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# Caddie User Guide

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Installation and Quick Start

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CHAPTER 1 - Installation and Quick Start

Advanced Computer Solutions Europe Ltd and Advanced Computer Solutions (Pty) Ltd - South Africa are continuously developing Caddie to include requests from users and to utilise up-to-date technology to improve the functionality of the software. Keeping your Caddie maintenance up to date ensures that you always have the latest software updates as and when they become available. This also entitles you to hot line telephonic, online and email support.

Terms used when referring to Caddie software

Dialogue: Caddie makes use of Form’s or Windows in which you set or change various options and/or values. These are called dialogues or dialogue boxes.

Checking an option means clicking on a check box so that there is a Tick in it. Un-check means clicking on the same box so that the Tick is removed. Check boxes are used to enable or disable options inclusively. In other words, more than one option can be selected or de-selected at the same time. Radio buttons control mutually exclusive options. In other words, only one option can be selected at a time.

Please note that all dialogue boxes are accurate as at the time of printing and may differ due to product changes and enhancements.

Installation

Installation requirements and procedure

Before installing Caddie onto your computer please ensure that your computer complies with the minimum hardware and software requirements to run Caddie successfully. Visit http://www.caddiesoftware.com for the minimum and recommended hardware and software requirements.

To install Caddie on your PC, insert the Caddie Installation DVD into your DVD-ROM drive. If the Setup Wizard does not start automatically then click on Start | Run and type in X:\Setupsplash (where X represents the drive letter of your DVD-ROM drive) and press [ENTER]. Alternatively you can click on the Browse button and browse to the location of the Caddie installation DVD. Install the full Caddie program and the Caddie 2D symbols. Follow the on-screen instructions to complete the installation.
Caddie security information and start-up
The Help | Dongle Info command displays all the information regarding the Hasp driver and Caddie security information.

Hasp Tools

![Hasp Tools Image]

Operation System: The installed Windows operating system, including the build number is queried and displayed.

Hasp Driver: The installed driver version is displayed. You can uninstall and reinstall the hasp driver if necessary with the Uninstall Hasp Driver and Reinstall Hasp Driver buttons.

Dongle Tools

![Dongle Tools Image]

Dongle Information: If the correct Hasp driver is installed, your Caddie dongle number will be displayed. You can manually check for the validity of other dongles without plugging them in by typing in the dongle numbers, one at a time.

Dongle Type: The dongle types can be a local dongle on the pc, a network dongle, software lock or evaluation. The Download the latest Security File button will download the latest file from www.caddiesoftware.com.
Software Activation

**Softlock Status:** This will indicate if the software lock or evaluation is activated and the evaluation time remaining.

**Automatic Activation:** If you have an Internet connection you can select the Activate Now button to do the activation directly through the Caddie website.

**Manual Activation:** You can activate your software manually by sending a request by email, to web or to file. Select the Send License button. You can also get the license from the web or from a file. If you do not have Internet connection, the activation will be from a file.

**Network Dongles**

The server with the network dongle can be detected automatically, by name or by IP address. Select TCP or UDP protocol and apply these settings. Contact Caddie Support should you require any help with a network dongle installation.
Chapter 1  Installation and Quick Start

Caddie first time start-up options

**Regional Settings:** Enter the region where you will be using Caddie most frequently. This determines from which Caddie server location files will be updated.

**Caddie License:** Select the Caddie license type applicable to you.

**Applications:** Select the application you want to load at start-up. You don’t have to load all the applications at start-up. You can simply right-click on an application when in Caddie and load another application.

**Maintenance Reminder**

You can set up the way in which Caddie reminds you or your maintenance status. This feature is merely a reminder of maintenance renewal and does not in any way influence the performance of Caddie. Use the Help  |  Maintenance Info command to set the maintenance expiry date and to remind you in a day, 7 days, 14 days or when you maintenance expires.
The Caddie Screen

Key areas of the Caddie screen

Caddie is a Windows based application and complies in its layout to Windows© standards when it comes to screen layout. Like any other applications, however, there are many aspects that are particular to Caddie. The essential components of the Caddie screen are illustrated below.

![Caddie Screen Diagram]

**Title bar:** The title bar is located at the very top of the screen. It consists of the application name – i.e. Caddie as well as the active drawing name. When you open Caddie, a new drawing is generated. If you are not using a default template to generate new drawings, the drawing will be entitled Caddie1 which you will see just to the right. If you are using a template, then the template name will appear here. When you save the drawing with a proper name, the name will appear in the title bar. If you have more than one drawing open, then whichever drawing is current will have its name displayed in the title bar.

**Menu bar:** Located just under the title bar. Many of the menu items can be dropped on any of the toolbars available in the Caddie environment (See later in Toolbar Customisation).

**Toolbars:** These are, by default, located just underneath the menu bar. Since all toolbars are fully customisable in Caddie, you can dock any toolbar on the Top, Bottom, Left or Right side of the screen as well as have them as floating toolbars.

**Application and command menus:** These are located at the extreme left of the screen. As you select command categories from the Application menu, Caddie displays all the commands pertaining to this menu just to the right of the Application menu. By right-clicking on either the Application or Command Menus, you can configure their appearance as you desire. You can also switch the display of these menus ON & OFF. If all commands are not visible at once, you can scroll the Application and Command Menus by using the mouse wheel or click on the UP or DOWN arrows.

You can view these two menus as your Caddie toolbox. The Applications menu is a categorised
listing of the toolboxes whilst the Command menu contains the actual tools you use to draw. You can configure these two menus in any number of ways in terms of position, layout and visibility.

**Drawing tabs (MDI Tabs):** By default, Caddie displays drawing tabs at the top of the drawing area. These tabs make it easier to activate drawings when you have several drawings opened simultaneously. You can relocate them to the bottom of the screen (right-click and select align bottom or top) as well as to switch them off altogether select the MDI Tabs option from the View menu to toggle the viewing of these tabs. You can also re-shuffle the order in which the tabs appear. If you have several drawings opened, click and drag a drawing tab to the left or right until it is in the desired new position. Right-click on a drawing tab to display options for controlling the tabs.

**Command line:** The command line is located right at the bottom of the screen. System prompts, measurement results and a history of all commands for a particular session of Caddie is displayed. You can resize this area by placing the mouse on the top edge of the command line and dragging it up or down to increase or decrease its size respectively.

**Drawing area:** This is the large area that fills up most of the screen. The drawing area is delimited by a rectangle representing the border size which you setup in the Modelspace settings.

**Themes** You can change the look of Caddie by making use of the Themes feature that is common to Windows® XP® and Vista®. To change the current theme for Caddie, select the Change Themes option from the View menu and make the desired changes. Note that changing themes here does NOT affect the Windows® environment.

**General information**

**The Help menu**

You can get useful information about Caddie from the Help menu.

**Send Report:** You can use this facility to send Caddie information about your system for assessment purposes. Please note that you need to have an open connection to the internet to use this feature effectively.

**Tip of the Day:** Scroll through a list of useful tips on Caddie. You can choose to have these tips displayed each time you launch Caddie by checking Show tips at start-up option.

**About Caddie:** Caddie displays the About Caddie dialogue. You can gather some useful information from this dialogue.

**Help:** Accesses Caddie’s help utility.

**Context Help:** Select the specific command you want help on.

**Visit the Caddie Website:** Go directly to Caddie’s website. You must have an open Internet connection for this feature to work.

**E-Mail Caddie:** Caddie launches your default mail client where you can compose a message to send to Caddie.

**History Logging:** Set up how you want Caddie to log command and event history.

**Dongle Info:** See earlier in this chapter for an explanation of this feature.

**Maintenance Info:** Caddie displays the maintenance information.

**System performance check:** Use the Help | System Performance Check to check the general performance of your computer as it relates to the usage of Caddie. This operation can take a short
while and it may appear that Caddie is not responding. After a while, Caddie displays information regarding the performance of your computer whether it meets the minimum requirements for optimised use with Caddie.

**Windows Explorer:** Launch the Windows Explorer from within Caddie.

**Setting up the Caddie Modelspace Workspace**

**What is the Modelspace?**

The modelspace is the working area in a drawing. It is merely the area (space) where you create a working model of your drawing objects. The modelspace has no size limit but you can setup a border space in conjunction with a desired scale to represent the extent of the drawing area. You are, however, not limited to draw within this pre-defined modelspace since you can create printing sheets from any part of the modelspace whether objects are drawn inside or outside of the modelspace border.

**Configuring the Modelspace Workspace**

We always draw 1:1 in the modelspace. The viewport we place on a sheet for plotting will have a scale. In order to work efficiently in the modelspace, you first need to set it up. Select the **Modelspace Workspace** option for the **Settings** menu.

![Modelspace Workspace Configuration](image)

**Border:** If you want to show the border you can enter a custom size, or you can select a sheet size at the required scale.

**Origin:** Choose bottom left or centre for the placement of the X,Y,Z – 0,0,0 – point. The axes icon is
displayed at this position.

**Base Units:** Choose the desired working units to be used in your drawing.

**Axes Icon:** Tick to show the icon at the origin.

**Colour:** Select the colours for the border, grid lines and background.

**Grid Parameters:** Set the X and Y increment spacing between the grid lines and enter the angle. Tick to display the grid and set the origin.

**Plot Setup:** Select the default printer you wish to use when printing a drawing from modelspace, the plot style, plot scale, plot offset and plot units.

**Introduction to using the Command Line**

Caddie makes use of the command line to communicate with while you draw. When you first start using Caddie, you will constantly need to refer to the command line located at the bottom of the screen. Caddie constantly prompts you with messages indicating the sequential steps in using commands.

**Implementation of Caddie Commands**

Some of the Caddie commands are recursive. This means that such commands can be repeated continuously until you terminate the command using the **Option End** command from the **Context Sensitive Menu**.

<table>
<thead>
<tr>
<th>Command Name</th>
<th>Brief explanation of command function.</th>
</tr>
</thead>
</table>

The **Command Name** is the name of the command as it appears on the **Application** and **Command** menus.

When you move the mouse over the command, a tool tip will show the name and short description of the command.

**Recursive commands in Caddie**

Many commands in Caddie are recursive. This means that you can execute them repeatedly. For example, you can draw several rectangles one after the other. Select **Option End** to terminate the command.

A non-recursive command only executes once and must be re-selected to execute it again.

**Special keys**

There are some keys that have special functions when working in Caddie. The five most notable keys are the **[Esc]**, **[ENTER]**, **[Spacebar]**, **[TAB]** & **[Ctrl]** keys.

**The [Esc] key:**

The **[Esc]** (escape) key is used to abort any command.

**The [ENTER] key:**

The **[ENTER]** key can be used to close dialogue boxes, locating the current position of the cursor in a drawing and accepting coordinates entered on the command line.

- Locating the current position of the cursor in a drawing: In commands where you are prompted
to indicate a starting point, or a **FROM** position, or any point in the drawing, pressing the **[ENTER]** key will automatically **Snap** to the current location of the cursor. For example, if you want to draw a rectangle and a circle in such a way that the centre of the circle is be located exactly on the last drawn corner of the rectangle, draw a rectangle using the **Rectangle** command from the **LINES** menu. You will notice that Caddie leaves the cursor on the last point you clicked to indicate the diagonally opposite corner of the rectangle. Now select the **C-2P** command from the **CURVES** menu. When Caddie prompts you to **Indicate the centre of the circle**, just press the **[ENTER]** key. This will locate the centre of the circle to be located on that corner of the rectangle.

- Accepting coordinates entered on the command line: Another way to indicate **FROM** and **TO** positions in a drawing is to enter the coordinates at the keyboard. The **REL/ABS** switch on the status bar at the bottom of the screen will determine if these coordinates are calculated from the origin (0,0,0) (**ABS**), or calculated from the last entered point (**REL**). Say you want to draw a line with a length of 5000mm in the + X direction. Make sure the status bar is set to relative (**REL**), indicate a start position on the screen and type in **5000,0,0 [ENTER]**. This will draw a horizontal line of 5000mm in the positive x – direction.

**The [Spacebar] Key:**

Use the **[Spacebar]** key to repeat the previously executed command. This is useful for commands that are not recursive when you need to execute them repetitively. Simply press the **[Spacebar]** and Caddie activates the last command executed.

It is worth noting that some commands in Caddie are non-repeatable so that you will find that making use of this feature will not work with every command.

**The [TAB] Key:**

**NOTE:** The commands mentioned in the following discussion are all explained in the relevant chapters of the user guide and refer to these chapters for detailed explanations of these commands.

- Constraining Angles: You can use this key to restrain the angle being swept through when drawing lines. Let’s say that you have selected the **Draw** command from the **LINES** menu, click at the position where you want the line to begin. You now want to draw the line at a -30° angle (CW) having a length of 5000mm. Press and hold the **[TAB]** key down. Move the mouse in a clockwise motion. As you move the mouse, you will notice that Caddie snaps to angle increments of 15° (Or to whatever the default drag angle has been set to. This value is setup in the **Drag Settings** option in the **Settings** menu). When it snaps to the desired angle (In increments of 15°), type in **5000** and press **[ENTER]** Caddie will draw a line of 5000mm at an angle of -30°.

- To draw construction rays, press the **[TAB]** key before selecting the **Con-X, Con-Y, Con-Z** or the **Pnt-Ang** commands from the **CnLIN** menu. For the **Con-X and Con-Y** commands you will prompted to indicate the **Position for UCS X (Y or X) construction line**. Click in the drawing to indicate the position of the construction line. Caddie then prompts you to **Indicate approximate extent**. The point that you click on will indicate the endpoint of the construction ray. For the **Pnt-Ang** command, enter the desired angle for the construction ray. Caddie then prompts you to indicate the **Position for construction**. The point you click on will determine the endpoint of the construction ray.

- Drawing Vertical or Horizontal Lines: The **Hold-HV** command will draw alternate vertical and horizontal lines joined head to tail. If you press the **[TAB]** key before activating this command, the vertical and horizontal lines will not be joined.
- Drawing a rectangle as lines or rectangle as polylines: If the settings for the LINES | Rectang command is set to draw with lines, you can press the [TAB] key while selecting the Rectang command to draw with polylines, and visa versa.

- Moving multiple elements: Press [TAB] when selecting the MveElm command in the LINES menu to select multiple elements for moving. Ordinarily, this command is used to move a single element.

- Toggle between marking and un-marking objects: Use the MrkObj command from the MARK menu to mark objects. While you are marking objects you can press the [TAB] key to unmark objects. When you release the [TAB] key you will be able to mark objects again.

- Entering Paragraph Text: If the Indicate paragraph width on enter is unchecked on the TEXT | ParSet dialogue box, you will be able to indicate the paragraph length of the entered text by pressing the [TAB] key when entering text.

- Toggling Patterns in Scope: The [TAB] key can be used to toggle the selection of patterns when executing the Pattern command in the SCOPE menu. Select the Pattern command from the SCOPE menu, press and hold down the [TAB] key while selecting the patterns. Release the [TAB] key before indicating the last pattern. Only the selected patterns will be displayed.

- Naming Patterns on Initialisation: You can name a pattern as you initialise it by pressing and holding down the [TAB] key and then pressing the [I] key. Caddie displays the Name Pattern dialogue. Enter the desired pattern name and click on OK.

- Merging multiple polygons, patterns or objects into current pattern: One of the methods to merge a pattern into the current pattern is to use the MergPol command from the PATTERN menu. Activating this command allows you to merge any number of objects enclosed within a specified polygon into the current pattern. If you want to merge more than one polygon into the current pattern in one operation you must press the [TAB] key before selecting the MergPol command. Caddie prompts you to indicate the First point of polygon to select objects. Click at a suitable point in the drawing to begin drawing the polygon. Caddie prompts you to indicate the Next point. Click at the next point of the polygon. Continue until all objects for merging are completely enclosed in the polygon. Select Option End to complete the polygon. Repeat the steps to include more objects enclosed by polygons. Select Option End after the last polygon again to terminate the command. To merge multiple patterns into the current pattern, simply click on all the patterns you wish to merge into the current pattern.

- Selecting and un-selecting patterns: Note that for this feature to work correctly, the mouse must be setup so that the left double click is set to Pattern selection (See mouse button setup later). Double clicking on a pattern will select it. To de-select the pattern, press and hold the [TAB] key down and double click on the pattern.

- Displaying a layer on which an object is located: The LayDisp command in the SCOPE menu will prompt you to select the layers you wish to have displayed whilst hiding all other layers. If you wish to display the layer on which a particular object is located, press and hold the [TAB] key down BEFORE selecting the LayDisp command. Only the layer on whichever object you select will be displayed.

The [Ctrl] Key:

- Moving or copying and dynamic rotation of objects: Two options on the context sensitive menu are Copy and Move. In order to be able to rotate the copied or moved objects after placement, proceed as follows, select all the objects you wish to move or copy. Right-click to display the context menu. Press and hold the [Ctrl] key down BEFORE selecting the move or copy option. Release the [Ctrl] key. Caddie prompts you to Pick start of translation. Click at a suitable point in
the drawing or type in the coordinates to indicate this point. Caddie prompts you to Indicate new position. Click at a suitable point in the drawing or type in the coordinates to indicate the new position for the objects being moved or copied. Caddie prompts you to Enter point for rotation handle. Click at a suitable point in the drawing or type in the coordinates to indicate the rotation handle. Now select the destination point for the rotation of the objects.

- Changing the default settings of the rectangle command. Press [Ctrl] while selecting the Rectang command. This changes the setting to draw with lines to polylines, or from polylines to lines.

Snap Modes

Drawing accuracy by using snap modes

In order for you to be able to draw accurately from one control point to another, Caddie makes extensive use of snap modes. Snap modes are designed to snap to control points located on objects. For example, you may want to draw a line from the end point of one line to the centre of a circle. Without the use of snap modes, this would be virtually impossible to achieve simply by moving the mouse near enough these control points and hoping that you will click at the end point and circle centre of the two objects.

This is obviously not acceptable from a drawing perspective. To snap accurately you must use specific snap modes.

When you right click on the screen, you can select Freehand, Auto Snap or Additional Snap Modes from the Context Sensitive menu. The snap modes have grey and teal backgrounds.

The snap modes with a grey background colour are permanent snap modes – these snap modes stay active until you or the system selects another snap mode.

The snap modes with a teal background are intermediate snap modes – they only stay active for one selection.

You can also select the relevant snap mode from the View | DockingSnapMenu. Position this menu on the screen. There are more intermediate snap modes on the DockingSnapMenu.
## Snap mode descriptions

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Icon" /></td>
<td>Restrict the motion of the object being translated to a specified angle.</td>
</tr>
<tr>
<td><img src="image2" alt="Icon" /></td>
<td>Restrict the motion of the object being translated to an indicated line. The origin you selected when prompted to indicate the point to translate will be located exactly on the line and dragging can only occur along the length of the line.</td>
</tr>
<tr>
<td><img src="image3" alt="Icon" /></td>
<td>Restrict the motion of the object being translated by indicating two points for the drag limitation.</td>
</tr>
<tr>
<td><img src="image4" alt="Icon" /></td>
<td>This will snap perpendicular to an arc or circle placing the object with its point of origin on the line of the arc/circle.</td>
</tr>
<tr>
<td><img src="image5" alt="Icon" /></td>
<td>This snap mode will seek the extrapolated intersection of two indicated lines – apparent intersection. Caddie moves the point of origin to the point where the two lines would intersect.</td>
</tr>
<tr>
<td><img src="image6" alt="Icon" /></td>
<td>Move the cursor position from one location to another.</td>
</tr>
<tr>
<td><img src="image7" alt="Icon" /></td>
<td>Snap tangentially to an indicated arc or circle.</td>
</tr>
<tr>
<td><img src="image8" alt="Icon" /></td>
<td>Snap to the centre of an indicated arc or circle.</td>
</tr>
<tr>
<td><img src="image9" alt="Icon" /></td>
<td>Snap to the midpoint of an indicated line/arc.</td>
</tr>
<tr>
<td><img src="image10" alt="Icon" /></td>
<td>Snap to the intersection of an indicated line and spline.</td>
</tr>
<tr>
<td><img src="image11" alt="Icon" /></td>
<td>Snap to the nearest end of an indicated line.</td>
</tr>
<tr>
<td><img src="image12" alt="Icon" /></td>
<td>Snap midway between two indicated points.</td>
</tr>
<tr>
<td><img src="image13" alt="Icon" /></td>
<td>Snap to the nearest line/arc.</td>
</tr>
<tr>
<td><img src="image14" alt="Icon" /></td>
<td>Snap perpendicular (or normal) to an indicated line/arc.</td>
</tr>
<tr>
<td><img src="image15" alt="Icon" /></td>
<td>Snap to the nearest Text origin.</td>
</tr>
</tbody>
</table>
### Icon Description

<table>
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<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Snap to the nearest Leader origin." /></td>
<td>Snap to the nearest Leader origin.</td>
</tr>
<tr>
<td><img src="image" alt="Snap to the nearest Hatch origin." /></td>
<td>Snap to the nearest Hatch origin.</td>
</tr>
<tr>
<td><img src="image" alt="Snap to the nearest Dimension origin." /></td>
<td>Snap to the nearest Dimension origin.</td>
</tr>
<tr>
<td><img src="image" alt="Hold X. Draw/Move whilst maintaining a motion in the X-direction." /></td>
<td>Hold X. Draw/Move whilst maintaining a motion in the X-direction.</td>
</tr>
<tr>
<td><img src="image" alt="Hold Y. Draw/Move whilst maintaining a motion in the Y-direction." /></td>
<td>Hold Y. Draw/Move whilst maintaining a motion in the Y-direction.</td>
</tr>
<tr>
<td><img src="image" alt="Hold Z. Draw/Move whilst maintaining a motion in the Z-direction." /></td>
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</tr>
<tr>
<td><img src="image" alt="Snap to the intersection points of the grid in a drawing. The background colour of this icon will turn green when selected. Re-select to switch the grid off. Grid tales preference above any other snap modes." /></td>
<td>Snap to the intersection points of the grid in a drawing. The background colour of this icon will turn green when selected. Re-select to switch the grid off. Grid tales preference above any other snap modes.</td>
</tr>
<tr>
<td><img src="image" alt="Snap to an indicated construction point." /></td>
<td>Snap to an indicated construction point.</td>
</tr>
<tr>
<td>![Snap to the snap modes that you specified with the Snap</td>
<td>AutoSnap Settings.](image)</td>
</tr>
<tr>
<td><img src="image" alt="Snap to the nearest intersection (any intersection except if it involves a spline)." /></td>
<td>Snap to the nearest intersection (any intersection except if it involves a spline).</td>
</tr>
<tr>
<td><img src="image" alt="Snap to an indicated construction intersection." /></td>
<td>Snap to an indicated construction intersection.</td>
</tr>
<tr>
<td><img src="image" alt="Snap to the nearest end point found." /></td>
<td>Snap to the nearest end point found.</td>
</tr>
<tr>
<td><img src="image" alt="Freehand snap mode. No snap mode is active - i.e. The point of origin is relocated to wherever you click in the drawing." /></td>
<td>Freehand snap mode. No snap mode is active - i.e. The point of origin is relocated to wherever you click in the drawing.</td>
</tr>
<tr>
<td><img src="image" alt="Option End." /></td>
<td>Option End.</td>
</tr>
</tbody>
</table>

### Setting the control points for the auto-snap mode

Amongst all the available snap modes, the auto-snap mode is the most versatile. As its name implies, it has the capability of snapping to just about any combination of the other snap modes. You will recognise when you are in auto snap mode by the shape of the screen cursor used to display this snap mode.
To set the manner in which the auto snap command behaves, select the **AutoSnap Settings** from the Snap menu.

By checking and un-checking the various options, you can get the auto-snap command to snap to any number of control points in a drawing. As the settings are indicated above, the auto-snap mode will snap to all the control points that are checked and NOT to those that are un-checked. This means that you don’t continuously have to change snap modes to snap to various control points in a drawing.

**Example:**

You may wish to draw a continuous line starting at the end point of an existing line, then to the midpoint of another line, then to the centre of a circle then normal to yet another line, etc. By using the auto-snap mode, you will not need to change to the relevant snap mode to achieve the desired results. This saves considerable time. As you move the mouse close to a control point, a yellow marker will be displayed to identify the type of the control point.

**Setting the control points for preset snaps**

The Auto-Snap mode settings can be refined with the **Settings | Configure Preset Snaps** command when inserting or editing dimensions, hatch, text and polylines, etc. For example, when you are in the auto-snap mode and you want to enter dimensions, update the preset snap for new dimensions to include dimension origins. You can easily line up dimensions as you are snapping to the dimension origin.
Coordinate Systems used in Caddie

Cartesian coordinates

It is important to get a clear picture of the coordinate system that Caddie uses when you are drawing. When drawing in 2D mode, the Cartesian coordinate system is represented by a horizontal and vertical line intersecting to form the X (Horizontal) and Y (Vertical) axis. In 2D mode, then, you draw using an X-Y pair of coordinates. When measuring objects in 2D, you are working only in two planes - i.e. the X and Y planes.

All the snap modes available in 2D are also available in 3D, as well as 3D specific snap modes.

The Surveyor’s coordinate system is discussed under Survey Commands later in the user guide.

For the purpose of this user guide, all measurements are taken as being made whilst drawing in 2D mode. This accepts a top or plan view of a drawing and using only the XY axes. Select the View|Top View command to ensure that you are in the correct view.
Drawing Mode

There are two modes in which you can draw when using the Cartesian coordinate system. These are Absolute (ABS) and relative (REL) you can switch between the two by toggling this switch which is located at the bottom right of the status bar.

**ABS Mode:** When drawing in ABS drawing mode, all measurements are made from the drawing origin.

**REL Mode:** When drawing in REL drawing mode, all measurements are made from the current location of the cursor.

**Angles**

Angle measured in an Anti-Clockwise (ACW) direction is given a positive value. You do not have to type in the ‘+’ (plus) sign in front of the angle.

Any angle measured in a Clockwise (CW) direction is always given a negative value. In this case you will type in the - (minus) sign in order for Caddie to interpret the direction of the angle correctly.

**Distances**

Any length that is either vertical or horizontal can be typed in at the keyboard without using the mouse.

If you want to draw a horizontal distance of 500mm to the right of the current cursor position, make sure you are in REL (relative mode) and using the Cartesian coordinate system and type in (500,0) at the keyboard. Type in (0,-500) to draw a vertically down.

You can also use the keyboard arrows to enter horizontal and vertical values. Enter the distance in the command line and press the keyboard for the direction. [←] – left, [→] – right, [↑] – up and [↓] – down.
Axis Indicator

By default, Caddie places the Axis indicator at the bottom left of the drawing area and looks something like the illustration (A) at left. This is in fact a top (plan) view on a drawing. The vertical line represents the Y-axis and the horizontal line represents the X-axis. The circular shape you see in bottom left corner is the representation of the Y-axis. This is the point where all three axes intersect (0,0,0).

If we tilt the drawing a fraction, we will see how the XYZ axes intersect to form the origin (0,0,0) of the drawing.

When drawing in normal 2D mode, you will mainly use X and Y coordinates and you will work using the top view on a drawing.

Whilst drawing in 2D mode, you can make use mainly of the X and Y measuring or dimensioning commands since any measurement along the Y-axis will result in a zero value.

Example:
In this example we will illustrate a useful feature of Caddie that allows you to draw interactively using both the mouse and keyboard. Say you are drawing a line, and you click in various points in your drawing to indicate the various points of the line, you can, at any time, type in a value representing the distance to the next point of the line. As per the example, select the LINES|Draw command and draw lines 1-2 and 2-3 using the Freehand snap mode. In stead of clicking the next point 3, type in 3000 and press the [Right-Arrow] key. Caddie draws the horizontal line 3-4. Now type 4000 as the next value and press the [Up-Arrow] key to draw the line 4-5. You can also enter angles via the keyboard. Type in 2400<135 and press [Enter]. The line 5-6 will be drawn with a distance of 2400 and a direction of 135°.

Being able to draw using the keyboard can often reduce drawing time in not having to insert construction lines and then drawing to the construction intersections. This functionality can be used in many other instances when drawing. You can use this feature whenever you need to
indicate a vertical or horizontal position, drawing a line, relocating objects, stretching parts of a drawing or repositioning the current location of the point of origin. These are a few examples of using the keyboard to manipulate a drawing.

**Entities in Caddie**

**General terminology**

Entity or object – This is the fundamental term used to describe anything that you draw in Caddie and can refer to lines, circles, text, hatch, construction lines, etc. An entity or object can also be referred to as an element when referring lines, line segments, circles and arcs.

Another term you will come across is finishing elements - This refers specifically and exclusively to text, leaders, hatch and dimensions.

Throughout this user guide, these terms are used interchangeably and in context to the parts of a drawing referred to.

Any entities you draw in Caddie are made up of one or more of the fundamental building blocks and can be categorised as follows.

- Construction objects - Lines, circles and points;
- Drawing objects - Lines, ellipses, splines, circles, arcs, etc;
- Finishing objects - Dimensions, text, leaders and hatching;
- 3D objects;

**Construction objects**

Construction lines, circles and points are used to help you draw accurately as well as speeding up drawing time. When you draw a construction line in a drawing, no matter what the angle, Caddie draws these lines right across the screen. You cannot enter a length for a construction line but you can draw them at specific angles. Construction circles, on the other hand, can be drawn to specific radii. By default, Caddie draws a small yellow filled rectangle for each construction point you place in a drawing. You can setup the way in which construction points are drawn. You can change the appearance of construction entities using the construction parameter set command (See later in the section on parameters).

Construction points are useful as markers in drawings when you need to identify specific points in a drawing as well as remembering the location of this point. A common use of construction points is when you need to locate the origin of a symbol. Before saving a symbol, place a construction point at the location of its origin. This way you will not need to try and remember where its origin is.

Construction lines and circles do **NOT** print so you don’t need to delete or hide them before printing a drawing. You can, however, print construction points. You can change the colour of construction entities using the construction parameter set command (See later in the section on parameters). You can also edit the colours of existing construction entities in a drawing.

**Drawing objects**

The bulk of any drawing is made up of drawing entities. This form the very substance of any drawing and by setting up pen type and line styles you can create different effects in a drawing thus producing the desired results when it comes to printing drawings. You can manipulate drawing entities in many different ways depending on the desired result. You will learn these various techniques as you progress through this user guide.
Finishing objects

There is a third group of entities in Caddie drawings that make up what are commonly known as finishing elements – text, dimensions, leaders and hatching. They are used to finish a drawing off.

The SCOPE of a Drawing

Visible objects versus hidden objects

The concept of SCOPE in a Caddie drawing is an easy one to understand. It is a concept that will crop up very regularly in all drawings that you create. It is imperative, then, that you understand this concept thoroughly.

In order better to understand this concept picture the following scenario – you are in a room with people inside sitting at desks behind room dividers. The door leading into the room is shut and all the curtains are drawn. Assume that you and all the people are seated at their desks and that you are unable to see them as a result of the room dividers (assume low room dividers that just cover the seated peoples’ heads). At that point, the scope of your vision is only what you can see in the room - a room full of dividers!

Let’s say that you now open one of the curtains. Immediately, the scope of your vision has changed as you can now see objects outside the room through the window. In other words, you have changed the scope of your vision.

Now you decide to open the door. Once again you have changed the scope of your vision to include any objects visible through the open doorway.

Ask one or two people to stand up and you will now be able to see them. Yet again, you have changed the scope of your vision to include the people that are now standing. Draw the curtain shut once again and you have limited the scope of your vision once more.

This scenario can be mirrored directly into Caddie’s concept of scope Caddie will only execute commands on what is within the scope of a drawing. In other words, what it cannot ‘SEE’ it will NOT be able to manipulate, edit, print, etc.

Hiding and displaying parts of a drawing

There are several reasons why you might want to hide parts of a drawing. You may only want to print parts of a drawing or isolate certain entities for selection or manipulation.

There are a number of ways in which you can hide parts of a drawing.
- By using the suppress switches on the scope toolbar;
- By switching the visibility of the layer off;
- By setting the scope of a drawing to an indicated pattern(s) only;
- By setting the scope of a drawing to selected or marked objects only;

Using the suppressing switches

When you have larger drawings that contain large areas of hatching as well as a lot of text and dimensions, the screen may be very busy. It is at times like these that you can make use of the available suppression switches. You will find these commands on the Scope toolbar.

If this toolbar is not visible, select the View | Scope command to switch it on. By default, all drawing entities are displayed in a drawing at all times. In order to speed up screen regeneration,
it is sometimes more practical temporarily to hide parts of a drawing by suppressing selected aspects of the drawing.

**Example 1:** To suppress hatching, click on the Hatches command on the Scope toolbar. You will notice that all hatching in the drawing **Disappears** from the screen. You will notice now that whenever you redraw the screen it is much quicker once the hatching and, for that matter, any other finishing entity has been suppressed. This is also practical when working of complex projects.

**Example 2:** Let’s say you have hatched parts of a drawing and you have suppressed all hatching in the drawing. You now decide to move, copy or stretch objects that have been hatched to a new location. Once the translation command has completed execution and you un-suppress all hatching, the hatched area will not have been affected and you will not have produced the desired results. As far as Caddie is concerned, entities you suppress in a drawing do not exist as they are outside the **SCOPE** of the drawing.

