Introduction to Finite Element Analysis Using SolidWorks® Simulation 2013

Randy H. Shih

SDC PUBLICATIONS
Schroff Development Corporation

Schroff Development Corporation
P.O. Box 1334, Mission, KS 66222
(913) 262-2664
www.SDCpublications.com
Publisher: Stephen Schroff

Trademarks

SolidWorks and *SolidWorks simulation* are trademarks of **Dassault Systèmes**. Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation. All other trademarks are trademarks of their respective holders.

Examination Copies:

Books received as examination copies are for review purposes only and may not be made available for student use. Resale of examination copies is prohibited.

Electronic Files:

Any electronic files associated with this book are licensed to the original user only. These files may not be transferred to any other party.

Shih, Randy H.
Introduction to Finite Element Analysis Using SolidWorks Simulation 2013
Randy H. Shih

ISBN 978-1-58503-772-8

The author and publisher of this book have used their best efforts in preparing this book. These efforts include the development, research and testing of the material presented. The author and publisher shall not be liable in any event for incidental or consequential damages with, or arising out of, the furnishing, performance, or use of the material.

Printed and bound in the United States of America.

Table of Contents

Preface i
Acknowledgments ii

Introduction

Introduction Intro-2
Development of Finite Element Analysis Intro-2
FEA Modeling Considerations Intro-3
Types of Finite Elements Intro-4
Finite Element Analysis Procedure Intro-6
Matrix Definitions Intro-6
Getting Started with *SolidWorks* Intro-9
Starting *SolidWorks* Intro-9
Units Setup Intro-11
SolidWorks Screen Layout Intro-13
- Menu Bar Toolbar Intro-13
- Menu Bar Pull-down Menus Intro-14
- Heads-up View Toolbar Intro-14
- Features Toolbar Intro-14
- Sketch Toolbar Intro-14
- Command Manager Intro-15
- Feature Manager Design Tree Intro-16
- Graphics Area Intro-17
- Reference Triad Intro-17
- Origin Intro-17
- Confirmation Corner Intro-17
- Graphics Cursor or Crosshairs Intro-17
- Message and Status Bar Intro-17

Basic Functions of Mouse Buttons Intro-18
[Esc] – Canceling Commands Intro-18
On-Line Help Intro-19
Leaving *SolidWorks* Intro-19
Creating a CAD Files Folder Intro-20

Chapter 1
The Direct Stiffness Method

Introduction 1-2
One-dimensional Truss Element 1-3
Example 1.1 1-5
Example 1.2 1-7

Basic Solid Modeling using SolidWorks 1-10
The Adjuster Design 1-10
Starting SolidWorks 1-10
Step 1: Creating a Rough Sketch 1-12
Graphics Cursors 1-12
Geometric Relation Symbols 1-14
Step 2: Apply/Modify Relations and Dimensions 1-15
Changing the Dimension Standard 1-16
Viewing Functions – *Zoom* and *Pan* 1-17
Modifying the Dimensions of the Sketch 1-18
Step 3: Completing the Base Solid Feature 1-19
Isometric View 1-20
Rotation of the 3D Model – *Rotate View* 1-20
Rotation and Panning – *Arrow Keys* 1-22
Dynamic Viewing – Quick Keys 1-23
Viewing Tools – Heads-up View Toolbar 1-25
View Orientation 1-26
Display Style 1-27
Orthographic vs. Perspective 1-27
Customizing the Heads-up View Toolbar 1-27
Sketch Plane 1-28
Step 4-1: Adding an Extruded Boss Feature 1-30
Step 4-2: Adding an Extruded Cut Feature 1-33
Save the Part and Exit 1-35
Questions 1-36
Exercises 1-37

Chapter 2
Truss Elements in Two-Dimensional Spaces

Introduction 2-2
Truss Elements in Two-Dimensional Spaces 2-2
Coordinate Transformation 2-5
EXAMPLE 2.1 2-9
Solution 2-10
Global Stiffness Matrix 2-10
EXAMPLE 2.2 2-13
Solution 2-13
Questions 2-19
Exercises 2-20

Chapter 3
2D Trusses in MS Excel and Truss Solver

Direct Stiffness Matrix Method using Excel	3-2
EXAMPLE 3.1	3-2
Establish the Global K Matrix for each Member	3-3
Assembly of the Overall Global Stiffness Matrix	3-8
Solving the Global Displacements	3-10
Calculating Reaction Forces	3-16
Determining the Stresses in Elements	3-18
The Truss Solver and the Truss View Programs	3-23
The Truss View Program	3-30
Questions	3-32
Exercises	3-33

Chapter 4
Truss Elements in SolidWorks Simulation

One-dimensional Line Elements	4-2
Starting *SolidWorks*	4-4
Units Setup	4-5
Creating the CAD Model – Solid Modeling Approach	4-6
A CAD Model is NOT an FEA Model	4-15
The SolidWorks Simulation Module	4-16
Creating an FEA Model	4-17
Assign the Element Material Property	4-19
Applying Boundary Conditions - Constraints	4-20
Applying External Loads	4-23
Create the FEA Mesh and Run the Solver	4-25
Viewing the Stress Results	4-26
Viewing the Displacement Results	4-28
Questions	4-29
Exercises	4-30

Chapter 5
SolidWorks Simulation Two-Dimensional Truss Analysis

Finite Element Analysis Procedure	5-2
Preliminary Analysis	5-3
Starting *SolidWorks*	5-4
Units Setup	5-5
Creating the CAD Model – Structural Member Approach	5-6
Creating Structural Members in SolidWorks	5-8
Weldment Profiles	5-9
Activate the SolidWorks Simulation Module	5-12
Setting Up Truss Elements	5-14

Assign the Element Material Property 5-15
Applying Boundary Conditions - Constraints and Loads 5-16
Applying External Loads 5-21
Create the FEA Mesh and Run the Solver 5-23
Viewing the Stress results 5-24
Viewing the Internal Loads of All members 5-26
Viewing the Reaction Forces at the supports 5-27
Questions 5-28
Exercises 5-29

Chapter 6
Three-Dimensional Truss Analysis

Three-Dimensional Coordinate Transformation Matrix 6-2
Stiffness Matrix 6-3
Degrees of Freedom 6-3
Problem Statement 6-5
Preliminary Analysis 6-5
Starting *SolidWorks* 6-7
Units Setup 6-8
Creating the CAD Model – Structural Member Approach 6-9
Creating Weldments Profiles *in SolidWorks* 6-12
Creating Structural Members using the New Profile 6-16
Editing the Dimensions of the New Profile 6-18
Activate the SolidWorks Simulation Module 6-19
Setting Up Truss Elements 6-21
Assign the Element Material Property 6-22
Applying Boundary Conditions - Constraints 6-23
Applying External Loads 6-24
Create the FEA Mesh and Run the Solver 6-26
Using the Probe Option to View Individual Stress 6-27
Viewing the Internal Loads of All Members 6-28
Questions 6-29
Exercises 6-30

Chapter 7
Basic Beam Analysis

Introduction 7-2
Modeling Considerations 7-2
Problem Statement 7-3
Preliminary Analysis 7-3
Starting *SolidWorks* 7-6
Units Setup 7-7

Creating the CAD Model – Structural Member Approach 7-8
Creating a Rectangular Weldment Profile 7-10
Creating Structural Members using the New Profile 7-14
Adjusting the Orientation of the Profile 7-15
Adding a Datum Point for the Concentrated Load 7-16
Activate the SolidWorks Simulation Module 7-18
Assign the Element Material Property 7-20
Applying Boundary Conditions - Constraints 7-21
Applying Concentrated Point Loads 7-24
Applying the Distributed Load 7-26
Create the FEA Mesh and Run the Solver 7-28
What Went Wrong? 7-29
Directions 1 and 2 in Shear and Moment Diagrams 7-32
Questions 7-34
Exercises 7-35

Chapter 8
Beam Analysis Tools

Introduction 8-2
Problem Statement 8-2
Preliminary Analysis 8-3
Stress Components 8-4
Starting *SolidWorks* 8-6
Creating the CAD Model – Structural Member Approach 8-7
Creating a Rectangular Weldment Profile 8-9
Creating Structural Members using the New Profile 8-13
Adjusting the Orientation of the Profile 8-14
Adding a Datum Point 8-15
Activate the SolidWorks Simulation Module 8-17
Assign the Element Material Property 8-19
Applying Boundary Conditions - Constraints 8-20
Applying the Distributed Load 8-23
Create the FEA Mesh and Run the Solver 8-25
Shear and Moment Diagrams 8-26
Using the Probe Option to Examine Stress at Point1 8-28
Questions 8-29
Exercises 8-30

Chapter 9
Statically Indeterminate Structures

Introduction 9-2
Problem Statement 9-3

Preliminary Analysis 9-3
Starting *SolidWorks* 9-6
Creating the CAD Model 9-7
Creating a Circular Weldment Profile 9-9
Creating Structural Members using the New Profile 9-13
Adding a Datum Point for the Concentrated Load 9-14
Activate the SolidWorks Simulation Module 9-16
Assign the Element Material Property 9-18
Applying Boundary Conditions - Constraints 9-19
Applying the Concentrated Point Load 9-23
Create the FEA Mesh and Run the Solver 9-24
Viewing the Internal Loads of All members 9-25
Shear and Moment Diagrams 9-26
Questions 9-28
Exercises 9-29

Chapter 10
Two-Dimensional Surface Analysis

Introduction 10-2
Problem Statement 10-3
Preliminary Analysis 10-3
- Maximum Normal Stress 10-3
- Maximum Displacement 10-4

Geometric Considerations of Finite Elements 10-5
Starting *SolidWorks* 10-6
Creating a CAD Model in SolidWorks 10-7
Activate the SolidWorks Simulation Module 10-10
Defining a Surface Model 10-12
Assign the Element Material Property 10-13
Applying Boundary Conditions - Constraints 10-14
Applying the External Load 10-17
H-Element versus P-Element 10-20
Create the first 2D Mesh –Coarse Mesh 10-21
Run the Solver 10-23
Refinement of the Mesh– Global Element Size 0.10 10-25
Refinement of the Mesh– Global Element Size 0.05 10-27
Refinement of the Mesh– Global Element Size 0.03 10-29
Refinement of the Mesh– Global Element Size 0.02 10-30
Comparison of Results 10-31
Questions 10-32
Exercises 10-33

Chapter 11
Three-Dimensional Solid Elements

Introduction 11-2
Problem Statement 11-3
Preliminary Analysis 11-4
Starting *SolidWorks* 11-7
Creating a CAD Model in SolidWorks 11-8
- Define the Sweep Path 11-8
- Define the Sweep Section 11-10
- Create the Swept Feature 11-12
- Create a Cut Feature 11-13

Activate the SolidWorks Simulation Module 11-15
Assign the Element Material Property 11-17
Applying Boundary Conditions - Constraints 11-18
Applying the External Load 11-19
Create the first FEA Mesh –Coarse Mesh 11-20
Run the Solver 11-22
Refinement of the Mesh– Global Element Size 0.10 11-24
Refinement of the FEA Mesh – Mesh Control Option 11-26
Refinement of the FEA Mesh – Automatic Transition 11-29
Comparison of Results 11-31
Notes on FEA Linear Static Analyses 11-32
Questions 11-33
Exercises 11-34

Chapter 12
3D Thin Shell Analysis

Introduction 12-2
Problem Statement 12-4
Preliminary Analysis 12-4
Starting *SolidWorks* 12-6
Creating a CAD Model in SolidWorks 12-7
Activate the SolidWorks Simulation Module 12-9
Assign the Element Material Property 12-11
Applying Boundary Conditions - Constraints 12-12
Applying the External Pressure 12-14
Create the first FEA Mesh –Coarse Mesh 12-15
Run the Solver and View the Results 12-16
Refinement of the Mesh– Global Element Size 0.0125 12-17
Starting a New 3D Surface Model 12-18
Starting a New FEA Study 12-21
Completing the Definition of the Surface Model 12-22
Assign the Element Material Property 12-23

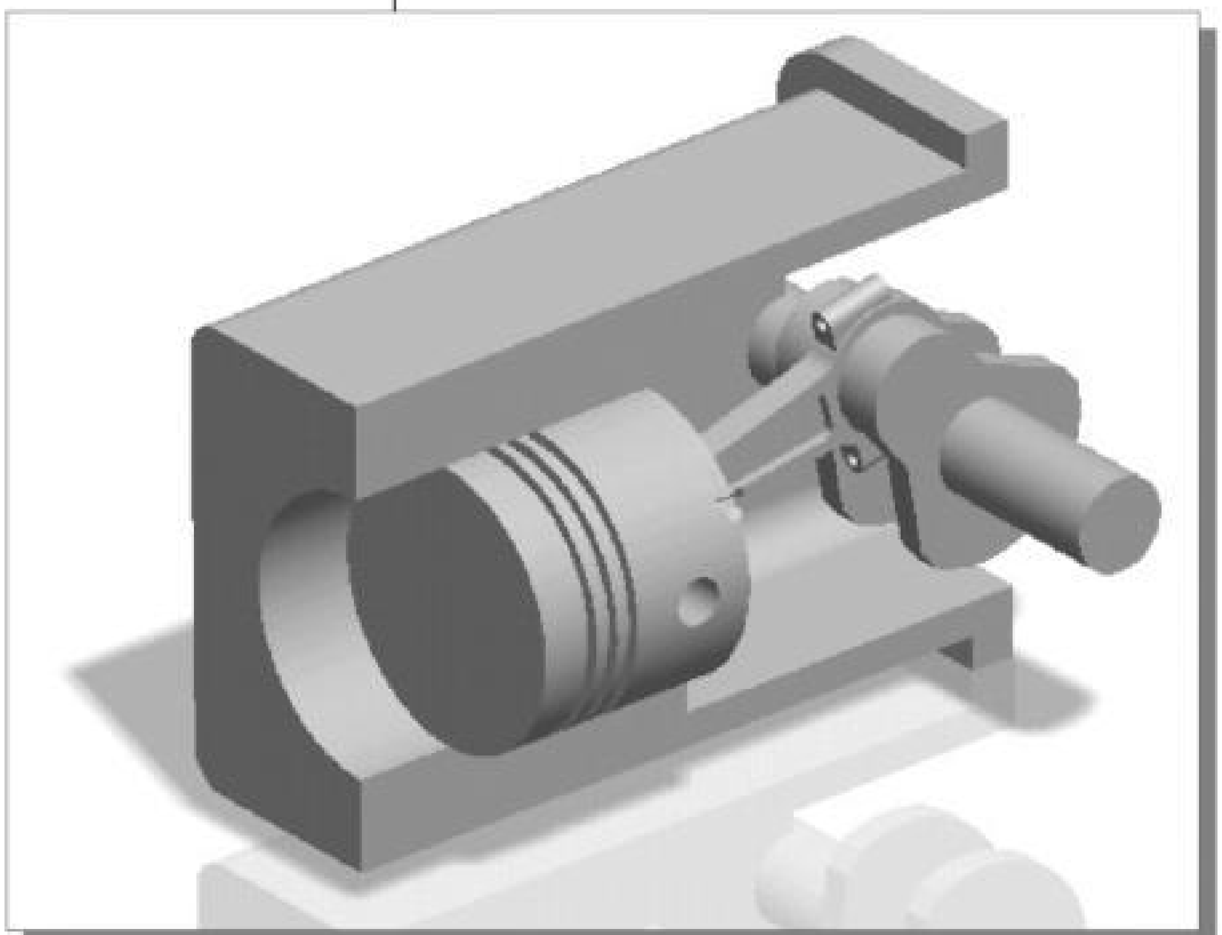
Applying Boundary Conditions - Constraints 12-24
Applying the External Pressure 12-28
Create the first FEA Mesh –Coarse Mesh 12-29
Run the Solver and View the Results 12-30
Refinement of the Mesh– Global Element Size 0.010 12-31
Questions 12-32
Exercises 12-33

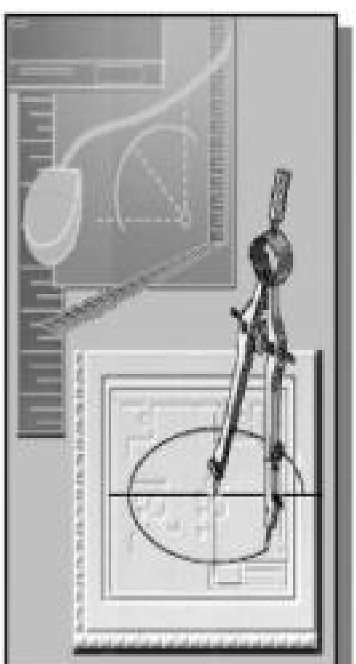
Chapter 13
Dynamic Modal Analysis

Introduction 13-2
Problem Statement 13-3
Preliminary Analysis 13-3
The Cantilever Beam Modal Analysis Program 13-6
Starting *SolidWorks* 13-9
Creating a CAD Model in SolidWorks 13-10
Activate the SolidWorks Simulation Module 13-12
Assign the Element Material Property 13-15
Create the first FEA Mesh 13-16
View the Results 13-18
Refinement of the Mesh– Global Element Size 0.15 13-21
Adding an Additional Mass to the System 13-23
One-Dimensional Beam Frequency Analysis 13-27
Conclusions 13-28
Questions 13-29
Exercises 13-30

Index

Introduction





Learning Objectives

- Development of Finite Element Analysis
- FEA Modeling Considerations
- Finite Element Analysis Procedure
- Getting Started with SolidWorks
- Startup Options and Units Setup
- SolidWorks Screen Layout
- Mouse Buttons
- SolidWorks On-Line Help

Introduction

Design includes all activities involved from the original concept to the finished product. Design is the process by which products are created and modified. For many years, designers sought ways to describe and analyze three-dimensional designs without building physical models. With the advancements in computer technology, the creation of three-dimensional models on computers offers a wide range of benefits. Computer models are easier to interpret and easily altered. Simulations of real-life loads can be applied to computer models and the results graphically displayed.

Finite Element Analysis (FEA) is a numerical method for solving engineering problems by simulating real-life-operating situations on computers. Typical problems solved by Finite Element Analysis include structural analysis, heat transfer, fluid flow, soil mechanics, acoustics, and electromagnetism. *SolidWorks* is an integrated package of mechanical computer aided engineering software tools developed by **Dassault Systèmes** (**DS**). *SolidWorks* is a suite of programs, including the *Finite Element Analysis module* (***SolidWorks Simulation***), which is used to facilitate a concurrent engineering approach to the design, analysis, and manufacturing of mechanical engineering products. This text focuses on basic structural analysis using the integrated ***SolidWorks*** and ***SolidWorks Simulation***.

Development of Finite Element Analysis

Finite element analysis procedures evolved gradually from the work of many people in the fields of engineering, physics, and applied mathematics. The finite element analysis procedure was first applied to problems of stress analysis. The essential ideas began to appear in publications during the 1940s. In 1941, Hrenikoff proposed that the elastic behavior of a physically continuous plate would be similar to a framework of one-dimensional rods and beams, connected together at discrete points. The problem could then be handled by familiar methods for trusses and frameworks. In 1943, Courant's paper detailed an approach to solving the torsion problem in elasticity. Courant described the use of piecewise linear polynomials over a triangularized region. Courant's work was not noticed and soon forgotten, since the procedure was impractical to solve by hand.

In the early 1950s, with the developments in digital computers, Argyris and Kelsey converted the well-established "framework-analysis" procedure into matrix format. In 1956, Turner, Clough, Matin, and Topp derived stiffness matrices for truss elements, beam elements and two-dimensional triangular and rectangular elements in plane stress. Clough introduced the first use of the phrase "finite element" in 1960. In 1961, Melosh developed a flat, rectangular-plate bending-element, followed by development of the curved-shell bending-element by Grafton and Strome in 1963. Martin developed the first three-dimensional element in 1961 followed by Gallagher, Padlog and Bijlaard in 1962 and Melosh in 1964.

From the mid-1960s to the end of the 1970s, finite element analysis procedures spread beyond structural analysis into many other fields of application. Large general purpose FEA software began to appear. By the late 1980s, FEA software became available on microcomputers, complete with automatic mesh-generation, interactive graphics, and pre-processing and post-processing capabilities.

In this text, we will follow a logical order, parallel to the historical development of the finite element analysis procedures, in learning the fundamental concepts and commands for performing finite element analysis using ***SolidWorks*** and ***SolidWorks Simulation***. We will begin with the one-dimensional truss element, beam element, and move toward the more advanced features of *SolidWorks Simulation*. This text also covers the general procedures of performing two-dimensional and three-dimensional solid FE analyses. The concepts and techniques presented in this text are also applicable to other FEA packages. Throughout the text, many of the classic strength of materials and machine design problems are used as examples and exercises, which hopefully will help build up the user's confidence on performing FEA analyses.

FEA Modeling Considerations

The analysis of an engineering problem requires the idealization of the problem into a mathematical model. It is clear that we can only analyze the selected mathematical model, and that all the assumptions in this model will be reflected in the predicted results. We cannot expect any more information in the prediction than the information contained in the model. Therefore it is crucial to select an appropriate mathematical model that will most closely represent the actual situation. It is also important to realize that we cannot predict the response exactly, because it is impossible to formulate a mathematical model that will represent all the information contained in an actual system.

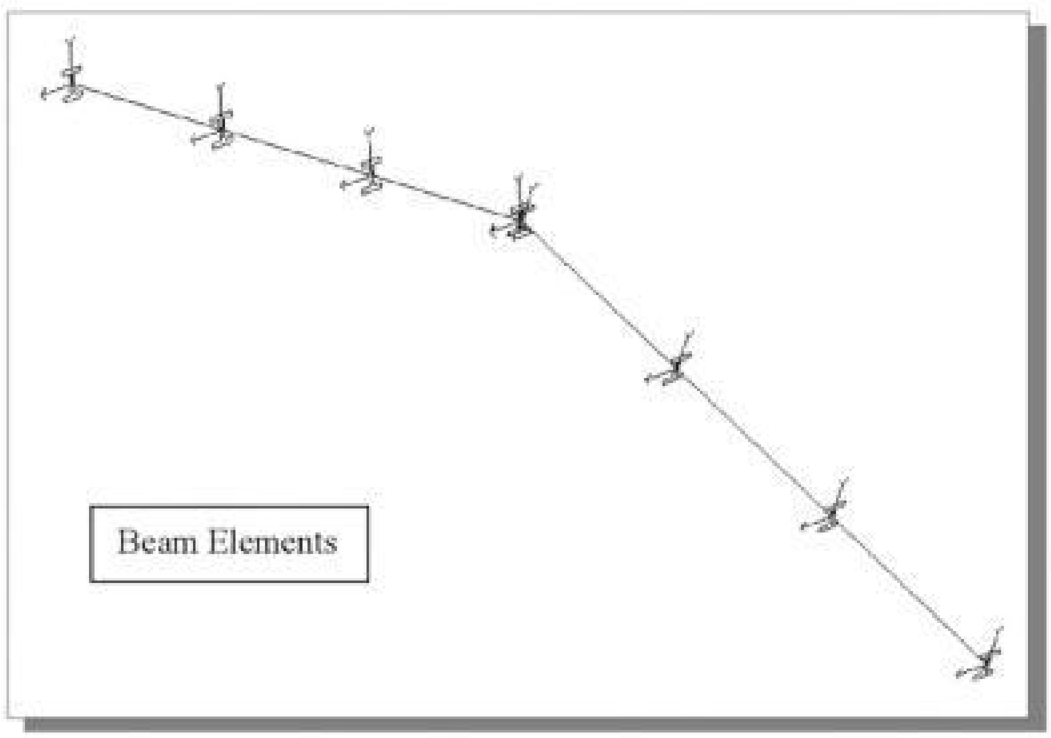
As a general rule, finite element modeling should start with a simple model. Once a mathematical model has been solved accurately and the results have been interpreted, it is feasible to consider a more refined model in order to increase the accuracy of the prediction of the actual system. For example, in a structural analysis, the formulation of the actual loads into appropriate models can drastically change the results of the analysis. The results from the simple model, combined with an understanding of the behavior of the system, will assist us in deciding whether and at which part of the model we want to use further refinements. Clearly, the more complicated model will include more complex response effects, but it will also be more costly and sometimes more difficult to interpret the solutions.

Modeling requires that the physical action of the problem be understood well enough to choose suitable kinds of analyses. We want to avoid the waste of time and computer resources caused by over-refinement and badly shaped elements. Once the results have been calculated, we must check them to see if they are reasonable. Checking is very important because it is easy to make mistakes when we rely upon the FEA software to solve complicated systems.

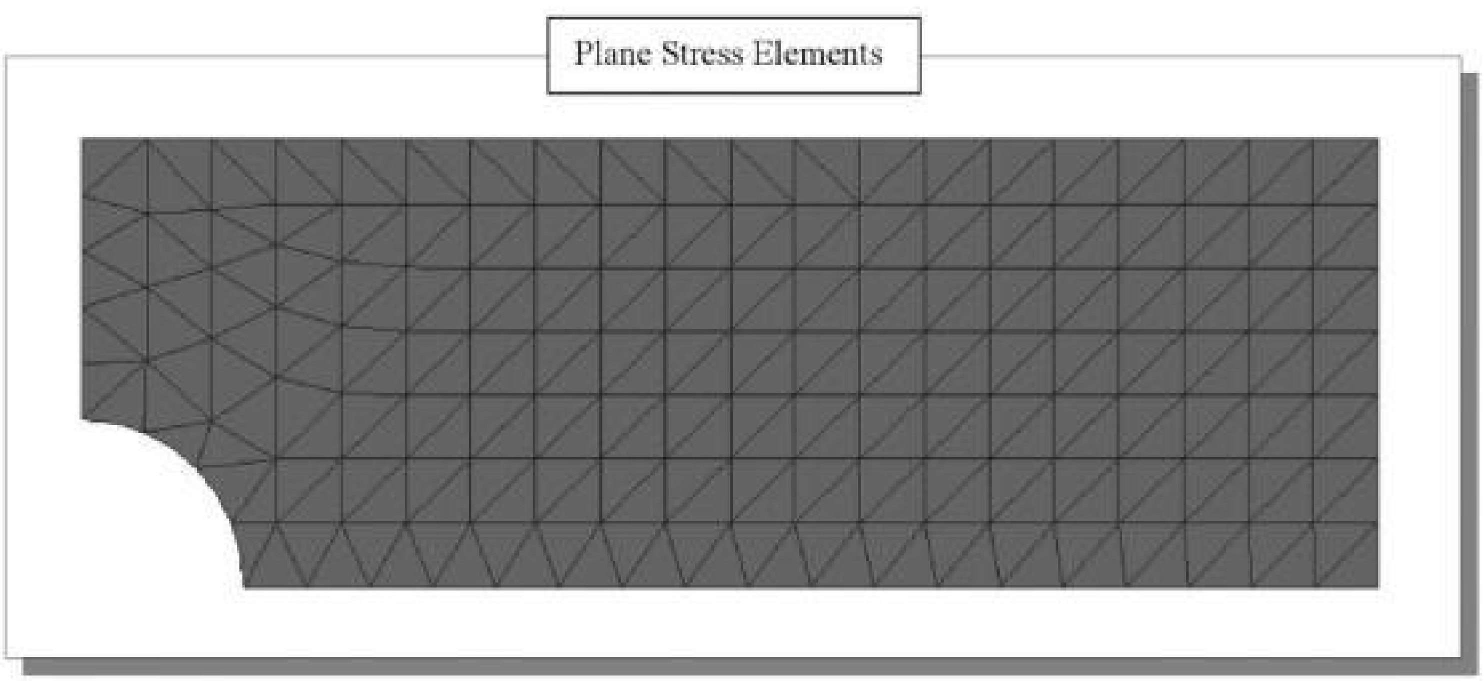
Types of Finite Elements

The finite element analysis method is a numerical solution technique that finds an approximate solution by dividing a region into small sub-regions. The solution within each sub-region that satisfies the governing equations can be reached much more simply than that required for the entire region. The sub-regions are called ***elements*** and the elements are assembled through interconnecting a finite number of points on each element called ***nodes***. Numerous types of finite elements can be found in commercial FEA software and new types of elements are being developed as research is done worldwide. Depending on the dimensions, finite elements can be divided into three categories:

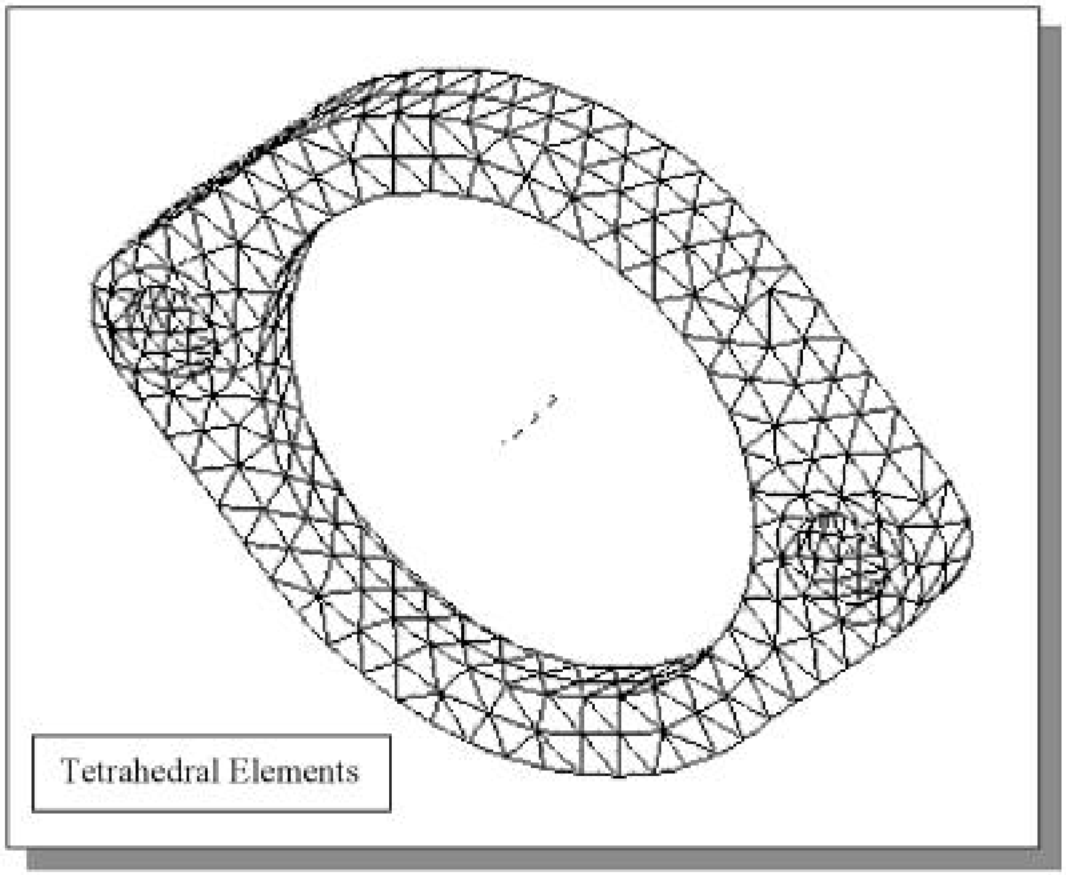
1. **One-dimensional** line elements: Truss, beam and boundary elements.



2. **Two-dimensional** plane elements: Plane stress, plane strain, axisymmetric, membrane and shell elements.



3. **Three-dimensional** volume elements: Tetrahedral, hexahedral, and brick elements.



Typically, finite element solutions using one-dimensional line elements are as accurate as solutions obtained using conventional truss and beam theories. It is usually easier to get FEA results than doing hand calculations using conventional theories. However, very few closed form solutions exist for two-dimensional elements and almost none exist for three-dimensional solid elements.

In theory, all designs could be modeled with three-dimensional volume elements. However, this is not practical since many designs can be simplified with reasonable assumptions to obtain adequate FEA results without any loss of accuracy. Using simplified models greatly reduces the time and effort in reaching FEA solutions.

Finite Element Analysis Procedure

Prior to carrying out the finite element analysis, it is important to do an approximate preliminary analysis to gain some insights into the problem and as a means of checking the finite element analysis results.

For a typical linear static analysis problem, the finite element analysis requires the following steps:

1. Preliminary Analysis.

2. Preparation of the finite element model:
   a. Model the problem into finite elements.
   b. Prescribe the geometric and material information of the system.
   c. Prescribe how the system is supported.
   d. Prescribe how the loads are applied to the system.

3. Perform calculations:
   a. Generate a stiffness matrix of each element
   b. Assemble the individual stiffness matrices to obtain the overall, or global, stiffness matrix.
   c. Solve the global equations and compute displacements, strains, and stresses.

4. Post-processing of the results:
   a. Viewing the stress contours and the displaced shape.
   b. Checking any discrepancy between the preliminary analysis results and the FEA results.

Matrix Definitions

The use of vectors and matrices is of fundamental importance in engineering analysis because it is with the use of these quantities that complex procedures can be expressed in a compact and elegant manner. One need not understand vectors or matrices in order to use FEA software. However, by studying the matrix structural analysis, we develop an understanding of the procedures common to the implementation of structural analysis as well as the general finite element analysis. The objective of this section is to present the fundamentals of matrices, with emphasis on those aspects that are important in finite element analysis. In the next chapter we will introduce the derivation of matrix structural analysis, the stiffness matrix method. *MATRIX Algebra* is a powerful tool for use in programming the FEA methods for electronic digital computers. Matrix notation represents a simple and easy-to-use notation for writing and solving sets of simultaneous algebraic equations.

- **Matrix** A **matrix** is a rectangular array of elements arranged in rows and columns. Applications in this text deal only with matrices whose elements are real numbers.

For example,

$$[\mathbf{A}] = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{21} & A_{22} & A_{23} \end{bmatrix}$$

[A] is a rectangular array of two rows and three columns, thus called a 2×3 matrix. The element A_{ij} is the element in the i^{th} row and j^{th} column.

- **Column Matrix (Row Matrix)** A **column (row) matrix** is a matrix having one column (row). A single-column array is commonly called a column matrix or *vector*. For example:

$$\{F\} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

- **Square Matrix** A **square matrix** is a matrix having equal numbers of rows and columns.

- **Diagonal Matrix** A **diagonal matrix** is a square matrix with nonzero elements only along the diagonal of the matrix.

- **Addition** The **addition of matrices** involves the summation of elements having the same "address" in each matrix. The matrices to be summed must have identical dimensions. The addition of matrices of different dimensions is not defined. Matrix addition is associative and commutative.

For example:

$$[\mathbf{A}] + [\mathbf{B}] = \begin{bmatrix} 1 & 2 \\ 3 & \boxed{4} \end{bmatrix} + \begin{bmatrix} 2 & 4 \\ 6 & \boxed{8} \end{bmatrix} = \begin{bmatrix} 3 & 6 \\ 9 & \boxed{12} \end{bmatrix}$$

- **Multiplication by a Constant** If a matrix is to be multiplied by a constant, every element in the matrix is multiplied by that constant. Also, if a constant is factored out of a matrix, it is factored out of each element. For example:

$$3 \times [\mathbf{A}] = 3 \times \begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix} = \begin{bmatrix} 3 & 6 \\ 9 & 12 \end{bmatrix}$$

- **Multiplication of Two Matrices** Assume that [C] = [A][B], where [A], [B], and [C] are matrices. Element C_{ij} in matrix [C] is defined as follows:

$$C_{ij} = A_{i1} \times B_{1j} + A_{i2} \times B_{2j} + \cdots + A_{ik} \times B_{kj}$$

For example:

$$[\,C\,] = [\,A\,]\ [\,B\,] = \begin{bmatrix} \overrightarrow{1 \quad 2} \\ 3 \quad 4 \end{bmatrix} \begin{bmatrix} 5 & 6 \\ 7 & 8 \end{bmatrix} = \begin{bmatrix} 19 & 22 \\ 43 & 50 \end{bmatrix}$$

$$C_{11} = 1 \times 5 + 2 \times 7 = 19, \quad C_{12} = 1 \times 6 + 2 \times 8 = 22$$
$$C_{21} = 3 \times 5 + 4 \times 7 = 43, \quad C_{22} = 3 \times 6 + 4 \times 8 = 50$$

- **Identity Matrix** An **identity matrix** is a diagonal matrix with each diagonal element equal to unity.

For example:

$$[\,I\,] = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

- **Transpose of a Matrix** The **transpose of a matrix** is a matrix obtained by interchanging rows and columns. Every matrix has a transpose. The transpose of a column matrix (vector) is a row matrix; the transpose of a row matrix is a column matrix.

For example:

$$[A] = \begin{bmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \end{bmatrix} \quad [A]^T = \begin{bmatrix} 1 & 4 \\ 2 & 5 \\ 3 & 6 \end{bmatrix}$$

- **Inverse of a Square Matrix** A square matrix *may* have an inverse. The product of a matrix and its inverse matrix yields the identity matrix.

$$[A]\,[A]^{-1} = [A]^{-1}\,[A] = [\,I\,]$$

The reader is referred to the following techniques for matrix inversion:

1. Gauss-Jordan elimination method
2. Gauss-Seidel iteration method

These techniques are popular and are discussed in most texts on numerical techniques.

Getting Started with *SolidWorks*

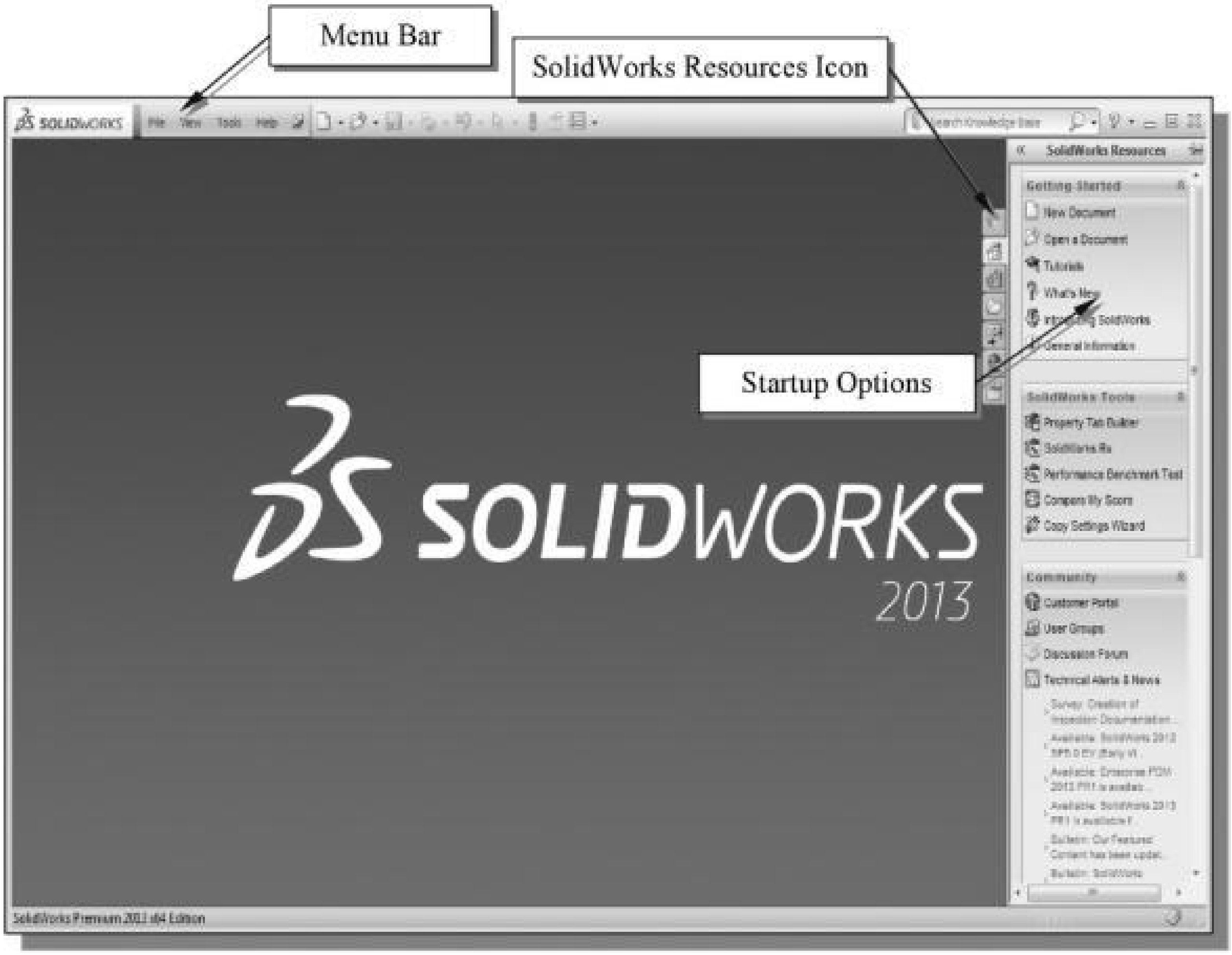
- *SolidWorks* is composed of several application software modules (these modules are called *applications*), all sharing a common database. In this text, the main concentration is placed on the solid modeling modules used for part design. The general procedures required in creating solid models, engineering drawings, and assemblies are illustrated.

Starting *SolidWorks*

How to start *SolidWorks* depends on the type of workstation and the particular software configuration you are using. With most *Windows* systems, you may select **SolidWorks** on the *Start* menu or select the **SolidWorks** icon on the desktop. Consult your instructor or technical support personnel if you have difficulty starting the software.

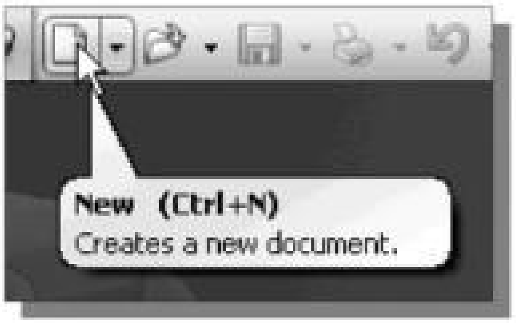
❖ The program takes a while to load, so be patient. The tutorials in this text are based on the assumption that you are using *SolidWorks'* default settings. If your system has been customized for other uses, some of the settings may appear differently and not work with the step-by-step instructions in the tutorials. Contact your instructor and/or technical support personnel to restore the default software configuration.

Once the program is loaded into memory, the *SolidWorks* program window appears. This screen contains the *Menu Bar* and the *Task Pane*. The *Menu Bar* contains a subset of commonly used tools from the *Menu Bar* toolbar (New, Open, Save, etc.), the *SolidWorks* menus, the *SolidWorks Search* oval, and a flyout menu of *Help* options. By default, the *Menu Bar* menus are hidden. To display them, move the cursor over or click the *SolidWorks* logo.



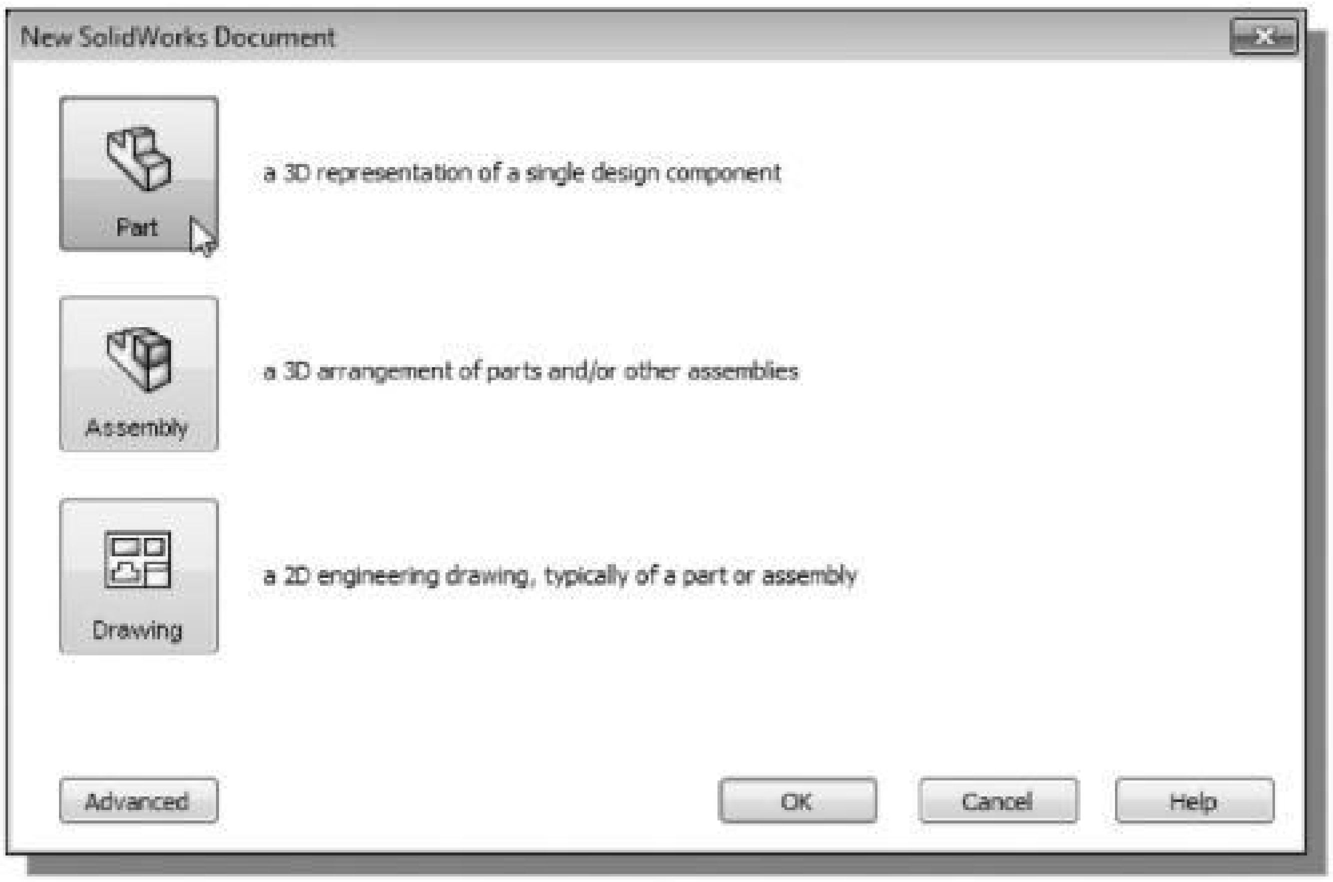
If the *task pane* does not appear to the right of the screen, click the SolidWorks Resources icon on the right side of the window. Other options for the *task pane* are the *Design Library* and the *File Explorer*. The icons for these options appear below the SolidWorks Resources icon. To collapse the *task pane*, click anywhere in the main area of the *SolidWorks* window.

The following two startup options are available: New and Open. The New option allows us to start a new modeling task. The Open option allows us to open an existing model file. These two commands can be executed in the *Getting Started* dialog box in the *SolidWorks Resources* task pane or on the *Menu Bar*. The *Getting Started* dialog box also contains a Tutorials option which provides some quick helps that illustrate the features and general procedures of using *SolidWorks*.



➢ Select the **New** icon on the *Menu Bar* with a single click of the left-mouse-button. The *New SolidWorks Document* dialog box appears.

Three icons appear in the *New SolidWorks Document* dialog box. Selecting the appropriate icon will allow creation of a new **Part**, **Assembly** or **Drawing** file. A part is a single three-dimensional (3D) solid model. Parts are the basic building blocks in modeling with *SolidWorks*. An assembly is a 3D arrangement of parts (components) and/or other assemblies (subassemblies). A drawing is a 2D representation of a part or assembly.

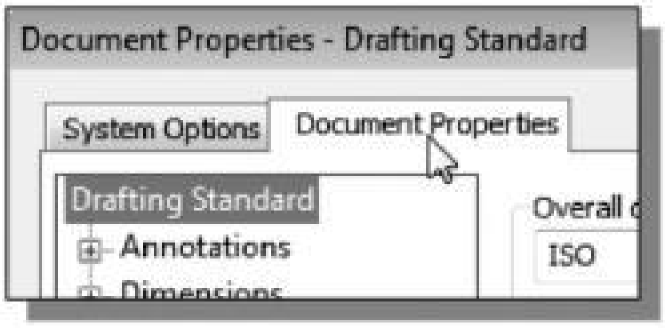


➢ Select the **Part** icon as shown. Click **OK** in the *New SolidWorks Document* dialog box to open a new part file.

Units Setup

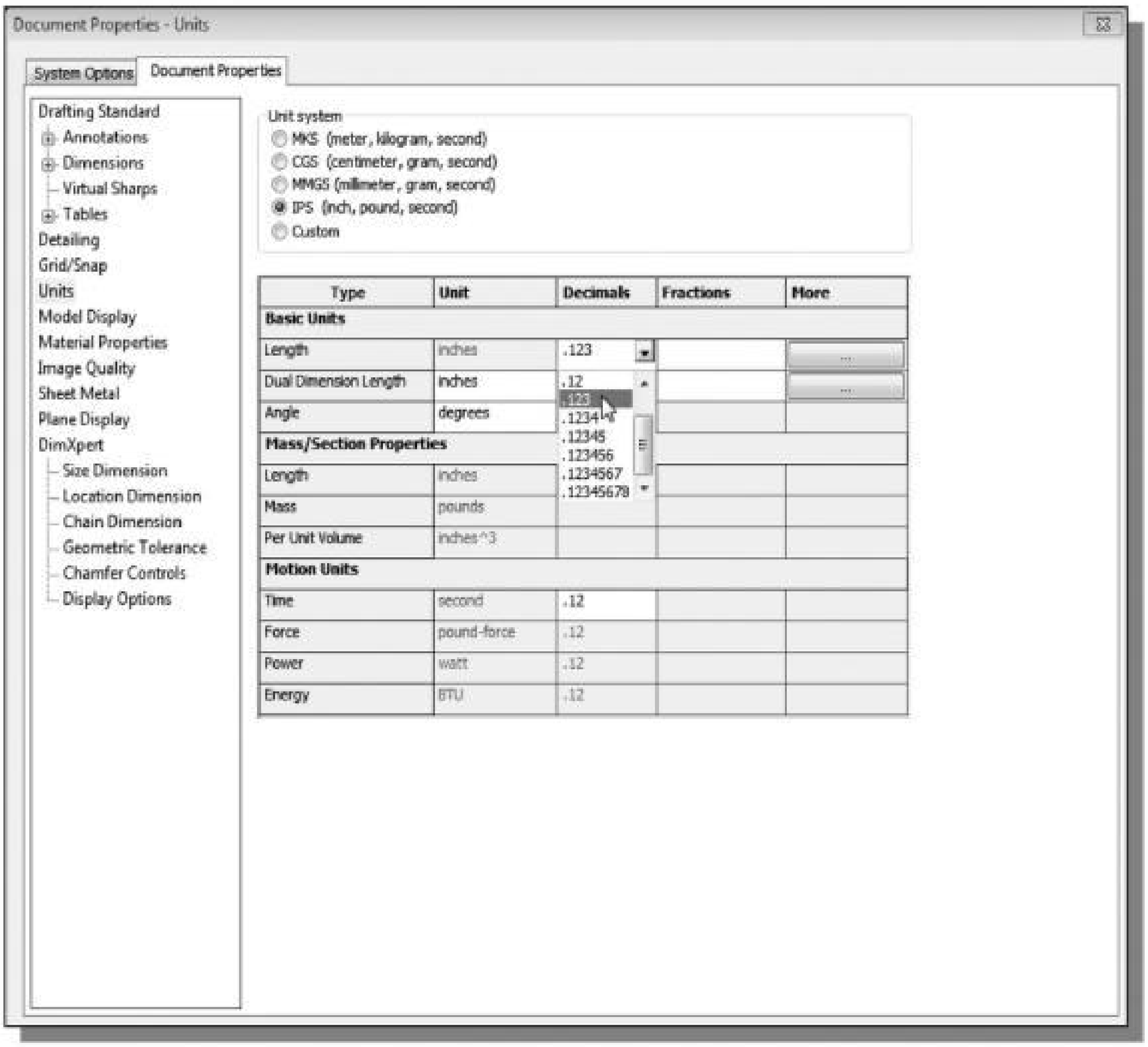
When starting a new CAD file, the first thing we should do is to choose the units we would like to use. We will use the English (feet and inches) setting for this example.

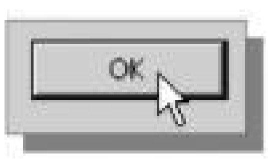
➢ Select the **Options** icon from the *Menu Bar* as shown to open the *Options* dialog box. (The *Options* dialog box can also be opened from the *Tools* pull-down menu.)



➢ When the *Options* dialog box opens, the **System Options** tab is active. The units setup is located under the Document Properties tab. Select the **Document Properties** tab as shown.

➢ Select **Units** on the left menu as highlighted below. Select **IPS (inch, pound, second)** under the *Unit system* options. Select **.123** in the *Decimals* spin box for the *Length* units as shown to define the degree of accuracy with which the units will be displayed.

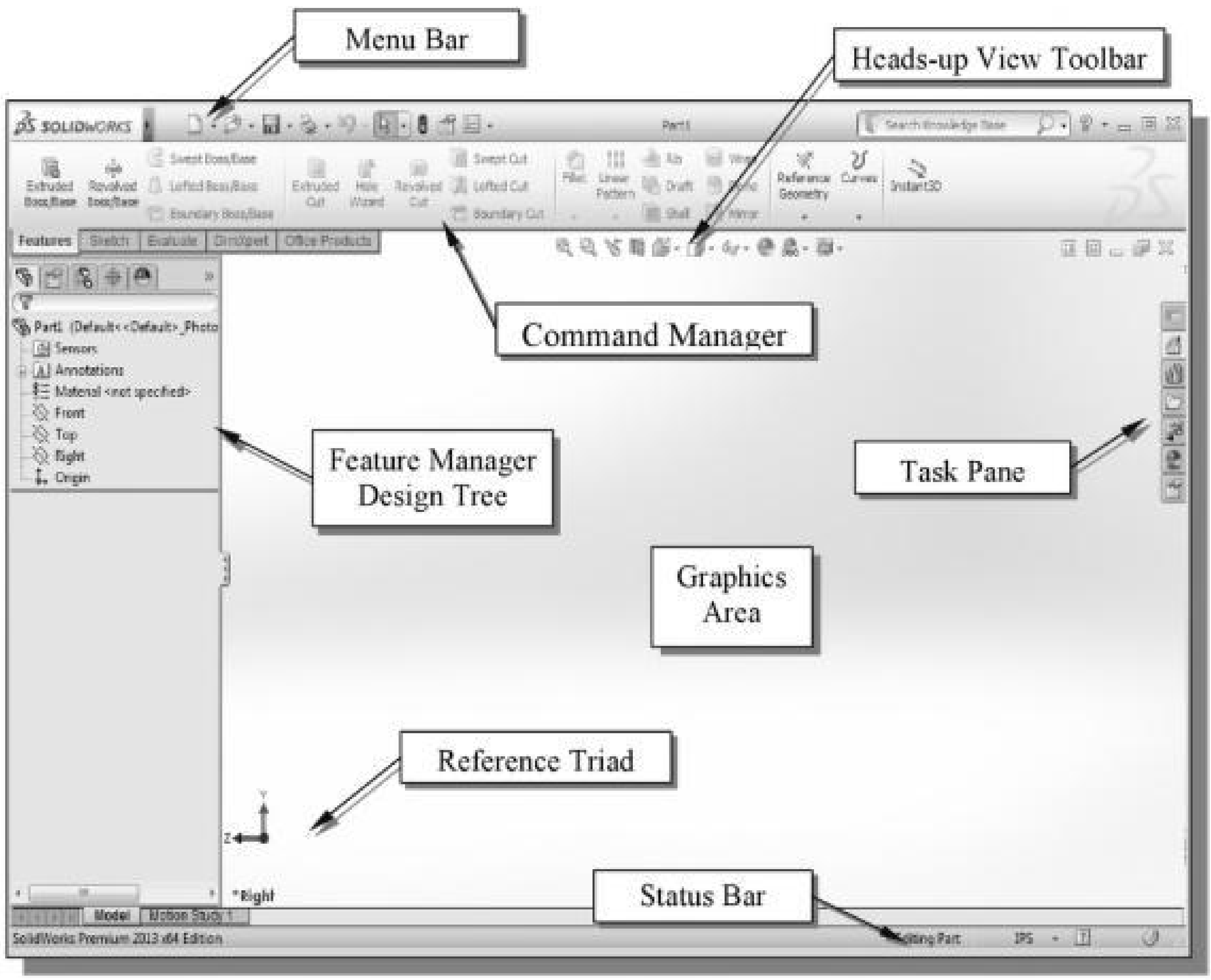




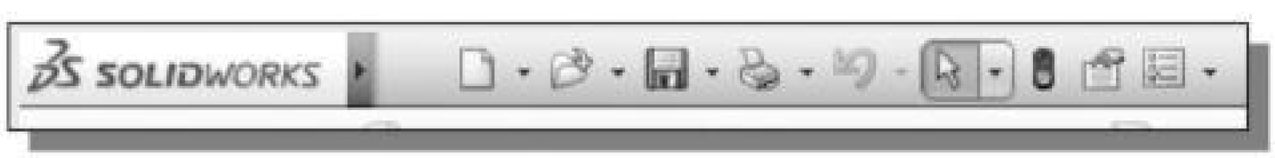
➢ Click **OK** at the bottom of the *Document Properties - Units* window to set the units.

SolidWorks Screen Layout

The default *SolidWorks* drawing screen contains the *Menu Bar*, the *Heads-up View* toolbar, the *Feature Manager Design Tree*, the *Features* toolbar (at the left of the window by default), the *Sketch* toolbar (at the right of the window by default), the graphics area, the *task pane* (collapsed to the right of the graphics area in the figure below), and the *Status Bar*. A line of quick text appears next to the icon as you move the *mouse cursor* over different icons. You may resize the *SolidWorks* drawing window by click and drag on the edge of the window, or relocate the window by click and drag on the *window title* area.



- **Menu Bar**



In the default view of the *Menu Bar*, only the toolbar options are visible. The default *Menu Bar* consists of a subset of frequently used commands from the *Menu Bar* toolbar as shown above.

- **Menu Bar Pull-down Menus**

To display the *pull-down* menus, move the cursor over or click the *SolidWorks* logo. The pull-down menus contain operations that you can use for all modes of the system.

- **Heads-up View Toolbar**

The *Heads-up View* toolbar allows us quick access to frequently used view-related commands, such as Zoom, Pan and Rotate. Note: You cannot hide or customize the *Heads-up View* toolbar.

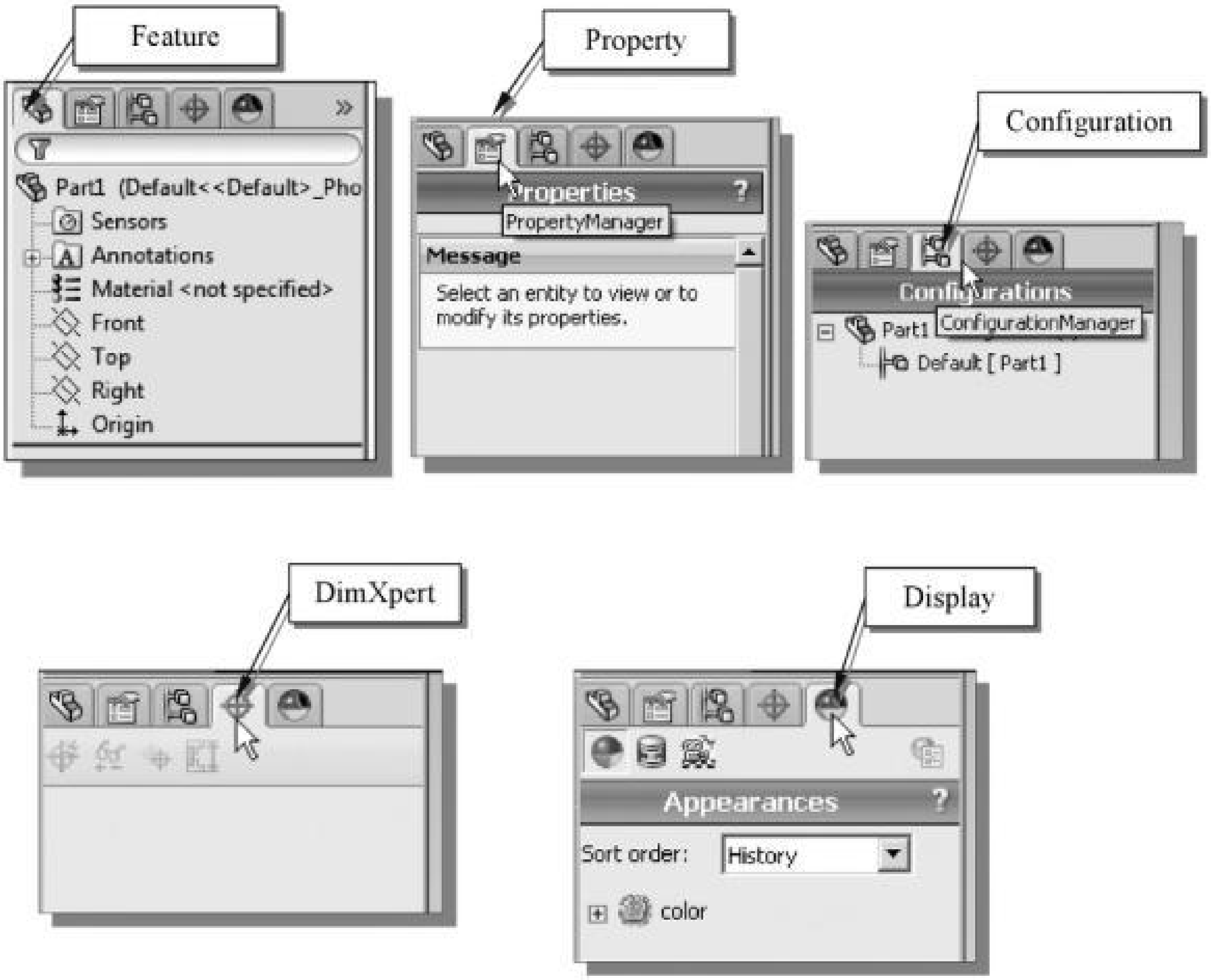
- **Features Toolbar**

By default, the *Features* toolbar is displayed vertically at the left of the *SolidWorks* window. The *Features* toolbar allows us quick access to frequently used feature-related commands, such as Extruded Boss/Base, Extruded Cut, and Revolved Boss/Base.

- **Sketch Toolbar**

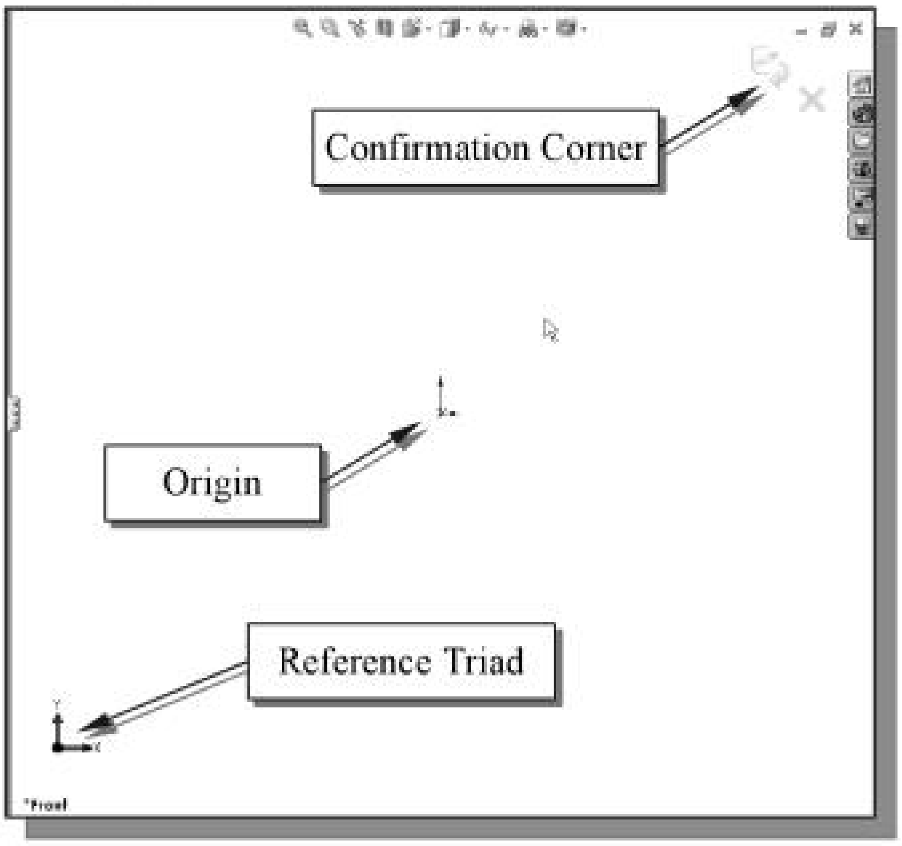
By default, the *Sketch* toolbar is displayed vertically at the right of the *SolidWorks* window. The *Sketch* toolbar provides tools for creating the basic geometry that can be used to create features and parts.

- **Feature Manager-Design Tree/Property Manager/Configuration Manager/ DimXpert Manager/Display Manager**



The left panel of the *SolidWorks* window is used to display the *Feature Manager Design Tree*, the *Property Manager*, the *Configuration Manager*, and the *DimXpert Manager*. These options can be chosen by selecting the appropriate tab at the top of the panel. The *Feature Manager Design Tree* provides an overview of the active part, drawing, or assembly in outline form. It can be used to show and hide selected features, filter contents, and manage access to features and editing. The *Property Manager* opens automatically when commands are executed or entities are selected in the graphics window, and is used to make selections, enter values, and accept commands. The *Configuration Manager* is used to create, select and view multiple configurations of parts and assemblies. The *DimXpert Manager* lists the tolerance features defined using the *SolidWorks* '*DimXpert for parts*' tools. The *Display Manager* lists the appearances, decals, lights, scene, and cameras applied to the current model. From the *Display Manager*, we can view applied content, and add, edit, or delete items. The *Display Manager* also provides access to *Photo View* options if the module is available.

- **Graphics Area**



The graphics area is the area where models and drawings are displayed.

- **Reference Triad**

The *Reference Triad* appears in the graphics area of part and assembly documents. The triad is shown to help orient the user when viewing models and is for reference only.

- **Origin**

The *Origin* represents the (0,0,0) coordinate in a model or sketch. A model origin appears blue; a sketch origin appears red.

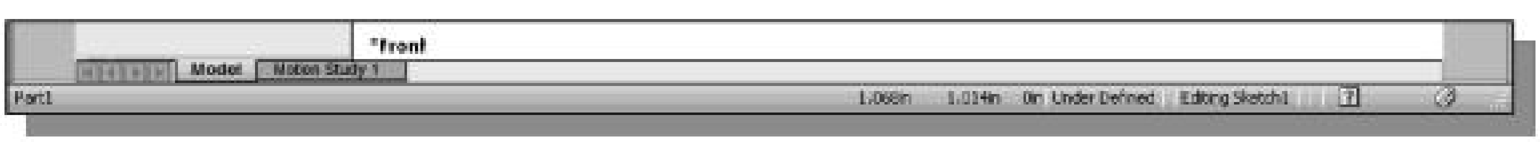
- **Confirmation Corner**

The *Confirmation Corner* offers an alternate way to accept features.

- **Graphics Cursor or Crosshairs**

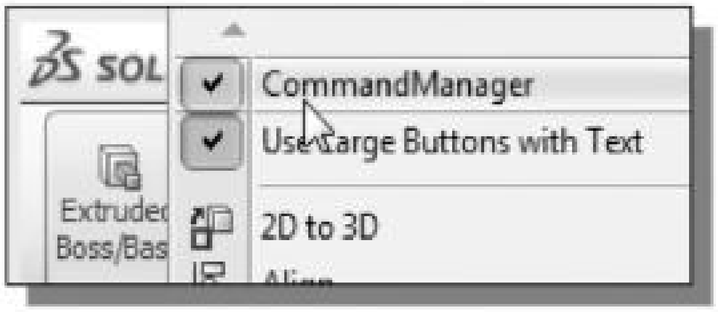
The *graphics cursor* shows the location of the pointing device in the graphics window. During geometric construction, the coordinate of the cursor is displayed in the *Status Bar* area, located at the bottom of the screen. The cursor's appearance depends on the selected command or option.

