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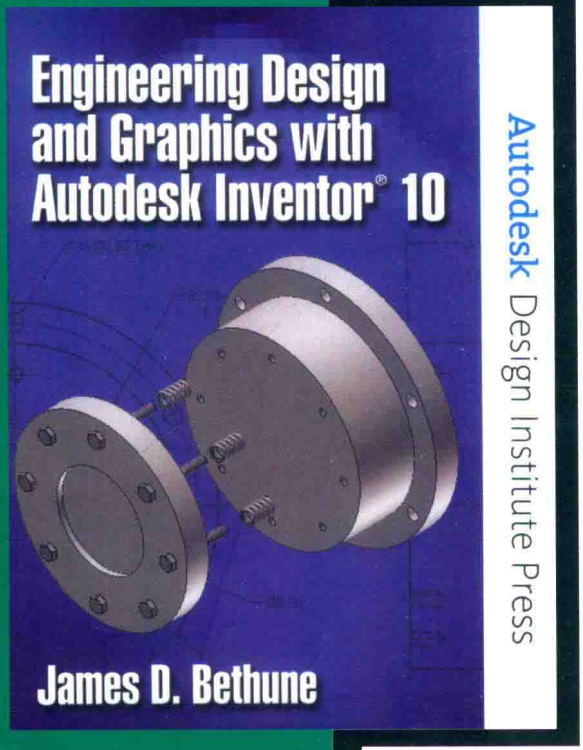
国外大学优秀教材 —— 工程图学系列(影印版)

James D. Bethune 著
窦忠强 改编

Autodesk Inventor® 10 工程设计绘图



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Autodesk Inventor® 10 工程设计绘图

Engineering Design and Graphics with Autodesk Inventor® 10

James D. Bethune 著

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清华大学出版社

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丛 书 序 言

本套丛书是由清华大学出版社和中国工程图学学会图学教育专业委员会共同策划的。

双语教学是近年来国内高校的教学改革热点之一，目前在数十所高校中已经开展了制图课程的双语教学。

从目前国内开展双语教学的高校使用的教材来看，大体上有以下几种情况：①直接选用欧美原版教材；②中国的制图教师根据我国的教学基本要求改编的原版教材，并以附录的形式讲解投影法和标准方面的差异；③中国的制图教师编写的英文教材；④中国的制图教师编写的中英文对照的双语教材等。

为了给我国高校的制图教师开展双语教学时提供更多的教材选择，也为了使我国高校的广大师生对美国制图课程的现状有更多的了解，清华大学出版社和中国工程图学学会图学教育专业委员会决定出版这套丛书。经过编委会一年多的分析与研究，我们从数十本美国原版教材中选择了 6 本构成了本套丛书，包括机械类的制图教材两本，近机械类与非机械类的制图教材两本，CAD 与计算机图形学方面的教材两本。需要说明的是美国的制图教材并未按照上述方式分类，所谓不同的类别是由本套丛书的编委会根据其内容来确定的。

由于美国原版教材的内容远远多于我国同类教材的内容，编委会根据我国的实际情况，以“教学基本要求”为依据，对其内容进行了删减，在这一过程中，未对原版教材作任何改写，以保证其“原汁原味”的风格。我们希望通过这种方法，给开展制图课双语教学的院校提供一套既能保持原版教材风貌，又符合我国实际情况的英语教材。

最后，清华大学出版社及本套丛书的编委会对积极提供样书供编委会选择的美国麦格劳-希尔公司和培生公司表示衷心的感谢，是他们的积极配合使得这套丛书得以顺利出版。

限于改编者的水平，书中不当之处在所难免，欢迎广大读者批评指正。

国外大学优秀教材——工程图学系列编委会

2007 年 3 月

影印版序言

本书原版有 14 章，加书后索引，共计 550 页。书的内容涉及 Autodesk Inventor® 10 的基本功能，既有基本操作的详述，又有结合工程实际问题的例题和习题，是一本教授和学习该软件的实用教材。

