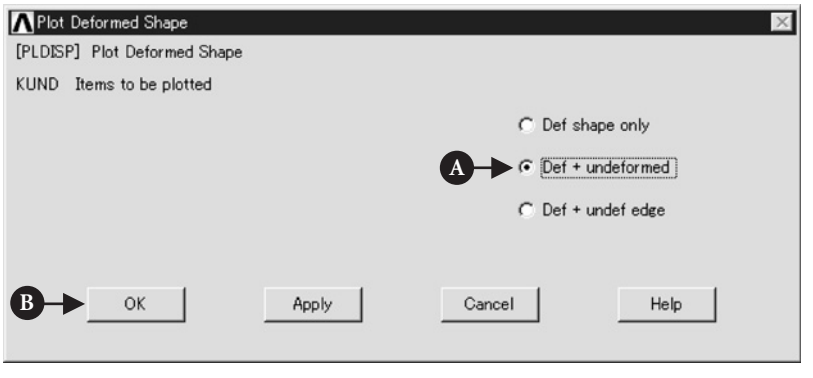


Figure 4.33 Window of **Plot Deformed Shape**.

4.2.5.3 READ THE CALCULATED RESULTS OF THE SECOND AND THIRD MODES OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → Next Set**

Follow the same steps outlined in Section 4.2.5.2 and calculated results for the second and third modes of vibration. Results are plotted in Figures 4.35 and 4.36. Resonant frequencies obtained by ANSYS show good agreement of those by analytical solution indicated in page 145 though they show slightly lower values.

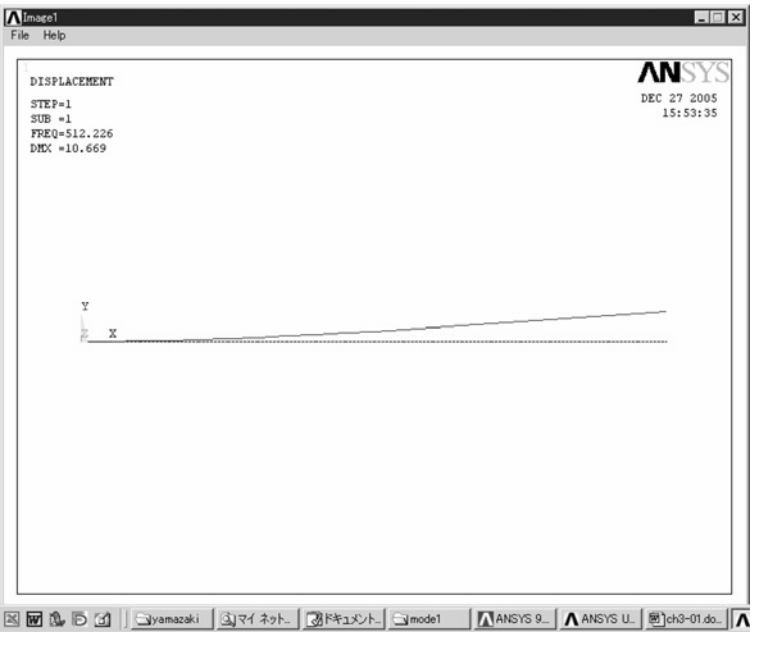


Figure 4.34 Window for the calculated result (the first mode of vibration).

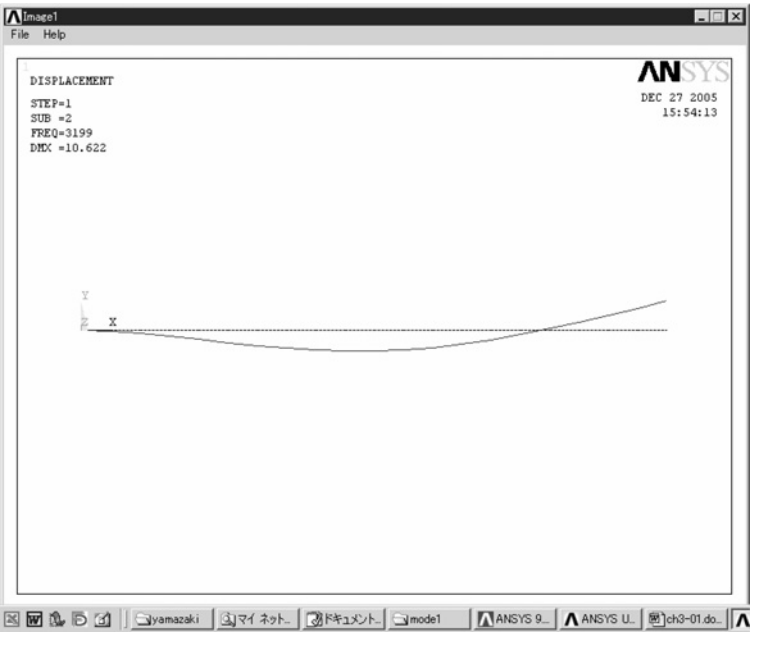


Figure 4.35 Window for the calculated result (the second mode of vibration).

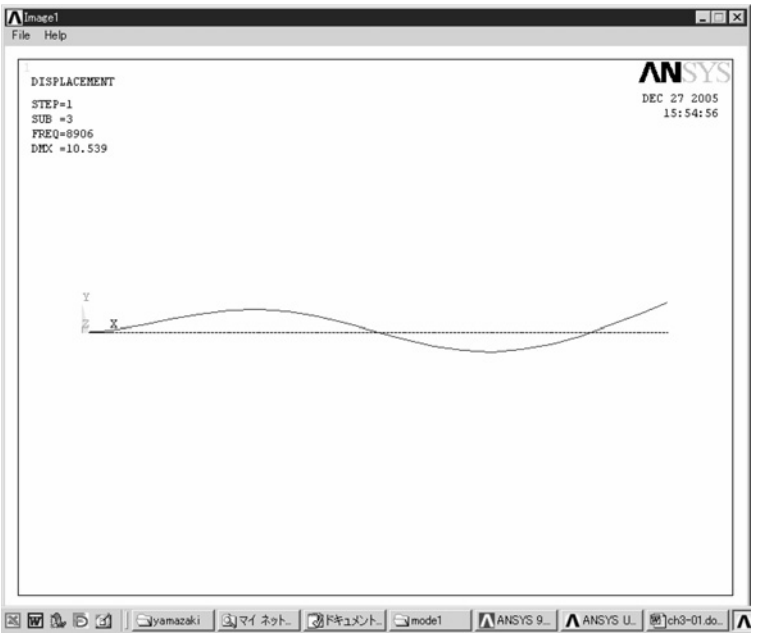


Figure 4.36 Window for the calculated result (the third mode of vibration).

4.3 Mode analysis of a suspension for hard-disc drive

4.3.1 Problem description

A suspension of hard-disc drive (HDD) has many resonant frequencies with various vibration modes and it is said that the vibration mode with large radial displacement causes the tracking error. So the suspension has to be operated with frequencies of less than this resonant frequency.

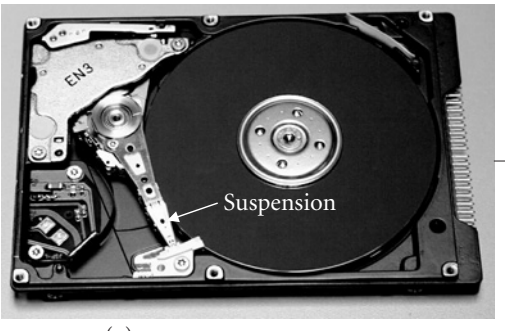
Obtain the resonant frequencies and determine the vibration mode with large radial displacement of the HDD suspension as shown in Figure 4.37:

- Material: Steel, thickness of suspension: 0.05×10^{-3}(m)
- Young's modulus, $E = 206$ GPa, Poisson's ratio $\nu = 0.3$
- Density $\rho = 7.8 \times 10^3$ kg/m^3
- Boundary condition: All freedoms are constrained at the edge of a hole formed in the suspension.

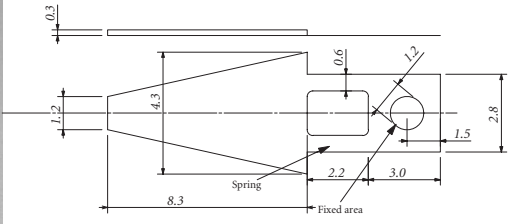
4.3.2 Create a model for analysis

4.3.2.1 Element type selection

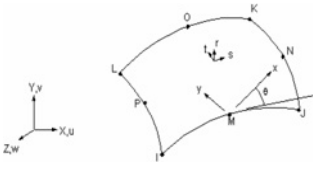
In this example, the two-dimensional elastic shell is selected for calculations as shown in Figure 4.37(c). Shell element is very suitable for analyzing the characteristics of thin material.



(a)



(b)



(c)

Figure 4.37 Mode analysis for an HDD suspension: (a) HDD (referenced from http://www-5.ibm.com/es/press/fotos/.informaticapersonal/i/hdd.jpg); (b) drawing of the analyzed suspension; and (c) shell element.

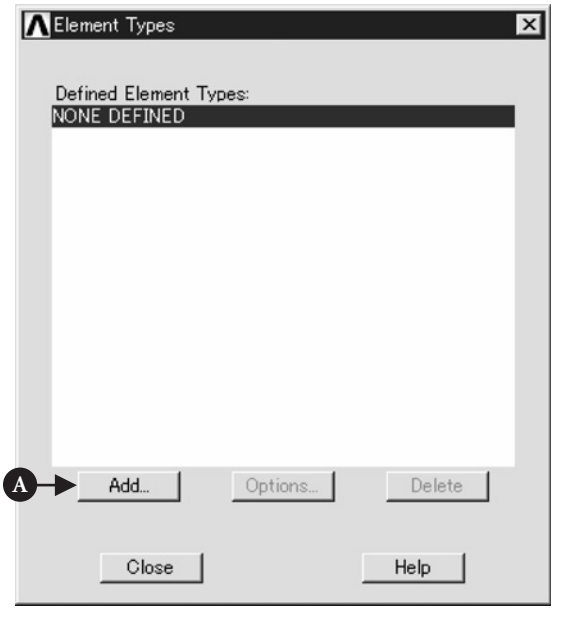


Figure 4.38 Window of **Element Types**.

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types**, as shown in Figure 4.38, is opened.

(1) Click [A] **add**. Then the window **Library of Element Types**, as shown in Figure 4.39, opens.

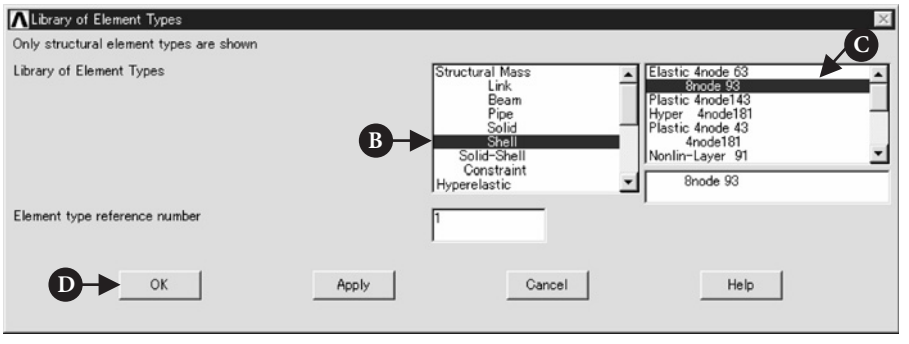


Figure 4.39 Window of **Library of Element Types**.

(2) Select [B] **Shell** in the table of **Library of Element Types** and, then, select [C] **Elastic 8node 93**.

(3) **Element type reference number** is set to **1**. Click [D]**OK** button to close the window **Library of Element Types**.

(4) Click [E] **Close** button in the window of Figure 4.40.

4.3.2.2 REAL CONSTANTS FOR BEAM ELEMENT

COMMAND | **ANSYS Main Menu → Preprocessor → Real Constants → Add/Edit/Delete → Add**

(1) The window **Real Constants** opens (Figure 4.41). Click [A]**add** button, and the window **Element Type for Real Constants** appears in which the name of element type selected is listed as shown in Figure 4.42.

(2) Click [B] **OK** button to input the values of real constants and the window **Real Constant Set Number1, for Shell** opens (Figure 4.43).

(3) Input the following values in Figure 4.43: [C] shell thickness at nodes I, J, K, and L = 0.05e−3. After inputting these values, click [D] **OK** button to close the window.

(4) Click [E] **Close** button in the window of **Real Constants** (Figure 4.44).

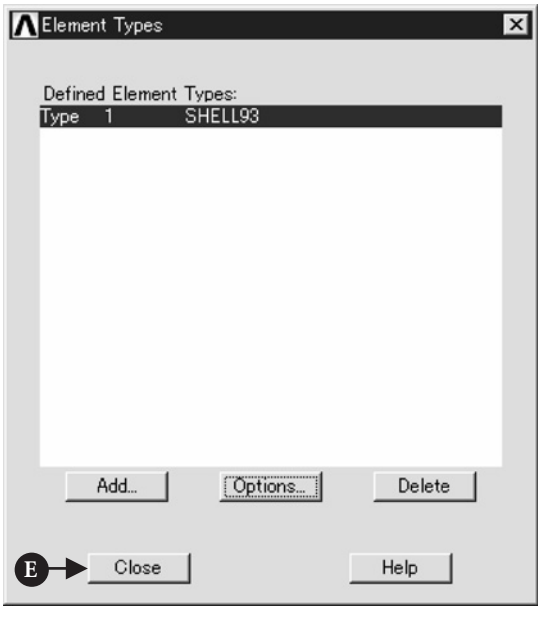


Figure 4.40 Window of **Library of Element Types**.

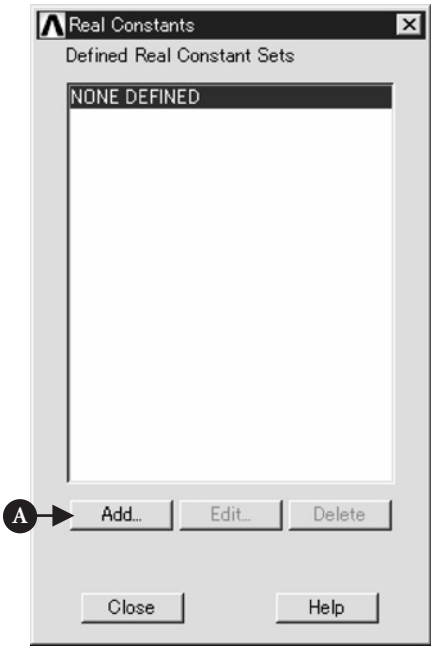


Figure 4.41 Window of **Real Constants**.

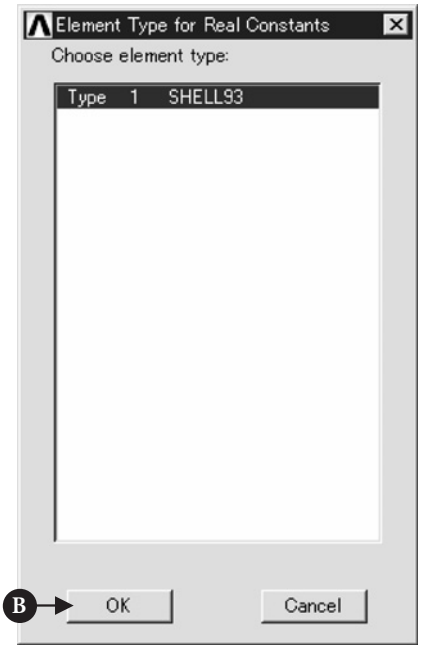


Figure 4.42 Window of **Real Constants**.

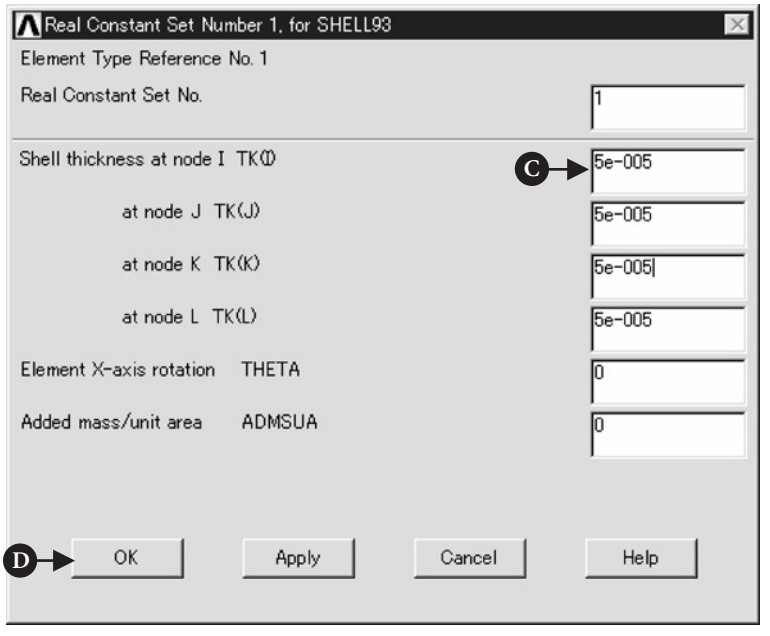


Figure 4.43 Window of **Real Constants Set Number 1, for Shell**.

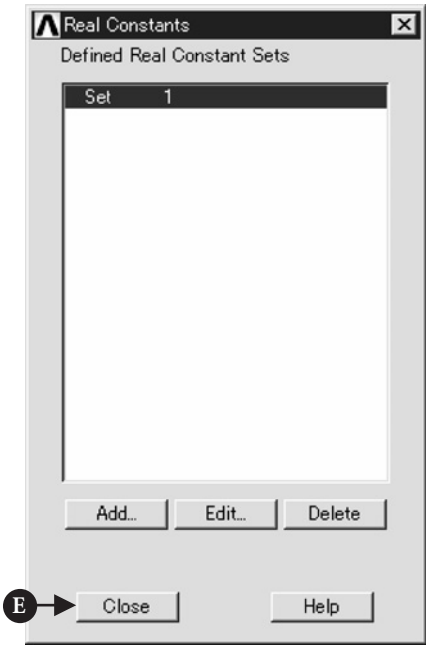


Figure 4.44 Window of **Real Constants**.

4.3.2.3 MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of shell element.

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) Click the above buttons in order and the window **Define Material Model Behavior**, as shown in Figure 4.12, opens .

(2) Double click the following terms in the window:

Structural → Linear → Elastic → Isotropic

Then the window of **Linear Isotropic Properties for Material Number 1** opens.

(3) Input Young's modulus of **206e9** to [A] **EX** box and Poisson ratio of **0.3** to [B] **PRXY** box. Then click [C] **OK** button (Figure 4.45).

Next, define the value of density of material.

(1) Double click the term **Density** and the window **Density for Material Number 1** opens (Figure 4.46).

(2) Input the value of Density, **7800** to [D] **DENS** box and click [E] **OK** button (Figure 4.46). Finally close the window **Define Material Model Behavior** by clicking **X** mark at the upper right end.

4.3.2.4 CREATE KEYPOINTS

To draw a suspension for the analysis, the method of keypoints is described in this section.

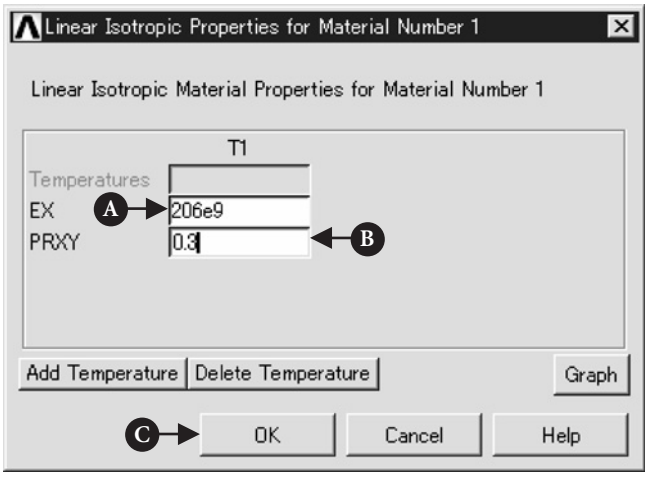


Figure 4.45 Window of **Linear Isotropic Properties for Material Number 1**.

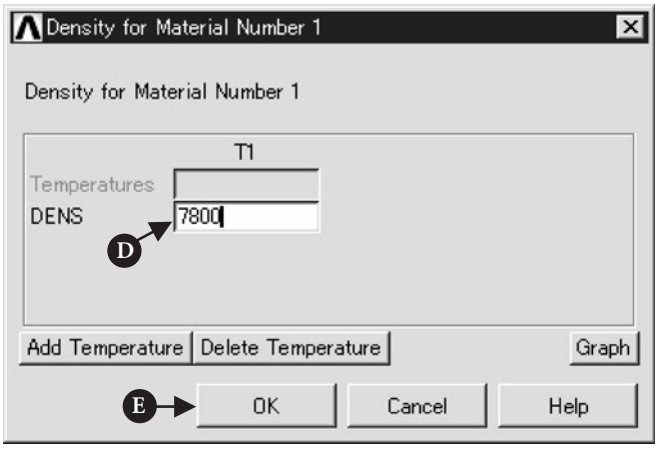


Figure 4.46 Window of **Density for Material Number 1**.

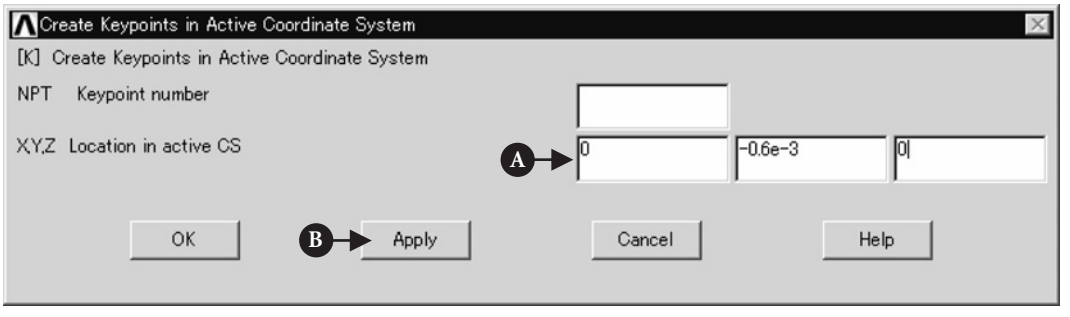


Figure 4.47 Window of **Create Keypoints in Active Coordinate System**.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

(1) The window **Create Keypoints in Active Coordinate System** opens (Figure 4.47).

(2) Input [A] **0, −0.6e−3, 0** to **X, Y, Z Location in active CS** box, and then click [B] **Apply** button. Do not click **OK** button at this stage. If a figure is not inputted into [C] **NPT Key Point number** box, the number of keypoint is automatically assigned in order.

(3) In the same window, input the values as shown in Table 4.1 in order. When all values are inputted, click **OK** button.

(4) Then all inputted keypoints appear on **ANSYS Graphics** window as shown in Figure 4.48.

Table 4.1 X, Y, and Z coordinates of keypoints for suspension

KP No.	X	Y	Z
1	0	−0.6e−3	
2	8.3e−3	−2.15e−3	
3	8.3e−3	−1.4e−3	
4	13.5e−3	−1.4e−3	
5	13.5e−3	1.4e−3	
6	8.3e−3	1.4e−3	
7	8.3e−3	2.15e−3	
8	0	0.6e−3	
9	8.3e−3	−0.8e−3	
10	10.5e−3	−0.8e−3	
11	10.5e−3	0.8e−3	
12	8.3e−3	0.8e−3	
13	0	−0.6e−3	0.3e−3
14	8.3e−3	−2.15e−3	0.3e−3
15	8.3e−3	2.15e−3	0.3e−3
16	0	0.6e−3	0.3e−3

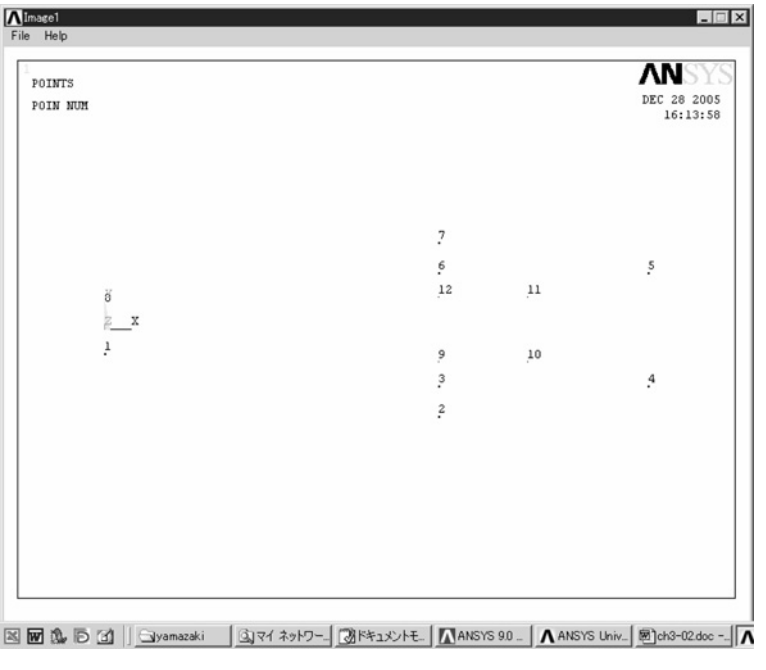


Figure 4.48 **ANSYS Graphics** window.

4.3.2.5 CREATE AREAS FOR SUSPENSION

Areas are created from keypoints by performing the following steps:

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → Through KPs**

(1) The window **Create Area thru KPs** opens (Figure 4.49).

(2) Pick the keypoints, [A] **9**, **10**, **11**, and **12** in Figure 4.50 in order and click [B] **Apply** button in Figure 4.49. An area is created on the window as shown in Figure 4.51.

(3) By performing the same steps, other areas are made on the window. Click keypoints listed in Table 4.2 and make other areas. When you make area No. 3 and 4, you have to rotate the drawing of suspension, using **PlotCtrls—Pan-Zoom-Rotate** in Utility Menu.

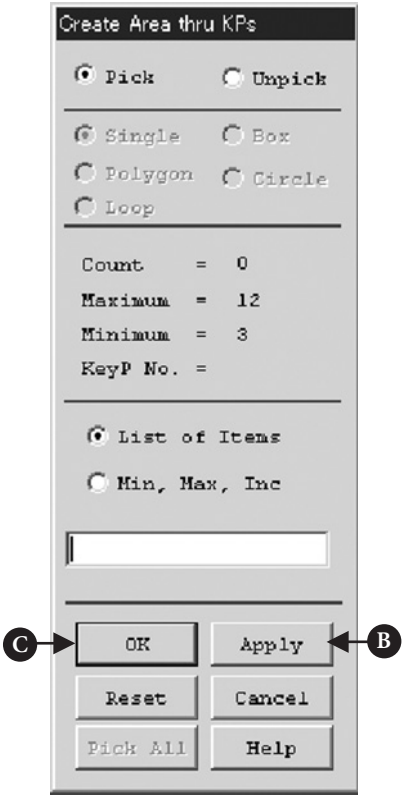


Figure 4.49 Window of **Create Area thru KPs**.

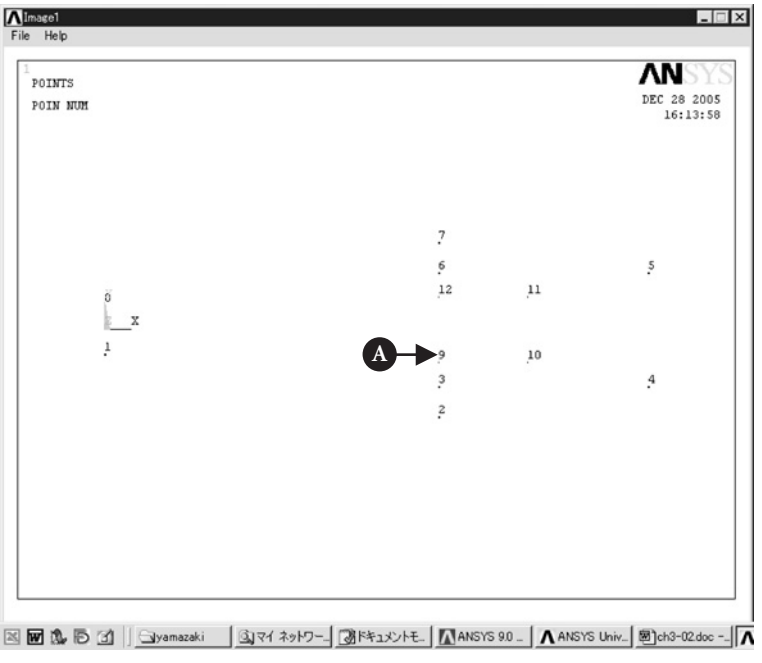


Figure 4.50 **ANSYS Graphics** window.

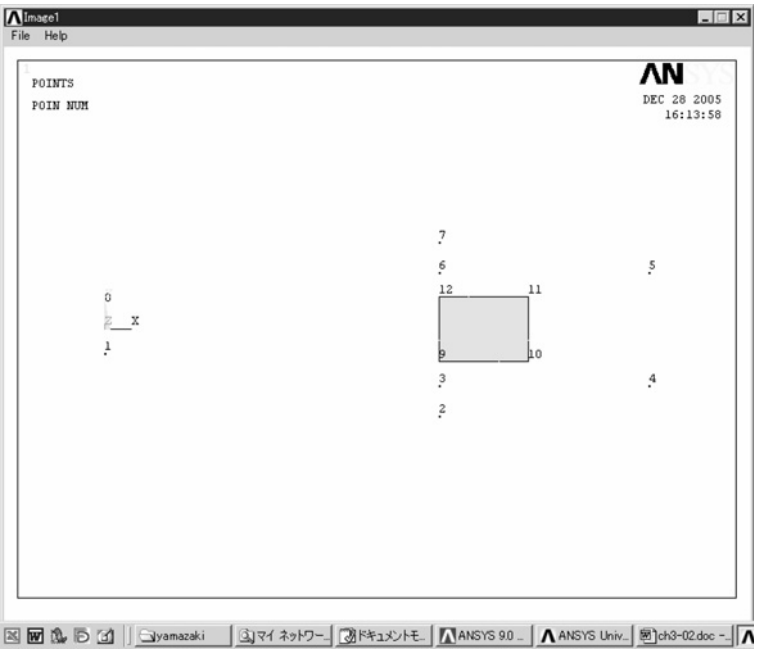


Figure 4.51 **ANSYS Graphics** window.

Table 4.2 Keypoint numbers for making areas of a suspension

Area No.	Keypoint number
1	9, 10, 11, 12
2	1, 2, 3, 4, 5, 6, 7, 8
3	1, 2, 14, 13
4	8, 7, 15, 16

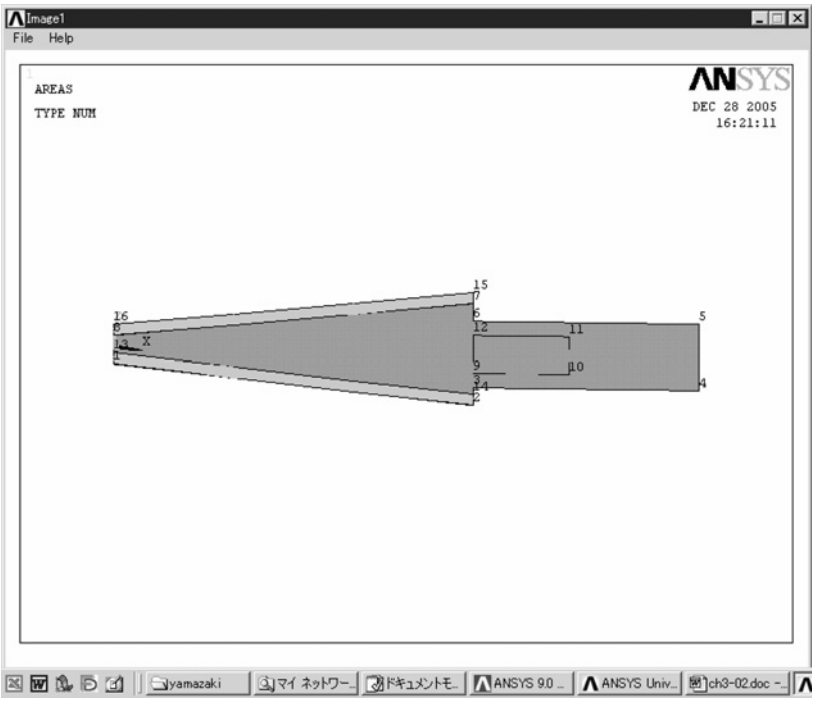


Figure 4.52 **ANSYS Graphics** window.

(4) When all areas are made on the window, click [C] **OK** button in Figure 4.49. Then the drawing of the suspension in Figure 4.52 appears.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

The window **Solid Circle Area** opens, Figure 4.53. Input [D]the values of **12.0e−3**, **0**, **0.6e−3** to **X**, **Y** and **Radius** boxes as shown in Figure 4.53, respectively, and click [E] **OK** button. Then the solid circle is made in the drawing of suspension as shown in Figure 4.54.

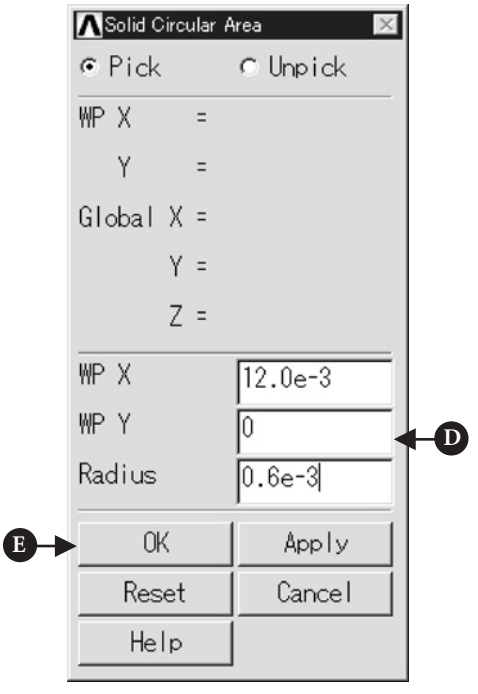


Figure 4.53 Window of **Solid Circular Area**.

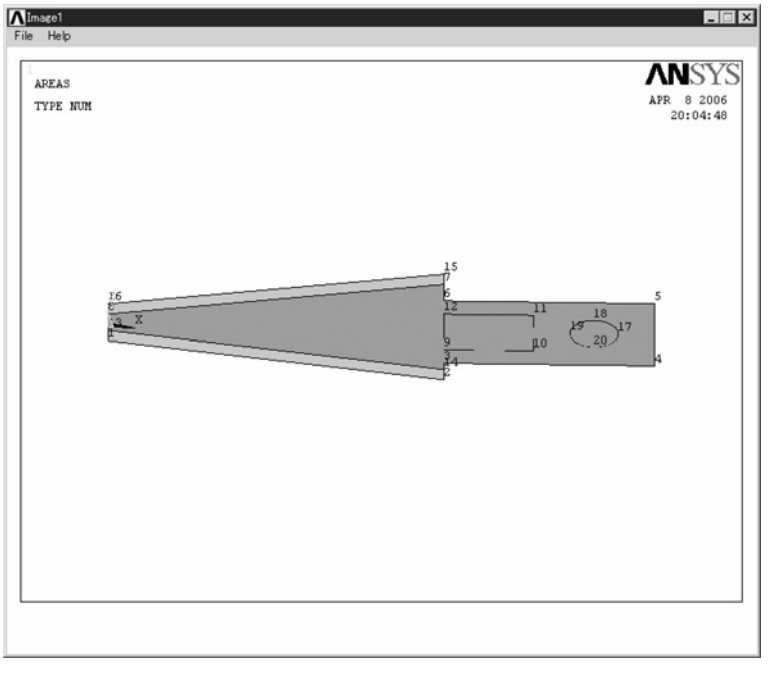


Figure 4.54 **ANSYS Graphics** window.

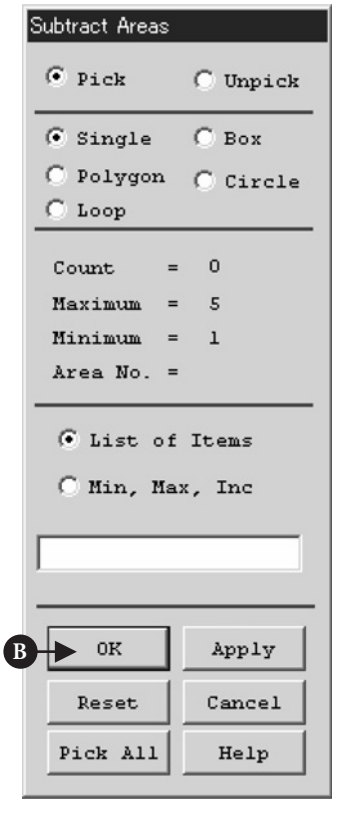


Figure 4.55 Window of **Subtract Areas**.

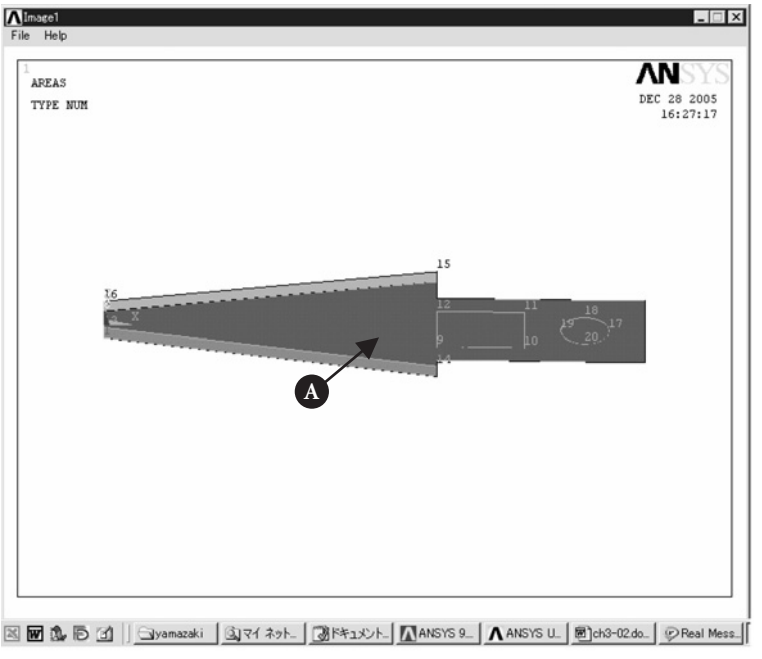


Figure 4.56 **ANSYS Graphics** window.

4.3.2.6 BOOLEAN OPERATION

In order to make a spring region and the fixed region of the suspension, the rectangular and circular areas in the suspension are subtracted by Boolean operation.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) The window **Subtract Areas** opens (Figure 4.55).

(2) Click [A] the area of the suspension in **ANSYS Window** and [B] **OK** button in Figure 4.56. Then click [C] rectangular and [D] circular areas as shown in Figure 4.57 and [B] **OK** button in Figure 4.54. The drawing of the suspension appears as shown in Figure 4.58.

(3) In analysis, all areas have to be glued.

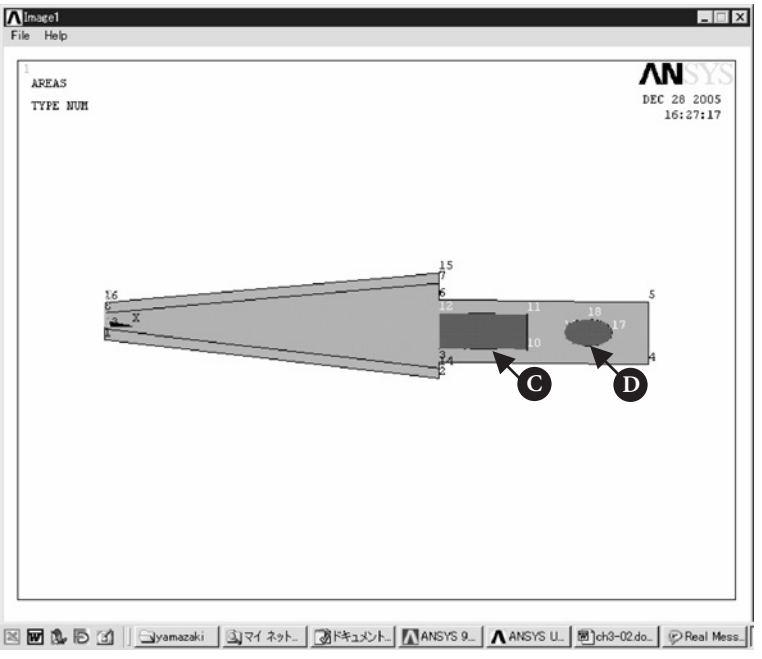


Figure 4.57 **ANSYS Graphics** window.

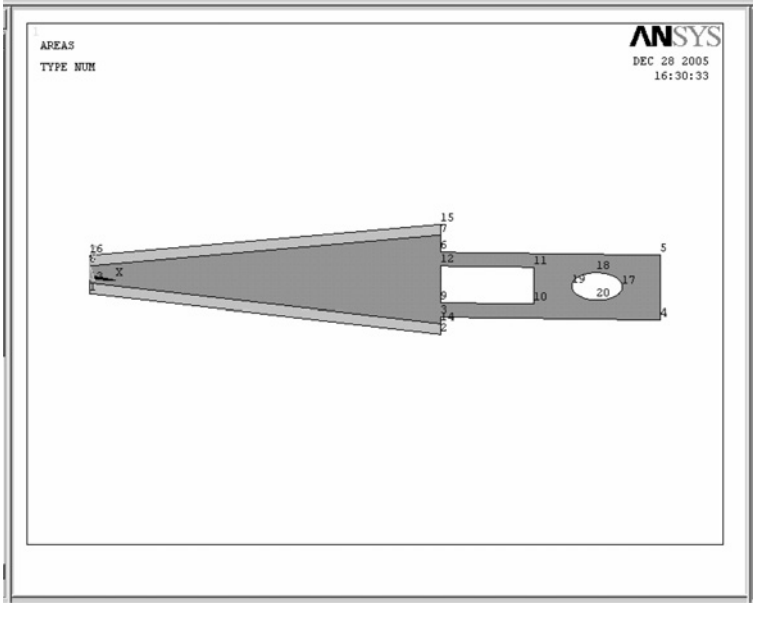


Figure 4.58 **ANSYS Graphics** window.

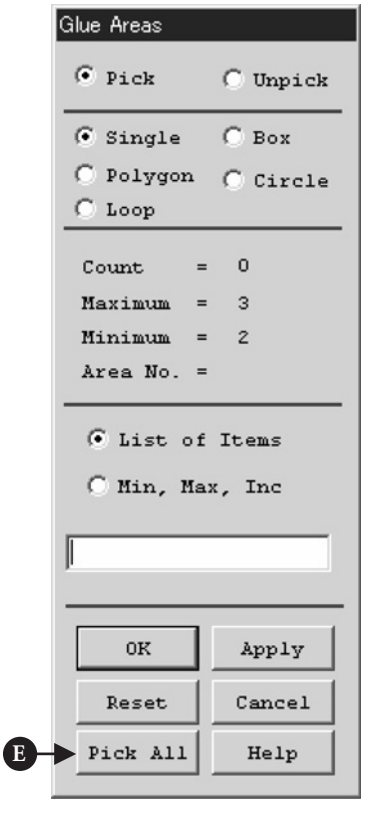


Figure 4.59 Window of **Glue Areas**.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Glue → Areas**

The window **Glue Areas** opens (Figure 4.59). Click [E] **Pick All** button.

4.3.2.7 CREATE MESH IN AREAS

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Size Cntrls → Manual Size → Areas → All Areas**

The window **Element Sizes on All Selected Areas** opens (Figure 4.60).

(1) Input [A] **0.0002** to **SIZE** box. This means that areas are divided by meshes of edge length of 0.0002 m.

(2) Click [B] **OK** button and close the window.

Figure 4.60 Window of **Element Sizes on All Selected Areas**.

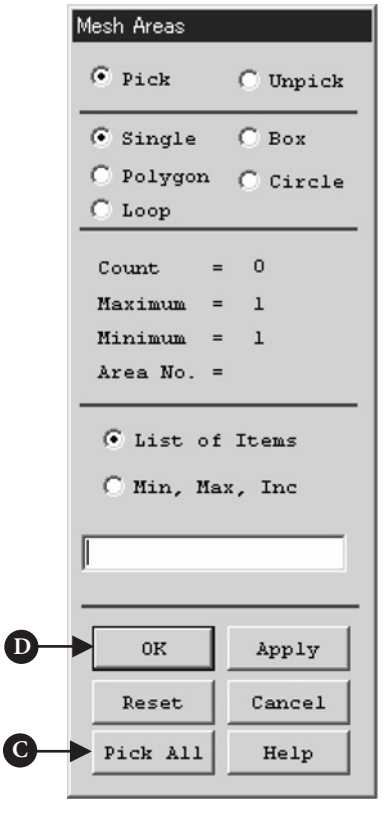


Figure 4.61 Window of **Mesh Areas**.

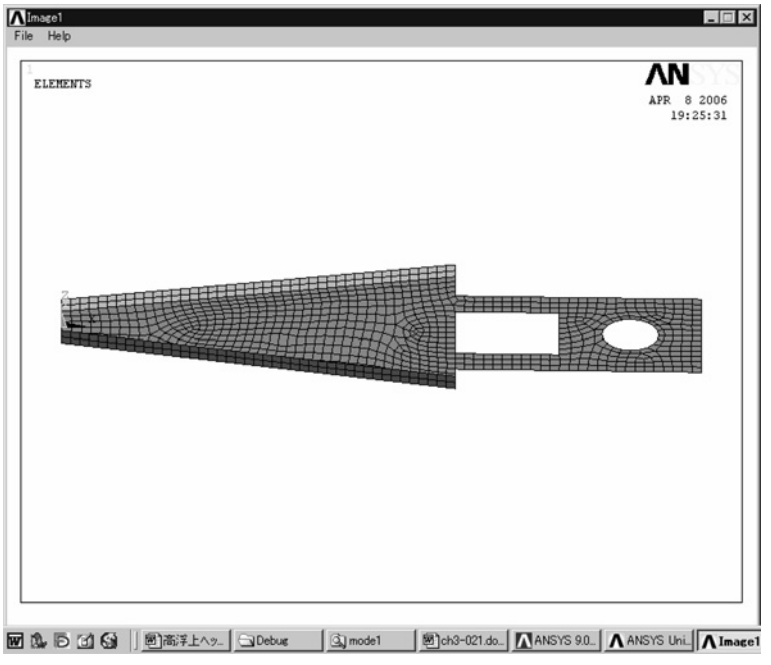


Figure 4.62 **ANSYS Graphics** window.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Areas → Free**

The window **Mesh Areas** opens (Figure 4.61).

(1) Click [C] **Pick ALL** and, then, [D] **OK** button to finish dividing the areas as shown in Figure 4.62.

4.3.2.8 BOUNDARY CONDITIONS

The suspension is fixed at the edge of the circle.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Lines**

The window **Apply U,ROT on Lines** opens (Figure 4.63).

(1) Pick [A] four lines included in the circle in Figure 4.64 and click [B] **OK** button in Figure 4.63. Then the window **Apply U,ROT on Lines** opens (Figure 4.65).

(2) In order to set the boundary condition, select [C] **All DOF** in the box of **Lab2**. Input [D] **0** to the box of **VALUE** and, then, click [E] **OK** button. After these steps, the **ANSYS Graphics** window is changed as shown in Figure 4.66.

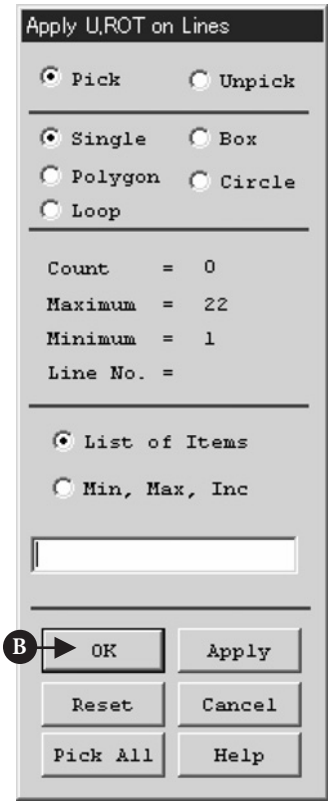


Figure 4.63 Window of **Apply U,ROT on Lines**.

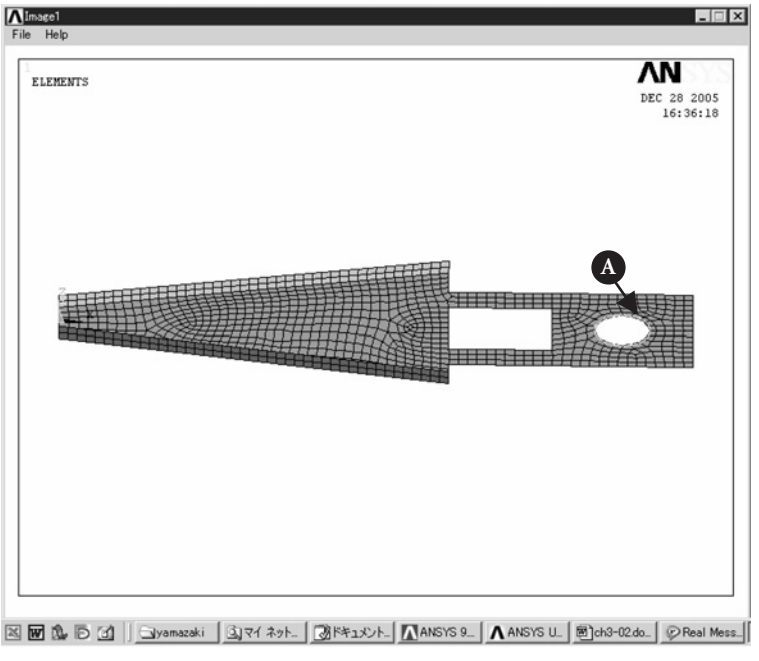


Figure 4.64 **ANSYS Graphics** window.

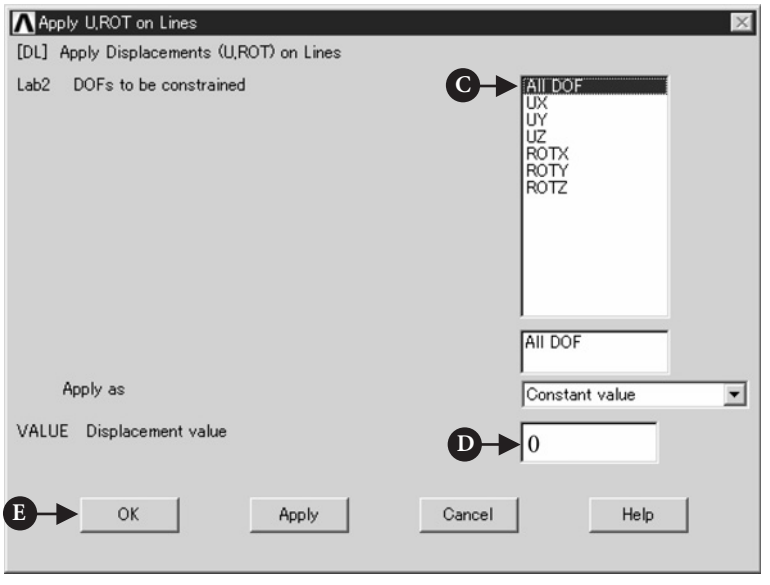


Figure 4.65 Window of **Apply U,ROT on Lines**.

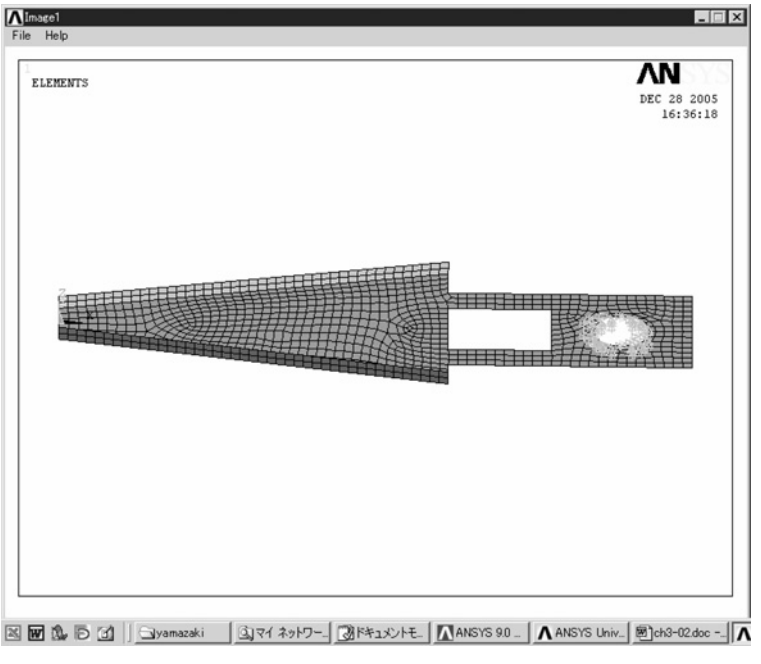


Figure 4.66 **ANSYS Graphics** window.

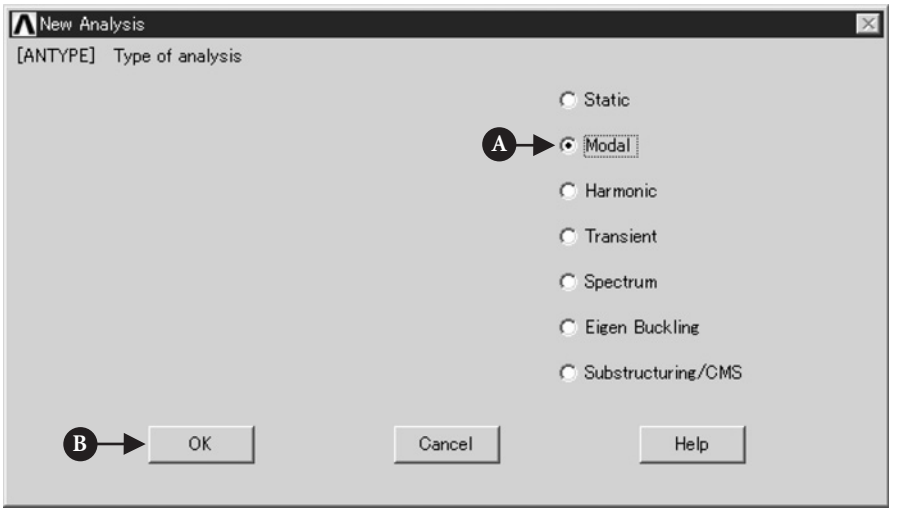


Figure 4.67 Window of **New Analysis**.

4.3.3 ANALYSIS

4.3.3.1 DEFINE THE TYPE OF ANALYSIS

The following steps are performed to define the type of analysis.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → New Analysis**

The window **New Analysis** opens (Figure 4.67).

(1) Check [A]**Modal** and, then, click [B]**OK** button.

In order to define the number of modes to extract, the following steps are performed.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → Analysis Options**

The window **Modal Analysis** opens (Figure 4.68).

(1) Check [C]**Subspace** of **MODOPT** and input [D] **10** in the box of **No. of modes to extract** and click [E] **OK** button.

(2) Then, the window **Subspace Modal Analysis** as shown in Figure 4.69 opens. Input [F] **20000** in the box of **FREQE** and click [G] **OK** button.

4.3.3.2 EXECUTE CALCULATION

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

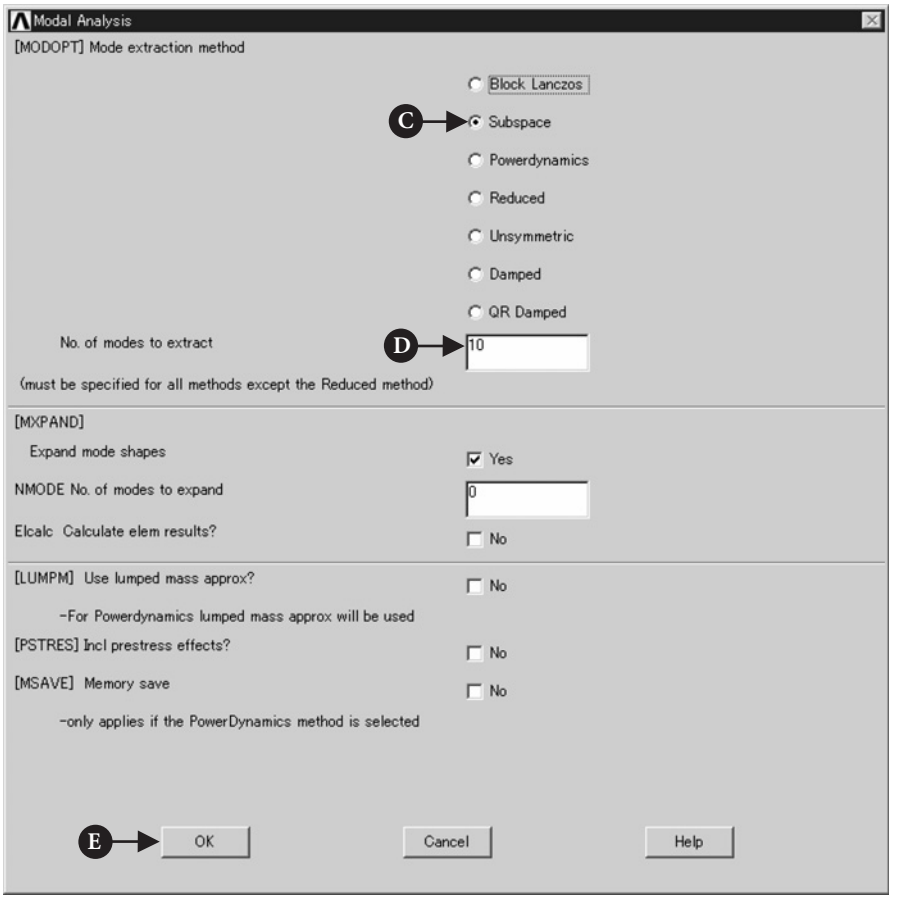


Figure 4.68 Window of **Modal Analysis**.

4.3.4 POSTPROCESSING

4.3.4.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → First Set**

4.3.4.2 PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape**

The window **Plot Deformed Shape** opens (Figure 4.70).

(1) Select [A] **Def+Undeformed** and click [B] **OK**.

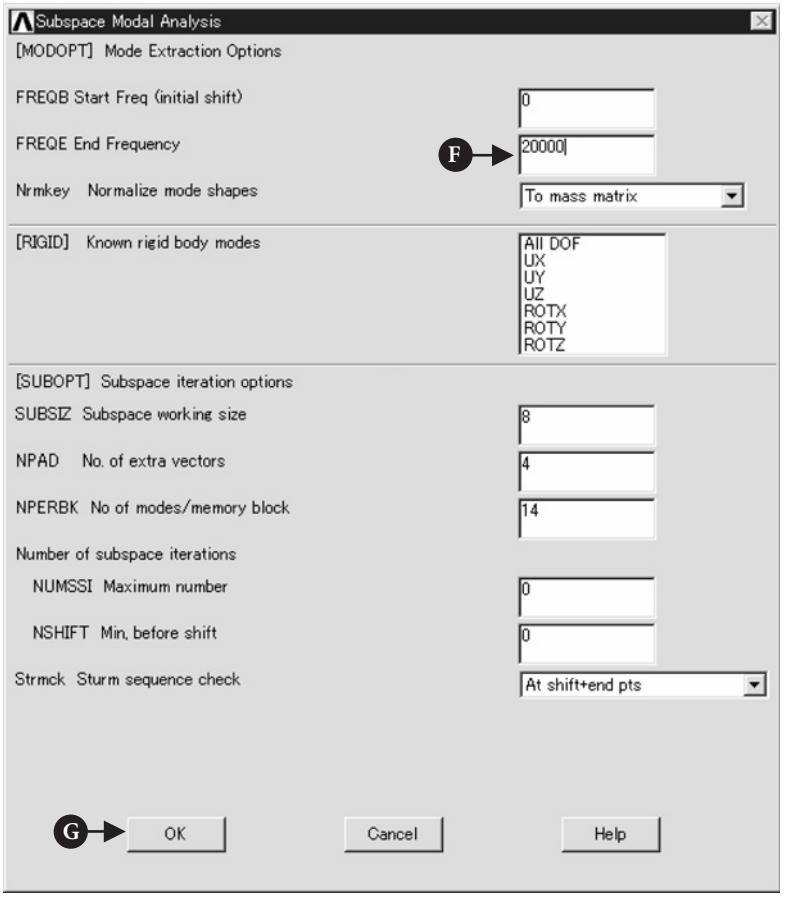


Figure 4.69 Window of **Subspace Modal Analysis**.

(2) The calculated result for the first mode of vibration appears on **ANSYS Graphics** window as shown in Figure 4.71. The resonant frequency is shown as FRQE at the upper left side on the window.

4.3.4.3 READ THE CALCULATED RESULTS OF HIGHER MODES OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → Next Set**

Perform the same steps described in Section 4.2.4.2 and the calculated results from the second mode to the sixth mode of vibration are displayed on the windows as shown in Figures 4.72–4.76. From these vibration modes, it is found that a large radial displacement appears at the fifth mode of vibration.

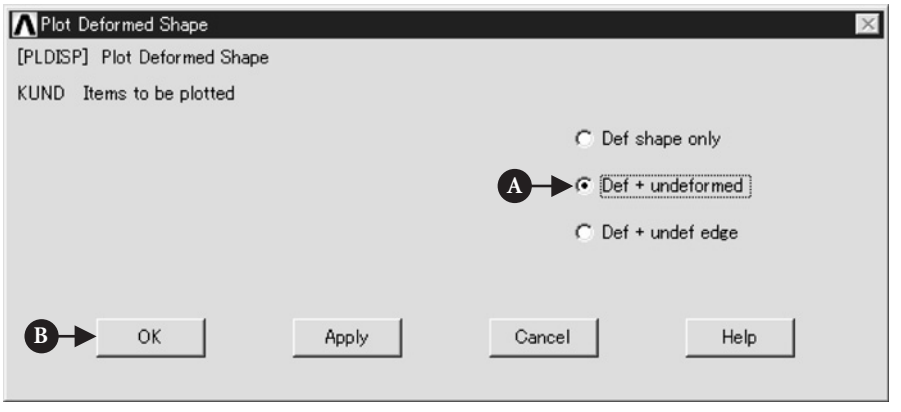


Figure 4.70 Window of **Plot Deformed Shape**.

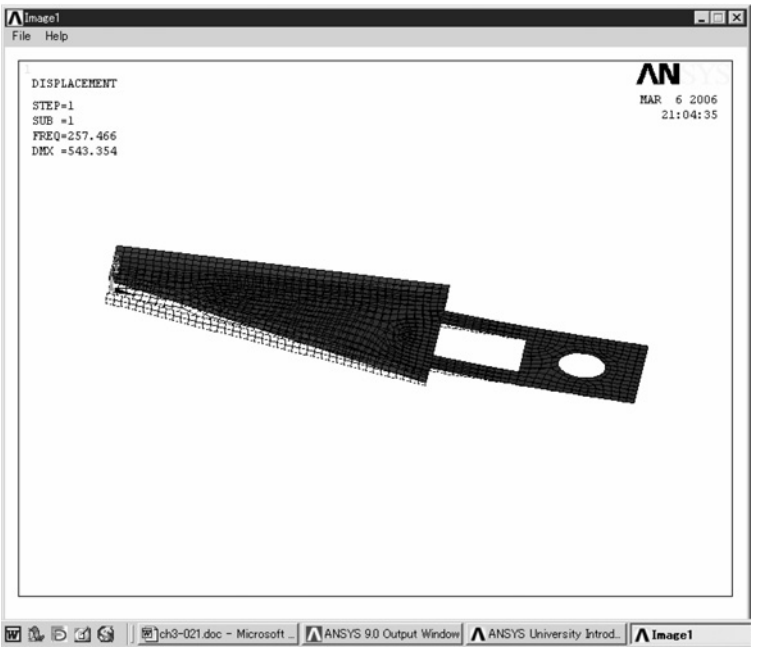


Figure 4.71 The first mode of vibration.

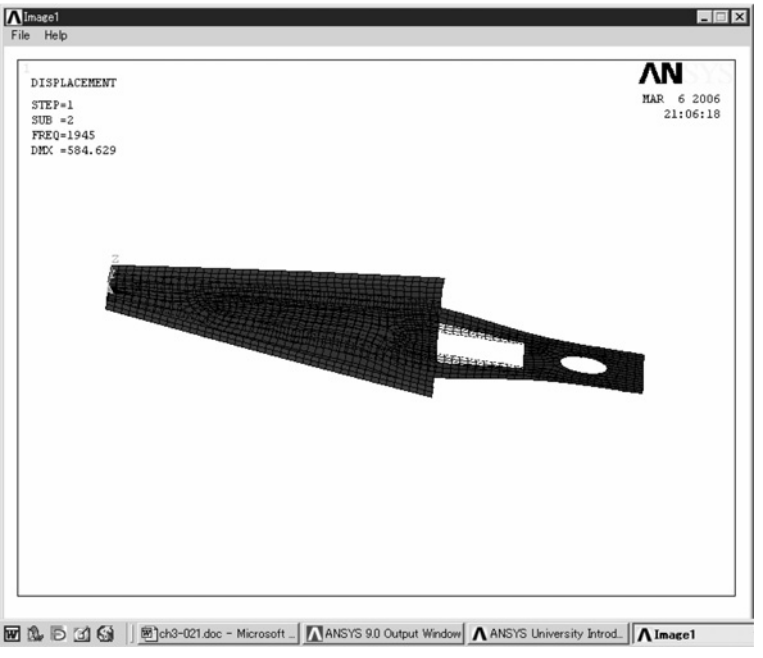


Figure 4.72 The second mode of vibration.

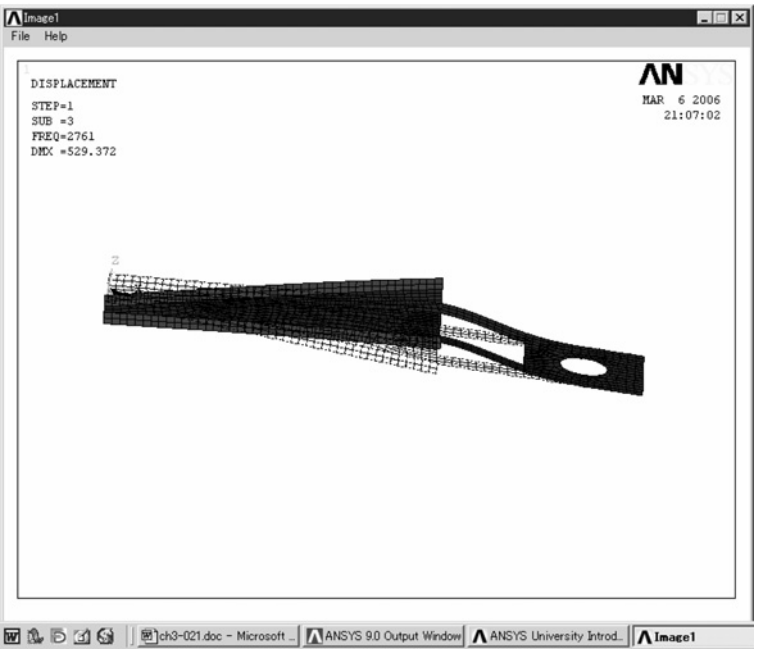


Figure 4.73 The third mode of vibration.

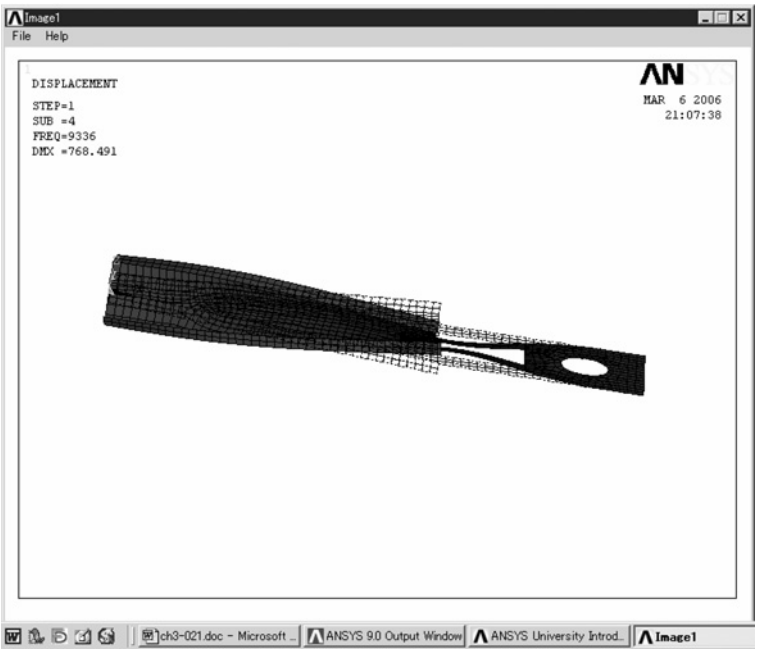


Figure 4.74 The fourth mode of vibration.

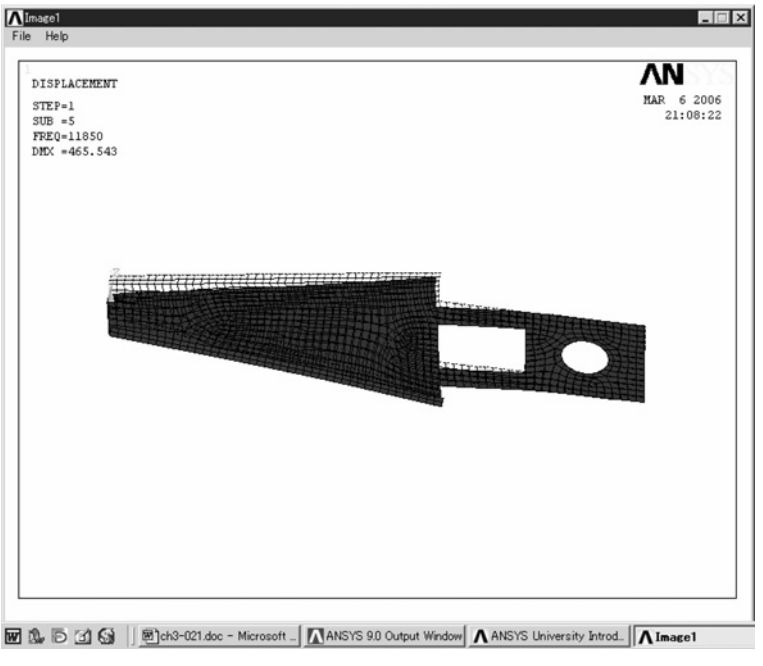


Figure 4.75 The fifth mode of vibration.

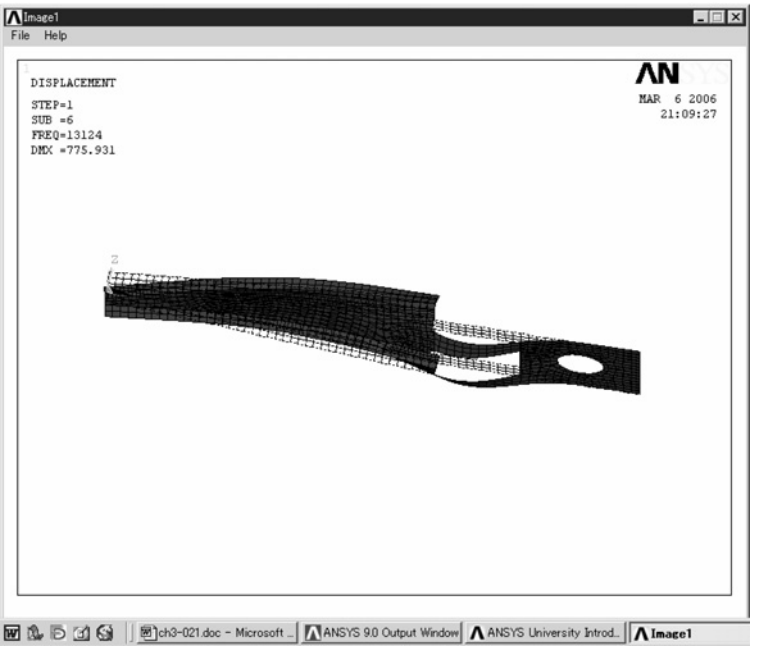


Figure 4.76 The sixth mode of vibration.

4.4 Mode analysis of a one-axis precision moving table using elastic hinges

4.4.1 Problem description

A one-axis table using elastic hinges has been often used in various precision equipment, and the position of a table is usually controlled at nanometer-order accuracy using a piezoelectric actuator or a voice coil motor. Therefore, it is necessary to confirm the resonant frequency in order to determine the controllable frequency region.

Obtain the resonant frequency of a one-axis moving table using elastic hinges when the bottom of the table is fixed and a piezoelectric actuator is selected as an actuator:

- Material: Steel, thickness of the table: 5 mm
- Young's modulus, $E = 206$ GPa, Poisson's ratio $\nu = 0.3$
- Density $\rho = 7.8 \times 10^3$ kg/m^3
- Boundary condition: All freedoms are constrained at the bottom of the table and the region A indicated in Figure 4.77, where a piezoelectric actuator is glued.

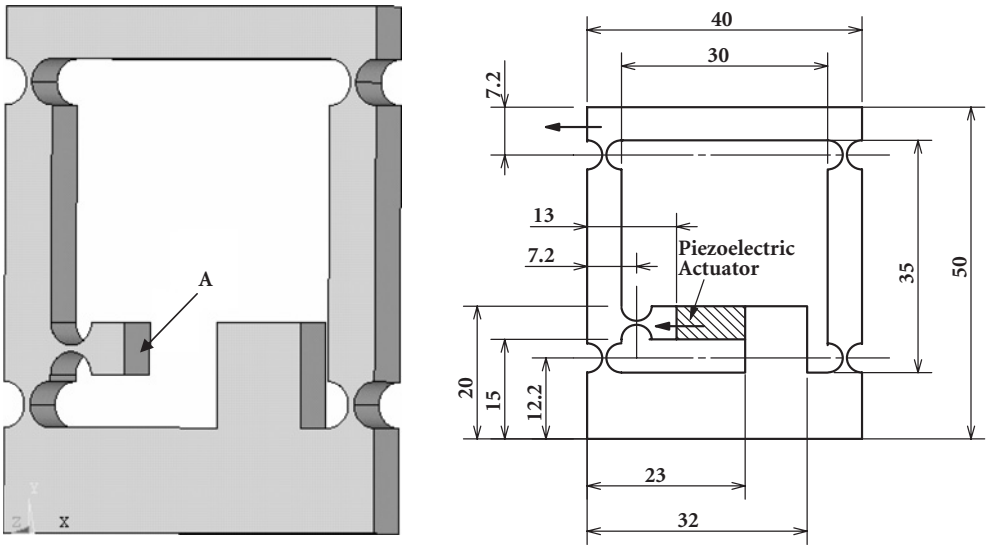


Figure 4.77 A one-axis moving table using elastic hinges.

4.4.2 CREATE A MODEL FOR ANALYSIS

4.4.2.1 SELECT ELEMENT TYPE

In this example, the solid element is selected to analyze the resonant frequency of the moving table.

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types** as shown in Figure 4.78 opens.

(1) Click [A] **add** button. Then the window **Library of Element Types** as shown in Figure 4.79 opens.

(2) Select [B] **Solid** in the table of **Library of Element Types** and, then, select [C] **Quad 4node 42** and **Element type reference number** is set to **1** and click [D] **Apply**. Select [E] **Brick 8node 45**, as shown in Figure 4.80.

(3) **Element type reference number** is set to **2** and click [F] **OK** button. Then the window **Library of Element Types** is closed.

(4) Click [G] **Close** button in the window of Figure 4.81 where the names of selected elements are indicated.

4.4.2.2 MATERIAL PROPERTIES

This section describes the procedure of defining the material properties of solid element.

Figure 4.78 Window of **Element Types**.

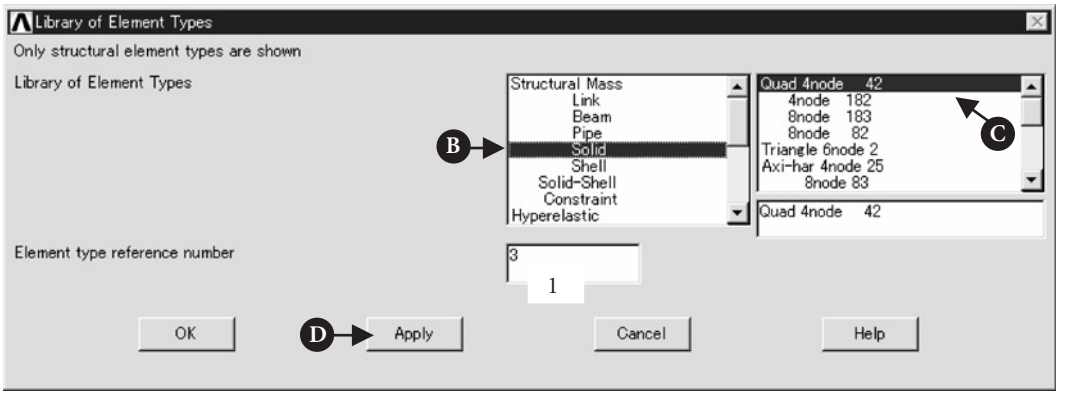


Figure 4.79 Window of **Library of Element Types**.

COMMAND | **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

(1) Click the above buttons in order and the window **Define Material Model Behavior** opens as shown in Figure 4.82.

(2) Double click the following terms in the window.

Structural → Linear → Elastic → Isotropic

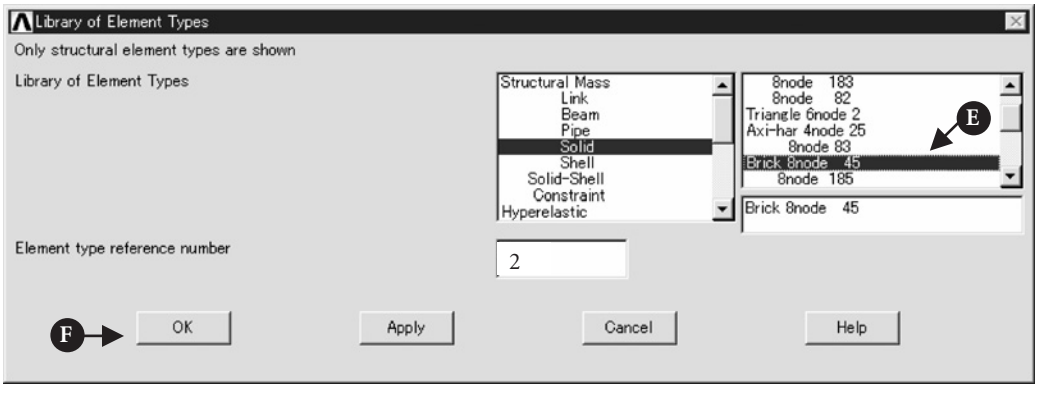


Figure 4.80 Window of **Library of Element Types**.

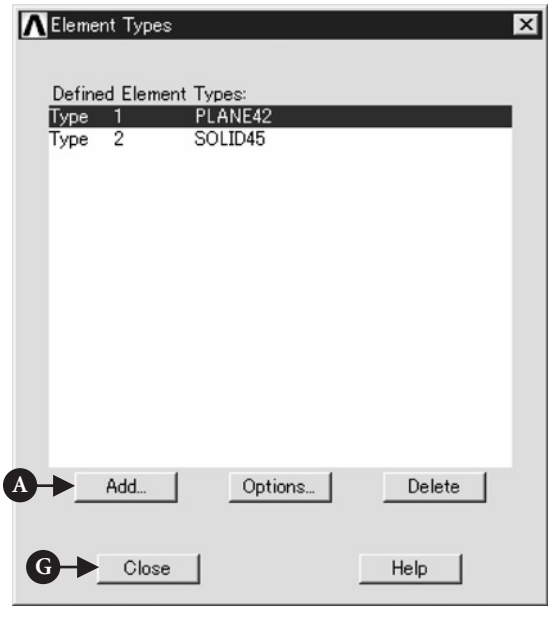


Figure 4.81 Window of **Element Types**.

Then the window **Linear Isotropic Properties for Material Number 1** opens.

(3) Input Young's modulus of **206e9** to **EX** box and Poisson's ratio of **0.3** to **PRXY** box.

Next, define the value of density of material.

(1) Double click the term of **Density**, and the window **Density for Material Number 1** opens.

(2) Input the value of Density, **7800** to **DENS** box and click **OK** button. Finally, close the window **Define Material Model Behavior** by clicking **X** mark at the upper right end.

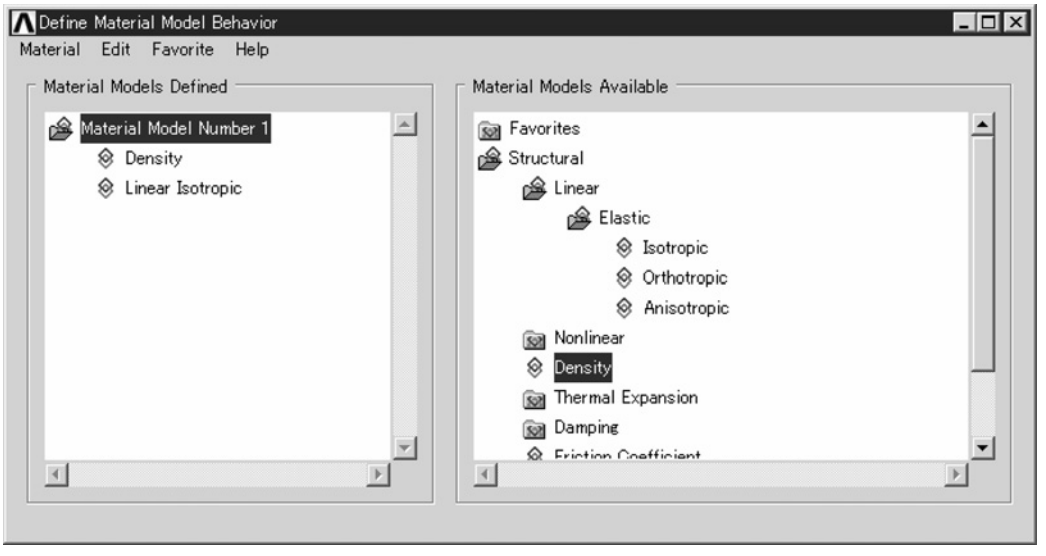


Figure 4.82 Window of **Define Material Model Behavior**.

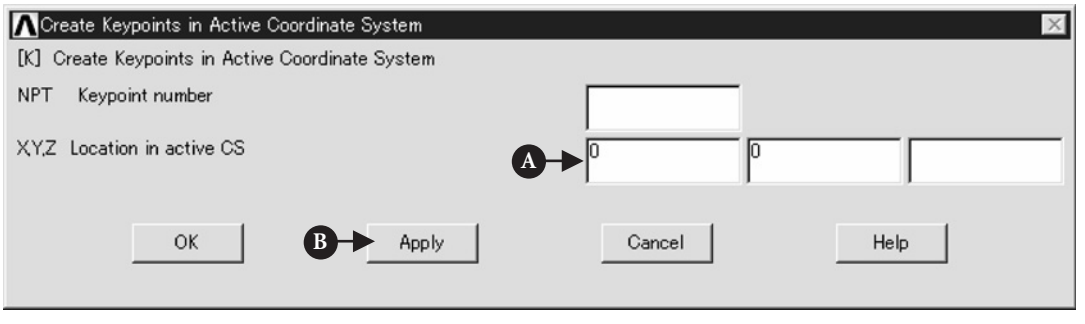


Figure 4.83 Window of **Create Keypoints in Active Coordinate System**.

4.4.2.3 CREATE KEYPOINTS

To draw the moving table for analysis, the method using keypoints is described in this section.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

The window **Create Keypoints in Active Coordinate System** opens (Figure 4.83).

(1) Input **A 0,0** to **X,Y,Z Location in active CS** box, and then click [B] **Apply** button. Do not click **OK** button at this stage.

Table 4.3 Coordinates of KPs

KP No.	X	Y
1	0	0
2	0.04	0
3	0.04	0.05
4	0	0.05
5	0.005	0.01
6	0.035	0.01
7	0.035	0.045
8	0.005	0.045
9	0.005	0.015
10	0.013	0.015
11	0.013	0.02
12	0.005	0.02
13	0.023	0.01
14	0.032	0.01
15	0.032	0.02
16	0.023	0.02

(2) In the same window, input the values as shown in Table 4.3 in order. When all values are inputted, click **OK** button.

(3) Then all input keypoints appear in **ANSYS Graphics** window as shown in Figure 4.84.

4.4.2.4 CREATE AREAS FOR THE TABLE

Areas are created from keypoints by proceeding the following steps.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Arbitrary → Through KPs**

The window **Create Area thru KPs** opens (Figure 4.85).

(1) Pick the keypoints, **1**, **2**, **3**, and **4** in order and click [A] **Apply** button in Figure 4.85. Then pick keypoints **5**, **6**, **7**, and **8** and click [B] **OK** button. Figure 4.86 appears.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(2) Click the area of KP No. **1–4** and **OK** button. Then click the area of KP No. **5–8** and **OK** button. The drawing of the table appears as shown in Figure 4.87.

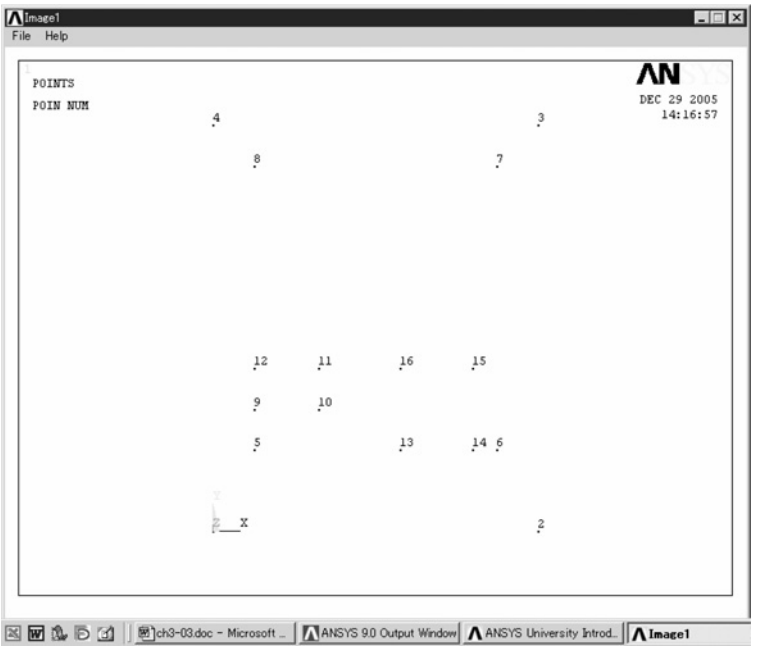


Figure 4.84 **ANSYS Graphics** window.

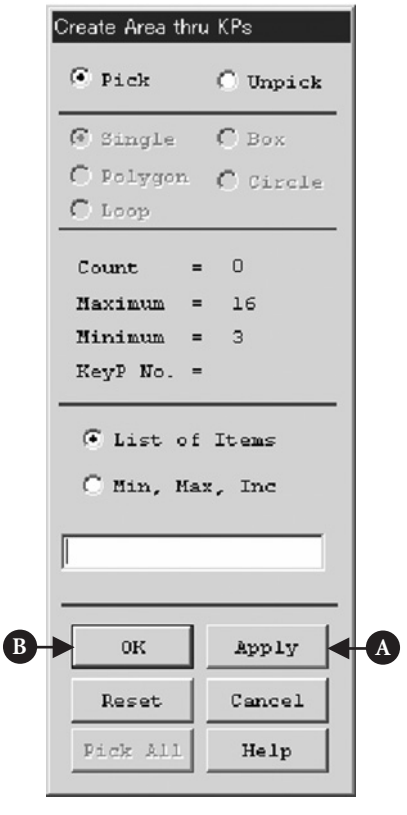


Figure 4.85 Window of **Create Area thru KPs**.

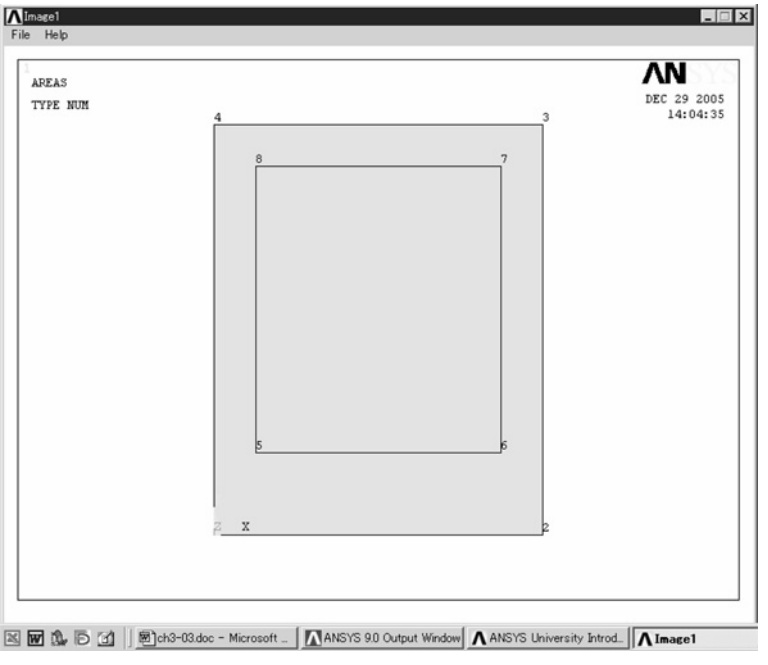


Figure 4.86 **ANSYS Graphics** window.

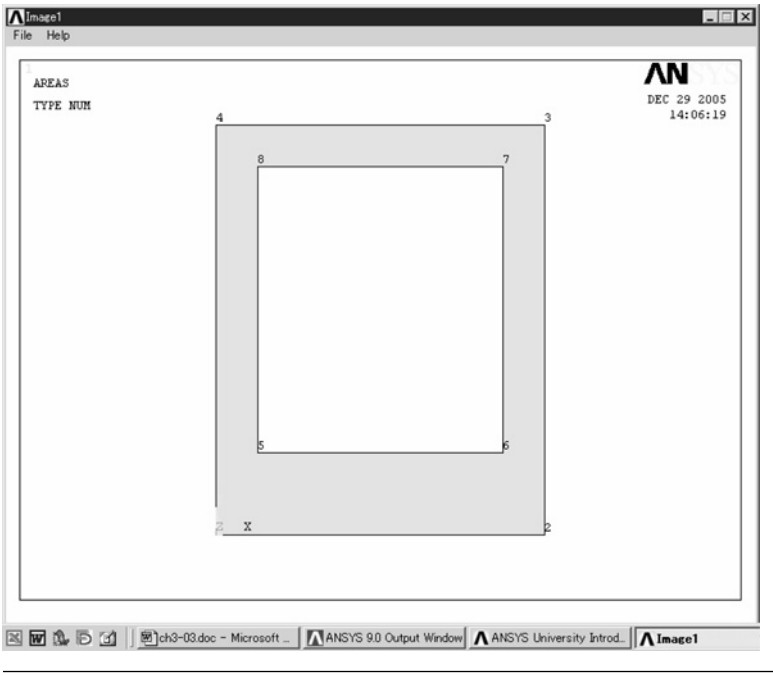


Figure 4.87 **ANSYS Graphics** window.

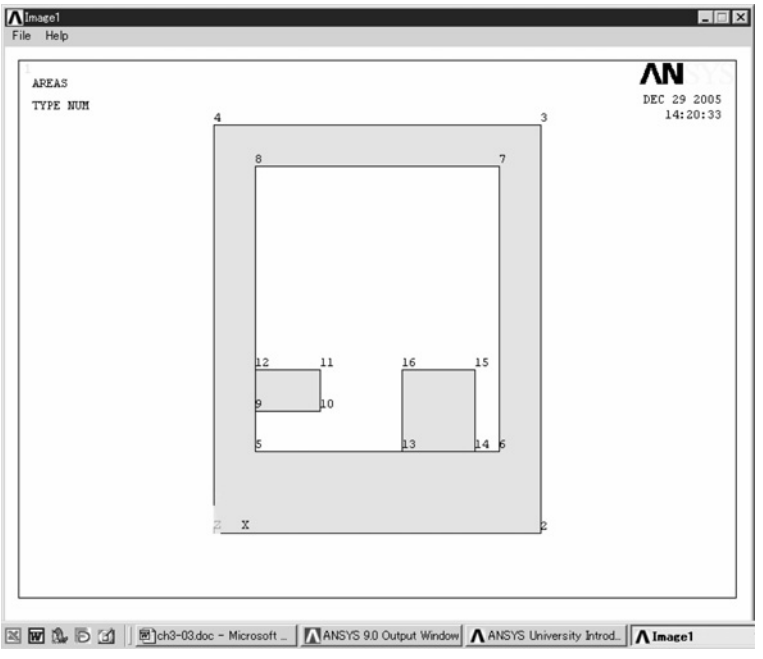


Figure 4.88 **ANSYS Graphics** window.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Arbitrary → Through KPs**

(3) Pick keypoints **9**, **10**, **11**, and **12** in order and click [A] **Apply** button in Figure 4.85. Then pick keypoints **13**, **14**, **15**, and **16** and click [B] **OK** button. Figure 4.88 appears.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas**

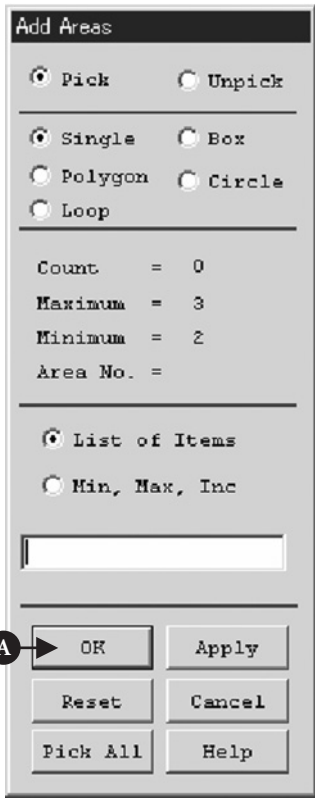
The window **Add Areas** opens (Figure 4.89).

(4) Pick three areas on **ANSYS Graphics** window and click [A] **OK** button. Then three areas are added as shown in Figure 4.90.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle**

The window **Solid Circle Area** opens (Figure 4.91).

(5) Input the values of **0**, **12.2e−3**, **2.2e−3** to **[X]**, **[Y]** and **Radius** boxes as shown in Figure 4.91 and click [A] **Apply** button. Then continue to input the coordinates



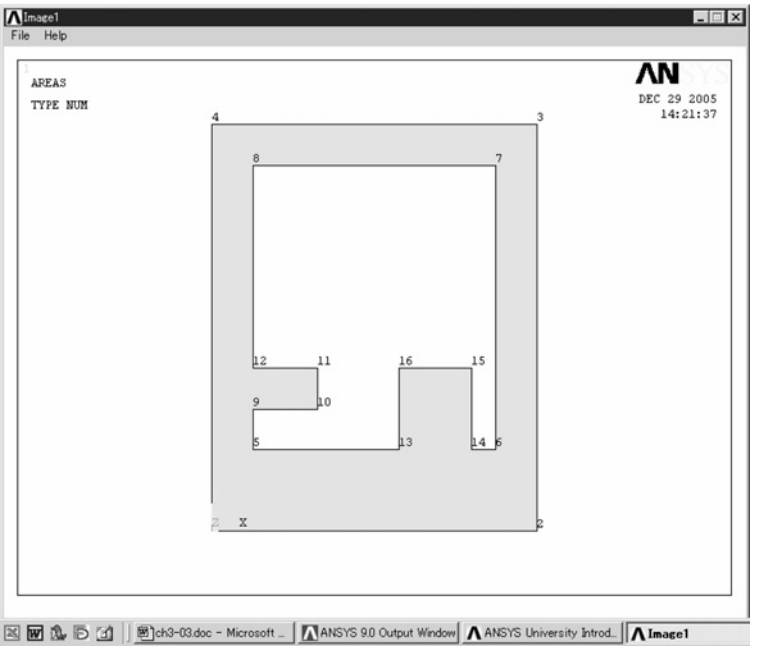


Figure 4.89 Window of **Add Areas**. **Figure 4.90** **ANSYS Graphics** window.

of the solid circle as shown in Table 4.4. The radius of all solid circles is 0.0022. When all values are inputted, the drawing of the table appears as shown in Figure 4.92. Click [B] **OK** button.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) Subtract all circular areas from the rectangular area by executing the above steps. Then Figure 4.93 is displayed.

4.4.2.5 CREATE MESH IN AREAS

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

The window **Mesh Tool** opens (Figure 4.94).

(1) Click [A] **Lines Set** and the window **Element Size on Picked Lines** opens (Figure 4.95).

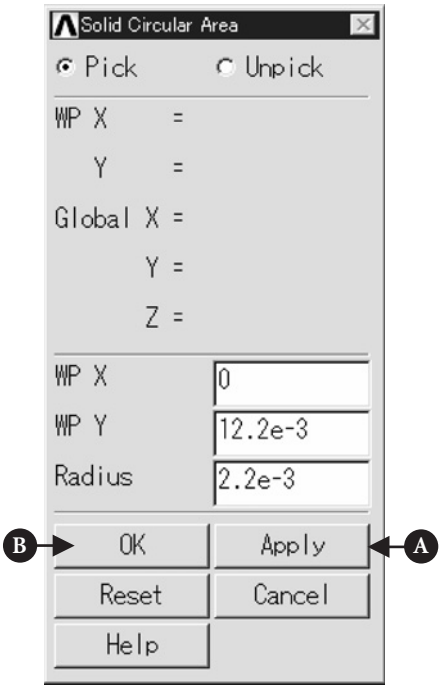


Figure 4.91 Window of **Solid Circular Area**.

Table 4.4 Coordinates of solid circles

No.	X	Y	Radius
1	0	0.0122	0.0022
2	0.005	0.0122	
3	0.035	0.0122	
4	0.04	0.0122	
5	0	0.0428	
6	0.005	0.0428	
7	0.035	0.0428	
8	0.04	0.0428	
9	0.0072	0.015	
10	0.0072	0.02	

(2) Click [B] **Pick All** button and the window **Element Sizes on Picked Lines** opens (Figure 4.96).

(3) Input [C] **0.001** to **SIZE** box and click [D] **OK** button.

(4) Click [E] **Mesh** of the window **Mesh Tool** (Figure 4.97) and, then, the window **Mesh Areas** opens (Figure 4.98).

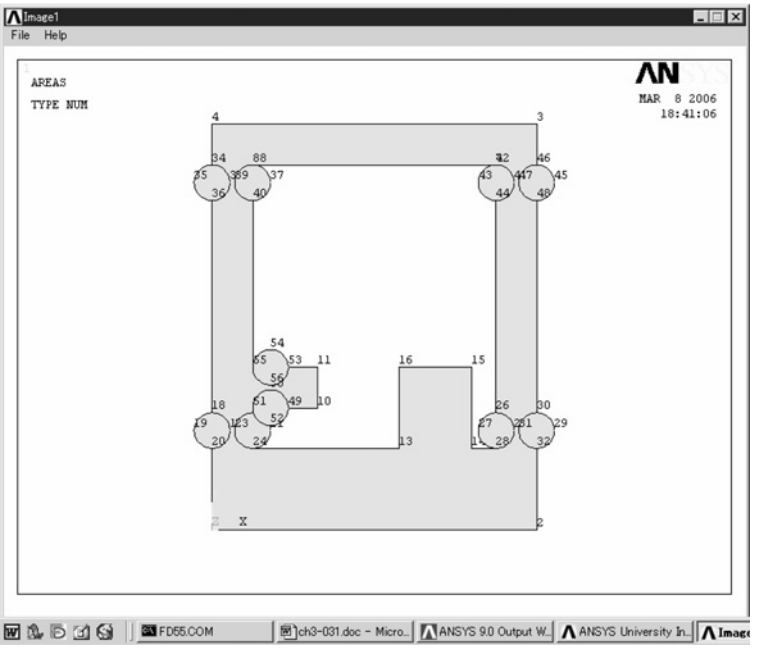


Figure 4.92 **ANSYS Graphics** window.