- **Message and Status Bar**

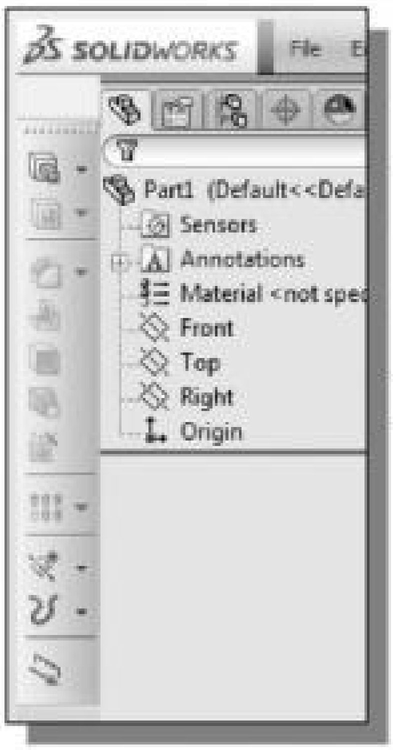


The *Message and Status Bar* area displays a single-line description of a command when the cursor is on top of a command icon. This area also displays information pertinent to the active operation. In the figure above, the cursor coordinates are displayed while in the *Sketch* mode.

Using the *SolidWorks* Command Manager

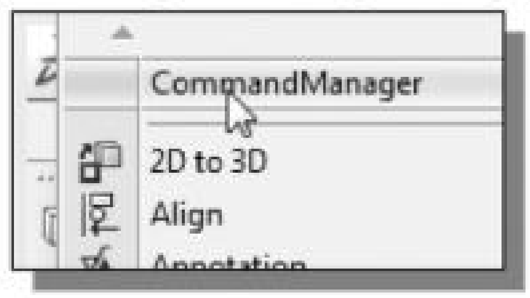


The *SolidWorks Command Manager* provides a convenient method for displaying the most commonly used toolbars. To turn on/off the *Command Manager*, **right click** on any toolbar icon, then select **Command Manager** from the top of the pop up menu list of toolbars.



You will notice that, when the *Command Manager* is turned off, the *Sketch* and *Features* toolbars appeared on the left and right edges of the display window.

To turn on the *Command Manager*, right click on the *Command Manager* (or any other) toolbar and toggle the *Command Manager* on by selecting it at the top of the pop-up menu.



The *Command Manager* is a context-sensitive toolbar that dynamically updates based on the user's selection. When you click a tab below the *Command Manager*, it updates to display the corresponding toolbar. For example, if you click the Sketches tab, the *Sketch* toolbar appears. By default, the *Command Manager* has toolbars embedded in it based on the document type.

The default display of the *Command Manager* is shown below. You will notice that, when the *Command Manager* is used, the *Sketch* and *Features* toolbars no longer appear on the left and right edges of the display window.

Important Note: The illustrations in this text use the *Command Manager*. If a user prefers to use the standard display of toolbars, the only change is that it may be necessary to activate the appropriate toolbar prior to selecting a command. For example, if the instruction is to "select *Extruded Boss* on the *Command Manager*" it may be necessary to first display the *Features* toolbar through the Customize Toolbar command.

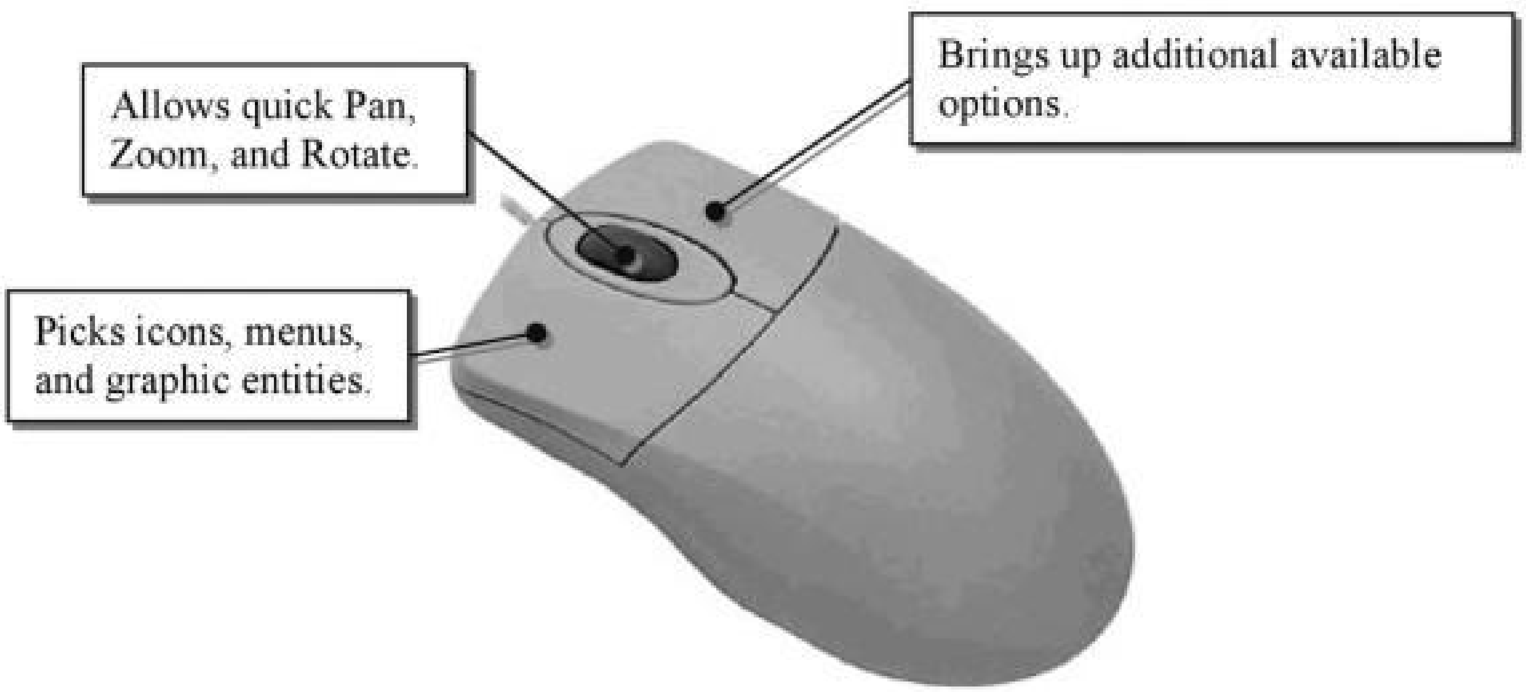
Mouse Buttons

SolidWorks utilizes the mouse buttons extensively. In learning *SolidWorks'* interactive environment, it is important to understand the basic functions of the mouse buttons.

- **Left mouse button**
  The **left-mouse-button** is used for most operations, such as selecting menus and icons, or picking graphic entities. One click of the button is used to select icons, menus and form entries, and to pick graphic items.

- **Right mouse button**
  The **right-mouse-button** is used to bring up additionally available options in a context-sensitive pop-up menu. These menus provide shortcuts to frequently used commands.

- **Middle mouse button/wheel**
  The middle mouse button/wheel can be used to Rotate (hold down the wheel button and drag the mouse), Pan (hold down the wheel button and drag the mouse while holding down the [**Ctrl**] key), or Zoom (hold down the wheel button and drag the mouse while holding down the [**Shift**] key) realtime. Spinning the wheel allows zooming to the position of the cursor.

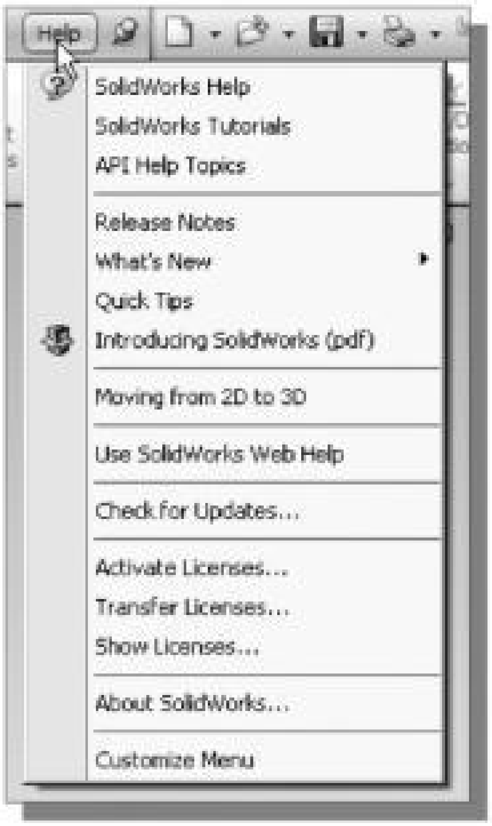


[Esc] – Canceling Commands

The [**Esc**] key is used to cancel a command in *SolidWorks*. The [**Esc**] key is located near the top left corner of the keyboard. Sometimes, it may be necessary to press the [**Esc**] key twice to cancel a command; it depends on where we are in the command sequence. For some commands, the [**Esc**] key is used to exit the command.

On-Line Help

❖ Several types of on-line help are available at any time during a *SolidWorks* session. *SolidWorks* provides many on-line help functions, such as:



- The **Help** menu: Click on the **Help** option in the *Menu Bar* to access the *SolidWorks Help* menu system. (Note: Move the cursor over the *SolidWorks* logo in the *Menu Bar* to display the pull-down menu options.) The **SolidWorks Help** option provides general help information, such as command options and command references. The **SolidWorks Tutorials** option provides a collection of tutorials illustrating different *SolidWorks* operations.

- The SolidWorks Tutorials can also be accessed from the *SolidWorks Resources* task pane on the right side of the screen by selecting **Tutorials**.

- The **SolidWorks Help** option can also be accessed by clicking on the **Help** icon at the right end of the *Menu Bar*.

Leaving *SolidWorks*

➢ To leave *SolidWorks*, use the left-mouse-button and click on **File** at the top of the *SolidWorks* screen window, then choose **Exit** from the pull-down menu. (Note: Move the cursor over the *SolidWorks* logo in the *Menu Bar* to display the pull-down menu options.)

Creating a CAD Files Folder

It is a good practice to create a separate folder to store your CAD files. You should not save your CAD files in the same folder where the *SolidWorks* application is located. It is much easier to organize and back up your project files if they are in a separate folder. Making folders within this folder for different types of projects will help you organize your CAD files even further. When creating CAD files in *SolidWorks*, it is strongly recommended that you *save* your CAD files on the hard drive.

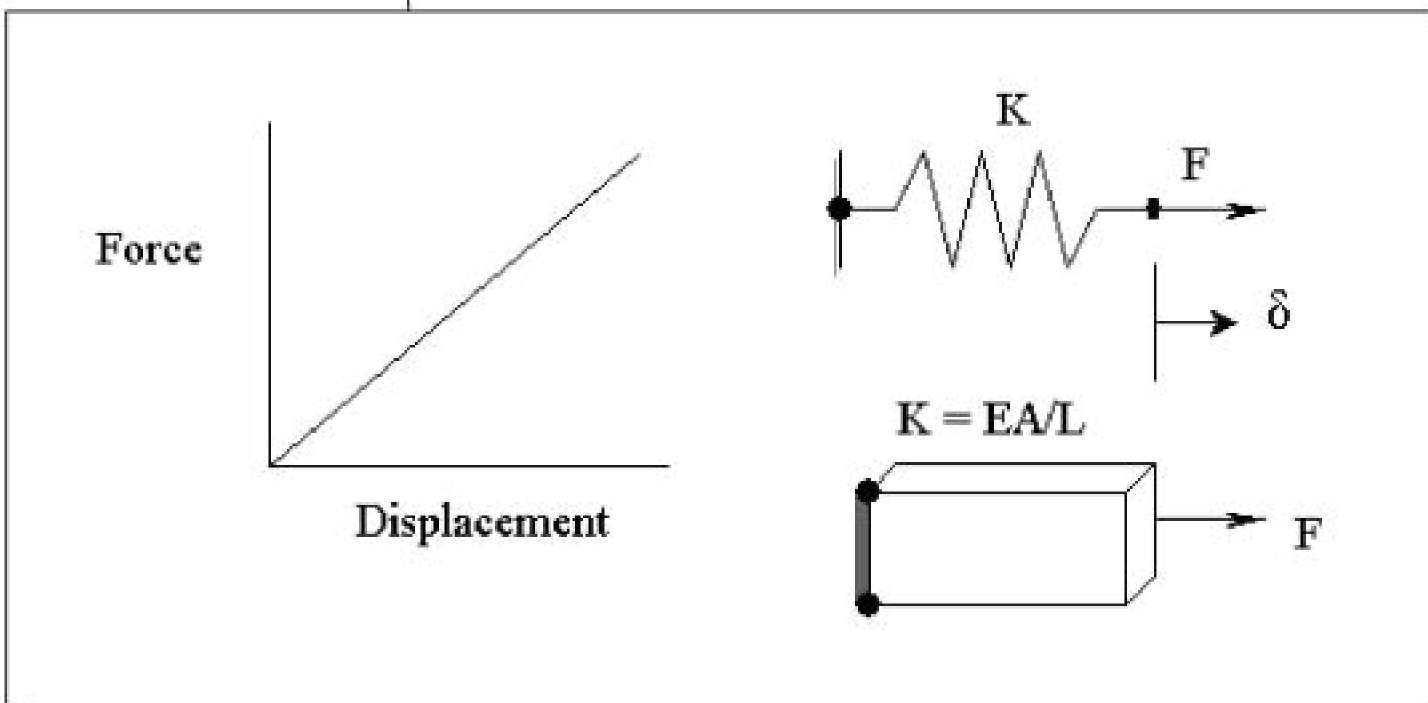
➢ To create a new folder in the *Windows* environment:

1. In *My Computer*, or start *Windows Explorer* under the *Start* menu, open the folder in which you want to create a new folder.

2. On the **File** menu, point to **New**, and then click **Folder**. The new folder appears with a temporary name.

3. Type a name, such as ***SolidWorks Projects***, for the new folder, and then press **[ENTER]**.

Chapter 1
The Direct Stiffness Method

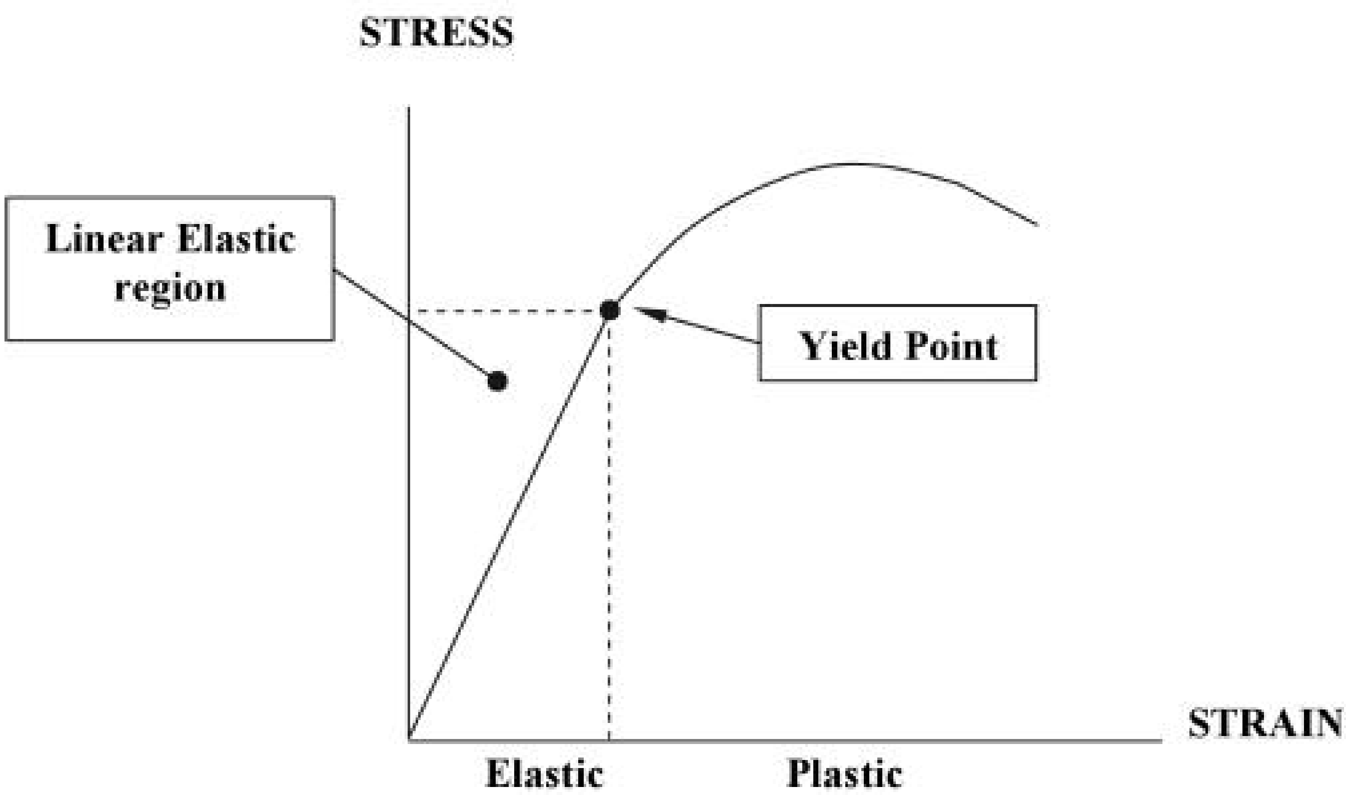


Learning Objectives

- Understand system equations for truss elements.
- Understand the setup of a Stiffness Matrix.
- Apply the Direct Stiffness Method.
- Create an Extruded solid model using SolidWorks.
- Use the SolidWorks 2D Sketch and Constraints tools.
- Use the Display Viewing commands.

Introduction

The **direct stiffness method** is used mostly for *Linear Static analysis*. The development of the direct stiffness method originated in the 1940s and is generally considered the fundamental of finite element analysis. Linear Static analysis is appropriate if deflections are small and vary only slowly. Linear Static analysis omits time as a variable. It also excludes plastic action and deflections that change the way loads are applied. The direct stiffness method for Linear Static analysis follows the *laws of Statics* and the *laws of Strength of Materials*.



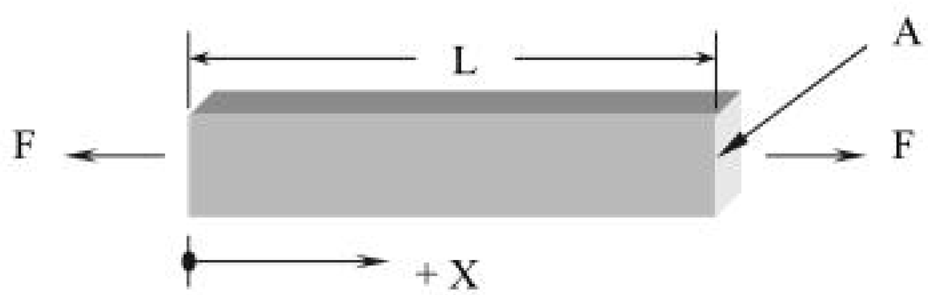
Stress-Strain diagram of typical ductile material

This chapter introduces the fundamentals of finite element analysis by illustrating an analysis of a one-dimensional truss system using the direct stiffness method. The main objective of this chapter is to present the classical procedure common to the implementation of structural analysis. The direct stiffness method utilizes *matrices* and *matrix algebra* to organize and solve the governing system equations. Matrices, which are ordered arrays of numbers that are subjected to specific rules, can be used to assist the solution process in a compact and elegant manner. Of course, only a limited discussion of the direct stiffness method is given here, but we hope that the focused practical treatment will provide a strong basis for understanding the procedure to perform finite element analysis with *SolidWorks Simulation*.

The later sections of this chapter demonstrate the procedure to create a solid model using *SolidWorks*. The step-by-step tutorial introduces the *SolidWorks* user interface and serves as a preview to some of the basic modeling techniques demonstrated in the later chapters.

One-dimensional Truss Element

The simplest type of engineering structure is the truss structure. A truss member is a slender (the length is much larger than the cross section dimensions) **two-force** member. Members are joined by pins and only have the capability to support tensile or compressive loads axially along the length. Consider a uniform slender prismatic bar (shown below) of length L, cross-sectional area A, and elastic modulus E. The ends of the bar are identified as nodes. The nodes are the points of attachment to other elements. The nodes are also the points for which displacements are calculated. The truss element is a two-force member element; forces are applied to the nodes only, and the displacements of all nodes are confined to the axes of elements.



In this initial discussion of the truss element, we will consider the motion of the element to be restricted to the horizontal axis (one-dimensional). Forces are applied along the X axis and displacements of all nodes will be along the X axis.

For the analysis, we will establish the following sign conventions:

1. Forces and displacements are defined as positive when they are acting in the positive X direction as shown in the above figure.

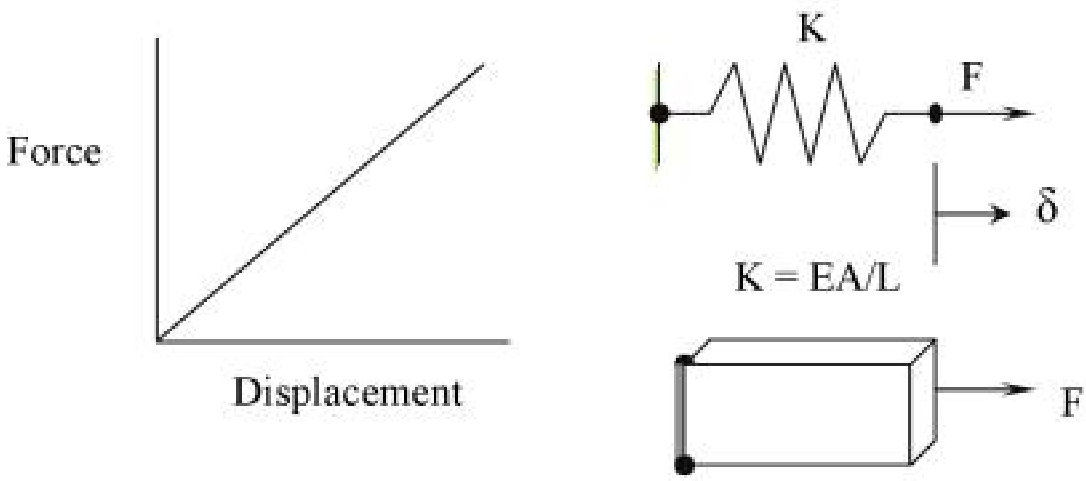
2. The position of a node in the undeformed condition is the finite element position for that node.

If equal and opposite forces of magnitude F are applied to the end nodes, from the elementary strength of materials, the member will undergo a change in length according to the equation:

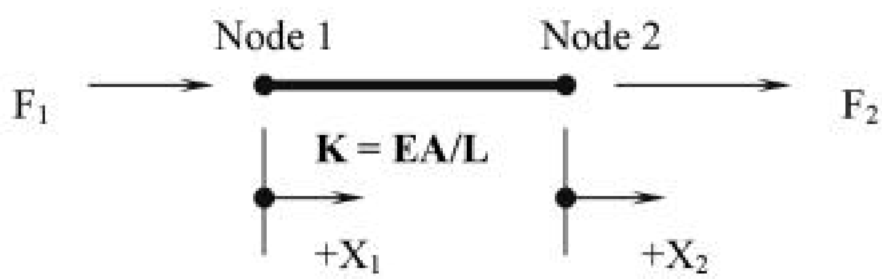
$$\delta = \frac{FL}{EA}$$

This equation can also be written as $\delta = F/K$, which is similar to *Hooke's Law* used in a linear spring. In a linear spring, the symbol K is called the **spring constant** or **stiffness** of the spring. For a truss element, we can see that an equivalent spring element can be used to simplify the representation of the model, where the spring constant is calculated as **K=EA/L**.

Force-Displacement Curve of a Linear Spring

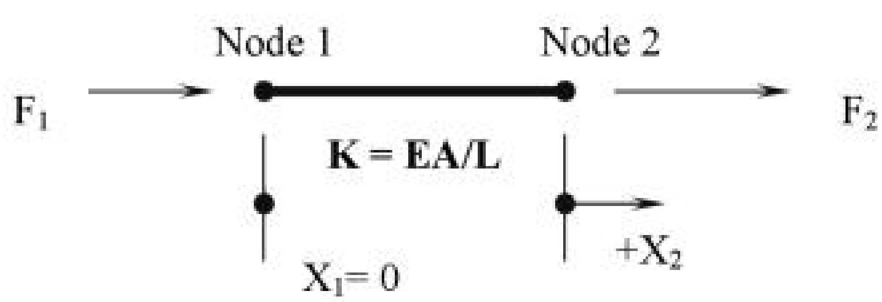


We will use the general equations of a single one-dimensional truss element to illustrate the formulation of the stiffness matrix method:



By using the *Relative Motion Analysis* method, we can derive the general expressions of the applied forces (F_1 and F_2) in terms of the displacements of the nodes (X_1 and X_2) and the stiffness constant (K).

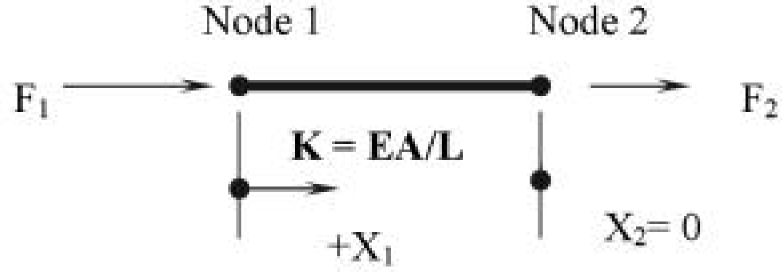
1. Let $X_1 = 0$,



Based on Hooke's law and equilibrium equation:

$$\begin{cases} F_2 = K\,X_2 \\ F_1 = -\,F_2 = -\,K\,X_2 \end{cases}$$

2. Let $X_2 = 0$,



Based on *Hooke's Law* and equilibrium:

$$\begin{cases} F_1 = K\,X_1 \\ F_2 = -\,F_1 = -\,K\,X_1 \end{cases}$$

Using the *Method of Superposition,* the two sets of equations can be combined:

$$\begin{aligned} F_1 &= \ \ K\,X_1 - K\,X_2 \\ F_2 &= -\,K\,X_1 + K\,X_2 \end{aligned}$$

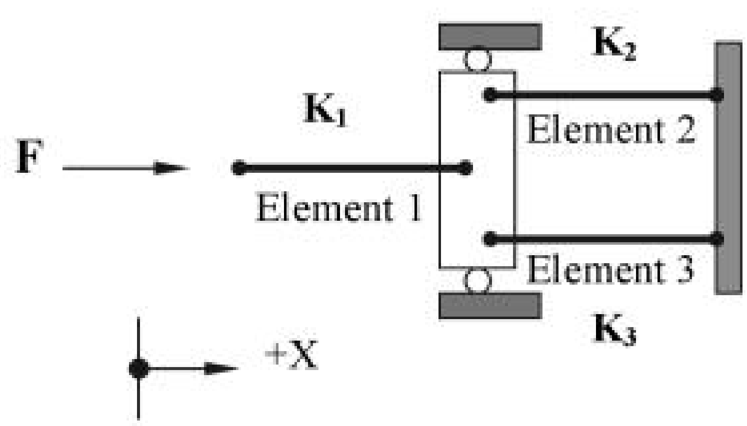
The two equations can be put into matrix form as follows:

$$\boxed{\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} +K & -K \\ -K & +K \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}}$$

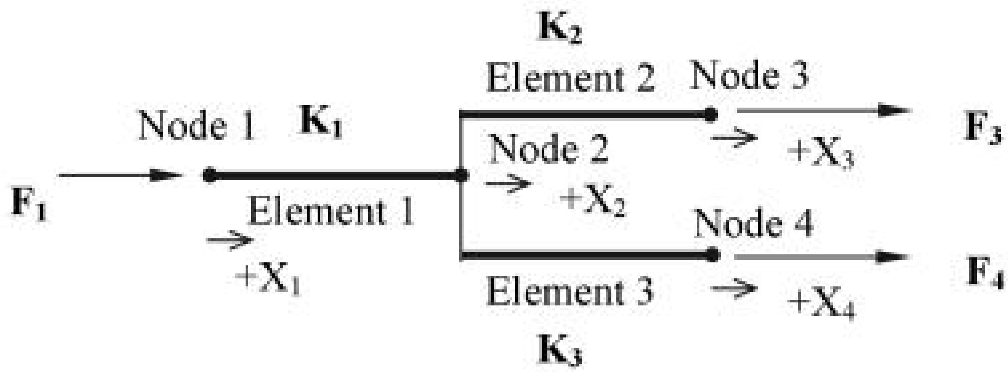
This is the general force-displacement relation for a two-force member element, and the equations can be applied to all members in an assemblage of elements. The following example illustrates a system with three elements.

Example 1.1:

Consider an assemblage of three of these two-force member elements. (Motion is restricted to one-dimension, along the X axis.)



The assemblage consists of three elements and four nodes. The *Free Body Diagram* of the system with node numbers and element numbers labeled:



Consider now the application of the general force-displacement relation equations to the assemblage of the elements.

Element 1:

$$\begin{Bmatrix} F_1 \\ F_{21} \end{Bmatrix} = \begin{bmatrix} +K_1 & -K_1 \\ -K_1 & +K_1 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

Element 2:

$$\begin{Bmatrix} F_{22} \\ F_3 \end{Bmatrix} = \begin{bmatrix} +K_2 & -K_2 \\ -K_2 & +K_2 \end{bmatrix} \begin{Bmatrix} X_2 \\ X_3 \end{Bmatrix}$$

Element 3:

$$\begin{Bmatrix} F_{23} \\ F_4 \end{Bmatrix} = \begin{bmatrix} +K_3 & -K_3 \\ -K_3 & +K_3 \end{bmatrix} \begin{Bmatrix} X_2 \\ X_4 \end{Bmatrix}$$

Expanding the general force-displacement relation equations into an *Overall Global Matrix* (containing all nodal displacements):

Element 1:

$$\begin{Bmatrix} F_1 \\ F_{21} \\ 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} +K_1 & -K_1 & 0 & 0 \\ -K_1 & +K_1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}$$

Element 2:

$$\begin{Bmatrix} 0 \\ F_{22} \\ F_3 \\ 0 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & +K_2 & -K_2 & 0 \\ 0 & -K_2 & +K_2 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}$$

Element 3:

$$\begin{Bmatrix} 0 \\ F_{23} \\ 0 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & +K_3 & 0 & -K_3 \\ 0 & 0 & 0 & 0 \\ 0 & -K_3 & 0 & +K_3 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}$$

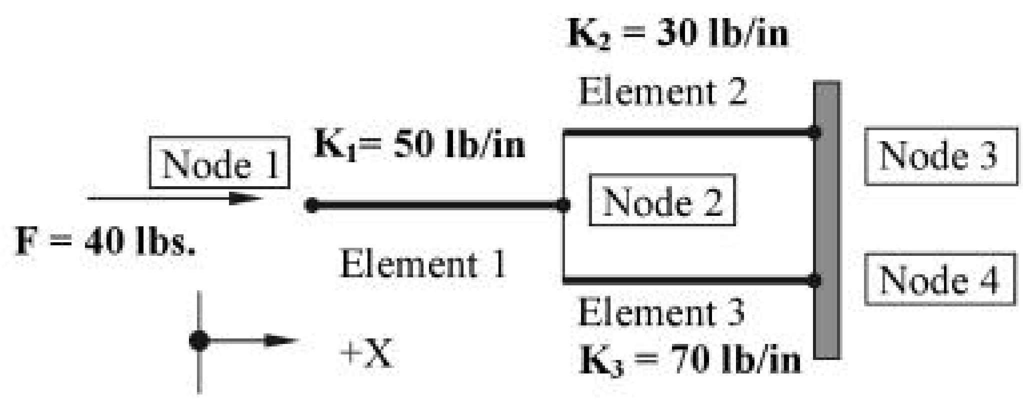
Summing the three sets of general equation: (Note $F_2 = F_{21} + F_{22} + F_{32}$)

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \underbrace{\begin{bmatrix} K_1 & -K_1 & 0 & 0 \\ -K_1 & (K_1+K_2+K_3) & -K_2 & -K_3 \\ 0 & -K_2 & K_2 & 0 \\ 0 & -K_3 & 0 & +K_3 \end{bmatrix}}_{\textbf{Overall Global Stiffness Matrix}} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}$$

Once the *Overall Global Stiffness Matrix* is developed for the structure, the next step is to substitute boundary conditions and solve for the unknown displacements. At every node in the structure, either the externally applied load or the nodal displacement is needed as a boundary condition. We will demonstrate this procedure with the following example.

Example 1.2:

Given:



Find: Nodal displacements and reaction forces.

Solution:

From Example 1.1, the overall global force-displacement equation set:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & (50+30+70) & -30 & -70 \\ 0 & -30 & 30 & 0 \\ 0 & -70 & 0 & 70 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}$$

Next, apply the known boundary conditions to the system: the right-ends of element 2 and element 3 are attached to the vertical wall; therefore, these two nodal displacements (X_3 and X_4) are zero.

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & (50+30+70) & -30 & -70 \\ 0 & -30 & 30 & 0 \\ 0 & -70 & 0 & 70 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ 0 \\ 0 \end{Bmatrix}$$

The two displacements we need to solve the system are X_1 and X_1. Remove any unnecessary columns in the matrix:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 50 & -50 \\ -50 & 150 \\ 0 & -30 \\ 0 & -70 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

Next, include the applied loads into the equations. The external load at *Node 1* is 40 lbs. and there is no external load at *Node 1*.

$$\begin{Bmatrix} 40 \\ 0 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 50 & -50 \\ -50 & 150 \\ 0 & -30 \\ 0 & -70 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

The Matrix represents the following four simultaneous system equations:

$$\begin{aligned} 40 &= 50\,X_1 - 50\,X_2 \\ 0 &= -\,50\,X_1 + 150\,X_2 \\ F_3 &= 0\,X_1 - 30\,X_2 \\ F_4 &= 0\,X_1 - 70\,X_2 \end{aligned}$$

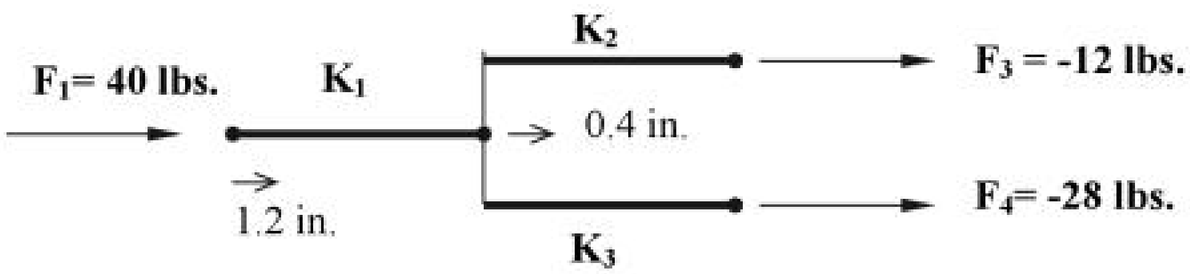
From the first two equations, we can solve for X_1 and X_2:

$$X_1 = 1.2 \text{ in.}$$
$$X_2 = 0.4 \text{ in.}$$

Substituting these known values into the last two equations, we can now solve for F_3 and F_4:

$$F_3 = 0\,X_1 - 30\,X_2 = -30 \times 0.4 = 12 \text{ lbs.}$$
$$F_4 = 0\,X_1 - 70\,X_2 = -70 \times 0.4 = 28 \text{ lbs.}$$

From the above analysis, we can now reconstruct the *Free Body Diagram* (*FBD*) of the system:



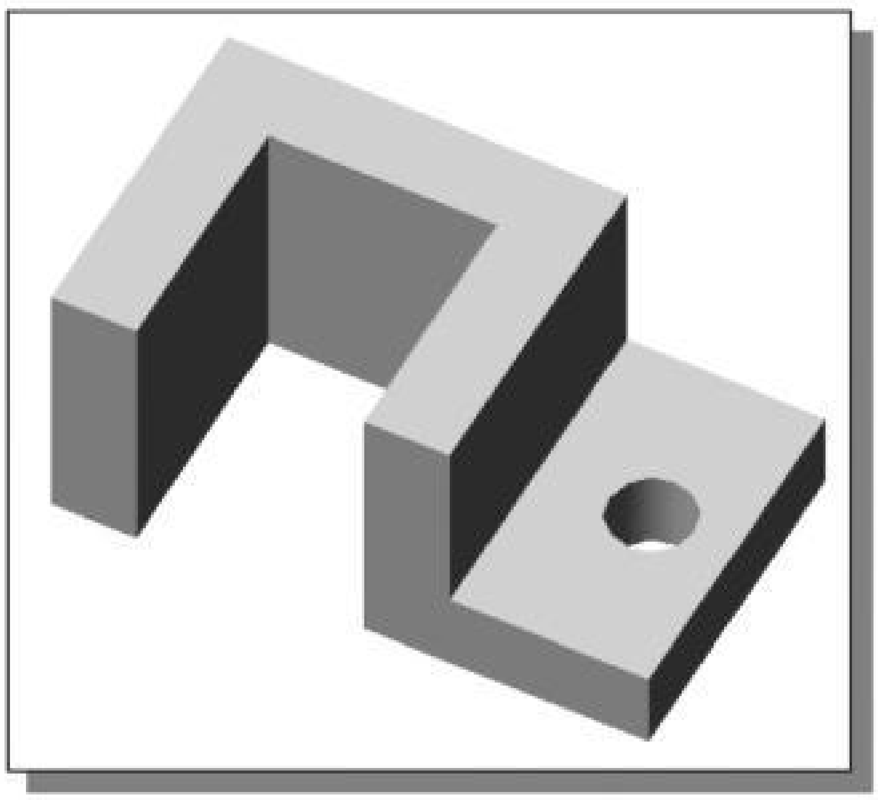
- The above sections illustrated the fundamental operation of the direct stiffness method, the classical finite element analysis procedure. As can be seen, the formulation of the global force-displacement relation equations is based on the general force-displacement equations of a single one-dimensional truss element. The two-force-member element (truss element) is the simplest type of element used in FEA. The procedure to formulate and solve the global force-displacement equations is straightforward, but somewhat tedious. In real-life applications, the use of a truss element in one-dimensional space is rare and very limited. In the next chapter, we will expand the procedure to solve two-dimensional truss frameworks.

  The following sections illustrate the procedure to create a solid model using *SolidWorks*. The step-by-step tutorial introduces the basic *SolidWorks's* user interface and the tutorial serves as a preview to some of the basic modeling techniques demonstrated in the later chapters.

Basic Solid Modeling using *SolidWorks*

One of the methods to create solid models in *SolidWorks* is to create a two-dimensional shape and then *extrude* the two dimensional shape to define a volume in the third dimension. This is an effective way to construct three-dimensional solid models since many designs are in fact the same shape in one direction. This method also conforms to the design process that helps the designer with conceptual design along with the capability to capture the design intent. *SolidWorks* provides many powerful modeling tools and there are many different approaches available to accomplish modeling tasks. We will start by introducing the basic two-dimensional sketching and parametric modeling tools.

The *Adjuster* Design

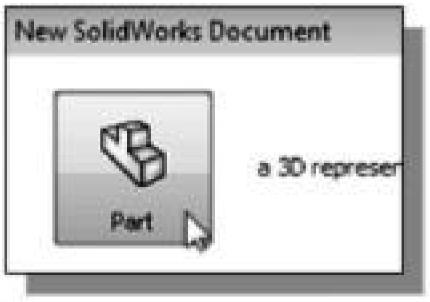


Starting *SolidWorks*

How to start *SolidWorks* depends on the type of workstation and the particular software configuration you are using. With most *Windows* and *Linux* systems, you may select **SolidWorks** on the *Start* menu or select the **SolidWorks** icon on the desktop. Consult your instructor or technical support personnel if you have difficulty starting the software.

1. Select the **SolidWorks** option on the *Start* menu or select the **SolidWorks** icon on the desktop to start *SolidWorks*. The *SolidWorks* main window will appear on the screen.

2. Click on the **New** icon, located in the *Standard* toolbar as shown.



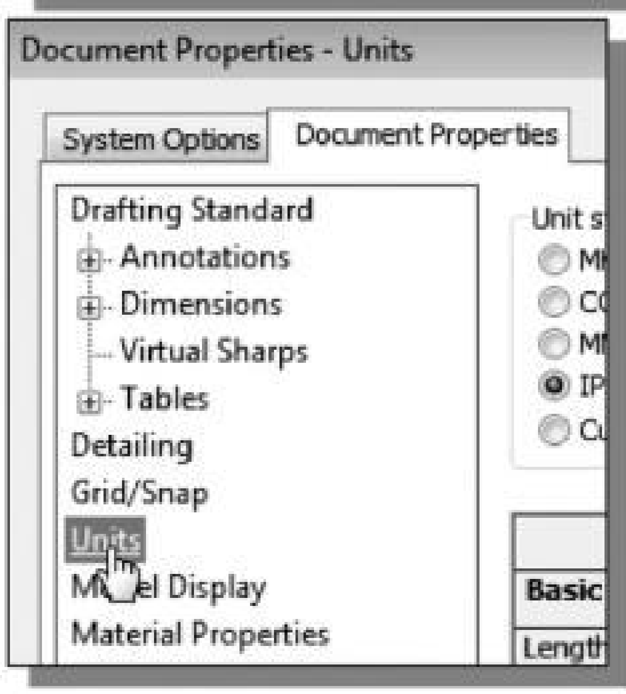
3. Select **Part** by clicking on the first icon in the *New SolidWorks Document* dialog box as shown.

4. Click on the **OK** button to accept the settings.

5. In the *Standard* toolbar area, **right-mouse-click** on any icon and select **Command Manager** in the option list as shown.

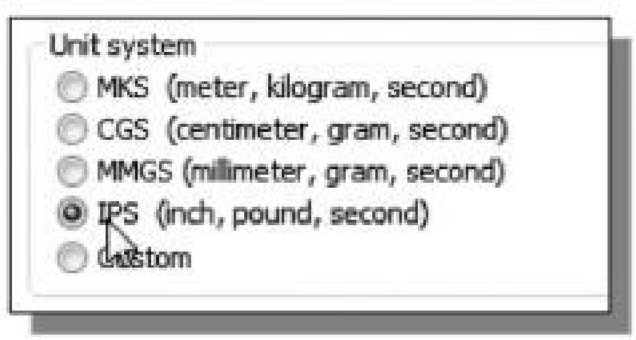
6. Select the **Options** icon from the *Menu* toolbar to open the *Options* dialog box.

7. Select the **Document Properties** tab as shown in the figure.



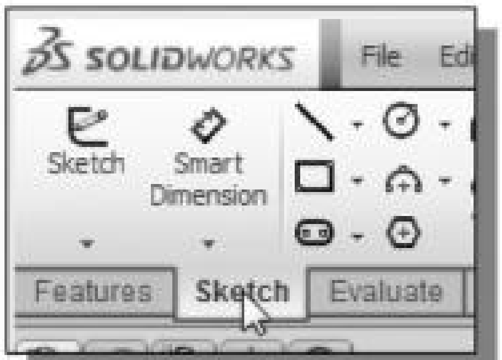
8. Click **Units** as shown in the figure.

9. Select **IPS (inch, pound, second)** under the *Unit system* options.

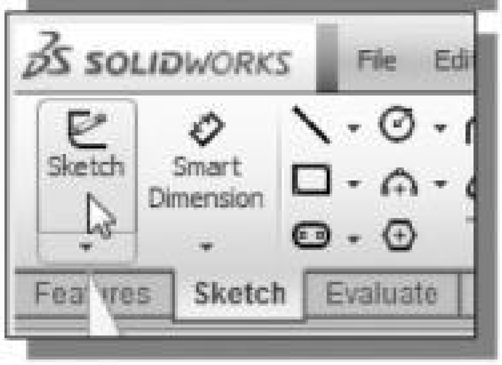


10. Click **OK** in the *Options* dialog box to accept the selected settings.

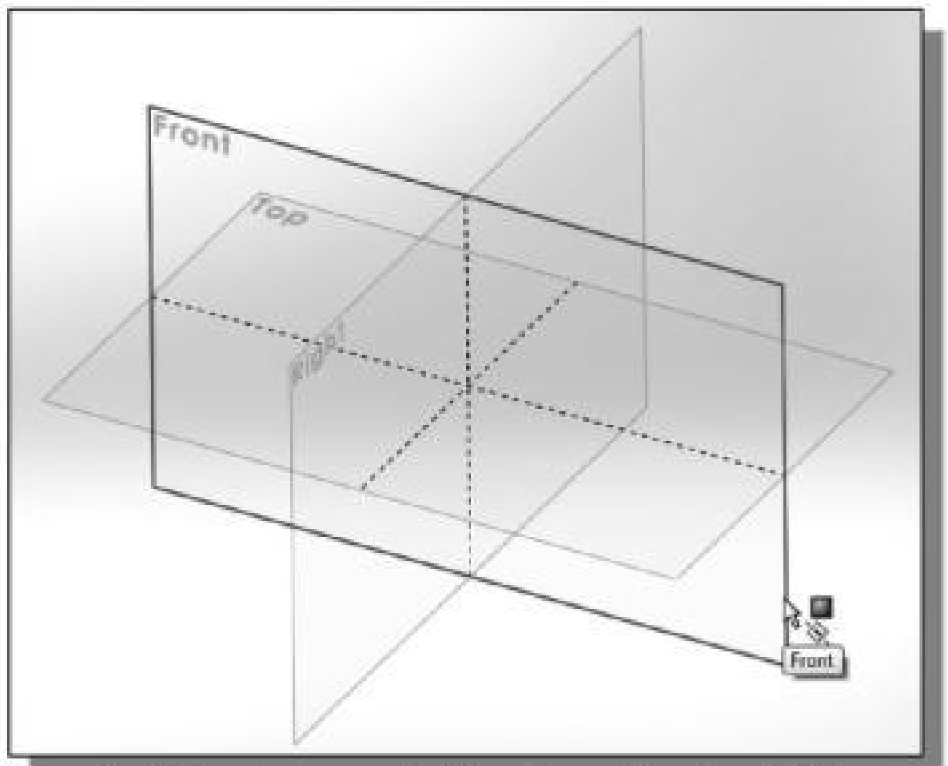
Step 1: Creating a Rough Sketch



1. Click on the **Sketch** tab in the *Command Manager* to switch to the *Sketch* toolbar.



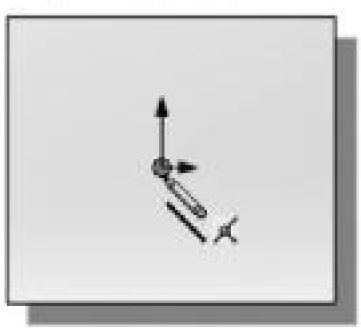
2. Select the **Sketch** button at the top of the *Sketch* toolbar to create a new sketch. Notice the left panel displays the *Edit Sketch Property Manager* with the instruction "*Select a plane on which to create a sketch for the entity.*"



3. Move the cursor over the edge of the **Front Plane** in the graphics area. When the **Front Plane** is highlighted, click once with the **left-mouse-button** to select the plane to align the sketching plane.

4. Select the **Line** icon on the *Sketch* toolbar by clicking once with the **left-mouse-button**; this will activate the Line command. The *Line Properties Property Manager* is displayed in the left panel.

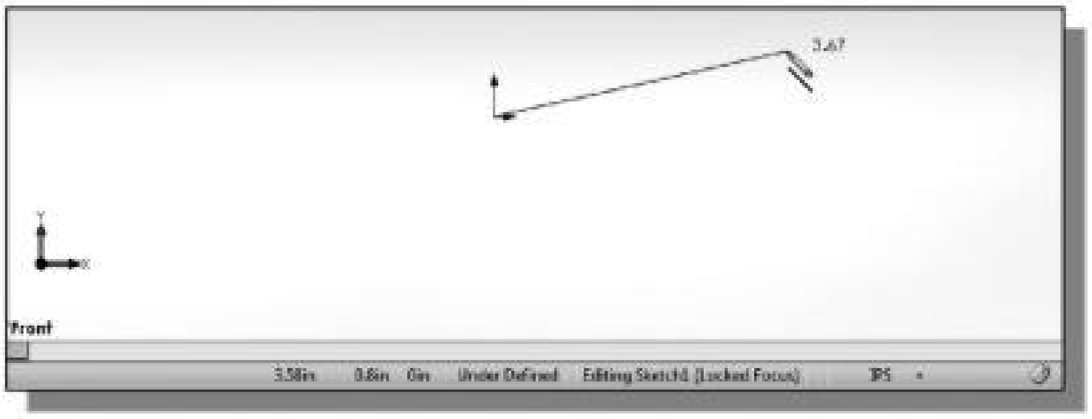
Graphics Cursors



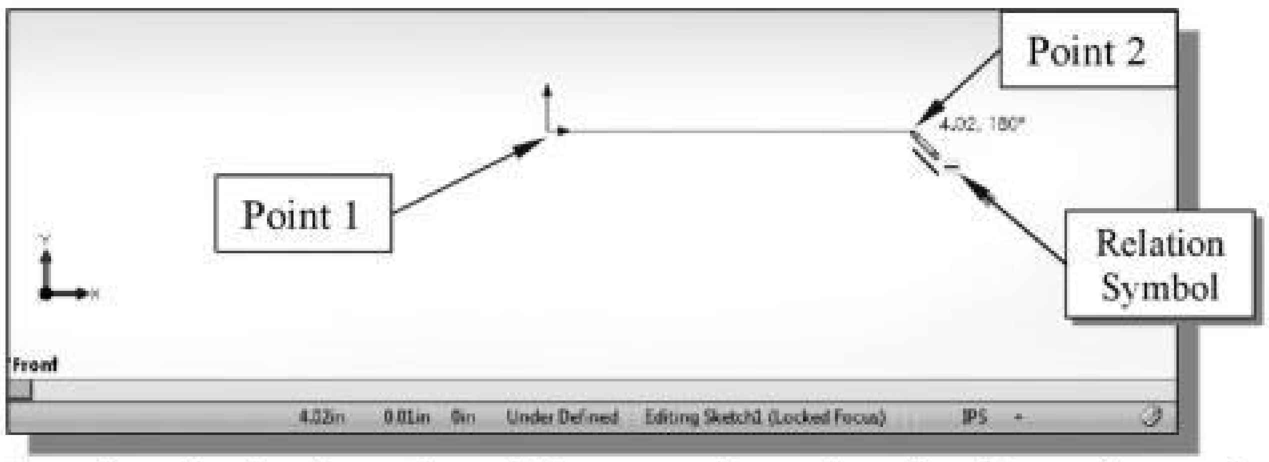
- Notice the cursor changes from an arrow to a pencil when a sketch entity is active.

1. Left-click on the **Origin** of the coordinate system to place the starting point of the line segments.

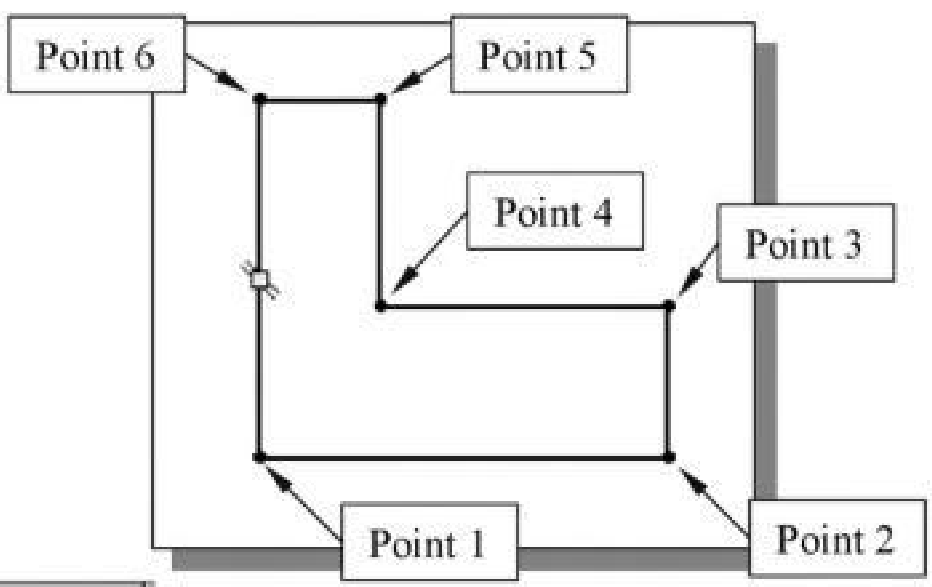
2. As you move the graphics cursor, you will see a digital readout next to the cursor. This readout gives you the line length. In the *Status Bar* area at the bottom of the window, the readout gives you the cursor location. Move the cursor around and you will notice different symbols appear at different locations.

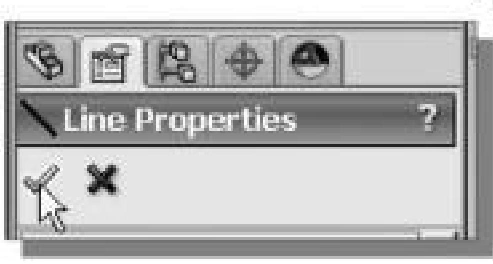


3. Move the graphics cursor toward the right side of the graphics window to create a horizontal line as shown below. Notice the geometric relation symbol displayed. When the **Horizontal** relation symbol is displayed, left-click to select **Point 2**.



4. Complete the sketch as shown below, creating a closed region ending at the starting point (**Point 1**). Do not be overly concerned with the actual size of the sketch. Note that all line segments are sketched horizontally or vertically.





5. Click the **OK** icon (green check mark) in the *Property Manager* to end editing of the current line, then click the **OK** icon again, or hit the [**Esc**] key once, to end the Sketch Line command.

Geometric Relation Symbols

SolidWorks displays different visual clues, or symbols, to show you alignments, perpendicularities, tangencies, etc. These relations are used to capture the *design intent* by creating relations where they are recognized. *SolidWorks* displays the governing geometric rules as models are built. To prevent relations from forming, hold down the [**Ctrl**] key while creating an individual sketch curve. For example, while sketching line segments with the Line command, endpoints are joined with a Coincident relation, but when the [**Ctrl**] key is pressed and held, the inferred relation will not be created.

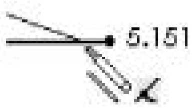
Vertical indicates a line is vertical

Horizontal indicates a line is horizontal

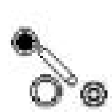
Dashed line indicates the alignment is to the center point or endpoint of an entity

Parallel indicates a line is parallel to other entities

Perpendicular indicates a line is perpendicular to other entities



Coincident indicates the endpoint will be coincident with another entity

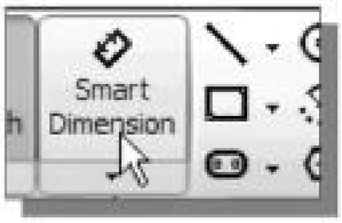


Concentric indicates the cursor is at the center of an entity

Tangent indicates the cursor is at tangency points to curves

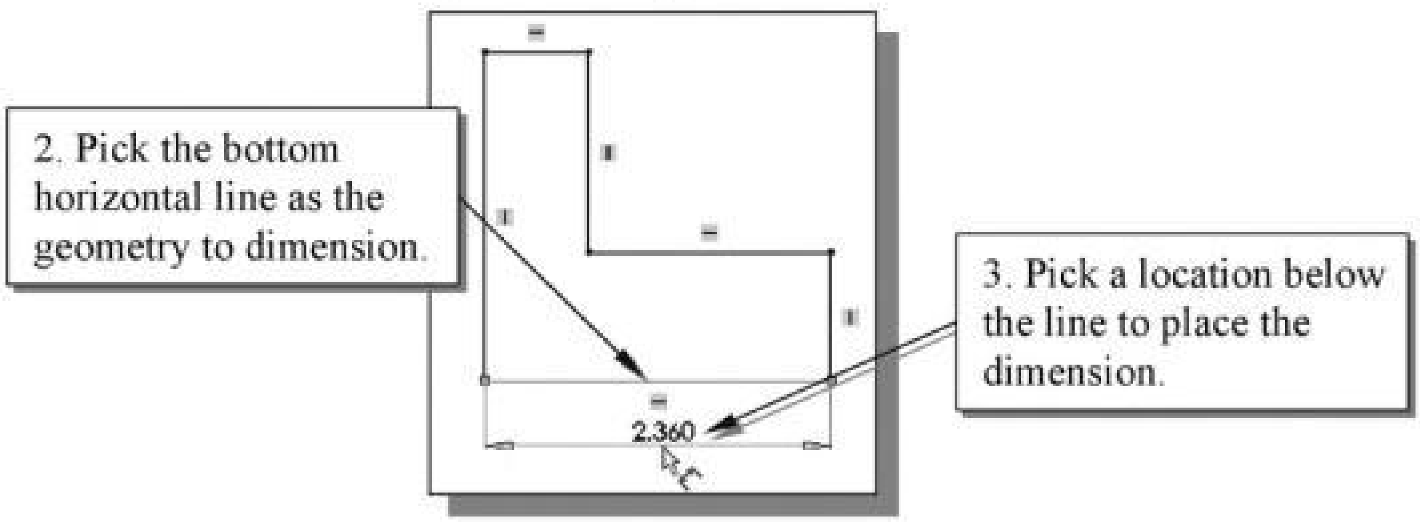
Step 2: Apply/Modify Relations and Dimensions

➢ As the sketch is made, *SolidWorks* automatically applies some of the geometric relations (such as Horizontal, Parallel and Perpendicular) to the sketched geometry. We can continue to modify the geometry, apply additional relations, and/or define the size of the existing geometry. In this example, we will illustrate adding dimensions to describe the sketched entities.

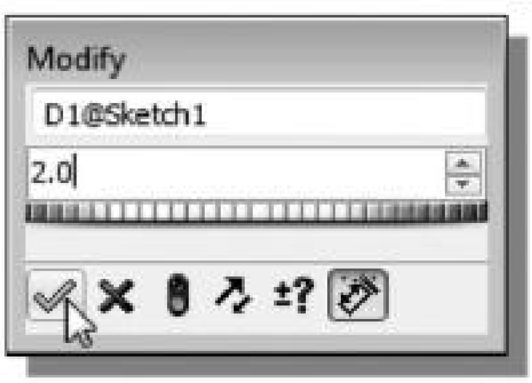


1. Move the cursor on top of the Smart Dimension icon on the *Sketch* toolbar. The Smart Dimension command allows us to quickly create and modify dimensions. Left-click once on the icon to activate the **Smart Dimension** command.

2. The message "*Select one or two edges/vertices and then a text location*" is displayed in the *Status Bar* area at the bottom of the *SolidWorks* window. Select the bottom horizontal line by left-clicking once on the line.



3. Move the graphics cursor below the selected line and left-click to place the dimension. (Note that the value displayed on your screen might be different than what is shown in the figure above.)



4. Enter ***2.0*** in the *Modify* dialog box.

5. Left click **OK** (green check mark) in the *Modify* dialog box to save the current value and exit the dialog.

6. Select the vertical line on the right.

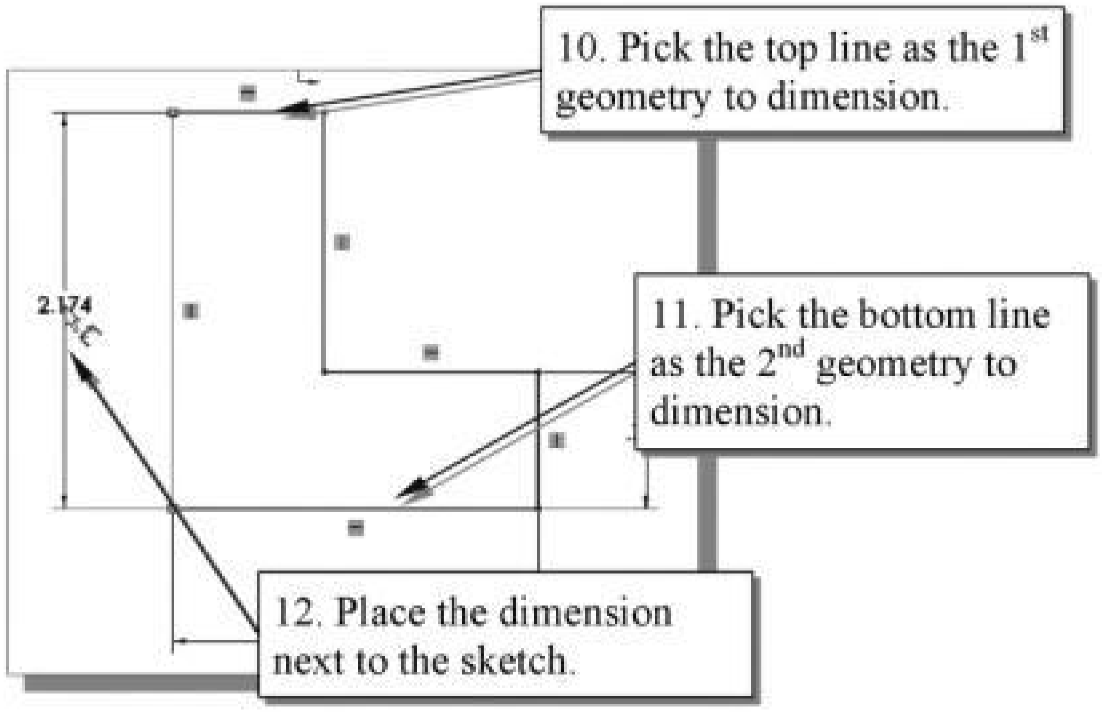
7. Pick a location toward the right of the sketch to place the dimension.

8. Enter ***0.75*** in the *Modify* dialog box and click **OK** in the *Modify* dialog box.

❖ The Smart Dimension command will create a length dimension if a single line is selected.

9. Select the top horizontal line as shown below.

10. Select the bottom horizontal line as shown below.



11. Pick a location to the left of the sketch to place the dimension.

12. Enter ***2.0*** in the *Modify* dialog box.

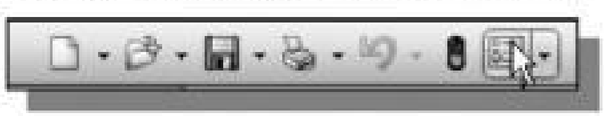
13. Click **OK** in the *Modify* dialog box.

❖ When two parallel lines are selected, the Smart Dimension command will create a dimension measuring the distance between them.

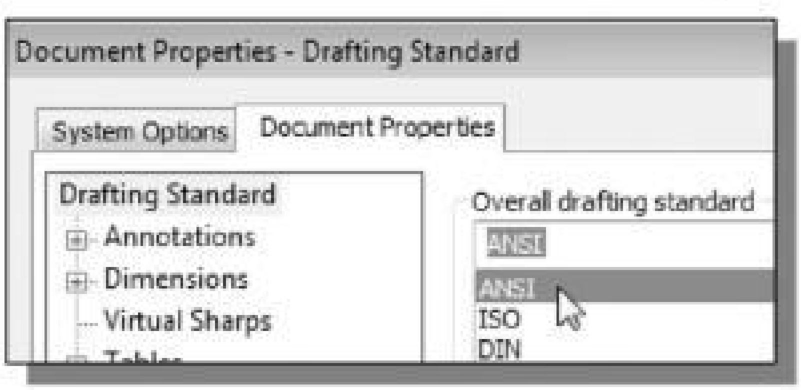
14. On you own, repeat the above steps and create an additional dimension for the top line. Make the dimension ***0.75***.

15. Click the **OK** icon in the *Property Manager* as shown, or hit the **[Esc]** key once, to end the Smart Dimension command.

Changing the Dimension Standard



1. Select the **Options** icon from the *Menu Bar* to open the *Options* dialog box.

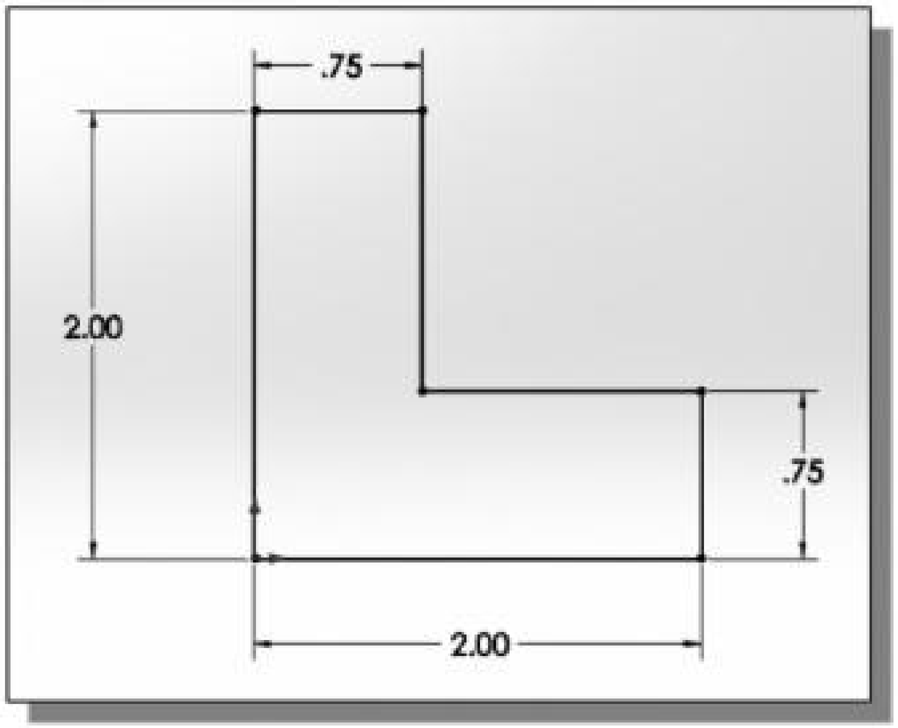


2. Select the **Document Properties** tab; also select **Drafting Standard** at the left.

3. Select **ANSI** in the pull-down selection list under the *Overall drafting standard* panel as shown.

4. Left-click **OK** in the *Options* dialog box to accept the settings.

- The sketch should now look as shown below. Notice the change in appearance of the dimensions.



Viewing Functions – *Zoom* and *Pan*

- *SolidWorks* provides a special user interface that enables convenient viewing of the entities in the graphics window. There are many ways to perform the **zoom** and **pan** operations.

1. Hold the [**Ctrl**] function key down. While holding the [**Ctrl**] function key down, press and drag with the mouse wheel to **pan** the display. This allows you to reposition the display while maintaining the same scale factor of the display.

2. Hold the [**Shift**] function key down. While holding the [**Shift**] function key down, **press and drag** with the mouse wheel to **zoom** the display. Moving downward will reduce the scale of the display, making the entities display smaller on the screen. Moving upward will magnify the scale of the display.

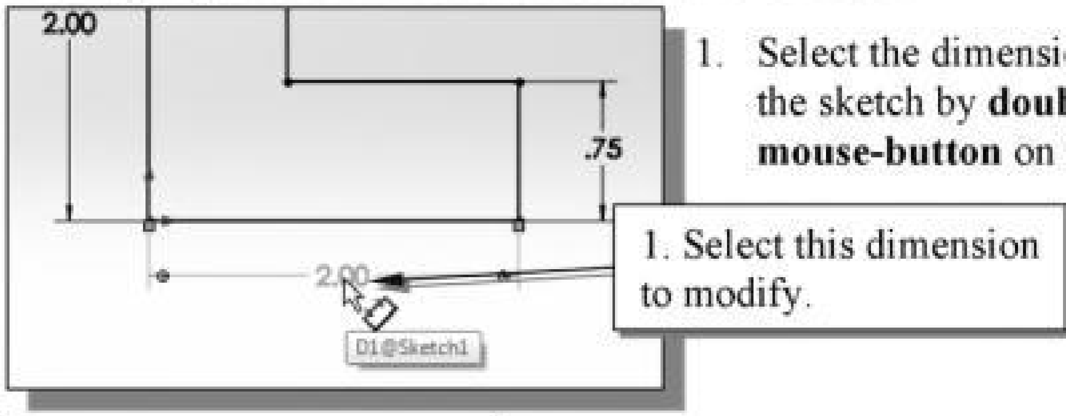
3. **Turning** the **mouse wheel** can also adjust the **scale** of the display. Turn the mouse wheel forward. Notice the scale of the display is reduced, making the entities display smaller on the screen.

4. Turn the **mouse wheel backward**. Notice scale of the display is magnified. (Note: Turning the mouse wheel allows zooming to the position of the cursor.)

