Preface

The purpose of writing this book is to produce a text suitable for those in Further and/or Higher Education who are required to learn how to use the CAD software package AutoCAD® 2007. Students taking examinations based on computer-aided design will find the contents of the book of great assistance. The book is also suitable for those in industry who wish to learn how to construct technical drawings with the aid of AutoCAD 2007 and those who, having used previous releases of AutoCAD, wish to update their skills in the use of AutoCAD.

The chapters dealing with two-dimensional (2D) drawing will also be suitable for those who wish to learn how to use AutoCAD LT 2007, the 2D version of this latest release of AutoCAD.

Many readers using AutoCAD 2002, 2004, 2005 or 2006 will find the book's contents largely suitable for use with those version of AutoCAD, although AutoCAD 2007 has enhancements over AutoCAD 2002, 2004, 2005 and 2006 (see Chapter 22).

The contents of the book are basically a graded course of work, consisting of chapters giving explanations and examples of methods of constructions, followed by exercises which allow the reader to practise what has been learned in each chapter. The first 12 chapters are concerned with constructing technical drawing in two dimensions (2D). These are followed by chapters detailing the construction of three-dimensional (3D) solid and surface model drawings and rendering. The two final chapters describe the Internet tools of AutoCAD 2007 and the place of AutoCAD in the design process. The book finishes with three appendices – printing and plotting; a list of tools with their abbreviations; and a list of some of the set variables upon which AutoCAD 2007 is based.

AutoCAD 2007 is a very complex computer-aided design (CAD) software package. A book of this size cannot possibly cover the complexities of all the methods for constructing 2D and 3D drawings available when working with AutoCAD 2007. However, it is hoped that by the time the reader has worked through the contents of the book, they will be sufficiently skilled with methods of producing drawing with the software, will be able to go on to more advanced constructions with its use and will have gained an interest in the more advanced possibilities available when using AutoCAD.

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Salisbury 2007

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CHAPTER 1

Introducing AutoCAD 2007

Aim of this chapter

The contents of this chapter are designed to introduce features of the AutoCAD 2007 window and methods of operating AutoCAD 2007.

Opening AutoCAD 2007

Fig. 1.1 The **AutoCAD 2007** shortcut icon on the Windows desktop

AutoCAD 2007 is designed to work in a Windows operating system. In general, to open AutoCAD 2007, either *double-click* on the **AutoCAD 2007** shortcut in the Windows desktop (Fig. 1.1), or *right-click* on the icon, followed by a *left-click* on **Open** in the menu which then appears (Fig. 1.2).

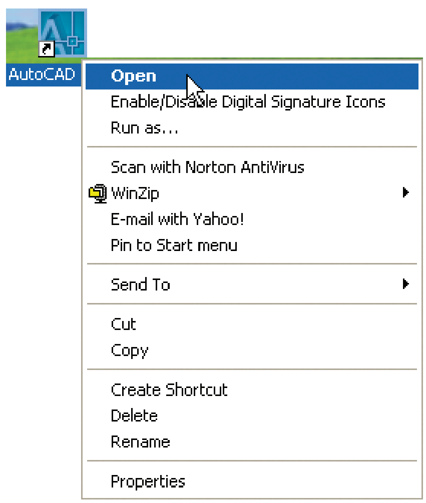


Fig. 1.2 The *right-click* menu which appears from the shortcut icon

When working in education or in industry, computers may be configured to allow other methods of opening AutoCAD, such as a list appearing on the computer in use when the computer is switched on, from which the operator can select the program they wish to use.

When AutoCAD 2007 is opened a window appears, depending upon whether a **Classic AutoCAD**, a **3D Modeling** or an **AutoCAD Default** workspace has been used previously. In this example the **Classic AutoCAD** workspace is shown and includes the drop-down menu from which a choice of the AutoCAD workspace to be opened can be made (Fig. 1.3). This **Classic AutoCAD** workspace shows:

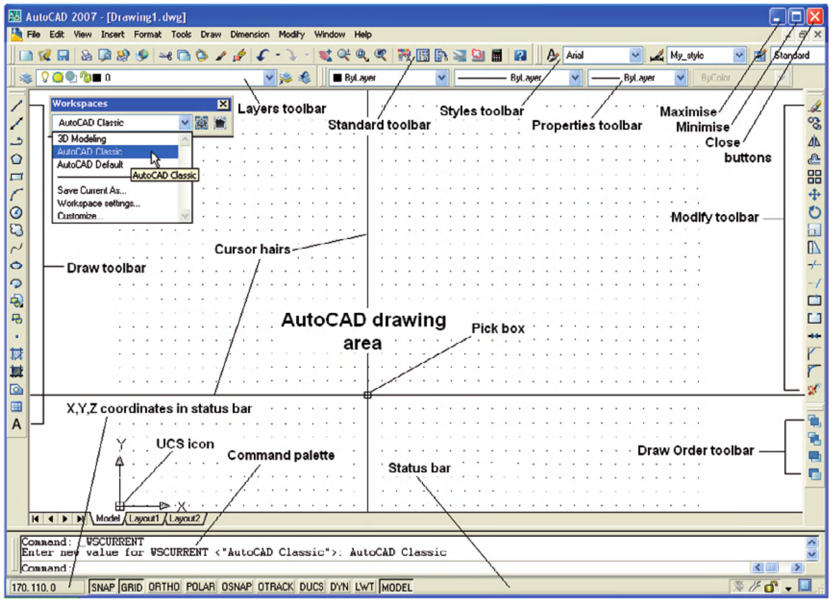


Fig. 1.3 The AutoCAD 2007 **Classic AutoCAD** workspace showing its various parts

Standard toolbar (Fig. 1.4) *docked* at the top of the workspace under the **Menu bar**.

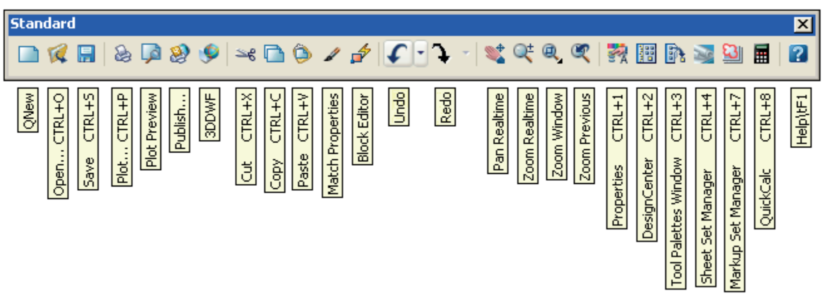


Fig. 1.4 The tools in the **Standard** toolbar

Workspaces toolbar (Fig. 1.5) usually within the workspace.
Styles toolbar *docked* to the right of the **Standard** toolbar.
Layers toolbar *docked* under the **Standard** toolbar.
Properties toolbar *docked* to the right of the **Layers** toolbar.

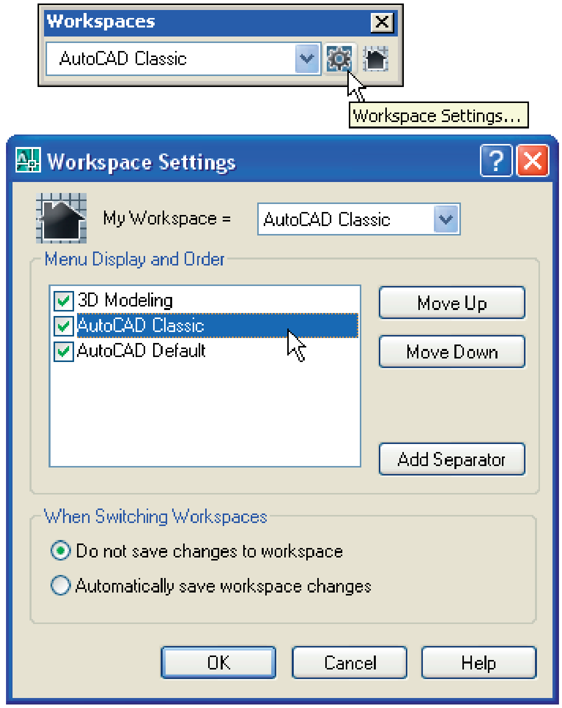


Fig. 1.5 The **Workspace Settings** dialog appearing when the **Workspace Settings . . .** icon of the **Workspaces** toolbar is *clicked*

Command palette can be *dragged* from its position at the bottom of the AutoCAD window into the AutoCAD drawing area, when it can be seen as a palette (Fig. 1.6). As with all palettes, an **AutoHide** icon and a *right-click* menu are included.

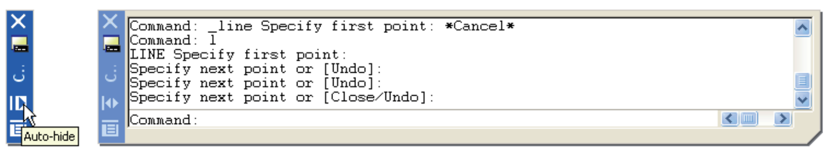


Fig. 1.6 The command palette when *dragged* from its position at the bottom of the AutoCAD window

Draw toolbar (Fig. 1.7) *docked* against the left-hand side of the workspace.

Modify toolbar (Fig. 1.7) *docked* against the right-hand side of the workspace.

Menu bar and menus: The **menu bar** is situated under the **title bar** and contains names of menus from which commands can be selected. Fig. 1.8 shows the **View** drop-down menu which appears with a *left-click* on the name. *Left-click* **3D Views** in the drop-down menu and a sub-menu appears, from which other sub-menus can be selected if required.

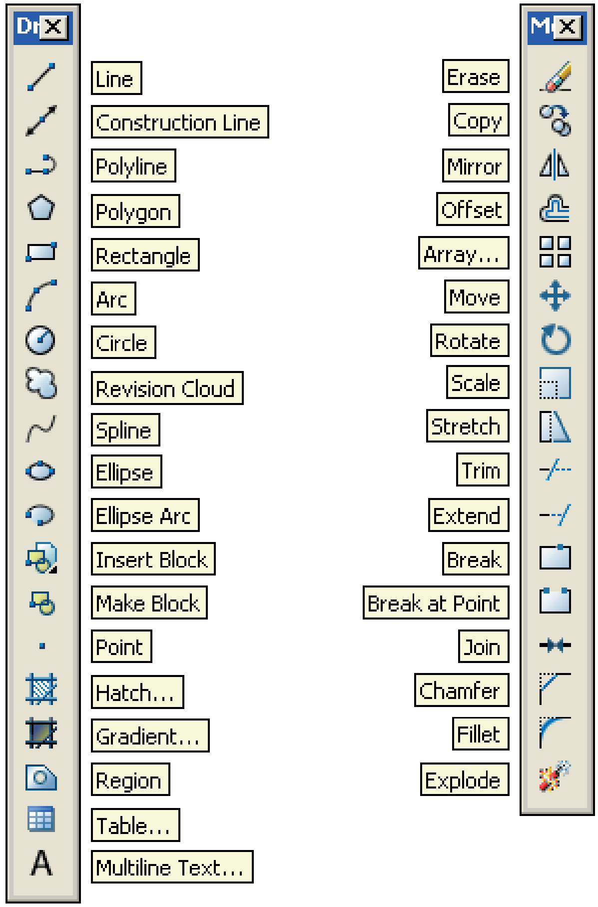


Fig. 1.7 The tools in the **Draw** and **Modify** toolbars

The mouse as a digitiser

Many operators working in AutoCAD will use a two-button mouse as the digitiser. There are other digitisers which may be used – pucks with tablets, a three-button mouse etc. Fig. 1.9 shows a mouse which has two buttons and a wheel.

To operate this mouse pressing the **Pick** button is a *left-click*. Pressing the **Return** button is a *right-click*. Pressing the **Return** button usually has the same result as pressing the **Enter** key of the keyboard.

When the **wheel** is pressed drawings in the AutoCAD screen can be panned. Moving the wheel forward enlarges (zooms in) the drawing on screen. Moving the wheel backwards reduces the size of a drawing.

The pick box at the intersection of the cursor hairs moves with the cursor hairs in response to movements of the mouse. The AutoCAD window as shown in Fig. 1.3 includes cursor hairs which stretch across the drawing in both horizontal and vertical directions. Some operators prefer cursors hairs

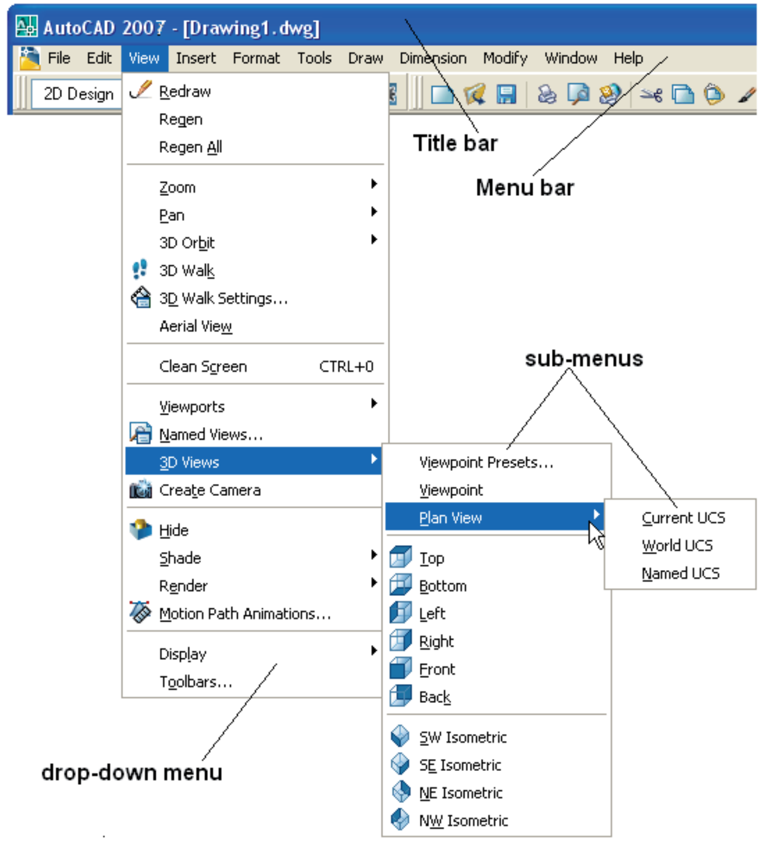


Fig. 1.8 Menus and sub-menus

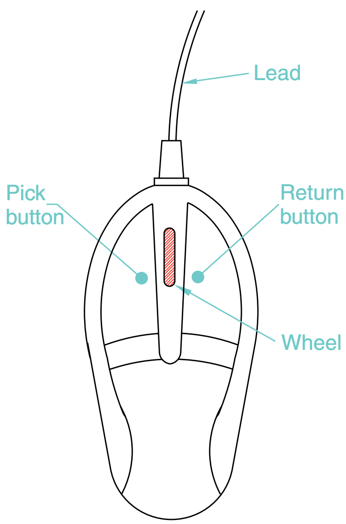


Fig. 1.9 A two-button mouse

to be shorter. The length of the cursor hairs can be adjusted in the **Options** dialog (page 10).

Palettes

A palette has already been shown – the **Command** palette. Two palettes which may be frequently used are the **DesignCenter** palette and the **Properties** palette. These can be called to screen from the **Tools** drop-down menu (Fig. 1.10).

DesignCenter palette: Fig. 1.11 shows the palette showing the **Block** drawings of metric fasteners from an AutoCAD directory **DesignCenter** from which the drawing file **Fasteners - Metric.dwg** has been selected. A fastener block drawing can be *dragged* from the **DesignCenter** for inclusion in a drawing under construction.

Properties palette: Fig. 1.12 shows the **Properties** palette in which the general and geometrical features of a selected polyline are shown. The polyline can be changed by the *entering* of new figures in the appropriate parts of the palette.

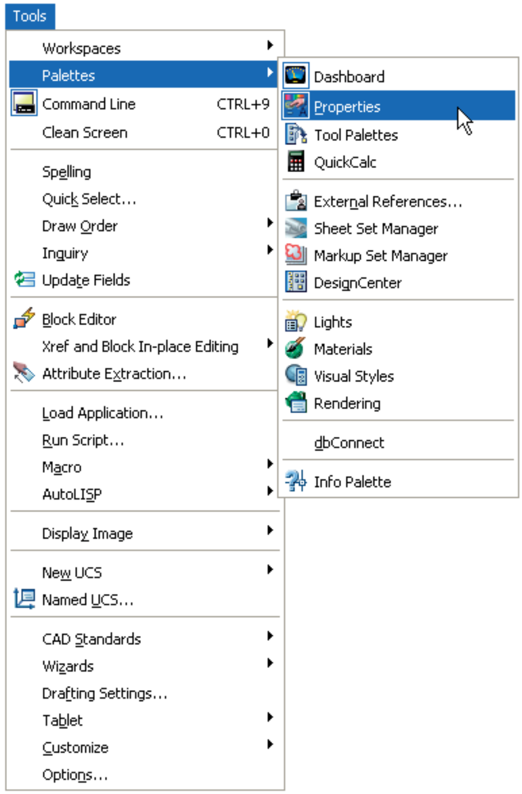


Fig. 1.10 Palettes can be called to screen from the **Palettes** sub-menu of the **Tools** drop-down menu

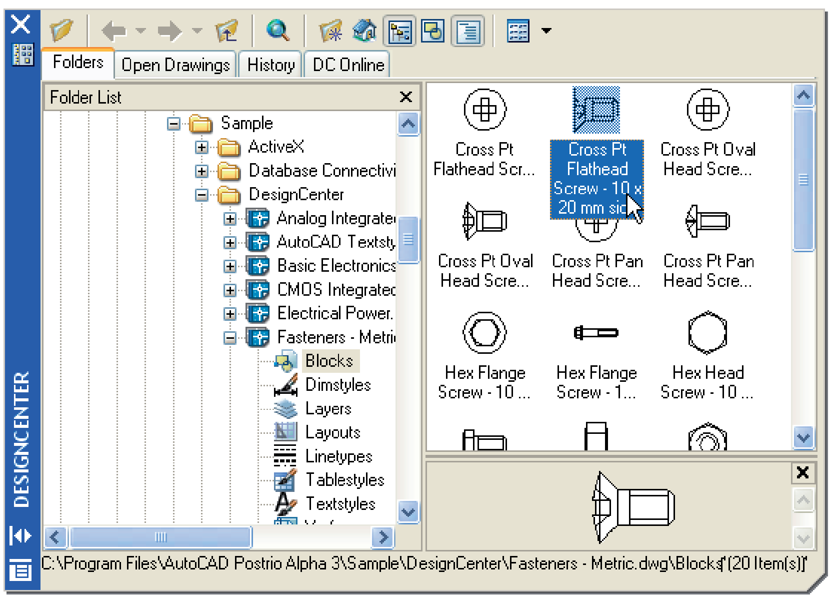


Fig. 1.11 The **DesignCenter** palette

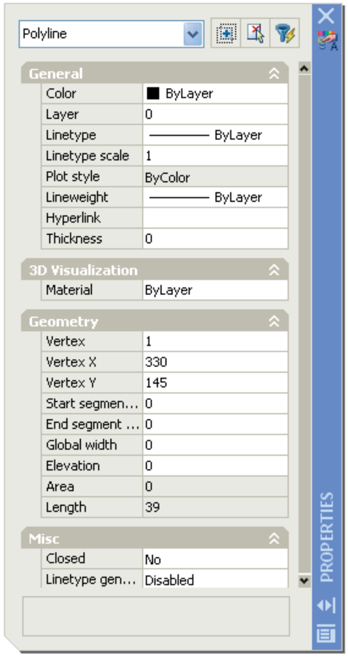


Fig. 1.12 The **Properties** palette

The DASHBOARD palette

Click on **Tools** in the menu bar and from the drop-down menu which appears *click* **Dashboard**. The **DASHBOARD** palette appears (Fig. 1.13). *Right-click* in the title bar of the palette and a popup menu appears. *Click* on **Control panels** and *click* against all the names which appear in the sub-menu except **2D Draw control panel**. Parts of the **DASHBOARD** disappear leaving only the **2D Draw control panel** dashboard. This can be reduced in size by *dragging* at corners or edges, or hidden by *clicking* on the **Auto-hide** icon, or moved by *dragging* on the **Move** icon. This panel holds all the tool icons contained in the **Draw** and **Modify** toolbars. The other panels in the **DASHBOARD** are for 3D modelling. These details are shown in Fig. 1.13.

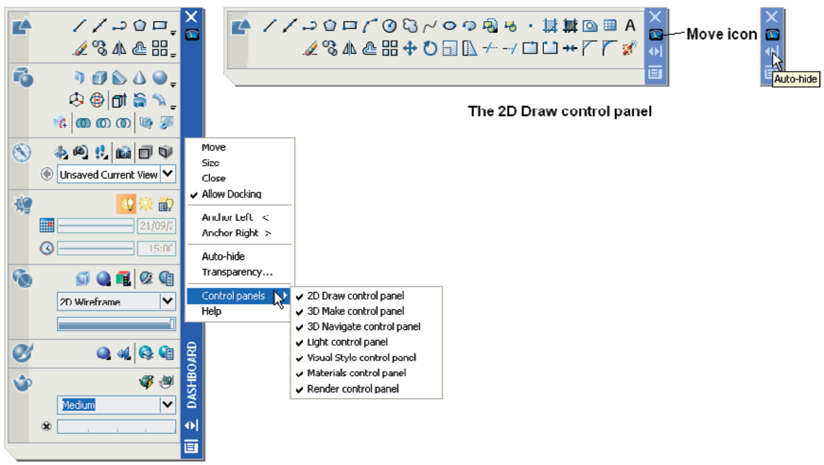


Fig. 1.13 The **DASHBOARD** palette

Dialogs

Dialogs are an important feature of AutoCAD 2007. Settings can be made in many of the dialogs, files can be saved and opened and changes can be made to variables.

Examples of the parts of dialogs are shown in Figs 1.14 and 1.15. The first example is taken from the **Select File** dialog (Fig. 1.14), opened with a *click* on **Open** . . . in the **File** drop-down menu (Fig. 1.16). The second example shows part of the **Options** dialog (Fig. 1.15) in which many settings can be made to allow operators the choice of their methods of constructing drawings. The **Options** dialog can be opened with a *click* on **Options** . . . in the *right-click* menu opened in the command window (Fig. 1.17).

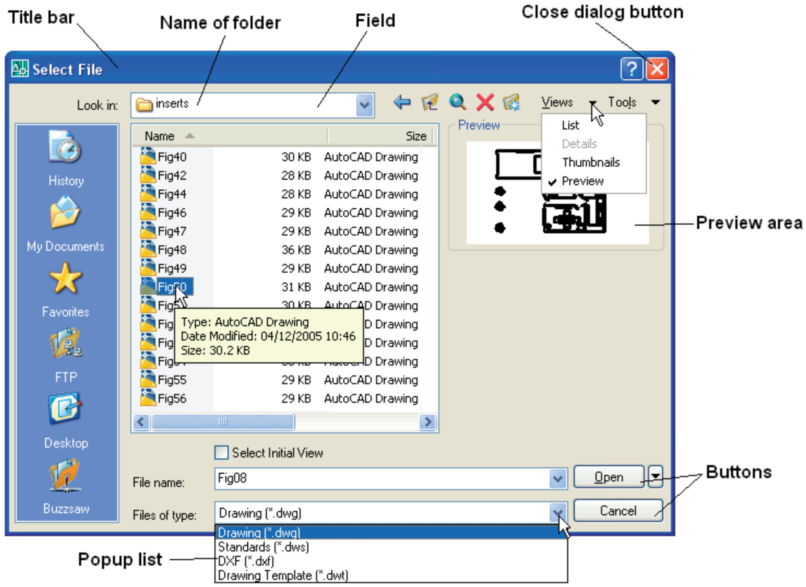


Fig. 1.14 The **Select File** dialog

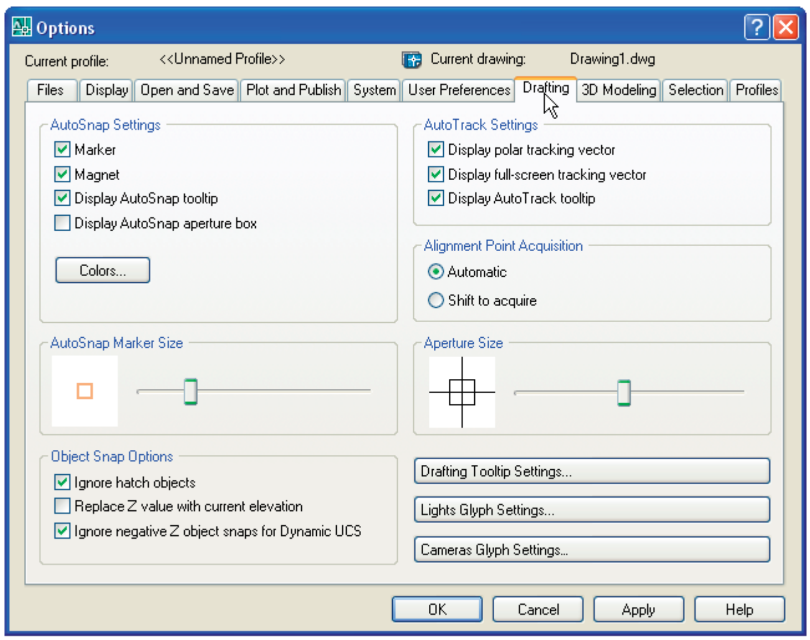


Fig. 1.15 Part of the **Options** dialog

Note the following parts in the dialog many of which are common to other AutoCAD dialogs:

Title bar: showing the name of the dialog.
Close dialog button: common to other dialogs.

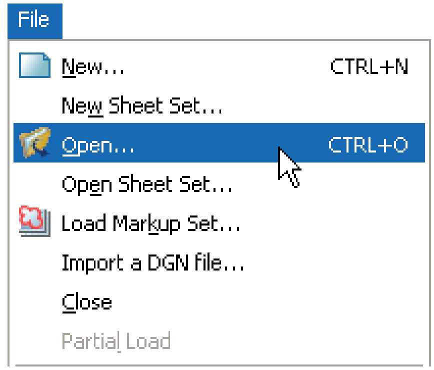


Fig. 1.16 Opening the **Select File** dialog from the **File** drop-down menu

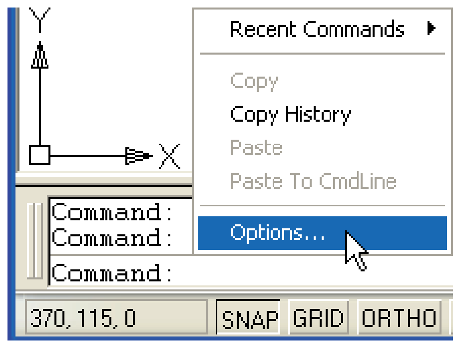


Fig. 1.17 The *right-click* menu in the command palette

Popup list: a *left-click* on the arrow to the right of the field brings down a popup list listing selections available in the dialog.

Buttons: a *click* on the **Open** button brings the selected drawing on screen. A *click* on the **Cancel** button closes the dialog.

Preview area: available in some dialogs – shows a miniature of the selected drawing or other feature, only part of which is shown in Fig. 1.15.

Note the following in the **Options** dialog:

Tabs: a *click* on any of the tabs in the dialog brings a sub-dialog on screen.

Check boxes: a tick appearing in a check box indicates the function described against the box is on. No tick indicates the function is off. A *click* in a check box toggles between the feature being off or on.

Radio buttons: a black dot in a radio button indicates the feature described is on. No dot and the feature is off.

Slider: a slider pointer can be *dragged* to change sizes of the feature controlled by the slider.

Buttons in the status bar

A number of buttons in the status bar can be used for toggling (turning on/off) various functions when operating within AutoCAD 2007 (Fig. 1.18). A *click* on a button turns that function on, if it is off, a *click* on a button when it is off turns the function back on. Similar results can be obtained by using function keys of the computer keyboard (keys **F1** to **F10**).

Fig. 1.18 The buttons in the status bar

SNAP: also toggled using the **F9** key. When set on, the cursor under mouse control can only be moved in jumps from one snap point to another. See also page 15.

GRID: also toggled using the **F7** key. When set on, a series of grid points appears in the drawing area. See also page 15.

ORTHO: also toggled using the **F8** key. When set on, lines etc. can only be drawn vertically or horizontally.

POLAR: also toggled using the **F10** key. When set on, a small tip appears showing the direction and length of lines etc. in degrees and units.

OSNAP: also toggled using the **F3** key. When set on, an osnap icon appears at the cursor pick box. See also page 41.

OTRACK: when set on, lines etc. can be drawn at exact coordinate points and precise angles.

DUCS: Dynamic UCS. Also toggled by the **F6** key. Used when constructing 3D solid models.

DYN: Dynamic Input. When set on, the **x,y** coordinates and prompts show when the cursor hairs are moved.

LWT: when set on, lineweights show on screen. When set off, lineweights only show in plotted/printed drawings.

When in **Paper Space** a button can toggle **Model Space** and **Paper Space** and a new button appears for toggling between **Maximizing** and **Minimizing** the workspace.

Note the square light-blue button at the right-hand end of the status bar – the **Clean Screen** button. *Left-click* this button and a screen clear of all but the menu bar and the command palette appears. When in the **Clean Screen** workspace another *click* on the button and the screen reverts to its original state.

Note

When constructing drawings in AutoCAD 2007 it is advisable to toggle between **Snap, Ortho, Osnap** and the other functions in order to make constructing easier.

The AutoCAD coordinate system

In the AutoCAD 2D coordinate system, units are measured horizontally in terms of X and vertically in terms of Y. A 2D point can be determined in terms of X,Y (in this book referred to as *x,y*). The coordinate point *x,y* = 0,0 is the **origin** of the system. The coordinate point *x,y* = 100,50 is 100 units to the right of the origin and 50 units above the origin. The point *x,y* = −100,−50 is 100 units to the left of the origin and 50 points below the origin. Fig. 1.19 shows some 2D coordinate points in the AutoCAD window.

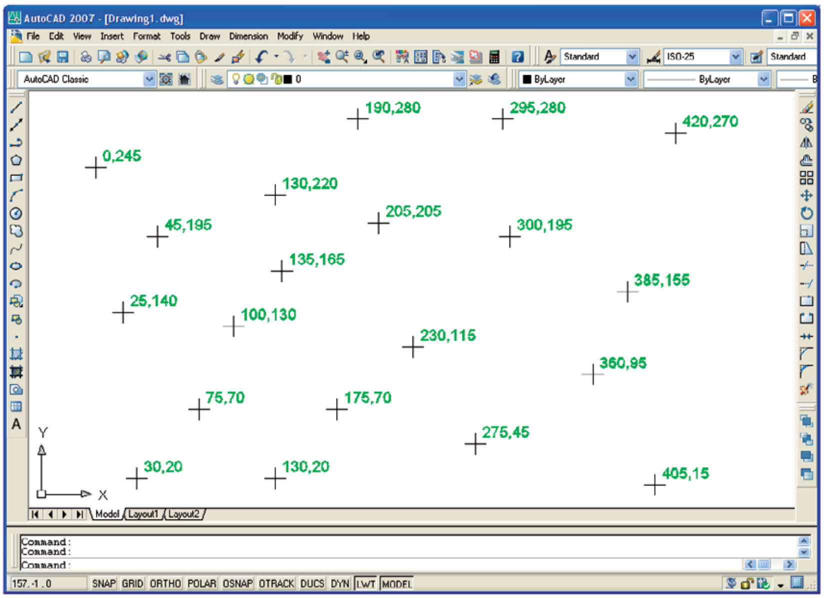


Fig. 1.19 The 2D coordinate points in the AutoCAD coordinate system

3D coordinates include a third coordinate (Z), in which positive Z units are towards the operator as if coming out of the monitor screen and negative Z units going away from the operator as if towards the interior of the screen. 3D coordinates are stated in terms of *x,y,z*. The coordinate point *x,y,z* = 100,50,50 is 100 units to the right of the origin, 50 units above the origin and 50 units towards the operator. A 3D model drawing as if resting on the surface of a monitor is shown in Fig. 1.20.

Drawing templates

Drawing templates are files with an extension **.dwt**. Templates are files which have been saved with predetermined settings – such as **Grid** spacing, **Snap**

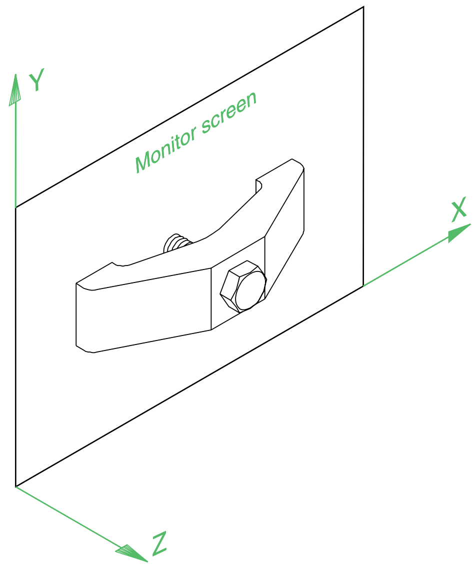


Fig. 1.20 A 3D model drawing showing the X,Y and Z coordinate directions

spacing etc. Templates can be opened from the **Select template** dialog (see Fig. 1.21) called by *clicking* **New** . . . in the **File** drop-down menu. An example of a template file being opened is shown in Fig. 1.21. In this example the template will be opened in Paper Space and is complete with a title block and borders.

When AutoCAD 2007 is used in European countries, the **acadiso.dwt** template automatically appears on screen. Throughout this book drawings

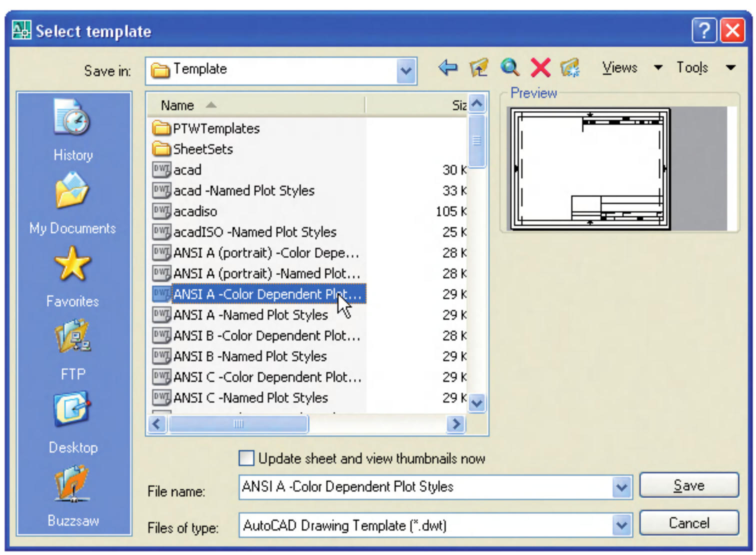


Fig. 1.21 A template selected for opening in the **Select template** dialog

will usually be constructed in an adaptation of the **acadiso.dwt** template. To adapt this template:

1. In the command palette *enter* (type) **grid** followed by a *right-click* (or pressing the **Enter** key). Then *enter* **10** in response to the prompt which appears, followed by a *right-click* (Fig. 1.22).

```
Command: grid
Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Follow/Aspect] <10>:
<Object Snap Tracking off>
Command:
```

Fig. 1.22 Setting **Grid** to **10**

2. In the command palette *enter* **snap** followed by *right-click*. Then *enter* **5** followed by a *right-click* (Fig. 1.23).

```
<Object Snap Tracking off>
Command: snap
Specify snap spacing or [ON/OFF/Aspect/Style/Type] <5>:
Command:
```

Fig. 1.23 Setting **Snap** to **5**

3. In the command palette *enter* **limits**, followed by a *right-click*. *Right-click* again. Then *enter* **420,297** and *right-click* (Fig. 1.24).

```
Command: limits
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0,0>:
Specify upper right corner <420,297>:
Command:
```

Fig. 1.24 Setting **Limits** to **420,297**

4. In the command window *enter* **zoom** and *right-click*. Then in response to the line of prompts which appears *enter* **a** (for All) and *right-click* (Fig. 1.25).

```
Command: zoom
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: a
Regenerating model.
Command:
```

Fig. 1.25 **Zooming** to **All**

5. In the command palette *enter* **units** and *right-click*. The **Drawing Units** dialog appears (Fig. 1.26). In the **Precision** popup list of the **Length** area of the dialog, *click* on **0** and then *click* the **OK** button. Note the change in the coordinate units showing in the status bar.

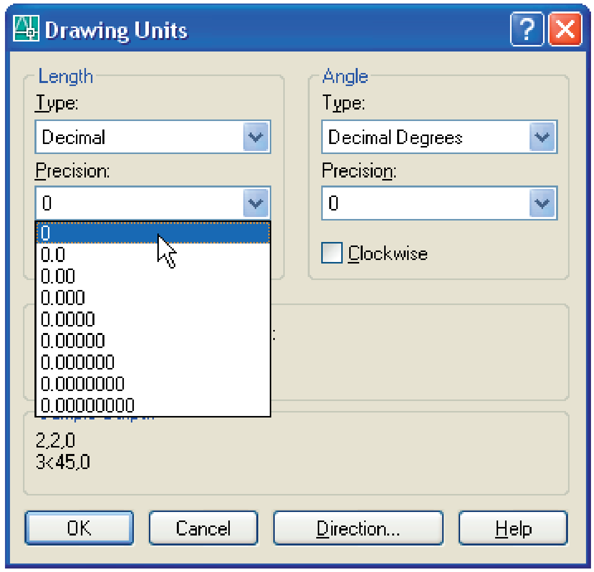


Fig. 1.26 Setting **Units** to **0**

6. *Click* **File** in the menu bar and *click* **Save As** . . . in the drop-down menu which appears. The **Save Drawing As** dialog appears. In the **Files of type** popup list select **AutoCAD Drawing Template (*.dwt)**. The templates already in AutoCAD are displayed in the dialog. *Click* on **acadiso.dwt**, followed by another *click* on the **Save** button.

Notes

1. Now when AutoCAD is opened the template saved as **acadiso.dwt** automatically loads with **Grid** set to **10, Snap** set to **5, Limits** set to **420,297** (size of an A3 sheet in millimetres) and with the drawing area zoomed to these limits, with **Units** set to **0**.
2. However if there are multiple users to the computer, it is advisable to save your template to another file name – I have used **my_template.dwt**.
3. Other features will be added to the template in future chapters.

Method of showing entries in the command palette

Throughout the book, where necessary, details *entered* in the command palette will be shown as follows:

At the command line:

Command: *enter* **zoom** *right-click*
Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: *enter* **a** (All) *right-click*
Regenerating model.
Command:

Note

In later examples this may be shortened to**:**

Command: *enter* **z** *right-click*
[prompts]: *enter* **a** *right-click*
Command:

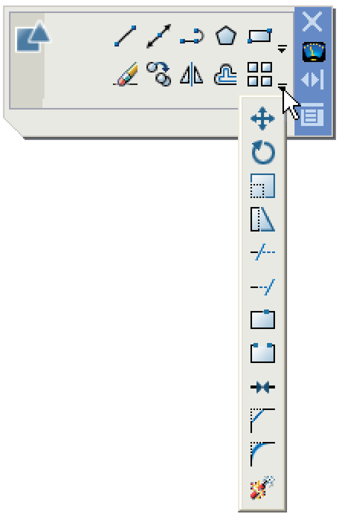


Fig. 1.27 Tool icons and a flyout in the **DASHBOARD** palette

Notes

1. In the above *enter* means type the given letter, word or words at the **Command:** prompt.
2. *Right-click* means press the **Return** (right) button of the mouse or press the **Return** key of the keyboard.

Tools and tool icons

An important feature of Windows applications are icons and tooltips. In AutoCAD 2007, tools are shown as icons in toolbars or in the **DASHBOARD** palette. When the cursor is placed over a tool icon a tooltip shows with the name of the tool as shown in the tooltips in the **Draw** and **Modify** toolbars (Fig. 1.7).

If a small arrow is included at the bottom right-hand corner of a tool icon, when the cursor is placed over the icon and the *pick* button of the mouse depressed and held, a flyout appears which includes other tool icons (Fig. 1.27). The example given in this illustration shows a flyout from the **2D Draw control panel**.

Another AutoCAD workspace

Click the **My Workspace** icon in the **Workspaces** toolbar (Fig. 1.28). *Click* on the **Clear screen** icon at the bottom right-hand corner of the AutoCAD window. All toolbars and palettes disappear from the screen except for the command window leaving a larger workspace. Open the

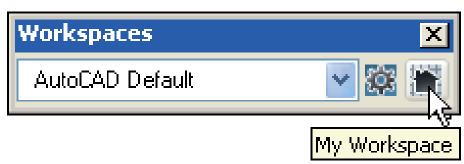


Fig. 1.28 Selecting **My Workspace** from the **Workspaces** toolbar

DASHBOARD with a *click* on its name in the **Palettes** sub-menu of the **Tools** drop-down menu and change it into the **2D Draw control panel**. A new screen appears with an enlarged working area (Fig. 1.29). Tools can be selected from the **2D Draw control panel** for the construction of drawings within this enlarged area.

Other workspaces can be designed as the operator wishes. This particular workspace has the advantage that it uses all the area available on the computer screen being used at the time. At any time another *click* on the **Clear Screen** icon causes the screen to revert to the **AutoCAD Classic**

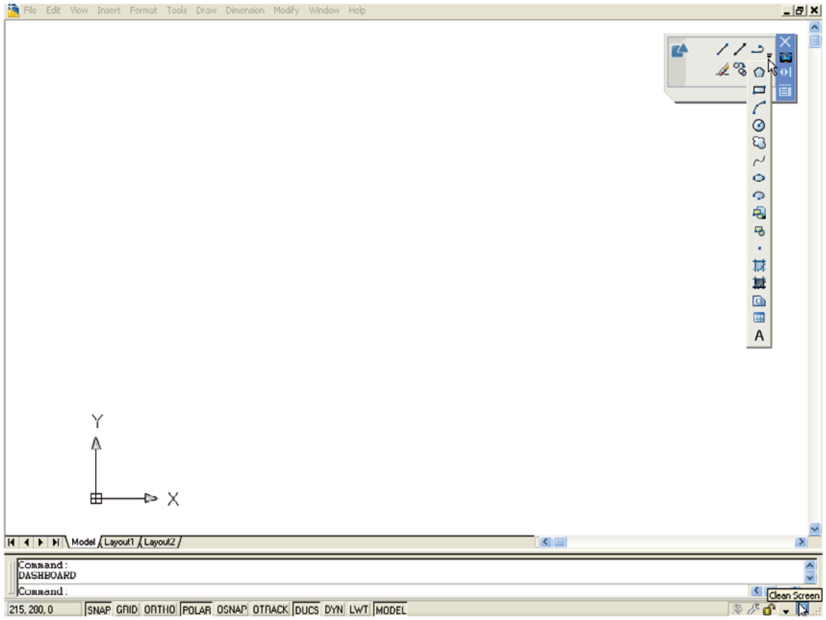


Fig. 1.29 **My workspace**

screen. When this happens it may be desirable to delete the **2D Draw control panel** with a *click* on its **Close** button.

Revision notes

1. A *double-click* on the **AutoCAD 2007** shortcut in the Windows desktop opens the AutoCAD window.
2. Or *right-click* on the shortcut, followed by a *left-click* on **Open** in the menu which then appears.
3. There are three main workspaces in which drawings can be constructed – the **Classic AutoCAD** workspace, the **3D Modeling** workspace and **My Workspace** workspace. From now on this part of the book (**Part I**) which deals with the construction of 2D drawings will show examples constructed mainly in the **Classic AutoCAD** screen, which opens showing the **Draw** and **Modify** toolbars.
4. A *left-click* on a menu name in the menu bar brings a drop-down menu on screen. In drop-down menus:
   (a) A small outward pointing arrow against a name means that a sub-menu will appear with a click on the name.
   (b) Three dots (. . .) following a name means that a *click* on the name will bring a dialog on screen.
5. All constructions in this book involve the use of a mouse as the digitiser. When a mouse is the digitiser:
   (a) A *left-click* means pressing the left-hand button (the **Pick**) button.
   (b) A *right-click* means pressing the right-hand button (the **Return**) button.

(c) A *double-click* means pressing the left-hand button twice in quick succession.
(d) *Dragging* means moving the mouse until the cursor is over an item on screen, holding the left-hand button down and moving the mouse. The item moves in sympathy with the mouse movement.
(e) To *pick* has a similar meaning to a *left-click*.

6. Palettes are a particular feature of AutoCAD 2007. The **Command** palette, the **DesignCenter** palette and the **Properties** palette will be in frequent use.
7. Tools are shown as icons in the toolbars and panels.
8. When a tool is *picked* a tooltip describing the tool appears.
9. Dialogs allow opening and saving of files and the setting of parameters.
10. A number of *right-click* menus are used in AutoCAD 2007.
11. A number of buttons in the status bar can be used to toggle features such as snap and grid. Functions keys of the keyboard can be also used for toggling most of these functions.
12. The AutoCAD coordinate system determines the position in units of any point in the drawing area (**Classic AutoCAD**) and any point in 3D space (**3D Modeling**).
13. Drawings are usually constructed in templates with predetermined settings. Some templates include borders and title blocks.

CHAPTER 2

Introducing drawing

Aims of this chapter

The contents of this chapter are designed to introduce:

1. The construction of 2D drawing in the **Classic AutoCAD** workspace.
2. The drawing of simple outlines using the **Line, Circle** and **Polyline** tools from the **Draw** toolbar or the **2D Draw control panel**.
3. Drawing to snap points.
4. Drawing to absolute coordinate points.
5. Drawing to relative coordinate points.
6. Drawing using the 'tracking' method.
7. The use of the **Erase, Undo** and **Redo** tools.

The 2D Classic AutoCAD workspace

Illustrations throughout this chapter will be shown using the **AutoCAD Classic** workspace. However the methods of construction will be the same if the reader wishes to work in other workspaces. If the **2D Draw control panel** is on screen, tools can be selected from the panel. In this chapter illustrations will show tools mainly selected from the **Draw** toolbar, but a few will show tools selected from the **2D Draw control panel**. Whether working with the **Draw** toolbar or the **2D Draw control panel**, the sequences and prompts which appear at the command line will be the same.

Drawing with the Line tool

First example – Line tool (Fig. 2.3)

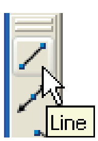


Fig. 2.1 The **Line** tool from the **Draw** toolbar

1. Open AutoCAD. The drawing area will show the settings of the **acadiso.dwt** template – **Limits** set to **420,297, Grid** set to **10, Snap** set to **5** and **Units** set to **0**.
2. *Left-click* on the **Line** tool in the **Draw** toolbar (Fig. 2.1).

Note

(a) The tooltip which appears when the tool icon is *clicked*.
(b) The prompt **Command:_line Specify first point:** which appears in the command window at the command line (Fig. 2.2).

3. Make sure **Snap** is on by either pressing the **F9** key or the **SNAP** button in the status bar. **<Snap on>** will show in the command window.

Fig. 2.2 The prompt appearing at the command line in the command palette when **Line** is 'called'

4. Move the mouse around the drawing area. The cursors' pick box will jump from point to point at 5 unit intervals. The position of the pick box will show as coordinate numbers in the status bar (left-hand end).
5. Move the mouse until the coordinate numbers show **60,240,0** and press the **Pick** button of the mouse (*left-click*).
6. Move the mouse until the coordinate numbers show **260,240,0** and *left-click*.
7. Move the mouse until the coordinate numbers show **260,110,0** and *left-click*.
8. Move the mouse until the coordinate numbers show **60,110,0** and *left-click*.
9. Move the mouse until the coordinate numbers show **60,240,0** and *left-click*. Then press the **Return** button of the mouse (*right-click*).

Fig. 2.3 appears in the drawing area.

Fig. 2.3 First example – **Line** tool

Second example – Line tool (Fig. 2.6)

1. Clear the drawing from the screen with a *click* on the **Close** drawing button (Fig. 2.4). Make sure it is not the AutoCAD 2007 window button.
2. The warning window Fig. 2.5 appears in the centre of the screen. *Click* its **No** button.

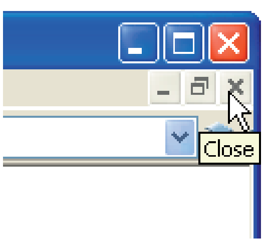


Fig. 2.4 The **Close** drawing button

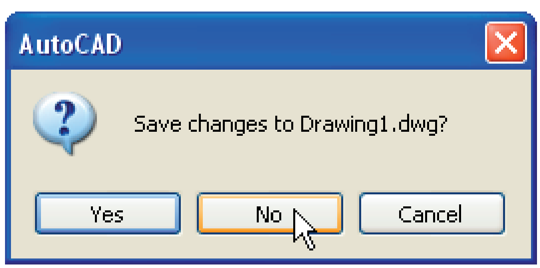


Fig. 2.5 The **AutoCAD** warning window

3. *Left-click* on **New** . . . in the **File** drop-down menu and from the **Select template** dialog which appears *double-click* on **acadiso.dwt**.
4. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: *enter* **80,235** *right-click*
Specify next point or [Undo]: *enter* **275,235** *right-click*
Specify next point or [Undo]: *enter* **295,210** *right-click*
Specify next point or [Close/Undo]: *enter* **295,100** *right-click*
Specify next point or [Close/Undo]: *enter* **230,100** *right-click*
Specify next point or [Close/Undo]: *enter* **230,70** *right-click*
Specify next point or [Close/Undo]: *enter* **120,70** *right-click*
Specify next point or [Close/Undo]: *enter* **120,100** *right-click*
Specify next point or [Close/Undo]: *enter* **55,100** *right-click*
Specify next point or [Close/Undo]: *enter* **55,210** *right-click*
Specify next point or [Close/Undo]: *enter* **c** (Close) *right-click*
Command:

The result is as shown in Fig. 2.6.

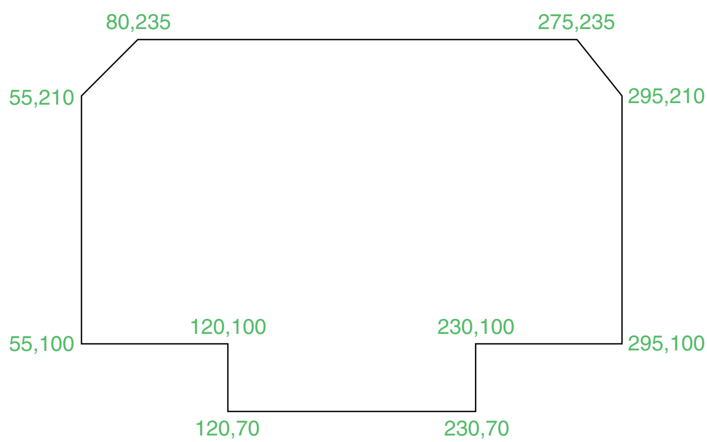


Fig. 2.6 Second example – **Line** tool

Third example – Line tool (Fig. 2.7)

1. Close the drawing and open a new **acadiso.dwt** window.
2. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: *enter* **60,210** *right-click*
Specify next point or [Undo]: *enter* **@50,0** *right-click*
Specify next point or [Undo]: *enter* **@0,20** *right-click*
Specify next point or [Close/Undo]: *enter* **@130,0** *right-click*
Specify next point or [Close/Undo]: *enter* **@0,−20** *right-click*
Specify next point or [Close/Undo]: *enter* **@50,0** *right-click*
Specify next point or [Close/Undo]: *enter* **@0,−105** *right-click*

Specify next point or [Close/Undo]: *enter* **@−50,0** *right-click*
Specify next point or [Close/Undo]: *enter* **@0,−20** *right-click*
Specify next point or [Close/Undo]: *enter* **@−130,0** *right-click*
Specify next point or [Close/Undo]: *enter* **@0,20** *right-click*
Specify next point or [Close/Undo]: *enter* **@−50,0** *right-click*
Specify next point or [Close/Undo]: *enter* **c** (Close) *right-click*
Command:

The result is as shown in Fig. 2.7.

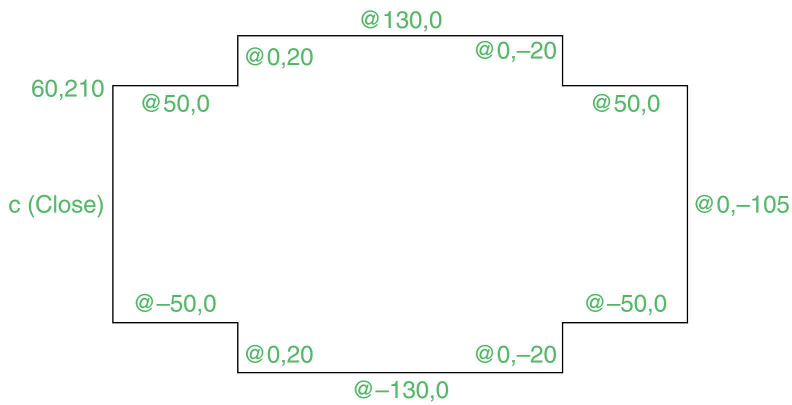


Fig. 2.7 Third example – **Line** tool

Notes

1. The figures typed at the keyboard determining the corners of the outlines in the above examples are two-dimensional (2D) **x,y** coordinate points. When working in 2D, coordinates are expressed in terms of two numbers separated by a comma.
2. Coordinate points can be shown as positive or negative numbers.
3. The method of constructing an outline as shown in the first two examples is known as the **absolute coordinate entry** method, where the **x,y** coordinates of each corner of the outlines are *entered* at the command line as required.
4. The method of constructing an outline as in the third example is known as the **relative coordinate entry** method – coordinate points are *entered* relative to the previous entry. In relative coordinate entry, the @ symbol is *entered* before each set of coordinates with the following rules in mind:

   +**ve** x entry is to the right
   −**ve** x entry is to the left
   +**ve** y entry is upwards
   −**ve** y entry is downwards.

5. The next example (the fourth) shows how lines at angles can be drawn taking advantage of the relative coordinate entry method. Angles in AutoCAD are measured in 360 degrees in a counter-clockwise (anticlockwise) direction (Fig. 2.8). The < symbol precedes the angle.

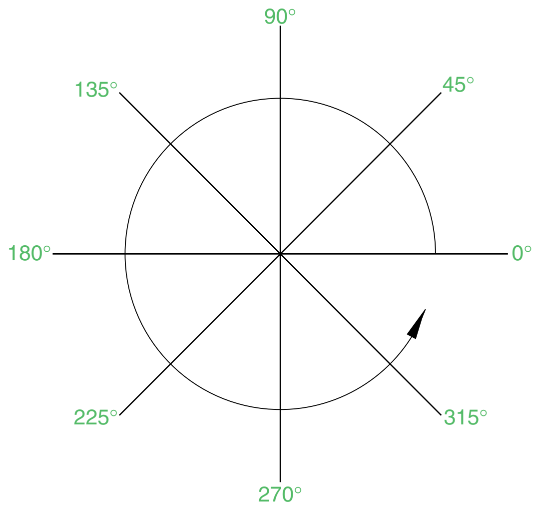


Fig. 2.8 The counter-clockwise direction of measuring angles in AutoCAD

Fourth example – Line tool (Fig. 2.9)

1. Close the drawing and open a new **acadiso.dwt** window.
2. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

Command:_line Specify first point: 70,230
Specify next point: @220,0
Specify next point: @0,−70
Specify next point or [Undo]: @115<225
Specify next point or [Undo]: @−60,0
Specify next point or [Close/Undo]: @115<135
Specify next point or [Close/Undo]: @0,70
Specify next point or [Close/Undo]: c (Close)
Command:

The result is as shown in Fig. 2.9

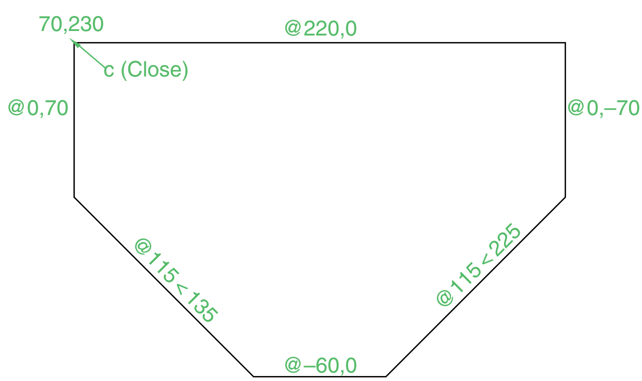


Fig. 2.9 Fourth example – **Line** tool

Fifth example – Line tool (Fig. 2.10)

Another method of constructing accurate drawings is by using a method known as **tracking**. When **Line** is in use, as each **Specify next point:** appears at the command line, a *rubber-banded* line appears from the last point *entered*. *Drag* the rubber-band line in any direction and *enter* a number at the keyboard, followed by a *right-click*. The line is drawn in the *dragged* direction of a length in units equal to the *entered* number.

In this example because all lines are drawn in either the vertical or the horizontal direction, either press the **F8** key or *click* the **ORTHO** button in the status bar.

1. Close the drawing and open a new **acadiso.dwt** window.
2. *Left-click* on the **Line** tool icon and *enter* figures as follows at each prompt of the command line sequence:

   Command:_line Specify first point: *enter* **65,220** *right-click*
   Specify next point: *drag* to right *enter* **240** *right-click*
   Specify next point: *drag* down *enter* **145** *right-click*
   Specify next point or [Undo]: *drag* left *enter* **65** *right-click*
   Specify next point or [Undo]: *drag* upwards *enter* **25** *right-click*
   Specify next point or [Close/Undo]: *drag* left *enter* **120** *right-click*
   Specify next point or [Close/Undo]: *drag* upwards *enter* **25** *right-click*
   Specify next point or [Close/Undo]: *drag* left *enter* **55** *right-click*
   Specify next point or [Close/Undo]: c (Close) *right-click*
   Command:

The result is as shown in Fig. 2.10.

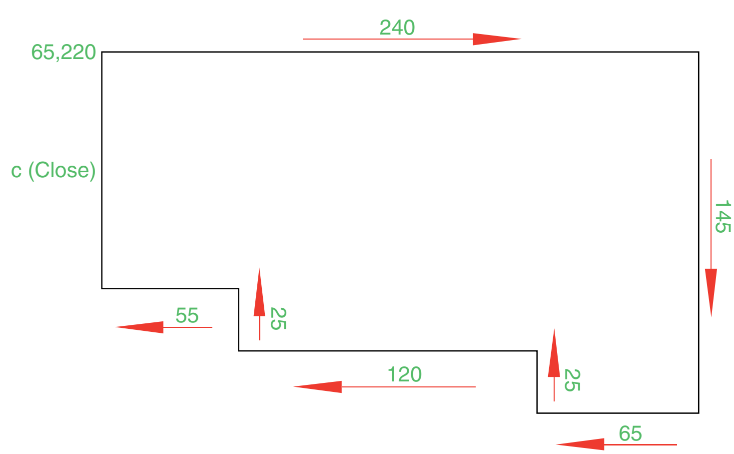


Fig. 2.10 Fifth example – **Line** tool

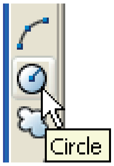
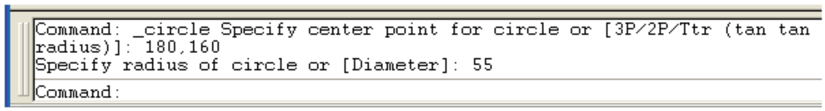


Fig. 2.11 The **Circle** tool from the **Draw** toolbar

Drawing with the Circle tool

First example – Circle tool (Fig. 2.13)

1. Close the drawing just completed and open the **acadiso.dwt** screen.
2. *Left-click* on the **Circle** tool icon in the **Draw** toolbar (Fig. 2.11).
3. *Enter* numbers against the prompts appearing in the command window as shown in Fig. 2.12, followed by *right-clicks*. The circle (Fig. 2.13) appears on screen.



```
Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan
radius)]: 180,160
Specify radius of circle or [Diameter]: 55
Command:
```

Fig. 2.12 First example – **Circle**. The command line prompts when **Circle** is called

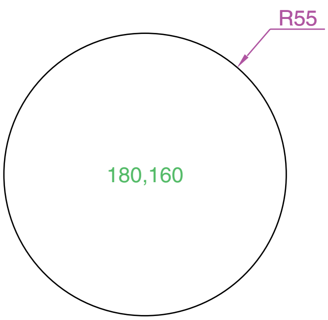


Fig. 2.13 First example – **Circle** tool

Second example – Circle tool (Fig. 2.14)

1. Close the drawing and open the **acadiso.dwt** screen.
2. *Left-click* on the **Circle** tool icon and construct two circles as shown in the drawing Fig. 2.14 in the positions and with the radii shown in Fig. 2.15.
3. *Click* the **Circle** tool again and against the first prompt *enter* **t** (the abbreviation for the prompt **tan tan radius**), followed by a *right-click*.

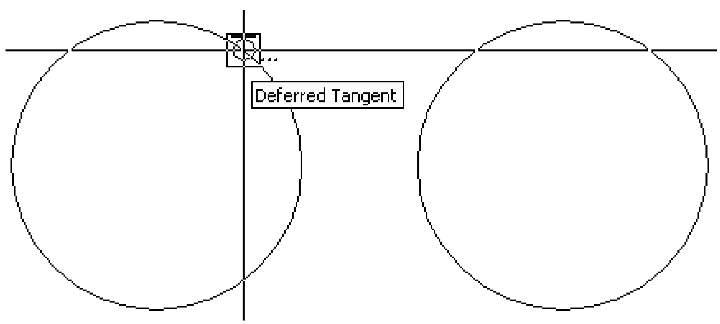


Fig. 2.14 Second example – **Circle** tool – the two circles of radius 50

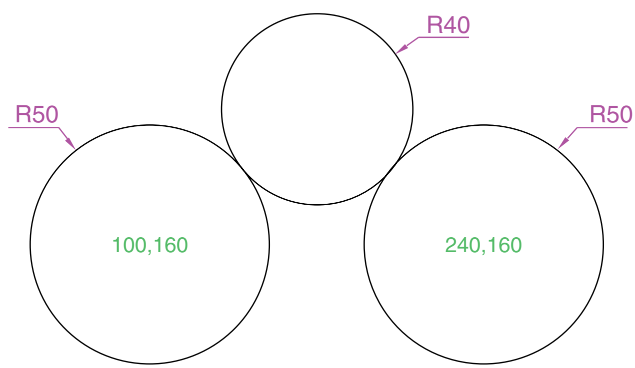


Fig. 2.15 Second example – **Circle** tool. The radius-40 circle tangential to the radius-50 circles

Command_circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *enter* **t** *right-click*
Specify point on object for first tangent of circle: *pick*
Specify point on object for second tangent of circle: *pick*
Specify radius of circle (50): *enter* **40** *right-click*
Command:

The radius-**40** circle tangential to the two circles already drawn then appears (Fig. 2.15).

Notes

1. When a point on either circle is picked a tip appears **Deferred Tangent**. This tip will only appear when the **OSNAP** button is set on with a *click* on its button in the status bar, or by pressing the **F3** key of the keyboard.
2. Circles can be drawn through 3 points or 2 points *entered* at the command line in response to prompts brought to the command line by using **3P** and **2P** in answer to the circle command line prompts.

The Erase tool

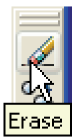


Fig. 2.16 The **Erase** tool icon from the **Modify** toolbar

If an error has been made when using any of the AutoCAD 2007 tools, the object or objects which have been incorrectly constructed can be deleted with the **Erase** tool. The **Erase** tool icon can be selected from the **Modify** toolbar (Fig. 2.16) or by *entering* **e** at the command line.

First example – Erase (Fig. 2.18)

1. With **Line** construct the outline in Fig. 2.17.

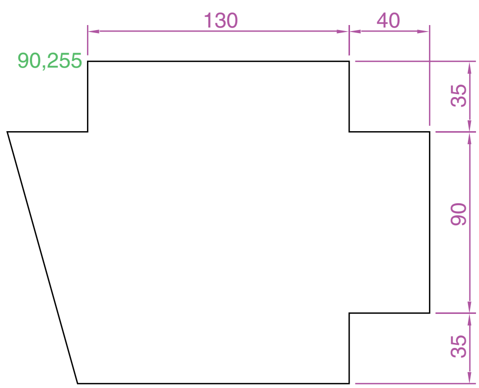


Fig. 2.17 First example – **Erase**. An incorrect outline

2. Assuming two lines of the outline have been incorrectly drawn, *left-click* the **Erase** tool icon. The command line shows:

Command:_erase
Select objects: *pick* one of the lines
Select objects: *pick* the other line

Select objects: *right-click*
Command:

And the two lines are deleted (right-hand drawing of Fig. 2.18).

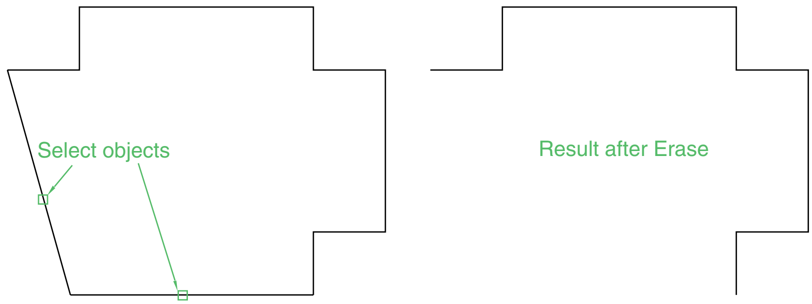


Fig. 2.18 First example – **Erase**

Second example – Erase (Fig. 2.19)

The two lines could also have been deleted by the following method:

1. *Left-click* the **Erase** tool icon. The command line shows:

Command:_erase
Select objects: *enter* c (Crossing)
Specify first corner: *pick* **Specify opposite corner**: *pick* **2 found**
Select objects: *right-click*
Command:

And the two lines are deleted as in the right-hand drawing of Fig. 2.18.

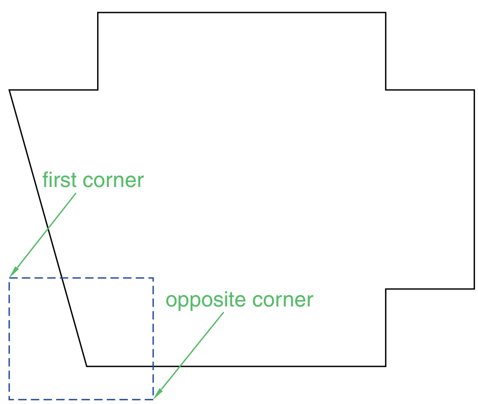


Fig. 2.19 Second example – **Erase**

Undo and Redo tools

Two other tools of value when errors have been made are the **Undo** and **Redo** tools. To undo the last action taken by any tool when constructing a drawing, either *left-click* the **Undo** tool in the **Standard** toolbar

Fig. 2.20 The **Undo** tool in the **Standard** toolbar

Fig. 2.21 The **Redo** tool in the **Standard** toolbar

Fig. 2.22 The popup menu from the **Redo** arrow

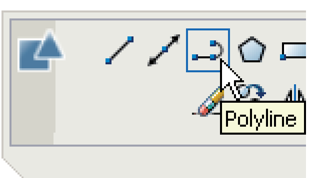


Fig. 2.23 The **Polyline** tool icon in the **2D Draw control panel**

(Fig. 2.20) or type **u** at the command line. No matter which method is adopted the error is deleted from the drawing.

Everything done during a session in constructing a drawing can be undone by repeated *clicking* on the **Undo** tool icon or by *entering* **u**'s at the command line.

To bring back objects that have just been removed by the use of **Undo**'s, *left-click* the **Redo** tool icon in the **Standard** toolbar (Fig. 2.21) or *enter* **redo** at the command line. Fig. 2.22 shows the popup menu of the **Redo** tool.

Drawing with the Polyline tool

When drawing lines with the **Line** tool, each line drawn is an object in its own right. A rectangle drawn with the **Line** tool is four objects. A rectangle drawn with the **Polyline** tool is a single object. Lines of different thickness, arcs, arrows and circles can all be drawn using this tool as will be shown in the examples describing constructions using the **Polyline** tool. Constructions resulting from using the tool are known as **polylines** or **plines**.

The **Polyline** tool can be called from the **2D Draw control panel** (Fig. 2.23) or from the **Draw** toolbar.

First example – Polyline tool (Fig. 2.24)

Note

In this example *enter* and *right-click* have not been included.

Left-click the **Polyline** tool (Fig. 2.23). The command line shows:

Command:_pline Specify start point: 30,250
Current line width is 0
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 230,250
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 230,120
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 30,120
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c (Close)
Command:

Fig. 2.24 First example – **Polyline** tool

Notes

1. Note the prompts – **Arc** for constructing pline arcs; **Close** to close an outline; **Halfwidth** to halve the width of a wide pline; **Length** to *enter* the required length of a pline; **Undo** to undo the last pline constructed; **Close** to close an outline.
2. Only the capital letter(s) of a prompt needs to be *entered* in upper or lower case to make that prompt effective.
3. Other prompts will appear when the **Polyline** tool is in use as will be shown in later examples.

Second example – Polyline tool (Fig. 2.25)

This will be a long sequence, but it is typical of a reasonably complex drawing using the **Polyline** tool. In the following sequences, when a prompt line is to be repeated, the prompts in square brackets ([ ]) will be replaced by **[prompts]**.

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 40,250
Current line width is 0
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: w (Width)
Specify starting width <0>: 5
Specify ending width <5>: *right-click*
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 160,250
Specify next point or [prompts]: h (Halfwidth)
Specify starting half-width <2.5>: 1
Specify ending half-width <1>: *right-click*
Specify next point or [prompts]: 260,250
Specify next point or [prompts]: 260,180
Specify next point or [prompts]: w (Width)
Specify starting width <1>: 10
Specify ending width <10>: *right-click*
Specify next point or [prompts]: 260,120
Specify next point or [prompts]: h (Halfwidth)

Fig. 2.25 Second example – **Polyline** tool

Specify starting half-width <5>: 2
Specify ending half-width <2>: *right-click*
Specify next point or [prompts]: 160,120
Specify next point or [prompts]: w (Width)
Specify starting width <4>: 20
Specify ending width <20>: *right-click*
Specify next point or [prompts]: 40,120
Specify starting width <20>: 5
Specify ending width <5>: *right-click*
Specify next point or [prompts]: c (Close)
Command:

Third example – Polyline tool (Fig. 2.26)

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 50,220
Current line width is 0
[prompts]: w (Width)
Specify starting width <0>: 0.5
Specify ending width <0.5>: *right-click*
Specify next point or [prompts]: 120,220
Specify next point or [prompts]: a (Arc)
Specify end point of arc or [prompts]: s (second pt)
Specify second point on arc: 150,200
Specify end point of arc: 180,220
Specify end point of arc or [prompts]: l (Line)
Specify next point or [prompts]: 250,220
Specify next point or [prompts]: 250,190
Specify next point or [prompts]: a (Arc)
Specify end point of arc or [prompts]: s (second pt)
Specify second point on arc: 240,170
Specify end point of arc: 250,150
Specify end point of arc or [prompts]: l (Line)
Specify next point or [prompts]: 250,150
Specify next point or [prompts]: 250,120

And so on until the outline in Fig. 2.26 is completed.

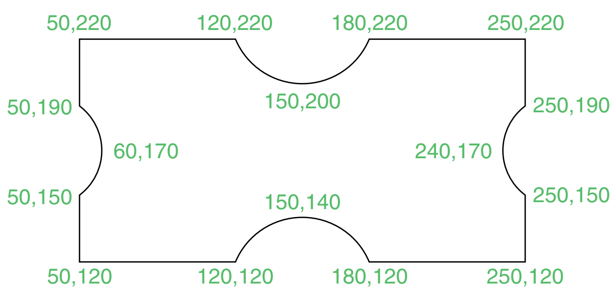


Fig. 2.26 Third example – **Polyline** tool

Fourth example – Polyline tool (Fig. 2.27)

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 80,170
Current line width is 0
Specify next point or [prompts]: w (Width)
Specify starting width <0>: 1
Specify ending width <1>: *right-click*
Specify next point or [prompts]: a (Arc)
Specify end point or arc or [prompts]: s (second pt)
Specify second point on arc: 160,250
Specify end point of arc: 240,170
Specify end point of arc or [prompts]: cl (CLose)
Command:

And the circle in Fig. 2.27 is formed.

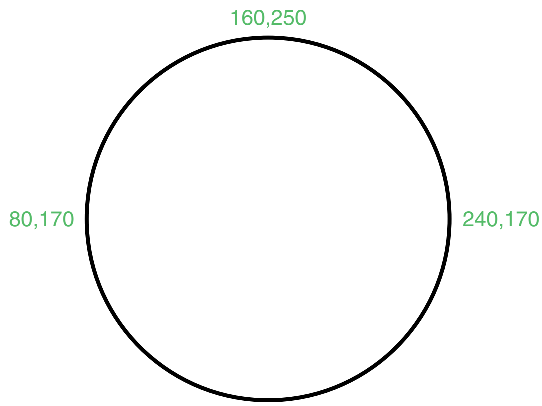


Fig. 2.27 Fourth example – **Polyline** tool

Fifth example – Polyline tool (Fig. 2.28)

Left-click the **Polyline** tool icon. The command line shows:

Command:_pline Specify start point: 60,180
Current line width is 0
Specify next point or [prompts]: w (Width)
Specify starting width <0>: 1
Specify ending width <1>: *right-click*
Specify next point or [prompts]: 190,180
Specify next point or [prompts]: w (Width)
Specify starting width <1>: 20
Specify ending width <20>: 0
Specify next point or [prompts]: 265,180
Specify next point or [prompts]: *right-click*
Command:

And the arrow in Fig. 2.28 is formed.

Fig. 2.28 Fifth example – **Polyline** tool

Revision notes

The following terms have been used in this chapter:

Left-click – press the left-hand button of the mouse.
Click – same meaning as *left-click*.
Double-click – press the left-hand button of the mouse twice in quick succession.
Right-click – press the right-hand button of the mouse; it has the same result as pressing the **Return** key of the keyboard.
Drag – move the cursor on to an object and, holding down the right-hand button of the mouse pull the object to a new position.
Enter – type the letters of numbers which follow at the keyboard.
Pick – move the cursor on to an item on screen and press the left-hand button of the mouse.
Return – press the ***Enter*** key of the keyboard. This key may also be marked with a left-facing arrow. In most cases (but not always) it has the same result as a *right-click*.
Dialog – a window appearing in the AutoCAD window in which settings can be made.
Drop-down menu – a menu appearing when one of the names in the menu bars is *clicked*.
Tooltip – the name of a tool appearing when the cursor is placed over a tool icon from a toolbar.
Prompts – text appearing in the command window when a tool is selected which advise the operator as to which operation is required.
Methods of coordinate entry – Three methods of coordinate entry have been used in this chapter:

1. **Absolute method** – the coordinates of points on an outline are *entered* at the command line in response to prompts.
2. **Relative method** – the distances in coordinate units are *entered* preceded by @ from the last point which has been determined on an outline. Angles which are measured in a counter-clockwise direction are preceded by >.
3. **Tracking** – the rubber band of the tool is *dragged* in the direction in which the line is to be drawn and its distance in units is *entered* at the command line followed by a *right-click*.

Line and Polyline tools – an outline drawn using the **Line** tool consists of a number of objects equal to the number of lines in the outline. An outline drawn using the **Polyline** is a single object no matter how many plines are in the outline.

Exercises

1. Using the **Line** tool construct the rectangle in Fig. 2.29.

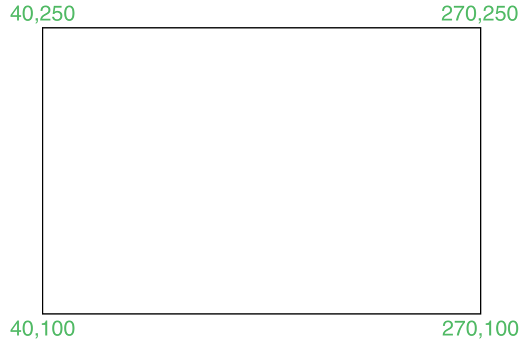


Fig. 2.29 Exercise 1

2. Construct the outline in Fig. 2.30 using the **Line** tool. The coordinate points of each corner of the rectangle will need to be calculated from the lengths of the lines between the corners.

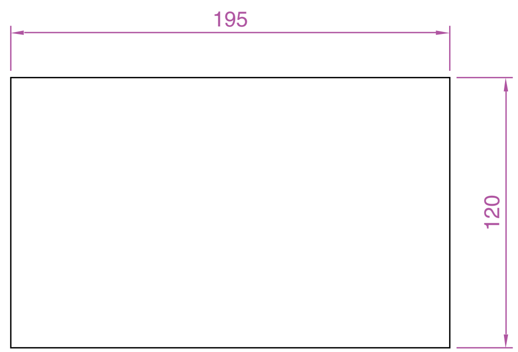


Fig. 2.30 Exercise 2

3. Using the **Line** tool, construct the outline in Fig. 2.31.

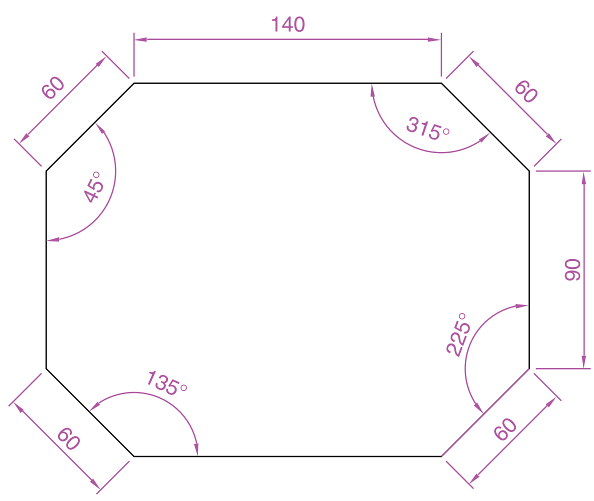


Fig. 2.31 Exercise 3

4. Using the **Circle** tool, construct the two circles of radius 50 and 30. Then, using the **Ttr** prompt, add the circle of radius 25 (Fig. 2.32).

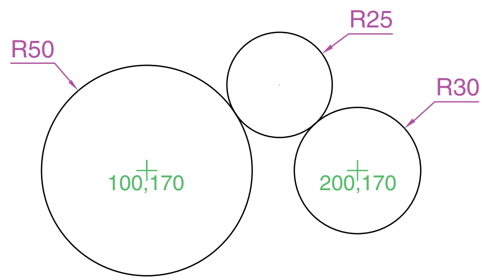


Fig. 2.32 Exercise 4

5. Fig. 2.33. In an **acadiso.dwt** screen and using the **Circle** and **Line** tools construct the line and the circle of radius 40. Then, using the **Ttr** prompt, add the circle of radius 25.

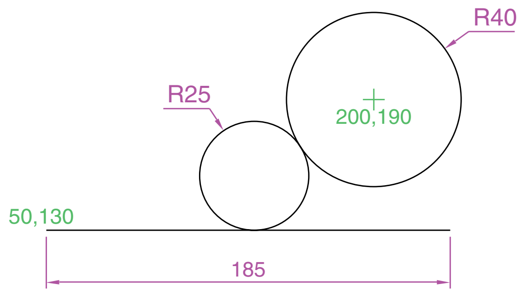


Fig. 2.33 Exercise 5

6. Using the **Line** tool construct the two lines at the length and angle as given in Fig. 2.34. Then with the **Ttr** prompt of the **Circle** tool, add the circle as shown.

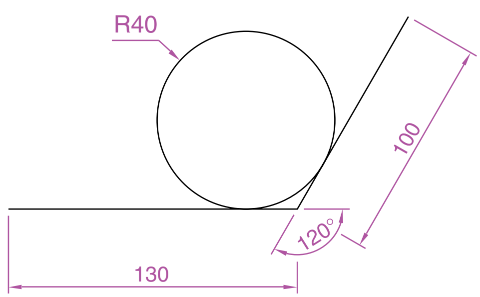


Fig. 2.34 Exercise 6

7. Using the **Polyline** tool, construct the outline given in Fig. 2.35.

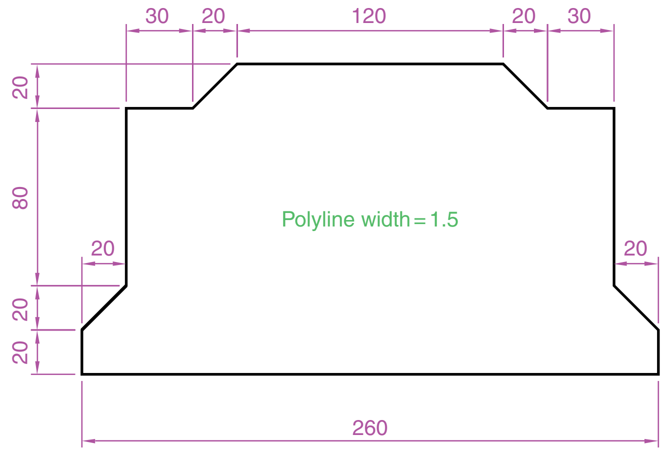


Fig. 2.35 Exercise 7

8. Construct the outline in Fig. 2.36 using the **Polyline** tool.

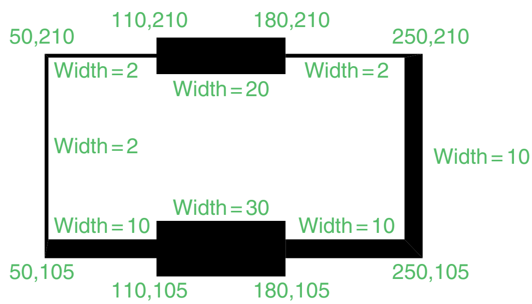


Fig. 2.36 Exercise 8

9. With the **Polyline** tool construct the arrows shown in Fig. 2.37.

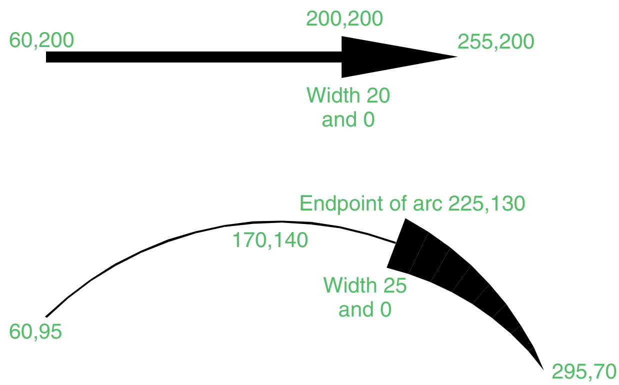


Fig. 2.37 Exercise 9

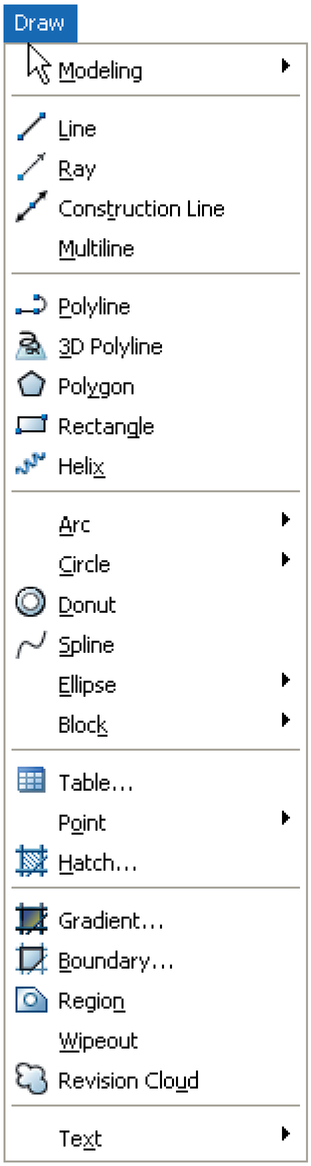


Fig. 3.1 The tool names in the **Draw** drop-down menu

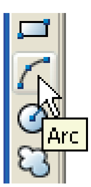


Fig. 3.2 The **Arc** tool icon in the **Draw** toolbar

CHAPTER 3

Osnap, AutoSnap and Draw tools

Aims of this chapter

1. To describe the use of the **Arc, Ellipse, Polygon** and **Rectangle** tools from the **Draw** toolbar.
2. To describe the uses of the **Polyline Edit** (pedit) tool.
3. To introduce the **AutoSnap** system and its uses.
4. To introduce the **Object Snap** (osnap) system and it uses.
5. To introduce the **Dynamic Input (DYN)** system and its uses.

Introduction

The majority of tools in AutoCAD 2007 can be called into use in any one of the following four ways:

1. With a *click* on the tool's icon in the **DASHBOARD** palette.
2. By *clicking* on the tool's name in an appropriate drop-down menu. Fig. 3.1 shows the tool names displayed in the **Draw** drop-down menu.
3. By *entering* an abbreviation for the tool name at the command line in the Command palette. For example the abbreviation for the **Line** tool is **1**, for the **Polyline** tool it is **pl** and for the **Circle** tool it is **c**.
4. By *entering* the full name of the tool at the command line.

In practice operators constructing drawings in AutoCAD 2007 may well use a combination of these four methods.

The Arc tool

In AutoCAD 2007, arcs can be constructed using any three of the following characteristics of an arc: its **Start** point; a point on the arc (**Second** point); its **Center**; its **End**; its **Radius**; **Length** of the arc; **Direction** in which the arc is to be constructed; **Angle** between lines of the arc.

In the examples which follow, *entering* initials for these characteristics in response to prompts at the command line when the **Arc** tool is called allows arcs to be constructed in a variety of ways.

To call the **Arc** tool *click* on its tool icon in the **Draw** toolbar (Fig. 3.2), or *click* on **Arc** in the **Draw** drop-down menu. A sub-menu shows the possible methods of constructing arcs (Fig. 3.3). The abbreviation for calling the **Arc** tool is **a**.

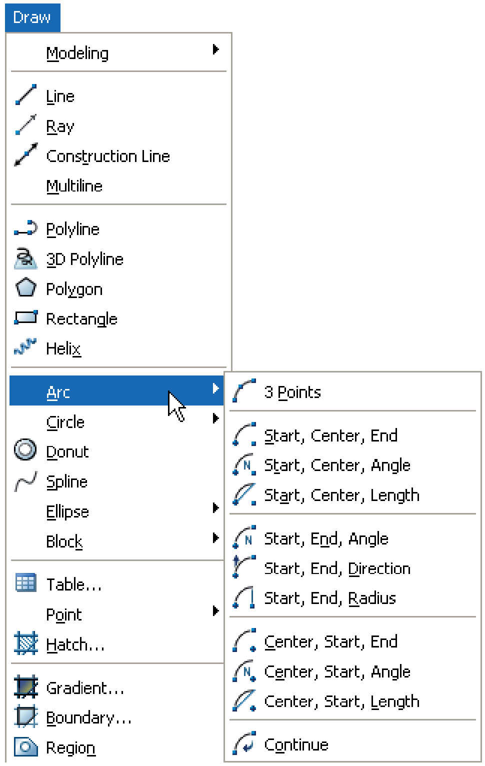


Fig. 3.3 The **Arc** sub-menu of the **Draw** drop-down menu

First example – Arc tool (Fig. 3.4)

Left-click the **Arc** tool icon. The command line shows:

Command:_arc Specify start point of arc or [Center]: 100,220
Specify second point of arc or [Center/End]: 55,250
Specify end point of arc: 10,220
Command:

Second example – Arc tool (Fig. 3.4)

Command: *right-click* brings back the **Arc** sequence
ARC Specify start point of arc or [Center]: c (Center)
Specify center point of arc: 200,190
Specify start point of arc: 260,215
Specify end point of arc or [Angle/chord Length]: 140,215
Command:

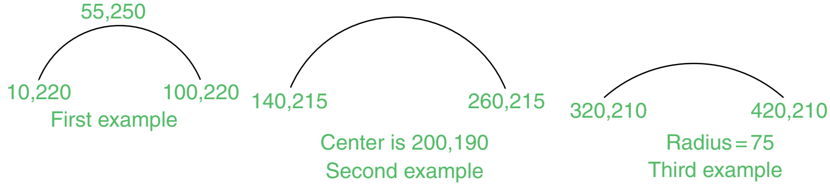


Fig. 3.4 Examples – **Arc** tool

Third example – Arc tool (Fig. 3.4)

Command: *right-click* brings back the **Arc** sequence
ARC Specify start point of arc or [Center]: 420,210
Specify second point of arc or [Center/End]: e (End)
Specify end point of arc: 320,210
Specify center point of arc or [Angle/Direction/Radius]: r (Radius)
Specify radius of arc: 75
Command:

The Ellipse tool

Ellipses can be regarded as what is seen when a circle is viewed from directly in front of the circle and the circle rotated through an angle about its horizontal diameter. Ellipses are measured in terms of two axes – a **major axis** and a **minor axis**; the major axis being the diameter of the circle and the minor axis being the height of the ellipse after the circle has been rotated through an angle (Fig. 3.5).

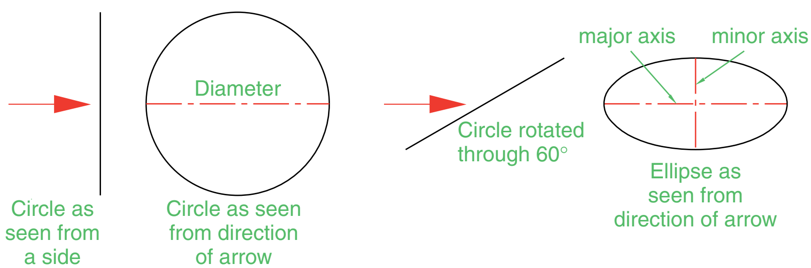


Fig. 3.5 An ellipse can be regarded as viewing a rotated circle

Fig. 3.6 The **Ellipse** tool icon in the **Draw** toolbar

To call the **Ellipse** tool, *click* on its tool icon in the **Draw** toolbar (Fig. 3.6) or *click* its name in the **Draw** drop-down menu. The abbreviation for calling the **Ellipse** tool is **el**.

First example – Ellipse (Fig. 3.7)

Left-click the **Ellipse** tool icon. The command line shows:

Command:_ellipse
Specify axis endpoint of elliptical arc or [Center]: 30,190
Specify other endpoint of axis: 150,190
Specify distance to other axis or [Rotation]: 25
Command:

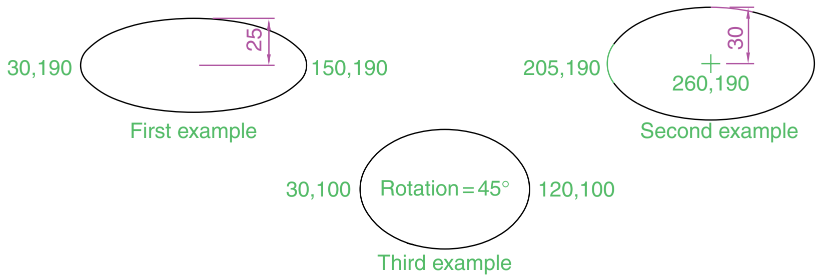


Fig. 3.7 Examples – **Ellipse**

Second example – Ellipse (Fig. 3.7)

In this second example, the coordinates of the centre of the ellipse (the point where the two axes intersect) are *entered*, followed by *entering* coordinates for the end of the major axis, followed by *entering* the units for the end of the minor axis.

Command: *right-click*
ELLIPSE
Specify axis endpoint of elliptical arc or [Center]: c
Specify center of ellipse: 260,190
Specify endpoint of axis: 205,190
Specify distance to other axis or [Rotation]: 30
Command:

Third example – Ellipse (Fig. 3.7)

In this third example, after setting the positions of the ends of the major axis, the angle of rotation of the circle from which an ellipse can be obtained is *entered*.

Command: *right-click*
ELLIPSE
Specify axis endpoint of elliptical arc or [Center]: 30,100
Specify other endpoint of axis: 120,100
Specify distance to other axis or [Rotation]: r (Rotation)
Specify rotation around major axis: 45
Command:

Saving drawings

Before going further it is as well to know how to save the drawings constructed when answering examples and exercises in this book. When a drawing has been constructed, *left-click* on **File** in the menu bar and on **Save As** . . . in the drop-down menu (Fig. 3.8). The **Save Drawing As** dialog appears (Fig. 3.9).

Unless you are the only person to use the computer on which the drawing has been constructed, it is best to save work to a floppy disk, usually held in the drive **A:**. To save a drawing to a floppy in drive **A:**

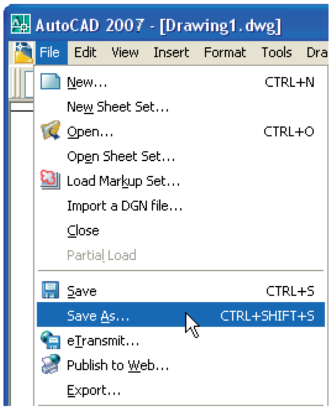


Fig. 3.8 Selecting **Save As** . . . in the **File** drop-down menu

1. Place a floppy disk in drive **A:**.
2. In the **Save in:** field of the dialog, *click* the arrow to the right of the field and from the popup list select **$3^1/_2$ Floppy [A:]**.

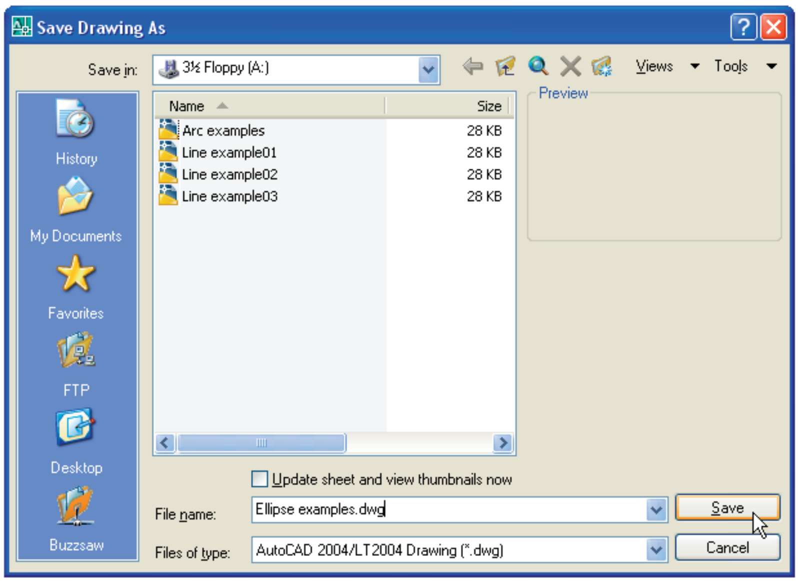


Fig. 3.9 The **Save Drawing As** dialog

3. In the **File name:** field of the dialog type a suitable name. The file name extension **.dwg** does not need to be typed because it will automatically be added to the file name.
4. *Left-click* the **Save** button of the dialog. The drawing will be saved to the floppy with the file name extension **.dwg** – the AutoCAD file name extension.

Osnap, AutoSnap and Dynamic Input

In previous chapters several methods of constructing accurate drawings have been described – using **Snap**; absolute coordinate entry; relative coordinate entry and tracking.

Other methods of ensuring accuracy between parts of constructions are by making use of **Object Snaps (Osnaps), AutoSnap** and **Dynamic Input (DYN)**.

Snap, Grid, Osnap and **DYN** can be set from the buttons in the status bar or by pressing the keys **F3 (Osnap), F7 (Grid), F9 (Snap)** and **F12 (DYN)**.

Object Snaps (Osnaps)

Osnaps allow objects to be added to a drawing at precise positions in relation to other objects already on screen. With osnaps, objects can be added to the end points, mid points, to intersections of objects, to centres and quadrants of circles, etc. Osnaps also override snap points even when snap is set on.

To set **Osnaps**, at the command line:

Command: *enter* **os**

And the **Drafting Settings** dialog appears. *Click* the **Object Snap** tab in the upper part of the dialog and *click* in each of the check boxes (the small squares opposite the osnap names). See Fig. 3.10.

When osnaps are set **ON**, as outlines are constructed using osnaps, osnap icons and their tooltips appear as indicated in Fig. 3.11.

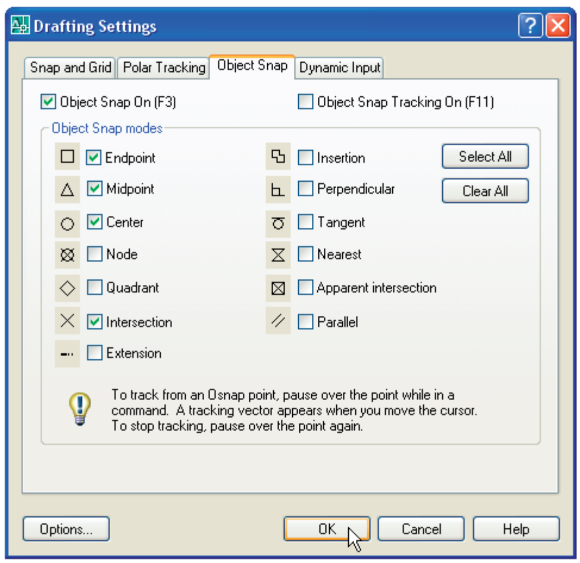


Fig. 3.10 The **Drafting Settings** dialog with some **Osnaps** set on

It is sometimes advisable not to have **Osnaps** set on in the **Drafting Settings** dialog, but to set **Osnap** off and use osnap abbreviations at the command line when using tools. The following examples show the use of some of these abbreviations.

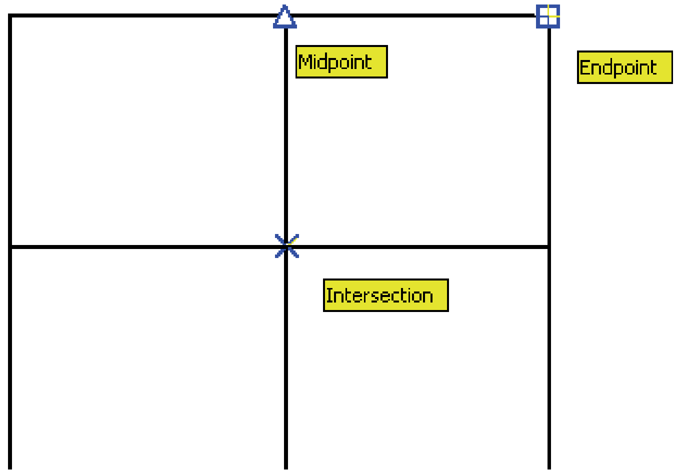


Fig. 3.11 Three osnap icons and their tooltips

First example – Osnap abbreviations (Fig. 3.12)

Call the **Polyline** tool:

Command:_pline
Specify start point: 50,230
[prompts]: w (Width)
Specify starting width: 1
Specify ending width <1>: *right-click*
Specify next point: 260,230
Specify next point: *right-click*
Command: *right-click*
PLINE
Specify start point: end **of** *pick* the right-hand end of the pline
Specify next point: 50,120
Specify next point: *right-click*
Command: *right-click*
PLINE
Specify start point: mid **of** *pick* near the middle of first pline
Specify next point: 155,120
Specify next point: *right-click*
Command: *right-click*
PLINE
Specify start point: int **of** *pick* the plines at their intersection
Specify start point: *right-click*
Command:

The result is shown in Fig. 3.12. In this illustration the osnap tooltips are shown as they appear when each object is added to the outline.

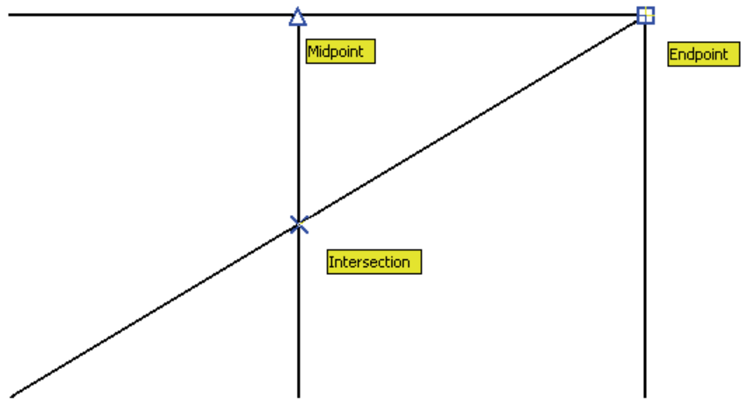


Fig. 3.12 First example – **Osnaps**

Second example – Osnap abbreviations (Fig. 3.13)

Call the **Circle** tool:

Command:_circle
Specify center point for circle: 180,170
Specify radius of circle: 60
Command: *enter* **l** (Line) *right-click*
Specify first point: *enter* **qua** *right-click*
of *pick* near the upper quadrant of the circle
Specify next point: *enter* **cen** *right-click*
of *pick* near the centre of the circle
Specify next point: *enter* **qua** *right-click*
of *pick* near right-hand side of circle
Specify next point: *right-click*
Command:

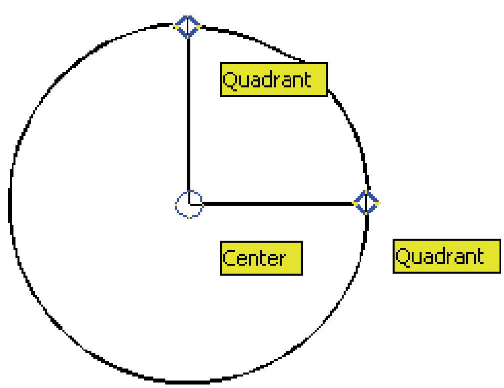


Fig. 3.13 Second example – **Osnaps**

Note

With osnaps off, the following abbreviations can be used:

end endpoint
int intersection
qua quadrant

ext	extension
mid	midpoint
cen	centre
nea	nearest

Using AutoSnap

AutoSnap is similar to **Osnap**. To set **AutoSnap**, *right-click* in the command window and from the menu which appears *click* **Options** . . . The **Options** dialog appears. *Click* the **Drafting** tab in the upper part of the dialog and set the check boxes against the **AutoSnap Settings** on (tick in boxes). These settings are shown on Figs 3.14 and 3.15.

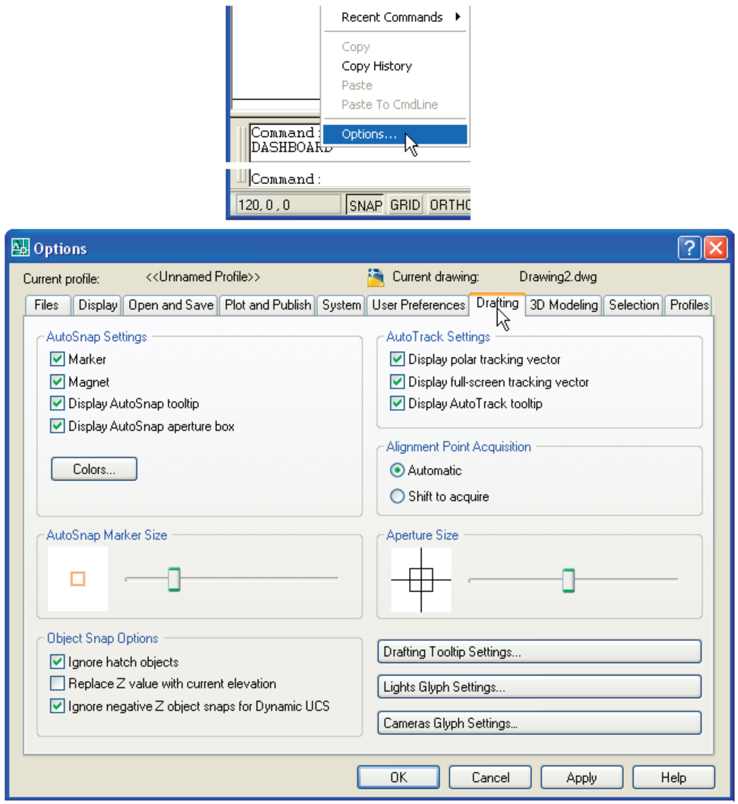


Fig. 3.14 Setting **AutoSnap** in the **Options** dialog

With **AutoSnap** set, each time an object is added to a drawing the **AutoSnap** features appear as indicated in Fig. 3.16.

Part of a drawing showing the features of a number of **AutoSnap** points is given in Fig. 3.17.

Note

OSNAP must be set **ON** for the AutoSnap features to show when constructing a drawing with their aid.

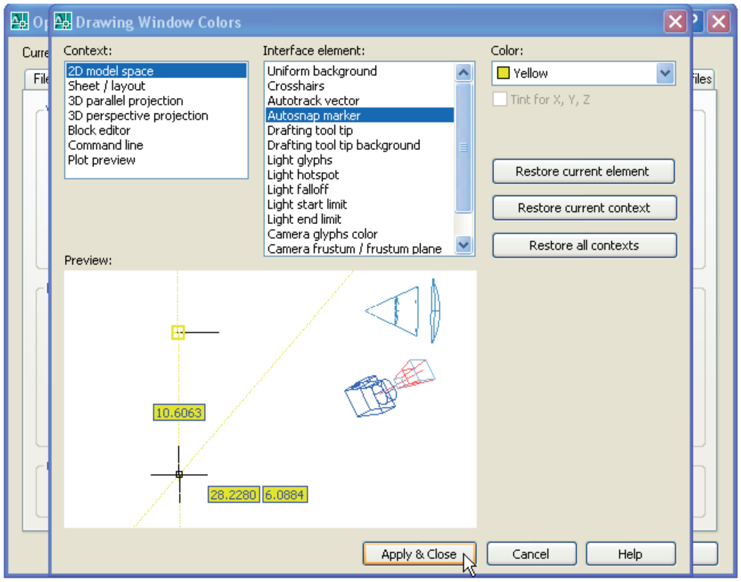


Fig. 3.15 Setting the colours of the parts of the **AutoSnap** features

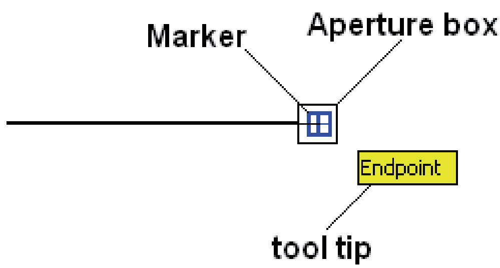


Fig. 3.16 The features of **AutoSnap**

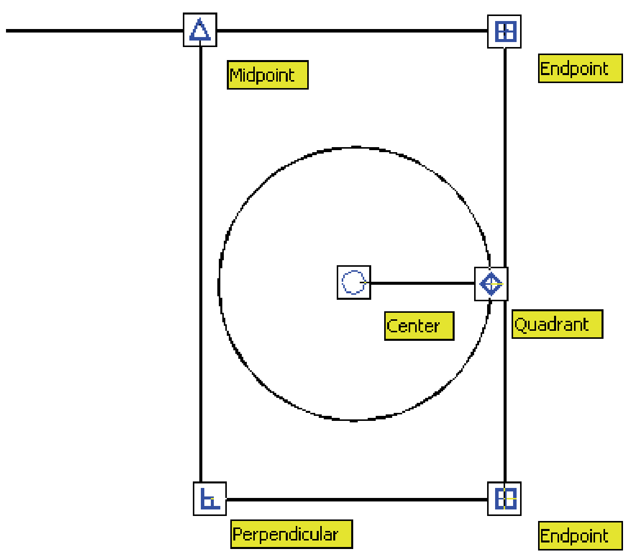


Fig. 3.17 A number of **AutoSnap** features

Dynamic Input

When **DYN** is set on by either pressing the **F12** key or with a *click* on the **DYN** button in the status bar, dimensions, coordinate positions and commands appear as tips when no tool is in action (Fig. 3.18).

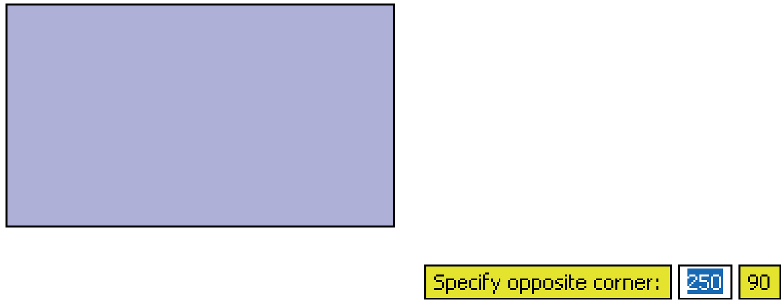


Fig. 3.18 The **DYN** tips appearing when no tool is in action and the cursor is moved

With a tool in action, as the cursor hairs are moved in response to movement of the mouse, **DYN** tips showing the coordinate figures for the point of the cursor hairs will show (Fig. 3.19), together with other details. To see the drop-down menu giving the prompts available with **DYN** press the down key of the keyboard and *click* the prompt to be used. Fig. 3.19 shows the **Arc** prompt as being the next to be used.