**Hiding and displaying layers**

There are times when you need to hide layers in a drawing to achieve special effects or to view different versions of parts of a drawing. If you have made judicious use of the layering system that Caddie offers, it will be easy for you achieve the desired results. It is imperative that you learn to work with layers in any drawing that you create.

Making use of layers in a drawing dramatically improves your ability to manage a drawing. This includes moving objects around, re-assigning layers to existing objects, printing or saving only parts of a drawing, to mention but a few.

Picture the following scenario. You have a full set of working drawings of the plans for a new shopping complex. The plans have been approved by your local municipal council or the organisation responsible for approving such plans. The project has the green light to go ahead. You now need to issue sets of drawings to the relevant contractors that are actually going to erect the structures involved. It would be desirable to give each contractor only what he needs to carry out his specific task in relation to the whole project.

For instance, the contractor, that is going to carry out the piling and foundation work, does not need to know the kind of flooring to be used or where furniture will be located. If you have made good use of layers, you can hide those layers that you do not wish to print by changing the scope of your drawing to include only those layers in the drawing that you want to print. Once you have done this, you can send the drawing to the printer with the result that only what is necessary for the contractor will be printed.

**Changing the scope to a pattern(s)**

You may want to inspect a single pattern in a drawing in order to change pen and line settings or you may want to find out what the extent of a particular pattern is. In the scope menu, you will find a command you can use to view only entities belonging to a single pattern. What this effectively accomplishes is to hide everything in the drawing that is not part of the pattern you indicate.

**Changing the scope to selected or marked objects**

By marking or selecting objects in a drawing, you can instruct Caddie to hide all entities in the drawing that are **NOT** marked or selected. By changing the scope of a drawing you are able to manage many different aspects of the drawing.
Component Manager

Select the Component Manager option from the View menu. The component manager enables you to retrieve certain existing information from existing drawings into the current drawing. Caddie displays the Component Manager located at the left of the screen.
Features of the component manager

Manage drawings on your computer by browsing the **Shell Tree**.
You can change the way in which drawings are listed by selecting a different view. Change view by selecting the **Choose View** button.
You can Show or Hide the drawing preview. Click on the **Show/Hide Preview** to toggle this option.
You can Show or Hide the drawing descriptions. Click on the **Show/Hide Description** button to toggle this option.
You can set the manner in which a drawing is inserted into the current drawing by setting it to either the **Original Layer** or the **Active Layer**. Click on the **Choose Layer** button to change this setting.
You can insert the selected blocks in a drawing or an entire drawing into the current drawing by clicking on the **Insert Drawing** button.
When this option is switched on, double-clicking on a drawing in the drawing list pane to the right of the **Shell Tree** pane will insert the selected drawing into the current drawing.
Caddie prompts you to **Select position for symbol**. Click on a suitable point in the drawing to indicate the location of the inserted symbol.
Continue clicking in the drawing to indicate other locations for the symbol.
Select **Option End** to terminate the command.
When the **Insert Drawing** is switched off, you can view properties of a drawing by clicking on the drawing name in the **Shell Tree** pane. In the following example, layer properties are being viewed for the selected drawing.

**Preview layers in the component manager:**

![Component Manager Screen](image)

Preview blocks in the component manager
Layers, Layouts, Linetypes, Tablestyles and Xrefs

The same way you insert blocks from an existing drawing into your current drawing, you can also retrieve all of the above-mentioned properties from one drawing into your current drawing.

Retrieving Layers

A layer with all the above-mentioned properties can be inserted into your current drawing with the component manager. See the Chapter 6 – Working with Layers for a detailed discussion on setting up layers.

Layouts

If you have an existing sheet with settings you want to re-use in your current drawing, you can retrieve that particular sheet from an existing drawing using your component manager. See the Chapter 3 – Working with Sheets for a detailed discussion on setting up sheets.

Linetypes

Sometimes when you open a drawing that was packed or purged, Caddie will only display a list of used line styles in your list of line styles. If you have a drawing with a line style that you want to use, you can retrieve the line style from the existing drawing into the current drawing with the component manager.

Tablestyles

On the pull down menu, under advanced objects, you have the option of drawing a table in a drawing. If you have an existing drawing with a table in it and you want draw a table having those particular styles, you can retrieve the table into the current drawing by using the component manager.

XRef

Drawings can consist of drawn entities and external reference drawings. If you have an existing drawing with Xrefs (referenced symbols), you can retrieve the XRef into your existing drawing with the component manager.
Mouse Button Set-up

The context sensitive menu enables you to make quick selections and transformations efficiently. You use the more traditional snap menu in conjunction with the mark commands to perform translations, or you can use the context sensitive menu. To setup these features, select the Mouse Button Setup from the Settings menu.

With the context menu setup as indicated in the previous dialogue, you can select objects simply by single clicking objects. You can also select multiple objects by tracing a rectangle around the objects. Once selected, the objects can then be manipulated by using options from the popup Context Sensitive menu.

Object Selection

When selecting objects with the mouse, please note the following:

If you move the mouse from **Left to Right**, only entire entities included in the selection rectangle will be selected. The selection rectangle will be a solid line.

If you move the mouse from **Right to Left**, all entities that are included in the selection rectangle as well as entities that the rectangle crosses will be selected. The selection rectangle will be a dashed line.

The Context Sensitive menu

The context menu is sensitive relevant to the current command or object selection. The following are examples of context menus:

- If no objects are selected (A);
- If an object has been selected (B);
- If you are in the process of editing a block (C);
- If you are in the process of copying or moving objects (D);
Quick Selection

You can select various objects in a drawing very quickly by using the Quick Select option.

**Selection:** Select which parts of the drawing will be affected.

**Object types:** All the object types in the drawing is listed. Select the object types you want to select. You can perform multiple selection sets by using a combination of the [Ctrl] and [Shift] keys.

**Properties:** Select the specific property of the objects you want to select. Only one property can be selected at a time.

**Operator:** Select the appropriate operator based on the properties selected. This can be equal, not equal, less than & greater than.

**Value:** Select or enter the desired value for the properties selected.

You can, for example, select all the circles in a drawing with a radius less than a certain value, or you can also use the quick select to select hatch of a certain colour or you can select objects of a certain z-height.
CHAPTER 2

Environment and Drawing Setup

- Setup Environment
- Setup Backup Options
- Setup Drawing Defaults
- Work with Templates
- Configure the Screen Layout
- Customise the Keyboard
- Customise Toolbars
CHAPTER 2 - Environment and Drawing Setup

The Environment Settings Dialogue

Select Environment Settings from the Settings menu to display the dialogue. There are a number of settings that will influence the drawing environment.

Clipboard

When copy and pasting to other software, you can specify the image type as Windows Metafile or Bitmap. You can choose if a plot style should be used or not.

Component Manager

When using the component manager you can specify that the drawing be inserted as blocks or symbols.

Control Points

Specify the colour and size of control points, for example hatch control points.

Cursor

You can specify if you want to use a long or a short cursor. You can specify the width, length, centre gap and colour for the long cursor and select if text (distances etc.) are shown on the cursor, or not.

Folders

Here is a list of the default folder locations Caddie uses.
Applications: This folder contains all compiled menus that are shipped with Caddie as well as any application menus developed by yourself or any 3rd party developer. It is advised not to change this location.

Auto Backup: This is the default folder where backups of your drawings will be stored. You can change this location to suit your needs.

Main Directory: This is the default folder containing the Caddie executable and other files. It is advised not to change this location.

Drawings: This is the default folder of all your drawings. You can change this location to be anywhere either on your local computer or on a computer located on your network. If you are connected to a network, it is recommended that you change this location to point to a location where you know that backups are being performed on a daily basis. This will ensure that you always have a backup copy of your drawings.

**KEY CONCEPT**

There is a difference between a normal Caddie drawing file (.DRW/.DWG) and a back up file. Do not use the Auto Backup feature as a means to saving drawings! Backup files have a different extension to drawings and are saved with a .DRB extension.

Fonts: This folder contains the Caddie shape fonts.

Images: The location where you store any images that you wish to load into Caddie. This can be a folder on your local computer or on a network drive.

Plot Styles: This is the location of all your different plot styles.

Referenced Symbols: This is the location of referenced symbols used in your drawings. It is advisable not to change the location unless you need to load similarly named referenced symbols from another location.

**KEY CONCEPT**

Be cautious when changing the location of RefSym folder. When you load a RefSym into a drawing, Caddie remembers the location of the RefSym. If you now change the location of the RefSym, Caddie will not be able to load the RefSym unless you have made a copy of the RefSym in the new location.

Symbols: The location of symbols used in drawings.

Hatch

The hatch settings control the density of hatch patterns that will be displayed, the scaling factor of hatch patterns for older Caddie drawings and the minimum hatch size to rescale.

Installation

You can set the country for the Caddie installation. This will influence location of Caddie downloads from the web.
Layering

You can specify if a layer definition file needs to be loaded at start-up, including the name and location of this file.

Object Properties

You can control the combo dropdown lengths of the colour, line type, line width and listbox options in object properties.

Online Updates

You can switch automatic notifications, software updates, maintenance dates and tips from Caddie on or off.

Patterns

You switch the initialisation of a pattern at start up on or off, and set the current pattern to be highlighted or not.

As a pattern can reside on more than one layer it is important to switch the initialisation of patterns on layer change and merging objects on if you wish to restrict patterns to reside only on one layer.

Selection

You can control all the normal grip point and AEC object grip point switches, sizes and colours to enable you easy identification on the screen.

Snap

Auto-enter on secondary snap modes: You can switch the automatic enter on secondary snap modes of the current cursor position on or off.

Snap Aperture size: This defines the search area when moving the mouse in close proximity to objects. The smaller the aperture size, the closer you need to move the mouse to an object to snap to the object.

Snap Marker size and colour: This controls the size and colour of the snap marker.

Translation

This controls the automatic unmarking switch. If it is switched on Caddie will automatically unmark the scope of the drawing when you select any of the transformation commands. The default is off.

Zoom

You can control the dynamic zoom factor and the mouse wheel factor as a percentage of the screen.

The Drawing Settings Dialogue

Select Drawing Settings from the Settings menu to display the dialogue. There are a number of settings that will influence the drawing settings.
Opaques, Pictures and Sheets:

You switch the boundary lines around opaques, pictures and sheets on or off.

Text:

You can switch the display of TrueType fonts filled, on or off. If the drawing contains a large number of text strings, the display will be faster if the filled TrueType font display is switched off.

The Auto Backup Settings Dialogue

Select **Automatic Backup Settings** from the **Settings** menu to display the dialogue. Caddie has a built-in feature whereby it can perform backups of drawings that you are currently working on.

**Automatic Backup Mode**

You can set Caddie to perform backups automatically, reminder only and reminder and do backup. You can also disable the Auto backup.

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**Disable Auto Backup WARNING**

*Working without any form of backup is strongly NOT recommended. Power failures can occur and therefore necessitates the need to have some kind of backup of your work.*
Retention Time

You can set the period of time you wish to retain a copy of backup files in the backup folder. This value is set in Days. You can also keep the backup files indefinitely. Bear in mind that this can cause your backup folder to grow rapidly if you are working with large drawings. Remember to delete older backup files manually through Windows Explorer.

Frequency

Set the time, in minutes, between backups. Use careful judgment here to suit your style of working. By default, Caddie sets this value to 10 minutes. Remember that you can do major changes to a drawing in 10 minutes so you might like to reduce this to 5 minutes.

Clear Backup Folder

When you select this button all the files in the current backup folder will be deleted.

Backup Folder

This is the location of the folder where your backup drawings will be saved. This can be on your local computer or on a network drive.

Auto Backup Naming Convention

Caddie uses a specific naming convention when making backups of drawings. Assume that you are working on a drawing with a name of Drawing101.dwg.

When it comes time to perform a backup of this drawing, Caddie will name the backup file using the following naming convention:

YYYYMMDD-hhmmss-DrawingName.drb

Where YYYY is the year, MM is the month, DD is the date – hh is the hours, mm is the minutes and ss is the seconds is added to the drawing name and given an extension .drb.

The backup of Drawing101.dwg can appear as 20100921-103010-Drawing101.drb.

The Drawing Defaults Dialogue

Select Drawing Defaults from the Settings menu to display the dialogue. These settings control
the drawing defaults when opening and saving drawings.

### Default Open Option

- **New Window**: When opening a drawing, Caddie defaults to open the drawing in a new window. This is the preferred option as the Replace Current and Merge Modelspace options have drastic implications on your current drawing.

- **Replace Current**: When you open a new drawing Caddie will prompt you to delete the current drawing before opening the new drawing in the current window. This method **DOES NOT SAVE** any changes when replacing the current drawing. You will have to save the current drawing BEFORE using this method to open a drawing.

- **Merge Modelspace**: When you open a new drawing Caddie will **MERGE** it with the current or active drawing.

### Drawing Preview

- **Current View**: The current view of the drawing on the screen will be used as the drawing preview.

- **Drawing Extents**: The preview will be generated from the extents of the drawing.

- **No Preview**: No preview will be generated from your drawing.

- **Background Colour**: You can set the background colour for the drawing preview for easy identification when listing drawings.

### Extension (Drawings)

You can set the default file extension for opening and saving drawings. The DRW and DWG extensions are normally used. At any time during an open you can select **Files of type** on the dialogue and change the drawing extension. (DRW, DWG, DXF, All drawing file or All files.) During a save you can select **Save as type** on the dialogue and change the drawing extension. (DRW, DWG, DXF or CTP.)
**Environment and Drawing Setup**

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**Extension (Symbols)**

You can set the default file extension for opening and saving symbols. This can be DRW or DWG. The symbols supplied with the software uses the DRW extension.

**DWG (DRW) Version**

This option determines the default file version in which Caddie drawings will be saved (DWG or DRW extension).

**General**

These general options determine some settings concerning the opened or saved drawing.

- **Auto Increment Revision Number:** If checked, Caddie will automatically increment the drawing revision number every time you save a drawing.
- **Retain last drawing path on exit:** Caddie will remember from which location you retrieved the last drawing you worked on.
- **Retain drawing name when reading V9 or earlier drawings:** Caddie will retain the drawing name of any Version 9 or earlier version drawing when you open such a drawing. Make a backup of your Caddie 9 (or earlier) drawings if this setting is switched on as it’s very easy to overwrite a Caddie 9 (or earlier) drawing.
- **Use recycle bin when overwriting local drawings:** Caddie will make a copy of the drawing in the recycle bin before overwriting a drawing when saving.
- **Create backup copy when overwriting drawings:** Caddie will make a backup of the current drawing before overwriting the drawing being saved. The location of this backup will be in the drawing folder. (.DR!)

**The Templates Dialogue**

Every time you start a new drawing, Caddie can load a pre-defined template as the preferred modelspace to use as the default. This is extremely useful as you can then set up a number of different modelspaces that Caddie can use when creating new drawings. This is also where you specify which titleshet drawing to use.
Start-up Defaults

**No template:** Caddie does not load any template and will use its own default settings for the drawing.

**Use default template:** Use the selected template as a default to load every time you create a new drawing or whenever you activate Caddie.

**Browse template:** If you want to select a different template every time you create a new drawing or whenever you activate Caddie, select this option. Caddie will then display all the available templates in the designated templates folder and you can then select the desired one to load as the default drawing.

Default template to load

If you want to use this functionality effectively, you first need to setup a modelspace the way you want it to be and then save it in the Templates folder of your Caddie installation. Select the **Browse** button to point to the desired file.

Template folder

If you prefer, you can change the location of the templates folder. Select the **Browse** button to change the folder. This is useful if you are working in an environment where several people need to access the same templates in a drawing office. By setting this folder to a common location on a network drive that is accessible to everyone, all users will have access to these templates.

**Titlesheets drawing**

Specify the location of the titlesheet drawings to be used when creating sheets for printing (see later in the discussion on printing in Caddie).

**The Selection Colour and Width dialogue**

Use this function under the Settings menu to change the way in which marked objects, selected objects, virtual objects and current pattern is displayed on the screen.

You can set the display colour, line width and line pattern of marked objects, selected objects and virtual objects for easy identification on the screen. By selecting a combination of these three options you can create a style suitable to you when visualising marked, selected and virtual objects in a drawing. The line pattern is represented by * for a space and _ for a line.
You can set a highlight colour for the current pattern.

**Use polygon pattern:** If you tick these options under Marked or Selected, hatches, faces, etc, will be displayed as diagonally patterned mesh (in the specified colour) for easy identification on the screen.

**Use highlighted settings at collision:** This setting is not currently in use.

### The Drag Settings dialogue

![Drag Settings Dialogue]

You can set the thickness and colour of your dragging, as well as the dragging angle when the [TAB] key is used in combination with commands such as lines, construction lines and transformations.

### The Mouse Button Setup dialogue

![Mouse Button Setup Dialogue]

The way the buttons on your mouse are configured will influence the way you work in Caddie.

The default installation setup is **Left Single Click** - Object selection; **Left Double Click** – Pattern selection; **Right Single Click** – Context sensitive menu.

**Left Single Click**

- **Object selection**: A single object will be selected.
- **Object properties**: The object properties dialogue will be displayed.
- **Pattern selection**: The pattern containing the indicated object will be selected.
- **None**: No selection will be made.
Left Double Click

- **Object selection**: A single object will be selected.
- **Object properties**: The object properties dialogue will be displayed.
- **Pattern selection**: The pattern containing the indicated object will be selected.
- **None**: No selection will be made.

Right Single Click

- **Context sensitive menu**: This selection will show the relevant option allowed with certain selections and commands. Assigning the context sensitive menu to the right click mouse button will give you access to a range of quick options that are only a click away.
- **Snap menu**: This selection will show the snap menu Caddie 9 and earlier versions used. This mouse button setup will also allow you the custom menu on the right double click.

**Important note**: Even if this setup is more familiar to some users, PLEASE use the right click Sensitive menu option – The intelligence of the range of right click options far outweigh the familiarity of the older snap menu.

The Favourite Colours dialogue

You can set your frequently used colours by selecting the colours in succession on the dialogue box. These colours will be displayed at the top of the pen colour list and make quick colour selection.

The Screen Layout

You can customise different screen layouts for different working environments. These screen
layouts can be saved to memory, recalled from memory; exported to a file, and imported from a file.

Remember to export your screen layout to a file for backup purposes and if you want to import the screen layout on another pc.

Customising the Caddie Screen Layout

When you install Caddie, the Applications toolbar is located at the extreme left of the screen with the Command menu immediately to the right of it.

By right clicking on either menu you can change the display of these two menus by selecting an appropriate option to suit your style of working. For example, you could set the Applications menu to Text only whilst setting the Command menu to Picture only.

- **Text Only**: Display menu items as text only.
- **Picture Only**: Display menu items as pictures only.
- **Text and Picture**: Display menu items as pictures with text under the picture.

You can increase or decrease the number of columns for the Command Menu by using the right-click decrease/increase options.

If you wish to relocate any of these menus, you can drag them to any position on the screen. You do this by moving the cursor to the title of the relevant menu and then click and drag it to the desired new position.

Select the View | Change Themes command to change the theme all menu displays.

Managing Applications

As you work in Caddie, you will often need to switch between applications depending on the commands you wish to execute. By default, the applications menu is located at the extreme left of the screen. If you have not loaded the required applications at installation time, you can simply right click anywhere on one of the loaded applications that is visible and select Load Application. Only the application available for loading will be listed. Similarly an application can be unloaded.

Command Line

The command line is Caddie’s means of communicating with you. As you execute commands, Caddie prompts you to respond in the execution of the command. As you learn more about Caddie and become more proficient in using Caddie, you will find that you will need to refer to the prompt line less and less. The command line is used to store a history of executed commands.

By right-clicking anywhere in the history buffer located just above the prompt line, you can perform the following actions:
You can select all and copy the text currently stored in the command buffer; you can also select just a value in the command line and copy this to another dialogue or application.

You can clear the contents of the command history buffer and you can save the contents of the command history as a text file for later use.

If you select the Buffer = option, Caddie will allow you to paste the most recent requested measurement to a dialogue or application.

You can display the most recent commands and you can also list the most used commands during your working session.

**Toolbar Customisation**

You can customise toolbars by adding, removing, reshuffling and re-locating them as you see fit. Take, for example, the Lines toolbar - if it is not visible, turn it on under the View menu - dragging this toolbar off its original docking position anywhere else onto the screen look something like the following illustration. To drag it to the docked position put the mouse over the vertical grey bar to the left end and holding the left button down while dragging.

You can now place this toolbar anywhere on the screen by dragging it around with the mouse (left button) to a new location. You can also resize it so that it is more compact such as the illustration below by dragging any of the edges of the box.

When you click on the down arrow at the top right of the toolbar, the **Add Remove Buttons** will appear. When you move your mouse to this option, the name of the toolbar – in this case **Lines** and **Customise** will appear. Click on **Customise** to display the **Customise** dialogue.

**Switching toolbars on or off**

The first tab on the customising dialogue displays a list of the available toolbars and their visibility. To change the display of any toolbar simply check or un-check it.

**Removing buttons from a toolbar**
If you want to remove buttons from a toolbar make sure that it is visible and simply drag the buttons that you no longer wish to be associated with the toolbar off the toolbar onto the screen. The button is now removed from the relevant toolbar.

**Adding buttons to a toolbar**

To add buttons to a toolbar you need to display the Commands tab in the customising dialogue. Select the application menu from which Categories list on the left from which you wish to copy commands. From the Commands list on the right; drag the button you wish to copy onto one of the visible toolbars. You can place the button anywhere on the toolbar. As you move over the target toolbar, you will see a Splitter cursor appear telling you where the button will appear once you drop it onto the toolbar. Continue this process until you have added all the buttons you need on a toolbar. To re-locate a button on a toolbar simply drag it to a new position. You can place buttons from any command category onto any toolbar. The concept of customisable dockable toolbars is extremely useful in grouping commands that are used in conjunction with each other.

**Copying buttons from one toolbar to another**

You can copy buttons from one toolbar to another without having to go through the customisation procedure. Make sure that both toolbars are visible. Move the mouse to the button on the toolbar you want to copy; press and hold down the [Ctrl] key; now drag the button onto the other toolbar and place it where you want it to appear.

**Toolbar appearance**

You can also change the way in which buttons are displayed on toolbars by first selecting the toolbar you want to change on the Customise dialogue, and then use any of the following options:

- **Small/Large:** Display small or large icons on your tool bars. (You cannot change the size of the standard Caddie toolbars.)
- **Text:** Display text only.
- **Picture:** Display picture only.
- **Text and Picture:** Display both text and picture.

**Create toolbars for frequently used commands**

You can create as many of your own customised toolbars as you wish. Whilst you can modify standard Caddie toolbars this is not advised as they may not transfer correctly to another version of Caddie. To create your own toolbar, make sure that the Toolbars tab is displayed. Let’s say you wish to create a new toolbar named Architectural. Click on the New button and type Architectural as the name of the new toolbar. When you click on OK, Caddie displays the new toolbar close to the Customising dialogue.

Let’s say that you want to create yourself a toolbar containing commands that you use frequently. For this exercise we will build a toolbar using the following commands:

- **LINES - Auto; LINES - Rectang; LINES - Parlel; CURVE - C-2P;**
- **CURVE - ArcL3Pt; TRANS - Stretch; DELETE – DelObj; DELETE – DelCon.**

Right-click on any toolbar and select the Customise option, click on New to create a new toolbar and type in the name **Utils** and Press [ENTER] or click on OK to continue. Caddie creates the new toolbar and places it close to the Customise dialogue. This new toolbar contains no buttons at this stage.
Let’s start with the **Auto** command; click on the **Command** tab; select the **Caddie|LINES** category to display all the **LINES** commands. Click and drag the **Auto** button onto the new toolbar and release the mouse button. The **Auto** command button appears on the new toolbar. Drag the **Rectang** and **Parleg** buttons onto the toolbar in the same manner.

Click on the **Caddie|CURVE** category and drag the **C-2Pt** and **ArcL3Pt** buttons onto the **Utils** toolbar; Click on the **Caddie|TRANS** category and drag the **Stretch** button onto the toolbar; Click on the **Caddie|DELETE** category and drag the **DelObj** and **DelCon** buttons onto toolbar.

This completes the customisation of the **Utils** toolbar. You can dock this toolbar on any side on the screen.

**The Docking Snap Menu**

One of the built-in toolbars is the Docking Snap Menu. This toolbar contains all the snap modes in Caddie on one toolbar. To display this toolbar, select the **DockingSnapMenu** option from the **View** menu. Just like any other toolbar, you can locate this toolbar on the side of the screen or as a floating menu. If you don’t use all the snap modes, create a new toolbar containing your favourite snap modes – **copy** the snap modes from the Docking Snap Menu to your new toolbar.

---

**WARNING**

Do not move any items from the Docking Snap Menu as you will have to re-install the Caddie software to restore them, copy buttons by pressing the [Ctrl] key.

---

**Keyboard Customisation**

An important area of customisation in Caddie is the ability to map commands to the keyboard. This enables you to speed up drawing time in that you do not have to access toolbars or scroll through menus to gain access to a specific command. It is a good idea to map the command that you use very frequently to your keyboard. This makes commands available simply by pressing a key (or combination of keys).

Just as you customise toolbars, right-click on any toolbar and click on the **Customise** option. From the **Customise** dialogue, select the **Keyboard** tab.
Managing shortcut keys is easy. In the example above, the key combination of **Ctrl-N** has been mapped to the **File | New** command. In other words, pressing **Ctrl-N** will invoke Caddie to create a new drawing. To customise your keyboard shortcut keys, simply select the **Category** and **Command** on the left; click in the **Press new shortcut key** edit box and then press the new shortcut key or combination of keys on your keyboard; select **Assign** to map the shortcuts to the keyboard. Caddie is installed with a range of default keyboard shortcut assignments. Some of these shortcuts are system defined and cannot be changed, for example the [Esc] button that is mapped to the **Abort** command.

### Default Keyboard Shortcuts

Here is a list of the default Caddie shortcut keys that are installed with the Caddie software:

<table>
<thead>
<tr>
<th>Shortcut keys</th>
<th>Command:</th>
<th>Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shift+F1</td>
<td>Help - Search</td>
<td></td>
</tr>
<tr>
<td>F1</td>
<td>Help</td>
<td></td>
</tr>
<tr>
<td>F7</td>
<td>Print</td>
<td>Print a drawing</td>
</tr>
<tr>
<td>Ctrl+V</td>
<td>Paste</td>
<td>Paste</td>
</tr>
<tr>
<td>Ctrl+C</td>
<td>Copy Selected</td>
<td>Copy Selected</td>
</tr>
<tr>
<td>Shift+C</td>
<td>CopyM</td>
<td>Copy Marked</td>
</tr>
<tr>
<td>Ctrl+X</td>
<td>Cut</td>
<td>Cut Selected</td>
</tr>
<tr>
<td>Shift+X</td>
<td>CutM</td>
<td>Cut Marked</td>
</tr>
<tr>
<td>I</td>
<td>Initial</td>
<td>Activate new pattern - deactivate all others</td>
</tr>
<tr>
<td>Ctrl+S</td>
<td>Save</td>
<td>Save active Drawing, current name</td>
</tr>
<tr>
<td>O</td>
<td>InsSym</td>
<td>Insert Symbol</td>
</tr>
<tr>
<td>Ctrl+H</td>
<td>DelHch</td>
<td>Delete a hatch definition</td>
</tr>
<tr>
<td>Keyboard Shortcut</td>
<td>Command</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------</td>
<td>--------------------------------------</td>
</tr>
<tr>
<td>Ctrl+Y</td>
<td>Redo</td>
<td>Redo</td>
</tr>
<tr>
<td>Ctrl+Z</td>
<td>Undo</td>
<td>Undo</td>
</tr>
<tr>
<td>A</td>
<td>LayAct</td>
<td>Set the active layer</td>
</tr>
<tr>
<td>Ctrl+E</td>
<td>MarkObj</td>
<td>Mark an object</td>
</tr>
<tr>
<td>Ctrl+P</td>
<td>MarkPat</td>
<td>Mark indicated patterns</td>
</tr>
<tr>
<td>Shift+A</td>
<td>SelectAll</td>
<td>Select All</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Context_selectall</td>
<td>Select All</td>
</tr>
<tr>
<td>Shift+U</td>
<td>UnSelectAll</td>
<td>Unselect All</td>
</tr>
<tr>
<td>Ctrl+U</td>
<td>Context_unselectall</td>
<td>Unselect All</td>
</tr>
<tr>
<td>R</td>
<td>Regen</td>
<td>Regenerate drawing</td>
</tr>
<tr>
<td>Ctrl+DOWN</td>
<td>Pan-D</td>
<td>Dynamic Pan DOWN</td>
</tr>
<tr>
<td>Ctrl+UP</td>
<td>Pan-U</td>
<td>Dynamic Pan UP</td>
</tr>
<tr>
<td>Ctrl+RIGHT</td>
<td>Pan-R</td>
<td>Dynamic Pan RIGHT</td>
</tr>
<tr>
<td>Ctrl+LEFT</td>
<td>Pan-L</td>
<td>Dynamic Pan LEFT</td>
</tr>
<tr>
<td>PGDN</td>
<td>Zm-Out</td>
<td>Zoom out twice</td>
</tr>
<tr>
<td>PGUP</td>
<td>Zm-In</td>
<td>Zoom in twice</td>
</tr>
<tr>
<td>P</td>
<td>Pan</td>
<td>Pan</td>
</tr>
<tr>
<td>Z</td>
<td>ZmBox</td>
<td>Zoom into the drawing</td>
</tr>
<tr>
<td>F6</td>
<td>ZmLast</td>
<td>Restore PREVIOUS zoom state</td>
</tr>
<tr>
<td>F5</td>
<td>ZmAll</td>
<td>Back zoom state</td>
</tr>
<tr>
<td>F4</td>
<td>ZmExts</td>
<td>Zoom to the drawing extents</td>
</tr>
<tr>
<td>F10</td>
<td>PnLnSet</td>
<td>Set the pen and line parameters</td>
</tr>
<tr>
<td>L</td>
<td>Draw</td>
<td>Draw a series of joined lines</td>
</tr>
<tr>
<td>DEL</td>
<td>Delete</td>
<td>Delete</td>
</tr>
<tr>
<td>F8</td>
<td>V9SPopup</td>
<td>Show V9 Popup snap menu</td>
</tr>
<tr>
<td>F9</td>
<td>V9MPopup</td>
<td>Show V9 Middle button menu</td>
</tr>
<tr>
<td>F11</td>
<td>Context</td>
<td>Context</td>
</tr>
<tr>
<td>Ctrl+N</td>
<td>New</td>
<td>Start New Drawing</td>
</tr>
<tr>
<td>Ctrl+O</td>
<td>Open</td>
<td>Open drawing</td>
</tr>
</tbody>
</table>

Once you have mapped your frequently used commands to shortcut keys and start to use them, you will see a marked improvement in your productivity.
Caddie Utilities

Caddie has a number of utilities to help you manage different styles and batch commands.

**Insert V9 Titleblock in current sheet**

You can insert a Caddie version 9 (or older) drawing into the current sheet. This drawing can be a title sheet drawing. This older drawing will be inserted and the drawing will be scaled so that the extents of the drawing fit into the current sheet. Please be aware that if elements reside outside the title sheet drawing it will cause an incorrect scaling of the title sheet border.

**Saved Preset Manager**

Use the saved preset manager to manage presets styles of various Caddie objects:

You can manage your styles with the Caddie Style Manager that allows you to delete and copy styles. You can also import styles from previous versions of Caddie - including Caddie version 9 (or older) style files.

**3D Arch Style Manager**

The 3D architectural style manager is used in conjunction with the AEC objects and is covered in the 3D section of the manual.

**Display Manager**

The display manager is used in conjunction with the AEC objects and is covered in the 3D section of the manual.
Chapter 2  Environment and Drawing Setup

Calculator

The calculator allows normal, scientific and brick size calculations.

System

Migrate Settings from Caddie #: You can import settings from a previous version of Caddie. You can select to import the registry settings, plot styles, vector fonts and the title sheet drawing from the specified folder.

Set system variable: This is mainly used by programmers to set system variables.

Reset visibility of objects: If you have switched the visibility of certain objects off on the Properties dialogue, the only way you can switch the visibility on is to use this utility. You will be prompted to indicate Scope, Polygon, Rectangle or Pattern.

Delete Hidden Hatches: Use this command to delete invisible hatches (or hidden hatches) from a drawing.

Update Weblinks: This command will update all weblinks to the latest setting on the Caddie website.

Wizards – Viewport Wizard

Use this command to create a viewport by following the step-by-step wizard. Read the instructions and follow the prompts to create a viewport from the modelspace.

Batch

Convert Symbol Folder to Block drawing: Use this command to convert all the drawings (symbols) in a folder to a named drawing containing these drawings (symbols) as blocks. The drawing name is used for the block name Caddie reads the symbols into the drawing and displays the Edit Block Description dialogue. Make any to the descriptions for these blocks in the drawing. You can use the component manager, discussed earlier in the first chapter, to load blocks from drawings.

Convert Symbol Folder to Object & preview colours: If you do not agree with any default colours as used in symbols, you can change their appearance. These can be done in batches using the specified folder. Any folder of Caddie drawings can also be changed with this command.

- Objects to change: Select the objects you want to change. If you want to select all the objects in the drawing, simply tick the All Lines / Curves box.

- Colour Mapping: Select the mapping of existing colours to new colours.