在本书影印时，考虑到我国的工科院校对 CAD 课程的教学要求、授课学时等情况，对原书作了必要的删节，具体如下：

(1) 删去第 8 章，共 82 页。这一章主要讲述尺寸公差和形状位置公差标注等，其中还有很多实际测量的方法介绍，对初学者的工程知识背景要求较高。

(2) 删去第 13 章，共 18 页。这一章介绍机械设计的过程和方法，与软件无直接关系。

(3) 删去第 14 章，共 18 页。这一章是机械和机构设计的练习题目，涉及较多的工程力学、机械制造的基础知识，不适于一般读者学习，与软件操作也无直接关联。

以上删除内容约占全书的 21%。

经过删节处理后，本书页眉保留原版书的页码，页脚是连续的新书页码。文中提到的页码均为原版书页码。本书保留原版书目录和索引，目录和索引的页码为原版书的页码，与正文页眉处的页码对应。有的内容或页码有可能已被删除从而无法找到，由此给读者带来不便，敬请谅解。

北京科技大学　窦忠强

2007 年 12 月

Preface

This book introduces Autodesk Inventor 10 and shows how to use Autodesk Inventor to create and document designs. The content of the book goes beyond the material normally presented in an engineering graphics text associated with CAD software to include exercises requiring students to design simple mechanisms. The book also presents a number of projects based on the concepts of Project-Based Learning (PBL). These projects not only serve to help students learn how to create drawings, but also help start an understanding of fundamental engineering design concepts.

All topics are presented using a step-by-step format so that the reader can work directly from the text to the screen. The book contains many sample problems that demonstrate the subject being discussed. Each chapter contains a variety of exercise problems that serve to reinforce the material just presented and allow the reader to practice the techniques described.

Chapters 1 and 2 present 2D sketching commands and the Extrude command. These chapters serve as an introduction to the program.

Chapter 3 demonstrates the commands needed to create 3D models, including the Shell, Rib, Split, Loft, Sweep, and Coil commands. Work points, work axis, and work planes are explained and demonstrated.

Chapter 4 shows how to create orthographic views from 3D models. The creation of isometric views, sectional views, and auxiliary views is also covered.

Chapter 5 shows how to create assembly drawings using both the bottom-up and top-down process. The chapter includes presentation drawings and exploded isometric drawings with title blocks, parts lists, revision blocks, and tolerances blocks. There is an extensive step-by-step example that shows how to create an animated assembly, that is, a drawing that moves on the screen.

Chapter 6 covers threads and fasteners. Drawing conventions and callouts are defined for both inch and metric threads. The chapter shows how to calculate thread lengths and how to choose the appropriate fastener from Inventor's Content Center. The Content Center also includes an extensive listing of nuts, setscrews, washers, and rivets.

Chapter 7 shows how to apply dimensions to drawings. Both ANSI and ISO standards are demonstrated. Different styles of dimensioning, including ordinate, baseline, and Inventor's Hole Table, are presented. Applying dimension to a drawing is considered an important skill, so many examples and sample problems are included.

Chapter 8 is an extensive discussion of tolerancing, including geometric tolerances. The chapter first shows how to use Inventor to apply tolerances to a drawing. The chapter then shows how to calculate tolerances in various design situations. Positional tolerances for both linear and geometric applications are included. The chapter introduces the Limits and Fits option of the Design Accelerator tool. The information contained in this option eliminates the need for an appendix that includes fit tables.

Chapter 9 presents bearings and shafts. The chapter shows how to calculate clearances for shafts and bearings and how to select bearings from manufacturers' catalogs and

from the Web. Shear and bending diagrams are introduced (algebra only) and the results are used to calculate minimum shaft diameters and critical speeds. The chapter shows how to use the Shaft option of the Design Accelerator tool to calculate shear and bending values and how to use the Beam option to calculate beam deflections.

Chapter 10 introduces gears. Gear ratios, gear trains, the Lewis equation, and forces in gears are covered. Center distance and backlash are included as part of an explanation of how to design gear boxes. The Spur Gears option of the Design Accelerator tool is presented in detail. This option allows students to select and create drawings of gears quickly and accurately. These gear drawings can be entered into drawings. The chapter also shows how to animate gears.

