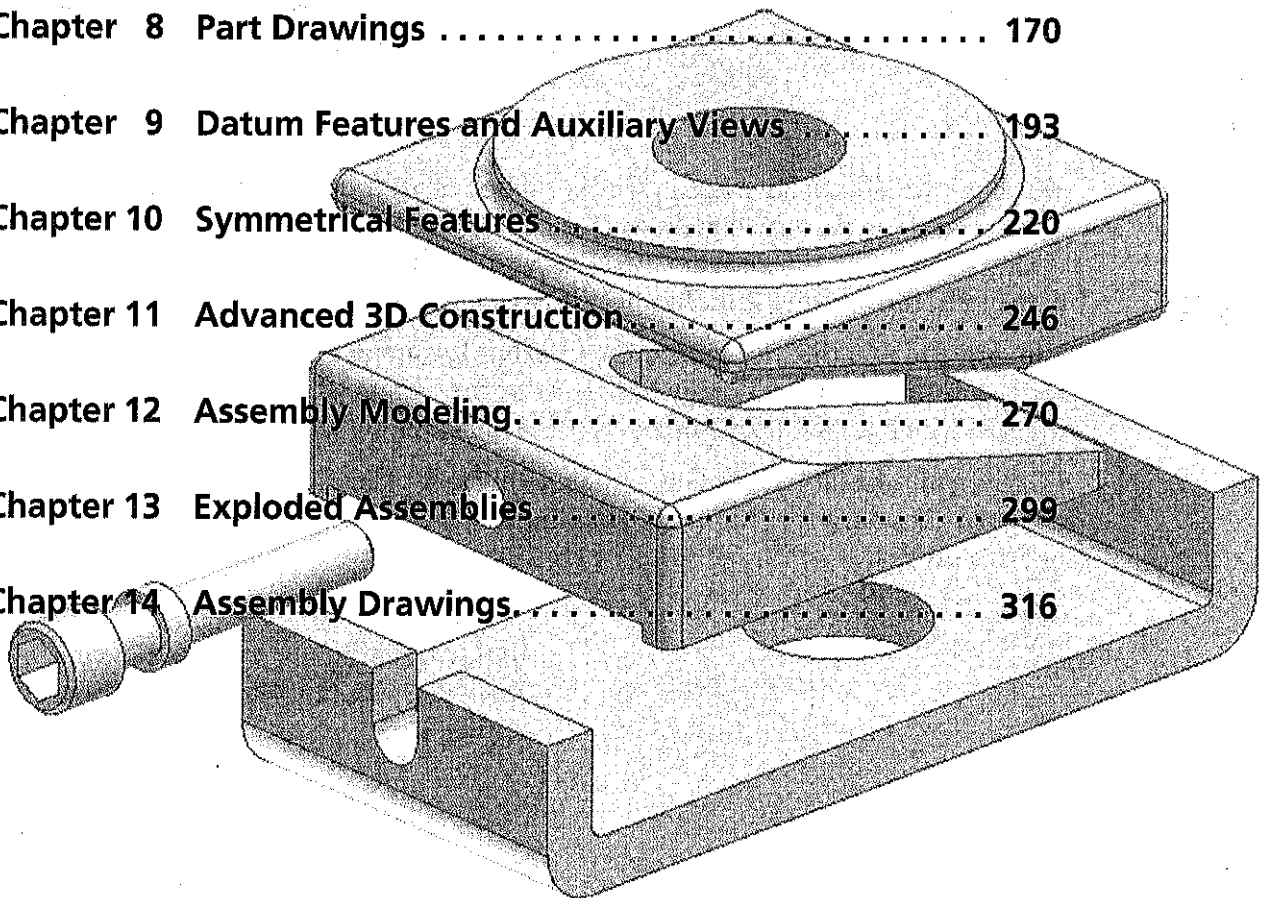
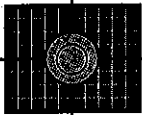


# Contents in Brief

Chapter 1	Getting Started .....	15
Chapter 2	Parametric Modeling.....	30
Chapter 3	Constructive Solid Geometry .....	58
Chapter 4	Model History Tree .....	77
Chapter 5	Parametric Constraints .....	96
Chapter 6	Geometric Construction .....	121
Chapter 7	Parent/Child Relationships .....	146
Chapter 8	Part Drawings .....	170
Chapter 9	Datum Features and Auxiliary Views .....	193
Chapter 10	Symmetrical Features .....	220
Chapter 11	Advanced 3D Construction.....	246
Chapter 12	Assembly Modeling.....	270
Chapter 13	Exploded Assemblies .....	299
Chapter 14	Assembly Drawings.....	316





# Acknowledgments

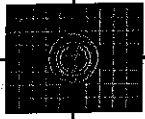


This book would not have been possible without a great deal of support. First, special thanks to two great teachers, Prof. George R. Schade of University of Nebraska-Lincoln and Mr. Denwu Lee, who showed me the fundamentals, the intrigue, and the sheer fun of computer-aided engineering.

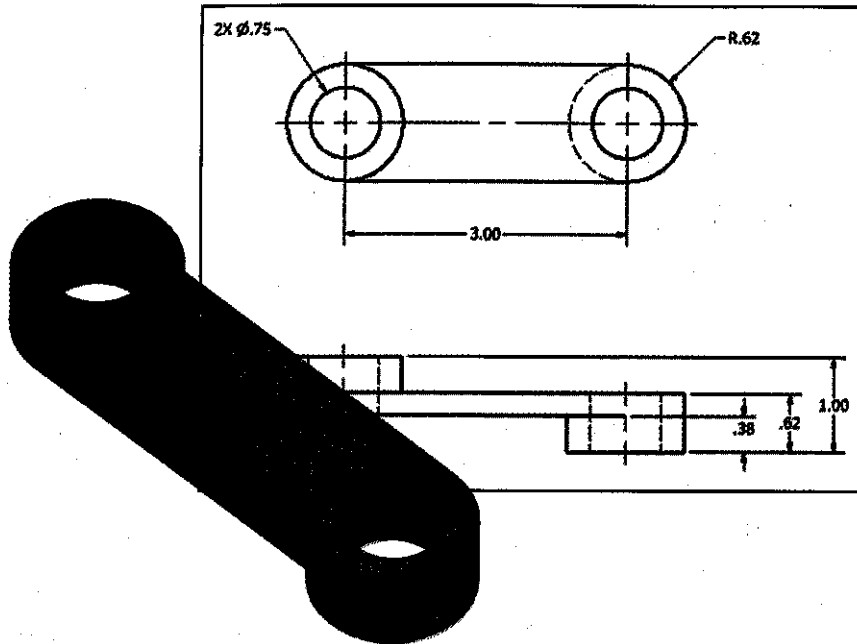
I am grateful that the Mechanical Engineering Technology Department of Oregon Institute of Technology has provided me with an excellent environment in which to pursue my interests in teaching and research.

Finally, truly unbounded thanks are due to my wife Hsiu-Ling and our daughter Casandra for their understanding and encouragement throughout this project.

Randy H. Shih  
Klamath Falls, Oregon  
Spring 2006



# Preface



The primary goal of *Designing with Inventor® 11* is to introduce the solid modeling and parametric modeling techniques used in the Inventor® 11 software. This textbook is intended to be used as a step-by-step training guide. The fourteen chapters proceed in a logical order to build your skills as you progress from constructing basic shapes to building intelligent solid models, creating multiview drawings, and building assembly models and drawings. Each chapter introduces a new set of commands and concepts, building on previous chapters.

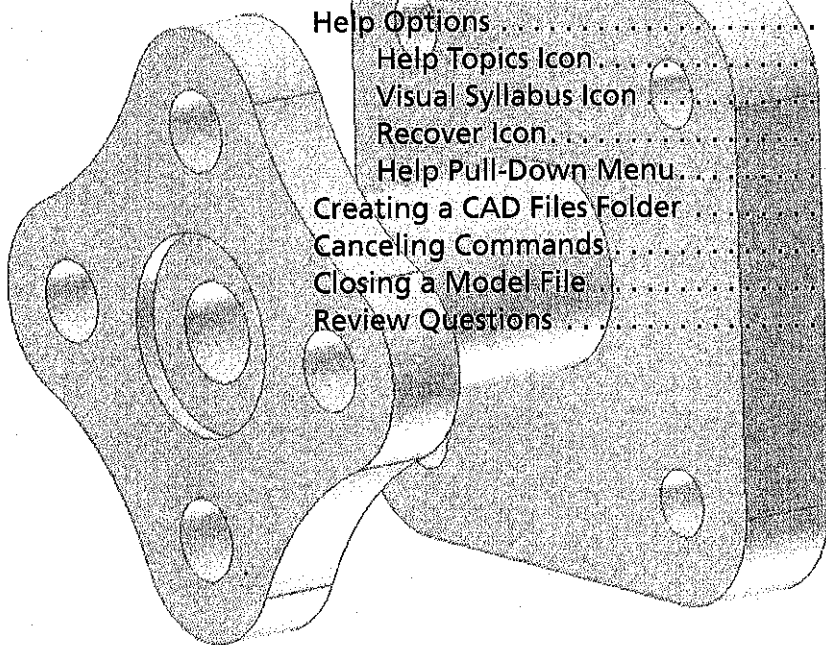
This hands-on, skills-based approach to parametric modeling techniques and concepts provides practical experience as you become familiar with Inventor. Many chapters encourage you to work out your own modeling strategies and attempt to build the models on your own before you work through the chapter material. Keep in mind that the more designs you create using Inventor, and the more experimentation you do, the more thoroughly you will learn the software.

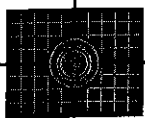
*Designing with Inventor® 11* does not attempt to cover all of the features in the Inventor software. Rather, it provides all of the basic skills and information you need to use the textbook as a springboard to promote your own skills and your career. It is intended to help you establish a good, solid basis for exploring and growing in the exciting field of computer-aided design and engineering.



