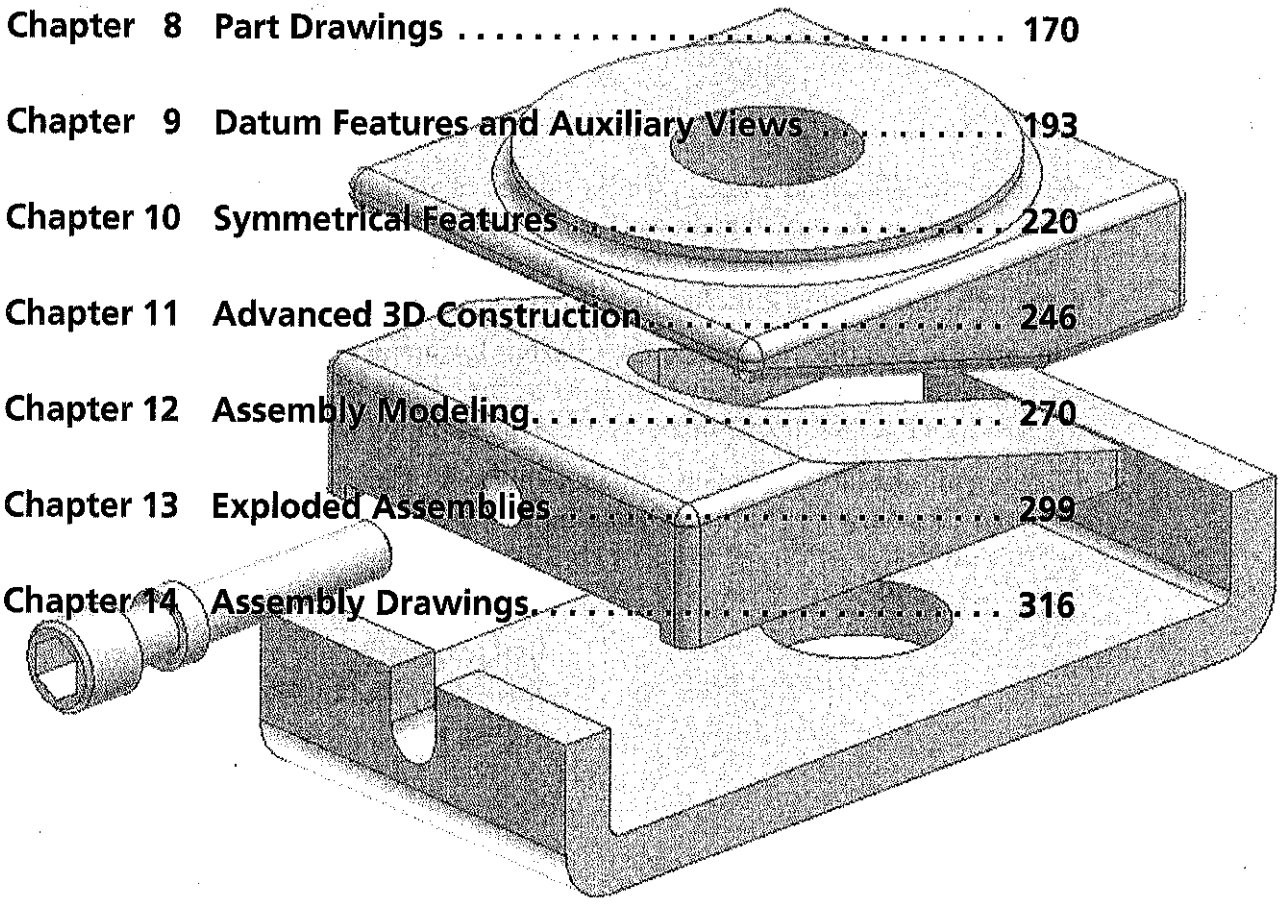
Contents in Brief



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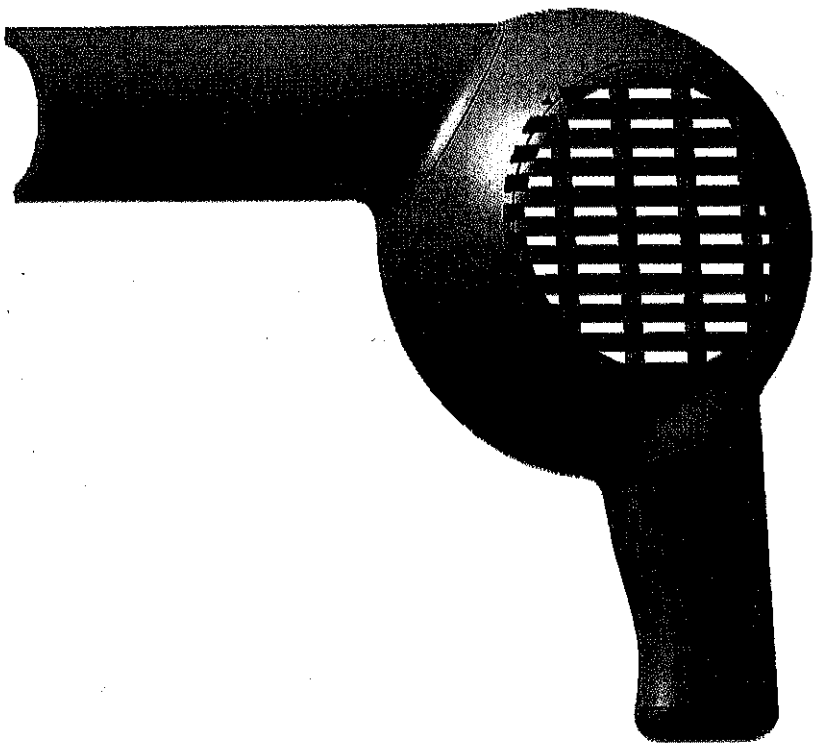


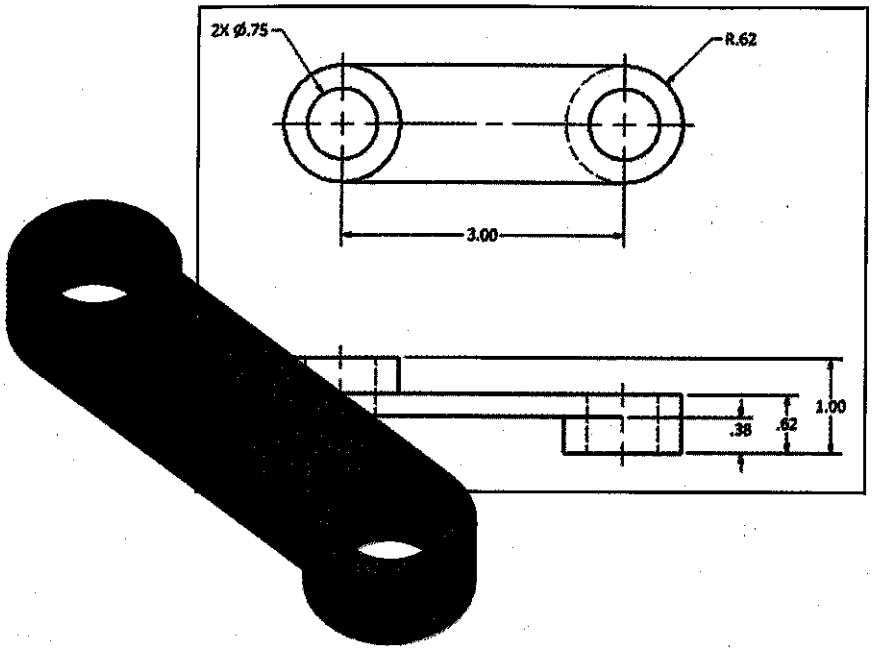
This book would not have been possible without a great deal of support. First, special thanks to two great teachers, Prof. George R. Schade of University of Nebraska-Lincoln and Mr. Denwu Lee, who showed me the fundamentals, the intrigue, and the sheer fun of computer-aided engineering.

