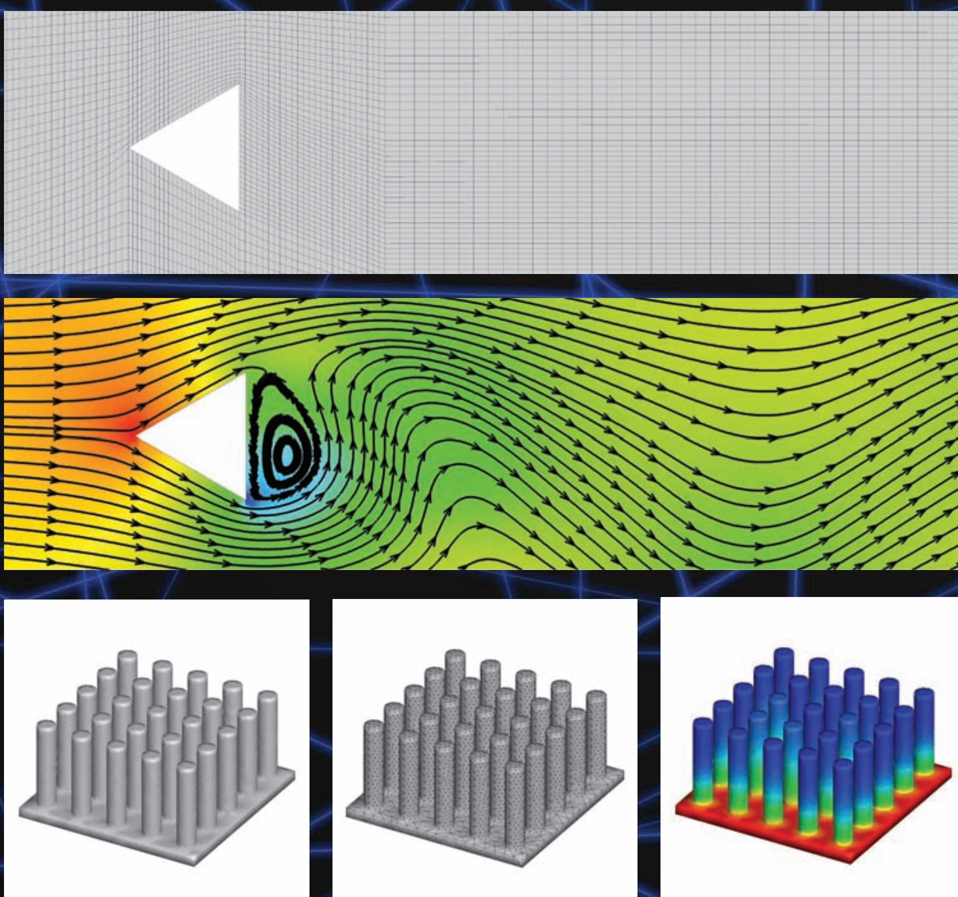
Finite Element Simulations Using ANSYS

Second Edition



Esam M. Alawadhi

CRC Press
Taylor & Francis Group

Finite Element Simulations Using ANSYS

Second Edition

This page intentionally left blank

Finite Element Simulations Using ANSYS

Second Edition

Esam M. Alawadhi

Kuwait University
Safat, Kuwait

CRC Press
Taylor & Francis Group
Boca Raton London New York

CRC Press is an imprint of the
Taylor & Francis Group, an **informa** business

CRC Press
Taylor & Francis Group
6000 Broken Sound Parkway NW, Suite 300
Boca Raton, FL 33487-2742

Version Date: 20150728

International Standard Book Number-13: 978-1-4822-6198-1 (eBook - PDF)

Visit the Taylor & Francis Web site at
http://www.taylorandfrancis.com

and the CRC Press Web site at
http://www.crcpress.com

Contents

This page intentionally left blank

Preface

Due to the complexity of modern-day problems in mechanical engineering, engineers, in practice, seldom rely only on theory or experiments. The use of engineering software is becoming prevalent among academics as well as practicing engineers. Commercial finite element software such as ANSYS (ANSYS, Canonsburg, Pennsylvania) and Abaqus (Dassault Systèmes, Vélizy Villacoublay, France) for analysis and design is commonly found in use in universities, industry, and research centers. This software has become more reliable, reputable, easy to use, and trustworthy, and it allows much time to be saved.

This book focuses on the use of ANSYS in solving practical mechanical engineering problems. ANSYS is extensively used in the design cycle by industry leaders in the United States and around the world. Additionally, ANSYS is available in computer labs for students in most universities around the world. Courses such as computer-aided design, modeling, and simulation, and major design courses all utilize ANSYS as a tool for analyzing various mechanical components. Graduate students also use ANSYS in their finite element courses as a complement for the theoretical study of the finite element method.

This book provides mechanical engineering students and engineers with the fundamental knowledge of numerical simulation using ANSYS. The book serves most of the disciplines of mechanical engineering: structure, solid mechanics, vibration, heat transfer, and fluid dynamics, with adequate background material to explain the physics behind the computations. Each physical phenomenon is treated independently in a way that enables readers to pick out a single subject or a related chapter and study it. Instructors can cover appropriate chapters depending on the objectives of the course. The required basic knowledge of the finite element method relevant to each physical phenomenon is illustrated at the beginning of the respective chapter. The general theory of the finite element, however, is presented briefly and concisely because the theory is well documented by other finite element books.

For example, in the heat transfer chapter, the theory is first explained, the governing equations are derived, the modeling techniques are presented, and finally practical problems are solved using ANSYS in a step-by-step technique. Each chapter independently discusses a single physical phenomenon, while the last chapter is devoted to multiphysics analyses and problems. The finite element solution is greatly affected by the quality of the mesh, and therefore, a separate chapter on meshing is included as a guide that emphasizes the basics of the meshing techniques. Practical end-of-chapter problems are provided in each chapter to challenge the reader's understanding.

Undergraduate and graduate engineers will use this book as a part of their courses, either when studying the basics of applied finite elements, or in mastering practical tools of engineering modeling. Engineers in industry can use this book as a guide for better design and analysis of their products. In all mechanical engineering curricula, junior- and senior-level courses use some type of engineering modeling software, and, oftentimes, this software is ANSYS. Senior students also use ANSYS in their senior design projects. Graduate-level finite element courses frequently use ANSYS to complement the theoretical analysis of finite elements. The course that uses this book should be taken after the introduction to design courses and the basic thermal–fluid courses. Courses such as that in senior design can be taken after this course.

In this second edition of the book, new sections are added, and ANSYS examples are modified to be in compliance with the new version of ANSYS. Most ANSYS examples in the first edition are replaced by more general, comprehensive, and easy-to-follow examples. In the finite element theoretical part, more details are added, especially for the heat transfer chapter. Additionally, open-ended problems are added at the end of each chapter, which can serve as class projects.

Acknowledgments

I profited greatly from discussion with faculty members and engineers at Kuwait University. I want to mention particularly Professor Ahmed Yigit.

This page intentionally left blank

About the author

Esam M. Alawadhi is a professor of mechanical engineering at Kuwait University, Kuwait City, Kuwait. He earned his doctor of philosophy in mechanical engineering from Carnegie Mellon University, Pittsburgh, Pennsylvania in May 2001. His research focuses on renewable energy, thermal management of electronics devices, energy conservation for buildings, fluid flow stability, and phase-change heat transfer.

This page intentionally left blank

CHAPTER ONE

Introduction to finite element method and ANSYS

1.1 The finite element method and structural analysis

The finite element method was developed in the 1950s and has been continuously enhanced since then. Rapid advances in computing power and the drastic drop in cost make the finite element method affordable. Now, it is a commonly used method for solving a wide range of problems, and the potential of the finite element method is enormous. The finite element method is typically found in the aerospace, automotive, electrical, hydraulic, biomedical, nuclear, and structural engineering fields, among many others. The first step in the finite element solution procedure is to divide the domain into elements, and this process is called the *domain discretization*. The variable distribution across each element can be defined by linear, quadratic, or trigonometric function. The elements distribution in the domain is called the *finite element mesh*. The elements are connected at points called *nodes*. For example, consider a section of a bridge, as shown in Figure 1.1a. The bridge is divided into linear elements and connected by nodes, as shown in Figure 1.1b.

After the region is discretized, the governing equations for the element must be established for the required physics. Material properties, such as modulus of elasticity for structural analysis, should be available. The equations are assembled to obtain the global equation for the mesh, which describes the behavior of the body as a whole. Generally, the global governing equation has the following form:

$$[K]\{U\} = \{F\} \tag{1.1}$$

where [K] is called the stiffness matrix; {U} is the nodal degree of freedom, such as the displacements for structural analysis; and {F} is the nodal external force, such as forces for structural analysis. The [K] matrix is a singular matrix, and therefore it cannot be inverted.

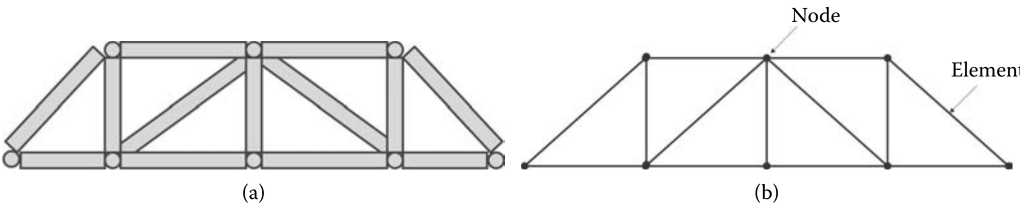


FIGURE 1.1 (a) A section of a bridge and (b) finite element mesh.

Consider a one-dimensional bar with initial length L that is subjected to a tensile force at its ends, as shown in Figure 1.2a. The cross-sectional area of the bar is A. The element can be modeled using a single element with two nodes, i and j, as shown in Figure 1.2b.

Assume that the displacement function of the bar u(x) varies linearly along the length of the bar. The expression of the displacement is represented as

$$u(x) = a + bx \tag{1.2}$$

The displacement at nodes i and j are u_i and u_j, respectively. Then

$$u_i(x) = a + bx_i \tag{1.3}$$

$$u_j(x) = a + bx_j \tag{1.4}$$

where x_i is the x-coordinate for node i, and x_j is the x-coordinate for node j. Solving for constants a and b, it is found that

$$a = (u_i x_j - u_j x_i)/L \tag{1.5}$$

$$b = (u_j - u_i) / L \tag{1.6}$$

where L is the initial length of the element and is equal to $(x_i - x_j)$. Substituting constants a and b into the displacement equation (Equation 1.2) and rearranging, the displacement function becomes

$$u(x) = \frac{x_j - x}{L} u_i + \frac{x - x_i}{L} u_j \tag{1.7}$$

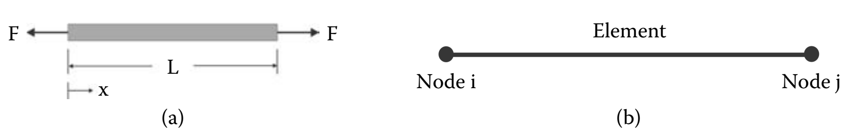


FIGURE 1.2 A one-dimensional bar element. (a) A single one-dimensional bar element and (b) an element with two nodes.

or

$$u(x) = N_i u_i - N_j u_j \tag{1.8}$$

where N_i and N_j are called the *shape functions* of the element. When the element is loaded, it will be in an equilibrium position. The sum of the strain energy γ and work w done by external forces is the potential energy π of the element. The potential energy at the equilibrium position has the minimum value, and the potential energy is defined as

$$\pi = \gamma - w \tag{1.9}$$

For a single bar element, the strain energy in the bar is given by

$$\gamma = \frac{1}{2}\int_{x_i}^{x_j} \sigma \varepsilon A \, dx \tag{1.10}$$

Since the strain is related to the stress by the modulus of elasticity ($\sigma = E\varepsilon$), the strain energy is expressed as

$$\gamma = \frac{AE}{2}\int_{x_i}^{x_j} \varepsilon^2 dx \tag{1.11}$$

The strain is equal to the elongation of the element in the x-direction

$$\varepsilon = (u_j - u_i) / L \tag{1.12}$$

Then, the strain energy in Equation 1.11 becomes

$$\gamma = \frac{AE}{2L}\left(u_j - u_i\right)^2 \tag{1.13}$$

The strain energy can be expressed in the matrix form as follows:

$$\gamma = \frac{AE}{2L}\begin{bmatrix} u_i & u_j \end{bmatrix}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \frac{1}{2}\{U\}^T [K]\{U\} \tag{1.14}$$

where [K] is the stiffness matrix for the element. The external forces are forces acting at the nodes. The work done by the applied forces at the nodes is expressed as

$$w = u_i F_i + u_j F_j = \{U\}^T\{F\} \tag{1.15}$$

Finally, the work done and strain energy are substituted into the potential energy for a single element. The potential energy is expressed as

$$\pi = \frac{1}{2}\{U\}^T[K]\{U\} - \{U\}^T\{F\} \tag{1.16}$$

For the minimum potential energy, the displacement must be equal to zero:

$$\frac{\partial \pi}{\partial \{U\}} = 0 \tag{1.17}$$

or

$$\frac{\partial \pi}{\partial u_i} = 0 \quad \text{and} \quad \frac{\partial \pi}{\partial u_j} = 0 \tag{1.18}$$

Equation 1.16 is differentiated using the expressions (1.18), and the results are

$$\frac{AE}{L}\left(u_i - u_j\right) - F_i = 0 \tag{1.19}$$

$$\frac{AE}{L}\left(-u_i + u_j\right) - F_j = 0 \tag{1.20}$$

Equations 1.19 and 1.20 are expressed in the matrix form as follows:

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} - \begin{Bmatrix} F_i \\ F_j \end{Bmatrix} = 0 \tag{1.21}$$

and symbolically,

$$[K]\{U\} - \{F\} = 0 \tag{1.22}$$

where

$$[K] = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{1.23}$$

The derivation of the stiffness matrix is valid only for one bar element. In practice, a model consists of many elements of different properties. The total potential energy of a number of elements is

$$\pi = \sum_{e=1}^{E}\left(\gamma^e - w\right) \tag{1.24}$$

Minimizing Equation 1.24, the result is

$$\sum_{e=1}^{E}\left[K^e\right]\{U\}-\{F\}=0 \tag{1.25}$$

1.2 Stress analysis of pin-jointed bar

Use the finite element method to calculate the maximum displacement of the bar assembly shown in Figure 1.3a. Given that A_1 = 90 mm^2, A_2 = 75 mm^2, A_3 = 50 mm^2, E_1 = 100 GPa, E_2 = 110 GPa, and E_3 = 180 GPa.

Before starting to solve the problem, all bar elements and nodes of the assembly should be numbered, as suggested in Figure 1.3b.

There are three main steps for solving this problem using the finite element method: (1) the stiffness matrix for all bar elements is determined, and then stiffness matrices are assembled to obtain the global stiffness matrix; (2) the boundary conditions are applied to the global stiffness matrix; and (3) rows and a column of the global stiffness matrix are eliminated to remove the singularity of the stiffness matrix, and finally we solve for the displacements.

Step 1: Construct the stiffness matrix for all bar elements

The stiffness matrix for element 1, which has nodes 1 and 2, is calculated using matrix (1.23):

$$\left[K^{(1)}\right]=\frac{90\times10^{-6}\left(100\times10^{9}\right)}{30\times10^{-3}}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=10^{8}\begin{bmatrix}3 & -3\\ -3 & 3\end{bmatrix}$$

The stiffness matrix for element 2, which has nodes 2 and 3, uses

$$\left[K^{(2)}\right]=\frac{75\times10^{-6}\left(110\times10^{9}\right)}{55\times10^{-3}}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=10^{8}\begin{bmatrix}1.5 & -1.5\\ -1.5 & 1.5\end{bmatrix}$$

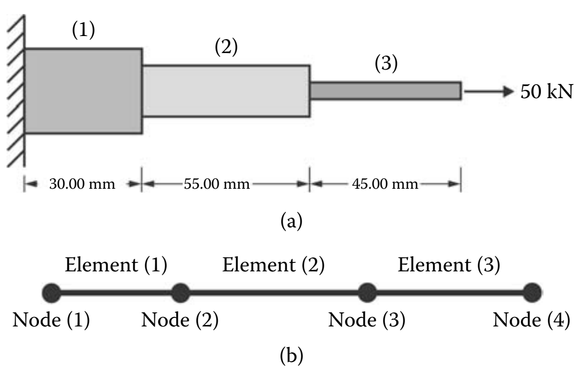


FIGURE 1.3 (a) Bar assembly. (b) Elements and nodes for the bar assembly.

The stiffness matrix for element 3, which has nodes 3 and 4, uses

$$\left[K^{(3)}\right] = \frac{50\times10^{-6}\left(180\times10^{9}\right)}{45\times10^{-3}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^{8}\begin{bmatrix} 2 & -2 \\ -2 & 2 \end{bmatrix}$$

Assembling the elements' stiffness matrices to form the global stiffness matrix,

$$[K] = \sum_{e=1}^{e=3}\left[K^{(e)}\right]$$

$$[K] = 10^{8}\begin{bmatrix} 3 & -3 & 0 & 0 \\ -3 & 4.5 & -1.5 & 0 \\ 0 & -1.5 & 3.5 & -2 \\ 0 & 0 & -2 & 2 \end{bmatrix}$$

Step 2: Apply the boundary conditions to the global stiffness matrix using Equation 1.22

$$10^{8}\begin{bmatrix} 3 & -3 & 0 & 0 \\ -3 & 4.5 & -1.5 & 0 \\ 0 & -1.5 & 3.5 & -2 \\ 0 & 0 & -2 & 2 \end{bmatrix}\begin{Bmatrix} u_1 = 0 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 = 0 \\ F_3 = 0 \\ F_4 = 50 \end{Bmatrix}$$

Step 3: Eliminate the first row and column from the global stiffness matrix to remove the singularity

$$10^{8}\begin{bmatrix} 4.5 & -1.5 & 0 \\ -1.5 & 3.5 & -2 \\ 0 & -2 & 2 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 50 \end{Bmatrix}$$

Solve for displacements:

$u_2 = 1.667 \times 10^{-7}$ m

$u_3 = 5.0 \times 10^{-7}$ m

$u_4 = 7.5 \times 10^{-7}$ m (Maximum)

1.3 The finite element method and thermal analysis

Heat transfer in a one-dimensional space can be easily analyzed using the finite element method. Consider a wall that is made by common bricks, as shown in Figure 1.4. The outdoor surface temperature is T_i, while the indoor surface temperature is T_j.

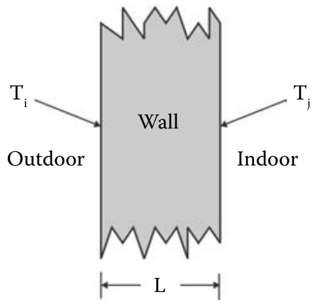


FIGURE 1.4 One-dimensional heat conduction in a wall.

The Fourier law indicates that the heat flow across the wall Q is proportional to temperature gradient dT/dx in the direction of heat transfer. That is,

$$Q = -kA\frac{dT}{dx} \tag{1.26}$$

where k is the thermal conductivity of the wall and A is the area perpendicular to the heat flow. For steady-state conditions, the temperature varies linearly along the length of the wall as follows:

$$\frac{dT}{dx} = \frac{T_j - T_i}{L} \tag{1.27}$$

Hence, the heat flow across the indoor and outdoor surfaces of the wall can be expressed as, respectively,

$$Q_i = -kA\frac{T_j - T_i}{L} \tag{1.28}$$

$$Q_j = -kA\frac{T_i - T_j}{L} \tag{1.29}$$

Equations 1.28 and 1.29 are expressed in the following matrix form:

$$\frac{kA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} T_i \\ T_j \end{Bmatrix} - \begin{Bmatrix} Q_i \\ Q_j \end{Bmatrix} = 0 \tag{1.30}$$

This is analogous to the bar element equation (Equation 1.21). The nodal forces are substituted by nodal heat flow, nodal displacements by nodal temperatures, and constant AE/L by kA/L; therefore,

$$[K]\{T\} - \{Q\} = 0 \tag{1.31}$$

where

$$[K] = \frac{kA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{1.32}$$

1.4 Heat transfer through a composite wall

Consider a wall that is constructed by stone bricks at the outdoor surface ($k = 1.25$ W/m · °C), common brick ($k = 0.75$ W/m · °C), and plaster at the indoor surface ($k = 0.9$ W/m · °C), as shown in Figure 1.5. The outdoor surface temperature is $T_o = 45$°C, while the indoor surface temperature is $T_i = 25$°C. Determine the temperature at the interfaces, and the results should be per unit area of the wall.

Step 1: Construct the stiffness matrix for all elements

The stiffness matrix for element 1, which has nodes 1 and 2, is calculated using expression (1.32):

$$\left[K^{(1)}\right] = \frac{1.25 \times 1}{5 \times 10^{-2}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \begin{bmatrix} 25 & -25 \\ -25 & 25 \end{bmatrix}$$

The stiffness matrix for element 2, which has nodes 2 and 3, uses

$$\left[K^{(2)}\right] = \frac{0.75 \times 1}{15 \times 10^{-2}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \begin{bmatrix} 5 & -5 \\ -5 & 5 \end{bmatrix}$$

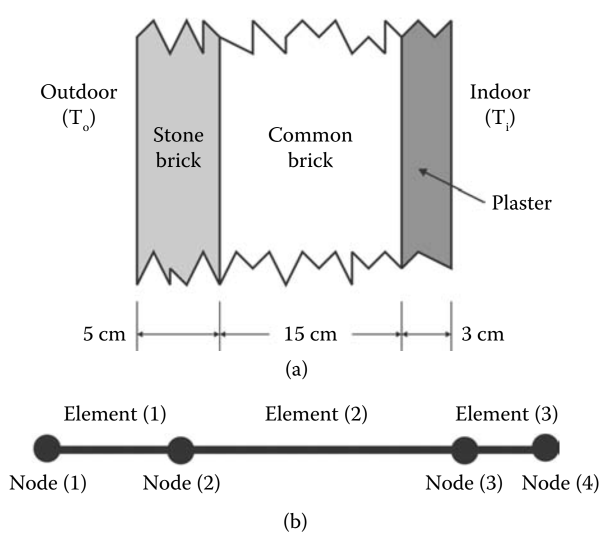


FIGURE 1.5 The wall composition and finite element mesh. (a) The wall composition and (b) the finite element presentation.

The stiffness matrix for element 3, which has nodes 3 and 4, uses

$$\left[K^{(3)}\right]=\frac{0.9\times 1}{3\times 10^{-2}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}=\begin{bmatrix} 30 & -30 \\ -30 & 30 \end{bmatrix}$$

Assembling the elements' stiffness matrices to form a global stiffness matrix,

$$[K]=\sum_{e=1}^{e=3}\left[K^{(e)}\right]$$

$$[K]=10^{8}\begin{bmatrix} 25 & -25 & 0 & 0 \\ -25 & 30 & -5 & 0 \\ 0 & -5 & 35 & -30 \\ 0 & 0 & -30 & 30 \end{bmatrix}$$

Step 2: Apply the boundary conditions to the global stiffness matrix using Equation 1.31

$$\begin{bmatrix} 25 & -25 & 0 & 0 \\ -25 & 30 & -5 & 0 \\ 0 & -5 & 35 & -30 \\ 0 & 0 & -30 & 30 \end{bmatrix}\begin{Bmatrix} T_1 = 45 \\ T_2 \\ T_3 \\ T_4 = 25 \end{Bmatrix}=\begin{Bmatrix} Q_1 \\ Q_2 = 0 \\ Q_3 = 0 \\ Q_4 \end{Bmatrix}$$

Then Q_1 and Q_2 are required to maintain the constant temperature at the external surfaces.

Step 3: Eliminate the rows and columns, numbers 1 and 4, from the global stiffness matrix to remove the singularity from the stiffness matrix

The temperatures at nodes 1 and 4 are known. Hence, only Equations 2 and 3 are needed to solve the problem:

$$\begin{bmatrix} 25 & -5 \\ -5 & 35 \end{bmatrix}\begin{Bmatrix} T_2 \\ T_3 \end{Bmatrix}=\begin{Bmatrix} 25\times 45 \\ 30\times 25 \end{Bmatrix}$$

Solving for temperatures,

$T_2 = 42.07°C$

$T_3 = 27.44°C$

1.5 Introduction to ANSYS software

The ANSYS software is the most advanced package for single- and multiphysics simulations, offering enhanced tools and capabilities that enable engineers to complete their jobs in an efficient manner. ANSYS includes significant capabilities, expanding functionality, and integration with almost all computer-aided design (CAD) drawing software, such as pro-engineers, AutoCAD, and Solid Edge. In addition, ANSYS has the best-in-class solver technologies, a coupled physics for complex simulations, meshing technologies customizable for physics, and computational fluid dynamics.

ANSYS can solve problems in structural, thermal, fluid, acoustics, and multiphysics:

Structural:

- Linear
- Geometric and material nonlinearities
- Contact
- Static
- Dynamic
- Transient, natural frequency, harmonic response, response spectrum, random vibration
- Buckling
- Topological optimization

Thermal:

- Steady state or transient
- Conduction
- Convection
- Radiation
- Phase change

Fluid mechanics:

- Steady state or transient
- Incompressible or compressible
- Laminar or turbulent
- Newtonian or non-Newtonian
- Free, forced, or mixed convection heat transfer
- Conjugate solid/fluid heat transfer
- Surface-to-surface radiation heat transfer

- Multiple species transport
- Free surface boundaries
- Fan models and distributed resistances
- Stationary or rotating reference frames

Acoustics:

- Fully coupled fluid/structural
- Near and far field
- Harmonic, transient, and modal

Multiphysics:

- Thermal–mechanical
- Thermal–electric
- Thermal–electric–structural
- Piezoelectric
- Piezoresistive
- Peltier effect
- Thermocouple effect
- Electromechanical circuit simulator

ANSYS is not a single program, but it is a family of CAD programs, as shown in Figure 1.6. The ANSYS/Multiphysics package consists of ANSYS/Emag, which is for magnetic field analysis, ANSYS/Flotran for fluid mechanics, ANSYS/LS-DYNA for dynamics analysis, and ANSYS/Mechanical for structural and thermal analyses.

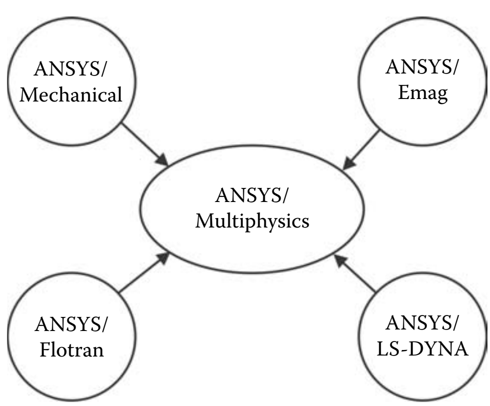


FIGURE 1.6 The ANSYS family.

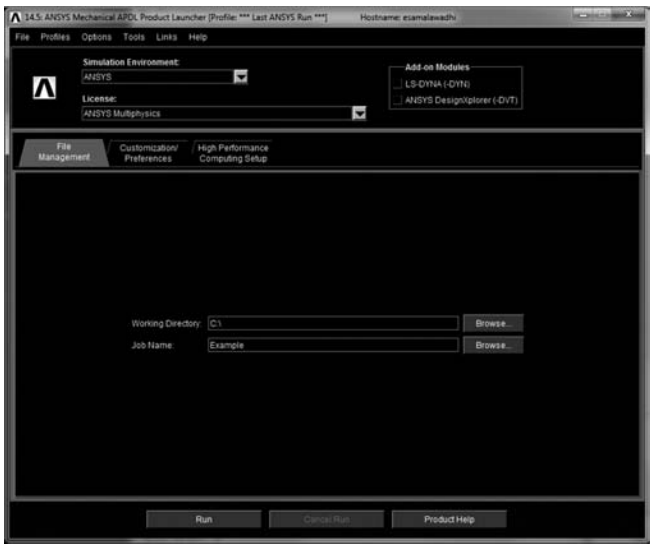
To start ANSYS:

Double click on the Mechanical APDL Product Launcher icon in the desktop

or, go to

Start > All Programs > ANSYS > Mechanical APDL Product Launcher

The ANSYS Product Launcher window will appear as shown. First, select the ANSYS Multiphysics in License. Then, type the working directory path for ANSYS files in the Working Directory. All ANSYS files will be stored in this directory, including images and AVI movies. The session name is specified in the Job Name. Finally, click on Run to start the ANSYS.



A select ANSYS Multiphysics in License

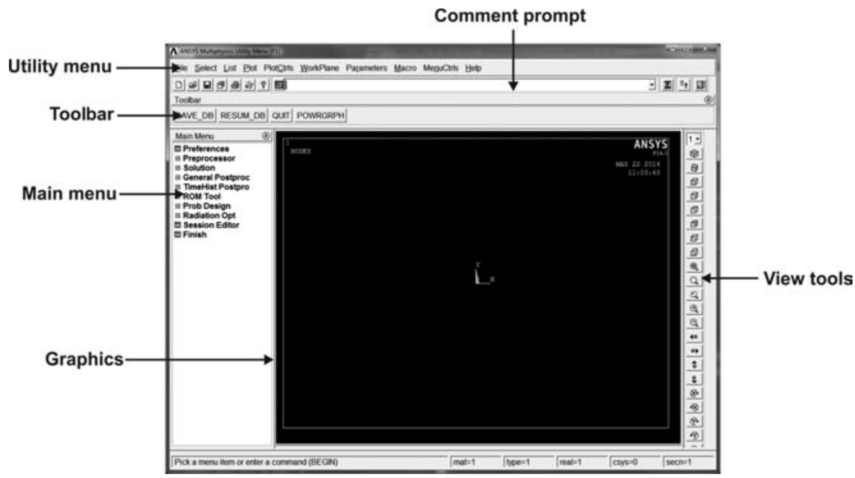
B change Working Directory to C:\, or any directory

C change the initial Job Name to Example, or any name

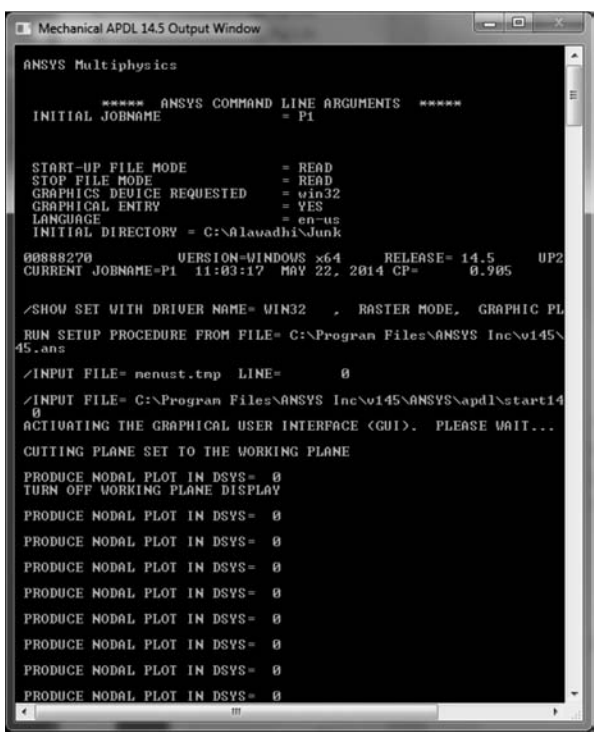
Run

The ANSYS interface has the following components:

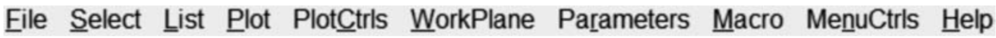
- Utility menu
- Toolbar
- Main menu
- View tools
- Graphics
- Comment prompt



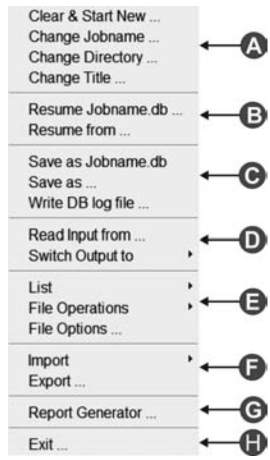
In addition, the ANSYS Output window will show up. The output window is dynamically listing important information during the preprocessor, solution, and postprocessor. Additionally, the calculated results in the postprocessor are shown in this window. Warnings in the ANSYS Output should be carefully considered to avoid unexpected errors.



Utility menu



The Utility menu is used for the file operations, listing and plotting, and changing display options. In the pull-down File menu, the following tasks can be performed:



A Clearing current job and starting a new job

B Resuming a job, and the database file can be located

C Saving ANSYS job

D Reading ANSYS input file

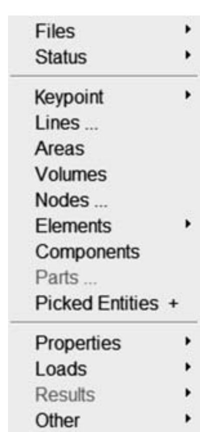
E Listing and file operations

F Importing and exporting files

G Generating a summary file

H Exiting ANSYS job

The pull-down List menu is for listing the model components, such as keypoints, areas, and nodes. In addition, the properties of the material can be listed. The boundary conditions imposed on the model can also be listed.



The pull-down Plot menu is similar to the List menu. Plotting geometry's components, such as keypoints and areas, can be performed in this menu. In the PlotCtrls menu, printing the model in the ANSYS graphics, changing the style of the ANSYS graphics, or changing the quality of the graphics can be done. The Workplane is for the grids setup.

ANSYS Main Menu

Most ANSYS jobs are done using the Main Menu, from building the model to viewing the results. In the preprocessor, the material properties and real constants are specified, element type is selected, and modeling operations and meshing tools are available. In the solution, the boundary conditions are imposed, and the solution setup parameters are specified. In the postprocessor, the ANSYS results are presented. List, plot results, and path operations can be performed.

The three tasks are summarized as follows:

Preprocessor:

1. Element type
2. Material properties
3. Real constants
4. Modeling
5. Meshing

Solution:

1. Boundary condition
2. Solution setup parameters

Postprocessor:

1. Plot results
2. List results
3. Path operation

PROBLEM 1.1

Use the finite element method to calculate the maximum displacement of the bar assemblies shown in Figures 1.7 to 1.9.

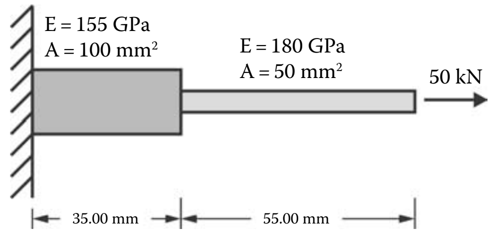


FIGURE 1.7 Two bars assembly with a tensile force for at one end.

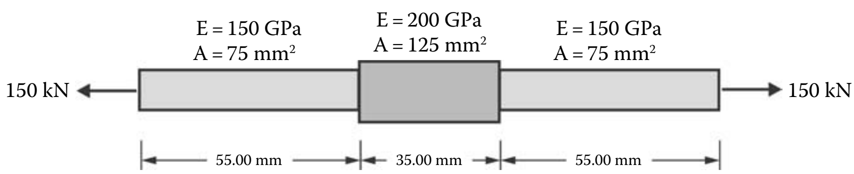


FIGURE 1.8 Three bars assembly with a tensile force for at both ends.

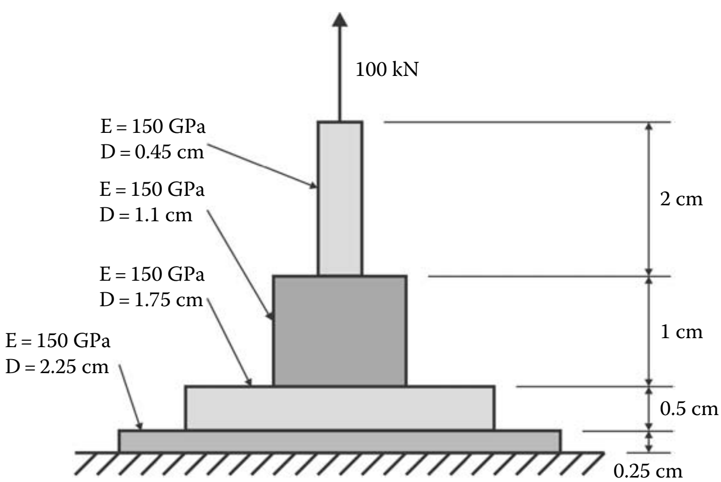


FIGURE 1.9 A structure with a vertical tensile force.

PROBLEM 1.2

Consider a wall shown in Figure 1.10. The outdoor surface temperature is T_o = 42°C, while the indoor surface temperature is T_i = 23.5°C. Determine the temperature at the interfaces, and the results should be per unit area of the wall. The thermal conductivity of stone brick, thermal insulation, common brick, and indoor finishing is 1.25 W/m · °C, 0.85 W/m · °C, 0.2 W/m · °C, and 1.75 W/m · °C, respectively.

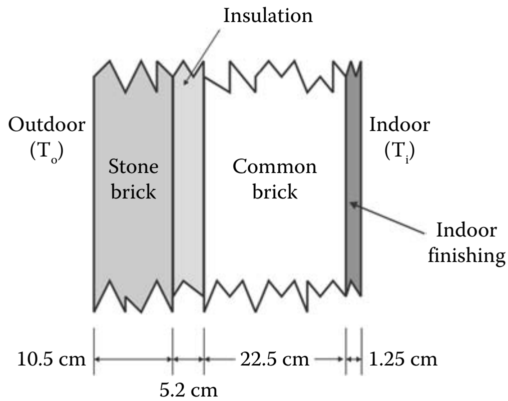


FIGURE 1.10 Heat transfer in a wall composed of four different materials.

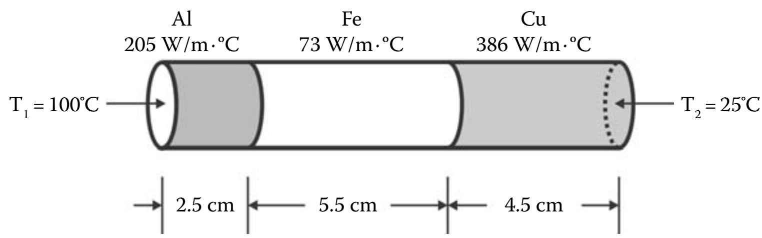


FIGURE 1.11 Heat transfer in a rod composed of three different materials.

PROBLEM 1.3

Consider a thermal rod that is composed of three different materials, as shown in Figure 1.11. The left circular area is maintained at a temperature of $T_1 = 100°C$, while the right one is maintained at a temperature of $T_2 = 25°C$. The entire lateral surface is well insulated. Determine the heat flow through the rod, if its diameter is 1.5 cm.

This page intentionally left blank

CHAPTER TWO

Bar and beam structures

2.1 Finite element method for a horizontal bar element

The bar structures are composed of straight bars and welded at the interactions. The bar structures can be found in bridges and roofs. The derivation of the stiffness matrix for a bar element is applicable to the pin-connected structures. The bar element is assumed to have a constant cross-sectional area A, uniform modulus of elasticity E, and initial length L. The bar is subjected to tensile forces along the local axis and applied at its ends. There are two coordinate systems: local $(\bar{x}, \bar{y})$ and global (x,y) coordinates. Figure 2.1 shows the local and global coordinate systems.

For each element, there are two local forces acting on the nodes: $\bar{F}_{1x}$ and $\bar{F}_{1y}$ are acting on node 1, and $\bar{F}_{2x}$ and $\bar{F}_{2y}$ are acting on node 2. The nodal degrees of freedom are the four local displacements, two at each node: $\bar{d}_{1x}$ and $\bar{d}_{1y}$ are at node 1, and $\bar{d}_{2x}$ and $\bar{d}_{2y}$ are at node 2. Figure 2.2 shows a bar element lying along the local x-coordinate.

The strain–displacement relationship is obtained from Hooke's law as follows:

$$\sigma_x = E\varepsilon_x \tag{2.1}$$

where

$$\varepsilon_x = \frac{d\bar{u}}{d\bar{x}} \tag{2.2}$$

and therefore,

$$A\sigma_x = \bar{F}_x \tag{2.3}$$

Then $\bar{u}$ is the local axial displacement in the $\bar{x}$-direction and $\bar{F}_x$ is the tensile force in the local x-direction. Note that the bar element cannot sustain shear forces. Substituting σ_x and ε_x into Hooke's law (2.1) yields:

$$\frac{d}{d\bar{x}}\left(AE\frac{d\bar{u}}{d\bar{x}}\right) = 0 \tag{2.4}$$

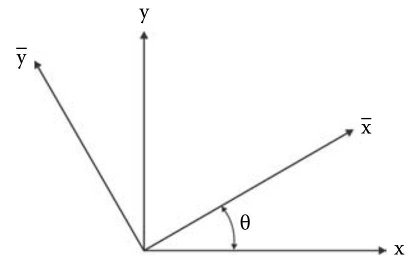


FIGURE 2.1 The local and global coordinate systems.

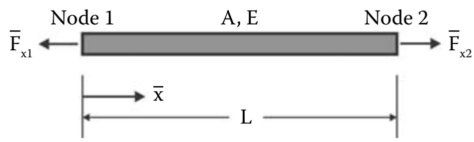


FIGURE 2.2 A bar element lying along the local x-coordinate.

Assuming a linear displacement along the local x-axis of the bar, the displacement function is expressed as

$$\bar{u}(\bar{x}) = a_1 + a_2\bar{x} \tag{2.5}$$

The displacement values at nodes are used to determine the values of the constants a_1 and a_2. Now, the displacement function is expressed as a function of the nodal displacement $\bar{d}_{1x}$ and $\bar{d}_{2x}$, which can be achieved by evaluating $\bar{u}$ at the nodes, and solving for a_1 and a_2 as follows:

$$\bar{u}(0) = \bar{d}_{1x} = a_1 \tag{2.6}$$

$$\bar{u}(L) = \bar{d}_{2x} = a_2L + \bar{d}_{1x} \tag{2.7}$$

and solving for a_2,

$$a_2 = \frac{\bar{d}_{2x} - \bar{d}_{1x}}{L} \tag{2.8}$$

Hence, the displacement function (2.5) becomes

$$\bar{u}(\bar{x}) = \left(\frac{\bar{d}_{2x} - \bar{d}_{1x}}{L}\right)\bar{x} + \bar{d}_{1x} \tag{2.9}$$

The displacement function (2.9) is expressed in the following form:

$$\bar{u}(\bar{x}) = N_1\bar{d}_1 + N_2\bar{d}_2 \tag{2.10}$$

where N_1 and N_2 are called the shape functions of the displacement function, which is associated with degrees of freedom $\bar{d}_1$ and $\bar{d}_2$. The shape functions are defined as

$$N_1 = 1 - \frac{\bar{x}}{L} \tag{2.11}$$

and

$$N_2 = \frac{\bar{x}}{L} \tag{2.12}$$

The displacement function can be written in the following matrix form:

$$\bar{u} = \begin{bmatrix} N_1 & N_2 \end{bmatrix} \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{2x} \end{Bmatrix} \tag{2.13}$$

The strain is the elongation in the x-direction, which is equal to the derivative of the displacement function as follows:

$$\varepsilon_x = \frac{\bar{u}(x + \Delta x) - \bar{u}(x)}{\Delta x} = \frac{d\bar{u}}{d\bar{x}} \tag{2.14}$$

In terms of the nodal displacement, the strain is written as

$$\varepsilon_x = \frac{d\bar{u}}{d\bar{x}} = \frac{\bar{d}_{2x} - \bar{d}_{1x}}{L} \tag{2.15}$$

The strain (2.15) is substituted into Hooke's law (2.1) to obtain the force:

$$\bar{F}_x = AE\left(\frac{\bar{d}_{2x} - \bar{d}_{1x}}{L}\right) \tag{2.16}$$

The nodal force at node 1 is acting in the negative x-direction and should have a negative sign as follows:

$$\bar{F}_{1x} = -\bar{F}_x \tag{2.17}$$

Hence, the nodal force at node 1 becomes

$$\bar{F}_{1x} = \frac{AE}{L}\left(\bar{d}_{1x} - \bar{d}_{2x}\right) \tag{2.18}$$

The nodal force at node 2 is acting in the positive x-direction and should have a positive sign as follows:

$$\bar{F}_{2x} = \bar{F}_x \tag{2.19}$$

$$\bar{F}_{2x} = \frac{AE}{L}\left(\bar{d}_{2x} - \bar{d}_{1x}\right) \tag{2.20}$$

The nodal force in the x-direction is expressed in a matrix form as follows:

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{2x} \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{2x} \end{Bmatrix} \tag{2.21}$$

and similarly for the nodal forces in the y-direction:

$$\begin{Bmatrix} \bar{F}_{1y} \\ \bar{F}_{2y} \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{d}_{1y} \\ \bar{d}_{2y} \end{Bmatrix} \tag{2.22}$$

Since $\{\bar{F}\} = [\bar{K}]\{\bar{d}\}$, the stiffness matrix for a bar element in local coordinates can be written as

$$[\bar{K}] = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{2.23}$$

The stiffness matrix can be determined using the strain energy expression along with Castigliano's theorem for a bar. The strain energy of a bar in tension is expressed as follows:

$$U = \int_0^L \frac{\bar{F}_x^2}{2EA} dx \tag{2.24}$$

In the theorem, the change of strain energy is equal to the virtual displacement times the external force at a point as follows:

$$\partial U = P_i \ \partial \bar{d}_i \tag{2.25}$$

Equation 2.25 in terms of external force is

$$P_i = \frac{\partial U}{\partial \bar{d}_i} \tag{2.26}$$

The axial force can be written in terms of displacement function as

$$\bar{F}_x = \sigma_x A = E\varepsilon_x A = EA\frac{\partial \bar{u}}{\partial \bar{x}} \tag{2.27}$$

Using Equation 2.10 in Equation 2.27, the axial force is expressed as

$$\bar{F}_x = EA\left(N_1' \ \bar{d}_{1x} + N_2' \ \bar{d}_{2x}\right) \tag{2.28}$$

The strain energy of a bar (2.24) in tension is expressed as

$$U = \frac{EA}{2}\int_0^L \left(N_1' \ \bar{d}_{1x} + N_2' \ \bar{d}_{2x}\right)^2 \ d\bar{x} \tag{2.29}$$

Using Castigliano's first theorem, the nodal force in the x-direction at node 1 is written as

$$\begin{aligned}\bar{F}_{1x} &= \frac{\delta \partial U}{\partial \bar{d}_{1x}} = EA\int_0^L \left(N_1' \,\bar{d}_{1x} + N_2' \,\bar{d}_{2x}\right) N_1' \,d\bar{x} \\ &= EA\left(\int_0^L N_1' \,N_1' \,\overline{dx}\right)\bar{d}_{1x} + EA\left(\int_0^L N_1' \,N_2' \,d\bar{x}\right)\bar{d}_{2x}\end{aligned} \tag{2.30}$$

and at node 2,

$$\begin{aligned}\bar{F}_{2x} &= \frac{\delta \partial U}{\partial \bar{d}_{2x}} = EA\int_0^L \left(N_1' \,\bar{d}_{1x} + N_2' \,\bar{d}_{2x}\right) N_2' \,d\bar{x} \\ &= EA\left(\int_0^L N_1' \,N_2' \,d\bar{x}\right)\bar{d}_{1x} + EA\left(\int_0^L N_2' \,N_2' \,d\bar{x}\right)\bar{d}_{2x}\end{aligned} \tag{2.31}$$

Nodal forces in the x-direction in the matrix form are expressed as

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{2x} \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{2x} \end{Bmatrix} \tag{2.32}$$

Or, symbolically,

$$\{\bar{F}_x\} = [K]\{\bar{d}_x\} \tag{2.33}$$

where [K] is the stiffness matrix, in which the matrix coefficients are defined as

$$k_{ij} = EA\int_0^L N_i' \,N_j' \,d\bar{x} \tag{2.34}$$

Substituting the shape functions into Equation 2.32, the stiffness equation is expressed as

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{2x} \end{Bmatrix} = \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{2x} \end{Bmatrix} \tag{2.35}$$

2.2 Finite element method for an arbitrary-oriented bar element

The local coordinate system is always chosen to represent an individual element, whereas the global coordinate system is chosen for the whole structure. In order to relate the global displacement components to a local one, the transformation matrix is used. Figure 2.3 shows a bar element lying along the local x-axis. The local x-axis is oriented at an angle θ

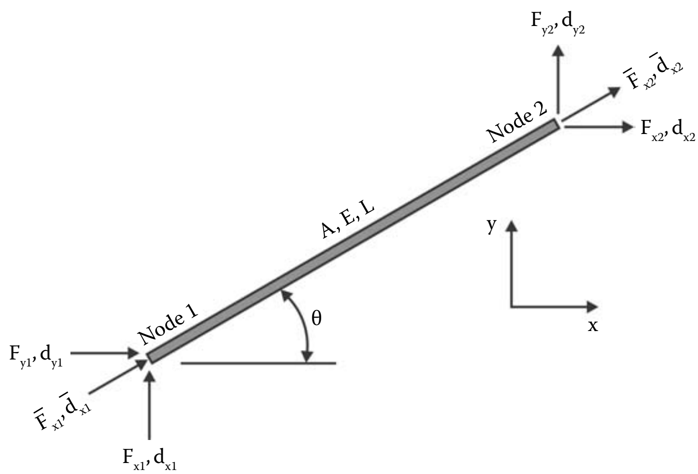


FIGURE 2.3 Relationship between the local and global displacements.

measured counterclockwise from the global x-axis. In the local coordinate, each node has a nodal force $\bar{F}_x$ and nodal displacement $\bar{d}_x$ along its axis. In the global coordinate, each node has a horizontal force F_x, vertical force F_y, horizontal displacement d_x, and vertical displacement d_y. Hence, each element has four degrees of freedom.

From Figure 2.3, the displacement in the x-direction for nodes 1 and 2 in a local coordinate system can be obtained from its global displacements as follows:

$$\bar{d}_{1x} = d_{1x}\cos\theta + d_{1y}\sin\theta \tag{2.36}$$

$$\bar{d}_{2x} = d_{2x}\cos\theta + d_{2y}\sin\theta \tag{2.37}$$

and in the y-direction,

$$\bar{d}_{1y} = -d_{1x}\sin\theta + d_{1y}\cos\theta \tag{2.38}$$

$$\bar{d}_{2y} = -d_{2x}\sin\theta + d_{2y}\cos\theta \tag{2.39}$$

The transformation for nodal displacements for each element can be written in matrix form as follows:

$$\begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{1y} \\ \bar{d}_{2x} \\ \bar{d}_{2y} \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \\ 0 & 0 & -\sin\theta & \cos\theta \end{bmatrix} \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{1y} \\ \bar{d}_{2x} \\ \bar{d}_{2y} \end{Bmatrix} \tag{2.40}$$

and symbolically,

$$\{\bar{d}\} = [T]\{d\} \tag{2.41}$$

The nodal force is transformed from the global to the local coordinate system in a similar manner:

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{1y} \\ \bar{F}_{2x} \\ \bar{F}_{2y} \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \\ 0 & 0 & -\sin\theta & \cos\theta \end{bmatrix} \begin{Bmatrix} F_{1x} \\ F_{1y} \\ F_{2x} \\ F_{2y} \end{Bmatrix} \tag{2.42}$$

and symbolically,

$$\{\bar{F}\} = [T]\{F\} \tag{2.43}$$

The global element nodal force vector is related to the global displacement vector using the global stiffness matrix as follows:

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{1y} \\ \bar{F}_{2x} \\ \bar{F}_{2y} \end{Bmatrix} = [K] \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{1y} \\ \bar{d}_{2x} \\ \bar{d}_{2y} \end{Bmatrix} \tag{2.44}$$

where

$$[K] = \frac{AE}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \tag{2.45}$$

Substituting $\{\bar{d}\} = [T]\{d\}$ and $\{\bar{F}\} = [T]\{F\}$ into $\{\bar{F}\} = [\bar{K}]\{\bar{d}\}$, the following equation is obtained:

$$[T]\{F\} = [\bar{K}][T]\{d\} \tag{2.46}$$

By multiplying both sides of the above equation by the inverse of the transformation matrix, the nodal force vector becomes

$$\{F\} = [T]^{-1}[\bar{K}][T]\{d\} \tag{2.47}$$

Since the transformation matrix is orthogonal, its transport and inverse are equal. The nodal force vector is expressed as

$$\{F\} = [T]^T \left[\bar{K}\right][T]\{d\} \tag{2.48}$$

Since the global coordinate force equation {F} = [K]{d} is equal to nodal force in Equation 2.48, the global stiffness matrix for an element is written as

$$[K] = [T]^T \left[\bar{K}\right][T] \tag{2.49}$$

Expanding the stiffness matrix yields

$$[K] = \frac{AE}{L}\begin{bmatrix} \cos\theta\cos\theta & \cos\theta\sin\theta & -\cos\theta\cos\theta & -\cos\theta\sin\theta \\ \cos\theta\sin\theta & \sin\theta\sin\theta & -\sin\theta\cos\theta & -\sin\theta\sin\theta \\ -\cos\theta\cos\theta & -\sin\theta\cos\theta & \cos\theta\cos\theta & \sin\theta\cos\theta \\ -\cos\theta\sin\theta & -\sin\theta\sin\theta & \sin\theta\cos\theta & \sin\theta\sin\theta \end{bmatrix} \tag{2.50}$$

Assemble the global stiffness and force matrices using the direct stiffness method to obtain matrices for the entire domain:

$$[K] = \sum_{e=1}^{N}\left[K^{(e)}\right] \tag{2.51}$$

$$\{F\} = \sum_{e=1}^{N}\left\{F^{(e)}\right\} \tag{2.52}$$

$$\{d\} = \sum_{e=1}^{N}\left\{d^{(e)}\right\} \tag{2.53}$$

The global stiffness matrix is related to the global nodal force matrix and global displacement matrix for the whole structure using the following expression:

$$\{F\}=[K]\{d\} \tag{2.54}$$

2.3 Analyzing a plane bar structure

The bar structure shown in Figure 2.4a is composed of three bars. All bars have the same length, cross-sectional area, and modulus of elasticity. A horizontal force is applied at Point 2 with F = 100 kN.

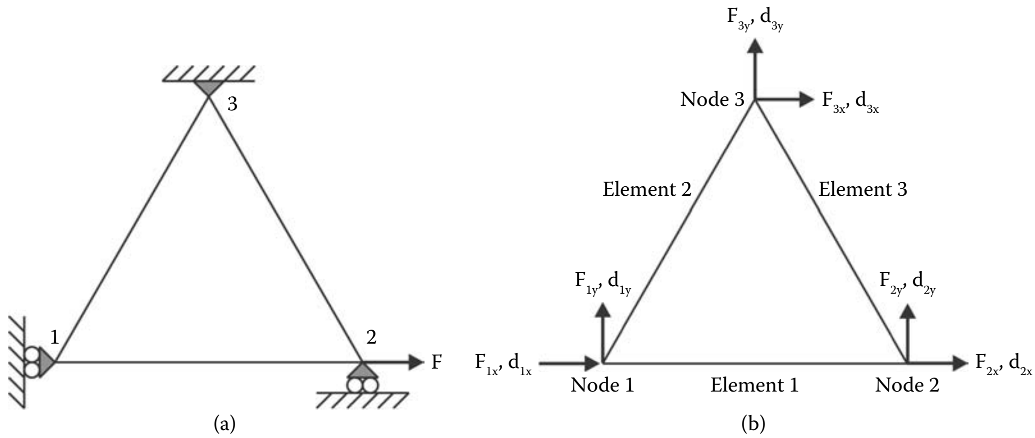


FIGURE 2.4 (a) A bar structure and (b) nodes and elements.

Determine the horizontal and vertical displacements at all connections, given that E = 100 GPa, $A = 1.0 \times 10^{-4}$ m^2, and L = 1 m.

Before starting to solve the problem, all elements and nodes of the structure should be numbered as suggested in Figure 2.4b.

Step 1: Construct the stiffness matrix for each element

The stiffness matrix for Element 1, which has nodes 1 and 2 with $\theta = 0^o$, is calculated using the following matrix (2.50):

$$\left[K^{(1)}\right] = \frac{AE}{4L} \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{2}} \end{array} \\ \begin{bmatrix} 4 & 0 & -4 & 0 \\ 0 & 0 & 0 & 0 \\ -4 & 0 & 4 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \end{array}$$

The stiffness matrix for Element 2, which has nodes 1 and 3 with $\theta = 60^o$, is calculated as follows:

$$\left[K^{(2)}\right] = \frac{AE}{4L} \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \begin{bmatrix} 1 & \sqrt{3} & -1 & -\sqrt{3} \\ \sqrt{3} & 3 & -\sqrt{3} & -3 \\ -1 & -\sqrt{3} & 1 & \sqrt{3} \\ -\sqrt{3} & -3 & \sqrt{3} & 3 \end{bmatrix} \end{array} \begin{array}{c} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

The stiffness matrix for Element 3, which has nodes 2 and 3 with $\theta = 120°$, is calculated as follows:

$$\left[K^{(3)}\right] = \frac{AE}{4L} \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{2}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \left[\begin{array}{cccc} 1 & -\sqrt{3} & -1 & \sqrt{3} \\ -\sqrt{3} & 3 & -\sqrt{3} & -3 \\ -1 & \sqrt{3} & 1 & -\sqrt{3} \\ \sqrt{3} & -3 & -\sqrt{3} & 3 \end{array}\right] \end{array} \begin{array}{c} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

Step 2: Assemble the elements' stiffness matrices to form a global stiffness matrix using Equation 2.51

$$[K] = \frac{AE}{4L} \begin{array}{c} \begin{array}{cccccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \left[\begin{array}{cccccc} 5 & \sqrt{3} & -4 & 0 & -1 & -\sqrt{3} \\ \sqrt{3} & 3 & 0 & 0 & -\sqrt{3} & -3 \\ -4 & 0 & 5 & -\sqrt{3} & -1 & \sqrt{3} \\ 0 & 0 & -\sqrt{3} & 3 & \sqrt{3} & -3 \\ -1 & -\sqrt{3} & -1 & \sqrt{3} & 2 & 0 \\ -\sqrt{3} & -3 & \sqrt{3} & -3 & 0 & 6 \end{array}\right] \end{array} \begin{array}{c} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

Step 3: Apply the boundary conditions to the global stiffness matrix using Equation 2.54

$$\left\{\begin{array}{c} F_{1x} \\ F_{1y} = 0 \\ F_{2x} = 10^5 \\ F_{2y} = 0 \\ F_{3x} \\ F_{3y} \end{array}\right\} = [K] \left\{\begin{array}{c} d_{1x} = 0 \\ d_{1y} \\ d_{2x} \\ d_{2y} = 0 \\ d_{3x} = 0 \\ d_{3y} = 0 \end{array}\right\}$$

Step 4: Eliminate rows and columns from the global stiffness matrix to remove the singularity

Since the stiffness matrix is singular, at a minimum, one equation must be eliminated to remove the singularity. The first, fourth, fifth, and sixth columns are multiplied by zero displacements, and therefore, they can be deleted with the corresponding rows, and the result is

$$\left\{\begin{array}{c} F_{1y} = 0 \\ F_{2x} = 10^5 \end{array}\right\} = \frac{1\times10^{-4}\left(100\times10^{9}\right)}{1\times4} \left[\begin{array}{cc} 3 & 0 \\ 0 & 5 \end{array}\right] \left\{\begin{array}{c} d_{1y} \\ d_{2x} \end{array}\right\}$$

Step 5: Solve for d_{1y} and d_{2x}

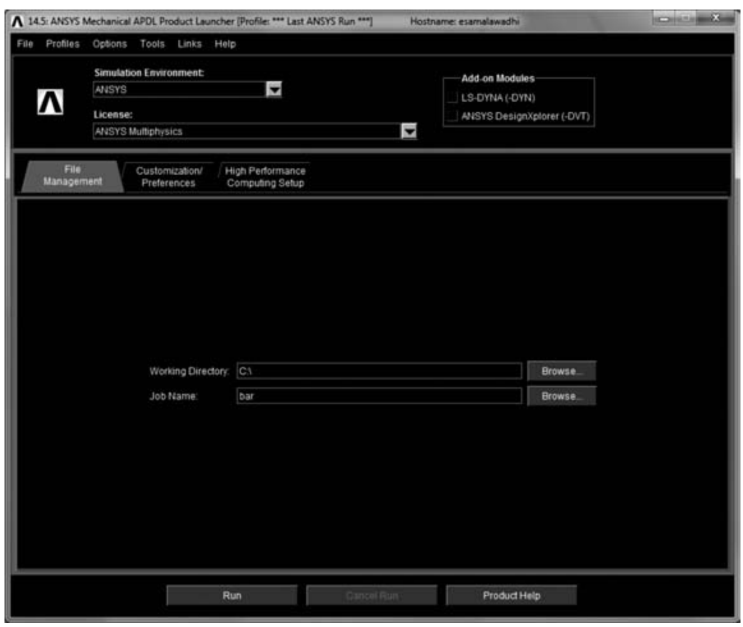
$d_{1y} = 0$ m

$d_{2x} = 0.008$ m

2.4 Analyzing a plane bar structure using ANSYS

The bar structure shown in Figure 2.4a is composed of three bars. All bars have the same length, cross-sectional area, and modulus of elasticity. A horizontal force is applied at Point 2, F = 100 kN. Determine using ANSYS the horizontal and vertical displacements at all connections and the reaction at the supports, given that E = 100 GPa, A = 1.0×10^{-4} m^2, and L = 1 m.

Double click on the Mechanical APDL Product Launcher icon

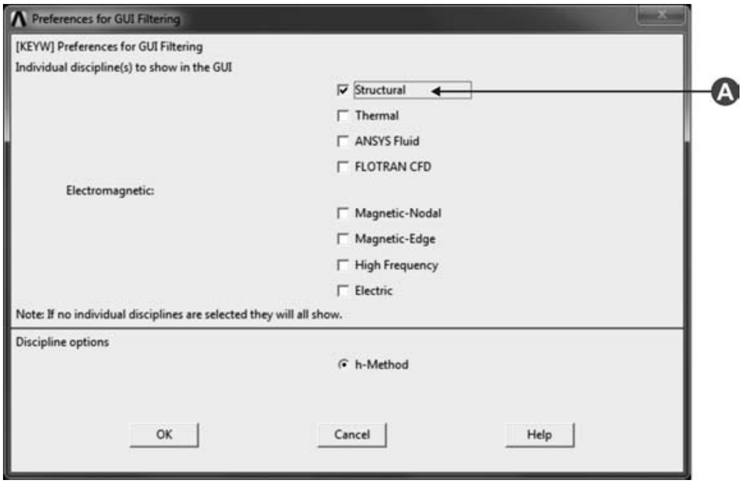


A select ANSYS Multiphysics in License
B change Working Directory to C:\ or any directory
C change the initial Job Name to bar, or any name

$\boxed{\textbf{Run}}$

This example is limited to structural analysis. Hence, select Structural in Preferences and leave other physics unselected. This will reduce selection options in the preprocessor and solution tasks.

Main Menu > Preferences



A select Structural in Preferences for GUI Filtering

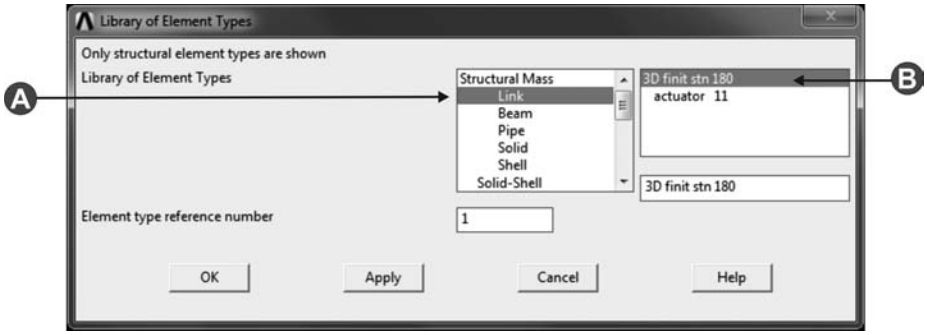
OK

Next, the element type is selected. For the present problem, the element type is Link, which is equivalent to a bar.

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add...

The selected link element will not support pressure on the elements or moment on the nodes. However, this element can be used for three-dimensional analyses.



A select Link
B select 3D finit stn 180

OK

Close

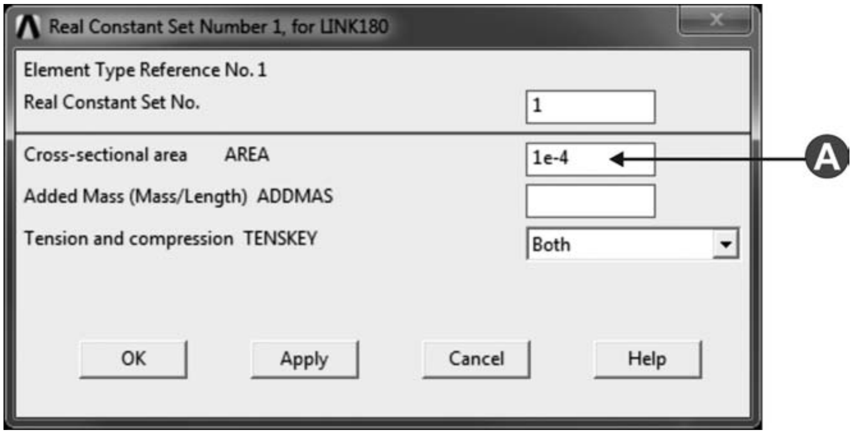
The cross-sectional area of the bars is required for the analysis. The initial strain is zero by default.

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

Add...

Notice that the selected element type is shown in the Element Type for Real Constants. Here, the cross-sectional area of the bars is specified.

OK



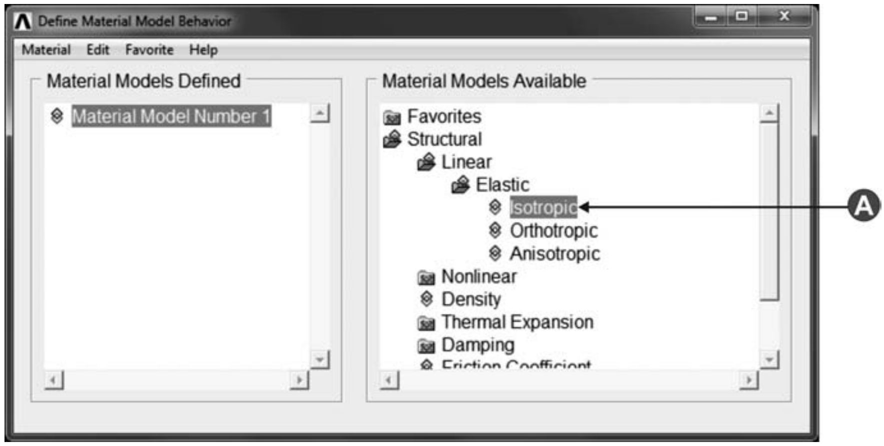
A type 1e-4 in Cross-sectional area AREA

OK

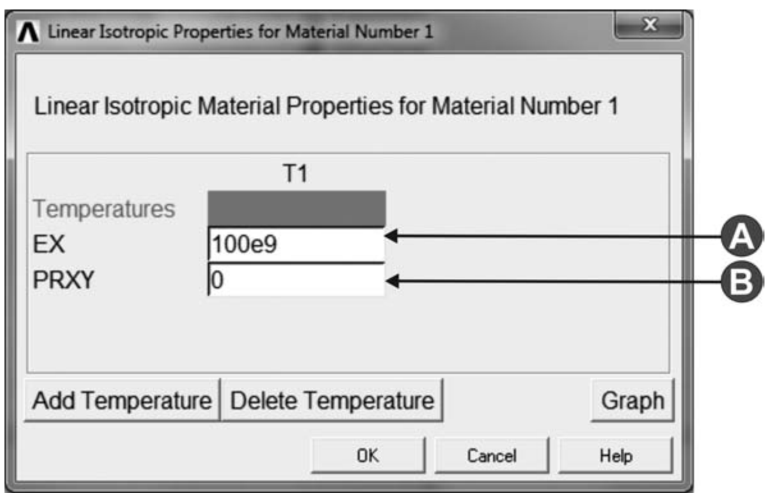
Close

Material properties of the bars are specified in the following steps. The bars are elastic and independent of the direction isotropic. Only the modulus of elasticity is required and the zero Poisson ratio, or any value, is just to avoid an error message from ANSYS.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic



A type 100e9 in EX

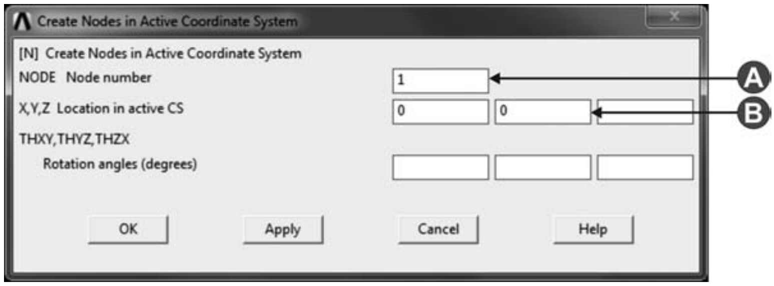
B type 0 in PRXY

OK

Close the Material Model Behavior window

The process of modeling the bar structure is started here. First, three nodes are created, followed by creation of the elements. The x- and y-coordinates of each node are specified in ANSYS. The coordinate for node 1 is (0,0), for node 2 is (1,0), and for node 3 is (0.5,0.866). The Apply button will not close the window, allowing for additional inputs, while the OK will close the window.

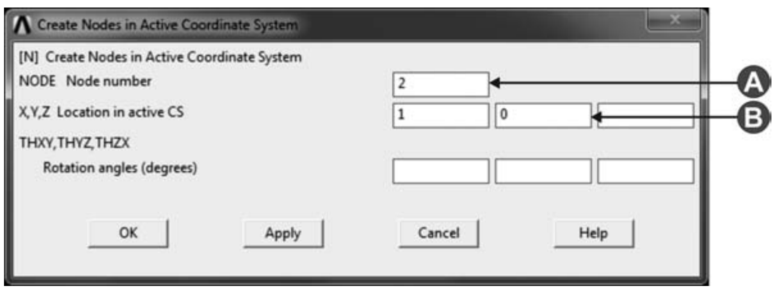
Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS



A type 1 in Node number

B type 0 and 0 in X,Y,Z Location in active CS

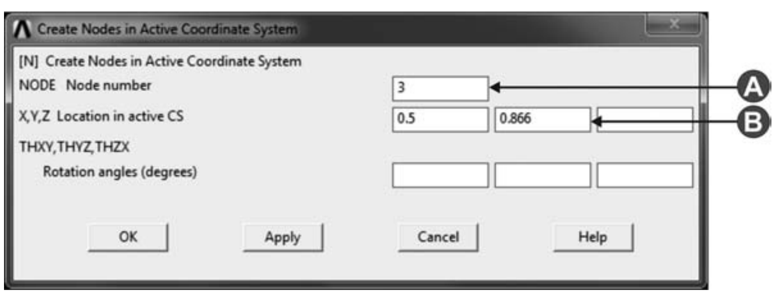
Apply



A type 2 in Node number

B type 1 and 0 in X,Y,Z Location in active CS

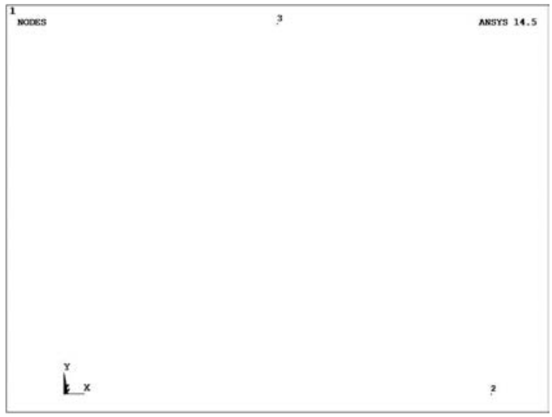
Apply



A type 3 in Node number

B type 0.5 and 0.866 in X,Y,Z Location in active CS

OK



ANSYS graphics show the created nodes

Main Menu > Modeling > Create > Elements > Auto Numbered > Thru Nodes

Click on node 1 then 2. Then in Elements from Nodes window, click on

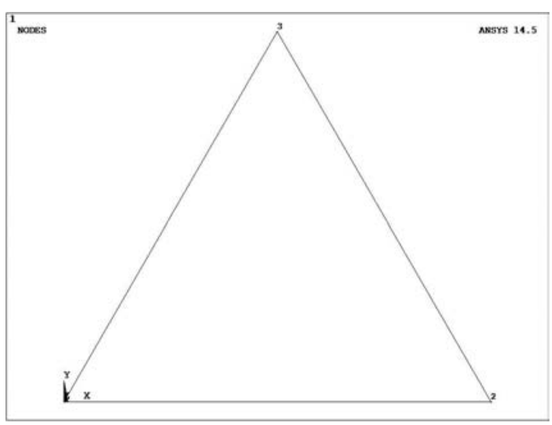
Apply

Click on node 2 then 3. Then in Elements from Nodes window, click on

Apply

Click on node 3 then 1. Then in Elements from Nodes window, click on

OK



ANSYS graphics show the three elements

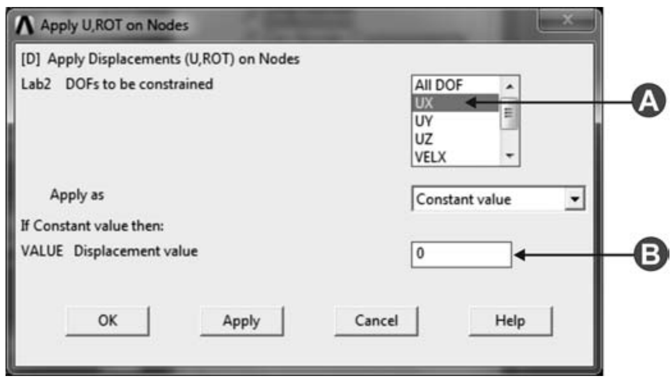
The preprocessor task is ended at this point. The solution task starts here. Nodal forces and displacements are applied. Starting with forces or

displacements will not affect the solution. A zero nodal displacement at a certain direction means that the node is fixed at that direction.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

Click on node 1. Then in Elements from Nodes window, click on

OK



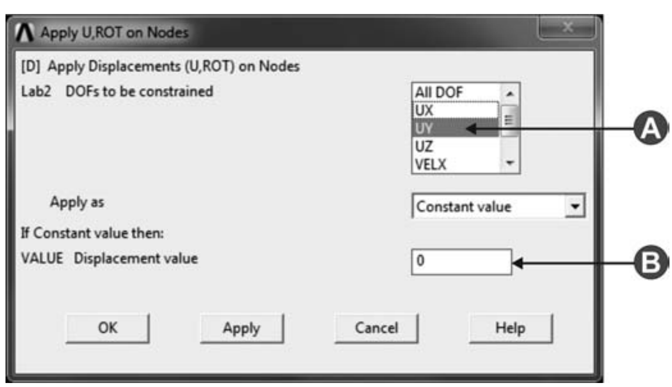
A select UX in Lab2 DOFs to be constrained

B type 0 in Displacement value

Apply

Click on node 2. Then in Elements from Nodes window, click on

OK



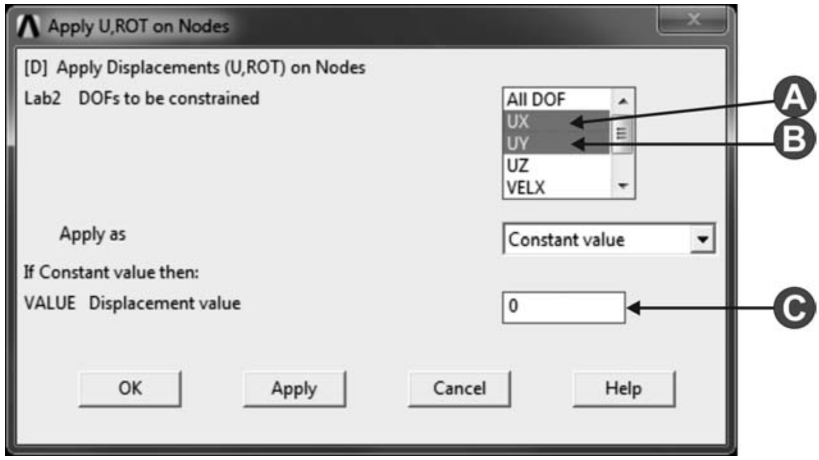
A select UY in Lab2 DOFs to be constrained

B type 0 in Displacement value

Apply

Click on node 3. Then in Elements from Nodes window, click on

OK



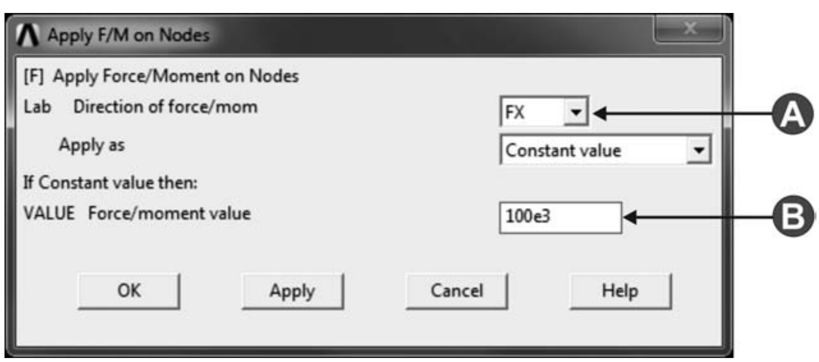
A select UX in Lab2 DOFs to be constrained
B select UY in Lab2 DOFs to be constrained
C type 0 in Displacement value

OK

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

Click on node 2. Then in Apply F/M on Nodes window, click on

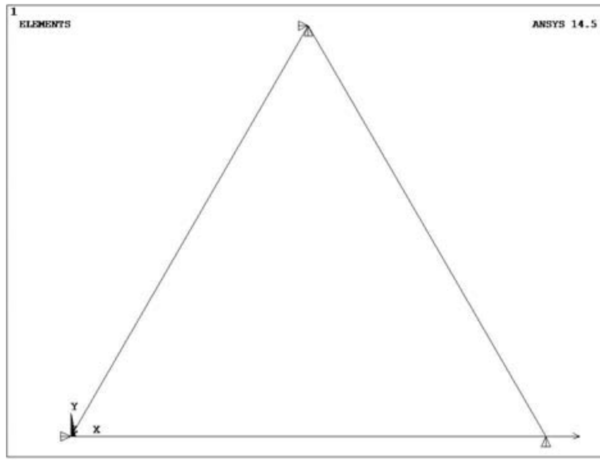
OK



A select FX in Direction of force/mom
B type 100e3 in Force/moment value

OK

The positive FX means that the force at node 2 is in the positive x-direction. The ANSYS graphics will show the applied force and its direction with a red arrow. Reapplying the force at node 2 will automatically delete the force and apply the new value.

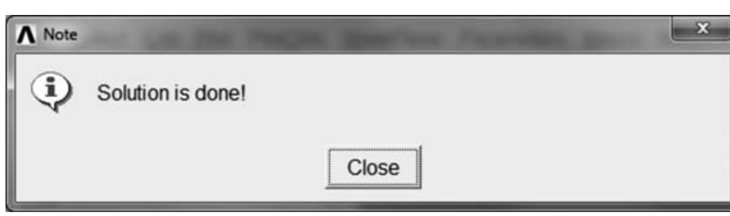


ANSYS graphics show the force with direction and the displacement constraints on nodes

The final step is to initiate the solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the equations.

Main Menu > Solution > Solve > Current LS

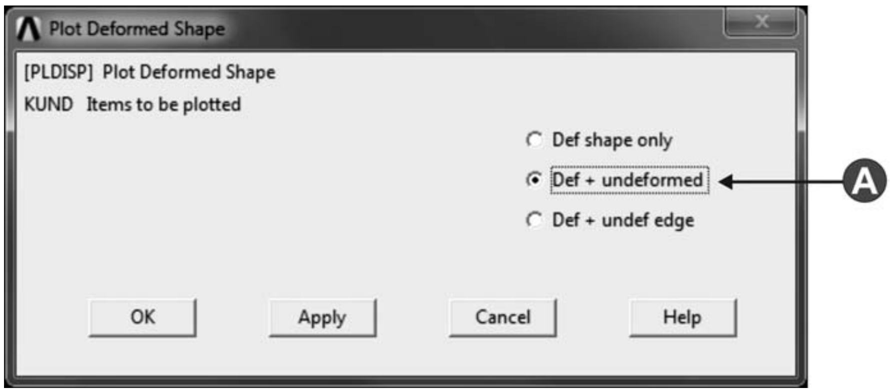
OK



Close

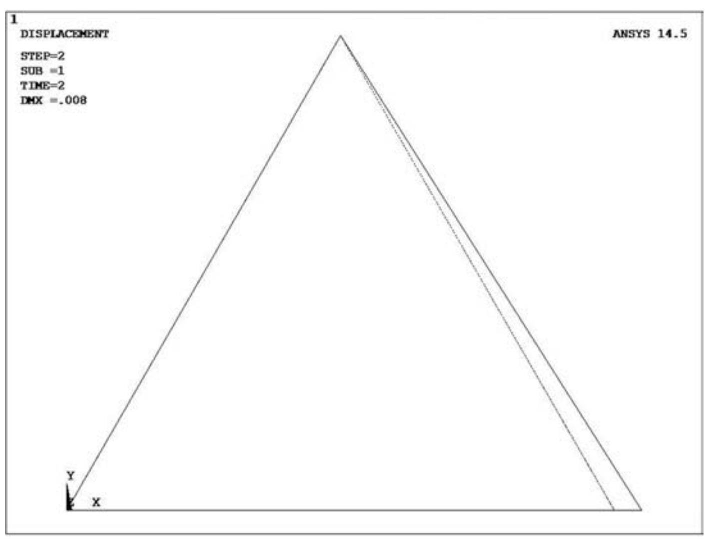
Results can be viewed in the general postprocessor task. Inspecting the deformation of the bar structure will help to identify if the problem is solved correctly. Node 2 should move to the right as a result of the applied force, while node 3 should be stationary.

Main Menu > General Postproc > Plot Results > Deformed Shape



A select Def + unreformed

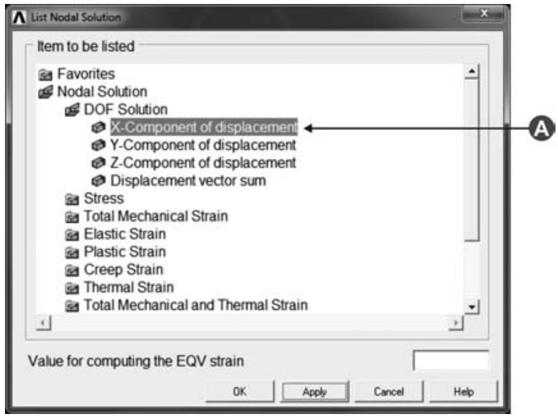
$\boxed{\textbf{OK}}$



ANSYS graphics show the bar before and after deformation

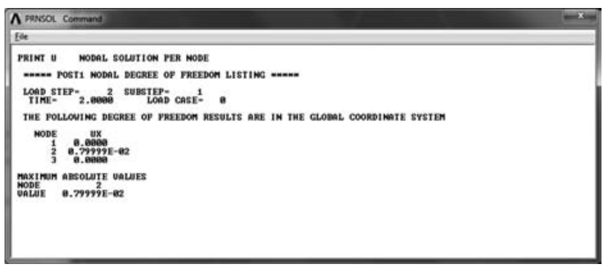
From the above figure, the result is as expected. Nodes 1 and 3 are fixed, while node 2 is moved in the direction of the applied force. In the following steps, the nodal results are presented. First, the nodal displacements are listed, and second, the reactions at the supports are listed.

Main Menu > General Postproc > List Results > Nodal Solution



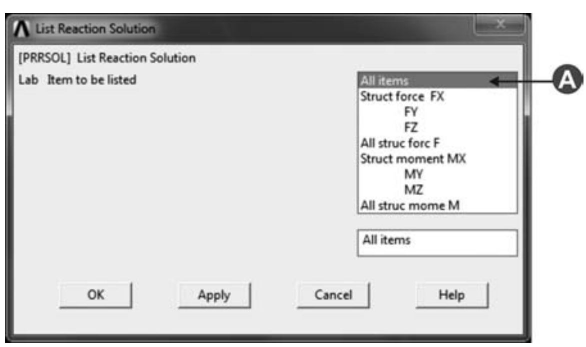
A click on Nodal Solution > DOF Solution > X-Component of displacement

OK



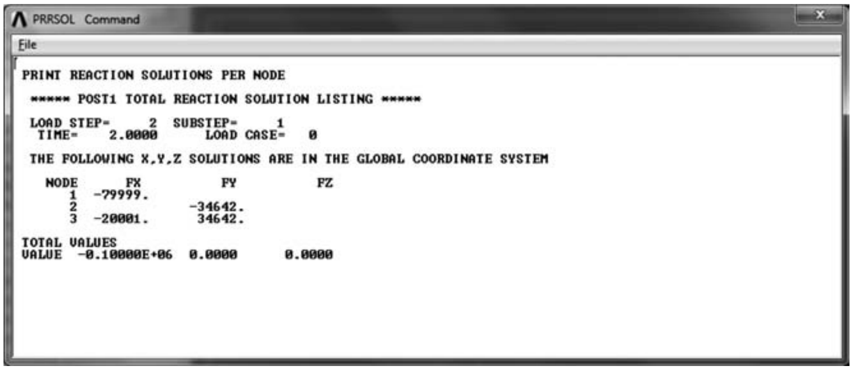
A list of nodal displacement in the x-direction is shown. In addition, the maximum nodal displacement is shown at the end. The nodal displacement in the y-direction can be determined by selecting Y-Component in the Nodal Solution. The results are identical to the previous example, and the reactions at the supports are listed as follows:

Main Menu > General Postproc > List Results > Reaction Solution



A select All items

OK




```
PRINT REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     2  SUBSTEP=     1
  TIME=    2.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

   NODE       FX           FY           FZ
      1   -79999.
      2                 -34642.
      3   -20001.        34642.

TOTAL VALUES
VALUE  -0.10000E+06  0.0000       0.0000
```

As shown, the forces are balanced in the x- and y-directions. The sum of the forces in the x-direction is equal to -100×10^3, which is equal to the applied force, and the sum of the forces in the y-direction is equal to zero.

2.5 Finite element method for a horizontal beam element

A beam is defined as a long and slender structural member that can be subjected to transverse loadings. Therefore, the beam can be twisted and bended. The bar elements can sustain the transverse loadings only. Consider a horizontal beam element, as shown in Figure 2.5. The beam has an initial length L, modulus of elasticity E, and moment of inertia I. The local displacement and rotation at nodes 1 and 2 are (d_{x1}, θ_1) and (d_{x2}, θ_2), respectively. The local nodal force and bending moment at nodes 1 and 2 are (F_{1y}, M_1) and (F_{2y}, M_2), respectively.

All moments and rotations are positive if their direction is counterclockwise and negative if their direction is in a clockwise direction. The horizontal forces are positive if the direction is positive in the x-direction and negative if the direction is negative in the x-direction. The vertical forces are positive if the direction is positive in the y-direction and negative if the direction is negative in the y-direction. Consider a differential beam element, as shown in Figure 2.6.

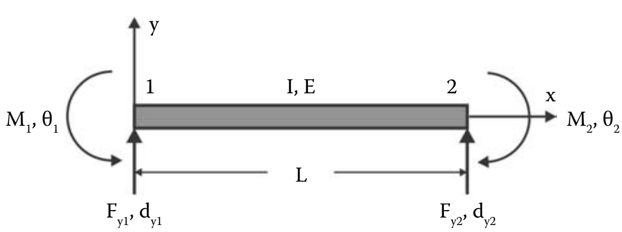


FIGURE 2.5 Beam element subjected to forces and moments.

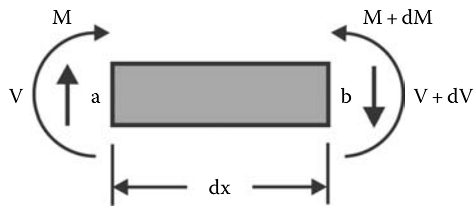


FIGURE 2.6 A differential beam element.

Applying the moments balance at Point b yield,

$$-Vdx + dM = 0 \tag{2.55}$$

or

$$V = \frac{dM}{dx} \tag{2.56}$$

The curvature of the beam is related to the applied moment. The θ is the rotation and the function v(x) is the transverse displacement in the y-direction. Figure 2.7 shows a section of a horizontal deflected beam.

The rotation of the deflection is expressed as

$$\theta = \frac{dv(x)}{dx} \tag{2.57}$$

The axial strain is related to the axial displacement using the following relationship:

$$\varepsilon_x = \frac{du}{dx} \tag{2.58}$$

On the other hand, the axial displacement is related to the transverse displacement by

$$u = -y\frac{dv}{dx} \tag{2.59}$$

Substituting the relationship (2.59) into (2.58) yields

$$\varepsilon_x = -y\frac{d^2v}{dx^2} \tag{2.60}$$

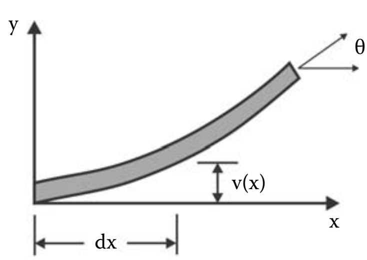


FIGURE 2.7 Horizontal deflected beam.

For a cross section of the beam shown in Figure 2.8, the differential force and moment along the center of the beam are expressed as, respectively,

$$dF = \sigma_x \, dA \tag{2.61}$$

$$dM = \sigma_x \, ydA \tag{2.62}$$

Using Hooke's equation, the differential moment becomes

$$dM = E\varepsilon_x ydA = Ey^2 \frac{d^2v}{dx} dA \tag{2.63}$$

Integrating Equation 2.63 for the beam section yields

$$M = E \frac{d^2v}{dx^2} \int y^2 dA \tag{2.64}$$

Since the first moment of inertia is equal to $\int y^2 \, dA$, Equation 2.64 is expressed as

$$M = EI \frac{d^2v}{dx^2} \tag{2.65}$$

Also, the shearing force (2.56) is expressed as

$$V = EI \frac{d^3v}{dx^3} \tag{2.66}$$

The variations of beam curvature are approximated by a third-order polynomial, and the constants in the polynomial are determined using the boundary conditions as follows:

$$v(x) = a_1x^3 + a_2x^2 + a_3x + a_4 \tag{2.67}$$

Using expression (2.57), the rotation of the deflection can be determined as

$$\theta(x) = 3a_1x^2 + 2a_2x + a_3 \tag{2.68}$$

Applying the boundary conditions,

$$v(0) = d_{y1} : a_4 = d_{y1} \tag{2.69}$$

$$v(L) = d_{y2} : a_1L^3 + a_2L^2 + a_3L + a_4 = d_{y2} \tag{2.70}$$

FIGURE 2.8 Beam cross section.

$$\theta(0) = \theta_1 : a_3 = \theta_1 \tag{2.71}$$

$$\theta(L) = \theta_2 : 3a_1L^2 + 2a_2L + a_3 = \theta_2 \tag{2.72}$$

Solving Equations 2.69–6.72 for a_1, a_2, a_3, and a_4 and substituting the solution into Equation 2.67 yields

$$\begin{aligned} v(x) = &\left[\frac{2}{L^3}\left(d_{1y} - d_{2y}\right) + \frac{1}{L^2}(\theta_1 + \theta_2)\right] x^3 \\ &+ \left[-\frac{3}{L^2}\left(d_{1y} - d_{2y}\right) - \frac{1}{L}(2\theta_1 - \theta_2)\right] x^2 + \theta_1 x + d_{1y} \end{aligned} \tag{2.73}$$

or

$$\begin{aligned} v(x) = &\ d_{1y} + \theta_1 x - \frac{3x^2}{L^2} d_{1y} - \frac{2x^2}{L}\theta_1 + \frac{3x^2}{L^2} d_{2y} - \frac{x^2}{L}\theta_2 \\ &+ \frac{2x^2}{L^3} d_{1y} + \frac{x^3}{L^2}\theta_1 - \frac{2x^2}{L^3} d_{2y} + \frac{x^3}{L^2}\theta_2 \end{aligned} \tag{2.74}$$

The displacement function v(x) can be expressed in terms of shape functions as

$$v(x) = N_1 d_{1y} + N_2\theta_1 + N_3 d_{2y} + N_1\theta_2 \tag{2.75}$$

and the shape functions are

$$N_1 = 1 - 3\left(\frac{x}{L}\right)^2 + 2\left(\frac{x}{L}\right)^3 \tag{2.76}$$

$$N_2 = x - 2\left(\frac{x^2}{L}\right) + 2\left(\frac{x^3}{L^2}\right) \tag{2.77}$$

$$N_3 = 3\left(\frac{x}{L}\right)^2 - 2\left(\frac{x}{L}\right)^3 \tag{2.78}$$

$$N_4 = -\left(\frac{x^2}{L}\right) + \left(\frac{x^3}{L^2}\right) \tag{2.79}$$

Finally, the element stiffness matrix is derived using the equilibrium approach. The nodal shear force and bending moment are calculated using Equations 2.65 and 2.66, respectively:

$$F_{1y} = EI\frac{d^3v(0)}{dx^3} = \frac{EI}{L^3}\left(12d_{1y} + 6L\theta_1 - 12d_{2y} + 6L\theta_2\right) \tag{2.80}$$

$$M_1 = -EI\frac{d^2v(0)}{dx^2} = \frac{EI}{L^3}\left(6d_{1y} + 4L^2\theta_1 - 6Ld_{2y} + 2L^2\theta_2\right) \tag{2.81}$$

$$F_{2y} = -EI\frac{d^3v(L)}{dx^3} = \frac{EI}{L^3}\left(-12d_{1y} - 6L\theta_1 + 12d_{2y} - 6L\theta_2\right) \tag{2.82}$$

$$M_2 = EI\frac{d^2v(L)}{dx^2} = \frac{EI}{L^3}\left(6Ld_{1y} + 2L^2\theta_1 - 6Ld_{2y} + 4L^2\theta_2\right) \tag{2.83}$$

The nodal forces and moments can be expressed in matrix form, {F} = [K]{d}, as follows:

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \end{Bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} d_{1y} \\ \theta_1 \\ d_{2y} \\ \theta_2 \end{Bmatrix} \tag{2.84}$$

where

$$[K] = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \tag{2.85}$$

Assemble the stiffness matrices and force and displacement vectors:

$$[K] = \sum_{e=1}^{N}\left[K^{(e)}\right] \tag{2.86}$$

$$\{F\} = \sum_{e=1}^{N}\left\{F^{(e)}\right\} \tag{2.87}$$

$$\{d\} = \sum_{e=1}^{N}\left\{d^{(e)}\right\} \tag{2.88}$$

The global stiffness matrix is used to relate the nodal force vector to nodal displacement vector for the whole structure:

$$\{F\} = [K]\{d\} \tag{2.89}$$

The stiffness equation for the horizontal beam element can also be obtained by using Castigliano's theorem. The strain energy for a beam with uniform material properties and cross-sectional area is

$$U = \frac{1}{2EI}\int_0^L (M)^2\,dx = \frac{EI}{2}\int_0^L \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx \tag{2.90}$$

The second derivative of displacement function is

$$v''(x) = N_1'' d_{1y} + N_2''\theta_1 + N_3'' d_{2y} + N_4''\theta_2 \tag{2.91}$$

and the second derivative of the shape functions is

$$N_1'' = -\frac{6}{L^2} + 12\frac{x}{L^3} \tag{2.92}$$

$$N_2'' = -\frac{4}{L} + 6\frac{x}{L^2} \tag{2.93}$$

$$N_3'' = \frac{6}{L^2} - 12\frac{x}{L^3} \tag{2.94}$$

$$N_4'' = -\frac{6}{L} + 6\frac{x}{L^2} \tag{2.95}$$

In Castigliano's first theorem, the change of strain energy is equal to the virtual displacement times external force at a point. Applying Castigliano's theorem at node 1 gives

$$F_{1y} = \frac{\partial U}{\partial d_{1y}} = \frac{\partial}{\partial d_{1y}}\left(\frac{EI}{2}\int_0^L \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx\right) \tag{2.96}$$

Substituting the second derivative of the displacement function into Equation 2.80 gives

$$F_{1y} = \frac{\partial}{\partial d_{1y}}\left(\frac{EI}{2}\int_0^L \left(N_1'' d_{1y} + N_2''\theta_1 + N_3'' d_{2y} + N_4''\theta_2\right)^2 dx\right) \tag{2.97}$$

Simplifying expression (2.97) yields

$$F_{1y} = \frac{EI}{2}\int_0^L 2\left(N_1'' d_{1y} + N_2''\theta_1 + N_3'' d_{2y} + N_4''\theta_2\right) N_1''\, dx \tag{2.98}$$

and finally,

$$F_{1y} = k_{11}v_1 + k_{12}\theta_1 + k_{13}v_2 + k_{14}\theta_2 \tag{2.99}$$

where

$$k_{ij} = EI\int_0^L N_i'' N_j''\, dx \tag{2.100}$$

The nodal force in the y-direction at node 1 is

$$F_{1y} = \frac{EI}{L^3}\left(12d_{1y} + 6L\theta_1 - 12d_{2y} + 6L\theta_2\right) \tag{2.101}$$

If instead of virtual displacement, the virtual rotation $\partial\theta$ at a point where moment M is applied, the moment in terms of strain energy is

$$M_1 = \frac{\partial U}{\partial \theta_i} \tag{2.102}$$

Applying Castigliano's theorem at node 1 gives

$$M_1 = \frac{\partial U}{\partial \theta_1} = \frac{\partial}{\partial \theta_1}\left(\frac{EI}{2}\int_0^L \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx\right) \tag{2.103}$$

Substituting the second derivative of the displacement function into Equation 2.103 gives

$$M_1 = \frac{\partial}{\partial \theta_1}\left(\frac{EI}{2}\int_0^L \left(N_1'' d_{1y} + N_2''\theta_1 + N_3'' d_{2y} + N_4''\theta_2\right)^2 dx\right) \tag{2.104}$$

Since the modulus of elasticity and moment are independent of x, Equation 2.104 becomes

$$M_1 = \frac{EI}{2}\int_0^L 2\left(N_1'' d_{1y} + N_2''\theta_1 + N_3'' d_{2y} + N_4''\theta_2\right) N_2''\, dx \tag{2.105}$$

or

$$M_1 = k_{21}v_1 + k_{22}\theta_1 + k_{23}v_2 + k_{24}\theta_2 \tag{2.106}$$

where

$$k_{ij} = EI\int_0^L N_i'' N_j''\, dx \tag{2.107}$$

Finally, the nodal moment at node 1 is expressed as

$$M_1 = \frac{EI}{L^3}\left(6d_{1y} + 4L^2\theta_1 - 6Ld_{2y} + 2L^2\theta_2\right) \tag{2.108}$$

By using the same procedure for nodal force and moment at node 2, the stiffness equation for a horizontal beam can be expressed as

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \end{Bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} d_{1y} \\ \theta_1 \\ d_{2y} \\ \theta_2 \end{Bmatrix} \tag{2.109}$$

2.6 Analyzing a horizontal beam structure

A horizontal beam structure, shown in Figure 2.9, is made of two solid cylinders with different materials and radii. Determine the displacement and slope at the points where force or moment is applied. For Beam 1: E = 210 GPa, D = 5 cm; and for Beam 2: E = 180 GPa, D = 4 cm, where D is the diameter of the cylinder.

There are unlimited options for elements and nodes distribution, and some of these options are shown in Figure 2.10. Increasing the number of elements will definitely enhance the accuracy of the results, but only up to a certain number of elements. After this number, the results become independent of the number of elements. The first mesh contains just two elements, which is the minimum to solve this problem. The second mesh contains 3 elements, the third mesh contains 6 elements, the fourth mesh contains 12 elements, and the fifth mesh contains 24 elements. For an illustration purpose, the first mesh is selected because it has the minimum number of elements.

The first moments of inertia of the first and second beams are required to solve the problem, and they are

$$I_1 = \frac{\pi}{4}R^4 = \frac{\pi}{4}\left(2.5\times10^{-2}\right)^4 = 3.067\times10^{-7}\,m^4$$

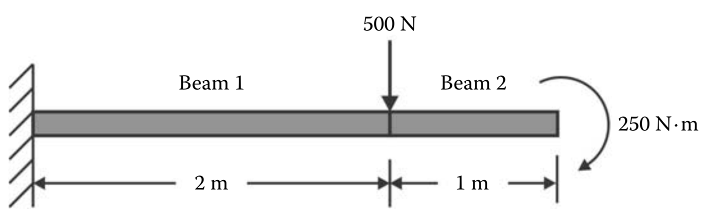


FIGURE 2.9 Beam structure.

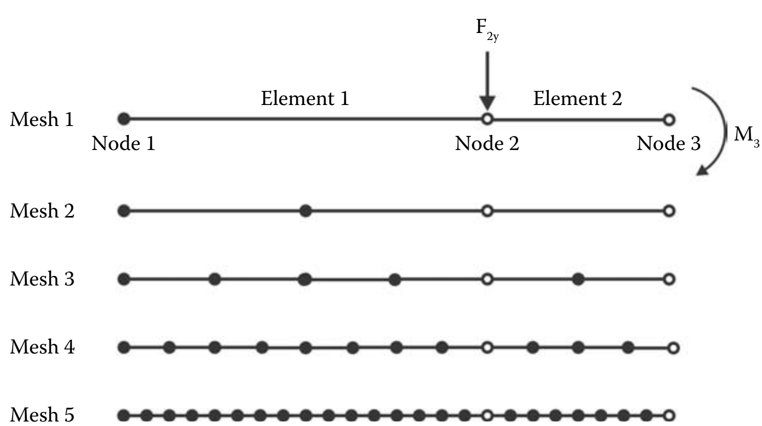


FIGURE 2.10 Meshes for the beam structure.

$$I_2 = \frac{\pi}{4}R^4 = \frac{\pi}{4}\left(2.0\times10^{-2}\right)^4 = 1.256\times10^{-7}\,\mathrm{m}^4$$

First, the stiffness matrix for each element is obtained using Equation 2.85. For the first element, which has nodes 1 and 2, the stiffness matrix is

$$[K^{(1)}] = \frac{210\times10^9\left(3.067\times10^{-7}\right)}{2^3}\begin{bmatrix} 12 & 12 & -12 & 12 \\ 12 & 16 & -12 & 8 \\ -12 & -12 & 12 & -12 \\ 12 & 8 & -12 & 16 \end{bmatrix}$$

$$[K^{(1)}] = 10^4 \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{2}} \end{array} \\ \begin{bmatrix} 9.66 & 9.66 & -9.66 & 9.66 \\ 9.66 & 12.88 & -9.66 & 6.44 \\ -9.66 & -9.66 & 9.66 & -9.66 \\ 9.66 & 6.44 & -9.66 & 12.88 \end{bmatrix} \end{array} \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \end{array}$$

For the second element, which has nodes 2 and 3, the stiffness matrix is

$$[K^{(2)}] = \frac{180\times10^9\left(1.256\times10^{-7}\right)}{1^3}\begin{bmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 2 & -6 & 4 \end{bmatrix}$$

$$[K^{(2)}] = 10^4 \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{2}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \begin{bmatrix} 27.13 & 13.56 & -27.13 & 13.56 \\ 13.56 & 9.04 & -13.56 & 4.52 \\ -27.13 & -13.56 & 27.13 & -13.56 \\ 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix} \end{array} \begin{array}{c} \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

Assembling $[K^{(1)}]$ and $[K^{(2)}]$ using Equation 2.86 yields:

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} = -500 \\ M_2 = 0 \\ F_{3y} = 0 \\ M_3 = -250 \end{Bmatrix} = 10^4 \begin{bmatrix} 9.66 & 9.66 & -9.66 & 9.66 & 0 & 0 \\ 9.66 & 12.88 & -9.66 & 6.44 & 0 & 0 \\ -9.66 & -9.66 & 36.79 & 3.9 & -27.13 & 13.56 \\ 9.66 & 6.44 & 3.9 & 21.88 & -13.56 & 4.56 \\ 0 & 0 & -27.13 & -13.56 & 27.13 & -13.56 \\ 0 & 0 & 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix} \begin{Bmatrix} d_{1y} = 0 \\ \theta_1 = 0 \\ d_{2y} \\ \theta_2 \\ d_{3y} \\ \theta_3 \end{Bmatrix}$$

The first and second columns and rows are deleted to remove the singularity from the stiffness matrix, and it becomes

$$\begin{Bmatrix} F_{2y} = -500 \\ M_2 = 0 \\ F_{3y} = 0 \\ M_3 = -250 \end{Bmatrix} = 10^4 \begin{bmatrix} 36.79 & 3.9 & -27.13 & 13.56 \\ 3.9 & 21.92 & -13.56 & 4.56 \\ -27.13 & -13.56 & 27.13 & -13.56 \\ 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix} \begin{Bmatrix} d_{2y} \\ \theta_2 \\ d_{3y} \\ \theta_3 \end{Bmatrix}$$

There are four equations and four unknowns, and solving for displacements and rotations, the results are

$$d_{2y} = -0.0279 \text{ m}$$

$$d_{3y} = -0.0562 \text{ m}$$

$$\theta_2 = -0.0227 \text{ rad}$$

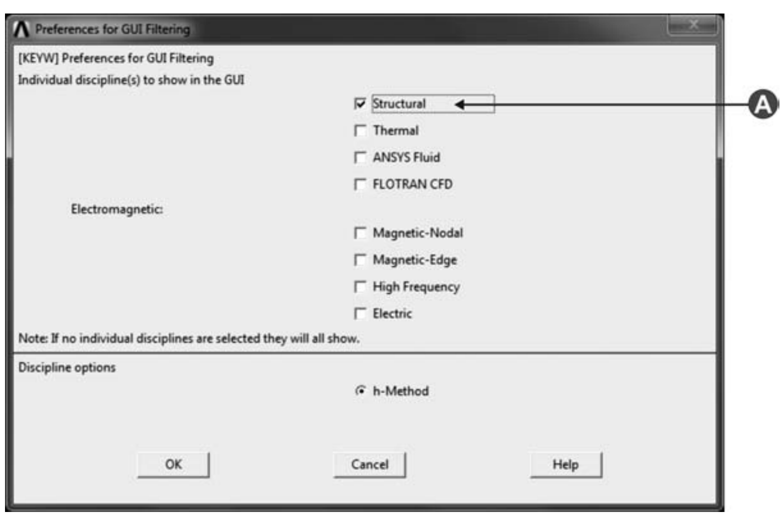
$$\theta_3 = -0.0337 \text{ rad}$$

2.7 Analyzing a horizontal beam structure using ANSYS

The horizontal beam structure shown in Figure 2.9 is made of two solid cylinders with different materials and radii. Determine the maximum displacement and reactions at the support using ANSYS. Divide each meter of the beam by 50 elements. For Beam 1: E = 210 GPa, D = 5 cm, and for Beam 2: E = 180 GPa, D = 4 cm.

Double Click on the Mechanical APDL Product Launcher icon

Main Menu > Preferences



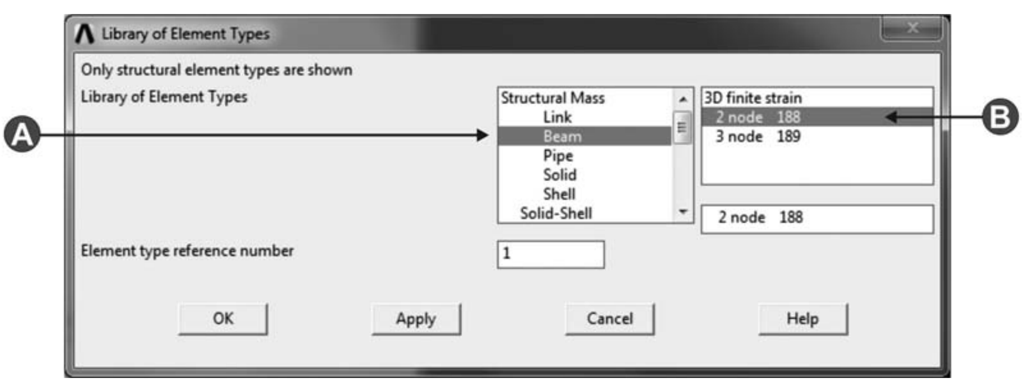
A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

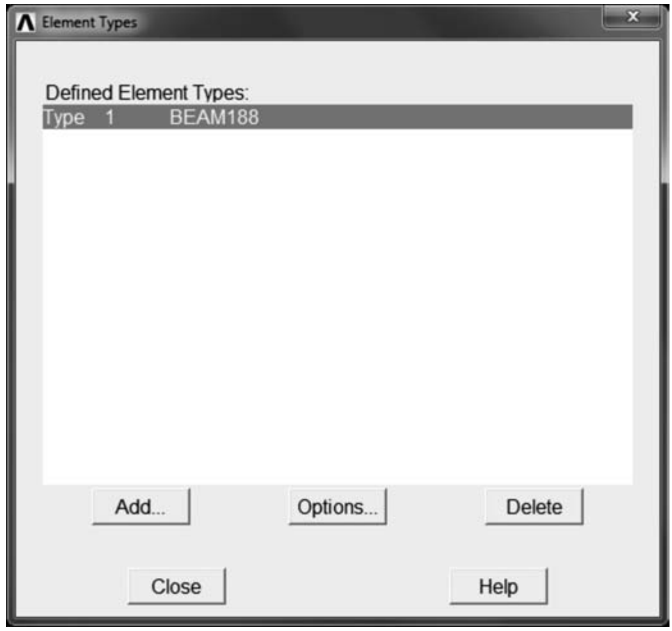
Add...

The beam element can support the moment and the selected element has two nodes.



