AUTODESK INVENTOR

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PREFACE

This work aims to be a hands-on, tutorial-driven introduction to *Autodesk Inventor*. Although often described as 3D parametric modeling software, it is perhaps more accurate to think of Inventor as product development software. Parametric modelers like Autodesk Inventor focus on creating virtual assemblies of parts (i.e., products) rather than standalone parts; for this reason the book is built around a couple of different product assemblies. The tutorials at the end of each chapter take the user through part and assembly modeling, drawing documentation, and finally, the simulation, analysis, and presentation of these products.

All of the print tutorials are duplicated in video on a CD that comes packaged with the book. This is done in part to accommodate different learning styles. Some users may well prefer the video approach to that of the text-based tutorials. Another reason for this approach is that the level of modeling difficulty encountered in these tutorials is perhaps more advanced than that typically found in an introductory CAD text. The built-in usability of the Inventor software, coupled with the step-by-step tutorials available both in print and video, should allow users to rapidly develop their modeling skills.

In another effort to accommodate different teaching and learning styles, the tutorials at the end of each chapter—while building on chapter content—can be done without prior reading of the chapter. With Inventor it is relatively easy to get new users up and running fast, allowing them to experience first-hand the thrill and excitement of 3D modeling.

Another characteristic of this book is an emphasis on *build strategy*. The steps required to complete each tutorial are summarized at the beginning of the tutorial. Taken together, these summary steps amount to a strategy for building each model. It is important for users new to 3D modeling to learn to think in terms of features, and to plan out a feature-based build strategy before starting to model. This format should also be helpful to users already familiar with 3D modeling. Rather than following the tutorials step-by-step, they can use the build strategy as a roadmap for building the part and assembly models.

To keep pace with future Inventor releases, a website is being maintained in conjunction with this book. Material on new tools and features, as well as other tools not addressed in the text, will be available for download from the website (www.mhhe.com/leake). Autodesk Inventor Release 6 was used in developing the material for this book.

In addition to the 3D parametric modeling material contained in the main body of each chapter, lecture notes in the form of PowerPoint slides are also included on the website. These slides cover a range of topics in engineering graphics, and are based on lecture notes currently used in an introductory engineering graphics course taught at the University of Illinois Urbana-Champaign.

The author and publisher would like to thank the following reviewers for their thoughtful comments on this text: Tom Bledsaw, ITT Educational Services; Joan Davis, Pellissippi State Technical Community College, Dan G. Dimitriu, San Antonio College, James B. Higley, Purdue University Calumet; Minh Quoc Pham, Houston Community Colleges/University of Houston; and James Shahan, Iowa State University.

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April 25, 2003

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CHAPTER 1

Getting Started

LEARNING OBJECTIVES

- Use the Open dialog box to access Inventor help, start new files, open existing Inventor files, and manage projects
- List the four file formats used in Inventor
- Create a new Inventor project using the Project Editor
- Locate Inventor tools from the panel bar and right-click context menus
- Identify the Browser bar on the Autodesk Inventor interface
- Access and list the seven default reference work features
- List several ways to access the Inventor Help system
- Use the viewing tools to control the viewpoint of the observer
- Use the display tools to change the display of the model

Introduction

Commercial Computer-Aided-Design (CAD) software has come a long way in twenty years. Originally developed to perform 2D manual drafting tasks, CAD software rapidly advanced in the 1990's to include 3D surface and solid modeling. Today mid-range Windows CAD software packages perform parametric assembly modeling, and are rapidly evolving into virtual product development tools. *Parametric solid modeling* employs parametric constraints to define part features, and to create relationships between these features in order to create intelligent part models. These parts can then be combined to form virtual assembly models.

Autodesk Inventor is a prime example of this next-generation CAD software. Inventor is the result of Autodesk's 1) twenty years' experience developing CAD products like AutoCAD and Mechanical Desktop, 2) making the decision to wipe the slate clean and start fresh, and 3) setting out to create a powerful, easy to use parametric solid modeler. Now with Release 6, Inventor also has added the capacity to handle hybrid (or nonmanifold) modeling, where freeform surfaces and parametric solids can be combined seamlessly to create complex parts.