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.

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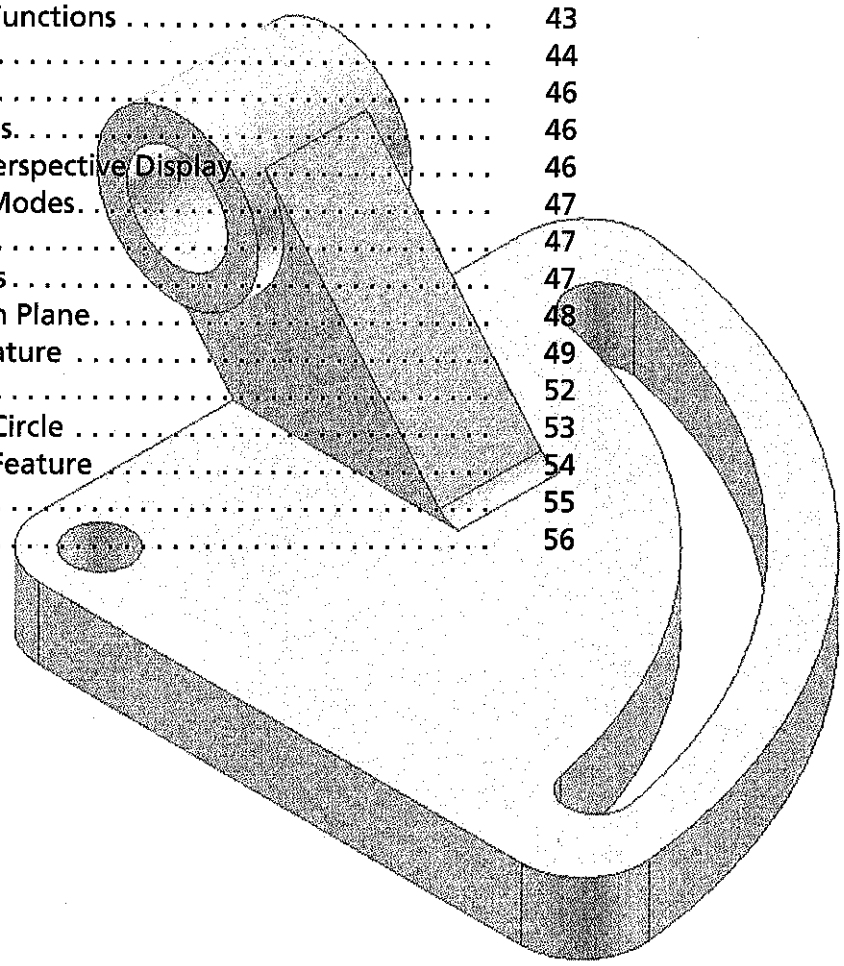

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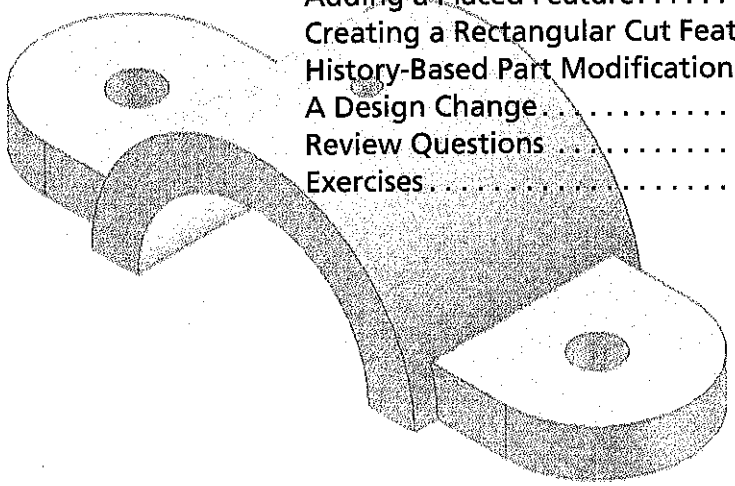

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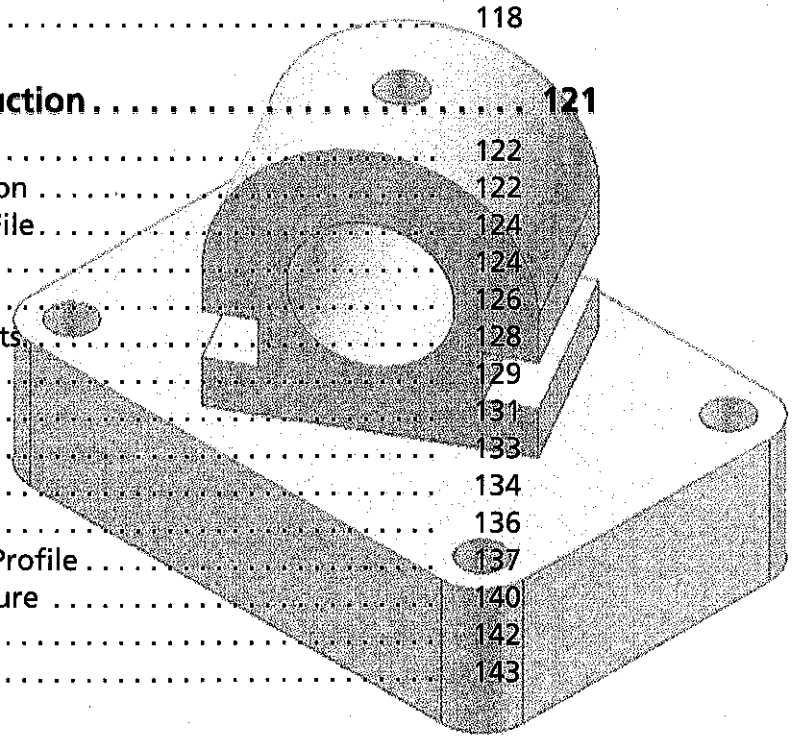

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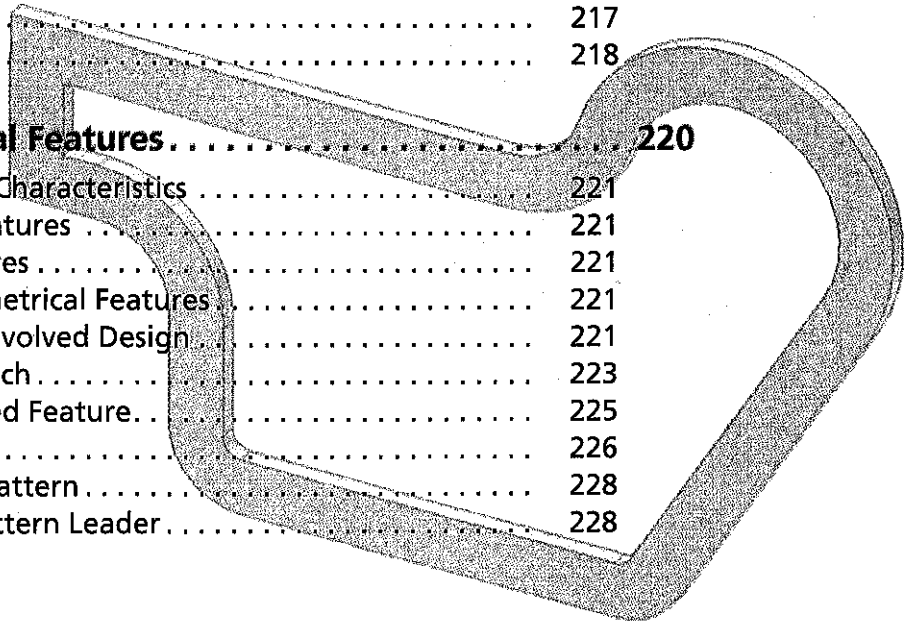