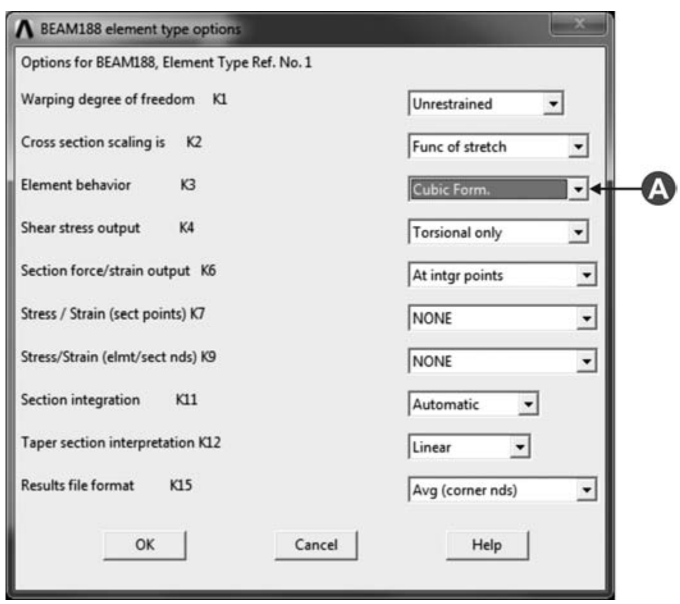
A select Beam
B select 2 node 188

OK



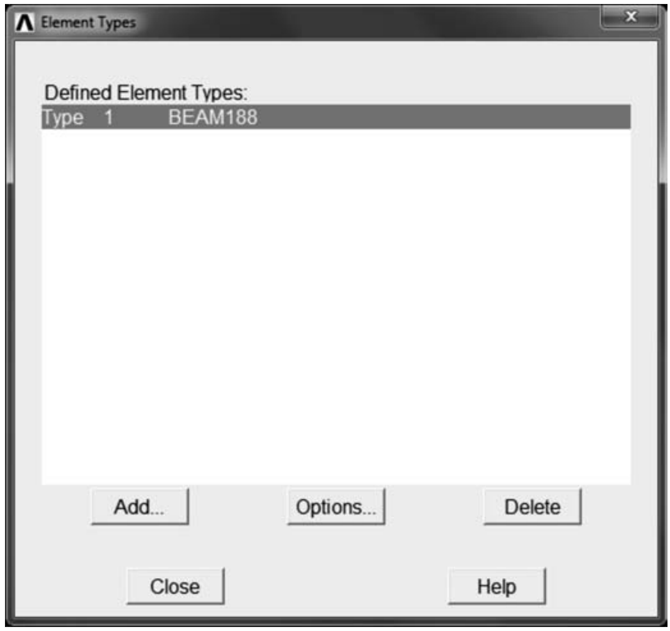
Options…

The illustrated theory for the beam in this chapter is based on the third-order polynomial for deflection function. Therefore, in options, the element behavior should be changed to a cubic form.



A select Cubic Form. in Element behavior K3

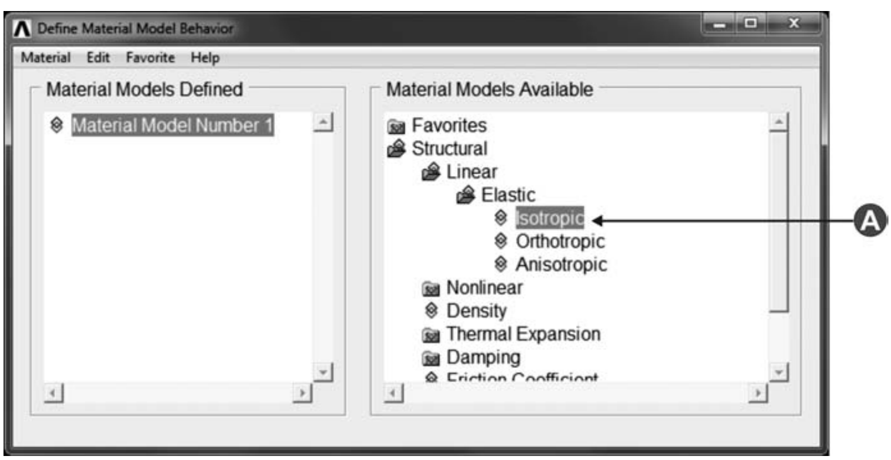
OK



Close

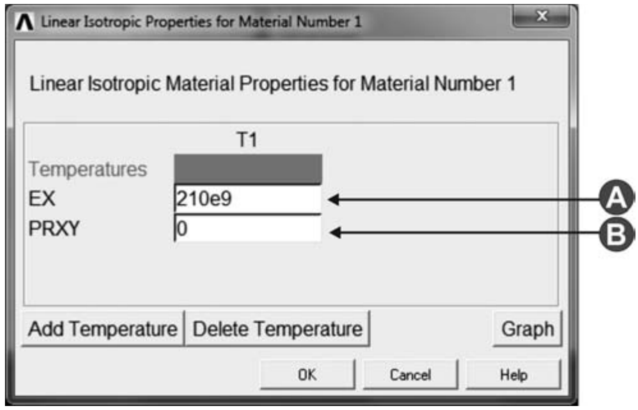
The beam cross-sectional area is specified in the beam section. In this example, two beams with different radii and material properties will be modeled.

Main Menu > Preprocessor > Material Props > Material Models



A Click on Structural > Linear > Elastic > Isotropic

The following window will appear, the modulus of elasticity for the first beam will be specified, and any value for the Poisson ratio is required to avoid an error message.



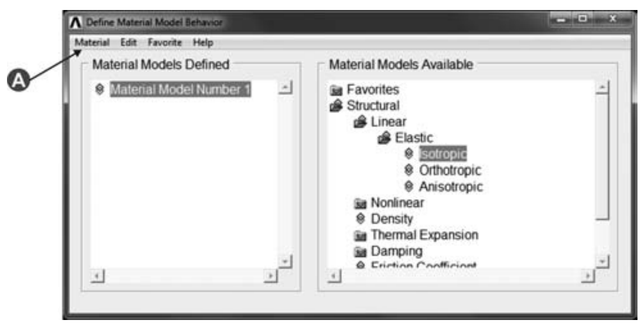
A type 210e9 in EX

B type 0 in PRXY

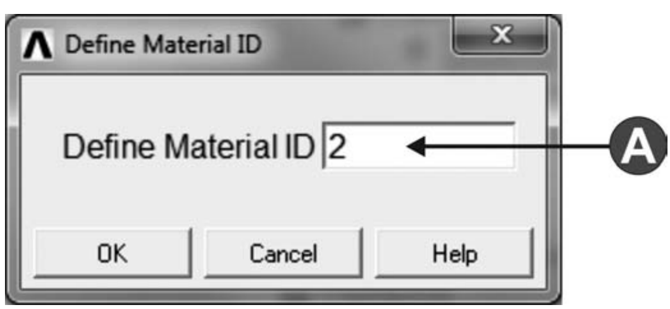
OK

In the following steps, the material of the second beam will be specified. The ID for the first beam is one, while the ID for the second beam is two.

In the Define Material Models Behavior

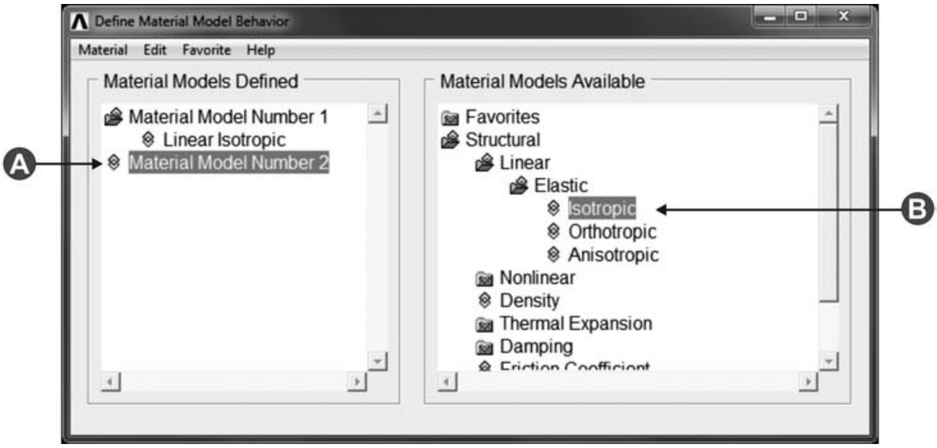


A click on Material, then New Model



A type 2 in Define Material ID

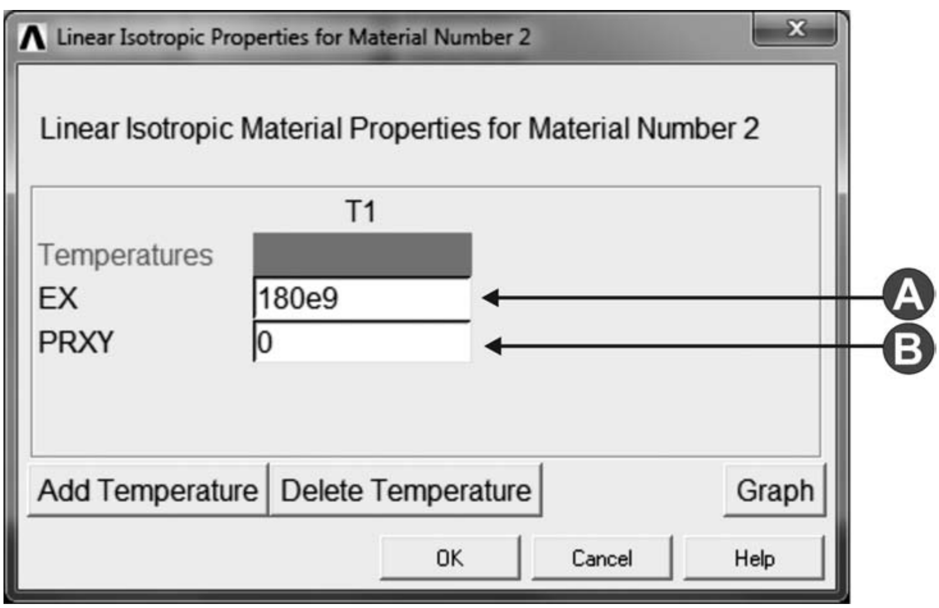
OK



A click Material Model Number 2

B click on Structural > Linear > Elastic > Isotropic

The following window will appear, the modulus of elasticity for the second beam will be specified, and any value for the Poisson ratio is required to avoid an error message.



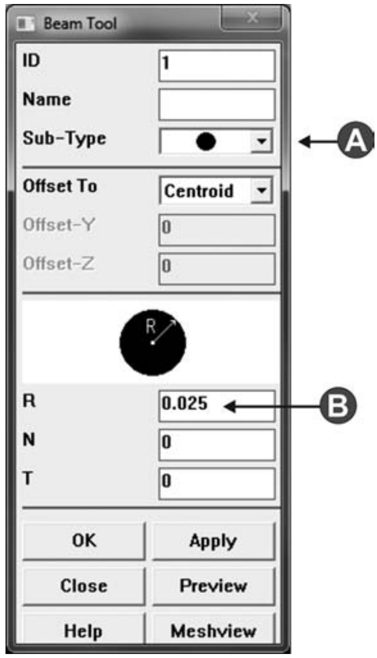
A type 180e9 in EX

B type 0 in PRXY

OK

Close the Material Model Behavior window

Main Menu > Preprocessor > Sections > Beam > Common Sections

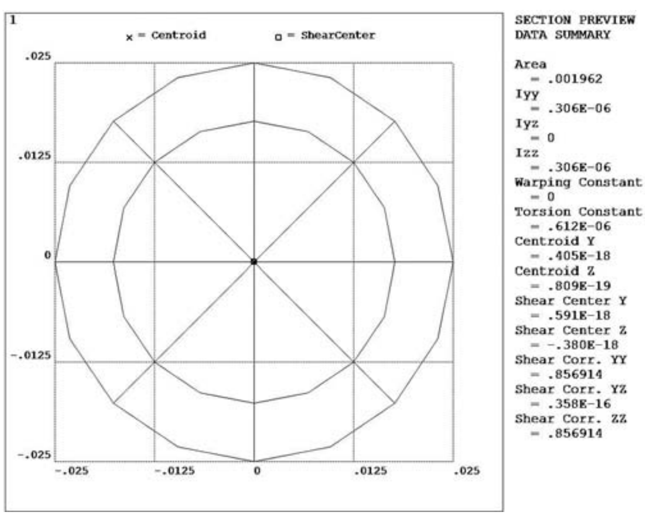


A select the solid cylinder in Sub-Type

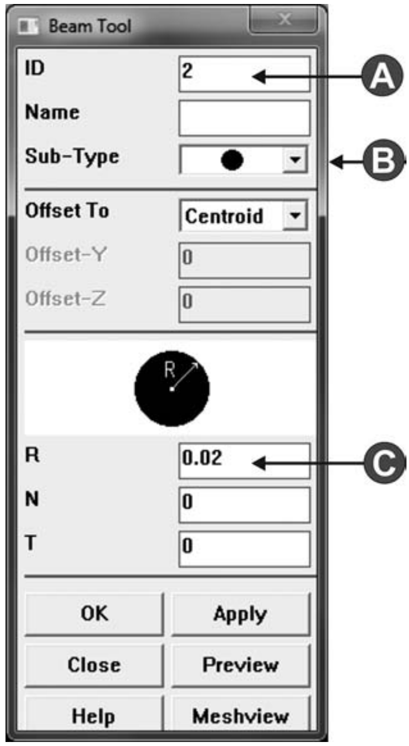
B type 0.025 in R

Apply

Meshview



ANSYS graphics show the properties of the solid cylinder



A type 2 in ID

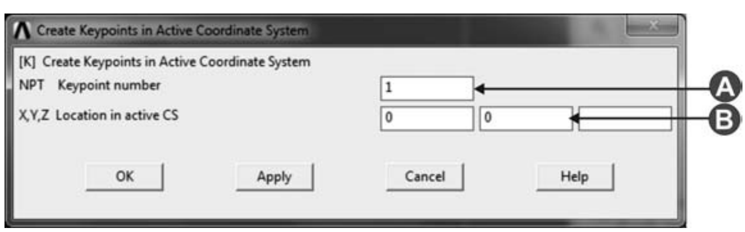
B select the geometry of a solid cylinder in Sub-Type

C type 0.02 in R

OK

The process of modeling the beam structure is started here. First, three keypoints are created followed by creating the lines. The x- and y-coordinates for all keypoints are specified: (0,0) is the coordinate for Keypoint 1, (0,2) for Keypoint 2, and (0,3) for Keypoint 3.

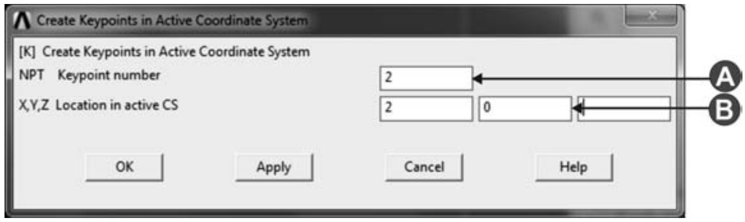
Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS



A type 1 in Keypoint number

B type 0 and 0 in X,Y,Z Location in active CS

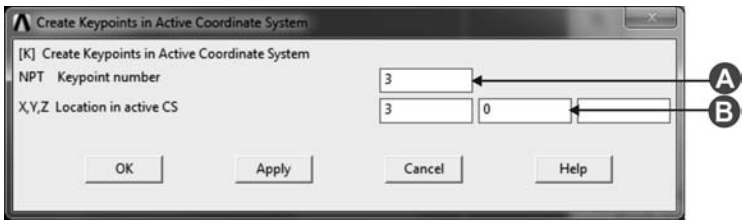
Apply



A type 2 in Keypoint number

B type 2 and 0 in X,Y,Z Location in active CS

Apply



A type 3 in Keypoint number

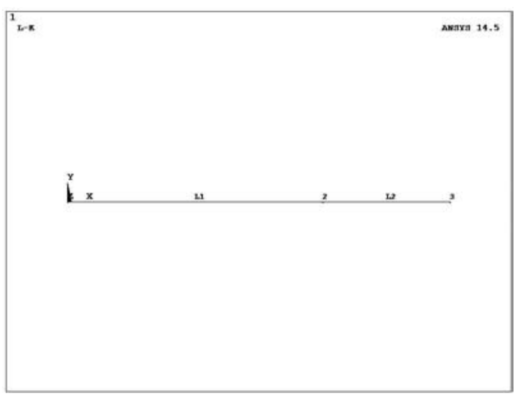
B type 3 and 0 in X,Y,Z Location in active CS

OK

Main Menu > Modeling > Create > Lines > Lines > Straight Line

Click on Keypoint 1 then 2, and then click on Keypoint 2 then 3. In Create Straight Line window, click on

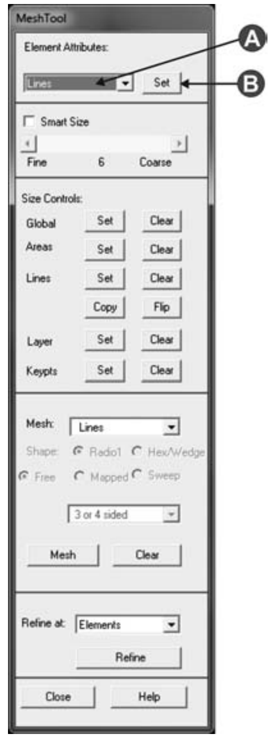
OK



ANSYS graphics show the three keypoints and two lines

The modeling task is ended at this point. Before meshing the lines, the radius and material properties should be specified for the two lines. All lines by default have ID number 1. Hence, only properties of Beam 2 should be changed to ID number 2. Each meter of the beam is divided into 50 elements. Therefore, the first beam will be 100 elements, while the second beam will be 50 elements.

Main Menu > Preprocessor > Meshing > Mesh Tool



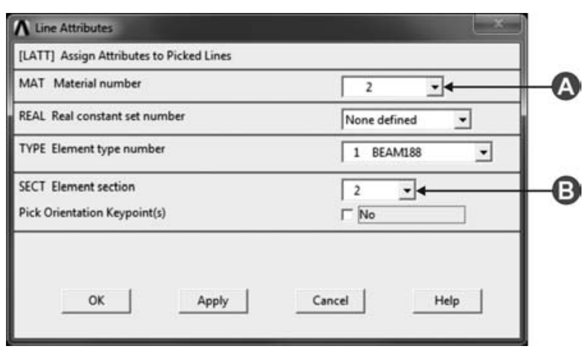
A select Lines

B click on Set

Select Line 2 only. In Line Attributes window, click on

OK

The following Line Attributes window will show up. By selecting number 2 for material number and element section, the properties of number 2 are assigned to the line number 2.

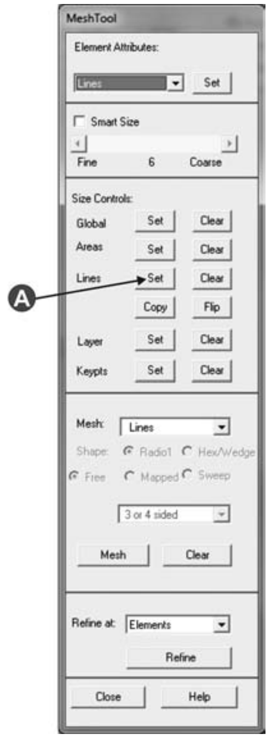


A select 2 in Material number

B select 2 in Element section

OK

Main Menu > Preprocessor > Meshing > Mesh Tool

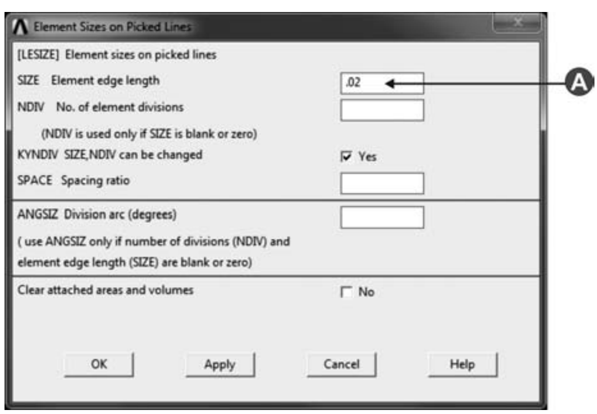


A click on Set in Lines

In Element Size on Picked Lines window, click on

Pick All

In Element Size on Picked Lines window, there are two options: specify the length of elements or the number of element divisions for a line. For the present example, each meter of the beam is divided into 50 elements. Or, the length of the elements is 0.02.



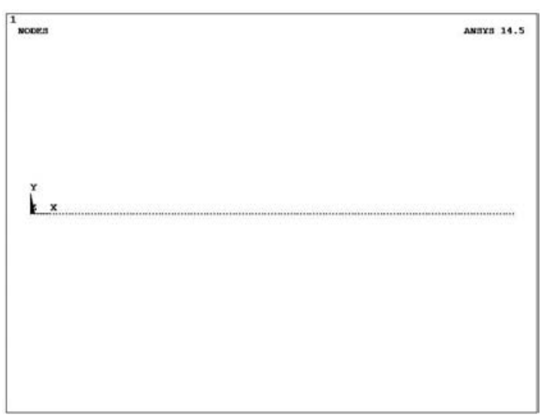
A type 0.02 in Element edge length

OK

A click on Mesh
In Mesh Lines window, click on

Pick All

Utility Menu > Plot > Nodes



ANSYS graphics show the nodes

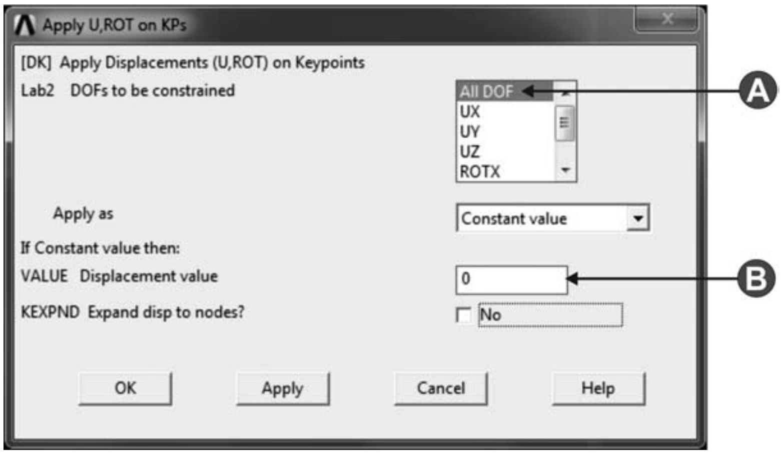
Modeling and meshing are completed at this point. Next, the boundary conditions are applied starting with the support and then the force and moment. This order is not important for the solution. Loads are

applied at the keypoints, which will automatically be transferred to the nodes.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Click on Keypoint number 1. In Apply U,ROT on KPs window, click on

OK



A select All DOF

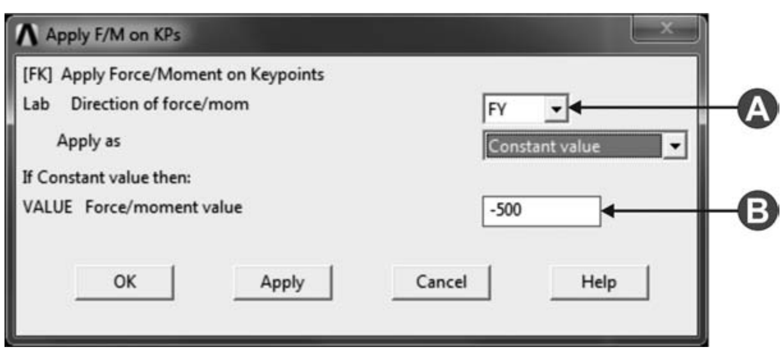
B type 0 in Displacement value

OK

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Click on Keypoint 2. Then, in Apply F/M on KPs window, click on

OK



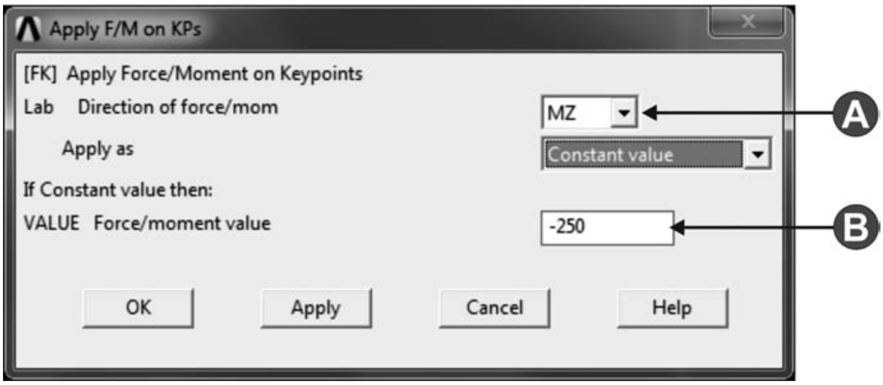
A select FY in Direction of force/mom

B type –500 in Force/moment value

Apply

Click on Keypoint 3. Then, in Apply F/M on KPs window, click on

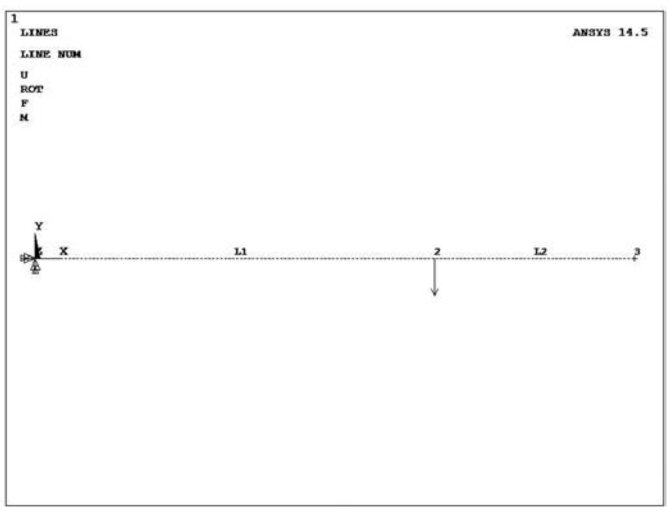
OK



A select MZ in the Direction of force/mom

B type –250 in the Force/moment value

OK



ANSYS graphics show the nodal force and moment with direction

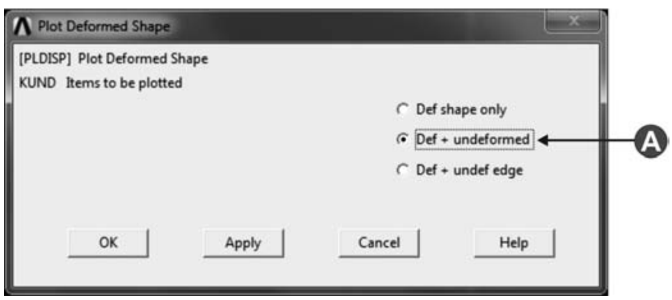
The final step is to run the ANSYS solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem.

Main Menu > Solution > Solve > Current LS

OK

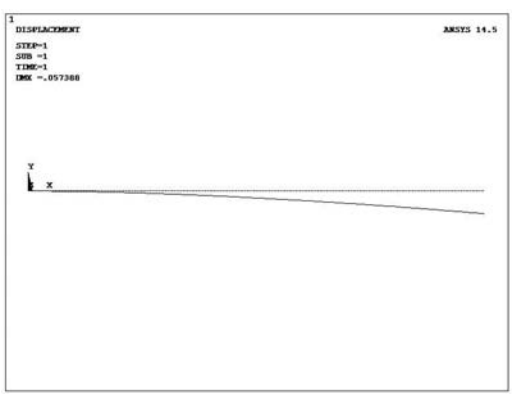
Close

Main Menu > General Postproc > Plot Results > Deformed Shape



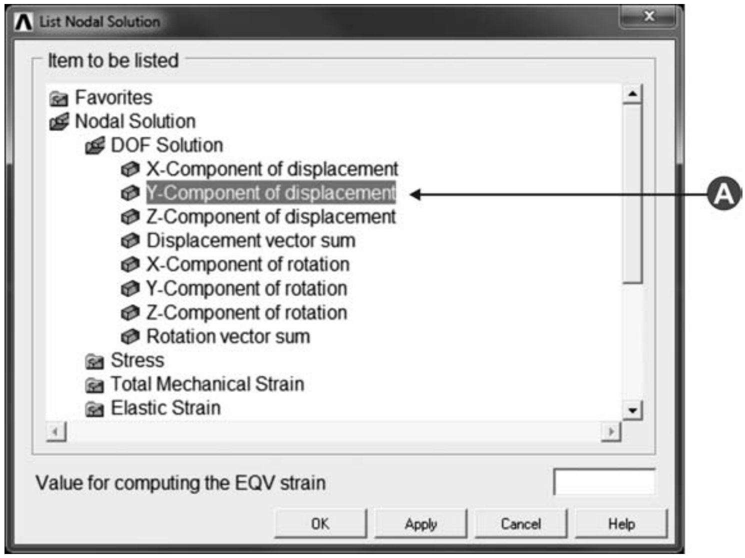
A select Def + undeformed

OK



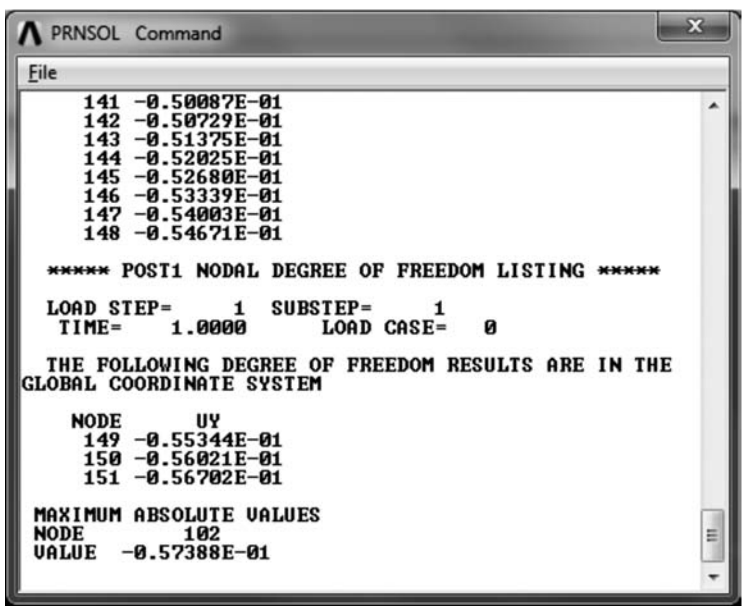
ANSYS graphics show the beams before and after applying the loads

Main Menu > General Postproc > List Results > Nodal Solution



A click on Nodal Solution > DOF Solution > Y-Component of displacement.

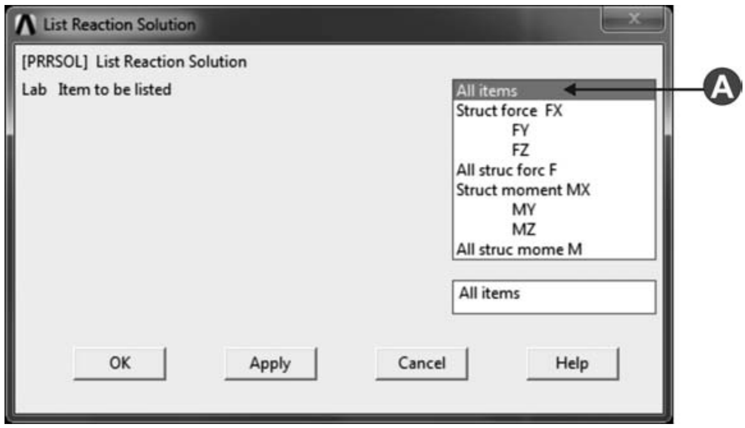
OK



A list of nodal displacements in the y-direction is shown. The maximum nodal displacement is shown at the end of the file, and node 102 has the highest deflection of –0.057388. Comparing with only two elements

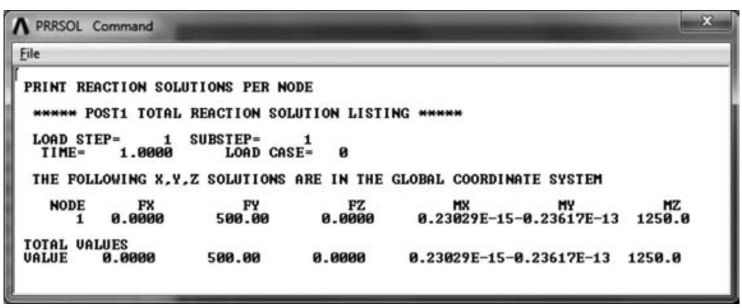
mesh solution in the previous example, maximum nodal displacement was −0.0562 m, which is close to the ANSYS result. The nodal reactions are determined in the following step:

Main Menu > General Postproc > List Results > Reaction Solution



A select All items

OK



The results are for node 1 only, the fixed node boundary, and the moment reaction is listed. Notice that the reaction force and moment at the fixed boundary are equal to in negative value the applied force and moment, which ensures the balance of force and moment in the structure.

2.8 Development of an arbitrary-oriented beam element

Practically, the beam elements are not horizontal, but they can be arbitrary oriented in two- or three-dimensional space to form a complex structure. For two-dimensional space, each node possesses three

degrees of freedom: x-displacement, y-displacement, and rotation. Additionally, there are forces in the x- and y-directions and a bending moment at each node. Figure 2.11 shows an arbitrary-oriented beam element at an angle φ with the global x-axis. At node 1, the local x-direction and y-direction forces are $\bar{F}_{1x}$ and $\bar{F}_{1y}$, respectively, and the bending moment is M_1. At node 2, the local x-direction and y-direction forces are $\bar{F}_{2x}$ and $\bar{F}_{2y}$, respectively, and the bending moment is M_2. These forces and bending moments cause the beam to bend. Node 1 is displaced in the local x- and y-directions $\bar{d}_{1x}$ and $\bar{d}_{1y}$, and rotated at an angle θ_1, while node 2 is displaced in the local x- and y-directions $\bar{d}_{2x}$ and $\bar{d}_{2y}$, and rotated at an angle θ_2.

The beam has modulus of elasticity E, uniform cross-sectional area A, moment of inertia I, and initial length L. Since the solution for axial displacements, the transverse deflection, and rotation is independent, the bar and horizontal beam are combined to form an axial flexural beam element. The bar element has a stiffness equation of

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{2x} \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{2x} \end{Bmatrix} \tag{2.110}$$

and a horizontal beam element has a stiffness equation of

$$\begin{Bmatrix} \bar{F}_{1y} \\ M_1 \\ \bar{F}_{2y} \\ M_2 \end{Bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} \bar{d}_{1y} \\ \theta_1 \\ \bar{d}_{2y} \\ \theta_2 \end{Bmatrix} \tag{2.111}$$

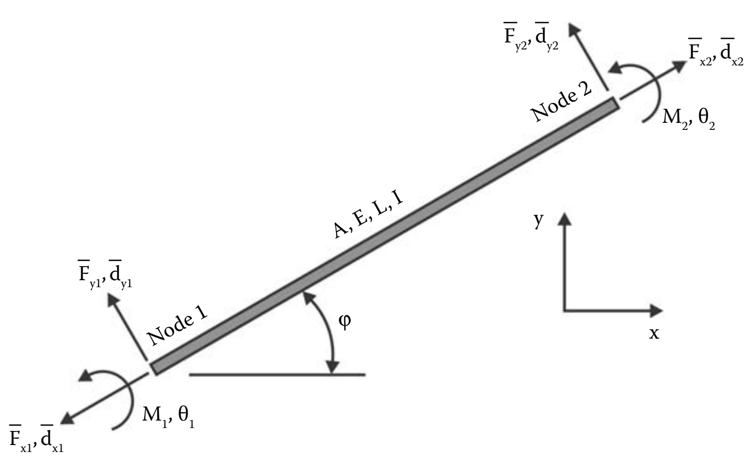


FIGURE 2.11 An arbitrary-oriented beam element at an angle φ.

Hence, the stiffness equations of bar and beam are combined as

$$\begin{Bmatrix} \bar{F}_{1x} \\ \bar{F}_{1y} \\ M_1 \\ \bar{F}_{2x} \\ \bar{F}_{2y} \\ M_2 \end{Bmatrix} = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EI}{L} & 0 & 0 & \frac{EI}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{Bmatrix} \bar{d}_{1x} \\ \bar{d}_{1y} \\ \theta_1 \\ \bar{d}_{2x} \\ \bar{d}_{2y} \\ \theta_2 \end{Bmatrix} \tag{2.112}$$

The axial flexural beam element is lying along the local x-axis and the local x-axis is oriented at an angle φ measured counterclockwise from the global x-axis. The transformation matrix [T] is used to transfer the local nodal forces and moments, and nodal displacements and rotation from local coordinate to the global one. The nodal displacements and rotation vector in a symbolic form are as follows:

$$\{\bar{d}\} = [T]\{d\} \tag{2.113}$$

and the nodal forces and bending moment vector in a symbolic form are

$$\{\bar{F}\} = [T]\{F\} \tag{2.114}$$

where

$$[T] = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & \sin\theta & 0 \\ 0 & 0 & 0 & -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{2.115}$$

Since $\{\bar{F}\} = [\bar{K}]\{\bar{d}\}$, and using the transformation matrix, the following equation is obtained:

$$[T]\{F\} = [\bar{K}][T]\{d\} \tag{2.116}$$

By multiplying both sides of Equation 2.116 by the inverse of the transformation matrix, the global force vector is expressed as

$$\{F\} = [T]^{-1}\left[\bar{K}\right][T]\{d\} \tag{2.117}$$

Since the transformation matrix is orthogonal, its transport and inverse are equal, and therefore,

$$\{F\} = [T]^{T}\left[\bar{K}\right][T]\{d\} \tag{2.118}$$

Since the global force vector, $\{F\} = [K]\{d\}$, is equal to the global force vector in Equation 2.118, the global stiffness matrix can be written as

$$[K] = [T]^{T}\left[\bar{K}\right][T] \tag{2.119}$$

The stiffness matrices, force vectors, and displacement vectors are assembled as follows:

$$[K] = \sum_{e=1}^{N}\left[K^{(e)}\right] \tag{2.120}$$

$$\{F\} = \sum_{e=1}^{N}\left\{F^{(e)}\right\} \tag{2.121}$$

$$\{d\} = \sum_{e=1}^{N}\left\{d^{(e)}\right\} \tag{2.122}$$

Finally, the global stiffness matrix is used to relate the global nodal force vector to global displacement vector for the entire structure by

$$\{F\} = [K]\{d\} \tag{2.123}$$

2.9 Distributed load on a beam element

Unlike the bar elements, beam elements can support a distributed load. The distributed load can be uniform, varying linearly, or randomly distributed. The work equivalence method is used to replace the distributed load by concentrated nodal forces and moments. Consider the beam subjected to a distributed load, as shown in Figure 2.12a.

The work due to the distributed load is expressed as

$$W_{dist} = \int_{0}^{L} P(x)\,v(x)\,dx \tag{2.124}$$

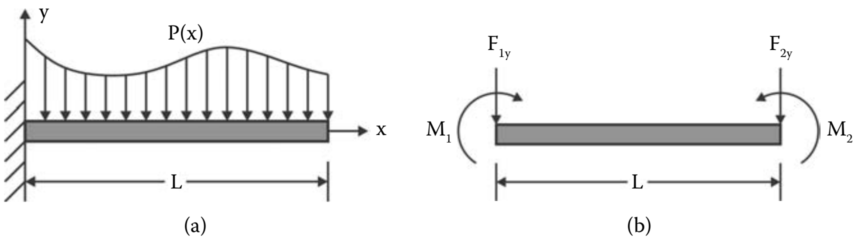


FIGURE 2.12 (a) Beam that is subjected to a distributed load and (b) its equivalent nodal forces and moments.

where P(x) is the distributed load on beam with length L and v(x) is the transverse displacement of a beam. The work due to the discrete nodal forces and moments is given by

$$W_{disc} = M_1\theta_1 + M_2\theta_2 + F_{1y}d_{1y} + F_{2y}d_{2y} \tag{2.125}$$

Hence, the distributed load can be replaced by equivalent nodal forces and moments, as shown in Figure 2.12b. M_1, M_2, F_{1y}, and F_{2y} are determined by setting $W_{dist} = W_{disc}$, as follows:

$$\int_0^L P(x)\left(\left[\frac{2}{L^3}\left(d_{1y}-d_{2y}\right)+\frac{1}{L^2}(\theta_1+\theta_2)\right]x^3+\left[-\frac{3}{L^2}\left(d_{1y}-d_{2y}\right)-\frac{1}{L}(2\theta_1-\theta_2)\right]x^2 \right.$$
$$\left. +\theta_1 x + d_{1y}\right)dx = M_1\theta_1 + M_2\theta_2 + F_{1y}d_{1y} + F_{2y}d_{2y} \tag{2.126}$$

For example, consider the case where the applied distributed pressure is uniform on a horizontal beam, $P(x) = -P_o$. Integrating Equation 2.126 with respect to x between 0 and L gives the following:

$$\begin{aligned}
&-\left[\frac{2}{L^3}\left(d_{1y}-d_{2y}\right)+\frac{1}{L^2}(\theta_1+\theta_2)\right] \\
&\quad \frac{L^4}{4}-\left[-\frac{3}{L^2}\left(d_{1y}-d_{2y}\right)-\frac{1}{L}(2\theta_1-\theta_2)\right] \\
&\quad \frac{L^3}{3}-\theta_1\frac{L^2}{2}-d_{1y}\,L = M_1\theta_1 + M_2\theta_2 + F_{1y}d_{1y} + F_{2y}d_{2y}
\end{aligned} \tag{2.127}$$

then,

$$F_{1y} = -\frac{P_oL}{2} \tag{2.128}$$

$$M_1 = -\frac{P_oL^2}{12} \tag{2.129}$$

$$F_{2y} = -\frac{P_o L}{2} \tag{2.130}$$

$$M_2 = \frac{P_o L^2}{12} \tag{2.131}$$

Or, in matrix form,

$$\{F_{dist}\} = \begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \end{Bmatrix} = \begin{Bmatrix} -\dfrac{P_o L}{2} \\ -\dfrac{P_o L^2}{12} \\ -\dfrac{P_o L}{2} \\ \dfrac{P_o L^2}{12} \end{Bmatrix} \tag{2.132}$$

In general, the concentrated nodal forces and moments, which are equivalent to the distributed load, are added to the applied nodal forces and moments to obtain the actual global nodal forces and moments as follows:

$$[K]\{d\} = \{F\} + \{F_{dist}\} \tag{2.133}$$

For a linearly increasing pressure along a beam, $P(x) = -P_o x$, the distributed load can be replaced by equivalent nodal forces and moments, M_1, M_2, F_{1y}, and F_{2y}. Equation 2.126 is integrated, and the equivalent nodal forces and moments are determined by setting $W_{dist} = W_{disc}$, and the final result is

$$\{F_{dist}\} = \begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \end{Bmatrix} + \begin{Bmatrix} -\dfrac{3\ PL}{20} \\ \dfrac{PL^2}{30} \\ -\dfrac{7PL}{20} \\ -\dfrac{5PL^2}{96} \end{Bmatrix} \tag{2.134}$$

2.10 Analyzing beam structure under a transient loading using ANSYS

For the three-dimensional beam structure shown in Figure 2.13, a transient harmonic force, with a frequency of $f = 1/500\ s^{-1}$ and amplitude of $A = 100$ kN, is applied at the shown location. Use ANSYS to determine

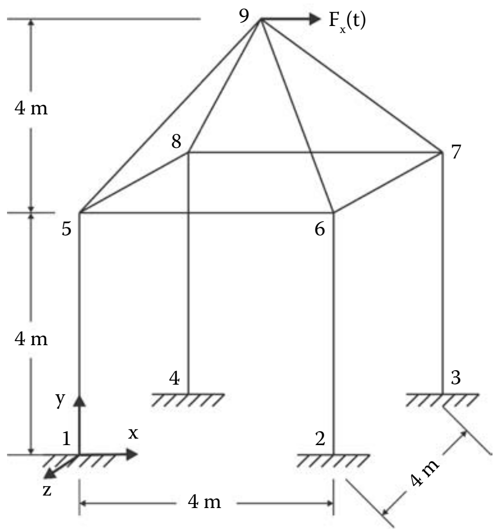


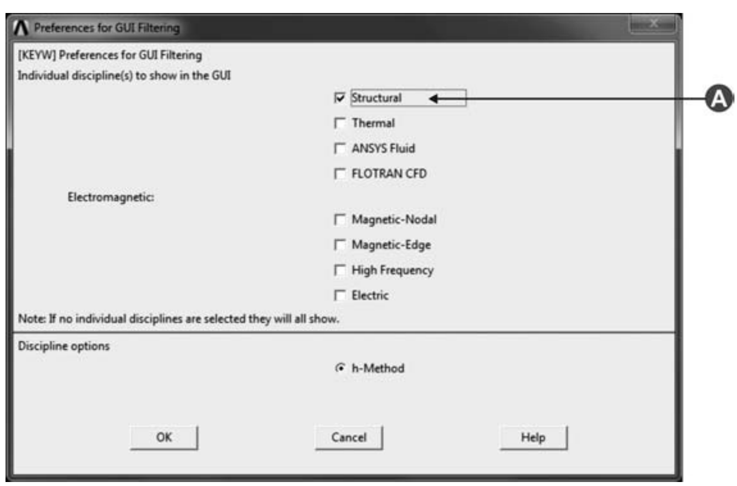
FIGURE 2.13 A structure under transient loading.

the displacement at the point where the force is applied as a function of time. Also, create an animation file. The total time for the loading process is 2500 seconds. The beams are made of circular cross-sectional pipe, E = 200 GPa, and with inner and outer radii of $R_i = 0.12$ m, $R_o = 0.15$ m, respectively. Mesh each beam by 10 elements. Use the following formula to simulate the applied transient force:

$$F_x(t) = A \sin(2\pi ft)$$

Double clicks on the Mechanical APDL Product Launcher icon

Main Menu > Preferences



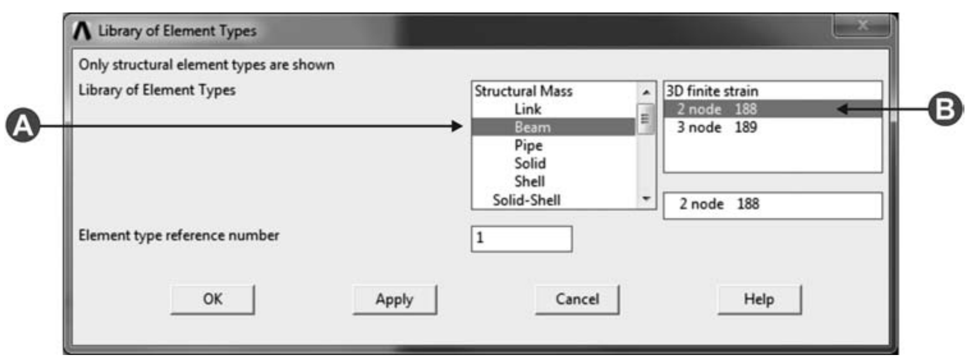
A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add...

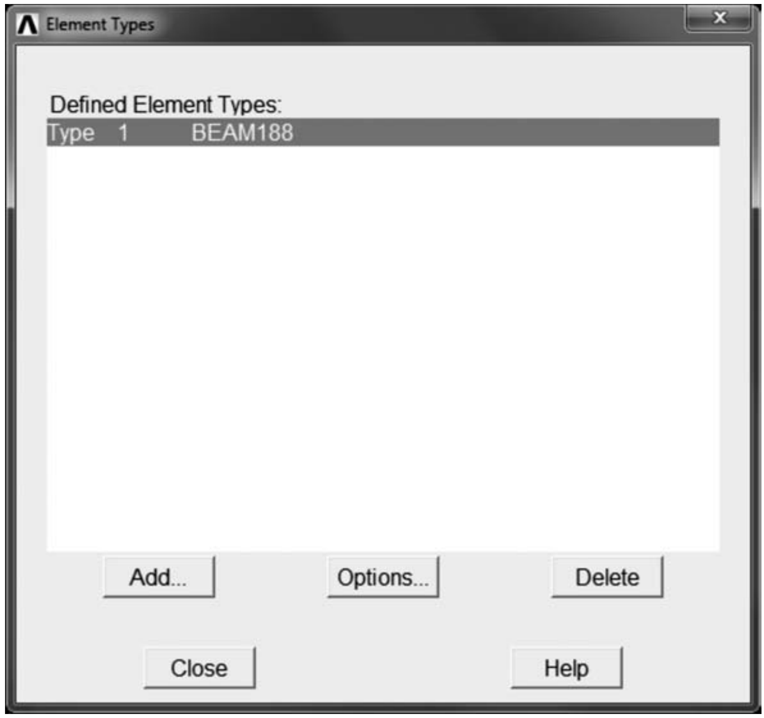
The beam elements can support the moment and have three-dimensional capability.



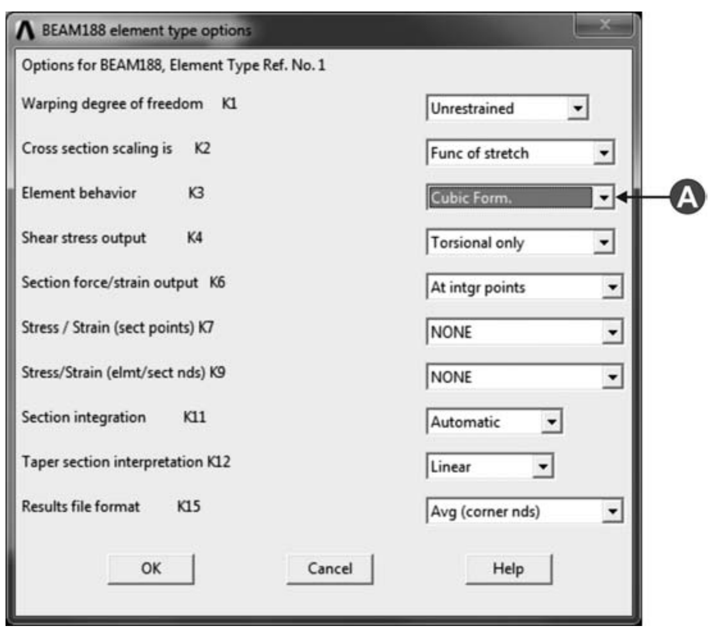
A select Beam

B select 2 node 188

OK

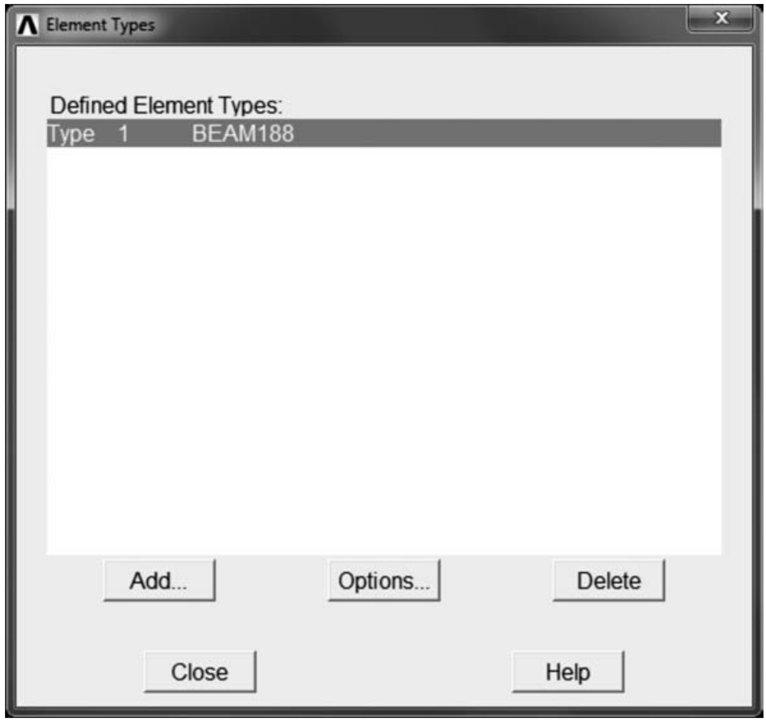


Options…



A select Cubic Form. in Element behavior K3

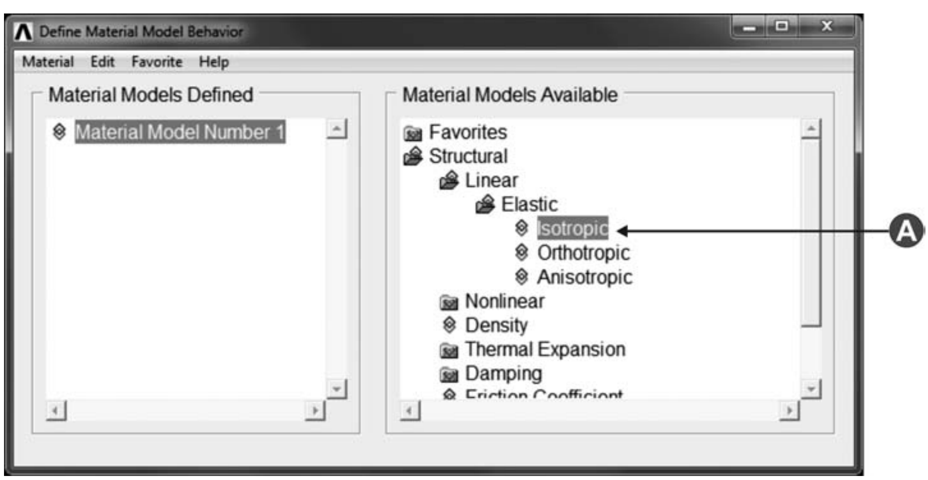
OK



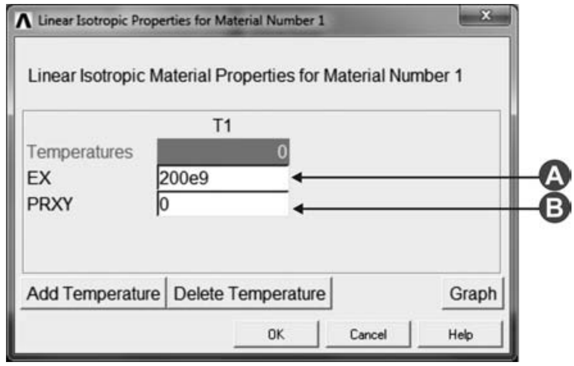
Close

For material properties, only the modulus of elasticity is required. In this example, all beams have the same geometry and material properties.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic



A type 200e9 in EX

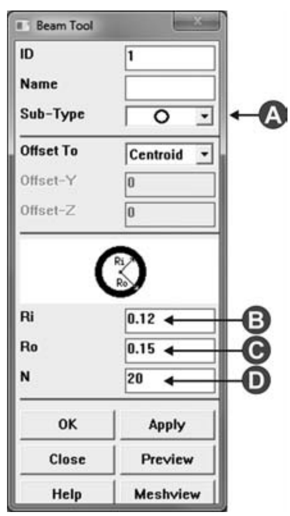
B type 0 in PRXY

OK

Close the Material Model Behavior window

The beam cross-sectional properties are specified in sections. The geometry of the cross-sectional area of the pipe will be selected and radii are specified. The number of divisions along the circumference is 20 to ensure a smooth circular cross section.

Main Menu > Preprocessor > Sections > Beam > Common Sections



A select the cross-sectional geometry of a pipe

B type 0.12 in Ri

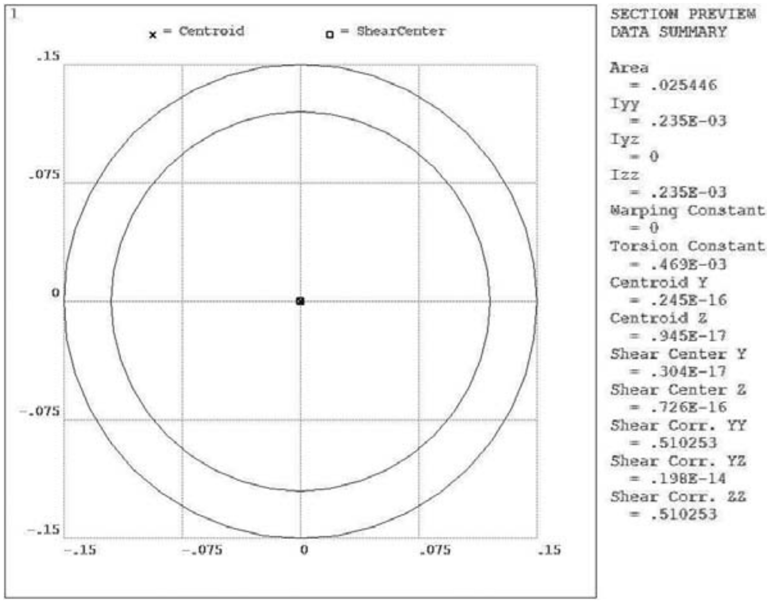
C type 0.15 in Ro

D type 20 in N

Apply

Meshview

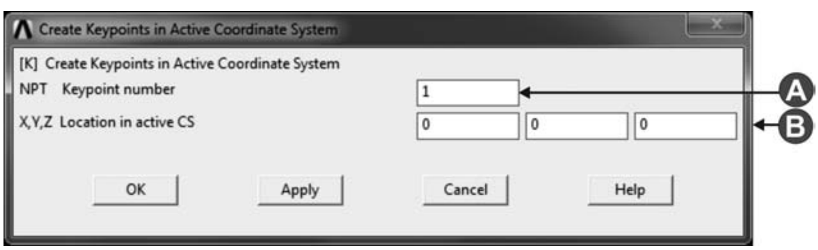
OK



ANSYS graphics will show geometrical properties of the pipe

The process of modeling the beam structure is started here. First, keypoints are created, followed by creating the line connecting the keypoints. The x- and y-coordinates of each keypoint are specified. The coordinate for Keypoint 1 is (0,0,0).

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS



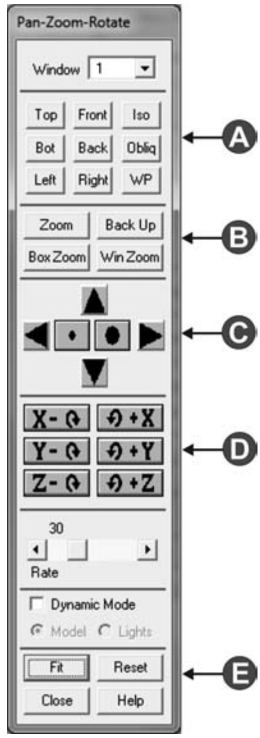
A type 1 in Keypoint number

B type 0, 0, and 0 in X,Y,Z Location in active CS

Apply

Similarly for other keypoints, (4,0,0) is the coordinate for Keypoint 2; (4,0,–4) is for Keypoint 3; (0,0,–4) is for Keypoint 4; (0,4,0) is for Keypoint 5; (4,4,0) is for Keypoint 6; (4,4,–4) is for Keypoint 7; (0,4,–4) is for Keypoint 8; and (2,8,–2) is for Keypoint 9. To view the keypoints at different directions, the Pan-Zoom-Rotate is used:

Utility Menu > PlotCtrls > Pan-Zoom-Rotate



A this group is for viewing a geometry at different directions

B this group is for zooming at a specific region in ANSYS graphics

C this group is for zooming and panning

D this group is for rotating the geometry

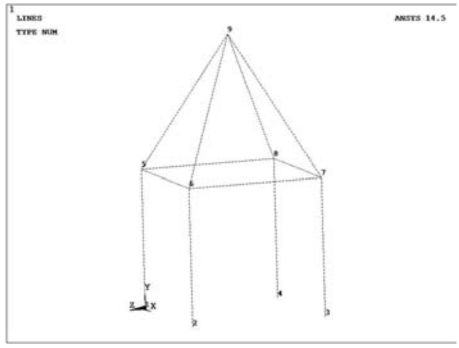
E this group is for fitting the geometry, resetting, closing, and helping

Close

Main Menu > Modeling > Create > Lines > Lines > Straight Line

click on Keypoint 1 then 5
click on Keypoint 2 then 6
click on Keypoint 3 then 7
click on Keypoint 4 then 8
click on Keypoint 5 then 6
click on Keypoint 6 then 7
click on Keypoint 7 then 8
click on Keypoint 8 then 5
click on Keypoint 5 then 9
click on Keypoint 6 then 9
click on Keypoint 7 then 9
click on Keypoint 8 then 9

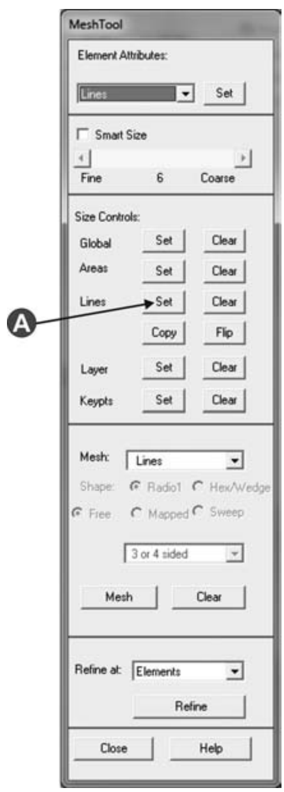
OK



ANSYS graphics show the keypoints and lines

The modeling task is ended at this point. Before meshing the lines, each meter of the beam is divided into 10 elements. Therefore, the beams' element length is 0.1 m.

Main Menu > Preprocessor > Meshing > Mesh Tool

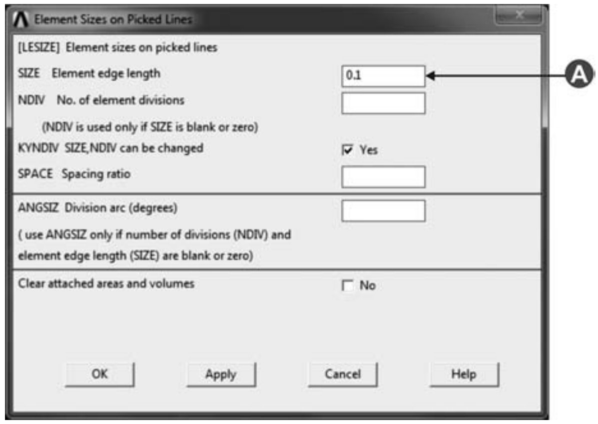


A click on Set in Lines
In Element Sizes on Picked Lines window, click on

Pick All

The following window will show up. In Element Sizes on Picked Lines window, there are two options, specifying either the length of

elements or the number of element divisions. For the present example, each meter of the beam should be divided into 10 elements. Alternatively, the length of the element is 0.1.



A type 0.1 in Element edge length

OK

A click on Mesh

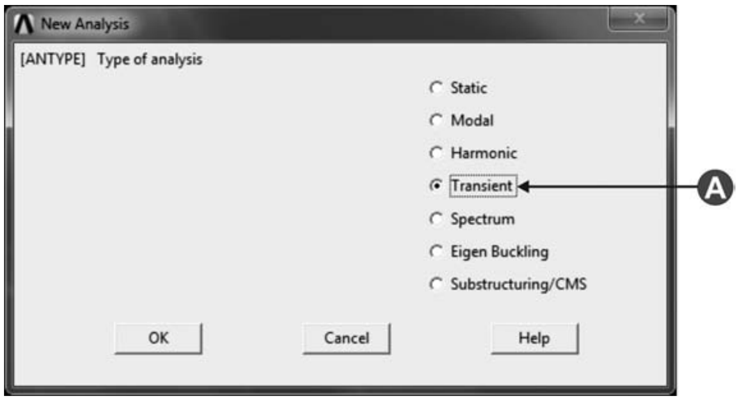
In Mesh Lines window, click on

Pick All

Utility Menu > Plot > nodes

Modeling and meshing is completed at this point. Next, the solution is switched to transient, and unsteady parameters will be specified. The boundary conditions are applied starting with the supports, and then the transient force. This order is not important for solution.

Main Menu > Solution > Analysis Type > New Analysis



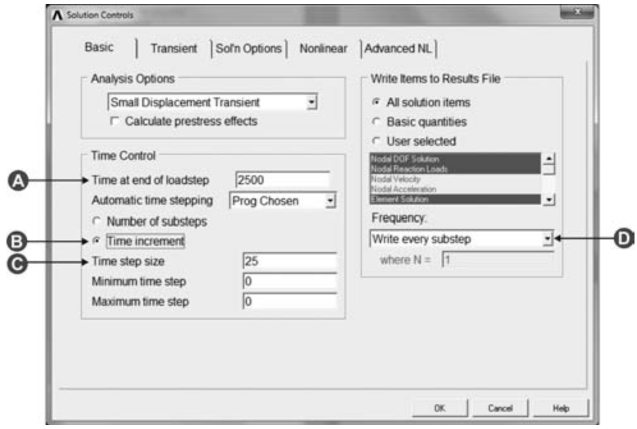
A select Transient

OK

OK

The total time duration for the process is 2500 seconds. To ensure that the obtained results are accurate, the total time duration is divided into 100 time steps. Hence, the time step for this problem is 25 seconds. Results for all time steps are stored by selecting Write every substep option in Frequency. Otherwise, only the result at time step 2500 seconds will be stored.

Main Menu > Solution > Analysis Type > Sol'n Control



A type 2500 in Time end of loadstep

B select Time increment

C type 25 in Time step size

D select Write every substep

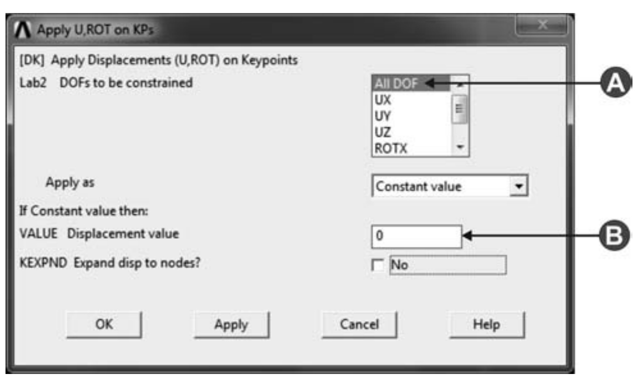
OK

The function editor is used to apply transient force formula as a nodal force. This technique is simple and convenient for this problem since an equation for the force is given. The function is plotted to ensure correct typing.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Click on Keypoint numbers 1, 2, 3, and 4. In Apply U,ROT on KPs window, click on

OK

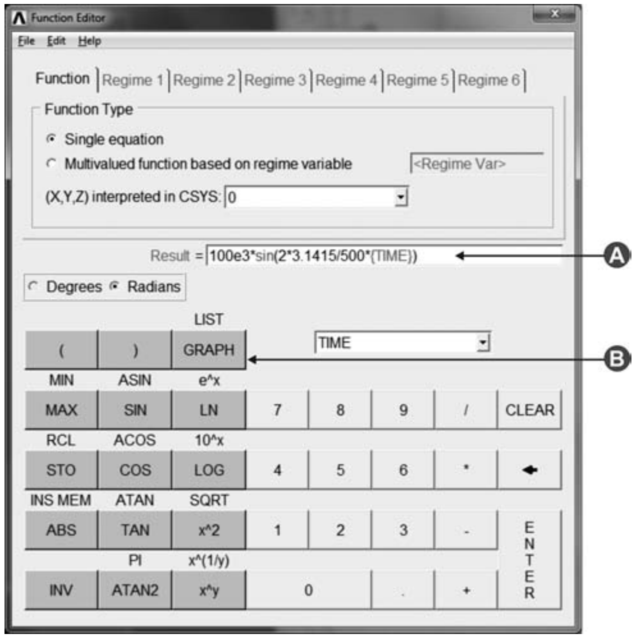


A select All DOF

B type 0 in Displacement value

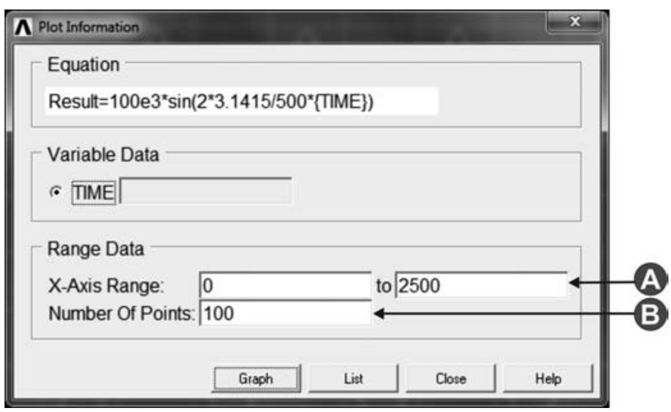
OK

Main Menu > Solution > Define Loads > Apply > Functions > Define/Edit



A type the equation: 100e3*sin(2*3.1415/500*{TIME})

B click on GRAPH



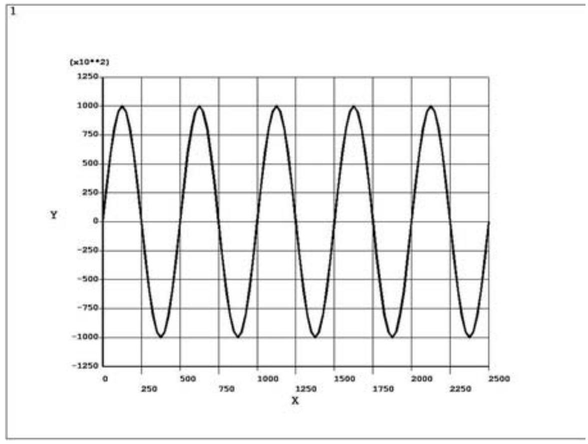
A type 0 and 2500 in X-Axis Range

B type 100 in Number Of Points

Graph

Close

Then 0 and 2500 are the range of the data on the x-axis, while 100 is the number of points to be plotted. Number Of Points has nothing to do with the accuracy of the results, and a higher number will just create a smooth plot. The equation should be saved.



ANSYS graphics show the oscillating function of the force

In Equation Editor window, click on File then Save

Save the file as ForceX, and this file name is optional. After saving the function, it is required to load it to the ANSYS solution using the read file.

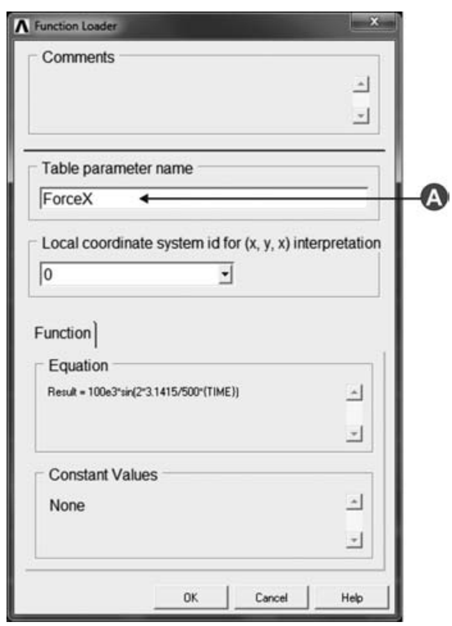
Save

Close the Function Editor window

Main Menu > Solution > Define Load > Apply > Functions > Read File

Select the file ForceX from Open window, and then click on

Open



A type ForceX in the Table parameter name. This name is optional, and should not be the same as the file name of the function.

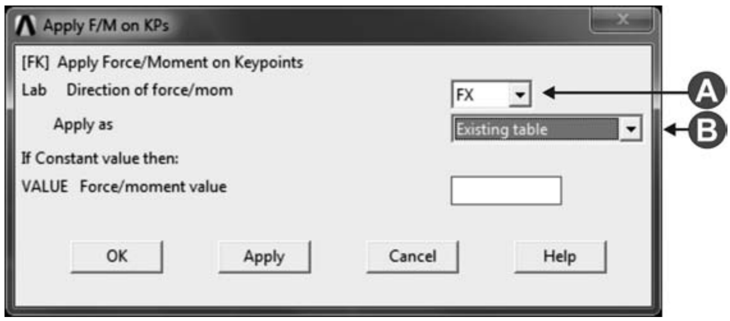
OK

Utility Menu > Plot > Lines

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Click on Keypoint number 9. In Apply F/M on KPs window, click on

OK

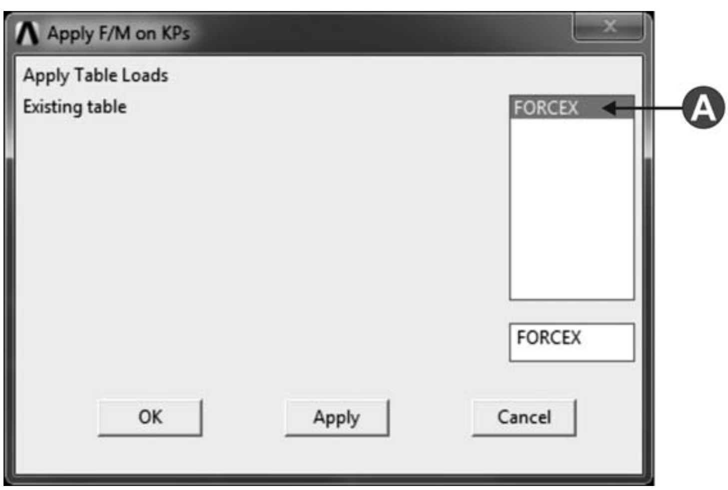


A select FX in the Direction of force/mom

B select Existing table

OK

The following window will show up to select the function.



A select FORCEX

OK

The final step is to run the ANSYS solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem.

Main Menu > Solution > Solve > Current LS

OK

Close

An animation of the deformed structure from time = 0 to 500 seconds can be easily accomplished using animate in the PlotCtrls option in Utility Menu. The Number of the frames in the animation over time is the number of pictures in the avi file, while the animation time delay is the display period between two pictures. Fifty frames produce a good resolution file. With 0.5 s, the file duration is 250 s (500 frames × 0.5 delay).

Main Menu > General Postproc

Utility Menu > PlotCtrls > Animate > Over time …

A type 50 in Number of animation frames

B select Time Range

C type 0 and 500 in Range Minimum, Maximum

D select DOF solution

E select Deformed Shape

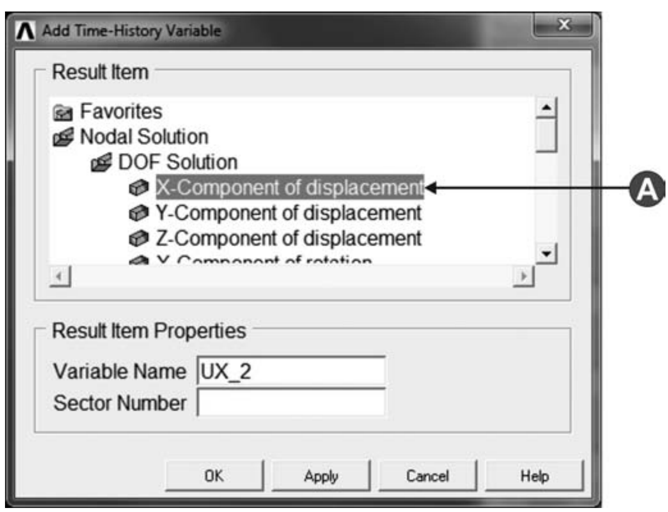
OK

ANSYS creates an animation for the deformation.

The displacement at any point in the structure can be displayed as a function of time using the time history postprocessor. For this problem, the displacement at the point where the force is applied as a function of time is required.

Main Menu > TimeHist Postpro

A click on the green + button

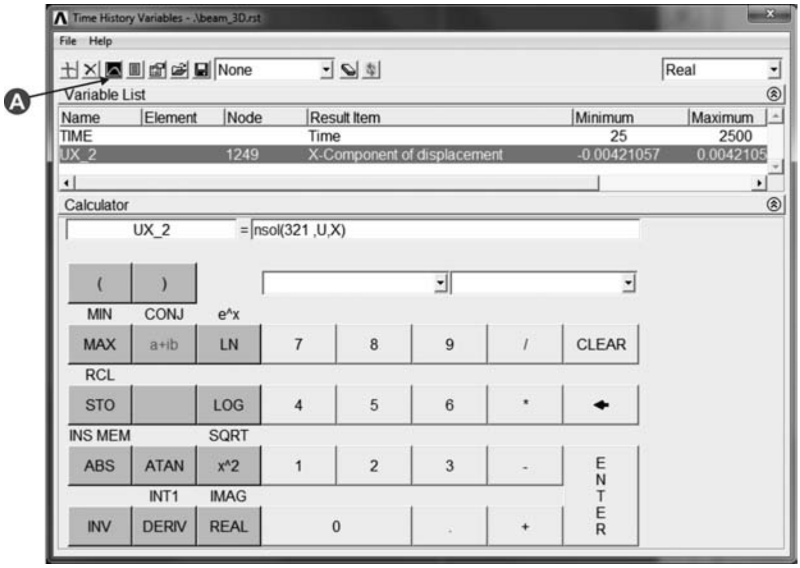


A click on Nodal Solution > DOF Solution > X-Component of displacement.

OK

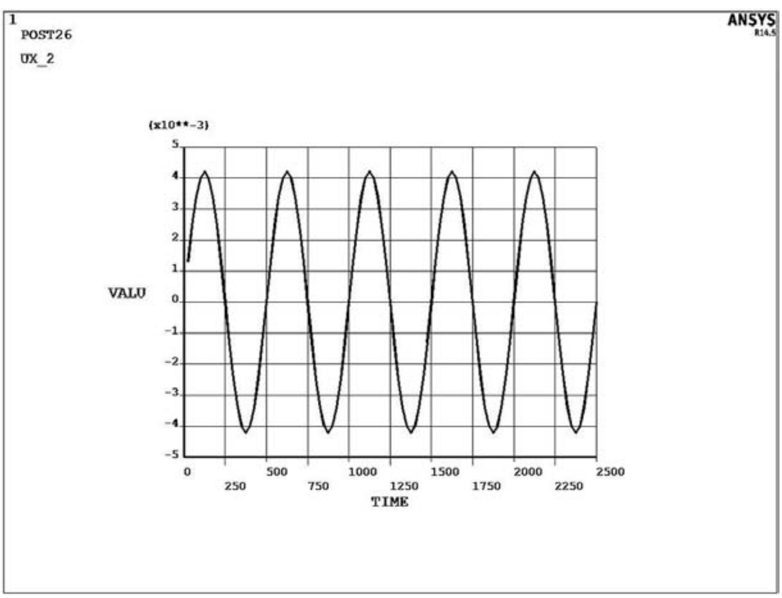
Click on the Keypoint number 9, where the force is applied, and in Node for Data window, click on

OK



A click on the graph button

OK



ANSYS graphics show the displacement history of the selected location

As shown, the maximum displacement in the x-direction is 0.00420747 m.

PROBLEM 2.1

For the bar structures shown in Figure 2.14a through 2.14d, determine the horizontal and vertical nodal displacements, using the finite element method, given that E = 210 GPa, A = 2.5 × 10^{-4} m^2, F = 0.5 kN, and L = 0.5 m.

PROBLEM 2.2

For the bar structures shown in Figure 2.14a through 2.14d, determine the horizontal and vertical nodal displacements, using ANSYS, given that E = 210 GPa, A = 2.5 × 10^{-4} m^2, F = 0.5 kN, and L = 0.5 m.

PROBLEM 2.3

The horizontal beam structures, shown in Figure 2.15a through 2.15d, are made of different materials and cross-sectional areas. Determine the nodal displacements and slopes, using the finite element method, given E_1 = 270 GPa, E_2 = 215 GPa, E_3 = 170 GPa, I_1 = 2.5 × 10^{-6} m^4, I_2 = 5 × 10^{-6} m^4, and I_3 = 7.5 × 10^{-6} m^4.

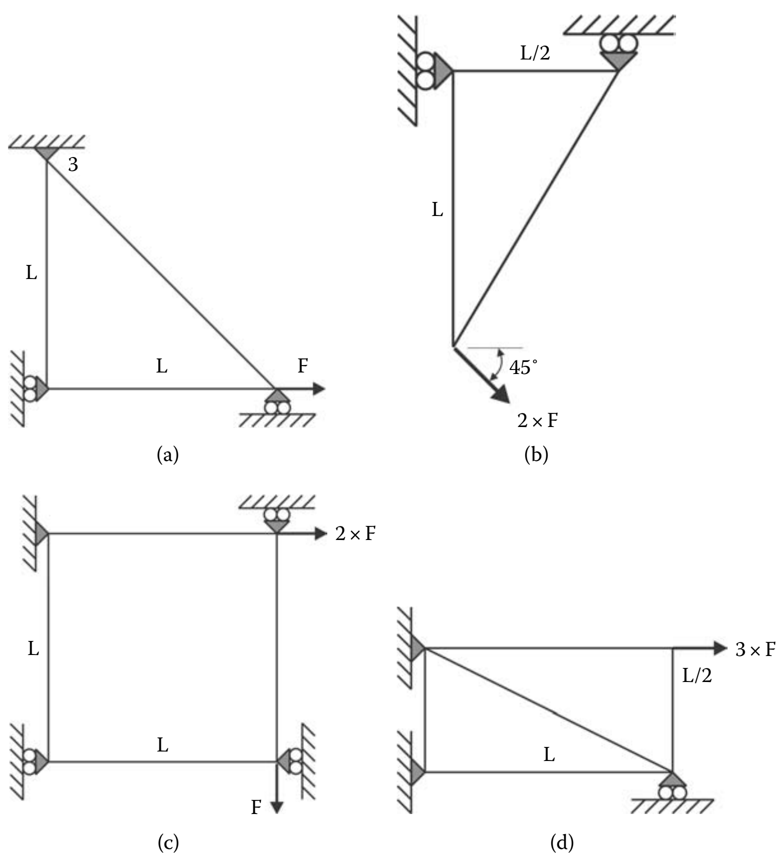


FIGURE 2.14 Bar structures.

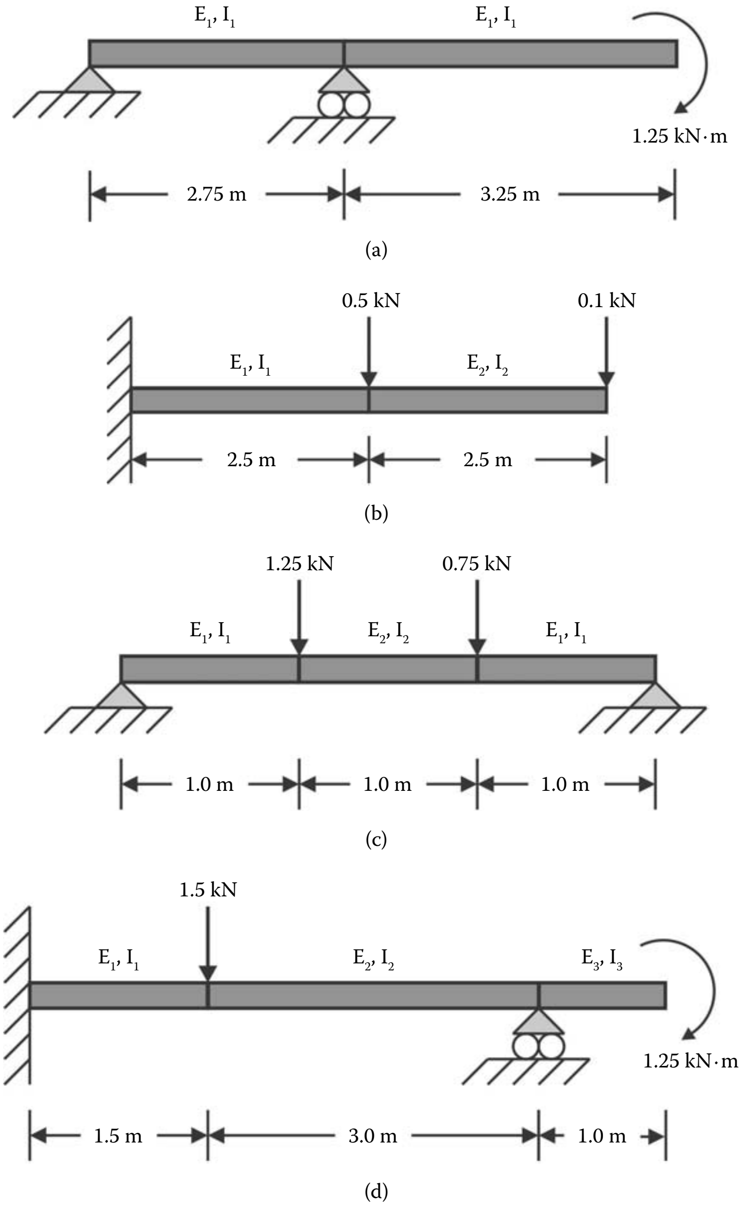


FIGURE 2.15 Horizontal beam structures.

PROBLEM 2.4

The beam structures, shown in Figure 2.16a through 2.16d, are made of solid cylinders with different materials and radii. Determine the maximum nodal displacement and reaction at the support(s) using ANSYS. Divide each meter of the beam by 25 elements, given $E_1 = 170$ GPa, $E_2 = 210$ GPa, $E_3 = 270$ GPa, $D_1 = 2$ cm, $D_2 = 5$ cm, and $D_3 = 7$ cm.

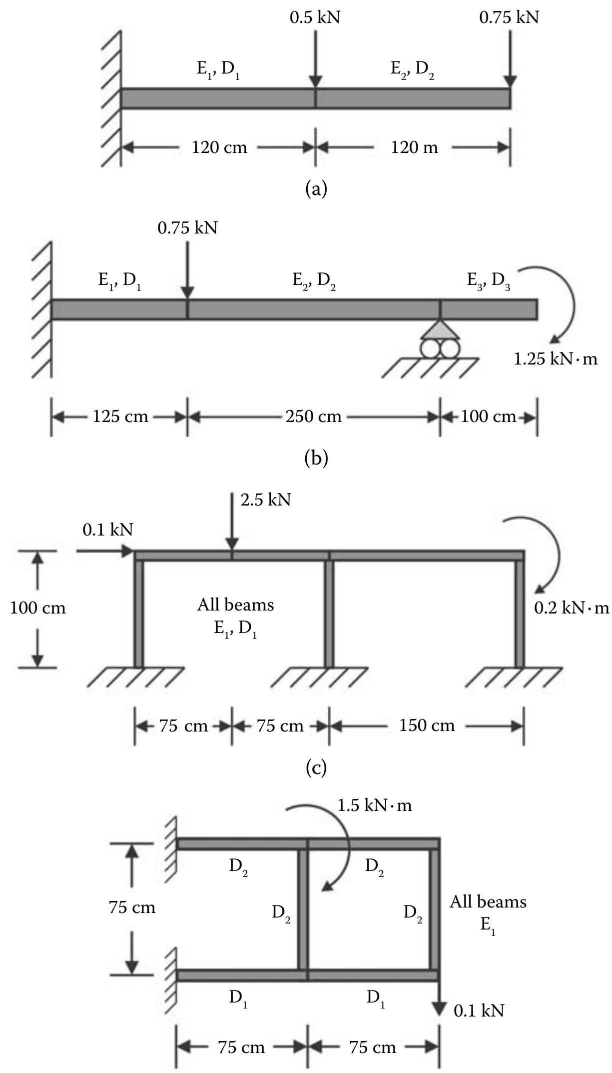


FIGURE 2.16 Two-dimensional beam structures.

PROBLEM 2.5

For the three-dimensional beam structure, shown in Figure 2.17, transient harmonic forces with a frequency of $f = 1/250\ s^{-1}$ and amplitude of $A = 0.85$ kN are applied at the shown locations. Use ANSYS to determine the displacement at the points where the forces are applied as a function of time. Also, create an animation for the loading process. The total time duration for the loading process is 1250 seconds. The beams are made of rectangular cross-sectional pipe, as shown in Figure 2.17, and E = 180 GPa. Mesh each beam by 10 elements. Use the following formula to simulate the applied transient forces:

$$F_x(t) = A \sin(2\pi f t)$$

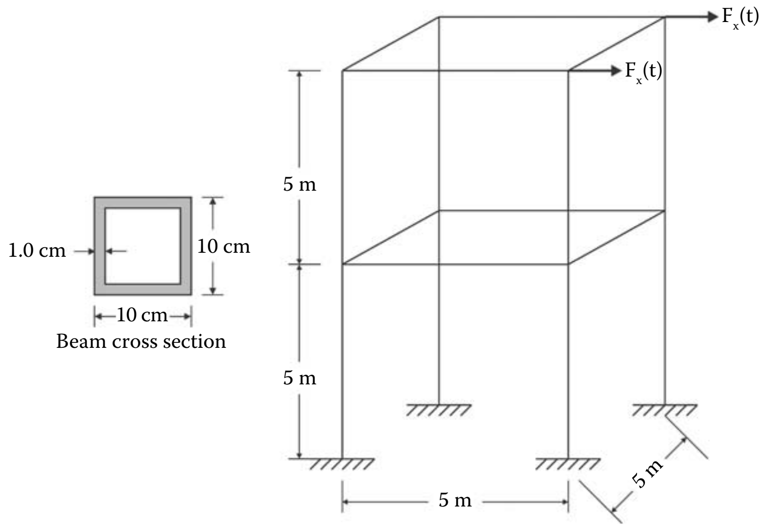


FIGURE 2.17 Three-dimensional beam structure with transient forces.

PROBLEM 2.6

For the beam structure shown in Figure 2.18, two transient harmonic forces are applied at the shown locations. Use ANSYS to determine the displacement at the points where the forces are applied as a function of time. Also, create an animation for the loading process. The total time duration for the loading process is 5000 seconds. The beams are made of circular cross-sectional pipe, E = 200 GPa, with inner and outer radii of $R_i = 0.1$ m, $R_o = 0.11$ m, respectively. Mesh each beam by 10 elements. Use the following formulas to simulate the applied transient forces:

$$F_1(t) = 100\sin\left(\frac{2\pi t}{500}\right)$$

$$F_2(t) = 200\sin\left(\frac{2\pi t}{1000}\right)$$

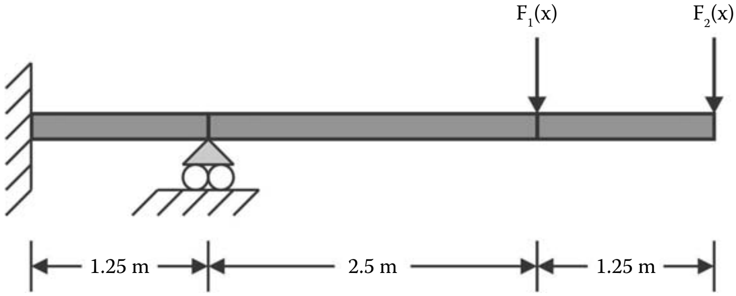


FIGURE 2.18 Horizontal beam structure with transient forces.

This page intentionally left blank

CHAPTER THREE

Solid mechanics

3.1 Stress–strain relations

In this chapter, finite element development of two-dimensional solid elements is described. The development is limited to plane stress–strain elements. The plane stress is defined as the state of stress in which the normal and shear stresses are perpendicular to the plane. Alternatively, the loads on the body are in the xy-plane only. The plane strain is defined as the state of strain in which the normal and shear strain are perpendicular to the plane. Stress is defined as the magnitude of force ΔF on a unit area ΔA, as the unit area approaches zero,

$$\sigma = \lim_{\Delta A \to 0} \frac{\Delta F}{\Delta A} \tag{3.1}$$

Figure 3.1 shows a two-dimensional stress acting on an element with width dx and height dy. The element is treated as two-dimensional with a unit depth. The normal stresses σ_x and σ_y are acting in the x- and y-directions, respectively. Shear stress τ_{xy} is acting in the y-direction and normal to the x-plane. Shear stress τ_{yx} is acting in the x-direction and normal to the y-plane. From the moment of equilibrium, τ_{xy} must be equal to τ_{yx}.

The equilibrium equation for two-dimensional stresses can be obtained, and the net force in the x-direction should be zero:

$$\left(\sigma_x + \frac{\partial \sigma_x}{\partial x}\right) dy - \sigma_x\, dy + \left(\tau_{yx} + \frac{\partial \tau_{yx}}{\partial y}\right) dx - \tau_{yx}\, dx \tag{3.2}$$

Simplifying Equation 3.2 yields

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} = 0 \tag{3.3}$$

Similarly, the equilibrium equation for two-dimensional stresses in the y-direction is

$$\frac{\partial \sigma_y}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} = 0 \tag{3.4}$$

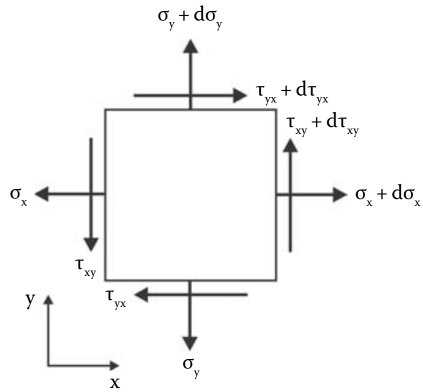


FIGURE 3.1 Two-dimensional state of stress.

Strain is defined as the magnitude of elongation $\Delta\delta$ due to stresses divided by its original length ΔL, as ΔL approaches zero,

$$\varepsilon = \lim_{\Delta L \to 0} \frac{\Delta\delta}{\Delta L} \tag{3.5}$$

Figure 3.2 shows a state of two-dimensional strain. There are two types of strain: longitudinal strain due to stress, as shown in Figure 3.2a, and shearing strain due to shear stress, as shown in Figure 3.2b. The shearing strain is defined as the change in value of the originally right angle in an unstrained state. In Figure 3.2a, u is the displacement in the x-direction and v is the displacement in the y-direction.

From Figure 3.2a, the strain in the x-direction can be expressed as

$$\varepsilon_x = \lim_{dx \to 0} \frac{\left(u + \frac{du}{dx}dx\right) - u}{dx} = \frac{du}{dx} \tag{3.6}$$

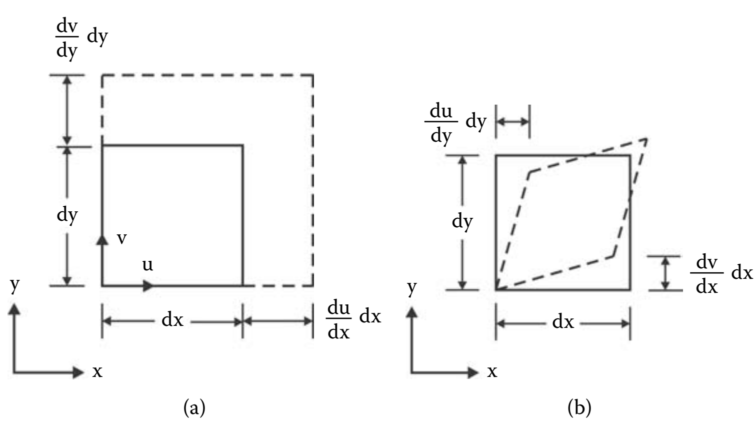


FIGURE 3.2 (a) Longitudinal and (b) shear strains.

Similarly in the y- and z-directions, the strains are, respectively,

$$\varepsilon_y = \lim_{dy\to 0} \frac{\left(v + \frac{dv}{dy}dy\right) - v}{dy} = \frac{dv}{dy} \tag{3.7}$$

$$\varepsilon_z = \lim_{dz\to 0} \frac{\left(w + \frac{dw}{dz}dz\right) - w}{dz} = \frac{dw}{dz} \tag{3.8}$$

As shown in Figure 3.2b, the shearing strain in the xy, xz-, and yz- planes is equal to the sum of changes in angle, respectively,

$$\gamma_{xy} = \gamma_{yx} = \frac{\frac{\partial v}{\partial x}dx}{dx} + \frac{\frac{\partial u}{\partial y}dy}{dy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \tag{3.9}$$

$$\gamma_{xz} = \gamma_{zx} = \frac{\frac{\partial w}{\partial x}dx}{dx} + \frac{\frac{\partial u}{\partial z}dz}{dz} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \tag{3.10}$$

$$\gamma_{yz} = \gamma_{zy} = \frac{\frac{\partial v}{\partial z}dz}{dz} + \frac{\frac{\partial w}{\partial y}dy}{dy} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \tag{3.11}$$

The stresses and strains can be presented in a vector form as follows:

$$\{\sigma\} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} \tag{3.12}$$

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \\ \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \end{Bmatrix} \tag{3.13}$$

According to Hooke's law, the stress and strain are related by the modulus of elasticity E. The stress–strain relation in the x-direction is expressed as

$$\sigma_x = E\varepsilon_x \tag{3.14}$$

Elongations in the y- and z-directions are caused by stress in the x-direction and stress lateral direction. Using Poisson's ratio v, which is defined as the lateral strain over the axial strain, the elongation in the y- and z-directions due to x-direction stresses can be determined as follows:

$$\varepsilon_y = \varepsilon_z = -v\varepsilon_x = -v\frac{\sigma_x}{E} \tag{3.15}$$

The shearing stress and the shearing strain are related by modulus of shear G. For the xy-plane,

$$\tau_{xy} = G\gamma_{xy} \tag{3.16}$$

where

$$G = \frac{E}{2(1+v)} \tag{3.17}$$

Hence, the generalized Hooke's laws are written as

$$\varepsilon_x = \frac{1}{E}\left(\sigma_x - v\left(\sigma_y + \sigma_z\right)\right) \tag{3.18}$$

$$\varepsilon_y = \frac{1}{E}\left(\sigma_y - v\left(\sigma_x + \sigma_z\right)\right) \tag{3.19}$$

$$\varepsilon_z = \frac{1}{E}\left(\sigma_z - v\left(\sigma_x + \sigma_y\right)\right) \tag{3.20}$$

$$\gamma_{xy} = \frac{2(1+v)}{E}\tau_{xy} \tag{3.21}$$

$$\gamma_{yz} = \frac{2(1+v)}{E}\tau_{yz} \tag{3.22}$$

$$\gamma_{zx} = \frac{2(1+v)}{E}\tau_{zx} \tag{3.23}$$

Expressing Equations 3.18–3.20 and 3.21–3.23 in terms of strain components as

$$\sigma_x = \frac{vE}{(1+v)(1-2v)}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right) + \frac{E}{1+v}\varepsilon_x \tag{3.24}$$

$$\sigma_y = \frac{vE}{(1+v)(1-2v)}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right) + \frac{E}{1+v}\varepsilon_y \tag{3.25}$$

$$\sigma_z = \frac{vE}{(1+v)(1-2v)}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right) + \frac{E}{1+v}\varepsilon_z \tag{3.26}$$

$$\tau_{xy} = \frac{E}{2(1+v)}\gamma_{xy} \tag{3.27}$$

$$\tau_{yz} = \frac{E}{2(1+v)}\gamma_{yz} \tag{3.28}$$

$$\tau_{zx} = \frac{E}{2(1+v)}\gamma_{zx} \tag{3.29}$$

For plane stress cases, all stresses in the z-direction are assumed zero, $\sigma_z = \tau_{yz} = \tau_{zx}$, and strain–stress relations are expressed as

$$\varepsilon_x = \frac{1}{E}(\sigma_x - v\sigma_y) \tag{3.30}$$

$$\varepsilon_y = \frac{1}{E}(\sigma_y - v\sigma_x) \tag{3.31}$$

$$\varepsilon_z = \frac{-v}{E}(\sigma_x + \sigma_y) \tag{3.32}$$

$$\gamma_{xy} = \frac{2(1+v)}{E}\tau_{xy} \tag{3.33}$$

Stress–strain relations are expressed as

$$\sigma_x = \frac{E}{1-v^2}(\varepsilon_x + v\varepsilon_y) \tag{3.34}$$

$$\sigma_y = \frac{E}{1-v^2}(\varepsilon_y + v\varepsilon_x) \tag{3.35}$$

$$\tau_{xy} = \frac{E}{2(1+v)}\ \gamma_{xy} \tag{3.36}$$

For the plane strain case, all strains in the z-direction are assumed to be zero, $\varepsilon_z = \gamma_{yz} = \gamma_{zx}$, and strain–stress relations are expressed as

$$\sigma_x = \frac{vE}{(1+v)(1-2v)}(\varepsilon_x + \varepsilon_y) + \frac{E}{1+v}\varepsilon_x \tag{3.37}$$

$$\sigma_y = \frac{vE}{(1+v)(1-2v)}(\varepsilon_x + \varepsilon_y) + \frac{E}{1+v}\varepsilon_y \tag{3.38}$$

$$\sigma_z = \frac{vE}{(1+v)(1-2v)} = v(\sigma_x + \sigma_y) \tag{3.39}$$

$$\tau_{xy} = \frac{E}{2(1+v)}\gamma_{xy} \tag{3.40}$$

Stress–strain relations are expressed as

$$\varepsilon_x = \frac{1+v}{E}((1-v)\sigma_x - v\sigma_y) \tag{3.41}$$

$$\varepsilon_y = \frac{1+v}{E}\left((1-v)\sigma_y - v\sigma_x\right) \quad (3.42)$$

$$\gamma_{xy} = \frac{2(1+v)}{E}\tau_{xy} \quad (3.43)$$

In matrix form, the stress–strain relations in the state of plane stress and plane strain are expressed as, respectively,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1-v^2)} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1-v}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (3.44)$$

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+v)(1-2v)} \begin{bmatrix} 1-v & v & 0 \\ v & 1-v & 0 \\ 0 & 0 & \frac{1-2v}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (3.45)$$

The stress–strain relationships are symbolically expressed as

$$\{\sigma\} = [D]\{\varepsilon\} \quad (3.46)$$

where [D] is called the stress–strain matrix, or simply the D matrix.

3.2 Development of triangular plane stress and plane strain element

Figure 3.3a shows a plate subjected to tensile stress. The plate is divided into linear triangular elements, as shown in Figure 3.3b. The linear triangular element has three nodes and nodes can be displaced, but the element's sides remain straight. As shown in Figure 3.3c, the nodes are named i, j, and m. At each node, there are two degrees of freedom, displacement in the x- and y-directions. The coordinates for the three nodes i, j, and k are (x_i,y_i), (x_j,y_j), and (x_k,y_k), respectively. The nodal displacement vector is given by

$$\{d\} = \begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_m \\ v_m \end{Bmatrix} \quad (3.47)$$

Therefore, each element has six degrees of freedom. Since it is a linear element, linear displacement functions are selected for the x- and y-directions as follows:

$$u(x,y) = a_1 + a_2x + a_3y \quad (3.48)$$

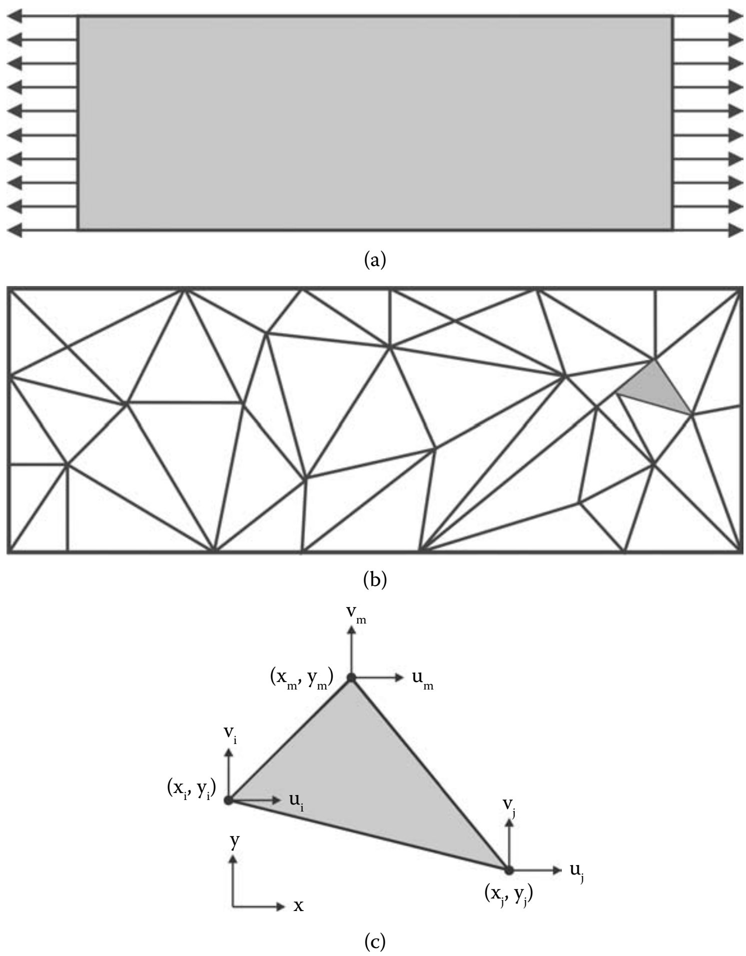


FIGURE 3.3 (a) Plate subjected to tensile stress, (b) dividing the plate into linear triangular elements, and (c) description of a linear triangular element.

$$v(x,y) = a_4 + a_5x + a_6y \tag{3.49}$$

The displacement equations can be solved because there are six constants, a_1 to a_6, and six equations. The nodal x- and y-displacements for all nodes are expressed in the following matrix form:

$$\begin{Bmatrix} u_i \\ u_j \\ u_m \end{Bmatrix} = \begin{bmatrix} 1 & x_i & y_i \\ 1 & x_j & y_j \\ 1 & x_m & y_m \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{3.50}$$

$$\begin{Bmatrix} v_i \\ v_j \\ v_m \end{Bmatrix} = \begin{bmatrix} 1 & x_i & y_i \\ 1 & x_j & y_j \\ 1 & x_m & y_m \end{bmatrix} \begin{Bmatrix} a_4 \\ a_5 \\ a_6 \end{Bmatrix} \tag{3.51}$$

and solving for a's,

$$\begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} \alpha_i & \alpha_j & \alpha_m \\ \beta_i & \beta_j & \beta_m \\ \gamma_i & \gamma_j & \gamma_m \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \\ u_m \end{Bmatrix} \tag{3.52}$$

and

$$\begin{Bmatrix} a_4 \\ a_5 \\ a_6 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} \alpha_i & \alpha_j & \alpha_m \\ \beta_i & \beta_j & \beta_m \\ \gamma_i & \gamma_j & \gamma_m \end{bmatrix}\begin{Bmatrix} v_i \\ v_j \\ v_m \end{Bmatrix} \tag{3.53}$$

where A is the area of the element, which is equal to

$$A = \frac{1}{2}\left[x_i\left(y_j - y_m\right) + x_j\left(y_m - y_i\right) + x_m\left(y_i - y_j\right)\right] \tag{3.54}$$

The α, β, and γ are defined as

$$\begin{aligned} &\alpha_i = x_j\, y_m - y_j\, x_m \qquad &&\alpha_j = x_m\, y_i - y_m\, x_i \qquad &&\alpha_m = x_i\, y_j - y_i\, x_j \\ &\beta_i = y_j - y_m &&\beta_j = y_m - y_i &&\beta_m = y_i - y_j \\ &\gamma_i = x_m - x_j &&\gamma_j = x_i - x_m &&\gamma_m = x_j - x_i \end{aligned} \tag{3.55}$$

Substituting the values of α's, β's, and γ's into Equations 3.48 and 3.49 yields

$$u(x,y) = \frac{1}{2A}\begin{bmatrix} \left(\alpha_i + \beta_i x + \gamma_i y\right)u_i + \left(\alpha_j + \beta_j x + \gamma_j y\right)u_j \\ +\left(\alpha_m + \beta_m x + \gamma_m y\right)u_m \end{bmatrix} \tag{3.56}$$

$$v(x,y) = \frac{1}{2A}\begin{bmatrix} \left(\alpha_i + \beta_i x + \gamma_i y\right)v_i + \left(\alpha_j + \beta_j x + \gamma_j y\right)v_j \\ +\left(\alpha_m + \beta_m x + \gamma_m y\right)v_m \end{bmatrix} \tag{3.57}$$

The strain vector for a two-dimensional element is given in Equation 3.13. Using the displacement in the v- and the u-directions, the strain vector is expressed as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1}{2A}\begin{Bmatrix} \beta_i u_i + \beta_j u_j + \beta_m u_m \\ \gamma_i v_i + \gamma_j v_j + \gamma_m v_m \\ \gamma_i u_i + \beta_i v_i + \gamma_j u_j + \beta_j v_j + \gamma_m u_m + \beta_m v_m \end{Bmatrix} \tag{3.58}$$

The strain vector can be expressed in a matrix form as follows:

$$\{\varepsilon\} = \frac{1}{2A}\begin{bmatrix} \beta_i & 0 & \beta_j & 0 & \beta_m & 0 \\ 0 & \gamma_i & 0 & \gamma_j & 0 & \gamma_m \\ \gamma_i & \beta_i & \gamma_j & \beta_j & \gamma_m & \beta_m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_m \\ v_m \end{Bmatrix} \tag{3.59}$$

The matrix (3.59) is symbolically expressed as

$$\{\varepsilon\} = [B]\{d\} \tag{3.60}$$

The stress–strain relationship is symbolically expressed as

$$\{\sigma\} = [D][B]\{d\} \tag{3.61}$$

The principle of the minimum potential energy theory is used to obtain the stiffness matrix [K]. The total potential energy is the sum of the strain energy U and the potential energy of the total applied nodal forces F_p. The strain energy is given as

$$U = \int_V \{\varepsilon\}^T \{\sigma\}\ dV \tag{3.62}$$

Using Equations 3.60 and 3.61 in Equation 3.62, the strain energy becomes

$$U = \int_V [B]^T \{d\}^T [D][B]\{d\}\ dV \tag{3.63}$$

The total applied nodal forces can be expressed as

$$\{F_p\} = -\{d\}^T\{F\} \tag{3.64}$$

Applying Castigliano's first theorem,

$$F_i = \frac{\partial U}{\partial d_i} \tag{3.65}$$

Using expressions (3.63) and (3.64) in the definition of Castigliano's first theorem yields

$$\frac{\partial U}{\partial\{d\}} \int_V [B]^T \{d\}^T [D][B]\{d\}\ dV = \{d\}^T\{F\} \tag{3.66}$$

Taking the first variation of the potential energy with respect to the nodal displacement vector yields

$$\int_V [B]^T [D][B]\{d\}\ dV = \{F\} \tag{3.67}$$

Since the nodal displacements vector is related to the nodal forces vector by [K]{d} = {F}, the stiffness matrix is expressed as

$$[K] = \int_V [B]^T[D][B]\ dV \tag{3.68}$$

For a constant thickness element t, Equation 3.68 is rewritten as

$$[K] = \int_A [B]^T [D][B] t \, dx \, dy \tag{3.69}$$

The matrices [D] and [B] are not functions of x or y, and thus the integration of dx dy is simply the area of the element A as follows:

$$[K] = tA[B]^T[D][B] \tag{3.70}$$

The global stiffness matrix is obtained by assembling the stiffness matrices for all elements as follows:

$$[K] = \sum_{e=1}^{N} \left[K^{(e)}\right] \tag{3.71}$$

and the nodal forces are also assembled to form a global force vector as follows:

$$[F] = \sum_{e=1}^{N} \left[F^{(e)}\right] \tag{3.72}$$

Also, the global displacements vector can be obtained by

$$[d] = \sum_{e=1}^{N} \left[d^{(e)}\right] \tag{3.73}$$

Finally,

$$\{F\} = [K]\{d\} \tag{3.74}$$

To determine the nodal displacements, the global stiffness matrix must be formulated. The following steps are for solving solid mechanics problems:

a. Divide the domain into elements and create a finite element mesh.
b. Formulate the [B] matrix for all elements.
c. Formulate the [D] matrix for all elements.
d. Formulate the [K] matrix for all elements using calculated [B] and [D] matrices.
e. Assemble the [K] matrices to create the global [K] matrix.
f. Formulate the stiffness equation {F} = [K]{d}.
g. Apply the boundary conditions to the stiffness equation to remove the singularity.
h. Solve the stiffness equations to determine the unknowns.