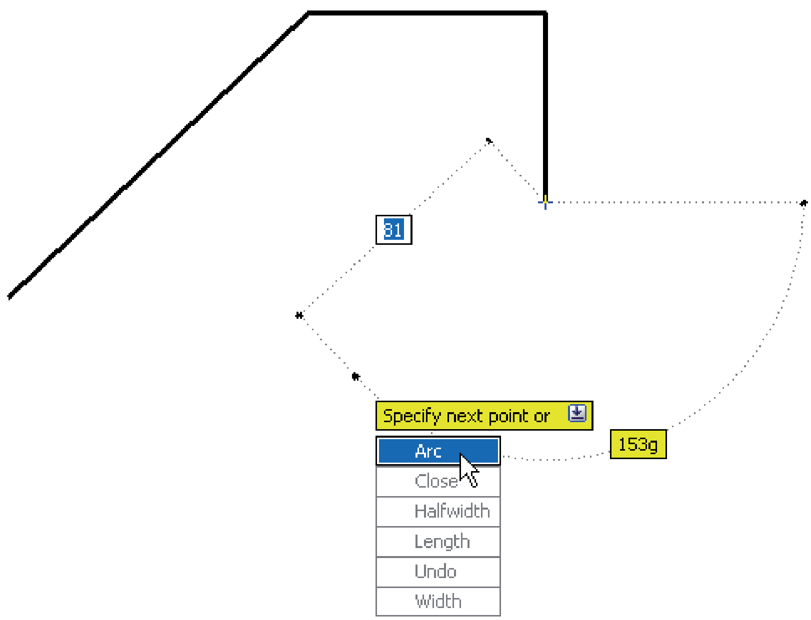


Fig. 3.19 Coordinate tips when **DYN** is in action

Notes on the use of DYN

1. A *click* on the **Clean Screen** icon in the bottom right-hand corner of the AutoCAD 2007 window produces an uncluttered workspace area with only the menu bar (Fig. 3.20). The command palette can also

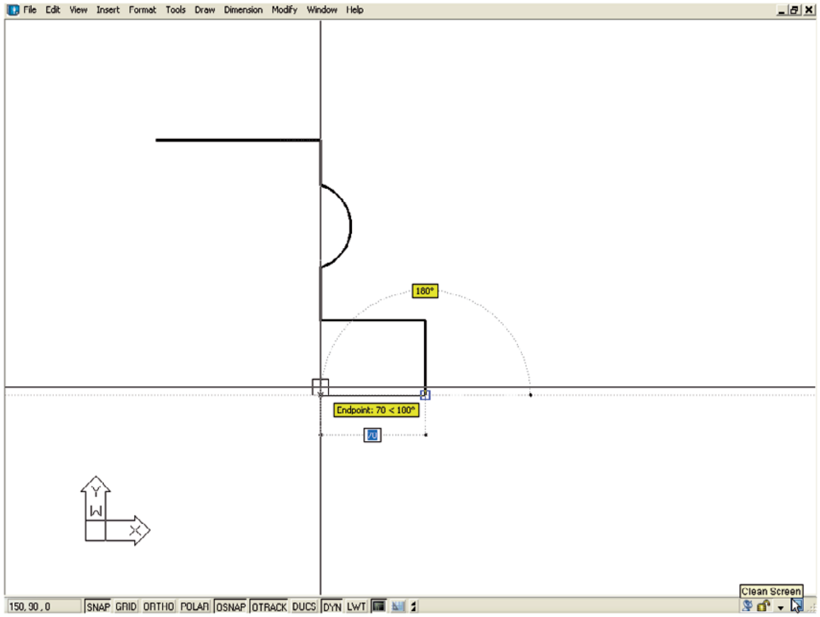


Fig. 3.20 Example – **Polyline Edit**

be cleared from screen by *entering* **commandlinehide** at the command line. To bring it back press the keys **Ctrl+9**. Some operators may well prefer working in such a larger than normal workspace. All the tool names or abbreviations can be *entered* at the keyboard. Thus working with **DYN** set on can be of benefit to those who prefer doing so.

2. Settings for **DYN** can be made in the **Drafting Settings** dialog (Fig. 3.21), brought to screen by *entering* **ds** at the command line.

An example of using DYN (Fig. 3.22)

This is a simple example of how **DYN** can be used to construct drawings in a **Clear Screen** workspace.

1. Turn **DYN** on with a *click* on its button in the status bar.
2. Turn **OSNAP** off with a *click* at the status bar.
3. *Click* the **Clear Screen** button at the bottom-right of the AutoCAD 2007 screen.
4. *Enter* **commandlinehide** to hide the command palette.
5. *Enter* **pl** (for **Polyline**) at the keyboard followed by pressing *Return*.
6. *Enter* **100,100** *Return*.
7. *Enter* **250,0** *Return*.
8. *Enter* **0,135** *Return*.
9. *Enter* **−250,0** *Return*.
10. *Enter* **0,−50** *Return*.
11. *Enter* **80,0** *Return*.
12. *Enter* **0,−35** *Return*.
13. *Enter* **−80,0** *Return*.
14. *Enter* **c** *Return*.

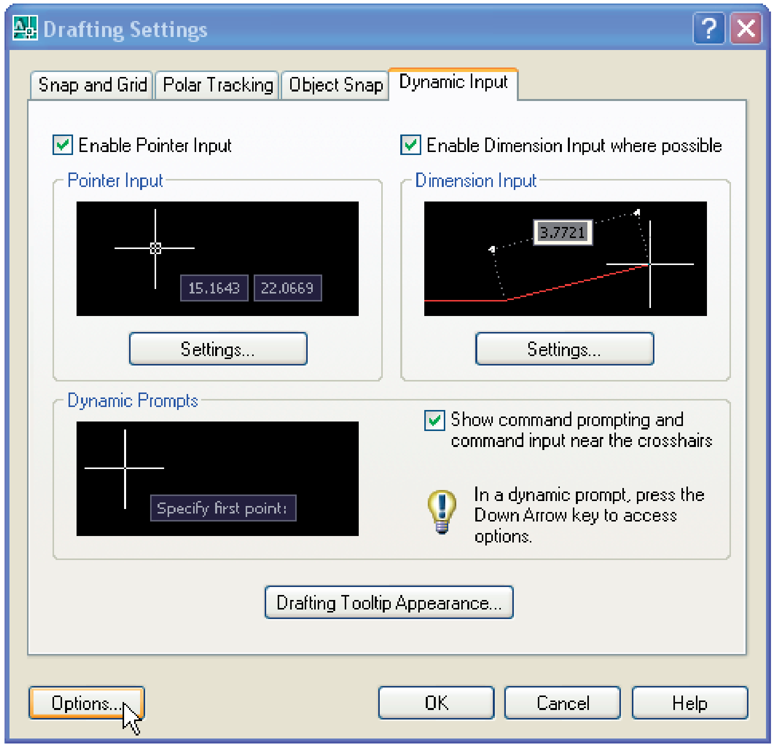


Fig. 3.21 Settings for **DYN** can be made in the **Drafting Settings** dialog

15. *Enter* **pe** (for **Polyline Edit**) *Return*.
16. Press the **down arrow** key of the keyboard.
17. In the menu which appears *click* **Width** *Return*.
18. *Click* on the pline just drawn and *enter* **2** *Return*.

The result is shown in Fig. 3.22.

Examples of using some Draw tools

First example – Polygon tool (Fig. 3.25)

1. Call the **Polygon** tool – either with a *click* on its tool icon in the **2D Draw control panel** (Fig. 3.23), by *entering* **pol** or **polygon** at the command line. Or it can be called from the **Draw** drop-down menu (Fig. 3.24). The command line shows:

Command:_polygon Enter number of sides <4>: 6
Specify center of polygon or [Edge]: 60,210
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: *right-click* (accept Inscribed)
Specify radius of circle: 60
Command:

2. In the same manner construct a **5**-sided polygon of centre **200,210** and radius **60**.

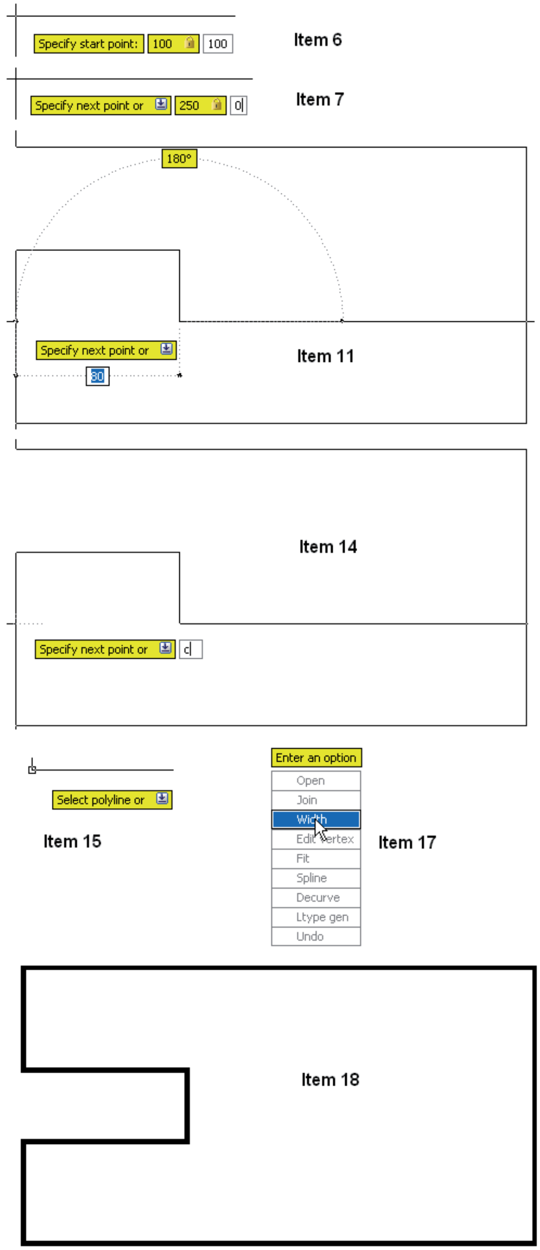


Fig. 3.22 An example of using **DYN** – stages in constructing the pline

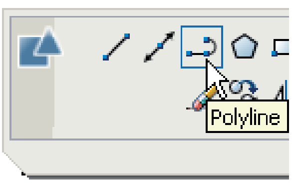


Fig. 3.23 The **Polygon** tool icon in the **2D Draw control panel**

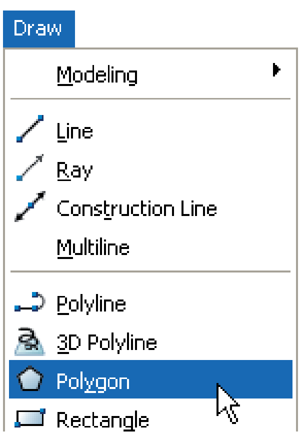


Fig. 3.24 Calling the **Polygon** tool from the **Draw** drop-down menu

3. Then, construct an **8**-sided polygon of centre **330,210** and radius **60**.
4. Repeat to construct a **9**-sided polygon circumscribed about a circle of radius **60** and centre **60,80**.
5. Construct yet another polygon with **10** sides of radius **60** and centre **200,80**.
6. Finally another polygon circumscribing a circle of radius **60**, of centre **330,80** and sides **12**.

The result is shown in Fig. 3.25.

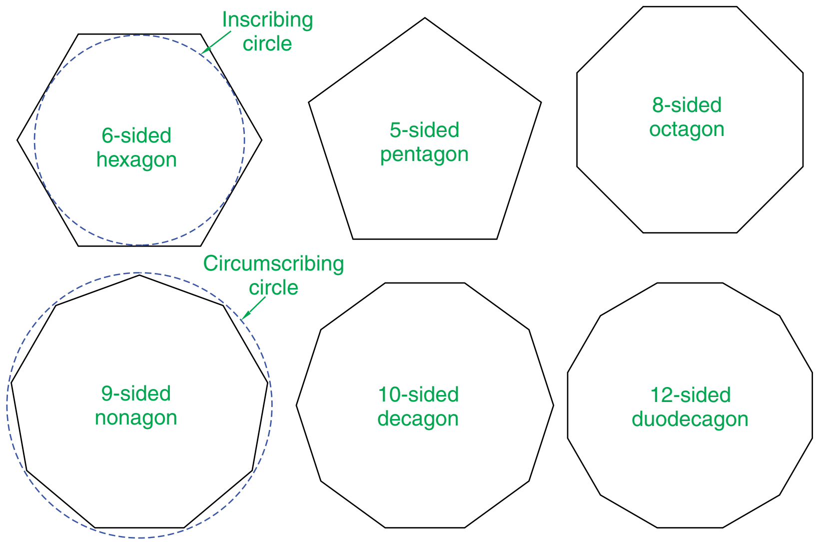


Fig. 3.25 First example – **Polygon** tool

Second example – Rectangle tool (Fig. 3.27)

Call the **Rectangle** tool – either with a *click* on its tool icon in the **Draw** toolbar (Fig. 3.26) or by *entering* **rec** or **rectangle** at the command line. Or it can be called from the **Draw** drop-down menu. The command line shows:

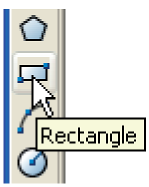


Fig. 3.26 The **Rectangle** tool icon and tooltip in the **Draw** toolbar

Command:_rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 25,240
Specify other corner point or [Area/Dimensions/Rotation]: 160,160
Command:

Third example – Rectangle tool (Fig. 3.27)

Command:_rectang
[prompts]: c (Chamfer)
Specify first chamfer distance for rectangles <0>: 15
Specify first chamfer distance for rectangles <15>: *right-click*
Specify first corner point: 200,240
Specify other corner point: 300,160
Command:

Fourth example – Rectangle (Fig. 3.27)

Command:_rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: w (Width)
Specify line width for rectangles <0>: 4
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: c (Chamfer)
Specify first chamfer distance for rectangles <0>: 10

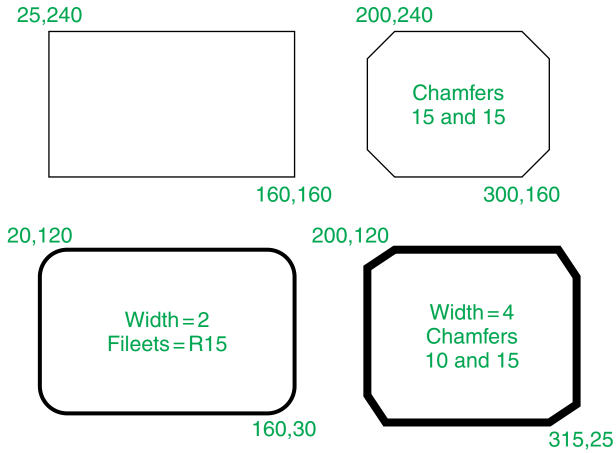


Fig. 3.27 Examples – **Rectangle** tool

Specify second chamfer distance for rectangles <10>: 15
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 200,120
Specify other corner point or [Area/Dimensions/Rotation]: 315,25
Command:

The Polyline Edit tool

Polyline Edit or **Pedit** is a valuable tool for editing plines.

First example – Polyline Edit (Figs 3.28 and 3.30)

1. With the **Polyline** tool construct the outlines **1** to **6** of Fig. 3.28.

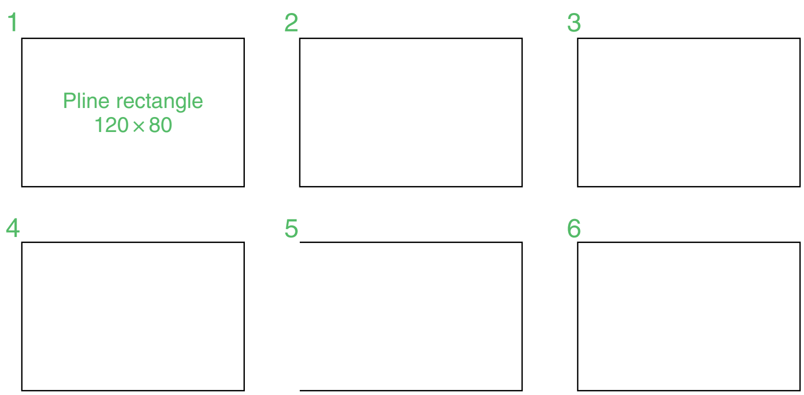


Fig. 3.28 Examples – **Edit Polyline** – the plines to be edited

2. Call the **Edit Polyline** tool – either from the **Modify** drop-down menu (Fig. 3.29), or by *entering* **pe** or **pedit** at the command line. The command line shows:

   Command: *enter* pe
   PEDIT Select polyline or [Multiple]: *pick* pline **2**

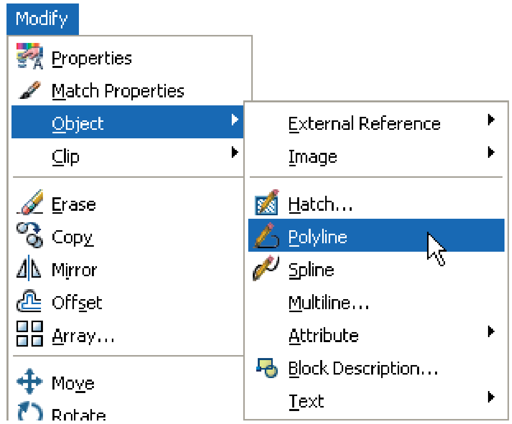


Fig. 3.29 Calling **Edit Polyline** from the **Modify** drop-down menu

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/ Ltype gen/Undo]: w (Width)
Specify new width for all segments: 2
Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/ Ltype gen/Undo]: *right-click*
Command:

3. Repeat with pline **3** and pedit to Width = **10**.
4. Repeat with pline **4** and *enter* **s** (Spline) in response to the prompt line:

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/ Ltype gen/Undo]:

5. Repeat with pline **5** and *enter* **j** (Join) in response to the prompt line:

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/ Ltype gen/Undo]:

The result is shown in pline **6**.
The resulting examples are shown in Fig. 3.30.

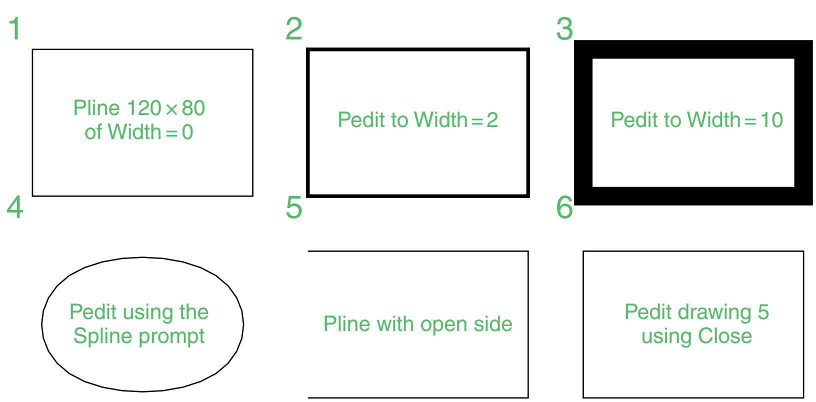


Fig. 3.30 Examples – **Edit Polyline**

Example – Multiple Polyline Edit (Fig. 3.31)

1. With the **Polyline** tool construct the left-hand outlines of Fig. 3.31.
2. Call the **Edit Polyline** tool. The command line shows:

Command: *enter* pe
PEDIT Select polyline or [Multiple]: m (Multiple)
Select objects: *pick* any one of the lines or arcs of the left-hand outlines of Fig. 3.31 **1 found**
Select objects: *pick* another line or arc **1 found 2 total**

Continue selecting lines and arcs as shown by the *pick* boxes of the left-hand drawing of Fig. 3.31 until the command line shows:

Select objects: *pick* another line or arc **1 found 24 total**
Select objects: *right-click*
[prompts]: w (Width)
Specify new width for all segments: 2
[prompts]: *right-click*
Command:

The result is shown in the right-hand drawing of Fig. 3.31.

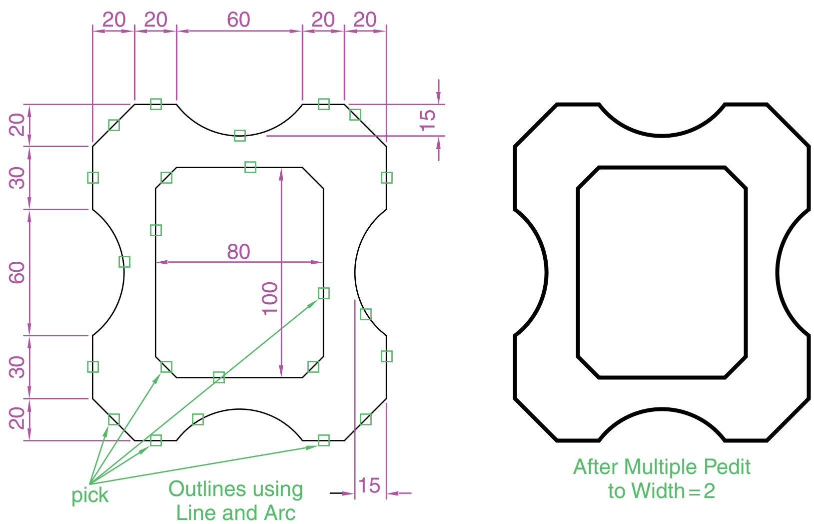


Fig. 3.31 Example – **Multiple Polyline Edit**

Transparent commands

When any tool is in operation it can be interrupted by prefixing the interrupting command with an apostrophe ('). This is particularly useful when the operator wishes to zoom when constructing a drawing (see page 61). As an example when the **Line** tool is being used:

Command:_line
Specify first point: 100,120

Specify next point: 190,120
Specify next point: *enter* 'z (Zoom)
>> Specify corner of window or [prompts]: *pick*
>>>>Specify opposite corner: *pick*
Resuming line command.
Specify next point:

And so on. The transparent command method can be used with any tool.

The set variable PELLIPSE

Many of the operations performed in AutoCAD are carried out under the settings of **set variables**. Some of the numerous set variables available in AutoCAD 2007 will be described in later pages. The variable **PELLIPSE** controls whether ellipses are drawn as splines or as polylines. It is set as follows:

Command: *enter* **pellipse** *right-click*
Enter new value for PELLIPSE <0>: *enter* **1** *right-click*
Command:

And now when ellipses are drawn they are plines. If the variable is set to **0**, the ellipses will be splines. The value of changing ellipses to plines is that they can then be edited using the **Edit Polyline** tool.

Revision notes

The following terms have been used in this chapter:

Field – a part of a window or dialog in which numbers or letters are *entered* or can be read.
Popup list – a list brought in screen with a *click* on the arrow often found at the right-hand end of a field.
Object – a part of a drawing which can be treated as a single object. For example a line constructed with the **Line** tool is an object; a rectangle constructed with the **Polyline** tool is an object; an arc constructed with the **Arc** tool is an object. It will be seen in Chapter 10 that several objects can be formed into a single object.
Toolbar – a collection of tool icons all of which have similar functions. For example in the **Classic AutoCAD** workspace the **Draw** toolbar contains tool icons of those tools which are used for drawing and the **Modify** toolbar contains tool icons of those tools used for modifying parts of drawings.
DASHBOARD palette – when working in either the **Classic AutoCAD** workspace, in the **3D Modeling** workspace, or in the **My Workspace** workspace, tool icons are held in the **DASHBOARD** palette.
Command line – a line in the command palette which commences with the word **Command:**

Snap, Grid and **Osnap** can be toggled with *clicks* on their respective buttons in the status bar.
These functions can also be set with function keys: **Snap – F9**; **Grid – F7** and **Osnap – F3**.
Osnaps ensure accurate positioning of objects in drawings.
AutoSnap can also be used for ensuring accurate positioning of objects in relation to other objects in a drawing.
Osnap must be set **ON** before **AutoSnap** can be used.
Osnap abbreviations can be used at the command line rather than setting it ON in the **Drafting Settings** dialog.
DYN – Dynamic input. Allows constructions in an enlarged workspace, without having to use the command palette.

Notes on tools

1. Polygons constructed with the **Polygon** tool are regular polygons – the edges of the polygons are all the same length and the angles are of the same degrees.
2. Polygons constructed with the **Polygon** tool are plines, so they can be acted upon with the **Edit Polyline** tool.
3. The easiest method of calling the **Edit Polyline** tool is to *enter* **pe** at the command line.
4. The **Multiple** prompt of the **pedit** tool saves considerable time when editing a number of objects in a drawing.
5. Transparent commands can be used to interrupt tools in operation by preceding the interrupting tool name with an apostrophe (').
6. Ellipses drawn when the variable **PELLIPSE** is set to **0** are splines, when **PELLIPSE** is set to **1**, ellipses are polylines. When ellipses are in polyline form they can be modified using the **pedit** tool.

Exercises

1. Using the **Line** and **Arc** tools, construct the outline given in Fig. 3.32.

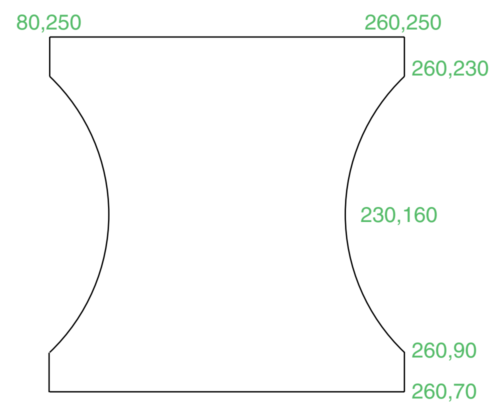


Fig. 3.32 Exercise 1

2. With the **Line** and **Arc** tools, construct the outline in Fig. 3.33.

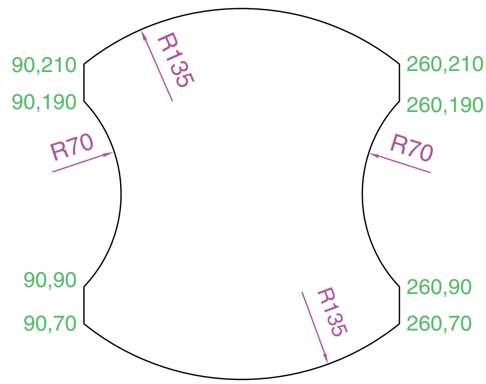


Fig. 3.33 Exercise 2

3. Using the **Ellipse** and **Arc** tools, construct the drawing in Fig. 3.34.

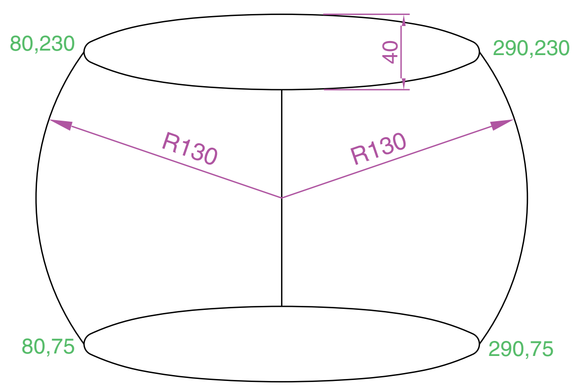


Fig. 3.34 Exercise 3

4. With the **Line, Circle** and **Ellipse** tools construct Fig. 3.35.

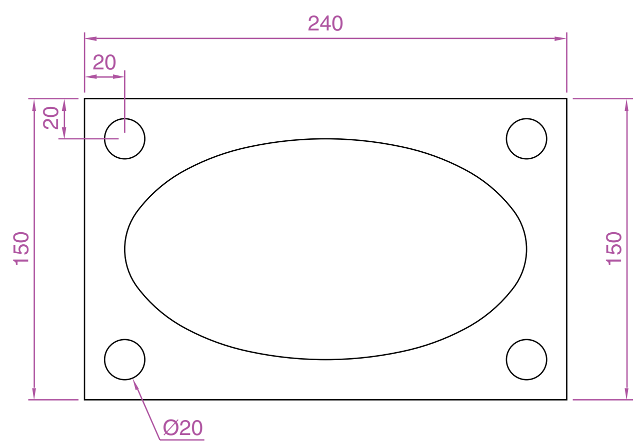


Fig. 3.35 Exercise 4

5. With the **Ellipse** tool, construct the drawing in Fig. 3.36.

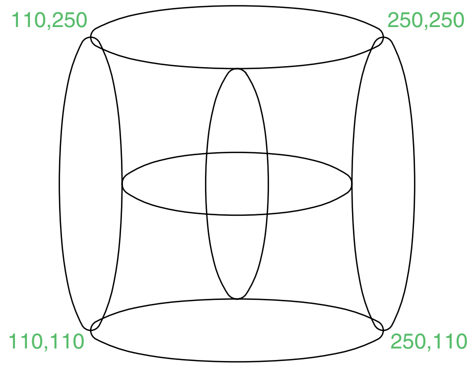


Fig. 3.36 Exercise 5

6. Fig. 3.37 shows a rectangle in the form of a square with hexagons along each edge. Using the **Dimensions** prompt of the **Rectangle** tool construct the square. Then, using the **Edge** prompt of the **Polygon** tool, add the four hexagons. Use the **Osnap endpoint** to ensure the polygons are in their exact positions.

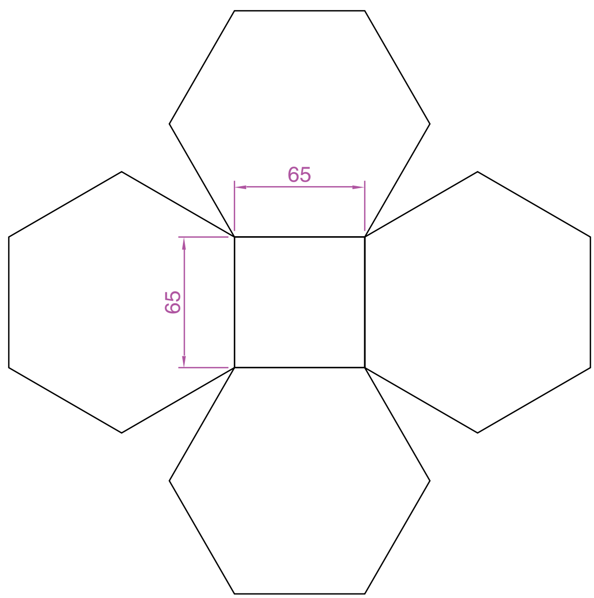


Fig. 3.37 Exercise 6

7. Fig. 3.38 shows seven hexagons with edges touching. Construct the inner hexagon using the **Polygon** tool, then with the aid of the **Edge** prompt of the tool, add the other six hexagons.

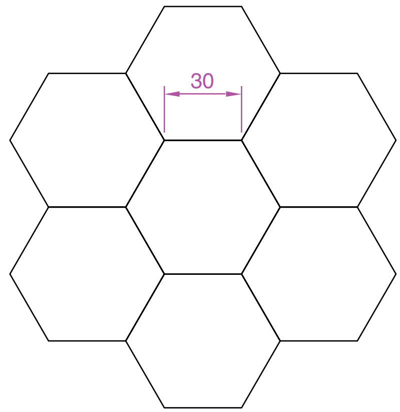


Fig. 3.38 Exercise 7

8. Fig. 3.39 was constructed using only the **Rectangle** tool. Make an exact copy of the drawing using only the **Rectangle** tool.

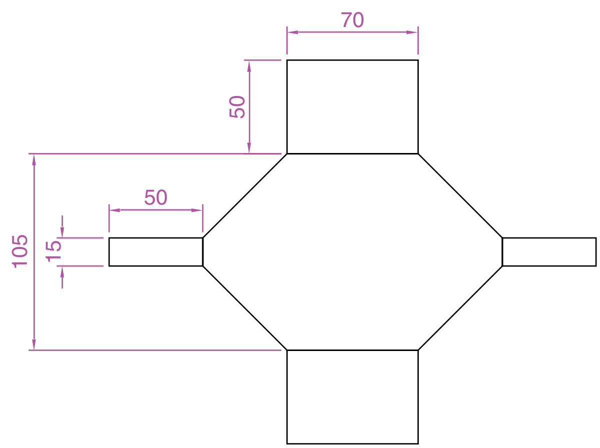


Fig. 3.39 Exercise 8

9. Construct the drawing in Fig. 3.40 using the **Line** and **Arc** tools. Then, with the aid of the **Multiple** prompt of the **Edit Polyline** tool change the outlines into plines of **Width=1**.

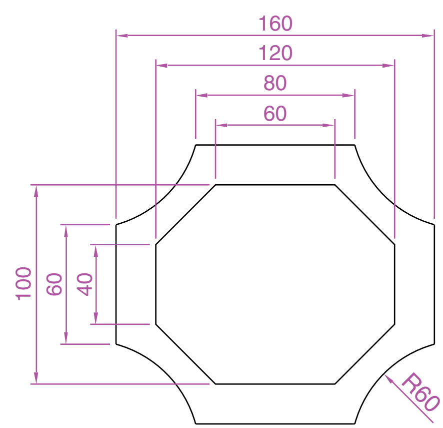


Fig. 3.40 Exercise 9

10. Construct Fig. 3.41 using the **Line** and **Arc** tools. Then change all widths of lines and arcs to a width of **2** with **Polyline Edit**.

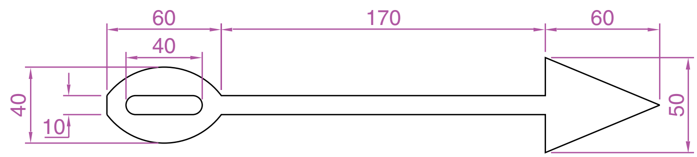


Fig. 3.41 Exercise 10

11. Construct the two outlines in Fig. 3.42 using the **Rectangle** and **Line** tools and then with **Edit Polyline** change the parts of the drawing to plines of widths as shown in Fig. 3.42.

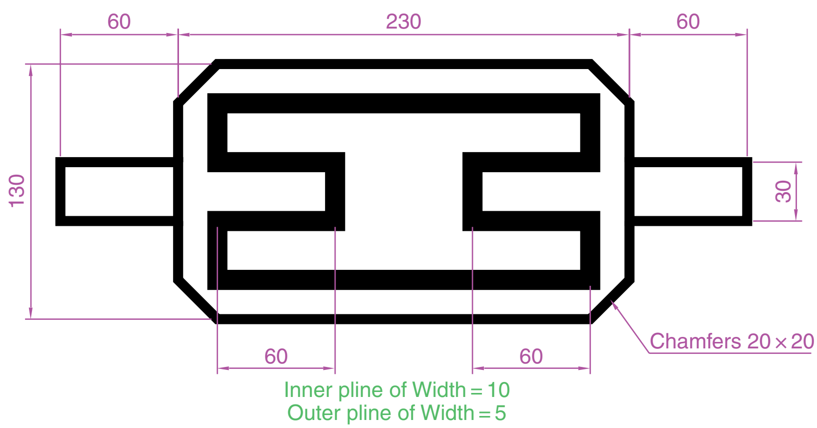


Fig. 3.42 Exercise 11

CHAPTER 4

Zoom, Pan and templates

Aims of this chapter

1. To demonstrate the value of the **Zoom** tools.
2. To introduce the **Pan** tool.
3. To describe the value of using the **Aerial View** window in conjunction with the **Zoom** and **Pan** tools.
4. To update the **acadiso.dwt** template.
5. To describe the construction and saving of drawing templates.

Introduction

The use of the **Zoom** tools allows not only the close inspection of the most minute areas of a drawing in the AutoCAD 2007 drawing area, but allows the construction of very small details accurately in a drawing.

The **Zoom** tools can be called from the **Zoom** sub-menu of the **View** drop-down menu (Fig. 4.1). However by far the easiest and

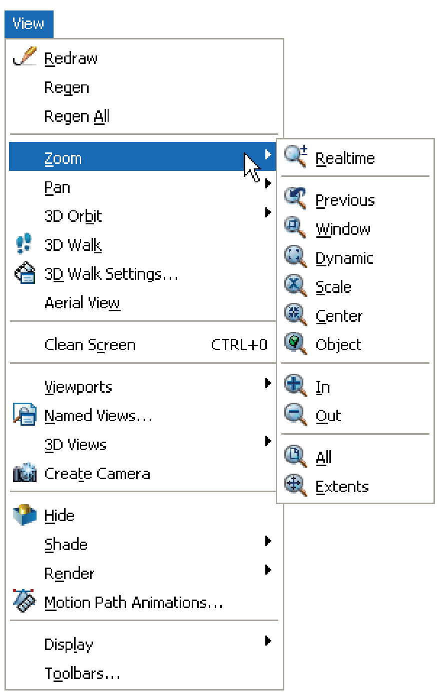


Fig. 4.1 Calling the **Zoom** tools from the **Zoom** sub-menu of the **View** drop-down menu

quickest method of calling **Zooms** is to *enter* **z** at the command line as follows:

Command: *enter* **z** *right-click*
ZOOM Specify corner of window, enter a scale factor (nX or nXP) or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:

This allows the different zooms:

Realtime – selects parts of a drawing within a window.
All – the screen reverts to the limits of the template.
Center – the drawing centres itself around a *picked* point.
Dynamic – a broken line surrounds the drawing which can be changed in size and repositioned to a part of the drawing.
Extents – the drawing fills the AutoCAD drawing area.
Previous – the screen reverts to its previous zoom.
Scale – entering a number or a decimal fraction scales the drawing.
Window – the parts of the drawing within a *picked* window appears on screen. The effect is the same as using **real time**.
Object – *pick* any object on screen and the object zooms.

The operator will probably be using **Realtime**, **Window** and **Previous** zooms most frequently.

Note the following illustrations: Fig. 4.2 – a drawing which has been constructed; Fig. 4.3 – a **Zoom Window** of part of the drawing allowing it to be checked for accuracy and Fig. 4.4 – a **Zoom Extents**.

It will be found that the **Zoom** tools are among those most frequently used when working in AutoCAD 2007.

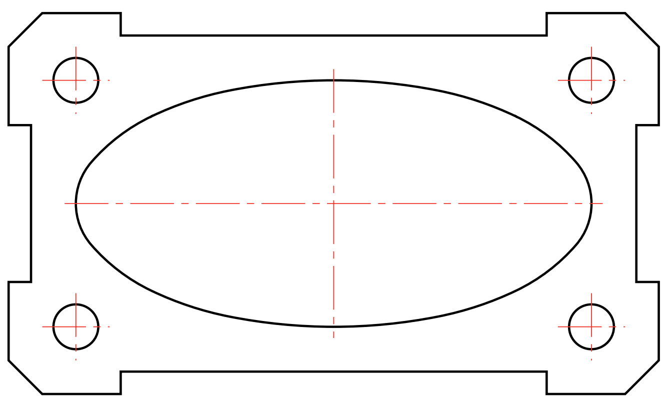


Fig. 4.2 A drawing constructed using the **Polyline** tool

The Aerial View window

Left-click on **Aerial View** from the **View** drop-down menu and the **Aerial View** window appears – usually in the bottom right-hand corner of the Auto-CAD 2007 window (Fig. 4.6). The **Aerial View** window shows the whole of a drawing with that part which is within the **Limits** of the drawing template

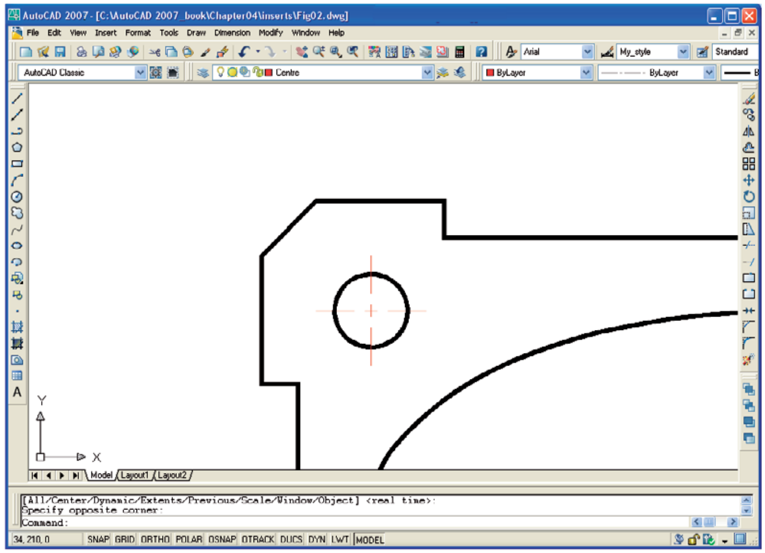


Fig. 4.3 A **Zoom Window** of part of the drawing in Fig. 4.2

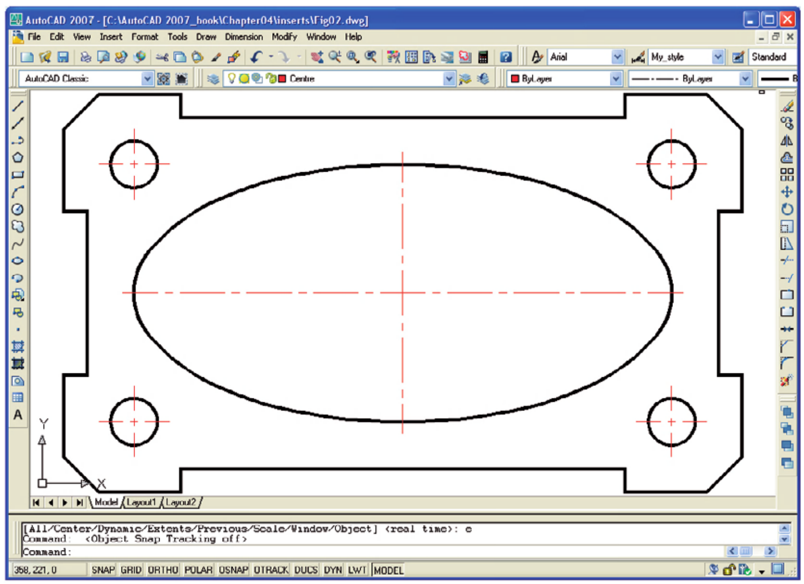


Fig. 4.4 A **Zoom Extents** of the drawing in Fig. 4.2

being used bounded with a thick black line. The **Aerial View** window is of value when dealing with large drawings – it allows that part of the window on screen to be shown in relation to other parts of the drawing. Fig. 4.5 shows the three-view orthographic projection of a small bench vice shown, in Figs 4.6 and 4.8.

The area of the drawing within a **Zoom** window in the drawing area is bounded by a thick black line in the **Aerial View** window.

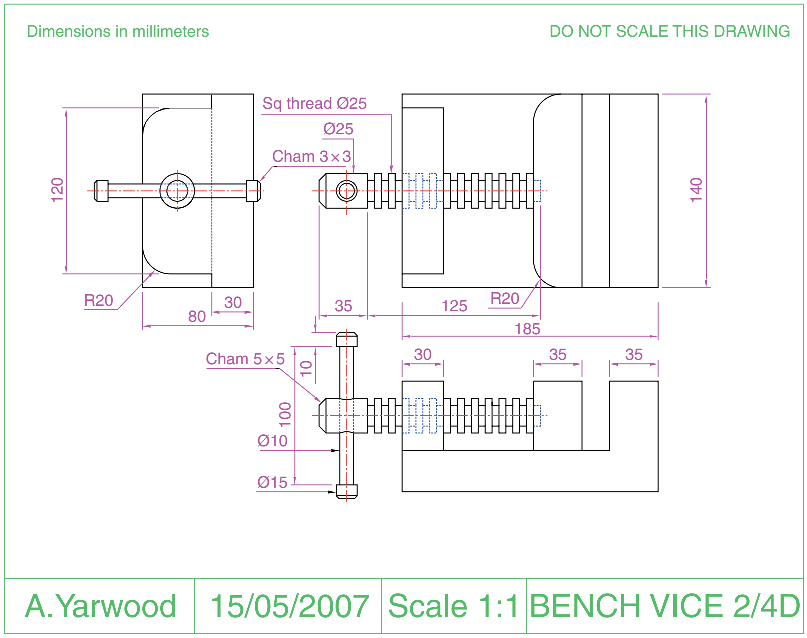


Fig. 4.5 The drawing used to illustrate Figs 4.6 and 4.8

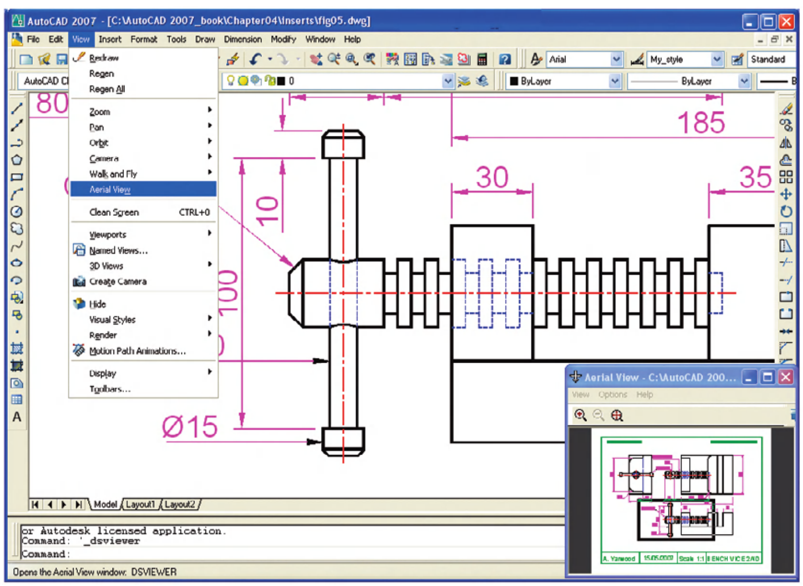


Fig. 4.6 A **Zoom Window** of the drawing in Fig 4.5 with its surrounding zoom rectangle showing in the **Aerial View** window

The Pan tool

The **Pan** tools can be called from the **Pan** sub-menu of the **View** drop-down menu (Fig. 4.7) or by *entering* **p** at the command line. When the tool is called, the cursor on screen changes to an icon of a hand. *Dragging* the hand across screen under mouse movement allows various parts of the

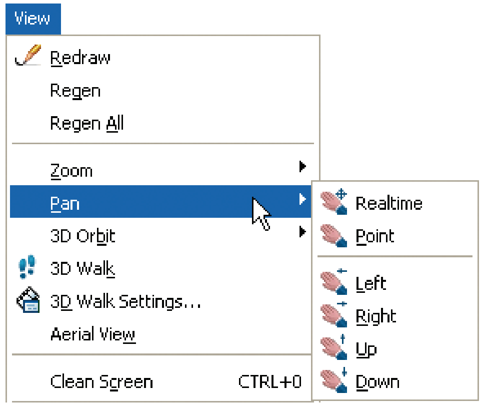


Fig. 4.7 Calling the **Pan** tools from the **Pan** sub-menu of the **View** drop-down menu

drawing not on screen to be viewed. As the *dragging* takes place, the black rectangle in the **Aerial View** window moves in sympathy (see Fig. 4.8). The **Pan** tool allows any part of the drawing to be viewed and/or modified. When that part of the drawing which is required is on screen a *right-click* calls up the menu as shown in Fig. 4.8, from which either the tool can be exited or other tools can be called.

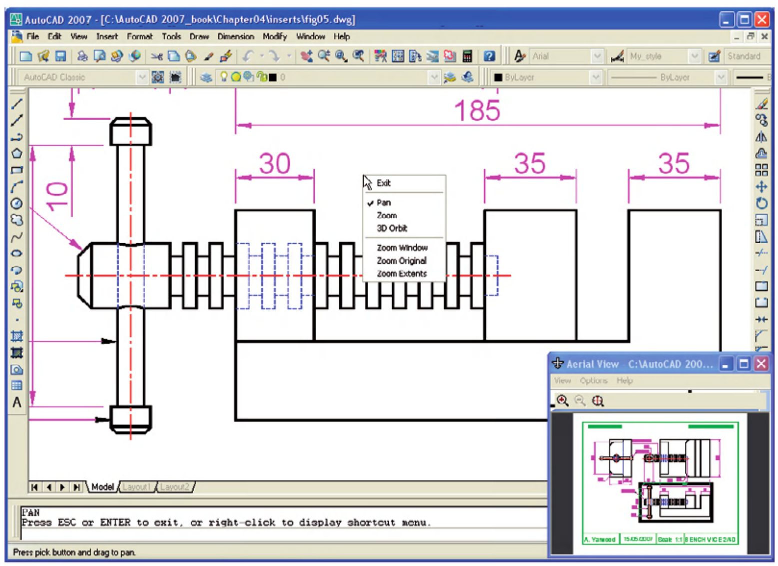


Fig. 4.8 The **Pan** tool in action showing a part of the drawing, while the whole drawing is shown in the **Aerial View** window

Notes

1. If using a mouse with a wheel both zooms and pans can be performed with the aid of the wheel. See page 6.
2. The **Zoom** tools are important in that they allow even the smallest parts of drawings to be examined and, if necessary, amended or modified.
3. The zoom tools can be called from the **Zoom** sub-menu of the **View** drop-down menu or by *entering* **zoom** at the command line. But easiest of all is to *enter* **z** at the command line followed by a *right-click*.

4. Similarly the easiest method of calling the **Pan** tool is to *enter* **p** at the command line followed by a *right-click*.
5. When constructing large drawings, the **Pan** tool and the **Aerial View** window are of value for allowing work to be carried out in any part of a drawing, while showing the whole drawing in the **Aerial View** window.

Drawing templates

In Chapters 1–3, drawings were constructed in the template **acadiso.dwt** which loaded when AutoCAD 2007 was opened. Now the default **acadiso** template is amended with **Limits** set to **420,297** (coordinates within which a drawing can be constructed), **Grid** set to **10, Snap** set to **5**, and the drawing area **Zoomed** to **All**.

Throughout this book most drawings will be based on an **A3** sheet, which measures 420 units by 297 units (the same as the **Limits**).

Note

As mentioned on page 16 if there are multiple users to the computer on which drawings are being constructed, it is as well to save the template being used to another file name or, if thought necessary, to a floppy disk. A file name **My_template.dwt**, as suggested earlier, or a name such as **book_template** can be given.

Adding features to the template

Four other features will now be added to our template:

1. **Text style** – set in the **Text Style** dialog.
2. **Dimension style** – set in the **Dimension Style Manager** dialog.
3. **Shortcutmenu variable** – set to **0**.
4. **Layers** – set in the **Layer Properties Manager** dialog.

Setting Text

1. At the command line:

   Command: *enter* **st** (Style) *right-click*

2. The **Text Style** dialog appears (Fig. 4.9). In the dialog, *enter* **6** in the **Height** field. Then *left-click* on **Arial** in the **Font Name** popup list. **Arial** font letters appear in the **Preview** area of the dialog.
3. *Left-click* the **New . . .** button and *enter* **ARIAL** in the **New Text Style** sub-dialog which appears (Fig. 4.10) and *click* the **OK** button.
4. *Left-click* the **Close** button of the **Text Style** dialog.

Setting Dimension style

Settings for dimensions require making *entries* in a number of sub-dialogs in the **Dimension Style Manager**. To set the dimensions style:

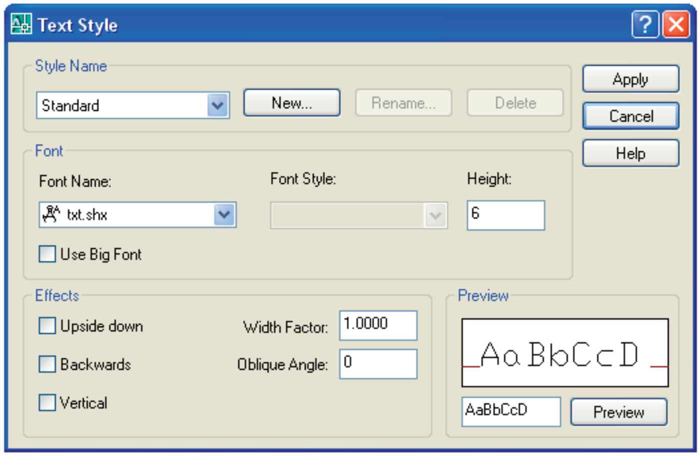


Fig. 4.9 The **Text Style** dialog

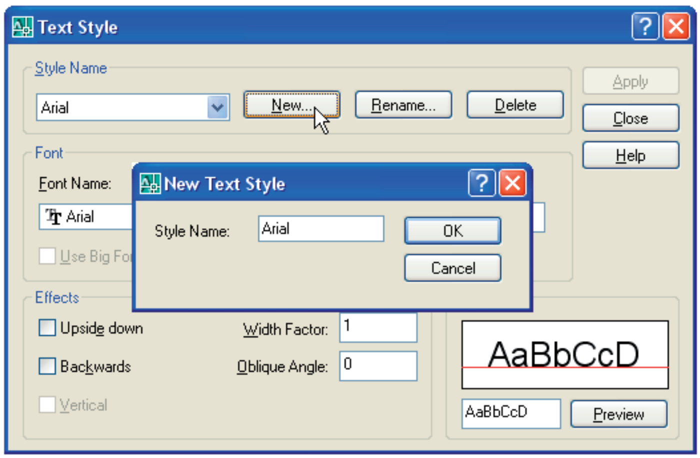


Fig. 4.10 The **New Text Style** sub-dialog

1. At the command line:

   Command: *enter* **d** *right-click*

   And the **Dimensions Style Manager** dialog appears (Fig. 4.11).
2. In the dialog, *click* the **New** . . . button. In the **Create New Dimension Style** sub-dialog which appears, *enter* **my_style** in the **New Style Name** field, followed by a *click* on the sub-dialog's **Continue** button.
3. The **New Dimension Style** sub-dialog appears (Fig. 4.12). In the dialog make settings as shown. Then *click* the **OK** button of that dialog.
4. The original **Dimension Style Manager** reappears. *Click* its **Modify** button.
5. In the **Modify Dimension Style** sub-dialog which appears (Fig. 4.13), *click* the **Text** tab at the top of the dialog. Then *click* the arrow to the right of the **Text style** field and select **Arial** from the popup list. *Enter* a height of **6** in the **Text height** field and **2** in the **Offset from dim line** field.

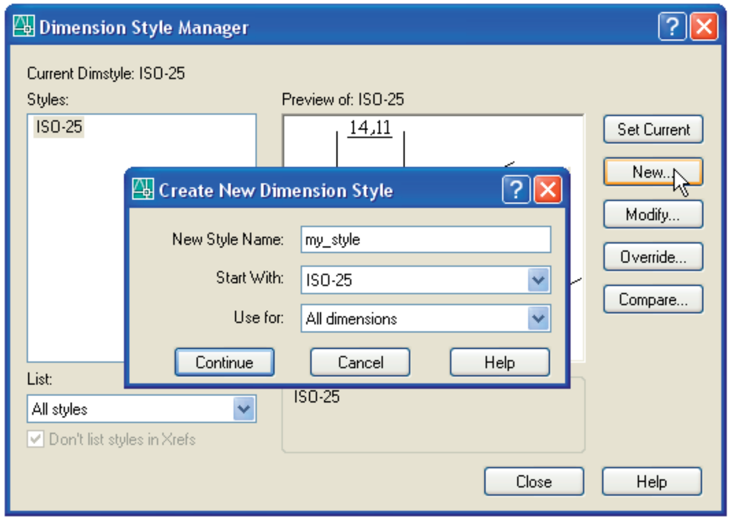


Fig. 4.11 The **Dimension Style Manager** dialog

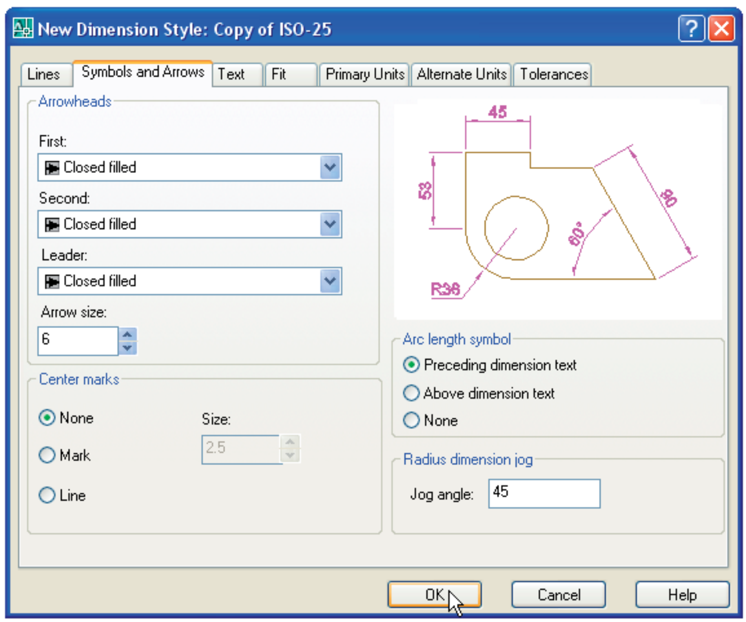


Fig. 4.12 Setting symbols and arrows in the **New Dimension Style** sub-dialog

6. Then *click* the **Primary Units** tab and set the units **Precision** in both **Linear** and **Angular dimensions** to **0**, that is no units after decimal point. *Click* the sub-dialogs **OK** button (Fig. 4.14). The **Dimension Style Manager** dialog reappears showing dimensions, as they will appear in a drawing, in the **Preview of: my_style** box. *Click* the **Set Current** button, followed by another *click* on the **Close** button (Fig. 4.15).

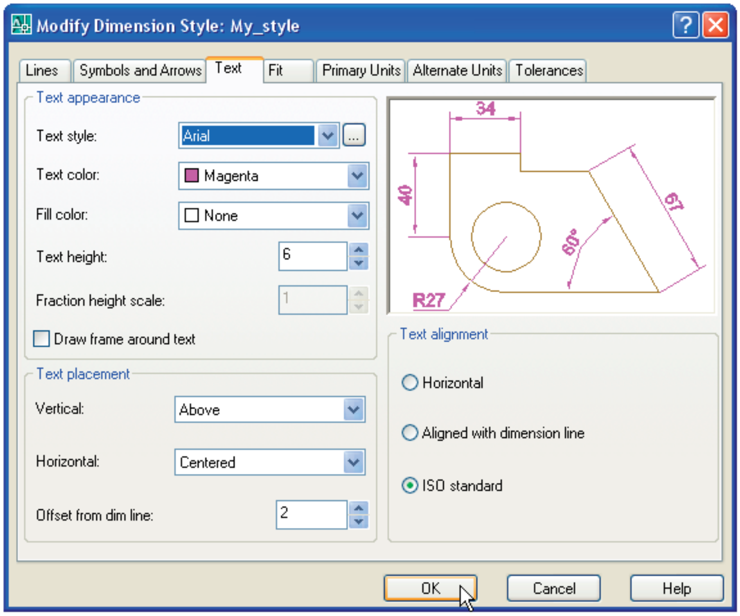


Fig. 4.13 Setting text style and height in the **Text** sub-dialog

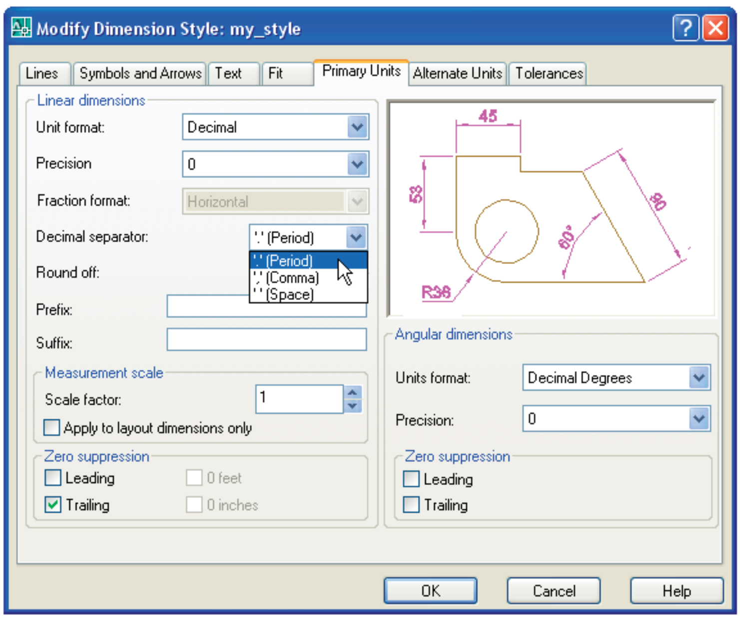


Fig. 4.14 Setting units in the **Primary Units** sub-dialog

Setting the shortcutmenu variable

Call the line tool, draw a few lines and then *right-click*. The *right-click* menu shown in Fig. 4.16 may well appear. The menu will also appear when any tool is called. Some operators prefer using this menu when constructing drawings. To stop this menu appearing:

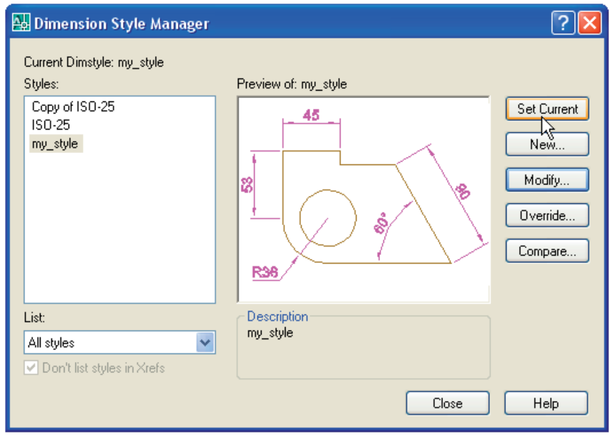


Fig. 4.15 *Click* the **Set Current** button

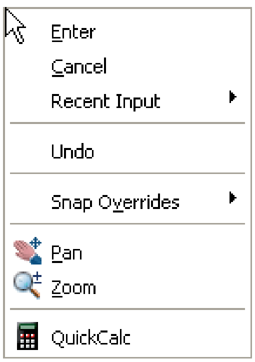


Fig. 4.16 The *right-click* menu

Command: *enter* **shortcutmenu** *right-click*
Enter new value for SHORTCUTMENU <12>: 0
Command:

And the menu will no longer appear when a tool is in action.

Setting Layers (see also page 121)

1. *Left-click* on the **Layer Properties Manager** tool icon in the **Layers** toolbar (Fig. 4.17).
   The **Layer Properties Manager** dialog appears on screen (Fig. 4.18).
2. *Click* the **New Layer** icon. A new layer appears. Overwrite the name **Layer1** by *entering* **Centre**.
3. Repeat step **2** four times and make four more layers titled **Construction, Dimensions, Hidden** and **Text**.

Fig. 4.17 The **Layer Properties Manager** tool icon in the **Layers** toolbar

4. *Click* against one of the squares under the **Color** column of the dialog. The **Select Color** dialog appears (Fig. 4.19). *Double-click* on one of the colours in the **Index Color** squares. The selected colour appears against the layer name in which the square was selected. Repeat until all the five new layers have a colour.
5. *Click* on the linetype **Continuous** against the layer name **Centre**. The **Select Linetype** dialog appears (Fig. 4.20). *Click* its **Load** . . . button and from the **Load or Reload Linetypes** dialog *double-click* **CENTER2**. The dialog disappears and the name appears in the **Select Linetype** dialog. *Click* the **OK** button and the linetype **CENTER2** appears against the layer **Centre**.

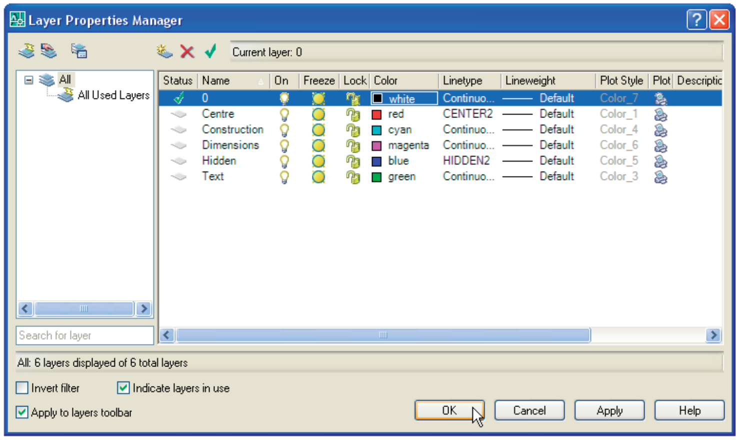


Fig. 4.18 The **Layer Properties Manager** dialog

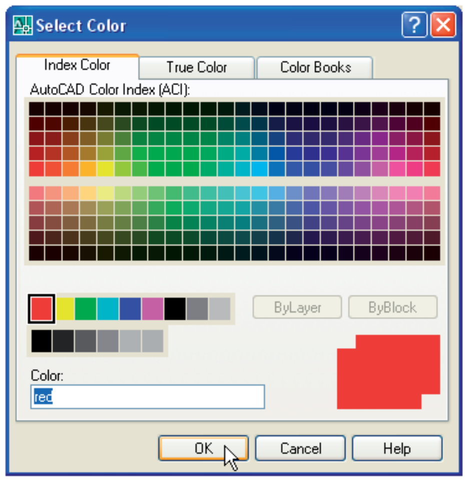


Fig. 4.19 The **Select Color** dialog

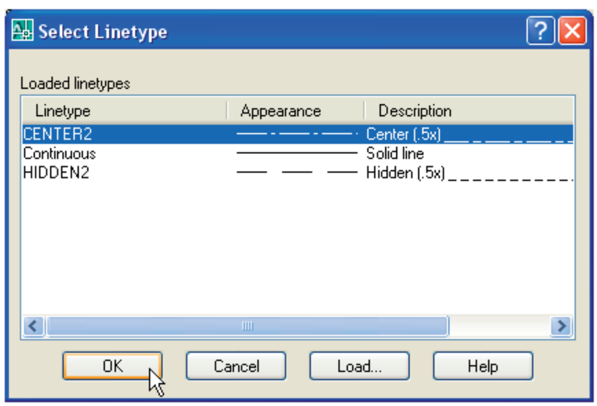


Fig. 4.20 The **Select Linetype** dialog

6. Repeat this with layer **Hidden** – load the linetype **HIDDEN2** and make the linetype against this layer **HIDDEN2**.
7. *Click* on any of the lineweights in the **Layer Properties Manager**. This brings up the **Lineweight** dialog (Fig. 4.21). Select the lineweight **0.3**. Repeat the same for all other layers. Then *click* the **OK** button of the **Layer Properties Manager**.

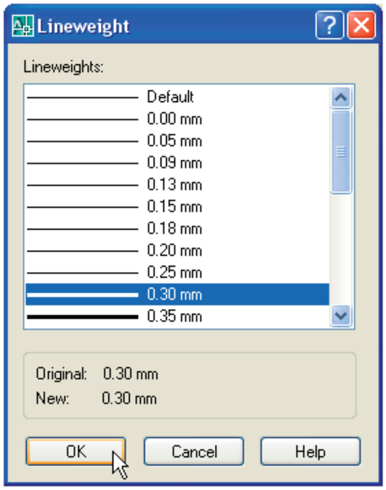


Fig. 4.21 The **Lineweight** dialog

Saving the template file

1. *Left-click* **Save As** . . . in the **File** drop-down menu.
2. In the **Save Drawing As** dialog which comes on screen (Fig. 4.22), *click* the arrow to the right of the **Files of type** field and in the popup

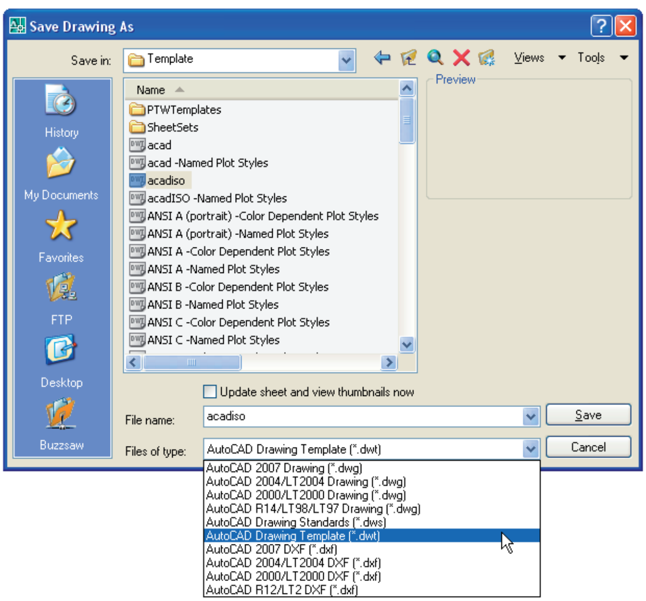


Fig. 4.22 Saving the template to the name **acadiso.dwt**

list associated with the field *click* on **AutoCAD Drawing Template (*.dwt)**. The list of template files in the **AutoCAD 2007/Template** directory appears in the file list.

3. *Click* on **acadiso** in the file list, followed by a *click* on the **Save** button.
4. A **Template Description** dialog appears. Make *entries* as indicated in Fig. 4.23, making sure that **Metric** is chosen from the popup list.

The template can now be saved to be opened for the construction of drawings as needed.

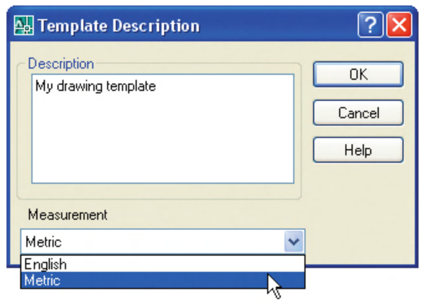


Fig. 4.23 The **Template Description** dialog

When AutoCAD 2007 is opened the template **acadiso.dwt** fills the drawing area.

Note

Please remember that if there are multiple users to the computer it is advisable to save the template to a name of your own choice.

Another template

A template A3_template.dwt – Fig. 4.26

In the **Select Template** dialog a *click* on any of the file names causes a preview of the template to appear in the **Preview** box of the dialog, unless the template is free of information – as is **acadiso.dwt**. To construct another template which includes a title block and other information based on the **acadiso.dwt** template:

1. In an **acadiso.dwt** template construct the required border, title block etc.
2. *Click* the **Layout1** tab in the status bar (Fig. 4.24). The screen is now in a **Paper Space** setting.
3. *Click* the **Maximize Viewport** icon in the status bar (Fig. 4.25).
4. **Zoom** to **Extents**.
5. It is suggested this template be saved as a **Paper Space** template with the name **A3_template.dwt**.

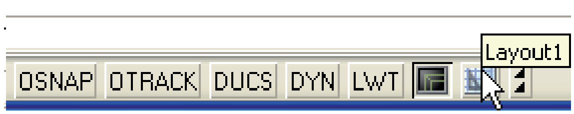


Fig. 4.24 The **Layout1** button on the status bar

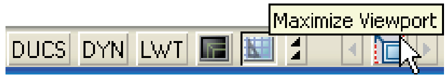


Fig. 4.25 The **Maximize Viewport** icon in the status bar

Notes

1. The outline for this template is a pline from **0,290** to **420,290** to **420,0** to **0,0** to **290,0** and of width **0.5** (Fig. 4.26).
2. The upper line of the title block is a pline from **0,20** to **420,20**.
3. **Pspace** is two-dimensional.
4. Further uses for **Layouts** and **Pspace** are given in Chapter 20.

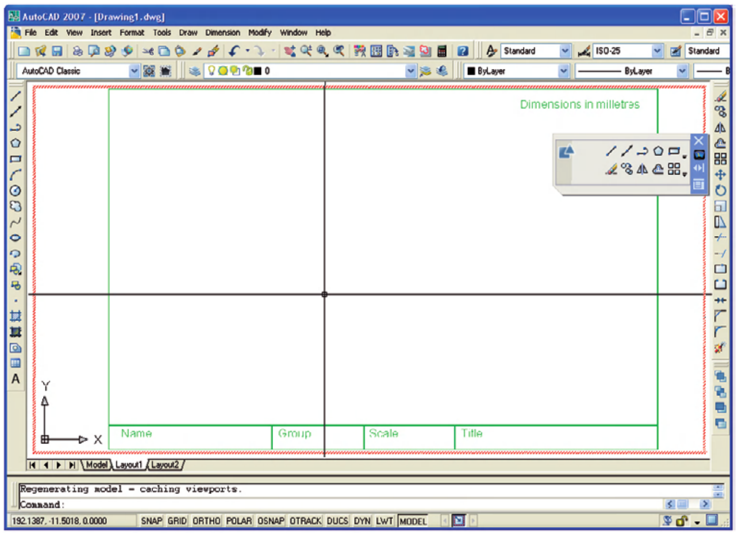


Fig. 4.26 The **A3_template.dwt**

Revision notes

1. The **Zoom** tools are important in that they allow even the smallest parts of drawings to be examined and, if necessary, amended or modified.
2. The **Zoom** tools can be called from the **Zoom** sub-menu of the **View** drop-down menu, or by *entering* **z** or **zoom** at the command line. The easiest is to *enter* **z** at the command line followed by a *right-click*.
3. There are four methods of calling tools for use – selecting a tool icon from a toolbar or from the **DASHBOARD** palette; *entering* the name of a tool in full at the command line; *entering* an abbreviation for a tool; or selecting from a drop-down menu.
4. When constructing large drawings, the **Pan** tool and the **Aerial View** window are of value for allowing work to be carried out in any part of a drawing, while showing the whole drawing in the **Aerial View** window.
5. An A3 sheet of paper is 420 mm by 297 mm. If a drawing constructed in the template **acadiso.dwt** is printed/plotted full size (scale 1:1), each unit in the drawing will be 1 mm in the print/plot.
6. When limits are set it is essential to call **Zoom** followed by **a** (All) to ensure that the limits of the drawing area are as set.
7. The *right-click* menu which appears when using tools can be aborted if required by setting the **SHORTCUTMENU** variable to **0**.

CHAPTER 5

The Modify tools

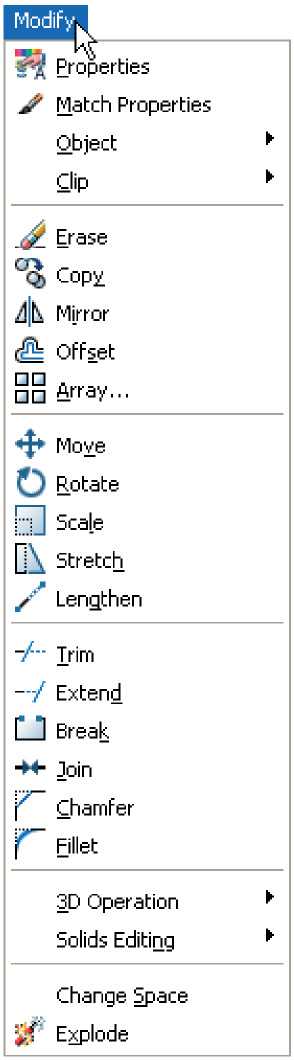


Fig. 5.1 The **Modify** tools in the **Modify** drop-down menu

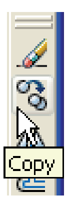


Fig. 5.2 The **Copy** tool in the **Modify** toolbar

Aim of this chapter

To describe the uses of tools for modifying parts of drawings.

Introduction

These tools are among the most frequently used tools of AutoCAD 2007. Their tool icons are found in the **Modify** drop-down menu (Fig. 5.1) or in the **Modify** toolbar in the **2D Classic AutoCAD** workspace (Fig. 5.2) or in the lower set of tool icons in the **Draw control panel**.

Using the **Erase** tool was described in Chapter 2. Examples of tools other than the **Explode** tool follow. See Chapter 10 for **Explode**.

The Copy tool

Example – Copy (Fig. 5.5)

1. Construct Fig. 5.3 using **Polyline**. Do not include the dimensions.
2. Call the **Copy** tool – either *left-click* on its tool icon in the **Modify** toolbar (Fig. 5.2), or *pick* **Copy** from the **Modify** drop-down menu, or *enter* **cp** or **copy** at the command line, or *left-click* its tool icon in the **2D Draw control panel** (Fig. 5.4). The command line shows:

Command:_copy
Select objects: *pick* the cross **1 found**
Select objects: *right-click*
Specify base point or [Displacement] <Displacement>: end **of** *pick*
Specify second point of displacement or <use first point as displacement>: *pick*
Specify second point or [Exit/Undo] <Exit>: *right-click*
Command:

The result is given in Fig. 5.5.

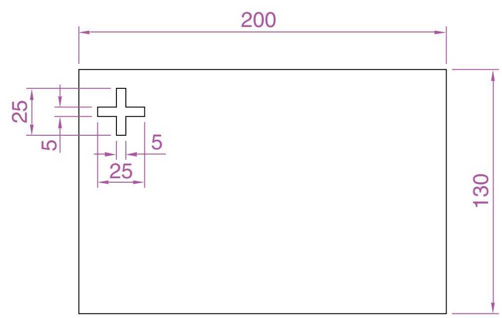


Fig. 5.3 Example – **Copy** – outlines

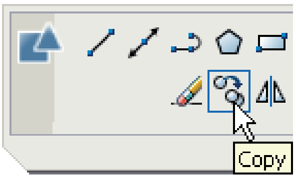


Fig. 5.4 The **Copy** tool icon from the **2D Draw control panel**

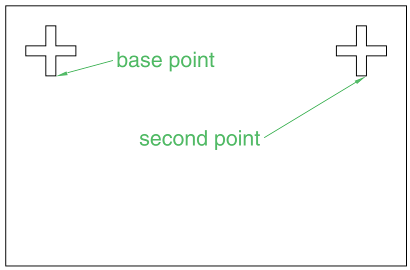


Fig. 5.5 Example – **Copy**

Example – Multiple copy (Fig. 5.6)

1. Erase the copied object.
2. Call the **Copy** tool. The command line shows:

Command:_copy
Select objects: *pick* **1 found**
Select objects: *right-click*
Specify base point or [Displacement] <Displacement>: *pick*
Specify second point of displacement or <use first point as displacement>: *pick*
Specify second point of displacement or <Exit/Undo>: *pick*
Specify second point of displacement or <Exit/Undo>: *right-click*
Command:

The result is shown in Fig. 5.6.

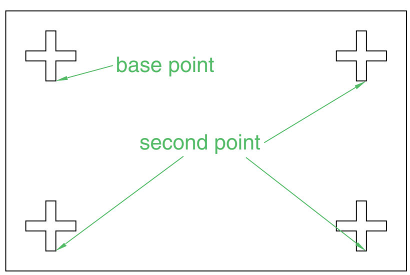


Fig. 5.6 Example – **Multiple Copy**

The Mirror tool

First example – Mirror (Fig. 5.9)

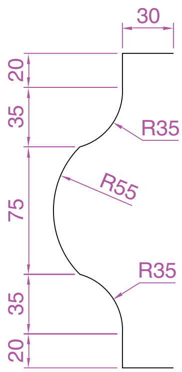


Fig. 5.7 First example – **Mirror** – outline

1. Construct the outline in Fig. 5.7 using **Line** and **Arc** tools.
2. Call the **Mirror** tool – either *left-click* on its tool icon in the **Modify** toolbar (Fig. 5.8), or *pick* **Mirror** from the **Modify** drop-down menu, or *enter* **mi** or **Mirror** at the command line. The command line shows:

Command:_mirror
Select objects: *pick* first corner **Specify opposite corner**: *pick* **7 found**
Select objects: *right-click*
Specify first point of mirror line: end **of** *pick*
Specify second point of mirror line: end **of** *pick*
Delete source objects [Yes/No] <N>: *right-click*
Command:

The result is shown in Fig. 5.9.

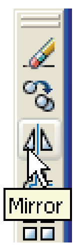


Fig. 5.8 The **Mirror** tool icon from the **Modify** toolbar

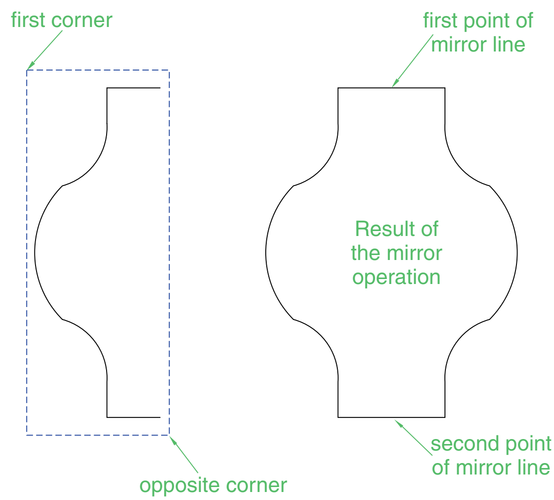


Fig. 5.9 First example – **Mirror**

Second example – Mirror (Fig. 5.10)

1. Construct the outline shown in the dimensioned polyline in the top drawing of Fig. 5.10.
2. Call **Mirror** and using the tool three times complete the given outline. The two points shown in Fig. 5.10 are to mirror the right-hand side of the outline.

Third example – Mirror (Fig. 5.11)

If text is involved when using the **Mirror** tool, the set variable **MIRRTEXT** must be set correctly. To set the variable:

Command: mirrtext
Enter new value for MIRRTEXT <1>: 0
Command:

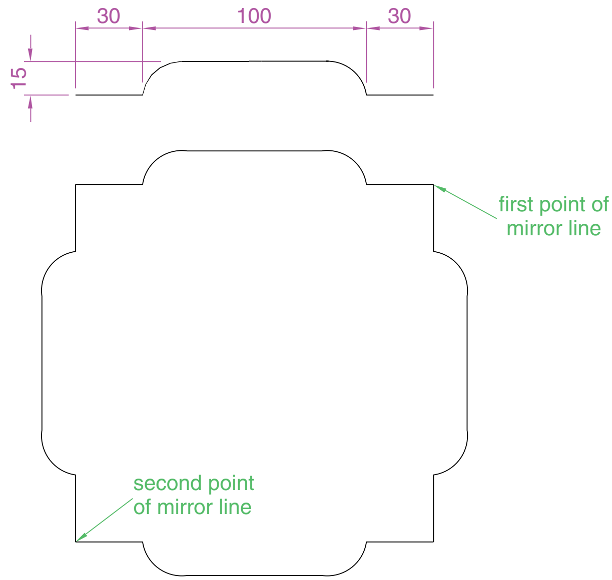


Fig. 5.10 Second example – **Mirror**

Fig. 5.11 Third example – **Mirror**

If set to **0** text will mirror without distortion. If set to **1** text will read backwards as indicated in Fig. 5.11.

The Offset tool

Examples – Offset (Fig. 5.14)

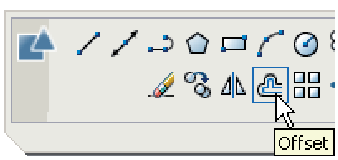


Fig. 5.12 The **Offset** tool from the **2D Draw control panel**

1. Construct the four outlines shown in Fig. 5.13.
2. Call the **Offset** tool – either *left-click* its tool icon in the **2D Draw control panel** (Fig. 5.12), or *pick* the tool name in the **Modify** drop-down menu, or *enter* **o** or **offset** at the command line. The command line shows:

   Command:_offset
   Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0
   Specify offset distance or [Through/Erase/Layer] <Through>: 10
   Select object to offset or [Exit/Undo] <Exit>: *pick* drawing **1**
   Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: *pick* inside the rectangle
   Select object to offset or [Exit/Undo] <Exit>: *right-click*
   Command:

3. Repeat the same for drawings **2, 3** and **4** in Fig. 5.13 as shown in Fig. 5.14.

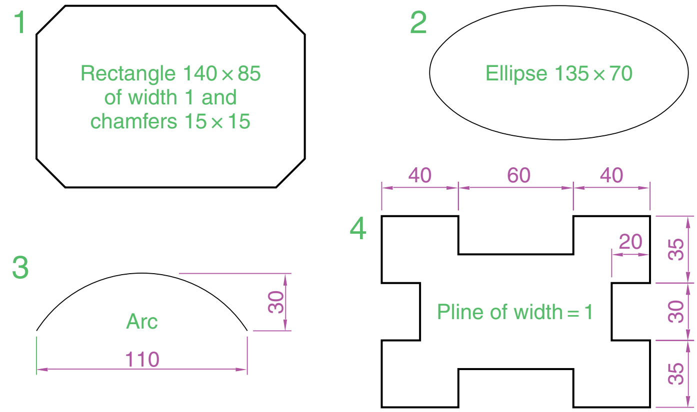


Fig. 5.13 Examples – **Offset** – outlines

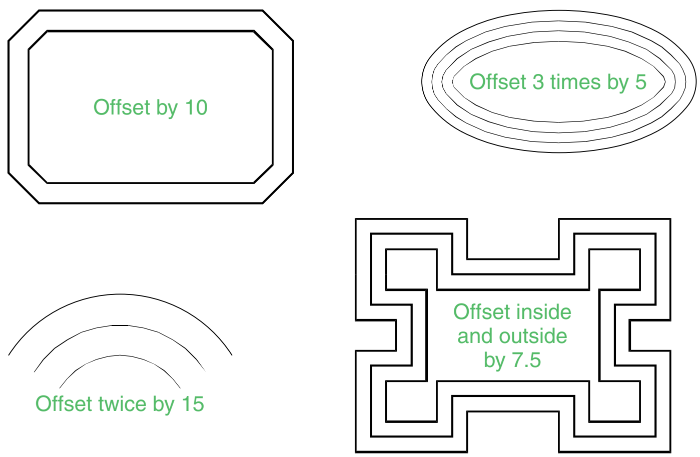


Fig. 5.14 Examples – **Offset**

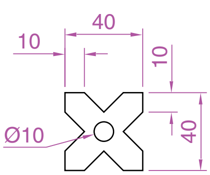


Fig. 5.15 First example – **Array** – drawing to be arrayed

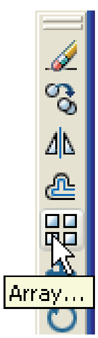


Fig. 5.16 The **Array** tool icon from the **Modify** toolbar

The Array tool

Arrays can be in either a **Rectangular** form or a **Polar** form as shown in the examples below.

First example – Rectangular Array (Fig. 5.17)

1. Construct the drawing in Fig. 5.15.
2. Call the **Array** tool – either *left-click* the **Array** tool icon in the **Modify** toolbar (Fig. 5.16), or *pick* **Array** . . . from the **Modify** drop-down menu, or *enter* **ar** or **array** at the command line. No matter which method is used the **Array** dialog appears (Fig. 5.17).
3. Make settings in the dialog:

   Rectangular Array radio button set on (dot in button).
   Rows field – *enter* **5**
   Columns field – *enter* **6**

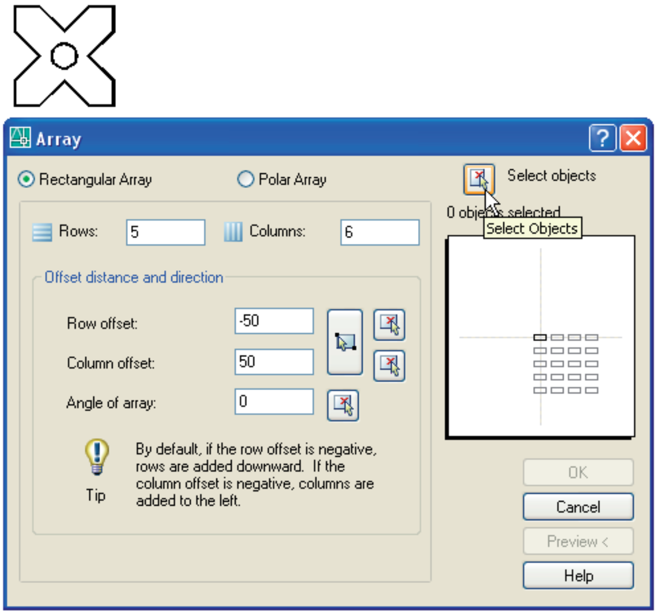


Fig. 5.17 First example – the **Array** dialog

Row offset field – *enter* **−50** (note the minus sign)
Column offset field – *enter* **50**

4. *Click* the **Select objects** button and the dialog disappears. *Window* the drawing. The dialog reappears.
5. *Click* the **Preview<** button. The dialog disappears and the array appears with a warning dialog in the centre of the array (Fig. 5.18).
6. If satisfied *click* the **Accept** button. If not *click* the **Modify** button and make revisions to the **Array** dialog fields.

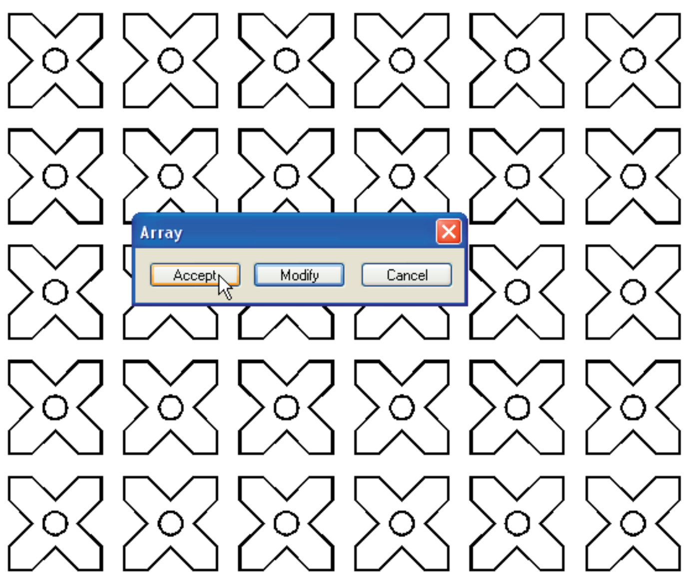


Fig. 5.18 First example – **Array**

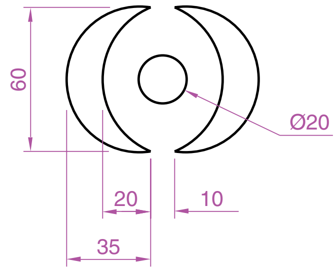


Fig. 5.19 Second example – **Array** – drawing to be arrayed

Second example – Polar Array (Fig. 5.22)

1. Construct the drawing in Fig. 5.19.
2. Call **Array**. The **Array** dialog appears. Make settings as shown in Fig. 5.20.

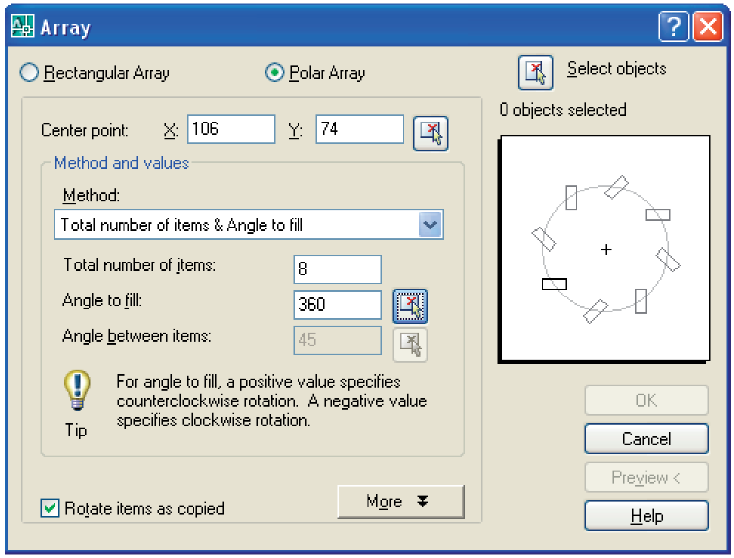


Fig. 5.20 Second example – **Array** – settings in the dialog

3. *Click* the **Select objects** button of the dialog and *window* the drawing. The dialog returns to screen. *Click* the **Pick Center Point** button (Fig. 5.21) and when the dialog disappears, *pick* a centre point for the array.

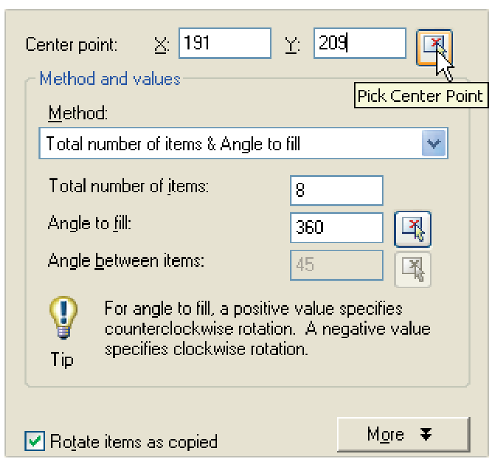


Fig. 5.21 Second example – **Array** – the **Pick Center Point** button

4. The dialog reappears. *Click* its **Preview<** button and when the array appears with its warning dialog, if satisfied with the result, *click* the **Accept** button of this dialog (Fig. 5.22).

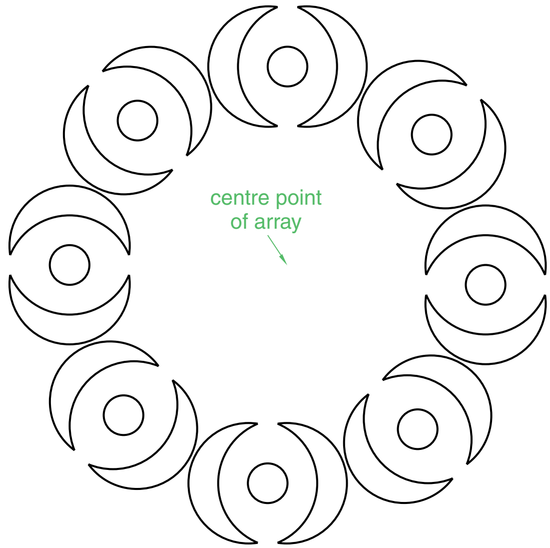


Fig. 5.22 Second example – **Array**

The Move tool

Example – Move (Fig. 5.25)

1. Construct the drawing in Fig. 5.23.

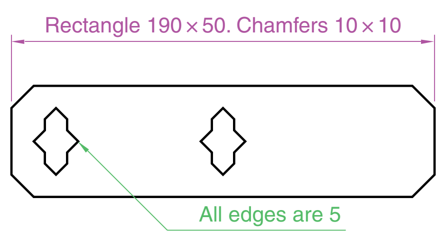


Fig. 5.23 Example – **Move** – drawing

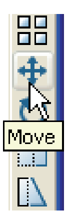


Fig. 5.24 The **Move** tool from the **Modify** toolbar

2. Call **Move** – either *click* the **Move** tool icon in the **Modify** toolbar (Fig. 5.24), or *pick* **Move** from the **Modify** drop-down menu, or *enter* **m** or **move** at the command line, which then shows:

Command:_move
Select objects: *pick* the shape in the middle of the drawing **1 found**
Select objects: *right-click*
Specify base point or [Displacement] <Displacement>: *pick*
Specify second point or <use first point as displacement>: *pick*
Command:

The result is given in Fig. 5.25.

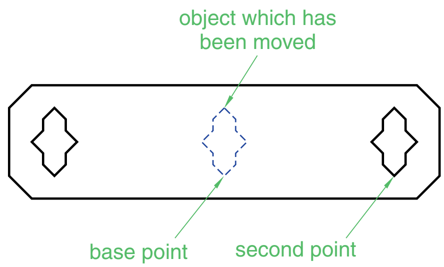


Fig. 5.25 Example – **Move**

The Rotate tool

When using the **Rotate** tool remember that the default rotation of objects within AutoCAD 2007 is counterclockwise (anticlockwise).

Example – Rotate (Fig. 5.27)

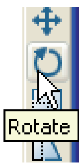


Fig. 5.26 The **Rotate** tool icon from the **Modify** toolbar

1. Construct drawing **1** of Fig. 5.27 with **Polyline**. Copy the drawing **1** three times – shown as drawings **2**, **3** and **4** in Fig. 5.27.
2. Call **Rotate** – either *left-click* its tool icon in the **Modify** toolbar (Fig. 5.26), or *pick* **Rotate** from the **Modify** drop-down menu, or *enter* **ro** or **rotate** at the command line. The command line shows:

   Command:_rotate
   Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0
   Select objects: *window* the drawing **3 found**
   Select objects: *right-click*
   Specify base point: *pick*
   Specify rotation angle or [Copy/Reference] <0>: 45
   Command:

   And the first copy rotates through the specified angle.
3. Repeat for drawings **3** and **4** rotating through angles as shown in Fig. 5.27.

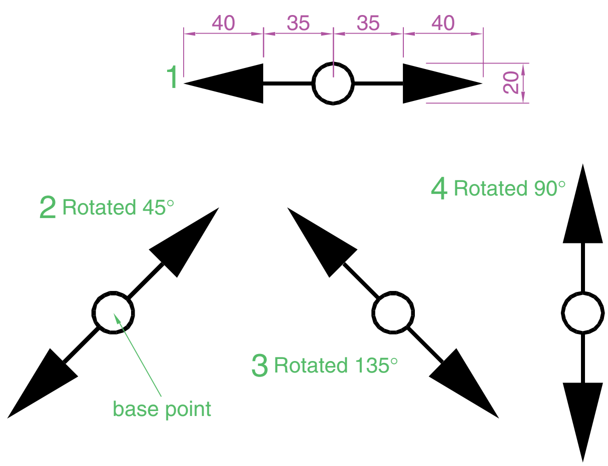


Fig. 5.27 Example – **Rotate**

The Scale tool

Examples – Scale (Fig. 5.29)

1. Using the **Rectangle** and **Polyline** tools, construct drawing **1** of Fig. 5.29. The **Rectangle** fillets are R10. The line width of all parts is **1**. Copy the drawing three times to give drawings **2**, **3** and **4**.
2. Call **Scale** – either *left-click* its tool icon in the **2D Draw control panel** (Fig. 5.28), or *pick* **Scale** from the **Modify** drop-down menu, or *click* its tool icon in the **Modify** toolbar, or *enter* **sc** or **scale** at the command line, which then shows:

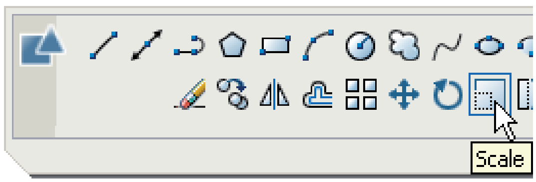


Fig. 5.28 The **Scale** tool icon from the **2D Draw control panel**

Command:_scale
Select objects: *window* drawing 2 **5 found**
Select objects: *right-click*
Specify base point: *pick*
Specify scale factor or [Copy/Reference] <1>: 0.75
Command:

3. Repeat for the other two drawings **3** and **4**, scaling them to the scales given with the drawings.

The results are shown in Fig. 5.29.

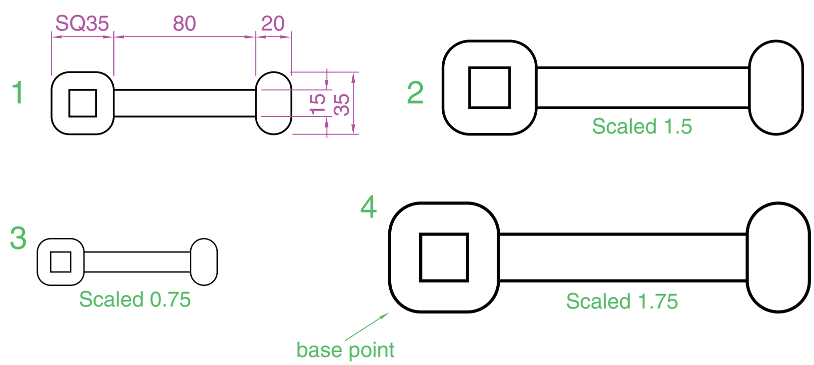


Fig. 5.29 Examples – **Scale**

The Trim tool

This tool is one which will be frequently used for the construction of drawings.

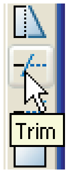


Fig. 5.30 The **Trim** tool icon from the **Modify** toolbar

First example – Trim (Fig. 5.31)

1. Construct the drawing **Original drawing** in Fig. 5.31.
2. Call **Trim** – either *left-click* its tool icon in the **Modify** toolbar (Fig. 5.30), or *pick* **Trim** from the **Modify** drop-down menu, or *enter* **tr** or **trim** at the command line, which then shows:

Command:_trim
Current settings: Projection UCS. Edge=None
Select cutting edges: *pick* the left-hand circle **1 found**
Select objects: *right-click*
Select objects to trim or shift-select to extend or [Fence/Project/Crossing/Edge/eRase/Undo]: *pick* one of the objects
Select objects to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: *pick* the second of the objects
Select objects to trim or shift-select to extend or [Project/Edge/Undo]: *right-click*
Command:

3. This completes the **First stage** as shown in Fig. 5.31. Repeat the **Trim** sequence for the **Second stage**.
4. The **Third stage** drawing of Fig. 5.31 shows the result of the trims at the left-hand end of the drawing.
5. Repeat for the right-hand end. The final result is shown in the drawing labelled **Result** in Fig. 5.31.

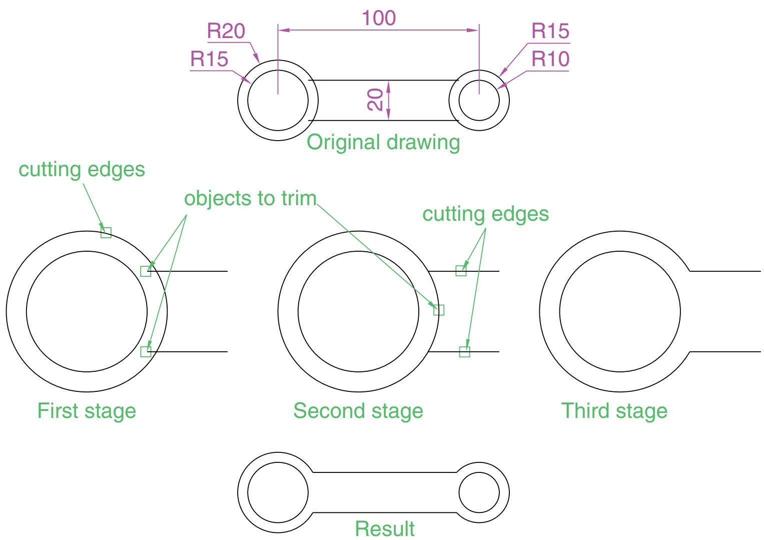


Fig. 5.31 First example – **Trim**

Second example – Trim (Fig. 5.32)

1. Construct the left-hand drawing of Fig. 5.32.
2. Call **Trim**. The command line shows:

Command:_trim
Current settings: Projection UCS. Edge=None

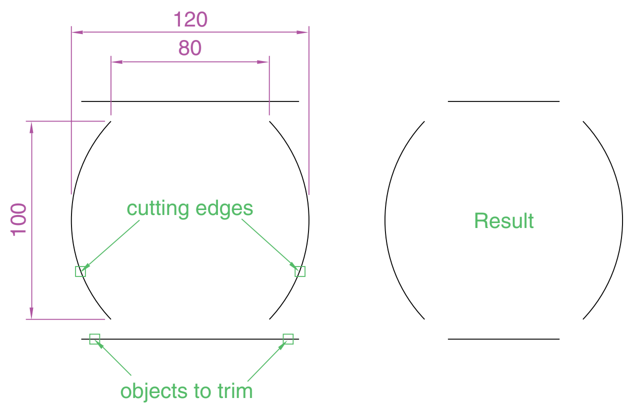


Fig. 5.32 Second example – **Trim**

Select cutting edges . . .
Select objects or <select all>: *pick* the left-hand arc **1 found**
Select objects: *right-click*
Select objects to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: e (Edge)
Enter an implied edge extension mode [Extend/No extend] <No extend>: e (Extend)
Select objects to trim: *pick*
Select objects to trim: *pick*
Select objects to trim: *right-click*
Command:

3. Repeat for the other required trims. The result is given in Fig. 5.32.

The Stretch tool

Examples – Stretch (Fig. 5.33)

As its name implies the **Stretch** tool is for stretching drawings or parts of drawings. The action of the tool prevents it from altering the shape of circles in any way. Only **crossing** or **polygonal** windows can be used to determine the part of a drawing which is to be stretched.

1. Construct the drawing labelled **Original** in Fig. 5.33, but do not include the dimensions. Use the **Circle, Arc, Trim** and **Polyline Edit** tools. The resulting outlines are plines of width=1. With the **Copy** tool make two copies of the drawing.

Note

In each of the three examples in Fig. 5.33, the broken lines represent the crossing windows required when **Stretch** is used.

2. Call the **Stretch** tool – either *click* on its tool icon in the **2D Draw control panel** (Fig. 5.34), or *left-click* on its tool icon in the **Draw** toolbar, or *pick* its name in the **Modify** drop-down menu, or *enter* **s** or **stretch** at the command line, which then shows:

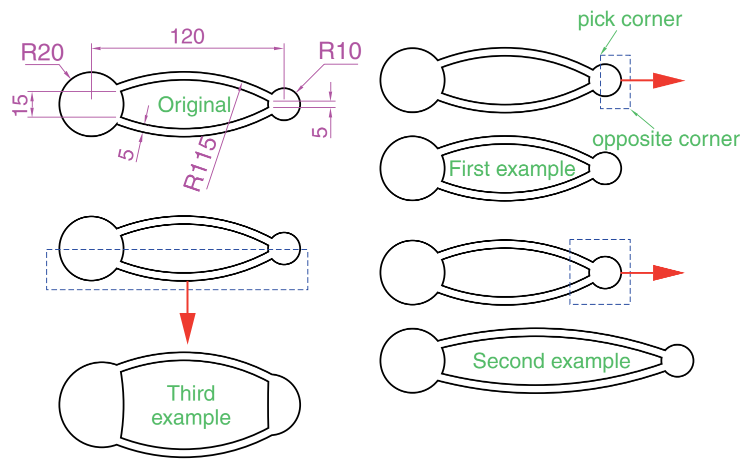


Fig. 5.33 Examples – **Stretch**

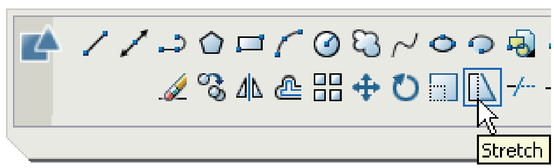


Fig. 5.34 The **Stretch** tool icon from the **2D Draw control panel**

Command:_stretch
Select objects to stretch by crossing-window or crossing-polygon . . .
Select objects: *enter* **c** *right-click*
Specify first corner: *pick* **Specify opposite corner**: *pick* **1 found**
Select objects: *right-click*
Specify base point or [Displacement] <Displacement>: *pick* beginning of arrow
Specify second point of displacement or <use first point as displacement>: *drag* in the direction of the arrow to the required second point and *right-click*
Command:

Notes

1. When circles are *windowed* with the crossing window, no stretching takes place. This is why, in the case of the first example in Fig. 5.33, when the **second point of displacement** was *picked*, there was no result – the outline did not stretch.
2. Care must be taken when using this tool as unwanted stretching can occur.

The Break tool

Examples – Break (Fig. 5.36)

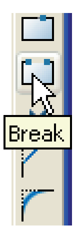


Fig. 5.35 The **Break** tool icon from the **Modify** toolbar

1. Construct the rectangle, arc and circle (Fig. 5.36).
2. Call **Break** – either *click* its tool icon in the **Modify** toolbar (Fig. 5.35), or *click* **Break** from the **Modify** drop-down menu, or *enter* **br** or **break** at the command line, which then shows as given below.

For drawings 1 and 2

Command:_break Select object *pick* at the point
Specify second break point or [First point]: *pick*
Command:

For drawing 3

Command:_break Select object *pick* at the point
Specify second break point or [First point]: *enter* **f** *right-click*
Specify first break point: *pick*
Specify second break point: *pick*
Command:

The results are shown in Fig. 5.36.

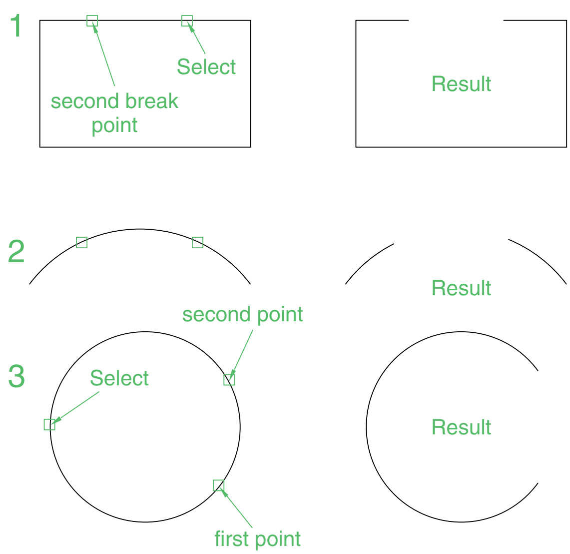


Fig. 5.36 Examples – **Break**

Note

Remember that the default rotation of AutoCAD 2007 is counterclockwise. This applies to the use of the **Break** tool.

The Join tool

The **Join** tool can be used to join plines providing their ends are touching; to join lines which are in line with each other; to join arcs; and to convert arcs to circles.

Examples – Join (Fig. 5.38)

1. Construct a rectangle from four separate plines – drawing **1** of Fig. 5.38; construct two lines – drawing **2** of Fig. 5.38 and an arc – drawing **3** of Fig. 5.38.

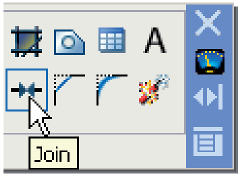


Fig. 5.37 The **Join** tool icon from the **2D Draw control panel**

2. Call the **Join** tool – either *click* the **Join** tool icon in the **2D Draw control panel** (Fig. 5.37), or *left-click* its tool icon in the **Modify** toolbar, or *enter* **join** or **j** at the command line. The command line shows:

   Command:_join Select source object:
   Select objects to join to source: *pick* a pline **1 found**
   Select objects to join to source: *pick* another **1 found, 2 total**
   Select objects to join to source: *pick* another **1 found, 3 total**
   Select objects to join to source: *right-click*
   3 segments added to polyline
   Command: *right-click*
   JOIN Select source object: *pick* one of the lines
   Select lines to join to source: *pick* the other **1 found**
   Select lines to join to source: *right-click*
   1 line joined to source
   Command: *right-click*
   JOIN Select source object: *pick* the *arc*
   Select arcs to join to source or [cLose]: *enter* **l** *right-click*
   Arc converted to a circle.
   Command:

The results are shown in Fig. 5.38.