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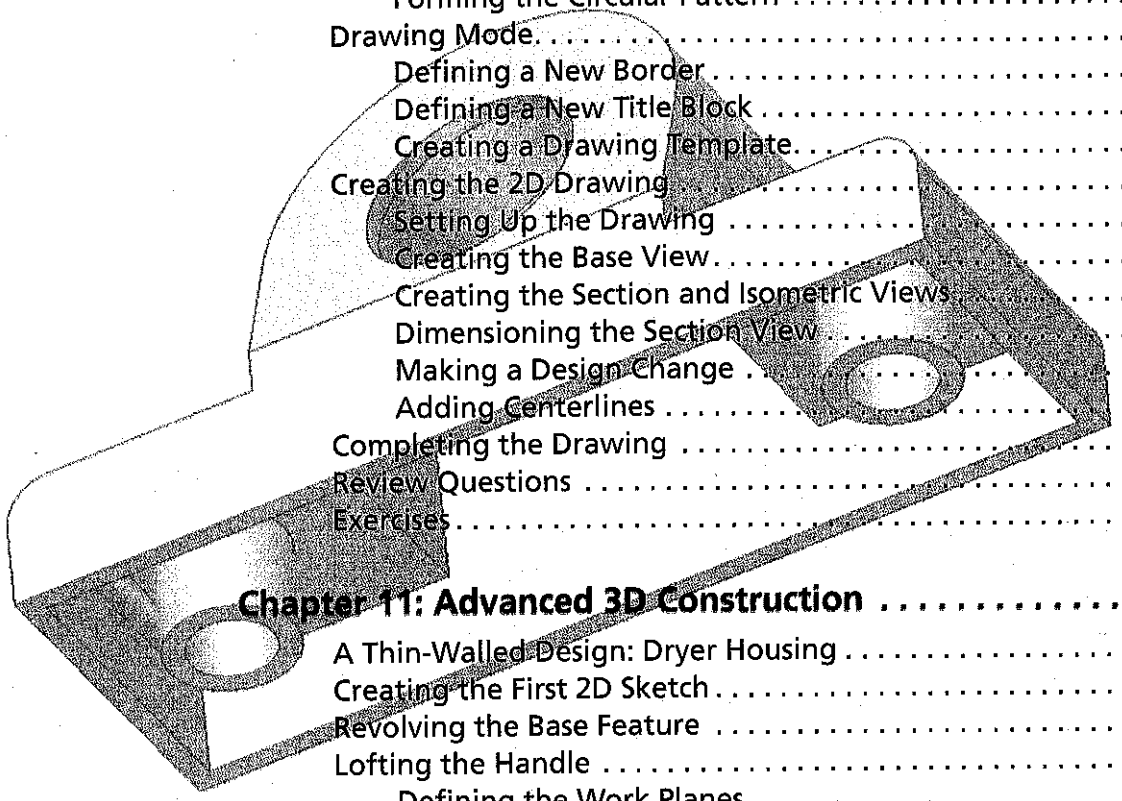

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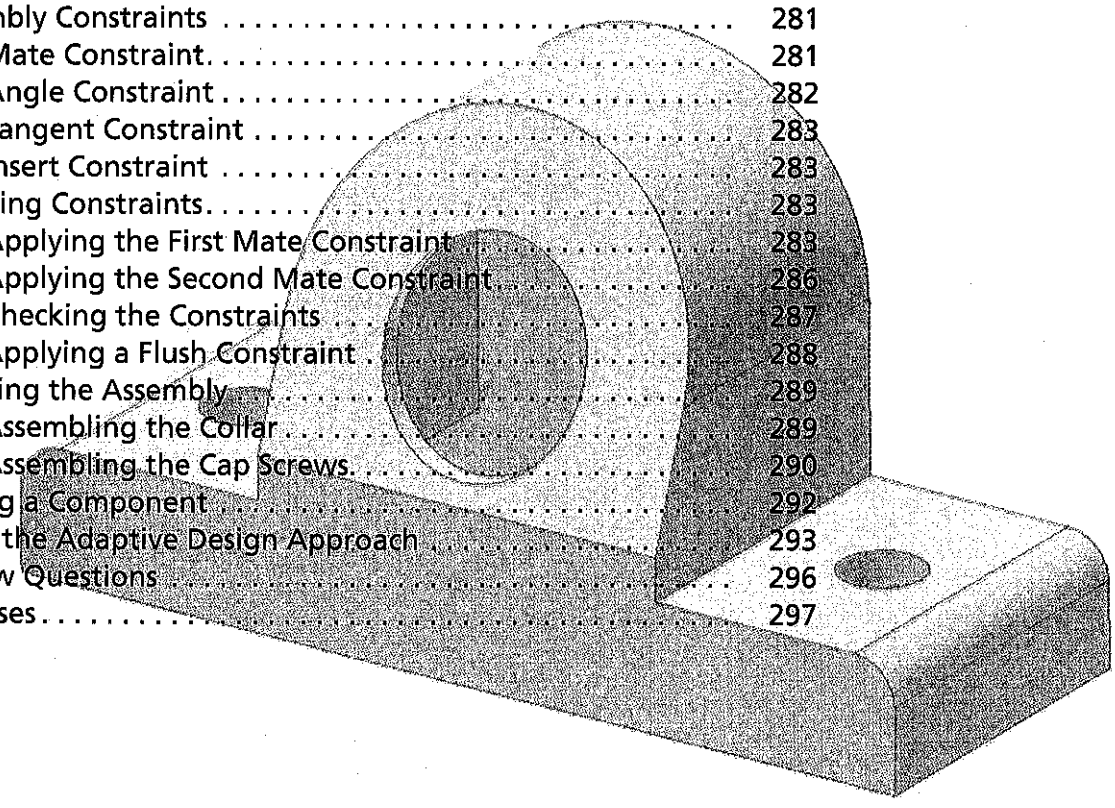