- Scaling: If you need to rescale the symbols, tick the Rescale box and enter a scaling factor.

- Units: You have the option to change the units of the symbols while converting them. This is handy when symbols have no units assigned, or even defaults to inches.

- Preview Background Colour: Select the preferred background colour for the symbols.
File Types to Convert: You can tick DRW to convert only native Caddie file formats; you can tick DWG to convert any DWG file formats; or you can tick both to convert all drawing files.

Blocks: You have to option to explode all blocks contained in the symbols.

Include Nested Blocks: This option allows you to explode all blocks and nested blocks contained in the symbols.

Folder to Convert: Browse to the selected folder for the batch symbol conversion.

Insert Geo-referenced ECW images: Browse to a folder that contains Geo-referenced ECW images. Caddie will insert all the images in the folder at the correct coordinates.

Browse

Common Application Data Folder: This command will browse to the folder on the computer that contains files common to all users on the computer like styles, plotstyles, hatch definitions etc.

User Application Data Folder: This command will browse to the folder on the computer that contains files that are user specific like backups and screen layouts.
CHAPTER 3

Creating and Managing Sheets

Create Sheets
Modify Sheet Properties
Insert title Blocks from Templates
Create View ports from Modelspaces
Rescale and Resize Viewports
Manage Viewports
Managing Sheets - Copying, Deleting, Reordering
CHAPTER 3 - Introduction to Sheets

A drawing consists of a modelspace and one or more sheets. The modelspace contains the drawing(s) and the sheet contains the title sheet drawing and the viewports reflecting the drawing from the modelspace to the sheet at specific scales.

Creating a new sheet

NewSht  Create a new sheet.

Caddie prompts you enter a name for the new sheet. Enter a valid name for the sheet. Note that names for sheets are unique for a drawing.

Setup of a sheet

SetSht  Setting up a sheet for printing.

Whenever you create a new drawing, Caddie creates two new blank sheets by default. These two sheets are named Sheet1 and Sheet2. Select the SetSht command from the Sheets menu to set up your sheet for printing and to add your titlesheet information.

Caddie display the Sheet Setup dialogue:

Sheet Settings

Sheet Name: Enter a name for the sheet. Use common sense here and name the sheets using practical meaningful names. If the sheet will print the floor plan of a building you could name it Floor Plan, for example. Try to keep the name as short as possible. The longer the name the larger the sheet tab and therefore less tabs can be displayed on the screen. You can change the order of these tabs with the Sheets | Reorder command.

Printer / Plotter: Select the printer to which the drawing will be sent for printing. It is necessary to set this up first as the printing preferences are used to determine the paper sizes available. If you
use this printer as your main printer you may like to tick the **Set printer as default for new sheets** tick box. Ticking this will save having to select a printer when ever you create a new sheet.

**Paper Size:** Select the actual paper size the drawing will be printed on.

**Orientation:** Select **Portrait** or **Landscape** orientation.

**Paper Units:** The paper units can be set to **Millimetres** or **Inches**. (The default is mm.)

**Plot Settings:**

**Plot Style:** Select a pre-defined plotstyle to use for printing the sheet. Go to **File | Configure Plotstyle...** command for setting up your plotstyles. If you use this plotstyle for most of your printing you may like to tick the **Set style as default for new sheets** tick box. Ticking this will save having to select a plotstyle when ever you create a new sheet.

**Plot Scale:** This indicates to Caddie the relative size at which to print the drawing. We normally add the scaling factor to the viewports on the sheet and using a plot scale of 1:1. This is a safe method to get an accurately scaled drawing to be printed. If you set the plot scale to 1:2, the printed drawing will be half the size and using a plot scale of 5:1 will result in an enlarged drawing by a factor of 5.

**Plot Offset:** It is most practical to set this value to **Centre Plot** so that the drawing is always centred on the printed sheet.

**View port line scaling:** You can select if the line scaling for broken lines must be on or off in the viewports. The default is to switch the line scaling on.

**Miscellaneous:** If the **Plot viewports first** option is selected, the viewport data will be printed before any sheet information. This is also the default.

**Preset Manager:** This functionality allows you to apply all the parameters of one sheet to other sheets, and you can also import the setup from another drawing.

---

**KEY CONCEPT**

Your sheets might already contain titlesheet information if you are using a template which has the titlesheet information added.

---

**Inserting a title block into a sheet**

<table>
<thead>
<tr>
<th>T-Sh</th>
<th>Insert a title block from a pre-defined titlesheet drawing into a sheet.</th>
</tr>
</thead>
</table>

Caddie supplies a sample titlesheet drawing with the system. This contains a number of pre-defined titlesheets for adding to your sheets.

This sample titlesheets drawing is **Titlesheets.drw**. You can customise this drawing by changing the titlesheets and re-saving it or you can replace the drawing with your own customised titlesheets drawing.

You can set the titlesheet drawing location with **Settings | Templates** command. Select the **Browse** button next to the **Titlesheets** drawing location to change it.

The default **Titlesheets.drw** displays the following pre-defined sheets:
Creating a custom titlesheets drawing

Open the default titlesheets drawing - \texttt{Titlesheets.drw} – and customise the different sheets to suit your requirements - you can draw your own titlesheet on a sheet and add it to the titlesheet drawing or you can also insert a Caddie 9 (or earlier version) titlesheet on the sheet, and then add it to the titlesheets drawing. In the example we will create a new \texttt{Titlesheets.drw} and add Caddie 9 titlesheet drawings to it.

\textbf{Open a Caddie 9 titlesheet drawing:} File | Open and select the Caddie 9 drawing.

\textbf{Setup your units:} Settings | Modelspace Workspace... and set the units to millimetres.

\textbf{Check the drawing measurements:} MEASR | Meas-X and Meas-Y, measure the borders of the titlesheet, if the measurements are not correct, use the Trans | Rescale command to scale the titlesheet to the correct 1:1 scale.

\textbf{Select and copy the drawing:} Select the titlesheet by indicating a rectangle around all the titlesheet elements; right-click and select Copy to Clipboard, and indicate an origin on the drawing.

\textbf{Open a new drawing:} Select File | New and open a new drawing. Click on Sheet 1 to activate the sheet.

\textbf{Setup the sheet:} Select the Sheets | SetSht command and do the sheet setup. It is important to give the sheet a name that easily identifies the sheet size, for example, A0 Title Project A. Pick the required printer and select the sheet size, and the plotstyle.
Paste the drawing from the clipboard: Right-click and select Paste from Clipboard and indicate the correct position on the sheet. Change or edit the titlesheet to suit your requirements.

This new drawing contains no elements in the modelspace, but has a titlesheet drawing on the first sheet.

Select File | Save As... and browse to the default Titlesheets.drw and overwrite it.

Add all your titlesheets in a similar fashion and update the Titlesheets.drw drawing.

When you select the Sheets | T-Sht command, your custom titlesheets will be displayed for selection:

Deleting sheets

DelSht Delete the current sheet.

Use this command to delete a sheet(s). The sheet to be deleted MUST be the currently selected sheet.

Reordering sheets

Reorder Change the order of sheets.

Use this command to change the order of sheets. Caddie displays the Reorder Sheets dialogue. To change the order of sheets, select the one you want to reorder and then use the Up and Down buttons to move the sheet up or down the order. Repeat this action for any other sheet you want to reorder.

Copying sheets

CpySht Copy the contents and properties of a sheet.

Use this command to copy the contents and properties of a sheet. Enter name of the new sheet and tick Copy viewports option if you want to copy the viewports as well.

Creating and Managing Viewports

What is a viewport?
In simple terms, a viewport is merely a window through which you view the drawing in the modelspace. Viewports can easily be moved, suppressed, resized and rescaled.

### Creating a new viewport

**NewVP** Create a new viewport on a sheet.

Caddie displays the **Plan Viewport** dialogue.

**Shape:** Note that the **Rectangle** option will enable the **Size** options where you can select from a pre-defined size, pick a size with the mouse or enter a custom size for the rectangular shape. The **User Defined** option will prompt you to trace a polygon defining the shape you want to describe for this option.

**Orientation:** Select the paper orientation for printing.

**Scale:** Select a suitable scale for the viewport. The drawing in the modelspace is always drawn at scale 1:1. The viewport scale is how near or how far away the viewport **window** through which you view the drawing in the modelspace is.

**Enable Relative Detailing:** Relevant detailing allows you to display the detail on specific layers on the drawing in the viewport. Refer to the **DetSet** command for the usage of relevant detailing.

Once you have setup the viewport, click on **OK** and follow the Caddie prompts based on the shape option selected.

### Scaling a viewport

**ScaleVP** Set the scale of a viewport.

Caddie prompts you to indicate the viewport to scale. Click inside the viewport you wish to re-scale. The current viewport scale is displayed. Select the new scale from the list. You can tick the **Re-centre viewport** option to enable you to indicate a new centre for the viewport.

### Moving a viewport

**MoveVP** Reposition a viewport.

Click inside the viewport you wish to move and indicate the position to move from and the new location for the viewport.
Resizing a viewport

ReszVP  Resize a viewport.

Indicate the viewport you want to resize. Control points are displayed around the viewport perimeter. Click on the control point and move it to the desired position.

Enable or disable layers in multiple viewports

E-D Lay  Enable or disable layers in multiple viewports.

You can enable or disable layers in all the viewports on the current sheet or on all the sheets.

**TIP:**

If a new layer is added to a drawing it is automatically included in the displayed layers of all the viewports. Use this command to disable the new layer in all the viewports and included the layer only in the required viewports with the LaysVP command.

Displaying layers in a viewport

LaysVP  Change the layers display status in a viewport.

When you create a viewport, Caddie retains all the layer information as specified in the modelspace. Use this command to set the layers for display for the indicated viewports. You can assign different layers for display to all the viewports on one sheet. Different viewports on the same sheet can have different layers set for display. You can also assign a different plotstyle to each viewport through the **Object Properties** dialogue.

Pan within a viewport

PanVP  Pan the drawing across in a viewport.

Use this command to pan the drawing across the viewport. This is handy to use if all the information in the modelspace is not visible in the viewport.

Zoom in or out within a viewport

ZmBxVP  Zoom in to a viewport.

ZmOutVP  Zoom out of a viewport.

You can zoom into the drawing in the viewport by indicating diagonal points. Please use this command with care as the scale of the viewport will be affected and result in a printed drawing that is not to a set scale. Use the **ScaleVP** to check the scale of the viewport.

When you want to zoom out in the viewport indicate the centre of the viewport. This has the effect of zooming out by a factor of 0.5 each time each time you click in the viewport. Use the **ScaleVP** to check the scale of the viewport.

Delete a viewport

DelVP  Delete a viewport.

Select the viewport to delete. Caddie regenerates the sheet and removes the viewport.

Setting the render mode of a viewport
### Creating and Managing Sheets

**Chapter 3**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>RendVP</strong></td>
<td>Setting a viewport render mode.</td>
</tr>
<tr>
<td><strong>ColIntVP</strong></td>
<td>Change the colour intensity of a viewport.</td>
</tr>
<tr>
<td><strong>DetSet</strong></td>
<td>Set the relevant detailing status of a viewport.</td>
</tr>
<tr>
<td><strong>ZmStVP</strong></td>
<td>Store the current zoom state for creating a viewport.</td>
</tr>
<tr>
<td><strong>CrStVP</strong></td>
<td>Create a viewport from a stored zoom state.</td>
</tr>
<tr>
<td><strong>ObjVP</strong></td>
<td>Define an area in the modelspace to be used to create a viewport.</td>
</tr>
<tr>
<td><strong>CrObVP</strong></td>
<td>Create a viewport from a pre-defined area in the modelspace.</td>
</tr>
</tbody>
</table>

Use this command to change the render mode of an indicated viewport. Just keep in mind that Cadie currently only print viewports in 2D wireframe mode.

**Changing the colour intensity of a viewport**

You can change the colour intensity of a viewport when viewing 3D objects.

**Set the relevant detailing status of a viewport**

You can specify certain layers for display at different scales. For example, you may only want to see objects in a viewport on layer 1-5 that were created at viewport scales ranging from 1:10 to 1:50. If the scale of the viewport is higher than 1:50, these relevant detailing layers will not be displayed.

**Storing a zoomed state for viewport creation**

We have already discussed one method for creating viewports by using the **NewVp** command. Another method for creating viewports is to store the current zoom state of the modelspace that can then be rendered into a viewport. Recall this stored zoom state with the **CrStVP** command.

**Creating a viewport from a stored zoom state**

Once you have stored a zoom state with the **ZmStVP** command, you can create a viewport from this stored zoom state. Select the sheet on which you want to create a viewport and set the scale for the viewport on the dialogue. Indicate the centre point for the viewport on the sheet. When using the Auto Fit scaling, indicate the dialogal position for the viewport.

**Defining an area from which to create a viewport**

An easy method to create a viewport is to indicate a specific area of the modelspace to be used as a viewport. The viewport can be a rectangle, a polygon or a circle. Place the viewport on the sheet with the **CrObVP** command.

**Creating a viewport from a pre-defined area**

Once you have defined an area from which to create a viewport with the **ObjVP** command, use this command to place the viewport on the sheet. Set the scale for the viewport and indicate the centre for the viewport. If you select the Auto Fit option, indicate the diagonal constrains for the viewport. Remember this viewport will not be to scale and you can check the scale of the viewport with the **ScaleVP** command.

**Show viewport positions in modelspace**
ShowVP  Show the viewport positions in the modelspace.

Use this command to show the viewports of a specific sheet in the modelspace. Select the sheet you require. Virtual rectangles will be drawn in the modelspace where the viewports are captured from.

Zoom to a viewport position

Zm_VP  Zoom to a viewport position in the modelspace.

Simply indicate the viewport on the sheet you want to zoom to. Caddie will zoom the modelspace to the viewport position. This is a handy command when working with large drawings.

Disable and enable all layers in a viewport

DisaVP  Disable all the layers in a viewport.

EnaVP  Enable all the layers in a viewport.

You can disable or enable all the layers in a viewport. Refer to the LaysVP command to disable or enable a selection of layers.

Using the viewport wizard

The viewport wizard is located under the Utilities button on the menu bar. This wizard enables you to create viewports quickly and effortlessly. Simply follow the prompts to create a viewport.

Plotstyle assignment per viewport

A sheet and viewports will be printed using the plotstyle assignment from the sheet. If you need to specify a different plotstyle in a viewport, use the Object Properties dialogue and select another plotstyle from the Plotstyle list.

Rotate viewports

If you need to rotate the drawing in a viewport, use the Object Properties dialogue and change the angle of the Rotation Angle entry.
CHAPTER 4

Printing in Caddie

Configure Plot Styles
Setting up Pen Colours and Line Styles
Defining Viewports from Model space
Creating Viewports on Sheets for Printing
Managing Viewports
Previewing and/or Printing Drawings
Batch Printing
CHAPTER 4 - Printing in Caddie

Introduction to printing

Any Caddie drawing consists of a modelspace and any number of sheets. The modelspace is the area in which you create your drawing, and the sheets are where you would normally print drawings from.

Configuring plotstyles

One important option in the sheet setup is the plotstyle to be used when printing a sheet. You can assign separate plotstyles for each viewport on a sheet via the Object Properties dialogue.

To create a plotstyle select the File | Configure Plotstyles... command.

Load: Load a predefined plotstyle.

New: Create a new customised plotstyle.

Copy: Create a copy of the currently loaded plotstyle.

Save: Save any changes made to the current plotstyle.

Delete: Delete a plotstyle.

Caddie Pen: This is a listing of all the available pens in Caddie. To configure a pen, simply scroll through the list of pens and click on the one you want to configure.

Description: Enter a brief description of the usage of the selected pen if necessary.

Device Properties

Colour: Use Object Colour to print objects in the pen’s screen colour setting and use Custom to specify a custom colour to use when printing.

Width: Use Object Pen Width to print objects using the width setting in the drawing and use Custom to specify the width for printing.
Line Type: For future use.

Set all Colours to selected: This will assign the current pen’s colour to all the Caddie pens.

Set all Widths to selected: This will assign the current pen’s width to all the Caddie pens.

Summary: The summary contains a listing of all pens and the current screen colour, print colour and width settings. You can expand or collapse this listing.

Maximum Pens to Show: This controls the speed of the refresh rate of the listing. Set the maximum no of pens to show in the summary list to suit your requirements, just keep in mind that a shorter list will result in a quicker refresh rate.

---

WARNING

Make sure that all pens in the plot styles that you are using for printing have a positive width set. If a pen has a zero width it will not print or be visible in the print preview window. If a pen is set to object width and it doesn’t print, check that the object width is not set to zero.

---

Printing a Drawing

Printing from the modelspace [F7]

You can print your drawing directly from the modelspace. Just bear in mind that the drawing in the modelspace is at scale 1:1. For example, if you want to print a drawing directly from the modelspace at 1:100 scale, set the plot scale to 1:100. Zoom into the area you want to print and select Current Window under the Print Extents options.

If you simply want to fit the drawing on a sheet (not to scale), select the Scaled To Fit option under the Plot Scale.
Printing from the sheet [F7]

We normally print a drawing from the sheets. The sheets with viewports and print settings are stored within the drawing. Viewports are dynamically linked to the modelspace which means that any changes to the modelspace will automatically be updated in the viewport.

Refer to Chapter 3 – Creating and Managing Sheets for a full explanation regarding sheets and viewports.

Once you have setup a sheet and added viewports, you are ready to print the sheet. Select the Print command from the File menu.

**Printer:** Select the printer driver you want to use. You can change printer properties by selecting the **Properties** button. The printable area is displayed below the printer name.

**Printer Settings:** Select the required orientation, number of copies, source and paper size for your printer. The available paper sizes are determined by the printer driver. If the required paper size is not available in the list, you will have to set it in your printer driver.

**Sheet Settings:** The default plot units are set to millimetres. Set the plot scale at which to print. You would normally plot at scale 1:1 and simply scale your viewport to see exactly on the screen what portion of the drawing in the modelspace will be printed. If you change the plot scale, Caddie will scale the sheet up or down by the scaling factor. If the scale is not on the list, select the **Custom** option and enter the required scale. You can enter an offset for the printer if necessary and you can specify the plotstyle to be used for printing.

**Print Extents:** This will determine what portion of the drawing will be printed. The default print extents are set to **Sheet**, but you can also print the **Current Window**, **Drawing Extents** or **Screen Rectangle**. When printing from a screen rectangle, click on the Pick button and indicate the diagonal positions on the drawing.

**Save Settings:** Tick the box to use the printer setup for new sheets.

**Batch printing of drawings**
You can use this feature to print several drawings in batches. **File | Batch Print...** allows you to print a number of drawings to different printers and paper sizes as specified in the sheet setups of each drawing. Once you have added the drawings to the batch print, the modelspace and sheets of each drawing will be listed. Delete the modelspaces or sheets from the list that is not to be printed.

Once you have setup a batch print set, you can save it to a file. Simply load this saved print set to print a batch of drawings at a later stage.

We can also print all the drawings in the print set to a single printer, specifying the sheet size and orientation. Tick the **Scale and print all drawings to** box. This is handy to use for creating PDF files for record keeping. Batch printing will lessen the load in the educational environment as assignments can be printed easily for grading.

### Change Print Set:
- The **Current** button will add the currently loaded drawing.
- **Add** enables you to browse to the disk and select drawing to load.
- **Delete** allows you to delete modelspaces and/or sheets from the print set.

### Include:
You can include the modelspace and sheets to the print set.

### Change Order:
Move the drawing up or down in the print set.

### Saved Print Sets:
You can save and load print sets that are printed regularly.

### Printing troubleshooting

**Objects do not print:**

- **Plotstyle:** In the plotstyle the pen width can be assigned to object width, and will not print if set to zero in Caddie. Change the plotstyle to a custom width, or change the width of the objects within Caddie.

- **ByLayer:** The objects widths are set to ByLayer and the layer’s width is set to zero. Use the **Scope | LaySet** command to change the layer width.

- **Non-plotable layering:** The layer can be set to non-plotable. Check the layers for plotability with the **Scope | LaySet** command.

- **Defpoints layer:** The objects can be located on the **Defpoints** layer and this layer does not print. Move all the objects to another layer.
Viewport layering: The viewport layer display setting will influence what is displayed and printed in the viewport. Check the layers for printing with the Sheets | LaysVP command.

Polygon viewport border prints:

Polyline parameters: If you use a polygon viewport a polyline is inserted on the viewport border. Select Object Properties and select the polyline. You can now change the Plotable option from Yes to No.

Hatch doesn’t print:

Hatch parameters: If hatch doesn’t print there might be a problem with the hatch parameters. Use the HATCH | HtcGet and HtcEdit commands to fix any hatch definition errors. Check for hidden hatches in imported drawings by using the Utilities | System | Delete Hidden Hatches command.

Hatch fill doesn’t print:

Viewport render mode: If hatch fill in a viewport doesn’t print, check that the Render Mode on the Object Properties dialogue is set to 2D Wireframe. Other settings will not display or print the fills.

Hatch fills prints black:

Hatch parameters: The hatch fill parameter can be set to Colour / Pen and not to RGB Colour. If this pen is assigned to print black in the plotstyle, the fills will also print black. Change the hatch parameters to use RGB Colour.
CHAPTER 5

Object Properties

- Setting, Getting and Editing Object Parameters
- Setting Element Parameters
- Setting Construction Parameters
- Setting Text Parameters
- Setting Leader Parameters
- Setting Dimension Parameters
- Setting Hatch Parameters
CHAPTER 5 - Object Properties

Every object that you draw has properties associated with it. These properties are also known as parameters.

Right-click on the mouse and select Object Properties from the Context Sensitive menu to display the properties of the selected object(s). Properties that can be changed on the fly on Object Properties is displayed in black, and properties that cannot be changed here are greyed out and can only be set through the Parameter Set commands.

Properties (or parameters) enable you to setup all objects in the way you wish them display and print. You can save these parameters in different preset styles for quick parameter retrieval.

Set, Get and Edit of parameters

There are three commands associated with parameter changes. Parameter Set is used to set object parameters before or during the creation of the object. Parameter Get is used to fetch parameters from an existing object in a drawing that already has parameter settings you wish to use. Parameter Edit applies the current parameter settings to indicated objects in a drawing.
Caddie enables you to change the parameters of objects on the fly. This means that you can change the pen and lines style while you draw lines, rectangles, circles, or any other object, as the parameter settings commands have a lower priority.

**LINES – Parameter set, get and edit**

- **PnLnSet** Set the pen and lines properties. [F10]
- **PnLnGet** Acquire the properties of an indicated drawing element.
- **PnLnEdit** Apply the current properties to the indicated drawing element.

We constantly change the drawing element parameters during the creation of a drawing.

**Pen /Colour:** Set the desired pen colour to be used for drawing objects. These colours are for easy identification on the screen. When the plotstyle is set to print the object colour, this colour will be used. Set the plotstyle to use custom colour if you want to override the pen colour.

**Line Style:** The line style is used to draw the line. If the drawing is imported or packed, all the line styles will not be displayed for selection. Use the **Settings | Drawing Elements | Import Line Style** command and select the range of line styles you want to import.

**Width:** Select the width for the line to be used. The pen width selected here can be used to display the line widths on screen – use the **DISPL | ShWdt** command. This width will also be used when the plotstyle is set to object width.

**TIP:** You don’t need to specify the pens widths if your plotstyle is configured to custom widths.

**Saved Presets:** Frequently used colour and line style parameters can be saved as presets. These presets can be loaded in the current drawing and is also available for loading in other drawings.

**CnLIN – Parameter set, get and edit**

- **Con-Set** Set the colour for construction lines, circles and points.
- **Con-Get** Acquire the properties of construction objects.
- **Con-Edit** Apply the current properties to the indicated construction objects.
Construction lines and circles:

**Colour:** Set the desired pen colour to be used for the objects.

**Width:** Set the width to be used to display the construction lines and circles on the screen.

**Construction Points:**

**Point Size:** Enter the pixel size between 1 and 100 for the display of the points on the screen.

**Point Style:** Select the display style on the dialogue.

**Colour:** Set the pen colour to be used for the points.

**Saved Presets:** Frequently used colour and line style parameters can be saved as presets. These presets can be loaded in the current drawing and is also available for loading in other drawings.

**TEXT – Parameter set, get and edit**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TxtSet</td>
<td>Set the text properties.</td>
</tr>
<tr>
<td>TxtGet</td>
<td>Acquire the text properties of an indicated text string.</td>
</tr>
<tr>
<td>TxtEdit</td>
<td>Apply the current text properties to the indicated text string.</td>
</tr>
</tbody>
</table>
Font: Select the desired font for the text. You can change the format of the text by right-clicking on the font type drop-down combo and selecting the desired formatting options like italic, bold and underlined. Two types of fonts can be selected – TrueType Fonts and shape files.

Width Factor: Caddie automatically calculates the width of text based on the height and on the font type used. The default value here is 1. You can create special effects by changing this value. For example, setting this value to 2 would create text that is elongated along the X-axis. On the other hand, setting this value to 0.5 would result in the text being elongated along the Y-axis.

Pen / Colour: Select the colour to be used for the text.

Height: Select the height of the text.

Scale: This scale setting in is a multiplier for the size. If you want your text to be 5mm when printed at 1:50 you could either set the height at 5mm and the scale 1:50.

Vertical Origin: Set the origin of the text to its Top, Middle or Bottom.

Horizontal Alignment: Set the origin of the text to its Left, Centre or Right.

Angle: Set the desired angle of the text to the UCS (User Coordinate System) X-axis. A positive angle will rotate the text anti clockwise whilst negative angles will rotate the text clockwise.

Indicate paragraph width on text enter: If you check this option, Caddie will prompt you to indicate the paragraph width when entering text in a drawing.

Text on UCS: Place text on the current UCS plane (working in 2D).

Style: Text in a drawing uses an internal text style. Use the Manage button to rename, delete and create new styles. These styles are saved in the drawing. The auto-name button will automatically name any styles when copying text from another drawing.

Auto Layering: Check this option if you want to place text automatically on a specified layer. Once checked, select the desired layer for auto placing of text.

Saved Presets: Frequently used text parameters can be saved as presets. These presets can be
loaded and is available for loading in other drawings.

**Balloon:** Tick the **Switch Balloon On** if a balloon around the text is required. Select the colour for the balloon and tick the **Use Background Colour** if you want the balloon to be opaqued. Enter the **Scale Factor** for the balloon. The scaling is relevant to the text height.

**TEXT – Leader parameter set, get and edit**

- **LdrSet** Set the leader properties.
- **LdrGet** Acquire the leader properties of an indicated leader.
- **LdrEdit** Apply the current leader properties to the indicated leader.

Leaders are indicator lines that you draw in order to point to objects when annotating a drawing.

<table>
<thead>
<tr>
<th>Leader Parameter Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameters</td>
</tr>
<tr>
<td>Current Pen : 1 Linestyle : 1</td>
</tr>
<tr>
<td>Pen / Colour:</td>
</tr>
<tr>
<td>[Image]</td>
</tr>
<tr>
<td>Line Style:</td>
</tr>
<tr>
<td>1 Continuous</td>
</tr>
<tr>
<td>Auto Layering</td>
</tr>
<tr>
<td>[Image]</td>
</tr>
<tr>
<td>Styles</td>
</tr>
<tr>
<td>Name:</td>
</tr>
</tbody>
</table>

- **Pen / Colour:** Select the colour to be used for the leader.
- **Header Type:** Select the desired shape of the leader head to be used.
- **Line Style:** Select the line style to be used to draw the leader.
- **Size:** Select the size of the leader head at the desired scale.
- **Auto Layering:** Check this option if you want to place leaders automatically on a specified layer. Once checked, select the desired layer for auto placing of leaders.
- **Saved Presets:** Frequently used leader parameters can be saved as presets. These presets can be loaded and is available for loading in other drawings.

**Dimension overview:**

Before we talk about the settings available for dimensions, we need to understand how we set the parameters of a particular dimension.

You can either set an individual dimension’s parameters directly, known as **Individual Mode**, or you can assign a dimension to use a preset style, known as a **Global Mode**.

If a dimension is set to a global mode, its parameters will be the same as any other dimension that uses the same global mode style. If the global mode style is edited, all the dimensions using that
global style will also reflect the changes.

You may only want to change the parameters of an individual dimension without affecting any other dimensions. To do this you can either edit the parameters by telling Caddie this will now be an individual dimension and all the parameters will be set explicitly for that dimension or create a new style with the required parameters and assign that style to only the required dimension.

Getting the parameters from a particular measurement using the DimGet command will get that dimensions mode. So if the selected dimension was set to use global mode then once the DimGet command was executed all following dimension commands will use the selected global mode. Similarly if the dimension that we were getting the parameters from was set to individual mode all further dimensions would be set to individual.

**DIMS – Dimension parameter set, get and edit**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DimSet</td>
<td>Set the dimension properties.</td>
</tr>
<tr>
<td>DimGet</td>
<td>Acquire the dimension properties of an indicated dimension.</td>
</tr>
<tr>
<td>DimEdit</td>
<td>Apply the current dimension properties to the indicated dimension.</td>
</tr>
</tbody>
</table>

There are a large number of properties that can be set for dimensions. The dialogue box is divided into a number of **TABS** and a common **bottom section**.

**Dimension set – basic settings**

If you are starting out with Caddie you may find that the only settings that need to be changed can be found on the **Basic Settings** tab (tick **Use basic settings**).

For advanced dimension settings the other tabs can be used when the **Use basic settings** box is un-ticked.

The basic setting allows you to change only a small number of the large number of dimension parameters that can be set. These will satisfy most users.

Edit your template file (*.CTP) and change the dimension parameters if you want to start-up with different pre-loaded settings.

If any setting has been changed on an advanced tab page and that parameter is not shown on the basic page these changed parameters will still stay set when you go back to the **Basic Settings** tab page. However if you want to set all the settings back to defaults tick the **Use basic settings** tick box. All the fields on the **Basic Settings** tab are now directly accessible. It is important to also note that as soon as this tick box is ticked then all settings NOT shown on this page will be re-set to the Caddie defaults.
General:

**Use basic settings:** Tick the box to use only the basic settings.

**Overall Scale:** This is the scale we want to print the drawing at. This is used by all other dimensions settings.

**Dimension Lines and Arrows:**

**Colour:** Sets the pen used for all dimension lines to the pen selected from this drop-down.

**Extend Beyond Ticks:** This distance is the length the leader lines extend beyond the leader head. This does not affect any of the arrow heads, large circle, large dot, origins, triangles, and boxes which always stop at the leaders head.

**Baseline spacing:** This sets the distance for Datum dimensions leaders from the previous one.

**Arrow Head:** Choose the required arrow head from this drop-down.

**Arrow Size:** Set the target size of the arrow head by typing in the require value.
Text:

Font: Choose the require text font from the drop-down. The only fonts available are those installed in the Windows fonts and the Caddie shape files (.shx)

Size: The height of the text, at the overall scale.

Colour: The pen used for the text.

Placement: Places the text at the position selected.

- Above - Places the text so the base of the text is next to the leader line and reads left to right and bottom to top.
- Centred - places the text so the centre line of the text breaks into the leader line and reads left to right and bottom to top.
- Outside - places the text so it is on the side of the leader line which is furthest away from the dimension measurement points.
- JIS - places the text to JIS standard

Extension Lines:

Colour: Sets the pen used for Extension (Witness) lines.

Extend beyond dim lines: This is the distance the extension lines extend beyond the top of the dimension line.

Offset from origin: Defines the gap (minimum gap if fixed length is used). The actual gap drawn will be the specified amount multiplied by the scale value.

Fixed Length: If ticked, the extension (witness) lines will have a maximum length of the specified value (multiplied by the scale value). However if the definition end of the line is closer than the offset value the extension line will be foreshortened.

Length: Sets the value used by fixed length extension lines, if used.

Dimension set – common settings

At the bottom of the dimension parameter set dialogue box is a section which is not only common to all the tabs but also sets the way in which the dimension will be drawn.

Type: This is how the dimension is drawn.

- Unchained – All dimensions placed will require a starting point and an end point.
- Chained – The first dimension will require a starting point, an end point and a text position. All subsequent dimensions will only require an endpoint.
- Datum - The first dimension point of a datum set is indicated followed by the end point of the first dimension and the position of the first text. You are now asked for the distance of the spacing for the next dimension.

Mode:

- Individual - If selected, any changes made will not affect any global style. The changes will only by applied to the indicated dimension when using the DimEdit command or to any newly drawn dimensions. Whenever a dimension with a global style is edited with DimEdit command and the individual flag set, it no longer uses the global style it original had.
**Global** - When selected all of the parameters set will update the currently selected global style. If you want to use another style, first set the global style you want to use from the drop-down and select **Set Current**. This will load the settings of the selected style and it will be this global style that is used until changed. To change a dimension that has been set to use an individual style to a global style re-open the **DimSet** window and select the style you want from the global style drop-down, and select global style. After closing the dialogue use **DimEdit** and select the dimension you want to change.

**Global styles:**

- **Set Current** - Sets the global style shown in the drop-down to the current selected global style.
- **Manage** - This dialogue allows you to create new styles or rename and delete the styles you already have.
- **Manage: New** - Create a new style. This can be based on any existing style from the **Based-On** selecting drop-down. Specify the name for the new style.
- **Manage: Rename** - Renames the selected style.
- **Manage: Delete** - Deletes an existing style.
- **Manage: Cancel** - Closes the dialogue box without saving the changes.