3.3 Analyzing rectangular plate subjected to forces

The thin plate, as shown in Figure 3.4a, is subjected to two forces at its right edges. Determine the nodal displacements at points where forces are applied, given E = 200 GPa, v = 0.3, and t = 0.01 m. Consider the problem as a plane stress.

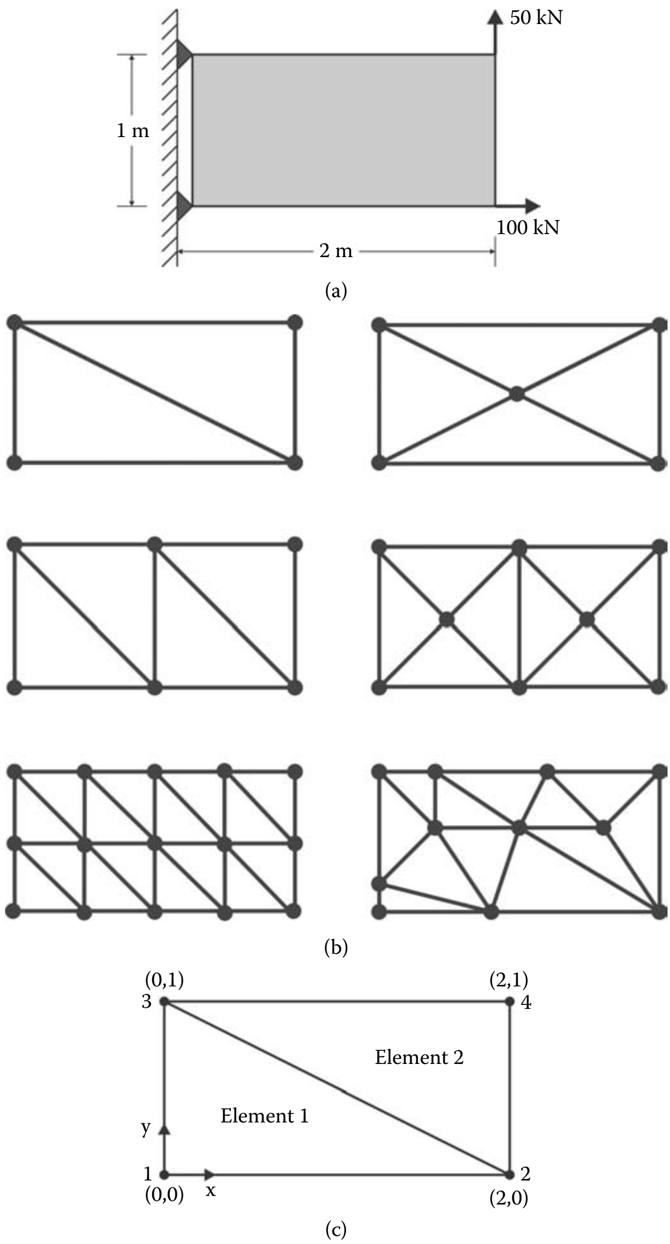


FIGURE 3.4 (a) Thin plate subjected to two forces, (b) element distribution for example, and (c) finite element mesh.

There are unlimited options for elements and nodes distributions, and some of these options are shown in Figure 3.4b. An increasing number of elements will definitely enhance the accuracy of the results, but up to a certain number of elements. After which, the results become independent of the number of the elements. The first mesh contains just two elements, which is the minimum number of elements to solve this problem. The second mesh contains four elements, the third mesh also contains four elements, but with a different elements distribution. The last mesh has random elements distribution. For illustration purposes, the first mesh is selected for this example, because it has the least number of elements. Figure 3.4c shows the finite element mesh for the present problem.

First, the [B], [D], and [K] matrices for Element 1 are formulated. Element 1 has the coordinate shown in Figure 3.4c. Node 1 is named (i) and its coordinate is (0,0), Node 2 is named (j) and its coordinate is (2,0), and Node 3 is named (m) and its coordinate is (0,1).

The β's and γ's are required for the [B] matrix, and they are calculated using Equations 3.55; A is calculated using (3.54). We have

$$\beta_i = y_j - y_m = 0 - 1 = -1$$

$$\beta_j = y_m - y_i = 1 - 0 = 1$$

$$\beta_m = y_i - y_j = 0 - 0 = 0$$

$$\gamma_i = x_m - x_j = 0 - 2 = -2$$

$$\gamma_j = x_i - x_m = 0 - 0 = 0$$

$$\gamma_m = x_j - x_i = 2 - 0 = 2$$

$$2A = 1 \text{ m}^2$$

Then, the [B] matrix is formulated using (3.59):

$$[B] = \frac{1}{2}\begin{bmatrix} -1 & 0 & 1 & 0 & 0 & 0 \\ 0 & -2 & 0 & 0 & 0 & 2 \\ -2 & -1 & 0 & 1 & 2 & 0 \end{bmatrix}$$

and [D] matrix is formulated using (3.44):

$$[D] = \frac{200 \times 10^9}{1 - 0.3^2}\begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & \dfrac{1-0.3}{2} \end{bmatrix}$$

Finally, the [K] matrix for Element 1 is obtained using expression (3.70):

$$\left[K^{(1)}\right] = 10^{11} \begin{array}{c} \begin{array}{cccccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \begin{bmatrix} 1.318 & 0.714 & -0.549 & -0.385 & -0.769 & -0.330 \\ 0.714 & 2.390 & -0.330 & -0.192 & -0.385 & -2.198 \\ -0.549 & -0.330 & 0.549 & 0 & 0 & 0.330 \\ -0.385 & -0.192 & 0 & 0.192 & 0.385 & 0 \\ -0.769 & -0.385 & 0 & 0.385 & 0.770 & 0 \\ -0.330 & -2.198 & 0.330 & 0 & 0 & 2.198 \end{bmatrix} \end{array} \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

Element 2 has the coordinate shown in Figure 3.4c. Node 2 is named (i) and its coordinate is (2,0), Node 4 is named (j) and its coordinate is (2,1), and Node 3 is named (m) and its coordinate is (0,1). The β's and γ's are

$$\beta_i = y_j - y_m = 1 - 1 = 0$$

$$\beta_j = y_m - y_i = 1 - 0 = 1$$

$$\beta_m = y_i - y_j = 0 - 1 = -1$$

$$\gamma_i = x_m - x_j = 0 - 2 = -2$$

$$\gamma_j = x_i - x_m = 2 - 0 = 2$$

$$\gamma_m = x_j - x_i = 2 - 2 = 0$$

$$2A = 1\ \text{m}^2$$

Then the [B] matrix is formulated using (3.59):

$$[B] = \frac{1}{2} \begin{bmatrix} 0 & 0 & 1 & 0 & -1 & 0 \\ 0 & -2 & 0 & 2 & 0 & 0 \\ -2 & 0 & 2 & 1 & 0 & -1 \end{bmatrix}$$

and [D] matrix is formulated using (3.44):

$$[D] = \frac{200 \times 10^9}{1 - 0.3^2} \begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & \dfrac{1 - 0.3}{2} \end{bmatrix}$$

Finally, the [K] matrix for Element 2 is obtained using expression (3.70):

$$\left[K^{(2)}\right]=10^{11}\begin{array}{c}\begin{array}{cccccc}\underline{2} & \underline{2} & \underline{4} & \underline{4} & \underline{3} & \underline{3}\end{array}\\ \left[\begin{array}{cccccc}0.769 & 0 & -0.769 & -0.385 & 0 & 0.385\\ 0 & 2.198 & -0.329 & -2.198 & 0.330 & 0\\ -0.769 & -0.330 & 1.318 & 0.714 & -0.549 & -0.385\\ -0.385 & -2.198 & 0.714 & 2.390 & -0.330 & -0.192\\ 0 & 0.330 & -0.549 & -0.330 & 0.549 & 0\\ 0.385 & 0 & -0.385 & -0.192 & 0 & 0.192\end{array}\right]\end{array}\begin{array}{c}\\ \underline{2}\\ \underline{2}\\ \underline{4}\\ \underline{4}\\ \underline{3}\\ \underline{3}\end{array}$$

Using Equation 3.71, the global matrix is formulated as shown:

$$[K]=10^{11}\begin{array}{c}\begin{array}{cccccccc}\underline{1} & \underline{1} & \underline{2} & \underline{2} & \underline{3} & \underline{3} & \underline{4} & \underline{4}\end{array}\\ \left[\begin{array}{cccccccc}1.318 & 0.714 & -0.549 & -0.385 & -0.769 & -0.330 & 0 & 0\\ 0.714 & 2.390 & -0.330 & -0.192 & -0.385 & -2.198 & 0 & 0\\ -0.549 & -0.330 & 1.319 & 0 & 0 & 0.715 & -0.769 & -0.385\\ -0.385 & -0.192 & 0 & 2.390 & 0.715 & 0 & -0.329 & -2.198\\ -0.769 & -0.385 & 0 & 0.714 & 1.319 & 0 & 0.549 & -0.330\\ 0.330 & -2.198 & 0.715 & 0 & 0 & 2.390 & -0.385 & -0.192\\ 0 & 0 & -0.769 & -0.330 & -0.549 & -0.385 & 1.318 & 0.714\\ 0 & 0 & -0.385 & -2.198 & -0.330 & -0.192 & 0.714 & 2.39\end{array}\right]\end{array}\begin{array}{c}\\ \underline{1}\\ \underline{1}\\ \underline{2}\\ \underline{2}\\ \underline{3}\\ \underline{3}\\ \underline{4}\\ \underline{4}\end{array}$$

The force–displacement equation is assembled using expression (3.74):

$$\left\{\begin{array}{c}F_{1x}\\ F_{1y}\\ F_{2x}=100\times10^{3}\\ F_{2y}=0\\ F_{3x}\\ F_{3y}\\ F_{4x}=0\\ F_{4y}=50\times10^{3}\end{array}\right\}=[K]\left\{\begin{array}{c}d_{1x}=0\\ d_{1y}=0\\ d_{2x}\\ d_{2y}\\ d_{3x}=0\\ d_{3y}=0\\ d_{4x}\\ d_{4y}\end{array}\right\}$$

Eliminating the first, second, fifth, and sixth columns and rows,

$$\left\{\begin{array}{c}F_{2x}=100\times10^{3}\\ F_{2y}\\ F_{4x}\\ F_{4y}=50\times10^{3}\end{array}\right\}=10^{11}\left[\begin{array}{cccc}1.319 & 0 & -0.769 & -0.385\\ 0 & 2.39 & -0.329 & -2.198\\ -0.769 & -0.330 & 1.318 & 0.714\\ -0.385 & -2.198 & 0.714 & 2.39\end{array}\right]\left\{\begin{array}{c}d_{2x}\\ d_{2y}\\ d_{4x}\\ d_{4y}\end{array}\right\}$$

Finally, solving the above equations, the nodal displacements are

$d_{2x} = 1.665 \times 10^{-6}$ m

$$d_{2y} = 2.836 \times 10^{-6}\ \text{m}$$

$$d_{4x} = 1.156 \times 10^{-8}\ \text{m}$$

$$d_{4y} = 3.082 \times 10^{-6}\ \text{m}$$

3.4 Development of rectangular plane stress element

The rectangular element has four nodes, and nodes can be displaced but sides remain straight. It is assumed that the stress components are constant within each element. At each node, there are two degrees of freedom, displacement in the x- and y-directions, as shown in Figure 3.5. Each node is subjected to two forces: vertical and horizontal. The width and height of the element are a and b, respectively. Node 1 has coordinate (0,0) and displacement (u_1,v_1), Node 2 has coordinate (a,0) and displacement (u_2,v_2), Node 3 has coordinate (a,b) and displacement (u_3,v_3), and Node 4 has coordinate (0,b) and displacement (u_4,v_4). The nodal displacement vector is given by

$$\{d\} = \left\{ \begin{matrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{matrix} \right\} \tag{3.75}$$

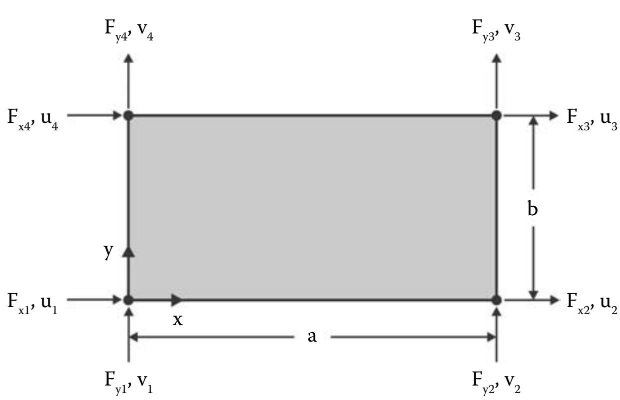


FIGURE 3.5 Rectangular element with displacements and forces at the nodes.

Therefore, there are eight degrees of freedom for each element, and the displacement functions for the x- and the y-directions are

$$u(x,y) = a_1 + a_2x + a_3y + a_4xy \tag{3.76}$$

$$v(x,y) = a_5 + a_6x + a_7y + a_8xy \tag{3.77}$$

The values of constants a_1 to a_8 are obtained by substituting the eight nodal coordinates in the displacement functions in terms of element dimensions, a and b. Rearranging the displacement functions as

$$u(x,y) = f_1u_1 + f_2u_2 + f_3u_3 + f_4u_4 \tag{3.78}$$

$$v(x,y) = f_1v_1 + f_2v_2 + f_3v_3 + f_4v_4 \tag{3.79}$$

where functions f_1, f_2, f_3, and f_4, are defined as follows:

$$f_1 = \left(1 - \frac{x}{a}\right)\left(1 - \frac{y}{b}\right) \tag{3.80}$$

$$f_2 = \frac{x}{a}\left(1 - \frac{y}{b}\right) \tag{3.81}$$

$$f_3 = \frac{xy}{ab} \tag{3.82}$$

$$f_4 = \frac{y}{b}\left(1 - \frac{x}{a}\right) \tag{3.83}$$

The strain vector for a two-dimensional element is given in Equation 3.13. Using the displacements v and u, the strain vector is expressed as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \rho_1 & 0 & \rho_2 & 0 & \rho_3 & 0 & \rho_4 & 0 \\ 0 & \mu_1 & 0 & \mu_2 & 0 & \mu_3 & 0 & \mu_4 \\ \mu_1 & \rho_1 & \mu_2 & \rho_2 & \mu_3 & \rho_3 & \mu_4 & \rho_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \tag{3.84}$$

where

$$\rho_1 = -\frac{1}{a}\left(1 - \frac{y}{b}\right) \tag{3.85}$$

$$\rho_2 = \frac{1}{a}\left(1 - \frac{y}{b}\right) \tag{3.86}$$

$$\rho_3 = \frac{y}{ab} \tag{3.87}$$

$$\rho_4 = -\frac{y}{ab} \tag{3.88}$$

$$\mu_1 = -\frac{1}{b}\left(1 - \frac{x}{a}\right) \tag{3.89}$$

$$\mu_2 = -\frac{x}{ab} \tag{3.90}$$

$$\mu_3 = \frac{x}{ab} \tag{3.91}$$

$$\mu_4 = \frac{1}{b}\left(1 - \frac{x}{a}\right) \tag{3.92}$$

The strain vector (3.84) is symbolically expressed as

$$\{\varepsilon\} = [B]\{d\} \tag{3.93}$$

and the stress–strain relationship is expressed as

$$\{\sigma\} = [D][B]\{d\} \tag{3.94}$$

The principle of the minimum potential energy theory is used to obtain the stiffness matrix [K]. The total potential energy π_p is the sum of the strain energy U and the potential energy of the total applied nodal forces F_p. For a constant thickness element t, the stiffness matrix is rewritten as

$$[K] = \int_A [B]^T [D][B] t \, dx \, dy \tag{3.95}$$

Using Equation 3.95, the stiffness equation for the rectangular element with four nodes in the state of plane stress element is calculated as

$$[K] = \begin{bmatrix} C_1 & C_2 & C_4 & C_5 & -C_1/2 & -C_2 & C_7 & -C_5 \\ C_2 & C_3 & -C_5 & C_6 & -C_2 & -C_3/2 & C_5 & C_8 \\ C_4 & -C_5 & C_1 & -C_2 & C_7 & C_5 & -C_1/2 & C_2 \\ C_5 & C_6 & -C_2 & C_3 & -C_5 & C_8 & C_2 & -C_3/2 \\ -C_1/2 & -C_2 & C_7 & -C_5 & C_1 & C_2 & C_4 & C_5 \\ -C_2 & -C_3/2 & C_5 & C_8 & C_2 & C_3 & -C_5 & C_6 \\ C_7 & C_5 & -C_1/2 & C_2 & C_4 & -C_5 & C_1 & -C_2 \\ -C_5 & C_8 & C_2 & -C_3/2 & C_5 & C_6 & -C_2 & C_3 \end{bmatrix} \tag{3.96}$$

where

$$C_1 = \left(\frac{b}{3a} + \frac{1-v}{6}\frac{a}{b}\right)\frac{Et}{1-v^2} \tag{3.97}$$

$$C_2 = \left(\frac{v}{4} + \frac{1-v}{8}\right)\frac{Et}{1-v^2} \tag{3.98}$$

$$C_3 = \left(\frac{b}{3a} + \frac{1-v}{6}\frac{b}{a}\right)\frac{Et}{1-v^2} \tag{3.99}$$

$$C_4 = \left(-\frac{b}{3a} + \frac{1-v}{12}\frac{a}{b}\right)\frac{Et}{1-v^2} \tag{3.100}$$

$$C_5 = \left(\frac{v}{4} + \frac{1-v}{8}\right)\frac{Et}{1-v^2} \tag{3.101}$$

$$C_6 = \left(\frac{a}{6b} + \frac{1-v}{6}\frac{b}{a}\right)\frac{Et}{1-v^2} \tag{3.102}$$

$$C_7 = \left(\frac{b}{6a} + \frac{1-v}{6}\frac{a}{b}\right)\frac{Et}{1-v^2} \tag{3.103}$$

$$C_8 = \left(-\frac{a}{3b} + \frac{1-v}{12}\frac{b}{a}\right)\frac{Et}{1-v^2} \tag{3.104}$$

The global stiffness matrix is obtained by assembling the stiffness matrices for all elements as follows:

$$[K] = \sum_{e=1}^{N}\left[K^{(e)}\right] \tag{3.105}$$

and the nodal forces are assembled to form a global force vector as follows:

$$[F] = \sum_{e=1}^{N}\left[F^{(e)}\right] \tag{3.106}$$

Also, the global displacement vector is obtained by

$$[d] = \sum_{e=1}^{N}\left[d^{(e)}\right] \tag{3.107}$$

Finally,

$$\{F\} = [K]\{d\} \tag{3.108}$$

3.5 Analyzing a plate with a hole subjected to tensile pressure using ANSYS

The square plate with a hole shown in Figure 3.6a is subjected to tensile pressure at both vertical sides. Use ANSYS to determine the maximum stress in the x-direction. Also, compare the ANSYS result with maximum stress using the stress concentration factor chart. The applied tensile

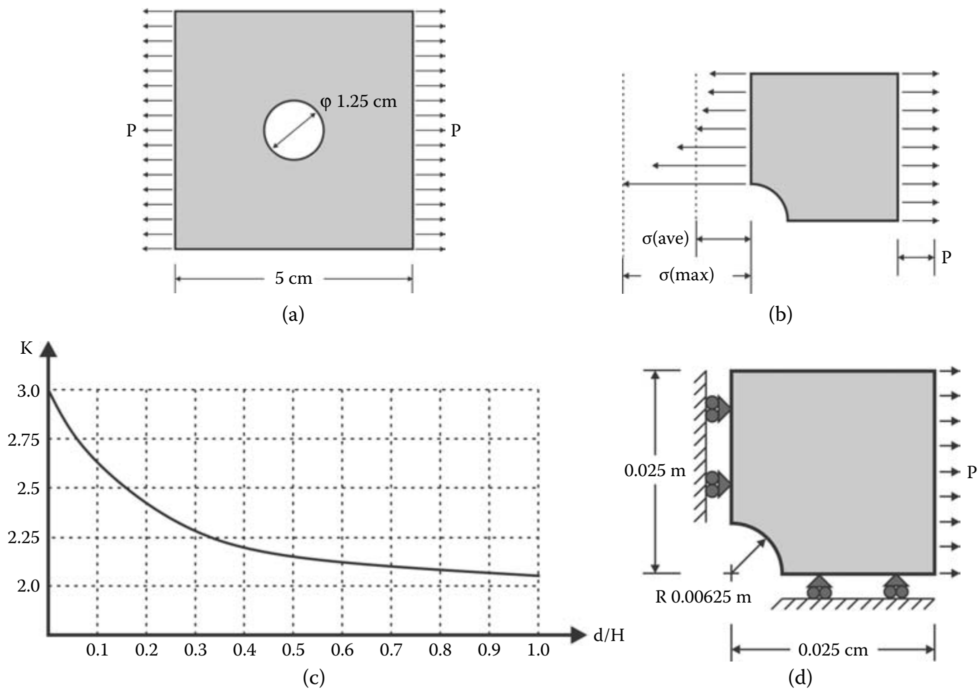


FIGURE 3.6 (a) A plate with a hole subjected to tensile pressure. (b) A plate with hole under stress, and stress distribution near the hole. (c) Stress concentration factor for a plate with a hole. (d) Symmetry boundary conditions for the problem.

pressure is 100 kN/m^2, and let E = 270 GPa, v = 0.3, and consider the plate as a plane stress with thickness of t = 0.005 m.

For the plate with a hole shown in Figure 3.6a, the stress along the vertical symmetry line is typically assumed uniform. However, from the experimental observations, the stress is not uniform, but it has a maximum value near the hole, and it is greater than the average stress, as illustrated in Figure 3.6b.

The complexity of maximum stress can be conveniently treated using the stress concentration factor K. The maximum stress is equal to the average stress multiplied by the stress concentration factor. The definition of the stress concentration factor is

$$K = \frac{\sigma(\max)}{\sigma(\text{ave})} \tag{3.109}$$

The average stress is calculated as

$$\sigma(\text{ave}) = \frac{P \times H}{(H - d)} \tag{3.110}$$

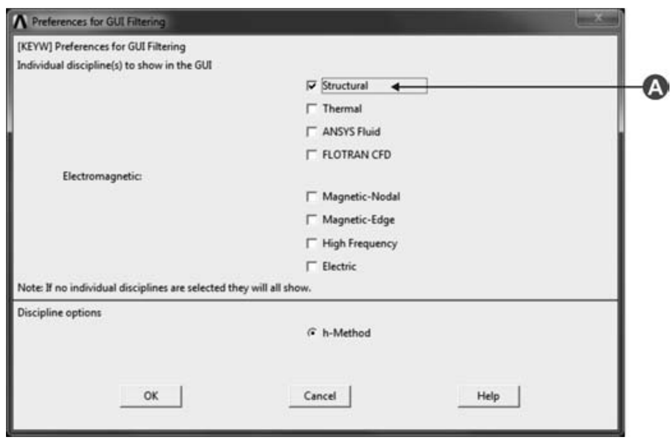
where H is the height of the plate, 5 cm for the present problem; P is plate thickness; and d is the hole diameter. The stress concentration for a solid plate with a hole can be obtained using Figure 3.6c. The x-axis represents the ratio of the diameter of the hole to the height of the plate,

while the y-axis is the corresponding stress concentration factor. Note that decreasing the diameter of the hole increases the stress concentration up to three times the average stress.

ANSYS is employed to determine the maximum stress. This example is limited to structural analysis. Hence, select Structural only in the Preferences. Solid element is used, and its shape is rectangular with four nodes.

Double click on the Mechanical APDL Product Launcher icon

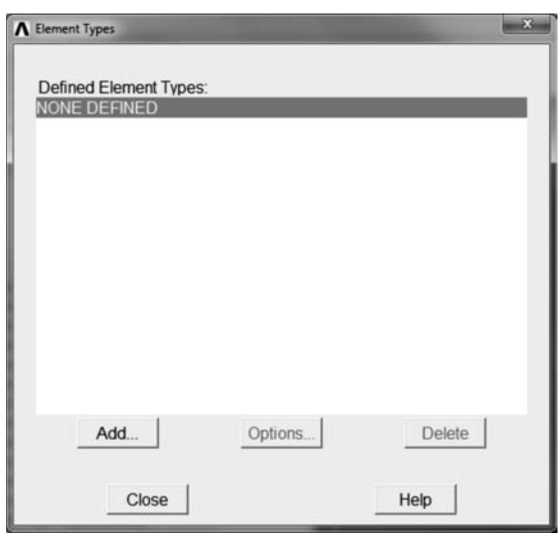
Main Menu > Preferences



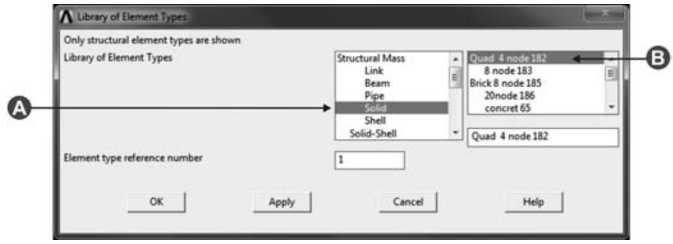
A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete



Add…



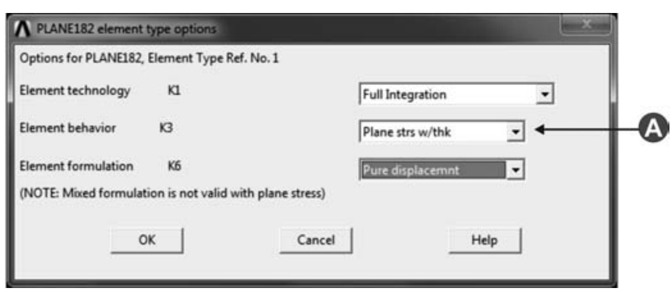
A select Solid

B select Quad 4 node 182

OK

The Quad 4 node 182 is a rectangular-shaped element with four nodes. Triangular elements are not available in the ANSYS elements library. A quadratic element with eight nodes is available. The plate has a thickness. In the option, select plane stress with thickness. In Real Constant window, the thickness is specified.

Options…



A select Plane strs w/thk

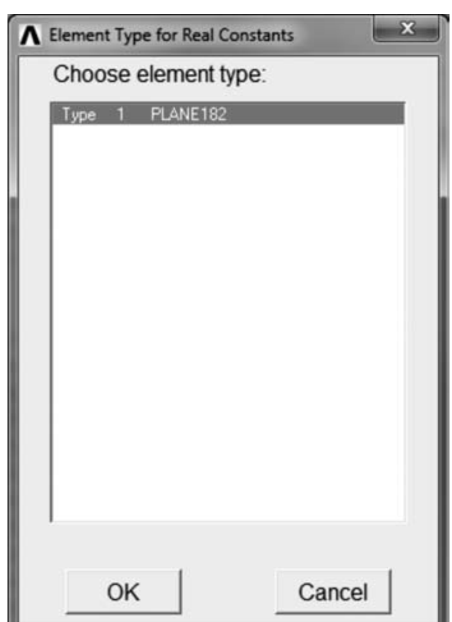
OK

Close the Element Type window

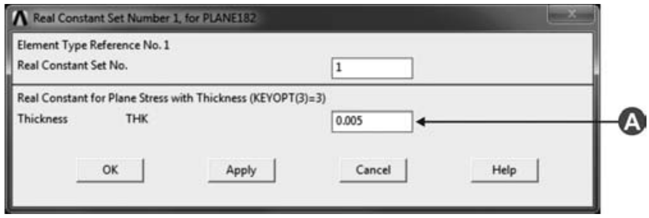
Close

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

Add…



OK



A type 0.005 in Thickness

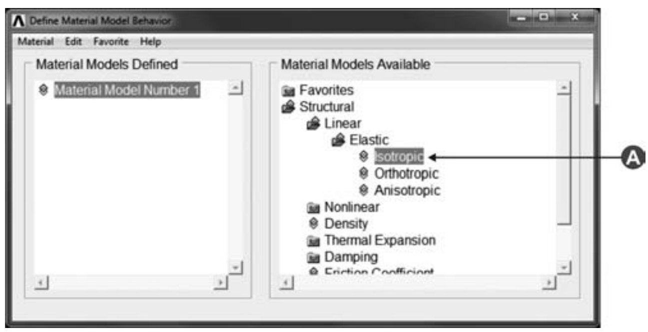
OK

Close the Element Type for Real Constants window

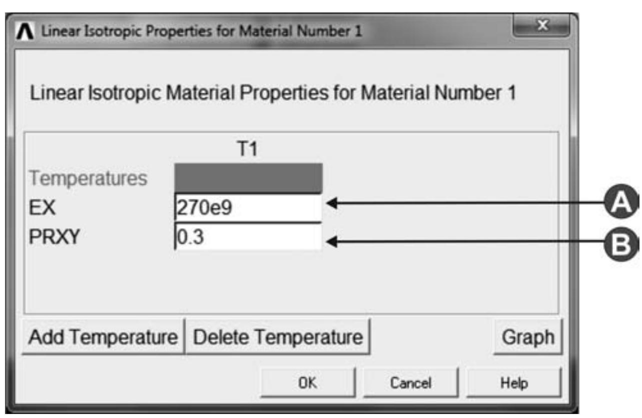
Close

For the material properties, the modulus of elasticity and Poisson's ratio are required to solve the problem.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic



A type 270e9 in EX

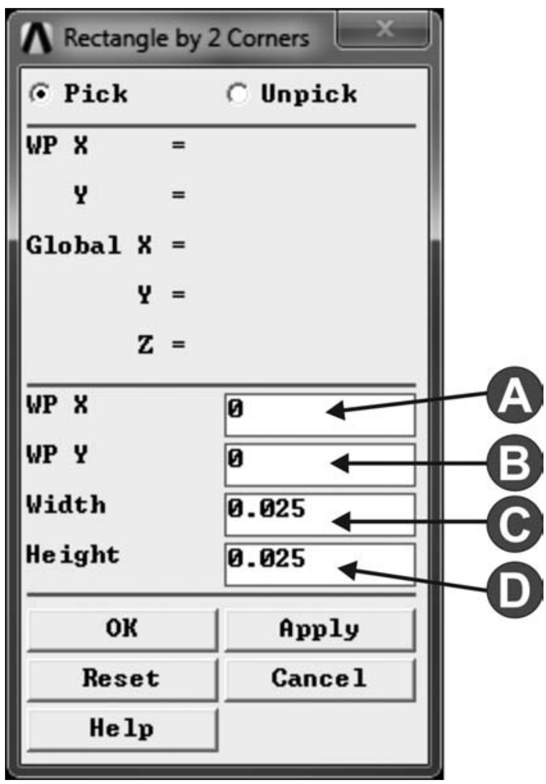
B type 0.3 in PRXY

OK

Close the Material Model Behavior window

The geometry is modeled by creating a square and a circle. Boolean operation is utilized to remove the circle from the square using subtraction. The advantage of symmetry in the problem is considered. Only the upper right quarter is considered. Figure 3.6d shows the considered geometry for the problem. Notice that the imposed boundary condition at the vertical symmetry line is zero displacement in the x-direction and zero displacement in the y-direction at the horizontal symmetry line.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

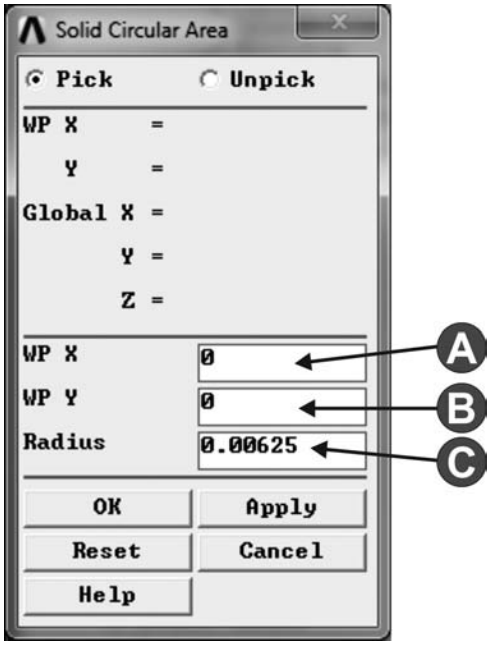
B type 0 in WP Y

C type 0.025 in Width

D type 0.025 in Height

OK

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle

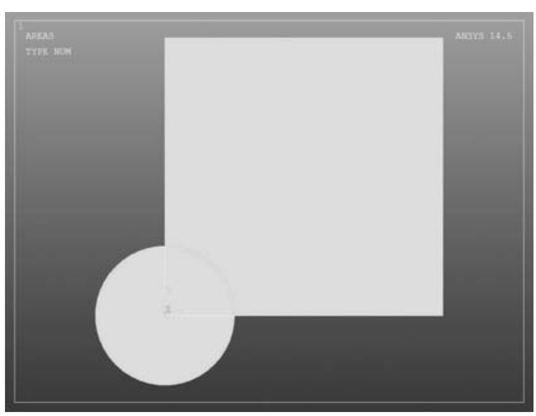


A type 0 in WP X

B type 0 in WP Y

C type 0.00625 in Radius

$\boxed{\textbf{OK}}$



ANSYS graphics show the square and the circle

Now, the circular area should be subtracted from the square area to form a plate with a hole. In Boolean operations, there are a number of operations such as adding, subtracting, dividing, and many more.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

Click on square area, and then in Subtract Areas window, click on

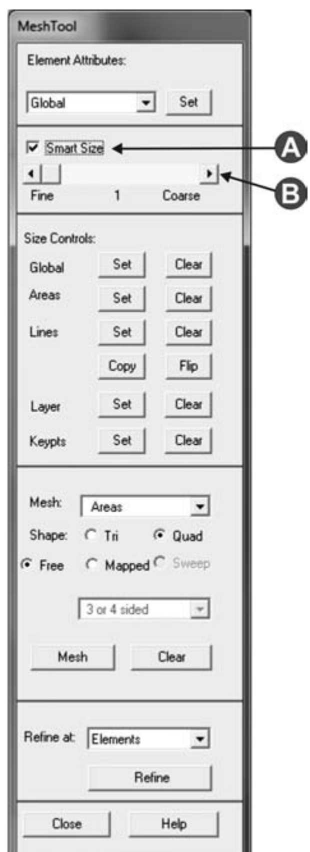
Apply

Click on circular area, and then in Subtract Areas window, click on

OK

A free mesh is generated using the smart mesh option, and the mesh refinement is 1. The high stresses are expected to be at a region close to the hole, and therefore, more elements will be added at that region using refinement at the line.

Main Menu > Preprocessor > Meshing > Mesh Tool



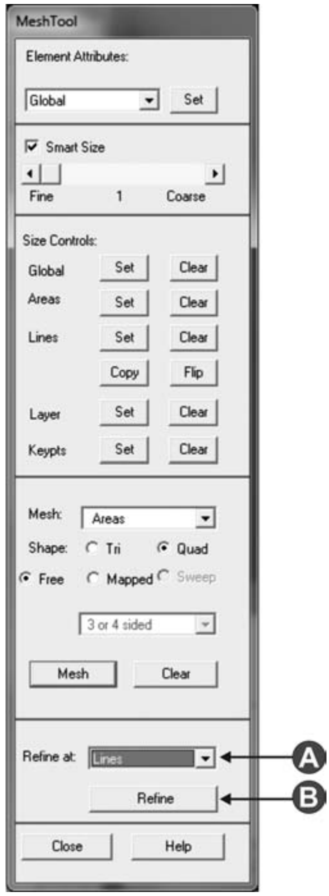
A select Smart Size

B set the level to 1

Mesh

In Mesh Areas window, click on

Pick All

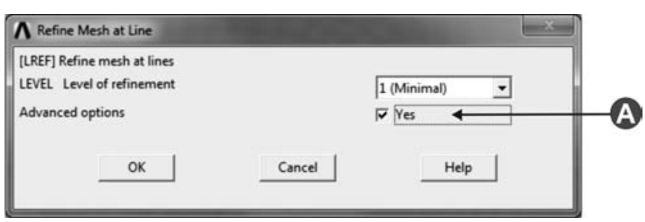


A select Lines

B click on Refine

Click on the curved line at the hole, and then in Refine Mesh at Line window, click on

OK

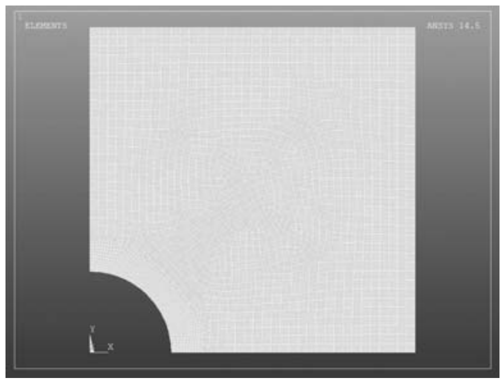


A turn on the Yes to activate Advanced options

OK

A type 3

OK

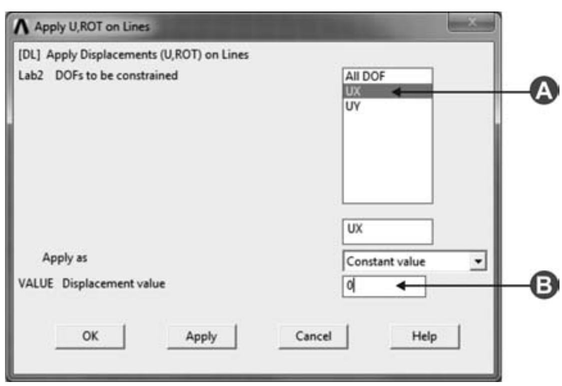


ANSYS graphics show the mesh

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines

In the ANSYS graphics, click on the left vertical line, where zero x-direction displacement is applied. Then, in Apply U,ROT on Lines window, click on

OK



OK

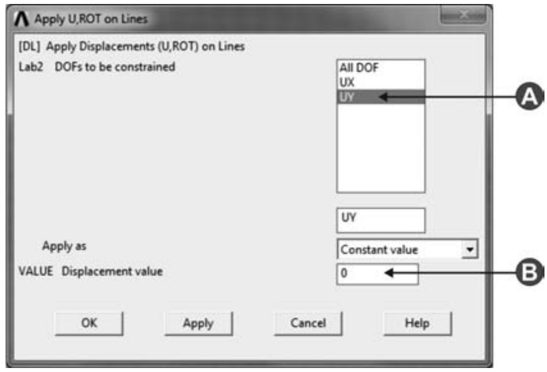
A select UX

B type 0 in Displacement value

Apply

In the ANSYS graphics, click on the horizontal bottom line, where zero y-direction displacement is applied. Then, in Apply U,ROT on Lines window, click on

OK



A select UY

B type 0 in Displacement value

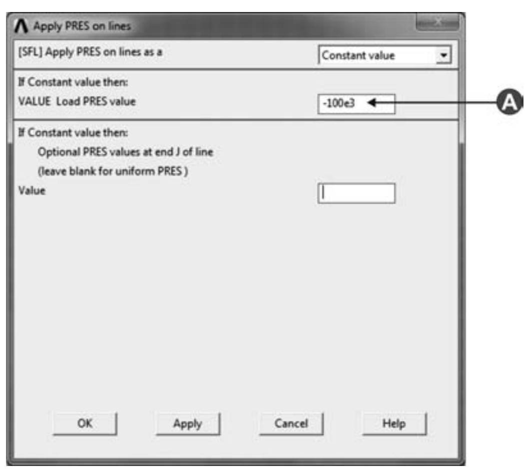
OK

Pressure is applied at the right vertical side of the plate. The negative pressure means that the pressure is tensile.

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Lines

In the ANSYS graphics, click on the right vertical line. Then, in Apply PRES on lines window, click on

OK



A type –100e3 in Load PRES value

OK

The final step is to run the solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem. Results can be plotted and listed in the General Preprocessor task.

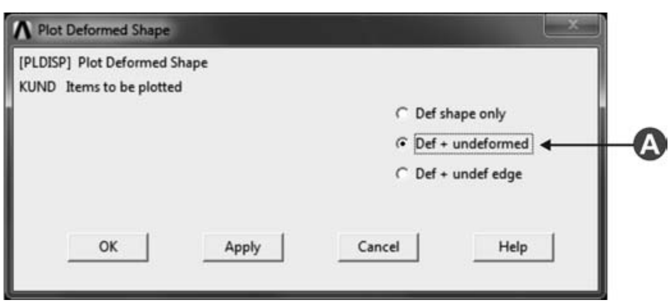
Main Menu > Solution > Solve > Current LS

OK

Close

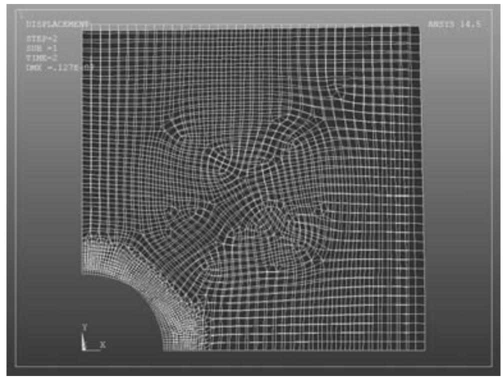
The above windows indicate that the solution task is completed successfully. The next step is for getting the results. The first figure shows the plate before and after deformation. Inspecting this figure is very important to determine if the problem is solved correctly. The second figure is the stress contours in the x-direction. This figure is used to determine the maximum stress in the x-direction, and the stress concentration factor.

Main Menu > General Postproc > Plot Results > Deformed Shape



A select Def + undeformed

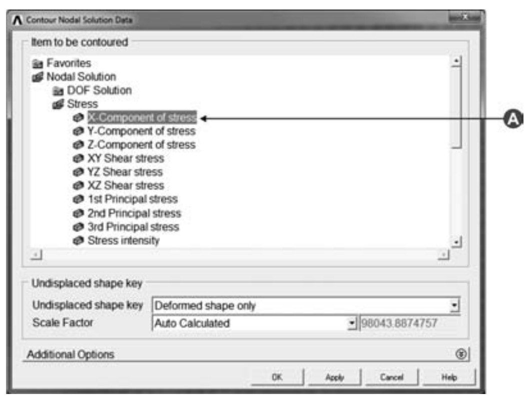
OK



ANSYS graphics show the plate before and after deformation

The above figure indicates that the left vertical and bottom horizontal lines are fixed, while the right vertical line is moved to the right in the direction of the applied pressure. The result is as expected.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > Stress > X-Component of stress

OK

ANSYS graphics show the x-direction stress contours

The stress contours show the location of the maximum stress in the x-direction. As expected, it is at the top of the hole. The accuracy of the result can additionally be improved if the number of elements in the model is increased. As shown in the stress contours, the maximum stress is 360,874 N/m^2. The maximum stress can also be obtained using the stress concentration in Figure 3.6c. The average stress is

$$\sigma_{ave} = \frac{100 \times 10^3 \times 0.025}{0.025 - 0.00625} = 133333.33 \text{ N/m}^2$$

Using Figure 3.6c, the stress concentration should be

$$K = 2.34$$

and the maximum stress is calculated as follows:

$$\sigma_{max} = K\sigma_{ave} = 2.34 \times 133333.33 = 312{,}000 \text{ N/m}^2$$

Comparing the ANSYS maximum stress with maximum stress obtained from Figure 3.6c, the error is calculated as

$$\text{Error} = \frac{|312000 - 360874|}{312400}(100) = 15.6\%$$

3.6 Axisymmetric elements

An axisymmetric solid that is subjected to axisymmetric loading is commonly found, and it is considered as a special case. Hence, a complex three-dimensional solid model is simplified to a two-dimensional model. Moreover, there are no additional degrees of freedom in the simplification. The axisymmetric solid model to be considered in this illustration is a cylinder with rectangular cross-sectional area, as shown in Figure 3.7a. Because of symmetry in the geometry and loads, no tangential degrees of freedom are assumed and, therefore, all derivations are independent of the circumferential angular variables. The plane strain finite element that is developed in this chapter can be extended to an axisymmetric element, by replacing the integration along the thickness to along the circumference. The cross-sectional area of the axisymmetric solid model is shown in Figure 3.7b.

A linear triangular element is used to mesh the cross-sectional area, as shown in Figure 3.7c. The nodal displacement functions in the r- and the z-directions are

$$u(r,z) = a_1 + a_2 r + a_3 z \tag{3.111}$$

$$w(r,z) = a_4 + a_5 r + a_6 z \tag{3.112}$$

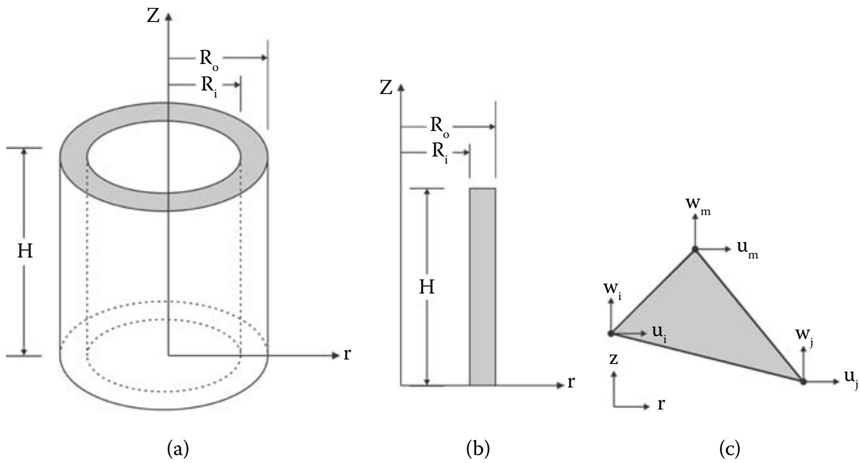


FIGURE 3.7 (a) Axisymmetric solid model, (b) cross-sectional area of the axisymmetric solid model, and (c) an axisymmetric element.

The cylindrical coordinates for the three nodes are (r_1, z_1), (r_2, z_2), and (r_3, z_3). The displacement functions can be expressed in matrix form for all nodes as

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ z_1 \\ z_2 \\ z_3 \end{Bmatrix} = \begin{bmatrix} 1 & r_1 & z_1 & 0 & 0 & 0 \\ 1 & r_2 & z_2 & 0 & 0 & 0 \\ 1 & r_3 & z_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & r_1 & z_1 \\ 0 & 0 & 0 & 1 & r_2 & z_2 \\ 0 & 0 & 0 & 1 & r_3 & z_3 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{Bmatrix} \tag{3.113}$$

and symbolically:

$$\{d\} = [B]\{a\} \tag{3.114}$$

The constants a's can be obtained by inverting the matrix [B] as follows:

$$\{a\} = [B]^{-1}\{d\} \tag{3.115}$$

The inverse of the matrix [B] is calculated as

$$[B]^{-1} = \frac{1}{\gamma} \begin{bmatrix} \omega_1 & \omega_2 & \omega_3 & 0 & 0 & 0 \\ \omega_4 & \omega_5 & \omega_6 & 0 & 0 & 0 \\ \omega_7 & \omega_8 & \omega_9 & 0 & 0 & 0 \\ 0 & 0 & 0 & \omega_1 & \omega_2 & \omega_3 \\ 0 & 0 & 0 & \omega_4 & \omega_5 & \omega_6 \\ 0 & 0 & 0 & \omega_7 & \omega_8 & \omega_9 \end{bmatrix} \tag{3.116}$$

where

$$\omega_1 = r_2z_3 - r_3z_2 \qquad \omega_2 = r_3z_1 - r_1z_3 \qquad \omega_3 = r_1z_2 - r_2z_1$$
$$\omega_4 = z_2 - z_3 \qquad \omega_5 = z_3 - z_1 \qquad \omega_6 = z_1 - z_2 \tag{3.117}$$
$$\omega_7 = r_3 - r_2 \qquad \omega_8 = r_1 - r_3 \qquad \omega_9 = r_2 - r_1$$
$$\gamma = r_1\,(z_2 - z_3) + r_2\,(z_3 - z_1) + r_3\,(z_1 - z_2)$$

The unit elongation of the element in radial direction is expressed as

$$\varepsilon_r = \frac{\partial u}{\partial r} \tag{3.118}$$

and the total tangential strain is given by

$$\varepsilon_\theta = \frac{u}{r} + \frac{\partial v}{r\,\partial\theta} \tag{3.119}$$

where the second term is the tangential strain due to the displacement in the y-direction. The shearing strain is expressed as

$$\gamma_{r\theta} = -\frac{v}{r} + \frac{\partial u}{r\,\partial\theta} + \frac{\partial v}{\partial r} \tag{3.120}$$

For an axisymmetric deformation, the displacement in the θ-direction is equal to zero, and hence, the strain and shearing strain are expressed as

$$\varepsilon_r = \frac{\partial u}{\partial r} = a_2 \tag{3.121}$$

$$\varepsilon_\theta = \frac{u}{r} = \frac{a_1}{r} + a_2 + \frac{a_3 z}{r} \tag{3.122}$$

$$\varepsilon_z = \frac{\partial w}{\partial z} = a_6 \tag{3.123}$$

$$\gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r} = a_3 + a_5 \tag{3.124}$$

$$\gamma_{r\theta} = 0 \tag{3.125}$$

$$\gamma_{\theta z} = 0 \tag{3.126}$$

Strain and shearing strain are expressed in matrix form as

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{rz} \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 1/r & 1 & z/r & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{Bmatrix} \tag{3.127}$$

Equation 3.127 is symbolically expressed as

$$\{\varepsilon\} = [G]\{a\} \tag{3.128}$$

Using Equation 3.115 in Equation 3.128 yields

$$\{\varepsilon\} = [A]\{d\} \tag{3.129}$$

$$[A] = [G][B]^{-1} \tag{3.130}$$

For an isotropic material, and the case of thermal strain due to temperature rise ΔT, the strain–stress relationship is expressed as

$$\begin{Bmatrix} \sigma_r \\ \sigma_\theta \\ \sigma_z \\ \tau_{rz} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1}{2}-\nu \end{bmatrix} \left[\begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{rz} \end{Bmatrix} - \begin{Bmatrix} \alpha \Delta T \\ \alpha \Delta T \\ \alpha \Delta T \\ 0 \end{Bmatrix} \right] \tag{3.131}$$

where α is the thermal coefficient of expansion. The symbolic expression for Equation 3.131 is

$$\{\sigma\} = [C][A]\{d\} - [C]\{\varepsilon^i\} \tag{3.132}$$

Castigliano's first theorem, which is the same as Equation 3.74 with an additional term for thermal strain, is expressed as follows:

$$\{F\} = [K]\{d\} - \{F^i\} \tag{3.133}$$

where the stiffness matrix is obtained as

$$[K] = \int [A]^T [C][A] \, dV \tag{3.134}$$

Putting Equation 3.130 into Equations 3.133 and 3.134 yields

$$\left[F^i\right] = \left([B]^{-1}\right)^T \int [G]^T [C]\left\{\varepsilon^i\right\} \, dV \tag{3.135}$$

Finally, the stiffness matrix is

$$[K] = \left([B]^{-1}\right)^T \left(\int [G]^T [C][G] \, dV \right) [B]^{-1} \tag{3.136}$$

3.7 Displacement analysis of a vessel under transient loading with ANSYS

The vessel shown in Figure 3.8a is subjected to internal transient pressure loading. Determine the maximum displacement in the radial direction of the vessel as a function of time. Take the advantage of symmetry in the problem to reduce the computational size. The inner and outer radii are 0.25 and 0.265 m, respectively, and the height of the vessel is 0.75 m. Let E = 180 GPa and v = 0.33. The upper and lower horizontal surfaces are fixed. The total time duration for the process is 1000 seconds. The internal pressure is a function of time according to the following equation:

$$P(t) = 100\times 10^3 + 100\times 10^3 \sin\left(\frac{2\pi}{100}(t+75)\right)$$

Double click on the Mechanical APDL Product Launcher icon

This example is limited to structural analysis. Hence, select Structural only. Solid element is used, and its shape is rectangular with four nodes. The following steps are for selecting the element type and its behavior:

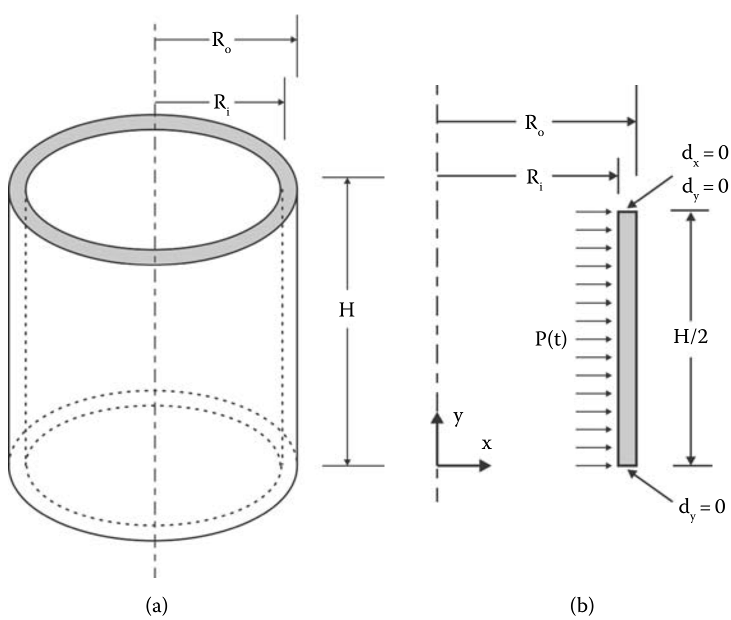
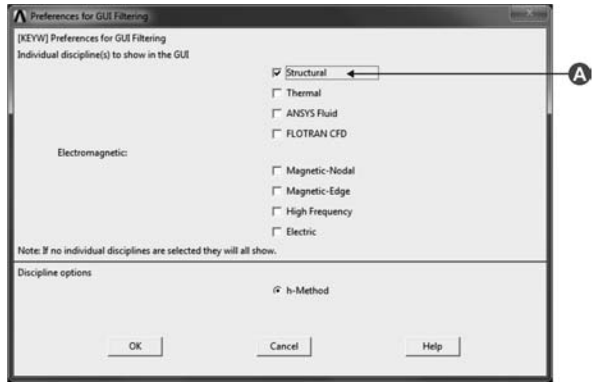


FIGURE 3.8 (a) A vessel subjected to time-dependent pressure at the inner surface. (b) The axisymmetric model for the problem.

Main Menu > Preferences

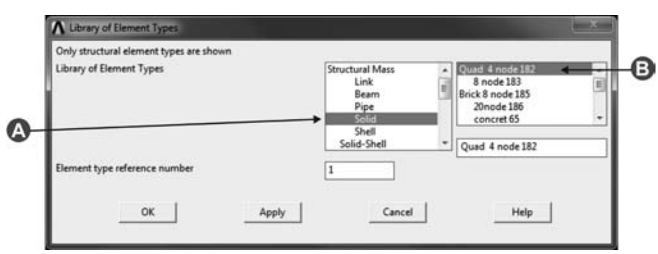


A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add…



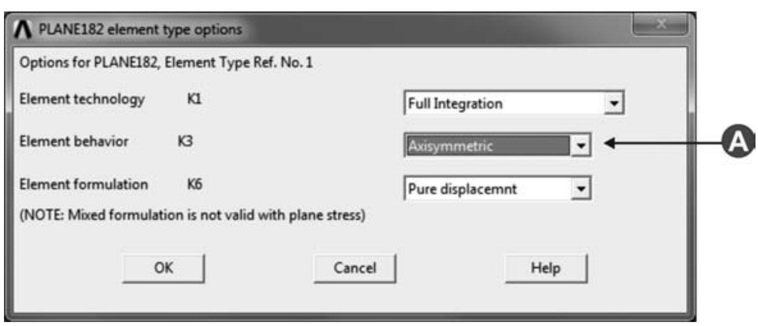
A select Solid

B select Quad 4 node 182

OK

In the element type, Quad means that a quadratic element is selected. The first digit of number 42 is the number of nodes in the element, and the second digit is the number of degrees of freedom in each node, which are the x- and y-displacements. In the following steps, the element behavior is changed from plane stress to axisymmetric. The results from the axisymmetric analysis should be more accurate than the results from three-dimensional elements, and the computational time will be less.

Options…



A select Axisymmetric in Element behavior

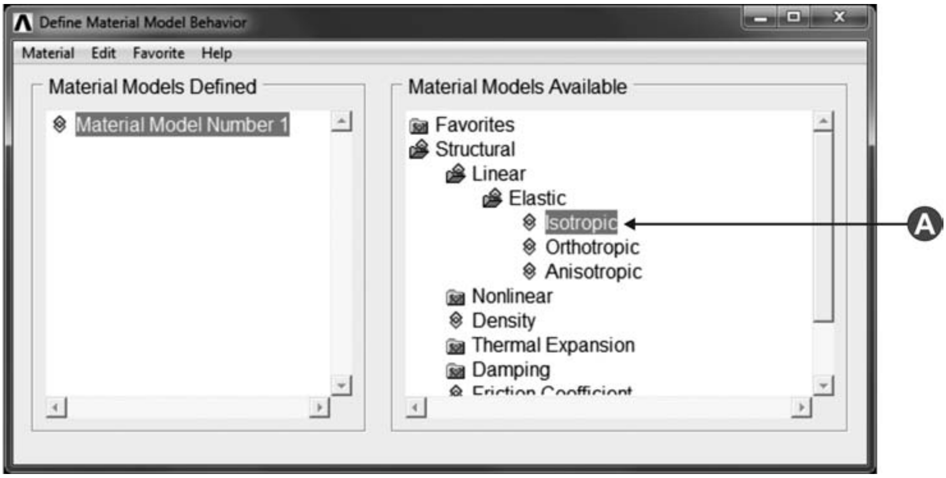
OK

Close the Element Types window

Close

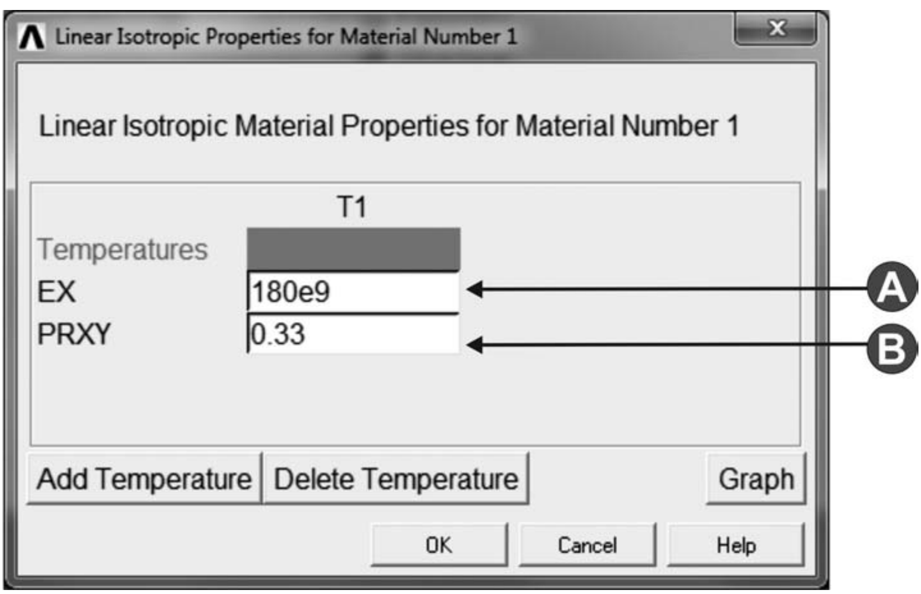
For the material properties, the elasticity and Poisson ratio are required to solve the problem.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

The following windows will show up.



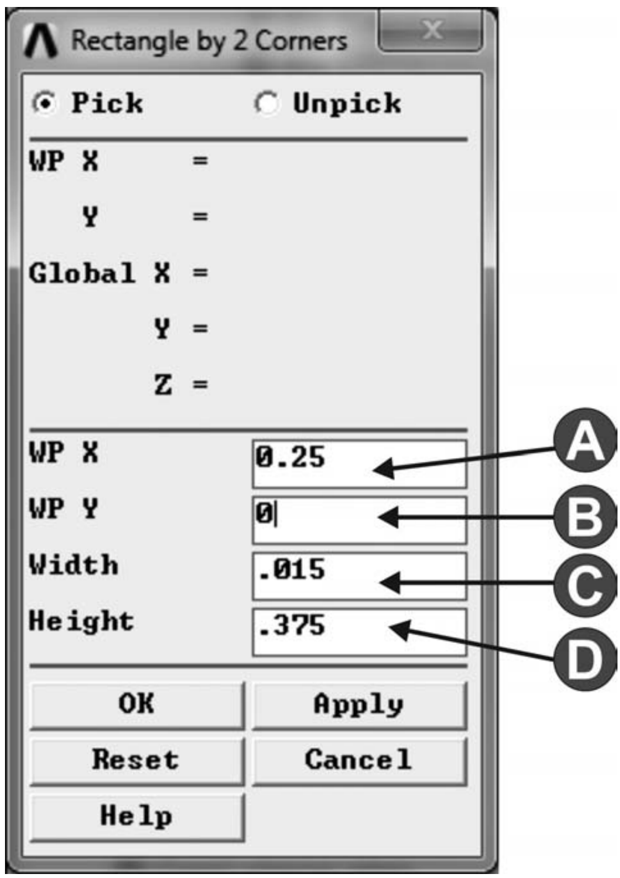
A type 180e9 in EX
B type 0.33 in PRXY

OK

Close the Material Model Behavior window

The geometry is simply a rectangle. The advantage of symmetry in the problem is considered, and only the cross-sectional area of the vessel is considered. Figure 3.8b shows the considered geometry used to solve the problem. Notice that the axis of rotation is the y-axis, and no geometry is allowed in the negative x-axis region. The geometry rotates about the positive y-axis. Additionally, only the upper half of the vessel will be modeled.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0.25 in WP X

B type 0 in WP Y

C type .015 in Width

D type .375 in Height

OK

Main Menu > Preprocessor > Meshing > Mesh Tool

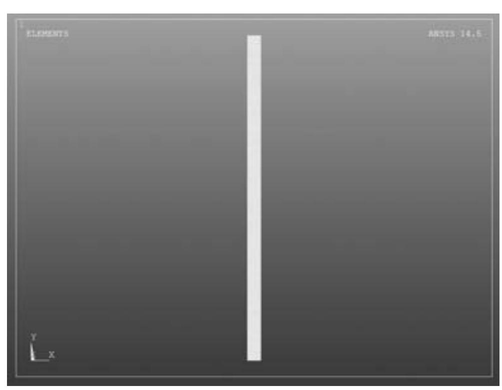
A select Smart Size

B set the level to 1

Mesh

In Mesh Areas window, click on

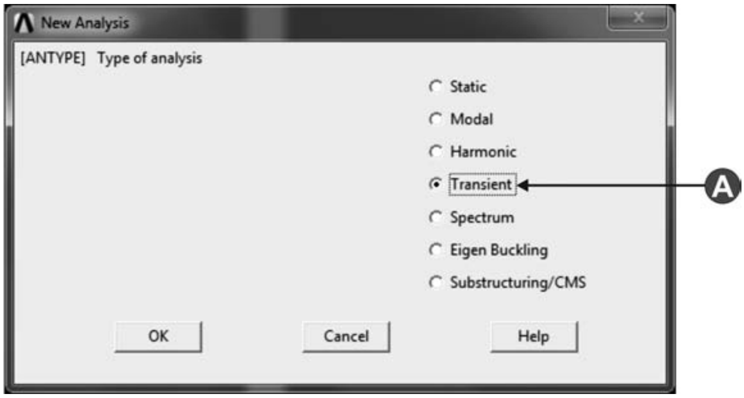
Pick All



ANSYS graphics show the mesh of the cross-sectional area of the vessel

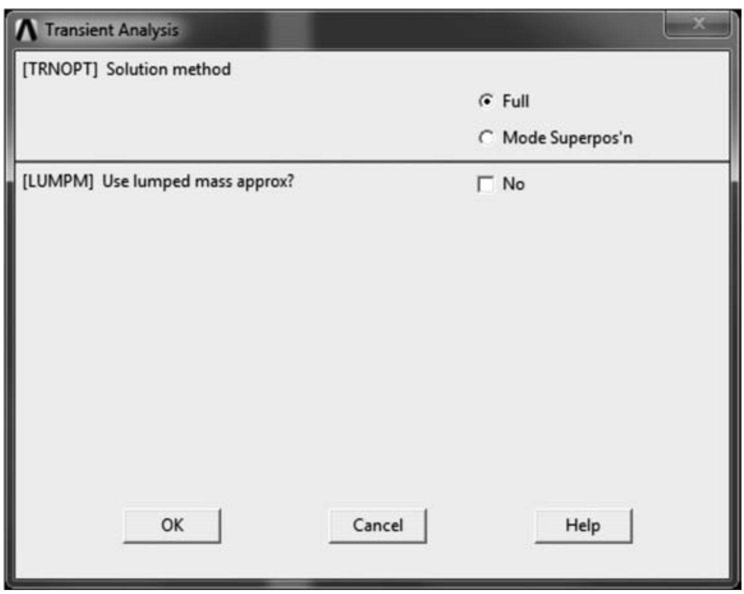
Modeling and meshing are completed at this point. Next, the solution is switched to transient, and unsteady parameters are specified. The boundary conditions are applied starting with the displacements, then the transient pressure. This order is not important for solution.

Main Menu > Solution > Analysis Type > New Analysis



A select Transient

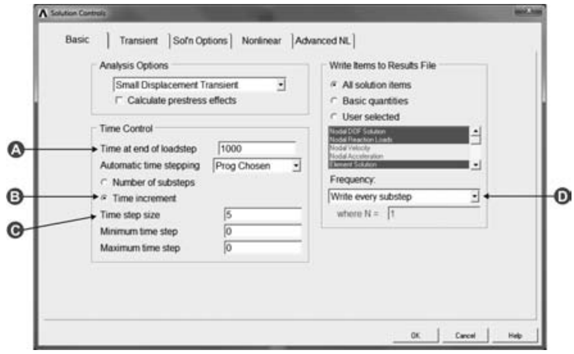
OK



OK

The total time duration for the process is 1000 seconds and to ensure obtaining accurate results, the process duration is divided into 200 time steps. Hence, the time step for this problem is 5 seconds. Results for all time steps are stored by selecting the Write every substep option in Frequency.

Main Menu > Solution > Analysis Type > Sol'n Control



A type 1000 in Time at end of loadstep

B select Time increment

C type 5 in Time step size

D select Write every substep

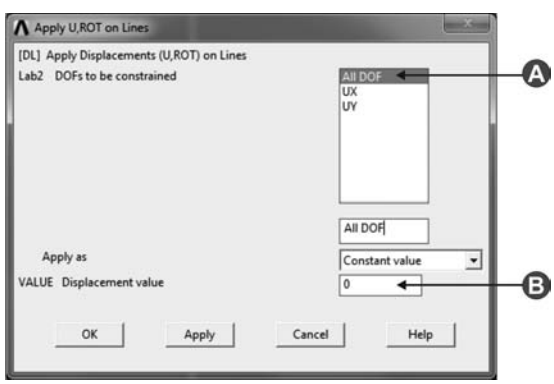
OK

The function editor is used to apply the transient pressure equation at the inner surface. This technique is simple and convenient for this problem because an equation for the pressure is given. The upper horizontal line is fixed, while the lower horizontal line is fixed in the y-direction only due to the symmetry. Transient pressure is applied at the left vertical line.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines

In the ANSYS graphics, click on the top lines that are fixed, and in Apply U,ROT on Lines window, click on

OK



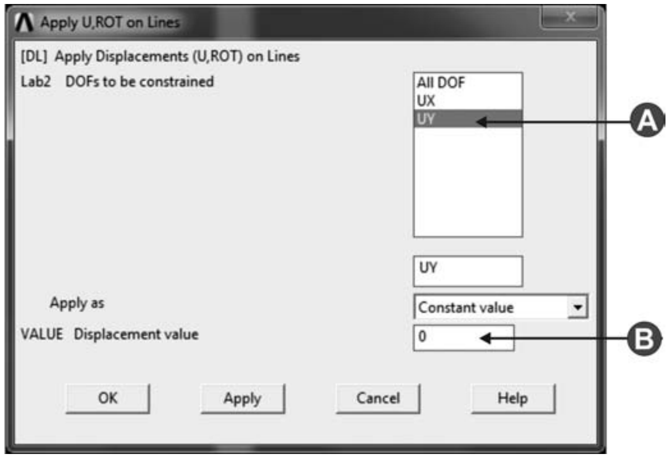
A select All DOF

B type 0 in Displacement value

Apply

In the ANSYS graphics, click on the bottom horizontal line, and in Apply U,ROT on Lines window, click on

OK

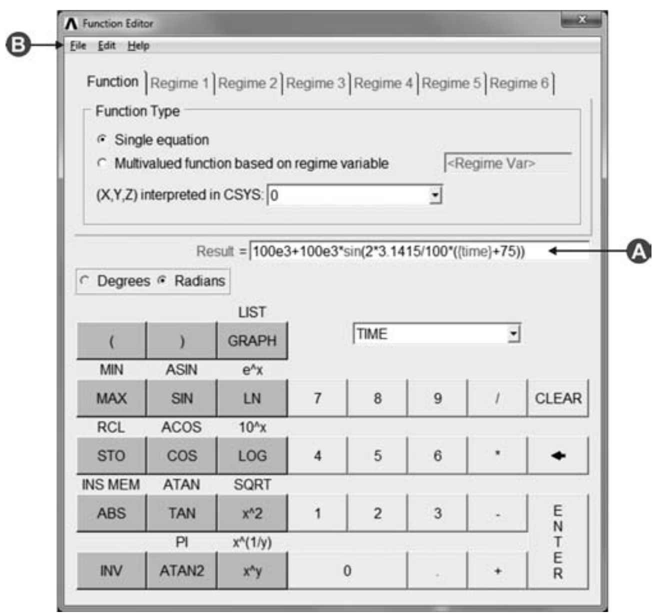


A select UY

B type 0 in Displacement value

OK

Main Menu > Solution > Define Loads > Apply > Functions > Define/Edit



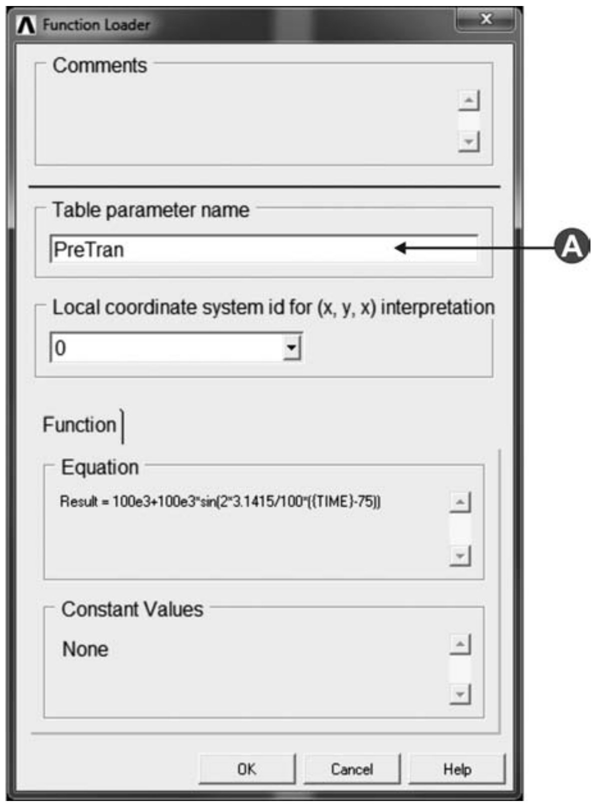
A Type the equation: 100e3+100e3*sin(2*3.1415/100*({time}+75))

B File → Save

Save the file as PreTran, and this file name is optional. After saving the function, it is required to load it to the ANSYS solution using the read file, and then close the Function Editor window.

Main Menu > Solution > Define Loads > Apply > Functions > Read File

Select the file PreTran, then open, and the following window will show up.



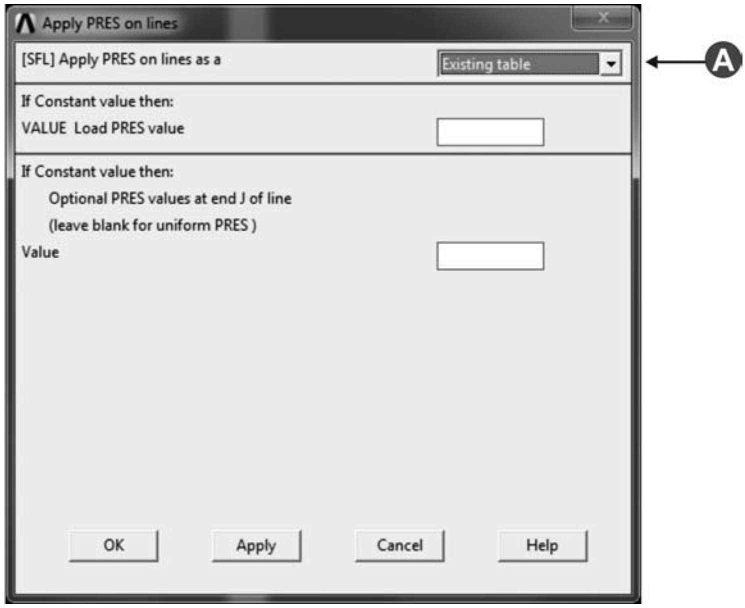
A type PreTran in Table parameter name and this name is optional, and it should not be the same as the file name of the function.

OK

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Lines

In the ANSYS graphics, click on the left vertical line where pressure is applied, and then in Apply PRES on lines window, click on

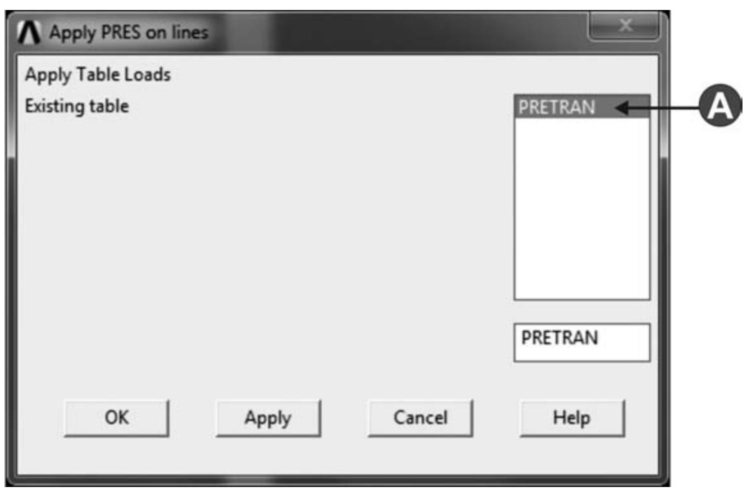
OK



A select Existing table

OK

The following window will show up to select the function.



A select PRETRAN

OK

The final step in the solution task is to run the solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem. Results can be plotted and listed in the General Preprocessor task.

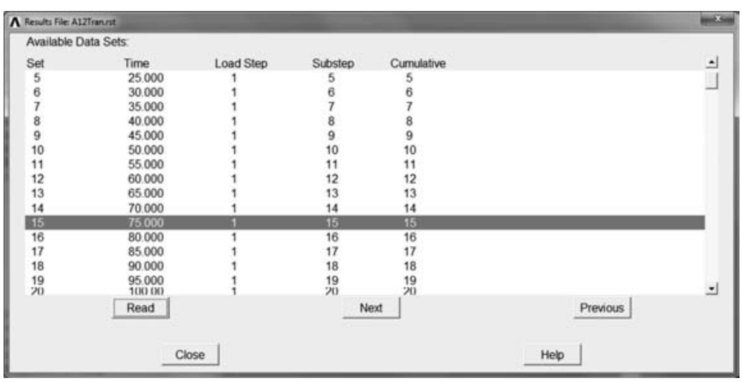
Main Menu > Solution > Solve > Current LS

OK

Close

The above window indicates that the solution task is successfully accomplished. The next step is for getting the results. First, the results at time of 75 seconds is uploaded, and the deformed shape and displacement contours in the x-direction are displayed.

Main Menu > General Postproc > Read Results > By Pick

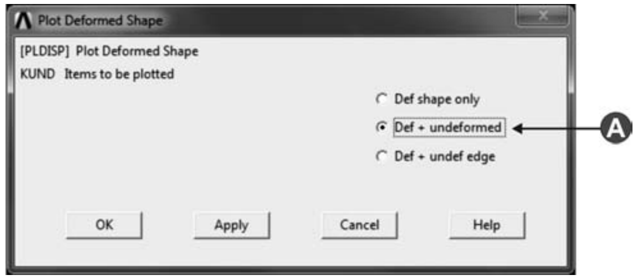


A select set 15

Read

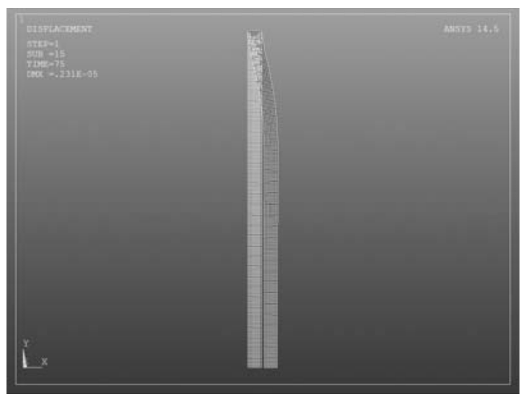
Close

Main Menu > General Postproc > Plot Results > Deformed Shape



A select Def + undeformed

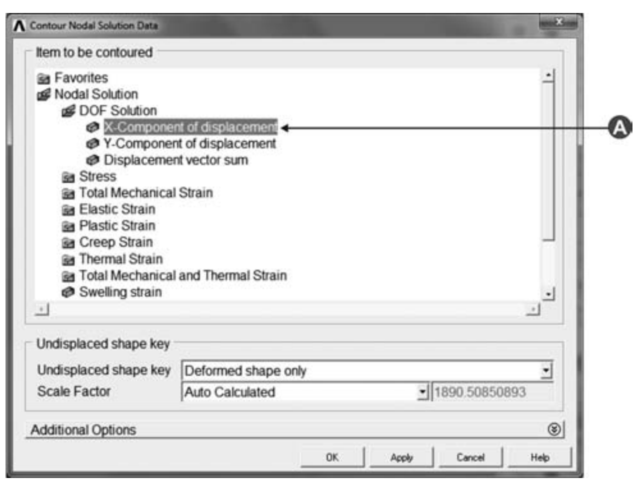
OK



ANSYS graphics show the vessel wall before and after deformation

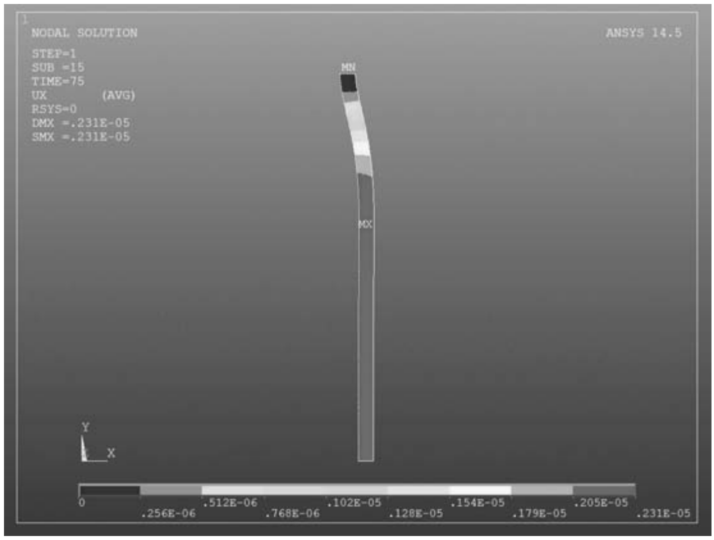
The above figure indicates that the upper line is fixed, while the bottom line is moved to the outside. The maximum displacement is exactly at the middle of the vessel, as expected.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution → DOF Solution → X-Component of displacement

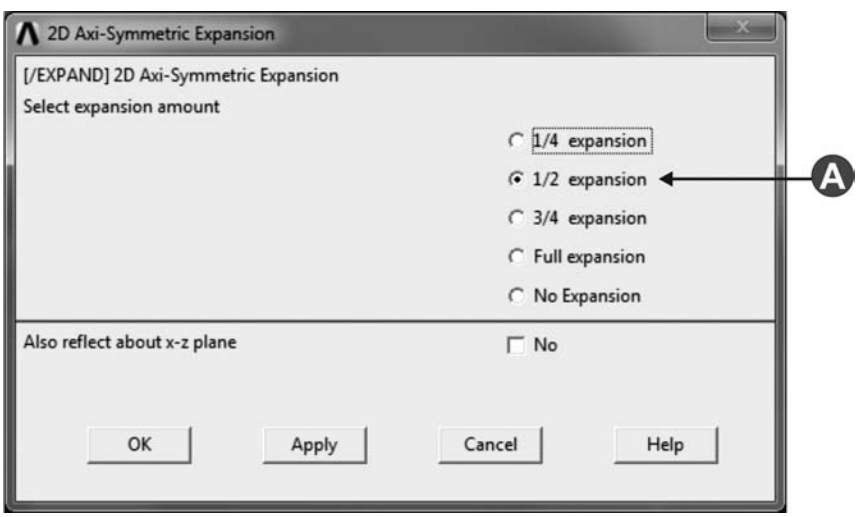
OK



ANSYS graphics show the contours of the displacement in the x-direction

The maximum displacement is equal to 0.231×10^{-5} m. To visualize the results in three-dimensional space, the vessel wall is partially expanded in 180° as follows:

Utility Menu > PlotCtrls > Style > Symmetry Expansion > 2D Axi-Symmetric

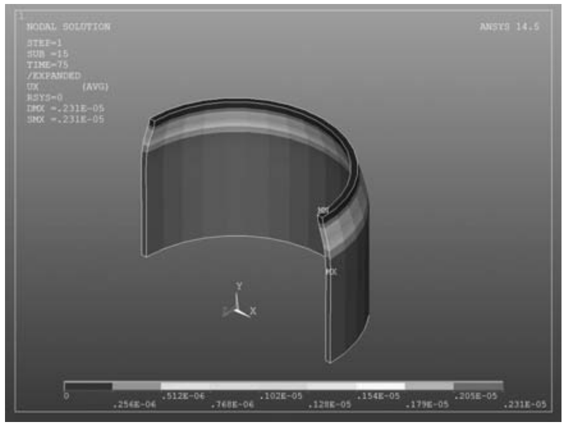


A select 1/2 expansion

OK

Utility Menu > PlotCtrls > Pan-Zoom-Rotate

Click on the isometric view.

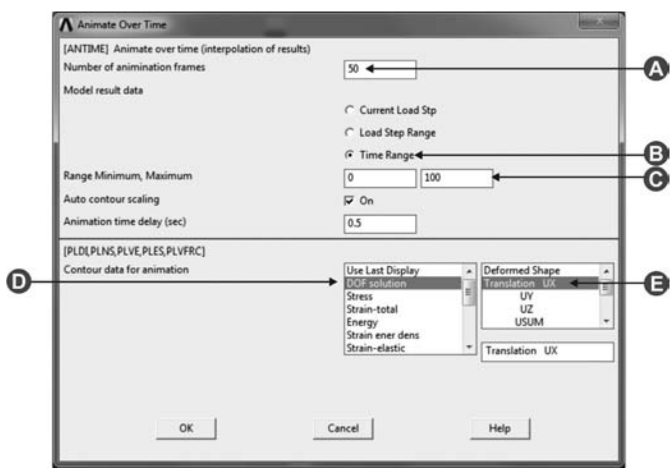


ANSYS graphics show the x-direction displacement contours in three dimensions

Animation of the deformation from time = 0–100 seconds, one cycle, can be easily done using animate in the PlotCtrol options. The Number of the frames in the animate over time is the number of pictures in the avi file, while the animation time delay is the display period between two pictures. Fifty animation frames produce a good resolution animation file and with 0.5 second, the animation file duration is 25 seconds (50 frames × 0.5 delay).

Main Menu > General Postproc

Utility Menu > PlotCtrls > Animate > Over time …



A type 50 in Number of animation frames

B select Time Range

C type 0 and 100 for Range Minimum, Maximum

D select DOF solution

E select Translation UX

$\boxed{\textbf{OK}}$

ANSYS creates an animation file for the deformation.

3.8 Contact element analysis

In this section, the contact pressure distribution between two bodies held in contact is determined. Figure 3.9 shows two parallel cylinders of length L in contact. The upper cylinder has a radius of r_1, while the lower cylinder has a radius of r_2. Force F is applied to the upper and lower cylinders. At the contact region, a semielliptical contact pressure distribution is developed within each cylinder. The width of the contact line is 2a, and the contact pressure varies from zero at the end of the contact line to the largest value P_o at the center of the contact line.

The contact width and maximum contact pressure can be expressed, respectively, as

$$a = \left[\frac{4Fr_1r_2}{\pi L(r_1 + r_2)}\left(\frac{1-v_1^2}{E_1} + \frac{1-v_2^2}{E_2}\right)\right]^{1/2} \tag{3.137}$$

$$P_o = \frac{2}{\pi}\frac{F}{aL} \tag{3.138}$$

where E is the modulus of elasticity and v is Poisson's ratio. If the two cylinders are made of the same material and Poisson's ratio is equal to 0.3, the expressions (3.137) and (3.138) are simplified as follows:

$$a = 1.52\left[\frac{F}{EL}\left(\frac{r_1\ r_2}{r_1 + r_2}\right)\right]^{1/2} \tag{3.139}$$

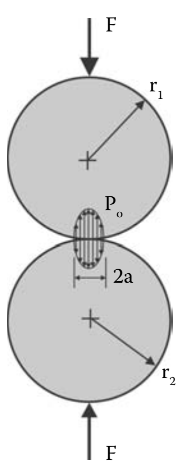


FIGURE 3.9 Two cylinders in contact.

$$P_o = 0.418\left[\frac{FE}{L}\left(\frac{r_1 + r_2}{r_1 r_2}\right)\right]^{1/2} \tag{3.140}$$

Expressions (3.139) and (3.140) can be used to determine the contact width and maximum pressure of a contact problem of a circular cylinder and a flat surface, as shown in Figure 3.10. If the radius of the second cylinder is very large, $r_2 = \infty$, the contact width and maximum contact pressure are expressed as

$$a = 1.52\left[\frac{Fr_1}{EL}\right]^{1/2} \tag{3.141}$$

$$P_o = 0.418\left[\frac{FE}{Lr_1}\right]^{1/2} \tag{3.142}$$

When two spheres are in contact with different materials and loaded with force F, the contact area will have a circular shape of diameter a. The contact area and maximum stress can be expressed as

$$a = \left[\frac{3F}{4}\frac{\left(1 - v_1^2\right)/E_1 + \left(1 - v_2^2\right)/E_2}{1/r_1 + 1/r_2}\right]^{1/3} \tag{3.143}$$

$$P_o = 1.5\frac{F}{\pi a^2} \tag{3.144}$$

If the sphere of radius r_1 contacting a large flat body of the same material, the expressions of the contact area, and the maximum stress for the contact can be obtained by substituting $r_2 = \infty$ in Equation 3.143, and then the contact diameter and maximum pressure become

$$a = 0.88\left[\frac{3Fr_1}{E}\right]^{1/3} \tag{3.145}$$

$$P_o = 0.62\left[\frac{FE^2}{4r^2}\right]^{1/3} \tag{3.146}$$

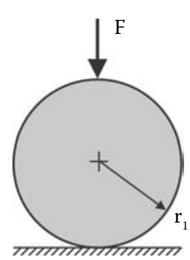


FIGURE 3.10 A cylinder and a flat plate in contact.

3.9 Two horizontal cylinders in contact analysis using ANSYS

Two horizontal cylinders are placed close to each other, as shown in Figure 3.11. The two cylinders have the same radius, r = 0.01 m, and force of 10 kN is applied to upper and lower cylinders, as shown in the figure. As a result, the cylinders will move and meet. Determine the maximum pressure at the contact region, given that E = 200 GPa and v = 0.3 for both cylinders.

Contact problems are highly nonlinear and require large computer resources to solve. Additionally, the physics behind the contact problem is relatively complex, and it is important to understand the physics of the problem to solve it as accurately as possible. Too many parameters are required and just using the ANSYS default settings will give a reasonable result. Contact problems present two main difficulties. First, the regions of contact cannot be accurately predicted. The contact region depends on the loads, materials, boundary conditions, and other factors. Second, most contact problems need to account for friction. There are several friction models to choose from, and all are nonlinear, which makes solution convergence even more difficult. The basic steps for performing a contact analysis are as follows:

1. Create the model geometry and mesh.
2. Identify the contact pairs. The contact pairs are the region in which contact might occur during the deformation of the model. Once potential contact surfaces are identified, they must be defined via target and contact elements.
3. Select contact and target surfaces. Contact elements are constrained against penetrating the target surface, but target elements can penetrate through the contact surface. If a convex surface is

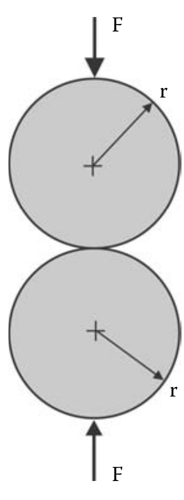


FIGURE 3.11 A contact element problem.

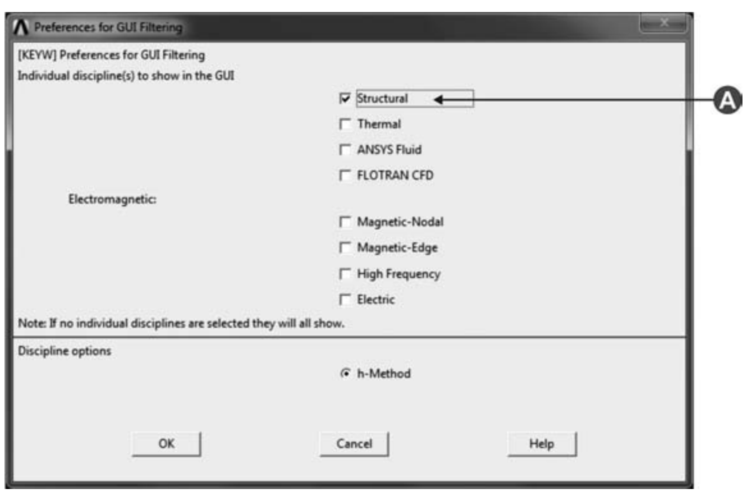
expected to meet a flat surface, the convex surface should be the contact surface. If one surface is smaller than the other, the smaller surface should be the contact surface.

4. Apply the boundary conditions. Any type of boundary conditions can be applied in the contact problems.
5. Define solution options and load steps. Convergence behavior for contact problems depends on the geometry and boundaries of the problem. The time step size must be small enough to ensure the convergence. The time step size is specified by a number of steps or the time step size itself. However, the best way to set an accurate time step size is to turn on the automatic time stepping in ANSYS solution options.
6. Solve the contact problem. Ensure that the problem is fully converged.
7. Finally, get the contact results. Results from a contact analysis consist mainly of displacements, stresses, strains, reaction forces, and the contact information, such as contact pressure.

Double click on the Mechanical APDL Product Launcher icon

This example is limited to structural analysis. Hence, select Structural in Preferences. Solid element is used, and its shape is rectangular with four nodes. The contact simulation requires another two element types: the target and contact elements. These two elements will be introduced into the model when the contact wizard tool is activated.

Main Menu > Preferences

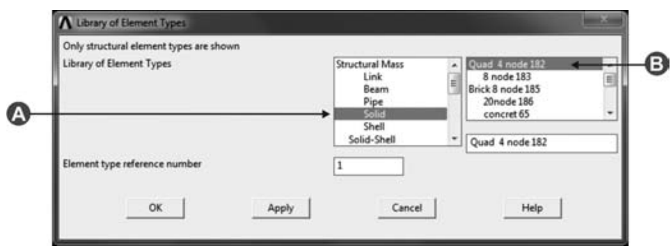


A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

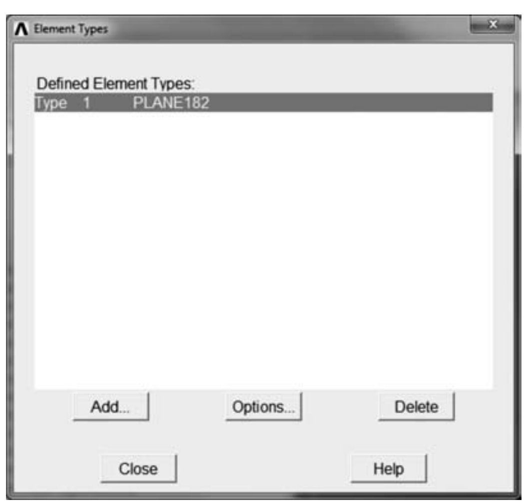
Add...



A select Solid

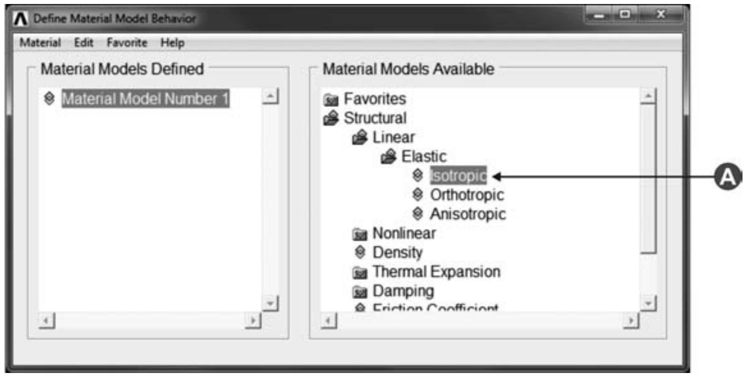
B select Quad 4 node 182

OK



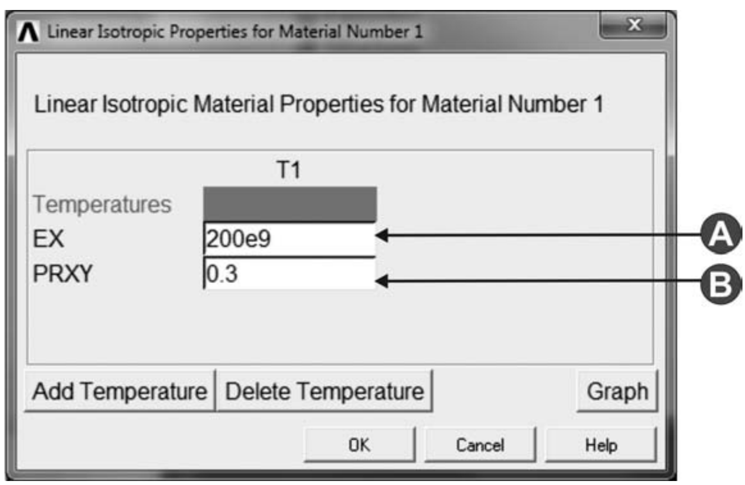
Close

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

The following window will show up. For the material properties, the modulus of elasticity and Poisson's ratio are required to solve the problem.



A type 200e9 in EX

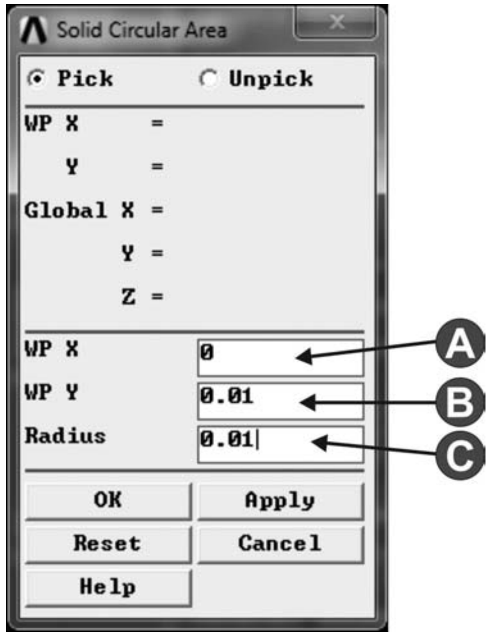
B type 0.3 in PRXY

OK

Close the Material Model Behavior window

Next, the geometry of the problem is created, which is two circles. The expected contact surfaces should be defined in the contact wizard. Additionally, the contact region should have a fine mesh for accurate numerical predication. Hence, a small circle of a radius of 0.002 m is created at the contact region and will have a fine mesh.

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle

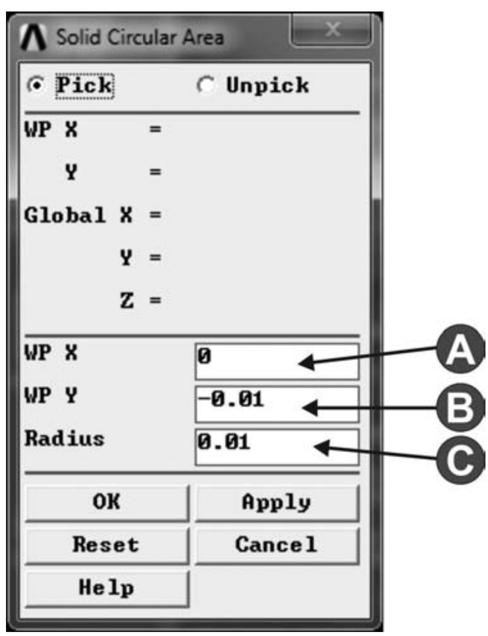


A type 0 in WP X

B type 0.01 in WP Y

C type 0.01 in Radius

Apply

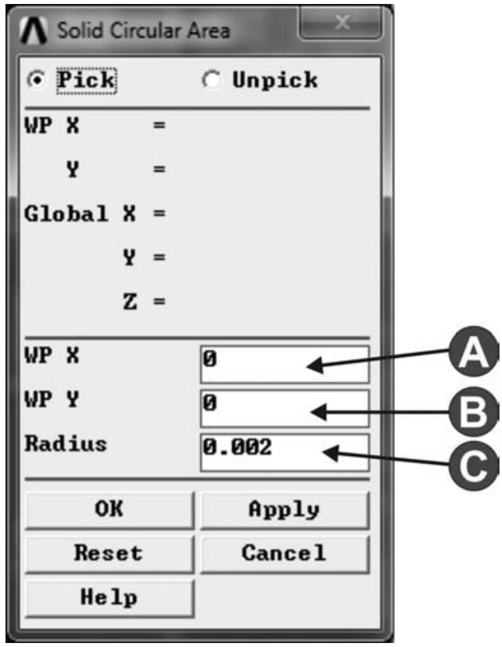


A type 0 in WP X

B type –0.01 in WP Y

C type 0.01 in Radius

Apply

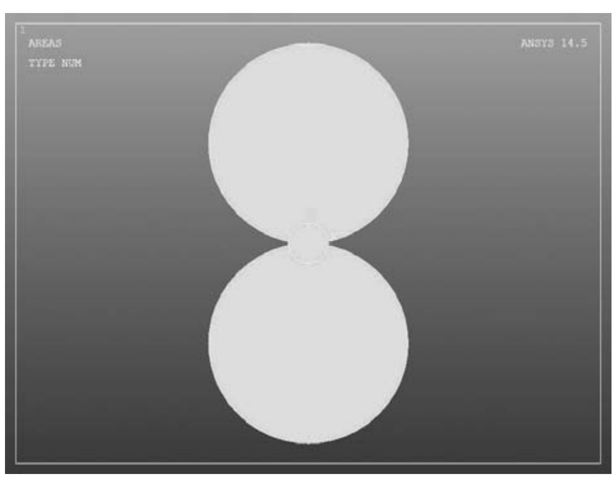


A type 0 in WP X

B type 0 in WP Y

C type 0.002 in Radius

OK



ANSYS graphics show the created three circles

First, the three areas are overlapped. Second, since there is symmetry in the problem, a vertical line is created to split the entire mode by half. Only the right symmetry side is considered. Finally, extra areas are deleted.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

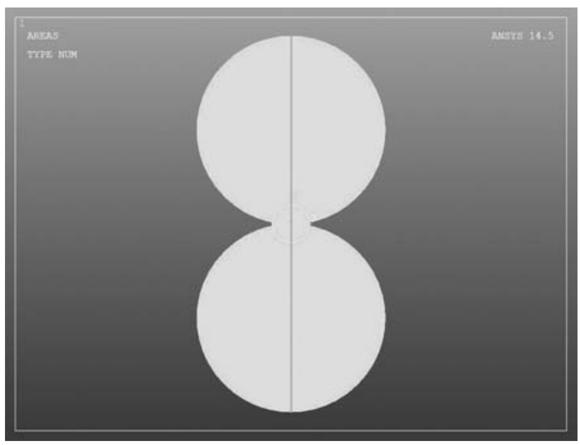
In Overlap Areas window, click on

Pick All

ANSYS Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Create a vertical line by clicking on two keypoints. The first at the top of the upper circle, and the second at the bottom of the lower circle, and then in create Straight Line window, click on

OK



ANSYS graphics show the created three overlapped circles and a vertical line

ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Line

To select all three circles, in Divide Area by Line window, click on

Pick All

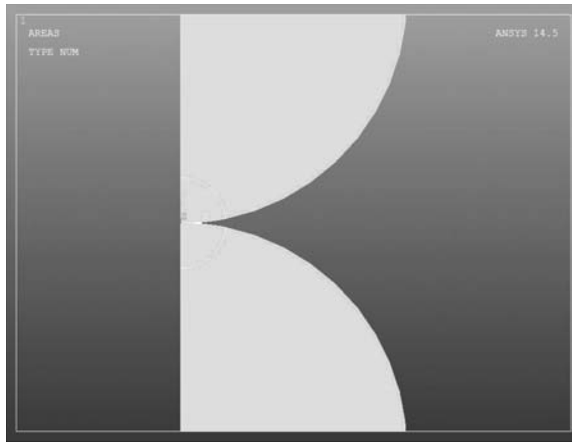
Click on the created vertical line. In Divide Area by Line window, click on

OK

Main Menu > Preprocessor > Modeling > Delete > Area and below

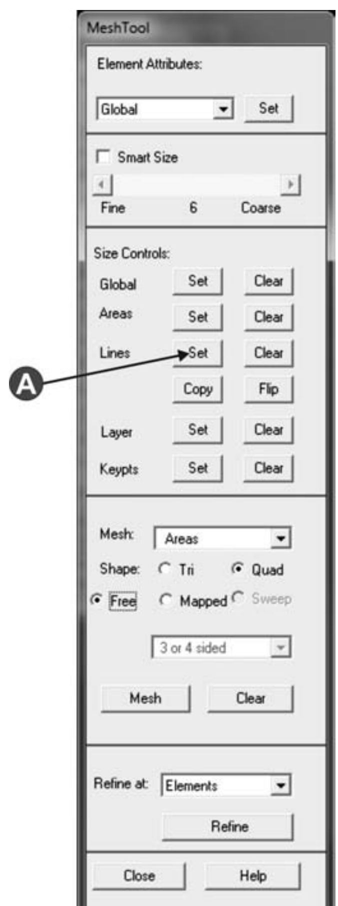
Select the left areas and the small area on the right side that is between the two circles. Then, in Delete Area and Below window, click on

OK



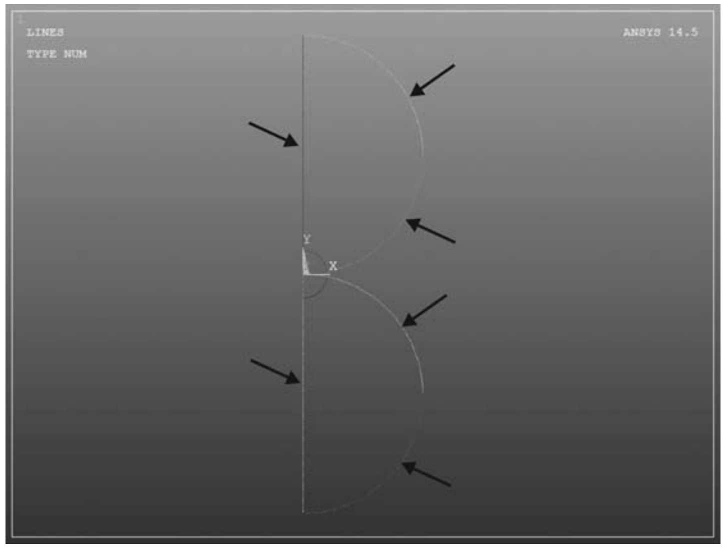
ANSYS graphics show the final geometry

Main Menu > Preprocessor > Meshing > Mesh Tool



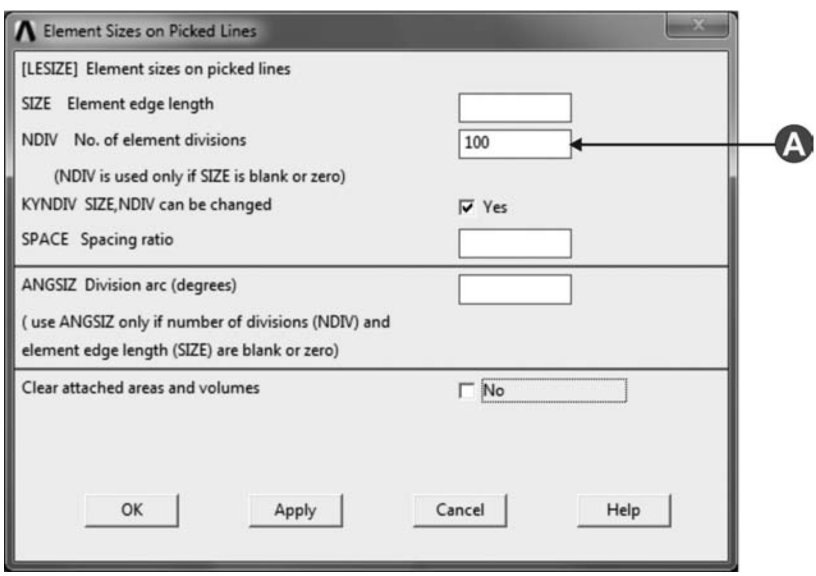
A Click on Set in Lines

Select the lines that are far away from the contact area, as illustrated in the following figure.



In Element Sizes on Picked Lines window, click on

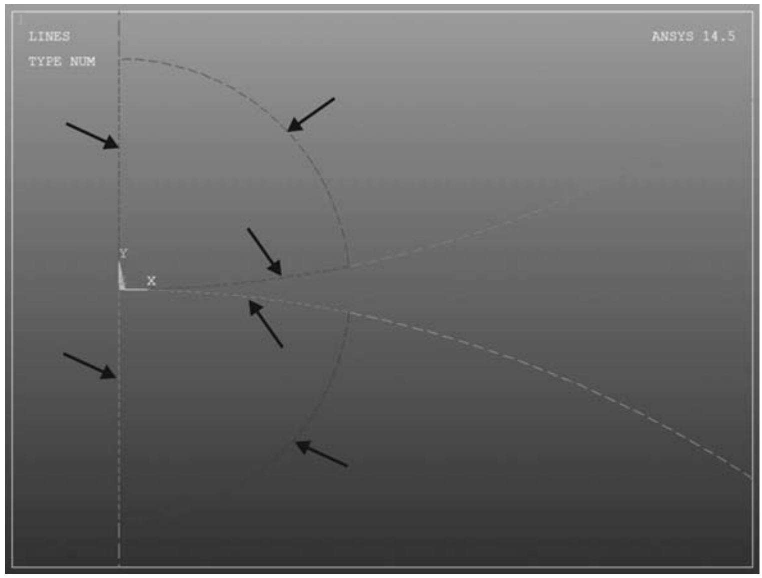
Apply



A type 100 in No. of element divisions

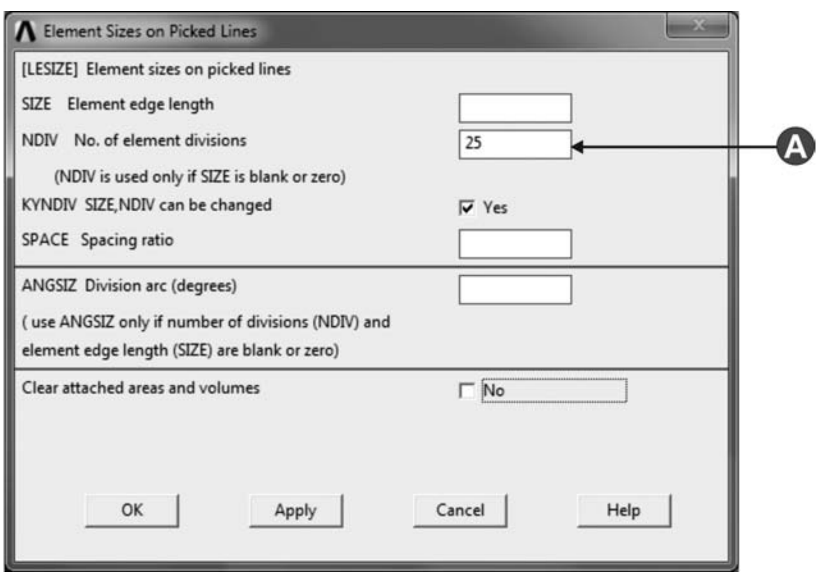
Apply

Select the lines that are close to the contact area, as illustrated in the following figure.