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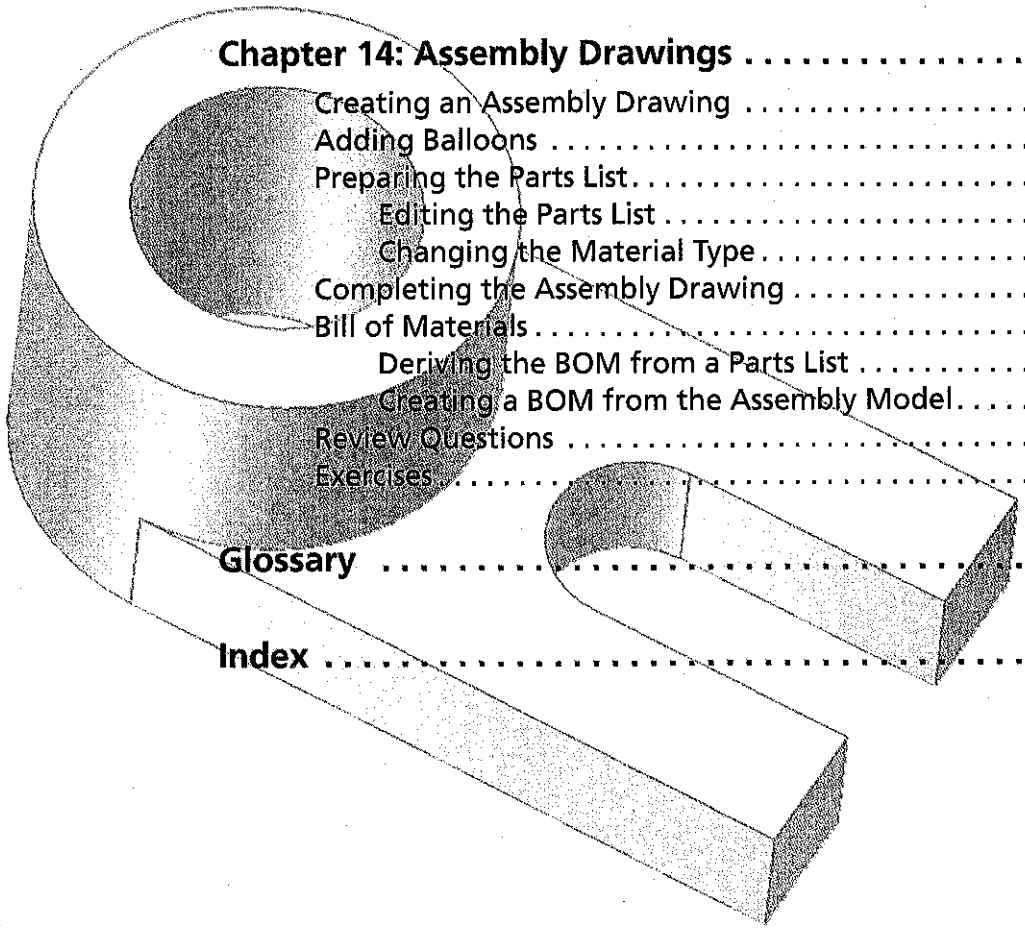
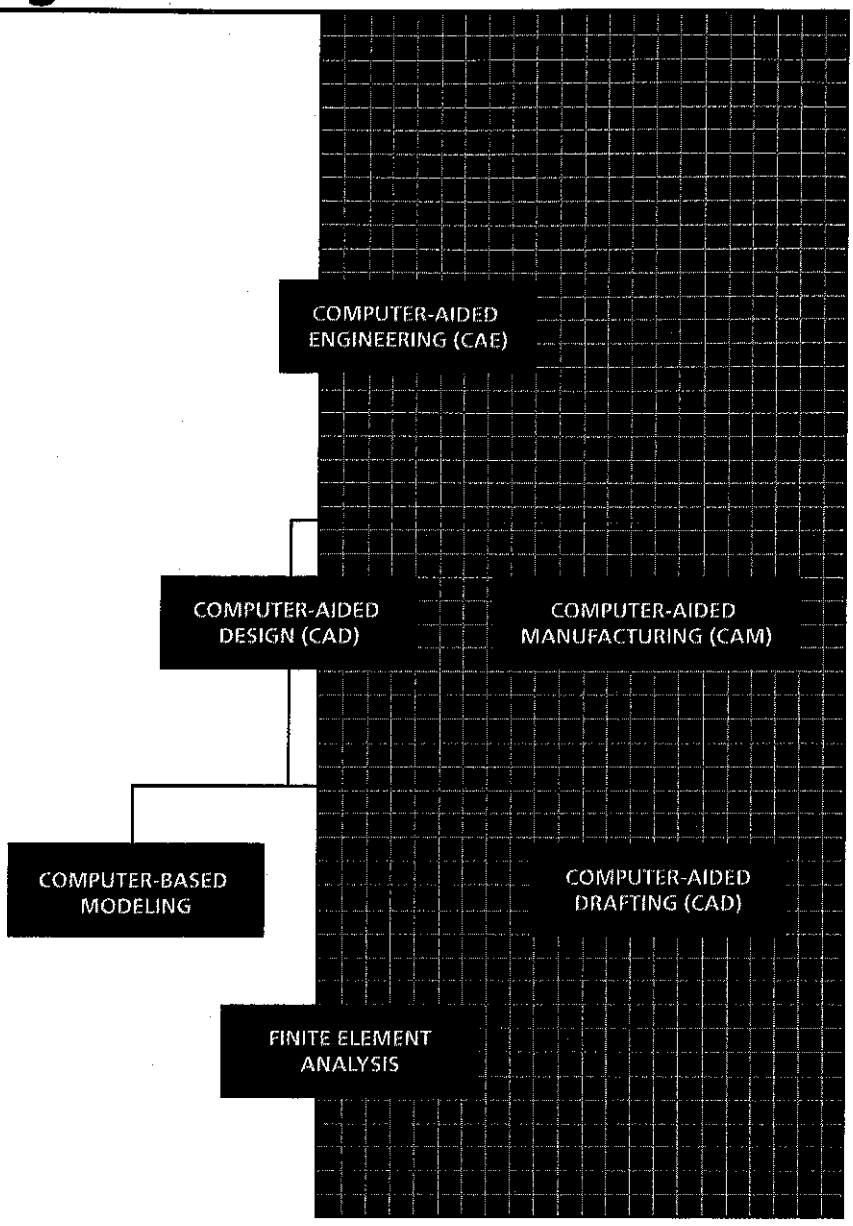


CHAPTER 1

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.



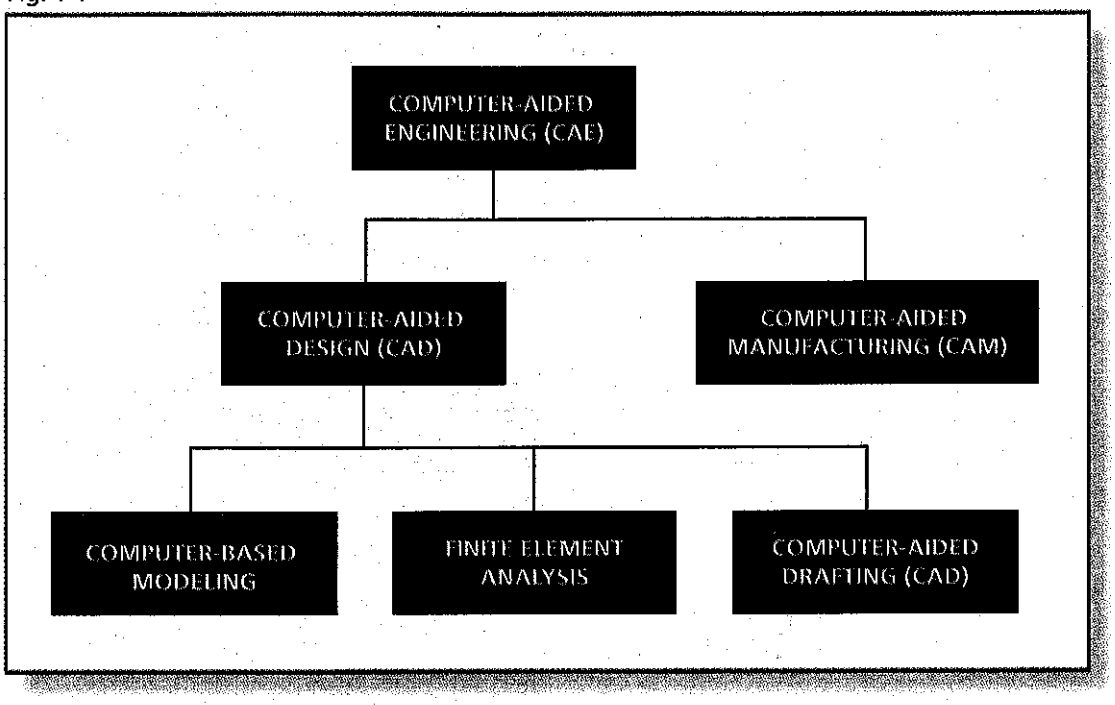
Vocabulary

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

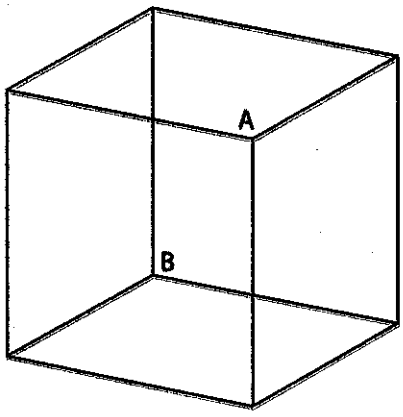
The first-generation **computer-aided drafting (CAD)** programs were simply 2D drafting programs, basically the electronic equivalent of the drafting board. For typical models, the use of this type of program would require that several to many views of the objects be created individually as they would 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