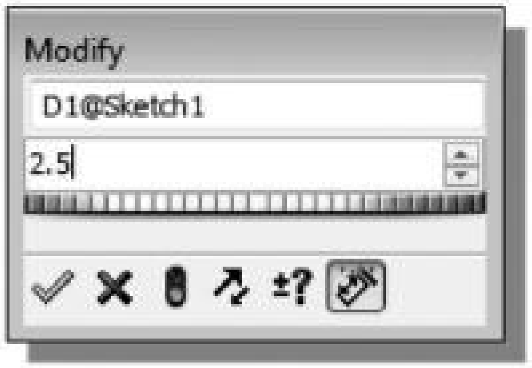
5. On your own, use the options above to change the scale and position of the display.

6. Press the **F** key on the keyboard to automatically **fit** the model to the screen.

Modifying the Dimensions of the Sketch

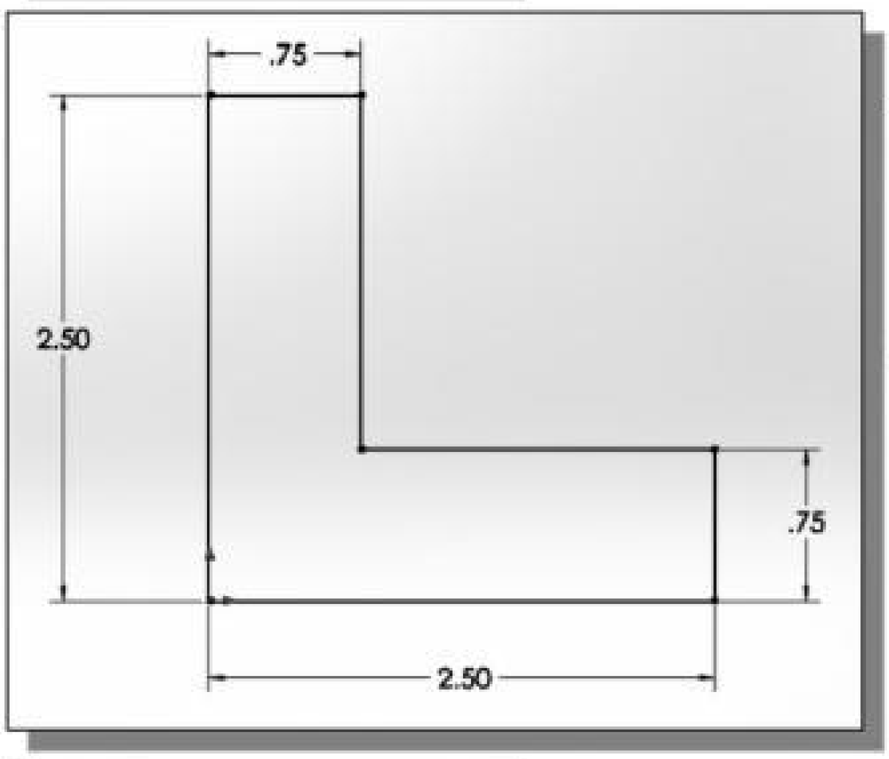


1. Select the dimension that is at the bottom of the sketch by **double-clicking** with the **left-mouse-button** on the dimension text.



2. In the *Modify* window, the current length of the line is displayed. Enter ***2.5*** to reset the length of the line.

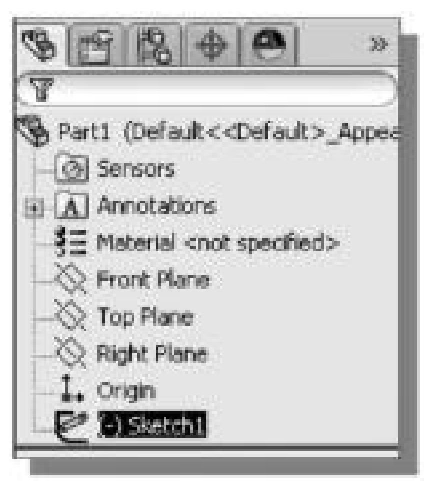
3. Click on the **OK** icon to accept the entered value.



4. On you own, repeat the above steps and adjust the left vertical dimension so that the sketch appears as shown.

5. Press the [**Esc**] key once to exit the Dimension command.

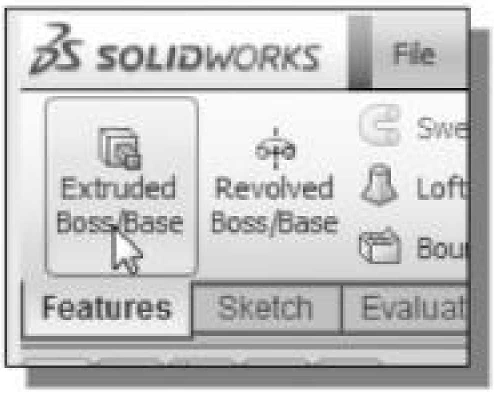
6. Click once with the **left-mouse-button** on the **Exit Sketch** icon on the *Sketch* toolbar to exit the sketch.



- Notice the newly created sketch is listed in the *Feature Manager Design Tree* as ***Sketch1***. Also notice that ***Sketch1*** is highlighted in the *Design Tree*, indicating that the sketch is currently 'selected'.

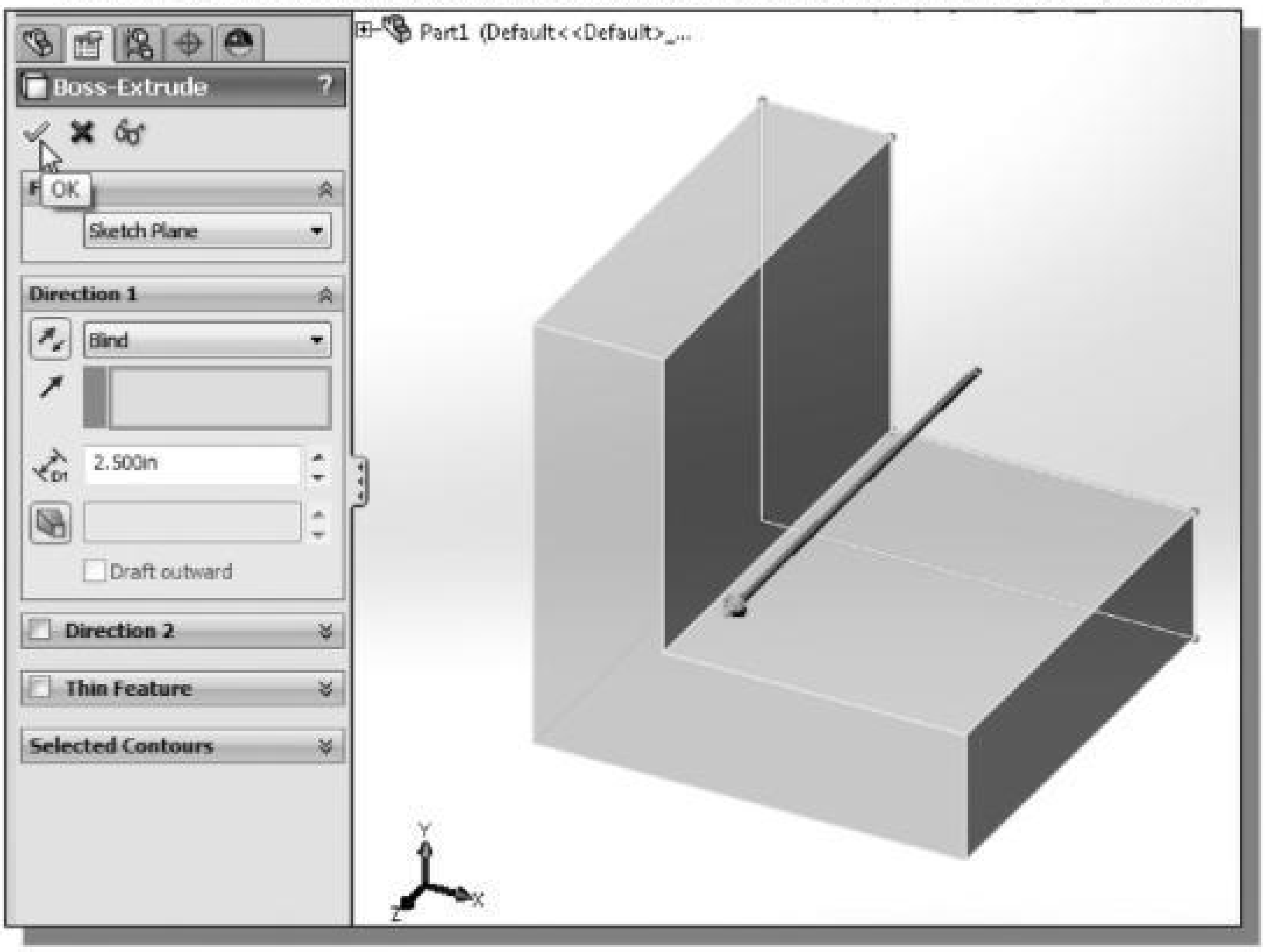
Step 3: Completing the Base Solid Feature

Now that the 2D sketch is complete, we will proceed to the next step: creating a 3D part from the 2D profile. Extruding a 2D profile is one of the common methods that can be used to create 3D parts. We can extrude planar faces along a path. We can also specify a height value and a tapered angle.



1. Switch to the **Features** toolbar (the first tab of the *Command Manager*), then select the **Extruded Boss/ Base** command by clicking once with the left-mouse-button on the icon. The *Extrude Property Manager* is displayed in the left panel.

2. In the *Boss-Extrude Property Manager* panel, enter ***2.5*** as the extrusion distance. Notice that the sketch region is automatically selected as the extrusion profile.



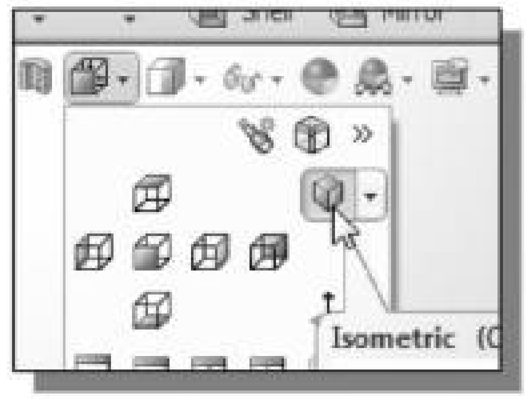
3. Click on the **OK** button to proceed with creating the 3D part.

➢ Note that all dimensions disappeared from the screen. All parametric definitions are stored in the ***SolidWorks* database** and any of the parametric definitions can be displayed and edited at any time.

Isometric View

❖ *SolidWorks* provides many ways to display views of the three-dimensional design. We will first orient the model to display in the *isometric view*, by using the *View Orientation* pull-down menu on the *Heads-up View* toolbar.

1. Select the **View Orientation** button on the *Heads-up View* toolbar by clicking once with the left-mouse-button.

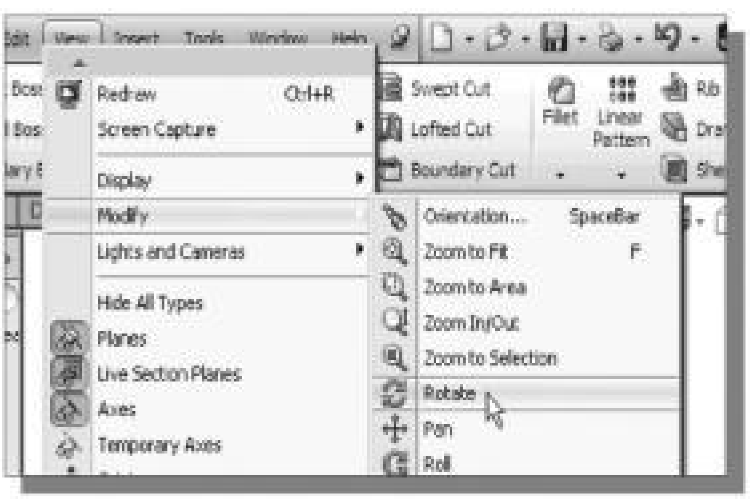


2. Select the **Isometric** icon in the *View Orientation* pull-down menu.

❖ Notice the other view-related commands that are available under the pull-down menu.

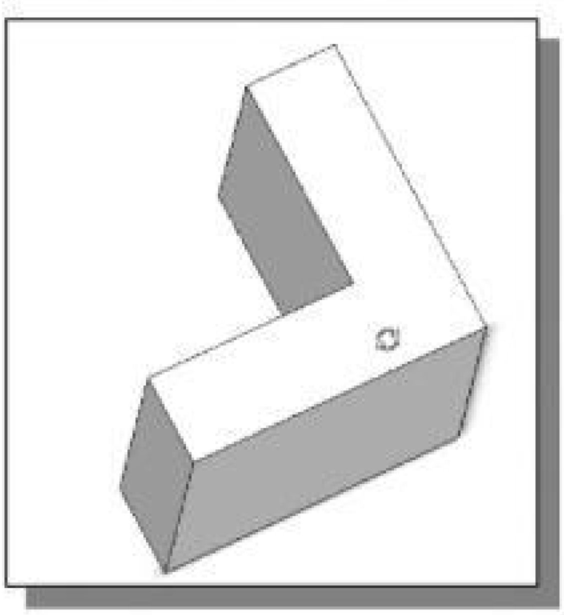
Rotation of the 3D Model – *Rotate View*

The Rotate View command allows us to rotate a part or assembly in the graphics window. Rotation can be around the center mark, free in all directions, or around a selected entity (vertex, edge or face) on the model.



1. Move the cursor over the SolidWorks logo to display the pull-down menus. Select **View → Modify → Rotate** from the pull-down menu as shown

2. Move the cursor inside the graphics area. Press down the left-mouse-button and drag in an arbitrary direction. The Rotate View command allows us to freely rotate the solid model.

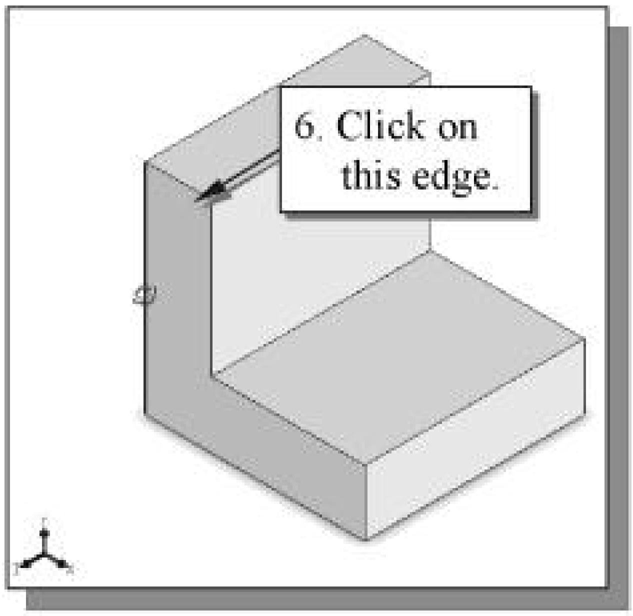


- The model will rotate about an axis normal to the direction of cursor movement. For example, drag the cursor horizontally across the screen and the model will rotate about a vertical axis.

3. Press the [**Esc**] key once to exit the Rotate View command.

4. Select the **Isometric** icon in the *View Orientation* pull-down menu (see steps 1 and 2 in the previous section) to reset the display to the isometric view.

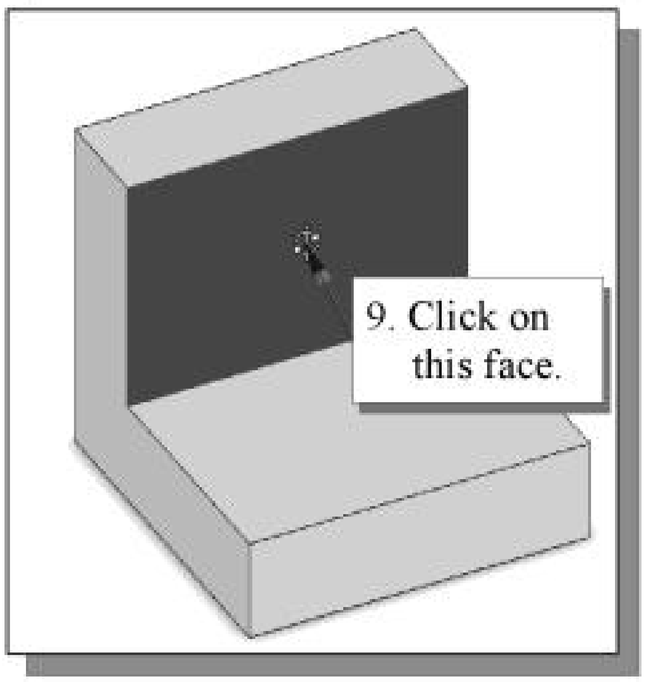
5. Execute the **Rotate View** option from the *View* pull-down menu (see step 1).



6. Move the cursor over the left edge of the solid model as shown. When the edge is highlighted, click the **left-mouse-button** once to select the edge.

7. Press down the left-mouse-button and drag. The model will rotate about this edge.

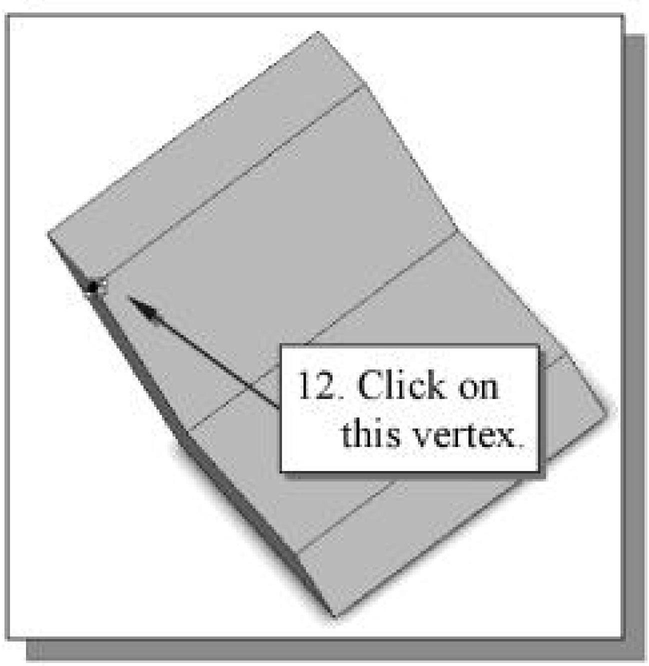
8. Left-click in the graphics area, outside the model, to deselect the edge.



9. Move the cursor over the upper front face of the solid model as shown. When the face is highlighted, click the **left-mouse-button** once to select the face.

10. Press down the left-mouse-button and drag. The model will rotate about the direction normal to this face.

11. Left-click in the graphics area, outside the model, to deselect the face.



12. Move the cursor over the upper front vertex as shown. When the vertex is highlighted, click the left-mouse-button once to select the vertex.

13. Press down the left-mouse-button and drag. The model will rotate about the vertex.

14. Left-click in the graphics area, outside the model, to deselect the vertex.

15. Press the [**Esc**] key once to exit the Rotate View command.

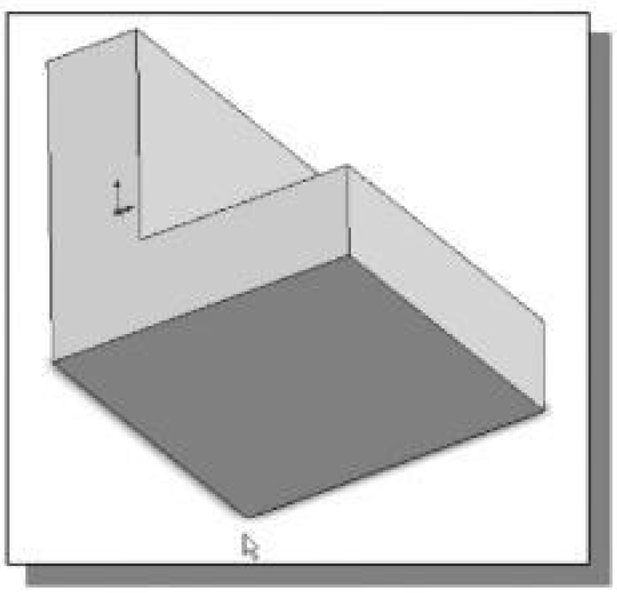
16. On your own, reset the display to the isometric view.

Rotation and Panning – *Arrow Keys*

SolidWorks allows us to easily rotate a part or assembly in the graphics window using the arrow keys on the keyboard.

- Use the arrow keys to rotate the view horizontally or vertically. The left-right arrow keys rotate the model about a vertical axis. The up-down keys rotate the model about a horizontal axis.

- Hold down the [**Alt**] key and use the left-right arrow keys to rotate the model about an axis normal to the screen, i.e., to rotate clockwise and counter-clockwise.



1. Hit the **left** arrow key. The model view rotates by a pre-determined increment. The default increment is 15°. (This increment can be set in the *Options* dialog box.) On your own, use the left-right and up-down arrow keys to rotate the view.

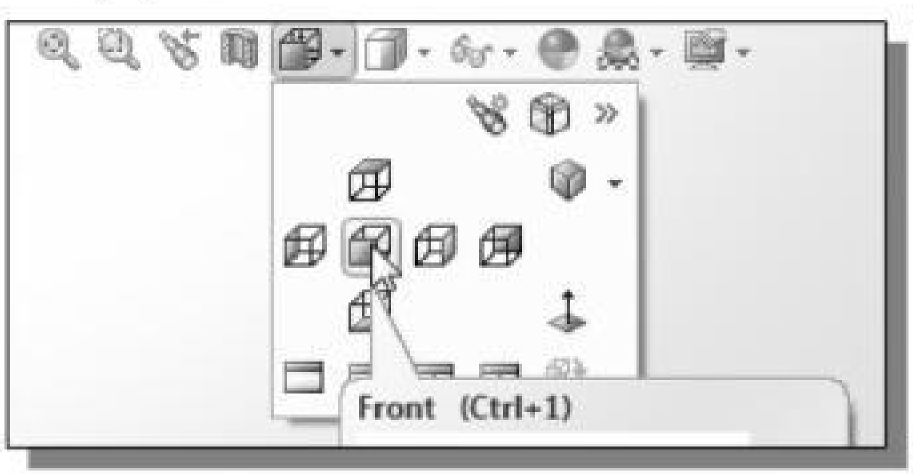
2. Hold down the [**Alt**] key and hit the **left** arrow key. The model view rotates in the clockwise direction. On your own use the left-right and up-down arrow keys, and the [**Alt**] key plus the left-right arrow keys, to rotate the view.

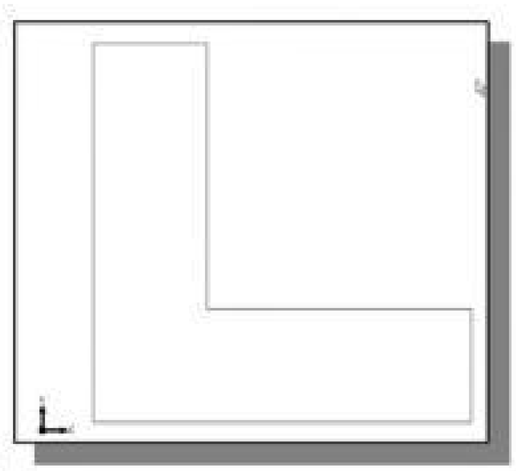
3. Reset the display to the isometric view.

- Hold down the [**Shift**] key and use the left-right and up-down arrow keys to rotate the model in 90° increments.

4. Hold down the [**Shift**] key and hit the right arrow key. The view will rotate by 90°. On your own use the [**Shift**] key plus the left-right arrow keys to rotate the view.

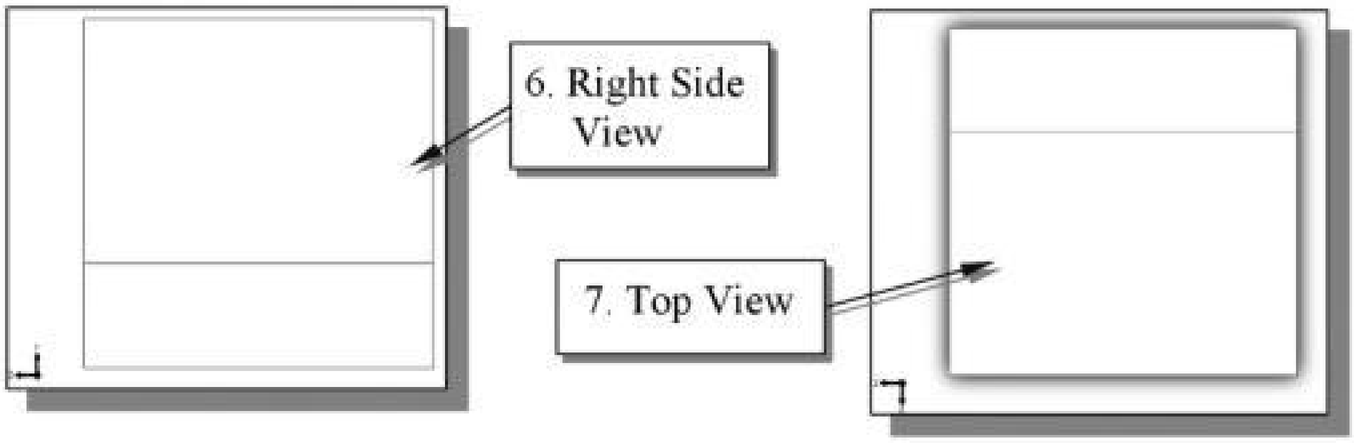
5. Select the **Front** icon in the *View Orientation* pull-down menu as shown to display the front view of the model.





6. Hold down the [**Shift**] key and hit the **left** arrow key. The view rotates to the right side view.

7. Hold down the [**Shift**] key and hit the **down** arrow key. The view rotates to the top view.



8. Reset the display to the isometric view.

- Hold down the [**Ctrl**] key and use the left-right and up-down arrow keys to pan the model in increments.

9. Hold down the [**Ctrl**] key and hit the **left** arrow key. The view pans, moving the model toward the left side of the screen. On your own use [**Ctrl**] key plus the left-right and up-down arrow keys to pan the view.

Dynamic Viewing – Quick Keys

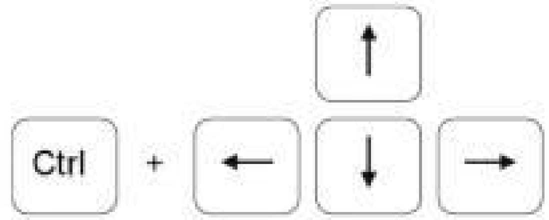
We can also use the function keys on the keyboard and the mouse to access the *Dynamic Viewing* functions.

❖ **Panning –**

(1) Hold [Ctrl] key; press and drag the mouse wheel

Hold the [**Ctrl**] function key down, and press and drag with the mouse wheel to pan the display. This allows you to reposition the display while maintaining the same scale factor of the display.

(2) Hold [Ctrl] key; use arrow keys

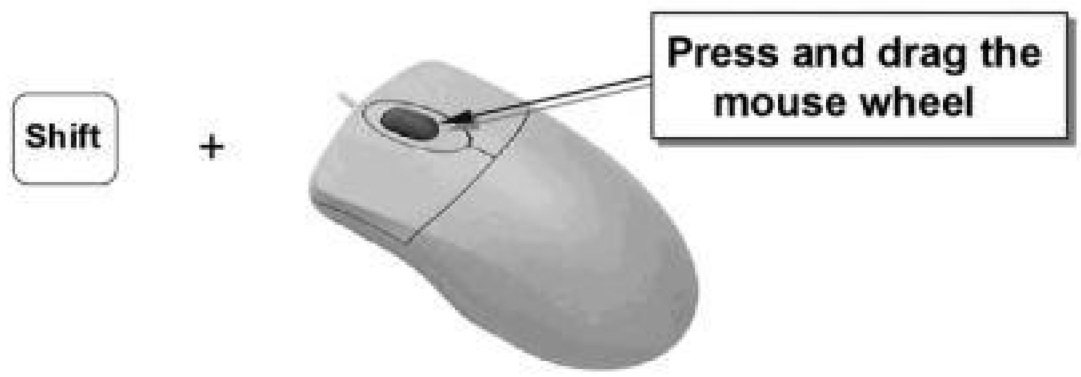


❖ Zooming

(1) Hold [Shift] key; press and drag the mouse wheel

Hold the [**Shift**] function key down, and press and drag with the mouse wheel to zoom the display. Moving downward will reduce the scale of the display, making the entities display smaller on the screen. Moving upward will magnify the scale of the display.

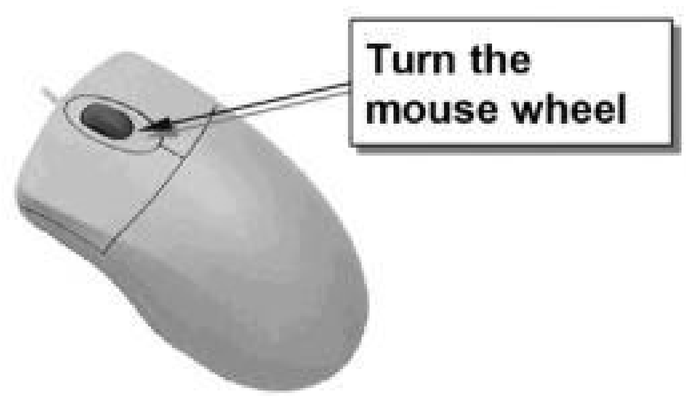
Zoom



(2) Turn the mouse wheel

Turning the mouse wheel can also adjust the scale of the display. Turning forward will reduce the scale of the display, making the entities display smaller on the screen. Turning backward will magnify the scale of the display.

- Turning the mouse wheel allows zooming to the position of the cursor.
- If the cursor is outside the graphics area, the wheel will allow zooming to the center of the graphics area.



(3) Z key or [Shift] + Z key

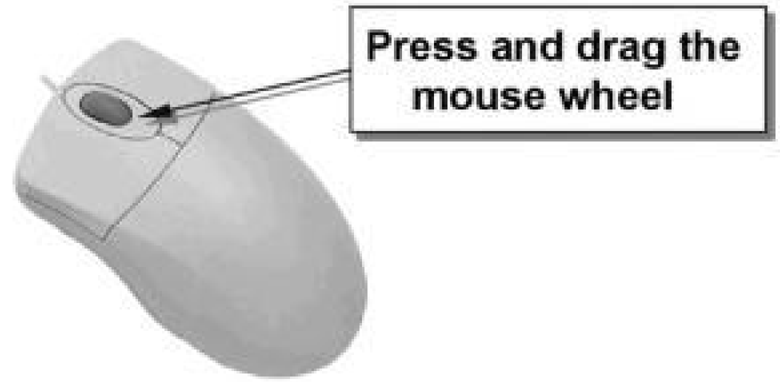
Pressing the **Z** key on the keyboard will zoom out. Holding the [**Shift**] function key and pressing the **Z** key will zoom in.

❖ **3D Rotation**

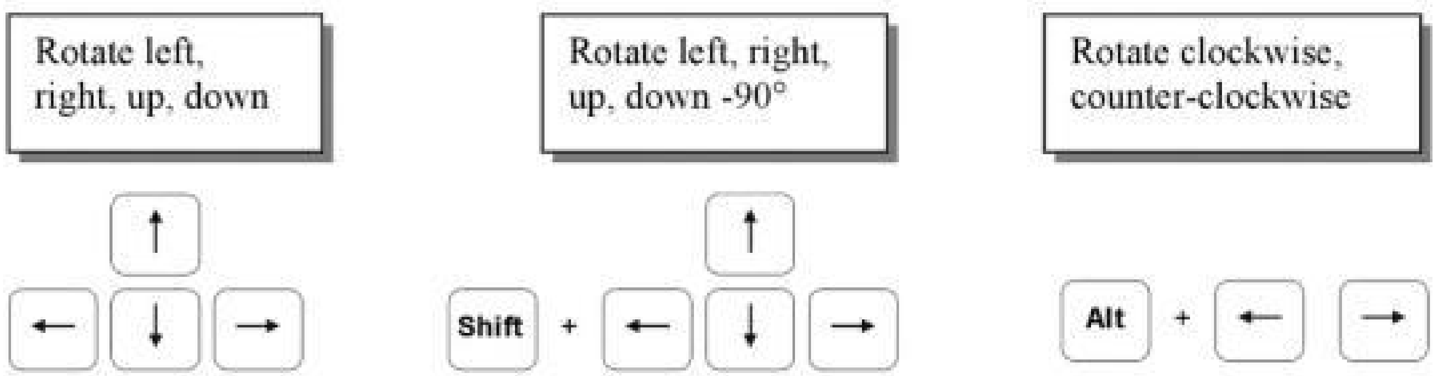
(1) Press and drag the mouse wheel

Press and drag with the mouse wheel to rotate the display.

Rotation

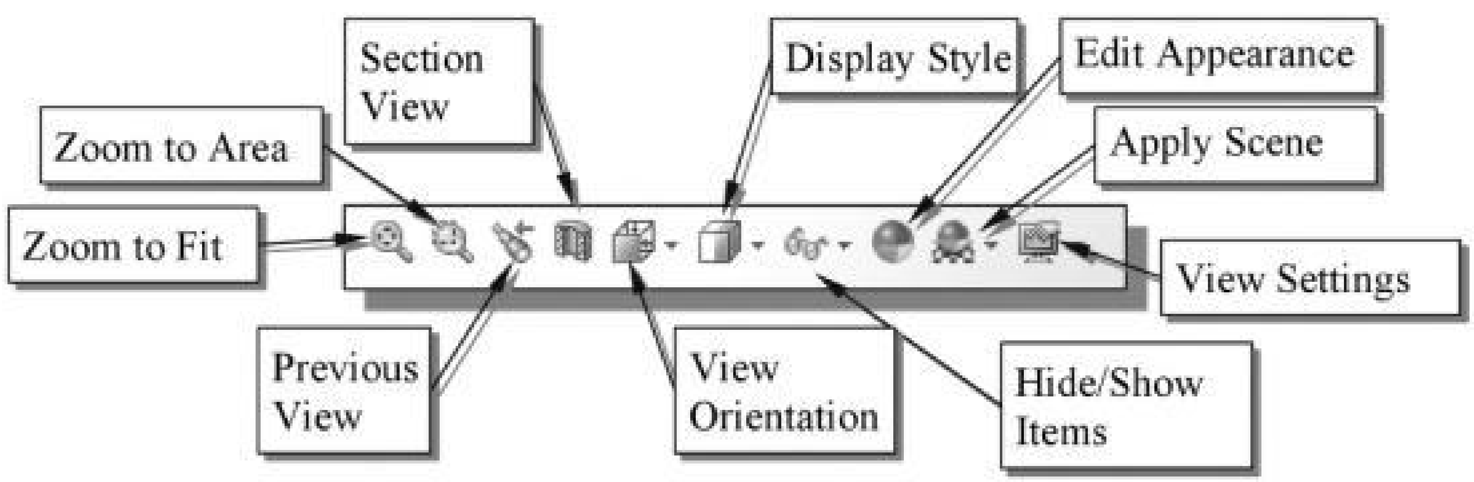


(2) Use arrow keys



Viewing Tools – Heads-up View Toolbar

The ***Heads-up View*** toolbar is a transparent toolbar which appears in each viewport and provides easy access to commonly used tools for manipulating the view. The default toolbar is described below.



Zoom to Fit – Adjusts the view so that all items on the screen fit inside the graphics window.

Zoom to Area – Use the cursor to define a region for the view; the defined region is zoomed to fill the graphics window.

Previous View – Returns to the previous view.

Section View – Displays a cutaway of a part or assembly using one or more section planes.

View Orientation – This allows you to change the current view orientation or number of viewports.

Display Style – This can be used to change the display style (shaded, wireframe, etc.) for the active view.

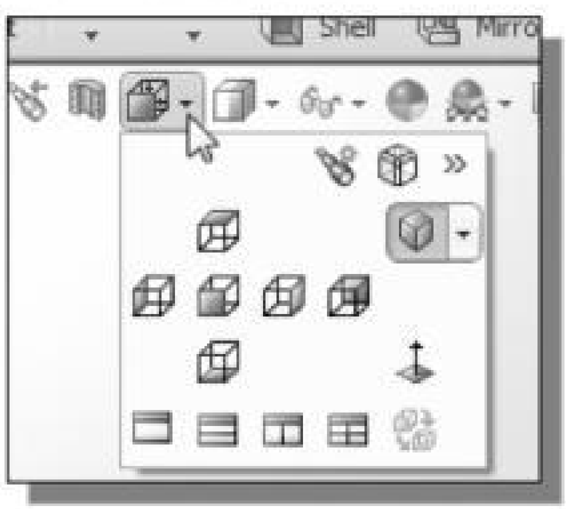
Hide/Show Items – The pull-down menu is used to control the visibility of items (axes, sketches, relations, etc.) in the graphics area.

Edit Appearance – Modify the appearance of entities in the model.

Apply Scene – Cycles through or applies a specific scene.

View Settings – Allows you to toggle various view settings (e.g., shadows, perspective).

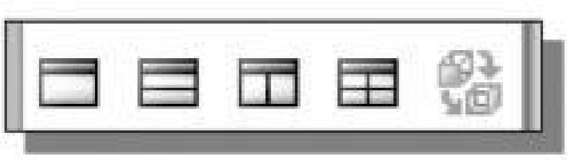
View Orientation



1. Click on the **View Orientation** icon on the *Heads-up View* toolbar to reveal the view orientation and number of viewports options.

Standard view orientation options – **Front**, **Back**, **Left**, **Right**, **Top**, **Bottom**, **Isometric**, **Trimetric** or **Dimetric** – icons can be selected to display the corresponding standard view. In the figure to the left, the isometric view is selected.

Normal to – In a part or assembly, zooms and rotates the model to display the selected plane or face. You can select the element either before or after clicking the Normal to icon.



❖ The icons across the bottom of the pull-down menu allow you to display a single viewport (the default) or multiple viewports.

Display Style

1. Click on the **Display Style** icon on the *Heads-up View* toolbar to reveal the display style options.

Shaded with Edges – Allows the display of a shaded view of a 3D model with its edges.

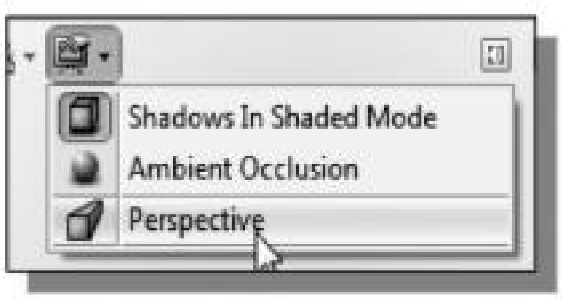
Shaded – Allows the display of a shaded view of a 3D model.

Hidden Lines Removed – Allows the display of the 3D objects using the basic wireframe representation scheme. Only those edges which are visible in the current view are displayed.

Hidden Lines Visible – Allows the display of the 3D objects using the basic wireframe representation scheme in which all the edges of the model are displayed, but edges that are hidden in the current view are displayed as dashed lines (or in a different color).

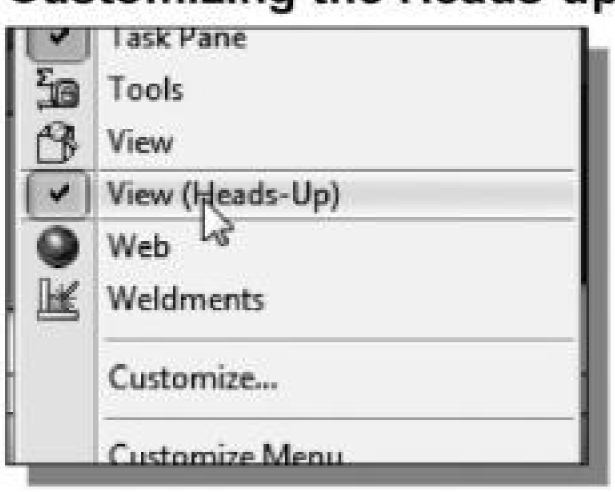
Wireframe – Allows the display of 3D objects using the basic wireframe representation scheme in which all the edges of the model are displayed.

Orthographic vs. Perspective



Besides the basic display modes, we can also choose an orthographic view or perspective view of the display. Clicking on the View Settings icon on the *Heads-up View* toolbar will reveal the Perspective icon. Clicking on the Perspective icon toggles the perspective view *ON* and *OFF*.

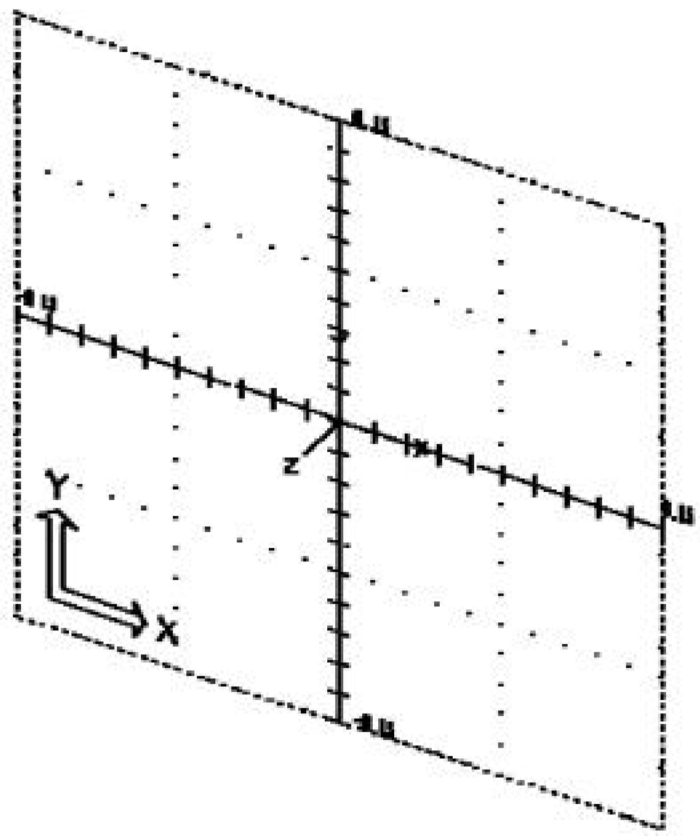
Customizing the Heads-up View Toolbar



The *Heads-up View* toolbar can be customized to contain icons for user-preferred view options. **Right-click** anywhere on the *Heads-up View* toolbar to reveal the *Display* menu option list. Click on the **View (Heads –up)** icon to turn *OFF* the display of the toolbar. Notice the **Customize** option is also available for add/remove different icons.

- On your own, use the different options described in the above sections to familiarize yourself with the 3D viewing/display commands. Reset the display to the standard **isometric view** before continuing to the next section.

Sketch Plane



Design modeling software is becoming more powerful and user friendly, yet the system still does only what the user tells it to do. When using a geometric modeler, we therefore need to have a good understanding of what its inherent limitations are. We should also have a good understanding of what we want to do and what to expect, as the results are based on what is available.

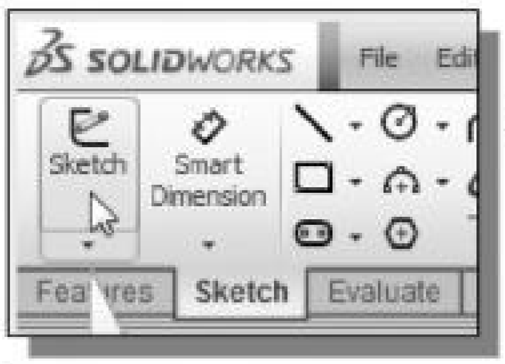
In most 3D geometric modelers, 3D objects are located and defined in what is usually called **world space** or **global space**. Although a number of different coordinate systems can be used to create and manipulate objects in a 3D modeling system, the objects are typically defined and stored using the world space. The world space is usually a **3D Cartesian coordinate system** that the user cannot change or manipulate.

In most engineering designs, models can be very complex, and it would be tedious and confusing if only the world coordinate system were available. Practical 3D modeling systems allow the user to define **Local Coordinate Systems** relative to the world coordinate system. Once a local coordinate system is defined, we can then create geometry in terms of this more convenient system.

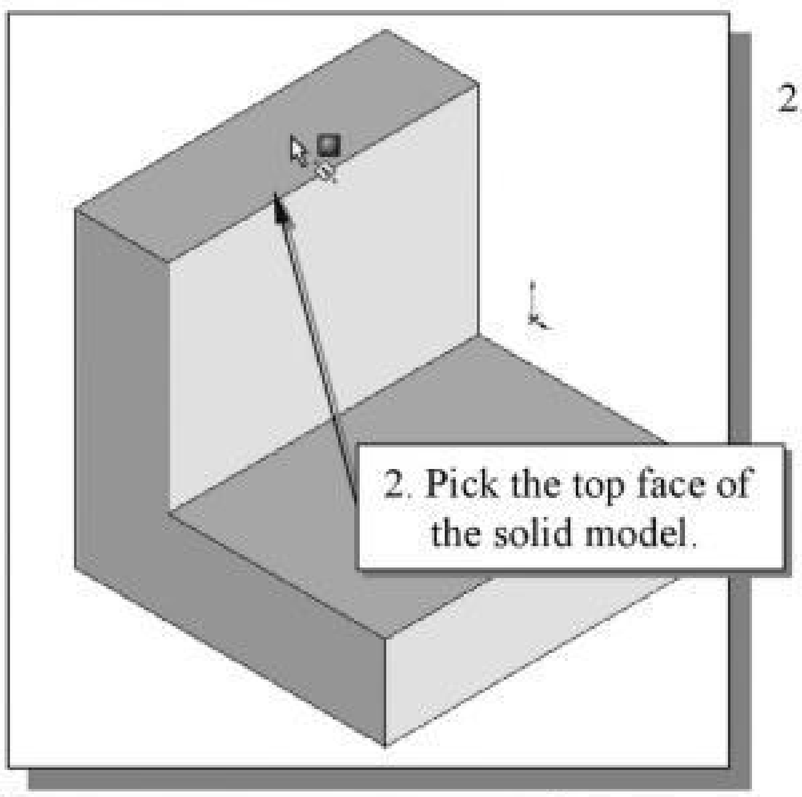
Although objects are created and stored in 3D space coordinates, most of the geometry entities can be referenced using 2D Cartesian coordinate systems. Typical input devices such as a mouse or digitizer are two-dimensional by nature; the movement of the input device is interpreted by the system in a planar sense. The same limitation is true of common output devices, such as screen displays and plotters. The modeling software performs a series of three-dimensional to two-dimensional transformations to correctly project 3D objects onto a 2D picture plane.

The *SolidWorks* ***sketch plane*** is a special construction tool that enables the planar nature of 2D input devices to be directly mapped into the 3D coordinate system. The *sketch plane* is a local coordinate system that can be aligned to the world coordinate system, an existing face of a part, or a reference plane. By default, the *sketch plane* is aligned to the world coordinate system.

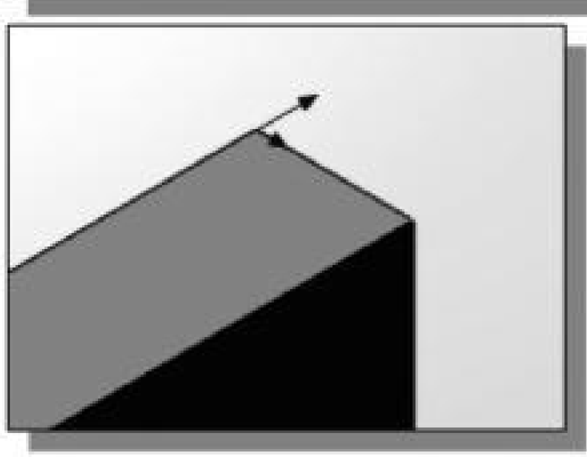
Think of a sketch plane as the surface on which we can sketch the 2D profiles of the parts. It is similar to a piece of paper, a white board, or a chalkboard that can be attached to any planar surface. The first profile we create is usually drawn on a sketch plane attached to a coordinate system such as the Front (XY), Top (XZ), and Right (YZ) sketch planes. Subsequent profiles can then be drawn on sketch planes that are defined on **planar faces of a part**, **work planes attached to part geometry**, or **sketch planes attached to a coordinate system**. The model we have created so far used the *SolidWorks* **Front Plane**, which is aligned to the XY plane of the world coordinate system.



1. Switch to the *Sketch* tab and select the **Sketch** button to create a new sketch.



2. In the *Edit Sketch Property Manager*, the message "*Select: 1) a plane, a planar face, or an edge on which to create a sketch for the entity*" is displayed. *SolidWorks* expects us to identify a planar surface where the 2D sketch of the next feature is to be created. Move the graphics cursor on the 3D part and notice that *SolidWorks* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Pick the top horizontal face of the 3D solid object.

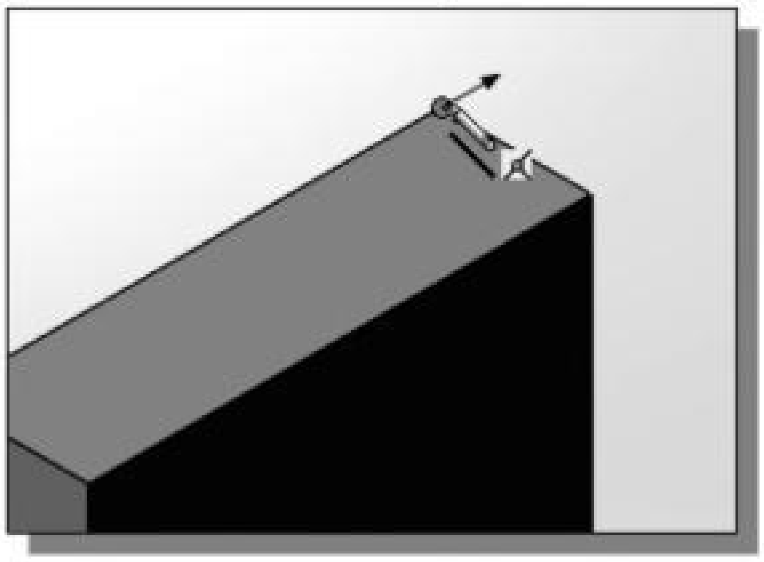


- Note that the sketch plane is aligned to the selected face. *SolidWorks* automatically establishes a local coordinate system, and records its location with respect to the part on which it was created.

Step 4-1: Adding an Extruded Boss Feature

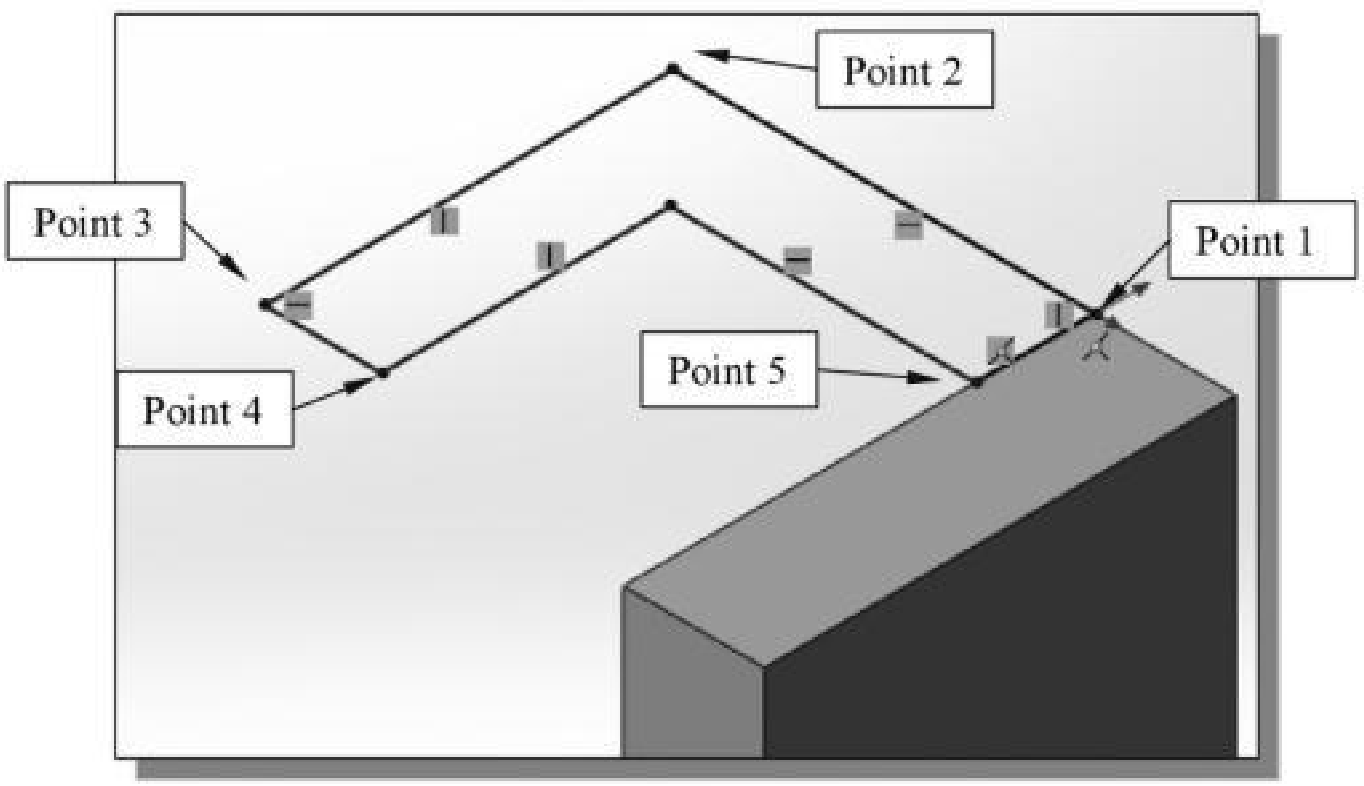
- Next, we will create and profile another sketch, a rectangle, which will be used to create another extrusion feature that will be added to the existing solid object.

1. Select the **Line** command by clicking once with the **left-mouse-button** on the icon in the *Sketch* toolbar.

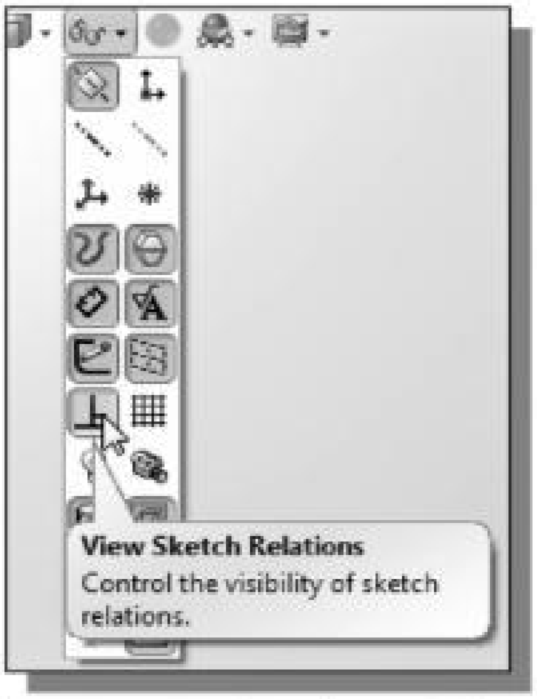


2. Move the cursor over the rear top vertex of the model. When the Coincident relation symbol appears as shown, click once with the **left-mouse-button**. This will start the first line, constraining its endpoint to be Coincident with the vertex.

3. Create a sketch with segments perpendicular/parallel to the existing edges of the solid model as shown below. **Close the sketch by ending at Point 1**. NOTE: Use the Pan and Zoom options discussed earlier to control the view, as needed.



4. Click the **OK** icon (green check mark) in the *Property Manager* twice, or hit the [**Esc**] key once, to end the Sketch Line command.

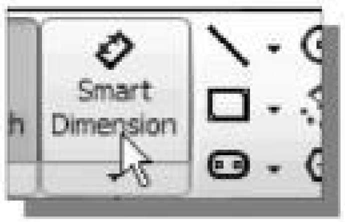


- We will hide the *Sketch Relation* icons in the sketch. The visibility of these and other items is controlled using the *Hide/Show Items* option on the *Heads-up View* toolbar.

5. Click on the **Hide/Show Items** icon on the *Heads-up View* toolbar to reveal the pull-down menu.

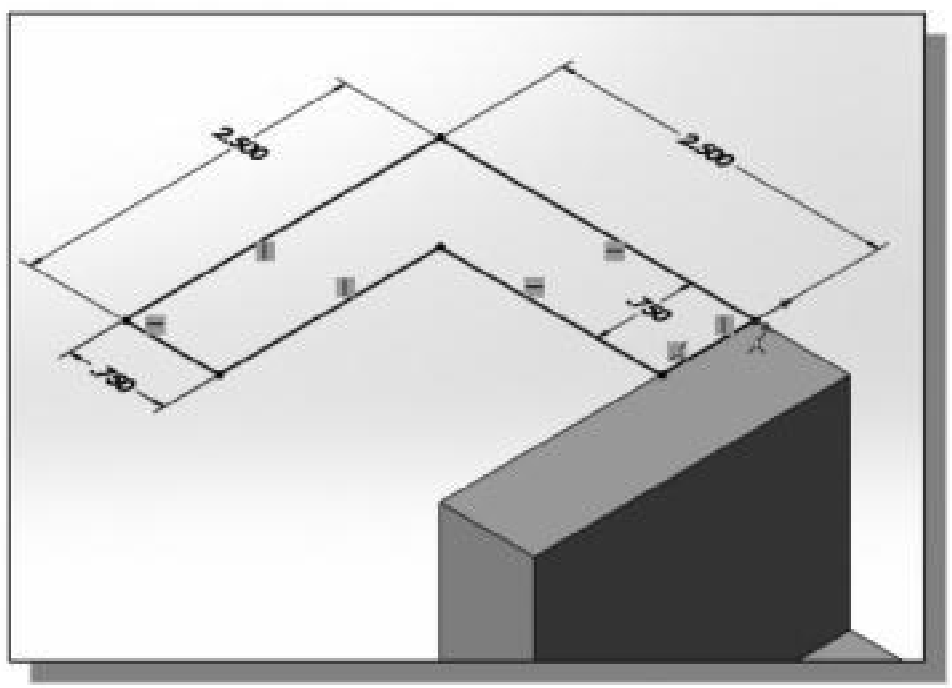
6. Click on the **View Sketch Relations** icon on the pull-down menu to toggle the sketch relation icon visibility *OFF*.

7. Click away from the pull-down menu to accept the settings.

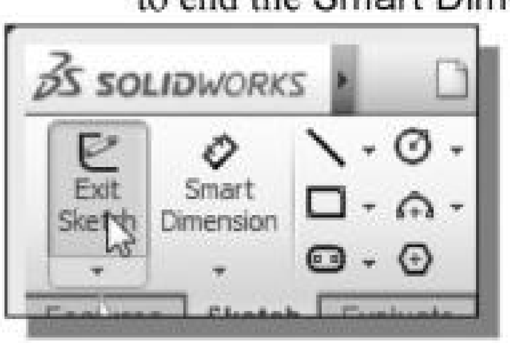


8. Select the **Smart Dimension** command in the *Sketch* toolbar. The Smart Dimension command allows us to quickly create and modify dimensions. Left-click once on the icon to activate the Smart Dimension command.

9. The message "*Select one or two edges/vertices and then a text location*" is displayed in the *Status Bar* area, at the bottom of the *SolidWorks* window. Create the four dimensions to describe the size of the sketch as shown in the figure, entering the values shown (***2.5***, ***2.5***, ***0.75*** and ***0.75***).



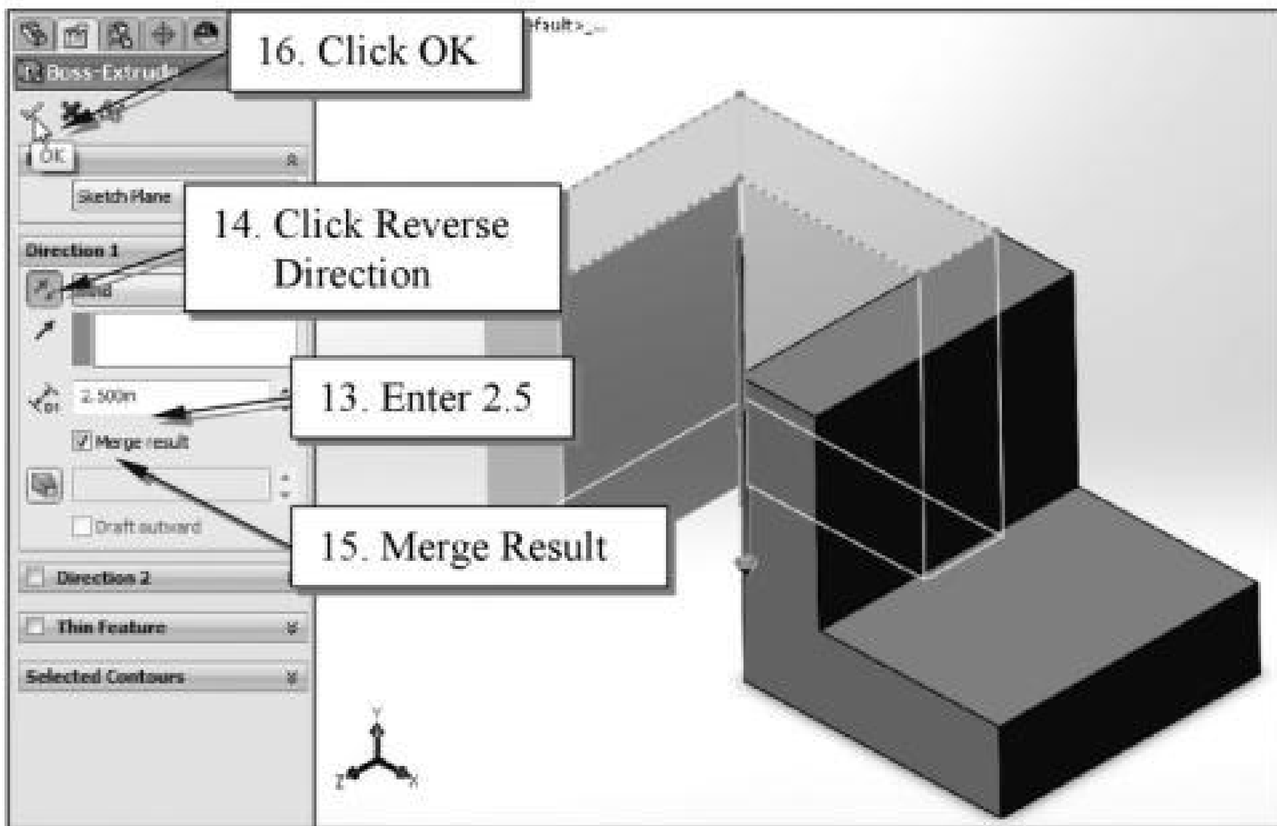
10. Click the **OK** icon in the *Property Manager* as shown, or hit the **[Esc]** key once, to end the Smart Dimension command.



11. Click once with the **left-mouse-button** on the **Exit Sketch** icon on the *Sketch* toolbar to exit the sketch.

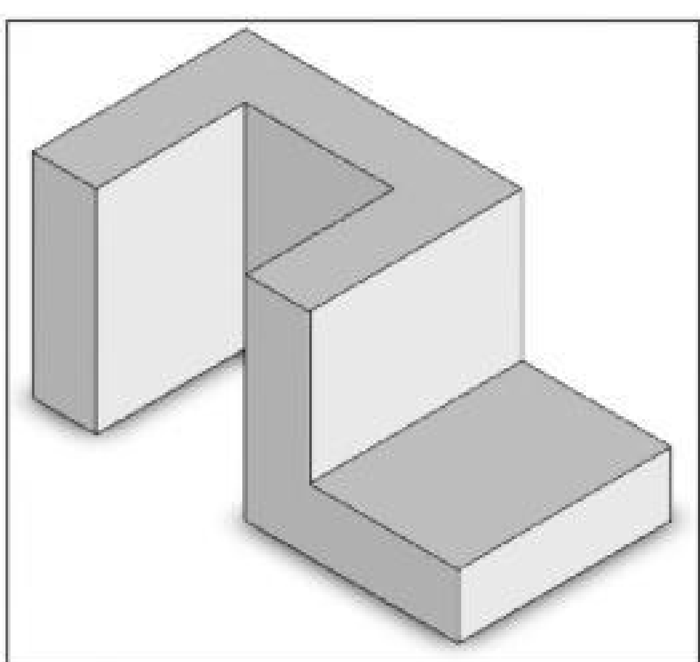
12. Switch to the *Features* toolbar (the first tab of the *Command Manager*), then select the **Extruded Boss/ Base** command by clicking once with the left-mouse-button on the icon.

13. In the *Extrude Property Manager* panel, enter **2.5** as the extrusion distance. Notice that the sketch region is automatically selected as the extrusion profile.



14. Click the **Reverse Direction** button in the *Property Manager* as shown. The extrude preview should appear as shown above.

15. Make sure the **Merge Result** option is selected (check box is checked).



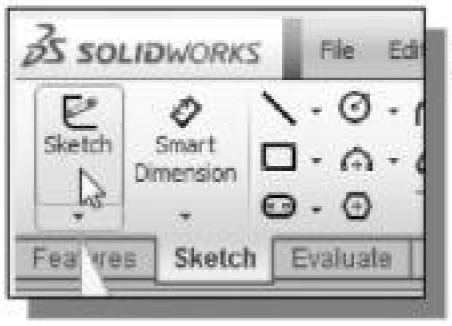
❖ With the *Merge Result* option selected, the resultant body is merged into the existing body (if possible). If the *Merge Result* option is not selected, the *Extruded Boss/Base* feature creates a distinct solid body.

16. Click on the **OK** button to proceed with creating the extruded feature.

Step 4-2: Adding an Extruded Cut Feature

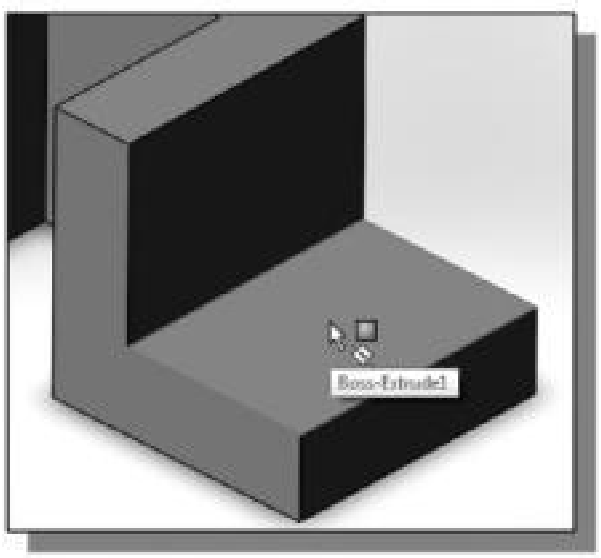
- Next, we will create and profile a circle, which will be used to create a cut feature that will be added to the existing solid object.

1. Click in the graphics area, away from the model, to ensure no items are selected.

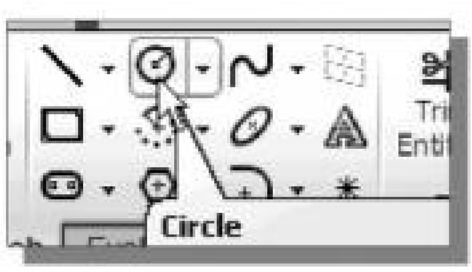


2. Switch to the *Sketch* tab and select the **Sketch** button to create a new sketch.

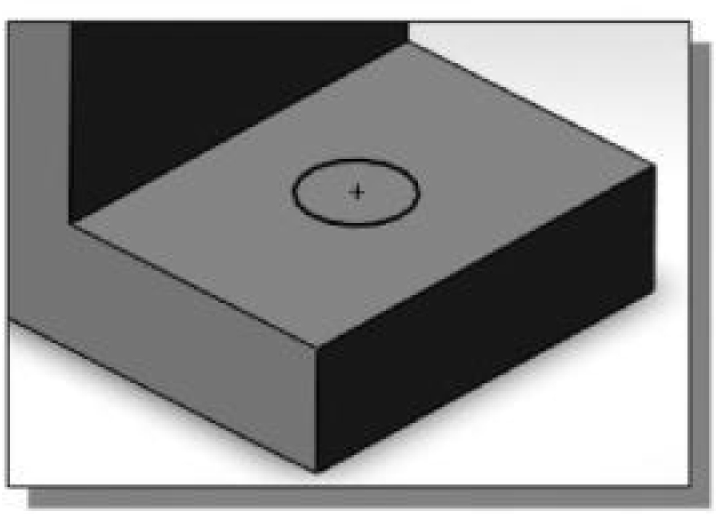
3. In the *Edit Sketch Property Manager*, the message "*Select: 1) a plane, a planar face, or an edge on which to create a sketch for the entity*" is displayed. *SolidWorks* expects us to identify a planar surface where the 2D sketch of the next feature is to be created. Move the graphics cursor on the 3D part and notice that *SolidWorks* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Pick the horizontal face of the 3D solid object.