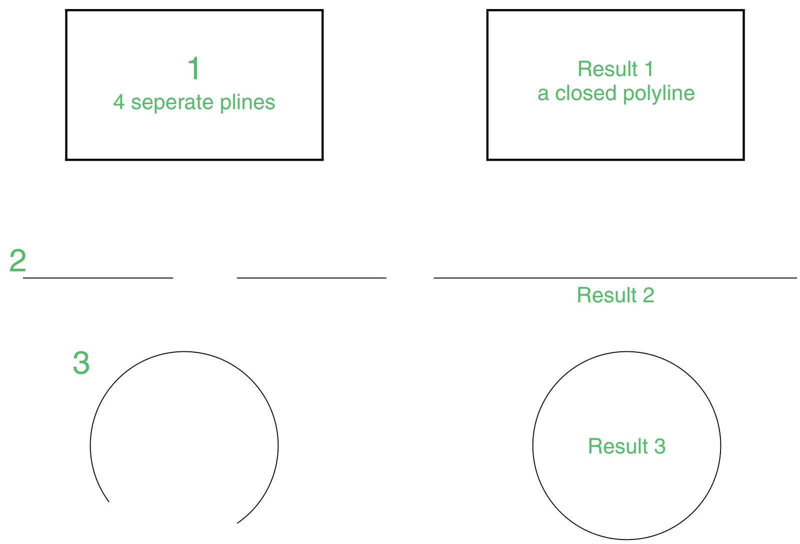


Fig. 5.38 Examples – **Join**

The Extend tool

Examples – Extend (Fig. 5.39)

1. Construct plines and a circles as shown in the left-hand drawings of Fig. 5.39.
2. Call **Extend** – either *click* the **Extend** tool icon in the **Modify** toolbar (Fig. 5.40), or *pick* **Extend** from the **Modify** drop-down

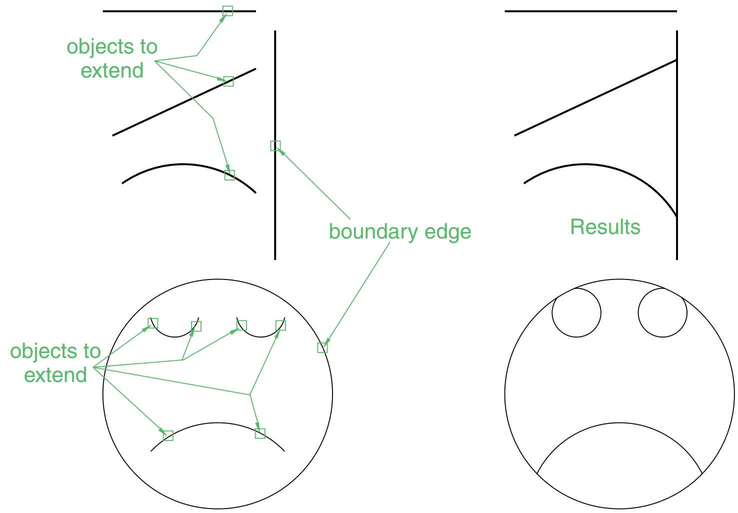


Fig. 5.39 Examples – **Extend**

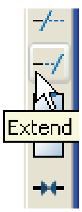


Fig. 5.40 The **Extend** tool icon from the **Modify** toolbar

menu, or *enter* **ex** or **extend** at the command line, which then shows:

Command:_extend
Current settings: Projection=UCS Edge=Extend
Select boundary edges . . .
Select objects or <select all>: <Snap off> *pick* **1 found**
Select objects: *pick*
Select objects: *right-click*
Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: *pick*

Repeat for each object to be extended. Then:

Select object to extend or shift-select to trim or [Project/Edge/Undo]: *right-click*
Command:

Note

Observe the similarity of the **Extend** and **No extend** prompts with those of the **Trim** tool.

The Chamfer and Fillet tools

There are similarities in the prompt sequences for these two tools. The major difference is that two settings (**Dist1** and **Dist2**) are required for a chamfer, but only one (**Radius**) for the fillet. The basic prompts for both are given below.

Chamfer

Command:_chamfer
(TRIM mode) Current chamfer Dist1=1, Dist2=1

Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: *enter* **d** (Distance) *right-click*
Specify first chamfer distance <1>: 10
Specify second chamfer distance <10>: *right-click*
Command:

Fillet

Command:_fillet
Current settings: Mode=TRIM, Radius=1
Select first object or [Polyline/Radius/Trim/mUltiple]: *enter* **r** (Radius) *right-click*
Specify fillet radius <1>: 15
Command:

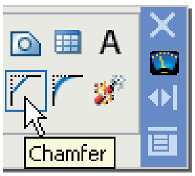


Fig. 5.41 The **Chamfer** tool icon from the **2D Draw control panel**

Examples – Chamfer (Fig. 5.42)

1. Construct three rectangles 100 by 60 using either the **Line** or the **Polyline** tool.
2. Call **Chamfer** – either *click* its tool icon in the **2D Draw control panel** (Fig. 5.41), or left-click on its tool icon in the **Modify** toolbar, or *pick* **Chamfer** from the **Modify** drop-down menu, or *enter* **cha** or **chamfer** at the command line which then shows:

Command:_chamfer
(TRIM mode) Current chamfer Dist1=1, Dist2=1
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: d (Distance)
Specify first chamfer distance <1>: 10
Specify second chamfer distance <10>: *right-click*
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: *pick* the first line for the chamfer
Select second line or shift-select to apply corner: *pick*
Command:

The other two rectangles are chamfered in a similar manner except that the **No trim** prompt is brought into operation with the bottom left-hand example (Fig. 5.42).

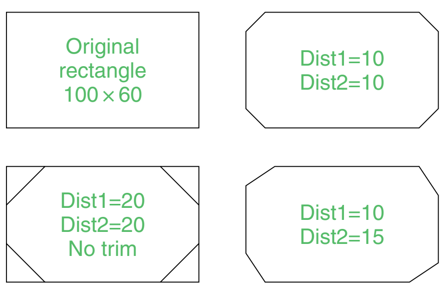


Fig. 5.42 Examples – **Chamfer**

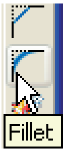


Fig. 5.43 The **Fillet** tool icon from the **Modify** toolbar

Examples – Fillet (Fig. 5.44)

1. Construct three rectangles as for the **Chamfer** examples.
2. Call **Fillet** – either *click* its tool icon in the **Modify** toolbar (Fig. 5.43), or *pick* **Fillet** from the **Modify** drop-down menu, or *left-click* its tool icon in the **2D Draw control panel**, or *enter* **f** or **fillet** at the command line which then shows:

Command:_fillet
Current settings: Mode=TRIM, Radius=1
Select first object or [Polyline/Radius/Trim/mUltiple]: r (Radius)
Specify fillet radius <1>: 15
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *pick*
Select second object or shift-select to apply corner: *pick*
Command:

Three examples are given in Fig. 5.44.

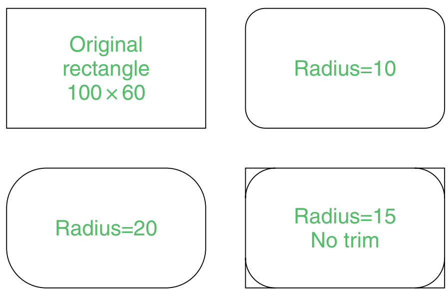


Fig. 5.44 Examples – **Fillet**

Revision notes

1. The **Modify** tools are among the most frequently used tools in AutoCAD 2007.
2. The abbreviations for the **Modify** tools are:
Copy – cp or **co**
Mirror – mi
Offset – o
Array – ar
Move – m
Rotate – ro
Scale – sc
Stretch – s
Trim – tr
Extend – ex
Break – br
Join – j
Chamfer – cha
Fillet – f

3. There are two other tools in the **Modify** toolbar or the **2D Draw control panel: Erase** – some examples were given in Chapter 2 and **Explode** – further details of this tool will be given in Chapter 10.

A note – selection windows and crossing windows

In the **Options** dialog, settings can be made in the **Selection** sub-dialog for **Visual Effects**. A *click* on the **Visual Effect Settings** . . . button brings up another dialog. If the **Area Selection Effect** settings are set on, a normal window from top left to bottom right will be coloured in a chosen colour (default blue). A crossing window – bottom left to top right – will be coloured red (default colour). Note also the highlighting – **Selection Preview Effect** allows objects to highlight if this feature is on. These settings are shown in Fig. 5.45.

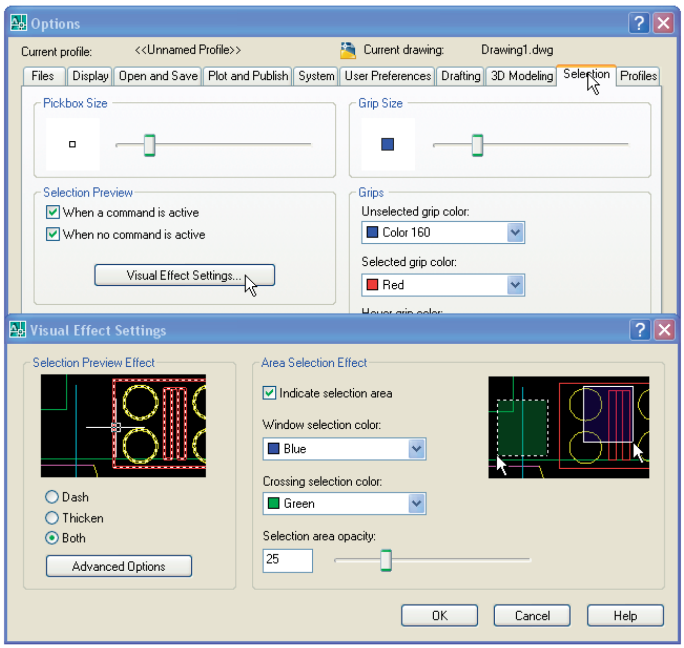


Fig. 5.45 **Visual Effect Settings** sub-dialog of the **Options** dialog

4. When using **Mirror**, if text is part of the area to be mirrored, the set variable **MIRRTEXT** will require setting – to either **1** or **0**.
5. With **Offset** the **Through** prompt can be answered by *clicking* two points in the drawing area – the distance of the desired offset distance.
6. **Polar Arrays** can be arrays around any angle set in the **Angle of array** field of the **Array** dialog.
7. When using **Scale**, it is advisable to practise the **Reference** prompt.
8. The **Trim** tool in either its **Trim** or its **No trim** modes is among the most useful tools in AutoCAD 2007.
9. When using **Stretch**, circles are unaffected by the stretching.

Exercises

1. Construct the drawing in Fig. 5.46. All parts are plines of width=0.7 with corners filleted R10. The long strips have been constructed using **Circle, Polyline, Trim** and **Polyline Edit**. Construct one strip and then copy it using **Copy**.

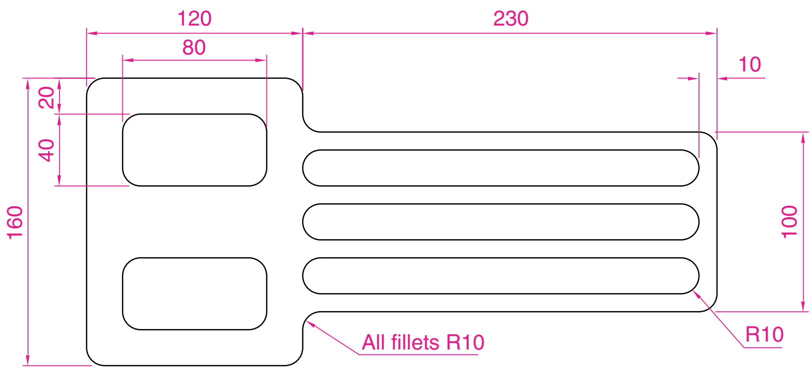


Fig. 5.46 Exercise 1

2. Construct the drawing in Fig. 5.47. All parts of the drawing are plines of width=0.7. The setting in the **Array** dialog is to be **180** in the **Angle of array** field.

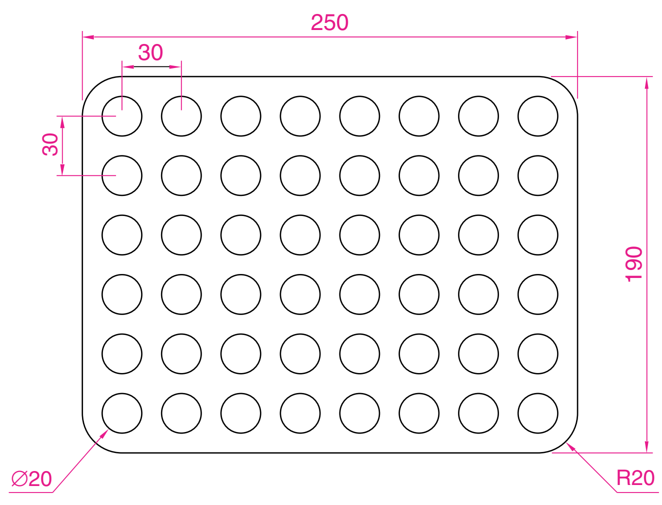


Fig. 5.47 Exercise 2

3. Using the tools **Polyline, Circle, Trim, Polyline Edit, Mirror** and **Fillet**, construct the drawing in Fig. 5.48.
4. Construct the circles and lines shown in Fig. 5.49. Using **Offset** and the **Ttr** prompt of the **Circle** tool, followed by **Trim**, construct one of the outlines arrayed within the outer circle. Then with **Polyline Edit** change the lines and arcs into a pline of width=0.3. Finally array the outline 12 times around the centre of the circles to produce Fig. 5.50.

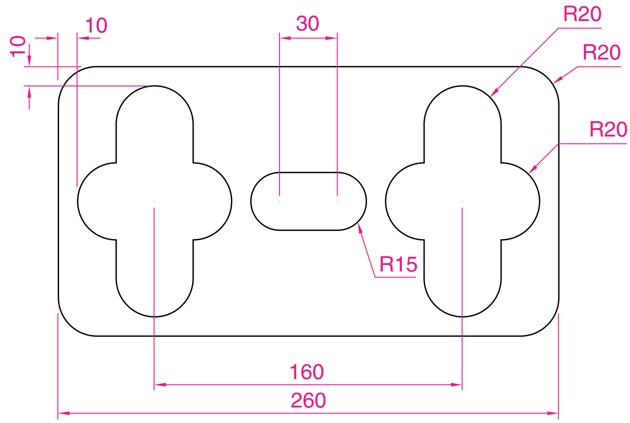


Fig. 5.48 Exercise 3

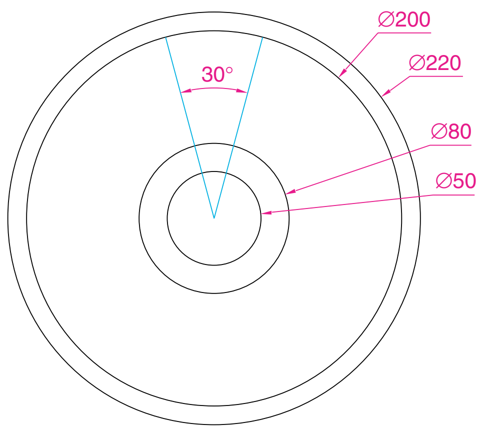


Fig. 5.49 Exercise 4 – circles and lines on which the exercise is based

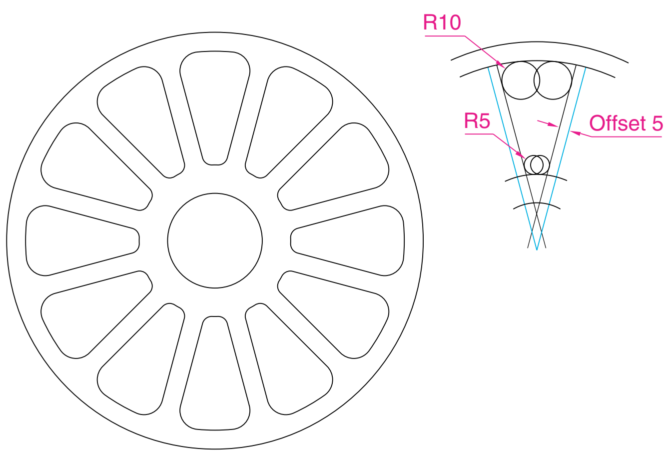


Fig. 5.50 Exercise 4

5. Construct the arrow (Fig. 5.51). Array the arrow around the centre of its circle eight times to produce the right-hand drawing of Fig. 5.51.

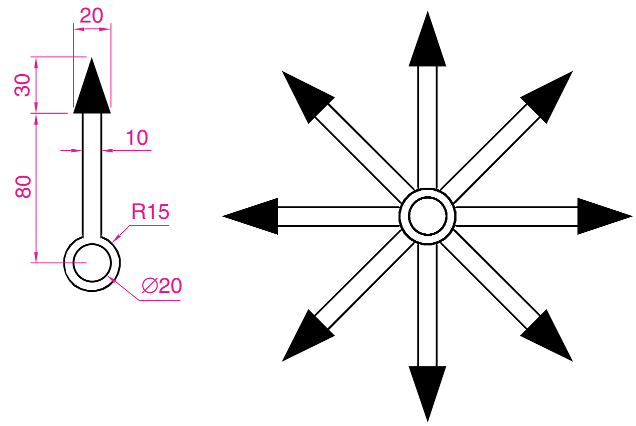


Fig. 5.51 Exercise 5

6. Construct the left-hand drawing of Fig. 5.52. Then, with **Move**, move the central outline to the top left-hand corner of the outer outline. Then with **Copy** make copies to the other corners.

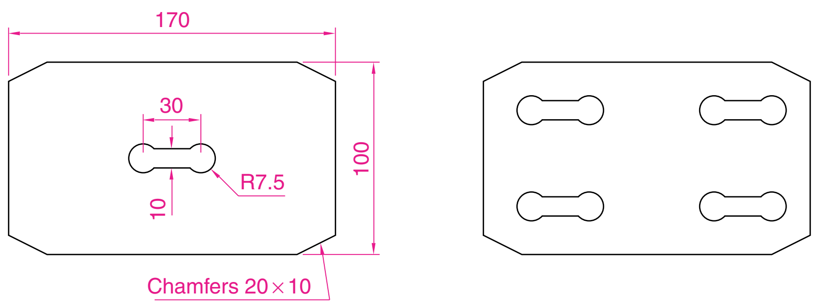


Fig. 5.52 Exercise 6

7. Construct the drawing in Fig. 5.53 and make two copies using **Copy**. With **Rotate**, rotate each of the copies to the angles as shown.

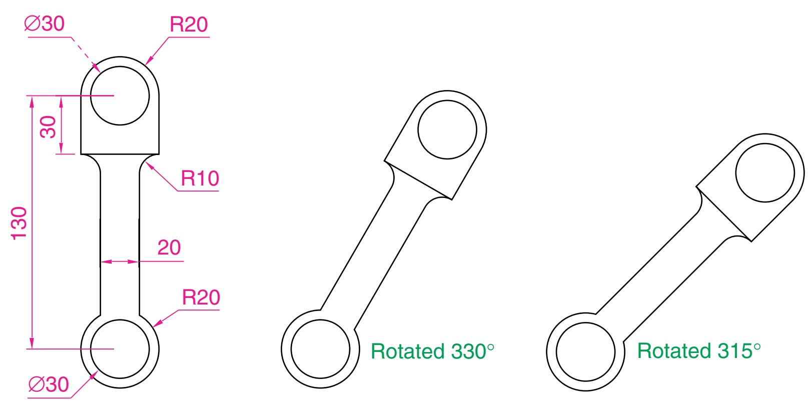


Fig. 5.53 Exercise 7

8. Construct the dimensioned drawing of Fig. 5.54. With **Copy**, copy the drawing. Then with **Scale** scale the drawing to a scale of **0.5**, followed by **Rotate** to rotate the drawing through an angle as shown. Finally scale the original drawing to a scale of **2:1**.

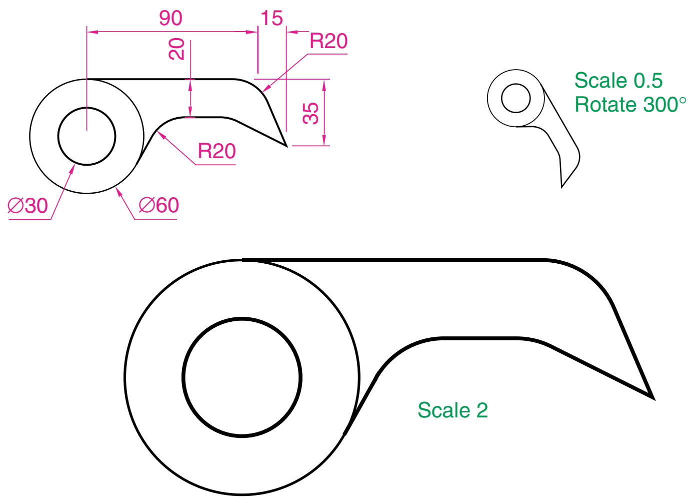


Fig. 5.54 Exercise 8

9. Construct the left-hand drawing of Fig. 5.55. Include the dimensions in your drawing. Then, using the **Stretch** tool, stretch the drawing, including its dimensions to the sizes as shown in the right-hand drawing. The dimensions are said to be **associative** (see Chapter 10).

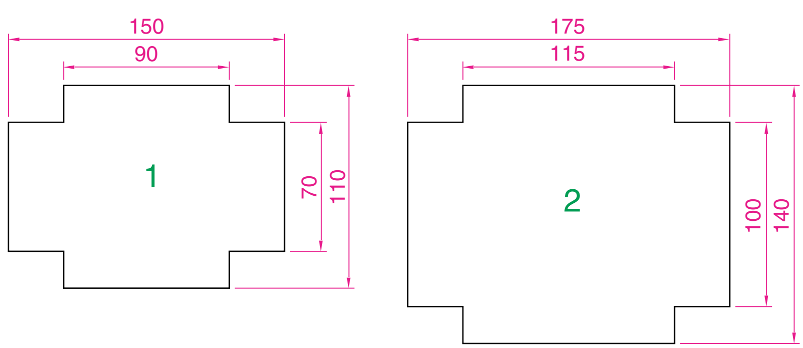


Fig. 5.55 Exercise 9

10. Construct the drawing in Fig. 5.56. All parts of the drawing are plines of width=0.7. The setting in the **Array** dialog is to be **180** in the **Angle of array** field.

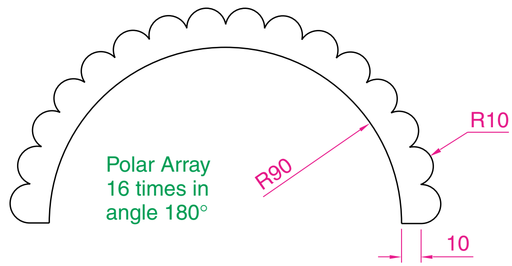


Fig. 5.56 Exercise 10

CHAPTER 6

Dimensions and Text

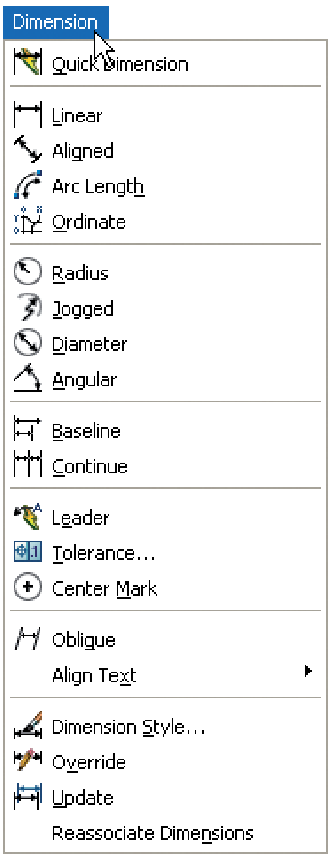


Fig. 6.1 The **Dimension** drop-down menu

Aims of this chapter

1. To describe the variety of methods of dimensioning drawings.
2. To describe methods of adding text to drawings.

Introduction

We have already set a dimension style (**My_Style**) in the **acadiso.dwt** template, so we can now commence adding dimensions to drawings using this dimension style.

The Dimension tools

There are several ways in which the dimension tools can be called.

1. *Click* **Dimension** in the menu bar. Tools can be selected from the drop-down menu which appears (Fig. 6.1).
2. See Fig. 6.2.
   (a) *Click* **Tools** in the menu bar, then *click* **Palettes** in the drop-down menu.
   (b) The **TOOL PALETTES – ALL PALETTES** palette appears. *Click* the **Command Tools** label.
   (c) *Drag* the resulting palette upwards from its bottom edge to produce a mini-palette containing the dimension tool icons.
   (d) Dimension tools can be selected from the flyout from the **Linear Dimension** icon in this palette.
3. From the **Dimension** toolbar (Fig. 6.3). The toolbar can be called to screen with a *right-click* in any toolbar on screen, followed by a *click* on **Dimension** in the popup menu which appears.
4. By *entering* an abbreviation for a dimension tool at the command line.

Any one of these methods can be used when dimensioning a drawing, but some operators may well decide to use a combination of the four methods.

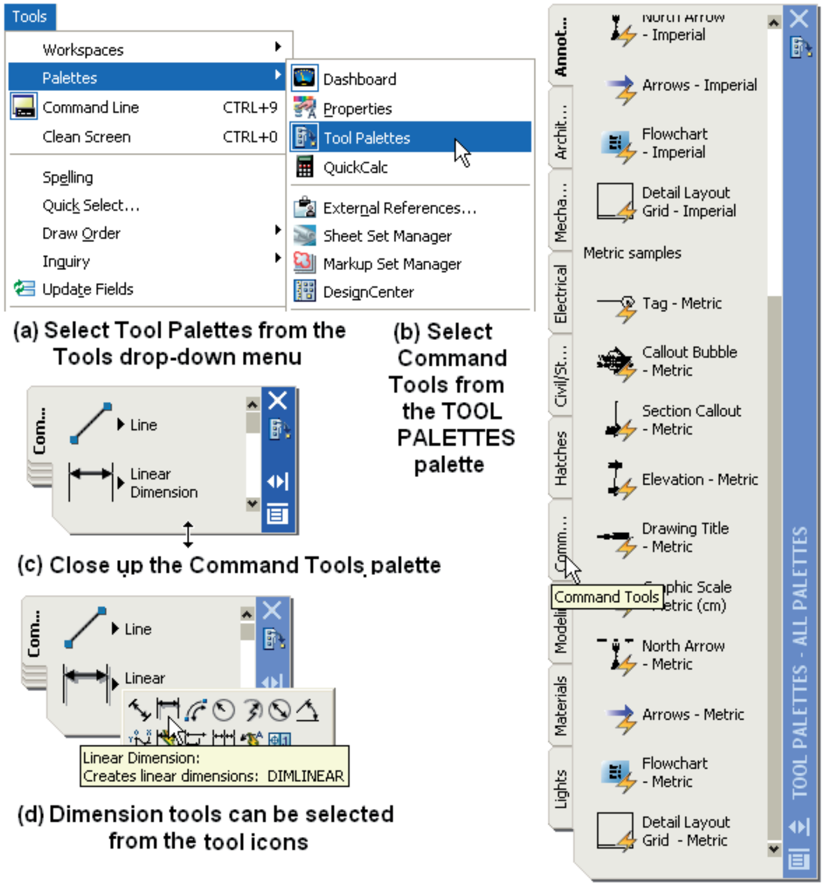


Fig. 6.2 Selecting a palette with dimension tool icons from the **Tools** drop-down menu

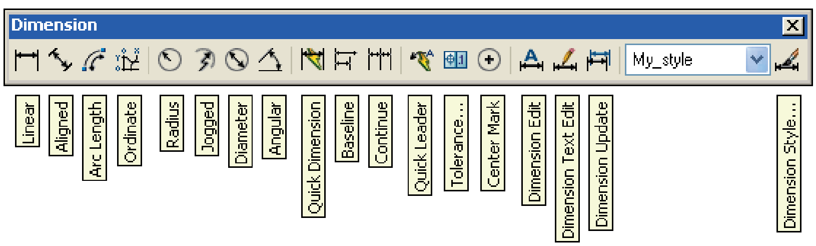


Fig. 6.3 The **Dimension** toolbar

Adding dimensions using the tools

First example – Linear Dimension (Fig. 6.4)

1. Construct a rectangle 180 × 110 using the **Polyline** tool.
2. *Left-click* on the **Linear Dimension** tool icon in the **Command Tools** palette or on **Linear** in the **Dimension** toolbar (Fig. 6.3). The command line shows:

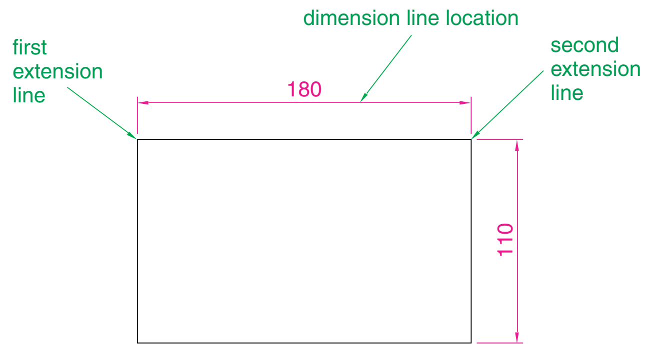


Fig. 6.4 First example – **Linear Dimension**

Command:_dimlinear
Specify first extension line origin or [select object]: *pick*
Specify second extension line origin: *pick*
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *pick*
Dimension text = 180
Command:

Fig. 6.4 shows the 180 dimension. Follow exactly the same procedure for the 110 dimension.

Notes

1. If necessary use **Osnaps** to locate the extension line locations.
2. The prompt **Specify first extension line origin or [select object]:** also allows the line being dimensioned to be *picked*.

Second example – Aligned Dimension (Fig. 6.6)

1. Construct the outline in Fig. 6.6 using the **Line** tool.
2. *Left-click* the **Aligned Dimension** tool icon (Fig. 6.5) and dimension the outline. The prompts and replies are similar to the first example.

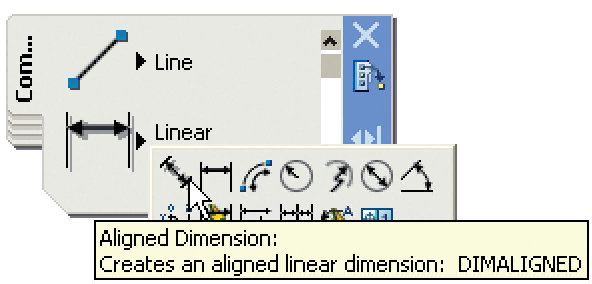


Fig. 6.5 The **Aligned Dimension** tool icon

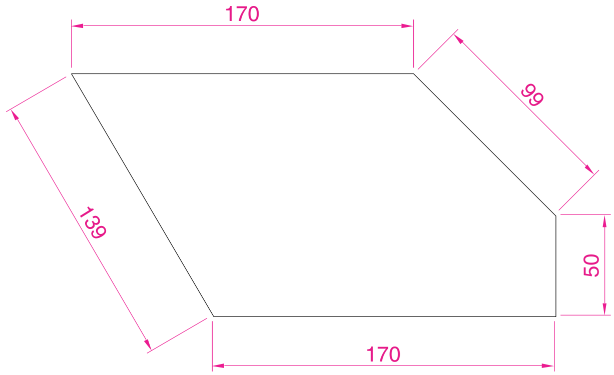


Fig. 6.6 Second example – **Aligned Dimension**

Third example – Radius Dimension (Fig. 6.8)

1. Construct the outline in Fig. 6.8 using the **Line** and **Fillet** tools.
2. *Left-click* the **Radius Dimension** (Fig. 6.7). The command line shows:

Command:_dimradius
Select arc or circle: *pick* one of the arcs
Dimension text = 30
Specify dimension line location or [Mtext/Text/Angle]: *pick*
Command:

3. Continue dimensioning the outline as shown in Fig. 6.8.

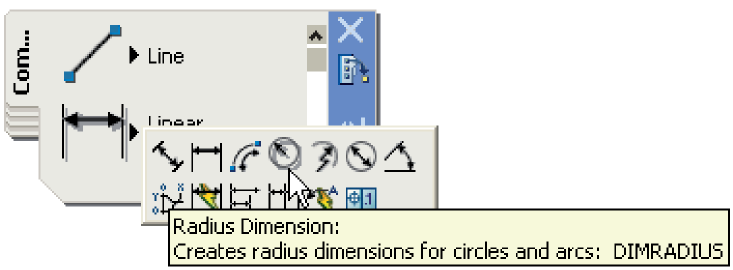


Fig. 6.7 The **Radius** dimension tool icon

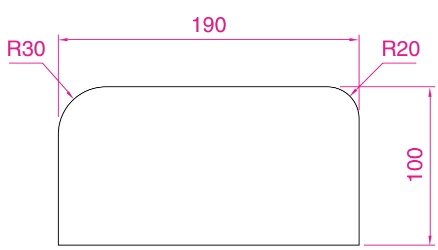


Fig. 6.8 Third example – **Radius Dimension**

Notes

1. At the prompt:
[Mtext/Text/Angle]:
If a **t** (Text) is *entered*, another number can be *entered*, but remember if the dimension is a radius the letter **R** must be *entered* as a prefix to the new number.
2. If the response is **a** (Angle), and an angle number is *entered*, the text for the dimension will appear at an angle. Fig. 6.9 show a radius dimension *entered* at an angle of 45°.
3. If the response is **m** (Mtext) the **Text Formatting** dialog appears together with a box in which new text can be *entered*. See page 141.
4. Dimensions added to a drawing using other tools from the **Command Tools** palette or from the **Dimensions** toolbar should be practised.

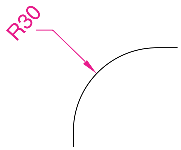


Fig. 6.9 A radius dimension at an angle of 45°

Adding dimensions from the command line

In the flyout from the **Linear tool** icon (Fig. 6.2) it will be seen that there are some dimension tools from this flyout which have not been described in examples. Some operators may prefer *entering* dimensions from the command line. This involves abbreviations for the required dimension such as:

For **Linear Dimension – hor** (horizontal) or **ve** (vertical)
For **Aligned Dimension – al**
For **Radius Dimension – ra**
For **Diameter Dimension – d**
For **Angular Dimension – an**
For **Dimension Text Edit – te**
For **Quick Leader – l**
And to exit from the dimension commands – **e** (Exit).

First example – hor and ve (Horizontal and vertical) – Fig. 6.11

1. Construct the outline in Fig. 6.10 using the **Line** tool. Its dimensions are shown in Fig. 6.11.
2. At the command line *enter* **dim**. The command line will show:

Command: *enter* **dim** *right-click*
Dim: *enter* **hor** (horizontal) *right-click*

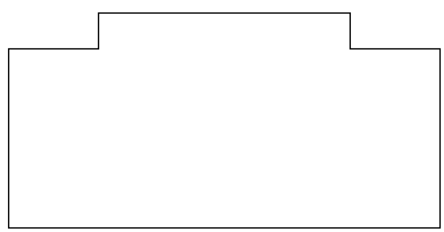


Fig. 6.10 First example – outline for dimensions

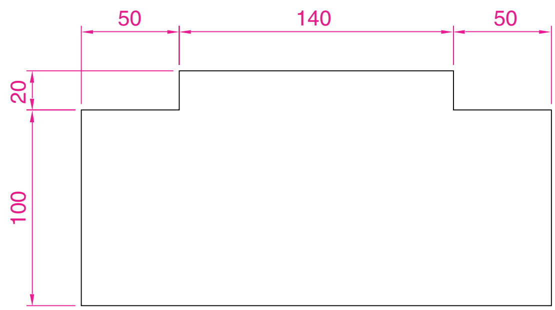


Fig. 6.11 First example – horizontal and vertical dimensions

Specify first extension line origin or <select object>: *pick*
Specify second extension line origin: *pick*
Specify dimension line location or [Mtext/Text/Angle]: *pick*
Enter dimension text <50>: *right-click*
Dim: *right-click*
HORIZONTAL
Specify first extension line origin or <select object>: *pick*
Specify second extension line origin: *pick*
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *pick*
Enter dimension text <140>: *right-click*
Dim: *right-click*

And the 50 and 140 horizontal dimensions are added to the outline.

3. Continue to add the right-hand 50 dimension. Then when the command line shows:

Dim: *enter* **ve** (vertical) *right-click*
Specify first extension line origin or <select object>: *pick*
Specify second extension line origin: *pick*
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *pick*
Dimension text <20>: *right-click*
Dim: *right-click*
VERTICAL
Specify first extension line origin or <select object>: *pick*
Specify second extension line origin: *pick*
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *pick*
Dimension text <100>: *right-click*
Dim: *enter* **e** (Exit) *right-click*
Command:

The result is shown in Fig. 6.11.

Second example – an (Angular) – Fig. 6.13

1. Construct the outline in Fig. 6.12 – a pline of width=1.
2. At the command line:

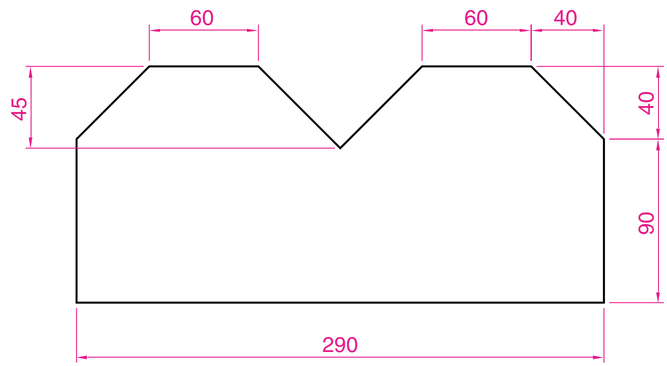


Fig. 6.12 Second example – outline for dimensions

Command: *enter* **dim** *right-click*
Dim: *enter* **an** *right-click*
Select arc, circle, line or <specify vertex>: *pick*
Select second line: *pick*
Specify dimension arc line location or [Mtext/Text/Angle]: *pick*
Enter dimension <90>: *right-click*
Enter text location (or press ENTER): *pick*
Dim:

And so on to add the other angular dimensions. The result is given in Fig. 6.13.

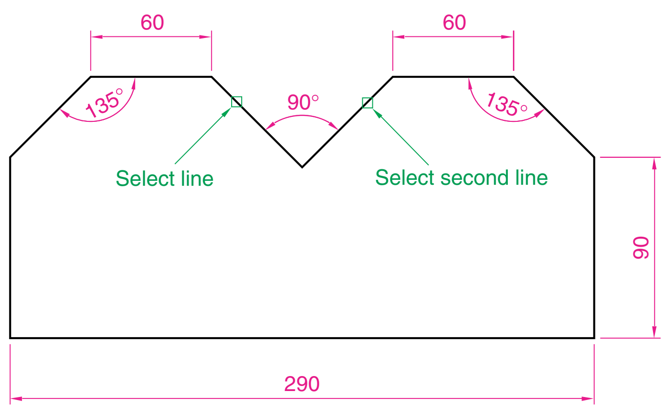


Fig. 6.13 Second example – **an** (Angle) dimension

Third example – l (Leader) – (Fig. 6.15)

1. Construct Fig. 6.14.
2. At the command line:

Command: *enter* **dim** *right-click*
Dim: *enter* **l** (Leader) *right-click*
Leader start: *enter* **nea** (osnap nearest) *right-click* **to** *pick* one of the chamfer lines
To point: *pick*
To point: *pick*

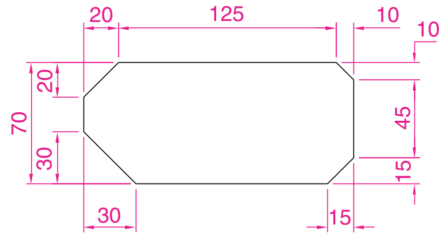


Fig. 6.14 Third example – outline for dimensioning

To point: *right-click*
Dimension text <0>: *enter* **CHA 10 × 10** *right-click*
Dim: *right-click*

Continue to add the other leader dimensions – Fig. 6.15.

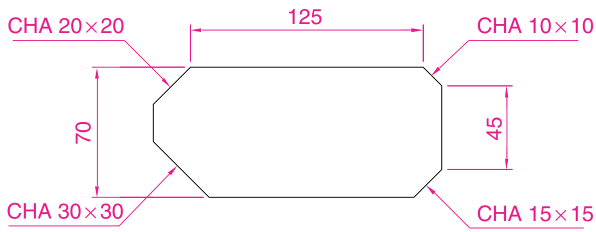


Fig. 6.15 Third example – **l** (Leader) dimensions

Fourth example – te (Dimension Text Edit) – (Fig. 6.17)

1. Construct Fig. 6.16.
2. At the command line:

Command: *enter* **dim** *right-click*
Dim: *enter* **te** (tedit) *right-click*
Select dimension: *pick* the dimension to be changed
Specify new location for text or [Left/Right/Center/Home/Angle]: either *pick* or *enter* a prompt's capital letter
Dim:

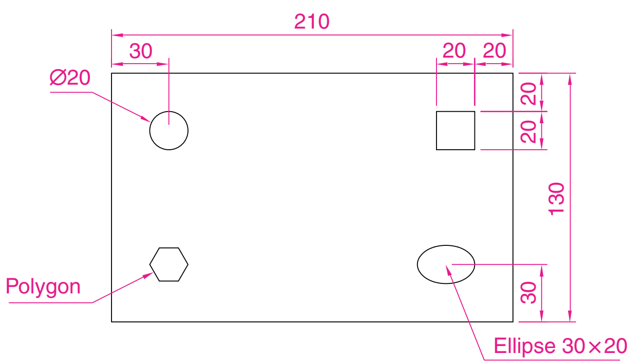


Fig. 6.16 Fourth example – dimensioned drawing

The results as given in Fig. 6.17 show dimensions which have been moved. The **210** dimension changed to the left-hand end of the dimension line, the **130** dimension changed to the left-hand end of the dimension line and the **30** dimension position changed.

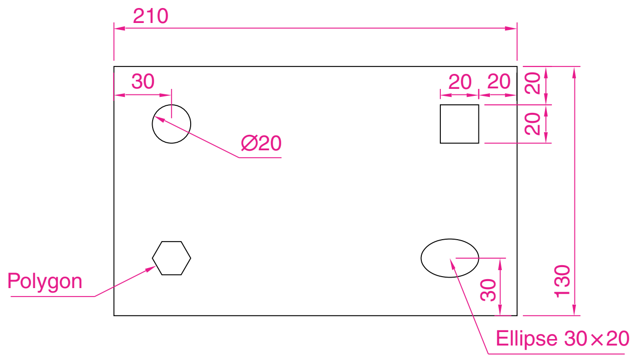


Fig. 6.17 Fourth example – dimensions amended with **tedit**

Fifth example – the Arc Length tool (Fig. 6.19)

1. Construct two arcs of different sizes as in Fig. 6.19.
2. Call the **Arc Length** tool with a *click* on its tool icon (Fig. 6.18), or with a *click* on **Arc Length** in the **Dimension** toolbar, or by *entering* **dimarc** at the command line. The command line shows:

   Command:_dimarc
   Select arc or polyline arc segment: *pick* an arc

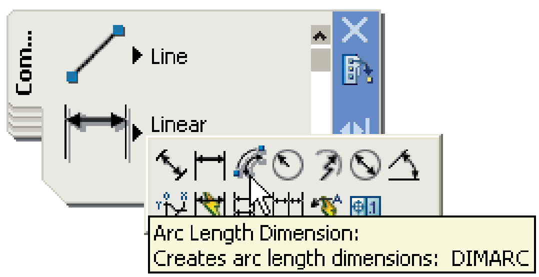


Fig. 6.18 The **Arc Length** tool

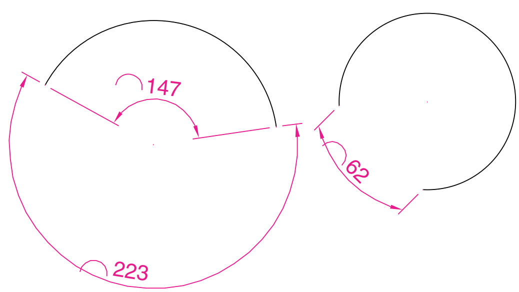


Fig. 6.19 Fifth example – **Arc Length** tool

Specify arc length dimension location, or [Mtext/Text/Angle/Partial/Leader]: *pick* a suitable position
Dimension text = 147
Command:

Examples on two arcs are shown in Fig. 6.19.

Sixth example – the Jogged tool (Fig. 6.21)

1. Draw a circle and an arc as indicated in Fig. 6.21.
2. Call the **Jogged** tool – either with a *left-click* on its tool icon (Fig. 6.20), or with a *click* on **Jogged** in the **Dimension** toolbar, or by *entering* **jog** at the command line. The command line shows:

Command:_dimjogged
Select arc or circle: *pick* the circle or the arc
Specify center location override: *pick*
Dimension text = 60
Specify dimension line location or [Mtext/Text/Angle]: *pick*
Specify jog location: *pick*
Command:

The results of placing a jogged dimension on a circle and an arc are shown in Fig. 6.21.

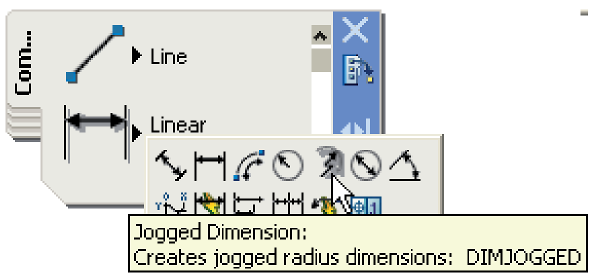


Fig. 6.20 The **Jogged** tool icon

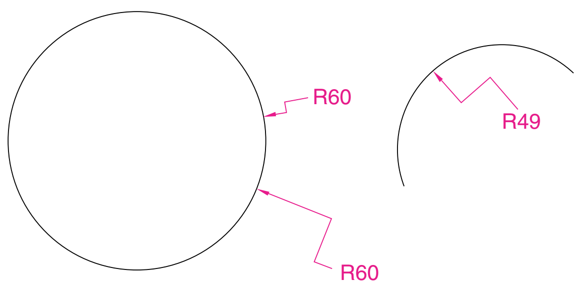


Fig. 6.21 Sixth example – the **Jogged** tool

Dimension tolerances

Before simple tolerances can be included with dimensions, new settings will need to be made in the **Dimension Style Manager** dialog as follows:

1. Open the dialog. The quickest way of doing this is to *enter* **d** at the command line followed by a *right-click*. This opens up the dialog.
2. *Click* the **Modify** . . . button of the dialog, followed by a *left-click* on the **Primary Units** tab and in the resulting sub-dialog make settings as shown in Fig. 6.22. Note the changes in the preview box of the dialog.
3. *Click* the **Tolerances** tab and in the resulting sub-dialog, make settings as shown in Fig. 6.23. *Left-click* the **OK** button, then in the

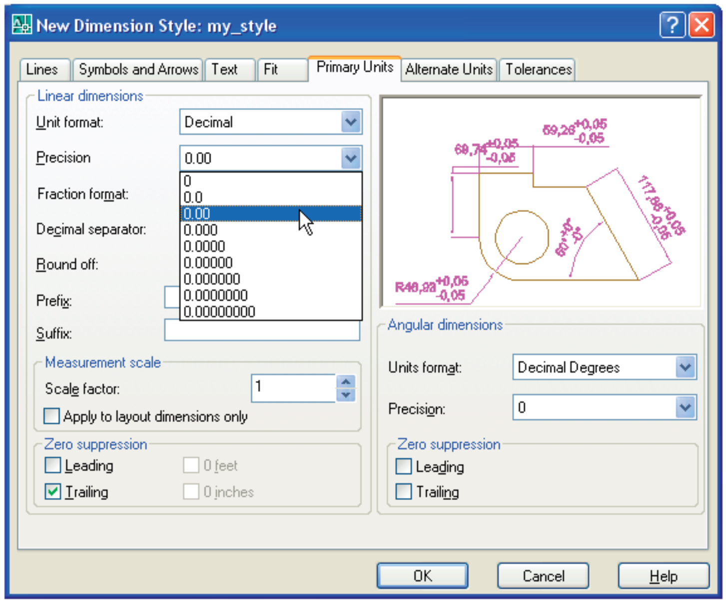


Fig. 6.22 The **Primary Units** sub-dialog of the **Dimension Style Manager**

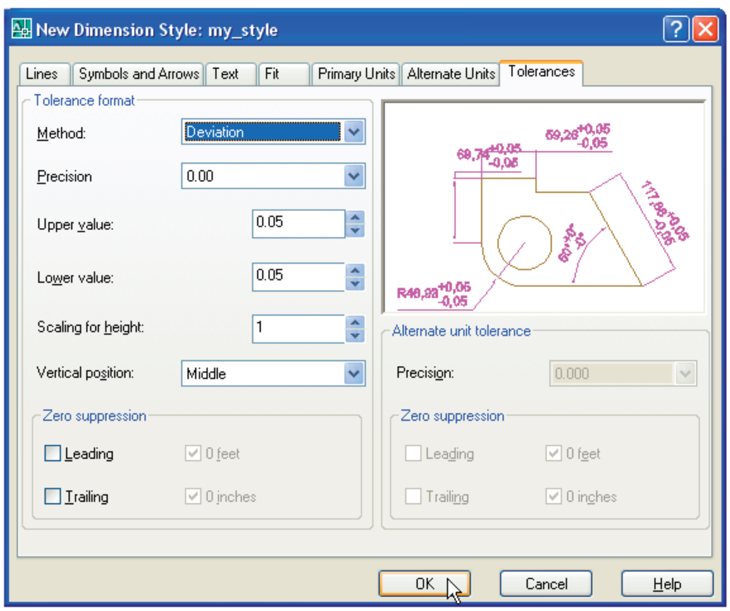


Fig. 6.23 The **Tolerances** sub-dialog of the **Dimension Style Manager**

main dialog, *click* the **Set Current** button, followed by a *left-click* on the **Close** button.

Example – simple tolerances (Fig. 6.25)

1. Construct the outline in Fig. 6.24

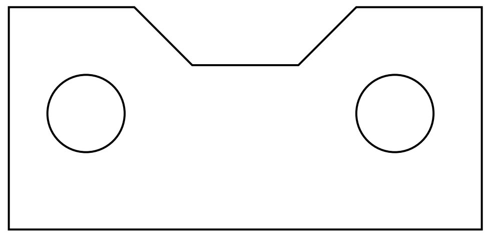


Fig. 6.24 First example – simple tolerances – outline

2. Dimension the drawing using tools from the **Dimension** toolbar or by *entering* abbreviations at the command line. Because tolerances have been set in the **Dimension Style Manager** dialog, the toleranced dimensions will automatically be added to the drawing (Fig. 6.25).

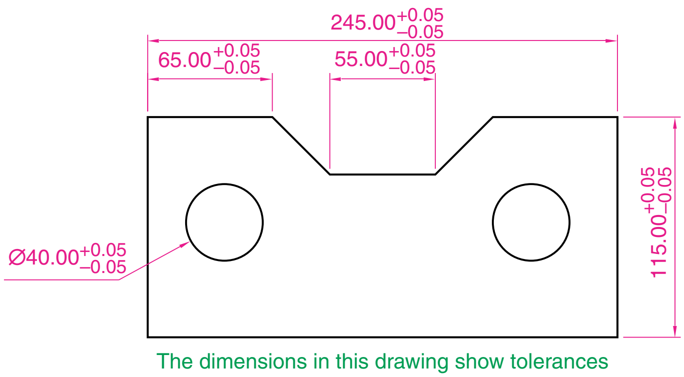


Fig. 6.25 Example – simple tolerances

Example – Geometric Tolerance (Fig. 6.30)

1. Construct the two rectangles with circles as in Fig. 6.26.
2. Add dimensions to the two circles.

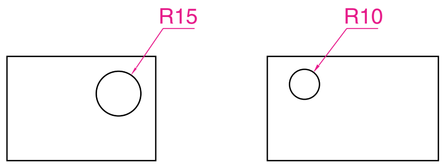


Fig. 6.26 Example – **Geometric Tolerance** dimensions to be toleranced

3. *Click* the **Tolerance** tool icon (Fig. 6.27). The **Geometric Tolerance** dialog (Fig. 6.28) appears.
4. In the dialog *click* the black box under **Sym**. The **Symbol** sub-dialog appears (Fig. 6.28) with a *click* on the top left-hand square.

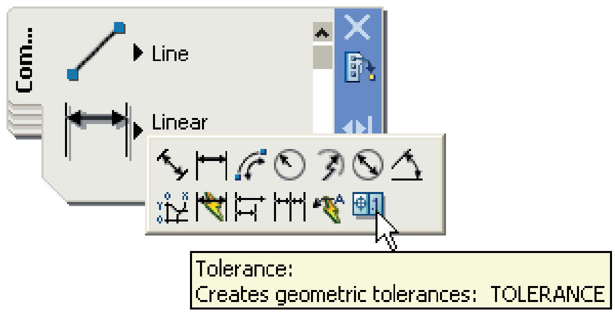


Fig. 6.27 The **Tolerance** tool icon

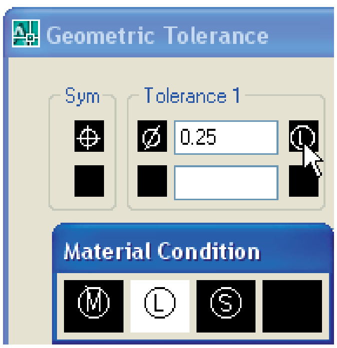
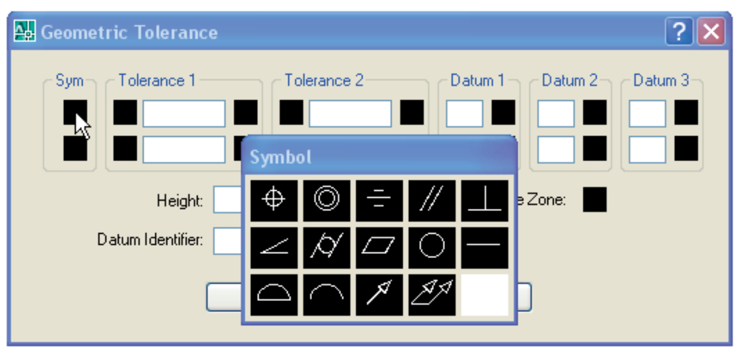


Fig. 6.28 The **Geometric Tolerance** dialog and the **Symbol** sub-dialog

5. Still in the dialog *click* the left-hand black square under **Tolerance 1**. The **Material Condition** dialog appears (Fig. 6.29). *Click* **L**. The letter appears in the top-right hand square of the dialog.
6. *Enter* **0.05** in the **Tolerance 1** field (Fig. 6.30), followed by a *click* on the dialog's **OK** button. The geometric tolerance appears. Move it to a position near the **R10** dimension in the drawing (Fig. 6.31).
7. Now add a geometric tolerance to the **15** dimension as shown in Fig. 6.31.

Fig. 6.29 The **Material Condition** dialog



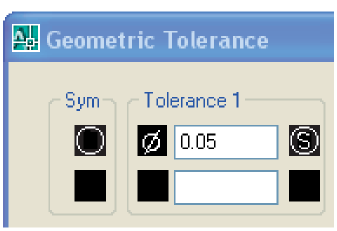


Fig. 6.30 *Enter* **0.05** in the **Tolerance 1** field

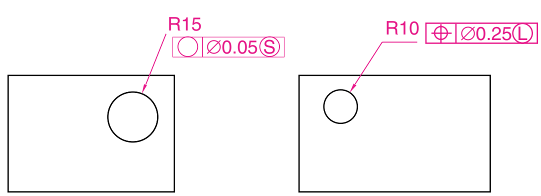


Fig. 6.31 Example – **Geometric Tolerances**

The meanings of the symbols

The **Material Condition** letters have the following meanings:

M – maximum amount of material
L – least amount of material
S – size within the limits.

Fig. 6.32 shows the meanings of the geometrical symbols.

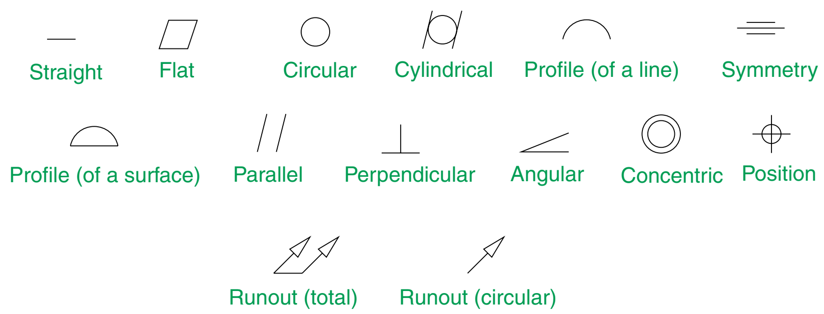


Fig. 6.32 The meanings of the symbols

Text

There are two main methods of adding text to drawings – **Multiline Text** and **Dynamic Text**.

Example – Dynamic Text (Fig. 6.25)

1. Open the drawing from the example on tolerances – Fig. 6.25.
2. At the command line *enter* **dt** (Dynamic Text) followed by a *right-click*:

Command: *enter* **dt** *right-click*
TEXT
Current text style "ARIAL" Text height: 8
Specify start point of text or [Justify/Style]: *pick*
Specify rotation angle of text <0>: *right-click*
Enter text: *enter* **The dimensions in this drawing show tolerances** *press* the **Return** key twice
Command:

The result is given in Fig. 6.25 on page 110.

Notes

1. When using **Dynamic Text**, the **Return** key of the keyboard is pressed when the text has been *entered* and **NOT** a *right-click*
2. At the prompt:

Specify start point of text or [Justify/Style]: *enter* **s** (Style) *right-click*
Enter style name or [?] <ARIAL>: *enter* **?** *right-click*
Enter text style(s) to list <*>: *right-click*

And an **AutoCAD Text Window** (Fig. 6.33) appears listing all the styles which have been selected in the **Text Style** (see page 67).

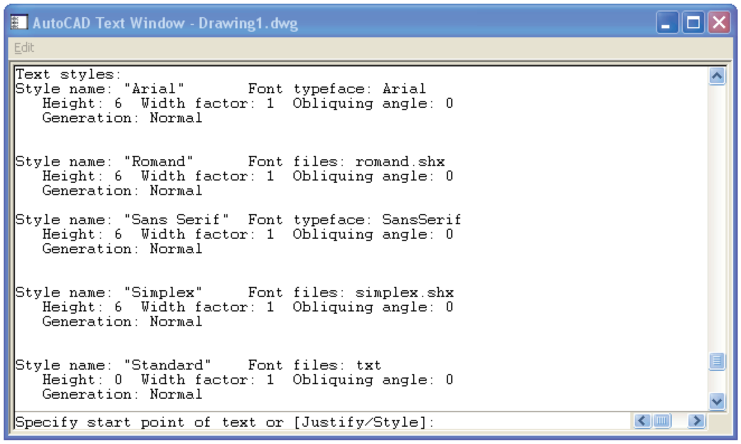


Fig. 6.33 The **AutoCAD Text Window**

3. In order to select the required text style its name must be *entered* at the prompt:

   Enter style name or [?] <ARIAL>: *enter* **Romand** *right-click*

   And the text *entered* will be in the **Romand** style of height **9**. But only if that style was previously selected in the **Text Style** dialog.
4. Fig. 6.34 shows some text styles from the **AutoCAD Text Window**.

Fig. 6.34 Some text styles

5. There are two types of text fonts available in AutoCAD 2007 – the **AutoCAD SHX** fonts and the **Windows True Type** fonts. The **ITALIC, ROMAND, ROMANS** and **STANDARD** styles shown in Fig. 6.34 are AutoCAD text fonts. The **TIMES** and **ARIAL** styles are **Windows True Type** fonts. Most of the **True Type** fonts can be *entered* in **Bold, Bold Italic, Italic** or **Regular** styles, but these variations are not possible with the AutoCAD fonts.
6. In the **Font name** popup list of the **Text Style** dialog, it will be seen that a large number of text styles are available to the AutoCAD 2007 operator. It is advisable to practise using a variety of these fonts to familiarise oneself with the text opportunities available with AutoCAD 2007.

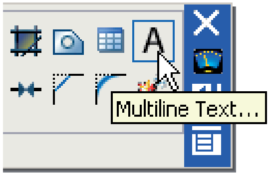


Fig. 6.35 The **Multiline Text** tool icon from the **2D Draw control panel**

Example – Multiline Text (Fig. 6.37)

1. Either *left-click* on the **Multiline Text** tool icon in the **2D Draw control panel** (Fig. 6.35) or *click* on **Multiline Text** . . . in the **Draw** toolbar or *enter* **t** at the command line:

Command:_mtext
Current text style: 'ARIAL' Text height: 8
Specify first corner: *pick*
Specify opposite corner or [Height/Justify/Line spacing/Rotation/ Style/Width]: *pick*

As soon as the **opposite corner** is *picked*, the **Text Formatting** dialog appears and the box changes as in Fig. 6.36. Text can now be *entered* as required within the box as indicated in this illustration. When all the required text has been *entered left-click* the **OK** button at the top right-hand corner of the **Text Formatting** dialog.

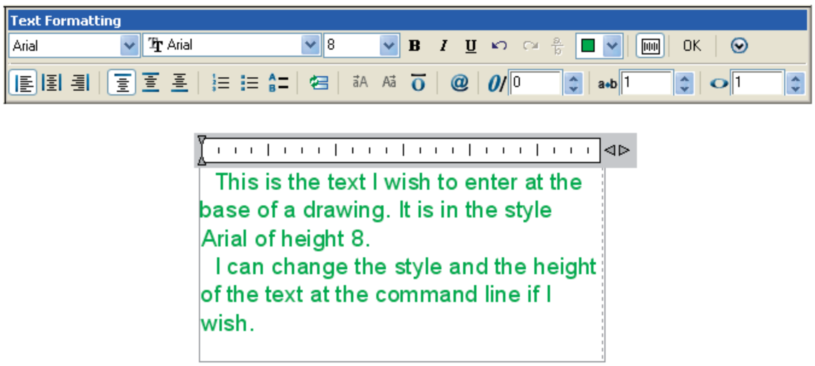


Fig. 6.36 Example – **Multiline Text** *entered* in the text box

2. Changes may be made to various aspects of the text being *entered* making choices from the various popup lists in the **Text Formatting** dialog. These popups are shown in Fig. 6.37.

Symbols used in text

When text has to be added by *entering* letters and figures as part of a dimension, the following symbols must be used:

To obtain **Ø75** *enter* **%%c75**
To obtain **55%** *enter* **55%%%**
To obtain **±0.05** *enter* **%%p0.05**
To obtain **90°** *enter* **60%%d**.

Checking spelling

There are two methods for the checking of spellings in AutoCAD 2007.

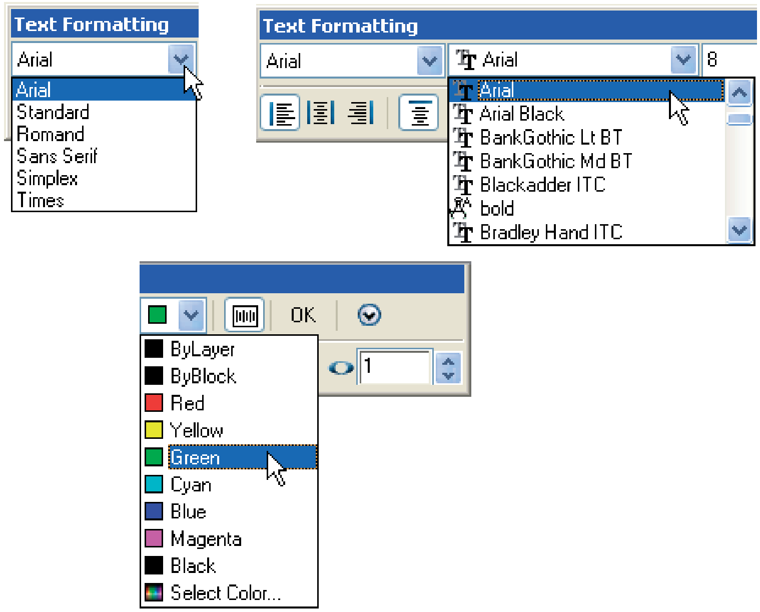


Fig. 6.37 The popups from the **Text Formatting** dialog

First example – the Spelling tool (Fig. 6.39)

1. *Enter* some badly spelt text as indicated in Fig. 6.39.
2. *Left-click* on **Edit** . . . from the **Text** sub-menu in the **Modify** drop-down menu (Fig. 6.38) or *enter* **ddedit** at the command line.
3. *Left-click* on the text. The text is highlighted. Edit the text as if working in a word processing application and when satisfied *click* the **Return** key of the keyboard (Fig. 6.39).

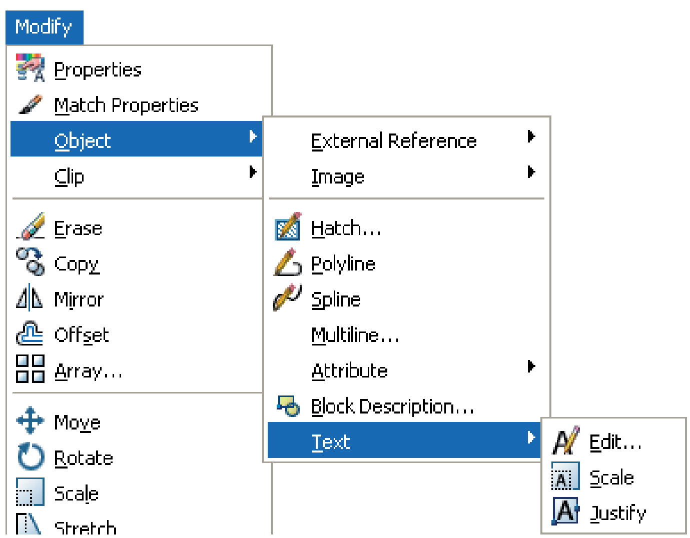


Fig. 6.38 Selecting **Edit** . . . from the **Text** sub-menu of the **Modify** drop-down menu

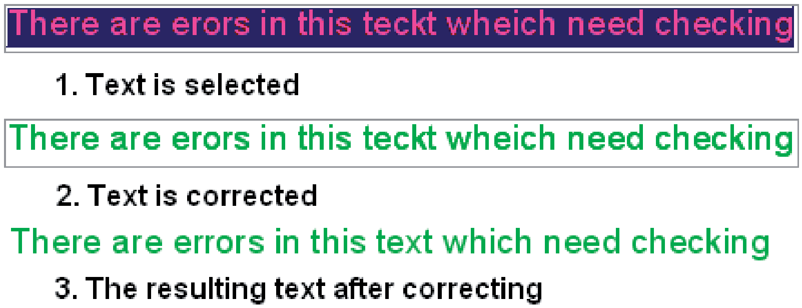


Fig. 6.39 The three stages in checking spelling using **Text Edit**

Second example – the Spelling tool (Fig. 6.43)

1. *Enter* some badly spelt text as indicated in Fig. 6.41.
2. Either *click* **Spelling** in the **Tools** drop-down menu (Fig. 6.40) or *enter* **spell** or **sp** at the command line.
3. *Click* the badly spelt text and *right-click*. The **Check Spelling** dialog appears (Fig. 6.41). Wrongly spelt words appear in the **Current work** field with words to replace them in the **Suggestions** field. Select the appropriate spelling as shown. Continue until all text is checked.

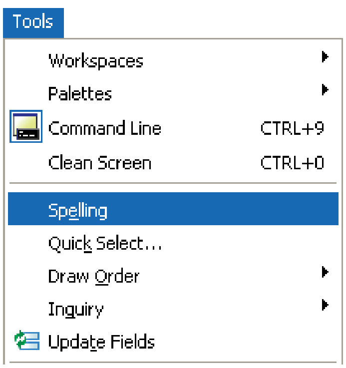


Fig. 6.40 **Spelling** in the **Tools** drop-down menu

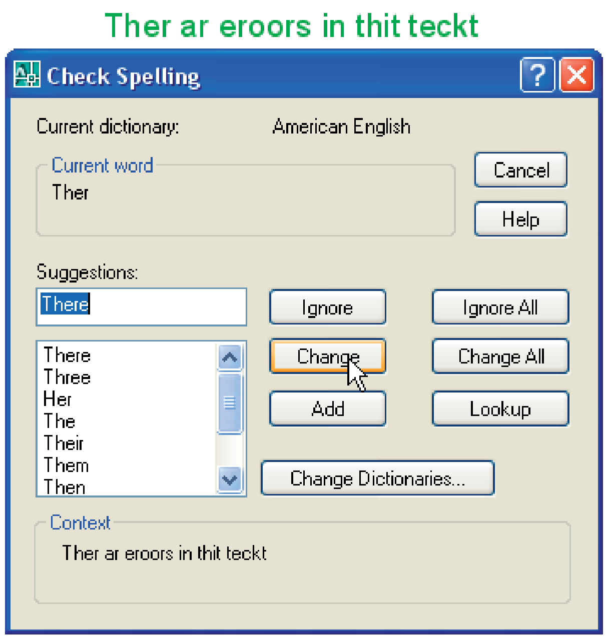


Fig. 6.41 Second example – the **Check Spelling** dialog

Revision notes

1. In the **Line and Arrows** sub-dialog of the **Dimension Style Manager** dialog, **Lineweights** were set to **0.3** (page 72). If these lineweights are

to show in the drawing area of AutoCAD 2007, the **LWT** button in the status bar must be set **ON**.

2. Dimensions can be added to drawings using the tools from the **Command Tools** palette, from the **Dimension** toolbar, or by *entering* **dim**, followed by abbreviations for the tools at the command line.
3. It is usually advisable to use osnaps when locating points on a drawing for dimensioning.
4. The **Style** and **Angle** of the text associated with dimensions can be changed during the dimensioning process.
5. When wishing to add tolerances to dimensions it will probably be necessary to make new settings in the **Dimension Style Manager** dialog.
6. There are two methods for adding text to a drawing – **Dynamic Text** and **Multiline Text**.
7. When adding text to a drawing, the **Return** key must be used and not the right-hand mouse button.
8. Text styles can be changed during the process of adding text to drawings.
9. AutoCAD 2007 uses two types of text style – **AutoCAD SHX** fonts and **Windows True Type** fonts.
10. Most **True Type** fonts can be in **bold**, ***bold italic***, *italic* or regular format. **AutoCAD** fonts can only be added in the single format.
11. When using **Multiline Text**, changes can be made by selection from the popup lists in the **Text Formatting** dialog.
12. To obtain the symbols **Ø;** ± ; °; % use %%**c**; %%**p**; %%**d**; %%% respectively, before or after the figures of the dimension.
13. Text spelling can be checked with by selecting **Text/Edit** . . . from the **Modify** drop-down menu or by *entering* **spell** or **sp** at the command line.

Exercises

1. Open any of the drawings previously saved from working through examples or as answers to exercises and add appropriate dimensions.
2. Construct the drawing in Fig. 6.42 but in place of the given dimensions add dimensions showing tolerances of 0.25 above and below.
3. Construct the two polygons in Fig. 6.43 and add all the diagonals. Then set the two osnaps **endpoint** and **intersection** and using the lines as in Fig. 6.43 construct the stars as shown using a polyline of width = 3. Next erase all unwanted lines. Finally dimension the angles labelled **A**, **B**, **C** and **D**.
4. Construct and dimension the drawing in Fig. 6.44.
5. Using the text style **Sans Serif** of height **20** and enclosing the wording within a rectangle of Width = 5 and Fillet = 10, construct the drawing in Fig. 6.45.

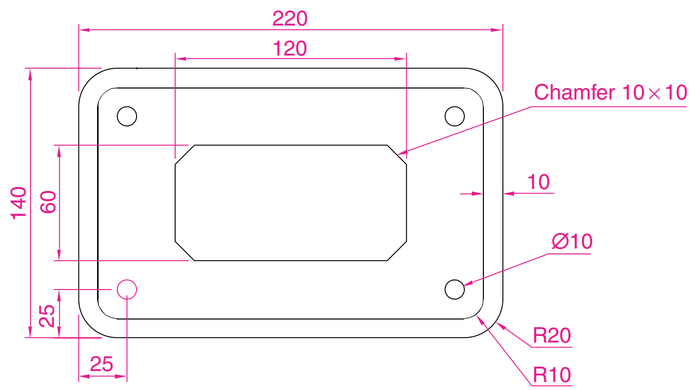


Fig. 6.42 Exercise 2

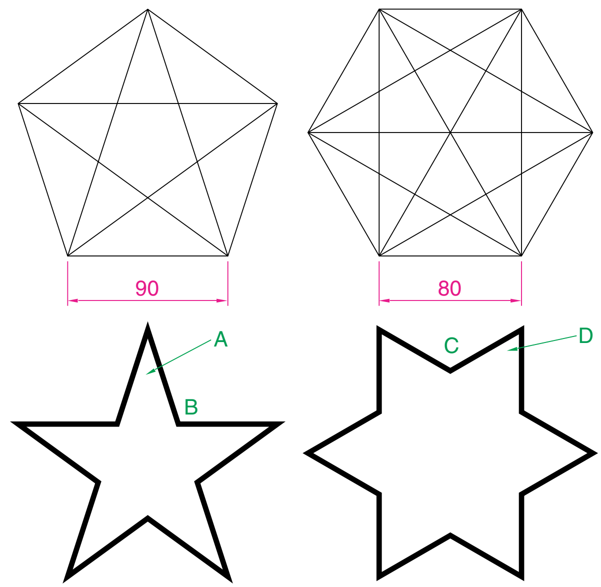


Fig. 6.43 Exercise 3

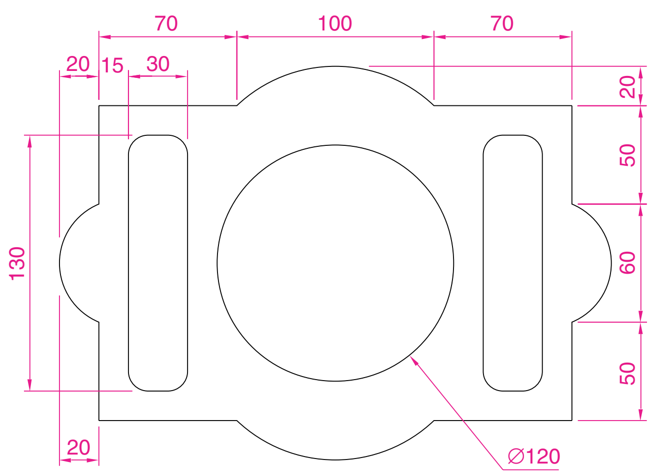


Fig. 6.44 Exercise 4

Fig. 6.45 Exercise 5

CHAPTER 7

Orthographic and isometric

Aim of this chapter

To introduce methods of constructing drawings of two types – orthographic projection and isometric drawings.

Orthographic projection

Orthographic projection involves viewing an article being described in a technical drawing from different directions – from the front, from a side, from above, from below or from any other viewing position. Orthographic projection often involves:

1. The drawing of details which are hidden, using hidden detail lines.
2. Sectional views in which the article being drawn is imagined as being cut through and the cut surface drawn.
3. Centre lines through arcs, circles, spheres and cylindrical shapes.

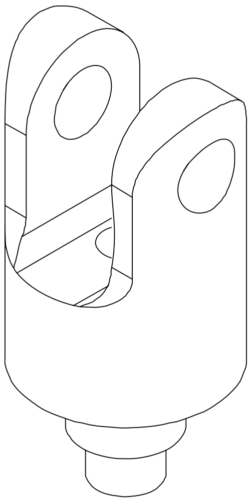


Fig. 7.1 Example – orthographic projection – the solid being drawn

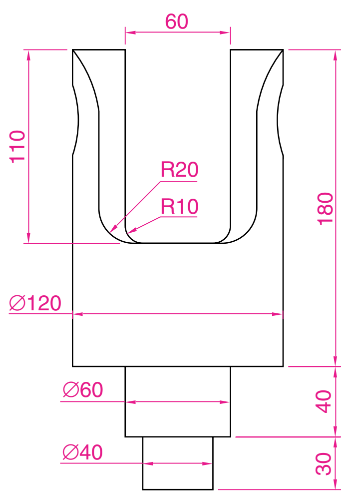


Fig. 7.2 The **front view** of the solid

An example of an orthographic projection

Taking the solid shown in Fig. 7.1, construct a three-view orthographic projection of the solid:

1. Draw what is seen when the solid is viewed from its left-hand side and regard this as the **front** of the solid. What is drawn will be a **front view** (Fig. 7.2).
2. Draw what is seen when the solid is viewed from the left-hand end of the front view. This produces an **end view**. Fig. 7.3 shows the end view alongside the front view.
3. Draw what is seen when the solid is viewed from above the front view. This produces a **plan**. Fig. 7.4 shows the plan below the front view.
4. Draw centre and hidden detail lines:
   (a) *Click* the arrow to the right of the **Layers** field to show all layers set in the **acadiso.dwt** template on which the drawing has been constructed (Fig. 7.5).
   (b) *Left-click* the **Centre** layer name in the layers list, making it the current layer. All lines will now be drawn as centre lines.
5. In the three-view drawing add centre lines.
6. Make the **Hidden** layer the current layer and add hidden detail lines.

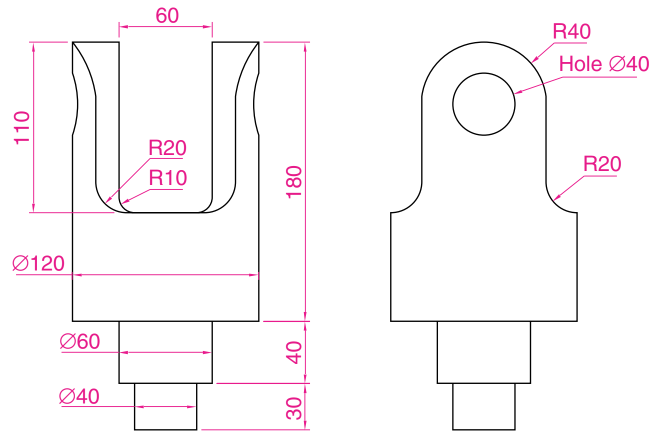


Fig. 7.3 **Front** and **end** views of the solid

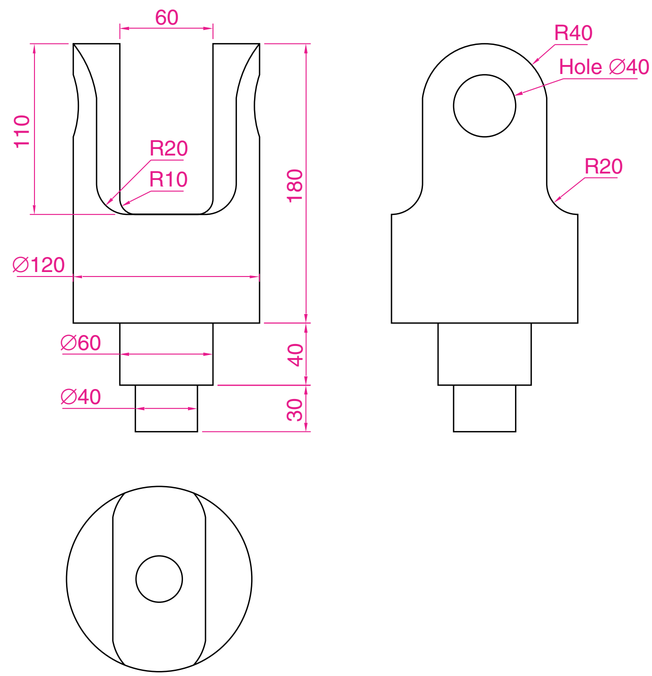


Fig. 7.4 **Front** and **end** views and **plan** of the solid

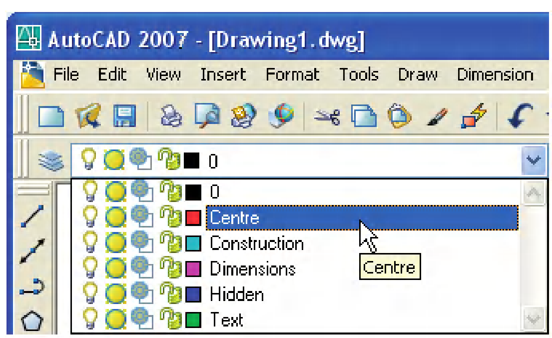


Fig. 7.5 Making the layer **Centre** current

7. Make the **Text** layer current and add border lines and a title block.
8. Make the **Dimensions** layer current and add all dimensions.

The completed drawing is shown in Fig. 7.6.

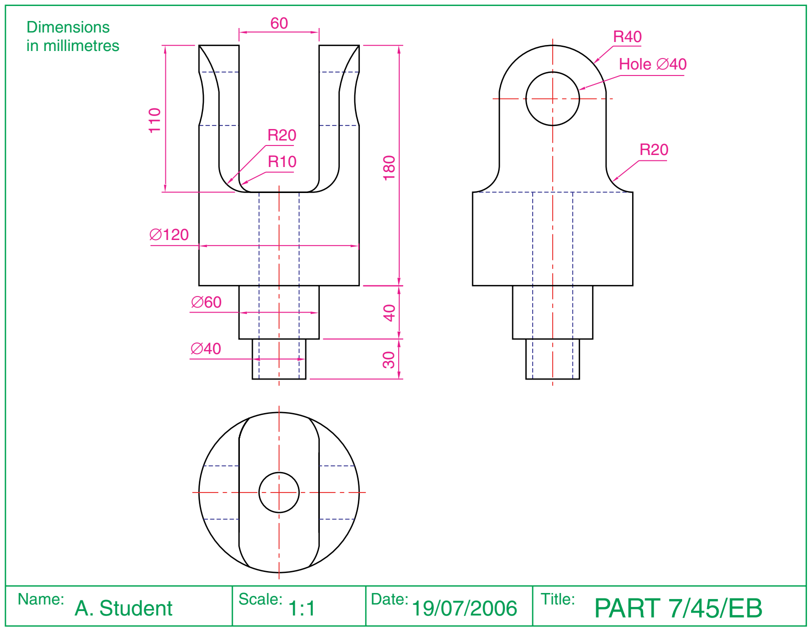


Fig. 7.6 The completed working drawing of the solid

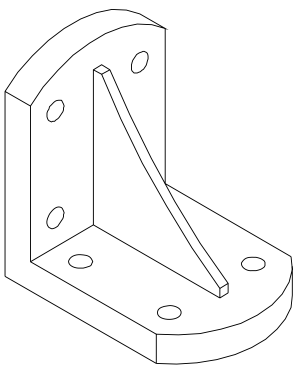
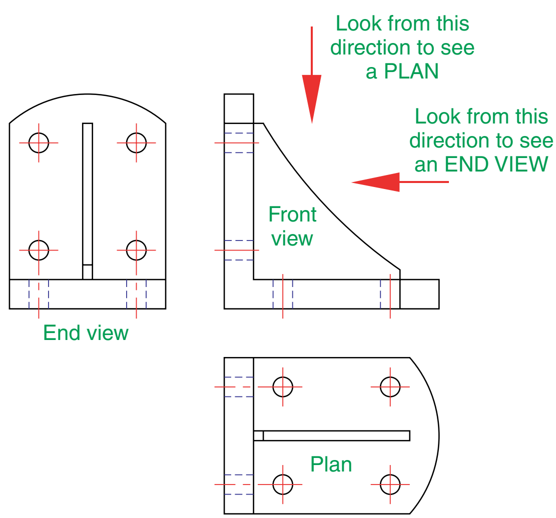


Fig. 7.7 The solid used to demonstrate first and third angles of projection

First angle and third angle

There are two types of orthographic projection – **first angle** and **third angle**. Fig. 7.7 is a pictorial drawing of the solid used to demonstrate the two angles. Fig. 7.8 shows a three-view **first angle projection** and Fig. 7.9 the same views in **third angle**.

Fig. 7.8 A **first angle** projection



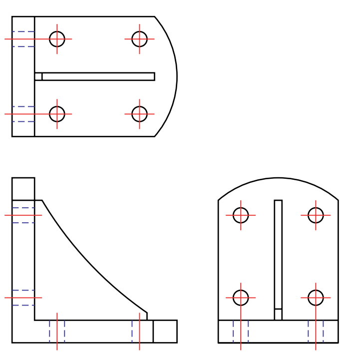


Fig. 7.9 A **third angle** projection

In both angles the viewing is from the same direction. The difference is that the view as seen is placed on the viewing side of the front view in **third angle** and on the opposite side to the viewing in **first angle**.

Sectional views

In order to show internal shapes of a solid being drawn in orthographic projection, the solid is imagined as being cut along a plane and the cut surface then drawn as seen. Common practice is to **hatch** the areas which then show in the cut surface. Note the section plane line, the section label and the hatching in the sectional view in Fig. 7.10.

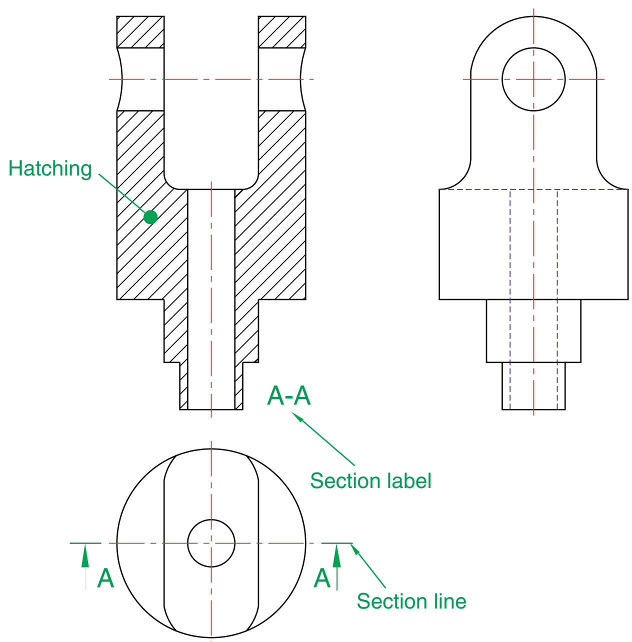


Fig. 7.10 A sectional view

Adding hatching

To add the hatching as shown in Fig. 7.10:

Fig. 7.11 The **Hatch** tool icon from the **2D Draw control panel**

1. Call the **Hatch** tool – either *left-click* on its tool icon in the **2D Draw control panel** (Fig. 7.11), or *click* the tool in the **Draw** toolbar, or *enter* **h** at the command line. ***Note*** – do not *enter* **hatch** as this gives a different result. The **Hatch and Gradient** dialog (Fig. 7.12) appears.
2. *Click* in the **Swatch** field. The **Hatch Pattern Palette** appears. *Left-click* the **ANSI** tab and from the resulting pattern icons *double-click* the **ANSI31** icon. The palette disappears and the **ANSI31** pattern appears in the **Swatch** field.

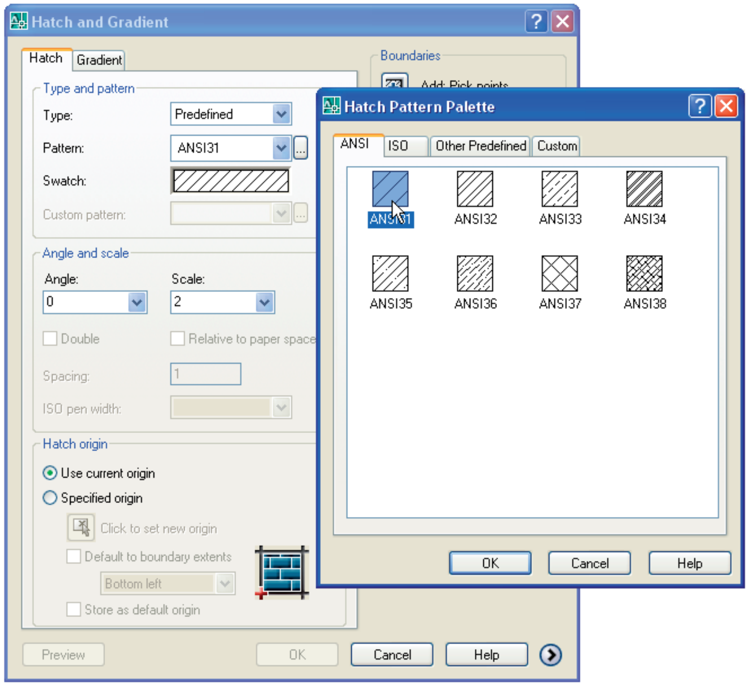


Fig. 7.12 The **Hatch and Gradient** dialog and the **ANSI Hatch Pattern Palette**

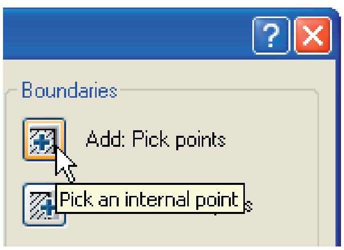


Fig. 7.13 The **Pick an internal point** button of the **Boundary Hatch and Fill** dialog

3. In the dialog *left-click* the **Pick an internal point** button (Fig. 7.13). The dialog disappears.
4. In the front view *pick* points as shown in the left-hand drawing of Fig. 7.14. The dialog reappears. *Click* the **Preview** button of the dialog and in the sectional view which reappears, check whether the hatching is satisfactory. In this example it may well be that the **Scale** figure in the dialog needs to be *entered* as **2** in place of the default **1**. Change the figure and **Preview** again. If satisfied *click* the **OK** button of the dialog.

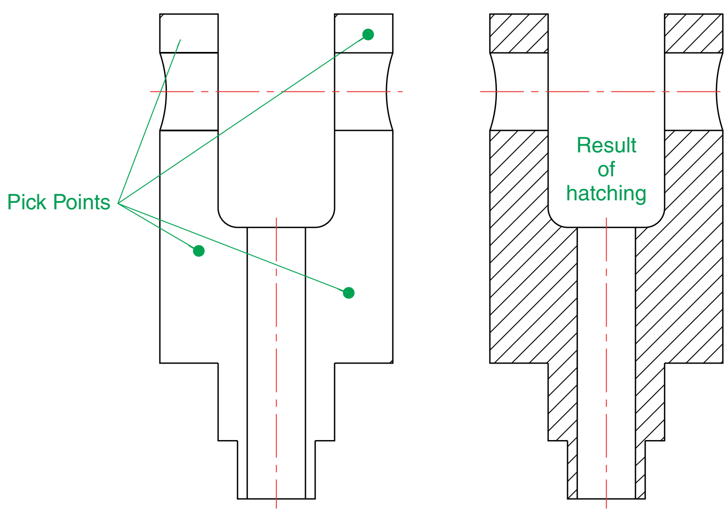


Fig. 7.14 The result of hatching

Isometric drawing

Isometric drawing must not be confused with solid model drawing, examples of which are given in Chapters 13 to 20. Isometric drawing is a 2D method of describing objects in a pictorial form.

Setting the AutoCAD window for isometric drawing

To set the AutoCAD 2007 window for the construction of isometric drawings:

1. At the command line:

Command: *enter* snap
Specify snap spacing or [On/Off/Aspect/Rotate/Style/Type] <5>: s (Style)
Enter snap grid style [Standard/Isometric] <S>: i (Isometric)
Specify vertical spacing <5>: *right-click*
Command:

Note the cursor hair lines which are set in an **Isometric Left** angle (Fig. 7.15).

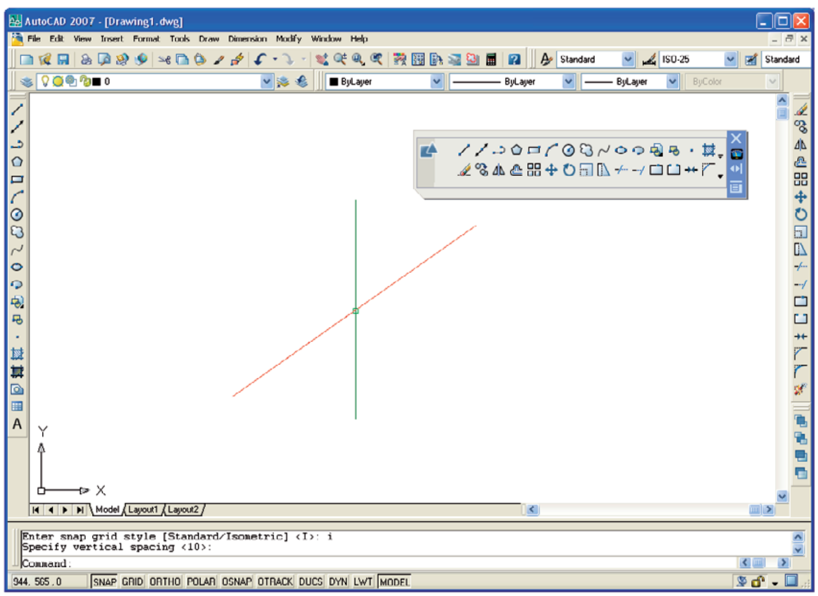


Fig. 7.15 Cursor hair lines set in an **Isometric Left** angle

2. There are three isometric angles – **Isoplane Top, Isoplane Left** and **Isoplane Right**. These can be set either by pressing the **F5** function key or by pressing the **Ctrl** and **E** keys. Repeated pressing of either of these 'toggles' between the three settings. Fig. 7.16 is an isometric view showing the three isometric planes.

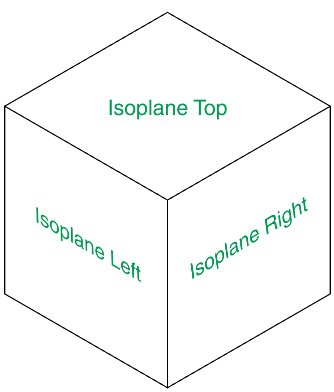


Fig. 7.16 The three isoplanes

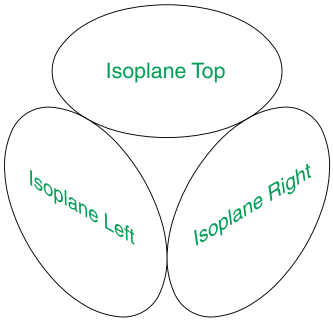


Fig. 7.17 The three isocircles

The isometric circle

Circles in an isometric drawing show as ellipses. To add an isometric circle to an isometric drawing, call the **Ellipse** tool. The command line shows:

Command: _ellipse
Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: *enter* **i** (Isocircle) *right-click*
Specify center of isocircle: *pick* or *enter* coordinates
Specify radius of isocircle or [Diameter]: *enter* a number
Command:

And the isocircle appears. Its isoplane position is determined by which of the isoplanes is in operation at the time the isocircle was formed. Fig. 7.17 shows these three isoplanes containing isocircles.

Examples of isometric drawings

First example – isometric drawing (Fig. 7.20)

1. Work to the shapes and sizes given in the orthographic projection in Fig. 7.18. Set Snap on (press the **F9** function key) and Grid on (**F7**).
2. Set Snap to Isometric and set the isoplane to Isoplane Top using **F5**.
3. With **Line**, construct the outline of the top of the model (Fig. 7.19) working to the dimensions given in Fig. 7.18.
4. Call **Ellipse** tool and set to isocircle, and add the isocircle of radius 20 centred in its correct position in the outline of the top (Fig. 7.19).
5. Set the isoplane to Isoplane Right and, with the **Copy** tool, copy the top with its ellipse vertically downwards three times as shown in Fig. 7.19.

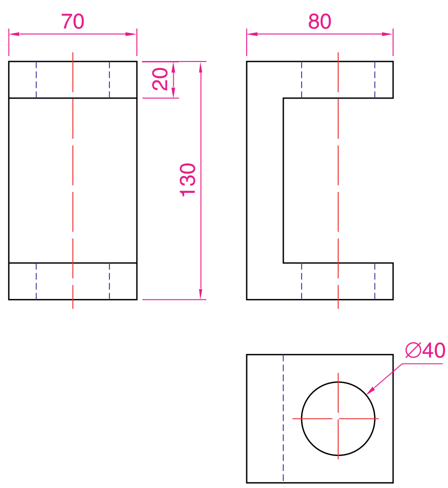
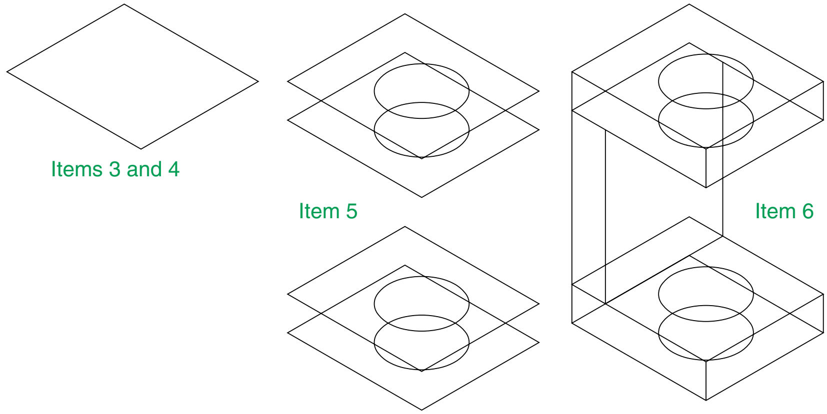


Fig. 7.18 First example – isometric drawing – the model

Fig. 7.19 First example – isometric drawing – items **3, 4, 5** and **6**



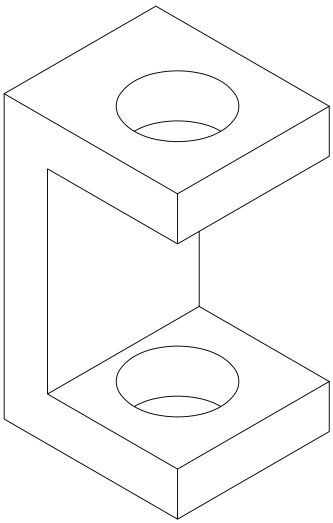


Fig. 7.20 First example – isometric drawing

6. Add lines as shown in Fig. 7.19.
7. Finally using **Trim** remove unwanted parts of lines and ellipses to produce Fig. 7.20.

Second example – isometric drawing (Fig. 7.22)

Fig. 7.21 is an orthographic projection of the model from which the isometric drawing is to be constructed. Fig. 7.22 shows the stages in its construction. The numbers refer to the items in the list below.

1. In **Isoplane Right** construct two isocircles of radii 10 and 20.
2. Add lines as in drawing **2** and trim unwanted parts of isocircle.
3. With **Copy**, copy three times as in drawing **3**.
4. With **Trim**, trim unwanted lines and parts of isocircle as in drawing **4**.

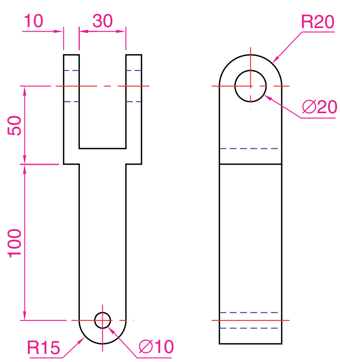


Fig. 7.21 Second example – isometric drawing – orthographic projection of model

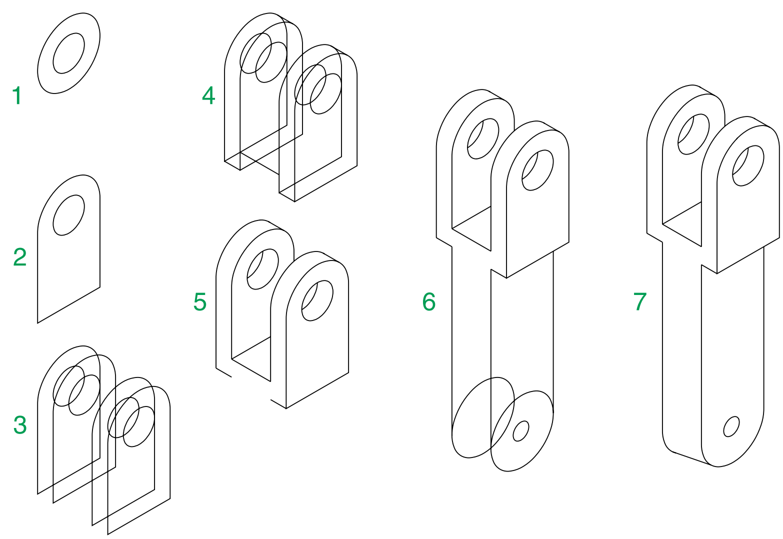


Fig. 7.22 Second example – isometric drawing – stages in the construction

5. In **Isoplane Left** add lines as in drawing **5**.
6. In **Isoplane Right** add lines and isocircles as in drawing **6**.
7. With **Trim**, trim unwanted lines and parts of isocircles to complete the isometric drawing – drawing **7**.

Revision notes

1. There are mainly two types of orthographic projection – first angle and third angle.
2. The number of views included in an orthographic projection depend upon the complexity of what is being drawn – a good rule to follow is to attempt fully describing the object being drawn in as few views as possible.
3. Sectional views allow parts of an object which are normally hidden from view to be more fully described in a projection.
4. When a layer is turned **OFF** all constructions on that layer disappear from the screen.
5. If a layer is locked, objects can be added to the layer but no further additions or modifications can be added to the layer. If an attempt is made to modify an object on a locked layer the command line shows:

   Command:_erase
   Select objects: *pick* **1 found**
   1 was on a locked layer

   And the object will not be modified.
6. Frozen layers cannot be selected, but note that layer **0** cannot be frozen.
7. Isometric drawing is a 2D pictorial method of producing illustrations showing objects. It is not a 3D method of showing a pictorial view.
8. When drawing ellipses in an isometric drawing, the **Isocircle** prompt of the **Ellipse** tool command line sequence must be used.
9. When constructing an isometric drawing **Snap** must be set to isometric mode before construction can commence.

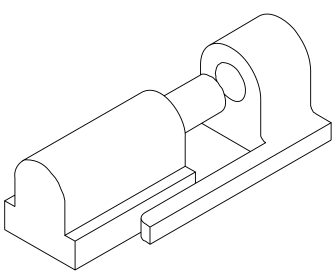


Fig. 7.23 Exercises 1, 2 and 3 – an isometric drawing of the three parts of the slider on which these exercises are based

Exercises

Fig. 7.23 is an isometric drawing of a slider fitment on which exercises **1, 2** and **3** are based.

1. Fig. 7.24 is a first angle orthographic projection of part of the fitment shown in the isometric drawing in Fig. 7.23. Construct a three-view third angle orthographic projection of the part.
2. Fig. 7.25 is a first angle orthographic projection of the other part of the fitment. Construct a three-view third angle orthographic projection of the part.

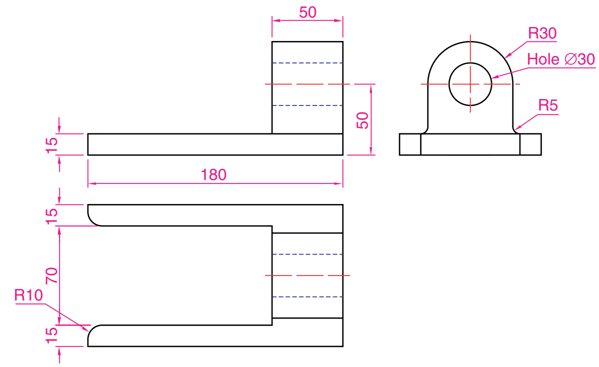


Fig. 7.24 Exercise 1

3. Construct an isometric drawing of the part shown in Fig. 7.25.

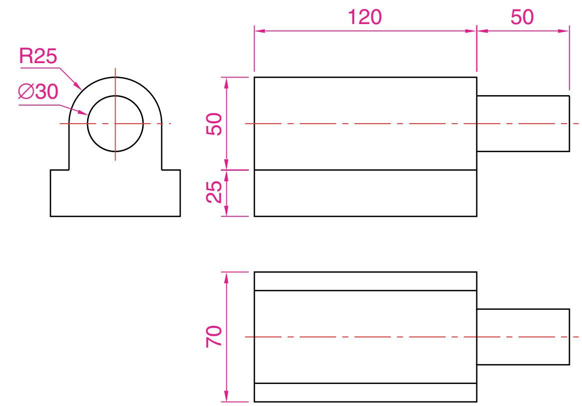


Fig. 7.25 Exercises 2 and 3

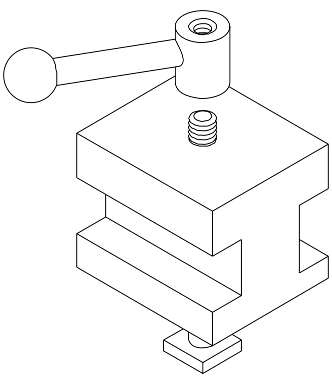


Fig. 7.26 Exercises 4 and 5 – an isometric drawing of the tool holder on which the two exercises are based

4. Construct a three-view orthographic projection in an angle of your own choice of the tool holder assembled as shown in the isometric drawing in Fig. 7.26. Details are given in Fig. 7.27.

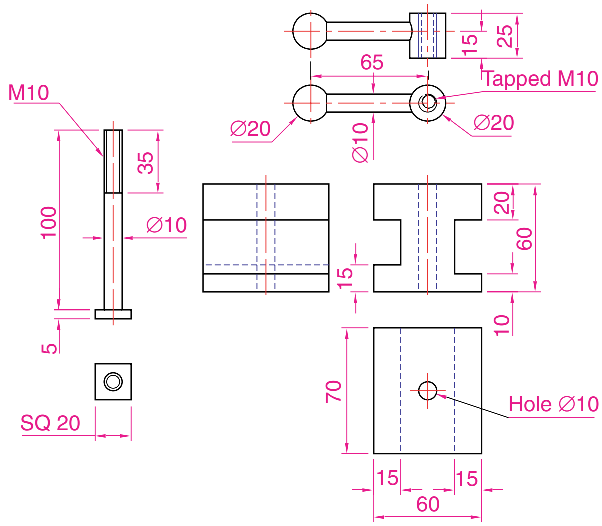


Fig. 7.27 Exercises 4 and 5 – orthographic projections of the three parts of the tool holder

5. Construct an isometric drawing of the body of the tool holder as shown in Figs 7.26 and 7.27.
6. Construct the orthographic projection given in Fig. 7.29.
7. Construct an isometric drawing of the angle plate shown in Figs 7.28 and 7.29.

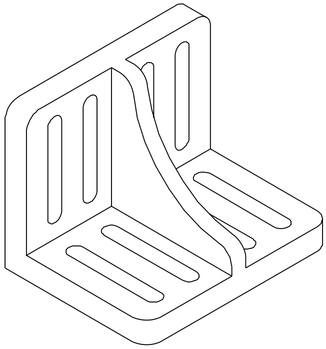


Fig. 7.28 An isometric drawing of the angle plate on which exercises 6 and 7 are based

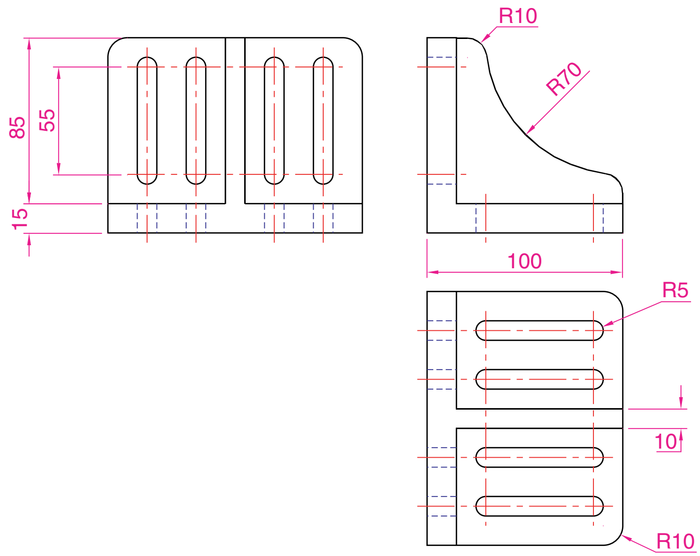


Fig. 7.29 Exercises 6 and 7 – an orthographic projection of the angle plate

8. Construct a third angle projection of the component shown in the isometric drawing in Fig. 7.30 and the three-view first angle projection in Fig. 7.31.

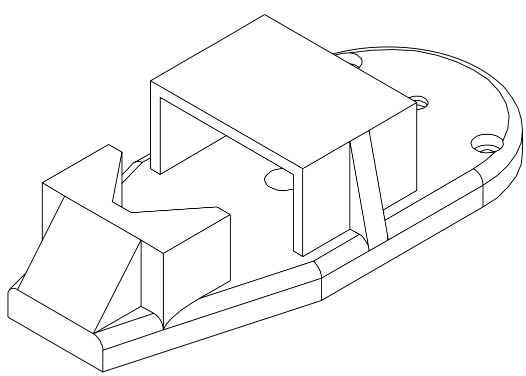


Fig. 7.30 Exercises 8 and 9 – an isometric drawing of the component for the two exercises

9. Construct the isometric drawing shown in Fig. 7.30 working to the dimensions given in Fig. 7.31.

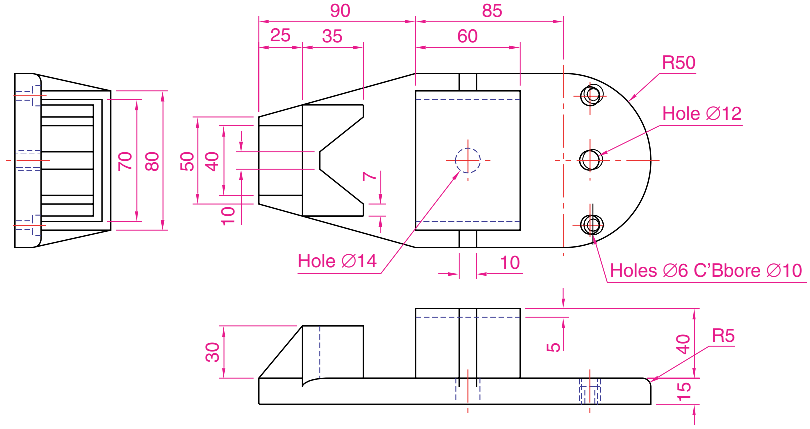


Fig. 7.31 Exercises 8 and 9

CHAPTER 8

Hatching

Aim of this chapter

To describe further examples of the use of hatching in its various forms.

Introduction

In Chapter 7 an example of hatching a sectional view in an orthographic projection was given. Further examples of the use of hatching will be described in this chapter.

There are a large number of hatch patterns available when hatching drawings in AutoCAD 2007. Some examples from the **Other Predefined** set of hatch patterns (Fig. 8.1) in the **Hatch Pattern Palette** sub-dialog are shown in Fig. 8.2.

Other hatch patterns can be selected from the **ISO** or **ANSI** hatch pattern palettes, or the operator can design their own hatch patterns and save them to the **Custom** hatch palette.