# Contents

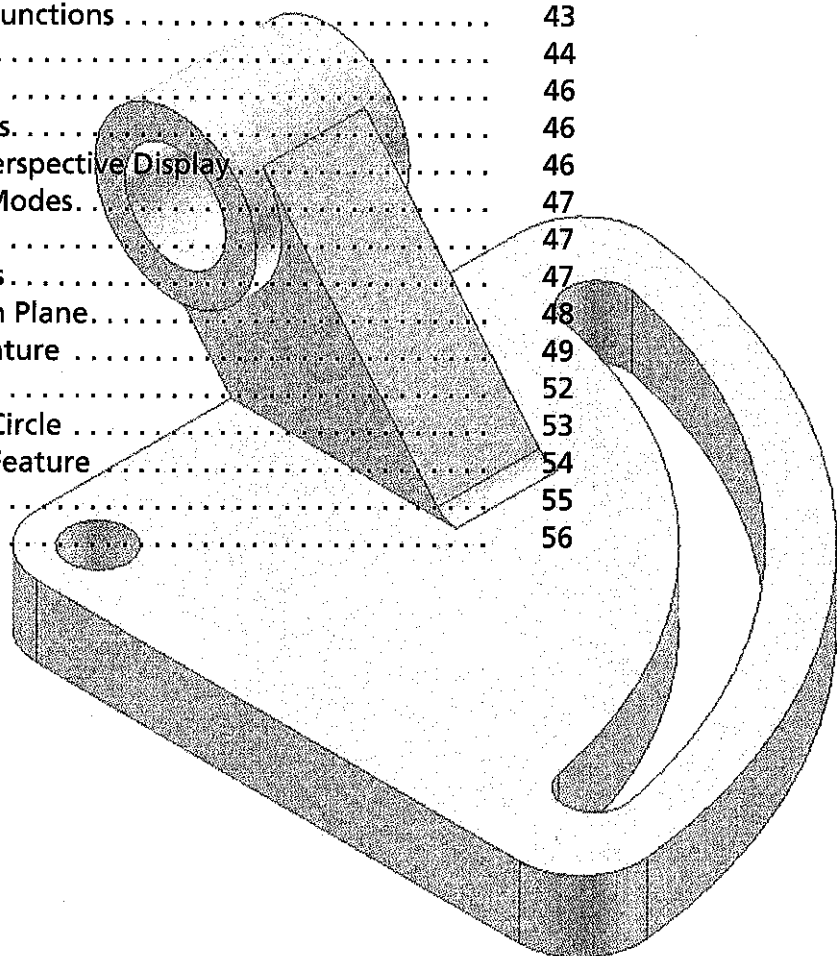
<b>Chapter 1: Getting Started</b> .....	<b>15</b>
Introduction .....	16
Computer-Based Modeling .....	16
Two-Dimensional Drawings .....	17
Three-Dimensional Wireframe Models .....	17
Surface Models .....	18
Solid Models .....	18
Parametric Modeling .....	18
Learning About Inventor .....	19
Starting Inventor .....	19
Starting a New Model File .....	20
Inventor Screen Layout .....	21
Pull-Down Menus .....	22
Inventor Standard Toolbar .....	22
2D Sketch Panel .....	22
Part Browser .....	23
Graphics Window .....	23
Crosshairs .....	24
Status Bar .....	24
Mouse Buttons .....	24
Left Mouse Button .....	24
Right Mouse Button .....	24
Middle Mouse Button or Wheel .....	24
Help Options .....	25
Help Topics Icon .....	25
Visual Syllabus Icon .....	26
Recover Icon .....	26
Help Pull-Down Menu .....	26
Creating a CAD Files Folder .....	27
Canceling Commands .....	28
Closing a Model File .....	28
Review Questions .....	29





# Contents

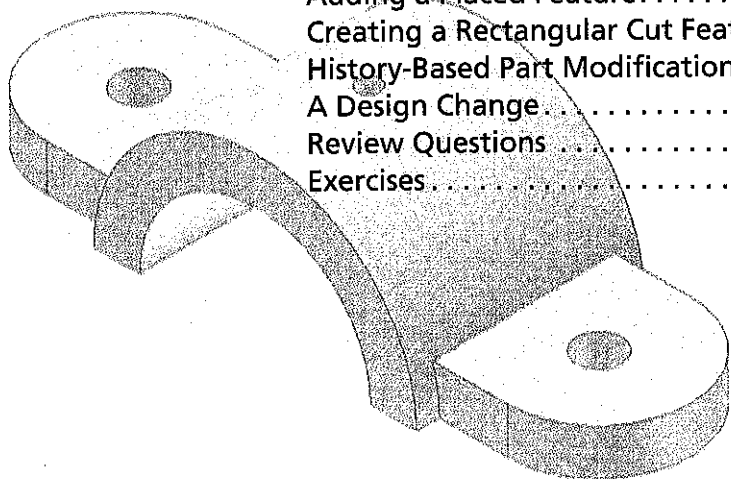
<b>Chapter 2: Parametric Modeling</b> .....	<b>30</b>
Parametric Part Modeling .....	31
Benefits of Parametric Modeling .....	31
The Parametric Part Modeling Process .....	31
Creating the Rough Sketch .....	32
Starting a New File .....	33
The Line Command .....	33
Geometric Constraint Symbols .....	35
Saving the File .....	36
Applying Constraints and Dimensions .....	37
Modifying Sketch Dimensions .....	40
Extruding the Base Solid Feature .....	40
Working with 3D Views .....	42
Isometric View .....	42
Dynamic Viewing Functions .....	43
Dynamic Rotation .....	44
Display Modes .....	46
Basic Display Modes .....	46
Orthographic vs. Perspective Display .....	46
Using the Display Modes .....	47
The Sketch Plane .....	47
Coordinate Systems .....	47
Aligning the Sketch Plane .....	48
Adding an Extruded Feature .....	49
Adding a Cut Feature .....	52
Dimensioning the Circle .....	53
Extruding the Cut Feature .....	54
Review Questions .....	55
Exercises .....	56

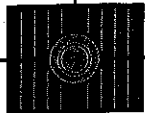




# Contents

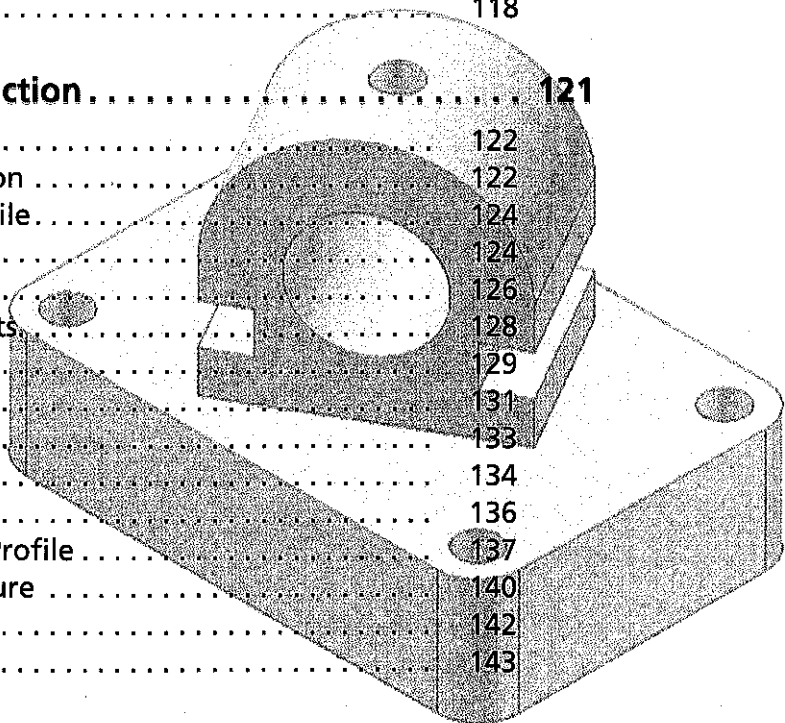
<b>Chapter 3: Constructive Solid Geometry</b> .....	<b>58</b>
Introduction to Constructive Solid Geometry .....	59
Binary Tree .....	60
Planning a Design with CSG .....	61
Setting Up the Locator File .....	63
Creating the Sketch .....	63
Rectangle Command .....	63
Dimension Format .....	64
Modifying the Sketch Dimensions .....	65
Repositioning Dimensions .....	65
Completing the Base Feature .....	66
Creating the Cylindrical Feature .....	67
Creating a Cut Feature .....	70
Creating a Placed Feature .....	72
Creating a Rectangular Cut Feature .....	73
Review Questions .....	75
Exercises .....	76
<b>Chapter 4: Model History Tree</b> .....	<b>77</b>
The History Tree .....	78
Planning Model Construction .....	79
The Part Browser .....	81
Creating the Base Feature .....	81
Adding the Second Solid Feature .....	83
Renaming the Part Features .....	85
Adjusting the Base Feature .....	85
Adding a Placed Feature .....	86
Creating a Rectangular Cut Feature .....	87
History-Based Part Modification .....	89
A Design Change .....	90
Review Questions .....	92
Exercises .....	93

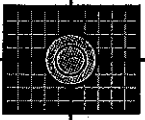




# Contents

<b>Chapter 5: Parametric Constraints</b> .....	<b>96</b>
Constraints and Relations .....	97
Automatic vs. Manual Constraints .....	97
Fully Constrained Geometry .....	97
Constraining a Model .....	98
Displaying Existing Constraints .....	98
Types of Geometric Constraints .....	99
Applying Geometric Constraints .....	101
Adding a Dimensional Constraint .....	103
Driven Dimensions .....	104
Deleting Existing Constraints .....	105
Using Auto Dimension .....	106
Checking Existing Constraints .....	106
Adding Constraints Manually .....	108
Constraint Settings .....	110
Parametric Relations .....	111
Dimensional Values and Dimensional Variables .....	112
Parametric Equations .....	114
Viewing Parameters and Relations .....	115
Review Questions .....	117
Exercises .....	118
<b>Chapter 6: Geometric Construction</b> .....	<b>121</b>
Introduction to Profiles .....	122
Planning Profile Creation .....	122
Setting Up the Model File .....	124
Sketching the Profile .....	124
Editing by Dragging .....	126
Adding Basic Constraints .....	128
Using Trim and Extend .....	129
Using Auto Dimension .....	131
Adding the Fillets .....	133
Checking the Constraints .....	134
Creating the Solid Model .....	136
Redefining the Sketch and Profile .....	137
Creating an Offset Cut Feature .....	140
Review Questions .....	142
Exercises .....	143