**Auto Layering:**

Choose the layer that you want to use for the dimension. You will need to create the layer if it doesn’t exist using the **SCOPE | LaySet** command. With auto layering enabled, all dimensions will be forced to the selected layer, regardless of the active layer.

**Saved presets:**

These are not the same as global styles that have already been discussed. Whereas global styles are part of the drawing and the dimension if used, saved presets are system based and are just a collection of settings that can be loaded and used either individually or to modify an existing global style or to create a new one.

**TABS:**

It is important to note that any settings changed on any **TAB** page with effect either an individual dimension or a global style dependant on the **Mode** selected.

Refer to the online help file for information regarding the advanced dimension settings. [F1]

**HATCH – Hatch parameter set, get and edit**

- **HtcSet** - Set the hatch properties.
- **HtcGet** - Acquire the hatch properties of an indicated hatch.
- **HtcEdit** - Apply the current hatch properties to the indicated hatch.
Chapter 5

Object Properties

Hatch Type:

There are 5 types of hatching you can use. You can hatch with line style, patterns, fills, gradient fills and custom patterns.

**Line Style:** Hatch with lines using a specified colour / pen and line style. Set the hatch spacing and angle.

**Pattern:** Hatch by selecting from a list of predefined patterns. Set the scale and angle.

**Fill:** Hatch using solid colour fills. You can set the colour / pen for the fill or you can tick the **Use RGB Colour** to ensure the correct colour is passed to the printer.

**Gradient fill:** Hatch that is composed of two colours and in between the colours is a smooth blending. The grading can be displayed as linear, cylinder, inverted cylinder, spherical, inverted spherical, hemispherical, inverted hemispherical, curved and inverted curve.

**Custom:** These are custom hatch pattern files.

Parameters:

The different hatch types have different parameters for selection.

**Plane:**

**Back and front:** Any object that you hatch has both a front and a back plane. By hatching either plane, you can create special effects by moving hatches in front of or behind other objects.
Auto Layering:

**Enable for Fill and Gradient Hatch:** You can specify a layer for any fills or gradient hatches. This layer will be used for these hatches regardless of the active layer setting.

**Enable for Line and Pattern Hatch:** Line style, pattern and custom hatch patterns will be placed on this layer.

**Saved Presets:**

Frequently used hatch parameters can be saved as presets. These presets can be loaded in the current drawing and is also available for loading in other drawings.
CHAPTER 6

Working with Layers

Create Layers
Activate Layers
Hide and Display Layers
Name Layers
Set Layer Attributes
Manage Layer Definition Files
Change the SCOPE of a Drawing
CHAPTER 6 - Working with Layers

Since the SCOPE menu is so intimately linked to layers, the whole of the SCOPE menu will be covered in the discussion on layers.

Working with layers and the SCOPE menu

Definition of a layer

The best way to understand the concept of layers is to imagine yourself drawing on sheets of tracing paper. As you are drawing you might like to keep certain parts of a drawing separate from other parts in order to make changes easier. By drawing on separate sheets of tracing paper you are able to keep these parts of the drawing separate. The analogy is the same in Caddie. Visualise the layers in a drawing as the sheets of tracing paper.

By default Caddie only creates 1 layer at startup - layer 0. Caddie can be set to load a layer definition table to create extra layers. Layers can be added at will, so it is advised that you work with only as many layers as it optimally required so as to render your drawing more manageable.

KEY CONCEPT

It is recommended that you work with as many layers as it optimally required in a drawing. This will render your drawing more manageable. Placing different parts of a drawing on different layers allows you to manipulate these parts of a drawing much more easily if they are located on different layers.

Naming layers

When naming layers use practical names for layers. Change the layer naming convention to suit your requirements.

Examples:

- Brick work Plan view; or 001 Brickwork Plan
- Brick work Elevations; 001 Brickwork Elevations
- Cold water reticulation; 101 Reticulation – Cold water
- Hot water reticulation; 101 Reticulation – Hot water

You can rename layers at any time. Simply select the LaySet command from the SCOPE menu, select the applicable layer and enter a new name for it. Remember to save your layer definition to a file for easy retrieval.

Setting up layers in a drawing

LaySet

Set up layers in a drawing.
Layers with an asterisk contain information. It can be visible or invisible drawing objects and/or data points.

**Layers used:** This indicates the number of layers currently in use in a drawing.

**Current VSE on:** This displays the layers that the VSE (View, Snap, Edit) is switched on.

**Active Layer:** This value indicates which layer is currently active.

**Sort by:** *Creation* - Sort layers by creation date and time; *Index* - Sort layers by layer number; *Name* - Sort layers by name.

**Show:** *Used* - Show only layers that are in use. This means that any layer that has been created but where no drawing entities have yet been drawn will NOT be included in this filter; *Visible* - Show only layers that are currently visible in the scope; *All* - Show all layers created.

**Security:** *Password* - Enter (and Retype) a password to lock a layer. *Protect/Unprotect* - Click on this button to protect or unprotect a layer.

**Create:** Click on this button to create a new layer. Enter the layer name in the Name button.

**Delete:** The selected layer in the dialogue will be deleted. If there are visible or invisible data on this layer it will not be deleted.

**Insert:** This enables you to insert a layer between existing layers. All the layers after the inserted layer will be re-indexed.

**Re-index:** You can enter a new index for an existing layer.

**View:** Toggle this option to render a layer visible or not.

**Snap:** Toggle this option to be able to snap to objects on this layer or not.

**Edit:** Toggle this option to be able to edit objects on a layer or not.

**Force VSE to all in list:** Set the View, Snap and Edit option of all the layers in the drawing to the current layer’s VSE setting.

**Enabled:** Toggle this option to enable or disable a layer.

**Enable/Disable All:** Click on either of these buttons to enable or disable all the layers in a drawing.

**Plotable:** Toggle this option to make a layer plotable or not.
Working with Layers

Chapter 6

KEY CONCEPT

The Delete option is only available if the selected layer is not the currently active layer or no objects yet exist on this layer. You will notice that all layers that currently have objects drawn on them have a * to the left of the layer no. This also applies to the active layer even though it may not yet have objects drawn on it. This indicates to you that any layer in the list having a * next to the layer number cannot be deleted.

Loading and/or saving layer definitions

<table>
<thead>
<tr>
<th>LayDef</th>
<th>Load or save a layer definition.</th>
</tr>
</thead>
</table>

The Caddie default layer definition file is Caddie.ldf. This layer definition file is loaded by default and only initialises the first 10 layers. You can create more than 1,000 layers, but it will not be necessary in most environments.

Once you have customised your layers, you can save the layer list to a file with the Save option, and load it into existing or new drawings. You can also save a Template with the correct layer definitions.

Establishing which layers are in use

<table>
<thead>
<tr>
<th>LayUsd</th>
<th>Show the layers used.</th>
</tr>
</thead>
</table>

When you select this command, Caddie displays a list of all the layers currently in use in a drawing. Only layers that have objects or information on them will be displayed here. The only exception to this rule is if you activate a new layer and have not yet drawn on this layer, then this layer will also be included in the list of used layers.

Use this command when working with imported drawings to determine the layer usage.

Displaying selected layers only

<table>
<thead>
<tr>
<th>LayDisp</th>
<th>Set to scope to a combination of layers.</th>
</tr>
</thead>
</table>

We can select a single layer or a range of layers for display. If you have a large number of layers, you can setup layer groups for easy selection. This will save time when working on large projects.

There will be times when you want to display only certain layers in a drawing. You may want to do this when wishing to print or save only parts of a drawing. Another reason might be that you want to change pen and line settings globally for objects on certain layers only.

TIP: If you want to set the display to one layer only and you know an object is located on that layer, press and hold the [TAB] key down and click on the LayDisp command. Caddie will prompt you to indicate the object to set layer display from. Caddie will change the scope of the drawing to the layer upon which the selected object is located.

NOTE: When your active layer is not included in the displayed layer list, and you start to draw, the objects will not be visible.
Pick: You can set the layers by selecting the objects directly from the drawing. Click on the Pick button and click on all the objects on the layers to be isolated. Right-click Option-End to finish the layer selection and to return you to the dialogue. The identified layers will be highlighted in the dialogue.

Update layers dynamically: Check this option if you want Caddie to update the layer display on the screen as you select them from the list in the dialogue.

Selection: If you know the layer index number you can type it in. Caddie highlights this layer in the list.

Layer Listing: You can select multiple layers for display by using a combination of the CTRL and SHIFT keys whilst making selections from the layer list. To deselect a layer, simply reselect it.

Saved Layer Groups: To make it easier and quicker to display layers on large projects, you can group regularly displayed layers together. Select the layers you want to include in the group and click on the Create button. Caddie adds a generic group named Group_X where X represents a generic auto-generated number. Double-click on the group to change the name. If you want to modify the layer group, simply make a new selection of layer and click on the Update button to update the group. Delete layer groups with the Remove button.

Displaying layers example

In the example below, 9 layers are currently in use. Layer 4 is the Active Layer. Visible layers are displayed as solid lines. Hidden layers are displayed as dotted lines.
Referring to diagrams below, let's have a look at how the displaying of different layers affects the scope of a drawing.

**SCOPE | LayDisp: 1 8-9**

Layers 2 to 7 are hidden. If you draw now and press R (redraw) the object(s) that you drew will disappear from the screen because they are drawn on a layer that is not within the scope of the drawing.

**SCOPE | LayDisp: 2-5 7 9**

All other layers are hidden.

**SCOPE | LayDisp**
Display layers 1 to 9 inclusive but hide layers 4 and 8. This method of notation is quicker than the equivalent: 1-3 5-7 9.

Beware of falling into the trap of drawing on a non-visible layer when the active layer is not in the scope of the drawing.

---

**Activating layers**

<table>
<thead>
<tr>
<th>LayAct</th>
<th>Set the active layer. [A]</th>
</tr>
</thead>
</table>

Select a layer to activate. This changes the currently active layer to the newly selected layer. Anything you draw from this point on will be associated with this newly selected layer. Only one layer can be active at given time.

**Refreshing the drawing area**

<table>
<thead>
<tr>
<th>RedrAll</th>
<th>Refresh the screen in all the windows.</th>
</tr>
</thead>
</table>

This command refreshes the screen for all open windows. This is useful when you are working on more than one drawing at a time and you have executed several copy and paste commands to and from open drawings.

**Refreshing the active window only**

<table>
<thead>
<tr>
<th>RedrWin</th>
<th>Refresh the screen in the active window only.</th>
</tr>
</thead>
</table>

This command refreshes the screen in the active window.
Displaying all layers

All-Lay  Set the SCOPE to all layers.

Use this command to display all the layers in a drawing. Set all the layers for display after you have isolated the layers with the LayDisp command.

Display the set layers

Layers  Reset scope to the previous displayed layer list.

You can recall the last layer displayed with this command. Use the Layers command in conjunction with LayDisp and All-Lay. Setup the layers for display with LayDisp. Use All-Lay to display all the layers and simply select Layers to display the layers that are currently set with LayDisp.

Displaying a single pattern

Pattern  Set the scope of the drawing to an indicated pattern.

Indicate the pattern to place in scope. Whichever pattern you click on becomes the scope of the drawing and all other entities disappear from the screen. Select the All-Lay command to display all the objects in the drawing.

If the pattern you click on is a parent pattern then Caddie displays a dialogue requesting you to indicate whether to include the parent pattern and all its child patterns or only one of the child patterns. If you select the parent pattern, then it as well as all its children will be displayed. If you select only one of the child patterns, then only that pattern remains visible on the screen.

If the object that you select is linked to more than one pattern, Caddie will display a list of patterns to which the selected one is linked. Select the desired pattern from the list.

Displaying a number of patterns

Pattern  Set the scope of the drawing to a number of indicated patterns.

If you want display more than one pattern in the scope, press and hold the [TAB] key down while indicating the patterns in the drawing. Release the [TAB] key when indicating the last pattern. The scope will be set to the selected patterns.

Displaying marked objects

Marked  Set the scope of a drawing to marked objects.

Caddie redraws the screen leaving only marked objects visible on the screen. Select the All-Lay command to display all the objects in the drawing.

List the pens in the current scope

PenUsd  List pens in the current scope.

A list of the pens used in the current scope is displayed on the screen.

Use the Edit | MapPen command is you need to remap the pens in the drawing.

Move and Copy the selection set to a specified layer

M>Lay  Move the selection set to a specified layer.

C>Lay  Copy the selection set to a specified layer.

Set the selection required and indicate the layer to move or copy the objects to.
Please note that you will have duplicate objects if the C>Lay was used.

**Move an entire layer to another layer**

**Delete all the objects in the scope of a drawing**

| Del-Scp | Delete the current scope. |

This command deletes the entire contents of the scope of a drawing. Please note that Caddie does **NOT** display a confirmation prompt when using this command. If you find that the result is undesirable, press **[Ctrl+Z]** to undo the command.

**Delete a selected layer**

| DelLay | Delete a selected layer |

This command deletes the selected layer(s) from the drawing. Once the objects have been deleted from the layer(s), you can confirm or deny the deletion of the layers from the drawing. Press **[Ctrl+Z]** to undo this command.

Please note that Caddie cannot delete the current layer or layer 0. Use **File | Drawing Utilities | Pack Drawing** to delete unused objects that might reside on the layer(s).

**Show objects/Hide objects**

| Objects | Set the scope to a selection of objects. |
| H-Objs | Remove objects from the scope. |

Show or hide objects in the scope. Indicate your selection set – Selected, Rectangular, Polygon or Object. These commands are also located on the **Context Sensitive** menu.

**Layering and display on the context sensitive menu**

**TIP:** Use the **Layering & Display...** option on the **Context Sensitive** menu for easy access to often used commands.
CHAPTER 7

Pattern Commands

- Initialise New Patterns
- Group Patterns
- Name Patterns
- Manipulate Patterns
- Move Patterns across Layers
- Save Patterns as Symbols
- Copy, Move and Repeat Patterns
CHAPTER 7 - Working with Patterns

Introduction to patterns

A pattern is one or more objects grouped together to enable easy manipulation and selection. Patterns can be grouped within patterns, meaning nested patterns are allowed. Patterns can also reside on more than one layer if necessary. There is no limit to the number of objects in a pattern. The default left double click on the mouse button is set the select a pattern. This enables you to easily manipulate pattern – right click Move, Copy, Rotate, Scale etc.

Settings | Environment Settings | Patterns controls the way patterns are handled in the drawing. You can switch the initialisation of a pattern at start up on or off. You can set the current pattern to be highlighted or not. As a pattern can reside on more than one layer it is important to switch the initialisation of patterns on layer change and merging objects on if you wish to restrict patterns to reside only on one layer.

The difference between patterns, blocks and referenced symbols

A pattern can be seen as a group of drawing objects related to each other that can be treated as a single unit. Single objects within a pattern can easily be edited, manipulated or deleted.

A more advanced way to handle objects in a drawing is blocks and referenced symbols. These are covered in Chapter 21 – File and Advanced Objects. Use blocks and referenced symbols when working with your projects to enable streamlined editing and revision handling.

Blocks are groups of objects, bound together under a single name. The block information is stored in the drawing. Each occurrence of a block object references an internal group of objects. In this way all occurrences of a block reference are identical in all but the position, scale and orientation. Blocks need to be edited to be changed. When the block edit is completed, you need to end the edit. Only at that time will all the block references with the same name be updated in the drawing.

Referenced symbols are symbols or entire drawings that are referenced to a file location every time the drawing is opened or updated. This means that the referenced symbols are not stored in the drawing, but on disk. Only the referenced symbol name, insertion point, scale and rotation are stored in the drawing. Reload a referenced symbol in a drawing when it is updated on disk.

Count named patterns

Count named patterns in a drawing.

You can count the occurrence of patterns in your drawing to a file. Use a naming convention for easy identification. Please refer to block in Chapter 21 – File and Advanced Objects for the scheduling of blocks.

Get pattern information

Get specific information about a pattern(s).

The pattern information available for the current pattern or indicated pattern is: pattern name, number of layers and layer names.

Naming a pattern

Set or change the name of a pattern.
Chapter 7

Pattern Commands

By default all patterns in a drawing are named. Use this command to name the current pattern or the indicated pattern. Setup a naming convention to easily identify patterns.

**Initialise a pattern**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Initial</strong></td>
<td>Initialise a new pattern. [I]</td>
</tr>
</tbody>
</table>

Whenever you want to draw something that must form a pattern on its own, you need to initialise a new pattern. Select this command or simply select [I] on the keyboard. A new pattern is created and automatically named by the system. If needed, you can name the pattern for counting or identification. Use the EndPat command to end the current pattern. Or select press [I] to end the current pattern and initialise a new one.

**Activate an existing pattern**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ActivD</strong></td>
<td>Activate an existing pattern whilst deactivating all other patterns.</td>
</tr>
</tbody>
</table>

Whenever you want to modify or add to a pattern in such a way that the modified or added entities form part of the pattern, it **MUST** be the current pattern. All other pattern will be deactivated.

**Deactivate a pattern**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>EndPat</strong></td>
<td>Deactivate the current pattern.</td>
</tr>
</tbody>
</table>

The current pattern will be deactivated.

**Initialising a new pattern and activate a new layer**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SetL2O</strong></td>
<td>Initialise and activate a new pattern on a new layer based on an existing pattern.</td>
</tr>
</tbody>
</table>

This command will initialise a new pattern on the layer of the indicated pattern. It you do not need to initialise a new pattern, you can use the SetL2O command.

**Merging objects into current pattern**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MergPol</strong></td>
<td>Merge objects into current pattern by polygon trace.</td>
</tr>
<tr>
<td><strong>MergPat</strong></td>
<td>Merge pattern into current pattern.</td>
</tr>
<tr>
<td><strong>MergObj</strong></td>
<td>Merge object into current pattern.</td>
</tr>
</tbody>
</table>

The objects will be merged into the current pattern. Caddie will break the link between the newly merged pattern and the original patterns.

**Move patterns to another layer**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MovLay</strong></td>
<td>Move an indicated pattern(s) to a specified layer.</td>
</tr>
</tbody>
</table>

Select the new layer from the list and indicate the patterns to move to that layer.

**Move a pattern on a plane**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>PatPln</strong></td>
<td>Move an indicated pattern on a plane.</td>
</tr>
</tbody>
</table>

Select the plane to move the pattern to - Front, Back, Forward, Backward – and indicate the pattern to move.
### Pattern Commands

#### Showing pattern extents

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ExtSngl</td>
<td>Show the extents of a single pattern as a virtual rectangle.</td>
</tr>
<tr>
<td>ExtAll</td>
<td>Show the extents of all the patterns as virtual rectangles.</td>
</tr>
</tbody>
</table>

Caddie displays the pattern extents as virtual rectangles on the screen. Press `[R]` on the keyboard to regenerate the drawing and remove the virtual rectangles.

#### Deleting the last pattern created

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeLast</td>
<td>Delete the last pattern created.</td>
</tr>
</tbody>
</table>

Caddie deletes the last pattern you created. Note that this command executes once **ONLY**. Executing this command repeatedly does **NOT** delete precedent patterns.

#### Deleting a pattern

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DelPat</td>
<td>Delete an indicated pattern(s).</td>
</tr>
</tbody>
</table>

Use this command to deleted indicated patterns. Caddie prompts you to indicate the pattern to delete.

#### Creating an owner pattern

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OwnPat</td>
<td>Create an owner pattern and add other patterns to it.</td>
</tr>
</tbody>
</table>

This command enables you to initialise a new pattern, name it and add other patterns to this newly created pattern. These indicated patterns are now owned by the owner pattern.

If you double click to select a pattern that forms part of an owner pattern, Caddie will prompt you to indicate which pattern you wish to select, the owner pattern or the nested pattern.

Using owner and nested patterns is a complex but powerful way to manage patterns in certain environments. If only basic usage of patterns is required, use the patterns on a single level only.

#### Removing a pattern from an owner pattern

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RemPat</td>
<td>Remove a pattern from an owner pattern.</td>
</tr>
</tbody>
</table>

Use the `RemPat` command to remove the indicated pattern from an owner pattern.

#### Adding a pattern to an owner pattern

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AddPat</td>
<td>Add a pattern to an owner patterns</td>
</tr>
</tbody>
</table>

Indicate the owner pattern and the pattern you wish to add to the owner pattern.

#### Activate an existing nested pattern

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Activ</td>
<td>Activate an existing nested pattern.</td>
</tr>
</tbody>
</table>

This command will activate the indicated pattern, **BUT** it will not deactivate currently active patterns.

#### Removing all objects from patterns

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Remove</td>
<td>Remove all objects from all patterns.</td>
</tr>
</tbody>
</table>

This command removes **ALL** patterns from all drawing objects. Please note that all pattern associability will be lost when you select this command.
**Merging objects into the current pattern – keeping existing patterns**

- **MrgPolK**  Merge objects into current pattern by polygon trace.
- **MrgPatK**  Merge pattern into current pattern.
- **MrgObjK**  Merge object into current pattern.

The objects will be merged into the current pattern. Caddie will **KEEP** the link between the newly merged pattern and the original patterns.

**Initialise pattern leaving other active patterns open**

- **InitM**  Initialise a new pattern, leaving other active patterns open.

The pattern is initialised on the active layer. Use the **ActivD** or **Activ** commands to add objects to an existing pattern.

**Change the layer to an indicated object without initialising a new layer**

- **SetL2O**  Change the layer to an object, without initialising a new pattern.

This command only changes the active layer to the indicated object’s layer. A new pattern will **NOT** be initialised.

**Moving patterns forward to backward**

- **ToFront**  Move pattern forward.
- **ToBack**  Move pattern backward.

Indicate the pattern to move forward or backwards on the plane.

**Reorder opaques in a pattern**

- **Reorder**  Reorder opaques in a pattern.

This command will reorder opaques in a pattern and move them to the back.
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CHAPTER 8

Line Commands

Draw Lines
Draw Single Line Elements
Draw Rectangles
Draw Multi Lines
Draw Parallel Lines
Draw Polygons
Embed Text on Lines
Draw Polylines
Modify Polylines
CHAPTER 8 - Line Commands

Drawing lines and polylines

The most used line commands are **Draw**, **Auto**, **P-Line** and **Rectang** as drawings largely consists of straight lines. The current pen and line settings are used when you draw, but you can change these on the fly by using the **LINES | PnLnSet** command, by pressing the **[F10]** key or selecting the pen and line from the **Standard** menu bar. The command is not interrupted as the settings buttons have a lower priority.

Draw joined lines

**Draw** Draw a series of joined lines

Indicate the start and consecutive end points for the joined lines. Select **Option End** on the **Context Sensitive** menu to end the command. **TIP**: Press the **[Spacebar]** to end the command and reactive the same command.

If you are drawing lines on the X and Y-axes, you can then make use of the keyboard to draw lines.

![Figure A](image1.png) ![Figure B](image2.png)

**Example 1**: Drawing lines at random points in a drawing. In **Figure A** above, click at point 1 as the starting point of the line. Now change to the **Freehand** snap mode and click on all the other points labelled 2-8. Once you have clicked on point 8, select **Option End** to terminate the **Draw** command.

**Example 2**: Drawing lines of known dimensions along the X and Y-axes. Referring to **Figure B** above, click at point 1 to indicate the start of the line. From this point on, you will use the keyboard to draw the rest of the lines. You can use one of two ways to draw lines using the keyboard.

**Using the keyboard arrow keys**: Enter the following values and press the arrow keys on the keyboard to the direction of the line.

2400 [→]; 1350 [↓]; 875 [←]; 1500 [↓]; 3000 [→]; 220 [↑]; 500 [→];

**Using relative coordinates**: Ensure that the coordinate entry mode at the bottom of the screen is set to **REL** – relative mode.

2400,0; 0,-135; -875,0; 0,-1500; 3000,0; 0,2200; 500,0;

**[TAB] key**: Press the **[TAB]** key to constrain the lines to the specified drag angle. This angle is set with **Settings | Drag Setting | Drag Angle**.
Drawing non-continuous lines

**Auto**  Draw a series of un-joined lines.

This command will prompt you for a start point and an end point for each line. The pen is lifted between points.

![Image of lines drawn with Auto command]

Drawing polylines

**P-Line**  Draw a series of polylines

Draw a series of continuous lines as a set of polylines. Use the snap modes, keyboard arrows or coordinates to draw the polylines. You can press and hold the [X] key down to create an arc segment. Indicate the arc end point and a point on the arc. Select Close on the Context Sensitive menu to close the polyline. When the polyline is selected all lines and arcs are highlighted as this forms one object. Refer to the Modify menu for a detailed explanation on how to modify polyline.

![Image of Object Properties dialog]

If you have drawn an open polyline and want to close it at a later date, simply select the Object Properties dialogue and change the Closed option from No to Yes.

Drawing horizontal or vertical lines

**Hold-HV**  Draw horizontal or vertical lines.

The placement of the second point determines which line, horizontal or vertical, will be drawn. The longest value will always be drawn first.

Draw a line at an angle

**Vector**  Draw a line with a specified length and angle.
Draw a line with a specified length at a certain angle. The angle can be entered as decimals. Degrees, minutes, seconds can be entered as follows: To enter a value of 45 degrees 15 minutes and 20 seconds, type the value as follows: 45+15/60+20/3600

The setting DEG/RAD (degrees/radians) at the bottom of the screen will influence all angle calculations.

**Drawing parallel lines**

**Par**

Draw a line(s) parallel to an existing line

Use this command to draw lines parallel to other indicated lines. Enter the distance in the dialogue and indicate the line on the correct side for the offset. When you right-click on the dialogue entry it gives you access to the previously entered distances and the buffer option when you have measured.

**Drawing a parallel line through an indicated point**

**Par-Pnt**

Draw a line(s) parallel to an existing line through a specified point.

Indicate the line for the parallel line placement and indicate the position on the screen for the line.

**Multiple parallel line offsets**

**ParMult**

Draw a set of multiple parallel lines

This command enables you to draw up to 10 parallel lines each at a different offset to an existing line.

**Type of Offsetting:**

**Incremental:** All the offsets lines will be measured from the previously drawn line.

**Base Line:** All the offset lines will be measured from the base line, that is the line closest to which you click is used as the base line.

**Repeat 1st Pen/Line:** All the offsets will be drawn according to the current pen and line settings.

**Repeat Pens/Lines:** You can set the pen and line settings for all the offset lines drawn.

**Multi Indicate:** Indicate the line/polyline on the correct side and indicate the position for the first parallel line. This point determines the parallel distance. Indicate the approximate position for the last line.

**Multi Distance:** Indicate the line/polyline on the correct side. The first offset parameter distance will be used to place the parallel lines. Indicate the approximate position for the last line.
Drawing orthogonal lines at an angle

**OrthAng** Draw Orthogonal lines at an indicated angle

This command will draw orthogonal lines at an indicated angle. The first two points will determine the angle for the orthogonal lines, and the points to follow will draw the orthogonal lines.

In the following illustration, line AB represent the reference line. Once you click on point B, Caddie uses the angle calculated by line AB as the angle for all the orthogonal lines to be drawn next. Starting at point 1, you can click at the points indicate 1 to 7. Caddie automatically draws subsequent lines normal to the previous one.

Drawing a rectangle by origin

**RectOrg** Draw a rectangle(s) by specifying its origin.

Use this command to draw a rectangle(s) specifying its size, orientation and origin.
The values in the dialogue is used in this example to draw the rectangle at a specific size, orientation and origin position.

### Drawing rectangles

**Rectang**  Draw a rectangle(s).

Use this command to draw rectangles of any size. The rectangle can be drawn as 4 lines, or 1 polyline. When drawing as lines, the prompt is *Indicate rectangle first corner*, and when drawing as polyline, the prompt changes to *Indicate polyline rectangle first corner*.

Every time you select this command the Rectangle dialogue is displayed. If you select not to show the dialogue, you can still change the format of the command by using the [TAB] and [Ctrl] keys.

- **[TAB] key**: Holding the [TAB] key down while selecting the **Rectang** command will switch to the other setting, lines to polyline, or polyline to lines for that single selection of the command.

- **[Ctrl] key**: Holding the [Ctrl] key down while selecting the **Rectang** command will change the default setting.

### Drawing polygons

**Polygon**  Draw a multi-faceted polygon(s)
Changing the outside or inside diameters or side length on the dialogue will change the other settings accordingly.

If you enter a bottom angle of 30°, Caddie would draw it as the figure on the right. Current pen and line settings are used to draw the polygon.

ID represents the inside diameter and OD represents the outside diameter.

**Drawing double lines**

*DblLine*  
Draw a set of double parallel lines.

Enter the offset distance on the dialogue and select the origin for the double line – left, centre or right. Set the origin of the starting point of the line. Visualise drawing the line vertically downwards.

**Cutting an opening between two lines**

*MultiCt*  
Cut an opening between two lines
Refer to the illustration in the following multi cut examples. The first break point is at position 1, the second break point is at position 2 and the second line is position 3.

**Figure A:** The two lines are parallel to one another.

**Figure B:** The two lines are NOT parallel to one another.

### Drawing multiple parallel lines

**MultiL2**  
Draw a set of multiple parallel lines.

You can draw up to twenty multiple parallel lines at once. Each offset can have a different distance, pen number and line style. Set the origin to the left or to the right. All the offsets are baseline dimensions. In other words, all the distances are calculated from your indicated positions. Select **Close** on the **Context Sensitive** menu to close the multiple parallel lines.

### Remove a gap in a line

**RemGap**  
Remove the gap in a line.

You can remove the gap in a line by indicating the two sides. If the lines are NOT parallel, the gap can still be removed, but you will have to confirm this removal.

![Warning - non parallel lines selected](image)

### Moving the end of a line

**MveLnE**  
Move the end of a line to a new location

Use this command to move the end of a line to a new location. Indicate the line to edit and indicate the new position for the line end.

**TIP:** You can also move a line end point by clicking on the line control point when the line is selected and moving it to a new position.

### Moving an element

**MveElm**  
Move an element

Use this command to move an element. Indicate the line to move, indicate the start of the
translate and indicate the new position.

**TIP:** You can also move a line by clicking on the centre control point when the line is selected and moving it to a new position.

### Drawing lines tangent to two circles

**Tan-2Cr** Draw a line(s) tangent to two circles

To use this command effectively, decide exactly how you want the line to join the two circles (arcs). Indicate the side of the circles or arcs for the tangency. Refer to the diagram below.

![Diagram showing two circles with tangents](image)

### Sketching lines

**Sketch** Draw a line(s) in freehand mode

You can use this command draw in free hand sketch mode. You can sketch as lines or as a polyline. The increment length determines the maximum length of the sketched elements.

**Sketch a Polyline:** The lines drawn will be treated as one element so if you use the **DelObj** command from the **DELETE** menu, the entire line will be deleted.

![Free Hand Sketch Mode dialog box](image)

### Deleting the last object drawn

**DelLst** Delete the object drawn.

Use this command to delete the last drawn object. Press `[Ctrl+Z]` to undo this action. Each time you select this command, Caddie cycles through the history of elements drawn and deletes the previous object.

### Deleting an object

**Del-Elm** Delete an element(s)

Use this command to delete indicated objects. These can be lines, polylines, arc and circles.

### Drawing orthogonal lines

**Ortho** Draw orthogonal lines normal to the X and Y axis on the UCS.

This command draws a series of joined lines normal to each other. The first line drawn is
horizontal and then the vertical line is drawn. From this point on, the same principle applies with each line drawn normal to the previous line. Select **Option End** to terminate the command.

### Drawing orthogonal lines at an angle

**Orth-TA** Trace a set of orthogonal lines at an indicated angle

This command will draw orthogonal lines at an indicated angle. The two points of the first line determines the angle. All lines following will be orthogonal to this line.

### Ink in around construction elements

**Ink-In** Draw a series of lines and/or arcs around a set of construction lines.

When you indicate the first point, Caddie sets the intersection snap mode. The snap mode is then automatically switched to freehand. Indicate the next points on the current path close to the intersections. Lines and arcs are drawn to enclose the area you want inked in.

### Parameter set, get and edit of lines

**PnLnSet** Set the pen and lines properties. **[F10]**

**PnLnGet** Acquire the properties of an indicated drawing element.

**PnLnEdit** Apply the current properties to the indicated drawing element.

![Drawing Element Parameter Settings](image)

Set the pen colour, line style and width you want to draw with. If all the line styles are not available in the drawing, use the **Settings | Drawing Elements | Import Line Style...** command and select the range of line styles you want to import.

**TIP:** You don’t need to specify the width of any pens if your plotstyle is configured to custom widths.

**Saved Presets:** Frequently used colour and line style parameters can be saved as presets. These presets can be loaded in the current drawing and is also available for loading in other drawings.
Quick multiple parallel lines

**Q-Mult**  Draw a set of parallel lines using the command line.

This command is a command line interface to draw multiple parallel lines. Select the type of offset you require on the popup.

**Incremental:** All the offsets lines will be measured from the previously drawn line.

**Base Line:** All the offset lines will be measured from the base line, that is the line closest to which you click is used as the base line.

**Repeat 1st Pen/Line:** All the offsets will be drawn according to the current pen and line settings.

**Repeat Pens/Lines:** You can set the pen and line settings for all the offset lines drawn.

**Multi Indicate:** Indicate the line/polyline on the correct side and indicate the position for the first parallel line. This point determines the parallel distance. Indicate the approximate position for the last line.