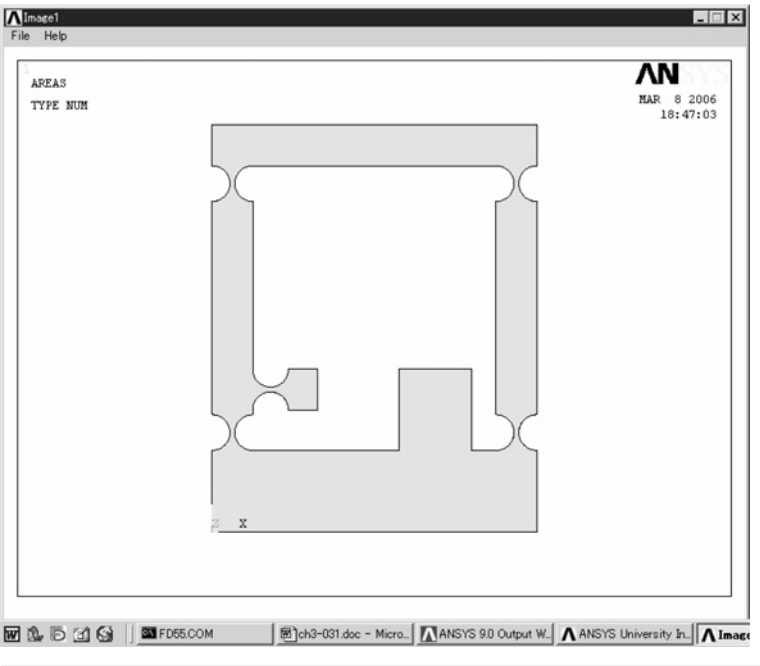


Figure 4.93 **ANSYS Graphics** window.

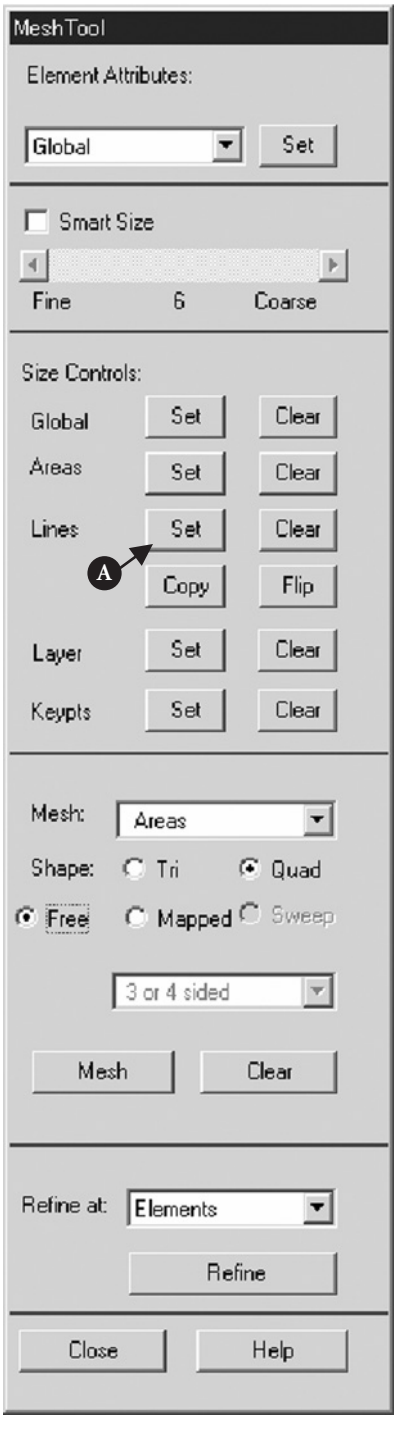


Figure 4.94 Window of **Mesh Tool**.

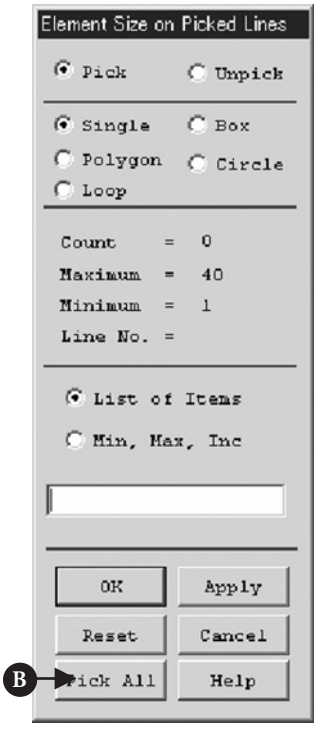


Figure 4.95 Window of **Element Size on Picked Lines**.

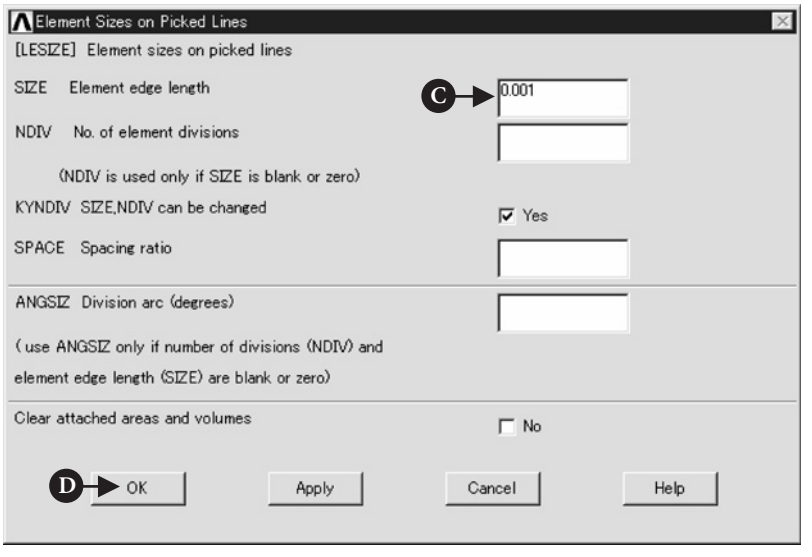


Figure 4.96 Window of **Element Sizes on Picked Lines**.

(5) Pick the area of the table on **ANSYS Graphics** window and click [F] **OK** button. Then the meshed drawing of the table appears on **ANSYS Graphics** window as shown in Figure 4.99.

Next, by performing the following steps, the thickness of 5 mm and the mesh size are determined for the drawing of the table.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Extrude → Elem Ext Opts**

The window **Element Extrusion Options** opens (Figure 4.100).

(1) Input [A] **5** to **VAL1** box. This means that the number of element divisions is 5 in the thickness direction. Then, click [B] **OK** button.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Areas → By XYZ Offset**

The window **Extrude Area by Offset** opens (Figure 4.101).

(1) Pick the area of the table on **ANSYS Graphics** window and click [A] **OK** button. Then, the window **Extrude Areas by XYZ Offset** opens (Figure 4.102).

(2) Input [B] **0,0,0.005** to **DX,DY,DZ** box and click [C] **OK** button. Then, the drawing of the table meshed in the thickness direction appears as shown in Figure 4.103.

4.4.2.6 BOUNDARY CONDITIONS

The table is fixed at both the bottom and the region A of the table.

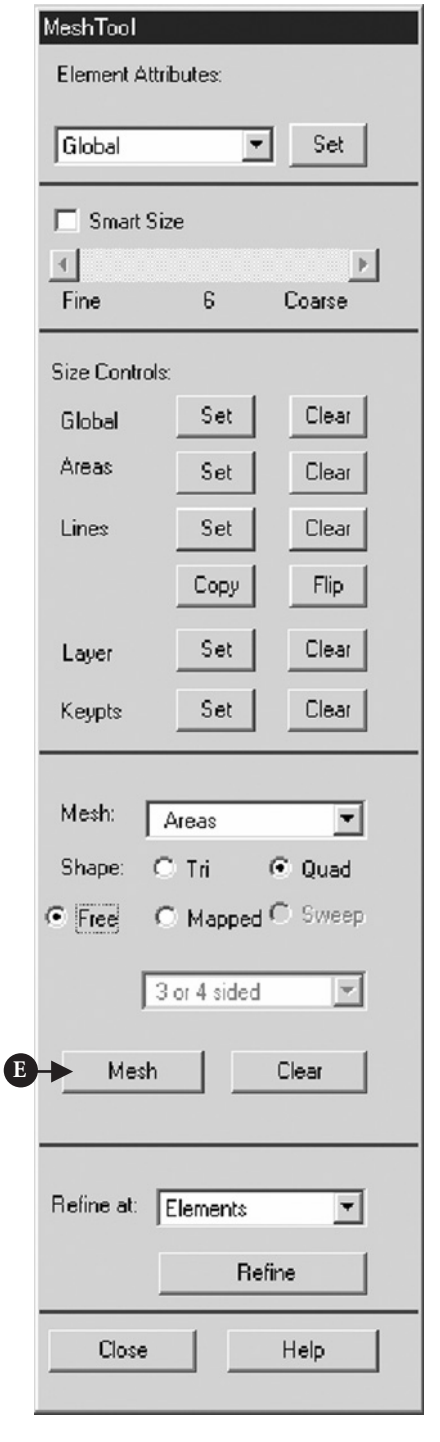


Figure 4.97 Window of **Mesh Tool**.

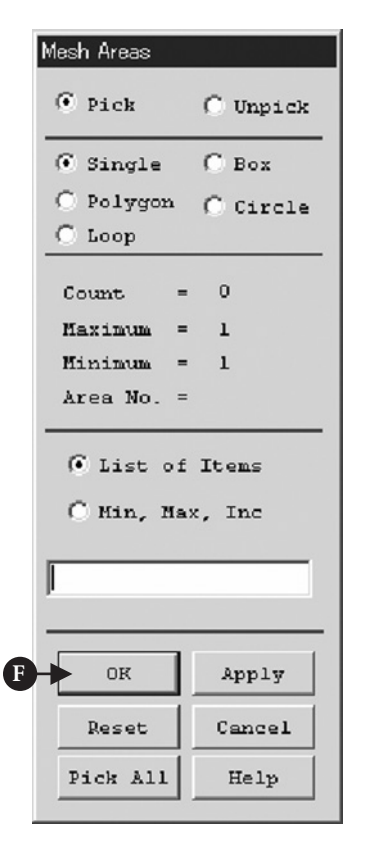


Figure 4.98 Window of **Mesh Areas**.

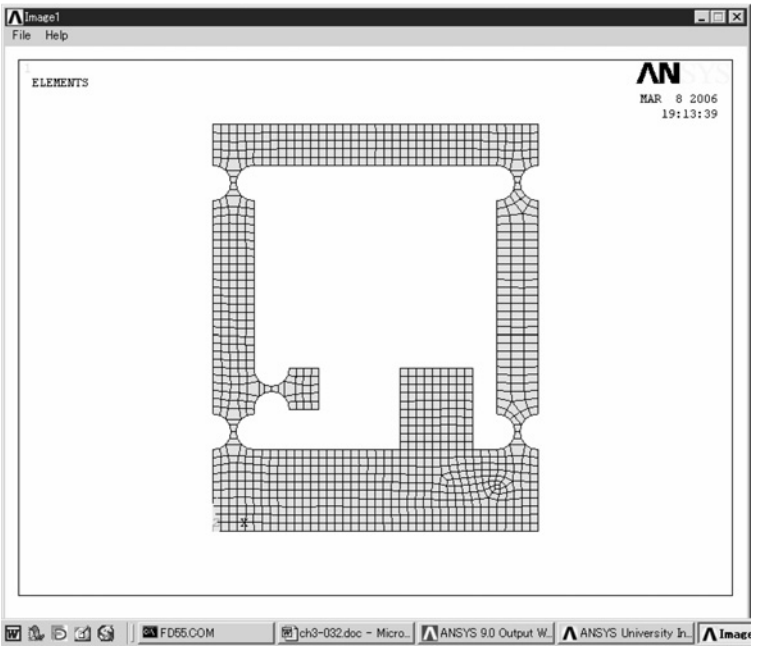


Figure 4.99 **ANSYS Graphics** window.

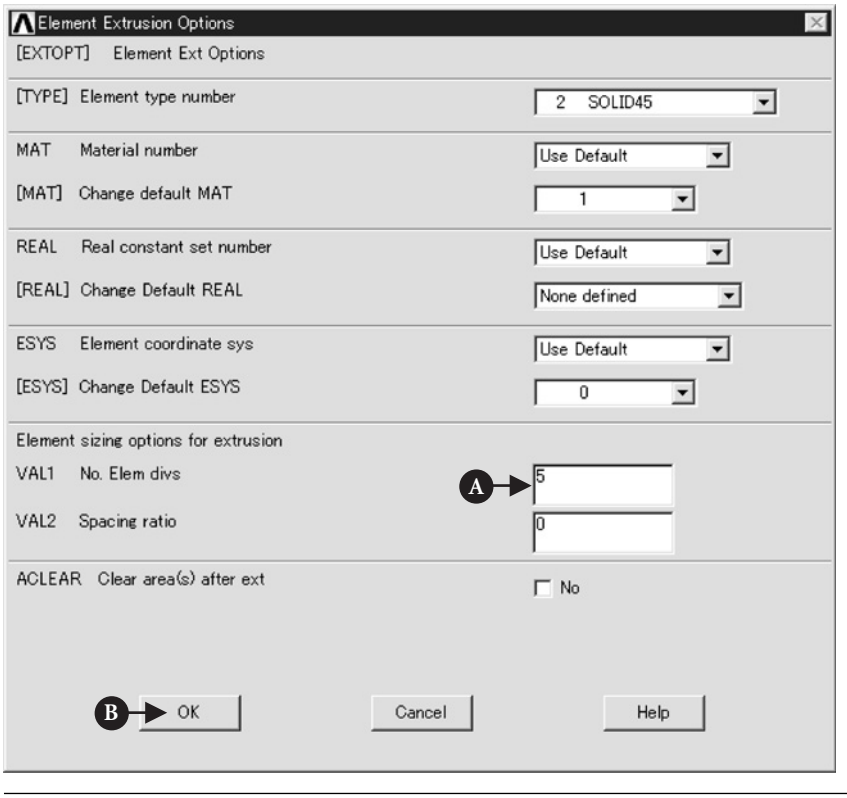


Figure 4.100 Window of **Element Extrusion Options**.

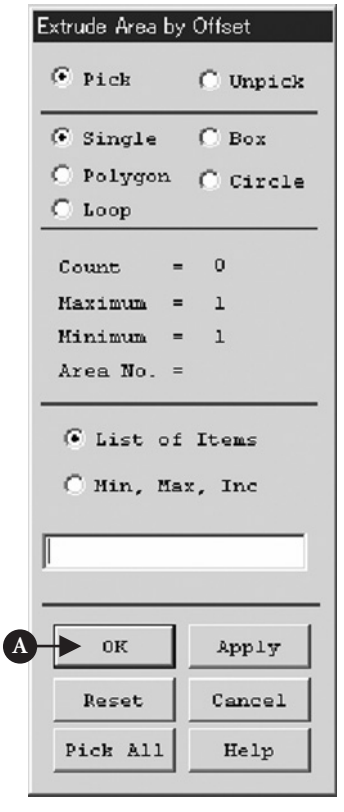


Figure 4.101 Window of **Extrude Area by Offset**.

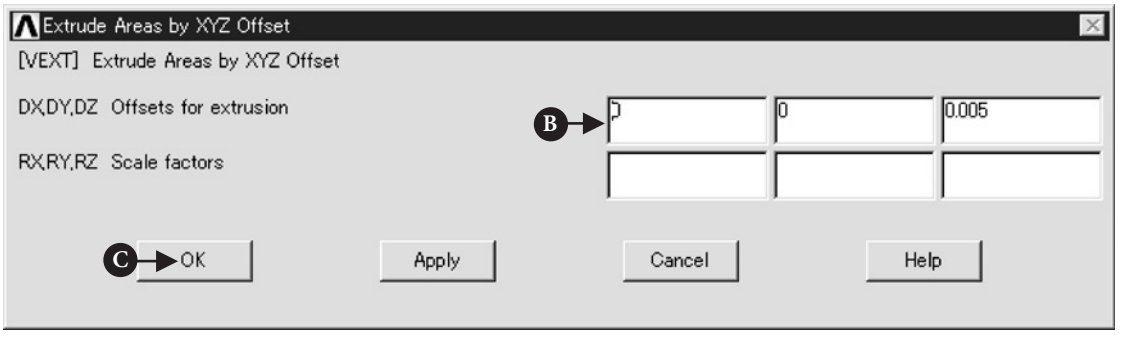


Figure 4.102 Window of **Extrude Areas by XYZ Offset**.

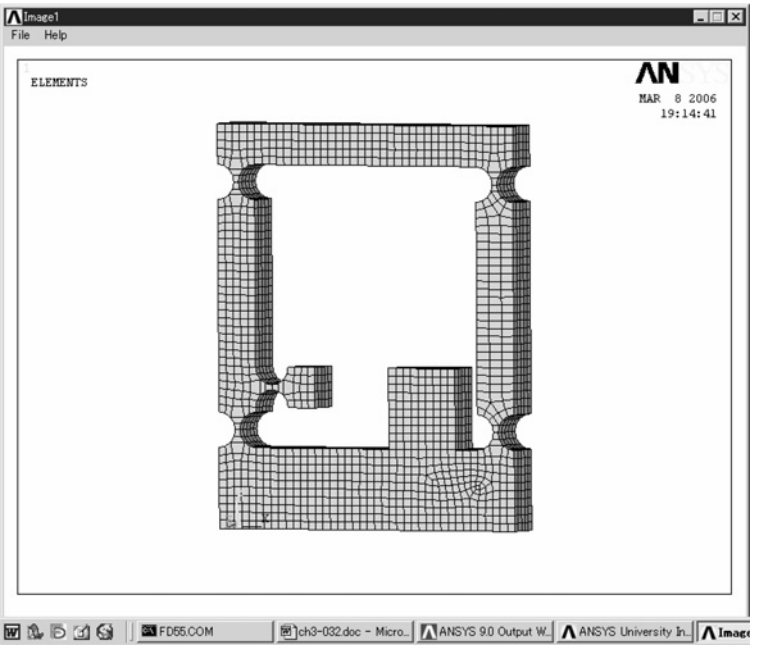


Figure 4.103 **ANSYS Graphics** window.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Areas**

The window **Apply U,ROT on Areas** opens (Figure 4.104).

(1) Pick [A] the side wall for a piezoelectric actuator in Figure 4.105 and [B] the bottom of the table. Then click [C] **OK** button and the window **Apply U,ROT on Areas** opens (Figure 4.106).

(2) Select [D] **All DOF** in the box of **Lab2** and, then, click [E] **OK** button. After these steps **ANSYS Graphics** window is changed as shown in Figure 4.107.

4.4.3 ANALYSIS

4.4.3.1 DEFINE THE TYPE OF ANALYSIS

The following steps are performed to define the type of analysis.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → New Analysis**

The window **New Analysis** opens (Figure 4.108).

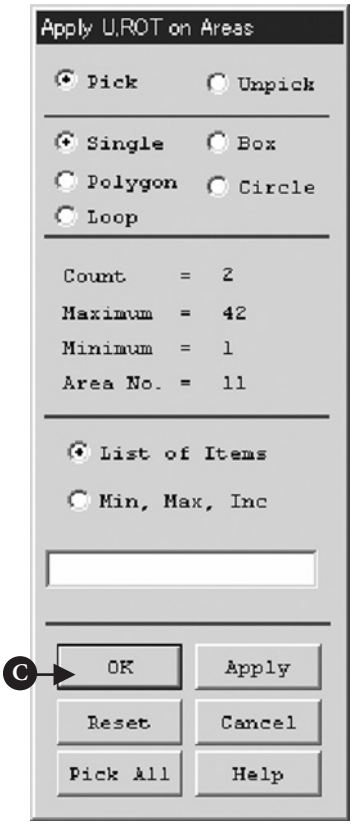


Figure 4.104 Window of **Apply U,ROT on Areas**.

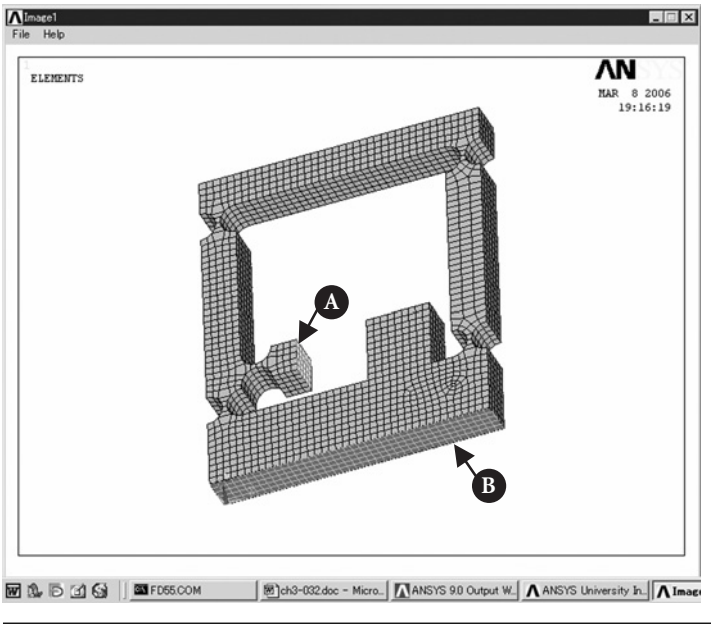


Figure 4.105 **ANSYS Graphics** window.

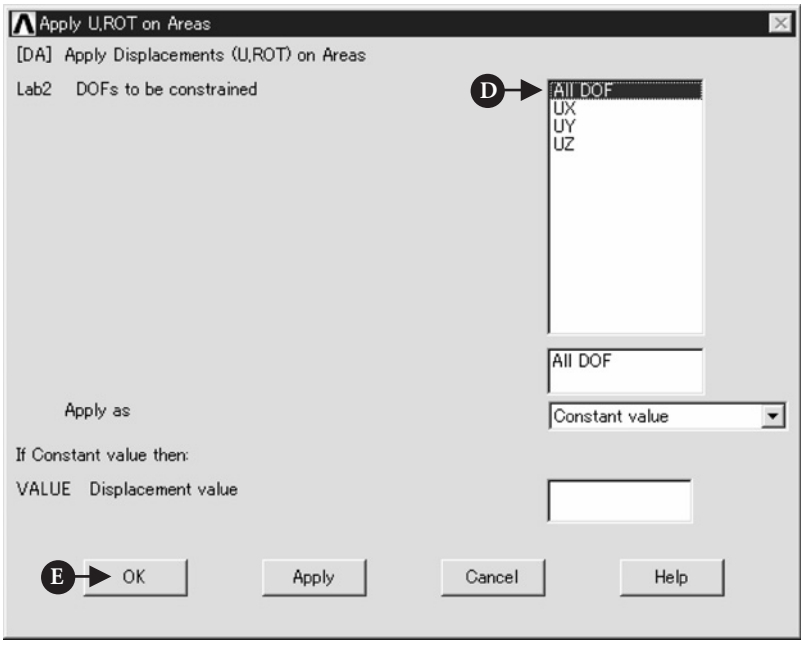


Figure 4.106 Window of **Apply U,ROT on Areas**.

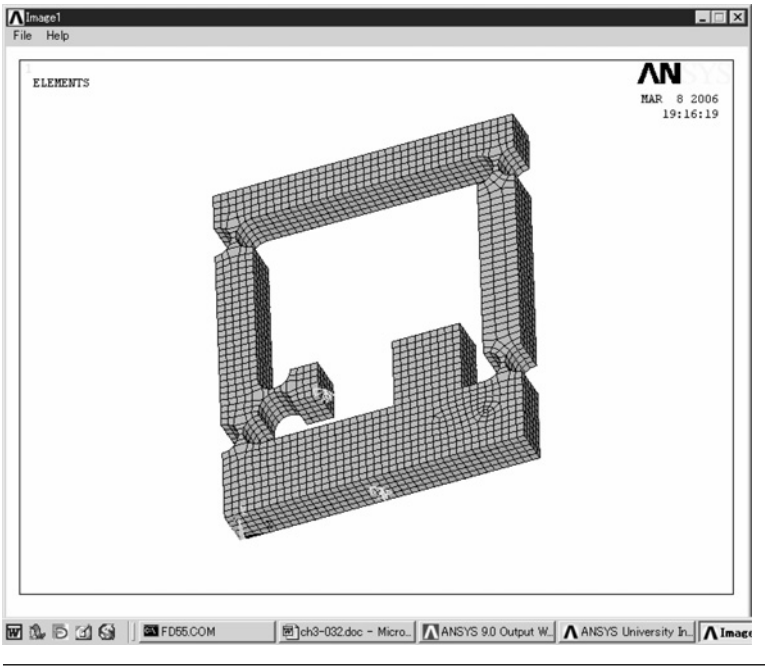


Figure 4.107 **ANSYS Graphics** window.

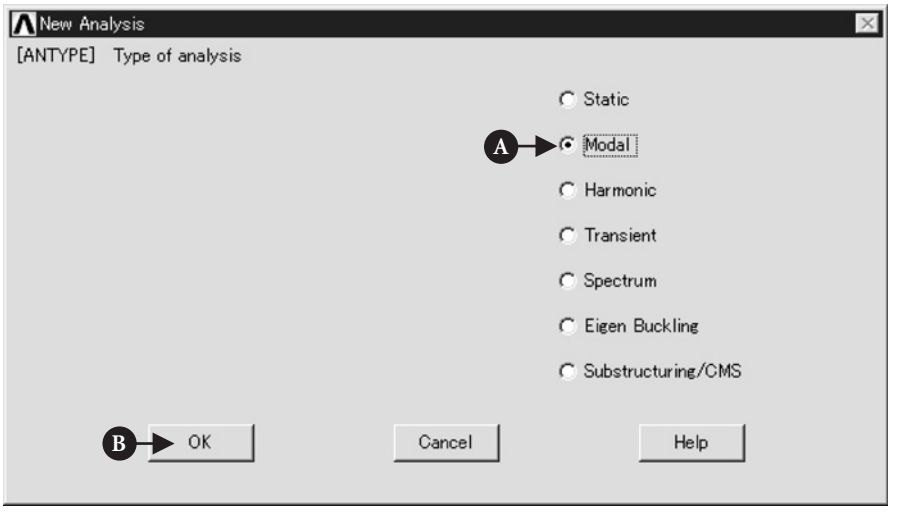


Figure 4.108 Window of **New Analysis**.

(1) Check [A] **Modal** and, then, click [B] **OK** button.

In order to define the number of modes to extract, the following steps are performed.

COMMAND | **ANSYS Main Menu → Solution → Analysis Type → Analysis Options**

The window **Modal Analysis** opens (Figure 4.109).

(1) Check [A] **Subspace** of **MODOPT** and input [B] **3** in the box of **No. of modes to extract** and click [C] **OK** button.

(2) Then, the window **Subspace Modal Analysis** as shown in Figure 4.110 opens. Input [D] **5000** in the box of **FREQE** and click [E] **OK** button.

4.4.3.2 EXECUTE CALCULATION

COMMAND | **ANSYS Main Menu → Solution → Solve → Current LS**

The window **Solve Current Load Step** opens.

(1) Click **OK** button and calculation starts. When the window **Note** appears, the calculation is finished.

(2) Click **Close** button and the window is closed. The window **/STATUS Command** is also open but this window can be closed by clicking the mark of **X** at the upper right side of the window.

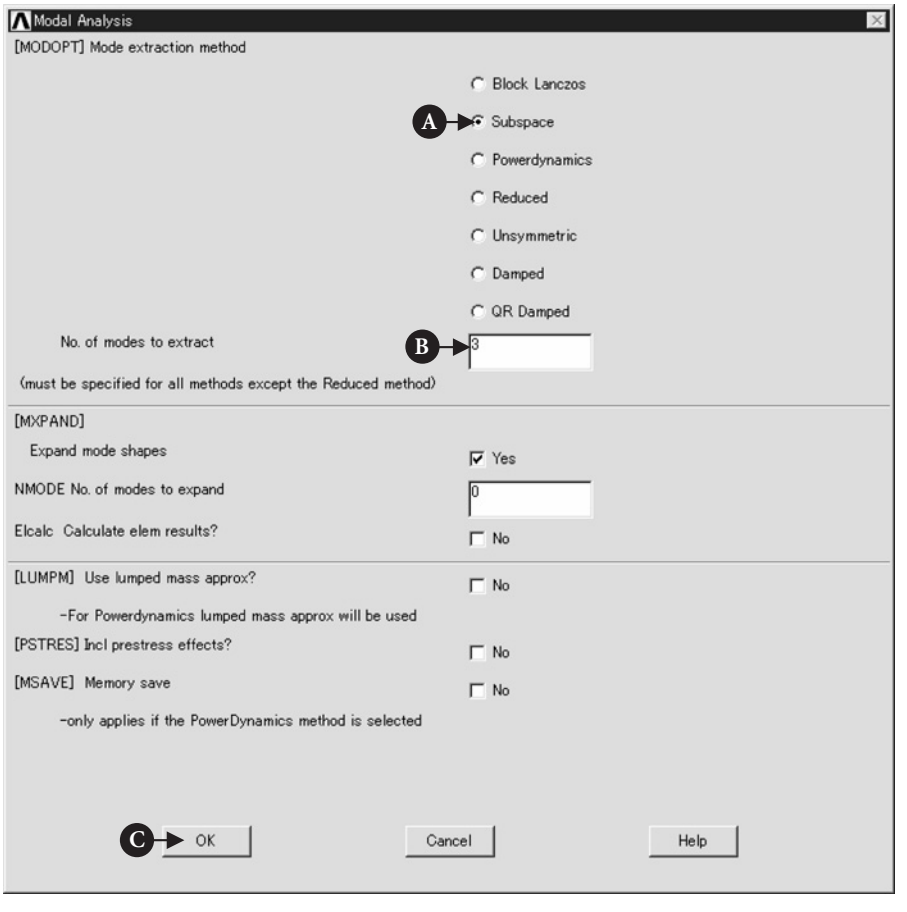


Figure 4.109 Window of **Modal Analysis**.

4.4.4 POSTPROCESSING

4.4.4.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → First Set**

4.4.4.2 PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape**

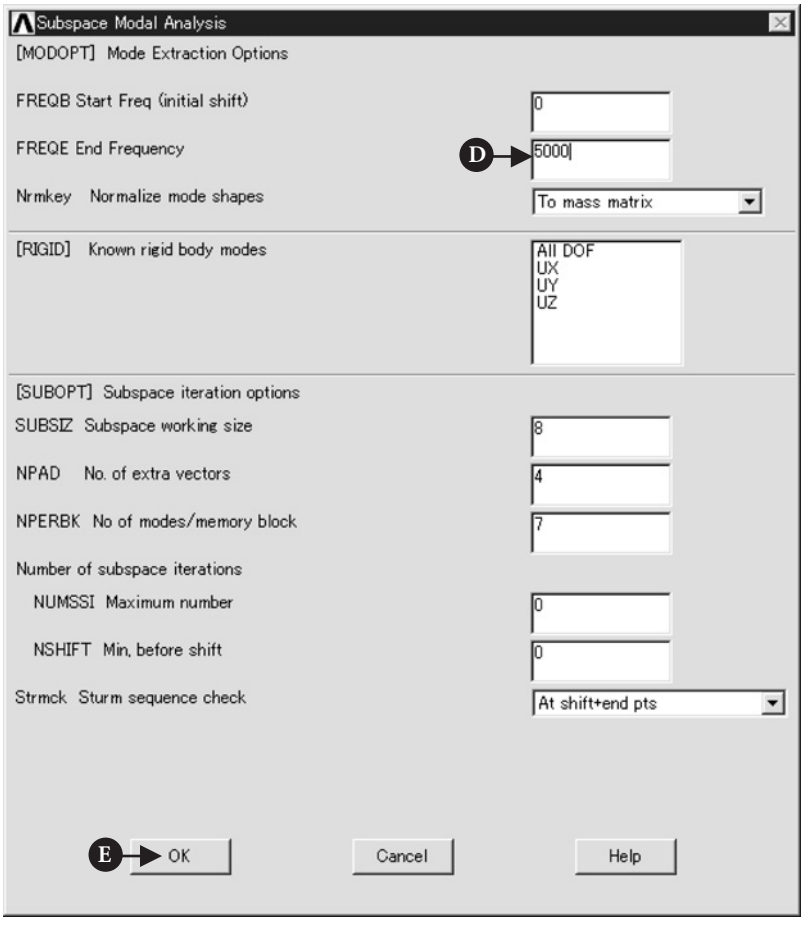


Figure 4.110 Window of **Subspace Modal Analysis**.

The window **Plot Deformed Shape** opens (Figure 4.111).

(1) Select [A] **Def+Undeformed** and click [B] **OK**.

(2) The calculated result for the first mode of vibration appears on **ANSYS Graphics** window as shown in Figure 4.112.

4.4.4.3 READ THE CALCULATED RESULTS OF THE SECOND AND THIRD MODES OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → Next Set**

Perform the same steps described in Section 4.4.4.2 and the results calculated for the higher modes of vibration are displayed as shown in Figures 4.113 and 4.114.

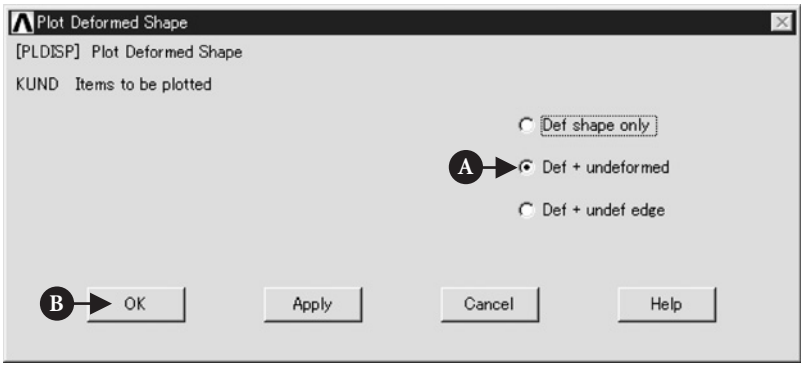


Figure 4.111 Window of **Plot Deformed Shape**.

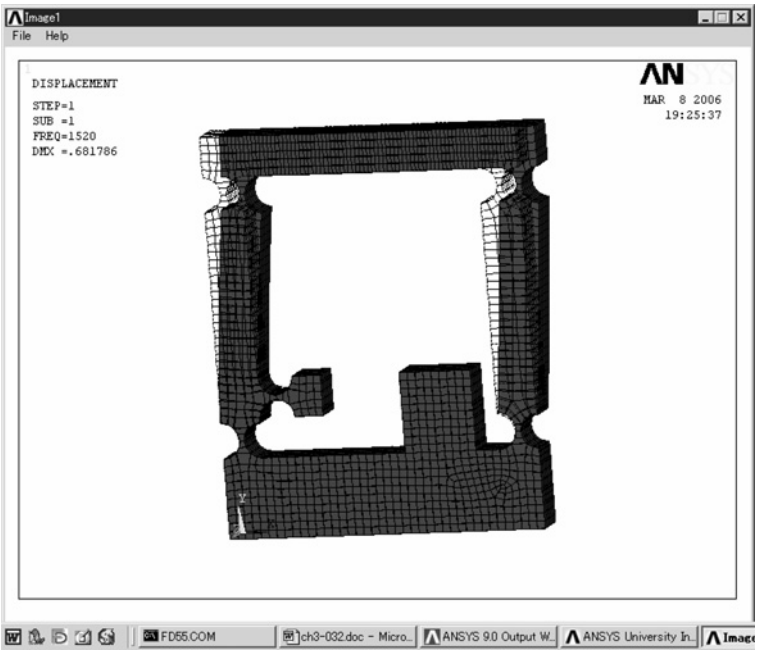


Figure 4.112 **ANSYS Graphics** window for the first mode.

4.4.4.4 ANIMATE THE VIBRATION MODE SHAPE

In order to easily observe the vibration mode shape, the animation of mode shape can be used.

COMMAND | **Utility Menu → PlotCtrls → Animate → Mode Shape**

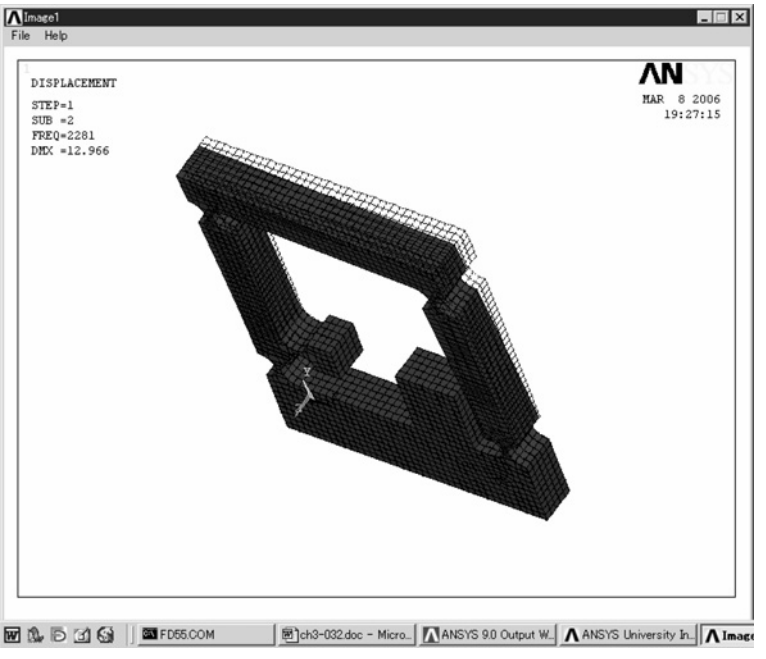


Figure 4.113 **ANSYS Graphics** window for the second mode.

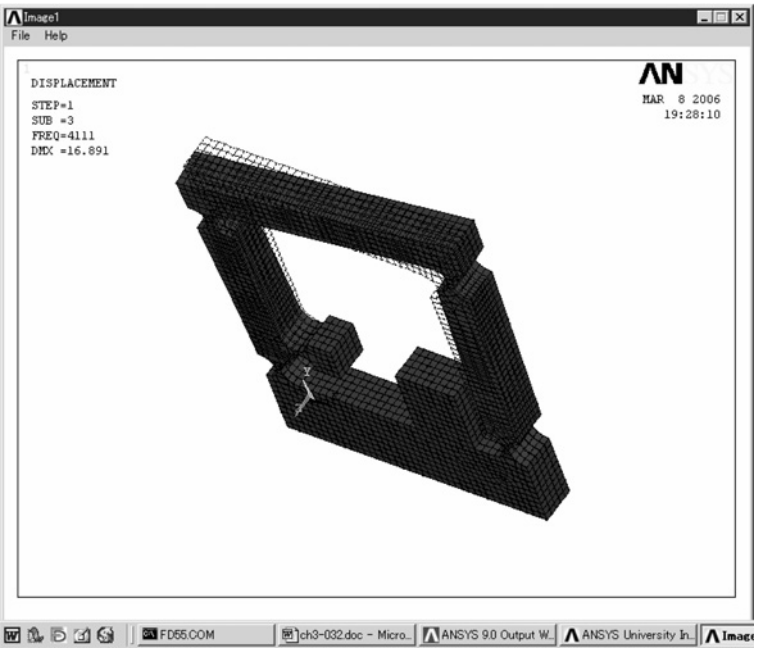


Figure 4.114 **ANSYS Graphics** window for the third mode.

The window **Animate Mode Shape** opens (Figure 4.115).

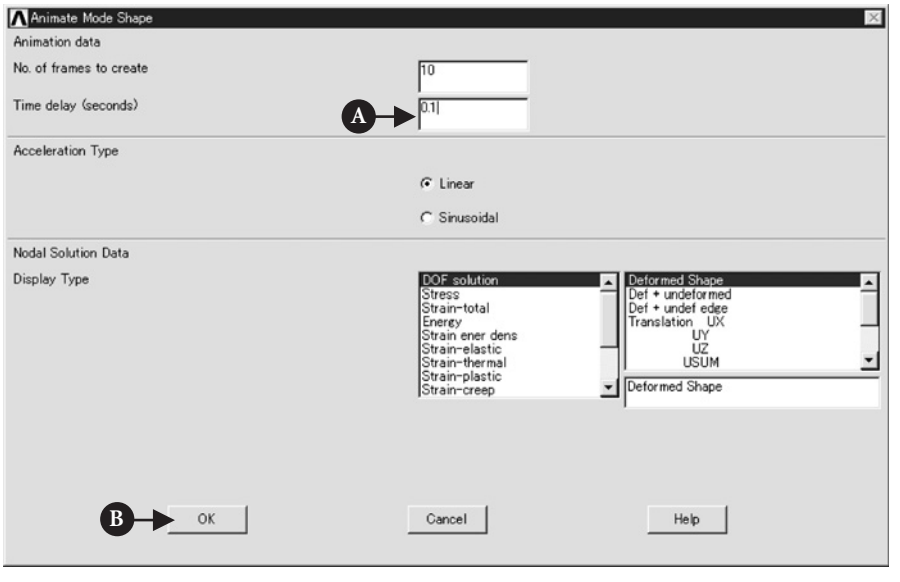


Figure 4.115 Window of **Animate Mode Shape**.

(1) Input [A] **0.1** to **Time delay** box and click [B] **OK** button. Then the animation of the mode shape is displayed in **ANSYS Graphics** window.

This page intentionally left blank

Chapter 5

Analysis for Fluid Dynamics

Chapter outline

5.1 Introduction 215
5.2 Analysis of flow structure in a diffuser 216
5.3 Analysis of flow structure in a channel with a butterfly valve 242

5.1 Introduction

Various fluids such as air and liquid are used as an operating fluid in a blower, a compressor, and a pump. The shape of flow channel often determines the efficiency of these machines. In this chapter, the flow structures in a diffuser and the channel with a butterfly valve are examined by using **FLOTRAN** which is an assistant program of **ANSYS**. A diffuser is usually used for increasing the static pressure by reducing the fluid velocity and the diffuser can be easily found in a centrifugal pump as shown in Figure 5.1.

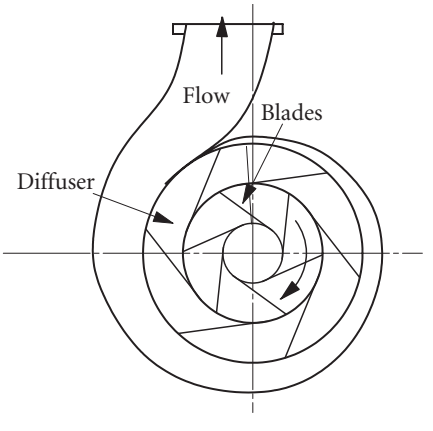


Figure 5.1 Typical machines for fluid.

5.2 Analysis of flow structure in a diffuser

5.2.1 Problem description

Analyze the flow structure of an axisymmetric conical diffuser with diffuser angle $2\theta = 6°$ and expansion ratio = 4 as shown in Figure 5.2.

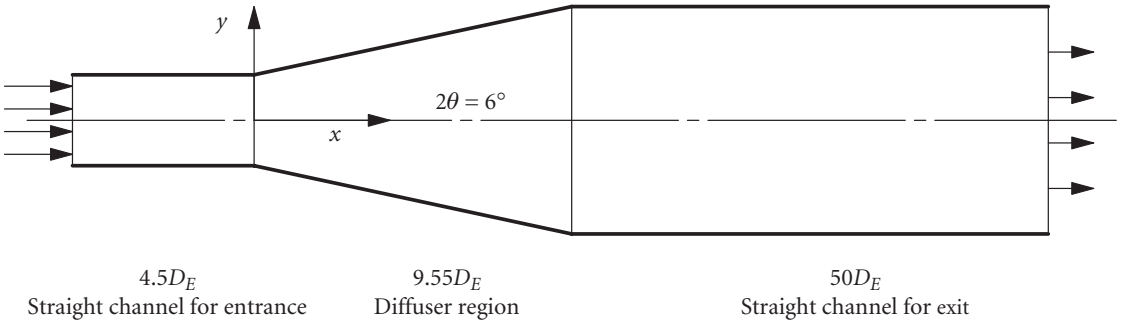


Figure 5.2 Axisymmetrical conical diffuser.

Shape of the flow channel:

(1) Diffuser shape is axisymmetric and conical, diffuser angle $2\theta = 6°$, expansion ratio = 4.
(2) Diameter of entrance of the diffuser: $D_E = 0.2$ m.
(3) Length of straight channel for entrance: $4.5D_E$.
(4) Length of diffuser region: $9.55D_E$.
(5) Length of straight channel for exit: $50.0D_E$.

Operating fluid: Air (300 K)

Flow field: Turbulence

Velocity at the entrance: 20 m/s

Reynolds number: 2.54×10^5 (assumed to set the diameter of the diffuser entrance to a representative length)

Boundary conditions:

(1) Velocities in all directions are zero on all walls.
(2) Pressure is equal to zero at the exit.
(3) Velocity in the y direction is zero on the x-axis.

5.2.2 Create a model for analysis

5.2.2.1 Select kind of analysis

Command | **ANSYS Main Menu → Preferences**

The window **Preferences for GUI Filtering** opens (Figure 5.3).

(1) Check [A] **FLOTRAN CFD** and click [B] **OK** button.

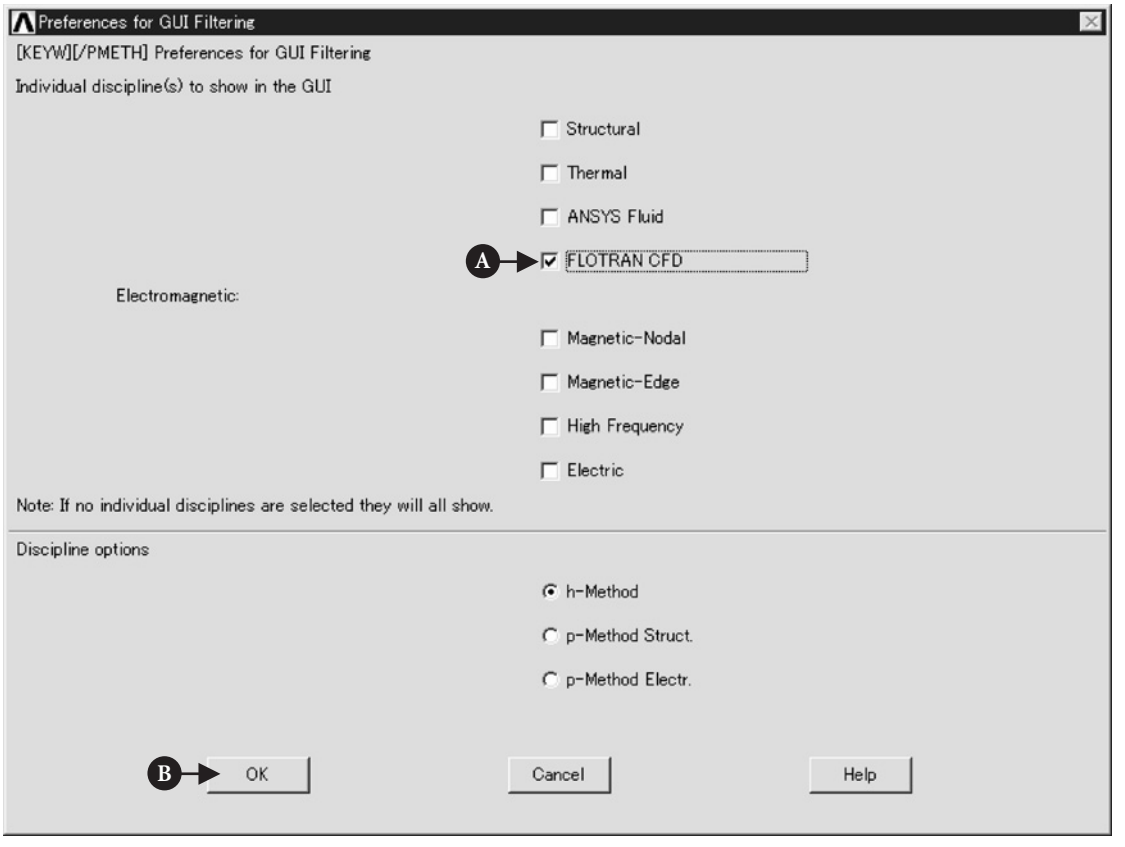


Figure 5.3 Window of **Preferences for GUI Filtering**.

5.2.2.2 ELEMENT TYPE SELECTION

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types** as shown in Figure 5.4 opens.

(1) Click [A] **add**. Then the window **Library of Element Types** as shown in Figure 5.5 opens.

(2) Select [B] **FLOTRAN CFD-2D FLOTRAN 141**.

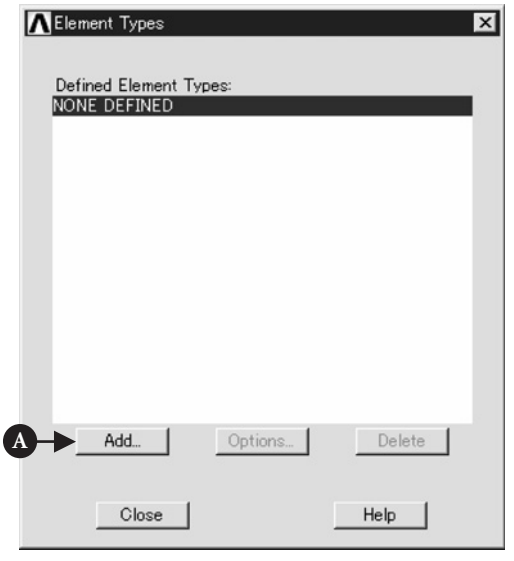


Figure 5.4 Window of **Element Types**.

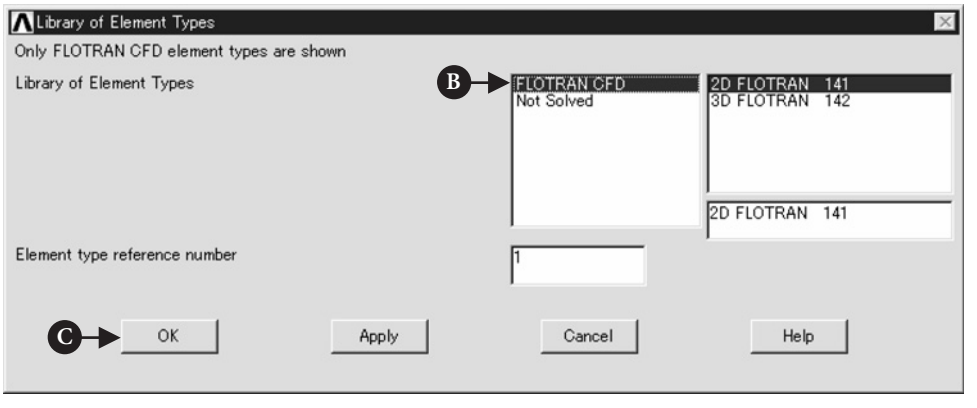


Figure 5.5 Window of **Library of Element Types**.

(3) Click [C] **OK** button and click [D] **Options** button in the window of Figure 5.6.

(4) The window **FLUID141 element type options** opens as shown in Figure 5.7. Select [E] **Axisymm about X** in the box of **Element coordinate system** and click [F] **OK** button. Finally click [G] **Close** button in Figure 5.6.

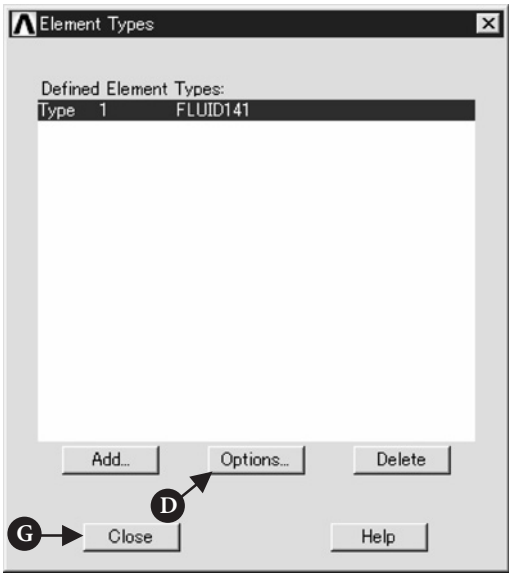


Figure 5.6 Window of **Element Types**.

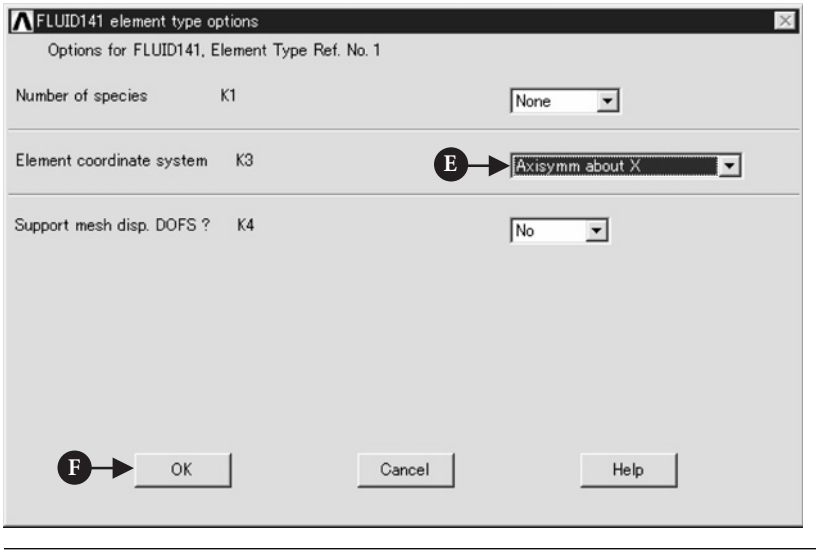


Figure 5.7 Window of **FLUID141 element type options**.

5.2.2.3 CREATE KEYPOINTS

To draw a diffuser for analysis, the method using keypoints on the window are described in this section.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

(1) The window **Create Keypoints in Active Coordinate System** opens (Figure 5.8).

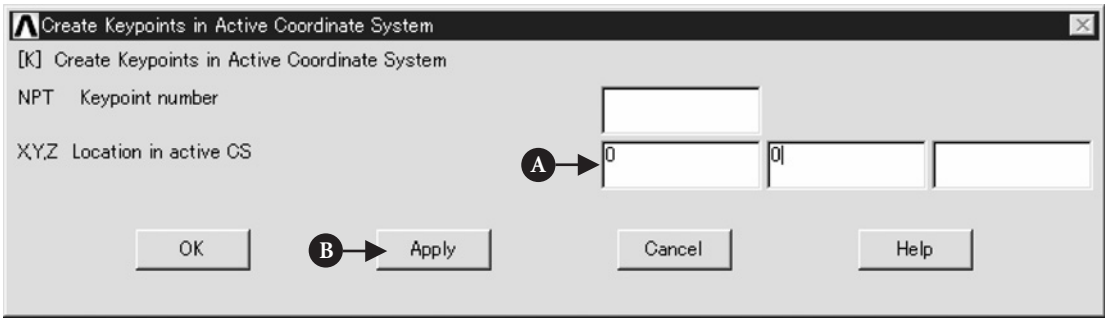


Figure 5.8 Window of **Create Keypoints in Active Coordinate System**.

(2) Input [A] **0, 0** to **X, Y, Z Location in active CS** box, and then click [B] **Apply** button. Do not click **OK** button at this stage. If **OK** button is clicked, the window will be closed. In this case, open the window **Create Keypoints in Active Coordinate System** again and then perform step (2).

(3) In the same window, input the values of keypoints indicated in Table 5.1.

Table 5.1 Coordinates of keypoints

KP No.	X	Y
1	0	0
2	0.9	0
3	0.9	0.1
4	0	0.1
5	2.81	0
6	2.81	0.2
7	12.81	0
8	12.81	0.2

(4) After finishing step (3), eight keypoints appear on the window as shown in Figure 5.9.

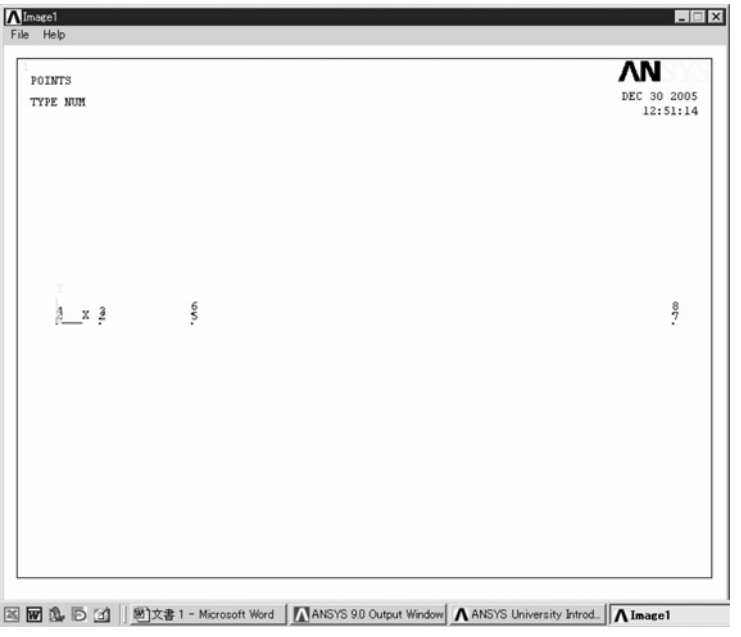


Figure 5.9 **ANSYS Graphics** window.

5.2.2.4 CREATE AREAS FOR DIFFUSER

Areas are created from the keypoints by performing the following steps.

COMMAND **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → Through KPs**

(1) The window **Create Area thru KPs** opens (Figure 5.10).

(2) Pick keypoints **1**, **2**, **3**, and **4** in Figure 5.9 in order and click [A] **Apply** button in Figure 5.10. One area of the diffuser is created on the window.

(3) Then another two areas are made on the window by clicking keypoints listed in Table 5.2.

(4) When three areas are made, click [B] **OK** button in Figure 5.10.

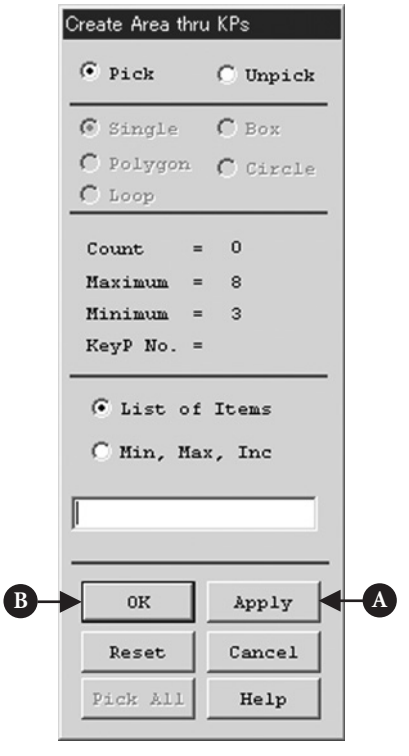


Figure 5.10 Window of **Create Area thru KPs**.

Table 5.2 Keypoint numbers for making areas

Area No.	KPs
1	1, 2, 3, 4
2	2, 5, 6, 3
3	5, 7, 8, 6

5.2.2.5 CREATE MESH IN LINES AND AREAS

COMMAND **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

The window **Mesh Tool** opens (Figure 5.11).

(1) Click [A] **Lines-Set** box. Then the window **Element Size on Picked Lines** opens (Figure 5.12).

(2) Pick **Line 1** and **Line 3** on **ANSYS Graphics** window (Line numbers and Keypoint numbers are indicated in Figure 5.13) and click [B] **OK** button.
The window **Element Sizes on Picked Lines** opens (Figure 5.14).

(3) Input [C] **15** to **NDIV** box and click [D] **OK** button.

(4) Click [A] **Lines-Set** box in Figure 5.11 and pick **Line 2**, Line 6, and Line 9 on **ANSYS Graphics** window. Then click **OK** button.

(5) Input [E] **50** to **NDIV** box and [F] **0.2** to **SPACE** in Figure 5.15. Then click [G] **OK** button. This means that the last dividing space between grids becomes one-fifth of the first dividing space on **Line 2**. When Line 2 was made according to Table 5.2, KP 2 was first picked and then KP 3. So the dividing space of grids becomes smaller toward KP 3.

(6) In order to mesh all lines, input the values listed in Table 5.3.

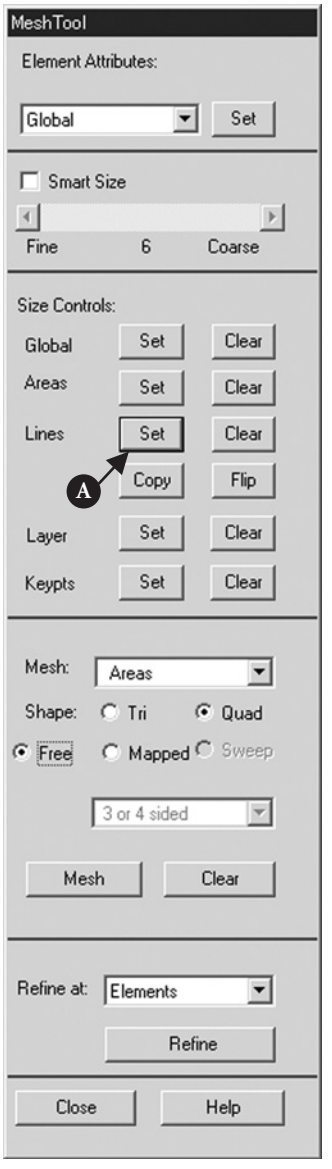


Figure 5.11 Window of **Mesh Tool**.

COMMAND **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

The window **Mesh Tool** opens (Figure 5.16).

(1) Click **Mesh** on the window **Mesh Tool** in Figure 5.11 and the window **Mesh Areas** opens (Figure 5.16). Click [A] **Pick All** button when all areas are divided into elements as seen in Figure 5.17.

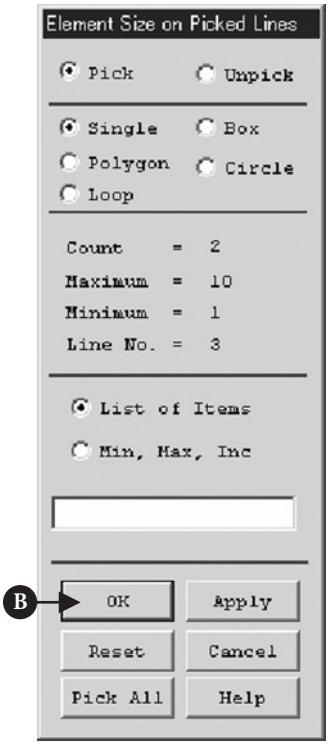


Figure 5.12 Window of **Element Size on Picked Lines**.

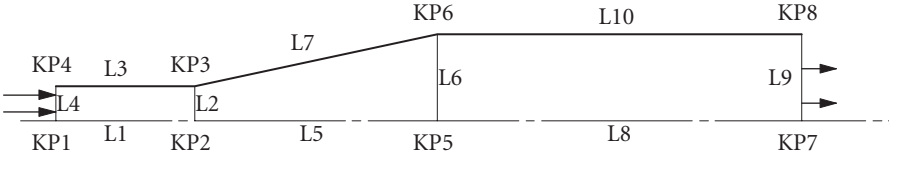


Figure 5.13 Keypoint and Line numbers for a diffuser.

COMMAND | **Utility Menu → PlotCtrls → Pan-Zoom-Rotate**

The window **Pan-Zoom-Rotate** opens (Figure 5.18).

(1) Click [A] **Box Zoom** and [B] make a box on **ANSYS Graphics** window to zoom up the area as shown in Figure 5.17. Then the enlarged drawing surrounded by the box appears on the window as shown in Figure 5.19.

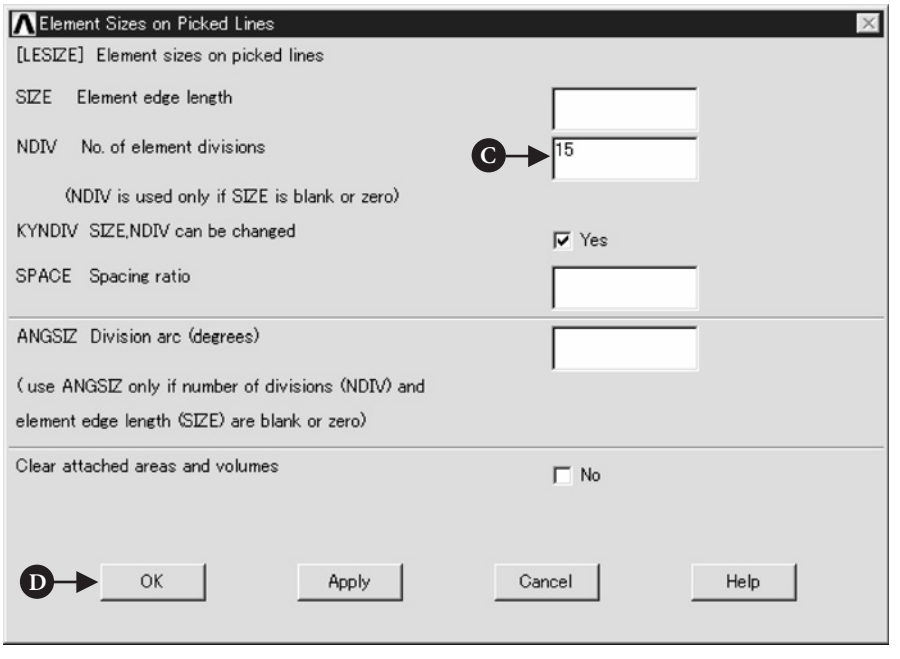


Figure 5.14 Window of **Element Sizes on Picked Lines**.

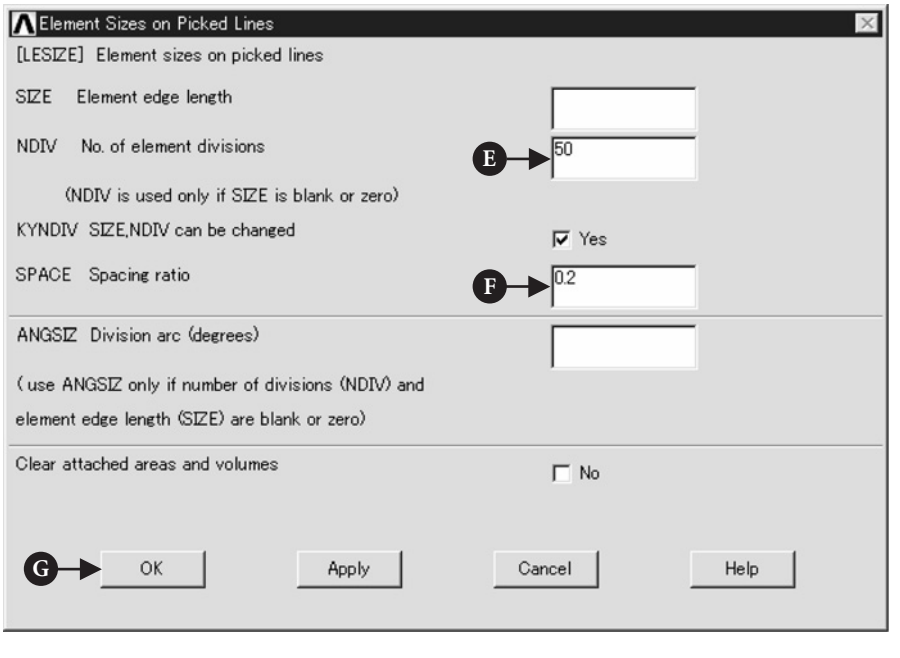


Figure 5.15 Window of **Element Sizes on Picked Lines**.

Table 5.3 Element sizes of picked lines

Line No.	NDIV	SPACE
1	15	
2	50	0.2
3	15	–
4	50	5
5	30	–
6	50	0.2
7	30	–
8	40	5
9	50	0.2
10	40	0.2

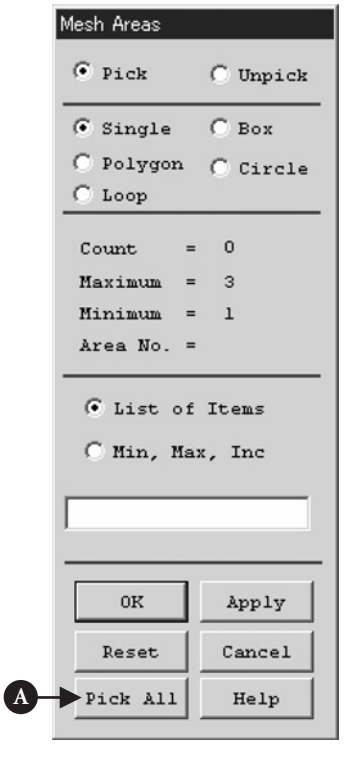


Figure 5.16 Window of **Mesh Areas**.

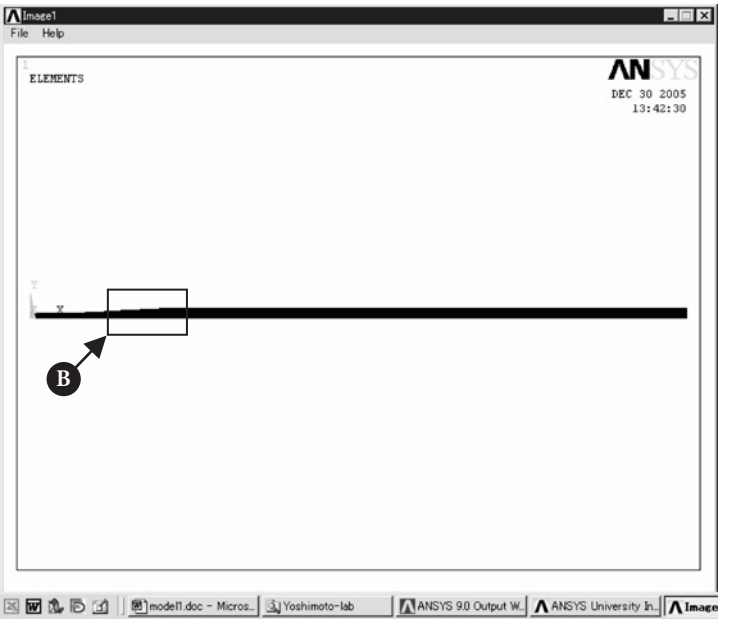


Figure 5.17 **ANSYS Graphics** window.

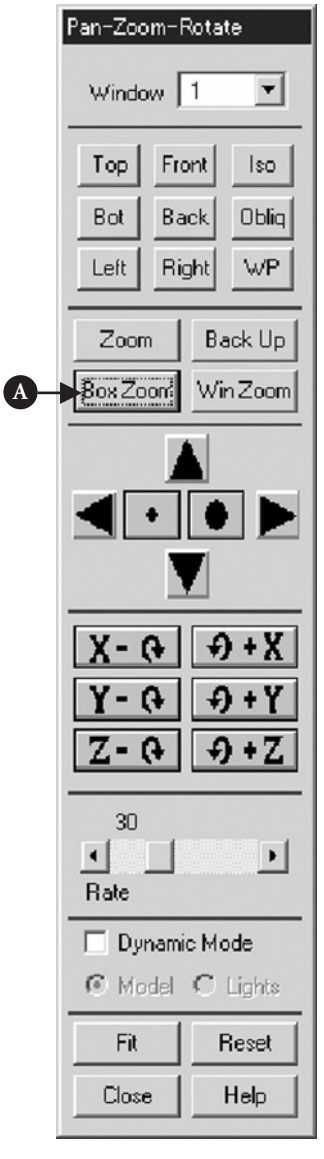


Figure 5.18 Window of **Pan-Zoom-Rotate**.

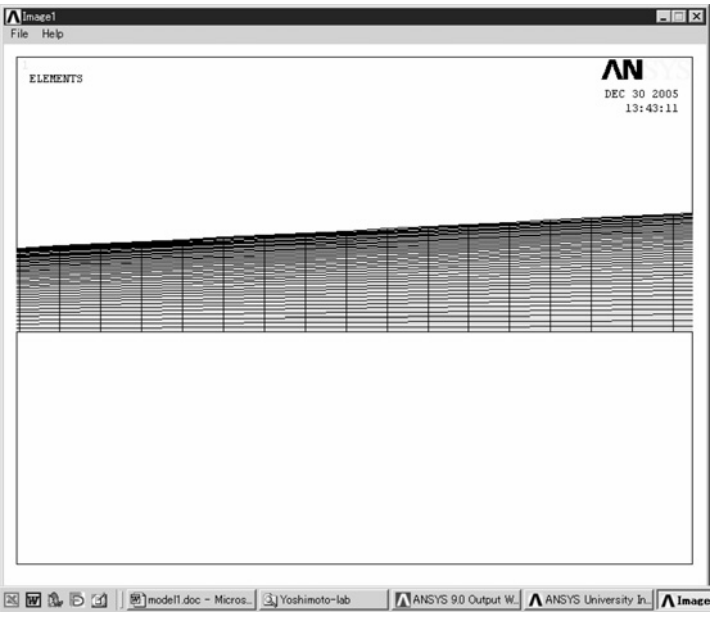


Figure 5.19 **ANSYS Graphics** window.

5.2.2.6 BOUNDARY CONDITIONS

In this section, boundary conditions listed in problem description (Section 5.2.1) are set. By performing the steps mentioned below, all lines need to appear on the window. Then set the boundary conditions.

COMMAND | **Utility Menu → Plot → Lines**

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Velocity → On Lines**

The window **Apply V on Lines** opens (Figure 5.20).

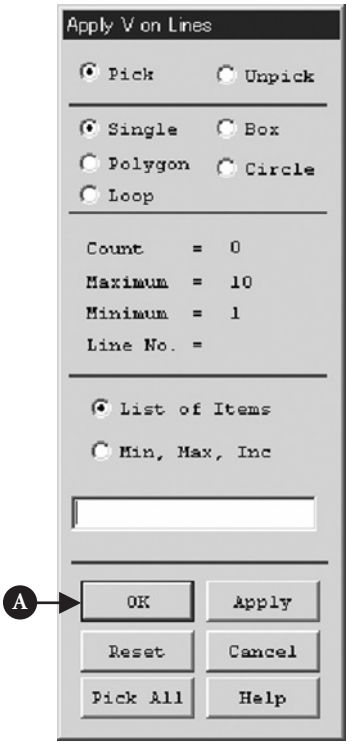


Figure 5.20 Window of **Apply V on Lines**.

(1) Pick **Line 4** at the entrance of the diffuser and click [A] **OK** button. Then the window **Apply VELO load on lines** as shown in Figure 5.21 opens.

(2) Input [B] **20** to **VX** box and [C] **0** to **VY** box. Then, click [D] **OK** button. The sign of setting boundary condition appears on **ANSYS Graphics** window as shown in Figure 5.22.

(3) Set boundary conditions to all lines listed in Table 5.4.

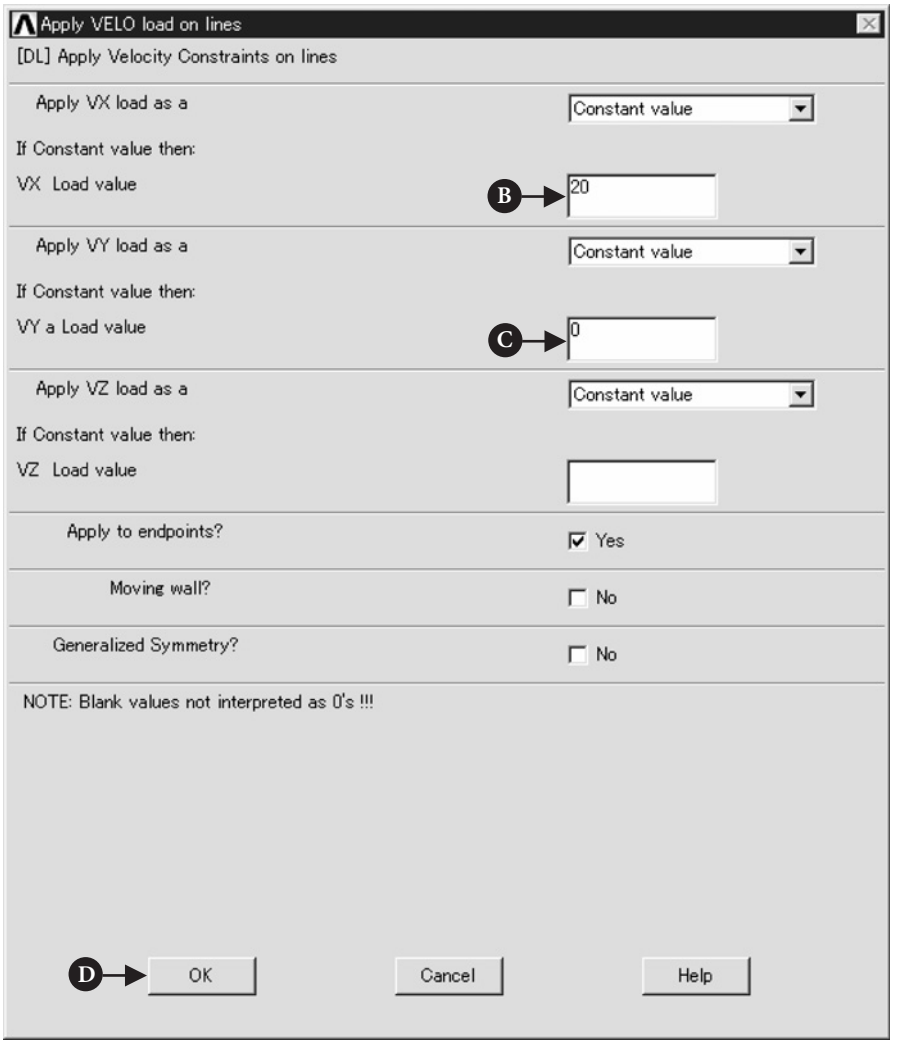


Figure 5.21 Window of **Apply VELO load on lines**.

COMMAND **ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Pressure DOF → On Lines**

The window **Apply PRES on Lines** opens (Figure 5.23).

(1) Pick **Line 9** on **ANSYS Graphics** window and click [A] **OK** button. Then the window **Apply PRES on lines** opens (Figure 5.24).

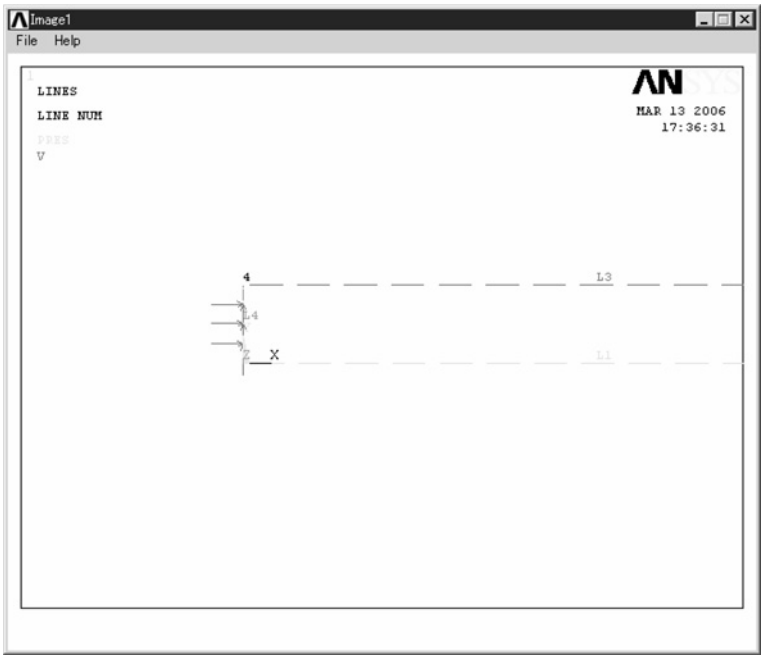


Figure 5.22 **ANSYS Graphics** window.

Table 5.4 Boundary conditions for all lines

Line No.	VX	VY
1	–	0
2	–	–
3	0	0
4	20	0
5	–	0
6	–	–
7	0	0
8	–	0
9	–	–
10	0	0

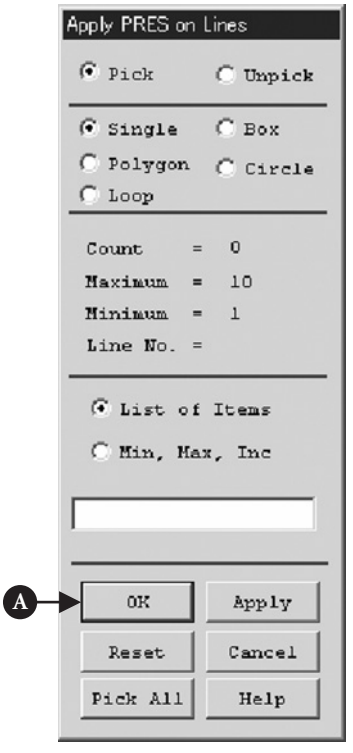


Figure 5.23 Window of **Apply PRES on lines**.

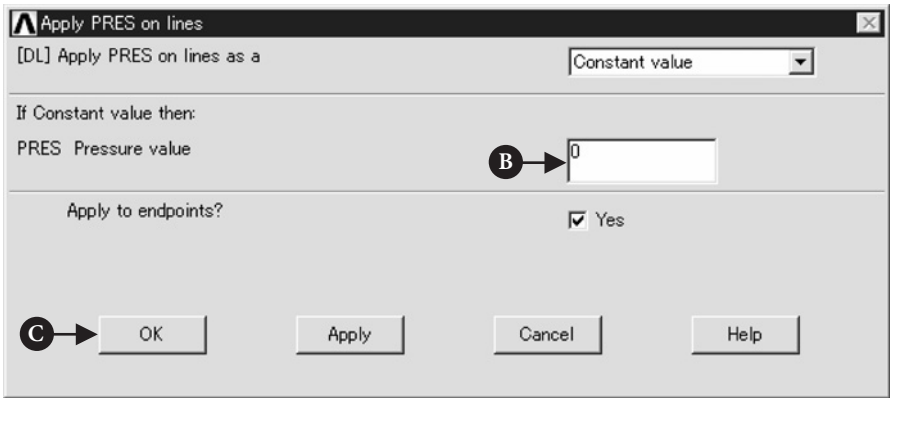


Figure 5.24 Window of **Apply PRES on lines**.

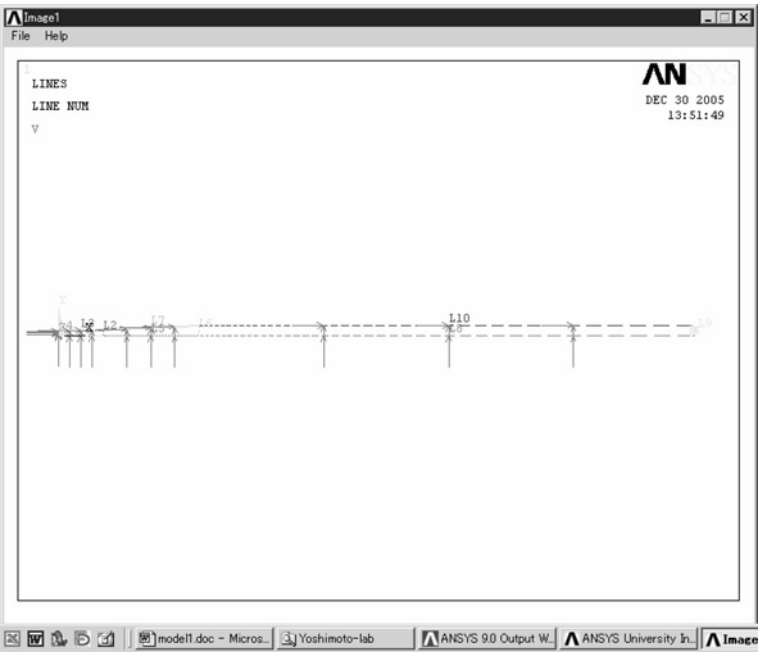


Figure 5.25 **ANSYS Graphics** window.

(2) Input [B] **0** to **PRES** box and close the window by clicking [C] **OK** button. The drawing on **ANSYS Graphics** window is displayed as shown in Figure 5.25.

5.2.3 EXECUTION OF THE ANALYSIS

5.2.3.1 FLOTRAN SET UP

The following steps are performed to set up the analysis of **FLOTRAN**.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Solution Options**

The window **FLOTRAN Solution Options** opens (Figure 5.26).

(1) Set [A] **Adiabatic**, **Turbulent**, and **Incompressible** to **TEMP**, **TURB**, and **COMP** boxes, and, then, click [B] **OK** button.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Execution Ctrl**

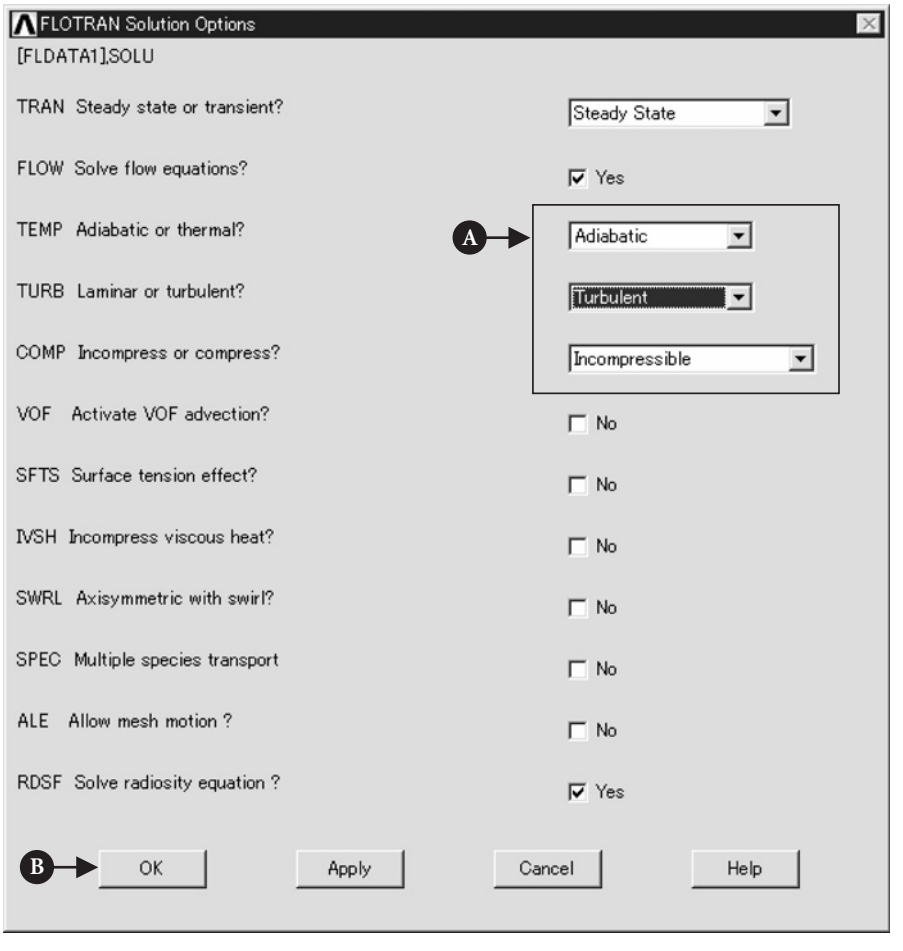


Figure 5.26 Window of **FLOTRAN Solution Options**.

The window **Steady State Control Settings** opens (Figure 5.27).

(1) Input [A] **200** to **EXEC** box and click [B] **OK** button.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Fluid Properties**

The window **Fluid Properties** opens (Figure 5.28).

(1) Select [A] **AIR-SI** in **Density**, **Viscosity**, **Conductivity**, and **Specific heat** boxes and click [B] **OK** button. Then the window **CFD Fluid Properties** as shown in Figure 5.29 opens and click [C] **OK** button.

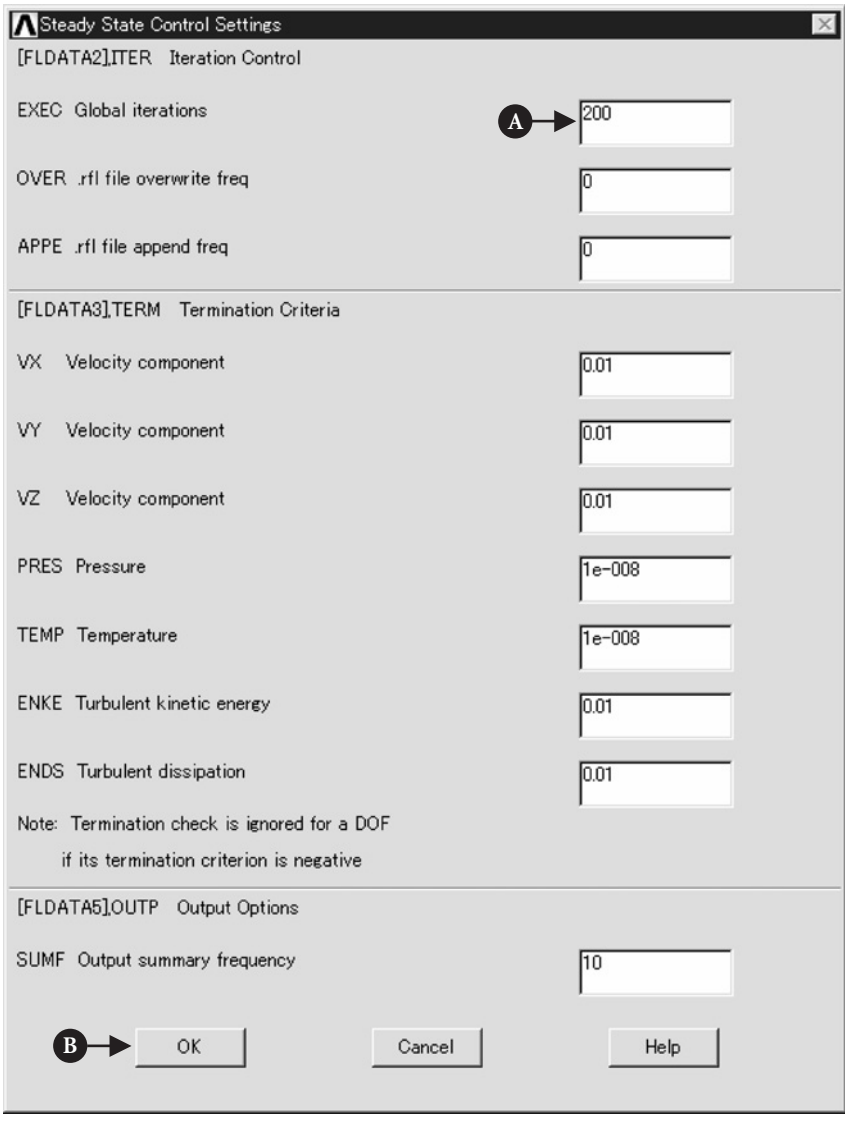


Figure 5.27 Window of **Steady State Control Settings**.

5.2.4 EXECUTE CALCULATION

COMMAND | **ANSYS Main Menu → Solution → Run FLOTRAN**

When the calculation is finished, the window **Note** as shown in Figure 5.30 opens.

(1) Click [A] **Close** button.

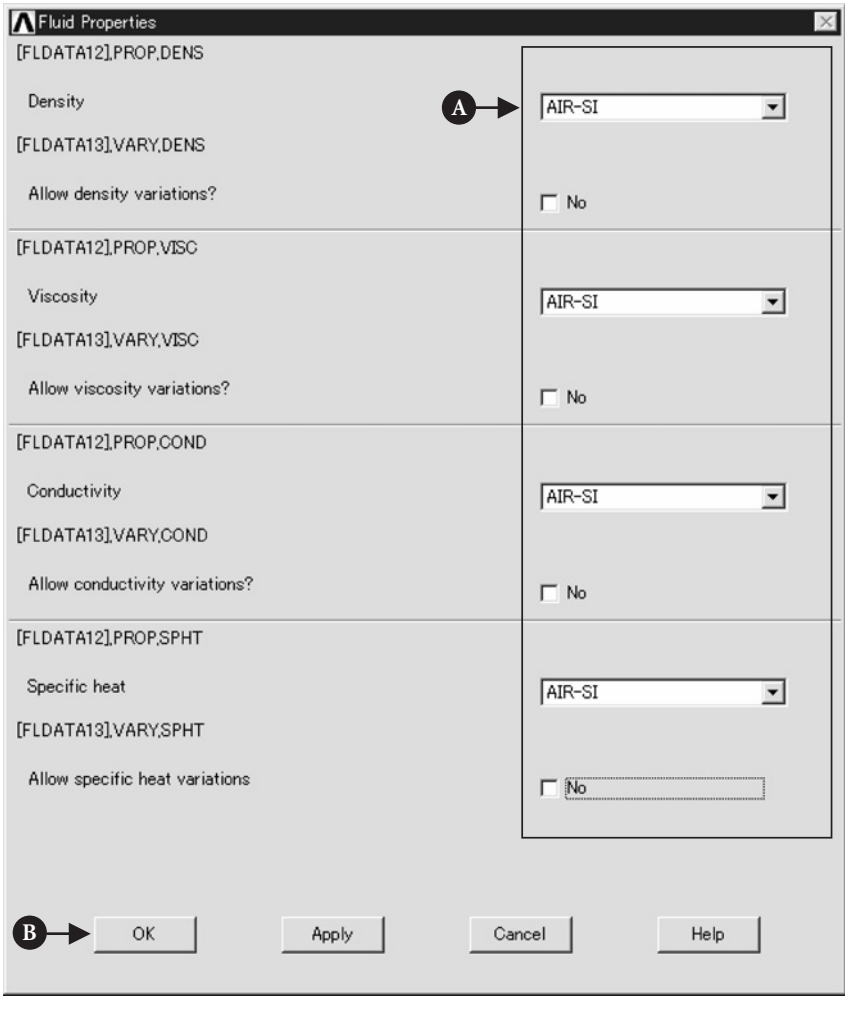


Figure 5.28 Window of **Fluid Properties**.

5.2.5 POSTPROCESSING

5.2.5.1 READ THE CALCULATED RESULTS OF THE FIRST MODE OF VIBRATION

COMMAND | **ANSYS Main Menu → General Postproc → Read Results → Last Set**

5.2.5.2 PLOT THE CALCULATED RESULTS

COMMAND | **ANSYS Main Menu → General Postproc → Plot Results → Vector Plot → Predefined**

Figure 5.29 Window of **CFD Fluid Properties**.

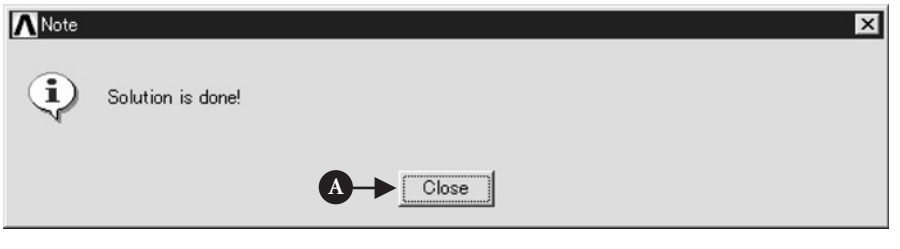


Figure 5.30 Window of **Note**.

The window **Vector Plot of Predefined Vectors** as shown in Figure 5.31 opens.

(1) Select [A] **DOF solution – Velocity V** and click [B] **OK**.

(2) The calculated result for velocity vectors appears on **ANSYS Graphics** window as shown in Figure 5.32.

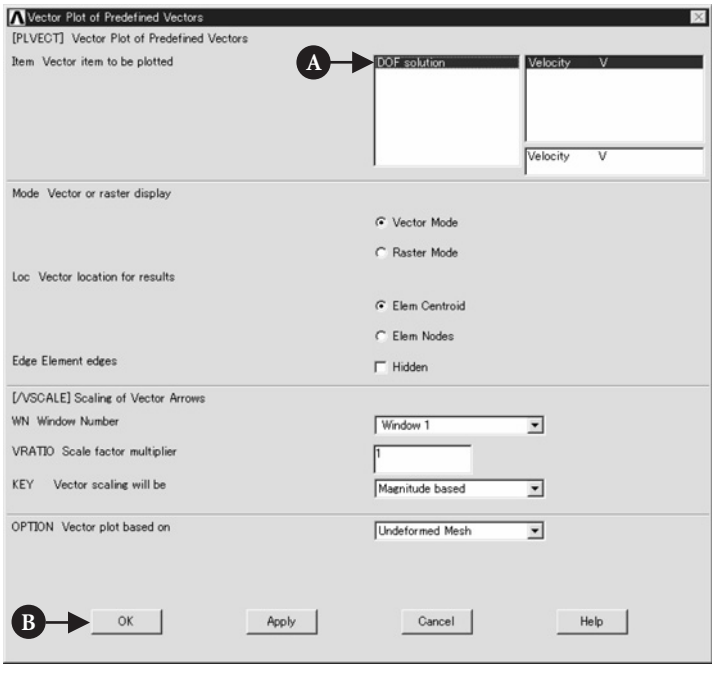


Figure 5.31 Window of **Vector Plot of Predefined Vectors**.

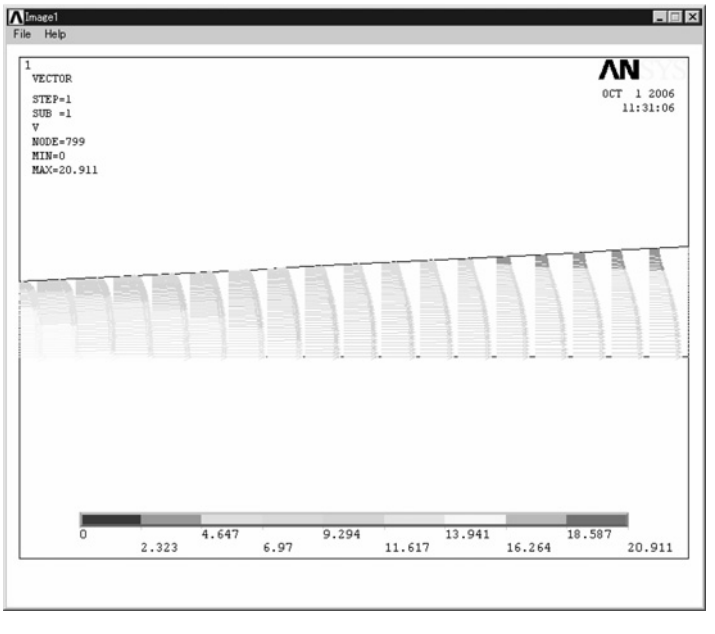


Figure 5.32 **ANSYS Graphics** window.

5.2.5.3 PLOT THE CALCULATED RESULTS BY PATH OPERATION

In order to plot the calculated results for a selected cross section, **Path Operation** command is used.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Define Path → By Nodes**

The window **By Nodes** opens (Figure 5.33).

(1) Check [A] **Pick** and pick two nodes on [B] the wall and [C] the x-axis as shown in Figure 5.34. If a wrong node on **ANSYS Graphics** window is selected, delete the picked node by using **Unpick**. Click [D] **OK** button.

(2) The window **By Nodes** as shown in Figure 5.35 opens. Input the path name [E] **vel01** to **Name** box [F] 50 to nDiv box and click [G] **OK** button. Then the window **PATH Command** appears as shown in Figure 5.36, which explains the content of the defined path. Close this window by clicking **X** mark at the upper right end.