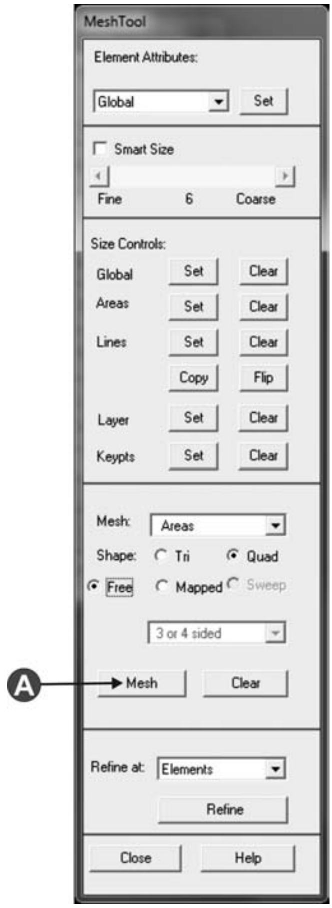
In Element Sizes on Picked Lines window, click on

OK



A type 25 in No. of element divisions

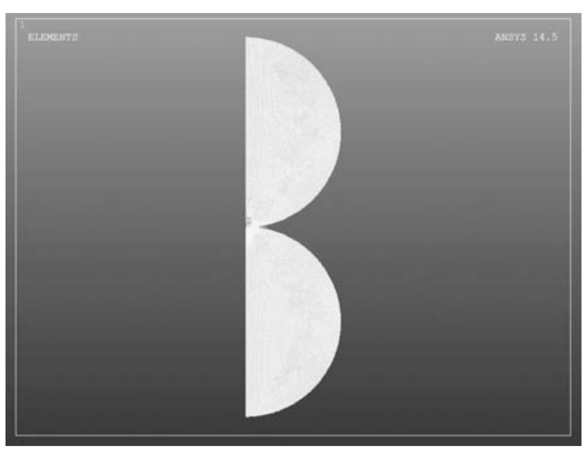
OK



A Mesh

In Mesh Areas, click on

Pick All

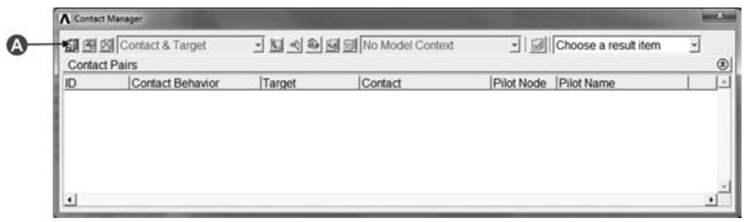


ANSYS graphics shows the mesh

In this problem, the top surface of the lower circle is the target surface because it is stationary, while the bottom surface of the upper circle is the contact surface because it is moving. Both contact and target surfaces are associated with the deformable bodies. These two surfaces together comprise the contact pair. The Contact Manager is very effective in defining, viewing, and editing the contact pairs. In addition, all contact pairs for the entire model can be managed. The Contact Wizard, which is accessed from the Contact Manager, makes the process of creating contact pairs very efficient.

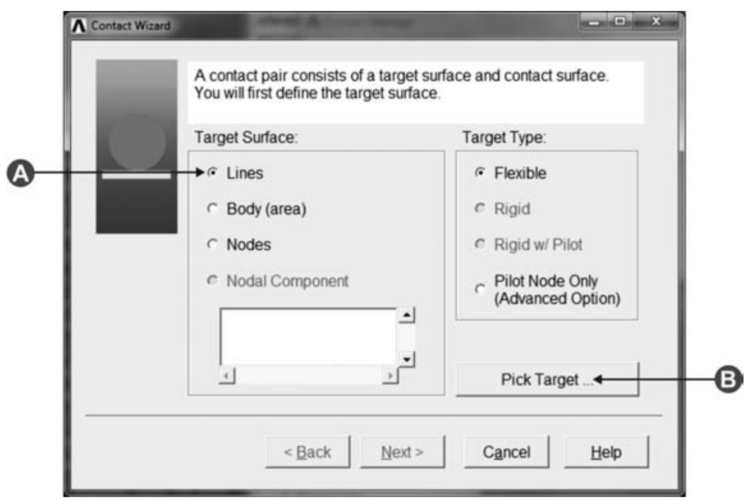
Utility Menu > Plot > Lines

Main Menu > Preprocessor > Modeling > Create > Contact Pair



A click on the contact wizard

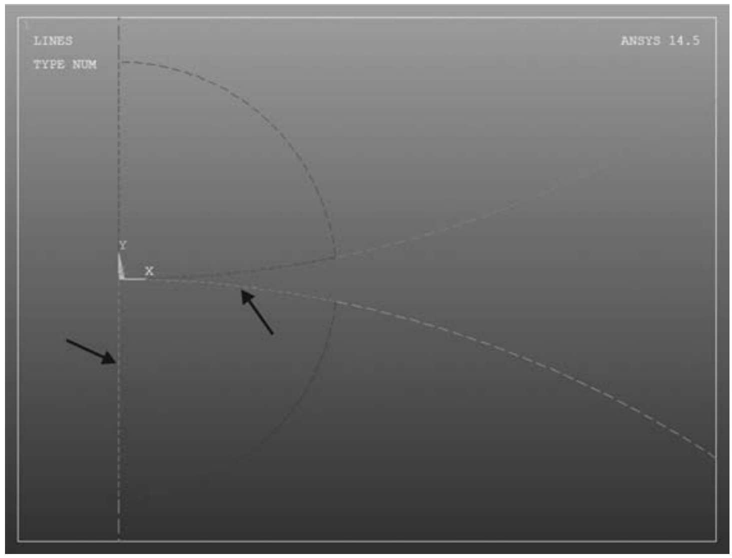
The contact wizard is used to select the target and contact surfaces. First, the target surface should be selected and then the contact surface.



A click on Lines

B click on Pick Target

Click on the top surfaces of the lower circle, as shown in the following figure.

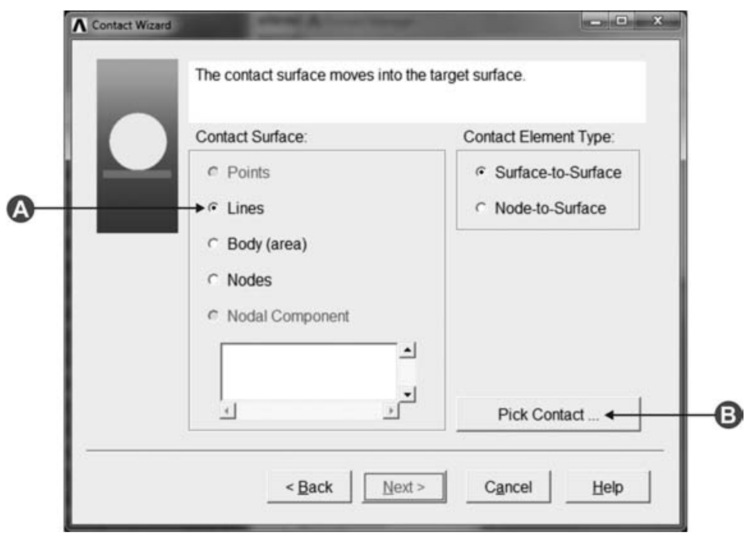


In Select Lines for Target window, click on

OK

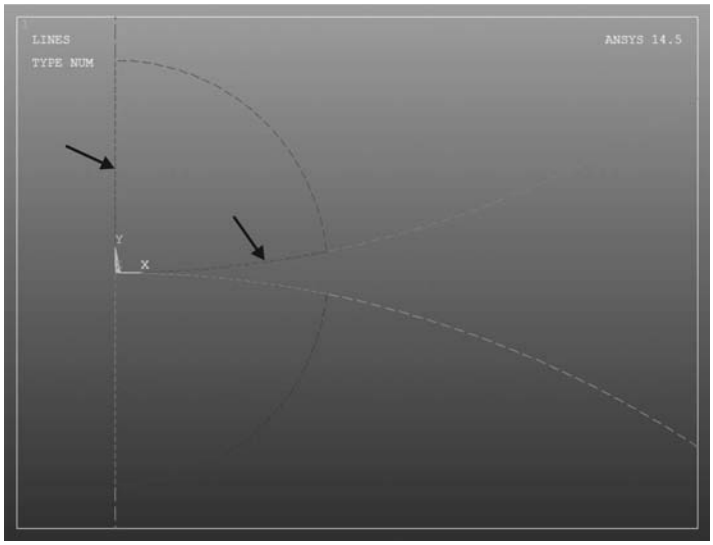
Then in the Contact Wizard window, click on

Next >



A click on Lines
B click on Pick Contact

Click on the bottom surfaces of the upper circle, as shown in the following figure.

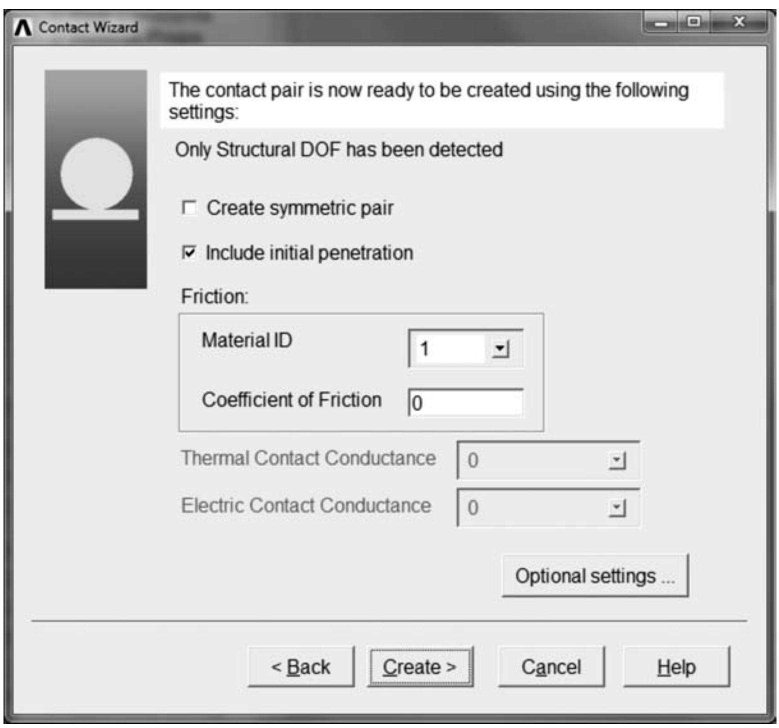


In Select Lines for Contact window, click on

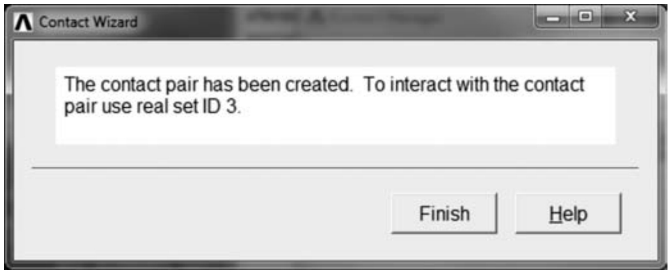
OK

Then in the Contact Wizard, click on

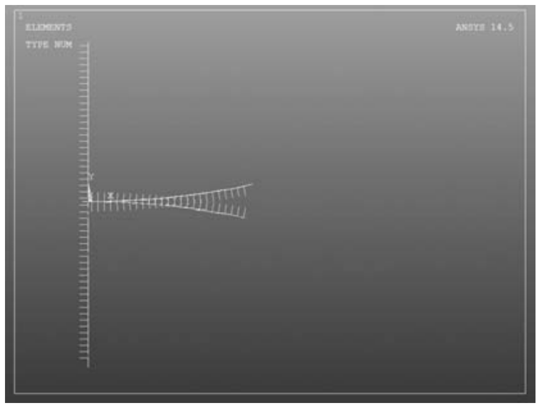
Next >



Create >



Finish



ANSYS graphics show the contact region

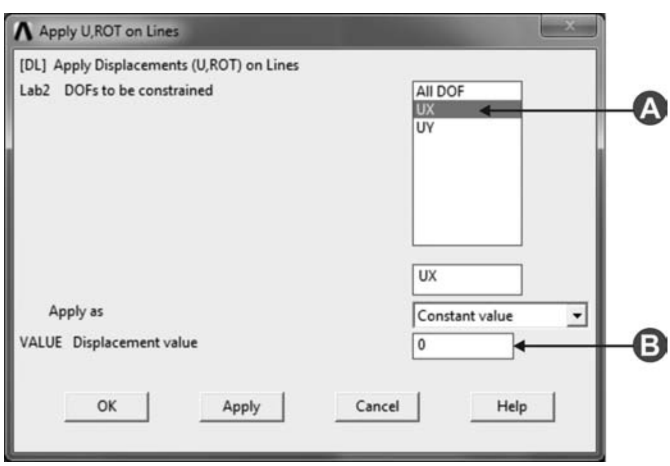
Close the Contact Manager window

Utility Menu > Plot > Lines

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines

In the ANSYS graphics, click on all vertical left lines, where zero x-displacement is applied, and then in Apply U,ROT on Lines, click on

OK



A select UX

B type 0 in Displacement value

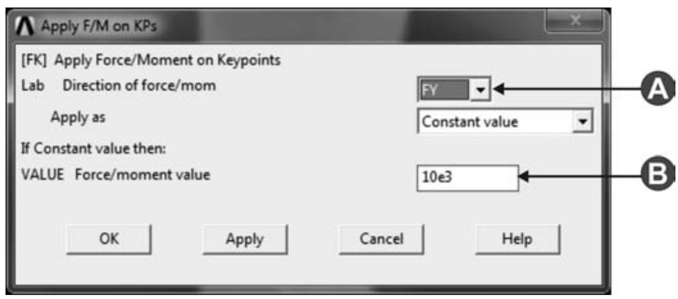
OK

Forces are applied at the top keypoint of the upper circle and bottom keypoint of the lower circle.

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

In the ANSYS graphics, click on the bottom keypoint at the lower circle, where the force is applied, and then in Apply F/M on KPs window, click on

OK



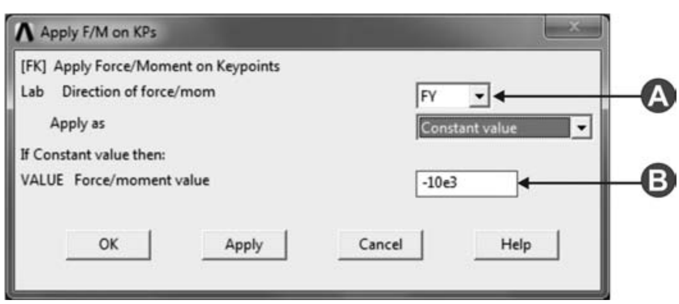
A select FY in Direction of force/mom

B type 10e3 in Force/moment value

Apply

In the ANSYS graphics, click on the top keypoint at the upper circle, where the force is applied, and then in Apply F/M on KPs window, click on

OK



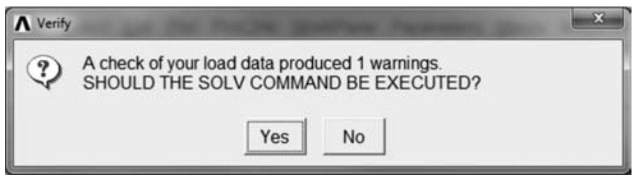
A select FY in Direction of force/mom

B type −10e3 in Force/moment value

OK

Main Menu > Solution > Solve > Current LS

OK

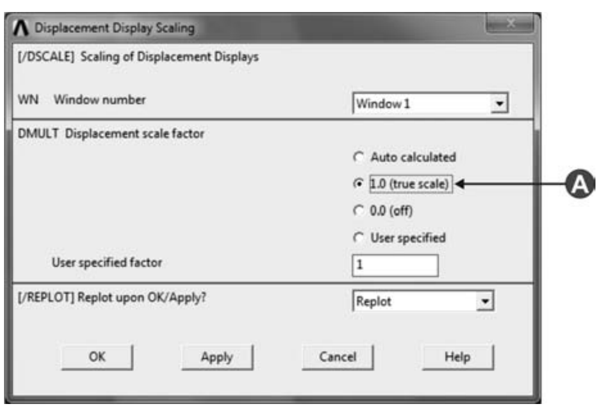


Yes

Close

The above window indicates that the solution task is accomplished successfully. The next step is getting the results. The scale is changed to true scale for actual dimensional display in ANSYS graphics.

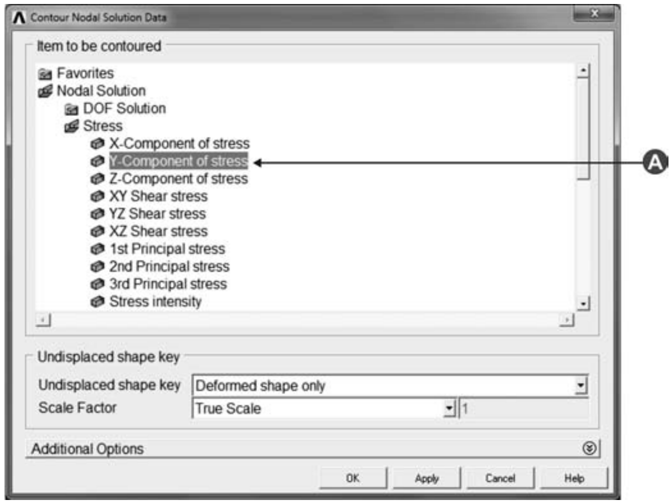
Utility Menu > PlotCtrls > Style > Displacement Scaling



A select 1.0 (true scale)

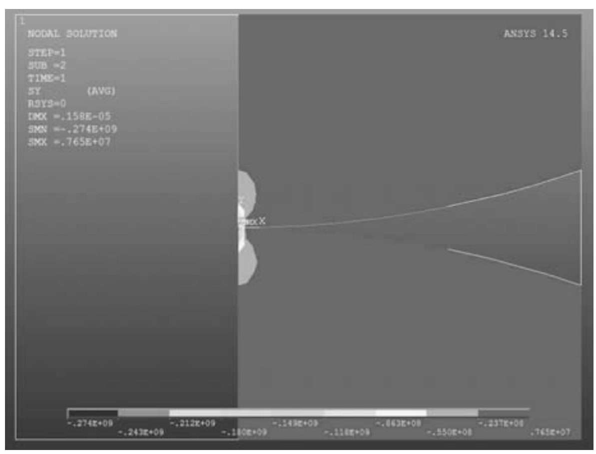
OK

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > Stress > Y-Component of stress

OK



ANSYS graphics show the contours of stress in the y-direction

The above figure indicates that the maximum stress in the y-direction is equal to 0.274×10^9 Pa. The maximum stress in the y-direction can be theoretically obtained using expression (3.140) as follows:

$$P_o = 0.418\sqrt{\frac{FE}{L}\frac{r_1 + r_2}{r_1 r_2}} = 0.418\sqrt{\frac{10\times10^3\left(200\times10^9\right)}{1}\frac{0.01+0.01}{0.01\times0.01}}$$

$$= 0.264\times10^9 \text{ Pa}$$

The theoretical maximum stress is very close to the result obtained by ANSYS with marginal error of 3.78%.

PROBLEM 3.1

The thin plates, as shown in Figure 3.12a and 3.12b, are subjected to force. Determine the nodal displacements at the point where the force is applied, using the finite element method, given E = 180 GPa, v = 0.33, and t = 0.02 m. Consider the plates as plane stress problems, and use only one triangular element.

PROBLEM 3.2

The thin plates, as shown in Figure 3.13a and 3.13b, are subjected to force(s). Determine the nodal displacements, using the finite element method, given E = 240 GPa, v = 0.3, and t = 0.01 m. Use linear triangular elements and the suggested two-element mesh. Consider the plates as plane stress problems.

PROBLEM 3.3

The thin plates, as shown in Figure 3.14a and 3.14b, are subjected to force(s). Determine the nodal displacements, using the finite element method, given E = 195 GPa, v = 0.3, and t = 0.015 m. Use linear triangular elements and the suggested finite element mesh. Consider them as plane stress problems.

PROBLEM 3.4

The square plate with a hole, as shown in Figure 3.15, is subjected to tensile pressure at both vertical sides. Use ANSYS to determine the maximum stress in the x-direction. Also, compare the ANSYS result with maximum stress using the stress concentration factor figure. The applied pressure is 75 kN/m^2, and let E = 230 GPa, v = 0.30, and consider the plate as a plane stress with thickness of t = 0.01 m. Take the advantage of symmetry in the problem to reduce the computational size.

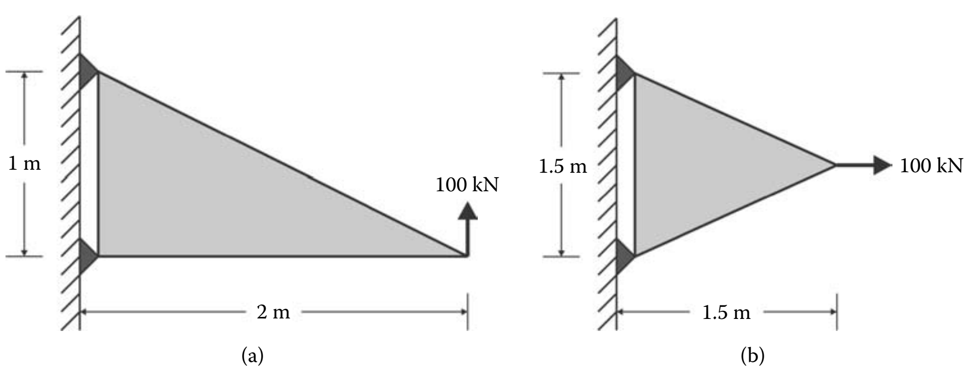


FIGURE 3.12 Thin plate subjected to force.

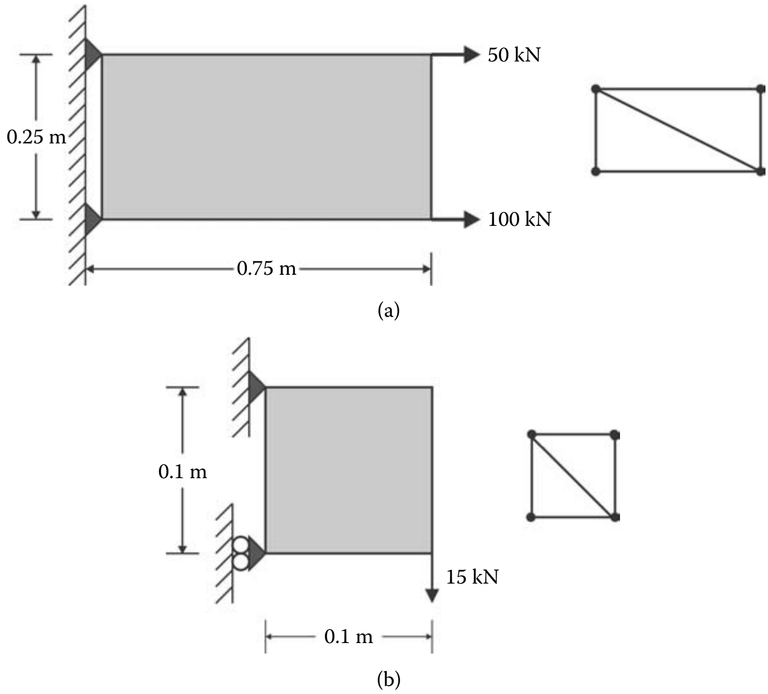


FIGURE 3.13 Thin plate subjected to force.

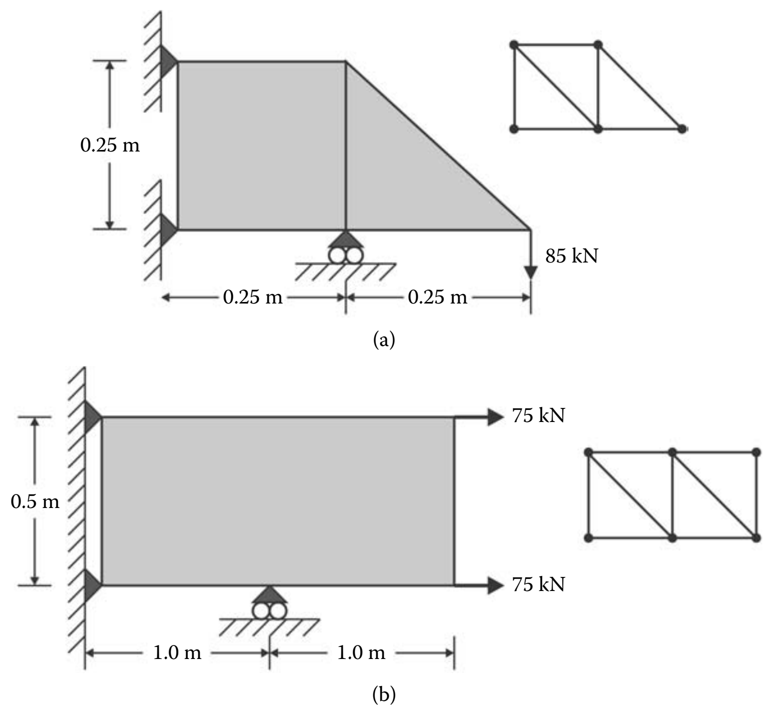


FIGURE 3.14 Thin plate subjected to force.

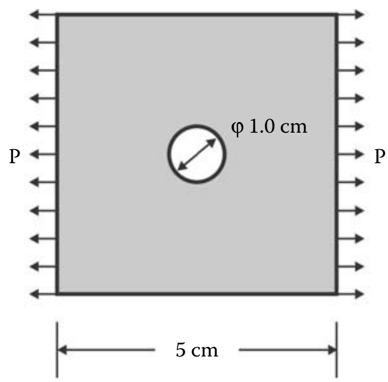


FIGURE 3.15 A plate with a hole subjected to tensile pressure.

PROBLEM 3.5

A notched rectangular plate is in tension, as shown in Figure 3.16. Pressure (P) is applied at the left and right vertical sides of the plate, given applied pressure P = 200 kPa, E = 180 GPa, v = 0.3, and t = 0.05 m, where t is the thickness of the plate. Determine maximum displacement in the x-direction and calculate the stress concentration factor (K) using the ANSYS result. Take advantage of symmetry in the problem to reduce the computational size.

PROBLEM 3.6

A flat plate is axially loaded, as shown in Figure 3.17. Pressure P = 150 kPa is applied at the left and right vertical sides of the plate. E = 210 GPa, v = 0.33, and t = 0.01 m, where t is the thickness of the plate. Determine using ANSYS the maximum displacement in the x-direction and the stress concentration factor (K). Take the advantage of symmetry in the problem to reduce the computational size.

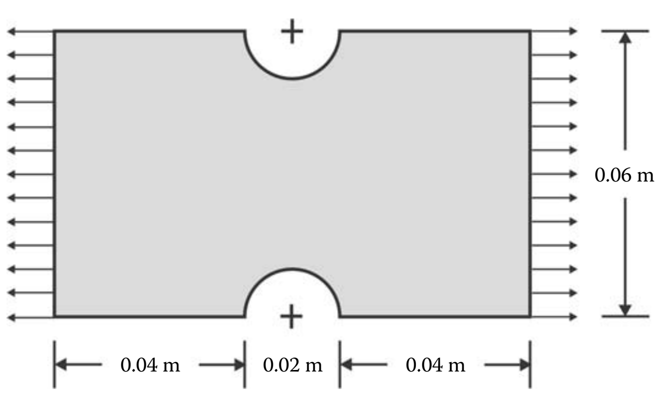


FIGURE 3.16 A notched rectangular plate subjected to tensile pressure.

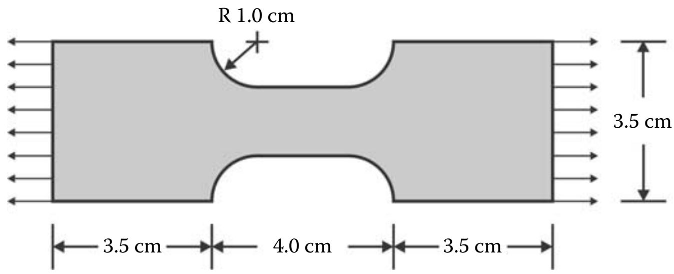


FIGURE 3.17 A plate subjected to tensile pressure.

PROBLEM 3.7

Determine the maximum displacement in the radial direction of the vessel shown in Figure 3.18. Take the advantage of symmetry in the problem to reduce the computational size. The inner and outer radii are 0.15 and 0.1625 m, respectively, and the height of the vessel is 0.95 m. Let E = 230 GPa and v = 0.3. Pressure of P = 1.5 MPa is applied at the inner surface of the vessel. The upper and lower horizontal surfaces are fixed.

PROBLEM 3.8

The square plate with a hole shown in Figure 3.15 is subjected to transient tensile pressure at both vertical sides. Use ANSYS to determine the maximum stress in the x-direction as a function of time, and let E = 230 GPa, v = 0.30, and consider the plate as a plane stress with

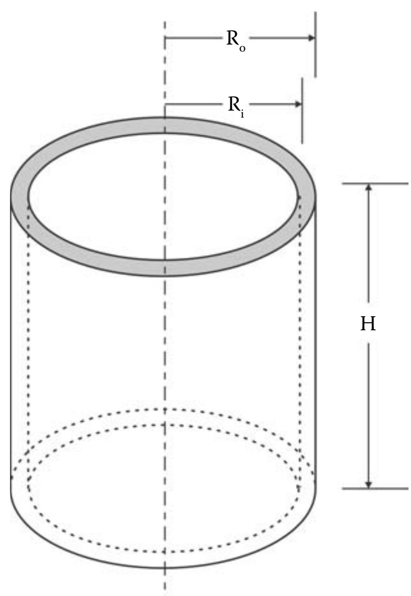


FIGURE 3.18 A vessel subjected to pressure at the inner surface.

thickness of t = 0.01 m. Take the advantage of symmetry in the problem to reduce the computational size. The total time duration for the process is 500 seconds. The applied pressure (in N/m^2) is a function of time according to the following expression:

$$P(t) = 75\times10^3 + 75\times10^3 \sin\left(\frac{2\pi}{50}t\right)$$

PROBLEM 3.9

The square plate geometry with a hole shown in Figure 3.16 is subjected to transient tensile pressure at both vertical sides. Use ANSYS to determine the maximum stress in the x-direction as a function of time, and let E = 180 GPa, v = 0.33 and consider the plate as a plane stress with thickness of t = 0.02 m. Take the advantage of symmetry in the problem to reduce the computational size. The total time duration for the process is 2500 seconds. The applied pressure (in N/m^2) is a function of time according to the following expression:

$$P(t) = 125\times10^3 + 125\times10^3 \sin\left(\frac{2\pi}{500}t\right)$$

PROBLEM 3.10

A horizontal cylinder is placed on a flat surface, as shown in Figure 3.19. The cylinder has a radius of r = 0.025 m, and a force of 25 kN is applied to the cylinder. The cylinder will compress and penetrate the flat surface. Determine the maximum stress at the contact region, given that E = 180 GPa and v = 0.3 for the cylinder and flat plate.

PROBLEM 3.11

Two horizontal cylinders are placed close to each other, as shown in Figure 3.20. The two cylinders have different radii, r_1 = 0.01 m and

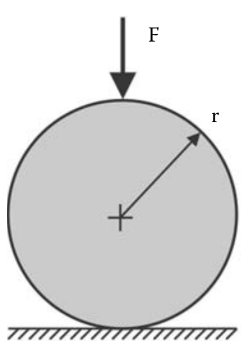


FIGURE 3.19 A horizontal cylinder on a flat plate.

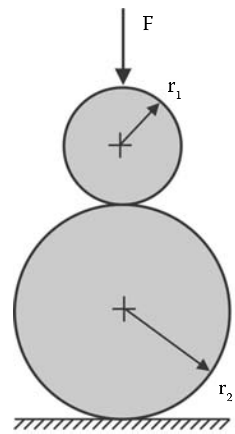


FIGURE 3.20 A contact element problem.

$r_2 = 0.02$ m, and force of 50 kN is applied only to the upper cylinder, while the lower cylinder is placed on a flat surface. As a result, the upper cylinder will move downward and meet the lower cylinder. Determine the maximum stress at the contact region. Given that E = 180 GPa and v = 0.33 for both cylinders.

CHAPTER FOUR

Vibration

4.1 Vibration analysis

In this chapter, the basic formulation of the motion of a single degree of freedom is presented. The aim of this introduction is to discuss some important concepts necessary for solving and understating vibration problems. Figure 4.1 shows a mass–spring system that is subjected to a time-dependent force F(t), where m is the mass of the system and k is the stiffness of the spring.

Applying Newton's second law, the equation of motion of the spring–mass element in the x-direction is expressed as

$$m\ddot{x} + kx = F(t) \tag{4.1}$$

Equation 4.1 can also be symbolically expressed as

$$[m]\{\ddot{x}\} + [k]\{x\} = \{F\} \tag{4.2}$$

where [k] is the stiffness matrix, [m] is mass matrix, {x} is the nodal displacement vector, and $\{\ddot{x}\}$ is the acceleration vector. If the applied force F(t) is equal to zero, then Equation 4.1 becomes a homogenous ordinary differential equation. The solution of this equation gives the natural frequency of the oscillating mass:

$$\omega^2 = \frac{k}{m}, \tag{4.3}$$

where ω is called the *natural frequency* of the free vibration of the mass. The natural frequency depends on the stiffness of the spring and the mass of the system. Substituting (4.3) into the equation of motion of free vibration (4.1) yields

$$\ddot{x} + \omega^2 x = 0 \tag{4.4}$$

The solution of Equation 4.4 is

$$x(t) = A\cos(\omega t) + B\sin(\omega t) \tag{4.5}$$

where the constants A and B can be determined using the initial conditions.

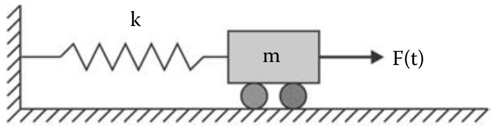


FIGURE 4.1 Spring–mass system subjected to a time-dependent force.

4.2 Development of stiffness equations for axial vibration of a bar element

Lumped mass formulation is one of the common ways to formulate the stiffness matrix for dynamics analysis. Figure 4.2 shows a bar with initial length L, modulus of elasticity E, cross-sectional area A, and density ρ. To account for the mass inertia of the bar, the mass is distributed and concentrated in the nodes, as shown in Figure 4.2.

The inertia forces are added to the nodal forces for the bar element equation, which is used in static analyses, as follows:

$$\begin{Bmatrix} F_1 - \dfrac{\rho AL}{2}\ddot{u}_1 \\ F_2 - \dfrac{\rho AL}{2}\ddot{u}_2 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} \tag{4.6}$$

The negative values of the inertial forces indicate that they are resisting the elastic forces F_1 and F_2. Equation 4.6 is rearranged as follows:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} + \frac{\rho AL}{2}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\begin{Bmatrix} \ddot{d}_1 \\ \ddot{d}_2 \end{Bmatrix} \tag{4.7}$$

or symbolically,

$$\{F\} = [k]\{d\} + [m]\{\ddot{d}\} \tag{4.8}$$

For free vibration, the nodal force vector is equal to zero. For a harmonic motion, it is assumed that the displacements vary harmonically with respect to time with frequency ω:

$$\{d\} = \{U\}\sin(\omega t) \tag{4.9}$$

The second derivative of the displacement vector is the acceleration:

$$\{\ddot{d}\} = -\omega^2\{U\}\sin(\omega t) \tag{4.10}$$

Expression (4.9) into (4.10) yields

$$\{\ddot{d}\} = -\omega^2\{d\} \tag{4.11}$$

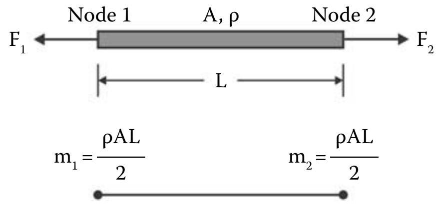


FIGURE 4.2 Bar element and the concentrated mass at the nodes.

Substituting the acceleration vector in Equation 4.7 yields

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} - \omega^2 \frac{\rho AL}{2}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} \quad (4.12)$$

Equation 4.12 can be rearranged as

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \left(\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2 \frac{\rho AL}{2}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\right)\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} \quad (4.13)$$

The global stiffness matrix can be obtained by assembling the stiffness matrix for each element as follows:

$$[K] = \sum_{e=1}^{N}\left[K^{(e)}\right] \quad (4.14)$$

The nodal forces are also assembled to form a global force vector:

$$[F] = \sum_{e=1}^{N}\left[F^{(e)}\right] \quad (4.15)$$

Similarly for the displacement vector,

$$[d] = \sum_{e=1}^{N}\left[d^{(e)}\right] \quad (4.16)$$

Finally, the nodal forces and the displacement relationship is expressed as

$$\{F\} = [K]\{d\} \quad (4.17)$$

4.3 Natural frequencies of axial vibration of a bar element

Determine the first two natural frequencies of a cantilever bar, as shown in Figure 4.3. The total length of the bar is 2 m. Model the bar with two elements only, and let $E = 200$ GPa, $A = 2 \times 10^{-3}$ m^2, and $\rho = 5000$ kg/m^3.

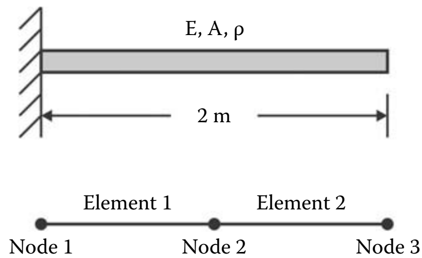


FIGURE 4.3 Cantilever bar and finite element mesh.

Element 1 has nodes 1 and 2, and its length is L = 1.0 m. Using Equation 4.13 for nodal forces and displacement relationship,

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \left[\frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2 \frac{\rho AL}{2} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \right] \begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix}$$

Nodal displacement at node 1 and nodal force at node 2 are equal to zero. Element 2 has nodes 2 and 3, and its length is L = 1 m. Using Equation 4.13,

$$\begin{Bmatrix} F_2 \\ F_3 \end{Bmatrix} = \left(\frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2 \frac{\rho AL}{2} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \right) \begin{Bmatrix} d_2 \\ d_3 \end{Bmatrix}$$

Nodal forces at node 3 are equal to zero. Assembling the nodal and displacement vectors and stiffness matrix for both elements yields

$$\begin{Bmatrix} F_1 \\ F_2 = 0 \\ F_3 = 0 \end{Bmatrix} = \left(\frac{EA}{L} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} - \omega^2 \frac{\rho AL}{2} \begin{bmatrix} 1 & 0 & 0 \\ 0 & 2 & 0 \\ 0 & 0 & 1 \end{bmatrix} \right) \begin{Bmatrix} d_1 = 0 \\ d_2 \\ d_3 \end{Bmatrix}$$

Since the first column is multiplied by zero displacement, it can be deleted. The first row can also be deleted, and then

$$\begin{Bmatrix} 0 \\ 0 \end{Bmatrix} = \left(\frac{EA}{L} \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2 \frac{\rho AL}{2} \begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix} \right) \begin{Bmatrix} d_2 \\ d_3 \end{Bmatrix}$$

The above equations are simplified as

$$\begin{Bmatrix} 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} - \lambda \begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix}$$

where λ is the eigenvalue,

$$\lambda = \omega^2 \frac{\rho L^2}{2E}$$

then,

$$\begin{Bmatrix} 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} 2-2\lambda & -1 \\ -1 & 1-\lambda \end{bmatrix}$$

The determinant of the above matrix yields the following equation:

$$2\lambda^2 - 4\lambda + 1 = 0$$

And the solution for the above equation is $\lambda_1 = 0.29289$ and $\lambda_2 = 1.70711$. The natural frequencies are calculated as

$$\omega_1 = \sqrt{\frac{2\lambda_1 E}{\rho L^2}} = 4840.58 \text{ Hz}$$

$$\omega_2 = \sqrt{\frac{2\lambda_2 E}{\rho L^2}} = 11686.23 \text{ Hz}$$

The exact solution for the present problem is

$$\omega = \frac{n\pi}{2L}\sqrt{\frac{E}{\rho}}$$

where n is the mode number. The exact solutions for the first and the second frequencies are ω_1 = 4967.15 Hz and ω_2 = 9934.3 Hz, respectively. The exact results are close to the finite element solution with error in the first and the second frequencies of 2.55% and 17.64%, respectively. The error can be reduced if the number of elements is increased.

4.4 Development of stiffness equations for flexural vibration of a beam element

Figure 4.4 shows a horizontal beam element. The variation of curvature of the beam is assumed to be a polynomial of the third order, and the constants in the polynomial are determined using the boundary conditions. There are four degrees of freedom for the element, two vertical displacements and two rotations. The displacement function is expressed as

$$d_y(x) = a_1x^3 + a_2x^2 + a_3x + a_4 \tag{4.18}$$

Using expression (2.68), the rotation is expressed as

$$\theta(x) = 3a_1x^2 + 2a_2x + a_3 \tag{4.19}$$

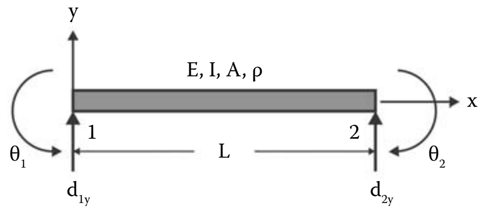


FIGURE 4.4 Horizontal beam element.

The constant a's are obtained by using the boundary conditions at the nodes as follows:

$$d_y(x = 0) = d_{1y} \tag{4.20}$$

$$d_y(x = L) = d_{2y} \tag{4.21}$$

$$\theta(x = 0) = \frac{\partial v}{\partial x} = \theta_1 \tag{4.22}$$

$$\theta(x = L) = \frac{\partial v}{\partial x} = \theta_2 \tag{4.23}$$

Therefore, the displacement function (4.18) is expressed as

$$\begin{aligned} d_y(x) = d_{1y} + \theta_1 x - \frac{3x^2}{L^2} d_{1y} - \frac{2x^2}{L}\theta_1 + \frac{3x^2}{L^2} d_{2y} - \frac{x^2}{L}\theta_2 \\ + \frac{2x^2}{L^3} d_{1y} + \frac{x^3}{L^2}\theta_1 - \frac{2x^2}{L^3} d_{2y} + \frac{x^3}{L^2}\theta_2 \end{aligned} \tag{4.24}$$

Rearranging the displacement function (4.24) gives

$$d(x) = f_1(x)d_1 + f_2(x)\theta_2 + f_3(x)d_2 + f_4(x)\theta_2 \tag{4.25}$$

where $f_1(x)$, $f_2(x)$, $f_3(x)$, and $f_4(x)$ are shape functions, and they are expressed as

$$f_1(x) = 1 - 3\left(\frac{x}{L}\right)^2 + 2\left(\frac{x}{L}\right)^3 \tag{4.26}$$

$$f_2(x) = 1 - 3\left(\frac{x^2}{L}\right) + 2\left(\frac{x^3}{L^2}\right) \tag{4.27}$$

$$f_3(x) = 3\left(\frac{x}{L}\right)^2 - 2\left(\frac{x}{L}\right)^3 \tag{4.28}$$

$$f_4(x) = -\left(\frac{x^2}{L}\right) + \left(\frac{x^3}{L^2}\right) \tag{4.29}$$

The kinetic and strain energies are, respectively,

$$T = \frac{\rho A}{2}\int_0^L \left(\frac{\partial v}{\partial x}\right)^2 dx \tag{4.30}$$

$$U = \frac{EI}{2}\int_0^L \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx \tag{4.31}$$

Using the Lagrange equation,

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \theta}\right) + \frac{\partial T}{\partial \theta} = F \tag{4.32}$$

Substituting kinetic and strain energies into the Lagrange equation gives

$$[m]\{\ddot{d}\} + [k]\{d\} = \{F(t)\} \tag{4.33}$$

In the case of free vibration, and with the expression $\{\ddot{d}\} = -\omega^2\{d\}$, the Lagrange equation (4.33) becomes

$$-\omega^2[m]\{d\} + [k]\{d\} = \{0\} \tag{4.34}$$

Expanding Equation 4.34 yields

$$-\omega^2 \frac{\rho AL}{420}\begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix}\begin{Bmatrix} d_{1y} \\ \theta_1 \\ d_{2y} \\ \theta_2 \end{Bmatrix}$$

$$+\frac{EI}{L}\begin{bmatrix} 12/L^2 & 6/L & -12/L^2 & 6/L \\ 6/L & 4 & -6/L & 2 \\ -12/L^2 & -6/L & -12/L^2 & -6/L \\ 6/L & 2 & -6/L & 4 \end{bmatrix}\begin{Bmatrix} d_{1y} \\ \theta_1 \\ d_{2y} \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \tag{4.35}$$

4.5 Natural frequencies of the flexural vibration of a beam element

Determine the first natural frequency of the beam shown in Figure 4.5a. The beam is 2 m in length and fixed at both ends. The beam has a square cross section with height and width of 0.05 m, ρ = 5000 kg/m^3, and E = 200 GPa.

The minimum number of nodes required to determine the natural frequency is 3, because two of the nodes are fixed. In this example,

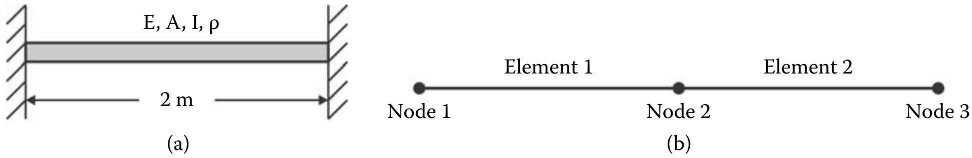


FIGURE 4.5 (a) Horizontal beam with fixed ends and (b) finite element mesh.

the beam is divided into two elements only. The finite element mesh for the horizontal beam is shown in Figure 4.5b. In the next ANSYS example, the beam is meshed with 50 elements. Element 1 has a length of L = 1 m, nodes 1 and 2, and its stiffness and mass matrices are, respectively,

$$[m_1] = \frac{\rho AL}{420}\begin{bmatrix} 156 & -22L \\ -22L & 4L^2 \end{bmatrix}$$

$$\left[K^{(1)}\right] = \frac{EI}{L}\begin{bmatrix} 12/L^2 & -6/L \\ -6/L & 4 \end{bmatrix}$$

Element 2 has a length of L = 1 m, nodes 2 and 3, and its stiffness and mass matrices are, respectively,

$$[m_2] = \frac{\rho AL}{420}\begin{bmatrix} 156 & 22L \\ 22L & 4L^2 \end{bmatrix}$$

$$\left[K^{(2)}\right] = \frac{EI}{L}\begin{bmatrix} 12/L^2 & 6/L \\ 6/L & 4 \end{bmatrix}$$

Applying Equation 4.34:

$$-\omega^2\frac{\rho AL}{420}\begin{bmatrix} 312 & 0 \\ 0 & 8L^2 \end{bmatrix}\begin{Bmatrix} d_{2y} \\ \theta_2 \end{Bmatrix} + \frac{EI}{L}\begin{bmatrix} 24/L^2 & 0 \\ 0 & 8 \end{bmatrix}\begin{Bmatrix} d_{2y} \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

Simplifying the above equation yields

$$-\omega^2\frac{\rho AL}{420}\begin{bmatrix} 312 & 0 \\ 0 & 8L^2 \end{bmatrix} + \frac{EI}{L}\begin{bmatrix} 24/L^2 & 0 \\ 0 & 8 \end{bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

Solving for the first natural frequency,

$$\omega_1 = \frac{5.68}{(L)^2}\left(\frac{EI}{\rho A}\right)^{1/2}$$

The exact solution for the first natural frequency is given by

$$\omega_1 = \frac{5.59}{(L)^2}\left(\frac{EI}{\rho A}\right)^{1/2}$$

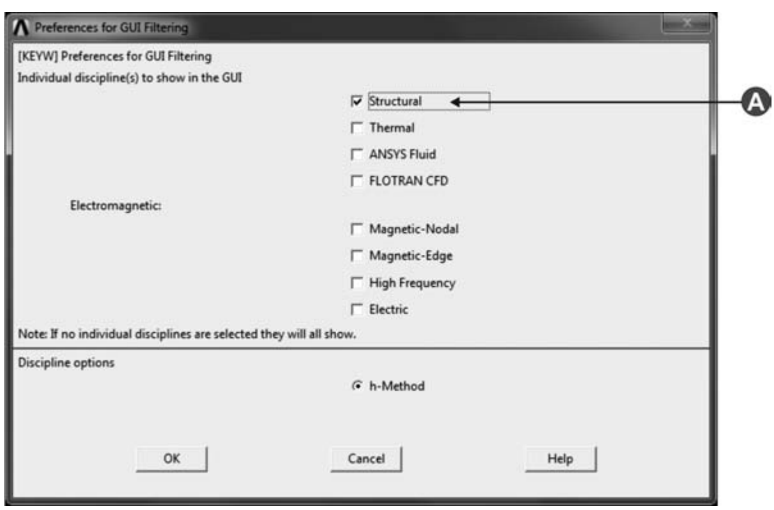
which is very close to the finite element result.

4.6 Natural frequencies of the flexural vibration of beam element using ANSYS

For a horizontal beam shown in Figure 4.5a, determine the first five natural vibration frequencies. Also, create animation for the third mode. Mesh the beam with 50 elements. The beam has a square cross section with height and width of 0.05 m, ρ = 5000 kg/m^3, and E = 200 GPa.

Double click on the Mechanical APDL Product Launcher icon

Main Menu > Preferences

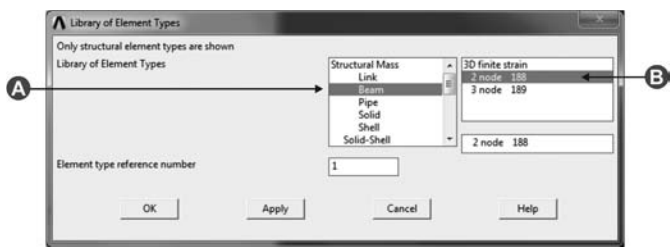


A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add...



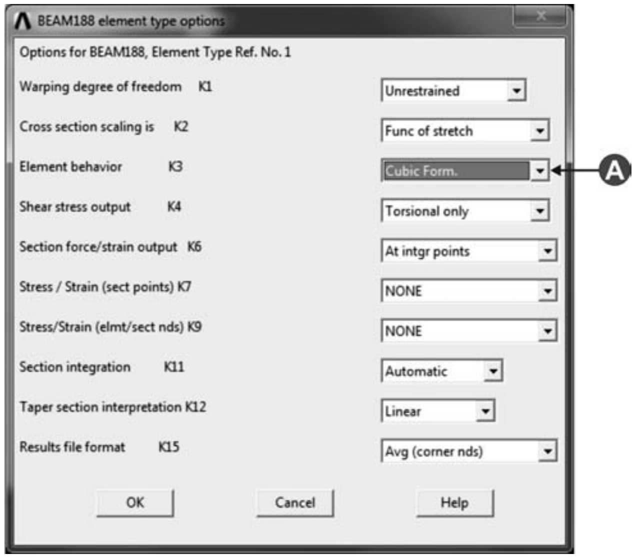
A select Beam

B select 2 node 188

OK

Options…

The illustrated theory for the beam in this chapter is based on the third-order polynomial for displacement function. Therefore, in the option, the element behavior should be changed to cubic form.



A select Cubic Form. in Element behavior K3

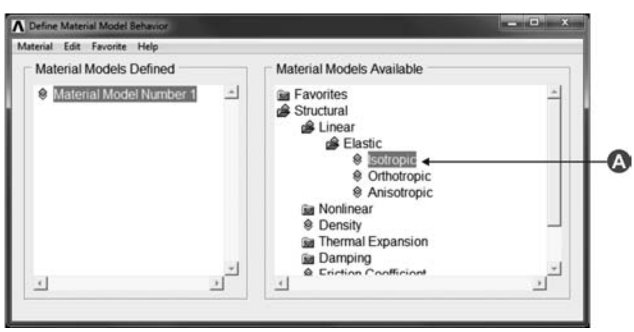
OK

Close the Element Type window

Close

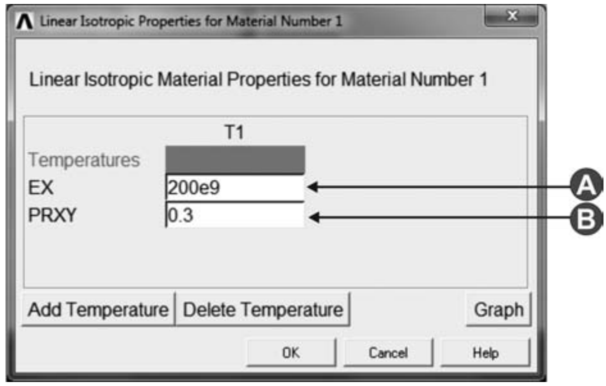
The beam cross section is specified in sections. For material properties, the modulus of elasticity and density are required.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

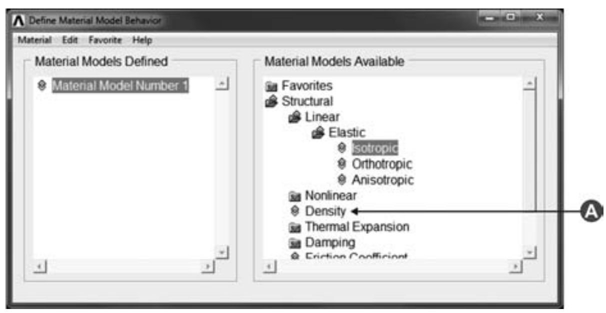
The following window will appear, the Young modulus for the beam is specified, and any value for the Poisson ratio is required to avoid an error message. Also, the density of the beam is required.



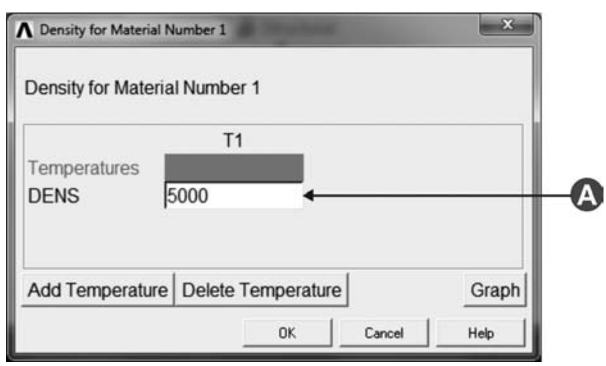
A type 200e9 in EX

B type 0.3 in PRXY

OK



A click on Structural > Density



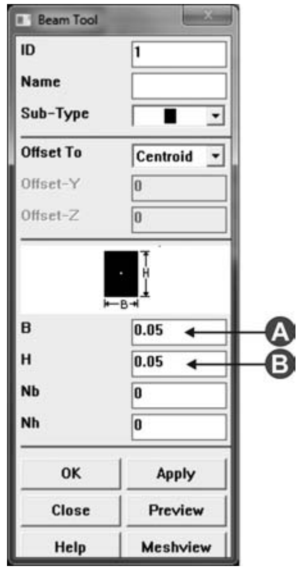
A type 5000 in DENS

OK

Close the Define Material Model Behavior window

The cross section of the beam is a square. In the cross section, the square section type is the default section. The width and height of the beam's cross section should be specified.

Main Menu > Preprocessor > Sections > Beam > Common Sections



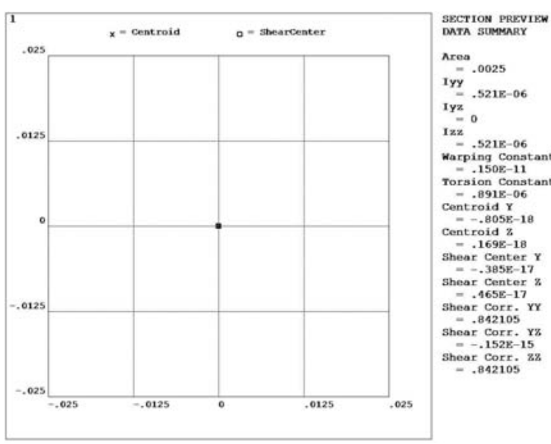
A type 0.05 in B

B type 0.05 in H

Apply

Review

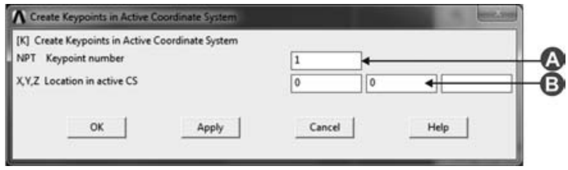
OK



ANSYS graphics will show the geometrical properties of the beam cross section

Modeling the beam structure takes place in the following steps. First, two keypoints are created. Then, a line that connects the keypoints is created. The x- and y-coordinates of each keypoint are specified for ANSYS. Then, (0,0) is the coordinate for Keypoint 1, and (0,2) is the coordinate for Keypoint 2.

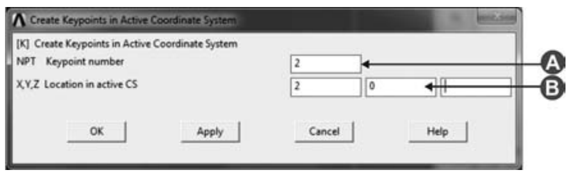
Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS



A type 1 in Keypoint number

B type 0 and 0 in X,Y,Z Location in active CS

Apply



A type 2 in Keypoint number

B type 2 and 0 in X,Y,Z Location in active CS

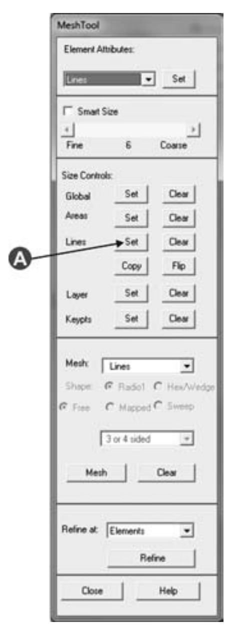
OK

Main Menu > Modeling > Create > Lines > Lines > Straight Line

Click on Keypoints 1 and 2. In Create Straight Line window, click on

OK

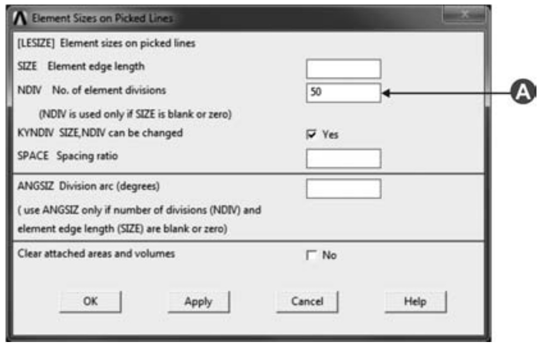
Main Menu > Preprocessor > Meshing > Mesh Tool



A click Set in Lines
In Element Sizes on Picked Lines window, click on

Pick All

The following window will show up. In Element Sizes on Picked Lines window, there are two options, either specifying the length of the elements or the number of element divisions. For the present example, the beam is divided into 50 elements.



A type 50 in No. of element divisions

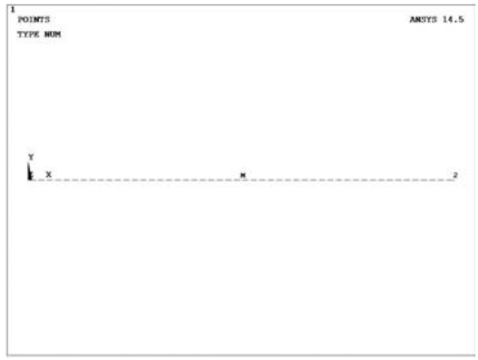
OK

A click on Mesh

In Mesh Lines window, click on

Pick All

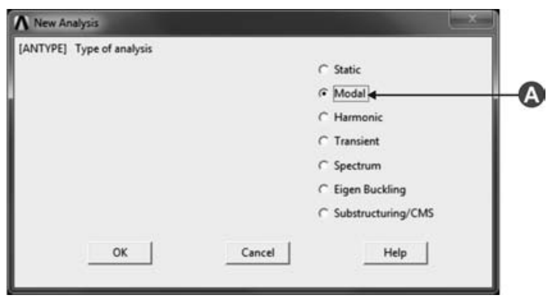
Utility Menu > Plot > Nodes



ANSYS graphics show the nodes

The analysis type is changed from Static to Modal in New Analysis. The required number of free vibration modes is five, and this is requested in analysis options.

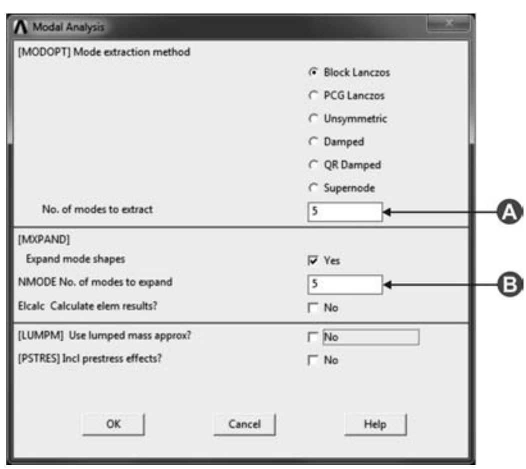
Main Menu > Solution > Analysis Type > New Analysis



A select Modal

OK

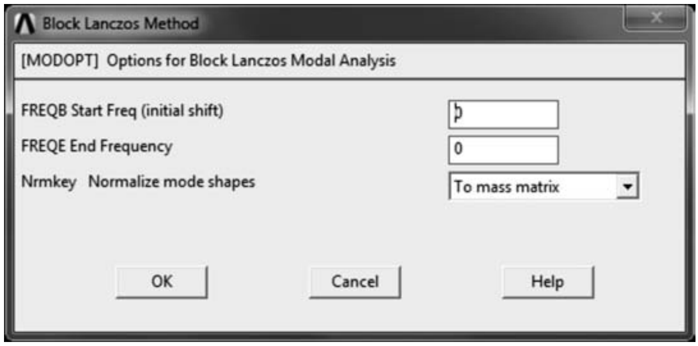
Main Menu > Solution > Analysis Type > Analysis Options



A type 5 in No. of modes to extract

B type 5 in NIMODE No. of modes to expand

OK



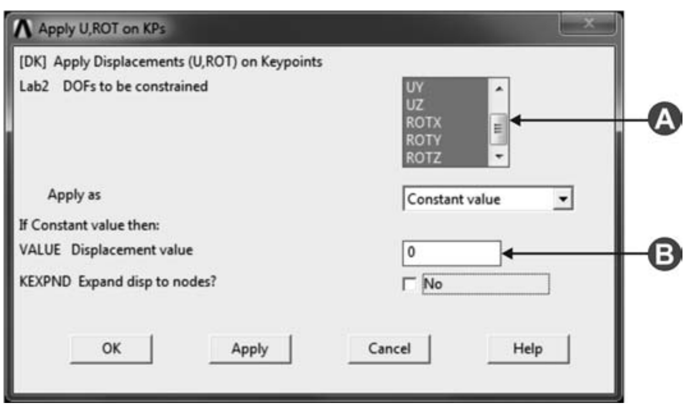
OK

Boundary conditions are specified in solution. Both ends of the beam are fixed and no forces are applied because it is a free vibration problem. The displacement is applied on the keypoints.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Click on Keypoints 1 and 2. Then, in Apply U,ROT on KPs window, click on

OK



A select UX, UY, UZ, ROTX, ROTY, and ROTZ

B type 0 in Displacement value

OK

The final step is to run the ANSYS solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem.

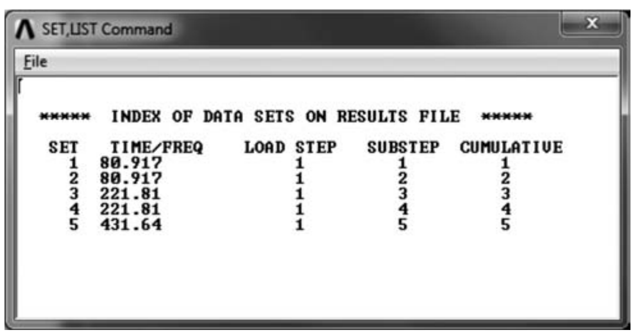
Main Menu > Solution > Solve > Current LS

OK

Close

In the following steps, the natural frequencies of the beam are listed. Notice that the frequencies 1 and 2 are the same, because the beam is vibrating similarly in the y- and the z-directions.

Main Menu > General Postproc > Results Summary


SET,LIST Command

File

***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	80.917	1	1	1
2	80.917	1	2	2
3	221.81	1	3	3
4	221.81	1	4	4
5	431.64	1	5	5

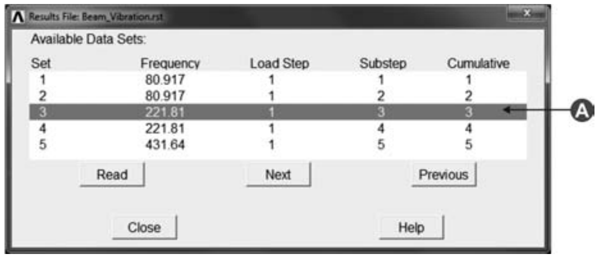
Close

The exact solution for the first natural frequency for this geometry is given by

$$\omega_1 = \frac{5.59}{(L/2)^2}\left(\frac{EI}{\rho A}\right)^{1/2}$$

where L is the beam length. The exact solution for the present geometry is ω_1 = 510.377 rad/s or 81.23 Hz, which is very close to the ANSYS results. Next, the deformation shape of the third mode is displayed.

Main Menu > General Postproc > Read Results > By Pick

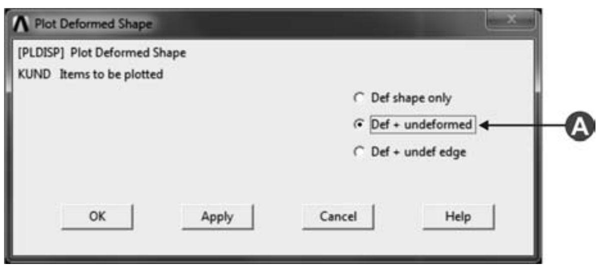


A select Set 3

Read

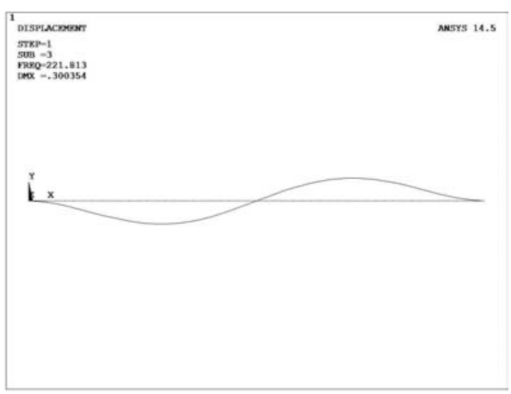
Close

Main Menu > General Postproc > Plot Results > Deformed Shape



A select Def + undeformed

OK

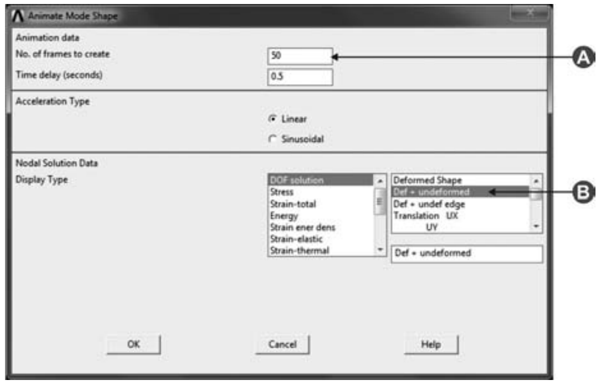


ANSYS graphics show the beam before and after deformation

The animation of the third vibration mode is displayed in the following steps. An AVI file is created and stored in the working directory. The number of frames is the number of screen shoots for the animation file. The 50-frame file is specified for this example and a higher value

will enhance the resolution of the animation, but the size of the file will be increased. The time delay is the time between each screen shot, and increasing the time delay will increase the duration of the animation.

Utility Menu > PlotCtrls > Animate > Mode Shape



A type 50 in No. of frames to create

B select Def + undeformed

OK

Animation will be shown on the screen for the third mode.

4.7 Development of stiffness equations for vibration of an oriented beam element

Figure 4.6 shows an axial flexural frame element oriented at an angle θ in the xy-plane. There are three degrees of freedom at each node, displacement in x-direction, displacement in y-direction, and rotation.

For a static beam, the stiffness equation is expressed as

$$\{F\} = [T]^T[K][T]\{d\} \tag{4.36}$$

The stiffness equation is extended for vibration by adding the $\omega^2[m]$ matrix to the stiffness matrix as follows:

$$\{F\} = [T]^T\left([K] - \omega^2[m]\right)[T]\{d\} \tag{4.37}$$

where

$$\{F\} = \begin{Bmatrix} F_{1x} \\ F_{1y} \\ M_1 \\ F_{2x} \\ F_{2y} \\ M_2 \end{Bmatrix} \tag{4.38}$$

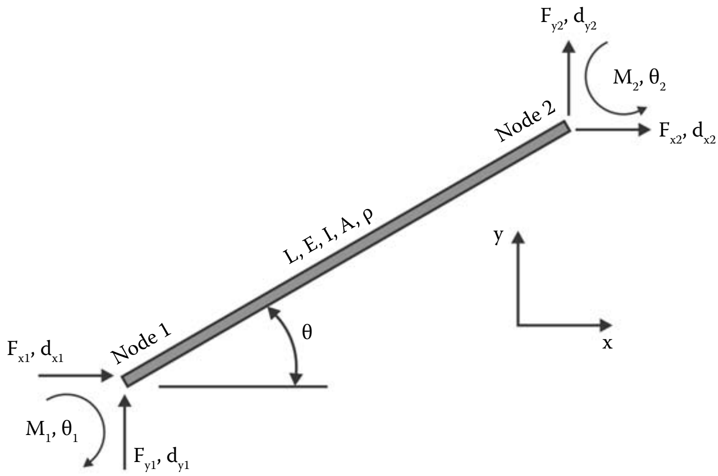


FIGURE 4.6 An axial flexural frame element.

$$[T] = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & \sin\theta & 0 \\ 0 & 0 & 0 & -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{4.39}$$

$$[K] = \frac{EI}{L}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 12/L^2 & 6/L & 0 & -12/L^2 & 6/L \\ 0 & 6/L & 4 & 0 & -6/L & 2 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -12/L^2 & -6/L & 0 & 12/L^2 & -6/L \\ 0 & 6/L & 2 & 0 & -6/L & 4 \end{bmatrix} \tag{4.40}$$

and

$$[m] = \frac{\rho AL}{420}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix} \tag{4.41}$$

The equation of motion for the two-dimensional flexural beam element is

$$\{F\} = \left([K] - \omega^2 [m]\right)\{d\} \tag{4.42}$$

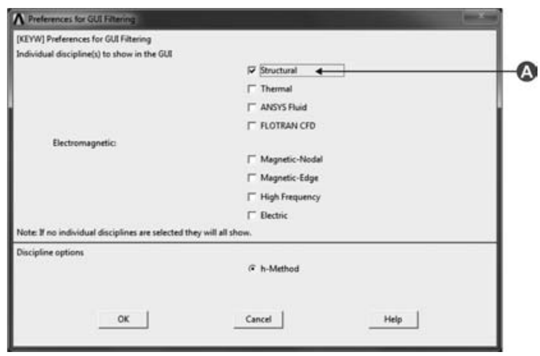
4.8 Harmonic vibration of a plate with holes using ANSYS

The rectangular plate with six holes, as shown in Figure 4.7a, is subjected to forcing function. The forcing function is shown in Figure 4.7b. Create a graph showing the relationship between the displacement at the point where the force is applied and the load's frequency. Let t = 0.001 m, E = 210 GPa, v = 0.25, and ρ = 5000 kg/m³. The frequency of the load is varied between 1 and 2000 Hz, and the number of subsets is 100.

Double click on the Mechanical APDL Product Launcher icon

This example is limited to vibration analysis. Hence, select Structural. The geometry is meshed with quadratic four-node elements.

Main Menu > Preferences



A select Structural

OK

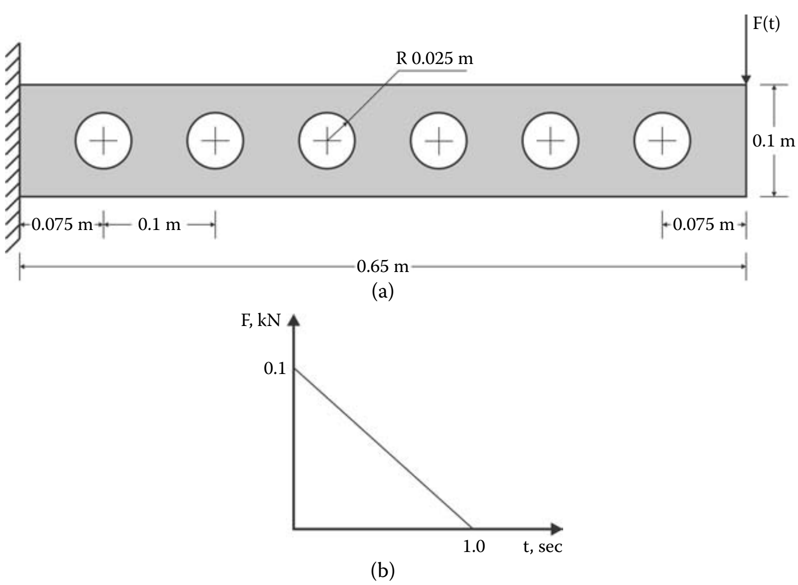
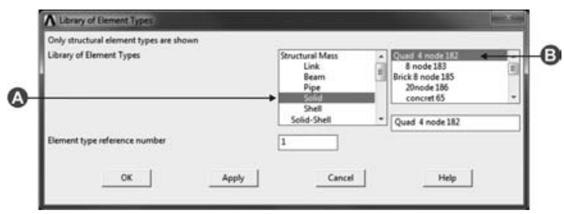


FIGURE 4.7 (a) A plate with six holes and (b) forcing function.

Material properties of the structure are defined in the following steps. The modulus of elasticity and density are required to solve this problem. In addition, the plate has a thickness, and the thickness is specified in real constants.

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

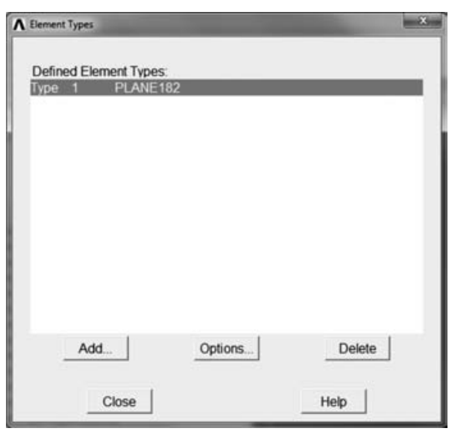
Add...



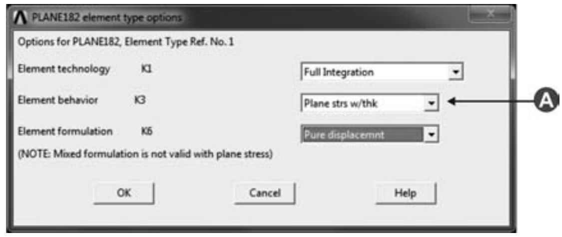
A select Solid

B select Quad 4 node 182

OK



Options…



A select Plane strs w/thk in Element behavior K3

OK

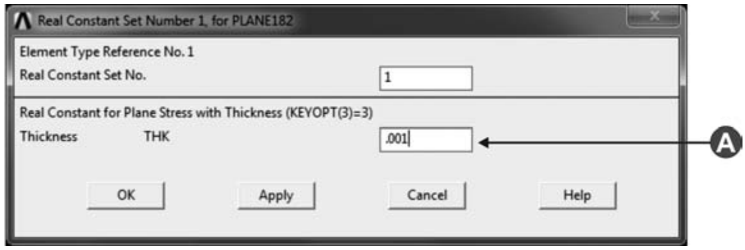
Close the Element Type window

Close

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

Add...

OK



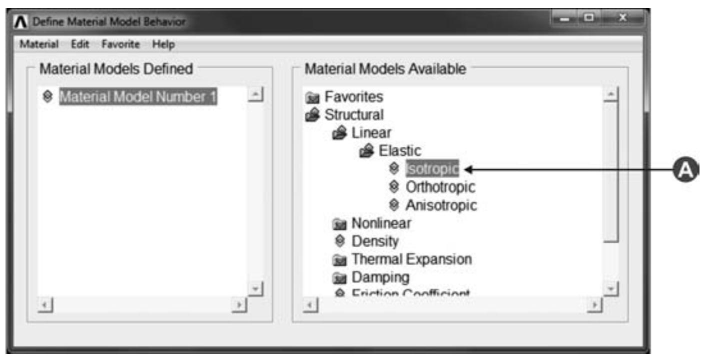
A type 0.001 in Thickness THK

OK

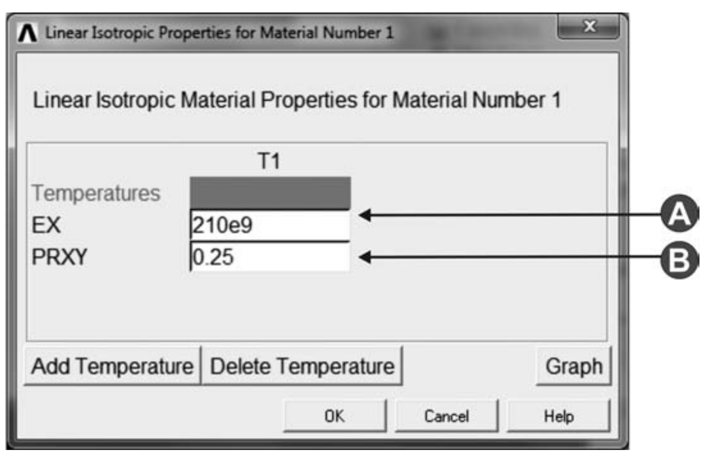
Close the Real Constants window

Close

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

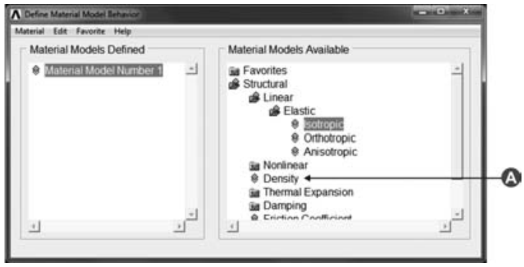


A type 210e9 in EX

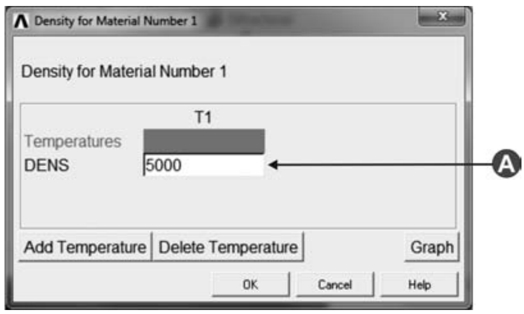
B type 0.25 in PRXY

OK

Main Menu > Material Props > Material Models



A click on Structural > Density



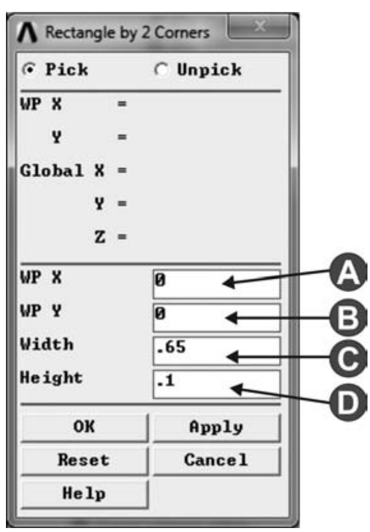
A type 5000 in DENS

OK

Close the Define Material Model Behavior window

The geometry is modeled by creating a rectangle and circles. Boolean operation is utilized to subtract the circles from the rectangle. Alternatively, the circles can be overlapped and then deleted.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

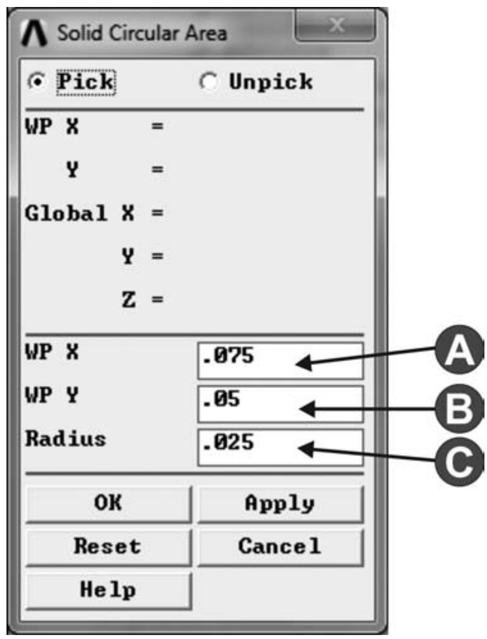
B type 0 in WP Y

C type .65 in Width

D type .1 in Height

OK

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle



A type .075 in WP X

B type .05 in WP Y

C type .025 in Radius

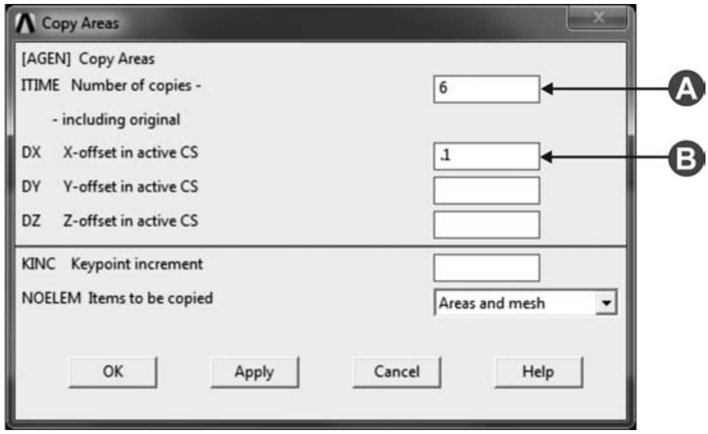
OK

The other circles are created using the copy area in the modeling. The number of circles including the original circle is six, and the distance between the circles is 0.1 m.

Main Menu > Preprocessor > Modeling > Copy > Areas

In the ANSYS graphics, select the solid circular area, and then in Copy Areas window, click on

OK



A type 6 in Number of copies
B type .1 DX X-offset in active CS

OK

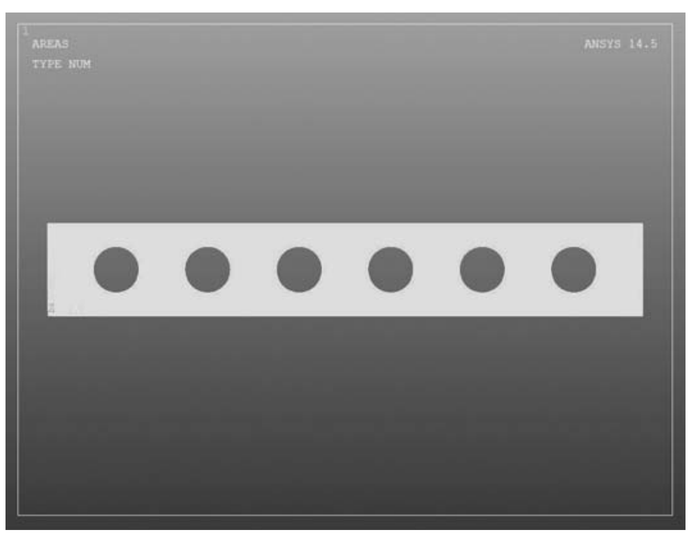
Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

Click on the center of the rectangular area to select it. Then, in Subtract Areas window, click on

OK

Click on all circular areas to select them. Then, in Subtract Areas window, click on

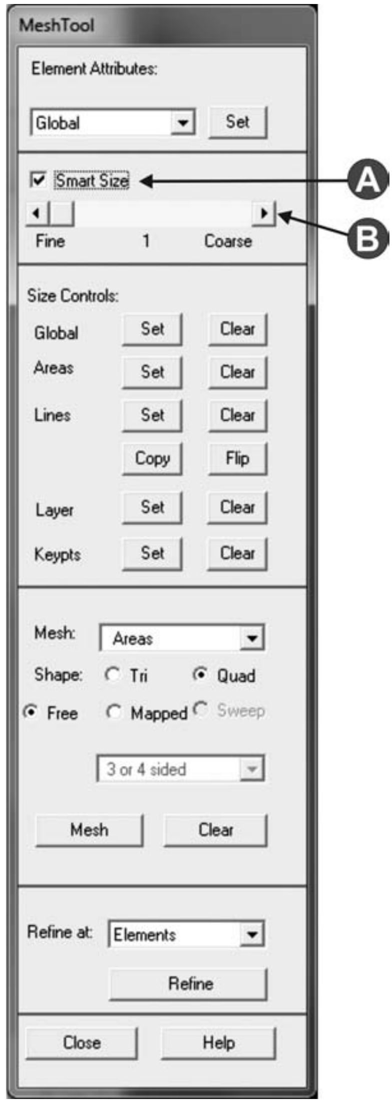
OK



ANSYS graphics showing the final geometry

The geometry is meshed with quadratic four-node elements. A free mesh is generated using the smart mesh option. The mesh refinement is 1.

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Smart Size

B set the level to 1

Then, in Mesh Tool, click on

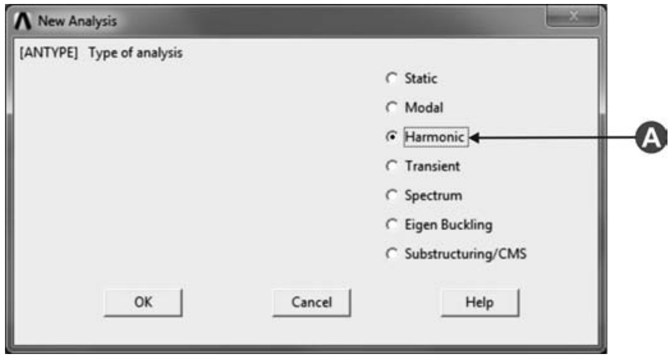
Mesh

In Mesh Areas, click on

Pick All

The analysis type is changed from Static to Harmonic. The frequency of the load is varied between 1 and 2000 Hz, and the number of subsets is 100.

Main Menu > Solution > Analysis Type > New Analysis



A select Harmonic

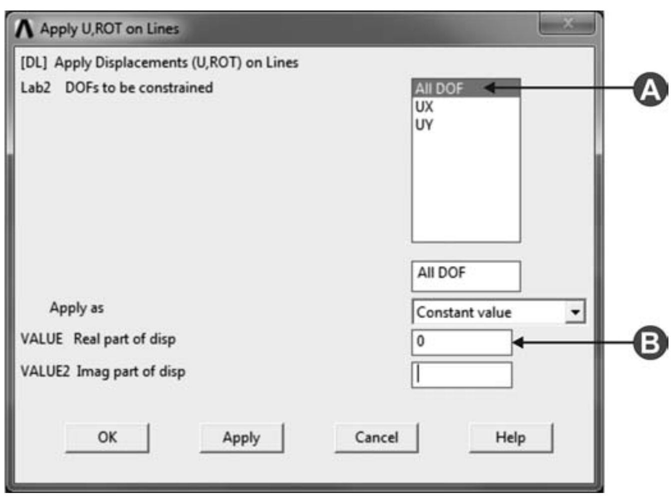
OK

Boundary conditions are applied as follows. The left vertical line of the beam is fixed, while the other lines are free. Force in the negative y-direction is applied at the keypoint at the upper right corner.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines

In the ANSYS graphics, click on the left line where displacement is applied. Then, in Apply U,ROT on Lines window, click on

OK



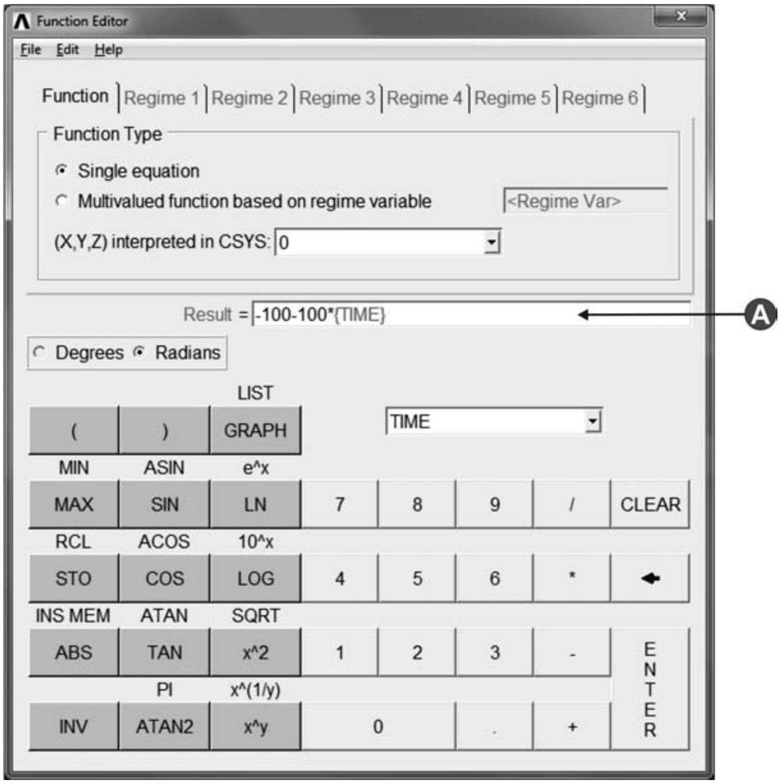
A select All DOF

B set VALUE to 0

OK

The Function Editor is used to apply the transient force formula as a nodal force. This technique is simple and convenient for this problem because an equation for the force is given. Notice that the force unit given in Figure 4.7b is in kN.

Main Menu > Solution > Define Loads > Apply > Functions > Define/Edit



A type the equation: −100−100*{TIME}

In the Equation Editors click on File then Save

Save the file as FY_Time, and the file name is optional. After saving the function, it has to be loaded to the ANSYS solution using the read file.

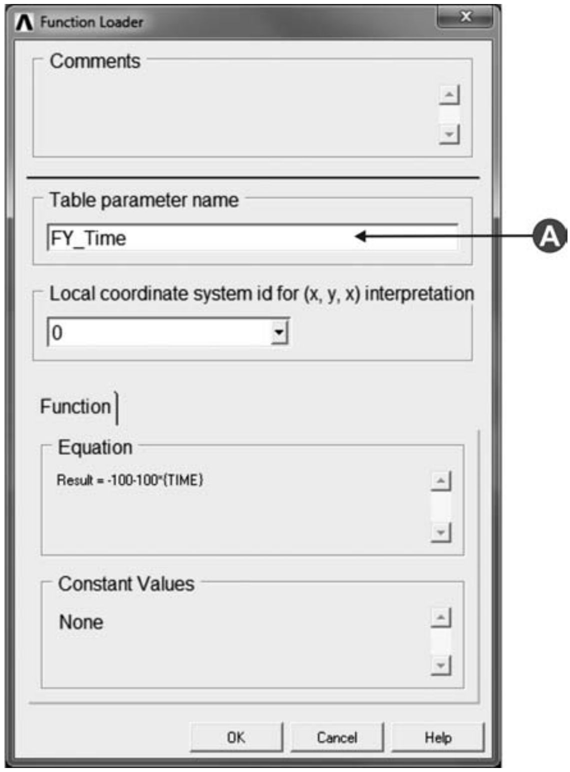
Save

Close the Function Editor

Main Menu > Solution > Define Loads > Apply > Functions > Read File

Select the file FY_Time.func then

Open



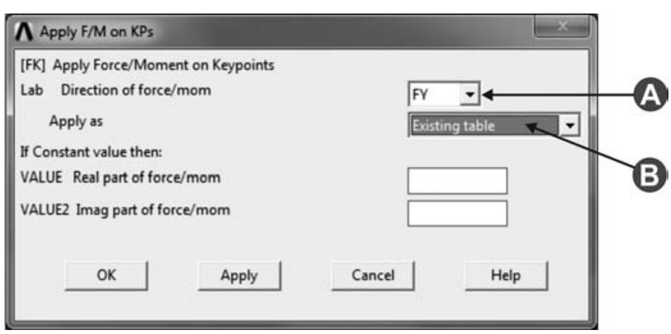
A type FY_Time in Table parameter name, and this name is optional

$\boxed{\textbf{OK}}$

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoint

Click on keypoint where the force is applied, and then in Apply F/M on KPs window, click on

$\boxed{\textbf{OK}}$

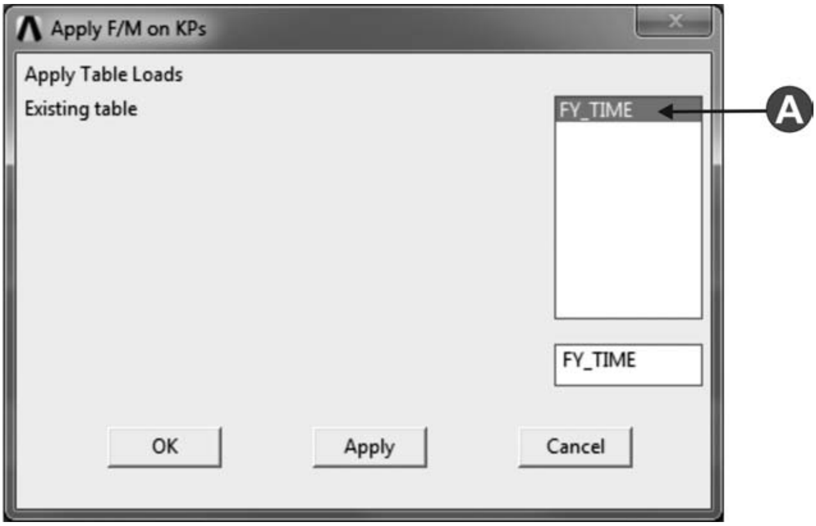


A select FY in Direction of force/mom

B select Existing table

$\boxed{\textbf{OK}}$

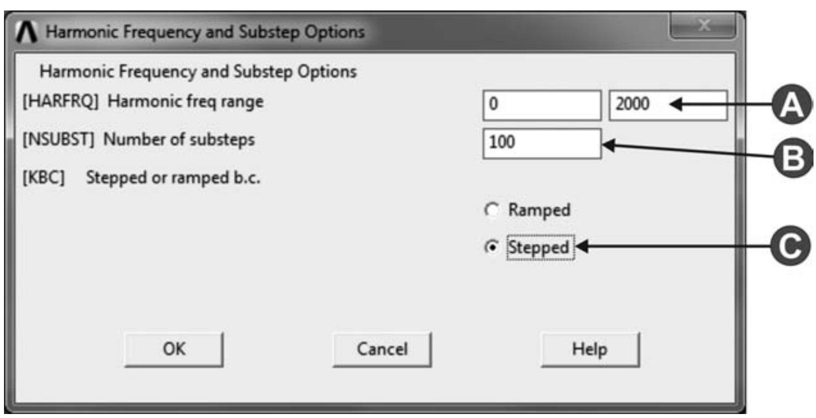
The following window will show up to select the function.



A select FY_TIME

OK

Main Menu > Solution > Load Step Opts > Time/Frequency > Freq and Substeps



A type 0 and 2000 in Harmonic freq range
B type 100 in Number of substeps
C select Stepped

OK

The final step is to run the ANSYS solution. ANSYS will assemble the stiffness matrices, apply the boundary conditions, and solve the problem.

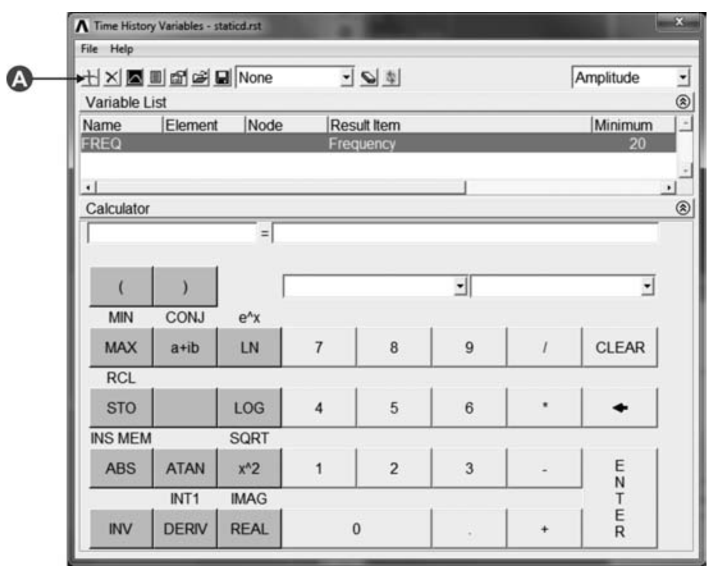
Main Menu > Solution > Solve > Current LS

OK

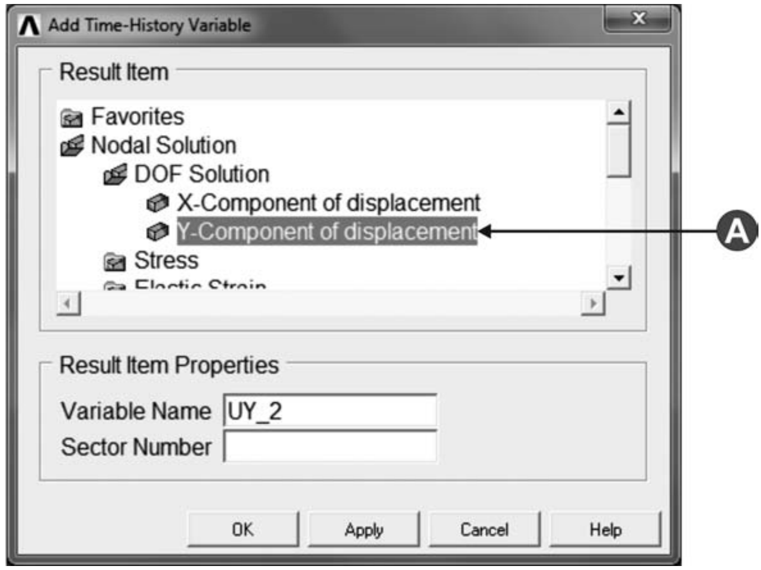
Close

A graph that shows the displacement at a specific location in the plate as a function of frequency can be done with the Time history. In the following steps, the displacement at the node where the force is applied is plotted as a function of frequency.

Main Menu > TimeHist Postpro



A click on the green + button

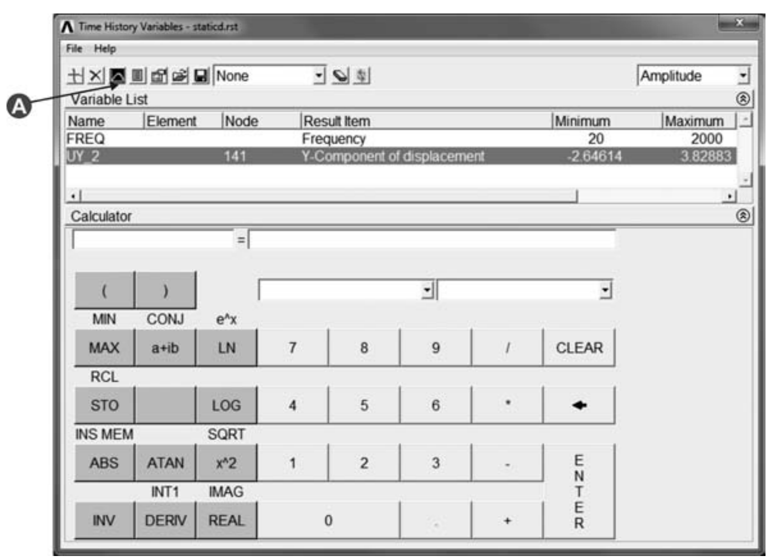


A click on Nodal Solution > DOF Solution > Y-Component of displacement

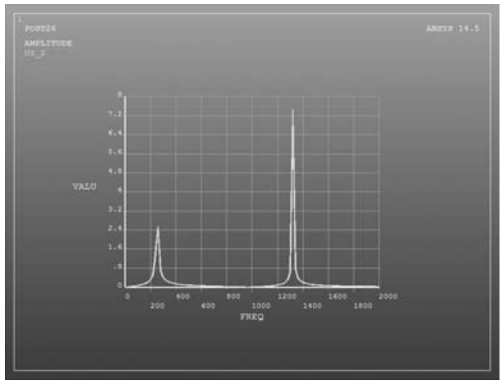
OK

In the ANSYS graphics, click on the node at the upper right corner of the plate and then in Node for Data window, click on

OK



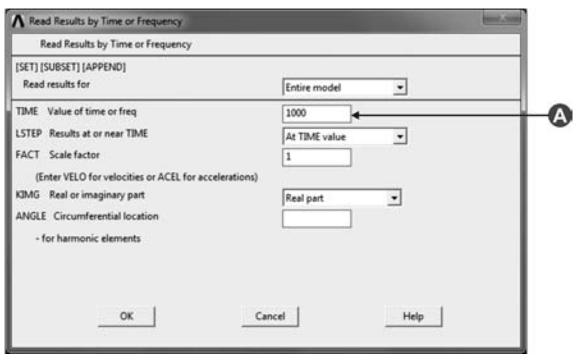
A click on the graph button



ANSYS graphics show the y-displacement as a function of frequency

The jumps in the displacement shown in the above figure occurred at the natural frequency of the first and the second modes of the plate. Next, an animation is created for the frequency of 1000 Hz.

Main Menu > General Postproc > Read Results > By Time/Freq

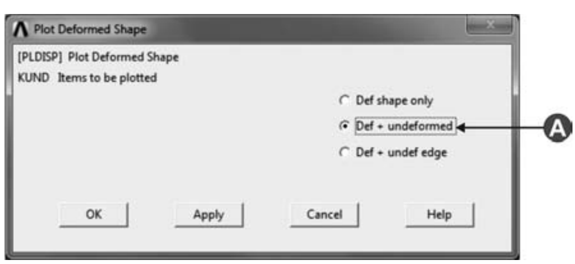


A type 1000 in TIME Value of time or freq

OK

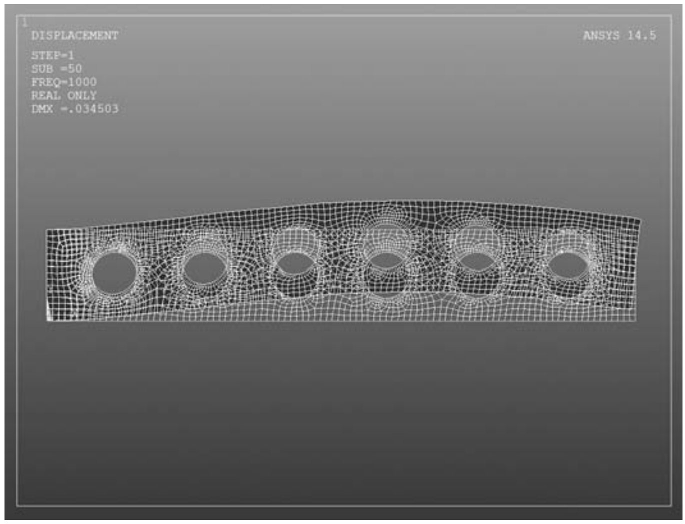
Utility menu > Plot > Areas

Main Menu > General Postproc > Plot Results > Deformed Shape



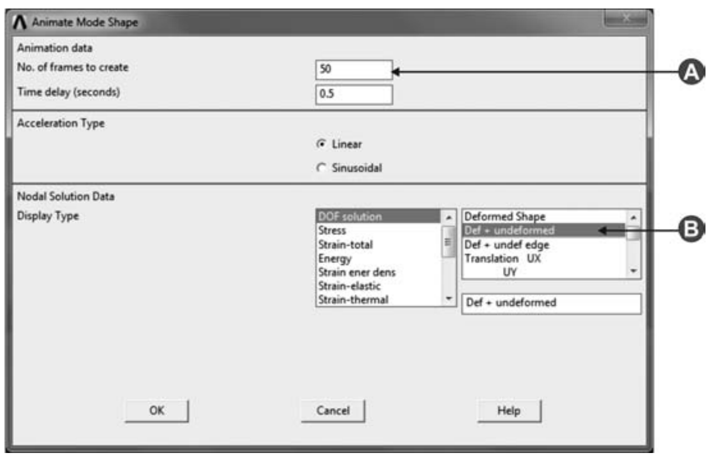
A select Def + undeformed

OK



ANSYS graphics show the plate before and after deformation

Utility Menu > PlotCtrls > Animate > Mode Shape



A type 50 in No. of frames to create

B select Def + undeformed

OK

Animation will be shown in ANSYS graphics.

4.9 Three-dimensional vibration of shaft with disks using ANSYS

For the geometry shown in Figure 4.8, the ends of the shaft are fixed, while the two disks and shaft are vibrating freely. The total shaft length is 0.655 m. Determine the first five natural vibration frequencies.

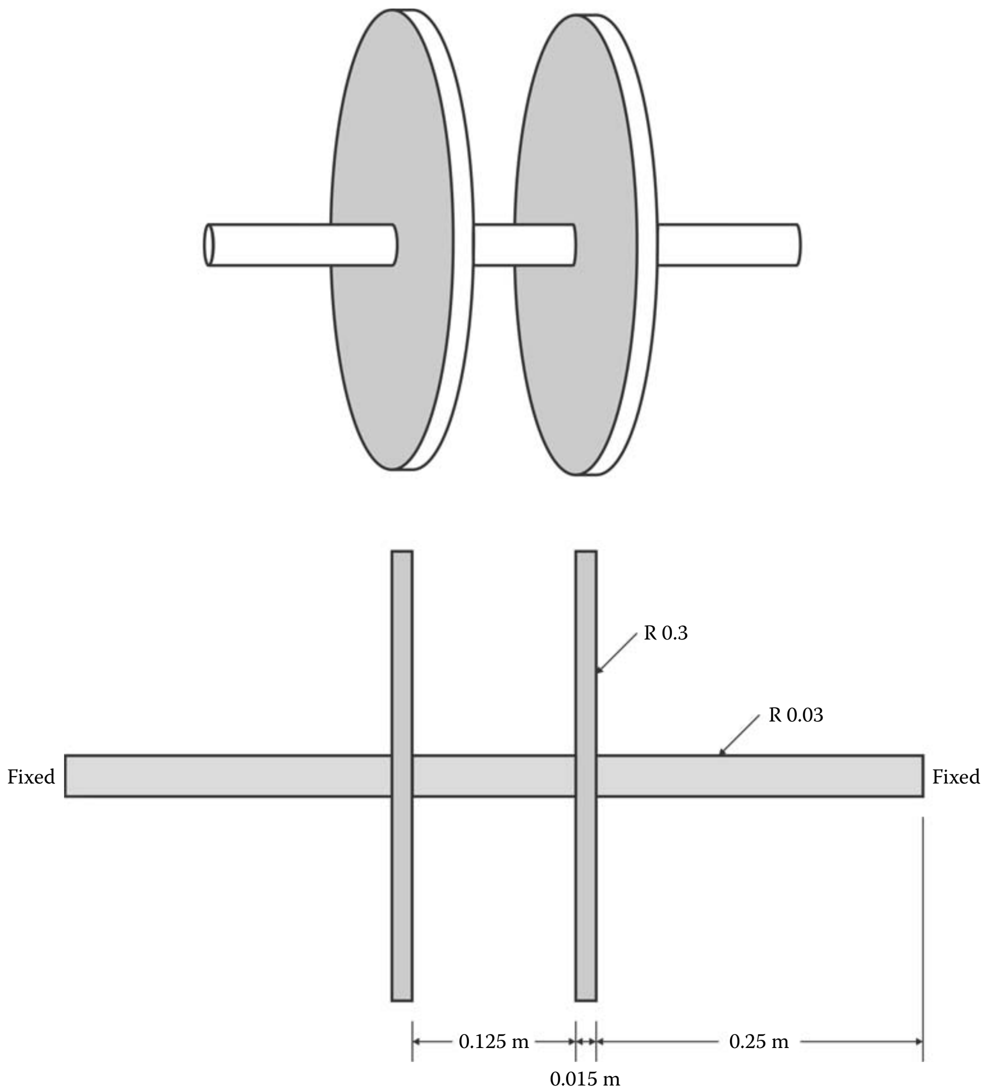


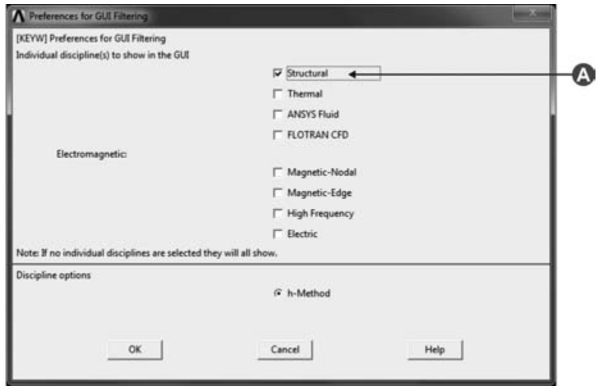
FIGURE 4.8 Shaft and two disks.

Also, create animation for the third mode. Let m, E = 180 GPa, v = 0.3, and ρ = 7000 kg/m^3. Use free mesh with Tet 4 node element and set smart size to 5.