**Multi Distance:** Indicate the line/polyline on the correct side. The first offset parameter distance will be used to place the parallel lines. Indicate the approximate position for the last line.

Enter the distances at the keyboard, separated by commas. Now you can enter the pen and line style for the parallel lines. For example: **3,1** where pen number 3 and line style 1 is used. Indicate the line on the correct side. The multiple parallel lines will be drawn.

Drawing rectangles using three points

**Rect3Pt**  Draw a rectangle(s) using three indicted points

Use this command to draw rectangles by indicating 3 points. The first 2 points determine the angle and the one side of the rectangle. The third point only determines the other side.

![Diagram of a rectangle drawn with three points](image)

Press and hold the **[TAB]** key down when selecting the Rect3Pt command to allow you to enter a specified area for the required rectangle. Indicate the first two points to determine the angle and the distance of the one side. A virtual rectangle or the area will be drawn. Indicate a side for this rectangle of a specific area.
**Draw a perpendicular line between 2 arc or circles**

**Cc-Line**  
Draw a perpendicular line between two arc or circles.

This command will draw a perpendicular or normal line between 2 indicated arcs or circles. The line is drawn from the circumference of the one circle to the circumference of the next.

**Embedding text in a line**

**TX-Line**  
Draw a line(s) with embedded text

Use this command to embed text in a line. Enter the text characters; minimum line length between the text characters and the pen number and line style. You have the option to draw full length lines and joined lines.
CHAPTER 9

Curve Commands

Circles
Concentric Arcs and Circles
Arcs
Ellipses
Splines
Spirals
CHAPTER 9 - CURVE Commands

Introduction to curve commands

The CURVE menu contains all the tools necessary to draw circles, arcs, ellipses, spirals and splines.

Drawing a circle specifying a radius and centre point

R----Cpt  Draw a circle specifying a radius and indicating its centre point.

Enter the radius of the circle and indicate the centre point on the drawing. You can continue to indicate centre points for more circles of the same radius.

Drawing a circle specifying a radius and origin

R----Org  Draw a circle specifying a radius and origin.

Use this command to draw a circle by specifying its radius and origin. Enter the radius on the dialogue and select the required origin position.

Drawing a circle specifying a radius tangent to 2 elements

R----2E  Draw a circle by specifying a radius and indicating two elements for tangency.

Use this command to draw a circle of specified radius touching two indicated elements. Enter the radius and indicate the 2 elements for the tangency.

Drawing a circle with a specific radius through 2 points

R-2P---  Draw a circle through two indicated points.
Enter the radius of the circle and indicate the first points through which to draw the circle. As you move the cursor position, Caddie will display virtual circles indicating the possible solutions. Indicate the second position and since there are 2 possible circles that can be drawn through these points, indicate the side for the required circle.

**Drawing a circle with a specific radius through a point and tangent to an element**

**R--P--E**  
Draw a circle with a specific radius through a point and tangent to an element.

Enter the radius on the dialogue and indicate the point through which to draw the circle. Indicate the element for the tangency.

If the distance between point 1 and 2 is too great, the system will return a message that no tangency could be generated.

![Diagram of circle with radius 15 and points 1 and 2]

---

**Drawing a circle through a point tangent to 2 elements**

**C--P-2E**  
Draw a circle through a point tangent to two elements.

Use this command to draw a circle through an indicated point and touching two indicated elements. Refer to the following illustration.

![Diagram of circle through point 1 and two elements]

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**Drawing a circle through 2 points tangent to an element**

**C-2P--E**  
Draw a circle through two points tangent to an element.

Use this command to draw a circle through an indicated point tangent to two elements. Refer to the following illustration.
Curve Commands

Chapter 9

Drawing a circle through 2 points

C-2P--- Draw a circle through two points.

Use this command to draw a circle through two indicated points. Indicate the centre of the circle on the drawing and indicate a point on the circumference of the circle.

Drawing a circle through 3 points

C-3P--- Draw a circle through three points.

Use this command to draw a circle through three indicated points.

Drawing a circle tangent to 3 elements

C----3E Draw a circle tangent to three elements.

Use this command to draw a circle touching three indicated elements. Indicate the three elements for the tangency. Refer to the following illustration.

Drawing parallel circles

Parlel Draw a circle or arc parallel to another circle or arc.
Use this command to draw circles or arcs parallel to other circles or arcs. Enter the distance in the dialogue and indicate the circle or arc on the correct side for the offset. When you right-click on the dialogue entry it gives you access to the previously entered distances and the buffer option when you have measured.

**Drawing parallel circles through a point**

**Par-Pnt** Draw a circle or arc parallel to another circle or arc, through a point.

Indicate the circle to draw a circle parallel to and indicate a position for the circle. Alternatively, you can type the offset at the keyboard and press [ENTER]. The value you type in is the offset distance between the two circles.

**Drawing multiple parallel circles**

**ParMult** Draw multiple parallel circles.

This command enables you to draw up to 10 parallel circles or arcs each at a different offset to an existing line.

**Type of Offsetting:**

**Incremental:** All the offsets will be measured from the previously drawn circle.

**Base Line:** All the offset circles will be measured from the base circle, that is the circle closest to which you click is used as the base circle.

**Repeat 1st Pen/Line:** All the offsets will be drawn according to the current pen and line settings.

**Repeat Pens/Lines:** You can set the pen and line settings for all the offsets drawn.

**Multi Indicate:** Indicate the circle or arc on the correct side and indicate the position for the first parallel circle. This point determines the parallel distance. Indicate the approximate position for the last circle.

**Multi Distance:** Indicate the circle on the correct side. The first offset parameter distance will be used to place the parallel circles. Indicate the approximate position for the last circle.
Drawing ellipses

**Ellipse** Draw an ellipse.

You can draw a true ellipse or a 4-arc ellipse. A true ellipse is created with one ellipse, and the 4arc ellipse consists of 4 arcs. Enter the ellipse data on the dialogue.

![Ellipse Diagram](image)

Tick the Indicate Major /Minor box if you want to indicate the major and minor axes on the drawing.

**Drawing an arc in a positive or negative direction**

**Arc-Pos** Draw an arc in a positive (ACW) angular direction.

**Arc-Neg** Draw an arc in a negative (CW) angular direction.

Use these commands to draw arcs by indicating the start point, centre point and end position.

![Arc Diagram](image)

Drawing an arc by angle
Chapter 9  
Curve Commands

**Arc-Ang**  
Draw an arc by specifying an angle.

Draw an arc by specifying an angle and direction. Caddie will display a virtual arc as you drag the mouse.

**Drawing an arc through 2 points**

**Arc-2Pt**  
Draw an arc through two points.

Use this command to draw an arc through two indicated points at a specified radius. Refer to the following illustration.

**Drawing an arc through 3 points**

**ArcL3Pt**  
Draw an arc through three points with the third point dynamic.

**ArcM3Pt**  
Draw an arc through three points with the second point dynamic.

Use these command to draw an arc through three indicated points.

**Drawing a series of arcs to fit on a path**

**Arc-Fit**  
Draw a series of arcs to fit along a path

Use this command to draw a series of arcs to fit on a path based on points indicated in a drawing.

**Drawing a spiral**

**Spiral**  
Draw a spiral.

Use this command to draw a spiral. Enter the number of spirals required, angle between spirals, distance between spirals and the direction for the spiral.
Deleting the last object drawn

**DelLst**  Delete the object drawn.

Use this command to delete the last drawn object. Press [Ctrl+Z] to undo this action. Each time you select this command, Caddie cycles through the history of elements drawn and deletes the previous object.

Deleting an object

**Del-Elm**  Delete an element(s)

Use this command to delete indicated objects. These can be lines, polylines, arc and circles.

Drawing quick multiple parallel circles or arcs

**Q-Mult**  Draw multiple parallel circles or arcs.

Use this command to draw multiple parallel circles. Enter the values for the parallel circles at the command line. Caddie does not display a dialogue, as is the case with the ParMult command. Follow the prompts for entering the required values.

Drawing a revision cloud

**R-Cloud**  Draw a revision cloud.

Use this command to draw a revision cloud around an indicated area. Use a revision cloud to highlight an area on the drawing.
Define the outline by specifying a minimum and maximum radius, pen colour and line style for the revision arcs. You can switch the visibility and plotabilité of the revision clouds on or off.

The background of the revision cloud can be filled with hatch and you also have the option to make it plotable. Revision notes can be added, including a leader arrow.

Switch **create on specified layer** on if you want the revision clouds to be created on a separate layer. Tick to create it as a new pattern.

If you use revision clouds on a regular basis, save presets to make the recall of parameters quick and easy.

**Drawing splines**

| Spline | Draw a spline(s) |

Splines can be closed or open objects. The shape of a spline is controlled by control points that can be forced or unforced. The current pen colour and line style will be used.

**Modifying splines**

| Modify | Modify a spline |

If you want to move a spline control point you simply select the spline and indicate the control point to move and the new position.

You can use the **Modify** command to modify aspects of a spline. Caddie displays a popup menu listing the possible functions to modify a spline. Un-force a forced spline, force an unforced spline, move a spline control point, edit the start angle of the first spline control point or change the end angle of the last control point.

**Deleting splines**

| Del-Spl | Delete a spline |

Use this command to delete splines. You can also select the spline and press the **[Delete]** key to delete it.
Exploding splines

**ExplSpl**  Explode a spline.

You can explode a spline into line segments or into a polyline. If you want to explode the spline as line segments, remember to select the **New Pattern** option to enable you to isolate the elements for easy selection.

Set the accuracy to use when exploding a spline. The smaller the value, the shorter the elements will be. This will result in a more accurate representation of the original spline but it comes at a cost in that more elements are used to build the exploded spline. An accuracy value of 50 should, in most cases, be sufficient to render an exploded spline. If you select the polyline option then the spline is exploded to form one single polyline entity.
CHAPTER 10

Construction Commands

- Draw Construction Lines
- Draw Construction Points
- Query Points
- Draw Construction Circles
- Draw Sheet Borders
CHAPTER 10 - Construction Elements

Introduction to construction elements

The CnLIN and CnCIR menus contain all the commands you will need to draw construction lines, points and construction circles. Construction elements are valuable drawing aids by enabling you to construct the rudiments of drawing objects around which you create actual objects. There are several advantages in using construction elements in a drawing:

- You can use as many construction elements as you wish.
- You suppress the display of construction elements using the Scope switches.
- You can use different colours for construction elements and you can change the colours of any existing construction elements.
- By default, construction lines do NOT print. This means that you don’t have to hide them before printing a drawing. Please note that construction points can be set to print.
- Construction lines can be drawn anywhere and at any angle.
- Construction elements can be used as reference points in a drawing (snapped to) anywhere along their length (Circumference for circles).

Construction line commands:

Tangential construction line between two circles

Tan-2Cr Draw a construction line tangential to two circles.

Indicate the correct side of the circles or arcs for the tangential construction line.

Tangential construction line at a specific angle

Tan-Ang Draw a construction line tangential to a circle, at a specified angle.

Enter the angle on the dialogue and indicate the correct side of the circle or arc for the tangential construction line.

X and Y axes construction lines

ConX&Y Draw a set of construction lines along the X and Y axes.

Indicate a position for the X and Y construction lines. For every point indicated in the drawing, Caddie draws a vertical and horizontal construction line – if your drawing is viewed from the top.

X, Y & Z axes construction lines

ConXYZ Draw construction lines along the X, Y and Z axes.

Indicate a position for the X, Y & Z construction lines to be drawn. If your drawing is viewed from the top, you will not see the Z axis construction line as it runs perpendicular to the X and Y axes.

X, Y or Z axes construction lines

Con-X Draw a construction line along the X axis.

Con-Y Draw a construction line along the Y axis.

Con-Z Draw a construction line along the Z axis.

Indicate a position for the X, Y or Z construction line.
If the [TAB] key is held down before selecting the command, Caddie allows you to indicate the end of the construction line and all subsequent construction lines until the command is stopped, restarted [Space Bar] or another command is selected.

**Parallel construction lines**

| Parlel       | Draw a parallel construction line(s). |

Use this command to draw up to parallel construction lines at a specified offset to an existing line or construction line.

**Parallel construction line through a point**

| Par-Pnt      | Draw a parallel construction line(s) though an indicated point. |

Use this command to draw a parallel construction line(s) through an indicated point on the screen.

**Construction line at zero offset**

| ProjLn       | Draw a construction line upon an existing line or construction line. |

Indicate the line upon which the construction line must be drawn. Caddie automatically obtains the angle of the selected line and the construction line is drawn on top of this line at the same angle.

**Parallel construction lines through two points**

| Par-2Pt      | Draw two parallel construction lines a set distance apart, through two points. |

Enter the distance between the parallel lines on the dialogue and indicate the point through which the first construction line must be drawn, and then the point through which the second construction line must be drawn. Two parallel construction lines, a set distance apart, will be drawn lines through these points.

**WARNING**

*If the two points you click on are closer together than the specified distance, Caddie displays a warning message to this effect. You will then need to select two new points. In other words, the distance between the two points you indicate MUST be greater than the distance specified.*

**Multiple parallel construction lines**

| ParMult      | Draw a set of multiple parallel construction lines at specified offsets. |

Caddie displays the ParMult dialogue.
Type of Offsetting:

**Incremental**: All the offsets lines will be measured from the previously drawn line.

**Base Line**: All the offset lines will be measured from the base line that is the line closest to which you indicate will be used as the base line.

**Repeat + use 1st Pen**: All the offsets will be drawn according to the current pen.

**Repeat + Pen**: You can set the pen setting for all the offset lines drawn.

**Multi Indicate**: The number of offsets is automatically reset to 0. Caddie will prompt you to Indicate the line on the correct side. Where you click determines the offset for all the other lines. Caddie then prompts you to Indicate the approximate position for the last line. Caddie draws as many offsets within this distance based on the first offset distance calculated.

**Multi Distance**: The number of offsets is automatically reset to 1. Caddie prompts you to Indicate the line on the correct side. Click at an appropriate point in the drawing. Caddie then prompts you to Indicate the approximate position of the last line. Caddie draws as many offsets within the distance based on the first offset you entered.

You can draw a maximum of 10 parallel lines at any one time. You do NOT have to draw all 10 every time. In the example above, only 6 parallel lines will be drawn. Caddie automatically disables any fields 7 to 10 and ignores any value you may have entered into fields 7 to 10.

**See also**: Q-Mult, the command line version of this command, further on in this chapter.

**Construction line through two points**

**Pnts-2** Draw a construction line(s) through two points.

Click on any two points in a drawing. Caddie draws the construction line through the two indicated points. After clicking on the first point, Caddie indicates the angle of the line as you move the mouse.
Chapter 10  Construction Commands

**Construction line at an angle**

**Pnt-Ang**  Draw a construction line(s) at a specified angle.

Use this command to draw a construction line at a specific angle. Enter the position on the drawing.

**Construction line normal to, and in the middle of a line**

**MdNmLn**  Draw a construction line(s) normal to another line, bisect the line.

Caddie draws the construction line perpendicular to the selected line and also bisects the line.

**Construction line at a reference angle**

**Ref-Ang**  Draw a construction line(s) at a specified angle relative to another line.

Use this command to draw a construction line at a specific angle from an existing line of construction line. Indicate on the screen the position for this construction line.

**Construction line normal to an indicated line**

**PtNmMd**  Draw a construction line(s) perpendicular to a line through a point.

You can draw a construction line normal (perpendicular) to an indicated line or construction line. You will have to indicate the positions for the line.

**Construction line normal to an indicated line**

**PtNmLn**  Draw a construction line(s) perpendicular to a line through a point.

You can draw a construction line normal (perpendicular) to an indicated line or construction line. You will have to indicate the positions on the line.

**Construction line normal to a circle**

**PtNmCr**  Draw a construction line(s) perpendicular to a circle or arc.

Indicate a circle or arc for the normal (perpendicular) construction line. A virtual construction line is dragged through the circle/arc centre point; indicate a position for the 2\textsuperscript{nd} point.

**Construction line bisecting two lines or construction lines**

**Bisect**  Draw a construction line(s) bisecting any two lines/construction lines.

Indicate the 2 lines or construction lines that are non-parallel. A construction line will be drawn bisecting the included angle. Caddie will also insert a normal (perpendicular) construction line to this bisecting construction line, at the intersection of the 2 lines.
### Dividing an angle into equal sections

**Mul-Ang**  
Draw a set of construction line(s) dividing the angle between two lines into equal angles.

Indicate the 2 non-parallel lines and enter the number of segments for the angle division. Construction lines will be inserted through the intersection point, dividing the angle into equal sections.

![Diagram of angle division](image)

### Delete construction objects

**DelCon**  
Delete a construction line or construction circle.

Use the **DelCon** command to delete construction lines and construction circles. Indicate in close proximity of the element to delete.

### Insert construction points

**Point**  
Insert construction points.

Indicate all the positions in the drawing for the construction points.

**PntIndx**  
Insert construction points with index numbers.

Enter the index number of the construction point on the dialogue, and indicate a position on the drawing. The index number of these construction points can be queries by using the **QPoint** command.

### Delete construction points

**DelPnt**  
Delete construction points.

You can delete construction points by indicating them on the drawing; you can also delete construction points by entering the point index number or you can delete all the construction points in the scope.

### Query construction point index number

**Qpoint**  
Query the index number of a construction point.

Click on the construction points to be queried. All points inserted using the **Point** command, have an index number of 0. All points inserted using the **PntIndx** command will have the index number you specified at the time of insertion.
Setting up construction lines, circles and points

**Con-Set**  
Setup the desired colours to be used for construction elements and points.

**Construction Lines & Circle** - Set the colour and width for the construction elements.

**Points** – Set the pixel size for the display of construction points; the point style and colour.

**Saved Presets** – You can save the most used settings for construction elements as a preset. Simply load this preset when needed.

Get and Edit of construction element settings

**Con-Get**  
Acquire current settings of an existing construction line.

**Con-Edt**  
Apply the current construction element settings to another.

Use **ConGet** to acquire the settings of an existing construction element. These settings will now be the current settings. Use the **ConEdt** to apply the current settings to existing construction elements in the drawing.

**Quick multiple parallel construction lines**

**Q-Mult**  
Draw multiple construction lines specifying type and offset values.

This command is a command line version of the **ParMult** command. Up to 30 values can be entered at the keyboard, separated by commas.

See also: ParMult command.

**Construction lines on the sheet border**

**BorCons**  
Draw a set of construction lines on the edges of the current sheet setting.
Caddie displays the **Border Construction Lines** dialogue. Select the required orientation, size and origin. If you want the absolute size of the sheet, use a scale of 1:1, otherwise enter the scale at which you will print your drawing. Caddie draws four construction lines, one each along the edges of the sheet as well as a horizontal (X) and vertical (Y) construction line intersecting at the centre (absolute zero) of the sheet.

**Construction circle commands:**

**Drawing a construction circle by radius**

**CnR-Cpt**  
Draw a construction circle by specifying a radius and centre point.

Enter the radius of the construction circle and indicate positions on the drawing for these circles.

**Drawing a construction circle by origin and radius**

**CnR-Org**  
Draw a construction circle by radius and origin.

Enter the radius and origin position for the construction circle. Indicate positions on the drawing for these circles.

**TANGENCY NOTE**

*If you get an error message displayed to the effect that ‘No tangency could be generated’, it means that Caddie cannot create the circle based on the three points you clicked on. You will need to re-select the points to draw the circle successfully.*

**Drawing a construction circle tangent to two elements**

**CnR-2E**  
Draw a construction circle tangent to two elements specifying a radius.

Enter the radius for the construction circle and indicate the 2 elements for tangency. Lines, circles, arcs and construction lines and circles can be indicated. Please note that it is important to indicate on the correct side of the elements to get the desired result.
Chapter 10  Construction Commands

**Drawing a construction circle through 2 points**

**CnR-2P**   Draw a construction circle through 2 points, specifying a radius.

Enter the radius and indicate the 2 points through which the construction circle must be drawn. As 2 circles can be drawn, you will have to indicate which one you want to be drawn.

**Drawing a construction circle through a point tangent to an element**

**CnR-P-E**   Draw a construction circle through a point, specifying a radius and tangent to an element.

Enter the radius on the dialogue. Indicate a point through which this construction circle must be drawn and the element to be used for tangency.

**Drawing a construction circle through a points tangent to 2 elements**

**CnC-P2E**   Draw a construction circle through a point, tangent to 2 elements.

Indicate a point on the drawing through which the construction circle must be drawn. Indicate the 2 elements for tangency.

**Construction circle through two points tangent to an element**

**CnC-2PE**   Draw a construction circle through 2 points, tangent to an element.

Indicate the 2 points through which to draw the construction circle and indicate the element for tangency.

**Construction circle indicating the centre point and point on the circumference**

**Cn-C-2P**   Draw a construction circle indicating the centre point and edge point.

Use this command to draw a construction circle through two indicated points. The first being the centre point and the second being a point on the circumference of the circle. You have the option to type in the radius of the circle in stead of indicating the second point.

**Construction circle through three points**

**Cn-C-3P**   Draw a construction circle through three indicated points.

Indicate 3 points on the drawing through which you want the circle to be drawn.

**Construction circle tangent to three elements**

**Cn-C-3E**   Draw a construction circle tangent to three indicated elements.

Indicate the 3 elements for tangency.

**Construction circle parallel to another circle**

**Cn-Par**    Draw a construction circle(s) parallel to another circle.

Enter a parallel distance on the dialogue and indicate the circle or arc on the correct side to insert the parallel construction circle.

**Construction circle parallel to another circle through a point**

**CnP-Pnt**   Draw a construction circle(s) parallel to another circle through an indicated point.

*Caddie* will ask you to indicate the circle from which you require a parallel construction circle to be
drawn, and to indicate a position on the screen.

**Multiple construction circles**

**Cn-PMlt**  Draw a set of multiple parallel construction circles at specified offsets.

Use the command to draw a set of multiple parallel construction circles at specified offsets.

**Type of Offsetting:**

- **Incremental** - All the offsets circles will be measured from the previously drawn circle.
- **Base Line** - All the offset circles will be measured from the base circle that is the circle closest to which you indicate will be used as the base circle.
- **Repeat + use 1st Pen** - All the offsets will be drawn according to the current pen.
- **Repeat + Pen** - You can set the pen setting for all the offset circles.
- **Multi Indicate** - The number of offsets is automatically reset to 0. Caddie will prompt you to **Indicate the circle on the correct side**. Where you click determines the offset for all the other circles. Caddie then prompts you to **Indicate the approximate position for the last circle**. Caddie draws as many offsets within this distance based on the first offset distance calculated.

- **Multi Distance** - The number of offsets is automatically reset to 1. Caddie prompts you to **Indicate the circle on the correct side**. Click at an appropriate point in the drawing. Caddie then prompts you to **Indicate the approximate position of the last circle**. Caddie draws as many offsets within the distance based on the first offset you entered.

You can draw a maximum of 10 parallel circles at any one time.

**See also:** Cn-QMult, the command line version of this command, further on in this chapter.

**Parallel radial construction circles**

**CnRad**  Draw a set of equally spaced parallel construction circle(s) radiating out from an inner circle.

Enter the offset distance between the concentric construction circles and indicate the inner circle position and the approximate outer circle position.

**Delete construction objects**

**DelCon**  Delete a construction line or construction circle.

Use the **DelCon** command to delete construction lines and construction circles. Indicate in close proximity of the element to delete.
**Set, get and edit up construction circles**

- **Con-Set**: Setup the desired colours to be used when drawing construction circles and points.
- **ConGet**: Acquire current settings of an existing construction line.
- **ConEdt**: Apply settings of a selected construction element to another construction line.

**Quick multiple parallel construction circles**

- **Cn-QMlt**: Draw multiple construction circles specifying type and offset values.

This command is a command line version of the **Cn-PMlt** command. Up to 30 values can be entered at the keyboard, separated by commas.

**See also**: Cn-PMlt command.
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CHAPTER 11

Display Commands

  - Zoom in and out of Drawings
  - Store and Recall Predefined Zooms
  - Pan across Drawings
  - Open Multiple Windows of one or more Drawings
CHAPTER 11 - Display Commands

Introduction to the display commands

Two of the most often performed operations in a drawing are zooming and panning. You can also display the actual line thicknesses in a drawing on the screen.

Zooming to a drawing sector

ZmSectr  
Zoom into a selected sector of a drawing.

Visualise the drawing as split up into 9 rectangles of 3 rows and 3 columns.

Each cell represents a sector to which you can zoom to. Select the desired sector to zoom to and click on OK.

Zoom recall

ZmRecal  
Recall a previously stored zoom.

Once you have stored a zoom area using the ZmStore command, you can zoom to this area quickly by executing the ZmRecal command.

Zoom store

ZmStore  
Store the current zoom.

If you find yourself having to zoom to a specific part of a drawing frequently, you can store this zoom and recall it quickly without the aid of scrolling and panning. You can only store one zoom area at a time as the current zoom is stored every time the ZmStore command is selected.

Refer to the DISPL | StoView command to store multiple zooms.

Zoom to the extents of a drawing

ZmExts  
Zoom to the extents of all entities in a drawing. [F4]

Caddie zooms in to or out of the drawing in such a way that the extreme left, top, right and bottom of all drawing entities are displayed to fill the drawing area. This allows you view every entity in a drawing and its location.

Zoom to a specified zoom factor

ZmScale  
Zoom to an area of a drawing specifying a zoom factor.

Enter a scaling factor for the zoom. Please note that Caddie does NOT rescale the drawing. This value is used to zoom in to or out of a drawing by the value you specify. Select the origin to be used for the zoom window and click on OK. Caddie uses the point where you click in the drawing as the origin of the new window and sets the zoom factor to the value you specify.
Chapter 11 Display Commands

**Zoom to full screen**

**ZmAll**  
Set the zoom factor to display the contents as set in the **Settings | Modelsapce Worksapce** border setup. [F5]

Select this command to view the contents of the entire drawing within the borders of the currently selected modelspace setup.

**Recall the last zoom state**

**ZmLast**  
Recall the last zoom state. [F6]

This command will display to the previous zoom.

**Set the zoom state to a box**

**ZmBox**  
Zoom in to a drawing by indicating a rectangle. [Z]

Indicate the zoom box on the drawing with the mouse.

**Store multiple zoom states**

**StoView**  
Store multiple zoom states by name.

Zoom into an area you want to store. Select this command and enter the name of the view. Continue this process until you have saved all the desired zoom states.

**Recall zoom states**

**RecView**  
Recall predefined zoom states

Recall saved zoom states by selecting the name from the list.

**Show zoom states**

**ShView**  
Show the predefined zoom states on the drawing.

The stored zoom states are indicated on screen by virtual rectangles.

**Delete zoom views**

**DelView**  
Delete a predefined zoom state(s).

You can delete a view or views you stored with the **StoView** command. Select the views and click on the **Delete Selected** button. Click on the **Delete All** button to delete all the views in the list.

**Refresh all windows**

**RedrAll**  
Refresh the screen in all open windows.

Caddie refreshes the screen in all the windows that are currently open, removing virtual lines.

**Refresh the contents of the current screen**

**RedrWin**  
Refresh the screen only in the currently window.

Caddie refreshes the screen in the current window only, removing virtual lines.

**Zoom in or out**

**Zm-In**  
Zoom in to a drawing by a factor of 2. [PageUp]

**Zm-Out**  
Zoom out of a drawing by a factor of 2. [PageDown]
Display Commands

Chapter 11

Zoom in or out of the drawing by a factor of 2.

Pan across a drawing

**Pan** Pan across a drawing. [P]

You will frequently need to pan across a drawing in order to bring into view other parts of a drawing not currently visible on the screen. Select the Pan command or [P] keyboard key and indicate the position from which to pan, and the position to pan to on the screen.

Dynamic Panning

You can pan dynamically by holding the \([\text{Alt} + \text{Left Mouse Button}]\) keyboard buttons down. Move the drawing to the desired position and release the buttons. You can also hold the [Scroll Button] of the mouse down to dynamically pan the drawing. Please note that if you have programmed the scroll wheel of your mouse to perform another action, this feature may not work.

Quick pan across a drawing

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pan-L</td>
<td>Pan LEFT across a drawing. [Ctrl+Left Arrow]</td>
</tr>
<tr>
<td>Pan-R</td>
<td>Pan RIGHT across a drawing. [Ctrl+Right Arrow]</td>
</tr>
<tr>
<td>Pan-U</td>
<td>Pan UP across a drawing. [Ctrl+Up Arrow]</td>
</tr>
<tr>
<td>Pan-D</td>
<td>Pan DOWN across a drawing. [Ctrl+Down Arrow]</td>
</tr>
</tbody>
</table>

Use these commands to pan quickly across a drawing by approximately a third of the width or height of the screen.

Show pen widths

**ShWdth** Show the drawing pens using actual thicknesses.

Caddie displays all the objects using the line widths as specified in the drawing. Please remember that this does not show the thickness as specified in the plot style – use the UpdWdt to update the widths from the plot style.

Hide pen widths

**HdWdth** Hide all drawing pen widths.

Caddie redraws all objects with all lines being the same thickness.

Update widths from plot styles

**UpdWdt** Update screen pen widths from the current plot styles.

This command is only effective if you have executed the ShWdth command described above to display pen thicknesses on the screen. If you have made changes to the plot styles, this command refreshes all drawing pens to reflect these changes.

Regenerate current window

**Regen** Regenerate the contents of the current window. [R]

Caddie regenerates the contents of the current window. Regen recalculates all the geometry and reloads the display list on the graphics card.
CHAPTER 12

Marking and Selecting Commands

Mark and UnMark Objects for Isolation and Manipulation

Select and UnSelect Objects for Isolation and Manipulation
CHAPTER 12 - Marking and Selecting Commands

Marking versus selection

Objects must be selected or marked for isolation so that you can manipulate and/or change the parts of the drawing.

Selection: Can be achieved in two ways:

1) Left clicking an object - when no command is active. The object colour will change to magenta (default colour for selection/highlighting). Right-click on the mouse to display the context sensitive menu – click the desired isolation and/or manipulation command.

2) Using the MARK/SELECT commands. When set to SELECT (set by the toggle switch T M>S or the TAB select – see below) the SELECT menu contains all the select commands. Selected objects are displayed in magenta (default colour for selected objects).

Marking: Achieved by using the MARK/SELECT commands.

1) When set to MARK (set by the toggle switch T M>S or the TAB select – see below) the MARK menu contains all the mark commands. Marked objects are displayed in cyan (default colour for marked objects).

Selection is a temporary isolation method as a refresh of the screen or a regen of the drawing will unselect the object(s). Marking is a more permanent isolation method as the marked objects stay marked until they are unmarked with the relevant command(s).

Toggle the functionality of the commands (MARK or SELECT)

<table>
<thead>
<tr>
<th>T M&gt;S</th>
<th>Toggles the functionality of the Commands between MARK and SELECT. The text on the Root tool will change to indicate the current selection.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>TIP:</strong> Press and hold [TAB] down when selecting the MARK button to change all the marking and unmarking functionality to selection and unselection. Press and hold [TAB] down and click on the MARK button again to change it back to marking and unmarking.</td>
</tr>
</tbody>
</table>

The use of ‘Highlight’ and ‘Un-highlight’ is used in this chapter, to refer to either MARK or SELECT depending on the current setting.

Marking or Selecting the scope of a drawing

<table>
<thead>
<tr>
<th>MSScp</th>
<th>Highlights the entire scope of a drawing.</th>
</tr>
</thead>
<tbody>
<tr>
<td>UnScp</td>
<td>Un-highlights the entire scope of a drawing.</td>
</tr>
</tbody>
</table>

Use this command to highlight every object that is within the current scope of the drawing. This implies that if you are suppressing part of a drawing either by activating one of the suppress commands or by hiding layers in the drawing, those parts of the drawing are considered to be outside of the scope of the drawing and will be excluded from marking.
## Marking or Selecting objects on a layer(s)

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSLay</strong></td>
<td>Highlights all the objects on a specified layer(s).</td>
</tr>
<tr>
<td><strong>UnLay</strong></td>
<td>Un-highlights all the objects on a specified layer(s).</td>
</tr>
</tbody>
</table>

Caddie displays a list of the existing layers in the drawing. Choose the layer to be highlighted and click on **OK**. Caddie marks all the objects on the indicated layer. You can highlight more than one layer at a time. Click on the first layer you wish to highlight. Now press and hold down the **[Ctrl]** key and click on all the other layers you wish to highlight. If you select a wrong layer by mistake, simply click on it again and it will be de-selected.

## Mark or Select entities inside or outside a rectangle

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSRect</strong></td>
<td>Highlight the objects inside or outside a rectangle.</td>
</tr>
<tr>
<td><strong>UnRect</strong></td>
<td>Un-highlight the objects inside or outside a rectangle.</td>
</tr>
</tbody>
</table>

Caddie displays a popup menu from which you choose the desired highlighting method. Patterns or elements can be deleted inside or outside the indicated rectangle.

## Mark or Select entities inside or outside a polygon

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSPol</strong></td>
<td>Highlight the objects inside or outside a polygon.</td>
</tr>
<tr>
<td><strong>UnPol</strong></td>
<td>Un-highlight the objects inside or outside a polygon.</td>
</tr>
</tbody>
</table>

Caddie displays a popup menu from which you select the desired highlighting method. Patterns or elements can be deleted inside or outside the indicated polygon.