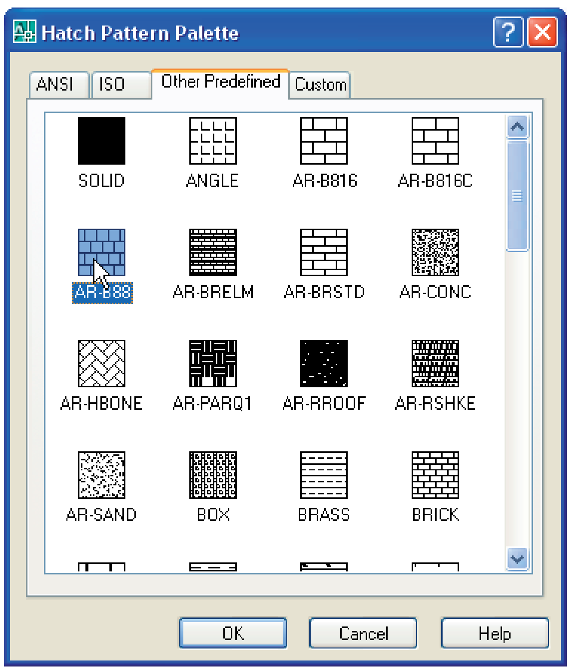


Fig. 8.1 The **Other Predefined** Hatch Pattern Palette

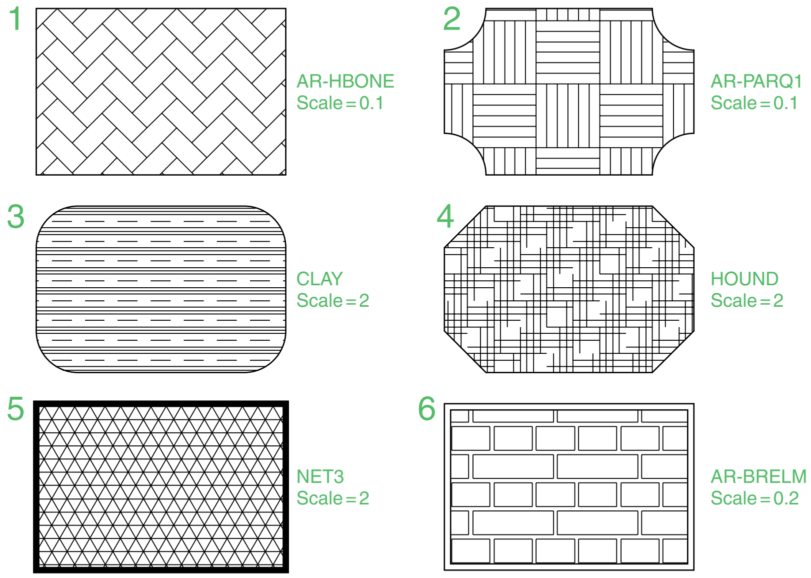


Fig. 8.2 Some hatch patterns from **Predefined** hatch patterns

First example – hatching a sectional view (Fig. 8.3)

Fig. 8.3 shows a two-view orthographic projection which includes a sectional end view. Note the following in the drawing:

1. The section plane line, consisting of a centre line with its ends marked **A** and an arrow showing the direction of viewing to obtain the sectional view.

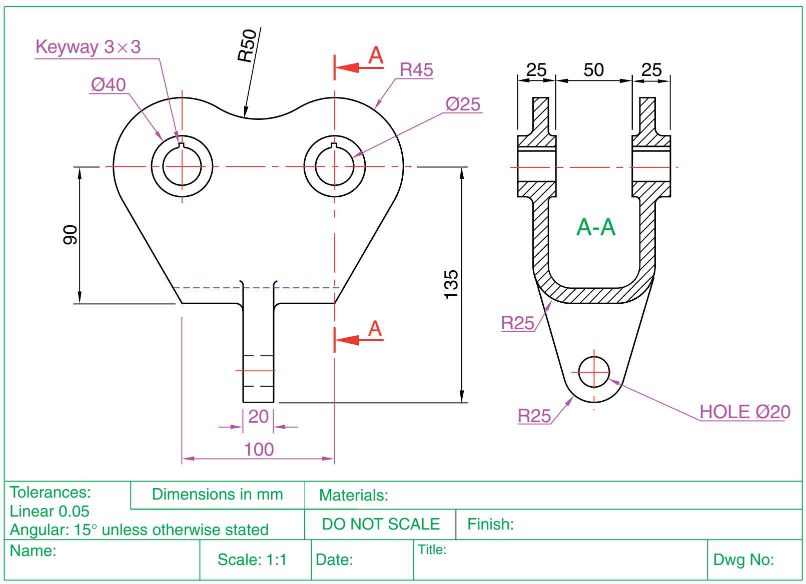


Fig. 8.3 First example – **Hatching**

2. The sectional view labelled with the letters of the section plane line.
3. The cut surfaces of the sectional view hatched with the **ANSI31** hatch pattern, which is in general use for the hatching of engineering drawing sections.

Second example – hatching rules (Fig. 8.4)

Fig. 8.4 describes the stages in hatching a sectional end view of a lathe tool holder. Note the following in the section:

1. There are two angles of hatching to differentiate in separate parts of the section.
2. The section follows the general rule that parts such as screws, bolts, nuts, rivets, other cylindrical objects, webs and ribs and other such features are shown within sections as outside views.

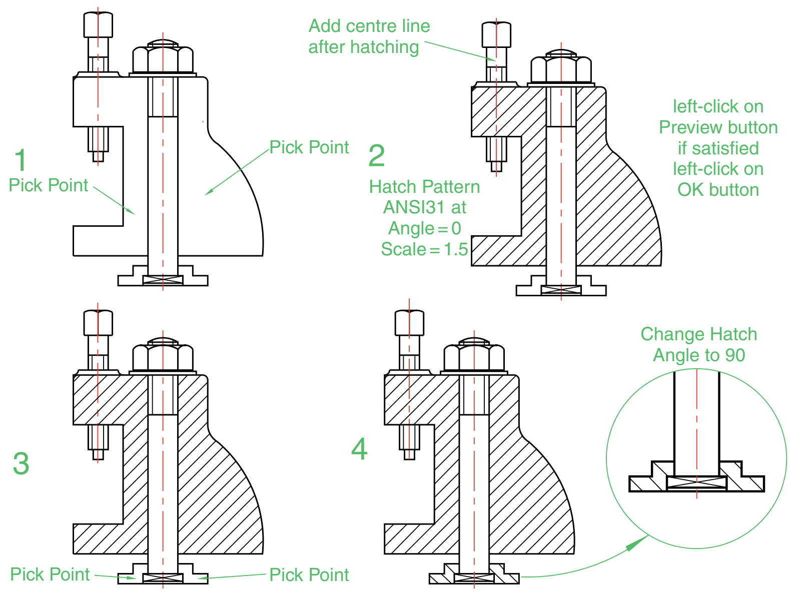


Fig. 8.4 Second example – hatching rules for sections

Third example – Associative hatching (Fig. 8.5)

Fig. 8.5 shows two end views of a house. After constructing the left-hand view, it was found that the upper window had been placed in the wrong position. Using the **Move** tool, the window was moved to a new position. The brick hatching automatically adjusted to the new position. Such **Associative** hatching is possible only if the check box against **Associative** in the **Options** area of the **Hatch and Gradient** dialog is **ON** – a tick in the check box (Fig. 8.6).

Fourth example – Colour gradient hatching (Fig. 8.9)

Fig. 8.8 shows two examples of hatching from the **Gradient** sub-dialog of the **Hatch and Gradient** dialog.

Fig. 8.5 Third example – **Associative** hatching

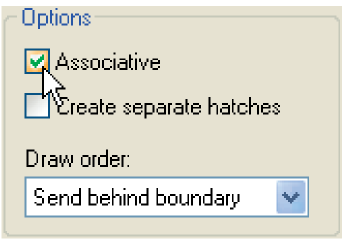


Fig. 8.6 **Associative** hatching set ON in the **Hatch and Gradient** dialog

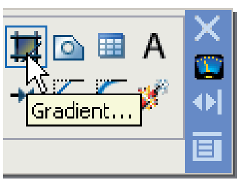


Fig. 8.7 The **Gradient Hatch** tool icon from the **2D Draw control panel**

1. Construct two outlines each consisting of six rectangles (Fig. 8.9).
2. *Click* the **Gradient** . . . tool icon in the **2D Draw control panel** (Fig. 8.7) or in the **Draw** toolbar. In the **Hatch and Gradient** dialog which appears (Fig. 8.8) *pick* one of the gradient choices, followed with a *click* on the **Pick an internal point** button. *Click* one of the color panels in the dialog and when the dialog disappears, *pick* a single area of one of the rectangles in the left-hand drawing, followed by a *click* on the dialog's **OK** button when the dialog reappears.
3. Repeat in each of the other rectangles of the left-hand drawing, changing the pattern in each of the rectangles.

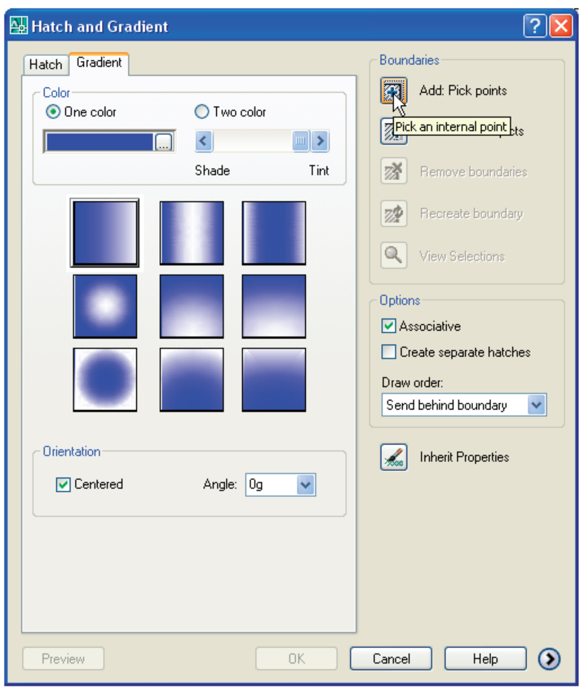


Fig. 8.8 The **Hatch and Gradient** dialog

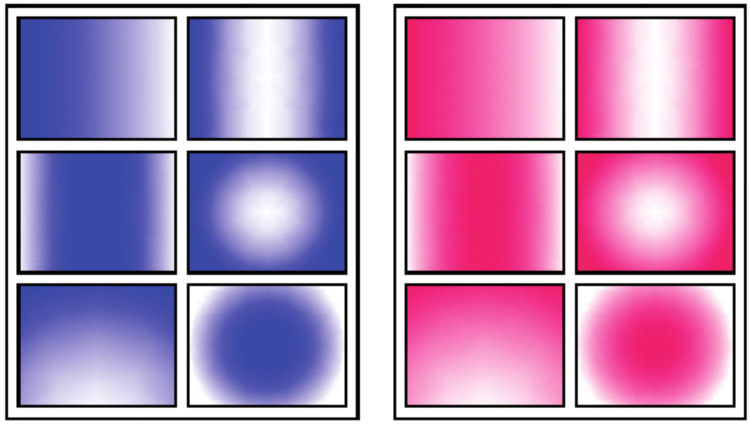


Fig. 8.9 Fourth example – **Colour gradient** hatching

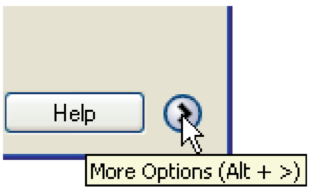


Fig. 8.10 The **More Options** arrow of the **Hatch and Gradient** dialog

4. *Click* the button (. . .) to the right of the **Color** field, select a new colour from the **Select Color** dialog which appears and repeat steps **3** and **4**.

The result is shown in Fig. 8.9.

Note

If the **Two color** radio button is set on (dot in circle) the colours involved in the gradient hatch can be changed by *clicking* the button marked with three dots (. . .) on the right of the colour field. This brings a **Select Color** dialog on screen, which offers three choices of sub-dialogs from which to select colours.

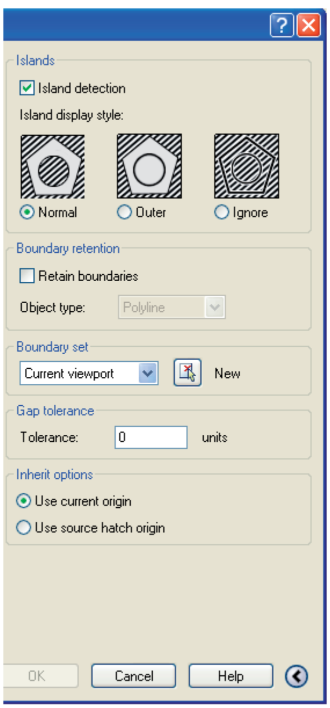


Fig. 8.11 The **Island display style** selections in the expanded **Hatch and Gradient** dialog

Fifth example – Advanced hatching (Fig. 8.12)

If the arrow at the bottom right-hand corner of the **Hatch and Gradient** dialog is *clicked* (Fig. 8.10) the dialog expands to show the **Island display** selections (Fig. 8.11).

1. Construct a drawing which includes three outlines as shown in the left-hand drawing of Fig. 8.12 and copy it twice to produce three identical drawings.
2. Select the hatch pattern **HONEY** at an angle of **0** and scale **1**.
3. *Click* in the **Normal** radio button of the **Island display style** area.
4. *Pick* a point in the left-hand drawing. The drawing hatches as shown.
5. Repeat in the centre drawing with the radio button of the **Outer** style set on (dot in button).
6. Repeat in the right-hand drawing with **Ignore** set on.

Sixth example – Text in hatching (Fig. 8.13)

1. Construct a pline rectangle using the sizes given in Fig. 8.13.
2. In the **Text Style Manager** dialog, set the text font to **Arial** and its **Height = 25**.
3. Using the **Dtext** tool *enter* the text as shown central to the rectangle.

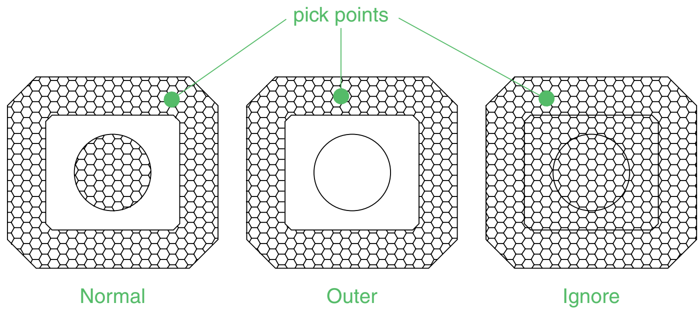


Fig. 8.12 Fifth example – **Advanced** hatching

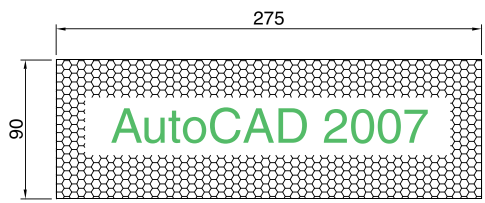


Fig. 8.13 Sixth example – Text in hatching

4. Hatch the area using the **HONEY** hatch pattern set to an angle of **0** and scale of **1**.

The result is shown in Fig. 8.13.

Note

Text will be entered with a surrounding boundary area free from hatching providing the **Advanced Normal** radio button is set on.

Seventh example – Advanced hatching (Fig. 8.20)

1. Open the **Layer Properties Manager** with a *click* on the **Layer . . .** icon in the **Layer** toolbar.

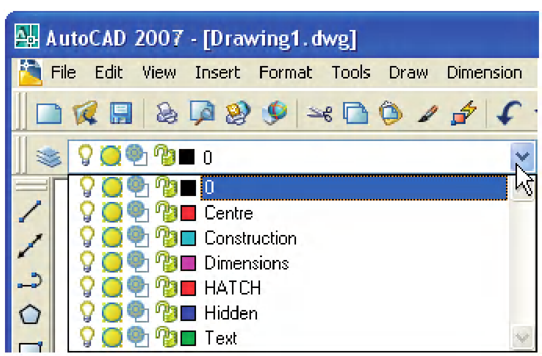


Fig. 8.14 Seventh example – the layers setup for the advanced hatch example

2. Make an extra layer (**HATCH**) as shown in Fig. 8.14.
3. With the layer **0** current construct the outline as given in Fig. 8.15.
4. Make layer **Text** current and construct the lines as shown in Fig. 8.16.

Fig. 8.15 Seventh example – construction on layer **0**

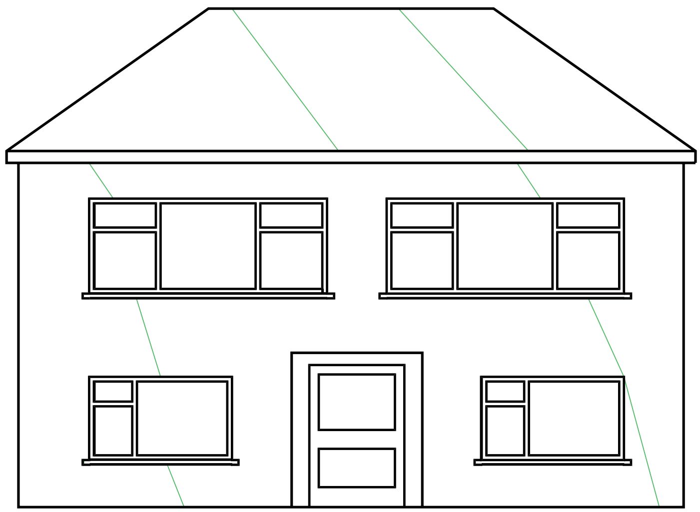


Fig. 8.16 Seventh example – construction on layer **Text**

5. Make the layer **HATCH** current and add hatching to the areas shown in Fig. 8.17 using the hatch patterns **ANGLE** at scale **2** for the roof and **BRICK** at a scale of **0.75** for the wall.
6. Finally turn the layer **Text** off. The result is given in Fig. 8.18.

Revision notes

1. A large variety of hatch patterns are available when working with AutoCAD 2007.
2. In sectional views in engineering drawings it is usual to show items such as bolts, screws, other cylindrical objects, webs and ribs as outside views.

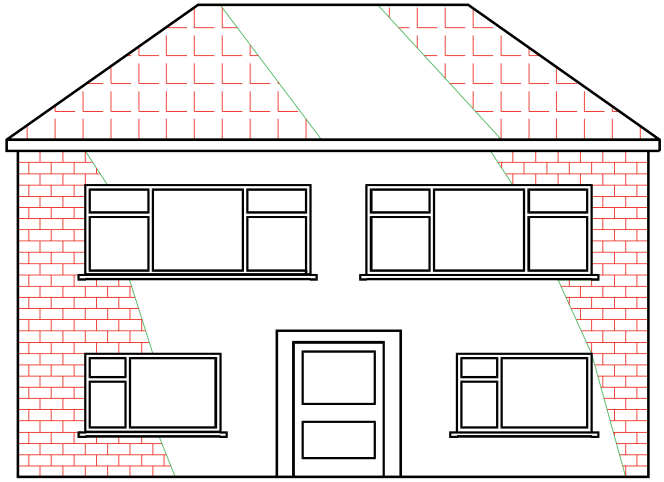


Fig. 8.17 Seventh example – construction on layer **HATCH**

Fig. 8.18 Seventh example – the finished drawing

3. When Associative hatching is set on and an object is moved within a hatched area, the hatching accommodates to fit around the moved object.
4. Colour gradient hatching is available in AutoCAD 2007.
5. When hatching takes place around text, a space around the text will be free from hatching.

Exercises

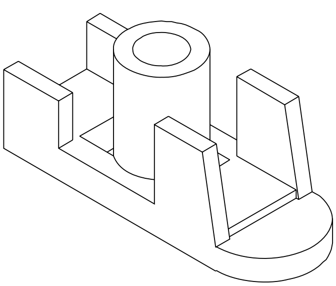


Fig. 8.19 Exercise 1 – a pictorial view

1. Fig. 8.19 shows a pictorial drawing of the component shown in the three-view orthographic projection in Fig. 8.20. Construct the three views, with the front view as a sectional view based on the section plane **A-A**.
2. Construct the three-view orthographic projection in Fig. 8.21 to the given dimensions with the front view as the sectional view **A-A**.

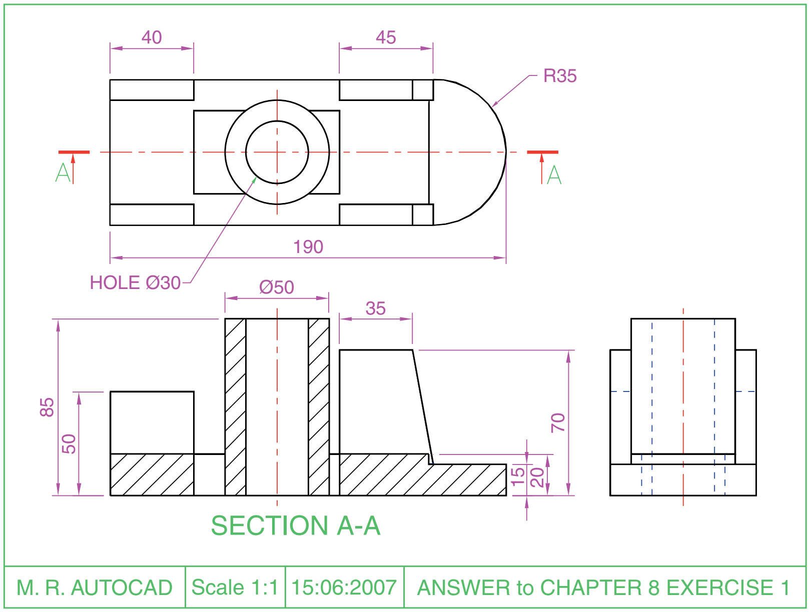


Fig. 8.20 Exercise 1

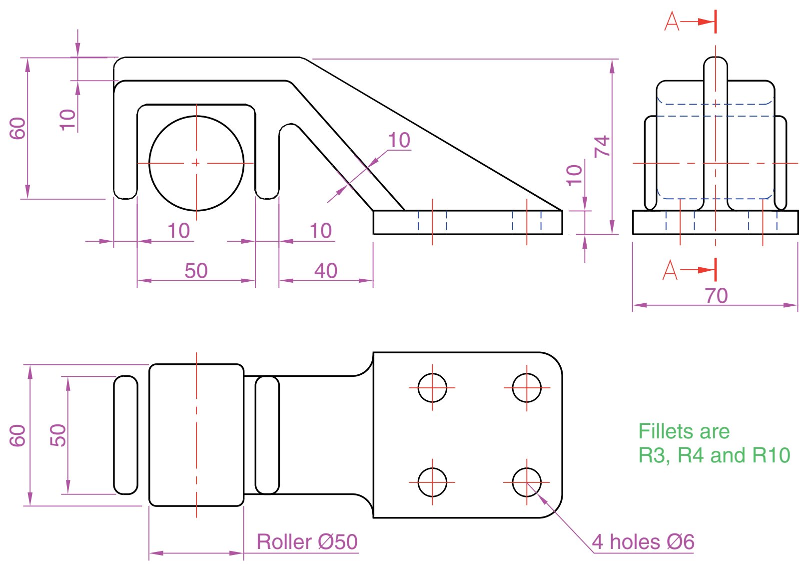


Fig. 8.21 Exercise 2

3. Construct the drawing in **Stage 5** following the descriptions of stages given in Fig. 8.22.
4. Fig. 8.23 is a front view of a car with parts hatched. Construct a similar drawing of any make of car, using hatching to emphasise the shape.
5. Working to the notes given with the drawing in Fig. 8.24, construct the end view of a house as shown. Use your own discretion about sizes for the parts of the drawing.
6. Working to dimensions of your own choice, construct the three-view projection of a two-storey house as shown in Fig. 8.25.

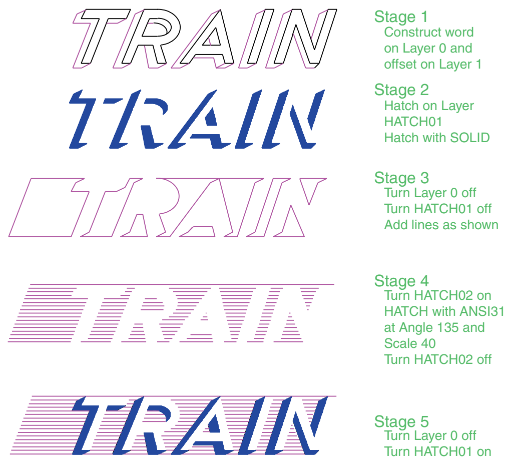


Fig. 8.22 Exercise 3

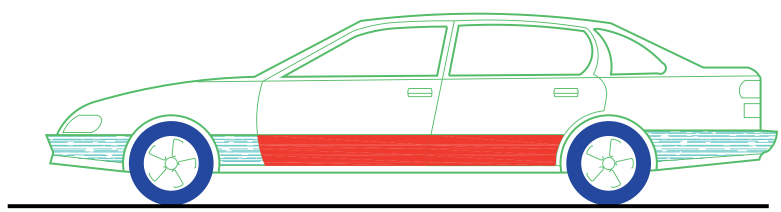


Fig. 8.23 Exercise 4

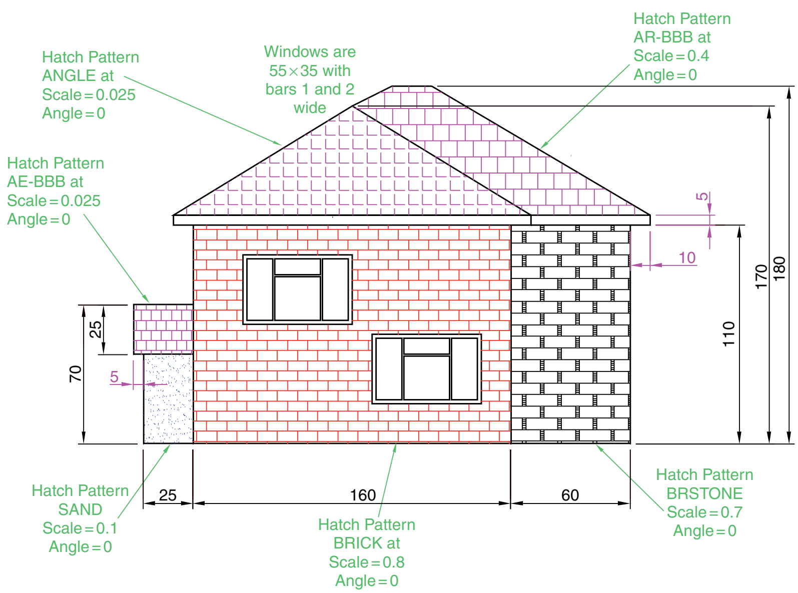


Fig. 8.24 Exercise 5

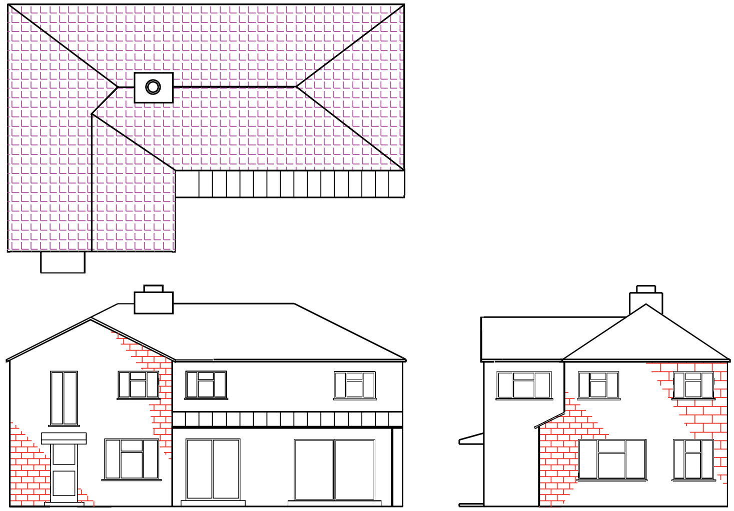


Fig. 8.25 Exercise 6

7. Construct Fig. 8.26 as follows:

(a) On layer **Text**, construct a circle of radius **90**.

(b) Make layer **0** current.

(c) Construct the small drawing to the details as shown and save as a block with a block name **shape** (see Chapter 9).

(d) Call the **Divide** tool by *entering* **div** at the command line:

Command: *enter* **div** *right-click*
Select object to divide: *pick* the circle
Enter number of segments or [Block]: *enter* **b** *right-click*
Enter name of block to insert: *enter* **shape** *right-click*
Align block with object? [Yes/No] <Y>: *right-click*
Enter the number of segments: *enter* **20** *right-click*
Command:

(e) Turn the layer **Text** off.

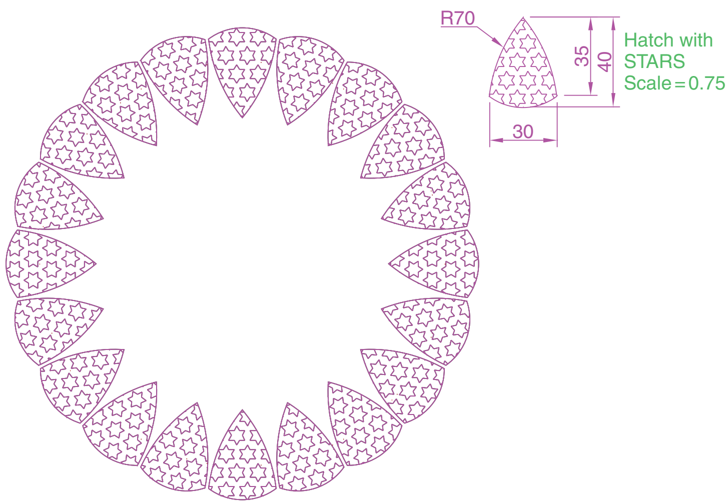


Fig. 8.26 Exercise 7

CHAPTER 9

Blocks and Inserts

Aims of this chapter

1. To describe the construction of **blocks** and **wblocks** (written blocks).
2. To introduce the insertion of blocks and wblocks into other drawings.
3. To introduce the use of the **DesignCenter** palette.
4. To explain the use of the **Explode** and **Purge** tools.

Introduction

Blocks are drawings which can be inserted into other drawings. Blocks are contained in the data of the drawing in which they have been constructed. Wblocks (written blocks) are saved as drawings in their own right, but can be inserted into other drawings if required.

Blocks

First example – Blocks (Fig. 9.3)

1. Construct the building symbols as shown in Fig. 9.1 to a scale of 1:50.
2. *Left-click* the **Make Block** tool (Fig. 9.2). The **Block Definition** dialog (Fig. 9.3) appears. To make a block from the **Double bed** symbol drawing:
   (a) *Enter* **double bed** in the **Name** field.
   (b) *Click* the **Select objects** button. The dialog disappears. *Window* the drawing of the double bed. The dialog reappears. Note the icon of the double bed in the top right-hand corner of the dialog.

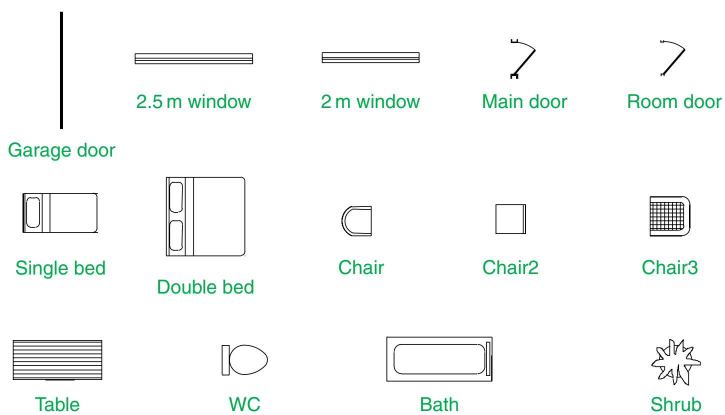


Fig. 9.1 First example – **Blocks** – symbols to be saved as blocks

(c) *Click* the **Pick point** button. The dialog disappears. *Click* a point on the double bed drawing to determine its **insertion point**. The dialog reappears. Note that the coordinates of the insertion point appears in the **Base point** area of the dialog.

(d) If thought necessary *enter* a description in the **Description** field of the dialog.

The drawing is now saved as a **block** in the drawing.

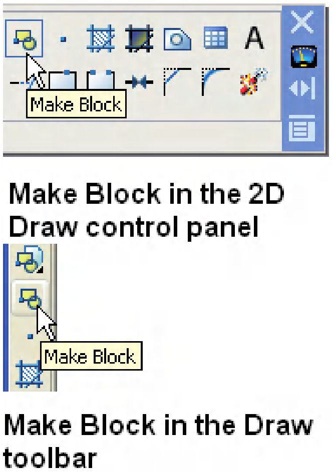


Fig. 9.2 *Click* the **Make Block** tool icon in either the **2D Draw control panel** or the **Draw** toolbar

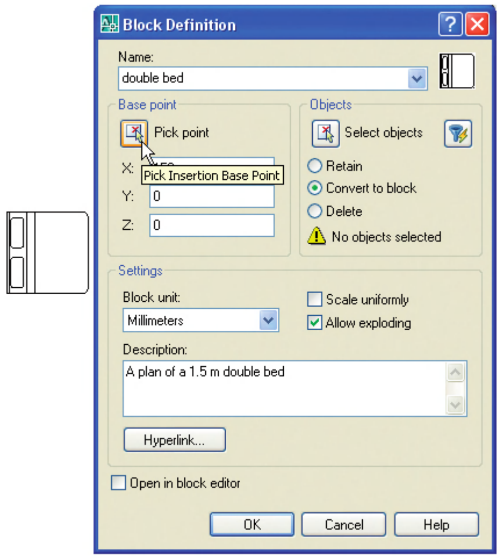


Fig. 9.3 The **Block Definition** dialog with entries for the **double bed**

3. Repeat steps **1** and **2** to make blocks of all the other symbols in the drawing.
4. Open the **Block Definition** dialog again and *click* the arrow on the right of the **Name** field. The blocks saved in the drawing appear in a popup list (Fig. 9.4).

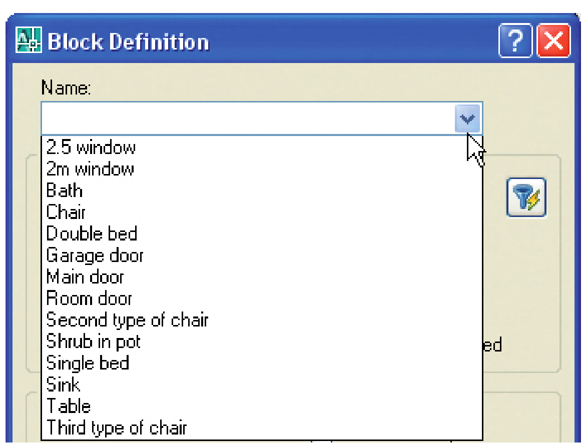


Fig. 9.4 The popup list in the **Name** field showing all blocks saved in the drawing

Inserting blocks into a drawing

There are two methods by which symbols saved as blocks can be inserted into another drawing.

Example – first method of inserting blocks

Ensuring that all the symbols saved as blocks using the **Make Block** tool are saved in the data of the drawing in which the symbols were constructed, erase the drawings of the symbols and in their place construct the outline of the plan of a bungalow to a scale of 1:50 (Fig. 9.5). Then:

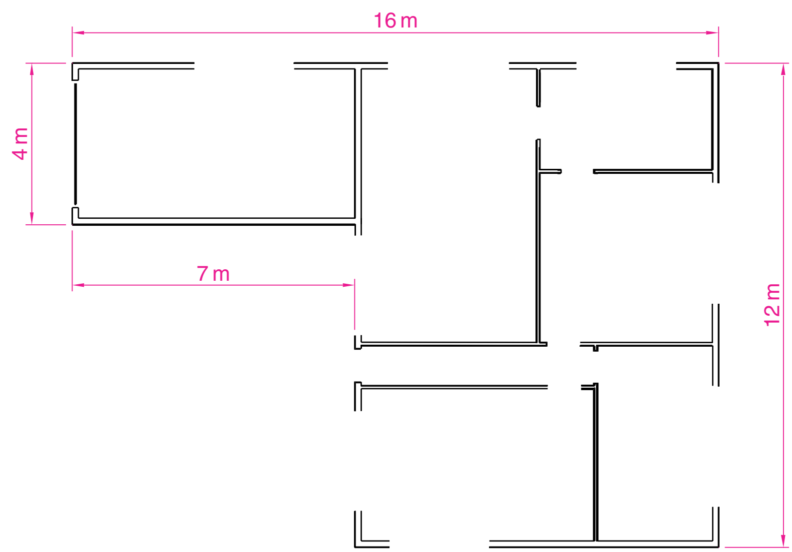


Fig. 9.5 First example – inserting blocks. Outline plan

1. *Left-click* the **Block** tool icon in the **2D Draw control panel** or the **Insert Block** tool in the **Draw** toolbar (Fig. 9.6). The **Insert** dialog appears on screen (Fig. 9.7). From the **Name** popup list select the name of the block which is to be inserted, in this example the **2.5 window**.

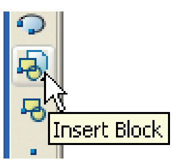


Fig. 9.6 The **Insert Block** tool icon in the **Draw** toolbar

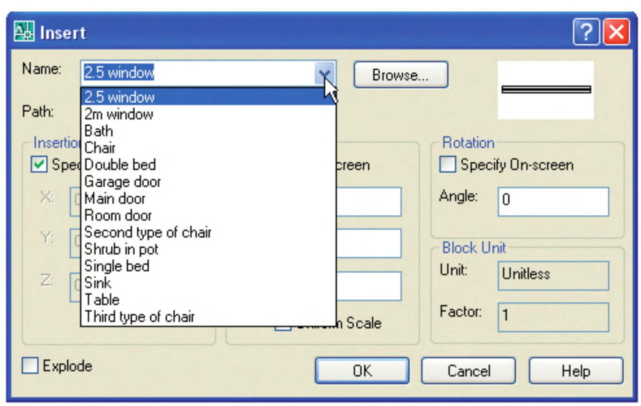


Fig. 9.7 The **Insert** dialog with its **Name** popup list displaying the names of all blocks in the drawing

2. Make sure the check box against **Explode** is off (no tick in box). *Click* the dialog's **OK** button, the dialog disappears. The symbol drawing appears with its insertion point at the intersection of the cursor hairs ready to be *dragged* into its position in the plan drawing.
3. Once all the block drawings are placed, their positions can be adjusted. Blocks are single objects and can thus be dragged into new positions as required under mouse control. Their angle of position can be amended at the command line, which shows:

   Command:
   INSERT
   Specify insertion point or [Basepoint/Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]: *enter* **r** (Rotate) *right-click*
   Specify insertion angle: *enter* **180** *right-click*
   Specify insertion point: *pick*
   Command:

   Selection from these prompts allows scaling, stretching along any axis, previewing, etc. as the block is inserted.
4. Insert all necessary blocks and add other details as required to the plan outline drawing. The result is given in Fig. 9.8.

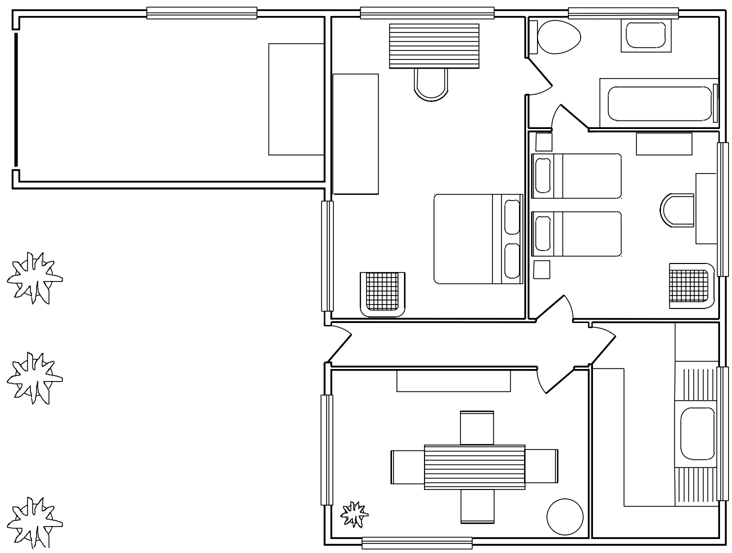


Fig. 9.8 Example – first method of inserting blocks

Example – second method of inserting blocks

1. Save the drawing which includes all the blocks to a suitable file name (building_symbols.dwg). Remember this drawing includes data of the blocks in its file.
2. *Left-click* **DesignCenter** in the **Palettes** sub-menu of the **Tools** drop-down menu (Fig. 9.9). The **DesignCenter** palette appears on screen (Fig. 9.10).

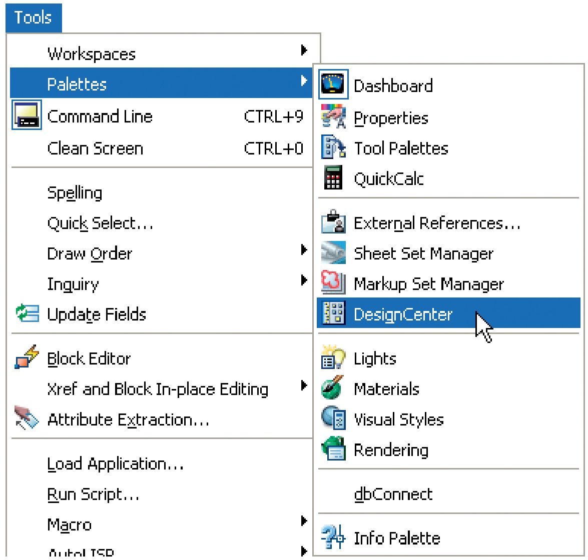


Fig. 9.9 Selecting **DesignCenter** from the **Palettes** sub-menu of the **Tools** drop-down menu

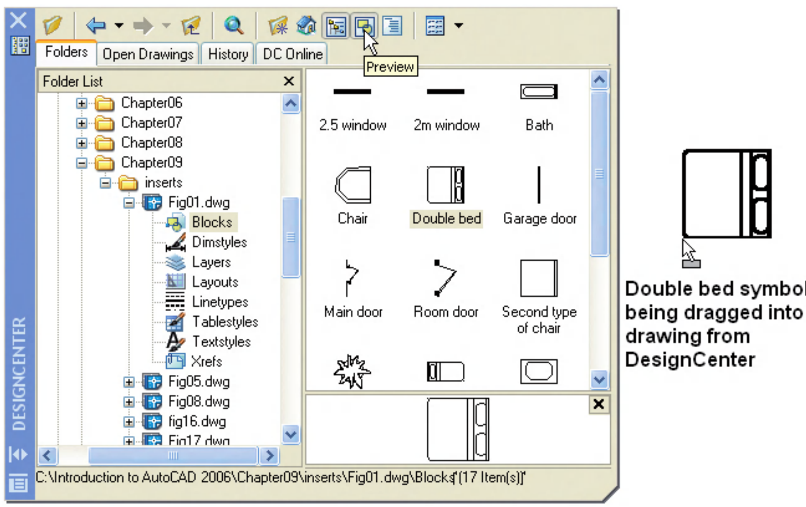


Fig. 9.10 The **DesignCenter** with the double bed block *dragged* on screen

3. With the outline plan (Fig. 9.5) on screen the symbols can all be *dragged* into position from the **DesignCenter**.

Notes about DesignCenter palette

1. As with other palettes, the **DesignCenter** palette can be re-sized by *dragging* the palette to a new size from its edges or corners.

2. *Clicks* on one of the three icons at the top-right corner of the palette (Fig. 9.11) have the following results:

(a) Tree View Toggle – changes from showing two areas – a **Folder List** and icons of the blocks within a file – to a single area showing the block icons.

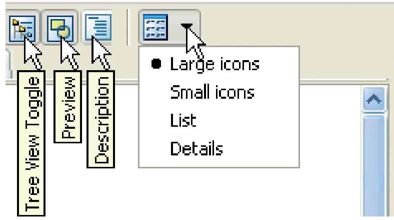


Fig. 9.11 The icons at the top-right corner of the **DesignCenter** palette

(b) Preview – a *click* on the icon opens a small area at the base of the palette showing an enlarged view of the selected block icon (Fig. 9.12).

(c) Description – a *click* on the icon opens another small area with a description of the block.

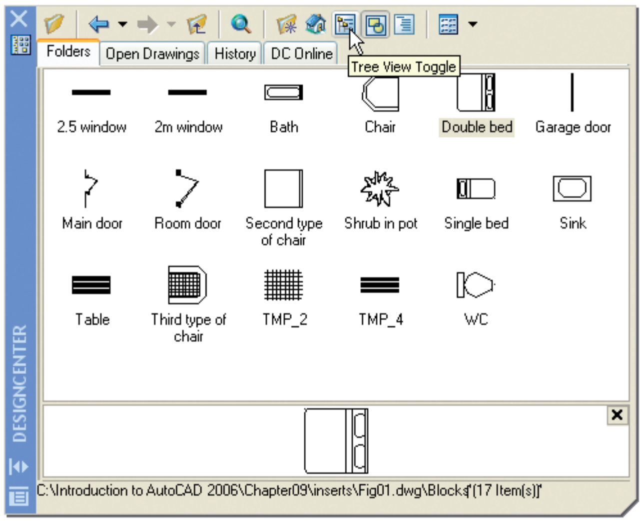


Fig. 9.12 The results of *clicks* on **Tree View Toggle** and **Preview**

The Explode tool

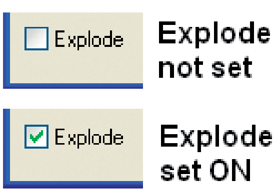


Fig. 9.13 The **Explode** check box in the **Insert dialog**

A block is a single object no matter from how many objects it was originally constructed. This enables a block to be *dragged* about the drawing area as a single object.

A check box in the bottom left-hand corner of the **Insert** dialog is labelled **Explode** (Fig. 9.13). If the check box is ticked, **Explode** will be set on and when a block is inserted it will be exploded into the objects from which it was constructed.

Another way of exploding a block would be to use the **Explode** tool from the **DASHBOARD** palette (Fig. 9.14). A *click* on the icon or *entering* **ex** at the command line brings prompts into the command line:

Command:_explode
Select objects: *pick* a block on screen **1 found**.
Select objects: *right-click*
Command:

And the *picked* object is exploded into its original objects.

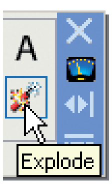


Fig. 9.14 The **Explode** tool icon in the **DASHBOARD** palette

The Purge tool

The **Purge** tool can be called by *entering* **pu** at the command line or from the **Drawing Utilities** sub-menu of the **File** drop-down menu (Fig. 9.15). When the tool is called the **Purge** dialog appears on screen (Fig. 9.16).

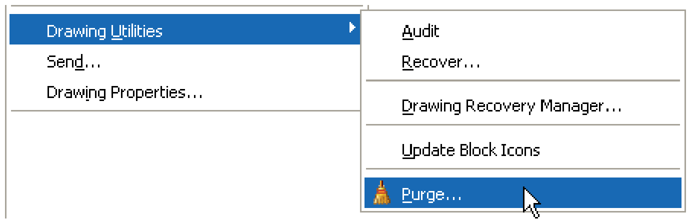


Fig. 9.15 Calling **Purge** from the **Drawing Utilities** sub-menu of the **File** drop-down menu

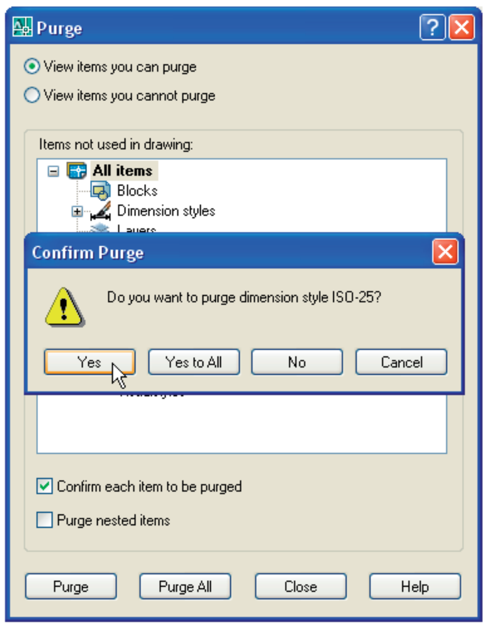


Fig. 9.16 The **Purge** dialog

The **Purge** tool can be used to remove the data of blocks within a drawing thus saving file space when a drawing which includes blocks is saved to disk.

To use the tool, in its dialog *click* the **Purge** button and a sub-dialog appears naming a block to be purged. A *click* on the **Yes** button clears the data of the block from the drawing. Continue until all blocks that are to be purged are removed.

Take the drawing in Fig. 9.8 (page 146) as an example. If all the blocks are purged from the drawing, the file will be reduced from **145** Kbytes to **67** Kbytes when the drawing is saved to disk.

Example using the DesignCenter (Fig. 9.19)

1. Construct the set of electric/electronic circuit symbols shown in Fig. 9.17 and make a series of blocks from each of the symbols.

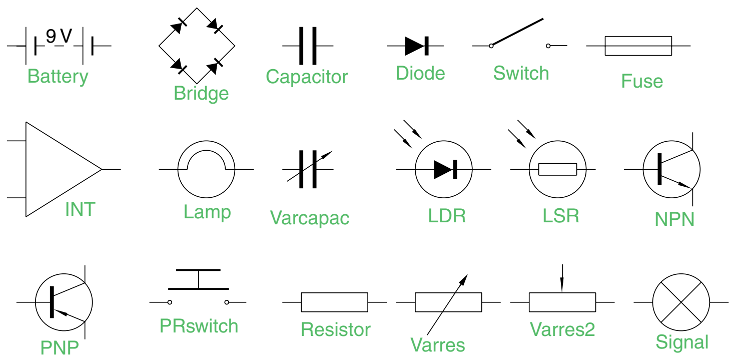


Fig. 9.17 Example using the **DesignCenter** – a set of electric/electronic symbols

2. The drawing has been saved to a file **Fig. 17.dwg**.
3. Open the **acadiso.dwt** template. Open the **DesignCenter** with a *click* on its icon in the **Standard** toolbar.
4. From the **Folder list** select the file **Fig. 17.dwg** and *click* on **Blocks** under its file name. Then *drag* the symbol icons from the **DesignCenter** into the drawing area as shown in Fig. 9.18. Ensure they are placed in appropriate positions in relation to each other to form a circuit. If necessary either **Move** or/and **Rotate** the symbols into correct positions.

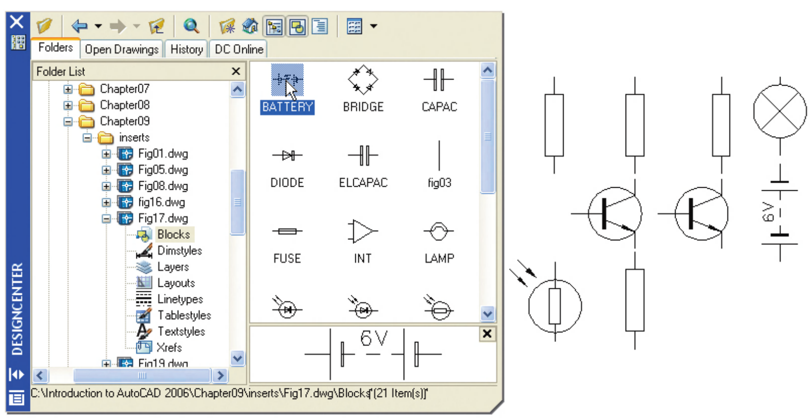


Fig. 9.18 Example using the **DesignCenter**

5. Close the **DesignCenter** palette with a *click* on the **X** in the top left-hand corner.
6. Complete the circuit drawing as shown in Fig. 9.19.

Note

Fig. 9.19 does not represent an authentic electronics circuit.

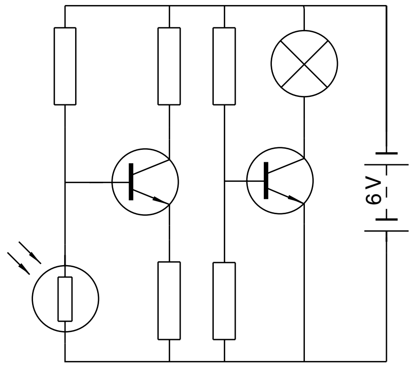


Fig. 9.19 Example using the **DesignCenter** – the completed circuit

Wblocks

Wblocks or written blocks are saved as drawing files in their own right and are not part of the drawing in which they have been saved.

Example – wblock (Fig. 9.20)

1. Construct a light emitting diode (**LED**) symbol and *enter* **w** at the command line. The **Write Block** dialog appears (Fig. 9.20).

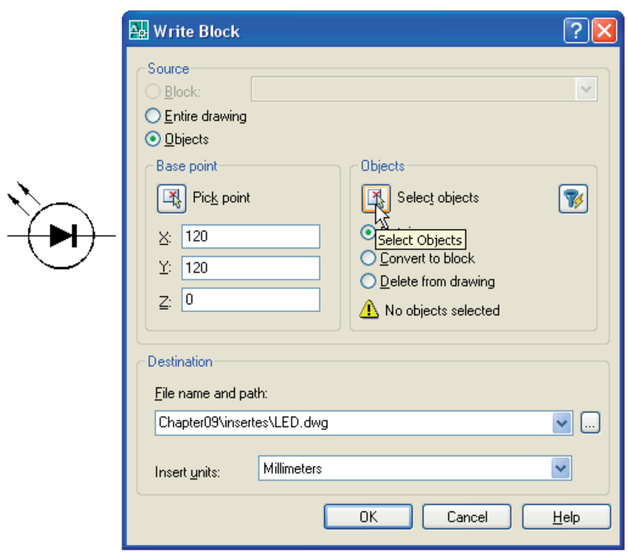


Fig. 9.20 Example – **Wblock**

2. *Click* the button marked with three dots (. . .) to the right of the **File name and path** field and from the **Browse for Drawing File** dialog which comes to screen select an appropriate directory. The directory name appears in the **File name and path** field. Add **LED.dwg** at the end of the name.
3. Make sure the **Insert units** is set to **Millimeters** in its popup list.

4. *Click* the **Select objects** button, *window* the symbol drawing and when the dialog reappears, *click* the **Pick point** button, followed by selecting the left-hand end of the symbol.
5. Finally *click* the **OK** button of the dialog and the symbol is saved in its selected directory as a drawing file **LED.dwg** in its own right.

Note on the DesignCenter

Drawings can be inserted into the AutoCAD window from the **DesignCenter** by dragging the icon representing the drawing into the window (Fig. 9.21).

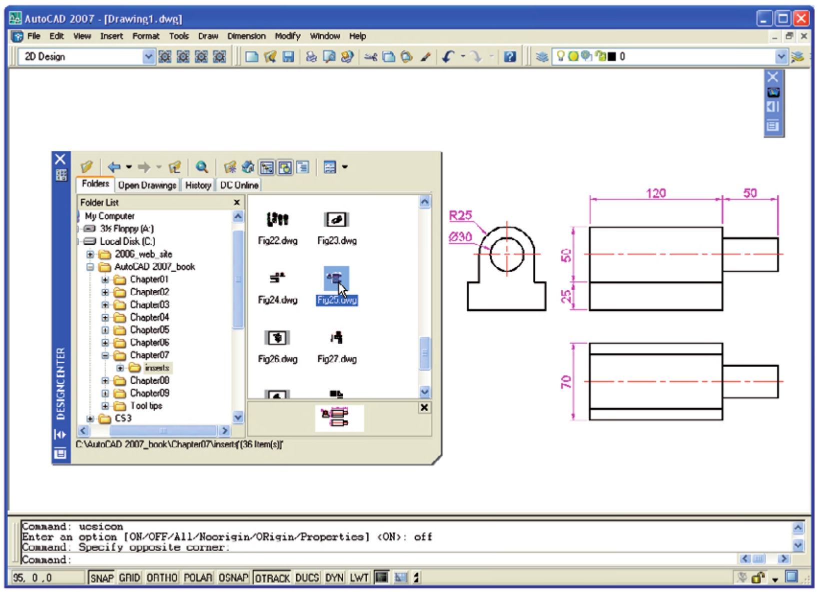


Fig. 9.21 An example of a drawing *dragged* from the **DesignCenter**

When such a drawing is *dragged* into the AutoCAD window, the command line shows a sequence such as:

Command:_INSERT Enter block name or [?] <Fig26>: 'Chapter07\inserts\Fig25.dwg'
Specify insertion point or [prompts]: *pick*
Enter X scale factor <1>: *right-click*
Enter Y scale factor <use X scale factor>: *right-click*
Specify rotation angle <0>: *right-click*
Command:

Revision notes

1. Blocks become part of the drawing file in which they were constructed.
2. Wblocks become drawing files in their own right.
3. Drawings or parts of drawings can be inserted into other drawings with the **Block** tool.

4. Inserted blocks or drawings are single objects unless either the **Explode** check box of the **Insert** dialog is checked or the block or drawing is exploded with the **Explode** tool.
5. Drawings can be inserted into the AutoCAD drawing area using the **DesignCenter**.
6. Blocks within drawings can be inserted into drawings from the **DesignCenter**.

Exercises

1. Construct the building symbols in Fig. 9.22 in a drawing saved as **symbols.dwg**. Then using the **DesignCenter** construct a building drawing of the first floor of the house you are living in, making use of the symbols. Do not bother too much about dimensions because this exercise is designed to practise using the idea of making blocks and using the **DesignCenter**.

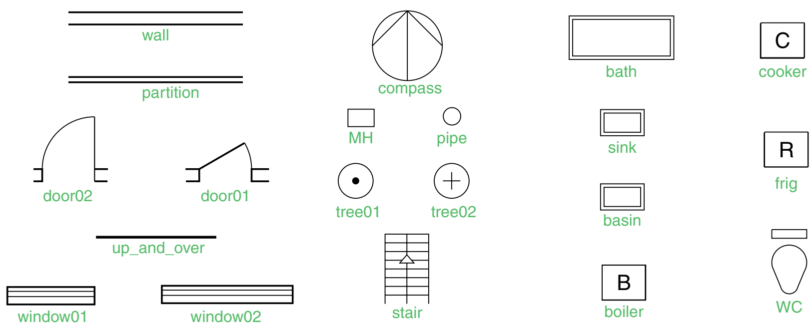


Fig. 9.22 Exercise 1

2. Construct drawings of the electric/electronics symbols in Fig. 9.17 (page 150) and save them as blocks in a drawing file **electronics.dwg**.
3. Construct the electronics circuit given in Fig. 9.23 from the file **electronics.dwg** using the **DesignCenter**.

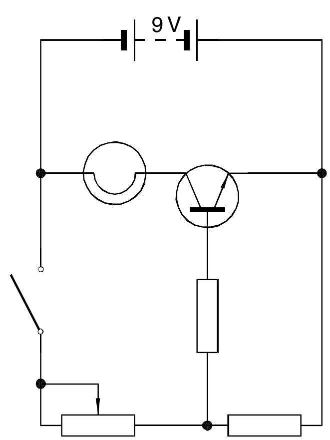


Fig. 9.23 Exercise 3

4. Construct the electronics circuit given in Fig. 9.24 from the file **Fig. 17.dwg** using the **DesignCenter**.

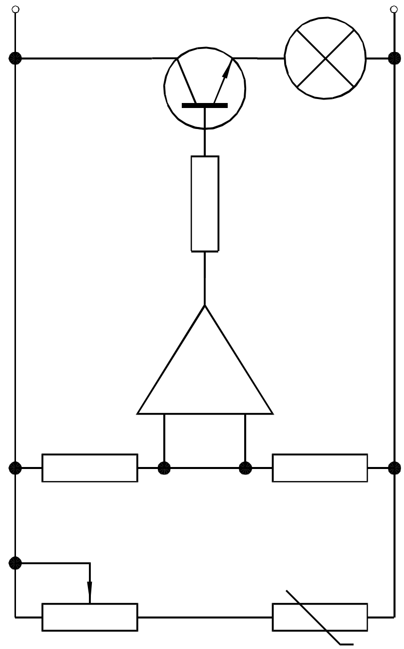


Fig. 9.24 Exercise 4

CHAPTER 10

Other types of file format

Aims of this chapter

1. To introduce Object Linking and Embedding (**OLE**) and its uses.
2. To introduce the use of Encapsulated Postscript (**EPS**) files.
3. To introduce the use of Data Exchange Format (**DXF**) files.
4. To introduce raster files.
5. To introduce **Xrefs**.

Object linking and embedding

First example – copying and pasting (Fig. 10.3)

1. Open any drawing in the AutoCAD 2007 window (Fig. 10.1).
2. *Left-click* **Copy Link** in the **Edit** drop-down menu (Fig. 10.1).
3. *Click* the AutoCAD 2007 **Minimize** button and open the **Clipboard** viewer. The copied drawing appears in the clipboard (Fig. 10.2).
4. Open **Microsoft Word** and *click* on **Paste** in the **Edit** drop-down menu (Fig. 10.3). The drawing from the **Clipboard** appears in the **Microsoft Word** document (Fig. 10.3).
5. Add text as required.

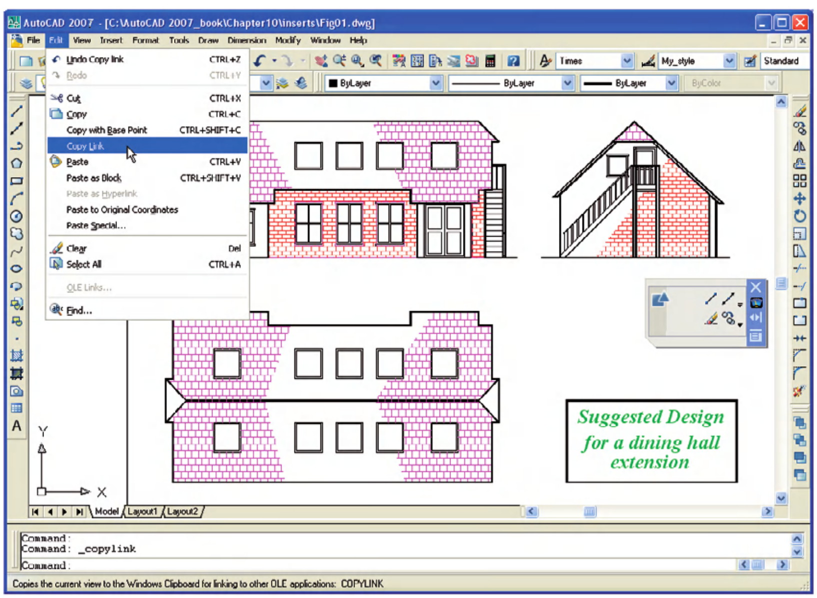


Fig. 10.1 A drawing in the AutoCAD 2007 window showing **Copy Link** selected from the **Edit** drop-down menu

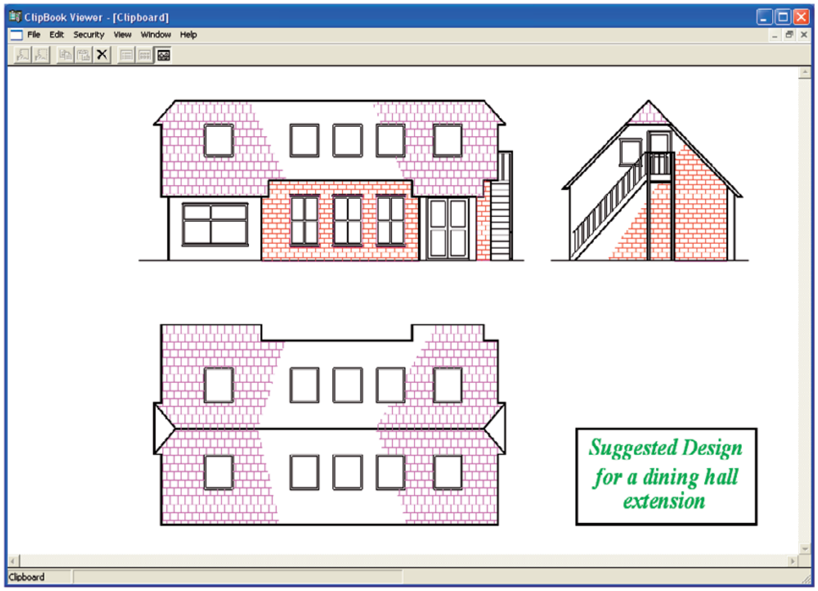


Fig. 10.2 The drawing from AutoCAD copied to the **Clipboard**

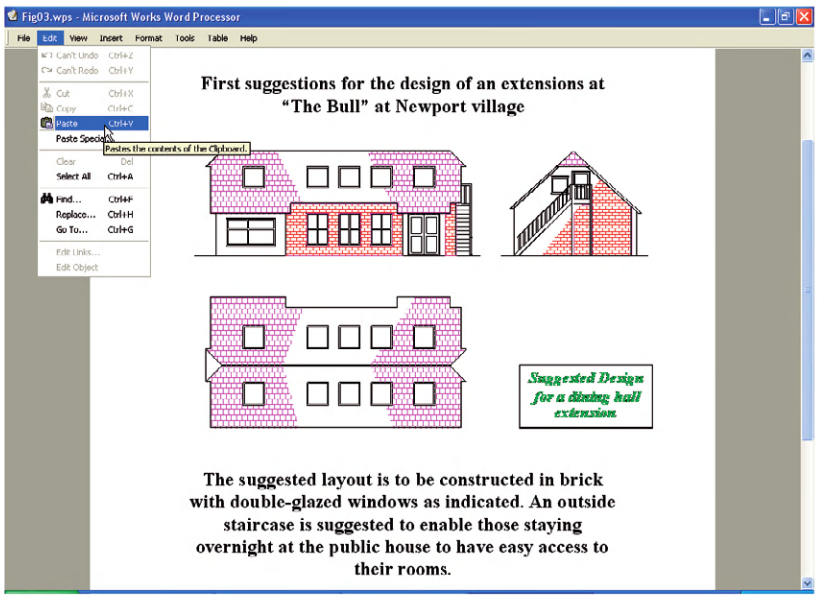


Fig. 10.3 Example – **copying and pasting**

Notes

1. It is not common practice to have a **Clipboard** window showing on screen, since it usually works in the background. It is shown opened here to display its use in acting as an agent for transposing drawings, etc. from one application to another.
2. Similar results can be obtained using the **Copy** and **Copy with Base Point** tools from the **Edit** drop-down menu of AutoCAD 2007.

3. The drawing could also be pasted back into the AutoCAD window – not that there would be much point in doing so, but anything in the **Clipboard** window can be pasted into other applications.

Second example – EPS file (Fig. 10.5)

1. With the same drawing on screen *click* on **Export** . . . in the **File** drop-down menu. The **Export Data** dialog appears (Fig. 10.4). *Pick* **Encapsulated PS (*.eps)** from the **Files of type** popup list, then *enter* a suitable file name (**building.eps**) in the **File name** field and *click* the **Save** button.

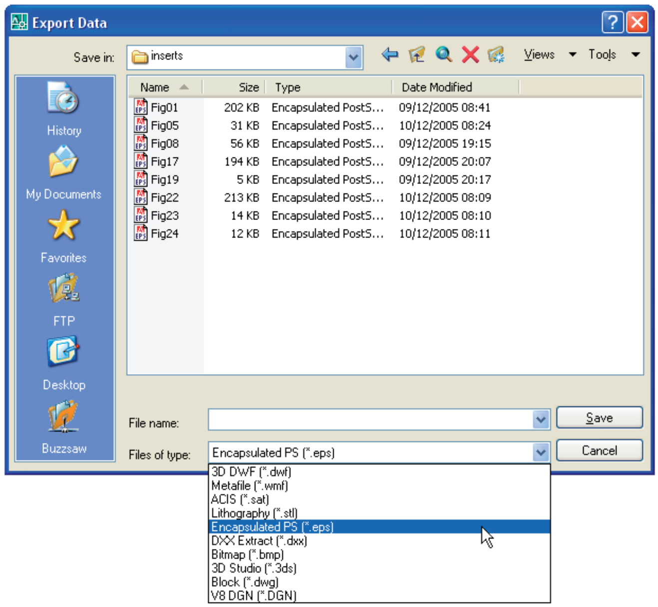


Fig. 10.4 The **Export Data** dialog of AutoCAD 2007

2. Open a desktop publishing application. That shown in Fig. 10.5 is **PageMaker**.
3. From the **File** drop-down menu *click* **Place** . . . A dialog appears listing files which can be placed in a PageMaker document. Among the files named will be **building.eps**. *Double-click* that file name and an icon appears, the placing of which determines the position of the ***.eps** file drawing in the PageMaker document (Fig. 10.5).
4. Add text as required.
5. Save the PageMaker document to a suitable file name.
6. Go back to the AutoCAD drawing and delete the title.
7. Make a new ***.eps** file with the same file name (**building.eps**).
8. Go back into **PageMaker** and *click* **Links Manager** . . . in the **File** drop-down menu. The **Links Manager** dialog appears (Fig. 10.6). Against the name of the **building.eps** file name is a dash and a note at the bottom of the dialog explaining that changes have taken place in

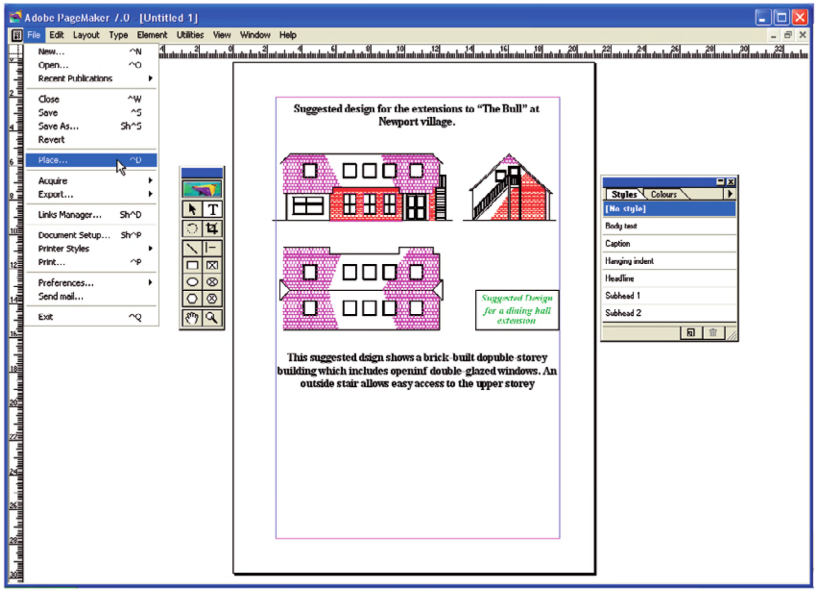


Fig. 10.5 An EPS file placed in position in a PageMaker document

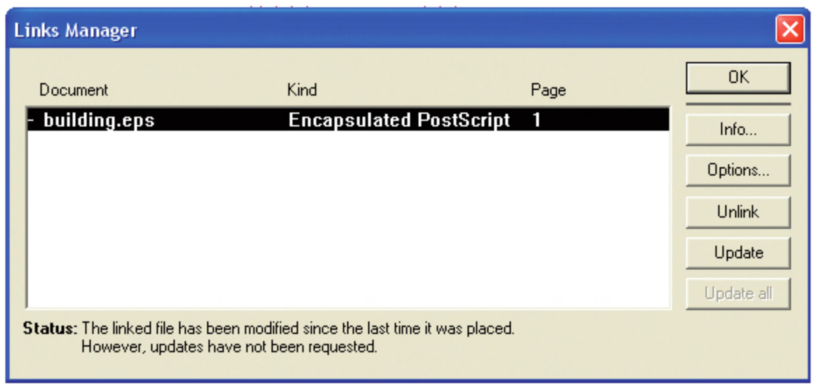


Fig. 10.6 The **Links Manager** dialog of PageMaker

the drawing from which the ***.eps** had been derived. *Click* the **Update** button and when the document reappears the drawing in PageMaker no longer includes the erased title.

Notes

1. This is **Object linking and embedding**. Changes in the AutoCAD drawing saved as an ***.eps** file are linked to the drawing embedded in another application document, so changes made in the AutoCAD drawing are reflected in the PageMaker document.
2. There is actually no need to use the **Links Manager** because if the file from PageMaker is saved with the old ***.eps** file in place, when it is reopened the file will have changed to the redrawn AutoCAD drawing, without the erased title.

DXF (Data Exchange Format) files

The ***.DXF** format was originated by Autodesk (publishers of AutoCAD), but is now in general use in most **CAD** software. A drawing saved to a ***.dxf** format file can be opened in most other CAD software applications. This file format is of great value when drawings are being exchanged between operators using different CAD applications.

Example – DXF file (Fig. 10.8)

1. Open a drawing in AutoCAD. This example is shown in Fig. 10.7.
2. *Click* on **Save As** . . . in the **File** drop-down menu and in the **Save Drawing As** dialog which appears, *click* **AutoCAD 2007 DXF [*.dxf]**.

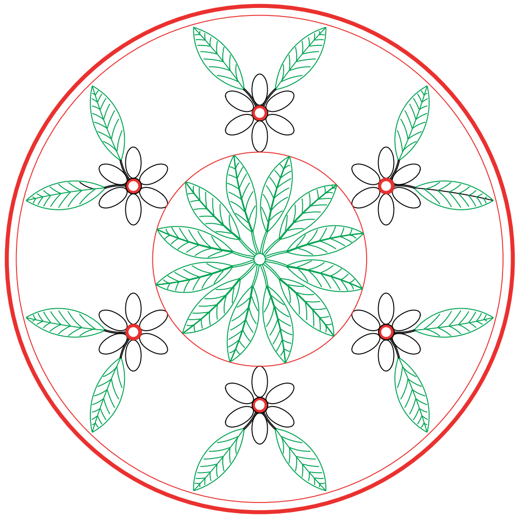


Fig. 10.7 Example – **DXF file**. Drawing to be saved as a dxf file

3. *Enter* a suitable file name. In this example this is Fig 07. The extension **.dxf** is automatically included when the **Save** button of the dialog is *clicked* (Fig. 10.8).
4. The **DXF** file can now be opened in the majority of CAD applications and then saved to the drawing file format of the CAD in use.

Note

To open a **DXF** file in AutoCAD 2007, select **Open** . . . from the **File** drop-down menu and select **AutoCAD 2007 DXF [*.dxf]** from the popup list of the **Files of type** field.

Raster images

A variety of raster files can be placed into AutoCAD 2007 drawings from the **Select Image File** dialog brought to screen with a *click* on **Raster Image** . . . in the **Insert** drop-down menu. In this example the selected raster file is a bitmap (extension ***.bmp**) of a rendered 3D model drawing

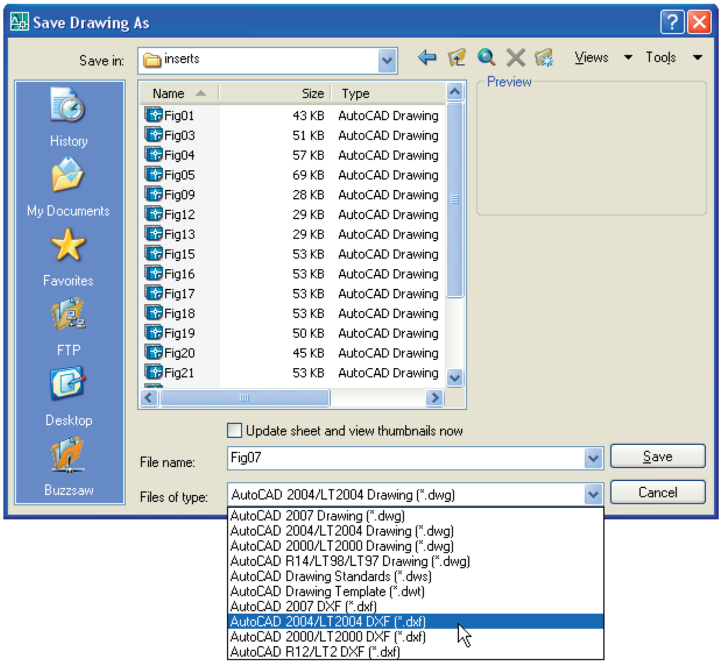


Fig. 10.8 The **Save Drawing As** dialog set up to save drawings in **DXF** format.

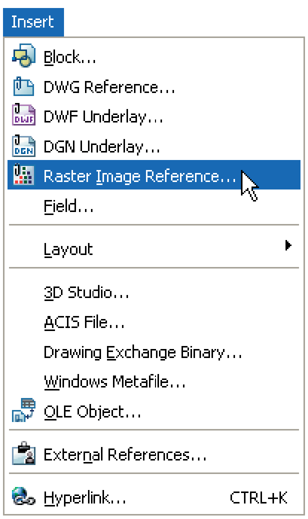


Fig. 10.9 Selecting **Raster Image Reference** . . . from the **Insert** drop-down menu

constructed to the views in an assembly drawing of a lathe tool post (see Chapter 16 about rendering 3D models).

Example – placing a raster file in a drawing (Fig. 10.13)

1. *Click* **Raster Image Reference** . . . from the **Insert** drop-down menu (Fig. 10.9). The **Select Image File** dialog appears (Fig. 10.10). *Click* the file name of the image to be inserted, in this example **rendering01.bmp**. A preview appears in the **Preview** area of the dialog.
2. *Click* the **Open** button of the dialog. The **Image** dialog appears (Fig. 10.11).
3. In the **Scale** field *enter* a suitable scale figure. The size of the image that will appear in the AutoCAD window can be seen with a *click* on the **Details** button which brings down an extension of the dialog which shows details about the resolution and size of the image.

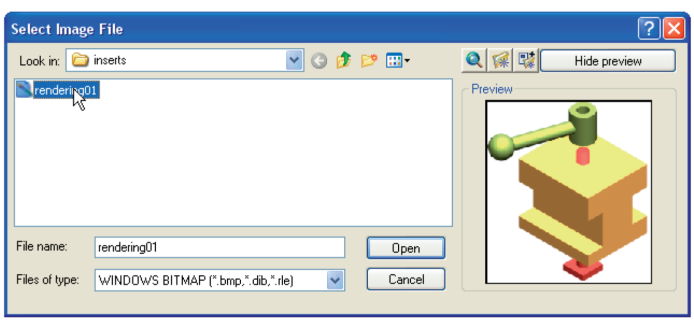


Fig. 10.10 The **Select Image File** dialog

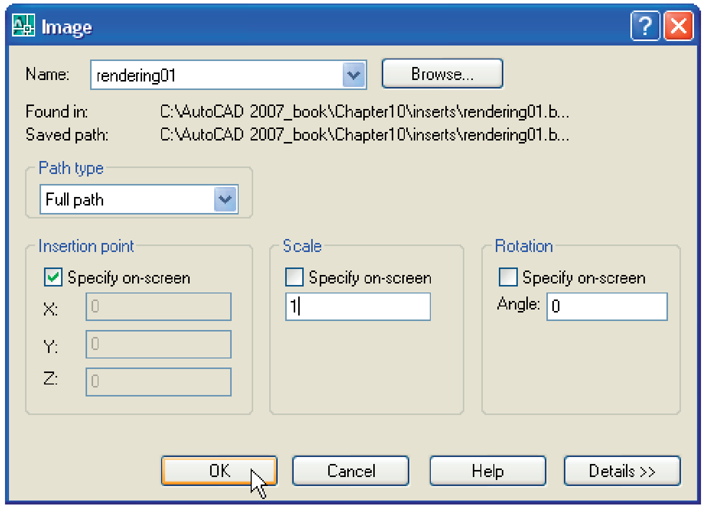


Fig. 10.11 The **Image** dialog

4. *Click* the **OK** button, the command line shows:

Command:_imageattach
Specify insertion point <0,0>: an outline of the image attached to the intersection of the cursor cross hairs appears *pick* a suitable point on screen
Command:

And the raster image appears at the *picked* point (Fig. 10.12).

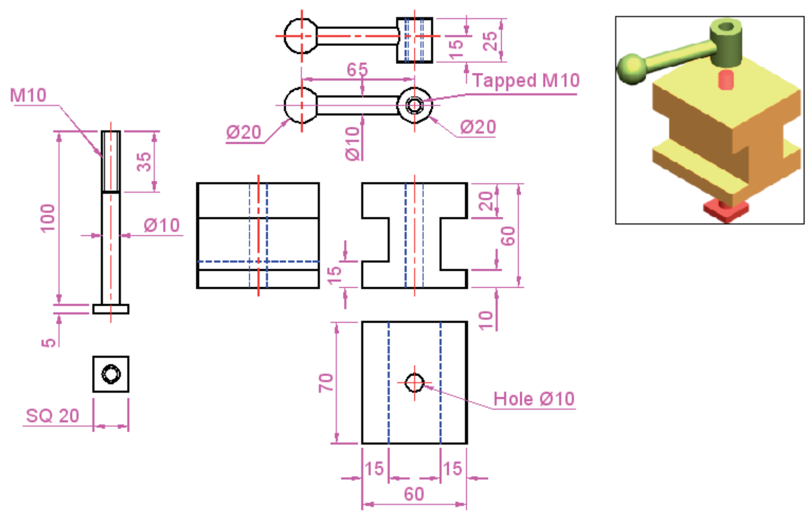


Fig. 10.12 Example – placing a raster file in a drawing

Notes

As can be seen from the **Insert** drop-down menu (Fig. 10.9) a variety of different types of raster and other images can be inserted into an AutoCAD drawing. Some examples are:

Blocks – see Chapter 9.
External References (Xrefs) – see later in this chapter.
Field name – a *click* on the name brings up the **Field** dialog. Practise inserting various categories of field names from the dialog.
Layout – a wizard appears allowing new layouts to be created and saved for new templates.
3D Studio – allows the insertion of images constructed in the Autodesk software **3D Studio** from files with the format ***.3ds**.
OLE Objects – allows raster images to be placed as OLE images from a variety of other applications.

External References (Xrefs)

If a drawing is inserted into another drawing as an external reference, any changes made in the original Xref drawing are automatically reflected in the drawing into which the Xref has been inserted.

Example – External References (Fig. 10.19)

1. Construct the three-view orthographic drawing in Fig. 10.15. Dimensions of this drawing will be found on page 263. Save the drawing to a suitable file name.
2. As a separate drawing construct Fig. 10.16. Save it as a wblock with the name of **Fig16.dwg** and with a base insertion point at one end of its centre line.
3. In the **Insert** drop-down menu *click* **External References** . . . The **EXTERNAL REFERENCES** palette appears (Fig. 10.13).
4. *Click* its **Attach** button and select **Attach DWG** from the popup list which appears when a *left-click* is held on the button. Select the

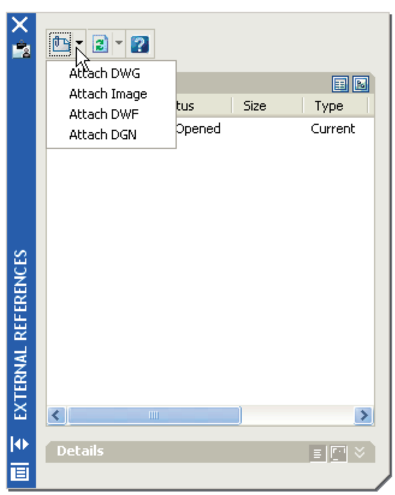


Fig. 10.13 The **EXTERNAL REFERENCES** palette

drawing of a spindle (**Fig16.dwg**) from the **Select Reference File** dialog which appears followed by a *click* on the dialog's **Open** button. This brings up the **External Reference** dialog (Fig. 10.14) showing **Fig16** in its **Name** filed. *Click* the dialog's **OK** button.

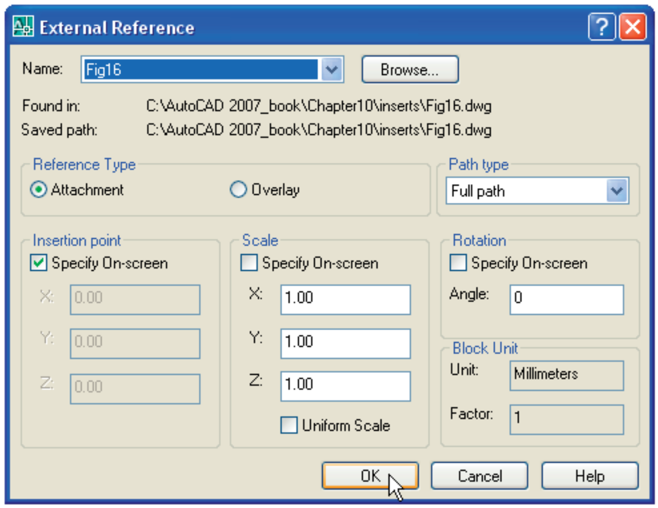


Fig. 10.14 The **External Reference** dialog

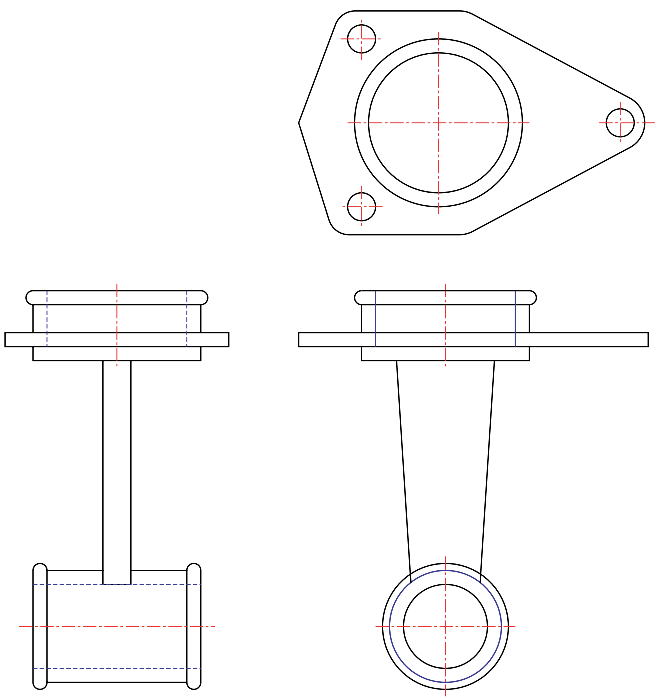


Fig. 10.15 Example – **External References** – original drawing

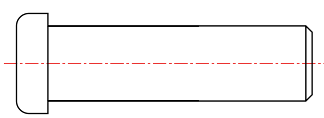


Fig. 10.16 The spindle drawing saved as **Fig16.dwg**

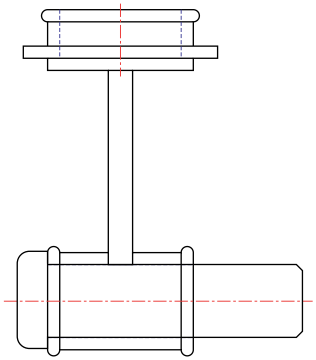


Fig. 10.17 The spindle in place in the original drawing

5. The spindle drawing appears on screen ready to be *dragged* into position. Place it in position as indicated in Fig. 10.17.
6. Save the drawing with its Xref to its original file name.
7. Open the drawing **Fig16.dwg** and make changes as shown in Fig. 10.18.
8. Now reopen the original drawing. The **external reference** within the drawing has changed in accordance with the alterations to the spindle drawing (Fig. 10.19).

Note

In this example, to ensure accuracy of drawing, the **external reference** will need to be exploded and parts of the spindle changed to hidden detail lines.

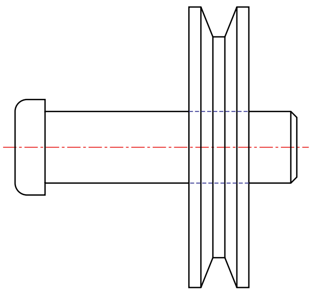


Fig. 10.18 The revised **spindle.dwg** drawing

Multiple Document Environment (MDE)

1. Open several drawings in AutoCAD, in this example four separate drawings have been opened.
2. In the **Window** drop-down menu, *click* **Tile Horizontally**. The four drawings rearrange as shown in Fig. 10.20.

Note

The names of the drawings appear in the **Window** drop-down menu, showing their directories, file names and file name extensions.

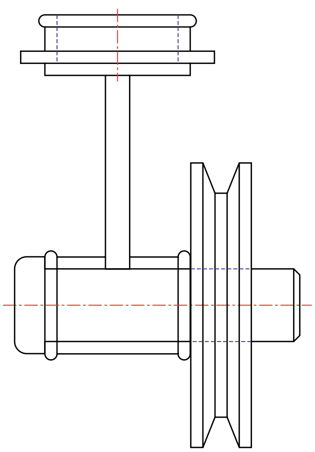


Fig. 10.19 Example – **Xrefs**

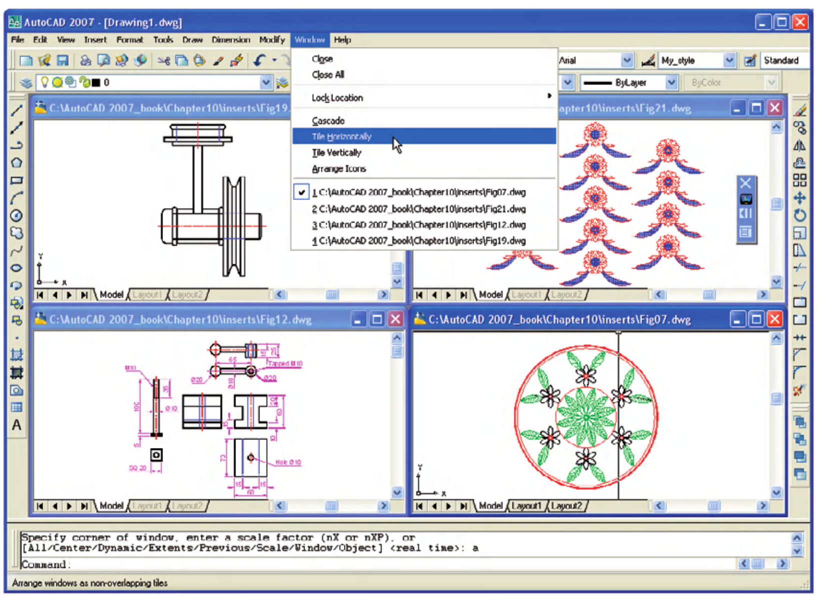


Fig. 10.20 Four drawings in the **Multiple Document Environment**

Revision notes

1. The **Edit** tools **Copy with Base Point, Copy** and **Copy Link** enable objects from AutoCAD 2007 to be copied for **Pasting** onto other applications.
2. Objects can be copied from other applications to be pasted into the AutoCAD 2007 window.
3. Drawings saved in AutoCAD as **DXF (*.dxf)** files can be opened in other CAD applications.
4. Similarly drawings saved in other CAD applications as ***.dxf** files can be opened in AutoCAD 2007.
5. **Raster** files of the format types ***.bmp**, ***.jpg**, ***.pcx**, ***.tga**, ***.tif** among other raster type file objects can be inserted into AutoCAD 2007 drawings.
6. Drawings saved to the Encapsulated Postscript (*.**eps**) file format can be inserted into documents of other applications.
7. Changes made in a drawing saved as an ***.eps** file will be reflected in the drawing inserted as an ***.eps** file in another application.
8. When a drawing is inserted into another drawing as an **external reference**, changes made to the inserted drawing will automatically be updated in the drawing into which it has been inserted.
9. A number of drawings can be opened in the AutoCAD 2007 window.

Exercises

1. Fig. 10.21 shows a pattern formed by inserting an **external reference** and then copying or arraying the **external reference**.

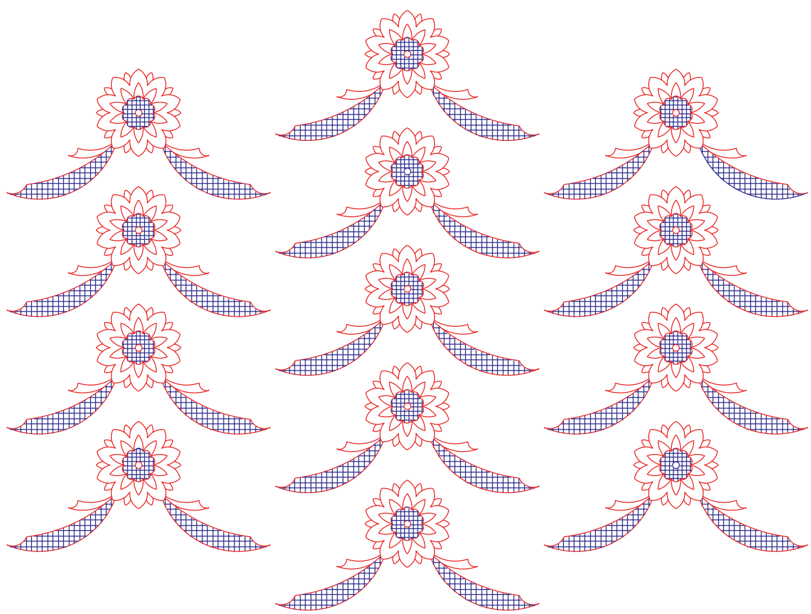


Fig. 10.21 Exercise 1 – original pattern

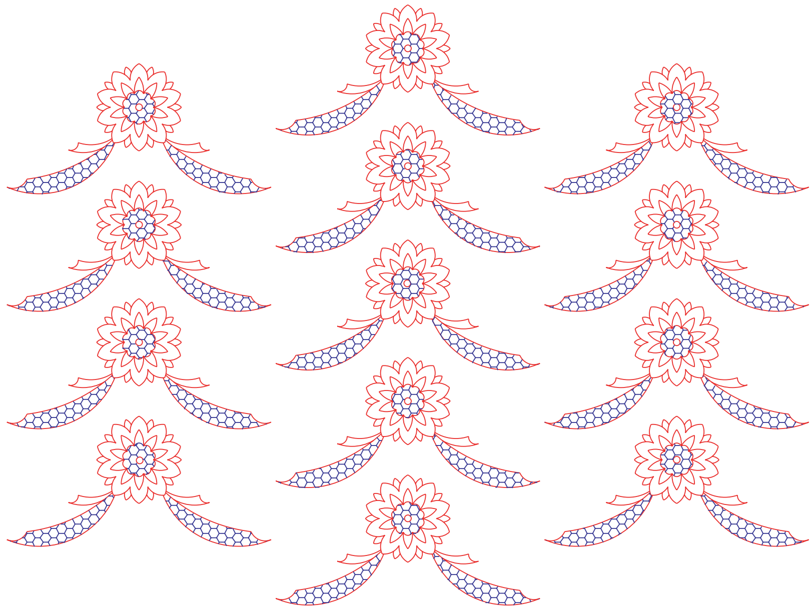


Fig. 10.22 Exercise 1

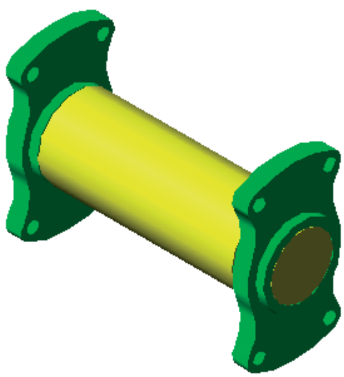


Fig. 10.23 Exercise 2 – a rendering of the holders and roller

The hatched parts of the **external reference** drawing were then changed using a different hatch pattern. The result of the change in the Xref is shown in Fig. 10.22.

Construct a similar **Xref** drawing, insert as an **Xref**, array or copy to form the pattern, then change the hatching, save the **Xref** drawing and note the results.

2. Fig. 10.23 is a rendering of a roller between two end holders. Fig. 10.24 gives details of the end holders and the roller in orthographic projections.

   Construct a full size front view of the roller and save to a file name **roller.dwg**. Then as a separate drawing construct a front view of the two end holders in their correct positions to receive the roller and save to the file name **assembly.dwg**.

   Insert the roller drawing into the assembly drawing as an **Xref**. Open the **roller.dwg** and change its outline as shown in Fig. 10.25. Save the drawing. Open the **assembly.dwg** and note the change in the inserted **Xref**.
3. *Click* **Raster Image** . . . in the **Insert** drop-down menu and insert a **JPEG** image (***.jpg** file) of a photograph into the AutoCAD 2007 window. An example is given in Fig. 10.26.
4. Using **Copy** from the **Insert** drop-down menu, copy a drawing from AutoCAD 2007 into a Microsoft Word document. An example is given in Fig. 10.27. Add some appropriate text.
5. The plan in Figs 10.1, 10.2 and 10.3 is incorrect in that some details have been missed from the drawing. Can you identify the error?

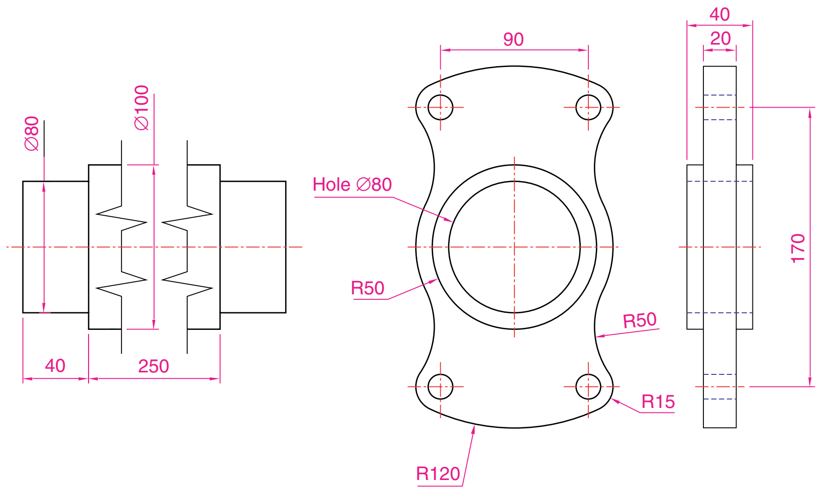


Fig. 10.24 Exercise 2 – details of the parts of the roller and holders

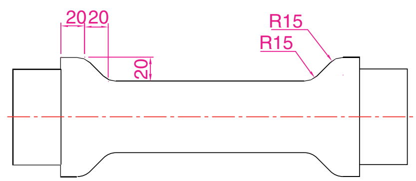


Fig. 10.25 The amended **Xref** drawing

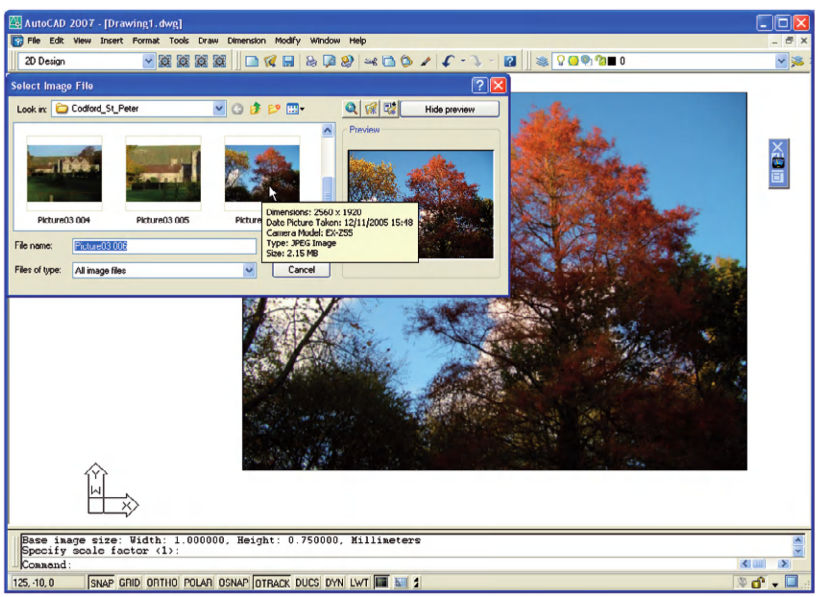


Fig. 10.26 Exercise 3 – example

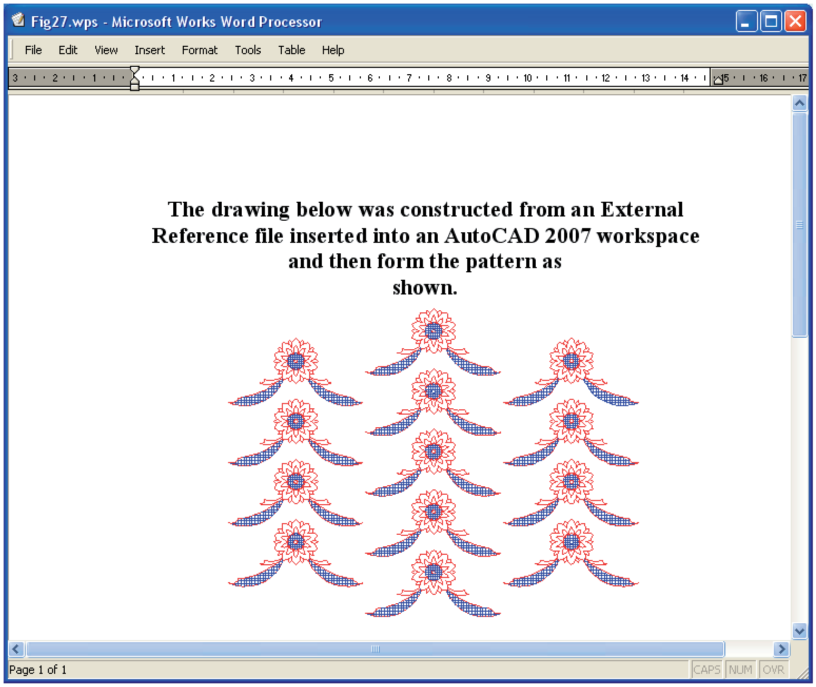


Fig. 10.27 Exercise 4 – an example

CHAPTER 11

Sheet sets

Aims of this chapter

1. To introduce sheet sets.
2. To describe working in the **Sheet Layout and Publishing** workspace.
3. To give an example of a sheet set based on the design of a two-storey house.

Sheet sets

When anything is to be manufactured or constructed, whether it be a building, an engineering design, an electronics device or any other form of manufactured artefact, a variety of documents, many in the form of technical drawings, will be needed to convey to those responsible for constructing the design information necessary to be able to proceed according to the wishes of the designer. Such sets of drawings may be passed between the people and the companies responsible for the construction, enabling all those involved to make adjustments or suggest changes to the design. In some cases there may well be a considerable number of drawings required in such sets of drawings. In AutoCAD 2007 all the drawings from which a design is to be manufactured can be gathered together in a **sheet set**. This chapter shows how a much reduced sheet set of drawings for the construction of a house at 62 Pheasant Drive can be formed. Some other drawings, particularly detail drawings, would be required in this example, but to save page space, the sheet set described here consists of only four drawings and a subset of another four.

Example – a sheet set for 62 Pheasant Drive

1. Construct a template **62 Pheasant Drive.dwt** based upon the acadisco.dwt template, but including a border and a title block. Save the template in a **Layout1** format. An example of the title block from one of the drawings constructed in this template is shown in Fig. 11.1.

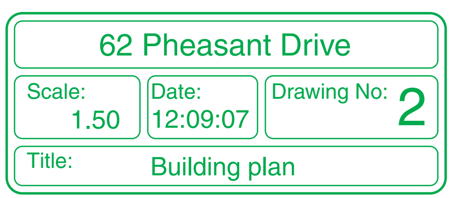


Fig. 11.1 The title block from Drawing number **2** of the sheet set

2. Construct each of the drawings which will form the sheet set in this drawing template. The whole set of drawings is shown in Fig. 11.2. Save the drawings in a directory – in this example this has been given the name **62 Pheasant Drive**.

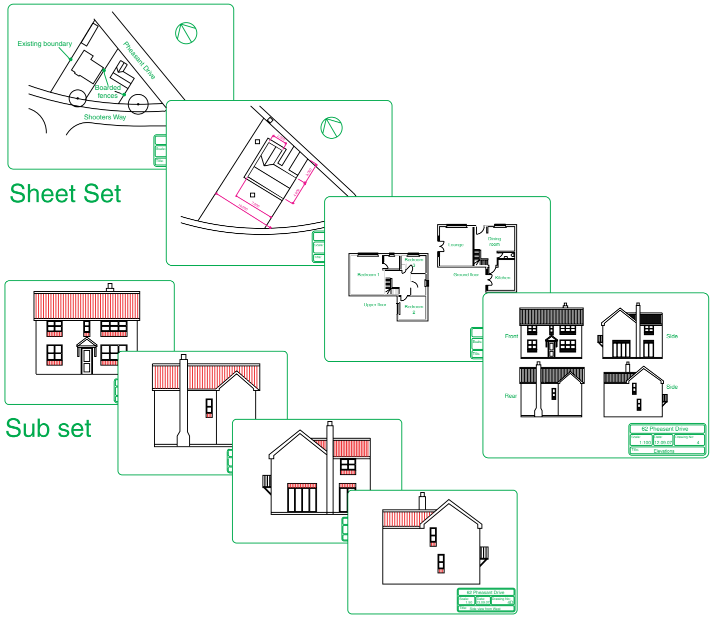


Fig. 11.2 The eight drawings in the **62 Pheasant Drive** sheet set

3. *Click* **New Sheet Set** in the **File** drop-down menu (Fig. 11.3). The first of a series of **Create Sheet Set** dialogs appears – the **Begin** dialog (Fig. 11.4). *Click* the radio button next to **Existing drawings**, followed

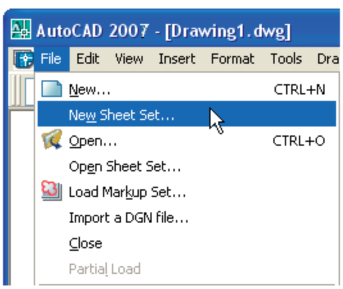


Fig. 11.3 Selecting **New Sheet Set**... from the **File** drop-down menu

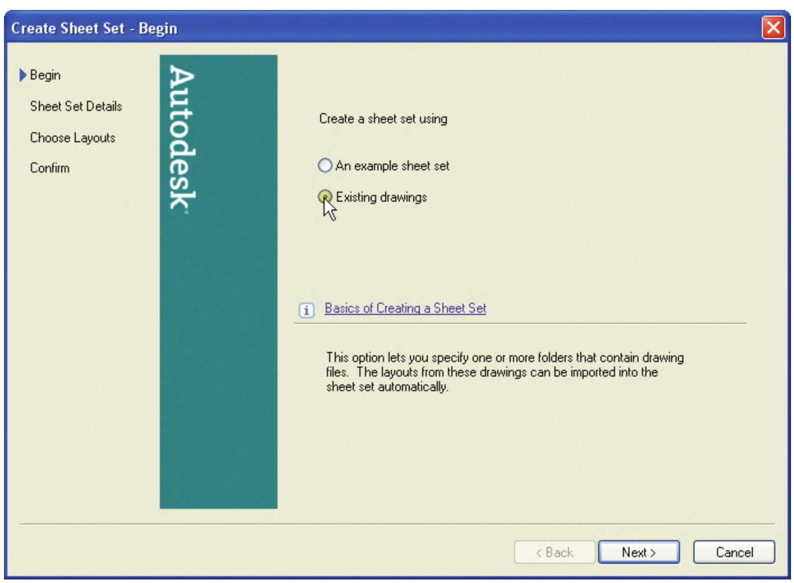


Fig. 11.4 The first of the **Create Sheet Set** dialogs – **Begin**

by a *click* on the **Next** button and the next dialog **Sheet Set Details** appears (Fig. 11.5).

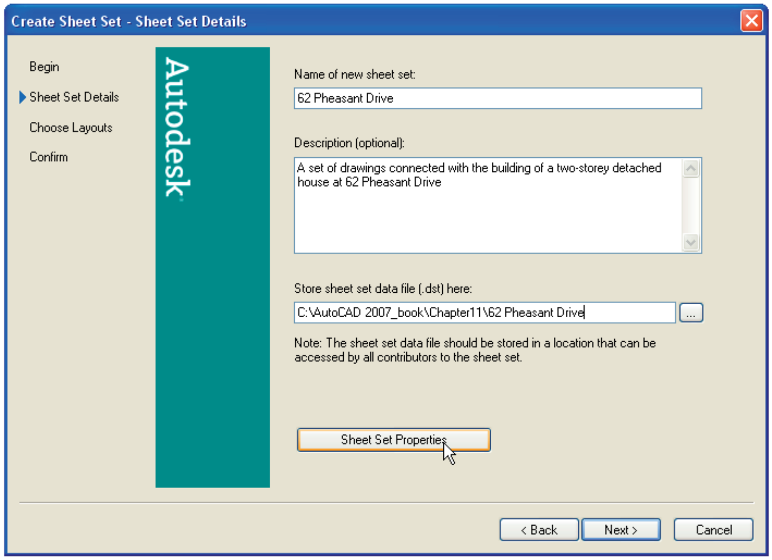


Fig. 11.5 The **Sheet Set Details** dialog

4. *Enter* details in the dialog as shown in Fig. 11.5. Then *click* the **Next** button to bring the **Choose Layouts** dialog to screen (Fig. 11.6).
5. *Click* its **Browse . . .** button and from the **Browse for Folder** list which comes to screen, *pick* the directory **62 Pheasant Drive**. *Click* the **OK** button and the drawings held in the directory appears in the

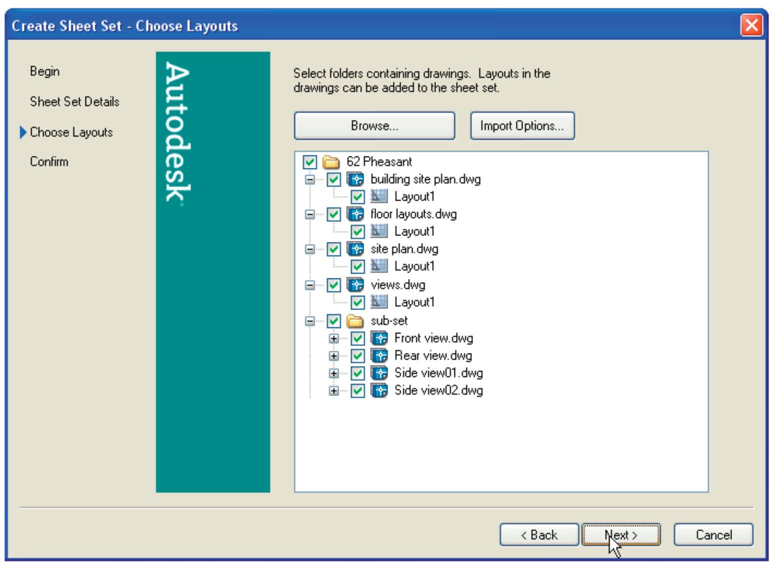


Fig. 11.6 The **Choose Layouts** dialog

Choose Layouts dialog (Fig. 11.6). If satisfied the list is correct, *click* the **Next** button. A **Confirm** dialog appears (Fig. 11.7). If satisfied *click* the **Finish** button and the **Sheet Set Manager** palette appears showing the drawings which will be in the **62 Pheasant Drive** sheet set (Fig. 11.8).