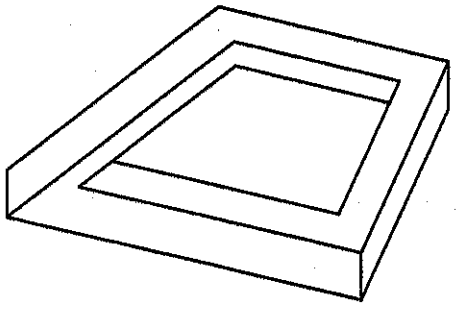
The development of three-dimensional modeling schemes started with 3D wireframes. **Wireframe models** are models consisting of points and edges, which are straight lines connecting appropriate points. The edges of wireframe models are used to represent transitions of surfaces and features, much as they 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



Wireframe ambiguity. Which corner is in front: A or B?



Non-realizable object. Wireframe models contain no surface definition.

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

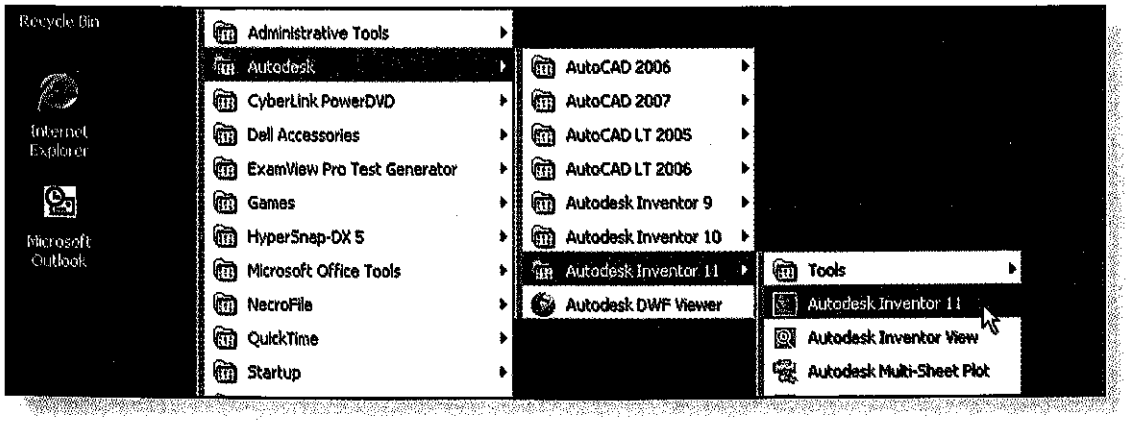
Inventor is composed of several application software modules called *applications* that share a common database. In this text, we will concentrate on the solid modeling modules used for part design and the general procedures required to create solid models, engineering drawings, and assemblies.

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.

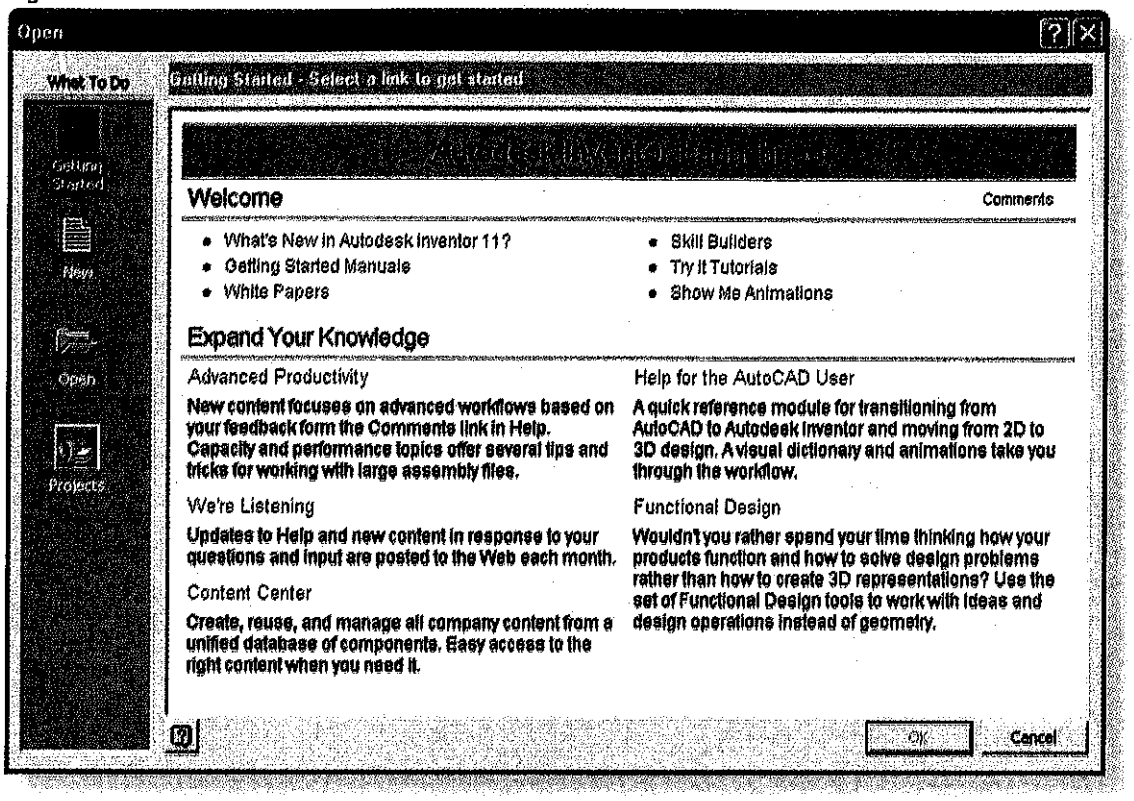
Fig. 1-3



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.

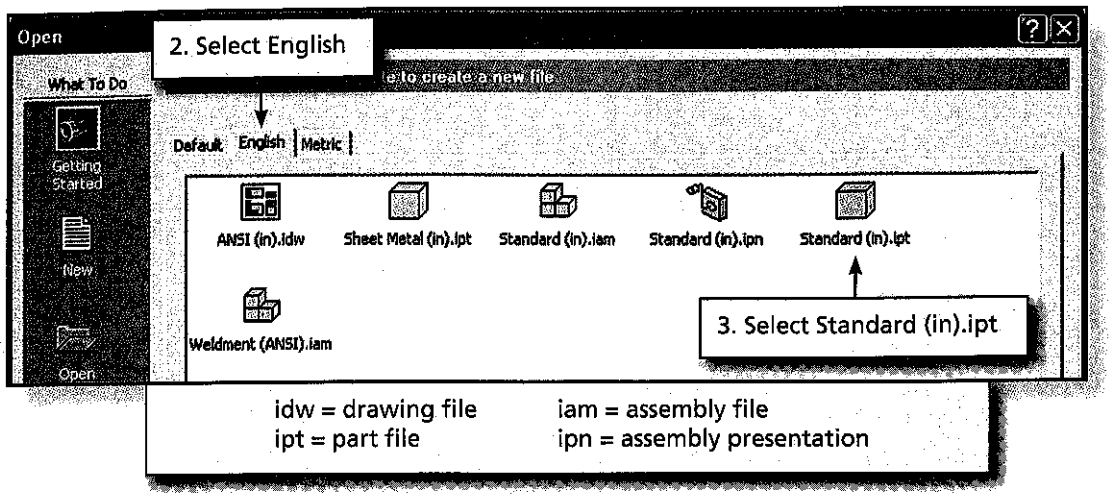
1. In the **Open** dialog box, select the **New** icon with a single click of the left mouse button.