Parametric modelers like Autodesk Inventor allow the user to model parts, and then combine these parts into an assembly. Further combinations of these component parts and sub-assemblies result in a virtual product model. The product examples

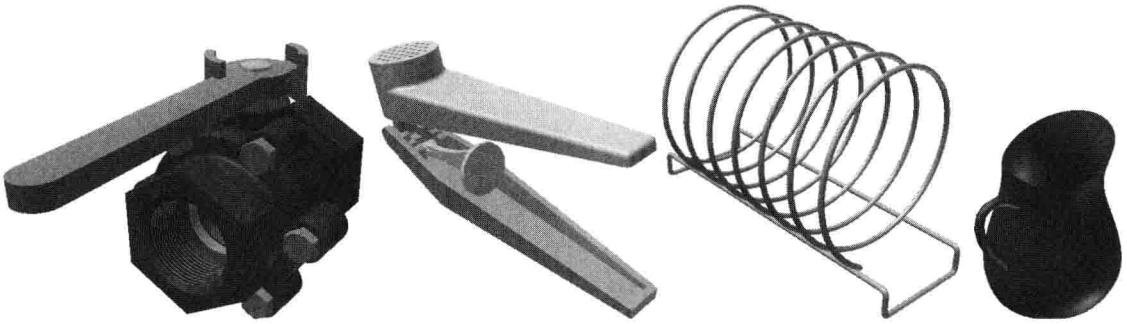

Figure 1.1 - Ball valve, garlic press, toast rack, creamer

shown in Figure 1.1 (ball valve, garlic press, toast rack, creamer) will all be modeled in the course of completing the tutorials that form the basis of this book.

Included among the by-products of the parametric modeling process are part and assembly drawings, exploded view drawings along with a Bill of Materials (BOM), and animation files showing either the assembly of a product, or the motion of a mechanism. Although not covered in this work, these same part and assembly files can then be used for such downstream applications as rapid prototyping, NC tool path generation, as well as finite element and kinematics analysis.

The fact that Inventor is a parametric modeler means that the parts and assemblies are easily edited and modified. This has a tremendous impact on the product development process. Families of parts can be created, products can be iteratively improved upon, multiple prototypes can be created and tested, etc.

Open Dialog Box

When Autodesk Inventor is first launched the Open dialog box appears, as shown in Figure 1.2. In the What To Do column on the left, four options are available: Getting Started, New, Open, and Projects. Several links appear on the Getting Started page. Of particular interest is the "Learn how to build models quickly" link, a gateway to several Inventor tutorials. The New option is used to start a new Inventor file, the Open option to open an existing Inventor file. The Projects option is used to open the Project Editor.

Inventor File Types

If New is selected from the Open dialog box, the right side of the dialog box changes to that seen in Figure 1.3. Three Tabs are now visible: Default on top, then English and Metric. Different icons are visible on the Default tab, each one corresponding to a template file. Inventor uses different file types to create and edit models. Each file type has a unique file format. These are (file extension in parenthesis):

- Part (ipt)
- Assembly (iam)
- Presentation (ipn)
- Drawing (idw)

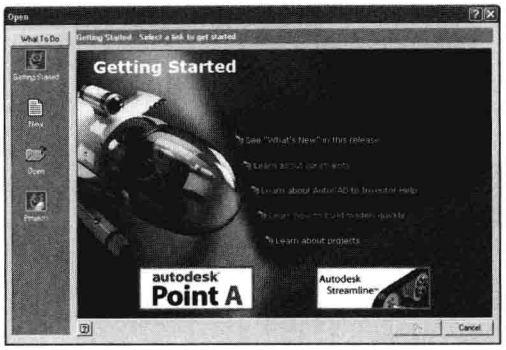

Figure 1.2 - Open dialog box: getting started

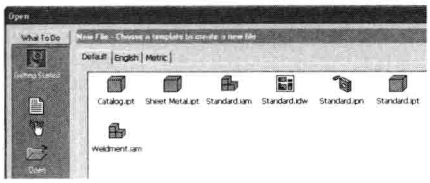

Figure 1.3 - Open dialog box: new

There are two kinds of part files, standard and sheet metal, as well as two kinds of assembly files, standard and weldment.

When Inventor is first installed the user is prompted to specify the default units system, either millimeters or inches. This selection determines the units of the templates available on the Default tab. Select either the Metric or the English tab and other template icons appear. Naturally the units of these templates correspond to that of the tab name. In this book metric predominates as the working unit.

Projects

Since Inventor employs several different file types, with each of these file types needing to share data with one another, file management becomes a significant issue. This is all the more the case in situations where many individuals work collaboratively on the same project. For the purposes of this book though, projects will simply be used to create a project folder (i.e., a workspace). When a given project is active, all file open and save operations default to this project folder. Only one project can be active at a time.