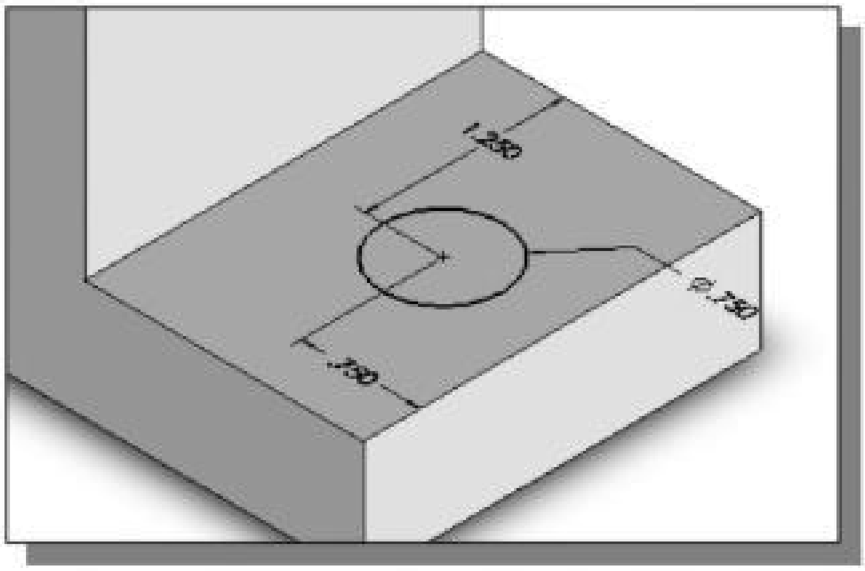
- Note that the sketch plane is aligned to the selected face.



4. Select the **Circle** command by clicking once with the **left-mouse-button** on the icon in the *Sketch* toolbar.



5. Create a circle of arbitrary size on the top face of the solid model as shown. Click once with the left-mouse-button to select the center of the circle; move and click again to set the radius. Press the **[Esc]** key once to end the Circle command.



6. On your own, create and modify the dimensions of the sketch as shown in the figure.

7. Click the **OK** icon in the *Property Manager*, or hit the [**Esc**] key once, to end the Smart Dimension command.

8. Click once with the **left-mouse-button** on the **Exit-Sketch** icon on the *Sketch* toolbar to exit the sketch.

9. In the *Features* toolbar (located at the left of the window), select the **Extruded Cut** command by clicking once with the left-mouse-button on the icon.

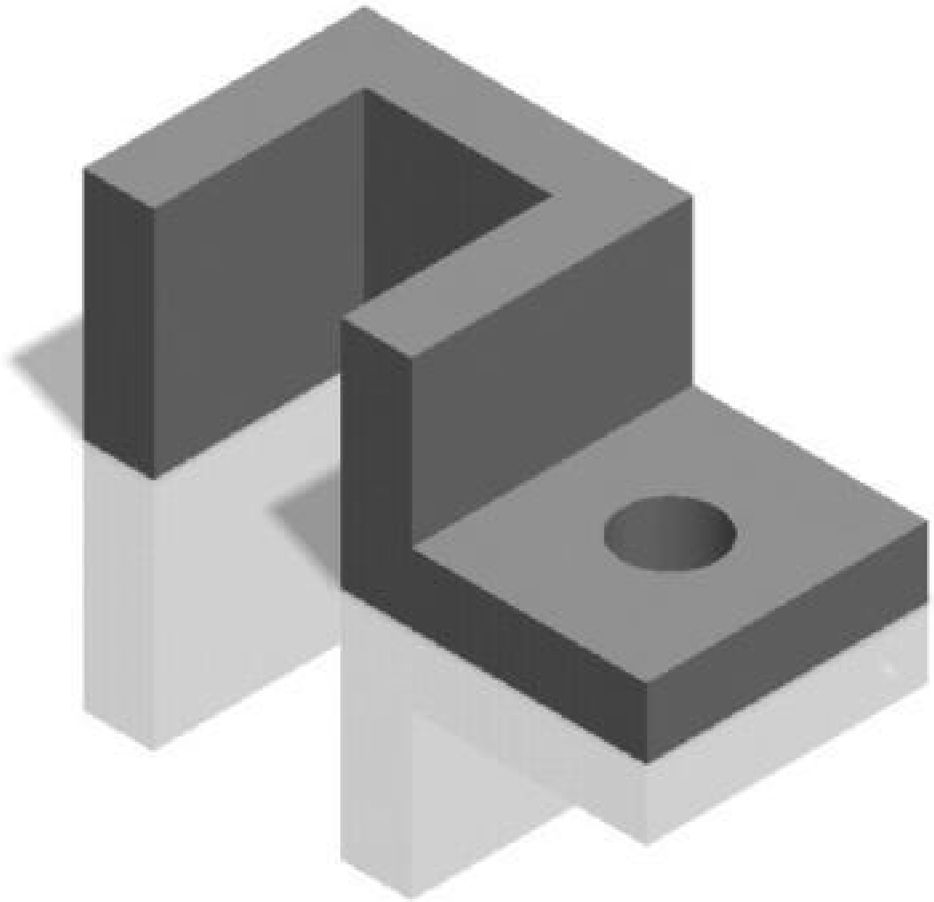
❖ The *Extrude Property Manager* is displayed in the left panel. Notice that the sketch region (the circle) is automatically selected as the extrusion profile.

10. In the *Cut-Extrude Property Manager* panel, click the arrow to reveal the pull-down options for the *End Condition* (the default end condition is Blind), and select **Through All** as shown.

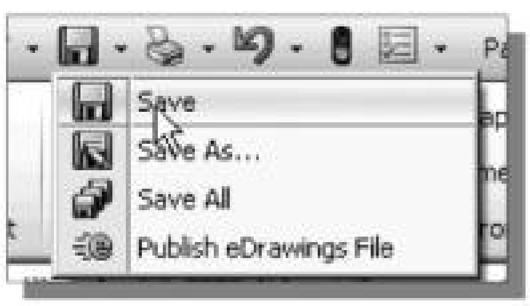
11. Click the **OK** button (green check mark) in the *Cut-Extrude Property Manager* panel.

12. Click in the graphics area, away from the model, to ensure no items are selected.

13. Press the **F** key on the keyboard to fit the model to the screen.



Save the Model



1. Select **Save** in the *Menu Bar* pull-down menu, or you can also use the [**Ctrl-S**] combination (hold down the [**Ctrl**] key and hit the [**S**] key once) to save the part.

2. In the popup window, select the directory to store the model in and enter ***Adjuster*** as the name of the file.

3. Click on the **Save** button to save the file.

❖ You should form a habit of saving your work periodically, just in case something might go wrong while you are working on it. In general, one should save one's work at an interval of every 15 to 20 minutes. One should also save before making any major modifications to the model.

Questions:

1. The truss element used in finite element analysis is considered as a two-force member element. List and describe the assumptions of a two-force member.

2. What is the size of the stiffness matrix for a single element? What is the size of the overall global stiffness matrix in example 1.2?

3. What is the first thing we should set up when building a new CAD model in *SolidWorks*?

4. How do we remove the dimensions created by the *system*?

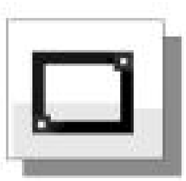
5. How do we modify existing dimensions?

6. Identify and describe the following commands:

(a)

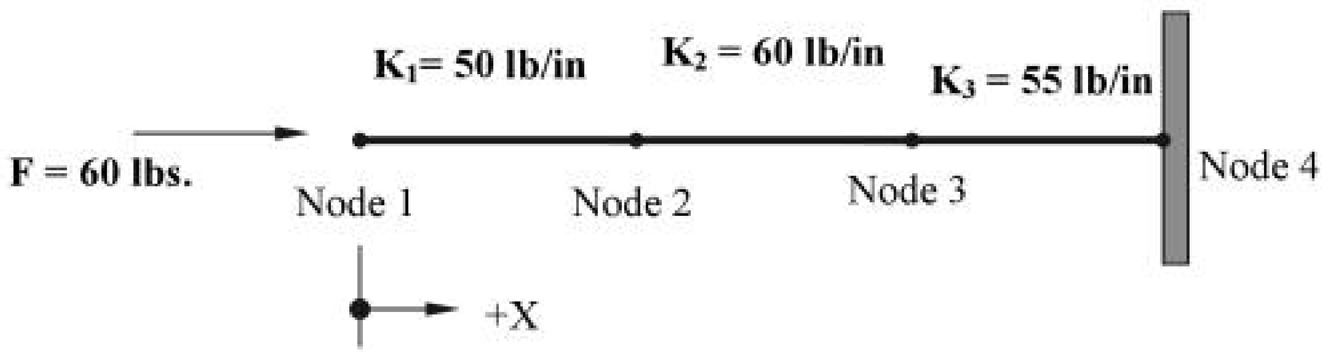
(b)

(c)

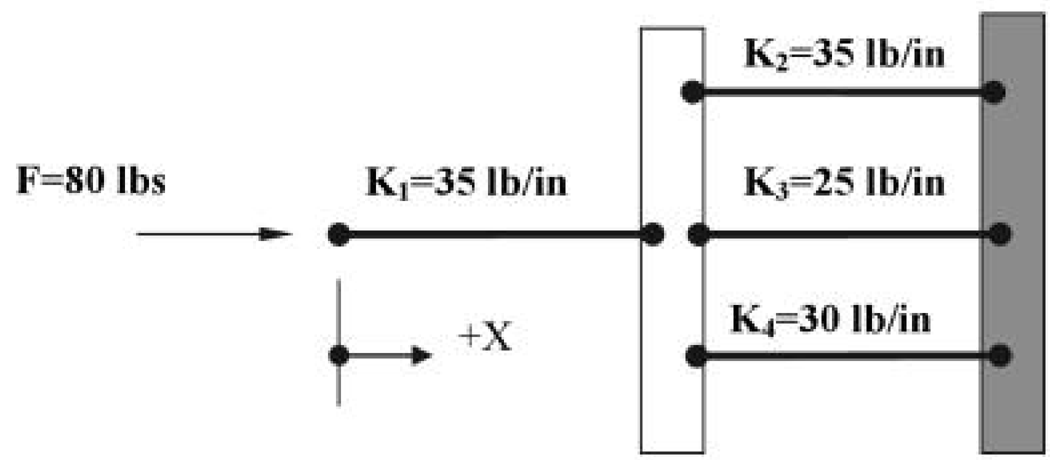


Exercises:

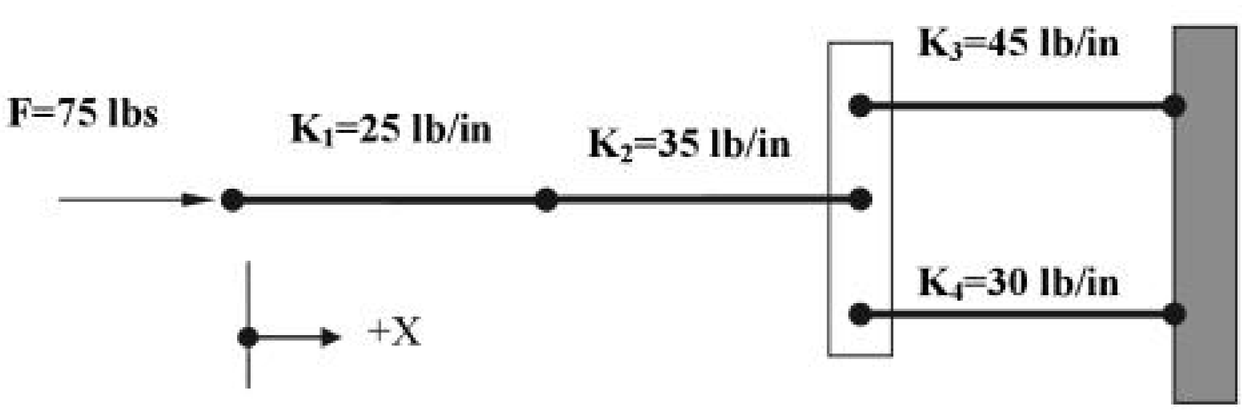
1. For the one-dimensional 3 truss-element system shown, determine the nodal displacements and reaction forces using the direct stiffness method.



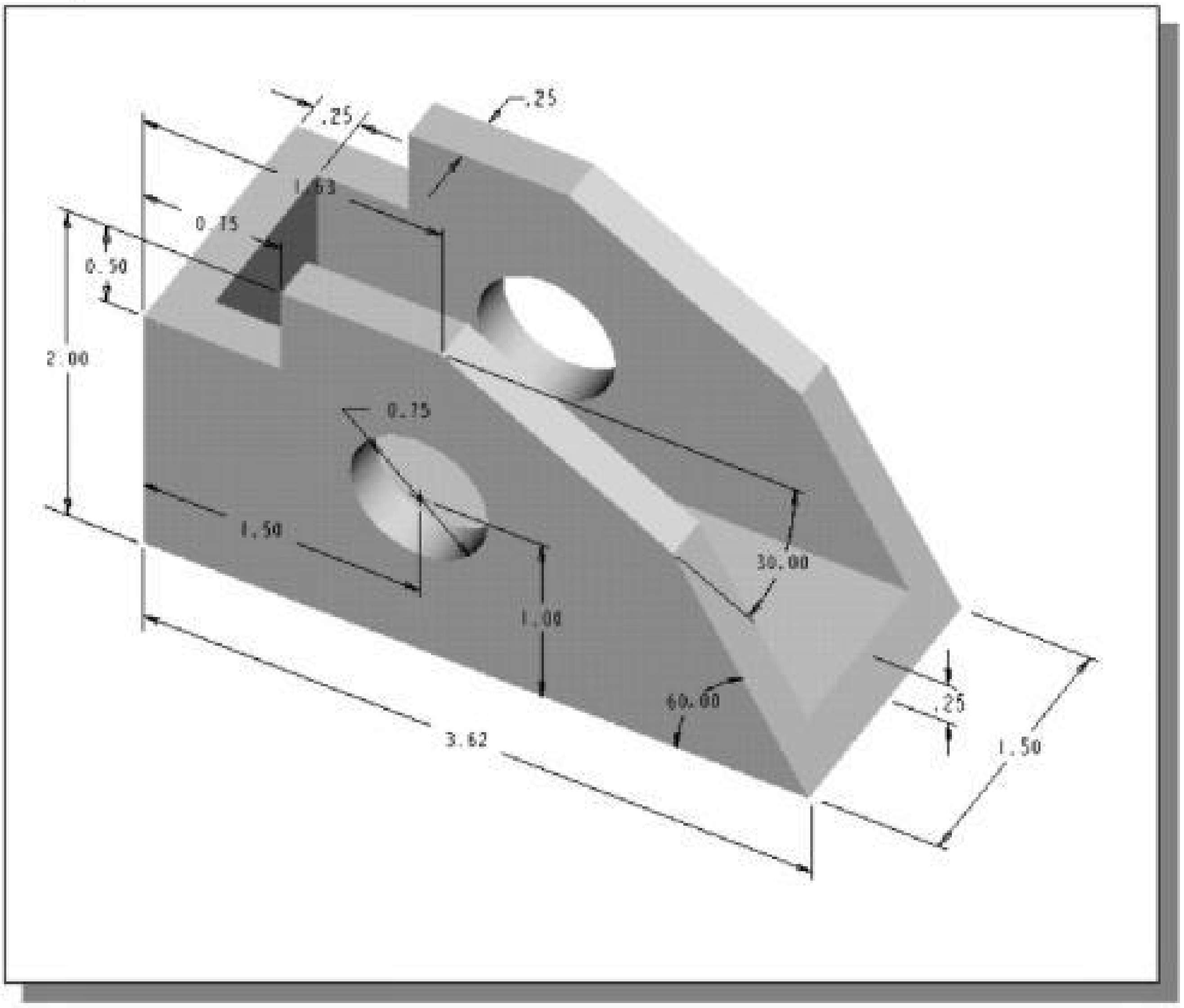
2. For the one-dimensional 4 truss-element system shown, determine the nodal displacements and reaction forces using the direct stiffness method.



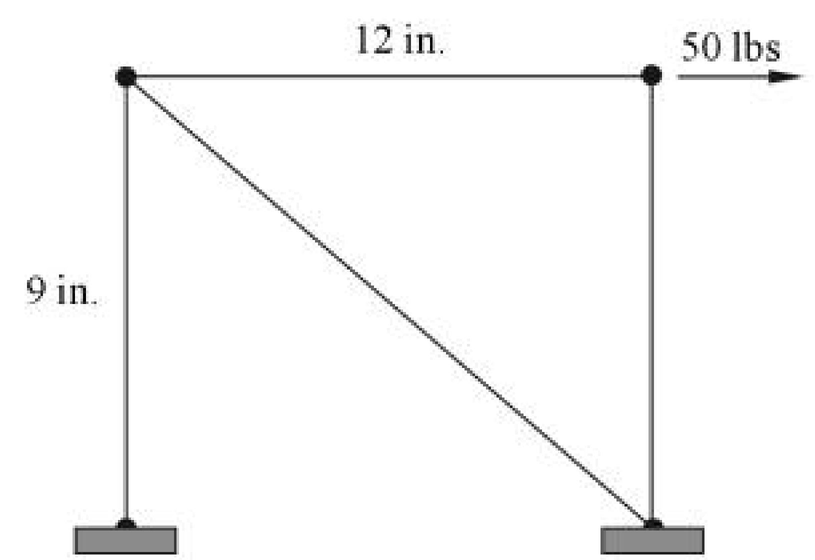
3. For the one-dimensional 4 truss-element system shown, determine the nodal displacements and reaction forces using the direct stiffness method



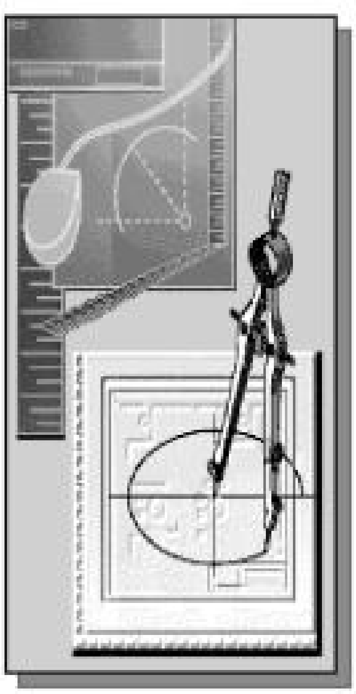
4. Using SolidWorks, create the 3D solid model as shown in the figure. (Dimensions are in inches.)



Chapter 2
Truss Elements in Two-Dimensional Spaces



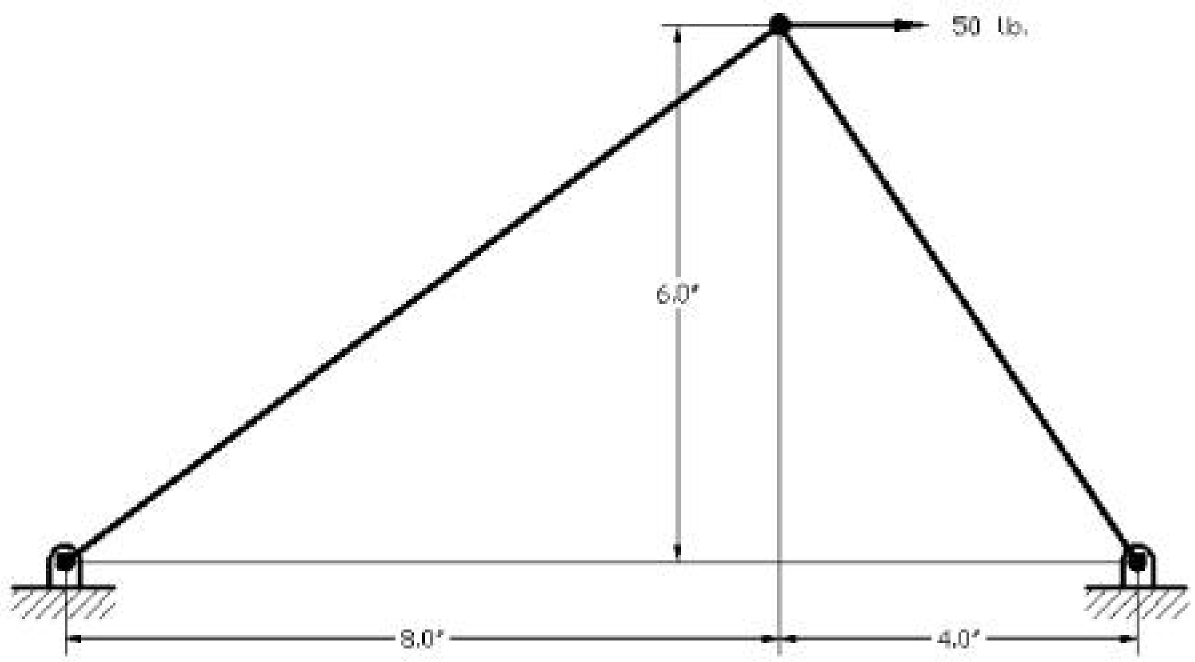
Learning Objectives



- Perform 2D Coordinate Transformation.
- Expand the Direct Stiffness Method to 2D Trusses.
- Derive the general 2D element Stiffness Matrix.
- Assemble the Global Stiffness Matrix for 2D Trusses.
- Solve 2D trusses using the Direct Stiffness Method.

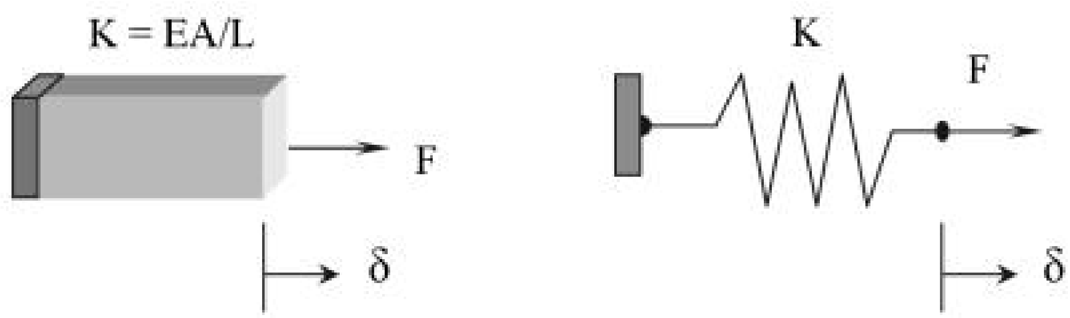
Introduction

This chapter presents the formulation of the direct stiffness method of truss elements in a two-dimensional space and the general procedure for solving two-dimensional truss structures using the direct stiffness method. The primary focus of this text is on the aspects of finite element analysis that are more important to the user than the programmer. However, for a user to utilize the software correctly and effectively, some understanding of the element formulation and computational aspects are also important. In this chapter, a two-dimensional truss structure consisting of two truss elements (as shown below) is used to illustrate the solution process of the direct stiffness method.

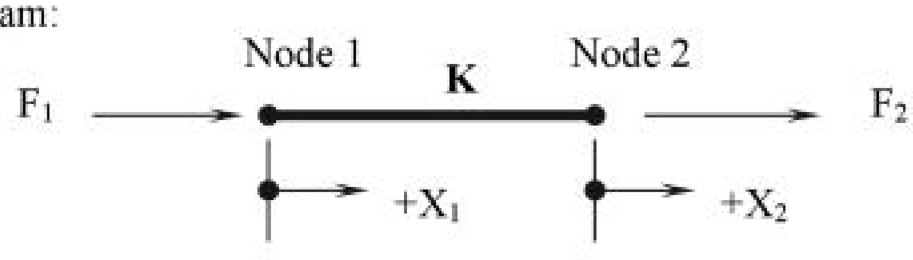


Truss Elements in Two-Dimensional Spaces

As introduced in the previous chapter, the system equations (stiffness matrix) of a truss element can be represented using the system equations of a linear spring in one-dimensional space.



Free Body Diagram:



The general force-displacement equations in matrix form:

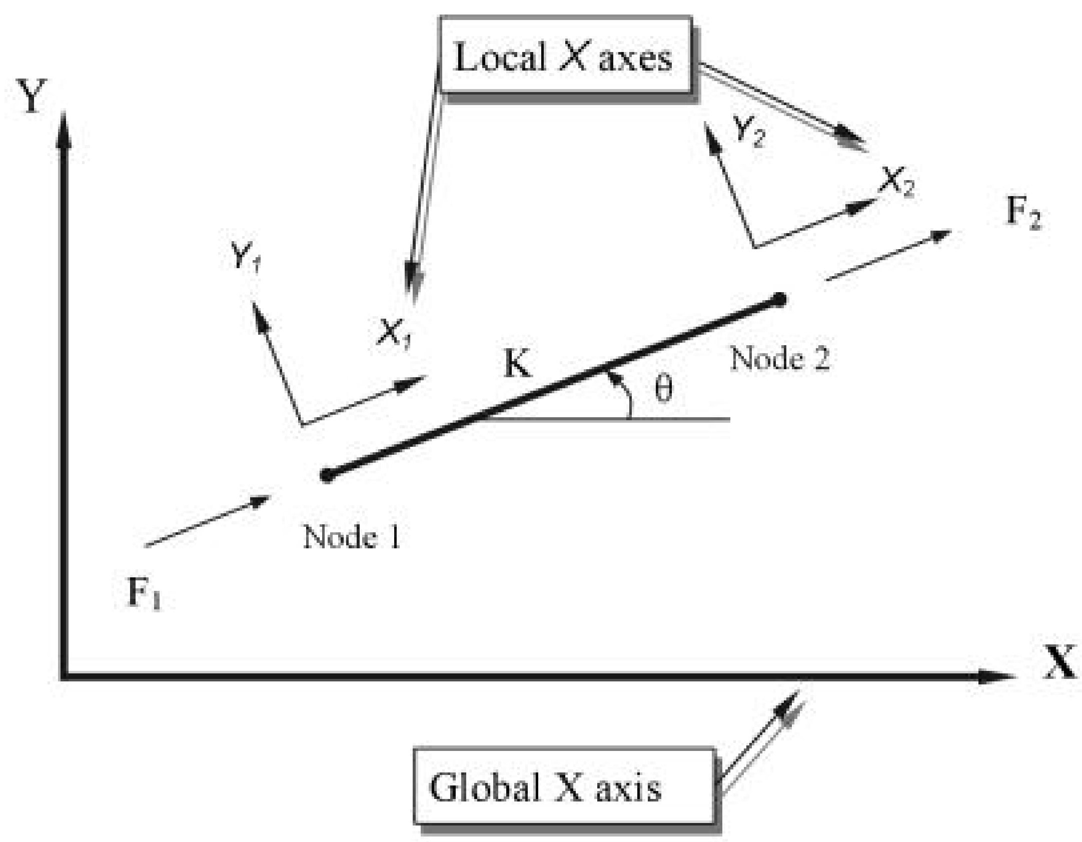
$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} +K & -K \\ -K & +K \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

For a truss element, **K = EA/L**

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{\mathbf{EA}}{\mathbf{L}} \begin{bmatrix} +1 & -1 \\ -1 & +1 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

For truss members positioned in two-dimensional space, two coordinate systems are established:

1. The global coordinate system (**X** and **Y** axes) chosen to represent the entire structure.
2. The local coordinate system (***X*** and ***Y*** axes) selected to align the ***X*** axis along the length of the element.



The force-displacement equations expressed in terms of components in the local *XY* coordinate system:

$$\begin{Bmatrix} F_{1X} \\ F_{2X} \end{Bmatrix} = \frac{\mathbf{EA}}{\mathbf{L}} \begin{bmatrix} +1 & -1 \\ -1 & +1 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix}$$

The above stiffness matrix (system equations in matrix form) can be expanded to incorporate the two force components at each node and the two displacement components at each node.

Force Components (Local Coordinate System)

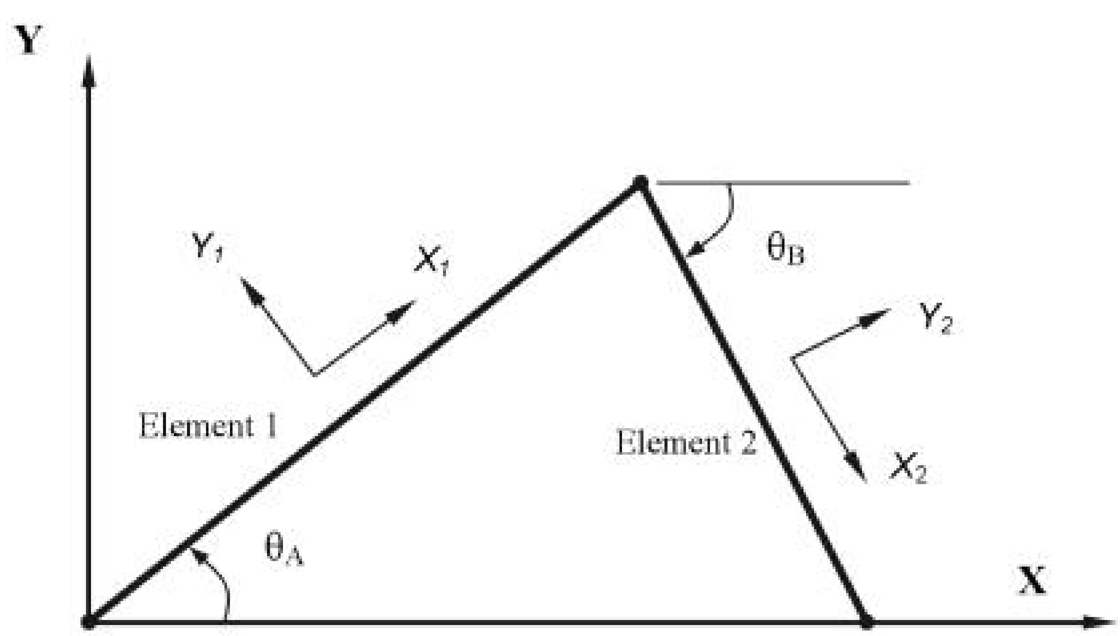
$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \end{Bmatrix} = \frac{\mathbf{EA}}{\mathbf{L}} \begin{bmatrix} +1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & +1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \end{Bmatrix}$$

Nodal Displacements (Local Coordinate System)

In regard to the expanded local stiffness matrix (system equations in matrix form):

1. It is always a square matrix.
2. It is always symmetrical for linear systems.
3. The diagonal elements are always positive or zero.

The above stiffness matrix, expressed in terms of the established 2D local coordinate system, represents a single truss element in a two-dimensional space. In a general structure, many elements are involved, and they would be oriented with different angles. The above stiffness matrix is a general form of a SINGLE element in a 2D local coordinate system. Imagine the number of coordinate systems involved for a 20-member structure. For the example that will be illustrated in the following sections, two local coordinate systems (one for each element) are needed for the truss structure shown below. The two local coordinate systems (X_1Y_1 and X_2Y_2) are aligned to the elements.

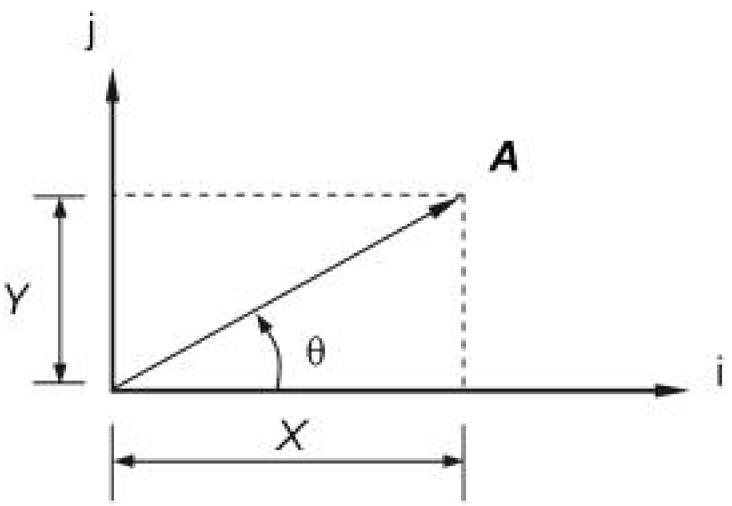


In order to solve the system equations of two-dimensional truss structures, it is necessary to assemble all elements' stiffness matrices into a **global stiffness matrix**, with all the equations of the individual elements referring to a common global coordinate system. This requires the use of *coordinate transformation equations* applied to system equations for all elements in the structure. For a one-dimensional truss structure (illustrated in chapter 2), the local coordinate system coincides with the global coordinate system; therefore, no coordinate transformation is needed to assemble the global stiffness matrix (the stiffness matrix in the global coordinate system). In the next section, the coordinate transformation equations are derived for truss elements in two-dimensional spaces.

Coordinate Transformation

A vector, in a two-dimensional space, can be expressed in terms of any coordinate system set of unit vectors.

For example,



Vector ***A*** can be expressed as:

$$\mathbf{A} = X\,i + Y\,j$$

Where i and j are unit vectors along the X and Y axes.

Magnitudes of X and Y can also be expressed as:

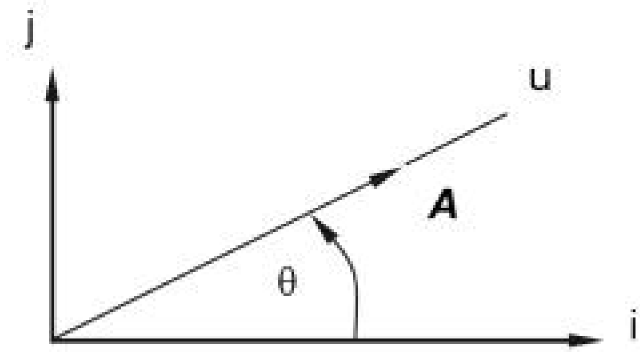
$$X = A \cos(\theta)$$
$$Y = A \sin(\theta)$$

Where X, Y *and* A are scalar quantities.

Therefore,

$$\mathbf{A} = X\,i + Y\,j = A\cos(\theta)\,i + A\sin(\theta)\,j \quad \text{----------- (1)}$$

Next, establish a new unit vector (u) in the same direction as vector $\boldsymbol{A}$.



Vector A can now be expressed as: $\boldsymbol{A} = A\,u$ -------------- (2)

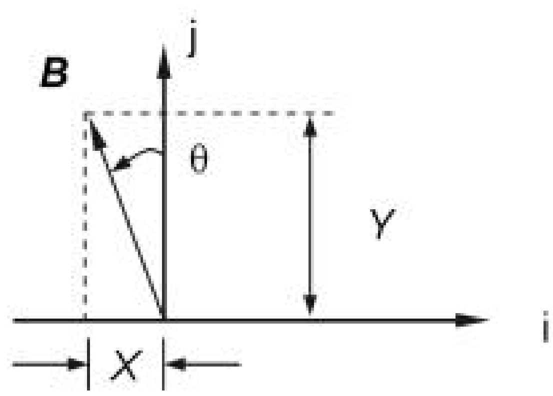
Both equations (the above (1) and (2)) represent vector $\boldsymbol{A}$:

$$\boldsymbol{A} = A\,u = A\cos(\theta)\,i + A\sin(\theta)\,j$$

The unit vector u can now be expressed in terms of the original set of unit vectors i and j:

$$\boxed{u = \cos(\theta)\,i + \sin(\theta)\,j}$$

Now consider another vector $\boldsymbol{B}$:



Vector $\boldsymbol{B}$ can be expressed as:

$$\boldsymbol{B} = -X\,i + Y\,j$$

Where i and j are unit vectors along the X- and Y-axes.

Magnitudes of X and Y can also be expressed as components of the magnitude of the vector:

$$X = B\sin(\theta)$$
$$Y = B\cos(\theta)$$

Where X, Y and B are scalar quantities.

Therefore,

$$\boldsymbol{B} = -X\,i + Y\,j = -B\sin(\theta)\,i + B\cos(\theta)\,j \quad \text{---------- (3)}$$

The transformation equations that enable us to transform any vector from a *LOCAL coordinate system* to the *GLOBAL coordinate system* become:

LOCAL coordinates to the GLOBAL coordinates:

$$\begin{Bmatrix} i \\ j \end{Bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) \\ \sin(\theta) & \cos(\theta) \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

The reverse transformation can also be established by applying the transformation equations that transform any vector from the *GLOBAL coordinate system* to the *LOCAL coordinate system*:

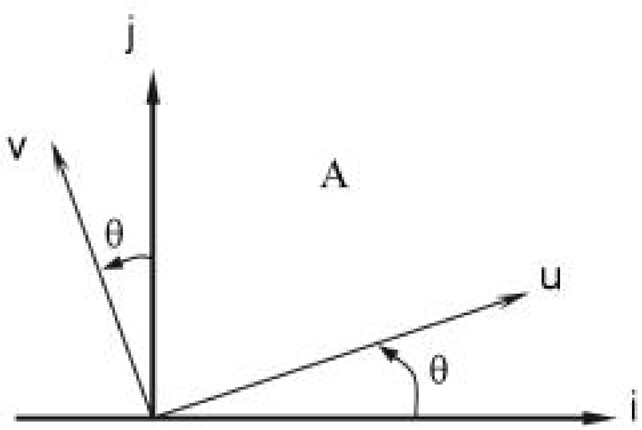
GLOBAL coordinates to the LOCAL coordinates:

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{Bmatrix} i \\ j \end{Bmatrix}$$

As it is the case with many mathematical equations, derivation of the equations usually appears to be much more complex than the actual application and utilization of the equations. The following example illustrates the application of the two-dimensional *coordinate transformation equations* on a point in between two coordinate systems.

EXAMPLE 2.1

Given:



The coordinates of point A: (20 i, 40 j).

Find: The coordinates of point A if the local coordinate system is rotated 15 degrees relative to the global coordinate system.

Solution:

Using the coordinate transformation equations (GLOBAL coordinates to the LOCAL coordinates):

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{Bmatrix} i \\ j \end{Bmatrix}$$

$$= \begin{bmatrix} \cos(15^\circ) & \sin(15^\circ) \\ -\sin(15^\circ) & \cos(15^\circ) \end{bmatrix} \begin{Bmatrix} 20 \\ 40 \end{Bmatrix}$$

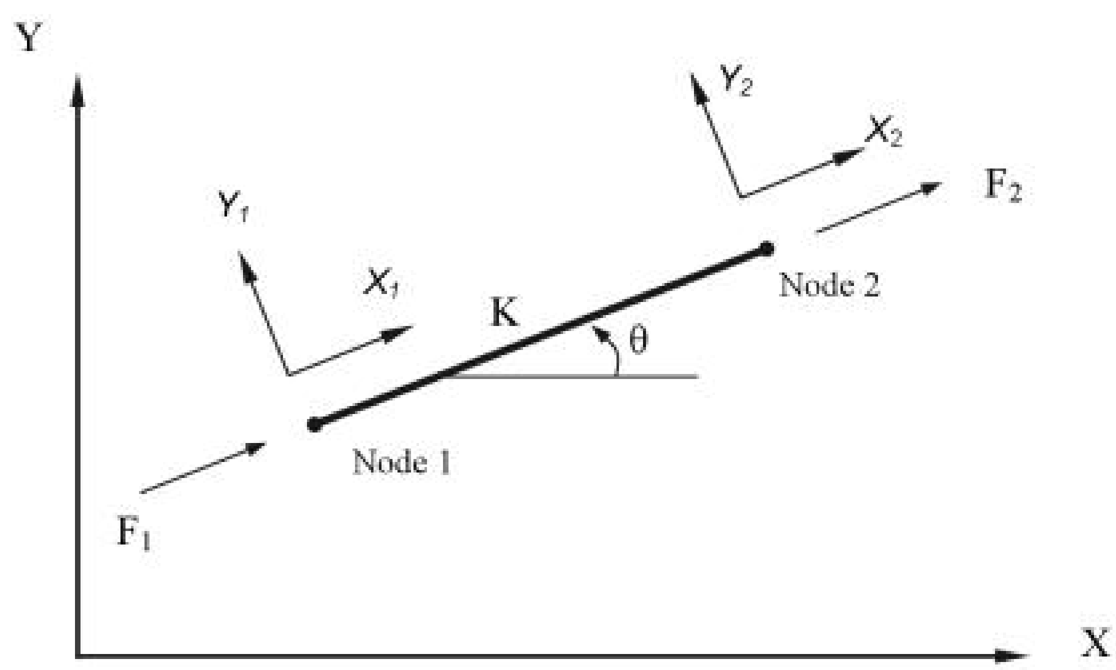
$$= \begin{Bmatrix} 29.7 \\ 32.5 \end{Bmatrix}$$

- On your own, perform a coordinate transformation to determine the global coordinates of point A using the *LOCAL* coordinates of (29.7,32.5) with the 15 degrees angle in between the two coordinate systems.

Global Stiffness Matrix

For a single truss element, using the coordinate transformation equations, we can proceed to transform the local stiffness matrix to the global stiffness matrix.

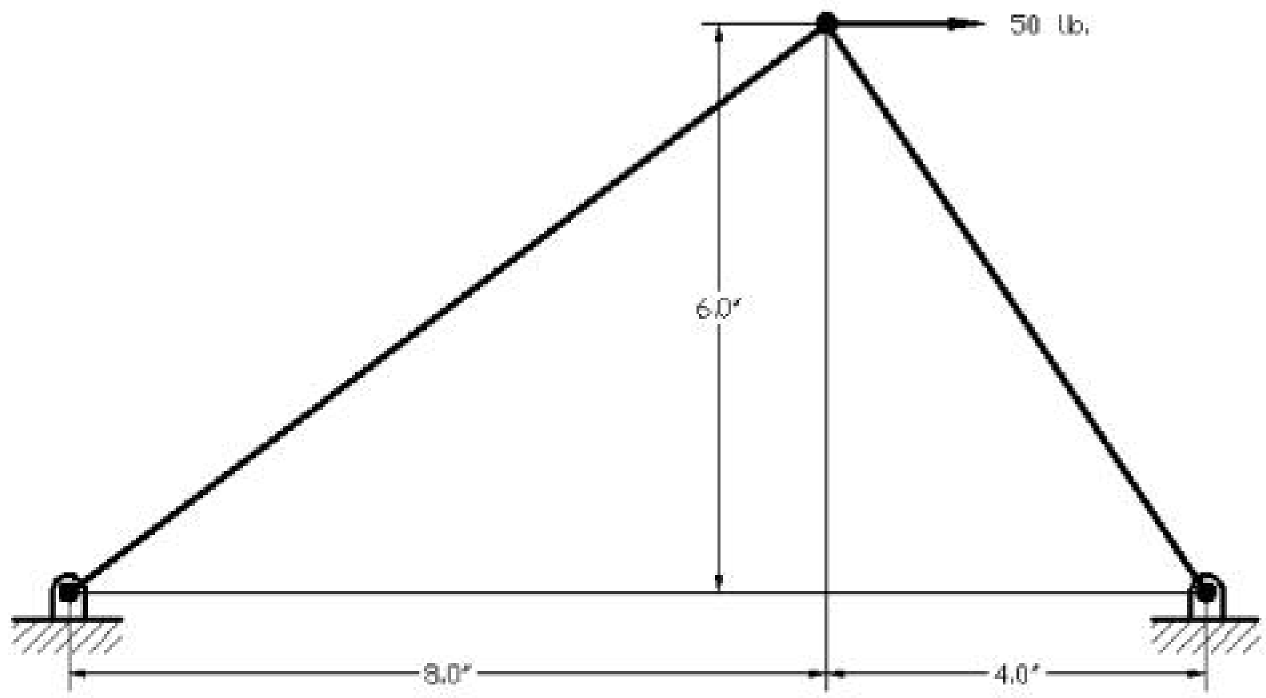
For a single truss element arbitrarily positioned in a two-dimensional space:



You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Example 2.2

Given: A two-dimensional truss structure as shown. (All joints are **Pin Joints**.)

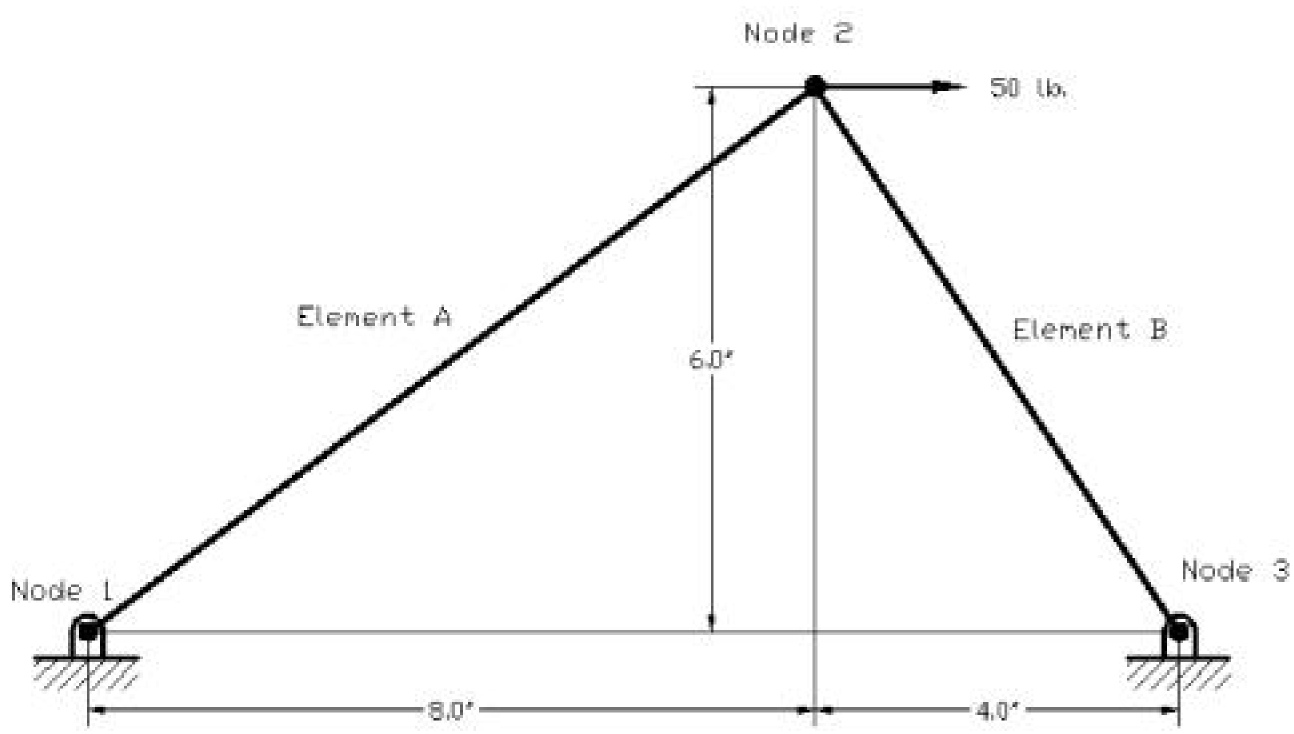


Material: Steel rod, diameter ¼ in.

Find: Displacements of each node and stresses in each member.

Solution:

The system contains two elements and three nodes. The nodes and elements are labeled as shown below.



$$\underbrace{\begin{Bmatrix} F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}}_{\text{Local system}} = \underbrace{\frac{EA}{L}\begin{pmatrix} +1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & +1 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix}}_{\text{Local Stiffness Matrix}} \underbrace{\begin{Bmatrix} X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{Bmatrix}}_{\text{Local system}}$$

Using the equations we derived in the previous sections, the GLOBAL system equation for *Element B* is:

$$\{ F \} = [K] \{ X \}$$

$$[K] = \frac{EA}{L}\begin{pmatrix} \cos^2(\theta) & \cos(\theta)\sin(\theta) & -\cos^2(\theta) & -\cos(\theta)\sin(\theta) \\ \cos(\theta)\sin(\theta) & \sin^2(\theta) & -\cos(\theta)\sin(\theta) & -\sin^2(\theta) \\ -\cos^2(\theta) & -\cos(\theta)\sin(\theta) & \cos^2(\theta) & \cos(\theta)\sin(\theta) \\ -\cos(\theta)\sin(\theta) & -\sin^2(\theta) & \sin(\theta)\cos(\theta) & \sin^2(\theta) \end{pmatrix}$$

Therefore,

$$\underbrace{\begin{Bmatrix} F_{2XB} \\ F_{2YB} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}}_{\text{Global}} = \mathbf{204216}\underbrace{\begin{pmatrix} 0.307 & -0.462 & -0.307 & 0.462 \\ -0.462 & 0.692 & 0.462 & -0.692 \\ -0.307 & 0.462 & 0.307 & -0.462 \\ 0.462 & -0.692 & -0.462 & 0.692 \end{pmatrix}}_{\text{Global Stiffness Matrix}} \overbrace{\begin{Bmatrix} X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{Bmatrix}}^{\text{Global}}$$

Now we are ready to assemble the overall global stiffness matrix of the structure.

Summing the two sets of global force-displacement equations:

$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = \begin{pmatrix} 94248 & 70686 & -94248 & -70686 & 0 & 0 \\ 70686 & 53014 & -70686 & -53014 & 0 & 0 \\ -94248 & -70686 & 157083 & -23568 & -62836 & 94253 \\ -70686 & -53014 & -23568 & 194395 & 94253 & -141380 \\ 0 & 0 & -62836 & 94253 & 62836 & -94253 \\ 0 & 0 & 94253 & -141380 & -94253 & 141380 \end{pmatrix} \begin{Bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{Bmatrix}$$

Next, apply the following known boundary conditions into the system equations:

(a) Node 1 and Node 3 are fixed-points; therefore, any displacement components of these two node-points are zero (X_1, Y_1 and X_3, Y_3).

(b) The only external load is at Node 2: $F_{2x} = 50$ lbs.

Therefore,

$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ 50 \\ 0 \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = \begin{pmatrix} 94248 & 70686 & -94248 & -70686 & 0 & 0 \\ 70686 & 53014 & -70686 & -53014 & 0 & 0 \\ -94248 & -70686 & 157083 & -23568 & -628360 & 94253 \\ -70686 & -53014 & -23568 & 194395 & 94253 & -141380 \\ 0 & 0 & -62836 & 94253 & 62836 & -94253 \\ 0 & 0 & 94253 & -141380 & -94253 & 141380 \end{pmatrix} \begin{Bmatrix} 0 \\ 0 \\ X_2 \\ Y_2 \\ 0 \\ 0 \end{Bmatrix}$$

The two displacements we need to solve are X_2 and Y_2. Let's simplify the above matrix by removing the unaffected/unnecessary columns in the matrix.

$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ 50 \\ 0 \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = \begin{pmatrix} -94248 & -70686 \\ -70686 & -53014 \\ 157083 & -23568 \\ -23568 & 194395 \\ -62836 & 94253 \\ 94253 & -141380 \end{pmatrix} \begin{Bmatrix} X_2 \\ Y_2 \end{Bmatrix}$$

Solve for nodal displacements X_2 and Y_2:

$$\begin{Bmatrix} 50 \\ 0 \end{Bmatrix} = \begin{bmatrix} 157083 & -23568 \\ -23568 & 194395 \end{bmatrix} \begin{Bmatrix} X_2 \\ Y_2 \end{Bmatrix}$$

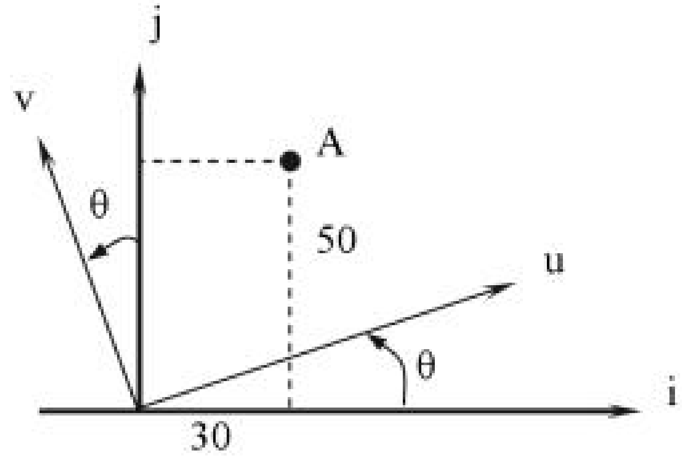
$$\mathbf{X_2 = 3.24\ e^{-4}\ in.}$$
$$\mathbf{Y_2 = 3.93\ e^{-5}\ in.}$$

Substitute the known X_2 and Y_2 values into the matrix and solve for the reaction forces:

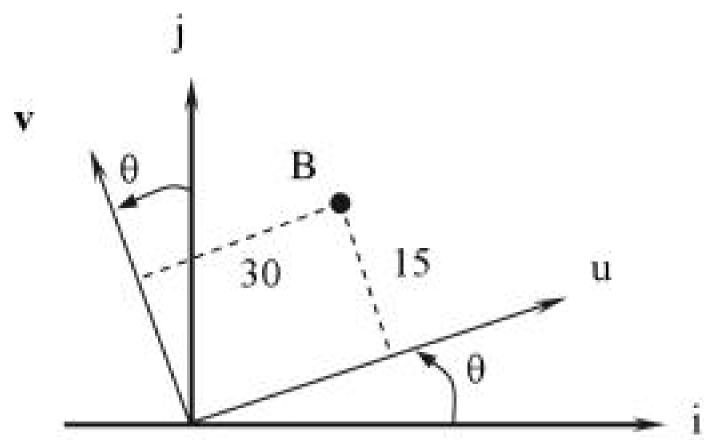
$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = \begin{pmatrix} -94248 & -70686 \\ -70686 & -53014 \\ -62836 & 94253 \\ 94253 & -141380 \end{pmatrix} \begin{Bmatrix} \mathbf{3.24\ e^{-4}} \\ \mathbf{3.93\ e^{-5}} \end{Bmatrix}$$

Questions:

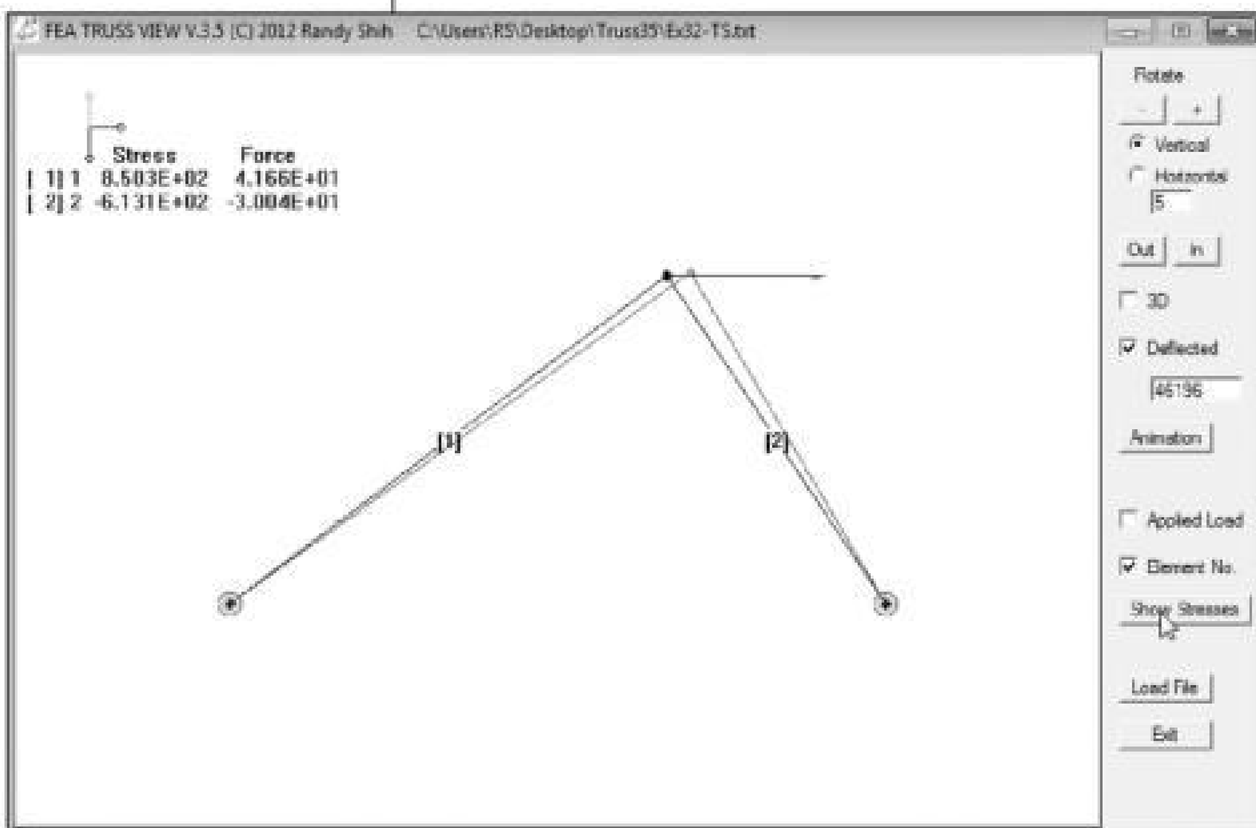
1. Determine the coordinates of point A if the local coordinate system is rotated 15 degrees relative to the global coordinate system. The global coordinates of point A: (30,50).

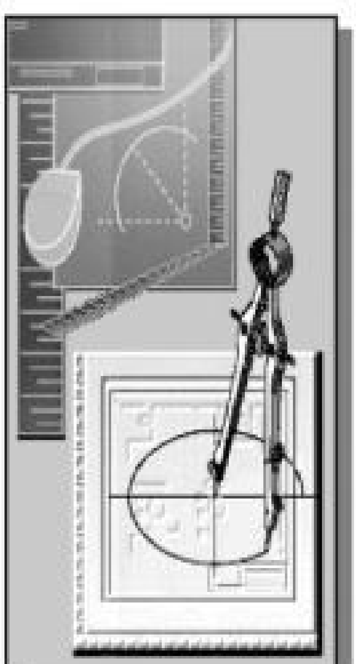


2. Determine the global coordinates of point B if the local coordinate system is rotated 30 degrees relative to the global coordinate system. The local coordinates of point B: (30,15).



Chapter 3
2D Trusses in MS Excel and Truss Solver



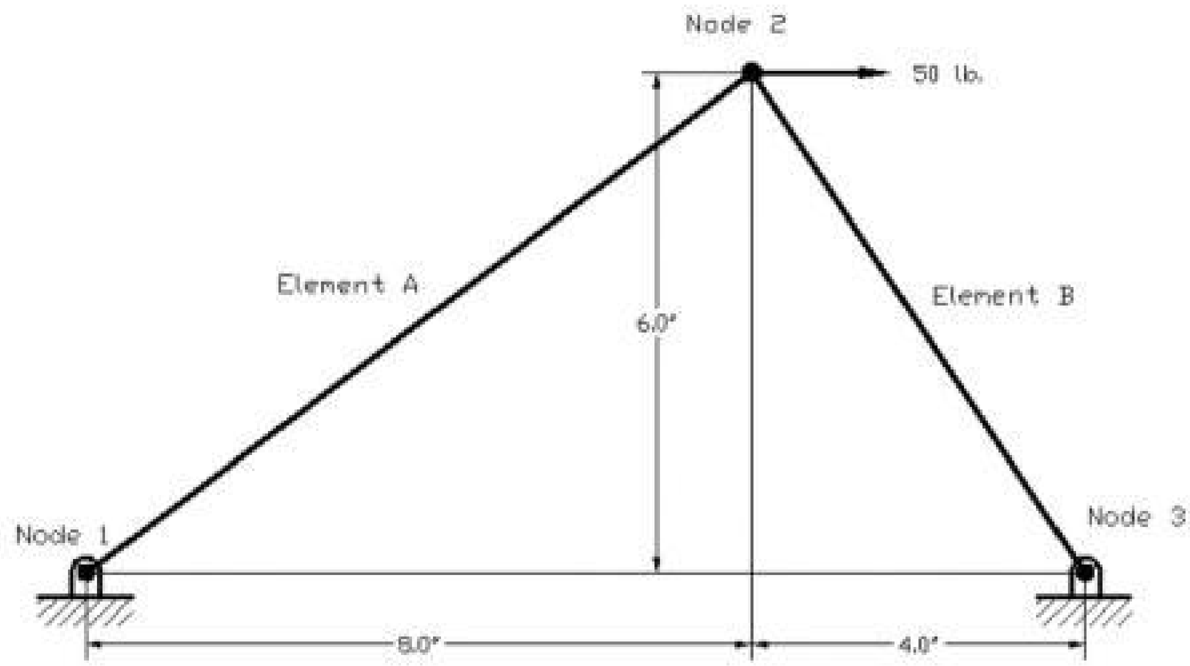


Learning Objectives

- **Perform the Direct Stiffness Matrix Method using MS Excel.**
- **Using MS Excel built-in commands to solve simultaneous equations.**
- **Solve 2D trusses using the Truss Solver program.**
- **Understand the general FEA procedure concepts.**

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

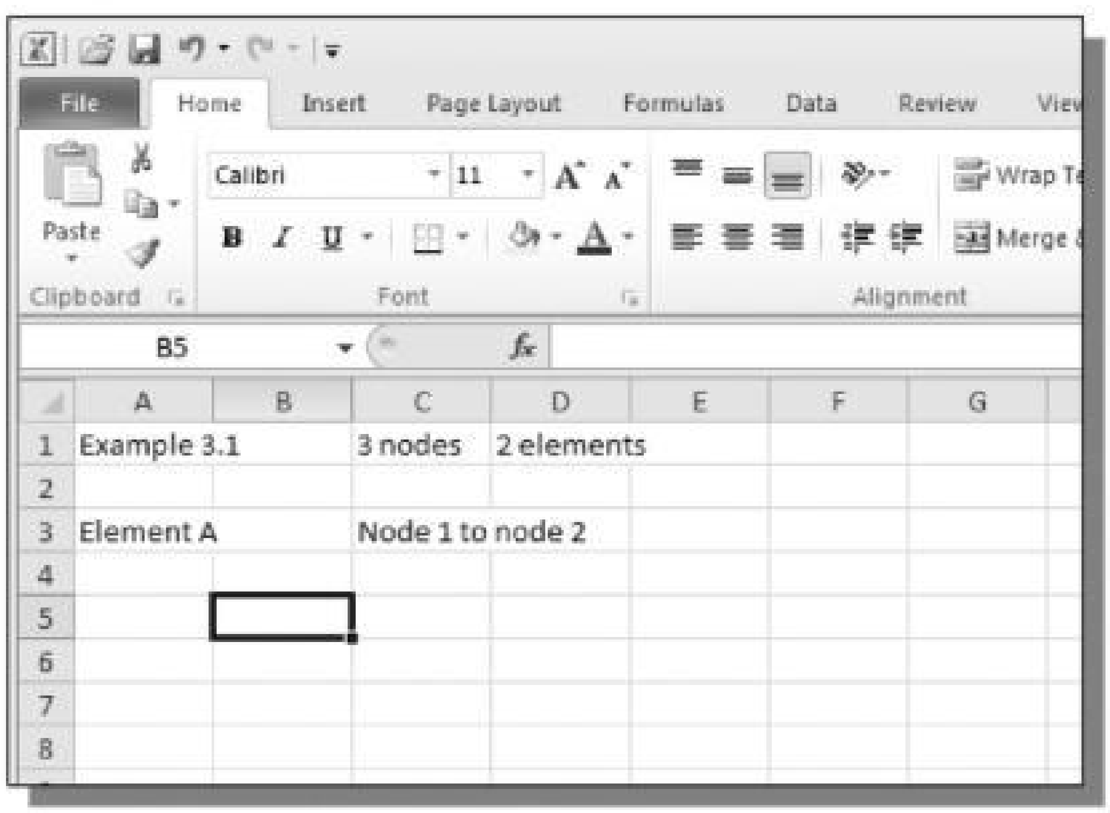
The system contains two elements and three nodes. The nodes and elements are labeled as shown below.



Establish the Global K matrix for each member

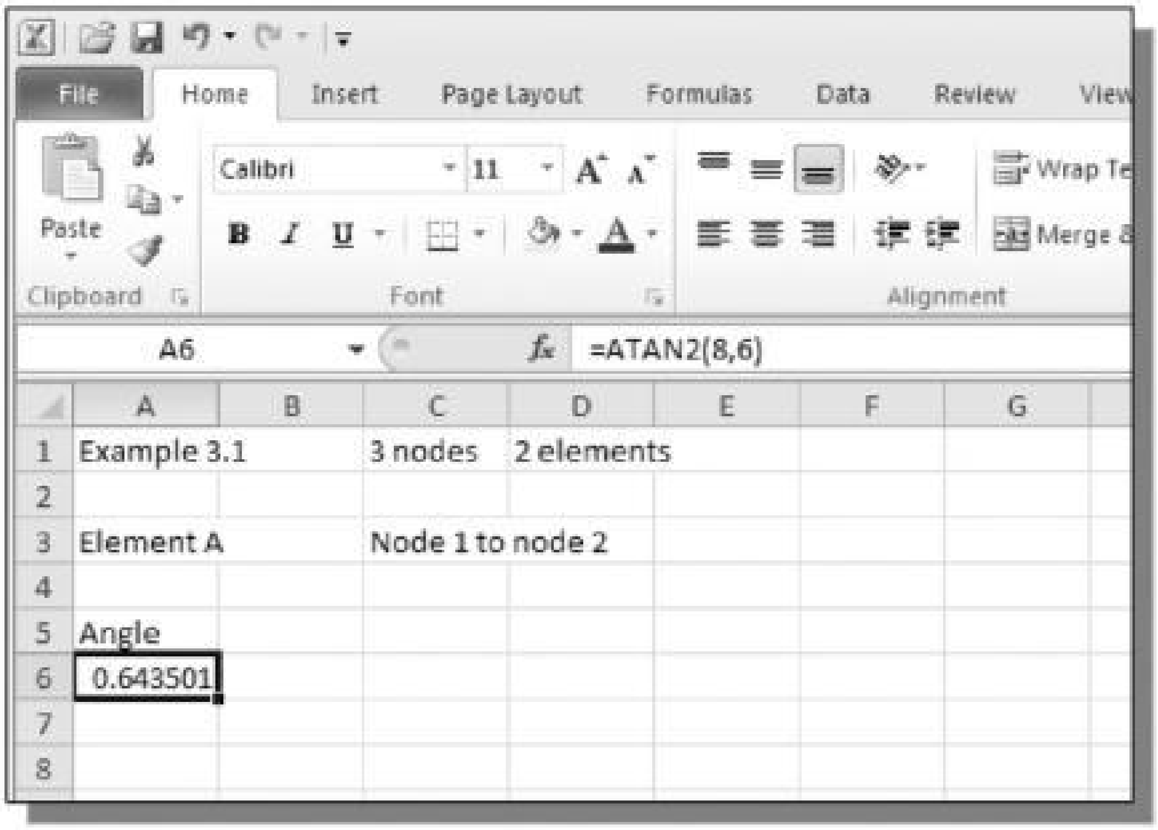
1. Start *Microsoft Excel* through the desktop icon or the *Explorer* toolbar.