## Mark or Select an indicated pattern(s)

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSPat</strong></td>
<td>Highlight an indicated pattern(s).</td>
</tr>
<tr>
<td><strong>UnPat</strong></td>
<td>Un-highlight an indicated pattern(s).</td>
</tr>
</tbody>
</table>

Click on all the patterns you wish to highlight. If the element you click is in more than one pattern, Caddie will display a dialogue listing the patterns. This list will usually contain the name of the parent pattern and the name of the pattern itself. Select the required pattern from the list.

## Mark or Select an indicated object(s)

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSObj</strong></td>
<td>Highlight an indicated object(s).</td>
</tr>
<tr>
<td><strong>UnObj</strong></td>
<td>Un-highlight an indicated object(s).</td>
</tr>
</tbody>
</table>

Select all the entities you wish to highlight. Select **Option End** to terminate the command.

Please note that you can un-highlight an object while you are busy highlighting by simply holding down the **[Tab]** key and clicking on the objects you wish to un-highlight. Release the **[Tab]** key to continue the highlighting process.

## Mark or Select a series of continuous lines

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MSTrc</strong></td>
<td>Highlight a series of continuous lines.</td>
</tr>
<tr>
<td><strong>UnTrc</strong></td>
<td>Un-highlight a series of continuous lines.</td>
</tr>
</tbody>
</table>

Indicate the object to start the outline trace and indicate the direction for trace. All the continuous lines will be highlighted.
Marking and Selecting Commands

Chapter 12

Mark or Select objects drawn with specific pen and/or line style

**MSP/L** Highlight the objects drawn in a specific pen type and/or line style.

**UnP/L** Un-highlight the objects drawn in a specific pen type and/or line style.

You have the option to highlight the objects with a specific pen type and line style; only a specific pen type; or only a specific line style.

Mark or Select construction points

**MSPts** Highlight a construction point(s).

**UnPts** Un-highlight a construction point(s).

You can highlight a construction point by indicating the point; entering the index number of the point; or you can delete all the construction points in the drawing.

Please note that only construction points inserted with the **CnLIN | PntIndx** command can be deleted by index.

Mark or Select a pattern by name

**MSPaN** Highlight pattern by its name.

**UnPaN** Un-highlight pattern by its name.

Caddie displays a dialogue listing all the patterns by name in the drawing. Select the pattern you wish to highlight.

Mark or Select the current pattern

**MSCPat** Highlight the current pattern.

**UnCPat** Un-highlight the current pattern.

Caddie highlights the currently active pattern. If no pattern is currently activated or you have just initialised a new pattern but have not yet drawn anything, this command has no effect.

Mark or Select by crossing

**MS-X** Highlight elements by line crossing.

**UnMS-X** Un-highlight elements by line crossing.

To use this command, simply draw virtual lines across any element you wish to highlight. Indicate the starting point of the crossing line. The next and subsequent points should all cross over the elements that you wish to highlight. Select **Option End** to terminate the command.

Swap Marked to Selected; Selected to Marked

**M<>Sel** Swap marked to selected; swap selected to marked.

Select this command to swap marked to selected, or selected to marked. This is a handy command to use when you are busy selecting and know that you want to change the drawing view and you will lose your selection set - simply swap selected to marked and once you have setup the different view – swap marked to selected, and continue your selection.
Mark or Select duplicate elements

**MSDupl** Highlight all duplicate elements in a drawing.

All duplicate objects will be highlighted. These can be deleted or moved to another layer.

Swapping marked and un-marked; selected and un-selected objects

**Swap** Swap marked and unmarked (or selected and un-selected) objects.

Caddie swaps the highlight status of all elements in the drawing. In other words, all marked elements are unmarked and vice versa or all selected elements are un-selected and vice versa.

Merging Marked or Selected objects

**Merge** Merge all highlighted elements into the current pattern.

Caddie will merge all highlighted objects into the current pattern.

Completion of marking/unmarking; selection/unselected

>**DONE**< Indicate that all marking and/or unmarking is completed.

When the **Auto** clear switch is on, objects and patterns are automatically unmarked when the translation commands are selected. This forces you to mark the objects and patterns during the translation function, and you need to select the >**DONE**< command to inform Caddie that marking/unmarking is completed so that you can return to the translation function.

When the **Auto** clear switch is off, **Man** will be displayed in the status controls. See below.

Deleting marked entities

**DelMrk** Delete all marked objects.

Caddie does NOT prompt you to confirm the deletion - select the **Edit | Undo** command [Ctrl+Z] to undo this action.

Mark or select a line and/or circle

**MSLnCr** Highlight indicated lines and/or circles.

This command is similar to the **MrkObj** command except that this command only highlights 2D objects and not 3D objects.

Mark or select text or Leaders

**MSTxLr** Highlight indicated text and/or leader arrows.

Indicate the text and/or leader arrows to be highlighted.

Mark or select hatching

**MSHtc** Highlight indicated hatching.

Indicate the hatching to be highlighted.
**Mark or select a dimension**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MSDim</td>
<td>Highlight an indicated dimension(s).</td>
</tr>
</tbody>
</table>

You have the option to highlight individual dimensions, or all the dimensions in the scope.

**Mark or select a construction element**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MSCon</td>
<td>Highlight a construction line or circle.</td>
</tr>
</tbody>
</table>

You have the option to highlight individual construction elements, or all the construction elements in the scope.

**Saving and recalling marked or selected status of objects**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MSStr</td>
<td>Store the highlight status of a drawing.</td>
</tr>
<tr>
<td>MSRcl</td>
<td>Recall the highlight status of a drawing.</td>
</tr>
</tbody>
</table>

There may be times when you wish to store and later recall the highlighted status of a drawing. Highlight all the required objects and choose the **MSStr** command. This stores the highlighted status of a drawing. If at a later stage you can restore this same highlighted status by choosing the **MSRcl** command. Both Marked and Selected are stored separately. One storage of each type of highlighted objects is allowed per drawing. A storage will overwrite the previous save settings.
CHAPTER 13

Transformation Commands

Manipulate Drawing Objects
Copy, Repeat and Move Objects
Rotate, Mirror and Scale Objects
Create a Matrix from Objects
Create Detailed Copies of Objects
Stretch Objects
CHAPTER 13 - Transformation Commands

Introduction to transforming objects

The TRANS menu contains the commands necessary to transform drawing objects. These commands enable you to manipulate objects in a drawing and include:

- Copying, repeating and moving objects.
- Rotating, mirroring and scaling objects.
- Creating a matrix to repeat existing drawing objects.
- Copying, rotating, scaling and fitting objects about a reference line.
- Creating detailed copies of objects in a drawing.
- Stretching parts of a drawing.
- Rescaling an entire drawing based on a given measurement.

You will notice that the three TRANS menu commands, Rotate, Scale and Mirror all have corresponding &rotate, &scale and &mirror commands. The Rotate, Scale and Mirror are high priority command used to perform specific transformations on indicated objects. The &rotate, &scale and &mirror commands have a lower priority and can be used in conjunction with another command. For example, Transl - copy a selection of objects a few times in the drawing, then select the &rotate command to rotate the dragged objects at a specific angle, continue to insert these objects in the drawing at an angle.

Transformation selection

The transformation commands will only affect objects that are set the Selection option on the dialogues.

- **Scope**: The transformation will affect all the objects in the scope of the drawing regardless of what is currently marked or selected.
- **Selected**: Transformation will only affect selected objects.
- **Marked**: Transformation will only affect marked objects.
- **Rectangle**: Transformation will only affect objects completely contained within the indicated rectangle.
- **Polygon**: Transformation will only affect objects completely contained within the indicated polygon.
- **Pattern**: Transformation will only affect the single indicated pattern.
- **Object**: Transformation will only affect the single indicated object.

<table>
<thead>
<tr>
<th>Rotating objects</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rotate</strong></td>
<td>Rotate objects.</td>
</tr>
<tr>
<td><strong>&amp;rotate</strong></td>
<td>And rotate objects.</td>
</tr>
</tbody>
</table>
Chapter 13  
Transformation Commands

Rotation Angle: Set the desired angle of rotation.

Dynamic Rotate: When you tick this option you can rotate the objects around a point on the screen.

Pick angle from elements: Tick this option to acquire the angle between 2 elements in the drawing for your rotation.

Rotation Direction: Set the rotation direction to either CW or ACW.

Rotate original: This will delete the original objects and relocate them in the newly rotated position.

Copy: The original objects being rotated will be retained and a copy of them will be generated in the newly rotated position.

Repeat: The original objects will be repeated a number of times through the angle of rotation in the specified direction of rotation. Enter the number of repeats.

Scaling objects

Scale  Scale objects.

&scale  And scale objects.

Proportional Scaling: Check this option if you want scaling to occur proportionally in all three axes (XYZ). Checking this option disables the Y and Z scale factors.

Dynamic Scaling: If you leave this option un-checked, you can enter a scaling factor to scale objects. To enlarge objects enter a factor value greater then 1. To scale objects down, enter a value between 0 and 1.

And adds the ‘Dynamic’ option (See below). If you want the scaling factor to be different on the X, Y and Z axes, un-check this option and enter the scaling factor to be used for the X, Y and Z axes.
If the Proportional Scaling is switched off you can enter different x, y and Z scales.

**Scale original**: This will delete the original objects and relocate them in the newly rotated position.

**Copy**: The original objects being rotated will be retained and a copy of them will be generated in the newly rotated position.

**Repeat**: The original objects will be repeated a number of times through the angle of rotation in the specified direction of rotation. Enter the number of repeats.

**Mirroring objects**

- **Mirror**
  - Mirror objects.
- **&mirror**
  - And mirror objects.

You can mirror objects around a specific angle, around the axis acquired from an object or 2 points.

**Mirror original**: This will delete the original objects being mirrored and relocate them in the newly mirrored position.

**Copy**: The original objects being mirrored will be retained and a copy of them will be generated in the newly mirrored position.

**Translating objects**

- **Transl**
  - Translate objects.

Use this command to translate objects in a drawing. This command is extremely useful and versatile as you can use it in conjunction with the **&rotate**, **&scale** and **&mirror** commands to manipulate objects in a drawing.
Move: Objects will be moved and original objects deleted.

Copy: Objects will be copied and original objects will be retained.

Skip: Objects will be moved but you can select multiple points before final placement of objects.

Repeat: Objects will be repeated a specified number of times. Enter the desired repeat value. The direction and distance between objects repeated will depend on the values you enter for the start of translation and position for next copy.

Repeat to Point: Objects will be repeated a specified distance and direction up to an indicated point in the drawing. Caddie will repeat the objects as many times as it can fit between the start and end points specified at the specified distance between each object.

Repeat Equally Spaced: Specify the maximum spacing between repeated objects. Indicate the start and end points to repeat the objects. Caddie will fit as many objects between the specified start and end points.

Moving patterns

PatMov Move an indicated pattern(s).

Click on the pattern you wish to move. If the pattern you click on is part of a group of patterns, Caddie displays a list of patterns linked to the one you clicked on. Select the relevant pattern from the list and indicate the start point and the end point for the move.

Copying patterns

PatCopy Copy an indicated pattern(s).

This command is similar to the PatMov command (See above) except that Caddie makes a copy of and retains the original pattern being copied.

Repeating patterns

PatRep Repeating an indicated pattern(s).

Click on the pattern you wish to repeat. Indicate the start of the translation. This is the point FROM which you want to repeat the pattern. Enter the new location. This is the point TO which you want to repeat the pattern. Caddie uses the distance between the two points indicated as the spacing between the repeated patterns at the angle subtended between the two points in the direction from the first point to the second point. Finally, Caddie prompts you to enter the
number of times to repeat the pattern. The number of resultant patterns is the number of repeats you entered plus 1 (this being the original pattern).

Creating a Matrix from objects

| Matrix | Create a regular row-column matrix from existing drawing objects. |

Enter the X and Y distances for the matrix offset, or click on the Pick button to indicate these distances on the drawing. Enter the number or offsets vertically and horizontally. You can also click on the Pick button to indicate the X and Y values on the drawing.

![Matrix Diagram]

Mirroring a pattern about a line

| PatLMir | Mirror a pattern about an indicated line. |

Indicate the pattern to mirror, and indicate the line for the angle of the mirror. Indicate the point about which you wish to mirror. You have the option to keep or discard the original pattern.

![Mirroring Diagram]

Fit patterns to a reference line

| FitPat | Fit selected objects to indicated reference lines. |

Indicate a reference line on the drawing and a line of the pattern. The pattern will be copied, rotated and scaled to fit onto the indicated reference line. This command can be used to assemble, for example, air-conditioning layouts.
Chapter 13 Transformation Commands

Copying, scaling and/or rotating selected entities

**Entity** Copy, scale and/or rotate drawing entities.

Caddie displays a pop-up menu. The top options act on single elements and the bottom options allow you to select multiple entities to be manipulated. This command is, in fact, just a variation of the **Transit**, **Rotate**, **Scale** and **Mirror** commands.

Creating a detailed copy of part of a drawing

**DetCopy** Generate a detailed copy of parts of a drawing.

This command enables you to cut out a section of a drawing and relocate to another area whilst at the same time rotating or scaling or mirroring the cut out section before placing it at the new location.

Rescale a drawing from a measurement

**Resize** Resize a drawing from a known measurement.

This function is handy to use for rescaling drawings. You can use this function to rescale older Caddie drawings or imported drawings. You can enter the scaling factor, or you can calculate the scaling factor from a line, 2 points or scale from one unit setting to another.

![Resize Drawing dialog box](image)

When you are rescaling from a line or 2 points, Caddie will display the distance in the dialogue box. Enter the correct distance between the given points. Caddie will scale the drawing accordingly.

**Rescale Finishes**: Check this option if you want to rescale finishes with the drawing.

**Indicate Origin**: Check this option if you want to indicate the origin for rescaling.

If you are scaling from one unit to another, set the modelspace units.

**TIP**: Use **MEASR | AltUnt** to check for alternative units when measuring.
Mirror about a line

**MirLine**  Mirror marked objects about an existing line.

Use this command to mirror objects about a specified angle.

Rotate a pattern

**Pat-Rot**  Mark and rotate patterns.

This command enables you to mark and rotate a single pattern. Indicate the pattern to rotate.

If the object you select is in more than one pattern, Caddie displays a dialogue listing the patterns in that group. Select the pattern you want to rotate. Indicate the position about which the pattern is to be rotated and an existing control point and a new position. The angle calculated between the 2 points determines the angle through which Caddie will sweep to rotate the pattern.

Stretching parts of a drawing

**Stretch**  Stretch parts of a drawing.

**QStrch**  Quick stretch parts of a drawing.

This command is one of the most powerful and useful commands in Caddie. It enables you to modify large sections of a drawing very rapidly. An important difference with the stretch command in comparison to other transformation commands is that entire entities do **NOT** have to be included in the traced outline to be affected by the stretch execution. Any end point that is included in the traced outline will be stretched.

**Rectangle:** Trace a rectangular shape around the area to be stretched.

**Polygon:** Trace a polygon shape around the area to be stretched.

**Drag Stretched Entities:** Caddie will use virtual lines to show the execution of the stretch command as you indicate the desired destination point for the stretched objects.

**Save Quick Button:** Set up frequently used settings and save to the quick button to eliminate the dialogue.
Stretching examples:

A1 → B1

A2 → B2

A3 → B3

A4 → B4
Stretching dimensions

Cutting through dimensions when stretching:

When cutting through dimensions, the stretch command behaves differently depending on which of the three Angle options you set. The following illustration demonstrates the result of these different options. Assume that all objects are stretched 10mm to the right in these examples. Two rectangles A and B, demarcated by dotted lines represent the area to be stretched.

**Fig A:** The original object before stretching.

**Fig B:** Stretch rectangle A with dimension angle set to Unchanged. Note that even if you had set this option to **Generate New Angle**, the result would have been identical since the dimensions are horizontal and the effect of the stretch execution is also horizontal. Notice that the dimension of 10mm has been automatically updated to 20mm to reflect the new dimension of the line.

**Fig D:** Stretch rectangle B with dimension angle set to **Leave Unchanged**. Notice that the dimension of 32mm has now been updated to 41mm to reflect the new dimension of the line but that the angle of the dimension is the same as that of the line before it was stretched. In other words, Caddie has retained the original angle of the dimension.

**Fig C:** Stretch rectangle B with dimension angle set to **Generate New Angle**. Notice that the dimension of 32mm has now been updated to 41mm to reflect the new dimension of the line and that the angle of the dimension is the same as that of the line.
Stretching complex shapes:

There are many times when you cannot use the rectangle option to stretch objects as this would give you undesirable results. In cases like these, you need to select the polygon option and trace an irregular path around the objects that you want to include in the stretch execution. Refer to the following illustration where an entire building needed to be stretched 1000mm to the right. The polygon trace line is shown in the left portion of the following illustration.

Note the following points:

The stretch execution cut through three dimensions and these have been automatically updated indicated by arrows.

The window at the bottom of the drawing moved across by 1000mm.

The entrance door to the office labelled DIRECTOR also moved across by 1000mm.

Examine the rest of the drawing and observe what was stretched/moved during the execution of the stretch command. This is a good example of where the stretch command can be used to stretch objects as well as to move them.

The rule here is that if objects are entirely enclosed within the trace, they will in fact move and not be stretched.
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CHAPTER 14

Dimension Commands

- Setup Dimension Parameters
- Change Parameters of Existing Dimensions
- Insert Dimensions in a Drawing
- Move and Align Dimensions
- Edit Dimension Text and Angle
- Dimension different Objects
CHAPTER 14 - Dimension Commands

Introduction to dimension commands

It is customary first to setup the dimension parameters before you actually insert dimensions into a drawing.

All dimensions use the same principle in laying them down. You indicate a FROM position and a TO position. It is up to you to decide what snap mode to use for these two points. By default, Caddie automatically changes the snap mode to Auto snap mode as soon as you click on one of the DIMS commands. The reason for this is that it makes it easier when you have many dimension points that constitute various control points. For example, you may need to dimension from an end point to an intersection, to the mid point of a line, to the centre of a circle, to another end point, and so forth.

It would be very frustrating to have to change to a different snap mode every time you indicate the control points.

Inserting Dimensions

Inserting horizontal, vertical and diagonal dimensions

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horiztl</td>
<td>Insert a horizontal set of dimensions.</td>
</tr>
<tr>
<td>Verticl</td>
<td>Insert a vertical set of dimensions.</td>
</tr>
<tr>
<td>Diagnal</td>
<td>Insert a diagonal set of dimensions.</td>
</tr>
</tbody>
</table>

All three above commands execute in exactly the same manner. It is only the resultant dimension that differs.

**Horiztl**: Gives the Horizontal (X) component of a dimension.

**Verticl**: Gives the Vertical (Y) component of a dimension.

**Diagnal**: the diagonal component of a dimension.

The **Horiztl** and **Verticl** commands will ALWAYS yield the horizontal (UCS X axis) and vertical (UCS Y axis) dimensions between indicated points. This means that no matter at what angle a line lies at, the resultant dimension will always be either horizontal or vertical depending on which of the two commands you selected. The **Diagnal** command will ALWAYS yield the actual length of a line.

In the illustration, dimensioning line AB - Using the **Horiztl** command will yield 35.47; using the **Verticl** command will yield 19.50; using the **Diagnal** command will yield 40.47.
Auto dimensioning

**AutoDim** Insert a set of horizontal, vertical and diagonal dimensions.

This command will insert a horizontal, vertical and diagonal dimension between 2 points or on a line. Enter the offset in actual units from the points. Indicate on the drawing the direction for the dimensions.

<table>
<thead>
<tr>
<th>Dimensions</th>
<th>Dimension Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical</td>
<td>Between 2 points</td>
</tr>
<tr>
<td>Horizontal</td>
<td>Dimension the Line</td>
</tr>
<tr>
<td>Diagonal</td>
<td></td>
</tr>
</tbody>
</table>

Offset from line: 2

Diametric and radial dimensioning

**Diam** Dimension the diameter of a circle or arc.

**Radius** Dimension the radius of a circle or arc.

These commands will dimension the diameter/radius of a circle or an arc. In the following illustration, the circles and the arcs happen to have the same diameter. It is only the position of the dimensions that is different. The positioning of the dimensions is determined when you are prompted for the dimension text position. If you click outside the circle/arc the dimension is placed outside the circle/arc. If you click inside the circle/arc, the dimension is placed inside the circle/arc.

Dimensioning the angle between two lines

**Angle** Dimension the angle between two lines.

This command will dimension the angle between two lines. Indicate the 2 lines and select the position for the dimension arc position.
Dimension Commands

**Dimensioning the circumference of an arc**

**Circum**  
Dimension the circumference of an arc.

Dimension to circumference of an arc. Indicate the 2 points for the dimension in a clockwise direction and indicate a position for the dimension arc.

In the illustration the circumference of arc AB is to be dimensioned. If you click first at end point A and secondly at end point B, the result is 29.53mm. If you click on end point B as the first point and end point A as the second point, the result is 68.80mm. The second result is the supplementary value. In other words, if you add these two values you will get the circumference of the complete circle.

**Aligning dimensions**

**Align**  
Aligning dimensions to one another.

There may be times when you want to align dimensions to one another. This situation usually occurs when dimensions are unchained and they are all staggered.

Change the **Auto Snap** mode to include dimension control points to snap accurately when aligning dimensions.

**Continue dimensioning**

**Continu**  
Continue dimensioning from an existing dimension set.

There will be times when you have had to interrupt a dimension set. To continue dimensioning along this dimension set, you do not need to select one of the dimension commands directly but you can carry on from an existing dimension set and continue the dimensioning. Caddie will automatically acquire the properties of the dimension you continue from.

**Move a dimension**

**MovDim**  
Move an entire dimension set.

There will often be times when you need to move dimensions to new locations. Use the **MovDim** command to achieve this. Indicate the dimension you want to move and indicate a new position for the dimension.

**Move dimension text**

**MovTxt**  
Move the text of a dimension.

You can move the dimension text to a new position; you can move the dimension text to a new position and add a leader arrow; you can reset the dimension text position to the original position.
**Dimension the angle between three points**

3P-Ang Dimension the angle between three points

Use this command to dimension the angle between three indicated points. The dimension arc is virtually displayed until you indicate the position on the drawing.

The lines in the following illustration are merely to highlight the angle and are not needed for the execution of the command.

![Angle between three points](image)

**Edit dimension text**

EditTxt Edit the text of a dimension.

Use this command to change the text of a dimension. The text position can be before a dimension, replace the dimension, after the dimension or replace the dimension text. If you want the text to appear on a separate line, prefix the text with \X (capital X).

**Insert horizontal and vertical witness line dimension**

WitLnV Insert a dimension whose witness lines are vertical.

WitLnH Insert a dimension whose witness lines are horizontal.

Using these dimension commands will result in dimensions with either vertical or horizontal witness lines. Ordinarily, Caddie draws witness lines perpendicular to the line being dimensioned.

**Dimension a chamfer**

Chamfer Dimension a chamfer.

This is a quick way to dimension a chamfer. Simply indicate the chamfer and the text position.

**Set the datum reference**

SetDatm Set the parameters for datum dimensions.

Enter the desired values for the datum group on the dialogue.
Height of datum: Set the base reference height to be used as the elevation for all the other datum points in the datum set. This value will be added to all points in this datum dimension set.

Unit Control: Set the base unit of measurement for this datum dimension set.

Datum Symbol Height in mm: Set the height of the datum symbol.

Datum Symbol Width in mm: Set the width of the datum symbol.

Datum Symbol Pen: Set the pen to be used to draw the dimension set.

Insert datum reference points

Insert new reference points in an existing set of datum points.

Use this command to insert additional levels in an existing set of datum reference points. Select any datum in the group and indicate the positions for the new datum reference points.

Edit datum base reference

Change the height reference of the base measurement of a datum dimension set.

You can change the base reference height of a datum dimension set using this command. You may need to do this if the original base reference level needs to be adjusted. All the datum levels associated with this group will be updated.

Delete a datum set

Delete an entire datum dimension set.

Use this command to delete an entire datum dimension set. Indicate any datum in the group you want to delete.

Delete a dimension set

Delete an entire dimension set.

Use this command to delete a complete dimension set. Click on the dimension that you wish to delete.
**Explode a dimension set**

**ExplDim**  
Explode a dimension set.

When Caddie draws dimensions, it treats all the components that make up the dimensions as one complete unit, although you can manipulate some components individually. Exploding a dimension set breaks it up into individual components that you can treat as separate objects. The dimension is converted to text, lines and blocks or solids for the headers, depending if the headers are open or filled.

**Changing the angle of dimension text**

**DimAngl**  
Change the angle of a dimension text.

This command enables you to change the angle at which the text of a dimension lies. Enter the new angle for the text. Caddie adjusts the display of the dimension line breaking it so that the text is clearly visible.

**Dimension X-Y coordinates**

**X-Y Pos**  
Insert dimension text displaying X and Y coordinates.

This command enables you to enter X-Y coordinates for selected points in a drawing. Indicate the coordinate position and indicate a position for the text.

The cross-haired circle within the rectangle represents the drawing origin. For the sake of clarity, the leader lines are not shown in the illustration. Once point D has been selected it is simply a matter of clicking on the remaining corners of the rectangle to insert their relative X-Y coordinates.

The top coordinates have been set to **Existing Origin** whilst the bottom coordinates have been set to **New Origin**. For the top coordinates, then, the X-Y coordinates are calculated from the drawing origin (The location of the cross-hair) whilst the bottom set of coordinates are measured from point D which was set as the new origin for all the other coordinates.
**Dimension Commands**

**Chapter 14**

**Stretch a dimension**

**DimStr**  Stretch objects by entering a new value for a dimension set.

This command is used to stretch objects by changing the value of an existing dimension set.

Indicate the dimension text on side to be stretched to indicate the direction for the stretch. Enter the new value for the dimension. This value does NOT represent how MUCH to stretch the dimension but rather what the new value for the dimension should be. Enter a polygon around the points you want to stretch and select **Option End** on the **Context Sensitive** menu to finish your selection.

**Dimension Parameters**

**DimSet**  Setup dimension parameters.

**DimGet**  Get dimension parameters from existing dimensions.

**DimEdit**  Change the parameters of existing dimensions.

See the **Chapter 5 - Object Properties** for a description of the basic dimension parameters.
CHAPTER 15

Measure Commands

Measure Horizontal, Vertical and Diagonal distances
Measure circle and arc diameters and radii
Measure areas
Measure circumferences
Convert decimal angles to degrees-minutes-seconds
Set alternative units
CHAPTER 15 - Measure Commands

Introduction to measuring

The results of the measuring commands are displayed in the command line. You can access the information here by clicking in the history buffer and highlighting text that you want to copy, right-click on the selected value and select Copy.

You can also access the last measurement result by right clicking on a value in a dialogue and selecting the Buffer option.

Measuring X and Y distances

**XY dist** Measure the horizontal, vertical and diagonal distance between two points.

Use this command to measure the X, Y and diagonal (r) distance between 2 points.

Measuring X-Y-Z Distances

**XYZ Dist** Measure the X, Y and Z distance between two points.

This command will measure X, Y and Z distance between 2 points.

Measuring diagonally

**Diagonal** Measure the diagonal distance between two points.

Indicate the 2 points for the diagonal measurement.

Measuring in the X direction

**Meas-X** Measure the distance between two points along the X-axis.

Indicate the 2 points for the X direction measurement. (Horizontal measurement when in top view.)
Chapter 15

Measure Commands

**Measuring in the Y direction**

**Meas-Y** Measure the distance between two points along the Y-axis.

Indicate the 2 points for the Y direction measurement. (Vertical measurement when in top view.)

**Measuring distances in 3D space**

**Meas-Z** Measure the distance between two points in 3D space.

Use this command when measuring in 3D space. Please refer to the Caddie 3D user guide for a detailed discussion of the use of the 3D commands.

**Measuring radii**

**Radius** Measure the radius of an arc or circle.

Indicate the circle, arc or construction circle for the radius measurement.

**Measuring the angle between two points**

**Ang2Pts** Measure the angle between two points from the UCS X-axis.

This command will measure the angle between two points – the resultant angle is relative to the X-axis.

**Measuring the angle of a line**

**Angle** Measure the angle of a line.

Use this command to measure the angle of a line relative to the X-axis.

**Measuring the angle between two lines**

**Ang-Lns** Measure the angle between 2 lines.

Use this command to measure the angle between 2 lines.

**Measuring the circumference, perimeter of a closed shape**

**Perim** Calculate the perimeter measurement of an arc, circle or shape.

This is a handy command to calculate the circumference of a circle or the length of an arc. You can select the measuring method on the dialogue – Endpoints, Inside, Manual, Track Individual or Track Continuous.

**Endpoints:** Use this option to measure the circumference of a closed shape constructed with endpoints.

**Inside:** Use this option to calculate the circumference of a shape formed by intersections and endpoints.

**Manual:** This option is used to calculate the circumference allowing you to indicate the perimeter (circumference) element by element.

**Track Individual:** This option allows you to indicate individual points. You will be prompted to indicate the first point and then the second point to be added to the perimeter.

**Track Continuous:** This option allows you to indicate continuous points for the distances to be added to the perimeter.
Measure Commands

Chapter 15

Measure a portion of an arc or circle

**Arc**
Measure a portion of an indicated arc or circle.

Indicate the first point to measure the length from and indicate the second point in a clockwise direction. Indicate the arc/circle circle centre position. The circle or arc length will be displayed in the command line.

Measuring area

**Area**
Measure the area of an enclosed area.

Use this command to measure the area of an enclosed area in a drawing. Select one of the following options:

Caddie displays a pop-up menu with three options.

**Endpoints**: Trace a closed shape by endpoints. Indicate the object to start the outline trace and indicate on the element for the trace direction. Use the same trace method to indicate an island(s) and select **Option End** when completed. The area measurement will be displayed in the command line.

**Inside**: Click inside areas to be traced for calculation. Indicate the island(s) to be subtracted from the area measurement.

**Manual**: Manually select control points enclosing an area to be measured. Select **Option End** when you have finished indicating the control points. – Subtraction of an island is not allowed.

**Closed Polyline**: Caddie will automatically measure the area of a closed polyline shape. – Subtraction of an island is not allowed.

Tracing complete arcs and circles

**PolyArc**
Trace around a complete arc or circle.

Use **PolyArc** to trace complete arcs or circles as part of complex shapes. The complete arc or circle will be used in the calculation.

Tracing portions of arcs and circles

**PolyTrk**
Trace around part of an arc or circle.

Use **PolyTrk** to trace around a portion of an arc or circle when working with complex shapes. Indicate the start point on the arc or circle and indicate the second point in a clockwise direction. Use the appropriate snap modes for accuracy. Indicate the arc or circle for the arc centre point. This allows a portion of an arc or circle to be calculated.

Totalling areas

**AreaAd**
Adding indicated areas to yield a total area.

Click inside each area for the total area calculation. As each area is indicated the unit area and the total area are displayed in the command line. If you have set the alternative units, these calculations will be displayed in brackets.

Measure the length of individual entities

**MeaItem**
Measure the length of a selected entity.

Indicate the object for the length measurement. Individual line, arc and circle lengths will be
measured. When you indicate a polyline, the total polyline length will be calculated.

**Measuring the length all the objects in a selection set**

**MeaSMIt**  Measure the length of all the objects in a selection set.

Select the required selection set on the dialogue. The total length of the objects will be calculated. The result is displayed in the command line.

**Converting angular units**

**DMS>Dc**  Convert degrees-minutes-seconds to decimal angles.

**Dc>DMS**  Convert decimal angles to degrees-minutes-seconds.

These two commands enable you to convert angular units.

**Example 1:** Convert 37° 48’ 18” to decimal format

Select the **DMS>Dc** command. Type 37-48-18 at the command line and press [ENTER]. The converted decimal angle **37.80500** is displayed.

**Example 2:** Convert 134.87° to **DMS** format.

Select the **Dc>DMS** command. Type 134.87 at the command line and press [ENTER]. The converted degrees-minutes-seconds angle **134-52-12** is displayed.

**Displaying alternate units**

**AltUnt**  Display alternative units for when making measurements.

Set the alternative units to display the measured values in the drawing units and the alternative units in brackets.
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CHAPTER 16

Text Commands

Enter Text in a Drawing
Import and Export Text
Edit Text
Convert Text Case
Change the Angle of Text
Copy and Move Text
Align Text
Delete and Explode Text
Insert Leaders in a Drawing
Make Global Changes to Text
Rescale Text
CHAPTER 16 - Text Commands

Introduction to text commands

All the annotation commands - text and leader arrows - are grouped together under the TEXT menu. Text is categorised as a finishing object. Text can be edited individually or globally.

Text settings are covered in Chapter 5 – Object Properties.

Entering text in a drawing via the command line

EntCLn  Enter text using the command line.

When entering the text at the command line, you need to press the [Enter] key for the next paragraph line and another [Enter] is needed to complete the command and enter the text on the drawing. The current text parameters will be used. Use this for entering single text strings.

Entering text in a drawing via the editor

EntEdtr  Enter text using the built-in text editor.

Larger text string can easily be entered by using the text editor.

The current text parameter settings will be used for the text. You can override these for this session by changing the parameters at the top of the dialogue. Two other features in the editor are Find and Replace text as well as a Spell Checker for the text contents in the editor. Refer to the FndTxt command further on in this chapter to find and replace text in the drawing.