Double click on the Mechanical APDL Product Launcher icon

This example is limited to structural analysis. Hence, select Structural only. The three-dimensional solid element must be used and the type of the element is tetrahedral with four nodes.

Main Menu > Preferences

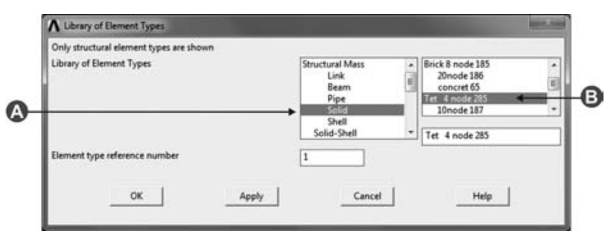


A select Structural

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add...



A select Solid

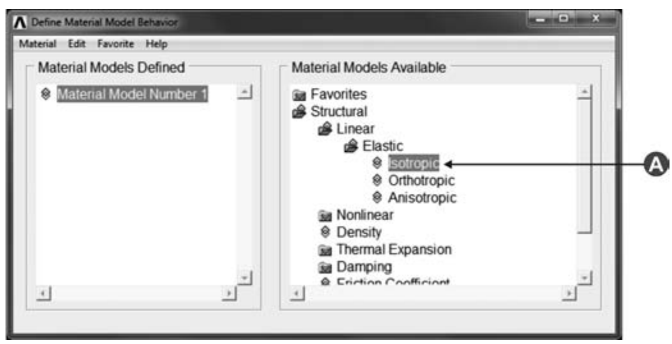
B select Tet 4 node 285

OK

Close

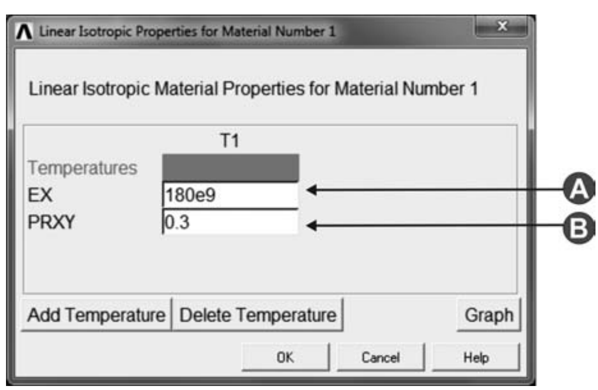
For the material properties, the modulus of elasticity and density are required to solve the problem.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

The following window will show up:

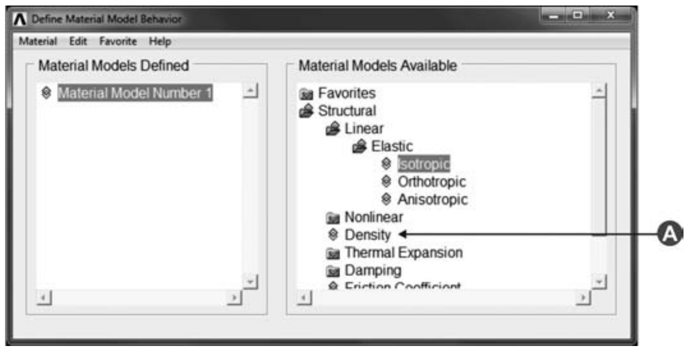


A type 180e9 in EX

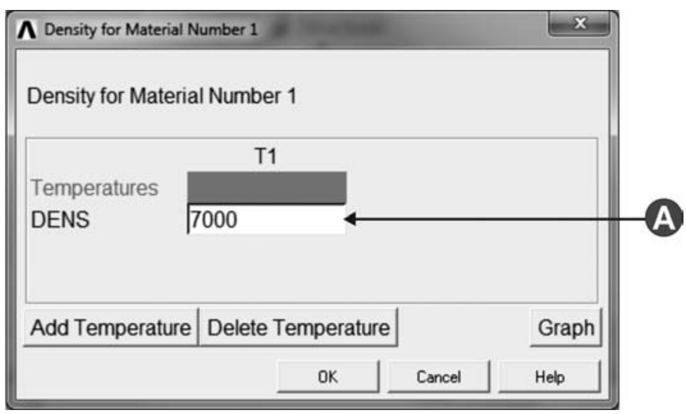
B type 0.3 in PRXY

OK

Main Menu > Material Props > Material Models



A click on Structural > Density



A type 7000 in DENS

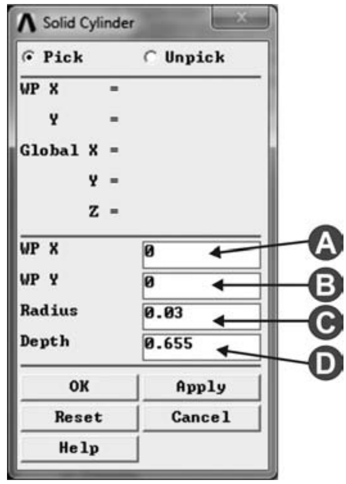
OK

Close the Material Model Behavior window

The used technique for modeling the geometry is quite simple. The following steps are performed:

1. The shaft is modeled as a long, solid cylinder.
2. The first disk is modeled as a thin, solid cylinder and then it is moved to its position in the shaft.
3. The second disk is created by copying the first disk.
4. The shaft and two disks are added to create one volume.

ANSYS Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder



A type 0 in WP X

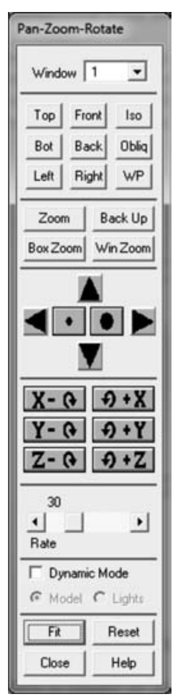
B type 0 in WP Y

C type 0.03 in Radius

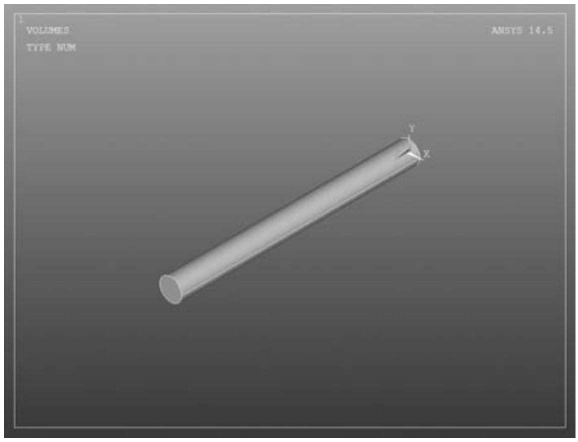
D type 0.655 in Depth

OK

ANSYS Utility Menu > PlotCtrls > Pan-Zoom-Rotate …

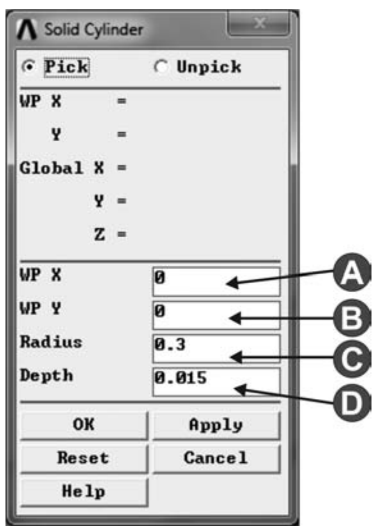


Click **Iso** button to show the isometric view of the shaft and zoom in and out.



ANSYS graphics show the isometric view of the shaft

ANSYS Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder



A type 0 in WP X

B type 0 in WP Y

C type 0.3 in Radius

D type 0.015 in Depth

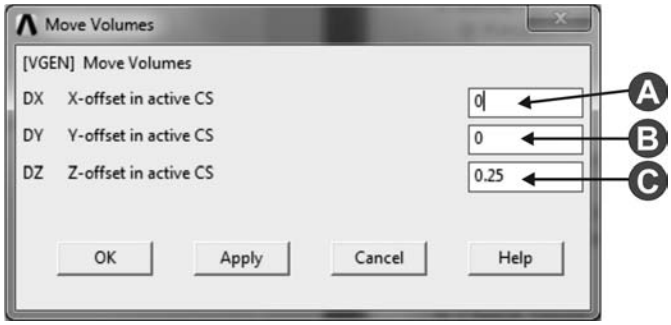
OK

The disk is not in its correct position, and it has to be moved along the z-axis to its position at a distance of 0.25 m from its current position.

ANSYS Main Menu > Preprocessor > Modeling > Move/Modify > Volume

Click on the disk and then in Move Volumes window, click on

OK



A type 0 in DX X-offset in active CS
B type 0 in DY Y-offset in active CS
C type 0.25 in DZ Z-offset in active CS

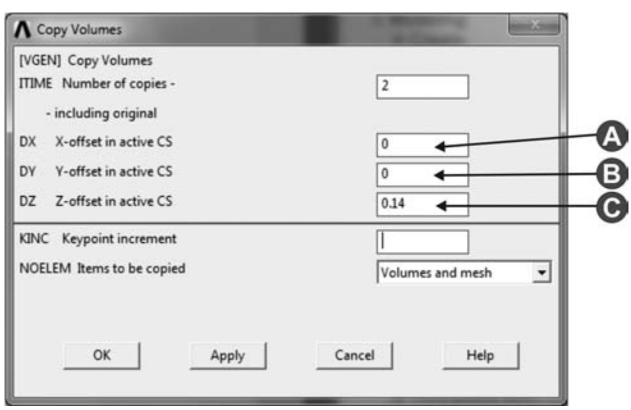
OK

The second disk is created by copying the first disk. The offset distance between the two disks is the spacing plus the thickness of the disk, which is 0.14 m.

ANSYS Main Menu > Preprocessor > Modeling > Copy > Volumes

Click on the disk, and then in Copy Volumes window, click on

OK



A type 0 in DX X-offset in active CS
B type 0 in DY Y-offset in active CS
C type 0.14 in DZ Z-offset in active CS

OK

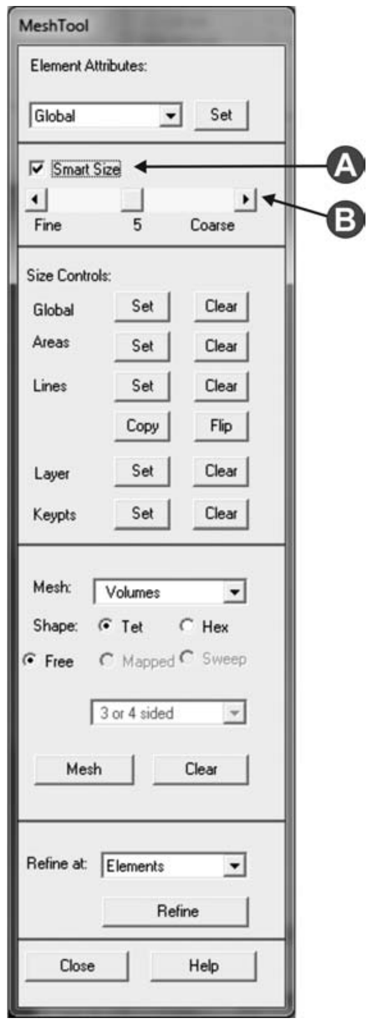
Next, the three volumes are added to create one volume.

ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Volumes

In Add Volumes windows, click on

Pick All

Main Menu > Preprocessor > Meshing > Mesh Tool



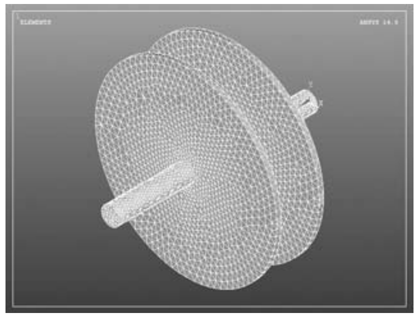
A select Smart Size

B set the level to 5

Mesh

In Meshed Areas window, click on

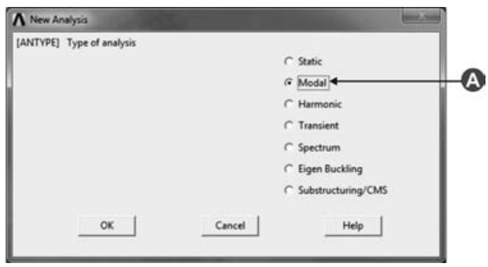
Pick All



ANSYS graphics show the mesh

The analysis type is changed from Static to Modal. The required number of free modes to be calculated is five.

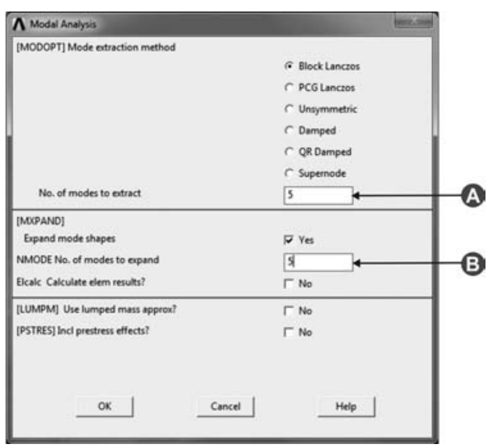
Main Menu > Solution > Analysis Type > New Analysis



A select Modal

OK

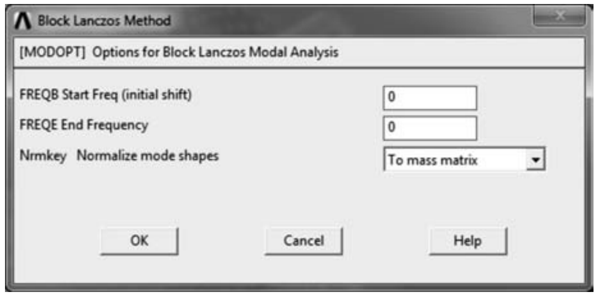
Main Menu > Solution > Analysis Type > Analysis Options



A type 5 in No. of modes to extract

B type 5 in NIMODE No. of modes to expand

OK



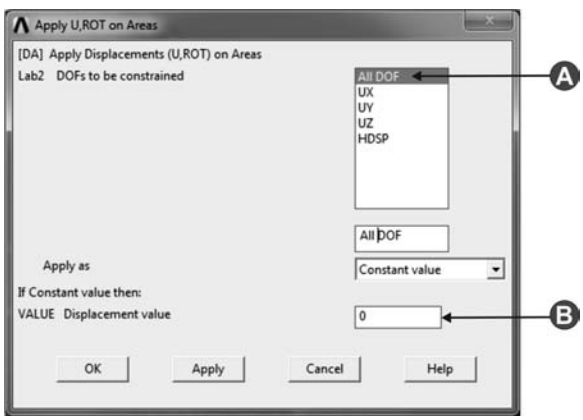
OK

The left and right surfaces of the shaft are fixed, while the other surfaces are free. No forces are applied.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Areas

In the ANSYS graphics, click on the left and right surfaces of the shaft, which are fixed. Then in Apply U,ROT on Areas window, click on

OK

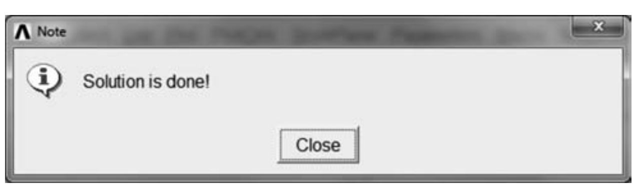


A select All DOF

B type 0 in Displacement value

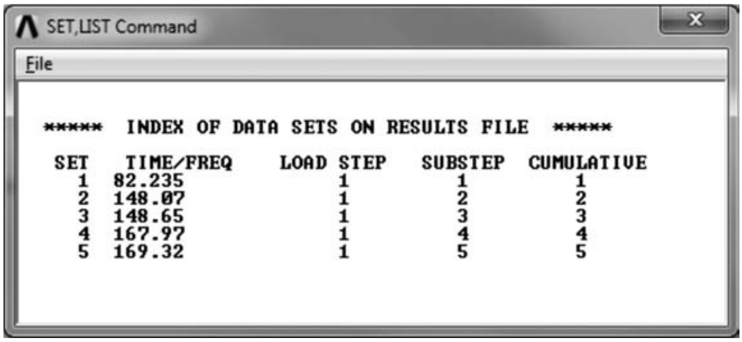
OK

Main Menu > Solution > Solve > Current LS



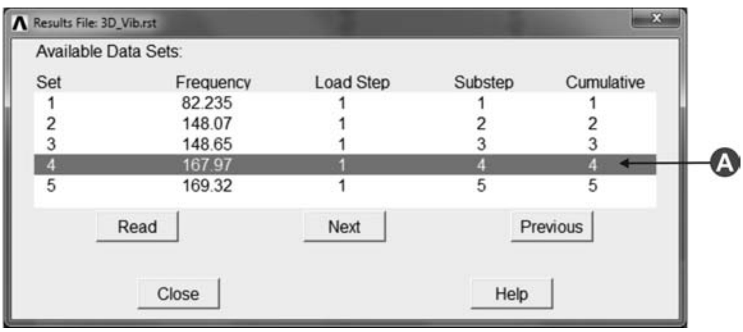
Close

Main Menu > General Postproc > Results Summary



ANSYS lists the natural frequencies of the shaft with two disks. Next, the deformed shape of the shaft for the mode number 4 is displaced, followed by an animation.

Main Menu > General Postproc > Read Results > By Pick

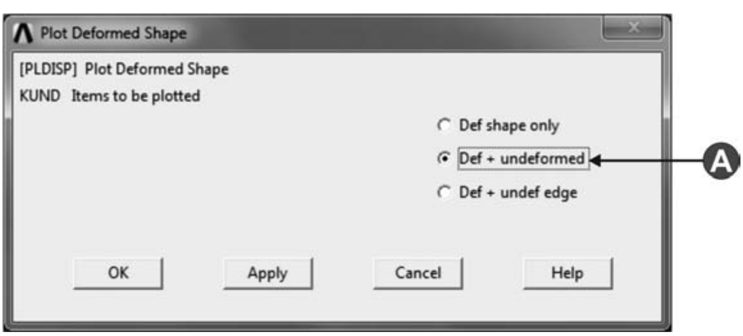


A select Set 4

Read

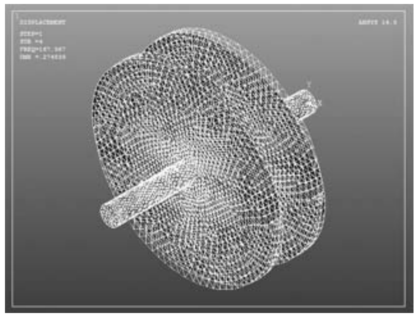
Close

Main Menu > General Postproc > Plot Results > Deformed Shape



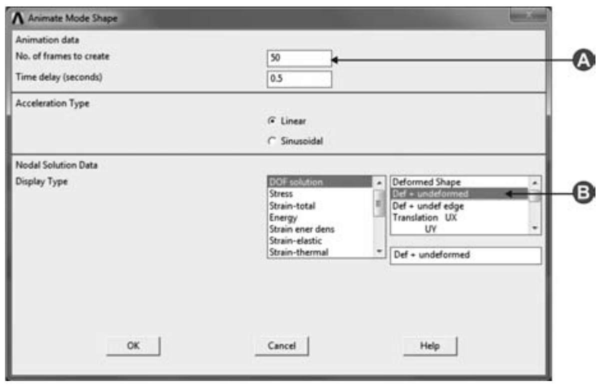
A select Def + undeformed

OK



ANSYS graphics show the vibrating shaft with two disks before and after deformation

Utility Menu > PlotCtrls > Animate > Mode Shape



A type 50 in No. of frames to create

B select Def + undeformed

$\boxed{\textbf{OK}}$

Animation will be shown for the third mode.

PROBLEM 4.1

Determine the first natural vibration frequency of a horizontal cantilever bar using finite element method, as shown in Figure 4.9. The total bar length is 3 m. Model the bar with three elements only and let E = 180 GPa, A = 4×10^{-3} m^2, and ρ = 7000 kg/m^3.

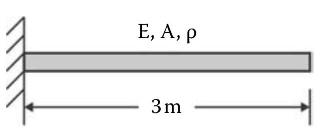


FIGURE 4.9 A simple cantilever bar.

PROBLEM 4.2

A horizontal cantilever bar is composed of two different bars of different cross sections and lengths, as shown in Figure 4.10. Determine the first natural vibration frequency of the bar using the finite element method. Model the bar with three elements only, as suggested in the figure, and let E = 210 GPa, ρ = 4500 kg/m^3, $A_1 = 4 \times 10^{-3}$ m^2, and $A_2 = 2 \times 10^{-3}$ m^2.

PROBLEM 4.3

A horizontal beam structure is composed of three different beams of different material properties, as shown in Figure 4.11. Determine the first natural vibration frequency of the structure using the finite element method. Model the structure with three elements only and let E_1 = 205 GPa, ρ_1 = 4250 kg/m^3, E_2 = 270 GPa, ρ_2 = 3000 kg/m^3, E_3 = 180 GPa, ρ_3 = 4000 kg/m^3, and the beams have a square cross-sectional area with height and width of 0.1 m.

PROBLEM 4.4

A horizontal cantilever bar structure is composed of two different bars of different cross-sectional areas and lengths, as shown in Figure 4.10. Determine the first five natural vibration frequencies of bar structure and let E = 210 GPa, ρ = 4500 kg/m^3; $A_1 = 4 \times 10^{-3}$ m^2, and $A_2 = 2 \times 10^{-3}$ m^2.

PROBLEM 4.5

A horizontal beam structure is composed of three different beams of different material properties, as shown in Figure 4.11. Determine the first five natural vibration frequencies of beam structure using ANSYS. Also, create an animation for the second mode. Mesh each

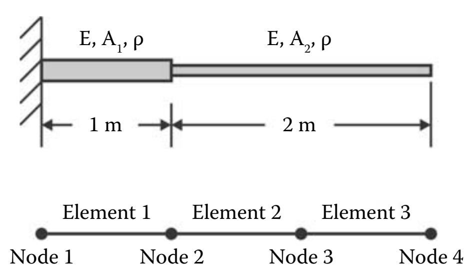


FIGURE 4.10 A cantilever bar made of two different bars.

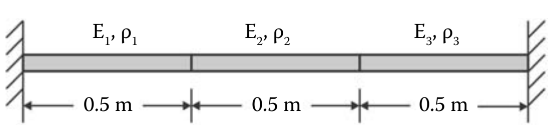


FIGURE 4.11 A horizontal beam structure made of three different beams.

beam with 50 elements and let E_1 = 205 GPa, ρ_1 = 4250 kg/m^3, E_2 = 270 GPa, ρ_2 = 3000 kg/m^3, E_3 = 180 GPa, ρ_3 = 4000 kg/m^3, and the beam has a square cross-sectional area with height and width of 0.01 m.

PROBLEM 4.6

For the plate with circular holes, as shown in Figure 4.12, the left vertical line is fixed. Determine the first five natural vibration frequencies of structure using ANSYS. Also, create an animation for the third mode. Let t = 0.002 m, E = 200 GPa, and ρ = 7500 kg/m^3.

PROBLEM 4.7

For the rectangular cross-sectional plate with square holes, as shown in Figure 4.13, both ends are fixed. Determine the first five natural vibration frequencies of the structure using ANSYS. Also, create an animation for the third mode. Let t = 0.001 m, E = 270 GPa, and ρ = 6000 kg/m^3.

PROBLEM 4.8

The geometry shown in Figure 4.14a is subjected to the transient force. The relationship between force and time is shown in Figure 4.14b. Create a graph showing the relationship between the displacement

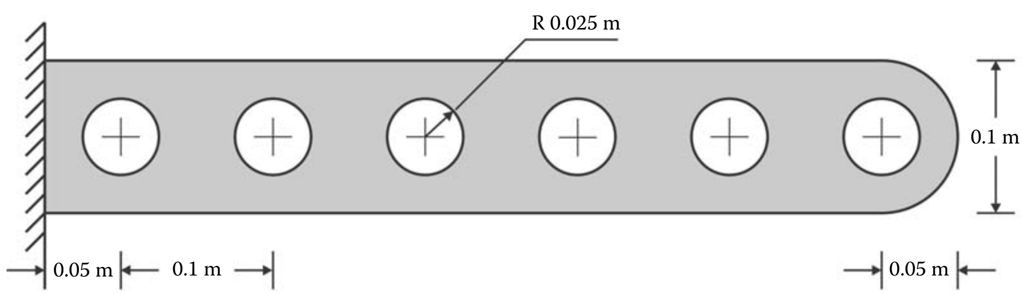


FIGURE 4.12 A plate with circular holes.

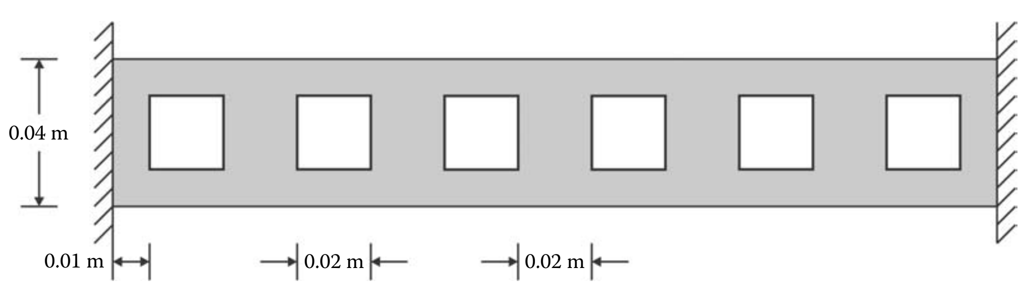


FIGURE 4.13 A rectangular plate with square holes.

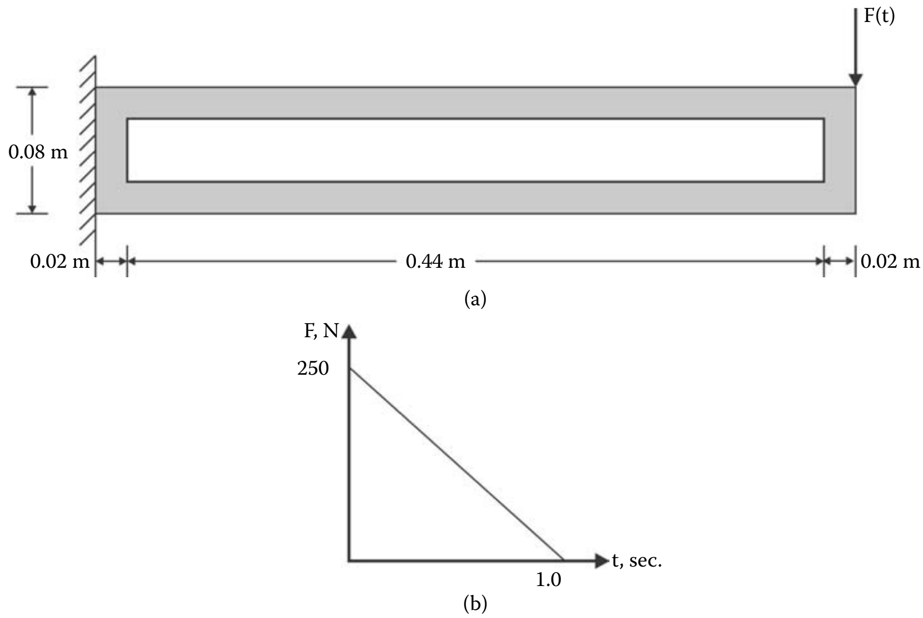


FIGURE 4.14 (a) A rectangular plate with a rectangular hole and (b) forcing function.

at the point where the force is applied and the load's frequency. Let t = 0.002 m, E = 230 GPa, and ρ = 4000 kg/m^3. The frequency of the load is varied between 1 and 5000 Hz, and the number of subset is 50.

PROBLEM 4.9

For the geometry shown in Figure 4.15, the ends of the shaft are fixed, while the disk and the shaft are vibrating freely. Determine the first five natural vibration frequencies for the shaft. Also, create an animation for the third mode. Let m, E = 180 GPa and ρ = 7000 kg/m^3. Use free mesh with Tet 4 node element, and set smart size to 5.

PROBLEM 4.10

For the airplane wing shown in Figure 4.16, determine the first five natural vibration frequencies. Let E = 180 GPa, v = 0.25, and ρ = 3500 kg/m^3. The wingspan is 5 m, and the left surface of the wing is fixed. Solve the problem as a three-dimensional problem. Use Tet 10-node elements with smart size set to 1.

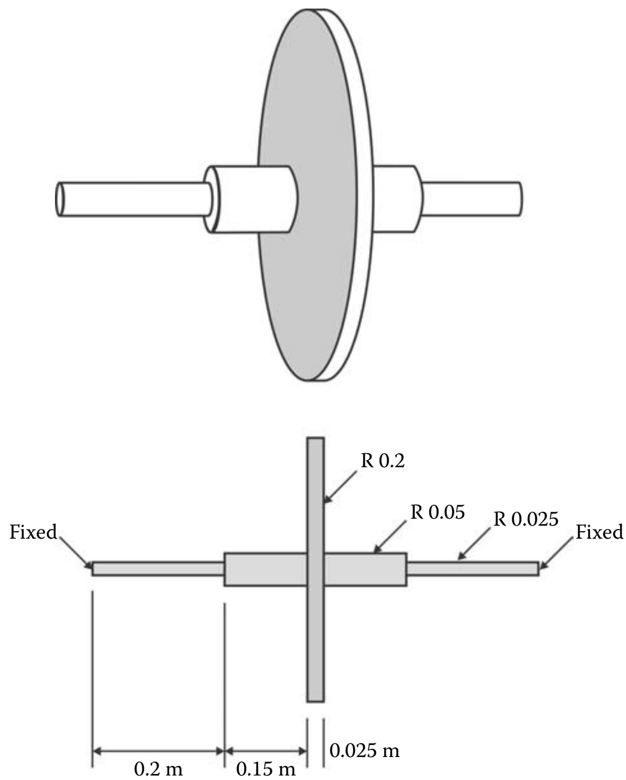


FIGURE 4.15 A shaft and disk.

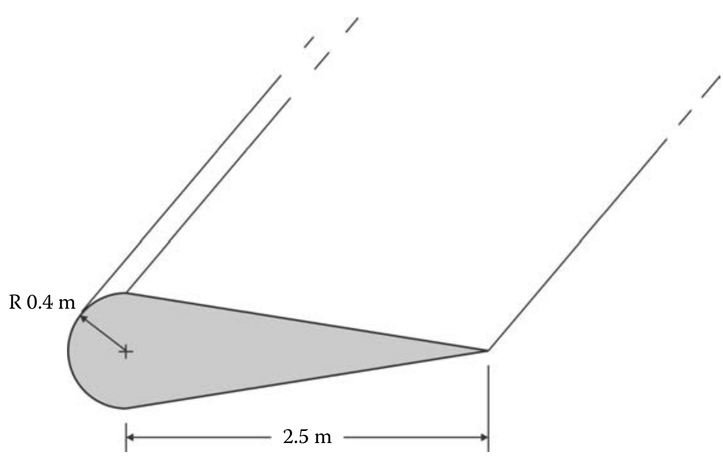


FIGURE 4.16 Airplane wing.

This page intentionally left blank

CHAPTER FIVE

Heat transfer

5.1 Introduction to heat conduction

Whenever a temperature gradient exists in a solid, heat will flow from the high-temperature region to the low-temperature region. The basic governing heat conduction equation is obtained by considering a plate with a surface area A and thickness Δx, as shown in Figure 5.1. One side is maintained at temperature T_1 and the other side is at temperature T_2. Experimental observation indicates that the rate of heat flow is directly proportional to the area and temperature difference, but inversely proportional to the plate thickness. The proportionality sign is replaced by an equal sign by introducing the constant k as follows:

$$Q = kA\frac{T_1 - T_2}{\Delta x} \tag{5.1}$$

The constant k is the thermal conductivity of the plate. This property depends on the type of the plate's material. Equation 5.1 is also called Fourier's law. Fourier's law can be expressed in differential form in the direction of the normal coordinate:

$$Q = -kA\frac{dT}{dn} \tag{5.2}$$

Also, Fourier's law can be expressed for multidimensional heat flux flow as follows:

$$q = -k\left(\frac{\partial T}{\partial x}\hat{i} + \frac{\partial T}{\partial y}\hat{j} + \frac{\partial T}{\partial z}\hat{k}\right) \tag{5.3}$$

An energy balance can be applied to a differential volume dx dy dz, for conduction analysis in a Cartesian coordinate, as shown in Figure 5.2. The objective of this energy balance is to obtain the temperature distribution within the solid. The temperature distribution is used to determine the heat flow at a certain surface, or to study the thermal stress. The heat flux perpendicular to the surface of the control volume is indicated

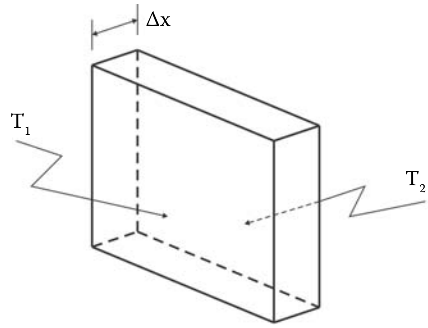


FIGURE 5.1 Heat transfer through a plate.

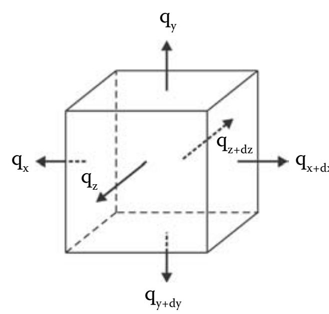


FIGURE 5.2 Differential control volume for the energy balance.

by the terms, q_x, q_y, and q_z. The heat flux at the opposite surface can then be expressed using the Taylor series expansion of the first order as follows:

$$q_{x+dx} = q_x + \frac{\partial q_x}{\partial x} dx \tag{5.4}$$

$$q_{y+dy} = q_y + \frac{\partial q_y}{\partial y} dy \tag{5.5}$$

$$q_{z+dz} = q_z + \frac{\partial q_z}{\partial z} dz \tag{5.6}$$

Energy can be generated in the medium, and the expression of the heat generation is

$$\dot{E}_{gen} = \dot{q}\, dx\, dy\, dz \tag{5.7}$$

where $\dot{q}$ is the generated heat per unit volume, W/m^3. If the heating process is unsteady, the total energy of the control volume can be increased or decreased. The energy storage term is expressed as

$$\dot{E}_{st} = \rho C_P \frac{\partial T}{\partial t} dx\, dy\, dz \tag{5.8}$$

The sum of the energy generation in the control volume and net heat flow should be equal to the energy stored in the control volume. The energy conservation can be expressed in the following mathematical form:

$$\dot{E}_{gen} + \left(\dot{E}_{in} - \dot{E}_{out}\right) = \dot{E}_{st} \tag{5.9}$$

Substituting expressions (5.4–5.8) into (5.9), the energy conservation equation becomes

$$\dot{q}\,dx\,dy\,dz + \left(\frac{\partial q_x}{\partial x}dy\,dz + \frac{\partial q_y}{\partial y}dx\,dz + \frac{\partial q_z}{\partial z}dx\,dy\right) = \rho C_P \frac{\partial T}{\partial t}\,dx\,dy\,dz \tag{5.10}$$

The Q''_x, Q''_y, and Q''_z are obtained from Fourier's law (5.2) as follows:

$$Q''_x = -k\frac{dT}{dx}\,dx \tag{5.11}$$

$$Q''_y = -k\frac{dT}{dy}\,dy \tag{5.12}$$

$$Q''_z = -k\frac{dT}{dx}\,dz \tag{5.13}$$

Finally, the conduction energy equation per unit volume, in a Cartesian coordinate, is expressed as

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) + \dot{q} = \rho C_p \frac{\partial T}{\partial t} \tag{5.14}$$

When the system reaches the steady-state condition, the term $\partial T/\partial t$ is equal to zero. If the thermal conductivity is independent of the direction, the conduction energy equation can be written in a simpler form as

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{\dot{q}}{k} = \frac{\rho C_P}{k}\frac{\partial T}{\partial t} \tag{5.15}$$

The energy equation is a partial differential equation with second order in space and first order in time. The boundary conditions along its surface as well as the initial condition must be specified. For the initial condition, the temperature distribution in the system must be provided. In heat transfer problems, there are three types of boundary conditions: temperature, heat flux, and convection.

The constant temperature, also called the Dirichlet condition, corresponds to a situation for which the surface is maintained at a fixed temperature. The mathematical expression for this boundary condition is

$$T(x,t) = T_s \tag{5.16}$$

The second type of boundary condition, which is called the Neumann condition, corresponds to a constant heat flux applied to a surface. The heat flux is related to the temperature gradient at the surface by Fourier's law as follows:

$$-k\frac{\partial T}{\partial x}=q \tag{5.17}$$

A special case of this boundary condition is an insulated boundary condition, and the heat flux should be zero:

$$-k\frac{\partial T}{\partial x}=0 \tag{5.18}$$

The third boundary condition corresponds to convection at a surface. The conduction–convection heat balance at the wall surface must be satisfied. The heat transfer coefficient (h) should be known, as well as the fluid bulk temperature (T_∞):

$$-k\frac{\partial T}{\partial x}=h\left[T_\infty-T(x,t)\right] \tag{5.19}$$

5.2 Finite element formulation for conductive heat transfer

The finite element method is an efficient way to solve conduction problems. The heat transfer solution can be used to estimate the heat flow at the boundaries or to determine the temperature distribution for thermal-stress analysis. In this chapter, the variation formulation is used to obtain the conductive equation. It is accomplished by minimizing the following potential function:

$$\begin{aligned} I=&\frac{1}{2}\int_V\left[k_x\left(\frac{\partial T}{\partial x}\right)^2+k_y\left(\frac{\partial T}{\partial y}\right)^2\right]dV-\int_V Q\,T\,dV-\int_S q\,T\,dS \\ &+\frac{1}{2}\int_S h(T-T_\infty)^2\,dS \end{aligned} \tag{5.20}$$

where the first term is heat conduction in the solid, the second term is volumetric heat generation, the third term is surface heat flux, and the fourth term is surface heat convection. For any surface, either surface heat flux or convection is applied. The temperature function T within each element can be expressed in terms of shape functions as

$$\{T\}=\begin{bmatrix} N_1 & N_2 & N_3 & \dots & N_i \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ \vdots \\ T_i \end{Bmatrix} \tag{5.21}$$

and the heat flux in the x- and y-directions is given by

$$\left\{\begin{array}{c} q_x \\ q_y \end{array}\right\} = -\left[\begin{array}{cc} k_x & 0 \\ 0 & k_y \end{array}\right]\left\{\begin{array}{c} \dfrac{\partial T}{\partial x} \\ \dfrac{\partial T}{\partial y} \end{array}\right\} \tag{5.22}$$

The temperature gradient vector can be expressed in terms of shape functions as

$$\{g\} = \left\{\begin{array}{c} \dfrac{\partial T}{\partial x} \\ \dfrac{\partial T}{\partial y} \end{array}\right\} = \left[\begin{array}{ccccc} \dfrac{\partial N_1}{\partial x} & \dfrac{\partial N_2}{\partial x} & \dfrac{\partial N_3}{\partial x} & \cdots & \dfrac{\partial N_i}{\partial x} \\ \dfrac{\partial N_1}{\partial y} & \dfrac{\partial N_2}{\partial y} & \dfrac{\partial N_3}{\partial y} & \cdots & \dfrac{\partial N_i}{\partial y} \end{array}\right]\left\{\begin{array}{c} T_1 \\ T_2 \\ T_3 \\ \vdots \\ T_i \end{array}\right\} \tag{5.23}$$

or symbolically,

$$\{g\} = [B]\{T\} \tag{5.24}$$

where [B] is a derivative of the shape functions. The thermal conductivity matrix is

$$[D] = \left[\begin{array}{cc} k_x & 0 \\ 0 & k_y \end{array}\right] \tag{5.25}$$

Hence, the heat flux vector can be written in terms of the temperature gradient vector and thermal conductivity matrix as follows:

$$\left\{\begin{array}{c} q_x \\ q_y \end{array}\right\} = [D]\{g\} \tag{5.26}$$

Potential function (5.20) can be expressed as

$$\begin{aligned} I = {} & \frac{1}{2}\int_V \left[\{g\}^T [D]\{g\}\right] dV - \int_V \{T\}^T [N]^T Q\, dV - \int_S \{T\}^T [N]^T q\, dS \\ & + \frac{1}{2}\int_S h\left[\left(\{T\}^T [N]^T - T_\infty\right)^2\right] dS \end{aligned} \tag{5.27}$$

Using the expression of the temperature gradient (5.24) and expanding the surface convection term, the potential function (5.27) can be written as

$$\begin{aligned} I = {} & \frac{1}{2}\{T\}^T \int_V [B]^T [D][B]\{T\} dV - \{T\}^T \int_V [N]^T Q\, dV - \{T\}^T \int_S [N]^T q\, dS \\ & + \frac{1}{2}\int_S h\left[\{T\}^T [N]^T [N]\{T\} - \left(\{T\}^T [N]^T + [N]\{T\}\right) T_\infty - T_\infty^2\right] dS \end{aligned} \tag{5.28}$$

Minimizing the potential equation (5.28) with respect to temperature vector, it gives

$$\frac{\partial I}{\partial\{T\}} = 0 = \int_V [B]^T [D][B]\{T\}\,dV - \int_V [N]^T Q\,dV - \int_S [N]^T q\,dS + \int_S h\Big[[N]^T [N]\{T\} - [N]^T T_\infty\Big]dS \tag{5.29}$$

Rearranging Equation 5.29 as

$$\int_V [B]^T [D][B]\{T\}\,dV + \int_S h[N]^T [N]\{T\}\,dS = \int_V [N]^T Q\,dV + \int_S [N]^T q\,dS + \int_S h[N]^T T_\infty\,dS \tag{5.30}$$

Equation 5.30 can be expressed in terms of conductive matrix, nodal temperature, and thermal forces as

$$[K]\{T\} = \{F_Q\} + \{F_q\} + \{F_c\} \tag{5.31}$$

where

$$[K] = \int_V [B]^T [D][B]\,dV + \int_S h[N]^T [N]\,dS \tag{5.32}$$

The first term for the conductive matrix (5.32) is for axial conduction through the element, while the second term is convection at the external surface of the element, and the thermal forces are

$$\{F_Q\} = \int_V [N]^T Q\,dV \tag{5.33}$$

$$\{F_q\} = \int_S [N]^T q\,dS \tag{5.34}$$

$$\{F_c\} = \int_S h[N]^T T_\infty\,dS \tag{5.35}$$

The term $\{F_Q\}$ is volumetric heat generation in the element, $\{F_q\}$ is applied heat flux on the external surface of the element, and $\{F_c\}$ is applied convection on the external surface of the element. The integrations are over the surface where convection or heat flux is applied. A symbolic expression for element conduction equation is

$$[K]\{T\} = \{F\} \tag{5.36}$$

where

$$\{F\} = \{F_Q\} + \{F_q\} + \{F_c\} \tag{5.37}$$

The global conductive matrix is obtained by assembling the conductive matrix for each element as follows:

$$[K] = \sum_{e=1}^{N} \left[K^{(e)}\right] \tag{5.38}$$

The thermal force vectors are assembled to form a global one as follows:

$$\{F\} = \sum_{e=1}^{N} \left\{F^{(e)}\right\} \tag{5.39}$$

Also, the temperature vectors are assembled to form a global one as follows:

$$\{T\} = \sum_{e=1}^{N} \left\{T^{(e)}\right\} \tag{5.40}$$

5.3 Finite element method for one-dimensional heat conduction

Consider one-dimensional heat conduction, as shown in Figure 5.3, where the length of the element is L. The temperature function is similar to the displacement function, which is

$$T(x) = N_1T_1 + N_2T_2 \tag{5.41}$$

where T_1 and T_2 are nodal temperatures, and N_1 and N_2 are shape functions, which are defined as

$$N_1 = 1 - \frac{x}{L} \tag{5.42}$$

$$N_2 = \frac{x}{L} \tag{5.43}$$

The shape functions can be presented in matrix form as

$$[N] = \begin{bmatrix} N_1 & N_2 \end{bmatrix} = \begin{bmatrix} 1 - \dfrac{x}{L} & \dfrac{x}{L} \end{bmatrix} \tag{5.44}$$

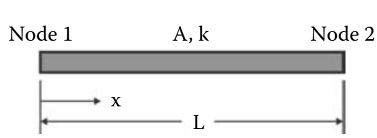


FIGURE 5.3 One-dimensional heat conduction.

The thermal conductivity of the material is assumed to be isotropic. The [B] and [D] matrices are calculated as

$$[B] = \begin{bmatrix} \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} \end{bmatrix} = \begin{bmatrix} \frac{1}{-L} & \frac{1}{L} \end{bmatrix} \tag{5.45}$$

$$[D] = \begin{bmatrix} k_x & 0 \\ 0 & k_y \end{bmatrix} = \begin{bmatrix} k & 0 \\ 0 & k \end{bmatrix} \tag{5.46}$$

In Equation 5.32, and since the element has a uniform cross-sectional area, the integral over differential volume dV is replaced by A dx, where A is the cross-sectional area of the element, and the integral over differential surface dS is replaced by P dx, where P is the circumference of the element. The element conductive matrix (5.32) is expressed as

$$\begin{aligned}[K] = &\int \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix}^T \begin{bmatrix} k & 0 \\ 0 & k \end{bmatrix} \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix} A\,dx \\ &+ \int h \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix}^T \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix} dA \end{aligned} \tag{5.47}$$

After performing matrix operations and integration, the element conductive matrix (5.47) is expressed as

$$[K] = \frac{Ak}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{hPL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} + hA\begin{bmatrix} 1 & 0 \\ 0 & 0 \end{bmatrix} \tag{5.48}$$

The first term of the conductive matrix is axial conduction, the second term is perimeter convection, and the third term is convection at the surface of node 1. If the convection is applied at node 2, the conductive matrix (5.48) becomes

$$[K] = \frac{Ak}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{hPL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} + hA\begin{bmatrix} 0 & 0 \\ 0 & 1 \end{bmatrix} \tag{5.49}$$

The thermal nodal forces can be calculated using expressions (5.33–5.35) as follows:

$$\{F_Q\} = \int \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix}^T QA\,dx = \frac{QAL}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \tag{5.50}$$

$$\{F_q\} = \int \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix}^T qP\,dx = \frac{qPL}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + qA\begin{Bmatrix} 1 \\ 0 \end{Bmatrix} \tag{5.51}$$

$$\{F_c\} = \int h \begin{bmatrix} 1-\dfrac{x}{L} & \dfrac{x}{L} \end{bmatrix}^T T_\infty\, P\, dx = \frac{hT_\infty PL}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + hT_\infty A\begin{Bmatrix} 1 \\ 0 \end{Bmatrix} \tag{5.52}$$

In Equations 5.51 and 5.52, the first term after integration is for heat flux or convection applied at the perimeter and the second term is for heat flux or convection applied at node 1. If the convection is applied at node 2, Equations 5.51 and 5.52 become, respectively,

$$\{F_q\} = \int \begin{bmatrix} 1-\dfrac{x}{L} & \dfrac{x}{L} \end{bmatrix}^T qP\, dx = \frac{qPL}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + qA\begin{Bmatrix} 0 \\ 1 \end{Bmatrix} \tag{5.53}$$

$$\{F_c\} = \int h \begin{bmatrix} 1-\dfrac{x}{L} & \dfrac{x}{L} \end{bmatrix}^T T_\infty\, P\, dx = \frac{hT_\infty PL}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + h\, T_\infty\, A\begin{Bmatrix} 0 \\ 1 \end{Bmatrix} \tag{5.54}$$

5.4 Heat transfer through a composite wall

Consider a composite wall shown in Figure 5.4. The wall is composed of three layers with different thermal conductivities: $k_A = 0.1$ W/m · °C, $k_B = 0.2$ W/m · °C, and $k_C = 0.25$ W/m · °C. A convection boundary condition is applied at the left surface, $h = 15$ W/m^2 · °C and $T_i = 25$°C, while the right surface is at a fixed temperature, $T_o = 50$°C. Calculate the temperature at the interfaces and heat flow through the wall. Given that $L_A = 5$ cm, $L_B = 15$ cm, and $L_C = 2.5$ cm.

The heat conduction in the wall is a one-dimensional heat transfer problem and can be solved using the finite element method using three elements only, as shown in Figure 5.4. Since the temperature variation is linear in each layer, having more than one element in each layer would not affect the results. The convection thermal load is applied at node 1 and temperature load at node 4. The results are per unit area A.

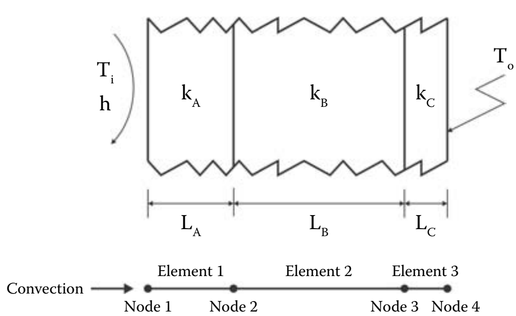


FIGURE 5.4 Heat conduction in a composite wall and finite element mesh.

For element 1, convection load is applied at node 1, and no nodal thermal load is applied at node 2, there is no convection or heat flux applied at the surface of the element, and no heat is generated in the element. The conductive matrix and nodal thermal load vector, respectively, are

$$\left[K^{(1)}\right]=\frac{k_A}{L_A}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}+0+h\begin{bmatrix}1 & 0\\ 0 & 0\end{bmatrix}$$

$$=\frac{0.1}{0.05}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}+15\begin{bmatrix}1 & 0\\ 0 & 0\end{bmatrix}=\begin{bmatrix}17 & -2\\ -2 & 2\end{bmatrix}$$

$$\left\{F^{(1)}\right\}=\left\{F_c^{(1)}\right\}=hT_\infty\begin{Bmatrix}1\\ 0\end{Bmatrix}=\begin{Bmatrix}375\\ 0\end{Bmatrix}$$

For elements 2 and 3, no nodal thermal loads are applied on nodes, there is no convection or heat flux applied at the element, and no heat is generated in the element. The conductive matrix and nodal thermal load vector are for elements 2 and 3, respectively,

$$\left[K^{(2)}\right]=\frac{k_B}{L_B}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{0.2}{0.15}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\begin{bmatrix}1.33 & -1.33\\ -1.33 & 1.33\end{bmatrix}$$

$$\left[K^{(3)}\right]=\frac{k_C}{L_C}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\frac{0.25}{0.025}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}=\begin{bmatrix}10 & -10\\ -10 & 10\end{bmatrix}$$

$$\left\{F^{(2)}\right\}=\begin{Bmatrix}0\\ 0\end{Bmatrix}$$

$$\left\{F^{(3)}\right\}=\begin{Bmatrix}0\\ 0\end{Bmatrix}$$

Assembling the conductive matrix and nodal thermal forces and temperatures yields, it gives

$$\begin{bmatrix}17 & -2 & 0 & 0\\ -2 & 3.33 & -1.33 & 0\\ 0 & -1.33 & 11.33 & -10\\ 0 & 0 & -10 & 10\end{bmatrix}\begin{Bmatrix}T_1\\ T_2\\ T_3\\ T_4\end{Bmatrix}=\begin{Bmatrix}375\\ 0\\ 0\\ Q_4\end{Bmatrix}$$

where Q_4 is heat flow at node 4, which is required to maintain the temperature at node 4 at 50°C. Since the temperature at node 4 is known, T_4 = 50°C, the last equation is modified, so that T_4 has a value of 50°C.

Consequently, the third equation number 3 is also modified. The conductive matrix and nodal thermal forces and temperatures are expressed as

$$\begin{bmatrix} 17 & -2 & 0 & 0 \\ -2 & 3.33 & -1.33 & 0 \\ 0 & -1.33 & 11.33 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 375 \\ 0 \\ 500 \\ 50 \end{Bmatrix}$$

Then, there are three equations and three unknowns, the solution for the equations is

$T_1 = 26.175°C$

$T_2 = 34.981°C$

$T_3 = 48.237°C$

5.5 Finite element method for two-dimensional heat conduction

The simplest type of element used for heat conduction is a three-node triangular element, as shown in Figure 5.5. The nodes are named i, j, and m, and the temperature is linearly varying across the element.

The temperature function is the same as the displacement function used in structural analysis:

$$\{T\} = \begin{bmatrix} N_i & N_j & N_m \end{bmatrix} \begin{Bmatrix} T_i \\ T_j \\ T_m \end{Bmatrix} \tag{5.55}$$

and the shape functions are given as

$$N_i = \frac{1}{2A}\left(\alpha_i + \beta_i x + \gamma_i y\right) \tag{5.56}$$

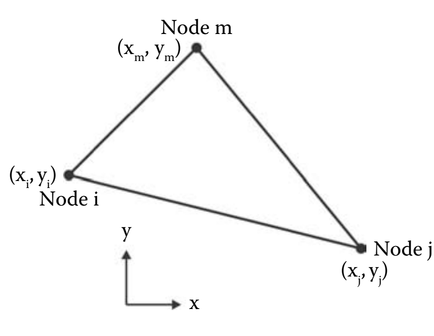


FIGURE 5.5 Three-node triangular element.

$$N_j = \frac{1}{2A}\left(\alpha_j + \beta_j x + \gamma_j y\right) \tag{5.57}$$

$$N_m = \frac{1}{2A}\left(\alpha_m + \beta_m x + \gamma_m y\right) \tag{5.58}$$

where α, β, and γ are defined as

$$\begin{array}{lll} \alpha_i = x_j y_m - y_j x_m & \alpha_j = x_m y_i - y_m x_i & \alpha_m = x_i y_j - y_i x_j \\ \beta_i = y_j - y_m & \beta_j = y_m - y_i & \beta_m = y_i - y_j \\ \gamma_i = x_m - x_j & \gamma_j = x_i - x_m & \gamma_m = x_j - x_i \end{array} \tag{5.59}$$

The [B] matrix is calculated using Equation 5.23 as

$$[B] = \begin{bmatrix} \dfrac{\partial N_i}{\partial x} & \dfrac{\partial N_j}{\partial x} & \dfrac{\partial N_m}{\partial x} \\ \dfrac{\partial N_i}{\partial y} & \dfrac{\partial N_j}{\partial y} & \dfrac{\partial N_m}{\partial y} \end{bmatrix} = \frac{1}{2A}\begin{bmatrix} \beta_i & \beta_j & \beta_m \\ \gamma_i & \gamma_j & \gamma_m \end{bmatrix} \tag{5.60}$$

where A is the area of the element. The thermal conductivity [D] matrix is

$$[D] = \begin{bmatrix} k_x & 0 \\ 0 & k_y \end{bmatrix} \tag{5.61}$$

The thermal conductive matrix (5.32)

$$[K] = \int_V [B]^T [D][B]\, dV + \int_S h\, [N]^T [N]\, dS$$

Assuming the element has a constant thickness t, the thermal conductive matrix can be written for a triangular element as

$$[K] = tA\,[B]^T [D][B] + \frac{hLt}{6}\begin{bmatrix} 2 & 1 & 0 \\ 1 & 2 & 0 \\ 0 & 0 & 0 \end{bmatrix} \tag{5.62}$$

where A is the area of the element and L is the element side length where the convection is applied. The second term in Equation 5.62 represents convection on the side of the element, and the volumetric heat generation can be considered as a nodal thermal force as follows:

$$\{F_Q\} = \int_V [N]^T Q\, dV = \frac{QAt}{3}\begin{Bmatrix} 1 \\ 1 \\ 1 \end{Bmatrix} \tag{5.63}$$

where the heat generation is equally distributed on the nodes. The heat flux on a surface is considered as nodal thermal load as

$$\{F_q\} = \int_S [N]^T q\,dS = \int_S \begin{bmatrix} N_i & N_j & N_m \end{bmatrix}^T q\,dS \tag{5.64}$$

If the heat flux is applied on a specific side of the element, the heat flux vector is

$$\{F_q\} = \frac{qL_{ij}}{2}\begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \text{ if heat flux applied between nodes i and j} \tag{5.65}$$

$$\{F_q\} = \frac{qL_{jm}}{2}\begin{Bmatrix} 0 \\ 1 \\ 1 \end{Bmatrix} \text{ if heat flux applied between nodes j and m} \tag{5.66}$$

$$\{F_q\} = \frac{qL_{mi}}{2}\begin{Bmatrix} 1 \\ 0 \\ 1 \end{Bmatrix} \text{ if heat flux applied between nodes m and i} \tag{5.67}$$

where L_{ij} is the side length between nodes i and j, L_{jm} is the side length between nodes j and m, and L_{mi} is the side length between nodes m and i. The convection on a surface is considered as nodal thermal load as follows:

$$\{F_c\} = \int_S [N]^T h\,T_\infty\,dS = \int_S \begin{bmatrix} N_i & N_j & N_m \end{bmatrix}^T h\,T_\infty\,dS \tag{5.68}$$

If the convection is applied on a specific side of the element, the convection vector is

$$\{F_c\} = \frac{hT_\infty L_{ij}}{2}\begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \text{ if convection applied between nodes i and j} \tag{5.69}$$

$$\{F_c\} = \frac{hT_\infty L_{jm}}{2}\begin{Bmatrix} 0 \\ 1 \\ 1 \end{Bmatrix} \text{ if convection applied between nodes j and m} \tag{5.70}$$

$$\{F_c\} = \frac{hT_\infty L_{mi}}{2}\begin{Bmatrix} 1 \\ 0 \\ 1 \end{Bmatrix} \text{ if convection applied between nodes m and i} \tag{5.71}$$

5.6 Heat conduction in a solid plane

The two-dimensional body, shown in Figure 5.6a, is subjected to convection at the vertical left side, with $T_\infty = 50°C$ and $h = 20\ W/m^2 \cdot °C$. At the right side, a fixed temperature boundary is imposed, with $T_o = 100°C$, while the two horizontal sides are well insulated. The sides' length of the body is 2 m, and it has a thermal conductivity of 25 W/m · °C. Determine the temperature at the center of the body.

There are unlimited options for elements distributions, and some of these options are shown in Figure 5.6b. Increasing the number of elements will definitely enhance the accuracy of the results, but up to a certain number of elements. After which, the results become independent of the number of the elements. The first mesh contains two elements, which is the minimum number of elements required to solve this problem. However, this mesh cannot predict the temperature at the center of the body because there is no node at the center of the body. The second mesh contains 4 elements, the third mesh contains 8 elements, and the fourth mesh contains 16 elements.

For an illustration purpose, the second mesh is selected because it has the least number of elements. Figure 5.6c shows the nodes and elements distribution for the geometry, and it consists of four elements and five nodes. First, the [B], [D], and [K] matrices for element 1 are formulated. The name of node 1 is (i) and its coordinate is (0,0), the name of node 2 is (j) and its coordinate is (2,0), and the name of node 5 is (m) and its coordinate is (1,1). The values of β's and γ's are required for the [B] matrix and are calculated using Equation 5.59, and A is the area of the element, so we have

$$\beta_i = y_j - y_m = 0 - 1 = -1$$

$$\beta_j = y_m - y_i = 1 - 0 = 1$$

$$\beta_m = y_i - y_j = 0 - 0 = 0$$

$$\gamma_i = x_m - x_j = 1 - 2 = -1$$

$$\gamma_j = x_i - x_m = 0 - 1 = -1$$

$$\gamma_m = x_j - x_i = 2 - 0 = 2$$

$$A = 0.5(2)1 = 1\ m^2$$

Then, the [B] matrix is formulated using Equation 5.60:

$$[B] = \frac{1}{2}\begin{bmatrix} -1 & 1 & 0 \\ -1 & -1 & 2 \end{bmatrix}$$

and the [D] matrix is formulated using Equation 5.61:

$$[D] = \begin{bmatrix} 25 & 0 \\ 0 & 25 \end{bmatrix}$$

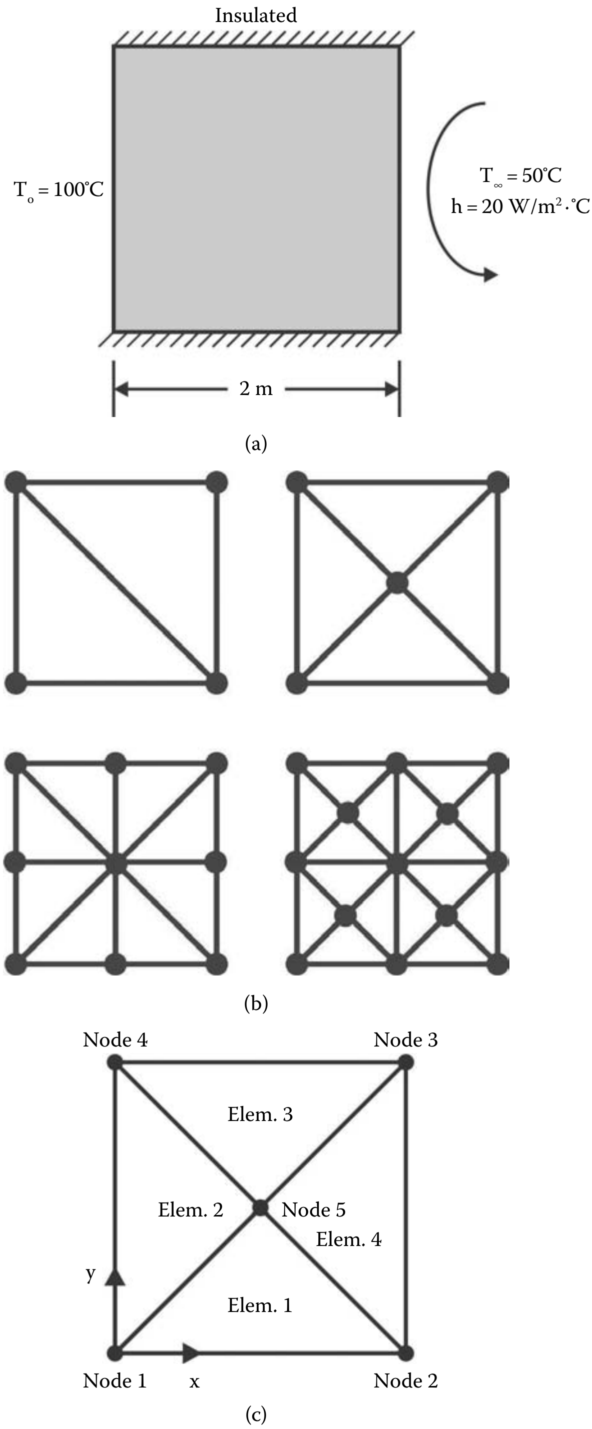


FIGURE 5.6 (a) The two-dimensional body subjected to convection and fixed temperature boundary conditions, (b) suggested finite element meshes, and (c) the elements distribution.

No convection is applied, and therefore, the convection term in Equation 5.62 is equal to zero. Finally, the [K] matrix for element 1 is calculated using Equation 5.62:

$$\left[K^{(1)}\right]=\begin{array}{c} \\ \\ \\ \\ \end{array}\begin{array}{cc} \begin{array}{ccc} \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{5}} \end{array} & \\ \begin{bmatrix} 12.5 & 0 & -12.5 \\ 0 & 12.5 & -12.5 \\ -12.5 & -12.5 & 25 \end{bmatrix} & \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{5}} \end{array} \end{array}$$

For element 2, the [B], [D], and [K] matrices are formulated. The name of node 1 is (i) and its coordinate is (0,0), the name of node 5 is (j) and its coordinate is (1,1), and the name of node 4 is (m) and its coordinate is (0,2). The values of β's and γ's are required for the [B] matrix, and calculated using Equation 5.59, and A is the area of the element. We then have

$$\beta_i = y_j - y_m = 1 - 2 = -1$$

$$\beta_j = y_m - y_i = 2 - 0 = 2$$

$$\beta_m = y_i - y_j = 0 - 1 = -1$$

$$\gamma_i = x_m - x_j = 0 - 1 = -1$$

$$\gamma_j = x_i - x_m = 0 - 0 = 0$$

$$\gamma_m = x_j - x_i = 1 - 0 = 1$$

$$A = 0.5(2)1 = 1\ m^2$$

Then, the [B] matrix is formulated using Equation 5.60:

$$[B]=\frac{1}{2}\begin{bmatrix} -1 & 2 & -1 \\ -1 & 0 & 1 \end{bmatrix}$$

and the [D] matrix is the same as for element 1. No convection is applied, and therefore, the convection term in Equation 5.62 is equal to zero. Finally, the [K] matrix for element 2 is calculated using Equation 5.62:

$$\left[K^{(2)}\right]=\begin{array}{cc} \begin{array}{ccc} \underline{\mathbf{1}} & \underline{\mathbf{5}} & \underline{\mathbf{4}} \end{array} & \\ \begin{bmatrix} 12.5 & -12.5 & 0 \\ -12.5 & 25 & -12.5 \\ 0 & -12.5 & 12.5 \end{bmatrix} & \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{5}} \\ \underline{\mathbf{4}} \end{array} \end{array}$$

For element 3, the [B], [D], and [K] matrices are formulated. The name of node 4 is (i) and its coordinate is (0,2), the name of node 5 is (j) and its coordinate is (1,1), and the name of node 3 is (m) and its coordinate is (2,0). The values of β's and γ's are required for

the [B] matrix, and are calculated using Equation 5.59, and A is the area of the element. We have

$\beta_i = y_j - y_m = 1 - 2 = -1$

$\beta_j = y_m - y_i = 2 - 2 = 0$

$\beta_m = y_i - y_j = 2 - 1 = 1$

$\gamma_i = x_m - x_j = 2 - 1 = 1$

$\gamma_j = x_i - x_m = 0 - 2 = -2$

$\gamma_m = x_j - x_i = 1 - 0 = 1$

$A = 0.5(2)1 = 1\ m^2$

Then, the [B] matrix is formulated using Equation 5.60:

$$[B] = \frac{1}{2}\begin{bmatrix} -1 & 0 & 1 \\ 1 & -2 & 1 \end{bmatrix}$$

and the [D] matrix is the same as for element 1. No convection is applied, and therefore, the convection term in Equation 5.62 is equal to zero. Finally, the [K] matrix for element 3 is calculated using Equation 5.62:

$$\left[K^{(3)}\right] = \begin{array}{c} \begin{array}{ccc} \underline{\mathbf{4}} & \underline{\mathbf{5}} & \underline{\mathbf{3}} \end{array} \\ \begin{bmatrix} 12.5 & -12.5 & 0 \\ -12.5 & 25 & -12.5 \\ 0 & -12.5 & 12.5 \end{bmatrix} \end{array} \begin{array}{c} \\ \underline{\mathbf{4}} \\ \underline{\mathbf{5}} \\ \underline{\mathbf{3}} \end{array}$$

For element 4, the [B], [D], and [K] matrices are formulated. The name of node 2 is (i) and its coordinate is (2,0), the name of node 3 is (j) and its coordinate is (0,2), and the name of node 5 is (m) and its coordinate is (2,0). The values of β's and γ's are required for the [B] matrix, and calculated using Equation 5.59, and A is area of the element. We have

$\beta_i = y_j - y_m = 2 - 1 = 1$

$\beta_j = y_m - y_i = 1 - 0 = 1$

$\beta_m = y_i - y_j = 0 - 2 = -2$

$\gamma_i = x_m - x_j = 1 - 2 = -1$

$\gamma_j = x_i - x_m = 2 - 1 = 1$

$\gamma_m = x_j - x_i = 2 - 0 = 2$

$A = 0.5(2)1 = 1\ m^2$

Then, the [B] matrix is formulated using Equation 5.60:

$$[B] = \frac{1}{2}\begin{bmatrix} 1 & 1 & -2 \\ -1 & 1 & 2 \end{bmatrix}$$

and the [D] matrix is the same as for element 1. Convection is applied on the side i–j of the element, and therefore, the convection term in Equation 5.62 is not equal to zero. Finally, the [K] matrix for element 4 is calculated using Equation 5.62:

$$\begin{array}{ccc} \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{5}} \end{array} \qquad\qquad \begin{array}{ccc} \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{5}} \end{array}$$

$$\begin{bmatrix} 12.5 & -12.5 & 0 \\ -12.5 & 25 & -12.5 \\ 0 & -12.5 & 12.5 \end{bmatrix} \begin{array}{c} \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{5}} \end{array} + \begin{bmatrix} 13.3 & 66.7 & 0 \\ 6.67 & 13.3 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{array}{c} \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{5}} \end{array}$$

Finally, the [K] matrix for the entire body is obtained by adding [K] matrices for elements 1, 2, 3, and 4 using the expression (5.38), and the result is

$$\begin{array}{ccccc} \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{4}} & \underline{\mathbf{5}} \end{array}$$

$$[K] = \sum_{i=1}^{5} \left[K^{(i)}\right] = \begin{bmatrix} 25 & 0 & 0 & 0 & -25 \\ 0 & 38.33 & 6.67 & 0 & -25 \\ 0 & 6.67 & 38.33 & 0 & -25 \\ 0 & 0 & 0 & 25 & -25 \\ -25 & -25 & -25 & -25 & 100 \end{bmatrix} \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{4}} \\ \underline{\mathbf{5}} \end{array}$$

Convection is applied on the side i–j of the element 4, the convection force vector is

$$\{F_c\} = \frac{hT_\infty L_{ij}}{2}\begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} = \begin{Bmatrix} 1000 \\ 1000 \\ 0 \end{Bmatrix}$$

Using total conductive matrix, [K], and nodal force in expression (5.36) yields

$$[K]\{T\} = \{F\}$$

$$\begin{bmatrix} 25 & 0 & 0 & 0 & -25 \\ 0 & 38.33 & 6.67 & 0 & -25 \\ 0 & 6.67 & 38.33 & 0 & -25 \\ 0 & 0 & 0 & 25 & -25 \\ -25 & -25 & -25 & -25 & 100 \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \\ T_5 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ 1000 \\ 1000 \\ Q_4 \\ 0 \end{Bmatrix}$$

where Q_1 and Q_4 are heat flow at nodes 1 and 4, respectively, which are required to maintain the temperature at the nodes. The temperature of nodes 1 and 4 are known, and therefore, the above equations should be modified. In nodal force vector, Q_1 and Q_4 are replaced by 50°C, and consequently, equations 1 and 4 are modified. The conductive matrix and nodal thermal forces and temperatures are expressed as

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 38.33 & 6.67 & 0 & -25 \\ 0 & 6.67 & 38.33 & 0 & -25 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & -25 & -25 & 0 & 100 \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \\ T_5 \end{Bmatrix} = \begin{Bmatrix} 100 \\ 1000 \\ 1000 \\ 100 \\ 5000 \end{Bmatrix}$$

There are three equations and three unknowns. The T_5 is the temperature at the center of the body, which is 84.62°C. The temperature at nodes 2 and 3 are equal to 69.33°C.

5.7 Thermal analysis of fin and chip using ANSYS

The fin shown in Figure 5.7 is used to manage the temperature of an electronic chip that generates heat. Heat is generated in the heat source within the chip, and its value is 25 Watts. The heat transfer process is steady. Heat convection is applied at the entire external surface, $h = 15\ W/m^2 \cdot °C$ and $T_o = 20°C$, while the bottom surface of the chip is well insulated. Determine the maximum and average temperatures at the bottom surface of the chip. The thermal conductivities of the used materials: $k_{fin} = 110\ W/m \cdot °C$, $k_{chip} = 1.25\ W/m \cdot °C$, and $k_{hs} = 2.5\ W/m \cdot °C$.

Double click on the Mechanical APDL Product Launcher icon

This example is limited to thermal analysis. Hence, select Thermal in the preferences. The Solid element is used, and its shape is Triangle with six nodes.

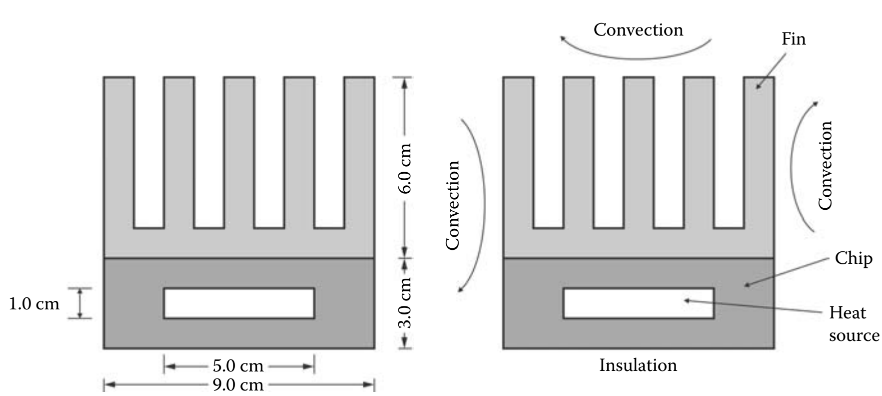
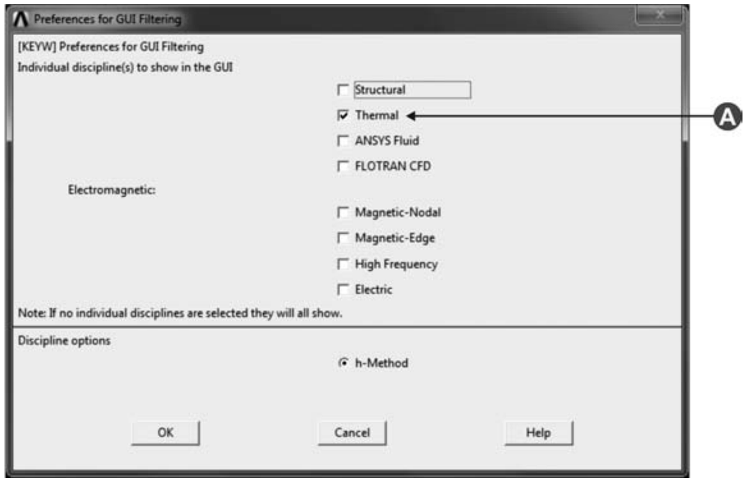


FIGURE 5.7 Heated chip with fin, and the boundary conditions.

Main Menu > Preferences

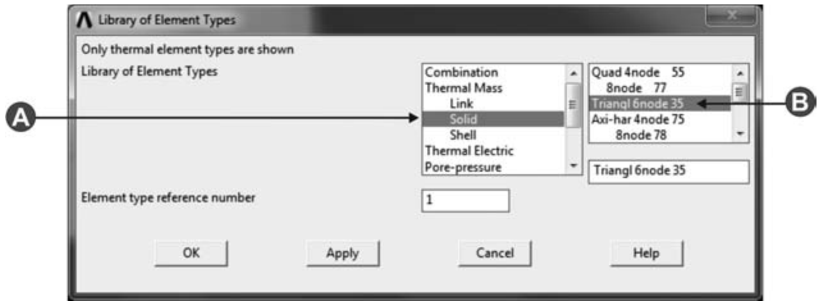


A select Thermal

OK

Main Menu > Preprocessor > Element Types > Add/Edit/Delete

Add...



A select Solid

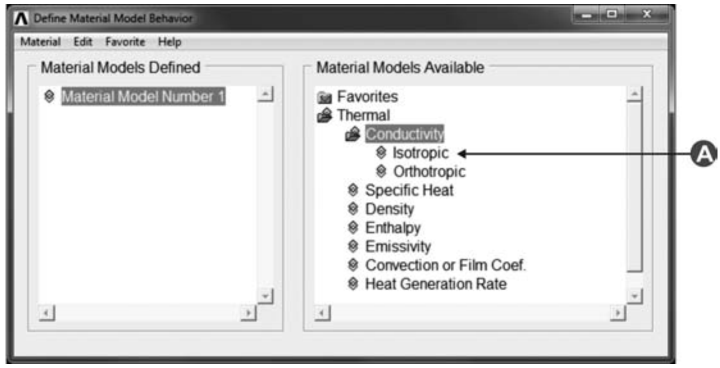
B select Triangl 6node 35

OK

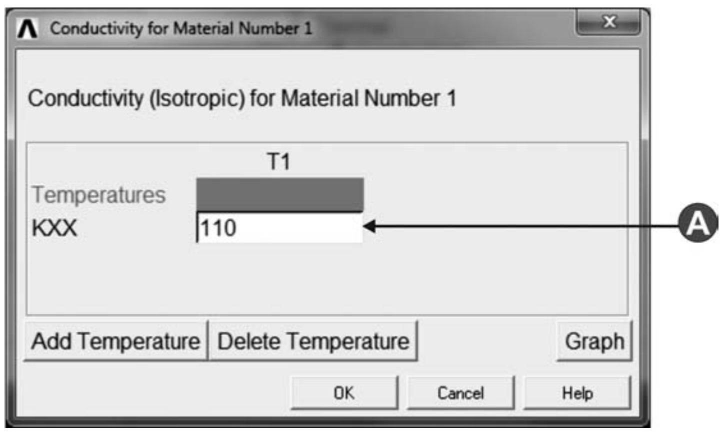
Close

Only the thermal conductivity is required to solve the problem. Note that the thermal conductivities of the fin, chip, and heater are different. By default, all areas will be assigned to material number 1. In this problem, 1 is the material number of the fin, 2 for the chip, and 3 for the heater.

Main Menu > Preprocessor > Material Props > Material Models



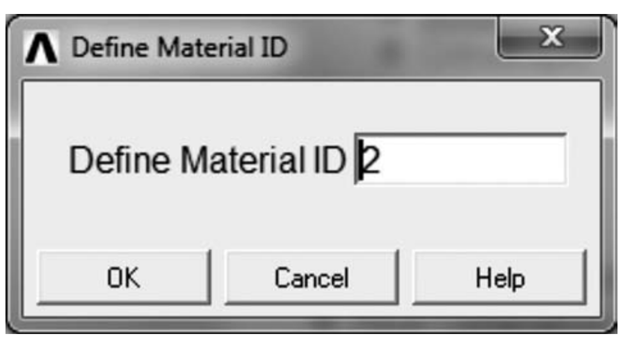
A click on Thermal > Conductivity > Isotropic



A type 110 in KXX

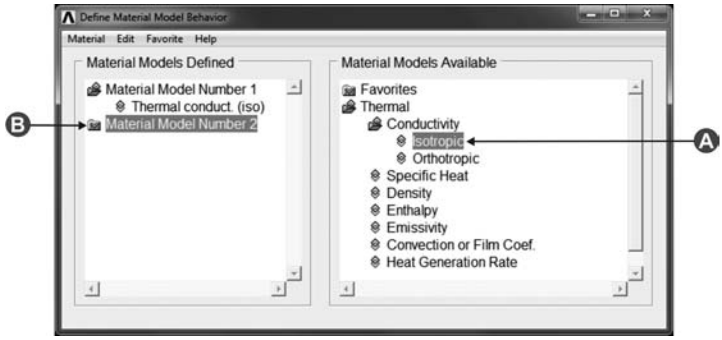
OK

In the Define Material Models Behavior: Material > New Model



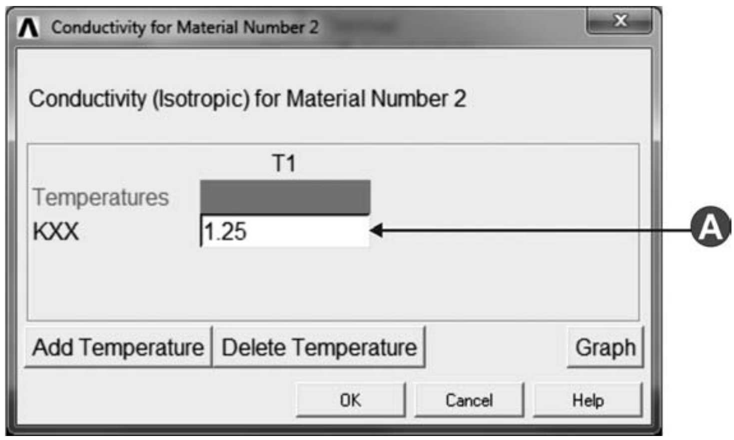
Make sure that the number 2 is in Define Material ID

OK



A select Material Model Number 2

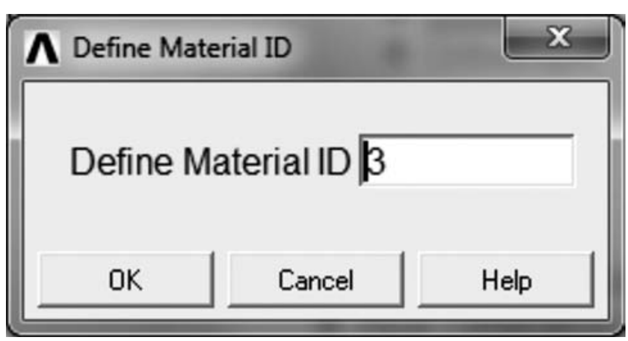
B click on Thermal > Conductivity > Isotropic



A type 1.25 in KXX

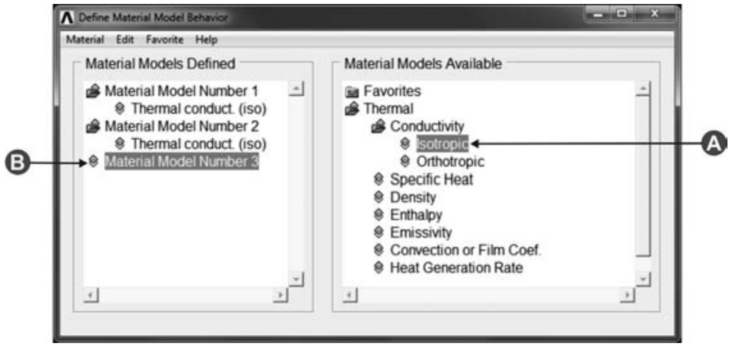
OK

In the Define Material Models Behavior: Material > New Model



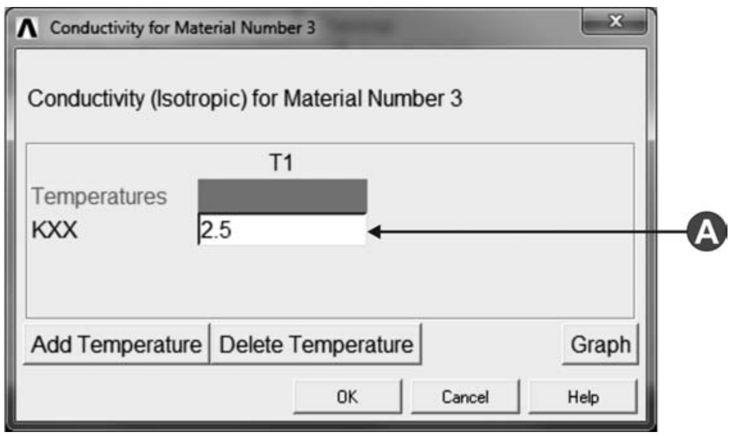
Make sure that the number 3 is in Define Material ID

OK



A select Material Model Number 3

B click on Thermal > Conductivity > Isotropic



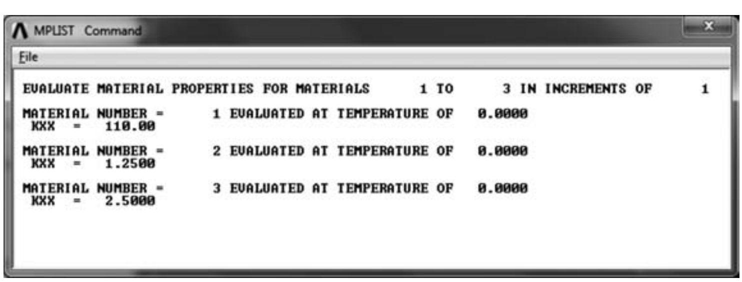
A type 2.5 in KXX

$\boxed{\textbf{OK}}$

Close the Material Model Behavior window

The thermal conductivities are listed using list properties in utility menu.

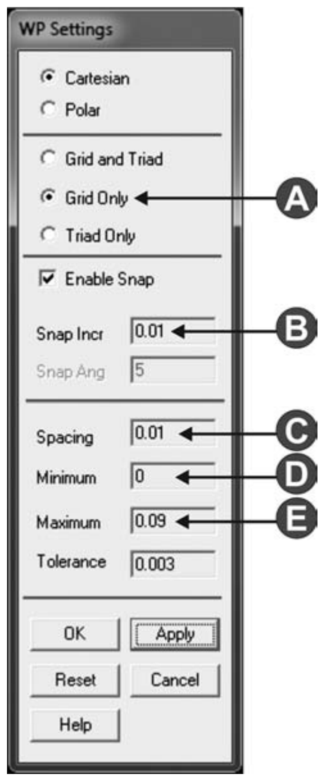
Utility Menu > List > Properties > All Materials



The geometry is created using the ANSYS graphics. Setting up the workspace is done using the WP Settings. Snap is enabled to allow the mouse-click on the ANSYS graphics with an increment. Spacing is the distance between the vertical or horizontal grids. The size of the grids is specified in the Minimum and Maximum. The space is divided into squares with side length of 0.01 m. Both the total width and the height of the grids are 0.09 m. This setup makes the modeling easy by creating keypoints, lines, and areas in the ANSYS graphics. The grids should be first activated by selecting Display Working Plane in the Utility Menu.

ANSYS Utility Menu > WorkPlane > Display Working Plane

Utility Menu > WorkPlane > WP Settings …



A select Grid Only

B type 0.01 in Snap Incr

C type 0.01 in Spacing

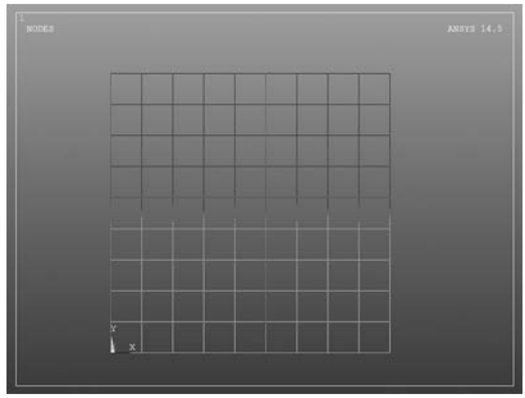
D type 0 in Minimum

E type 0.09 in Maximum

OK

ANSYS Utility Menu > PlotCtrls > Pan-Zoom-Rotate …

Click on zoom in and out, until the ANSYS graphics show all the grids.



ANSYS graphics show the grids

The keypoints are created first, and then lines. Finally, fin, heater, and chip areas are created. The thermal conductivities of the chip, heater, and fin are assigned using element attribute in the meshing tools window.

ANSYS Main Menu > Preprocessor > Modeling > Create > Keypoints > On Working Plane

Click on the ANSYS graphics window at the location of the keypoints, as shown in the following figure.

$\boxed{\textbf{OK}}$

Now, the grids are deactivated by selecting Display Working Plane in Utility Menu.

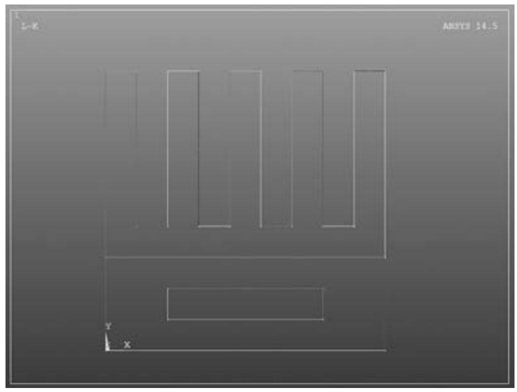
ANSYS Utility Menu > WorkPlane > Display Working Plane

ANSYS graphics show the keypoints

ANSYS Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Click on two keypoints to create a line, and continue to create all lines. The created lines are shown in the following figure.

OK



ANSYS graphics show the lines

ANSYS Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines

Click on chip lines to create a large rectangle area, and then in Create Area by Lines window, click on

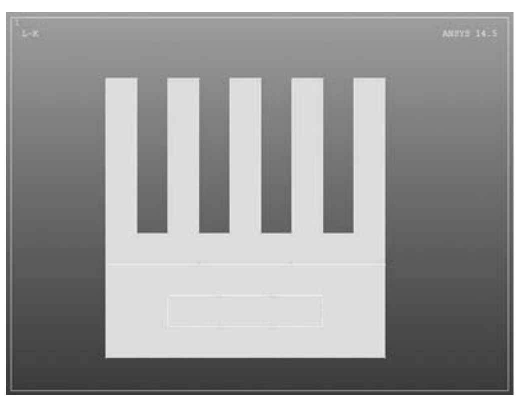
Apply

Click on heater lines to create a small rectangle area, and then in Create Area by Lines window, click on

Apply

Click on fin lines to create the fin area, and then in Create Area by Lines window, click on

OK



ANSYS graphics show the areas

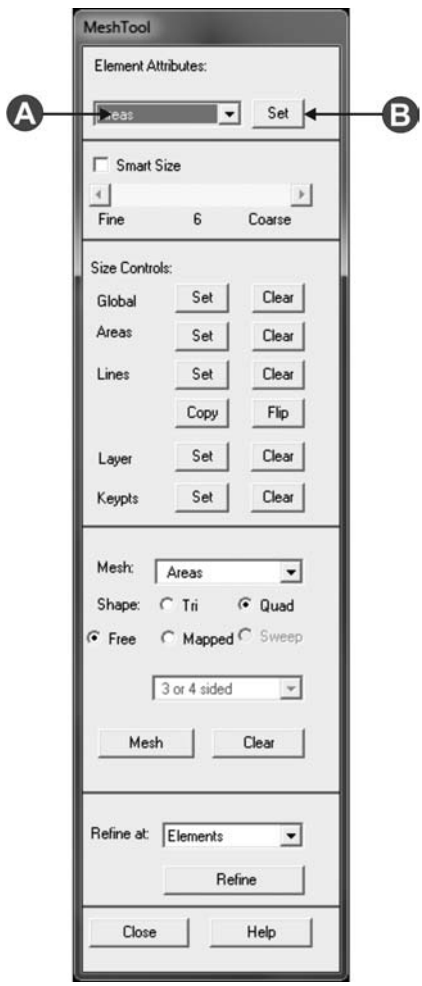
The area between the heater and the chip is separated, and the area of the heater is just over the area of the chip. The two areas should be connected by common lines. Overlap merges the two areas, and the boundary lines of the heater are sheared with the chip.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

In Overlap Areas window, click on

Pick All

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Area

B click on Set

Click the chip's area only

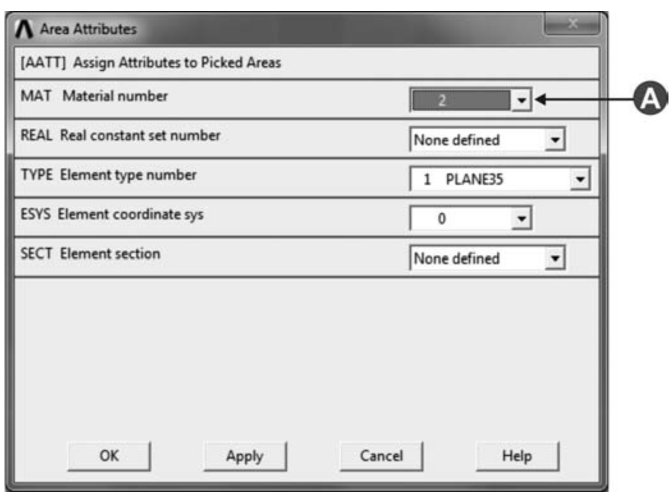
The above window will show up. Because the chip and heater have the same centroid, ANSYS is inquiring about which area should be selected. The chip or heater selection can be switched by clicking on **Prev** and **Next** buttons in the Multiple_Entities window. Make sure that the chip's area is highlighted. In Multiple_Entities window, click on

OK

In Area Attributes window, click on

OK

By selecting number 2, the properties of number 2 in the material model are assigned to the chip. The fin by default has the properties of number 1 in the material model.



A select 2 in Material number

Apply

Click the heater area only

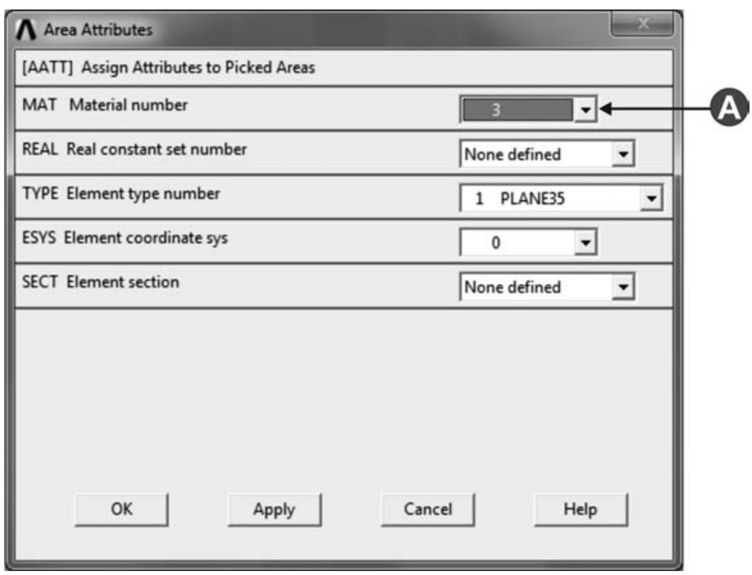
The chip or heater selection can be switched by clicking on **Prev** and **Next** buttons in the Multiple_Entities window. Make sure that the heater's area is highlighted.

OK

In Area Attributes window, click on

OK

By selecting number 3, the properties of number 3 in the material model are assigned to the heater.



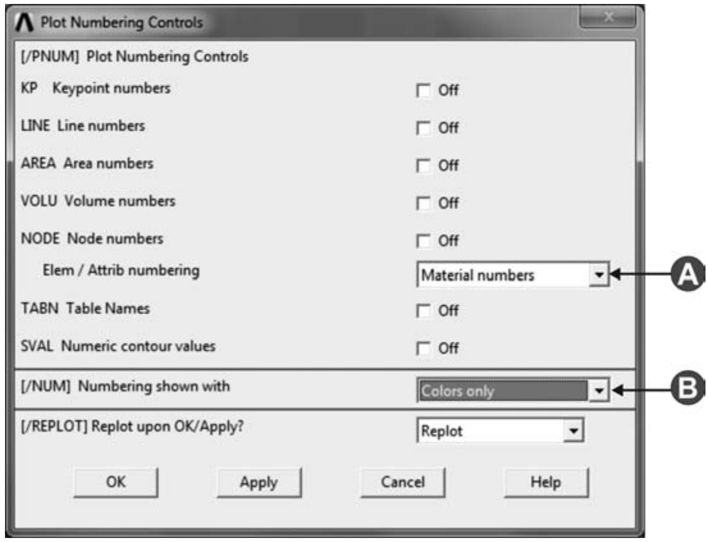
A select 3 in Material number

OK

To ensure that thermal conductivity of the fin and chip are assigned correctly, the components are colored according to their material number.

The material coloring has no effect on the solution. The geometry is meshed with triangular six-node elements. A free mesh is generated using the smart mesh option. The mesh refinement is 1.

Utility Menu > PlotCtrls > Numbering …

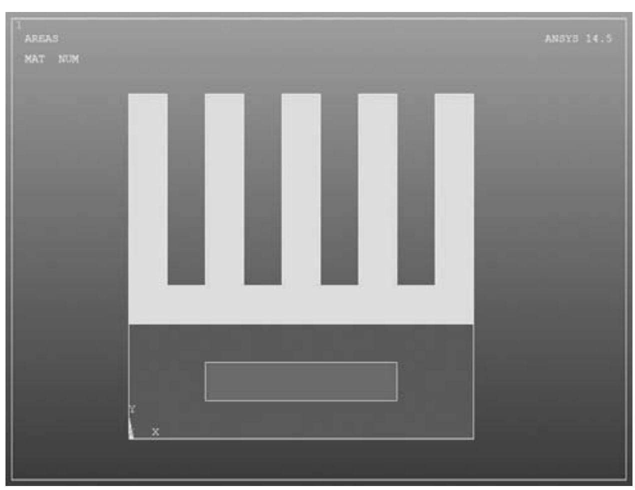


A select Material numbers

B select Colors only

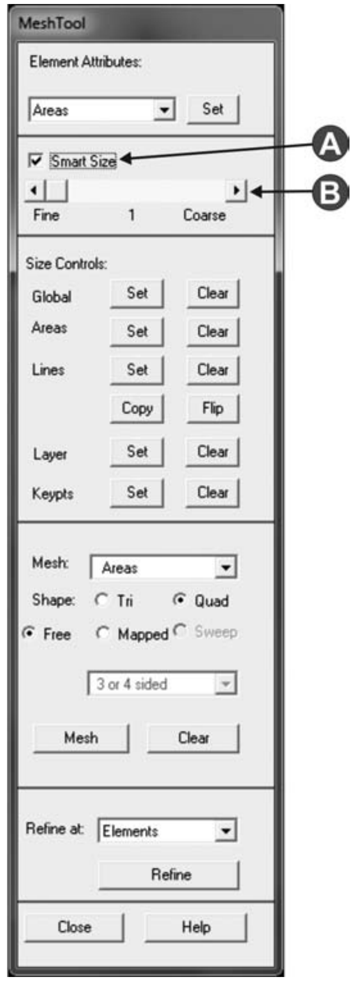
OK

Utility Menu > Plot > Areas



ANSYS graphics show the heater, fin, and chip with different colors

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Smart Size

B set the level to 1

Mesh

In Mesh Areas window, click on

Pick All

Close the Mesh Tool window

Close

Boundary conditions are applied in the solution task. A volumetric heat generation is applied to the heater only. The convection boundary condition is applied to the fin surface and the vertical sides of the chip. The bottom surface of the chip is well insulated, and the

zero heat flux simulates the insulation boundary condition. However, if no boundary condition is specified at any external surface, ANSYS will consider it as an insulated boundary condition. No boundary conditions are applied at a common line between the chip and fin, and the chip and heater. The heat generation must be per unit volume. The applied heat generation is divided by the area of the heater because the problem is two dimensional. The heater volumetric heat generation is calculated as

$$Q = \frac{25}{0.05 \times 0.01 \times 1} = 50{,}000\,\mathrm{W/m^3}$$

Main Menu > Solution > Define Loads > Apply > Thermal > Heat Generat > On Areas

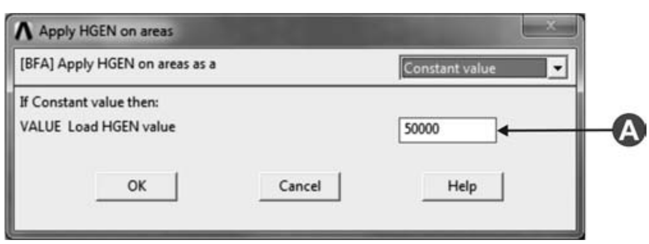
Click on the heater area, where heat generation is applied.

Because the chip and heater are overlapped, and having the same centroid, ANSYS is inquiring about which area should be selected. The chip or heater selection can be switched by clicking on **Prev** and **Next** buttons in the Multiple_Entities window. Make sure that the heater's area is highlighted.

OK

In Apply HGEN on areas window, click on

OK



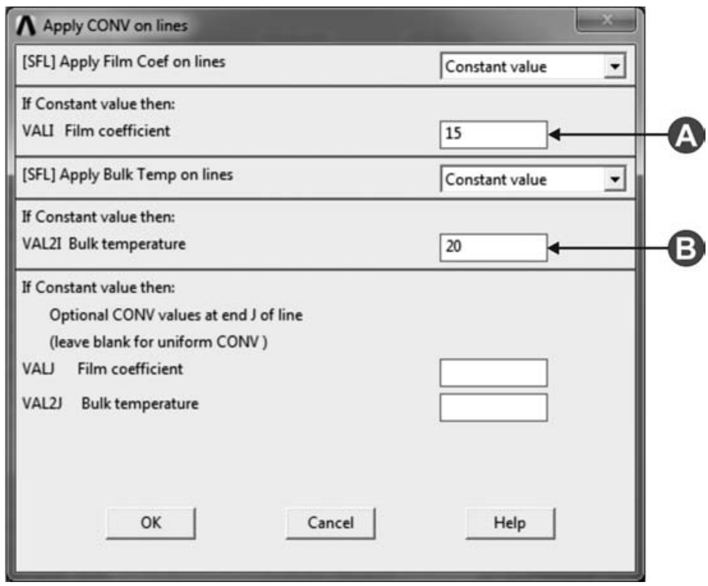
A type 50000 in Load HGEN value

OK

Main Menu > Solution > Define Load > Apply > Thermal > Convection > On Lines

Click on fin external surfaces and two vertical surfaces of the chip where the convection boundary condition is applied. Then in Apply CONV on lines window, click on

OK



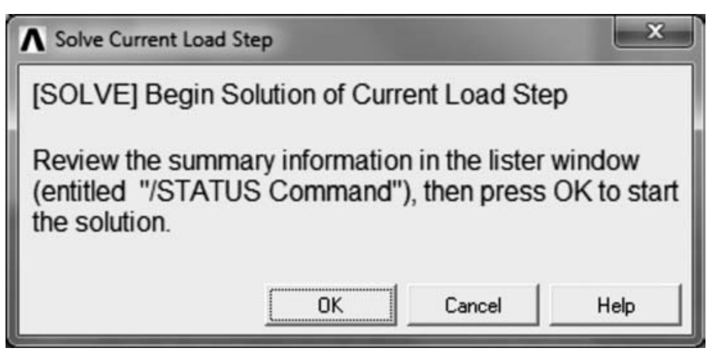
A type 15 in Film coefficient

B type 20 in Bulk temperature

OK

The preprocessor and solution tasks are completed.

Main Menu > Solution > Solve > Current LS

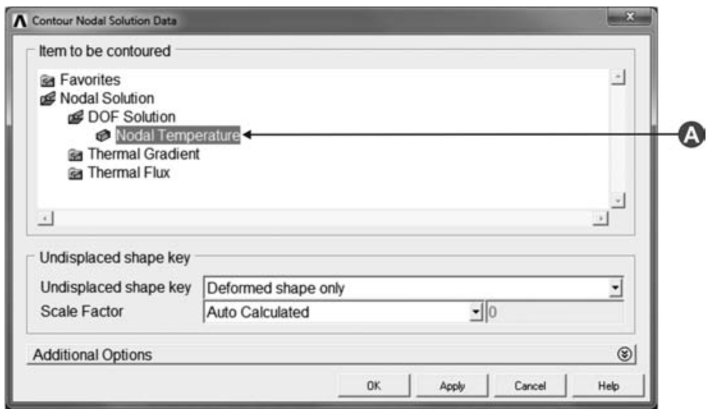


OK

Close

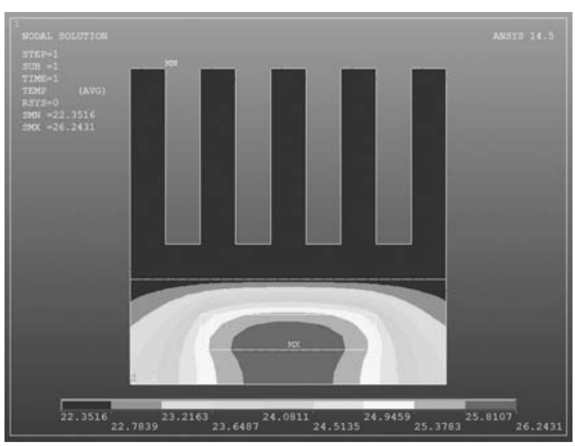
The solution is successfully completed, and no error messages are posted. In the postprocessor, the temperature contours should be carefully inspected to ensure that the boundary conditions are applied correctly.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > DOF Solution > Nodal Temperature

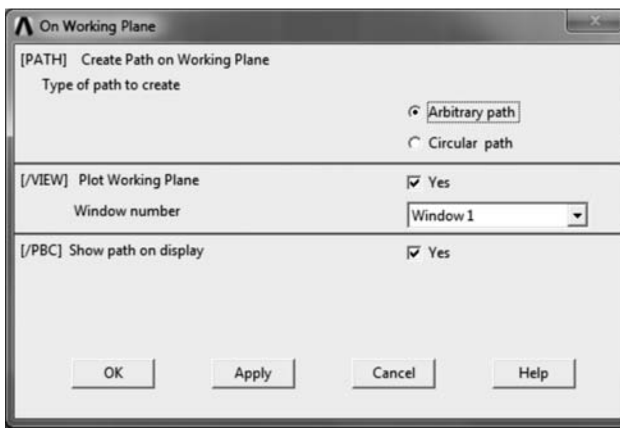
OK



ANSYS graphics show the temperature contours

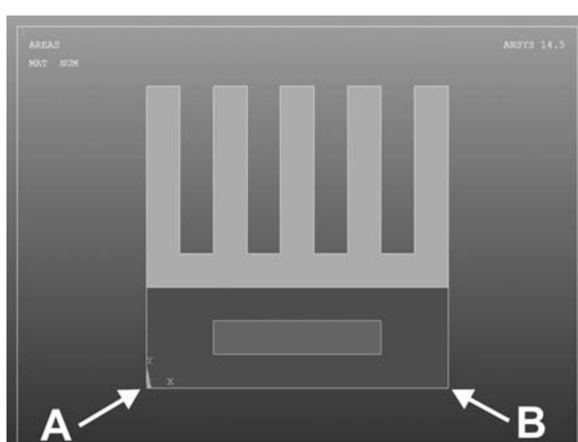
The temperature contours indicate that the maximum temperature is located at the bottom surface of the chip, and is equal to 26.431°C. The default number of contours is 9, and this number can be increased to a higher value for better data analysis. First, the graphics device must be changed to win32C, and then the number of contours can be increased up to 114 contours. Notice that increasing the number of contours does not mean that the accuracy of the result is improved. The vector plot showing the heat flow from the chip to the fin can be presented. The red arrow is for high value of the heat flux. The average temperature at the base of the chip is calculated using the path operation. To create a path, there are two options: Arbitrary and Circular paths. The Arbitrary path can be made from straight-line segments by clicking on the ANSYS graphics, and the grids should be enabled. For this example, the Arbitrary path is utilized.

Main Menu > General Postproc > Path Operations > Define Path > On Working Plane



OK

Click on the ANSYS graphics window at the right and left bottom corners of the chip, as shown in the following figure.

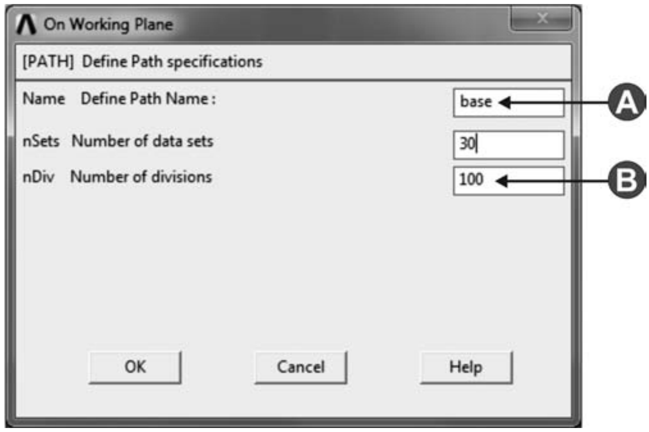


A click on left bottom corner of the chip

B click on right bottom corner of the chip

Then in On Working Plane window, click on

OK



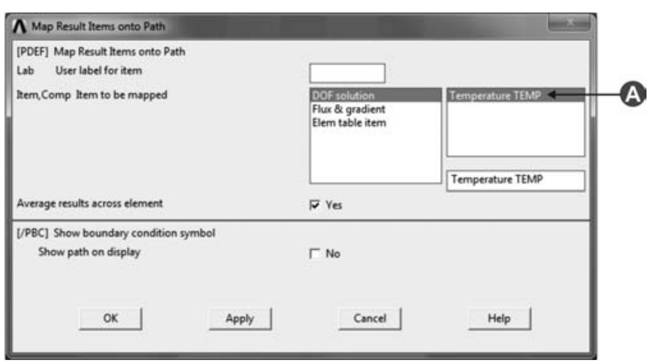
A name the path as base

B type 100 in Number of divisions

OK

The name of the path is optional. The number of the data set is the maximum number of field variables. The number of division is 20 by default, and increasing this number to 100 will produce a smoother plot. Next, the field variable is assigned to the path for plotting. This can be accomplished by using the Map onto Path in the path operation. Only one variable can be selected. For this example, the temperature is selected.

Main Menu > General Postproc > Path Operations > Map onto Path

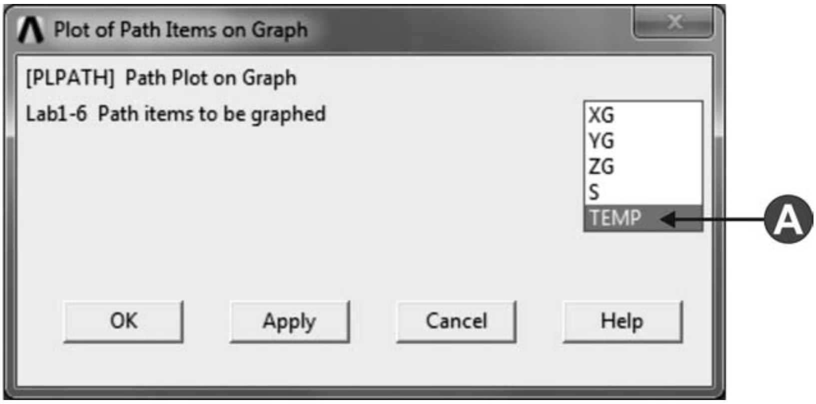


A select Temperature TEMP

OK

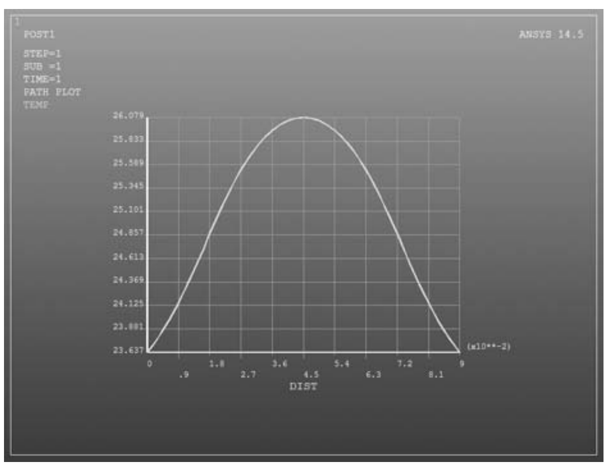
Now, the temperature is ready to be plotted. In the Plot path item, there are two options. The temperature can either be plotted or listed. The list results can be exported to another software, such as EXCEL.