Chapter 11 presents cams, springs, and keys. This chapter makes extensive use of the Design Accelerator tool. Displacement diagrams can easily be created and the resulting cam entered directly into the drawing. Likewise, springs can easily be designed and entered into drawings. The chapter includes a discussion of various types of keys and how they are toleranced and applied to keyways.

Chapter 12 introduces sheet metal and weldments drawings. The chapter shows how to draw basic sheet metal parts and how to redesign existing parts as weldments. Only fillet and groove welds are used.

Chapter 13 shows how to manage the design process. The process starts with concept sketches and evaluation of various design ideas to select the optimal design. Evaluation matrices, team calendars, responsibility charts, and Gantt charts are applied to sample design problems. A sample design problem is taken from the initial problem statement to one possible solution.

Chapter 14 presents 10 project design problems. This chapter is intended to support the concept of project-based learning (PBL). Rather than have students solve a given exercise problem, students are assigned a project. These projects are competitive in that students compete against each other for the best performance. The idea behind PBL is that students will learn better when trying to solve a specific real-life problem and that they will start to understand basic engineering concepts such as tolerances, friction, energy transfer, materials, packaging, and ease of construction.

Online Instructor's Manual

An online Instructor's Manual is available to qualified instructors for downloading. To access supplementary materials online, instructors need to request an instructor access code. Go to **www.prenhall.com**, click the **Instructor Resource Center** link, and then click **Register Today** for an instructor access code. Within 48 hours after registering, you will receive a confirming e-mail including an instructor access code. Once you have received your code, go to the site and log on for full instructions on downloading the materials you wish to use.

Autodesk Learning License

Through a recent agreement with Autodesk Inventor publisher, Autodesk®, Prentice Hall now offers the option of purchasing *Engineering Design and Graphics with Autodesk Inventor® 10* with either a 180-day or a 1-year student software license agreement. This provides adequate time for a student to complete all the activities in this book. The software is functionally identical to the professional license, but it is intended **for student or faculty personal use only**. It is not for professional use. For more information about this book and the Autodesk Learning License, contact your local Pearson Prentice Hall sales representative, or contact our National Marketing Manager, Jimmy Stephens, at 1-800-228-7854, X3725 or at Jimmy_Stephens@prenhall.com. For the name and number of your sales rep, please contact Prentice Hall Faculty Services at 1-800-526-0485.

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James D. Bethune
Boston University

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CHAPTER 1

Getting Started

1-1 INTRODUCTION

This chapter presents a step-by-step introduction to Inventor 10.0. When the program is first accessed, the New File dialog box will appear. See Figure 1-1. If the drawing screen does not look like Figure 1-1, click the New tool in the What To Do box.

There are seven options that will create drawings using four different types of files. The files are categorized using four different extensions. The extensions are defined as follows.

.ipt: part files for either 3D model drawings or sheet metal drawings. These files are for individual parts.