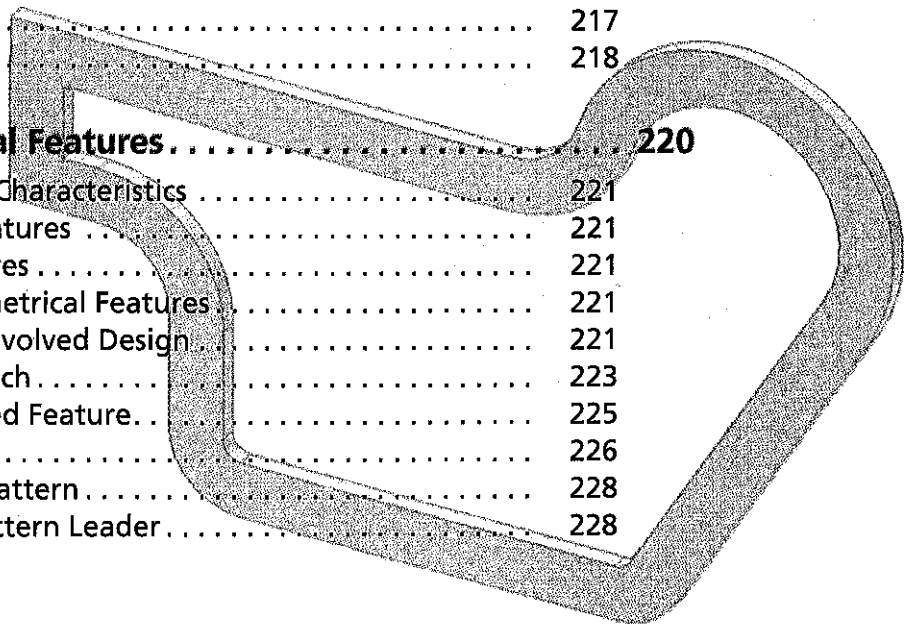
# Contents

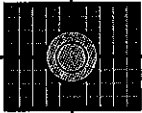
<b>Chapter 7: Parent/Child Relationships</b> .....	<b>146</b>
Introduction to Parent/Child Relationships .....	147
The BORN Technique .....	148
Setting the Default Sketch Plane .....	149
Applying the BORN Technique .....	150
Creating the Base Feature .....	151
The Implied Parent/Child Relationships .....	154
Creating the Second Solid Feature .....	155
Creating the First Cut Feature .....	157
Creating the Second Cut Feature .....	158
Examining the Parent/Child Relationships .....	160
Modifying a Parent Dimension .....	161
Feature Suppression .....	161
Enabling the Center_Drill Feature .....	163
Suppressing the Rect_Cut Feature .....	164
Creating a Circular Cut Feature .....	164
A Flexible Design Approach .....	165
Review Questions .....	166
Exercises .....	167
<b>Chapter 8: Part Drawings</b> .....	<b>170</b>
Part Drawings in Inventor .....	171
Creating a 2D Drawing .....	172
Working with Drawing Sheets .....	173
Using Predefined Drawing Sheets .....	175
Manipulating Drawing Sheets .....	176
Adding a Base View .....	177
Creating Projected Views .....	178
Adjusting the View Scale .....	178
Repositioning Views .....	179
Displaying Feature Dimensions .....	180
Repositioning and Hiding Dimensions .....	182
Adding Reference Dimensions .....	183
Adding Center Marks and Centerlines .....	184
Completing the Title Block .....	186
Modifying Feature Dimensions .....	187
Review Questions .....	190
Exercises .....	191



# Contents

<b>Chapter 9: Datum Features and Auxiliary Views</b> . . . . .	<b>193</b>
Working with Complex Models . . . . .	194
Datum Features . . . . .	194
Auxiliary Views . . . . .	194
The Rod Guide Design . . . . .	194
File Setup . . . . .	196
Creating the Base Feature . . . . .	196
Constructing the Second Feature . . . . .	198
Creating the Angled Work Plane . . . . .	198
Projecting the Geometry . . . . .	200
Creating the Sketch . . . . .	200
Completing the Solid Feature . . . . .	203
Constructing the Cut Feature . . . . .	204
Creating the 2D Drawing Layout . . . . .	206
Adding the Base View . . . . .	207
Adding the Auxiliary View . . . . .	208
Adding the Isometric View . . . . .	209
Displaying Feature Dimensions . . . . .	210
Adjusting the Views . . . . .	211
Adding Dimensions . . . . .	211
Adding Center Marks and Centerlines . . . . .	212
Shading the Isometric View . . . . .	215
Completing the Drawing Sheet . . . . .	215
Review Questions . . . . .	217
Exercises . . . . .	218
<b>Chapter 10: Symmetrical Features</b> . . . . .	<b>220</b>
Symmetrical Design Characteristics . . . . .	221
Symmetrical Features . . . . .	221
Revolved Features . . . . .	221
Modeling Symmetrical Features . . . . .	221
The Pulley: A Revolved Design . . . . .	221
Creating the 2D Sketch . . . . .	223
Creating the Revolved Feature . . . . .	225
Mirrored Features . . . . .	226
Creating a Circular Pattern . . . . .	228
Creating the Pattern Leader . . . . .	228



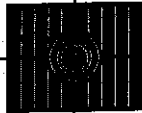


# Contents

Forming the Circular Pattern .....	231
Drawing Mode .....	232
Defining a New Border .....	232
Defining a New Title Block .....	234
Creating a Drawing Template .....	235
Creating the 2D Drawing .....	236
Setting Up the Drawing .....	236
Creating the Base View .....	236
Creating the Section and Isometric Views .....	238
Dimensioning the Section View .....	240
Making a Design Change .....	241
Adding Centerlines .....	242
Completing the Drawing .....	243
Review Questions .....	244
Exercises .....	245

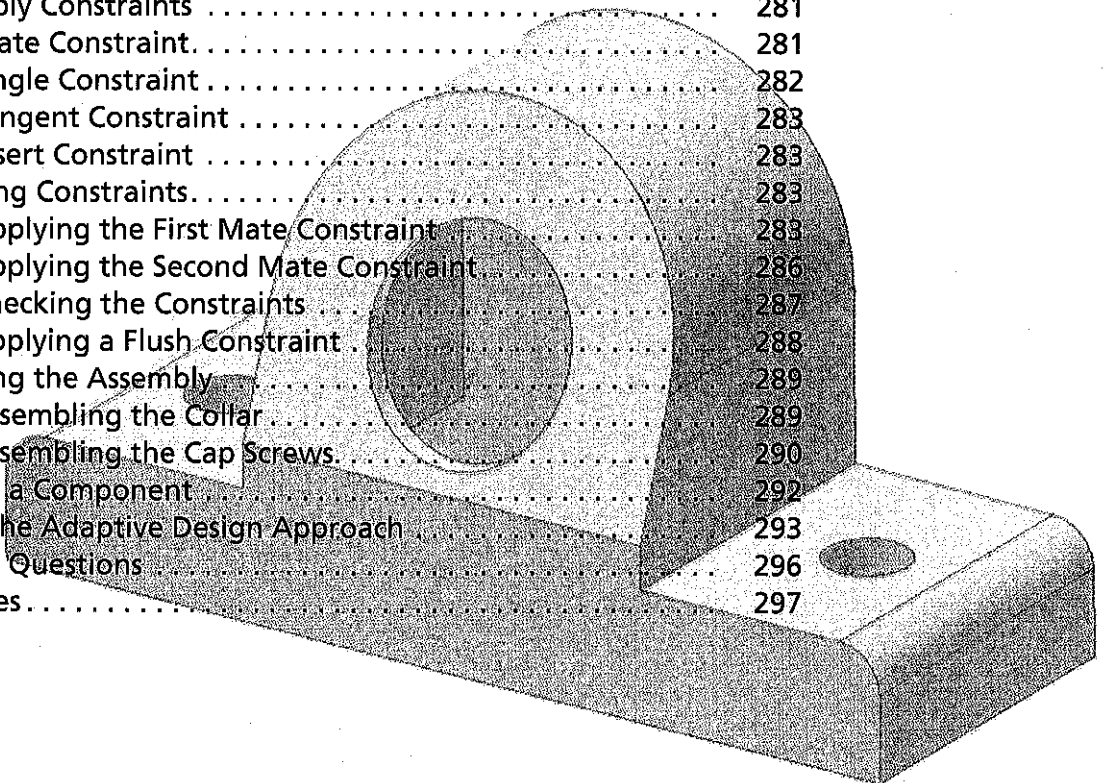
## **Chapter 11: Advanced 3D Construction .....** 246

A Thin-Walled Design: Dryer Housing .....	247
Creating the First 2D Sketch .....	248
Revolving the Base Feature .....	249
Lofting the Handle .....	250
Defining the Work Planes .....	250
Creating the Profile Sketches .....	251
Finishing the Lofted Feature .....	253
Building the Nozzle .....	254
3D Rounds and Fillets .....	256
Shelling the Dryer Housing .....	257
Creating the Grill Pattern .....	258
Creating the Pattern Leader .....	258
Creating the Rectangular Array .....	260
Creating a Swept Feature .....	262
Defining the Sweep Path .....	262
Defining the Sweep Section .....	263
Completing the Sweep .....	264
Review Questions .....	266
Exercises .....	267