An Inventor project is really an ASCII text file, with an ipj file extension. This means that Inventor projects can be created and modified outside Inventor in a text editor. You will notice these ipj project files in your project folders.

User Interface

The composition of the Autodesk Inventor user interface is shown in Figure 1.4. Although this figure depicts the part file interface, the other Inventor file type interfaces are similarly arranged and configured.

Panel Bar

Regardless of file type, Inventor tools are most conveniently accessed from the Panel Bar. One of the nice things about Inventor is the clean, uncluttered interface. This is mostly due to the modularized "similar but different" work environments. Different tool sets are displayed in the Panel Bar depending upon the type of file (i.e., part, assembly, drawing, presentation) open, and also according to the active work environment (e.g., 2D sketch, 3D sketch, feature). The Panel Bar is normally docked in the upper left of the interface. Shown in Figure 1.5 are the sketch and feature tools as they appear in the (floating, expanded) Panel Bar for a part file.

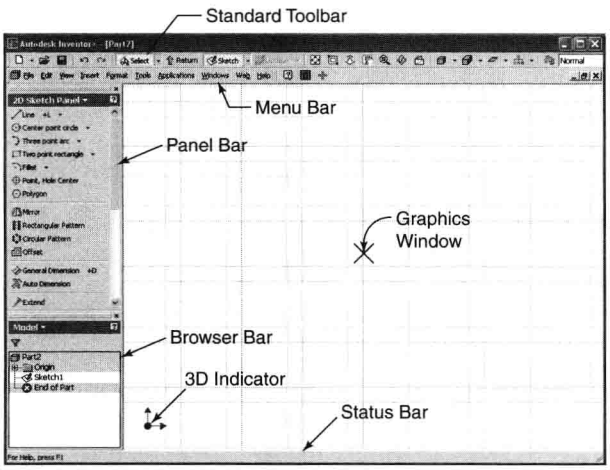

Figure 1.4 - Inventor interface

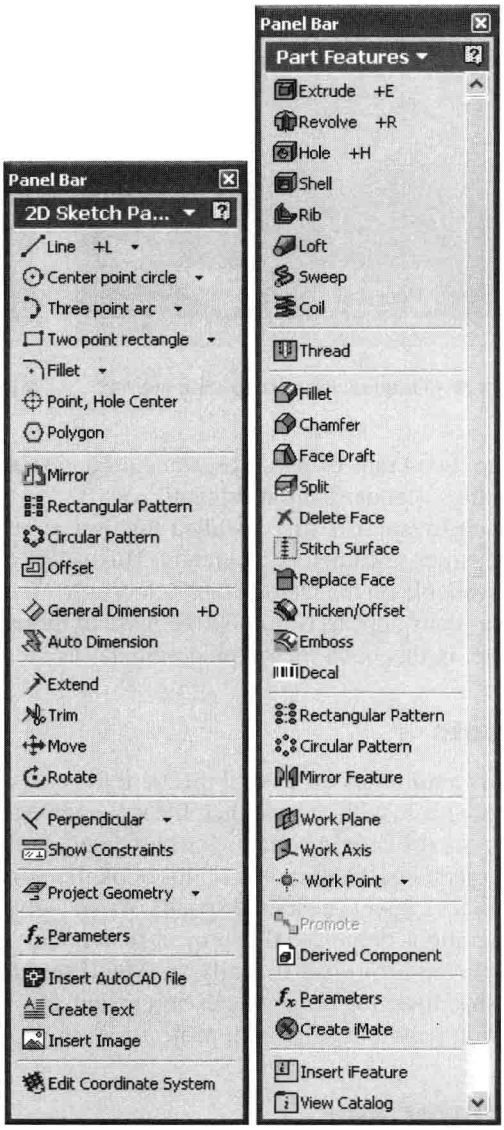

Figure 1.5 - Panel bars: 2D sketch and feature

In addition to the Panel Bar, Inventor tools are also frequently accessed from right-click context menus. Two such context menus are shown in Figure 1.6.

Browser Bar

The Browser Bar, as it is called in Inventor, is a hierarchical tree structure used to display file information. Most parametric modelers employ tree structures similar to the Browser Bar. The Inventor browser changes according to file type (i.e., part, assembly, drawing, presentation). In the case of a part file, the browser would generally be referred to as a feature tree. In Inventor the part file feature tree is called the Part Browser.