Figure 1-1

1

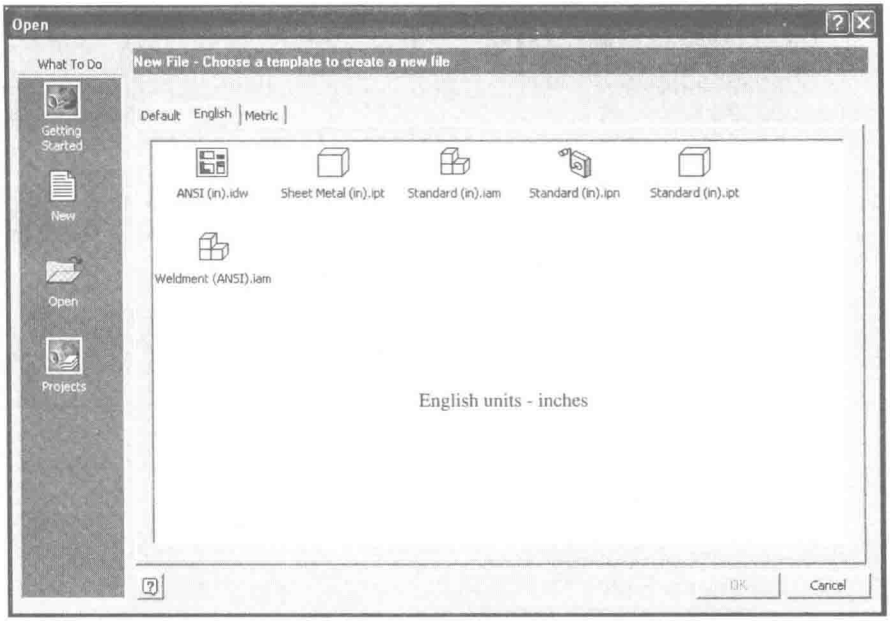


Figure 1-2

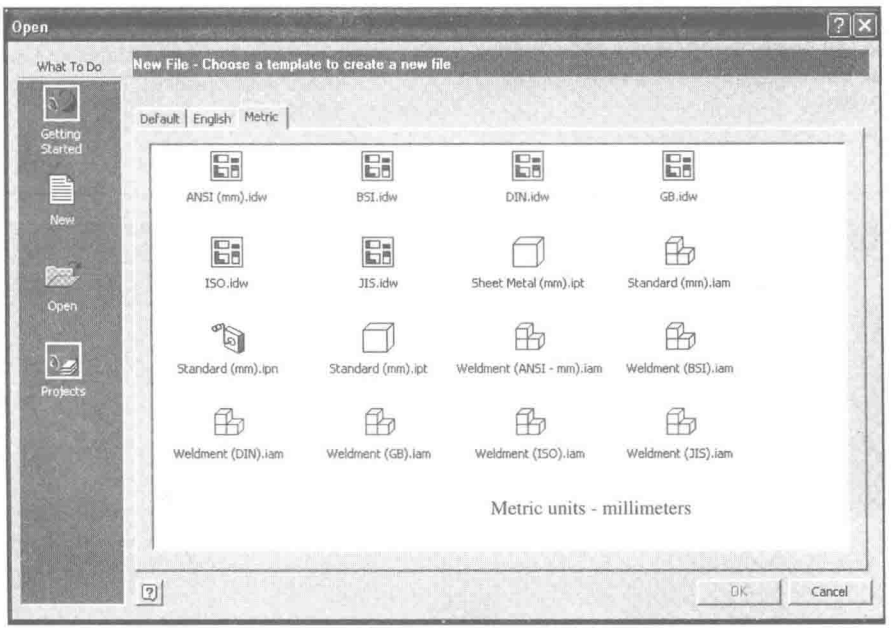


Figure 1-3

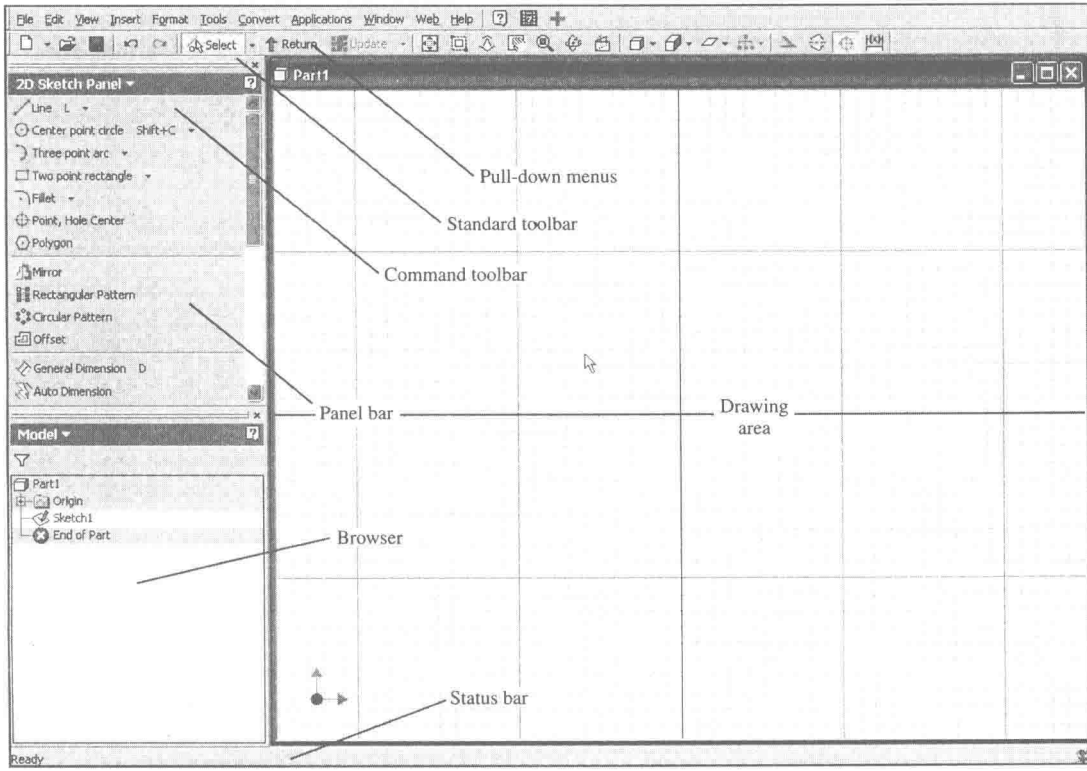


Figure 1-4

.iam: assembly drawings and weldments. Assembly drawings are formed by combining .ipt files.

.ipn: presentation files. These files are used to create exploded assembly drawings.

.idw: drawing layout files. These files are used to create orthographic views from already created assembly and presentation files.

Figure 1-2 shows the New File – English option. This option is used to create drawings in English units (inches) that conform to ANSI (American National Standards Institute) standards.

Figure 1-3 shows the New File – Metric option. This option is used to create metric drawings (millimeters).

1-2 CREATING A FIRST SKETCH

This section shows how to set up, create, and save a first drawing. The intent is to walk through a simple drawing in order to start to understand how inventor functions.

1. Select the Metric tab from the New File dialog box.
2. Select the Standard (mm) .ipt tool, then OK.