# Contents

<b>Chapter 12: Assembly Modeling</b> .....	<b>270</b>
Introduction to Assembly Modeling .....	271
Bidirectional Associative Functionality .....	271
Direct Adaptive Assembly .....	271
Adaptive Design .....	271
Assembly Modeling Methods .....	272
The Bottom-Up Approach .....	272
The Top-Down Approach .....	272
The Middle-Out Approach .....	272
Creating the Part Files .....	272
The Collar Model .....	273
The Bearing Model .....	274
The Base Plate Model .....	275
The Cap Screw Model .....	276
Creating the Assembly Model .....	277
Placing the Base Component .....	277
Placing the Bearing .....	279
Degrees of Freedom and Constraints .....	280
Assembly Constraints .....	281
Mate Constraint .....	281
Angle Constraint .....	282
Tangent Constraint .....	283
Insert Constraint .....	283
Applying Constraints .....	283
Applying the First Mate Constraint .....	283
Applying the Second Mate Constraint .....	286
Checking the Constraints .....	287
Applying a Flush Constraint .....	288
Finishing the Assembly .....	289
Assembling the Collar .....	289
Assembling the Cap Screws .....	290
Editing a Component .....	292
Using the Adaptive Design Approach .....	293
Review Questions .....	296
Exercises .....	297





# Contents

## Chapter 13: Exploded Assemblies ..... 299

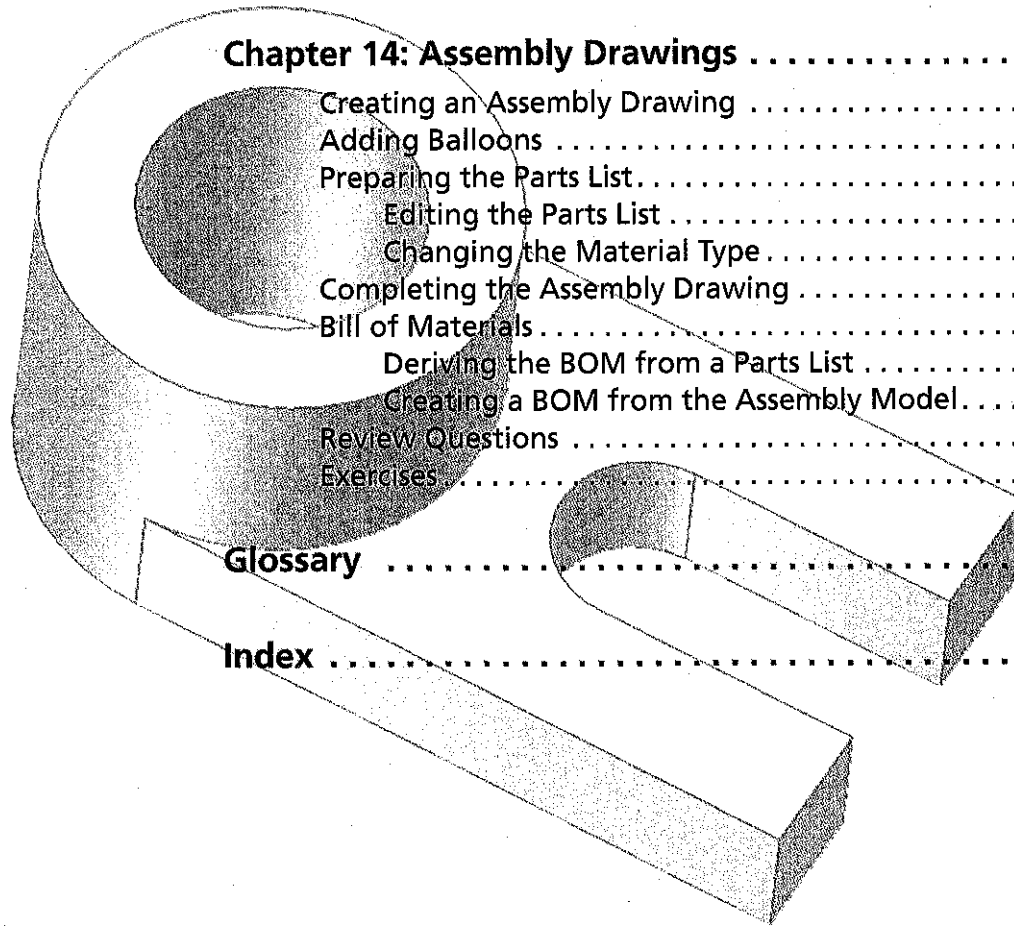
Exploding an Assembly .....	300
The Press Assembly .....	300
Creating the Base .....	300
Creating the Top Bar .....	301
Creating the Plate .....	302
Creating the Screw .....	303
Creating the Handle .....	304
Creating the Assembly Model .....	304
Using the Assembly Modeler .....	306
Using the Presentation Modeler .....	307
Creating the View Automatically .....	307
Creating the View Manually .....	308
Review Questions .....	311
Exercises .....	312

## Chapter 14: Assembly Drawings ..... 316

Creating an Assembly Drawing .....	317
Adding Balloons .....	318
Preparing the Parts List .....	320
Editing the Parts List .....	322
Changing the Material Type .....	325
Completing the Assembly Drawing .....	326
Bill of Materials .....	328
Deriving the BOM from a Parts List .....	328
Creating a BOM from the Assembly Model .....	328
Review Questions .....	330
Exercises .....	331

## Glossary ..... 335

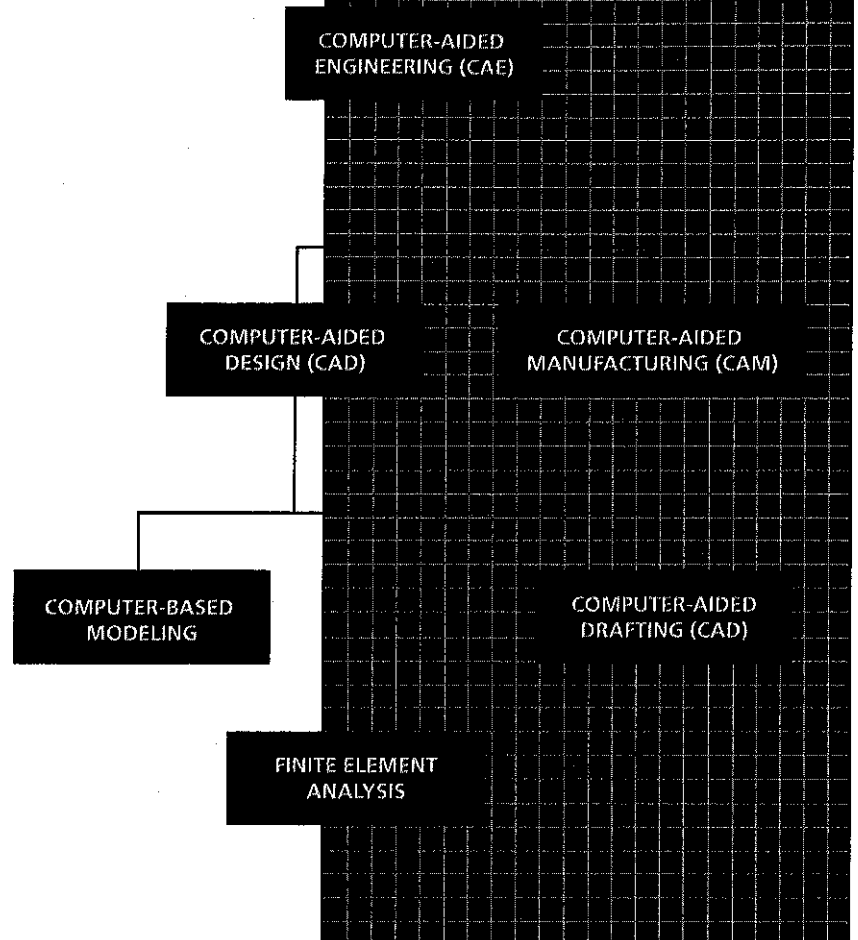
## Index ..... 354



# Getting Started

## Objectives

- Explore the development of computer-based geometric modeling.
- Describe the characteristics of feature-based parametric solid modeling.
- Create a model file using drawing units that are appropriate for the current project.
- Explore Inventor's screen layout.
- Identify the functions of the mouse buttons in Inventor.
- Use Inventor's help features.
- Create a folder to store your Inventor files.
- Save and close an Inventor model file.