Main Menu > General Postproc > Path Operations > Plot path Items > On Graph



A select TEMP

OK

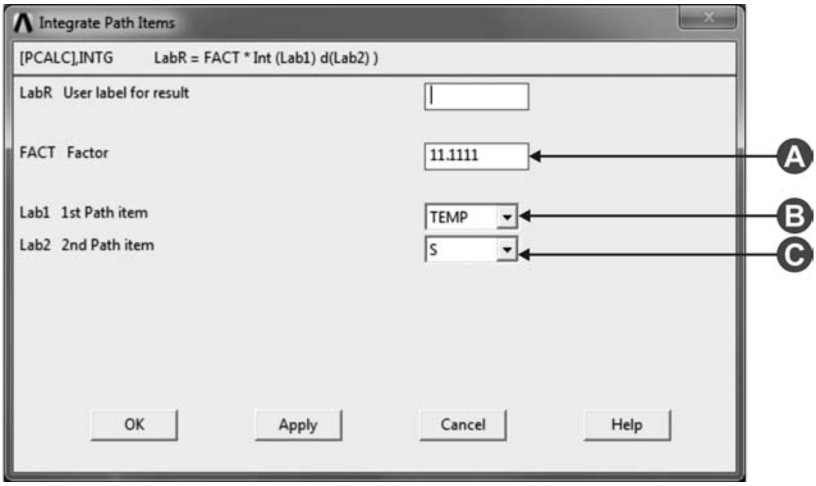


ANSYS graphics show the temperature distribution at the base

The temperature at the exit is perfectly parabolic due to the symmetry in the problem. The above graph indicates that the maximum temperature at the bottom surface of the chip is 26.079°C. The average temperature at the base can be determined using the integration in the path operation. The value of the integration must be divided by the path length to get the average value of the variable, and the path length

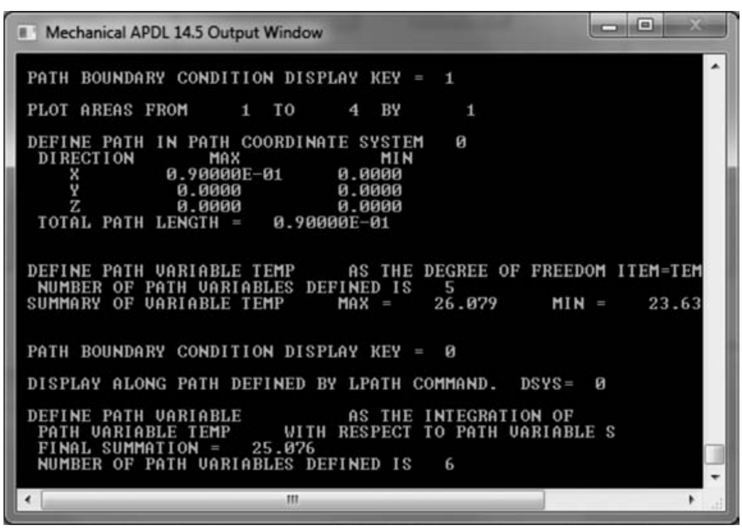
is 0.09 m. Hence, number 11.111 is entered in the FACT and will be multiplied by the integration result. Selecting S in the Lab2 means that the integration is performed along the path.

Main Menu > General Postproc > Path Operation > Integrate



A type 11.1111 in FACT Factor
B select TEMP in Lab1 1st Path item
C select S in Lab2 2nd Path item

OK



The ANSYS output window shows the value of the average temperature at the base which is 25.076°C.

5.8 Finite element method for transient heat transfer

Finite element formulation for transient conduction heat transfer is presented in this section. The conduction energy equation in per unit volume and in Cartesian coordinate can be expressed as

$$\frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_z \frac{\partial T}{\partial z}\right)+\dot{q}=\rho C_p \frac{\partial T}{\partial t} \tag{5.72}$$

where C_p is the specific heat of the material. The finite element method is an efficient way to solve transient conduction problems. The heat transfer solution can be used to estimate the heat flow at the system's boundary, or to determine the temperature distribution for thermal-stress analysis. In this section, the variation formulation is used to obtain the conductive equation, and it is accomplished by minimizing the following potential function:

$$I=\frac{1}{2}\int_V\left[k_x\left(\frac{\partial T}{\partial x}\right)^2\right]dV-\int_V\left(Q-\rho C_p\frac{\partial T}{\partial t}\right)T\,dV-\int_S qT\,dS+\frac{1}{2}\int_S h(T-T_\infty)^2\,dS \tag{5.73}$$

The second term in Equation 5.73 adds an additional contribution for transient as follows:

$$\Omega_Q=-\int_V\left(Q-\rho C_p\dot{T}\right)T\,dV \tag{5.74}$$

where $\dot{T}=\partial T/\partial t$. The temperature at nodes can be expressed in terms of the shape functions as follows:

$$\{T\}=[N_i]\{\dot{T}_i\} \tag{5.75}$$

Substituting temperature function (5.75) into the transient term in the heat conduction equation (5.74) yields

$$\Omega_Q=-\int_V\left([N_i]\{T_i\}Q-\rho\,C_p[N_i]\{T_i\}[N_i]\{\dot{T}_i\}\right)dV \tag{5.76}$$

Minimizing the transient term with respect to the nodal temperature is as follows:

$$\frac{\partial\Omega_Q}{\partial\{T_i\}}=-\int_V\left([N_i]Q-\rho\,C_p[N_i][N_i]\{\dot{T}_i\}\right)dV \tag{5.77}$$

The second term in Equation 5.77 is an additional term that should be added to the conductive matrix, and it is expressed as

$$\int_V \left(\rho C_p [N_i][N_i]\right) dV \left\{\dot{T}_i\right\} = [m]\left\{\dot{T}_i\right\} \quad (5.78)$$

where [m] is the element mass matrix. Equation 5.78 is added to the conductive matrix to account for transient heat transfer as

$$[K]\{T\} + [m]\left\{\dot{T}_i\right\} = \{F\} \quad (5.79)$$

where

$$[K] = \int_V [B]^T [D][B]\, dV + \int_S h[N]^T [N]\, dS \quad (5.80)$$

and

$$\{F\} = \{F_Q\} + \{F_q\} + \{F_c\} \quad (5.81)$$

$\{F_Q\}$ is volumetric heat generation in the element, $\{F_q\}$ is applied heat flux on the external surface of the element, and $\{F_c\}$ is applied convection on the external surface of the element. The integration is over the surface in which convection or heat flux is applied. The global conductive matrix can be obtained by assembling the conductive matrixes as follows:

$$[K] = \sum_{e=1}^{N} \left[K^{(e)}\right] \quad (5.82)$$

and the nodal heat force are also assembled to form a global one as follows:

$$\{F\} = \sum_{e=1}^{N} \left\{F^{(e)}\right\} \quad (5.83)$$

Also, the global temperature vector can be obtained as

$$\{T\} = \sum_{e=1}^{N} \left\{T^{(e)}\right\} \quad (5.84)$$

5.9 Unsteady thermal analyses of a masonry brick using ANSYS

The cross-sectional area of a masonry brick, as shown in Figure 5.8a, is made of cement and hollow cylinders. The purpose of the numerical

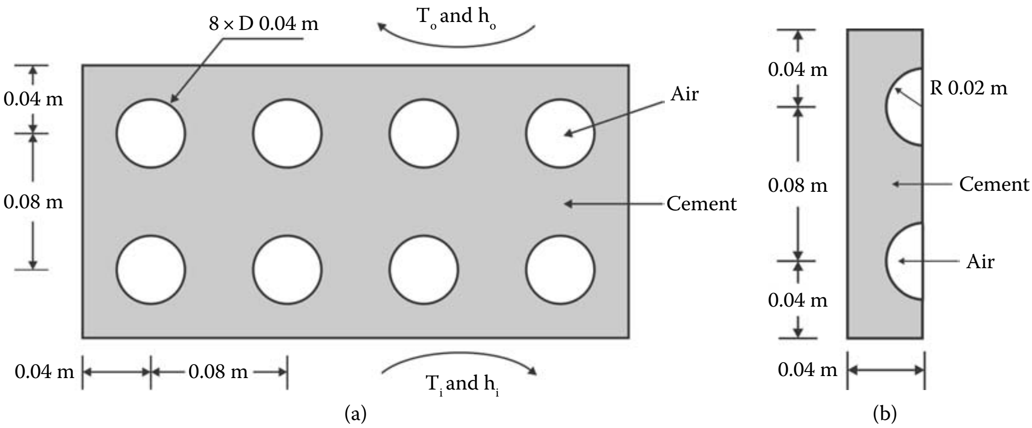


FIGURE 5.8 (a) Masonry brick with boundary conditions. (b) The considered geometry for numerical simulation.

simulation is to study the heat flow from an outdoor to the indoor space. The initial temperature of the brick is 25°C.

Property	Air	Cement
Density (kg/m^3)	1.125	2500
Conductivity (W/m·K)	0.025	0.8
Specific heat (J/kg·K)	1005	750

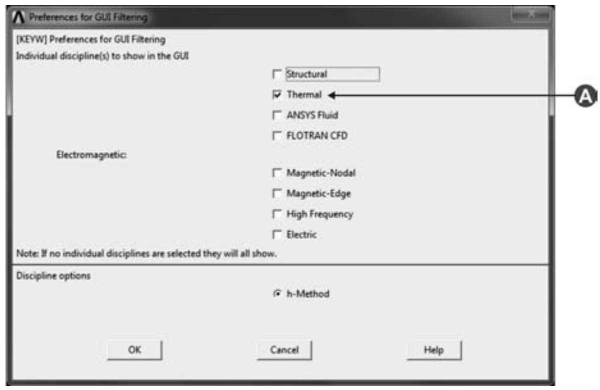
At the indoor surface of the brick, the convection boundary condition is applied with time-independent temperature and heat transfer coefficient, T_i = 20°C and h_i = 10 W/m^2·°C. At the outdoor surface, the heat transfer coefficient is time independent, h_o = 25 W/m^2·°C, but the temperature is time dependent. The following expression is used to simulate the outdoor temperature variation:

$$T(t) = 35 + 5\,\mathrm{Sin}\left(\frac{2\pi}{86400}t\right)$$

To ensure having a periodic condition, the simulation is kept running for 3 days, and the results for the last day are presented. Since there is symmetry in the geometry and boundary conditions, only one-eighth of the geometry is considered. Figure 5.8b shows the considered geometry for numerical simulation. Outdoor convection is applied at the upper lateral line, and indoor convection is applied at the lower lateral line. The two vertical lines are well insulated, which show temperature history at the indoor and outdoor surfaces of the brick for 1 day.

Double click on the Mechanical APDL Product Launcher icon

Main Menu > Preferences

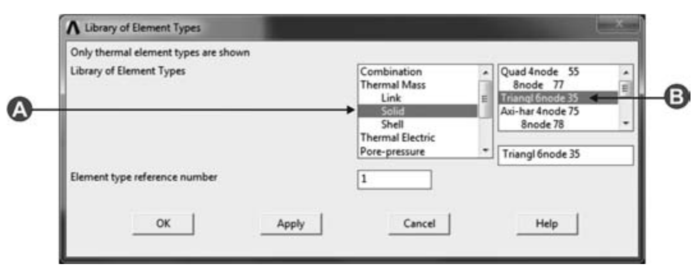


A select Thermal

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add…



A select Solid

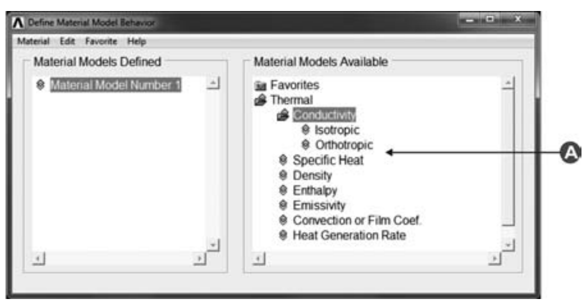
B select Triangl 6node 35

OK

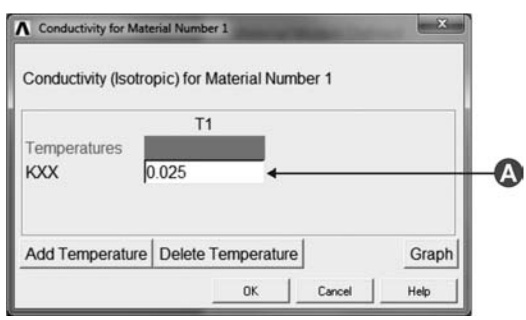
Close

For the material properties, thermal conductivity, specific heat, and density are required to solve the problem because the problem is unsteady. Note that the material properties of the air and cement are different. By default, all areas will be assigned to material number 1. In this problem, the material number of the air is 1, and 2 for the cement.

Main Menu > Preprocessor > Material Props > Material Models



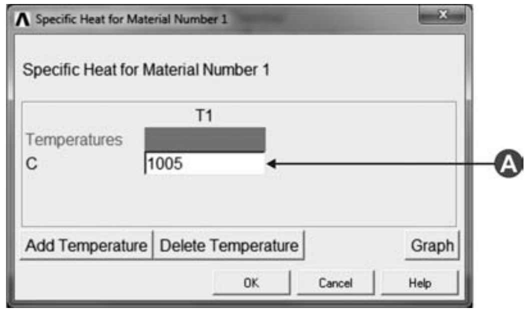
A click on Thermal > Conductivity > Isotropic



A type 0.025 in KXX

OK

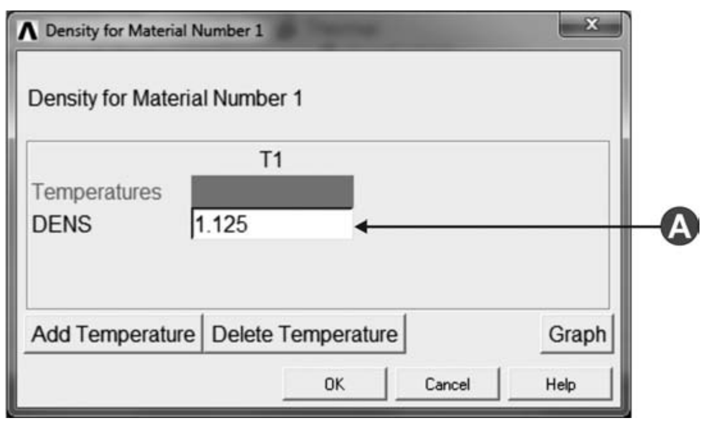
Click on Thermal > Specific Heat



A type 1005 in C

OK

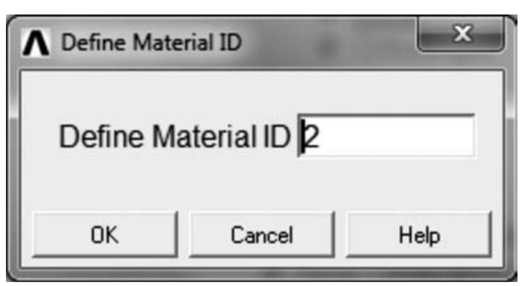
Click on Thermal > Density



A type 1.125 in DENS

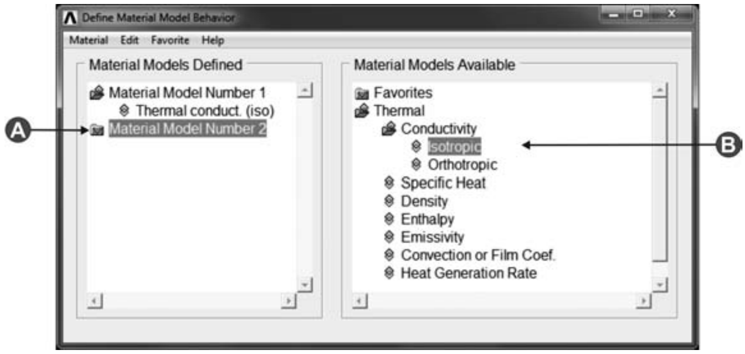
OK

In the Define Material Models Behavior: Material > New Model



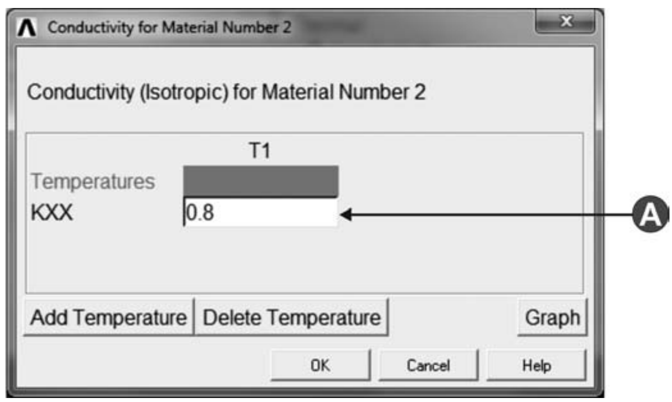
Make sure that the number 2 is in Define Material ID

OK



A select Material Model Number 2

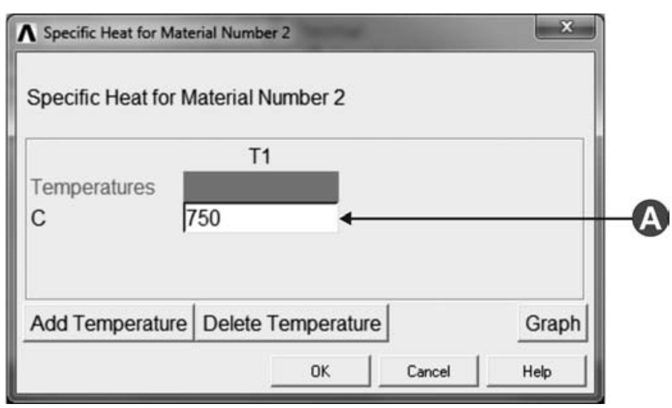
B click on Thermal > Conductivity > Isotropic



A type 0.8 in KXX

OK

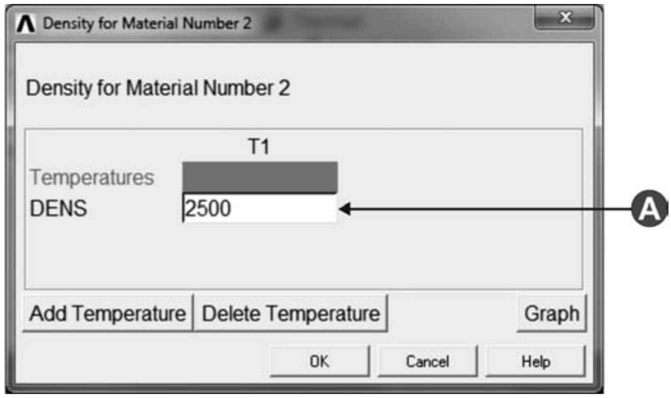
Click on Thermal > Specific Heat



A type 750 in C

OK

Click on Thermal > Density

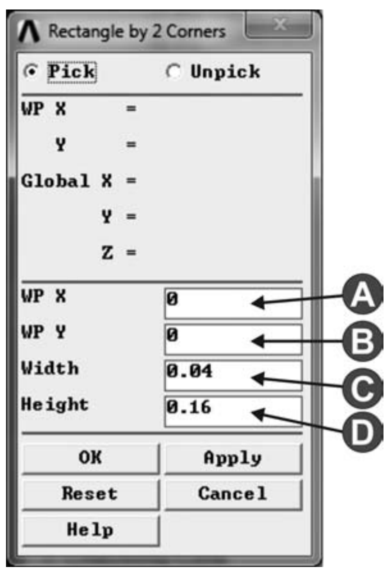


A type 2500 in DENS

OK

The geometry is modeled by creating a rectangle and circles. The Boolean operation is utilized to merge the areas.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

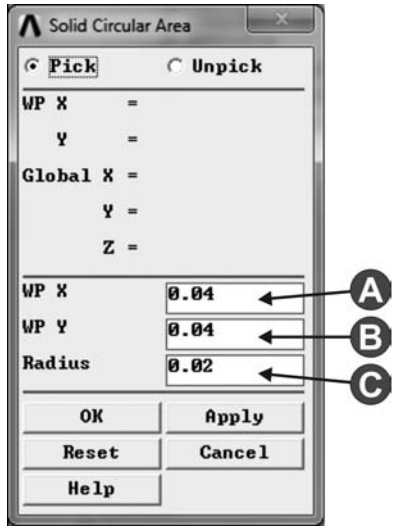
B type 0 in WP Y

C type 0.04 in Width

D type 0.16 in Height

OK

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle

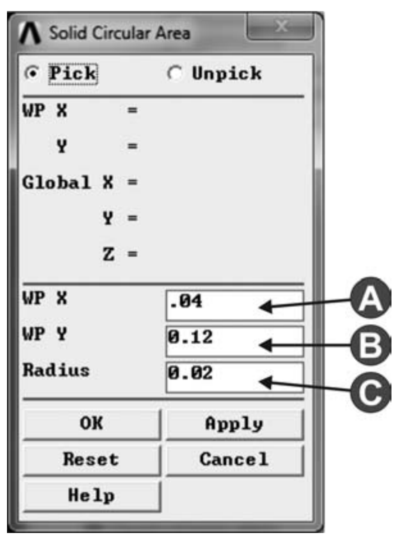


A type 0.04 in WP X

B type 0.04 in WP Y

C type 0.02 in Radius

Apply



A type 0.04 in WP X

B type 0.12 in WP Y

C type 0.02 in Radius

OK

ANSYS graphics show the areas

The cement and the air areas are separated, and the area of the air is just over the area of the cement. The two areas should be connected by common lines. Overlap operation merges the areas, and the boundary lines of the air will be sheared with the cement. Then, the extra areas of the air are deleted.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

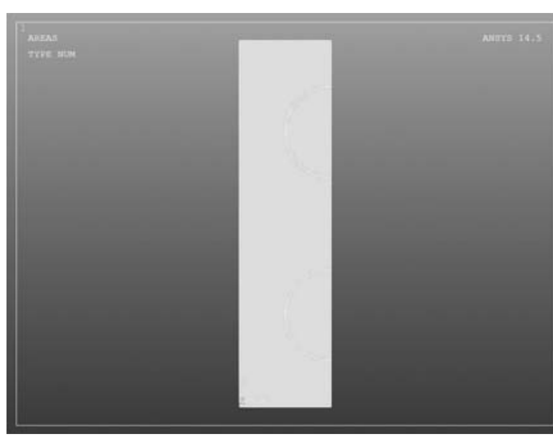
In Overlap Areas window, click on

Pick All

Main Menu > Preprocessor > Modeling > Delete > Area and Below

Click on extra air areas to select them, and in Delete Area and Below window, click on

OK



ANSYS graphics show the final geometry

Main Menu > Preprocessor > Meshing > Mesh Tool

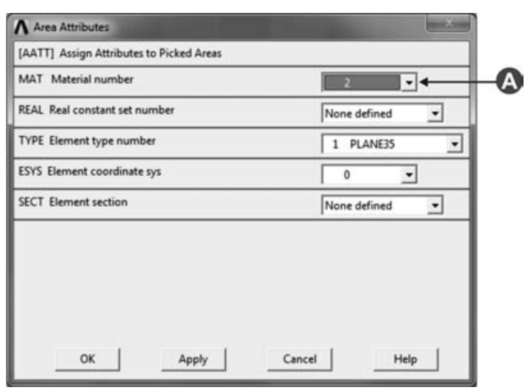
A select Areas

B click on Set

Select the brick area only. In Area Attributes window, click on

OK

The following windows will show up. By selecting number 2, the properties of number 2 in the material model are assigned to the cement. The air by default has the properties of number 1 in the material model.

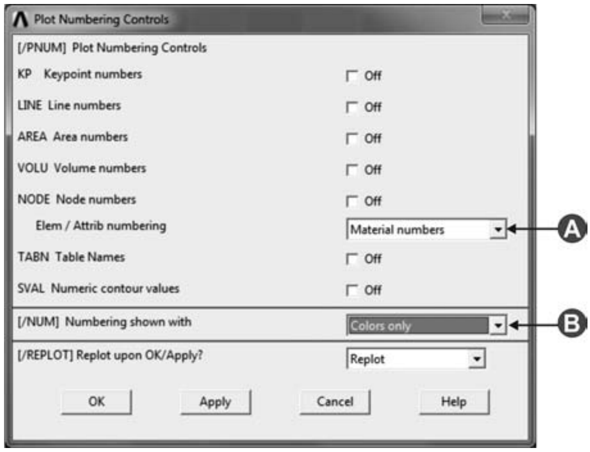


A select 2 in Material number

OK

To ensure that thermal conductivities of the air and cement are assigned correctly, the components are colored according to their material number. This has no effect on the solution. The geometry is meshed with triangular six-node elements. A free mesh is generated using the smart mesh option. The mesh refinement is 1.

Utility Menu > PlotCtrls > Numbering …



A select Material numbers

B select Colors only

OK

Main Menu > Preprocessor > Meshing > Mesh Tool

A select Smart Size

B set the level to 1

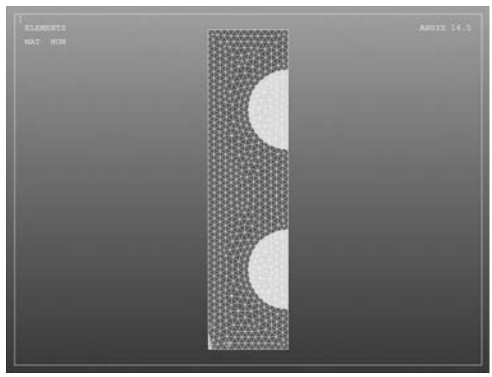
Mesh

In Mesh Areas window, click on

Pick All

Close the Mesh Areas window

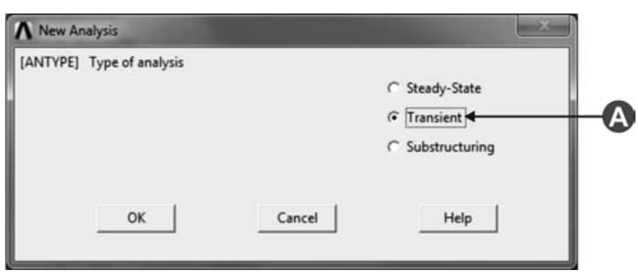
Close



ANSYS graphics show the brick and air with different colors, and finite element mesh

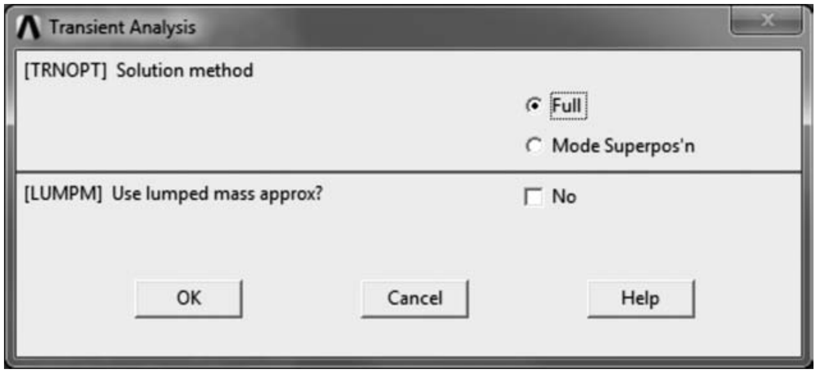
The type of the analysis is changed to Transient, and a Full solution method is selected. This solution method fully imposes the boundary conditions at time 0. The solution control window is used to specify the total time duration, time step, and number of output results to be stored for the postprocessor. The total time duration is 3 days, or 259,200 seconds, and the total time duration is solved by approximately 1000 steps, or the solution time step size is 250. If less time step size is used, such as 125 seconds, the results will be more accurate, but the computational time will be doubled. Data will be stored at every two subset. Hence, there will be 518 data sets available for the postprocessor task.

Main Menu > Solution > Analysis Type > New Analysis



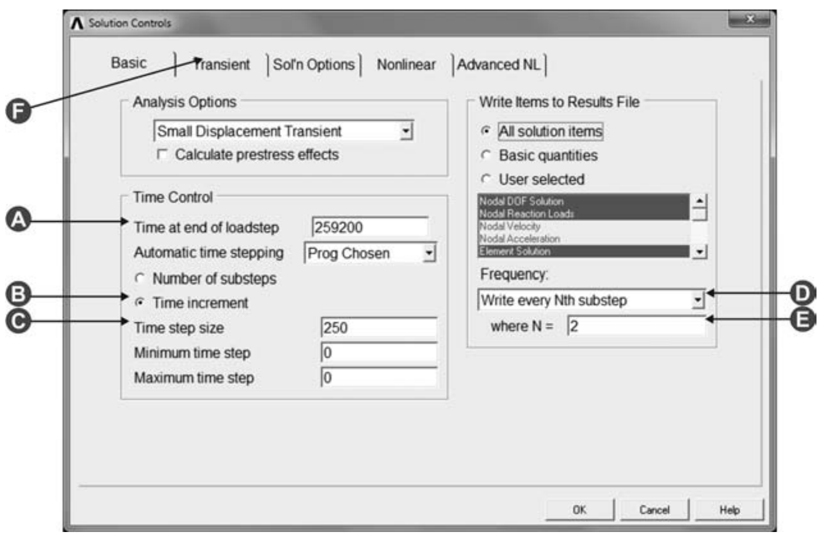
A select Transient

OK



OK

Main Menu > Solution > Analysis Type > Sol'n Controls



A type 259200 in the Time at end of loadstep

B select Time increment

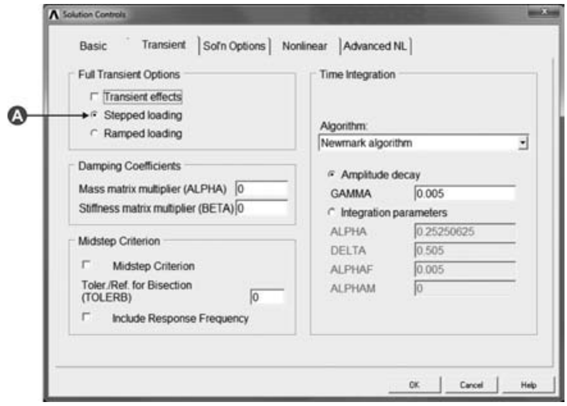
C type 250 in Time step size

D select Write every Nth substep

E type 2 in where N =

F click on Transient tab

In the Transient tab, selection between stepped loading or ramped loading is specified. If stepped loading is selected, loads are fully applied at the time = 0. If ramped loading is selected, loads are linearly increasing over the entire process. In this example, the stepped loading is selected.

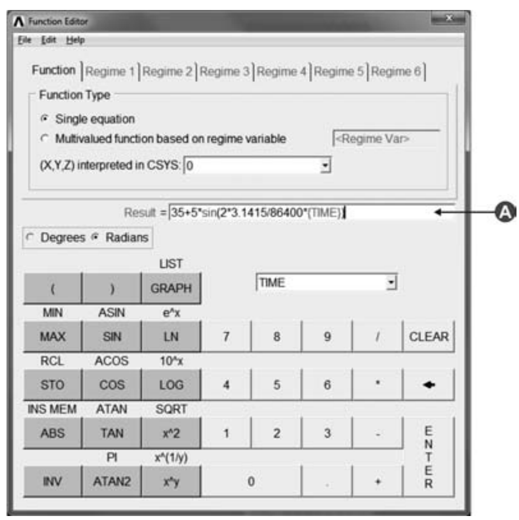


A select Stepped loading.

OK

Boundary conditions and initial conditions are applied in the solution task. The function editor is used to apply a transient temperature formula for a convective boundary condition on the upper horizontal surface. This technique is convenient for this problem since an equation for the temperature is given.

Main Menu > Solution > Define Loads > Apply > Functions > Define/Edit



A type the equation: 35+5*sin(2*3.1415/86400*{TIME})

In Equation Editor, click on File then Save

Save the file as Tout, and this file name is optional. After saving the function, it is required to load it to the ANSYS solution using the read file.

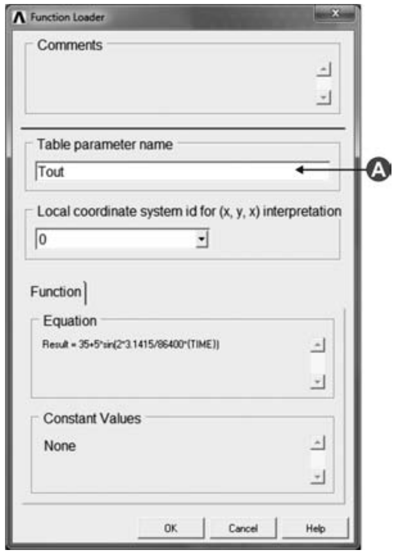
Save

Close the Equation Editor window

Main Menu > Solution > Define Loads > Apply > Functions > Read File

Select the file Tout.func, then

Open

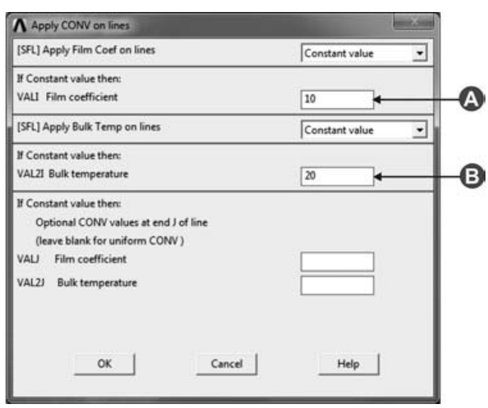


A type Tout in the Table parameter name, and this name is optional, and should not be the same as the file name of the function.

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

Click on the bottom line where the indoor convective boundary condition is applied, and then in Apply CONV on lines window, click on

OK



A type 10 in the VALI Film coefficient

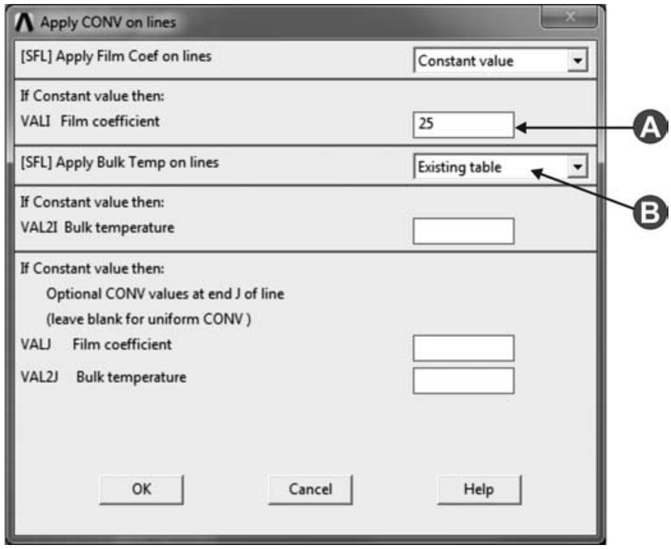
B type 20 in the VAL2I Bulk temperature

OK

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

Click on the upper line where the outdoor convective boundary condition is applied, and then in Apply CONV on lines window, click on

OK

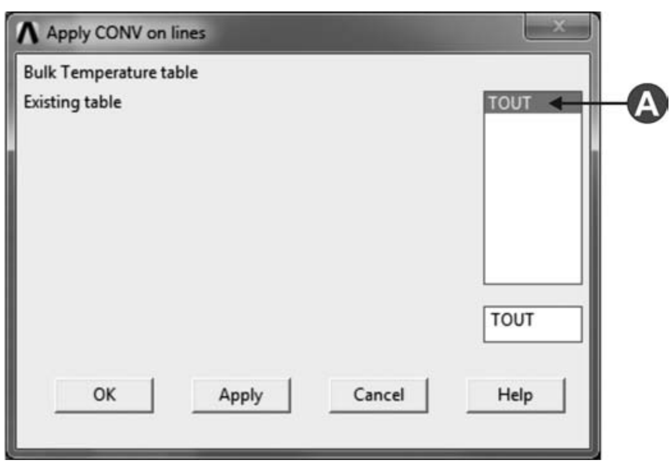


A type 25 in Film coefficient

B select Existing table

OK

The following window will show up to select the function.



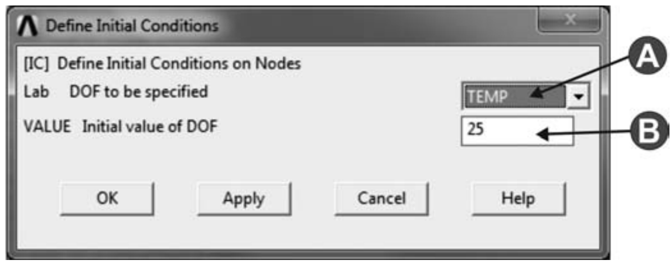
A select TOUT

OK

Main Menu > Solution > Define Loads > Apply > Initial Condit'n > Define

To select all nodes in the domain, in Define Initial Conditions window, click on

Pick All



A select TEMP in DOF to be specified

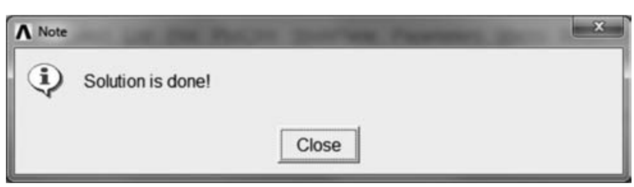
B type 25 in the VALUE Initial value of DOF

OK

The solution task is now completed, and the model is now ready to be solved. During the solution task, ANSYS output windows will show the progress of the solution, and carefully monitor the run.

Main Menu > Solution > Solve > Current LS

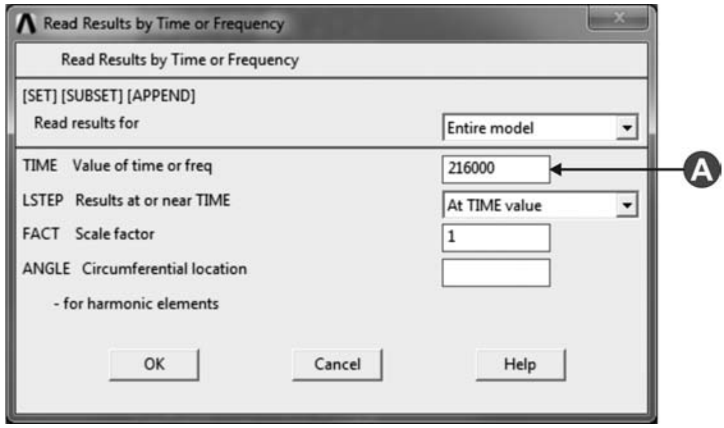
OK



Close

The temperature contours are presented at 12 PM of the third day, or at time = 216,000 seconds. First, the time step is loaded, and then the temperature contours are plotted.

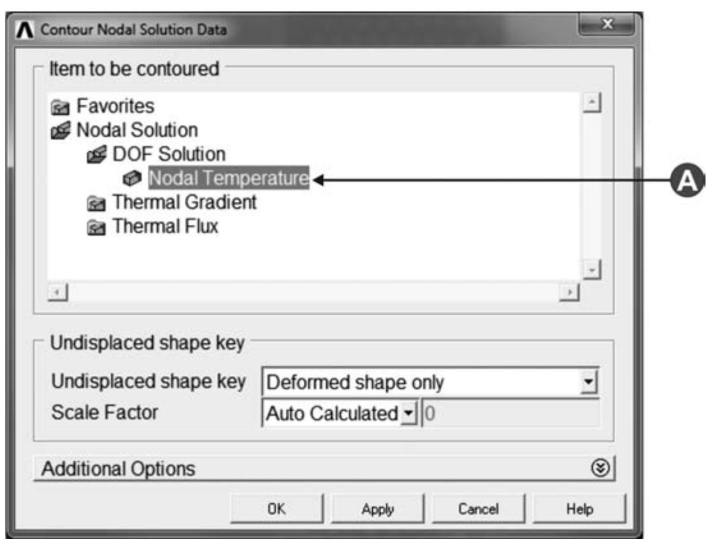
Main Menu > General Postproc > Read Results > By Time/Freq



A type 216000 in TIME Value of time or freq

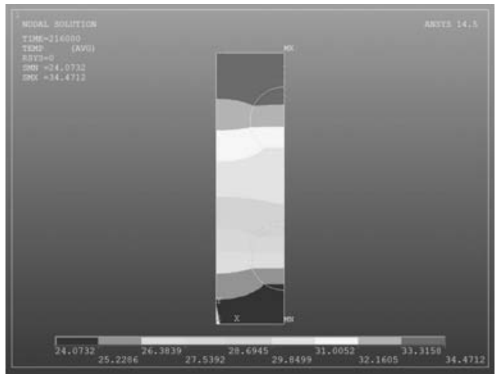
OK

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > DOF Solution > Nodal Temperature

OK

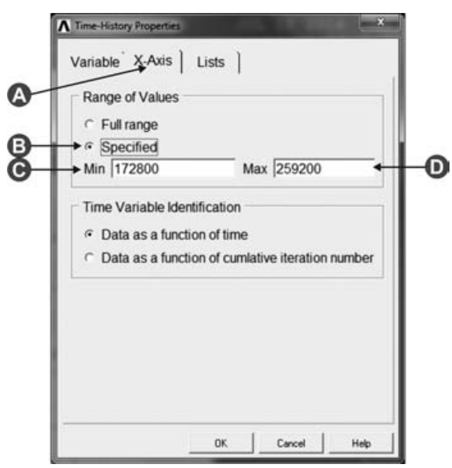


ANSYS graphics show the temperature contours at time = 216,000 seconds

Determining the temperature history at a specific location in the domain is required. Here, the temperature history at the upper and bottom surfaces is presented in graphical form. Results are presented for the third day only. Hence, the time range is from 172,800 to 259,200 seconds as follows:

Main Menu > TimeHist Postpro

A click on the data properties button



A select X-Axis

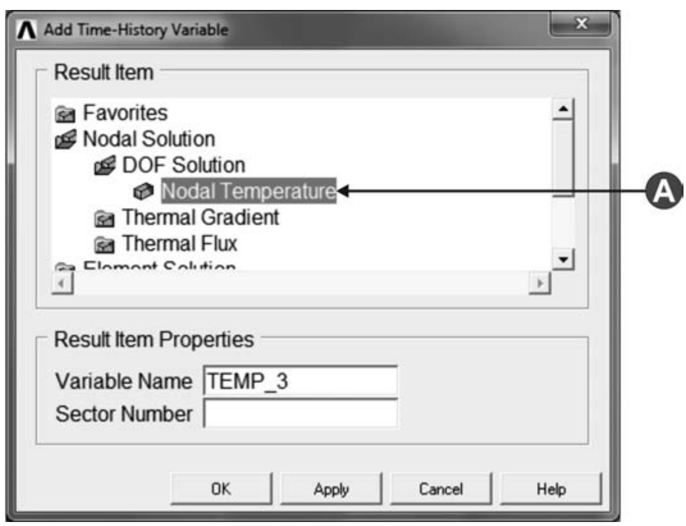
B select Specified

C type 172800 in Min

D type 259200 in Max

OK

A click on green + button

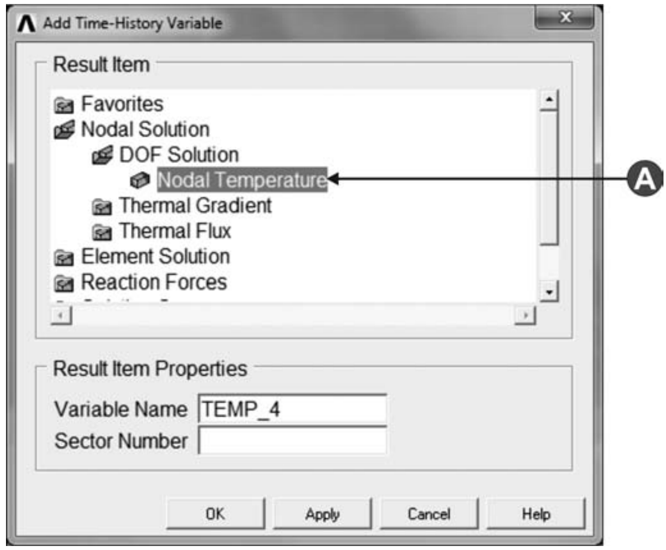


A click on Nodal Solution > DOF Solution > Nodal Temperature

Apply

Click at right bottom corner, and in Node for Date window, click on

Apply

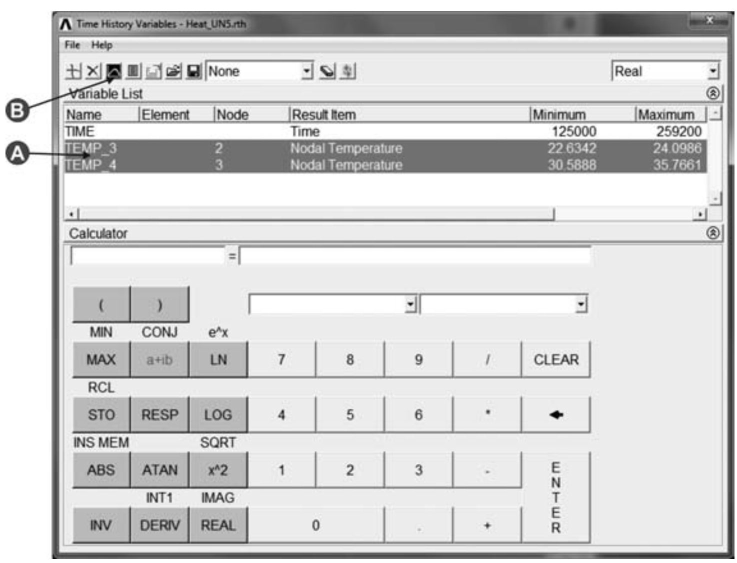


A click on Nodal Solution > DOF Solution > Nodal Temperature

OK

Click at right upper corner, and in Node for Date window, click on

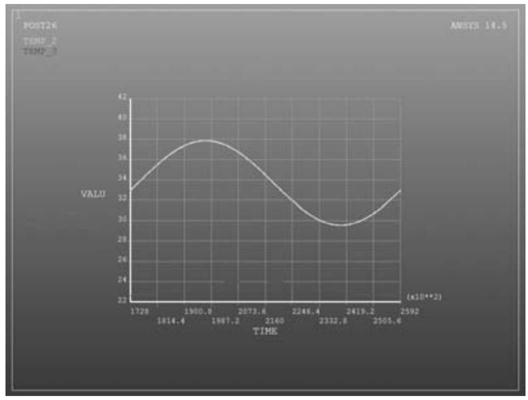
OK



A click on Ctrl key, and then click on TEMP_3 and TEMP_4 to select both temperatures

B click on the graph button

OK

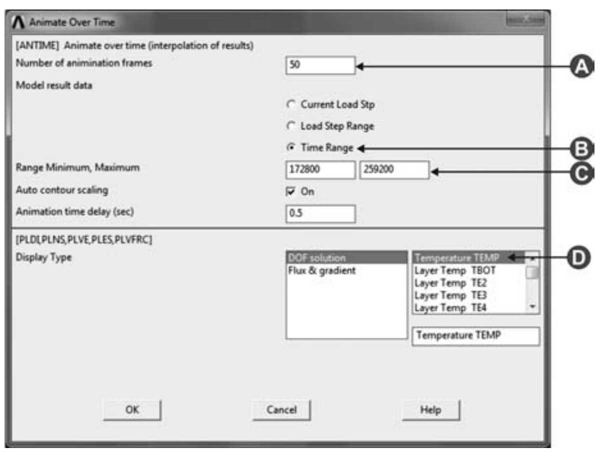


ANSYS graphics show the temperature history at the selected locations

Animation of the temperature contours from time = 172,800 to 259,200 seconds can be easily accomplished using animate in the PlotCtrls. The number of the frames in the animate over time is the number of pictures in the avi file, while the animation time delay is the display period between two pictures.

Main Menu > General Postproc

Utility Menu > PlotCtrls > Animate > Over time …



A type 50 in Number of animation frames

B select Time Range

C type 172800 and 259200 in Range Minimum, Maximum

D select Temperature TEMP

OK

The ANSYS will show an animation of the heating process of the brick. The animation file will be stored in the working directory, and its format is avi.

PROBLEM 5.1

Consider a composite wall shown in Figure 5.9. The wall is composed of four layers with different thermal conductivities: $k_A = 0.22$ W/m · °C, $k_B = 0.16$ W/m · °C, $k_C = 0.25$ W/m · °C, and $k_D = 0.35$ W/m · °C. A convective boundary condition is applied at the left surface, $h_i = 7.5$ W/m^2 · °C and $T_i = 22$°C, and at the right surface, h = 15 W/m^2 · °C and $T_i = 45$°C. Calculate the temperature at the interfaces, and heat flow through the wall using the finite element method. Given that $L_A = 2.5$ cm, $L_B = 15$ cm, $L_C = 20$ cm, and $L_D = 2.5$ cm.

PROBLEM 5.2

Consider a composite pipe shown in Figure 5.10. The pipe is composed of three pipes with different thermal conductivities ($k_A = 0.22$ W/m · °C, $k_B = 0.16$ W/m · °C, and $k_C = 0.25$ W/m · °C), and radii ($r_A = 0.03$ m, $r_B = 0.02$ m, and $r_C = 0.01$ m). A convective boundary condition is applied at the right surface: $h_1 = 10$ W/m^2 · °C and $T_1 = 25$°C, and fixed temperature at the left surface, $T_2 = 45$°C. The external surface of the pipe is well insulated. Calculate the temperature at the interfaces, and heat flow through the pipe using finite element method. Given that $L_A = 0.05$ m, $L_B = 0.075$ m, and $L_C = 0.025$ m.

PROBLEM 5.3

The two-dimensional body shown in Figure 5.11 is subjected to a fixed temperature boundary at the left and right vertical sides, $T_L = 100$°C and $T_R = 50$°C, respectively. The two horizontal sides are well insulated. The side length of the body is 1 m, and it has thermal conductivity of 10 W/m · °C. Determine the temperature at the center of the body using the finite element method. Consider the suggested four elements mesh.

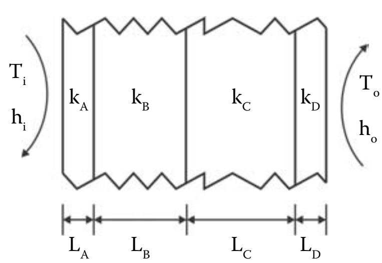


FIGURE 5.9 Heat conduction in a composite wall.

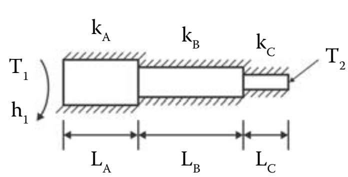


FIGURE 5.10 Heat conduction in a composite pipe.

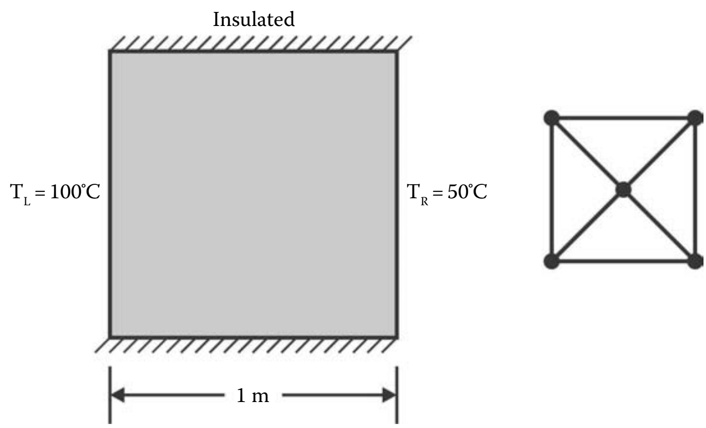


FIGURE 5.11 Two-dimensional body subjected to fixed temperature boundary conditions and suggested finite element mesh.

PROBLEM 5.4

The two-dimensional body shown in Figure 5.12 is subjected convection at the inclined side, with $T_R = 50°C$ and $h = 20\ W/m^2 \cdot °C$. At the left vertical side, a fixed temperature boundary is imposed, $T_L = 100°C$, while the two horizontal sides are well insulated. It has thermal conductivity of 10 W/m · °C. Determine the temperature at the inclined surface of the body using the finite element method. Consider the suggested three elements mesh.

PROBLEM 5.5

The fin shown in Figure 5.13 is used to manage the temperature of an electronic chip that generates heat. Heat is only generated in the chip, and its value is 15 Watts. The heat transfer process is steady. Heat convection is applied along the entire external surfaces, $h = 10\ W/m^2 \cdot °C$ and $T_o = 25°C$, while the bottom surface of the chip is well insulated. Determine the maximum and average temperatures at the bottom surface of the chip, given $k_{fin} = 75\ W/m \cdot °C$, and $k_{chip} = 0.95\ W/m \cdot °C$.

PROBLEM 5.6

Four cylindrical heaters are embedded in a conductive epoxy for heating purposes. An aluminum alloy fin is used to enhance heat flow out of the heaters. Figure 5.14 shows the geometry and the thermal conductivity of the components. Each heater generates 12.5 Watts, and free convection, $h = 10\ W/m^2 \cdot °C$ and $T = 20°C$, is applied at all external surfaces. Determine:

1. Maximum operating temperature in the device.
2. Temperature distribution a long path A–A.
3. Average temperature a long path A–A.

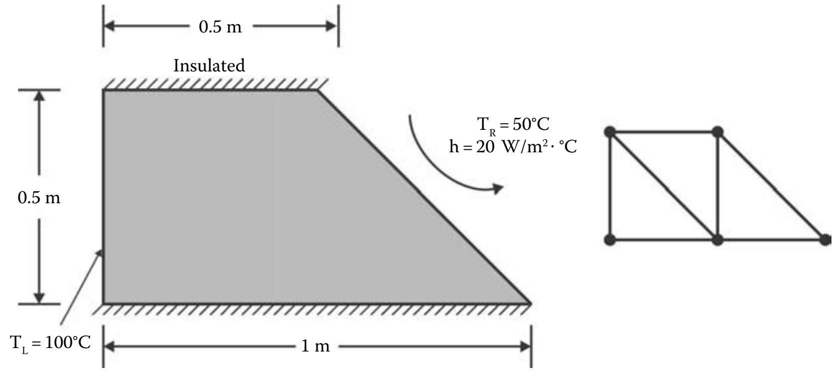


FIGURE 5.12 Two-dimensional body subjected convection and fixed temperature boundary conditions and the suggested finite element mesh.

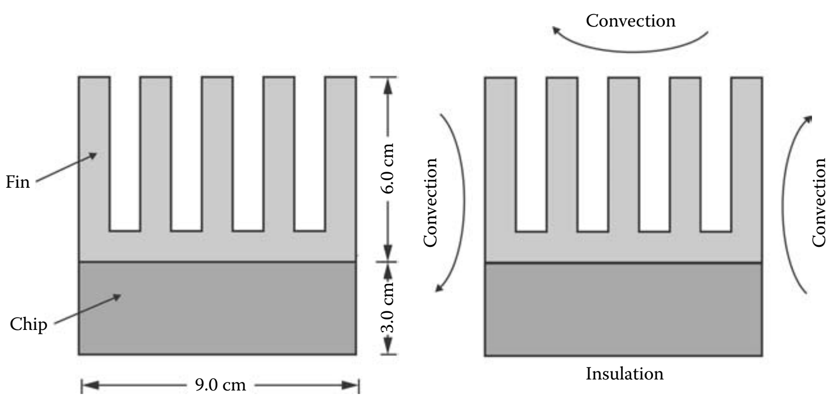


FIGURE 5.13 Heated chip with fin, and the boundary conditions.

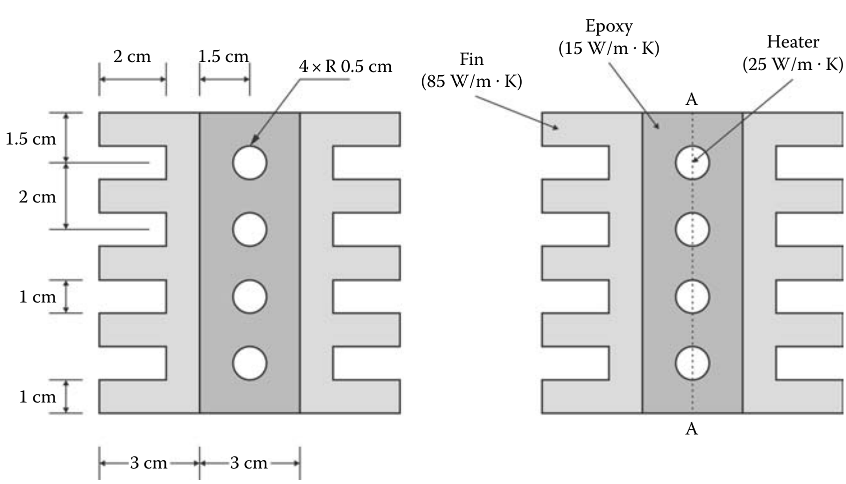


FIGURE 5.14 Cylindrical heaters with fin.

PROBLEM 5.7

A masonry brick, as shown in Figure 5.15, is made of cement and air holes. At the indoor surface, convective boundary conditions are applied with time-independent temperature and heat transfer coefficient, $T_i = 22.5°C$ and $h_i = 7.5\ W/m^2 \cdot °C$. At the outdoor surface, heat transfer coefficient is time independent, $h_o = 27.5\ W/m^2 \cdot °C$, but the temperature is time dependent. The initial temperature of the brick is 20°C.

Property	Air	Cement
Density (kg/m^3)	1.125	2400
Conductivity (W/m · K)	0.025	0.82
Specific heat (J/kg · K)	1005	740

The following expression simulates the outdoor temperature:

$$T(t) = 30 + 10\ \mathrm{Sin}\left(\frac{2\pi}{86400}\,t\right)$$

To ensure having a periodic condition, the simulation should be kept running for 3 days, and the results for the last day should be presented. Show temperature histories at the indoor and outdoor surfaces of the brick for the last day.

PROBLEM 5.8

For the electronic board shown in Figure 5.16, the initial temperature of the entire system is 20°C. At time = 0, 24 Watts are generated in the processor, and convection is applied to the boundaries with $h = 5\ W/m^2 \cdot °C$ and $T_o = 20°C$, while the board bottom surface is well insulated. The total

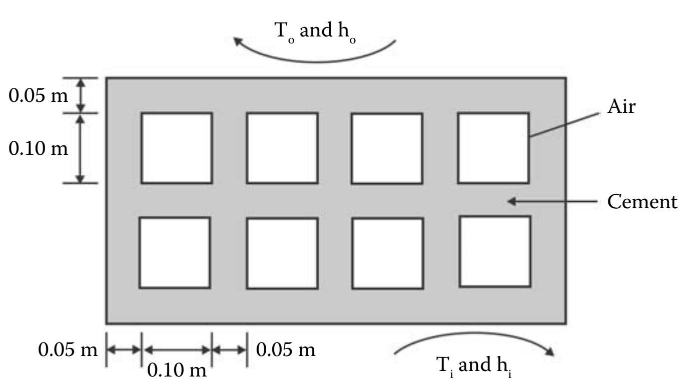


FIGURE 5.15 Masonry brick and the boundary conditions.

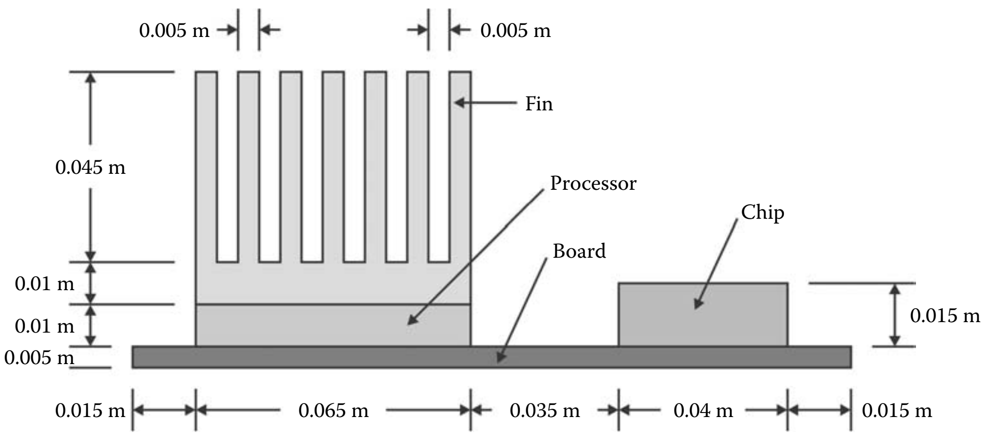


FIGURE 5.16 Electronic board.

time duration of the device is 10 hours. Solve this transient process using a time step of 25 seconds. Determine the following:

a. Does the system reach the steady-state condition?
b. What is the maximum temperature of the processor and chip at time = 5 hours?
c. Show a temperature history at the center of the processor and chip.
d. Show the temperature distribution along the external surface of the chip at time = 5 and 7.5 hours.

Property	Chip	Fin	Processor	Board
Density (kg/m^3)	900	2050	920	910
Conductivity (W/m·K)	2.1	45	2.1	3.2
Specific heat (J/kg·K)	2200	4500	1300	1150

This page intentionally left blank

CHAPTER SIX

Fluid mechanics

6.1 Governing equations for fluid mechanics

The mass conservation equation in a differential form can be obtained by applying the mass conservation principle on a differential control volume as shown in Figure 6.1. Considering the control volume, the net mass flow rate in the x-, y-, and z-directions can be expressed as

$$\text{x-direction: } \frac{\partial}{\partial x}(\rho u) dx\ dy\ dz \tag{6.1}$$

$$\text{y-direction: } \frac{\partial}{\partial y}(\rho v) dx\ dy\ dz \tag{6.2}$$

$$\text{z-direction: } \frac{\partial}{\partial z}(\rho w) dx\ dy\ dz \tag{6.3}$$

The rate of change of mass inside the control volume can be obtained from the Reynolds transport theory as follows:

$$\int_{CV} \frac{\partial \rho}{\partial t} dV = \frac{\partial \rho}{\partial t} dx\ dy\ dz \tag{6.4}$$

The net mass flux into the control volume should be equal to the rate of change of mass inside the control volume. The mass conservation in differential form can be expressed as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \tag{6.5}$$

Newton's second law on a differential control volume, as shown in Figure 6.1, can be used to obtain the conservation of momentum equation. The net forces on the control volume should be balanced with the acceleration of the control volume times its mass as follows:

$$\vec{a}\ dm = \sum \overrightarrow{dF} \tag{6.6}$$

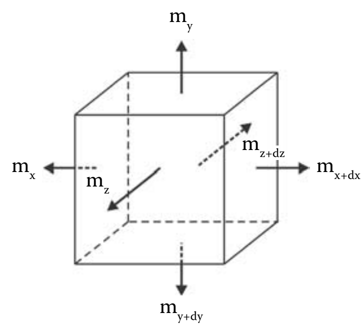


FIGURE 6.1 Fixed control volume.

The acceleration vector field $\vec{a}$ is obtained from the total time derivative of the velocity vector

$$\vec{a} = \frac{d\mathbf{V}}{dt} = \frac{du}{dt}\mathbf{i} + \frac{dv}{dt}\mathbf{j} + \frac{dw}{dt}\mathbf{k} \tag{6.7}$$

Each component of the velocity fields is a function of the space and time. Using the chain rule, the scalar time derivative can be obtained

$$\frac{du(x,y,z,t)}{dt} = \frac{\partial u}{\partial t} + \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\frac{dz}{dt} \tag{6.8}$$

where $u = dx/dt$ is the local velocity component in the x-direction, $v = dy/dt$ is the local velocity component in the y-direction, and $w = dw/dt$ is the local velocity component in the z-direction. The total derivative of u is the acceleration in the x-direction

$$a_x = \frac{du(x,y,z,t)}{dt} = \frac{\partial u}{\partial t} + u\frac{du}{dx} + v\frac{du}{dy} + w\frac{du}{dz} \tag{6.9}$$

The acceleration in the y- and z-directions can be expressed as, respectively,

$$a_y = \frac{dv(x,y,z,t)}{dt} = \frac{\partial v}{\partial t} + u\frac{dv}{dx} + v\frac{dv}{dy} + w\frac{dv}{dz} \tag{6.10}$$

$$a_z = \frac{du(x,y,z,t)}{dt} = \frac{\partial w}{\partial t} + u\frac{dw}{dx} + v\frac{dw}{dy} + w\frac{dw}{dz} \tag{6.11}$$

The mass of the control volume must be equal to volume times the density as follows:

$$dm = \rho\, dx\, dy\, dz \tag{6.12}$$

The forces on the control volume are of two types: body and surface. The body force is due to the gravity:

$$dF_b = \rho \vec{g}\, dx\, dy\, dz \tag{6.13}$$

while the surface forces are due to the surface stresses, including normal and parallel stresses. The surface stresses in the x-, y-, and z-directions are as follows:

$$dF_{sx} = \left(\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx\, dy\, dz \tag{6.14}$$

$$dF_{sy} = \left(\frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) dx\, dy\, dz \tag{6.15}$$

$$dF_{sz} = \left(\frac{\partial \sigma_{zz}}{\partial x} + \frac{\partial \tau_{xz}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right) dx\, dy\, dz \tag{6.16}$$

The equations of motion in the x-, y-, and z-directions can be expressed as, respectively,

$$\begin{aligned} \text{x-direction: } \rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) \\ = \left(\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) \end{aligned} \tag{6.17}$$

$$\begin{aligned} \text{y-direction: } \rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) \\ = \left(\rho g_y + \frac{\partial \sigma_{yy}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) \end{aligned} \tag{6.18}$$

$$\begin{aligned} \text{z-direction: } \rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) \\ = \left(\rho g_z + \frac{\partial \sigma_{zz}}{\partial x} + \frac{\partial \tau_{xz}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right) \end{aligned} \tag{6.19}$$

For Newtonian fluid, the stress components are obtained from the theory of elasticity, and they are

$$\sigma_{xx} = -P + 2\mu \frac{\partial u}{\partial x} \tag{6.20}$$

$$\sigma_{yy} = -P + 2\mu \frac{\partial v}{\partial y} \tag{6.21}$$

$$\sigma_{zz} = -P + 2\mu \frac{\partial w}{\partial z} \tag{6.22}$$

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{6.23}$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \tag{6.24}$$

$$\tau_{zy} = \tau_{yz} = \mu\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) \tag{6.25}$$

Substituting the stress equations into the equations of motion, we have

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \rho g_x \tag{6.26}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) + \rho g_y \tag{6.27}$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + \rho g_z \tag{6.28}$$

Equations 6.26–6.28 are called the Navier–Stokes equations. They are nonlinear and nonhomogenous partial differential equations.

6.2 Finite element method for fluid mechanics

The finite element method is utilized to solve the governing equations and to discretize the computational domain for fluid flow problems. Four nodes of quadrilateral elements were used for the numerical discretization. ANSYS has only this type of element in its library for fluid dynamics. Weighted integral statements of the mass, momentum, and energy conservations over a typical element Ω are given by

$$\int_{\Omega} w_1 f_1 \, d\Omega \tag{6.29}$$

$$\int_{\Omega} w_2 f_2 \, d\Omega \tag{6.30}$$

$$\int_{\Omega} w_3 f_3 \, d\Omega \tag{6.31}$$

where f_1, f_2, and f_3 are mass, momentum, and energy conservations, respectively. Then w_1, w_2, and w_3 are weight functions, which are equal to the interpolation functions. The choice of the weight function is restricted to the spaces of approximation functions used for pressure, velocity fields, and temperature. The pressure, velocity fields, and temperature are approximated as follows:

$$P(x,t) = \sum_{l=1}^{4} \Phi_l(x) P_l(t) = \Phi^T \{P\} \tag{6.32}$$

$$u_i(x,t) = \sum_{n=1}^{4} \Psi_n(x) u_i^n(t) = \Psi^T \{u_i\} \tag{6.33}$$

$$T(x,t) = \sum_{m=1}^{4} \Theta_m(x) T_m(t) = \Theta^T \{T\} \tag{6.34}$$

where Φ, ψ, and θ are vectors of the shape functions, and P, u_i, and T are vectors of nodal value of the pressure, velocity components, and temperature, respectively. The weight functions have the following correspondence:

$$w_1 \cong \Phi,\ w_2 \cong \Psi,\ w_3 \cong \Theta \tag{6.35}$$

The mass and momentum conservations can be symbolically written in the following matrix form:

$$[A^T]\ \{u\} = 0 \tag{6.36}$$

$$[C]\ \{u\} + [K]\ \{u\} - [A]\ \{P\} = \{F\} \tag{6.37}$$

$$[D]\ \{T\} + [L]\ \{T\} = \{G\} \tag{6.38}$$

The coefficients are defined by

$$A_i = \int_{\Omega} \frac{\partial \psi}{\partial x_i} \Phi^T d\Omega \tag{6.39}$$

$$C_i(u_j) = \int_{\Omega} \rho \psi u_i \frac{\partial \psi^T}{\partial x_i} d\Omega \tag{6.40}$$

$$K_{ij} = \int_{\Omega} \mu \frac{\partial \psi}{\partial x_j} \frac{\partial \psi^T}{\partial x_i} d\Omega \tag{6.41}$$

$$F_i = \int_{\Gamma} \psi \tau_i d\Gamma \tag{6.42}$$

$$D_i\left(u_j\right) = \int_{\Omega} \rho\ C_P \psi u_i \frac{\partial \Theta^T}{\partial x_i} d\Omega \tag{6.43}$$

$$L_{ij} = \int_{\Omega} k \frac{\partial \Theta}{\partial x_i} \frac{\partial \Theta^T}{\partial x_j} d\Omega \tag{6.44}$$

$$G = \int_{\Omega} \Theta Q''' d\Omega + \int_{\Gamma} \Theta q'' d\Gamma \tag{6.45}$$

where k is the thermal conductivity, C_p is the specific heat, and τ_i is the component of the total boundary stress, which is the sum of the viscous boundary stress and the hydrostatic boundary stress. The q'' is the heat flux applied at the boundary of the elements.

6.3 Entrance length in developing flow in a channel using ANSYS

Water flow in a two-dimensional channel, as shown in Figure 6.2, develops in the axial direction. At the inlet, the flow is uniform and the flow velocity is 0.005 m/s. The exit condition is a reference zero pressure. The density and viscosity of water are $\rho = 998.3$ kg/m^3 and $\mu = 1.002 \times 10^{-3}$ (1.002e-3) Pa·s, respectively. Determine the entrance length and show that the mass conservation principle is satisfied.

Double click on the Mechanical APDL Product Launcher icon

This example is limited to fluid analysis. Hence, select Fluid FLOTRAN. The 2D FLOTRAN 141 element is used, and its shape is rectangle with

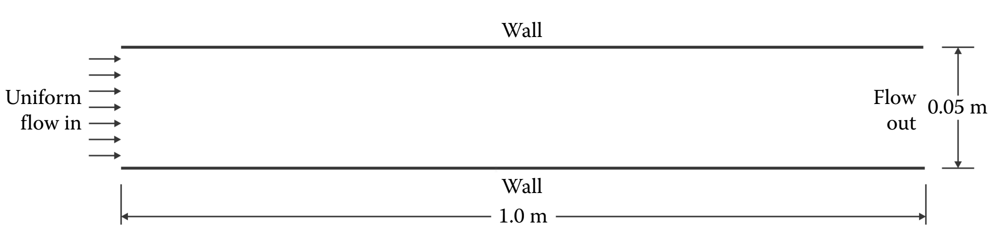
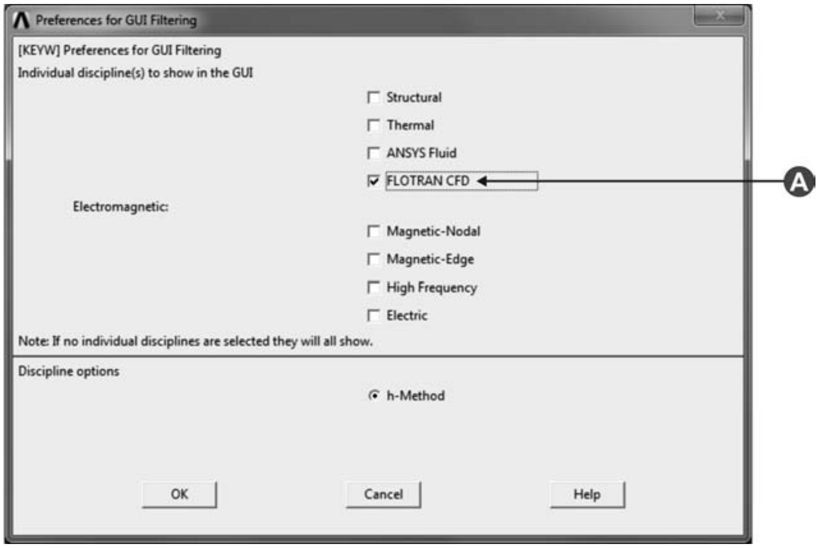


FIGURE 6.2 Channel with developing flow.

four nodes. The 3D FLOTRAN 142 element is for three-dimensional analysis. The density and viscosity are the only needed properties to solve this problem.

Main Menu > Preferences

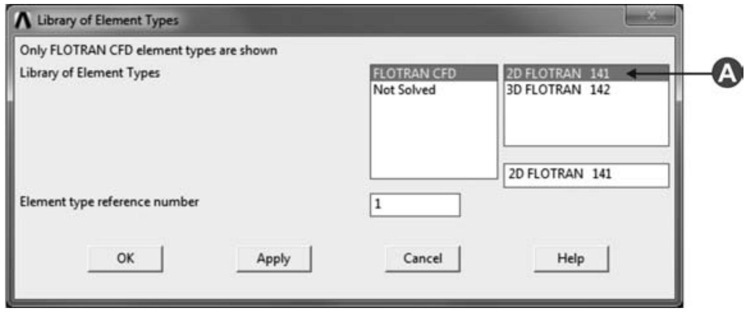


A select FLOTRAN CFD

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add…

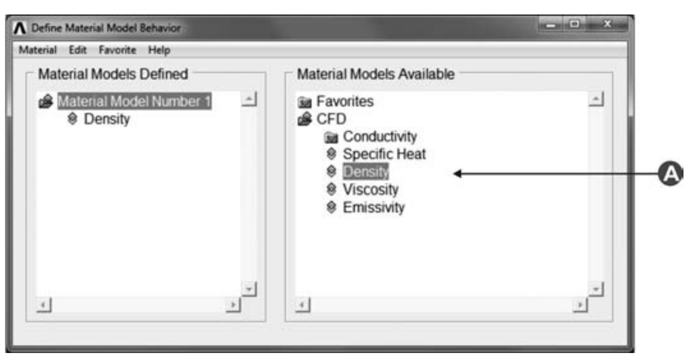


A select 2D FLOTRAN 141

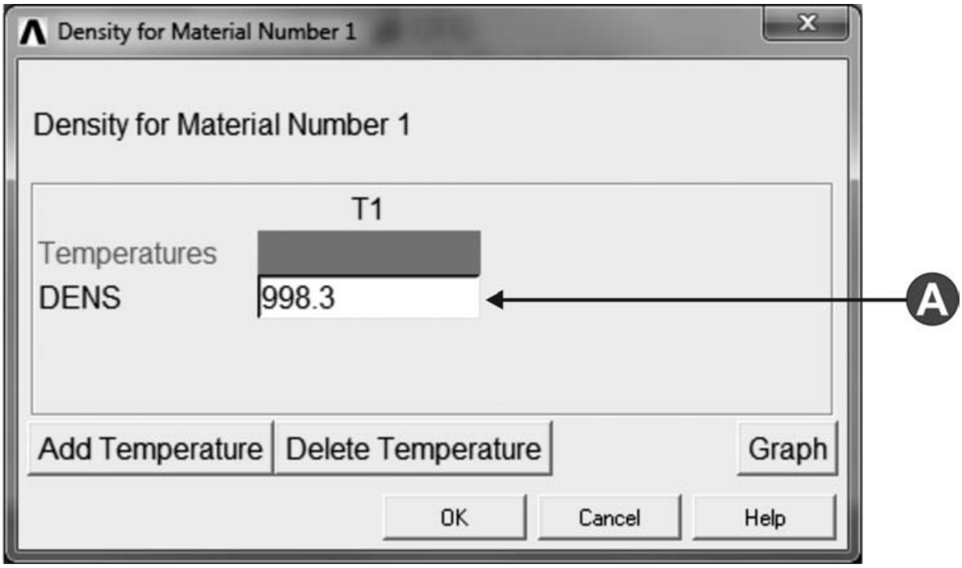
OK

Close

Main Menu > Preprocessor > Material Props > Material Models



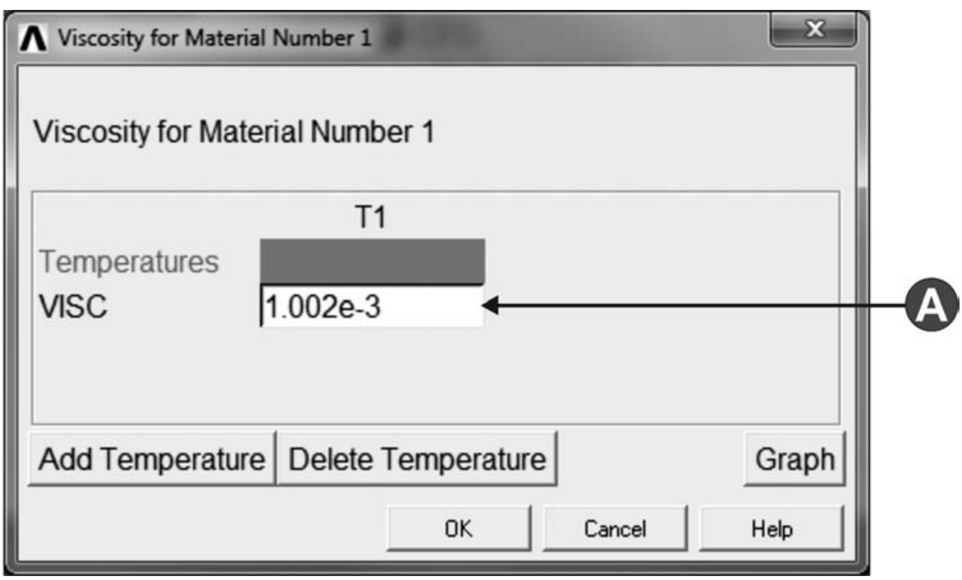
A click on CFD > Density



A type 998.3 in DENS

OK

Click on CFD > Viscosity

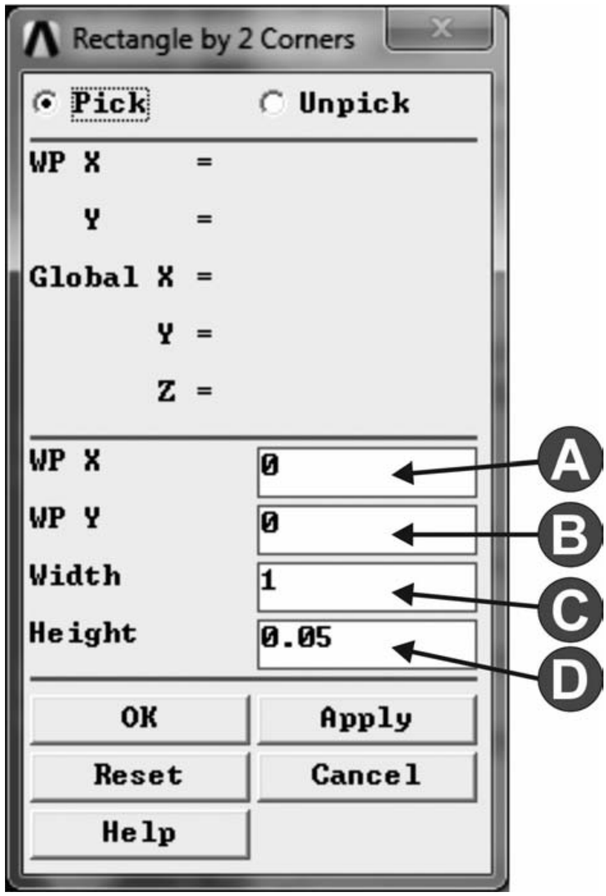


A type 1.002e-3 in VISC

OK

Close the Define Material Model Behavior window

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

B type 0 in WP Y

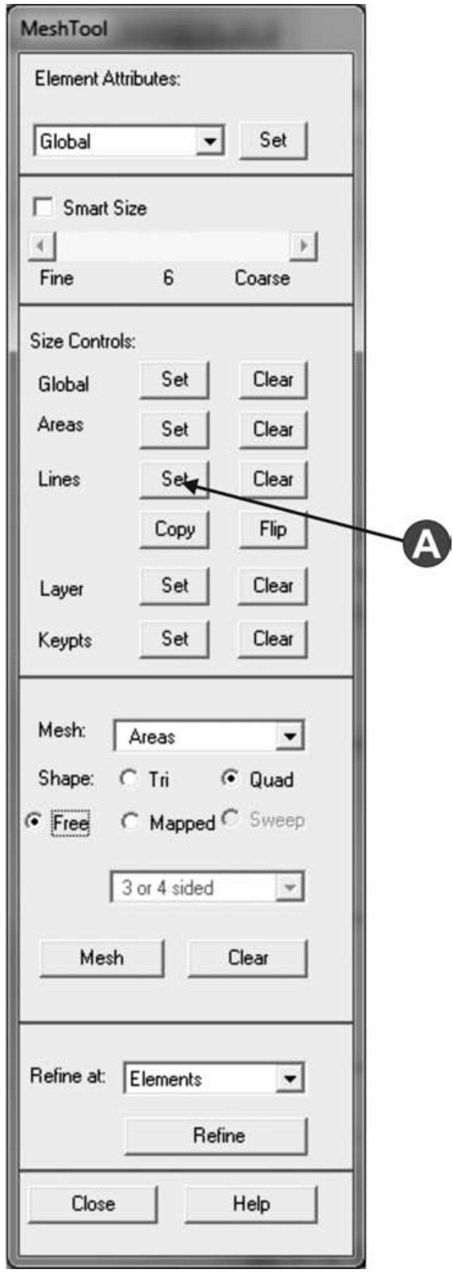
C type 1 in Width

D type 0.05 in Height

OK

The smart mesh number of one will produce insufficient mesh density to have accurate results. The elements in the domain can be additionally increased by using the lines size control in the Mesh Tool. Lines are divided into segments, which will be elements in these lines. The lines are divided by either specifying the number of divisions or the lengths of the segments. In this example, the number of divisions is specified. The vertical and lateral lines will be divided by 25 and 500 segments, respectively.

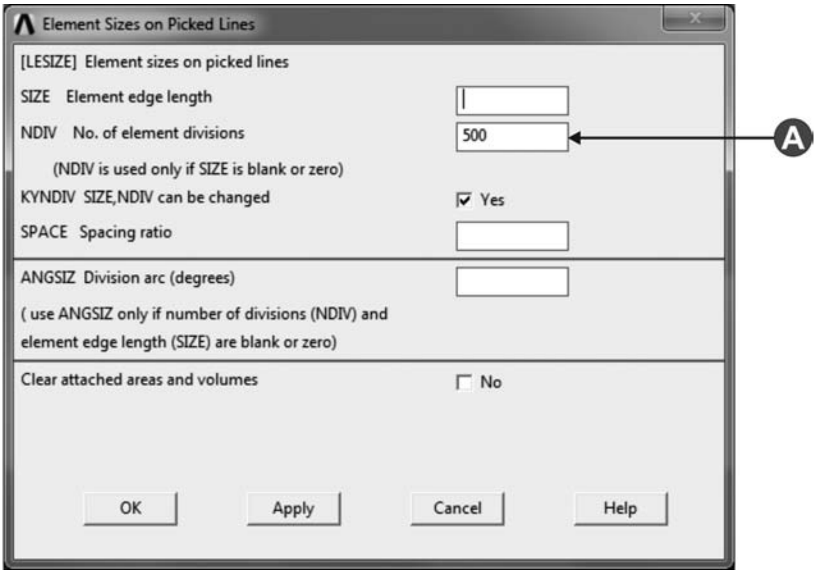
Main Menu > Preprocessor > Meshing > Mesh Tool



A click on Set in Lines

Click on two horizontal lines. Then, in Element Sizes on Picked Lines window, click on

OK

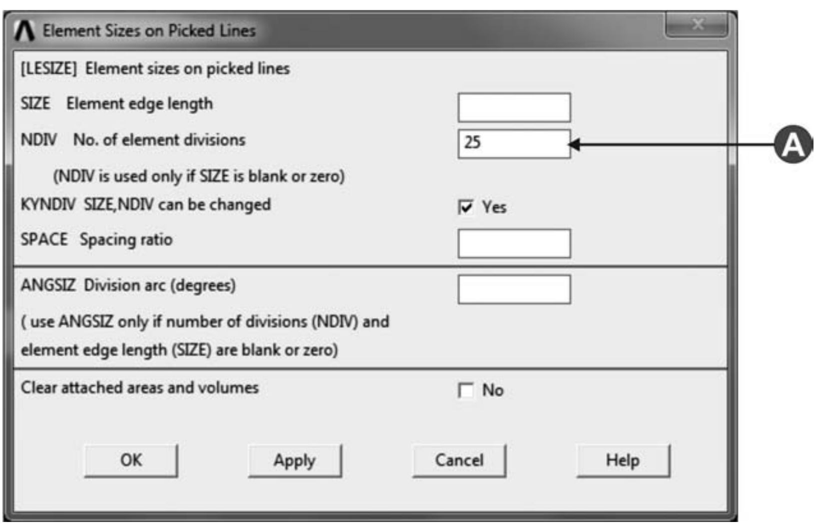


A type 500 in NDIV No. of element divisions

Apply

Click on two vertical lines. Then, in Element Sizes on Picked Lines window, click on

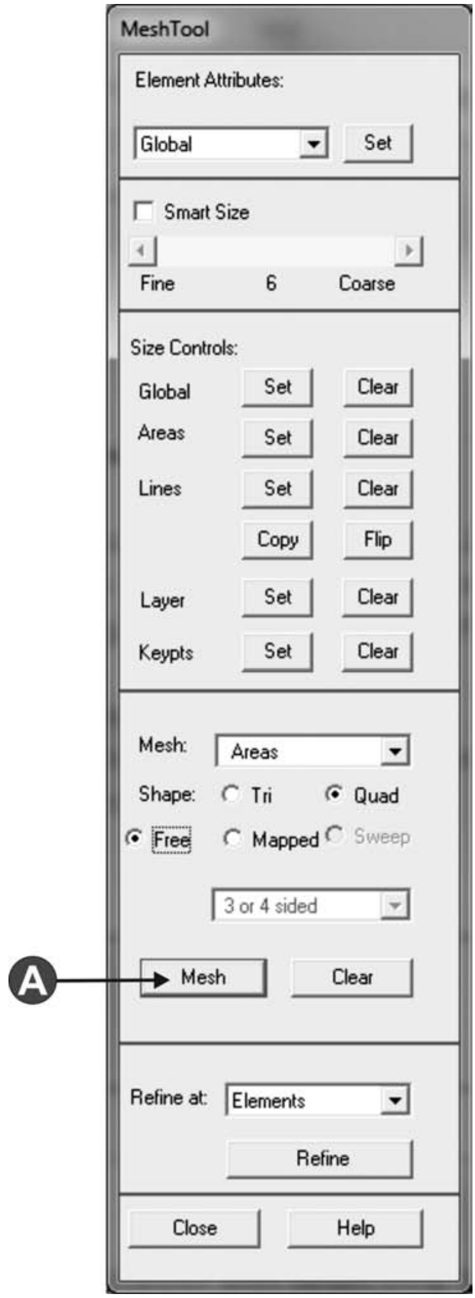
OK



A type 25 in NDIV No. of element divisions

OK

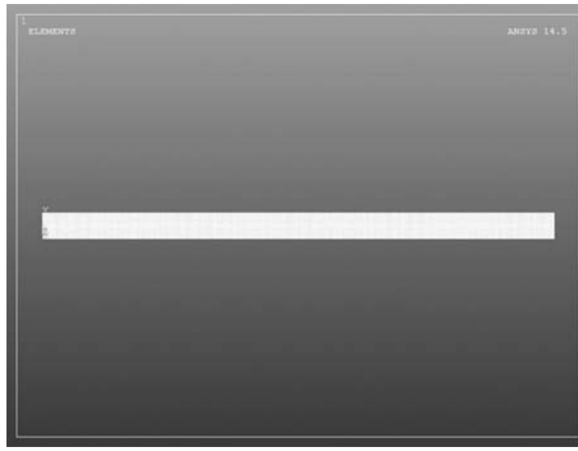
Main Menu > Preprocessor > Meshing > Mesh Tool



A click on Mesh

In Mesh Areas window, click on

Pick All



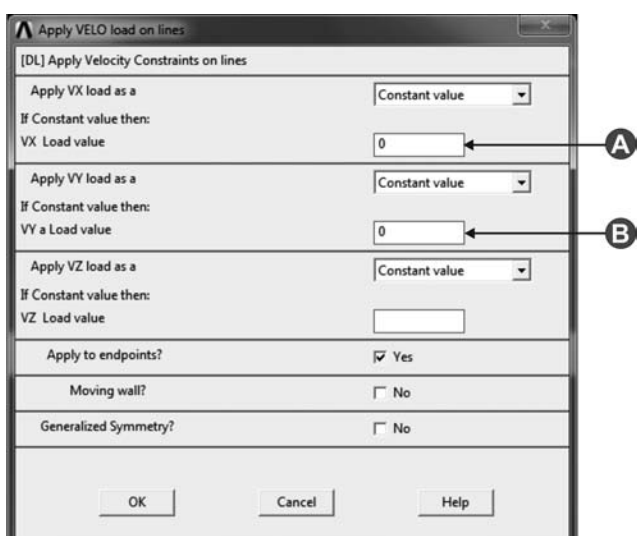
ANSYS graphics show the mesh

Zero velocity is imposed at the upper and lower walls to simulate wall boundary condition. At the inlet, the velocity is uniform. At the exit, zero pressure is imposed to simulate the exit boundary condition.

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click on the two horizontal lines. Then in Apply VELO load on lines window, click on

OK



A type 0 in VX Load value

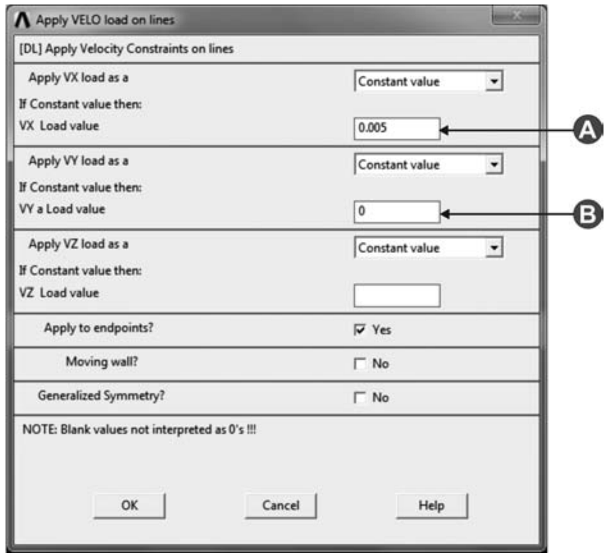
B type 0 in VY a Load value

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click at the inlet line. Then, in Apply VELO load on lines window, click on

OK



A type 0.005 in VX Load value

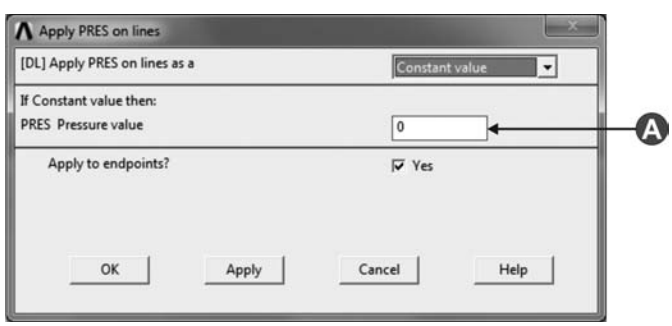
B type 0 in VY a Load value

OK

Main Menu > Solution > Define Load > Apply > Fluid/CFD > Pressure DOF> On Lines

Click on the exit line, and then in Apply PRES on lines window, click on

OK

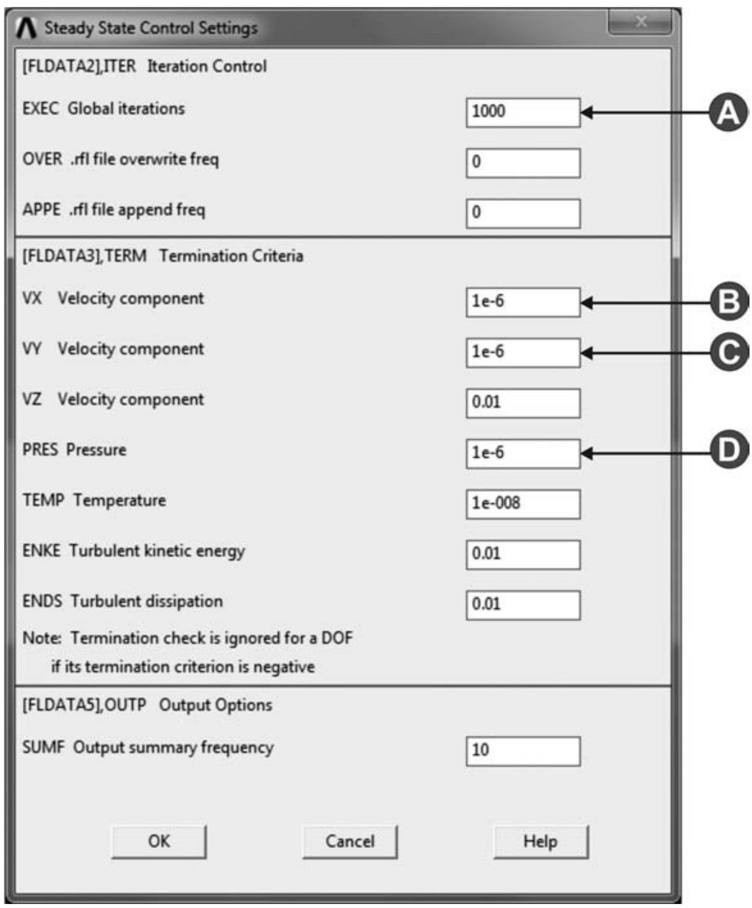


A type 0 in PRES Pressure value

OK

The present problem is steady state and adiabatic. Hence, keep the default setting in execution control. Notice that ANSYS is capable of simulating turbulent and compressible flow. The maximum number of iterations is 1000, and an additional 1000 iterations are required if the termination criterion is not satisfied. The termination criterion for the velocity components and pressure is 1×10^{-6}. The iterations will stop if the maximum number of iterations is reached or the termination criterion is satisfied. The material properties are MP, which means that the ANSYS will use properties stored in the Material Properties in the preprocessor task to solve the problem.

Main Menu > Solution > FLOTRAN Set Up > Execution Ctrl



A type 1000 in EXEC Global iteration

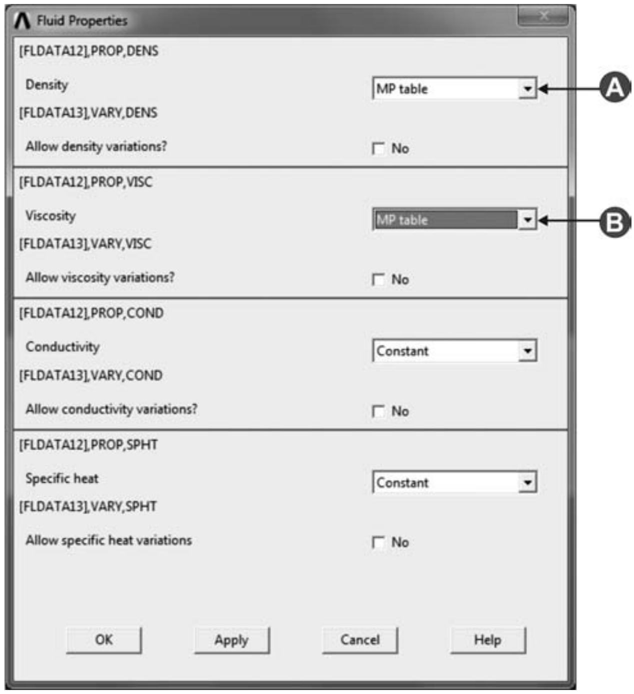
B type 1e-6 in the termination criterion for VX Velocity component

C type 1e-6 in the termination criterion for VY Velocity component

D type 1e-6 in the termination criterion for PRES Pressure

OK

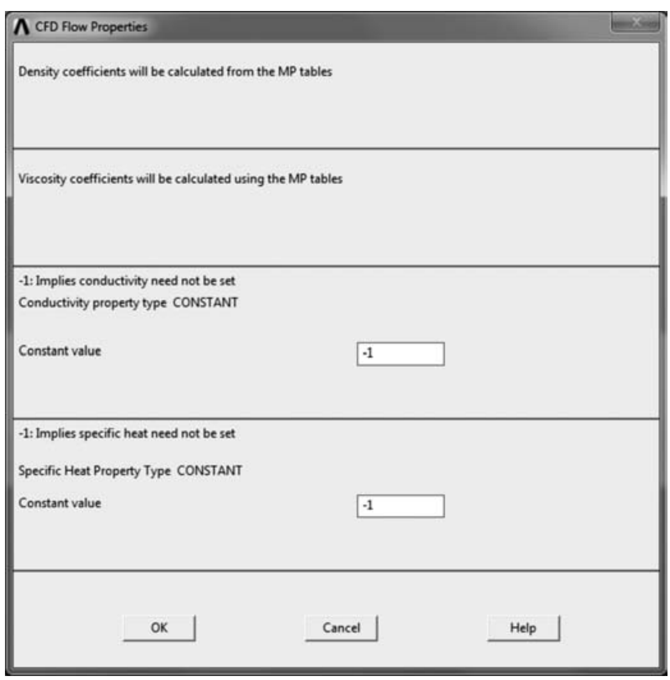
Main Menu > Solution > FLOTRAN Set Up > Fluid Properties



A select MP table in Density

B select MP table in Viscosity

$\boxed{\textbf{OK}}$



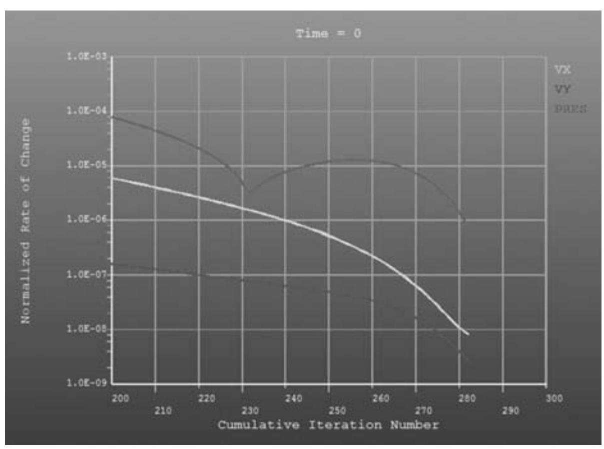
A confirmation window will show up. Read it carefully to avoid an unexpected error. The value –1 indicates that the property is not available. The conductivity and specific heat are not required to solve the problem.

OK

The following step will initiate the numerical iterations. Carefully examine the normalized rate of change for all field variables. The normalized rate of change should reach termination criterion for all field variables to declare the convergence. Otherwise, additional iterations are required. When the normalized rate of change is decreasing, the solution process is approaching the convergence. If the solution is diverged, there are either incorrect boundary conditions or fluid properties.

Main Menu > Solution > FLOTRAN Set Up > Run FLOTRAN

Close

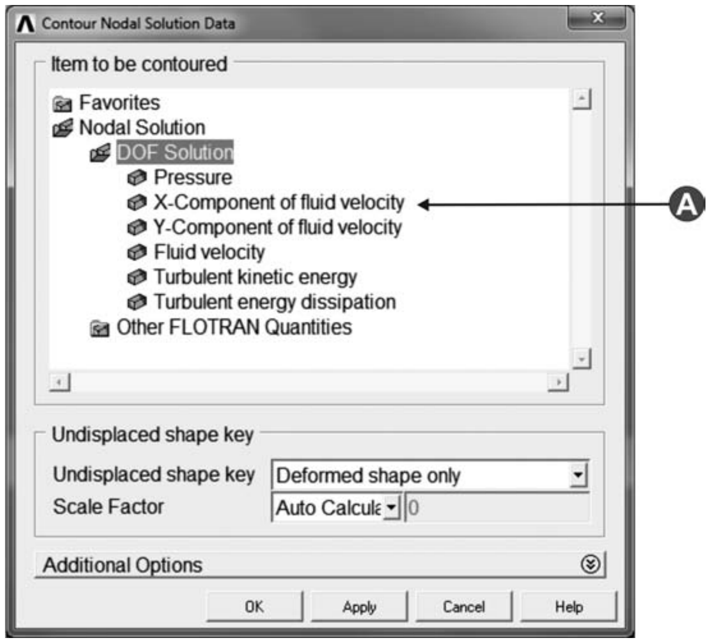


ANSYS graphics show the solution convergence

The normalized rate of change reaches the termination criterion. Hence, the convergence is reached. The Last Set is selected in the Read Results to ensure that the data from the last iteration set are loaded for the postprocessor. Otherwise, no results will be shown in the postprocessor. A plot of velocity vectors is presented in the postprocessor task, followed by velocity profile along the centerline of the channel and at the exit.

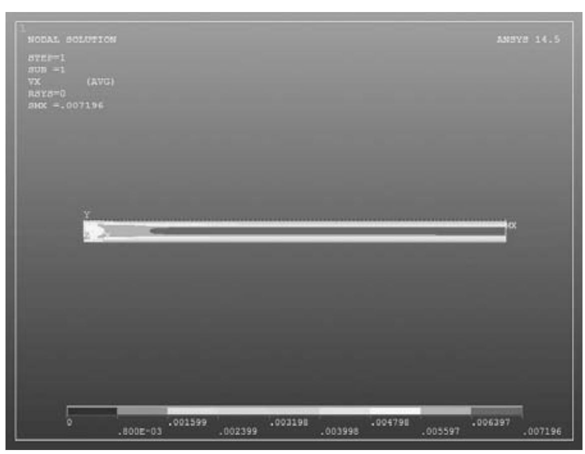
Main Menu > General Postproc > Read Results > Last Set

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A select X-Component of fluid velocity

$\boxed{\textbf{OK}}$



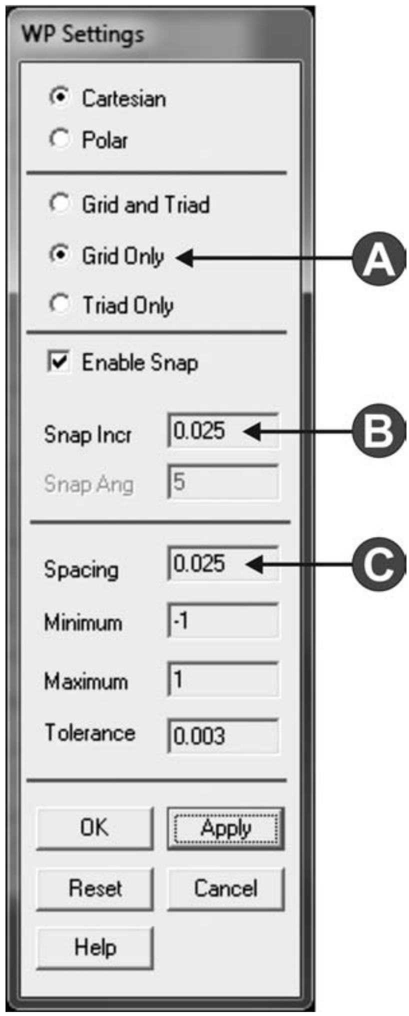
ANSYS graphics show the vector for the velocity

The red contours are for the maximum velocity in the channel, while the blue contours are for minimum velocity. The velocity is maximum at the centerline and zero at the wall. The developing region is clearly visible at the entrance region of the channel. Notice that the velocity has a parabolic velocity profile at the exit of the channel. To determine

the entrance length, a plot of the x-velocity component along the channel's centerline is created using the path operation in the postprocessor. The x-velocity component should be increased from its initial value at the inlet until it becomes unchanged. The path is created by specifying two points, one at the inlet and the other at the exit along the centerline. The number of division is the number of points used to create the plot. A higher number of divisions will create a smoother plot.

ANSYS Utility Menu > WorkPlane > Display Working Plane

Utility Menu > WorkPlane > WP Setting



A select Grid Only

B type 0.025 in Snap Incr

C type 0.025 in Spacing

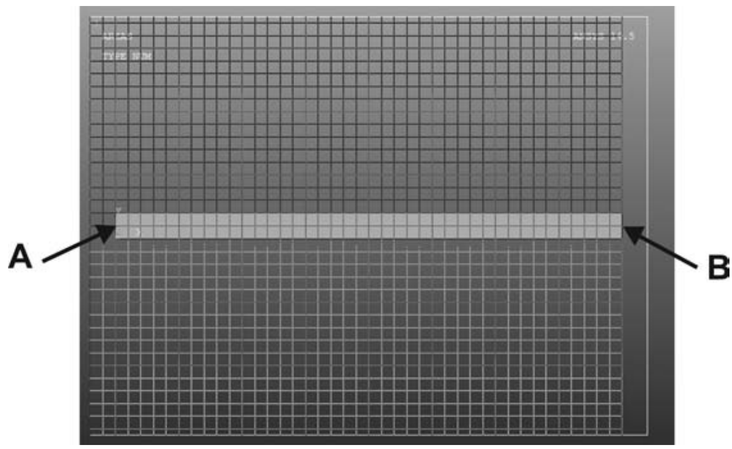
$\boxed{\textbf{OK}}$

Main Menu > General Postproc > Path Operations > Define Path > On Working Plane

The path is arbitrary. In On Working Plane window, click on

OK

Click on the ANSYS graphics window at the right and left ends of the centerline of the channel as shown below.

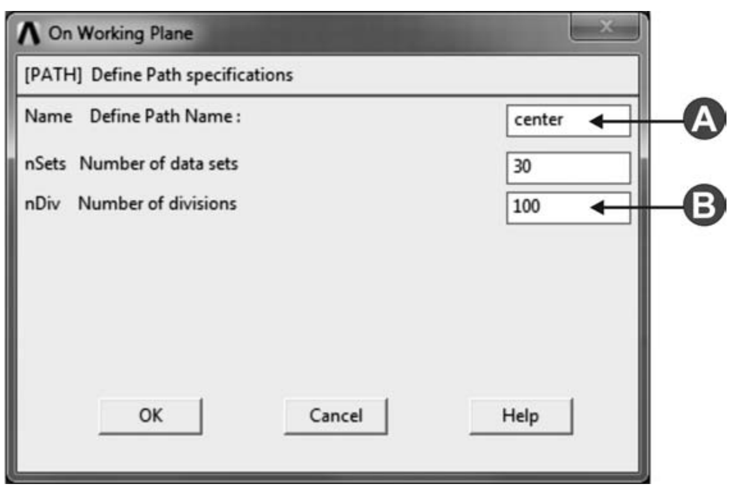


A click on left end of the centerline

B click on right end of the centerline

In On Working Plane window, click on

OK

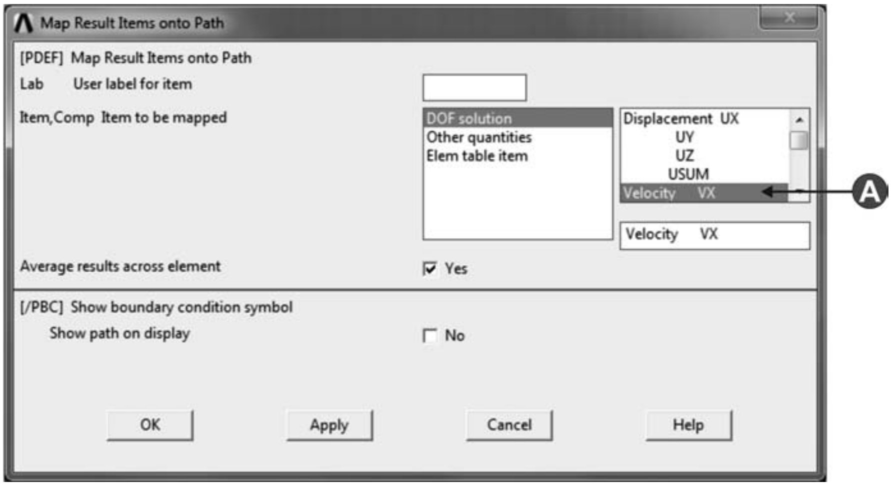


A type center in Define Path Name; the name of the path is optional

B type 100 in Number of divisions

OK

Main Menu > General Postproc > Path Operation > Map onto Path



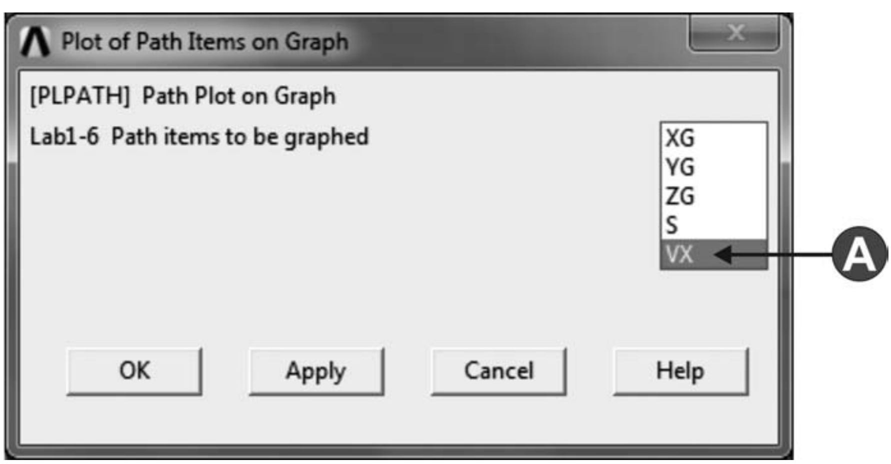
A select Velocity VX

OK

The velocity in the x-direction is ready to be plotted. In the Plot Path Item, there are two options. The velocity can be either plotted or listed. The list results can be exported to another graphical software such as EXCEL. First, the grids are removed from ANSYS graphics.