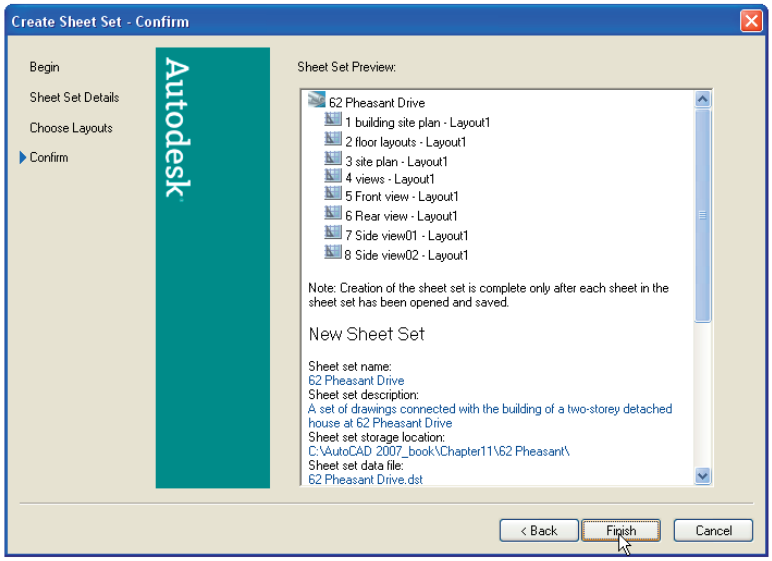


Fig. 11.7 The **Confirm** dialog

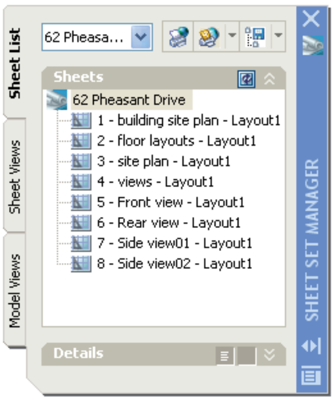


Fig. 11.8 The **Sheet Set Manager** palette for **62 Pheasant Drive**

Notes

1. The eight drawings in the sheet set are shown in Fig. 11.8. If any of the drawings in the sheet set are subsequently amended or changed, when the drawing is opened again from the **62 Pheasant Drive** Sheet Set Manager palette, the drawing will include any changes or amendments.
2. Drawings can only be placed into sheet sets if they have been saved in a **Layout** screen. Note that all the drawings shown in the **62 Pheasant Drive** Sheet Set Manager have **Layout1** after the drawing names because each has been saved after being placed in a **Layout1** screen.
3. Sheet sets in the form of **DWF** (Design Web Format) files can be sent via email to others who are using the drawings or placed on an intranet. The method of producing a **DWF** for the **62 Pheasant Drive** Sheet Set follows.

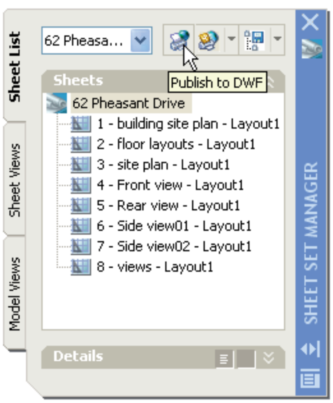


Fig. 11.9 The **Publish to DWF** icon in the **Sheet Set Manager**

62 Pheasant Drive DWF

1. In the **62 Pheasant Drive** Sheet Set Manager *click* the **Publish to DWF** icon (Fig. 11.9). The **Select DWF File** dialog appears (Fig. 11.10).

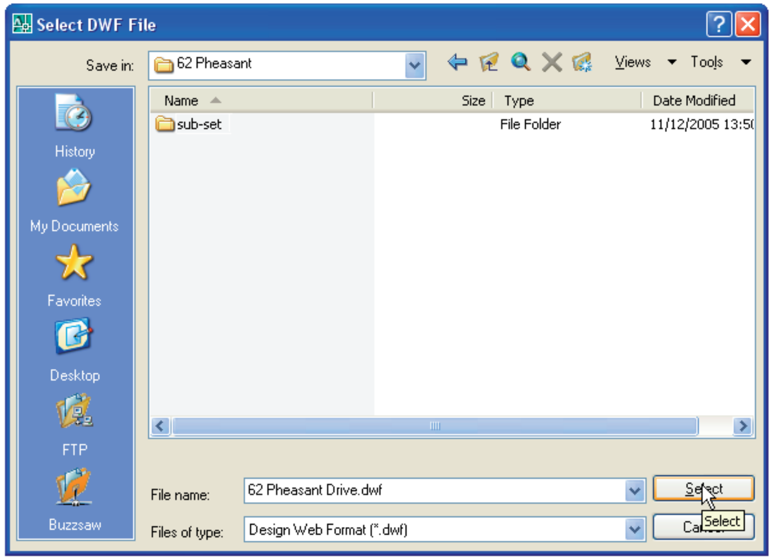


Fig. 11.10 The **Select DWF File** dialog

Enter **62 Pheasant Drive** in the **File name** field followed by a *click* on the **Select** button. The **Processing Background Job** dialog appears (Fig. 11.11). *Click* its **OK** button. The **Publish Job in Progress** icon at the bottom right-hand corner of the AutoCAD 2007 window starts fluctuating in shape showing that the DWF file is being processed (Fig. 11.12). When the icon becomes stationary *right-click* the icon

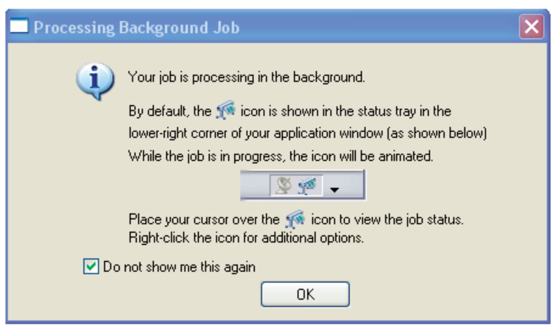


Fig. 11.11 The **Processing Background Job** dialog

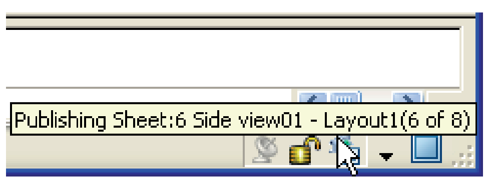


Fig. 11.12 The **Publish Job in Progress** icon

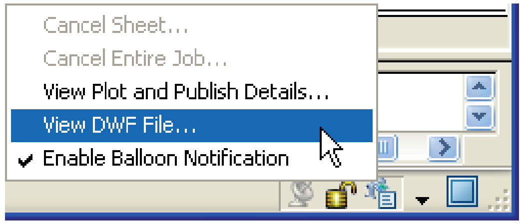


Fig. 11.13 The *right-click* menu of the icon

and *click* **View DWF file** . . . in the *right-click* menu which appears (Fig. 11.13).

2. The **Autodesk DWF Viewer** window appears showing the **62 Pheasant Drive.dwf** file (Fig. 11.14). *Click* in any of the icons of the thumbnails of the drawings in the viewer and the drawing appears in the right-hand area of the viewer.

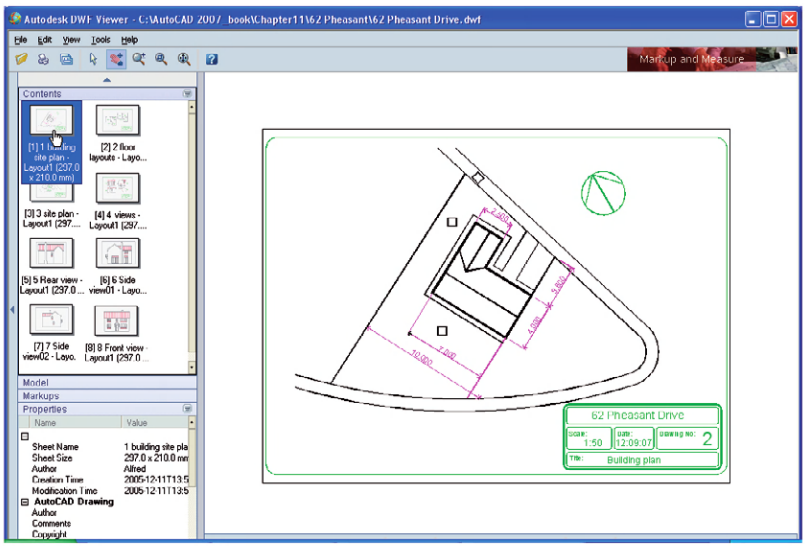


Fig. 11.14 The **Autodesk DWF Viewer** showing details of the **62 Pheasant Drive.dwf** file

3. If required the DWF Viewer file can be sent between people by email as an attachment, opened in a company's intranet or, indeed, included within an Internet web page.

Revision notes

1. To start off a new sheet set, *click* **New Sheet Set** . . . in the **File** drop-down menu.
2. Sheet sets can only contain drawings saved in **Layout** form.
3. Sheet sets can be published as **Design Web Format (*.dwf)** files which can be sent between offices by email, published on an intranet or published on a web page.

4. Sub sets can be included in sheet sets.
5. Changes or amendments made to any drawings in a sheet set are reflected in the sheet set drawings when the sheet set is opened.

Exercises

1. Fig. 11.15 is an exploded orthographic projection of the parts of a piston and its connecting rod. There are four parts in the assembly. Small drawings of the required sheet set are shown in Fig. 11.16.

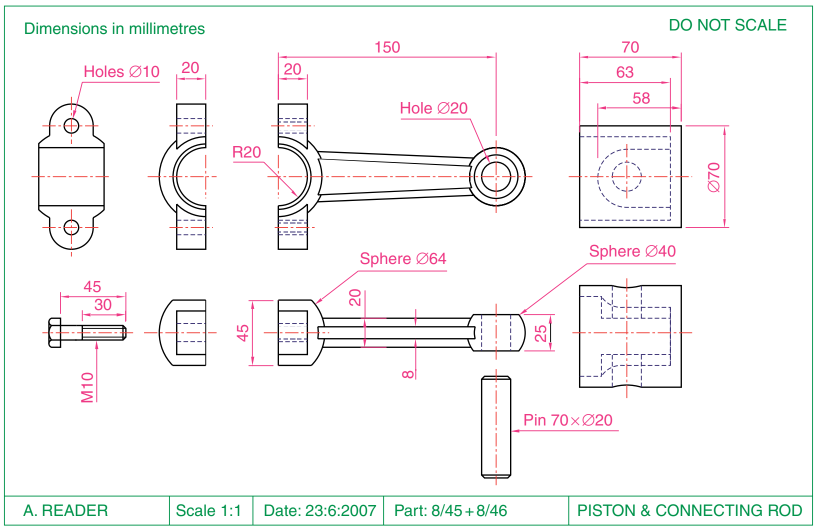


Fig. 11.15 Exercise 1 – the exploded orthographic projection

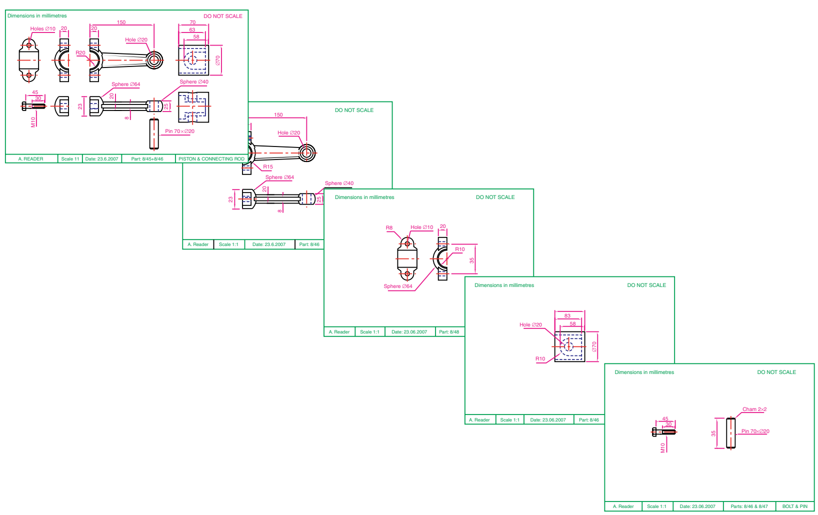


Fig. 11.16 Exercise 1 – the five drawings in the sheet set

Construct the drawing in Fig. 11.15 and also the four drawings of its parts. Save each of the drawings in a **Layout1** format and construct the sheet set which contains the five drawings.

Construct the **DWF** file of the sheet set. Experiment sending it to a friend via email as an attachment to a document, asking them to return the whole email to you without changes. When the email is returned, open its DWF file and *click* each drawing icon in turn to check the contents of the drawings.

Note

Fig. 11.17 shows a DWF for the sheet set from exercise 1 with the addition of a sixth drawing which is a 3D exploded model drawing of the five parts of the piston and connecting rod which has been Gouraud shaded – see Chapter 16. This illustration has been included here to show that such shaded 3D models can be included in a sheet set.

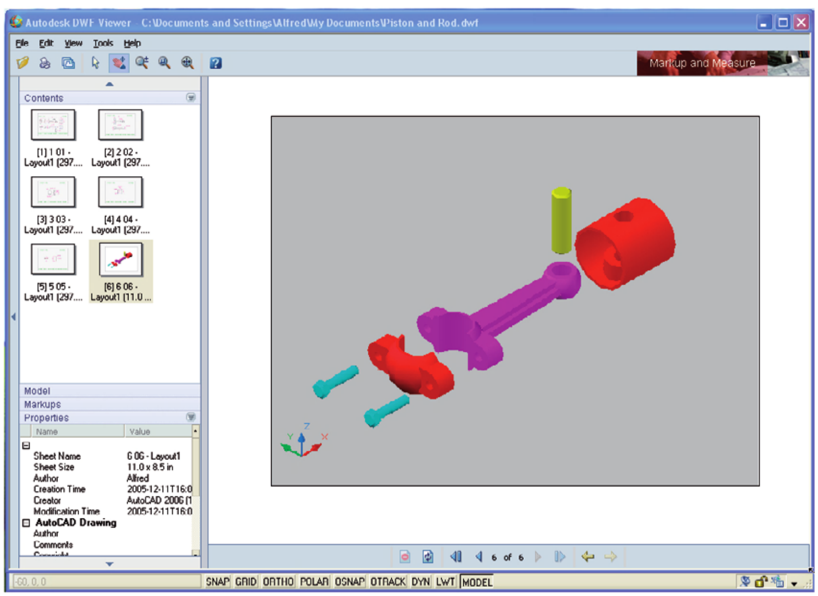


Fig. 11.17 The **DWF** for exercise **1**

2. Construct a similar sheet set as in the answer to exercise **1** from the exploded orthographic drawing of a **Machine adjusting spindle** given in Fig. 11.18.

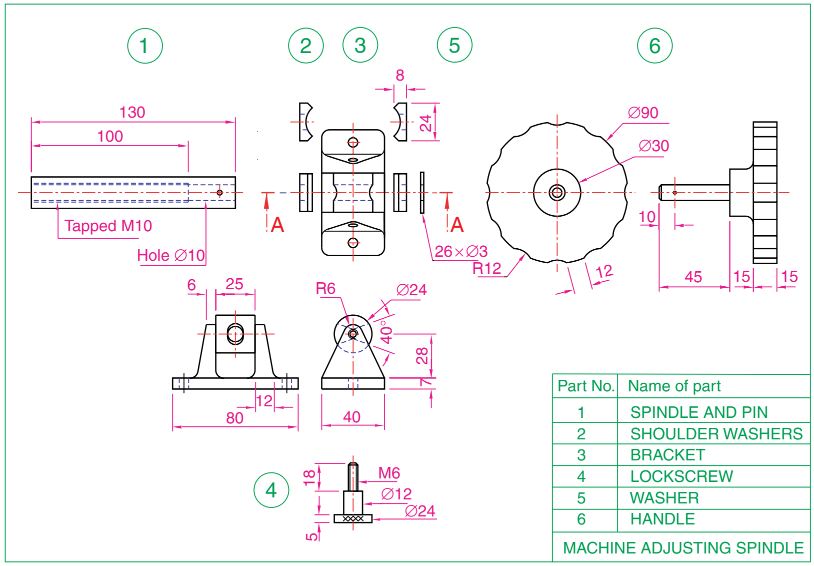


Fig. 11.18 Exercise 2

CHAPTER 12

Building drawing

Aim of this chapter

To show that AutoCAD 2007 is a suitable CAD software package for the construction of building drawings.

Building drawings

There are a number of different types of drawings related to the construction of any form of building. As fairly typical examples of a set of building drawings, in this chapter, seven drawings are shown related to the construction of an extension to an existing two-storey house (44 Ridgeway Road). These show:

1. A site plan of the original two-storey house, drawn to a scale of **1:200** (Fig. 12.1).
2. A site layout plan of the original house, drawn to a scale of **1:100** (Fig. 12.2).

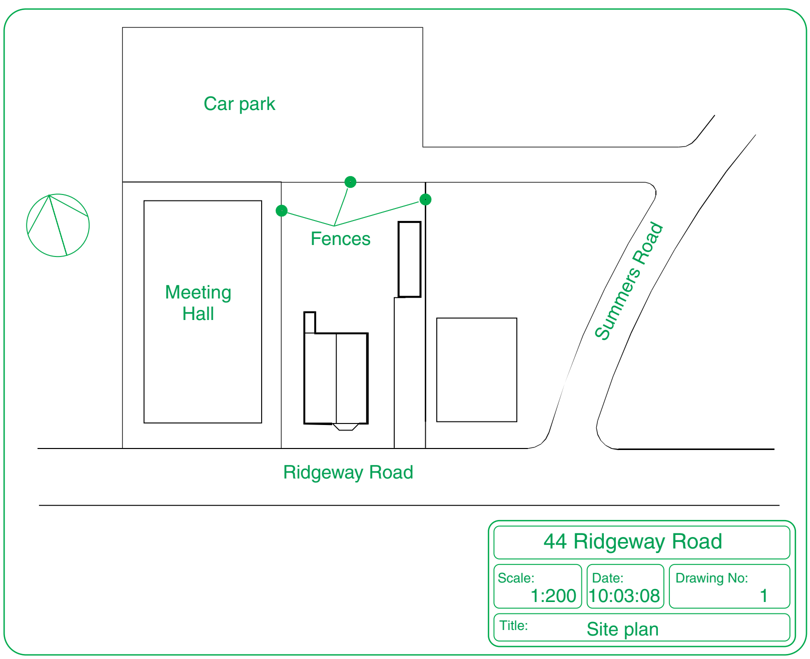


Fig. 12.1 A site plan

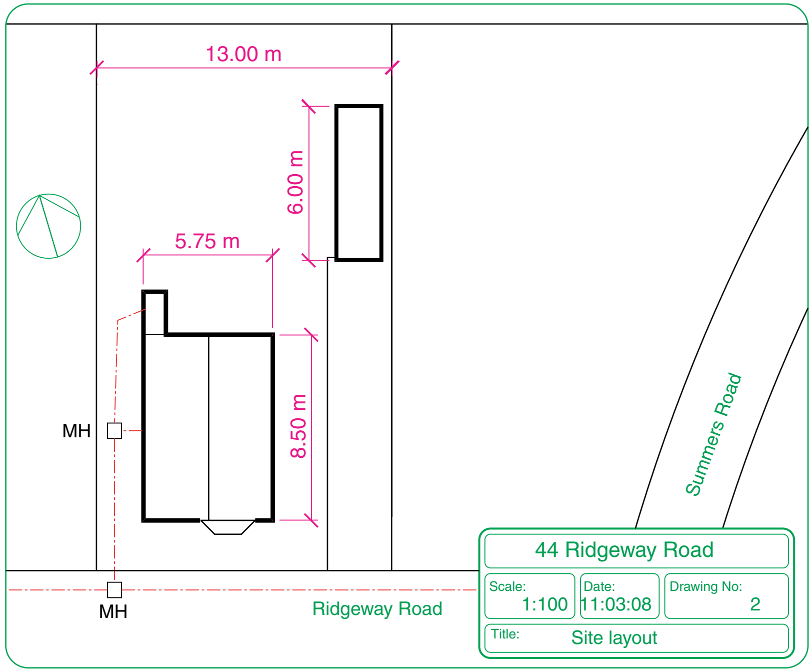


Fig. 12.2 A site layout plan

3. Floor layouts of the original house, drawn to a scale of **1:50** (Fig. 12.3).

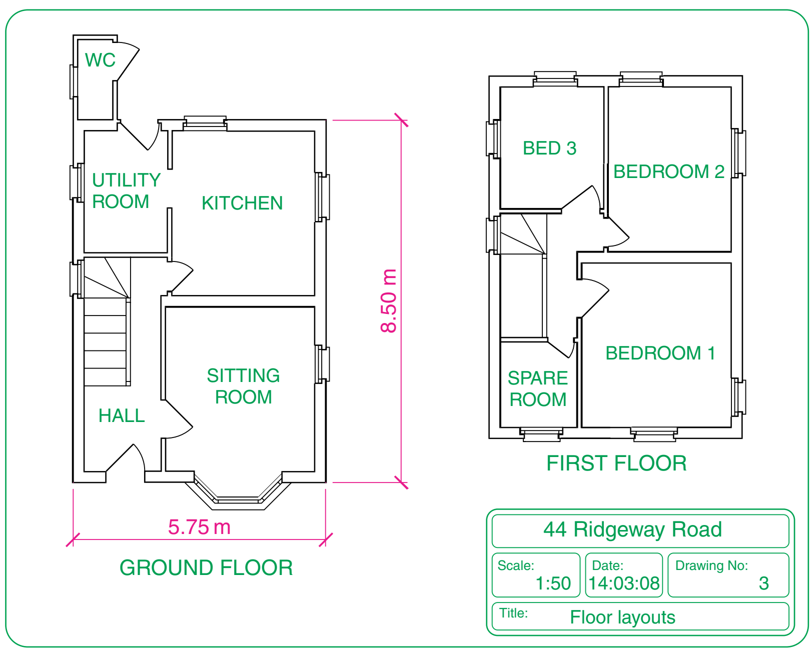


Fig. 12.3 Floor layouts drawing of the original house

Fig. 12.4 Views of the original house

4. Views of all four sides of the original house drawn to a scale of **1:50** (Fig. 12.4).
5. Floor layouts including the proposed extension, drawn to a scale of **1:50** (Fig. 12.5).

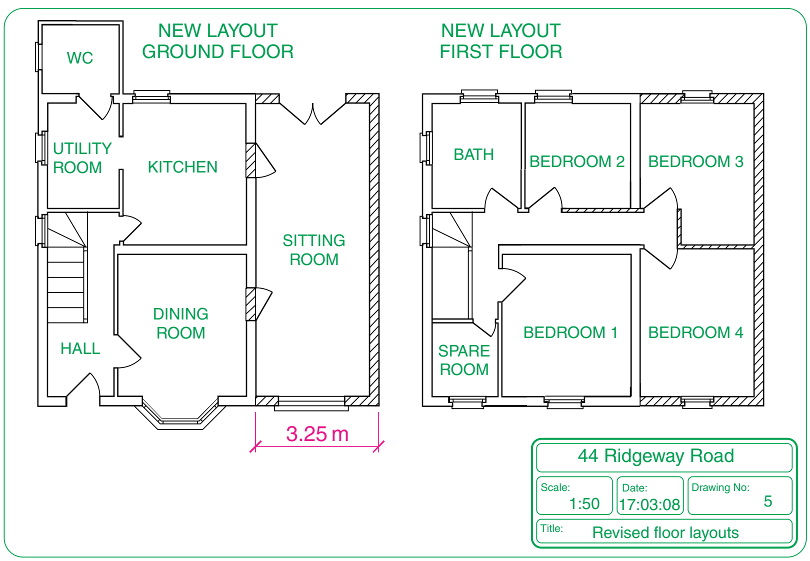


Fig. 12.5 Floor layouts drawing of the proposed extension

6. Views of all four sides of the house including the proposed extension, drawn to a scale of **1:50** (Fig. 12.6).
7. A sectional view through the proposed extension, drawn to a scale of **1:50** (Fig. 12.7).

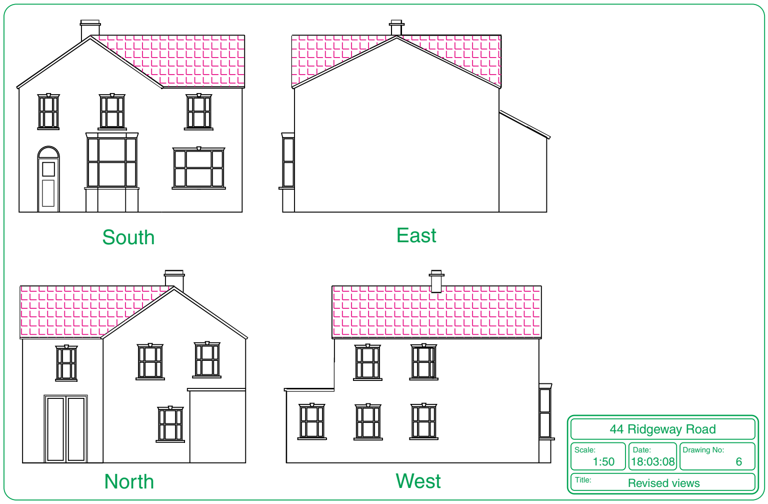


Fig. 12.6 Views including the proposed extension

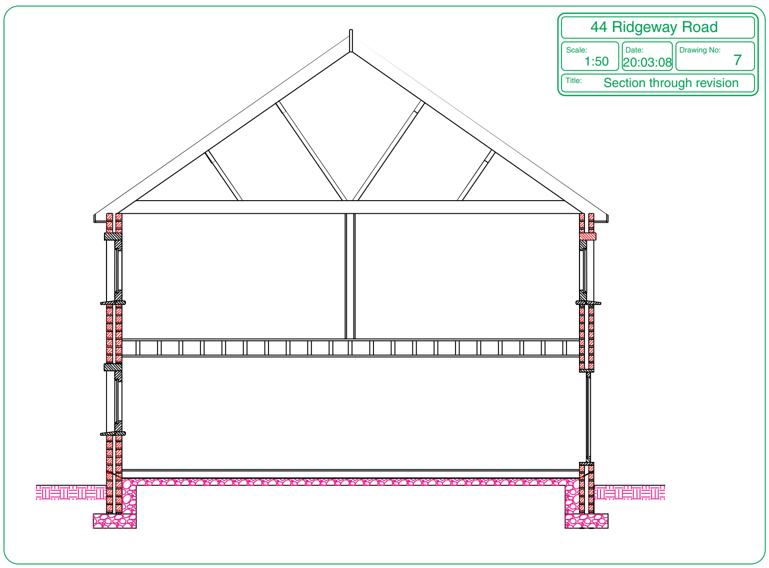


Fig. 12.7 A section through the proposed extension

Notes

1. Other types of drawings will be constructed which show the details of parts such as doors, windows, floor structures, etc. These are often shown in sectional views.
2. Although the seven drawings related to the proposed extension of the house at 44 Ridgeway Road are shown here as having been constructed on either A3 or A4 layouts, it is common practice to include several types of building drawings on larger sheets such as A1 sheets of a size 820 mm by 594 mm.

Floor layouts

When constructing floor layout drawings it is advisable to build up a library of block drawings of symbols representing features such as doors, windows, etc. These can then be inserted into layouts from the DesignCenter. A suggested small library of such block symbols is shown in Fig. 12.8.

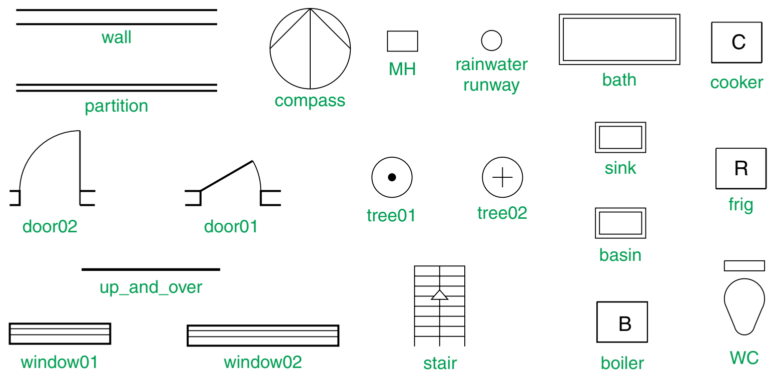


Fig. 12.8 A small library of building symbols

Revision notes

There are a number of different types of building drawings – site plans, site layout plans, floor layouts, views, sectional views, detail drawings. AutoCAD 2007 is a suitable CAD program to use when constructing building drawings.

Exercises

1. Fig. 12.9 is a site plan drawn to a scale of 1:200 showing a bungalow to be built in the garden of an existing bungalow.

Construct the library of symbols shown in Fig. 12.8 and by inserting the symbols from the DesignCenter construct a scale 1:50 drawing of the floor layout plan of the proposed bungalow.

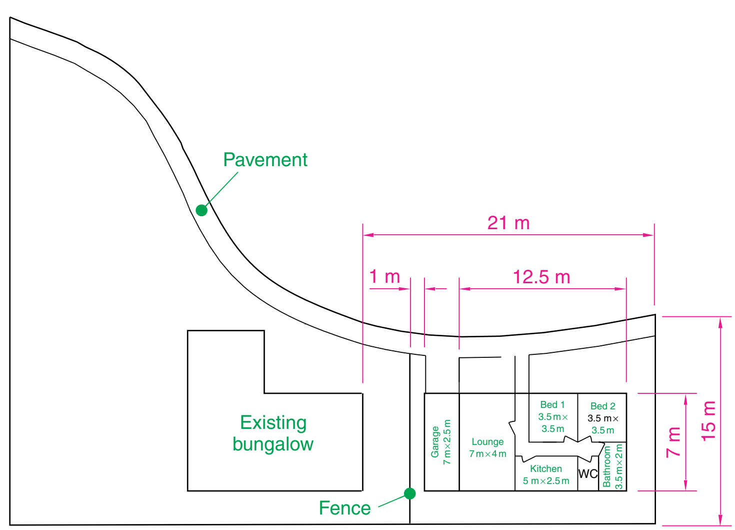


Fig. 12.9 Exercise 1

2. Fig. 12.10 is a site plan of a two-storey house to be built on a building plot.

Design and construct to a scale 1:50, a suggested pair of floor layouts for the two floors of the proposed house.

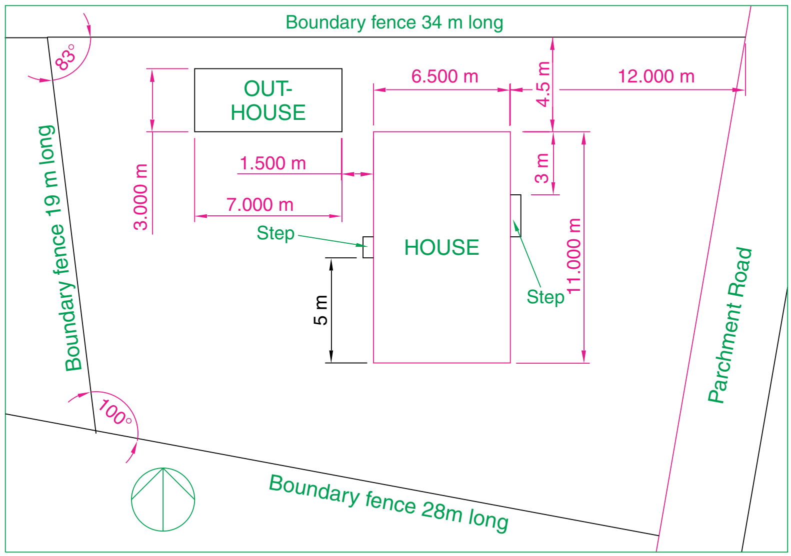


Fig. 12.10 Exercise 2

3. Fig. 12.11 shows a scale 1:100 site plan for the proposed bungalow at 4 Caretaker Road. Construct the floor layout for the proposed house shown in the drawing in Fig. 12.12.

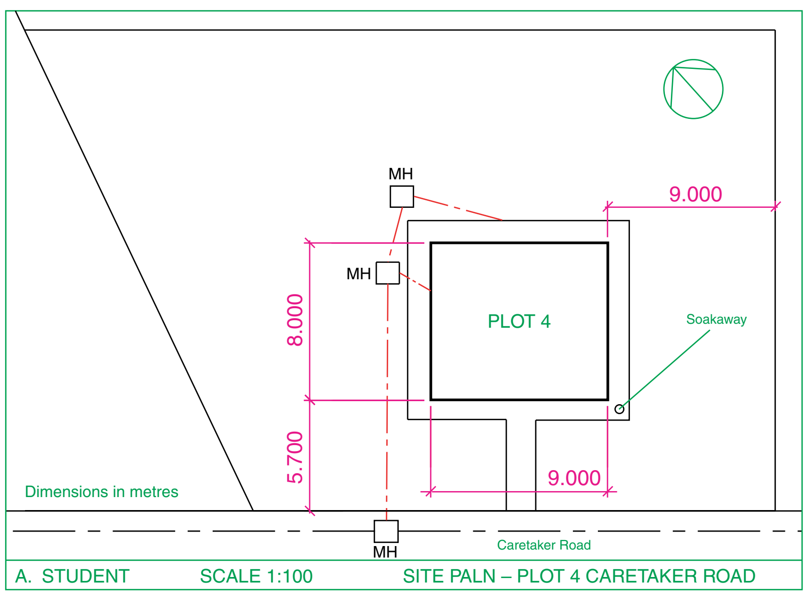


Fig. 12.11 Exercise 3 – site plan

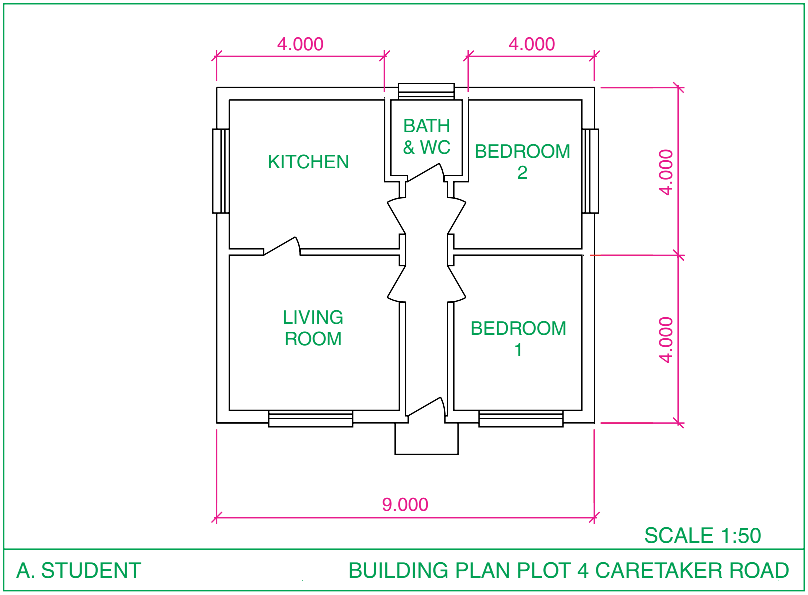


Fig. 12.12 Exercise 3

CHAPTER 13

Introducing 3D modelling

Aims of this chapter

1. To introduce the tools used for the construction of 3D solid models.
2. To introduce the use of the **3D face** tool.
3. To give examples of the construction of 3D solid models using tools from the **Modeling** toolbar or the **3D Make control panel**.
4. To give examples of 2D outlines suitable as a basis for the construction of 3D solid models.
5. To give examples of constructions involving the Boolean operators – **Union, Subtract** and **Intersect**.

Introduction

As shown in Chapter 1 the AutoCAD coordinate system includes a third coordinate direction **Z**, which, when dealing with 2D drawing in previous chapters, has not been used. 3D model drawings make use of this third **Z** coordinate.

The 3D Modeling workspace

It is possible to construct 3D model drawings in the **AutoCAD Classic** workspace, but in this part of the book we will be working in the **3D Modeling** workspace. To set this workspace *left-click* the **3D Modeling** icon in the **Workspaces** toolbar and the AutoCAD screen changes as shown in Fig. 13.1. In this example the four main 3D modelling toolbars have been shown on screen, together with the **3D Make control panel**. It is not necessary to have the toolbars or control panel on screen. It is up to the operator to decide which of the available methods for calling tools for 3D modelling they wish to use. Toolbars can be called to screen with a *right-click* in any toolbar on screen, followed by selecting the required toolbar from the menu which then appears.

To bring the **3D Make control panel** from the **DASHBOARD**, *right-click* in the title bar (blue area to the right of the palette) and make sure a tick is in place only against the **3D Make control panel** (Fig. 13.2) and not against any of the other control panel names in the sub-menu. In Fig. 13.1 the **3D Navigate control panel** is shown in the **DASHBOARD** with the **Parallel Projection** icon selected.

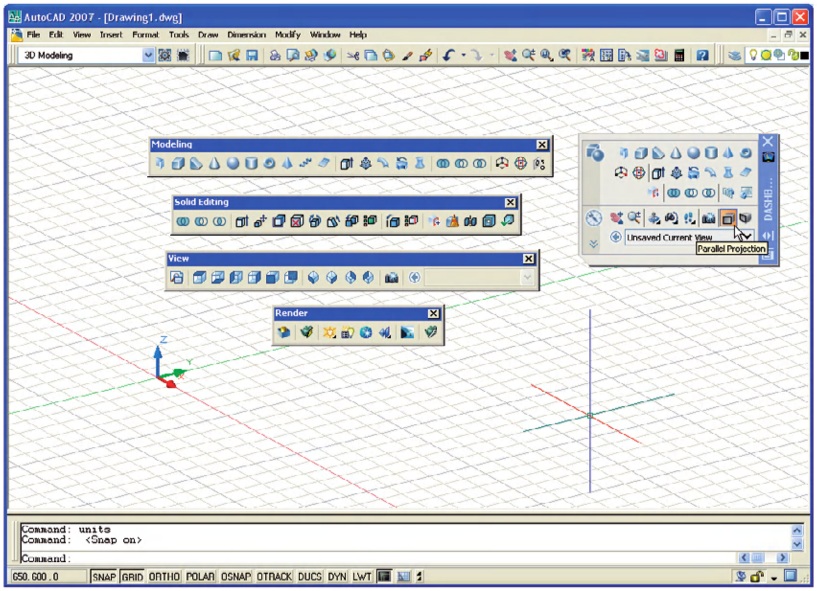


Fig. 13.1 The **3D Modeling** workspace showing four toolbars and the **3D Make control panel**

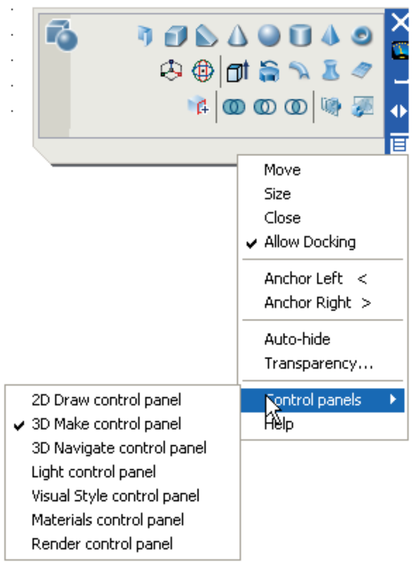


Fig. 13.2 Ensuring only the **3D Make control panel** is showing from the **DASHBOARD** palette

Methods of calling tools for 3D modelling

When calling the tools for the construction of 3D model drawings, the four same methods apply as can be used when constructing 2D drawings:

1. A *click* on a tool icon in the **Modeling** toolbar or in the appropriate **3D Make control panel** brings the selected tool into action (Fig. 13.3).
2. A *click* on the name of a tool from a drop-down menu brings the tool into action.

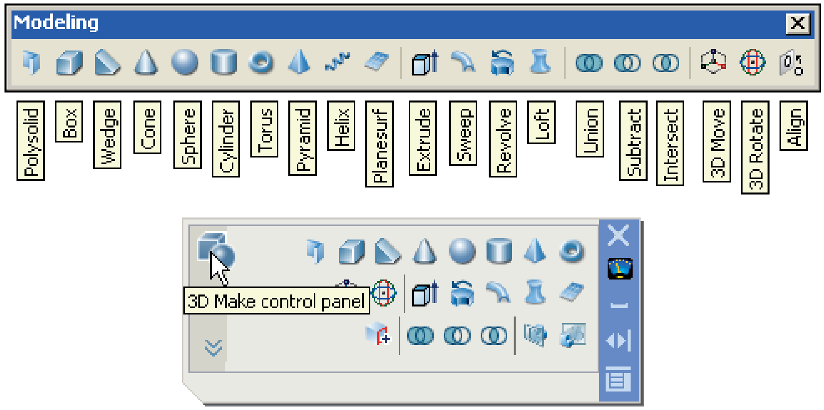


Fig. 13.3 The tool icons in the **Modeling** toolbar and **3D Make control panel**

3. *Entering* the tool name at the command line in the command window, followed by pressing the *Return* button of the mouse or the **Return** key of the keyboard, brings the tool into action.
4. Some of the 3D tools have an abbreviation which can be *entered* at the command line instead of its full name.

Notes

1. As when constructing 2D drawings no matter which method is used and most operators will use a variety of these four methods, the result of calling a tool results in prompts sequences appearing at the command prompt as in the following example:

   Command: *enter* **box** *right-click*
   Specify corner of box or [CEnter]: *enter* **90,120** *right-click*
   Specify corner or [Cube/Length]:

   Or, if the tool is called from its tool icon or from a drop-down menu:

   Command:_box
   Specify corner of box or [CEnter]: *enter* **90,120** *right-click*
   Specify corner or [Cube/Length]:

2. In the following pages, if the tool's sequences are to be repeated, they may be replaced by an abbreviated form such as:

   Command: box
   [prompts]: 90,120
   [prompts]:

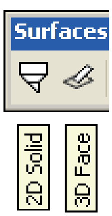


Fig. 13.4 The **3D Face** tool icon in the **Surfaces** toolbar

Examples of 3D drawings using the 3D Face tool

The following two examples demonstrate the construction of 3D faces using the **3D Face** tool from the **Surfaces** toolbar (Fig. 13.4). A 3D face is a triangular or quadrilateral flat (planar) or non-planar surface – i.e. a surface with either three or four edges in 3D space. Lines or other surfaces behind 3D faces can be hidden by using the **Hide** tool (Fig. 13.5).

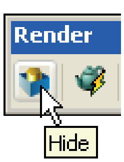


Fig. 13.5 The **Hide** tool icon in the **Render** toolbar

First example – 3D Face tool (Fig. 13.6)

1. At the command line:

   Command: *enter* **3dface** *right-click*
   Specify first point or [Invisible]: *enter* **60,230** *right-click*
   Specify second point or [Invisible]: *enter* **60,110** *right-click*
   Specify third point or [Invisible] <exit>: *enter* **190,110,150** *right-click*
   Specify fourth point or [Invisible] <create three-sided face>: *enter* **190,230,150** *right-click*
   Specify third point or [Invisible] <exit>: *right-click*
   Command:

2. Call the **Mirror** tool and mirror the 3D face about **190,110** and **190,230**.
3. *Click* on **View** in the menu bar and on **3D Views** in the drop-down menu which appears, and *click* again on **Southwest Isometric** in the sub-menu which comes on screen alongside the drop-down menu.
4. *Click* on the **Hide** tool from the **Render** toolbar (Fig. 13.5) or from the **View** drop-down menu.

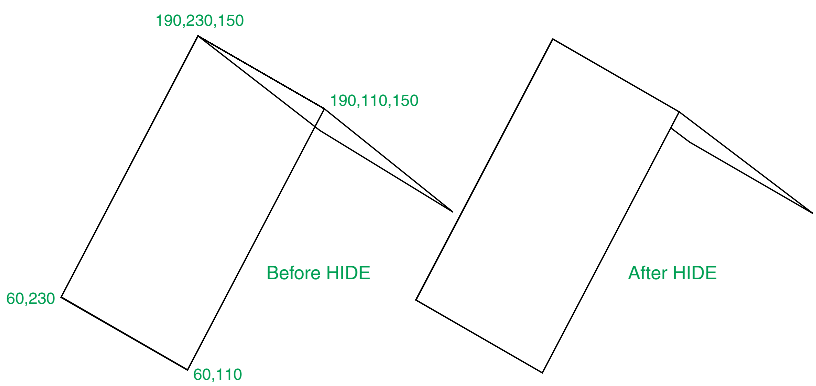


Fig. 13.6 First example – **3D Face tool**

After **HIDE**, part of the right-hand 3D face has been hidden behind the left-hand face (Fig. 13.6).

Note

3D Views can be called from the **3D Navigate control panel** (Fig. 13.7). Open the panel in the **DASHBOARD** (see Fig. 13.2, page 188) and *left-click* on the arrow to the right of the popup list name.

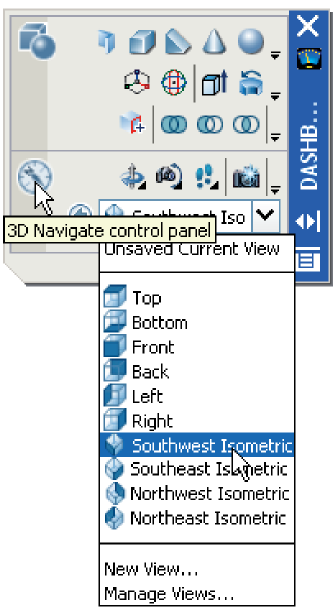


Fig. 13.7 Selecting **Southwest Isometric** from the popup list in the **3D Navigate control panel**

Second example – 3D Face tool (Fig. 13.8)

It is assumed in this example that the reader understands how to *enter* 3D coordinates and how to place the screen in various **3D Views** from the **View** drop-down menu.

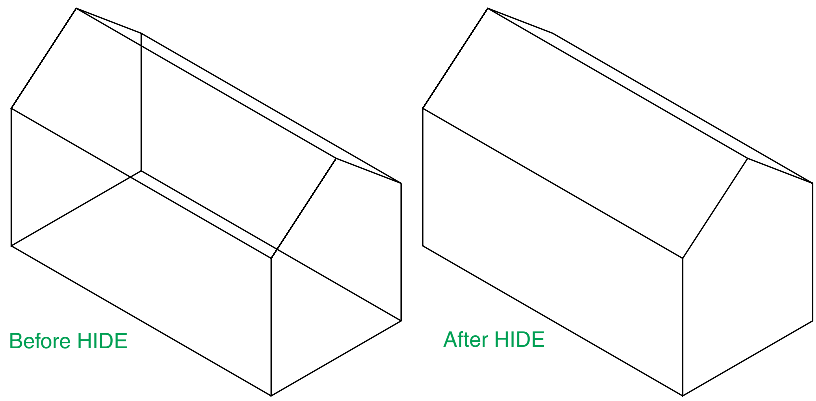


Fig. 13.8 Second example – **3D Face tool**

1. *Click* on **View** in the menu bar and again on **3D Views** in the drop-down menu. *Click again* on **Front** in the sub-menu which appears.
2. Call **Zoom** and zoom to **1**.
3. Call the **3D Face** tool:

   Command: 3dface
   [prompts]: 130,0
   [prompts]: 250,0
   [prompts]: i (for Invisible)
   [prompts]: 250,110
   [prompts]: 130,110
   [prompts]: 190,250
   [prompts]: 250,110
   [prompts]: *right-click*
   Command:

4. *Click* on **View** in the menu bar and again on **3D Views** in the drop-down menu. *Click* again on **Right** in the sub-menu which appears.
5. Call **Zoom** and zoom to **1**.
6. With **Mirror**, mirror the face about the line **320,0** and **320,160**.
7. *Click* on **View** and on **3D Views** and select **SW Isometric** from the sub-menu. **Zoom** to **1**.
8. Call **3D Face** and with the aid of the osnap **endpoint** construct four 3dfaces linking the two previously constructed faces.
9. Call **Hide**.

2D outlines suitable for 3D models

When constructing 2D outlines suitable as a base for constructing some forms of 3D model, use the **Line** or the **Polyline** tools. If constructed with the **Line** tool, before being of any use for 3D modelling, the outline must be changed to a region using the **Region** tool. Polylines can be used without change.

First example – Line outline and Region (Fig. 13.9)

1. Construct the left-hand drawing of Fig. 13.9 using the **Line** tool.

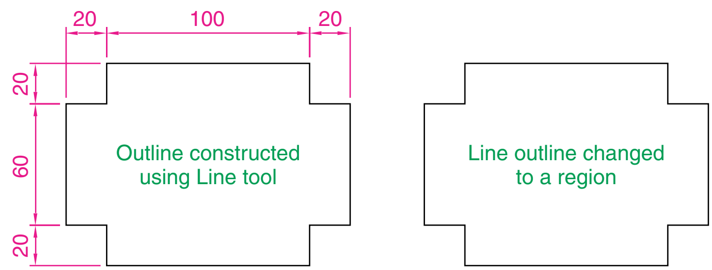


Fig. 13.9 First example – Line outline and Region

2. *Left-click* on **Region** tool in the **Draw** drop-down menu, or *enter* **reg** at the command line. The command line shows:

   Command:_region
   Select objects: *window* the drawing **12 found**
   Select objects: *right-click*
   1 loop extracted
   1 Region created
   Command:

And the **Line** outline is changed to a region.

Second example – Union, Subtract and regions (Fig. 13.11)

1. Construct drawing **1** of Fig. 13.11 and with the **Copy** tool, copy the drawing three times to produce drawings **2, 3** and **4**.
2. With the **Region** tool change all the outlines into regions.
3. Drawing **2** – call the **Union** tool from the **3D Make control panel** (Fig. 13.10). The command line shows:

   Command:_union
   Select objects: *pick* the left-hand region **1 found**
   Select objects: *pick* the circular region **1 found, 2 total**
   Select objects: *pick* the right-hand region **1 found, 3 total**
   Command:

4. Drawing **3** – with the **Union** tool form a union of the left-hand region and the circular region.

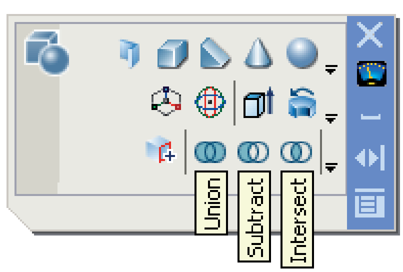


Fig. 13.10 The **Union, Subtract** and **Intersect** tool icons in the **3D Modeling control panel**

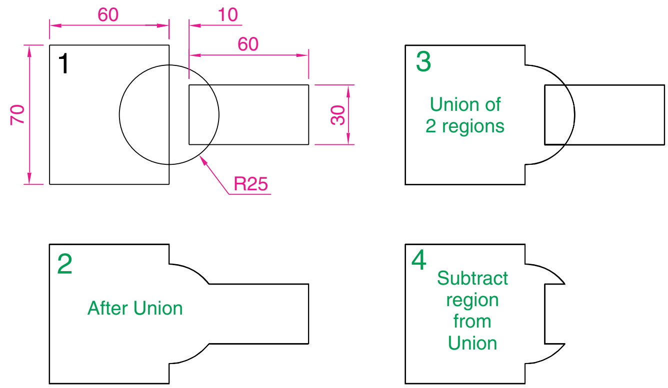


Fig. 13.11 Second example – Union, Subtract and regions

5. Drawing **4** – call the **Subtract** tool (Fig. 13.10). The command line shows:

Command:_subtract Select solids and regions to subtract from . . .
Select objects: *pick* the region just formed **1 found**
Select objects: *right-click*
Select solids and regions to subtract: *pick* the right-hand region **1 found**
Select objects: *right-click*
Command:

Third example – Intersection and regions (Fig. 13.12)

1. Construct drawing **1** of Fig. 13.12.
2. With the **Region** tool, change the three outlines into regions.
3. With the **Copy** tool, copy the three regions.

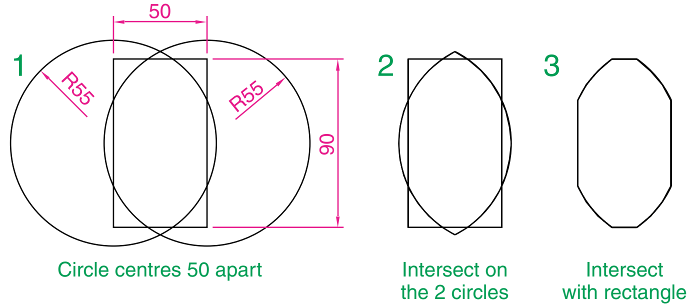


Fig. 13.12 Third example – Intersection and regions

4. Drawing **2** – call the **Intersect** tool from the **3D Modeling control panel** (Fig. 13.10). The command line shows:

Command:_intersect
Select objects: *pick* one of the circles **1 found**
Select objects: *pick* the other circle **1 found, 2 total**

Select objects: *right-click*
Command:

And the two circular regions intersect with each other to form a region.

5. Drawing **3** – repeat using the **Intersect** tool on the intersection of the two circles and the rectangular region.

The Extrude tool

The **Extrude** tool can be called with a *click* on its tool icon in the **Modeling** toolbar (Fig. 13.3), from the **3D Make control panel**, by *entering* **extrude** or its abbreviation **ext** at the command line.

Note

In this chapter, 3D models are shown in illustrations as they appear in the **3D Modeling** workspace. From Chapter 14 onwards, such 3D models are sometimes shown in outline only. This is to allow the reader to see the parts of 3D models in future chapters more clearly in the illustrations.

Examples of the use of the Extrude tool

The three examples of forming regions given in Figs 13.9, 13.11 and 13.12 are used here to show the results of using the **Extrude** tool.

First example – Extrude (Fig. 13.13)

From the first example of forming a region:

1. Call the **Extrude** tool. The command line shows:

Command:_extrude
Current wire frame density: ISOLINES=4
Select objects to extrude: *pick* region **1 found**
Select objects to extrude: *right-click*
Specify height of extrusion or [Direction/Path/Taper angle] <45>: *enter* **50** *right-click*
Command:

2. *Click* **View** in the menu bar, followed by a *click* on **3D Views** in the drop-down menu which appears, followed by another *click* on **SW Isometric** in the **3D Views** sub-menu. The extrusion appears in an isometric view.
3. Call **Zoom** and zoom to **1**.

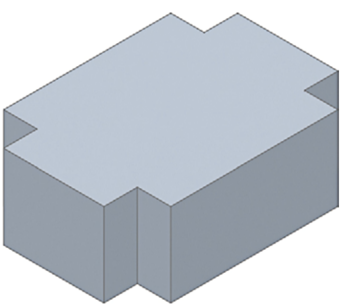


Fig. 13.13 First example – **Extrude**

Notes

1. In the above example we made use of one of the isometric views possible using the **3D Views** sub-menu of the **View** drop-down menu or from the **3D Navigate control panel** (Fig. 13.7). These views will be

used frequently in examples to show 3D solid model drawings in a variety of positions in 3D space.

2. Note also the use of the **Hide** tool. Extruded polylines or regions are made up from 3D faces (3D meshes). When **Hide** is called, lines behind the 3D meshes become invisible on screen.
3. **Hide** can also be called from the **View** drop-down menu, but the quickest method of calling the tool is to *enter* **hi** at the command line, followed by a *right-click*.
4. Note the **Current wire frame density: ISOLINES=4** in the prompts sequence when **Extrude** is called. The setting of **4** is suitable when extruding plines or regions consisting of straight lines, but when arcs are being extruded it may be better to set **ISOLINES** to a higher figure as follows:

Command: *enter* isolines *right-click*
Enter new value for ISOLINES <4>: *enter* **16** *right-click*
Command:

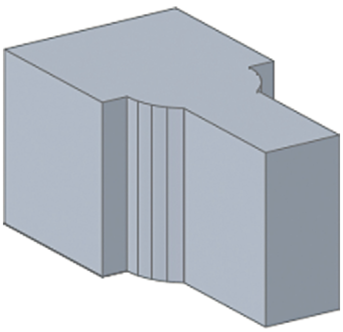


Fig. 13.14 Second example – **Extrude**

Second example – Extrude (Fig. 13.14)

From the second example of forming a region:

1. Set **ISOLINES** to **16**.
2. Call the **Extrude** tool. The command line shows:

Command:_extrude
Current wire frame density: ISOLINES=16
Select objects to extrude: *pick* the region **1 found**
Select objects to extrude: *right-click*
Specify height of extrusion or [Direction/Path/Taper angle]: *enter* **t** *right-click*
Specify angle of taper for extrusion: *enter* **5** *right-click*
Specify height of extrusion or [Direction/Path/Taper angle]: *enter* **100** *right-click*
Command:

3. *Click* **SW Isometric** in the **3D Views** sub-menu of the **View** drop-down menu.
4. **Zoom** to **1**.

Third example – Extrude (Fig. 13.16)

From the third example of forming a region:

1. Place the screen in the **3D Views/Front** view from the **View** drop-down menu (Fig. 13.15).
2. With the **Move** tool, move the arc to the centre of the region.
3. Place the screen in the **3D View/SW Isometric** view.
4. Set **ISOLINES** to **24**.
5. Call the **Extrude** tool. The command line shows:

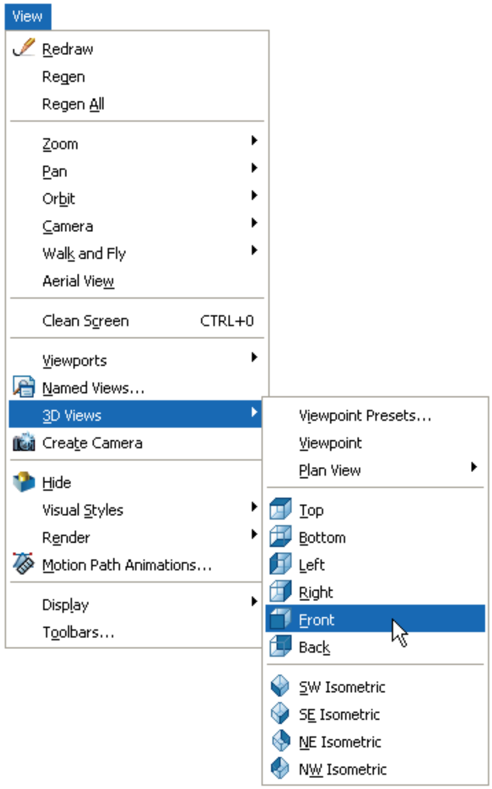


Fig. 13.15 Calling **Front** from the **3D Views** sub-menu of the **View** drop-down menu

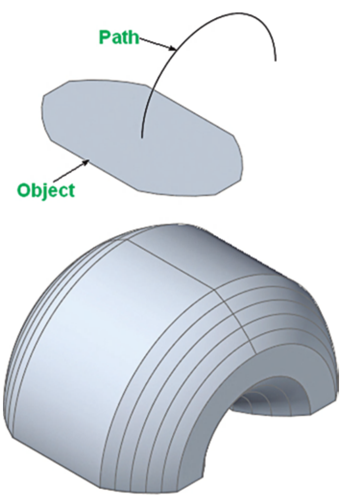


Fig. 13.16 Third example – **Extrude**

Command:_extrude
Current wire frame density: ISOLINES=24
Select objects to extrude: *pick* the region **1 found**
Select objects to extrude: *right-click*
Specify height of extrusion or [Direction/Path/Taper angle] <100>: *enter* **p** *right-click*
Select extrusion path or [Taper angle]: *pick* the path
Command:

The Revolve tool

The **Revolve** tool can be called with a *click* on its tool icon in the **Modeling** toolbar, or by a *click* on its tool icon in the **3D Make control panel**, or by a *click* on its name in the **Modeling** sub-menu of the **Draw** drop-down menu, or by *entering* **revolve**, or its abbreviation **rev**, at the command line.

Examples of the use of the Revolve tool

Solids of revolution can be constructed from closed plines or from regions.

First example – Revolve (13.19)

1. Construct the closed polyline in Fig. 13.17.
2. Set **ISOLINES** to **24**.

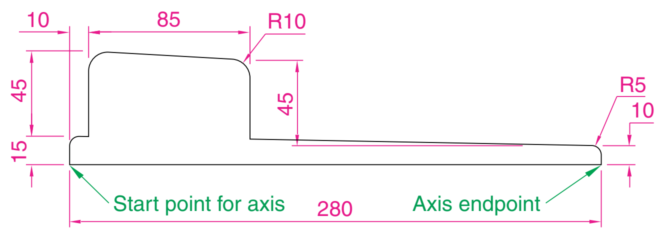


Fig. 13.17 First example – **Revolve**. The closed pline

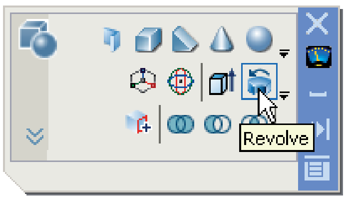


Fig. 13.18 Calling **Revolve** from its tool icon in the **3D Make control panel**

3. Call the **Revolve** tool either from the **Modeling** toolbar or from the **3D Make control panel** (Fig. 13.18). The command line shows:

Command:
Command:_revolve
Current wire frame density: ISOLINES=24
Select objects to revolve: *pick* the polyline **1 found**
Select objects to revolve: *right-click*
Specify axis start point or define axis by [Object/X/Y/Z] <Object>: *pick*
Specify axis endpoint: *pick*
Specify angle of revolution or [STart angle] <360>: *right-click*
Command:

4. Place in the **3D Views/SW Isometric** view.

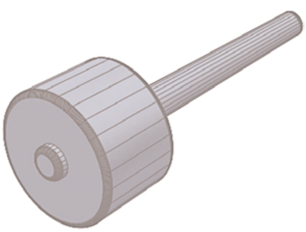


Fig. 13.19 First example – **Revolve**

Second example – Revolve (Fig. 13.21)

1. Place the screen in the **3D Views/Front** view. **Zoom** to **1**.
2. Construct the pline outline (Fig. 13.20).
3. Set **ISOLINES** to **24**.
4. Call the **Revolve** tool and construct a solid of revolution.
5. Place the screen in the **3D Views/SW Isometric** view.

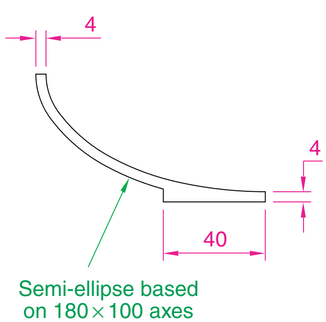


Fig. 13.20 Second example – **Revolve**. The pline outline

Third example – Revolve (Fig. 13.23)

1. Construct the pline in Fig. 13.22. The drawing must be either a closed pline or a region.
2. Call **Revolve** and form a solid of revolution through 180°.
3. Place the model in the **NE Isometric** view.

3D objects

At the command line:

Command: *enter* **3d** *right-click*
Enter an option [Box/Cone/Dlsh/DOme/Mesh/Pyramid/Sphere/Torus/Wedge]:

Fig. 13.21 Second example – **Revolve**

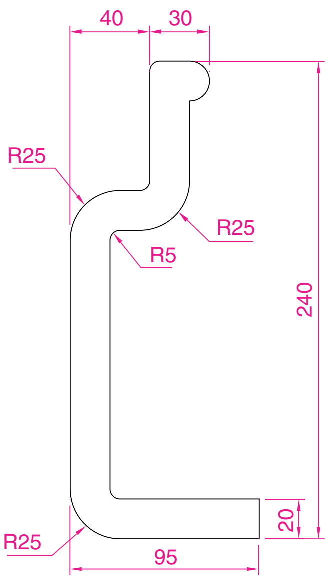


Fig. 13.22 Third example – **Revolve**. The outline to be revolved

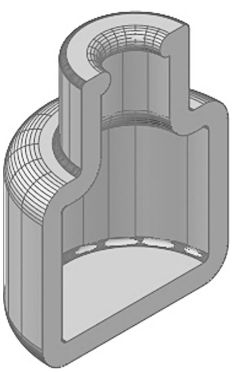


Fig. 13.23 Third example – **Revolve**

And the 3D objects' names appear. Any one of the 3D objects can then be called by *entering* the capital letter of the name of the 3D object. The 3D objects can also be called from the **Modeling** toolbar, from the **3D Make control panel**, or by *entering* the name (e.g. **box**) at the command line.

First example – 3D objects (Fig. 13.24)

1. Place the screen in the **3D Views/Front** view.
2. *Click* the **Box** tool icon in the **Modeling** toolbar or in the **3D Make control panel**. The command line shows:

Command:_box
Specify first corner or [Center]: *enter* **90,90** *right-click*
Specify other corner or [Cube/Length]: *enter* **110,−30** *right-click*
Specify height or [2Point]: *enter* **75** *right-click*
Command: *right-click*
BOX Specify first corner or [Center]: 110,90
Specify other corner or [Cube/Length]: 170,70
Specify height or [2Point]: 75
Command: *right-click*
BOX Specify first corner or [Center]: 110,−10
Specify other corner or [Cube/Length]: 200,−30
Specify height or [2Point]: 75
Command:

3. Place in the **3D Views/SE Isometric View** and **Zoom** to **1**.
4. Call the **Union** tool from the **Modeling** toolbar or from the **3D Make control panel**. The command line shows:

Command:_union
Select objects: *pick* one of the boxes **1 found**
Select objects: *pick* the second box **1 found, 2 total**
Select objects: *pick* the third box **1 found, 3 total**
Select objects: *right-click*
Command:

And the three boxes are joined in a single union.

Second example – 3D objects (Fig. 13.25)

1. Set **ISOLINES** to **16**.
2. *Click* the **Sphere** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

Command:_sphere
Specify center point or [3P/2P/Ttr]: 180,170
Specify radius or [Diameter]: 50
Command:

3. *Click* the **Cylinder** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

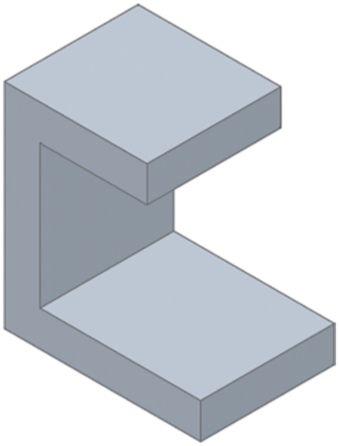


Fig. 13.24 First example – **3D Objects**

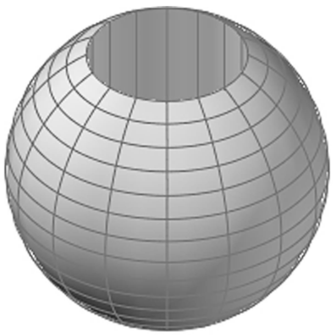


Fig. 13.25 Second example – **3D Objects**

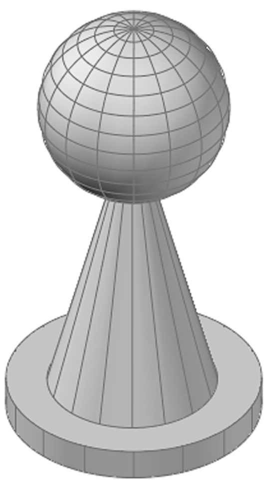


Fig. 13.26 Third example – **3D Objects**

Command:_cylinder
Specify center point of base or [3P/2P/Ttr/Elliptical]: 180,170
Specify base radius or [Diameter]: 25
Specify height or [2Point/Axis endpoint]: 110
Command:

4. Place the screen in the **3D Views/Front** view and **Zoom** to **1**.
5. With the **Move** tool, move the cylinder vertically down so that the bottom of the cylinder is at the bottom of the sphere.
6. *Click* the **Subtract** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

Command:_subtract Select solids and regions to subtract from . . .
Select objects: *pick* the sphere **1 found**
Select objects: *right-click*
Select solids and regions to subtract
Select objects: *pick* the cylinder **1 found**
Select objects: *right-click*
Command:

7. Place the screen in the **3D Views/SW Isometric** view and **Zoom** to **1**.

Third example – 3D objects (Fig. 13.26)

1. Call the **Cylinder** tool and with a centre **170,150** construct a cylinder of radius **60** and height **15**.
2. *Click* the **Cone** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

Command:_cone
Specify center point of base or [3P/2P/Ttr/Elliptical]: 170,150
Specify base radius or [Diameter]: 40
Specify height or [2Point/Axis endpoint/Top radius]: 150
Command:

3. Call the **Sphere** tool and construct a sphere of centre **170,150** and radius **45**.
4. Place the screen in the **3D Views/Front** view and, with the **Move** tool, move the cone and sphere so that the cone is resting on the cylinder and the centre of the sphere is at the apex of the cone.
5. Place in the **3D Views/SW Isometric** view and **Zoom** to **1** and with **Union** form a single 3D model from the three objects.

Fourth example – 3D objects (Fig. 13.27)

1. *Click* the **Box** tool icon in the **Modeling** toolbar or **3D Make control panel** and construct two boxes – the first of corners **70,210** and **290,120** and of height **10**, the second of corners **120,200,10** and **240,130,10** and of height **80**.
2. Place the screen in the **3D Views/Front** view and **Zoom** to **1**.

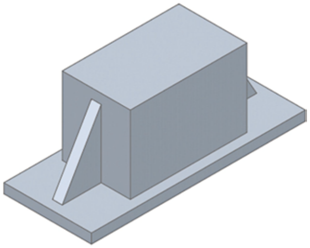


Fig. 13.27 Fourth example – **3D Objects**

3. *Click* the **Wedge** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

Command:_wedge
Specify first corner or [Center]: 120,160,10
Specify other corner or [Cube/Length]: 80,170,10
Specify height or [2Point]: 70
Command: *right-click*
WEDGE
Specify first corner of wedge or [Center]: 240,170,10
Specify corner or [Cube/Length]: 280,160,10
Specify height or [2Point]: 70
Command:

4. Place the screen in the **3D Views/SW Isometric** view and **Zoom** to **10**.
5. Call the **Union** tool from the **Modeling** toolbar or **3D Make control panel** and in response to the prompts in the tool's sequences *pick* each of the four objects in turn to form a union of the four objects.

Fifth example – 3D objects (Fig. 13.28)

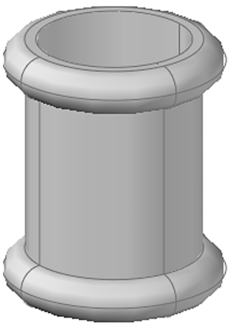


Fig. 13.28 Fifth example – **3D Objects**

1. Using the **Cylinder** tool from the **Modeling** toolbar or **3D Make control panel**, construct a cylinder of centre **180,160**, radius **40** and height **120**.
2. *Click* the **Torus** tool icon in the **Modeling** toolbar or **3D Make control panel**. The command line shows:

Command:_torus
Specify center point or [3P/2P/Ttr]: 180,160,10
Specify radius or [Diameter]: 40
Specify tube radius or [2Point/Diameter]: 10
Command: *right-click*
TORUS
Specify center point or [3P/2P/Ttr]: 180,160,110
Specify radius or [Diameter] <40>: *right-click*
Specify tube radius or [2Point/Diameter] <10>: *right-click*
Command:

3. Call the **Cylinder** tool and construct another cylinder of centre **180,160**, radius **35** and height **120**.
4. Place in the **3D Views/SW Isometric** view and **Zoom** to **1**.
5. *Click* the **Union** tool icon in the **Modeling** toolbar or **3D Make control panel** and form a union of the larger cylinder and the two torii.
6. *Click* the **Subtract** tool icon in the **Modeling** toolbar or **3D Make control panel** and subtract the smaller cylinder from the union.

The Chamfer and Fillet tools

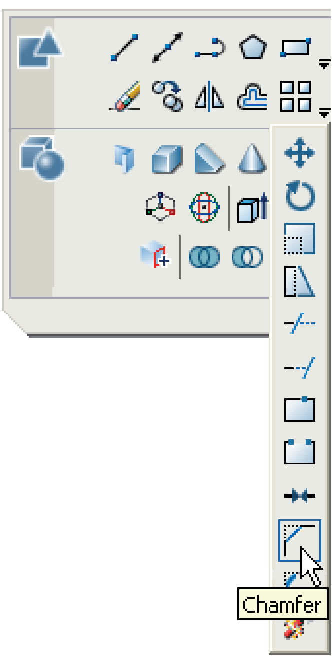


Fig. 13.29 The **Chamfer** tool in the **2D Draw control panel**

The **Chamfer** and **Fillet** tools from the **Modify** toolbar or **2D Draw control panel** (Fig. 13.29), which are used to create chamfers and fillets in 2D drawings in AutoCAD 2007, can just as well be used when constructing 3D models.

Example – Chamfer and Fillet (Fig. 13.32)

1. Using the **Box** and **Cylinder** tools, construct the 3D model in Fig. 13.30.

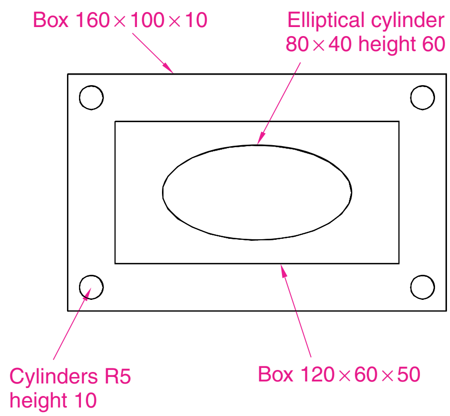


Fig. 13.30 Example – **Chamfer and Fillet** – the model before using the tools

2. Place in the **3D Views/SW Isometric** view (Fig. 13.31). **Union** the two boxes and with the **Subtract** tool, subtract the cylinders from the union.

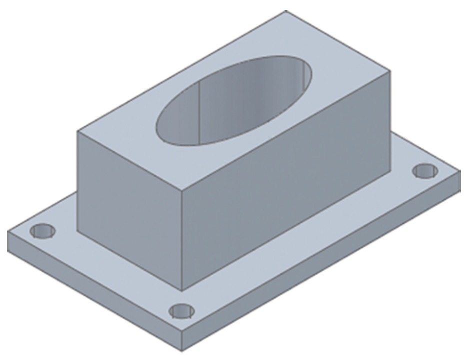


Fig. 13.31 Example – isometric view – **Chamfer and Fillet** – the model before using the tools

Note

To construct the elliptical cylinder:

Command:_cylinder
Specify center point of base or [3P/2P/Ttr/Elliptical]: *enter* **e** *right-click*
Specify endpoint of first axis or [Center]: 130,160
Specify other endpoint of first axis: 210,160
Specify endpoint of second axis: 170,180
Specify height or [2Point/Axis endpoint]: 50
Command:

3. *Click* the **Fillet** tool icon in the **Modify** toolbar or **2D Draw control panel**. The command line shows:

Command:_fillet
Current settings: Mode = TRIM. Radius = 1

Specify first object or [Undo/Polyline/Radius/Trim/Multiple]: *enter* **r** (Radius) *right-click*
Specify fillet radius <1>: 10
Select first object: *pick* one corner
Select an edge or [Chain/Radius]: *pick* a second corner
Select an edge or [Chain/Radius]: *pick* a third corner
Select an edge or [Chain/Radius]: *pick* the fourth corner
Select an edge or [Chain/Radius]: *right-click*
4 edge(s) selected for fillet.
Command:

4. *Click* the **Chamfer** tool icon in the **Modify** toolbar. The command line shows:

Command:_chamfer
(TRIM mode) Current chamfer Dist1 = 1, Dist2 = 1
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple] *enter* **d** (Distance) *right-click*
Specify first chamfer distance <1>: 10
Specify second chamfer distance <10>: 10
Select first line: *pick* one corner One side of the box highlights
Base surface selection . . . **Enter surface selection [Next/OK (current)] <OK>:** *right-click*
Specify base surface chamfer distance <10>: *right-click*
Specify other surface chamfer distance <10>: *right-click*
Select an edge or [Loop]: *pick* the edge again **Select an edge:** *pick* he second edge
Select an edge [or Loop]: *right-click*
Command:

And two edges are chamfered. Repeat to chamfer the other two edges (Fig. 13.32).

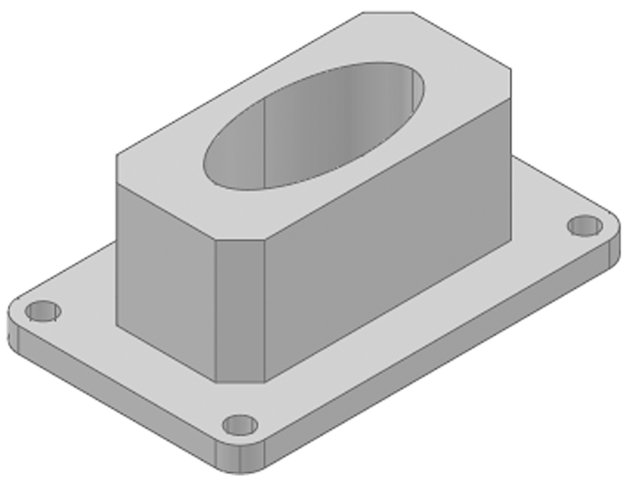


Fig. 13.32 Example – **Chamfer and Fillet**

Note on the tools Union, Subtract and Intersect

The tools **Union, Subtract** and **Intersect** found in the **Solids Editing** toolbar are known as the **Boolean** operators after the mathematician Boolean. They can be used to form unions, subtractions or intersections between extrusions, solids of revolution or any of the 3D Objects.

Note on using Modify tools on 3D models

As was seen above while using the **Move, Chamfer** and **Fillet** tools from the **Modify** toolbar, so also can other tools like **Copy, Mirror, Rotate** and **Scale** from the toolbar be used in connection with the construction of 3D models.

Constructing 3D surfaces using the Extrude tool

In this example of the construction of a 3D surface model the use of the **DYN** (Dynamic input) method of construction will be shown.

1. Place the AutoCAD drawing area in the **SW Isometric** view.
2. *Click* the **DYN** button in the status bar to make dynamic input active.

Example (Fig. 13.34)

1. Using the line tool construct the outline in Fig. 13.33.
2. Call the **Extrude** tool and *window* the line outline.
3. Extrude to a height of **100**.

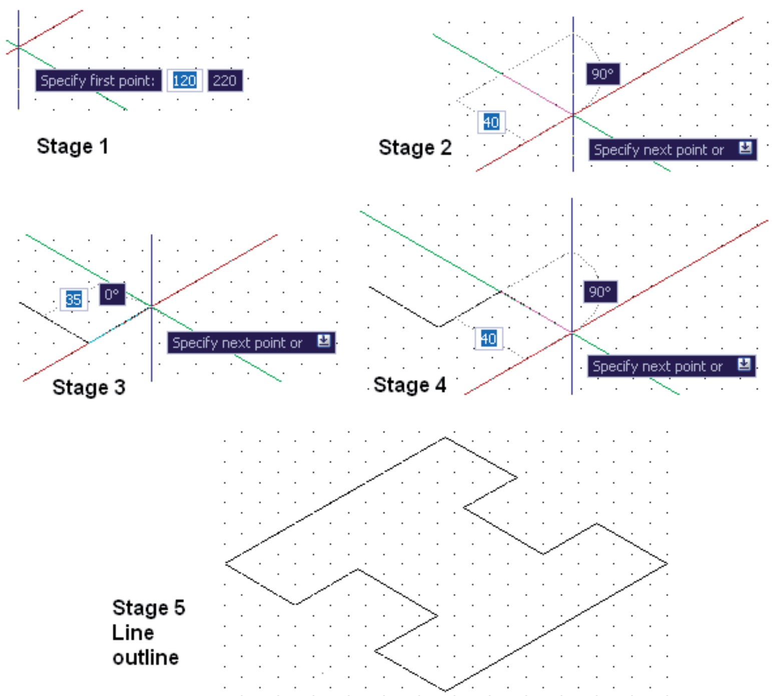


Fig. 13.33 Example – constructing the **Line** outline

The stages of producing the extrusion are shown in Fig. 13.34. The resulting 3D model is a surface model.

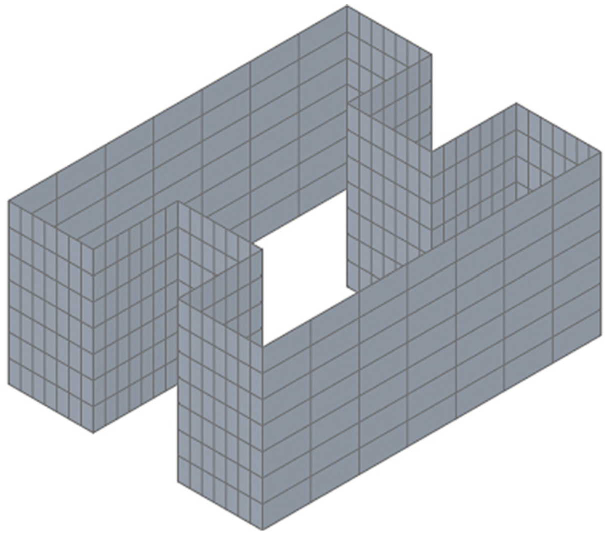


Fig. 13.34 Example – constructing a 3D surface using the **Extrude** tool

The Sweep tool

The modelling tool **Sweep** is new to AutoCAD 2007. To call the tool *click* on its tool icon in the **Modeling** toolbar or **3D Make control panel** (Fig. 13.35).

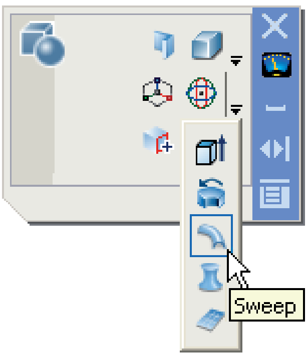


Fig. 13.35 The **Sweep** tool icon in the **3D Make control panel**

Example – Sweep (Fig. 13.38)

1. Construct the pline outline in Fig. 13.36 in the **3D Views/Top** view.
2. Change to the **3D Views/Front** view, **Zoom** to **1** and construct a pline as shown in Fig. 13.37 as a path central to the ellipse.
3. Place the window in the **3D Views/SW Isometric** view and *click* the **Sweep** tool icon. The command line shows:

   Command:_sweep
   Current wire frame density: ISOLINES=4
   Select objects to sweep: *pick* the ellipse **1 found**
   Select objects to sweep: *right-click*
   Select sweep path or [Alignment/Base point/Scale/Twist]: *pick* the pline
   Command:

The result is shown in Fig. 13.38.

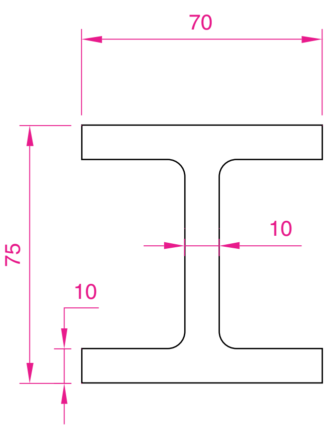


Fig. 13.36 Example **Sweep** – the outline to be swept

The Loft tool

The modelling tool **Loft** is new to AutoCAD 2007. To call the tool *click* on its tool icon in the **Modeling** toolbar or **3D Make control panel** (Fig. 13.39).

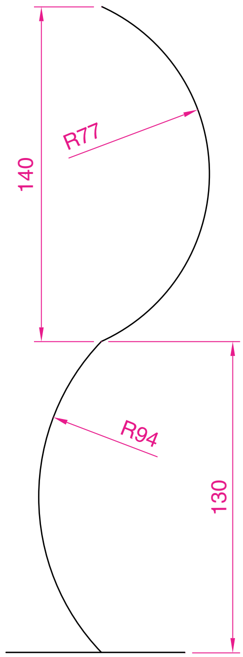


Fig. 13.37 Example **Sweep** – the pline path

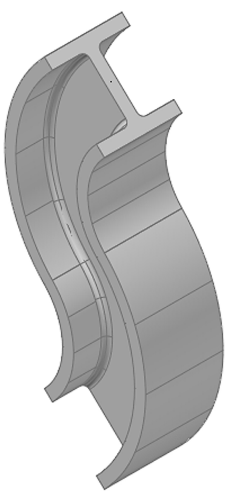


Fig. 13.38 Example – **Sweep**

Example – Loft (Fig. 13.42)

1. Construct the seven circles shown in Fig. 13.40 at vertical distances of **30** units apart.
2. Place the drawing area on the **3D Views/SW Isometric** view (Fig. 13.40).
3. Call the **Loft** tool with a *click* on its tool icon in the **Modeling** toolbar or **3D Make control panel** (Fig. 13.39).
4. The command line shows:

Command:_loft
Select cross-sections in lofting order: <Snap off> *pick* the bottom circle **1 found**
Select cross-sections in lofting order: *pick* the next circle **1 found, 2 total**
Select cross-sections in lofting order: *pick* the next circle **1 found, 3 total**
Select cross-sections in lofting order: *pick* the next circle **1 found, 4 total**
Select cross-sections in lofting order: *pick* the next circle **1 found, 5 total**
Select cross-sections in lofting order: *pick* the next circle **1 found, 6 total**
Select cross-sections in lofting order: *pick* the next circle **1 found, 7 total**
Select cross-sections in lofting order: *right-click*

And the **Loft Settings** dialog appears (Fig. 13.41).

5. *Click* the **Smooth Fit** button, followed by a *click* on the **OK** button. The loft appears.

The result is shown in Fig. 13.42.

Revision notes

1. In the AutoCAD 3D coordinate system, positive Z is towards the operator away from the monitor screen.
2. A 3D face is a mesh behind which other details can be hidden.
3. The **Extrude** tool can be used for extruding closed plines or regions to stated heights, to stated slopes or along paths.
4. The **Revolve** tool can be used for constructing solids of revolution through any angle up to 360°.
5. 3D models can be constructed from the 3D objects **Box, Sphere, Cylinder, Cone, Torus** and **Wedge**. Extrusions and/or solids of revolutions may form part of models constructed using the 3D objects.
6. Tools such as **Chamfer** and **Fillet** from the **Modify** toolbar can be used when constructing 3D models.

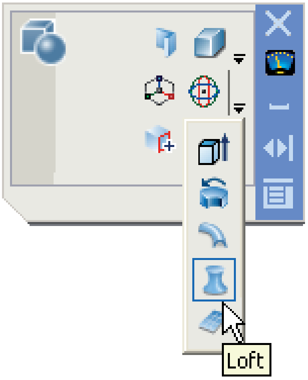


Fig. 13.39 The **Loft** tool icon in the **3D Make control panel**

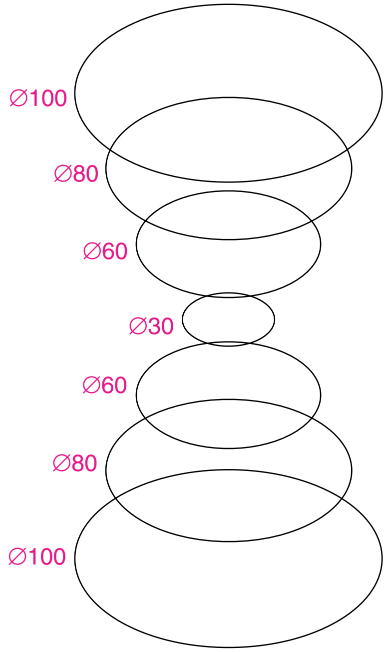


Fig. 13.40 Example **Loft** – the cross sections

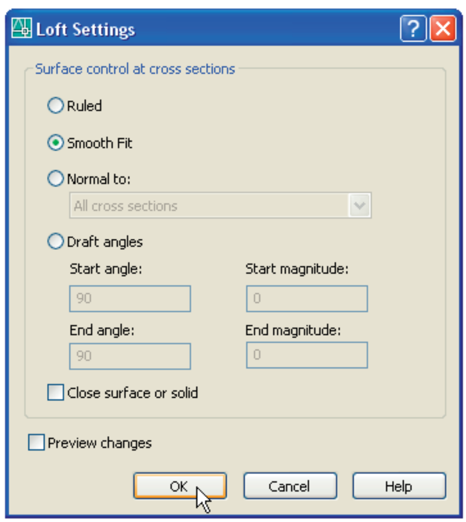


Fig. 13.41 The **Loft Settings** dialog

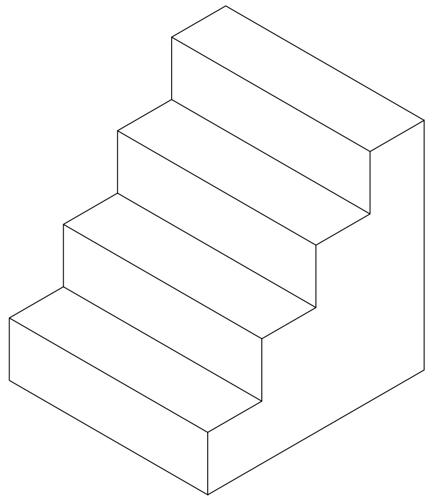
Fig. 13.42 Example – **Loft**

7. The tools **Union, Subtract** and **Intersect** are known as the **Boolean** operators.
8. When outlines which are not closed polylines or regions are acted upon by the **Modeling** tool **Extrude** the resulting models will be 3D Surface models and not 3D solid models.

Exercises

The first three exercises given below give practice in the use of the **3D Face** and **Hide** tools. A variety of 3D viewing positions selected from **Views/3D Views** will need to be used when constructing the 3D model drawings in answer to these three exercises.

1. Construct the four steps shown in Fig. 13.43, working to the given sizes.

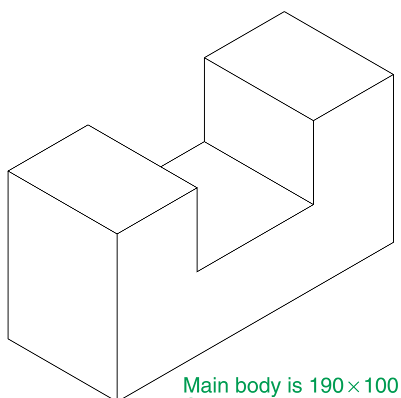


Each step is 110 long
by 30 deep and 30 high

Fig. 13.43 Exercise 1

2. Construct the 3D model given in Fig. 13.44, working to the sizes given with the drawing.

3. Construct the 3D model drawing of a V block shown in Fig. 13.45, working to the sizes given in the right-hand drawing. The V block is 100 deep.



Main body is 190 × 100 × 75
Cut out portion is 80 × 75 × 50

Fig. 13.44 Exercise 2

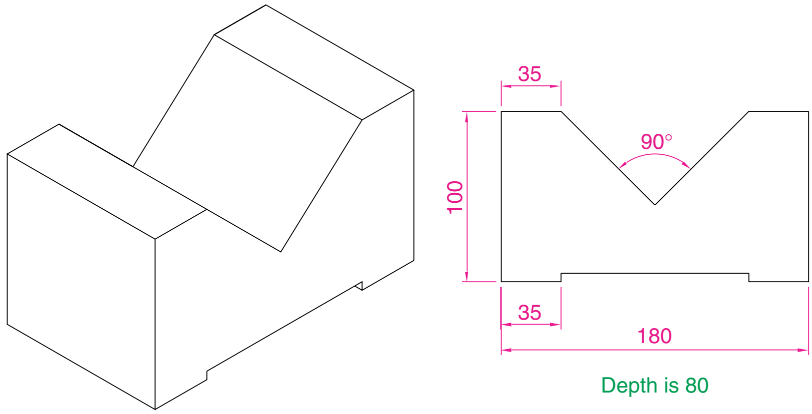
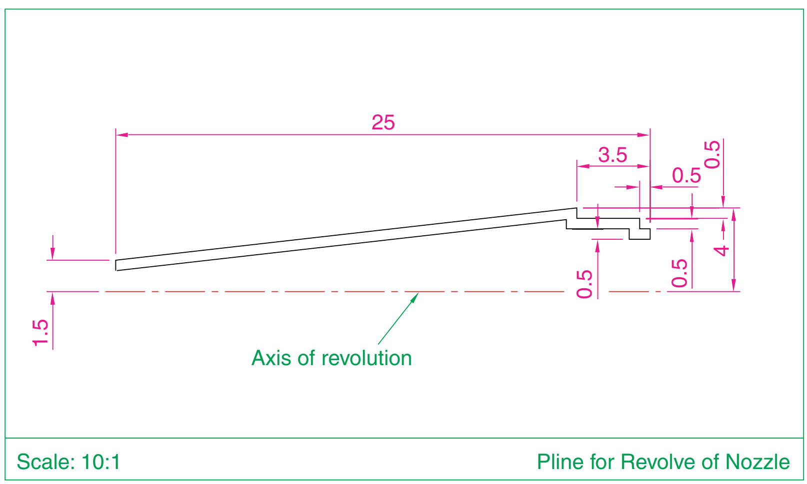


Fig. 13.45 Exercise 3

The exercises which follow require using the 3D objects tools and the tools **Revolve, Extrude, Union** and **Subtract**.

4. Fig. 13.46 shows the outline from which a solid of revolution can be constructed. Using the **Revolve** tool construct the solid of revolution.

Fig. 13.46 Exercise 4



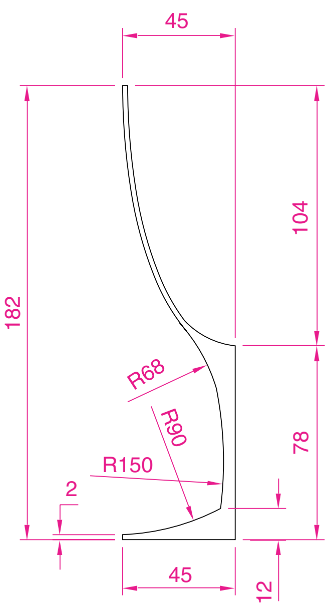


Fig. 13.47 Exercise 5

5. Construct the 3D model of a wine glass, working to the dimensions given in the outline drawing in Fig. 13.47.

You will need to construct the outline and change it into a region before being able to change the outline into a solid of revolution using the **Revolve** tool. This is because the semi-elliptical part of the outline has been constructed using the **Ellipse** tool, resulting in part of the outline being a spline, which cannot be acted upon by **Polyline Edit** to form a closed pline.

6. Construct a 3D solid model of a bracket, working to the information given in Fig. 13.48.

7. Working to the dimensions given in Fig. 13.49, construct an extrusion of the plate to a height of **5** units.

8. Working to the details given in the orthographic projection in Fig. 13.50, construct a 3D model of the assembly.

After constructing the pline outline(s) required for the solid(s) of revolution, use the **Revolve** tool to form the 3D solid.

9. Working to the polylines shown in Fig. 13.51, construct the **Sweep** shown in Fig. 13.52.

10. Construct the cross sections as shown in Fig. 13.53, working to suitable dimensions. From the cross sections construct the lofts shown in Fig. 13.54. The lofts are topped with a sphere constructed using the **Sphere** tool.

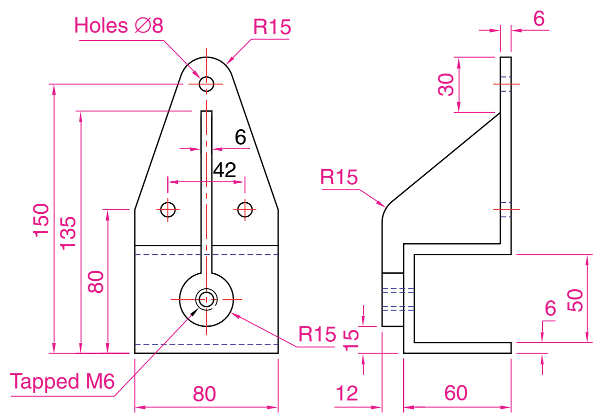


Fig. 13.48 Exercise 6

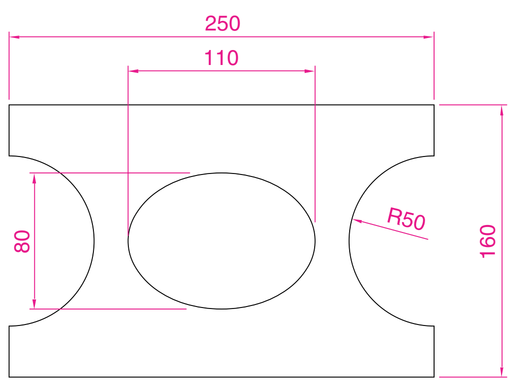


Fig. 13.49 Exercise 7

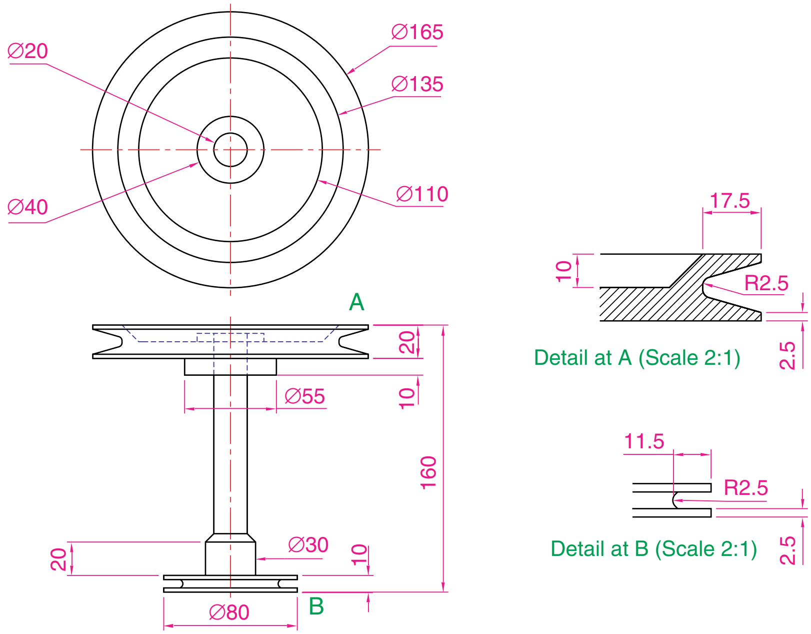


Fig. 13.50 Exercise 8

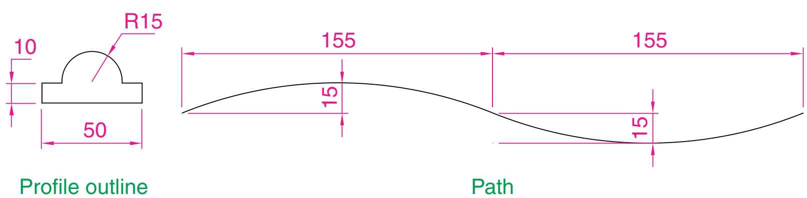


Fig. 13.51 Exercise 9 – profile and path dimensions

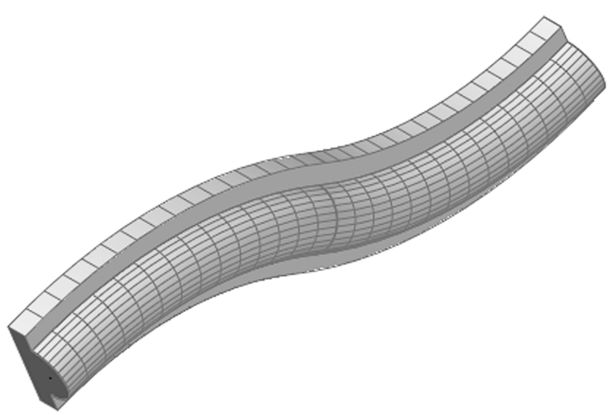


Fig. 13.52 Exercise 9

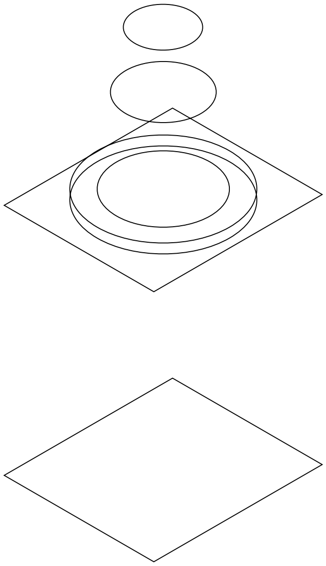


Fig. 13.53 The cross sections for Exercise 10

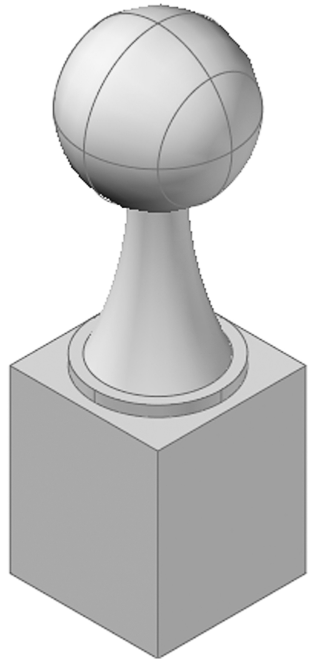


Fig. 13.54 Exercise 10

CHAPTER 14

3D models in viewports

Aim of this chapter

To give examples of 3D solid models constructed in multiple viewport settings.

Setting up viewport systems

One of the better methods of constructing 3D models is in different viewport settings. This allows what is being constructed to be seen from a variety of viewing positions. To set up a new viewport system:

1. *Click* **View** in the menu bar and from the drop-down menu which appears *click* **Viewports** and in the sub-menu which then appears *click* **New Viewports** . . . (Fig. 14.1). The **Viewports** dialog appears (Fig. 14.2).
2. *Click* the **New Viewports** tab and a number of named viewports systems appear in the **Standard viewports** list in the dialog.

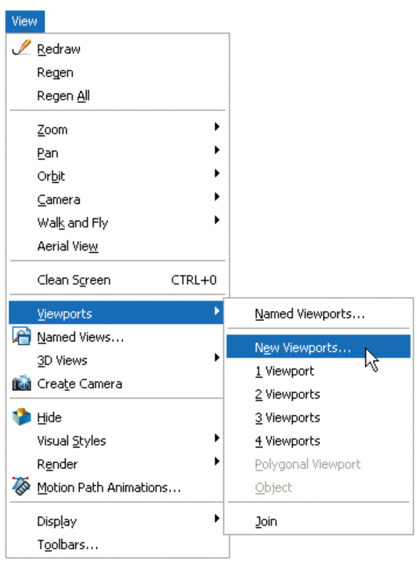


Fig. 14.1 Selecting **New Viewports** . . . from the **View** drop-down menu

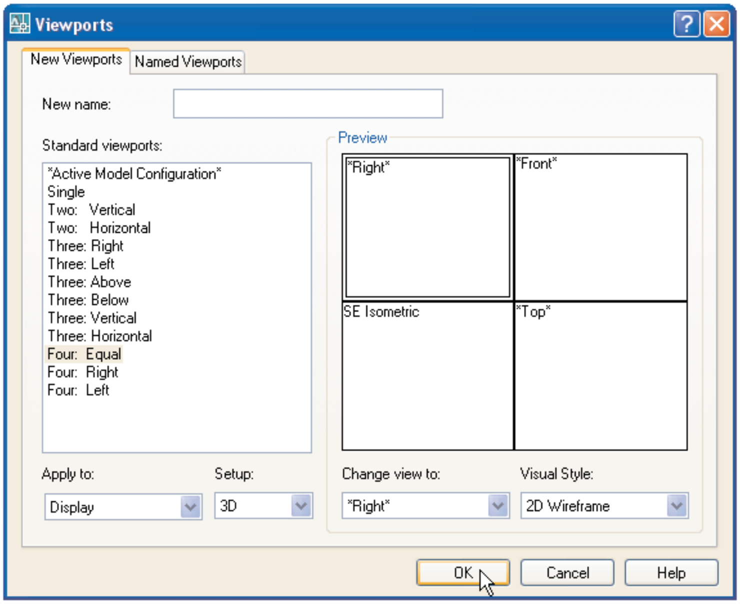


Fig. 14.2 The **Viewports** dialog

3. *Click* the name **Four: Equal**, followed by a *click* on **3D** in the **Setup** popup list. A preview of the **Four: Equal** viewports screen appears showing the views appearing in each of the four viewports.
4. *Click* the **OK** button of the dialog and the AutoCAD 2007 drawing area appears showing the four viewport layout (Fig. 14.3).

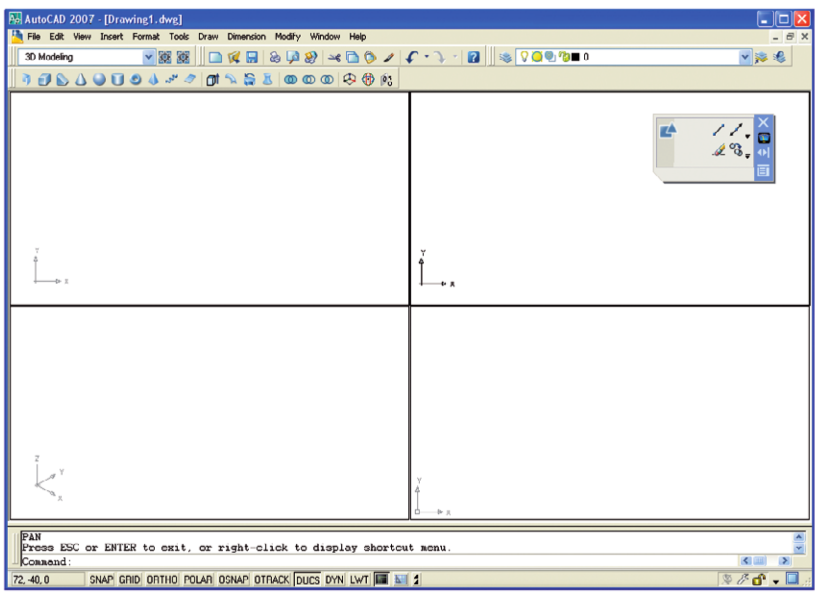


Fig. 14.3 The **Four: Equal** viewports layout

First example – Four: Equal viewports (Fig. 14.7)

Fig. 14.4 shows a first angle orthographic projection of a support. To construct a **Scale 1:1** 3D model of the support in a **Four: Equal** viewport setting:

1. *Click* **View** in the menu bar, followed by a *click* on **Viewports** in the drop-down menu, followed by another *click* on **New Viewports** . . . in the **Viewports** sub-menu. Make sure the **3D** option is selected from the **Setup** popup list and *click* the **OK** button of the dialog. The AutoCAD 2007 drawing area appears in a **Four: Equal** viewport setting.
2. *Click* in the **Top** viewport (bottomright-hand corner viewport) to make it current.
3. Set **ISOLINES** to **4**.
4. Using the **Polyline** tool, construct the outline of the plan view of the plate of the support, including the holes (Fig. 14.5). Note the views in the other viewports.
5. Call the **Extrude** tool from the **Solids** toolbar and extrude the plan outline and the circles to a height of **20**.
6. With the **Subtract** tool from the **Solids Editing** toolbar, subtract the holes from the plate (Fig. 14.6).
7. Call the **Box** tool and in the centre of the plate construct a box of Width = **60**, Length = **60** and Height = **30**.
8. Call the **Cylinder** tool and in the centre of the box construct a cylinder of Radius = **20** and Height = **30**.
9. Call **Subtract** and subtract the cylinder from the box.
10. *Click* in the **Right** viewport and with the **Move** tool, move the box and its hole into the correct position with regard to the plate.
11. With **Union**, form a union of the plate and box.