## Vocabulary

computer-based modeling  
 computer-aided drafting  
 (CAD)  
 wireframe models  
 surface models  
 solid models

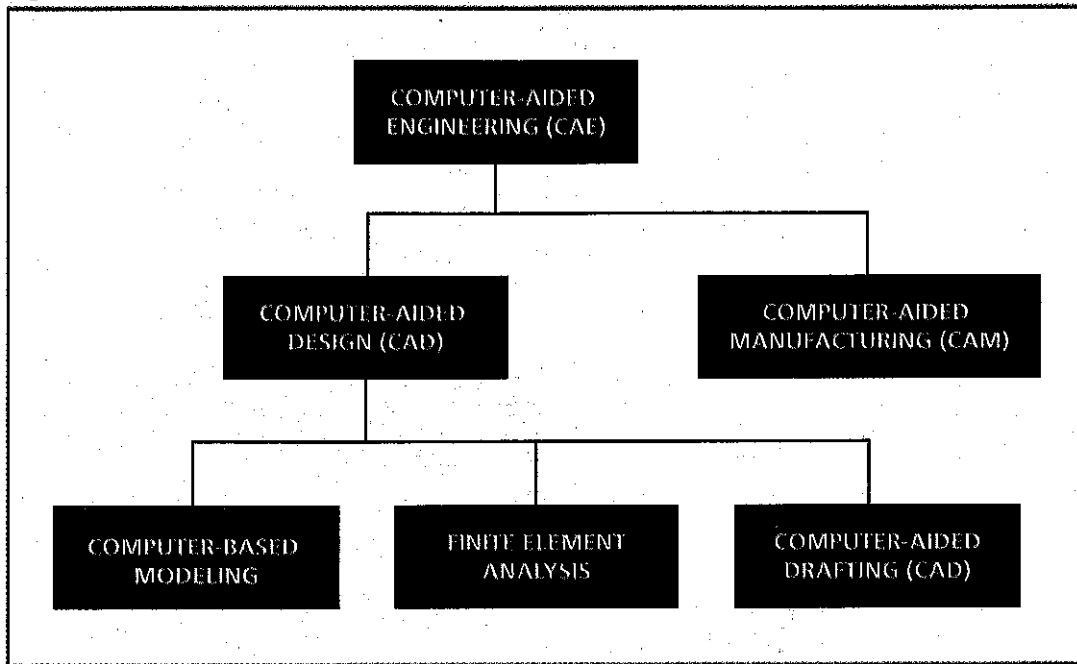
constructive solid  
 geometry (CSG)  
 primitive  
 boundary representation  
 concurrent engineering  
 parametric modeling

features  
 part  
 crosshairs  
 pan  
 zoom

## Introduction

In the computer-aided engineering (CAE) field, engineering design centers around one or more three-dimensional (3D) computer models. Rapid changes in the field of computer-aided engineering have brought exciting advances in the engineering community. CAE has become the core of concurrent engineering and is aimed at reducing design time, producing prototypes faster, and achieving higher product quality. The flowchart in Fig. 1-1 shows the relationship of various computer-based processes to CAE.

Fig. 1-1



Autodesk Inventor 11 is an integrated package of mechanical computer-aided engineering software tools developed by Autodesk, Inc. Inventor facilitates a concurrent engineering approach to the design and stress analysis of mechanical engineering products. Computer models created using Inventor can also be used by manufacturing equipment such as machining centers, lathes, mills, and additive fabrication systems to manufacture products. In this text, we will be dealing only with the solid modeling modules used for part design, part drawings, and assembly creation.

## Computer-Based Modeling

Creating geometrically accurate 3D models on a computer, known as **computer-based modeling**, is a relatively new technology. Its rapid expansion in the last 50 years is truly amazing. Computer modeling technology has advanced along with the development of computer hardware.

## Two-Dimensional Drawings

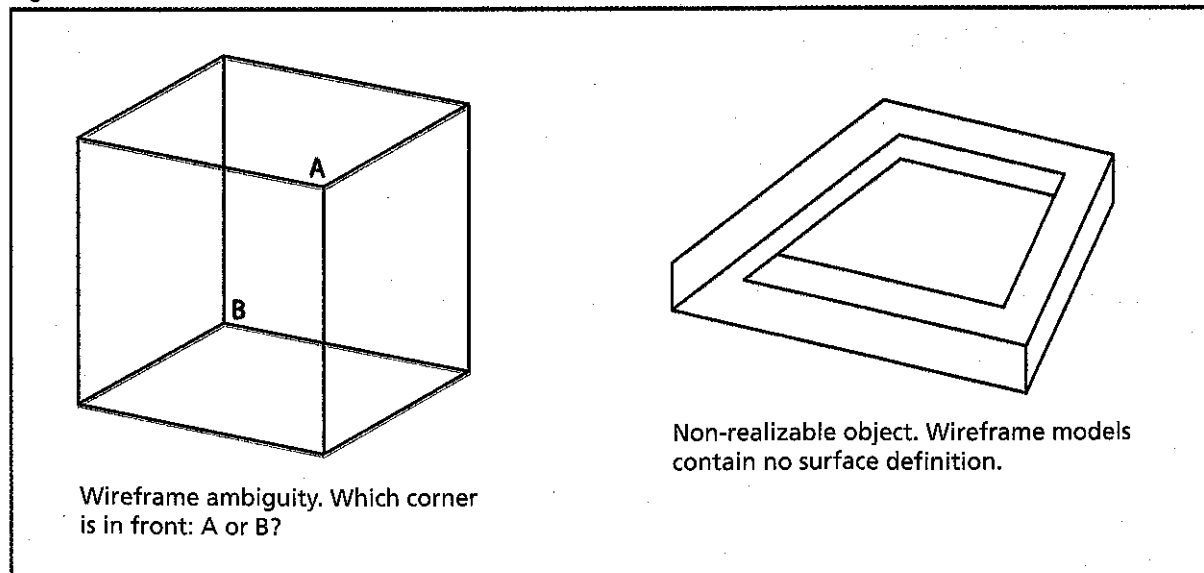
The first-generation **computer-aided drafting (CAD)** programs were simply 2D drafting programs, basically the electronic equivalent of the drafting board. For typical models, the use of this type of program would require that several to many views of the objects be created individually as they would be on the drafting board. The 3D designs remained in the designer's mind, not in the computer database. Mental translations of 3D objects to 2D views are required throughout the use of these packages. Although such systems have some advantages over traditional board drafting, they are still tedious and labor-intensive. The need for the development of 3D modelers came quite naturally, given the limitations of the 2D drafting packages.

## Three-Dimensional Wireframe Models

The development of three-dimensional modeling schemes started with 3D wireframes. **Wireframe models** are models consisting of points and edges, which are straight lines connecting appropriate points. The edges of wireframe models are used to represent transitions of surfaces and features, much as they are in 2D drawings. The use of lines and points is a very economical way to represent 3D designs.

The computer database in a 3D wireframe modeler contains the locations of all the points in 3D space coordinates, and it is typically sufficient to create just one model rather than multiple views of the same model. This single 3D model can then be viewed from any direction as needed. In comparison to other types of 3D modelers, the 3D wireframe modelers require very little computing power and generally can be used to achieve reasonably good representations of 3D models. However, because surface definition is not part of a wireframe model, all wireframe images have the inherent problem of ambiguity. Two examples of such ambiguity are illustrated in Fig. 1-2.

Fig. 1-2



---

## Surface Models

Surface modeling is the next logical development in computer geometry modeling. In **surface models**, edges that define polygonal surfaces are organized and grouped. Surface modeling describes the part's surfaces, but not its interior. Designers are still required to examine surface models interactively to ensure that the various surfaces on a model are contiguous throughout.

## Solid Models

Many of the concepts used in 3D wireframe and surface modelers are incorporated into solid modeling, but solid modeling offers more advantages as a design tool. In **solid models**, the model definition includes nodes, edges, and surfaces. A solid model is a complete and unambiguous mathematical representation of a precisely enclosed and filled volume. Unlike surface models, solid models start with a solid or use topology rules to guarantee that all of the surfaces are stitched together properly. Two predominant methods for representing solid models are constructive solid geometry representation and boundary representation.

**Constructive solid geometry (CSG)** representation can be defined as the combination of 3D solid primitives. A **primitive** is a basic solid from which other, more complex solids can be built. Predefined primitives vary somewhat with the software, but most programs include a minimum of a rectangular prism, a cylinder, a cone, a wedge, and a sphere. The underlying concept of the CSG representation method is very straightforward: we simply add or subtract one primitive from another.

In the **boundary representation** method, objects are represented in terms of their spatial boundaries. This method, commonly known as *B-rep*, defines the points, edges, and surfaces of a volume, and/or issues commands that sweep or rotate a defined face into a third dimension to form a solid. The object is then made up of the unions of these surfaces to enclose a volume completely and precisely.

## Parametric Modeling

By the 1980s, a new paradigm called **concurrent engineering** had emerged. In concurrent engineering, designers, design engineers, analysts, manufacturing engineers, and management engineers all work together closely right from the initial stages of the design. In this way, all aspects of the design can be evaluated and any potential problems can be identified throughout the design process.

Using the principles of concurrent engineering, a new type of feature-based computer modeling technique known as **parametric modeling** appeared. The basic principle of feature-based modeling is to build models by adding simple features one at a time. The word *parametric* means that the geometric definitions of the design, such as dimensions, can be changed at any time in the design process. **Features** are predefined pieces or construction tools for which users define the key parameters. A **part** is a sequence of engineering features that can be modified at any time. You will learn more about parametric modeling techniques in Chapter 2.

The key advantage of feature-based parametric modeling is its ability to produce very flexible designs. Changes can be made easily and design alternatives can be evaluated with minimum effort. The result is a flexible design defined by its design variables and parametric features. Parametric modeling is therefore one of the key elements of the Inventor software.

Another key feature of Inventor is the use of an assembly-based fit function that allows users to concentrate on the design without depending on the associated parameters or constraints. Users can specify how parts fit together, and Inventor automatically determines the sizes and positions of the parts. This unique approach, which we will call the *adaptive assembly* approach, defines part relationships with no order dependency. The adaptive assembly approach is a design methodology that can only be found in Inventor. The goal of this methodology is to improve the design process and to allow you, the designer, to design the way you think.

## Learning About Inventor

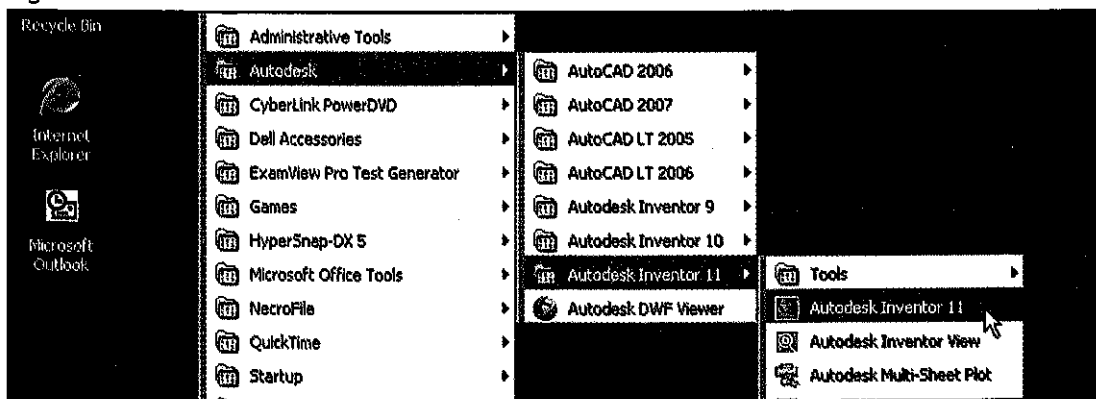
Inventor is composed of several application software modules called *applications* that share a common database. In this text, we will concentrate on the solid modeling modules used for part design and the general procedures required to create solid models, engineering drawings, and assemblies.