The drawing screen should change and look like the screen shown in Figure 1-4. The Inventor drawing screen includes a set of pull-down menus, the Standard toolbar, and the Command toolbar at the top of the screen. The browser area is at the lower left of the screen and contains a running list of how the drawing was created. The browser

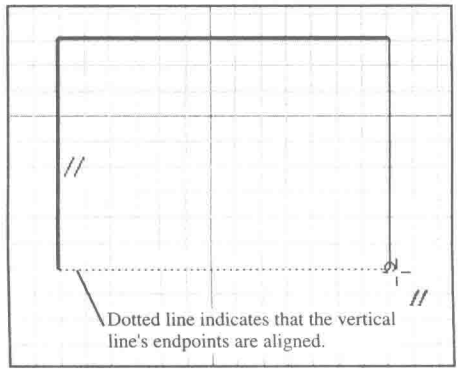


Figure 1-5

area information is used to edit models both during their creation and after they are created.

The panel bar contains command tools used to create drawings. The tool listing will change according to the operating mode selected.

To sketch a 30 × 40 rectangle

1. Select the Line tool on the 2D Sketch Panel bar.

Inventor does not use command line prompts, and there is no coordinate value input or axis reference. All work is done on the drawing screen. Each model generates its own set of reference values.

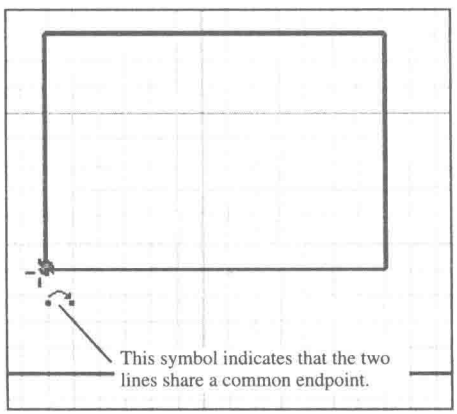


Figure 1-6

Lines are first sketched, that is, drawn without dimensions, and then modified to the required size.

2. Sketch a vertical line anywhere on the screen.

As the line is drawn, if it is vertical, a small symbol will appear next to the line indicating that the line is vertical.