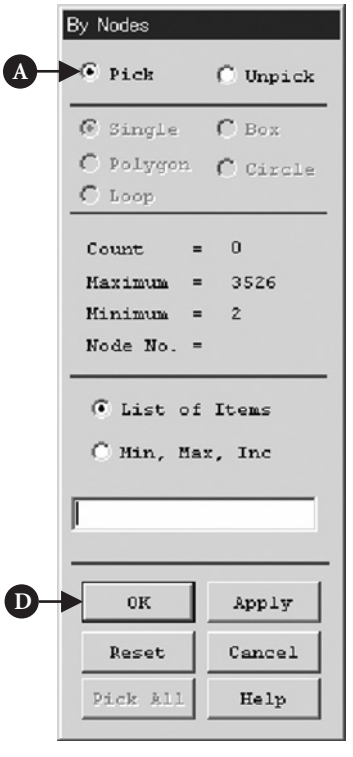


Figure 5.33 Window of **By Nodes**.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Map onto Path**

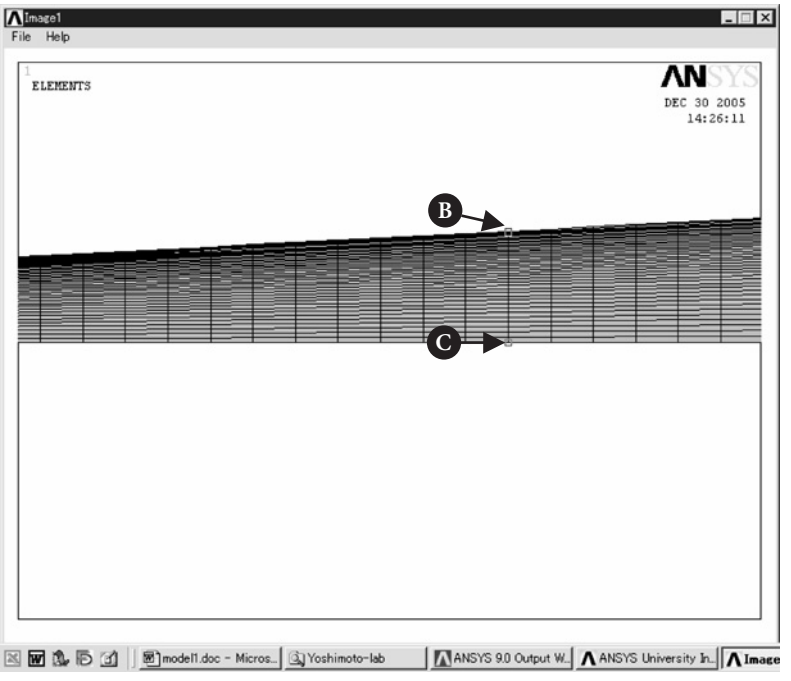


Figure 5.34 **ANSYS Graphics** window.

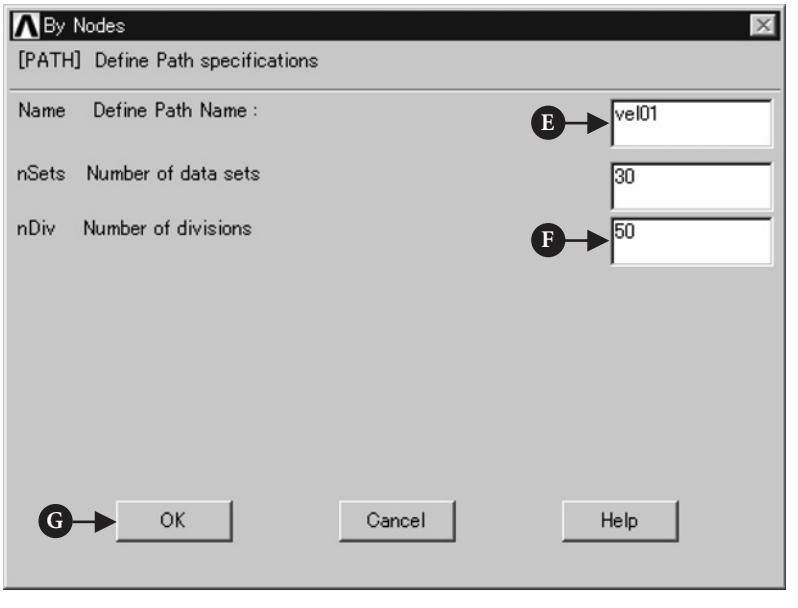


Figure 5.35 Window of **By Nodes**.

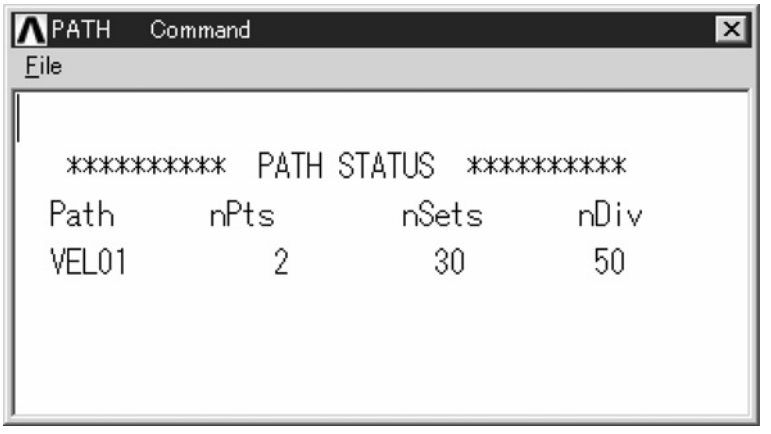


Figure 5.36 Window of **PATH Command**.

The window **Map Result Items onto Path** opens (Figure 5.37).

(1) Input [A] **vel01** to **Lab** box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

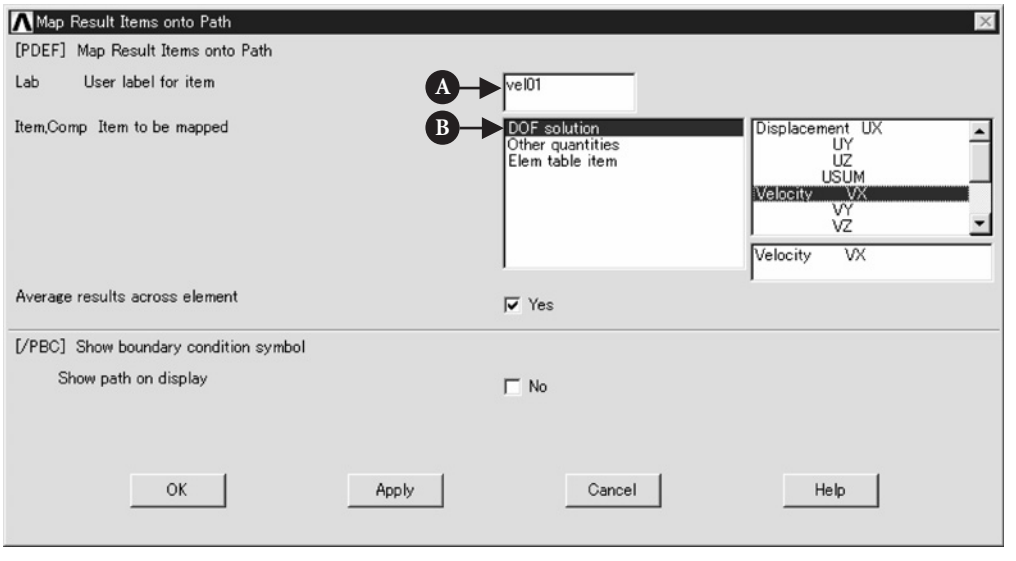


Figure 5.37 Window of **Map Result Items onto Path**.

COMMAND **ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph**

The window **Plot of Path Items on Graph** opens (Figure 5.38).

(1) Select [A] **VEL01** in **Lab1-6** box and click **OK** button. Then the calculated result for the defined path appears on **ANSYS Graphics** window as shown in Figure 5.39.

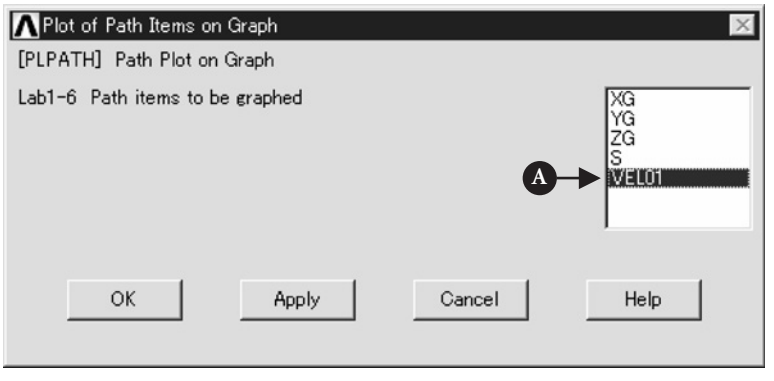


Figure 5.38 Window of **Plot of Path Items on Graph**.

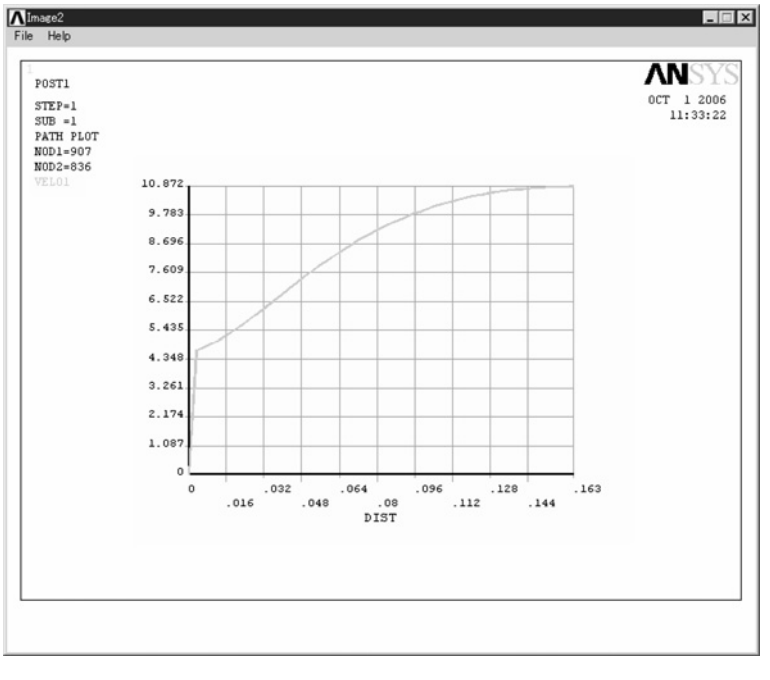


Figure 5.39 **ANSYS Graphics** window.

In addition, when the values of the defined path are needed, the following steps can be used.

COMMAND | **ANSYS Main Menu → General Postproc → List Results → Path Items**

The window **List of Path Items** opens (Figure 5.40).

(1) Select [A] **VEL01** in **Lab1-6** box and click **OK** button. Then the list for the defined path appears as shown in Figure 5.41.

Figure 5.40 Window of **List Path Items**.

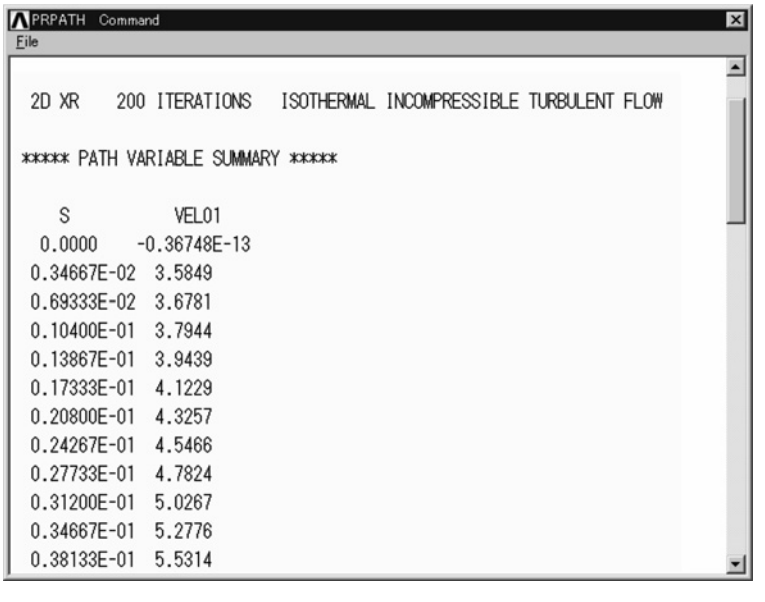


Figure 5.41 Window of **PRPATH Command**.

5.3 Analysis of flow structure in a channel with a butterfly valve

5.3.1 Problem description

Analyze the flow structure around a butterfly valve as shown in Figure 5.42.

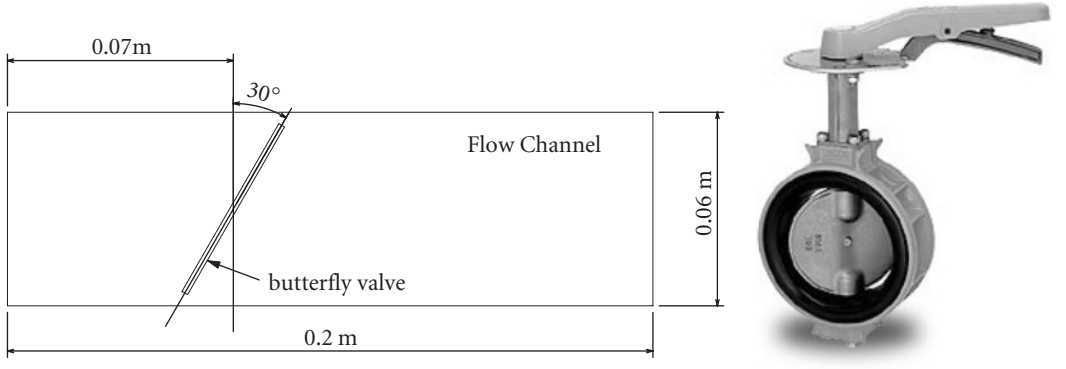


Figure 5.42 Flow channel with a butterfly valve (photo is quoted from http://www.kitz.co.jp/product/bidg-jutaku/kyutouyou/index.html).

Shape of the flow channel:

(1) Diameter of the flow channel: $D_E = 0.06$ m.
(2) Diameter of a butterfly valve: $D_E = 0.06$ m.
(3) Tilt angle of a butterfly valve: $\alpha = 30°$.
Operating fluid: water
Flow field: turbulent
Velocity at the entrance: 0.01 m/s

Boundary conditions:

(1) Velocities in all directions are zero on all walls.
(2) Pressure is equal to zero at the exit.
(3) Velocity in the y direction is zero on the x-axis.

5.3.2 Create a model for analysis

5.3.2.1 Select kind of analysis

Command | **ANSYS Main Menu → Preferences**

5.3.2.2 SELECT ELEMENT TYPE

COMMAND | **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

Then the window **Element Types** opens.

(1) Click **add** and then the window **Library of Element Types** opens.

(2) Select **FLOTRAN CFD-2D FLOTRAN 141**. Click **OK** button.

(3) Click **Options** button in the window **Element Types**.

(4) The window **FLUID141 element type options** as shown in Figure 5.43 opens. Select [A] **Cartesian** in the box of **Element coordinate system** and click **OK** button. Finally click **Close** button in the window **Element Types**.

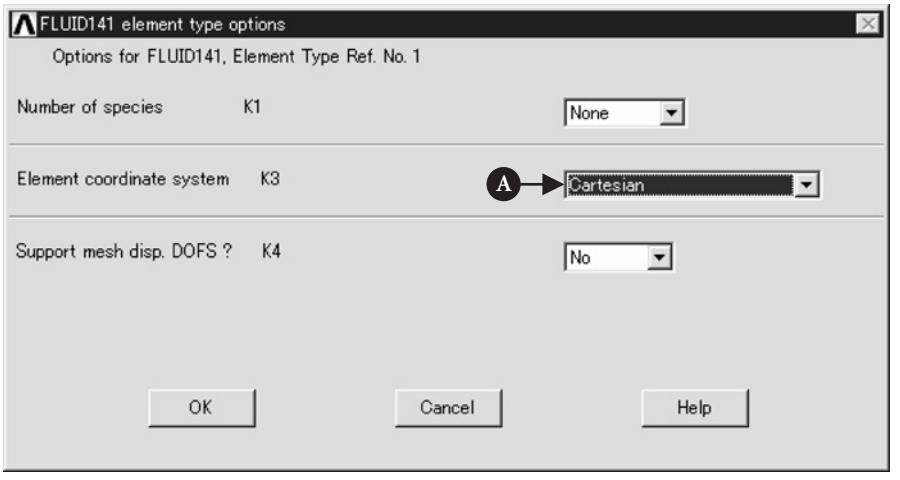


Figure 5.43 Window of **FLUID141 element type options**.

5.3.2.3 CREATE KEYPOINTS

To draw a flow channel with a butterfly valve for analysis, keypoints on the window are used to describe it in this section.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS**

(1) The window **Create Keypoints in Active Coordinate System** opens.

(2) Input the X and Y coordinate values to **X, Y, Z Location in active CS** box listed in Table 5.5.

Table 5.5 X and Y coordinates of KPs for a flow channel

KP No.	X	Y
1	0	0
2	0	−0.03
3	0.2	−0.03
4	0.2	0
5	0.2	0.03
6	0	0.03
7	0.05587	−0.02648
8	0.0716	0
9	0.08587	0.02548
10	0.08413	0.02648
11	0.06885	0
12	0.05413	−0.02548

(3) After finishing step (2), 12 keypoints appear on the window as shown in Figure 5.44. Four keypoints on the *x*-axis are made to use variable grids of which distance becomes smaller as a grid approaches the wall.

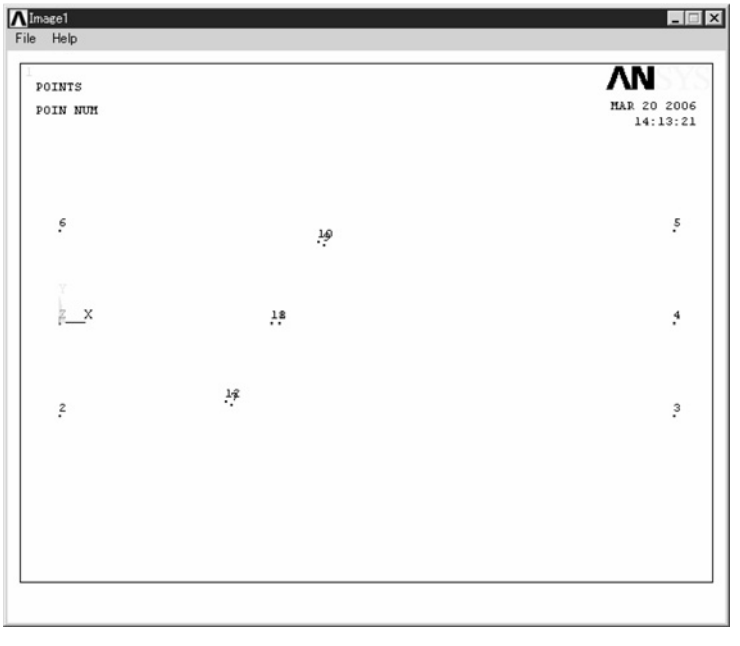


Figure 5.44 **ANSYS Graphics** window.

5.3.2.4 CREATE AREAS FOR FLOW CHANNEL

Areas are created from keypoints by performing the following steps.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → Through KPs**

(1) The window **Create Area thru KPs** opens.
(2) Pick keypoints **1, 2, 3, 4, 5, 6** and **11, 12, 7, 8, 9, 10** in order, and, then, two areas are created on the window as shown in Figure 5.45.
(3) When two areas are made, click **OK** button in the window **Create Area thru KPs**.

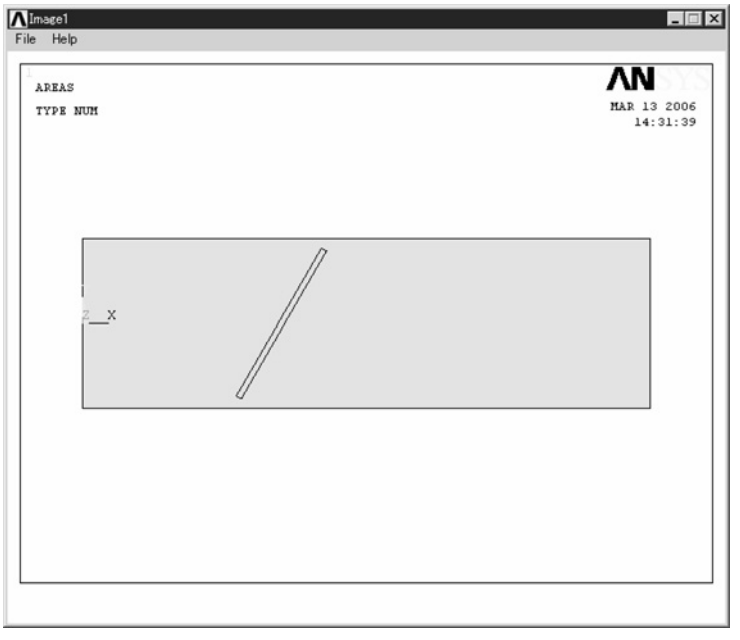


Figure 5.45 **ANSYS Graphics** window.

5.3.2.5 SUBTRACT THE VALVE AREA FROM THE CHANNEL AREA

According to the following steps, the valve area is subtracted from the channel area.

COMMAND | **ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas**

(1) The window **Subtract Areas** opens (Figure 5.46).

(2) Click the channel area displayed on **ANSYS Graphics** window and **OK** button in Figure 5.46. Then click the valve area and **OK** button in Figure 5.46. The drawing of the channel appears as shown in Figure 5.47.

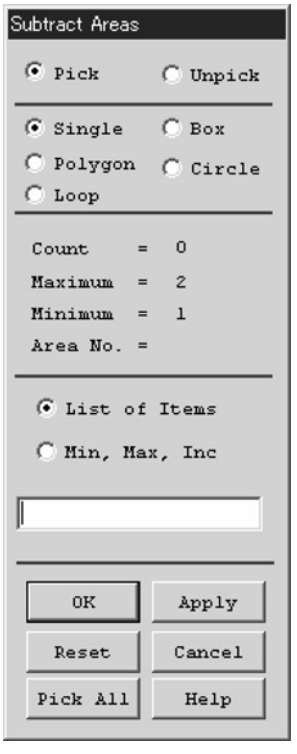


Figure 5.46 Window of **Subtract Areas**.

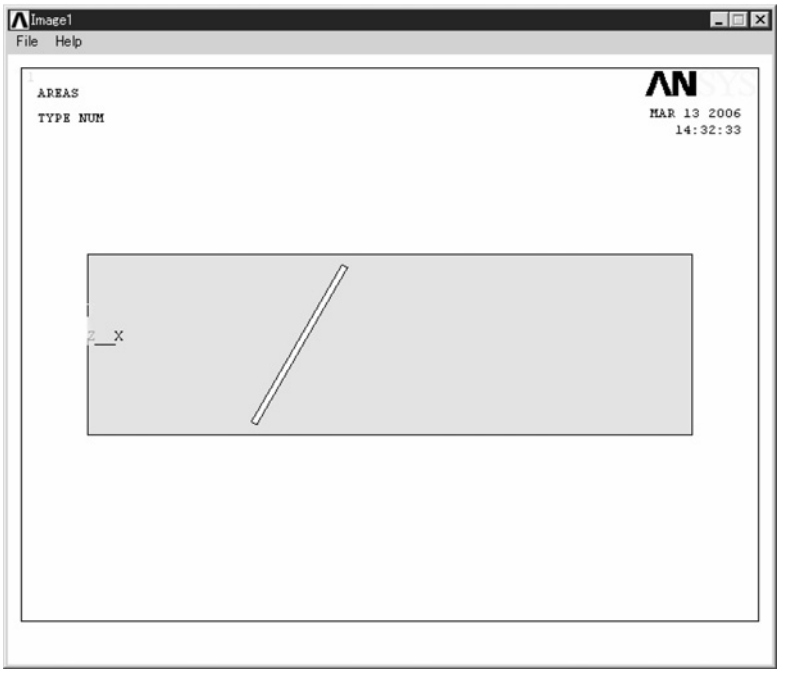


Figure 5.47 **ANSYS Graphics** window.

5.3.2.6 CREATE MESH IN LINES AND AREAS

First, in order to indicate the number of keypoints and lines on **ANSYS Graphics** window, perform the following steps.

COMMAND | **Utility Menu → Plot → Lines**

COMMAND | **Utility Menu → PlotCtrls → Numbering**

(1) Check the boxes of **KP** and **LINE** of the window **Plot Numbering Controls** and click **OK** button.

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

(1) Click **Lines Set** box in the window Mesh Tool. Then the window **Element size on Picked Lines** as shown in Figure 5.48 opens.

(2) Pick lines **1, 4, 7, 10** on **ANSYS Graphics** window and click **OK** button in Figure 5.48. The window **Element Sizes on Picked Lines** as shown in Figure 5.49 opens.

(3) Input **20** to **NDIV** box and **0.2** to **SPACE** box and, then, click **Apply** button. Next, input figures to these boxes according to Table 5.6.

(4) When all figures are inputted, click **OK** button of the window **Element Size on Picked Lines**.

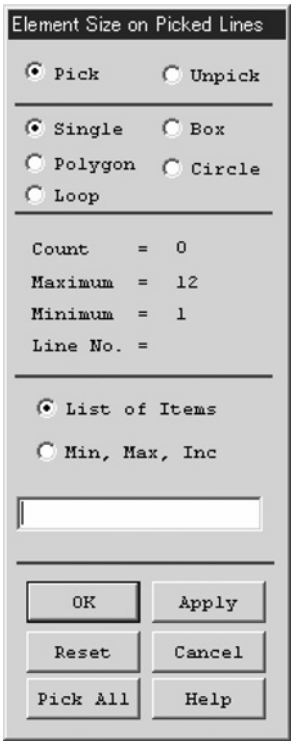


Figure 5.48 Window of **Element Size on Picked Lines**.

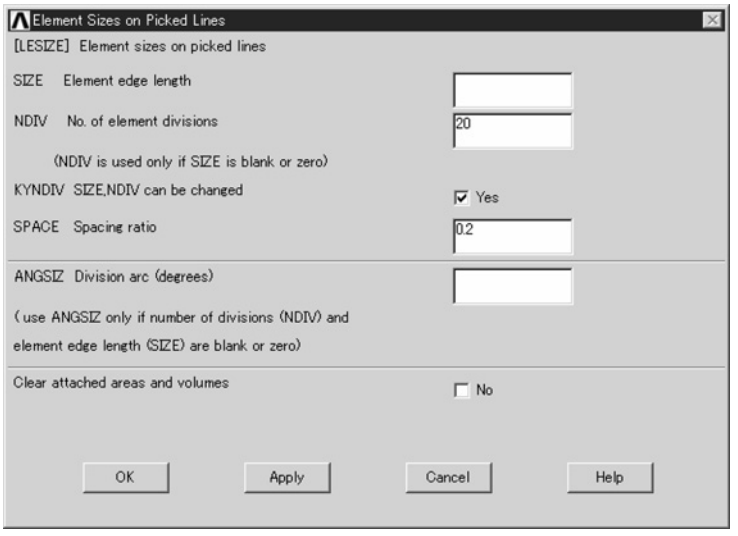


Figure 5.49 Window of **Element Sizes on Picked Lines**.

Table 5.6 Element sizes of picked lines

Line No.	NDIV	SPACE
1, 4, 7, 10	20	0.2
3, 6, 9 12	20	5
2, 5	100	–
8, 11	2	–

COMMAND | **ANSYS Main Menu → Preprocessor → Meshing → Mesh Tool**

The window **Mesh Tool** opens.

(1) Click **Mesh** on the window **Mesh Tool** and the window **Mesh Areas** opens. Click the channel area on **ANSYS Graphics** window and, then, click **OK** button of the window **Mesh Areas**. Meshed area appears on **ANSYS Graphics** window as shown in Figure 5.50.

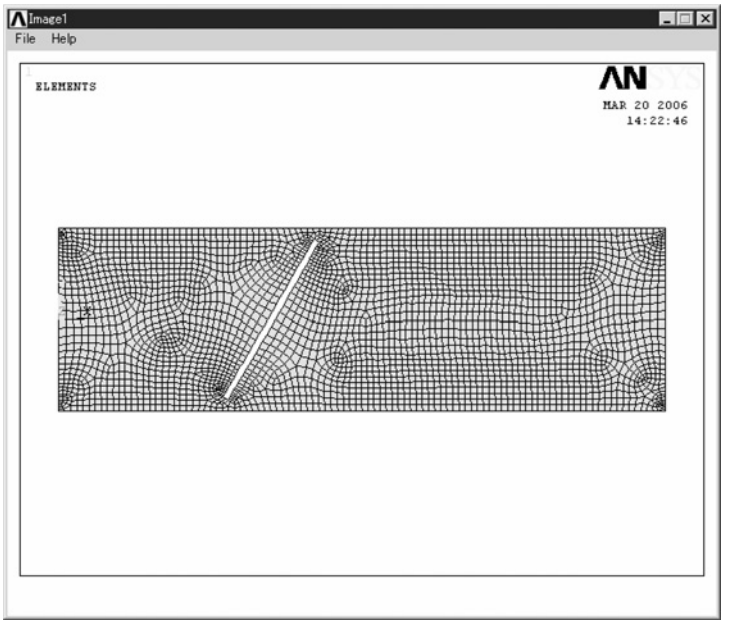


Figure 5.50 **ANSYS Graphics** window.

5.3.2.7 BOUNDARY CONDITIONS

The boundary conditions can be set by the following steps.

COMMAND | **ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Velocity → On Lines**

The window **Apply V on Lines** opens.

(1) Pick **Line 1** and **Line 6** at the entrance of the channel on **ANSYS Graphics** window and click **OK** button. Then the window **Apply VELO load on lines** opens (Figure 5.51).

(2) Input [A] **0.01** to **VX** box and [B] **0** to **VY** box. Then, click [C] **OK** button.

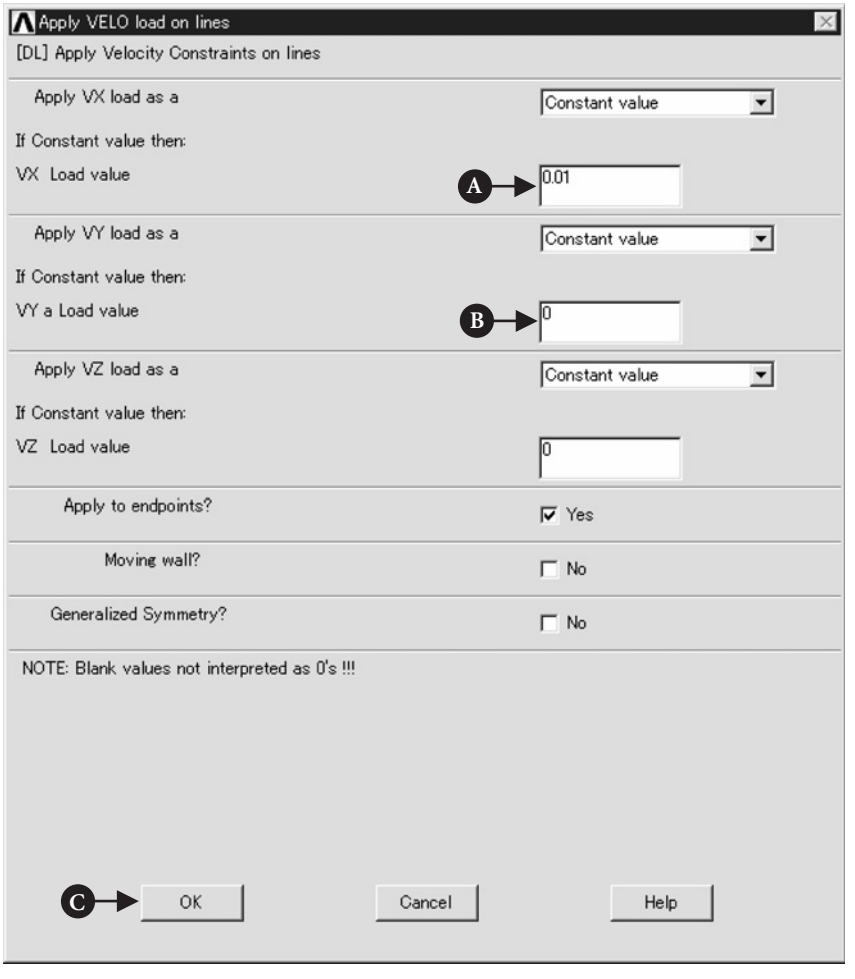


Figure 5.51 Window of **Apply VELO load on lines**.

(3) Pick Line Number of **2, 5, 7, 8, 9, 10, 11, 12** and click **OK** button. Input **0** to **VX** and **VY** boxes in Figure 5.51 and click **OK** button.

COMMAND **ANSYS Main Menu → Solution → Define Loads → Apply → Fluid/CFD → Pressure DOF → On Lines**

The window **Apply PRES on Lines** opens (Figure 5.52).

(1) Pick **Line 3** and **Line 4** on **ANSYS Graphics** window and click **OK** button. Then the window **Apply PRES on lines** opens (Figure 5.53).

(2) Input [B] **0** to **PRES** box and close the window by clicking [C] **OK** button.

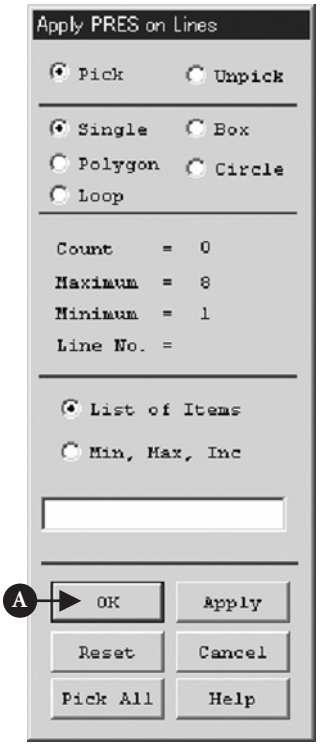


Figure 5.52 Window of **Apply PRES on lines**.

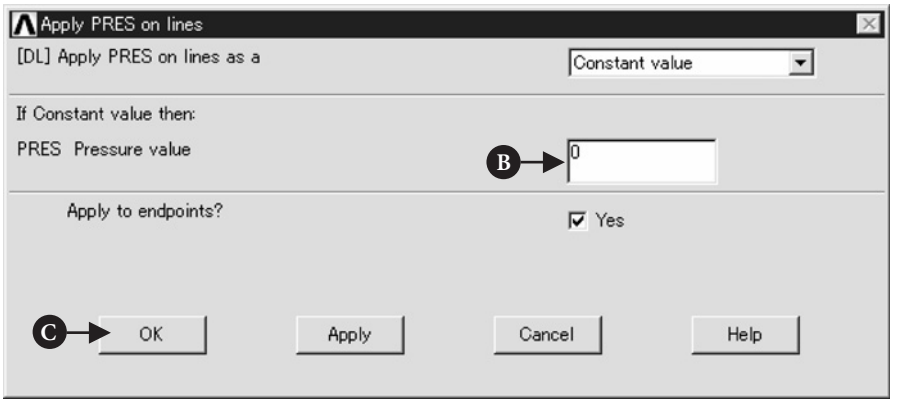


Figure 5.53 Window of **Apply PRES on lines**.

5.3.3 EXECUTION OF THE ANALYSIS

5.3.3.1 FLOTRAN SET UP

The following steps are performed to set up the analysis of **FLOTRAN**.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Solution Options**

The window **FLOTRAN Solution Options** opens (Figure 5.54).

(1) Set [A] **Adiabatic**, **Turbulent**, and **Incompressible** to **TEMP**, **TURB**, and **COMP** boxes, and, then, click [B] **OK** button.

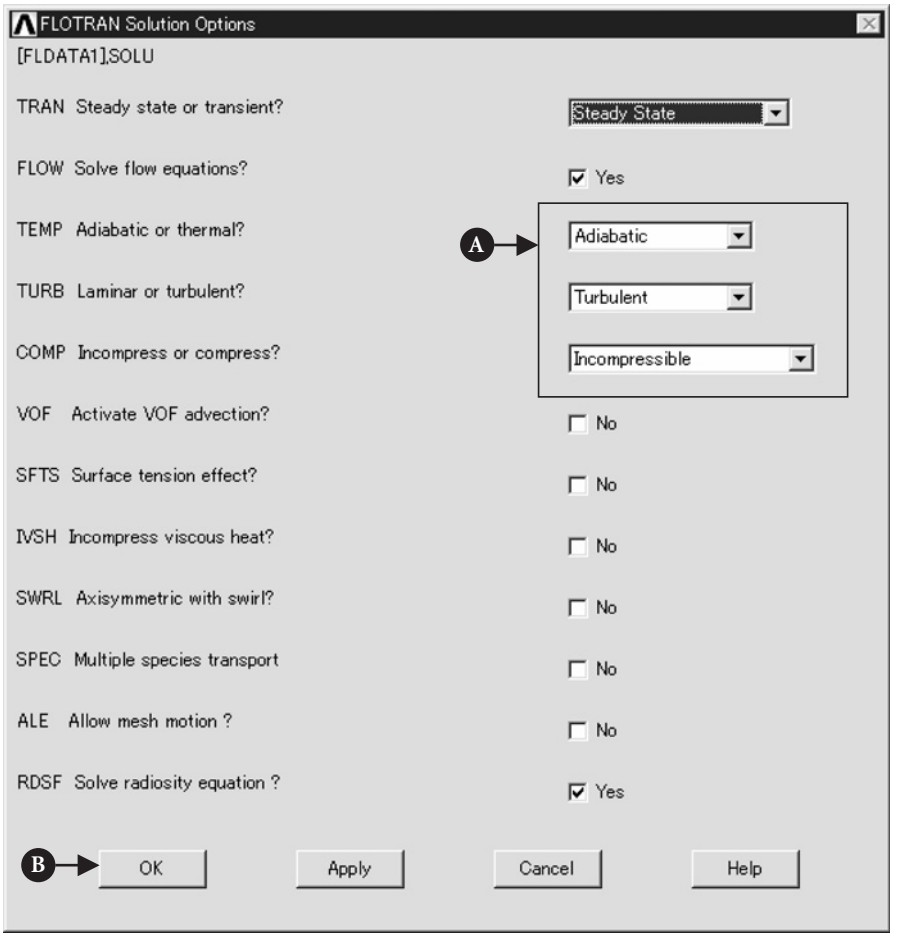


Figure 5.54 Window of **FLOTRAN Solution Options**.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Fluid Properties**

The window **Fluid Properties** opens (Figure 5.55).

(1) Select [A] **Liquid** in **Density**, **Viscosity**, and **Conductivity** boxes and click [B] **OK** button. Then the window **CFD Flow Properties** as shown in Figure 5.56 opens. Input [C] **1000** to **D0** box and [D] **0.001** to **V0** box. Then click [E] **OK** button.

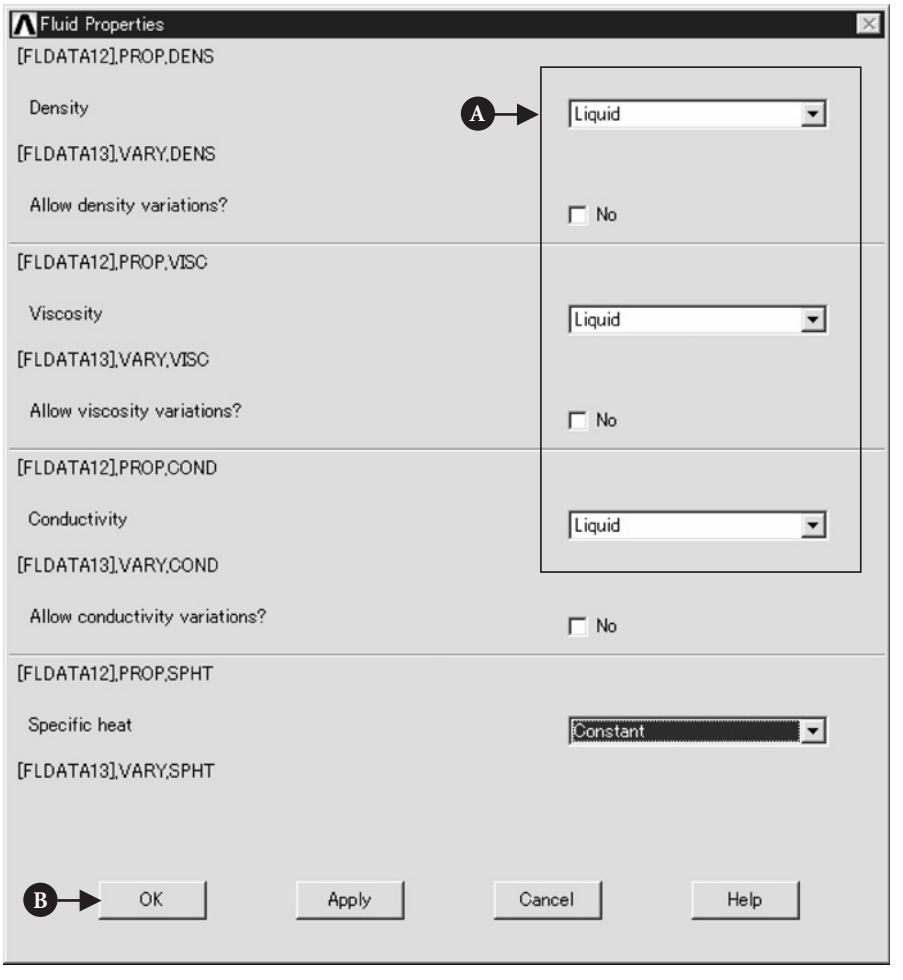


Figure 5.55 Window of **Fluid Properties**.

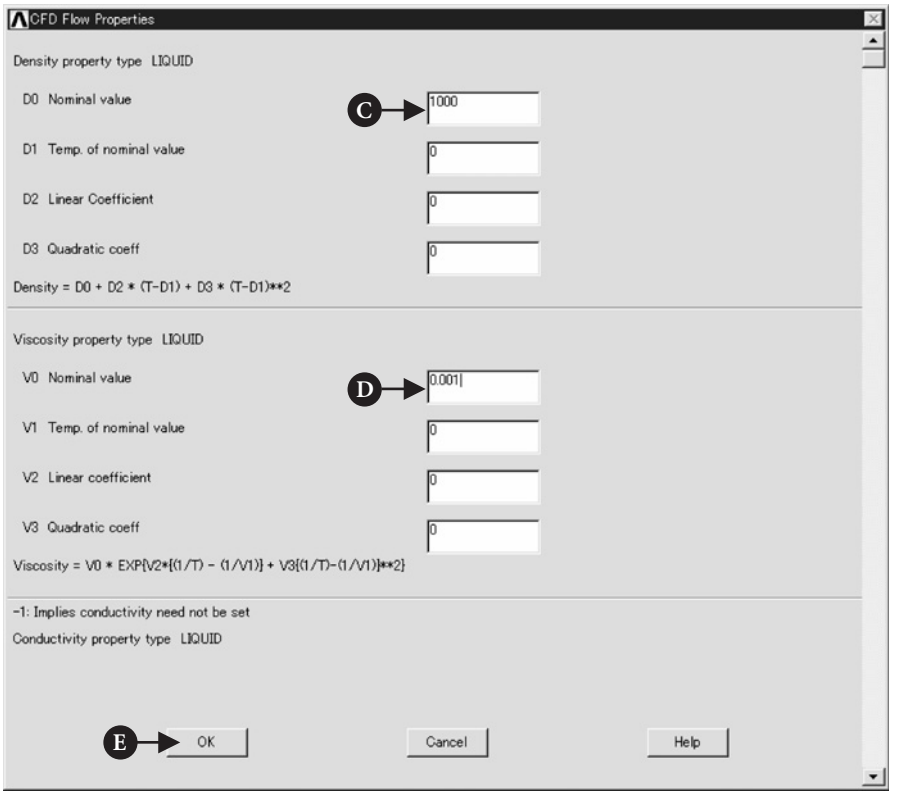


Figure 5.56 Window of **CFD Flow Properties**.

COMMAND | **ANSYS Main Menu → Solution → FLOTRAN Set Up → Execution Ctrl**

The window **Steady State Control Settings** opens (Figure 5.57).

(1) Input [A] **200** to **EXEC** box and click [B] **OK** button.

5.3.4 EXECUTE CALCULATION

COMMAND | **ANSYS Main Menu → Solution → Run FLOTRAN**

When the calculation is finished, the window **Note** opens.

(1) Click **Close** button.

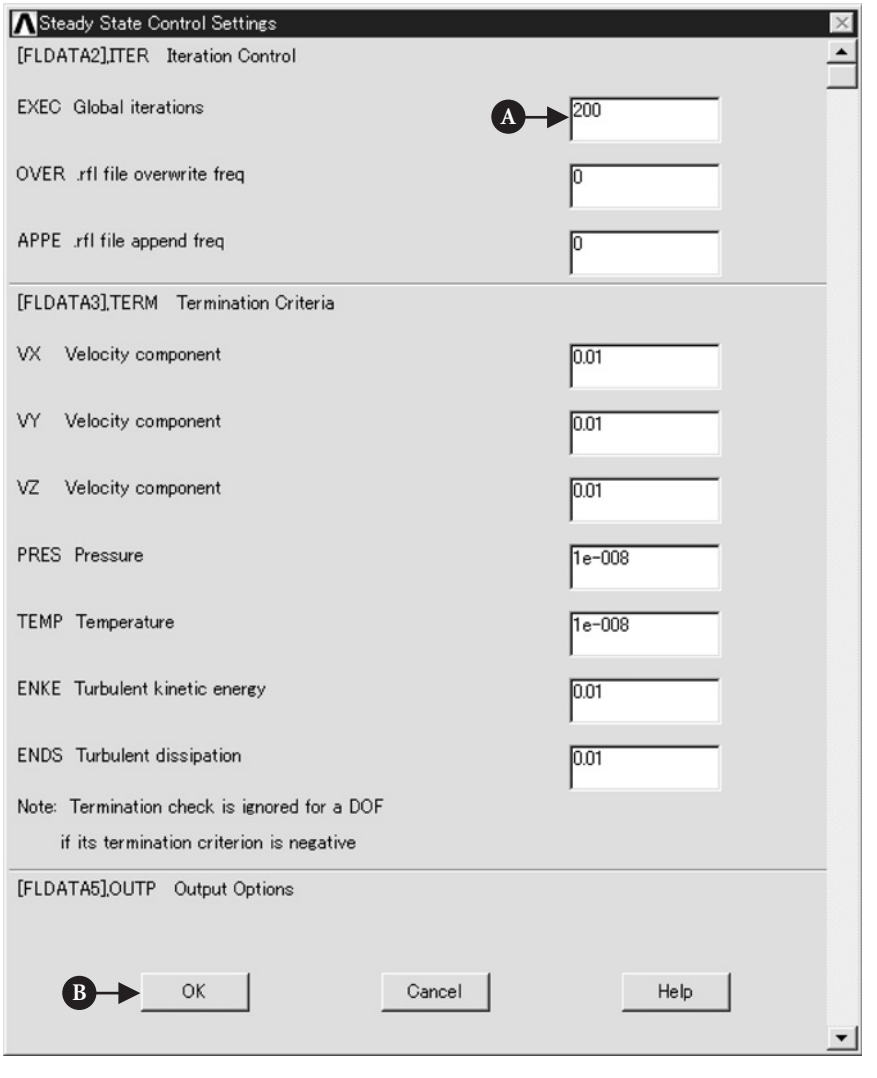


Figure 5.57 Window of **Steady State Control Settings**.

5.3.5 Postprocessing

5.3.5.1 Read the calculated results

Command | **ANSYS Main Menu → General Postproc → Read Results → Last Set**

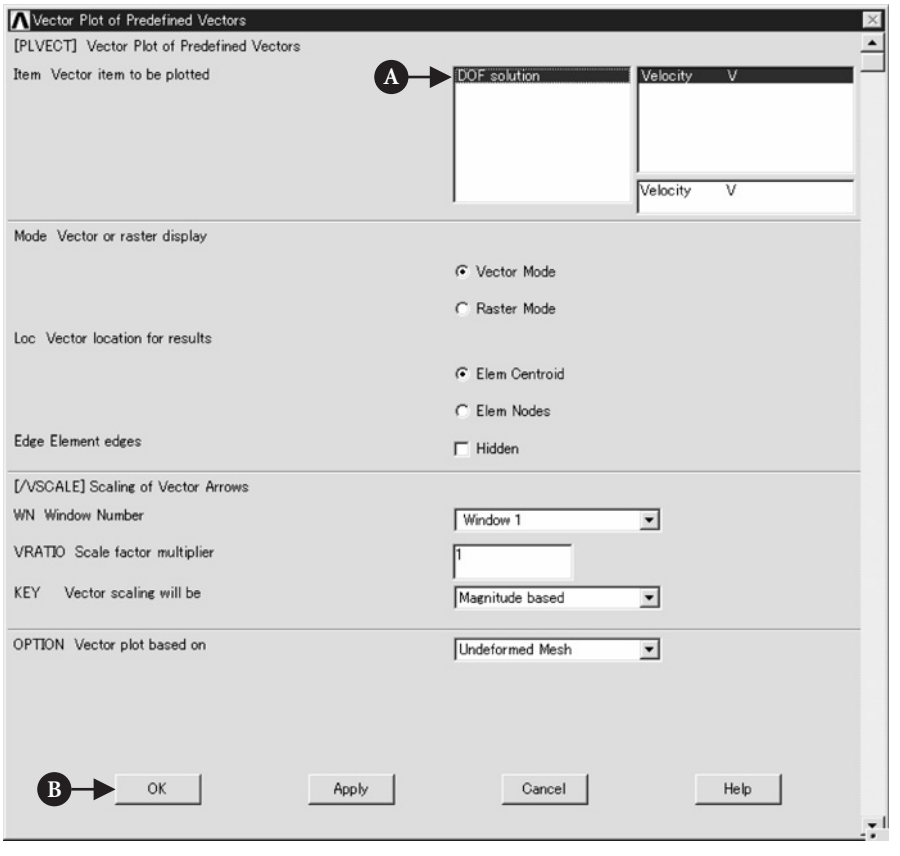


Figure 5.58 Window of **Vector Plot of Predefined Vectors**.

5.3.5.2 PLOT THE CALCULATED RESULTS

COMMAND **ANSYS Main Menu → General Postproc → Plot Results → Vector Plot → Predefined**

The window **Vector Plot of Predefined Vectors** opens (Figure 5.58).

(1) Select [A] **DOF solution** and **Velocity V** and click [B] **OK**.

(2) The calculated result for velocity vectors appears on **ANSYS Graphics** window as shown in Figure 5.59.

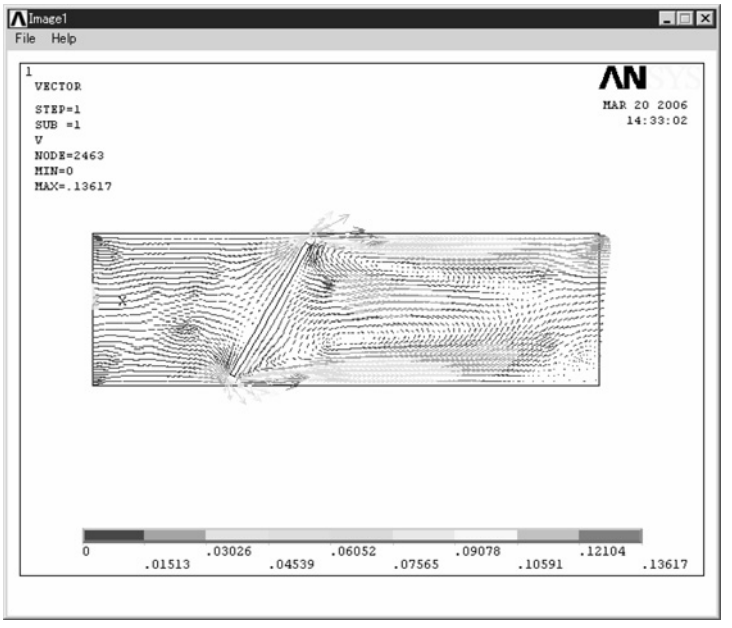


Figure 5.59 **ANSYS Graphics** window.

5.3.5.3 DETAILED VIEW OF THE CALCULATED FLOW VELOCITY

Near separation points in front of or the back of the valve surface, the flow velocity is very small and it is very difficult to distinguish the flow directions even if the area near the valve is enlarged as shown in Figure 5.60. Therefore, the size of vector arrows becomes large by the following steps.

COMMAND | **Utility Menu → PlotCtrls → Style → Vector Arrow Scaling**

The window **Vector Arrow Scaling** opens (Figure 5.61).

(1) Input [A] **5** to **VRATIO** box. This means that the length of arrows becomes five times larger as long as those in Figure 5.60. Click [B] **OK** button. Then the detailed flow view near the valve is displayed as shown in Figure 5.62.

In ANSYS, contour intervals can be changed by performing the following steps.

COMMAND | **Utility Menu → PlotCtrls → Style → Contours → Uniform → Contours**

The window **Uniform Contours** opens (Figure 5.63).

(1) Pick [A] **Contour intervals – User specified**. Input **0** and **0.03** to **VMIN** and **VMAX** boxes and then click [C] **OK** button. This means that the velocity arrows

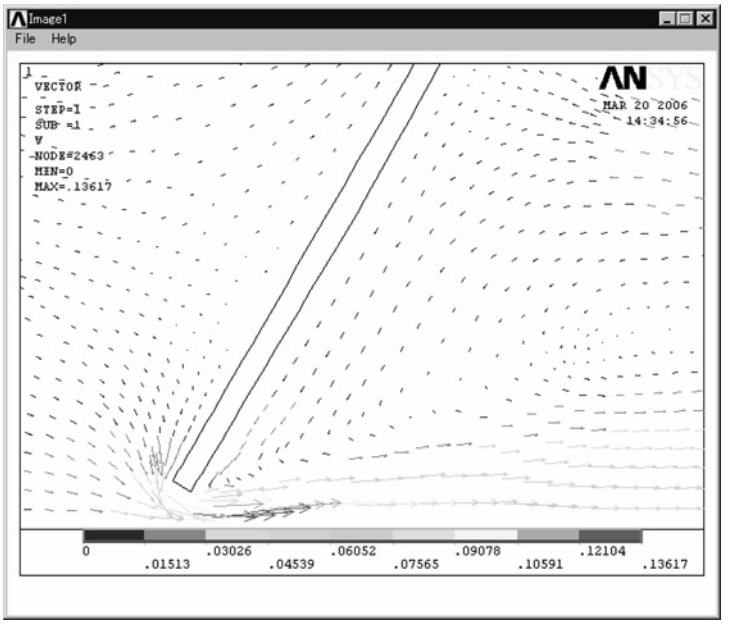


Figure 5.60 Enlarged view near the valve in **ANSYS Graphics** window.

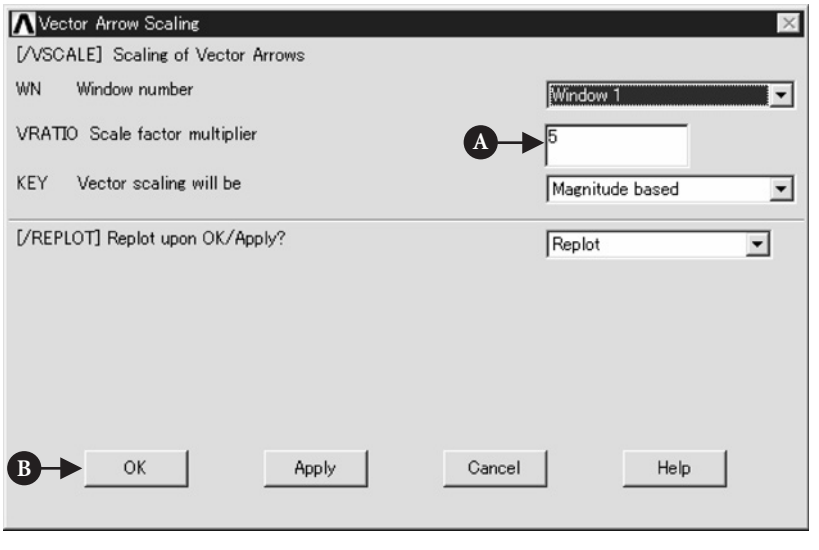


Figure 5.61 Window of **Vector Arrow Scaling**.

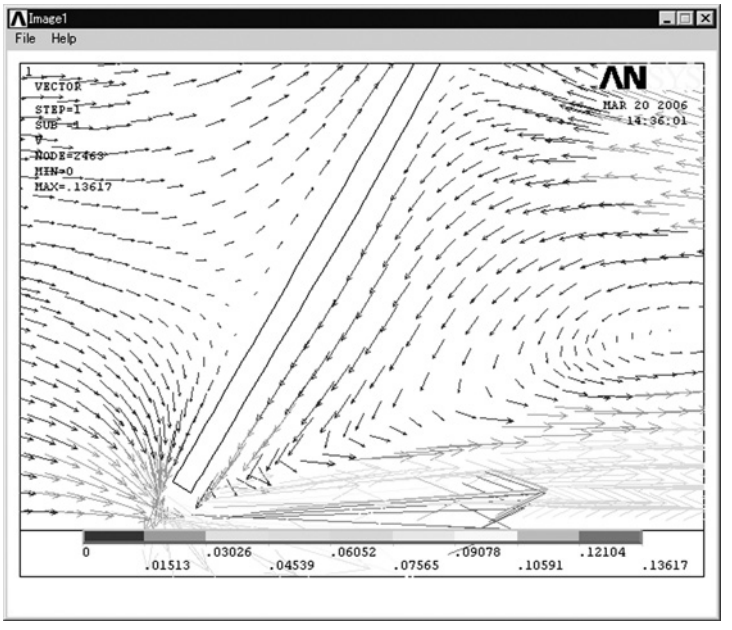


Figure 5.62 **ANSYS Graphics** window.

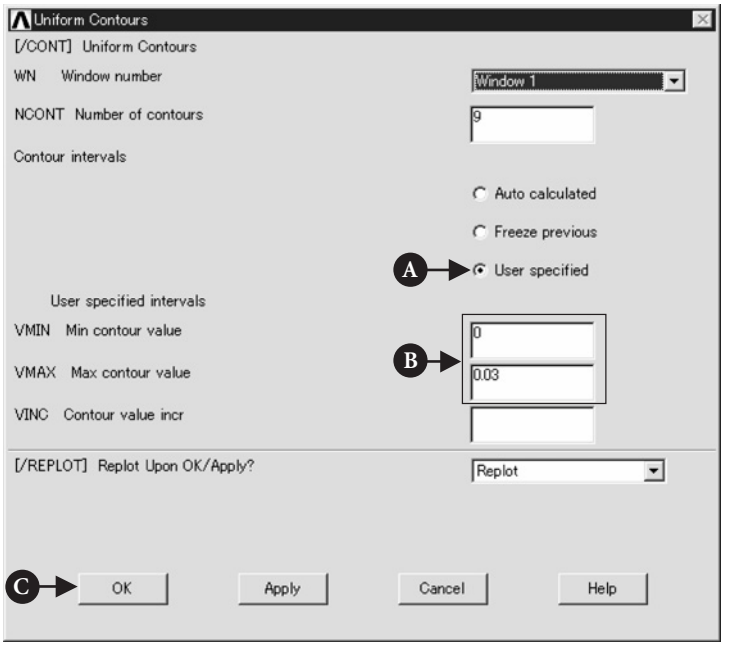


Figure 5.63 Window of **Uniform Contours**.

displayed on **ANSYS Graphics** window are colored only from **0** to **0.03** as shown in Figure 5.64.

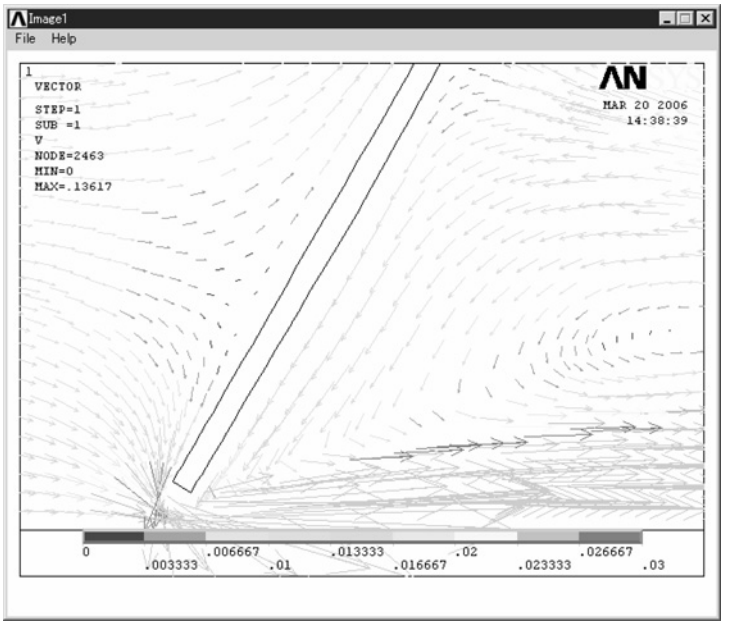


Figure 5.64 **ANSYS Graphics** window.

5.3.5.4 PLOT THE CALCULATED RESULTS BY PATH OPERATION

The continuity of volume flow can be confirmed by using **Path Operation** command and estimating the velocity distributions at the entrance and the exit of the channel. The steps for **Path Operation** are described in Section 5.2.5.3.

COMMAND **ANSYS Main Menu → General Postproc → Path Operations → Define Path → By Nodes**

The window **By Nodes** opens.

(1) Check **Pick** and pick two end nodes of the entrance of the channel. Click **OK** button.

(2) The window **By Nodes** opens. Input the path name **aa1** to **Name** box, the number of divisions 40 to nDiv box, and click **OK** button. The window **PATH Command** appears and then close this window by clicking **X** mark at the upper right end.

(3) Click **By Nodes** in **ANSYS Main Menu**. Pick two end nodes of the exit of the channel. Click **OK** button. The window **By Nodes** opens. Input the path name **aa2** to **Name** box and click **OK** button.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Map onto Path**

The window **Map Result Items onto Path** opens.

(1) Input **aa** to **Lab** box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

(2) Input **aa2** to **Lab** box and select [B] **DOF solution** and **Velocity VX**. Then click **OK** button.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph**

The window **Plot of Path Items on Graph** opens.

(1) Select **AA** in **Lab1-6** box and click **OK** button. Then the calculated result for the defined path **AA2** appears on **ANSYS Graphics** window as shown in Figure 5.65.

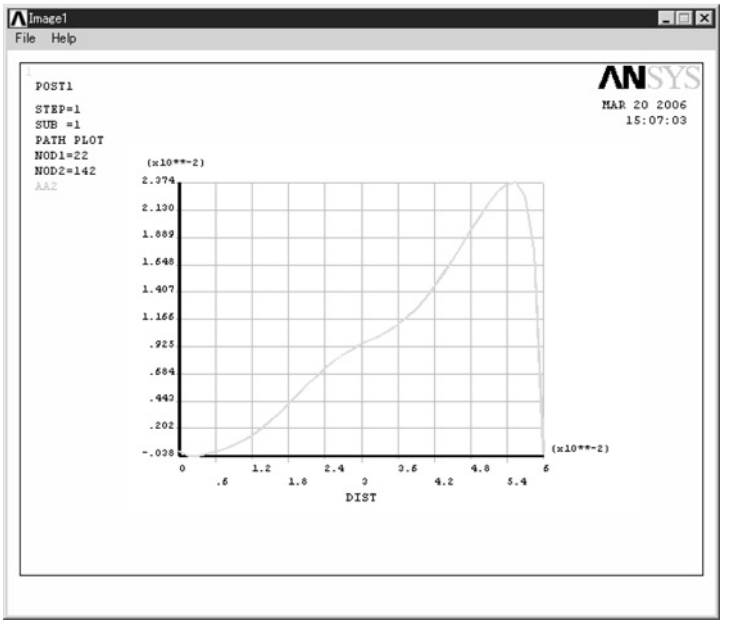


Figure 5.65 **ANSYS Graphics** window.

For the path of **AA1**:

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Recall Path**

(1) Select **AA1** in **NAME** box and click **OK** button.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Map onto Path**

(1) Input **aa** to **Lab** box and select **DOF solution** and **Velocity VX**. Then click **OK** button.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → On Graph**

(1) Select **AA** in **Lab1-6** box and click **OK** button. Then the calculated result for the defined path **AA1** is displayed on **ANSYS Graphics** window as shown in Figure 5.66.

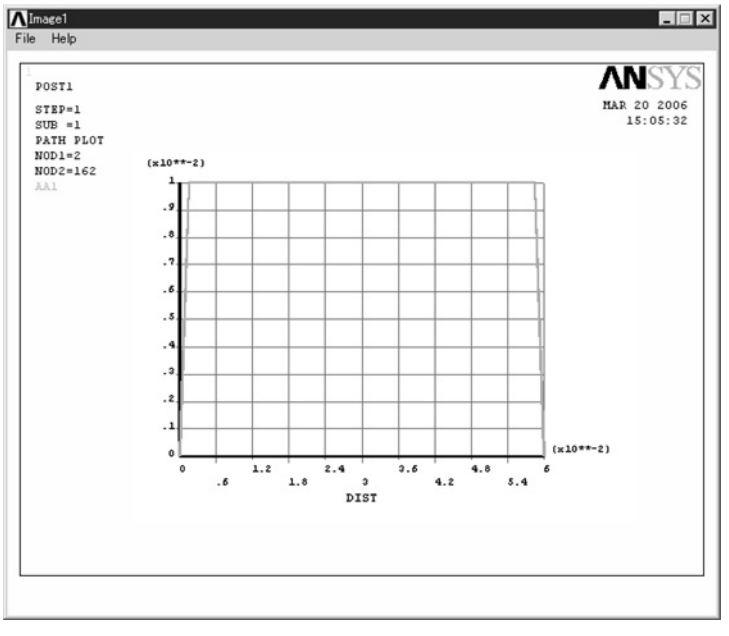


Figure 5.66 **ANSYS Graphics** window.

The values of flow velocities of the defined paths can be listed by the following commands.

COMMAND | **ANSYS Main Menu → General Postproc → Path Operations → Plot Path Item → List Path Items**

The window **List of Path Items** opens.

(1) Select **AA** in **Lab1-6** box and click **OK** button. Then the list for the defined path appears. When the path is changed, use **Recall Path** command.

CHAPTER 6

APPLICATION OF ANSYS TO THERMO MECHANICS

CHAPTER OUTLINE

6.1 GENERAL CHARACTERISTIC OF HEAT TRANSFER PROBLEMS 263
6.2 HEAT TRANSFER THROUGH TWO WALLS 265
6.3 STEADY-STATE THERMAL ANALYSIS OF A PIPE INTERSECTION 285
6.4 HEAT DISSIPATION THROUGH RIBBED SURFACE 312

6.1 GENERAL CHARACTERISTIC OF HEAT TRANSFER PROBLEMS

The transfer of heat is normally from a high-temperature object to a lower-temperature object. Heat transfer changes the internal energy of both systems involved according to the first law of thermodynamics.

Heat may be defined as energy in transit. An object does not possess "heat"; the appropriate term for the microscopic energy in an object is internal energy. The internal energy may be increased by transferring energy to the object from a higher-temperature (hotter) object – this is called heating.

A convenient definition of temperature is that it is a measure of the average translation kinetic energy associated with the disordered microscopic motion of atoms and molecules. The flow of heat is from a high-temperature region toward a lower-temperature region. The details of the relationship to molecular motion are dealt with by the kinetic theory. The temperature defined from kinetic theory is called

the kinetic temperature. Temperature is not directly proportional to internal energy since temperature measures only the kinetic energy part of the internal energy, so two objects with the same temperature do not, in general, have the same internal energy.

Internal energy is defined as the energy associated with the random, disordered motion of molecules. It is separated in scale from the macroscopic ordered energy associated with moving objects. It also refers to the invisible microscopic energy on the atomic and molecular scale. For an ideal monoatomic gas, this is just the translational kinetic energy of the linear motion of the "hard sphere" type atoms, and the behavior of the system is well described by the kinetic theory. However, for polyatomic gases there is rotational and vibrational kinetic energy as well. Then in liquids and solids there is potential energy associated with the intermolecular attractive forces.

Heat transfer by means of molecular agitation within a material without any motion of the material as a whole is called conduction. If one end of a metal rod is at a higher temperature, then energy will be transferred down the rod toward the colder end because the higher speed particles will collide with the slower ones with a net transfer of energy to the slower ones. For heat transfer between two plane surfaces, such as heat loss through the wall of a house, the rate of conduction could be estimated from,

$$\frac{Q}{t} = \frac{\kappa A(T_{hot} - T_{cold})}{d}$$

where the left-hand side concerns rate of conduction heat transfer; κ is the thermal conductivity of the barrier; A is the area through which heat transfer takes place; T is the temperature; and d is the thickness of barrier.

Another mechanism for heat transfer is convection. Heat transfer by mass motion of a fluid such as air or water when the heated fluid is caused to move away from the source of heat, carrying energy with it is called convection. Convection above a hot surface occurs because hot air expands, becomes less dense, and rises. Convection can also lead to circulation in a liquid, as in the heating of a pot of water over a flame. Heated water expands and becomes more buoyant. Cooler, more dense water near the surface descends and patterns of circulation can be formed.

Radiation is heat transfer by the emission of electromagnetic waves which carry energy away from the emitting object. For ordinary temperatures (less than red hot), the radiation is in the infrared region of the electromagnetic spectrum. The relationship governing radiation from hot objects is called the Stefan–Boltzmann law:

$$P = e\sigma A(T^4 - T_c^4)$$

where P is the net radiated power; A is the radiating area; σ is the Stefan's constant; e is the emissivity coefficient; T is the temperature of radiator; T_c is the temperature of surroundings.

6.2 Heat transfer through two walls

6.2.1 Problem description

A furnace with dimensions of its cross-section specified in Figure 6.1 is constructed from two materials. The inner wall is made of concrete with a thermal conductivity, $k_c = 0.01\,\text{W/m K}$. The outer wall is constructed from bricks with a thermal conductivity, $k_b = 0.0057\,\text{W/m K}$. The temperature within the furnace is 673 K and the convection heat transfer coefficient $k_1 = 0.208\,\text{W/m}^2\,\text{K}$. The outside wall of the furnace is exposed to the surrounding air, which is at 253 K and the corresponding convection heat transfer coefficient, $k_2 = 0.068\,\text{W/m}^2\,\text{K}$.

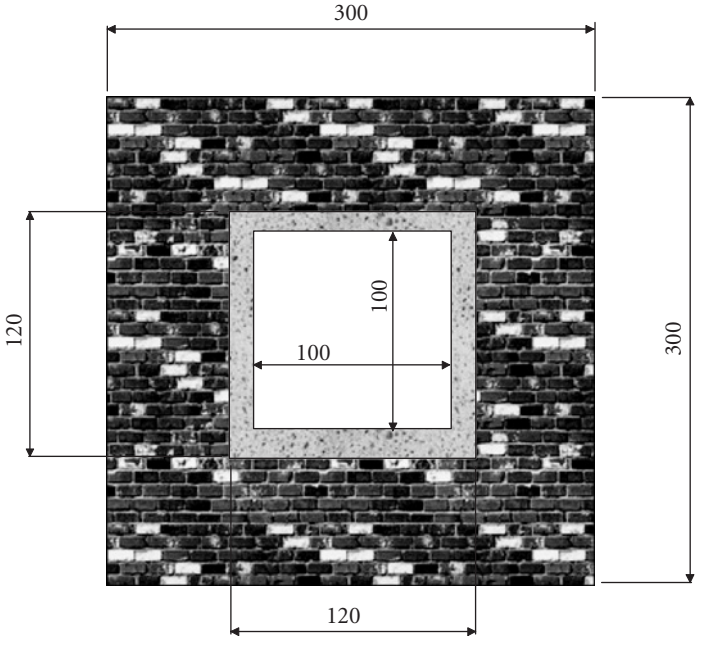


Figure 6.1 Cross-section of the furnace.

Determine the temperature distribution within the concrete and brick walls under steady-state conditions. Also determine the heat fluxes through each wall.

This is a two-dimensional (2D) problem and will be modeled using graphical user interface (GUI) facilities.

6.2.2 Construction of the model

From **ANSYS Main Menu** select **Preferences**. This frame is shown in Figure 6.2.

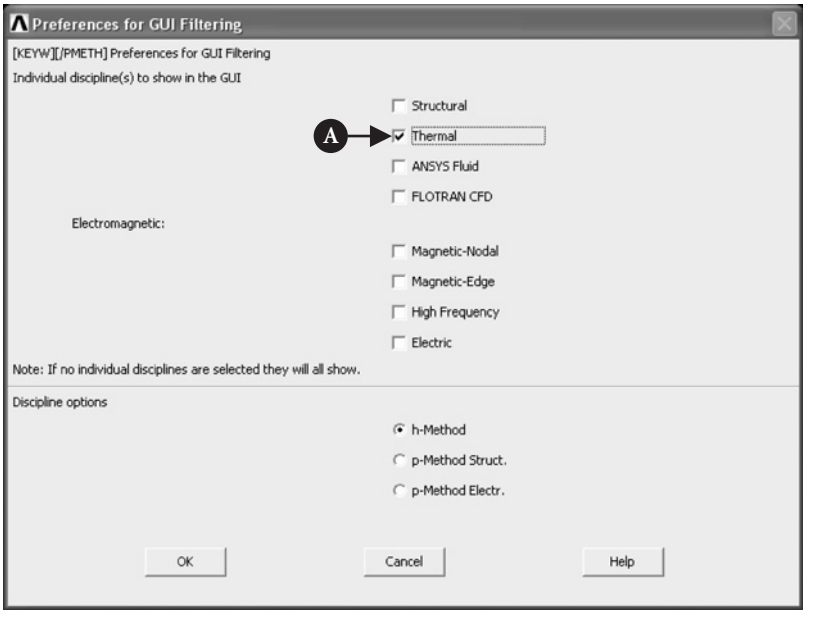


Figure 6.2 Preferences: Thermal.

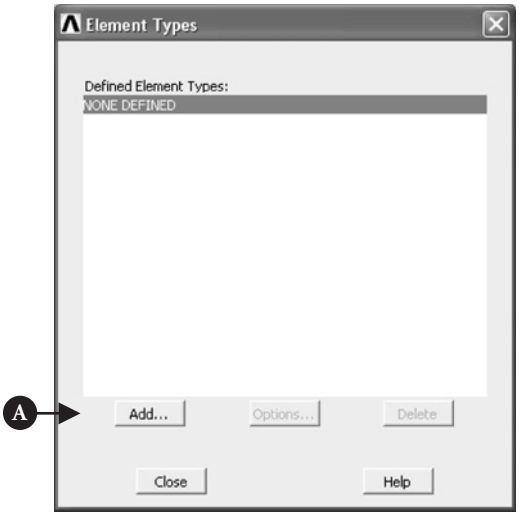


Figure 6.3 Element Types selection.

Depending on the nature of analysis to be attempted an appropriate analysis type should be selected. In the problem considered here [A] **Thermal** was selected as shown in Figure 6.2.

From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 6.3 appears.

Clicking [A] **Add** button activates a new set of options which are shown in Figure 6.4.

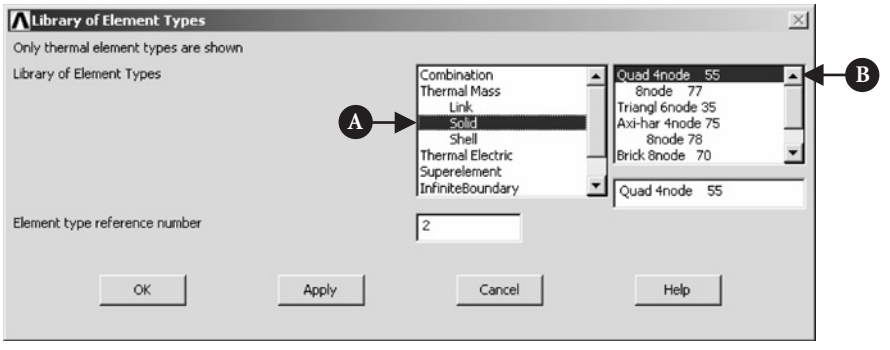


Figure 6.4 Library of Element Types.

Figure 6.4 shows that for the problem considered the following were selected: [A] **Thermal Mass → Solid** and [B] **4node 55**. This element is referred to as Type 1 PLANE55.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**. Figure 6.5 shows the resulting frame.

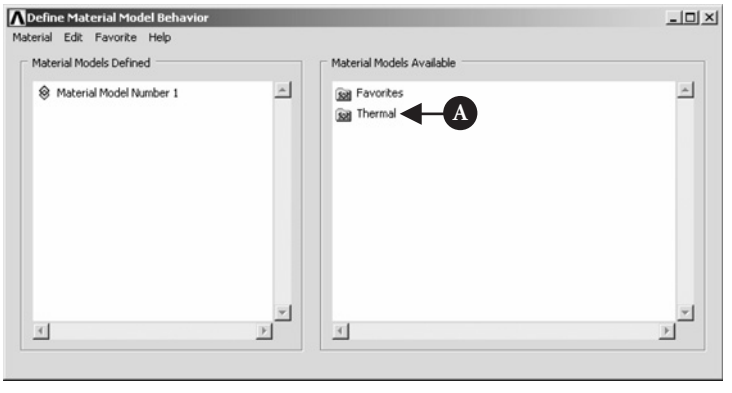


Figure 6.5 Define Material Model Behavior.

From the right-hand column select [A] **Thermal → Conductivity → Isotropic**. As a result, the frame shown in Figure 6.6 appears. Thermal conductivity [A] **KXX** = 0.01 W/m K, was selected as shown in Figure 6.6.

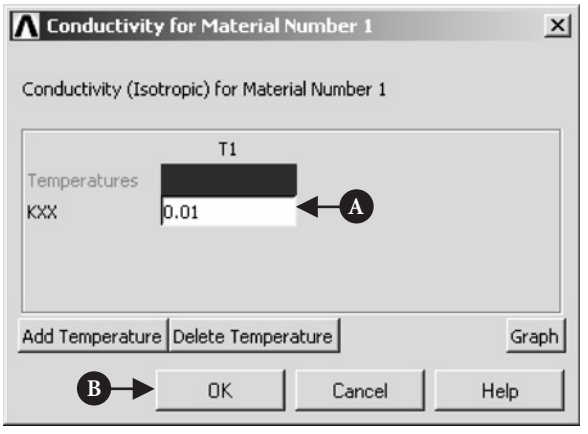


Figure 6.6 Conductivity for Material Number 1.

Clicking [B] **OK** button ends input into Material Number 1. In the frame shown in Figure 6.7 select from the top menu [A] **Material → New Model**. Database for Material Number 2 is created.

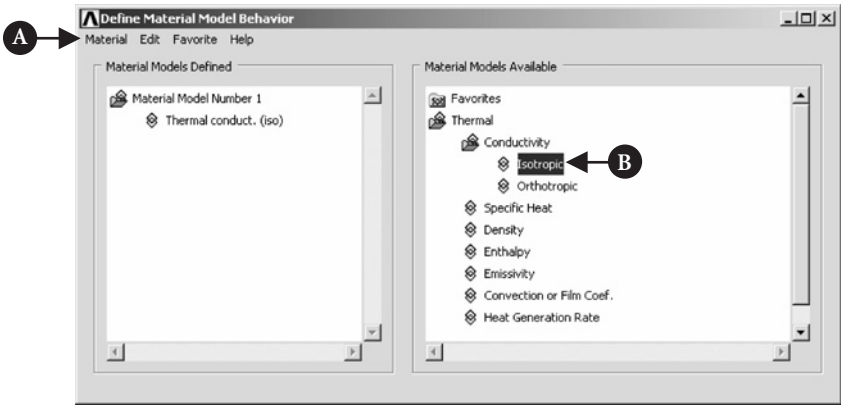


Figure 6.7 Define Material Model Behavior.

As in the case of Material Number 1 select [B] **Thermal → Conductivity → Isotropic**. The frame shown in Figure 6.8 appears. Enter [A] **KXX** = 0.0057 W/m K and click [B] **OK** button as shown in Figure 6.8.

In order to have created primitives numbered from **ANSYS Utility Menu** select **PlotCtrls → Numbering** and check the box area numbers.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Figure 6.9 shows the resulting frame.

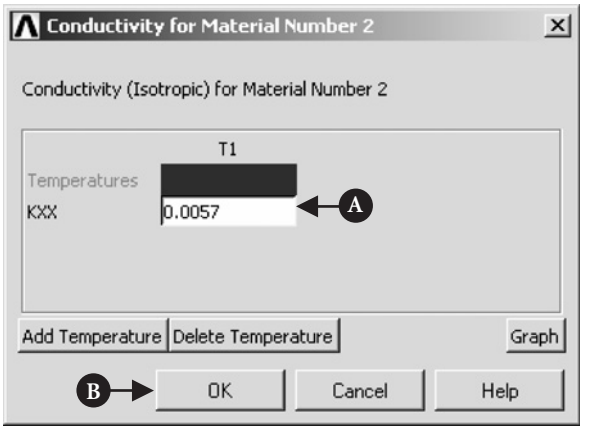


Figure 6.8 Conductivity for Material Number 2.

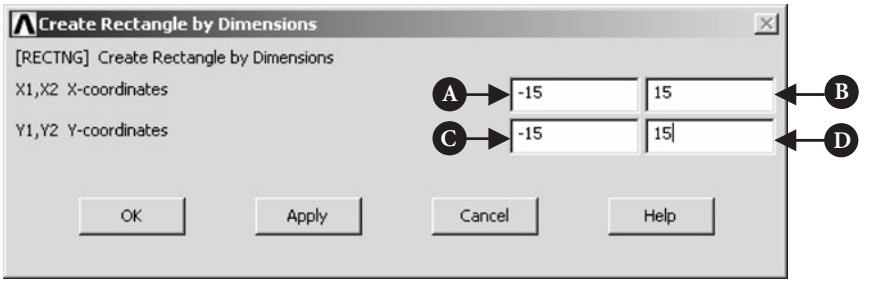


Figure 6.9 Create Rectangle by Dimensions.

Inputs [A] **X1** $= -15$; [B] **X2** $= 15$; [C] **Y1** $= -15$; and [D] **Y2** $= 15$ to create outer wall perimeter are shown in Figure 6.9. Next, the perimeter of inner wall is created in the same way. Figure 6.10 shows the frame with appropriate entries.

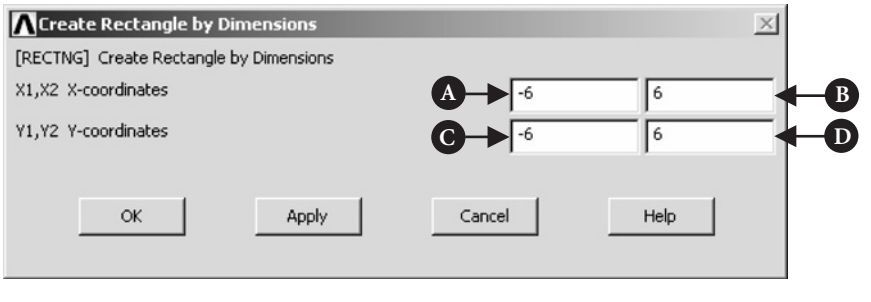


Figure 6.10 Create Rectangle by Dimensions.

In order to generate the brick wall area of the chimney, subtract the two areas which have been created. From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. Figure 6.11 shows the resulting frame.

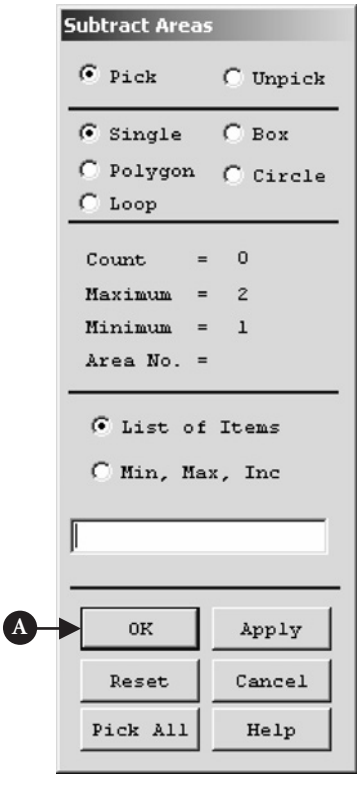


Figure 6.11 Subtract Areas.

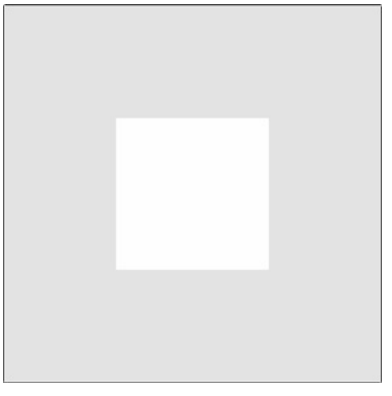


Figure 6.12 Brick wall outline.

First select the larger area (outer brick wall) and click [A] **OK** button in the frame of Figure 6.11. Next, select the smaller area (inner concrete wall) and click [A] **OK** button. The smaller area is subtracted from the larger area and the outer brick wall is produced. It is shown in Figure 6.12.

Using a similar approach, the inner concrete wall is constructed. From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Figure 6.13 shows the resulting frame with inputs: [A] **X1** = −6; [B] **X2** = 6; [C] **Y1** = −6, and [D] **Y2** = 6. Pressing [E] **OK** button creates the rectangular area A1.

Again, from **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Frame with inputs: [A] **X1** = −5; [B] **X2** = 5; [C] **Y1** = −5, and [D] **Y2** = 5 is shown in Figure 6.14.

Clicking [E] **OK** button creates the rectangular area A2. As before, to create the concrete area of the furnace, subtract area A2 from area A1. From **ANSYS Main Menu**

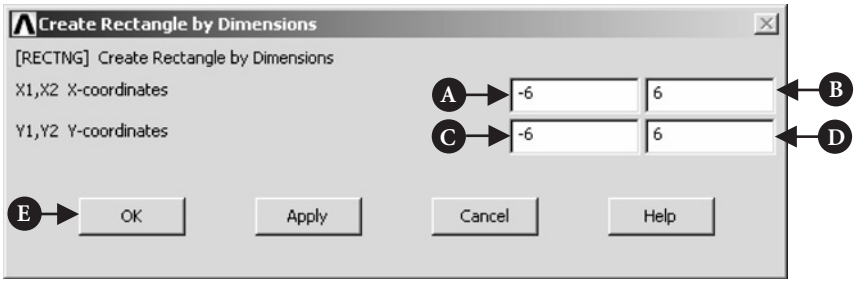


Figure 6.13 Create Rectangle by Dimensions.

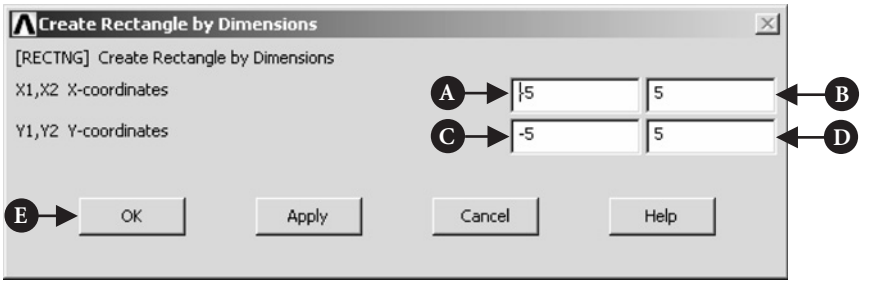


Figure 6.14 Create Rectangle by Dimensions.

select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. The frame shown in Figure 6.11 appears. Select area A1 first and click [A] **OK** button. Next select area A2 and click [A] **OK** button. As a result, the inner concrete wall is created. This is shown in Figure 6.15.

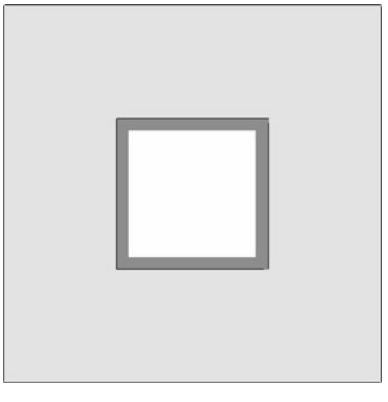


Figure 6.15 Outline of brick and concrete walls.

From **ANSYS Main Menu** select **Preprocessor → Meshing → Size Cntrls → ManualSize → Global → Size**. As a result of this selection, the frame shown in Figure 6.16 appears.

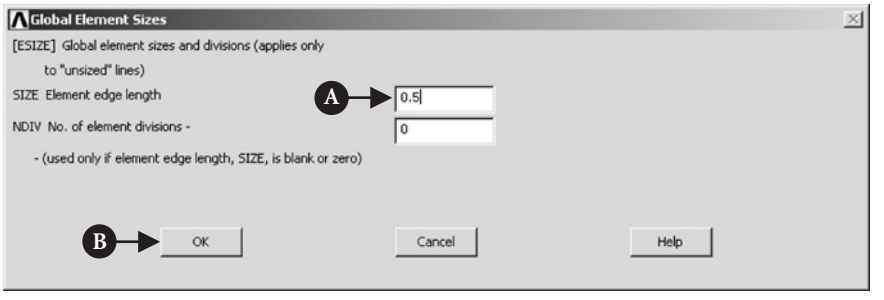


Figure 6.16 Global Element Sizes.

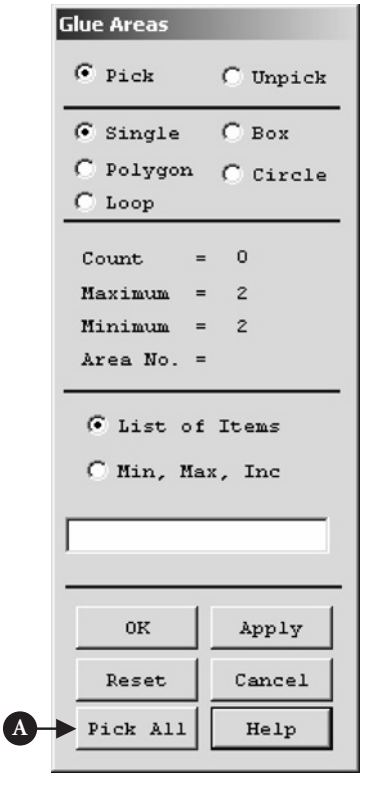


Figure 6.17 Glue Areas.

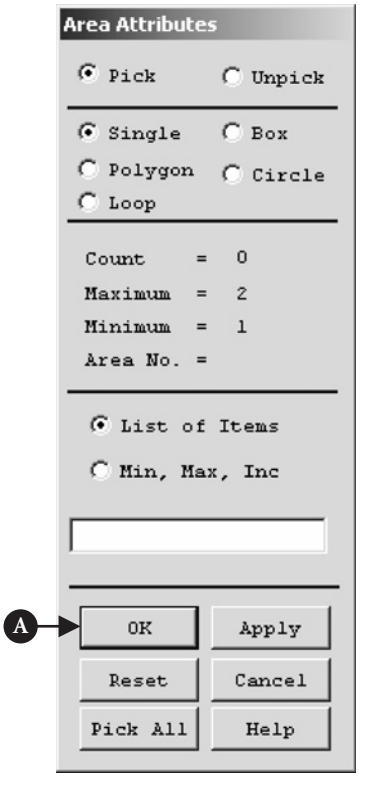


Figure 6.18 Area Attributes.

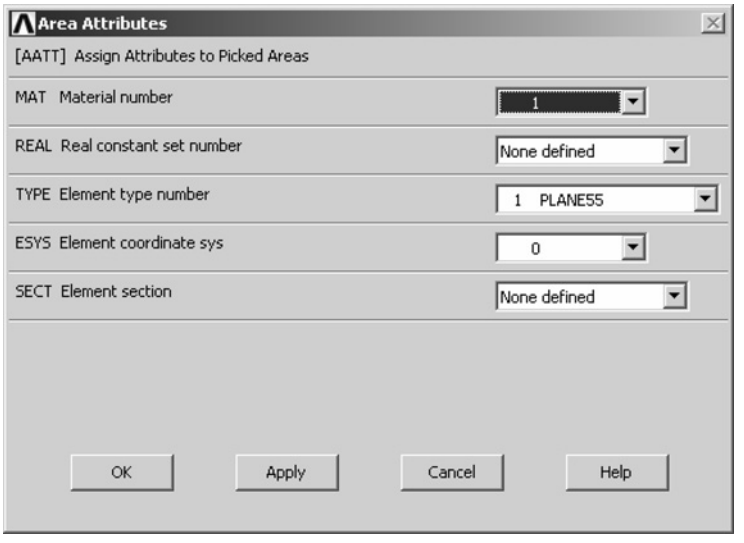


Figure 6.19 Area Attributes (concrete wall).

Input for the element edge length [A] **SIZE** = 0.5 and click [B] **OK** button.

Because the outer brick wall and the inner concrete wall were created as separate entities, therefore, it is necessary to "glue" them together so that they share lines along their common boundaries. From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Boolean → Glue → Areas**. The frame shown in Figure 6.17 appears.

Select [A] **Pick All** option in the frame of Figure 6.17 to glue the outer and inner wall areas. Before meshing is done, it is necessary to specify material numbers for the concrete and the brick walls.

From **ANSYS Main Menu** select **Preprocessor → Meshing → Mesh Attributes → Picked Areas**. The frame shown in Figure 6.18 is created.

Select first the concrete wall area and click [A] **OK** button in the frame of Figure 6.18. A new frame is produced as shown in Figure 6.19.

Material Number 1 is assigned to the concrete inner wall as shown in Figure 6.19.

Next, assign Material Number 2 to the brick outer wall following the procedure outlined above that is recall frame of Figure 6.19 and select brick outer wall. Figure 6.20 shows the frame with appropriate entry.

Now meshing of both areas can be carried out. From **ANSYS Main Menu** select **Preprocessor → Meshing → Mesh → Areas → Free**. The frame shown in Figure 6.21 appears.

Select [A] **Pick All** option shown in Figure 6.21 to mesh both areas.

In order to see both areas meshed, from **Utility Menu** select **PlotCtrls → Numbering**. In the appearing frame, shown in Figure 6.22, select [A] **Material numbers** and click [B] **OK** button.

As a result of that, both walls with mesh are displayed (see Figure 6.23).

Figure 6.20 Area Attributes.

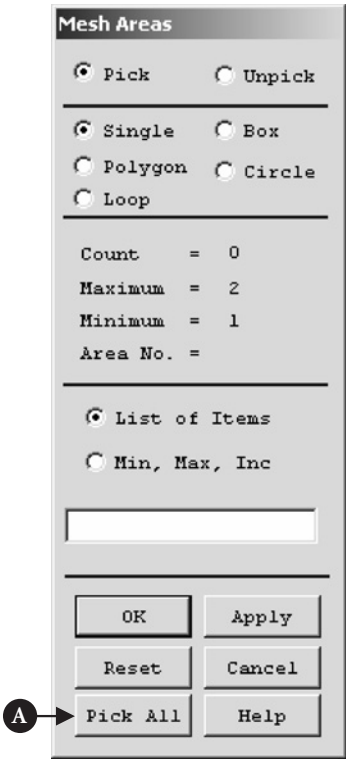


Figure 6.21 Mesh Areas.

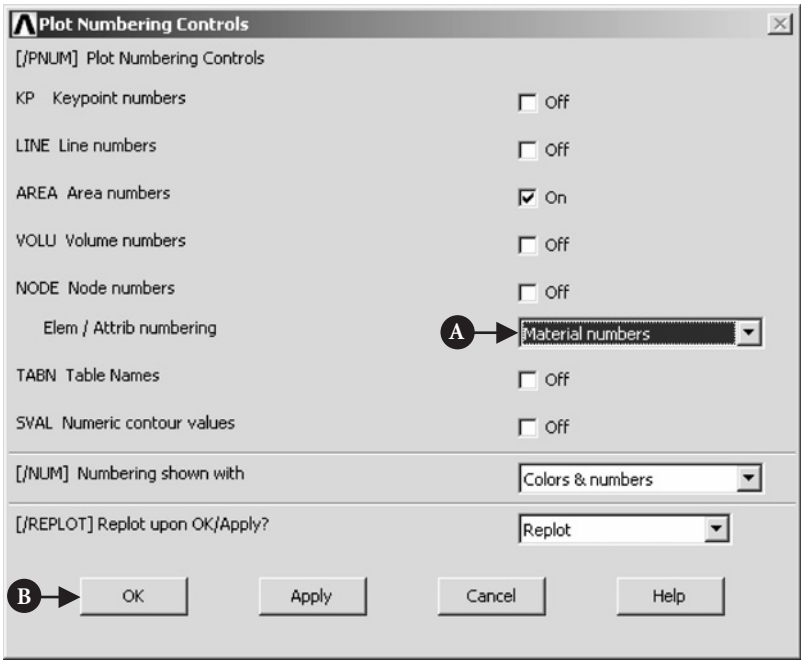


Figure 6.22 Plot Numbering Controls.

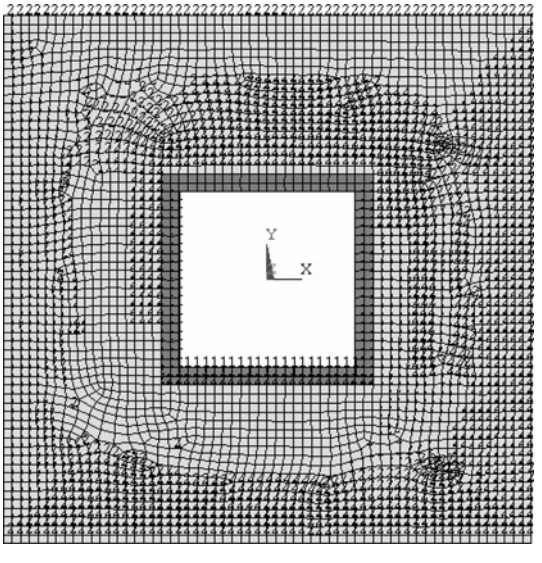


Figure 6.23 Outer and inner walls of the furnace meshed.

6.2.3 Solution

Before a solution can be run boundary conditions have to be applied. From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Convection → On Lines**. This selection produces the frame shown in Figure 6.24.

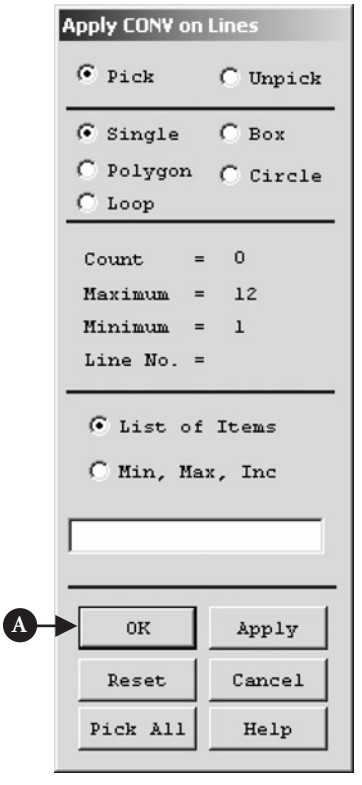


Figure 6.24 Apply CONV on Lines.

First pick the convective lines (facing inside the furnace) of the concrete wall and press [A] **OK** button. The frame shown in Figure 6.25 is created.

As seen in Figure 6.25, the following selections were made: [A] **Film coefficient** = 0.208 W/m^2 K and [B] **Bulk temperature** = 673 K, as specified for the concrete wall in the problem formulation.

Again from **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Convection → On Lines**. The frame shown in Figure 6.24 appears. This time pick the exterior lines of the brick wall and press [A] **OK** button. The frame shown in Figure 6.26 appears.

For the outer brick wall, the following selections were made (see the frame in Figure 6.26): [A] **Film coefficient** = 0.068 W/m^2 K and [B] **Bulk temperature** = 253 K as specified for the brick wall in the problem formulation.