The tutorials in this text are based on the assumption that you are using Inventor's default settings. If your system has been customized for other uses, contact your technical support personnel to restore the default software configuration.

## Starting Inventor

How you start Inventor depends on the type of workstation and the particular software configuration you are using. With most Windows systems, you may select Autodesk Inventor 11 on the Start menu (Fig. 1-3) or select the Autodesk Inventor 11 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.

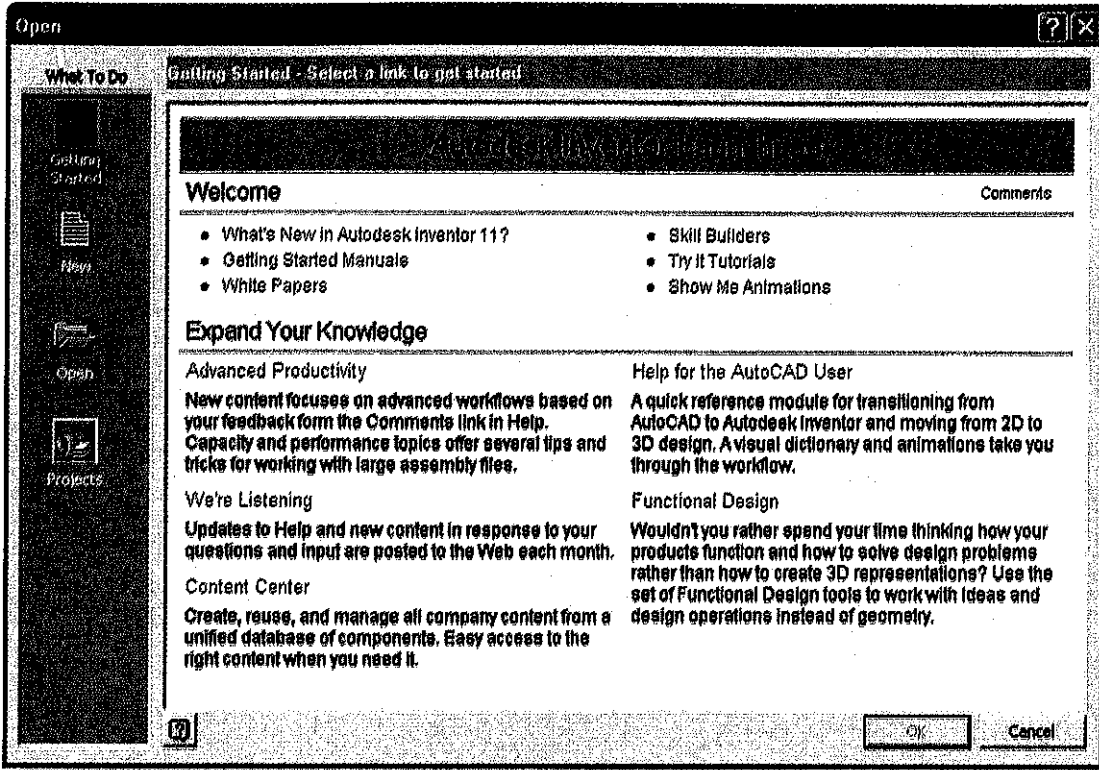
Fig. 1-3



## Starting a New Model File

When Inventor first loads, the Open dialog box appears. The four options shown in the What To Do area on the left side of the dialog box are Getting Started, New, Open, and Projects. The Getting Started option is active in Fig. 1-4.

Fig. 1-4



The Getting Started option provides some quick helps that illustrate the features and general procedures for using Inventor. The New option allows you to start a new modeling task. The Open option allows you to open an existing model file. Projects opens a list of defined projects from which you can select an existing project or create a new one. Projects are used only for designs that require multiple files.

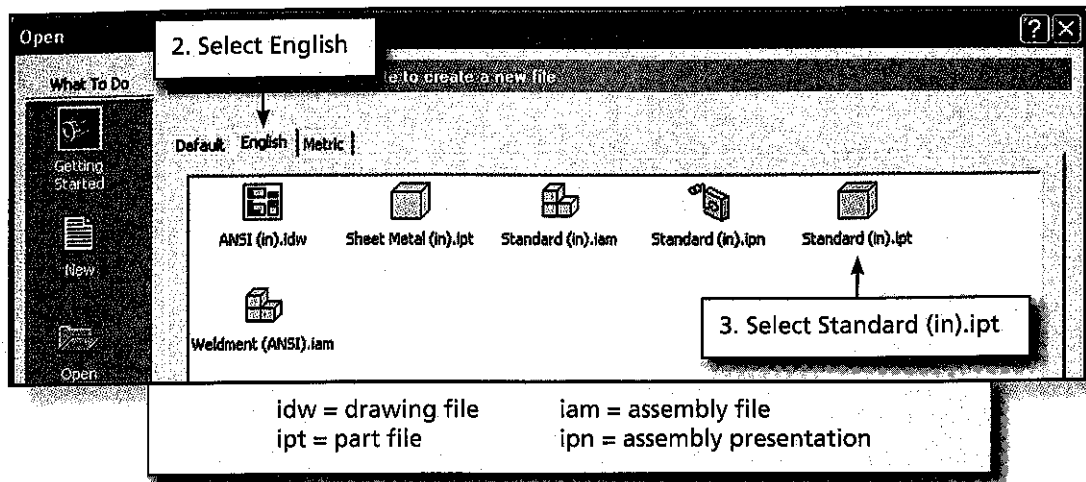
1. In the Open dialog box, select the New icon with a single click of the left mouse button.

The New File page appears, prompting you to select a template for the new file. Notice the three tabs at the top of the New File page: Default, English, and Metric. Each tab contains template files that are set up in the specified units. (The Default templates use the units that were chosen as default units when Inventor was first installed.) When starting a new file, the first thing you should do is choose the units you want to use. In this exercise, we will use English units (feet and inches).

2. Select the English tab. Fig. 1-5.

The icons that appear represent the templates for different modeling tasks. Notice that the names of the templates have different file extensions.

Fig. 1-5

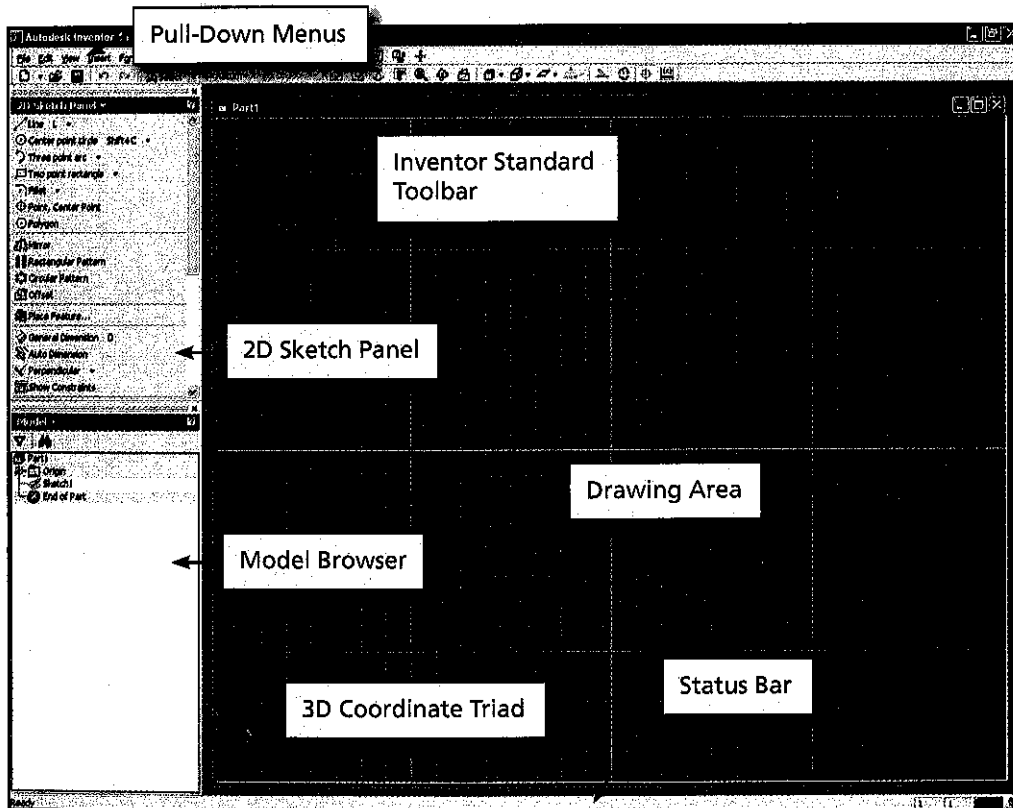


3. Select the Standard (in).ipt icon and pick OK to open the template. Fig. 1-5.

## Inventor Screen Layout

The default Inventor drawing screen contains the pull-down menus, the Inventor Standard toolbar, the Sketch Panel, the Model browser area, the drawing area, and the status bar. Fig. 1-6. A line of quick text appears next to each icon as you move the mouse cursor over it. Use this text to become familiar with what each icon does.

Fig. 1-6



## Pull-Down Menus

The pull-down menus at the top of the main window contain operations that you can use for all modes of the system. Fig. 1-7. When an operation is not available at a given time, it grays out so that it cannot be used.