3. Left-click the mouse and continue, sketching a horizontal line.

As the line is sketched a perpendicular symbol will appear if the horizontal line is perpendicular to the vertical line.

4. Left-click the mouse and continue, sketching a second vertical line.

As the second vertical line is sketched two parallel symbols will appear, one next to the line being sketched and the second next to the first vertical line, indicating that the lines are parallel.

When the endpoint of the second vertical line is aligned with the starting point of the first vertical line a broken line will appear. See Figure 1-5.

5. Sketch the second vertical line equal in length to the first vertical line.
6. Sketch a second horizontal line and locate its endpoint on the starting point of the first vertical line.

When the two points are aligned the cursor dot will change its color, and a small arclike symbol will appear. See Figure 1-6.

7. Right-click the mouse and select the Done option.

To delete lines

Lines and other objects may be deleted from a sketch.

1. Select the line to be deleted.
2. Right-click the mouse.

A dialog box will appear on the screen. See Figure 1-7.

3. Select the Delete option.

The line will disappear.

To undo a command

The Undo command will undo the last command entered.

1. Click on the Undo tool located at the top of the screen on the Standard toolbar.

The line will reappear.

To size the rectangle

1. Select the General Dimension tool from the 2D Sketch Panel bar.

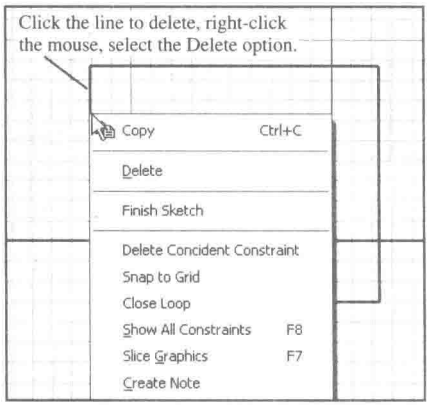


Figure 1-7

2. Select the left vertical line, then move the created dimension to the left of the object and click the left mouse button.

A small dialog box will appear containing the distance on the sketched line. See Figure 1-8.

3. Press the Delete key to remove the value, and type in 30, the required length.
4. Click on the check mark on the dialog box.

The line will change length.

5. Repeat the procedure for one of the horizontal lines, changing the sketched value to 40.
6. Right-click the mouse button and select the Done option.

Figure 1-9 shows the resulting 30 × 40 rectangle.

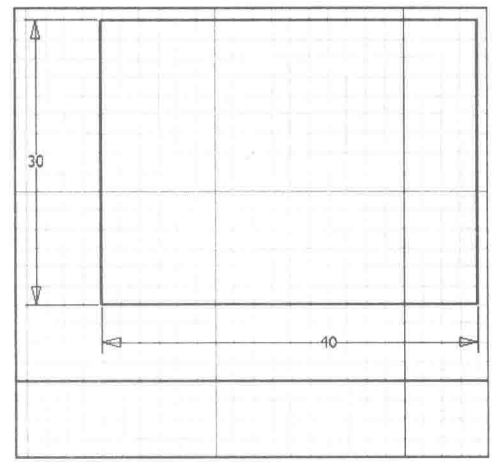


Figure 1-9

1-3 CREATING A SOLID MODEL

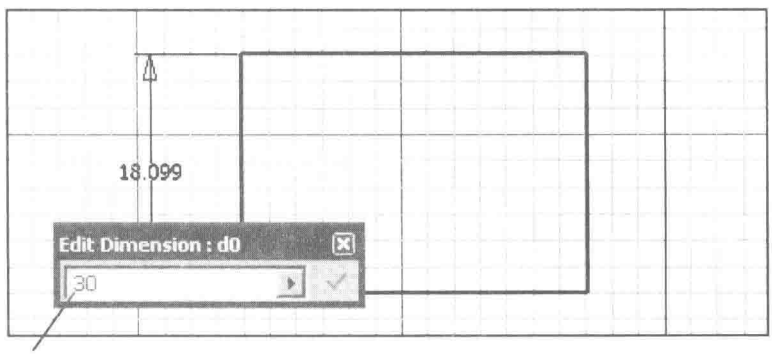
The 30 × 40 rectangle sketched in Section 1-2 will now be used to create a 3D solid model.