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

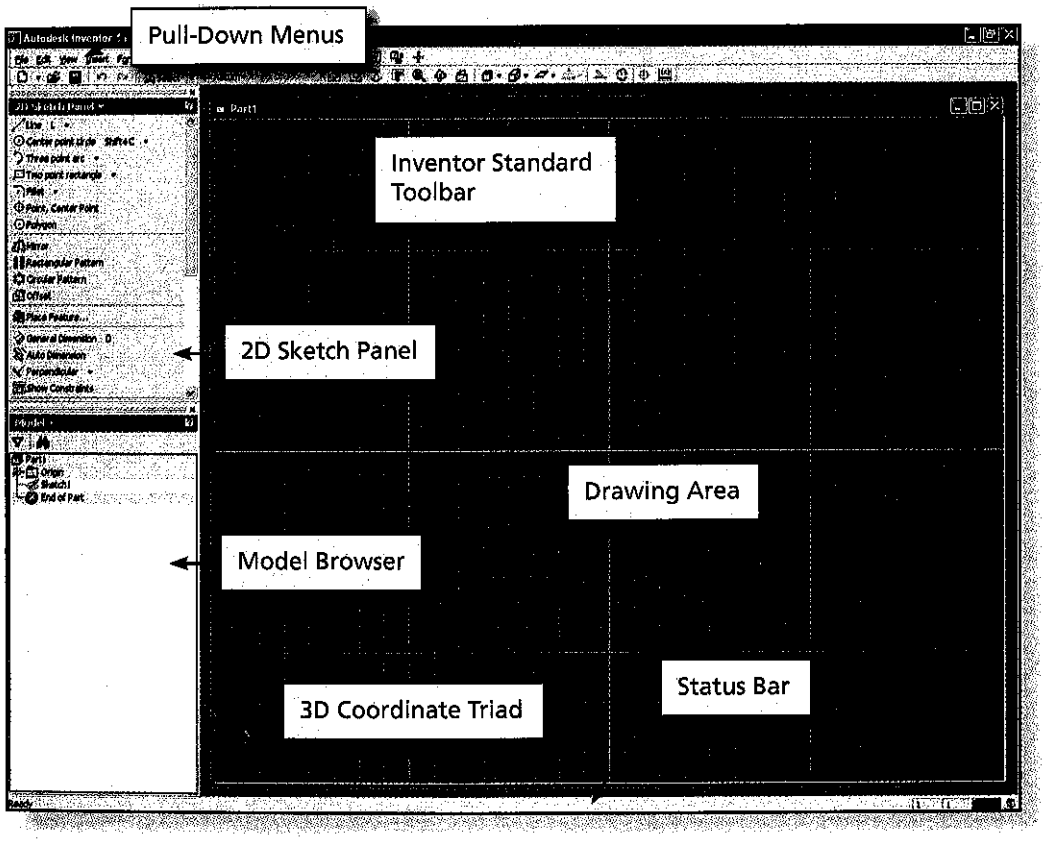


3. Select the **Standard (in).ipt** icon and pick **OK** to open the template. Fig. 1-5.

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.

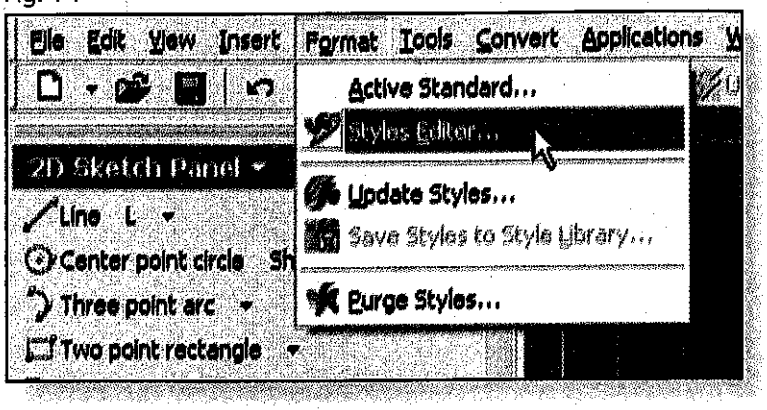
Fig. 1-6



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.

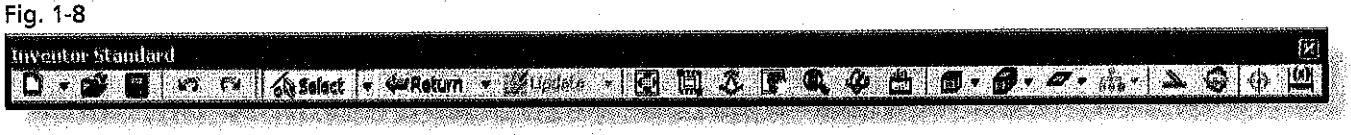
Fig. 1-7



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. 1-8



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

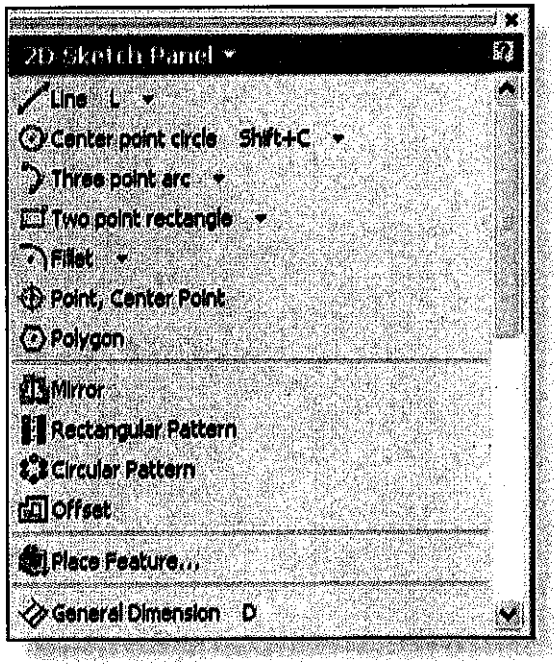
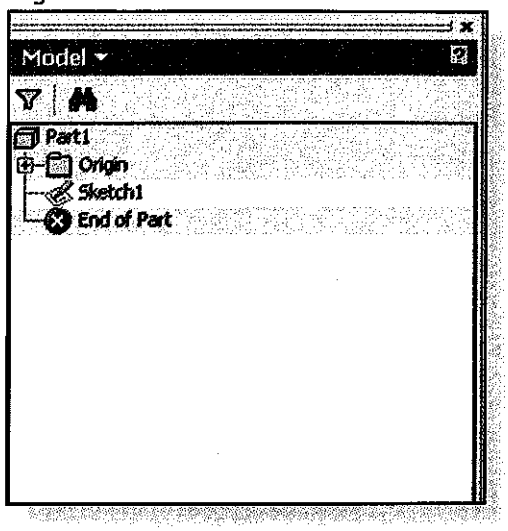