TIP: The visibility of both the Panel Bar and the Browser Bar, as well as other toolbars, can all be controlled using View > Toolbar from the menu bar.

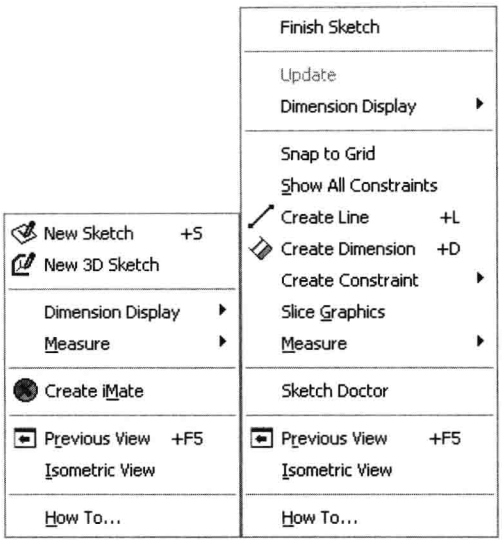

Figure 1.6 - Right-click context menus

Default Reference Work Features

The first element in the Browser Bar (part browser) is the Origin folder. If the Origin folder is expanded by clicking on the + symbol at the left, the folder expands to reveal three work planes (YZ, XZ, and XY), three work axes (X, Y, and Z), and a work point (Center Point). By default, these reference work features are not visible. By moving the mouse over one of the features, however, the feature becomes visible in the graphics window. By right-clicking on any of these work features and selecting Visibility, the feature remains visible in the graphics area. Figure 1.7 shows an expanded Origin folder with visible default work features, and an isometric view. From this it can be seen that the work planes and axes are all mutually perpendicular and that the Center Point is at their intersection.

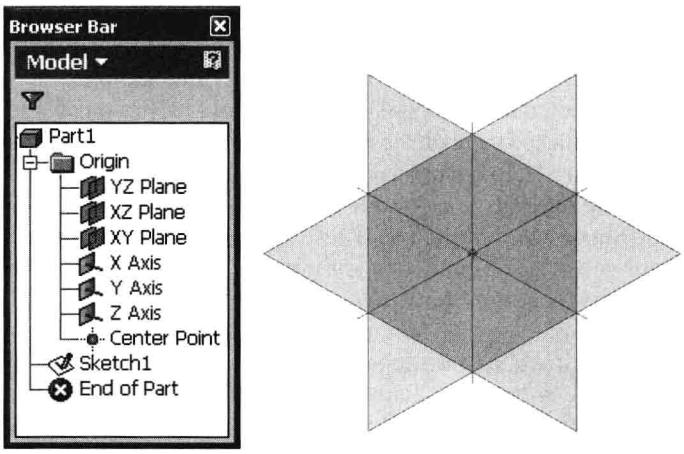

Figure 1.7 - Expanded origin folder with visible work features

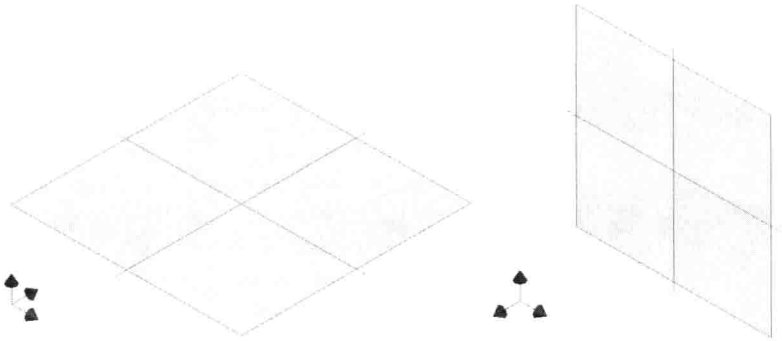

Figure 1.8 - Isometric view of XY Plane; metric template on left, English template on right

These reference work features will be used extensively in this book. The default work planes are useful as sketch planes, or as an aid in creating other (e.g., offset, inclined) work planes. In addition, by constraining sketch geometry to the default work features, the construction of symmetrical parts is considerably simplified. Using the Project Geometry tool, the default work axes and center point can be projected on any sketch plane.

By default, new part files open in 2D sketch mode. This default sketch plane is the XY Plane seen in the Origin folder. In fact though, any of the three reference work planes can be used for the first (or subsequent) sketches. In this book the first part sketch will almost always be drawn on the default XY Plane.