## Inventor Standard Toolbar

The Inventor Standard toolbar at the top of the screen allows quick access to frequently used commands. Fig. 1-8. For example, the view-related commands, such as Zoom, Pan, and Rotate, are tools to help manipulate views of graphic objects.

Fig. 1-7

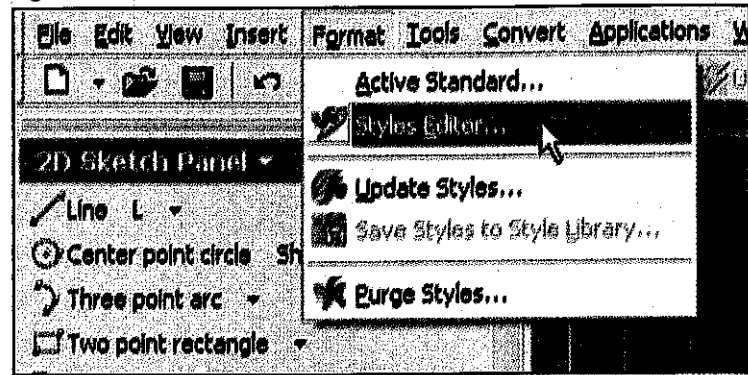


Fig. 1-8



## 2D Sketch Panel

Located near the top of the left side of the drawing area, the 2D Sketch Panel provides tools to create the basic geometry that is used to create features and parts. Fig. 1-9. These tools include commands to create points, lines, circles, arcs, rectangles, and other polygons. It also includes editing tools, such as mirroring, offsetting, trimming, and rotating, as well as dimensioning tools.

When the operations you are performing require a large amount of screen space in the drawing area, you can close the 2D Sketch Panel and open the 2D Sketch Panel toolbar instead. Fig. 1-10. Exactly the same tools are provided on the toolbar, but it takes up less space because it shows the tools as icons only. Fig. 1-10. To display the 2D Sketch Panel toolbar, right-click on any icon in the Inventor Standard toolbar and pick 2D Sketch Panel from the list.

Fig. 1-9

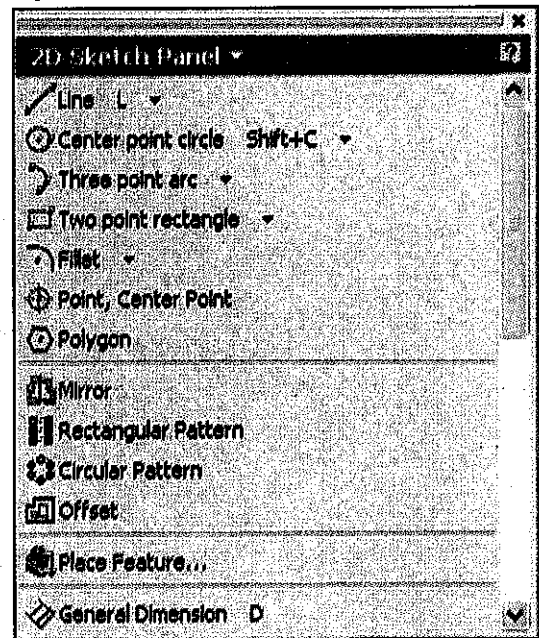


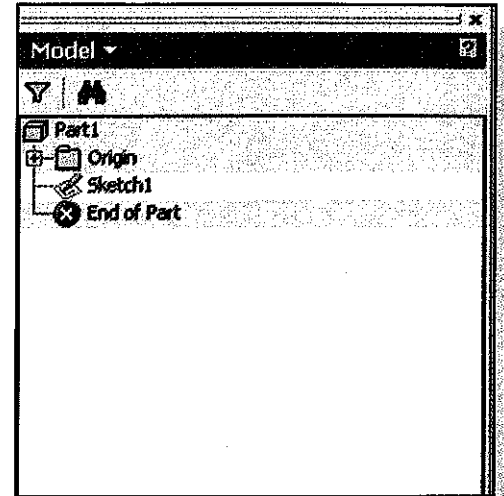
Fig. 1-10



## Part Browser

By default, the part browser is located just below the 2D Sketch Panel. When you first create a new part, it will be titled Model, as shown in Fig. 1-11. The part browser is a tree view that shows the operations and features that have been created in the current file. It allows you to show, hide, and work with selected features. To work with a specific sketch or feature, right-click it in the part browser to see a list of operations you can perform. The available operations depend on the type of feature, as well as other existing features and geometry. Like the 2D Sketch Panel, the part browser can be closed when you need more screen space to work on the current part or assembly.

Fig. 1-11



## Graphics Window

The graphics window is the area in which models and drawings are displayed. When you first open a new file, it contains a square reference grid and the 3D coordinate triad. Fig. 1-12. Neither of these elements is part of the drawing, and they do not print. The reference grid gives you an idea of the sizes of parts at the current magnification, and the 3D coordinate triad shows the current orientation of the object in 3D space.

Fig. 1-12



## Crosshairs

When a command is active, the mouse cursor (pointer) changes to a crosshairs. Fig. 1-13. The **crosshairs** is a more precise cursor that allows you to pick a point exactly at the intersection of the horizontal and vertical lines of the cursor.

## Status Bar

The status bar at the bottom of the screen shows a single-line help tip when you rest the cursor over any icon. This area also displays information pertinent to the active operation. For example, in Fig. 1-14, the coordinates and length information of a line are displayed while the Line command is active.

Fig. 1-13

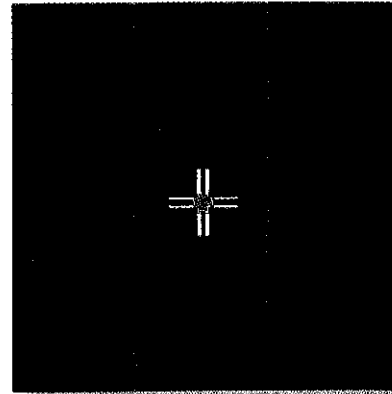


Fig. 1-14



## Mouse Buttons

While using Inventor, you will use the mouse buttons extensively. It is therefore important to understand the basic function of each of the mouse buttons. Fig. 1-15.

### 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, form entries, and graphic items on the screen.

### Right Mouse Button

The right mouse button is used to bring up additional options that are available at any given time. The software also allows you to use the right mouse button the same as you would the ENTER key. You can right-click to enter commands, accept default settings, or end a process.

### Middle Mouse Button or Wheel

The middle mouse button or wheel can be used to pan and zoom within the drawing. To **pan** (move to a different place at the same magnification), hold down the wheel button and drag the mouse. To **zoom** (change to a different drawing magnification), rotate the wheel.

Wheel allows quick panning and zooming.

Right button displays option menu; also used as alternative to the ENTER key.

Left button picks icons, menus, and entities.

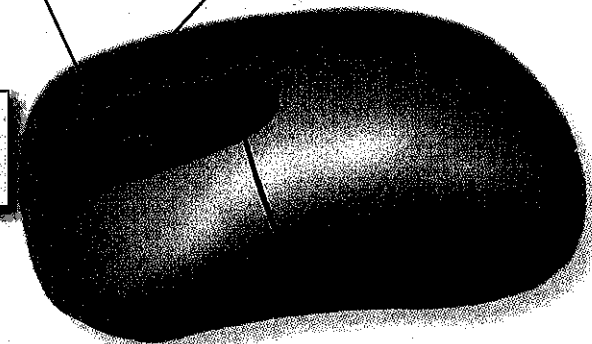
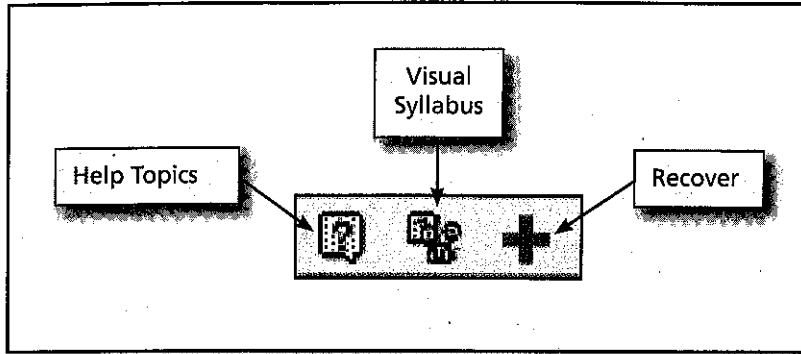


Fig. 1-15

# Help Options

The three icons to the right of the pull-down menus offer important help options. From left to right, they include the Help Topics icon, the Visual Syllabus icon, and the Recover icon. Fig. 1-16.