2. First enter the general problem information and element labels near the top of the spreadsheet.



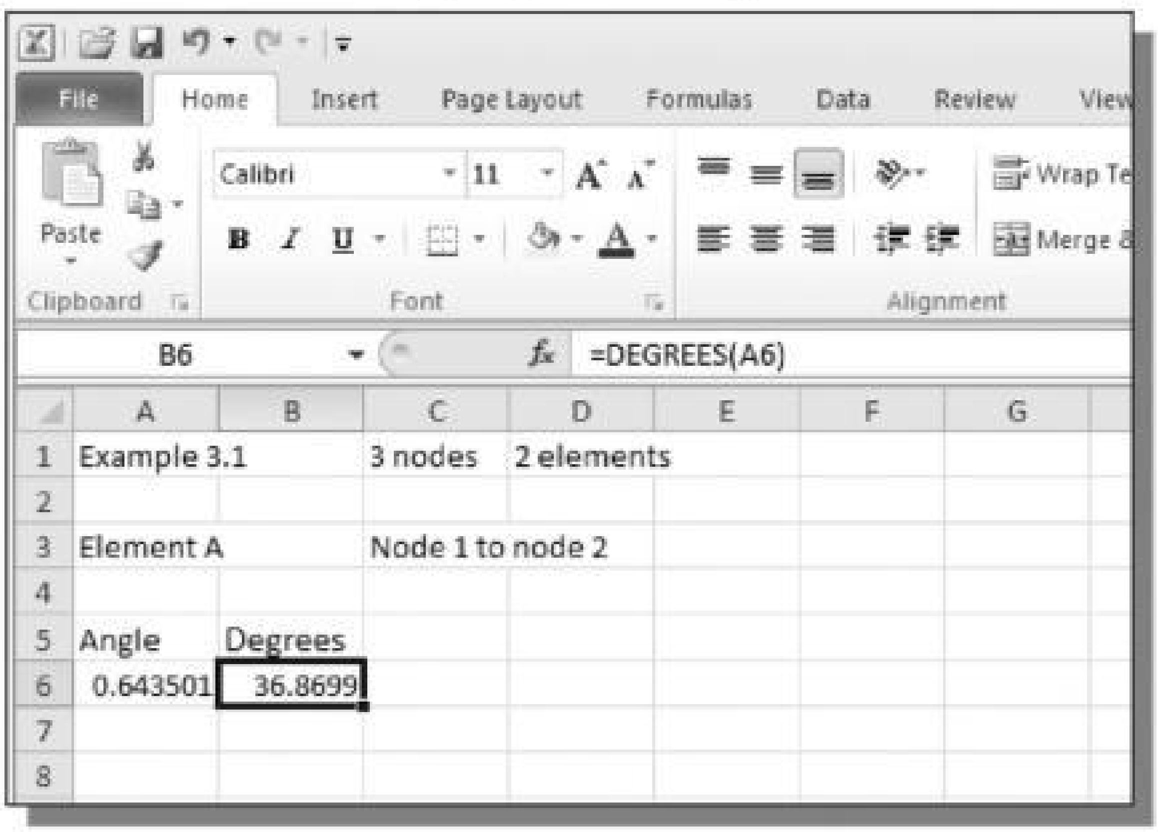
❖ Note that we will be following the same convention as outlined in Example 2.2.

3. To calculate the angle of *Element A*, enter the formula: **=ATAN2(8,6)**; the resulting angle is in **radians**.



❖ Note the equal sign is used to identify the formula; the **ATAN2** function will calculate the arctangent function in all four quadrants.

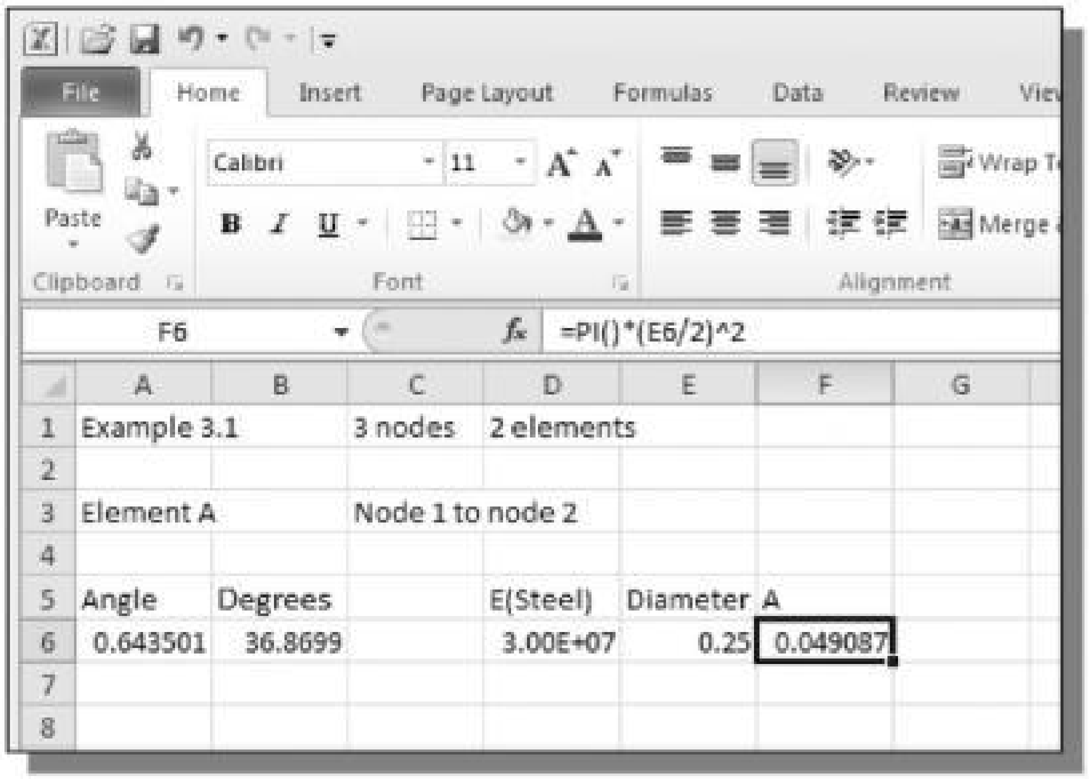
4. To convert the radians to degrees, use the **Degrees** function as shown.



❖ Note that the conversion from radians to degrees can also be calculated using the following equation:

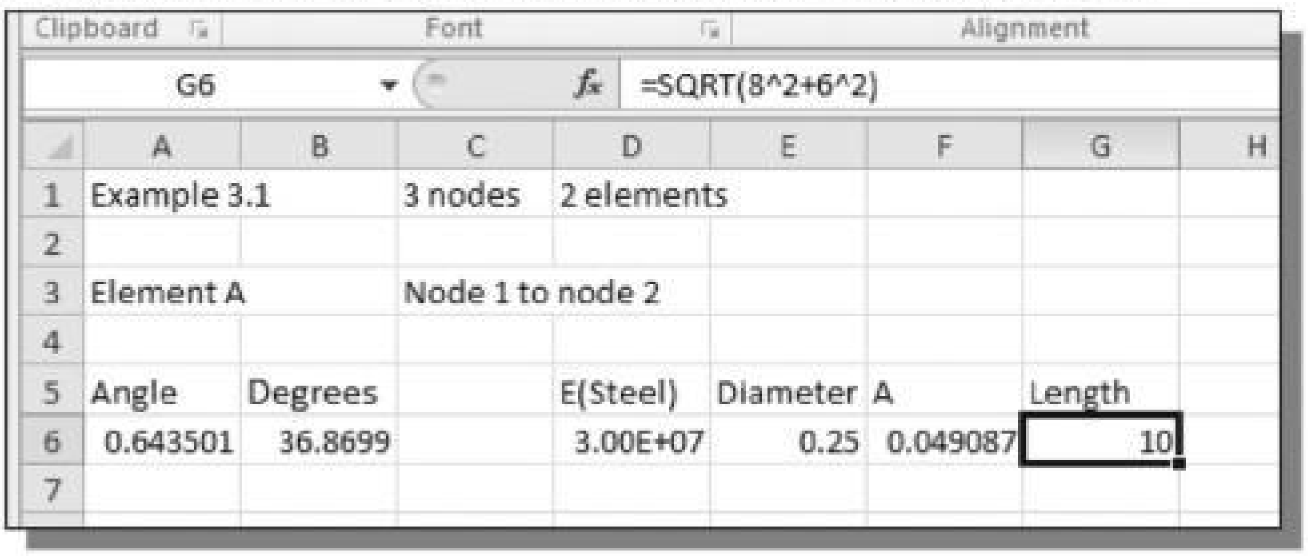
$$\textbf{Degrees} = \textbf{Radians} \times 180/\pi$$

5. Next enter the modulus of elasticity (E), and cross-section area based on the diameter of the truss member.



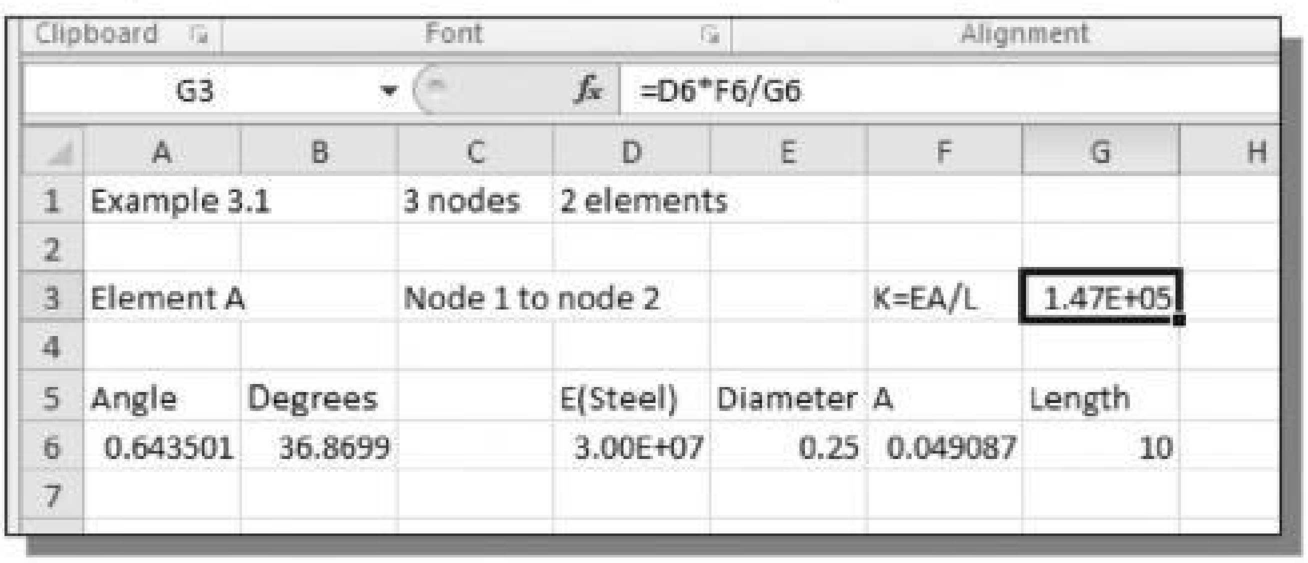
❖ Note the PI() function gives the π value in *Excel*.

6. To calculate the length of the member, enter the formula as shown.

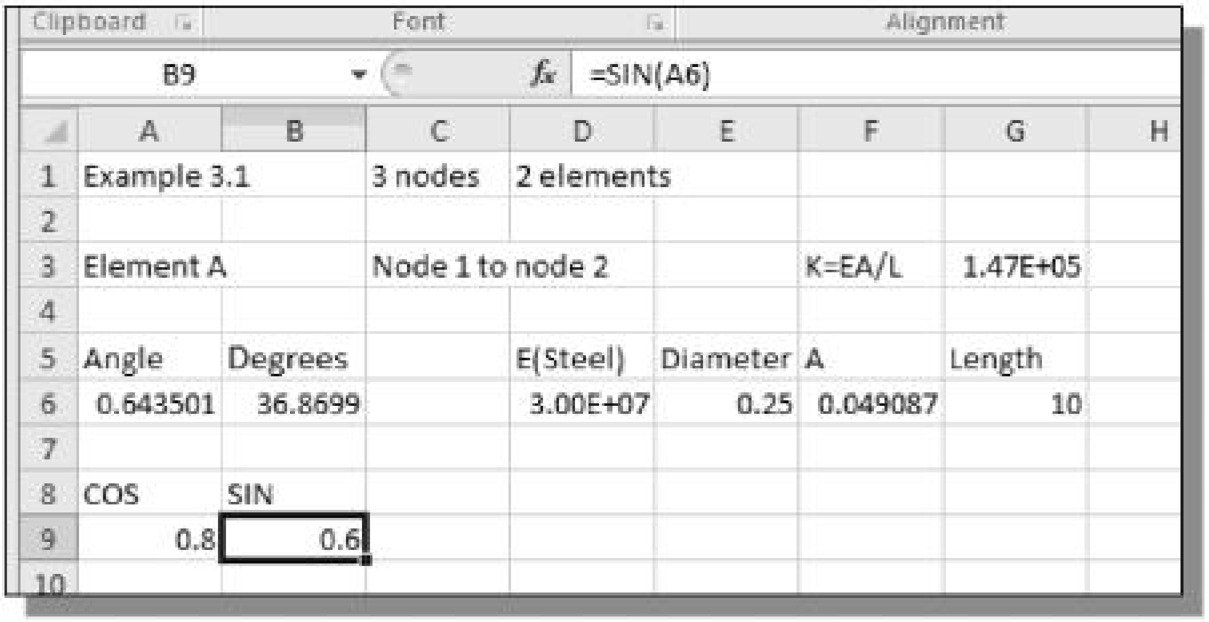


❖ The **SQRT** function can be used to calculate the square root value.

7. Now, we can calculate the k value. Note the equation is **k=EA/L**.



8. Next calculate the cosine and sine of the radian angle as shown.



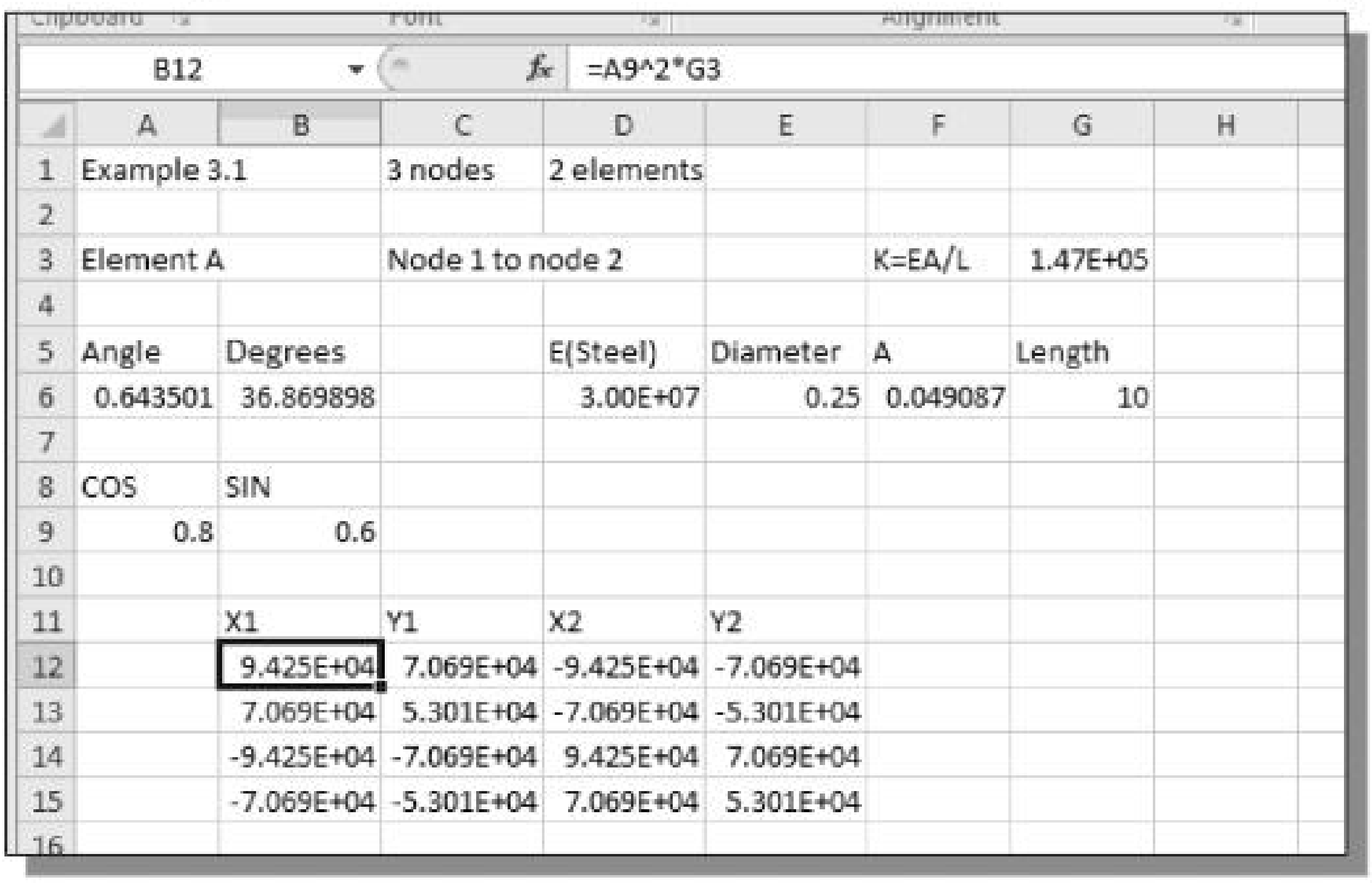
Clipboard | Font | Alignment

B9 =SIN(A6)

	A	B	C	D	E	F	G	H
1	Example 3.1		3 nodes	2 elements				
2								
3	Element A		Node 1 to node 2			K=EA/L	1.47E+05	
4								
5	Angle	Degrees		E(Steel)	Diameter	A	Length	
6	0.643501	36.8699		3.00E+07	0.25	0.049087	10	
7								
8	COS	SIN						
9	0.8	0.6						
10								

9. Next calculate the 4x4 global K matrix using the cosine and sine values.

$$[K] = \frac{EA}{L}\begin{pmatrix} \cos^2(\theta) & \cos(\theta)\sin(\theta) & -\cos^2(\theta) & -\cos(\theta)\sin(\theta) \\ \cos(\theta)\sin(\theta) & \sin^2(\theta) & -\cos(\theta)\sin(\theta) & -\sin^2(\theta) \\ -\cos^2(\theta) & -\cos(\theta)\sin(\theta) & \cos^2(\theta) & \cos(\theta)\sin(\theta) \\ -\cos(\theta)\sin(\theta) & -\sin^2(\theta) & \sin(\theta)\cos(\theta) & \sin^2(\theta) \end{pmatrix}$$



Clipboard | Font | Alignment

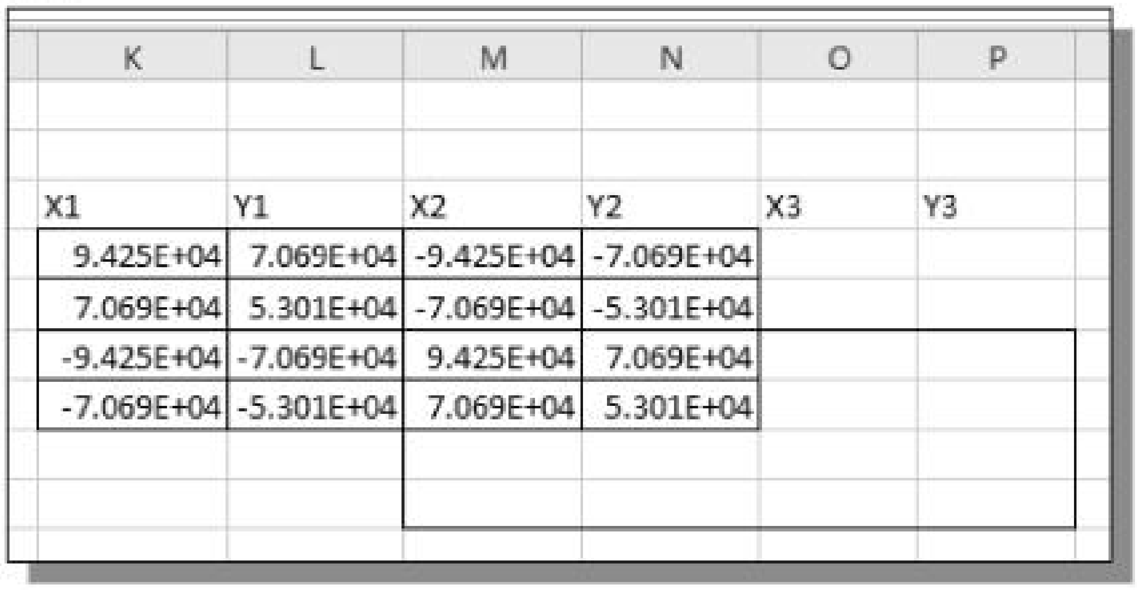
B12 =A9^2*G3

	A	B	C	D	E	F	G	H
1	Example 3.1		3 nodes	2 elements				
2								
3	Element A		Node 1 to node 2			K=EA/L	1.47E+05	
4								
5	Angle	Degrees		E(Steel)	Diameter	A	Length	
6	0.643501	36.869898		3.00E+07	0.25	0.049087	10	
7								
8	COS	SIN						
9	0.8	0.6						
10								
11		X1	Y1	X2	Y2			
12		9.425E+04	7.069E+04	-9.425E+04	-7.069E+04			
13		7.069E+04	5.301E+04	-7.069E+04	-5.301E+04			
14		-9.425E+04	-7.069E+04	9.425E+04	7.069E+04			
15		-7.069E+04	-5.301E+04	7.069E+04	5.301E+04			
16								

K_{11}=k(cos(θ))^2, K_{12}=kcos(θ)sin(θ),
K_{13}=-k(cos(θ))^2, K_{14}=-kcos(θ)sin(θ),
K_{21}=kcos(θ)sin(θ), K_{22}=k(sin(θ))^2........

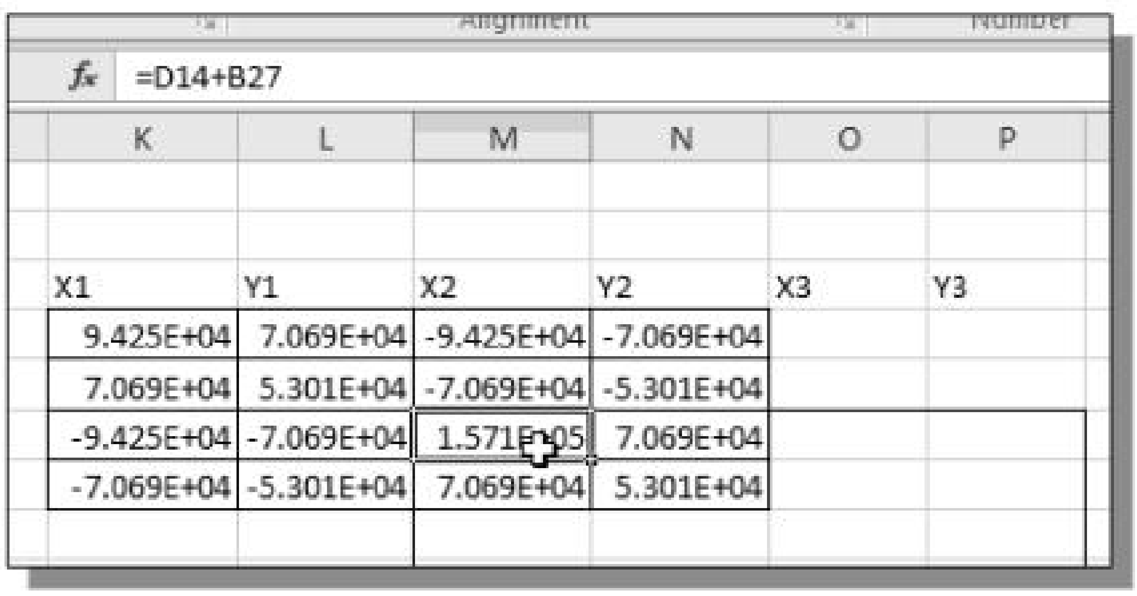
❖ On your own, compare the calculated values to the numbers shown on page 2-15.

3. Repeat the last step to fill in the first rectangle, which are simply referencing the cells in the *Element A* matrix. (For the cell locations, refer to the image shown on page 3-6.)



K	L	M	N	O	P
X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04		
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04		
-9.425E+04	-7.069E+04	9.425E+04	7.069E+04		
-7.069E+04	-5.301E+04	7.069E+04	5.301E+04		

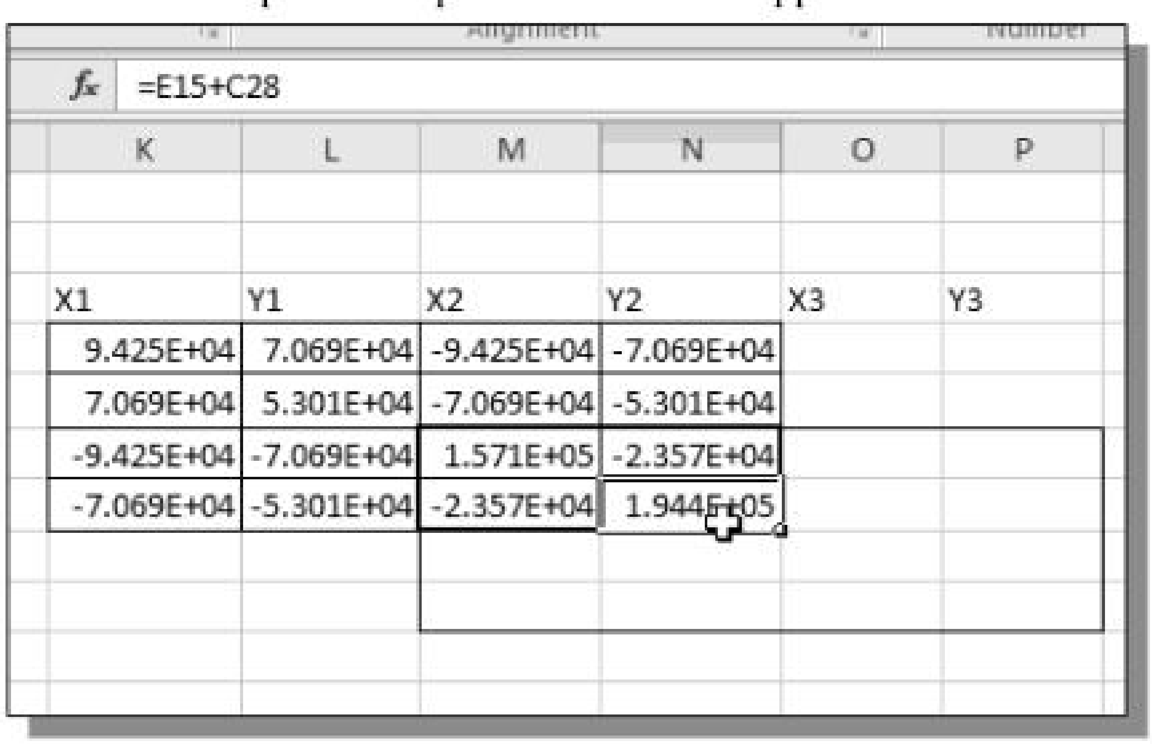
4. Edit the third cell in the X2 column; this is where the first overlap occurs. Edit the cell, so that it includes the first cell of the *Element B* matrix: **=D14+B27**



f_x =D14+B27

K	L	M	N	O	P
X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04		
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04		
-9.425E+04	-7.069E+04	1.571E+05	7.069E+04		
-7.069E+04	-5.301E+04	7.069E+04	5.301E+04		

5. Repeat the above step and complete the four overlapped cells as shown.



f_x =E15+C28

K	L	M	N	O	P
X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04		
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04		
-9.425E+04	-7.069E+04	1.571E+05	-2.357E+04		
-7.069E+04	-5.301E+04	-2.357E+04	1.944E+05		

6. Fill in the rest of the second rectangle by referencing to the *Element B* matrix; the results should appear as shown.

f_x =E27

K	L	M	N	O	P
X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04		
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04		
-9.425E+04	-7.069E+04	1.571E+05	-2.357E+04	-6.284E+04	9.425E+04
-7.069E+04	-5.301E+04	-2.357E+04	1.944E+05	9.425E+04	-1.414E+05
		-6.284E+04	9.425E+04	6.284E+04	-9.425E+04
		9.425E+04	-1.414E+05	-9.425E+04	1.414E+05

7. Enter eight **zeros** in the remaining blank cells. Note that these zeros are necessary for the calculations of the reaction forces, which we will do once the global displacements are calculated.

f_x 0

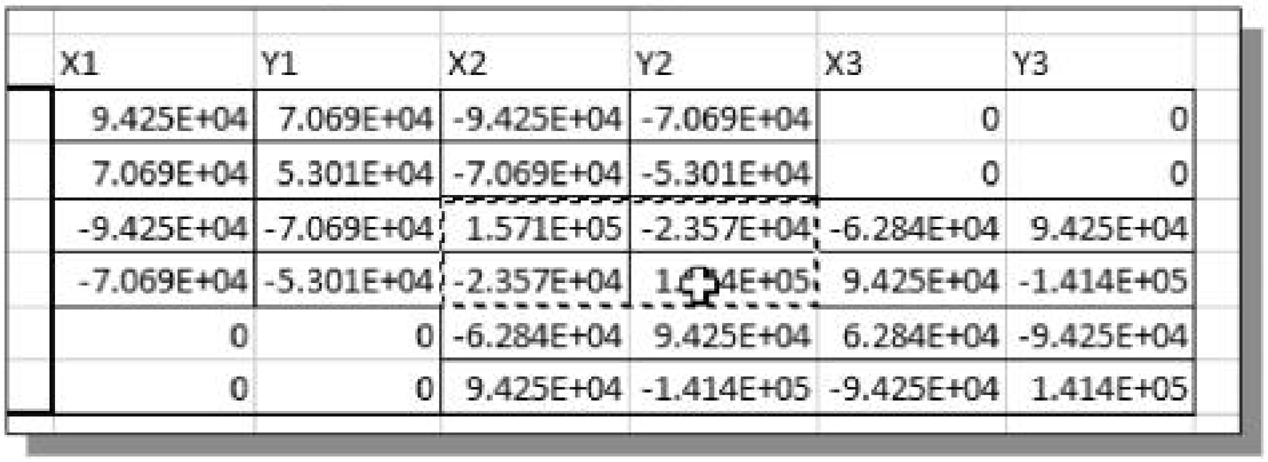
K	L	M	N	O	P
X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04	0	0
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04	0	0
-9.425E+04	-7.069E+04	1.571E+05	-2.357E+04	-6.284E+04	9.425E+04
-7.069E+04	-5.301E+04	-2.357E+04	1.944E+05	9.425E+04	-1.414E+05
0	0	-6.284E+04	9.425E+04	6.284E+04	-9.425E+04
0	0	9.425E+04	-1.414E+05	-9.425E+04	1.414E+05

Solving the Global Displacements

1. Label the global displacements to the right of the overall global matrix; enter the four zeros as shown. (Node 1 and Node 3 are fixed points.)

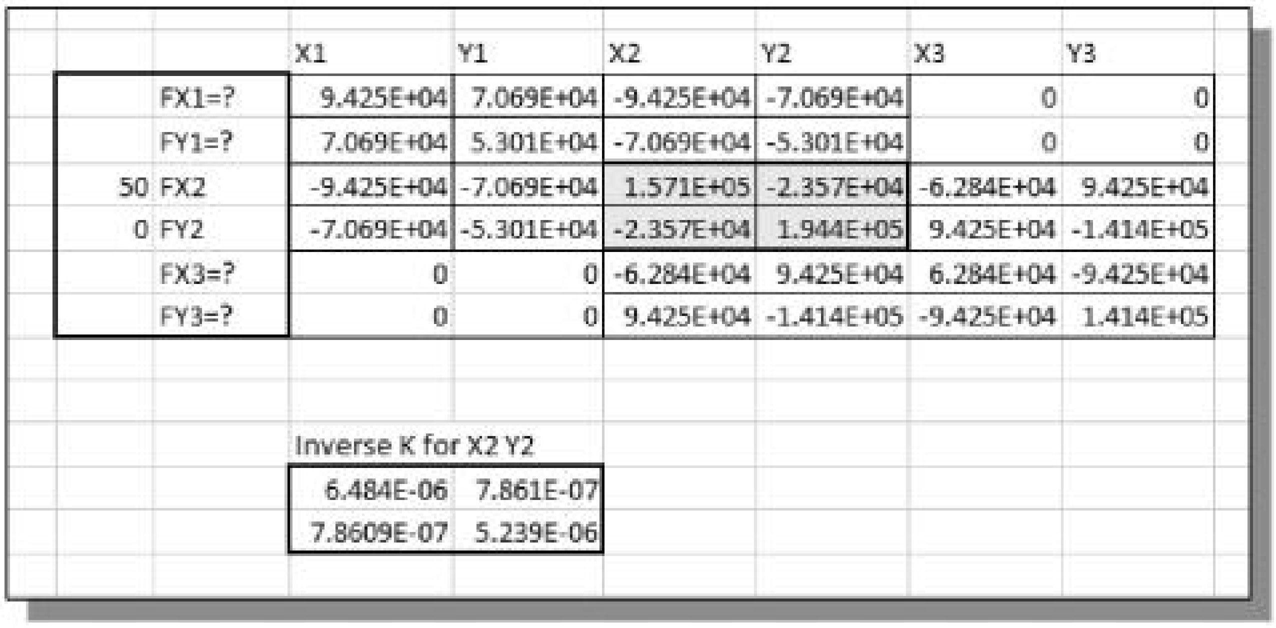
	X3	Y3			
04	0	0		X1	0
04	0	0		Y1	0
04	-6.284E+04	9.425E+04		X2=?	
05	9.425E+04	-1.414E+05		Y2=?	
04	6.284E+04	-9.425E+04		X3	0
05	-9.425E+04	1.414E+05		Y3	0

9. Select the 2x2 array as shown in the figure below.



X1	Y1	X2	Y2	X3	Y3
9.425E+04	7.069E+04	-9.425E+04	-7.069E+04	0	0
7.069E+04	5.301E+04	-7.069E+04	-5.301E+04	0	0
-9.425E+04	-7.069E+04	1.571E+05	-2.357E+04	-6.284E+04	9.425E+04
-7.069E+04	-5.301E+04	-2.357E+04	1[illegible]4E+05	9.425E+04	-1.414E+05
0	0	-6.284E+04	9.425E+04	6.284E+04	-9.425E+04
0	0	9.425E+04	-1.414E+05	-9.425E+04	1.414E+05

10. It is important to note that when using the matrix commands in *Excel*, we must press **CTRL+SHIFT+ENTER** to perform the calculations. The Inverse K is calculated as shown in the figure below.

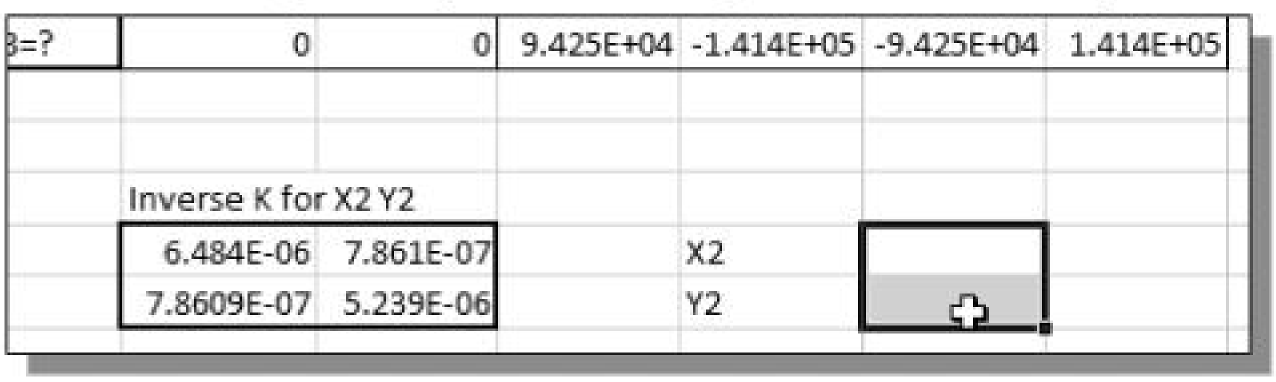


		X1	Y1	X2	Y2	X3	Y3
	FX1=?	9.425E+04	7.069E+04	-9.425E+04	-7.069E+04	0	0
	FY1=?	7.069E+04	5.301E+04	-7.069E+04	-5.301E+04	0	0
50	FX2	-9.425E+04	-7.069E+04	1.571E+05	-2.357E+04	-6.284E+04	9.425E+04
0	FY2	-7.069E+04	-5.301E+04	-2.357E+04	1.944E+05	9.425E+04	-1.414E+05
	FX3=?	0	0	-6.284E+04	9.425E+04	6.284E+04	-9.425E+04
	FY3=?	0	0	9.425E+04	-1.414E+05	-9.425E+04	1.414E+05

Inverse K for X2 Y2

6.484E-06	7.861E-07
7.8609E-07	5.239E-06

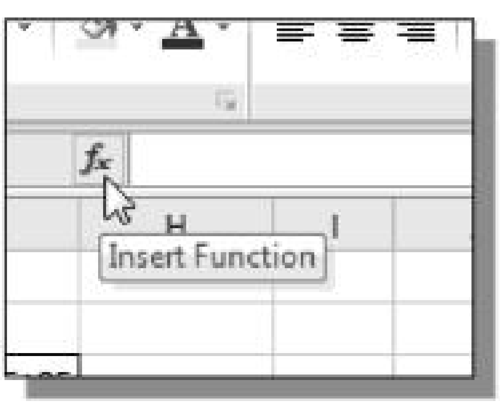
11. Label the two global displacements, and pre-select the 2x1 array as shown.



3=?	0	0	9.425E+04	-1.414E+05	-9.425E+04	1.414E+05

Inverse K for X2 Y2

6.484E-06	7.861E-07	X2	
7.8609E-07	5.239E-06	Y2	



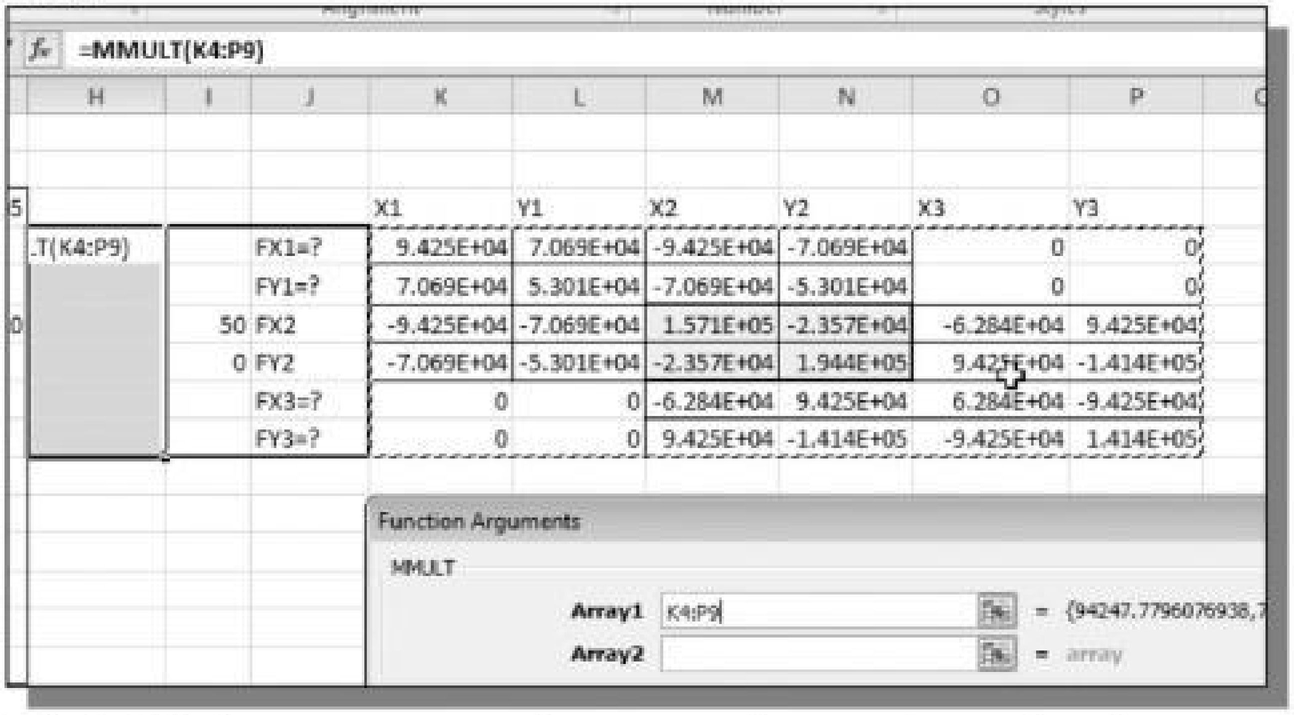
12. Click on the **Insert Function** icon that is located in front of the input edit box as shown in the figure

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

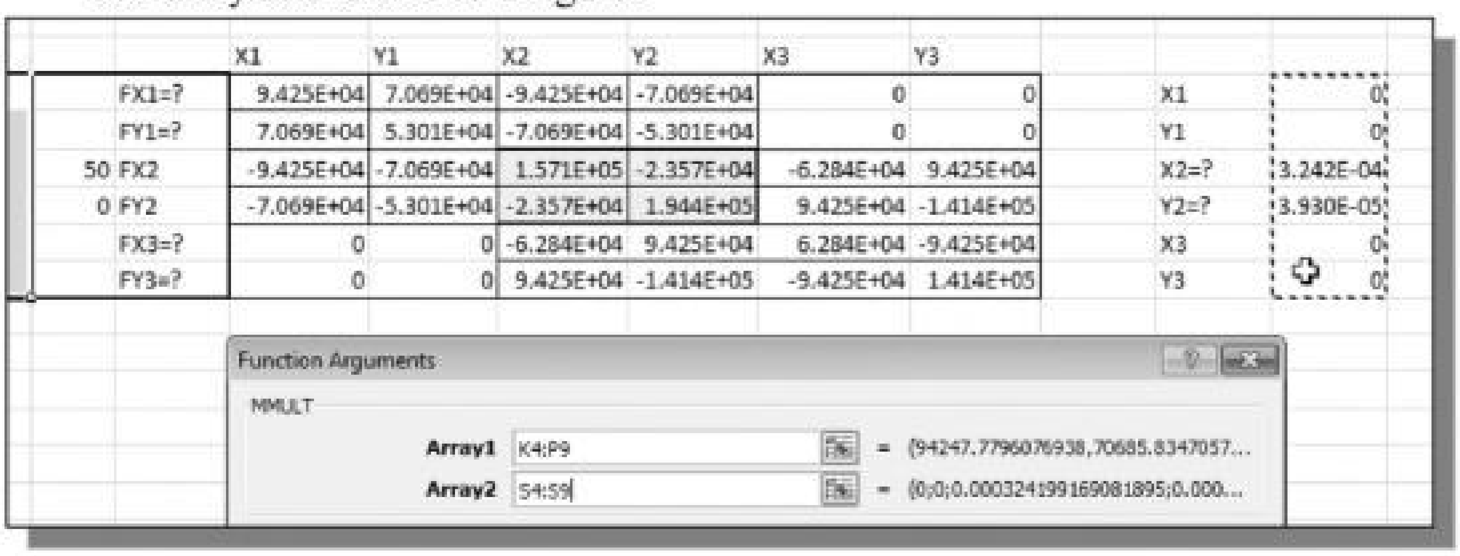
5. The ***Function Arguments*** dialog box appears on the screen with a brief description of the function. Note that two arrays are required for this function.

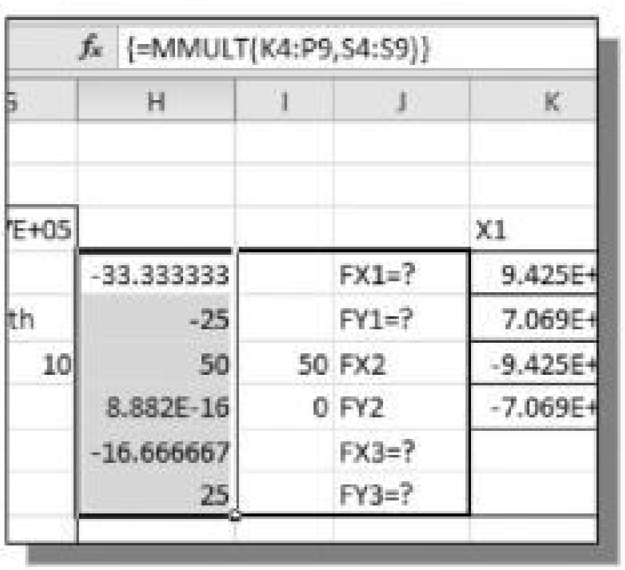
6. Select the **overall global [k] matrix**, which is the 6x6 array, as shown as the first array.



7. Hit the **Tab** key once to move the cursor to the second array input box.

8. Select the **overall global displacement matrix** as the second array. This is the 6x1 array as shown in the figure.

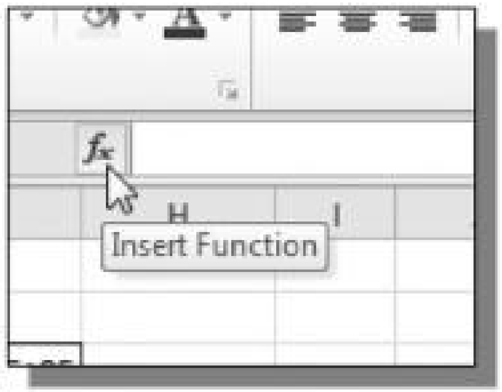




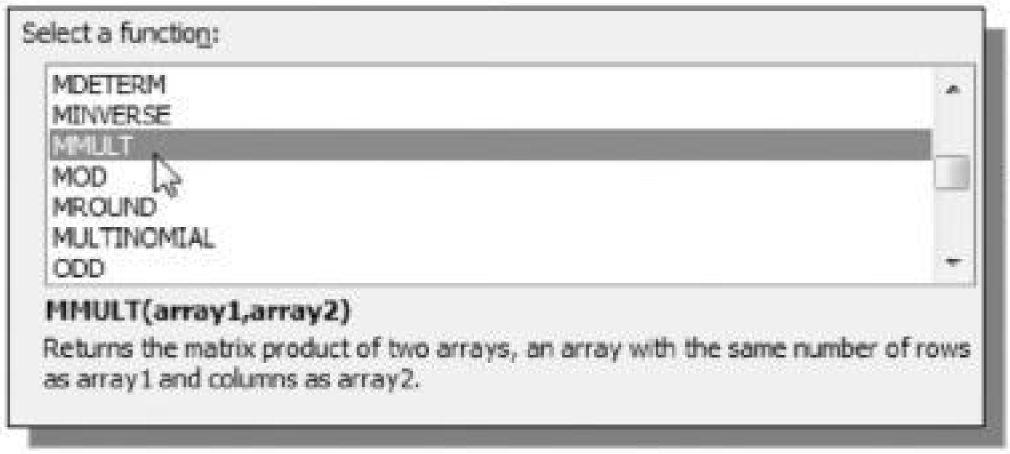
9. It is important to note that when using the matrix commands in *Excel*, we must press **CTRL+SHIFT+ENTER** to perform the calculations. The global forces are calculated as shown in the figure.

❖ Note this calculation also provides a quick check against the initial forces at node 2. The small FY2 represents the rounding error that exists in computer software.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



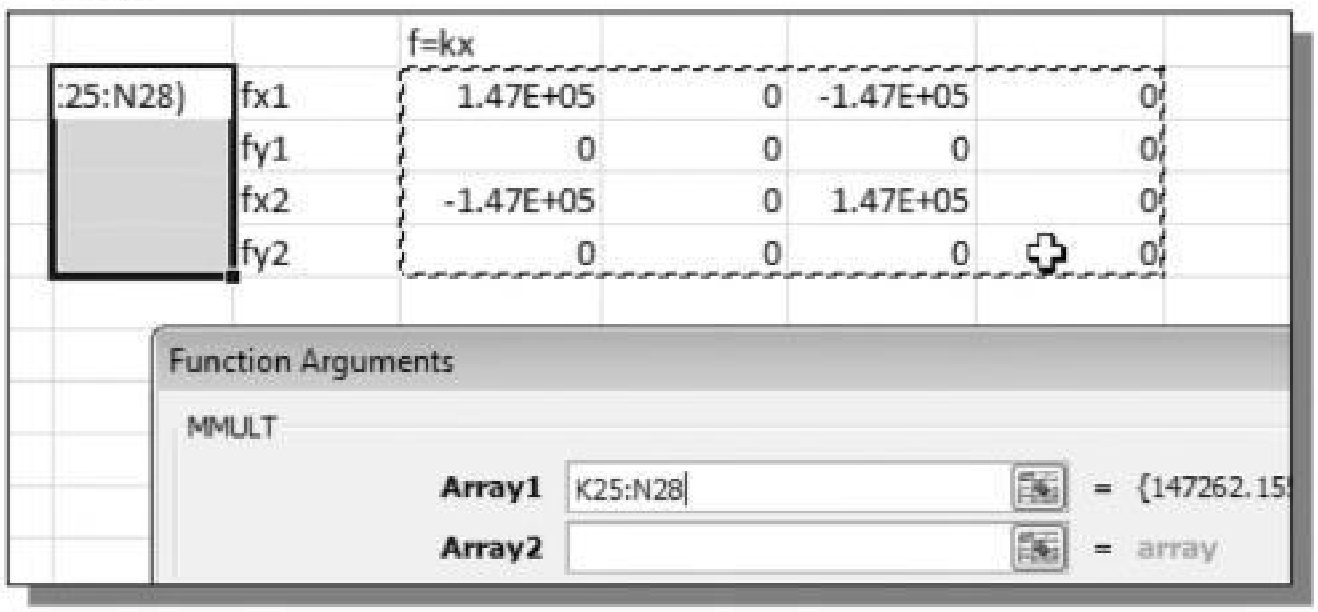
12. Click on the **Insert Function** icon that is located in front of the input edit box as shown in the figure.



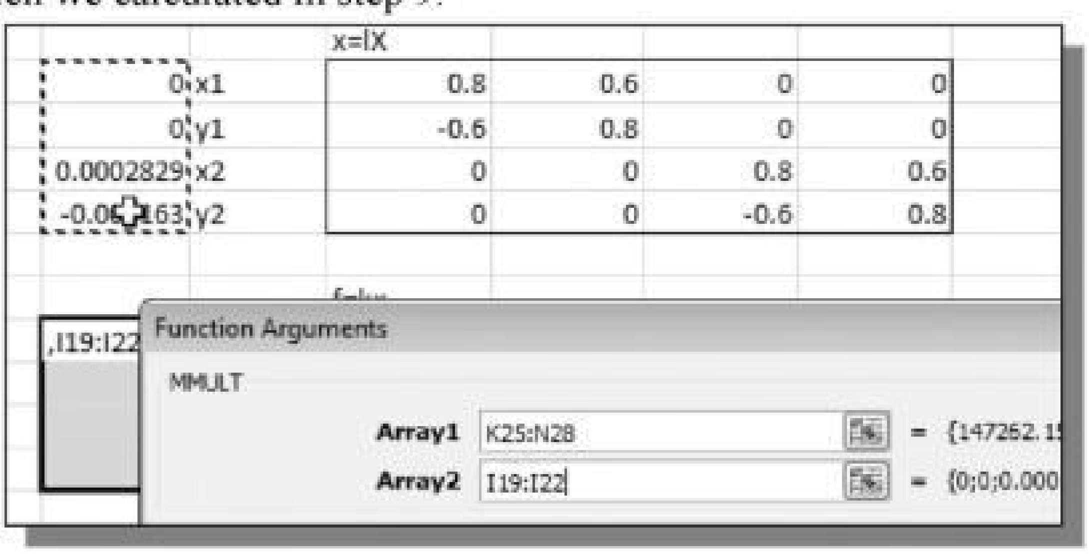
13. Select the **MMULT** command as shown.

14. Click **OK** to proceed with the **MMULT** command.

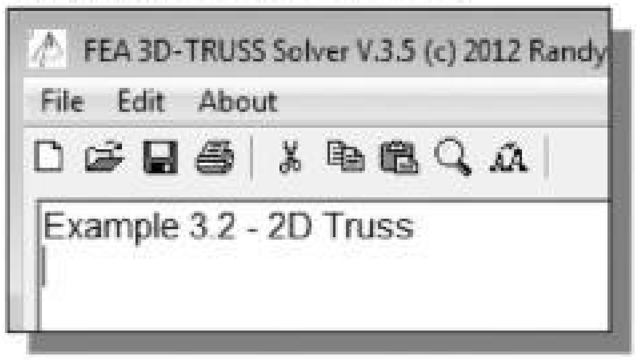
15. Select the **local k matrix**, which is the 4x4 array, which we just set up as the first array.



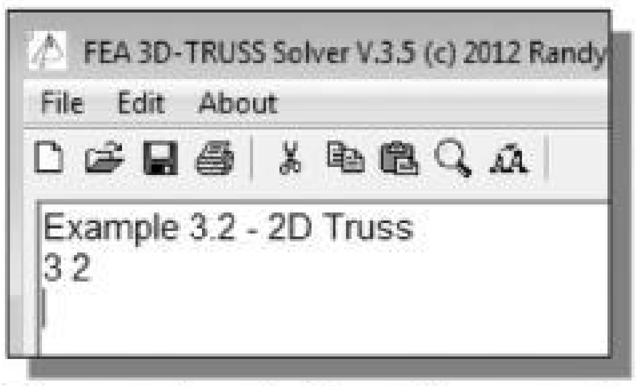
16. Select the **local displacement matrix**, as the second array. This is the 4x1 array which we calculated in step 9.



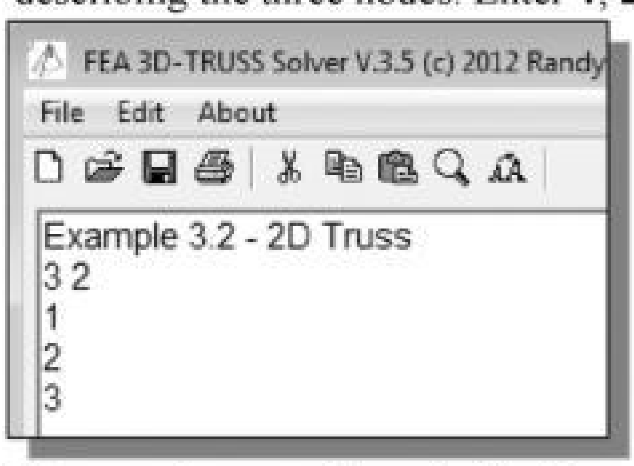
6. The first line is a comment line, which can be used to describe any general information about the truss system being solved. For this example, enter: **Example 3.2, 2D truss**.



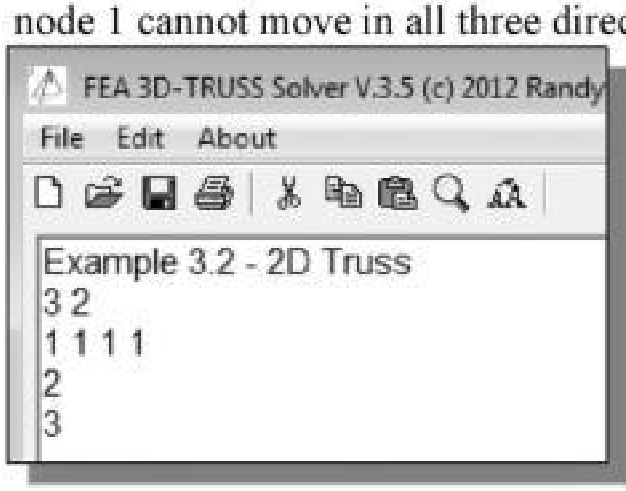
7. The second line contains two numbers: the number of nodes and the number of elements in the system. For our example, we enter **3 2** (the two numbers are separated by a space) to indicate the system has 3 nodes and 2 elements.



8. Since we have indicated there are three nodes, the next three lines will be describing the three nodes. Enter **1**, **2** and **3** as the first number of the three lines.

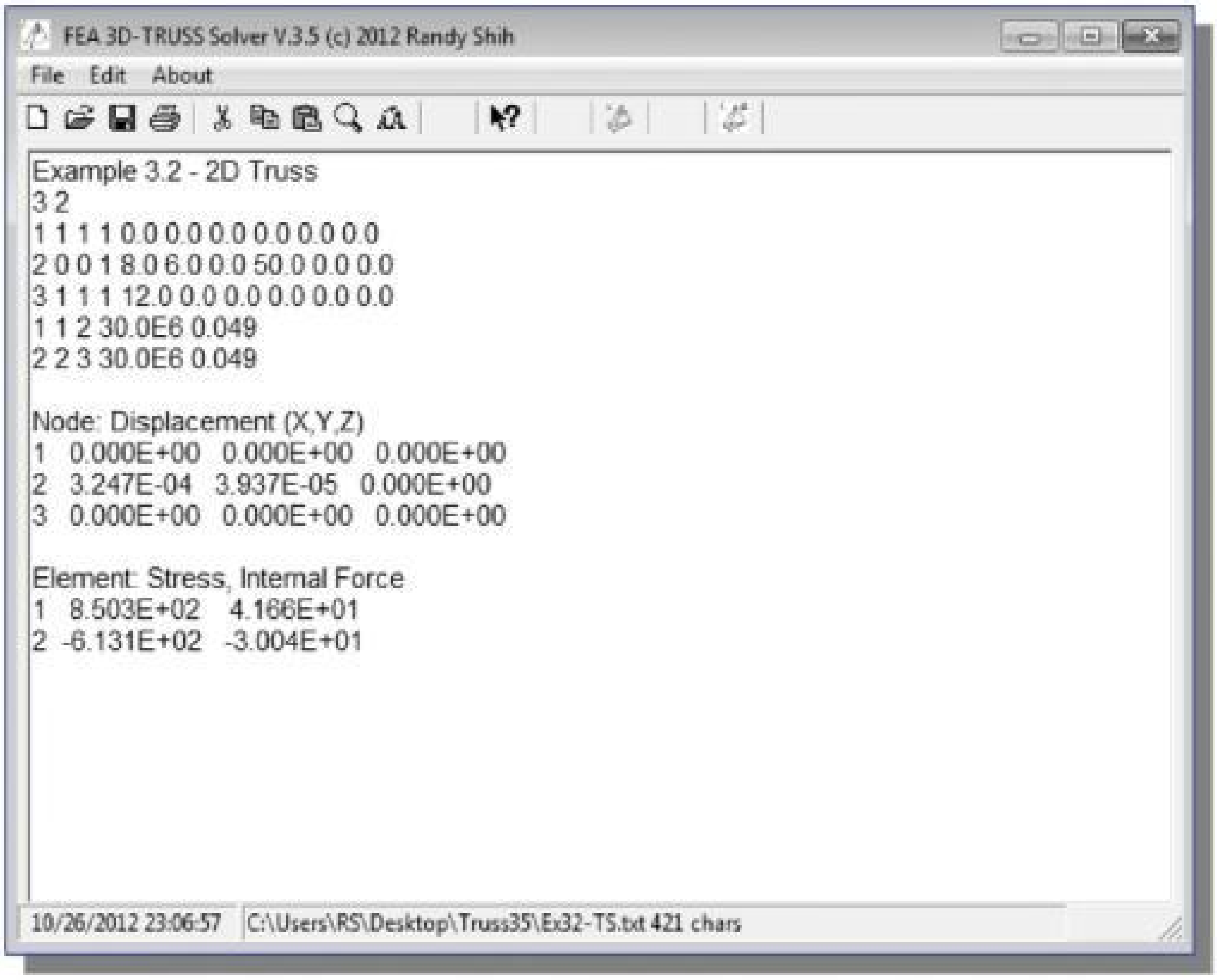


9. The next three numbers, behind the node number, are used to identify the restraints of the node in X, Y and Z directions. Use 1 to indicate it cannot move in that direction; use 0 if it is free to move in that direction. Enter **1 1 1** to indicate node 1 cannot move in all three directions.

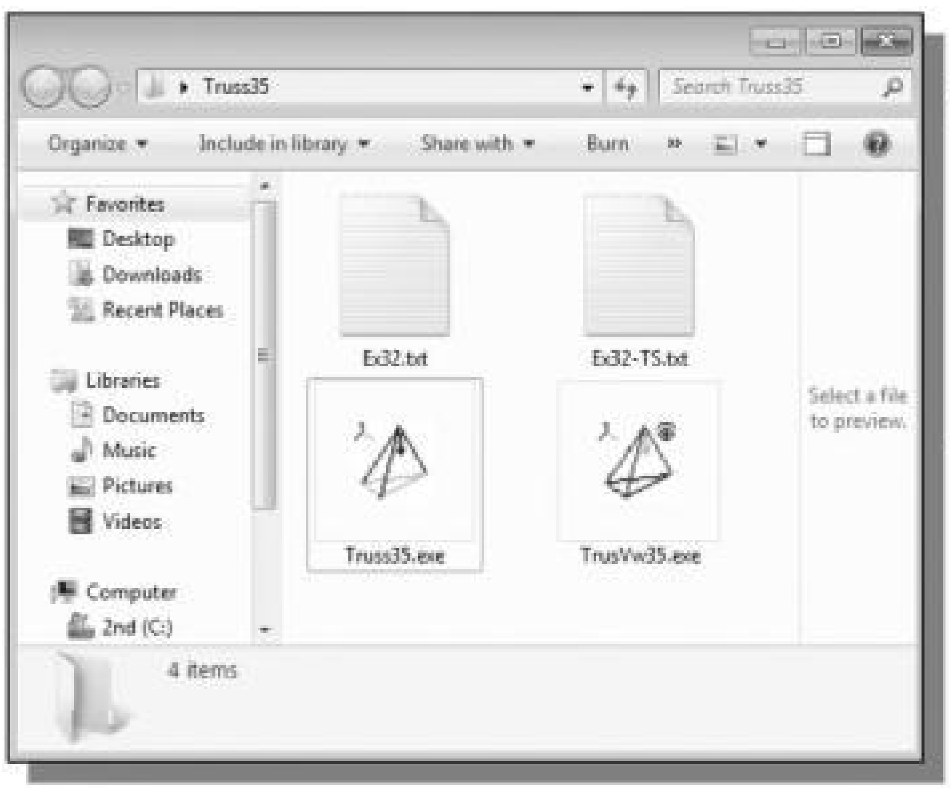


You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

21. The *Truss Solver* will then process the data and the solution is displayed inside the built-in editor.



22. The solution contains two sections: (1) displacements of the nodes in X, Y and Z directions, and (2) the stresses and internal forces of the elements.



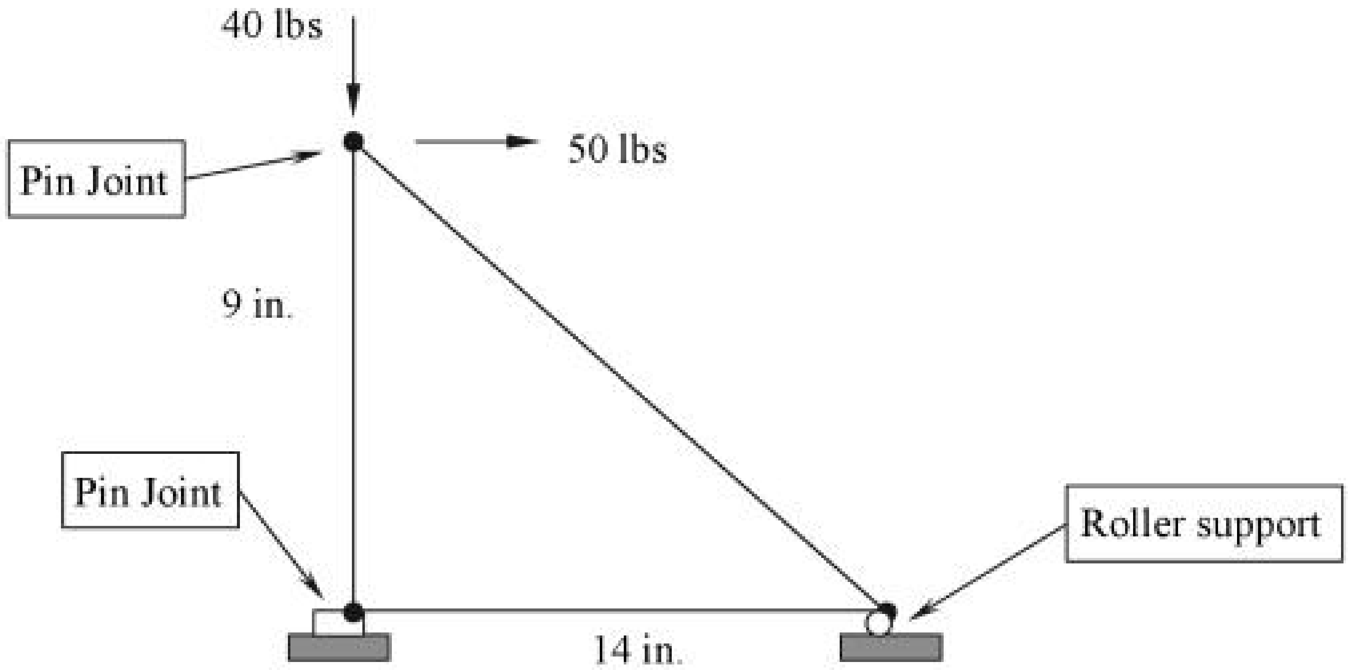
➤ Note the results of the *Truss Solver* are also stored in the *Ex32-TS.txt* file. Same filename as the input file, with a different filename extension.

❖ Also note that both *Ex32.txt* and *Ex32-TS.txt* are plain text files, which can be opened with any text editor, such as *Windows Notepad*.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Exercises: Solve the following problems using *MS Excel* and the *Truss Solver* program.

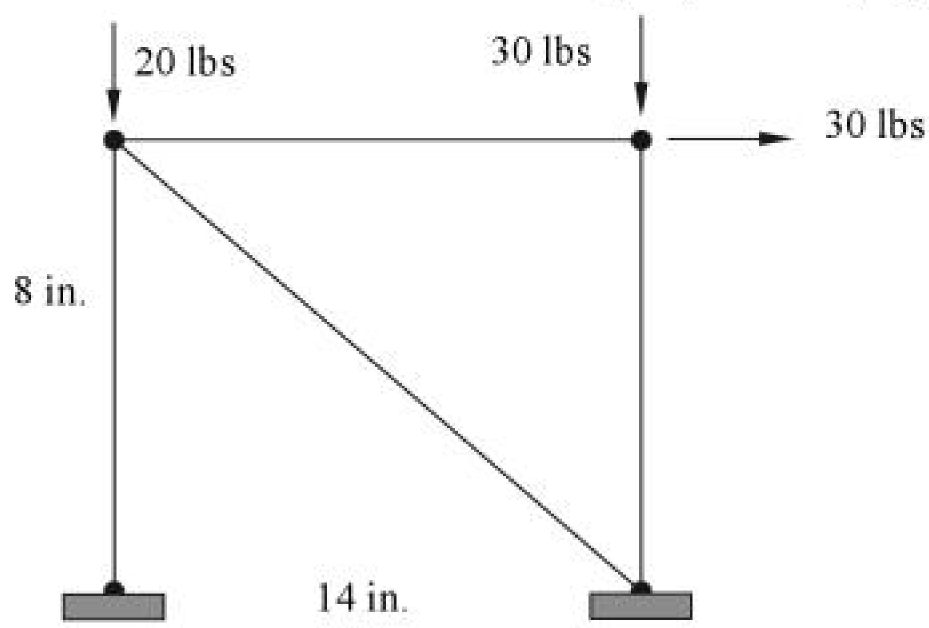
1. Given: two-dimensional truss structure as shown.



Material: Steel, diameter ¼ in.

Find: (a) Displacements of the nodes.
(b) Normal stresses developed in the members.

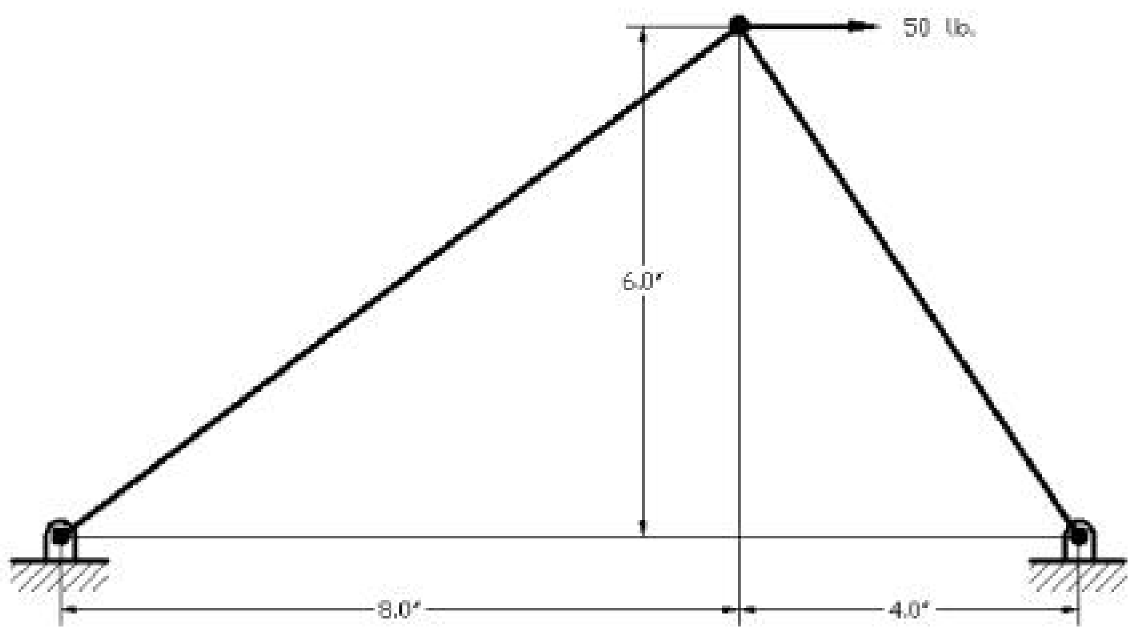
2. Given: Two-dimensional truss structure as shown (All joints are pin joints).