Another point to note is that the XY Plane in an Inventor metric template corresponds to a top, or plan view (see Figure 1.8 on left), whereas in an English template the XY Plane corresponds with a front view (see Figure 1.8 on right).

Autodesk Inventor Help

Just as CAD systems have made significant advances in the past twenty years, so have the software Help systems that support them. Autodesk's built-in Help for Inventor is excellent, so good in fact that it significantly impacts the format of this book. Rather than repeat what is already embedded in the software help, readers are strongly advised to make use of what is already available. This book in turn attempts to cut through the confusing array of multiple options, and provide straightforward hands-on examples of part and assembly modeling, as well as drawing documentation.

We have already noted that Autodesk has provided fairly extensive tutorials available from the Getting Started page on the Open dialog box. These same tutorials can also be accessed by selecting Help > Tutorials from the Menu Bar, as shown in Figure 1.9.

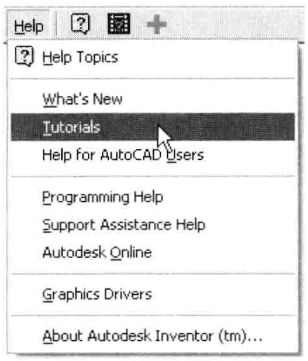

Figure 1.9 - Help from Menu Bar

In addition to the tutorials, the Autodesk Inventor help system also includes literally hundreds of Show Me animations. These brief step-by-step animations demonstrate how to complete a task or understand a concept. The Visual Syllabus icon on the Menu Bar is the gateway to the Show Me animations (see Figure 1.10).

The Visual Syllabus palette is shown in Figures 1.11 and 1.12. The Syllabus organizes the Show Me animations into separate palettes according to the different work environments available in Inventor: Part Modeling, Sheet Metal, Assembly Modeling, Presentations, and Drawings (Figure 1.11). Each icon on a given palette either directly launches an animation, or opens a window with a menu of animations (Figure 1.12).

The main Inventor Help system can be accessed using the Help Topics icon on the menu bar. The Inventor Help Topics, shown in the Figure 1.13, are similar in