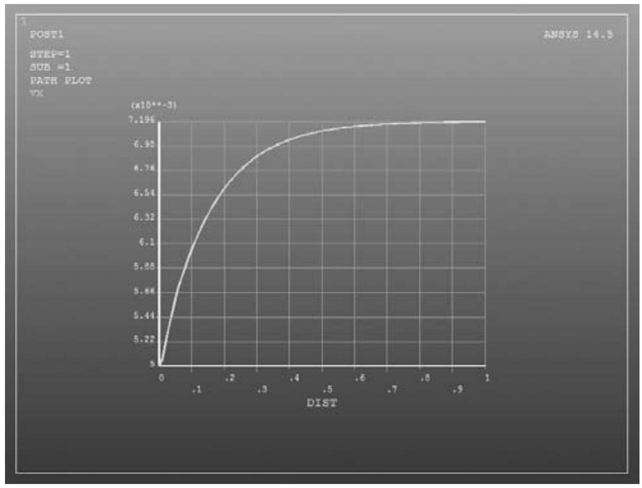
ANSYS Utility Menu > WorkPlane > Display Working Plane

Main Menu > General Postproc > Path Operation > Plot Path Item > On Graph



A select VX

OK



ANSYS graphics show the x-velocity profile along the centerline of the channel

The figure indicates that the flow is fully developed at the exit of the channel. The x-velocity component becomes invariant at a distance of 0.75 m from the entrance. Experimentally, the entrance length can be determined using the following equation:

L = 0.056 Re H

where Re is the Reynolds number and H is the channel's height. The Reynolds number is calculated using the following expression:

$$Re = \frac{\rho VH}{\mu} = \frac{998.3 \times 0.005 \times 0.05}{1.002 \times 10^{-3}} = 249.07$$

Then, the entrance length is 0.697 m, which is close to the ANSYS results. The error between the two methods can be additionally reduced if a finer mesh is used. To determine the average velocity at the exit, the path operation is used to plot the velocity profile with integration.

Utility Menu > Plot > Areas

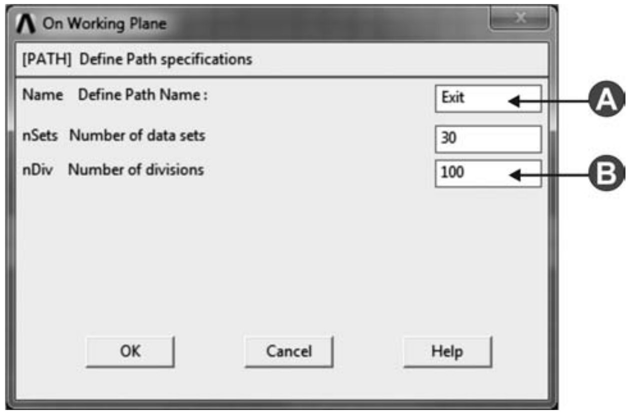
Main Menu > General Postproc > Path Operation > Define Path > On Working Plane

The path is arbitrary. In On Working Plane window, click on

OK

Click on the ANSYS graphics window at the top and bottom corners at the exit of the channel, and then in On Working Plane window, click on

OK

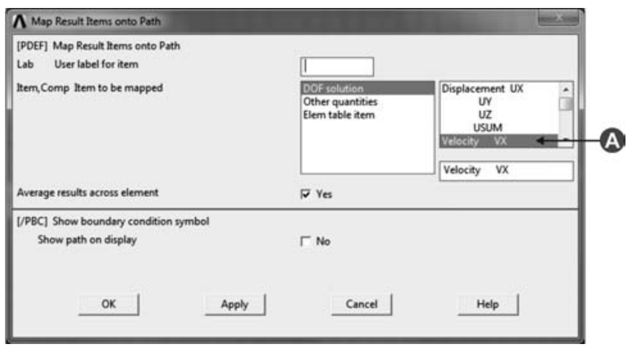


A type Exit in Define Path Name; the name of the path is optional

B type 100 in Number of divisions

OK

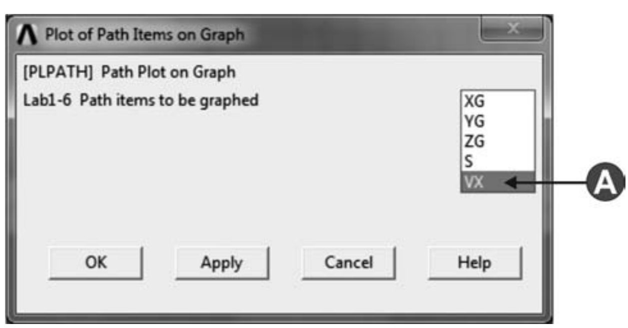
Main Menu > General Postproc > Path Operation > Map onto Path



A select Velocity VX

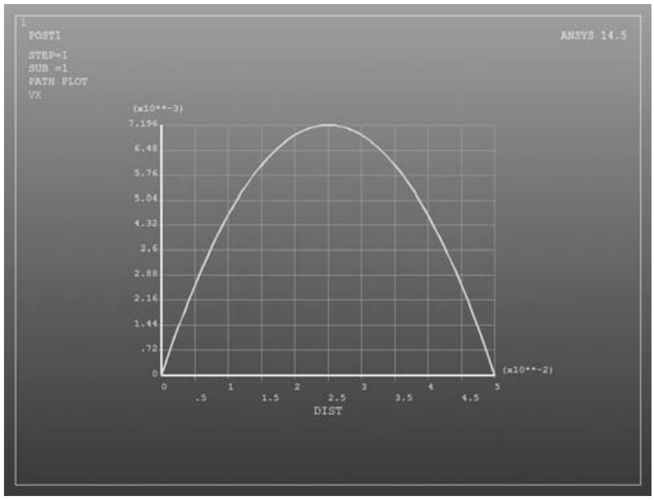
OK

Main Menu > General Postproc > Path Operation > Plot Path Item > On Graph



A select VX

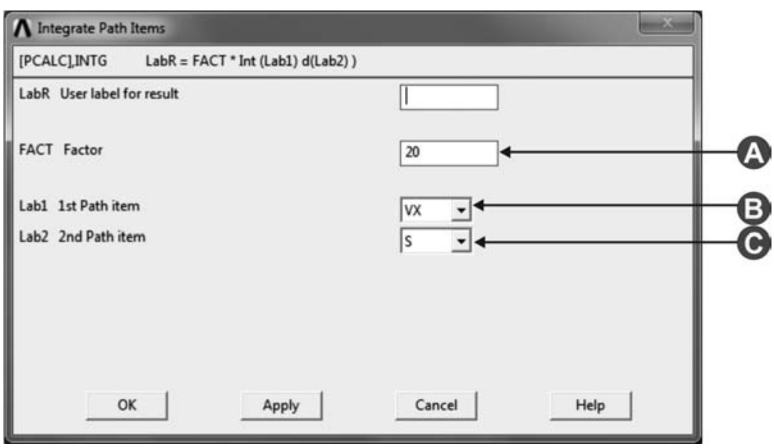
OK



ANSYS graphics show the x-velocity profile at the exit of the channel

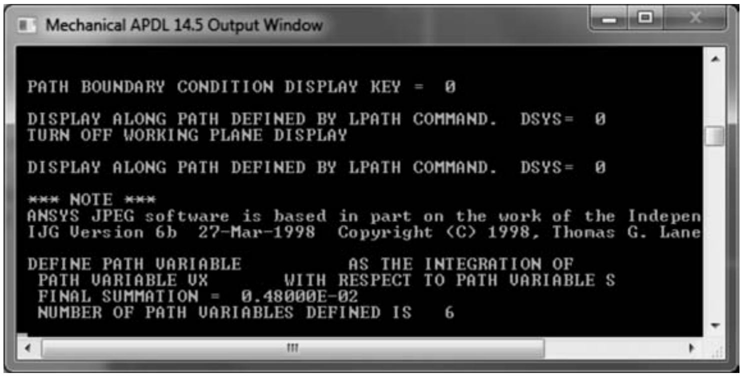
The velocity profile at the exit is perfectly parabolic. This graph can be compared to the analytical velocity profile of a fully developed flow to ensure that the obtained solution is accurate. The average velocity at the exit can be determined using the integration in the path operation. The value of the integration must be divided by the path length to get the average value. The number 20 in the Factor is the inverse of the path length. Selecting S in the Lab2 means that the integration is performed along the path.

Main Menu > General Postproc > Path Operation > Integrate



A type 20 in FACT Factor
B select VX in 1st Path item
C select S in 2nd Path item

OK



```
Mechanical APDL 14.5 Output Window

PATH BOUNDARY CONDITION DISPLAY KEY =  0

DISPLAY ALONG PATH DEFINED BY LPATH COMMAND.  DSYS=  0
TURN OFF WORKING PLANE DISPLAY

DISPLAY ALONG PATH DEFINED BY LPATH COMMAND.  DSYS=  0

*** NOTE ***
ANSYS JPEG software is based in part on the work of the Indepen
IJG Version 6b  27-Mar-1998  Copyright (C) 1998, Thomas G. Lane

DEFINE PATH VARIABLE          AS THE INTEGRATION OF
 PATH VARIABLE UX       WITH RESPECT TO PATH VARIABLE S
 FINAL SUMMATION =  0.48000E-02
 NUMBER OF PATH VARIABLES DEFINED IS   6
```

The ANSYS Output window shows the average velocity at the exit, which is 0.48×10^{-2} m/s. The mass flow rate at the exit and inlet should be the same as follows:

$$\dot{m}_i = \dot{m}_e$$

or

$$(\rho VA)_i = (\rho VA)_e$$

Since the density and cross-sectional area of the inlet and exit are the same, then

$$V_i = V_e$$

Comparing the inlet velocity, which is 0.005 m/s, to the exit velocity, which is 0.48×10^{-2} m/s, the two velocities are close to each other with an error of 4%.

6.4 Studying flow around a half cylinder in a channel using ANSYS

Air at low velocity enters a channel as shown in Figure 6.3. A half cylinder with a circular cross section is placed at the bottom wall of the channel. The inlet velocity has fully developed velocity profile with an average velocity of $U_m = 0.005$ m/s. The exit condition is a reference zero pressure. Air's density and viscosity are $\rho = 1.25$ kg/m^3 and

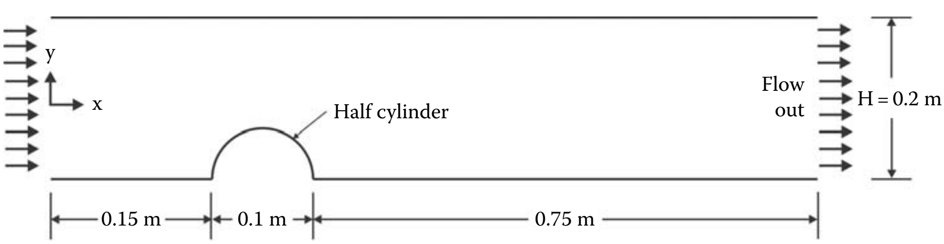


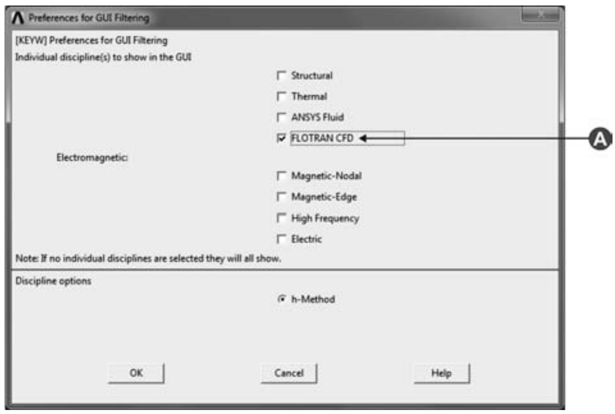
FIGURE 6.3 Flow over a half cylinder in a channel.

μ = 17.7 × 10^{-6} Pa·s, respectively. Determine the pressure drop in the channel and drag force on the half cylinder. Use the following equation for the velocity profile at the inlet:

$$u(y) = \frac{3}{2} U_m \left[1 - \left(\frac{2y}{H}\right)^2\right]$$

Double click on the Mechanical APDL Product Launcher icon

Main Menu > Preferences

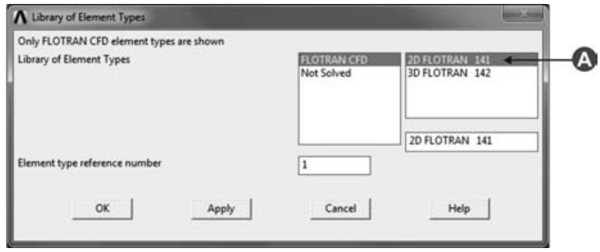


A select FLOTRAN CFD

OK

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Add...



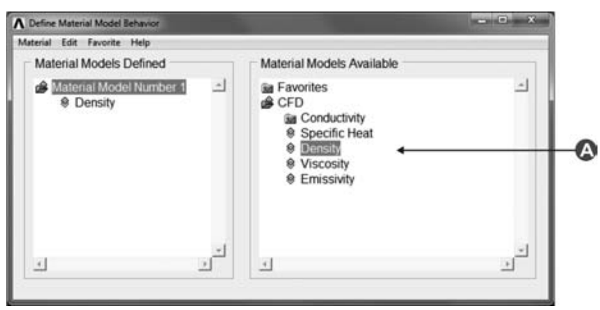
A select 2D FLOTRAN 141

OK

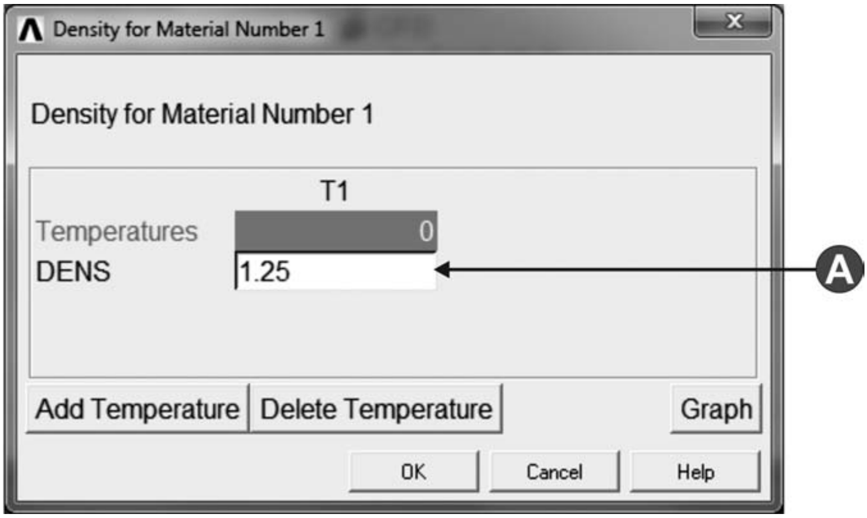
Close

Since this problem is fluid dynamics, density and viscosity are required to solve this problem. The geometry is modeled by creating a rectangle and a circle. The Boolean operation is utilized to remove the circle from the rectangle using subtraction.

Main Menu > Preprocessor > Material Props > Material Models



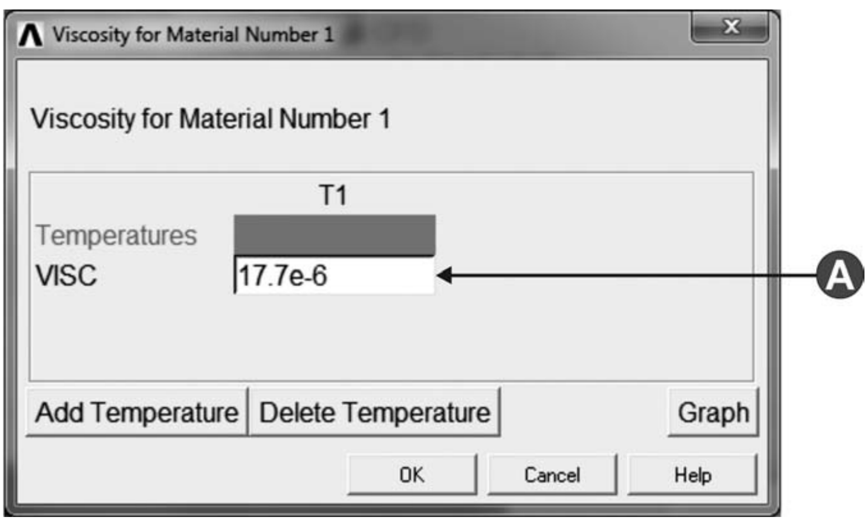
A click on CFD > Density



A type 1.25 in DENS

OK

Click on CFD > Viscosity

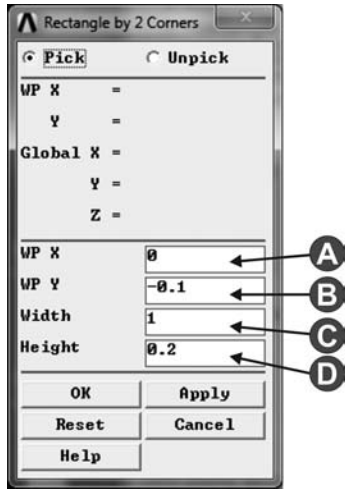


A type 17.7e-6 in VISC

OK

Close the Define Material Model Behavior window

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

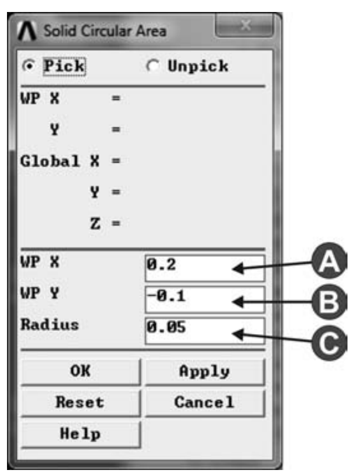
B type –0.1 in WP Y

C type 1 in Width

D type 0.2 in Height

OK

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle



A type 0.2 in WP X

B type –0.1 in WP Y

C type 0.05 in Radius

OK

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

Click on rectangle area, and then in Subtract Area window, click on

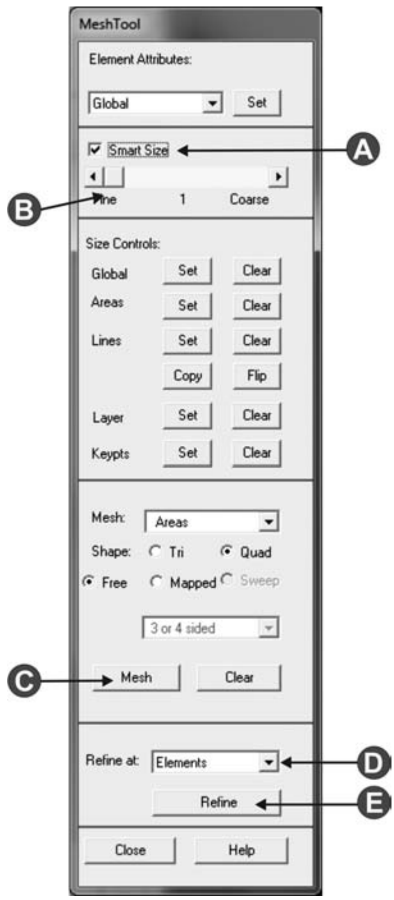
Apply

Click on circular area, and then in Subtract Area window, click on

OK

A free mesh is generated using the smart mesh option, and the mesh refinement is 1. More elements will be added to the computational domain using refinement at elements.

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Smart Size

B set the level to 1

C Click on Mesh

In Mesh Areas window, click on

Pick All

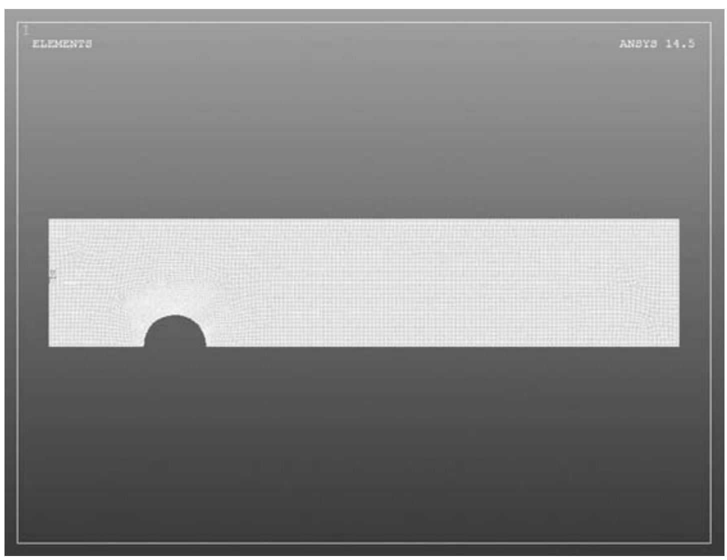
D select Elements

E click on Refine

In Refine Mesh at Area window, click on

Pick All

OK

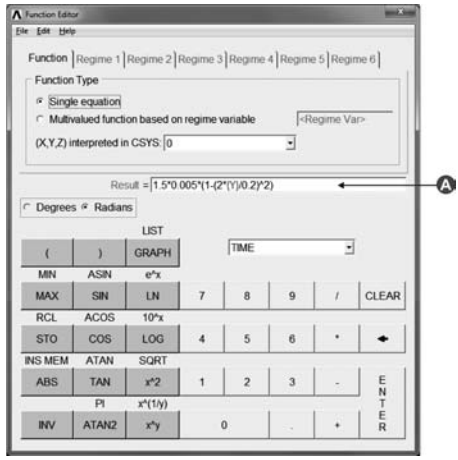


ANSYS graphics show the final mesh

The modeling and meshing tasks are now completed. The hydrodynamics boundary conditions are applied. At the inlet, Vx is a function of the y-direction and the Vy is 0. At the exit, zero pressure is imposed to simulate a free exit boundary condition. Zero velocity components are applied at the surface of the cylinder and lateral surfaces of the channel to simulate a wall boundary condition.

The inlet velocity profile can be applied easily with the function editor of ANSYS. ANSYS will create a data table from the velocity function and apply it to the selected lines.

Main Menu > Solution > Define Loads > Apply > Functions > Define/Edit



A type the equation 1.5*0.005*(1–(2*{Y}/0.2)^2)

In Equation Editor window, click on File then Save

Save the file as VinY. The file name is optional.

Save

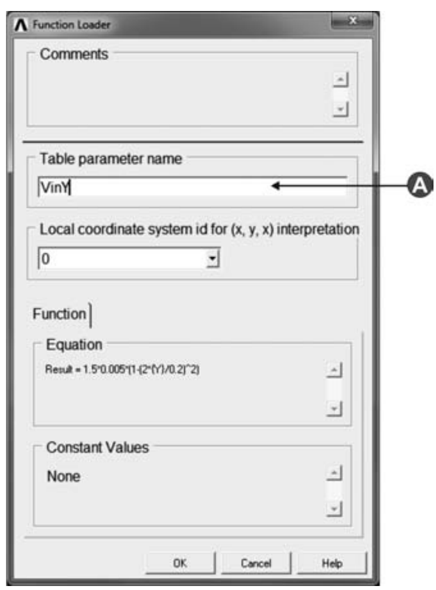
Close the Function Editor window

After saving the function, it is required to load the function to the ANSYS solution using the read file in function.

Main Menu > Solution > Define Loads > Apply > Functions > Read File

Select VinY.func

Open



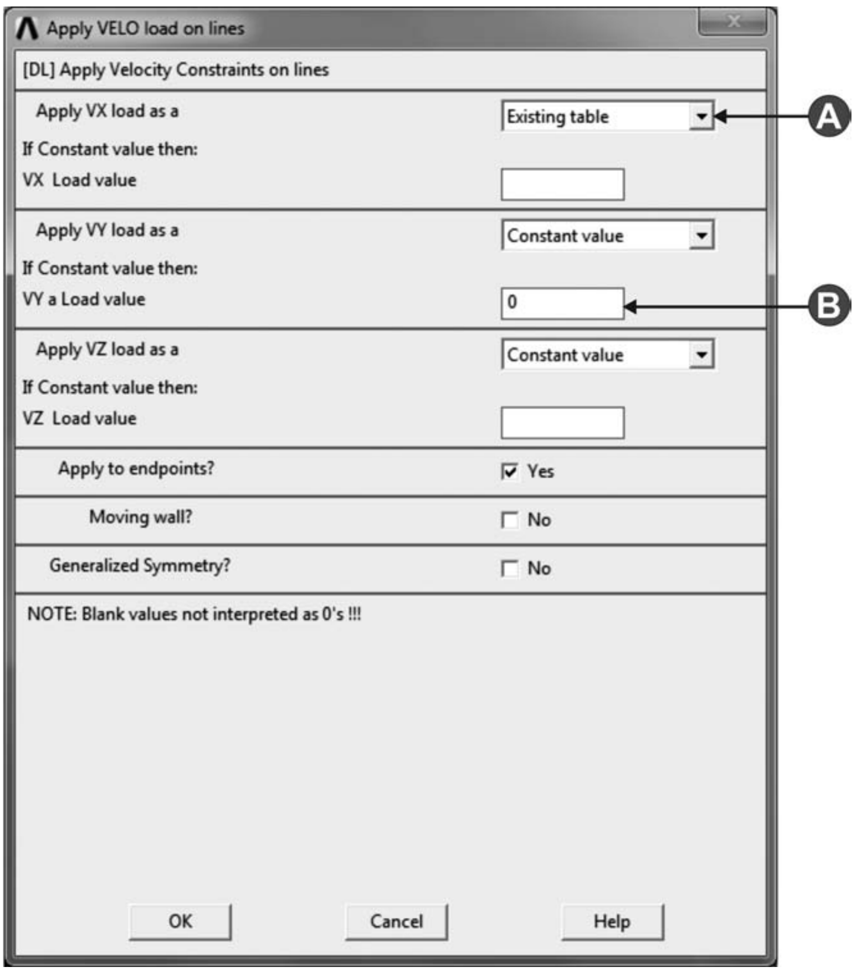
A type VinY in the Table parameter name. The name of the table is optional. The function "VinY" will be shown later when the boundary conditions are applied.

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click on the channel entrance line, and then in Apply VELO load on lines window, click on

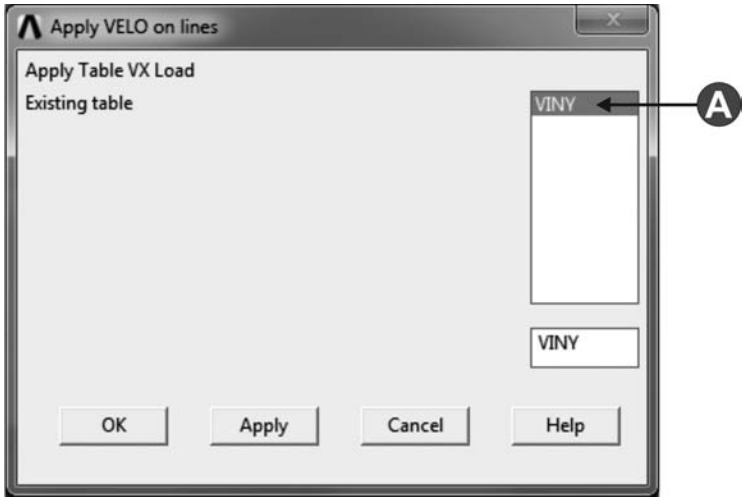
OK



A select Existing table in Apply VX load as a

B type 0 in VY a Load value

OK



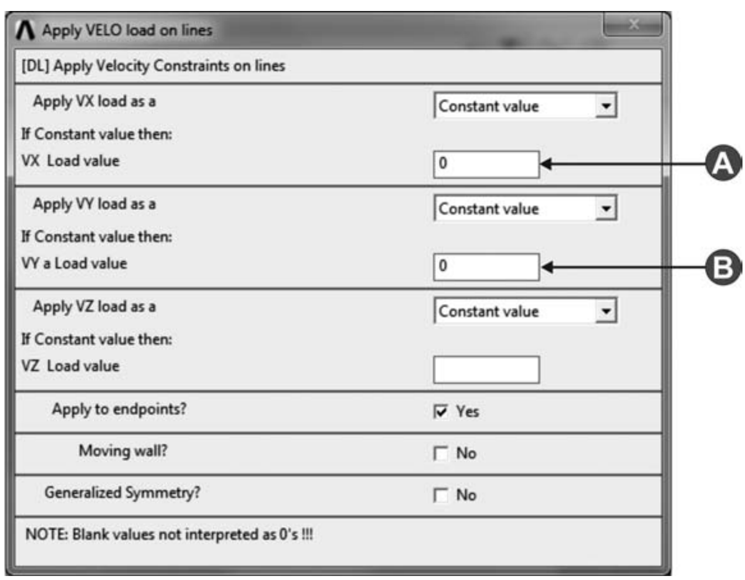
A select VINY

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click on the upper and lower lines of the channel, and the surface of the cylinder, and then in Apply VELO load on lines window, click on

OK



A type 0 in VX Load value

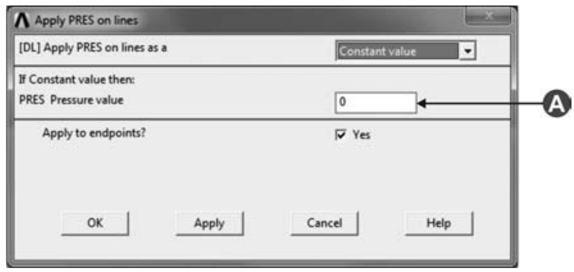
B type 0 in VY a Load value

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Pressure DOF > On Lines

Click on the channel's exit line, and then on Apply PRES on lines window, click on

OK

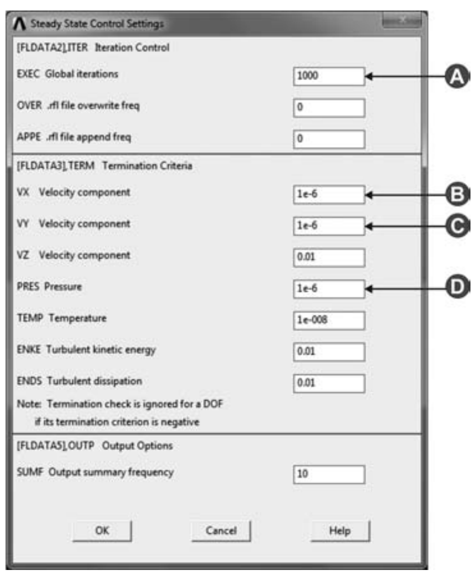


A type 0 in PRES Pressure value

OK

The maximum number of iterations is 1000, and an additional 1000 iterations is required if the termination criterion was not satisfied. The termination criterion for the velocity components and pressure is 1×10^{-6}. The iterations will stop if the maximum number of iterations is reached or the termination criterion is satisfied for all field variables.

Main Menu > Solution > FLOTRAN Set Up > Execution Ctrl



A type 1000 in EXEC Global iterations

B type 1e-6 in VX Velocity component

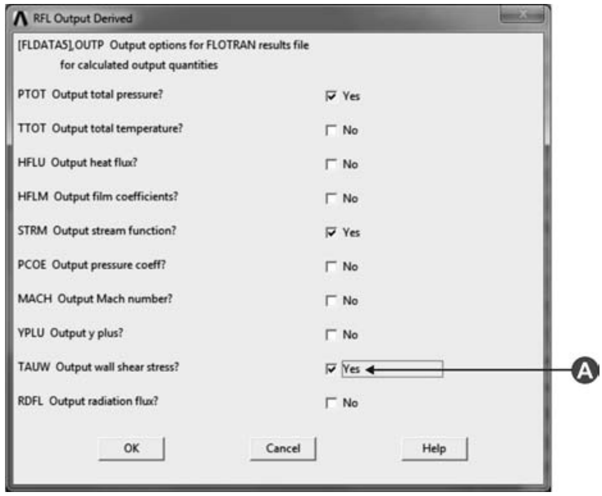
C type 1e-6 in VY Velocity component

D type 1e-6 in PRES Pressure

OK

By default, the shear stress will not be calculated by ANSYS, which is required to calculate the lift and drag forces. To calculate the shear stress, it must be selected in the additional output in the FLOTRAN setup.

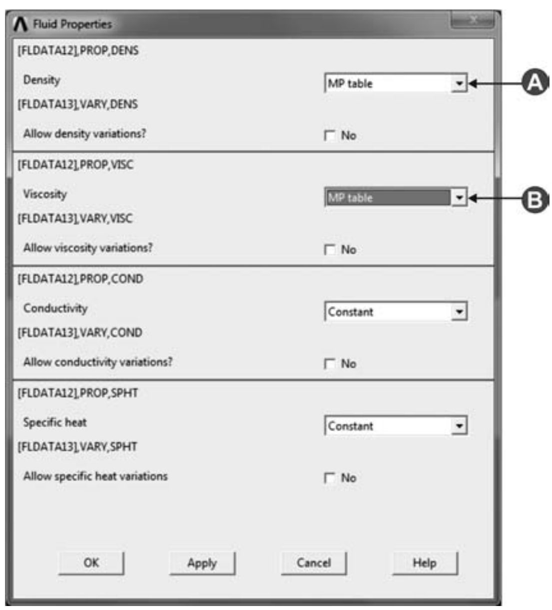
Main Menu > Solution > FLOTRAN Set Up > Additional Out > RFL Output Derived



A select TAUW Output wall shear stress?

OK

Main Menu > Solution > FLOTRAN Set Up > Fluid Properties



A select MP table in Density

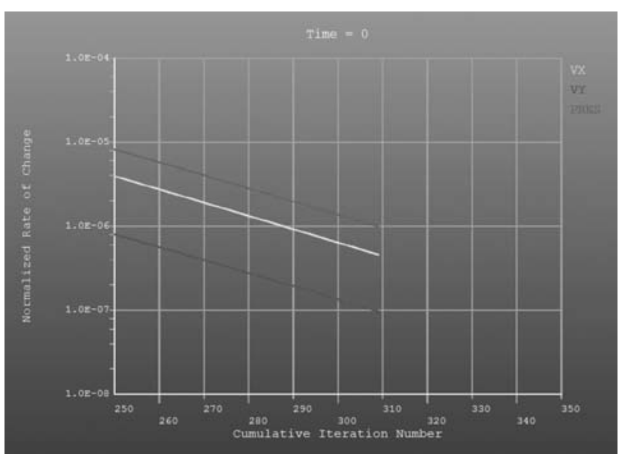
B select MP table in Viscosity

OK

OK

Main Menu > Solution > FLOTRAN Set Up > Run FLOTRAN

OK

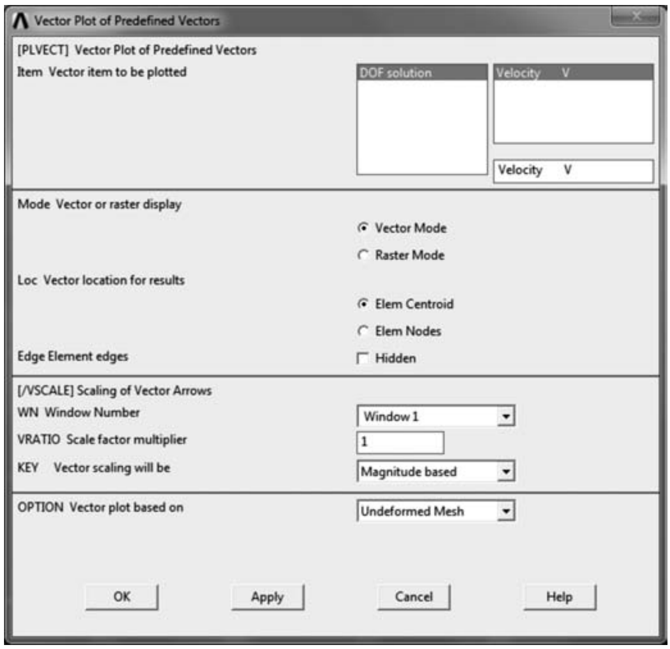


ANSYS graphics show the solution convergence

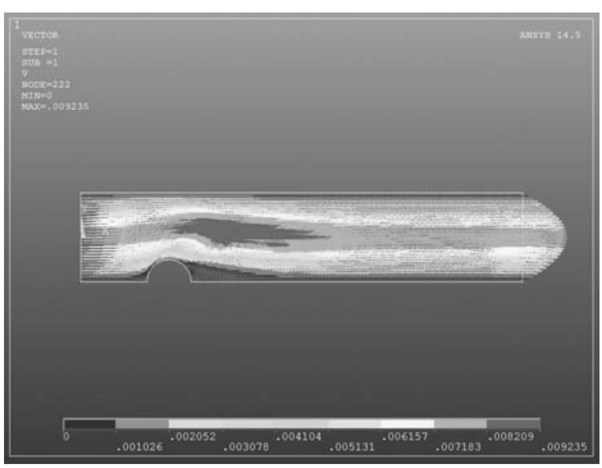
As shown in the normalized rate of change, the solution did not reach the maximum number of iterations, which is 1000, and all field variables reach the termination criterion, which is 1×10^{-6}. Therefore, the solution is converged. A plot of velocity vectors is presented in the postprocessor task, followed by pressure drop, and lift and drag calculations.

Main Menu > General Postproc > Read Results > Last Set

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined



OK



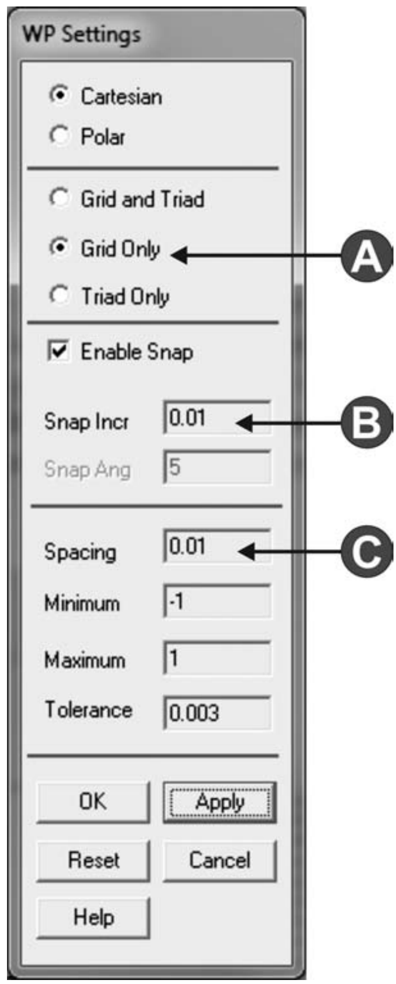
ANSYS graphics show the vector for the velocity

The red arrows are for the maximum velocity in the channel, while the blue arrows are for minimum velocity. The wake flow behind the cylinder is visible in the figure, and the flow is very slow at this region. The flow velocity is maximum above the cylinder. Notice that the velocity has a parabolic profile at the inlet and exit of the channel. To calculate the pressure drop, the average pressure at the inlet is determined using path operation in the postprocessor. The path at the inlet is created by specifying two points. The grids should be enabled.

Utility Menu > Plot > Areas

Utility Menu > WorkPlane > Display Working Plane

Utility Menu > WorkPlane > WP Setting



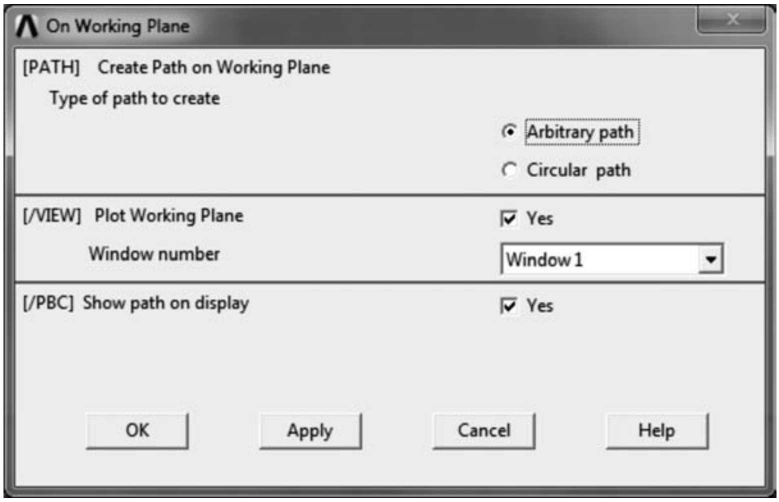
A select Grid Only

B type 0.01 in Snap Incr

C type 0.01 in Spacing

OK

Main Menu > General Postproc > Path Operation > Define Path > On Working Plane



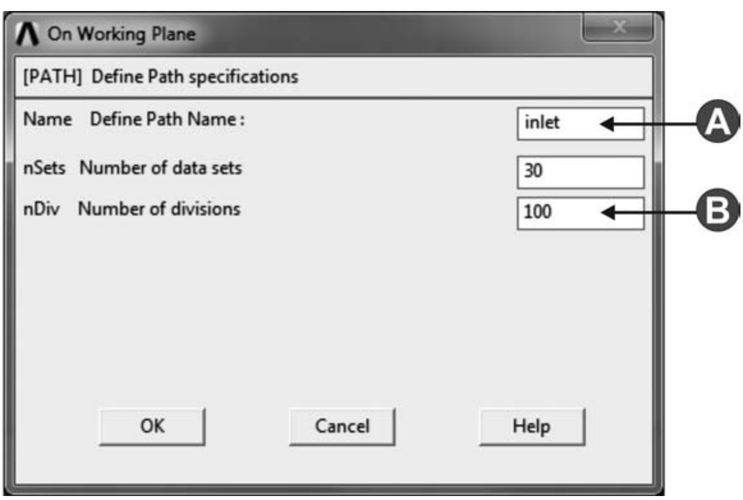
OK

ANSYS Utility Menu > PlotCtrls > Pan-Zoom-Rotate …

Click on zoom in and out until the ANSYS graphics show all grids.

Click on the ANSYS graphics window at the top and bottom corners at the inlet of the channel, and then in On Working Plane window, click on

OK

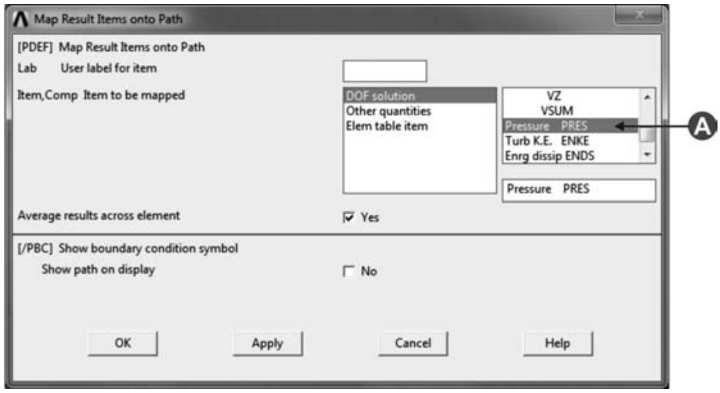


A type inlet in Define Path Name. The name of the path is optional

B type 100 in Number of divisions

OK

Main Menu > General Postproc > Path Operation > Map onto Path

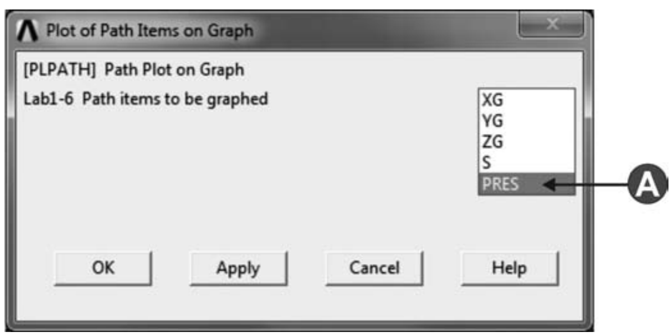


A select Pressure PRES

OK

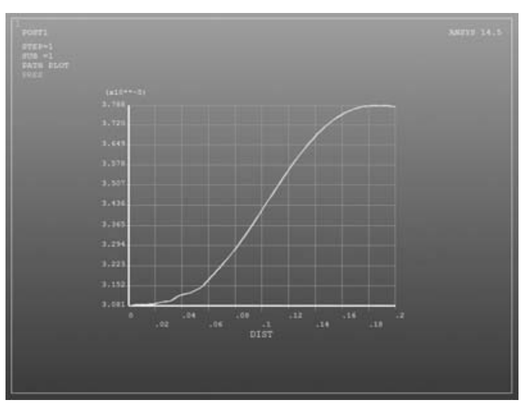
Utility Menu > WorkPlane > Display Working Plane

Main Menu > General Postproc > Path Operation > Plot path Item > On Graph



A select PRES

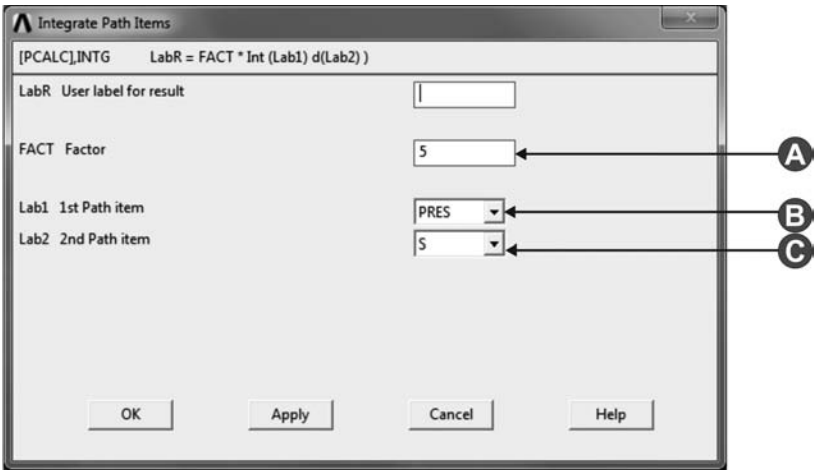
OK



ANSYS graphics show the pressure distribution at the inlet of the channel

The average pressure at the inlet can be determined using the integration in the path operation. The value of the integration must be divided by the path length to get the average value of the variable. The number 5 in the Factor is the inverse of the path length. Selecting S in the Lab2 means that the integration is performed along the path.

Main Menu > General Postproc > Path Operation > Integrate

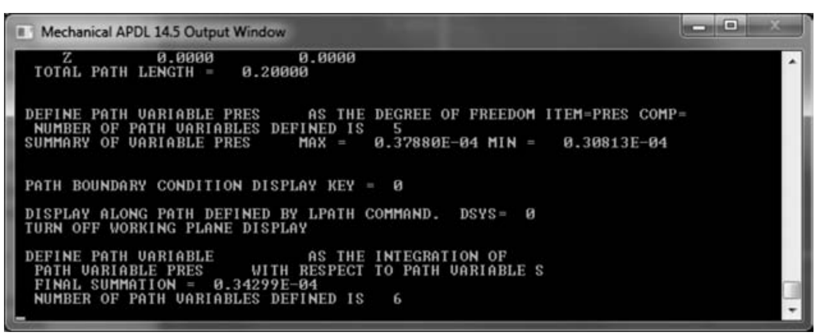


A type 5 in FACT Factor

B select PRES in Lab1 1st Path item

C select S in Lab2 2nd Path item

OK



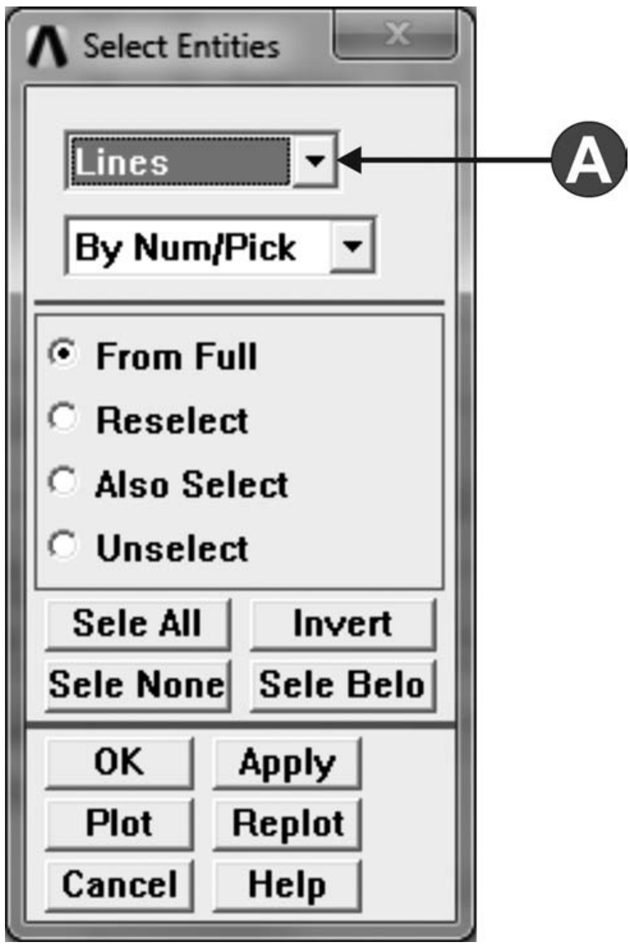
The ANSYS Output window shows the average pressure at the inlet, which is 0.34299×10^{-4} N/m^2. At the exit, the pressure is specified as a boundary condition and equal to zero. Hence, the pressure drop in the channel is equal to 0.34299×10^{-4} N/m^2. To determine the lift and drag forces on the surface of the cylinder, nodes along

the surface of the cylinder should be selected first. Then, ANSYS automatically integrates the pressure and shear stress along the surface of the cylinder. The nodal selection is performed in the following steps:

Utility Menu > Plot > Areas

Utility Menu > PlotCtrls > Pan-Zoom-Rotate …
Zoom the area around the cylinder.

Utility Menu > Select > Entities



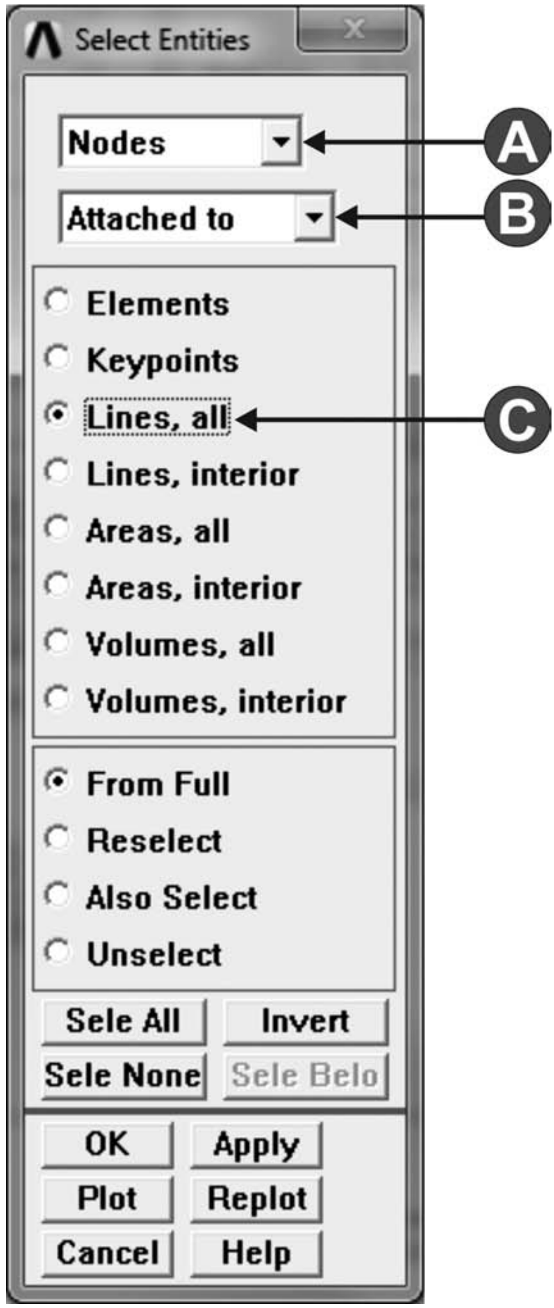
A select Lines

OK

Carefully select the two lines at the surface of the cylinder, and then in Select Lines window, click on

OK

Utility Menu > Select > Entities



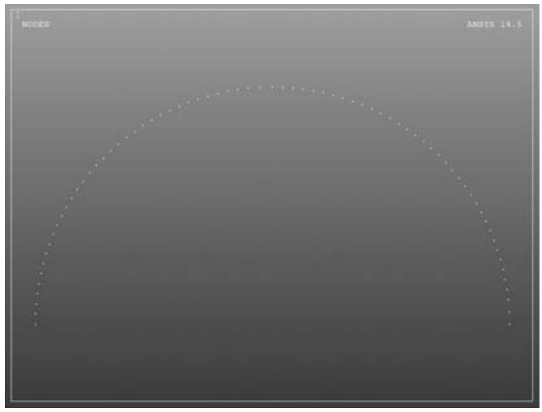
A select Nodes

B select Attached to

C select Lines, all

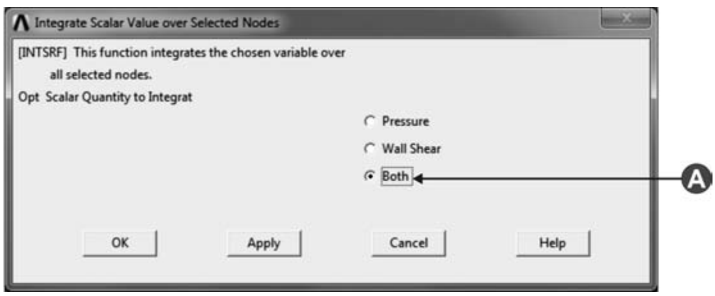
OK

Utility Menu > Plot > Nodes



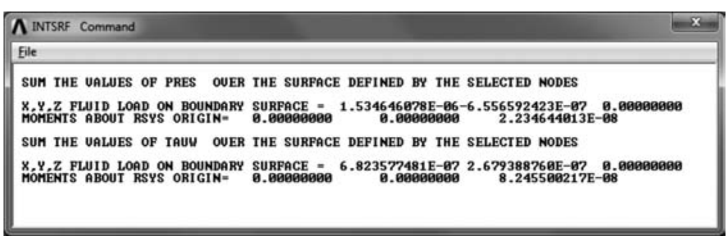
ANSYS graphics show the nodes at the cylinder

Main Menu > General Postproc > Nodal Calcs > Surface Integral



A select Both

OK



ANSYS shows the net pressure in the x- and y-directions, which are

$P_x = 1.534646078 \times 10^{-6}\ \text{N/m}^2$
$P_y = -6.556592423 \times 10^{-7}\ \text{N/m}^2$

Notice that P_y has a negative value because the pressure in the y-direction is downward. The net shear stresses in the x- and y-directions are

$\tau_x = 6.823577481 \times 10^{-7}\ \text{N/m}^2$
$\tau_y = 2.67938876 \times 10^{-7}\ \text{N/m}^2$

Hence, the net drag force is $(P_x + \tau_x) \times S$, and the net lift force is $(P_y + \tau_y) \times S$, where S is the circumference of the half cylinder, and it is equal to 0.05π. To visualize the flow streamlines, the flow is traced with particles. First, the trace points will be defined at the entrance and behind the cylinder to capture the wake flow. Then, the trace particles are plotted. Finally, an animation is created.

Utility Menu > Select > Everything

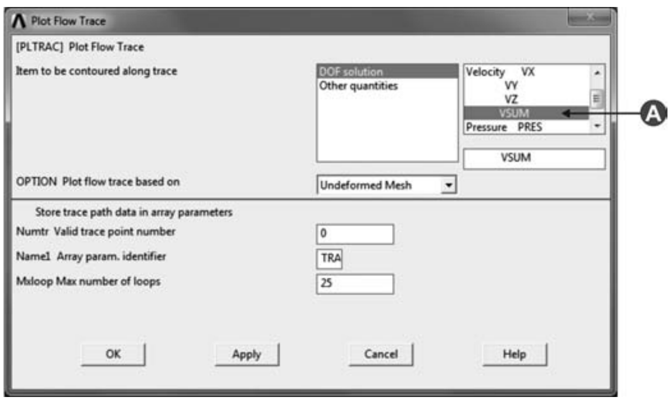
Utility Menu > Plot > Areas

Main Menu > General Postproc > Plot Results > Defi Trace Pt

Click on all grid points at the entrance and just one grid point behind the cylinder, and then in Define Trace Point window, click on

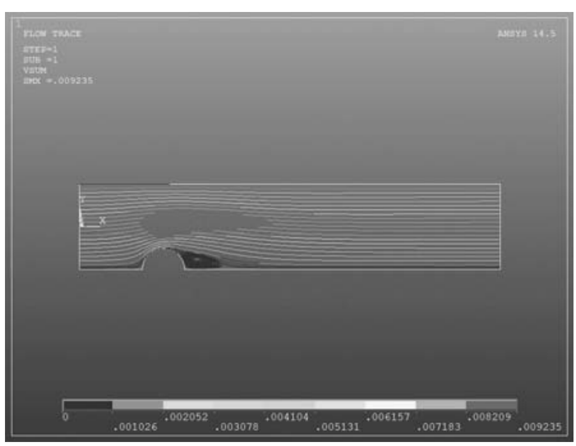
OK

Main Menu > General Postproc > Plot Results > Plot Flow Tra



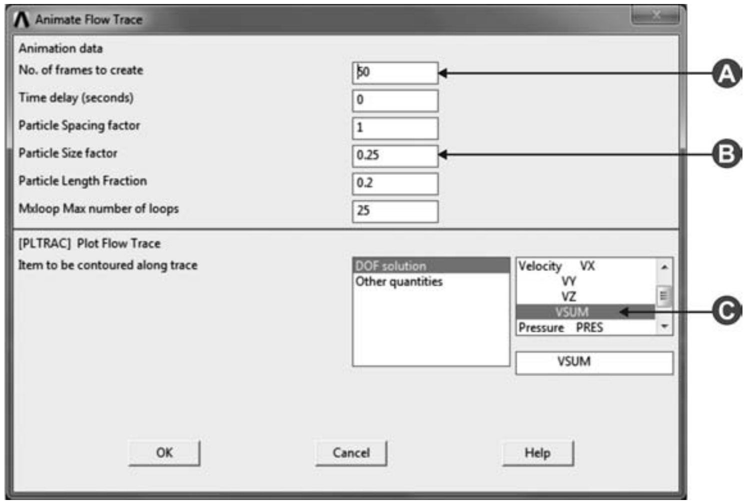
A select VSUM

OK



ANSYS graphics show the streamlines

Utility Menu > PlotCtrls > Animate > Particle Flow



A type 50 in No. of frames to create
B type 0.25 in Particle Size factor
C select VSUM in Plot Flow Trace

OK

ANSYS graphics show the animation of the particles flow in the channel.

PROBLEM 6.1

Air at low velocity enters a channel as shown in Figure 6.4. An object is placed in the channel, at the location shown. The inlet velocity is uniform, 0.005 m/s. The exit condition is a reference zero pressure. Let $\rho = 1.25$ kg/m^3 and $\mu = 17.7 \times 10^{-6}$ Pa·s. Determine the pressure drop in the channel and net drag force on the object. Also show that the mass conservation principle is satisfied.

PROBLEM 6.2

Water at low velocity enters a channel as shown in Figure 6.5. A triangular cross-sectional area cylinder is placed in the channel, at the location shown. The inlet velocity has a fully developed profile. The exit condition is a reference zero pressure. The properties of water are

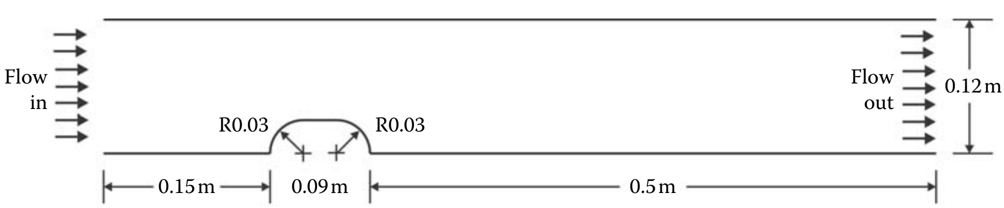


FIGURE 6.4 Flow over an object in a channel.

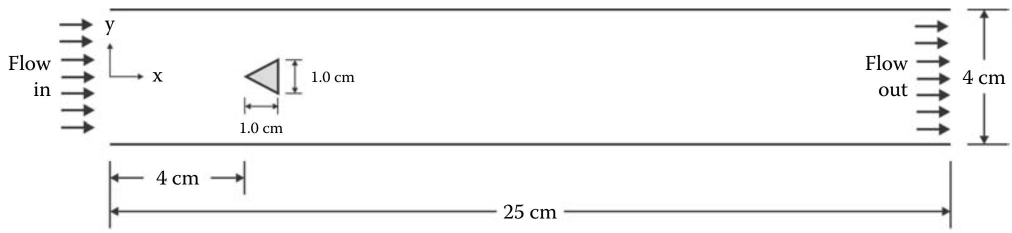


FIGURE 6.5 Channel with a triangular cylinder.

ρ = 998.3 kg/m^3 and μ = 1.002 × 10^{-3} Pa·s. Use the following equation for the velocity profile at the inlet:

$$u(y) = \frac{3}{2}\,0.0001\left[1 - \left(\frac{2y}{H}\right)^2\right]$$

where H = 0.04 m. Determine the pressure drop in the channel, and drag and lift forces on the cylinder, and show that the mass conservation principle is satisfied.

PROBLEM 6.3

Two cylinders in a wind tunnel, as shown in Figure 6.6, are used to simulate heat exchanger tubes. The inlet velocity is 0.0025 m/s, and air is the working fluid (μ = 20 × 10^{-6} Pa·s, ρ = 1.25 kg/m^3). The exit condition is a reference zero pressure. Determine the following:

a. Is mass balance satisfied?
b. What is the maximum shear stress on the first and second cylinders?
c. What is the average shear stress on the first and second cylinders?
d. What is the pressure drop in the channel?
e. Is the exit flow fully developed?

PROBLEM 6.4

Water at low velocity enters a backward-facing step channel as shown in Figure 6.7. The height in the channel is increased at a distance 0.5 m from the entrance as shown in the figure. The inlet velocity has a fully

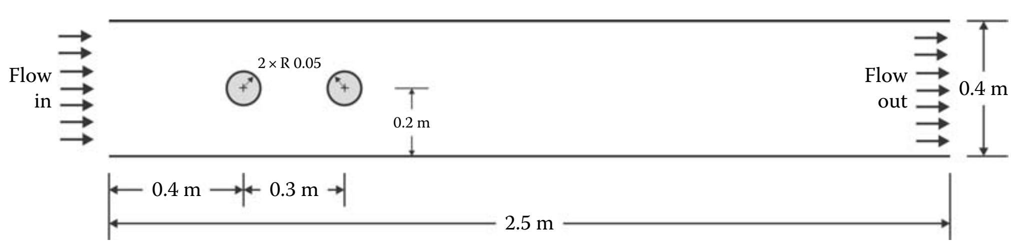


FIGURE 6.6 Two cylinders in a wind tunnel.

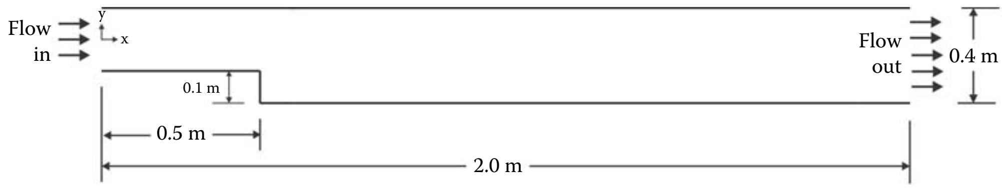


FIGURE 6.7 Flow over a backward-facing step.

developed profile. The exit condition is a reference zero pressure. Water is the working fluid, and its properties are $\rho = 997.1$ kg/m^3 and $\mu = 1.003 \times 10^{-3}$ Pa·s. Use the following equation for the velocity profile at the inlet:

$$u(y) = \frac{3}{2}\,0.001\left[1 - \left(\frac{2y}{H}\right)^2\right]$$

where H = 0.15 m. Flow over a backward-facing step generates a recirculation zone due to the separation flow obtained from the adverse pressure gradients in the fluid flow, and the flow reattached again. Determine the reattachment length.

CHAPTER SEVEN

Multiphysics

7.1 Introduction

In practice, the engineering problems to be simulated are multiphysics, and there has not been sufficient capability in the past to address the full physics of the problems. Therefore, there has been a tendency to simplify the problems by either focusing on the primary physics or decoupling the physics. For example, for heat exchanger analysis, fluid flow is solved first, followed by the thermal solution. Recently, significant improvements have been achieved in both hardware and software capabilities that made the multiphysics simulations inexpensive and accurate. As computing power increases, the ability to model the full physics is becoming a practical possibility. Hence, there is an increasing demand for multiphysics simulation software. ANSYS has the capability to model such complicated simulations. ANSYS can effectively simulate the thermal–structural, thermal–fluid, and fluid–structural problems.

In this chapter, the setting up of problems involving multiphysics is introduced. The level of coupling for multiphysics simulations is illustrated. Examples of thermal–structural and thermal–fluid systems are presented. The coupling between the physical phenomena can be classified into three levels: low, medium, and high. With a low level, it may be sufficient to use a simple one-way coupling with file transfer between two physics, as analysis is thermal–structural and thermal–fluid with temperature-dependent properties. In the thermal–structural system, the thermal analysis is done first, and then the temperature distribution is transferred to the structural analysis as a boundary condition. This analysis is fast, and the thermal and structural analysis is completely separate, because the structural deformation has a negligible effect on the temperature distribution. The process is shown in Figure 7.1.

The problem with a medium level of coupling is that the solution of each physics depends on another but requires no mesh movement, such as thermal–fluid analysis with temperature-dependent properties.

In the thermal–fluid analysis, the properties of the fluid, such as the density and viscosity, are a function of temperature. In this analysis, fluid is solved first, followed by thermal solution. The properties of the fluid are updated, and then the fluid is solved again. This process is continued until the solution is fully converged. The process is shown in Figure 7.2. If all properties of the fluid are independent of the temperature, the heat transfer and fluid flow can be solved separately, and the simulation is considered as a low-level analysis.

The fluid–structural analysis is considered as a high-level analysis. It requires a degree of compatibility in the solver technologies and often involves a mesh movement, which may be accounted for using the Arbitrary Lagrangian Eulerian method. The computation time for this analysis is high, and it requires high-performance computers. The process is shown in Figure 7.3. There are two types of fluid–structural analyses. In the first type, the solid is moving in a specified way and the fluid pressure has no effect on the movement of the solid. In the second type, which is much more complex, the fluid pressure affects the movement of the structure, and possible structural deformation occurs.

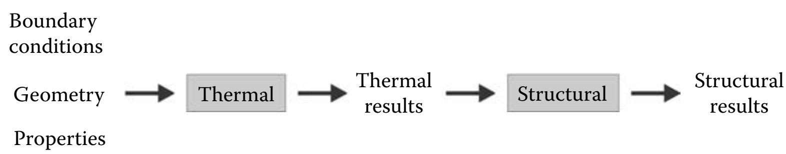


FIGURE 7.1 Low-level analysis.

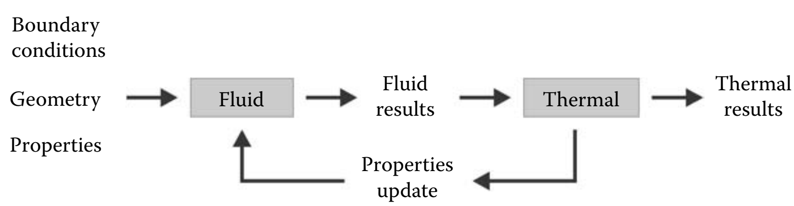


FIGURE 7.2 Medium-level analysis.

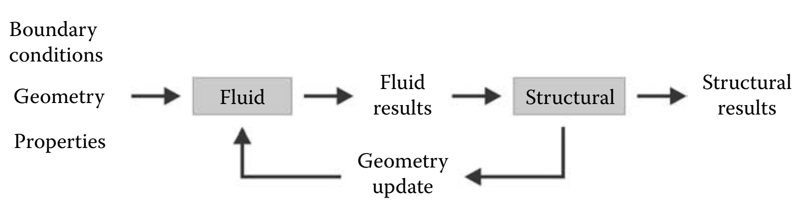


FIGURE 7.3 High-level analysis.

7.2 Thermal and structural analysis of a thermocouple using ANSYS

Furnaces use thermocouples to control their temperature, and they come in various designs. The common design of thermocouples is the two-plate design. It consists of aluminum and carbon steel plates attached to each other as shown in Figure 7.4. The left end is fixed and maintained at 100°C, and other surfaces are exposed to a free convection boundary condition with h = 5 $W/m^2 \cdot °C$ and T_∞ = 15°C. Determine the maximum displacement in the y-direction. The thermophysical properties of the aluminum and carbon steel are shown in Table 7.1.

This problem is considered as multiphysics, and physics are not coupled because the structure and heat transfer are solved separately. The deformation of the thermocouple has no effect on the heat transfer. The heat transfer must be solved first, and then the structure.

Double click on the Mechanical APDL Product Launcher icon

A solid thermal element is used, and its shape is quadratic with four nodes. This element will be replaced by a structural element when the thermal part is completely solved.

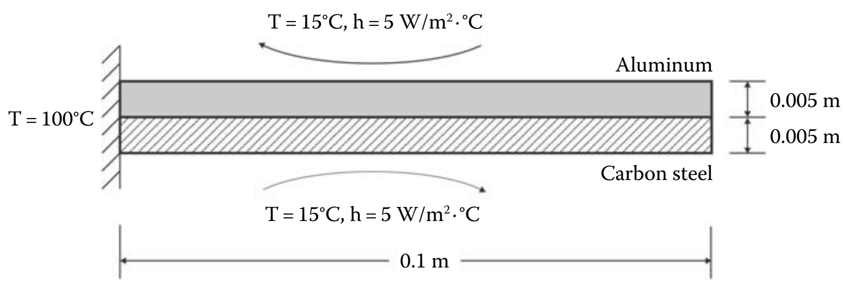
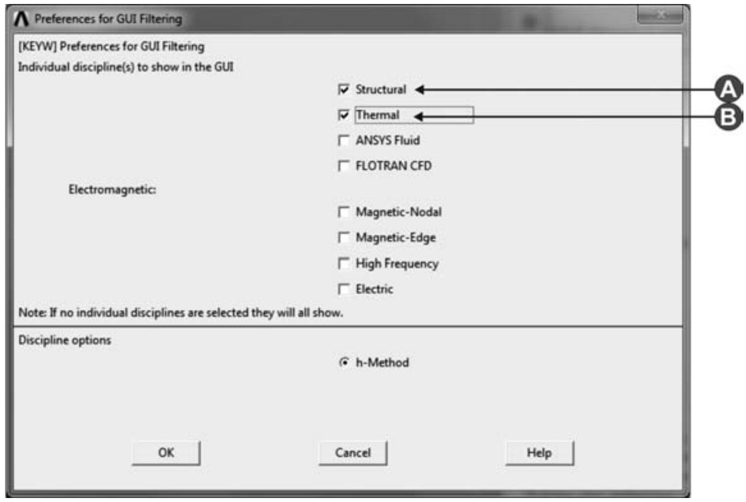


FIGURE 7.4 A thermocouple consists of aluminum and carbon steel plates.

Table 7.1 Thermophysical properties of aluminum and carbon steel

Property	Aluminum	Carbon steel
Thermal conductivity (W/m·K)	83	111
Young modulus (Pa)	70×10^9	210×10^9
Poisson's ratio	0.33	0.29
Thermal expansion (1/K)	23×10^{-6}	12×10^{-6}

Main Menu > Preferences



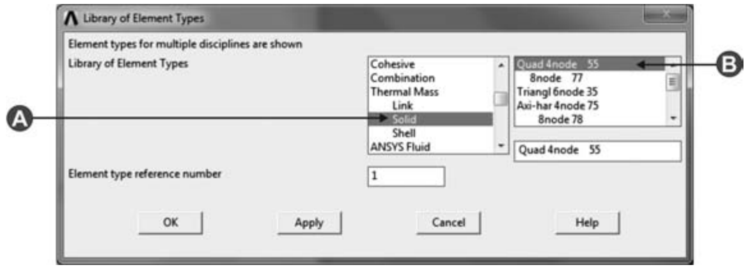
A select Structural

B select Thermal

OK

Main Menu > Preprocessor > Element Types > Add/Edit/Delete

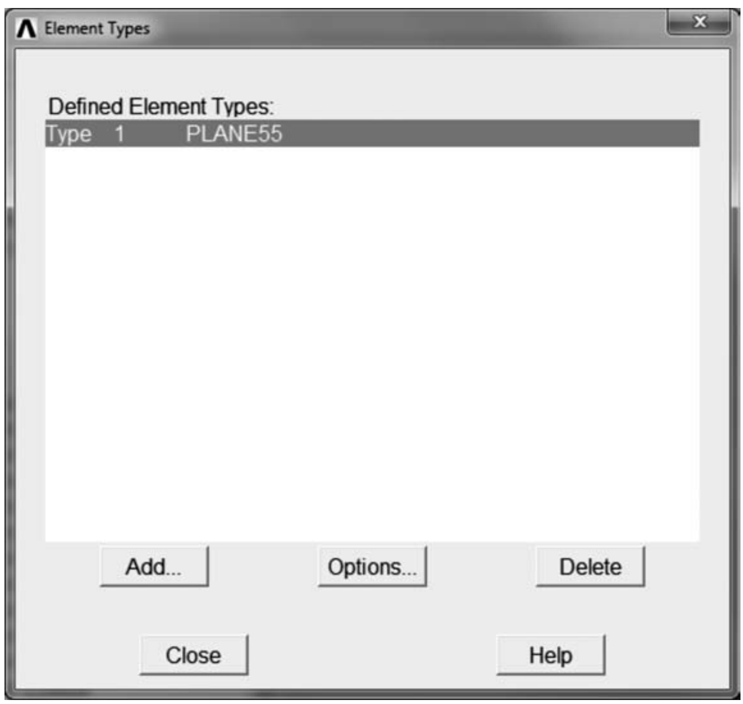
Add...



A select Solid in Thermal Mass

B select Quad 4 node 55

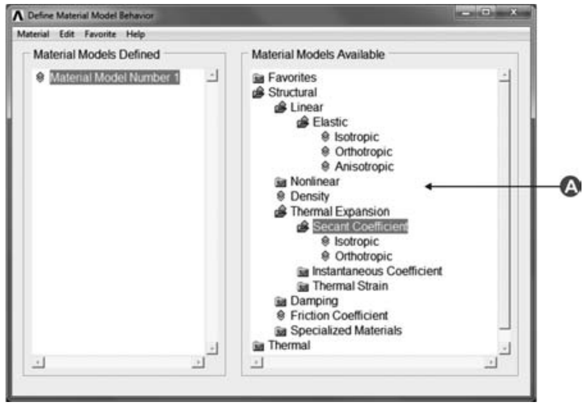
OK



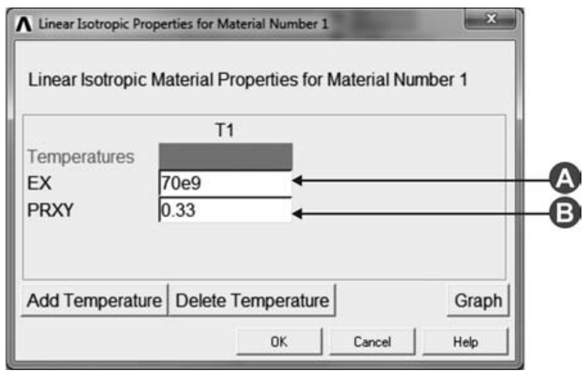
Close

The thermal conductivity is required for the thermal part, and the modulus of elasticity and Poisson's ratio are required for the structural part. Additionally, the thermal expansion of both materials is important to account for the deflection of the thermocouple. There will be two different sets of properties for aluminum and carbon steel. The material number 1 in the Define Material Model Behavior is for aluminum, while material number 2 is for carbon steel.

Main Menu > Preprocessor > Material Props > Material Models



A click on Structural > Linear > Elastic > Isotropic

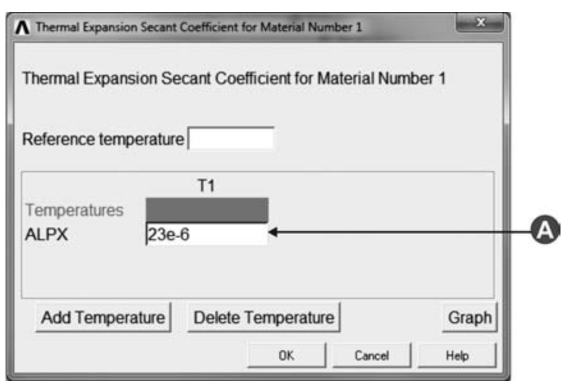


A type 70e9 in EX

B type 0.33 in PRXY

OK

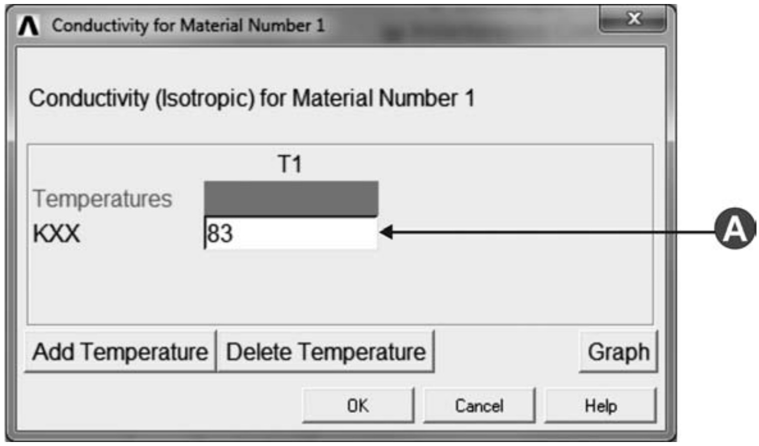
Click on Structural > Thermal Expansion > Secant Coefficient > Isotropic



A type 23e-6 in ALPX

OK

Click on Thermal > Conductivity > Isotropic



A type 83 in KXX

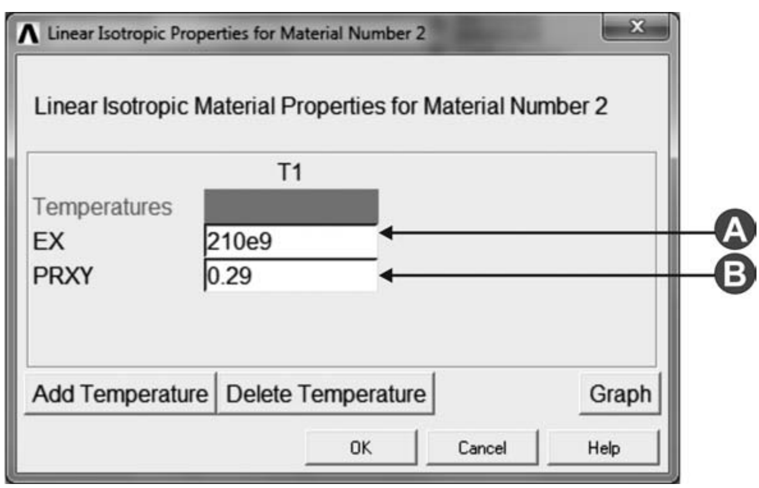
OK

In the Define Material Models Behavior menu: Material > New Model

OK

Select material number 2 in the Material Models Defined, and then

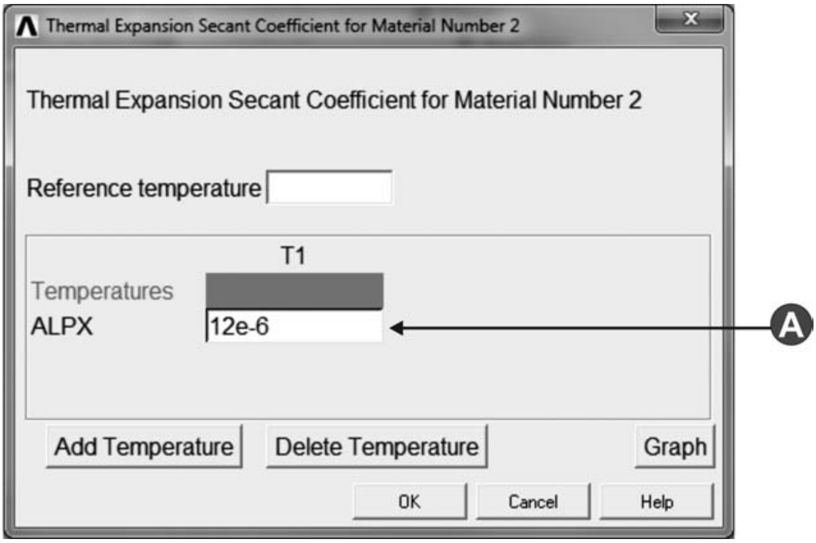
Click on Structural > Linear > Elastic > Isotropic



A type 210e9 in EX
B type 0.29 in PRXY

OK

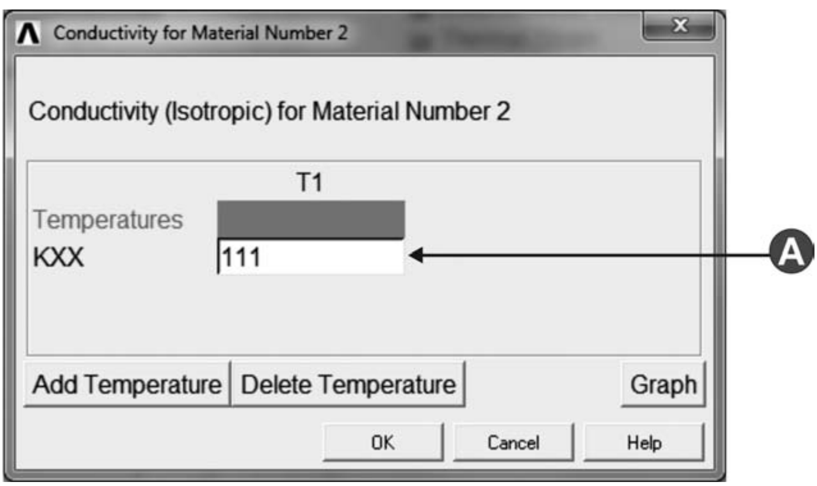
Click on Structural > Thermal Expansion > Secant Coefficient > Isotropic



A type 12e-6 in ALPX

OK

Click on Thermal > Conductivity > Isotropic



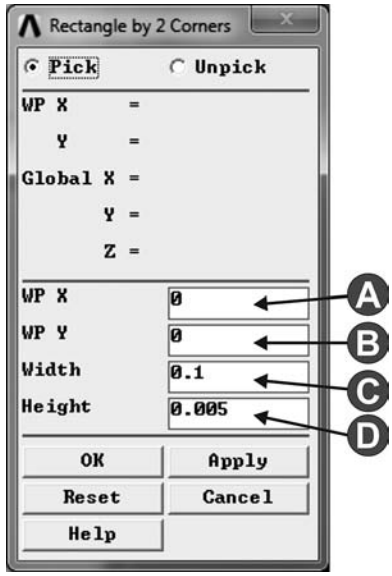
A type 111 in KXX

OK

Close the Material Models Behavior window

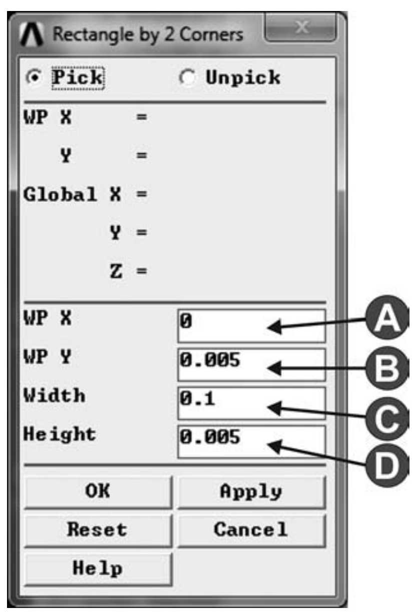
The geometry of the problem is relatively simple. Two rectangles are created, and then they are glued.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle by 2 Corners



A type 0 in WP X
B type 0 in WP Y
C type 0.1 in Width
D type 0.005 in Height

$\boxed{\textbf{Apply}}$



A type 0 in WP X
B type 0.005 in WP Y

C type 0.1 in Width

D type 0.005 in Height

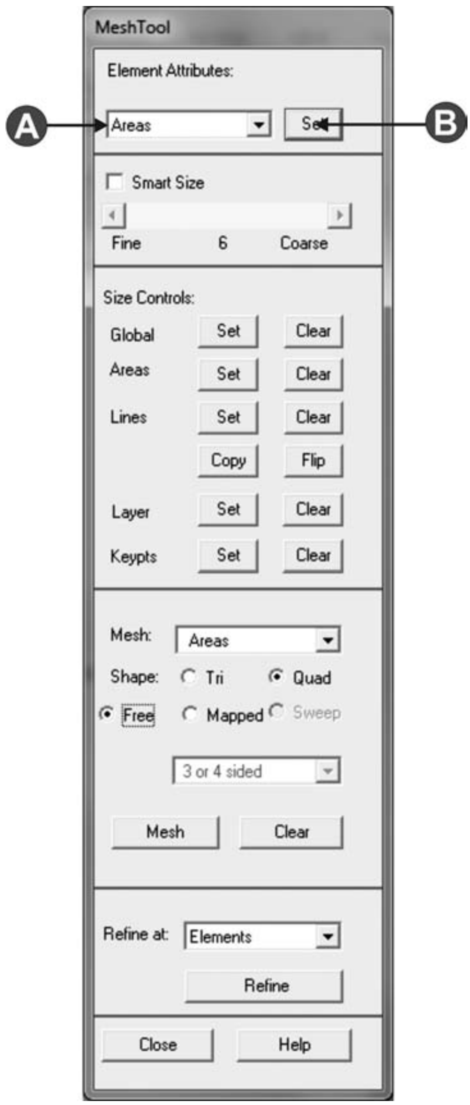
OK

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

In Glue Areas window, click on

Pick All

Main Menu > Preprocessor > Meshing > Mesh Tool

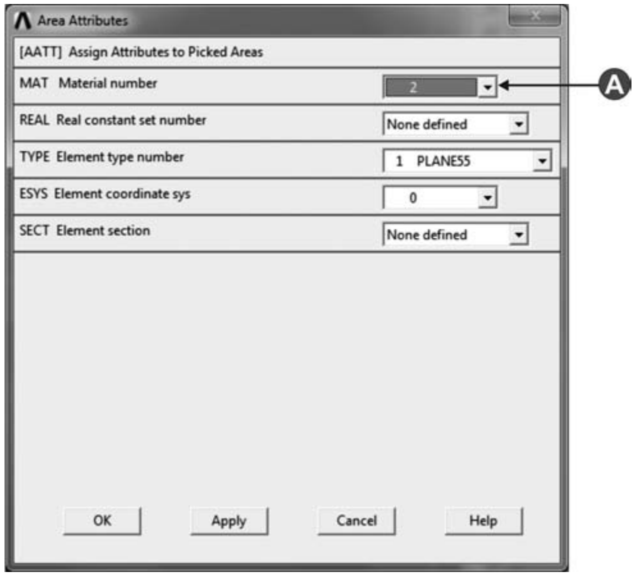


A select Areas in Element Attributes

B click on Set in Element Attributes

select the carbon steel area only, and then in Area Attributes window, click on

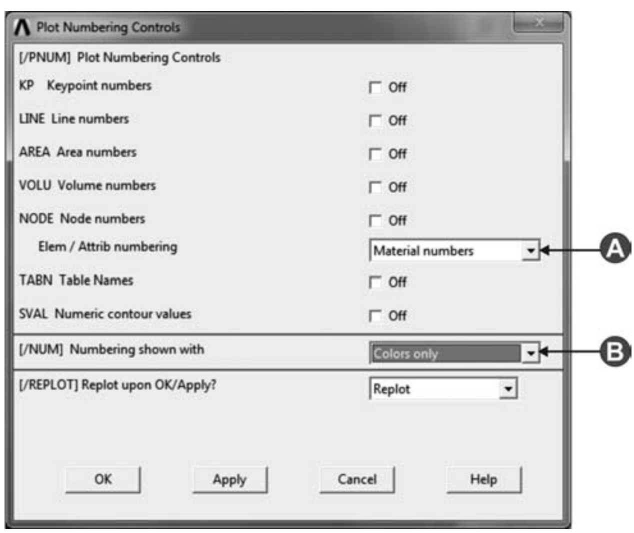
OK



A select 2 in Material number

OK

Utility Menu > PlotCtrls > Numbering …

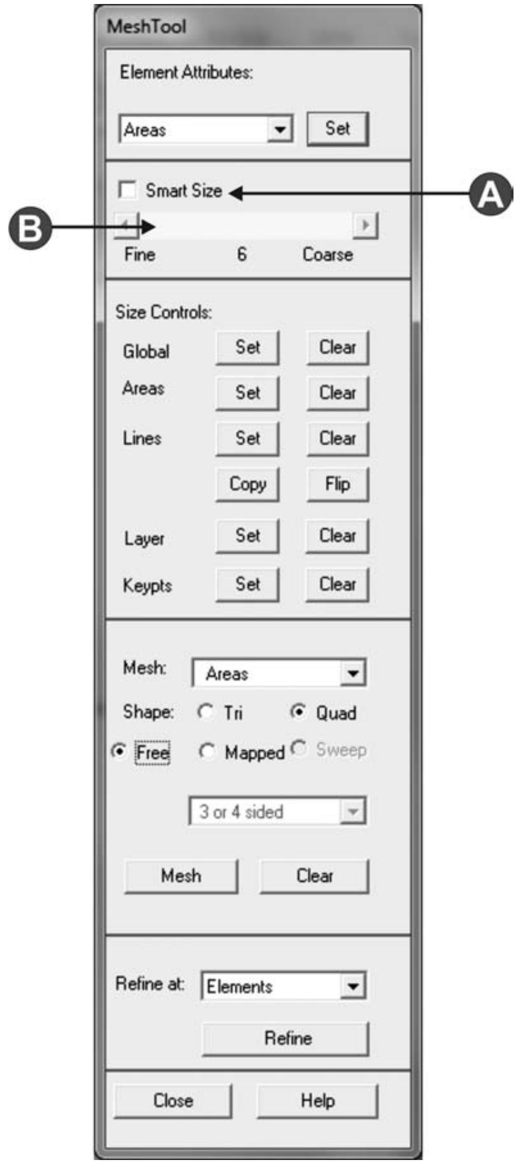


A select Material numbers in Elem/Attrib numbering

B select Colors only in Numbering shown with

OK

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Smart Size

B set the level to 1

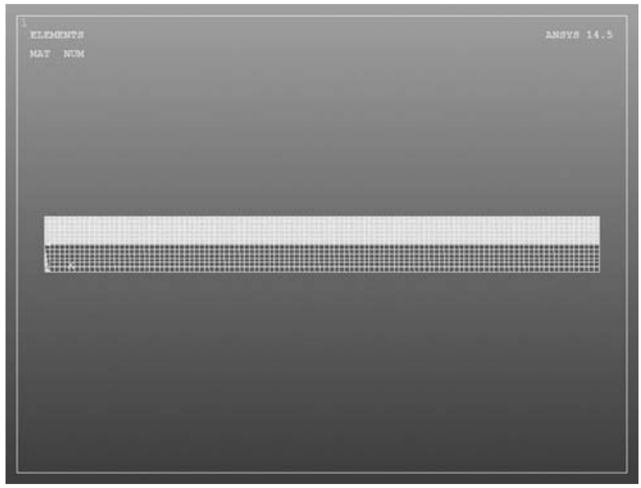
Mesh

In Mesh Area window, click on

Pick All

Close the Mesh Tool window

Close

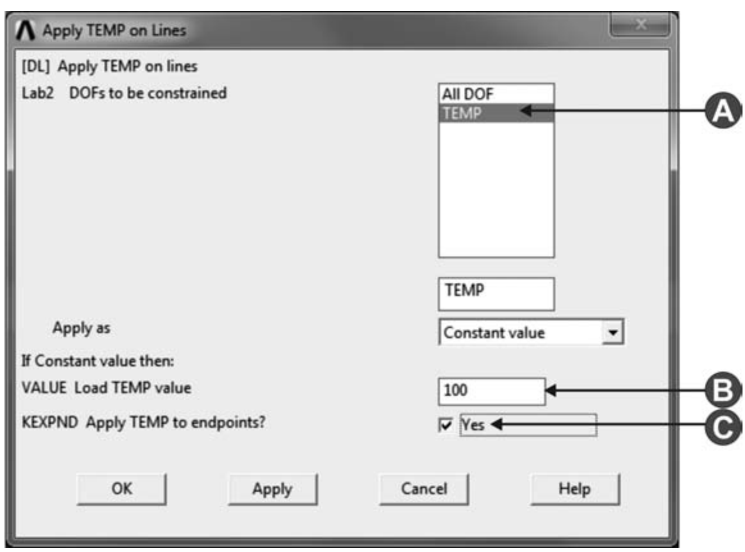


ANSYS graphics show the mesh

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature > On Lines

Click on the two left vertical lines where the temperature boundary condition is applied, and then in Apply TEMP on Lines window, click on

OK



A select TEMP

B type 100 in VALUE Load TEMP value

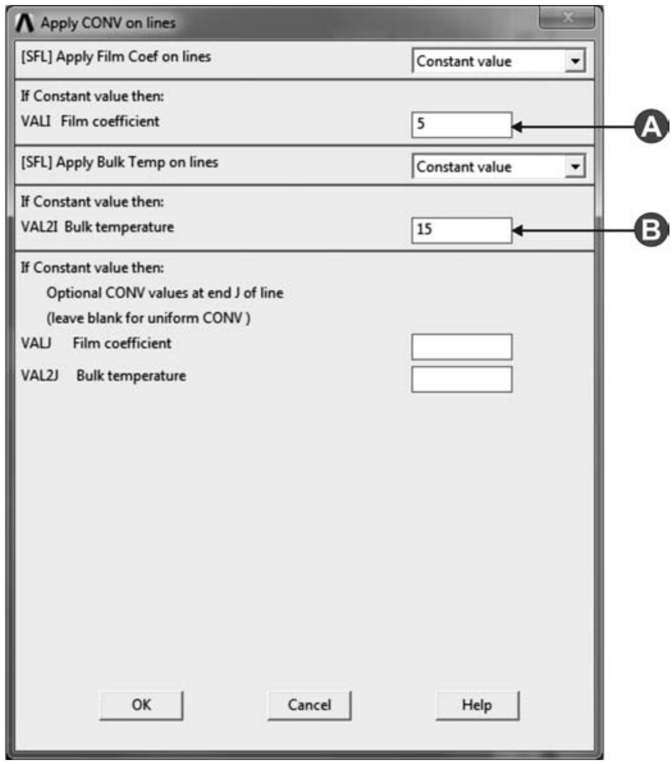
C select Yes

OK

Main Menu > Solution > Define Loads > Apply > Thermal > Convection > On Lines

Click on lines where the convection boundary condition is applied: the two external horizontal lines and the right two vertical lines. Do not click on the line between the two materials. Then in Apply CONV on lines window, click on

OK



A type 5 in Film coefficient

B type 15 in Bulk temperature

OK

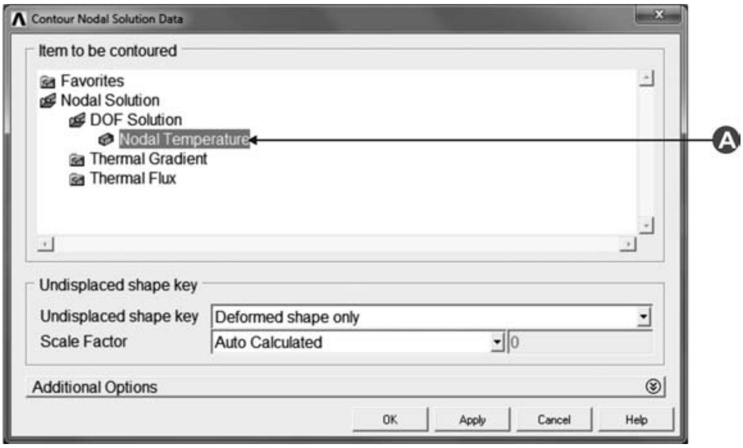
Now, the problem is ready to be solved as a heat transfer problem.

Main Menu > Solution > Solve > Current LS

Close

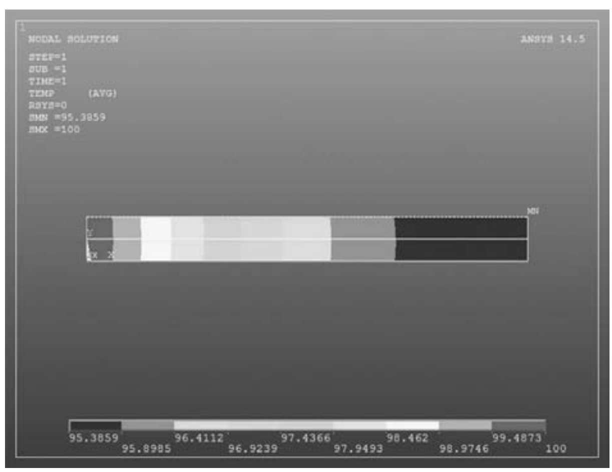
The thermal solution is now completed. The thermal elements must be replaced by structural elements with the same element type. This can be done using the Switch element type in the preprocessor. The temperature contours are plotted to ensure that the thermal solution is correct before solving it as a structural problem.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click Nodal Solution > DOF Solution > Nodal Temperature

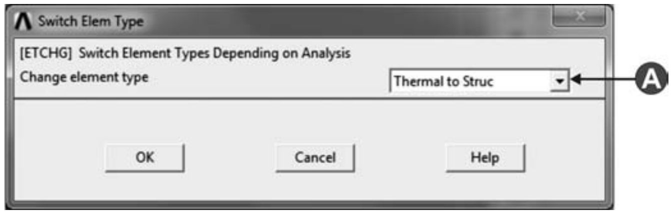
OK



ANSYS graphics show the temperature contours

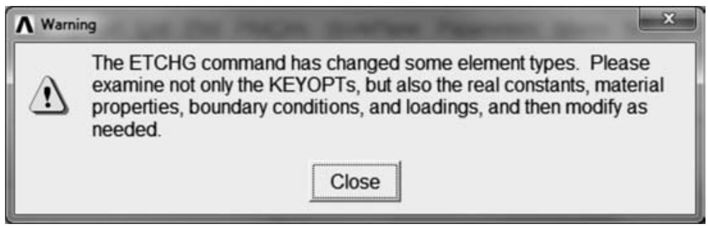
The temperature contours show the left side is maintained at 100°C, and the temperature is decreasing in the positive x-direction due to the convective cooling. In general, the temperature contours are as expected. Next, the element type is switched to structural type, and the nodal temperature solution from thermal analysis is loaded to the structural nodes.

Main Menu > Preprocessor > Element Type > Switch Elem Type



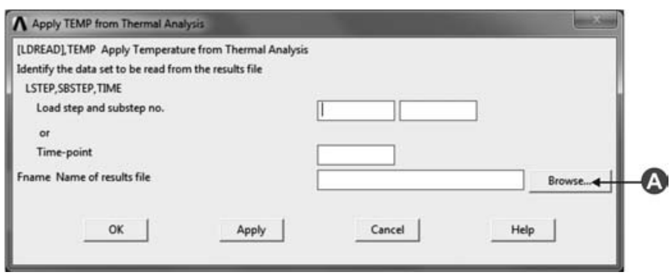
A select Thermal to Struc in Change element type

OK



Close

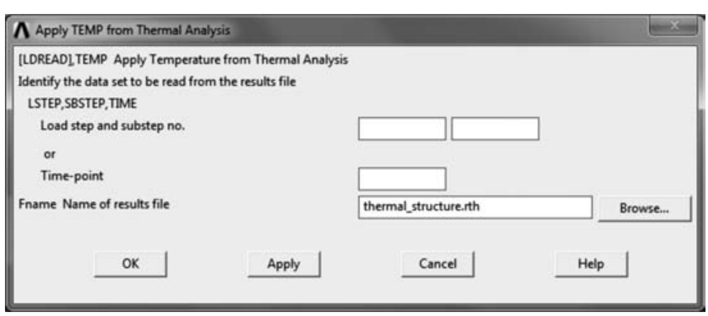
Main Menu > Solution > Define Loads > Apply > Structure > Temperature > from Thermal Analysis



A click on Browse …

Select from the thermal results file that has an extension of rth, and then click on

Open



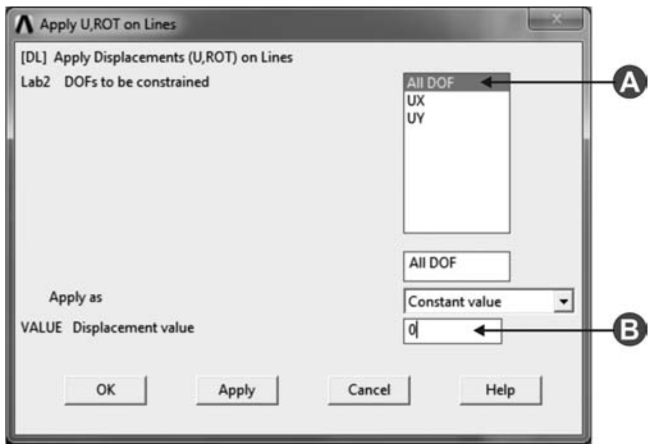
OK

The structural boundary condition is the zero displacement at the left side of the thermocouple.

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines

In the ANSYS graphics, click on the two vertical left lines where zero displacement is applied. Then, in Apply U,ROT on Lines window, click on

OK



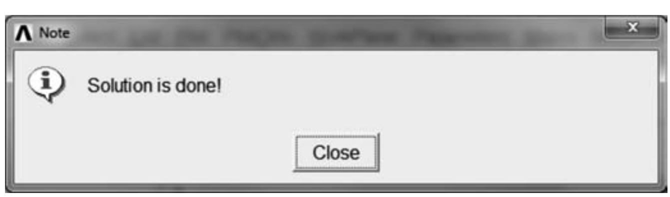
A select All DOF

B type 0 in VALUE Displacement value

OK

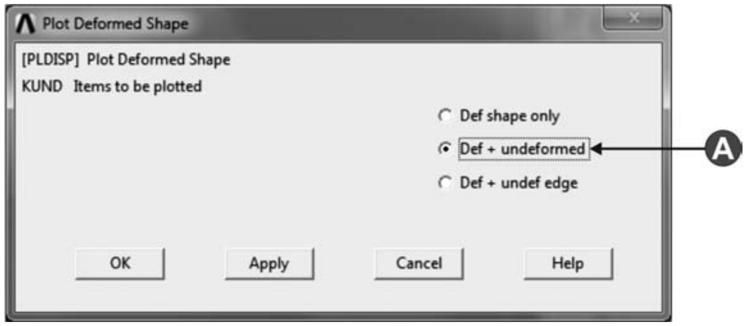
Main Menu > Solution > Solve > Current LS

OK



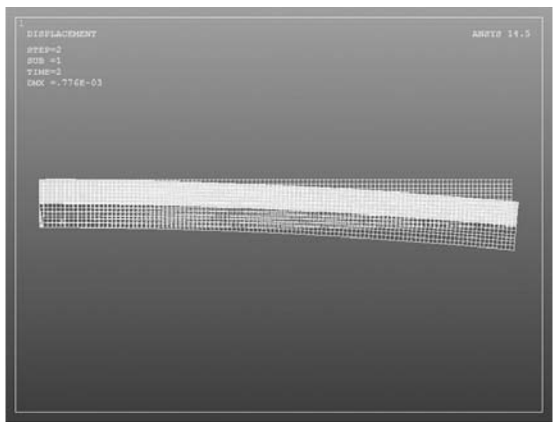
Close

Main Menu > General Postproc > Plot Results > Deformed Shape



A select Def + undeformed

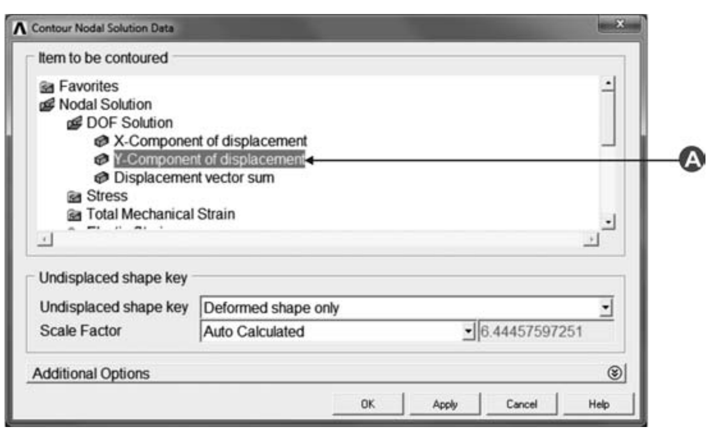
OK



ANSYS graphics show the thermocouple before and after deformation

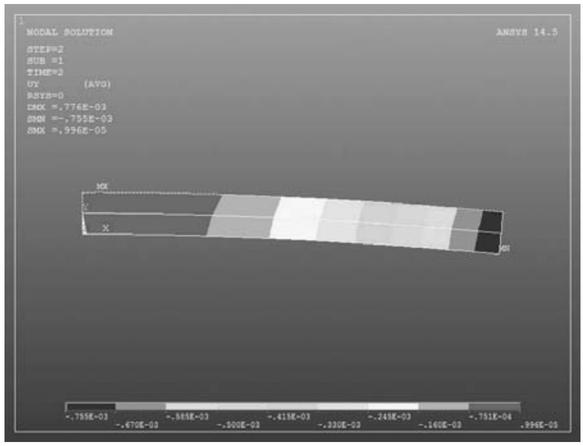
The plot shows downward deflection of the thermocouple because the thermal expansion of the aluminum is higher than that of the carbon steel.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > DOF Solution > Y-Component of displacement

$\boxed{\textbf{OK}}$



ANSYS graphics show the displacement in the y-direction

The ANSYS results indicate that the maximum displacement in the y-direction is equal to 0.755×10^{-3} m.

7.3 Chips cooling in a channel using ANSYS

Study the thermal characteristics of two electronic chips mounted on a channel's bottom wall using ANSYS. The configuration is shown in Figure 7.5. The working fluid is air, and thermophysical properties of the chips are listed in Table 7.2. The inlet velocity is 0.01 m/s, while the exit condition is a reference zero pressure. The inlet temperature is 25°C, and 5 W are generated in each chip. Consider the problem as a steady fluid flow and heat transfer. Determine the following:

1. Maximum velocity in the x-direction, and temperature in the domain
2. Satisfaction in mass and energy balances
3. Heat transfer coefficient distribution at the surface of the first chip

Electronic manufacturers are continuously providing the market with high-performance devices, but with high heat dissipation. Heat highly affects the performance and durability of electronic devices more than any other factor, and operating electronic devices at a temperature higher

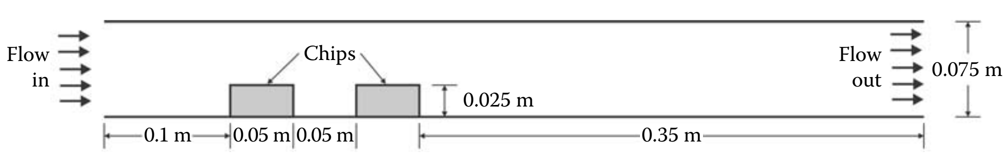


FIGURE 7.5 Channel with two chips.

Table 7.2 Thermophysical properties of the air and electronic chips

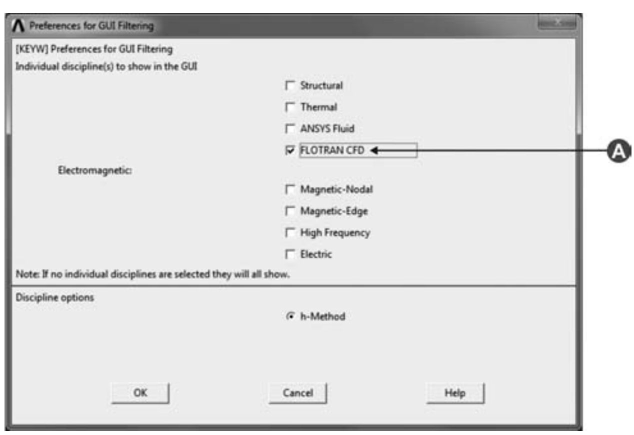
Property	Air	Chips
Density (kg/m^3)	1.25	2500
Conductivity (W/m · °C)	0.025	0.25
Specific heat (J/kg · °C)	1006.2	850
Viscosity (N · s/m^2)	18.5×10^{-6}	—

than its recommended operating value will significantly affect their reliability and functionality. Thermal analysis of a channel containing multiple heated blocks subjected to forced convection flow is extensively addressed in the literature because it simulates integrated circuit chips placed on a horizontal board. In this analysis, researchers are focusing on the temperature distribution within the chips, the maximum temperature of the chips, and distribution of the local Nusselt number along the chips' surface. Temperature distribution within the chips is typically used to predict the reliability of some of its components, such as the solder joints, and to establish a guide for safe operating conditions. The Nusselt number is used to estimate the heat flow out of the chips and to determine the required cooling load. In addition, researchers are interested in the friction factors on the surface of the chips, and the pressure drop in the channel, which are used to measure the required fan power.

Double click on the Mechanical APDL Product Launcher icon

This problem is considered as multiphysics and physics are not coupled because the fluid flow and heat transfer are solved separately. ANSYS will solve the fluid flow first, and then it uses the velocity components to solve for the temperature field. ANSYS Flotran CFD can solve fluid dynamics problems, as well as the thermal problems. The selected FLOTRAN 141 element has temperature as a degree of freedom.