Material: Steel, diameter ¼ in.

Find: (a) Displacements of the nodes.
(b) Normal stresses developed in the members.

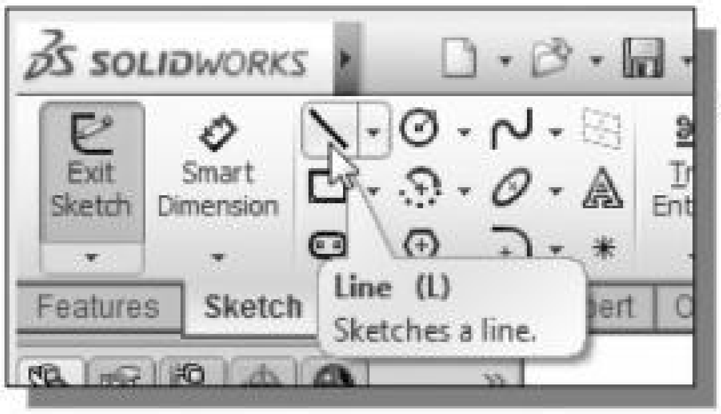
In this chapter, we will examine the process of creating, solving and viewing the results of a finite element analysis on a two-dimensional truss structure using the FEA application software *SolidWorks Simulation*, which is available in *SolidWorks*. Note that several options are available in *SolidWorks* to create truss/beam systems. The same 2D truss system that was analyzed in the previous chapter will be used as the first FEA example.



To create trusses and beams models in *SolidWorks*, two different approaches are available. We can either: (1) create the truss and beam members using the standard parametric **solid modeling tools**, an approach which treats the members the same as any other solid model; or (2) use the **Structural Members** tool specifically designed for creating long and slender members. In this chapter, we will create and analyze the truss system using the solid modeling approach. The **Structural Members** tool approach will be demonstrated in the next chapter.

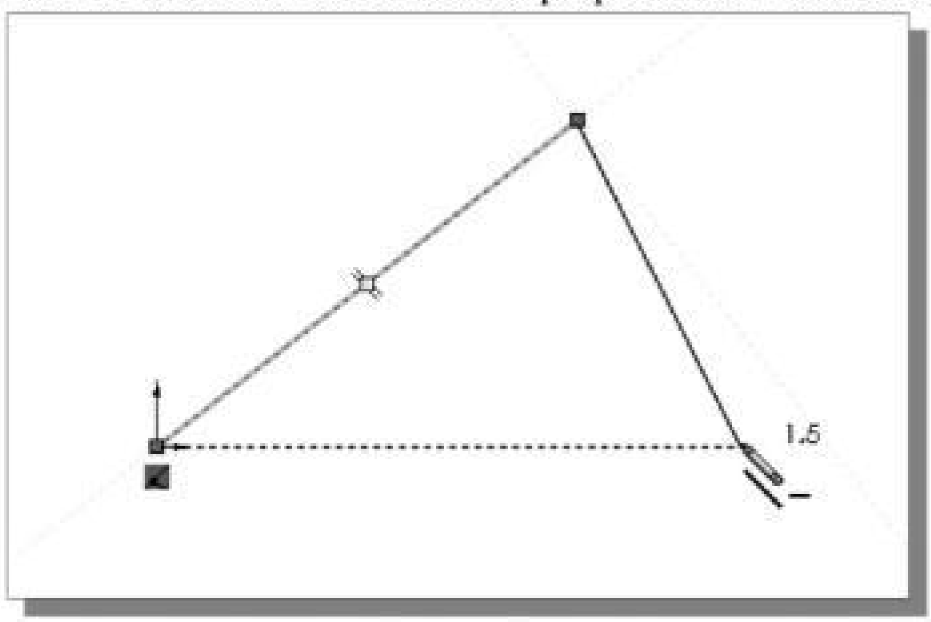
The typical procedure of creating and analyzing an FEA truss or beam element model in *SolidWorks* involves the following steps:

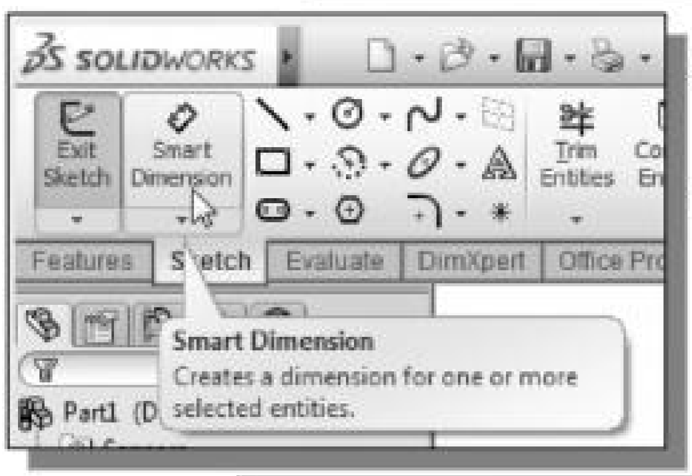
1. Create the solid models of the truss or beam systems using either the standard parametric modeling tools or the Structural Members tool.
2. Transfer the CAD model into *SolidWorks Simulation*.
3. Define the type of finite element analysis to be performed.
4. Select the proper element type and create the FEA model.
5. Assign material properties to the FEA model.
6. Prescribe how the system is supported.
7. Prescribe how the loads are applied to the system.
8. Run the *FEA Solver* to compute displacements, strains and stresses.
9. View the results of the FEA procedure and confirm the results of the analysis.



4. Click the **Line** icon in the ***Sketch*** toolbar as shown.

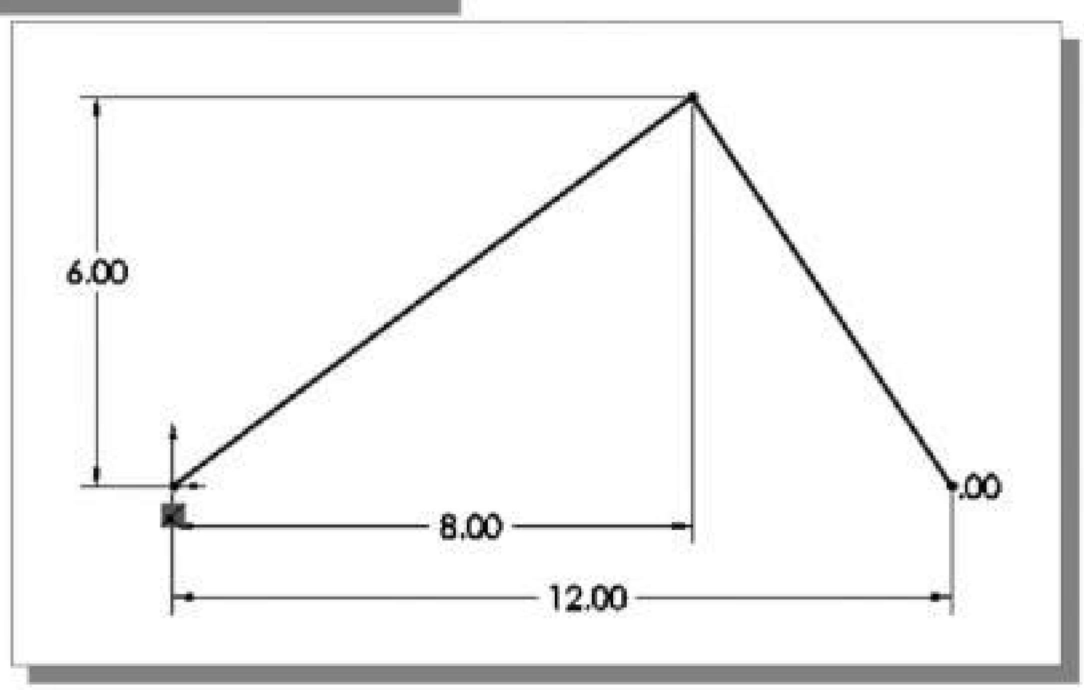
5. Start the line segment at the origin and create the two connected line segments as shown. (Note: **Do not** make the two lines perpendicular to each other.)

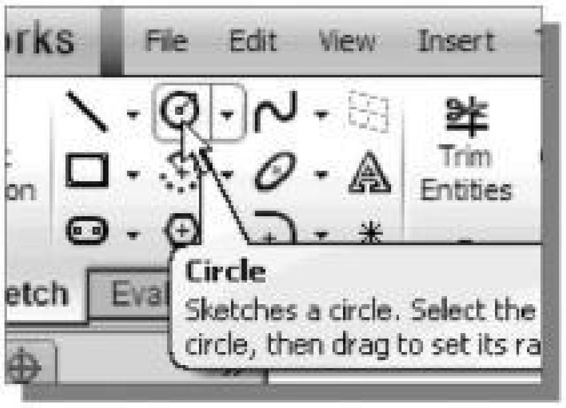




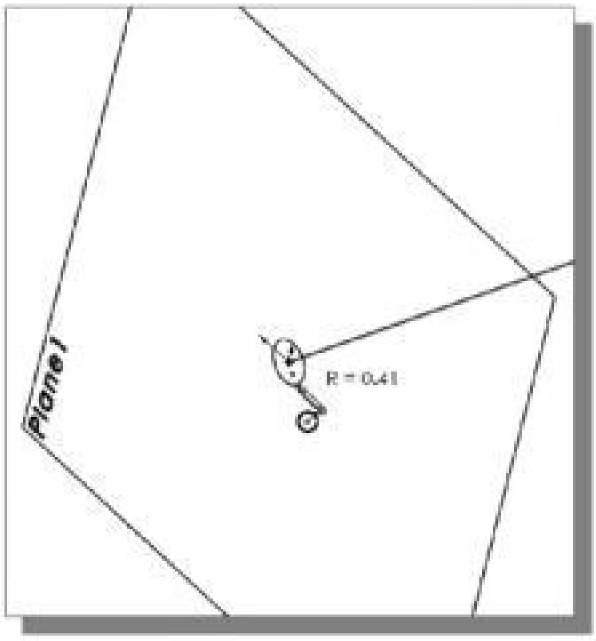
6. Click the **Smart Dimension** icon in the ***Sketch*** toolbar as shown.

7. On your own, create the four dimensions to adjust the locations of the endpoints as shown. Note the **0.00** dimension is used to align the last endpoint of the line segments to the origin in the vertical direction.



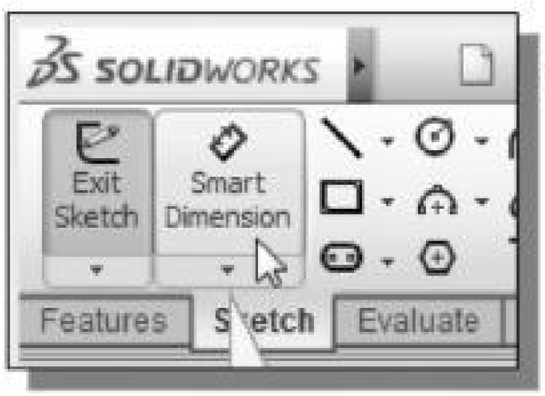


18. Click the **Circle** icon in the ***Sketch*** toolbar as shown.



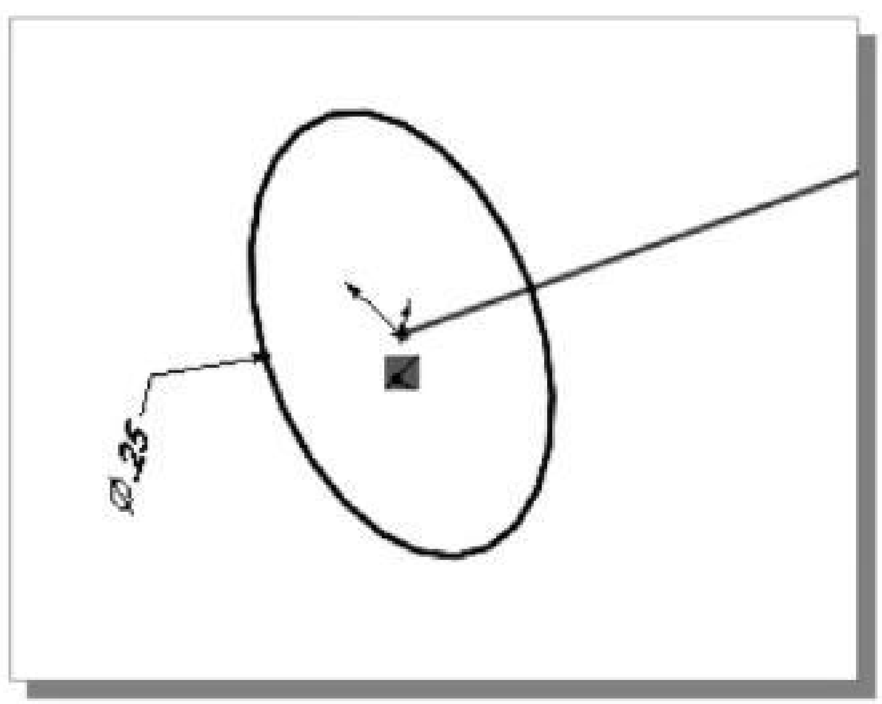
19. Select the bottom endpoint of the left line segment to align the center of the circle.

20. On your own, create a circle of arbitrary size.



21. Click the **Smart Dimension** icon in the ***Sketch*** toolbar as shown.

22. On your own, create and adjust the diameter dimension to ***0.25*** as shown.

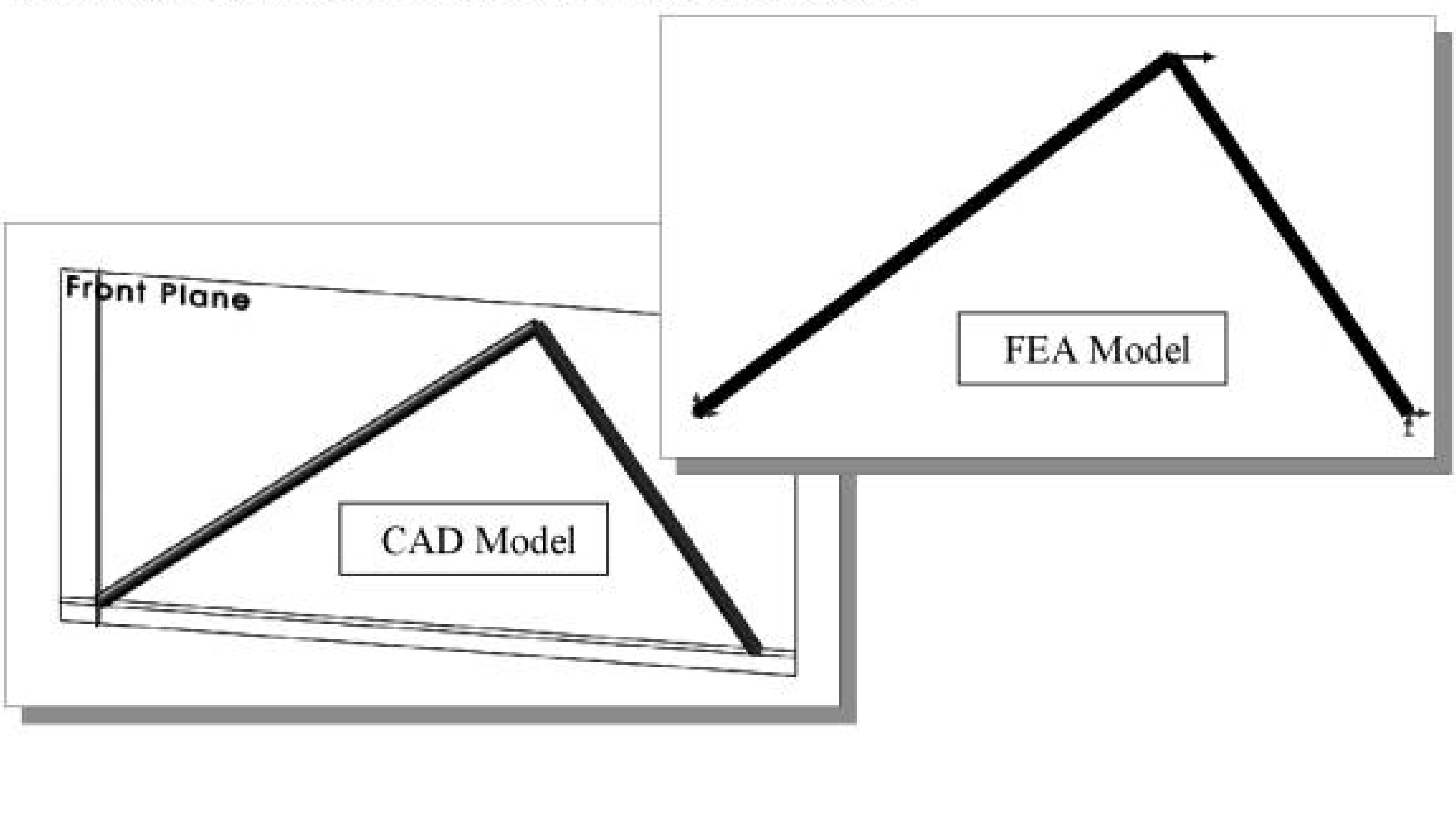


You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

A CAD Model is NOT an FEA Model

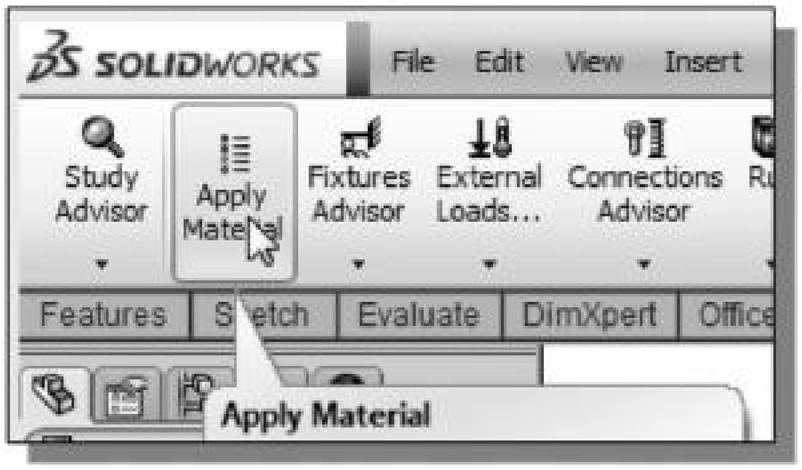
The two extruded features we just created represent the solid model of the truss system that we will be analyzing. This model is created under *SolidWorks*, and it is a CAD model that contains geometric information about the system. The CAD model can be used to provide geometric information needed in the finite element model. However, the CAD model is not an FEA model. In the previous chapters, we have examined the FEA procedure, which concentrated on the nodes and elements. So far, none of the nodes or elements exists in our CAD model. The geometric information of the system is necessary for the FEA analysis and what we have established in the CAD model is just the geometric information.

The CAD model is typically developed to provide geometric information necessary for manufacturing. All details must be specified and all dimensions are required. The manufactured part and the CAD model are identical in terms of geometric information. The FEA model uses the geometric information of the CAD model as the starting point, but the FEA model usually will adjust some of the basic geometric information of the CAD model. The FEA model usually will contain additional nodes and elements. Idealized boundary conditions and external loads are also required in the model. The goal of finite element analysis is to gain sufficient reliable insights into the behaviors of the real-life system. Many assumptions are made in the finite element analysis procedure to simplify the analysis, since it is not possible or practical to simulate all factors involved in real-life systems. The finite element analysis procedure provides an idealized approximation of the real-life system. It is therefore not practical to include all details of the system in the FEA model; the associated computational cost cannot be justified in doing so. It is a common practice to begin with a more simplified FEA model. Once the model has been solved accurately and the result has been interpreted, it is feasible to consider a more refined model in order to increase the accuracy of the prediction of the actual system. In *SolidWorks Simulation*, creating an FEA model by using the geometric information of the CAD model is known as *idealization*.



Assign the Element Material Property

➢ Next we will set up the *Material Property* for the elements. The *Material Property* contains the general material information, such as *Modulus of Elasticity*, *Poisson's Ratio*, etc. that are necessary for the FEA analysis.

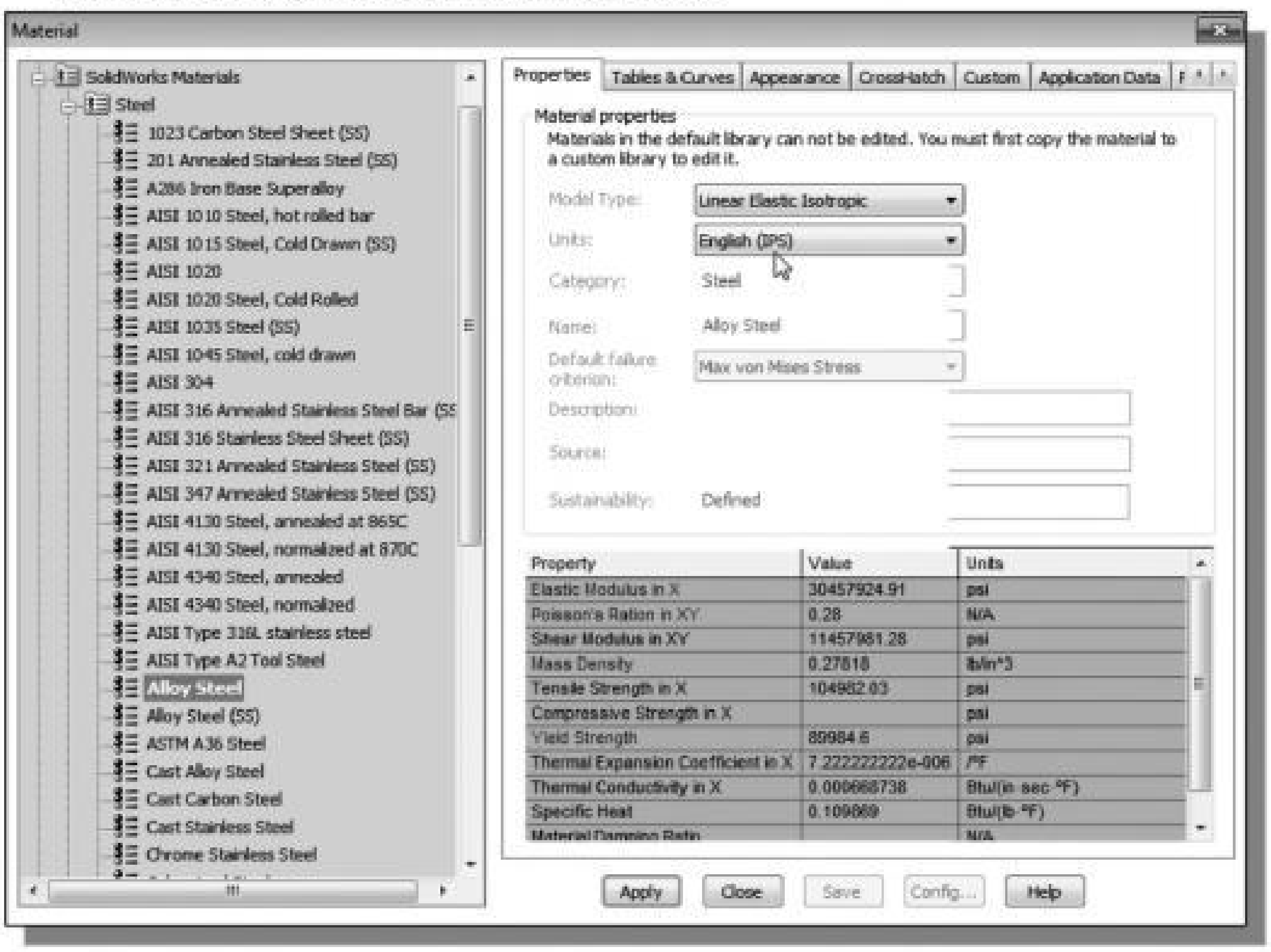


1. Choose the **Apply Material** option from the pull-down menu as shown.

❖ Note the default list of materials, which are available in the pre-defined *SolidWorks Simulation* material library, is displayed.

2. Select **ALLOY STEEL** in the *Material* list as shown.

3. Set the **Units** option to display **English (IPS)** to make the selected material available for use in the current FEA model.

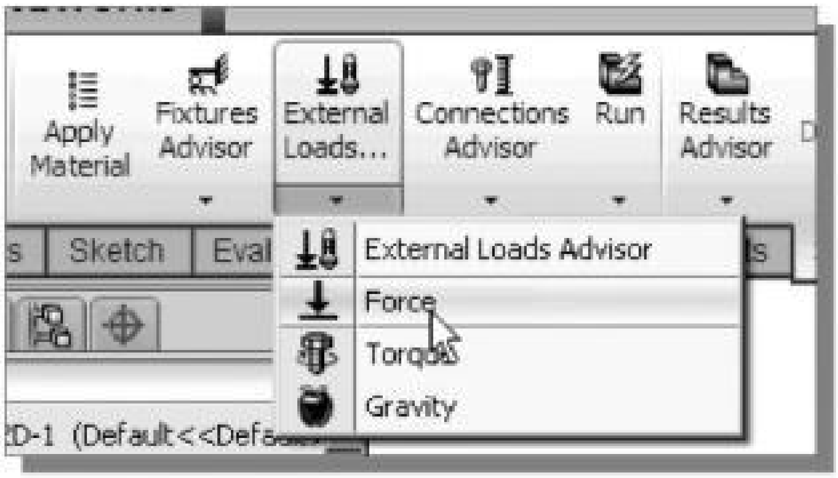


4. Click **Apply** to assign the material property, then click **Close** to exit the Material Assignment command.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Applying External Loads



1. Choose **External Loads → Force** by clicking the icon in the toolbar as shown.

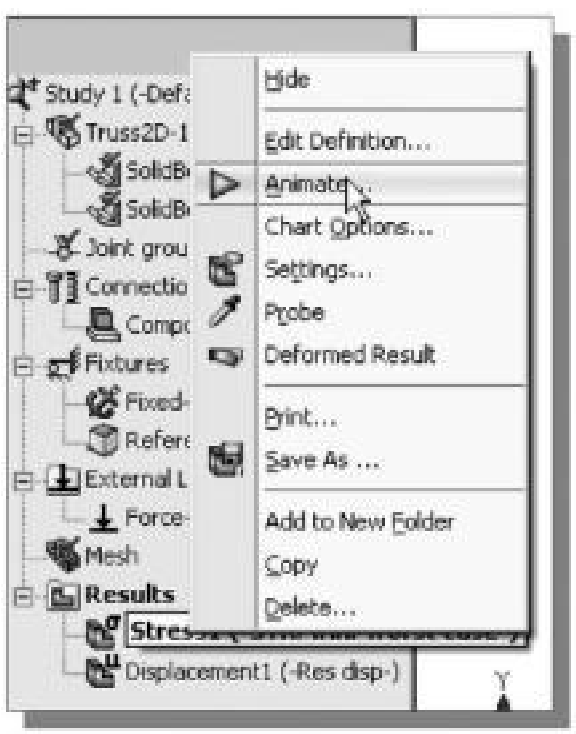
2. Change the *Force/Torque* option to **Joints** as shown.

3. Select the **top node point** as shown.

4. Activate the ***Direction Reference*** list option box by clicking on the inside of the ***Reference*** list box as shown.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

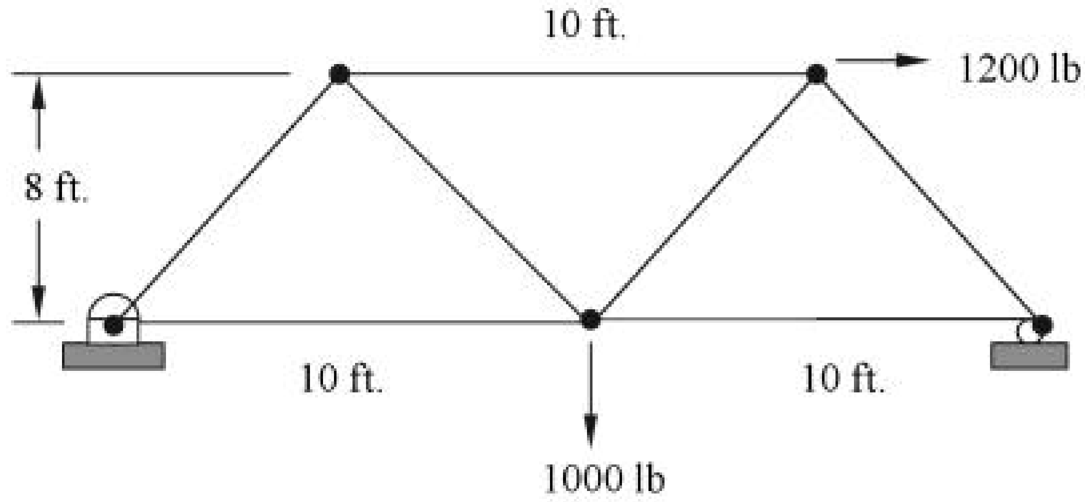
- *SolidWorks Simulation* calculated the stresses for the two members as (1) left member: **848.8** psi in tension and (2) right member: **612.1** psi in compression. These values match with the results of the *Excel* analysis on page 3-22.



5. In the *FEA Study* window, click once with the right-mouse-button on the **Result/Stress** item to display the option list and select **Animate** as shown.

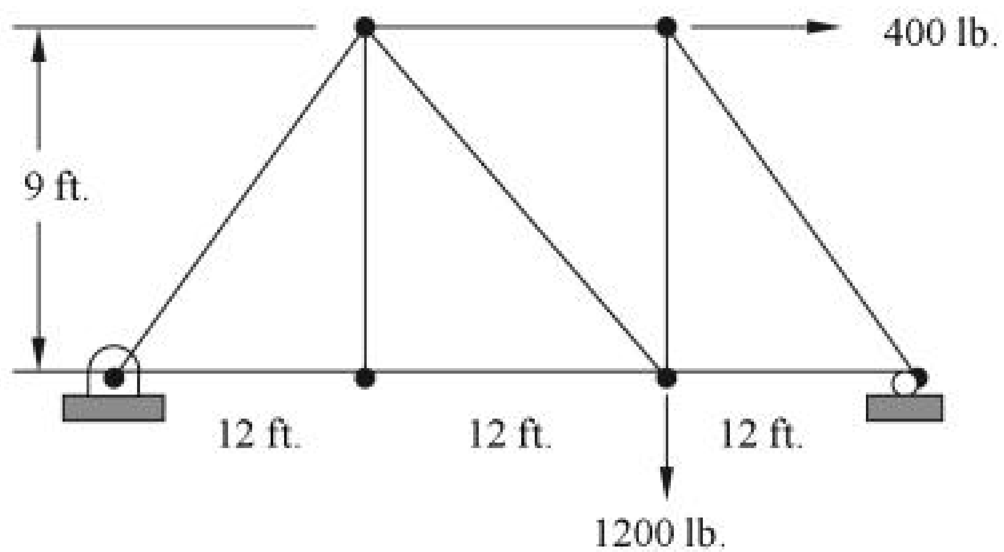
6. On your own, adjust the speed by dragging the slide bar as shown.

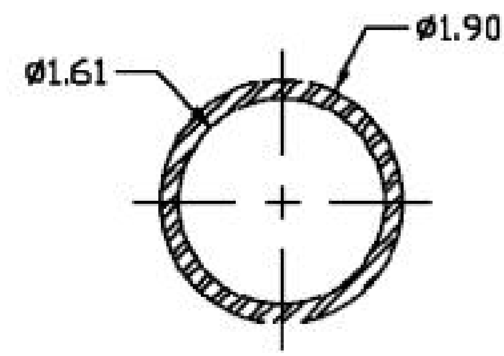
3. Material: Steel,
Diameter: 2.5 in.



Preliminary Analysis

Determine the normal stress in each member of the truss structure shown.

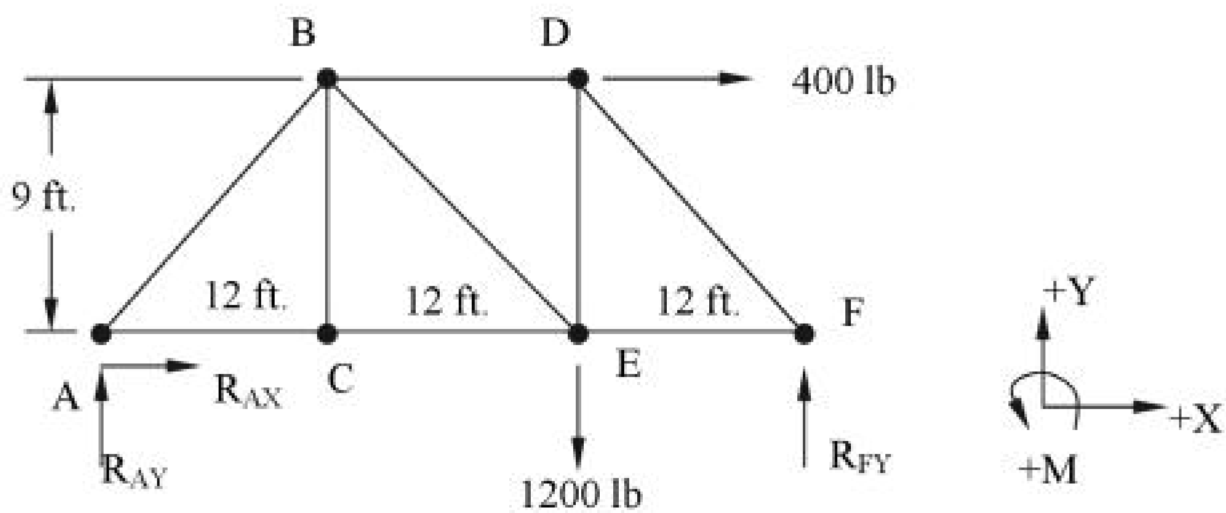




Material: Steel pipe,
NPS 1½″, Schedule 40

Prior to carrying out the finite element analysis, it is important to do an approximate preliminary analysis to gain some insights into the problem and as a means to check the finite element analysis results.

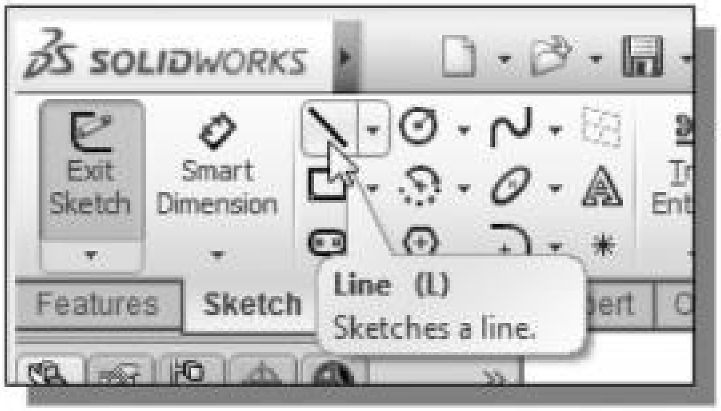
Free Body Diagram of the structure:



By inspection, member BC can be identified as a ZERO-FORCE member. Therefore, the stress in BC will be zero.

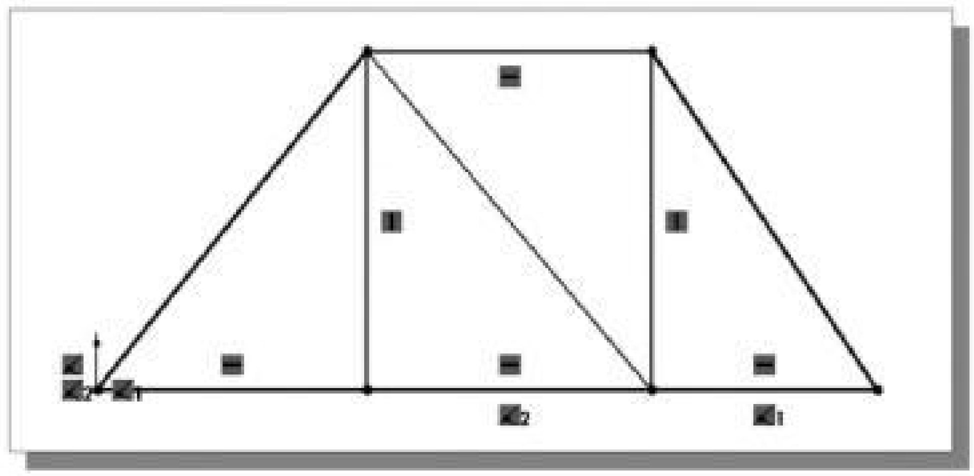
$\Sigma M_A = 36 \times R_{FY} - 24 \times 1200 - 9 \times 400 = 0$

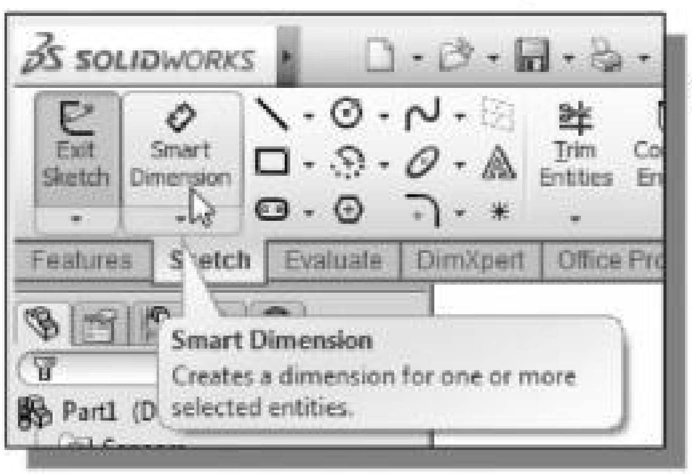
Solving for R_{FY}: $R_{FY} = 900$ lb.



4. Click the **Line** icon in the *Sketch* toolbar as shown.

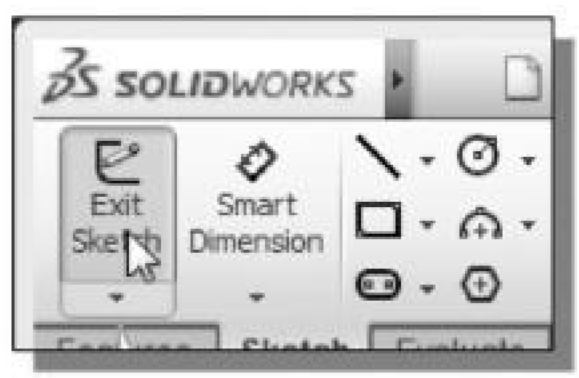
5. Start the line segment at the origin and create the **nine** connected line segments as shown.



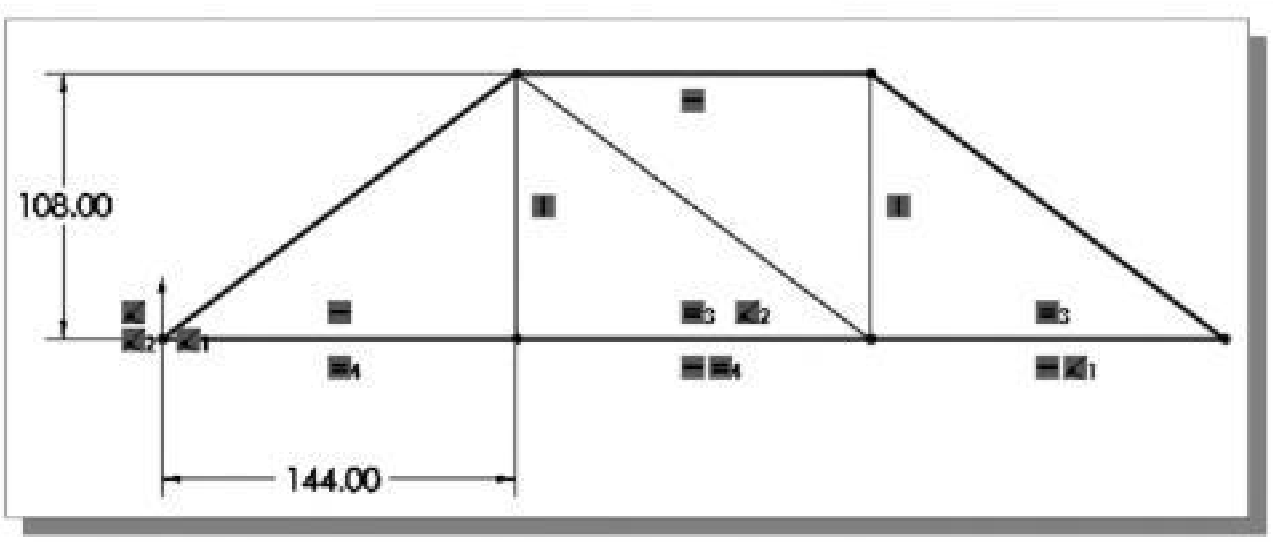


6. On your own, apply the **Equal** constraint to the bottom three members and use **Smart Dimension** to adjust the sketch as shown.

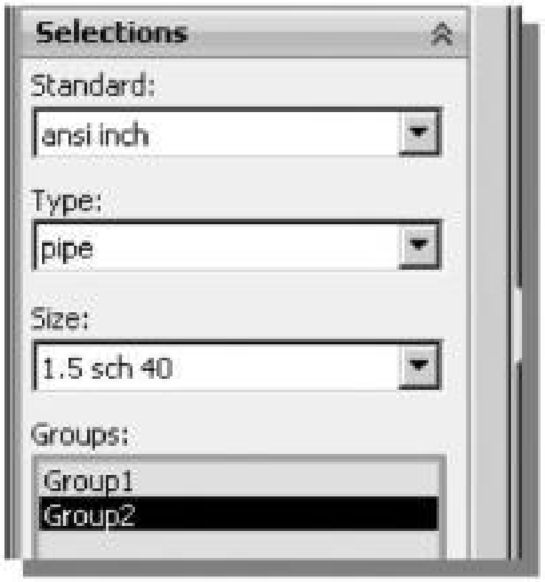
7. Click the **Exit sketch** icon in the *Sketch* toolbar as shown.



❖ Note the dimensions disappeared from the screen. The nine lines we just created represent the center axes of the nine truss members.



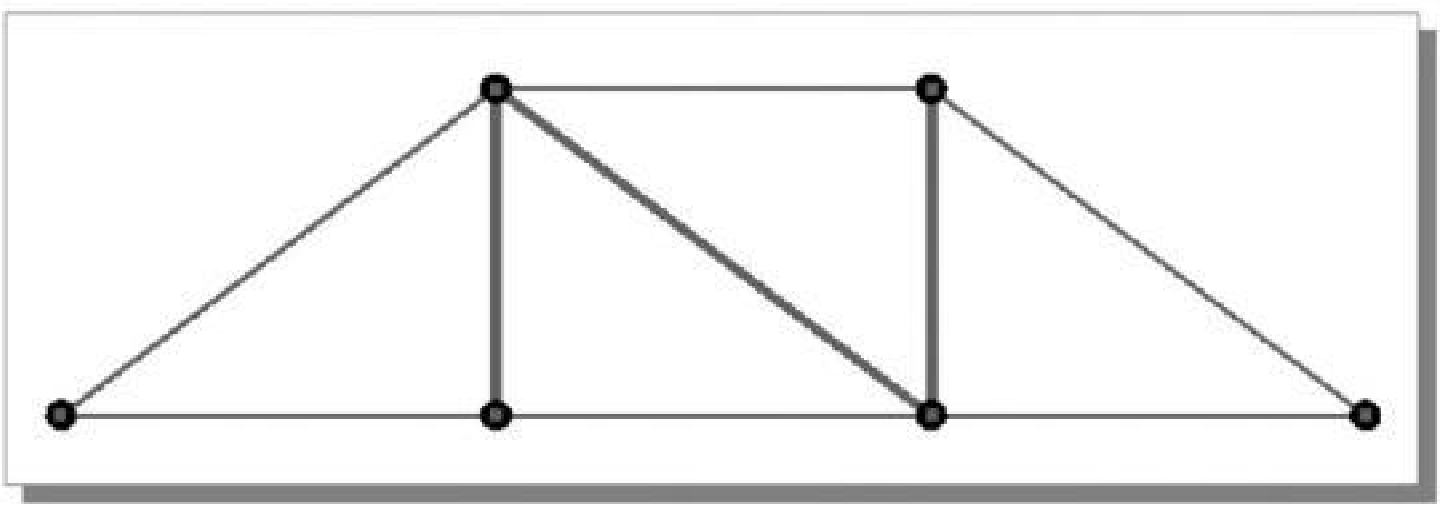
7. Click on the **New Group** button to create a new group under the same structural member.



8. Confirm the new group is highlighted and also all of the settings for the weldment profile remain the same as *Group1* as shown.

❖ All groups in a single structural member will use the same profile.

9. Select the **three line segments**, forming the letter N, that are on the inside loop of the structure as shown.



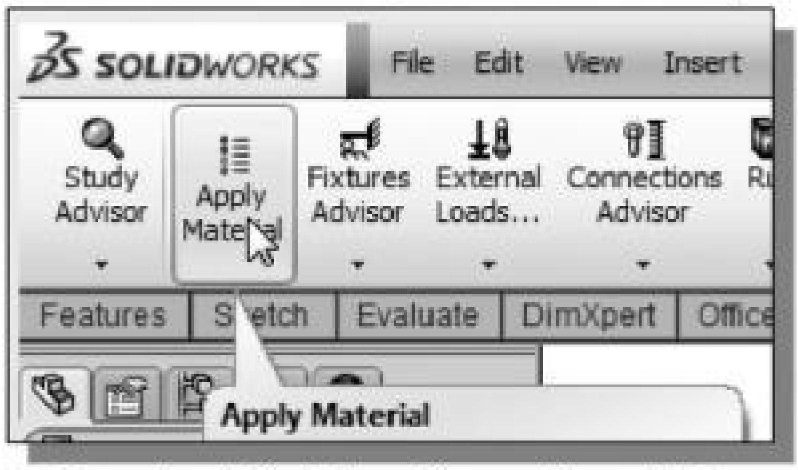
10. In the ***Settings*** option list, switch ***OFF*** the *Apply corner treatment* option as shown.

11. Click **OK** to create the structural member, which contains two groups with the same profile.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Assign the Element Material Property

➢ Next we will set up the *Material Property* for the elements. The *Material Property* contains the general material information, such as *Modulus of Elasticity*, *Poisson's Ratio*, etc. that is necessary for the FEA analysis.

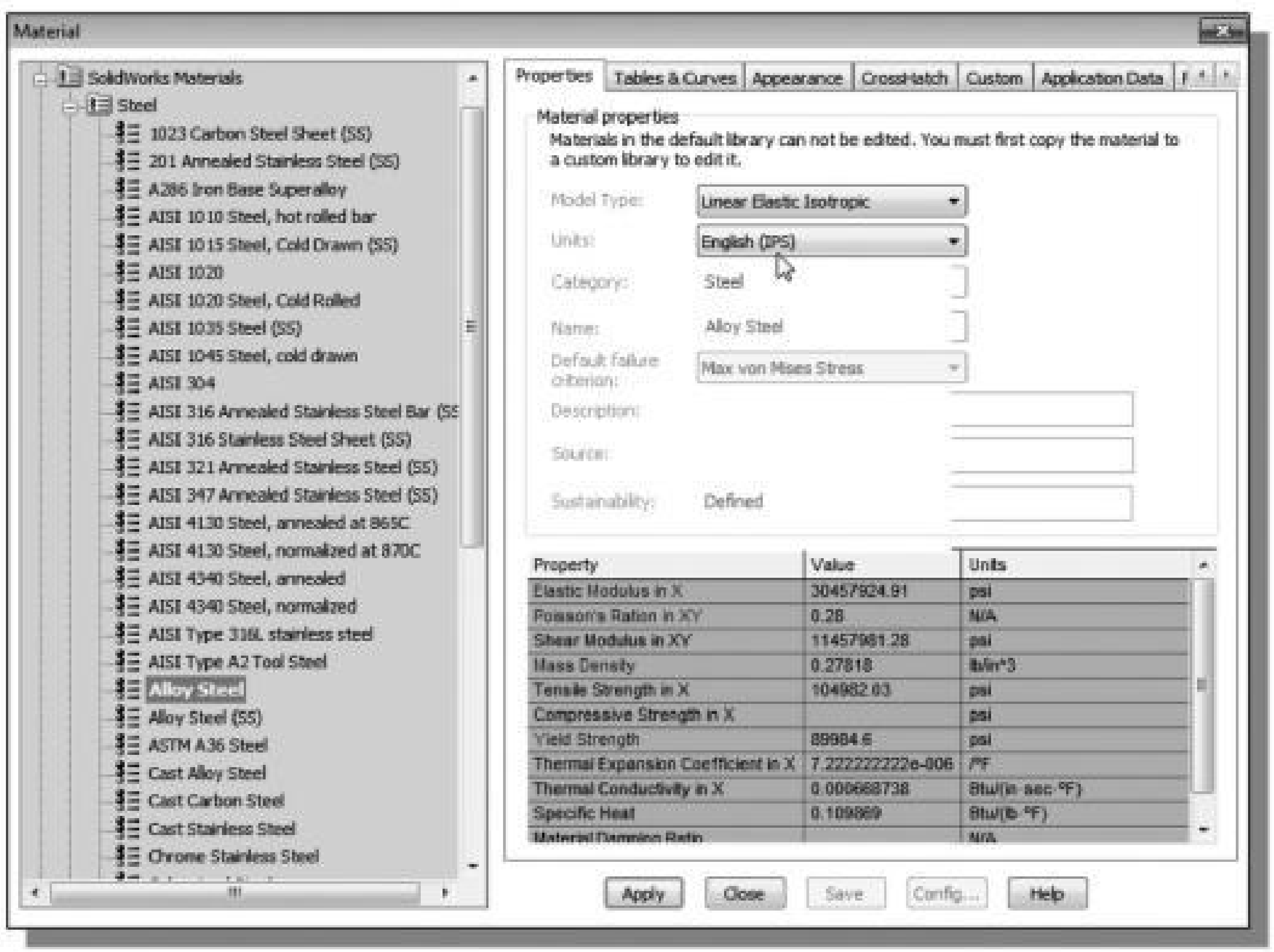


1. Choose the **Apply Material** option from the pull-down menu as shown.

❖ Note the default list of materials, which are available in the predefined *SolidWorks Simulation* material library, is displayed.

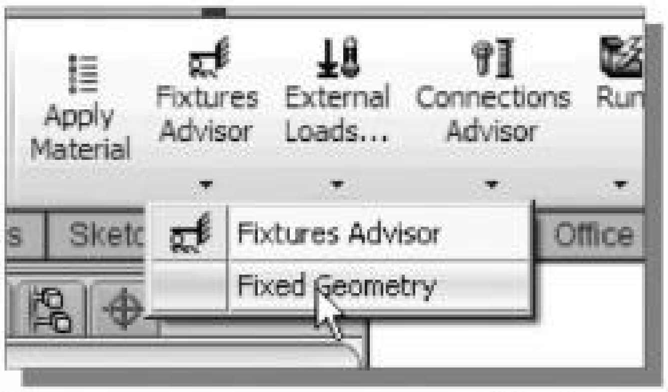
2. Select **Alloy Steel** in the *Material* list as shown.

3. Set the **Units** option to display **English (IPS)** to make the selected material available for use in the current FEA model.

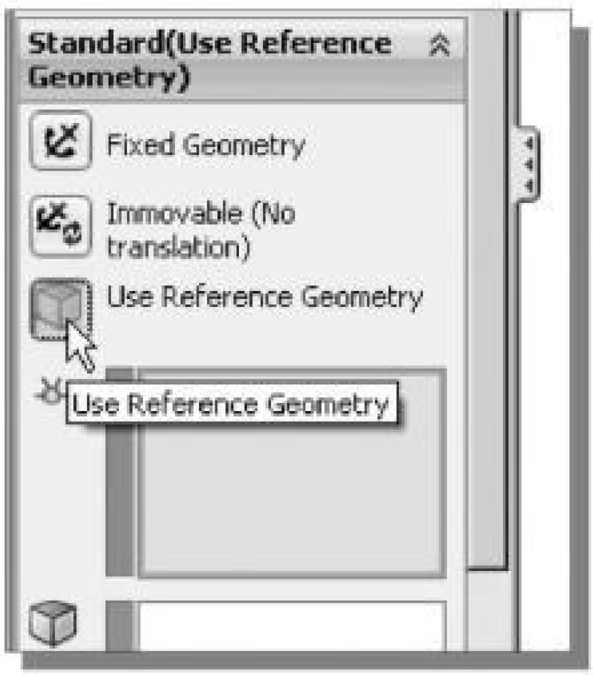


4. Click **Apply** to assign the material property then click **Close** to exit the Material Assignment command.

❖ Note that for 2D truss systems, any translation motion in the Z direction is not allowed. We will need to add another constraint set to restrict the motion of the unconstrained nodes in the Z direction. Not restricting the movement in the Z direction will result in an erroneous FEA analysis.

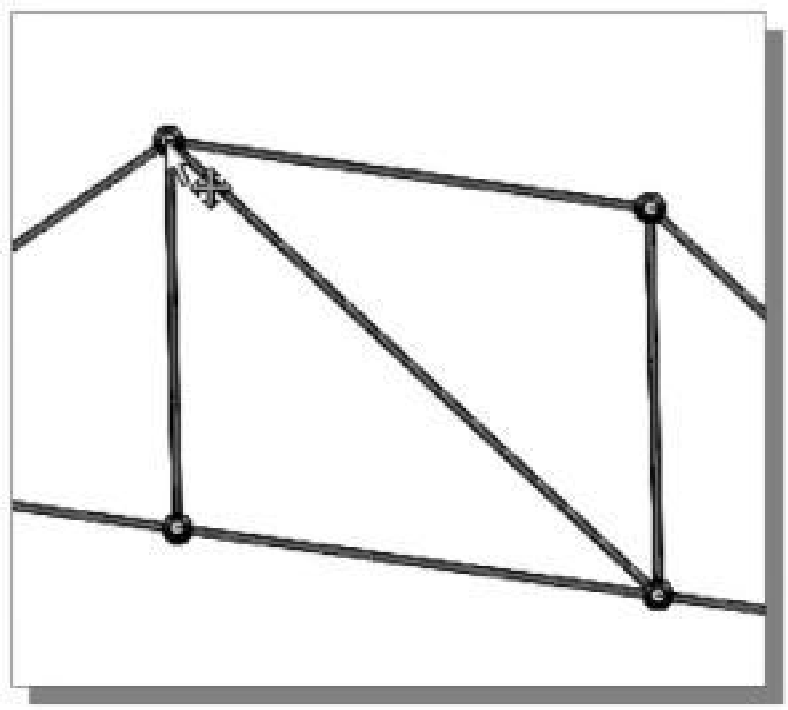


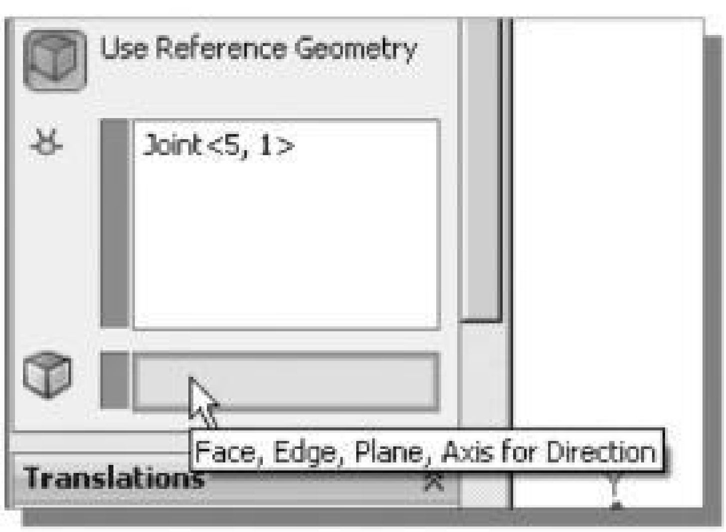
16. Choose **Fixed Geometry** by clicking the icon in the toolbar as shown.



17. Change the *Standard (Fixed Geometry)* option to **Use Reference Geometry** as shown.

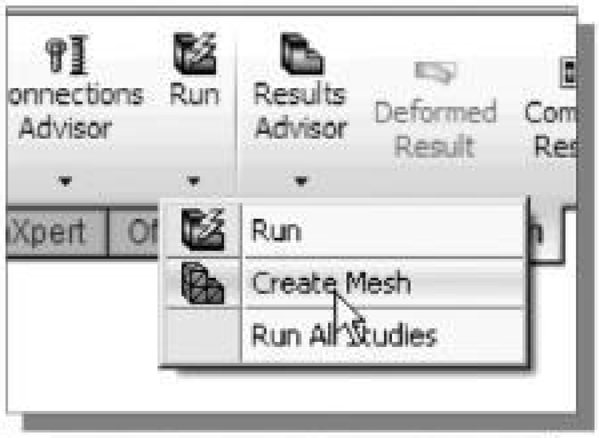
18. Select the **four nodes in the middle** as shown.



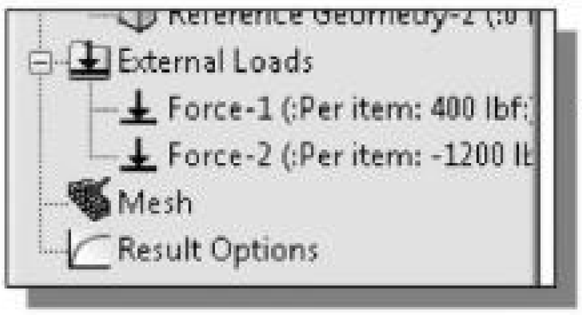


19. Activate the *Direction Reference* list option box by clicking on the inside of the **Reference** list box as shown.

Create the FEA Mesh and Run the Solver

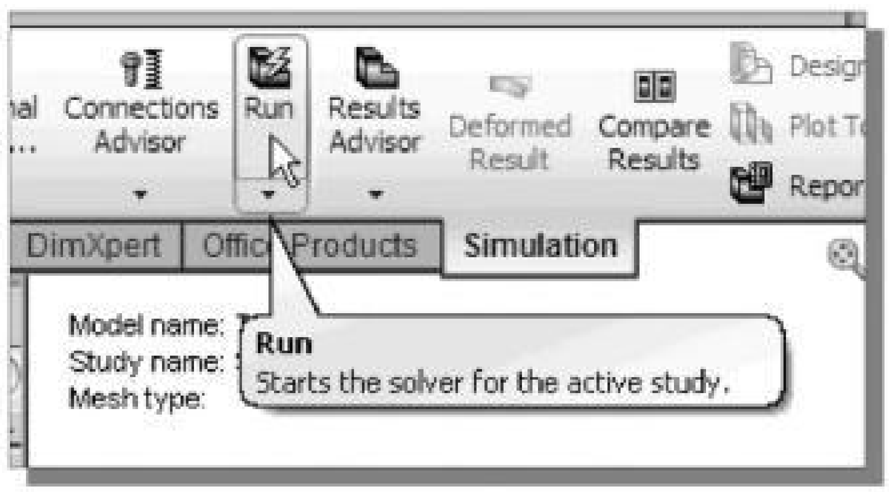


1. Choose **Create Mesh** by clicking the icon in the toolbar as shown.

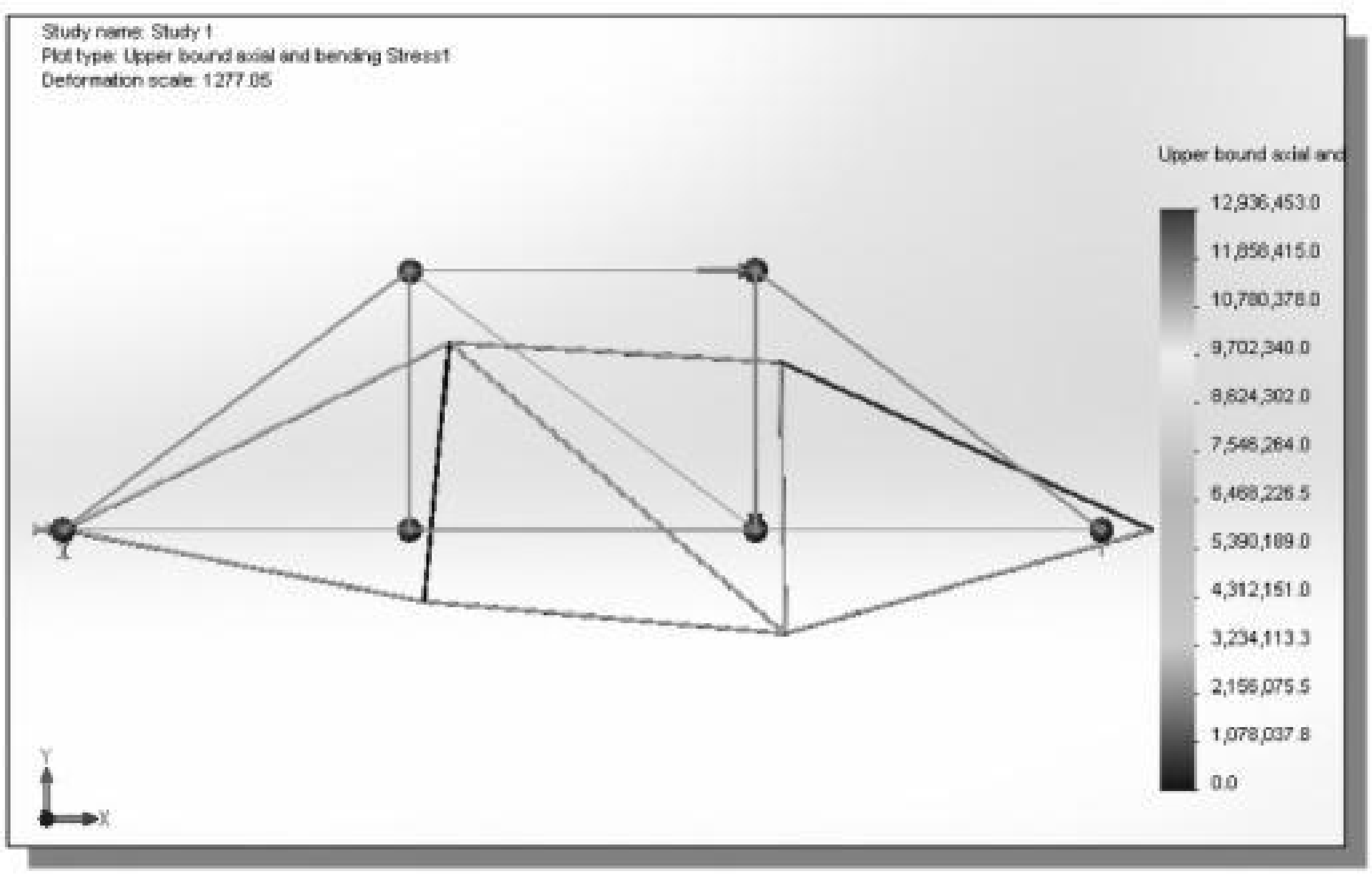


❖ Note the **Mesh** icon has changed, which indicates the FEA Mesh has been created.

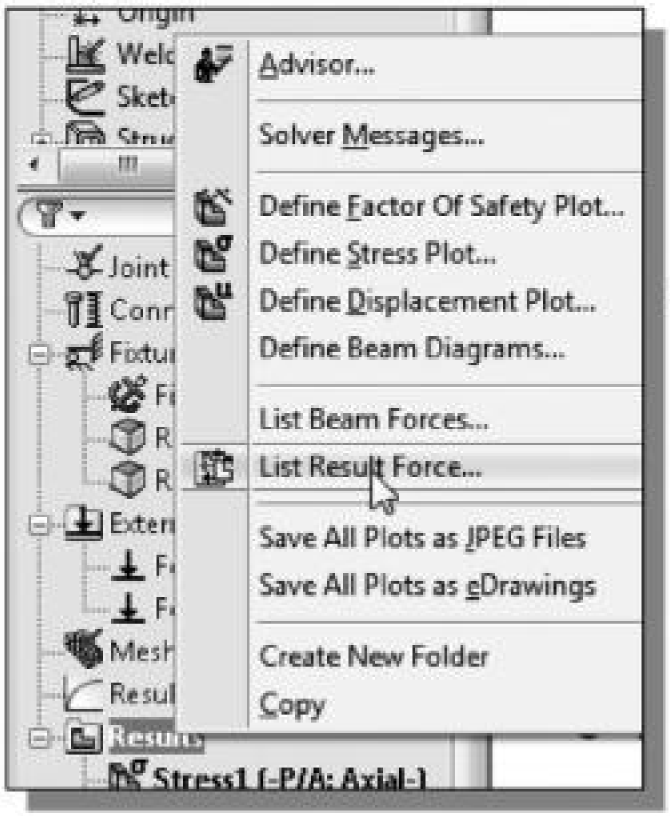
2. Click on the **Run** button to start the *FEA Solver* to calculate the results.



❖ Note the stress results are displayed when the *Solver* has completed the FEA calculations.

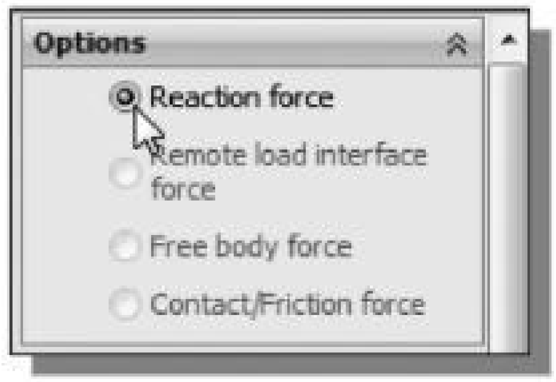


Viewing the Reaction Forces at the supports



1. In the *FEA Study* window, click once with the right-mouse-button on the ***Results*** item to display the option list and select **List Result Forces** as shown.

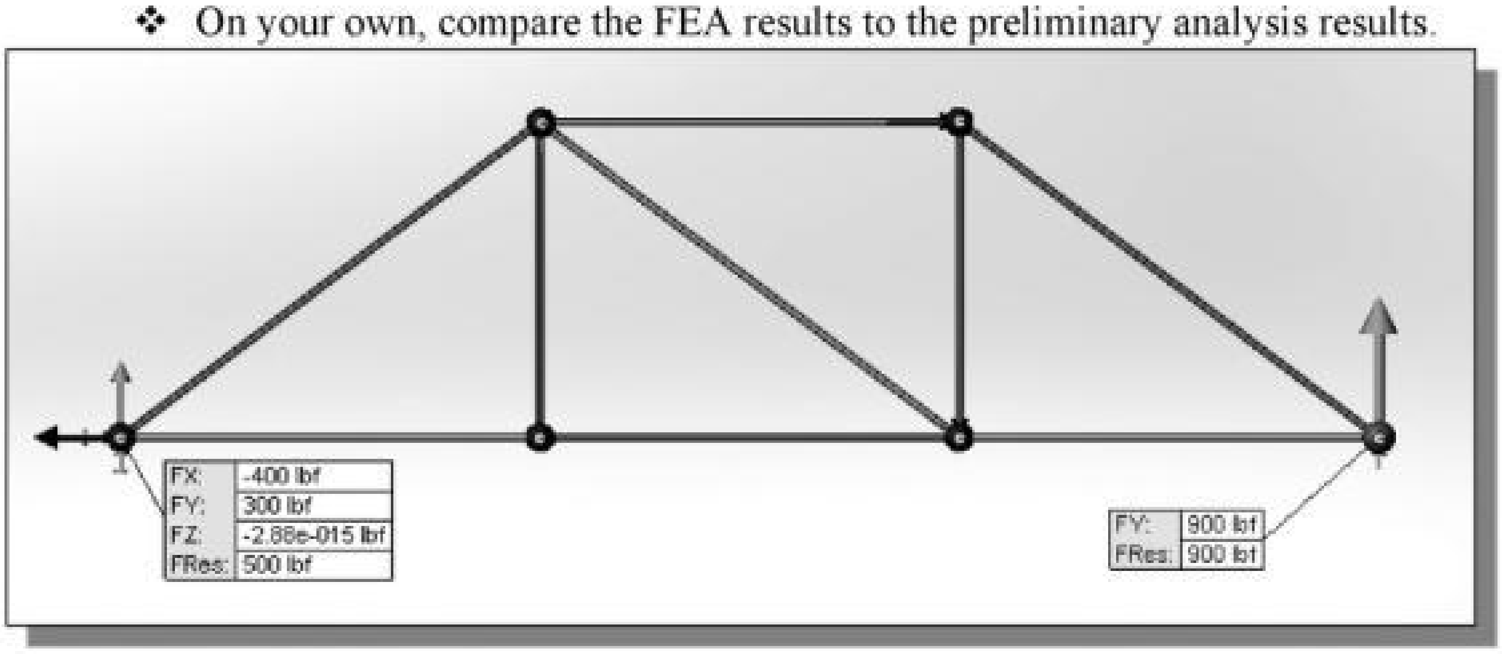
2. Confirm the Option is set to **Reaction Force**.