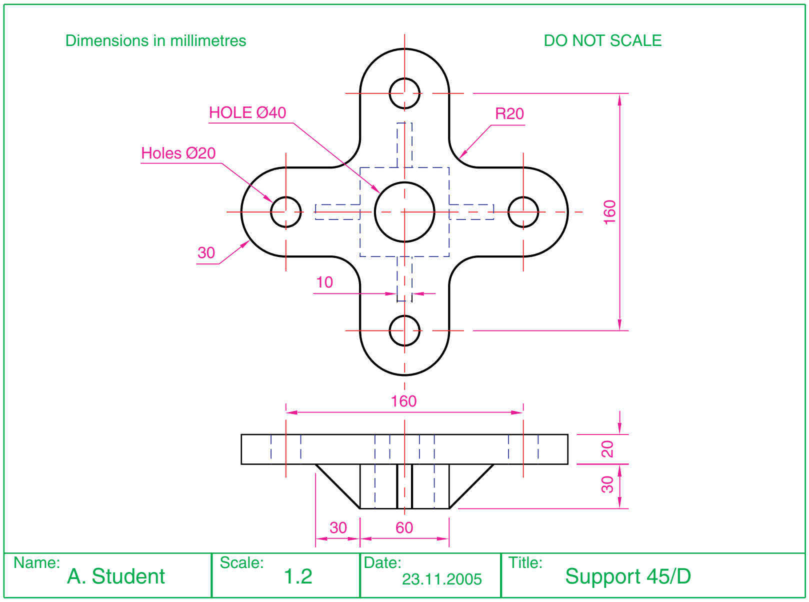


Fig. 14.4 Orthographic projection of the support for the first example

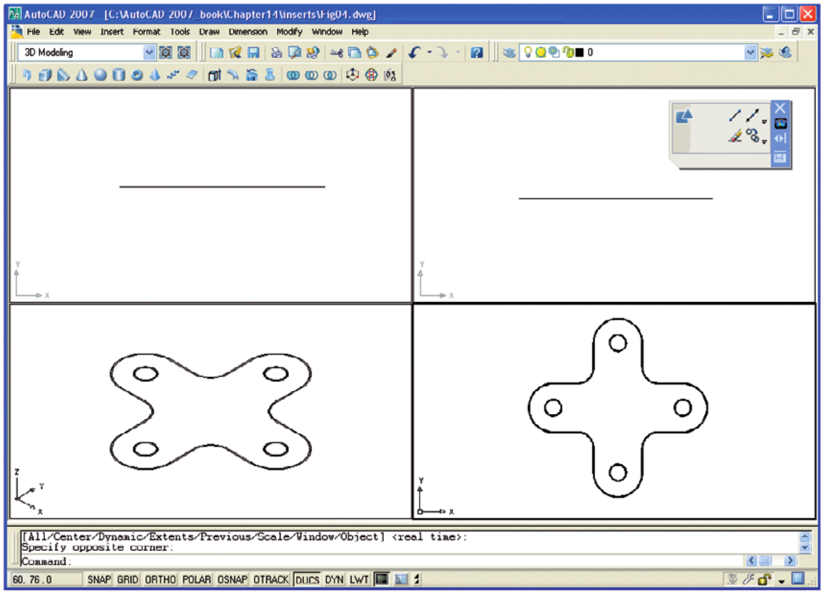


Fig. 14.5 The plan view drawn

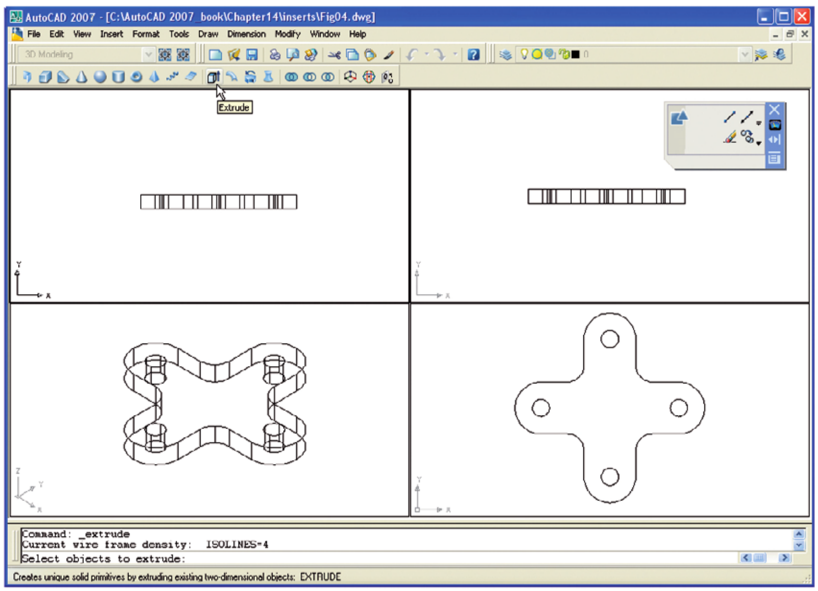


Fig. 14.6 The four views after using the **Extrude** and **Subtract** tools

12. *Click* in the **Front** viewport and construct a triangle for one of the webs attached between the plate and the box. With **Extrude**, extrude the triangle to a height of **10**. With the **Mirror** tool, mirror the web to the other side of the box.
13. *Click* in the **Right** viewport and with the **Move** tool, move the two webs into their correct position between the box and the plate. Then, with **Union**, form a union between the webs and the 3D model.

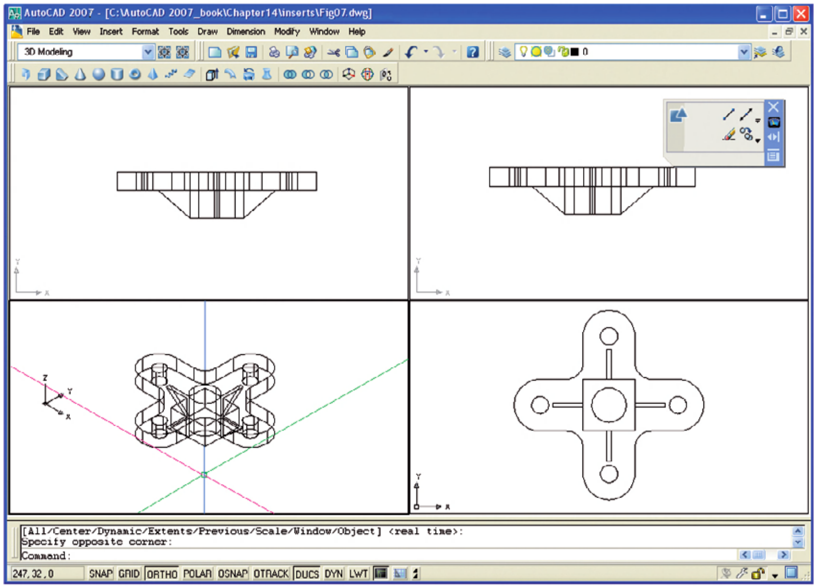


Fig. 14.7 First example – **Four: Equal viewports**

14. While in the **Right** viewport, construct the other two webs and in the **Front** viewport, move, mirror and union the webs as in steps **12** and **13**.

Fig. 14.7 shows the resulting four-viewport scene.

Second example – Four: Left viewports (Fig. 14.9)

1. Open the **Four: Left** viewport layout from the **Viewports** dialog.
2. In the **Top** viewport construct an outline of the web of the Support Bracket shown in Fig. 14.8. With the **Extrude** tool, extrude the parts of the web to a height of **20**.

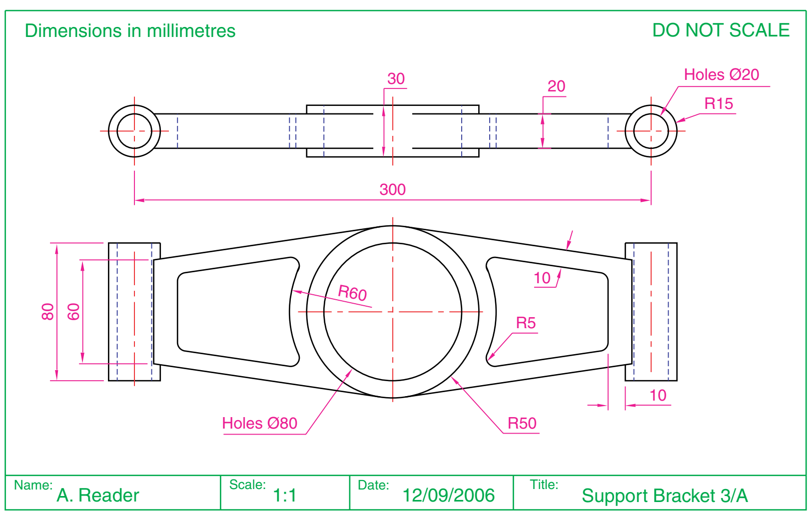


Fig. 14.8 Working drawing for the second example

3. With the **Subtract** tool, subtract the holes form the web.
4. In the **Top** viewport, construct two cylinders central to the extrusion, one of radius **50** and height **30**, the second of radius **40** and height **30**. With the **Subtract** tool, subtract the smaller cylinder from the larger.
5. *Click* in the **Front** viewport and move the cylinders vertically by **5** units. With **Union** form a union between the cylinders and the web.
6. Make the **Front** viewport active and at one end of the union, construct two cylinders, the first of radius **10** and height **80**, the second of radius **15** and height **80**. Subtract the smaller from the larger.
7. With the **Mirror** tool, mirror the cylinders to the other end of the union.
8. Make the **Top** viewport current and with the **Move** tool, move the cylinders to their correct position at the ends of the union. Form a union between all parts on screen.
9. Make the **SE Isometric** viewport current. At the command line *enter* **3dorbit**. *Right-cick* in the viewport and select **Visual Styles/Conceptual** from the *right-click* menu (see Chapter 16 about **3D Orbit**).

Fig. 14.9 shows the result.

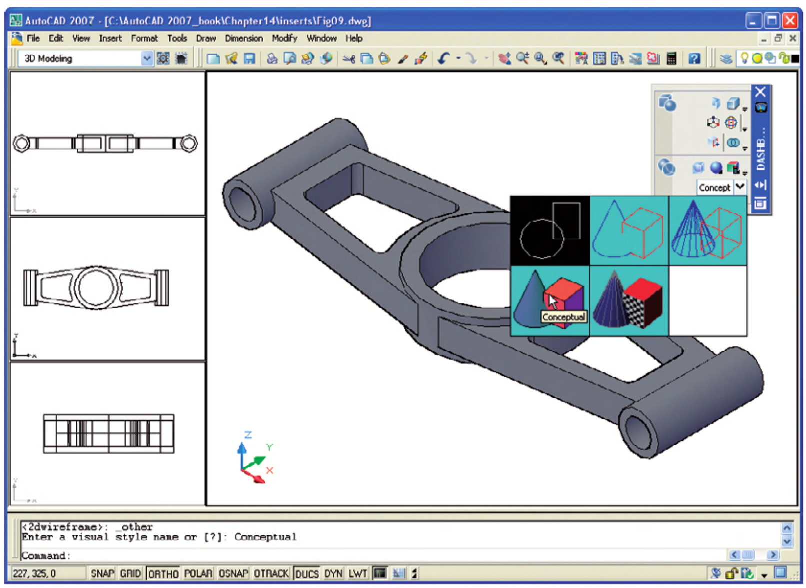


Fig. 14.9 Second example – **Four: Left viewports**

Third example – Three: Right viewports (Fig. 14.11)

1. Open the **Three: Right** viewport layout from the **Viewports** dialog. Make sure **3D** setup is chosen.
2. *Click* in the **Top** viewport (top left-hand viewport) and change it to a **3D Views/Right** view from the **View** drop-down menu.
3. In what is now the **Right** viewport construct a pline outline to the dimensions in Fig. 14.10.

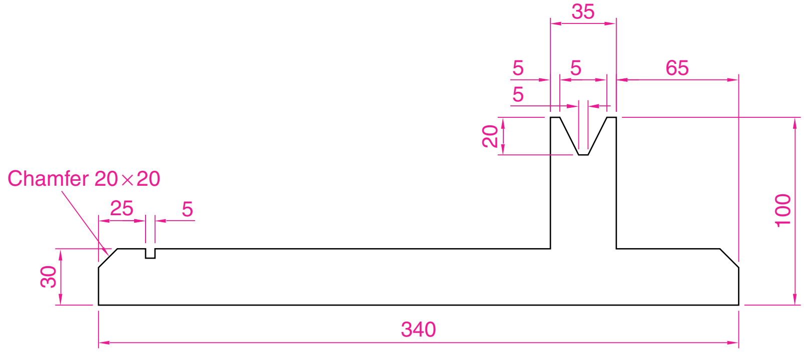


Fig. 14.10 Third example – outline for solid of revolution

4. Call the **Revolve** tool from the **Solids** toolbar and revolve the outline through 360°.
5. Make the **SE Isometric** viewport current. At the command line *enter* **3dorbit**. *Right-click* in the viewport and select **Visual Styles/Conceptual** from the *right-click* menu.

The result is shown in Fig. 14.11.

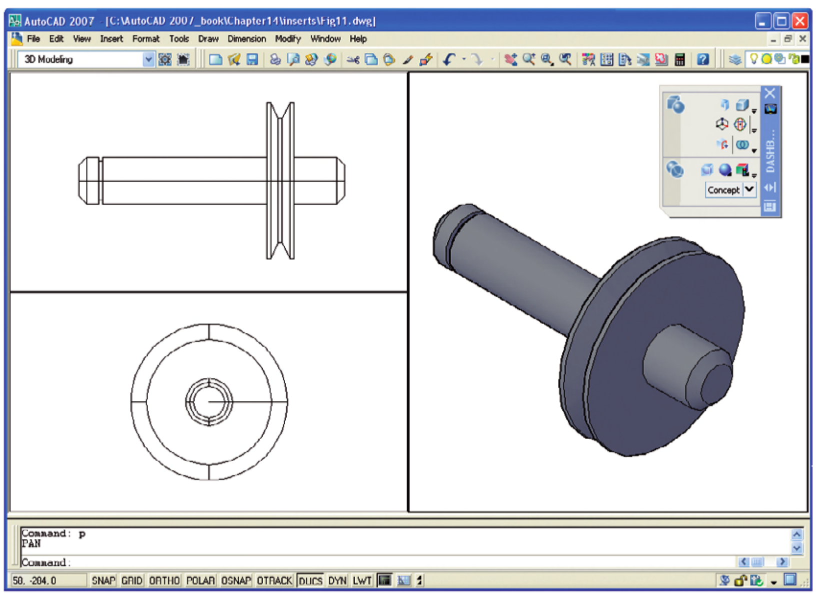


Fig. 14.11 Third example – **Three: Right viewports**

Notes

1. When working in viewport layouts such as in the above three examples, it is important to make good use of the **Zoom** tool, mainly because the viewports are smaller than the single viewport when working in AutoCAD 2007.

2. As in all other forms of constructing drawings in AutoCAD 2007 frequent toggling of **SNAP, ORTHO** and **GRID** will allow speedier and more accurate working.

Revision notes

1. Outlines suitable for use when constructing 3D models can be constructed using the 2D tools such as **Line, Arc, Circle** and **Polyline**. Such outlines must be changed either to closed polylines or to regions before being incorporated in 3D models.
2. The use of multiple viewports can be of value when constructing 3D models in that various views of the model appear enabling the operator to check the accuracy of the 3D appearance throughout the construction period.

Exercises

1. Using the **Cylinder, Box, Sphere, Wedge** and **Fillet** tools together with the **Union** and **Subtract** tools and working to any sizes thought suitable, construct the 'head' in the **Three: Right** viewport as shown in Fig. 14.12.
2. Using the tools **Sphere, Box, Union** and **Subtract** and working to the dimensions given in Fig. 14.13, construct the 3D solid model as shown in the isometric drawing in Fig. 14.14.

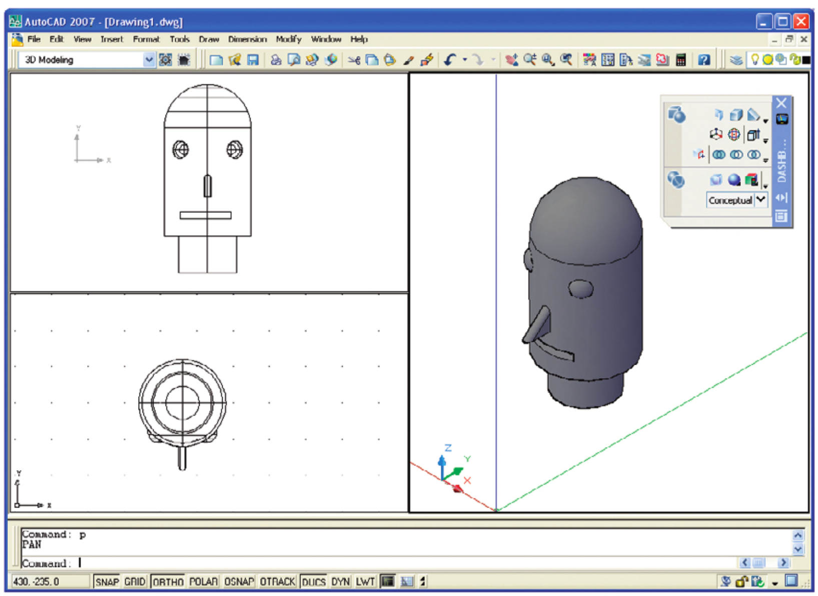


Fig. 14.12 Exercise 1

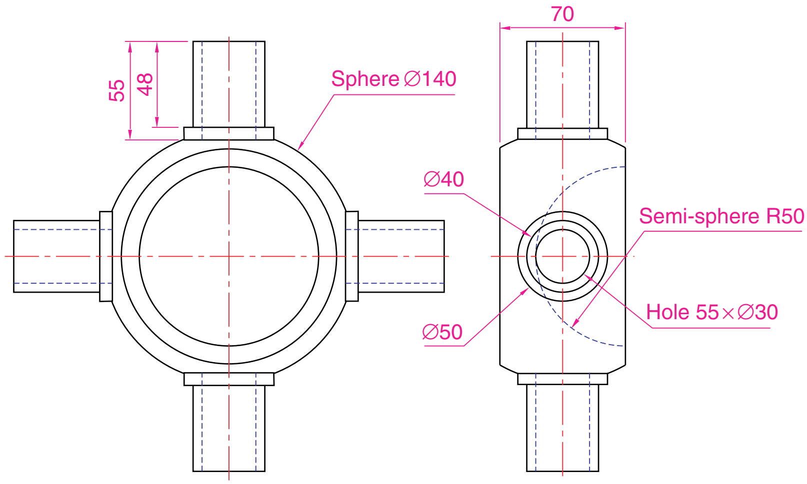


Fig. 14.13 Exercise 2 – working drawing

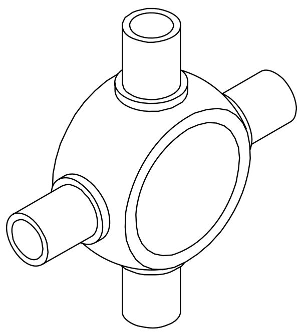


Fig. 14.14 Exercise 2

3. Each link of the chain shown in Fig. 14.15 has been constructed using the **Extrude** tool, extruding a small circle along an elliptical path. Copies of the link were then made, half of which were rotated in a **3D Views/Right** view and then moved into their positions relative to the other links. Working to suitable sizes, construct a link and from the link construct the chain as shown.
4. A two-view orthographic projection of a rotatable lever from a machine is given in Fig. 14.16 together with an isometric drawing

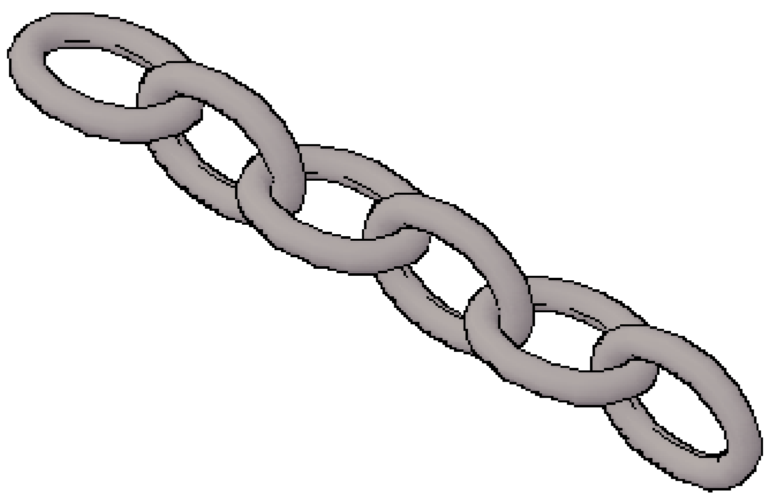


Fig. 14.15 Exercise 3

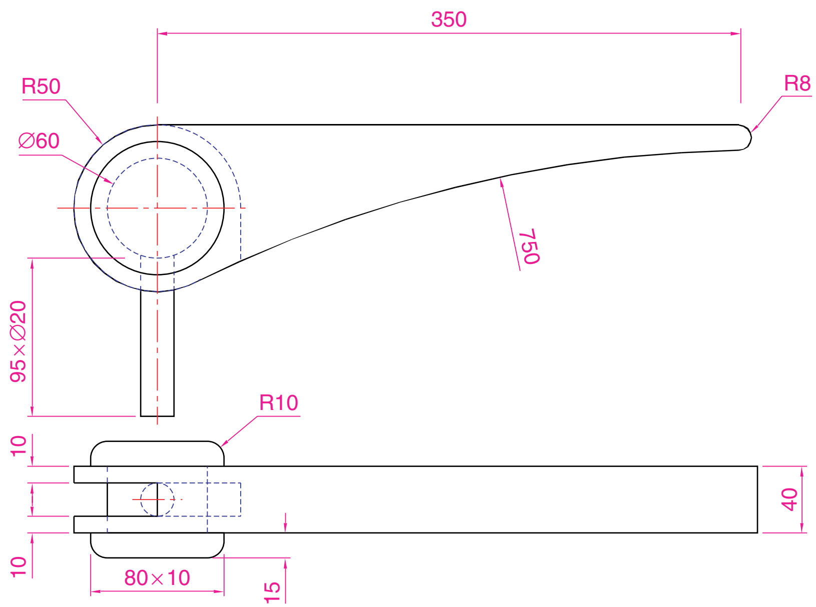


Fig. 14.16 Exercise 4 – orthographic projection

of the 3D model constructed to the details given in the drawing in Fig. 14.17. Construct the 3D model drawing in a **Four: Equal** viewport setting.

5. Working in a **Three: Left** viewport setting, construct a 3D model of the faceplate to the dimensions given in Fig. 14.18. With the **Mirror** tool, mirror the model to obtain an opposite facing model. In the **Isometric** viewport call the **Hide** tool (Fig. 14.19).

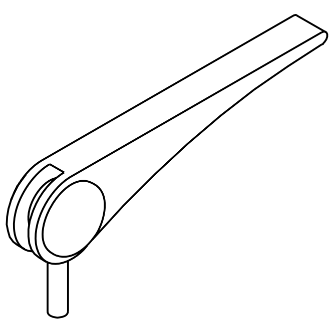


Fig. 14.17 Exercise 4

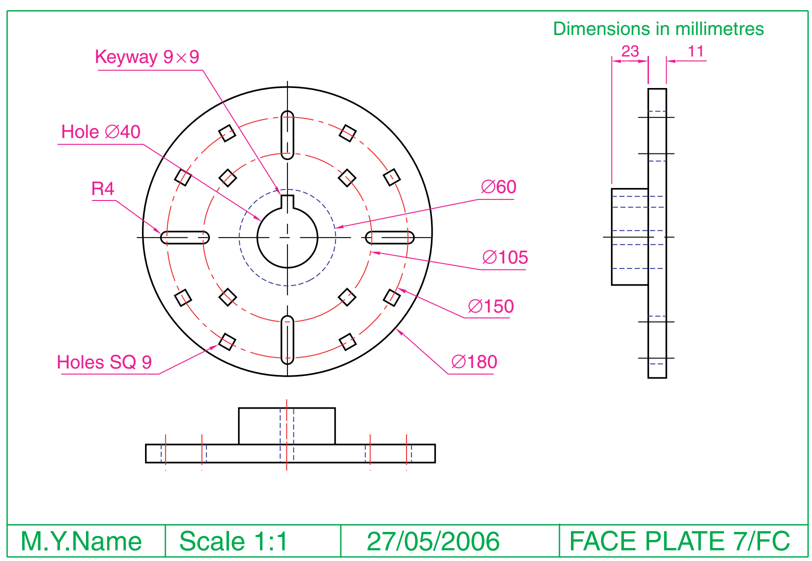


Fig. 14.18 Exercise 5 – dimensions

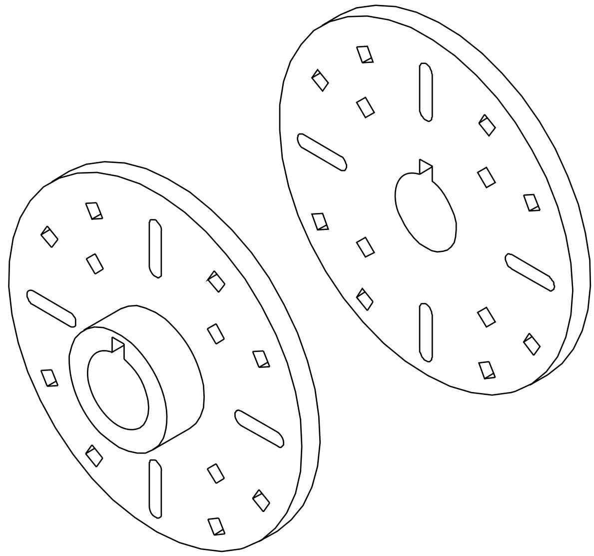


Fig. 14.19 Exercise 5

CHAPTER 15

The modification of 3D models

Aims of this chapter

1. To demonstrate how 3D models can be saved as blocks for insertion into other drawings via the DesignCenter.
2. To show how a library of 3D models in the form of blocks can be constructed to enable the models to be inserted into other drawings.
3. To give examples of the use of the tools from the **Operations** sub-menu from the **Modify** drop-down menu:

   3D Array – Rectangular and Polar 3D arrays
   Mirror 3D
   Rotate 3D.

4. To give examples of the use of the following tools from the **Solids** toolbar:

   Slice
   Section.

5. To give examples of the use of the **Helix** tool.
6. To give a further example of construction involving the **DYN** method.
7. To show how to obtain different views of 3D models in 3D space using:

   3D Views from the **View** drop-down menu
   Viewpoint Presets.

Creating 3D model libraries

In the same way as 2D drawings of parts such as electronics symbols, engineering parts, building symbols and the like can be saved in a file as blocks and then opened into another drawing by *dragging* the appropriate block drawing from the DesignCenter, so can 3D models.

First example – inserting 3D blocks (Fig. 15.4)

1. Construct individual 3D models of the parts for a lathe milling wheel holder to the dimensions given in Fig. 15.1 on layers of different colours.
2. Save each of the 3D models of the parts to file names as given in the drawing in Fig. 15.1 as blocks using the **Make Block** tool from the **Draw** toolbar. When all seven blocks have been saved, the drawings on

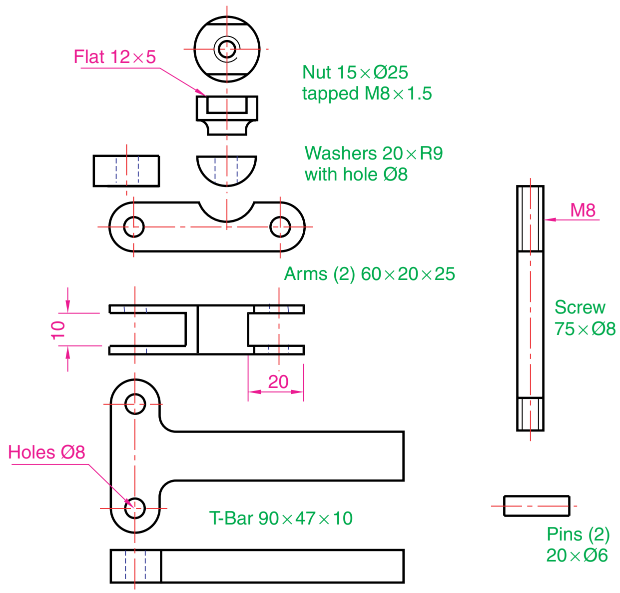


Fig. 15.1 The components of a lathe milling wheel holder

the screen can be deleted. Save the drawing with its blocks to a suitable file name. In this example this is **Fig01.dwg**.

3. Set up a **Four: Equal** viewports setting.
4. Open the **DesignCenter** with a *click* on its name in the **Palettes** sub-menu in the **Tools** drop-down menu (Fig. 15.2), or by pressing the **Ctrl** and **2** keys of the keyboard.
5. In the **DesignCenter** *click* the directory **Chapter 15**, followed by another *click* on **Fig03.dwg** and yet another *click* on **Blocks**. The saved blocks appear as icons in the right-hand area of the **DesignCenter**.

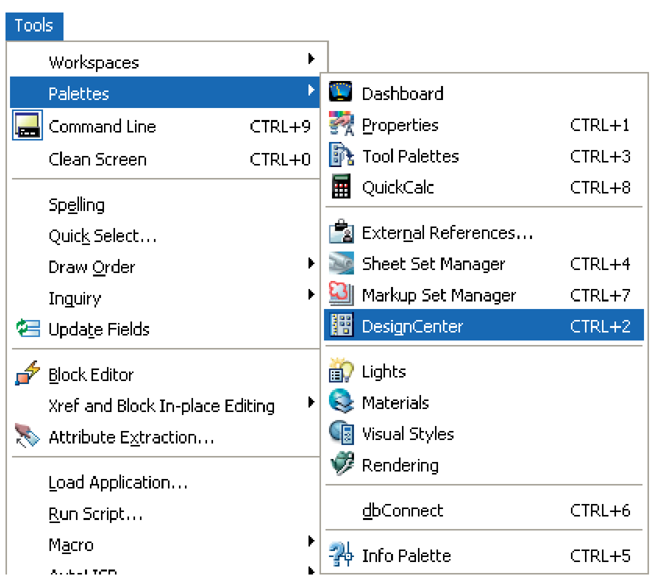


Fig. 15.2 Calling the **DesignCenter** to screen

6. *Drag* and *drop* the blocks one by one into one of the viewports on screen Fig. 15.3 shows the **Nut** block ready to be *dragged* into position in the **Front** viewport. As the blocks are *dropped* on screen, they will need moving into their correct positions in relation to other parts of the assembly by using the **Move** tool from the **Modify** toolbar in suitable viewports.
7. Using the **Move** tool, move the individual 3D models into their final places on screen and render the **SE Isometric** viewport. Shade using **3D orbit/Visual Styles/Realistic** (Fig. 15.4).

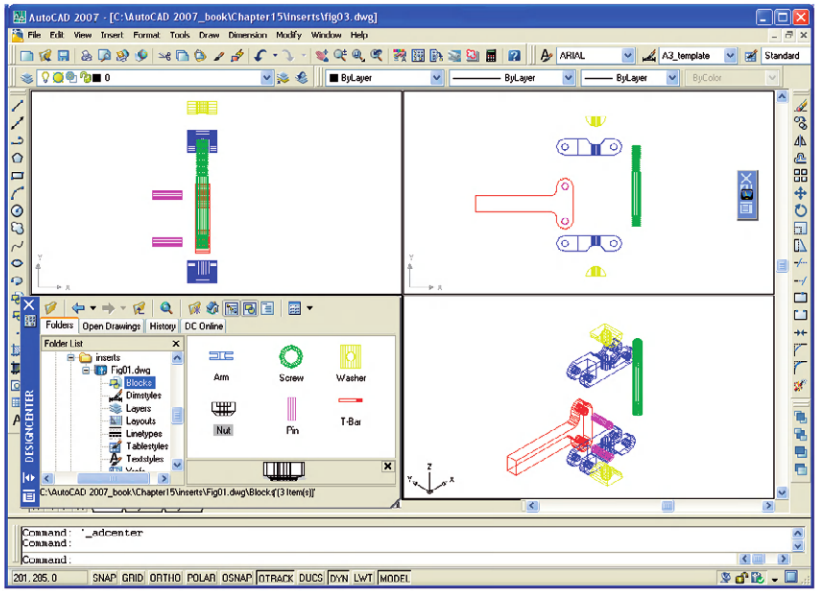


Fig. 15.3 First example – **inserting 3D blocks**

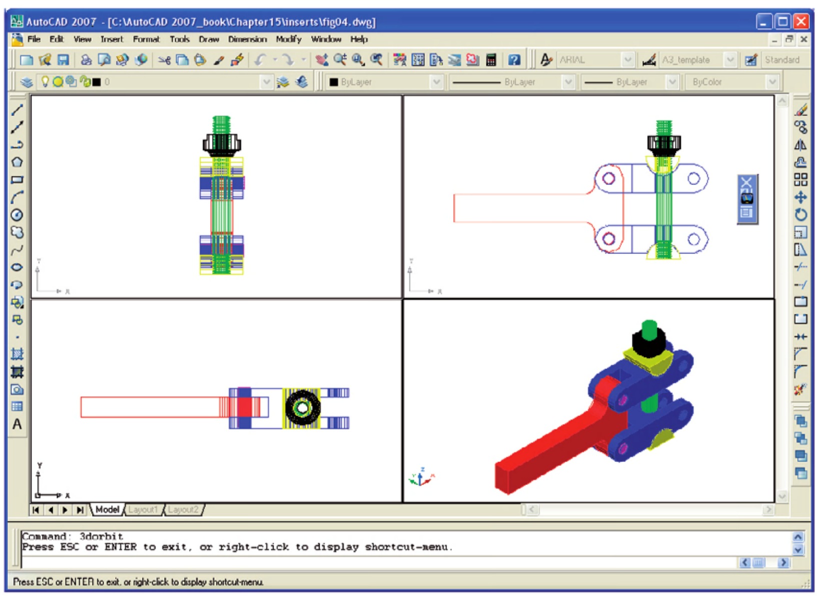


Fig. 15.4 First example – **inserting 3D blocks**

Notes

1. It does not matter which of the four viewports any one of the blocks is *dragged* and *dropped* into – The part automatically assumes the view of the viewport.
2. If a block destined for layer **0** is *dragged* and *dropped* into the layer **Centre** (which in our **acadiso.dwt** is of colour **red** and linetype **CENTER2**), the block will take on the colour (red) and linetype of that layer (CENTER2).
3. In this example, the blocks are 3D models and there is no need to use the **Explode** tool option.

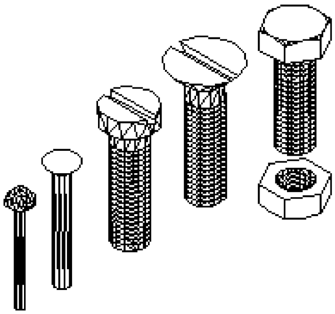


Fig. 15.5 Second example – the five fastenings

Second example – a library of fastenings (Fig. 15.6)

1. Construct a number of engineering fastenings. The number constructed does not matter. In this example only five have been constructed – a 10 mm round head rivet, a 20 mm countersunk head rivet, a cheese head bolt, a countersunk head bolt and a hexagonal head bolt together with its nut (Fig. 15.5). With the **Make Block** tool save each separately as a block, erase the original drawings and save the file to a suitable file name – in this example this is **Fig05.dwg**.
2. Open the DesignCenter, *click* on the **Chapter15** directory, followed by a *click* on **Fig05.dwg**. Then *click* again on **Blocks** in the content list of **Fig05.dwg**. The five 3D models of fastenings appear as icons in the right-hand side of the DesignCenter (Fig. 15.6).
3. Such engineering fastenings can be *dragged* and *dropped* into position in any engineering drawing where the fastenings are to be included.

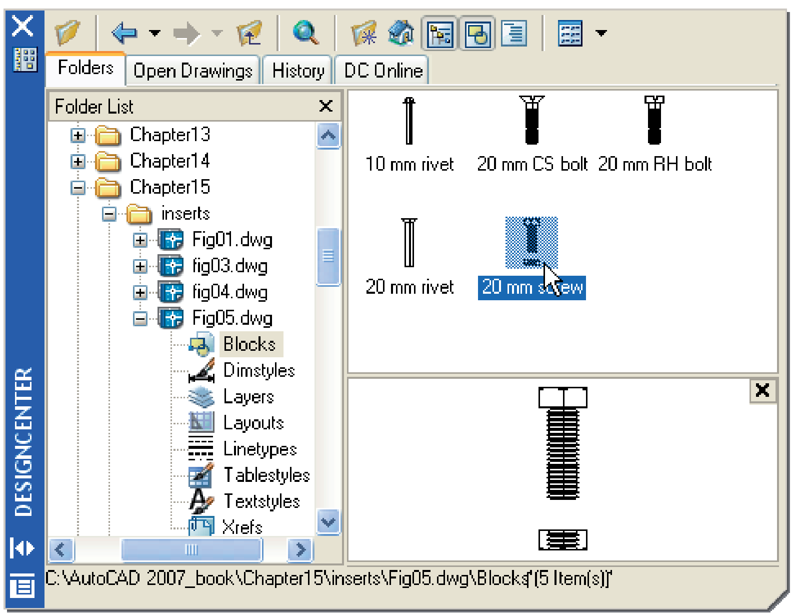


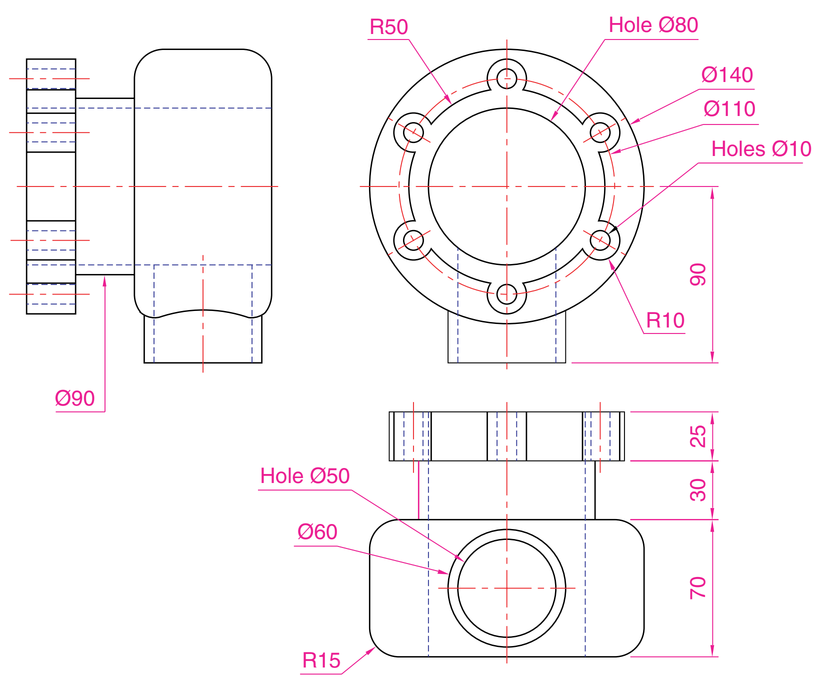
Fig. 15.6 Second example – a library of fastenings

An example of constructing a 3D model (Fig. 15.9)

A three-view projection of a pressure head is shown in Fig. 15.7. To construct a 3D model of the head:

1. From **3D Views** select the **Front** view.
2. Construct the outline on a layer colour magenta to be formed into a solid of revolution (Fig. 15.8) and with the **Revolve** tool, produce the 3D model of the outline.
3. Place the screen in the **3D Views/Top** view and with the **Cylinder** tool, construct cylinders as follows:
   (a) In the centre of the solid already constructed – radius **50** and height **50**.
   (b) With the same centre – radius **40** and height **40**. Subtract this cylinder from that of radius **50**.
   (c) At the correct centre – radius **10** and height **25**.
   (d) At the same centre – radius **5** and height **25**. Subtract this cylinder from that of radius **10**.
4. With the **Array** tool, form a **6** times polar array of the last two cylinders based on the centre of the 3D model.
5. Place the drawing in the **Front** view.
6. With the **Move** tool, move the array and the other two cylinders to their correct positions relative to the solid of revolution so far formed.
7. With the **Union** tool form a union of the array and other two solids.

Fig. 15.7 Orthographic drawing for the example of constructing a 3D model



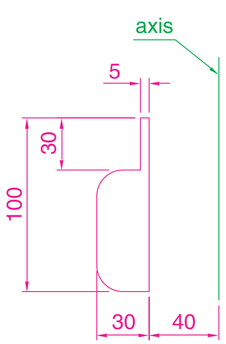


Fig. 15.8 Example of constructing a 3D model – outline for solid of revolution

8. Place the screen in the **3D Views/Right** view.
9. Construct a cylinder of radius **30** and height **25** and another of radius **25** and height **60** central to the lower part of the 3D solid so far formed.
10. Place the screen in the **3D Views/Top** view and with the **Move** tool move the two cylinders into their correct positions relative to the 3D solid.
11. With **Union**, form a union between the radius **30** cylinder and the 3D model and with **Subtract**, subtract the radius **25** cylinder from the **3D** model.
12. Call **3D Orbit** and *click* **Visual Styles/Conceptual** in the *right-click* menu of the tool (Fig. 15.9).

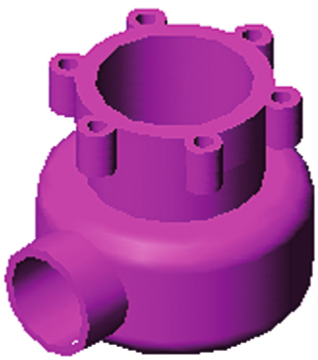


Fig. 15.9 Example of constructing a 3D model

Note

This 3D model could equally as well have been constructed in a three or four viewports setting.

The 3D Array tool

First example – a Rectangular Array (Fig. 15.12)

1. Construct the star-shaped pline on a layer colour green (Fig. 15.10) and extrude it to a height of **20**.
2. *Click* on **Modify** in the menu bar and in the drop-down menu which appears *click* on **3D Operation**, followed by another *click* on **3D Array** in the sub-menu which appears (Fig. 15.11). The command line shows:

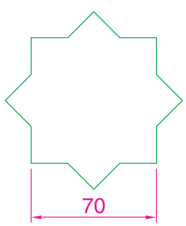


Fig. 15.10 Example – 3D Array – the star pline

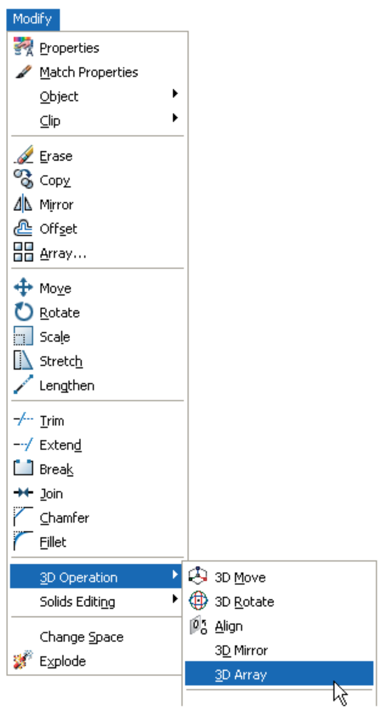


Fig. 15.11 Selecting **3D Array** from the **Modify** drop-down menu

Command:_3darray
Select objects: *pick* the extrusion **1 found**
Select objects: *right-click*
Enter the type of array [Rectangular/Polar] <R>: *right-click*
Enter the number of rows (—) <1>: *enter* **3** *right-click*
Enter the number of columns (III): *enter* **3** *right-click*
Enter the number of levels (. . .): *enter* **4** *right-click*
Specify the distance between rows (—): 100
Specify the distance between columns (III): 100
Specify the distance between levels (. . .): 300
Command:

3. Place the screen in the **3D views/SW Isometric** view.
4. Shade using **3D Orbit/Visual Styles/Realistic** (Fig. 15.12).

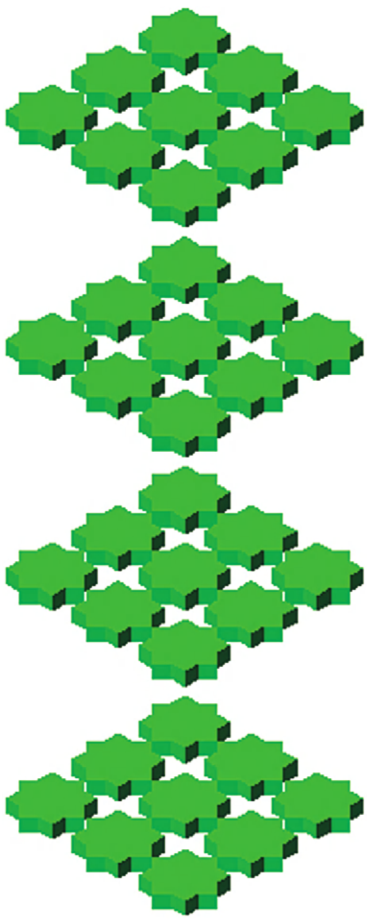


Fig. 15.12 First example – a **Rectangular Array**

Second example – a Polar Array (Fig. 15.13)

1. Use the same star-shaped 3D model.
2. Call the **3D Array** tool again. The command line shows:

Command:_3darray
Select objects: *pick* the extrusion **1 found**
Select objects: *right-click*
Enter the type of array [Rectangular/Polar] <R>: *enter* **p** (Polar) *right-click*
Enter number of items in the array: 12
Specify the angle to fill (+=ccw, −=cw) <360>: *right-click*
Rotate arrayed objects? [Yes/No] <Y>: *right-click*
Specify center point of array: 235,125
Specify second point on axis of rotation: 300,200
Command:

3. Place the screen in the **3D Views/SW Isometric** view.
4. Call **3D Orbit** and *click* **Visual Styles/Conceptual** in the *right-click* menu of the tool (Fig. 15.13).

Third example – a Polar Array (Fig. 15.15)

1. Working on a layer of colour red, construct a solid of revolution in the form of an arrow to the dimensions as shown in Fig. 15.14.
2. Call **3D Array** from the **Modify** drop-down menu. The command line shows:

Command:_3darray
Select objects: *pick* the arrow **1 found**
Select objects: *right-click*
Enter the type of array [Rectangular/Polar] <R>: *enter* **p** *right-click*
Enter the number of items in the array: *enter* **12** *right-click*
Specify the angle to fill (+=ccw, −=cw) <360>: *right-click*
Rotate arrayed objects? [Yes/No] <Y>: *right-click*

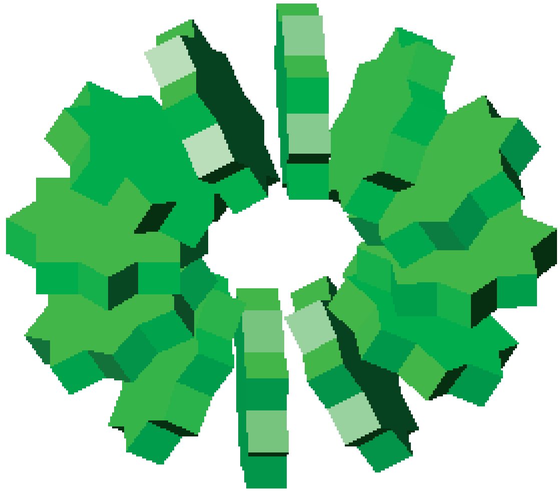


Fig. 15.13 Second example – a **Polar Array**

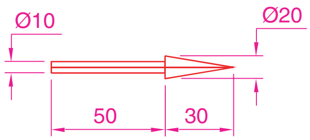


Fig. 15.14 Third example – a **Polar Array** – the 3D model to be arrayed

Specify center point of array: *enter* **40,170,20** *right-click*
Specify second point on axis of rotation: *enter* **60,200,100** *right-click*
Command:

3. Place the array in the **3D Views/SW Isometric** view and shade to **3D Orbit/Visual Styles/Realistic**. The result is shown in Fig. 15.15.

Fig. 15.15 Third example – a **Polar Array**

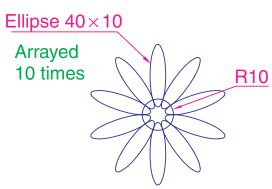


Fig. 15.16 First example – **Mirror 3D** – plan of object to be mirrored

The Mirror 3D tool

First example – Mirror 3D (Fig. 15.18)

1. Working on a layer colour blue, construct the array of ellipses and circle as shown (Fig. 15.16).
2. Change the 11 objects into regions, form a union of the ellipses and subtract the circle from the ellipses.
3. Extrude the region to a height of **5** and render. A **3D Orbit/Conceptual** style shading is shown in Fig. 15.17.
4. *Click* on **Mirror 3D** in the **3D Operation** sub-menu of the **Modify** drop-down menu. The command line shows:

Command:_mirror3d
Select objects: *pick* the extrusion **1 found**
Select objects: *right-click*

Fig. 15.17 First example – **Mirror 3D** the object to be mirrored

Specify first point of mirror plane (3 points): 80,130,100
Specify second point on mirror plane: 170,220,50
Specify third point on mirror plane: 180,160,20
Delete source objects? [Yes/No] <N>: *right-click*
Command:

The result is shown in Fig. 15.18.

Fig. 15.18 First example – **Mirror 3D**

Second example – Mirror 3D (Fig. 15.20)

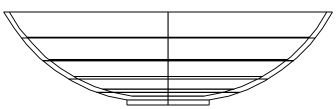


Fig. 15.19 Second example **Mirror 3D** – the 3D model

1. Construct a solid of revolution in the shape of a bowl in the **3D Views/Front** view (Fig. 15.19).
2. *Click* **Mirror 3D** in the **3D Operations** sub-menu of the **Modify** drop-down menu. The command line shows:

   Command:_mirror3d
   Select objects: *pick* the bowl **1 found**
   Select objects: *right-click*
   Specify first point on mirror plane (3 points): *pick*
   Specify second point on mirror plane: *pick*
   Specify third point on mirror plane: *enter* **.xy** *right-click* **(need Z):** *enter* **1** *right-click*
   Delete source objects? [Yes/No] <N>: *right-click*
   Command:

Note

The line in the illustration shows the top edge of an imaginary plane vertical to the plane on which the drawing is being constructed.

3. Place in the **3D Views/SW Isometric** view.
4. Call **3D Orbit** and *click* **Visual Styles/Conceptual** in the *right-click* menu of the tool. The result is shown in Fig. 15.20.

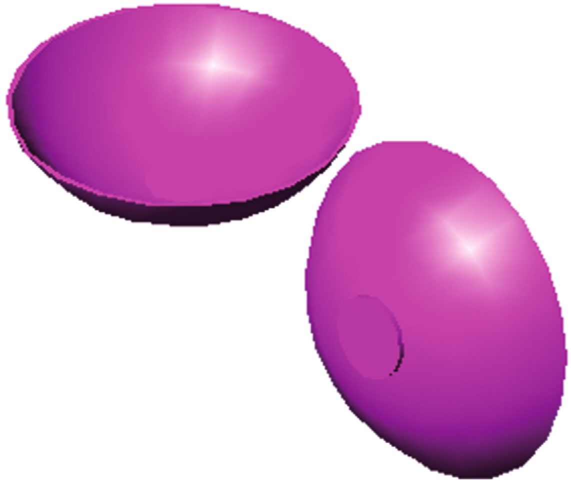


Fig. 15.20 Second example – **Mirror 3D** – the result in a front view

The Rotate 3D tool

Example – Rotate 3D (Fig. 15.21)

1. Use the same 3D model of a bowl as for the last example. Call the **Rotate 3D** tool from the **3D Operations** sub-menu of the **Modify** drop-down menu.
2. The command line shows:

Command:_3DROTATE
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0
Select objects: *pick* the bowl **1 found**
Select objects: *right-click*
Specify base point: *pick* the centre bottom of the bowl
Specify rotation angle or [Copy/Reference] <0>: *enter* **60** *right-click*
Command:

3. Place in the **SW Isometric** view and call **3D Orbit** and *click* **Visual Styles/Conceptual** in the *right-click* menu of the tool.

The result is shown in Fig. 15.21.

Fig. 15.21 Example – **Rotate 3D**

The Slice tool

First example – Slice (Fig. 15.25)

1. Construct a 3D model of the rod link device shown in the two-view projection (Fig. 15.22).
2. Place the 3D model in the **3D Views/Top** view.
3. Call the **Slice** tool from the **Modify/3D Operation** sub-menu (Fig. 15.23). The command line shows (Fig. 15.24 shows the *picked* points):

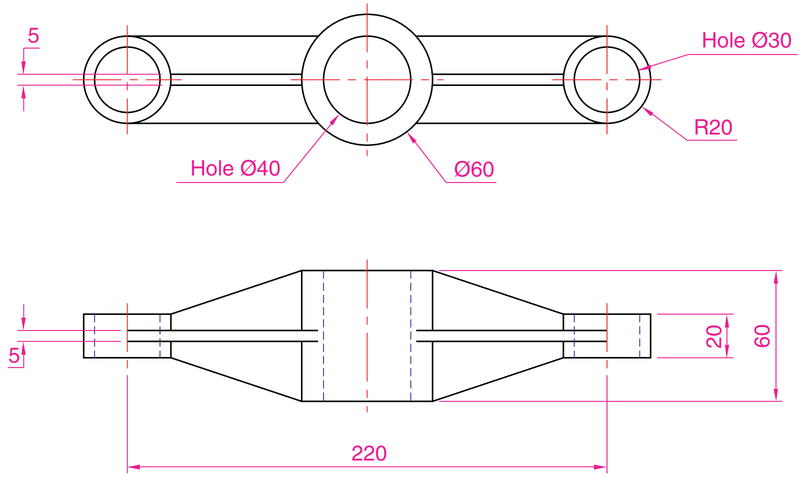


Fig. 15.22 First example – **Slice** – the two-view drawing

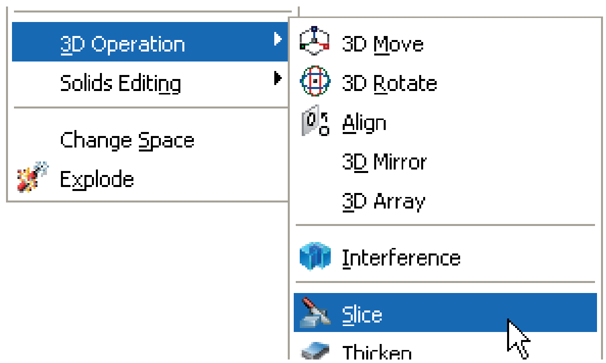


Fig. 15.23 The **Slice** tool icon from the **Modify** toolbar

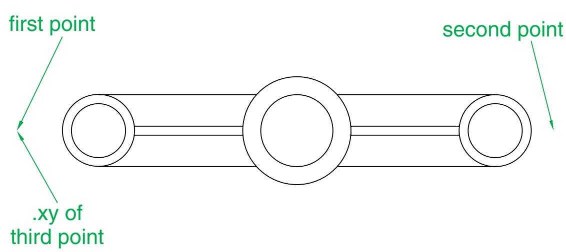


Fig. 15.24 First example – **Slice** – the *pick* points

Command:_slice
Select objects: *pick* the 3D model
Select objects: *right-click*
Specify first point on slicing plane <3points>: *pick*
Specify second point on slicing plane <3points>: *pick*
Specify third point on slicing plane <3points>: *enter* **.xy** *right-click*
of *pick* first point again **(need Z):** *enter* **1** *right-click*
Specify a point on desired side of the plane or [keep Both sides]: *enter* **b** (Both) *right-click*
Command:

Fig. 15.25 First example – **Slice**

4. With the **Move** tool, move the lower half of the sliced model away from the upper half.
5. Place the 3D model(s) in a **SW Isometric** view.
6. Call **3D Orbit** and *click* **Visual Styles/Conceptual** in the *right-click* menu of the tool (Fig. 15.25).

Second example – Slice (Fig. 15.26)

1. Construct the closed pline (Fig. 15.26) and with the **Revolve** tool, form a solid of revolution from the pline.

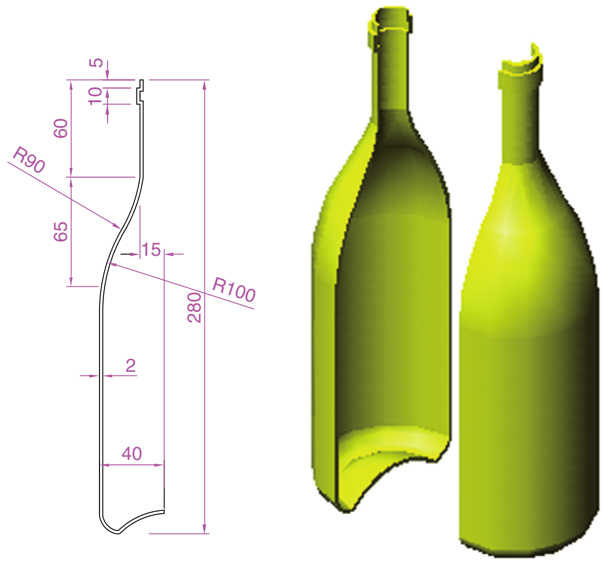


Fig. 15.26 Second example – **Slice**

2. With the **Slice** tool and working to the same sequence as for the first **Slice** example, form two halves of the 3D model and render.

The Section tool

First example – Section (Fig. 15.29)

1. Construct a 3D model to the information given in Fig. 15.27 on layer **0**. Note there are three objects in the model – a box, a lid and a cap.
2. Place the model in the **3D Views/Top** view.
3. *Click* in the layer field of the **Layers** toolbar (Fig. 15.28) and *click* again on **Construction** to make it the current layer. Its colour is cyan.
4. *Click* the **Section** tool icon in the **3D Make control panel**. The command line shows:

Command:_section
Select objects: *window* the 3D model **3 found**
Select objects: *right-click*

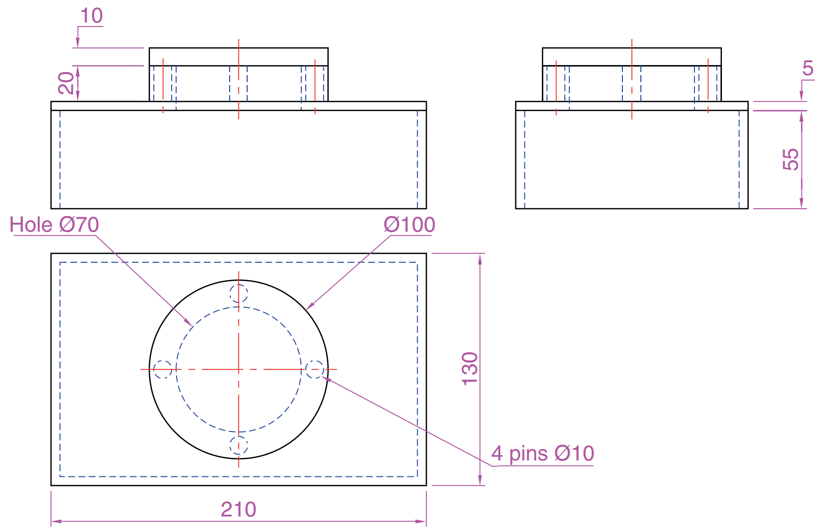


Fig. 15.27 First example – **Section** – orthographic projection

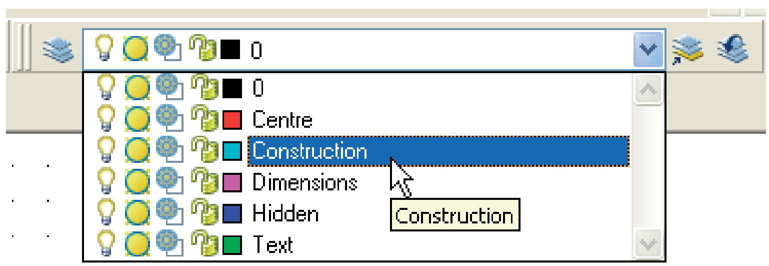


Fig. 15.28 Making layer **Construction** current

Specify first point on Section plane <3points>: *pick*
Specify second point on plane: *pick*
Specify third point on plane: *enter* **.xy** *right-click*
of *pick* first point **(need Z):** *enter* **1** *right-click*
Command:

And a cyan line showing the top edge of the section plane appears in the view.

Note

The three points *picked* above are similar to those given in the previous examples of using the **Slice** tool.

5. Turn **Layer 0** (on which the 3D model was constructed) off, leaving only the cyan line showing on screen.
6. Place the screen in the **3D Views/Front** view and **Zoom** to **1**. The outlines of the section appear.

Note

The sectional view as given in the outlines (Fig. 15.29) will not be a correct view as required in general engineering drawing practice for the following reasons:

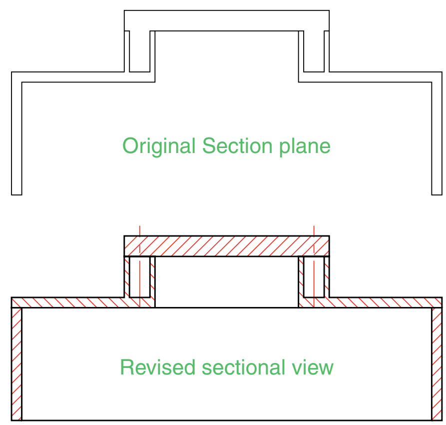


Fig. 15.29 First example – **Section**

(a) There are no hatch lines. This is usually acceptable because in some circumstances hatching of sections is not expected, but in this example hatch lines are to be included.
(b) In engineering sectional views, parts such as the pins holding the cap onto the lid would usually be shown by outside views in a sectional view.

7. Amend the drawing by adding lines as necessary and hatching using the **ANSI31** hatch pattern as shown in the lower drawing of Fig. 15.29.

Second example – Section (Fig. 15.31)

1. Open the drawing of the lathe tool holder constructed in answer to the first example in this chapter (Fig. 15.4). The drawing is in a **Four: Equal** viewports setting. *Click* in the **Top** viewport and from the **View** drop-down menu *click* **1 Viewport** in the **Viewports** sub-menu (Fig. 15.30). The assembly appears in a full size single viewport.
2. Make the layer **Construction** current.
3. At the command line:

Command: *enter* section

Then proceed as in the **First example – section**.
4. Turn off all layers other than **Construction**. The resulting section is shown in Fig. 15.31.

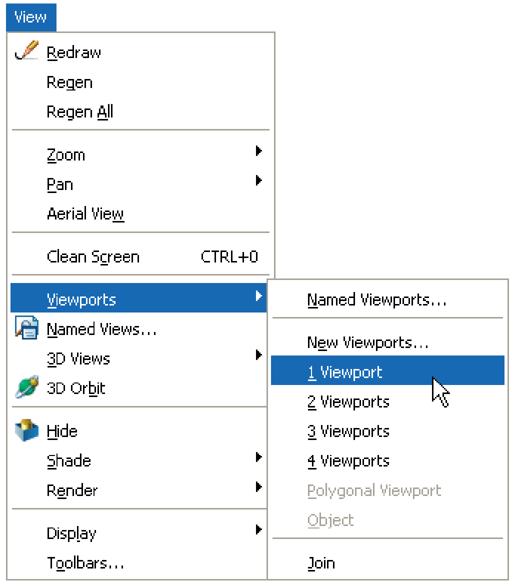


Fig. 15.30 Calling **1 Viewport** from the **View** drop-down menu

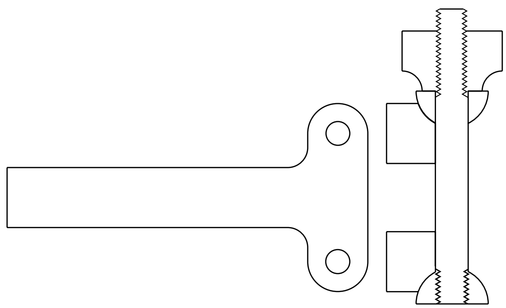


Fig. 15.31 Second example – **Section**

Views of 3D models

Fig. 15.32 is a two-view projection of a model of an arrow.

Some of the possible viewing positions of a 3D model which can be obtained by using the **3D Views** positions have already been seen in this book. The views in Fig. 15.33 show all of the viewing positions of the 3D model of the arrow using the viewing positions from the **3D Views** sub-menu of the **Views** drop-down menu.

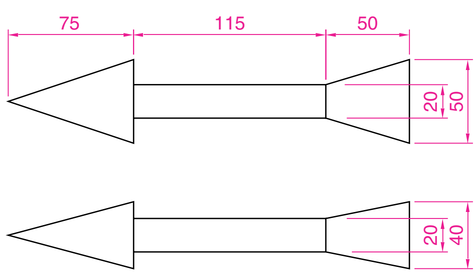


Fig. 15.32 Two views of the arrow

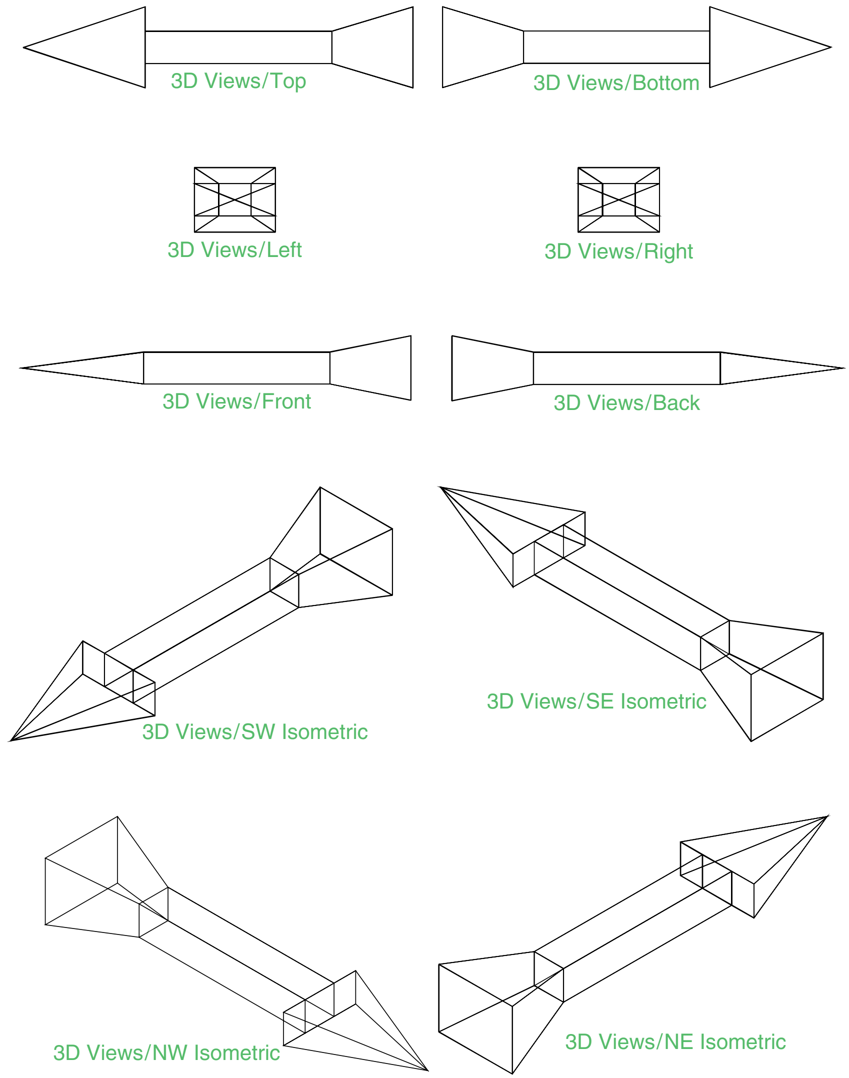


Fig. 15.33 The views from **3D Views**

The Viewpoint Presets dialog

There are other methods of obtaining a variety of viewing positions of a 3D model. One method – using the **UCS** (User Coordinate System) – will be described in a later chapter. Another method is by using the **Viewpoint Presets** dialog called with a *click* on **Viewpoint Presets** . . . in the **3D Views** sub-menu of the **View** drop-down menu (Fig. 15.34).

When the dialog appears with a 3D model on screen, *entering* figures for degrees in the **X Axis** and **XY Plane** fields, followed by a *click* on the dialog's **OK** button causes the model to take up the viewing position indicated by these two angles.

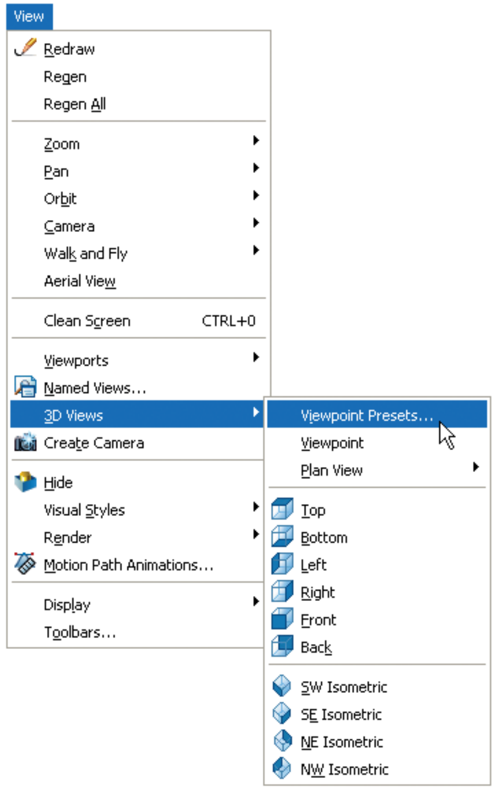


Fig. 15.34 Calling the **Viewpoint Presets** dialog

Note

The **Relative to UCS** radio button must be checked to allow the 3D model to position along the two angles.

Example – Viewpoint Presets

1. With the 3D model of the arrow on screen, *click* **Viewpoint Presets . . .** in the **3D Views** sub-menu of the **Views** drop-down menu. The dialog appears.
2. *Enter* **330.0** in the **From X Axis** and −30.0 in the **From XY Plane** fields and *click* the **OK** button of the dialog.
3. The 3D model takes up the viewing position indicated by the two angles (Fig. 15.35).

The Helix tool

The **Helix** tool can be called with a *click* on its tool icon in the extension of the **3D Make control panel** (Fig. 15.36).

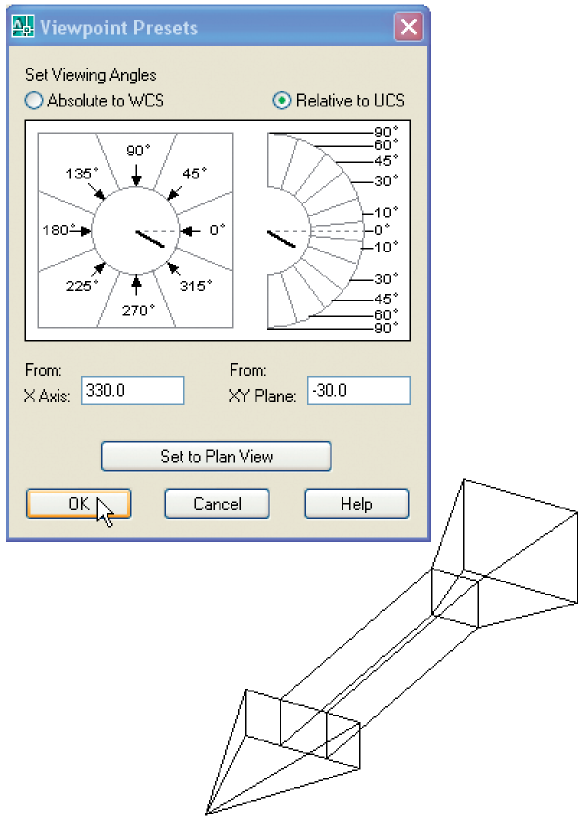


Fig. 15.35 Example – **Viewpoint Presets**

Fig. 15.36 The extension of the **3D Make control panel**

First example – Helix (Fig. 15.39)

1. Construct the triangular outline shown in Fig. 15.37 using the **Polyline** tool. Make sure the pline outline is placed at right angles to the bottom end of the helix as shown in Fig. 15.37. This may mean moving and rotating the outline in a selection of the **3D Views**.
2. Call the **Helix** tool from the **3D Make control panel** (Fig. 15.38) or from the **Modeling** toolbar. The command line shows:

Command:_Helix
Number of turns = 3 Twist=CCW
Specify center point of base: *enter* **160, 160** *right-click*
Specify base radius or [Diameter] <1>: *pick* **160, 200**
Specify top radius or [Diameter] <40>: *right-click*
Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>: *enter* **t** (Turns) *right-click*
Enter number of turns <3>: *enter* **10** *right-click*
Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>: *enter* **100** *right-click*
Command:

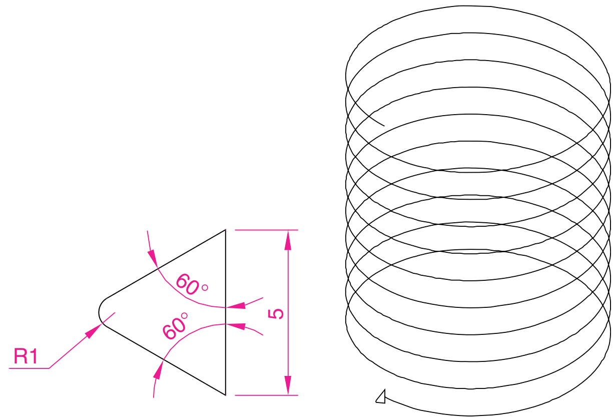


Fig. 15.37 First example **Helix** – the polyline outline and the helix

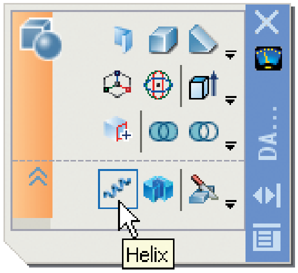


Fig. 15.38 The **Helix** tool icon in the extension of the **3D Make control panel**

3. Call the **Extrude** tool from the **3D Make control panel** and extrude the outline along the path of the helix. The command line shows:

   Command:_extrude
   Current wire frame density: ISOLINES=4
   Select objects to extrude: *pick* the outline **1 found**
   Select objects to extrude: *right-click*
   Specify height of extrusion of [Direction/Path/Taper angle]: *enter* **p** (path) *right-click*
   Select extrusion path or [Taper angle]: *pick* the helix
   Command:

   The result is shown in Fig. 15.39.

4. Add three cylinders, one to fit inside the helix, the second to form the shank of the screw, the third for the head of the screw. Subtract a box from the head for the screw slot. Then union the four parts of the screw.
5. Call **3D Orbit** and shade the screw using the **Visual Style/Conceptual** form of shading.

The result is shown in Fig. 15.40.

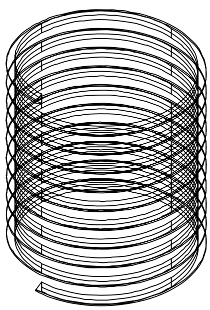


Fig. 15.39 First example **Helix** – the resulting helix

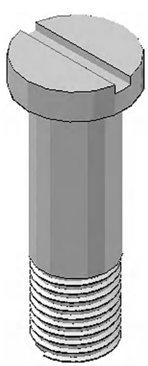


Fig. 15.40 First example – **Helix**

Second example – Helix (Fig. 15.41)

Fig. 15.41 shows a 3D hidden view model of a helix formed from a circle of **5** units radius extruded along a helical path of **6** turns and a radius of **40** and height of **100**.

Using DYN

As with all other tools (commands) in AutoCAD 2007, a helix can be formed working with the **DYN** (Dynamic Input) system. Fig. 15.42 shows the stages (**1** to **5**) in the construction of the helix in the second example.

Set **DYN** on with a *click* on its button in the status bar.

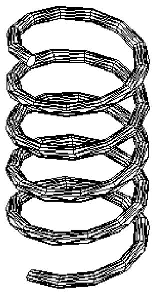


Fig. 15.41 Second example – **Helix**

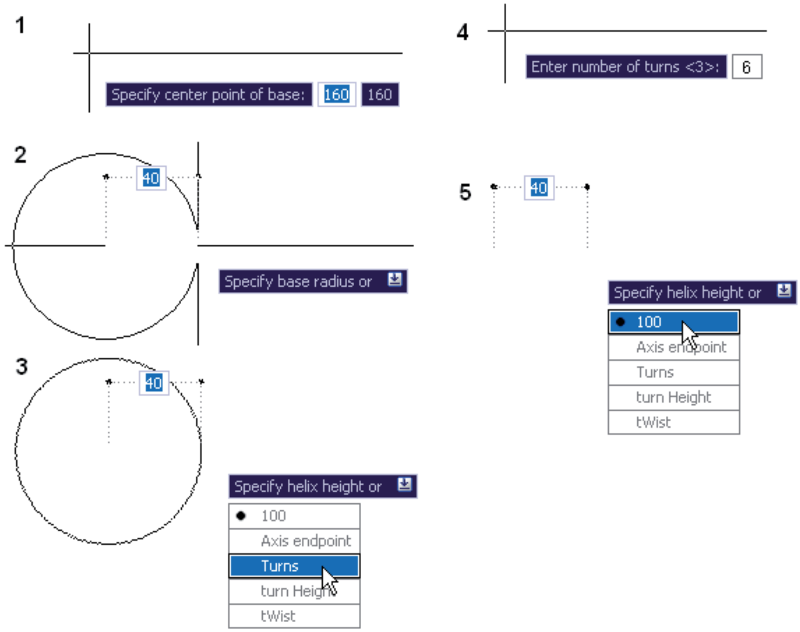


Fig. 15.42 Constructing the helix for the second example with the aid of **DYN**

1. *Click* the **Helix** tool icon in the **3D Make control panel** or the **Modeling** toolbar. The first of the prompts in **DYN** form appears. *Enter* **160,160** at the command line or *drag* the cursor until **160,160** appears in the **DYN** tip and *right-click*.
2. Move the cursor until the dimension **40** shows and *right-click*.
3. Press the down arrow key of the keyboard and *click* **Turns** in the menu which appears.
4. *Enter* **6** and *right-click*.
5. Press the down arrow key of the keyboard and *enter* **100** in the menu as shown.

Revision notes

1. 3D models can be saved as blocks in a similar manner to the method of saving 2D drawings as blocks.
2. Libraries can be made up from 3D model drawings.
3. 3D models saved as blocks can be inserted into other drawings via the DesignCenter.
4. Arrays of 3D model drawings can be constructed in 3D space using the **3D Array** tool.
5. 3D models can be mirrored in 3D space using the **Mirror 3D** tool.
6. 3D models can be rotated in 3D space using the **Rotate 3D** tool.
7. 3D models can be cut into parts with the **Slice** tool.
8. Sectional views can be obtained from 3D model using the **Section** tool.
9. Helices can be constructed using the **Helix** tool. The helices so formed can be used as paths for extruding outlines.

10. Both **3D Views** viewing positions and **Viewpoint Presets** can be used for the placing of 3D models in different viewing positions in 3D space.
11. The **DYN** (Dynamic Input) method of construction can be used equally as well when constructing 3D model drawings as when constructing 2D drawings.

Exercises

1. Fig. 15.43 shows a shaded view of the 3D model for this exercise. Fig. 15.44 is a three-view projection of the model. Working to the details given in Fig. 15.44, construct the 3D model.

Fig. 15.43 Exercise 1 – a shaded view

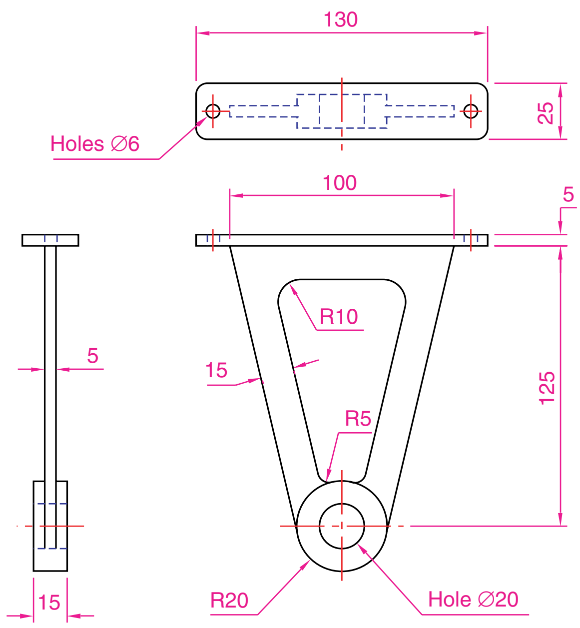


Fig. 15.44 Exercise 1 – three-view projection

2. Construct a 3D model drawing of the separating link shown in the two-view projection (Fig. 15.45). With the **Slice** tool, slice the model into two parts and remove the rear part. Place the front half in a suitable isometric view from the **3D Views** sub-menu. Shade the resulting model.
3. Working to the dimensions given in the two orthographic projections (Fig. 15.47), construct an assembled 3D model of the one part inside the other.

   With the **Slice** tool, slice the resulting 3D model into two equal parts, place in an isometric view and call the **Hide** tool as indicated in Fig. 15.46.

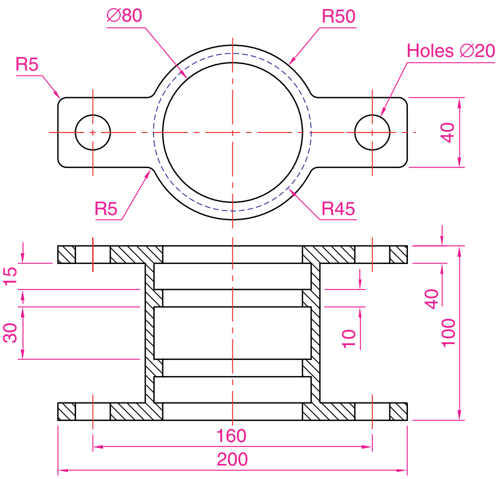


Fig. 15.45 Exercise 2

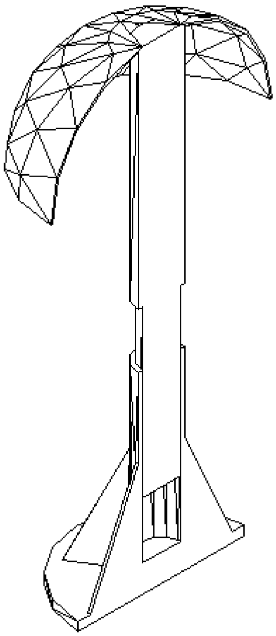


Fig. 15.46 Exercise 3

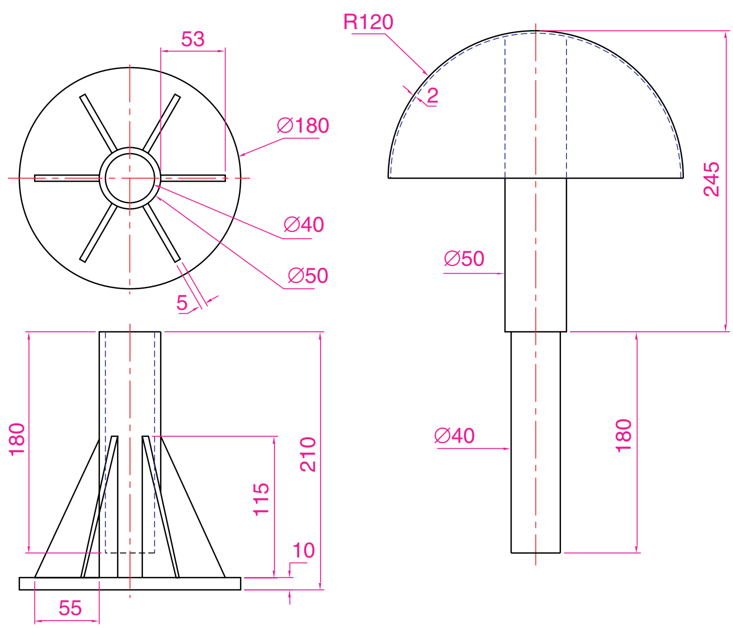


Fig. 15.47 Exercise 3 – orthographic projections

4. Construct a solid of revolution of the jug shown in the orthographic projection (Fig. 15.48). Construct a handle from an extrusion of a circle along a semicircular path. Union the two parts. Place the 3D model in a suitable isometric view and render.

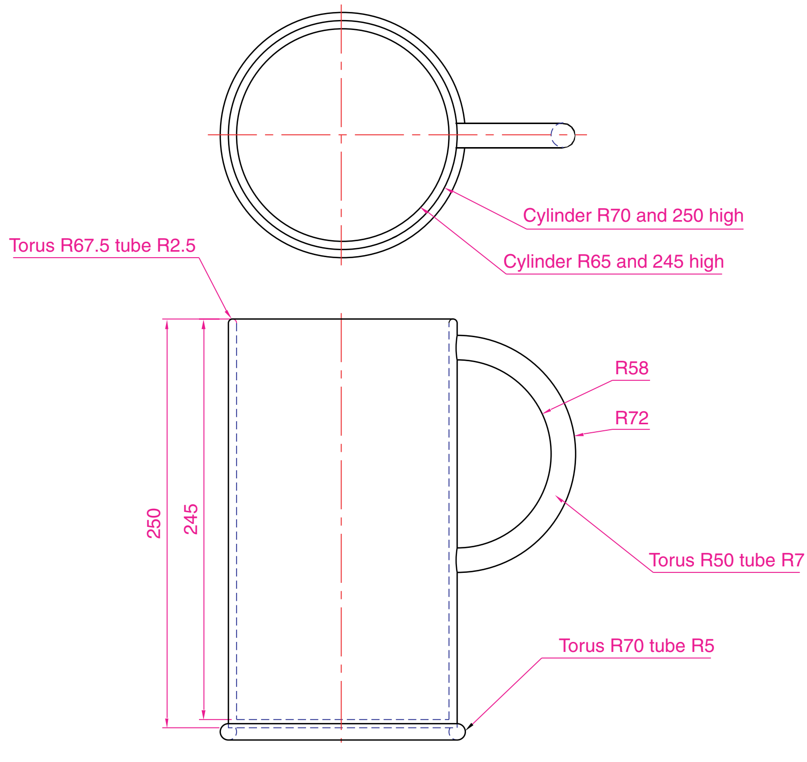


Fig. 15.48 Exercise 4

CHAPTER 16

Rendering

Aims of this chapter

1. To construct a template for 3D Modelling to be used as the drawing base for further work in 3D in this book.
2. To introduce the use of the **Render** tools in producing photographic like images of 3D solid models.
3. To show how to illuminate a 3D solid model to obtain good lighting effects when rendering.
4. To give examples of the rendering of 3D solid models.
5. To introduce the idea of adding materials to 3D solid models in order to obtain a realistic appearance to a rendering.
6. To demonstrate the use of the forms of shading available while using the **3D Orbit** tool.
7. To demonstrate methods of printing rendered 3D solid models.

Setting up a new 3D template

So far in the earlier chapters of this book, we have been constructing both 2D and 3D drawings in the **acadiso.dwt** template. Now we will be constructing 3D model drawings in the **acadiso3D.dwt** template. To prepare this template for the remaining drawings in this book:

1. *Click* **New** . . . in the **File** drop-down menu, followed by a *click* on **acadiso3D** in the file list (Fig. 16.1).
2. When the template appears on screen, ensure the following **control panels** are showing in the **DASHBOARD – 3D Make, 3D Navigate** and **Visual Styles**.
3. *Click* the arrow to the right of the **Visual Styles** field in the **Visual Styles control panel** and select **3D Wireframe** from the icons which appear in a popup (Fig. 16.2).
4. Open the **Options** dialog with a *right-click* in the command window. *Click* the **Display** tab followed by a *click* on the **Colors** . . . button and then set **all** the background colours to **White** (Fig. 16.3).
5. The AutoCAD window should appear as in Fig. 16.4.
6. In the **Options** dialog *click* the **Files** label and *click* **Default Template File Name for QNEW** (Fig. 16.5), followed by a *click* on the **Browse** . . . button which brings up the **Select template** dialog, from which the **acadiso3D.dwt** can be selected. Now when

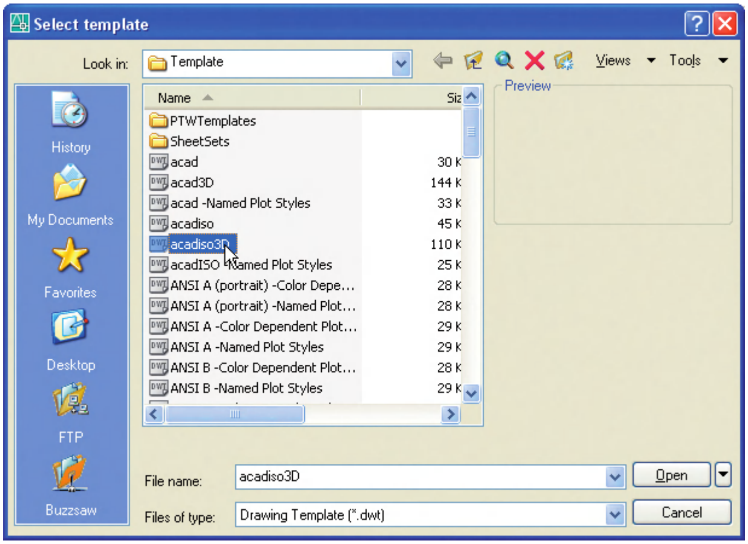


Fig. 16.1 Selecting **acadiso3D** from the **Select template** dialog

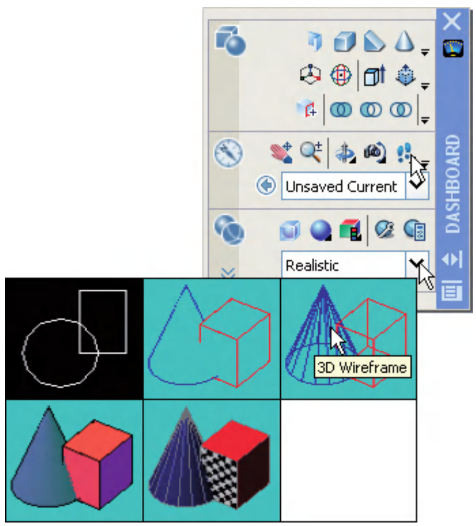


Fig. 16.2 Selecting **3D Wireframe** from **Visual Styles**

AutoCAD 2007 is opened from the desktop, the **acadiso3D.dwt** template will open.

7. Set **Snap** to **5, Grid** to **10** and **Units** to no figures after the decimal point.
8. Set up five layers of different colours. In the author's template these have been named after the colours.
9. Save the template to the name **acadiso3D** and then *enter* a suitable name in the **Template Definition** dialog.
10. Note (Fig. 16.4) that the screen is in **Perspective** projection. In later chapters it will often be set to **Parallel** projection.

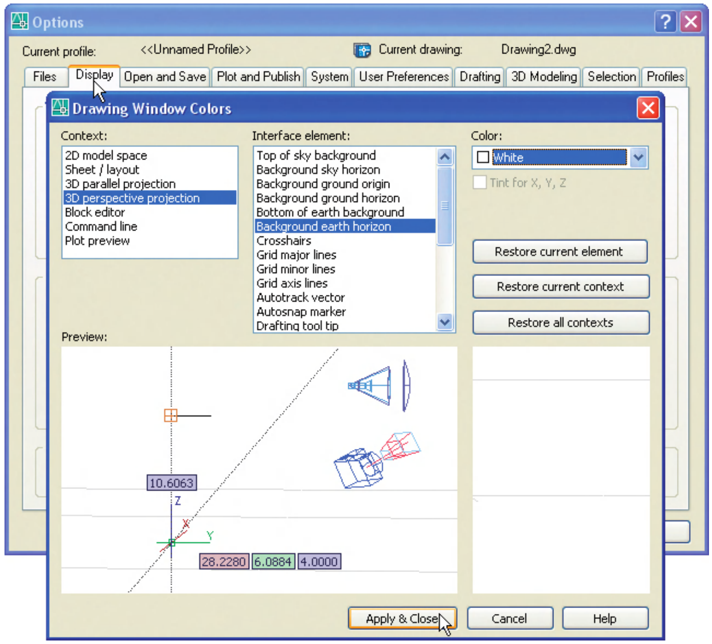


Fig. 16.3 Set **all** background colours to **White**

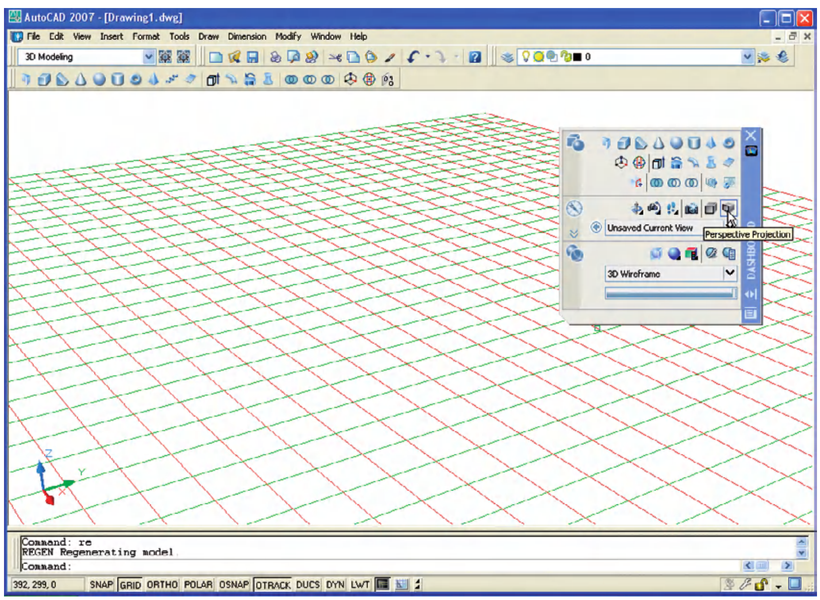


Fig. 16.4 The **acadiso3D** screen

The Render tools

The tools in the **Render** toolbar are shown in Fig. 16.6. Note that a *click* on the arrow below the **Lights** tool icon brings down a flyout showing the light tools available for lighting a 3D model (Fig. 16.7).

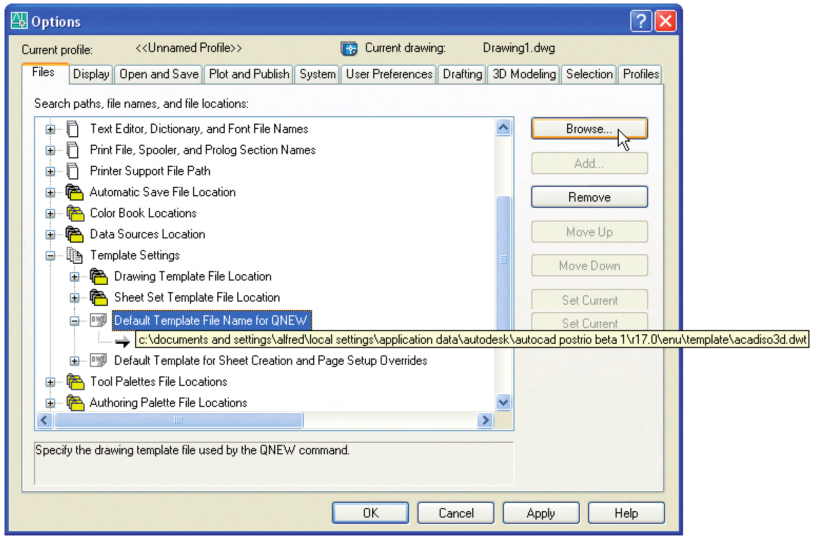


Fig. 16.5 Setting the default window in the **Options** dialog

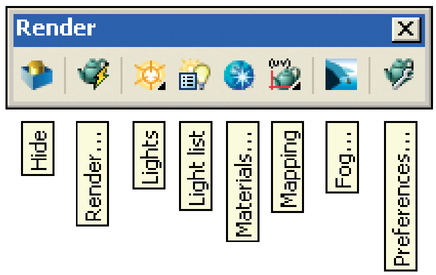


Fig. 16.6 The tools in the **Render** toolbar

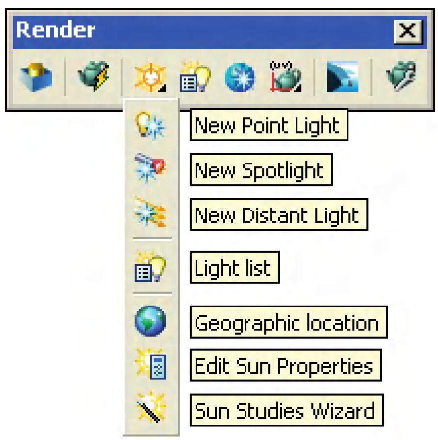


Fig. 16.7 The **Light** tools in the flyout from the **Lights** tool icon in the **Render** toolbar

Render lights

There are four types of lights available when using AutoCAD 2007 – **Ambient** lights, **Point** lights, **Distant** lights and **Spotlights**.

1. **Ambient** lighting is taken as the general overall light that is all around and surrounding any object.
2. **Point** lights shed light in all directions from the position in which the light is placed.
3. **Distant** lights send parallel rays of light from their position in the direction chosen by the operator.
4. **Spotlights** illuminate as if from a spotlight. The light is in a direction set by the operator and is in the form of a cone, with a "hotspot" cone giving a brighter spot on the model being lit.

Placing lights to illuminate a 3D model

Any number of the three types of lights – **Point, Distant** and **Spotlight** – can be positioned in 3D space as wished by the operator.

In general reasonably good lighting effects can be obtained by placing a **Point** light high above the object(s) being illuminated, with a **Distant** light placed pointing towards the object at a distance from the front and above the general height of the object(s) and with a second **Distant** light pointing towards the object(s) from one side and not as high as the first **Distant** light. If desired, **Spotlights** can be used either on their own or in conjunction with the other two forms of lighting.

Setting rendering background colour

The default background colour for rendering in AutoCAD 2007 is usually black. In this book, all renderings are shown on a white background in the viewport in which the 3D model drawing was constructed. To set the background to white for renderings:

1. At the command line:

   Command: *enter* **view** *right-click*

   The **View Manager** dialog appears (Fig. 16.8). *Click* **Model Views** in the **Views** list, followed by a *click* on the **New** . . . button.
2. The **New View** dialog (Fig. 16.9) appears. *Enter* **current** in the **View name** field and **model** in the **View category** field. Then *click* the **Override default background** check box. The **Background** dialog appears (Fig. 16.10).
3. In the **Background** dialog *click* in the **Color** field. The **Select Color** dialog appears (Fig. 16.11).
4. In the **Select Color** dialog *drag* the **Red, Green** and **Blue** sliders as far as possible to the right to change the colour to white **(255,255,255)**. Then *click* the dialog's **OK** button. The **Background** dialog reappears showing white in the **Color** field (Fig. 16.12). *Click* the dialog's **OK** button.
5. The **New View** dialog reappears showing the background as white. *Click* the dialog's **OK** button (Fig. 16.13). The **View Manager** dialog reappears. *Click* its **OK** button.
6. In the **Render control panel** of the **DASHBOARD** *click* the **Advanced Render Settings** . . . icon (Fig. 16.14). The **ADVANCED RENDER SETTINGS** palette appears (Fig. 16.15).

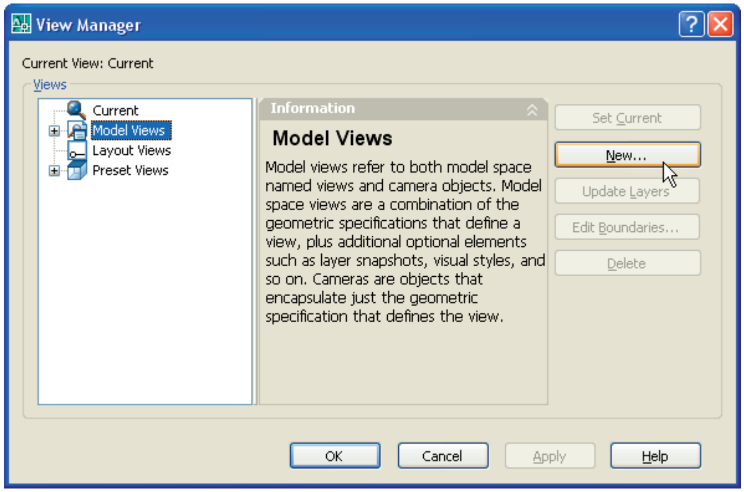


Fig. 16.8 The **View Manager** dialog

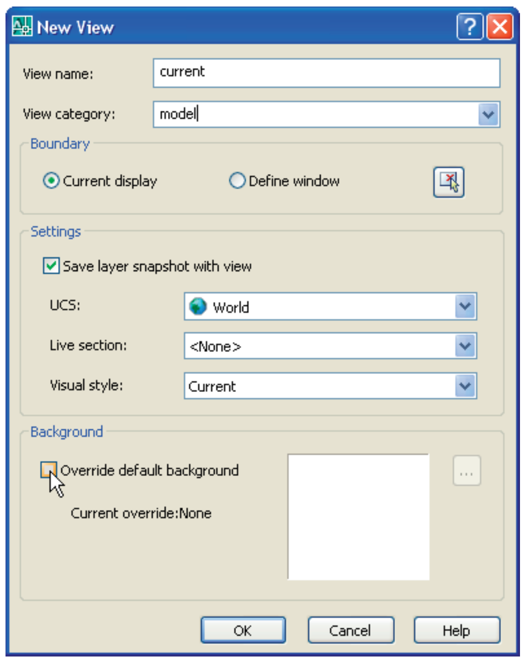


Fig. 16.9 The **New View** dialog

7. In the palette *click* in the **Destination** field and select **Viewport** as the rendering destination (Fig. 16.15).
8. Close the palette and save the screen with the new settings as the template **3dacadiso.dwt**. This will ensure renderings are made in the viewport in which the 3D model was constructed – on a white background.

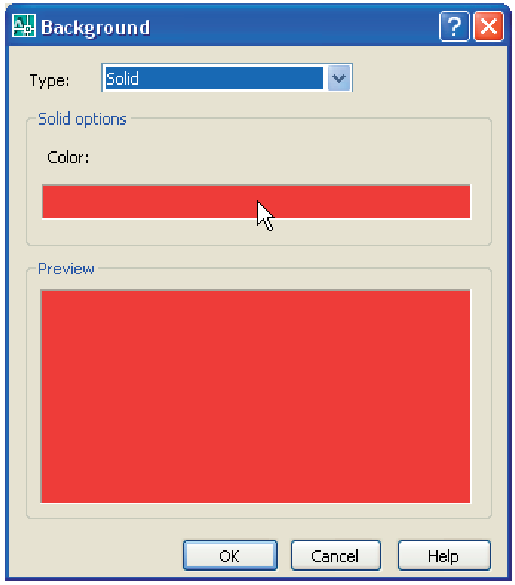


Fig. 16.10 The **Background** dialog

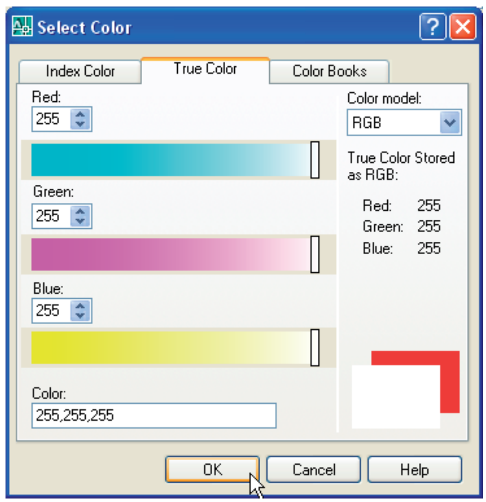


Fig. 16.11 The **Select Color** dialog

First example – rendering a 3D model (Fig. 16.21)

1. Construct a 3D model of the wing nut shown in the two-view projection in Fig. 16.16.
2. Place the 3D model in the **3D Views/Top** view, **Zoom** to **1** and with the **Move** tool, move the model to the upper part of the AutoCAD drawing area.
3. *Click* the **New Point Light** tool icon in the **Lights** flyout from the **Render** toolbar (Fig. 16.7). The **Viewport Lighting Mode** warning window appears (Fig. 16.17). *Click* its **Yes** button.
4. A **New Point Light** icon appears and the command line shows:

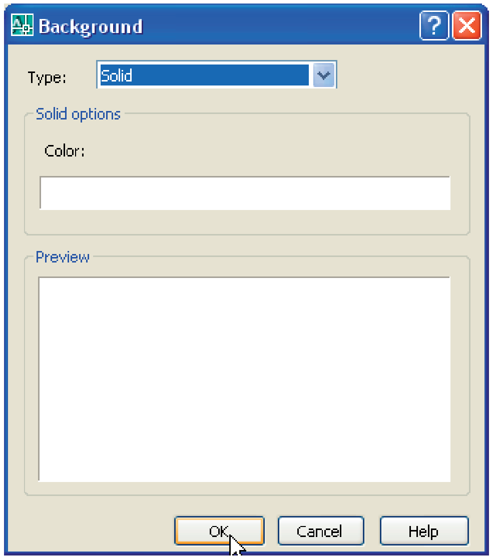


Fig. 16.12 The **Background** dialog

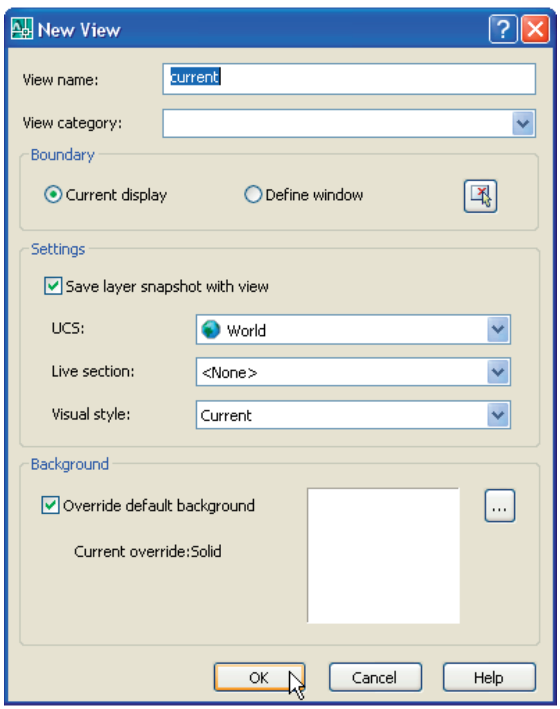


Fig. 16.13 The **New View** dialog

Command:_pointlight
Specify source location <0,0,0>: *enter* **.xy** *right-click*
of *click* at centre of model **(need Z):** *enter* **500** *right-click*
Enter an option to change [Name/Intensity/Status/shadoW/Attenuation/Color/eXit] <eXit>: *enter* **n** (Name) *right-click*
Enter light name <Pointlight1>: *right-click*

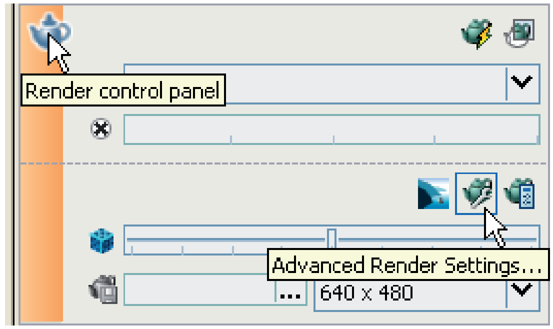


Fig. 16.14 The **Advanced Render Settings** . . . icon in the **Render control panel**

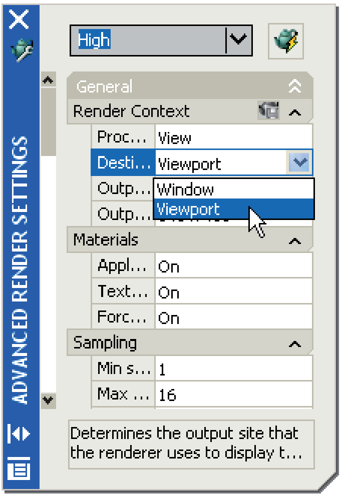


Fig. 16.15 The **ADVANCED RENDER SETTINGS** palette

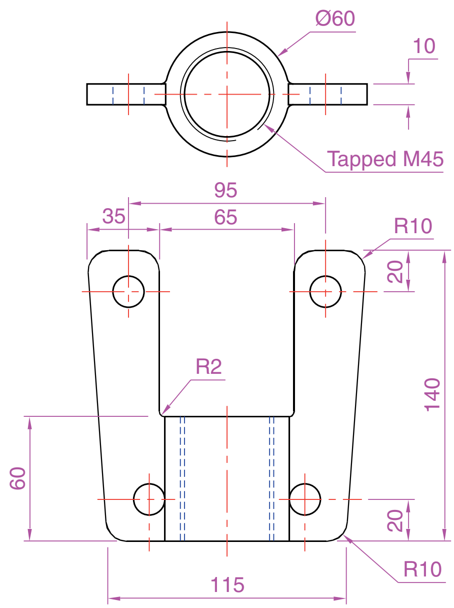


Fig. 16.16 First example – **Rendering** – two-view projection

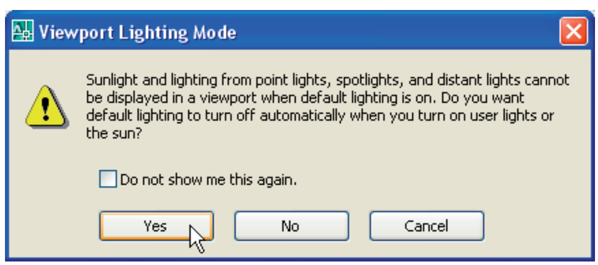


Fig. 16.17 The **Viewport Lighting Mode** warning window

Enter an option to change [Name/Intensity/Status/shadoW/Attenuation/Color/eXit] <eXit>: *right-click*
Command:

5. *Click* the **New Distant Light** tool icon in the flyout from the **Lights** tool icon. The command line shows:

Command:_distantlight
Specify light direction FROM <0,0,0> or [Vector]: *enter* **5,0,400** *right-click*
Specify light direction TO <1,1,1>: *enter* **150,200,70** *right-click*
Enter an option to change [Name/Intensity/Status/shadoW/Color/eXit] <eXit>: *enter* **n** *right-click*
Enter light name <Distantlight1>: *right-click*
Enter an option to change [Name/Intensity/Status/shadoW/Color/eXit] <eXit>: *right-click*
Command:

6. Place another **Distant Light** (Distantlight2) in the same position **TO** and **FROM** −80,−20,300.

Note

The **Intensity** of the lights can be set, **Shadow** can be set off or on in a **Sharp** or **Soft** setting, the **Color** of a light can be changed as needed in response to the prompts appearing when a light is added to a view.

7. When the model has been rendered if a light requires to be changed in intensity, shadow, position or colour, *click* the **Light List** icon in the **Render** toolbar and the **LIGHTS IN MODEL** palette appears. *Click* a light name and the **PROPERTIES** palette for the light appears in which modifications can be made (Fig. 16.18). Make amendments as thought necessary.