Load existing Style: Load a Saved Preset.

Indicate Angle: When you click OK, Caddie will prompt you to indicate a second point for the angle. As you move the mouse you will see a virtual line being dragged. The angle between the 2 points will determine the angle of the text.

Indicate Line: Click on an existing line in the drawing for Caddie to calculate the angle for the text.

None: You don’t have to indicate an angle, the current settings will be used.

Use Caddie settings for display: Click this option if you want the editor window background settings to be the same as in the drawing.
Chapter 16

Text Commands

**Word Wrap:** Switch word wrap on to display long strings of text in the editor to make the display and editing thereof easier.

![Zoom In and Out]

You can zoom in and out of the text editor to make the text larger or smaller.

**NOTE:** You can change the way in which Caddie displays TrueType fonts on the screen. If you have a large number of text strings in the drawing, you can display the TrueType fonts as outline text to speed up the text regeneration. Use **Settings | Drawing Settings | Text.**

![Drawing Settings]

**TIP:** Use the copy **[Ctrl+C]** and paste **[Ctrl+V]** functions to copy and paste text from other application.

**Editing text in a drawing via the command line**

**EditCLn** Edit text using the command line.

Select the text you want to edit. Edit the text string in the command line and press the **[Enter]** key for the next paragraph line and another **[Enter]** is needed to complete the command and enter the text on the drawing.

**Editing text in a drawing via the editor**

**EdtEdtr** Edit text using the built-in text editor.

Indicate the text to edit. When you click on the text it will be displayed in the text editor. Modify the text and select **OK** to apply the changes.

**Importing and exporting text**

**Imp/Exp** Import text into and export text from Caddie.

Use this command to export or import ASCII text. Importing text can save you time in preventing you from re-typing large blocks of text whilst exporting text can be used to add to reports and other documentation.
You can also copy and paste non-ASCII text import text from other programs into Caddie by using the text editor.

**Importing Text:** The current text settings will be used. The text can be imported as single lines of text or as a paragraph. Enter suitable line spacing when single lines are specified and browse to the ASCII file location.

**Exporting Text:** Type in the name of the text file you want to export text to. Indicate the paragraph or text string to export. Select **More** on the popup if you want to indicate another text string and **Done** when completed.

**Entering text on an element**

- **On-Elm** Enter text directly onto an element(s).

Use this command to enter text directly onto a line, circle or arc.

- **Line Selected:** Enter the text through the text editor and indicate a position for the text. The current text settings will be used to insert the text at the line’s angle.

- **Arc Selected:** Caddie displays the Arc Aligned Text dialogue. The arc alignment can be fitted between 2 points, or left/centre/right aligned. You have to specify if the text must be written on the inside or the outside of the arc, and if the text must be read from the inside or the outside. You can also mirror the text. If the characters are not fitted between the points, you have to specify the character spacing.
Entering text tags on the drawing

**Tags** Enter text tags on the drawing.

Use these tags to customise your drawing by placing personalised text strings on your drawing. The current text parameters will be used when you enter tags. Redraw the screen to see the effects of tags on your drawing. The text tags are useful when you want to see the drawing name, description, path, date, time, etc. on the plotted drawing.

<table>
<thead>
<tr>
<th>Date Normal</th>
<th>Date Full</th>
</tr>
</thead>
<tbody>
<tr>
<td>Date Numeric</td>
<td>Date Month , Year</td>
</tr>
<tr>
<td>Time</td>
<td>Drawing Name Full + Path</td>
</tr>
<tr>
<td></td>
<td>Drawing Name No + Path</td>
</tr>
<tr>
<td></td>
<td>Drawing Description</td>
</tr>
<tr>
<td></td>
<td>Drawing Revision</td>
</tr>
<tr>
<td></td>
<td>Current Pattern Name</td>
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<td></td>
<td>Print Time</td>
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<tr>
<td></td>
<td>Drawing Date</td>
</tr>
<tr>
<td>Drawing Time</td>
<td>Drawing Size</td>
</tr>
<tr>
<td>Sheet Name</td>
<td>Cancel</td>
</tr>
</tbody>
</table>

Converting the case of text

**Up-Low** Convert the case of text.

Use this command to change the case of existing text. Simply indicate the text string to change the text to uppercase or lowercase.

Globally scaling of text

**ScpScal** Resize all text in the scope of a drawing.

You can use this command to scale text up or down by specifying a factor. Use a factor greater than one to increase the text and a factor smaller than 1 to reduce the text.

Changing global properties of text

**ScpEdt** Change selected properties of all text in the scope of a drawing.

This powerful command allows you to change only selected parameters of text globally in the drawing.
If the text of an imported drawing runs over the edges of the drawing, you can simply change the text in the scope of the drawing to less than 1.

**Changing the angle of text**

**RotTxt** Change the angle of selected text.

You can change the angle of text in a drawing through this command. You specify a set angle for all the text, or you can add an angle to the existing text angle.

**Moving and copying text**

**Move** Move text to a new location.

**Copy** Copy text and place the copy in a new location.

The **Move** command deletes the original text and places it at the new position. **Copy** will leave the original text and make a copy of the text at the new position.

**Copying and modify text**

**CpyExt** Copy and modify text.

**Copy and Edit**: Indicate the text to copy and edit the text in the editor. Indicate a new position for the text.

**Copy and Increment**: This command allows you to copy and increment the text. This is handy to use when any numbering needs to be done on the drawing. Text can have 3 incrementations separated by spaces or decimal points. Each increment can be specified and can be different. For a field to remain unchanged, you can increment it with zero (0) and enter a negative increment to reduce the text.
Original text refers to the text you selected in the drawing and resultant text shows you how the increment of the text string will look.

**Copy Multiple:** This powerful command allows you to edit existing text on the drawing. If you need to renumber parking, select the text and add or subtract a value. You can also copy, for example, existing level values to another layer and edit these by adding or subtracting a value.

**Resetting paragraph width**

**RelWdth** Reset the width of a paragraph text.

Use this command to change the width of paragraph text. Indicate the text to change and indicate a position for the width of the paragraph.

**Aligning text**

**TxtAlign** Aligning text vertically or horizontally.

Use this command to align text paragraphs in the drawing.

**Align Paragraph:** Select this option to change the alignment of text within indicated paragraph text. Click on one of the check boxes to select the desired text alignment.
Align Separate Paragraphs: Select this option to change the alignment of a number of text paragraphs relative to one another. Indicate the text for the alignment change: left, centre or right; and top, middle or bottom.

Deleting text

DelTxt Delete selected text.

Indicate the text string to be deleted. Press [Ctrl+Z] to undo this action.

Exploding text

ExplTxt Exploding multi-line text to single line text.

Explode multi-line text to single line text. By default we work with multi-line text (MText).

Merging text to multi-line text

T->MT Change single line text to multi-line text.

By default Caddie uses multi-line text (MText) and some commands can only be executed on multi-line text.

Setting and changing text parameters

TxtSet Set text parameters.
TxtGet Get text parameters.
TxtEdit Edit text parameters.

See Chapter 5 – Object Properties for a description of these commands.

Drawing leader arrows

Leader Draw a leader arrow(s).

Use this command to insert leader arrows into a drawing. Indicate the positions on the screen and select Option End on the Context Sensitive menu to end the command and insert the header. Please note that the header will not be inserted if the leader length is too short. Make the header smaller or extend the leader to display the header.

Moving leader control points

MvLdrPt Move control points of a leader arrow.

Use this command to move nodes on a leader arrow. Indicate the control point to move and indicate the new position.

Drawing continuous leader arrows

CnLead Draw a leader with headers at each control point.

This command allows you to draw a continuous leader with headers at each control point.

Inserting notations

Notate Add a notation comment(s) to a drawing.

This command combines a leader arrow with a text string to notate a drawing. Enter the notation on the dialogue and select the angle for the leader arrow. Indicate the position for the text on the
screen and the position for the leader arrow. You can continue to place this notation on the drawing by indicating the 2 points. The current text and leader arrow setting will be used.

**Deleting leaders**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DelLdr</td>
<td>Delete an entire leader arrow.</td>
</tr>
</tbody>
</table>

Use this command to delete leader arrows. You can click close to any control point of the leader arrow to delete it. Press [Ctrl+Z] to undo this action.

**Exploding leaders**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ExpLdr</td>
<td>Explode a leader(s).</td>
</tr>
</tbody>
</table>

Use this command to convert leaders to elements. The leader lines are converted to lines and the headers are converted to solid or block, depending if the header is filled or not.

**Setting and editing leader parameters**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LdrSet</td>
<td>Set leader parameters.</td>
</tr>
<tr>
<td>LdrGet</td>
<td>Get leader parameters.</td>
</tr>
<tr>
<td>LdrEdit</td>
<td>Edit leader parameters.</td>
</tr>
</tbody>
</table>

Refer to Chapter 5 – Object Properties for a detailed discussion on setting and editing leader parameters.

**Marking a text string**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MrkTxt</td>
<td>Mark text in a drawing based on a string filter.</td>
</tr>
</tbody>
</table>

This command enables you to mark all occurrences of a text string in the drawing. The text can be case sensitive, or not. Set SCOPE | Marked to display only the marked text.

**Mirroring of text**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MirTxt</td>
<td>Mirror text.</td>
</tr>
</tbody>
</table>

Use this command to mirror selected text. Only text in the selection will be affected. The text can be mirrored horizontally, vertically or around a specified UCS angle.
Finding and replacing text

Findtxt Find and replace text in a drawing.

Use this command to search for all occurrences of a specified string of text. You can also replace a text string with another string of text.

Only text in the selection set will be searched. The specified text string(s) will be selected and a virtual cross is placed on the text origin for identification. Use the Layering & Display option on the Context Sensitive menu to isolate the text.

Calculations of numbers

No- Func Add, multiply or average sets of numbers.

Use this function to quickly add, multiply or average a set of values in the drawing. You can indicate the text one by one, or you can mark the text beforehand and select the Marked option. Indicate a position on the drawing for the result.

Underlining text

UndLin Underline text.

Use this command to underline indicated text. A line element is added to the drawing and does not form part of the text string. If the text is deleted, the line will remain in the drawing.

Set auto-size of text

SizeAut Set text size from a predefined listing.

Use this command to set the size of text to a predefined size according to the text listing.
Select the desired height for the text, the scale for the text and the width factor. Set the pen colour for the text. This is a quick way to set up text for annotations. You can also use the **Saved Presets** on the **TxtSet** dialogue.

### Insert text boxes

**Box**  
Insert text boxes around existing text strings.

You can add text boxes to existing text in the drawing; update and delete them.

Select the **Add Box** option to define the text box and the selection. The type of text box can be a rectangle with rounded corners, a rectangle with square corners or a circle. You can specify the pen colour and the scale for the text box. The minimum scale will draw the box on the extents of the text and the maximum scale will draw the box the distance equal to the text height away from the text.

Select the **Update Box** to update the inserted text boxes if the text has changed, moved or rotated, or if the box parameters where changed.

**Delete box** allows you to delete text boxes in the drawing.
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CHAPTER 17

Delete Commands

Deleting Objects in a Drawing
CHAPTER 17 - Delete Commands

Deleting objects from a drawing is common practice. In the DELETE menu you will find all the commands you will need to delete any entity in a drawing. You can quickly delete objects by selecting them with the pointing device and pressing the Delete key on the keyboard. You can toggle the selection of objects, if you click on a selected object, it will become unselected.

Undoing and redoing commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo</td>
<td>Undo drawing commands. [Ctrl+Z]</td>
</tr>
<tr>
<td>Redo</td>
<td>Redo drawing commands. [Ctrl+Y]</td>
</tr>
</tbody>
</table>

As the name of the commands imply, these commands are used to Undo and Redo drawing commands.

Deleting by rectangle or polygon

RecPoly 
Delete drawing objects by rectangle, polygon or by a trace method.

Use this command to delete objects in a drawing, inside or outside defined boundaries.

Outline:

Rectangle: Delete objects by indicating by a rectangular area.

Polygon: Delete objects by indicating by a polygon shape around the objects.

Traced Inside: Delete objects by indicating a traced element boundary.

Traced Endpoints: Delete objects by indicating traced endpoints.

Selection:

Inside: Delete inside the outline.

Outside: Delete outside the outline.

Boundary:

Elements: Delete only entirely included or excluded elements.

Clipped: Clip and elements the boundary crosses.

Save Quick Button:

Use the current setting for the QRecPly button. This command will not prompt you for any
settings; it will simply ask you to indicate the boundary.

**Deleting marked objects**

- **DelMrk**  Delete all marked objects.

Use this command to delete all marked objects from a drawing. Remember too, that if there are more objects to be marked for deletion than there are un-marked, you could first mark all objects you do NOT want to delete and then use the **Swap** command from the **MARK** menu.

**Deleting patterns**

- **DelPat**  Delete a pattern(s).

Use this command to delete patterns from a drawing. Take note that this command only works on patterns. If an object is not part of a pattern Caddie will display an error message informing you that the **Entity is not part of a pattern**. If this is the case you will need to use an alternative delete command. If the indicated pattern is part of a group of patterns, Caddie will list the patterns and you will have to select the required pattern to delete.

**Deleting referenced symbols**

- **DelRef**  Delete a referenced symbol(s).

Use this command to delete the indicated referenced symbols from a drawing.

**Deleting leaders**

- **DelLdr**  Delete leaders.

Use this command to delete leaders from a drawing.

**Deleting hatching**

- **DelHch**  Delete hatching.  
  
  Use this command to delete hatching from a drawing.

**Deleting text**

- **DelTxt**  Delete text.

Use this command to delete text from a drawing.

**Deleting dimensions**

- **DelDim**  Delete dimensions.

Use this command to delete dimensions from a drawing.

**Deleting layers**

- **DelLay**  Delete layers.

Use this command to delete layers from a drawing. Select a layer from the dialogue.

**Deleting points**

- **DelPnt**  Delete construction points.

Use this command to delete construction points from a drawing. You can indicate the point to delete; you can enter the index-number of the point or you can delete all the points in the scope.
Delete Commands

Chapter 17

Deleting splines

DelSpl  Delete splines.

Use this command to delete splines from a drawing.

Deleting construction lines

DelCon  Delete construction lines and circles.

Use this command to delete construction lines and circles from a drawing.

Deleting all construction elements

AllCon  Delete all construction lines and circles from the drawing.

Use this command to delete all construction lines and circles from the drawing. Construction points will not be deleted by this command.

Deleting last object

DelLst  Delete the last object created.

Use this command to delete the last created object from a drawing.

Deleting objects

DelObj  Delete objects.

Use this command to delete objects from a drawing. This command can be used to delete any object in a drawing.

Deleting segments

DelSeg  Delete a segment.

Use this command to delete line or arc segments from elements in a drawing. A segment in can be a complete element or an element which is bisected by a construction line, line, arc or circle.

Deleting duplicate objects

DelDup  Delete duplicate objects.

Use this command to delete duplicated objects from a drawing. The command will report the number of duplicate objects in the drawing.

Deleting pictures

DelPic  Delete a picture.

Use this command to delete pictures from a drawing.

Deleting all drawing entities

DelAll  Delete all drawing entities.

Use this command to delete ALL entities and settings from a drawing. This command effectively deletes the entire drawing and sets the settings of the drawing to the default settings of a new Caddie drawing - with no template.
CHAPTER 18

Hatch Commands

- Hatch Simple Areas using Auto Hatching
- Hatch Complex Areas by Association
- Change Hatch Origin
- Add, Delete and Move Control Points
- Stretch Hatching
- Move Hatch across Planes
- Trace Islands
- Trace Arcs and Circles
CHAPTER 18 - Hatch Commands

Introduction to hatch commands

The HATCH menu contains all the tools to hatch any area in a drawing, even areas that are not enclosed. Please refer to Chapter 5 – Object Properties for the hatch parameters settings.

Automatic hatch of areas

Auto Hatch inside areas as well as identifying islands.

Hatch enclosed areas with or without included islands. The sample drawings were hatched using line style 1 at an angle of 45°.

Hatching areas without islands:

If the sample drawing is a lines rectangle, select the Inside option. If the sample drawing is a polygon, select the Closed Object option.

If you had several areas to hatch, you can simply click inside all the areas or indicate all the closed objects for the hatch.

Hatching areas with islands:

In this sample drawing there are two islands that we want to exclude from the hatching process. Click inside the outline and then inside the islands.

Hatching complex areas:

Use the Auto command to indicate the outline and the islands for the sample drawing.

Auto hatching closed objects:

Use the Closed Object to indicate the outline and islands of closed objects like polygons.

Abort hatch process

Abort Abort any hatch process.

Use this command to abort any hatch process.

Inside hatch

Inside Hatch inside indicated areas, no island definitions.
Use this command to hatch inside objects. No island definitions are added. Use the Island command to add an island if needed.

**Hatching by block**

**RecDia**  Hatch inside areas by indicating a rectangular shape.

Use this command if you want to hatch rectangular shapes without tracing outlines. Indicate the diagonal positions for the hatch.

**Adding hatch control points**

**Cnr-Add**  Add a corner control point to a hatch.

This command is used to add control points to an existing hatched area. There may be times when you need to modify a hatched area whereby you need to add new control points to the area.

The rectangle ABCD on the left has been hatched. This object needs to be modified to yield the object to the right in which points E and F have been added and the area ADEF now needs to be included in the hatched area. Indicate the hatch to be edited. You have to indicate the existing two control points between which you want to add one or more new control points. End the new control point selection with **Option End** on the **Context Sensitive** menu.

**Deleting hatch control points**

**Cnr-Del**  Delete a corner control point of a hatch.

Indicate the hatch definition to edit and indicate the node(s) to delete.

**Move hatch control points**

**Cnr-Mov**  Move a corner (Control point) of a hatch.

Indicate the hatch to edit and indicate the node(s) to move and indicate the new position for the control point.

**Change the origin of a hatch**

**OrgEdit**  Change the origin of a hatch.

Use this command to change the origin of a hatched area. This may sometimes be necessary when you use hatch patterns or lines and you want the pattern to fit correctly within the bounds of the hatched area. Indicate the hatch for the origin edit and indicate the position for the origin.
Hatch by association

Asociat | Hatch an area by association.

The Auto, Inside and RecDia commands will cover most of your hatch requirements, but there might be times when islands and outlines overlap, or not all islands must be excluded. At times like these you need to be able to use the Asociat command. It is a longer process to hatch an area but you have a lot more control in the path that the hatch traces.

The name Associative hatching derives from the fact that, when using this technique, you associate various control points with each other to define the boundaries of both the outline of an area to be hatched as well as any islands you want to exclude from the hatching.

If you wish to hatch the area bounded by points 1, 2, 3, 4, 5 and 6, you need to understand the principles of associative hatching so we use this method to hatch this area. There are two islands to exclude from the hatching. One is the circle labelled C1 and the other the area bounded by rectangle ABCD but tracking around the circumference of the circle labelled C2. Let’s see how to achieve this.

For this exercise, we will use the Auto Snap mode set to freehand, intersection, quadrant and end point snaps. Snap | Autosnap Settings.

Define the outline:

Select the Asociat command and start at point 1 and indicate point 2 as the next point.

We now need to indicate the segment of arc A1 from point 2 to point 3 with the PolyTrk command in a counter clockwise direction. While still busy with the Asociat command, select the PolyTrk command and indicate the centre point of the arc by clicking on the circumference of the arc. Indicate the end of the arc at point 3 and select an anti-clockwise direction for the arc.

Indicate point 4 and 5 for the next positions.

We now need to track the full arc A2 with the PolyArc command. Indicate the arc on the circumference. Point 6 is automatically selected.

Select Option End on the Context Sensitive menu to end the definition of the outline.

Define the island C1:

The Asociat command will now prompt you to indicate the islands, but the cursor must be positioned at the start of the island, so snap to the quadrant of arc C1 at the 3 o’clock position. Select PolyArc and indicate arc C1.
Select **Option End** on the **Context Sensitive** menu to end the first island definition.

**Define the island C2:**

Position the cursor at position C where the circle and the rectangle intersect. Select the **PolyTrk** command and indicate the centre for C2 and the end position at D. The arc direction is clockwise. Continue with the island definition by indicating the rectangle corner positions.

Select **Option End** on the **Context Sensitive** menu to end the second island definition. Select **Option End** again on the **Context Sensitive** menu to end the hatch definition.

**Adding or delete islands**

<table>
<thead>
<tr>
<th>Island</th>
<th>Add islands to or remove islands from existing hatching.</th>
</tr>
</thead>
</table>

Use this command to add islands to existing hatched areas in a drawing. Islands can be added by indicating end points, manually indicated positions or by inside tracing. Simply click on the island if you want to remove it.

**Tracking arcs and circles**

<table>
<thead>
<tr>
<th>PolyArc</th>
<th>Trace around <strong>COMPLETE</strong> arcs or circles.</th>
</tr>
</thead>
<tbody>
<tr>
<td>PolyTrk</td>
<td>Trace around <strong>PARTIAL</strong> arcs or circles.</td>
</tr>
</tbody>
</table>

These commands are used in conjunction with other hatch commands to trace around arcs and circles. See the discussion on the **Asociat** command earlier in this chapter for details on the usage of these two commands.

**Moving hatches within and between planes**

<table>
<thead>
<tr>
<th>Mv2FrPl</th>
<th>Move hatch to front plane.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mv2BkPl</td>
<td>Move hatch to back plane.</td>
</tr>
<tr>
<td>MvForw</td>
<td>Move hatch to front of object.</td>
</tr>
<tr>
<td>MvBckw</td>
<td>Move hatch to back of object.</td>
</tr>
</tbody>
</table>

These commands enable you to move hatched areas forwards and backwards relative to other objects. Refer to the example below.

**Mv2FrPln:** Clicking on the object with the black background in Hatch A yields Hatch B. If you now click on the shape with the white background, Hatch C is the result.
**Hatch Commands**

**Chapter 18**

**MvForw:** Clicking on the object with the black background in Hatch A will not result in any visible difference on the screen since you are merely moving the hatch forward in a plane relative the object itself and not to any of the other objects such as is the case with the **Mv2FrPlan** and **Mv2BkPln** commands.

**Deleting hatch**

**DelHch**  
Delete hatching.

Use this command to delete hatch. If the incorrect hatch definition is deleted use **[Ctrl+Z]** to undo this action.

**Convert hatch patterns or lines to line elements**

**ExplHtc**  
Explode hatching.

Use this command to convert line and pattern hatch definitions to line elements. You can convert single hatch to the original/current pattern or the entire hatch in the scope to original/current patterns.

**Hatch parameter set, get and edit**

**HtcSet**  
Set the hatch properties.

**HtcGet**  
Acquire the hatch properties of an indicated hatch.

**HtcEdit**  
Apply the current hatch properties to the indicated hatch.

---

**Hatch Parameter Settings**

- **Hatch Type**
  - **Type:**
    - [ ] All
    - [ ] Solid
    - [ ] Chain
    - [ ] Cross
    - [ ] Dashed
    - [ ] Dotted
    - [ ] Double Dashed
    - [ ] Double Dotted
  - **Space:**
    - [ ] Select
    - [ ] Horizontal
    - [ ] Vertical
    - [ ] Angle
    - [ ] Radius
  - **Scale:**
    - [ ] Select
    - [ ] Horizontal
    - [ ] Vertical
    - [ ] Angle
    - [ ] Radius
  - **Angle:**
    - [ ] Select
    - [ ] Horizontal
    - [ ] Vertical
    - [ ] Angle
    - [ ] Radius
  - **Auto Layering:**
    - [ ] Enable for Fill and Gradient Hatch
    - [ ] Enable for Line and Pattern Hatch
  - **Saved Presets**:
    - **Name:**
      - [ ] Load
      - [ ] Save

**Hatch Preview**

- **Colour / Pen:**
  - [ ] Red
  - [ ] Green
  - [ ] Blue
  - [ ] Black
  - [ ] White
  - [ ] Yellow
  - [ ] Cyan
  - [ ] Magenta
  - [ ] Gray
  - [ ] Beige
  - [ ] Pink
  - [ ] Aqua
  - [ ] Purple
  - [ ] Olive
  - [ ] Olive Green
  - [ ] Olive Drab
  - [ ] Reddish Brown
  - [ ] Brown
  - [ ] Greenish Brown
  - [ ] Green
  - [ ] Yellowish Green
  - [ ] Lime Green
  - [ ] Greenish Yellow
  - [ ] Blue
  - [ ] Greenish Blue
  - [ ] Brownish Blue
  - [ ] Turquoise
  - [ ] Dark Turquoise
  - [ ] Sky Blue
  - [ ] Light Blue
  - [ ] Light Sky Blue
  - [ ] Blueish Green
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Hatch Type:

There are 5 types of hatching you can use. You can hatch with line style, patterns, fills, gradient fills and custom patterns.

Line Style: Hatch with lines using a specified colour / pen and line style. Set the hatch spacing and angle.

Pattern: Hatch by selecting from a list of predefined patterns. Set the scale and angle.

Fill: Hatch using solid colour fills. You can set the colour / pen for the fill or you can tick the Use RGB Colour to ensure the correct colour is passed to the printer.

Gradient fill: Hatch that is composed of two colours and in between the colours is a smooth blending. The grading can be displayed as linear, cylinder, inverted cylinder, spherical, inverted spherical, hemispherical, inverted hemispherical, curved and inverted curve.

Custom: These are custom hatch pattern files.

Parameters:

The different hatch types have different parameters for selection.

Plane:

Back and front: Any object that you hatch has both a front and a back plane. By hatching either plane, you can create special effects by moving hatches in front of or behind other objects.

Auto Layering:

Enable for Fill and Gradient Hatch: You can specify a layer for any fills or gradient hatches. This layer will be used for these hatches regardless of the active layer setting.

Enable for Line and Pattern Hatch: Line style, pattern and custom hatch patterns will be placed on this layer.

Saved Presets:

Frequently used hatch parameters can be saved as presets. These presets can be loaded in the current drawing and is also available for loading in other drawings.

Swap the direction of line hatch

SwapDir  Swap the direction of line hatch.

Use this command if you need to swap the direction for broken line hatch definitions for display purposes.

Vectorised hatching

Vector  Hatch an area using lines and not hatch.

This command uses lines instead of hatch definitions. No hatch commands can be executed on this type of hatching.

Create a boundary around text or dimension text

TxtBlok  Draw a rectangular border around text.

DimBlok  Draw a rectangular border around dimension text.
If you do not want to use opaques to blank the fill below text or dimension text, you can add rectangle boundaries around the text and dimension text to use as island definitions when hatching.

**Calculate a hatch area**

### ClcArea

Calculate a hatch area.

This command enquires the area of a hatch definition. The island definitions will be subtracted for the calculation. Switch alternative units on with the **MEASR | AltUnt** command. For example, calculate the area in working units (mm) and alternative units (m).

**Hatch an arc or circle**

### Circle

Quick hatch of arcs and circles.

Use this command to hatch arc and circle quickly. You can hatch a circular area by indicating the centre and edge points manually. You can also hatch the full circle, closed arc or wedged arc by indicating the arcs and circles.
CHAPTER 19

Modify Commands

Modify Segments
Trim Elements
Create Fillets and Chamfers
Break Elements
Divide Elements and Between Points
Draw Multiple Offsets
Insert Symbol and Text Tokens
Cleanup Wall Crossings
Insert Token Symbols
CHAPTER 19 - Modify Commands

Introduction to modify commands

The Modify menu has a host of commands that are useful when modifying a drawing. Create your own toolbar with the most frequently used modify commands for quick access.

Editing segments

**EdtSeg** Change the pen number, line style and line width of segments.

Set the pen colour, line style and line width on the dialogue and indicate the segment of the line, arc of circle you wish to edit.

Modify polylines

**ModPol** Modify a polyline.

There are a range of functions to modify polylines.

- You can merge an element into a polyline.
- You can add nodes to polyline ends, insert and delete nodes.
- You can change a line segment to an arc, or change an arc into a line segment and mirror an arc segment.
- You can move polyline control points. This function also allows you to change the widths of the polyline by specifying the start and end widths for each segment.
- The complete polyline direction can be reversed.

Make a polyline(s)

**MkPol** Create a polyline from existing lines and arc elements in the drawing.

You can create a polyline or polylines from existing lines and arcs in the drawing.

Selection:

The selection set will determine which elements will be converted to polylines. All the objects in the scope, selected objects, marked objects, included in a rectangle, included in a polygon or an indicated pattern will be converted to polylines.

If the Trace method is selected, you have to indicate the elements to start the trace and the direction for the trace.
New Objects:

You can keep the original colour and layer for the new polylines or you can set the new colour and layer on the dialogue. The created polylines can be included in the current pattern, added to a new pattern, no pattern or it can be added to the original patterns.

Polyline creation settings:

The maximum gap option enables you to control the joining of elements that have a space between them due to inaccuracy. The original objects can be kept or erased.

Trimming elements to a point

TrmElPt   Trim an element or elements to a point.

Use this command to trim or extend elements to an indicated point on the drawing. Indicate a trim/extend point on the drawing and continue to indicate elements for trimming/extend.

Trimming elements to another element

TrmElm   Trim an element or elements to another element.

Use this command to trim or extend elements to an element in the drawing. You can continue indicating elements to trim/extend to this element.

TIP: If you have to trim or extend a large number of elements you can indicate crossing lines running across all the elements you wish to trim or extend. Select Option End to end the crossing selection.
Example:

In the illustration above, we need to trim and extend the tops of all the balusters so that they all follow the curvature of the curved ones. Insert a temporarily arc by selecting the CURVE – Parlel command.

Use the TrmElm command to trim and extend the balusters to the arc. Indicate the temporary arc and select a point to the left of the first baluster and a second point to the right of the last baluster, crossing all relevant lines. Select Option End to end the command. All lines crossed will be trimmed or extended to the indicated element.

Trimming two element to each other

TrmElEl Trim two elements to each other.

This command will trim or extend two elements to one another. Continue indicating the first and second elements to trim to each other.

Fillet and chamfering of elements

The illustration below demonstrates the use of the Fillet, ChmSym, ChmOfs and ChmAng commands. Please note that the elements being filleted or chamfered do not have to intersect. You have the option to trim (extend) neither element, trim only the first element, trim only the second element of trim both elements.
Filleting corners

Fillet Insert a fillet on a corner.

Use this command to insert a fillet at the intersection or apparent intersection point of two elements. Note that these elements do not have to intersect. (Refer to D in the illustration.)

Symmetrical chamfer

ChmSym Insert a symmetrical chamfer on a corner using a diagonal measurement.

Use this command to insert a symmetrical chamfer at the intersection or apparent intersection of two elements. (Refer to A in the illustration.)

Chamfer with offsets

ChmOfs Insert a chamfer on a corner using offset measurements.

Use this command to insert a chamfer with offset measurements. Enter the 2 offsets and the trim settings on the dialogue. (Refer to B in the illustration.)

Chamfer with an offset and an angle

ChmAng Insert a chamfer on a corner at an angle with an offset measurement.

Use this command to insert a chamfer at a specified angle with an offset distance. Enter these values on the dialogue and indicate the two elements on the correct side. (Refer to C in the illustration.)

Breaking elements

There are many situations that will require you to break up elements into smaller segments. The original element will replaced by a number segment.

Breaking elements

BrElm Break an element (line, arc or circle).

Use this command to break an element up into separate elements.

Break the element AB at point B:

Break Point

A B A B C

Original Result

Breaking an element between points

BrOuPts Break an element between two points (Line, arc or circle).

Use this command to break out an element between two indicated points on the element. Once the command has completed its execution, Caddie removes the element segment between the two indicated points.

Break out an element from element AB between indicated points:
Breaking an element between two elements

**BrOutElm**  Break out an element between two elements.

Use this command to break out an element between two other elements.

Break out an element between elements AB and DC:

Break and divide elements equally

**DivBrk**  Break and divide an element into equal segments.

Use this command break an element and divide it up into equal segments. A point will be placed at the break point for easy identification.

Dividing elements

There are many situations that will require you to divide elements into segments. The divide functionality does not break elements apart; it simply divides the elements into a number of segments using construction lines or points.

Dividing elements with construction lines

**DivElCl**  Divide an element with construction lines.

Use this command to divide an element into equal segments and insert construction lines normal to the elements at the division points.

Dividing elements with construction points

**DivElPt**  Divide an element with construction points.

Use this command to divide an element into equal segments and draw construction points at the division points.
Chapter 19
Modify Commands

Divide the space between two elements with construction lines

**Div2PCI**  Divide the space between two points with construction lines.

Use this command to divide the space between two indicated points into equal segments and draw construction lines at the division points.

Divide the space between two elements with construction points

**Div2Pts**  Divide the space between two points with construction points.

Use this command to divide the space between two indicated points into equal segments and insert construction points at the division points.

Inserting a point on an element at a specified distance

**DistPt**  Insert a construction point on an element at a specified distance from a certain position on the element.

This command enables you to insert a construction point on an element at a specified distance from another point.

![Diagram](image)

Indicate the line or arc for the distance measurement. Indicate the position on the element from which to measure and indicate an approximate position for the point to get the direction for the measurement. Enter the distance measurement on the dialogue.

Caddie will insert a construction point at the set distance. Use the **CnLIN | Con-Set** command to set the construction point’s parameters for the display.