Fig. 1-10

Part Browser

Fig. 1-11

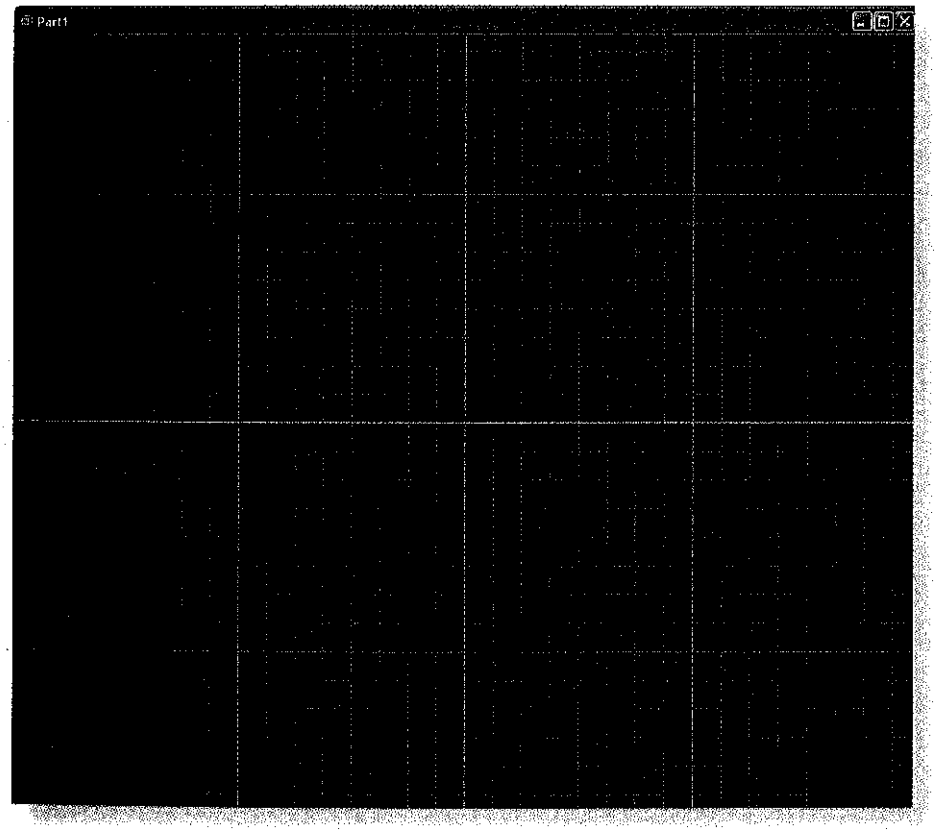


By default, the part browser is located just below the **2D Sketch Panel**. When you first create a new part, it will be titled **Model**, as shown in Fig. 1-11. The part browser is a tree view that shows the operations and features that have been created in the current file. It allows you to show, hide, 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.

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.

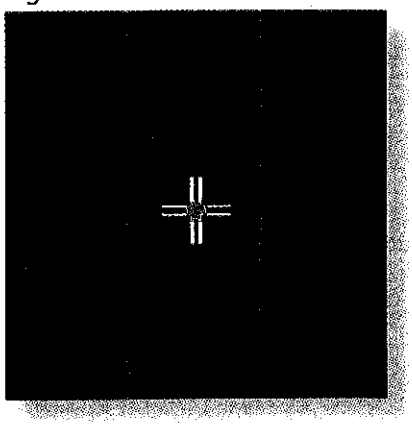
Fig. 1-12



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.

Fig. 1-13



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. 1-14

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.

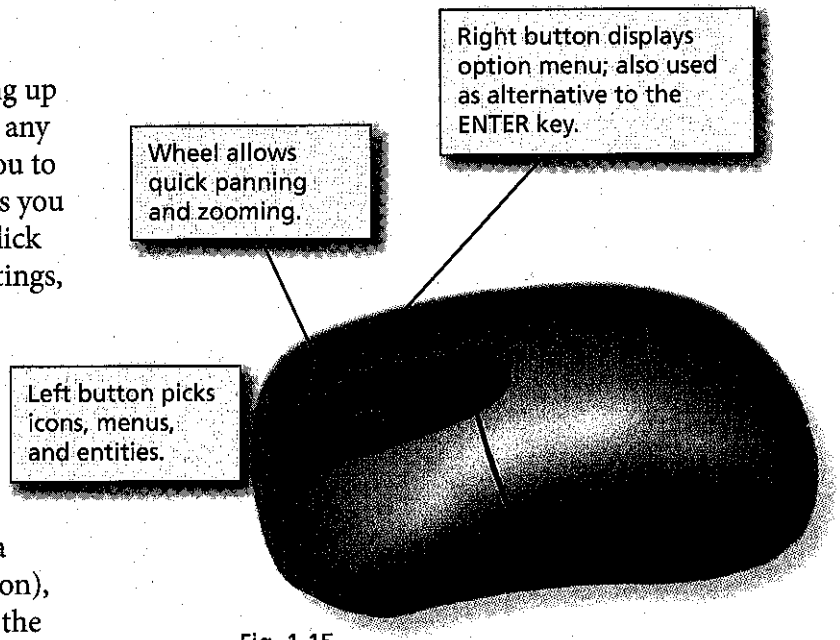
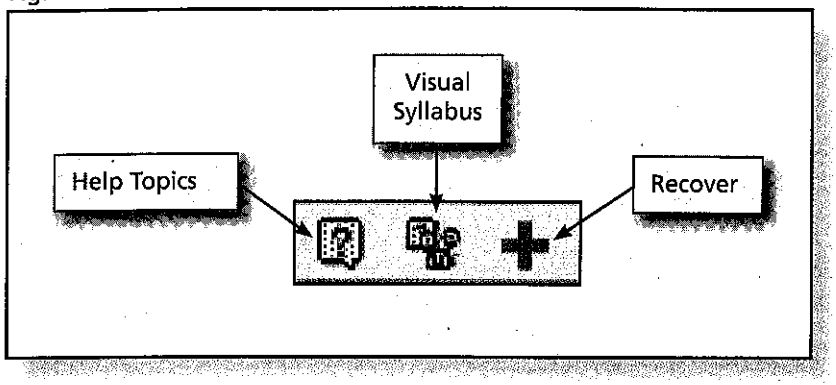


Fig. 1-15

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.