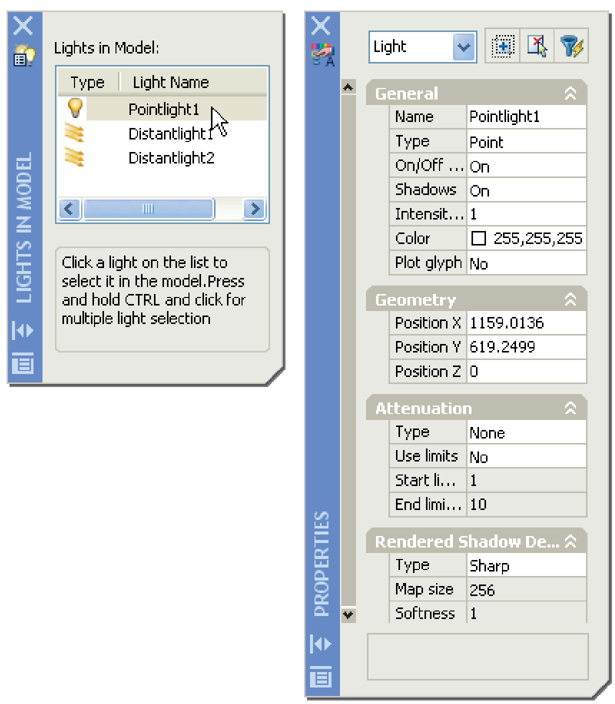


Fig. 16.18 The **LIGHTS IN MODEL** and **PROPERTIES** palettes

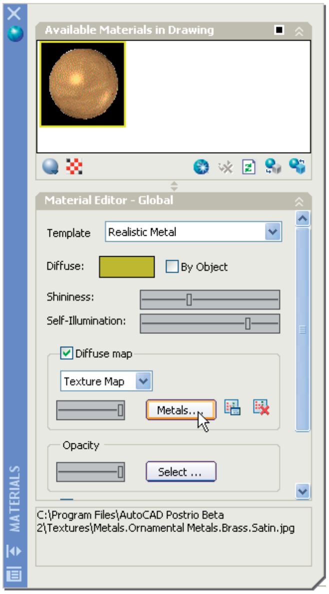
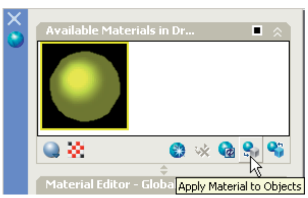


Fig. 16.19 The **MATERIALS** palette



1Fig. 16.20 The **Apply Material to Objects** button in the **MATERIALS** palette

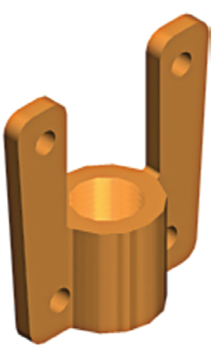


Fig. 16.21 First example – **Rendering**

Adding a material to the model

1. *Click* the **Materials** . . . tool icon in the **Render** toolbar. The **MATERIALS** palette appears (Fig. 16.19). *Click* the **Diffuse** box and from the **Select Color** dialog which appears, choose a colour most suited to a brass finish.
2. Adjust the sliders **Shininess** and **Self-Illumination** until satisfied, the desired material appears in the **Swatch Geometry** box at the top of the palette.
3. *Click* the **Select** . . . button next to the **Textures** field and from the **Select Image File** dialog which appears (not illustrated) select a suitable metal texture. Adjust the slider to amend as needed in the Swatch geometry box.
4. Continue making adjustments until satisfied, the material appears as wished.
5. *Click* the **Apply Material to Objects** button (Fig. 16.20) and *click* the objects in the 3D model drawing to which the material is to be applied.
6. *Click* the **Render** tool icon in the **Render** toolbar and the 3D model renders (Fig. 16.21).

Notes

The limited descriptions of rendering given in these pages do not show the full value of different types of lights, materials and rendering methods. The reader is advised to experiment with the facilities available for rendering.

Second example – rendering a 3D model (Fig. 16.23)

1. Construct 3D models of the two parts of the stand and support given in the projections in Fig. 16.22 with the two parts assembled together.

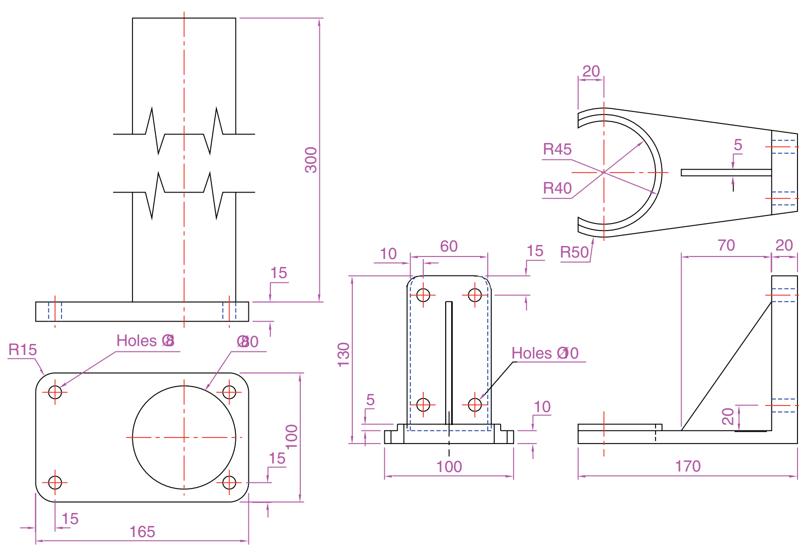


Fig. 16.22 Second example – **Rendering** – projections of the two parts

2. Place the scene in the **3D Views/Top** view and add lighting.
3. Add two materials to the parts of the assembly and render the result. Fig. 16.23 shows the resulting rendering.

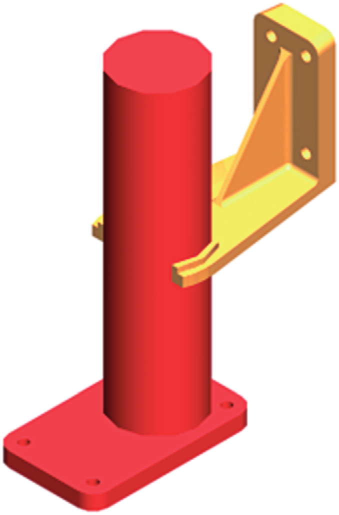


Fig. 16.23 Second example – **Rendering**

The 3D Orbit tool

At the command line *enter* **3dorbit**. The command line shows:

Command: 3dorbit
Press ESC or ENTER to exit, or right-click to display shortcut-menu.

Right-click anywhere on screen and the **3dorbit** *right-click* menu appears (Fig. 16.24). *Click* **Free Orbit** and a circle appears surrounding a 3D model on screen. The position and angle of the model can be adjusted by either *clicking* in one of the four outer small circles or by *clicking* outside the main circle and moving the mouse.

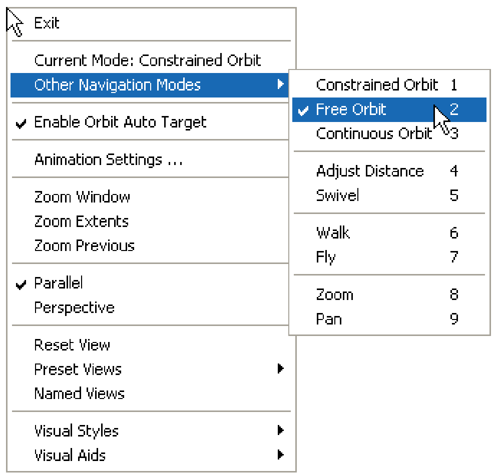


Fig. 16.24 The **3dorbit** *right-click* menu

Example – 3D Orbit (Fig. 16.25)

This is another tool for the manipulation of 3D models into different positions within 3D space.

1. Open the file of the second example of rendering (Fig. 16.23).
2. *Enter* **3dorbit** at the command line.
3. With the cursor outside the circle move the mouse. The 3D model rotates within the circle.
4. With the cursor inside the circle move the mouse. The 3D model rotates around the screen.
5. With the cursor inside any one of the small quadrant circles the 3D model can be moved vertically or horizontally as the mouse is moved.
6. *Right-click* anywhere in the screen. A *right-click* menu appears. *Click* on **Visual Styles** and again on **Realistic** in the sub-menu which appears. The parts of the 3D model are shaded in the materials which

have been applied to the parts of the model. The result is given in Fig. 16.25.

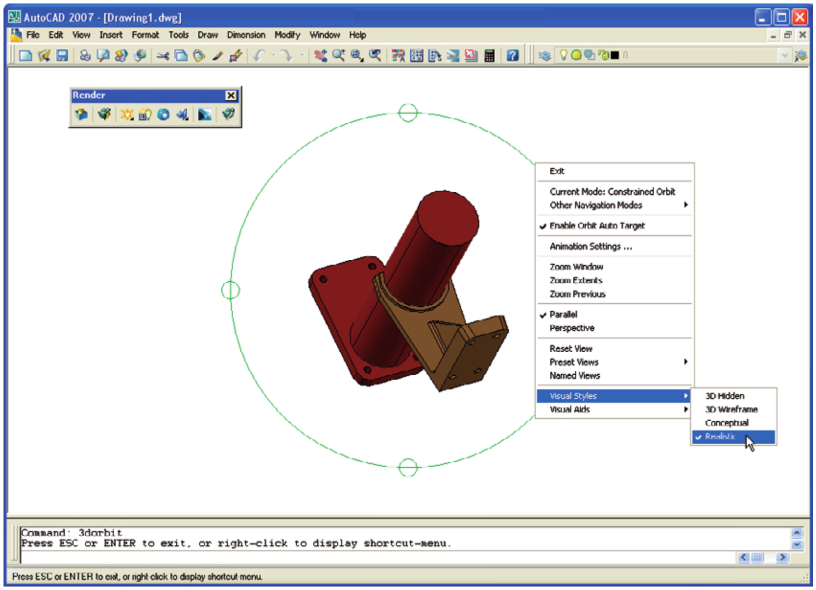


Fig. 16.25 The **3dorbit Visual Styles** menu

7. Fit the 3D model into a **Four: Equal** viewports setting. Note that the **Realistic Visual Style** mode still shows in each of the four viewports (Fig. 16.26).

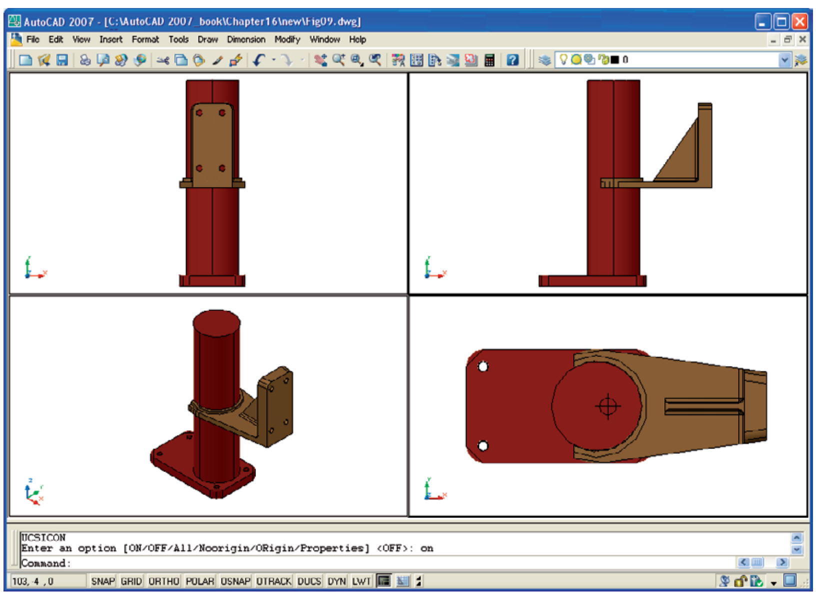


Fig. 16.26 The **Realistic Style** 3D model in a **Four: Equal** viewports setting

Producing hardcopy

Printing or plotting a drawing on screen using AutoCAD 2007 can be carried out from either the **Model Space** or the **Paper Space**. In versions of AutoCAD before AutoCAD 2004, it was necessary to print or plot from **PSpace**.

First example – printing a single copy (Fig. 16.28)

Fig. 16.27 The **Plot** tool icon in the **Standard** toolbar

1. With a drawing to be printed or plotted on screen *click* the **Plot** tool icon in the **Standard** toolbar (Fig. 16.27).
2. The **Plot** dialog appears. Set the **Printer/Plotter** to a printer or plotter currently attached to the computer and the **Paper Size** to a paper size to which the printer/plotter is set.
3. *Click* the **Preview** button of the dialog and if the preview is OK, *right-click* and in the *right-click* menu which appears, *click* **Plot**. The drawing plots producing the necessary 'hardcopy' (Fig. 16.28).

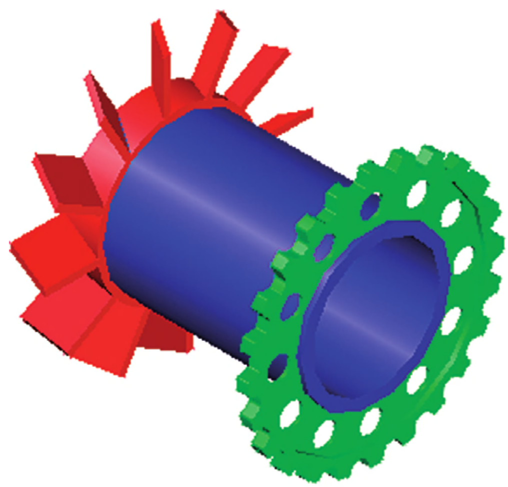


Fig. 16.28 First example – **Printing a single copy**

Second example – multiple view copy (Fig. 16.29)

A 3D model to be printed is a **Realistic** view of a 3D model which has been constructed on three layers – **Red, Blue** and **Green** in colour. To print a multiple view copy:

1. Place the drawing in a **Four: Equal** viewports setting.
2. Make a new layer **vports** of colour cyan and make it the current layer.
3. *Click* the **Layout** button in the status bar. The drawing appears in **Pspace**. A view of the 3D model appears within a cyan coloured viewport (Fig. 16.29).
4. *Click* the **Plot** tool icon in the **Standard** toolbar. Make sure the correct **Printer/Plotter** and **Paper Size** settings are selected and *click* the **Preview** button of the dialog.
5. A preview of the 3D model appears.

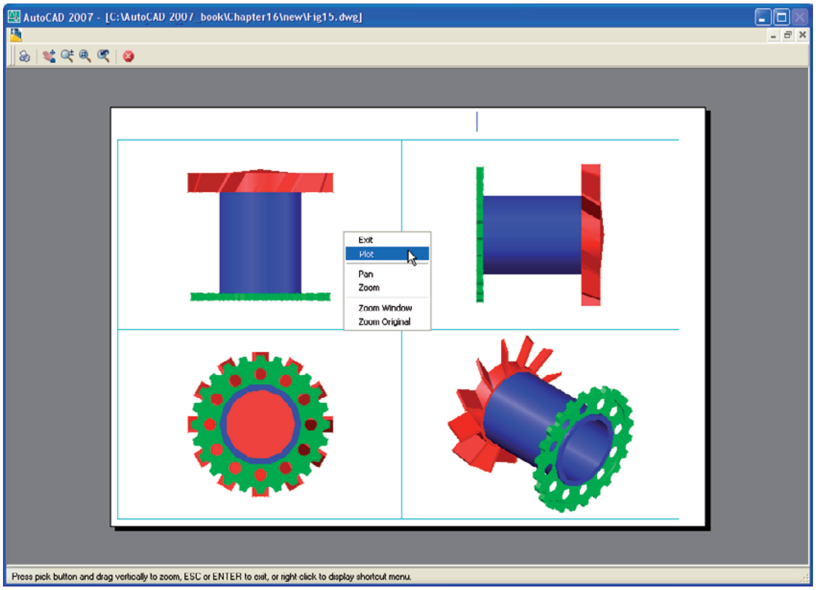


Fig. 16.29 Second example **Multiple view copy** – the print **Preview**

6. If the preview is satisfactory (Fig. 16.29), *right-click* and from the *right-click* menu *click* **Plot**. The drawing plots to produce the required four-viewport hardcopy.

Other forms of hardcopy

When working in AutoCAD 2007, several different forms of hardcopy can be printed or plotted determined by the settings in the **3D Orbit/Visual Styles** settings (Fig. 16.29). As an example a single view plot preview of the same 3D model is shown in the **Hidden** shading form (Fig. 16.30).

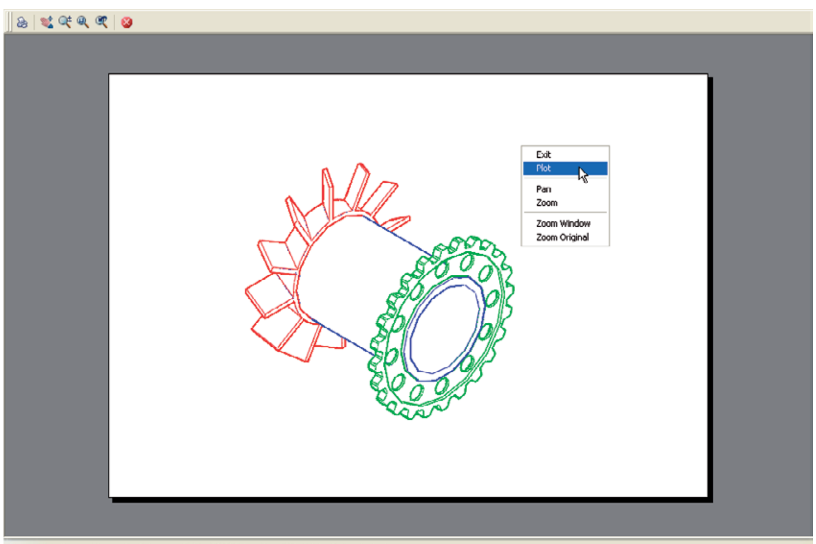


Fig. 16.30 An example of a **Hidden Style** plot **Preview**

Saving and opening 3D model drawings

3D model drawings are saved and/or opened in the same way as are 2D drawings. To save a drawing *click* **Save As** . . . in the **File** drop-down menu and save the drawing in the **Save Drawing As** dialog and *enter* a file name in the **File Name** field of the dialog before *clicking* the **Save** button. To open a drawing which has been saved *click* **Open** . . . in the **File** drop-down menu, and in the **Select File** dialog which appears select a file name from the file list.

There are differences between saving a 2D and a 3D drawing, in that when a 3D model drawing is shaded by using a shading mode from the **3D Orbit/Shading Modes** sub-menu, the shading is saved with the drawing.

Exercises

1. A rendering of an assembled lathe tool holder is shown in Fig. 16.31. The rendering includes different materials for each part of the assembly. Working to the dimensions given in the parts orthographic drawing (Fig. 16.32), construct a 3D model drawing of the assembled lathe tool holder on several layers of different colours, add lighting and materials and render the model in an isometric view.

Shade with **3DOrbits/Hidden** and print or plot a **SW Isometric** view of the model drawing.

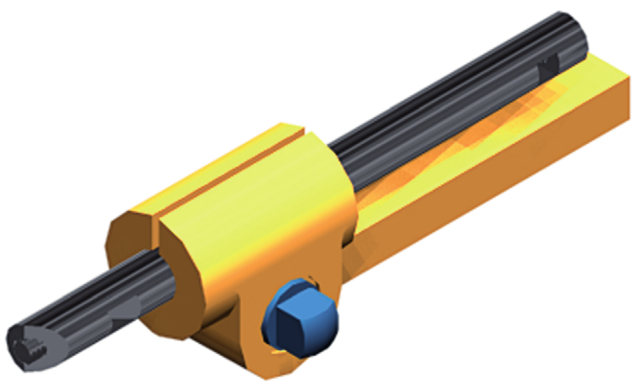


Fig. 16.31 Exercise 1 – a rendering

2. Working to the sizes given in Fig. 16.33, construct a 3D model drawing of the drip tray from an engine. Add lighting and a suitable material, place the model in an isometric view and render (Fig. 16.34).

3. A three-view drawing of a hanging spindle bearing in third angle orthographic projection is shown in Fig. 16.35. Working to the dimensions in the drawing, construct a 3D model drawing of the bearing.

Add suitable lighting and a suitable material, place in an isometric view and render the model.

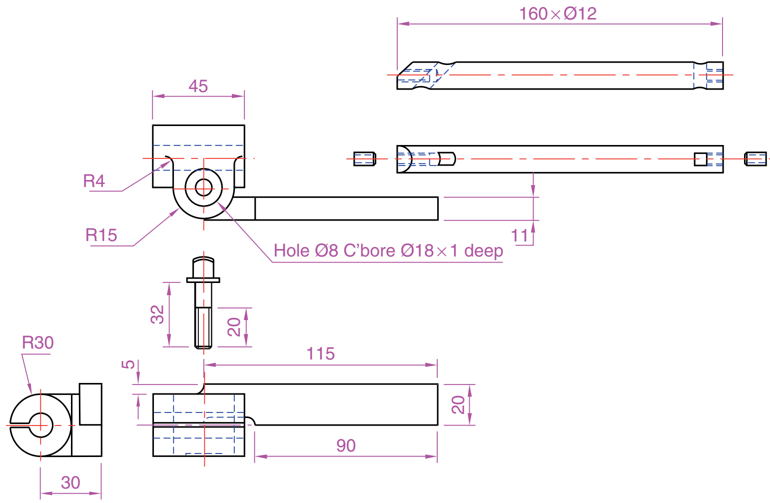


Fig. 16.32 Exercise 1 – parts drawing

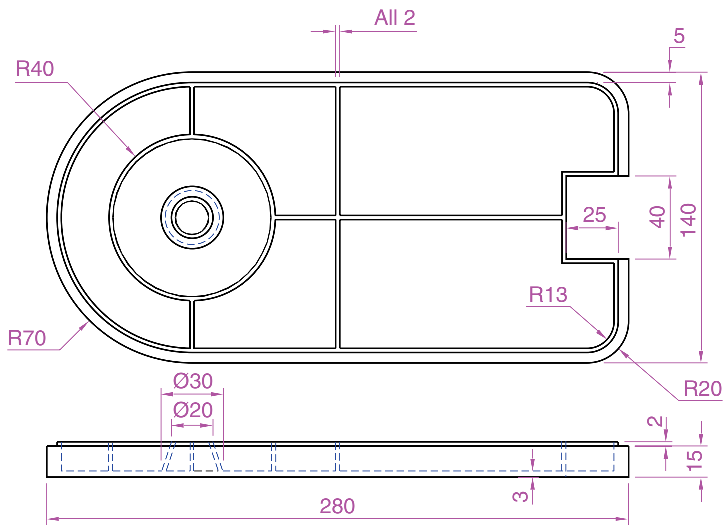


Fig. 16.33 Exercise 2 – orthographic projection

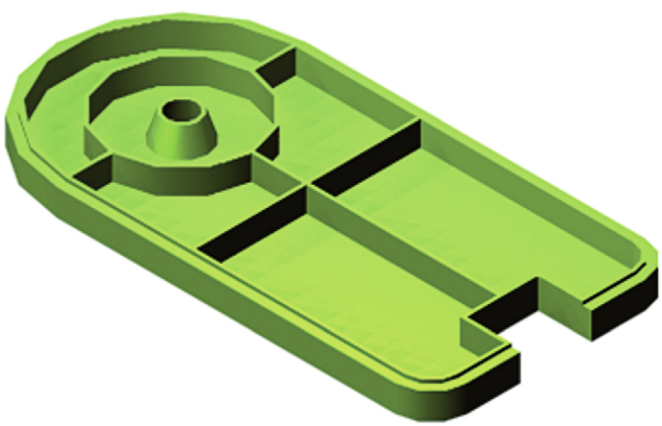


Fig. 16.34 Exercise 2

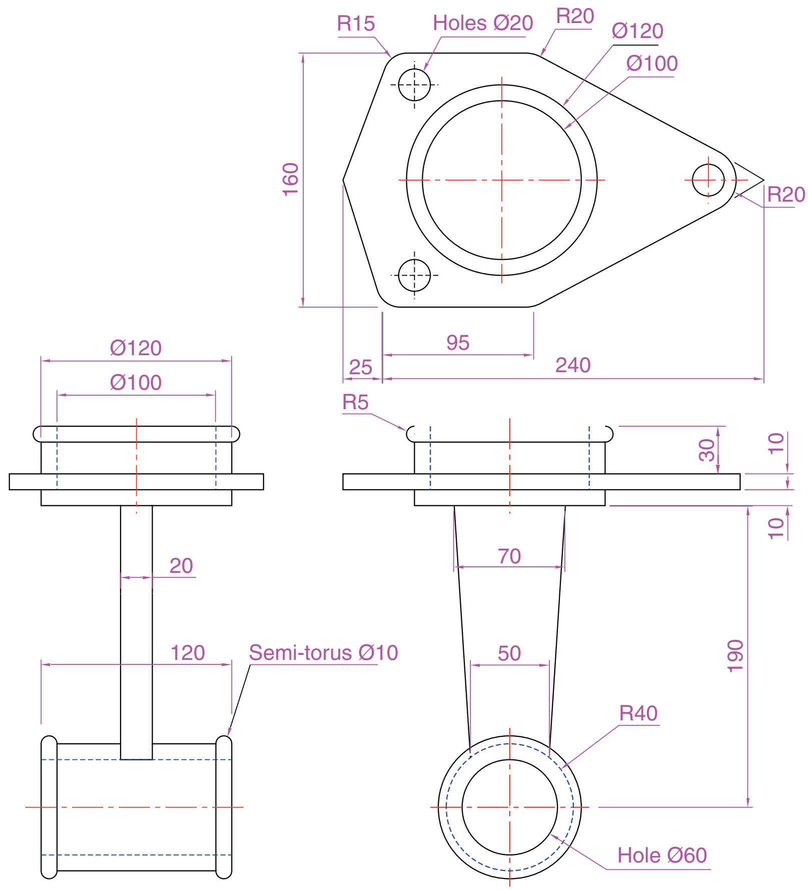


Fig. 16.35 Exercise 3

CHAPTER 17

3D space

Aim of this chapter

To show in examples the methods of manipulating 3D models in 3D space using tools from the **UCS** toolbars or from the command line.

3D space

So far in this book, when constructing 3D model drawings, they have been constructed on the AutoCAD 2007 coordinate system which is based upon three planes: the **XY Plane** – the screen of the computer; the **XZ Plane** at right angles to the **XY Plane** and as if coming towards the operator of the computer; and a third plane (**YZ**) is lying at right angles to both the other two planes (Fig. 17.1).

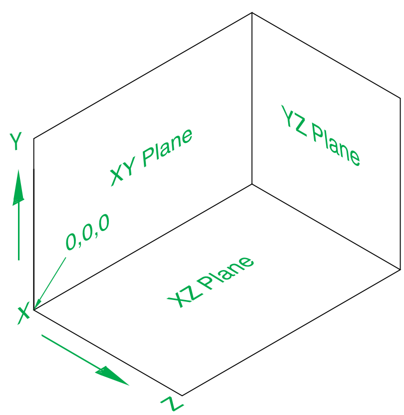


Fig. 17.1 The 3D space planes

In earlier chapters in order to view 3D objects which have been constructed on these three planes at other angles, we have used presets from the **3D Views** sub-menu of the **Views** drop-down menu and have indicated other methods of rotating the model in 3D space and placing the model in other viewing positions using the **View point Presets** dialog.

Note

The **XY** plane is the basic **UCS** plane, which in terms of the ucs is known as the ***WORLD*** plane.

The User Coordinate System (UCS)

The **UCS** allows the operator to place the AutoCAD coordinate system in any position in 3D space using a variety of **UCS** tools (commands). Features of the **UCS** can be called either by *entering* **ucs** at the command line, or by selection from the **Tools** drop-down menu (Fig. 17.2), or from the two **UCS** toolbars – **UCS** and **UCS II** (Fig. 17.3).

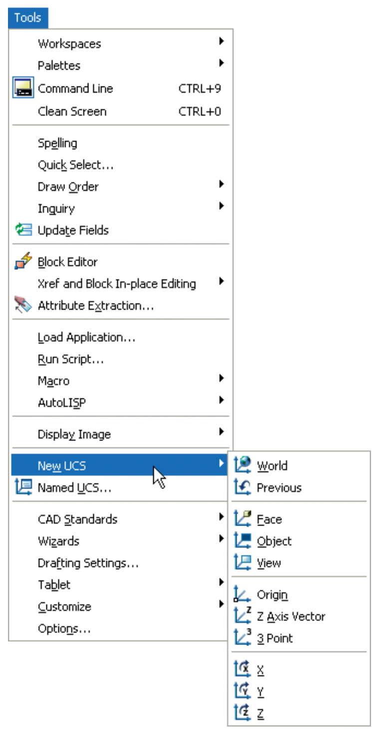


Fig. 17.2 The **New UCS** sub-menu from the **Tools** drop-down menu

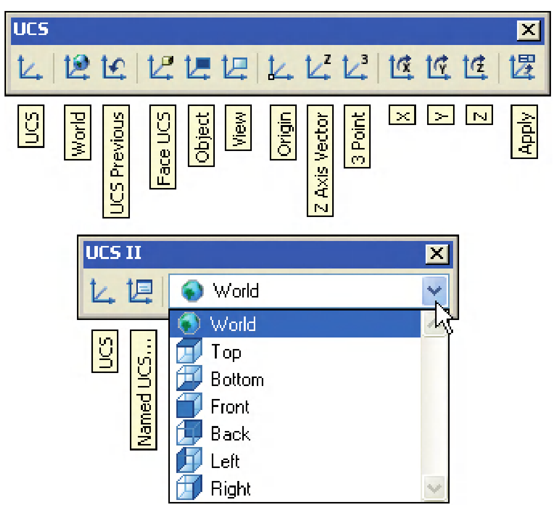


Fig. 17.3 The tools from the two **UCS** toolbars

If **ucs** is *entered* at the command line, it shows:

Command: *enter* **ucs** *right-click*
Current use name: *WORLD*
Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <World>: *enter* **n** (New) *right-click*
Specify origin of UCS or [ZAxis/3point/OBject/Face/View/X/Y/Z] <0,0,0>:

And from these prompt lines a selection can be made.

The variable UCSFOLLOW

UCS planes can be set from either of the two **UCS** toolbars (Figs 17.2 and 17.3). For the UCS to operate from the command line, the variable **UCSFOLLOW** must first be set on as follows:

Command: *enter* **ucsfollow** *right-click*
Enter new value for UCSFOLLOW <0>: *enter* **1** *right-click*
Command:

The UCS icon

The **UCS** icon which indicates the direction of the three coordinate axes **X, Y** and **Z** is by default shown in the AutoCAD drawing area as arrows pointing in the directions of the axes. When working in 2D, only the **X** and **Y** axes are showing, but when the drawing area is in a 3D view all three coordinate arrows are showing, except when the model is in the **XY** plane. The icon can be turned off as follows:

Command: *enter* **ucsicon** *right-click*
Enter an option [ON/OFF/Noorigin/ORigin/Properties] <ON>:

To turn the icon off, *enter* **off** in response to the prompt line and the icon disappears from the screen.

The appearance of the icon can be changed by *entering* **p** (Properties) in response to the prompt line. The **UCS Icon** dialog appears in which changes can be made to the shape, line width and colour of the icon if wished.

Types of UCS icon

The shape of the icon can be varied partly when changes are made in the **UCS Icon** dialog but also according to whether the AutoCAD drawing area is in 2D, 3D or Paper Space (Fig. 17.4).

Examples of changing planes using the UCS

First example – changing UCS planes (Fig. 17.6)

1. Set **UCSFOLLOW** to **1** (ON).
2. Place the screen in **3D Views/Front** and **Zoom** to **1**.
3. Construct the pline outline in Fig. 17.5 and extrude to a height of **120**.

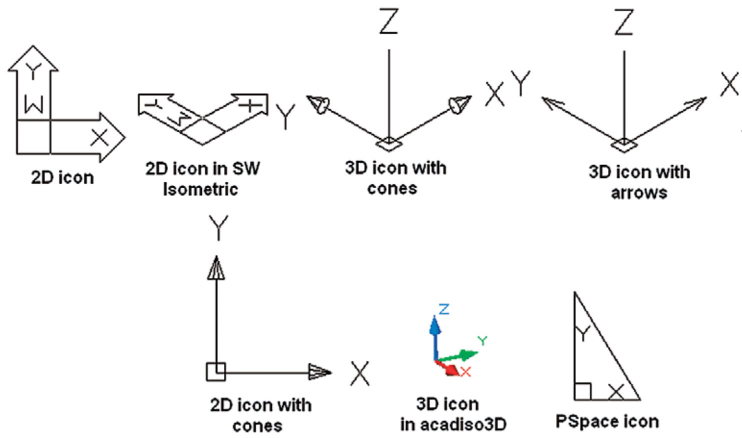


Fig. 17.4 Types of **UCS** icon

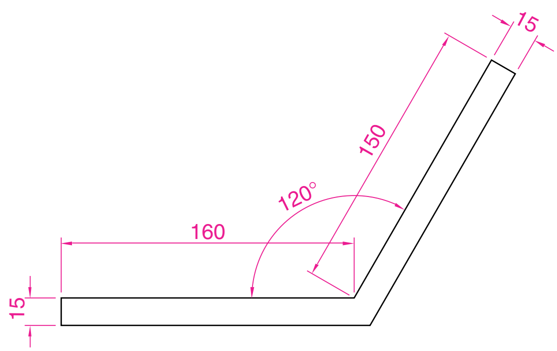


Fig. 17.5 First example – **Changing UCS planes** – pline for extrusion

4. Set **UCSFOLLOW** to **1**.
5. Place in the **SW Isometric** view and **Zoom** to **1**.
6. With the **Fillet** tool, fillet corners to a radius of **20**.
7. At the command line:

   Command: *enter* **ucs** *right-click*
   Current ucs name: *WORLD*
   Enter an option New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <World>: *enter* **n** (New) *right-click*
   Specify origin of new UCS or [ZAxis/3point/OBject/Face/View/X/Y/Z/]: *enter* **f** (Face) *right-click*
   Select face of solid object: *pick* the sloping face – its outline highlights
   Enter an option [Next/Xflip/Yflip] <accept>: *right-click*
   Command:

   And the 3D model changes its plane so that the sloping face is now on the new UCS plane. **Zoom** to **1**.
8. On this new UCS, construct four cylinders of radius 7.5 and height –15 (note the minus) and subtract them from the face.
9. *Enter* **ucs** at the command line again and *right-click* to place the model in the ***WORLD* UCS** – that being the plane in which the construction commenced.

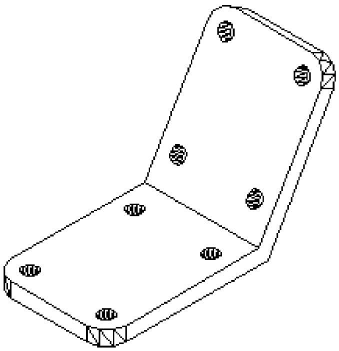


Fig. 17.6 First example – **changing UCS planes**

10. Place four cylinders of the same radius and height into position in the base of the model and subtract them from the model.
11. Place the 3D model in the **SW Isometric** view and call **Hide** (Fig. 17.6).

Second example – UCS (Fig. 17.9)

The 3D model for this example is a steam venting valve from a machine, the two-view third angle projection of which is shown in Fig. 17.7.

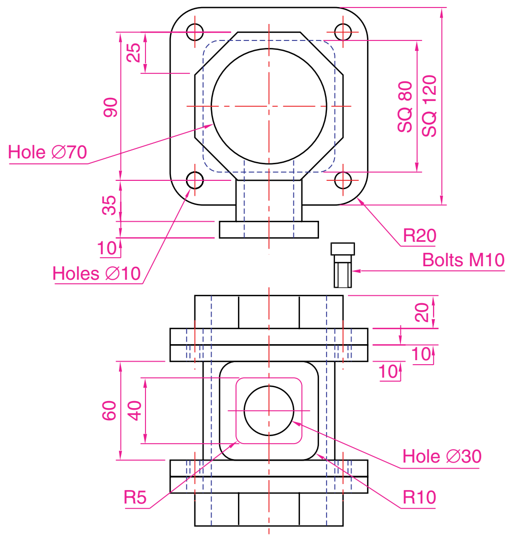


Fig. 17.7 **Second example UCS** – the orthographic projection of a steam venting valve

1. Make sure that **UCSFOLLOW** is set to **1**.
2. The **UCS** plane is the ***WORLD*** plane. Construct the **120** square plate at the base of the central portion of the valve. Construct five cylinders for the holes in the plate. Subtract the five cylinders from the base plate.
3. Construct the central part of the valve – a filleted **80** square extrusion with a central hole.
4. Place the models in the **UCS orthoGonal/Front** plane.
5. With the **Move** tool, move the central portion vertically up by **10**.
6. With the **Copy** tool, copy the base up to the top of the central portion.
7. With the **Union** tool form a single 3D model of the three parts.
8. Make the layer **Construction** current.
9. Place the model in the **UCS *WORLD*** plane. Construct the separate top part of the valve – a plate forming a union with an hexagonal plate and with holes matching those of the other parts.
10. Place the scene so far in the **UCS/orthoGonal/Front** plane and move the parts of the top into their correct positions relative to each other and with **Union** and **Subtract** tools, complete the part. This will be made easier if the layer **0** is turned off.
11. Turn layer **0** on and move the top into its correct position relative to the main part of the valve. Then with the **Mirror** tool, mirror the top to produce the bottom of the assembly (Fig. 17.8).

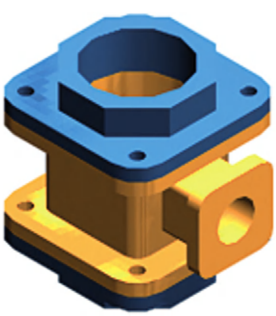


Fig. 17.8 **Second example UCS** – construction up to step **11** + rendering

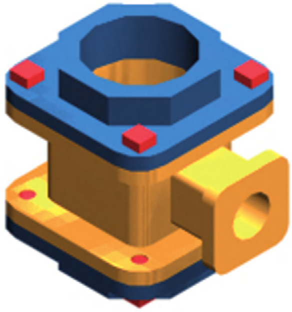


Fig. 17.9 **Second example UCS – steps 12 and 13 + rendering**

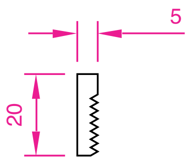


Fig. 17.10 **Second example UCS** – pline for the bolt

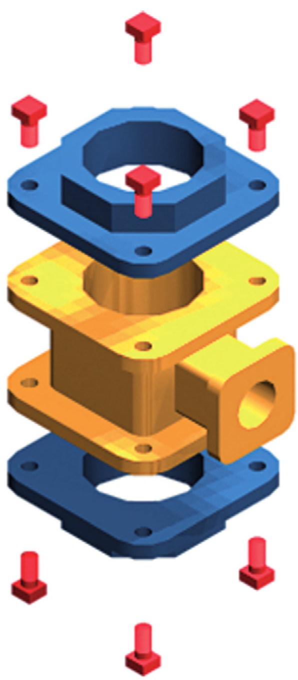


Fig. 17.11 **Second example – UCS**

12. While in the ***FRONT* UCS** construct the three parts of a 3D model of the extrusion to the main body.
13. In the **UCS *WORLD*** move the parts into their correct positions relative to each other and with **Union** form a union of the two filleted rectangular extrusions and the main body. Then with **Subtract**, subtract the cylinder from the whole (Fig. 17.9).
14. In the **UCS *FRONT*** plane, construct one of the bolts as shown in Fig. 17.10, forming a solid of revolution from a pline. Then add a head to the bolt and with **Union** add it to the screw.
15. With the **Copy** tool, copy the bolt seven times to give eight bolts. With **Move**, and working in the **UCS *WORLD*** and ***FRONT*** planes, move the bolts into their correct positions relative to the 3D model.
16. Add suitable lighting and attach materials to all parts of the assembly and render the model.
17. Save the model to a suitable file name.
18. Finally move all the parts away from each other to form an exploded view of the assembly (Fig. 17.11).

Third example – UCS (Fig. 17.15)

1. Set **UCSFOLLOW** to **1**.
2. Place the drawing area in the **UCS FRONT** view.
3. Construct the outline in Fig. 17.12 and extrude to a height of **120**.
4. Either *click* the **3 Point UCS** tool icon in the **UCS** toolbar (Fig. 17.13) or at the command line:

   Command: *enter* **ucs** *right-click*
   Current ucs name: *RIGHT*
   Enter an option [prompts]: *enter* **n** (New) *right-click*
   Specify origin of UCS or [prompts]: *enter* **3** (3point) *right-click*
   Specify new origin point: *pick*
   Specify point on positive portion of X-axis: *pick*
   Specify point on positive-Y portion of UCS XY plane:
   enter **.xy** *right-click*
   of *pick* **(need Z):** *enter* **−1** (note the minus sign) *right-click*
   Regenerating model.
   Command:

   And the model regenerates in this new 3point plane.

Note

Fig. 17.14 shows the three **UCS** points.

5. On the face of the model construct a rectangle **80** × **50** central to the face of the front of the model, fillet its corners to a radius of **10** and extrude to a height of **10**.
6. Place the model in the **SW Isometric** view and fillet the back edge of the second extrusion to a radius of **10**.
7. Subtract the second extrusion from the first.
8. Add lights and a suitable material, and render the model (Fig. 17.15).

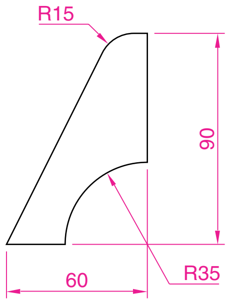


Fig. 17.12 Third example **UCS** – outline for 3D model

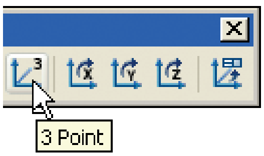


Fig. 17.13 The **3 Point UCS** icon in the **UCS** toolbar

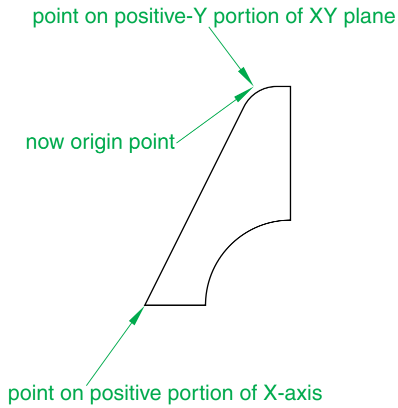


Fig. 17.14 Third example **UCS** – the three UCS points

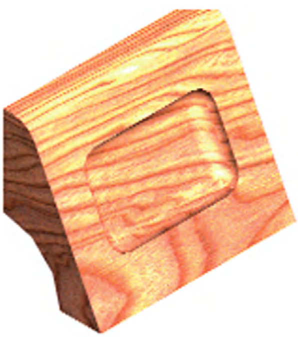


Fig. 17.15 Third example **UCS**

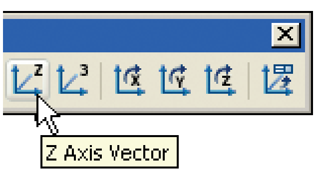


Fig. 17.16 The **Z Axis Vector UCS** icon in the **UCS** toolbar

Fourth example – UCS (Fig. 17.17)

1. With the last example still on screen, place the model in the **UCS *WORLD*** view.
2. *Click* the **Z Axis Vector UCS** tool icon in the **UCS** toolbar (Fig. 17.16). The command line shows:

Command:_ucs
Current ucs name: *WORLD*
Enter an option [prompts] <World>:_zaxis
Specify a new origin point: *enter* **40, 60** *right-click*
Specify point on positive portion of Z-axis <40, 60, 0>: *enter* **.xy** *right-click*
of *enter* **170,220** *right-click* **(need Z):** *enter* **1** *right-click*
Regenerating model.
Command:

3. Render the model in its new **UCS** plane (Fig. 17.17).

Fig. 17.17 Fourth example **UCS**

Saving UCS views

If a number of different **UCS** planes are used in connection with the construction of a 3D model, each can be saved to a different name and recalled when required. To save the UCS plane in which a 3D model drawing is being constructed, either *click* the **UCS** tool icon in the **UCS** toolbar (Fig. 17.18) or *enter* **ucs** at the command line:

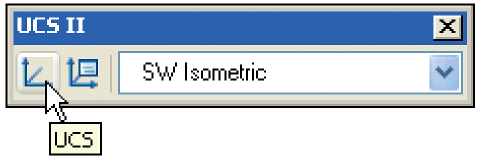


Fig. 17.18 The **UCS** icon in the **UCS** toolbar

Command:_ucs
Current ucs name: NW Isometric
Enter an option [prompts]: *enter* **s** (Save) *right-click*
Enter name to save current UCS or [?]: *enter* **SW Isometric** *right-click*
Regenerating drawing.
Command:

Now *click* the **Named UCS** . . . tool icon in the **UCS II** toolbar (Fig. 17.19) and the **UCS** dialog appears (Fig. 17.20) showing the names of the views saved in the current drawing.

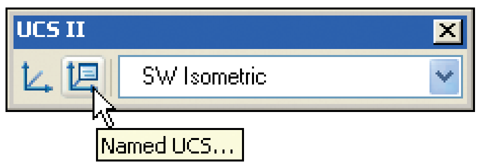


Fig. 17.19 The **Named UCS** . . . tool icon in the **UCS II** toolbar

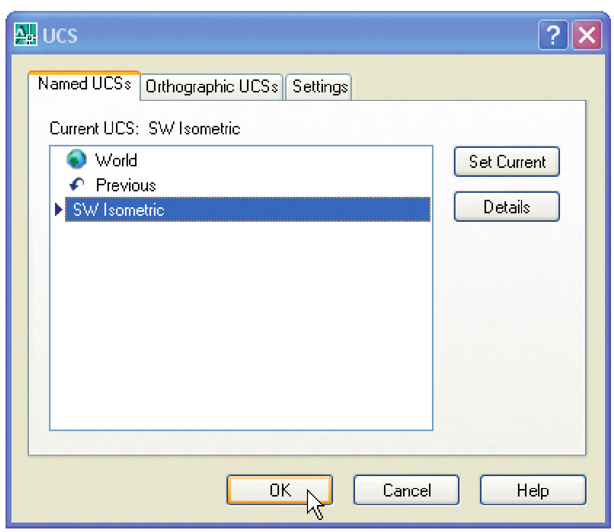


Fig. 17.20 The **UCS** dialog

Constructing 2D objects in 3D space

In previous chapters of this book there have been examples of 2D objects constructed with the **Polyline, Line, Circle** and other 2D tools to form the outlines for extrusions and solids of revolution. These outlines have been drawn on planes set either from the **3D Views** sub-menu of the **Views** drop-down menu, or in **UCS** planes such as the **UCS *RIGHT*, *FRONT*** and ***LEFT*** planes.

First example – 2D outlines in 3D space (Fig. 17.23)

1. Construct a **3point UCS** to the following points:

   origin point: 80,90
   X-axis point: 290,150
   positive-Y point: .xy of **80,90**
   (need Z): *enter* **1**.

2. On this **3point UCS** construct a 2D drawing of the plate to the dimensions given in Fig. 17.21, using the **Polyline, Ellipse** and **Circle** tools.

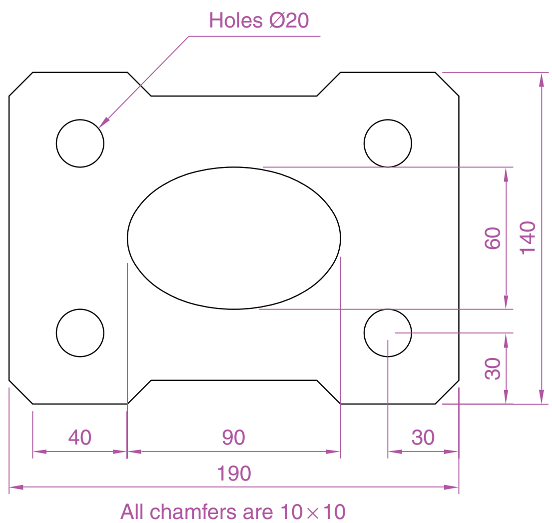


Fig. 17.21 First example – **2D outlines in 3D space**

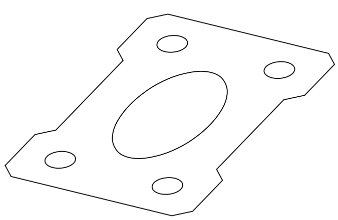


Fig. 17.22 First example – **2D outlines in 3D space** – the outline in a SW Isometric view

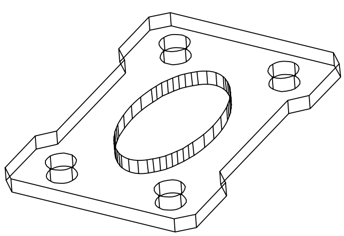


Fig. 17.23 First example – **2D outlines in 3D space**

3. Save the **UCS** plane in the **UCS** dialog to the name **3point**.
4. Place the drawing area in the **SW Isometric** view (Fig. 17.22).
5. With the **Region** tool form regions of the six parts of the drawing and with the **Subtract** tool, subtract the circles and ellipse from the main outline.
6. Extrude the region to a height of **10** (Fig. 17.23).

Second example – 2D outlines in 3D space (Fig. 17.26)

1. Place the drawing area in the **UCS *FRONT*** view, **Zoom** to **1** and construct the outline in Fig. 17.24.

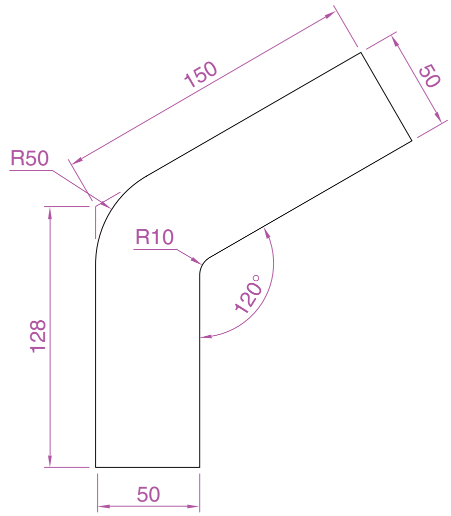


Fig. 17.24 Second example – **2D outlines in 3D space** – outline to be extruded

2. Extrude the outline to a height of **150**.
3. Place in the **3D Views/SW Isometric** view and **Zoom** to **1**.
4. *Click* the **Face UCS** tool icon in the **UCS** toolbar (Fig. 17.25) and place the 3D model in the ucs plane shown in Fig. 17.26, selecting the sloping face of the extrusion for the plane and again **Zoom** to **1**.

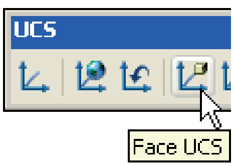


Fig. 17.25 The **Face UCS** icon from the **UCS** toolbar

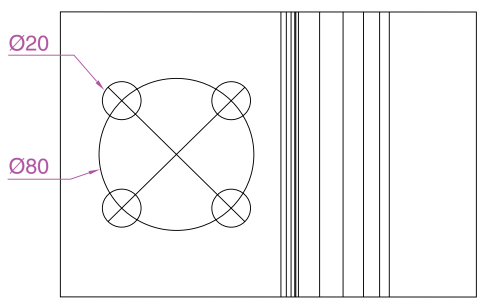


Fig. 17.26 Second example – **2D outlines in 3D space** – the circles in the new UCS face

5. With the **Circle** tool draw five circles as shown in Fig. 17.26.
6. Form a region from the five circles and with **Union** form a union of the regions.
7. Extrude the region to a height of **−60** (note the minus) – higher than the width of the sloping part of the 3D model.
8. Place the model in the **SW Isometric** view and subtract the extruded region from the model.

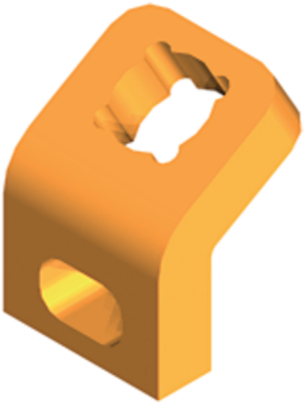


Fig. 17.27 Second example – **2D outlines in 3D space**

9. With the **Fillet** tool, fillet the upper corners of the slope of the main extrusion to a radius of **30**.
10. Place the model into another **UCS FACE** plane and construct a filleted pline of sides **80** and **50** and filleted to a radius of **20**. Extrude to a height of −**60** and subtract the extrusion from the 3D model.
11. Place in the **SW Isometric** view, add lighting and attach a material and render (Fig. 17.27).

Revision notes

1. The **UCS** (User Coordinate System) tools can be called from the two toolbars **UCS** and **UCS II** or from sub-menus from the **Tools** drop-down menu or by *entering* **ucs** at the command line.
2. The variable **UCSFOLLOW** must first be set on (to **1**) before operations of the UCS can be brought into action.
3. There are several types of **UCS** icon – 2D (different types), 3D (different types) and Pspace.
4. The position of the plane in 3D space on which a drawing is being constructed can be varied using tools from the **UCS**.
5. The different planes on which drawings are constructed in 3D space can be saved in the **UCS** dialog.

Fig. 17.28 Exercise 1–a rendering

Exercises

1. The two-view projection in Fig. 17.29 shows an angle bracket in which two pins are placed in holes in each of the arms of the bracket.
   Construct a 3D model of the bracket and its pins.
   Add lighting to the scene and materials to the parts of the model and render (Fig. 17.28).
2. The two-view projection (Fig. 17.31) shows a stand consisting of two hexagonal prisms. Circular holes have been cut right through each

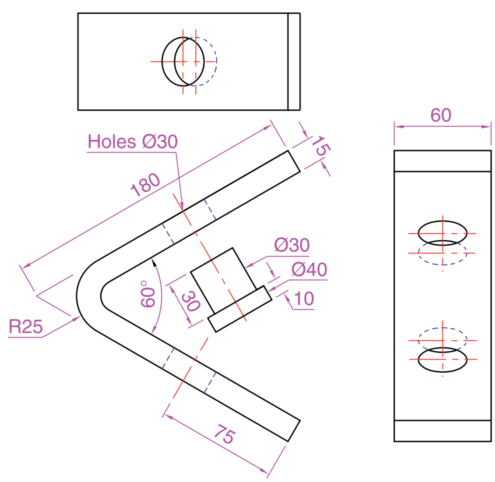


Fig. 17.29 Exercise 1 – details of the shapes and sizes

face of the smaller hexagonal prism and rectangular holes with rounded ends have been cut right through the faces of the larger.

Construct a 3D model of the stand. When completed add suitable lighting to the scene. Then add a material to the model and render (Fig. 17.30).

3. The two-view projection in Fig. 17.32 shows a ducting pipe. Construct a 3D model drawing of the pipe. Place in a **SW Isometric** view, add lighting to the scene and a material to the model and render.

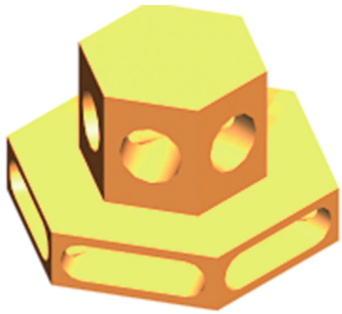


Fig. 17.30 Exercise 2 – a rendering

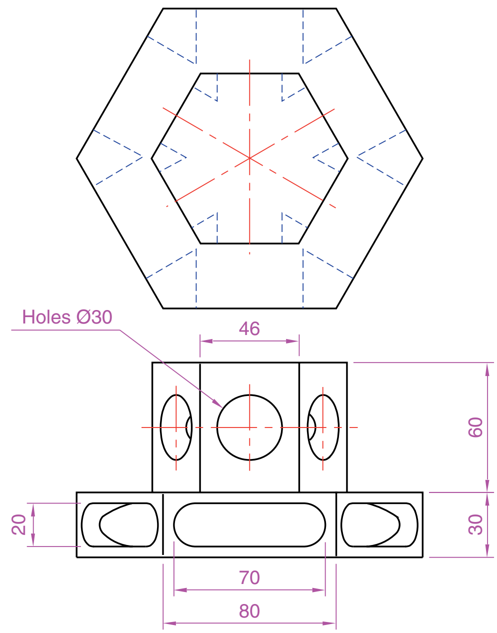


Fig. 17.31 Exercise 2 – details of the shapes and sizes

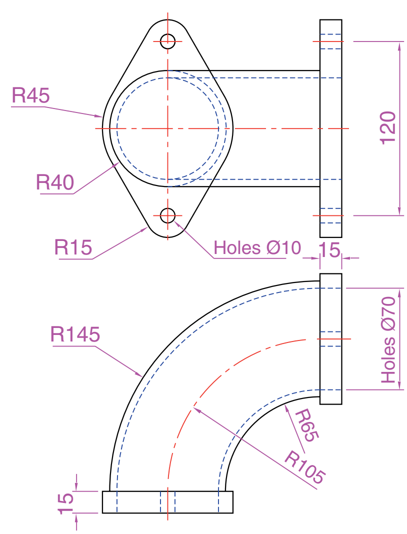


Fig. 17.32 Exercise 3 – details of the shapes and sizes

4. A point marking device is shown in the two-view projection in Fig. 17.33. The device is composed of three parts – a base, an arm and a pin.

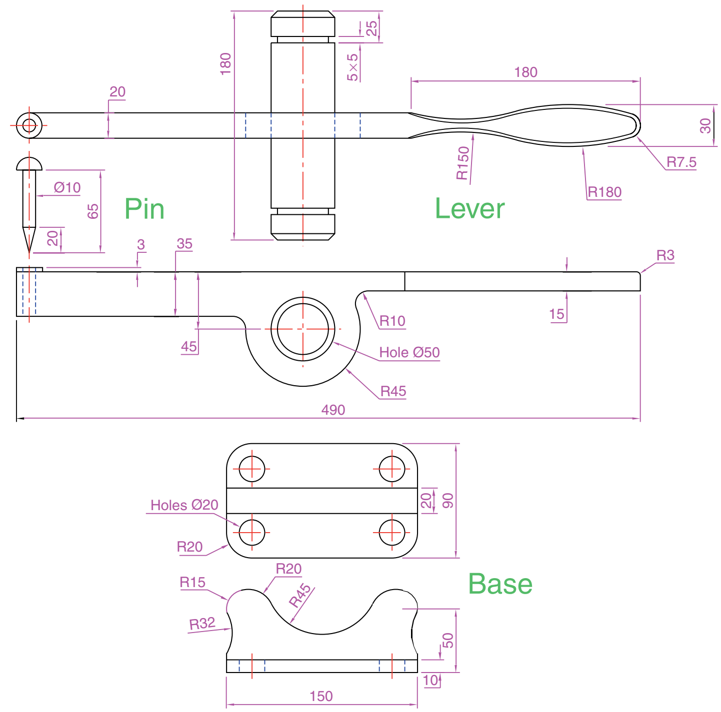


Fig. 17.33 Exercise 4 – details of the shapes and sizes

Construct a 3D model of the assembled device and add appropriate materials to each part. Then add lighting to the scene and render in a **SW Isometric** view (Fig. 17.34).

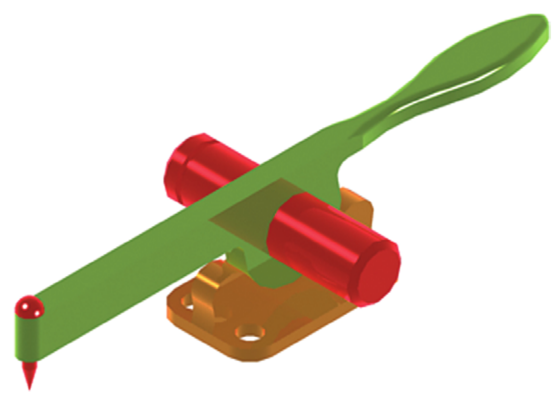


Fig. 17.34 Exercise 4 – a rendering

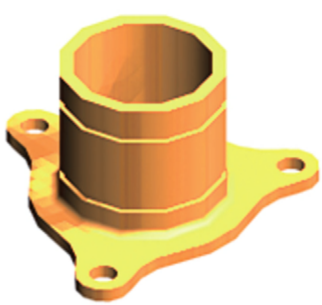


Fig. 17.35 Exercise 5 – a rendering

5. Fig. 17.35 shows the rendering of a 3D model drawing of the connecting device shown in the orthographic projection in Fig. 17.36. Construct the 3D model drawing of the device and add a suitable lighting to the scene.

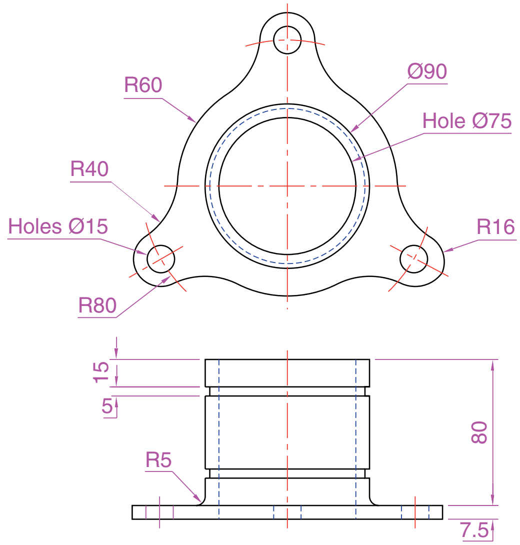


Fig. 17.36 Exercise 5 – two-view drawing

Then place in a **SE Isometric** view, add a material to the model and render.

6. A fork connector and its rod and are shown in a two-view projection in Fig. 17.37. Construct a 3D model drawing of the connector with its rod in position. Then add lighting to the scene, place in an **Isometric** viewing position, add materials to the model and render.
7. An orthographic projection of the parts of a lathe steady are given in Fig. 17.39. From the dimensions shown in the drawing, construct an assembled 3D model of the lathe steady.

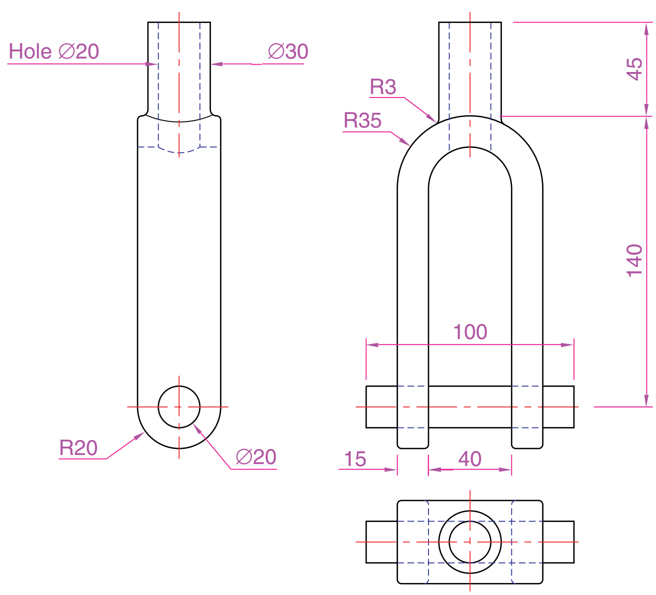


Fig. 17.37 Exercise 6

When the 3D model has been completed, add suitable lighting and materials and render the model as shown in Fig. 17.38.

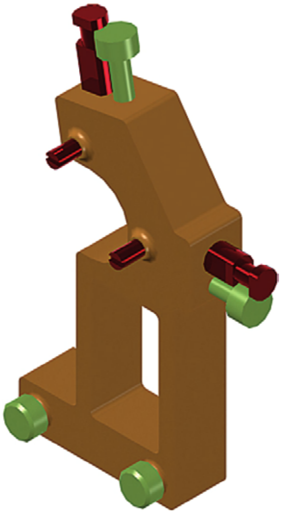


Fig. 17.38 Exercise 7 – a rendering

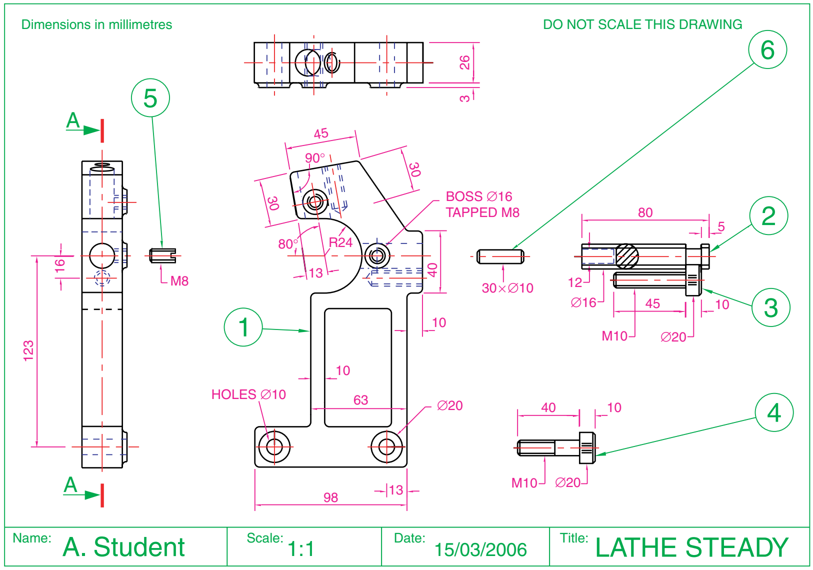


Fig. 17.39 Exercise 7 – details

CHAPTER 18

3D surface models

Aims of this chapter

1. To introduce the idea of 3D surfaces.
2. To compare 3D surface models with 3D solid drawing models.
3. To give examples of 3D surface models.

3D surface meshes

3D surface models will be introduced in this chapter. The 3D models described in earlier chapters have all been constructed using the tools from the **3D Make control panel** or the **Modeling** toolbar. 3D surface models are constructed using the surface commands. 3D surface model drawings are constructed from 3D surface meshes. In some instances it appears as if 3D solid models are the same as 3D surface models, but in fact there are distinct differences.

1. The surface meshes of 3D solid models are controlled by the settings of the variable **ISOLINES**.
2. The surface meshes of 3D surface models are controlled by either the number of segments or the two variables **SURFTAB1** and/or **SURFTAB2**.
3. The Boolean operators **Union, Subtract** and **Intersect** can be used to join, subtract or intersect 3D model objects, but have no action when used with 3D surface objects.

Setting the 3D Modeling screen to 2D Wireframe

All the surface model drawings in this chapter have been constructed in a **2D Wireframe** format on a **Parallel** grid system which can be set up as indicated in Fig. 18.1.

Comparisons between Solids and Surfaces tools

First example – comparing 3D solid and 3D surface models (Fig. 18.2)

The three 3D cubes shown in the upper drawing of Fig. 18.2 have been constructed as follows:

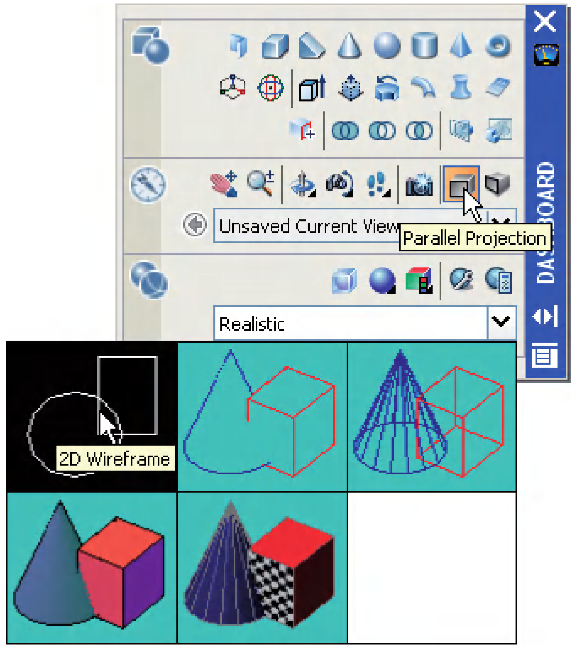


Fig. 18.1 Setting **2D Wireframe** and **Parallel Projection**

Left-hand cube: using the **Line** tool
Central cube: using the **Box** tool from the **Modeling** toolbar
Right-hand cube: a box constructed using the surface command **ai_box**.

The lower three drawings of Fig. 18.2 show the results of calling the **Hide** tool on each of the three upper drawings.

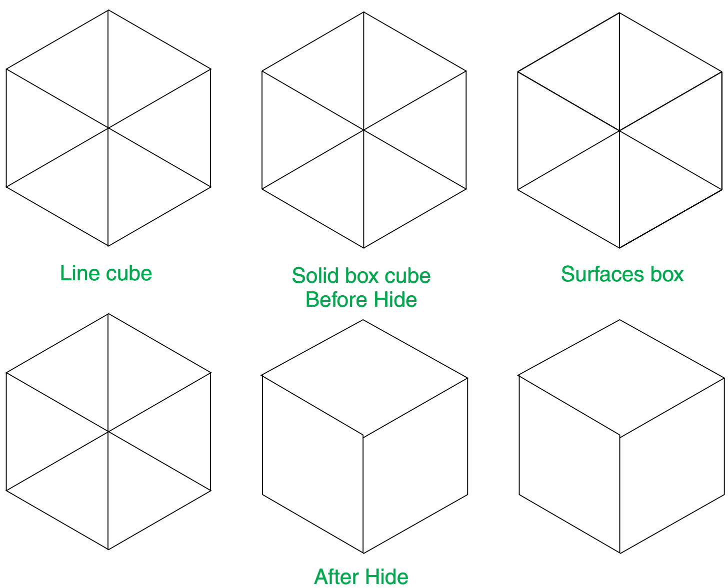


Fig. 18.2 First example – comparing **3D solid** and **3D surface** models

Second example – Surfaces – Cone (Fig. 18.3)

The three cones were all constructed using the surface command **ai_cone**. When using the **Cone** tool, the last prompt line is:

Enter number of segments for surface of cone: *enter* a figure *right-click*

The upper three cones show the number of segments entered in response to this prompt line.

Note that when **Hide** is called, the upper surface of the cones do not have surface meshes and so are open.

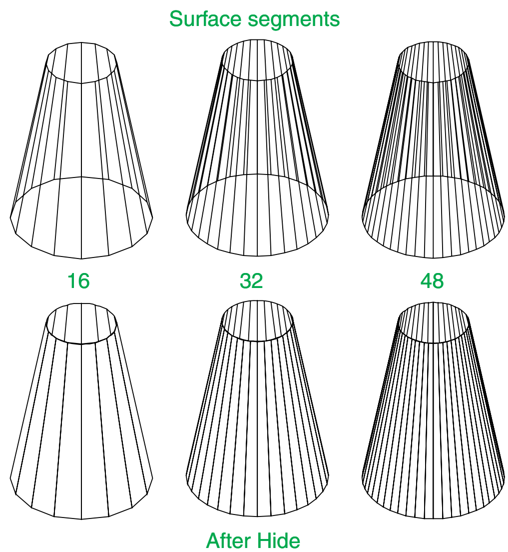


Fig. 18.3 Second example – **Surface** command – **ai_cone**

Third example – Solids tool – Cylinder (Fig. 18.4)

The upper three cylinders in this example were constructed using the **Cylinder** tool from the **Modeling** toolbar.

The lower three drawings show the results of calling **Hide** to the three cylinders in the upper drawings.

Note that the upper surface of the three cylinders hide the rest of the lines behind the top and that all three cylinders assume the same mesh appearance when **Hide** is called.

Fourth example – Surfaces tool – Edgesurf (Fig. 18.5)

When constructing 3D surface models using tools such as the **Edge Surface** tool, both the **Surftab1** and **Surftab2** variables may need resetting.

The upper three drawings of Fig. 18.5 show 3D edgesurf models with a note under each stating the **Surftab** settings; the lower three models show the three surface models after **Hide** has been called.

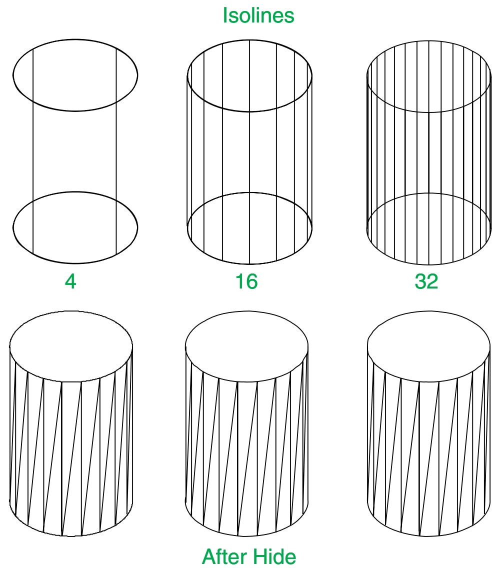


Fig. 18.4 Third example – Solids tool – **Cylinder**

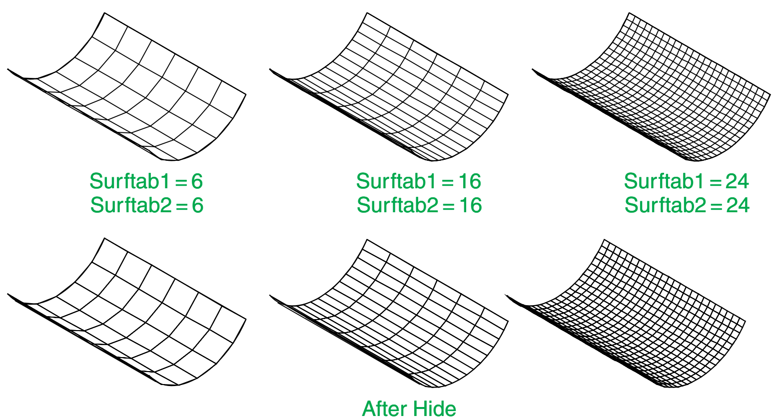


Fig. 18.5 Fourth example – **Surfaces tool – Edgesurf**

The Surface tools

The following examples show the use of many of the tools in the **Surfaces** toolbar.

First example – 3D Surfaces model (Fig. 18.6)

1. Make the layer **Magenta** current.
2. *Enter* **ai_cone** at the command line which then shows:

Command:_ai_cone
Specify centre point for base of cone: *enter* **190,170** *right-click*
Specify radius for base of cone: *enter* **30** *right-click*
Specify radius for top of cone: *enter* **30** *right-click*
Specify height of cone: *enter* **80** *right-click*
Enter number of segments for surface of cone <16>: *right-click*
Command:

3. *Enter* **ai_box** at the command line which then shows:

Command:_ai_box
Specify corner point of box: *enter* **160,140,80** *right-click*
Specify length of box: *enter* **60** *right-click*
Specify width of box: *enter* **60** *right-click*
Specify height of box: *enter* **30** *right-click*
Specify rotation angle of box about the Z axis: *enter* **0** *right-click*
Command:

4. *Enter* **ai_dome** at the command line which then shows:

Command:_ai_dome
Specify centre point of dome: *enter* **190,170,110** *right-click*
Specify radius of dome: *enter* **30** *right-click*
Enter number of longitudinal segments for surface of dome <16>: *right-click*
Enter number of latitudinal segments for surface of dome <8>: *enter* **16** *right-click*
Command:

5. Place the model in the **SW Isometric** view.

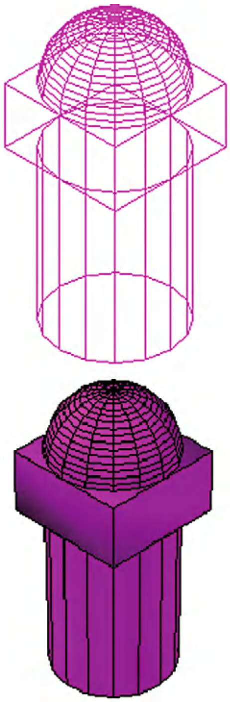


Fig. 18.6 First example – **3D Surfaces model**. The model was constructed on layer **magenta**

The resulting 3D surfaces solid is shown in the upper illustration of Fig. 18.6. The lower illustration of Fig. 18.6 shows the surface model in a **Perspective** setting shaded in **Conceptual** mode.

Note

Try forming a union of the three 3D surface objects by *clicking* the **Union** tool from the **Solids Editing** toolbar.

Command:_union
Select objects: *pick* the cylinder **1 found**
Select objects: *pick* the box **1 found, 2 total**
Select objects: *pick* the dome **1 found, 3 total**
Select objects: *right-click*
At least 2 solids or coplanar regions must be selected.
Command:

It will be seen that the three surface models will not form a union. Test by moving any one of the three surface models.

Second example – Surfaces model (Fig. 18.7)

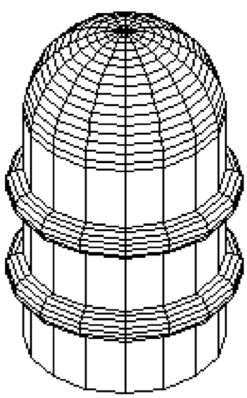


Fig. 18.7 Second example – **Surfaces model**

1. Construct the same cylinder (using the **Cone** tool from **Surfaces**) as for the first example.
2. *Enter* **ai_torus** at the command line which then shows:

Command:_ai_torus
Specify center point of torus: *enter* **190,170,25** *right-click*
Specify radius of torus: *enter* **35** *right-click*
Specify radius of tube: *enter* **5** *right-click*

Enter number of segments around the tube circumference <16>: *right-click*
Enter number of segments around torus circumference <16>: *right-click*
Command:

3. Construct a similar torus at a height of **55** above the **XY** plane.
4. Add a **Dome** of radius **30** at a height of **80** above the **XY** plane.
5. Place the model in the **SW Isometric** view. **Zoom** to **1**.
6. Call **Hide**.

Third example – Surfaces model (Fig. 18.9)

1. Make a new layer of colour **30**.
2. Construct the plines as shown in Fig. 18.8.

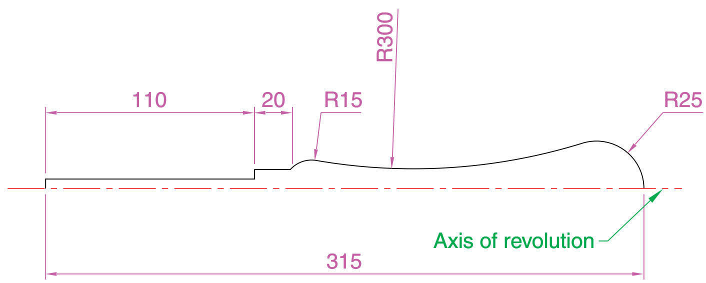


Fig. 18.8 Third example – **Surfaces model** – pline outline

3. Set **SURFTAB1** to **24**.
4. Set **SURFTAB2** to **6**.
5. *Enter* **revsurf** at the command line which then shows:

Command:_revsurf
Current wire frame density: SURFTAB1=24 SURFTAB2=6
Select object to revolve: *pick* the curved pline
Select object that defines the axis of revolution: *pick*
Specify start angle <0>: *right-click*
Specify included angle <360>: *right-click*
Command:

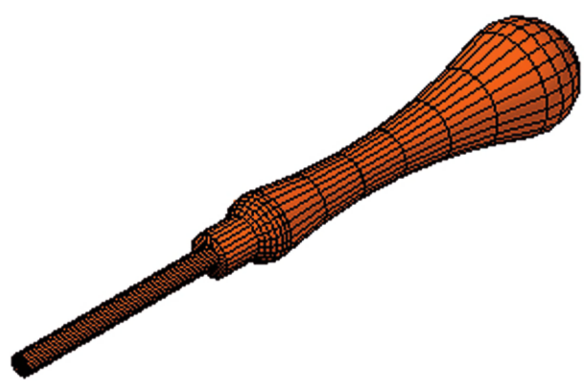


Fig. 18.9 Third example – **Surface model**

6. Erase the axis of revolution pline.
7. Place in a **SW Isometric** view, **Zoom** to **1** and set the **Visual Style** to **Realistic**.

Fourth example – Surfaces model (Fig. 18.10)

1. Set **Surftab1** to **2**.
2. In the **UCS *WORLD*** construct a hexagon of edge length **35**.
3. In the **UCS *FRONT*** and in the centre of the hexagon construct a pline of height **100**.
4. Place the drawing in the **SW Isometric** view.
5. *Enter* **tabsurf** at the command line which then shows:

Command:_tabsurf
Current wire frame density: SURFTAB1=6
Select objects for path curve: *pick* the hexagon
Select object for direction vector: *pick* the pline
Command:

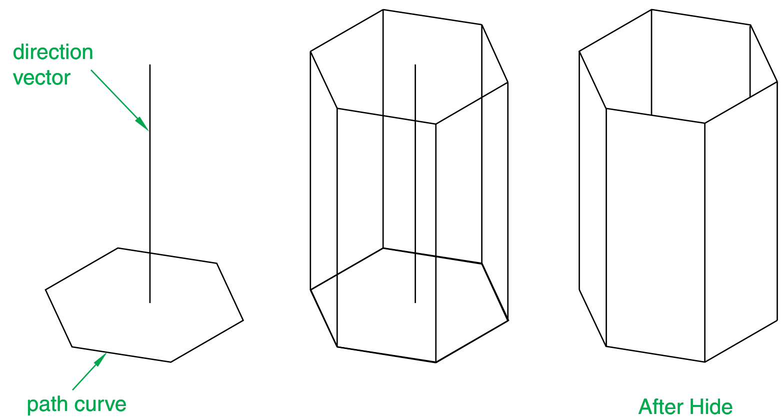


Fig. 18.10 Fourth example – **Surfaces model**

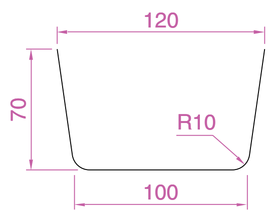


Fig. 18.11 Fifth example – **Surfaces model** – pline outline

Fifth example – Surfaces model (Fig. 18.12)

1. In the **UCS *FRONT*** construct the pline as shown in Fig. 18.11.
2. In the **UCS *WORLD*, Zoom** to **1** and copy the pline to a vertical distance of **120**.
3. Place in the **SW Isometric** view and **Zoom** to **1**.
4. Set **SURFTAB1** to **32**.
5. *Enter* **rulesurf** at the command line which then shows:

Command:_rulesurf
Current wire frame density: SURFTAB1=32
Select first defining curve: *pick* one of the plines
Select second defining curve: *pick* the other pline
Command:

6. Call **Hide**.

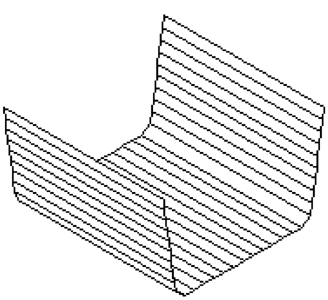


Fig. 18.12 Fifth example – **Surfaces model**

Sixth example – Surfaces model (Fig. 18.15)

1. Place the drawing area in the **UCS *RIGHT*. Zoom** to **1**.
2. Construct the polyline to the sizes and shape as shown in Fig. 18.13.

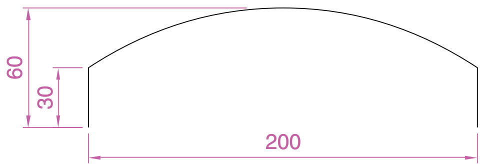


Fig. 18.13 Sixth example – **Surfaces model** – pline outline

3. Place the drawing area in the **UCS *WORLD*. Zoom** to **1**.
4. Copy the pline to the right by **250**.
5. Place the drawing in the **SW Isometric** view and **Zoom** to **1**.
6. With the **Line** tool, draw lines between the ends of the two plines using the **endpoint** osnap (Fig. 18.14). Note that if polylines are drawn they will not be accurate at this stage.
7. Set **SURFTAB1** to **32** and **SURFTAB2** to **64**.
8. *Enter* **edgesurf** at the command line which then shows:

Command:_edgesurf
Current wire frame density: SURFTAB1=32 SURFTAB2=64
Select object 1 for surface edge: *pick* one of the lines (or plines)
Select object 1 for surface edge: *pick* the next adjacent line (or pline)
Select object 1 for surface edge: *pick* the next adjacent line (or pline)
Select object 1 for surface edge: *pick* the last line (or pline)
Command:

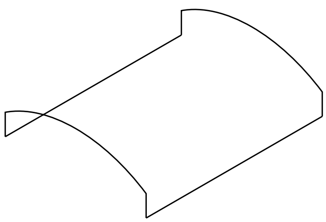


Fig. 18.14 Sixth example – **Surfaces model** – adding lines joining the outlines

9. Call **Hide**.

The result is shown in Fig. 18.15.

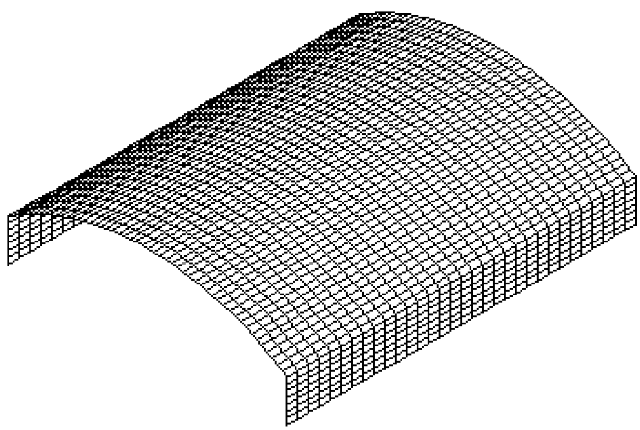


Fig. 18.15 Sixth example – **Surfaces model**

Rendering of 3D Surface models

As with 3D solid model drawings, 3D surface model drawings can be rendered if thought necessary. By adding lights and materials to the surfaces, surface models can be rendered as effectively as can 3D solid models.

Revision notes

1. To follow the construction of the examples given in this chapter the screen should be set to **2D Wireframe** in **Parallel Projection** mode.
2. The density of 3D surface meshes of surface models are controlled by the two set variables **SURFTAB1** and **SURFTAB2**.
3. The **Modeling** tools **Union, Subtract** and **Intersect** (the Boolean operators) cannot be used with surface models.
4. When constructing a surface 3D cylinder use the **Cone** tool command.
5. Surfaces formed with the **Ruled Surface** tool require two outlines from which the surface can be obtained.
6. Surfaces formed with the **Edge Surface** tool require four edges from which to form the surface. The edges must meet each other at their ends for the surface to form.
7. 3D surface models can be rendered using the same methods as are used when rendering 3D solid models.

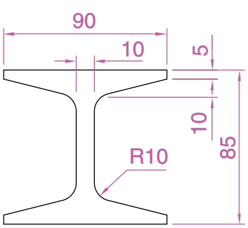


Fig. 18.16 Exercise 1 – the outline from which the surface model is formed

Exercises

1. In the **UCS *WORLD*** construct the polyline outline as shown in Fig. 18.16.

   With the **Tabulated Surface** tool create a girder of length **200** from the outline (left-hand drawing of Fig. 18.17). Using the **Edge Surface**

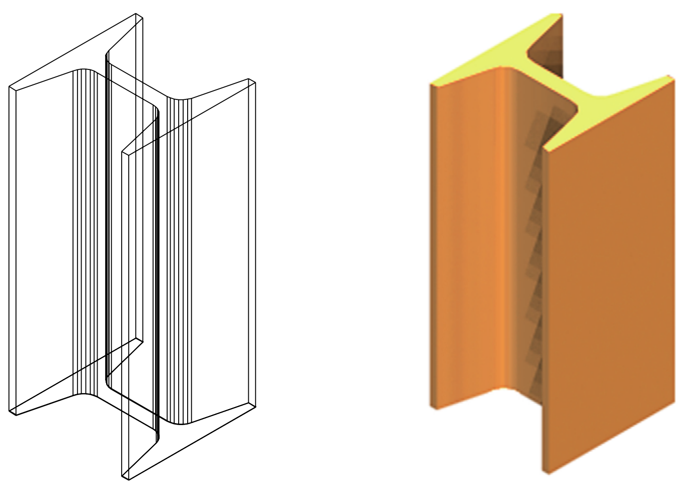


Fig. 18.17 Exercise 1 – the **Tabulated Surface** and the rendering

tool, construct a surface to cover the top end of the girder. Render the girder after adding suitable lights and a material.

2. Using the **Polyline** tool construct arcs as shown in Fig. 18.18. Using the **UCS 3point** option construct semicircles joining both ends of the arcs as shown in Fig. 18.19.

   With the **Edge Surface** tool and with the **SURFTAB** variables set to suitable sizes, construct a surface from the four arcs (Fig. 18.20).

   Using the **Mirror** tool, mirror the surface twice to obtain the hook shape. Render the hook after adding lighting and a material (Fig. 18.21).

3. Construct a surface of revolution from the polyline outline given in the left-hand drawing of Fig. 18.22. Add lighting and a suitable material and render.

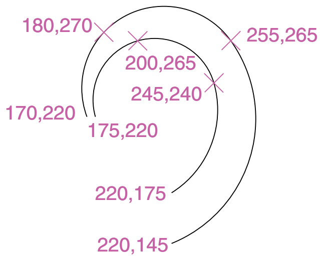


Fig. 18.18 Exercise 2 – the arcs on which the exercise is based

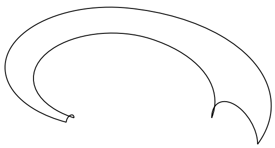


Fig. 18.19 Exercise 2 – semi-circles as end of arcs

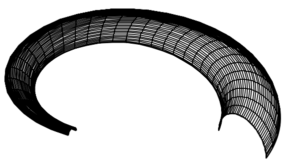


Fig. 18.20 Exercise 2 – a surface

Fig. 18.21 Exercise 2

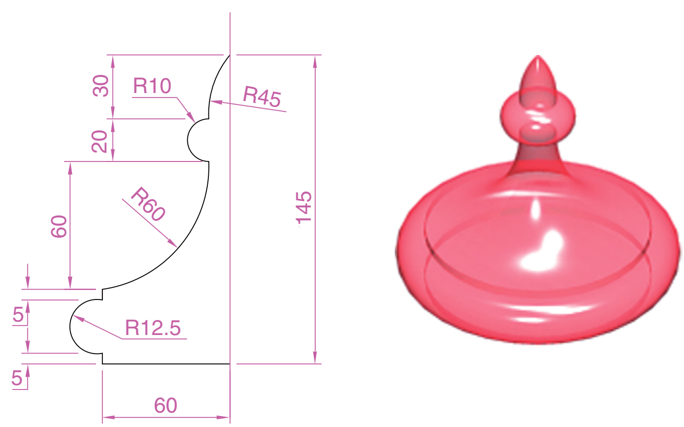


Fig. 18.22 Exercise 3

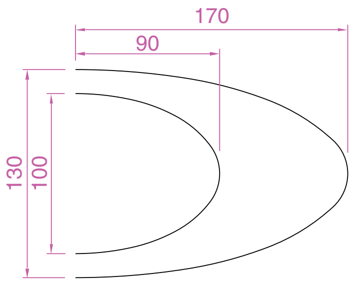


Fig. 18.23 Exercise 4 – the two semi-ellipses

4. In the **UCS *WORLD*** plane construct two semi-ellipses to the sizes as given in Fig. 18.23. Place in the **UCS *FRONT*** plane and move the smaller semi-ellipse vertically upwards by **200** units.

   Using the **Ruled Surface** tool form a surface between the two semi-ellipses. Add suitable lighting and a suitable material and render the surface (Fig. 18.24).
5. Using the **Pyramid** and **Box** tools from the **Surfaces** toolbar construct the 3D model in Fig. 18.25. Then place the model in the **SW Isometric** view and using the **Realistic** shading from the **3D Orbit** *right-click* menu, hide lines behind the front faces (Fig. 18.26).
6. Working to the sizes given in the front view in Fig. 18.27, construct the 3D surface model shown in the rendering in Fig. 18.28. ***(Note:** the base is **60** deep).

Fig. 18.24 Exercise 4

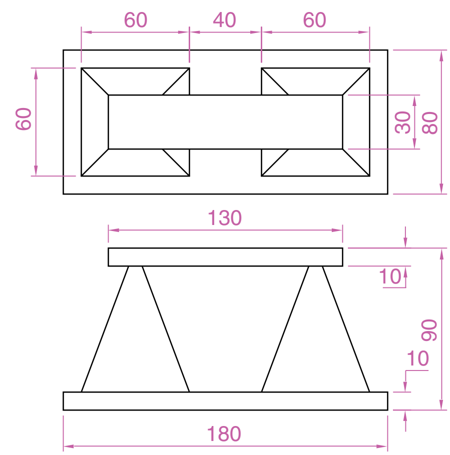


Fig. 18.25 Exercise 5 – sizes of the parts of the model

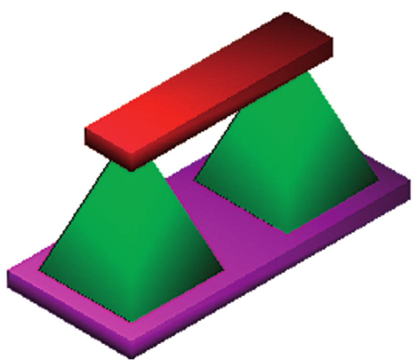


Fig. 18.26 Exercise 5

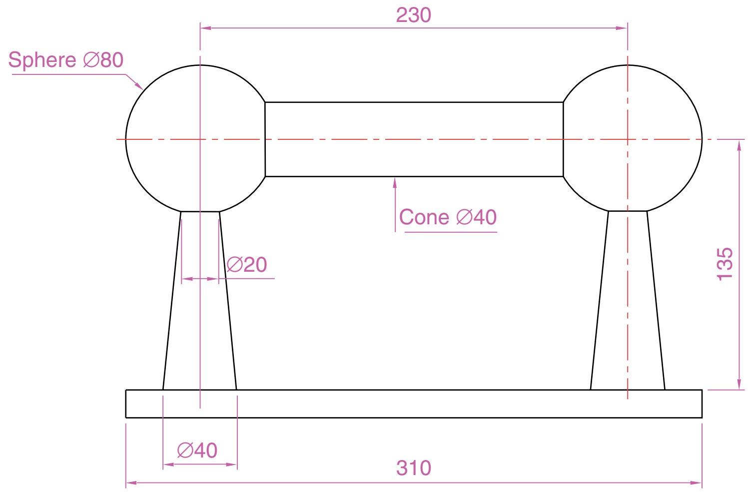


Fig. 18.27 Exercise 6 – front view

Fig. 18.28 Exercise 6

CHAPTER 19

Editing 3D solid models

Aims of this chapter

1. To introduce the use of tools from the **Solids Editing** toolbar.
2. To give examples of the use of the tools from the **Solids Editing** toolbar.
3. To show examples of a variety of 3D solid and 3D surface models.

The Solids Editing tools

Examples of the results of using some of the tools from the **Solids Editing** toolbar are shown in this chapter. These tools are of value if the design of a 3D solid model requires to be changed (edited), although some have a value in constructing solids which cannot easily be constructed using other tools. The **Solid Editing** tools have no effect when used on 3D surface models.

First example – Extrude faces tool (Fig. 19.3)

1. Set **ISOLINES to** 24.
2. In the **UCS *RIGHT*** plane construct a cylinder of radius **30** and height **30** (Fig. 19.1).

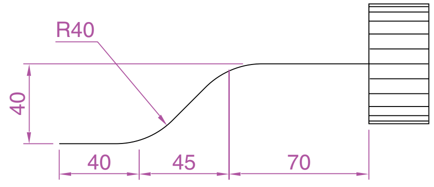


Fig. 19.1 First example – **Extrude faces tool** – first stages

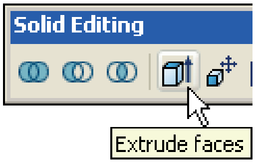


Fig. 19.2 The **Extrude faces** tool icon

3. In the **UCS *FRONT*** view construct the pline in Fig. 19.1. Mirror the pline to the other end of the cylinder.
4. In the **UCS *WORLD*** move the pline to lie central to the cylinder.
5. Place the screen in the **SW Isometric** view.
6. *Click* the **Extrude faces** tool icon in the **Solids Editing** toolbar (Fig. 19.2). The command line shows:

Command:_solidedit
Solids editing automatic checking: SOLIDCHECK=1

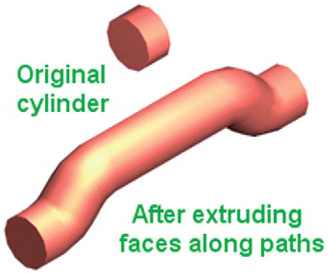


Fig. 19.3 First example – **Extrude faces** tool

Enter a solids editing option [Face/Edge/Body/Undo/eXit]<eXit>: _face
Enter a face editing option [Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>:_extrude
Select faces or [Undo/Remove]: <Snap off> *pick* a face **2 faces found**.
Select faces or [Undo/Remove/ALL]: *enter* **r** *right-click*
Remove faces or [Undo/Add/ALL]: 2 faces found, 1 removed.
Remove faces or [Undo/Add/ALL]: *right-click*
Specify height of extrusion or [Path]: *enter* **p** *right-click*
Select extrusion path: *pick* the path
Path was moved to the center of the profile.
Solid validation started.
Solid validation completed.

7. Repeat the operation using the plane at the other end of the cylinder.
8. Render the resulting edited 3D model (Fig. 19.3).

Notes

1. Note the prompt line which includes the statement **SOLIDCHECK=1**. If the variable **SOLIDCHECK** is set on (to **1**) the prompt lines include the lines **SOLIDCHECK=1, Solid validation started** and **Solid validation completed**. If set to **0** these two lines do not show.
2. When a face is *picked* other faces become highlighted; using the **Remove** option of the line **Select faces or [Undo/Remove/ALL]** allows faces which are not to be extruded to be removed from the operation of the tool.

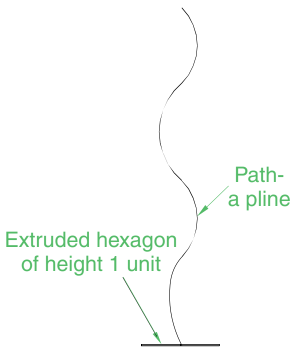


Fig. 19.4 Second example – **Extrude faces** tool – pline for path

Second example – Extrude faces tool

1. Construct a hexagonal extrusion just **1** unit high in the **UCS *WORLD*** plane.
2. Change to the **UCS *FRONT*** plane and construct the curved pline in Fig. 19.4.
3. Back in the **UCS *WORLD*** move the pline to lie central to the extrusion.
4. Place in the **SW Isometric** view and extrude the top face of the extrusion along the path of the curved pline.
5. Add lighting and a material to the model and render (Fig. 19.5).

Fig. 19.5 Second example – **Extrude faces** tool

Note

This example shows that a face of a 3D solid model can be extruded along any suitable path curve. If the polygon on which the extrusion is based had been turned into a region, no extrusion could have taken place. The polygon had to be extruded to give a face to a 3D solid.

Third example – Move faces tool (Fig. 19.7)

1. Construct the 3D solid drawing shown in the left-hand drawing of Fig. 19.7 from three boxes which have been merged using the **Union** tool.
2. *Click* on the **Move faces** tool in the **Solids Editing** toolbar (Fig. 19.6).

The command line shows:

Command:_solidedit
[prompts]_face
Enter a face editing option
[prompts]:_move
Select faces or [Undo/Remove]: *pick* face **1 face found**
Select faces or [Undo/Remove/ALL]: *right-click*
Specify a base point or displacement: *pick*
Specify a second point of displacement: *pick*
[further prompts]:

And the *picked* face is moved – right-hand drawing of Fig. 19.7.

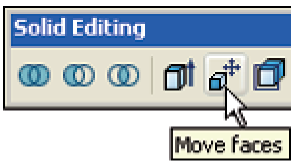


Fig. 19.6 The **Move faces** tool icon

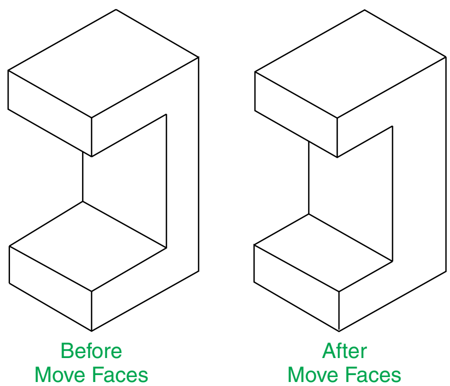


Fig. 19.7 Third example – **Move faces** tool

Fourth example – Offset faces (Fig. 19.9)

1. Construct the 3D solid drawing shown in the left-hand drawing of Fig. 19.9 from a hexagonal extrusion and a cylinder which have been merged using the **Union** tool.
2. *Click* on the **Offset faces** tool icon in the **Solids Editing** toolbar (Fig. 19.8). The command line shows:

Command:_solidedit
[prompts]:_face
[prompts]
[prompts]:_offset
Select faces or [Undo/Remove]: *pick* the bottom faces of the 3D model **2 faces found**.
Select faces or [Undo/Remove/All]: *enter* **r** *right-click*

Fig. 19.8 The **Offset faces** tool icon

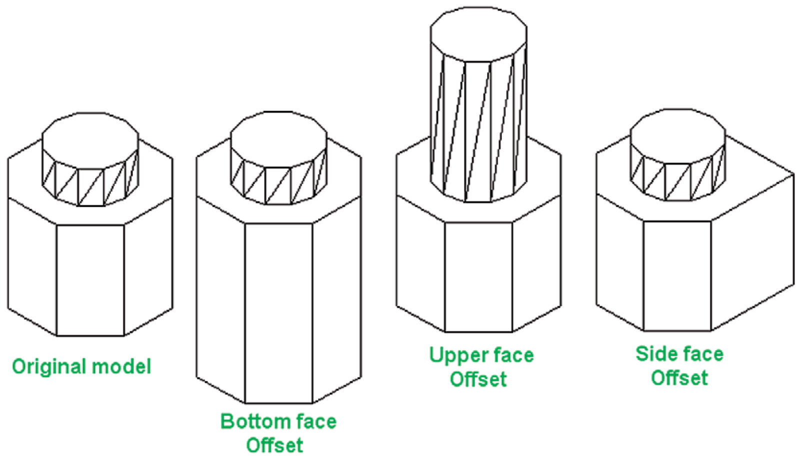


Fig. 19.9 Third example – **Offset faces** tool

Select faces or [Undo/Remove/All]: *pick* highlighted faces other than the bottom face **2 faces found, 1 removed**
Select faces or [Undo/Remove/All]: *right-click*
Specify the offset distance: *enter* **30** *right-click*

3. Repeat by offsetting the upper face of the cylinder by **50** and the right-hand face of the lower extrusion by **15**.

Fifth example – Taper faces tool (Fig. 19.10)

1. Construct the 3D model as in the left-hand drawing of Fig. 19.10. Place in the **SW Isometric** view.

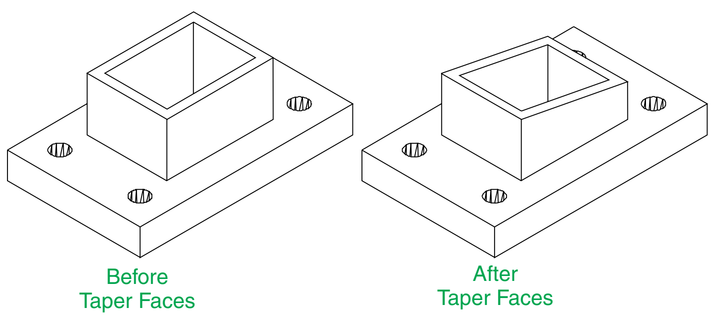


Fig. 19.10 Fifth example – **Taper faces** tool

2. Call **Taper faces**. The command line shows:

Command:_solidedit
[prompts]:_face
[prompts]
[prompts]:_taper
Select faces or [Undo/Remove]: *pick* the upper face of the base **2 faces found**
Select faces or [Undo/Remove/All]: *enter* **r** *right-click*

Select faces or [Undo/Remove/All]: *pick* highlighted faces other than the upper face **2 faces found, 1 removed**
Select faces or [Undo/Remove/All]: *right-click*
Specify the base point: *pick* a point on left-hand edge of the face
Specify another point along the axis of tapering: *pick* a point on the right-hand edge of the face
Specify the taper angle: *enter* **10** *right-click*

And the selected face tapers as indicated in the right-hand drawing (Fig. 19.10).

Sixth example – Copy faces tool (Fig. 19.13)

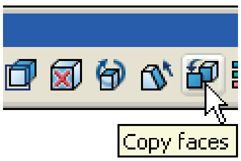


Fig. 19.11 The **Copy faces** tool icon from the **Solids Editing** toolbar

1. Construct a 3D model to the sizes as given in Fig. 19.12.
2. *Click* on the **Copy faces** tool in the **Solids Editing** toolbar (Fig. 19.11).

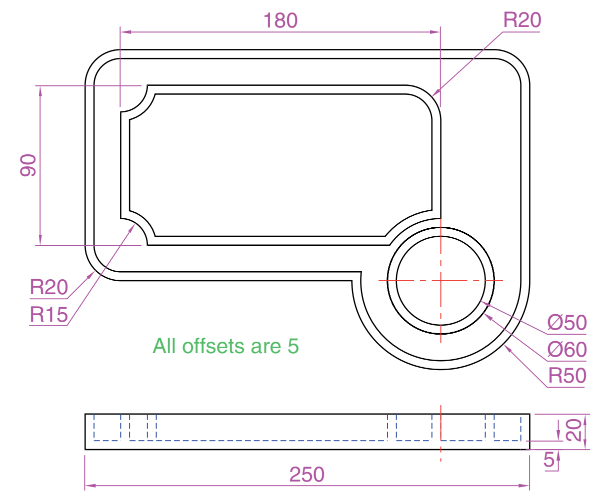


Fig. 19.12 Sixth example – **Copy Faces** tool – details of the 3D solid model

The command line shows:

Command:_solidedit
[prompts]:_face
[prompts]
[prompts]:_copy
Select faces or [Undo/Remove]: *pick* the upper face of the solid model **2 faces found**
Select faces or [Undo/Remove/All]: *enter* **r** *right-click*
Select faces or [Undo/Remove/All]: *pick* highlighted face not to be copied **2 faces found, 1 removed**
Select faces or [Undo/Remove/All]: *right-click*
Specify a base point or displacement: *pick* anywhere on the highlighted face
Specify a second point of displacement: *pick* a point some 50 units above the face

Fig. 19.13 Sixth example – **Copy faces** tool

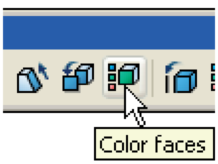


Fig. 19.14 The **Color Faces** tool icon from the **Solids Editing** toolbar

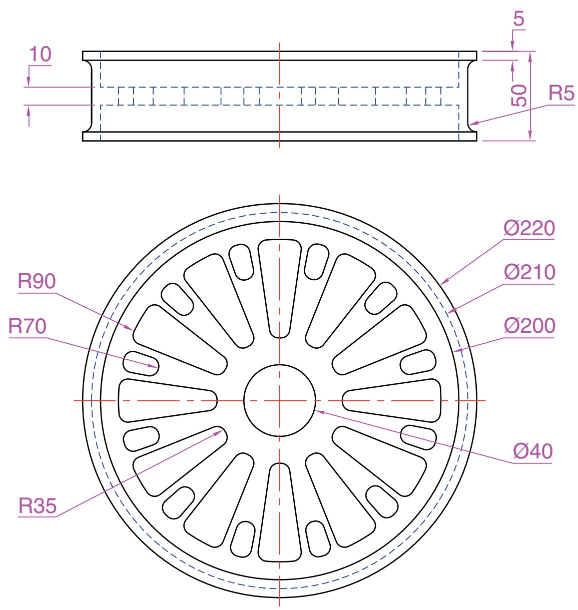


Fig. 19.15 Seventh example – **Color faces** tool – details of the 3D model

3. Add lights and a material to the 3D model and its copied face and render (Fig. 19.13).

Seventh example – Color faces tool (Fig. 19.16)

1. Construct a 3D model of the wheel to the sizes as shown in Fig. 19.15.
2. *Click* the **Color faces** tool icon in the **Solids Editing** toolbar (Fig. 19.14). The command line shows:

Command:_solidedit
[prompts]:_face
[prompts]
[prompts]:_color
Select faces or [Undo/Remove]: *pick* the inner face of the wheel **2 faces found**
Select faces or [Undo/Remove/All]: *enter* **r** *right-click*
Select faces or [Undo/Remove/All]: *pick* highlighted faces other than the required face **2 faces found, 1 removed**
Enter new color <ByLayer>: *enter* **1** (which is red) *right-click*

3. Add lights and a material to the edited 3D model and render (Fig. 19.16).

Examples of more 3D models

These 3D models can be constructed in the **acadiso3D.dwt** screen. The descriptions of the stages needed to construct these 3D models have been reduced from those given in earlier pages, in the hope that readers have already acquired a reasonable skill in the construction of such drawings.

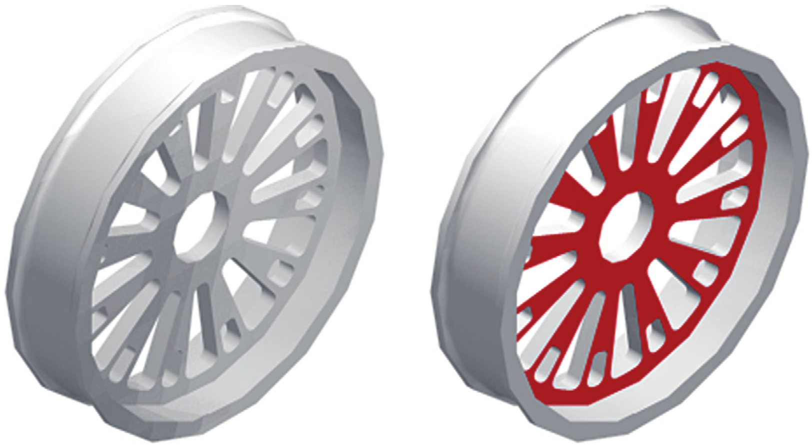


Fig. 19.16 Eighth example – **Color faces** tool

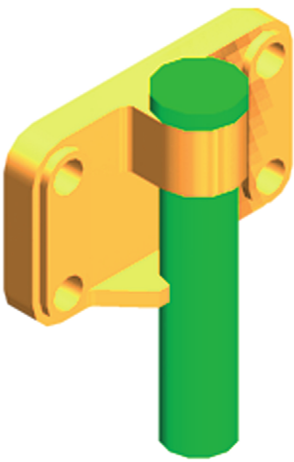


Fig. 19.17 First example of 3D models

First example (Fig. 19.17)

1. ***FRONT*** plane. Construct the three extrusions for the back panel and the two extruding panels to the details given in Fig. 19.18.
2. ***WORLD*** plane. Move the two panels to the front of the body and union the three extrusions. Construct the extrusions for the projecting parts holding the pin.
3. ***FRONT*** plane. Move the two extrusions into position and union them to the back.
4. ***WORLD*** plane. Construct two cylinders for the pin and its head.
5. ***FRONT*** plane. Move the head to the pin and union the two cylinders.
6. ***WORLD*** plane. Move the pin into its position in the holder. Add lights and materials.
7. **SW Isometric** view. Render. Adjust lighting and materials as necessary (Fig. 19.17).