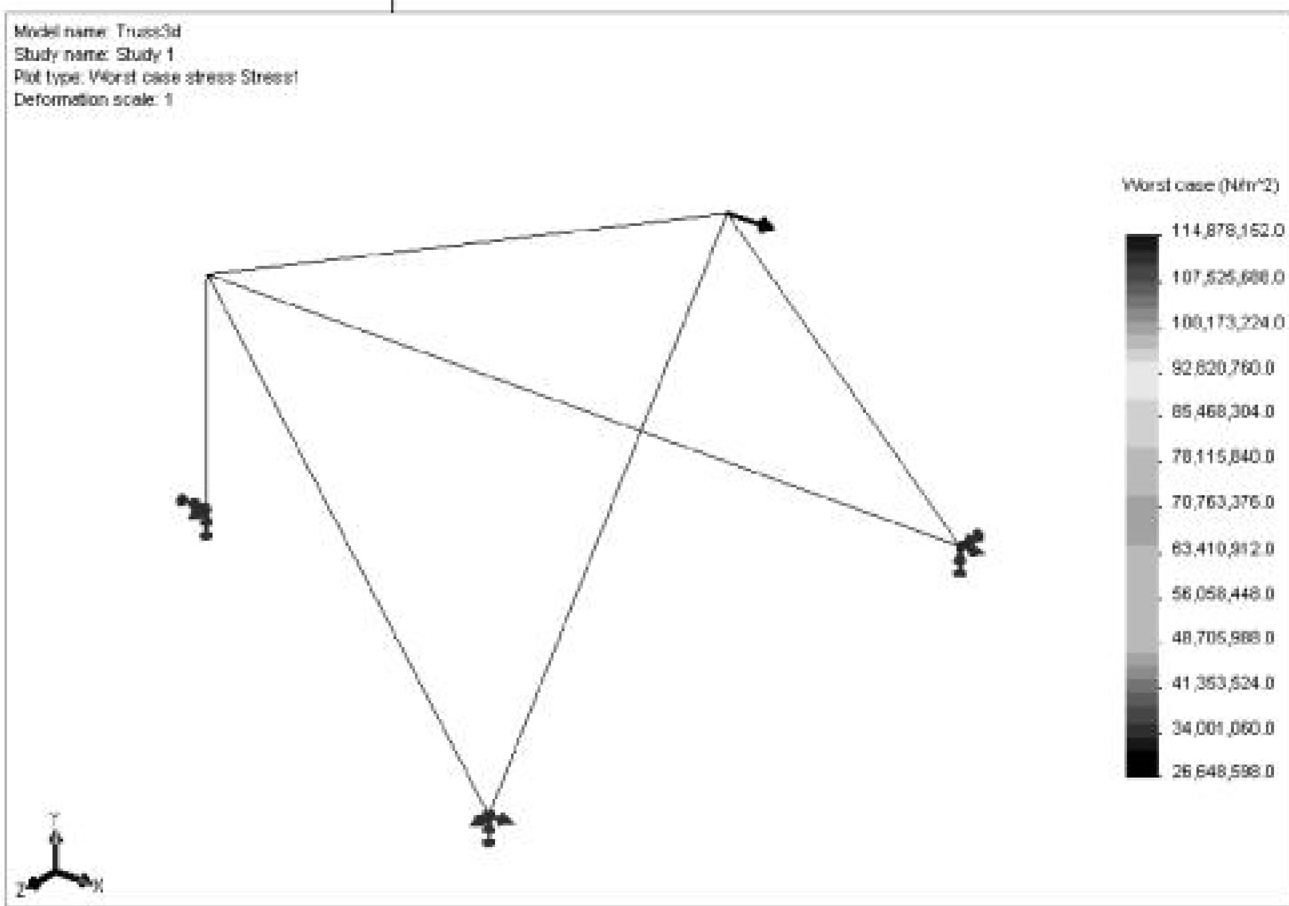
3. Set the *Units* to **English (IPS)** as shown.

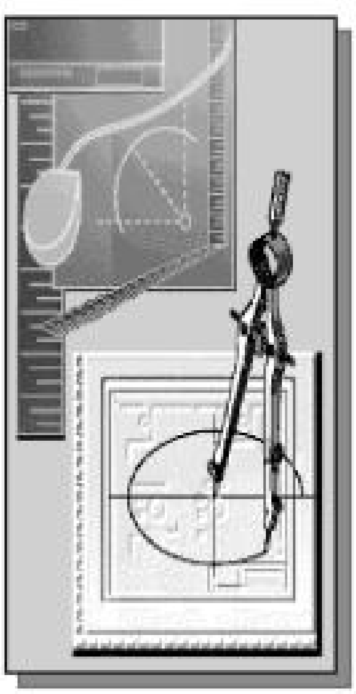
4. Click **Update** to display the results on the screen.

❖ On your own, compare the FEA results to the preliminary analysis results.



Chapter 6
Three-Dimensional Truss Analysis





Learning Objectives

- Determine the Number of Degrees of Freedom in Elements.
- Create 3D FEA Truss Models.
- Apply proper boundary conditions to FEA Models.
- Use SolidWorks Simulation Solver for 3D Trusses.
- Use SolidWorks Simulation to determine Axial Loads.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Unit vectors along AB, AC and AD:

$$u_{AB} = \frac{(-5)\,i + (-2)\,j}{\sqrt{(-5)^2+(-2)^2}} = -\,0.9285\,i - 0.371\,j$$

$$u_{AC} = \frac{(-4)\,j + (-3)\,k}{\sqrt{(-4)^2+(-3)^2}} = -\,0.8\,j - 0.6\,k$$

$$u_{AD} = \frac{(-4)\,j + (3)\,k}{\sqrt{(-4)^2+(3)^2}} = -\,0.8\,j + 0.6\,k$$

Forces in each member:

$$\mathrm{F}_{AB} = F_{AB}\,u_{AB} = F_{AB}\,(-\,0.9285\,i - 0.371\,j\,)$$
$$\mathrm{F}_{AC} = F_{AC}\,u_{AC} = F_{AC}\,(-\,0.8\,j - 0.6\,k\,)$$
$$\mathrm{F}_{AD} = F_{AD}\,u_{AD} = F_{AD}\,(-\,0.8\,j + 0.6\,k\,)$$

(F_{AB}, F_{AC} and F_{AD} are magnitudes of vectors F_{AB}, F_{AC} and F_{AD})

Applying the equation of equilibrium at node A:

$$\begin{aligned}\sum \mathrm{F}_{@A} &= 0 = 6000\,i + \mathrm{F}_{AB} + \mathrm{F}_{AC} + \mathrm{F}_{AD}\\ &= 6000\,i - 0.9285\,F_{AB}\,i - 0.371\,F_{AB}\,j - 0.8\,F_{AC}\,j - 0.6\,F_{AC}\,k\\ &\quad - 0.8\,F_{AD}\,j + 0.6\,F_{AD}\,k\\ &= (6000 - 0.9285\,F_{AB})i + (-\,0.371\,F_{AB} - 0.8\,F_{AC} - 0.8\,F_{AD})j\\ &\quad + (-\,0.6\,F_{AC} + 0.6\,F_{AD})k\end{aligned}$$

Also, since the structure is symmetrical, $F_{AC} = F_{AD}$

Therefore,

$\mathbf{6000 - 0.9285}\,F_{AB} = 0$, $\boxed{F_{AB} = \mathbf{6462\ N}}$

$\mathbf{-0.371}\,F_{AB} - \mathbf{0.8}\,F_{AC} - \mathbf{0.8}\,F_{AD} = 0$,

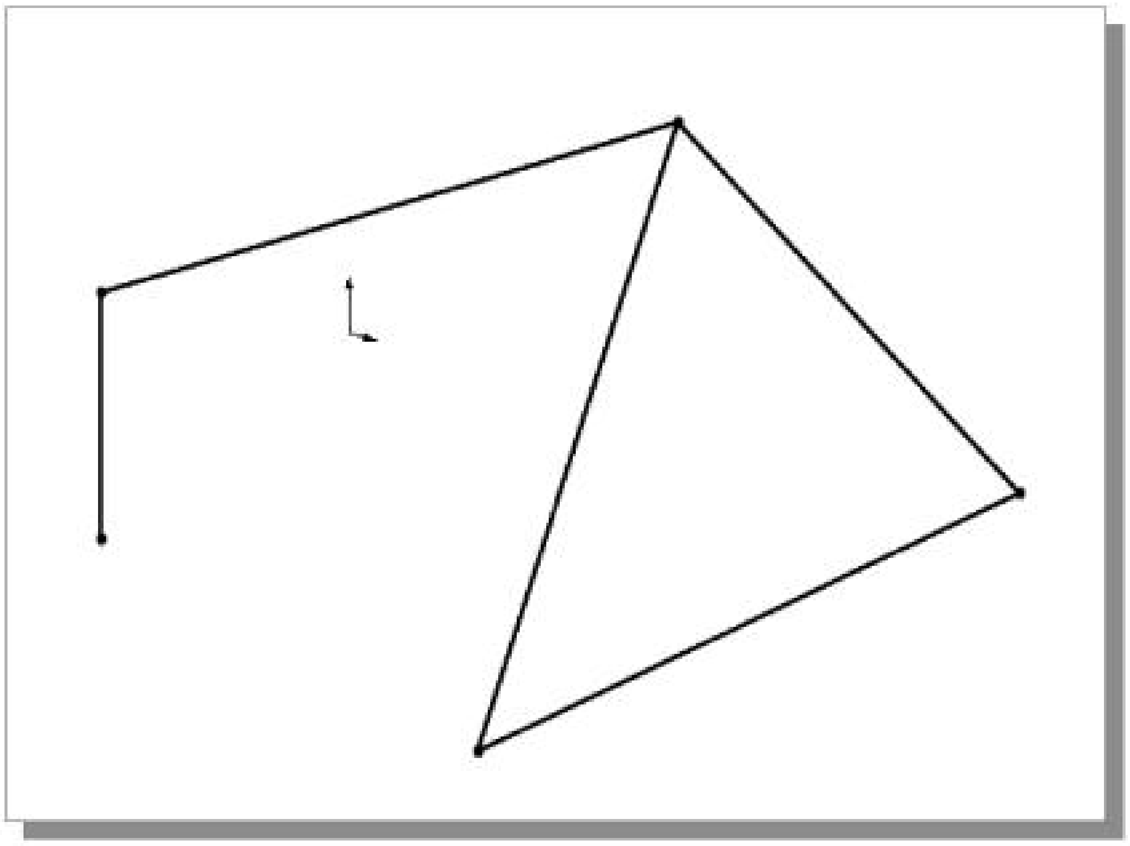
$\boxed{F_{AC} = F_{AD} = \mathbf{-\,1500\ N}}$

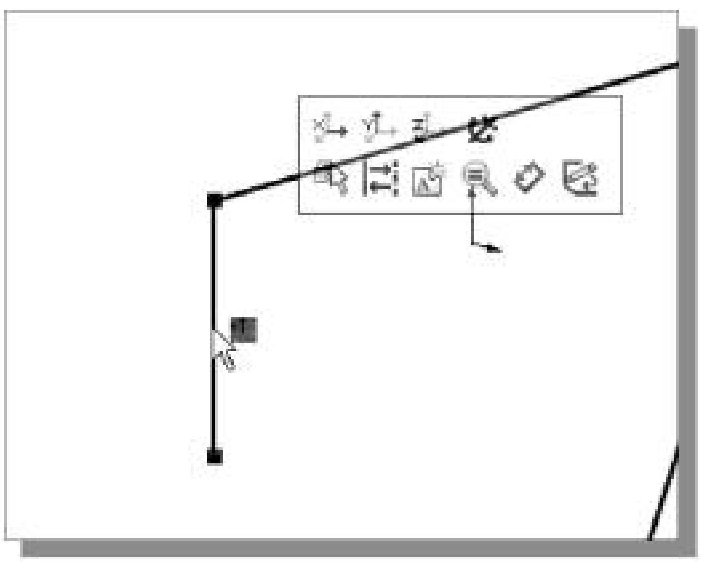
The stresses:

$$\boxed{\begin{aligned}&\sigma_{AB} = 6460\,/\,(5.63 \times 10^{-5}) = \mathbf{115\ MPa}\\ &\sigma_{AC} = \sigma_{AD} = -\,1500\,/\,(5.63 \times 10^{-5}) = \mathbf{-26.7\ MPa}\end{aligned}}$$

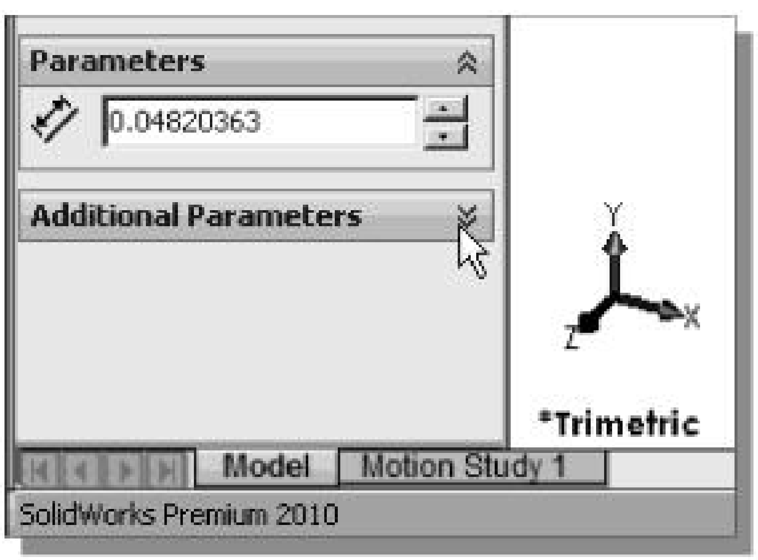
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

4. Start the line segment at the left side; create the vertical segment from the bottom up and create the five connected line segments as shown. Note the extra member is created to aid the 3D construction of the necessary members.

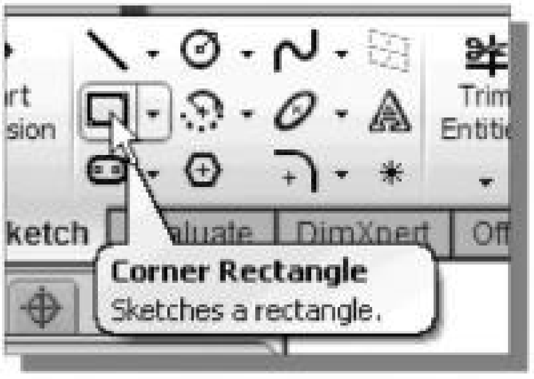




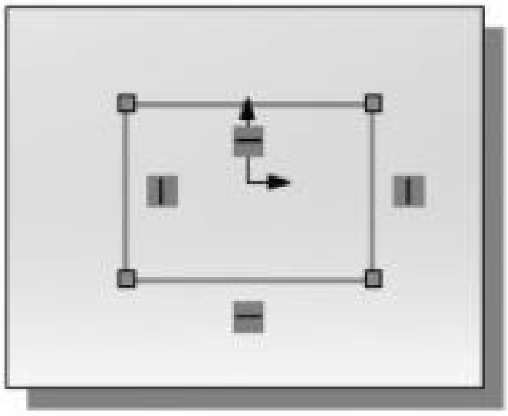
5. On your own, click on the left vertical line and confirm the **Vertical (Along Y)** constraint has been applied to this left member.



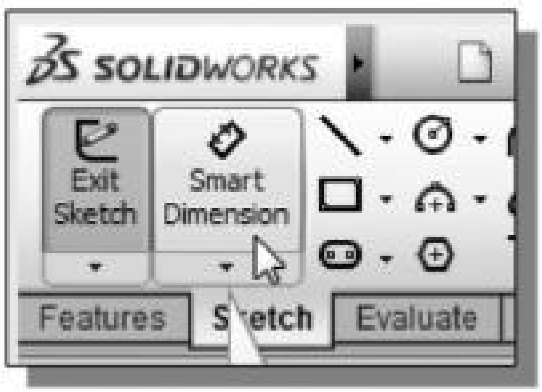
6. Click on the down arrows to open up the *Additional Parameters* panel, which contains the coordinates of the endpoints of the selected line.



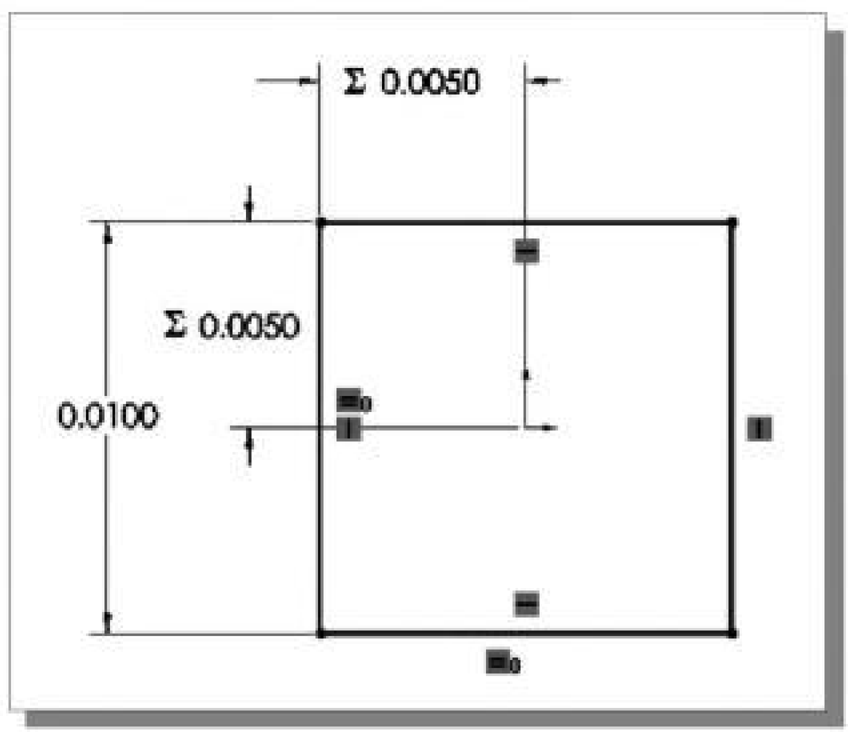
12. Click the **Corner Rectangle** icon to activate the **Rectangle** command.



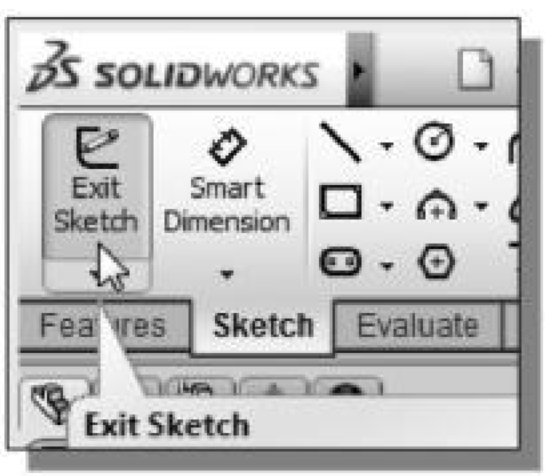
13. On your own, create a rectangle roughly centered at the origin of the world coordinate system.



14. On your own, apply the **Equal** constraint to the sides of the rectangle and use **Smart Dimension** to adjust the sketch as shown.



15. On your own, create the equations as shown.



16. Click the **Exit Sketch** icon in the *Sketch* toolbar as shown.

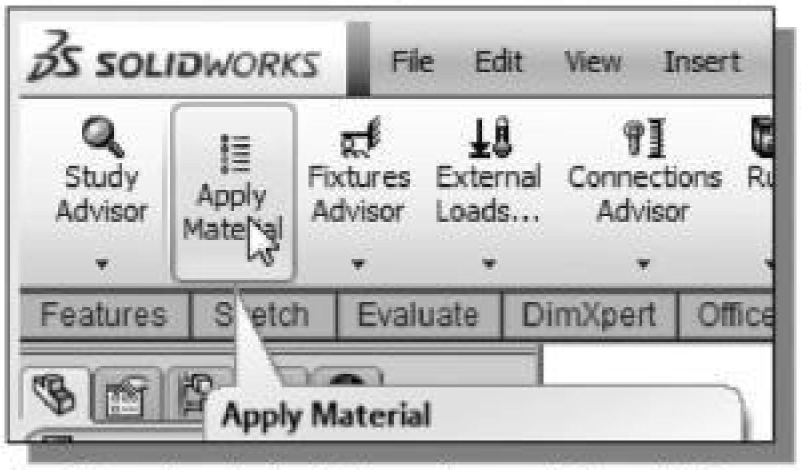
❖ Next, we will save the completed 2D sketch as a library file.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Assign the Element Material Property

➢ Next we will set up the *Material Property* for the elements. The *Material Property* contains the general material information, such as *Modulus of Elasticity*, *Poisson's Ratio*, etc. that is necessary for the FEA analysis.

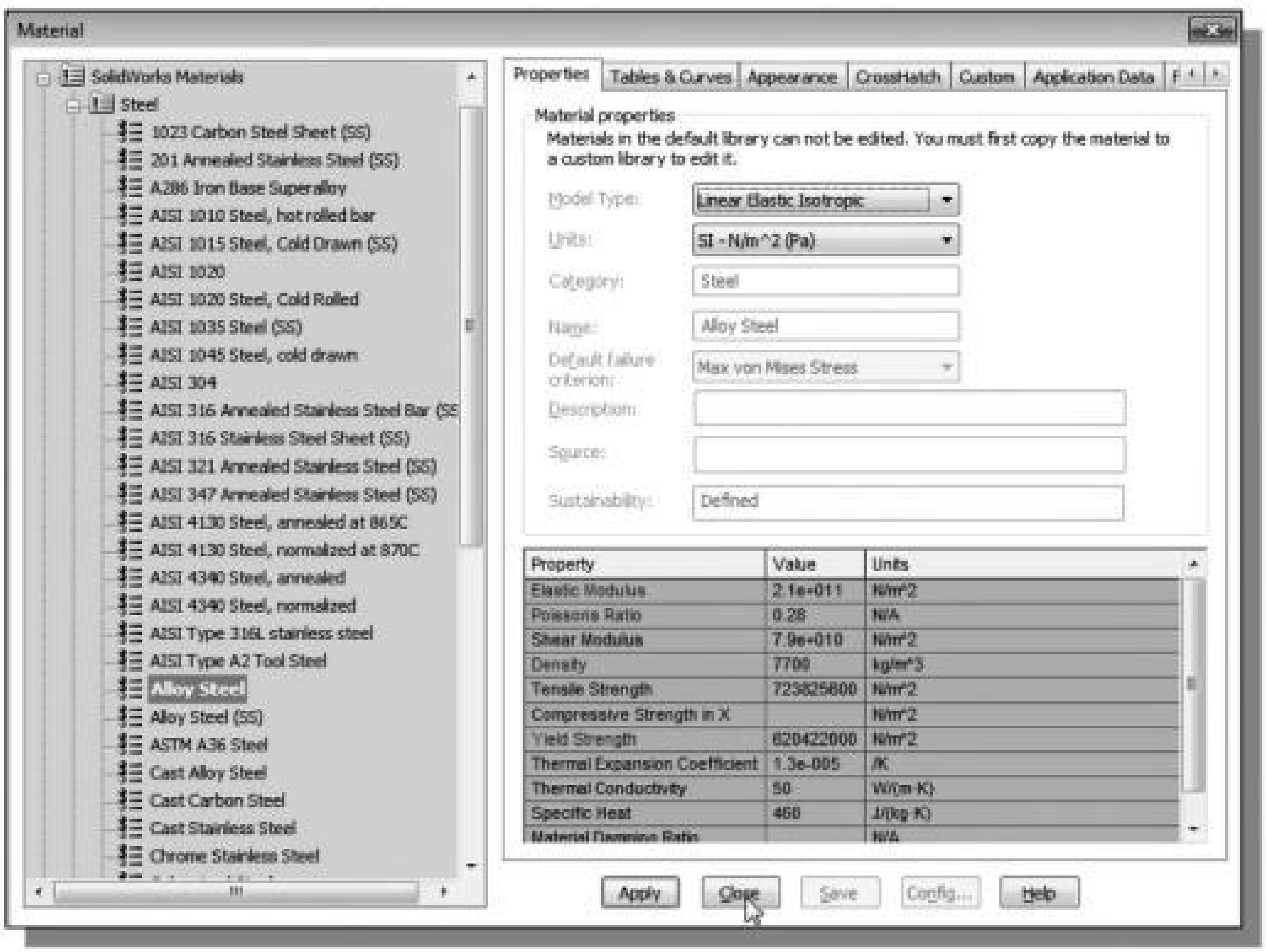


1. Choose the **Apply Material** option from the pull-down menu as shown.

❖ Note the default list of materials, which are available in the pre-defined *SolidWorks Simulation* material library, is displayed.

2. Select **Alloy Steel** in the *Material* list as shown.

3. Confirm the **Units** option to display **SI – N/m^2 (Pa)**.



4. Click **Apply** to assign the material property then click **Close** to exit the Material Assignment command.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

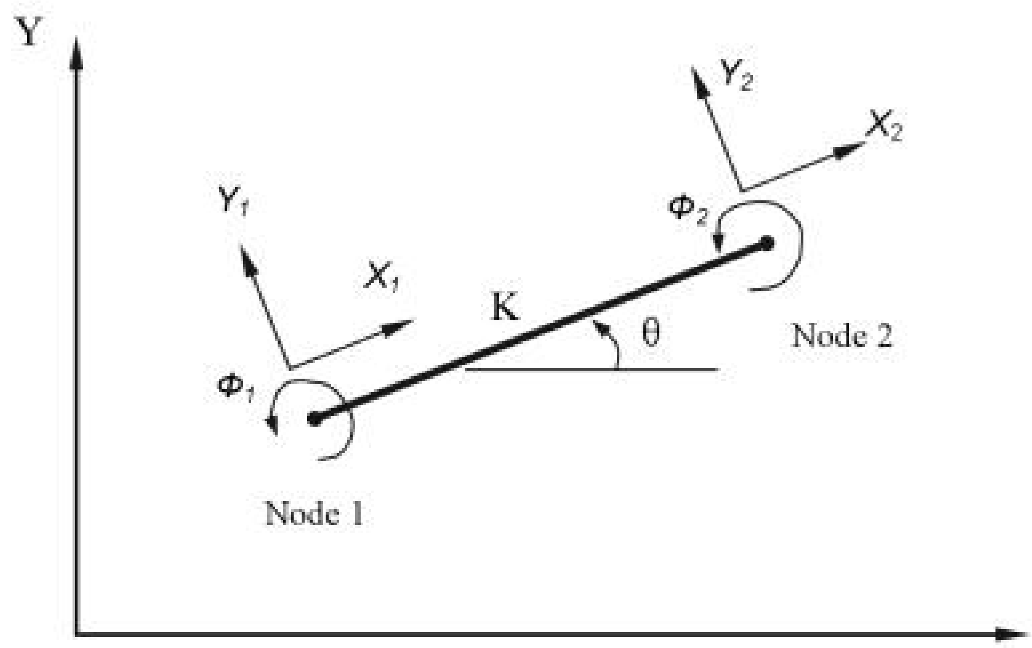
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Introduction

The truss element discussed in the previous chapters does have a practical application in structural analysis, but it is a very limiting element since it can only transmit axial loads. The second type of finite element to study in this text is the *beam element*. Beams are used extensively in engineering. As the members are rigidly connected, the members at a joint transmit not only axial loads but also bending and shear. This contrasts with truss elements where all loads are transmitted by axial force only. Furthermore, beams are often designed to carry loads both at the joints and along the lengths of the members, whereas truss elements can only carry loads that are applied at the joints. A beam element is a long, slender member generally subjected to transverse loading that produces significant bending effects as opposed to axial or twisting effects.

Modeling Considerations

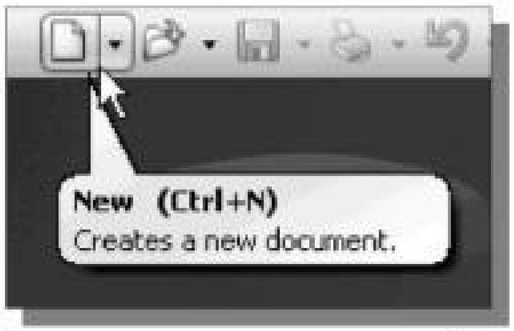
Consider a simple beam element positioned in a two-dimensional space:



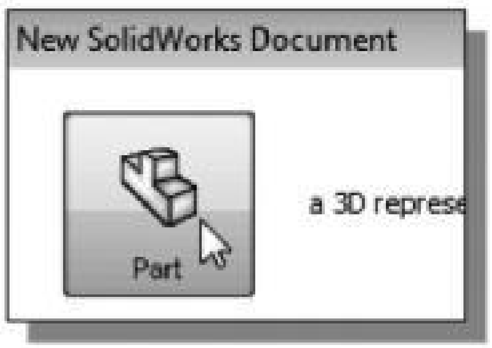
A beam element positioned in two-dimensional space is typically modeled to possess three nodal displacements (two translational and one rotational) at each node: three degrees of freedom at each node. Each element therefore possesses six degrees of freedom. A finite element analysis using beam elements typically provides a solution to the displacements, reaction forces, and moments at each node. The formulation of the beam element is based on the elastic beam theory, which implies the beam element is initially straight, linearly elastic, and loads (forces and moments) are applied at the ends. Therefore, in modeling considerations, place nodes at all locations that concentrated forces and moments are applied. For a distributed load, most finite element procedures replace the distributed load with an equivalent load set, which is applied to the nodes available along the beam span. Accordingly, in modeling considerations, place more nodes along the beam spans with distributed loads to lessen the errors.

Starting *SolidWorks*

1. Select the **SolidWorks** option on the *Start* menu or select the **SolidWorks** icon on the desktop to start *SolidWorks*. The *SolidWorks* main window will appear on the screen.



2. Click on the **New** icon located in the *Standard* toolbar as shown.

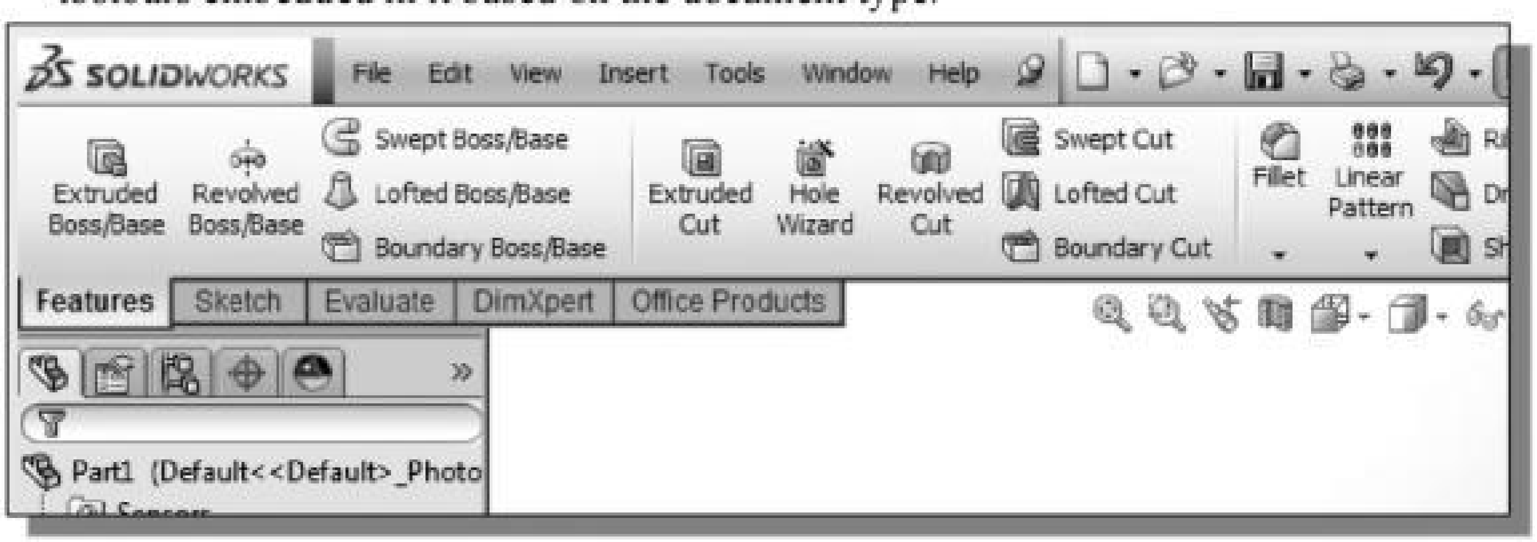


3. Select **Part** by clicking on the first icon in the *New SolidWorks Document* dialog box as shown.

4. Click on the **OK** button to accept the settings.

5. In the *Standard* toolbar area, right-mouse-click on any icon and select **Command Manager** in the option list as shown.

❖ The ***Command Manager*** is a context-sensitive toolbar that dynamically updates based on the user's selection. When you click a tab below the *Command Manager*, it updates to display the corresponding toolbar. By default, the *Command Manager* has toolbars embedded in it based on the document type.

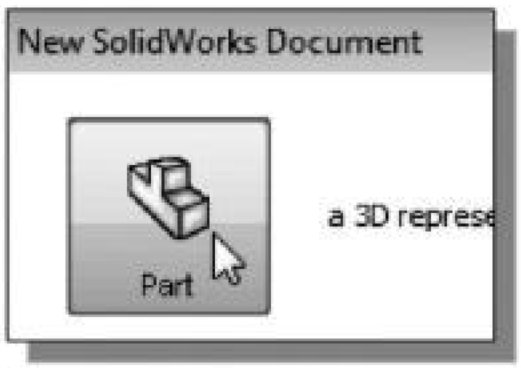


You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Creating a Rectangle *Weldment Profile*

❖ In *SolidWorks*, all structural members use **weldment profiles**, which provide the definitions of the properties of the associated cross sections. *Weldment Profiles* are identified by **Standard**, **Type**, and **Size**. In this section, we will illustrate the procedure to create new profiles and add them to the existing library.

1. Click on the **New** icon, located in the *Standard* toolbar as shown.

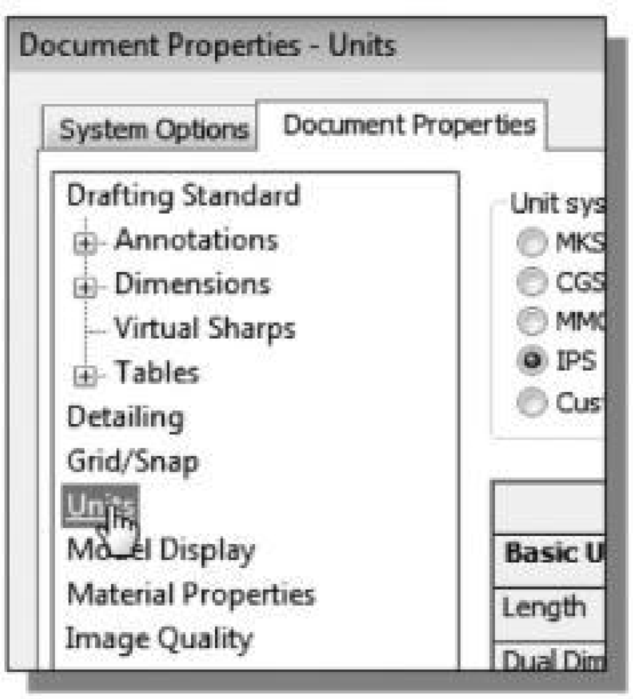


2. Select **Part** by clicking on the first icon in the *New SolidWorks Document* dialog box as shown.

3. Click on the **OK** button to accept the settings.

4. Click on the **Options** icon from the *Menu Bar* toolbar to open the *Options* dialog box.

5. Select the **Document Properties** tab as shown in the figure.



6. Click **Units** as shown in the figure.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Creating Structural Members using the new profile

1. Select **Insert → Weldments → Structural Member** in the pull-down menu as shown.

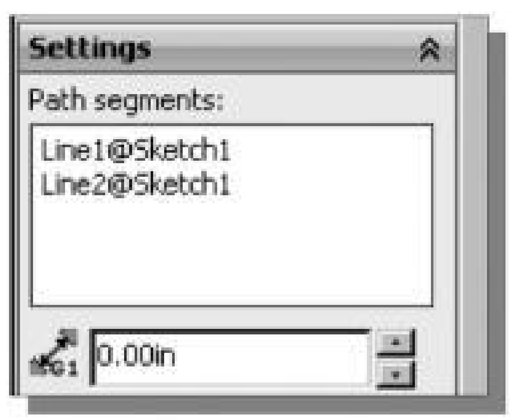
❖ Note a *Weldment* feature is automatically added in the *Model History Tree* when the **Structural Member** command is activated.

2. In the *Structural Member Property Manager* dialog box, set the *Standard* option to **ansi inch** as shown.

3. In the *Structural Member Property Manager* dialog box, set the *Type* option to **solid** as shown.

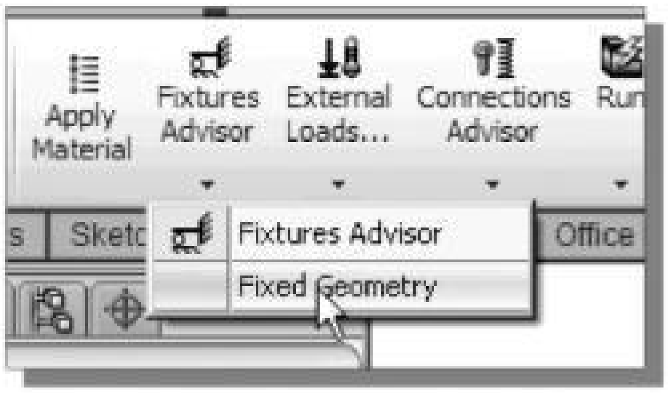
4. In the *Structural Member Property Manager* dialog box, set the *Size* option to **Rectangular** as shown.

5. Select the **two line segments** of the structure as shown.

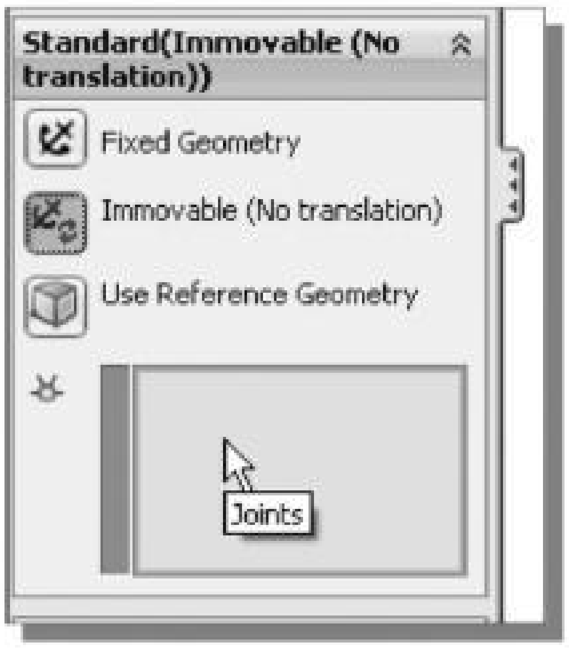


6. Click **OK** to create the structural member, which contains two segments with same profile.

Applying Boundary Conditions – Constraints



1. Choose **Fixed Geometry** by clicking the icon in the toolbar as shown.

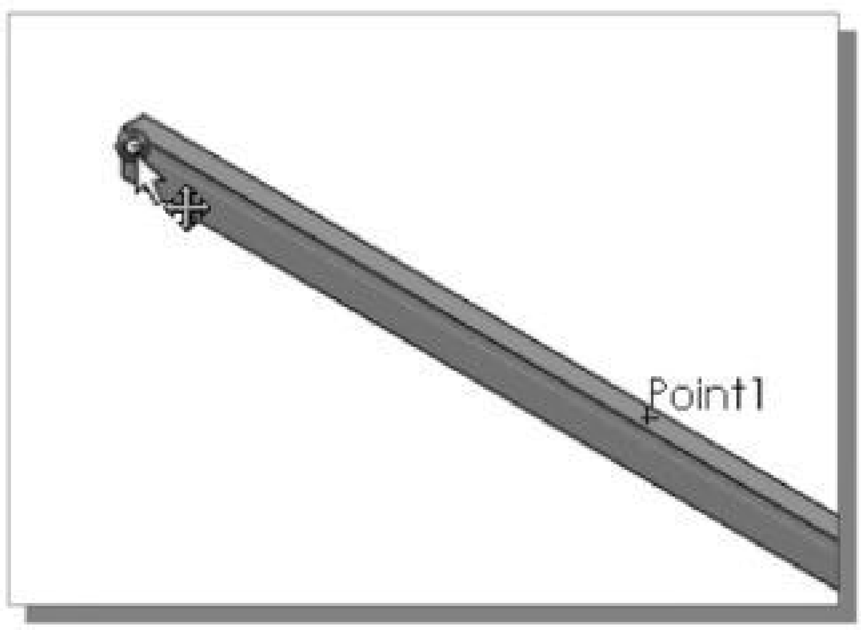


2. In the *Standard* option list, choose **Immovable (No translation)**.

❖ For beam systems, all joints are treated as **fixed** by default, locking all six degrees of freedom.

3. Activate the *Joints* list option box by clicking on the inside of the **Joints** list box as shown.

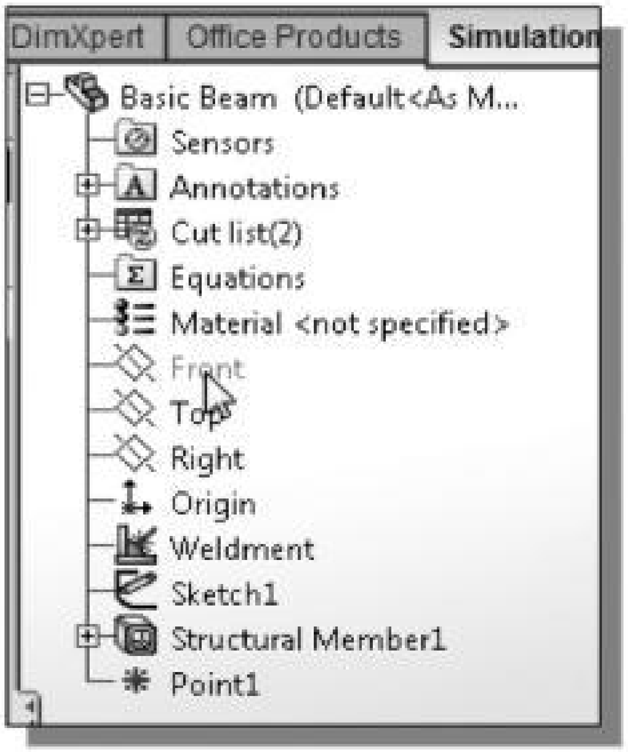
4. Select the **left node** as shown.



5. Click on the **OK** button to accept the first **Fixture** constraint settings.

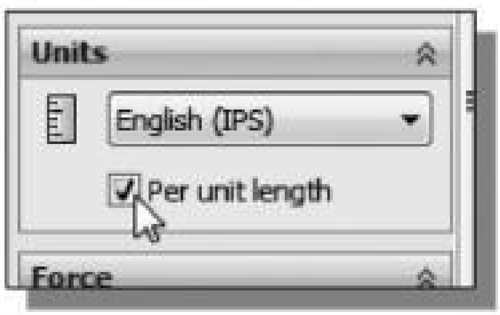
❖ The small arrows indicate constraints have been applied to the associated node.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



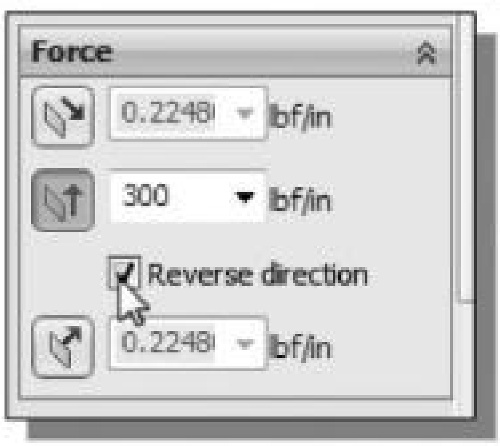
5. Select *Front Plane* as the *direction reference* as shown.

❖ Note the selected plane is highlighted; the constraints we set will be using the selected reference to determine the constraint direction.



6. Set the *Units* option to **IPS** to match with the systems units we are using, as shown.

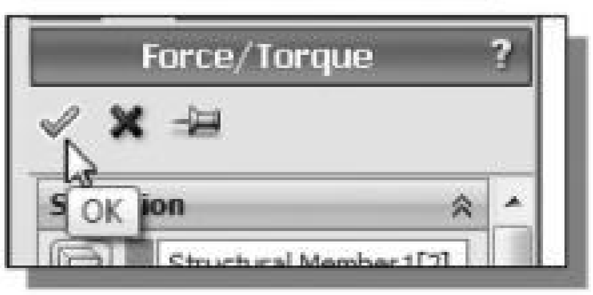
7. Activate the **Per unit length** option as shown.



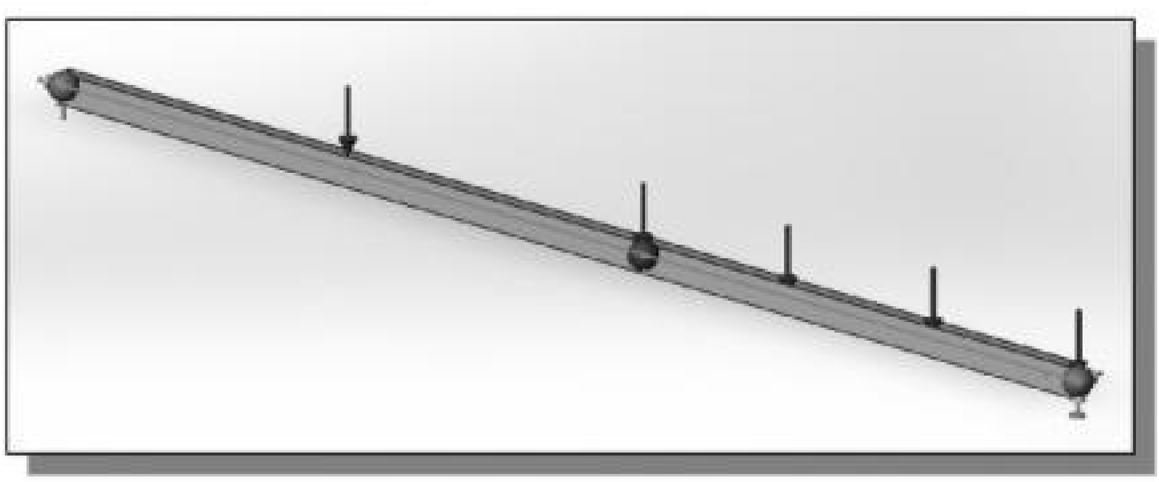
8. Click on the **Along Plane Dir 2** icon to activate the force direction.

9. Set the *Force* to **300** as shown.

10. Activate the **Reverse direction** option.



11. Click on the **OK** button to accept the *Force/Torque* settings.



Questions:

1. For a beam element in three-dimensional space, what is the number of degrees of freedom it possesses?

2. What are the assumptions for the beam element?

3. What are the differences between Truss Element and Beam Element?

4. What are the relationships between the shear diagram and the moment diagram?

5. Can we apply both a point load and a distributed load to the same beam element?

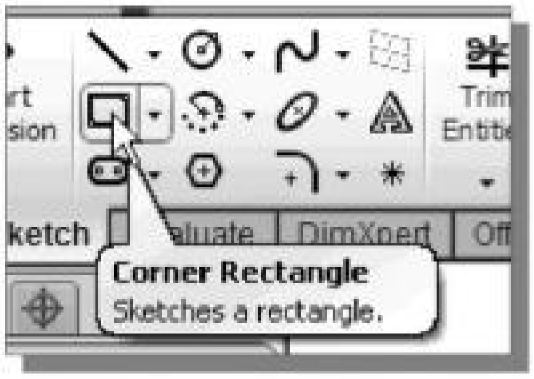
6. For a 2D roller support in a beam system, how should we set the constraints in *SolidWorks Simulation*?

7. List and describe the general procedure to display the moment diagram in *SolidWorks Simulation*.

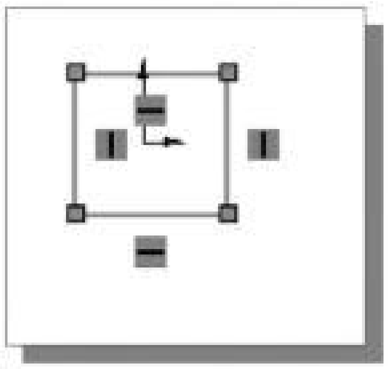
8. How are Moment in Dir 2 and Shear Force in Dir 1 determined in *SolidWorks Simulation*?

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

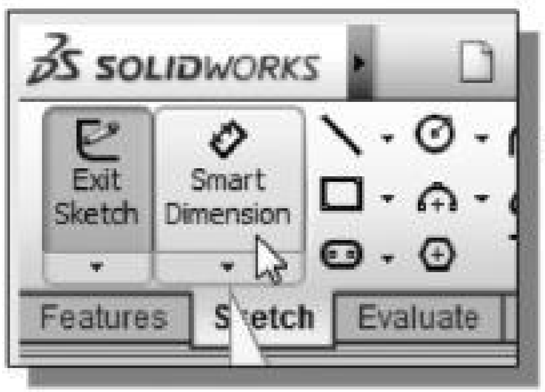
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



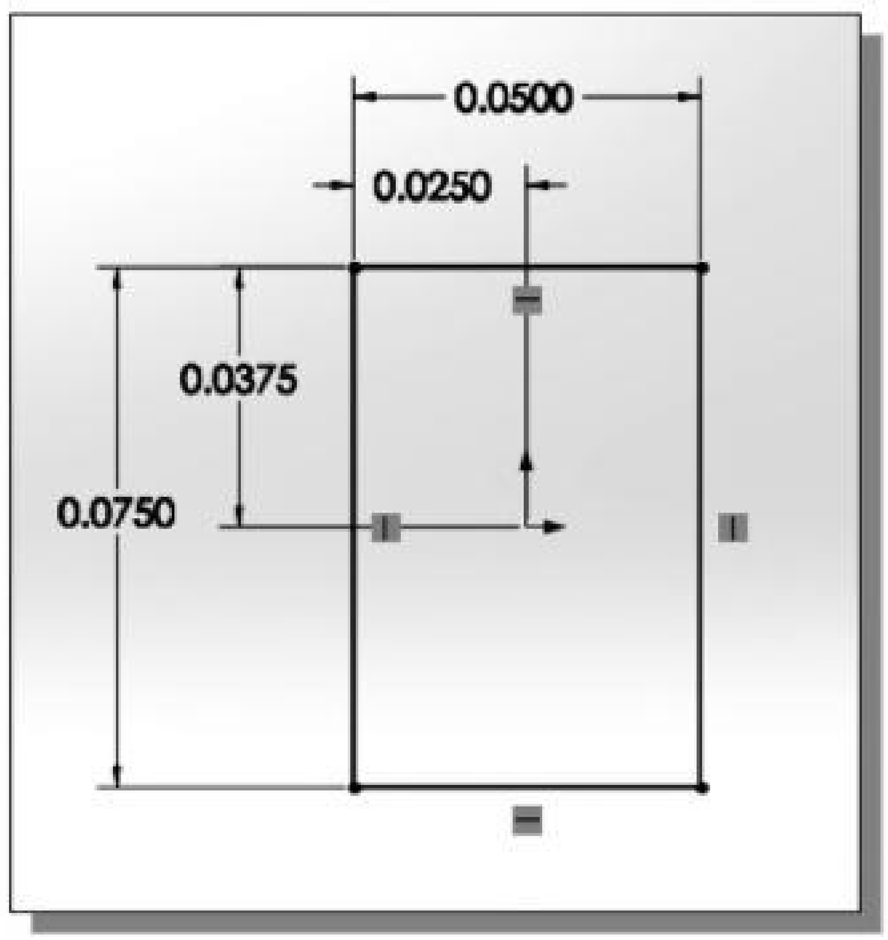
12. Click the **Corner Rectangle** icon to activate the **Rectangle** command.

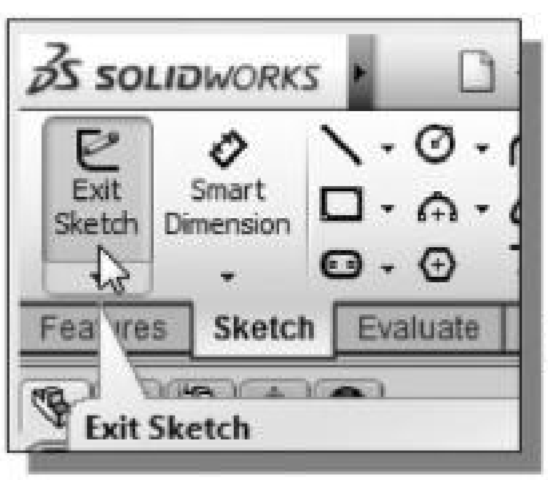


13. On your own, create a rectangle roughly centered at the origin of the world coordinate system.



14. On your own, use the **Smart Dimension** to adjust the sketch as shown.





15. Click the **Exit Sketch** icon in the *Sketch* toolbar as shown.

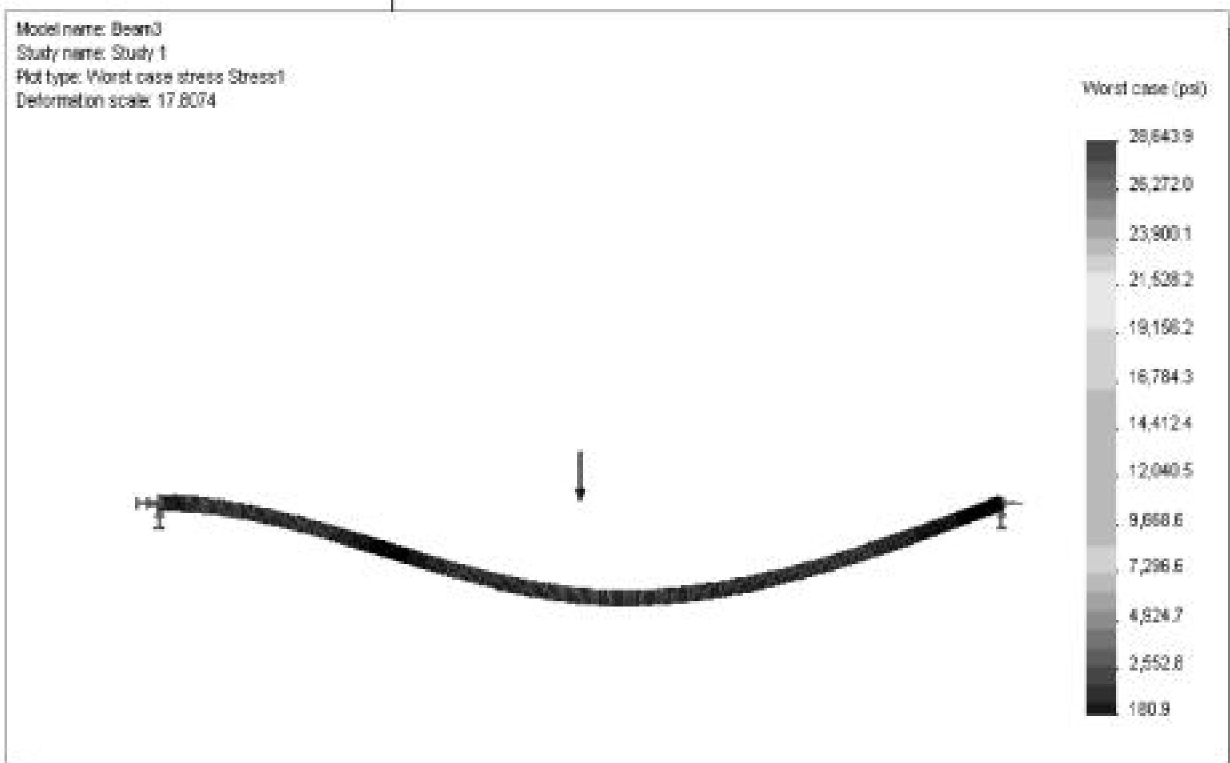
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Chapter 9
Statically Indeterminate Structures

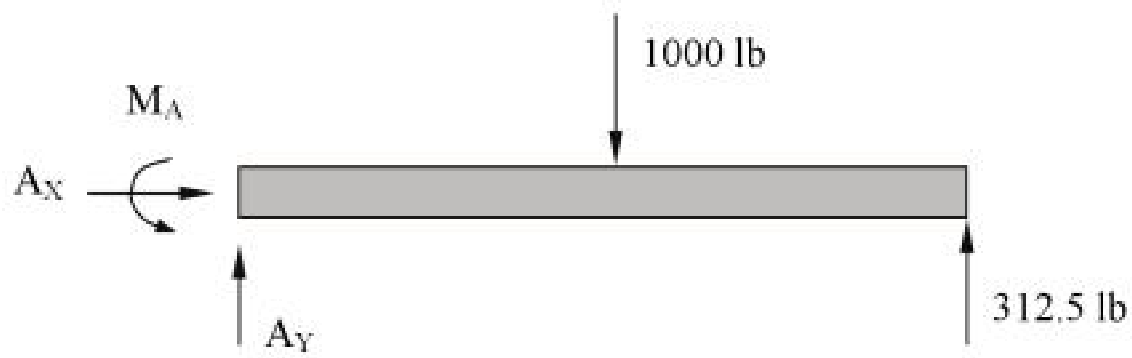


Learning Objectives

- Perform Statically Indeterminate Beam Analysis.
- Understand and Apply the Principle of Superposition.
- Identify Statically Indeterminate Structures.
- Apply and Modify Boundary Conditions on Beams.
- Generate Shear and Moment diagrams.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Free Body Diagram of the system:



The reactions at A can now be solved:

$$\sum F_X = 0, \quad A_X = 0$$

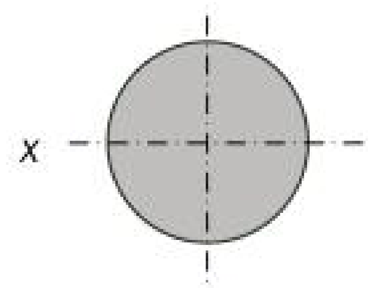
$$\sum F_Y = 0, \quad 312.5 - 1000 + A_Y = 0, \quad A_Y = 687.5 \text{ lb.}$$

$$\sum M_{@A} = 0, \; 312.5 \times 10 - 1000 \times 5 + M_A = 0, \; M_A = 1875 \text{ ft-lb}$$

The maximum normal stress at point A is from the bending stress:

$$\sigma = \frac{MC}{I}$$

for the circular cross section:



$$I_{X\text{-}X} = \frac{\pi r^4}{4}$$

$$C = r$$

Therefore,

$$\sigma_A = \frac{MC}{I} = \frac{4\,M\,r}{\pi\, r^4} = 4.125 \times 10^6 \text{ lb/ft}^2 = 28648 \text{ lb/in}^2$$

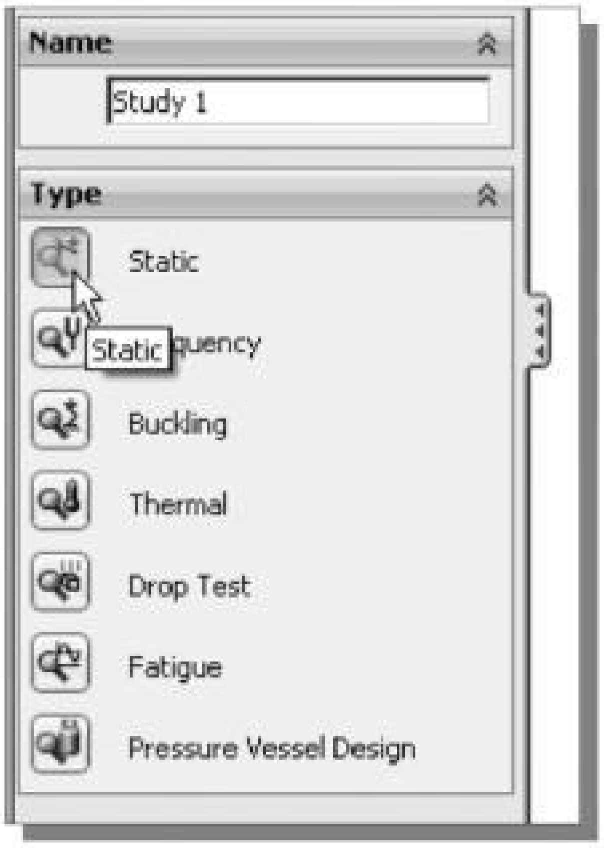
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

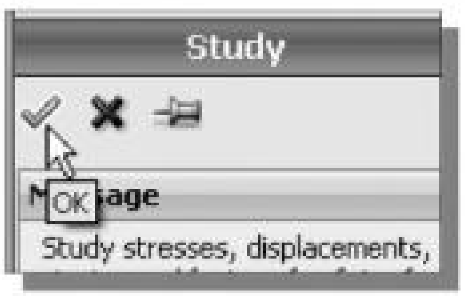
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



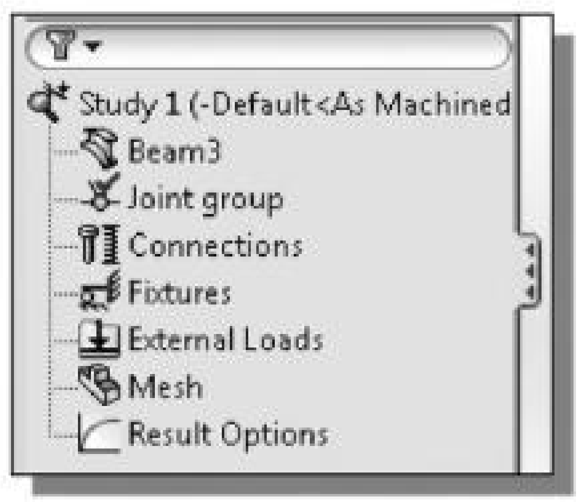
5. Select **Static** as the type of analysis to be performed with *SolidWorks Simulation*.

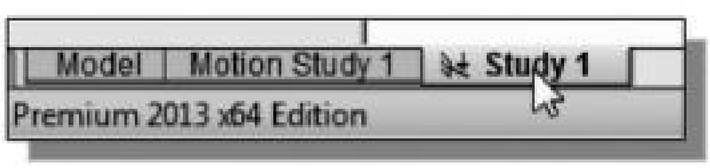
❖ Note that different types of analyses are available, which include both structural static and dynamic analyses, as well as the thermal analysis.



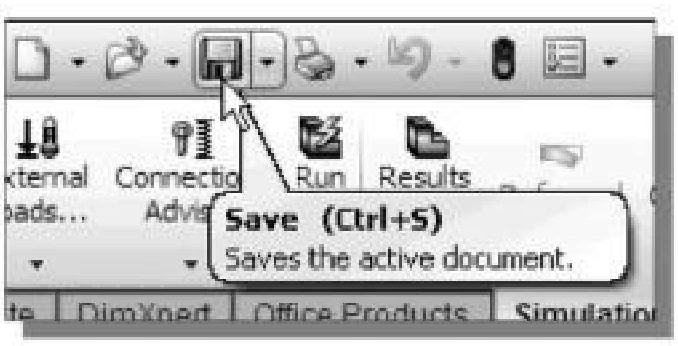
6. Click **OK** to start the definition of a structural static analysis.

❖ In the *Feature Manager* area, note that a new panel, the *FEA Study* window, is displayed with all the key items listed.





❖ Also note the **Study 1** tab is activated, which indicates the use of the FEA model.



7. On your own, use the **Save** command to save the current model (**Beam3**).

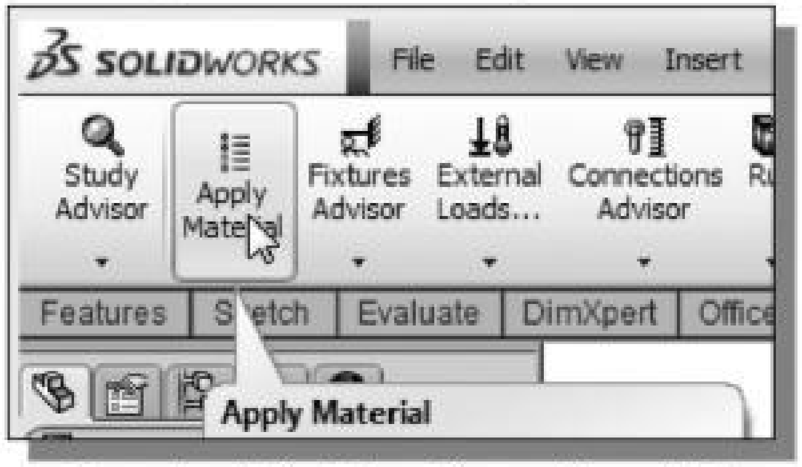
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Assign the Element Material Property

➢ Next we will set up the *Material Property* for the elements. The *Material Property* contains the general material information, such as *Modulus of Elasticity*, *Poisson's Ratio*, etc. that is necessary for the FEA analysis.

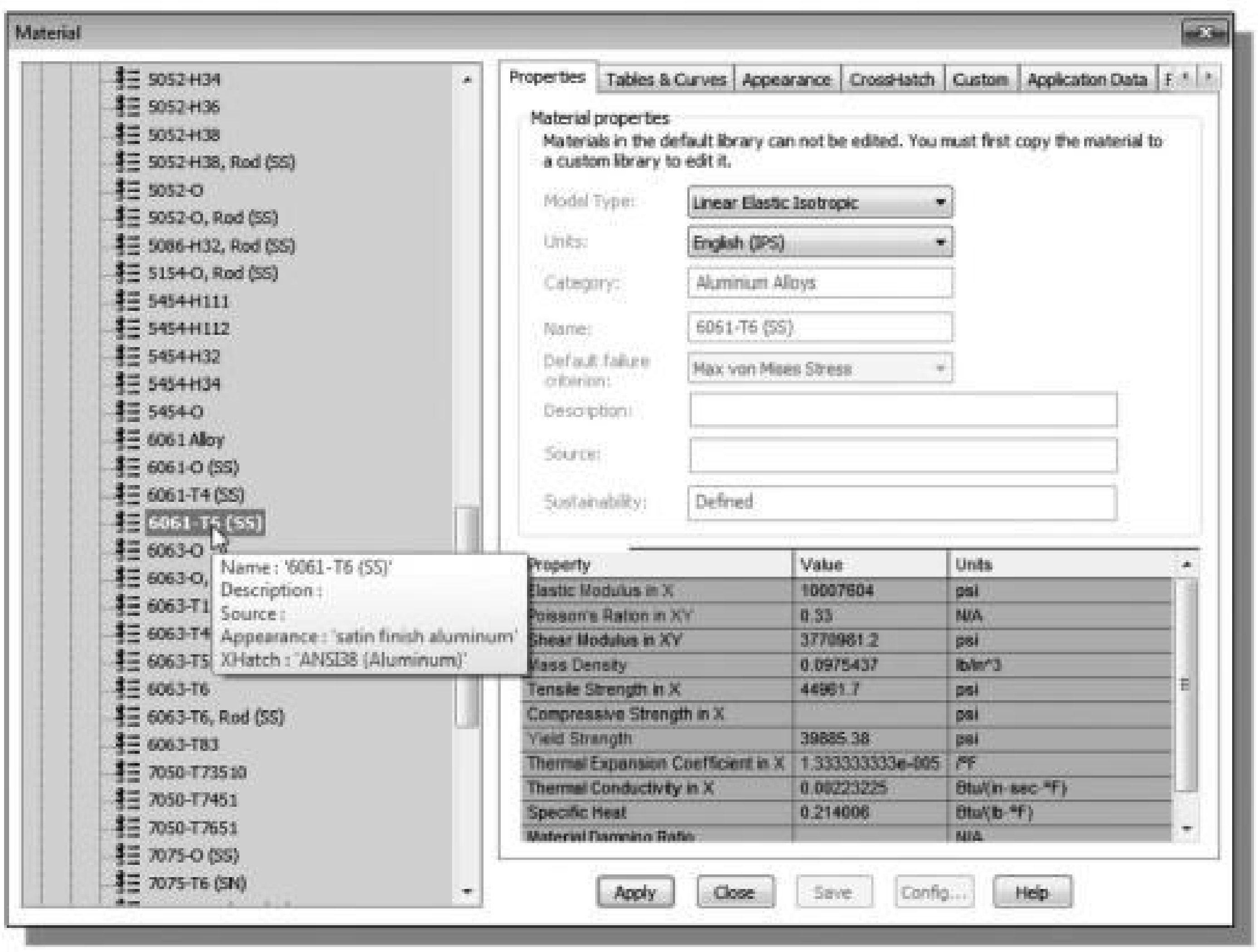


1. Choose the **Apply Materials** option from the pull-down menu as shown.

❖ Note the default list of materials, which are available in the pre-defined *SolidWorks Simulation* material library, is displayed.