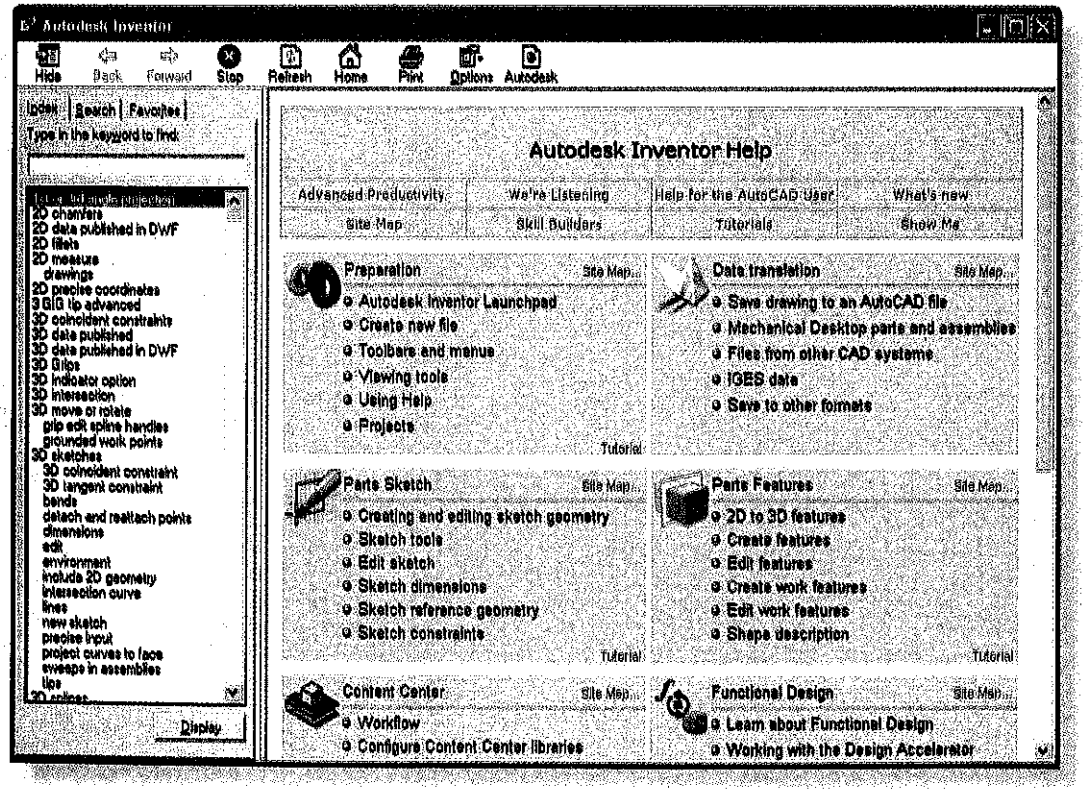
Fig. 1-16



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.

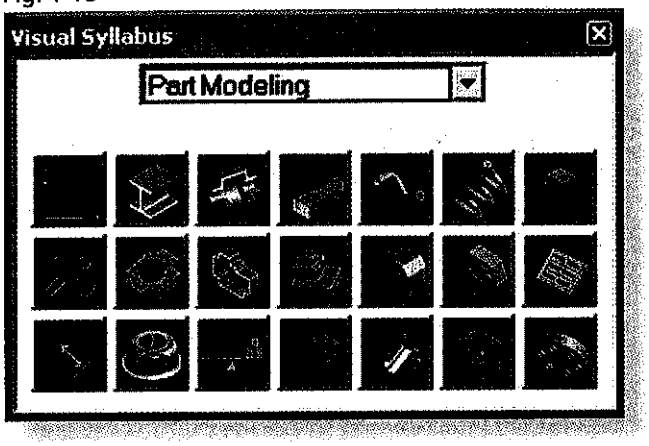
Fig. 1-17



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



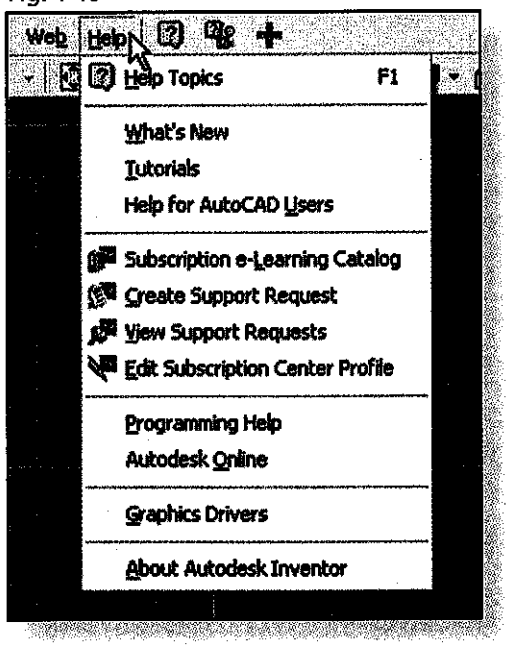
Recover Icon

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

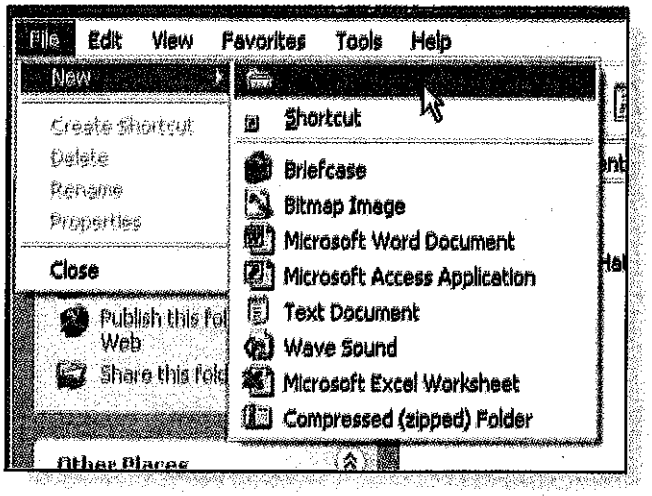
When you create and save files in Inventor, it is strongly recommended that you save them on the hard drive of your computer. It is a good practice to create a separate folder to store your CAD files. You should not save your files in the same folder in which the Inventor application files are located.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

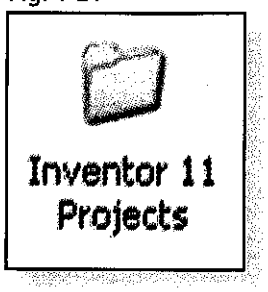
A new folder appears with the temporary name **New Folder**.

Fig. 1-20



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

Fig. 1-21



Canceling Commands

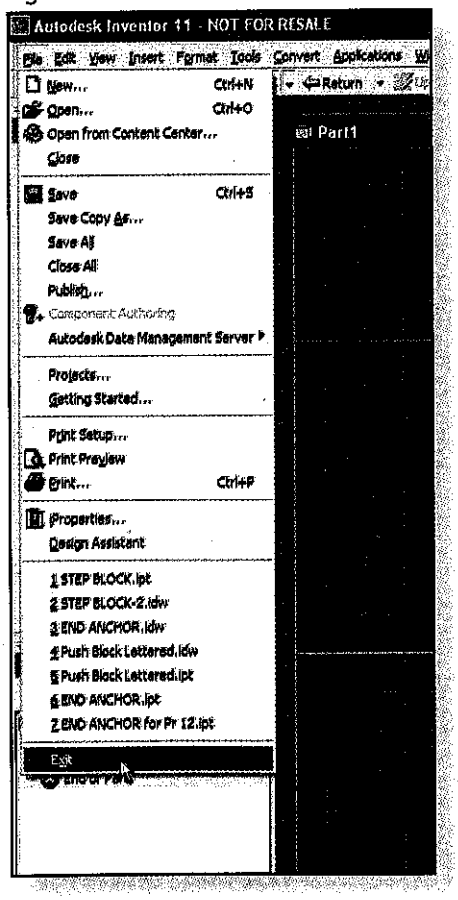
The ESCAPE key (ESC) is used to cancel a command in Inventor. The ESC key is located near the top left corner of the keyboard. It may sometimes be necessary to press the ESC key twice to cancel a command; it depends on where you are in the command sequence. For some commands, the ESC key is also used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

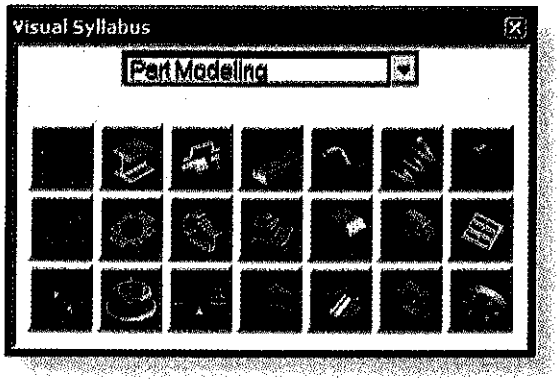
1. From the File pull-down menu, use the left mouse button to pick Save (for a single part or project) or Save All to save the current project. In this case, since you created a new model file in this chapter, pick Save and enter the name Model Example 1.
2. Move the cursor again to the File pull-down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