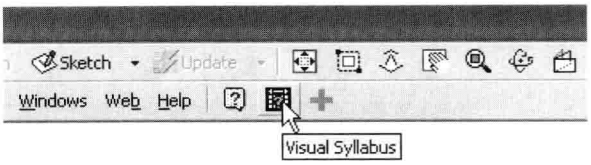

Figure 1.10 - Visual syllabus icon on Inventor Menu Bar

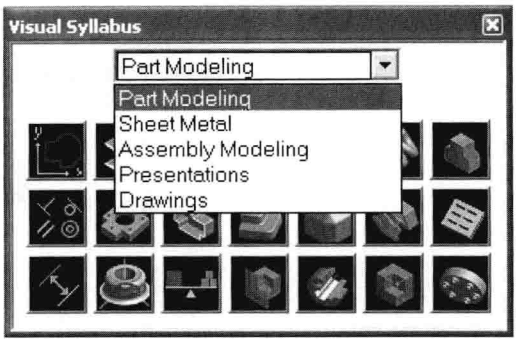

Figure 1.11 - Visual syllabus palette options

Figure 1.12 - Visual Syllabus icons and Show Me animation menus

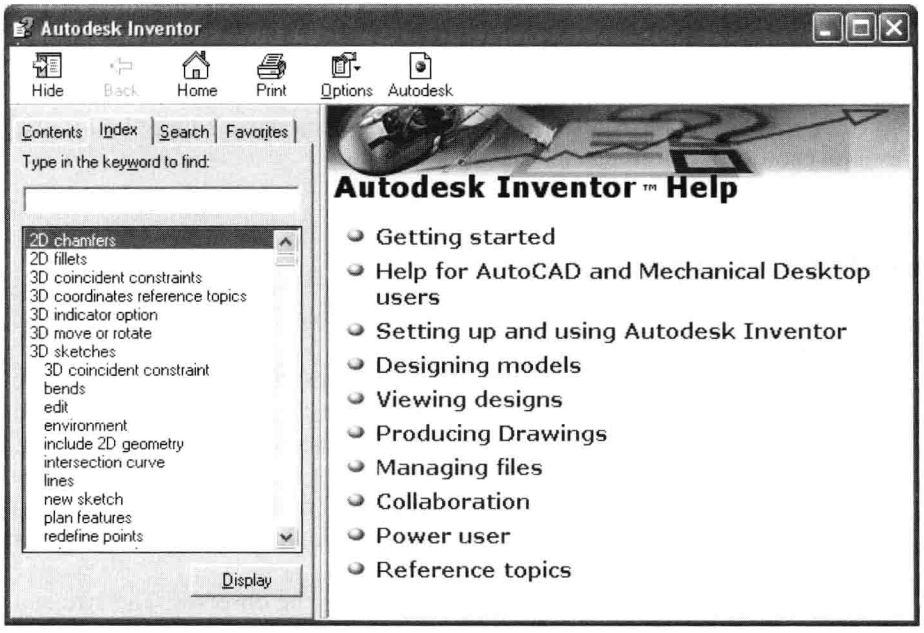

Figure 1.13 - Help topics

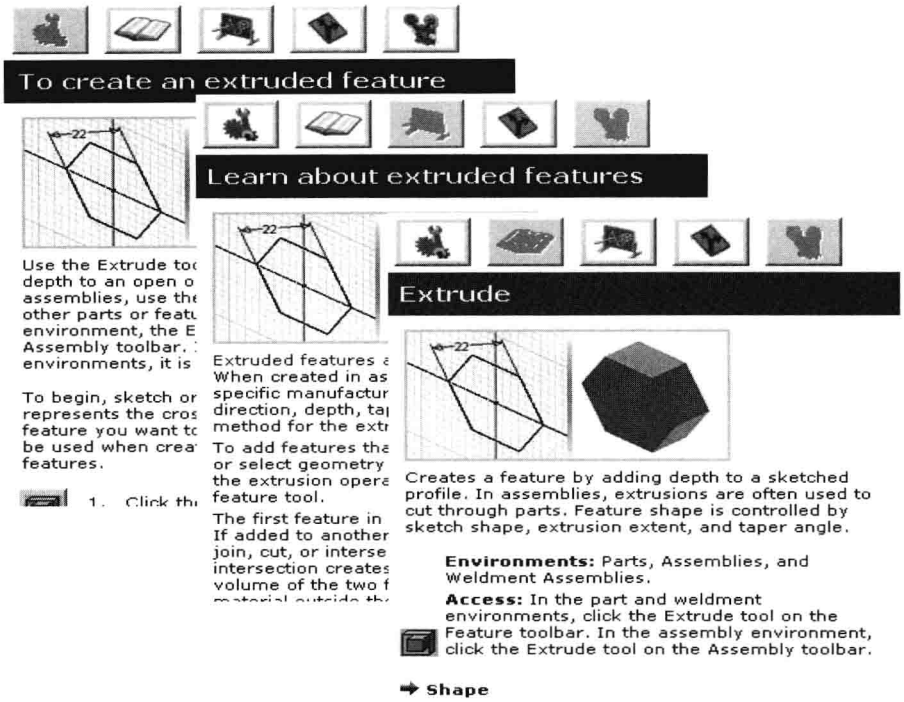

Figure 1.14 - Color coded help page

organization to other Windows software programs. Four tabs are provided in the Help window: Contents, Index, Search, and Favorites. The Contents tab gives access to a Table of Contents format, Index to an Index format, and Search to a search engine format. Favorites can be used to bookmark particularly useful help pages.

The different Help Topics pages are organized into one of three different categories; How To, Learn about, and Reference. Figure 1.14 shows the three different categories of help for a single topic, extruded features. Note that the three different Help Topic categories are color-coded; "How to" pages are a navy color, "Learn about" pages a rust color, and "Reference" pages a teal color.

Viewing and Display Tools

The viewing tools are available from the Standard toolbar. The names of the different viewing tools are indicated in Figure 1.15.

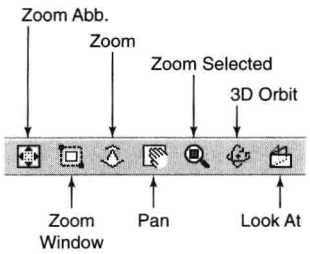

Figure 1.15 - Viewing tools

Information on using the different viewing tools can be found by entering "viewing tools" from the Index tab of Autodesk Inventor Help, then clicking the Display button. This brings up the Topics Found dialog box. The "Viewing tools reference" page, shown highlighted in Figure 1.16, gives an overview of the viewing tools. The other topic links shown below demonstrate how to use the different tools. The "Rotate the view in the graphics window" page also has a Show Me link to an animation demonstrating the use of the 3D Rotate tool.

Viewing tools are used to control the viewpoint of the observer. The size and/or location of a model are not changed by the different viewing commands.