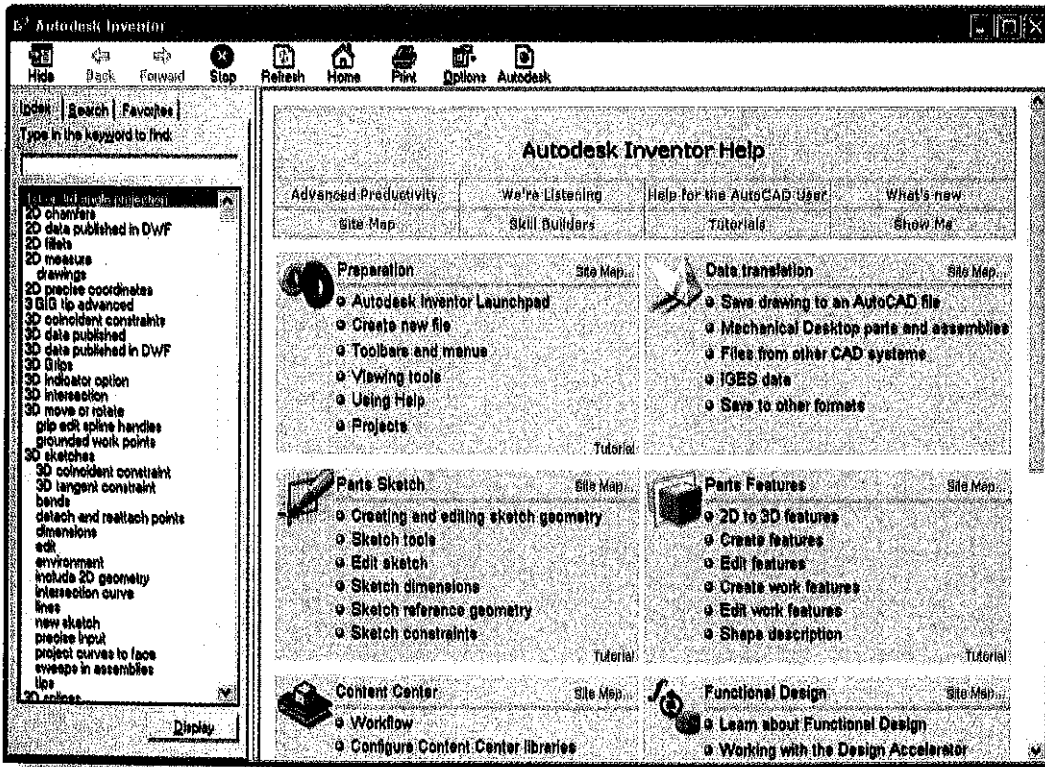
Fig. 1-16



## Help Topics Icon

Picking the Help Topics icon displays the Autodesk Inventor Help dialog box. Fig. 1-17. This dialog box provides general help information, such as command options and command references. You can also access this information by pressing the F1 function key.

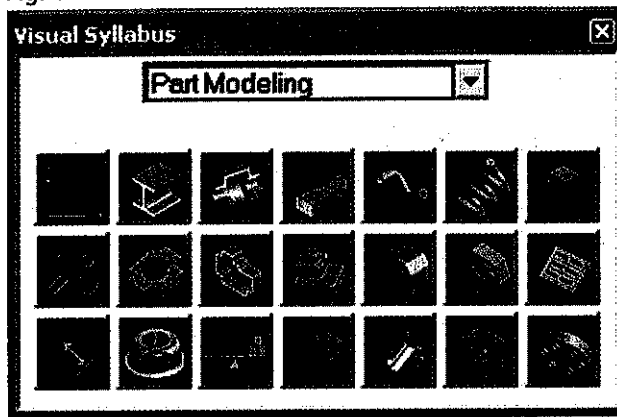
Fig. 1-17



## Visual Syllabus Icon

The Visual Syllabus is a dialog box that contains visual icons representing many different types of operations in Inventor. Fig. 1-18. Picking an icon provides a tutorial illustrating the associated operation. The icons are arranged on pages according to type of operation. Figure 1-18 shows the Part Modeling page. Other pages include Sheet Metal, Assembly Modeling, Presentations, and Drawings. To reach these pages, pick the down arrow to the right of Part Modeling.

Fig. 1-18



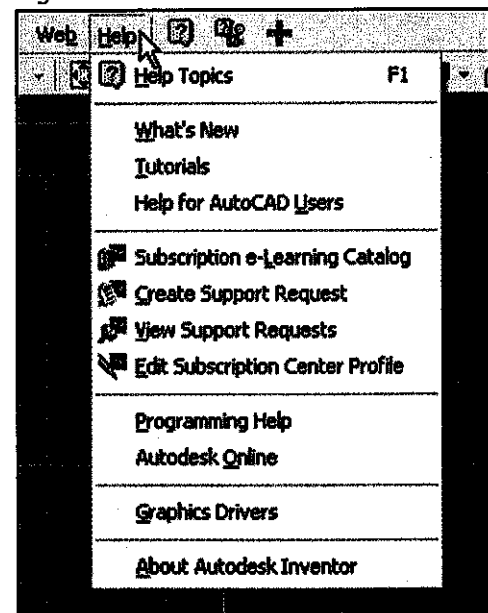
## Recover Icon

Inventor contains a feature called Design Doctor that automatically diagnoses problems in the current model and recommends solutions to them. Such problems might occur, for example, if you make changes to one feature of a model that are not consistent with other features. The Recover icon becomes active when Design Doctor discovers errors or problems in a model, allowing you to fix the problem immediately or wait until later. If you choose to wait, you can either pick the Recover icon or right-click on the affected feature and select Recover.

## Help Pull-Down Menu

In addition to the help buttons, Inventor provides a detailed Help pull-down menu. Fig. 1-19. This menu allows you to access the Autodesk Inventor Help dialog box, just as if you had clicked the Help Topics button. However, it also offers many other options. You can view tutorials similar to those represented in the Visual Syllabus, subscribe to an e-learning catalog, request support on-line, or choose from several other activities.

Fig. 1-19



## Creating a CAD Files Folder

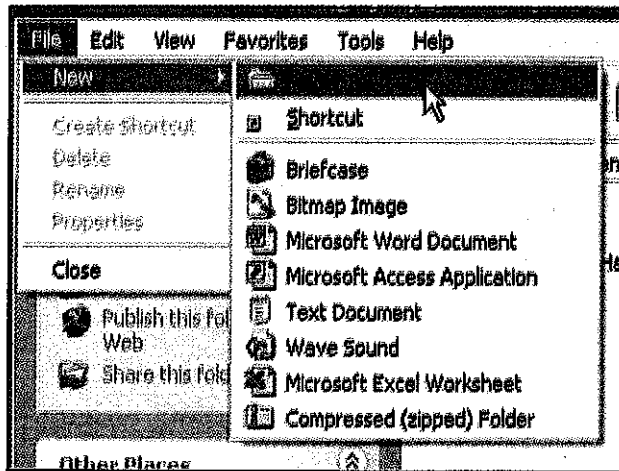
When you create and save files in Inventor, it is strongly recommended that you save them on the hard drive of your computer. It is a good practice to create a separate folder to store your CAD files. You should not save your files in the same folder in which the Inventor application files are located. 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.

For now, you only need to create one folder for your files. To create a new folder in the Windows environment, perform the following steps.

1. Minimize the Inventor program.
2. Open My Computer or My Documents and navigate to the folder in which you want to create a new folder for your CAD files.
3. On the File menu, pick New and then Folder. Fig. 1-20.

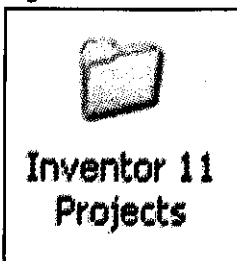
A new folder appears with the temporary name New Folder.

Fig. 1-20



4. Type a name for the new folder and press ENTER. Fig. 1-21.

Fig. 1-21



## Canceling Commands

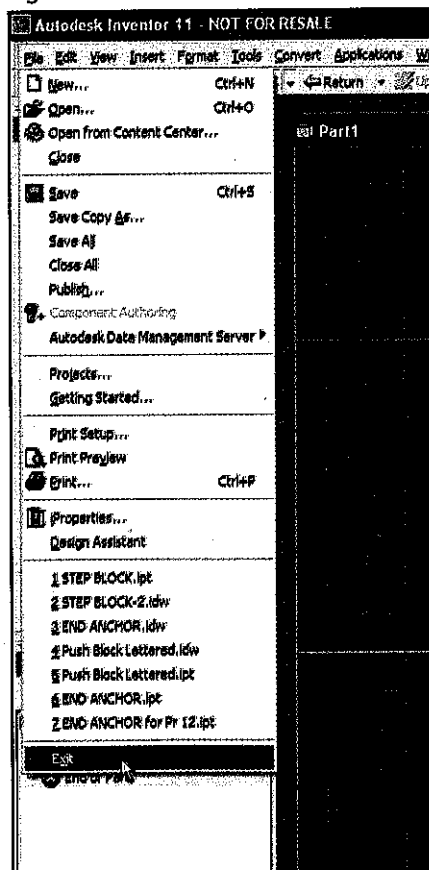
The ESCAPE key (ESC) is used to cancel a command in Inventor. The ESC key is located near the top left corner of the keyboard. It may sometimes be necessary to press the ESC key twice to cancel a command; it depends on where you are in the command sequence. For some commands, the ESC key is also used to exit the command.

## Closing a Model File

At the end of your Inventor session, you should save all open files before quitting the software. Follow the steps below.

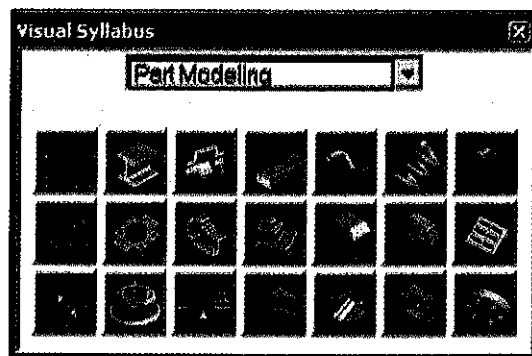
1. From the File pull-down menu, use the left mouse button to pick Save (for a single part or project) or Save All to save the current project. In this case, since you created a new model file in this chapter, pick Save and enter the name Model Example 1.
2. Move the cursor again to the File pull-down menu and choose Exit to close Inventor. Fig. 1-22.

Fig. 1-22



## Review Questions

1. Briefly describe the purposes for which Inventor models are commonly used.
2. What was the greatest limitation of the first-generation CAD programs?
3. What is the difference between a wireframe model and a surface model?
4. What is the difference between a surface model and a solid model?
5. What is parametric modeling?
6. What decision must you make before you create a new model file in Inventor? Explain.
7. What is the purpose of the Inventor Standard toolbar?
8. What can you display instead of the 2D Sketch Panel if you need more space in the graphics area?
9. To work with a specific feature in the part browser, how should you proceed?
10. What information does the status bar show?
11. Pressing the F1 function key on the keyboard brings up the Autodesk Inventor help dialog box. Name two other methods you can use to display this important dialog box.
12. How can you display the dialog box shown below?



13. How do you cancel a command in Inventor?
14. What should you always do before you close Inventor?