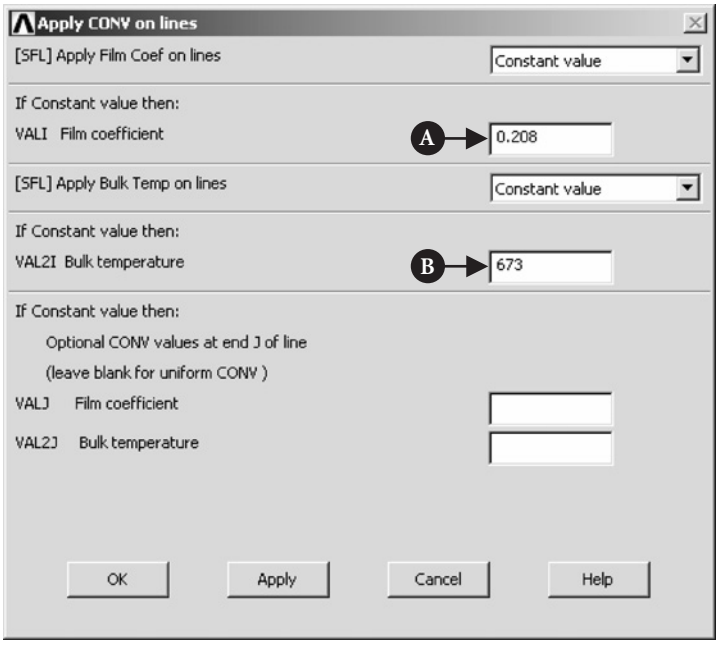


Figure 6.25 Apply CONV on lines (the inner wall).

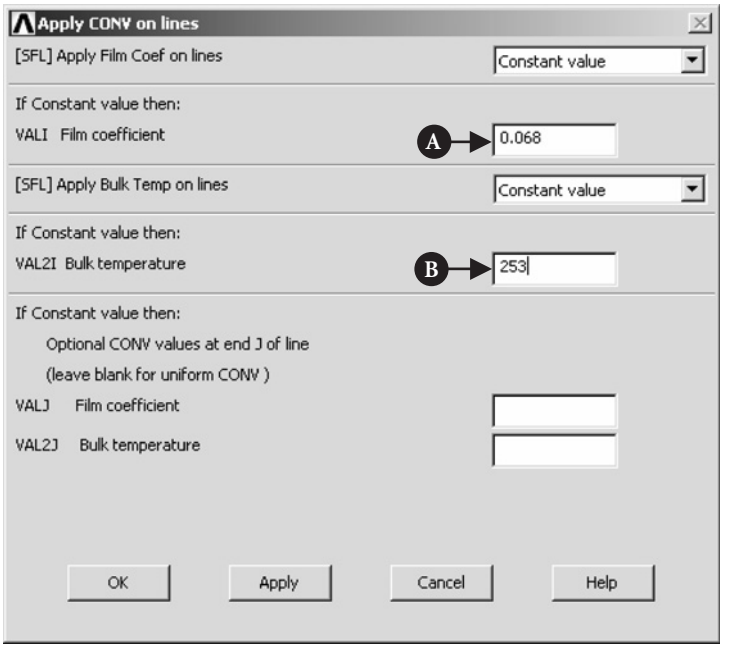


Figure 6.26 Apply CONV on lines (the outer wall).

Finally, to see the applied convective boundary conditions from **Utility Menu** select **PlotCtrls → Symbols**. The frame shown in Figure 6.27 appears.

In the frame shown in Figure 6.27 select [A] **Show pres and convect as = Arrows** and click [B] **OK** button.

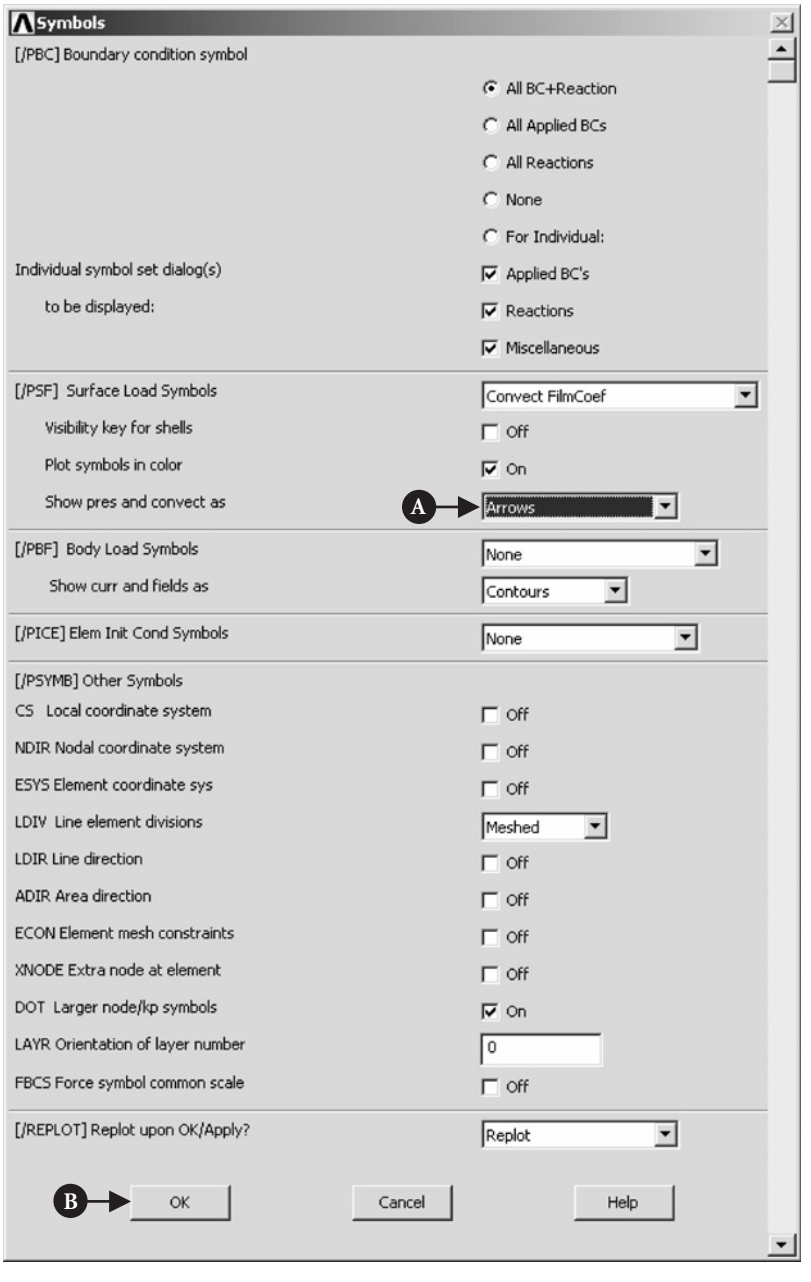


Figure 6.27 Symbols.

From **Utility Menu** select **Plot → Lines** to produce an image shown in Figure 6.28.

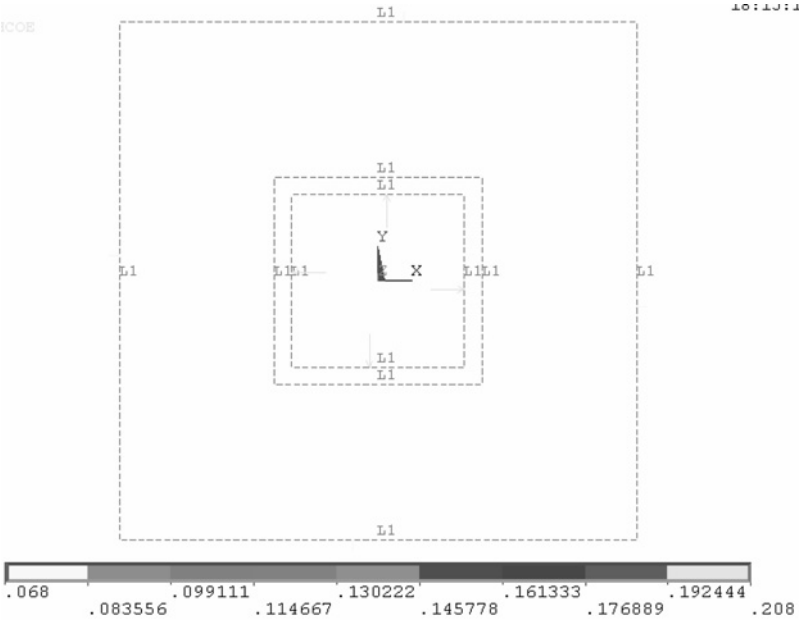


Figure 6.28 Applied convective boundary conditions.

To solve the problem select from **ANSYS Main Menu**, **Solution → Solve → Current LS**. Two frames appear. One gives summary of solution options. After checking correctness of the options, it should be closed using the menu at the top of the frame. The other frame is shown in Figure 6.29. Clicking [A] **OK** button initiates solution process.

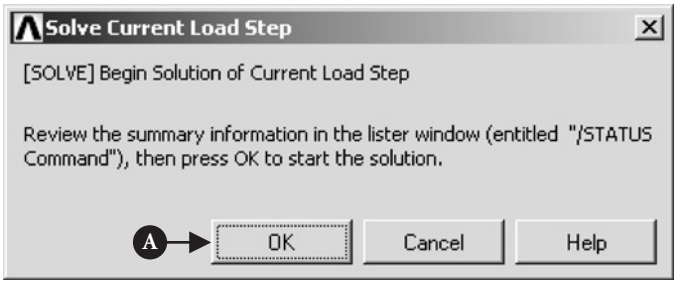


Figure 6.29 Solve Current Load Step.

6.2.4 Postprocessing

The end of a successful solution process is denoted by the message "solution is done." The postprocessing phase can be started. First it is necessary to obtain information about temperatures and heat fluxes across the furnace's walls.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 6.30 appears.

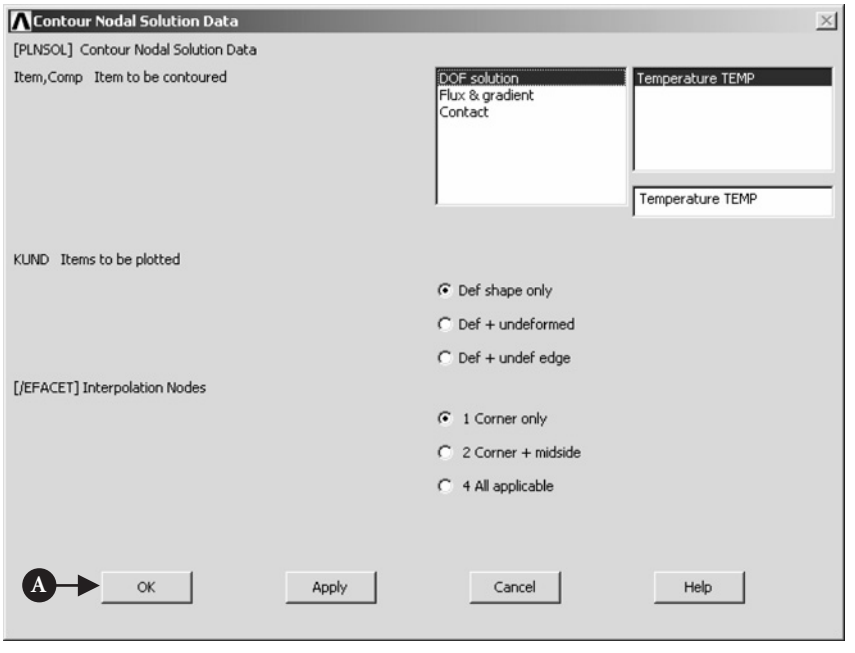


Figure 6.30 Contour Nodal Solution Data.

Selections made are shown in Figure 6.30. Clicking [A] **OK** button results in the graph shown in Figure 6.31.

In order to observe the heat flow across the walls the following command should be issued: **General Postproc → Plot Results → Vector Plot → Predefined**. This produces the frame shown in Figure 6.32.

Clicking [A] **OK** button produces a graph shown in Figure 6.33.

In order to observe temperature variations across the walls, it is necessary to define the path along which the variations are going to be determined. From **Utility Menu** select **Plot → Areas**. Next, from **ANSYS Main Menu** select **General Postproc → Path Operations → Define Path → On Working Plane**. The resulting frame is shown in Figure 6.34.

By activating [A] **Arbitrary path** button and clicking [B] **OK**, another frame is produced and is shown in Figure 6.35.

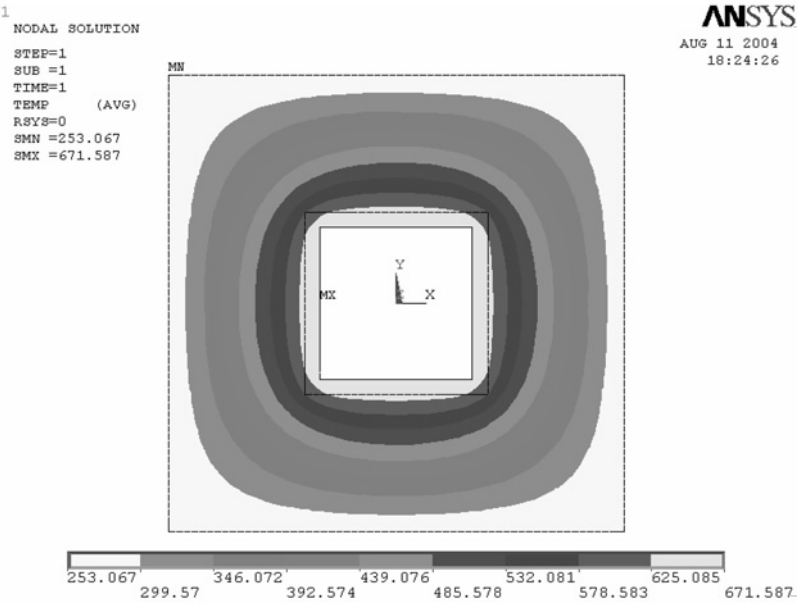


Figure 6.31 Temperature distribution in the furnace as a contour plot.

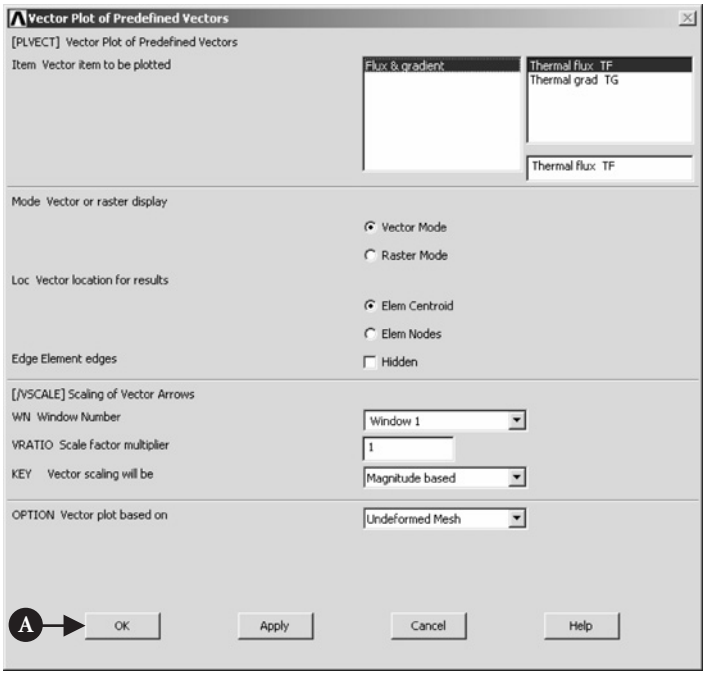


Figure 6.32 Vector Plot of Predefined Vectors.

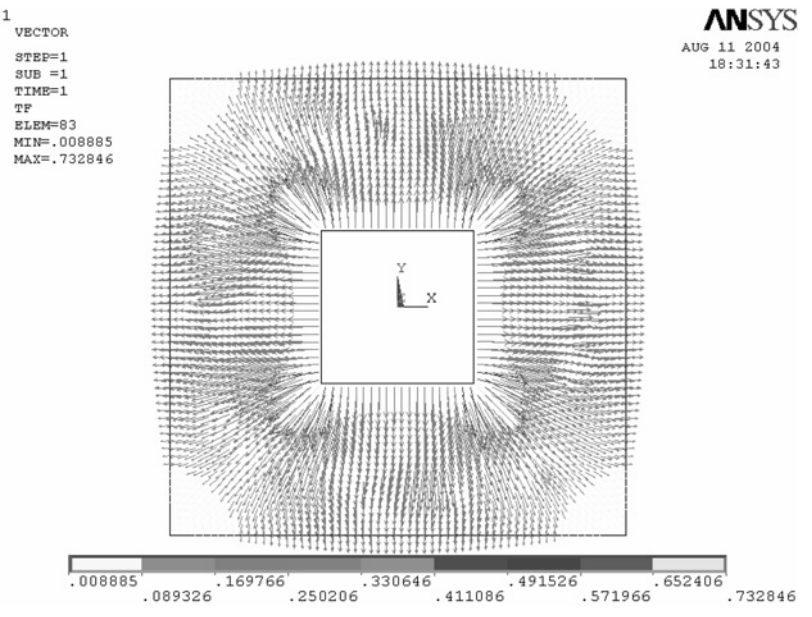


Figure 6.33 Heat flow across the wall plotted as vectors.

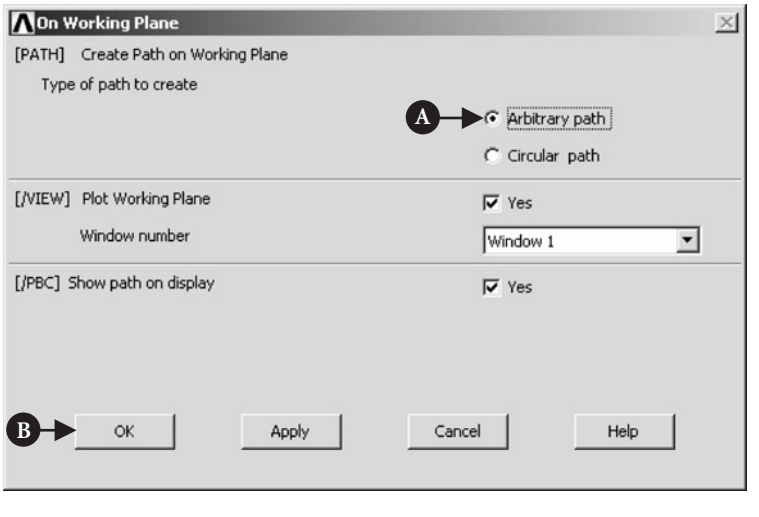


Figure 6.34 On Working Plane (definition of the path).

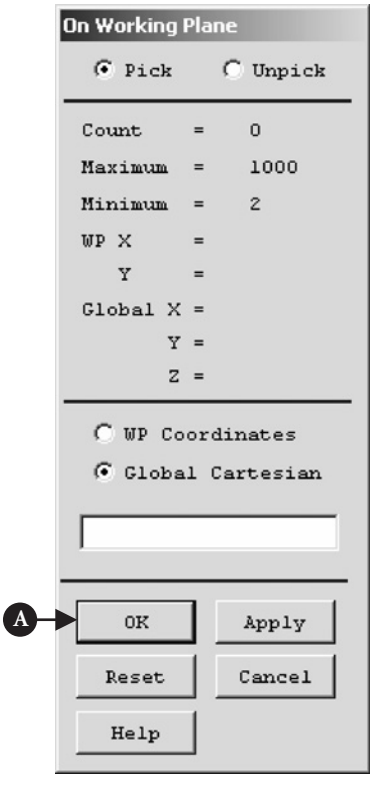


Figure 6.35 On Working Plane (selection of two points defining the path).

Two points should be selected by clicking on the inner line of the concrete wall and moving in Y direction at the right angle by clicking on the outer line of the brick wall. As a result of clicking [A] **OK** button frame shown in Figure 6.36 appears.

In the box [A] **Define Path Name**, write **AB** and click [B] **OK** button.

From **ANSYS Main Menu** select **General Postproc → Path Operations → Map onto Path**. The frame shown in Figure 6.37 appears.

In Figure 6.37, the following selections are made: [A] **Flux & gradient** and [B] **Thermal grad TGX**. By repeating steps described above, recall the frame shown in Figure 6.37. This time select the following: [A] **Flux & gradient** and [B] **Thermal grad TGY**. Finally, recall the frame shown in Figure 6.37 and select: [A] **Flux & gradient** and [B] **Thermal grad TGSUM** as shown in Figure 6.38 and click [C] **OK** button.

From **ANSYS Main Menu** select **General Postproc → Path Operations → Plot Path Item → On Graph**. The frame shown in Figure 6.39 appears.

The selections made [A] are highlighted in Figure 6.39. Pressing [B] **OK** button results in a graph shown in Figure 6.40.

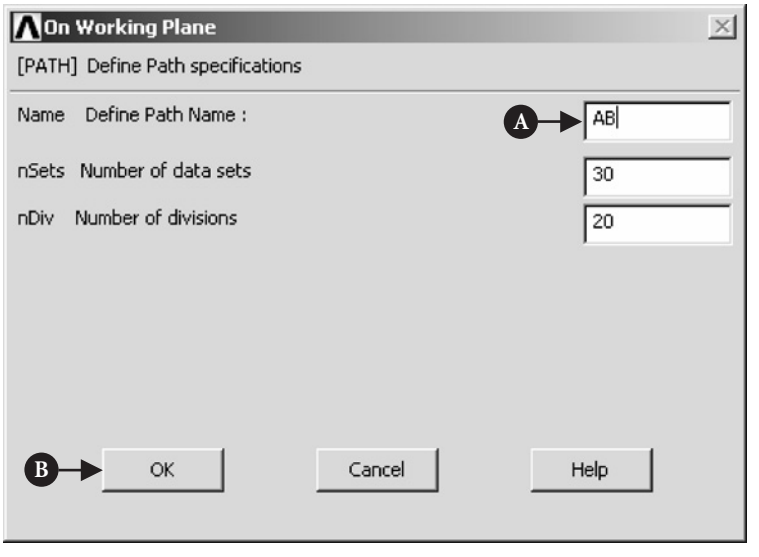


Figure 6.36 On Working Plane (path name: AB).

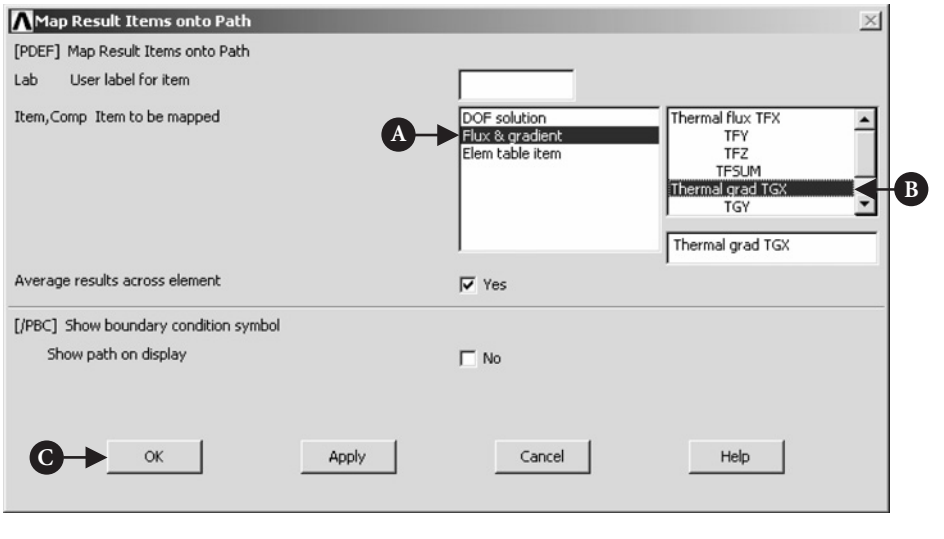


Figure 6.37 Map Result Items onto Path (AB path).

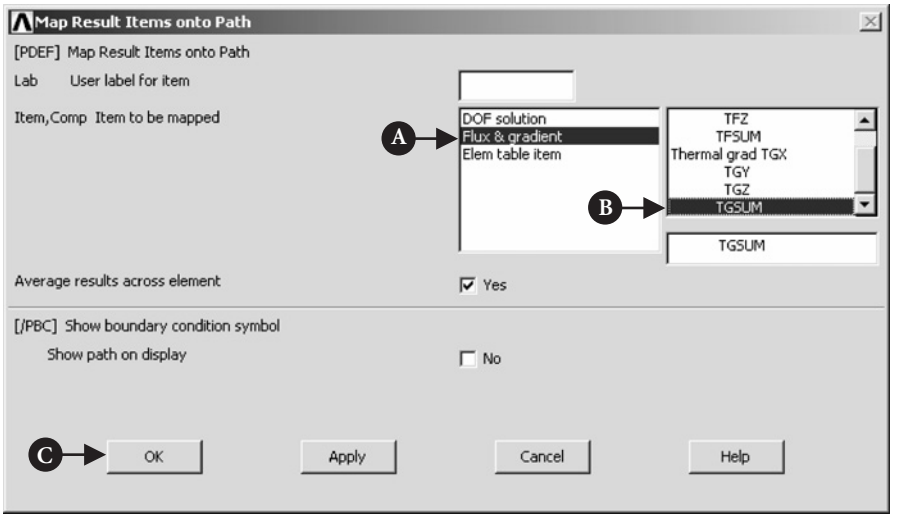


Figure 6.38 Map Result Items onto Path (AB path).

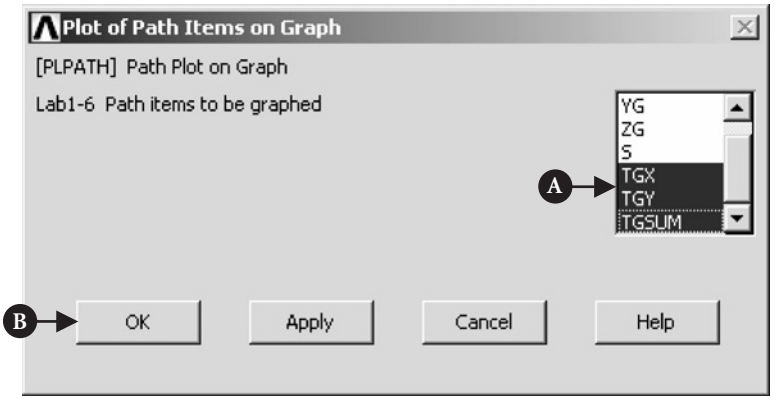


Figure 6.39 Plot of Path Items on Graph.

6.3 Steady-state thermal analysis of a pipe intersection

6.3.1 Description of the problem

A cylindrical tank is penetrated radially by a small pipe at a point on its axis remote from the ends of the tank, as shown in Figure 6.41.

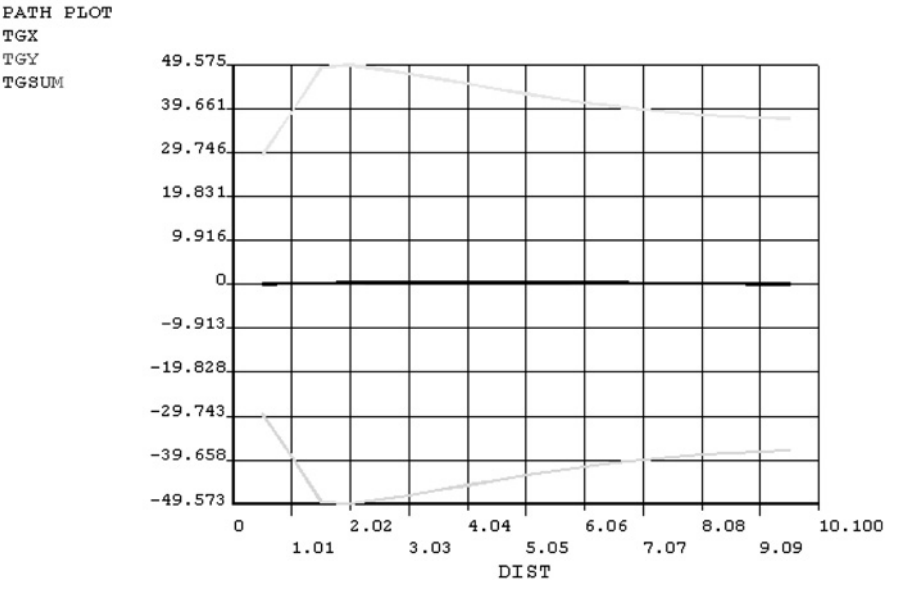


Figure 6.40 Variations of temperature gradients along path AB.

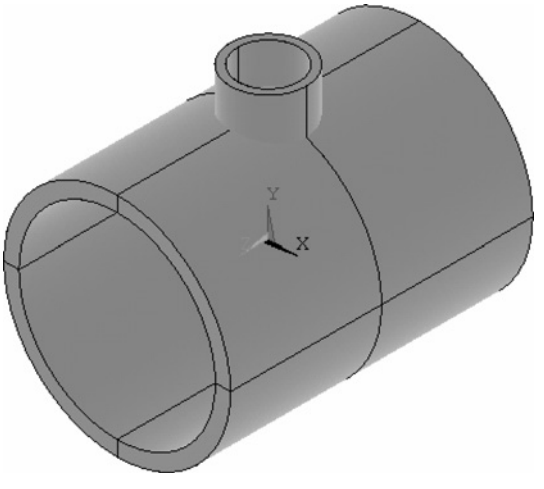


Figure 6.41 Pipe intersection.

The inside of the tank is exposed to a fluid with temperature of 232°C. The pipe experiences a steady flow of fluid with temperature of 38°C, and the two flow regimes are isolated from each other by means of a thin tube. The convection (film) coefficient in the pipe varies with the metal temperature and is thus expressed as a

material property. The objective is to determine the temperature distribution at the pipe–tank junction.

The following data describing the problem are given:

- Inside diameter of the pipe = 8 mm
- Outside diameter of the pipe = 10 mm
- Inside diameter of the tank = 26 mm
- Outside diameter of the tank = 30 mm
- Inside bulk fluid temperature, tank = 232°C
- Inside convection coefficient, tank = 4.92 W/m^2 °C
- Inside bulk fluid temperature, pipe = 38°C
- Inside convection coefficient (pipe) varies from about 19.68 to 39.36 W/m^2 °C, depending on temperature.

Table 6.1 provides information about variation of the thermal parameters with temperature.

Table 6.1 Variation of the thermal parameters with temperature

Temperature [°C]	**21**	**93**	**149**	**204**	**260**
Convection coefficient [W/m^2 °C]	41.918	39.852	34.637	27.06	21.746
Density [kg/m^3]	7889	7889	7889	7889	7889
Conductivity [J/s m °C]	0.2505	0.267	0.2805	0.294	0.3069
Specific heat [J/kg °C]	6.898	7.143	7.265	7.448	7.631

The assumption is made that the quarter symmetry is applicable and that, at the terminus of the model (longitudinal and circumferential cuts in the tank), there is sufficient attenuation of the pipe effects such that these edges can be held at 232°C.

The solid model is constructed by intersecting the tank with the pipe and then removing the internal part of the pipe using Boolean operation.

Boundary temperatures along with the convection coefficients and bulk fluid temperatures are dealt with in the solution phase, after which a static solution is executed.

Temperature contours and thermal flux displays are obtained in postprocessing.

Details of steps taken to create the model of pipe intersecting with tank are outlined below.

6.3.2 PREPARATION FOR MODEL BUILDING

From **ANSYS Main Menu** select **Preferences**. This frame is shown in Figure 6.42.

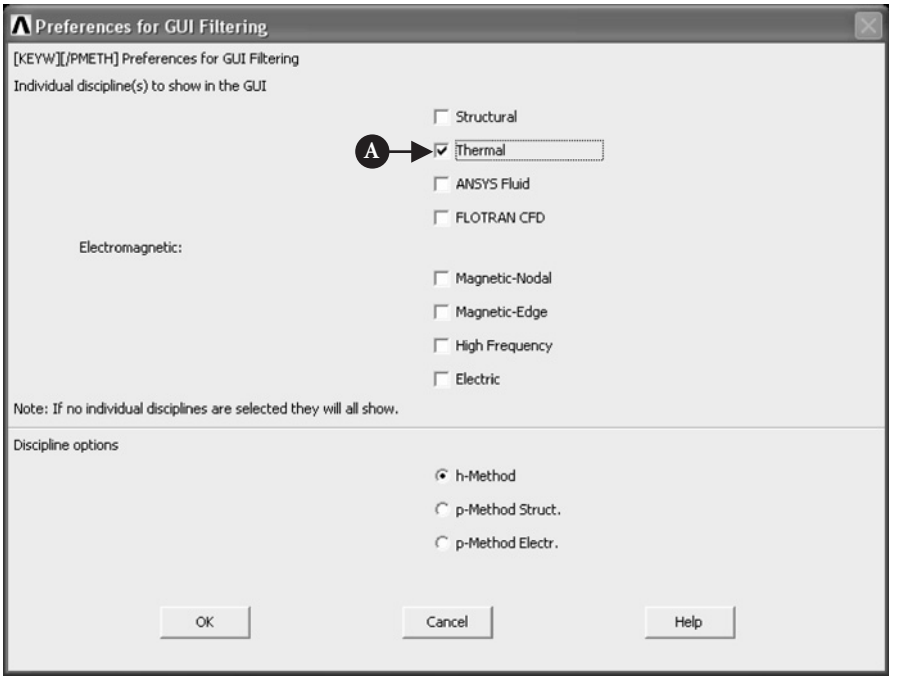


Figure 6.42 Preferences: Thermal.

Depending on the nature of analysis to be attempted an appropriate analysis type should be selected. In the problem considered here [A] **Thermal** was selected as shown in Figure 6.42.

From **ANSYS Main Menu** select **Preprocessor** and then **Element Type** and **Add/Edit/Delete**. The frame shown in Figure 6.43 appears.

Clicking [A] **Add** button activates a new set of options which are shown in Figure 6.44.

Figure 6.44 indicates that for the problem considered here the following was selected: [A] **Thermal Mass → Solid** and [B] **20node 90**.

From **ANSYS Main Menu** select **Material Props** and then **Material Models**. Figure 6.45 shows the resulting frame.

From the options listed on the right hand select [A] **Thermal** as shown in Figure 6.45.

Next select [B] **Conductivity, Isotropic**. The frame shown in Figure 6.46 appears.

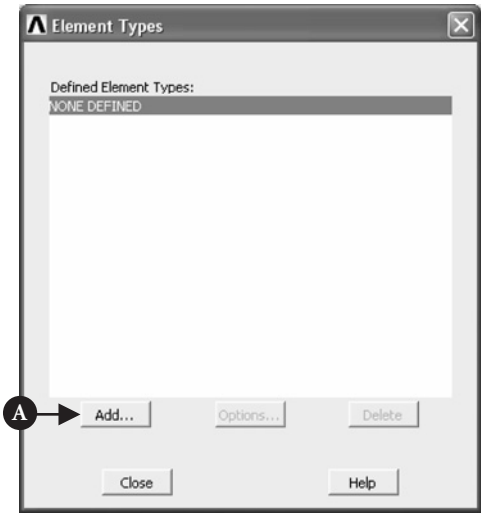


Figure 6.43 Element Types selection.

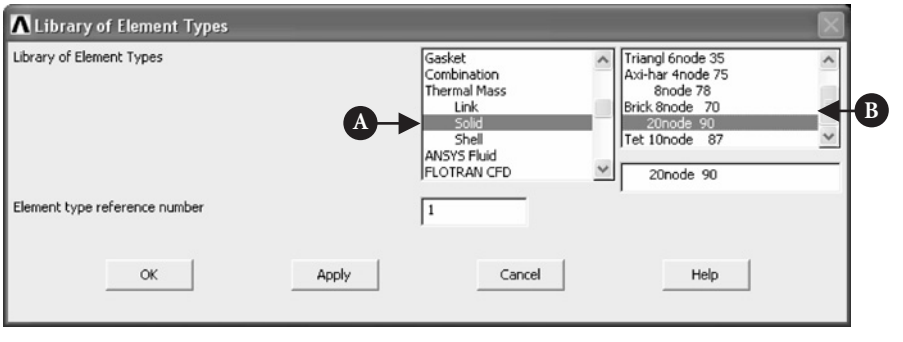


Figure 6.44 Library of Element Types.

Then, using conductivity versus temperature values, listed in Table 6.1, appropriate figures should be typed in as shown in Figure 6.46.

By selecting [C] **Specific Heat** option on the right-hand column (see Figure 6.45), the frame shown in Figure 6.47 is produced.

Appropriate values of specific heat versus temperature, taken from Table 6.1, are typed as shown in Figure 6.47.

The next material property to be defined is density. According to Table 6.1, density is constant for all temperatures used. Therefore, selecting [D] **Density**

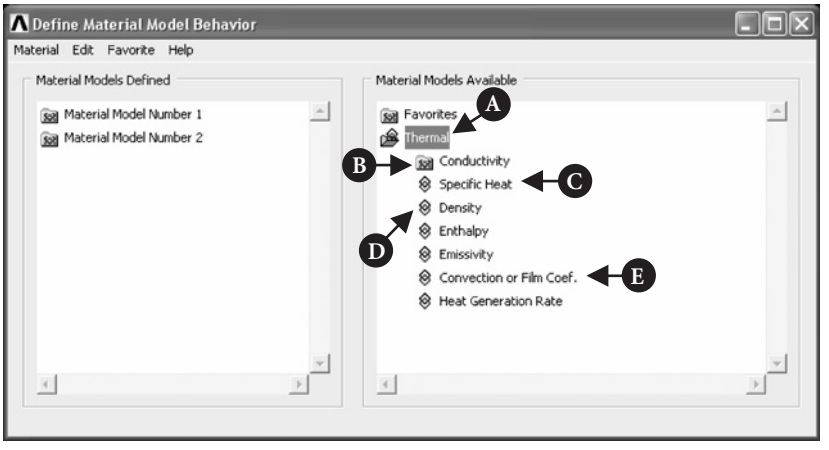


Figure 6.45 Define Material Model Behavior.

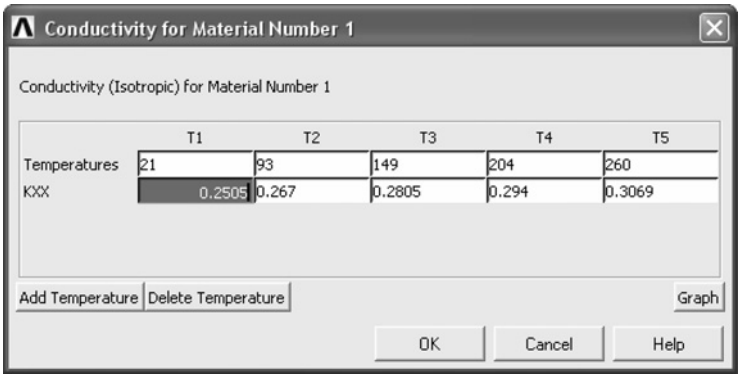


Figure 6.46 Conductivity for Material Number 1.

from the right-hand column (see Figure 6.45), results in the frame shown in Figure 6.48.

Density of 7888.8 kg/m^3 is typed in the box shown in Figure 6.48.

All the above properties were used to characterize Material Number 1. Convection or film coefficient is another important parameter characterizing the system being analyzed. However, it is not a property belonging to Material Number 1 (material of the tank and pipe) but to a thin film formed by the liquid on solid surfaces. It is a different entity and, therefore, is called Material Number 2.

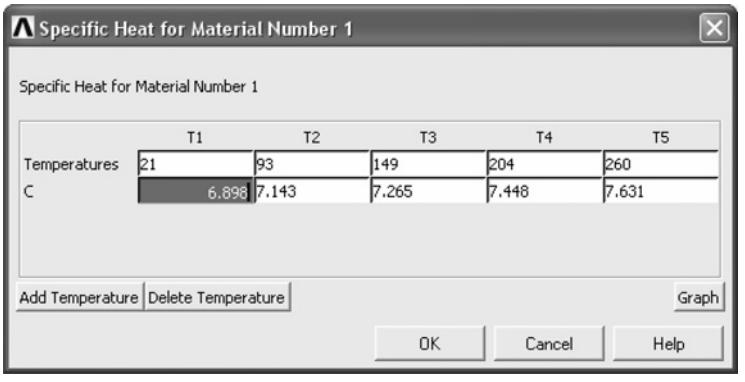


Figure 6.47 Specific Heat for Material Number 1.

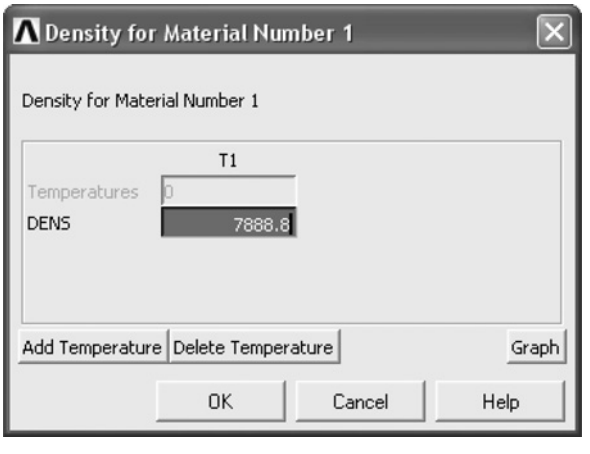


Figure 6.48 Density for Material Number 1.

Selecting [E] **Convection or Film Coef.** (see Figure 6.45) results in the frame shown in Figure 6.49. Appropriate values of film coefficient for various temperatures, taken from Table 6.1, are introduced as shown in Figure 6.49.

6.3.3 CONSTRUCTION OF THE MODEL

The entire model of the pipe intersecting with the tank is constructed using one of the three-dimensional (3D) primitive shapes, that is cylindrical. Only one-quarter of

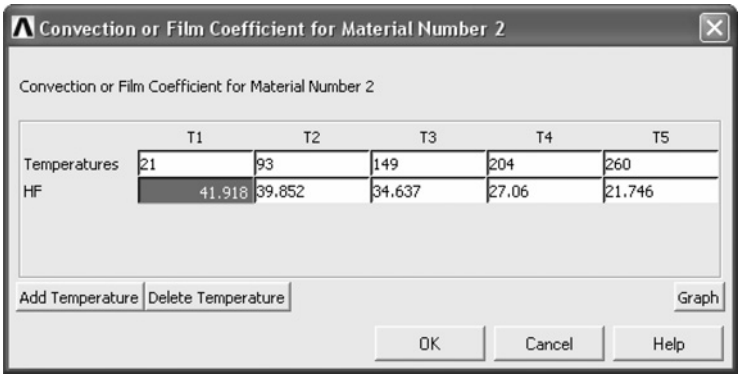


Figure 6.49 Convection or Film Coefficient for Material Number 2.

the tank–pipe assembly will be sufficient to use in the analysis. From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. Figure 6.50 shows the resulting frame.

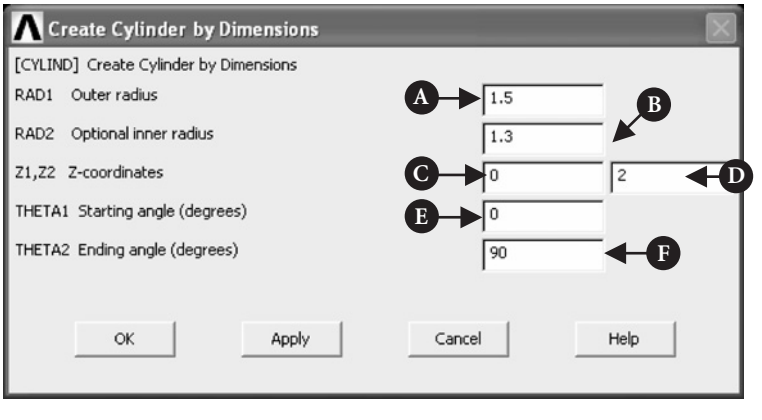


Figure 6.50 Create Cylinder by Dimensions.

In Figure 6.50, as shown, the following inputs are made: [A] **RAD1** = 1.5 cm; [B] **RAD2** = 1.3 cm; [C] **Z1** = 0; [D] **Z2** = 2 cm; [E] **THETA1** = 0; [F] **THETA2** = 90.

As the pipe axis is at right angle to the cylinder axis, therefore it is necessary to rotate the working plane (WP) to the pipe axis by 90°. This is done by selecting from **Utility Menu WorkPlane → Offset WP by Increments**. The resulting frame is shown in Figure 6.51.

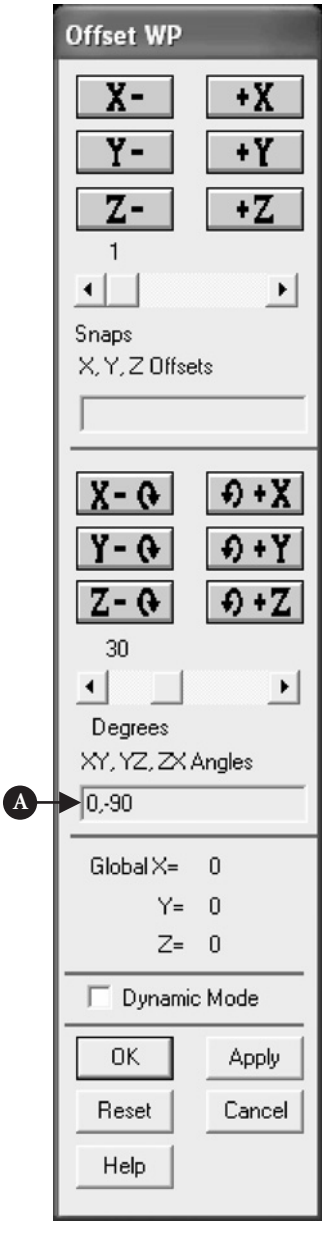


Figure 6.51 Offset WP by Increments.

In Figure 6.51, the input is shown as [A] **XY** = 0; **YZ** = −90 and the **ZX** is left unchanged from default value. Next, from **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. Figure 6.52 shows the resulting frame.

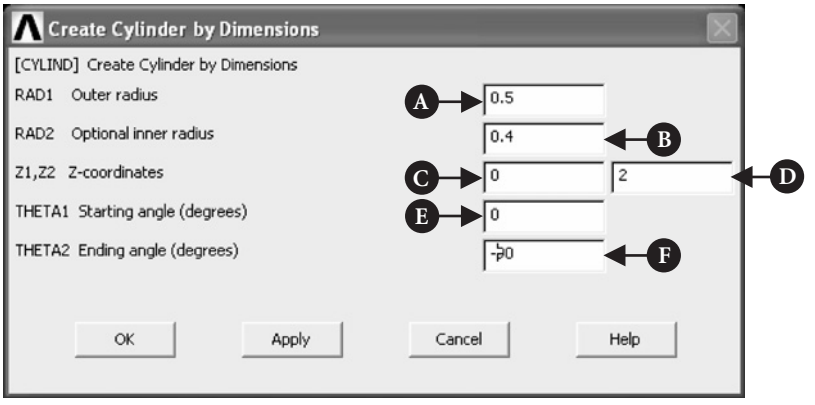


Figure 6.52 Create Cylinder by Dimensions.

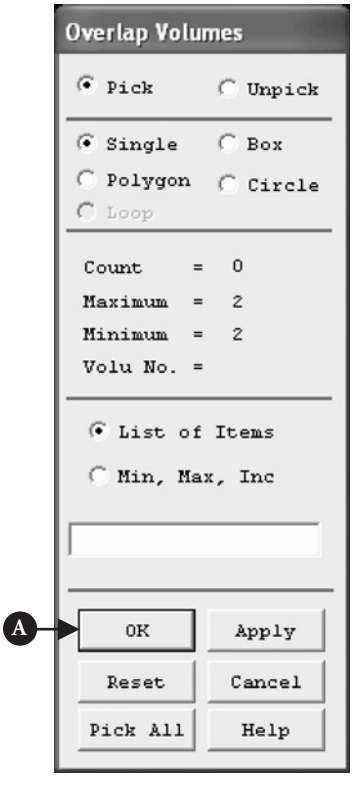


Figure 6.53 Overlap Volumes (Booleans Operation).

In Figure 6.52, as shown, the following inputs are made: [A] **RAD1** = 0.5 cm; [B] **RAD2** = 0.4 cm; [C] **Z1** = 0; [D] **Z2** = 2 cm; [E] **THETA1** = 0; [F] **THETA2** = −90.

After that the WP should be set to the default setting by inputting in Figure 6.51 YZ = 90 this time. As the cylinder and the pipe are separate entities, it is necessary to overlap them in order to make the one component. From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Operate → Booleans → Overlap → Volumes**. The frame shown in Figure 6.53 is created.

Pick both elements, that is cylinder and pipe, and press [A] **OK** button to execute the selection. Next activate volume numbering which will be of help when carrying out further operations on volumes. This is done by selecting from **Utility Menu PlotCtrls → Numbering** and checking, in the resulting frame, **VOLU** option.

Finally, 3D view of the model should be set by selecting from **Utility Menu** the following: **PlotCtrls → View Settings**. The resulting frame is shown in Figure 6.54.

The following inputs should be made (see Figure 6.54): [A] **XV** = −3; [B] **YV** = −1; [C] **ZV** = 1 in order to plot the model as shown in Figure 6.55. However, this is not the only possible view of the model and any other preference may be chosen.

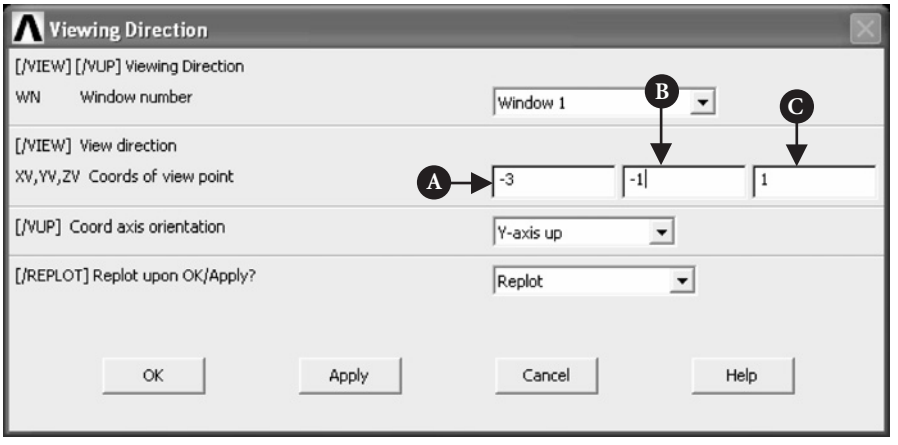


Figure 6.54 View Settings.

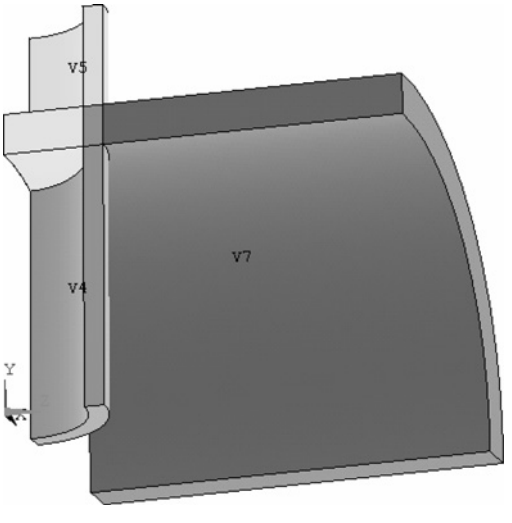


Figure 6.55 Quarter symmetry model of the tank–pipe intersection.

Certain volumes of the models, shown in Figure 6.55, are redundant and should be deleted. From **ANSYS Main Menu** select **Preprocessor → Modelling → Delete → Volume and Below**. Figure 6.56 shows the resulting frame.

Volumes V4 and V3 (a corner of the cylinder) should be picked and [A] **OK** button pressed to implement the selection. After the delete operation, the model looks like that shown in Figure 6.57.

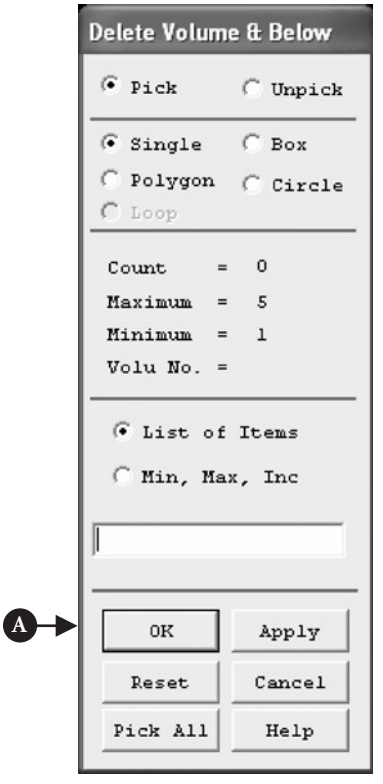


Figure 6.56 Delete Volume and Below.

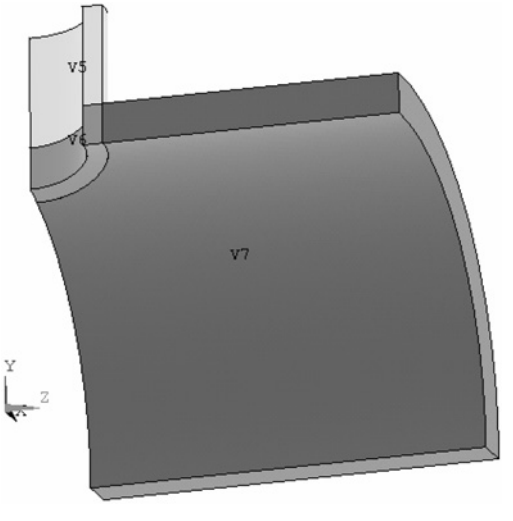


Figure 6.57 Quarter symmetry model of the tank–pipe intersection.

Finally, volumes V5, V6, and V7 should be added in order to create a single volume required for further analysis. From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Add → Volumes**. The resulting frame asks for picking volumes to be added. Pick all three volumes, that is V5, V6, and V7, and click **OK** button to implement the operation. Figure 6.58 shows the model of the pipe intersecting the cylinder as one volume V1.

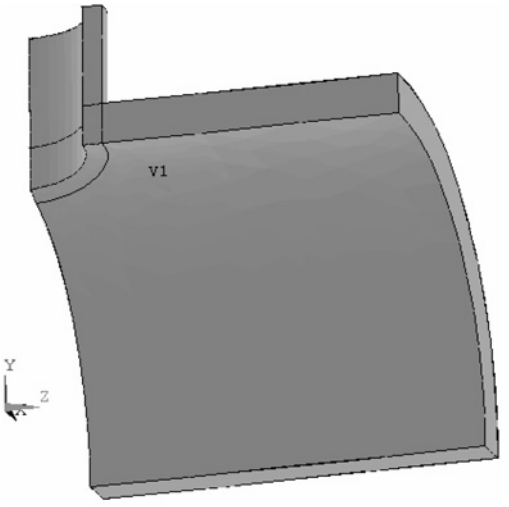


Figure 6.58 Quarter symmetry model of the tank–pipe intersection represented by a single volume V1.

Meshing of the model usually begins with setting size of elements to be used. From **ANSYS Main Menu** select **Meshing → Size Cntrls → SmartSize → Basic**. A frame shown in Figure 6.59 appears.

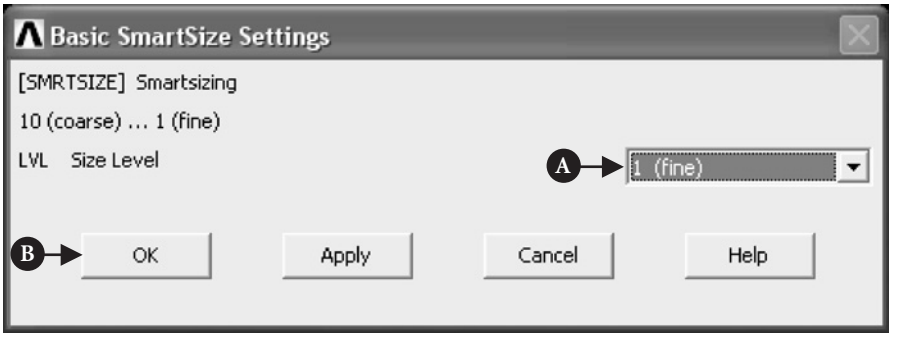


Figure 6.59 Basic SmartSize Settings.

For the case considered, [A] **Size Level** – 1 (fine) was selected as shown in Figure 6.59. Clicking [B] **OK** button implements the selection. Next, from **ANSYS Main Menu** select **Mesh → Volumes → Free**. The frame shown in Figure 6.60 appears. Select the volume to be meshed and click [A] **OK** button.

The resulting network of elements is shown in Figure 6.61.

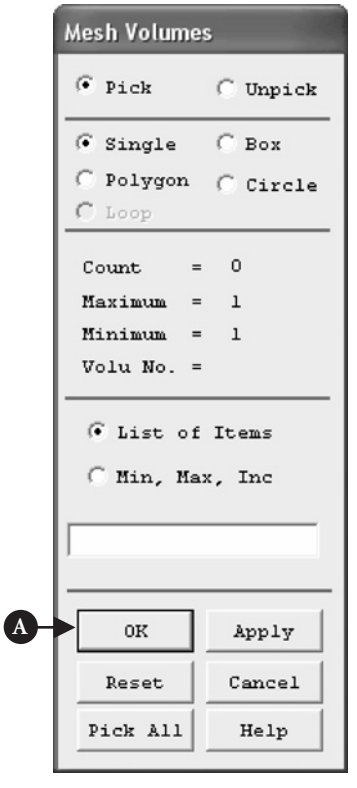


Figure 6.60 Mesh Volumes frame.

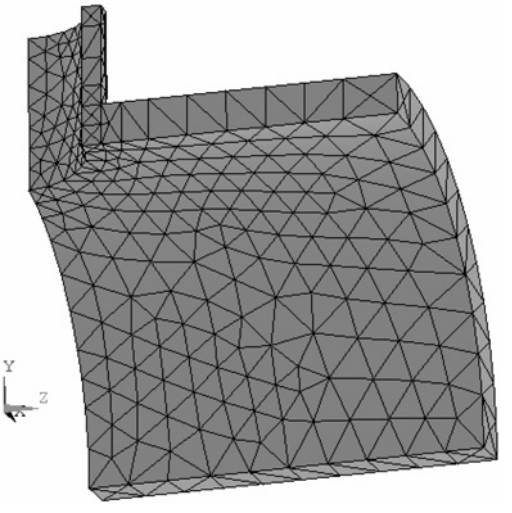


Figure 6.61 Meshed quarter symmetry model of the tank–pipe intersection.

6.3.4 Solution

The meshing operation ends the model construction and the Preprocessor stage. The solution stage can now be started. From **ANSYS Main Menu** select **Solution → Analysis Type → New Analysis**. Figure 6.62 shows the resulting frame.

Activate [A] **Steady-State** button. Next, select **Solution → Analysis Type → Analysis Options**. In the resulting frame, shown in Figure 6.63, select [A] **Program chosen** option.

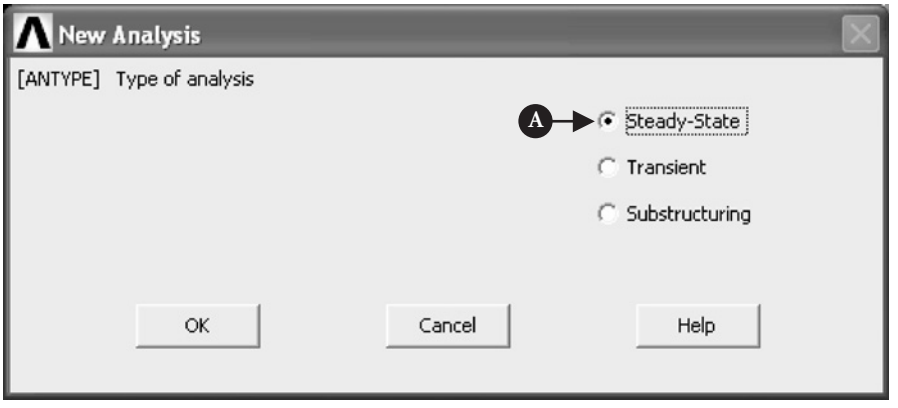


Figure 6.62 New Analysis window.

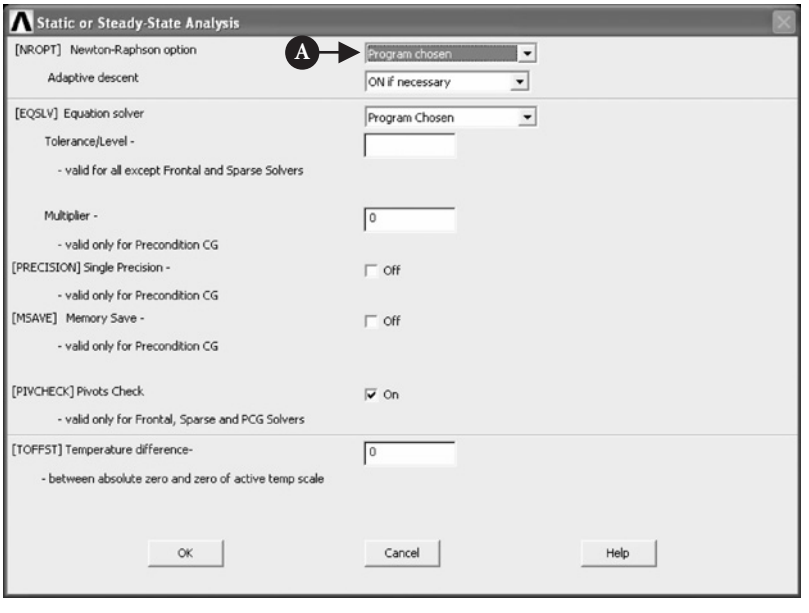


Figure 6.63 Analysis Options.

In order to set starting temperature of 232°C at all nodes select **Solution → Define Loads → Apply → Thermal → Temperature → Uniform Temp**. Figure 6.64 shows the resulting frame. Input [A] **Uniform temperature** = 232°C as shown in Figure 6.64.

From **Utility Menu** select **WorkPlane → Change Active CS to → Specified Coord Sys**. As a result of that the frame shown in Figure 6.65 appears.

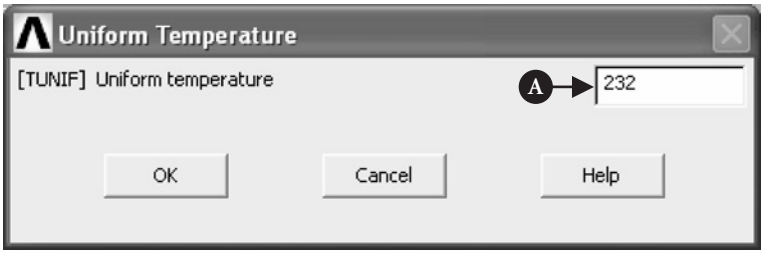


Figure 6.64 Temperature selection.

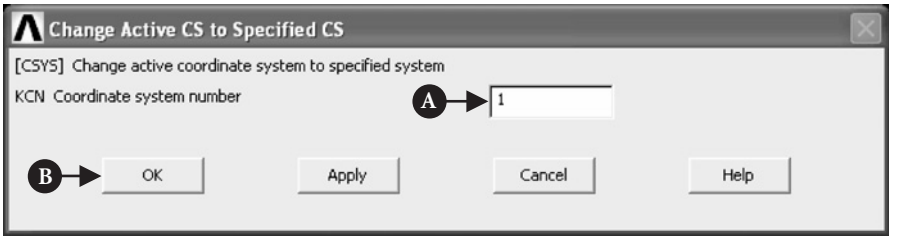


Figure 6.65 Change Coordinate System.

In order to re-establish a cylindrical coordinate system with Z as the axis of rotation select [A] **Coordinate system number** = 1 and press [B] **OK** button to implement the selection.

Nodes on inner surface of the tank ought to be selected to apply surface loads to them. The surface load relevant in this case is convection load acting on all nodes located on inner surface of the tank. From **Utility Menu** select **Select → Entities**. The frame shown in Figure 6.66 appears.

From the first pull down menu select [A] **Nodes**, from the second pull down menu select [B] **By Location**. Also, activate [C] **X coordinates** button and enter [D] **Min,Max** = 1.3 (inside radius of the tank). All the four required steps are shown in Figure 6.66. When the subset of nodes on inner surface of the tank is selected then the convection load at all nodes has to be applied. From **ANSYS Main Menu** select **Solution → Define Load → Apply → Thermal → Convection → On nodes**. The resulting frame is shown in Figure 6.67.

Press [A] **Pick All** in order to call up another frame shown in Figure 6.68.

Inputs into the frame of Figure 6.68 are shown as: [A] **Film coefficient** = 4.92 and [B] **Bulk temperature** = 232. Both quantities are taken from Table 6.1.

From **Utility Menu** select **Select → Entities** in order to select a subset of nodes located at the far edge of the tank. The frame shown in Figure 6.69 appears.

From the first pull down menu select [A] **Nodes**, from the second pull down menu select [B] **By Location**. Also, activate [C] **Z coordinates** button and [D] enter **Min,Max** = 2 (the length of the tank in Z-direction). All the four required steps are

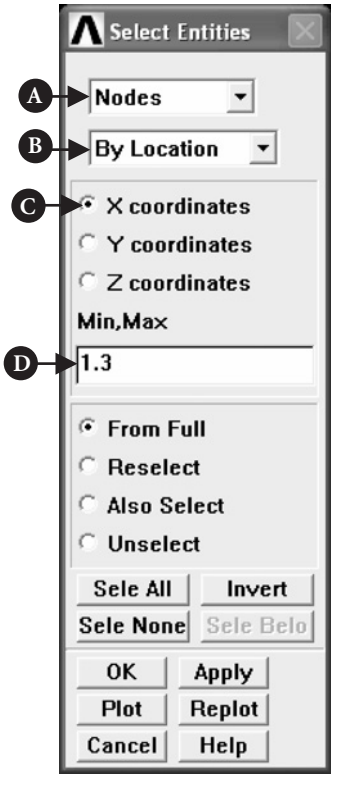


Figure 6.66 Select Entities.

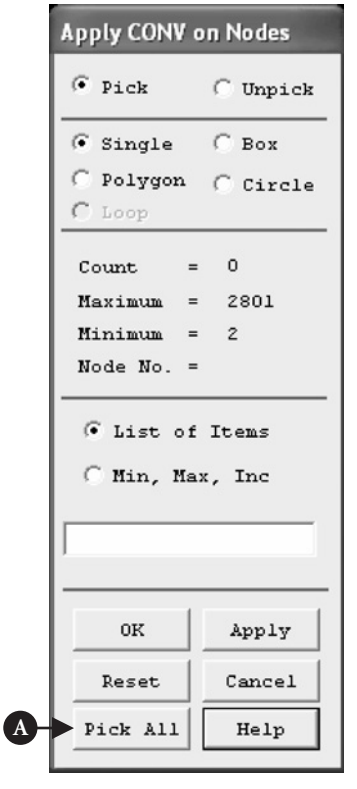


Figure 6.67 Apply Thermal Convection on Nodes.

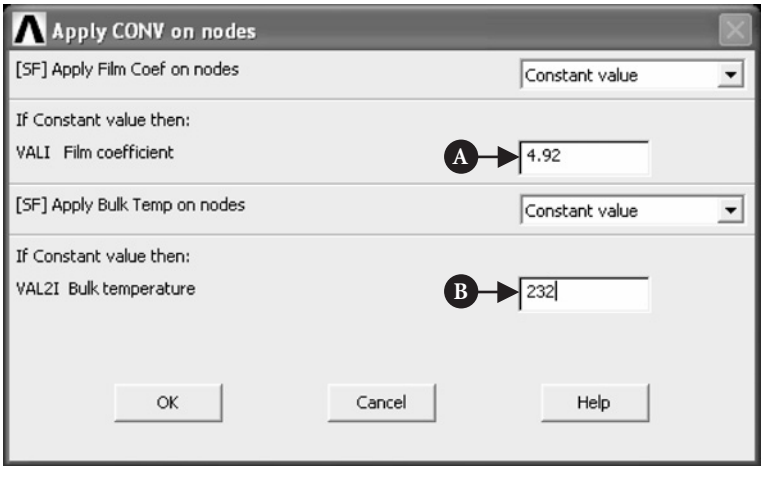


Figure 6.68 Select All Nodes.

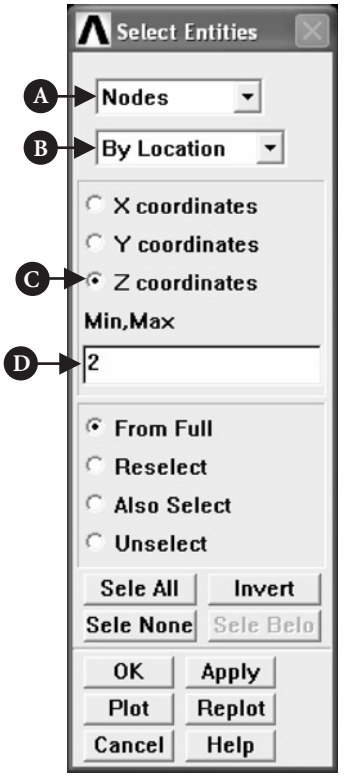


Figure 6.69 Select Entities.

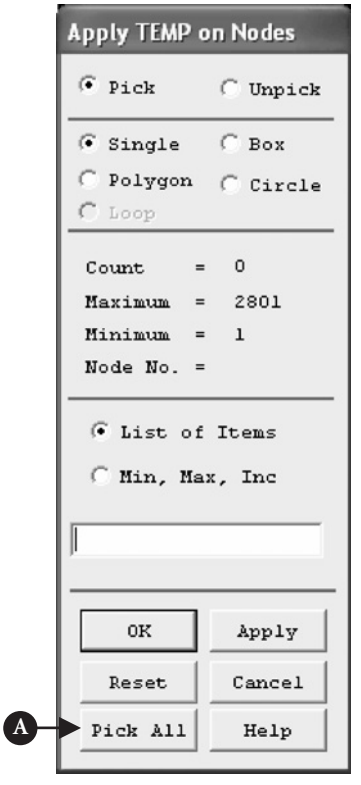


Figure 6.70 Select All Nodes.

shown in Figure 6.69. Next, constraints at nodes located at the far edge of the tank (additional subset of nodes just selected) have to be applied. From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Temperature → On Nodes**. The frame shown in Figure 6.70 appears.

Click [A] **Pick All** as shown in Figure 6.70. This action brings another frame shown in Figure 6.71.

Activate both [A] **All DOF** and **TEMP** and input [B] **TEMP value** = 232°C as shown in Figure 6.71. Finally, click [C] **OK** to apply temperature constraints on nodes at the far edge of the tank. The steps outlined above should be followed to apply constraints at nodes located at the bottom of the tank. From **Utility Menu** select **Select → Entities**. The frame shown in Figure 6.72 appears.

From the first pull down menu select [A] **Nodes**, from the second pull down menu select [B] **By Location**. Also, activate [C] **Y coordinates** button and [D] enter **Min,Max** = 0 (location of the bottom of the tank in Y-direction). All the four required steps are shown in Figure 6.72. Next, constraints at nodes located at the bottom of the tank (additional subset of nodes selected above) have to be applied. From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Temperature → On Nodes**. The frame shown in Figure 6.70 appears. As shown in

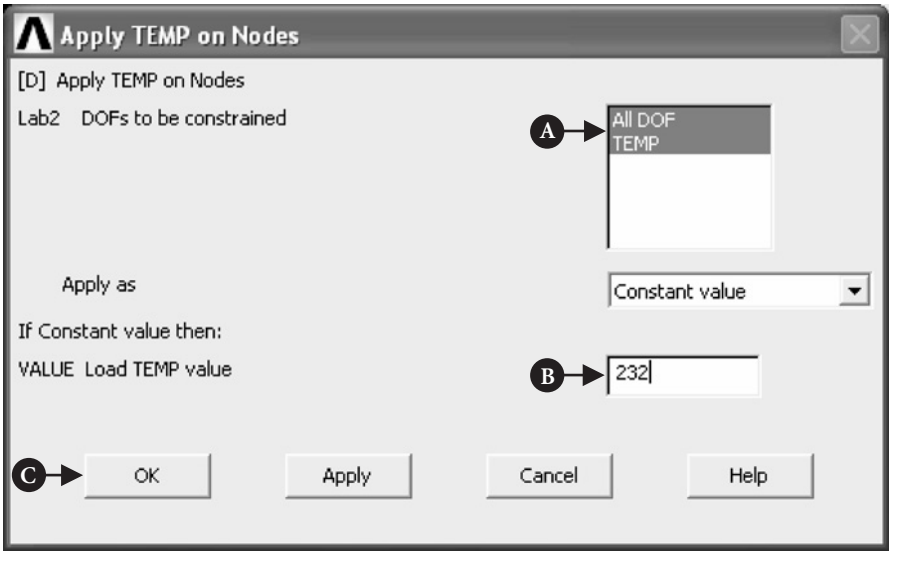


Figure 6.71 Apply Temperature to All Nodes.

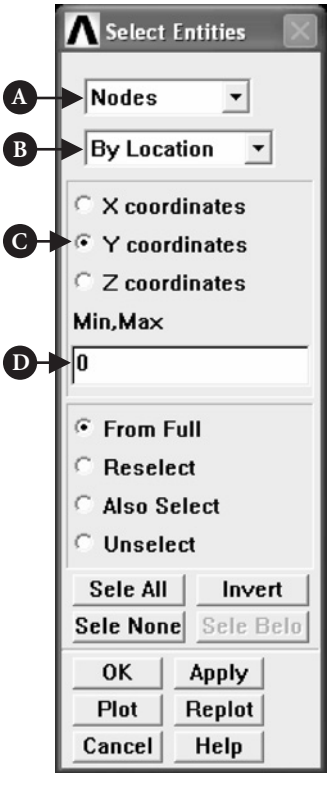


Figure 6.72 Select Entities.

Figure 6.70, click [A] **Pick All** in order to bring the frame shown in Figure 6.71. As before, activate both [A] **All DOF** and **TEMP** and input [B] **TEMP value** = 232°C. Clicking [C] **OK** applies temperature constraints on nodes at the bottom of the tank.

Now, it is necessary to rotate the WP to the pipe axis. From **Utility Menu** select **WorkPlane → Offset WP by Increments**. Figure 6.73 shows the resulting frame.

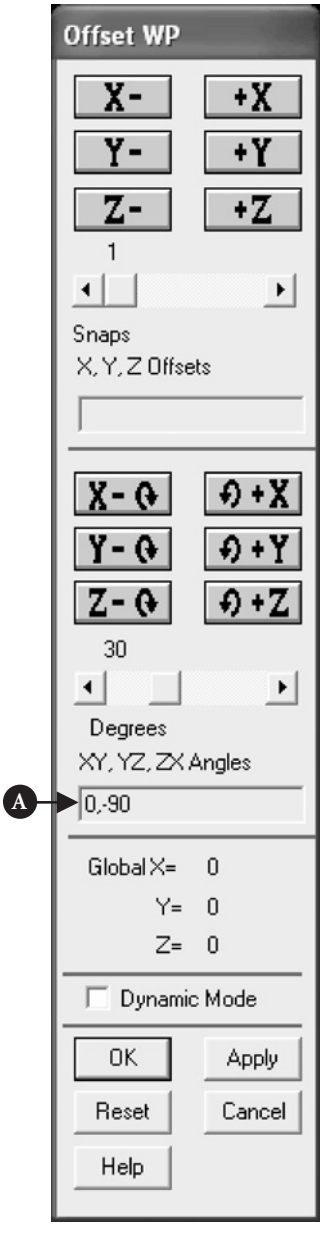


Figure 6.73 Offset WP by Increments.

In degrees box input [A] **XY** = 0 and **YZ** = −90 as shown. Having WP rotated to the pipe axis, a local cylindrical coordinate system has to be defined at the origin of the WP. From **Utility Menu** select **WorkPlane → Local Coordinate Systems → Create local CS → At WP Origin**. The resulting frame is shown in Figure 6.74.

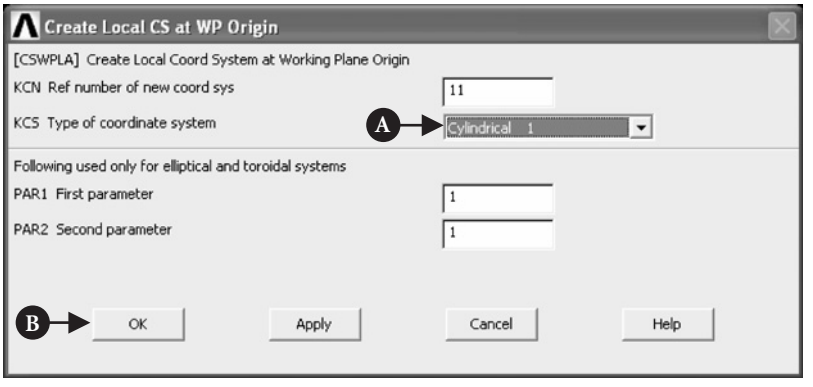


Figure 6.74 Create Local CS.

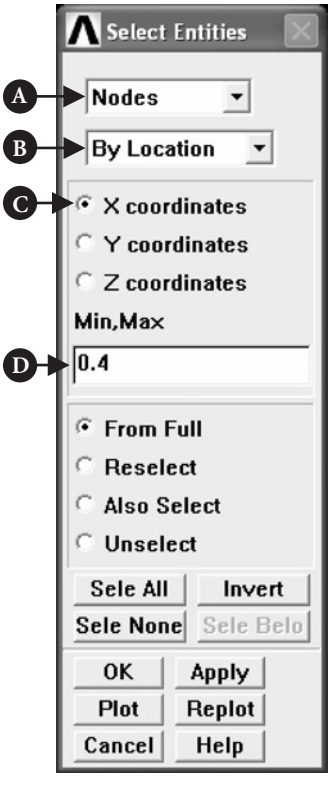


Figure 6.75 Select Entities.

From the pull down menu select [A] **Cylindrical 1** and click [B] **OK** button to implement the selection. The analysis involves nodes located on inner surface of the pipe. In order to include this subset of nodes, from **Utility Menu** select **Select → Entities**. Figure 6.75 shows the resulting frame.

From the first pull down menu select [A] **Nodes**, from the second pull down menu select [B] **By Location**. Also, activate [C] **X coordinates** button and [D] enter **Min,Max** = 0.4 (inside radius of the pipe). All the four required steps are shown in Figure 6.75. From **ANSYS Main Menu** select **Solution → Define Load → Apply → Thermal → Convection → On nodes**. In the resulting frame (shown in Figure 6.67), press [A] **Pick All** and the next frame, shown in Figure 6.76, appears.

Input [A] **Film coefficient** = −2 and [B] **Bulk temperature** = 38 as shown in Figure 6.76. Pressing [C] **OK** button implements the selections. The values inputted are taken from Table 6.1. The final action is to select all entities involved with a single command. Therefore, from **Utility Menu** select **Select → Everything**. For the loads to be applied to tank and pipe surfaces in the form of arrows from **Utility Menu**

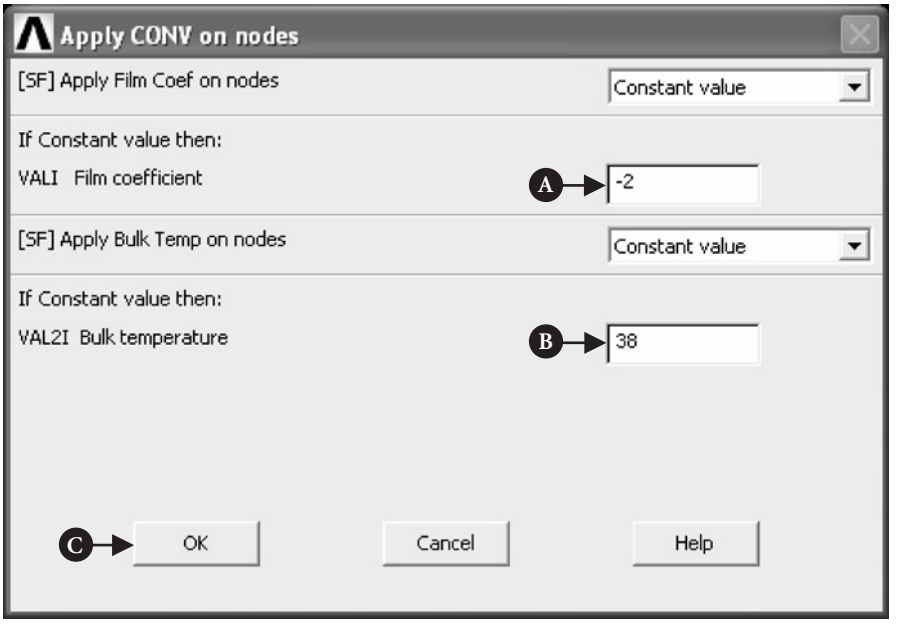


Figure 6.76 Apply CONV on Nodes.

select **PlotCtrls → Symbols**. The frame in Figure 6.77 shows the required selection: [A] **Arrows**.

From **Utility Menu** selecting **Plot → Nodes** results in Figure 6.78 where surface loads at nodes as shown as arrows.

From **Utility Menu** select **WorkPlane → Change Active CS to → Specified Coord Sys** in order to activate previously defined coordinate system. The frame shown in Figure 6.79 appears.

Input [A] **KCN** (coordinate system number) = 0 to return to Cartesian system. Additionally from **ANSYS Main Menu** select **Solution → Analysis Type → Sol'n Controls**. As a result, the frame shown in Figure 6.80 appears.

Input the following [A] **Automation time stepping** = On and [B] **Number of substeps** = 50 as shown in Figure 6.80. Finally, from **ANSYS Main Menu** select **Solve → Current LS** and in the appearing dialog box click **OK** button to start the solution process.

6.3.5 POSTPROCESSING STAGE

When the solution is done, the next stage is to display results in a form required to answer questions posed by the formulation of the problem.

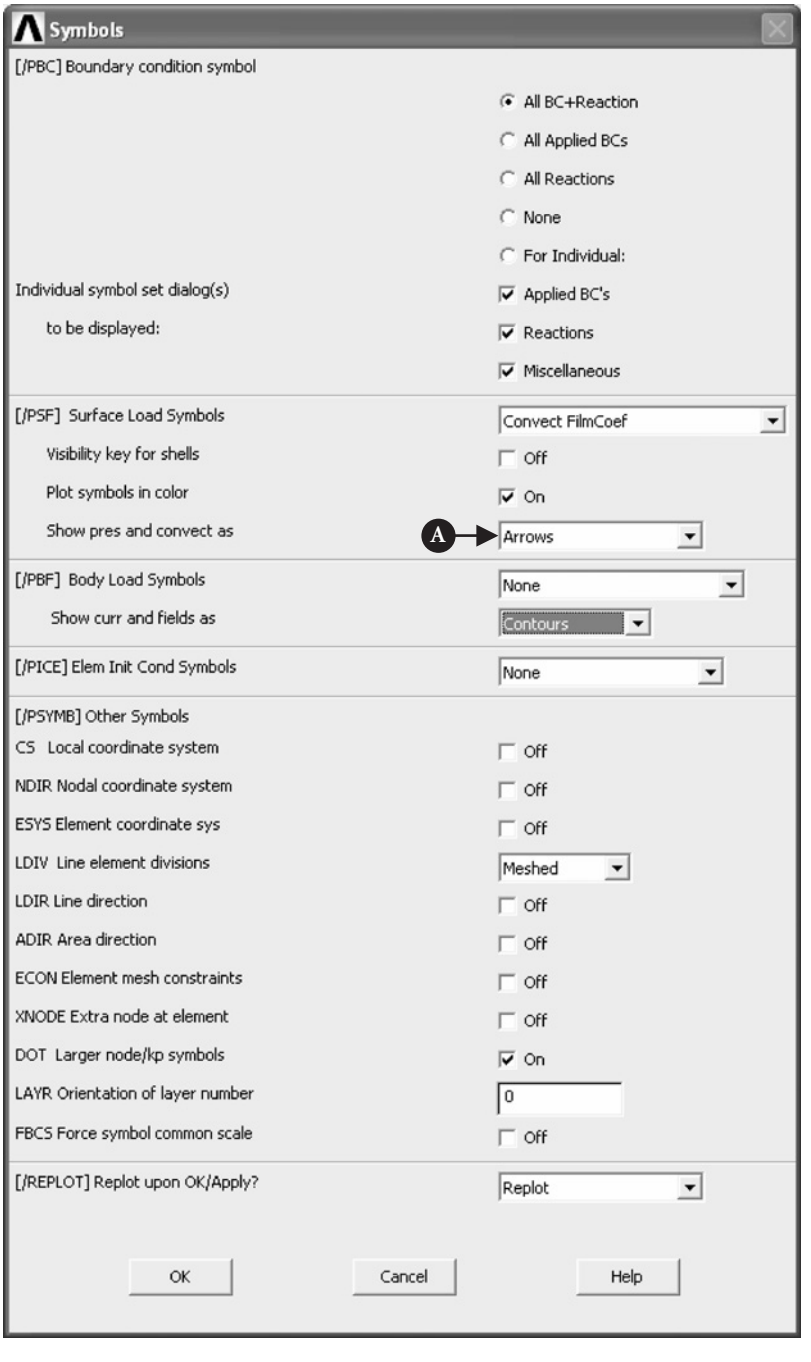


Figure 6.77 Symbols.

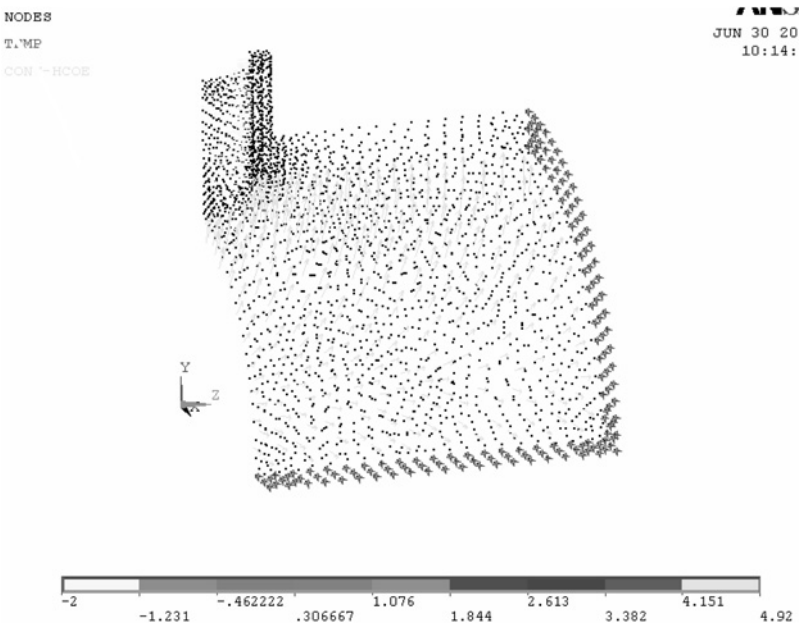


Figure 6.78 Convection surface loads displayed as arrows.

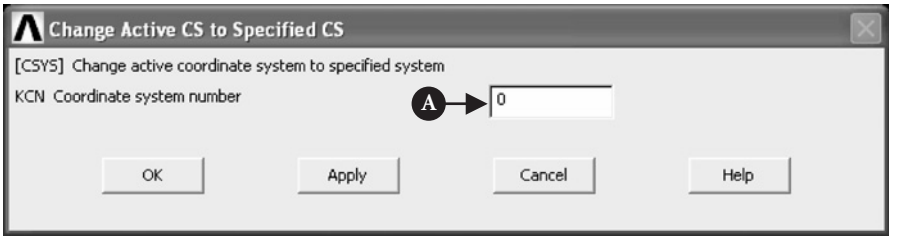


Figure 6.79 Change Active CS to Specified CS.

From **Utility Menu** select **PlotCtrls → Style → Edge Options**. Figure 6.81 shows the resulting frame.

Select [A] **All/Edge only** and [B] press **OK** button to implement the selection which will result in the display of the "edge" of the object only. Next, graphic controls ought to be returned to default setting. This is done by selecting from **Utility Menu PlotCtrls → Symbols**. The resulting frame, as shown in Figure 6.82, contains all default settings.

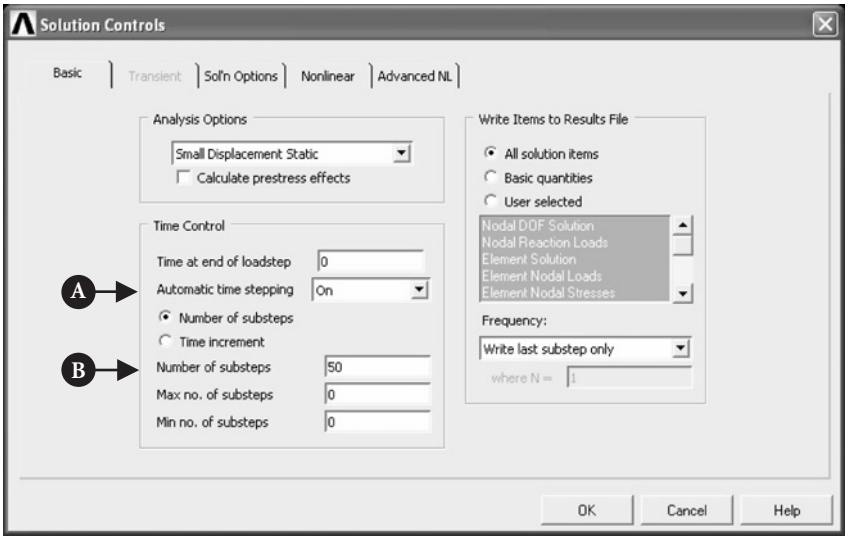


Figure 6.80 Solution Controls.

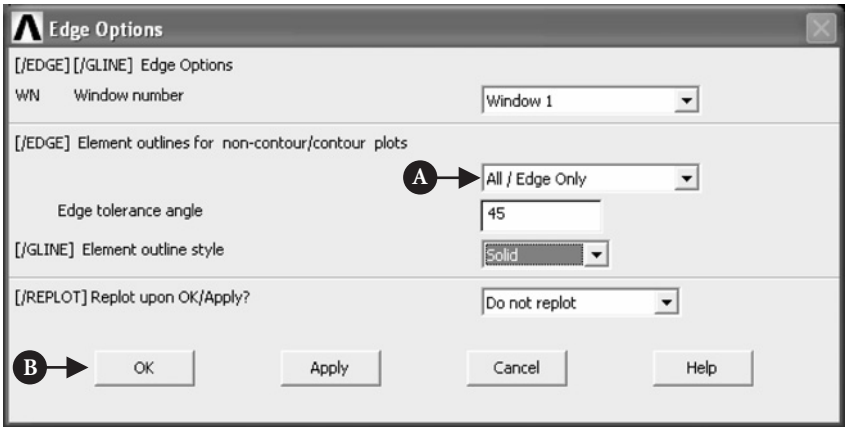


Figure 6.81 Edge Options.

The first plot is to show temperature distribution as continuous contours. From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The resulting frame is shown in Figure 6.83.

Select [A] **Temperature** and press [B] **OK** button as shown in Figure 6.83. The resulting temperature map is shown in Figure 6.84.

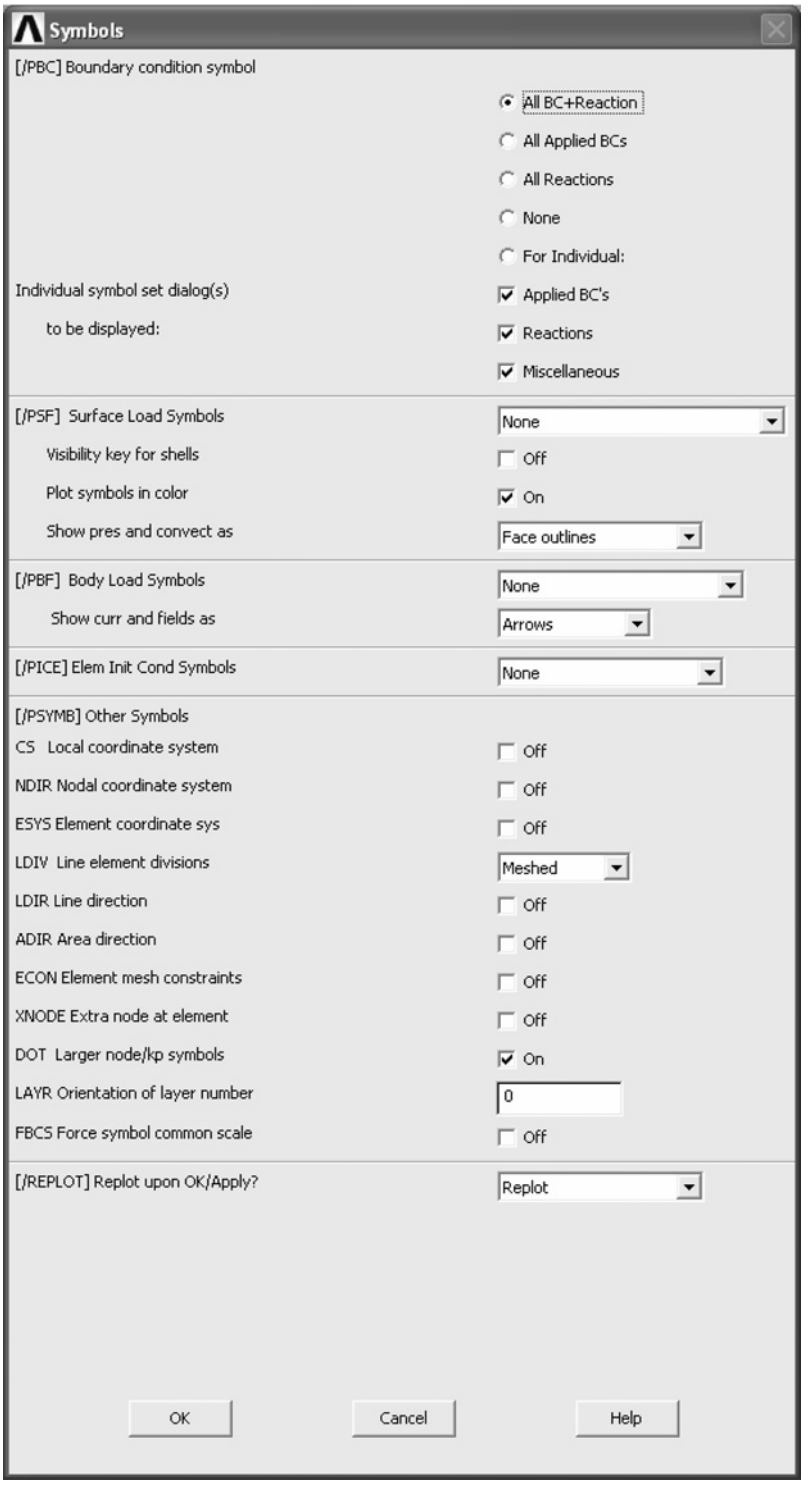


Figure 6.82 Symbols.

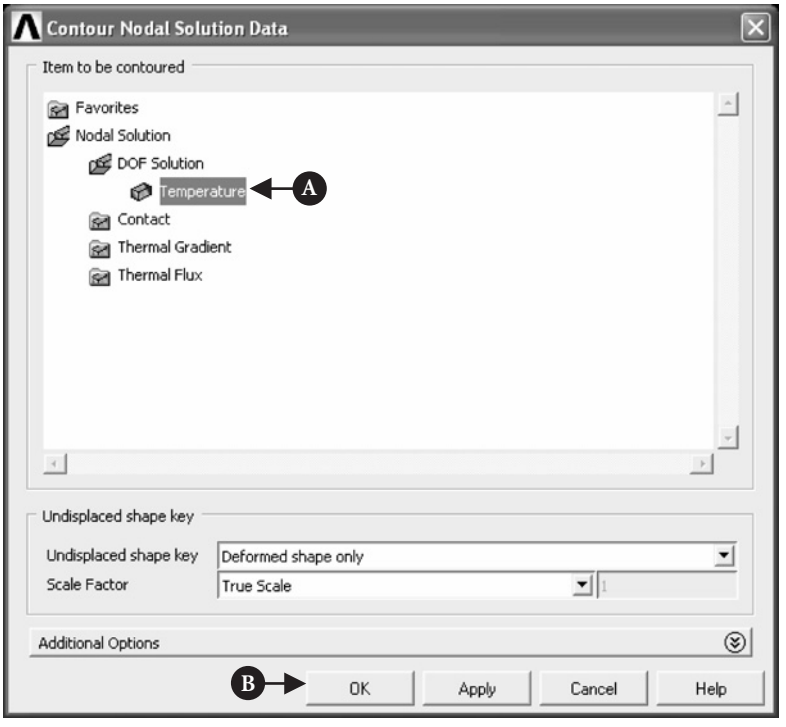


Figure 6.83 Contour Nodal Solution Data.

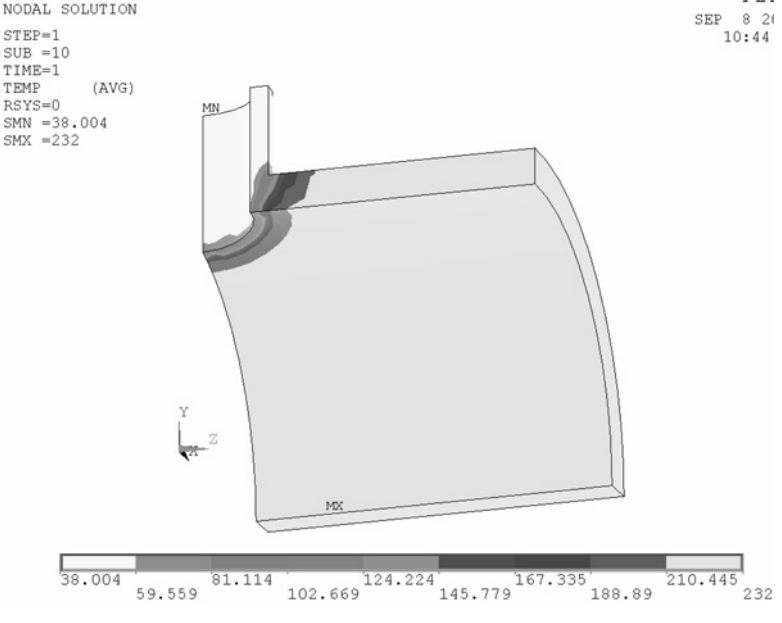


Figure 6.84 Temperature map on inner surfaces of the tank and the pipe.

The next display of results concerns thermal flux at the intersection between the tank and the pipe. From **ANSYS Main Menu** select **General Postproc → Plot Results → Vector Plot → Predefined**. The resulting frame is shown in Figure 6.85.

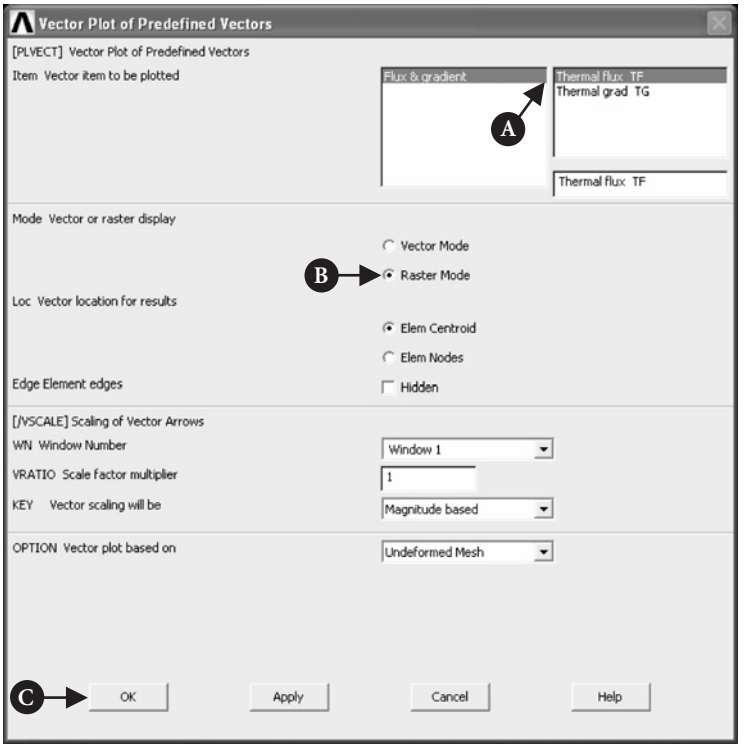


Figure 6.85 Vector Plot Selection.

In Figure 6.85, select [A] **Thermal flux TF** and [B] **Raster Mode**. Pressing [C] **OK** button implements selections and produces thermal flux as vectors. This is shown in Figure 6.86.