2. Select **6061-T6 Aluminum** in the *Material* list as shown.

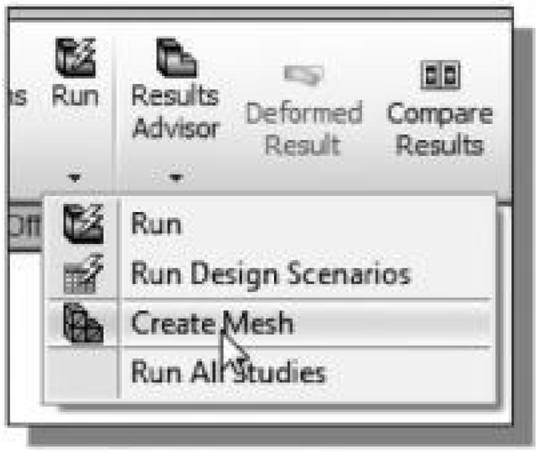
3. Set the **Units** option to display **English (IPS)** to make the selected material available for use in the current FEA model.



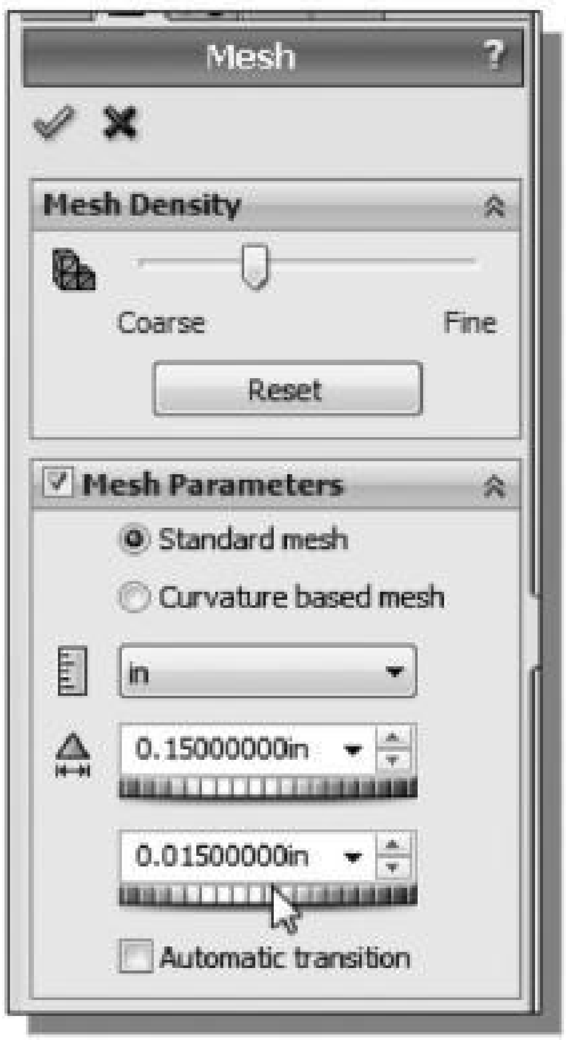
4. Click **Apply** to assign the material property then click **Close** to exit the material assignment command.

Create the first FEA Mesh – Coarse Mesh

- As a rule in creating the first FEA mesh in using the H-element approach is to start with a relatively small number of elements and progressively move to more refined models. The main objective of the first coarse mesh analysis is to obtain a rough idea of the overall stress distribution. In most cases, use of a complex and/or a very refined FEA model is not justifiable since it most likely provides computational accuracy at the expense of unnecessarily increased processing time.



1. Choose **Create Mesh** by clicking the icon in the toolbar as shown.



2. Switch on the **Mesh Parameters** options to show the additional control options.
3. Set the *Units* to **inches** as shown.
4. Enter ***0.15*** inch as the *Global Element size*.
5. Enter ***0.015*** inch as the *Size tolerance*.

- A rule of thumb to use as the first mesh is to have at least 4 to 5 elements on the edges of the model. The shortest edge in our model is about 0.6 inches, so we will use 0.15 as the element size.

6. Click on the **OK** button to accept the *Mesh* settings.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

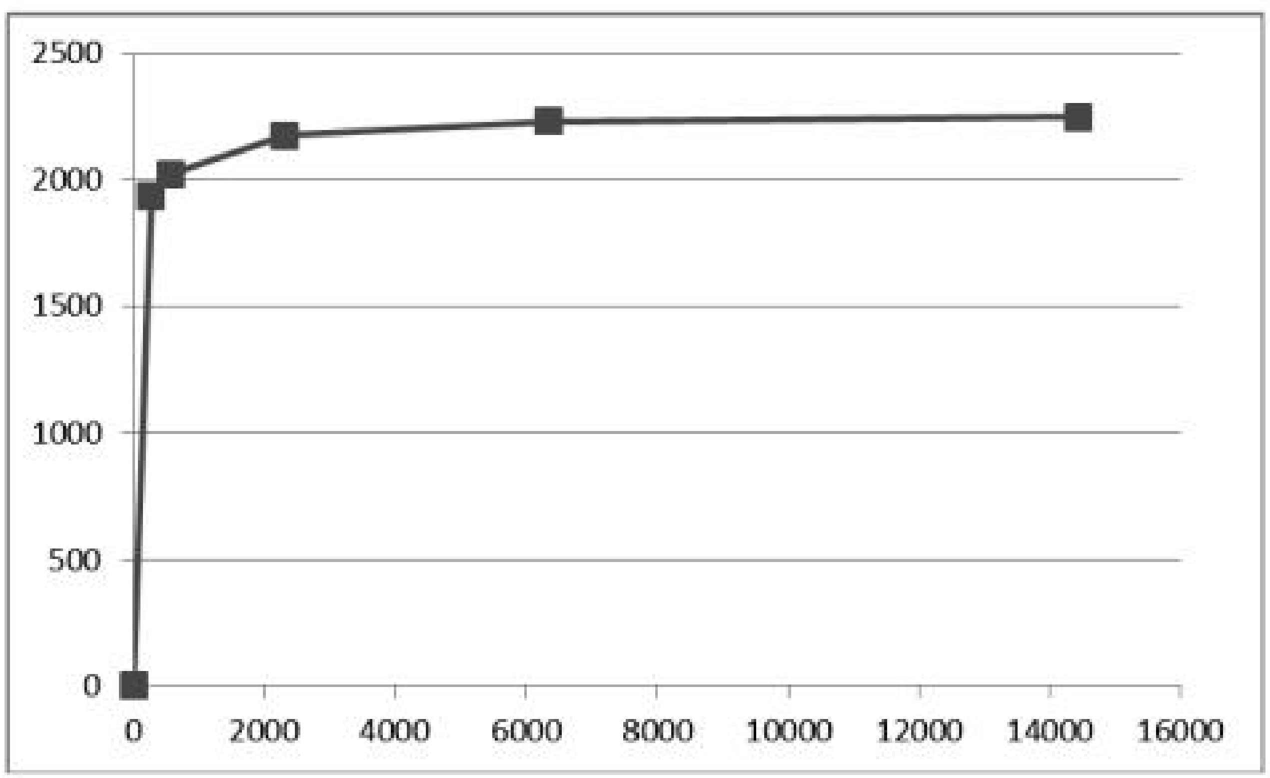
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Comparison of Results

The accuracy of the *SolidWorks Simulation* results for this problem can be checked by comparing them to the analytical results presented earlier. In the Preliminary Analysis section, the maximum stress was calculated using a stress concentration factor and the value obtained was **2180 psi**. One should realize the analytical result is obtained through the use of charts from empirical data and therefore involves some degrees of error. The maximum stress obtained by finite element analysis using *SolidWorks Simulation* ranging from **1938** to **2249 psi**. In the *Preliminary Analysis* section, the maximum displacement was also estimated to be around **1.94E-4 inches**, measured from the center of the hole to one end of the plate. The maximum displacement obtained by finite element analysis using *SolidWorks Simulation* was around **2.050E-4 inches**. The agreement between the analytical results and those from *SolidWorks Simulation* demonstrates the potential of *SolidWorks Simulation* as a very powerful design tool.

In FEA, the process of mesh refinement is called convergence analysis, or H-convergence. For our analysis, the refinement of the mesh does show the FEA results converging near the analytical results. The refinement to the size of **0.03~0.05 inches** is quite adequate for our analysis. Any further refinement does not provide any additional insight and is therefore not necessary.

Global Element size	Number of Elements	σ_{max} (psi)	D_{max} (in)
0.15	270	1938	2.057e-4
0.10	580	2023	2.054e-4
0.05	2314	2176	2.050e-4
0.03	6363	2232	2.048e-4
0.02	14448	2249	2.047e-4



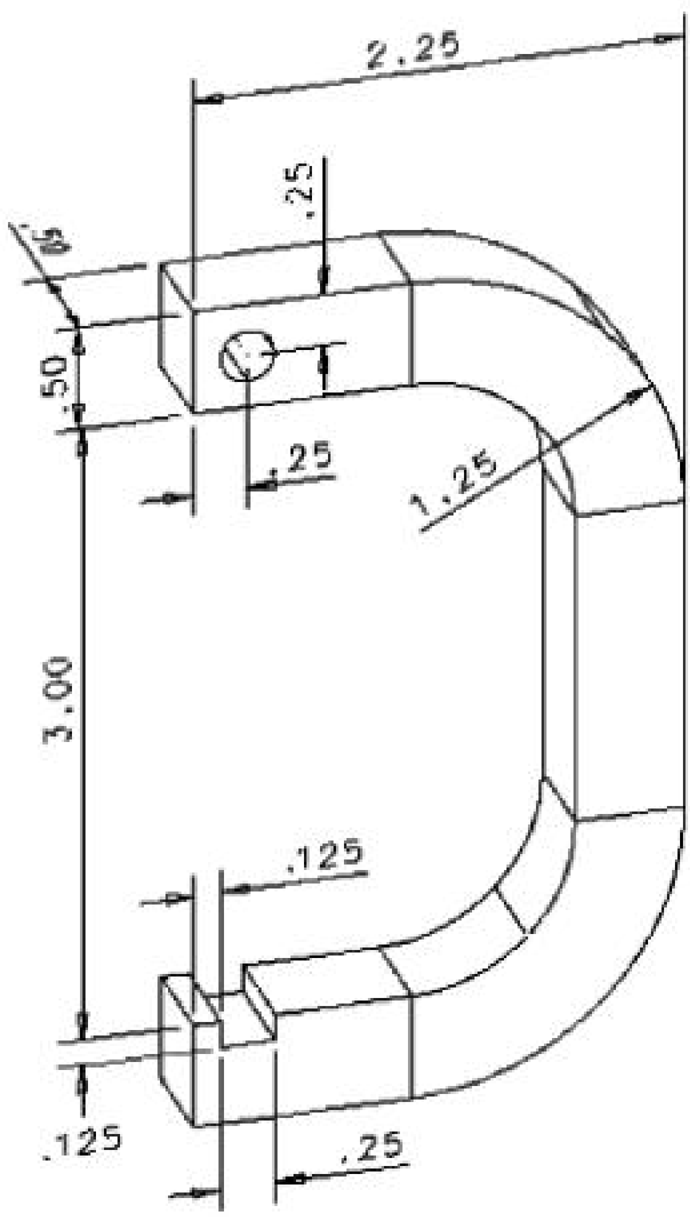
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

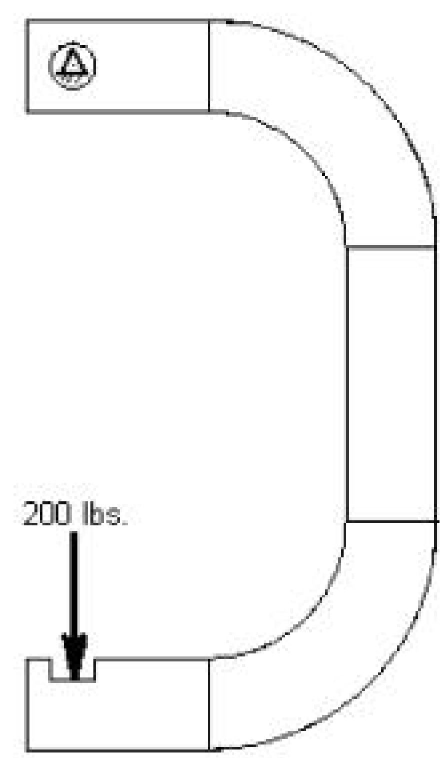
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

This chapter illustrates the general FEA procedure of using three-dimensional solid elements. The creation of a solid model is first illustrated and *solid elements* are generated using the *SolidWorks Simulation* mesh commands. In theory, all designs could be modeled with three-dimensional solid elements. The three-dimensional solid element is the most versatile type of element compared to the more restrictive one-dimensional or two-dimensional elements. The procedure involved in performing a three-dimensional solid FEA analysis is very similar to that of a two-dimensional solid FEA analysis, as was demonstrated in Chapter 10. As one might expect, the number of node-points involved in a typical three-dimensional solid FEA analysis is usually much greater than that of a two-dimensional solid FEA analysis.

Problem Statement

Determine the maximum normal stress in the AL6061 member shown; the c-link design is assembled to the frame at the upper hole (Ø 0.25) and a vertical load of 200 lbs. is applied at the top of the notch on the lower arm as shown.

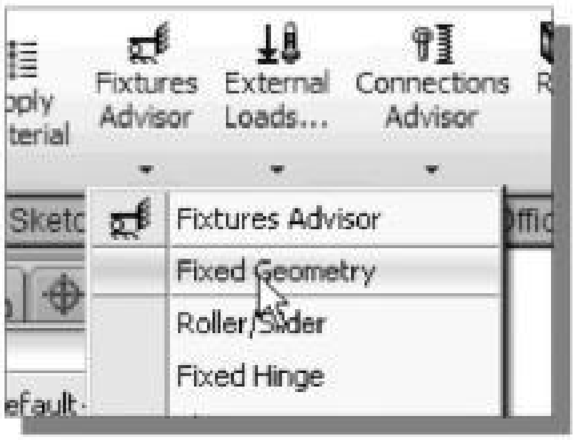




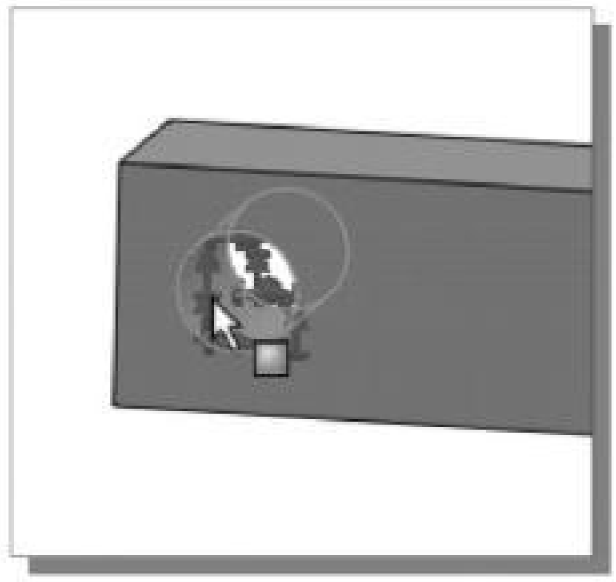
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

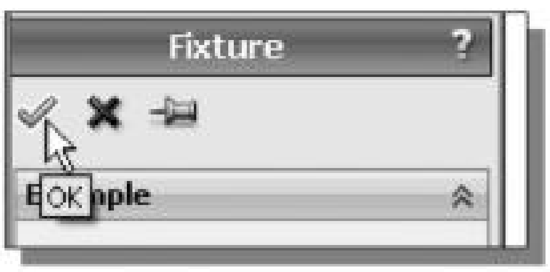
Applying Boundary Conditions – Constraints



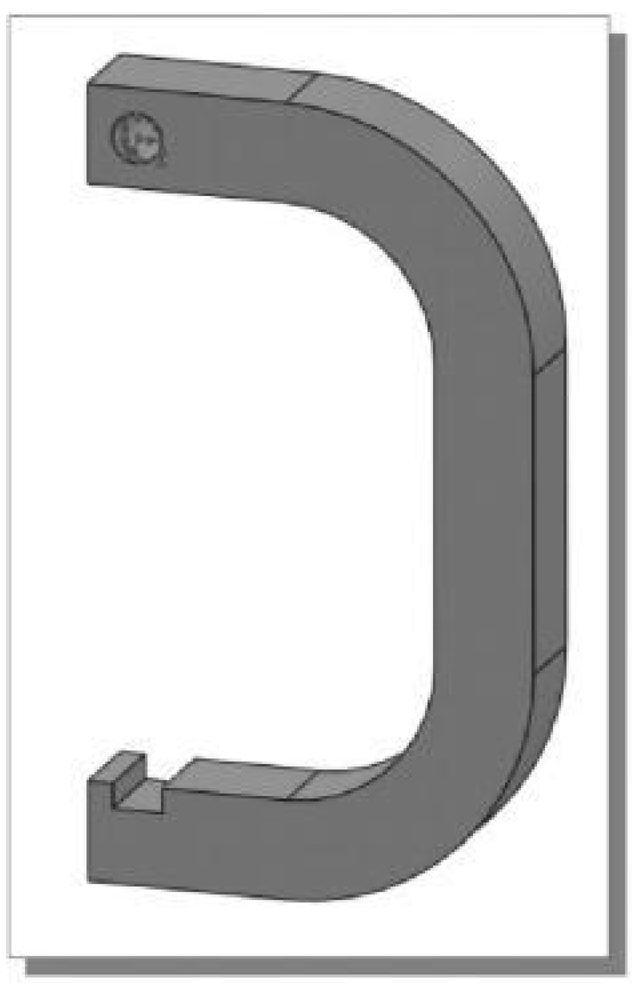
1. Choose **Fixed Geometry** by clicking the fixture icon in the toolbar as shown.



2. Select the **cylindrical surface** of the top portion of the model as the entity to apply the **Fixed** constraint.



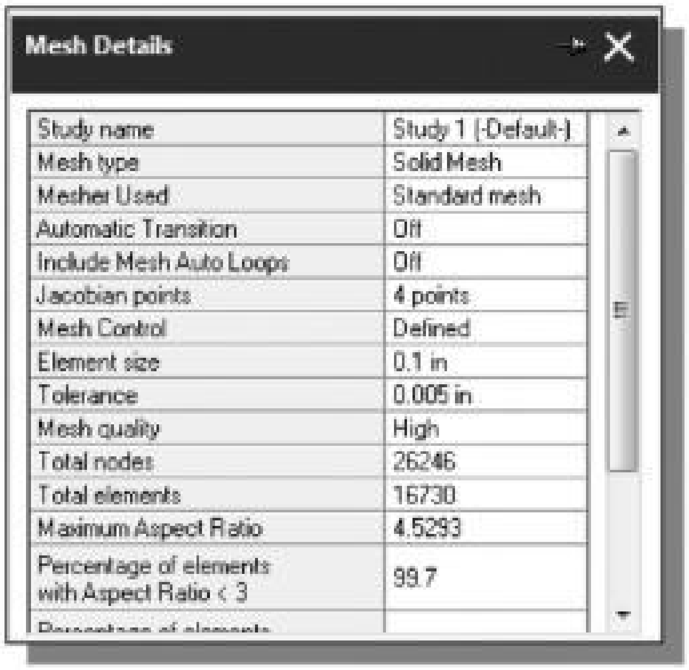
3. Click on the **OK** button to accept the Fixture constraint settings.



You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

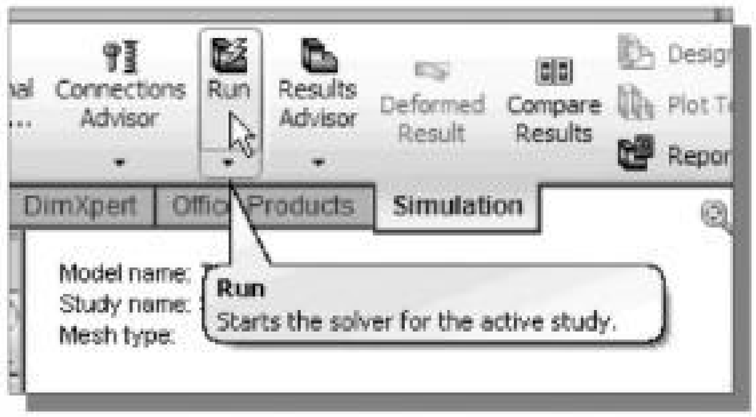
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



10. On your own, examine the details of the refinement.

❖ The current coarse mesh consists of **26246** nodes and **16730** solid elements, which is about 44% increase of elements, compared to the last mesh.

11. Click on the **Run** button to start the *FEA Solver* to calculate the results.



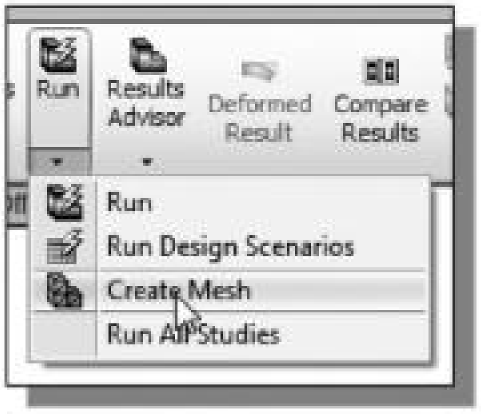
❖ The *FEA Solver* calculated the Max. Von Mises Stress with the refinement to be **21124.4 psi**. The refinement only changes the stress value by 1.6%, which implies the FEA mesh is quite adequate.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

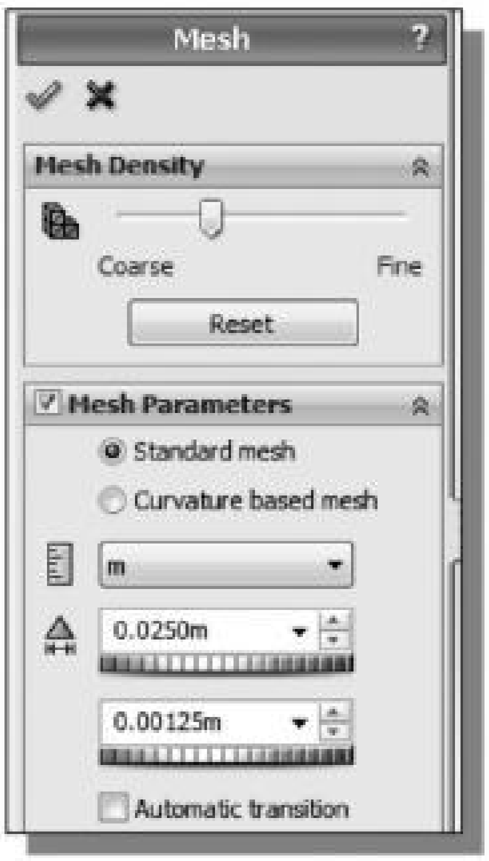
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Create the first FEA Mesh – Coarse Mesh

➢ As a rule in creating the first FEA mesh in using the H-element approach is to start with a relatively small number of elements and progressively move to more refined models. The main objective of the first coarse mesh analysis is to obtain a rough idea of the overall stress distribution.



1. Choose **Create Mesh** by clicking the icon in the toolbar as shown.



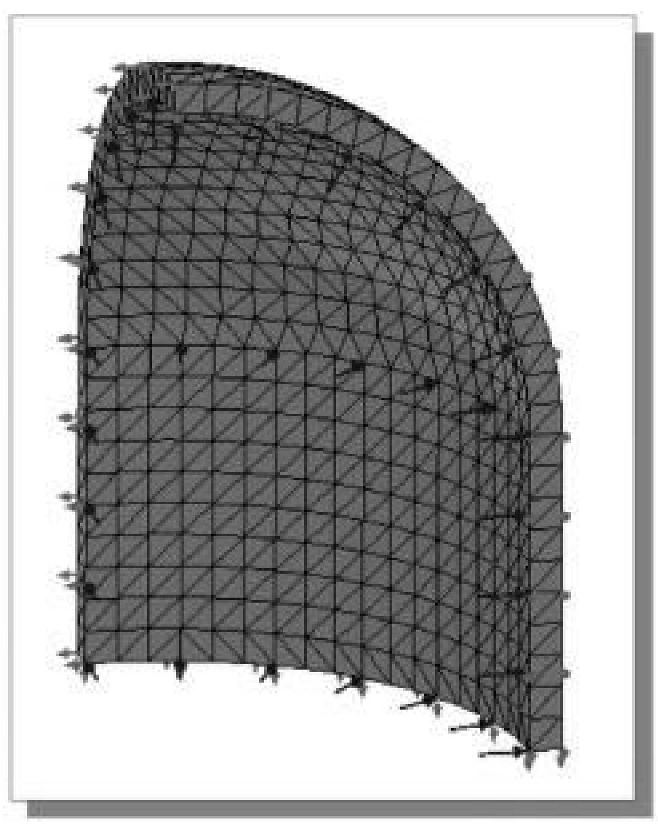
2. Switch on the **Mesh Parameters** options, to show the additional control options.

3. Set the mesh option to **Standard mesh**.

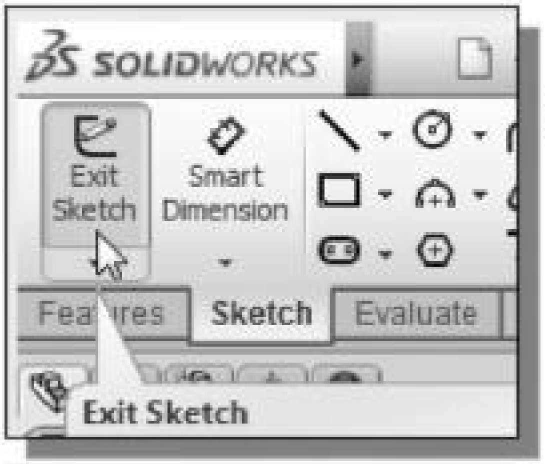
4. Set the *Units* to **meters** as shown.

5. Enter **0.025m** as the *Global Element size*.

❖ In general, a good rule of thumb to follow in creating the first mesh is to have about 3 to 4 elements on the edges of the model. Since our model has fairly small cross section, we will use the wall thickness as the element size.



6. Click on the **OK** button to accept the *Mesh* settings.



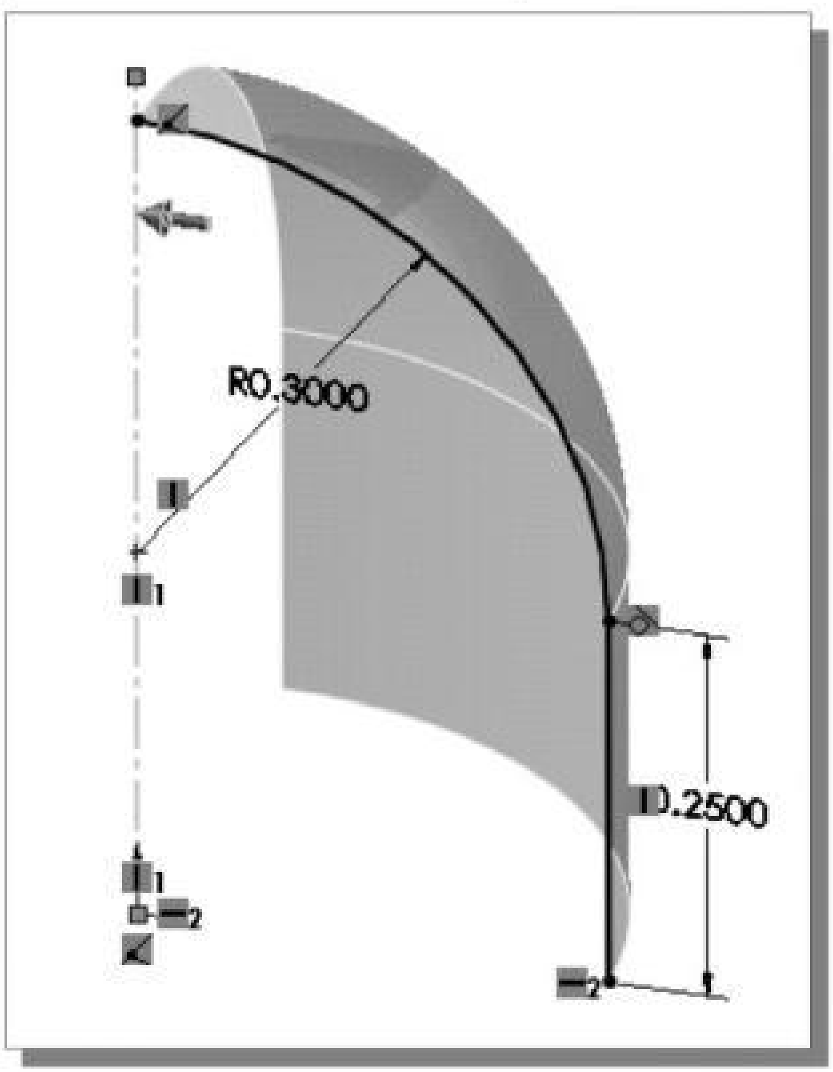
9. Click the **Exit Sketch** icon in the *Sketch* toolbar to exit the *2D Sketch* mode.

10. Confirm the center line is selected as the axis of rotation.

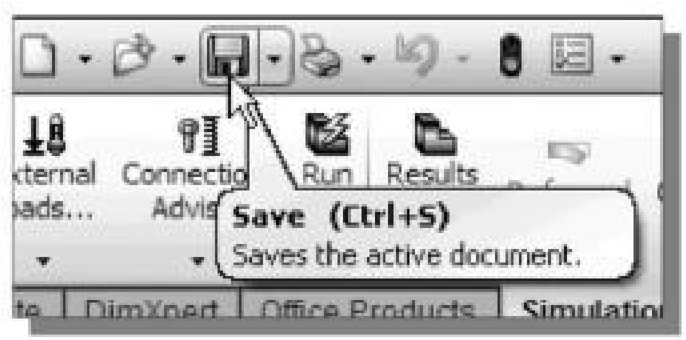
11. If necessary, set the rotation direction by clicking on the **Reverse** icon.

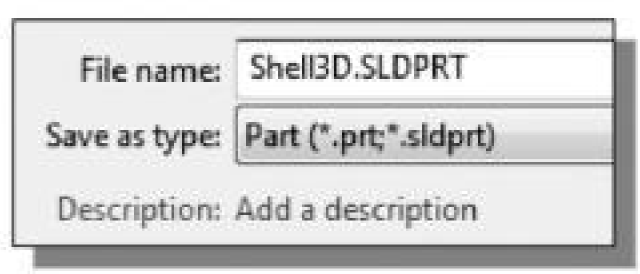
12. Set the *Revolve Angle* to ***90.00* deg** as shown.

13. Click **OK** to proceed with the settings and create the revolved feature.



14. On your own, save a copy of the current model.



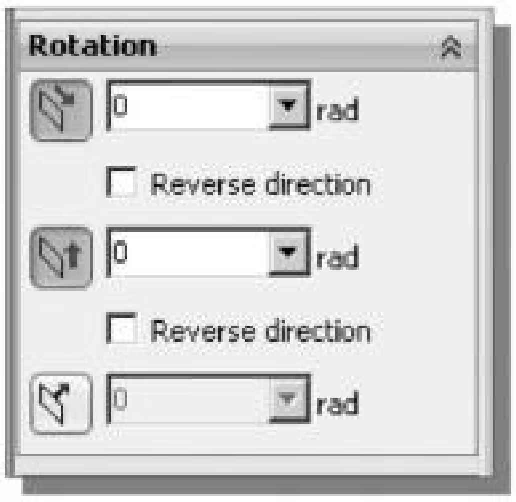


You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

5. Set the distance measurement to **meters**, to match with the systems units we are using.

6. In the *Translation constraints* list, click on the **Normal to Plane** icon to activate the constraint.

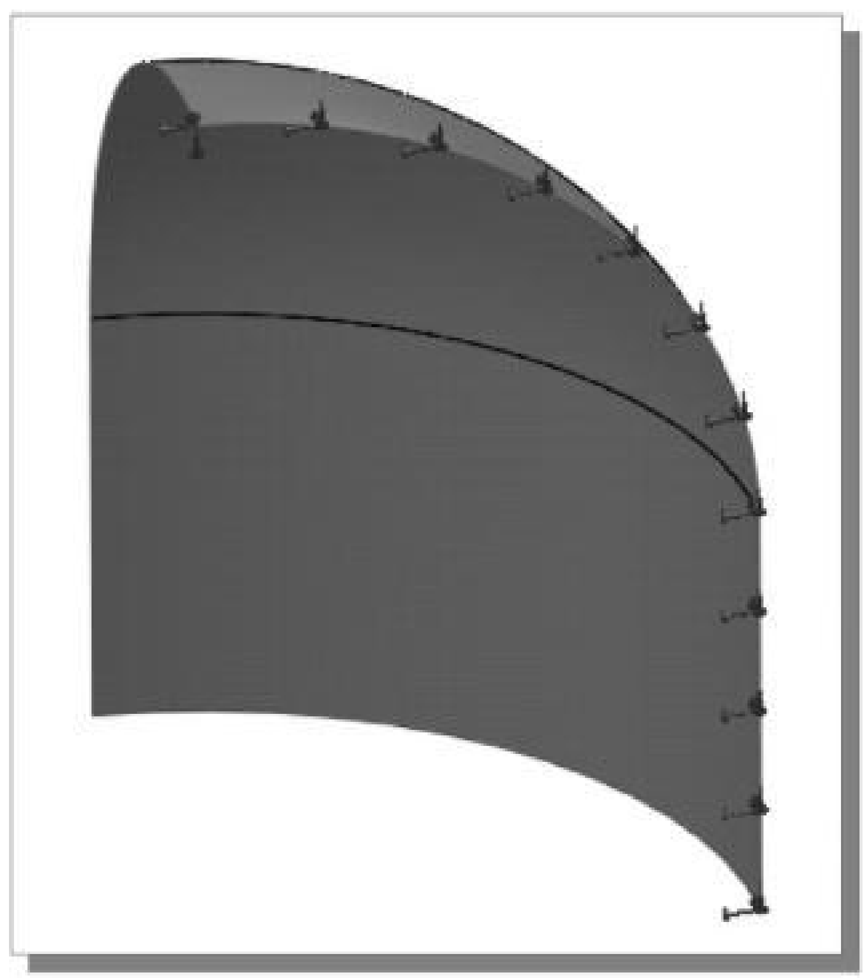
7. Set the *Normal to Plane* distance to **0** as shown.



8. Click on the **Along Plane Dir 1** and **Along Plane Dir 2** icons to activate the constraint.

9. Set the *Along Plane Dir 1* and *Along Plane Dir 2 angles* to **0** as shown.

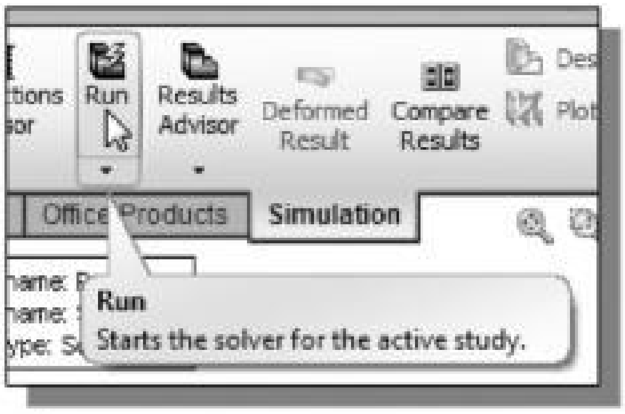
10. Click on the **OK** button to accept the Fixture constraint settings.



You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

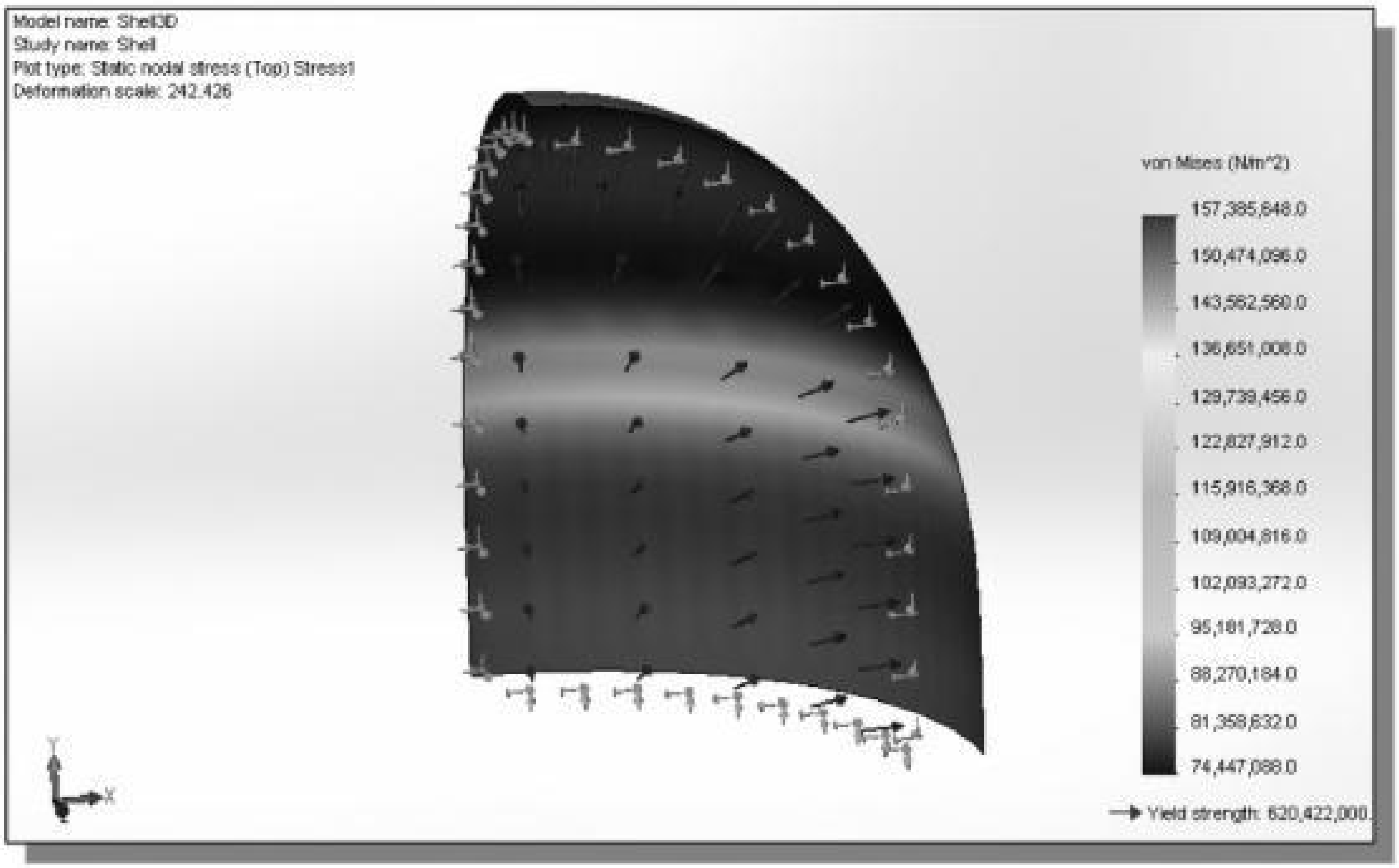
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Run the Solver and view the results



1. Click on the **Run** button to start the *FEA Solver* to calculate the results.

❖ Once the solver has completed the calculations, the display will switch to the stress distribution.



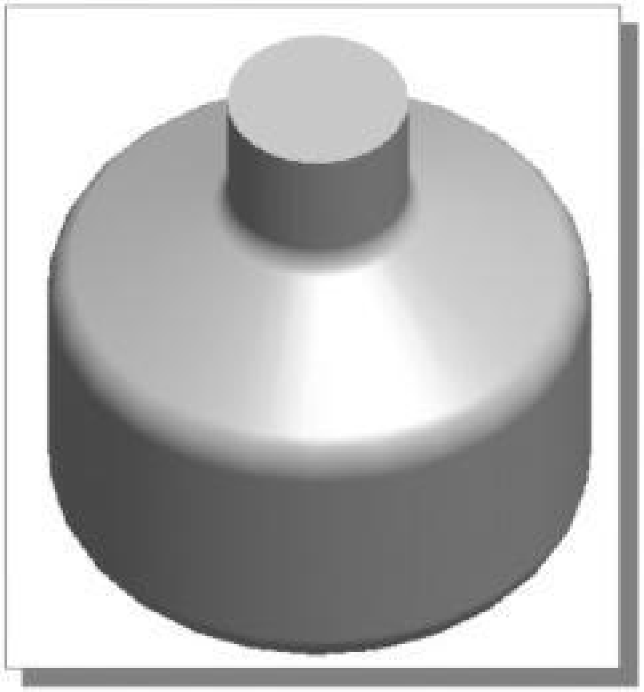
❖ Note the FEA calculated Max. Stress is about 4% lower than the result from the preliminary analysis on page 12-5.

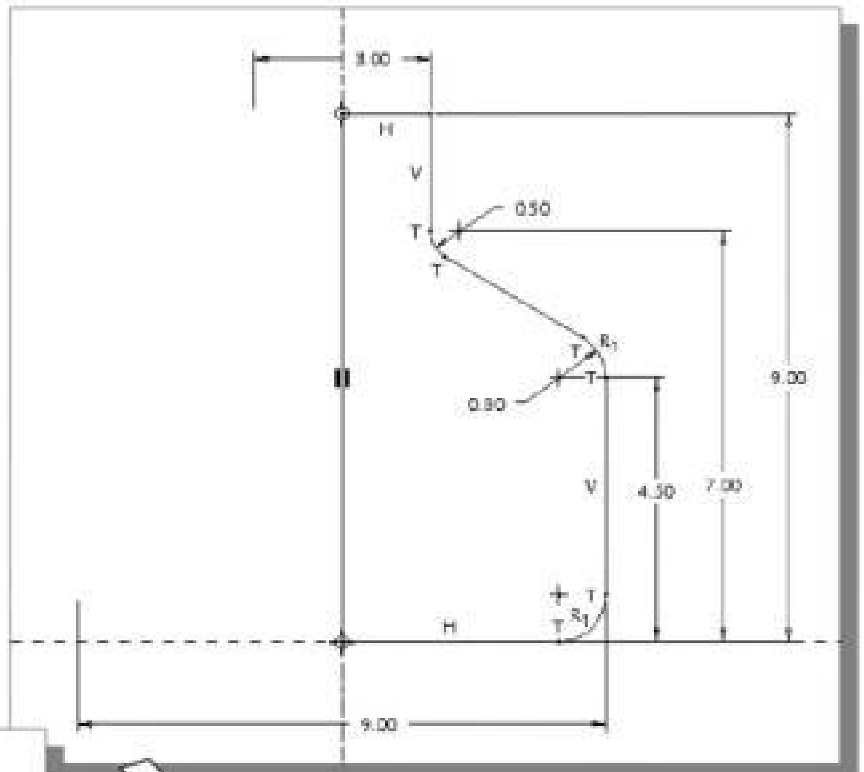
Mesh Details

Study name	Shell (-Default-)
Mesh type	Shell Mesh Using S
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian check for shell	On
Element size	0.025 m
Tolerance	0.00125 m
Mesh quality	High
Total nodes	1672
Total elements	797
Time to complete mesh(hh:mm:ss)	00:00:01

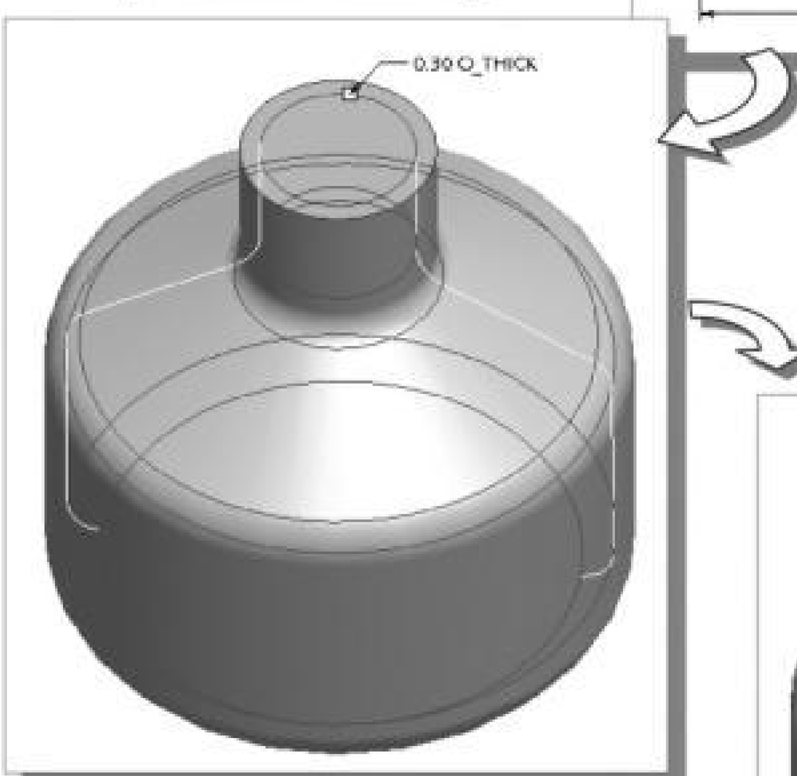
2. On your own, examine the mesh details. Current mesh consists of **1672** nodes and **797** elements, which are only about one quarter of the elements in the solid mesh performed in the previous section. This also indicates the solution time is much less than the solid analysis.

2. Determine the **maximum Von Mises stress** of the thin-wall flat-bottom cylindrical pressure vessel shown in the figure below; dimensions are in inches. The pressure vessel is made of steel and is subject to an **internal pressure of 45Psi**. For the FEA analyses, set all degrees of freedom to **Fixed** at the top edge of the opening.

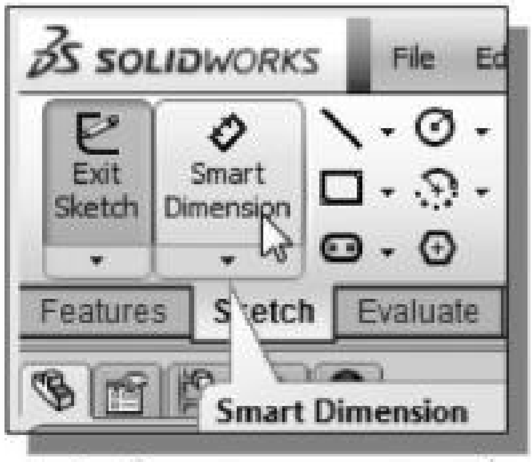




Shell thickness:
0.3

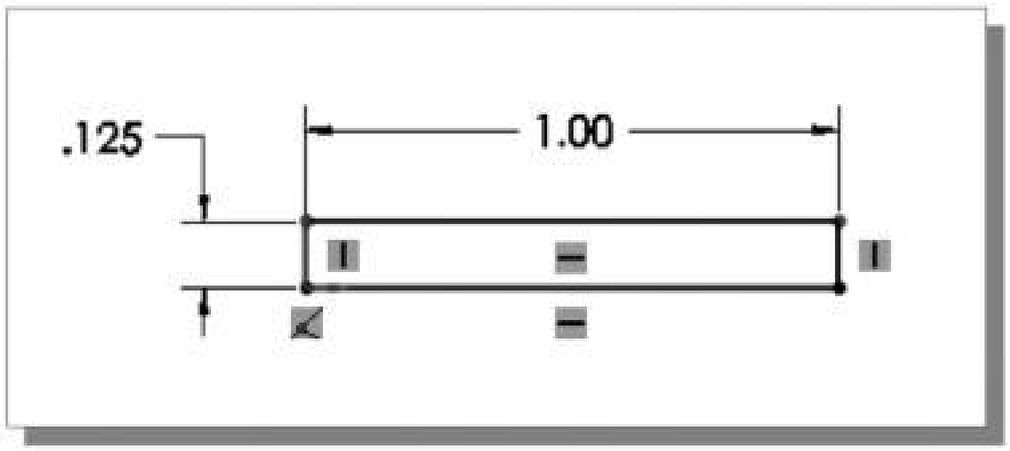


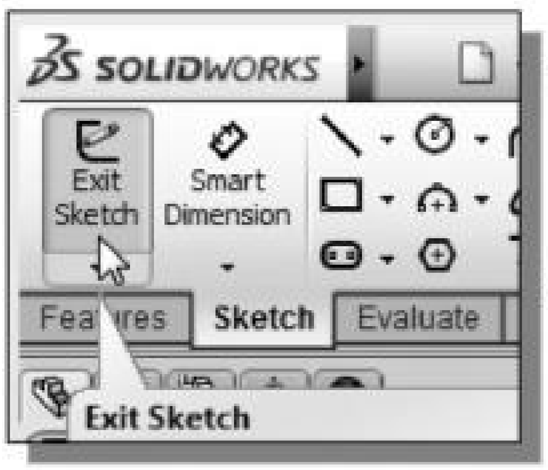
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



5. Click the **Smart Dimension** icon in the *Sketch* toolbar as shown.

6. On your own, create the dimensions and adjust the sketch as shown.





7. Click the **Exit Sketch** icon in the *Sketch* toolbar to exit the *2D Sketch* mode.

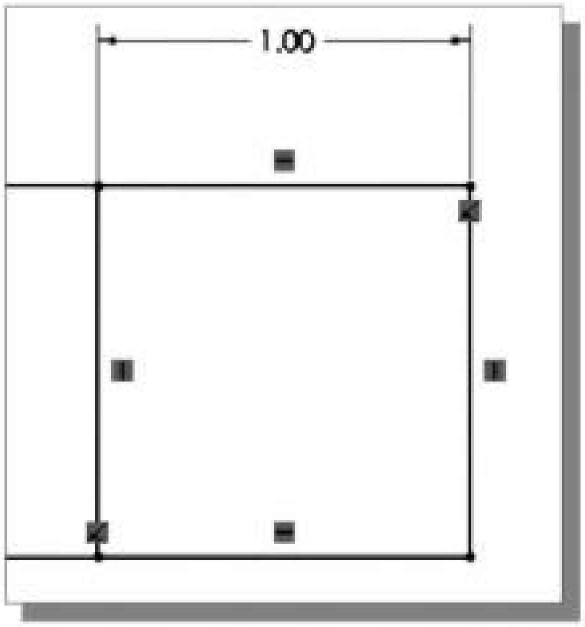
8. On your own, using the extrusion distance of **14.5** in, create the solid feature as shown.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

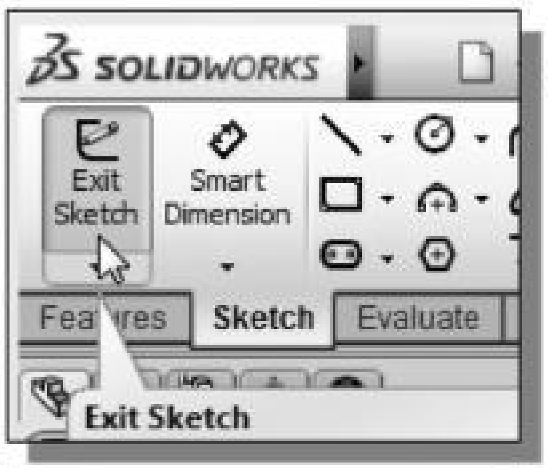
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.



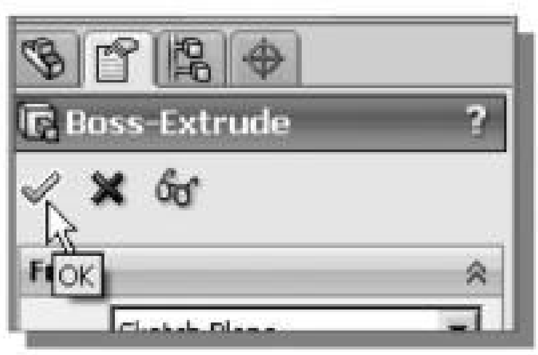
7. On your own, create and adjust the width dimension of the rectangle to **1"**.

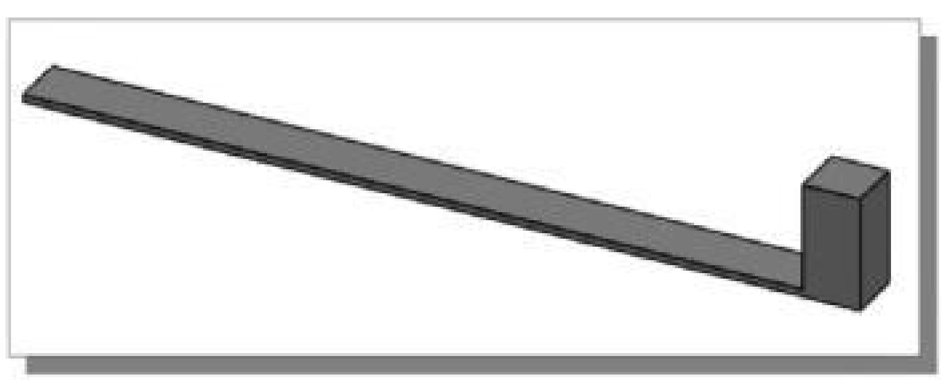


8. Click **Exit Sketch** to accept the completed sketch.

9. Set the *extrusion distance* to **1.77** as shown.

10. Click **OK** to accept the settings and complete the feature.





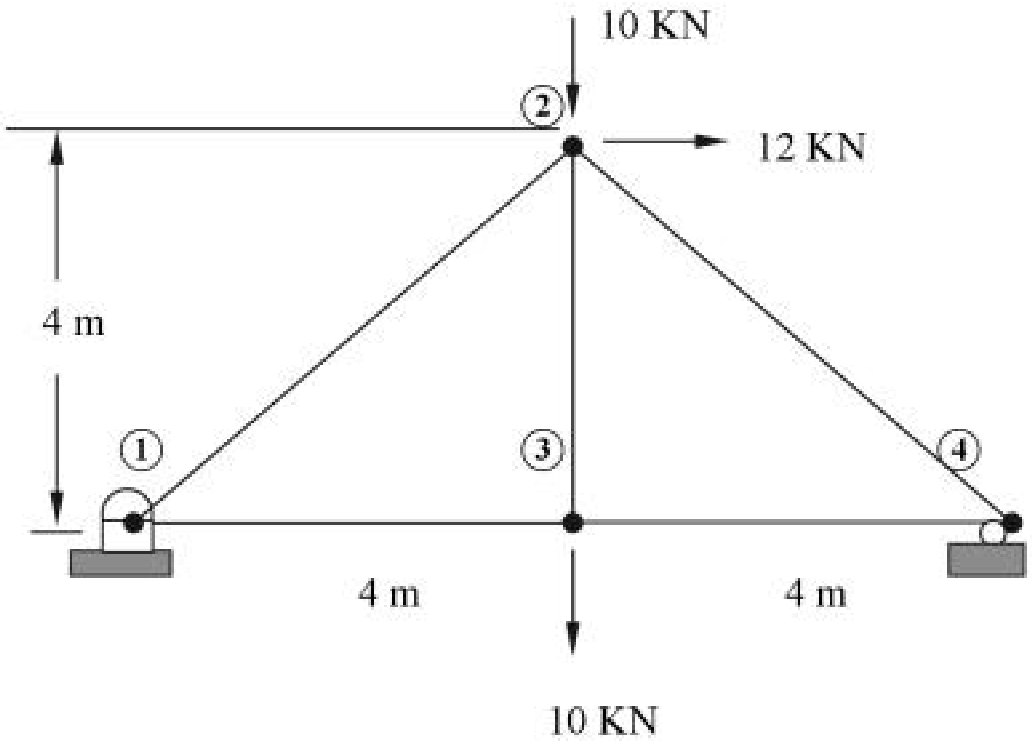
You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Ch. 4 Exercise 2:

2. Material: Steel,
Diameter: 3 cm.



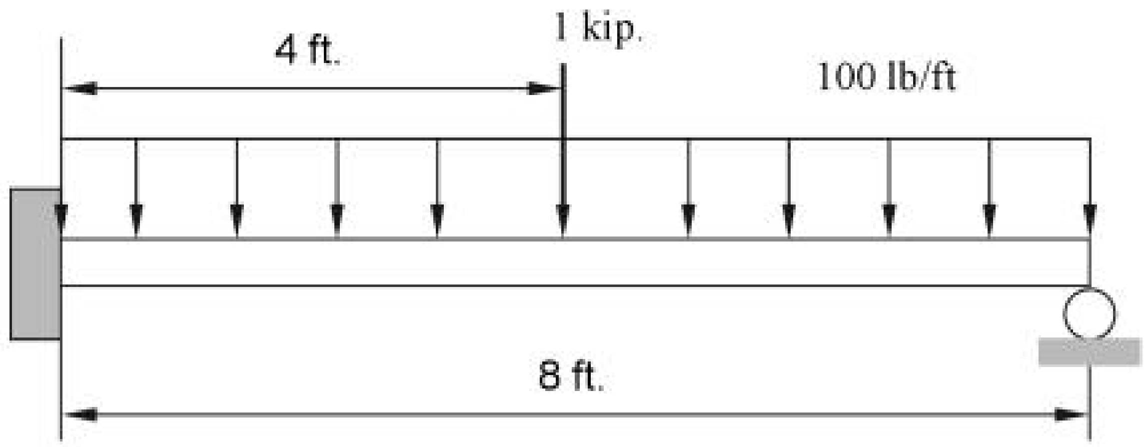
Answers:

X2= 9.326E-04 m, Y2= -1.252E-03 m
X3= 4.526E-04 m, Y3= -1.535E-03 m
X4= 9.052E-04 m, Y4= 0 m
Stress in Element 12 = - 8.001E+06 Pa
Stress in Element 23 = 1.414E+07 Pa
Stress in Element 13 = 2.263E+07 Pa
Stress in Element 24 = -3.200E+07 Pa
Stress in Element 34 = 2.263E+07 Pa

You have either reached a page that is unavailable for viewing or reached your viewing limit for this book.

Ch. 9 Exercise 2:

2. Material: Steel
   Diameter 2.0 in.



Reaction at the roller:
R = 612.5 lbs

INDEX

A
Analysis Procedure, Intro-6
Animate, 4-27
Apply Materials, 4-19
ATAN2, Excel, 3-3
Automatic Transition, 11-29
Axisymmetric, 9-2

B
Beam Bending, 8-3
Beam elements, 7-18
Bending Stress, 8-4
Buttons, Mouse, Intro-18

C
CAD Model, 4-15
Cancel, [Esc], Intro-18
Centroid, 4-2
Circle, 1-33
Coarse Mesh, 10-21
Command manager, 4-3
Constraints, Fixture, 4-20
Considerations, Modeling, Intro-3
Convergence study, 10-31
Coordinates transformation, 2-5
Create Mesh, 4-24
Create Shell Mesh, 12-16
Create Solid Mesh, 10-20
Curved beam, 11-2
Customize Heads-up View Tools, 1-27
Cut, Extrude, 1-34

D
Datum Planes, 4-8
Datum Points, 7-16
Degrees of freedom, 6-3
Dimension, Smart, 1-15
Displacement Constraint, 4-20
Displacement Results, 4-28
Display Style, 1-27
Direct Stiffness Method, 2-2
Direction Cosines, 2-8
Direction Reference, 4-23
Display Modes, 1-27
Display Options, 1-27
Distributed Load, 8-24
Dynamic, Modal, 13-2
Dynamic Orientation, 1-26
Dynamic Pan, 1-17
Dynamic Rotation, 1-20
Dynamic Viewing, 1-23
Dynamic Zoom, 1-17

E
Edit Definition, 4-18
ELEMENT, Intro-4
Element types, Intro-4
Excel, MS, 3-2
EXIT, Intro-19
External Loads, 4-23
Extrude, 1-19
Extruded Cut, 1-34

F
Failure Criteria, 11-2
FEA Elements, Intro-4
File Folder, Intro-20
Finite elements, types of, Intro-4
Finite element analysis, Intro-2
Finite element procedure, 5-2
Fixed Geometry, 4-20
Fixture, Constraints, 4-20
Force/torque, 6-24

G
Geometric Considerations, 10-5
Geometric Constraints, 1-14
Geometric Factor, 10-2
Global Space, 1-28
Global stiffness matrix, 1-7
Global stiffness matrix, 2D, 2-10
Graphics Cursors, 1-12
Graphics Window, Intro-13

H

H-method, 10-20
Heads-up View Toolbar, 1-25
Heads-up View Tools, Customize, 1-27
Help, On-Line, Intro-19
Hide/Show Items, 1-31
Hidden-lines Removed, 1-27
History, FEA development, Intro-2

I

Insert Columns, 3-16
Insert Function, Excel, 3-10
IPS (inch, pound, second), 4-5
Isometric View, 1-20

L

Line, 1-30
Linear Statics Analysis, 1-2
List Beam Forces, 5-26
Local coordinate system, 1-28
Longitudinal stress, 12-5

M

Material Properties, 4-19
Matrix
  Addition, Intro-7
  Column Matrix, Intro-7
  Definitions, Intro-6
  Diagonal Matrix, Intro-7
  Identity Matrix, Intro-8
  Inverse of a Square Matrix, Intro-8
  Multiplication by a constant, Intro-7
  Multiplication of Two Matrices, Intro-8
  Row Matrix, Intro-7
  Square Matrix, Intro-7
  Transpose of a Matrix, Intro-8
Merge Result, 1-32, 4-13
Mesh Control, 11-26
Mesh, Shell, 12-22
MINVERSE, 3-12
MMULT, 3-14
Modal analysis, 13-2
Modal Analysis program, 13-6
Mode Shapes, 13-20
Modeling considerations, Intro-3
Modify Dimensions, 1-18
Modulus of Elasticity, 1-2
Moment in Dir 2, 7-32
Moment Diagram, 7-5, 8-27
Mouse Buttons, Intro-18

N

Natural frequency, 13-2
Neutral axis, 7-3
New Part, 4-4
New Study, 4-16
No Hidden-Edge Display, 1-27
NODE, Intro-4
Normal To, 1-26

O

Orient, Beam Section, 7-15
Orthographic, 1-27

P

P-method, 10-21
Pan, 1-17
Perspective, 1-27
Plane Strain, 10-2
Plane Stress, 10-2
Plane Stress element, 10-2
Pressure Load, 12-15
Principle of Superposition, 9-3
Probe, 6-24
Procedure, Analysis, Intro-6

Q

Quick keys, 1-23

R

Redundant, 9-2
Reference Geometry, 4-8
Refinement, 10-25
Relative Motion Analysis, 1-4
Results, 4-26
Resonance, 13-2
Revolved Boss/Base, 12-8
Rotate, 1-20
Rough Sketches, 1-12
Run, 4-25

S

Screen Layout, Intro-13
Shaded Solid, 1-27
Shear & Moment, 7-5
Shear Diagram, 7-5, 8-26
Shear Force in Dir 1, 7-32
Shell, 12-22
Shell Mesh, 12-22
Sketch, 1-33
Sketch plane, 1-28
Smart Dimension, 1-15
Solid Modeling, 1-10
SolidWorks Simulation, 4-16
Solver, 4-25
Spring Constant, 1-2
Starting SolidWorks, Intro-12
Statically Indeterminate system, 9-2
Stiffness, 1-2
Stress Concentration Factor, 10-2
Stress-Strain Diagram, 1-2
Structural Member, 5-6
Study, Simulation, 4-16
Superposition, Principle, 9-3
Surface Mesh, 10-22
Sweep, 11-8
  Sweep Path, 11-8
  Sweep Section, 11-10
Symmetrical features, 12-2

T

Tangential stress, 12-5
Thickness, Shell, 12-22
Thin-Shell, 3D, 12-3
Treat selected bodies as beams, 4-17
Tresca Yield Criterion, 11-2
Through All, 1-34
Truss Element, 1-D, 1-2
Truss Element, 2-D, 2-2
Truss Solver Program, 3-23
Truss Viewer Program, 3-23

U

Units Setup, Intro-11
User Coordinate system, 1-28

V

Viewing functions, 1-20
View Sketch Relations, 1-31
View Orientation, 1-26
Von Mises Stress, 11-2
Von Mises Yield Criterion, 11-2

W

Weldments, 5-8
Weldment Profile, 5-9
Wireframe model, Intro-2
World Coordinate System, 1-28

Y

Young's Modulus, 1-2

Z

Zero force members, 4-29
ZOOM, dynamic, 1-17

Introduction to Finite Element Analysis Using SolidWorks® Simulation 2013

- Uses step-by-step tutorials to introduce users to SolidWorks Simulation 2013
- Incorporates theoretical aspects of Finite Element Analysis
- Covers all the most important Finite Element Analysis techniques and concepts

Description

The primary goal of Introduction to Finite Element Analysis Using SolidWorks Simulation 2013 is to introduce the aspects of Finite Element Analysis (FEA) that are important to engineers and designers. Theoretical aspects of FEA are also introduced as they are needed to help better understand the operation.

The primary emphasis of the text is placed on the practical concepts and procedures needed to use SolidWorks Simulation in performing Linear Static Stress Analysis and basic Model Analysis. This text covers SolidWorks Simulation and the lessons proceed in a pedagogical fashion to guide you from constructing basic truss elements to generating three-dimensional solid elements from solid models. This text takes a hands-on, exercise-intensive approach to all the important FEA techniques and concepts.

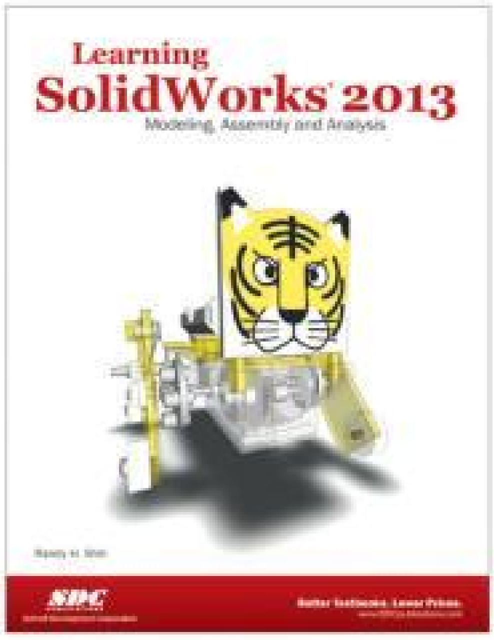
This textbook contains a series of thirteen tutorial style lessons designed to introduce beginning FEA users to SolidWorks Simulation. The basic premise of this book is that the more designs you create using SolidWorks Simulation, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons.

Table of Contents

1. The Direct Stiffness Method
2. Truss Elements in Two-Dimensional Spaces
3. 2D Trusses in MS Excel and the Truss Solver
4. Truss Elements in SolidWorks Simulation
5. SolidWorks Simulation Two-Dimensional Truss Analysis
6. Three-Dimensional Truss Analysis
7. Basic Beam Analysis
8. Beam Analysis Tools
9. Statically Indeterminate Structures
10. Two-Dimensional Surface Analysis
11. Three-Dimensional Solid Elements
12. Three-Dimensional Thin Shell Analysis
13. Dynamic Model Analysis

Index

Also Available



ISBN: 978-1-58503-772-8
9 781585 037728

SDC PUBLICATIONS

Better Textbooks. Lower Prices.
www.SDCpublications.com