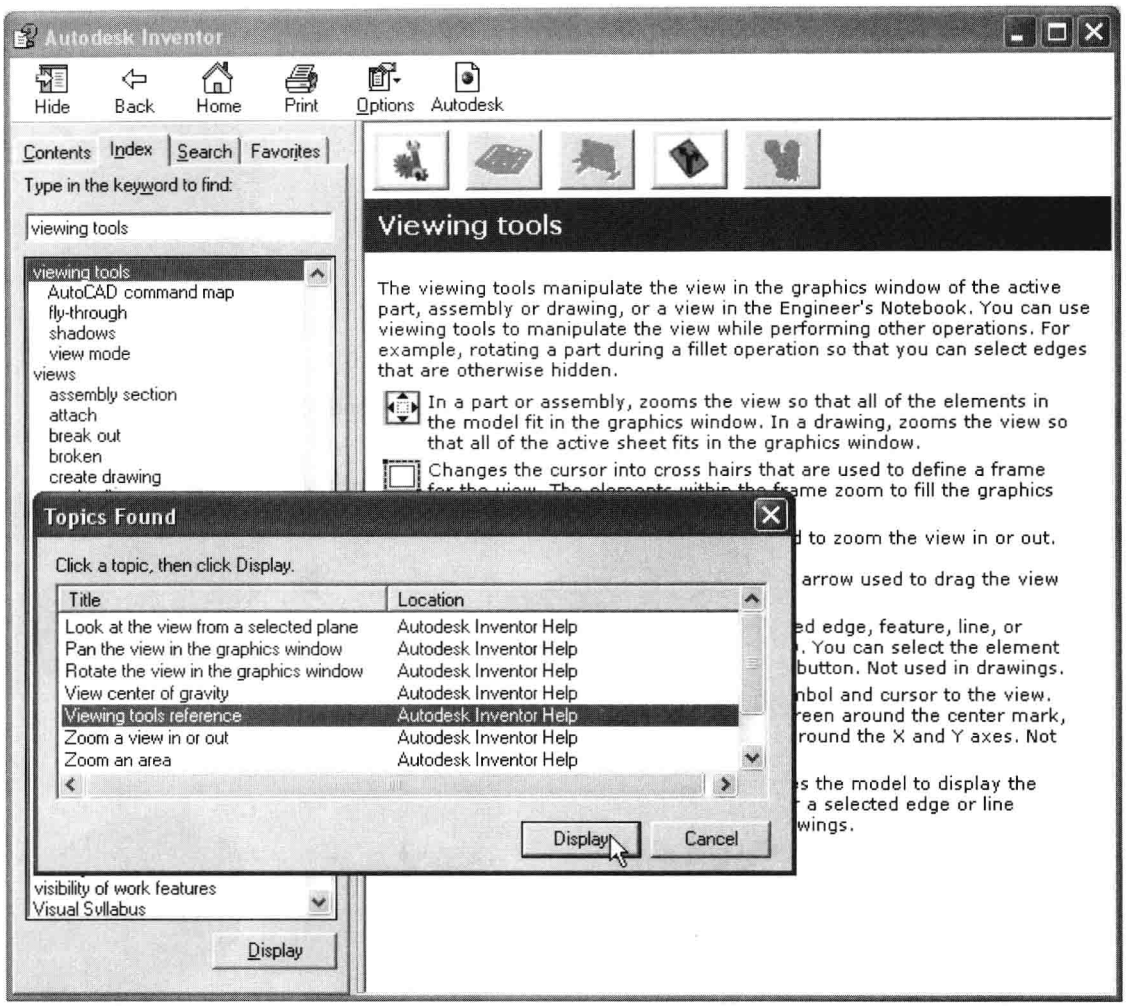

Figure 1.16 - Viewing tools help

Viewing tools can be used in the middle of another operation. Once the view has been changed, the operation resumes.

Right-clicking in the graphics area opens a context menu. Select Isometric View, as shown in Figure 1.17, and an isometric view is composed and centered within the graphics window.

With the 3D Orbit tool in operation, the SPACE bar acts as a toggle between the Free Rotate and Common View modes (Figure 1.18).

Common View can be used to access the eight different *isometric* and six different *principal views*. Highlight one of the Common View arrows so that its color turns red, and then left-click to change to one of these standard views.

While Common View is in operation, it is also possible to redefine the default isometric view. For example, it may be desirable to change the default isometric view on the left in Figure 1.19 to that shown on the right.