6.4 Heat dissipation through ribbed surface

6.4.1 Problem description

Ribbed or developed surfaces, also called fins, are frequently used to dissipate heat. There are many examples of their use in practical engineering applications such as computers, electronic systems, radiators, just to mention a few of them.

Figure 6.87 shows a typical configuration and geometry of a fin made of aluminum with thermal conductivity coefficient $k = 170\,\text{W/m K}$.

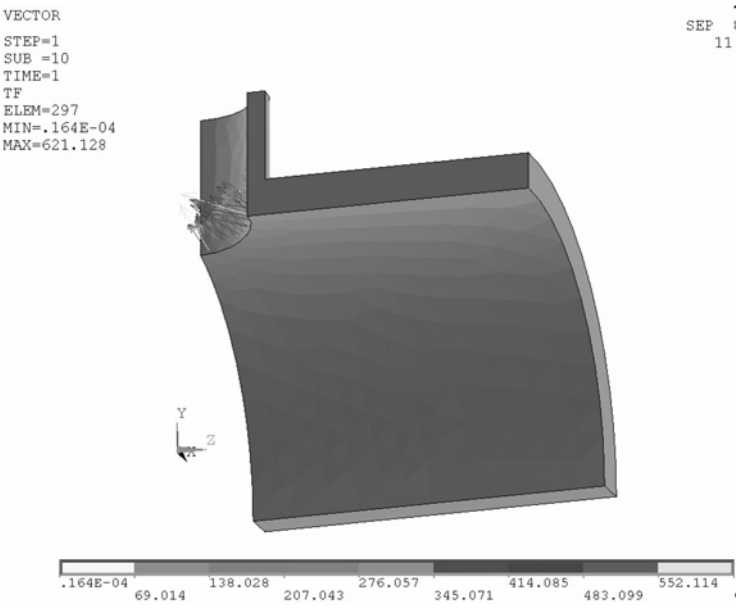


Figure 6.86 Distribution of thermal flux vectors at the intersection between the tank and the pipe.

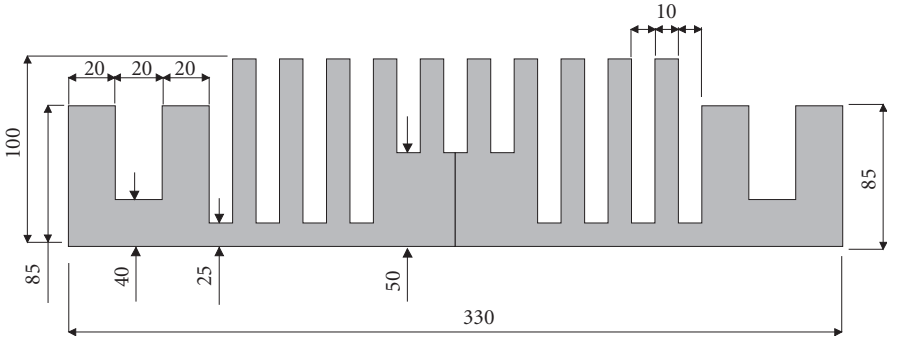


Figure 6.87 Cross-section of the fin.

The bottom surface of the fin is exposed to a constant heat flux of $q = 1000\,\mathrm{W/m}$. Air flows over the developed surface keeping the surrounding temperature at 293 K. Heat transfer coefficient between the fin and the surrounding atmosphere is $h = 40\,\mathrm{W/m^2\,K}$.

Determine the temperature distribution within the developed surface.

6.4.2 CONSTRUCTION OF THE MODEL

From **ANSYS Main Menu** select **Preferences** to call up a frame shown in Figure 6.88.

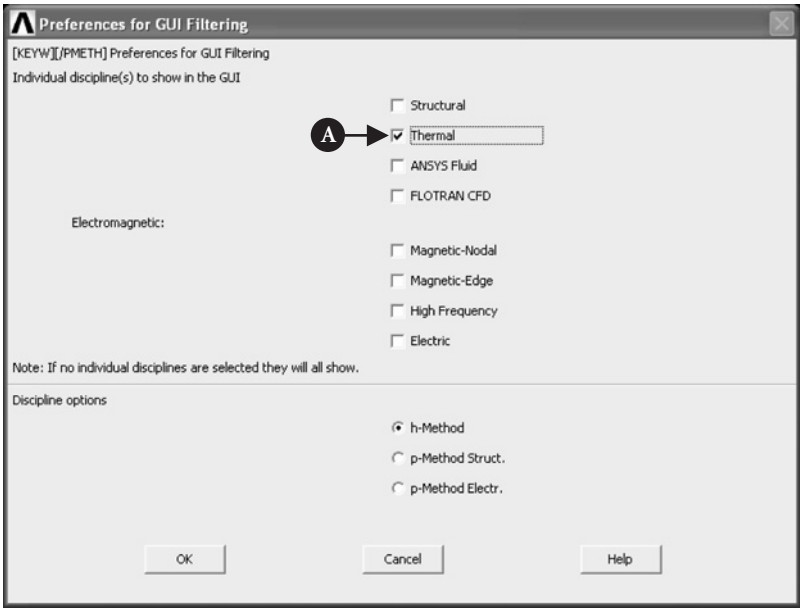


Figure 6.88 Preferences: Thermal.

Because the problem to be solved is asking for temperature distribution, therefore [A] **Thermal** is selected as indicated in the figure. Next, from **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 6.89 appears.

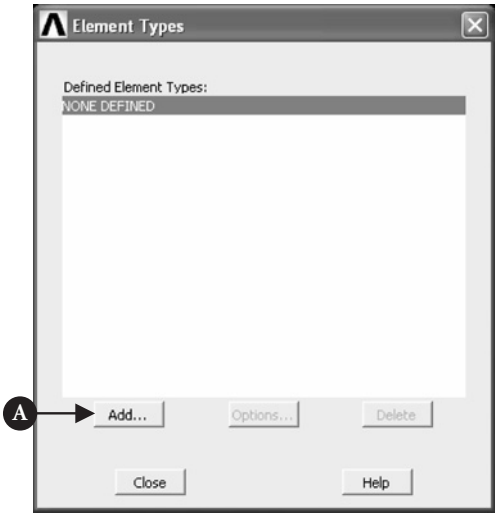


Figure 6.89 Define element type.

Click [A] **Add** button to call up another frame shown in Figure 6.90.

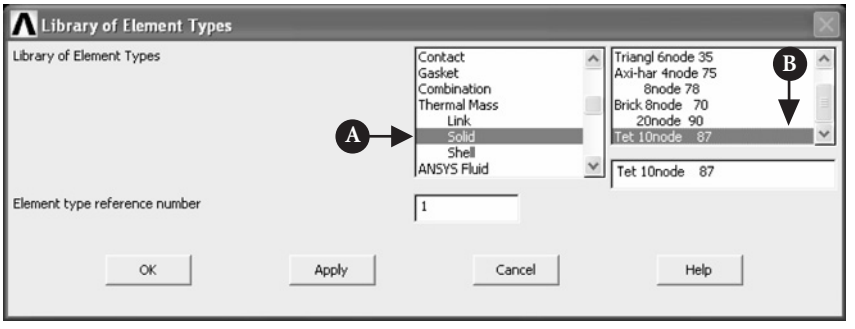


Figure 6.90 Library of Element Types.

In Figure 6.90, the following selections are made: [A] **Thermal Mass → Solid** and [B] **Tet 10node 87**. From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**. Figure 6.91 shows the resulting frame.

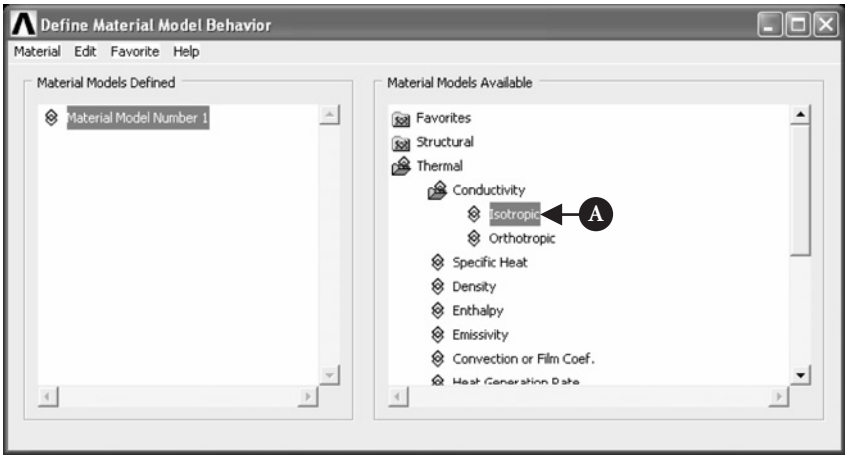


Figure 6.91 Define Material Model Behavior.

From the right-hand column select [A] **Thermal → Conductivity → Isotropic**. In response to this selection another frame, shown in Figure 6.92, appears.

Thermal conductivity [A] **KXX** = 170 W/m K is entered and [B] **OK** button clicked to implement the entry as shown in the figure.

The model of the developed area will be constructed using primitives and it is useful to have them numbered. Thus, from **ANSYS Utility Menu** select **PlotCtrls → Numbering** and check [A] the box area numbers on as shown in Figure 6.93.

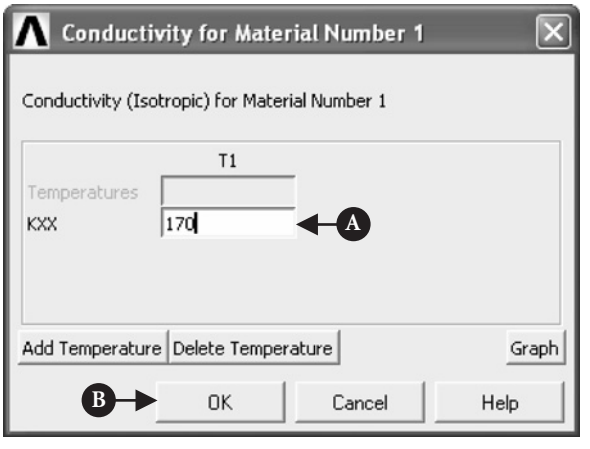


Figure 6.92 Conductivity coefficient.

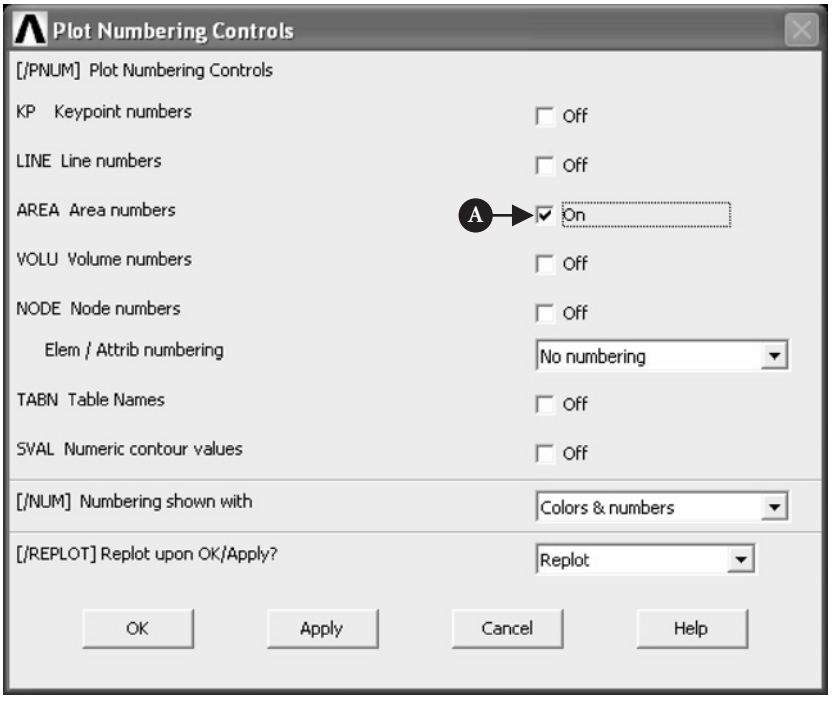


Figure 6.93 Numbering Controls.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Figure 6.94 shows the resulting frame.

Input [A] **X1** = −165; [B] **X2** = 165; [C] **Y1** = 0; [D] **Y2** = 100 to create rectangular area (A1) within which the fin will be comprised. Next create two rectangles

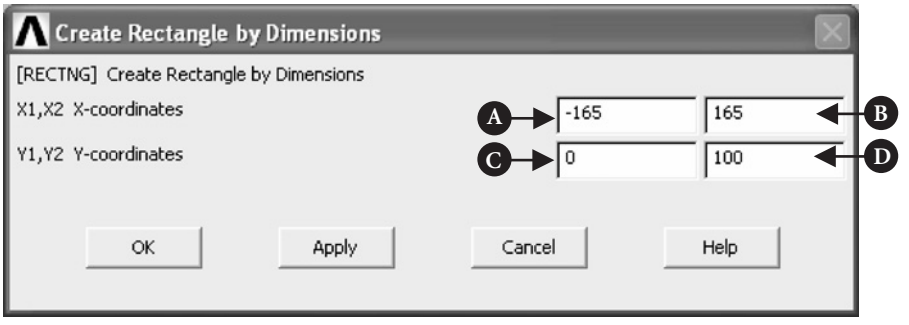


Figure 6.94 Create Rectangle by Dimensions.

at left and right upper corner to be cut off from the main rectangle. From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Figure 6.95 shows the resulting frame.

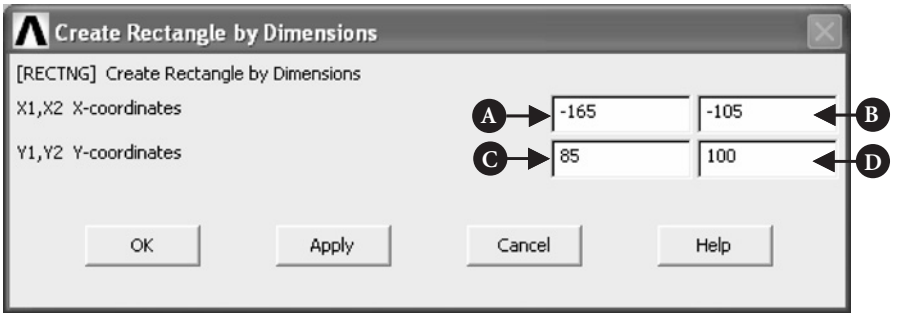


Figure 6.95 Rectangle with specified dimensions.

Figure 6.95 shows inputs to create rectangle (A2) at the left-hand upper corner of the main rectangle (A1). They are: [A] **X1** = −165; [B] **X2** = −105; [C] **Y1** = 85; [D] **Y2** = 100. In order to create right-hand upper corner rectangles (A3) repeat the above procedure and input: [A] **X1** = 105; [B] **X2** = 165; [C] **Y1** = 85; [D] **Y2** = 100. Now, areas A2 and A3 have to be subtracted from area A1. From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. Figure 6.96 shows the resulting frame.

First, select area A1 (large rectangle) to be subtracted from and [A] click **OK** button. Next, select two smaller rectangles A2 and A3 and click [A] **OK** button. A new area A4 is created with two upper corners cut off. Proceeding in the same way, areas should be cut off from the main rectangle in order to create the fin shown in Figure 6.87.

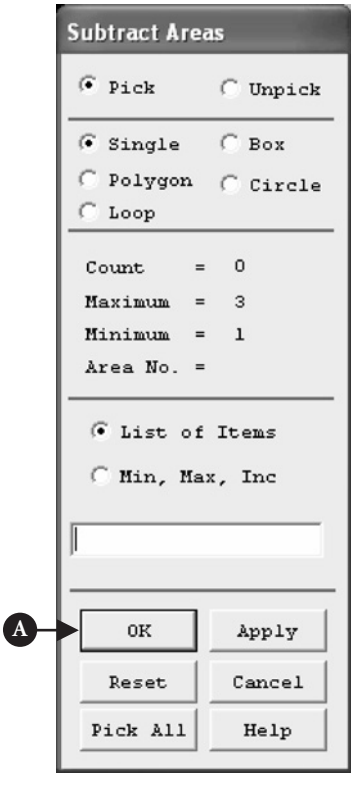


Figure 6.96 Subtract Areas.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Figure 6.97 shows the frame in which appropriate input should be made.

In order to create area A1 input: [A] **X1** = −145; [B] **X2** = −125; [C] **Y1** = 40; [D] **Y2** = 85. In order to create area A2 input: [A] **X1** = 125; [B] **X2** = 145; [C] **Y1** = 40; [D] **Y2** = 85. In order to create area A3 input: [A] **X1** = −105; [B] **X2** = −95; [C] **Y1** = 25; [D] **Y2** = 100. In order to create area A5 input: [A] **X1** = 95; [B] **X2** = 105; [C] **Y1** = 25; [D] **Y2** = 100.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. The frame shown in Figure 6.96 appears. Select first area A4 (large rectangle) and click [A] **OK** button. Next, select areas A1, A2, A3, and A5 and click [A] **OK** button. Area A6 with appropriate cut-outs is created. It is shown in Figure 6.98.

In order to finish construction of the fin's model use the frame shown in Figure 6.97 and make the following inputs: [A] **X1** = −85; [B] **X2** = −75; [C] **Y1** = 25; [D] **Y2** = 100. Area A1 is created. Next input: [A] **X1** = −65; [B]

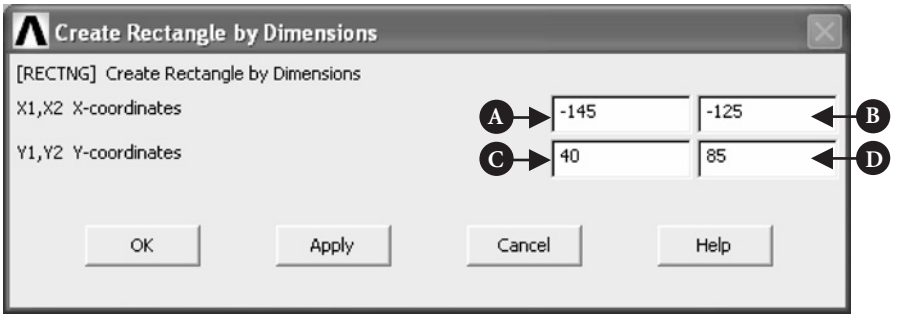


Figure 6.97 Create rectangle by four coordinates.

X2 = −55; [C] **Y1** = 25; [D] **Y2** = 100 to create area A2. Next input: [A] **X1** = −45; [B] **X2** = −35; [C] **Y1** = 25; [D] **Y2** = 100 to create area A3. Appropriate inputs should be made to create areas, to be cut out later, on the right-hand side of the fin. Thus inputs: [A] **X1** = 85; [B] **X2** = 75; [C] **Y1** = 25; [D] **Y2** = 100 create area A4. Inputs: [A] **X1** = 65; [B] **X2** = 55; [C] **Y1** = 25; [D] **Y2** = 100 create area A5. Inputs

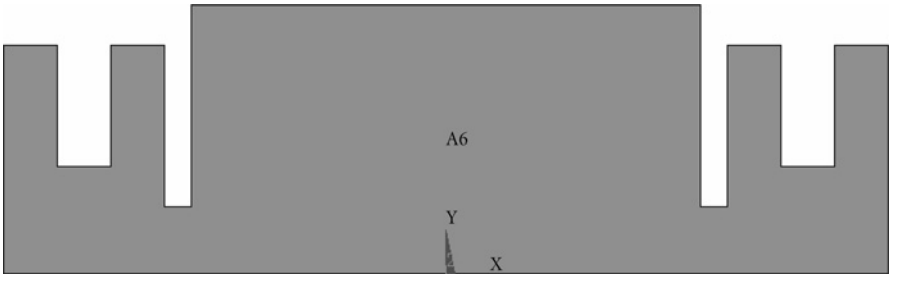


Figure 6.98 Image of the fin after some areas were subtracted.

[A] **X1** = 45; [B] **X2** = 35; [C] **Y1** = 25; [D] **Y2** = 100 create area A7. Next, from **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. The frame shown in Figure 6.96 appears. Select first area A6 and click [A] **OK** button. Then, select areas A1, A2, A3, A4, A5, and A7. Clicking [A] **OK** button implements the command and a new area A8 with appropriate cut-outs is created. In order to finalize the construction of the model make the following inputs to the frame shown in Figure 6.97 to create area A1: [A] **X1** = −25; [B] **X2** = −15; [C] **Y1** = 50; [D] **Y2** = 100. Inputs: [A] **X1** = −5; [B] **X2** = 5; [C] **Y1** = 50; [D] **Y2** = 100 create area A2. Finally input [A] **X1** = 15; [B] **X2** = 25; [C] **Y1** = 50; [D] **Y2** = 100 to create area A3. Again from **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. The frame shown in Figure 6.96 appears. Select first area A8 and click [A] **OK** button. Next, select areas A1, A2, and A3. Clicking [A] **OK** button produces area A4 shown in Figure 6.99. Figure 6.99 shows the final shape of the fin with dimensions as specified in Figure 6.87. It is, however, a 2D model. The width of the fin is 100 mm and this dimension can be used to create 3D model.

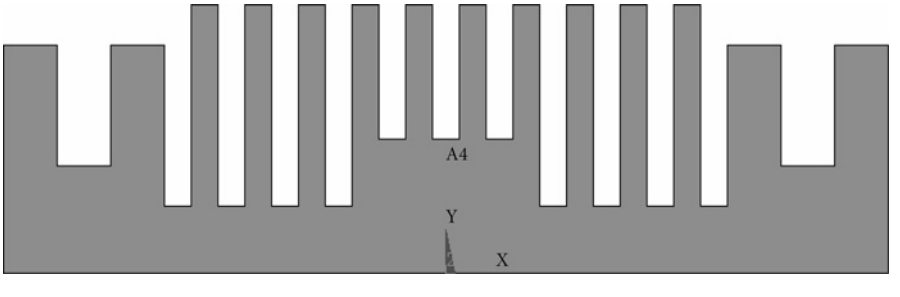


Figure 6.99 Two-dimensional image of the fin.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Extrude → Areas → Along Normal**. Select Area 4 (to be extruded in the direction normal to the screen, i.e., *z*-axis) and click **OK** button. In response, the frame shown in Figure 6.100 appears.

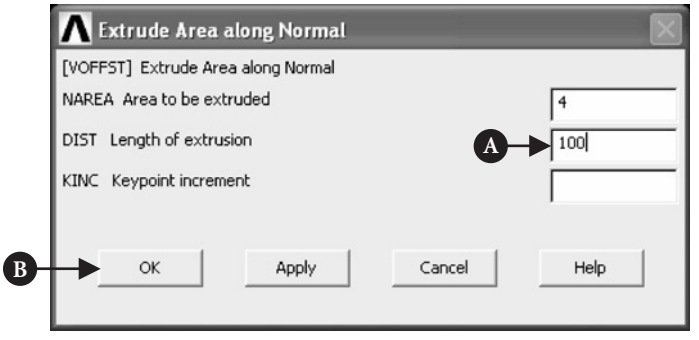


Figure 6.100 Extrude area.

Input [A] **Length of extrusion** = 100 mm and [B] click **OK** button. The 3D model of the fin is created as shown in Figure 6.101.

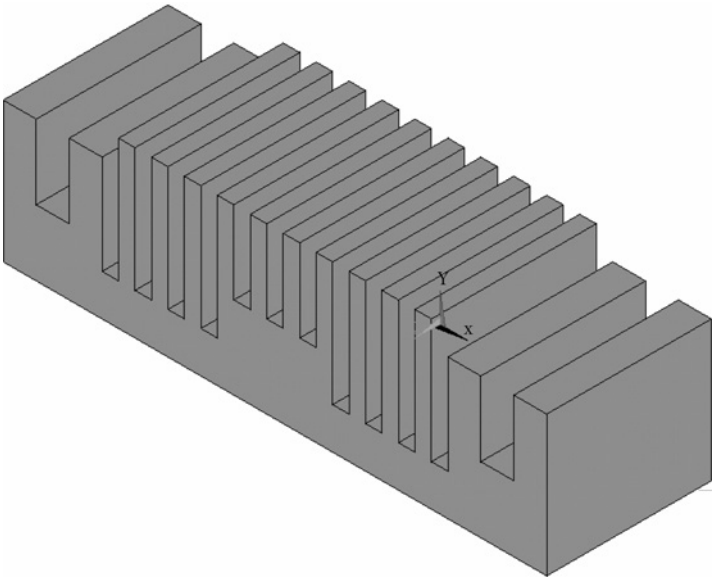


Figure 6.101 Three-dimensional (isometric) view of the fin.

The fin is shown in isometric view without area numbers. In order to deselect numbering of areas refer to Figure 6.93 in which box **Area numbers** should be checked off.

From **ANSYS Main Menu** select **Preprocessor → Meshing → Mesh Attributes → Picked Volumes**. The frame shown in Figure 6.102 is created.

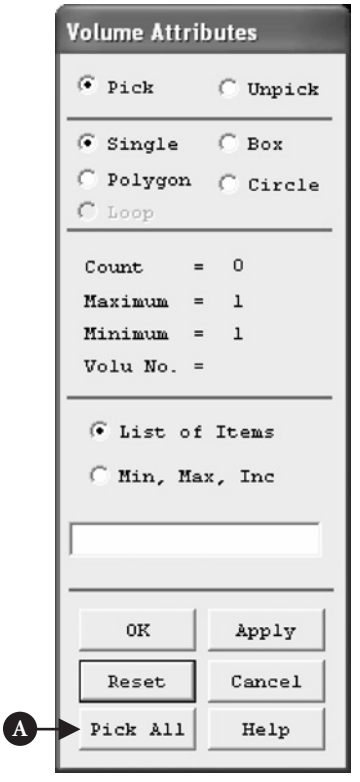


Figure 6.102 Volume Attributes.

Select [A] **Pick All** and the next frame, shown in Figure 6.103, appears.

Material Number 1 and element type SOLID87 are as specified at the beginning of the analysis and in order to accept that click [A] **OK** button.

Now meshing of the fin can be carried out. From **ANSYS Main Menu** select **Preprocessor → Meshing → Mesh → Volumes → Free**. The frame shown in Figure 6.104 appears.

Select [A] **Pick All** option, as shown in Figure 6.104, to mesh the fin. Figure 6.105 shows the meshed fin.

6.4.3 Solution

Prior to running solution stage boundary conditions have to be properly applied. In the case considered here the boundary conditions are expressed by the heat transfer coefficient which is a quantitative measure of how efficiently heat is transferred from fin surface to the surrounding air.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Convection → On Areas**. Figure 6.106 shows the resulting frame.

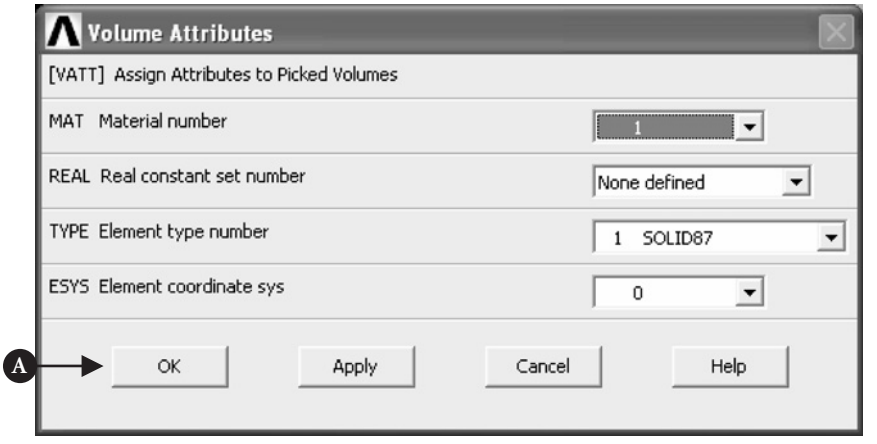


Figure 6.103 Volume attributes with specified material and element type.

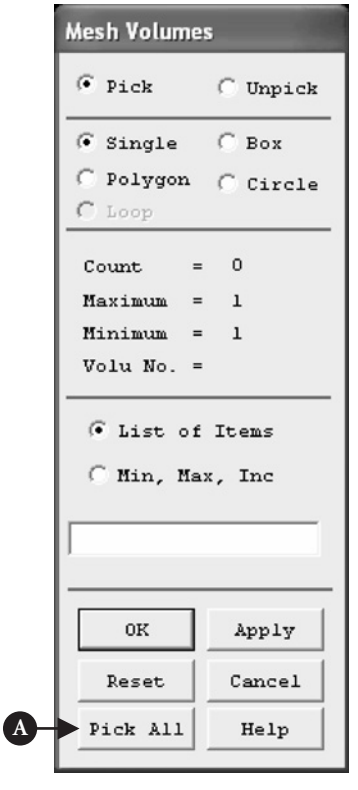


Figure 6.104 Mesh Volumes.

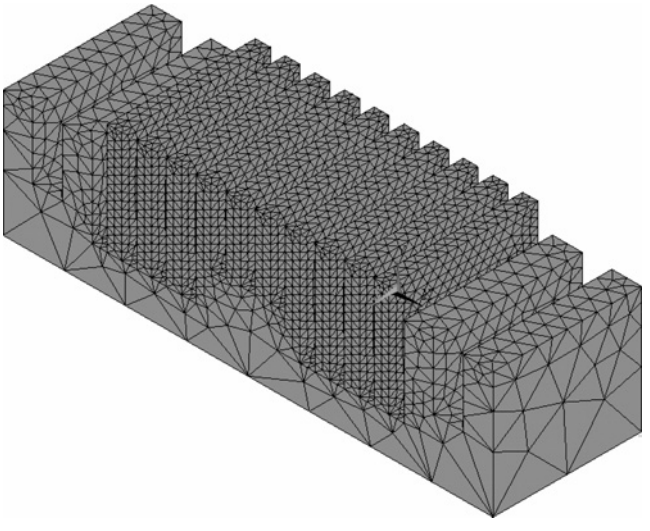


Figure 6.105 View of the fin with mesh network.

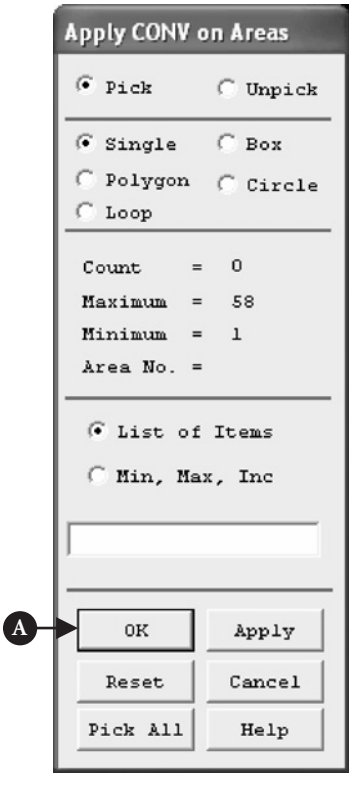


Figure 6.106 Apply boundary conditions to the fin areas.

Select all areas of the fin except the bottom area and click [A] **OK** button. The frame created as a result of that action is shown in Figure 6.107.

Input [A] **Film coefficient** = 40 W/m^2 K; [B] **Bulk temperature** = 293 K and click [C] **OK** button. Next a heat flux of intensity 1000 W/m has to be applied to the base of the fin. Therefore, from **ANSYS Main Menu** select **Solution → Define Loads → Apply → Thermal → Heat Flux → On Areas**. The resulting frame is shown in Figure 6.108.

Select the bottom surface (base) of the fin and click [A] **OK** button. A new frame appears (see Figure 6.109) and the input made is as follows: [A] **Load HFLUX value** = 1000 W/m. Clicking [B] **OK** button implements the input.

All required preparations have been made and the model is ready for solution. From **ANSYS Main Menu** select **Solution → Solve → Current LS**. Two frames appear. One gives a summary of solution options. After checking correctness of the options, it should be closed using the menu at the top of the frame. The other frame is shown in Figure 6.110.

Clicking [A] **OK** button starts the solution process.

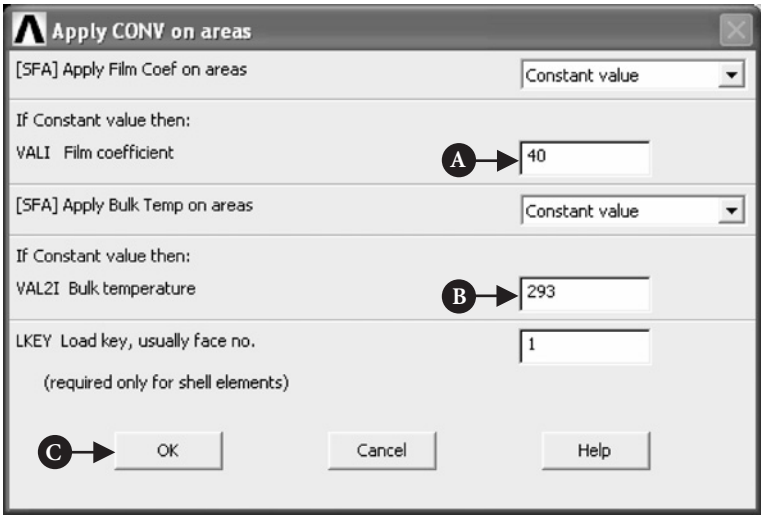


Figure 6.107 Apply heat transfer coefficient and surrounding temperature.

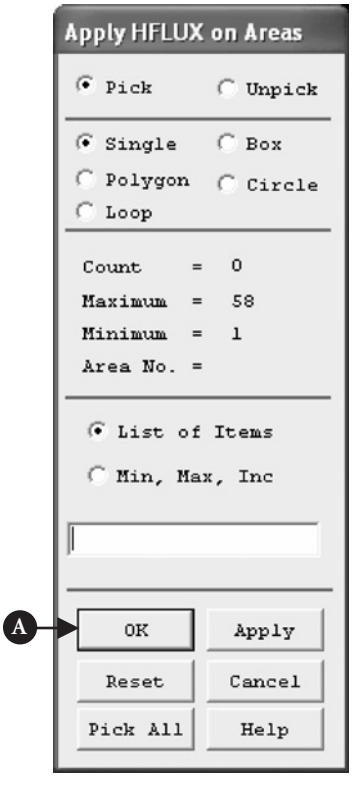


Figure 6.108 Apply heat flux on the fin base.

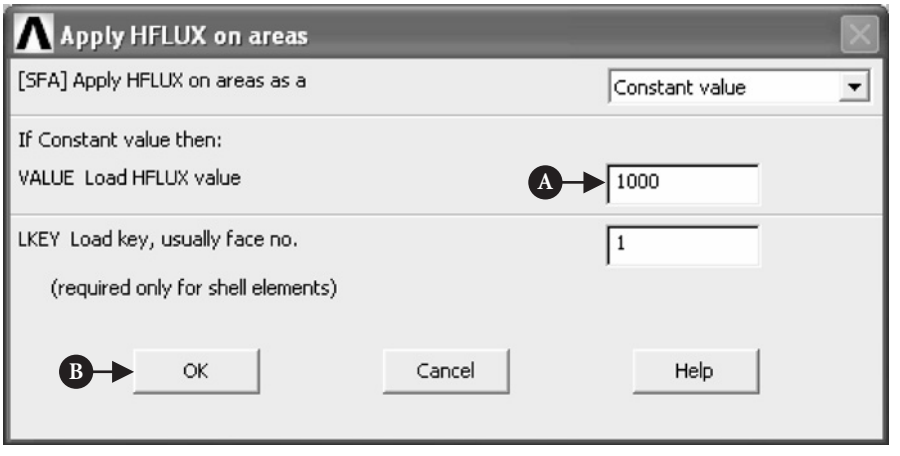


Figure 6.109 Apply heat flux value on the fin base.

Figure 6.110 Solve the problem.

6.4.4 POSTPROCESSING

Successful solution is signaled by the message "solution is done." The postprocessing phase can be initiated now in order to view the results. The problem asks for temperature distribution within the developed area.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solution**. The frame shown in Figure 6.111 appears.

Select [A] **Thermal Flux**; [B] **thermal flux vector sum** and click [C] **OK** to produce the graph shown in Figure 6.112.

In order to observe how the temperature changes from the base surface to the top surface of the fin a path along which the variations take place has to be determined.

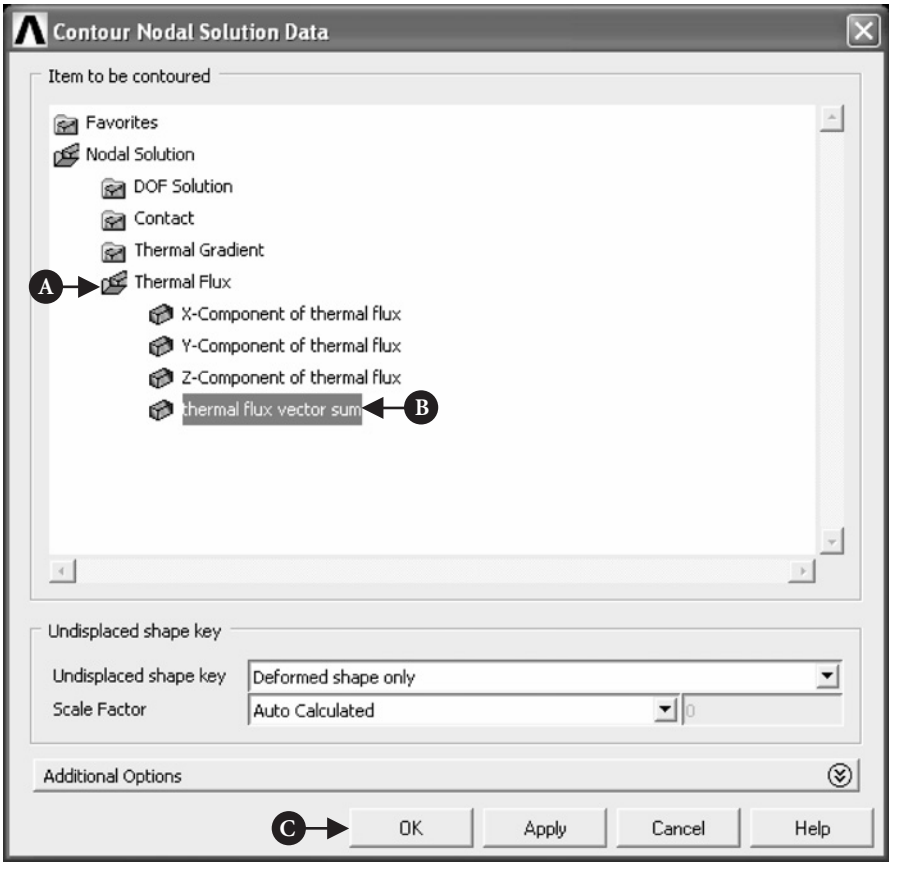


Figure 6.111 Contour Nodal Solution Data.

From **ANSYS Main Menu** select **General Postproc → Path Operations → Define Path → On Working Plane**. The resulting frame is shown in Figure 6.113.

By activating [A] **Arbitrary path** button and clicking [B] **OK** button another frame, shown in Figure 6.114, is produced.

Two points should be picked that is on the bottom line at the middle of the fin and, moving vertically upward, on the top line of the fin. After that [A] **OK** button should be clicked. A new frame appears as shown in Figure 6.115.

In the box [A] **Define Path Name**, write AB and click [B] **OK** button.

From **ANSYS Main Menu** select **General Postproc → Path Operations → Map onto Path**. The frame shown in Figure 6.116 appears.

Select [A] **Flux & gradient**; [B] **TGSUM** and click [C] **OK** button. Next, from **ANSYS Main Menu** select **General Postproc → Path Operations → Plot Path Item → On Graph**. Figure 6.117 shows the resulting frame.

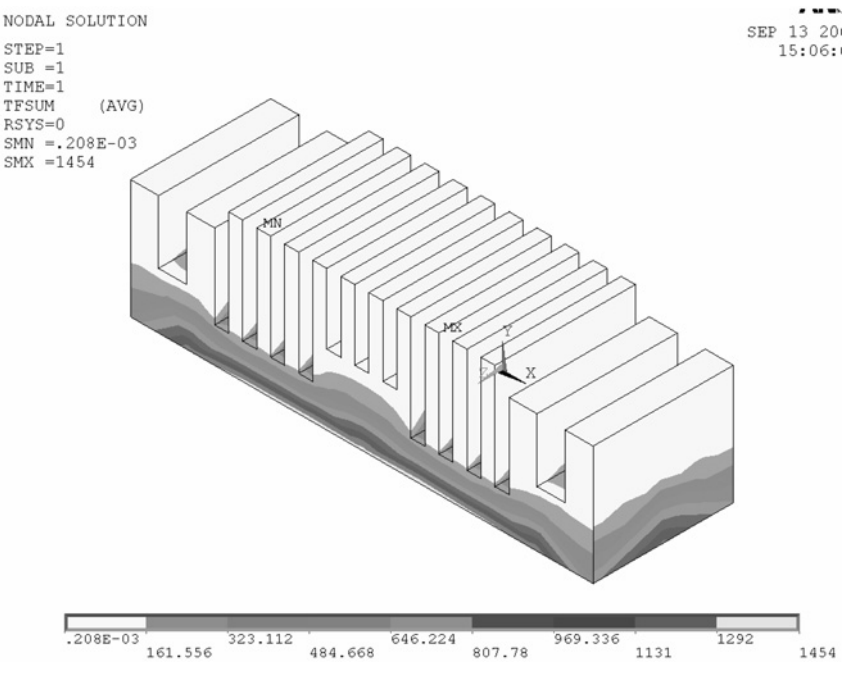


Figure 6.112 Heat flux distribution.

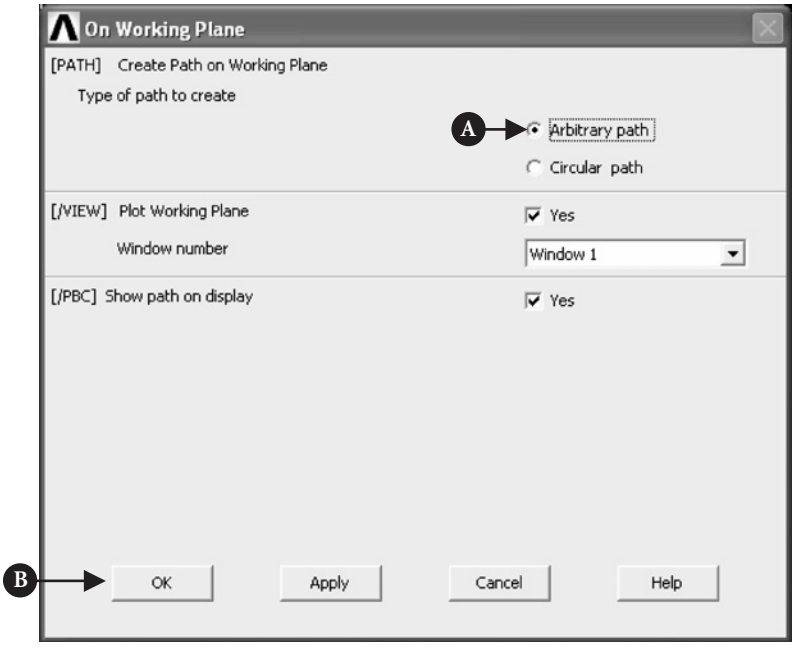


Figure 6.113 Arbitrary path selection.

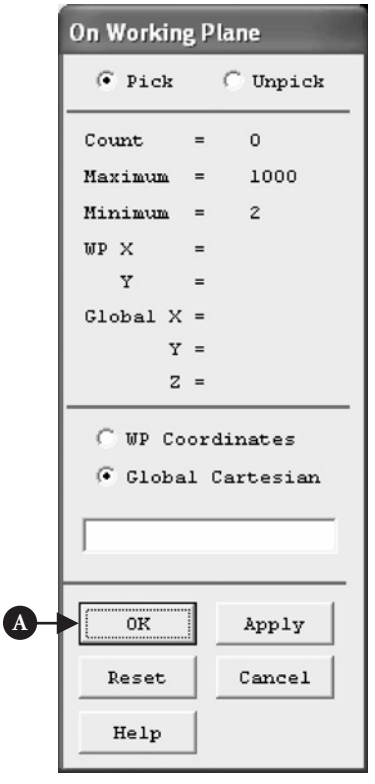


Figure 6.114 Arbitrary path on working plane.

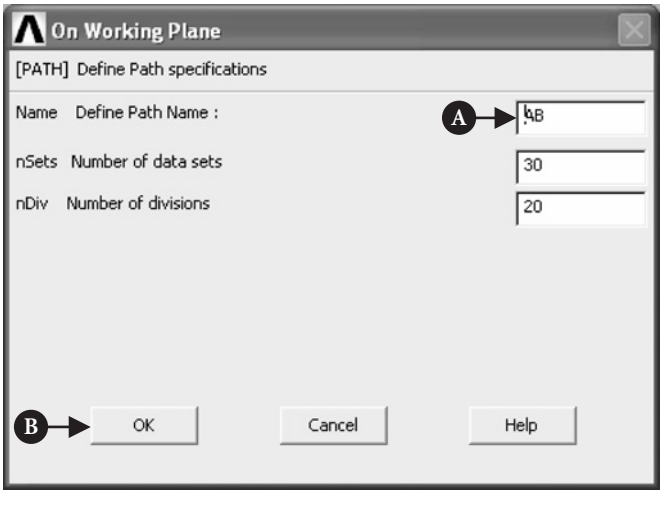


Figure 6.115 Path name definition.

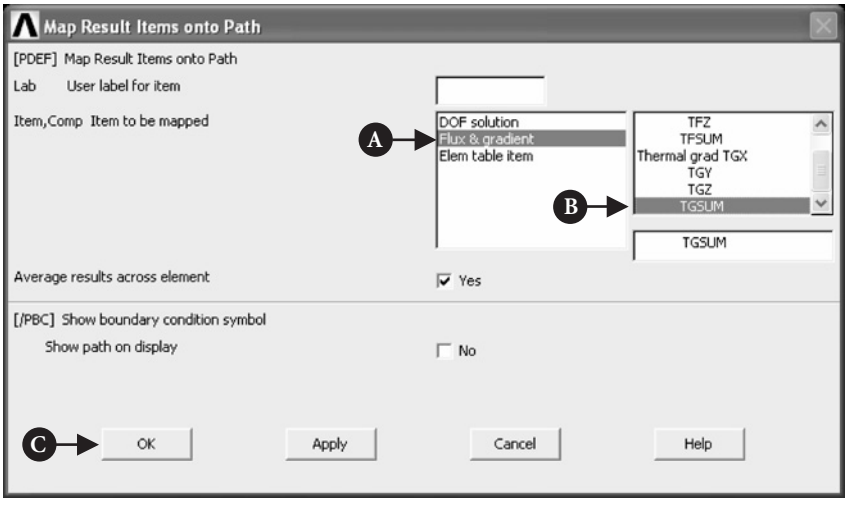


Figure 6.116 Map results on the path.

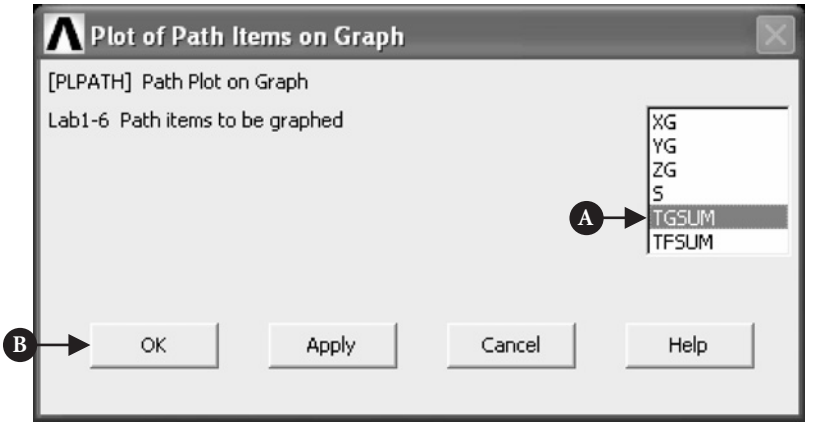


Figure 6.117 Selection of items to be plotted.

Select [A] **TGSUM** and click [B] **OK** button to obtain a graph shown in Figure 6.118.

The graph shows temperature gradient variation as a function of distance from the base of the fin.

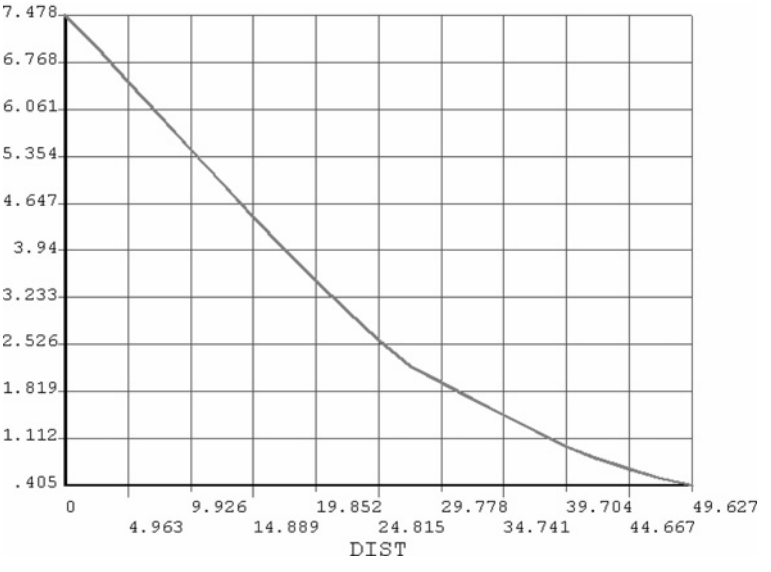


Figure 6.118 Temperature gradient plot as a function of distance from the fin base.

CHAPTER 7

Application of ANSYS to Contact Between Machine Elements

Chapter outline

7.1 General characteristics of contact problems 331
7.2 Example problems 332

7.1 General characteristics of contact problems

In almost every mechanical device, constituent components are in either rolling or sliding contact. In most cases, contacting surfaces are non-conforming so that the area through which the load is transmitted is very small, even after some surface deformation, and the pressures and local stresses are very high. Unless purposefully designed for the load and life expected of it, the component may fail by early general wear or by local fatigue failure. The magnitude of the damage is a function of the materials and the intensity of the applied load as well as the surface finish, lubrication, and relative motion.

The intensity of the load can usually be determined from equations, which are functions of the geometry of the contacting surfaces, essentially the radii of curvature, and the elastic constants of the materials. Large radii and smaller modules of elasticity give larger contact areas and lower pressures.

A contact is said to be conforming (concave) if the surfaces of the two elements fit exactly or even closely together without deformation. Journal bearings are an example of concave contact. Elements that have dissimilar profiles are considered to be non-conforming (convex). When brought into contact without deformation they touch first at a point, hence *point contact* or along line, *line contact*. In a ball bearing, the ball makes point contact with the inner and outer races, whereas in a roller bearing the roller makes line contact with both the races. Line contact arises when the profiles of the elements are conforming in one direction and non-conforming in the perpendicular direction. The contact area between convex elements is very small compared to the overall dimensions of the elements themselves. Therefore, the stresses are high and concentrated in the region close to the contact zone and are not substantially influenced by the shape of the elements at a distance from the contact area.

Contact problem analyses are based on the Hertz theory, which is an approximation on two counts. First, the geometry of general curved surfaces is described by quadratic terms only and second, the two bodies, at least one of which must have a curved surface, are taken to deform as though they were elastic half-spaces. The accuracy of Hertz theory is in doubt if the ratio a/R (a is the radius of the contact area and R is the radius of curvature of contacting elements) becomes too large. With metallic elements this restriction is ensured by the small strains at which the elastic limit is reached. However, a different situation arises with compliant elastic solids like rubber. A different problem is encountered with conforming (concave) surfaces in contact, for example, a pin in a closely fitting hole or by a ball and socket joint. Here, the arc of contact may be large compared with the radius of the hole or socket without incurring large strains.

Modern developments in computing have stimulated research into numerical methods to solve problems in which the contact geometry cannot be described adequately by the quadratic expressions used originally by Hertz. The contact of worn wheels and rails or the contact of conforming gear teeth with Novikov profile are the typical examples. In the numerical methods, contact area is subdivided into a grid and the pressure distribution represented by discrete boundary elements acting on the elemental areas of the grid. Usually, elements of uniform pressure are employed, but overlapping triangular elements offer some advantages. They sum to approximately linear pressure distribution and the fact that the pressure falls to zero at the edge of the contact ensures that the surfaces do not interfere outside the contact area. The three-dimensional (3D) equivalent of overlapping triangular elements is overlapping hexagonal pyramids on an equilateral triangular grid.

An authoritative treatment of contact problems can be found in the monograph by Johnson [1].

7.2 Example problems

7.2.1 Pin-in-hole interference fit

7.2.1.1 Problem description

One end of a steel pin is rigidly fixed to the solid plate while its other end is force fitted to the steel arm. The configuration is shown in Figure 7.1.

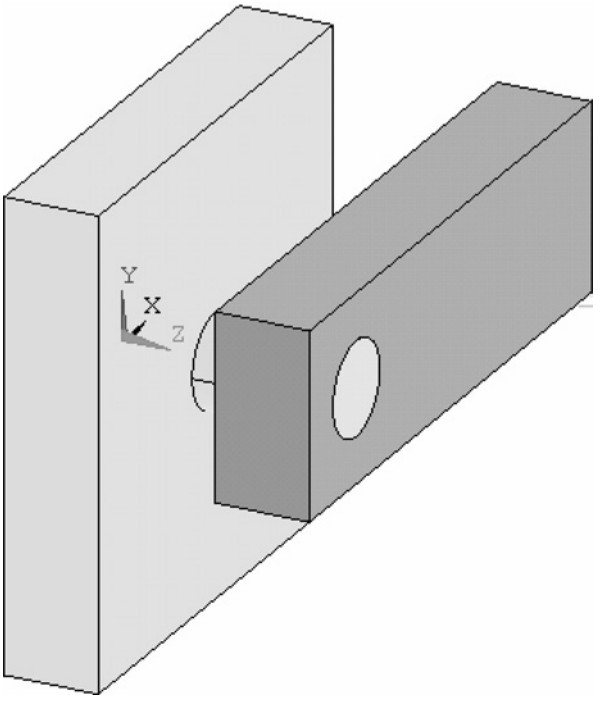


Figure 7.1 Illustration of the problem.

This is a 3D analysis but because of the inherent symmetry of the model, analysis will be carried out for a quarter-symmetry model only. There are two objectives of the analysis. The first is to observe the force fit stresses of the pin, which is pushed into the arm's hole with geometric interference. The second is to find out stresses, contact pressures, and reaction forces due to a torque applied to the arm (force acting at the arm's end) and causing rotation of the arm. Stresses resulting from shearing of the pin and bending of the pin will be neglected purposefully.

The dimensions of the model are as follows: pin radius = 1 cm, pin length = 3 cm; arm width = 4 cm, arm length = 12 cm, arm thickness = 2 cm; and hole in the arm: radius = 0.99 cm, depth = 2 cm (through thickness hole).

Both the elements are made of steel with Young's modulus = 2.1×10^9 N/m^2, Poisson's ratio = 0.3 and are assumed to be elastic.

7.2.1.2 Construction of the model

In order to analyze the contact between the pin and the hole, a quarter-symmetry model is appropriate. It is shown in Figure 7.2.

In order to create a model shown in Figure 7.2, two 3D primitives are used, namely block and cylinder. The model is constructed using graphical user interface (GUI) only. It is convenient for carrying out Boolean operations on volumes to have

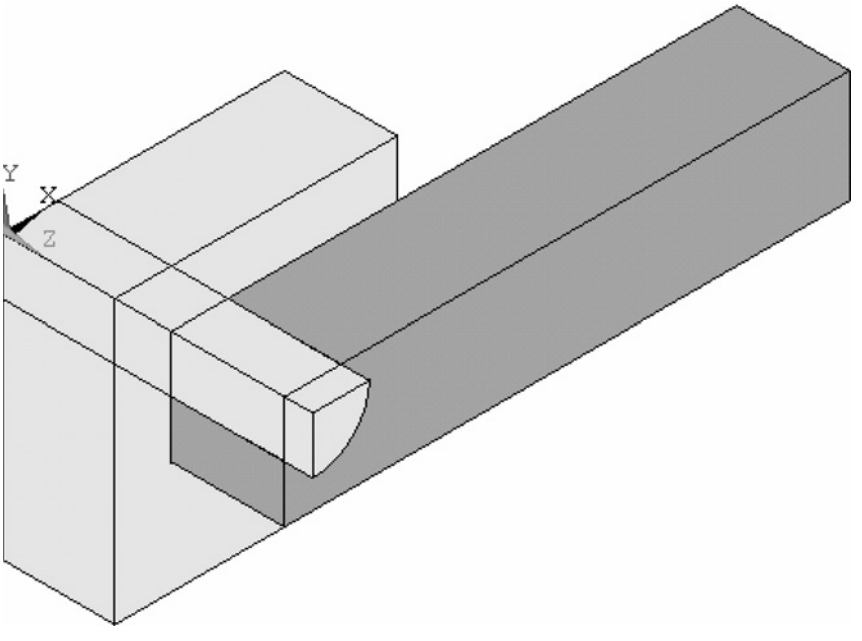


Figure 7.2 A quarter-symmetry model.

them numbered. This can be done by selecting from **Utility Menu → PlotCtrls → Numbering** and checking appropriate box to activate **VOLU** (volume numbers) option.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Block → By Dimensions**. In response, a frame shown in Figure 7.3 appears.

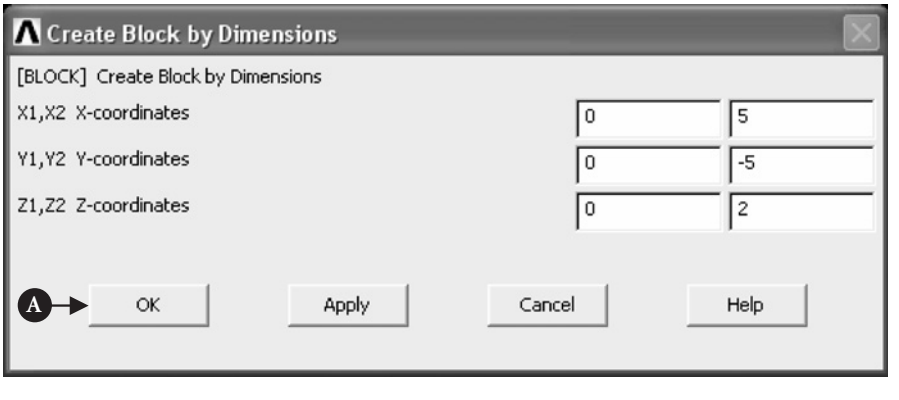


Figure 7.3 Create Block by Dimensions.

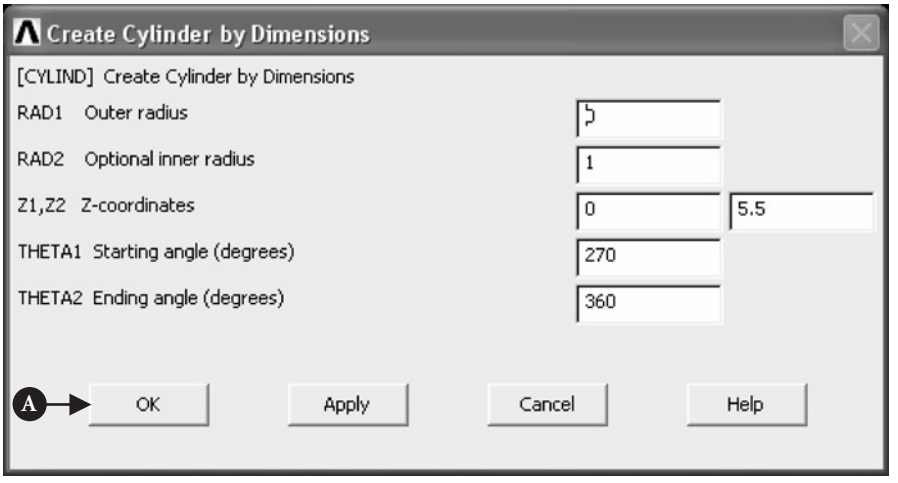


Figure 7.4 Create Cylinder by Dimensions.

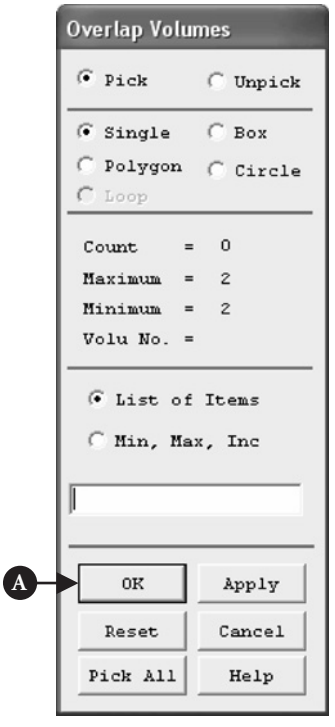


Figure 7.5 Overlap Volumes.

It can be seen from Figure 7.3 that appropriate X, Y, and Z coordinates were entered. Clicking [A] **OK** button implements the entries. A block with length 5 cm, width 5 cm, and thickness 2 cm (vol. 1) is created.

Next, from **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. In response, a frame shown in Figure 7.4 appears.

The inputs are shown in Figure 7.4. Clicking [A] **OK** button implements the entries and creates a solid cylinder sector with radius 1 cm, length 5.5 cm, starting angle 270°, and ending angle 360° (vol. 2).

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Overlap → Volumes**. The frame shown in Figure 7.5 appears.

Block (vol. 1) and cylinder (vol. 2) should be picked and [A] **OK** button pressed. As a result of that block and cylinder are overlapped.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Block → By Dimensions**. The frame shown in Figure 7.6 appears.

Coordinates X, Y, and Z were used as shown in Figure 7.6. Clicking [A] **OK** button implements the entries and, as a result, a block volume was created with length 10 cm, width 2 cm, and thickness 2 cm (vol. 2).

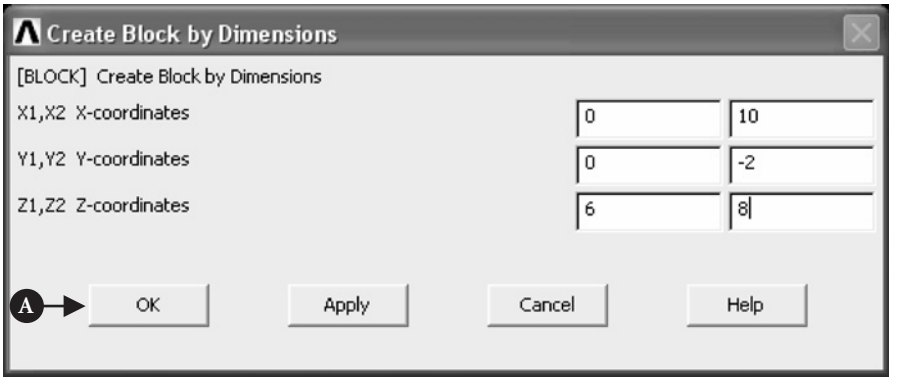


Figure 7.6 Create Block by Dimensions.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. The frame shown in Figure 7.7 appears.

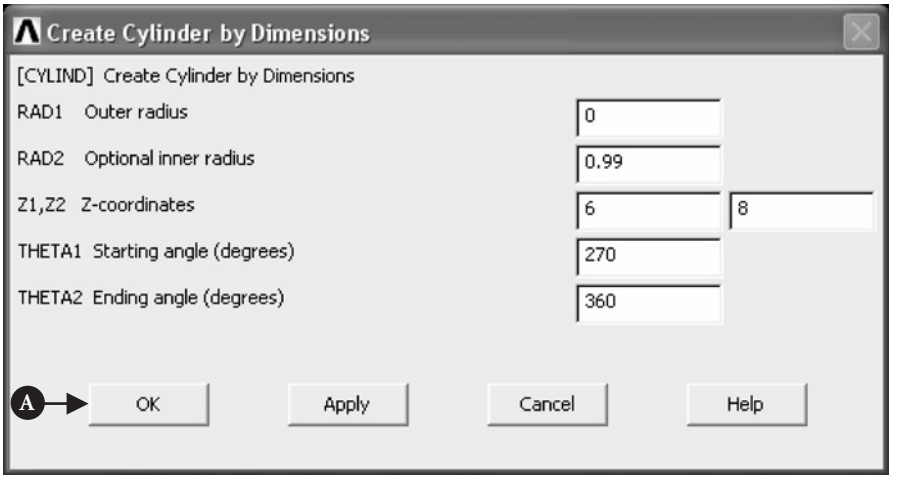


Figure 7.7 Create Cylinder by Dimensions.

Input data entered are shown in Figure 7.7. Clicking [A] **OK** button implements the entries. As a result solid cylinder sector with radius 0.99 cm, length 2 cm, starting angle 270°, and ending angle 360° (vol. 2) is produced. Next, volume 2 must be subtracted from volume 1 to produce a hole in the arm with the radius of 0.99 cm, which is smaller than the radius of the pin. In this way, an interference fit between the pin and the arm is created.

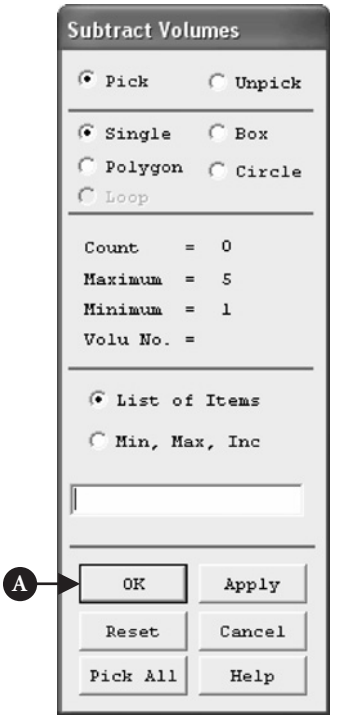


Figure 7.8 Subtract Volumes.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Volumes.** The frame shown in Figure 7.8 appears.

Volume 2 (short solid cylinder sector with radius 0.99 cm) is subtracted from volume 1 (the arm) by picking them in turn and pressing [A] **OK** button. As a result volume 6 is created.

From **ANSYS Main Menu** select **Modelling → Move/Modify → Volumes**. Then pick volume 6 (the arm), which is to be moved, and click **OK**. The frame shown in Figure 7.9 appears.

In order to move the arm (vol. 6) in required position, coordinates shown in Figure 7.9 should be used. Clicking [A] **OK** button implements the move action.

From **Utility Menu** select **Plot → Replot** to view the arm positioned in required location. Finally from **Utility Menu** select **PlotCtrls → View Settings → Viewing Direction**. The frame shown in Figure 7.10 appears.

By selecting coordinates X, Y, and Z as shown in Figure 7.10 and activating [A] **Plot → Replot** command (Utility Menu), a quarter-symmetry model, as shown in Figure 7.2, is finally created.

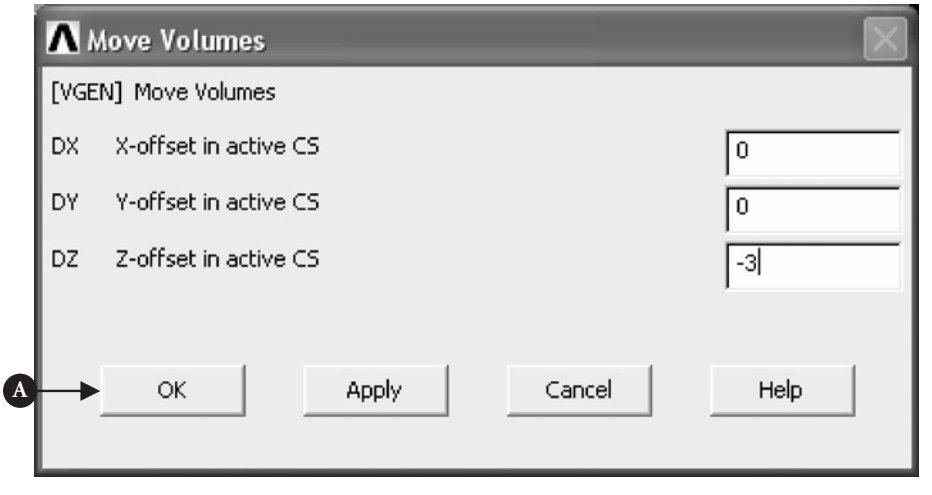


Figure 7.9 Move Volumes.

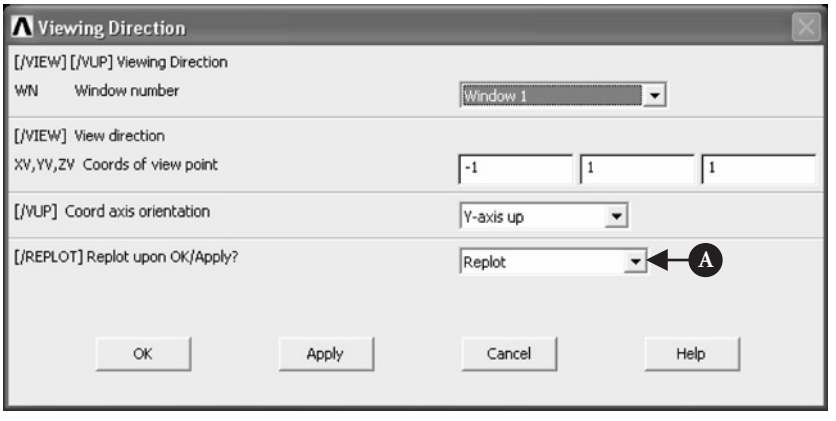


Figure 7.10 Viewing Direction.

7.2.1.3 MATERIAL PROPERTIES AND ELEMENT TYPE

The next step in the analysis is to define the properties of the material used to make the pin and the arm.

From **ANSYS Main Menu** select **Preferences**. The frame shown in Figure 7.11 is produced.

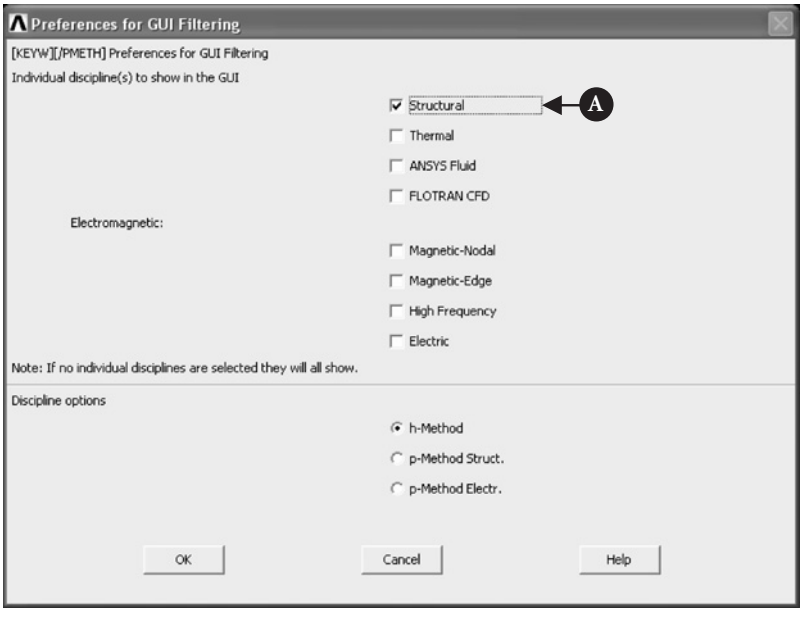


Figure 7.11 Preferences.

From the Preferences list [A] **Structural** option was selected as shown in Figure 7.11.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**.

Double click **Structural → Linear → Elastic → Isotropic**. The frame shown in Figure 7.12 appears.

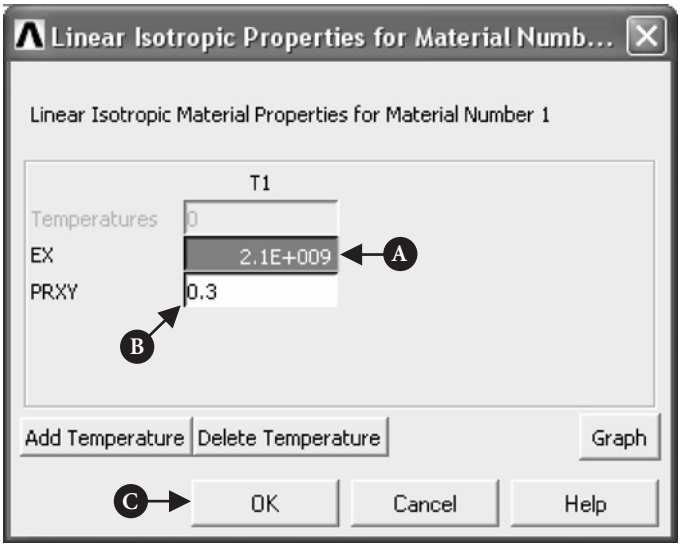


Figure 7.12 Material Properties.

Enter [A] **EX** $= 2.1 \times 10^9$ for Young's modulus and [B] **PRXY** $= 0.3$ for Poisson's ratio. Then click [C] **OK** and afterward **Material → Exit**.

After defining the properties of the material, the next step is to select the element type appropriate for the analysis.

From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.13 appears.

Click [A] **Add** in order to pull down another frame as shown in Figure 7.14.

In the left column click [A] **Structural Solid** and in the right column click [B] **Brick 8node 185**. After that click [C] **OK** and [B] **Close** in the frame shown in Figure 7.13. This completes the element type selection.

7.2.1.4 MESHING

From **ANSYS Main Menu** select **Preprocessor → Meshing → MeshTool**.

The frame shown in Figure 7.15 appears.

There are a number of options available. First step is to go to [A] **Size Control → Lines** option and click [B] **Set** button. This opens another frame (shown in Figure 7.16) prompting to pick lines on which element size is going to be controlled.

Figure 7.13 Element Types.

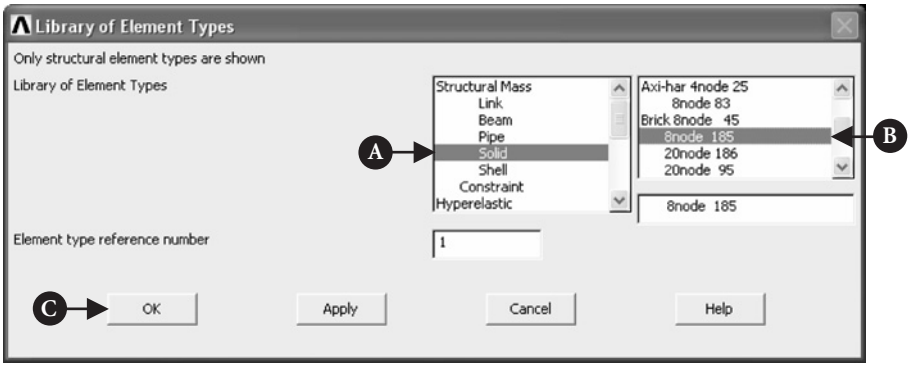


Figure 7.14 Library of Element Types.

Pick the horizontal and vertical lines on the front edge of the pin and click [A] **OK**. The frame shown in Figure 7.17 appears. In the box, [A] **No. of element divisions** type 3 and change selection [B] **SIZE, NDIV can be changed** to **No**. by checking the box and, finally, click [C] **OK**.

Using **MeshTool** frame again (as shown in Figure 7.18) click button [A] **Set** in the **Size Controls → Lines** option and pick the curved line on the front of the arm. Click

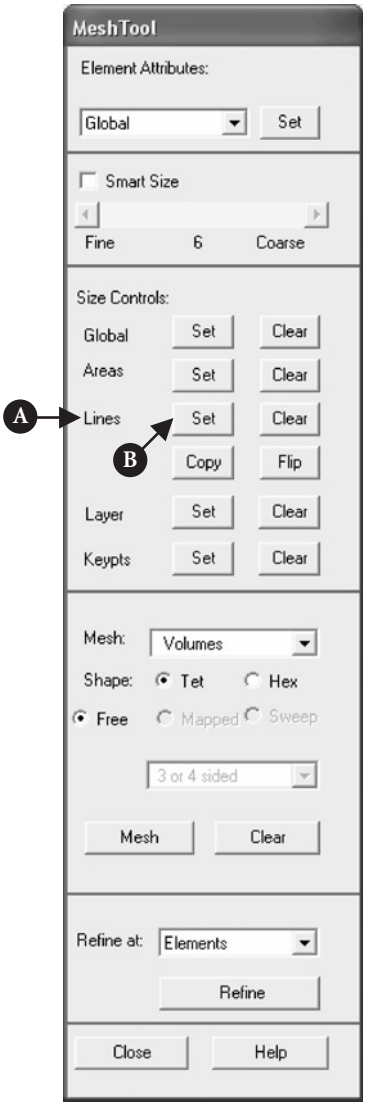


Figure 7.15 MeshTool options.

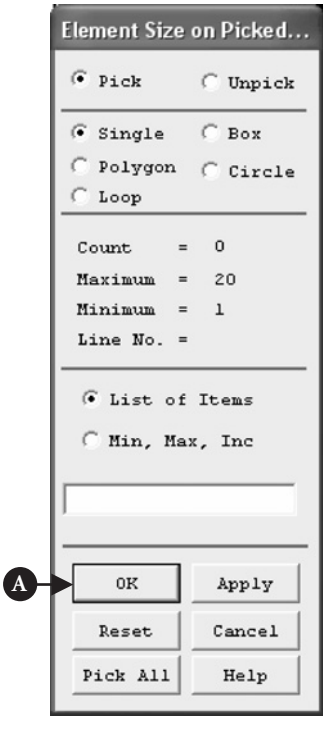


Figure 7.16 Element Size on Picked Lines.

OK afterward. The frame shown in Figure 7.17 appears. In the box **No. of element divisions** type 4 and press [C] **OK** button.

In the frame **MeshTool** (see Figure 7.18) pull down [B] **Volumes** in the option **Mesh**.

Check [C] **Hex/Wedge** and [D] **Sweep** options. This is shown in Figure 7.18.

Pressing [E] **Sweep** button brings another frame asking to pick the pin and the arm volumes (see Figure 7.19).

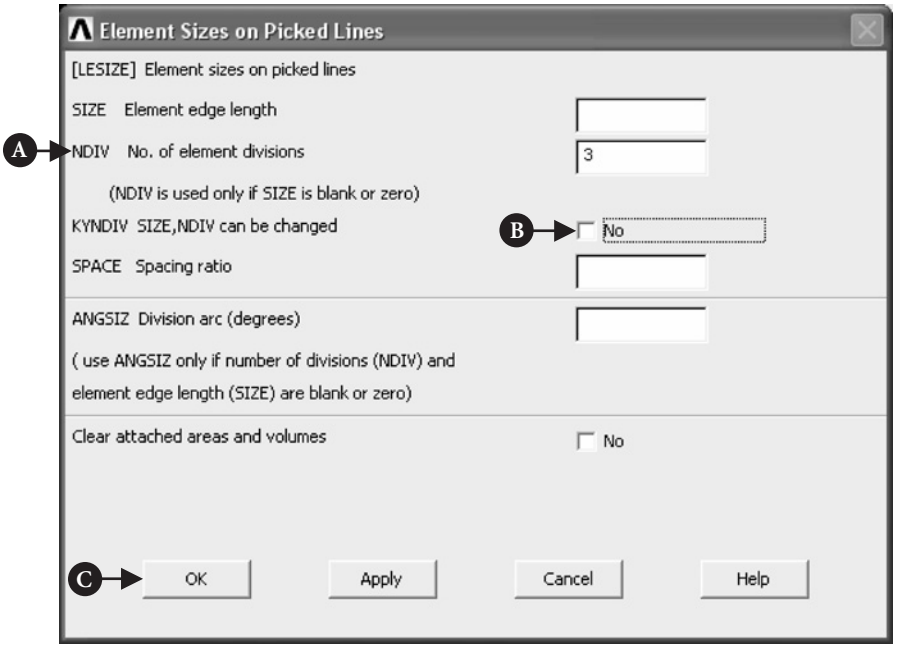


Figure 7.17 Element Sizes on Picked Lines.

Pressing [A] **OK** button initiates meshing process. The model after meshing looks like the image in Figure 7.20.

Pressing [F] **Close** button on **MeshTool** frame (see Figure 7.18) ends mesh generation stage.

After meshing is completed, it is usually necessary to smooth element edges in order to improve graphic display. It can be accomplished using **PlotCtrls** facility in the Utility Menu.

From **Utility Menu** select **PlotCtrls → Style → Size and Shape**. The frame shown in Figure 7.21 appears.

From option [A] **Facets/element edge** select **2 facets/edge**, which is shown in Figure 7.21.

7.2.1.5 CREATION OF CONTACT PAIR

In solving the problem of contact between two elements, it is necessary to create contact pair. Contact Wizard is the facility offered by ANSYS.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Contact Pair**. As a result of this selection, a frame shown in Figure 7.22 appears.

Location of [A] **Contact Wizard** button is in the upper left-hand corner of the frame. By clicking on this button a new frame (shown in Figure 7.23) is produced.

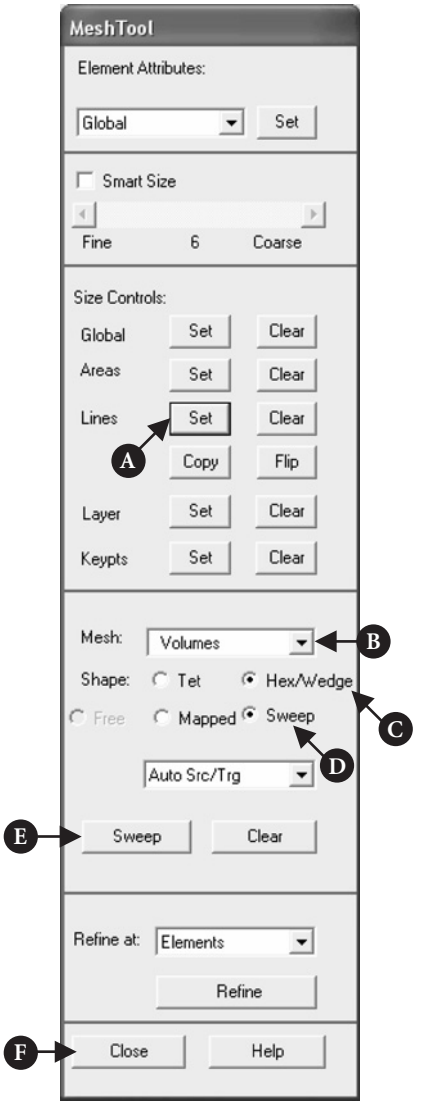


Figure 7.18 Checked Hex and Sweep options.

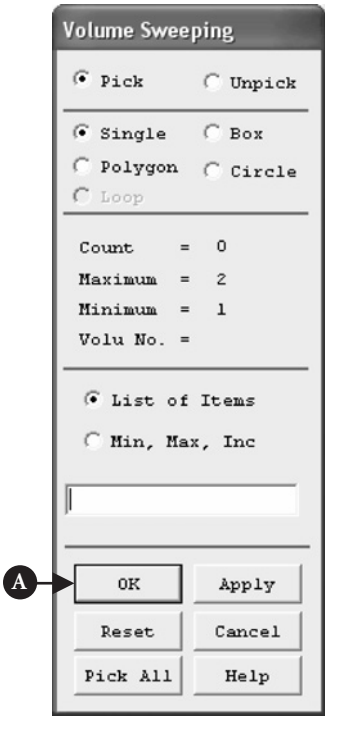


Figure 7.19 Volume Sweeping.

In the frame shown in Figure 7.23, select [A] **Areas**, [B] **Flexible**, and press the button [C] **Pick Target**. As a result of this selection, the frame shown in Figure 7.24 is produced.

The target area is the surface of pinhole in the arm and it should be picked and [A] **OK** button pressed. In **Contact Wizard** frame (see Figure 7.23) press [D] **Next** button (which should be highlighted when the target area is picked) to obtain another frame as shown in Figure 7.25.

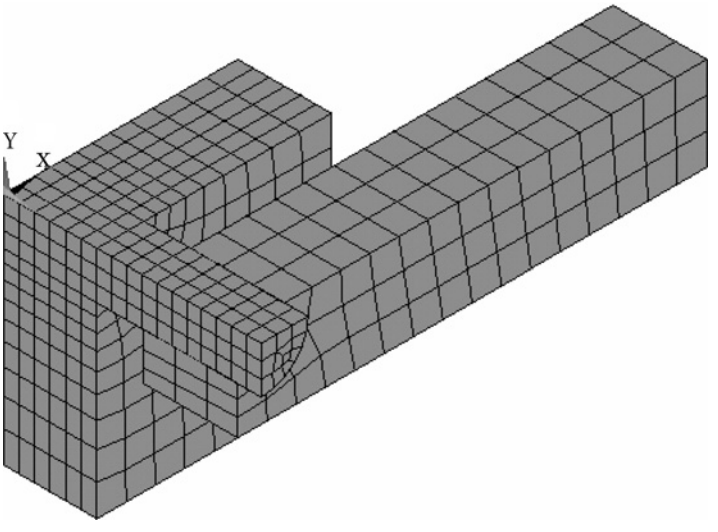


Figure 7.20 Model after meshing process.

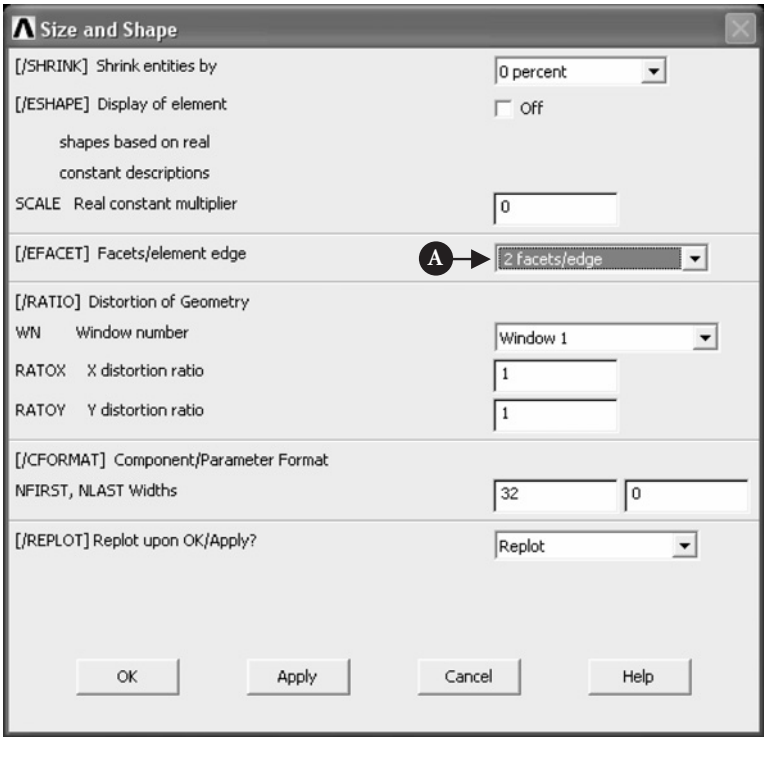


Figure 7.21 Size and Shape control of element edges.

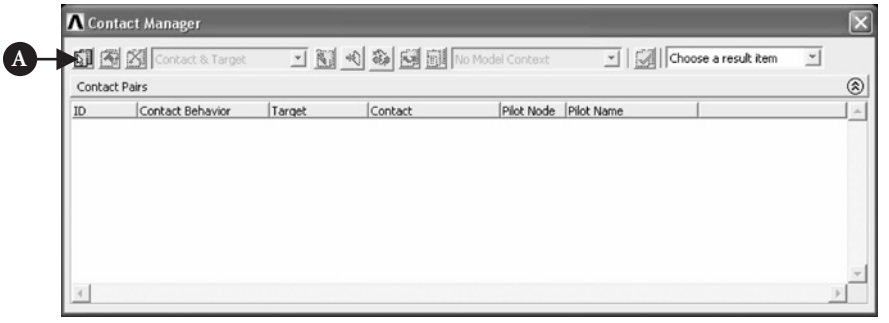


Figure 7.22 Contact Manager.

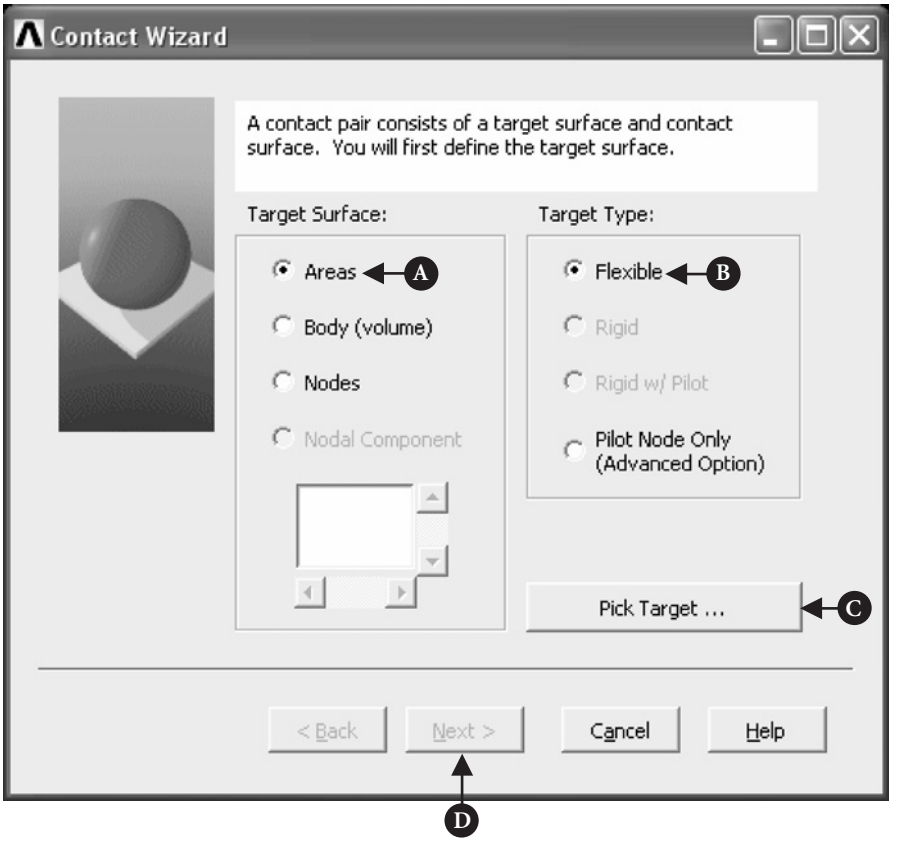


Figure 7.23 Contact Wizard.

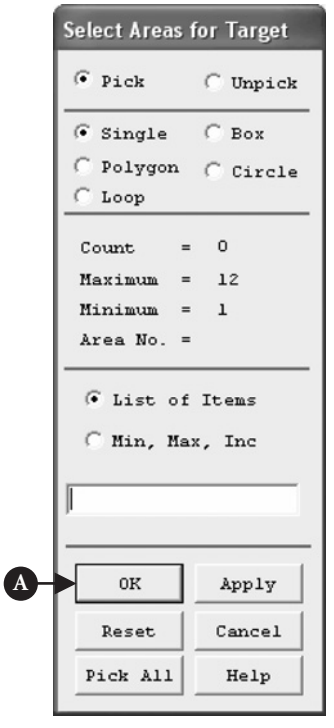


Figure 7.24 Select Areas for Target.

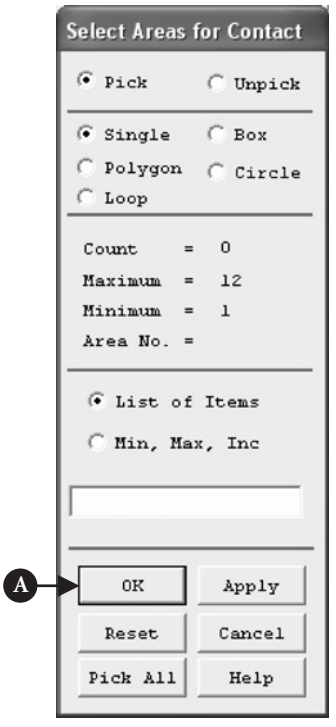


Figure 7.25 Select Areas for Contact.

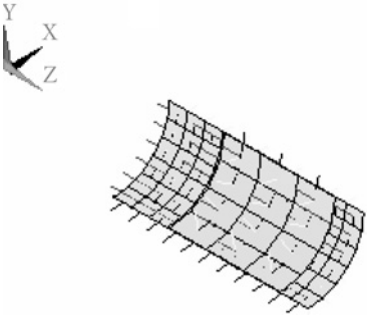


Figure 7.26 Pin contact area.

Surface area of the pin should be picked as the area for contact. When it is done and the [A] **OK** button pressed, **Contact Wizard** frame appears (see Figure 7.23). Pressing **Next** [D] button produces a frame in which **Material ID = 1**, **Coefficient of friction = 0.2** should be selected. Also, **Include Initial penetration** option should be checked. Next, **Optional settings** button should be pressed in order to further refine contact parameters. In the new frame, **Normal penalty stiffness = 0.1** should be selected. Also, **Friction** tab located in the top of the frame menu should be activated and **Stiffness matrix = Unsymmetric** should be selected. Afterward, pressing the button **OK** and next **Create** results in the image shown in Figure 7.26.

Also, **Contact Wizard** frame appears in the form shown in Figure 7.27.

The message is that the contact pair has been created. Pressing [A] **Finish** button closes the **Contact Wizard** tool.

Contact Manager frame appears again with the information pertinent to the problem considered. It is shown in Figure 7.28.

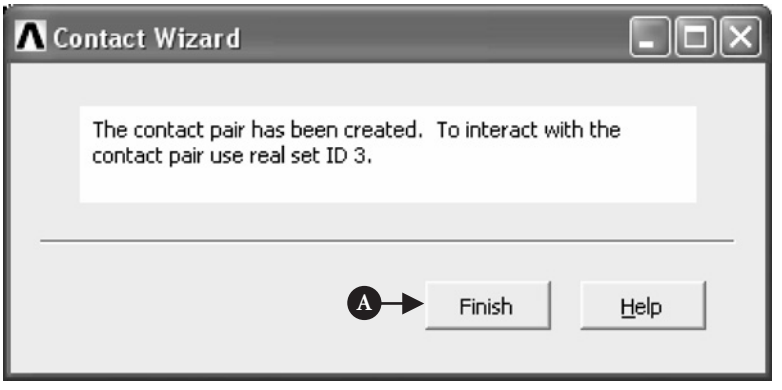


Figure 7.27 Contact Wizard final message.

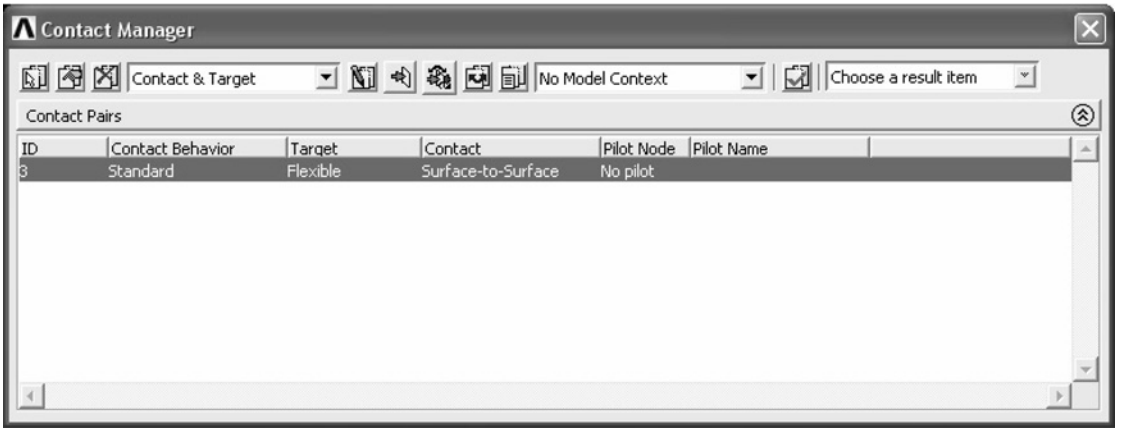


Figure 7.28 Contact Manager summary information.

7.2.1.6 SOLUTION

In the solution stage, solution criteria have to be specified first. As a first step in that process, symmetry constraints are applied on the quarter-symmetry model.

From **ANSYS Main Menu** select, **Solution → Define Loads → Apply → Structural → Displacement → Symmetry BC → On Areas**. The frame shown in Figure 7.29 appears.

Four areas which were created when the full configuration model was sectioned to produce quarter-symmetry model should be picked as shown in Figure 7.30. When that is done, click [A] **OK** button in the frame shown in Figure 7.29.

The next step is to apply boundary constraints on the block of which the pin is an integral part.

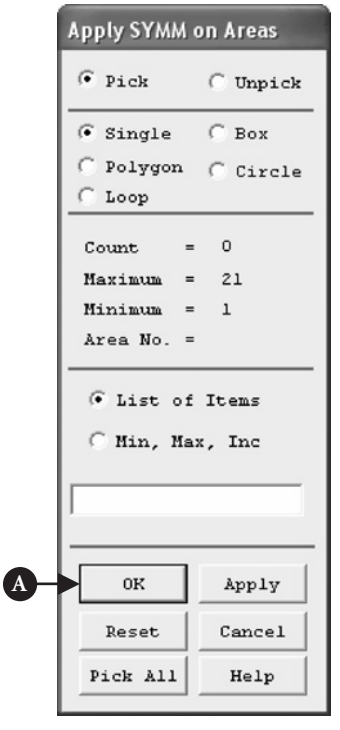


Figure 7.29 Apply SYMM on Areas.

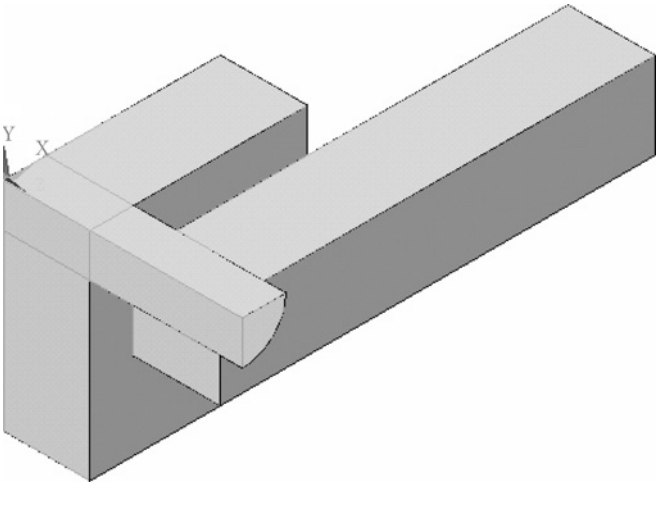


Figure 7.30 Selected areas on which symmetry constraints are applied.

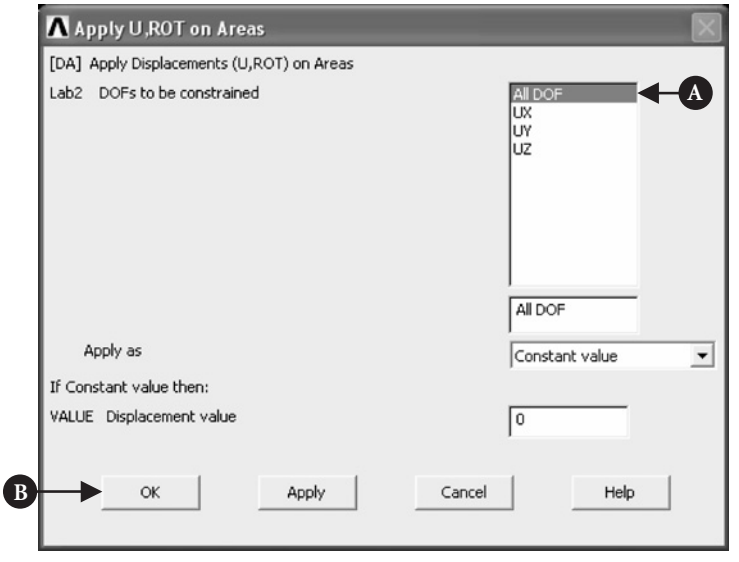


Figure 7.31 Apply U,ROT on Areas.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Areas**.