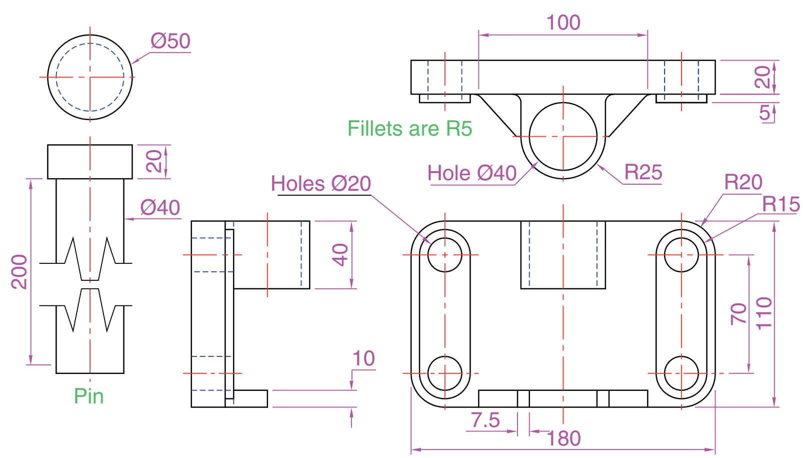


Fig. 19.18 First example 3D models – details of sizes and shapes

8. *Enter* **re** to regenerate the model back to its 3D wireframe state.
9. Press and hold the **Ctrl** key. The **Selecting Subobjects on Solids** warning window (Fig. 19.19) appears. *Click* its **Close** button. Then

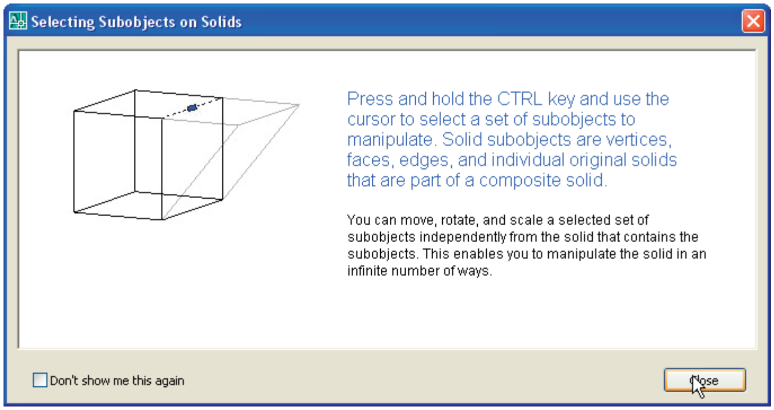


Fig. 19.19 The **Selecting Subobjects on Solids** warning window

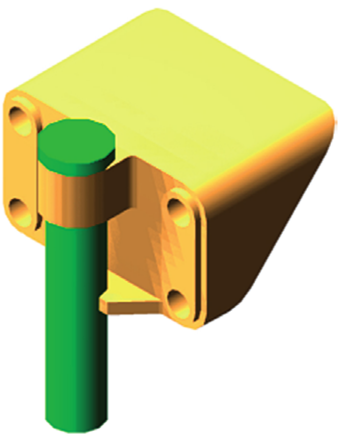


Fig. 19.20 The stretched first example

while still holding the **Ctrl** key down *pick* the top centre point of the back of the holder. The command line shows:

Command:
****STRETCH****
Specify stretch point or [Base point/Undo/eXit]: *pick* a point to the rear of the holder
Command:

The 3D model can again be rendered and then appears as shown in Fig. 19.20.

Note

When using this method of holding down the **Ctrl** key when a 3D model is on screen, and if the **Return** key of the keyboard is pressed repeatedly, the command line shows:

****MOVE****
Specify move point or [Base point/Undo/eXit]:
****ROTATE****
Specify rotation angle or [Base point/Undo/Reference/eXit]:
****SCALE****
Specify scale factor or [Base point/Undo/Reference/eXit]:
****MIRROR****
Specify second point or [Base point/Undo/eXit]:

This allows the operator to use any of these modify commands on the 3D model.

Second example (Fig. 19.22)

1. ***WORLD*** (Fig. 19.21). Construct polyline outlines for the body extrusion and the solids of revolution for the two end parts. Extrude the body and subtract its hole and using the **Revolve** tool form the two end solids of revolution.

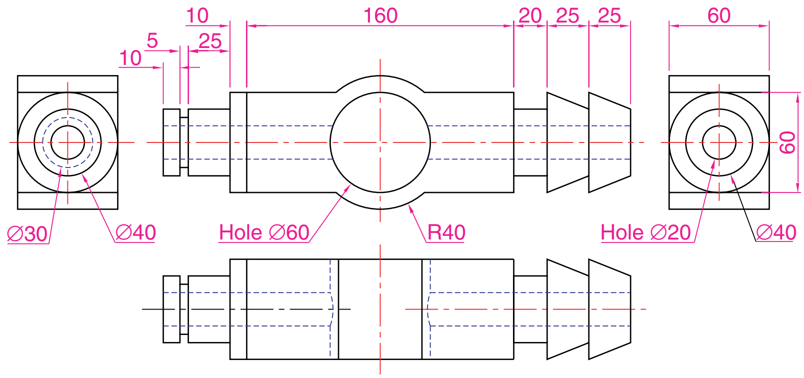


Fig. 19.21 Second example – dimensions

2. ***RIGHT***. Move the two solids of revolution into their correct positions relative to the body and union the three parts. Construct a cylinder for the hole through the model.
3. ***FRONT***. Move the cylinder to its correct position and subtract from the model.
4. ***WORLD***. Add lighting and a material.
5. Render (Fig. 19.22).

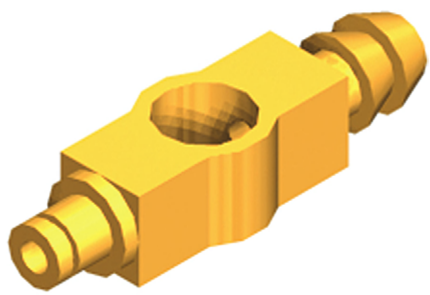


Fig. 19.22 Second example of 3D models

Third example (Fig. 19.24)

1. ***FRONT***. Construct the three plines needed for the extrusions of each part of the model (details in Fig. 19.23). Extrude to the given heights. Subtract the hole from the **20** high extrusion.
2. ***WORLD***. Move the **60** extrusion and the **10** extrusion into their correct positions relative to the **20** extrusion. With **Union** form a single 3D model from the three extrusions.

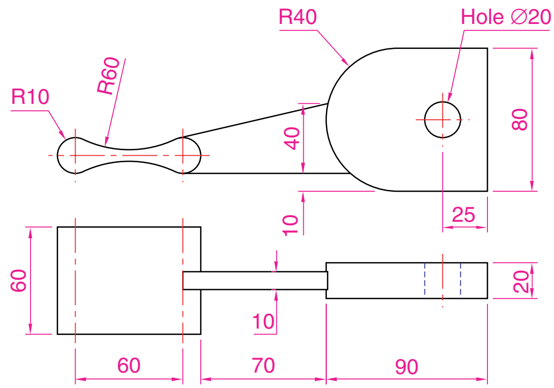


Fig. 19.23 Third example of 3D models – details of shapes and sizes

3. Add suitable lighting and a material to the model.
4. **SW Isometric**. Render (Fig. 19.24).

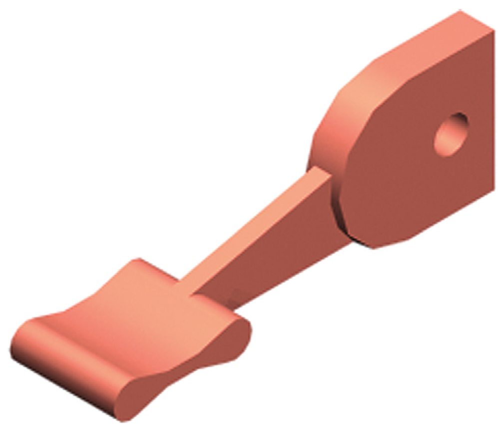


Fig. 19.24 Third example of 3D models

Fourth example (Fig. 19.25)

1. ***FRONT***. Construct the polyline – left-hand drawing of Fig. 19.25.
2. With the **Revsurf** tool form a surface of revolution from the pline.
3. ***WORLD***. Add suitable lighting and a coloured glasslike material.
4. **SW Isometric**. Render (right-hand illustration of Fig. 19.25).

Exercises

1. Construct suitable polylines to sizes of your own discretion in order to form the two surfaces to form the box shape shown in Fig. 19.26 with the aid of the **Rulesurf** tool. Add lighting and a material and render the surfaces so formed.

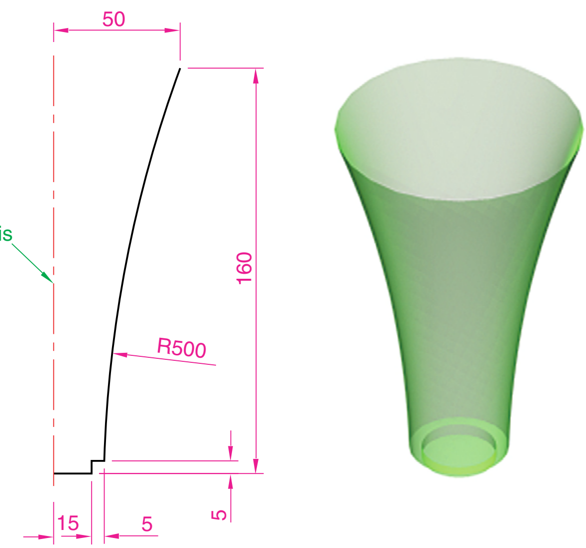


Fig. 19.25 Fourth example of 3D models

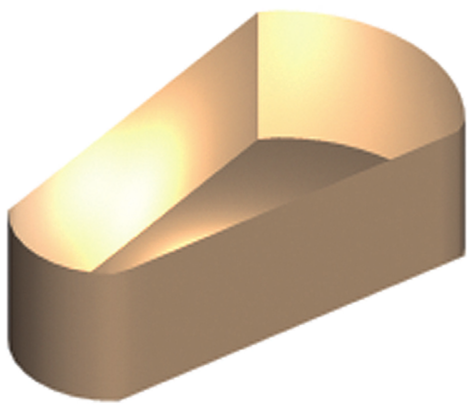


Fig. 19.26 Exercise 1 – first part

Construct another three **Edgesurf** surfaces to form a lid for the box. Place the surface in a position above the box, add a material and render (Fig. 19.27).

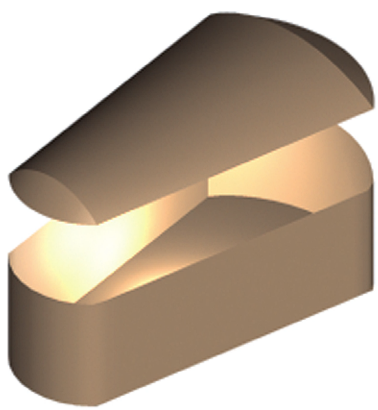


Fig. 19.27 Exercise 1 – second part

2. Working to the dimensions given in the orthographic projections of the three parts of the 3D model in Fig. 19.29, construct the assembled part as shown in the rendered 3D model in Fig. 19.28.

   Add suitable lighting and materials, place in one of the isometric viewing positions and render the model.

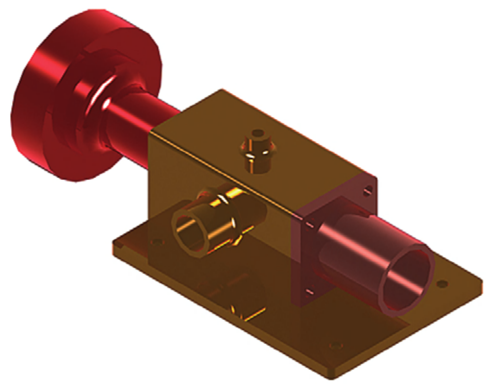


Fig. 19.28 Exercise 2

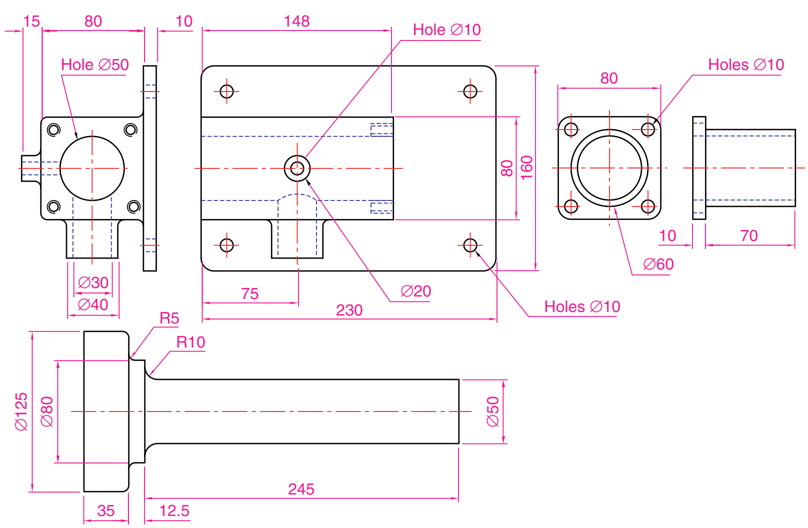


Fig. 19.29 Exercise 2 – details of shapes and sizes

3. Construct the 3D model shown in the rendering in Fig. 19.30 from the details given in the parts drawing in Fig. 19.31.

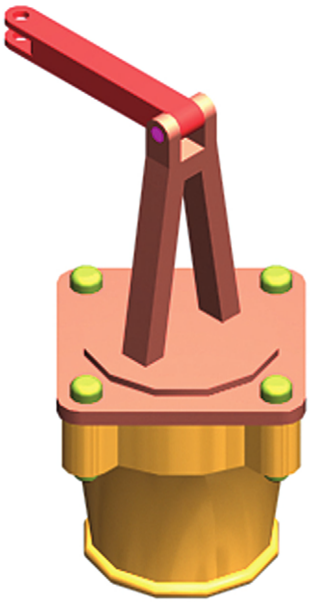


Fig. 19.30 Exercise 3

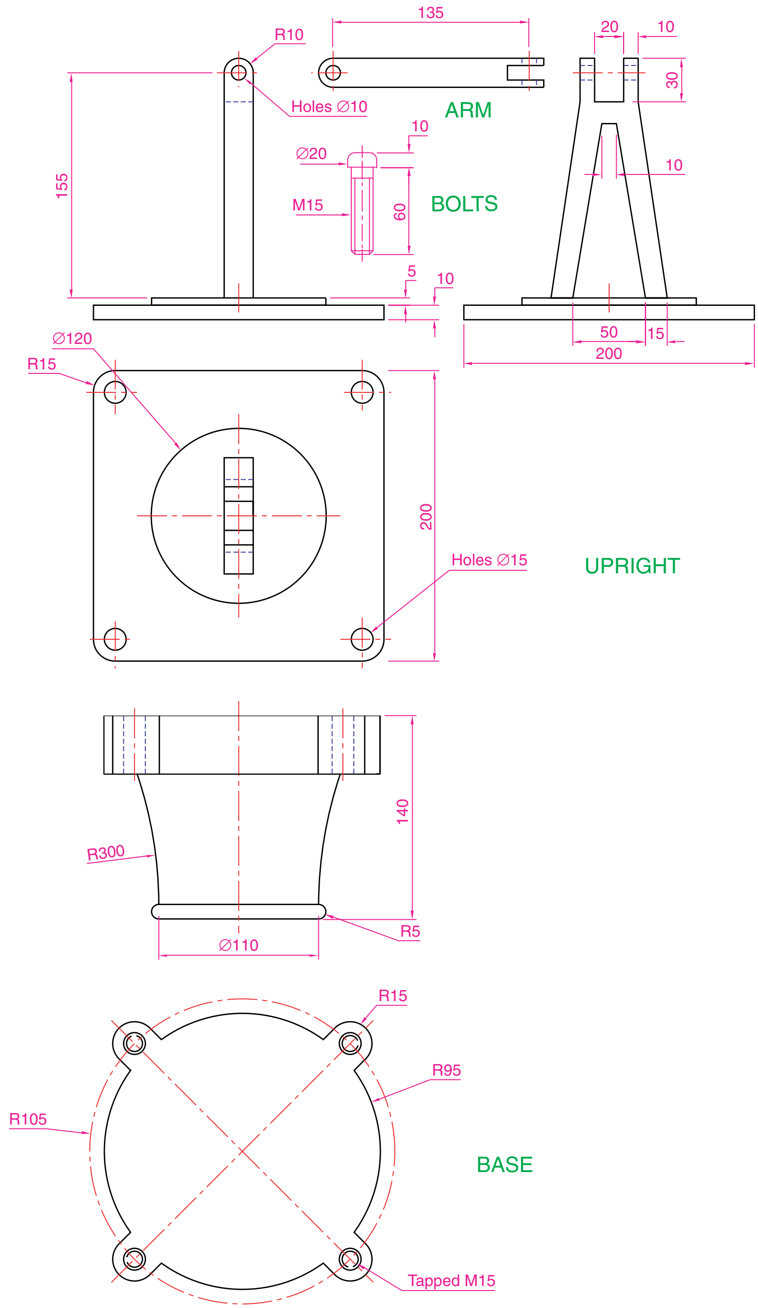


Fig. 19.31 Exercise 3 – the parts drawing

4. A more difficult exercise.
A rendered 3D model of the parts of an assembly are shown in Fig. 19.35.

Working to the details given in the three orthographic projections in Figs 19.32, 19.33 and 19.34, construct the two parts of the 3D model, place them in suitable positions relative to each other, add lighting and materials and render the model.

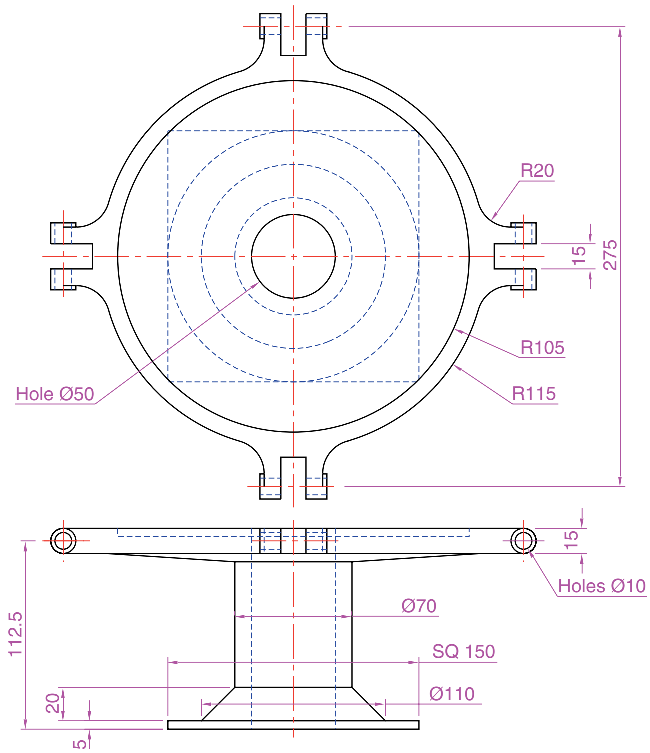


Fig. 19.32 Exercise 4 – first orthographic projection

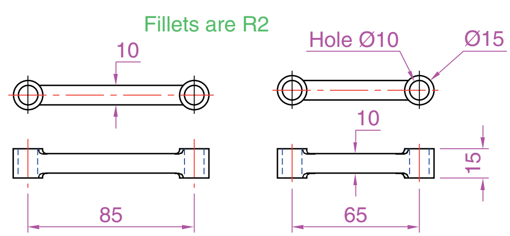


Fig. 19.33 Exercise 4 – second orthographic projection

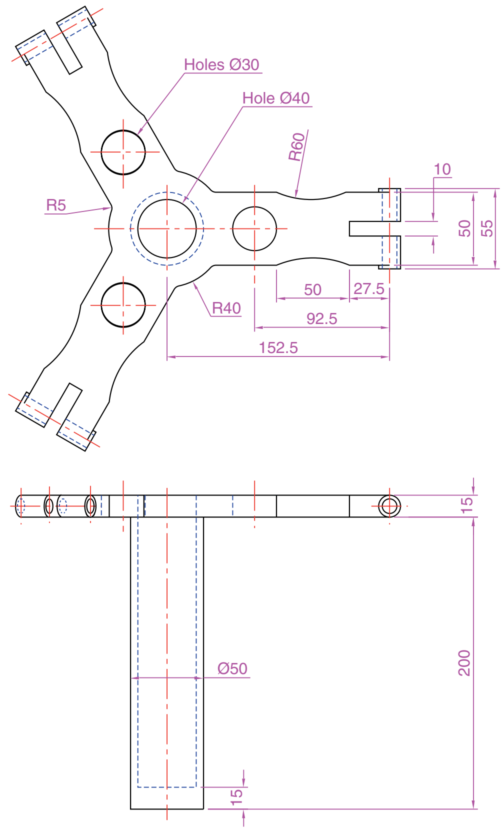


Fig. 19.34 Exercise 4 – third orthographic projection

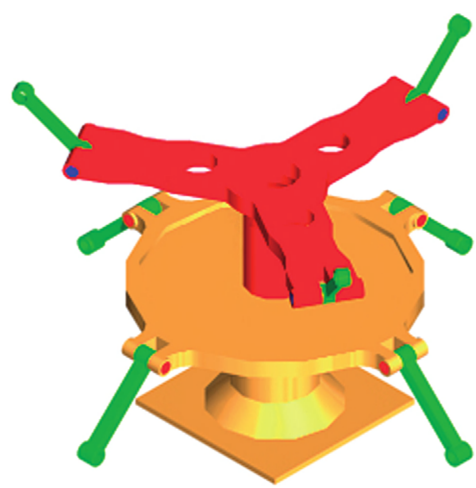


Fig. 19.35 Exercise 4

CHAPTER 20

Other features of 3D modelling

Aims of this chapter

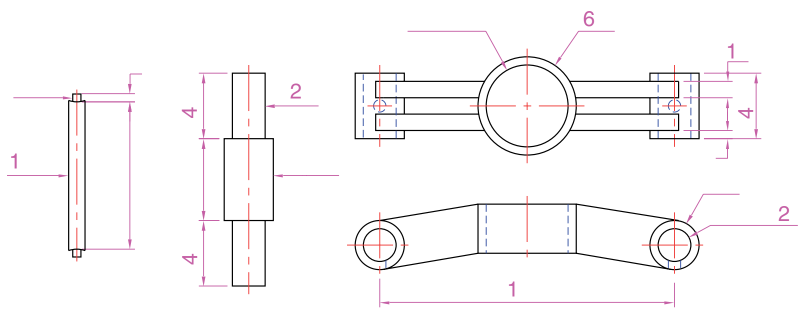
1. To give a further example of placing raster images in an AutoCAD drawing.
2. To give examples of methods of printing or plotting not given in previous chapters.
3. To give examples of polygonal viewports.

Raster images in AutoCAD drawings

Example – Raster image in a drawing (Fig. 20.5)

This example shows the raster file **12.bmp** of the 3D model constructed to the details given in the drawing in Fig. 20.1.

Fig. 20.1 Example – **Raster image in a drawing** – details



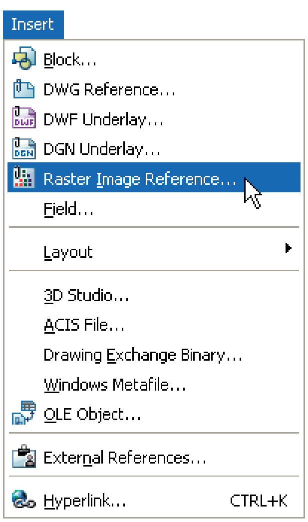


Fig. 20.2 Selecting **Raster Image Reference** . . . from the **Insert** drop-down menu

Raster images are graphics images such as those taken from files ending with the file extensions ***.bmp**; ***.pcx**; ***.tif** and the like. The types of graphics files which can be inserted into AutoCAD drawings can be seen by first *clicking* on **Raster Image Reference** . . . in the **Insert** drop-down menu (Fig. 20.2), which brings the **Select Image File** dialog (Fig. 20.3) on screen. In the dialog *click* the arrow to the right of the **Files of type** field and the popup list appears which lists the types of graphics files which can be inserted into AutoCAD drawings. Such graphics files can be used to describe in 3D the details shown in 2D by a technical drawing.

1. Construct the 3D model to the shapes and sizes given in Fig. 20.1, working in four layers each of a different colour.

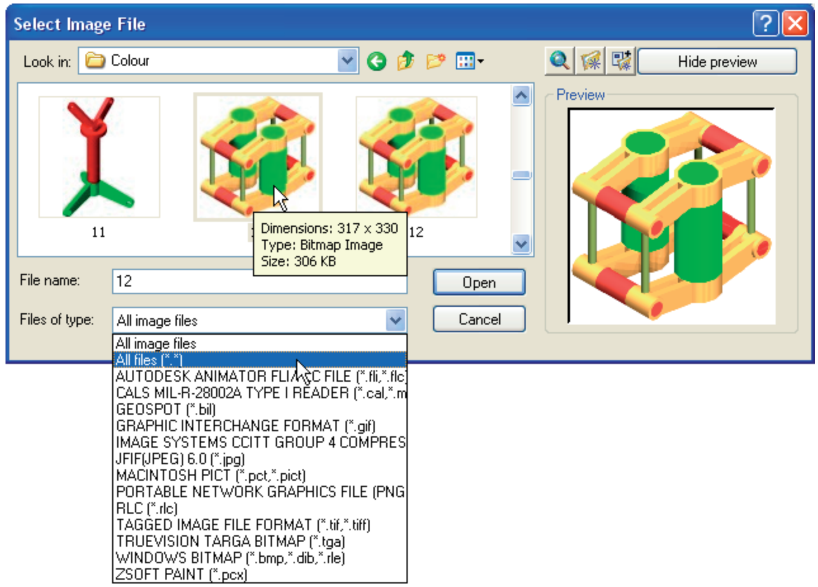


Fig. 20.3 The **Select Image File** dialog

2. Place in the **SW Isometric** view.
3. Shade the 3D model using **Realistic** shading from **Visual Styles**.
4. **Zoom** the shaded model to a suitable size and press the **Print Scr** key of the keyboard.
5. Open the Windows **Paint** application and *click* **Edit** in the menu bar, followed by another *click* on **Paste** in the drop-down menu. The whole AutoCAD screen which includes the shaded 3D assembled model appears.
6. *Click* the **Select** tool icon in the toolbar of **Paint** and window the 3D model. Then *click* **Copy** in the **Edit** drop-down menu.
7. *Click* **New** in the **File** drop-down menu, followed by a *click* on **No** in the warning window which appears.
8. *Click* **Paste** in the **Edit** drop-down menu. The shaded 3D model appears. *Click* **Save As** . . . from the **File** drop-down menu and save the bitmap to a suitable file name – in this example – **12.bmp**.
9. Open the orthographic projection drawing in AutoCAD.
10. Open the **Select Image File** dialog and from the **Look in** field select the raster file **12.bmp** from the file list (Fig. 20.3). Another dialog (**Image**) opens (Fig. 20.4) showing the name of the raster image file. *Click* the **OK** button of the dialog and a series of prompts appear at the command line requesting position and scale of the image. *Enter* appropriate responses to these prompts and the image appears in position in the orthographic drawing (Fig. 20.5).

Notes

1. It will normally be necessary to *enter* a scale in response to the prompt lines otherwise the raster image may appear very small on screen. If it does it can be zoomed anyway.

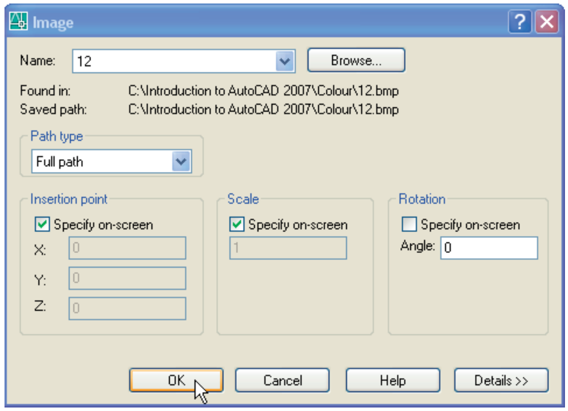


Fig. 20.4 Example – **Raster image in a drawing** – the **Image** dialog

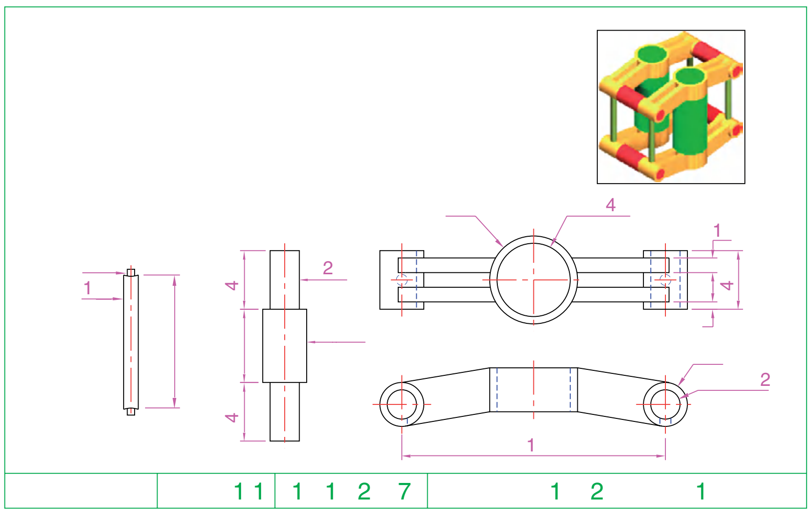


Fig. 20.5 Example – **Raster image in a drawing**

2. Place the image in position in the drawing area. In Fig. 20.5 the orthographic projections have been placed within a margin and a title block has been added.

Printing/Plotting

Hardcopy (prints or plots on paper) from a variety of AutoCAD drawings of 3D models can be obtained. Some of this variety has already been shown on pages 259–260 in Chapter 16.

First example – Printing/Plotting (Fig. 20.8)

If an attempt is made to print a multiple viewport screen with all viewport drawings appearing in the plot, only the current viewport will be printed. To print or plot all viewports:

1. Make sure **UCSFOLLOW** is set off (to **1**).
2. Open a four-viewport screen of the assembled 3D model shown in the first example (page 306).
3. Make a new layer **vports** of colour yellow. Make this layer current.
4. *Right-click* on the **Layout1** tab and *click* **Rename** in the menu which appears. Then in the **Rename Layout** dialog which comes to screen, *enter* a new name in the **Name** field. *Click* the **OK** button and the **Layout1** tab is renamed (Fig. 20.6).

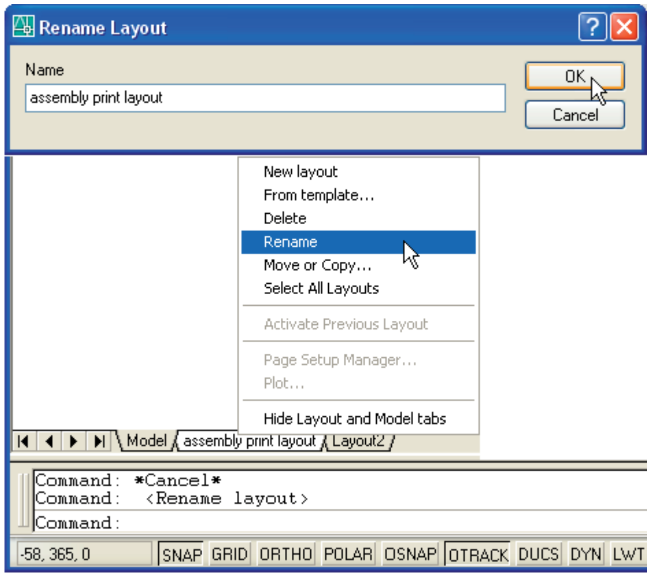


Fig. 20.6 First example **Printing/Plotting** – renaming the **Layout1** tab

5. *Click* the renamed **Layout** tab. The screen changes to a **PSpace** layout.
6. The Paper Space layout appears with the current viewport outlined in yellow (the colour of the **vports** layer). Using the **Erase** tool, erase the viewport with a *click* on its boundary line. The viewport and its contents disappear.
7. At the command line:

Command: *enter* **mv** (Mview) *right-click*
MVIEW
Specify corner of viewport or [ON/OFF/Shadeplot/Lock/Object/Polygonal/Restore/2/3/4]: *enter* **4** *right-click*
Specify first corner of viewport or [Fit] <Fit>: *right-click*
Regenerating model.
Command:

And four viewports reappear with the 3D model drawing in each.

8. Turn the layer **vports** off with a click on its **Turn a layer On or Off** icon.
9. *Click* the **Plot** tool icon in the **Standard New** toolbar (Fig. 20.7). A **Plot** dialog appears.

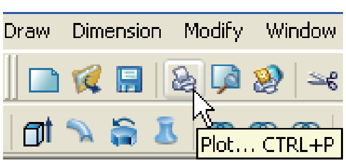


Fig. 20.7 The **Plot** tool icon from the **Standard New** toolbar

10. Check that the printer/plotter is correct and the paper size is also correct.
11. Click the **Preview** button. The full preview of the plot appears (Fig. 20.8).
12. *Right-click* anywhere in the drawing and *click* on **Plot** in the *right-click* menu which then appears.
13. The drawing plots (or prints).

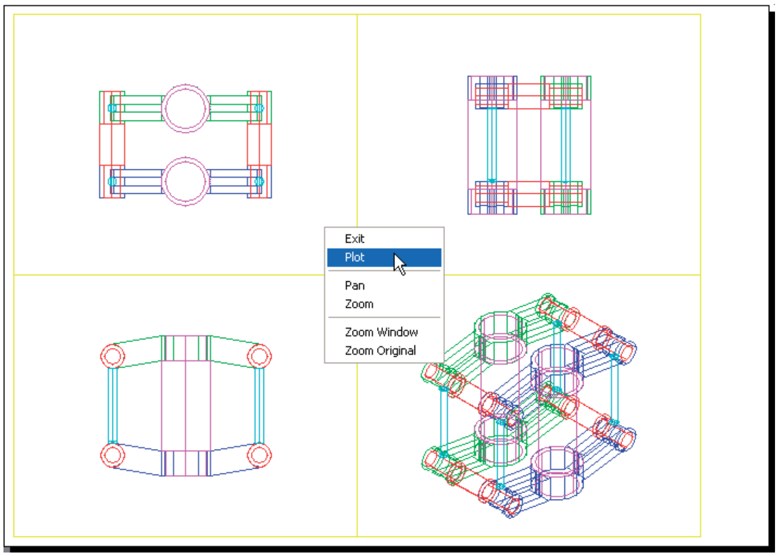


Fig. 20.8 First example – **Printing/Plotting**

Second example – Printing/Plotting (Fig. 20.9)

1. Open the orthographic drawing with its raster image (Fig. 20.5).
2. While still in **Model Space** *click* the **Plot** tool icon. The **Plot** dialog appears. Check that the required printer/plotter and paper size have been chosen.
3. *Click* the **Preview** button.
4. If satisfied with the preview (Fig. 20.9), *right-click* and in the menu which appears *click* the name **Plot**. The drawing plots.

Third example – Printing/Plotting (Fig. 20.10)

1. Open the 3D model drawing of the assembly shown in Fig. 20.8 in a single **SW Isometric** view.
2. While in **MSpace**, *click the* **Plot** tool icon. The **Plot** dialog appears.
3. Check that the plotter device and sheet sizes are correct. *Click* the **Preview** button.
4. If satisfied with the preview (Fig. 20.10), *right-click* and *click* on **Plot** in the menu which appears. The drawing plots.

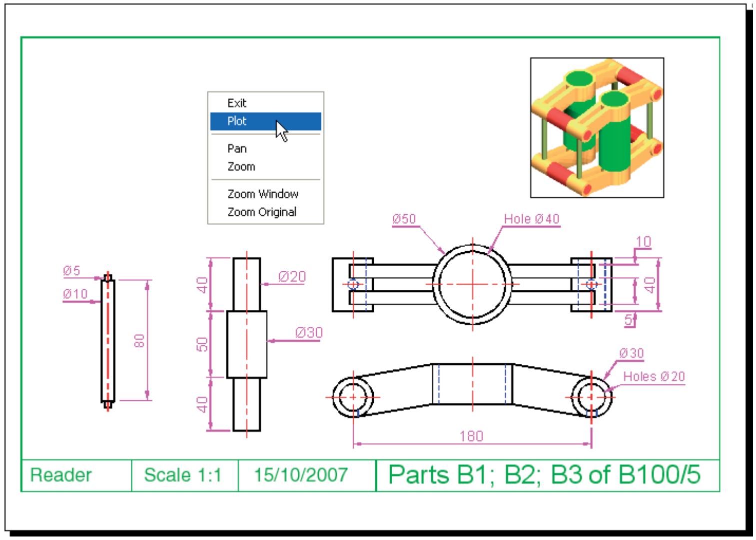


Fig. 20.9 Second example – **Printing/Plotting**

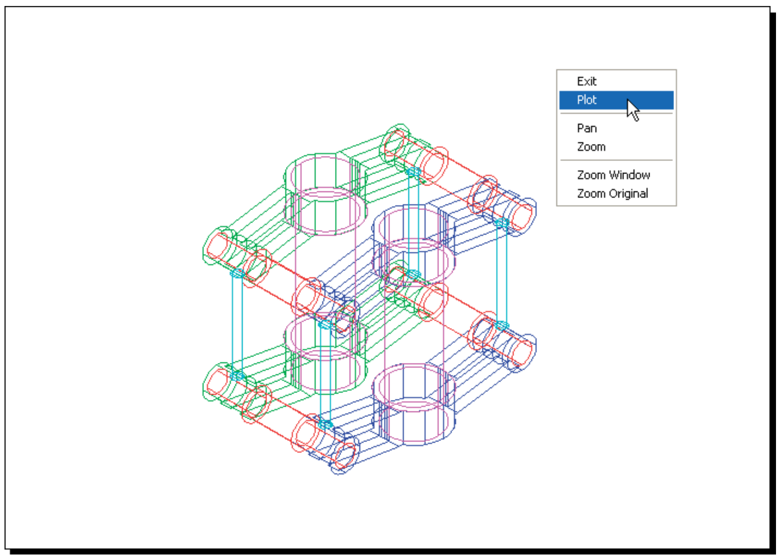


Fig. 20.10 Third example – **Printing/Plotting**

Polygonal viewports (Fig. 20.11)

The example to illustrate the construction of polygonal viewports is based upon exercise **7** (page 318). When the 3D model for this assignment has been completed in **Model Space**:

1. Make a new layer **Yellow** of colour yellow and make this layer current.
2. *Click* the **Layout1** tab.
3. Erase the viewport with a *click* on its bounding line. The outline and its contents are erased.

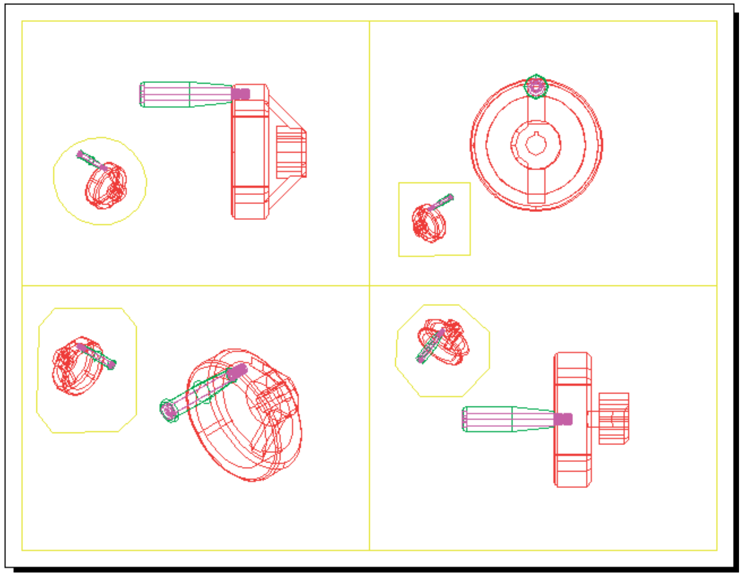


Fig. 20.11 **Polygonal viewports** – plot preview

4. At the command line:

   Command: *enter* **mv** *right-click*
   [prompts]: *enter* **4** *right-click*
   [prompts]: *right-click*
   Regenerating model.
   Command:

   And the model appears in a four-viewport layout.
5. Make sure the variable **UCSFOLLOW** is **OFF** (set to **0**).
6. *Click* the **PAPER** button in the status bar to turn it to **MODEL**. With a *click* in each viewport in turn and using the **3D Views** settings, set viewports in **Front, Right, Top** and **SW Isometric** views respectively.
7. **Zoom** each viewport to **All**.
8. *Click* the **MODEL** button to go back to **PAPER**.
9. Enter **mv** at the command line, which shows:

   Command: *enter* **mv** *right-click*
   MVIEW
   [prompts]: *enter* **p** (Polygonal) *right-click*
   Specify start point: In the top-right viewport *pick* one corner of a square
   Specify next point or [Arc/Close/Length/Undo]: *pick* next corner for the square
   Specify next point or [Arc/Close/Length/Undo]: *pick* next corner for the square
   Specify next point or [Arc/Close/Length/Undo]: *enter* **c** (Close) *right-click*
   Regenerating model.
   Command:

And a square viewport outline appears in the top-right viewport within which is a copy of the model.

10. Repeat in each of the viewports with different shapes of polygonal viewport outlines (Fig. 20.11).
11. *Click* the **PAPER** button to change to **MODEL**.
12. In each of the polygonal viewports make a different isometric view. In the bottom-right viewport change the view using the **3D Orbit** tool.
13. Turn the layer **Yellow** off. The viewport borders disappear.
14. Now plot the drawing. The **Preview** appears (Fig. 20.12).

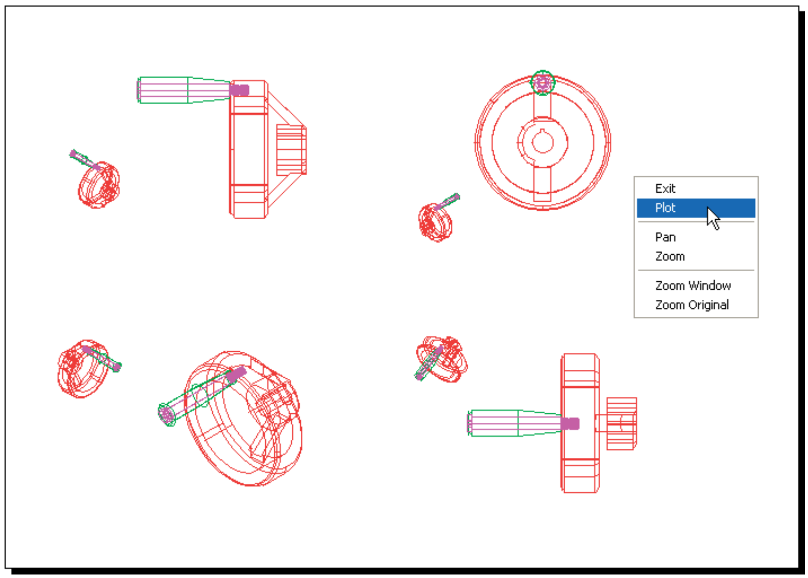


Fig. 20.12 **Polygonal viewports** – plot preview after turning the layer **Yellow** off

Exercises

1. Fig. 20.13 shows a polyline for each of the four objects from which the surface shown in Fig. 20.14 was obtained. Construct the surface and shade with **Realistic** shading.
2. Working to the sizes given in Fig. 20.16, construct an assembled 3D model drawing of the spindle in its two holders and render (Fig. 20.15).
3. A partial front view of a stand is shown by an isometric drawing (Fig. 20.17). From the details given in the drawing in Fig. 20.18, construct a 3D model drawing of the stand.

   Using appropriate lighting and material, construct a rendering of the 3D model which has been constructed.

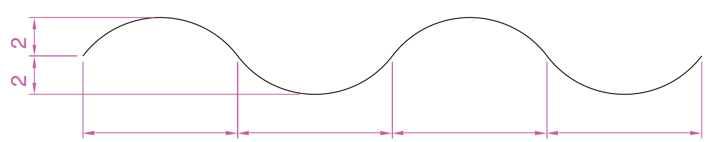
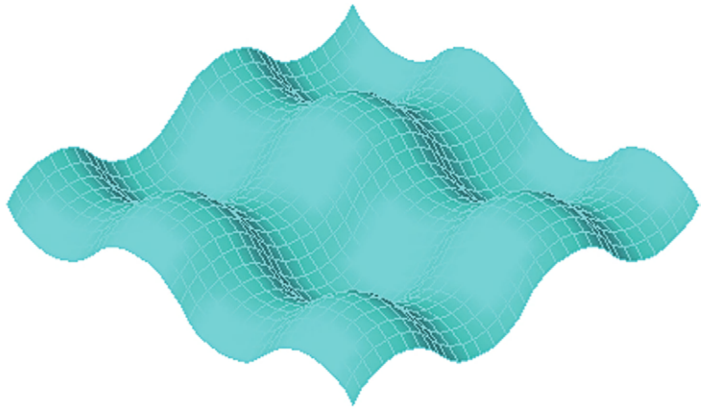


Fig. 20.13 Exercise 1 – the pline for each of the four objects

Fig. 20.14 Exercise 1



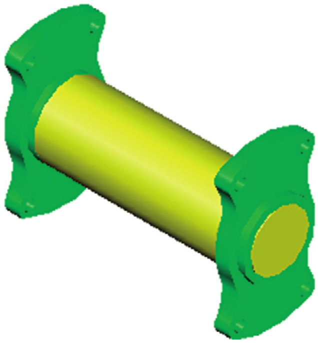
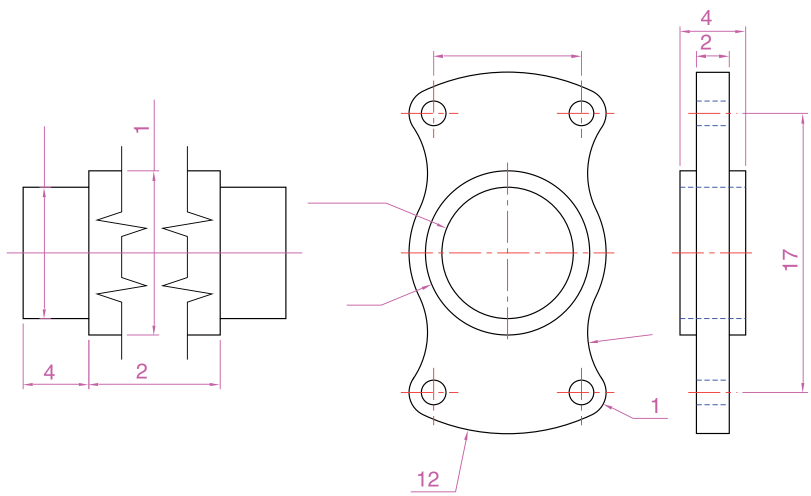


Fig. 20.15 Exercise 2

Fig. 20.16 Exercise 2 – details of shapes and sizes



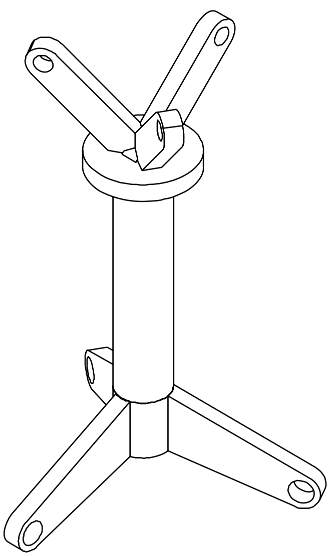


Fig. 20.17 Exercise 3 – isometric drawing

4. Construct an assembled 3D model drawing, working to the details given in Fig. 20.19.

   When the drawing has been constructed disassemble the parts as shown in the given isometric drawing (Fig. 20.20).
5. The surface model for this assignment was constructed from three edgesurf surfaces, working to the suggested objects for the surface as shown in Fig. 20.23. The sizes of the outlines of the objects in each case are left to your discretion. Fig. 20.21 shows the completed surface model. Fig. 20.22 shows the three surfaces of the model separated from each other.
6. Working to the details shown in Fig. 20.24, construct an assembled 3D model, with the parts in their correct positions relative to each

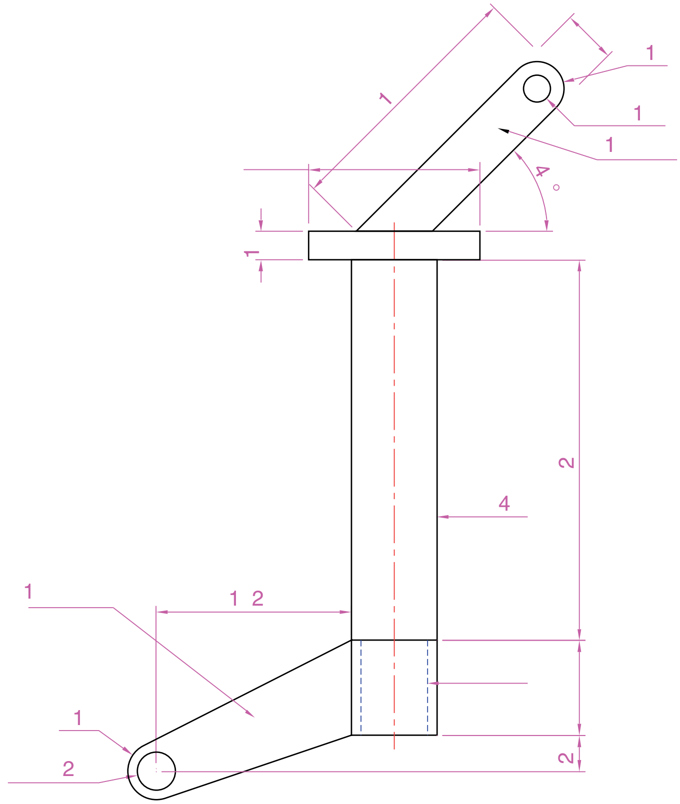


Fig. 20.18 Exercise 3

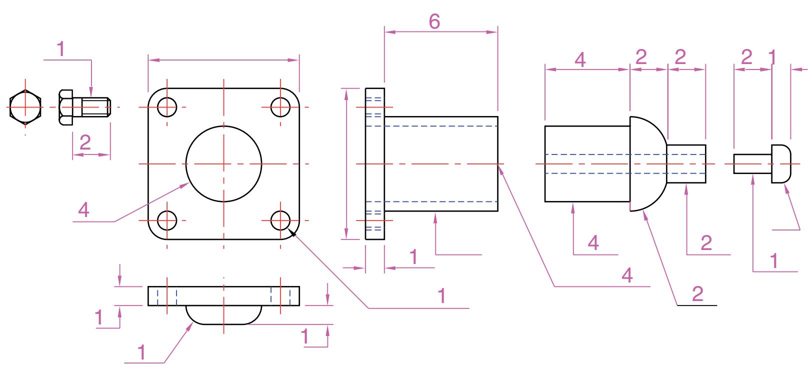


Fig. 20.19 Exercise 4 – details of shapes and sizes

other. Then separate the parts as shown in the isometric drawing in Fig. 20.25. When the 3D model is complete add suitable lighting and materials and render the result.

7. Working to the details shown in Fig. 20.26, construct a 3D model of the parts of the wheel with its handle.

   Two renderings of 3D models of the rotating handle are shown in Fig. 20.27 – one with its parts assembled and the other with the parts in an exploded position relative to each other.

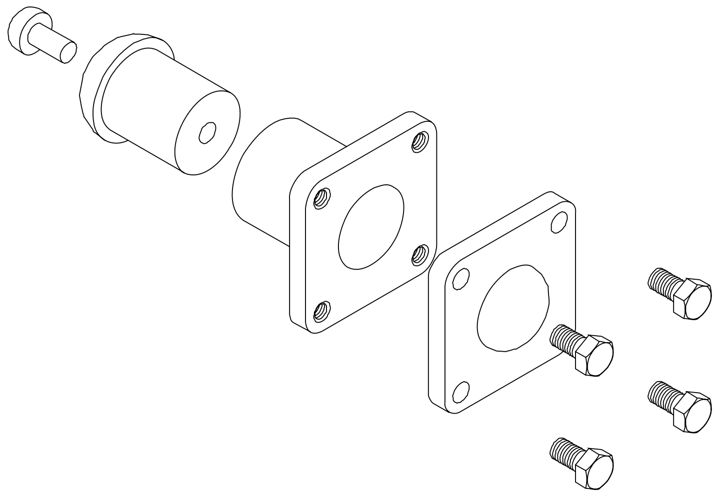


Fig. 20.20 Exercise 4 – an exploded isometric drawing

Fig. 20.21 Exercise 5

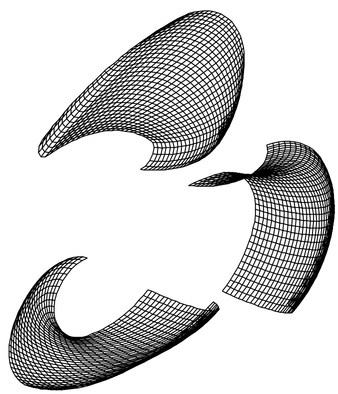


Fig. 20.22 Exercise 5 – surfaces separated

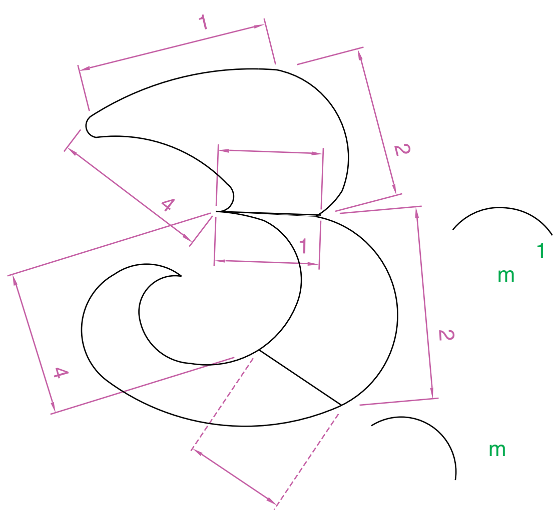


Fig. 20.23 Exercise 5 – objects for edgesurf surface model

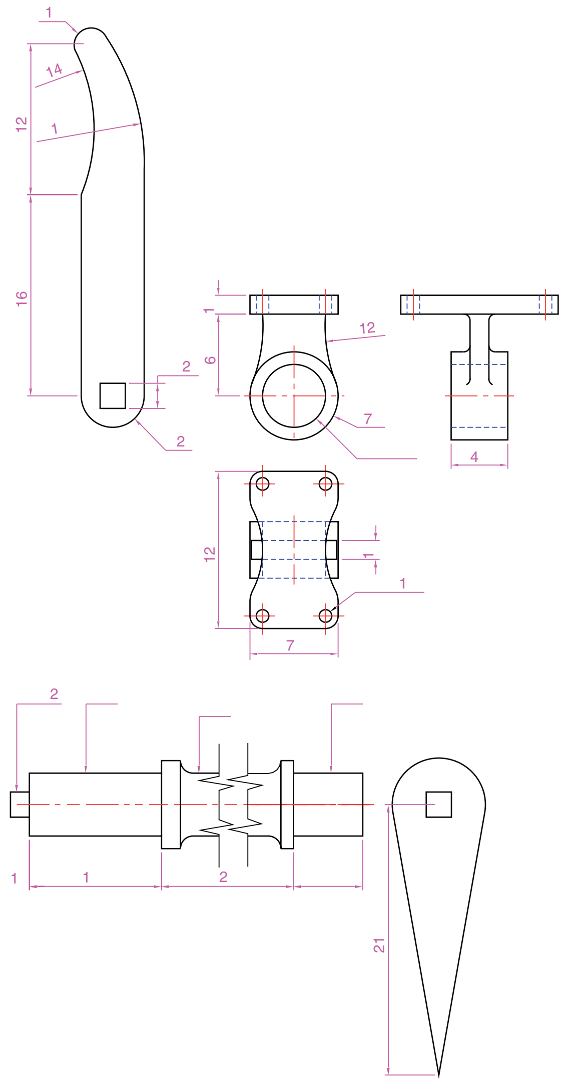


Fig. 20.24 Exercise 6 – details drawing

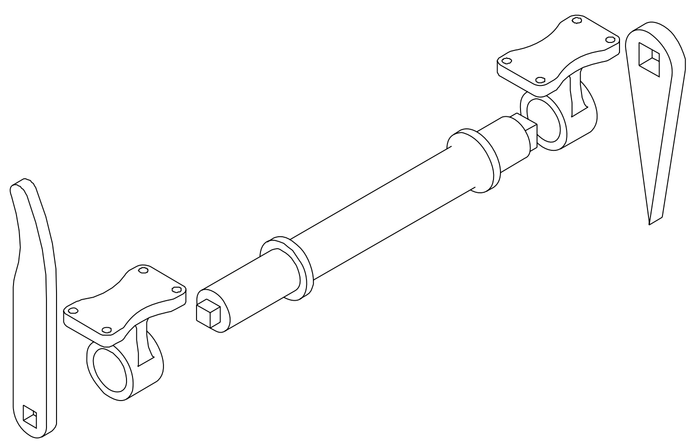


Fig. 20.25 Exercise 6 – exploded isometric drawing

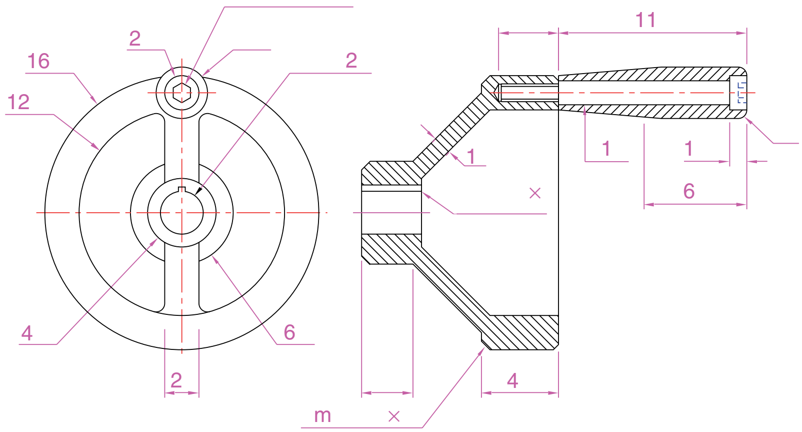


Fig. 20.26 Exercise 7 – details drawing

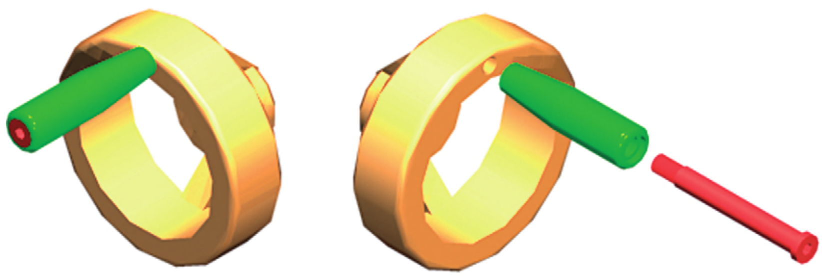


Fig. 20.27 Exercise 7 – renderings

CHAPTER 21

Internet tools

Aim of this chapter

The purpose of this chapter is to introduce the tools which are available in AutoCAD 2007 which make use of facilities available on the Word Wide Web (www).

Emailing drawings

As with any other files which are composed of data, AutoCAD drawings can be sent by email as attachments. If a problem of security of the drawings is involved they can be encapsulated with a password as the drawings are saved prior to being attached in an email. To encrypt a drawing with a password, *click* **Tools** in the **Save Drawing As** dialog and from the popup list which appears *click* **Security Options** . . . (Fig. 21.1). Then in the **Security Options** dialog which appears (Fig. 21.2) *enter* a password in the **Password or phrase to open this drawing** field. After *entering* a password *click* the **OK** button and *enter* the password again in the **Confirm Password** dialog which appears. The drawing then cannot be opened until the password is *entered* in the **Password** dialog which appears when an attempt is made to open the drawing by the person receiving the email (Fig. 21.3).

There are many reasons why drawings may require to be password encapsulated in order to protect confidentiality of the contents of drawings.

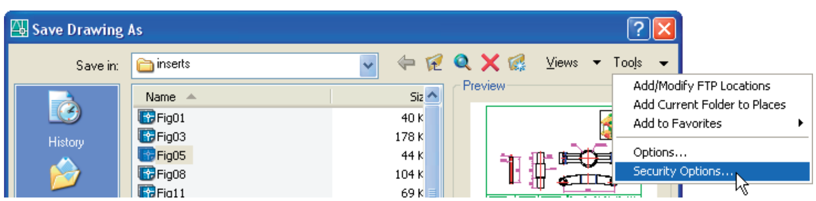


Fig. 21.1 Selecting **Security Options** in the **Save Drawing As** dialog

Example – creating a web page (Fig. 21.5)

To create a web page which includes AutoCAD drawings *left-click* **Publish to Web** . . . from the **File** drop-down menu. A series of **Publish to Web** dialogs appear, the second of which is shown in Fig. 21.4. After making entries in the dialogs which come on screen after each **Next** button is *clicked*, the resulting web page such as that shown in Fig. 21.5 will

Fig. 21.2 *Entering* and confirming a password in the **Security Options** dialog

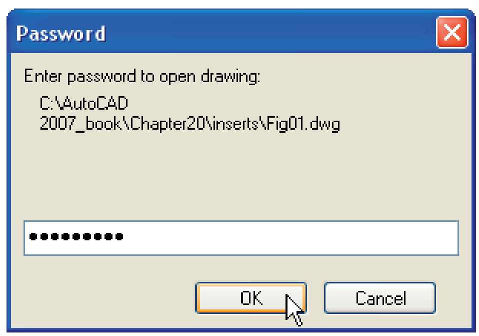


Fig. 21.3 The **Password** dialog appearing when a password encrypted drawing is about to be opened

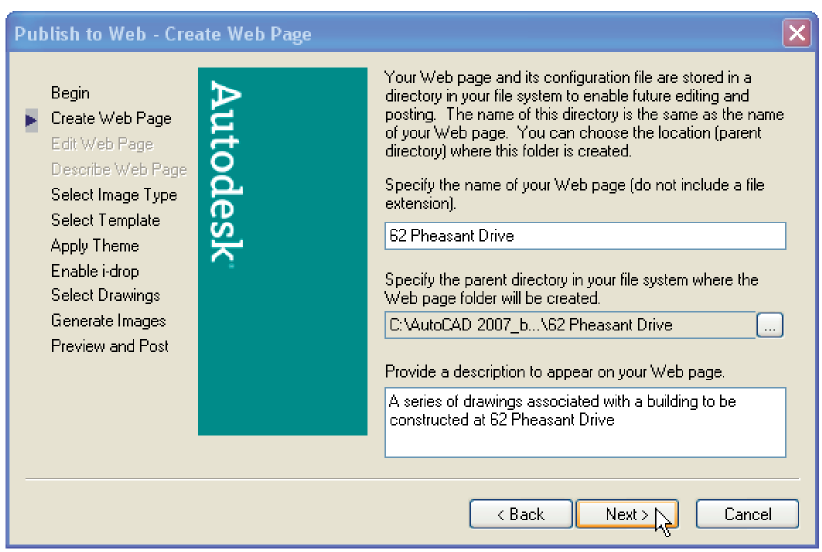


Fig. 21.4 One of the **Publish to Web** dialogs

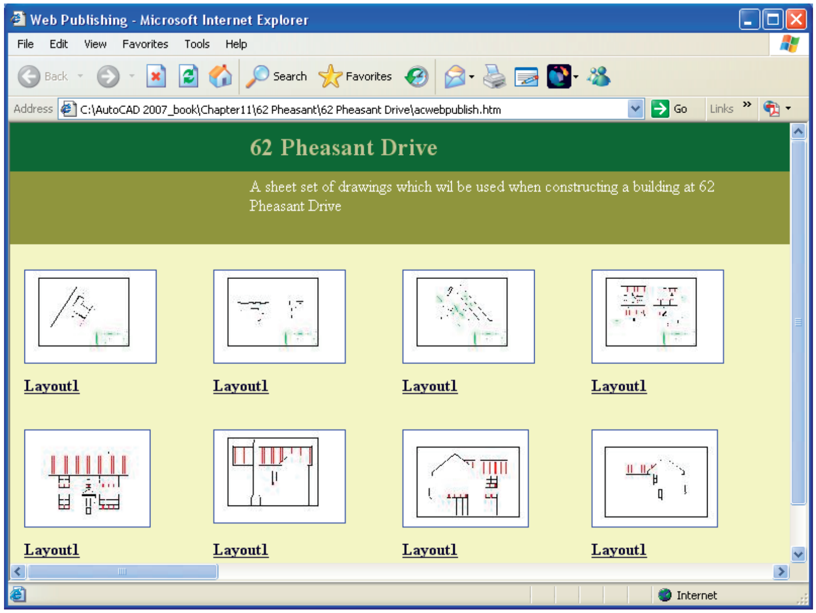


Fig. 21.5 The web page resulting from completing the **Publish to Web** series of dialogs

be seen (which can be saved as an ***.htm** file). A *double-click* in any of the thumbnail views in this web page and another page appears showing the selected drawing in full (Fig. 21.6).

In this example the drawings are the same as those which were included in a **DWF** file for the set of drawings shown in the sheet set for **62 Pheasant Drive**. The drawings from the **DWF** file are shown in Fig. 21.7 as seen in the **Autodesk DWF Composer**.

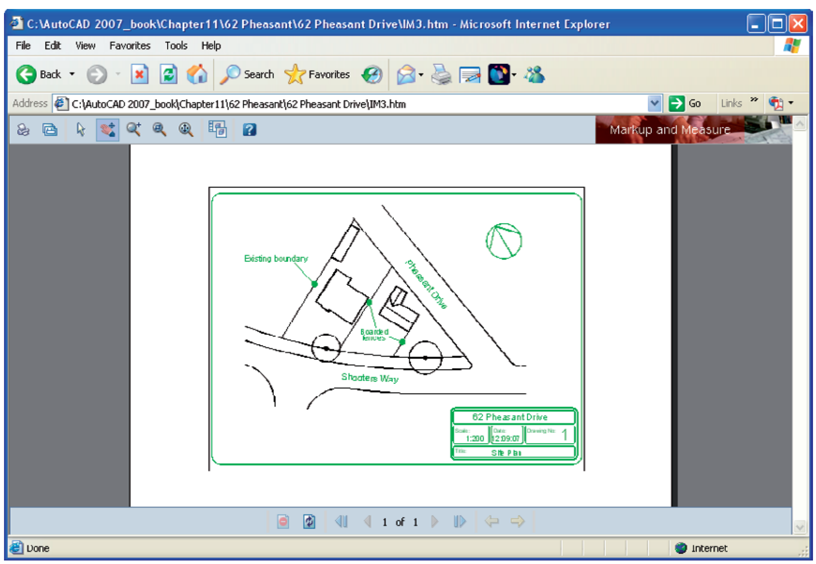


Fig. 21.6 The web page resulting from a *double-click* on a thumbnail

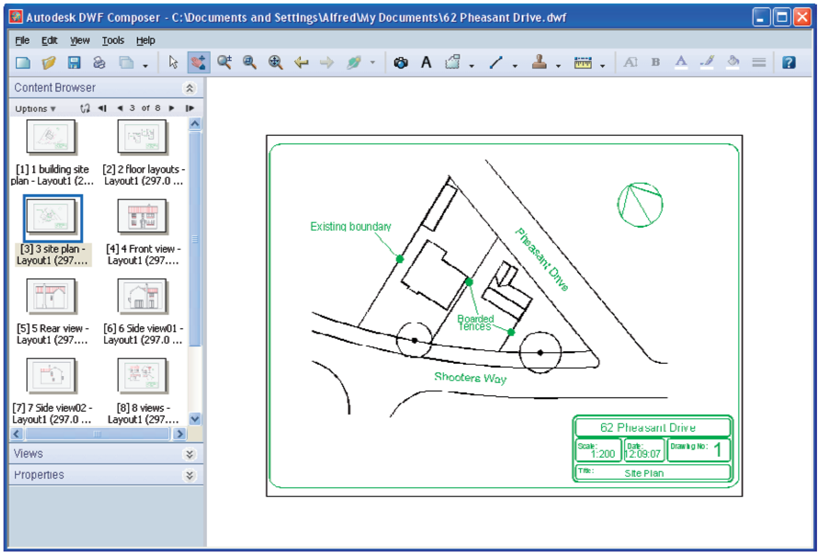


Fig. 21.7 The DWF file opened in the **AutoCAD DWF Composer**

The eTransmit tool

Click **eTransmit** . . . in the **File** drop-down menu and the **Create Transmittal** dialog appears (Fig. 21.8). The transmittal shown in this example is the drawing on screen at the time. Fill in details as necessary and *click* the **OK** button and a **zip** file is formed from the drawing file (Fig. 21.9). This **zip** file is easier and quicker to email than the drawing file. The AutoCAD drawing can be obtained by unzipping the **zip** file at the receiving end.

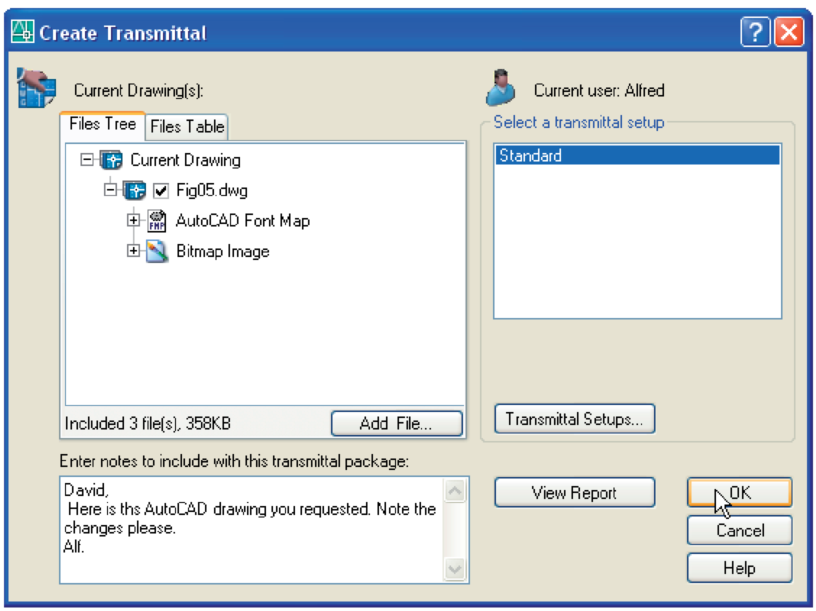


Fig. 21.8 The **Create Transmittal** dialog

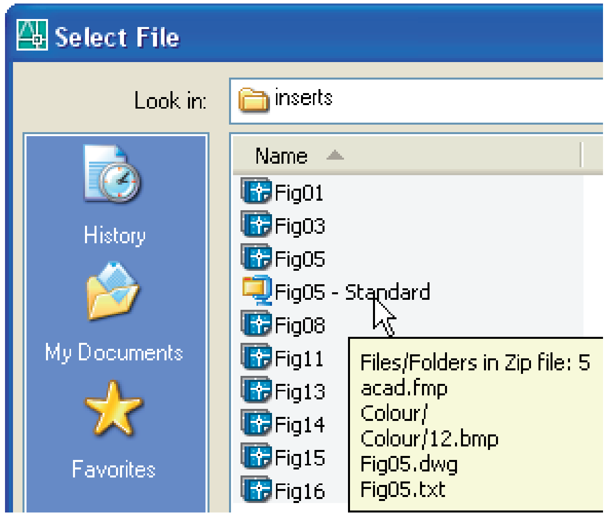


Fig. 21.9 The **zip** file created from the **Create Transmittal** dialog

CHAPTER 22

Design and AutoCAD 2007

Ten reasons for using AutoCAD

1. A CAD software package such as AutoCAD 2007 can be used to produce any form of technical drawing.
2. Technical drawings can be produced much more speedily using AutoCAD than when working manually – probably as much as ten times as quickly when used by skilled AutoCAD operators.
3. Drawing with AutoCAD is less tedious than drawing by hand – features such as hatching, lettering, adding notes, etc. are easier, quicker and indeed more accurate.
4. Drawings or parts of drawings can be moved, copied, scaled, rotated, mirrored and inserted into other drawings without having to redraw.
5. AutoCAD drawings can be saved to a file system without necessarily having to print the drawing. This can save the need for large drawing storage areas.
6. The same drawing or part of a drawing need never be drawn twice, because it can be copied or inserted into other drawings with ease. A basic rule when working with AutoCAD is: Never draw the same feature twice.
7. New details can be added to drawings or be changed within drawings without having to mechanically erase the old details.
8. Dimensions can be added to drawings with accuracy reducing the possibility of making errors.
9. Drawings can be plotted or printed to any scale without having to redraw.
10. Drawings can be exchanged between computers and/or emailed around the world without having to physically send the drawing.

The place of AutoCAD 2007 in designing

The contents of this book are designed to help only those who have limited (or no) knowledge and skills in the construction of technical drawing using AutoCAD 2007. However it needs to be recognised that the impact of modern computing on the methods of designing in industry has been immense. Such features as analysis of stresses, shear forces, bending forces and the like can be carried out more quickly and accurately using computing methods. The storage of data connected with a design and the ability to recover the data speedily are carried out much easily using computing methods than it had been, prior to the introduction of computing.

AutoCAD 2007 can play an important part in the design process because technical drawings of all types are necessary for achieving well-designed artefacts whether it be an engineering component, a machine, a building, an electronics circuit or any other design project.

In particular, 2D drawings which can be constructed in AutoCAD 2007 are still of great value in modern industry. AutoCAD 2007 can also be used to produce excellent and accurate 3D models, which can be rendered to produce photographic like images of a suggested design. Although not dealt with in this book, data from 3D solid model drawings constructed in AutoCAD 2007 can be taken for use in computer-aided machining (CAM).

At all stages in the design process, either (or both) 2D or 3D drawings can play an important part in aiding those engaged in designing to assist in assessing the results of their work at various stages. It is in the design process that drawings constructed in AutoCAD 2007 play an important part.

In the simplified design process chart shown in Fig. 22.1 an asterisk (*) has been shown against those features where the use of AutoCAD 2007 can be regarded as being of value.

The design chart (Fig. 22.1)

The simplified design chart in Fig. 22.1 shows the following features:

Design brief: A design brief is a necessary feature of the design process. It can be in the form of a statement, but it is usually much more. A design brief can be a written report which not only includes a statement made of the problem which the design is assumed to be solving, but includes preliminary notes and drawings describing difficulties which may be encountered in solving the design and may include charts, drawings, costings, etc. to emphasise some of the needs in solving the problem for which the design is being made.

Research: The need to research the various problems which may arise when designing is often much more demanding than that shown in the chart (Fig. 22.1). For example the materials being used may require extensive research as to costing, stress analysis, electrical conductivity, difficulties in machining or in constructional techniques and other such features.

Ideas for solving the brief: This is where technical and other drawings and sketches play an important part in designing. It is only after research that designers can ensure the brief will be fulfilled.

Models: These may be constructed models in materials representing the actual materials which have been chosen for the design, but in addition 3D solid model drawings such as those which can be constructed in AutoCAD 2007 can be of value. Some models may also be made in the materials from which the final design is to be made so as to allow testing of the materials in the design situation.

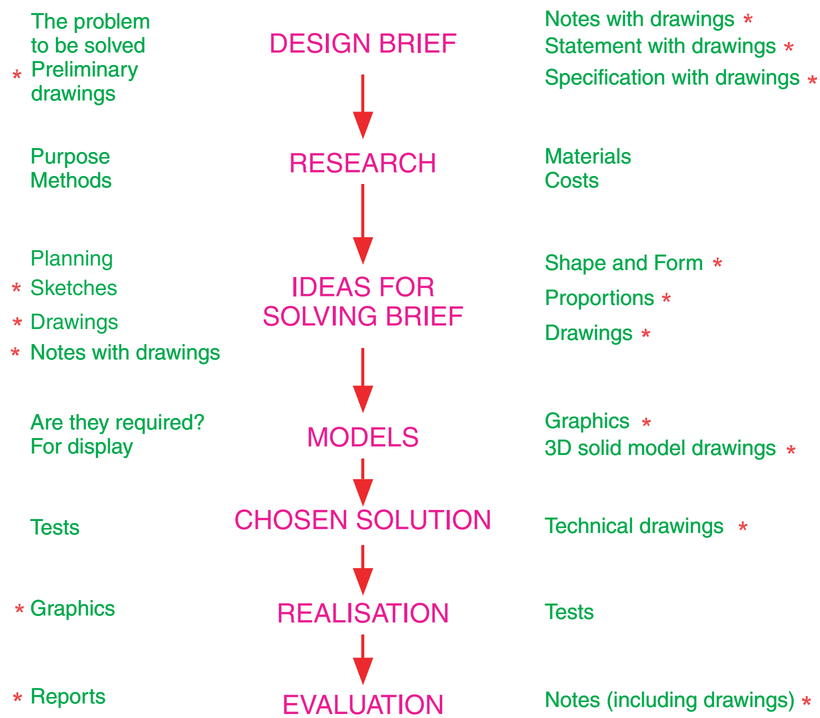


Fig. 22.1 A simplified design chart

Chosen solution: This is where the use of drawings constructed in AutoCAD 2007 are of great value. 2D and 3D drawings come into their own here. It is from such drawings that the final design will be manufactured.

Realisation: The design is made. There may be a need to manufacture a number of the designs in order to enable evaluation of the design to be fully assessed.

Evaluation: The manufactured design is tested in situations such as it is liable to be placed in use. Evaluation will include reports and notes which could include drawings with suggestions for amendments to the working drawings from which the design was realised.

Enhancements in AutoCAD 2007

AutoCAD 2007 contains major enhancements over previous releases, particularly when working in a **3D** environment. Please note that not all the enhancements in AutoCAD 2007 are described in this book. Among the most important enhancements are the following:

1. The introduction of **Workspaces**. The AutoCAD window can be in either a **2D** or a **3D** workspace. The **acadiso.dwt** template is the basis for a **2D** workspace and the **acadiso3D.dwt** is the basis for a **3D** workspace.
2. The introduction of the **DASHBOARD** palette which can be configured to show one, some or all of the **control panels** in which

the various 2D and 3D tools are held, allowing all tools in use to be held in one palette.

3. Control panels include **2D Draw, 3D Make, 3D Navigate, Light, Visual Styles, Materials** and **Render**.
4. Radical changes to methods of rendering 3D solid model drawings, with the introduction of new tools, new methods of lighting, adding materials, shading and rendering. Changes in methods of applying materials and lighting, including sun lighting.
5. New sub-dialogs in the **Options** dialog.
6. The introduction of some new **Solids** tools – **Helix, Sweep, Loft, Planesurf** and **3D Polylines**.
7. Changes in the 3D solids tools **Extrude** and **Revolve**, allowing either solids or surface to be extruded from 2D outlines.
8. Changes in the Solids tool **Pyramid**, allowing a pyramid of a chosen number of sides to be created.
9. Surfaces can be extruded from outlines which are not closed.
10. Enhancement of the **DYN** (Dynamic Input) system, particularly with respect to working in 3D.
11. The introduction of **DUCS** (Dynamic UCS) which allows work-spaces to be switched automatically when working in 3D.
12. Enhancements in the **3D Orbit** command.
13. The introduction of the **Camera** command, allowing the creation of cameras for taking views of a solid model.
14. The introduction of the **3Dwalk** command, allowing a walkthrough of a 3D model.
15. Advanced rendering methods.
16. **Parallel** and **Perspective** layouts are easily changed.
17. Drawings can be exchanged between MicroStation ***.dgn** formats and AutoCAD ***.dwg** formats.
18. The **Sheet Set Manager** has been updated.
19. Many new 3D commands introduced.
20. Changes to the **Slice** and **Section** tools, allowing slicing and sections along planes other than flat planes.
21. Introduction of animations.
22. The ***.dwg** format has been updated.

System requirements for running AutoCAD 2007

Graphics card: An AutoCAD certified graphics card. Details can be found on the web page **AutoCAD Certified Hardware XML Database**.

Operating system: Windows XP Professional, Windows XP Home, Windows 2000 or Windows XP Tablet PC Edition.

Microsoft Internet Explorer 6.0.

Processor: Pentium III 800 Mhz.

Ram: At least 128 MB.

Monitor screen: 1024 × 768 VGA with True Colour as a minimum.

Hard disk: A minimum of 300 MB.

APPENDIX A

Printing/Plotting

Introduction

Some suggestions for printing/plotting of AutoCAD drawings have already been given (pages 259–260). Plotters or printers can be selected from a wide range and are used for printing or plotting drawings constructed in AutoCAD 2007. The example given in this appendix is from a print using one of the default printers connected to the computer used by the author. However if another plotter or printer is connected to the computer, its driver can be set by first opening the Windows **Control Panel**, and with a *double-click* on the **Autodesk Plotter Manager** icon the **Plotters** dialog appears (Fig. A.1).

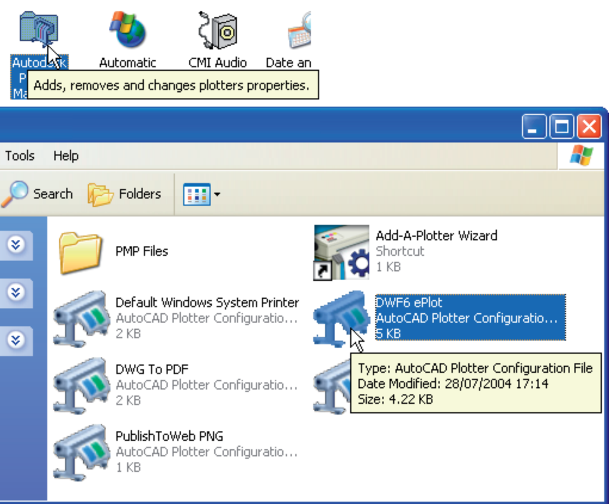


Fig. A.1 The **Plotters** window

Double-click on the **Add-A-Plotter Wizard** icon and the first of the **Add Plotter** series of dialogs appears (Fig. A.2). *Click* the **Next** button and the second of the dialogs (**Begin**) appears (Fig. A.3). *Click* in the radio button **My Computer** (in this case), followed by a *click* on the dialog's **Next** button.

The third of the dialogs appears – the **Plotter Model** dialog (Fig. A.4). Select names in the **Manufacturers** list and a selection can be made from the **Models** list associated with the chosen manufacturer's name.

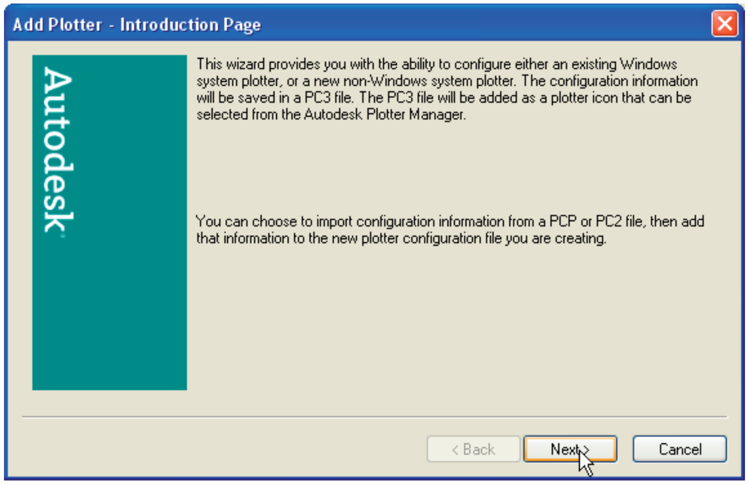


Fig. A.2 The first of the series of **Add Plotter** dialogs

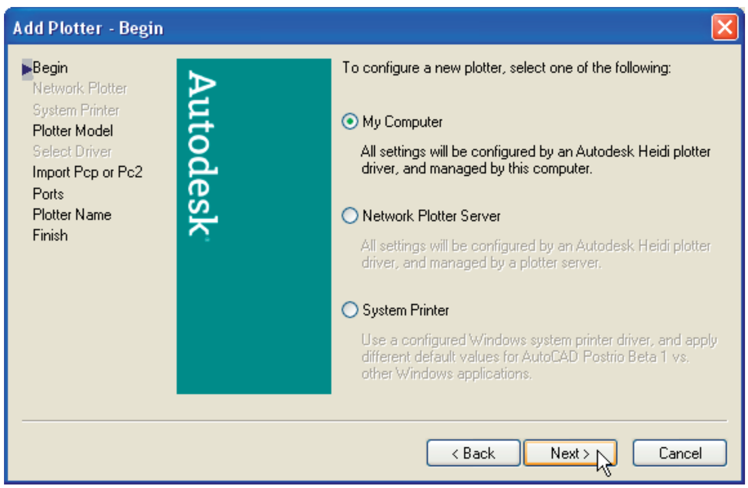


Fig. A.3 The **Add Plotter – Begin** dialog

There are several more dialogs in the series in which selections have to be made before completing the setting up of a printer or plotter for the printing of AutoCAD drawings.

Notes

1. AutoCAD drawings can be printed from the default printers already installed in the Windows system of the computer on which AutoCAD 2007 is loaded.
2. Plots or prints from drawings constructed in AutoCAD 2007 can be made from either Model Space or Paper Space.

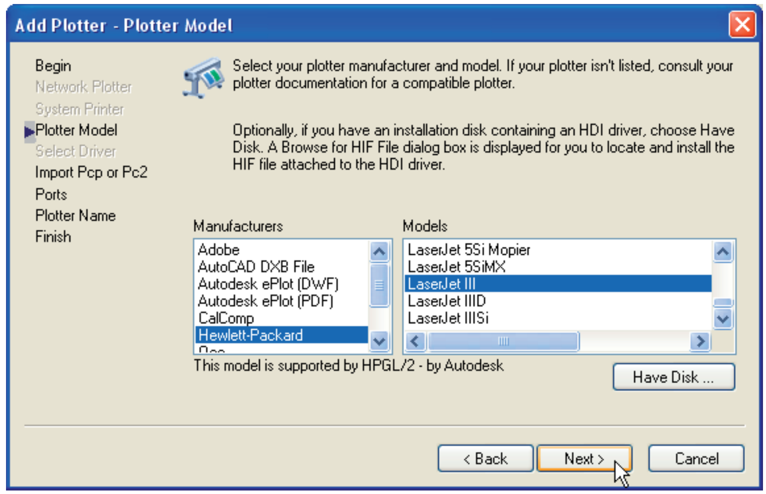


Fig. A.4 The **Add Plotter – Plotter Model** dialog

An example of a printout

1. Either select **Plot** . . . with a *click* on its tool icon in the **Standard New** toolbar (Fig. A.5) or from the **File** drop-down menu. The **Plot** dialog appears (Fig. A.6).
2. There are two parts in the **Plot** dialog. Fig. A.6 shows both the parts. A *click* on the arrow at the bottom right-hand corner of the dialog closes to reveal only the left-hand part and vice versa.
3. Select an appropriate printer or plotter from the **Printer/Plotter** list. In this example this is a colour printer. Then select the correct paper size

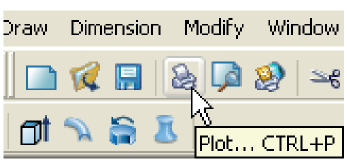


Fig. A.5 The **Plot** tool icon in the **Standard New** toolbar

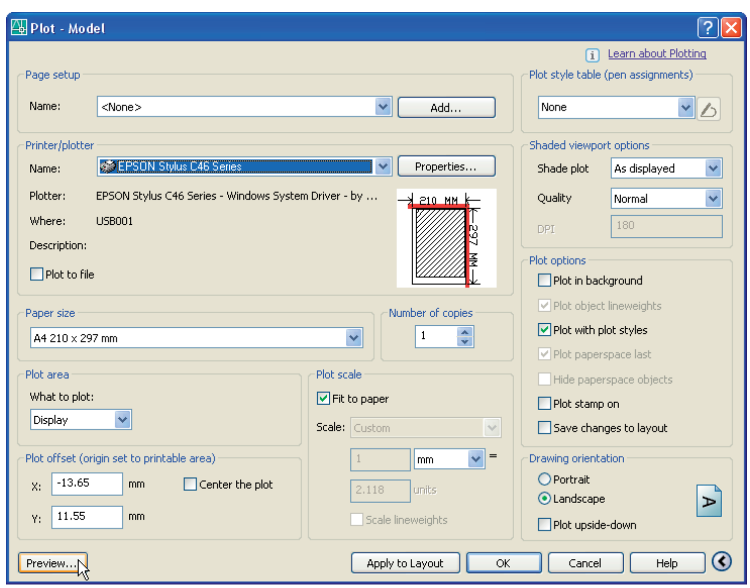


Fig. A.6 The **Plot** dialog

from the **Paper size** popup list. Then select what is to be printed/plotted from the **What to plot** popup list – in the example shown this is **Display**. Make sure the **Landscape** button is showing a dot (on). Then *click* the **Preview** button.

4. A preview of the drawing to be printed/plotted appears (Fig. A.7). If satisfied with the preview, *right-click* and in the menu which appears *click* **Plot**. If not satisfied *click* **Exit**. The preview disappears and the **Plot** dialog reappears. Make changes as required from an inspection of the preview and carry on in this manner until a plot can be made.

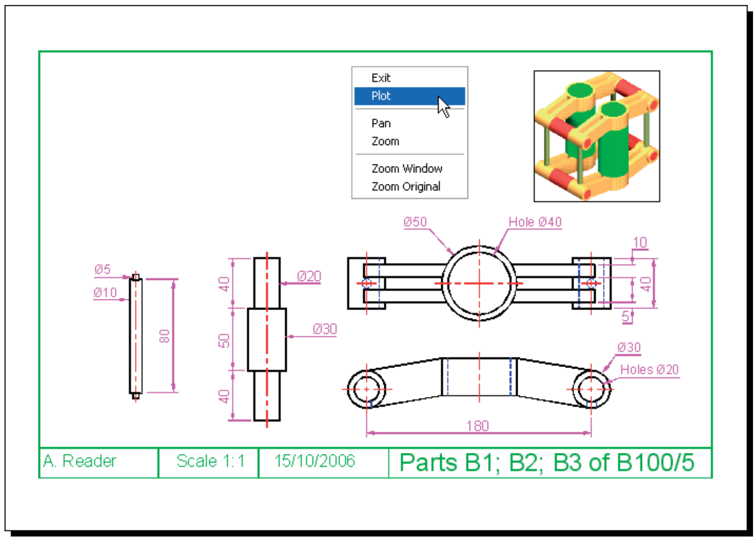


Fig. A.7 The **Plot Preview** window with its *right-click* menu

APPENDIX B

List of tools

Introduction

AutoCAD 2007 allows the use of over 300 tools. Some operators prefer using the word 'commands', although command as an alternative to tool is not in common use today. The majority of these tools are described in this list. Many of the tools described here have not been used in this book, because this book is an introductory text designed to initiate readers into the basic methods of using AutoCAD 2007. It is hoped the list will encourage readers to experiment with those tools not described in the book. The abbreviations for tools which can be abbreviated are included in brackets after the tool name. Tool names can be *entered* in upper or lower case.

A list of 2D tools is followed by a list of 3D tools. Internet tools are described at the end of this listing.

2D tools

About – Brings the **About AutoCAD** bitmap on screen
Appload – Brings the **Load/Unload Applications** dialog to screen
Adcenter (dc) – Brings the **DesignCenter** palette on screen
Align (al) – Aligns objects between chosen points
Arc (a) – Creates an arc
Area – States in square units the area selected from a number of points
Array (ar) – Creates **Rectangular** or **Polar** arrays in 2D
Ase – Brings the **dbConnect Manager** on screen
Attdef – Brings the **Attribute Definition** dialog on screen
Attedit – Allows editing of attributes from the Command line
Audit – Checks and fixes any errors in a drawing
Bhatch (h) – Brings the **Boundary Hatch** dialog on screen
Block – Brings the **Block Definition** dialog on screen
Bmake (b) – Brings the **Block Definition** dialog on screen
Bmpout – Brings the **Create Raster File** dialog
Boundary (bo) – Brings the **Boundary Creation** dialog on screen
Break (br) – Breaks an object into parts
Cal – Calculates mathematical expressions
Chamfer (cha) – Creates a chamfer between two entities
Chprop (ch) – Brings the **Properties** window on screen
Circle (c) – Creates a circle

Copytolayer – Copies objects from one layer to another
Copy (co) – Creates a single or multiple copies of selected entities
Copyclip (Ctrl+C) – Copies a drawing or part of a drawing for inserting into a document from another application
Copylink – Forms a link between an AutoCAD drawing and its appearance in another application such as a word processing package
Customize – Brings the **Customize** dialog to screen, allowing the customisation of toolbars, palettes, etc.
Dashboard – Brings the **DASHBOARD** palette to screen
Dashboardclose – Closes the **DASHBOARD** palette
Dblist – Creates a database list in a Text window for every entity in a drawing
Ddattdef (at) – Brings the **Attribute Definition** dialog to screen
Ddatte (ate) – Edits individual attribute values
Ddatext – Brings the **Attribute Extraction** dialog on screen
Ddcolor (col) – Brings the **Select Color** dialog on screen
Ddedit (ed) – The **Text Formatting** dialog box appears on selecting text
Ddim (d) – Brings the **Dimension Style Manager** dialog box on screen
Ddinsert (i) – Brings the **Insert** dialog on screen
Ddmodify – Brings the **Properties** window on screen
Ddosnap (os) – Brings the **Drafting Settings** dialog on screen
Ddptype – Brings the **Point Style** dialog on screen
Ddrmodes (rm) – Brings the **Drafting Settings** dialog on screen
Ddunits (un) – Brings the **Drawing Units** dialog on screen
Ddview (v) – Brings the **View** dialog on screen
Del – Allows a file (or any file) to be deleted
Dim – Starts a session of dimensioning
Dimension tools – The **Dimension** toolbar contains the following tools: **Linear, Aligned, Arc Length, Ordinate, Radius, Jogged, Diameter, Angular, Quick Dimension, Baseline, Continue, Quick Leader, Tolerance, Center Mark, Dimension Edit, Dimension Edit Text, Update** and **Dimension Style**
Dim1 – Allows the addition of a single dimension to a drawing
Dist (di) – Measures the distance between two points in coordinate units
Distantlight – Creates a distant light
Divide (div) – Divides an entity into equal parts
Donut (do) – Creates a donut
Dsviewer – Brings the **Aerial View** window on screen
Dtext (dt) – Creates dynamic text. Text appears in drawing area as it is entered
Dxbin – Brings the **Select DXB File** dialog on screen
Dxfin – Brings the **Select File** dialog on screen
Dxfout – Brings the **Save Drawing As** dialog on screen
Ellipse (el) – Creates an ellipse
Erase (e) – Erases selected entities from a drawing
Exit – Ends a drawing session and closes AutoCAD 2007
Explode (x) – Explodes a block or group into its various entities
Explorer – Brings the Windows **Explorer** on screen

Export (exp) – Brings the **Export Data** dialog on screen
Extend (ex) – To extend an entity to another
Fillet (f) – Creates a fillet between two entities
Filter – Brings the **Object Selection Filters** dialog on screen
Gradient – Brings the **Hatch and Gradient** dialog on screen
Group (g) – Brings the **Object Grouping** dialog on screen
Hatch – Allows hatching by the *entry* responses to prompts
Hatchedit (he) – Allows editing of associative hatching
Help – Brings the **AutoCAD 2007 Help: User Documentation** dialog on screen
Hide (hi) – To hide hidden lines in 3D models
Id – Identifies a point on screen in coordinate units
Imageadjust – (iad) Allows adjustment of images
Imageattach (iat) – Brings the **Select Image File** dialog on screen
Imageclip – Allows clipping of images
Import – Brings the **Import File** dialog on screen
Insert (i) – Brings the **Insert** dialog on screen
Insertobj – Brings the **Insert Object** dialog on screen
Isoplane (Ctrl/E) – Sets the isoplane when constructing an isometric drawing
Join (j) – Joins lines which are in line with each other or arcs which are from the same centre point
Laycur – Changes layer of selected objects to current layer
Laydel – Deletes and purges a layer with its contents
Layer (la) – Brings the **Layer Properties Manager** dialog on screen
Layout – Allows editing of layouts
Lengthen (len) – Lengthen an entity on screen
Limits – Sets the drawing limits in coordinate units
Line (l) – Creates a line
Linetype (lt) – Brings the **Linetype Manager** dialog on screen
List (li) – Lists in a text window details of any entity or group of entities selected
Load – Brings the **Select Shape File** dialog on screen
Ltscale (lts) – Allows the linetype scale to be adjusted
Measure (me) – Allows measured intervals to be placed along entities
Menu – Brings the **Select Menu File** dialog on screen
Menuload – Brings the **Menu Customization** dialog on screen
Mirror (mi) – Creates an identical mirror image to selected entities
Mledit – Brings the **Multiline Edit Tools** dialog on screen
Mline (ml) – Creates mlines
Mlstyle – Brings the **Multiline Styles** dialog on screen
Move (m) – Allows selected entities to be moved
Mslide – Brings the **Create Slide File** dialog on screen
Mspace (ms) – When in PSpace changes to MSpace
Mtext (mt or t) – Brings the **Multiline Text Editor** on screen
Mview (mv) – To make settings of viewports in Paper Space
Mvsetup – Allows drawing specifications to be set up
New (Ctrl+N) – Brings the **Select template** dialog on screen

Notepad – For editing files from the Windows 95 **Notepad**
Offset (o) – Offsets selected entity by a stated distance
Oops – Cancels the effect of using **Erase**
Open – Brings the **Select File** dialog on screen
Options – Brings the **Options** dialog to screen
Ortho – Allows ortho to be set ON/OFF
Osnap (os) – Brings the **Drafting Settings** dialog to screen
Pagesetup – Brings either the **Page Setup Model** or **Page Setup – Layout1** dialog to screen for setting print/plot parameters
Pan (p) – *Drags* a drawing in any direction
Pbrush – Brings Windows **Paint** on screen
Pedit (pe) – Allows editing of polylines. One of the options is **Multiple** allowing continuous editing of polylines without closing the command.
Pline (pl) – Creates a polyline
Plot (Ctrl+P) – Brings the **Plot** dialog to screen
Point (po) – Allows a point to be placed on screen
Polygon (pol) – Creates a polygon
Polyline (pl) – Creates a polyline
Preferences (pr) – Brings the **Options** dialog on screen
Preview (pre) – Brings the print/plot preview box on screen
Properties – Brings the **Properties** palette on screen
Psfill – Allows polylines to be filled with patterns
Psout – Brings the **Create Postscript File** dialog on screen
Purge (pu) – Purges unwanted data from a drawing before saving to file
Qsave – Saves the drawing file to its current name in AutoCAD 2007
Quickcalc (qc) – Brings the **QUICKCALC** palette to screen
Quit – Ends a drawing session and closes down AutoCAD 2007
Ray – A construction line from a point
Recover – Brings the **Select File** dialog on screen to allow recovery of selected drawings as necessary
Rectang (rec) – Creates a pline rectangle
Redefine – If an AutoCAD command name has been turned off by **Undefine, Redefine** turns the command name back on
Redo – Cancels the last **Undo**
Redraw (r) – Redraws the contents of the AutoCAD 2007 drawing area
Redrawall (ra) – Redraws the whole of a drawing
Regen (re) – Regenerates the contents of the AutoCAD 2007 drawing area
Regenall (rea) – Regenerates the whole of a drawing
Region (reg) – Creates a region from an area within a boundary
Rename (ren) – Brings the **Rename** dialog on screen
Replay – Brings the **Replay** dialog on screen from which bitmap image files can be selected
Revcloud – Forms a cloud-like outline around objects in a drawing to which attention needs to be drawn
Save (Ctrl+S) – Brings the **Save Drawing As** dialog box on screen
Saveas – Brings the **Save Drawing As** dialog box on screen
Saveimg – Brings the **Save Image** dialog on screen
Scale (sc) – Allows selected entities to be scaled in size – smaller or larger

Script (scr) – Brings the **Select Script File** dialog on screen
Setvar (set) – Can be used to bring a list of the settings of set variables into an AutoCAD Text window
Shape – Inserts an already loaded shape into a drawing
Shell – Allows MS-DOS commands to be entered
Sketch – Allows freehand sketching
Solid (so) – Creates a filled outline in triangular parts
Spell (sp) – Brings the **Check Spelling** dialog on screen
Spline (spl) – Creates a spline curve through selected points
Splinedit (spe) – Allows the editing of a spline curve
Stats – Brings the **Statistics** dialog on screen
Status – Shows the status (particularly memory use) in a Text window
Stretch (s) – Allows selected entities to be stretched
Style (st) – Brings the **Text Style** dialog on screen
Tablet (ta) – Allows a tablet to be used with a pointing device
Tbconfig – Brings the **Customize** dialog on screen to allow configuration of a toolbar
Text – Allows text from the Command line to be entered into a drawing
Thickness (th) – Sets the thickness for the Elevation command
Tilemode – Allows settings to enable Paper Space
Tolerance – Brings the **Geometric Tolerance** dialog on screen
Toolbar (to) – Brings the **Toolbars** dialog on screen
Trim (tr) – Allows entities to be trimmed up to other entities
Type – Types the contents of a named file to screen
UCS – Allows selection of **UCS** (User Coordinate System) facilities
Undefine – Suppresses an AutoCAD command name
Undo (u) (Ctrl+Z) – Undoes the last action of a tool
View – Brings the **View** dialog on screen
Vplayer – Controls the visibility of layers in Paper Space
Vports – Brings the **Viewports** dialog on screen
Vslide – Brings the **Select Slide File** dialog on screen
Wblock (w) – Brings the **Create Drawing File** dialog on screen
Wmfin – Brings the **Import WMF File** dialog on screen
Wipeout – Forms a polygonal outline within which all crossed parts of objects are erased
Wmfopts – Brings the **Import Options** dialog on screen
Wmfout – Brings the **Create WMF** dialog on screen
Xattach (xa) – Brings the **Select Reference File** dialog on screen
Xline – Creates a construction line
Xref (xr) – Brings the **Xref Manager** dialog on screen
Zoom (z) – Brings the zoom tool into action

3D tools

3darray – Creates an array of 3D models in 3D space
3dface (3f) – Creates a 3- or 4-sided 3D mesh behind which other features can be hidden
3dmesh – Creates a 3D mesh in 3D space

3dcorbit – Allows a continuous movement and other methods of manipulation of 3D models on screen
3ddistance – Allows the controlling of the distance of 3D models from the operator
3dfly – Allows walkthroughs in any 3D plane
3dforbit – Controls the viewing of 3D models without constraint
3dmove – Shows a 3D move icon
3dorbit (3do) – Allows methods of manipulating 3D models on screen
3dorbitctr – Allows further and a variety of other methods of manipulation of 3D models on screen
3dpan – Allows the panning of 3D models vertically and horizontally on screen
3drotate – Displays a 3D rotate icon
3dsin – Brings the **3D Studio File Import** dialog on screen
3dsout – Brings the **3D Studio Output File** dialog on screen
3ddwf – Brings up the **Export 3D DWF** dialog
3dwalk – Starts walk mode in 3D
Align – Allows selected entities to be aligned to selected points in 3D space
Ameconvert – Converts AME solid models (from Release 12) into AutoCAD 2000 solid models
anipath – Opens the **Motion Path Animation** dialog
Box – Creates a 3D solid box
Cone – Creates a 3D model of a cone
convertoldlights – Converts lighting from previous releases to AutoCAD 2007 lighting
convertoldmaterials – Converts materials from previous releases to AutoCAD 2007 materials
convtosolid – Converts plines and circles with thickness to 3D solids
convtosurface – Converts objects to surfaces
Cylinder – Creates a 3D cylinder
Dducs (uc) – Brings the **UCS** dialog on screen
Edgesurf – Creates a 3D mesh surface from four adjoining edges
Extrude (ext) – Extrudes a closed polyline
Flatshot – Creates a flattened view
Helix – Constructs a helix
Interfere – Creates an interference solid from selection of several solids
Intersect (in) – Creates an intersection solid from a group of solids
Light – Brings the **Lights** dialog on screen
Lightlist – Opens the **Lights in Model Space** palette
Loft – Activates the **Loft** command
Materials – Opens the **Materials** palette
Matlib – Brings the **Materials Library** dialog on screen
Mirror3d – Mirrors 3D models in 3D space in selected directions
Mview (mv) – When in PSpace brings in MSpace objects
Pface – Allows the construction of a 3D mesh through a number of selected vertices
Plan – Allows a drawing in 3D space to be seen in plan (UCS World)

Planesurf – Creates a planar surface
Pointlight – Creates a **Point** light
Pspace (ps) – Changes MSpace to PSpace
Pyramid – Creates a pyramid
Renderpresets – Opens the **Render Presets Manager** dialog
Renderwin – Opens the **Render** window
Revolve (rev) – Forms a solid of revolution from outlines
Revsurf – Creates a solid of revolution from a pline
Rmat – Brings the **Materials** dialog on screen
Rpref (rpr) – Brings the **Rendering Preferences** dialog on screen
Rulesurf – Creates a 3D mesh between two entities
Scene – Brings the **Scenes** dialog on screen
Section (sec) – Creates a section plane in a 3D model
Shade (sha) – Shades a selected 3D model
Slice (sl) – Allows a 3D model to be cut into several parts
Solprof – Creates a profile from a 3D solid model drawing
Sphere – Creates a 3D solid model sphere
Spotlight – Creates a spotlight
Stlout – Saves a 3D model drawing in ASCII or binary format
Subtract (su) – Subtracts one 3D solid from another
Sunproperties – Opens the **Sun** palette
Sunstudywizard – Creates a sun study through a wizard
Sweep – Creates a 3D model from a 2D outline along a path
Tabsurf – Creates a 3D solid from an outline and a direction vector
Torus (tor) – Allows a 3D torus to be created
Ucs – Allows settings of the UCS plane
Union (uni) – Unites 3D solids into a single solid
Visualstyles – Opens the **Visual Styles** palette
Vpoint – Allows viewing positions to be set from *x, y* and *z* entries
Vports – Brings the **Viewports** dialog on screen
Wedge (we) – Creates a 3D solid in the shape of a wedge
Xedges – Creates a 3D wireframe for a 3D solid

Internet tools

Browse the Web – Brings **Autodesk home page** from the Internet on screen
Etransmit – Brings the **Create Transmittal** dialog to screen
Publish – Brings the **Publish to Web** dialog to screen

APPENDIX C

Some of the set variables

Introduction

AutoCAD 2007 is controlled by a large number of variables (over 440 in number), the settings of many of which are determined when making entries in dialogs. Others have to be set at the command line. Some are read-only variables which depend upon the configuration of AutoCAD 2007 when it was originally loaded into a computer (default values).

A list of those set variables follows which are of interest in that they often require setting by *entering* figures or letters at the command line. To set a variable, enter its name at the command line and respond to the prompts which arise.

To see all set variables, *enter* **set** (or **setvar**) at the command line:

Command: *enter* **set** *right-click*
SETVAR Enter variable name or ?: *enter* ?
Enter variable name to list <*>: *right-click*

And a Text window opens showing a first window with a list of the first of the variables. To continue with the list press the **Return** key when prompted and at each press of the **Return** key another window opens.

Some of the set variables

ANGDIR – Sets angle direction. **0** counterclockwise; **1** clockwise
APERTURE – Sets size of pick box in pixels
BLIPMODE – Set to **1** marker blips show; set to **0** no blips
COMMANDLINE – Opens the command line palette
COMMANDLINEHIDE – Closes the command line palette

Note

DIM variables – There are over 70 variables for setting dimensioning, but most are in any case set in the **Dimension Style** dialog or as dimensioning proceeds. However one series of the **Dim** variables may be of interest:

DMBLOCK – Sets a name for the block drawn for an operator's own arrowheads. These are drawn in unit sizes and saved as required
DIMBLK1 – Operator's arrowhead for first end of line
DIMBLK2 – Operator's arrowhead for other end of line

DRAGMODE – Set to **0** no dragging; set to **1** dragging on; set to **2** automatic dragging

DRAG1 – Sets regeneration drag sampling. Initial value is 10

DRAG2 – Sets fast dragging regeneration rate. Initial value is 25

FILEDIA – Set to **0** disables **Open** and **Save As** dialogs; set to **1** enables these dialogs

FILLMODE – Set to **0** hatched areas are filled with hatching. Set to **0** hatched areas are not filled. Set to **0** and plines are not filled

GRIPS – Set to **1** and grips show. Set to **0** and grips do not show

MBUTTONPAN – Set to **0** no *right-click* menu with the Intellimouse. Set to **1** Intellimouse *right-click* menu on

MIRRTEXT – Set to **0** text direction is retained; set to **1** text is mirrored

PELLIPSE – Set to **0** creates true ellipses; set to **1** polyline ellipses

PICKBOX – Sets selection pick box height in pixels

PICKDRAG – Set to **0** selection windows picked by two corners; set to **1** selection windows are dragged from corner to corner

RASTERPREVIEW – Set to **0** raster preview images not created with drawing. Set to **1** preview image created

SHORTCUTMENU – For controlling how *right-click* menus show:
0 all disabled; **1** default menus only; **2** edit mode menus; **4** command mode menus; **8** command mode menus when options are currently available. Adding the figures enables more than one option

SURFTAB1 – Sets mesh density in the M direction for surfaces generated by the **Surfaces** tools

SURFTAB2 – Sets mesh density in the N direction for surfaces generated by the **Surfaces** tools

TEXTFILL – Set to **0** True Type text shows as outlines only; set to **1** True Type text is filled

TILEMODE – Set to **0** Paper Space enabled; set to **1** tiled viewports in Model Space

TOOLTIPS – Set to **0** no tooltips; set to **1** tooltips enabled

TPSTATE – Set to **0** and the Tool Palettes window is inactive. Set to **1** and the Tool Palettes window is active

TRIMMODE – Set to **0** edges not trimmed when **Chamfer** and **Fillet** are used; set to **1** edges are trimmed

UCSFOLLOW – Set to **0** new UCS settings do not take effect; set to **1** UCS settings follow requested settings

UCSICON – Set **OFF** UCS icon does not show; set to **ON** it shows

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