To accomplish this, proceed as shown in Figure 1.20. In Figure 1.20A, with Common View in operation, select the highlighted arrow. The view changes to an

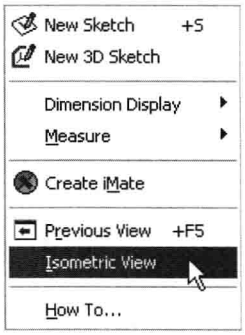

Figure 1.17 - Isometric view access using the context menu

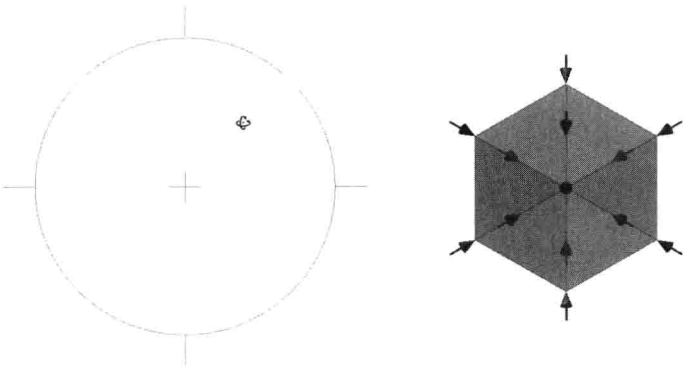

Figure 1.18 - 3D Orbit: free rotate and common view

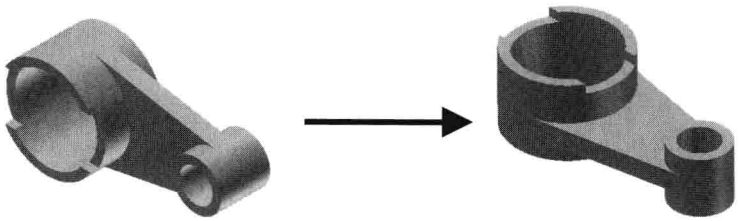

Figure 1.19 - Two different isometric views

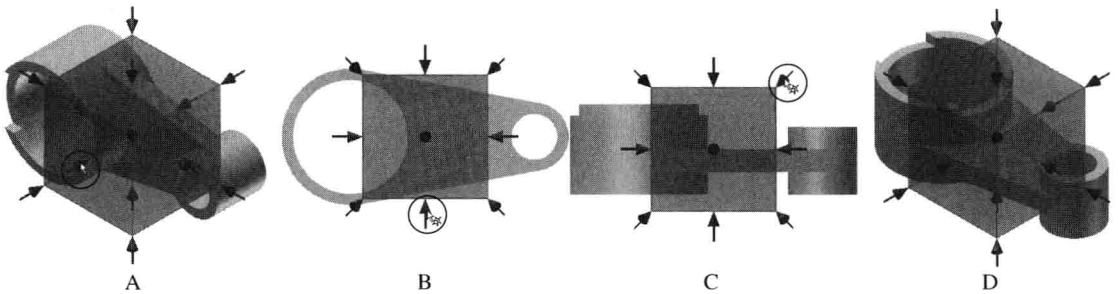

Figure 1.20 - Use of common view tool

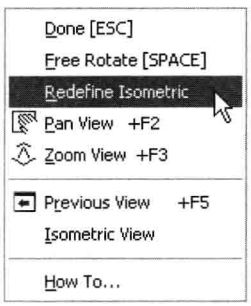

Figure 1.21 - Redefine isometric

orthographic view (Figure 1.20B). Clicking the highlighted arrow shown Figure 1.20B causes the view to change again, to that shown in Figure 1.20C. Now select the isometric view arrow shown in Figure 1.20C. The resulting view is shown in Figure 1.20D.

Finally, to redefine the default isometric view, right-click. The context menu shown in Figure 1.21 appears. Select Redefine Isometric.

The display, camera view, and shadow tools available in Autodesk Inventor are accessed from the Standard toolbar, and are shown in Figure 1.22.

Inventor display options include shaded, hidden edge, and wireframe, as shown in Figure 1.23.

By default, Inventor uses *orthographic projection*. It is also possible to view models using *perspective projection* techniques. Switching between orthographic and perspective projection is also done from the Standard toolbar, as shown in Figure 1.24.

New with Release 6 is the ability to display shadows of part and assembly models. These options are also accessed from the Standard toolbar, as shown in Figure 1.25.