Backside of the block should be picked and then **OK** button pressed. The frame shown in Figure 7.31 appears.

All degrees of freedom [A] **All DOF** should be constrained with the displacement value equal to zero (see Figure 7.31). Clicking [B] **OK** button applies the constraints.

Because the original problem formulation asks for stress analysis when the arm is pulled out of the pin, the analysis involves a large displacement effects. The first type of load results from the interference fit between the pin and the arm.

From **ANSYS Main Menu** select **Solution → Analysis Type → Sol'n Controls**. The frame shown in Figure 7.32 appears. In the pull down menu select [A] **Large Displacement Static**. Further selected options should be [B] **Time at end of load step** = 100; [C] **Automatic time stepping** (pull down menu) = Off; and [D] **Number of substeps** = 1. All specified selections are shown in Figure 7.32. Pressing [E] **OK** button applies the settings and closes the frame.

The next action is to solve for the first type of load, i.e., interference fit.

From **ANSYS Main Menu** select **Solution → Solve → Current LS**. A frame showing review of information pertaining to the planned solution action appears. After checking that everything is correct, select **File → Close** to close that frame. Pressing **OK** button starts the solution. When the solution is completed, press **Close** button.

In order to return to the previous image of the model, select **Utility Menu → Plot → Replot**.

The second type of load is created when the arm is pulled out of the pin. A number of actions have to be taken in order to prepare the model for solution. First action is to apply a displacement along Z-axis equal to 2 cm (thickness of the arm) to all nodes on the front of the pin in order to observe this effect.

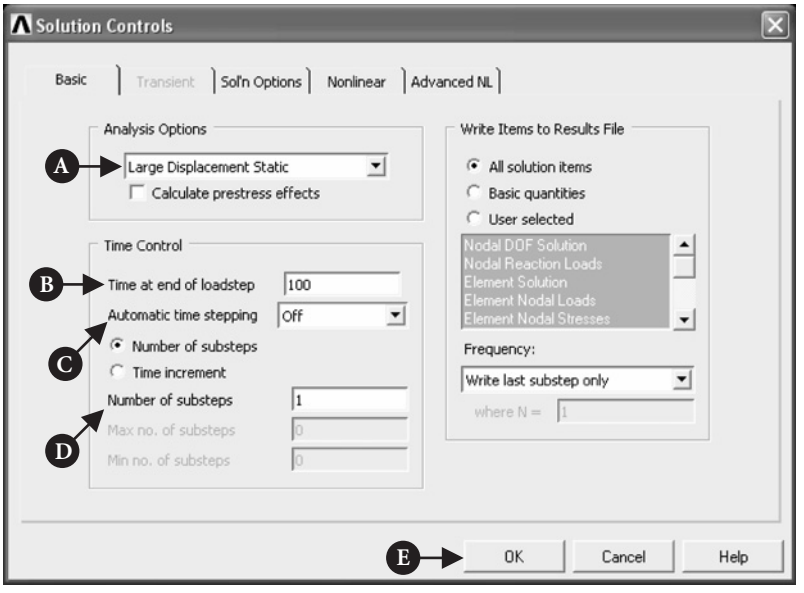


Figure 7.32 Solution Controls.

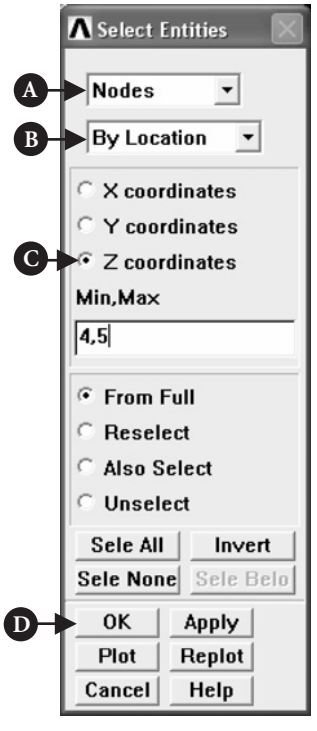


Figure 7.33 Select Entities.

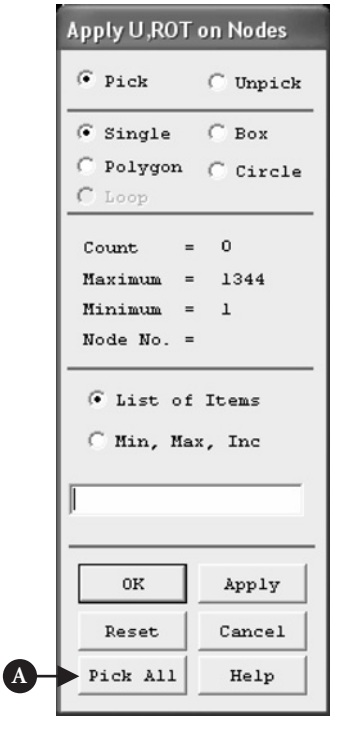


Figure 7.34 Apply U,ROT on Nodes.

From **Utility Menu → Select → Entities**. The frame shown in Figure 7.33 appears.

In this frame the following selections should be made: [A] **Nodes** (from pull down menu); [B] **By Location** (pull down menu); [C] **Z coordinates** (to be checked); **Min, Max** = 4,5. Pressing [D] **OK** implements the selections made.

Next, degrees of freedom in the Z-direction should be constrained with the displacement value of 2 (thickness of the arm).

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Nodes**. In response, a frame shown in Figure 7.34 appears.

By pressing [A] **Pick All** button, a frame shown in Figure 7.35 is called up.

As shown in Figure 7.35, [A] **DOF to be constrained** = **UZ** and the [B] **Displacement value** = 2. Pressing [C] **OK** button applies selected constraints.

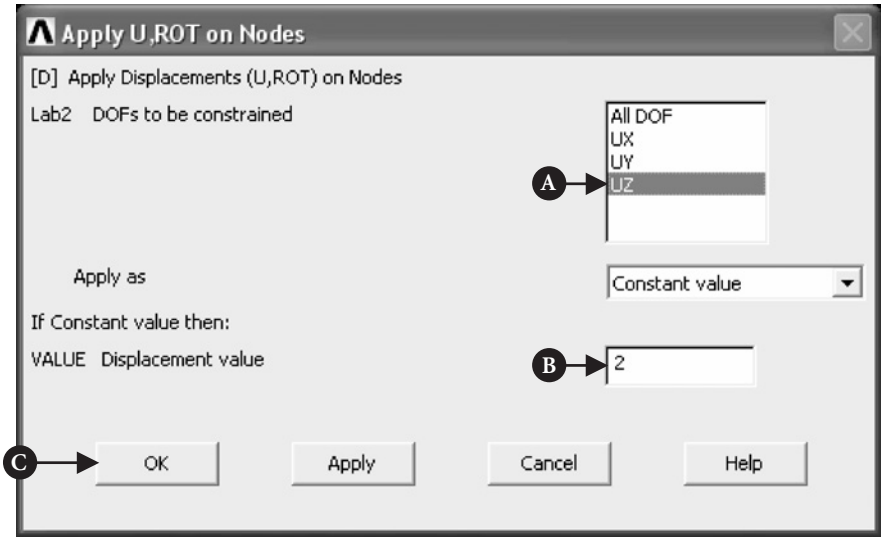


Figure 7.35 Apply U,ROT on Nodes.

Options for the analysis of pull-out operation have to be defined now.

From **ANSYS Main Menu** select **Solution → Analysis Type → Sol'n Controls**. The frame shown in Figure 7.36 appears in response.

As shown in Figure 7.36, the following selections defining solution controls were made: [A] **Time at end of load step** = 200; [B] **Automatic time stepping** (pull down menu) = On; [C] **Number of substeps** = 100; [D] **Max no. of substeps** = 10,000; [E] **Min no. of substeps** = 10; and **Frequency** (pull down menu) = Write every Nth substep, where $N = -10$. Pressing [F] **OK** button applied selected controls.

Now the model is ready to be solved for the load resulting from pulling the arm out.

From **ANSYS Main Menu** select **Solution → Solve → Current LS**. A frame giving summary information pertinent to the solution appears. After reviewing the information select **File → Close** to close the frame. After that, pressing **OK** button starts the solution. When the solution is done, press **Close** button.

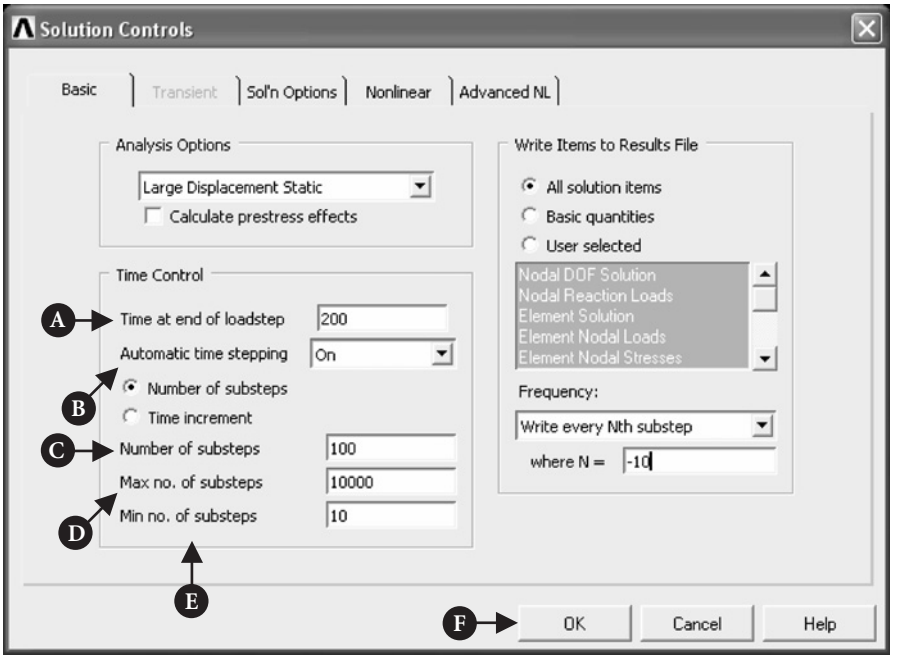


Figure 7.36 Solution Controls.

During solution process warning messages could appear. In order to make sure that the solution is done, it is practical to issue the command in the input box: **/NERR, 100, 100, 0FF**. This ensures that ANSYS program does not abort if it encounters a considerable number of errors.

7.2.1.7 POSTPROCESSING

Postprocessing stage is used to display solution results in a variety of forms.

The first thing to do is to expand the quarter-symmetry model into full configuration model.

From **Utility Menu** select **PlotCtrls → Style → Symmetry Expansion → Periodic/ Cyclic Symmetry**. Figure 7.37 shows the resulting frame.

Click [A] **¼ Dihedral Sym** button as shown in Figure 7.37 and press [B] **OK**.

From **Utility Menu** select **Plot → Elements**. An image of the full configuration model appears as shown in Figure 7.38.

The first set of results to observe in the postprocessing stage is to look at the state of stress due to interference fit between the pin and the hole in the arm.

From **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. The frame shown in Figure 7.39 is produced.

The selection [A] **Load step number** = 1 is shown in Figure 7.39. By clicking [B] **OK** button the selection is implemented.

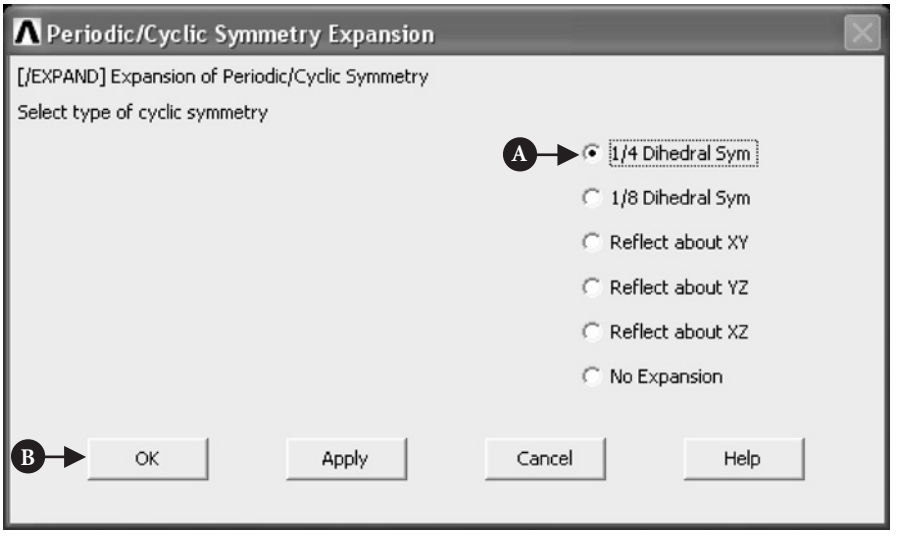


Figure 7.37 Periodic/Cyclic Symmetry Expansion.

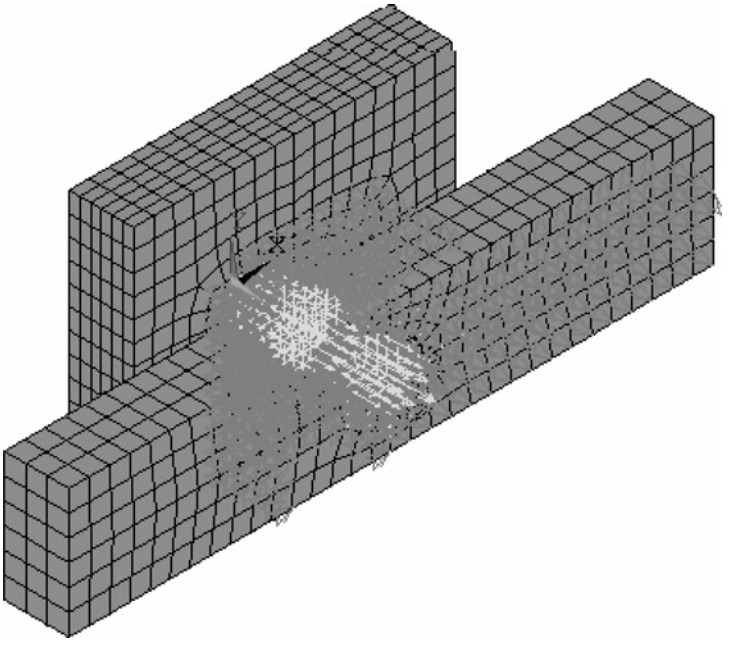


Figure 7.38 Full model with mesh of elements and applied constraints.

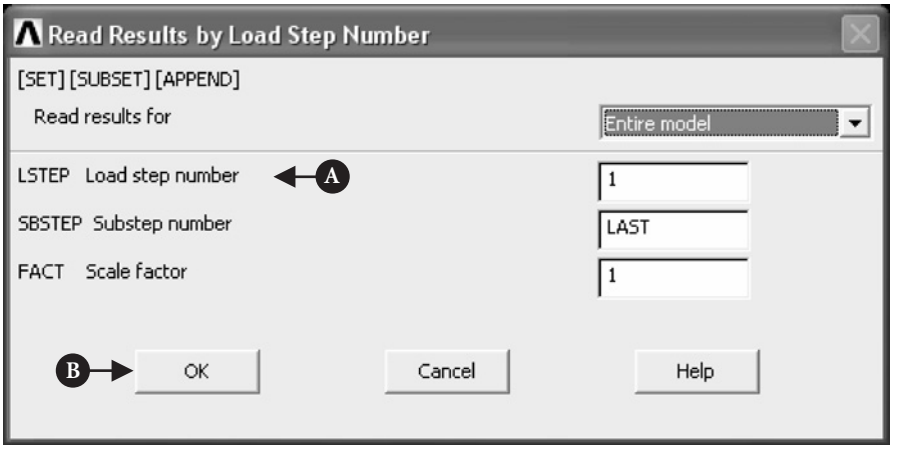


Figure 7.39 Read Results by Load Step Number.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. In the resulting frame, see Figure 7.40, the following selections

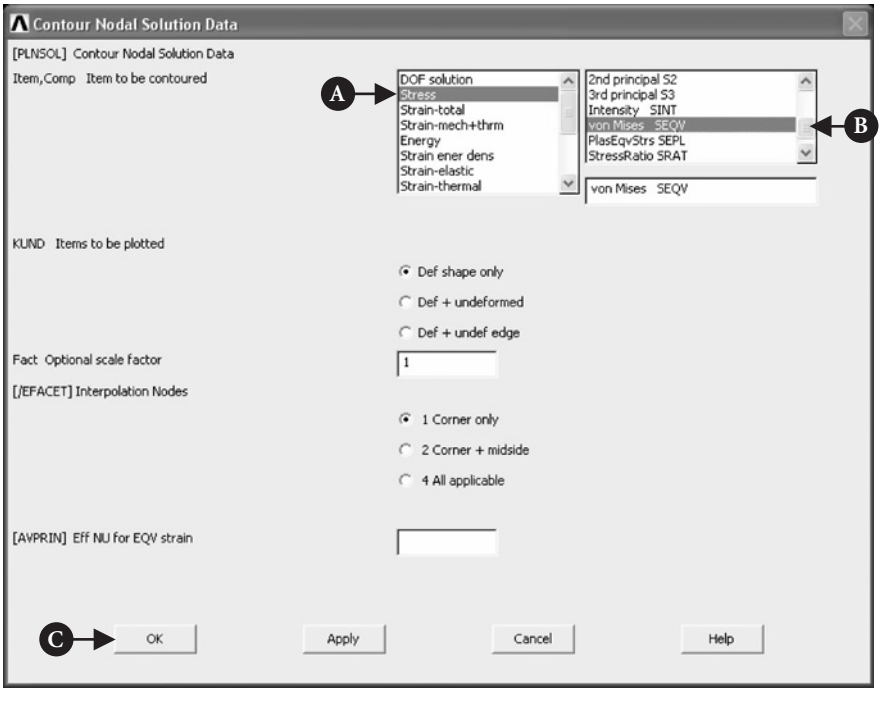


Figure 7.40 Contour Nodal Solution Data.

are made: [A] **Item to be contoured** = Stress and [B] **Item to be contoured** = von Mises (SEQV). Pressing [C] **OK** button implements selections.

Contour plot of von Mises stress (nodal solution) is shown in Figure 7.41.

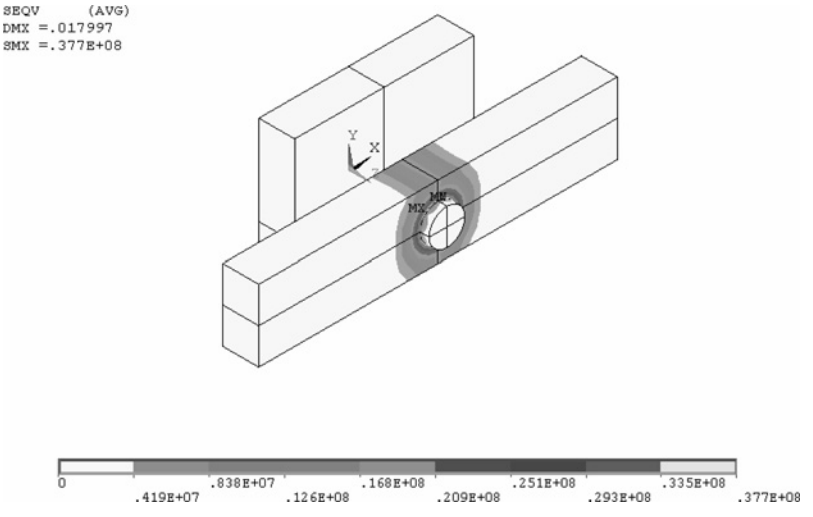


Figure 7.41 Contour plot of nodal solution (von Mises stress).

Figure 7.41 shows stress contour plot for the assembly of the pin in the hole. In order to observe contact pressure on the pin resulting from the interference fit, it is required to read results by time/frequency.

From **ANSYS Main Menu** select **General Postproc → Read Results → By Time/Freq**.

In the resulting frame, shown in Figure 7.42, the selection to be made is: [A] **Value of time or freq.** = 120. Pressing [B] **OK** implements the selection.

From **Utility Menu** choose, **Select → Entities**. The frame shown in Figure 7.43 appears.

In the frame shown in Figure 7.43, the following selections are made: [A] **Elements** (from pull down menu); [B] **By Elem Name** (from pull down menu); and [C] **Element Name** = 174. The element with the number 174 was introduced automatically during the process of creation of contact pairs described earlier. It is listed in the **Preprocessor → Element Type → Add/Edit/Delete** option. Selections are implemented by pressing [D] **OK** button.

From **Utility Menu** select **Plot → Elements**. Image of the pin with surface elements is produced (see Figure 7.44).

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 7.45 appears.

In the frame shown in Figure 7.45, the following selections are made: [A] **Contact** and [B] **Pressure**. These are items to be contoured. Pressing [C] **OK** implements selections made. Figure 7.46 shows image of the pin with pressure contours.

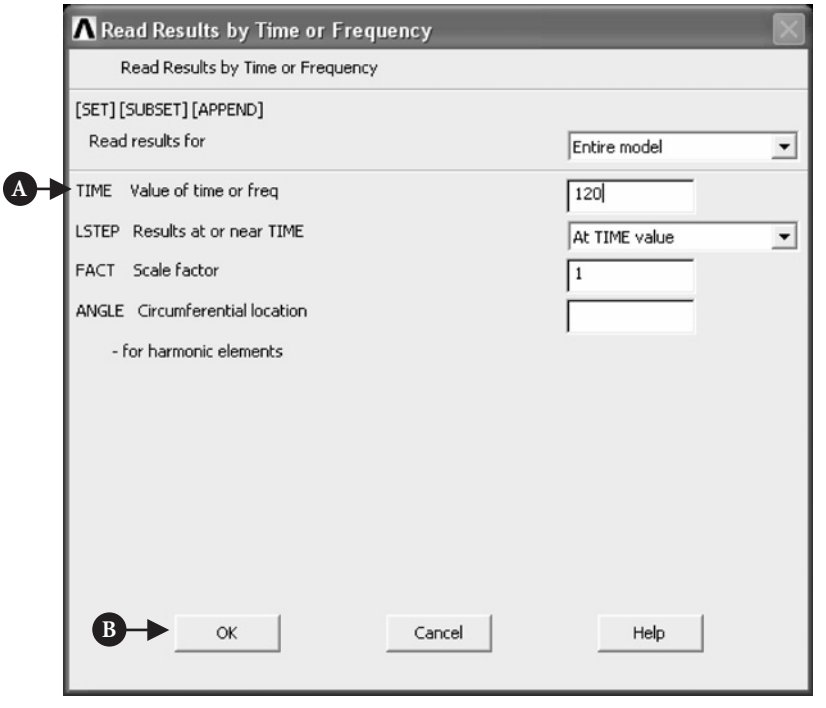


Figure 7.42 Read Results by Time or Frequency.

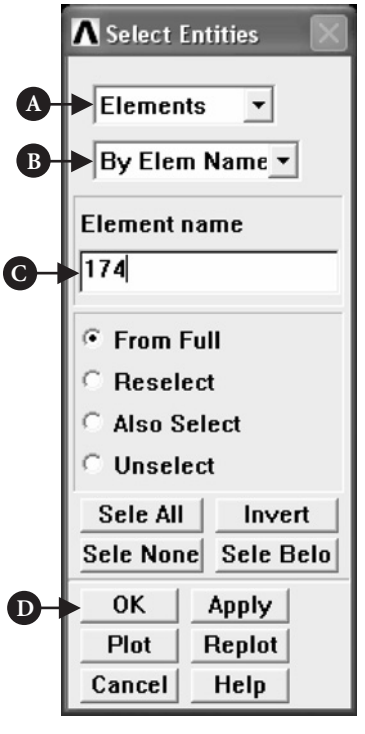


Figure 7.43 Select Entities.

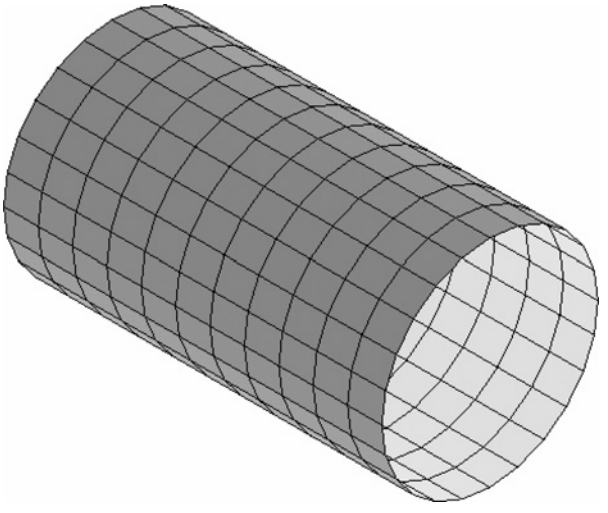


Figure 7.44 Pin with surface elements.

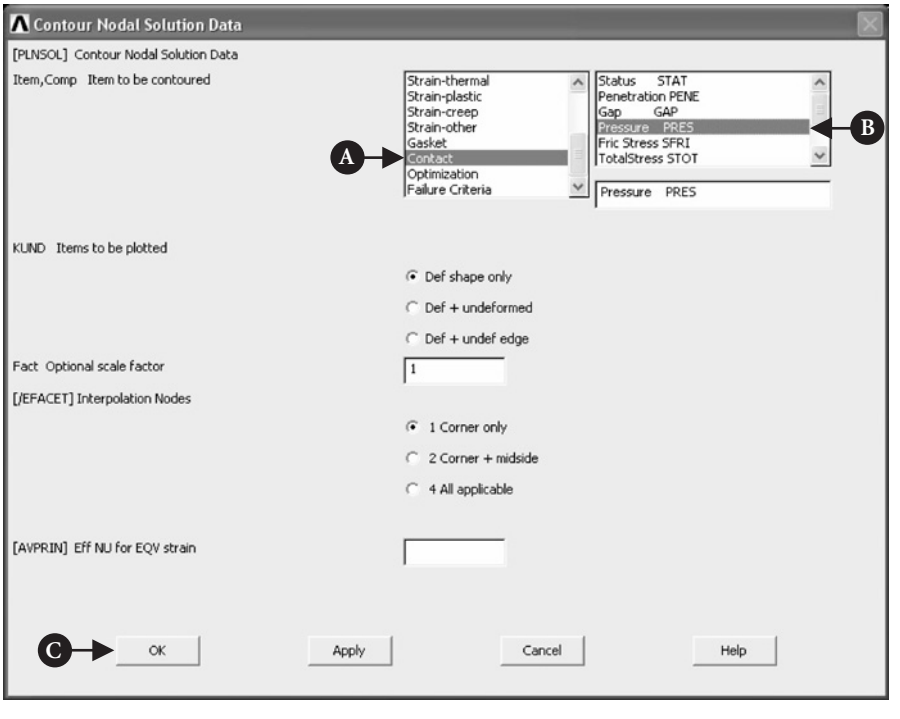


Figure 7.45 Contour Nodal Solution Data.

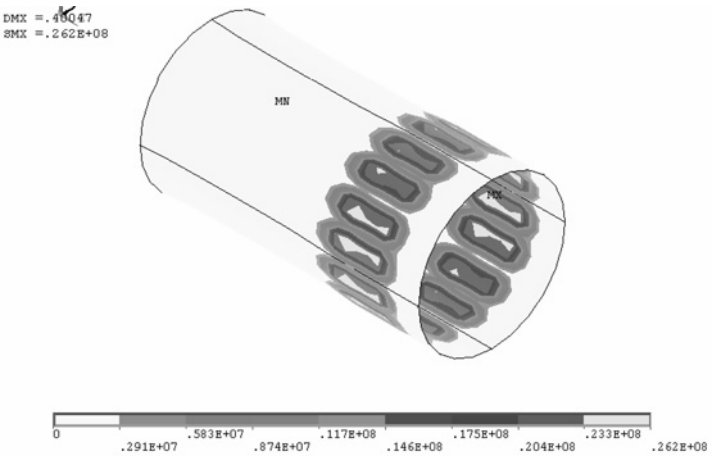


Figure 7.46 Contact pressure contours on surface of the pin.

The last action to be taken is to observe state of stress resulting from pulling out of the arm from the pin.

From **Utility Menu** choose **Select → Everything**. Next, from **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. The frame shown in Figure 7.47 appears.

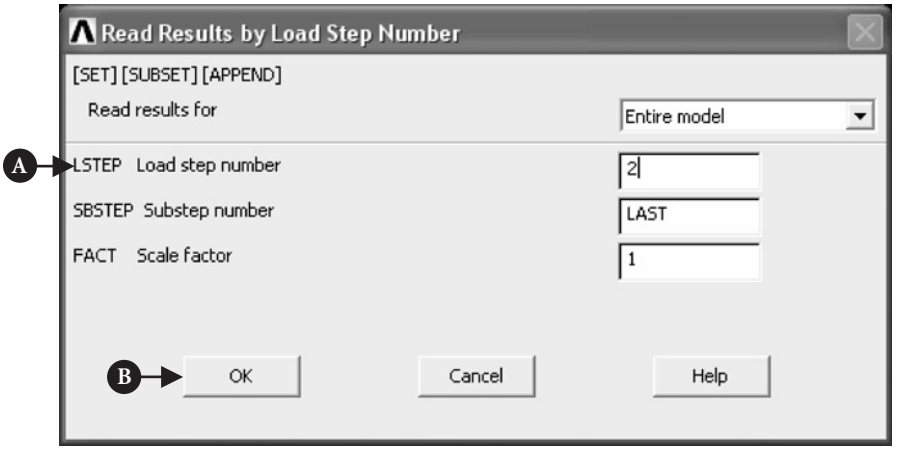


Figure 7.47 Read Results by Load Step Number.

As shown in Figure 7.47, [A] **Load step number** = 2 was selected. Pressing [B] **OK** implements the selection.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. In appearing frame (see Figure 7.40), the following are selected

as items to be contoured: [A] **Stress** and [B] **von Mises (SEQV)**. Pressing [C] **OK** implements selections made. Figure 7.48 shows stress contours on the pin resulting from pulling out the arm.

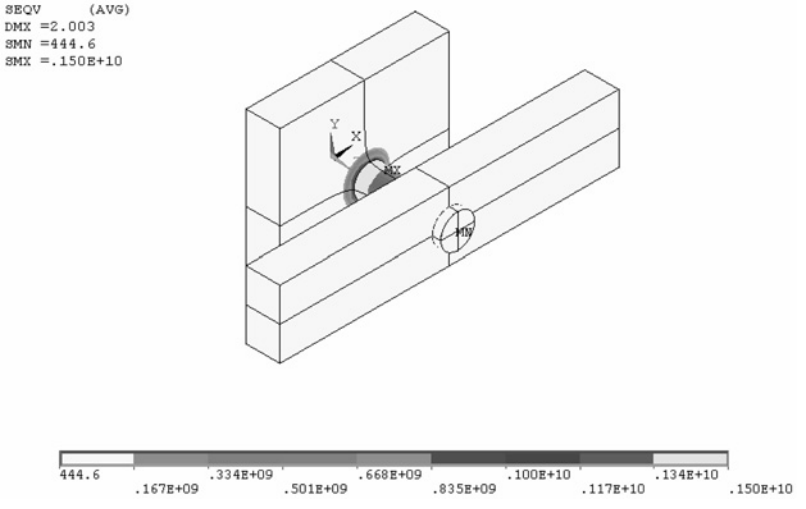


Figure 7.48 Pull-out stress contours on the pin.

7.2.2 Concave contact between cylinder and two blocks

7.2.2.1 Problem description

Configuration of the contact between cylinder and two blocks is shown in Figure 7.49.

This is a typical contact problem, which in engineering applications is represented by a cylindrical rolling contact bearing. Also, the characteristic feature of the contact is that, nominally, surface contact takes place between elements. In reality, this is never the case due to surface roughness and unavoidable machining errors and dimensional tolerance. There is no geometrical interference when the cylinder and two blocks are assembled.

This is a 3D analysis and advantage could be taken of the inherent symmetry of the model. Therefore, the analysis will be carried out on a half-symmetry model only. The objective of the analysis is to observe the stresses in the cylinder when the initial gap between two blocks is decreased by 0.05 cm.

The dimensions of the model are as follows: cylinder radius = 0.5 cm; cylinder length = 1 cm; block length = 2 cm; block width = 1 cm; and block thickness = 0.75 cm. Both blocks are geometrically identical. All elements are made of steel with Young's modulus = 2.1×10^9 N/m^2, Poisson's ratio = 0.3 and are assumed elastic. Friction coefficient at the interface between cylinder and the block is 0.2.

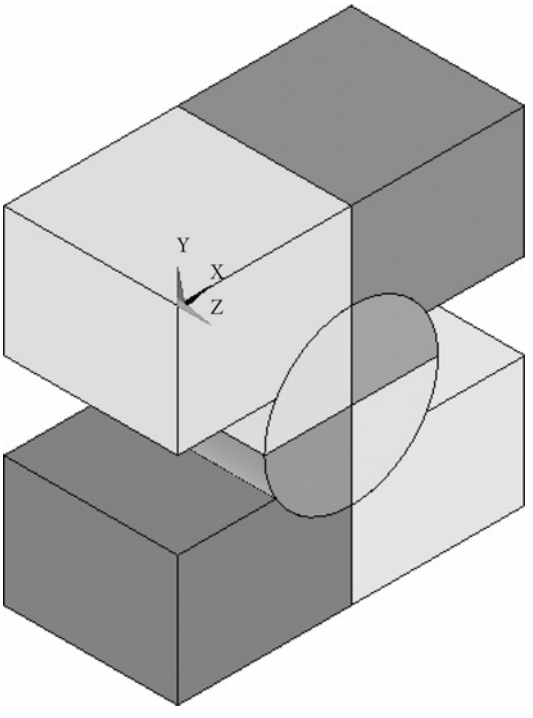


Figure 7.49 Configuration of the contact between cylinder and two blocks.

7.2.2.2 MODEL CONSTRUCTION

For the intended analysis a half-symmetry model is appropriate. It is shown in Figure 7.50.

In order to create a model shown in Figure 7.50, the use of two 3D primitives, namely block and cylinder, is made. The model is constructed using GUI facilities only. When carrying out Boolean operations on volumes it is quite convenient to have them numbered. This is done by selecting from the **Utility Menu → PlotCtrls → Numbering** and checking appropriate box to activate **VOLU** (volume numbers) option.

From **ANSYS Main Menu** select, **Preprocessor → Modelling → Create → Volumes → Block → By Dimensions**. In response, a frame shown in Figure 7.51 appears.

It can be seen from Figure 7.51 that appropriate X, Y, and Z coordinates were entered to create a block (vol. 1) with the length 2 cm ([A] X1 = −1; X2 = 1), width 1 cm ([B] Z1 = 0; Z2 = 1), and thickness 0.75 cm ([C] Y1 = −0.25; Y2 = −1). Next, from **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. In response, a frame shown in Figure 7.52 appears.

The input into the frame, shown in Figure 7.52, created a solid cylinder sector with the [A] radius 0.5 cm, [B] length 1 cm, [C] starting angle 180°, and [D] ending angle 360° (vol. 2).

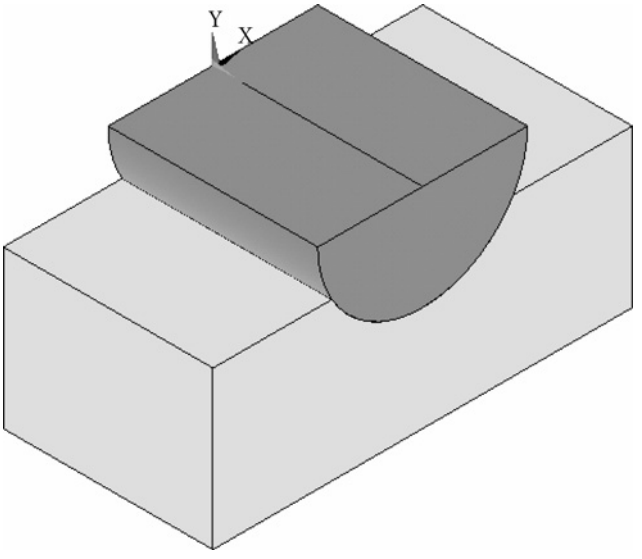


Figure 7.50 A half-symmetry model.

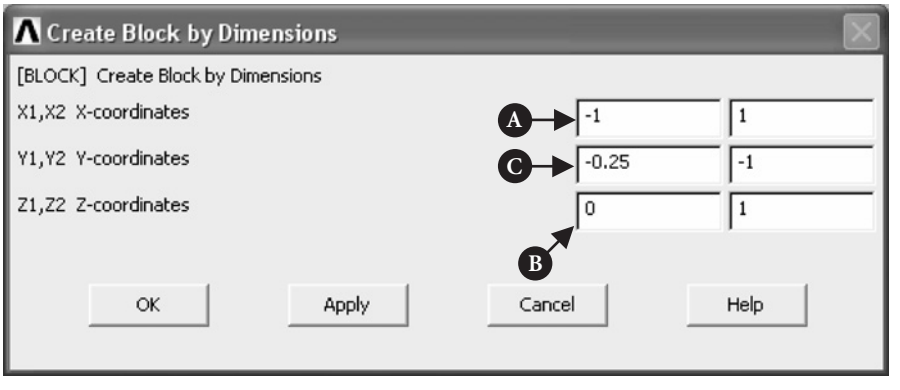


Figure 7.51 Create Block by Dimensions.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Overlap → Volumes**. The frame shown in Figure 7.53 appears.

Block (vol. 1) and cylinder (vol. 2) should be picked and [A] **OK** button pressed. As a result of that, block and cylinder are overlapped and three volumes created, i.e., volume 5 (block), volume 3 (section of the cylinder within the block), and volume 4 (remaining of the cylinder after a section of it has been subtracted).

From **ANSYS Main Menu** select **Modelling → Delete → Volume and Below**. The frame shown in Figure 7.54 appears.

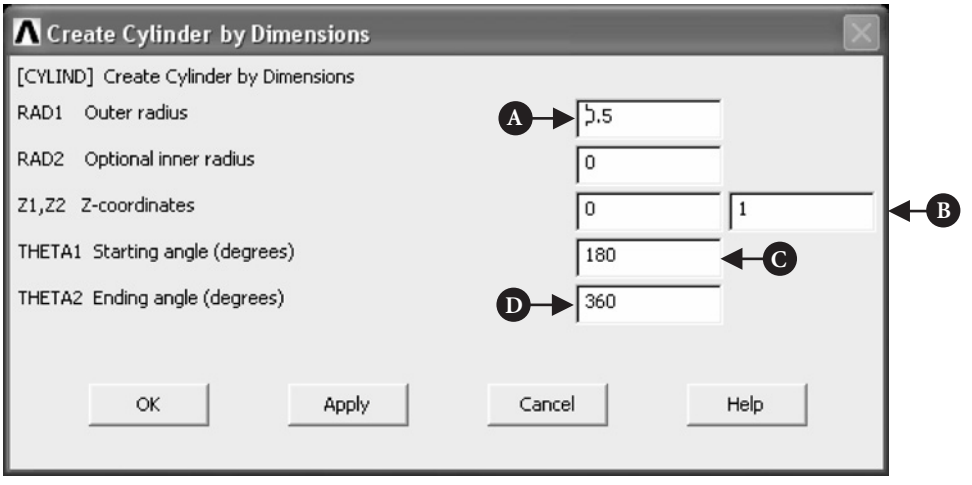


Figure 7.52 Create Cylinder by Dimensions.

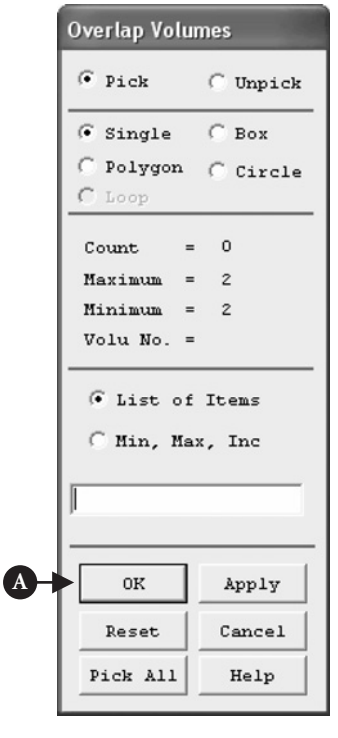


Figure 7.53 Overlap Volumes.

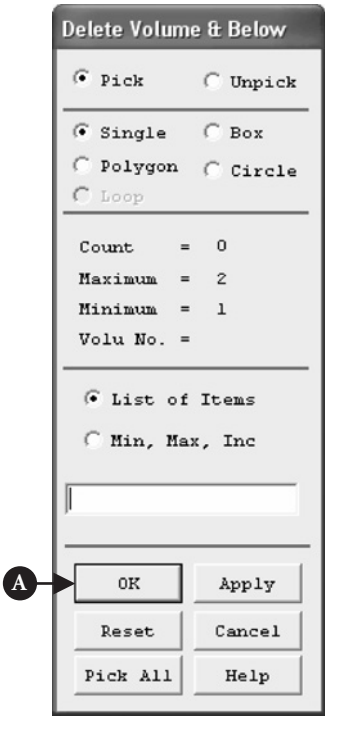


Figure 7.54 Delete Volume & Below.

Picking volume 4 and clicking [A] **OK** deletes it. The same operation should be repeated to deleted volume 3. As a result of that a block should be produced. Front view of the block is shown in Figure 7.55.

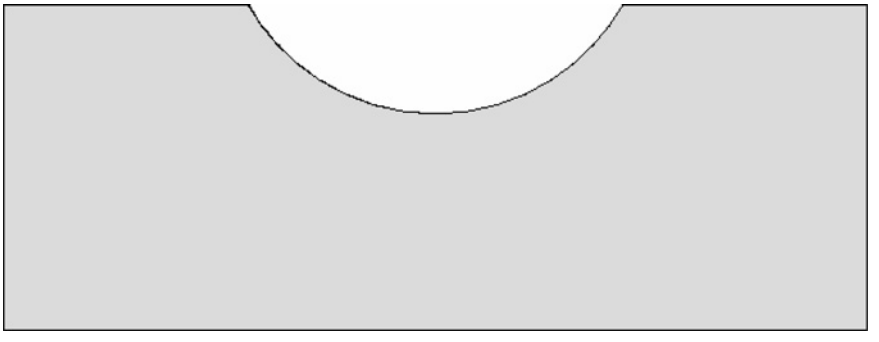


Figure 7.55 Block with a cylindrical cut-out.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Volumes → Cylinder → By Dimensions**. In response, a frame shown in Figure 7.56 appears.

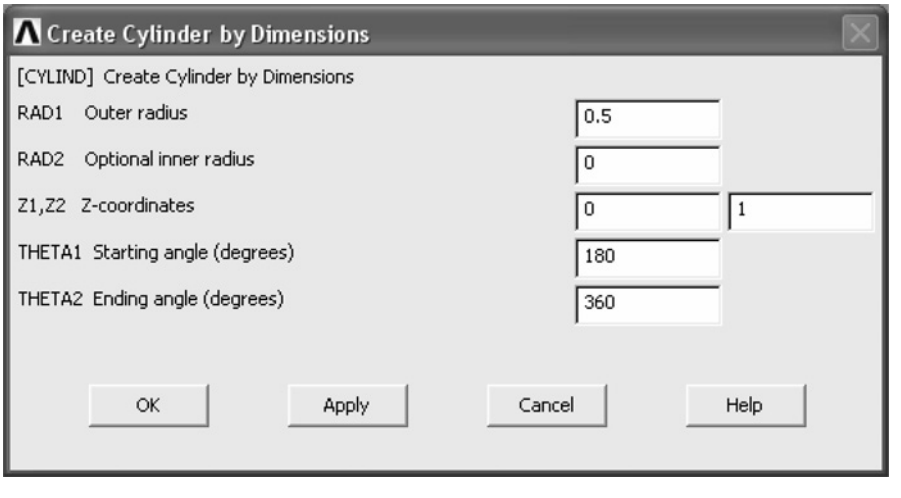


Figure 7.56 Create Cylinder by Dimensions.

A cylinder created earlier (see Figure 7.52 for inputs) is reproduced (vol. 1).

In order to create loading conditions at the contact interface, the cylinder is moved toward the block by 0.05 units.

From **ANSYS Main Menu** select **Modelling → Move/Modify → Volumes**. The frame shown in Figure 7.57 appears.

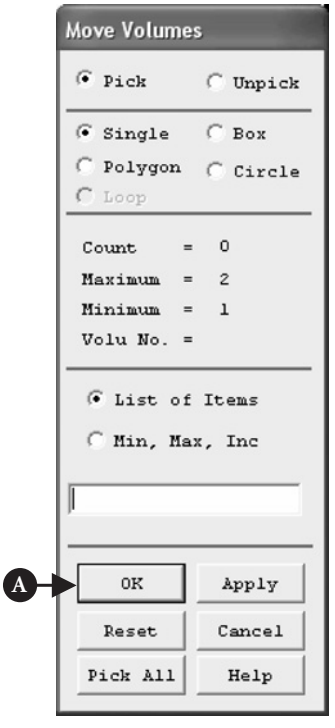


Figure 7.57 Move Volumes.

Selecting volume 1 and clicking [A] **OK** button calls up another frame shown in Figure 7.58.

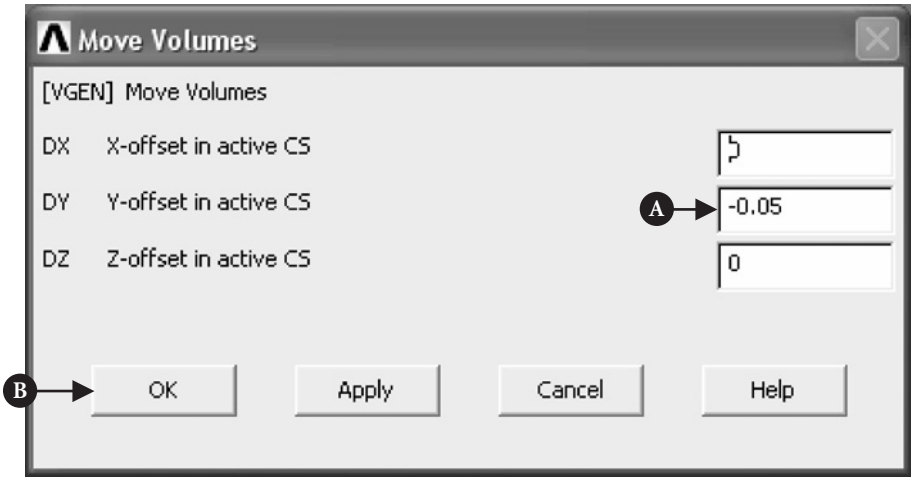


Figure 7.58 Move Volumes.

Figure 7.58 shows that the cylinder was moved by [A] 0.05 cm downward, i.e., toward the block, after clicking [B] **OK**.

From **Utility Menu** select **Plot → Replot** to view the cylinder positioned in required location. Finally, from **Utility Menu** select **PlotCtrls → View Settings → Viewing Direction**. The frame shown in Figure 7.59 appears.

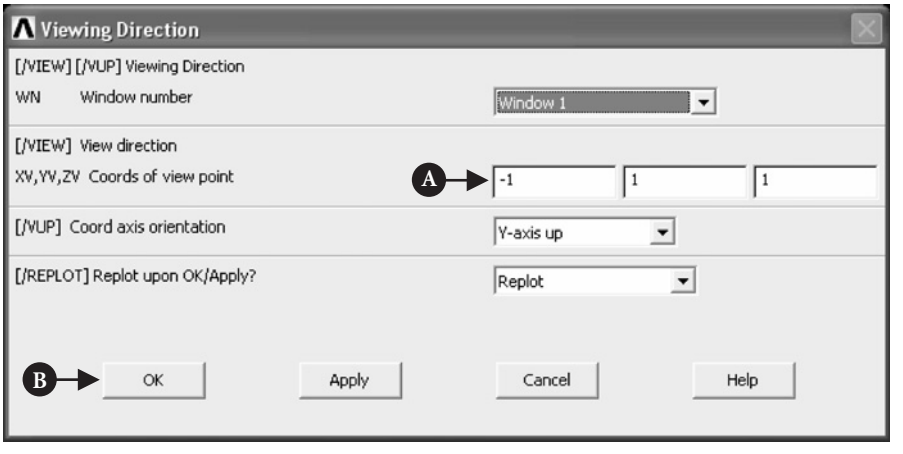


Figure 7.59 Viewing Direction.

By selecting [A] X, Y, and Z, coordinates as shown in Figure 7.59, clicking [B] **OK** button, and activating **Plot → Replot** command (Utility Menu), a half-symmetry model, shown in Figure 7.50, is finally created.

7.2.2.3 Material properties

Before any analysis is attempted, it is necessary to define properties of the material to be used.

From **ANSYS Main Menu** select **Preferences**. The frame in Figure 7.60 is produced.

From the Preferences list [A] **Structural** option was selected as shown in Figure 7.60. From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**. Double click **Structural → Linear → Elastic → Isotropic**. The frame shown in Figure 7.61 appears.

Enter [A] **EX** $= 2.1 \times 10^9$ for Young's modulus and [B] **PRXY** $= 0.3$ for Poisson's ratio. Then, click [C] **OK** and afterward **Material: Exit**. After defining properties of the material, the next step is to select element type appropriate for the analysis performed. From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.62 appears.

Click [A] **Add** in order to pull down another frame, shown in Figure 7.63.

In the left column click [A] **Structural Solid** and in the right column click [B] **Brick 8node 185**. After that click [C] **OK** and **Close** in order to finish element type selection.

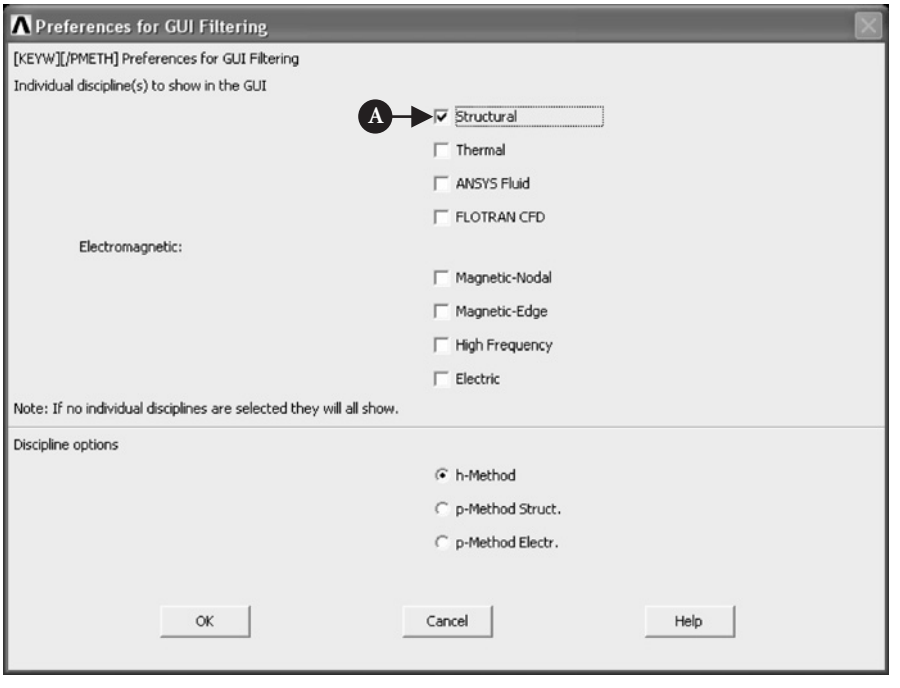


Figure 7.60 Preferences: Structural.

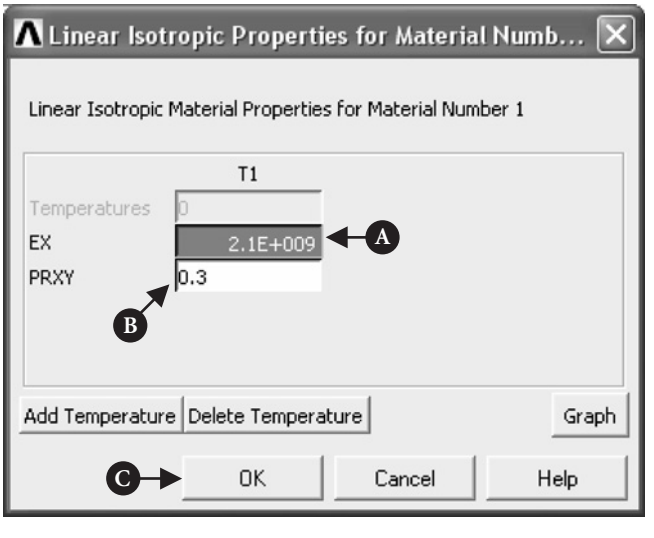


Figure 7.61 Material Properties.

Figure 7.62 Element Types.

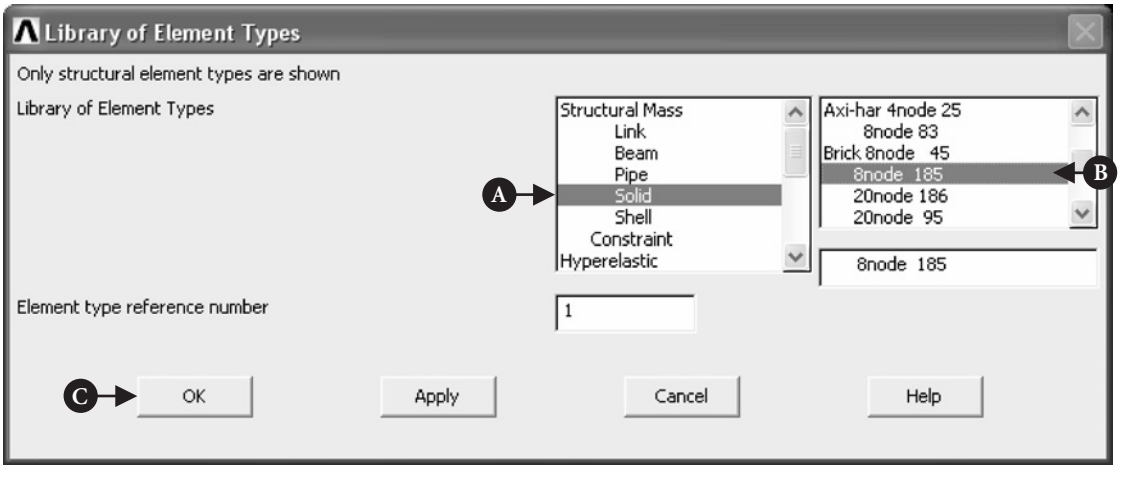


Figure 7.63 Library of Element Types.

7.2.2.4 MESHING

From **ANSYS Main Menu** select **Preprocessor → Meshing → MeshTool**. The frame shown in Figure 7.64 appears.

There are a number of options available. First step is to go to **Size Control: Lines** option and click [A] **Set** button. This opens another frame, as shown in Figure 7.65, prompting to pick lines on which element size is going to be controlled.

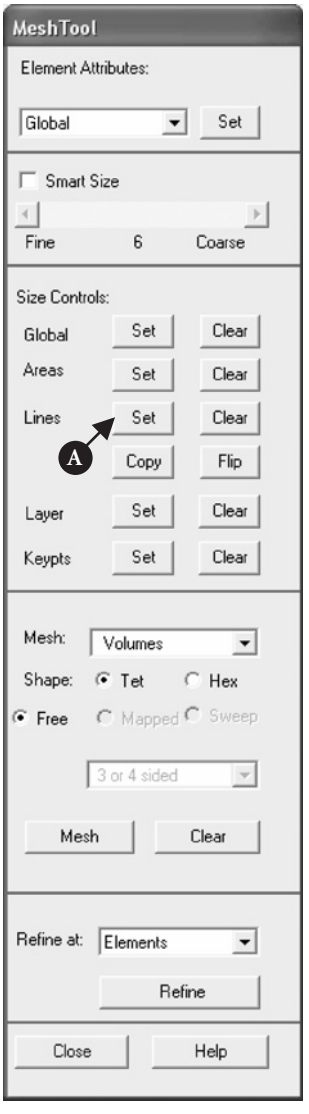


Figure 7.64 MeshTool options.

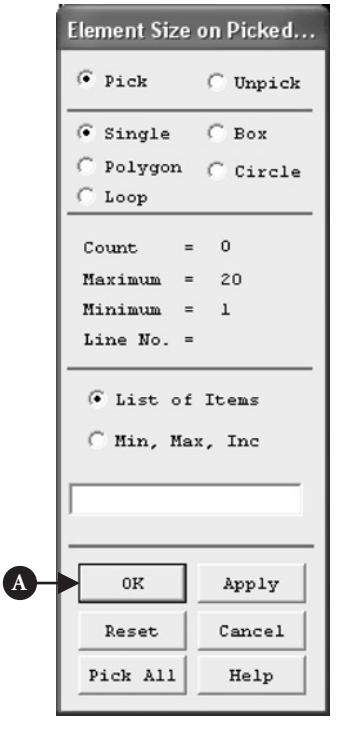


Figure 7.65 Element Size on Picked Lines.

Pick two horizontal lines on the front edge of the cylinder and click [A] **OK**. The frame shown in Figure 7.66 appears.

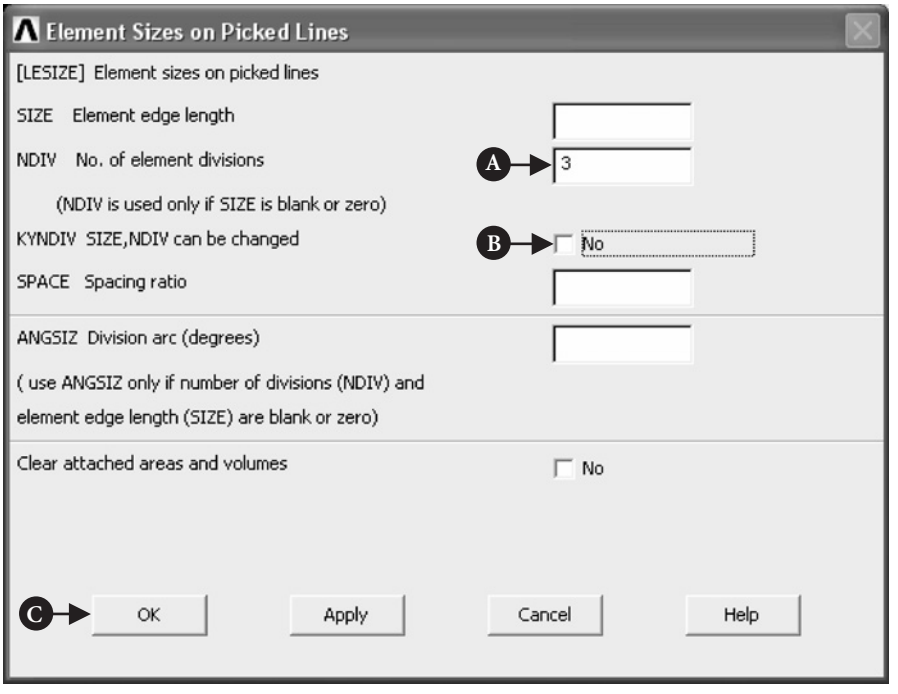


Figure 7.66 Element Sizes on Picked Lines.

In the box [A] **No. of element divisions** type 3 and change selection [B] **SIZE, NDIV can be changed** to **No.** by checking the box and click [C] **OK**. Both selections are shown in Figure 7.66.

Similarly, using **MeshTool** frame, click **Set** button in the **Size Controls → Lines** option and pick the curved line on the front of the block. Click **OK** afterward. The frame shown in Figure 7.66 appears. In the box [A] **No. of element divisions** type 4 this time and press [C] **OK** button.

In the frame **MeshTool** (see Figure 7.67) pull down [A] **Volumes** in the option **Mesh**.

Check [B] **Hex/Wedge** and [C] **Sweep** options. This is shown in Figure 7.67.

Pressing [D] **Sweep** button brings another frame asking to pick volumes to be swept (see Figure 7.68).

Pressing [A] **Pick All** button initiates meshing process. The model after meshing looks like the image in Figure 7.69. Pressing [E] **Close** button in **MeshTool** frame ends mesh generation stage.

After meshing completed, it is usually necessary to smooth element edges in order to improve graphic display. It can be accomplished using **PlotCtrls** facility in

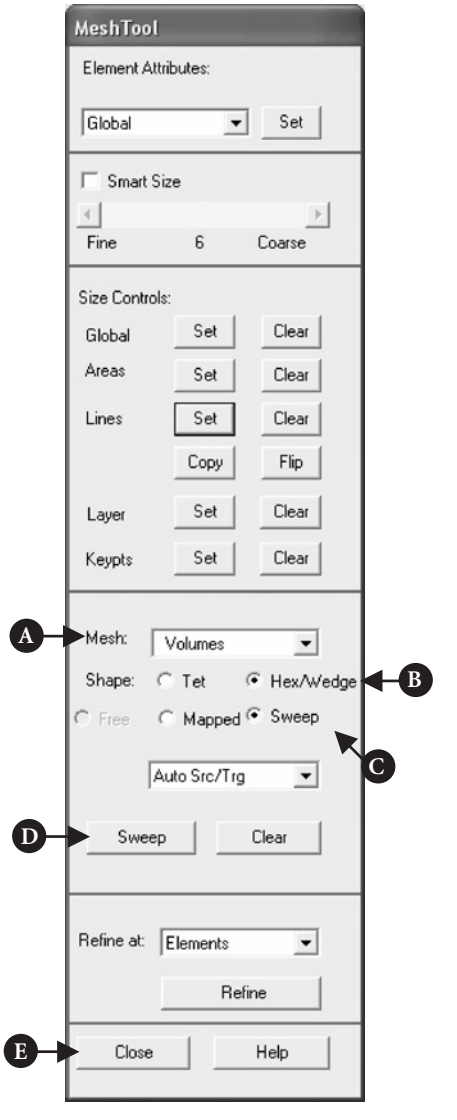


Figure 7.67 Checked Hex and Sweep options.

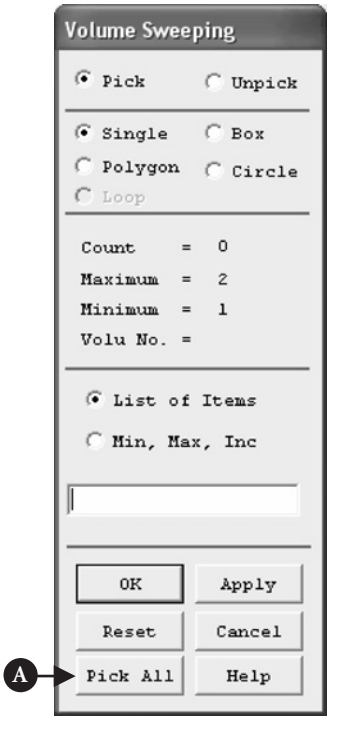


Figure 7.68 Volume Sweeping.

the Utility Menu. From **Utility Menu** select **PlotCtrls → Style → Size and Shape**. The frame shown in Figure 7.70 appears.

In the option [A] **Facets/element edge** select **2 facets/edge** and click [B] **OK** button to implement the selection as shown in Figure 7.70.

In solving the problem of contact between two elements, it is necessary to create contact pair. Contact Wizard is the facility offered by ANSYS.

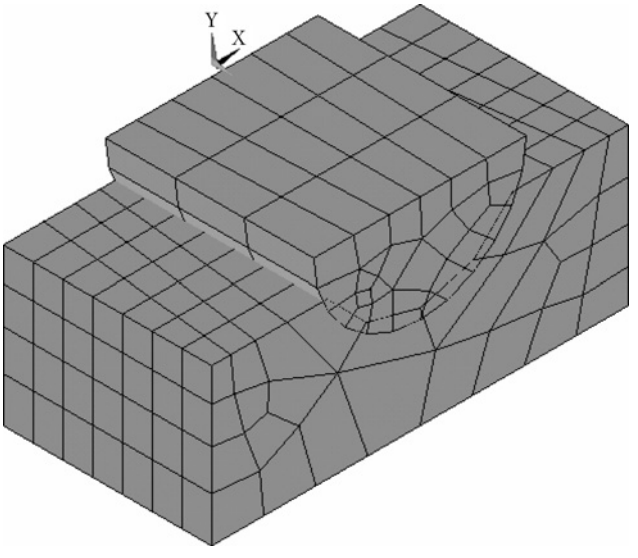


Figure 7.69 Model after meshing process.

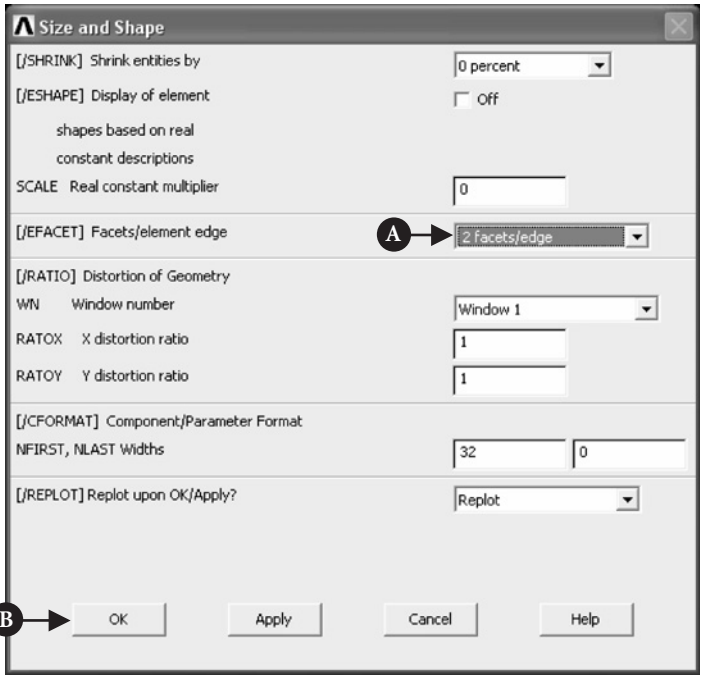


Figure 7.70 Control of element edges.

7.2.2.5 CREATION OF CONTACT PAIR

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Contact Pair**. As a result of this selection, a frame shown in Figure 7.71 appears.

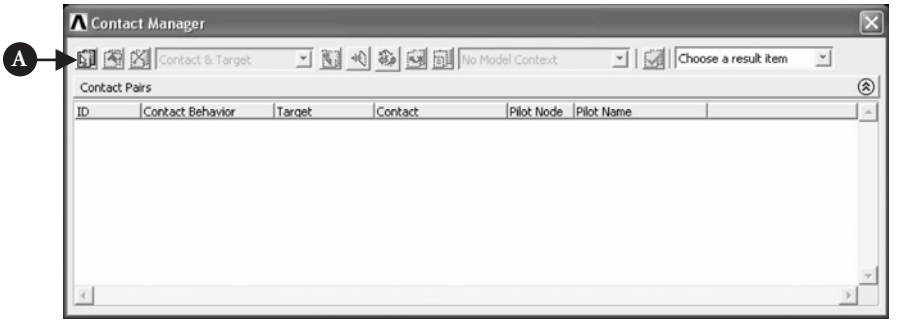


Figure 7.71 Contact Manager.

Contact Wizard button is located in the upper left-hand corner of the frame. By clicking [A] on this button a Contact Wizard frame, as shown in Figure 7.72, is produced.

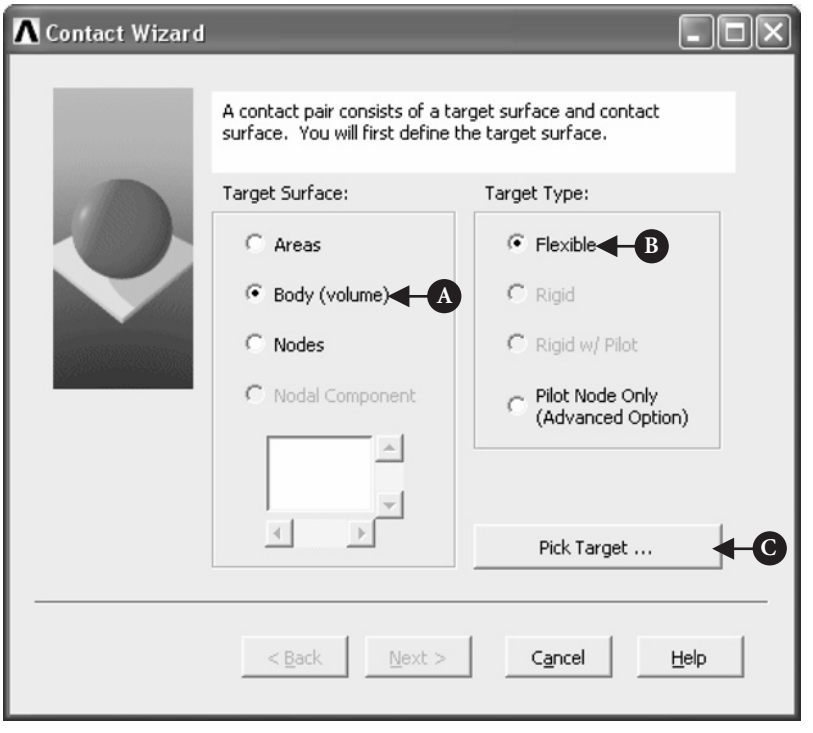


Figure 7.72 Contact Wizard (target).

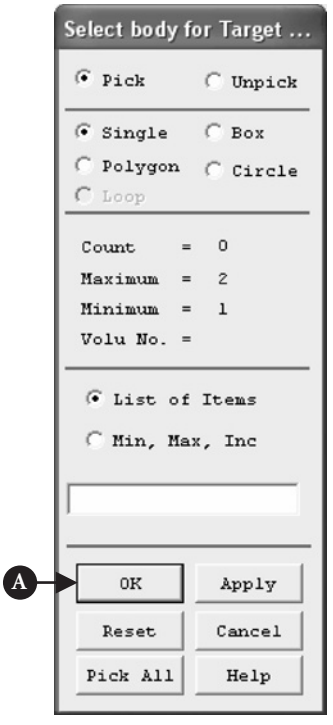


Figure 7.73 Select body for Target.

In the frame shown in Figure 7.72 select [A] **Body** (volume), [B] **Flexible**, and press [C] **Pick Target**. The frame shown in Figure 7.73 is produced.

Select block as target by picking on it and press [A] **OK** button in the frame of Figure 7.73. Again Contact Wizard frame appears and this time **Next** button should be pressed to obtain the frame shown in Figure 7.74.

Press [A] **Pick Contact** button to create the frame in Figure 7.75.

Pick cylinder as contact and press **OK** button. Again Contact Wizard frame appears where **Next** button should be clicked. The frame of Figure 7.76 appears.

In this frame select [A] **Coefficient of Friction** = 0.2 and check box [B] **Include initial penetration**. Next press [C] **Optional settings** button to call up another frame. In the new frame (Figure 7.77), **Normal penalty stiffness = 0.1** should be selected. Also, **Friction** tab located in the top of the frame menu should be activated and **Stiffness matrix = Unsymmetric** selected.

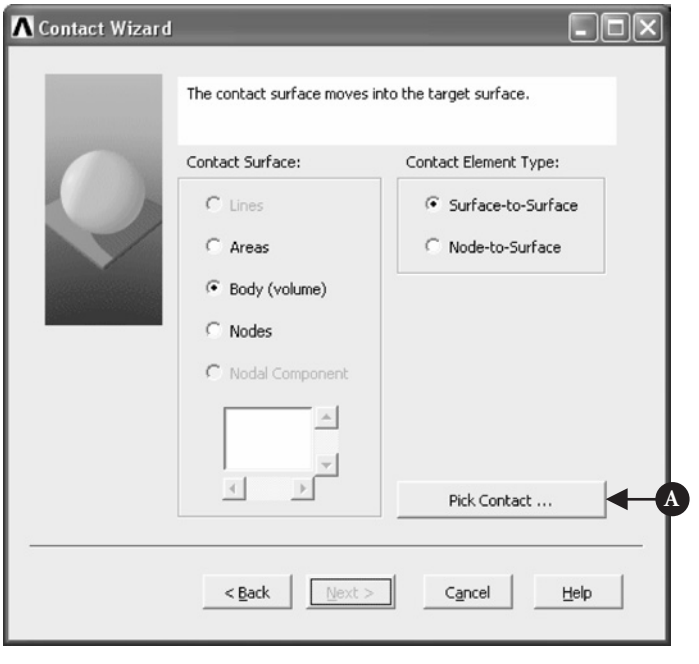


Figure 7.74 Contact Wizard (contact).

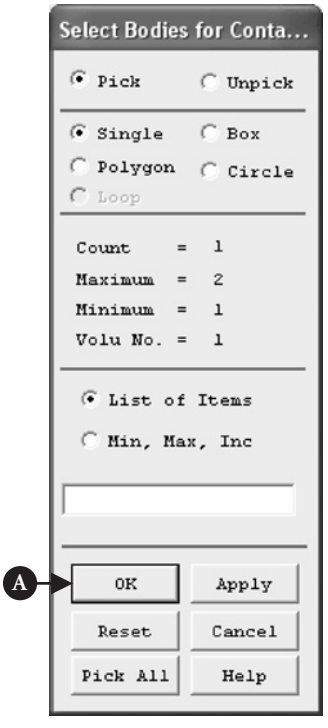


Figure 7.75 Select Bodies for Contact.

Pressing [A] **OK** button brings back **Contact Wizard** frame (see Figure 7.76) where the [D] **Create** button should be pressed.

Created contact pair is shown in Figure 7.78.

Finally, **Contact Wizard** frame should be closed by pressing **Finish** button. Also, **Contact Manager** summary information frame should be closed.

7.2.2.6 SOLUTION

Before the solution process can be attempted, solution criteria have to be specified. As a first step in that process, symmetry constraints are applied on the half-symmetry model.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → Symmetry BC → On Areas**. The frame shown in Figure 7.79 appears.

Three horizontal surfaces should be selected by picking them and then clicking [A] **OK**. As a result, image shown in Figure 7.80 appears.

The next step is to apply constraints on the bottom surface of the block. Form **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Areas**. The frame shown in Figure 7.81 appears.

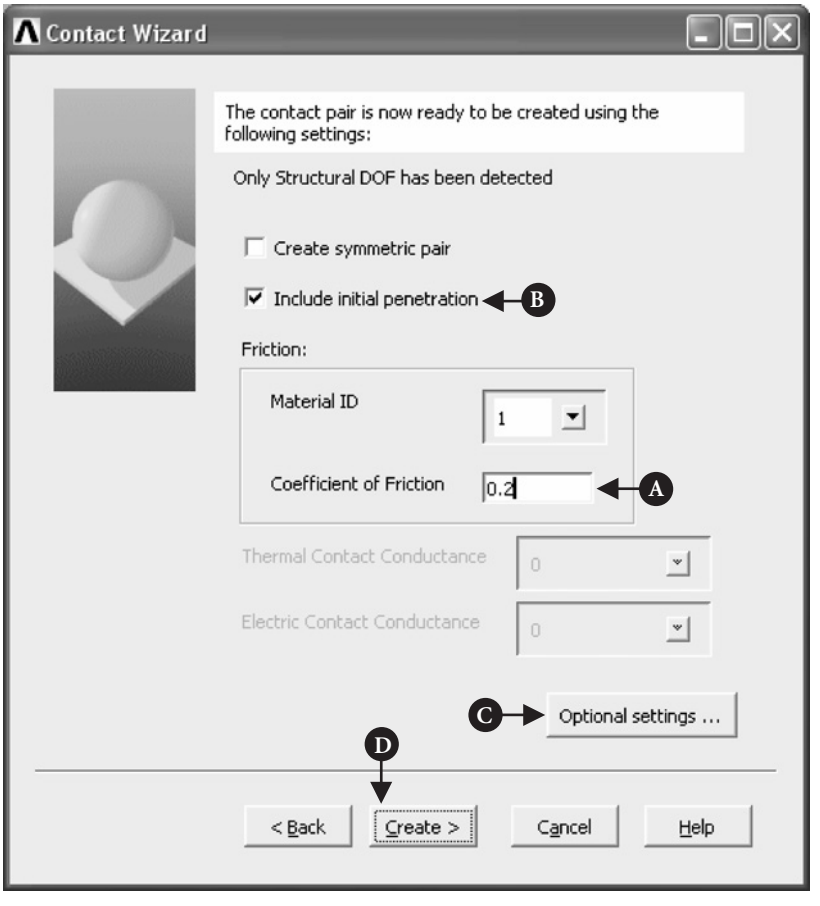


Figure 7.76 Contact Wizard (optional settings).

After selecting required surface (bottom surface of the block) and pressing [A] **OK** button, another frame appears in which the following should be selected: **DOFs to be constrained** = **All DOF** and **Displacement value** = 0. Selections are implemented by pressing **OK** button in the frame.

Because the cylinder has been moved toward the block by 0.05 cm, in order to create interference load, the analysis involves a large displacement effects.

From **ANSYS Main Menu** select **Solution → Analysis Type → Sol'n Controls**. The frame shown in Figure 7.82 appears.

In the pull down menu select [A] **Large Displacement Static**. Further selected options should be: [B] **Time at end of load step** = 100; [C] **Automatic time stepping** (pull down menu) = Off; and [D] **Number of substeps** = 1. All specified selections are shown in Figure 7.82. Pressing [E] **OK** button implements the settings and closes the frame.

Now the modeling stage is completed and the solution can be attempted. From **ANSYS Main Menu** select **Solution → Solve → Current LS**. A frame showing review of information pertaining to the planned solution action appears. After checking that

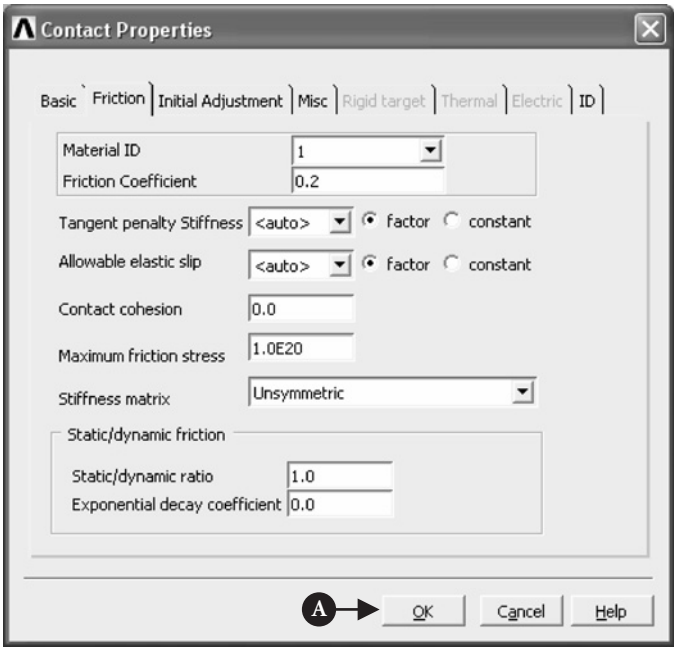


Figure 7.77 Contact Properties (optional settings).

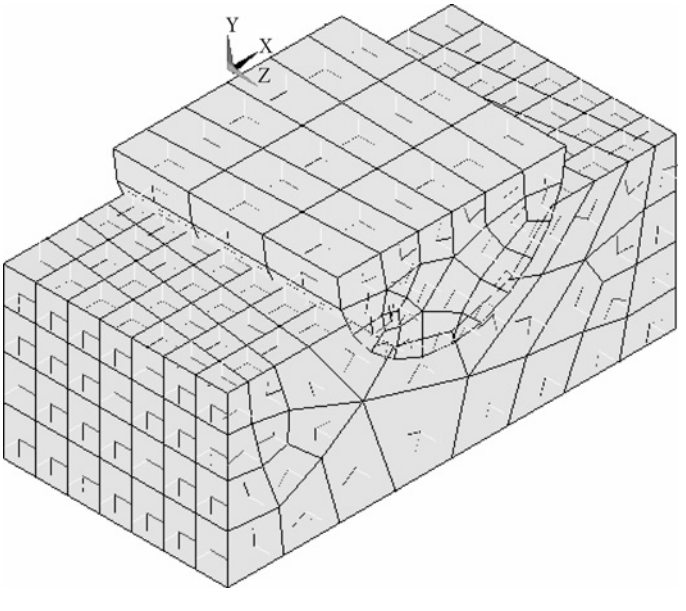


Figure 7.78 Contact Pair created by Contact Wizard.

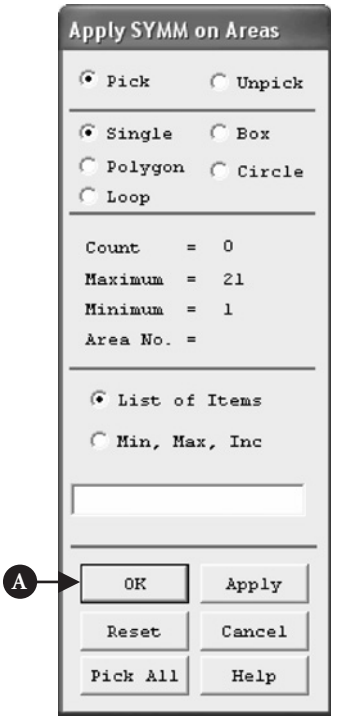


Figure 7.79 Apply SYMM on Areas.

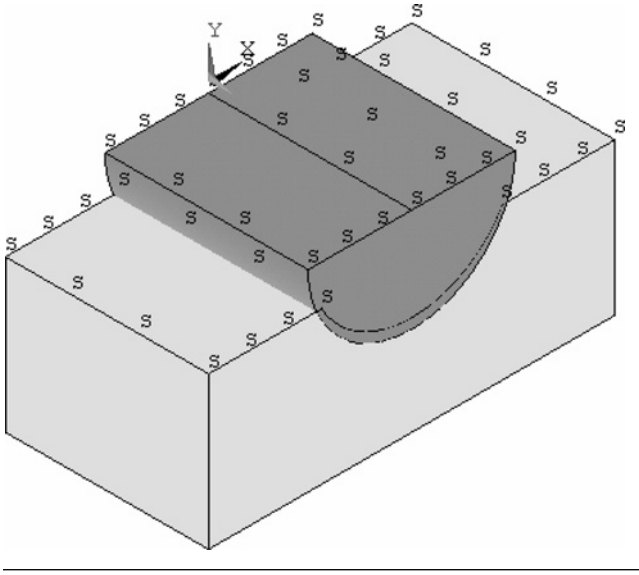


Figure 7.80 Symmetry constraints applied on three horizontal areas.

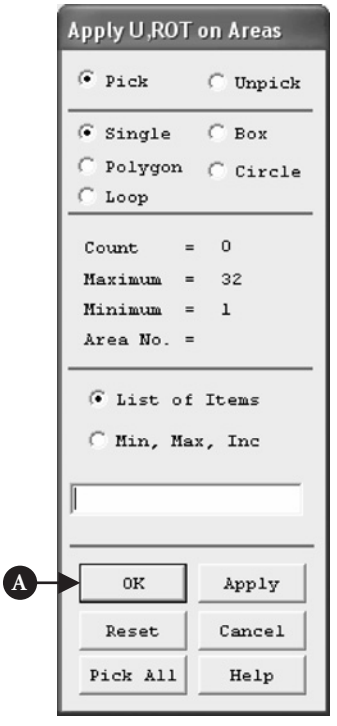


Figure 7.81 Apply U,ROT on Areas.

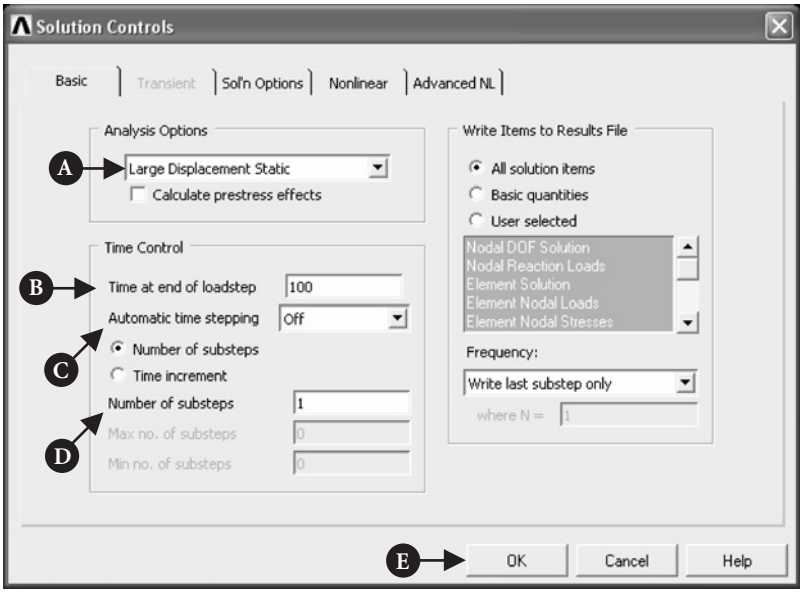


Figure 7.82 Solution Controls.

everything is correct, select **File → Close** to close that frame. Pressing **OK** button starts the solution. When the solution is completed, press **Close** button.

In order to return to the previous image of the model select **Utility Menu → Plot → Replot**.

7.2.2.7 POSTPROCESSING

Solution results can be displayed in a variety of forms using postprocessing facility. For the results to be viewed for the full model, the half-symmetry model used for analysis has to be expanded.

From **Utility Menu** select **PlotCtrls → Style → Symmetry Expansion → Periodic/Cyclic Symmetry**. Figure 7.83 shows the resulting frame.

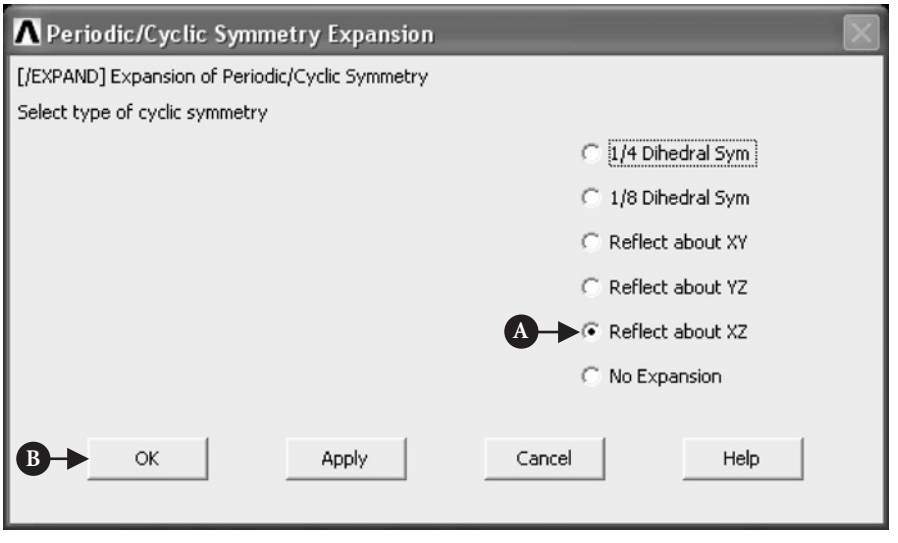


Figure 7.83 Periodic/Cyclic Symmetry Expansion.

In the frame shown in Figure 7.83 [A] **Reflect about XZ** was selected. After clicking [B] **OK** button in the frame and selecting from **Utility Menu**, **Plot → Elements**, an image of full model, as shown in Figure 7.84, is produced.

The objective of the analysis presented here was to observe stresses in the cylinder produced by the reduction of the initial gap between two blocks by 0.05 cm (an interference fit). Therefore, form **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. The frame shown in Figure 7.85 is produced.

The selection [A] **Load step number =** 1, shown in Figure 7.85, is implemented by clicking [B] **OK** button.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. In the resulting frame (see Figure 7.86) the following selections are made: [A] **Item to be contoured = Stress** and [B] **Item to be contoured = von Mises (SEQV)**. Pressing [C] **OK** implements selections.

Contour plot of von Mises stress (nodal solution) is shown in Figure 7.87.

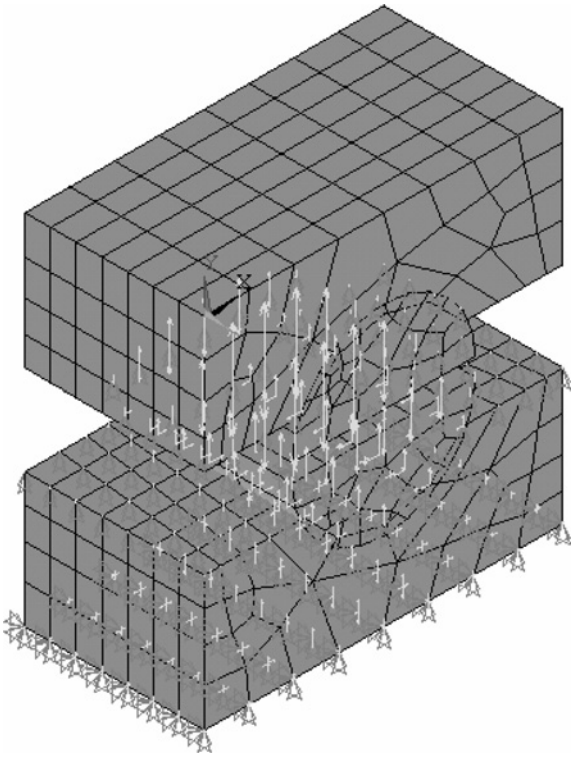


Figure 7.84 Full model with mesh of elements and applied constraints.

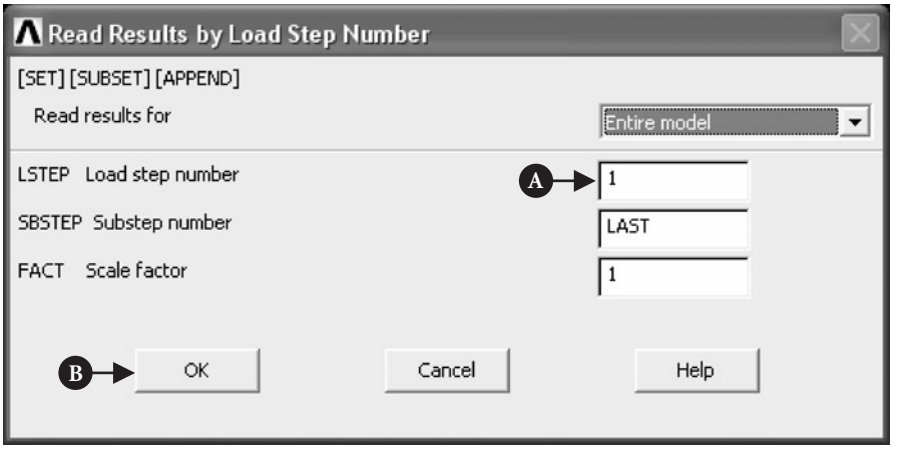


Figure 7.85 Read Results by Load Step Number.

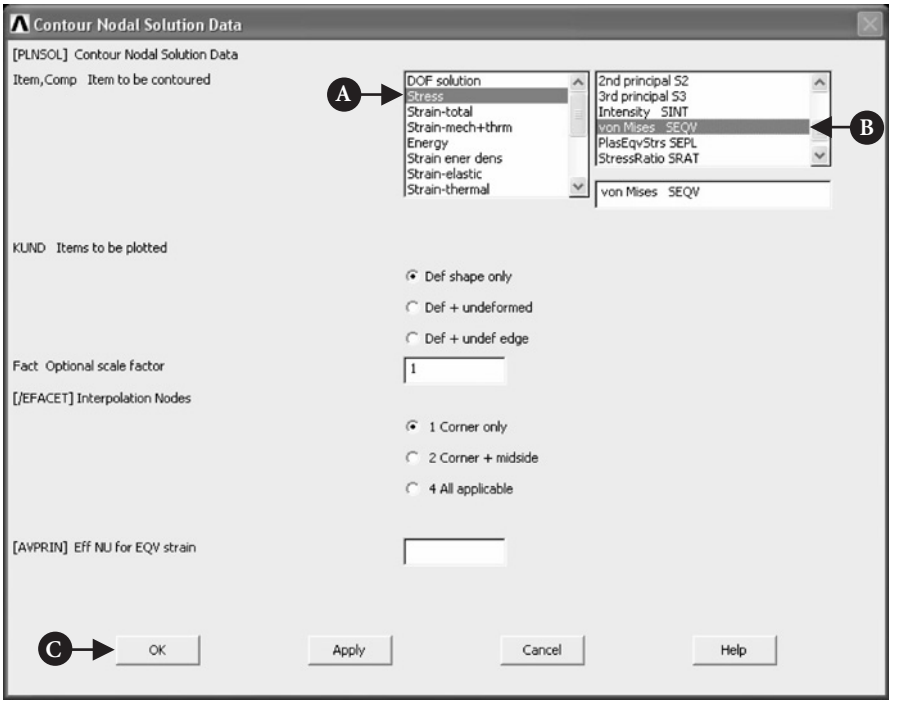


Figure 7.86 Contour Nodal Solution Data.

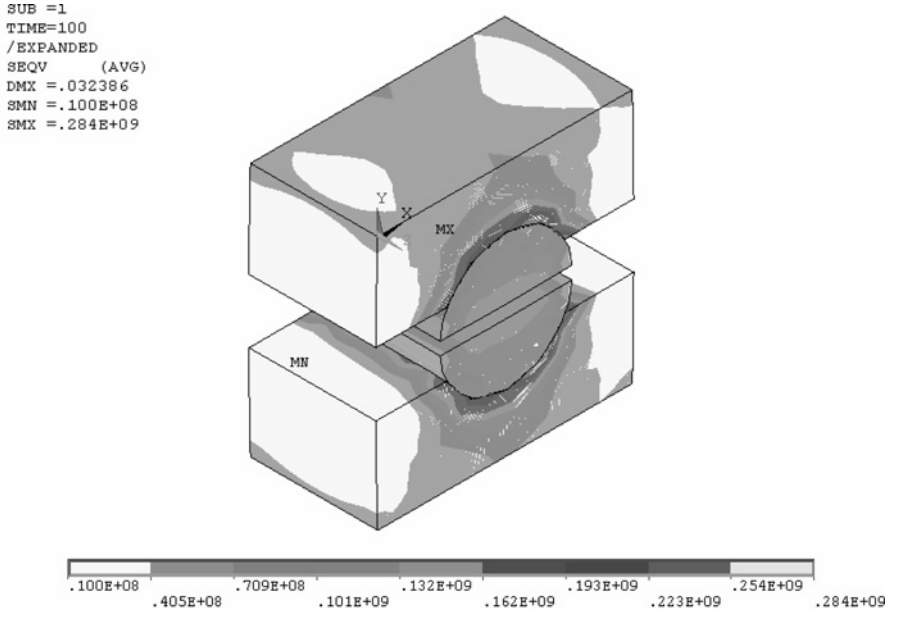


Figure 7.87 Contour plot of nodal solution (von Mises stress).

Figure 7.87 shows von Mises stress contour for the whole assembly. If one is interested in observing contact pressure on the cylinder alone then a different presentation of solution results is required.

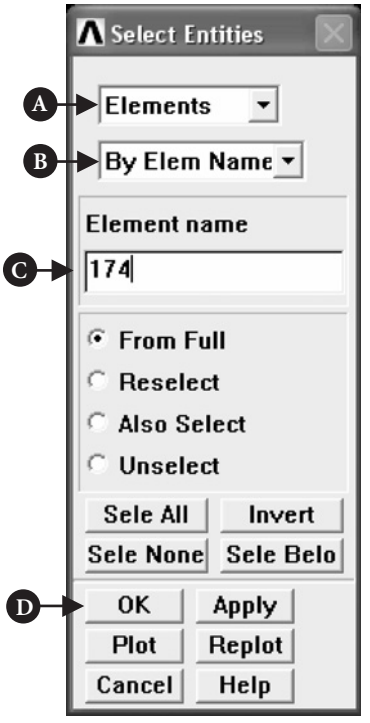


Figure 7.88 Select Entities.

From **Utility Menu** choose **Select → Entities**. The frame shown in Figure 7.88 appears.

In the frame shown in Figure 7.88, the following selections are made: [A] **Elements** (first pull down menu); [B] **By Elem Name** (second pull down menu); and [C] **Element Name =** 174. The element with the number 174 was introduced automatically during the process of creation of contact pairs described earlier. It is listed in the **Preprocessor → Element Type → Add/Edit/Delete** option. Selections are implemented by pressing [D] **OK** button.

From **Utility Menu** select **Plot → Elements**. Image of the cylinder with mesh of elements is produced (see Figure 7.89).

It is seen that the gap equal to 0.05 units exists between two half of the cylinder. It is the result of moving half of the cylinder toward the block (by 0.05 cm) in order to create loading at the interface.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 7.90 appears.

In the frame shown in Figure 7.90, the following selections are made: [A] **Contact** and [B] **Pressure**. These are items to be contoured. Pressing [C] **OK** implements selections made. In response to that an image of the cylinder with pressure contours is produced as shown in Figure 7.91.

7.2.3 WHEEL-ON-RAIL LINE CONTACT

7.2.3.1 PROBLEM DESCRIPTION

Configuration of the contact to be analyzed is shown in Figure 7.92.

This contact problem, which in practice is represented by a wheel-on-rail configuration, is well known in engineering. Also, the characteristic feature of the contact is that, nominally, contact between elements takes place along line. In reality, this is never the case due to unavoidable elastic deformations and surface roughness. As a consequence of that, surface contact is established between elements.

This is a 3D analysis and advantage could be taken of the inherent symmetry of the model. Therefore, the analysis will be carried out on a quarter-symmetry model only. The objective of the analysis is to observe the stresses in the cylinder and the rail when an external load is imposed on them.

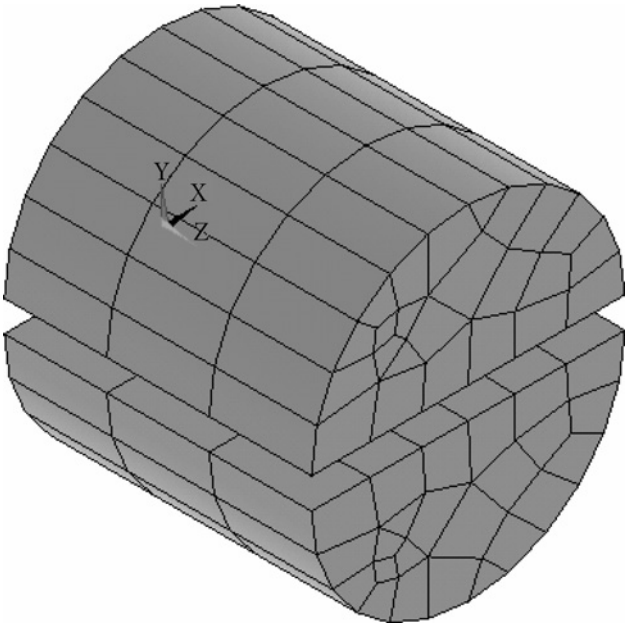


Figure 7.89 Cylinder with surface elements (174).

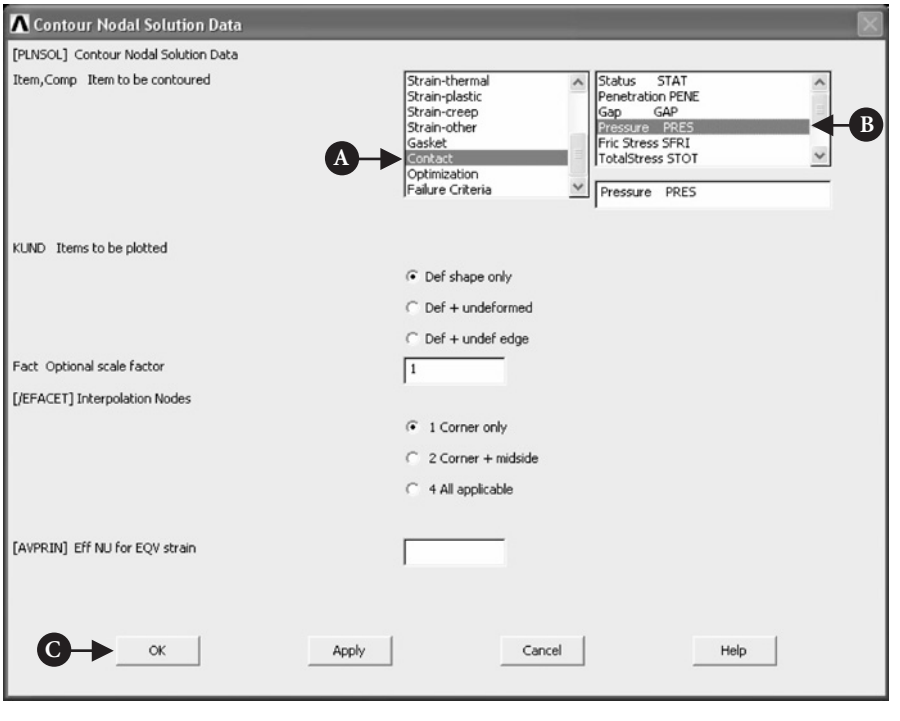


Figure 7.90 Contour Nodal Solution Data.