Main Menu > Preferences

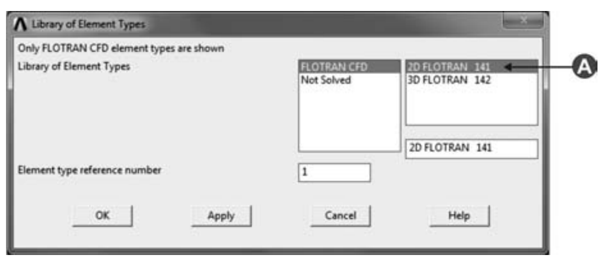


A select FLOTRAN CFD

OK

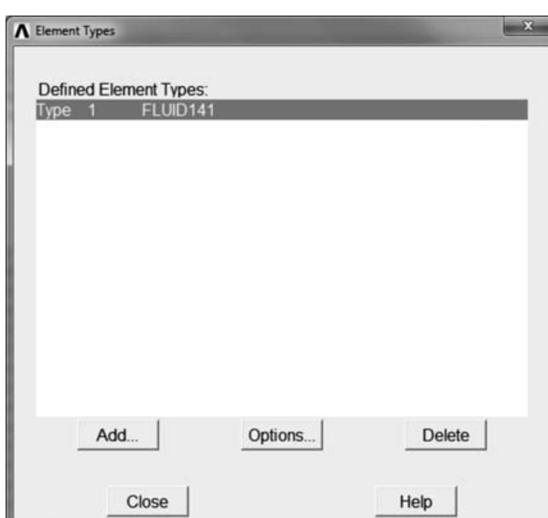
Main Menu > Preprocessor > Element Types > Add/Edit/Delete

Add...



A select 2D FLOTRAN 141

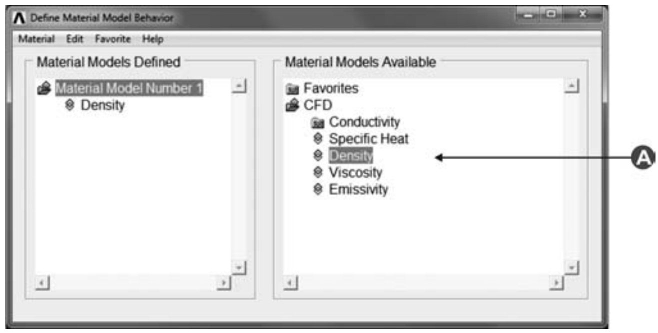
OK



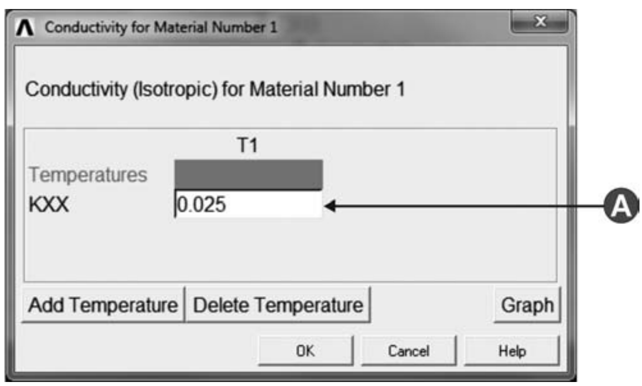
Close

There will be different sets of properties for air and chips. The material number 1 in the Material Model Behavior is for the fluid only, and material numbers 2 to 10 are for the solids.

Main Menu > Preprocessor > Material Props > Material Models



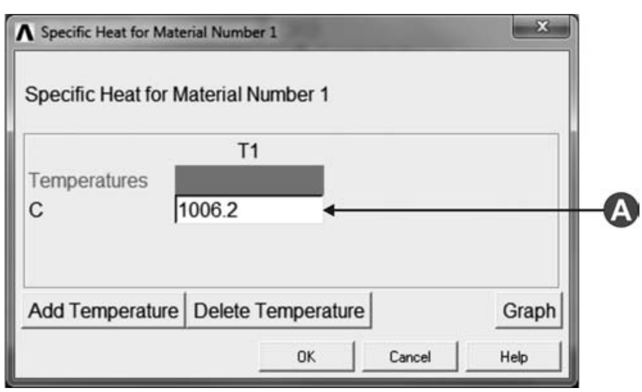
A click on CFD > Conductivity > Density



A type 0.025 in KXX

OK

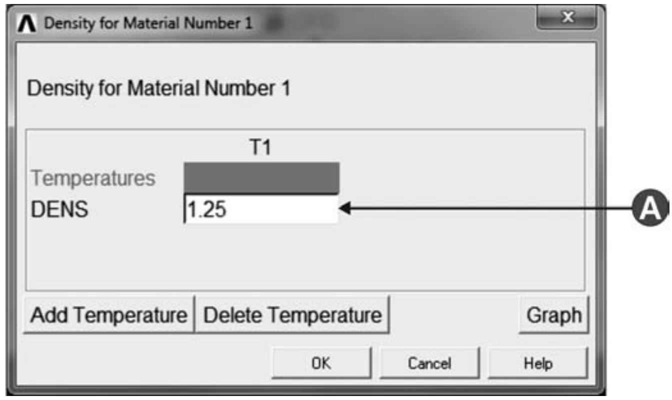
Click on CFD > Specific Heat



A type 1006.2 in C

OK

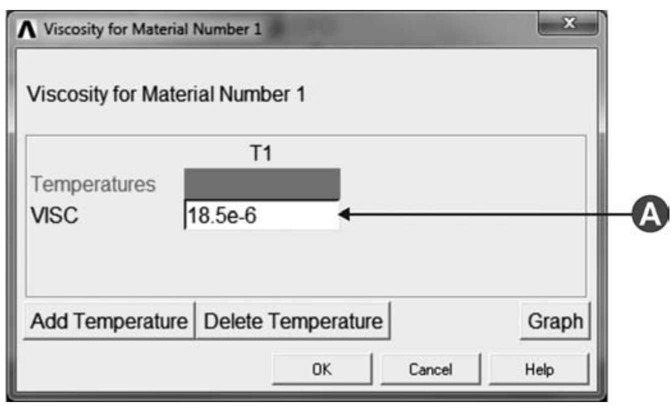
Click on CFD > Density



A type 1.25 in DENS

OK

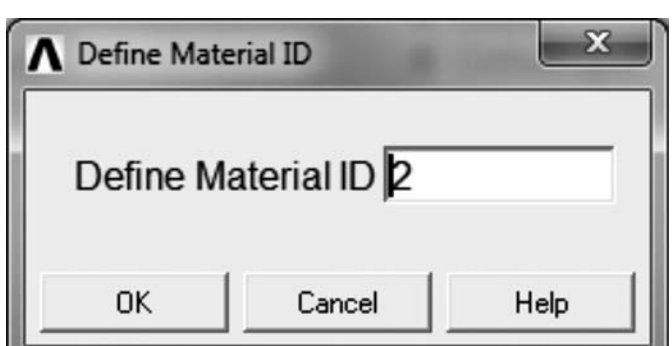
Click on CFD > Viscosity



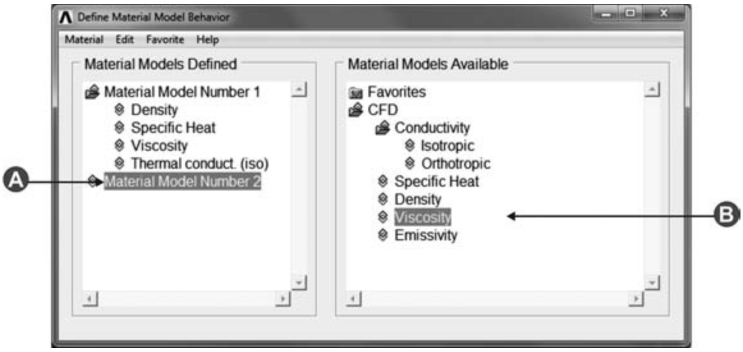
A type 18.5e-6 in VISC

OK

In the Define Material Model Behavior menu: Material > New Model

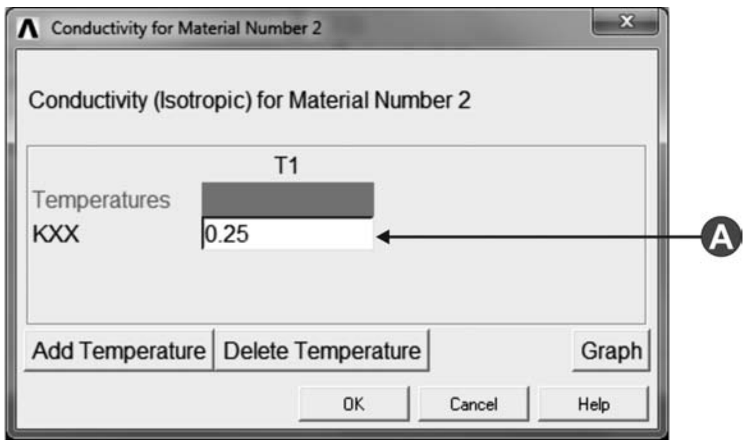


OK



A select Material Model Number 2

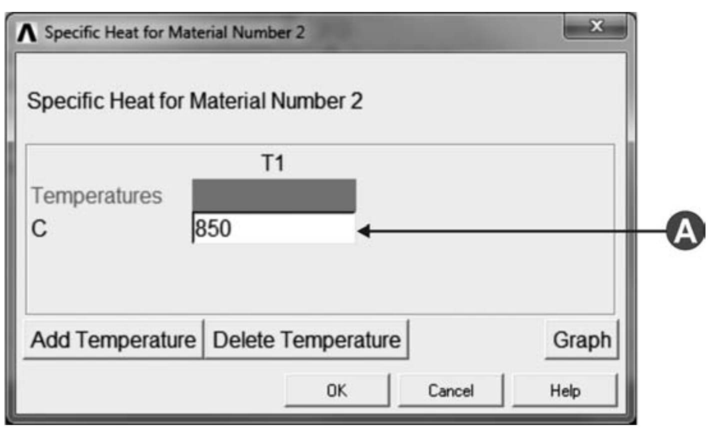
B click on CFD > Conductivity > Viscosity



A type 0.25 in KXX

OK

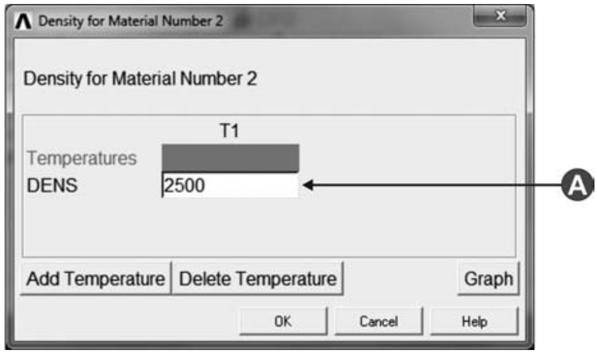
Click on CFD > Specific Heat



A type 850 in C

OK

Click on CFD > Density



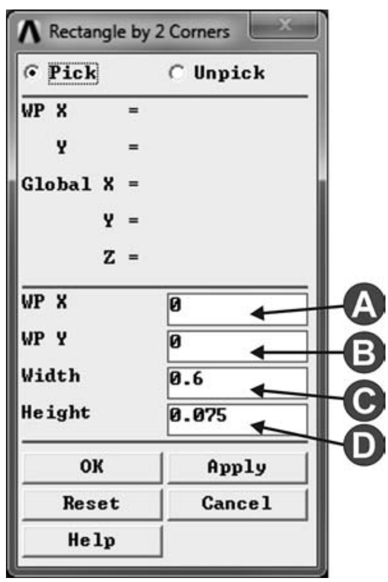
A type 2500 in DENS

OK

Close the Define Material Model Behavior window

The geometry is created by first creating a large rectangular area for fluid flow. Then, the two chips are created. Overlap is utilized to insert the chips into the fluid rectangular area.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle by 2 Corners



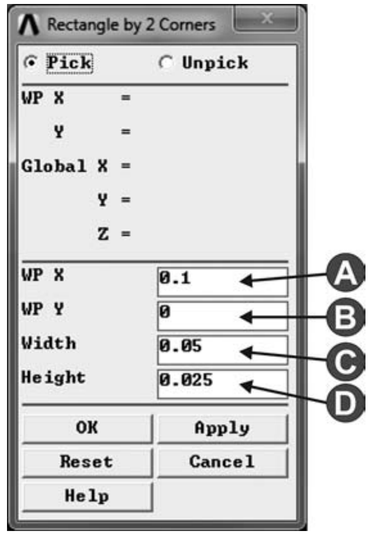
A type 0 in WP X

B type 0 in WP Y

C type 0.6 in Width

D type 0.075 in Height

Apply



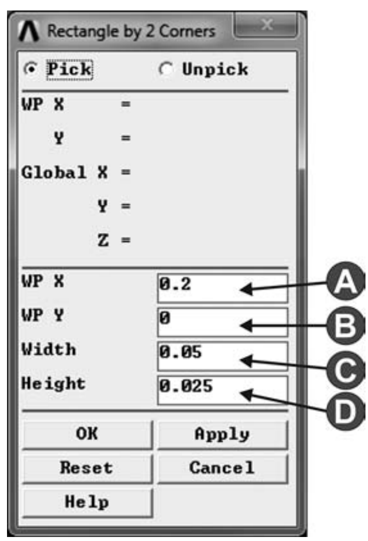
A type 0.1 in WP X

B type 0 in WP Y

C type 0.05 in Width

D type 0.025 in Height

Apply



A type 0.2 in WP X

B type 0 in WP Y

C type 0.05 in Width

D type 0.025 in Height

OK

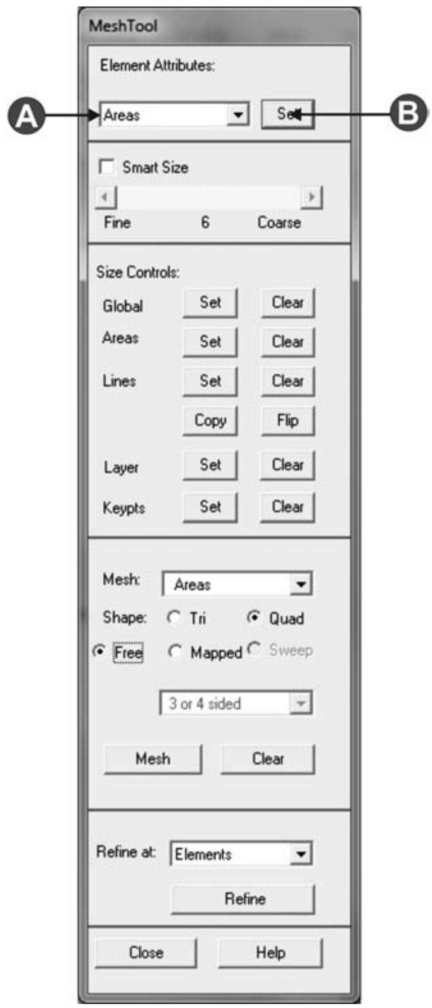
Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

In Overlap Areas window, click on

Pick All

The following step is for changing the material properties of the chips from number 1 to 2. By selecting number 2, the properties of number 2 in the material model are assigned to the chips. Air by default has the properties of number 1 in the material model.

Main Menu > Preprocessor > Meshing > Mesh Tool

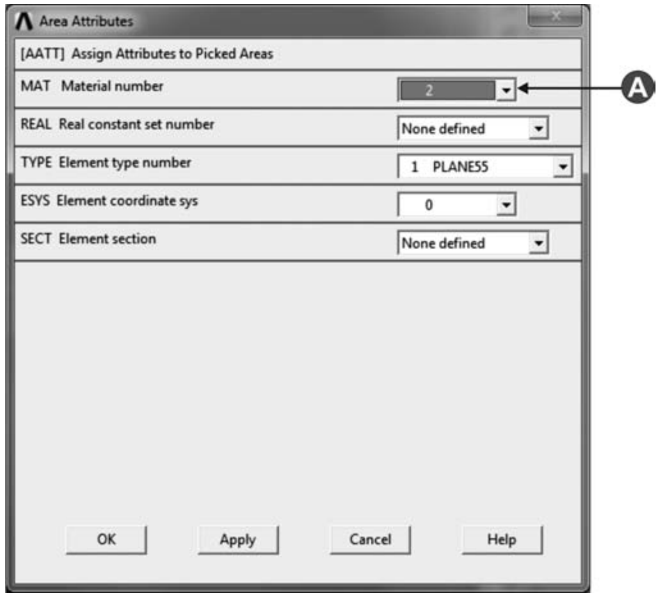


A select Areas

B click on Set

Select both chips, and then in Area Attributes window, click on

OK

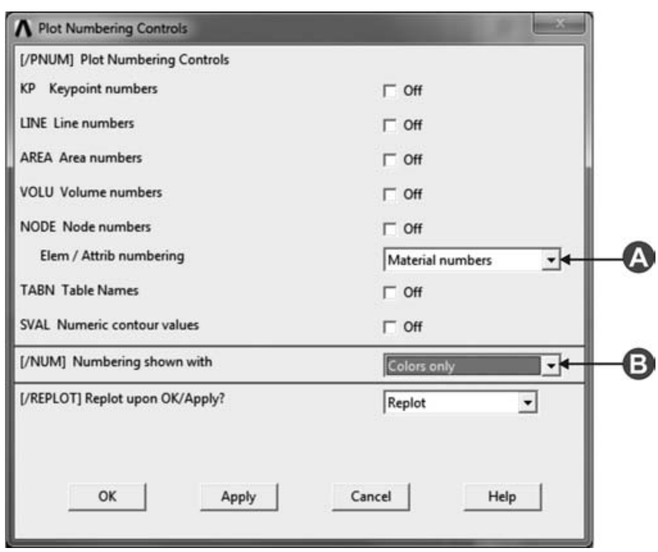


A select 2 in Material number

OK

To ensure that the properties of air and chips are assigned correctly, the air and chips are colored according to their material number in the material model. This step has no effect on the solution.

Utility Menu > PlotCtrls > Numbering …



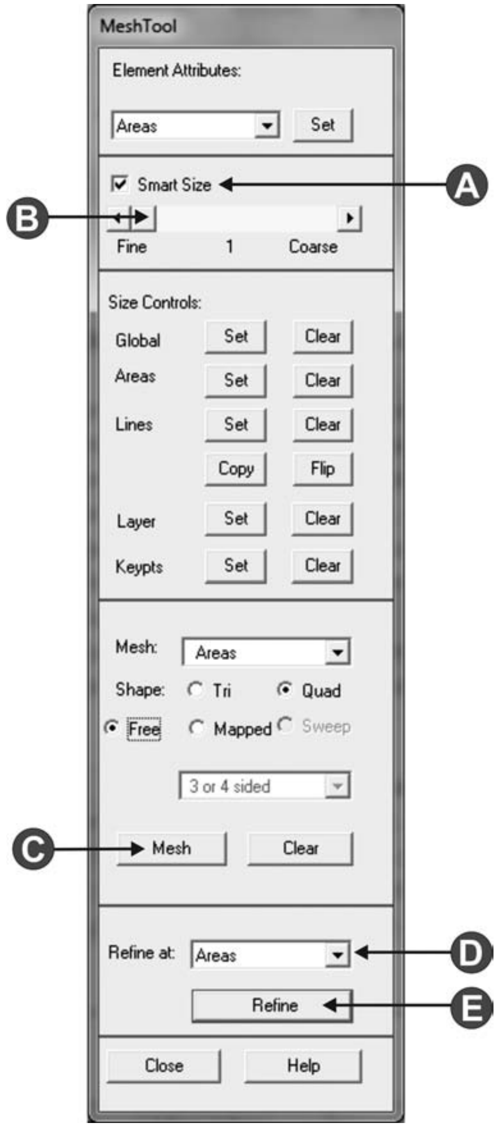
A select Material numbers in Elem/Attrib numbering

B select Colors only in Numbering shown with

OK

The smart mesh number 1 has insufficient mesh density to have accurate results for fluid dynamics problems. The number of elements in the domain is increased by using the area size control in the Mesh Tool.

Main Menu > Preprocessor > Meshing > Mesh Tool



A select Smart Size

B set the level to 1

C Mesh

In Mesh Area window, click on

Pick All

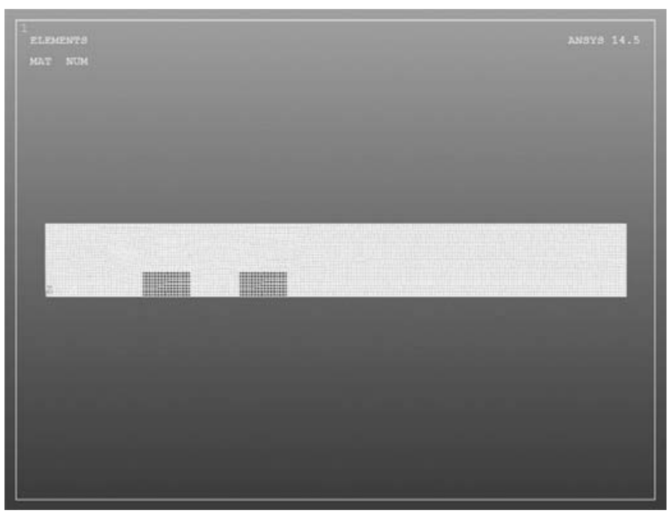
D select Areas in Refine at

E click on Refine

In Refine Mesh at Area window, click on

Pick All

OK



ANSYS graphics show the mesh

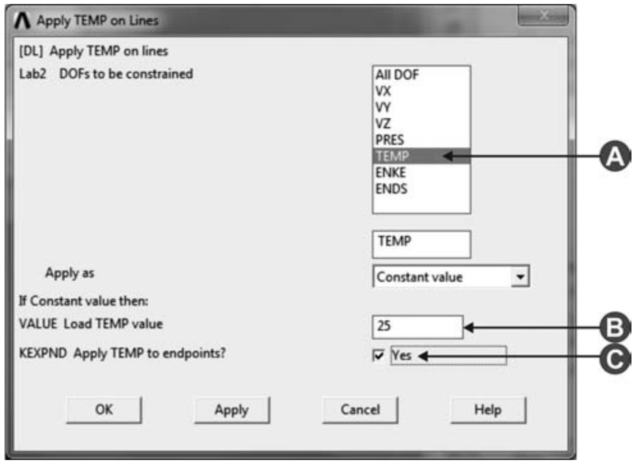
Using refine at elements instead of refine at areas will generate a similar mesh. Notice that the ANSYS properties of the material model behavior will be used for air and chips.

Close the Mesh Tool Window

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature> On Lines

Click at the inlet line of the channel to specify the inlet temperature, and in Apply TEMP on Lines window, click on

OK



A select TEMP

B type 25 in VALUE Load TEMP value

C select Yes

OK

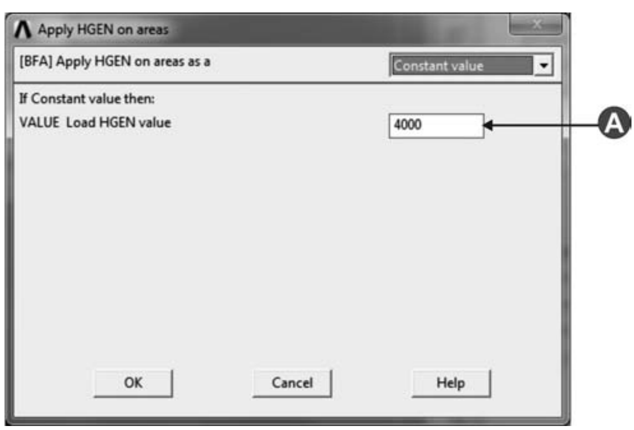
The heat generation must be per unit volume. The applied heat generation is divided by the area of the chip because the problem is two dimensional. The chip's volumetric heat generation is calculated as follows:

$$Q = \frac{5}{0.05 \times 0.025} = 4000 \text{ W/m}^2$$

Main Menu > Solution > Define Loads > Apply > Thermal > Heat Generat > On Areas

Click on both chips. Then, in Apply HGEN on ARs window, click on

OK



A type 4000 in Load HGEN value

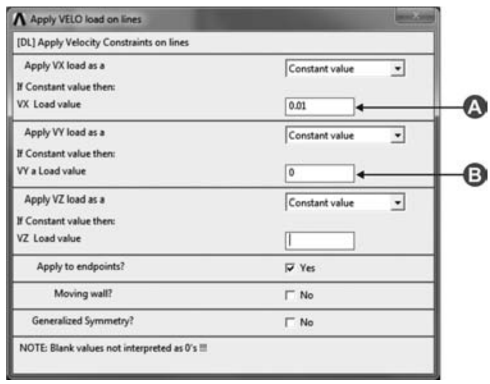
OK

The top and bottom boundaries are wall and insulated. Zero x- and y-velocities should be imposed. There is no need to impose a velocity boundary at the bottom surface of the chips. By default, any unassigned thermal boundary conditions will be considered as insulated. Do not impose any boundary conditions on the surface of the chips that are exposed to the flow.

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click on the inlet of the channel. Then in Apply VELO load on lines window, click on

OK



A type 0.01 in VX Load value

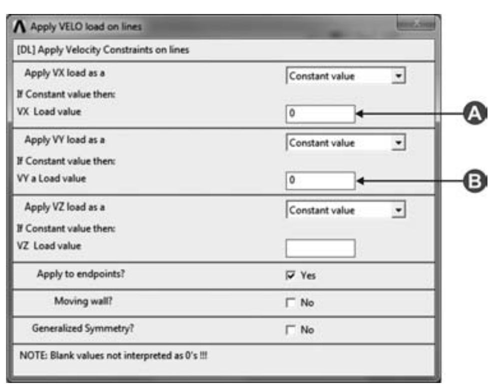
B type 0 in VY a Load value

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Velocity > On Lines

Click on the top and bottom walls. Then, in Apply VELO load on lines window, click on

OK



A type 0 in VX Load value

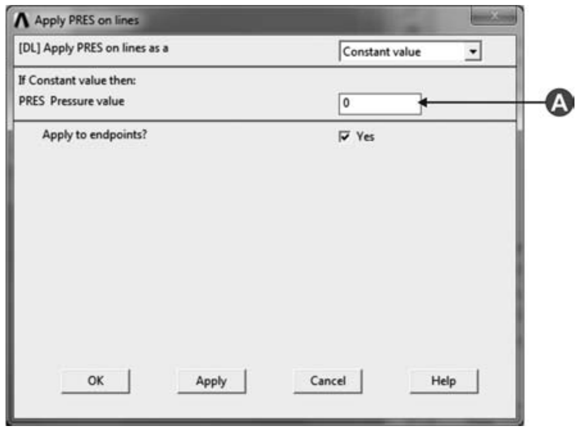
B type 0 in VY a Load value

OK

Main Menu > Solution > Define Loads > Apply > Fluid/CFD > Pressure DOF > On Lines

Click on channel's exit. Then, in Apply PRES on lines window, click on

OK

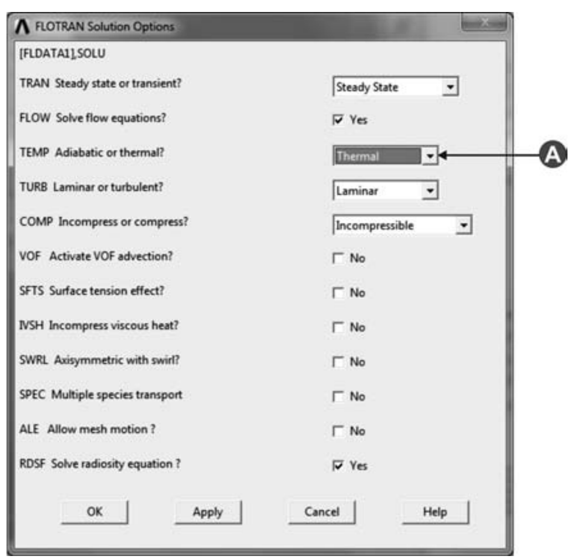


A type 0 in PRES Pressure value

OK

The fluid flow and heat transfer are solved at the same time. In solution options, the problem will be solved as steady state, and the system is thermal. Flow can be changed from laminar to turbulent and from incompressible to compressible in this window.

Main Menu > Solution > FLOTRAN Set Up > Solution Options

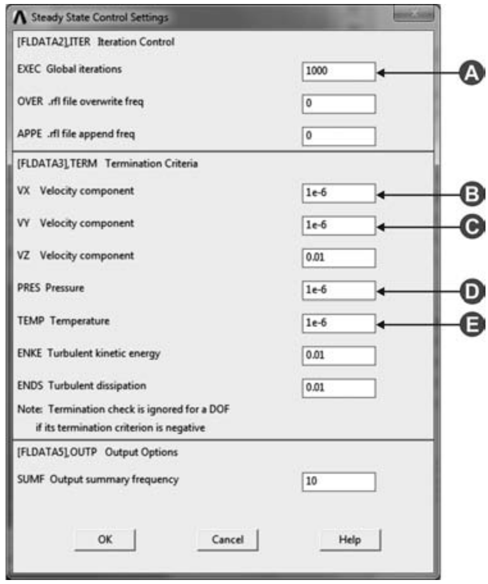


A change to thermal in TEMP Adiabatic or thermal?

OK

In execution control, it is required to specify the number of iterations for fluid flow. Then 1000 iterations will be sufficient to reach the convergence criteria for all field variables, which is 1×10^{-6} (1e-6).

Main Menu > Solution > FLOTRAN Set Up > Execution Ctrl



A type 1000 in EXEC Global iterations

B change the termination criteria for VX Velocity component to 1e-6

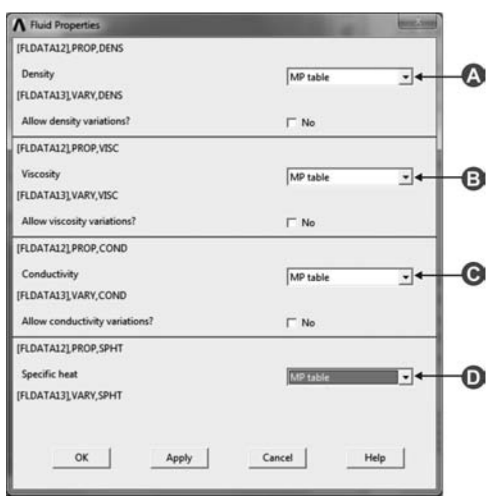
C change the termination criteria for VY Velocity component to 1e-6

D change the termination criteria for Pressure to 1e-6

E change the termination criteria for Temperature to 1e-6

OK

Main Menu > Solution > FLOTRAN Set Up > Fluid Properties



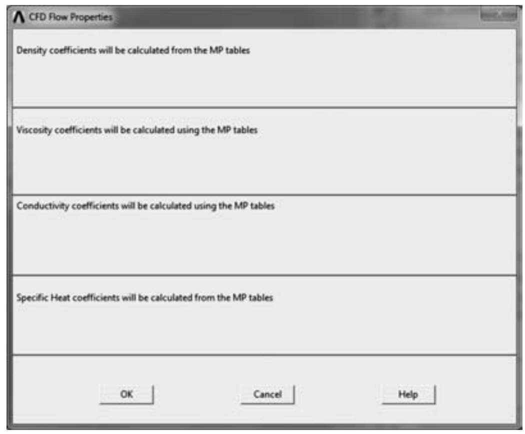
A select MP table in Density

B select MP table in Viscosity

C select MP table in Conductivity

D select MP table in Specific heat

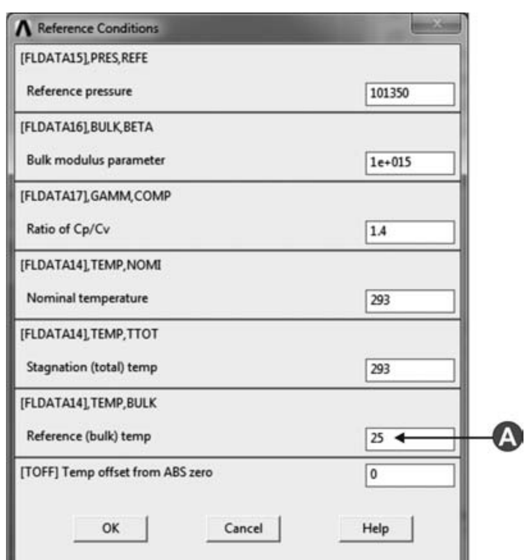
OK



OK

For the heat transfer coefficient calculation, the flow bulk temperature is required. In the reference condition, the bulk temperature is specified, which is equal to the flow inlet temperature.

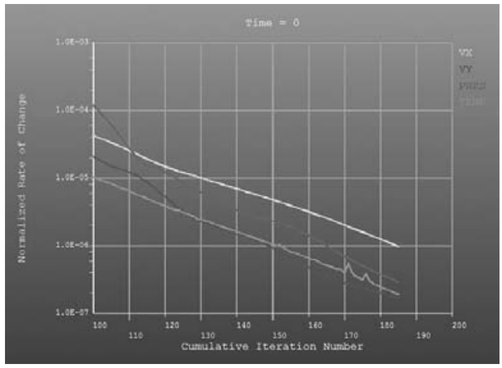
Main Menu > Solution > FLOTRAN Set Up > Flow Environment > Ref Conditions



A type 25 in Reference (bulk) temp

OK

Main Menu > Solution > FLOTRAN Set Up > Run FLOTRAN



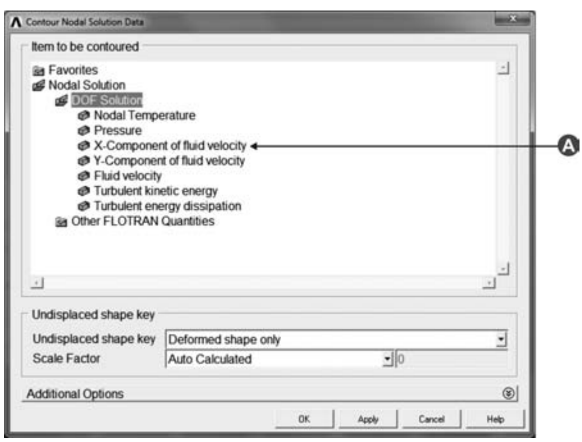
ANSYS graphics show a normalized rate of change for field variables

Close

ANSYS graphics show the normalized rate of change of the field variables. Note the solution is fully converged. All field variables reach 1×10^{-6} at the iteration number 185, which is less than the specified 1000 iterations. In read results, the last set must be read to have the results from the last iteration.

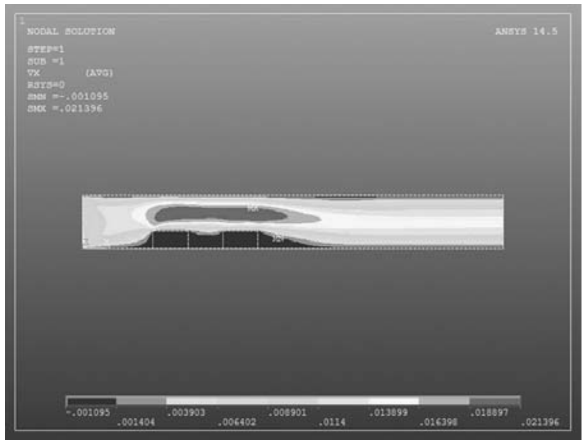
Main Menu > General Postproc > Read Results > Last Set

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > DOF Solution > X-Component of fluid velocity

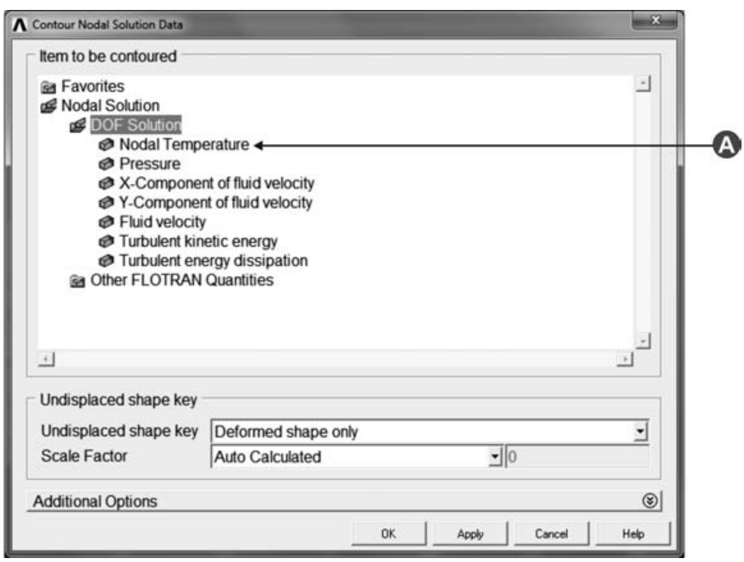
OK



ANSYS graphics show the x-component velocity contours

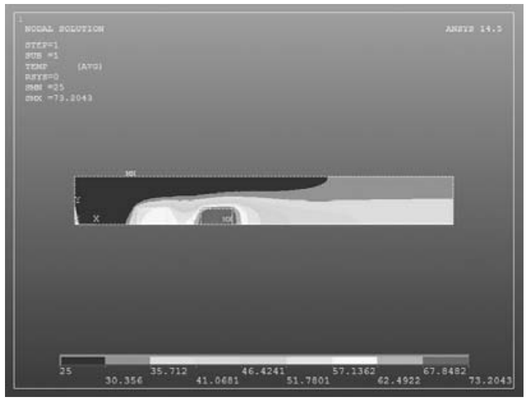
For the obtained results, the maximum and average velocity in the x-direction occurred above the chips, and the maximum velocity is 0.021396 m/s. This is expected because this region has the smallest cross-sectional area that forces the flow to accelerate. At the walls, the velocity is 0. Notice that the velocity has a parabolic profile at the exit of the channel.

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solution



A click on Nodal Solution > DOF Solution > Nodal Temperature

OK

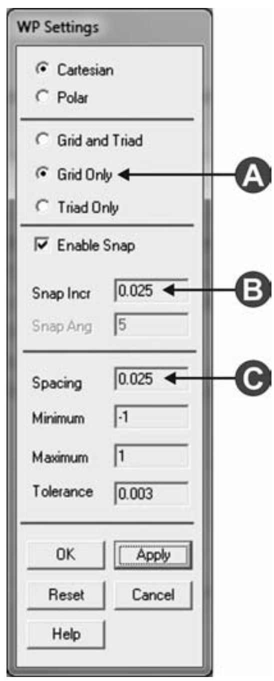


ANSYS graphics show the temperature contours

As shown in the temperature contours, the maximum temperature is 73.2043°C, and this occurred at the second chip. To ensure that mass and energy balances are satisfied, the average velocity in the x-direction and temperature at the exit are determined using the path operation in the postprocessor. The path at the exit is created by specifying two points.

ANSYS Utility Menu > WorkPlane > Display Working Plane

Utility Menu > WorkPlane > WP Setting



A select Grid Only

B type 0.025 in Snap Incr

C type 0.025 in Spacing

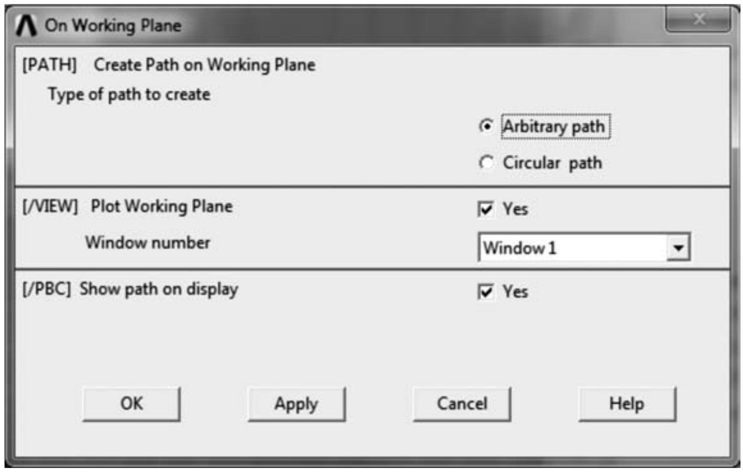
OK

Utility Menu > Plot > Areas

Utility Menu > PlotCtrls > Pan-Zoom-Rotate …

Click on zoom in and out until the ANSYS graphics show all grids.

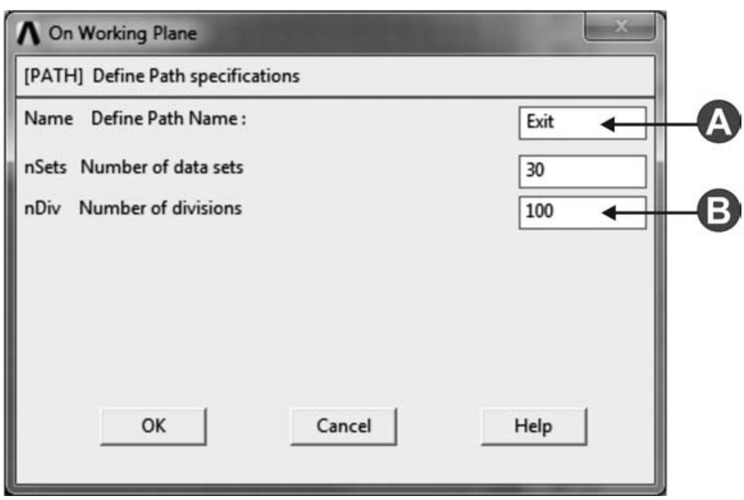
Main Menu > General Postproc > Path Operations > Define Path > On Working Plane



OK

Click on the ANSYS Graphics window at the exit's top and bottom corners of the channel, and then in On Working Plane window, click on

OK

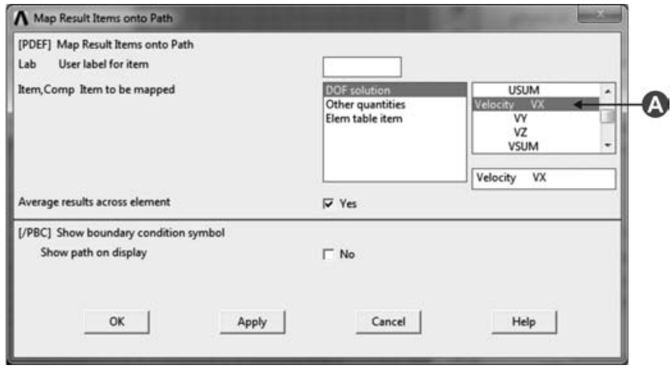


A type Exit in Define Path Name; the name of the path is optional

B type 100 in Number of divisions

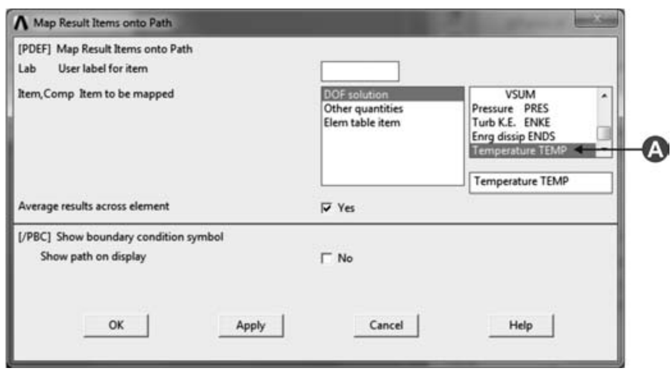
OK

Main Menu > General Postproc > Path Operations > Map onto Path



A select Velocity VX

Apply

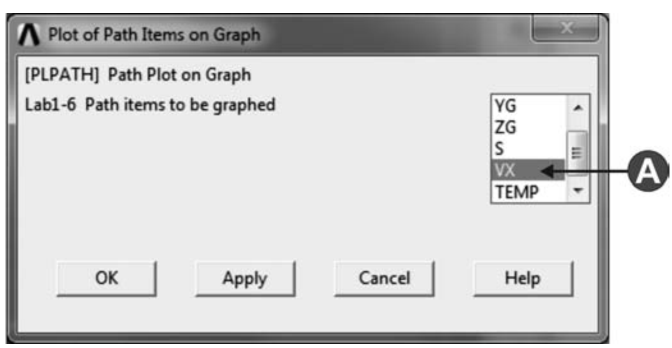


A select Temperature TEMP

OK

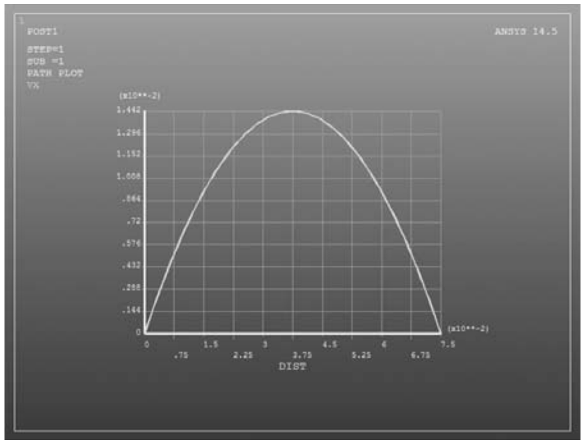
ANSYS Utility Menu > WorkPlane > Display Working Plane

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph



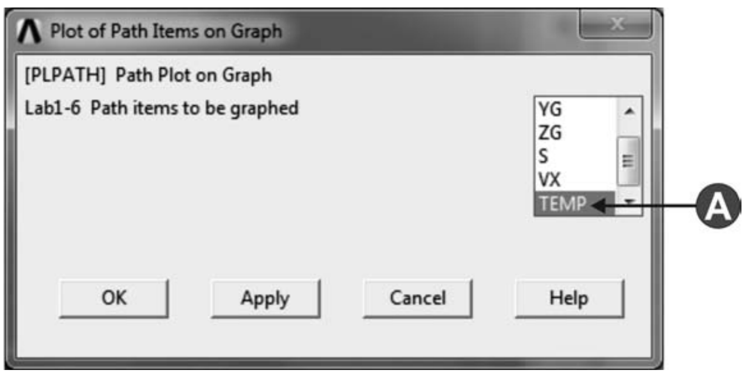
A select VX

OK



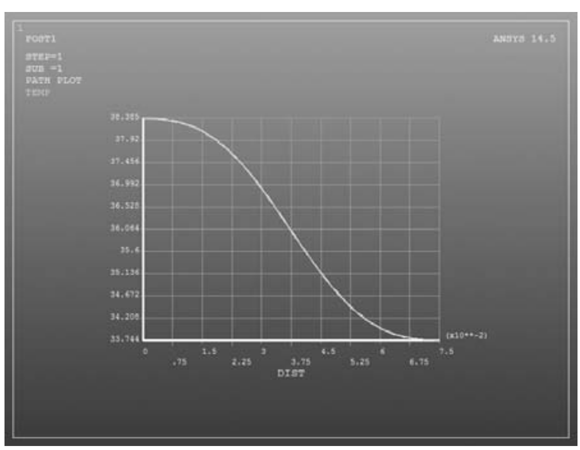
ANSYS graphics show the velocity profile at the exit of the channel

Main Menu > General Postproc > Path Operation > Plot Path Items > On Graph



A select TEMP

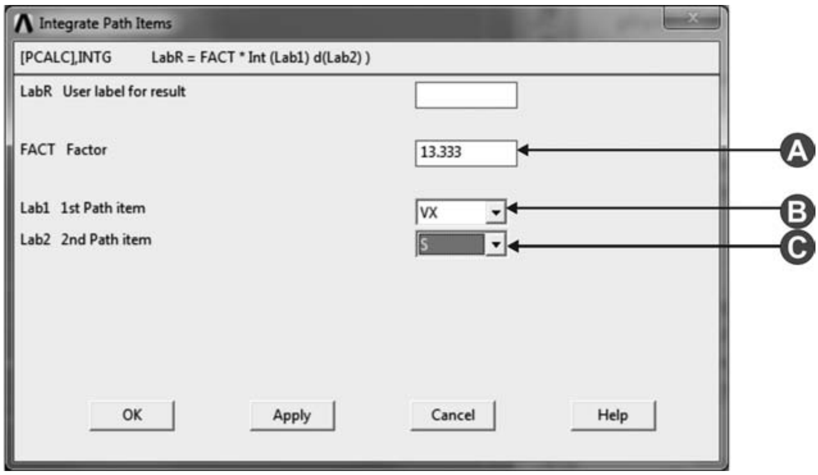
OK



ANSYS graphics show the temperature profile at the exit of the channel

The average velocity and temperature at the exit are determined using the integration in the path operation. The value of the integration must be divided by the path length to get the average value of the variable. The number 13.333 in the Factor is the inverse of the path length. Selecting S in the Lab2 means that the integration is performed along the path.

Main Menu > General Postproc > Path Operations > Integrate

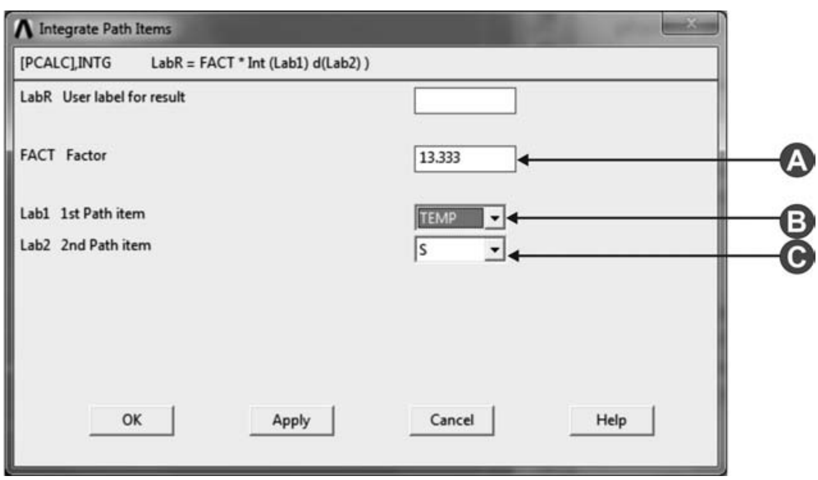


A type 13.333 in Factor

B select VX in 1st Path item

C select S in 2nd Path item

Apply



A type 13.333 in Factor

B select TEMP in 1st Path item

C select S in 2nd Path item

OK

The ANSYS Output window shows the average velocity and temperature at the exit, which are 0.95993×10^{-2} m/s and 36.052°C, respectively. The mass flow rate at the exit and inlet should be the same as follows:

$$\dot{m}_i = \dot{m}_e$$

or

$$(\rho\ V\ A)_i = (\rho\ V\ A)_e$$

Since the density and cross-sectional area of the inlet and exit are the same, then

$$V_i = V_e$$

Comparing the inlet velocity, which is 0.01 m/s, to the exit velocity, which is 0.0095993 m/s, the two velocities are close to each other with an error of 4%. The heat gained by air in the channel can be determined using the first law of thermodynamics:

$$Q = \dot{m}\ Cp\left(T_e - T_i\right)$$

or

$$Q = (\rho\ V\ A)\ Cp(T_e - T_i)$$

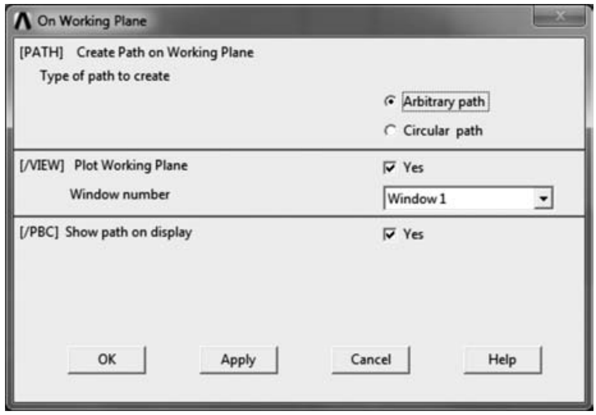
$$Q = (1.25 \times 0.01 \times 0.075) \times 1006.2 \times (36.052 - 25) = 10.425 \text{ Watts}$$

The heat gained Q should be equal to the generated heat in the two chips. Comparing the heat gain to the generated heat in the two chips, which is 10 Watts, the energy balance is satisfied with an error of 4.25%. The error found in mass and energy balances can be reduced if the computational domain is meshed with a finer mesh. The path operation is performed in the following steps to obtain the distribution of the average heat transfer coefficient around the first chip. Here, the film coefficient is selected.

Utility Menu > Plot > Areas

ANSYS Utility Menu > WorkPlane > Display Working Plane

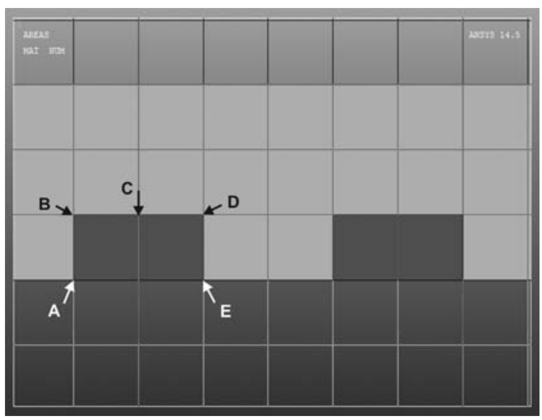
Main Menu > General Postproc > Path Operations > Define Path > On Working Plane

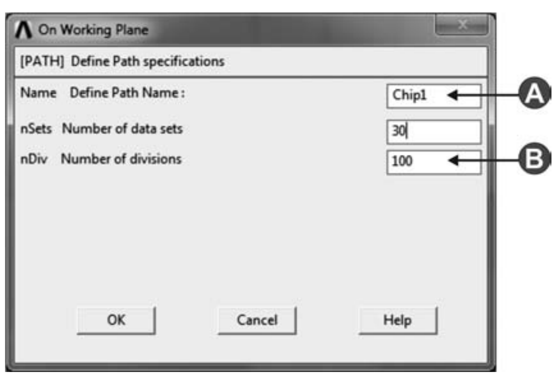


OK

Click on the ANSYS Graphics window at locations A, B, C, D, and E, as shown in the following figure, and then in On Working Plane window, click on

OK



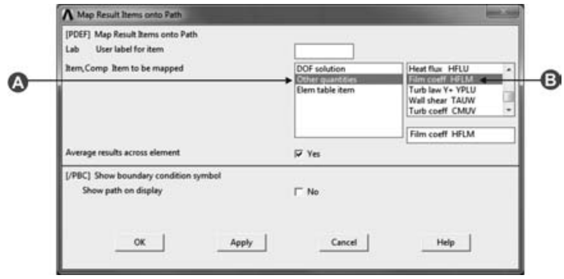


A type Chip1 in Define Path Name; the name of the path is optional

B type 100 in Number of divisions

OK

Main Menu > General Postproc > Path Operations > Map onto Path



A select Other quantities

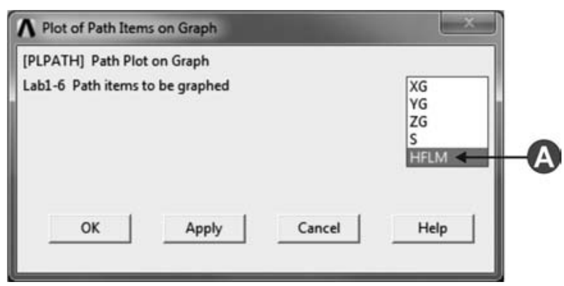
B select Film coeff HFLM

OK

Now, the stored variable, HFLM, is ready to be plotted. In the Plot path item, there are two options. The stored data can be either plotted or listed. The list results can be exported into other software such as EXCEL.

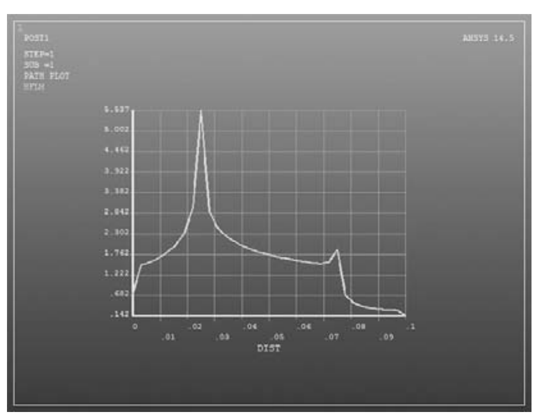
ANSYS Utility Menu > WorkPlane > Display Working Plane

Main Menu > General Postproc > Path Operations > Plot of Path Item > On Graph



A select HFLM

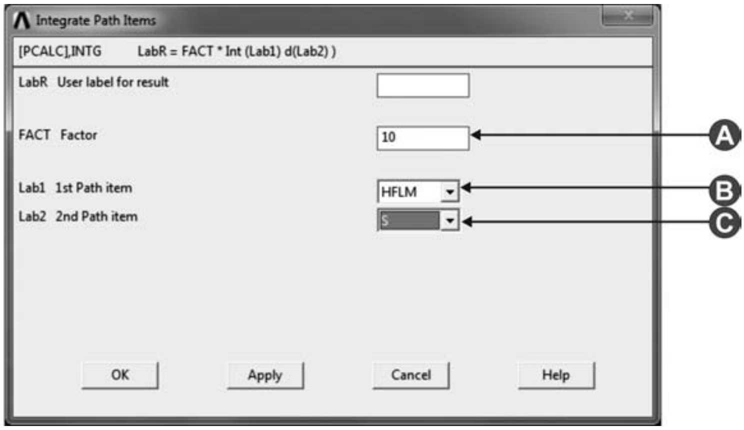
OK



ANSYS graphics show the heat transfer coefficient around the first chip

The average heat transfer coefficient at the chip surface can be determined using the integration in the path operation. The value of the integration must be divided by the path length to get the average value of the variable. Selecting S in the Lab2 means that the integration is performed along the path.

Main Menu > General Postproc > Path Operation > Integrate

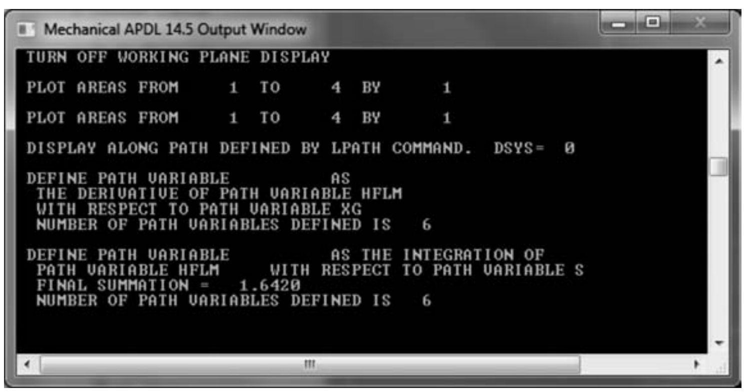


A type 10 in Factor

B select HFLM in 1st Path item

C select S in 2nd Path item

OK



The ANSYS Output window shows the value of the average heat transfer coefficient around the first chip, which is 1.642 W/m$^2\cdot$°C.

PROBLEM 7.1

The thermocouple shown in Figure 7.6 consists of copper and carbon steel plates attached to each other. The left end is fixed and maintained at 200°C, and the thermocouple is exposed to a free convection boundary condition with h = 15 W/m$^2\cdot$°C and T_∞ = 20°C. Display the y-displacement contours and determine the maximum displacement in

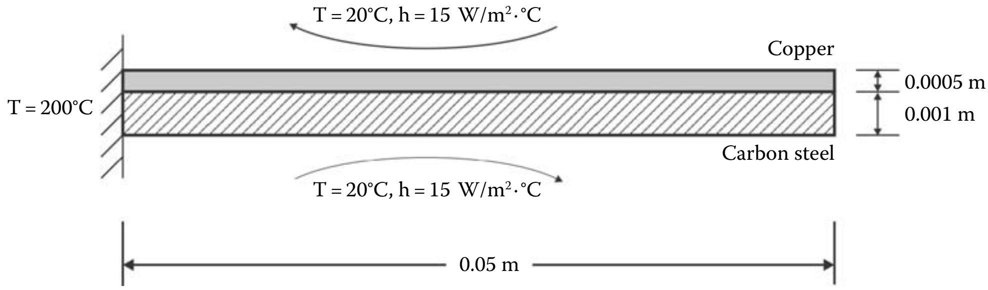


FIGURE 7.6 A thermocouple consists of copper and carbon steel plates.

the y-direction. The thermophysical properties of the used materials are shown Table 7.3.

PROBLEM 7.2

A pipeline used for transporting highly corrosive liquid is made of concentric carbon steel and aluminum pipes as shown in Figure 7.7. The inner pipe is made of aluminum to prevent corrosion, while the outer pipe is made of carbon steel to add strength to the pipeline in Table 7.4. At the inner surface of the pipe, the temperature is 100°C, while on the outer surface the temperature is 25°C. Due to thermal expansion variation between the carbon steel and aluminum, a thermal stress is developed at the contact surface that could damage the pipeline. Determine the maximum stress in the pipe. The displacement at inner surface of the pipe is fixed.

Table 7.3 Thermophysical properties of the used materials

Property	Copper	Carbon steel
Thermal conductivity (W/m·K)	401	111
Young's modulus (GPa)	140	210
Poisson's ratio	0.34	0.29
Thermal expansion (1/K)	16.5×10^{-6}	12×10^{-6}

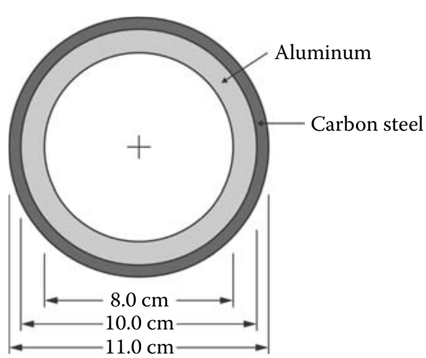


FIGURE 7.7 A composite pipeline.

Table 7.4 Thermophysical properties of aluminium and carbon steel

Property	Aluminum	Carbon steel
Thermal conductivity (W/m·°C)	83	111
Young's modulus (Pa)	70×10^9	210×10^9
Poisson's ratio	0.33	0.29
Thermal expansion (1/K)	23×10^{-6}	12×10^{-6}

PROBLEM 7.3

Study the thermal characteristics of an electronic chip mounted on a channel using ANSYS. The configuration is shown in Figure 7.8. The working fluid is air, and thermophysical properties of the chip and air are listed in Table 7.5. The inlet velocity is 0.0125 m/s, while the exit condition is a reference zero pressure. The inlet temperature is 20°C, and 5 Watts are generated in a heat source. Consider the problem as a steady-state heat transfer. Determine the following:

a. Maximum velocity in the x-direction and temperature in the domain
b. Satisfaction of the mass and energy balances
c. Heat transfer coefficient distribution at the surface of the chip.

PROBLEM 7.4

The two cylinders in a wind tunnel, as shown in Figure 7.9, are used to simulate heat exchanger tubes. The inlet velocity and temperature are 0.002 m/s and 25°C, respectively. Air is the working fluid ($\mu = 20 \times 10^{-6}$ Pa·s, $C_p = 1000$ J/kg·°C, k = 0.025 W/m·°C, and $\rho = 1.25$ kg/m^3). The surfaces of the cylinders are maintained at 50°C. The exit condition is a reference zero pressure.

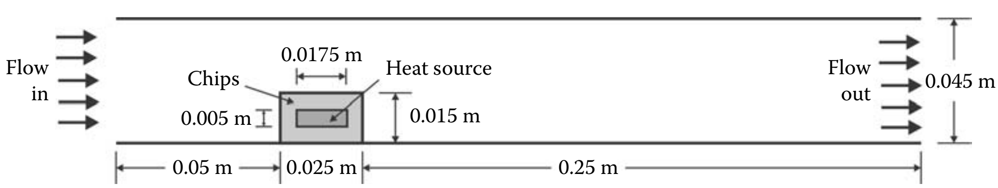


FIGURE 7.8 A channel with a chip.

Table 7.5 Thermophysical properties of the chip and air

Property	Air	Chip
Density (kg/m^3)	1.25	2250
Conductivity (W/m·°C)	0.025	0.4
Specific heat (J/kg·°C)	1005	950
Viscosity (N s/m^2)	20×10^{-6}	—

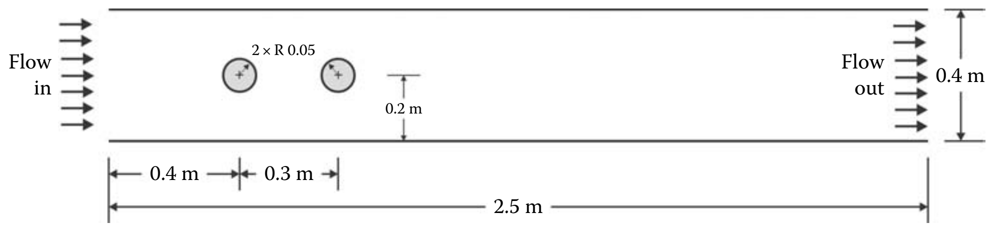


FIGURE 7.9 Two cylinders in a wind tunnel.

a. Show that the mass balance is satisfied.
b. Ensure maximum shear stress on the first and second cylinders.
c. Find the average heat transfer coefficient on the first and second cylinders.
d. Determine pressure drop in the channel.

PROBLEM 7.5

Water at low velocity enters a channel as shown in Figure 7.10. A triangular cross-sectional area cylinder is placed in the channel, at the shown location. The inlet velocity has a fully developed profile, and its temperature is uniform at 25°C. The exit condition is a reference zero pressure. The properties of water are $\rho = 997.1$ kg/m^3, $C_p = 4180$ J/kg °C, k = 0.6076 W/m·°C, and $\mu = 0.891 \times 10^{-3}$ Pa·s. Use the following equation for the velocity profile at the inlet:

$$u(y) = \frac{3}{2}\, 0.0001\left[1 - \left(\frac{2y}{H}\right)^2\right]$$

where H = 0.04. Ten Watts are generated in the triangular cylinder, which has properties of $\rho = 4600$ kg/m^3, $C_p = 1055$ J/kg·°C, and k = 76.5 W/m·°C.

a. Show that the mass balance is satisfied.
b. Show that the energy balance is satisfied.
c. Determine the maximum shear stress at the surface of the triangular cylinder.
d. Determine the average heat transfer coefficient in a triangular cylinder.
e. Determine pressure drop in the channel.

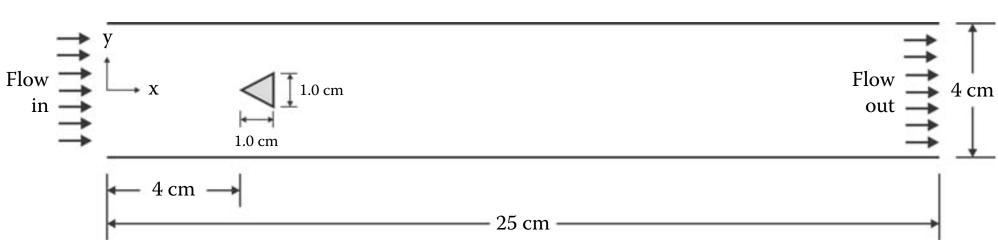


FIGURE 7.10 Channel with a triangular object.

This page intentionally left blank

CHAPTER EIGHT

Meshing guide

8.1 Mesh refinement

The accuracy of the finite element model can be enhanced by increasing either the number or the order of the elements. An increasing number of elements is called h-refinement, while increasing the order of the elements is called p-refinement. Increasing the number of elements or their order will lead to a significant increase in the computational time and required memory to solve the problem. Consider the stress analysis of a plate shown in Figure 8.1a. A linear quadratic element is used to mesh the domain. The stress concentration factor is required, and the exact solution is available from theory of elasticity. Figure 8.1b shows the finite element mesh. The mesh consists of 55 elements. To study the effect of the number of elements on the solution, the number of elements is increased and the stress concentration is obtained for the corresponding elements number.

Figure 8.2 shows the error of the stress concentration for different numbers of elements using linear quadratic elements. The domain is meshed with three different meshes, A, B, and C. The number of elements of mesh C is higher than that of mesh B, and the number of elements of mesh B is higher than that of mesh A. The figure indicates that as the number of elements increases, the relative error is decreased to approach a fixed value. Adding more elements to the mesh C will insignificantly reduce the error.

When the number of elements has no effect on solution, the solution is called a mesh-independent solution. On the other hand, increasing the order of the elements reduces the error for the same mesh size. Figure 8.3 indicates that replacing the linear quadratic elements with cubical ones reduces the percentage of the error by about 10%.

With high-order elements, the mesh-independent process is faster than with low-order elements, as shown in Figure 8.4. In addition, a mesh with the linear quadratic elements is becoming mesh independent at a very slow rate.

The higher order elements lead to more accurate results than lower order elements. However, the time required to complete the solution is important, especially for very large meshes. In addition to time, the higher order elements need a large amount of memory that could not

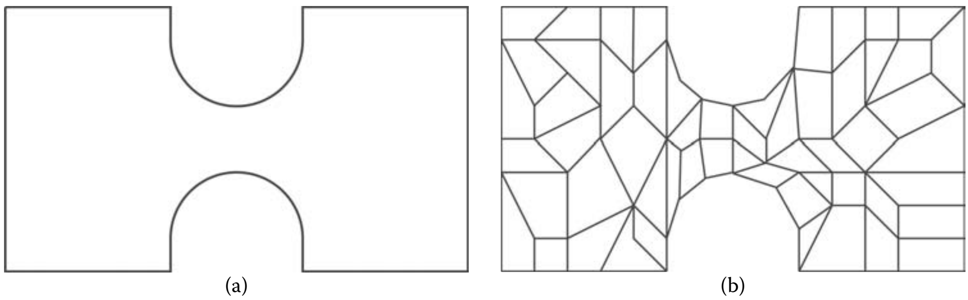


FIGURE 8.1 (a) Geometry of a plate and (b) finite element mesh.

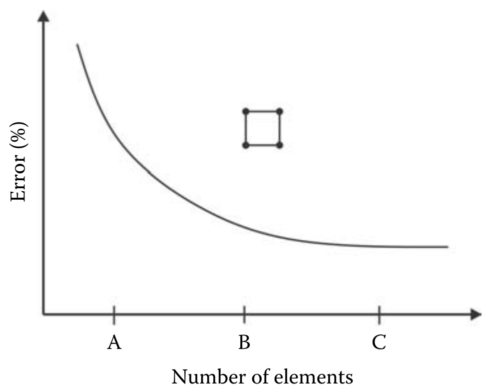


FIGURE 8.2 Error of the stress concentration for different mesh sizes using linear quadratic elements.

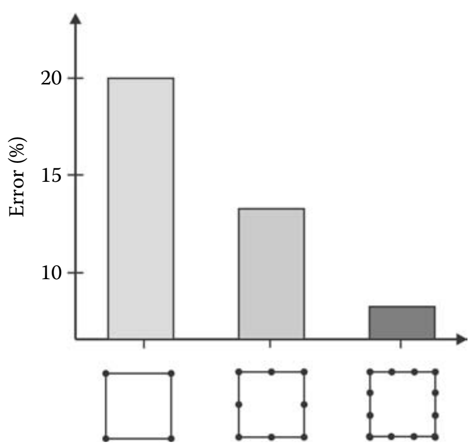


FIGURE 8.3 Percentage of error with different element orders.

be afforded. Figure 8.5 indicates that in spite of a low mesh convergence rate of linear quadratic elements, the time required to solve the problem is nearly independent of the number of elements in the mesh. In addition, increasing the order of the elements significantly increases the time required to solve the problem.

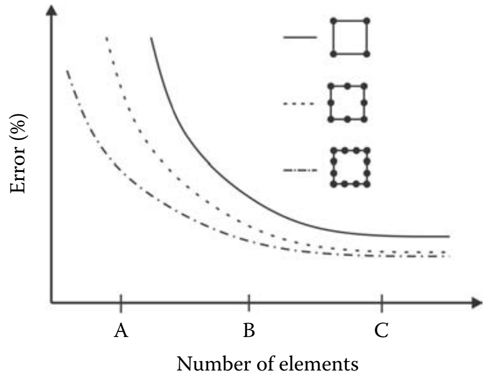


FIGURE 8.4 Error of the stress concentration for different numbers of elements and orders.

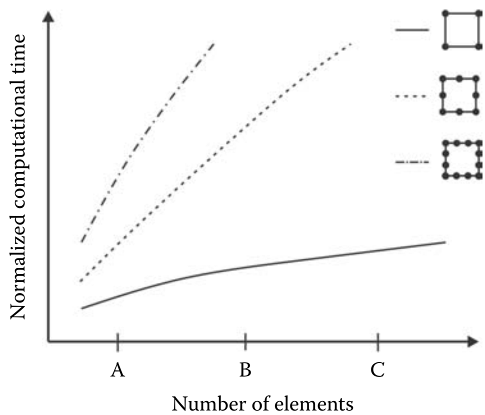


FIGURE 8.5 Normalized computational time for different quadrilateral elements.

8.2 Element distortion

When a complex geometry is meshed using an automatic mesh generator, elements at some critical regions in the geometry are distorted and the accuracy of the results is significantly decreased at these regions. Hence, this section concerns the element distortions and identifies these elements. In addition, guidelines about good element shapes are also introduced. In general, there are four types of elements distortion:

1. Mid-side node off-center
2. Taper
3. Skew
4. Aspect ratio

The mid-side node off-center type is when side nodes are shifted from the center of the side. As the node moves away from the center,

the errors increase. In practice, the mid-side node should not be shifted more than one-eighth of the side's length. Rectangular elements can be skewed or tapered. The angles of the rectangular elements should be 90°. For a rectangular element, and when the height-to-width ratio exceeds 2, the element has an aspect-ratio problem. It turns out that the aspect ratio for a perfectly shaped rectangular element is 1 and 1.15 for an equilateral triangle element. Aspect ratios higher than the recommended value can only occur in noncritical regions of the model, such as in uniform temperature or stress regions. Figure 8.6 shows the major types of element distortions. Element distortions are not limited to two-dimensional spaces, they can also occur in three-dimensional spaces. The most common one is the warping. Warping occurs when the face of a solid element does not lie in the same plane, and the warping angle should not be more than 10°.

8.3 Mapped mesh

Meshes can be classified as free or mapped meshes. The elements of the free meshes are randomly distributed in the domain with different sizes and shapes, and element distortions are most likely to appear in these meshes. Figure 8.7 shows a free mesh of a corrugated channel used for fluid flow analysis. Most of the finite element software generates this type of mesh. Poor results are always associated with free meshes. The free meshes should be only limited to very complex geometries.

Mapped meshes have organized distributed elements, and the meshes are almost free of the element distortions. Figure 8.8 shows a mapped mesh of a corrugated channel. The mesh designers can fully control the

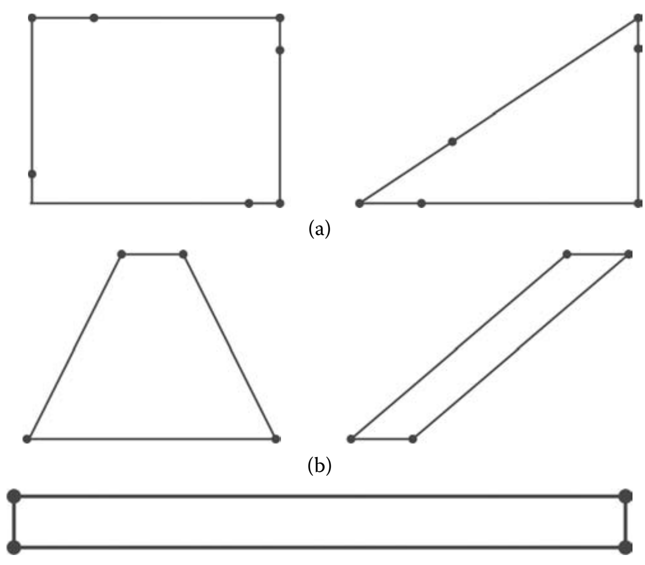


FIGURE 8.6 Major types of element distortions: (a) mid-side node off-center, (b) taper and skew, and (c) aspect ratio.

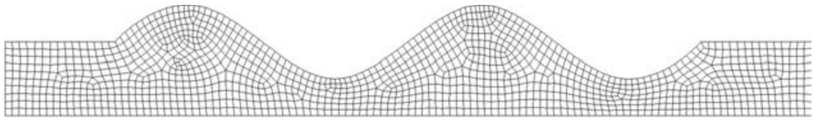


FIGURE 8.7 Free mesh of a corrugated channel.

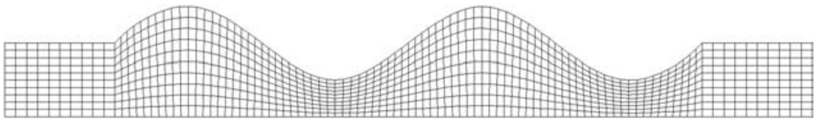


FIGURE 8.8 Mapped mesh of a corrugated channel.

size of the elements and their distribution in the computational domain. These meshes are typically used in finite element simulations because of their flexibility and ability to mesh complex geometries. Mesh designers should have a strong understanding of the physics of the problem before generating the mapped meshes. For example, for solid mechanics problems, the areas with high and uniform stress gradients should be identified, and the mesh should be designed accordingly.

For fluid problems, the regions close to the walls have very high velocity gradients due to shear forces, and elements should be dense at the region close to the walls. The geometry shown in Figure 8.9 is fin, and a thermal analysis is required. High temperature gradients at the upper region are expected. Therefore, elements must be concentrated at this region. The geometry is meshed with three different mapped meshes. The mesh in Figure 8.9a is unacceptable because the elements are not concentrated in the upper region, and poor results are expected. In the second mesh (Figure 8.9b), the upper region of the geometry has more

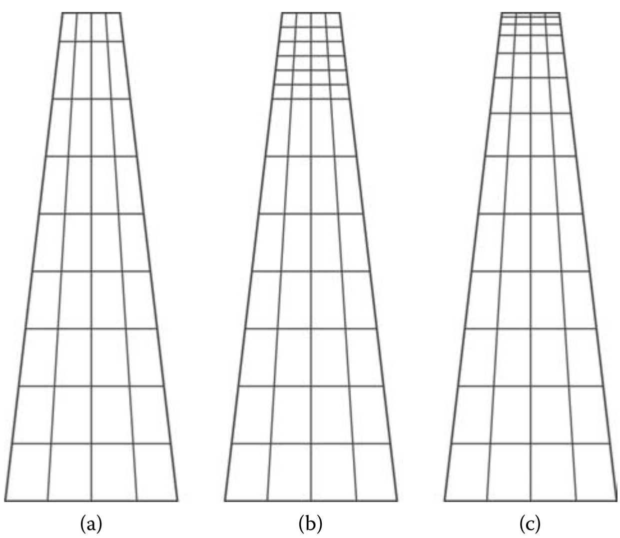


FIGURE 8.9 A fin (a) without elements concentration, (b) with only elements concentration at the upper region, and (c) with elements concentration at the upper region with gradual changes.

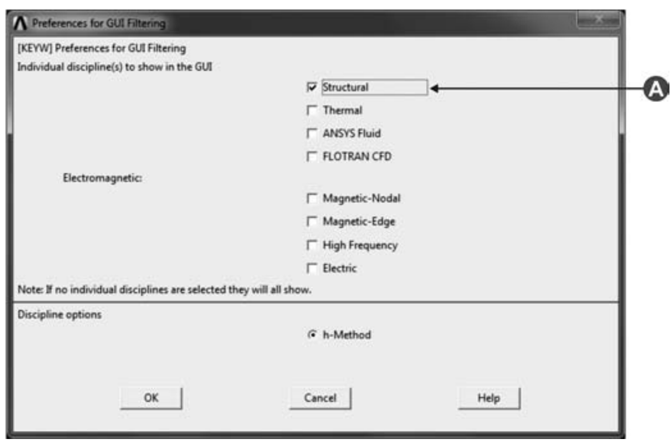
elements than the first mesh, but an abrupt change in element size should be avoided, because errors in the large elements will be transferred to the small elements. In addition, performing matrix operations of these meshes is difficult. The third mesh, Figure 8.9c, is the ideal mesh. Elements are concentrated at the upper region, and the change from the large elements in the lower region to small elements in the upper region is gradual.

8.4 Mapped mesh with ANSYS

The square plate with a hole shown in Figure 8.10 is subjected to tensile pressure at both vertical sides. Use ANSYS to create a mapped mesh of the plate and concentrate elements at the region close to the hole where high stress gradients are expected. Use quadratic elements with four nodes to mesh the plate.

Double click on the Mechanical APDL Product Launcher icon

Main Menu > Preferences



A select Structural

OK

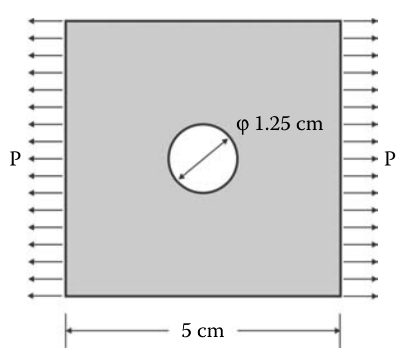
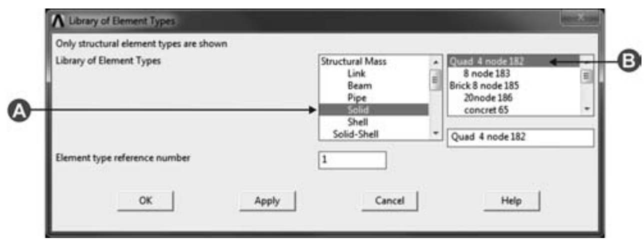


FIGURE 8.10 Square plate with a hole.

Main Menu > Preprocessor > Element Types > Add/Edit/Delete

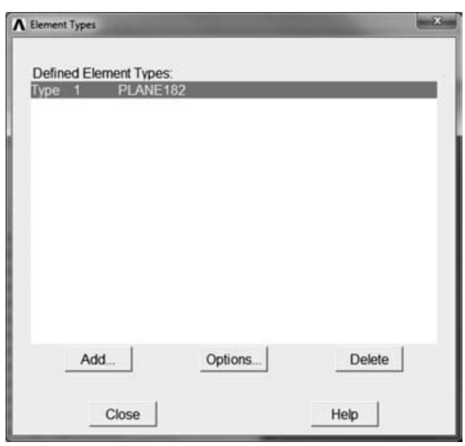
Add...



A select Solid

B select Quad 4 node 182

OK

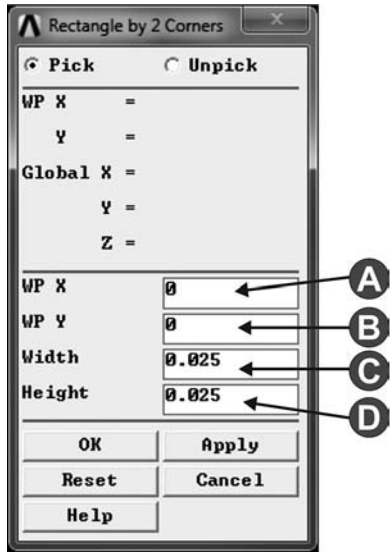


Close

The geometry is modeled by creating a square and a circle. A Boolean operation is utilized to remove the circle from the square using subtraction. The advantage of symmetry in the problem is considered. Only the upper right quarter is considered. To create a mapped mesh with

ANSYS, the computational domain must be divided into areas, and each area must have four sides only.

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners



A type 0 in WP X

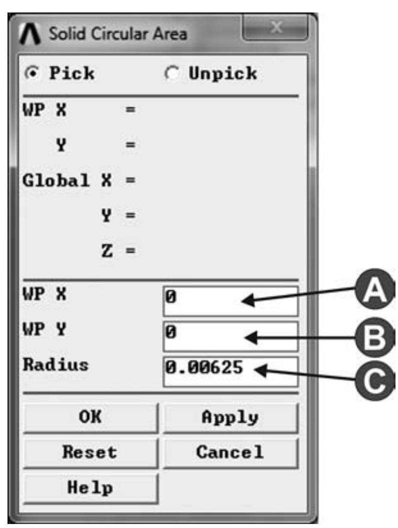
B type 0 in WP Y

C type 0.025 in Width

D type 0.025 in Height

OK

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle



A type 0.0 in WP X

B type 0.0 in WP Y

C type 0.00625 in Radius

OK

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

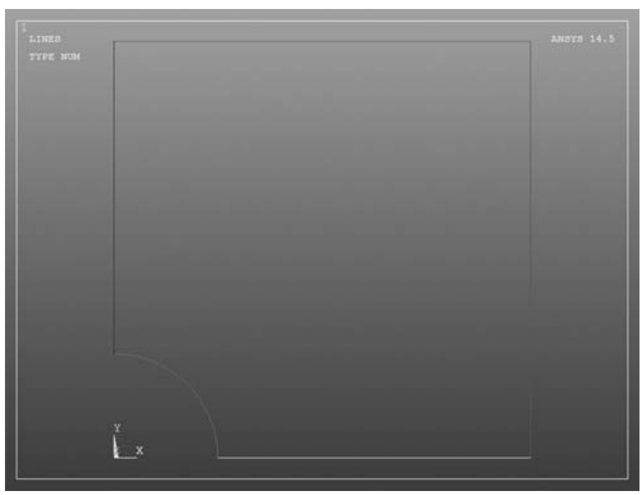
Click on square area, and then in Subtract Areas window, click on

Apply

Click on circular area, and then in Subtract Areas window, click on

OK

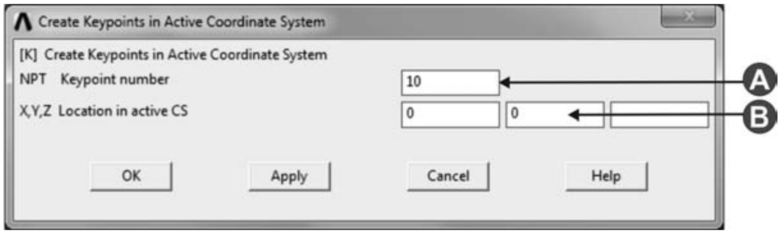
Utility Menu > Plot > Lines



ANSYS graphics show the lines of the geometry

As shown in the figure, the geometry is one area which is composed of five lines. Therefore, the mapped mesh cannot be created. The area should be divided into areas that each should be composed of four sides, and there are many options. As a suggestion for this example, the area is divided into two areas along the symmetry line of the geometry. First, a keypoint is created at the origin. Second, a line is created between the keypoints at the origin and upper right corner of the plate. Third, the area is divided by a line, which has just been created. The new Keypoint will have number 10, and this number is selected to avoid a conflict with existing keypoints.

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS



A type 10 in NPT Keypoint number

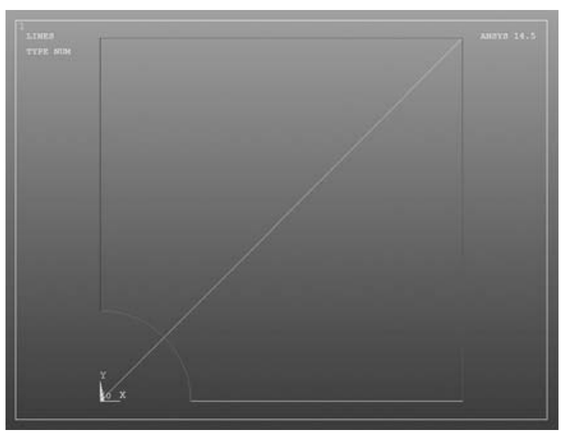
B type 0 and 0 in X,Y,Z Location in active CS

OK

ANSYS Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Click on keypoints at the origin, and at the origin and upper right corner of the geometry. Then, in Create Straight Line window, click on

OK



ANSYS graphics show the created line

ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Line

Click on area to select it. Then, in Divide Area by Line window, click on

Apply

Click on the just created line to select it. Then, in Divide Area by Line window, click on

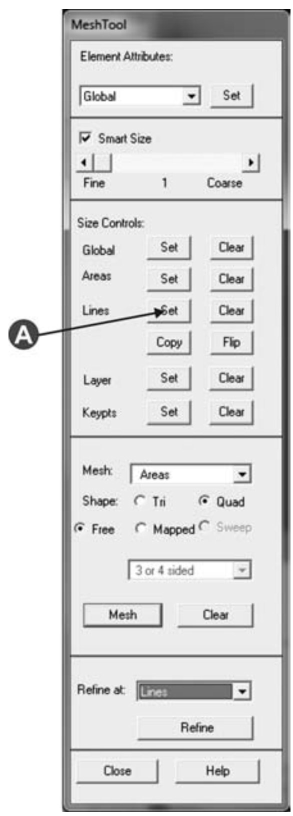
OK

Utility Menu > Plot > Areas

ANSYS graphics show the areas

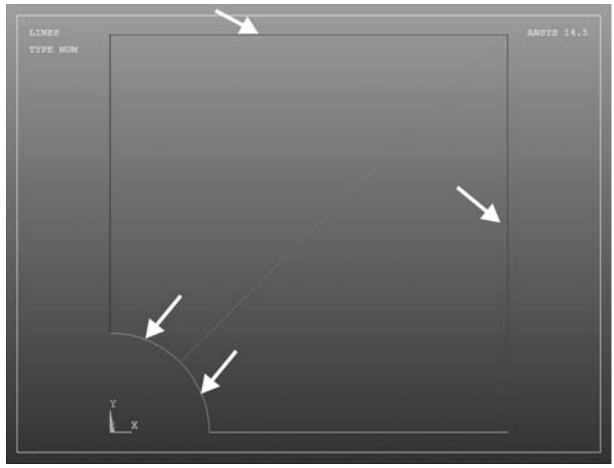
Now the geometry is ready to be meshed. It consists of two areas, and each area is composed of four lines. Elements should be concentrated at the region close to the hole since high stress gradients are expected at this region. The lines should be divided into segments, and the two opposite lines of an area must have the same number of division.

Main Menu > Preprocessor > Meshing > Mesh Tool



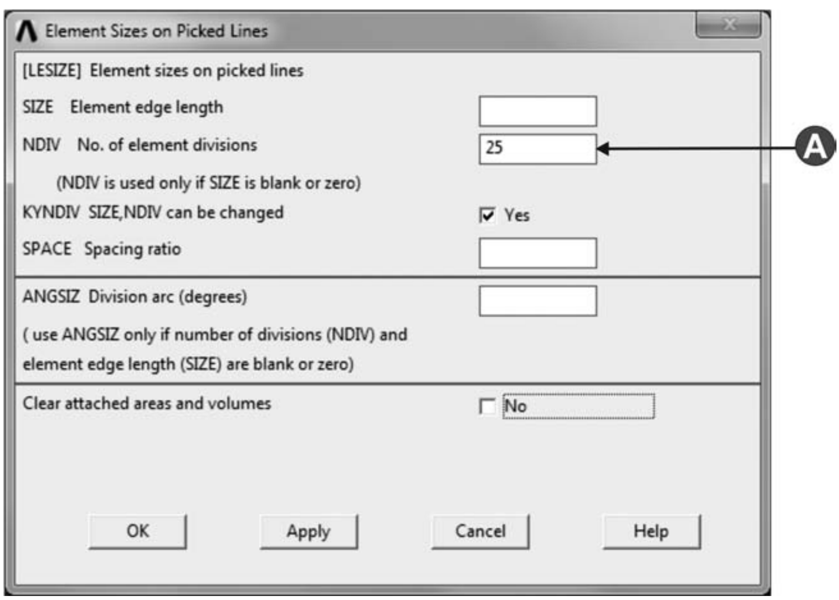
A click on Set in Lines

The lines indicated in the following figure should have the same line division and spacing ratio.



Select the lines as shown in the figure. Then in Element Sizes on Picked Lines window, click on

Apply

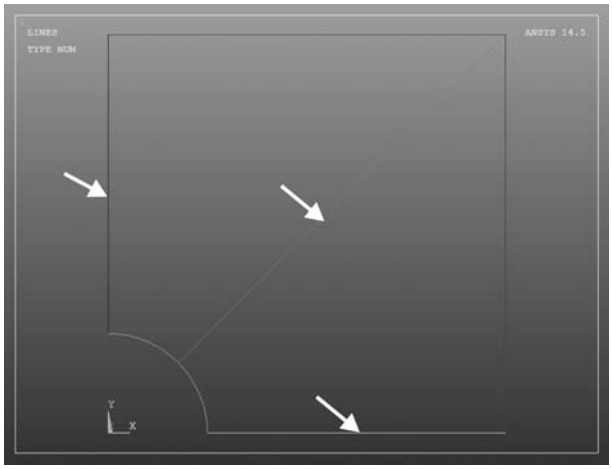


A type 25 in the NDIV

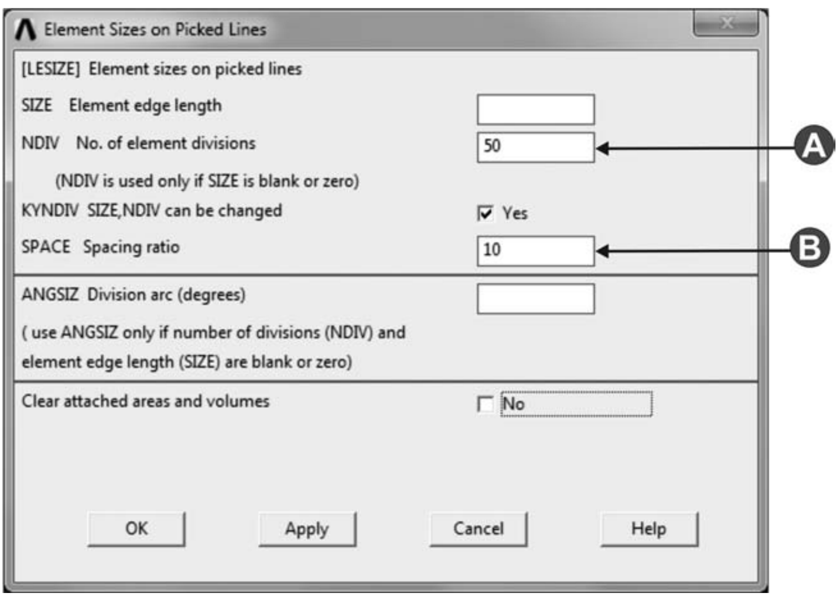
The number 25 means that the selected lines will be divided into 25 segments, and each segment will be one element.

Apply

Select the lines, as shown in the following figure. Then in Element Sizes on Pick Lines window, click on



OK



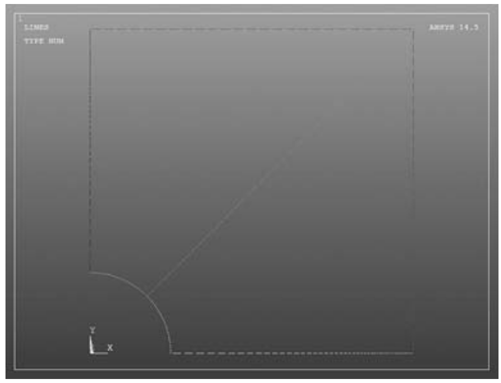
A type 50 in NDIV

B type 10 in SPACE

The number 50 means that the selected lines will be divided into 50 segments, and the segments are gradually increasing by a spacing ratio of 10.

OK

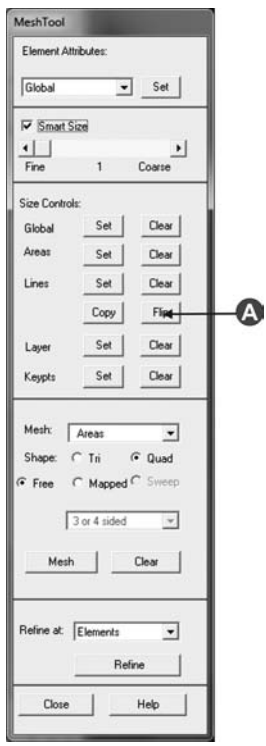
Utility Menu > Plot > Lines



ANSYS graphics show the lines division

Notice that the division of the horizontal and vertical lines is wrongly implemented. The segment lengths are increasing toward the hole, and they should be reversed. The reversion can be easily done using flip in Mesh Tool as follows:

Main Menu > Preprocessor > Meshing > Mesh Tool

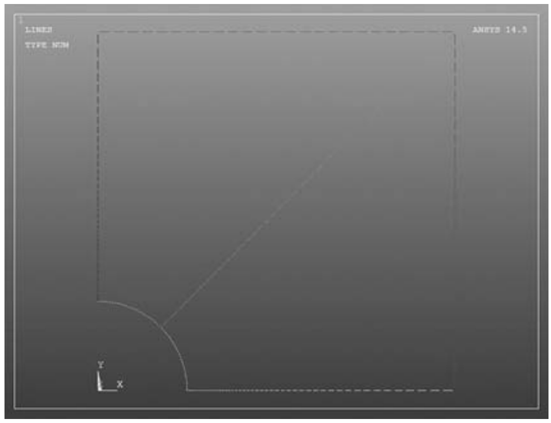


A click on Flip in Lines

Select the horizontal and vertical lines, which have a wrong division. Then in Flip Line Bias window, click on

OK

Utility Menu > Plot > Lines



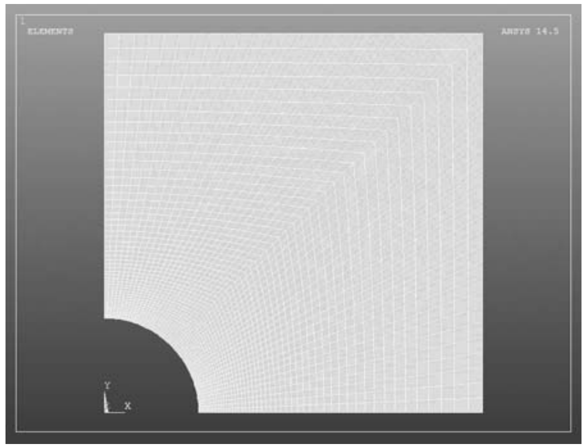
ANSYS graphics show the lines division

A select Mapped

Mesh

In Mesh Areas window, click on

Pick All



ANSYS graphics show a mapped mesh

PROBLEM 8.1

Create a mapped mesh for the geometries shown in Figure 8.11, and the geometries are used for solid mechanics.

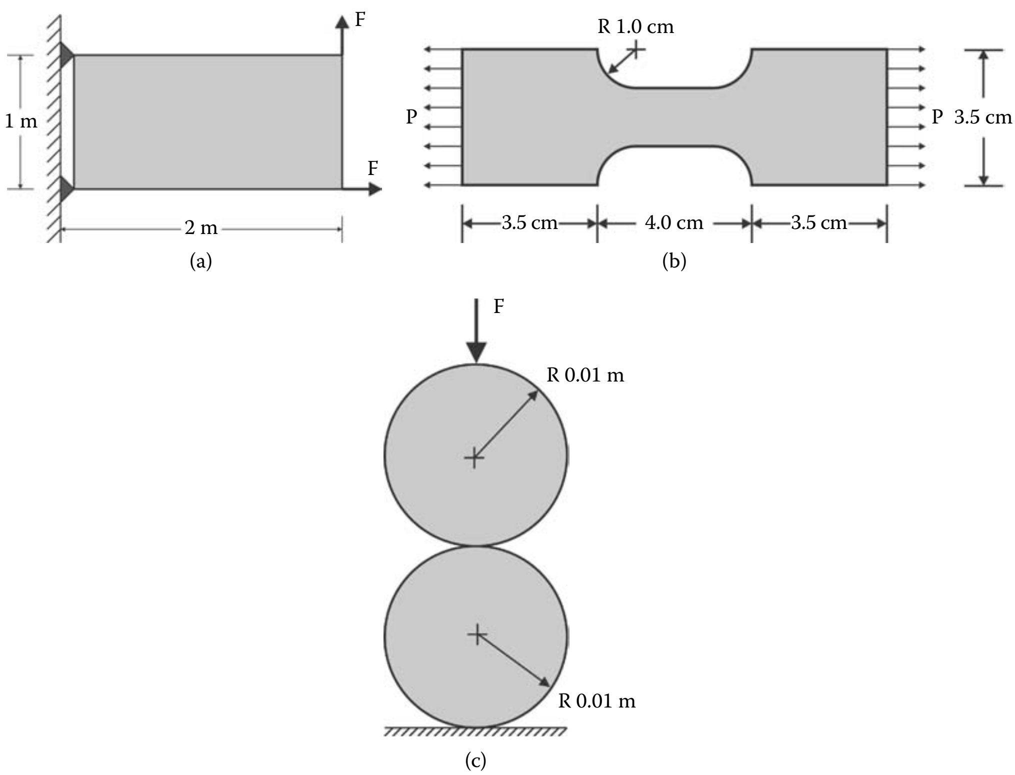


FIGURE 8.11 Geometries for solid mechanics: (a) a rectangular plate subjected to vertical and horizontal forces, (b) a notched rectangular plate subjected to a tensile pressure, and (c) two horizontal cylinders on a flat plate with a force.

PROBLEM 8.2

Create a mapped mesh for the geometries shown in Figure 8.12, and the geometries are used for heat transfer.

PROBLEM 8.3

Create a mapped mesh for the geometries shown in Figure 8.13, and the geometries are used for fluid mechanics.

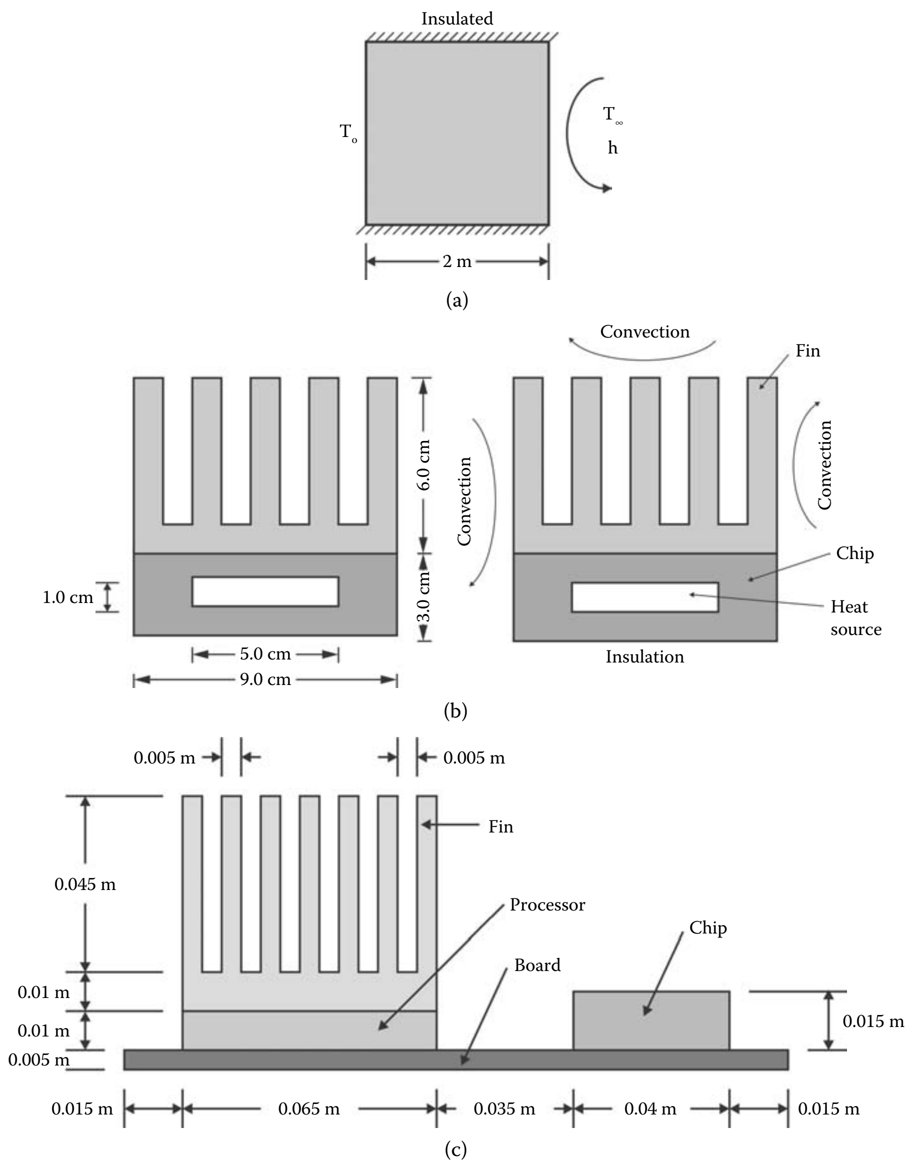


FIGURE 8.12 Geometries for heat transfer: (a) two-dimensional body subjected to fixed temperature and convection boundary conditions, (b) a chip and fin with heat generation in the heat source, and (c) electronic board assembly.

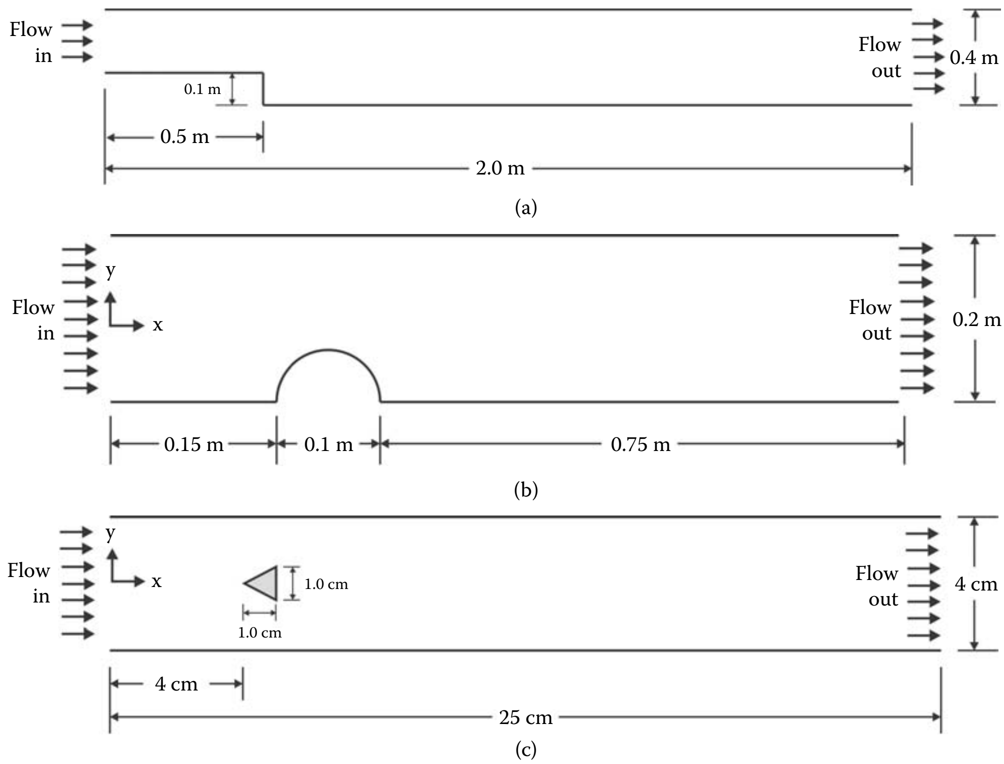


FIGURE 8.13 Geometries for fluid mechanics: (a) flow over a backward-facing step, (b) flow over a half cylinder in a channel, and (c) flow over a triangular cylinder in a channel.

Bibliography

V. Adams and A. Askenazi. *Building Better Product with Finite Element Analysis*. ONWORD Press, Santa Fe, NM.

Ansys Theory. ANSYS online manual. ANSYS, Canonsburg, PA.

T. Chandrupatla and A. Belegundu. *Introduction to Finite Elements in Engineering*. 3rd edition. Prentice Hall, Upper Saddle River, NJ.

M. Fagan. *Finite Element Analysis: Theory and Practice*. Pearson Education, Harlow, Essex, England.

S. Moaveni. *Finite Element Analysis: Theory and Application with ANSYS*. Prentice Hall, Upper Saddle River, NJ.

J. Reddy. *An Introduction to the Finite Element Method*. 2nd edition. McGraw-Hill, New York, NY.

E. Thompson. *Introduction to the Finite Element Method: Theory, Programming, and Applications*. Wiley, Hoboken, NJ.

T. Yang. *Finite Element Structural Analysis*. Prentice Hall, Englewood Cliffs, NJ.

This page intentionally left blank

This page intentionally left blank

ENGINEERING – GENERAL

"A must read for those interested in exploring the possibilities of using the finite element method (FEM) as a practical analysis tool for structural mechanics, stress analysis, vibration, heat transfer, and fluid dynamic problems. … Alawadhi systematically introduces the theory of FEA and demonstrates a step-by-step procedure for practical analysis using ANSYS. This book is a must-have reference for students, academics, and practitioners in the field of mechanical, civil, environmental, and design engineering."
—Dr. Arun Arjunan, University of Wolverhampton, UK

Uses a Step-By-Step Technique Directed with Guided Problems and Relevant Screen Shots

Simulation use is on the rise, and more practicing professionals are depending on the reliability of software to help them tackle real-world mechanical engineering problems. *Finite Element Simulations Using ANSYS, Second Edition* offers a basic understanding of the principles of simulation in conjunction with the application of ANSYS. Employing a step-by-step process, the book presents practical end-of-chapter problems that are solved using ANSYS and explains the physics behind them. The book examines structure, solid mechanics, vibration, heat transfer, and fluid dynamics. Each topic is treated in a way that allows for the independent study of a single subject or related chapter.

What's New in the Second Edition:

- Introduces the newest methods in modeling and meshing for finite element analysis
- Modifies ANSYS examples to comply with the newest version of ANSYS
- Replaces many ANSYS examples used in the first edition with more general, comprehensive, and easy-to-follow examples
- Adds more details to the theoretical material on the finite element
- Provides increased coverage of finite element analysis for heat transfer topics
- Presents open-ended, end-of-chapter problems tailored to serve as class projects

Finite Element Simulations Using ANSYS, Second Edition functions as a fundamental reference for finite element analysis with ANSYS methods and procedures, as well as a guide for project and product analysis and design.

K24312

ISBN: 978-1-4822-6197-4
90000
9 781482 261974

CRC Press
Taylor & Francis Group
an informa business
www.crcpress.com

6000 Broken Sound Parkway, NW
Suite 300, Boca Raton, FL 33487
711 Third Avenue
New York, NY 10017
2 Park Square, Milton Park
Abingdon, Oxon OX14 4RN, UK