To change to an isometric view

1. Right-click the mouse and select the Isometric View option.

See Figure 1-10.

The screen will rotate into an isometric view orientation. Use the center mouse button to zoom the sketch to an acceptable size on the screen. See Figure 1-11.



Enter the desired 30 value. The vertical sides of the rectangle will change to 30 mm.

Figure 1-8

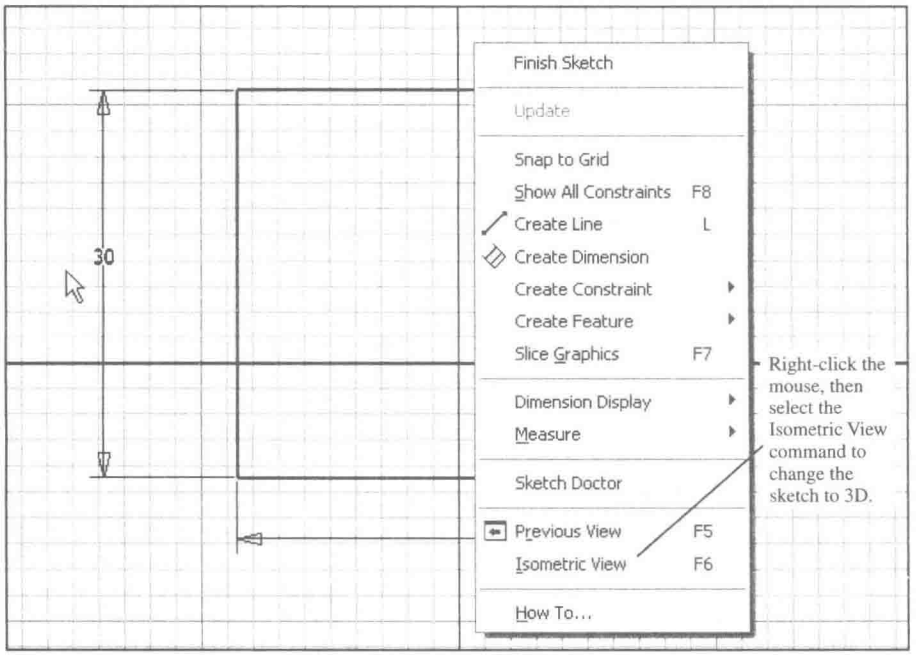


Figure 1-10

To create a solid model

1. Click the Return tool located on the Command toolbar or right-click the mouse and select the Finish Sketch option.

   The panel bar will change to a listing of Part Features tools. See Figure 1-12.

2. Select the Extrude tool from the Panel bar.

   The Extrude dialog box will appear. See Figure 1-13.

3. Change the Extents value to 15, then select OK.

   Figure 1-14 shows the results.

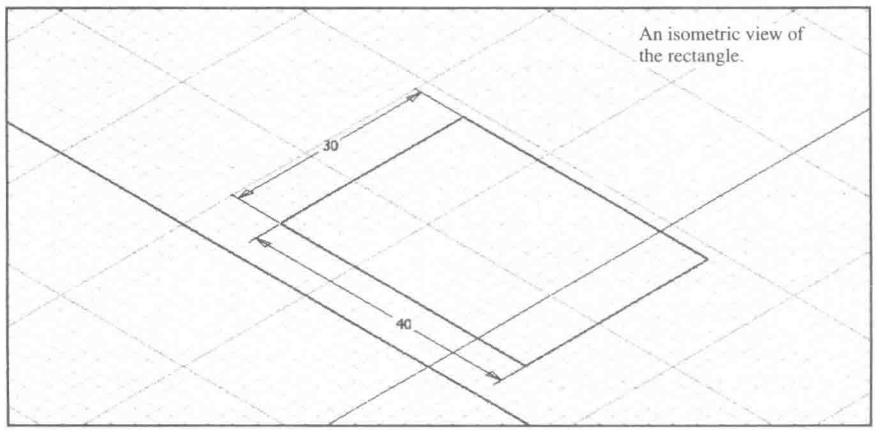


Figure 1-11