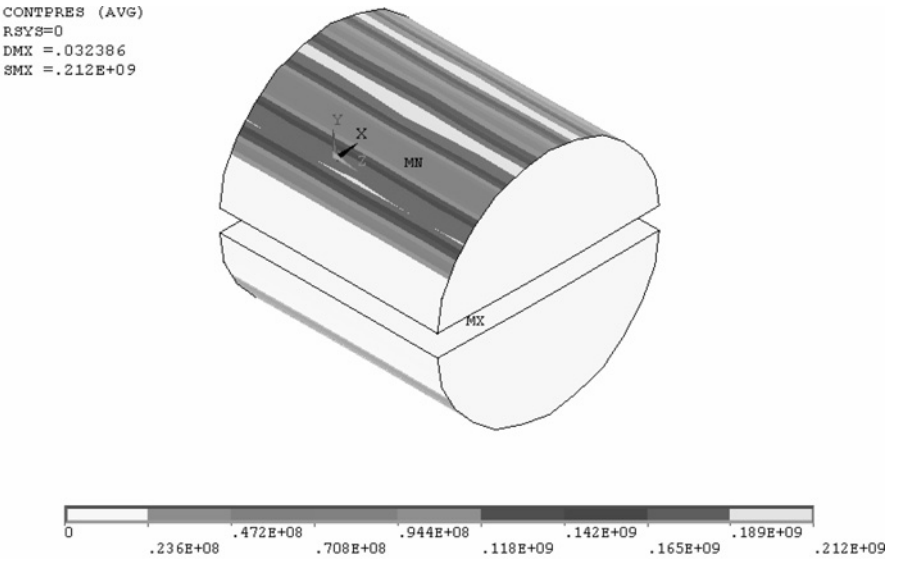


Figure 7.91 Contact pressure contours on the cylinder.

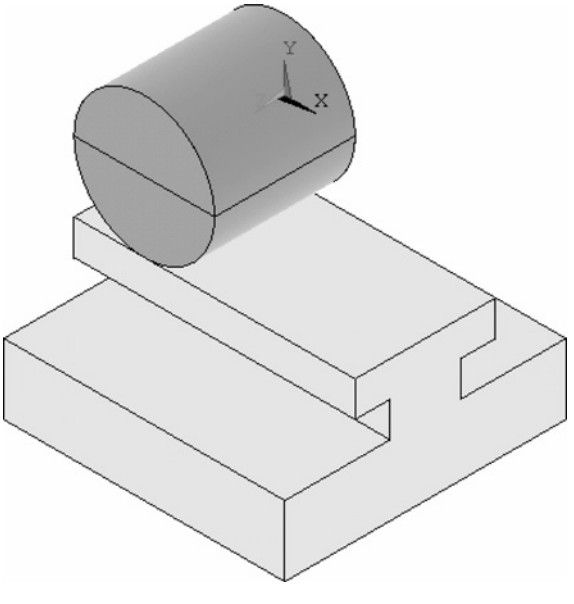


Figure 7.92 Configuration of the contact.

The dimensions of the model are as follows: diameter of the cylinder = 1 cm and cylinder length = 2 cm. Rail dimensions: base width = 4 cm; head width = 2 cm; head thickness = 0.5 cm; and rail height = 2 cm.

Both elements are made of steel with Young's modulus = 2.1×10^9 N/m^2, Poisson's ratio = 0.3 and are assumed elastic. Friction coefficient at the interface between cylinder and the rail is 0.1.

7.2.3.2 Model construction

For the intended analysis a quarter-symmetry model is appropriate. It is shown in Figure 7.93.

The model is constructed using GUI facilities only. First, a two-dimensional (2D) model is created (using rectangles and circle as primitives). This is shown in Figure 7.94.

Next, using "extrude" facility, areas are converted into volumes and a 3D model constructed. When carrying out Boolean operations on areas or volumes, it is convenient to have them numbered. This is done by selecting from the **Utility Menu →**

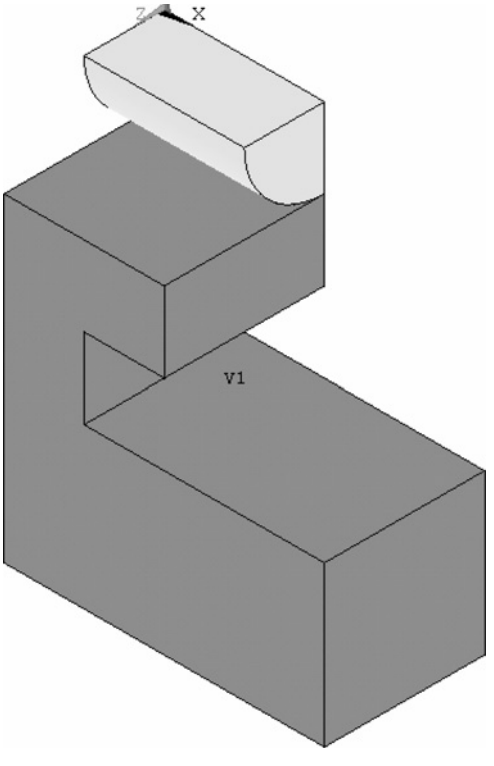


Figure 7.93 A quarter-symmetry model.

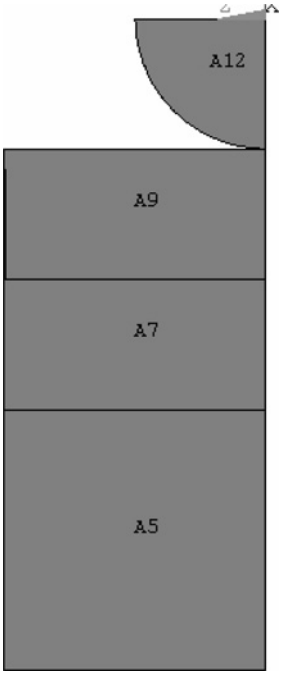


Figure 7.94 2D image (front) of the quarter symmetry model. Involved areas are numbered.

PlotCtrls → Numbering and checking appropriate box to activate **AREA** (area numbers) or **VOLU** (volume numbers) option.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. The frame shown in Figure 7.95 appears.

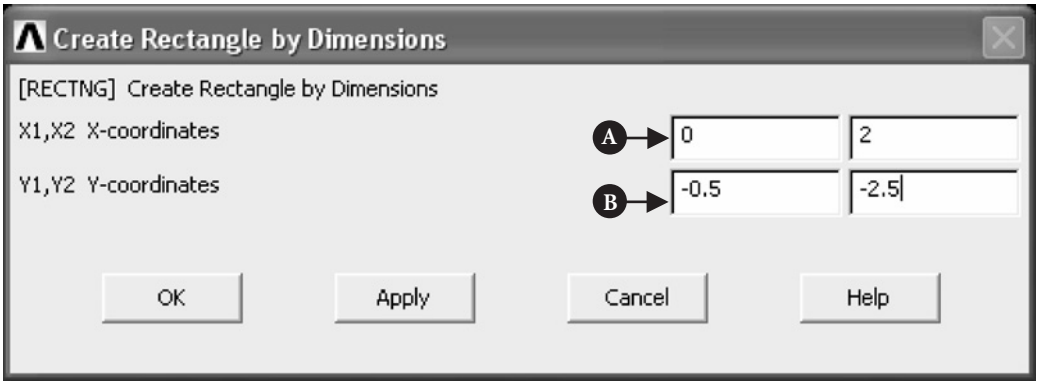


Figure 7.95 Create Rectangle by Dimensions.

Entered coordinates, [A] (X1 = 0; X2 = 2) and [B] (Y1 = −0.5; Y2 = −2.5), are shown in Figure 7.95. This creates area, A1.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Frame with entered coordinates, [A] (X1 = 1; X2 = 2) and [B] (Y1 = −0.5; Y2 = −1.5), is shown in Figure 7.96. This creates area, A2.

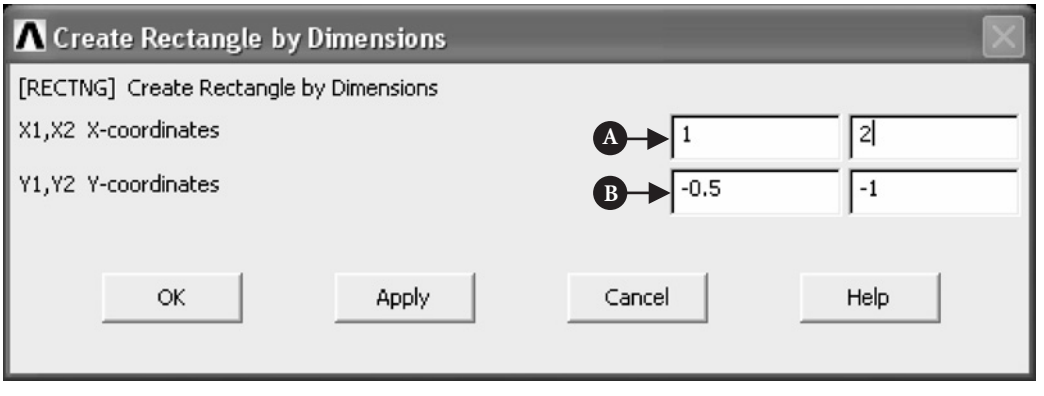


Figure 7.96 Create Rectangle by Dimensions.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Rectangle → By Dimensions**. Frame with entered coordinates, [A] (X1 = 0.5; X2 = 2) and [B] (Y1 = −1; Y2 = −1.5), is shown in Figure 7.97. This creates area, A3.

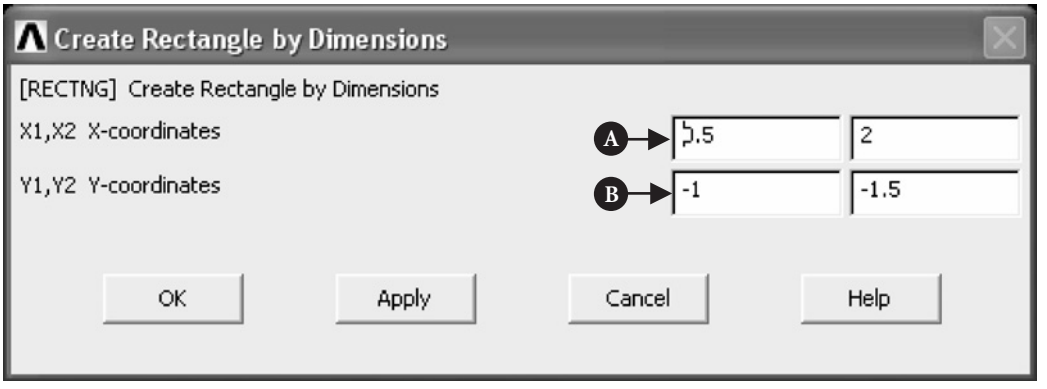


Figure 7.97 Create Rectangle by Dimensions.

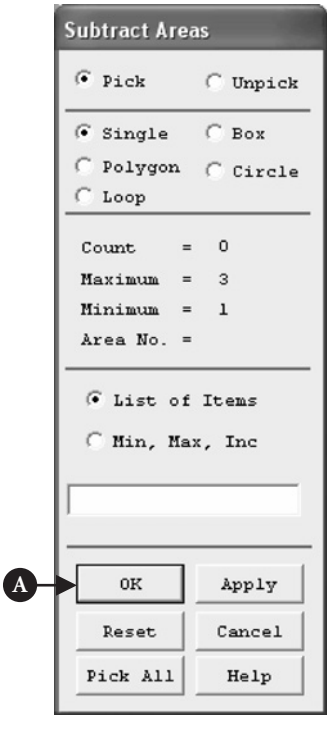


Figure 7.98 Subtract Areas.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Booleans → Subtract → Areas**. Frame shown in Figure 7.98 appears asking for selection of areas to be subtracted.

Select first area, A1, by clicking on it. Then click [A] **OK** button in the frame of Figure 7.98. Next, select area A3, by clicking on it and pressing [A] **OK** button. Area A3 is subtracted from area A1. This operation creates area, A4.

Repeat all the above steps in order to subtract area A3 from area A4. As a result of this operation, a cross-section of half of the rail's area is created. Its assigned number is A1.

Next, a quarter of a circle is created, which will be extruded into a quarter of the cylinder. To do that, it is necessary to offset **WP** (work plane) by 90°, so that the quarter circle will have a required orientation.

From **Utility Menu** select **WorkPlane → Offset WP by Increments**. Figure 7.99 shows the frame resulting from the selection.

It is seen from Figure 7.99 that ZX plane was offset by [A] 90° clockwise (clockwise direction is negative).

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Circle → By Dimensions**. Frame shown in Figure 7.100 appears where appropriate data were entered to create a solid quarter of a circle with [A] outer radius of 0.5 cm, [B] starting angle 270°, and [C] ending angle 360°.

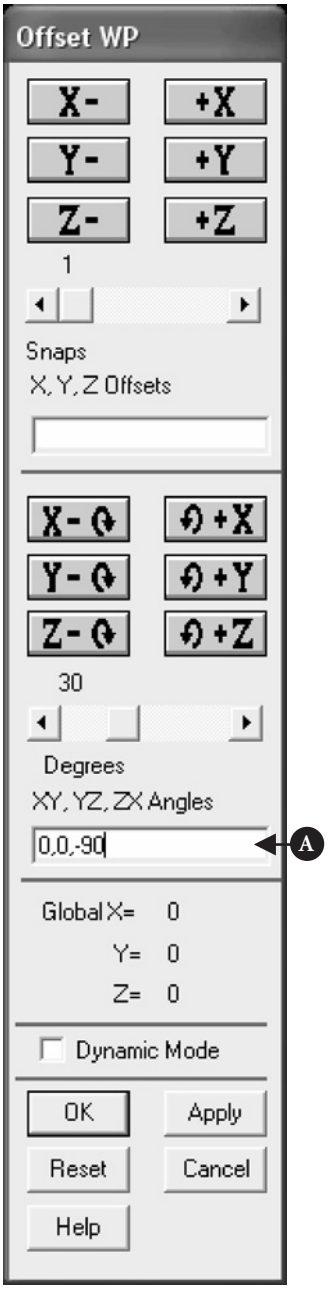


Figure 7.99 Offset WP.

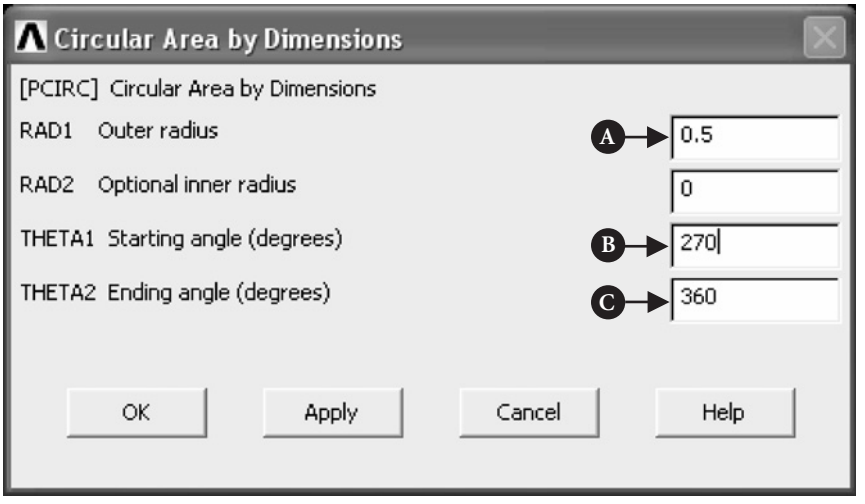


Figure 7.100 Circular Area by Dimensions.

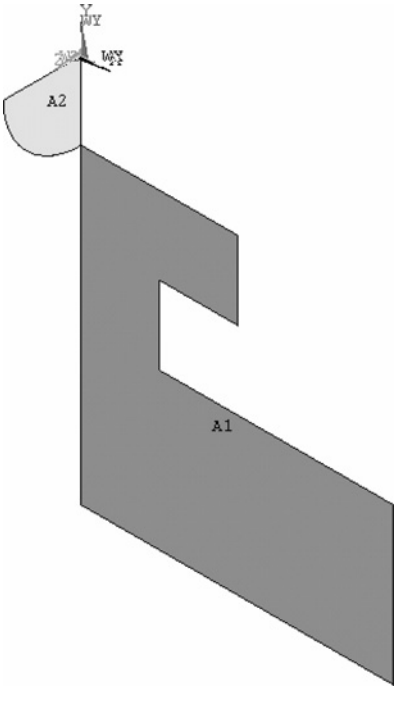


Figure 7.101 Isometric view of the 2D quarter model.

When the solid quarter circular area is created, it is important to restore **WP** offset to its original setting. This can be done by following steps associated with Figure 7.99 and selecting XY, YZ, and ZX angles as 0, 0, and 90, respectively.

In an isometric view, the 2D quarter model is shown in Figure 7.101.

It can be seen that the cross-section of the rail is assigned number A1 and the quarter of the solid circle is given number A2.

The next step is to extrude both areas, A1 and A2, into volumes.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Operate → Extrude → Areas → Along Normal**. The frame shown in Figure 7.102 appears.

Pick area A1 and click [A] **OK** button to pull down another frame shown in Figure 7.103.

Selections made are shown in Figure 7.103. By clicking [A] **OK** button, 2D cross-section of the rail is extruded by [B] 1 cm (length of extrusion) into a volume. Direction of extrusion is normal to the rail's cross-section.

In a similar way, quarter solid circle can be extruded to create a quarter of the cylinder with 1 cm length. Figure 7.104 shows the frame and selections made.

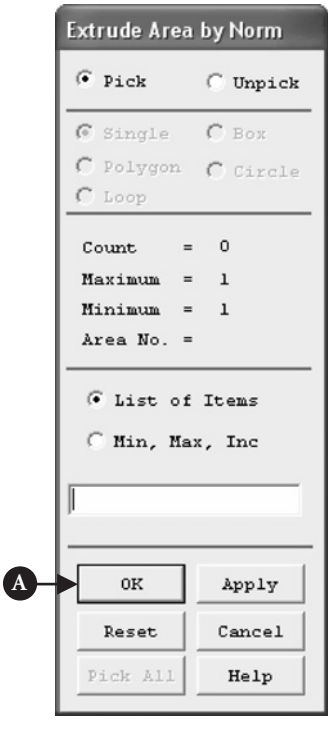


Figure 7.102 Extrude Areas by Norm.

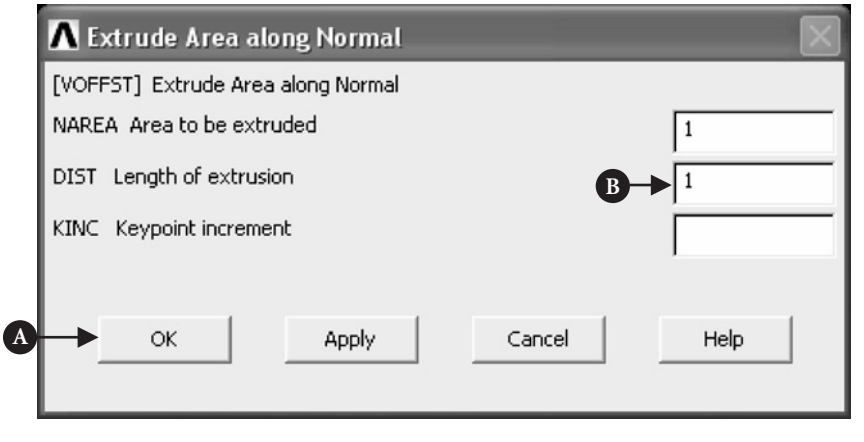


Figure 7.103 Extrude Area along Normal.

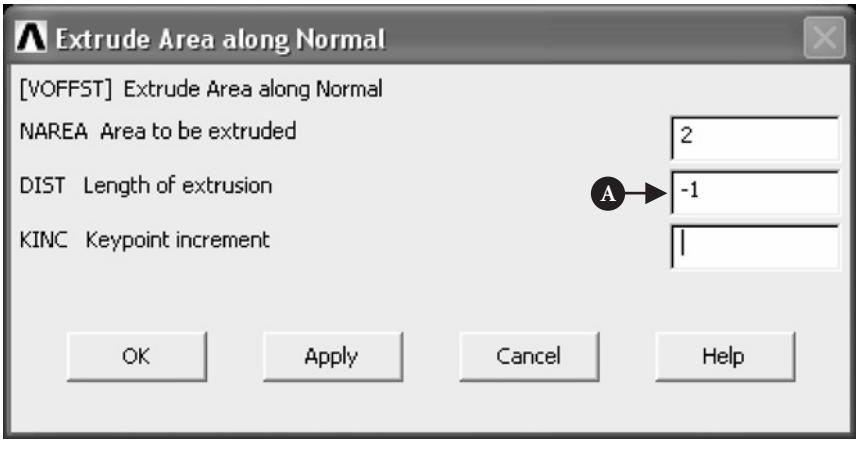


Figure 7.104 Extrude Area along Normal.

It should be noted that in order to have the quarter cylinder oriented as required, [A] length of extrusion = −1 cm should be selected. The minus sign denotes direction of extrusion.

From **Utility Menu** select **PlotCtrls → Numbering** and check in **VOLU** and check out **AREA**. This will change the system of numbering from areas to volumes. Rail is allocated number V1 and cylinder number V2. This completes the construction of a quarter-symmetry model, as shown in Figure 7.93.

7.2.3.3 PROPERTIES OF MATERIAL

Before any analysis is attempted, it is necessary to define properties of the material to be used.

From **ANSYS Main Menu** select **Preferences**. The frame in Figure 7.105 is produced.

As shown in Figure 7.105, [A] **Structural** was the option selected.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models**. Double click **Structural → Linear → Elastic → Isotropic**. The frame shown in Figure 7.106 appears.

Enter [A] **EX** = 2.1×10^9 for Young's modulus and [B] **PRXY** = 0.3 for Poisson's ratio. Then click [C] **OK** button and afterward **Material → Exit**.

After defining properties of the material, the next step is to select element type appropriate for the analysis performed.

From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.107 appears.

Click [A] **Add** in order to pull down another frame, as shown in Figure 7.108.

In the left column click [A] **Structural Solid** and in the right column click [B] **Brick 8node 185**. After that click [C] **OK** and **Close** in order to finish element-type selection.

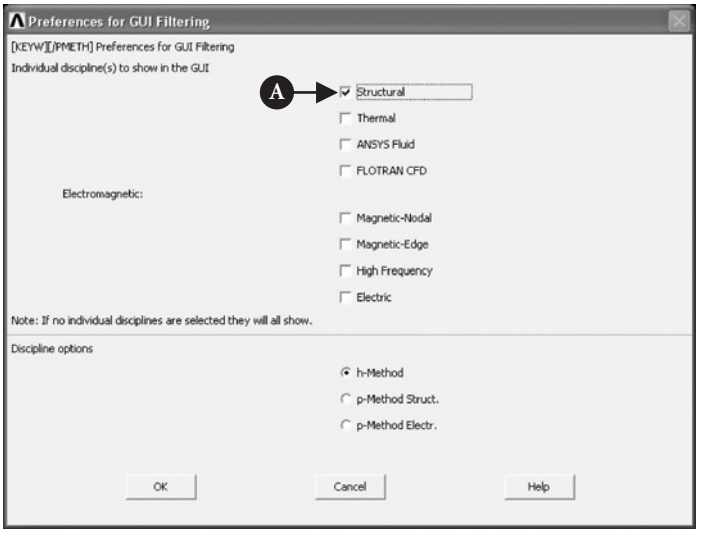


Figure 7.105 Preferences: Structural.

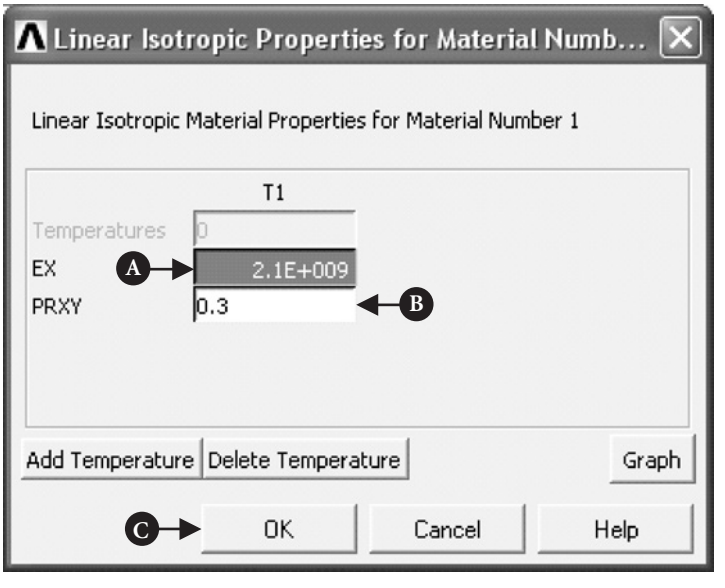


Figure 7.106 Material Properties.

7.2.3.4 MESHING

From **ANSYS Main Menu** select **Preprocessor → Meshing → MeshTool**. Then a frame shown in Figure 7.109 appears.

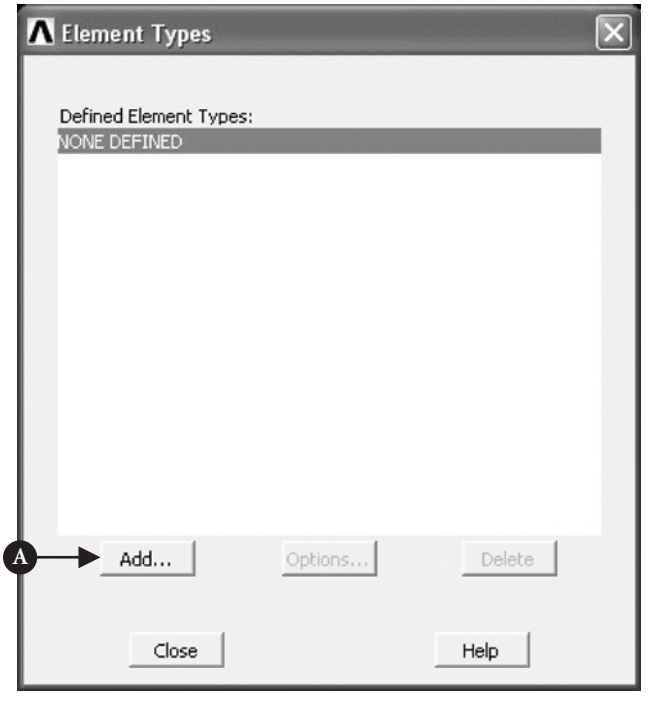


Figure 7.107 Element types.

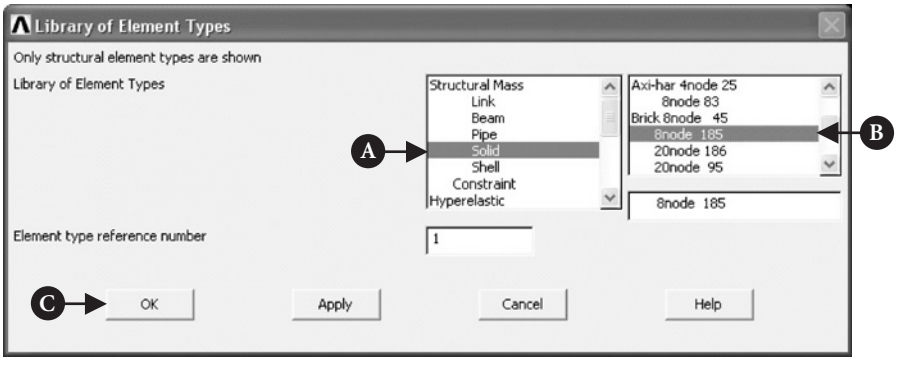


Figure 7.108 Library of Element Types.

There are a number of options available. First step is to go to **Size Control → Lines** option and click [A] **Set** button. This opens another frame (shown in Figure 7.110) prompting to pick lines on which element size is going to be controlled.

Pick two lines, one arcuate and the other horizontal in contact with the top surface of the rail, located on the front side of the cylinder and click [A] **OK**. The frame shown in Figure 7.111 appears.

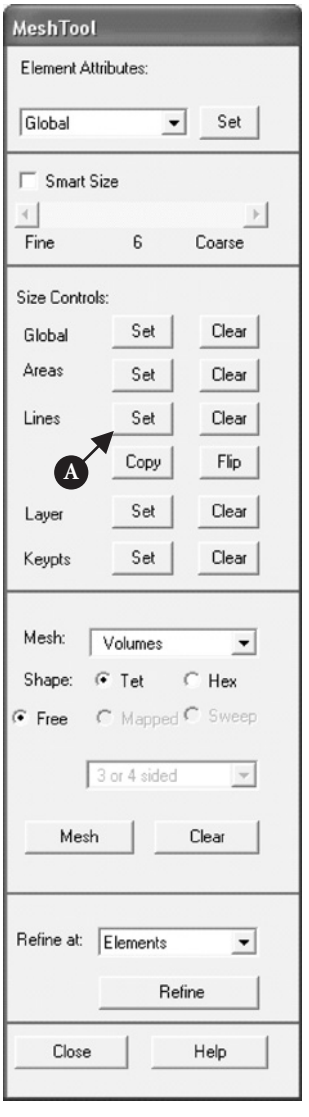


Figure 7.109 MeshTool.

In the box [A] **No. of element divisions** type 30 and change selection [B] **SIZE, NDIV can be changed** to **No.** by checking the box out and then click [C] **OK**. Both selections are shown in Figure 7.111.

Similarly, using **MeshTool** frame (see Figure 7.109) click [A] **Set** in the **Size Controls: Lines** option and pick two lines located on top surface of the rail: one coinciding with the line previously picked and the other at the right angle to the first one. Click [A] **OK** as shown in Figure 7.110. The frame shown in Figure 7.111 appears again. In the box [A] **No. of element divisions** type 30 and press [C] **OK** button.

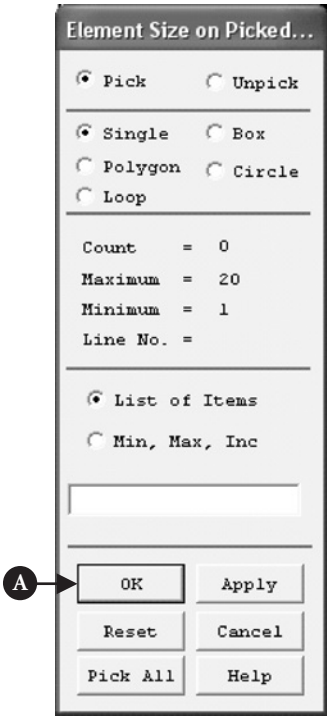


Figure 7.110 Element Size on Picked Lines.

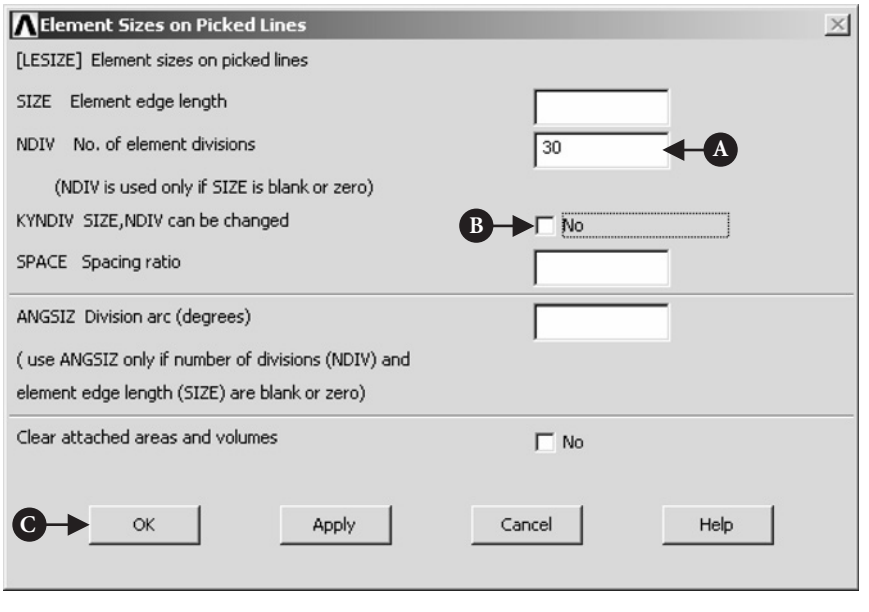


Figure 7.111 Element Sizes.

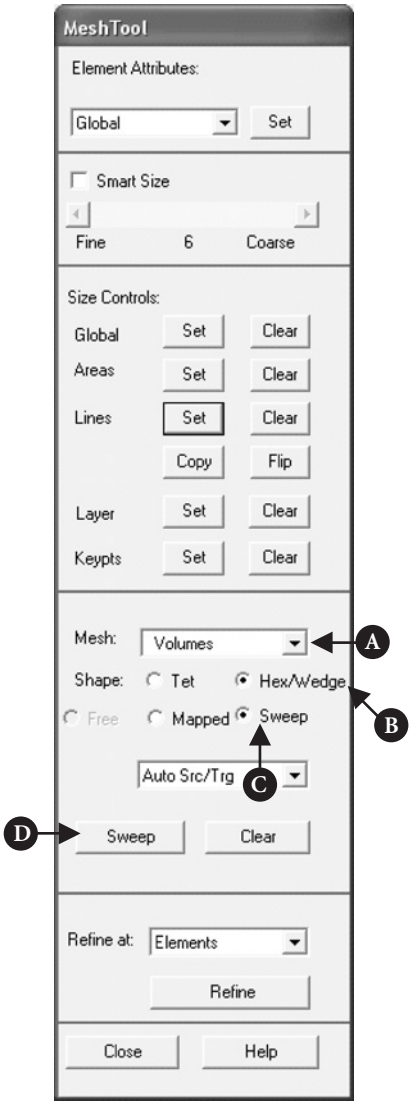


Figure 7.112 MeshTool (checked Hex/Wedge and Sweep options).

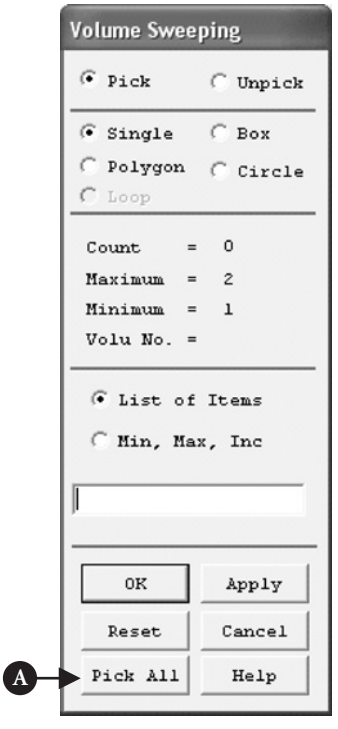


Figure 7.113 Volume Sweeping.

In the frame **MeshTool** (see Figure 7.112) pull down [A] **Volumes** in the option **Mesh**. Check [B] **Hex** and [C] **Sweep** options.

Pressing [D] **Sweep** button brings another frame, as shown in Figure 7.113, asking to pick volumes to be swept.

Pressing [A] **Pick All** button initiates meshing process. The model after meshing looks like the image in Figure 7.114.

Pressing **Close** button on **MeshTool** frame, ends mesh generation stage.

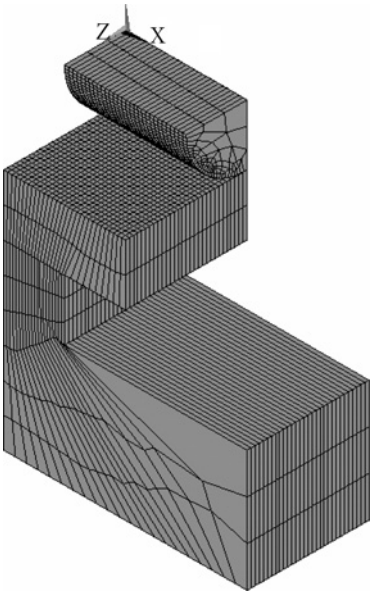


Figure 7.114 Model after meshing process.

After meshing being completed, it is usually necessary to smooth element edges in order to improve graphic display. It can be accomplished using **PlotCtrls** facility in the Utility Menu.

From **Utility Menu** select **PlotCtrls → Style → Size and Shape**. The frame shown in Figure 7.115 appears.

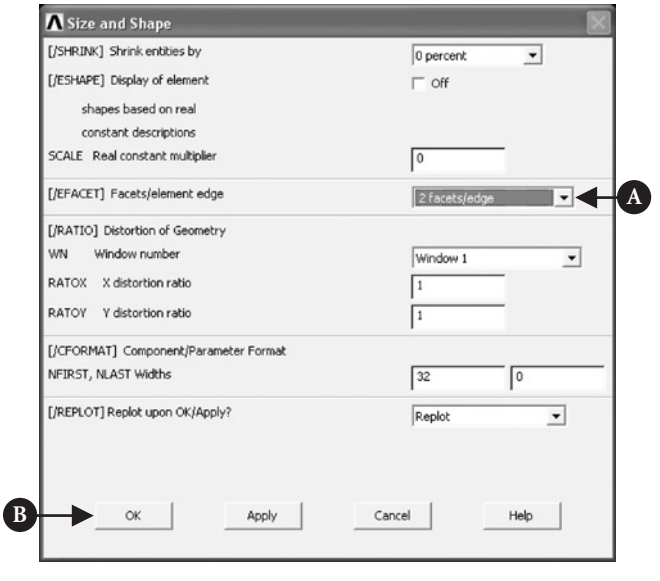


Figure 7.115 Size and Shape.

In the option [A] **Facets/element edge** select **2 facets/edge** and click [B] **OK** button as shown in Figure 7.115.

7.2.3.5 CREATION OF CONTACT PAIR

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Contact Pair**. As a result of this selection, a frame shown in Figure 7.116 appears.

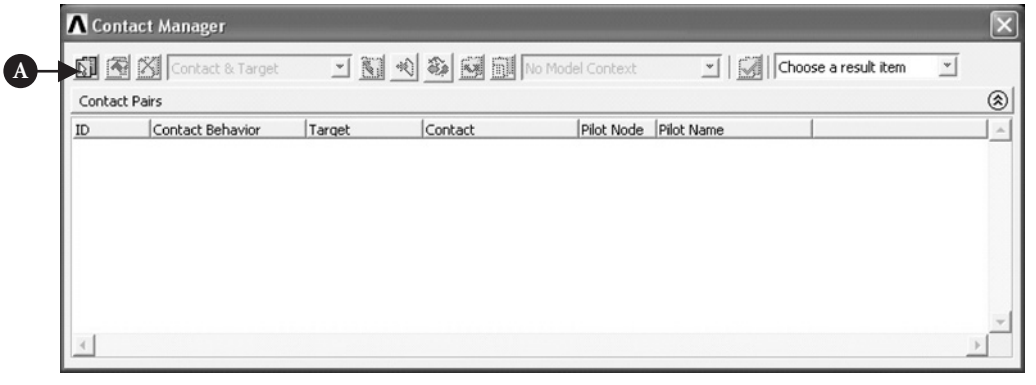


Figure 7.116 Contact Manager.

Upper left-hand corner of the frame contains [A] Contact Wizard button. By clicking on this button a Contact Wizard frame, as shown in Figure 7.117, is produced.

In the frame shown in Figure 7.117 select: [A] **Areas**, [B] **Flexible**, and press [C] **Pick Target**. The frame shown in Figure 7.118 is produced.

Select top area of the rail by picking on it and pressing [A] **OK** button in the frame of Figure 7.118. Again Contact Wizard frame appears and this time **Next** button should be pressed to obtain the frame shown in Figure 7.119.

Press [A] **Pick Contact** button to create the frame in Figure 7.120.

Select curved surface of the cylinder and press [A] **OK** button. Contact Wizard frame appears again and **Next** button should be pressed to obtain the frame shown in Figure 7.121.

In the [A] **Optional Settings** frame select: set **Coefficient of Friction** = 0.1 and check out box – **Include initial penetration**. Next, press [A] **Optional Settings** button to call up another frame. In the new frame, **Normal penalty stiffness = 0.1** should be selected. Also, **Friction** tab located in the top of the frame menu should be activated and **Stiffness matrix = Unsymmetric** selected. Tab **Initial Adjustment,** also located at the top of the menu, should be pressed and in the box [A] **Automatic Contact Adjustment** selection **Close gap** should be made as shown in Figure 7.122.

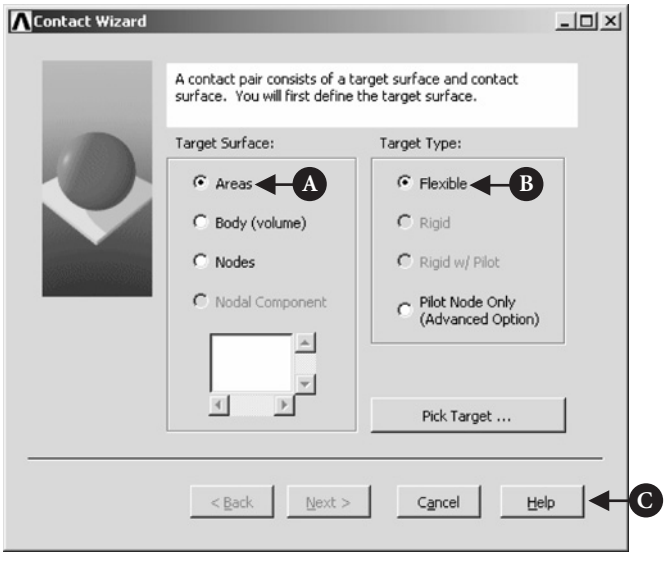


Figure 7.117 Contact Wizard.

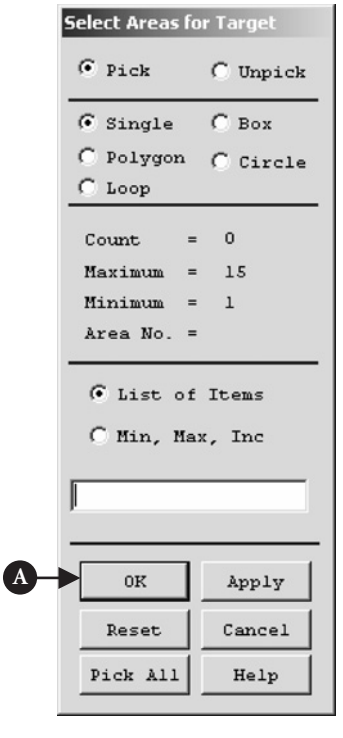


Figure 7.118 Select Areas for Target.

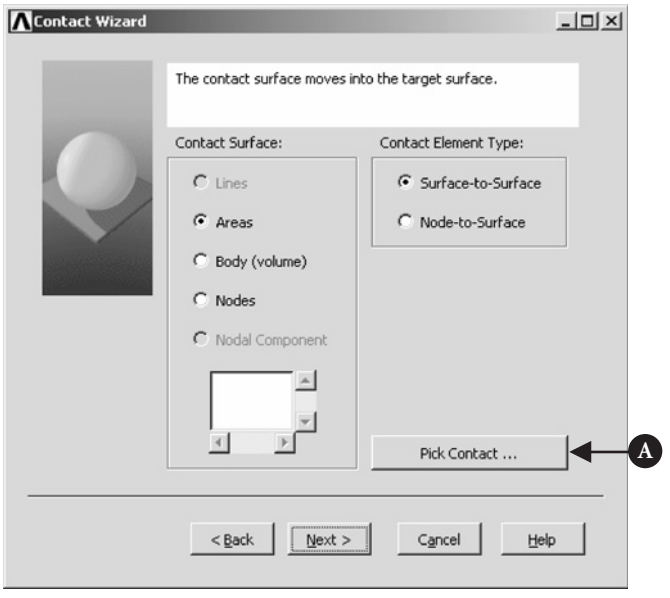


Figure 7.119 Contact Wizard (contact).

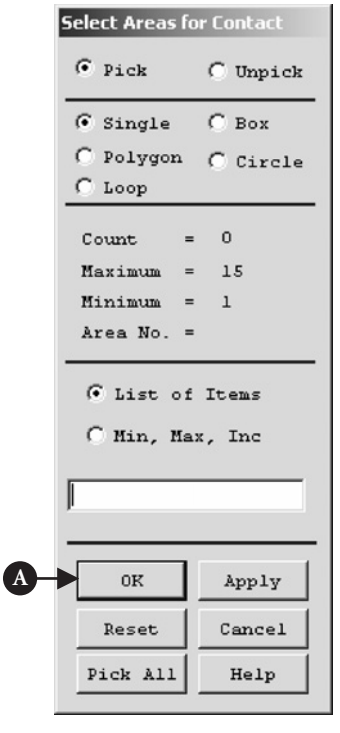


Figure 7.120 Select Areas for Contact.

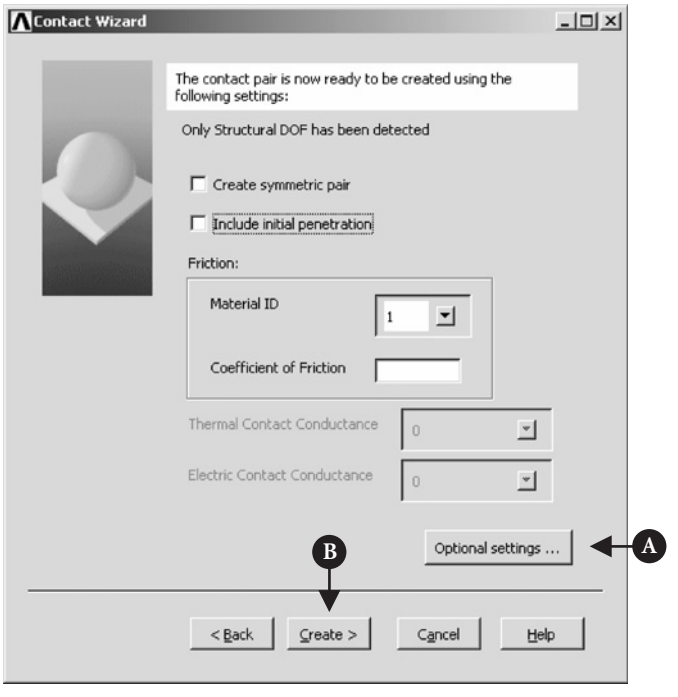


Figure 7.121 Contact Wizard (optional settings).

This will prevent solution crushing due to the gap between target and contact surfaces greater than allowed by the program.

Pressing [B] **OK** button brings back Contact Wizard frame (see Figure 7.121), where the [B] **Create** button should be pressed. Created contact pair is shown in Figure 7.123.

Finally, Contact Wizard frame should be closed by pressing **Finish** button. Also, Contact Manager summary information frame should be closed.

7.2.3.6 SOLUTION

Before the solution can be attempted, solution criteria have to be specified. As a first step in that process, symmetry constraints are applied on the quarter-symmetry model.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → Symmetry BC → On Areas**. The frame shown in Figure 7.124 appears.

Four surfaces at the back of the quarter-symmetry model should be selected and button [A] **OK** clicked. Symmetry constraints applied to the model are shown in Figure 7.125.

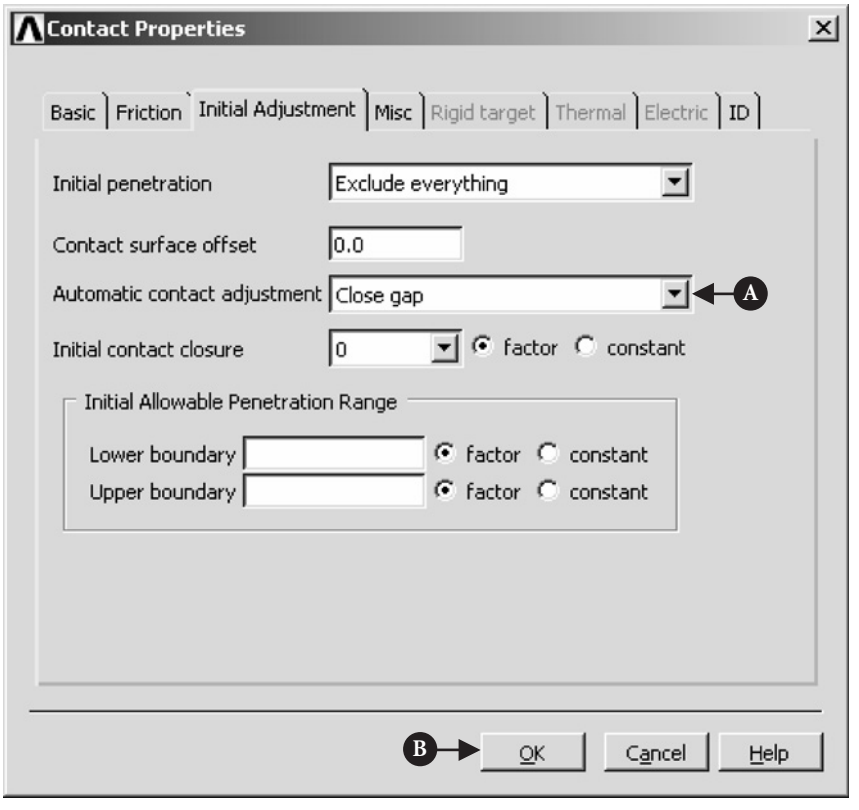


Figure 7.122 Contact Properties (optional settings).

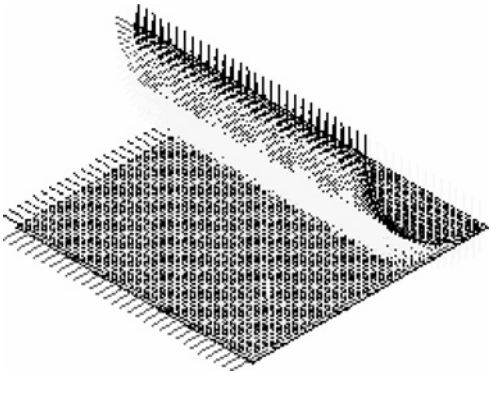


Figure 7.123 Contact pair (surface).

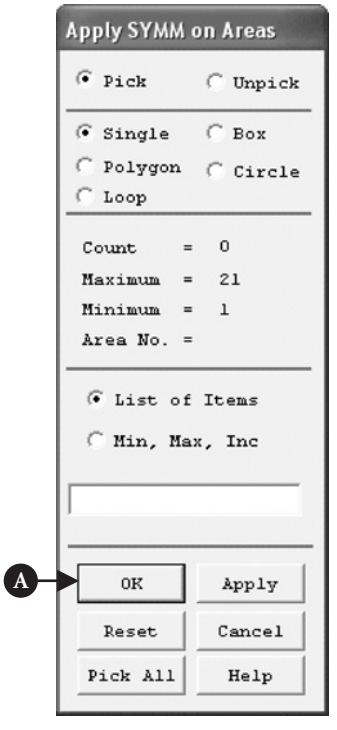


Figure 7.124 Apply SYMM on Areas.

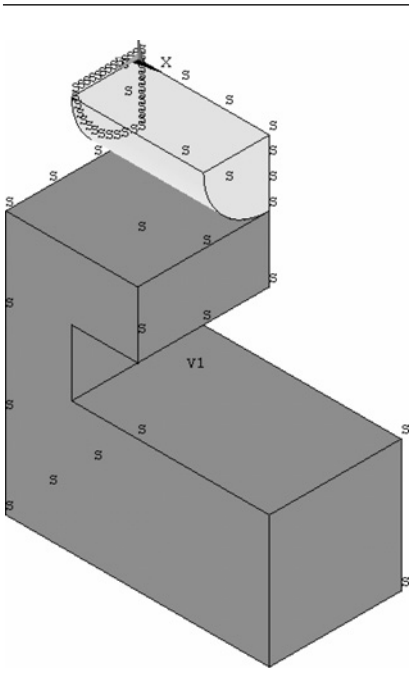


Figure 7.125 Symmetry constraints applied to the model.

The next step is to apply constraints on the bottom surface of the block. From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Areas**. The frame shown in Figure 7.126 appears.

The bottom surface of the rail should be selected. After selecting required surface and pressing [A] **OK** button, another frame appears in which the following should be selected: **DOFs to be constrained =** All DOF and **Displacement value =** 0. Selections are implemented by pressing **OK** button in the frame.

The final action is to apply external loads. In the case considered here a pressure acting on the top surface of the cylinder will be used.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Pressure → On Areas**. The frame shown in Figure 7.127 appears.

Top surface of the cylinder should be selected and [A] **OK** button pressed to pull down another frame, as shown in Figure 7.128.

It is seen from Figure 7.128 that the [A] constant pressure of 0.5 MPa was applied to the selected surface. Figure 7.129 shows the model ready for solution with constraints and applied load.

Now the modeling stage is completed and the solution can be attempted. From **ANSYS Main Menu** select **Solution → Solve → Current LS**. A frame showing the

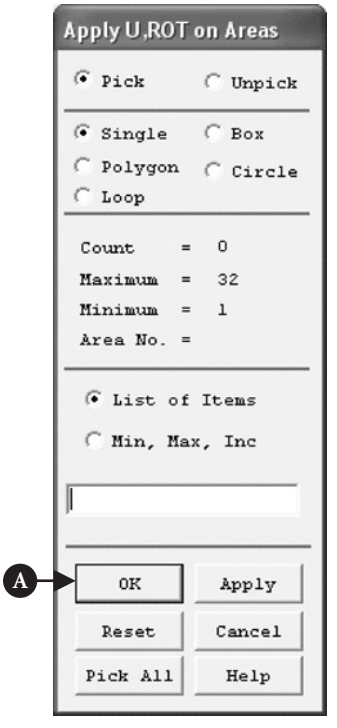


Figure 7.126 Apply U,ROT on Areas.

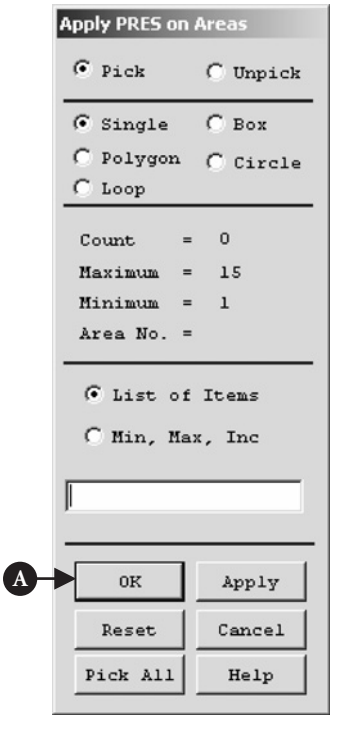


Figure 7.127 Selection on area.

review of information pertaining to the planned solution action appears. After checking that everything is correct, select **File → Close** to close that frame. Pressing **OK** button starts the solution. When the solution is completed, press **Close** button.

In order to return to the previous image of the model, select **Utility Menu → Plot → Replot**.

7.2.3.7 POSTPROCESSING

In order to display solution results in a variety of forms, postprocessing facility is used. In the example solved here, there is no need to expand the quarter-symmetry model into half-symmetry or full model because the contact stresses are best observed from a quarter-symmetry model. Furthermore, the isometric viewing direction so far used should be changed in the following way. From **Utility Manu** select **PlotCtrls → View Settings → Viewing Direction**. In the resulting frame, as shown in Figure 7.130, set **View Direction**: [A] XV = −1, [B] YV = 1, [C] ZV = −1, and click [D] **OK** button.

Quarter-symmetry model in selected viewing direction is shown in Figure 7.131.

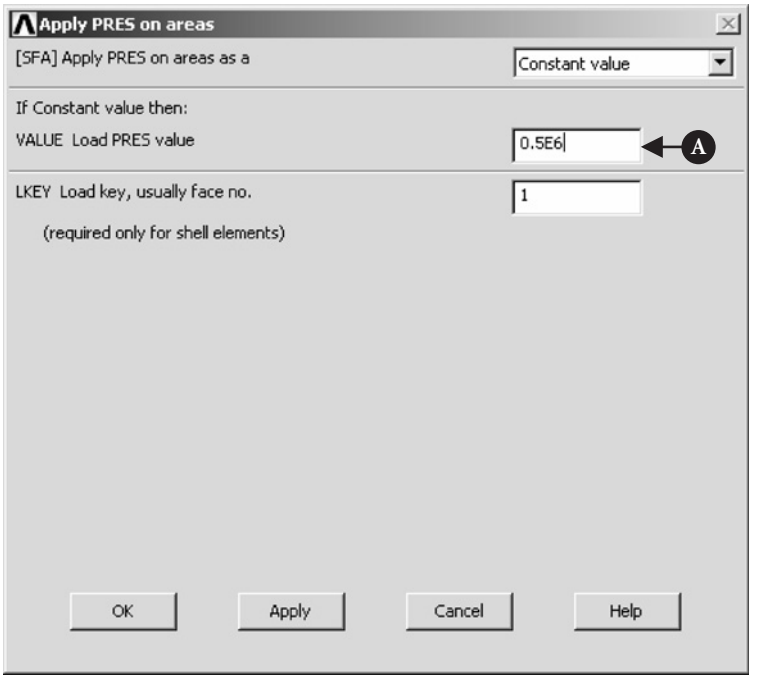


Figure 7.128 Apply PRES on areas (magnitude).

From **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. The frame shown in Figure 7.132 is produced. The selection [A] **Load step number =** 1, as shown in Figure 7.132, is implemented by clicking [B] **OK**.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. In the resulting frame the following selections are made: [A] **Item to be contoured = Stress** and [B] **Item to be contoured = von Mises** (SEQV) (see Figure 7.133). Pressing [C] **OK** implements selections.

Contour plot of von Mises stress (nodal solution) is shown in Figure 7.134.

Figure 7.134 shows von Mises stress contour for both the rail and cylinder. If one is interested in observing contact pressure on the cylinder surface alone then a different presentation of solution results is required.

From **Utility Menu** choose **Select → Entities**. The frame shown in Figure 7.135 appears.

In the frame shown in Figure 7.135, the following selections are made: [A] **Elements** (first pull down menu); [B] **By Elem Name** (second pull down menu); and [C] **Element Name** = 174. The element with the number 174 was introduced automatically during the process of creation of contact pairs described earlier. It is listed in the **Preprocessor → Element Type → Add/Edit/Delete** option. Pressing **OK** button in the frame shown in Figure 7.135 implements the selections made.

From **Utility Menu** select **Plot → Elements**. Image of the cylinder with mesh of elements is produced (see Figure 7.136).

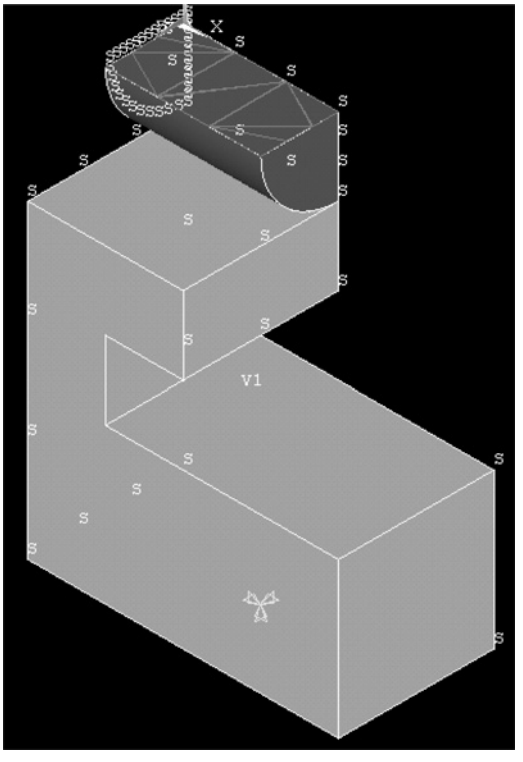


Figure 7.129 Constraints and loads applied to the model.

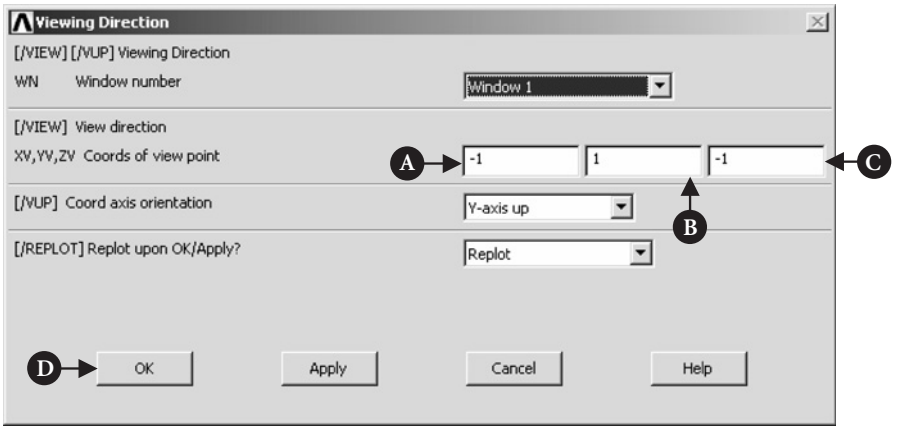


Figure 7.130 Viewing Direction.

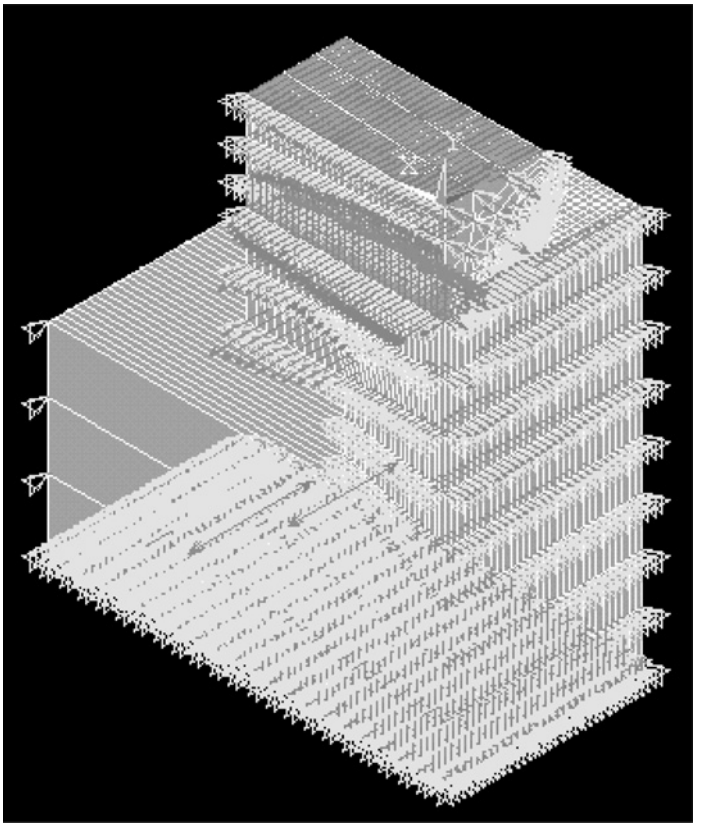


Figure 7.131 Quarter symmetry model with elements, constraints, and loads.

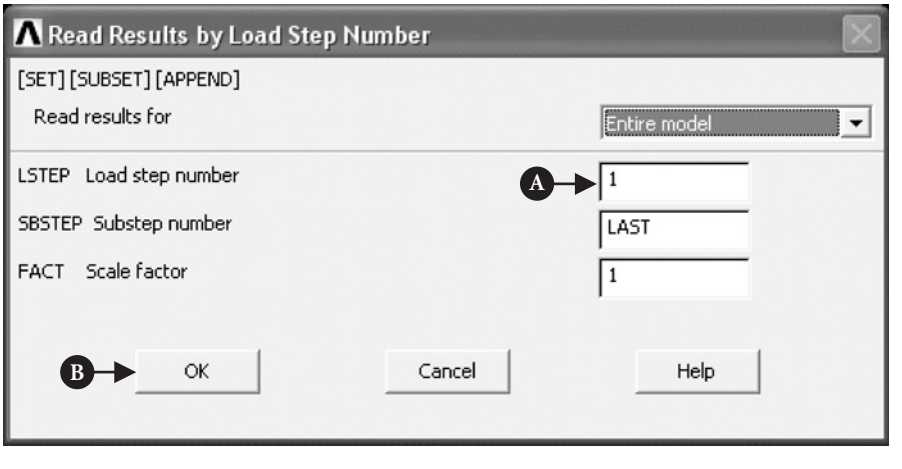


Figure 7.132 Read Results by Load Step Number.

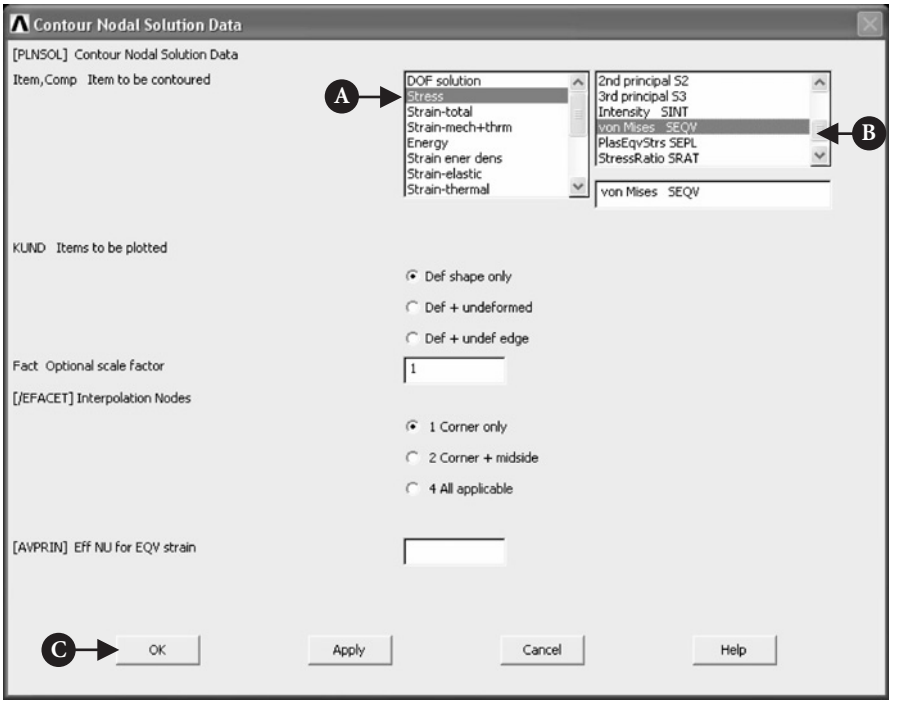


Figure 7.133 Contour Nodal Solution Data.

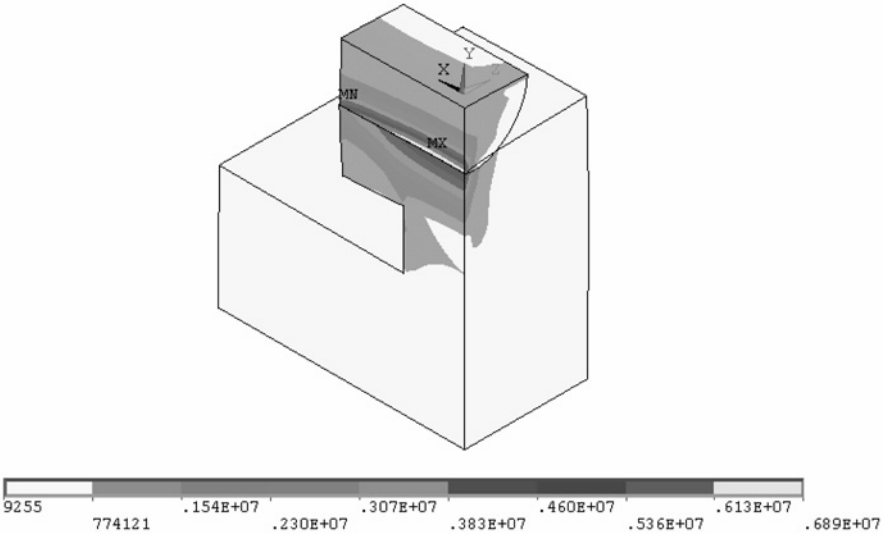


Figure 7.134 Contour plot of nodal solution (von Mises stress).

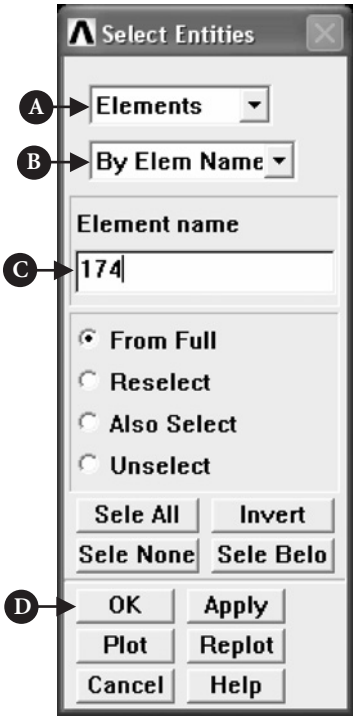


Figure 7.135 Select Entities.

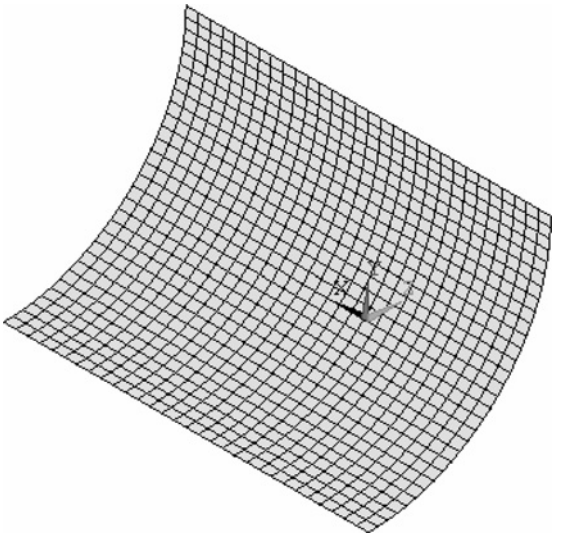


Figure 7.136 Surface of the cylinder with contact elements.

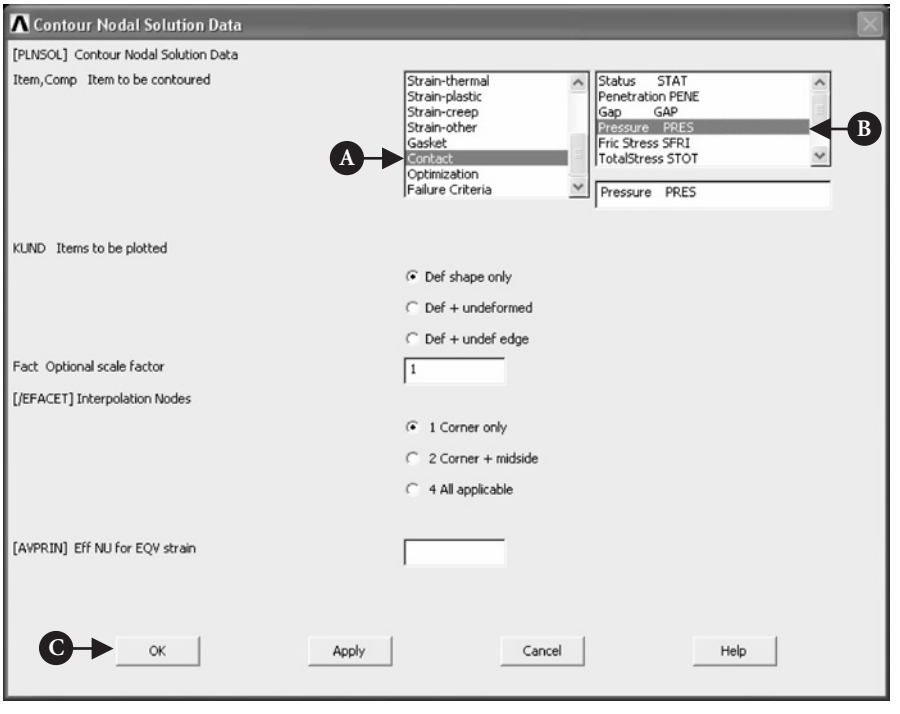


Figure 7.137 Contour Nodal Solution Data.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 7.137 appears.

In the frame shown in Figure 7.137 the following selections are made: [A] **Contact** and [B] **Pressure**. These are items to be contoured. Pressing [C] **OK** implements selections made. In response to this, an image of the cylinder surface with pressure contours is produced as shown in Figure 7.138.

7.2.4 O-RING ASSEMBLY

7.2.4.1 PROBLEM DESCRIPTION

Configuration of the contact between an O-ring made of rubber (hyper-elastic material) and the groove is shown in Figure 7.139.

An O-ring of solid circular cross-section is forced to conform to the shape of a rectangular groove by a moving wall as shown, schematically, in Figure 7.139.

Following the initial squeeze of the O-ring (through movement of the wall), pressure is applied to the surface of the O-ring. Because of the sealing provided by the intrusion of the side walls, the pressure is only effective over less than 180° of the

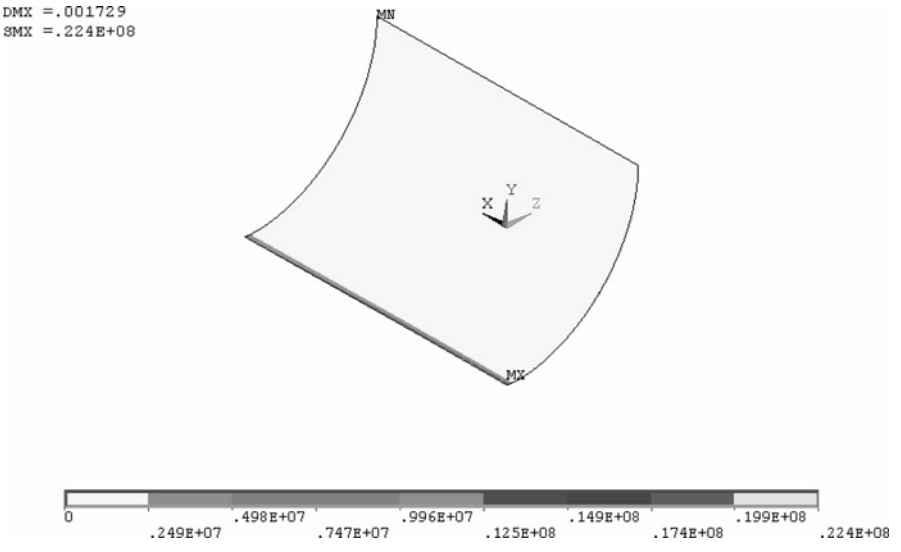


Figure 7.138 Contact pressure on the cylinder surface.

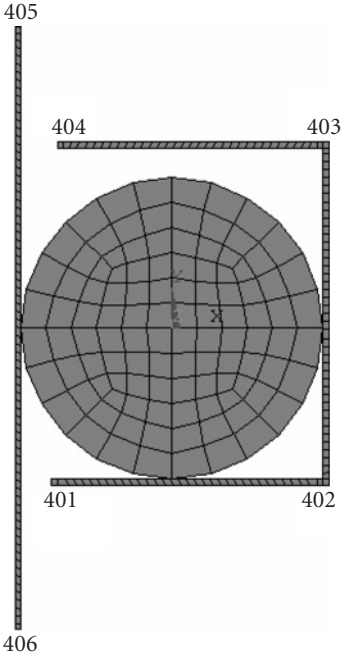


Figure 7.139 Configuration of the contact between an O-ring and the groove.

O-ring top surface. It is required to observe the conformity of the O-ring with the groove walls and stresses created by the pressure acting over its top surface.

The contact is characterized by the following data:

Young's modulus of the wall material $= 2.1 \times 10^9$ Pa.

Surface pressure applied to the O-ring $= 0.1 \times 10^6$ Pa.

Material of the O-ring is modeled as hyper-elastic material of Mooney–Rivlin type with constants: $C_1 = 0.01044$ and $C_2 = 0.1416$.

Poisson's ratio for O-ring $= 0.499$.

Coefficient of friction between O-ring and wall $= 0.1$.

Normal contact stiffness $= 5 \times 10^3$ N/m.

Wall movement $= 0.1$ mm.

Radius of the O-ring $= 2.5$ mm.

Depth of the groove $= 4.5$ mm.

Width of the groove $= 5.5$ mm.

Length of the wall $= 10$ mm.

7.2.4.2 MODEL CONSTRUCTION

The O-ring is constructed using a hyper-elastic element (Mooney–Rivlin), and the groove and movable wall, both considered to be rigid, are constructed using 2D (link or spar) elements. However, spars are used only for contact element generation and not for any structural rigidity of their own. The contact elements are constructed using 2D node-to-surface approach. The loads are applied by wall motion and groove cavity pressurization. The pressure sealing on the O-ring is assumed to take place at 15° off horizontal. The model is constructed using GUI facilities only.

From **ANSYS Main Menu** select **Preferences** and check **Structural** option. Next, elements to be used in the analysis are selected.

From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.140 appears.

Pressing [A] **Add** button calls another frame shown in Figure 7.141.

Select [A] **Mooney-Rivlin** and [B] **2D 8node U-P 74** element and click [C] **OK** button. This creates a frame shown in Figure 7.142, where Type 1 element, **HYPER74**, is shown.

Next, contact element type should be selected.

From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.142 appears. Click [A] **Add** and make selection of the element as shown in Figure 7.143.

Selections [A] **Contact** and [B] **2D pt-to-surf 48** were made and are shown in Figure 7.143.

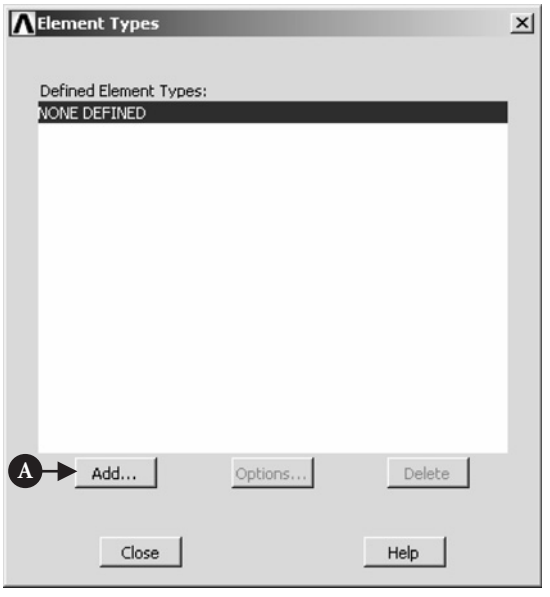


Figure 7.140 Element Types to be selected.

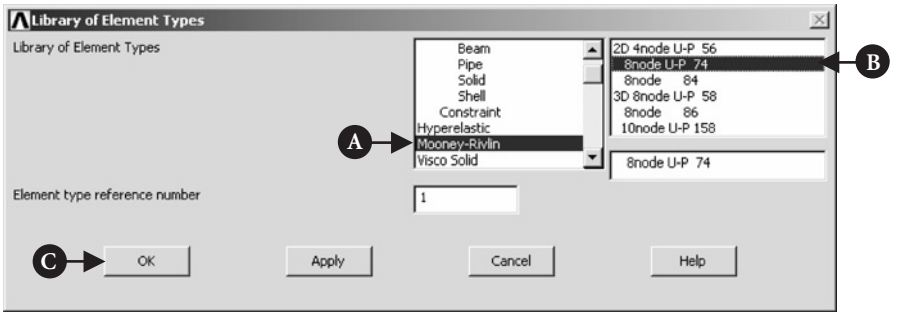


Figure 7.141 Library of Element Types.

The last element type to be selected is the link element. Link element represents in the analysis the movable wall and the groove. From **ANSYS Main Menu** select **Preprocessor → Element Type → Add/Edit/Delete**. The frame shown in Figure 7.144 appears.

Click [A] **Add** button and select element type as shown in Figure 7.145.

Selections [A] **Link** and [B] **2D spar 1** were made as shown in Figure 7.145.

7.2.4.3 SELECTION OF MATERIALS

The next step is to establish database for materials used.

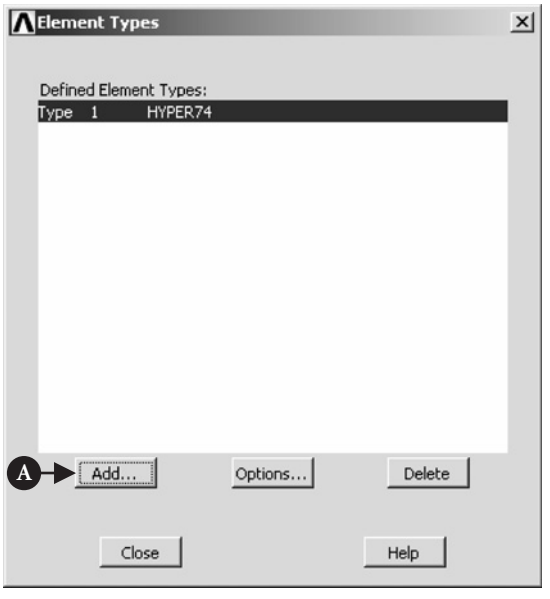


Figure 7.142 Defined Element Types – HYPER74.

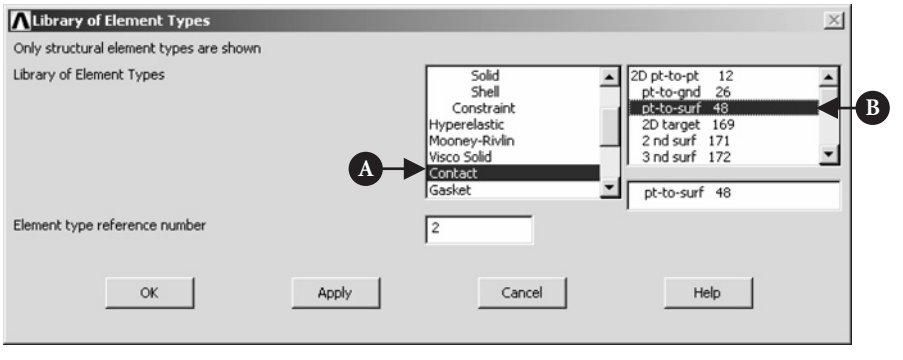


Figure 7.143 Library of Element Types.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models → Structural → Nonlinear → Elastic → Hyperelastic → Mooney-Rivlin** (**TB**, **MOON**). As a result of the selection, a frame shown in Figure 7.146 appears.

Next, double click on [A] **Mooney-Rivlin** (TB, MOON) to call up frame shown in Figure 7.147.

Values for **C1** and **C2** coefficients should be entered and [A] **OK** clicked as shown in Figure 7.147.

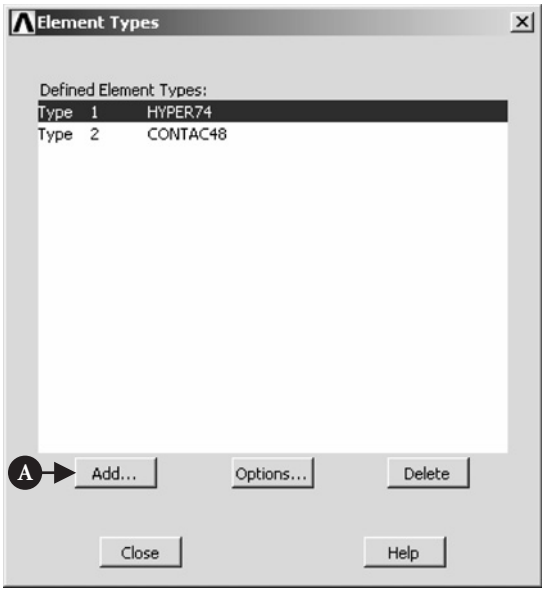


Figure 7.144 Element Types to be defined.

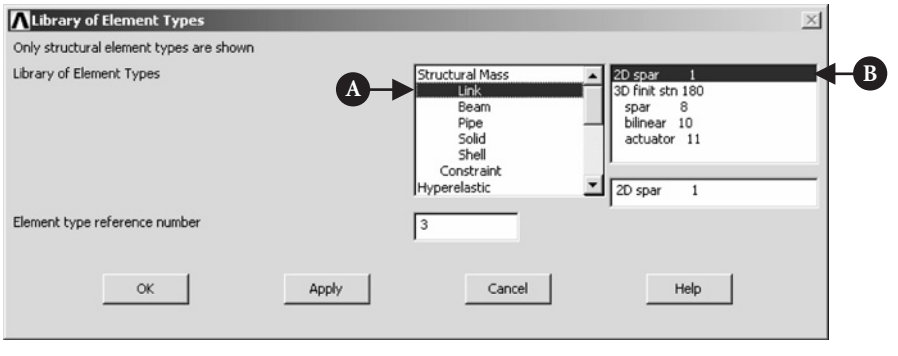


Figure 7.145 Library of Element Types.

From **ANSYS Main Menu** select **Preprocessor → Material Props → Material Models → Structural → Linear → Elastic → Isotropic**. The frame shown in Figure 7.148 appears. Poisson's ratio for O-ring should be entered, **PRXY** = 0.499 as shown in Figure 7.148.

By clicking [A] **OK**, the frame shown in Figure 7.149 appears. Material model number 1 characterizes the behavior of the O-ring component of the contact assembly.

Select [A] **Material** from the menu at the top of the frame, as shown in Figure 7.149, and click **New Model** option. In response, a frame of Figure 7.150 appears.

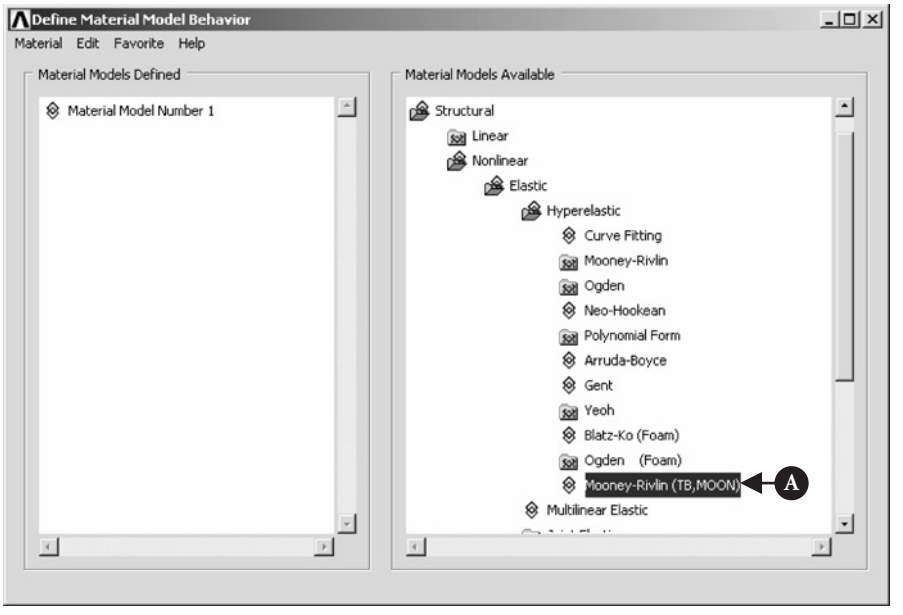


Figure 7.146 Define Material Model Behavior.

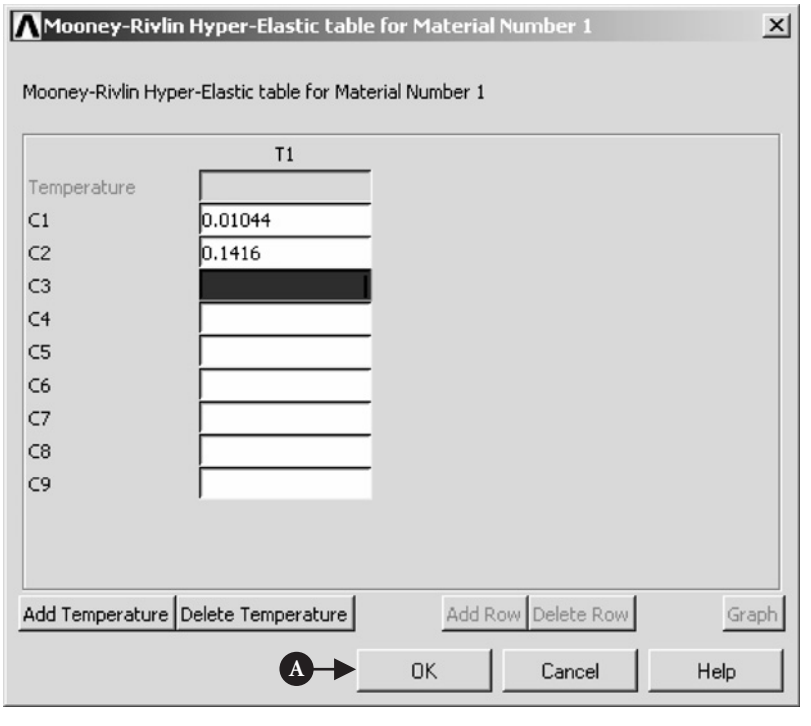


Figure 7.147 Mooney–Rivlin Hyper-Elastic table for Material Number 1.

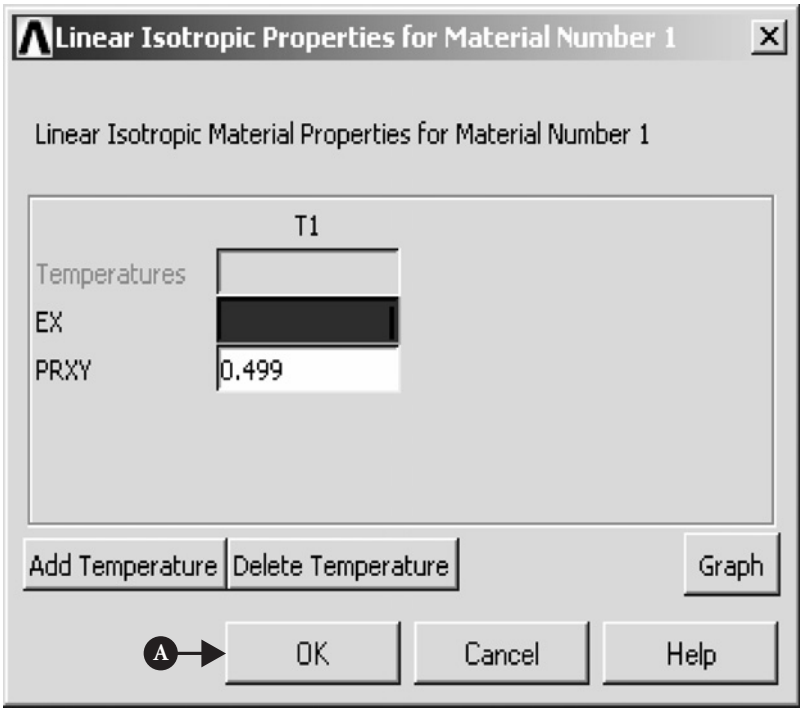


Figure 7.148 Linear Isotropic Properties for Material Number 1.

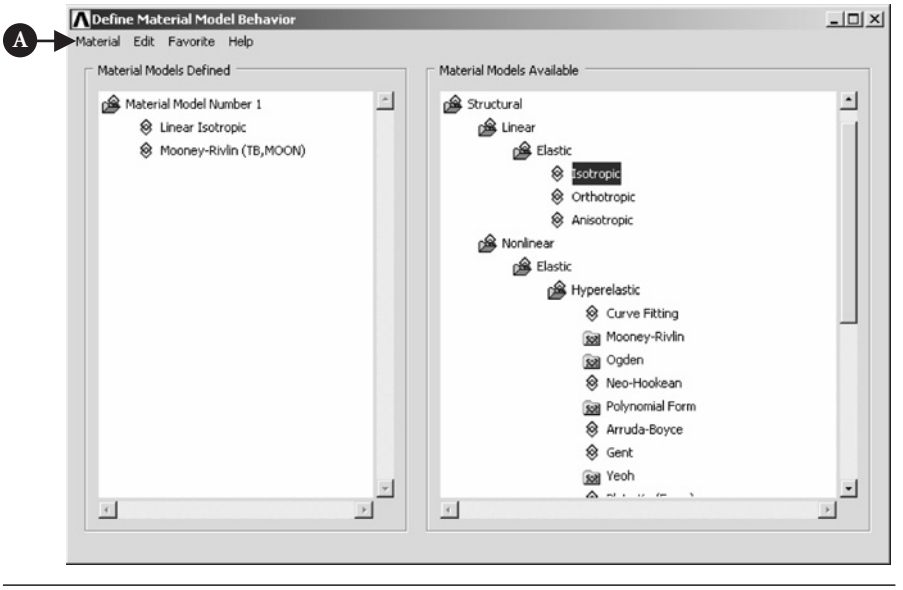


Figure 7.149 Define Material Model Behavior.

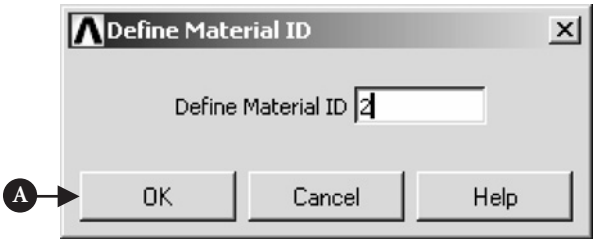


Figure 7.150 Define Material ID.

By clicking [A] **OK** button, a new material model number 2 is created as shown in Figure 7.151.

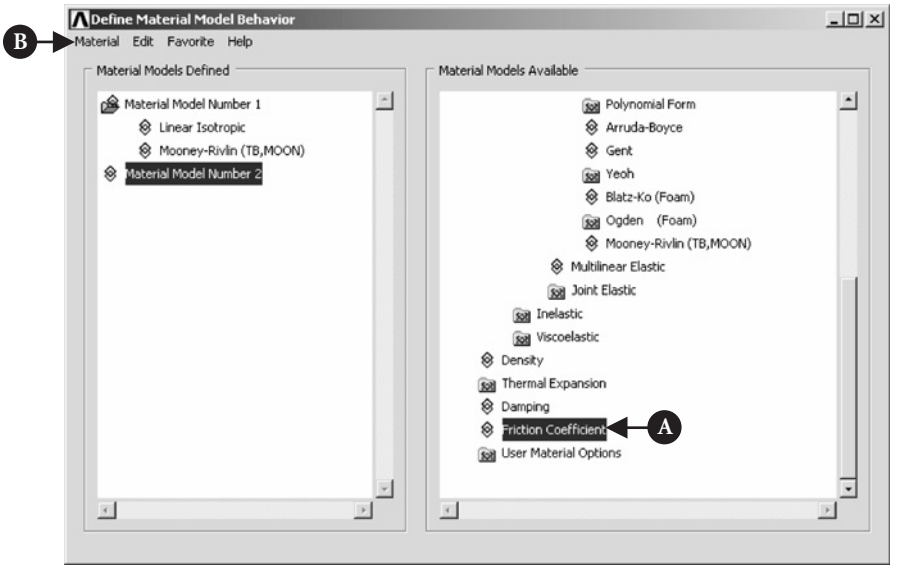


Figure 7.151 Define Material Model Behavior.

From the right-hand column, [A] **Friction Coefficient** should be selected by clicking on it twice. The frame shown in Figure 7.152 appears, where value of friction coefficient, [A] **MU** = 0.1, should be entered and [B] **OK** button clicked as shown in Figure 7.152. Material Model number 2 characterizes the friction between O-ring and the other two components (the groove and the wall) of the contact assembly.

Select [B] **Material** from the menu at the top of the frame, shown in Figure 7.151, and click **New Model** option. In response, the frame shown in Figure 7.153 appears.

By clicking [A] **OK**, a new material model number 3 is created as shown in Figure 7.154.

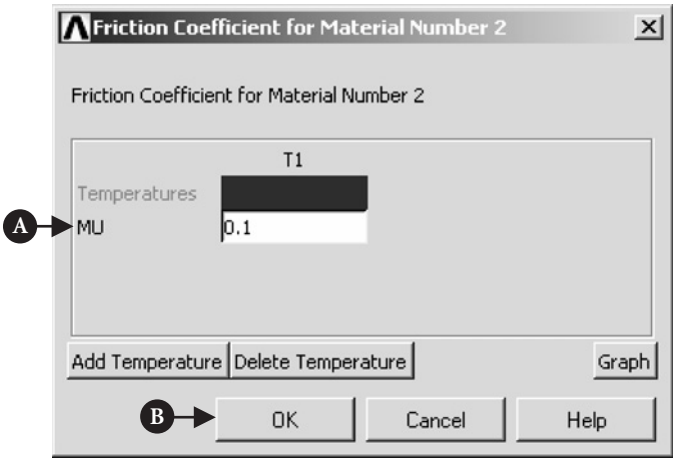


Figure 7.152 Friction Coefficient for Material Number 2.

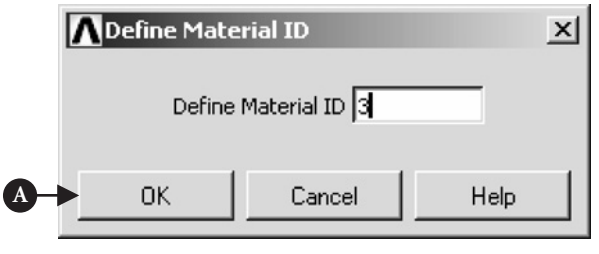


Figure 7.153 Define Material ID.

From the right-hand column, [A] **Isotropic** should be selected by clicking on it twice. This action produces a frame shown in Figure 7.155.

Material Model number 3 characterizes the groove and the wall components of the contact assembly. Young's modulus, [A] **EX** $= 2.1 \times 10^9$ Pa and Poisson's ration, [B] **PRXY** $=$ 0.33 should be entered and [C] **OK** button clicked as shown in Figure 7.155.

Having all the three materials involved in the contact assembly characterized, as shown in Figure 7.156, the frame should be closed by selecting [A] **Material** and **Exit**.

The next step in the model creation process is to define real constants, which associate material models with element types.

From **ANSYS Main Menu** select **Preprocessor → Real Constants → Add/Edit/Delete**. The frame shown in Figure 7.157 appears.

Clicking [A] **Add** produces another frame, as shown in Figure 7.158.

Element Type 1 [A] (HYPER74) does not require any real constant to be defined. Only Type 2 [B] (CONTACT48) and Type 3 [C] (LINK1) should be selected and real constants assigned to them. In Figure 7.158, Type 2 [B] (CONTACT48) is selected by highlighting it. By clicking [D] **OK**, a frame shown in Figure 7.159 is created.

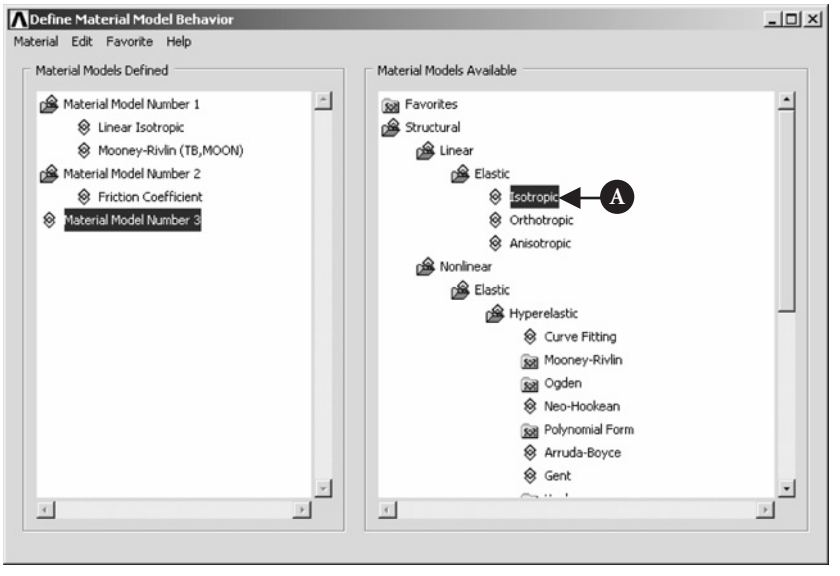


Figure 7.154 Define Material Model Behavior.

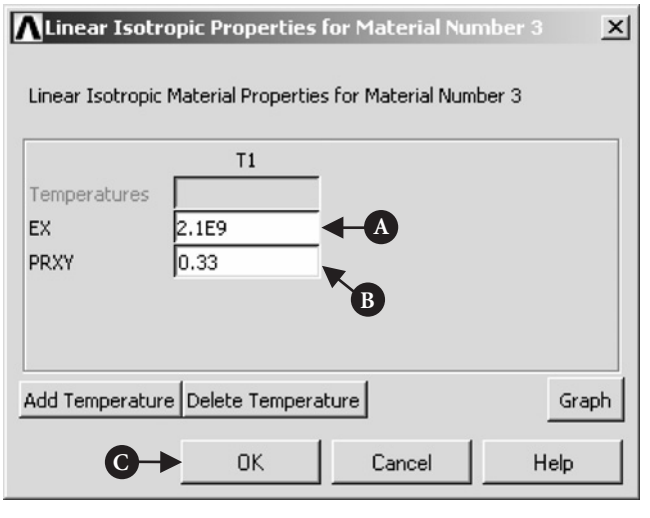


Figure 7.155 Linear Isotropic Properties for Material Number 3.

As seen in Figure 7.159, normal contact stiffness, **KN** $= 5 \times 10^3$ N/m and sticking (tangential) contact stiffness, **KT** $= 50$ N/m were entered. Also, real constant set was allocated number 2. This set refers to the contact element at the groove. The tangential stiffness equal to 50 N/m is a default value, which is usually taken to be KN/100.

Clicking [A] **OK** in frame of Figure 7.159 results in a frame shown in Figure 7.160.

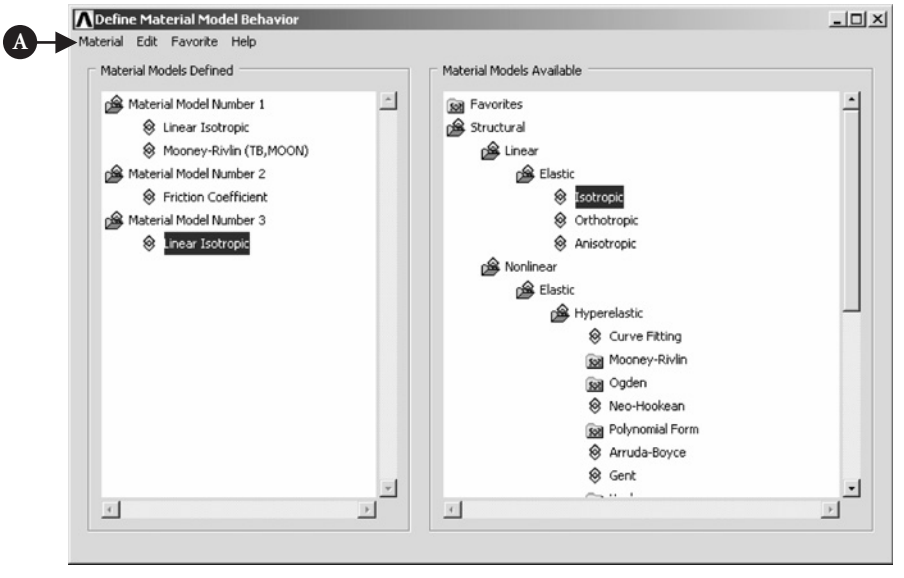


Figure 7.156 Define Material Model Behavior.

Figure 7.157 Real Constants.

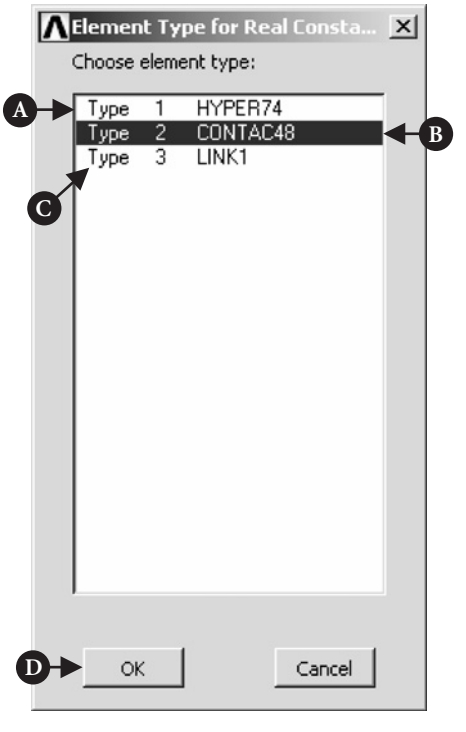


Figure 7.158 Element Type for Real Constant.

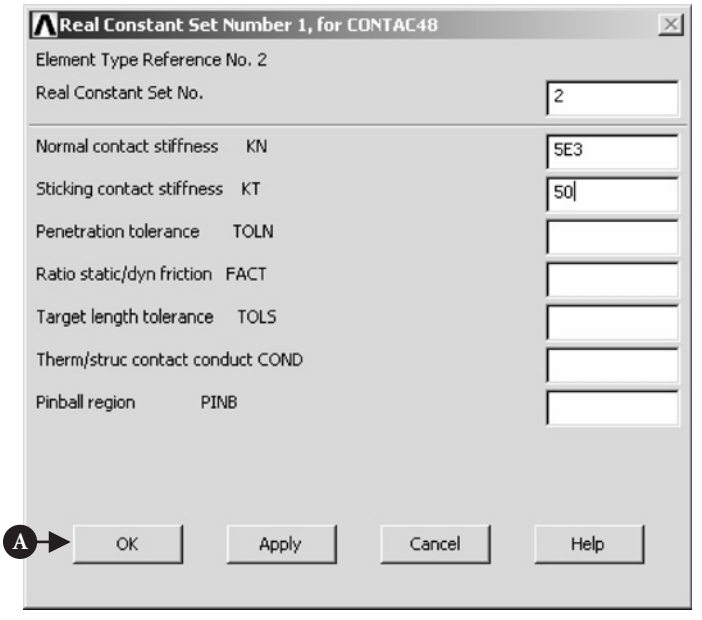


Figure 7.159 Real Constant Set for CONTACT48.

Figure 7.160 Real Constants (Set No. 2 shown).

Clicking [A] **Add** button calls up frame shown in Figure 7.158. Once again Type 2 [B] (CONTACT48) is selected by highlighting it and clicking [D] **OK**. As a result of this selection, a frame of Figure 7.161 is produced.

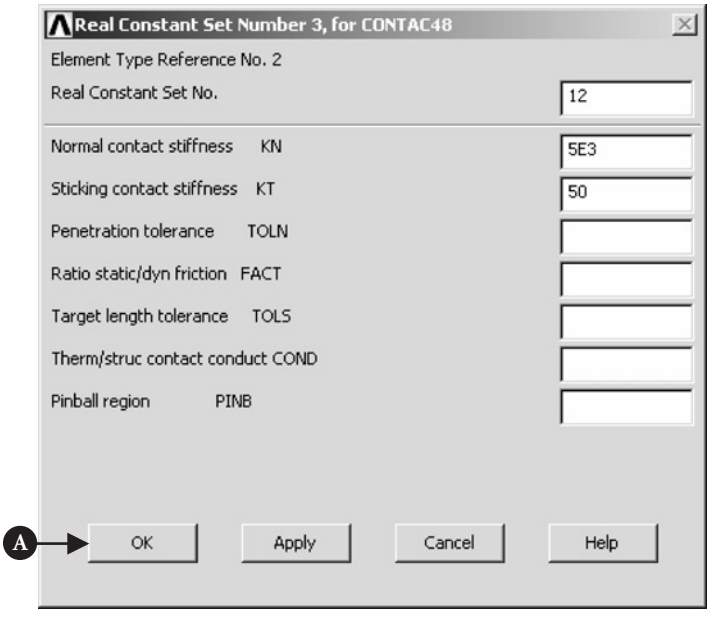


Figure 7.161 Real Contact Set for CONTACT48.

The entries shown in Figure 7.161 are similar to those of Figure 7.159. The only difference is that the set was allocated number 12 (it could be any number different from 2 already allocated). This set is linked to the contact element at the wall. Clicking [A] **OK** results in frame as shown in Figure 7.162.

The last set to be defined refers to the Type 3 (LINK1) element, representing wall and groove in the model.

In frame shown in Figure 7.162, click [A] **Add** button and select Type 3 [A] **(LINK 1)** in the frame shown in Figure 7.163.

Clicking [B] **OK** produces the frame shown in Figure 7.164.

The set was allocated number 3 and the **AREA** = 1 entered as shown in Figure 7.164. Clicking [A] **OK** button creates a frame (Figure 7.165) showing all three real constants, with numbers 2, 3, and 12.

7.2.4.4 GEOMETRY OF THE ASSEMBLY AND MESHING

The next phase in the model creation process is to draw elements involved.

From **ANSYS Main Menu** select **Preprocessor → Modelling → Create → Areas → Circle → By Dimensions**. As a result of this selection, the frame shown in Figure 7.166 appears.

By entering **RAD1** = 2.5, **RAD2** = 0, **THETA1** = 0, **THETA2** = 360, and clicking [A] **OK** button, a solid circular area is created.

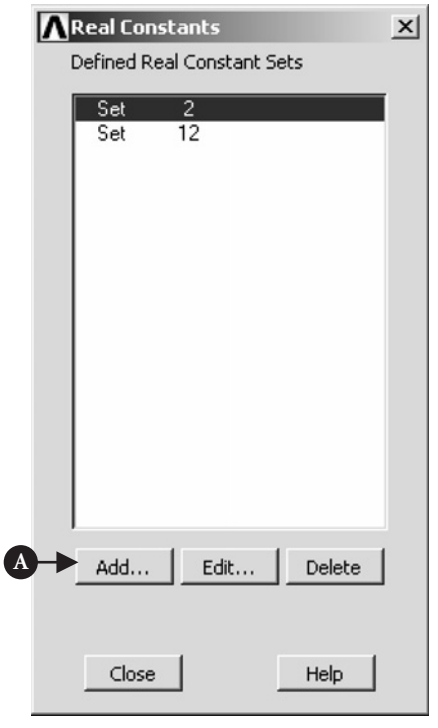


Figure 7.162 Real Constants (Sets No. 2 and 12 shown).

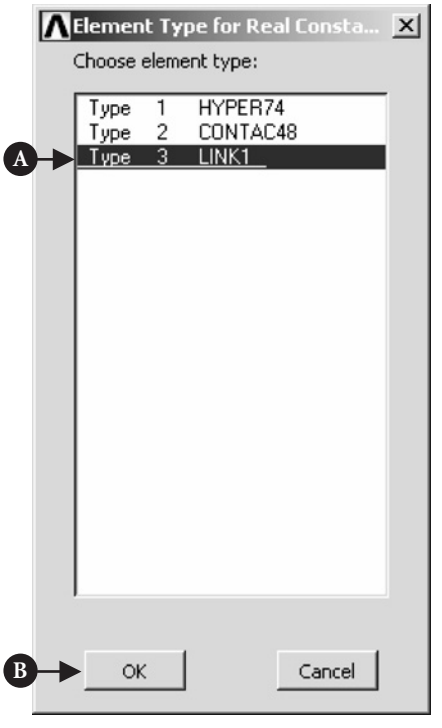


Figure 7.163 Element Type for Real Constant.

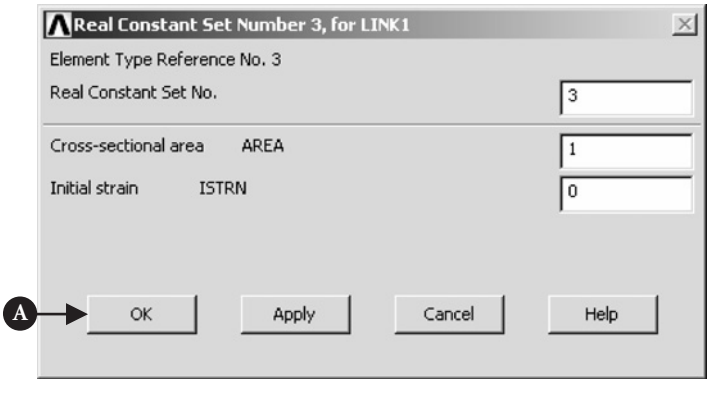


Figure 7.164 Real Constant Set for LINK1.

Next, the circular area, representing O-ring, is meshed.

From **ANSYS Main Menu** select **Preprocessor → Meshing → MeshTool**. Figure 7.167 shows the resulting frame.

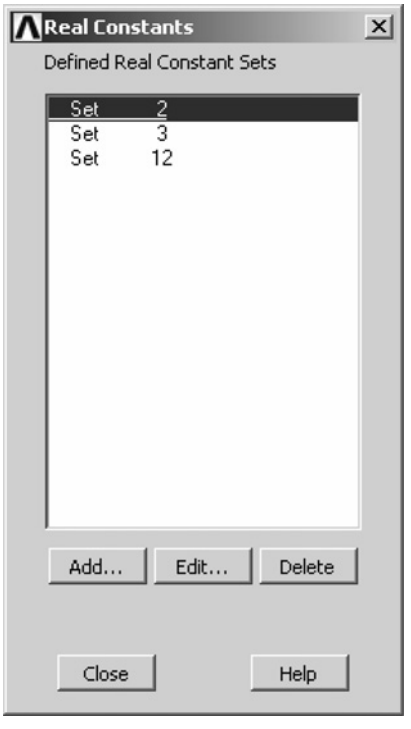


Figure 7.165 Real Constants (Sets No. 2, 3, and 12 shown).

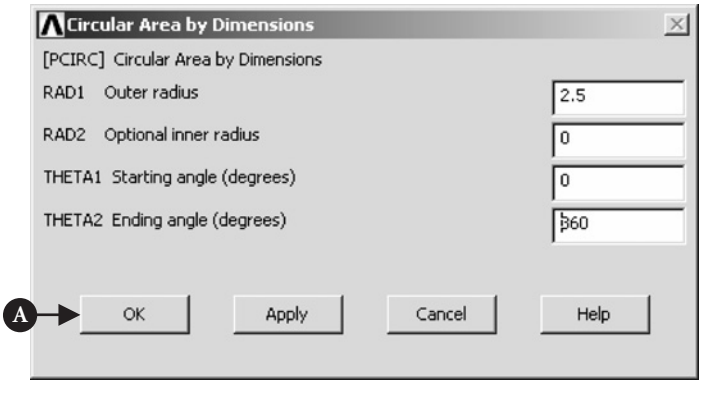


Figure 7.166 Circular Area by Dimensions.

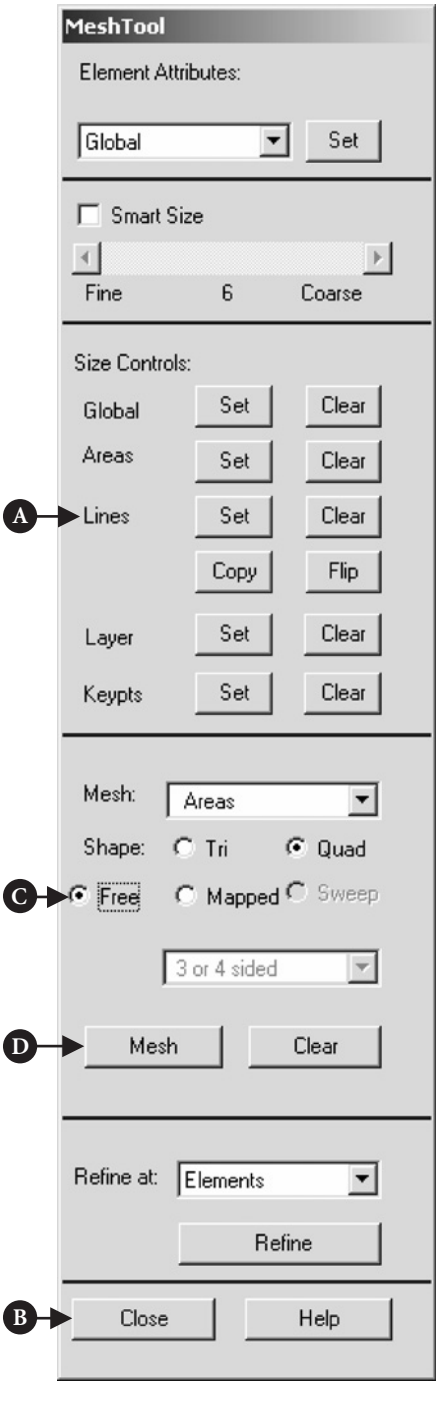


Figure 7.167 MeshTool.

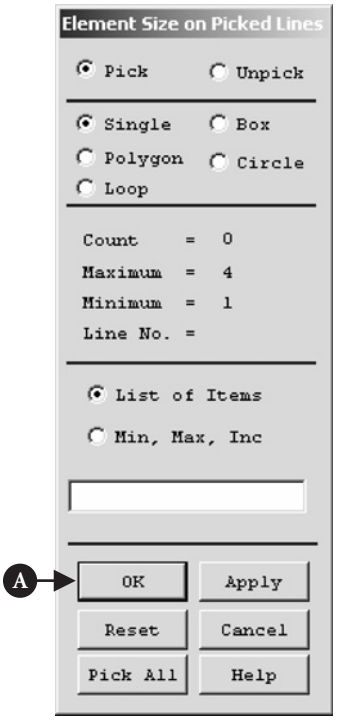


Figure 7.168 Element Size on Picked Lines.

In the frame of Figure 7.167 select [A] **Lines – Set** and [B] **Close**. A new frame is produced (see Figure 7.168).

Pick all four arcuate segments on the circumference of the circular area and click [A] **OK**. A new frame shown in Figure 7.169 appears.

In the frame of Figure 7.169 enter: number of element divisions, [A] **NDIV** = **6** and uncheck box [B] **NDIV can be changed**. Clicking [C] **OK** implements the selections made.

In the frame shown in Figure 7.167 (MeshTool), activate [C] **Free** button and click [D] **Mesh**. In the appearing frame (Figure 7.170) click [A] **Pick All** to have the circular area meshed.

Figure 7.171 shows the circular area after meshing process.

7.2.4.5 CREATING CONTACT INTERFACE

Next, the wall and the groove are modeled as nodal components with an area equal to 1.

From **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Nodes → In Active CS**. The frame shown in Figure 7.172 appears.

The input into the frame of Figure 7.172 is as follows: **node number =** 401 (it is arbitrary selection but it has to be greater than any number allocated to existing nodes), $\mathbf{X} = -2, \mathbf{Y} = -2.5$.

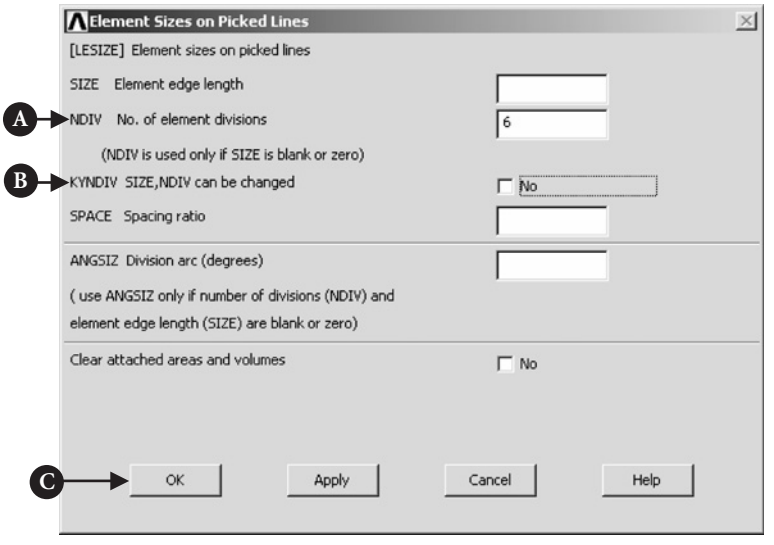


Figure 7.169 Element Sizes on Picked Lines.

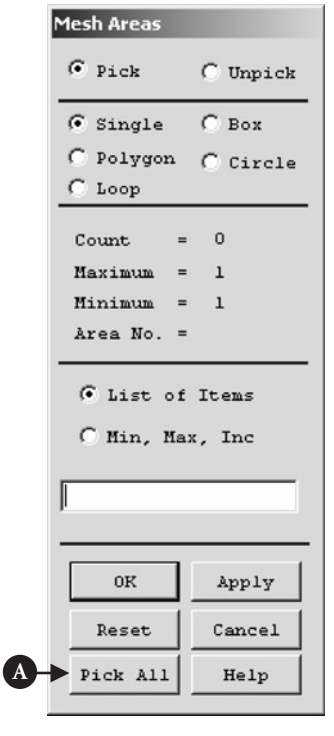


Figure 7.170 Mesh Areas.

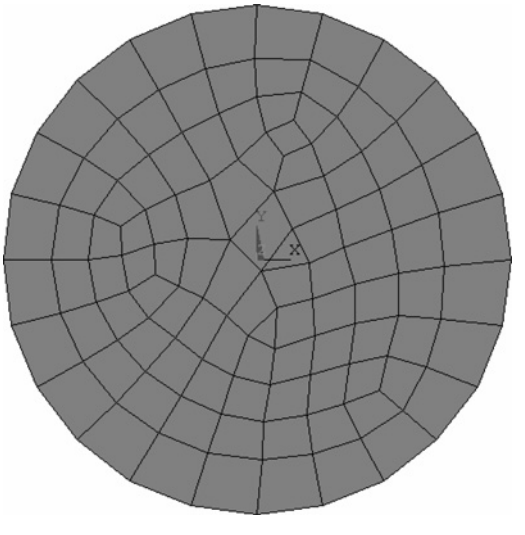


Figure 7.171 Meshed O-ring.

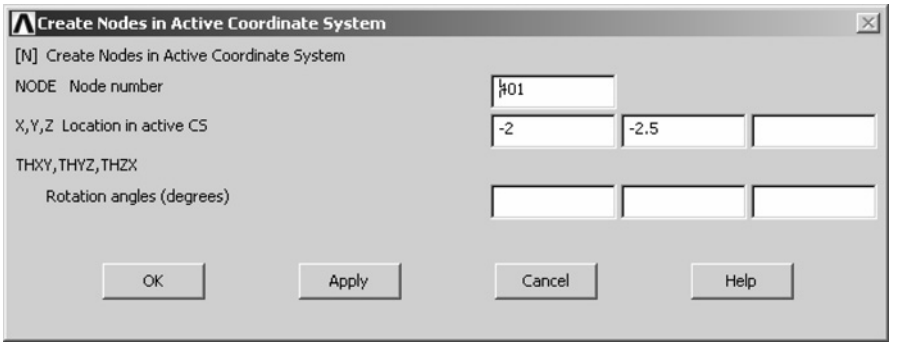


Figure 7.172 Create Nodes in Active Coordinate System.

In a similar way, the other nodes required for the groove and the wall as nodal components are created. The input coordinates are as follows: Node number = 402, X = 2.5, Y = −2.5; node number = 403, X = 2.5, Y = 3; node number = 404, X = −2, Y = 3; node number = 405, X = −2.5, Y = 5; node number = 406, X = −2.5, Y = −5. The nodes are shown in Figure 7.139.

Next, the groove and the wall elements are to be created using existing nodes with numbers from 401 to 406.

From **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Elements → Element Attributes**. The frame shown in Figure 7.173 appears.

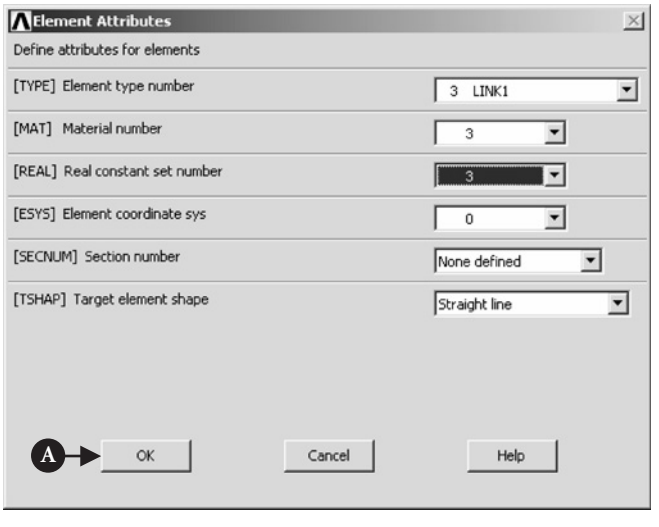


Figure 7.173 Element Attributes.

In the frame of Figure 7.173, the following selections were made: element type number, **TYPE = 3 LINK1**; material number, **MAT = 3**; and real constant set number,

REAL = 3. The selections are implemented by clicking on [A] **OK** button. All these data refer to the wall and the groove for which LINK1 was selected as the element type at the beginning of the analysis.

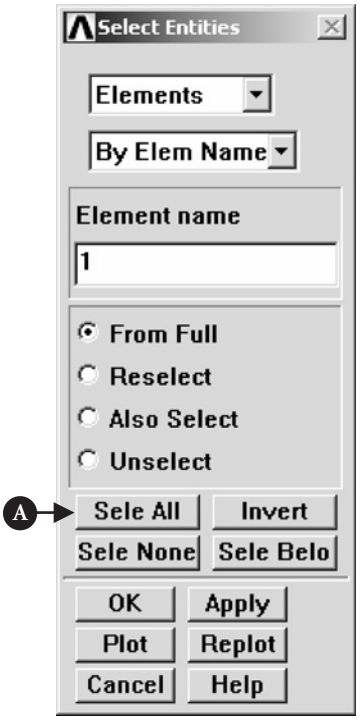


Figure 7.174 Select Entities (elements of Type 1: HYPER74).

Now, the nodal components are going to be created.

O-ring as a component

From **ANSYS Utility Menu** select **Select → Entities**. The frame shown in Figure 7.174 appears.

The selections made are shown in Figure 7.174. Pressing [A] **Sele All** button implements the selections made.

Next, from **ANSYS Utility Menu** select **Select → Entities**. The frame shown in Figure 7.175 appears.

Selections made are shown in Figure 7.175. This time [A] **Nodes** which are attached to elements already selected by action described above (see Figure 7.174) are selected. That is why the selection in Figure 7.175 is [B] **Attached to** and [C] **Elements** is activated. The selection of nodes takes place from a full set of elements; therefore, [D] **From Full** button is activated. Pressing [E] **OK** implements the selections.

Finally, from **ANSYS Utility Menu** select **Select → Entities**. Figure 7.176 shows the frame resulting from the selection.

This time [A] **Nodes** located on [B] **Exterior** of the circular area (representing O-ring) are selected. The selection of nodes takes place from the set of nodes already selected in the process described above (see Figure 7.175). This is why, [C] **Reselect** button is activated. Pressing [D] **OK** implements the selections made.

From **ANSYS Utility Menu** select **Select → Comp/Assembly → Create Component**. In response to this selection, the frame shown in Figure 7.177 appears.

As it is seen in Figure 7.177, component name, **Cname = O-ring** was entered. Pressing [A] **OK** creates nodal component with the name O-ring.

From **ANSYS Utility Menu** select **Select → Entities** to call up frame shown in Figure 7.178.

In order to select all elements involved in the model, [A] **Sele All** should be pressed followed by [B] **OK**. Pressing **OK** creates another frame of Figure 7.179.

[A] **Pick All** should be clicked in order to implement the selection of all nodes.

Groove as a component

From **ANSYS Utility Menu** select **Select → Entities**. The frame shown in Figure 7.180 appears.

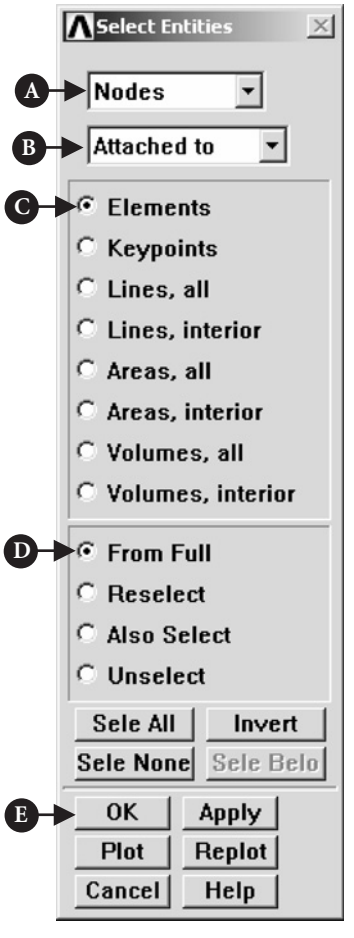


Figure 7.175 Select Entities (nodes attached to elements).

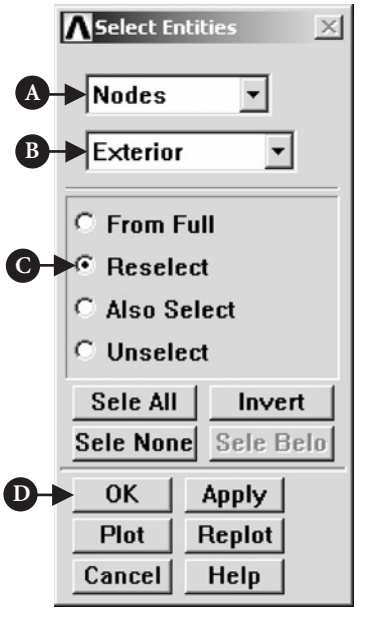


Figure 7.176 Select Entities (nodes, exterior).

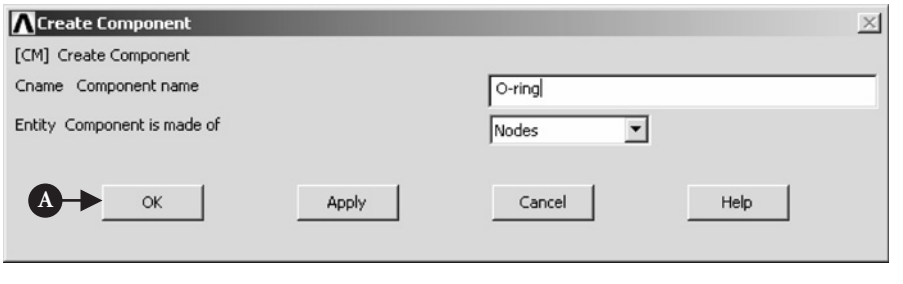


Figure 7.177 Create Component (O-ring).

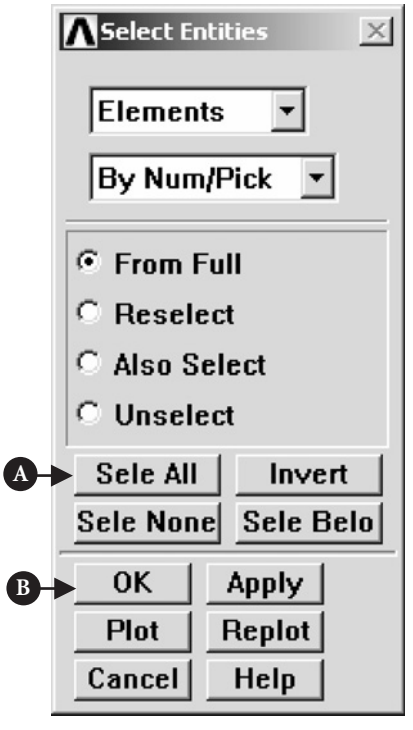


Figure 7.178 Select Entities (all elements of Type 1: HYPER74).

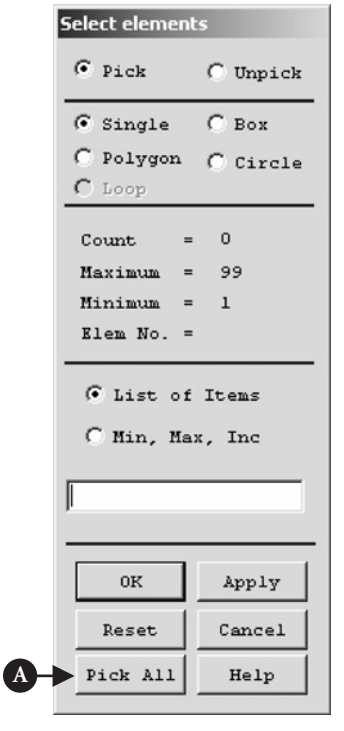


Figure 7.179 Select elements.

Nodes and **By Num/Pick** should be selected. Pressing [A] **OK** creates another frame shown in Figure 7.181.

Nodes from 401 to 404, belonging to the groove, should be picked and afterward [A] **OK** button pressed to implement the selection.

Next, from **ANSYS Utility Menu** select **Select → Comp/Assembly → Create Component**. The frame shown in Figure 7.182 appears.

Enter **Groove** in the component name box and press [A] **OK** to create nodal component called "groove."

Wall as a component

From **ANSYS Utility Menu** select **Select → Entities**. The frame shown in Figure 7.180 appears. As shown in Figure 7.180, **Nodes** and **By Numb/Pick** should again be selected and implemented by pressing [A] **OK**. This recalls frame shown in Figure 7.181. This time nodes 405 and 406, belonging to the wall, should be picked and [A] **OK** pressed.

Next, from **ANSYS Utility Menu** select **Select → Comp/Assembly → Create Component**. Then, a frame shown in Figure 7.183 appears.

As shown in Figure 7.183, component name is **Wall**. Pressing [A] **OK** implements the selection.

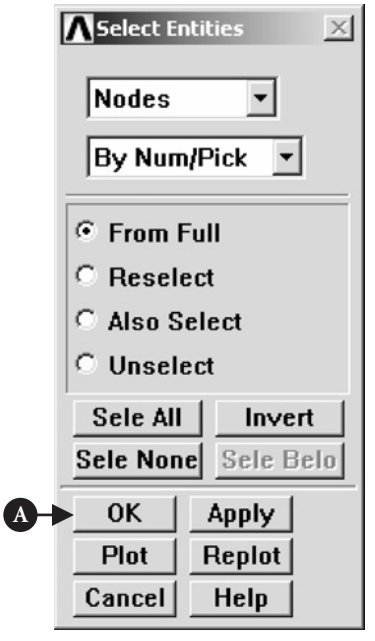


Figure 7.180 Select Entities.

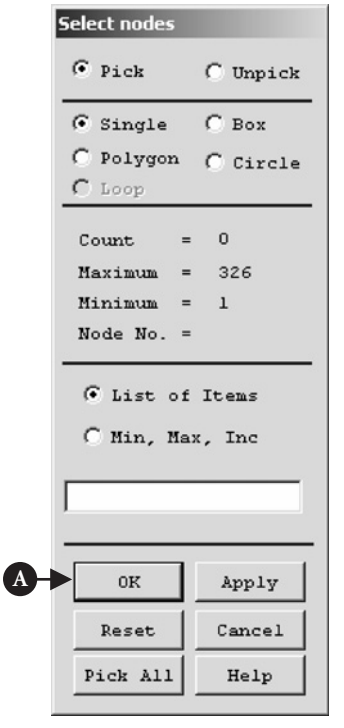


Figure 7.181 Select nodes (nodes 401 to 404 defining groove).

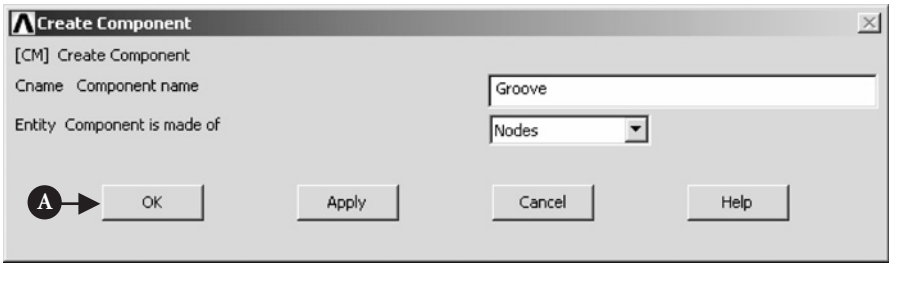


Figure 7.182 Create Component (groove).

Finally, from **ANSYS Utility Menu** select **Select → Entities** and in the frame enter selections as shown in Figure 7.184.

Click [A] **Sele All** button and in the next appearing frame press **Pick All** in order to select all nodes belonging to the model of the O-ring assembly.

This action ends the process of creating the model of the O-ring assembly. The plot of all elements involved looks like that shown in Figure 7.185.

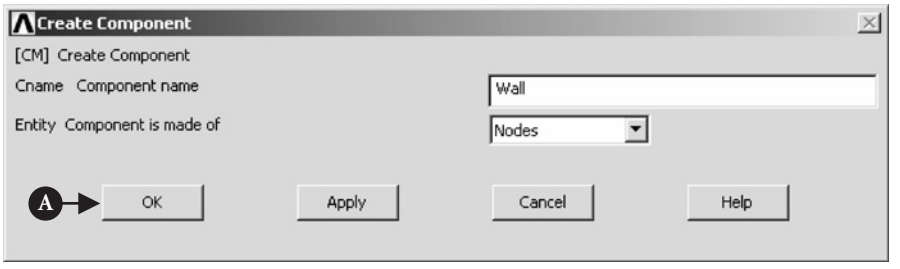


Figure 7.183 Create Component (wall).

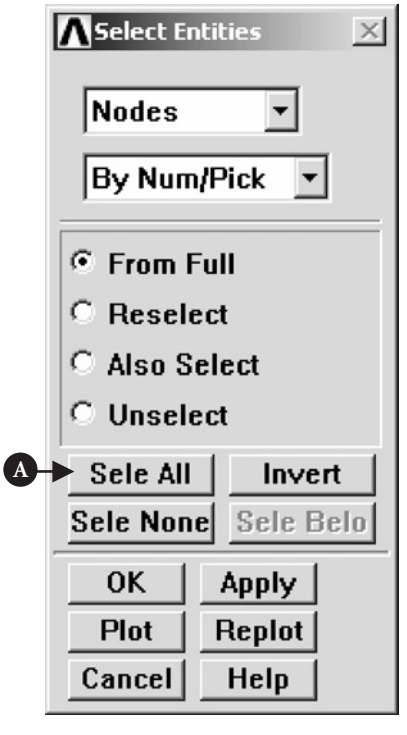


Figure 7.184 Select Entities (all nodes).

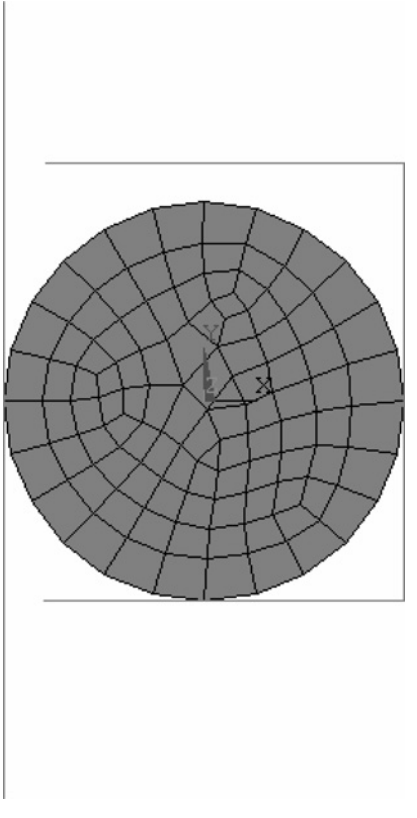


Figure 7.185 Plot of elements in the O-ring assembly.

Contact elements

The final stage in the modeling process is the creation of contact elements.

From **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Elements → Element Attributes**. As a result of this selection, the frame shown in Figure 7.186 appears.

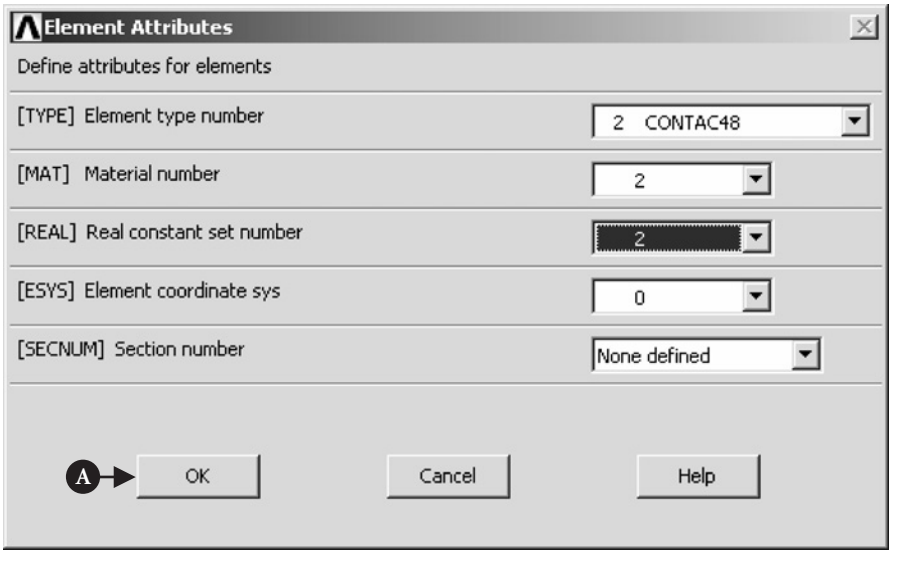


Figure 7.186 Element Attributes.

It is seen in Figure 7.186 that Type **2 CONTACT48** elements were selected as well as material number, **MAT = 2** and the real constant set number, **REAL = 2**. This selection is pertinent to contact elements at the groove and is implemented by pressing [A] **OK** button. Next, from **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Elements → Surf/Contact → Node to Surf**. This selection creates a frame shown in Figure 7.187.

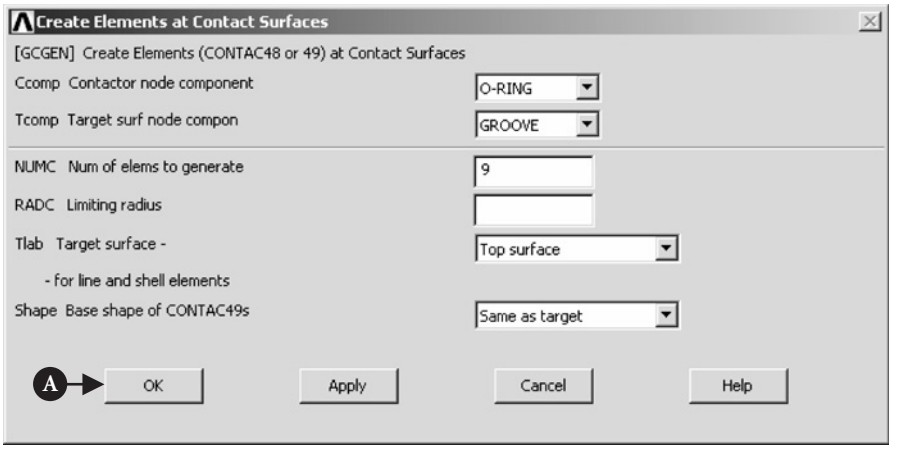


Figure 7.187 Create Elements at Contact Surfaces (the groove).

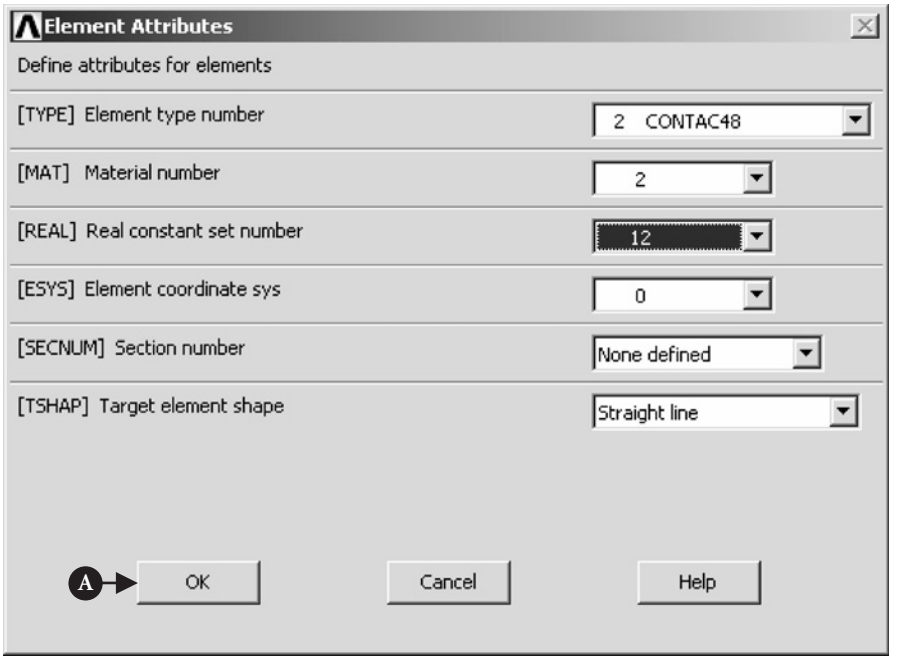


Figure 7.188 Element Attributes (the wall).

Selections made and shown in Figure 7.187 are as follows: contactor node component, **Ccomp = O-RING**; target surf node compon, **Tcomp = GROOVE**; and number of elements to generate, **NUMC = 9**. Selections are implemented by clicking [A] **OK**.

From **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Elements → Element Attributes**. As a result of this selection, the frame shown in Figure 7.188 appears.

The selections made, as shown in Figure 7.188, are as follows: element type number, **TYPE 2 = CONTACT48**; material number, **MAT = 2**; and real constant set number, **REAL = 12**. All selections made are associated with the wall. By clicking [A] **OK** selections made are implemented.

Next, from **ANSYS Main Menu** select **Preprocessing → Modelling → Create → Elements → Surf/Contact → Node to Surf**. This creates a frame shown in Figure 7.189. Inputs into the frame are shown and they are implemented by clicking [A] **OK**.

The final action is to reorder elements in the X-direction.

From **ANSYS Main Menu** select **Preprocessor → Numbering Ctrls → Element Reorder → Reorder by XYZ**. This creates a frame shown in Figure 7.190.

In the pull down menu [A] **Coord direction for sort** the option [B] **X direction only** was selected.

7.2.4.6 SOLUTION

In the solution stage, various selections are made affecting execution of the solution.

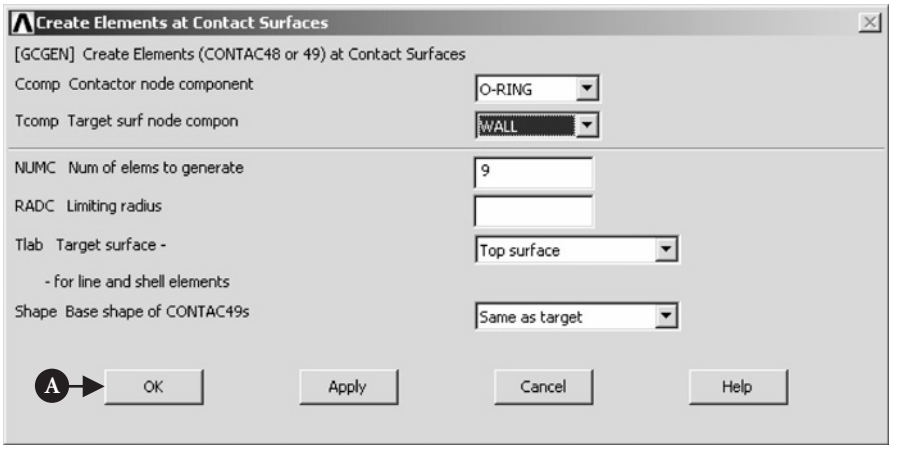


Figure 7.189 Create Elements at Contact Surfaces (the wall).

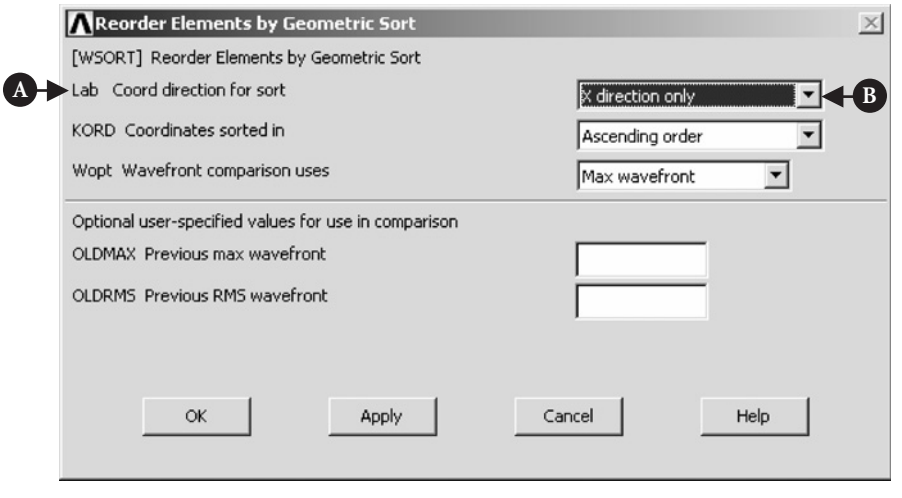


Figure 7.190 Reorder Elements by Geometric Sort.

From **ANSYS Main Menu** select **Solution → Analysis Type: Sol'n Controls**. In response, the frame shown in Figure 7.191 appears.

As shown in Figure 7.191, the following selections are made: **Analysis Options = Large Displacement Static** (because a hyper-elastic material is involved); **Time at end of load step = 1**; and **Automatic time stepping = On**. [A] **Time increment** should be activated and then, **Time step size** $= 1 \times 10^{-2}$; **Minimum time step** $= 1 \times 10^{-4}$. In the [B] **Frequency tab**, select **Write *N* number of substeps** and use $N = 5$.

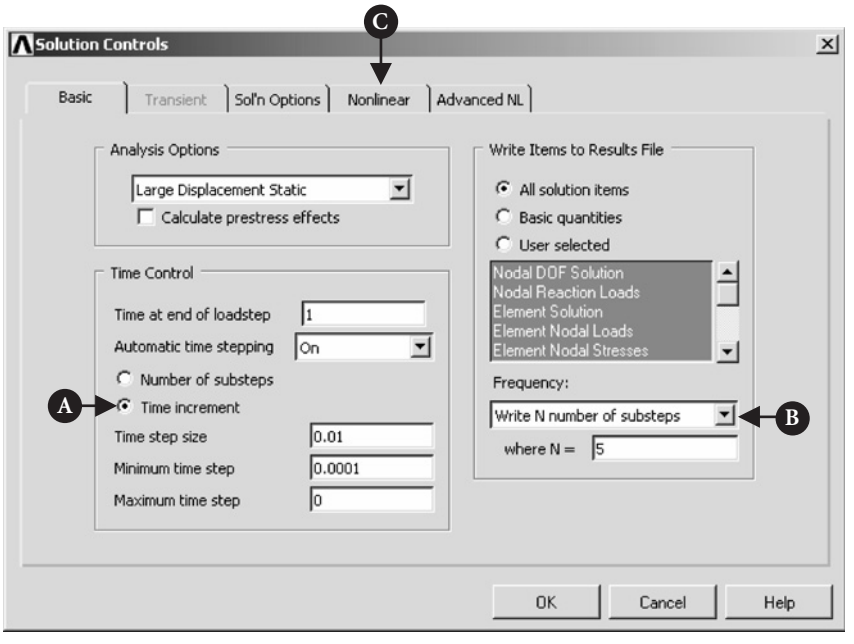


Figure 7.191 Solution Controls.

Next, press [C] **Nonlinear** tab located at the top of the frame of Figure 7.191. New frame shown in Figure 7.192 appears.

In this frame, showing nonlinear options, the following selection is made: **DOF solution predictor = On after 1 substep**. Additionally, [A] **Set convergence criteria** button ought to be pressed in order to set convergence value appropriate for the analysis to be performed here. The frame shown in Figure 7.193 is produced.

Button [A] **Replace** should be pressed in order to modify the default convergence criteria. The frame shown in Figure 7.194 is produced.

In the box minimum reference value, [A] **MINREF = 0.1**, should be typed and [B] **OK** button pressed to implement the selection.

After defining solution options, loads acting on the O-ring should be applied. There are two types of loading to be considered here. First type of loading is produced by moving the wall 0.2 units in X-direction. This results in squeezing of the O-ring. The second type of loading is produced by simultaneously applying squeeze and pressure over the upper surface of the O-ring.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Nodes**. The frame shown in Figure 7.195 appears. Nodes belonging to the wall and the groove should be picked (nodes with numbers 401–406).

When that is done, [A] **OK** button should be pressed and the frame shown in Figure 7.196 appears.

As it is seen in Figure 7.196, **All DOF** option is selected and the displacement value, **VALUE = 0** entered. Pressing [A] **OK** button implements the selection, which means that both the groove and the wall are initially constrained in all direction.

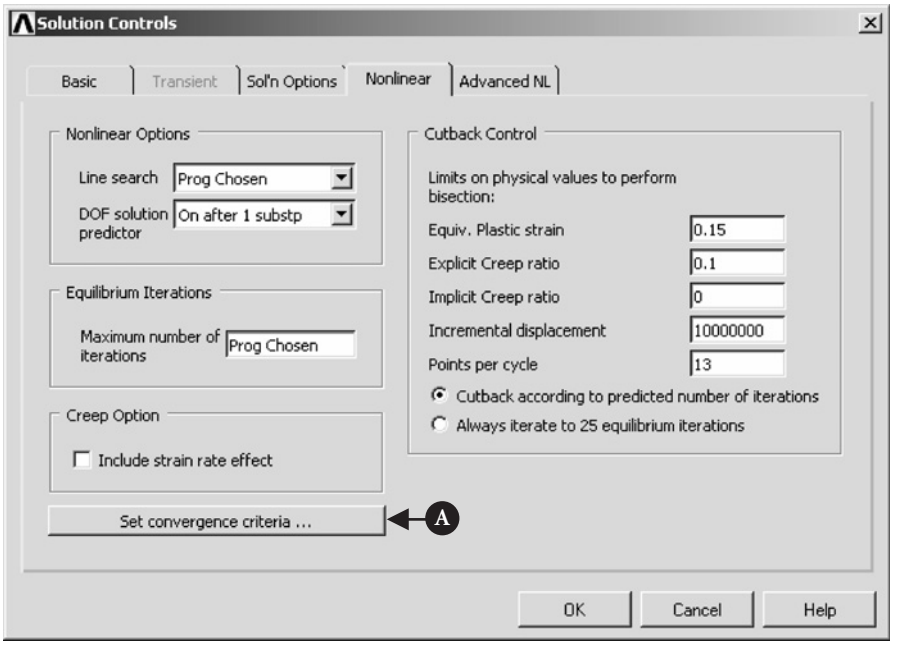


Figure 7.192 Solution Controls.

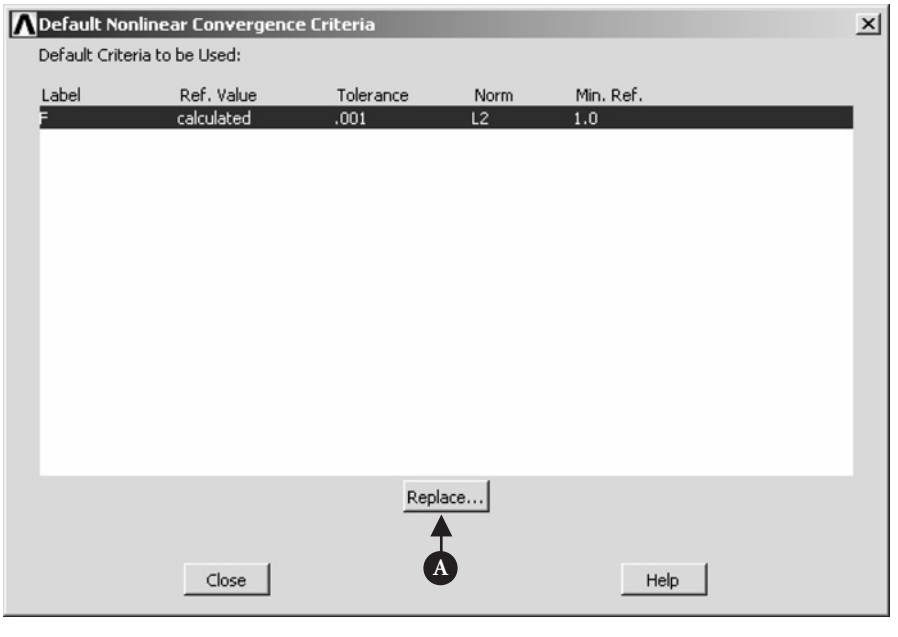


Figure 7.193 Default Nonlinear Convergence Criteria.

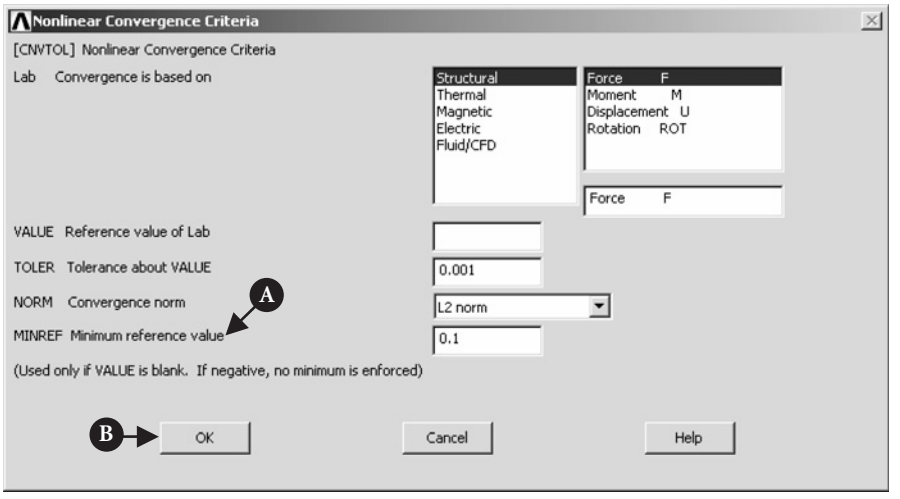


Figure 7.194 Nonlinear Convergence Criteria (modified).

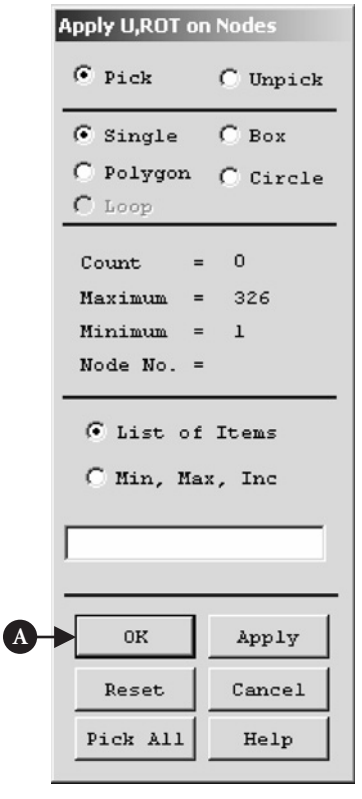


Figure 7.195 Apply U,ROT on Nodes.

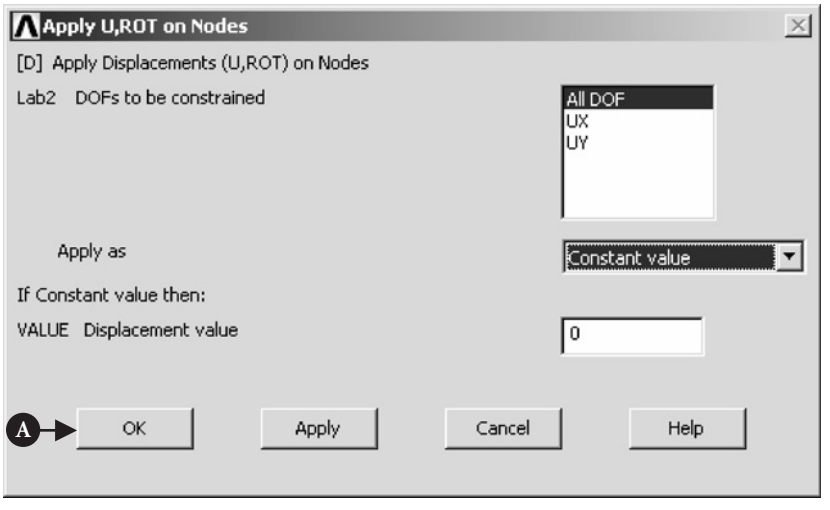


Figure 7.196 Apply U,ROT on Nodes (All DOF selected).

Next, the wall should be moved by 0.2 units in the X-direction to squeeze the O-ring and, through that, apply load on it.

From **ANSYS Main Menu** select **Solution → Define Loads → Apply → Structural → Displacement → On Nodes**. The frame shown in Figure 7.195 appears. This time nodes 405 and 406 (belonging to the wall) should be picked and [A] **OK** button pressed. This action produces a frame shown in Figure 7.197.

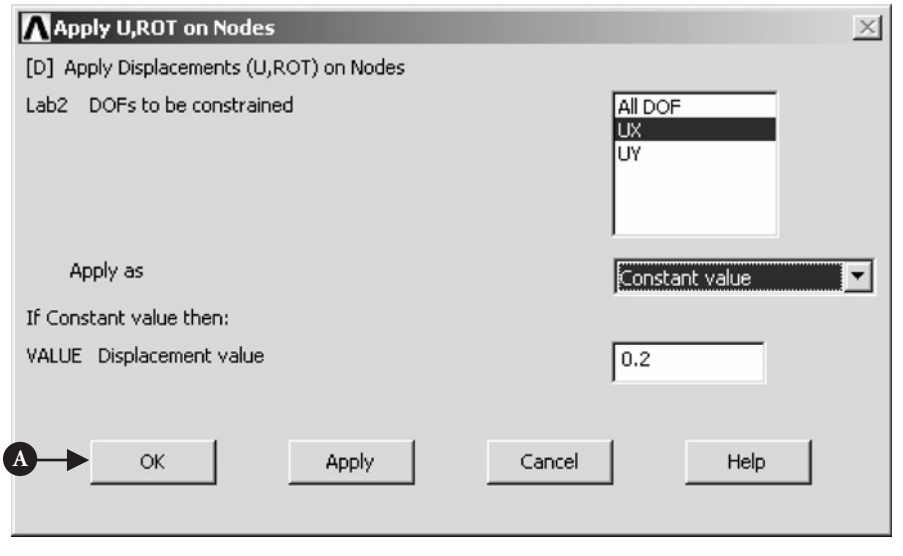


Figure 7.197 Apply U,ROT on Nodes.

Selections made and shown in Figure 7.197 are as follows: **DOFs to be constrained = UX**; and displacement value, **VALUE = 0.2**. Clicking [A] **OK** button implements the selections made.

First load step (solution)

Load on the O-ring is due to the movement of the wall in X-direction. This load was applied in the way described above. Now, the solution stage ought to be initiated.

From **ANSYS Main Menu** select **Solution → Solve → Current LS**. A frame shown in Figure 7.198 appears together with another frame, which gives a summary of solution options selected.

Figure 7.198 Solve Current Load Step.

After checking the correctness of information it should be closed by selecting **File → Close**. Next, [A] **OK** button should be pressed to initiate the solution.

7.2.4.7 POSTPROCESSING (FIRST LOAD STEP)

In order to observe deformations and stresses produced by the load applied to the O-ring through the movement of the wall in X-direction by 0.2 units, a postprocessing facilities of ANSYS should be used.

From **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. Figure 7.199 shows the resulting frame.

Entries to the frame are shown in Figure 7.199. Pressing [A] **OK** button implements the selections made.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 7.200 appears.

Selections made and shown in Figure 7.200 are as follows: **Item to be contoured = Stress** and **von Mises SEQV**. Clicking [A] **OK** button results in the plot shown in Figure 7.201.

In order to see, simultaneously, deformed and undeformed shapes, a button [B] **Def + undef edge** should be activated in the frame shown in Figure 7.200 and [A] **OK** tab pressed. Figure 7.202 shows the resulting image.

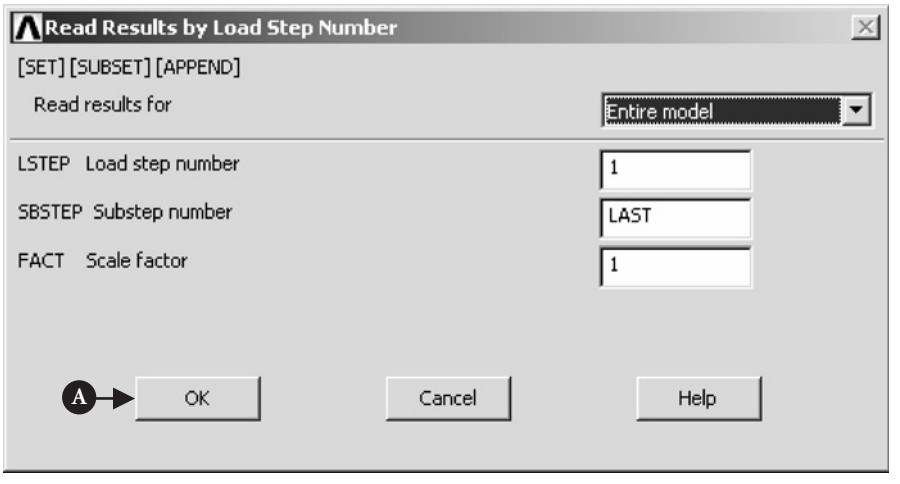


Figure 7.199 Read Results by Load Step Number.

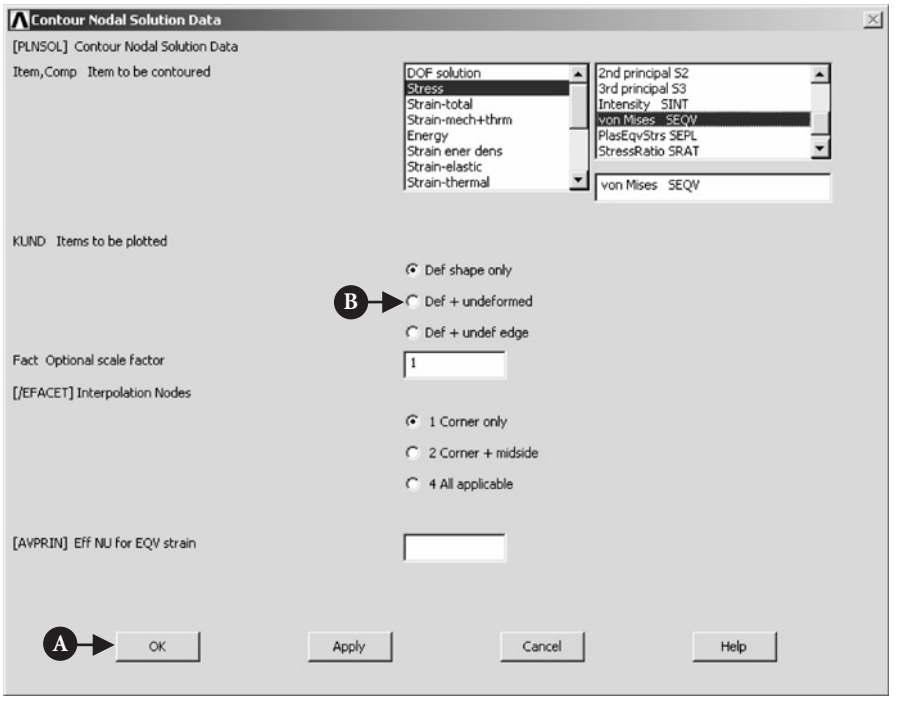


Figure 7.200 Contour Nodal Solution Data.

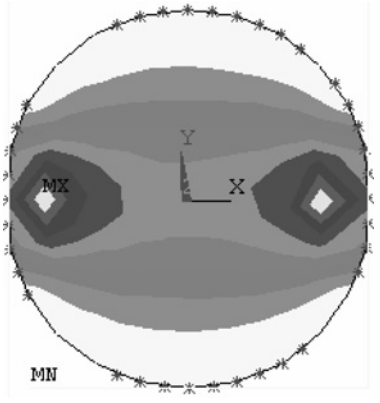


Figure 7.201 von Mises stress in O-ring (deformed shape only).

7.2.4.8 SOLUTION (SECOND LOAD STEP)

The second load step involves applying, in addition to the load due to the wall movement, pressure acting over the top surface of the O-ring. Because the pressure effectively acts over the angle from 14° to 166°, therefore, to apply it properly it is convenient to change coordinate system from Cartesian to Polar.

From **ANSYS Utility Menu** select **WorkPlane → WP Settings**. The frame shown in Figure 7.203 appears.

In the frame [A] **Polar** should be activated and [B] **OK** tab pressed to implement the selection.

Next, from **ANSYS Utility Menu** select **WorkPlane → Change Active CS to: Working Plane**. This selection makes sure that the active coordinate system is identical with that of WP coordinate system, which is polar system.

From **ANSYS Utility Menu** select **Select → Entities**. This selection produces a frame shown in Figure 7.204.

In order to select all external nodes belonging to the O-ring (located on its periphery) the following selections, shown in Figure 7.204, are made: [A] **Nodes**, [B] **Exterior**, and [C] **From Full**. Pressing [D] **OK** implements the selections made.

Next, from **ANSYS Utility Menu** select **Select → Entities**. In response, a frame shown in Figure 7.205 appears.

As shown in Figure 7.205, the following selections are made to apply pressure acting on external nodes between 14° and 166°: **Nodes**; **By Location**; and **Min, Max = 14**, **166**. Also **Y** and **Reselect** are activated. The reason for that is that Y-direction is now measured in degrees (polar coordinate system) and, reselect, option is used to pick up nodes at which pressure is applied from the external nodes already selected. Pressing [A] **OK** implements the selections made.

Next, all associated elements with nodes selected should be picked up. Therefore, from **ANSYS Utility Menu** select **Select → Entities**. In response, a frame shown in Figure 7.206 appears.

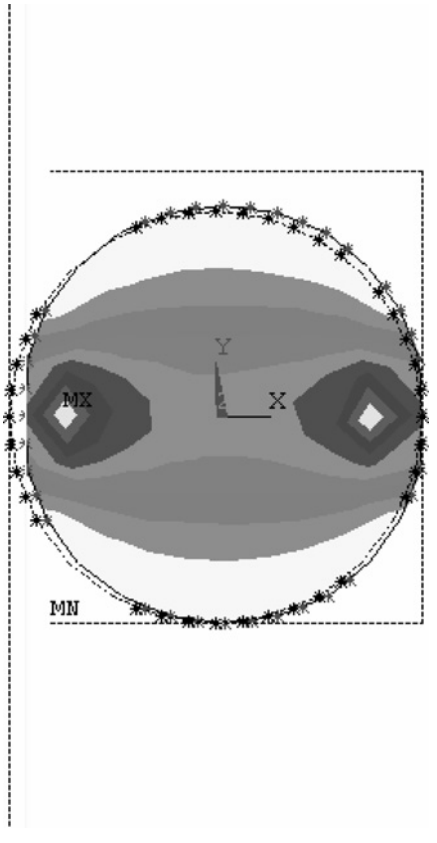


Figure 7.202 von Mises stress in O-ring (deformed shape and undeformed edge).

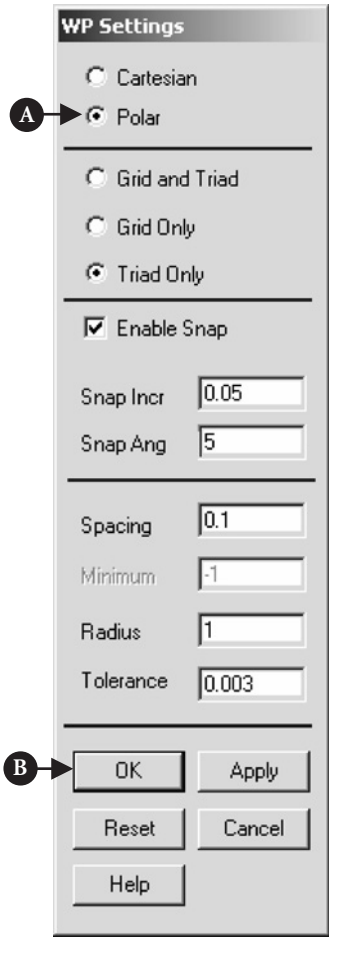


Figure 7.203 WP Settings (Polar system is selected).

Selections made are shown in Figure 7.206 and are implemented by clicking [A] **OK**.

It is now necessary to deselect all contact elements, Type 2 CONTACT48, from being involved in the load transmission, therefore, from **ANSYS Utility Menu** select **Select → Entities**. Figure 7.207 shows the resulting frame. Activate [A] **Unselect** button and click on [B] **OK** to implement the action.

In order to apply pressure to selected external nodes belonging to the O-ring the following steps should be taken.

From **ANSYS Main Menu** select **Preprocessor → Solution → Define Loads → Apply → Structural → Pressure → On Nodes**. As a result of this selection, a frame shown in Figure 7.208 appears.

Pressing [A] **Pick All** creates another frame shown in Figure 7.209.

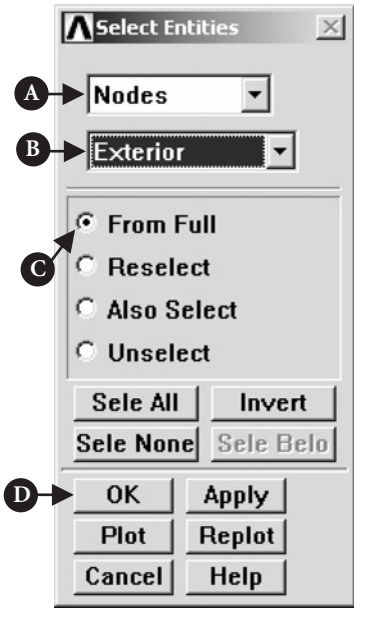


Figure 7.204 Select Entities.

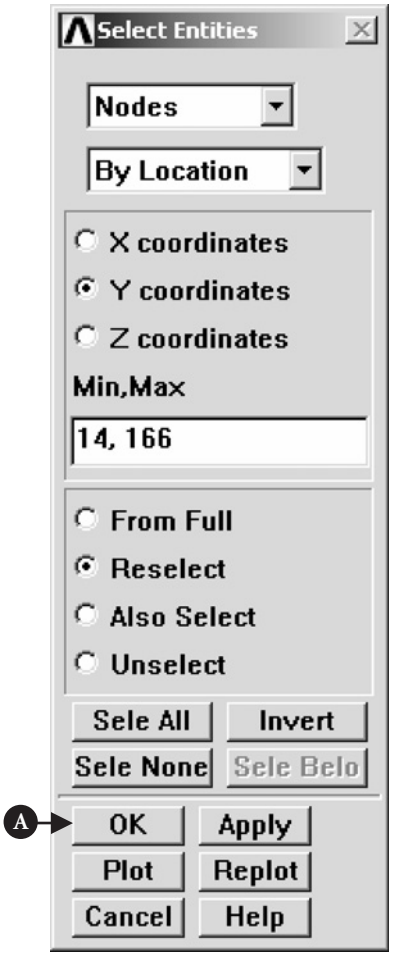


Figure 7.205 Select Entities (nodes).

Load PRES value, [A] **VALUE = 0.1** MPa is entered in the frame shown in Figure 7.209. Pressing [B] **OK** implements the selection.

Finally, from **ANSYS Utility Menu** select **Select → Entities**. Figure 7.210 shows the appearing frame.

Select [A] **Elements**, [B] **By Num/Pick**, and press [C] **Sele All** in order to select all elements involved in the model of the O-ring assembly. Clicking [D] **OK** implements the selections made.

In a similar way all nodes involved in the model of the O-ring assembly should be selected. From **ANSYS Utility Menu** select **Select → Entities**. Then the frame shown in Figure 7.211 appears.

The following selections should be made: **Nodes**, and **By Num/Pick**. Pressing [A] **Sele All** button selects all nodes involved. Finally, [B] **OK** should be pressed to implement the selection.

Before the solution is activated, solution options should be modified.

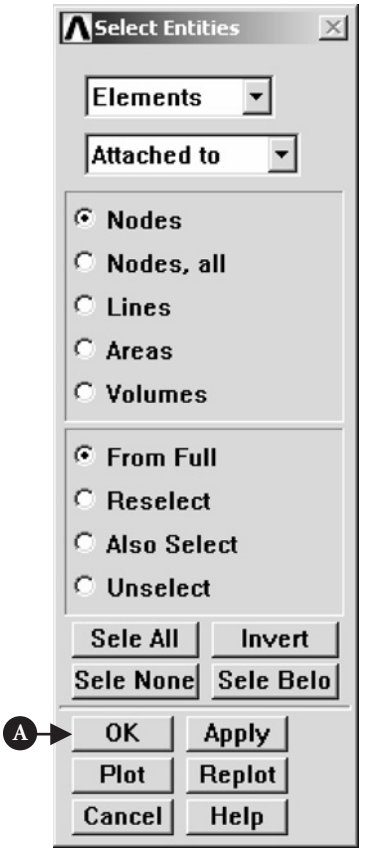


Figure 7.206 Select Entities (elements attached to nodes).

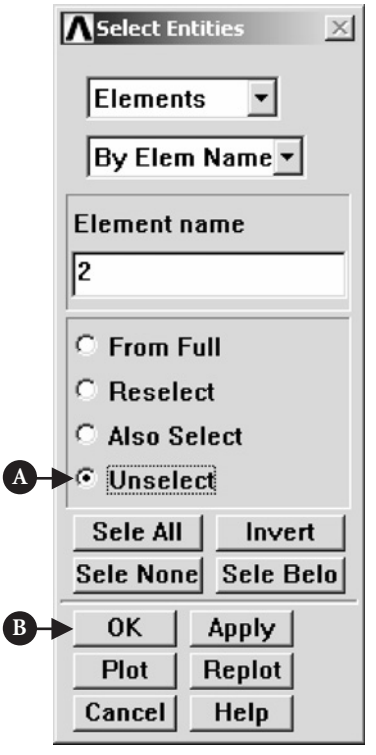


Figure 7.207 Select Entities (contact elements, Type 2, unselected).

From **ANSYS Main Menu** select **Preprocessor → Solution → Analysis Type → Sol'n Controls.** As a result of this selection, a frame shown in Figure 7.212 appears.

It is seen in Figure 7.212 that following entries are required: **Large Displacement Static**; **Time at end of load step** = 2; **Automatic time stepping** = On; **time step size** = 0.1; **Minimum time step** = 0.001; and **Write *N* number of substeps**, $N = 15$.

These are all modifications to the solution options required to run solution for the second load step.

From **ANSYS Main Menu** select **Preprocessor → Solution → Define Loads → Apply → Structural → Displacement → On Nodes**. As a result of this selection, a frame shown in Figure 7.213 appears.

Nodes 405 and 406 (belonging to the wall) should be picked and [A] **OK** button pressed. As a result of that another frame, as shown in Figure 7.214, is produced.

Selections made are shown in Figure 7.214. This time, when constant pressure of magnitude 0.1 MPa is applied to the upper surface of the O-ring, the movement of the wall in X-direction is 0.1 unit. Therefore, [A] **VALUE** = 0.1. Clicking on [B] **OK** finishes the operation.

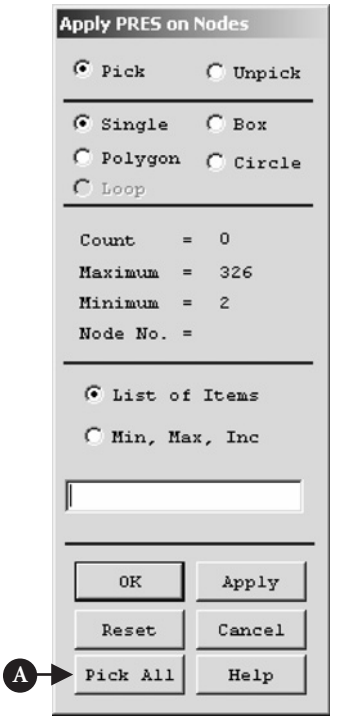


Figure 7.208 Apply PRES on Nodes.

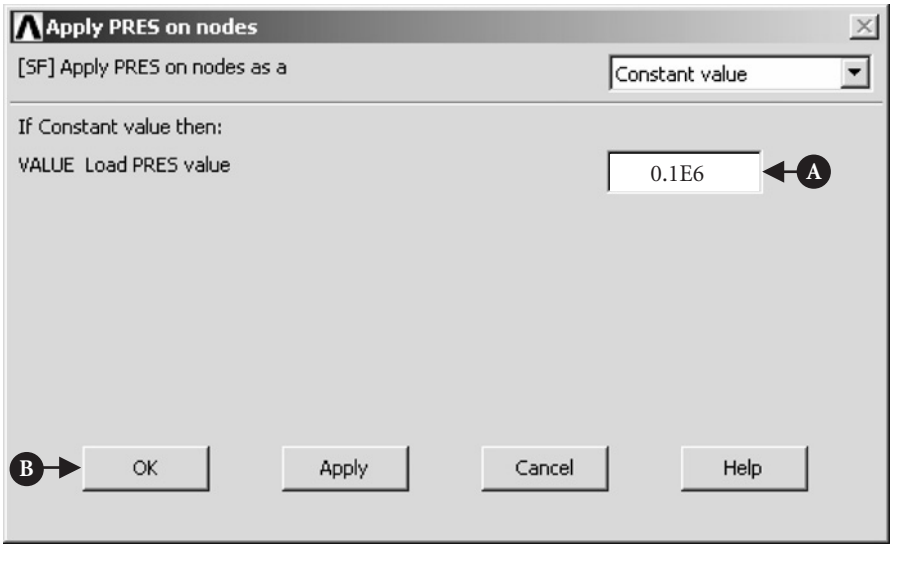


Figure 7.209 Apply PRES on nodes.

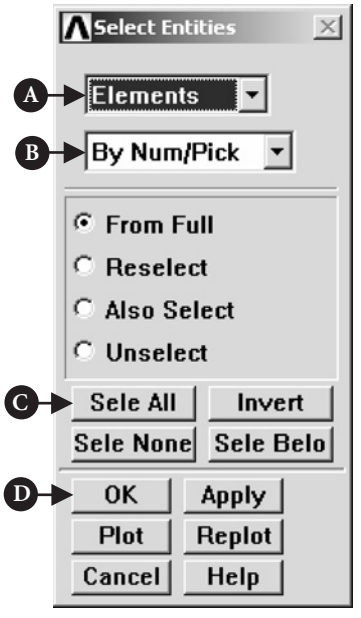


Figure 7.210 Select Entities (all elements).

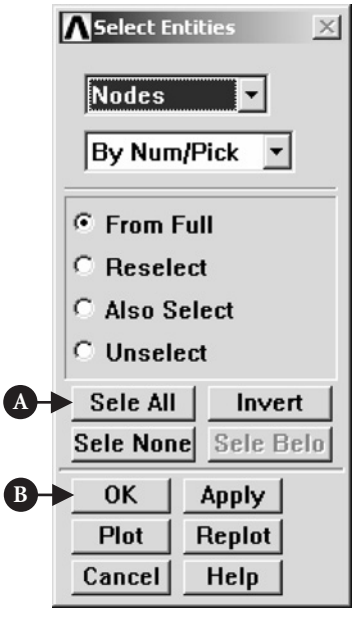


Figure 7.211 Select Entities (all nodes).

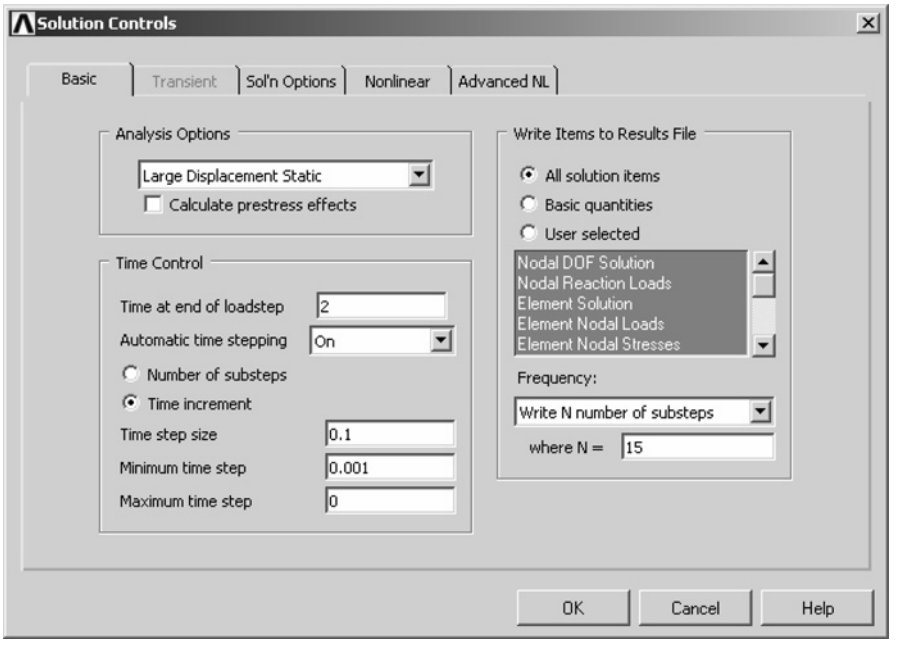


Figure 7.212 Solution Controls.

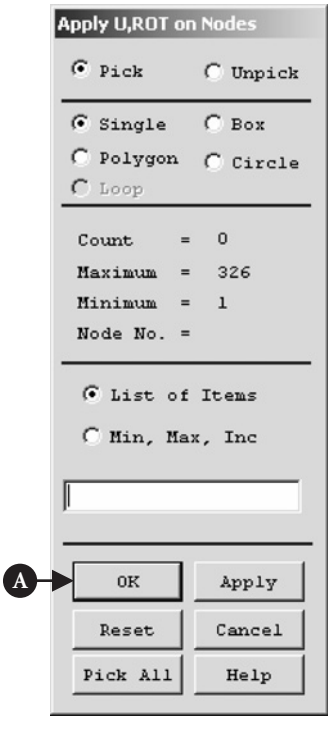


Figure 7.213 Apply U,ROT on Nodes.

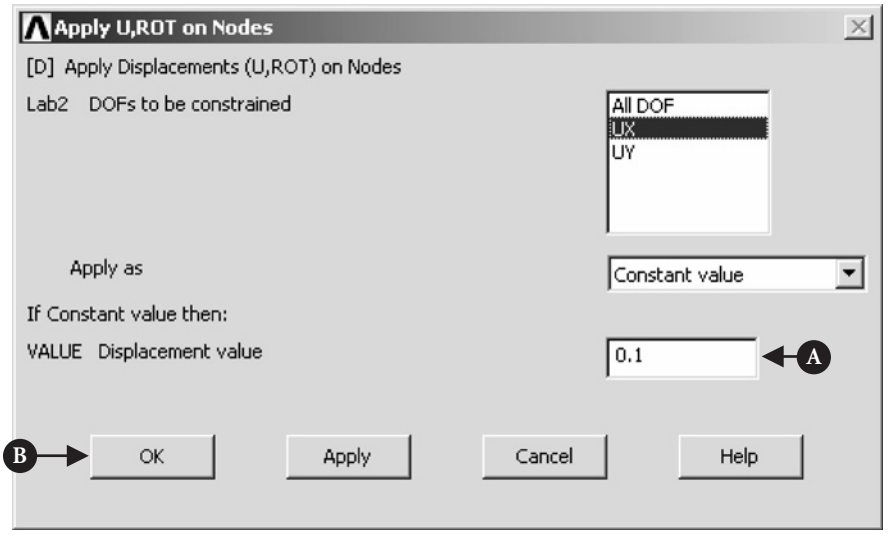


Figure 7.214 Apply U,ROT on Nodes.

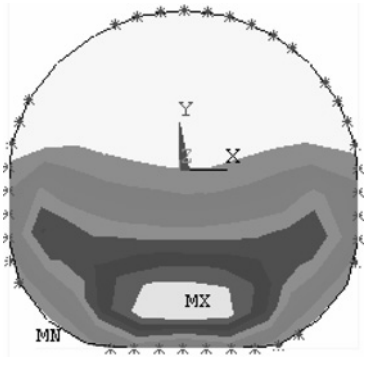


Figure 7.215 von Mises stress due to squeeze and external pressure (deformed shape only).

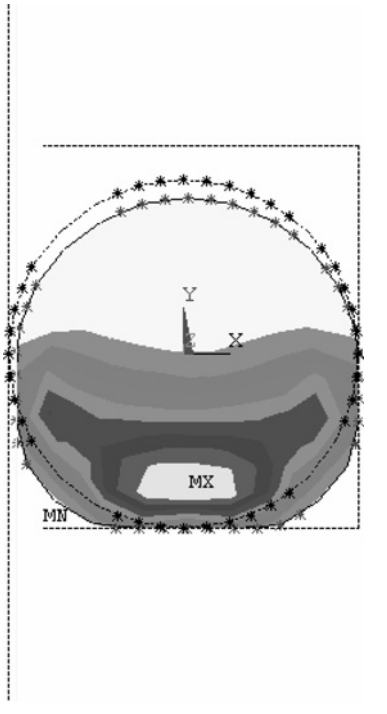


Figure 7.216 von Mises stress due to squeeze and external pressure (deformed shape and undeformed edge).

Now, the solution stage ought to be initiated. From **ANSYS Main Menu** select **Solution → Solve → Current LS**. The frame shown in Figure 7.198 appears together with another frame, which gives a summary of solution options selected. After checking the correctness of information, it should be closed by selecting **File → Close**. Next, [A] **OK** should be pressed to initiate the solution.

7.2.4.9 POSTPROCESSING (SECOND LOAD STEP)

As in the case of the first load step, from **ANSYS Main Menu** select **General Postproc → Read Results → By Load Step**. Figure 7.199 shows the resulting frame. Pressing [A] **OK** button initiates reading results produced during solution stage.

From **ANSYS Main Menu** select **General Postproc → Plot Results → Contour Plot → Nodal Solu**. The frame shown in Figure 7.200 appears. As before, **Stress** and **von Mises SEQV** options should be selected. Clicking [A] **OK** results in plot shown in Figure 7.215.

In order to see, simultaneously, deformed and undeformed shapes, a [B] **Def + undef edge** should be activated in the frame shown in Figure 7.200 and [A] **OK** tab pressed. The resulting image is shown in Figure 7.216.

REFERENCES

1. K. L. Johnson, *Contact Mechanics*, Cambridge University Press (1985).

This page intentionally left blank

INDEX

2D elastic 3, 146
2D FLOTRAN 141, 217

A
Acoustic problems, 37
Adiabatic, 231
Animate, 211
Anisotropic, 46
Apply PRES, 228
Apply VELO load, 227
Arbitrary path, 328
Area, 110, 111, 112, 114, 121, 122, 123, 124, 125
  add —s, 80
  display — number, 84
  subtract —, 82, 83
  subtract the elliptic —, 97
Area moment of inertia of the cross section, 53
Arm, 332, 338

B
Beam, 146
BEAM3, 147
Bending stress, 74
Body force, 16, 21, 25, 26, 31
Boolean operation, 80, 82, 94, 287, 360, 385
Booleans, 175
Boundary conditions, 276
Brick 8node, 189
Butterfly valve, 242

C
Cantilever beam, 144
Cauchy relation, 20
Center-cracked tension plate, 106, 107, 110, 114, 115, 116, 118
Command files, 38
Conduction, 264
Conductivity, 289
Constitutive relation, 17, 25
Constraint condition
  apply —, 99, 113, 133
  clear —, 66
  delete the —, 66
  definition of —, 63
  imposing of —, 65, 88, 99, 113, 133
Constraints, 49
Contact, 345, 346
  manager, 345, 346, 372
  pair, 370, 398, 405
  pick, 373, 398
  pressure, 355, 358, 384, 411
  properties, 376
  wizard, 345, 346, 370, 372
Contact stress (see also Hertz contact stress), 51, 120, 121, 123, 125, 127, 129, 131, 133, 135, 136, 137, 138, 139, 140
Contour plot, 355, 381, 408
Convection, 264
  coefficient, 286
  film, 290
Convective lines, 276
Convergence criteria, 439
Coordinate system, 306
  Cartesian, 444
  polar, 444
Coupled field loads, 49
Crack, 27, 106, 107, 108, 109, 110, 111, 112, 113, 114, 115, 116, 117, 118, 119, 120, 142
Create, 317
  block, 334
  component, 431, 433
  cylinder, 335, 362, 363
  elements, 437
  nodes, 429
  rectangle, 317, 386
Create Area thru KPs, 171
Current LS, 159
Cylindrical coordinates, 305

D
Database, 41
Define Loads, 154
Define Path, 237
Deformed Shape, 161, 442

Delete volume, 296, 362
Density, 149
Differential equation, 1, 2, 3, 5, 6, 14, 15, 19, 21
Diffuser, 216
DOF, 49

E
Edge option, 309
Effective stress, 79
Eight-node isoparametric element, 57, 59, 86, 98, 110, 123, 124
Elastic 8node, 165
Electromagnetic problems, 37
Element, 1, 2, 3, 4, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 18, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 34, 35, 37, 51, 52, 53, 57, 58, 59, 60, 61, 62, 74, 76, 77, 78, 79, 84, 86, 87, 88, 93, 98, 99, 107, 110, 111, 112, 113, 123, 124, 125, 130131 132 133 134 135 136
  attributes, 46, 429, 435, 436
  edges, 371
  size, 60, 61, 77, 78, 79, 87, 98, 111, 124, 125, 342, 395, 427
  surface, 357
  type, 37, 44, 340, 365
Element matrix, 12, 13
Element stiffness matrix, 15, 26, 27, 28, 30, 31
Entities, 303, 350, 356
Equations of equilibrium, 16, 18, 19, 21, 25
Equivalent nodal force, 25, 26, 31
Equivalent stress, 81
Extrude, 201
Extrude area, 320, 389, 390

F
Finite difference method, 1
Finite element, 37
Finite element method, 1, 2, 4, 6, 7, 8, 9, 10, 11, 12, 13, 14, 16, 18, 20, 22, 24, 26, 28, 30, 32, 34, 35, 52
Finite volume method (see also sub-domain method), 2
FLOTRAN, 215
FLOTRAN set up, 231
Fluid problems, 37
Fluid properties, 232
Four-node linear rectangular element, 59
Functional, 6, 7, 8

G
Galerkin method, 4, 5, 7, 8, 9
General postprocessor, 50
Generalized Hooke's law, 17, 18
Geometric properties, 37
Geometrical boundary condition, 15, 20, 21
Global coordinate, 8
Global matrix, 13
Global stiffness equations, 27, 29
Glue, 177, 273
Graphical representation, 73
Graphical user interface (GUI), 38, 385
Graphics window, 39
GUI see Graphical user interface

H
Hard copy, 42
Heat flow, 280
Heat flux, 280, 323
Heat transfer, 37, 263, 264
Hertz contact stress (see also contact stress), 136
Hyper-elastic material, 410, 412, 416

I
In Active CS, 149
Incompressible, 231
Input line, 39
Interference fit, 336
Internal energy, 263, 264
International system of units (SI), 44
Interpolation function, 8, 9, 10, 11, 15, 22, 24
Isida's solution, 118
Isometric view, 389
Isotropic, 46, 47, 268, 149

J
Jobname, 40

K
Keypoints, 149
Kinetic energy, 264

L
Launcher, 40
Library of elements, 367, 393, 413
Line contact, 382
Linear isotropic, 417
Lines, 151
List results, 42

Load, 49
apply, 403
body, 49
convection, 300
inertia, 49
step, 49, 358, 443
surface, 49
Local coordinate, 9
Log file, 41

M
Main window, 38
Map onto Path, 237
Material, 37
model, 147, 416, 418, 421
properties, 37, 46, 339
Material Props, 168
Mechanical boundary condition, 15, 20, 21, 25
Mesh, 37
areas, 37, 428
generation, 342, 369
lines, 37
tool, 197, 341, 368, 394
volumes, 37, 364
Meshing, 152, 273
process, 396
Method of weighted residuals, 1, 2, 7, 35
Modal Analysis, 157
Mode I, 106, 116, 118, 119
Mooney-Rivlin, 412, 416
Move volume, 337
Moving table, 188

N
Nodal, 379
contour, 381, 443
solution, 379, 405, 408
Nodal displacement, 22, 23, 24, 29
Node, 47, 129, 130, 131, 138
reselection of —, 129
selection of —, 63, 65, 130
Nonlinear, 46
Notch, 100, 101, 102

O
Offset WP (work plane), 293, 387, 388
Operate, 201
Orthotropic, 46
Output window, 38
Overlap volumes, 294, 335, 362

P
Package, 37
Pan-Zoom-Rotate, 171
Path name, 259
Path operation, 237
Pin, 332, 338, 355
Plane strain approximation, 18
Plot results, 42, 161
PlotCtrls, 171
Poisson's ratio, 18, 19, 53, 56, 57, 78, 79, 85, 86, 94, 98, 106, 110, 120, 123, 138, 140, 149, 365, 391
Postprocessing, 38, 404
Preferences, 365
PREP7 preprocessor, 43
Preprocessing, 37
Primitive shape, 291
Principle of the minimum potential energy, 8
Principle of virtual work, 21
Print, 42
Properties
elastic —, 56, 86, 97, 110, 123
linear isotropic —, 57, 86, 98, 110, 123
— of the materials, 17

R
Rayleigh-Ritz method, 1, 4, 5, 6, 7
Read results, 161, 354
Real constants, 147, 421, 424
Rectangular area, 270
Reference number, 46
Reorder elements, 437
Residual, 1, 2, 3, 6, 7, 8, 25, 35
Resonant frequency, 144
Restore WP, 389
Resume, 40

S
Select, 382
entities, 382, 409, 431, 446
nodes, 433
Shape function, 8, 24
Shear modulus, 18, 19
Shell, 165
Size Cntrls, 152
Solid, 189
Solid Circle Area, 173
Solution, 38, 49
controls, 350, 378, 438, 449
nodal, 354
processor, 48
stage, 298

Space, 222
Specific heat, 289
Static/dynamic, 37
Stationary condition, 6, 7
Stefan-Boltzmann law, 264
Straight bar, 144
Straight Lines, 152
Strain, 14, 15, 16, 17, 18, 19, 21, 22, 24, 25, 26, 31, 34, 74, 107, 110, 117, 124
Strain-displacement
— matrix, 21, 24
— relation, 14, 15, 16
Stress analysis, 51
Stress intensity factor, 104, 106, 116, 118, 119, 141
Stress singularity, 51, 106, 107, 109, 111, 113, 115, 117, 119
Stress–strain matrix, 25
Stress–strain relation, 14, 15, 17, 18
Structural analysis, 37
Sub-domain method (see also finite volume method), 2, 4
Subspace, 157
Substeps, 49
Subtract, 175, 270
area, 270, 318, 387
volumes, 337
Suspension, 163
Symbols, 310
Symmetry, 353
constraints, 374, 377, 403
expansion, 353, 379

T
Target, 346, 373
pick, 373, 398
Temperature, 263
constraints, 302, 304
distribution, 287
gradient, 329
map, 309, 311
Thermal, 264
conductivity, 264, 267, 312, 315
flux, 312
Time history postprocessor, 50
Toolbar, 39
Traction, 20, 21, 25
Triangular constant strain element, 2, 24
Turbulent, 231

U
Undeformed shape, 442
Units, 43
Utility menu, 39, 40

V
Vector Arrow Scaling, 256
Vector plot, 312
View setting, 295
Viewing direction, 365, 406
Virtual displacement, 25, 26
Volume sweeping, 343, 370
von Mises stress, 78, 79, 381, 405, 444, 451

W
Weighting function, 2, 4, 9
Working plane (WP), 283, 292, 305
WP settings, 445

X
XYZ Offset, 204

Y
Young's modulus, 18, 19, 53, 56, 57, 78, 79, 85, 86, 94, 98, 106, 110, 117, 120, 123, 138, 140, 149, 365, 391