Drawing offsets

**Offset**  Draw a set of parallel lines or curves at specified offsets

Use this command to draw a number of offset parallel lines, arcs or polylines to an indicated line, arc or polyline.
Offset Parameters:
Enter distances for the various offsets, select the pen type and line style for each offset.

Trace Method:
The objects can be traced by endpoints, inside a closed area or by tracing intersection points.

Pattern:
These new offset lines can be created in the current pattern, stored as a new pattern or not made part of a pattern.

Offsets number:
Set the desired number of offsets to draw.

Distances:
The offsets can be calculated from the base line of incrementally from the previously placed element.

Offset:
You can choose if the offsetting must be on one side or both sides of the elements. If you select one sided offsetting, indicate the side of the traced element(s) for placement of the offset lines.

Offset Type:
The offset elements can be inserted as line and arcs, or polylines.

Corner Type:
The corner type only affects the corners of offsets that radiate outward from the object and they can be Lines to follow the original path of the object(s), Chamfer to chamfer the corners and Fillet to fillet the corners.

Offsetting examples:

<table>
<thead>
<tr>
<th>Figure A</th>
<th>Figure B</th>
<th>Figure C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trace Method = Inside Offset = Both Sides</td>
<td>Trace Method = Endpoints Offset = One Side</td>
<td>Trace Method = Intersections Offset = One Side</td>
</tr>
</tbody>
</table>

Repeat symbol or text on an outline

| IntrSym | Repeat a symbol or text along an outline at a specified interval. |

You can place a symbol or a text string along an indicated outline at a specific distance. If you want to place a symbol select the Browse button to browse to the required symbol. This symbol will be placed along the outline. Select the Text option and type in a text string to be placed along the outline.

Use the Modify | SwapDir command to reverse the element direction if the chainage is not started from the correct side.
Repeat chainages around an outline

**IntTxCr** Repeat text, circles and points around an outline.

Use this command to add chainages – text, circles and points – around an outline. If the chainage is started from the incorrect side, use **SwapDir** to reverse the element direction.

**Interval parameters:**

The interval parameters determine the distance for the chainage, the circle diameters and chainage number. The pattern name of the circle and/or tick that is placed at each interval is determined by the point name : distance : chainage number.

**Pens:**

Select the circle and chainage circle pen colours.

**Decimals:**

The decimal setting is used when the distance values are written at the intervals.

**Intervals:**

You can insert text and/or a tick at the intervals.

**Example:**

Using the values in the **Modify | IntTxCr** dialogue sample above, the results are shown in the following illustration.
At the end of the execution of the command, Caddie will display resultant perimeter information listing the number of points, interval distance, total length and the remainder distance.

Cleaning up of wall crossings or intersections

When drawing walls, you don’t have to snap to elements, you can end the walls roughly and tidy the corners and intersections at a later stage.

Cleaning wall intersections

<table>
<thead>
<tr>
<th>X-Cros</th>
<th>Cleanup an X-intersection.</th>
</tr>
</thead>
<tbody>
<tr>
<td>OT Cros</td>
<td>Cleanup an open T-intersection.</td>
</tr>
<tr>
<td>CT Cros</td>
<td>Cleanup a closed T-intersection.</td>
</tr>
<tr>
<td>L Cros</td>
<td>Cleanup an L-intersection.</td>
</tr>
<tr>
<td>Y Cros</td>
<td>Cleanup a Y-intersection.</td>
</tr>
</tbody>
</table>
In the example these wall crossing commands are used to clean up the wall intersections.

**Inserting line break symbol**

**BrkObj** Draw a line break.

Select the pen number and linestyle for the line break on the dialogue. Enter the break height and width in mm and select the required scale for the symbol. Indicate the start and end point for the line break.

**Generating cylinder ends**

**CylBreak** Generate a symbol for a cylinder end on an elevation view.

Use this command to generate a symbol to fit on the end of a cylinder on side view. Indicate the terminal left and right points on the cylinder.
Generating a cross section symbol

| X-Pipe   | Generate a symbol for a cylinder end cross section. |

Use this command to generate a symbol to indicate a cylinder cross section. Indicate the circle. The cross section will be inserted and the left portion of the circle will be filled with the current hatch fill settings.

Reversing the direction of an Element

| SwapDir | Reverse the direction of an element. |

The direction of an element is determined by the order of the positions you indicate when drawing the element.

If you need to reverse the direction of an element, use the `SwapDir` command.

The direction of the elements will influence the start point of the chainages when using the `Modify` | `IntrSym` and `IntTxCr` commands. `SwapDir` will reverse the direction and the chainages will be inserted from the correct side.
CHAPTER 20

Editing Commands

- Change Global Pen and Line settings
- Re-scale Finishes
- Explode Entities Globally
- Change Entity Colours
- Move Objects between Planes
CHAPTER 20 - Global Editing Commands

Introduction to global editing commands

The Edit menu contains tools to make global changes to the drawing - this is useful for making changes to a number of objects simultaneously without having to select all these objects individually.

Exercise caution when using these commands as they make widespread changes to a drawing by a single execution.

Changing global pen and line settings

**Glob-PL** Change the pen colour and linestyle of objects in the drawing.

Use this command to change the pen types and linestyles of objects in the drawing. All the objects included by the selection will be changed.

![Global Pen & Linestyle Edit](image)

**Changing pen colour**

**Glob-P** Change the pen colour of objects in the drawing.

Use this command to change only the pen colour of objects in a drawing.

![Global Pen Edit](image)

Tick the **ByLayer** option on the dialogue if you want to include objects that are set to ByLayer.

**Changing linestyle**

**Glob-L** Change the linestyle of objects in a drawing.

Use this command to change only the linestyle of objects in a drawing.
Tick the **ByLayer** option on the dialogue if you want to include objects that are set to ByLayer.

### Changing fonts

**GlobFnt**  Globally change the fonts in a drawing.

Use this command to change the fonts of text objects in a drawing.

You have the option to substitute one font for another, or you can change all the text objects to the new font.

### Rescaling of finishes

**Glob-Sc**  Rescale selected finishes in a drawing.

When a drawing was rescaled without the finishes you can use this command to only rescale the finishes. Indicate which selection you want to apply this to and select text, leaders, dimension or hatch, or all of these. Enter the scaling factor for the rescale.

**TIP:**

Refer to the **TRANS | Rescale** command that allows you to rescale finishes while rescaling the drawing.
Exploding objects globally

**Glob-Ex**  Explode all objects in a drawing.

This command is used to explode the selected objects into editable objects.

As objects can be nested, more than one level of explode might be needed to convert objects into editable objects. For example, a block that contains a nested block (and this nested block contains 2D and 3D polylines) – means that the first explode will result in a block, the second explode will result in 2D and 3D polylines and only the third explode will turn these objects into editable lines and arcs.

If the **Explode to a new patterns** tick box is selected, the objects will be part of new patterns.

The number of objects exploded will be reported in the command line.

It is important to keep in mind that exploding objects in a drawing will cause loss of intelligence in the drawing.

By their nature referenced symbols are not stored in the drawing, they are referenced to a drawing that exists on the disk. Only the insertion point, scale and rotation are stored in the drawing. The referenced drawing needs to be edited before the main drawing can be updated with the new referenced symbol display.

It also means that if the drawing is sent to a third party, the referenced symbol needs to accompany the drawing. If you explode the referenced symbols in a drawing, they will form part of the drawing and will thus not be updated if the original referenced drawing on disk is updated.

Remapping of pens in a drawing

**MapPen**  Remap all pens in a drawing.

You can use this command to remap a set of pens in the drawing. If you need to remap certain pens regularly, you can save the pen mapping to a file for later retrieval.

Only the pens used in the current drawing is listed under **Old Pen**. Click on the **New Pen** to change the pen and select the new pen number from the **Select new pen** list.

You can specify entities for the remapping of pen by selecting lines and curves, text, 3D polylines, hatch, dimensions and leaders. Tick the option **Remap ByLayer Pens** to include pens that are set to ByLayer.
If you use the same remapping of pens on a regular basis, you have the option to save the map file for future usage.

Remapping line styles in a drawing

MapLay    Remap line styles in a drawing.

You can use this command to remap line styles in the drawing. If you need to remap lines styles regularly, you can save the line style mapping as a Saved Preset for later retrieval.

Remapping layers in a drawing

MapLay    Remap selected layers in a drawing.

You can use this command to remap a set of layers in the drawing. If you need to remap certain layers regularly, you can save the layer mapping as a Saved Preset for later retrieval.

Only the layers used in the current drawing is listed under Current Layer. Click on the New Layer to change the layer and select the new layer from the Select new layer list.
If you use the same remapping of layers on a regular basis, simply load the Saved Preset and remap the layers.

**Changing an object’s colour**

**ChgCol** Change the colour of an object to the current colour.

Use this command to change the colour of an object(s) to that of the currently selected colour.

**Cloning object properties**

**Clone** Clone specific properties of an object or objects.

The Clone command can be used to change the properties of a large number objects in the drawing in one action. The command will prompt you to indicate the object to acquire properties from. The dialogue displays the object’s properties.

The basic settings of object are displayed on the top of the dialogue. These are the pen colour, linestyle, linestyle scale, layer and thickness. If you want to clone specific properties of the object, click on the Advanced button to the left of the OK button to display the specific properties at the bottom of the dialogue.

The general properties will be applied to all the objects in the selection set. When you only select the specific properties, and no general properties, only similar objects to the indicated object will be affected.

The Clone command is handy to use when you want to change a large number of dimensions in the drawing.

**TIP:** Use the Text | ScpScl and ScpEdit command if you want to clone text in the drawing.
Chapter 20

Editing Commands

Moving objects across planes

Obj2FrPl  Move an object to the front plane.
Obj2BkPl  Move an object to the back plane.
ObForw    Move an object in front of another object.
ObBckw    Move an object behind another object.

Use the Obj2FrPl and Obj2BkPl commands to move objects to the front or back plane. Use the ObForw and ObBckw commands to move objects in front of behind other objects.

Please refer to the Edit | Reorder command for the reordering of a group of elements in the drawing.

Forcing objects onto the WCS

Obj>WCS   Force objects onto the WCS plane.

Use this command to force objects onto the WCS (Z=0) plane. Only use this command only when you are working in a 2D environment. The system will force all possible objects to the WCS. Objects contained within blocks cannot be forced to the WCS – you will have to edit the block and force the objects to WCS - remember to end the edit of the block.

Reordering of objects

Reorder   Change the order of objects in the drawing.

You can use this command to change the order of objects in a drawing. The order in which objects are drawn or inserted in the drawing determines the order of the objects. Only objects in the indicated selection set (scope, selected, marked, rectangle, polygon or pattern), will be affected.
Click on the object you want to reorder and click on **Forward** or **Back** buttons.

**Edit commands on the menu bar**

There are a number of edit commands that are easily accessible from the **Menu Bar** at the top of the screen.
### Selecting and unselecting objects

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select All</td>
<td>Select all the objects in the drawing. [Ctrl+A]</td>
</tr>
<tr>
<td>UnSelect All</td>
<td>Unselect all the objects in the drawing. [Ctrl+U]</td>
</tr>
<tr>
<td>Swap Selected/Unselected</td>
<td>Swap selected to unselected and unselected to selected.</td>
</tr>
<tr>
<td>Store Selected</td>
<td>Current selection will be stored.</td>
</tr>
<tr>
<td>Recall Selected</td>
<td>Recall the stored selection set.</td>
</tr>
</tbody>
</table>

Use these selection commands to accelerate the editing process of drawings.

### Copy and paste of selected objects

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Place selected object in the clipboard and remove it from the drawing. [Ctrl+X]</td>
</tr>
<tr>
<td>Copy</td>
<td>Place selected object in the clipboard. [Ctrl+C]</td>
</tr>
<tr>
<td>Paste</td>
<td>Insert the objects from the clipboard to the drawing. [Ctrl+V]</td>
</tr>
</tbody>
</table>

The Cut, Copy and Paste commands can be used to cut, copy and paste object in a specific drawing or between drawings. You can also copy the selected objects to other applications.

The **Settings | Environment Settings | Clipboard** controls the data that will be pasted to the other application. The image type can be Windows Metafile or Bitmap and you have the option to apply the current plotstyle.

### Mark and unmarking objects

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mark All</td>
<td>Mark all the objects in the drawing. [Shift+A]</td>
</tr>
<tr>
<td>UnMark All</td>
<td>Unmark all the objects in the drawing. [Shift+U]</td>
</tr>
<tr>
<td>Swap Marked/Unmarked</td>
<td>Swap marked to unmarked and unmarked to marked.</td>
</tr>
</tbody>
</table>

Use these marking commands to accelerate the editing process of drawings.

### Copy and paste of marked objects

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut Marked</td>
<td>Place selected object in the clipboard and remove it from the drawing. [Shift+X]</td>
</tr>
<tr>
<td>Copy Marked</td>
<td>Place selected object in the clipboard. [Shift+C]</td>
</tr>
</tbody>
</table>

The marked cut and copy, and paste commands can be used to cut, copy and paste marked object in a specific drawing or between drawings. These marked objects can also be pasted to other applications.

### Make a detailed copy of part of the drawing

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detail Copy</td>
<td>Create copy of a portion of the drawing.</td>
</tr>
</tbody>
</table>

This command allows you to clip objects when indicating the polygon outline. Indicate to origin and the new location for the detail copy. This copy can be placed on the original layers or active layer.
And scale during translation

&Scale  Scale objects during a transformation.

The &Scale command is used in conjunction with other transformation commands.

Refer to the Transformation Commands in Chapter 13.

And rotate during translation

&Rotate  Rotate objects during a transformation.

The &Rotate command is used in conjunction with other transformation commands.

Refer to the Transformation Commands in Chapter 13.

And mirror during translation

&Mirror  Mirror objects during a transformation.

The &Mirror command is used in conjunction with other transformation commands.

Refer to the Transformation Commands in Chapter 13.
CHAPTER 21

Files and Advanced Objects

Open Drawings
Save Drawings
Save only part of a Drawing
Merge Drawings
Backup and Restore Drawings
Print Drawings
Insert Images into Drawings
Move, Copy and Scale Images
CHAPTER 21 - File and Advanced Objects

Introduction to the file commands

The File menus contain commands enabling you to open drawings, symbols and reference symbols. You can also change the way in which these operations are executed whilst performing them, save drawings, parts of drawings, symbols, reference symbols, backup drawings, configure plot styles, print drawings, insert and manipulate pictures.

Creating a new drawing

New Create a new drawing. [Ctrl+N]

Use this command to create a new drawing. If you have selected to use a template, it will be opened.

Opening drawings

Open Open an existing drawing. [Ctrl+O]

Use this command to open an existing drawing.

- Look in: Browse to the location of the drawing you wish to open.
- File Name: The name of the drawing you wish to open.
- Files of Type: The type of file that you can open using Caddie is .drw (Caddie), .dwg (AutoCAD Drawing Files), .dxf (AutoCAD DXF), .drb (Caddie backup file).
- Properties - Options: You can open a drawing completely or retrieve a pattern, into a new window or merge with the current drawing or replace the current drawing.
### Chapter 21

**Files and Advanced Objects**

**Open the last drawing saved**

**LastDrw**  
Open the last drawing you saved.  

Caddie will automatically open the last drawing that was saved and zoom in or out to the same position you were in when the drawing was saved.

**Open and position a drawing into the current drawing**

**Locate**  
Open a drawing and locate it manually in an existing drawing.  

Use this command to merge a drawing with the current drawing in such a way that you can locate it in a new position in the current drawing.

**Open and place a drawing at the origin of the current drawing**

**Absolute**  
Open a drawing located in its original location.  

Use this command to merge a drawing with the current drawing in such a way that it will be positioned in the current drawing in the same location.

**Restoring backed up drawings**

**Restore**  
Restore a previously backed up drawing.  

Use this command to restore a previously backed up drawing. Select the drawing you want to restore. Bear in mind the naming convention that Caddie uses to name backup files. This will enable you to find the specific backup drawing you are looking for.

**Saving a drawing**

**Save**  
Save a drawing. [Ctrl+S]  

Use this command to save a drawing. The drawing will be saved with the existing name in the existing location. If you are working on a new drawing, Caddie will display the Save As dialogue to enable you to select a location and enter a name and file format.

**Saving a drawing with a new name**

**Save As**  
Save a drawing with a new name.  

Use this command to save a drawing with a new name. You can specify a different location, name and/or file format.

**Save as type:** The type of file that you can save is .drw (Caddie), .dwg (AutoCAD Drawing Files), .dxf (AutoCAD DXF), .ctp (Caddie template).

**Properties – Option:** You can specify the DWG version for the file. Please note the earlier DWG versions might not support only one sheet layout.

**Properties – Specific:** You can automatically increment the revision number, add a description, and enter the revision number.

**Saving the scope as a drawing**

**SveScp**  
Save the scope as a drawing.  

This command enables you to save only the current scope of a drawing.
Files and Advanced Objects

Chapter 21

Back up drawings

Backup
Create a backup of a drawing.

Use this command to make a backup of a drawing in between the pre-defined auto-backup period.

Insert a symbol

InsSym
Insert a symbol. [O]

Use this command to open a symbol drawing and insert into the drawing. Browse to the folder containing symbols. You can change the background colour of the symbols in the dialogue and the symbols drawings can be displayed as large icons, small icons, details or thumbnails.

There is an option to insert the drawings as a symbol, a referenced symbol or as a block.

You can enter a rotation angle for the symbol and you can tick the Align box to enable you to rotate the symbol to an existing object in the drawing.

There is an option to scale the symbol automatically or to keep the original scale.

You can select to insert the symbol on the active layer, or to retain the original layers or the symbol.

Save a symbol

SavSym
Save a symbol from the drawing.

Use this command to save a symbol for later retrieval in another drawing. Enter the origin (insertion point or handle) for the symbol and enter the symbol name and file format on the dialogue.

Insert a referenced symbol

InsRef
Insert a referenced symbol.

Use this command to open a referenced symbol and insert into the drawing. Browse to the folder containing referenced symbols. You can change the background colour of the referenced symbols in the dialogue and they can be displayed as large icons, small icons, details or thumbnails.

You can enter a rotation angle for the referenced symbol and you can tick the Align box to enable you to rotate the referenced symbol to an existing object in the drawing.

There is an option to scale it automatically or to keep the original scale.

You can select to insert the referenced symbol on the active layer, or to retain the original layers or the symbol.
A second dialogue will be displayed where you can change the referenced symbol properties. The file name and saved path is displayed for your information.

**Path type:** The path type can be **Absolute**, **Relative** or **None**. The path is crucial when moving projects on the server or on disk.

**Position:** You can enter X, Y and Z coordinates for the referenced symbol or you can tick the **Pick position** button to place it on the drawing.

**Scale:** You can enter different scale factors for the X, Y and Z axes or you can tick the **Proportional scaling** box to scale the referenced symbol proportional.

**Rotation:** You can enter a rotation angle, or if you ticked the **Align** box on the first dialogue, the indicated object angle is displayed and can be changed.

**Reference Type:** Select the required reference type, **Attachment** or **Overlay**.

**Manage referenced symbols**

| ManRef | Manage the referenced symbols in a drawing. |

Use this command to manage the referenced symbols in the drawing. A list of the referenced symbols is displayed. The status, size, date, type and path are displayed. Select the referenced symbol(s) you want to manage.

They can be inserted, removed, hidden, shown, merged, merged and exploded, replaced or reloaded. You can also print a list of the referenced symbols to an ASCII file.
Save a referenced symbol

**SavRef**  
Save a referenced symbol.

Use this command to save a referenced symbol for later retrieval in another drawing. Enter the origin (insertion point or handle) for the symbol and enter the symbol name and file format on the dialogue.

Explode a referenced symbol

**ExpRef**  
Explode a referenced symbol.

You can explode a single referenced symbol to a block or to objects or you can explode all the referenced symbols to individual blocks or objects.

Inserting pictures

**InsPic**  
Insert a picture into a drawing.

Use this command to insert pictures into a drawing. Caddie supports the .gif, .tiff, .bmp, .jpg, .jpeg, .png and .ecw image formats.

Managing pictures

**MngPic**  
Managing pictures in a drawing.

Use this command to manage existing pictures in a drawing. The dialogue display a list of the pictures in the drawing. The file name, visibility, size in kB, modification date, file type and path is displayed.

You can insert a new picture into the drawing, remove the selected picture from the drawing, hide the selected picture (the picture will not show on the screen or print), show the picture and replace it.

Convert a raster image to vectors

**Rs>Vec**  
Convert a raster image to vectors in insert it.

This command will convert a monochrome raster image to lines and insert it in the drawing at the indicated position.
Packing a drawing

Pack Drawing Pack the current drawing.

Use this command to pack the current drawing. Packing a drawing simply means to remove extraneous objects that are no longer in the drawing database.

![Unused Objects in Drawing](image)

You can select the items to pack or you can select **All Items**. You have an option to confirm the deletion. Unused block, layers and styles will be deleted from the drawing.

Exporting drawings

Export drawings to different formats.

You can export drawings in image format. These formats include BMP (Windows Bitmap), GIF (Graphics Interchange Format), JPG or JPEG (Joint Photographic Experts Group), PNG (Portable Network Graphics), TIF or TIFF (Tagged Information Format File) and DWF (Drawing Web Format).

File commands on the menu bar

Opening recently used drawings

Recent files Open a drawing from a list of recently opened drawings.

Caddie displays a list of recently opened drawings. Click on the one you want to open.

Save all the drawings currently open

Save All Drawings Save all the drawings currently open.

All the drawings currently open will be saved. If there are no name drawings open, the system will inform you that they are not saved.

Save a copy of the current drawing

Save Copy Save a copy of the current drawing.

Save a copy of the current drawing with a different name or path, without changing the current drawing’s name or path. Subsequent saves will use the original drawing name and path. If you use the **File | Save As...** command, the current drawing name or path changes.
### Save scope as a symbol

**Scope as Symbol**  
Save the scope as a symbol.

The current scope will be saved as a symbol for later retrieval. Indicate the origin of the symbol and enter a name and file format on the dialogue.

### Save the selection as a symbol

**Selection as Symbol**  
Save the selection as a symbol.

The current selection will be saved as a symbol for later retrieval. Indicate the origin of the symbol and enter a name and file format on the dialogue.

### Save the drawing as a PDF file

**Save as PDF**  
Save the drawing as a PDF (Portable Document Format) file.

You can save your drawing as a PDF file. You can specify the paper size and the plot style to use. Select the orientation, what to save and the extents of the drawing.

You can include the layer information so that the viewer can display the different layers.

### Add sheet information to the titlesheets drawing

**Add sheet to titlesheets drawing**  
Add a sheet to the titlesheets drawing.

You can add the current drawing’s sheet objects to your titlesheets drawing. Enter the new layout name for this new titlesheet layout.

### Create a SWF – ShockWave Flash video file

**Make SWF**  
Create a ShockWave Flash video file from images.

Add .jpg image files in the correct order. You can also add all the images in a specified folder. Manage the images by moving them up or down, or deleting them. Enter the frames per second. Browse to the correct folder where the .swf file is created.

### Emailing drawings

**Email**  
Email a drawing to a recipient.

Use this command to e-mail a drawing to a specified recipient.
Drawing to send:

You can email the current drawing or you can browse to a drawing on disk to email.

Email Header:

Enter the recipient address or click on the Caddie button to forward the drawing to Caddie support. Enter a subject for the email.

Parameters:

Tick the options zip the file(s) and specify a zip file name. Include all the relevant files for inclusions such as referenced symbols, pictures, plotstyles and Vio custom materials.

Close the drawing

Close

Close the current drawing.

This command will close the current drawing. If the drawing has not been saved, you will be prompted if you want to save it.

Checking and fixing drawings

Check and Fix Drawing

Check current drawing for errors and fix them.

Use this command to check the active drawing and fix any errors found. This command is useful to correct any errors when importing drawings. You can list these errors for reference.

Recovering damaged drawings

Recover Damaged Drawing

Recover a damaged drawing.

Use this command to recover a drawing that might be corrupted. Select the drawing on the dialogue to open, and enter a new name for the recovered drawing. The recovery is done in the background and you will be able to open the recovered file if the process was successful.
Importing drawings from Caddie 9 and earlier

Configure Settings for Older Caddie drawings

Use this command to setup how you want objects from Caddie 9 and earlier versions of Caddie to be imported into Caddie. In the dialogue you can setup the way in which drawings created in previous versions of Caddie are imported into Caddie.

Plotstyles and printing

Refer to Chapter 4 – Printing for a full explanation regarding printing and plotstyles.

Introduction to the advanced objects commands

The AdvObj menu contains commands enabling you to rotate, scale, mirror and clip pictures. It also contains the block and opaque commands.

Rotating a picture

RotPic  Rotate a picture.

Click on the picture you wish to rotate and enter the angle for the rotation. Tick the Dynamic box to rotate the image dynamically by indicating the first rotation point and the new location.

Mirroring a Picture

MirPic  Mirror a picture.

Indicate the picture you wish to mirror. You can mirror around a specific angle, an axis from an object or 2 indicated points. You can also select to mirror the original picture or you can mirror a copy of the picture.

Moving and copying a picture

MovPic  Move a picture.

CopPic  Copy a picture.

Use these commands to move or copy a picture respectively.

Scaling a picture

SclPic  Scale a picture in a drawing.

Use this command to scale a picture in a drawing. You can scale the picture proportional entering a scaling factor, by indicating different scale factors for the X, Y and Z axes or scaling the picture dynamically by indicating a scale base point and scaled size on the drawing.

Change the plane of a picture

PlnPic  Change the plane of a picture in a drawing.

Use this command to change the plane of a picture in a drawing. You can move the picture to the front plane, to the back plane, one plane forward or one plane backwards.
Clipping a picture

ClpPic Clip or Un-Clip a picture in a drawing.

Use this command to clip or un-clip a picture in a drawing. Indicate a clipping boundary and select Option End to close the trace. The picture outside the clipping boundary is hidden.

Merge images into one image

M-Imgs Merge images into one image.

Indicate the image on the screen you want to merge. Enter an image file name and location. The individual images in the drawing is not deleted or replaced.

You can delete these images and replace it with the single combined image.

Inserting a block into a drawing

InsBlck Insert a block into a drawing.

Use this command to insert a block into a drawing.

Enter the name of the block you want to insert or select it from the drop-down list. Note that Caddie will only list blocks that already exist in the current drawing. If you wish to retrieve a block from another drawing, click on the Browse button and locate the drawing from which you want to retrieve the block.

If you want to scale the block being inserted, type in the relevant scaling factors for the X, Y & Z components of the block. If you want to maintain proportionality for scaling, check the Proportional Scaling option. This will disable the Y and Z scale settings. The value in the X scale setting will be used to re-scale the block proportionately. Set the desired angle and direction of rotation. Set this value to 0 for no rotation.

Caddie prompts you to indicate the Position for block. Click at the desired position to insert the block. Click at more locations to insert more blocks and Select Option End to terminate the command.

Making a block

MkBlock Make a block in a drawing.

Use this command to make a block in a drawing.
Specify the required selection set for the making of a block.

Enter the desired name to refer to the block and pick the origin point on the drawing or enter the X, Y, Z coordinates for the block.

There are 3 options regarding the original objects: you can replace the original objects with the new block, delete the original objects in the block or retain original objects in the block. Enter a description for the block for easy reference.

**Edit and end the edit of a block**

**EdtBlck**  Edit a block.

**EEdBlck**  End the edit of a block

A block is unique and cannot be edited like other drawing objects. We need to edit the block before any changes or additions to the block can be made. Remember to end the edit of the block before continuing with your drawing.

**Merging objects into a block**

**mrgObj**  Merge existing objects into a block.

Use this command to merge objects into an existing block. **NOTE:** You must be editing a block for this command to work.

**Add text attributes**

**Attrib**  Add text attributes to your drawing.

Use this command add text attributes to the drawing. You can create, edit and move attributes, and you can also create text strings from attributes.

**Managing blocks**

**MnBlock**  Manage blocks in a drawing.

Use this command to manage blocks in a drawing.
Select the block to be managed or click on Pick to select it from the drawing. Enter a new block name if you wish to change the name of the block and add a description.

You can duplicate a block with options to insert, insert and edit it. You can also change the name and description or save the block to a new drawing.

**Replacing blocks**

**RplBck** Replace a block with another block.

Use this command to replace one block with another block. Enter or select the block you want to be replaced and indicate or enter the block to replace with.

**Schedule blocks**

**CtBlks** Display scheduled information about blocks in a drawing.

Use this command to display information about blocks in a drawing in tabular format.
**Selection:** Select which parts of the drawing to affect.

**Column Headers:** Enter the desired headings for the column headers of the table.

**Table settings:** Specify the row height and table width in pixels.

**Output:** The block information can be displayed as a table on the drawing, or exported to an ASCII (.txt) or an Excel (.xls) file.

**Blocks:** Caddie lists all blocks in the drawing. Select the blocks whose information is to be included in the table schedule and click on the > button to transfer them to the **Blocks to schedule** list.

**Layers to schedule:** Caddie will only schedule blocks on the selected layers.

**Clipping blocks and reference symbols**

<table>
<thead>
<tr>
<th>ClipRef</th>
<th>Clip a block or a reference symbol.</th>
</tr>
</thead>
</table>

Use this command to clip blocks and reference symbols. Define the area of a block or reference symbol to clip by using the polygon trace method. Once a block or reference symbol has been clipped, you can switch the clipping on or off. You can show or hide the clipping boundary of a block or reference symbol that has been clipped. You can also delete the clipping boundary.

**Creating opaque areas**

<table>
<thead>
<tr>
<th>Opaque</th>
<th>Create a region(s) of a drawing that is opaque.</th>
</tr>
</thead>
</table>

Use this command to create opaque regions in a drawing. You can insert opaques by tracing end points or intersections or by indicating points manually on the screen.
Change the ByLayer properties of objects in a block

**BLayer** Change the ByLayer properties of objects in a block.

You can use this command to edit the properties of any objects set to ByLayer. Select the desired layer for these objects from the dropdown and indicate the block to edit.

Advanced object commands on the draw and modify menus

**Inserting OLE Objects**

**Insert OLE Object** Insert an OLE object into a drawing.

Use this command to insert an OLE (Object Linked and Embedded) into a drawing.

Choose the object type you want to insert from the **Object Type** list.

Caddie will display a browse dialogue from which you can migrate to the folder where the object is located.

You can manipulate the OLE object like any other object by copying, moving, rotating and scaling the object.

**Editing OLE Objects**

**Edit OLE Object** Edit an OLE object into a drawing.

Use this command to edit an inserted OLE object. It will be opened in the software that is associated with the type of file.

**Creating Tables**

**Create Table** Create a table in a drawing.

Use this command to create a table of specified size in a drawing.
You can indicate a size for the table on the screen or you can specify the origin for the table. Specify the number of columns, column width, number of rows and row height. Tick the option if you want to include a title row.

Use the **Modify | Advanced Objects | Edit Table** command to insert and edit the table properties.

**Editing tables**

**Edit Table** Edit properties of a table in a drawing.

Use this command to edit the properties of a table in a drawing.

**Resizing tables**

**Resize Table** Resize a table in a drawing.

Use this command to resize a table in a drawing. Indicate the table to resize and click on a control points to move. The table will be resized accordingly.
Resizing a cell in a table

**Resize Cell**  Resize a cell within a table.

Use this command to resize the cell within a table in a drawing. Indicate the cell to resize and click on a control points to move. The cell in the table will be resized accordingly.

**Change table text**

**Change Table Text**  Change the text styles for the table text.

Use this command to change the text style as used in the table.

**Export table to Excel**

**Export Table to Excel**  Export the values in the table to Excel.

Use this command to export the content of a table to an Excel (.xls) file. Enter the location and file name.
CHAPTER 22

Working with Symbols and Referenced Symbols

- Insert Symbols into a Drawing
- Manipulate Symbols
- Distinguish Between Symbols and Referenced Symbols
- Create and Save Symbols
- Modify Symbols
CHAPTER 22 - Working with Symbols

Introduction to symbols

What is a symbol?

Symbols and Referenced Symbols are ordinary drawings. It is simply the method you use to read them into the drawing that defines them differently. When you insert symbols in a drawing, the elements from the symbol form part of the drawing. These elements can be change or edited.

When you insert referenced symbols in a drawing, the elements from the referenced symbols do not form part of the drawing. You cannot change or edit elements of inserted referenced symbols, you can only delete or explode it. If you need to change an element of a referenced symbol you must open the referenced symbol as a drawing, make the necessary changes and re-save the drawing as a referenced symbol. The next time you open the drawing containing these referenced symbols, they are loaded from disk and the updated referenced symbols will be displayed at all the positions in which they are used.

Creating a symbol

The easiest way to create symbols is to create them on their own. In other words, create them on a blank modelspace. Before creating a symbol, it is advised that you first carefully plan how the components of the symbol are related to each other. Take into account the following:

- Origin;
- Orientation;
- Pen types and Line styles;
- Patterns;
- Layers.

Origin: The insertion point of a symbol is determined by the location of the origin of the symbol. If you start the symbol on a blank modelspace, make sure the origin of the drawing (0,0) is also the origin of the symbol.

Orientation: Draw the symbol orientated in such a way that it is already correctly orientated when you insert it into a drawing.

Pen types and Line styles: Set the Pen types and Line styles to those you want them to be when inserting the symbol thus reducing the time taken to change these settings once you have inserted the symbol.

Patterns: If the symbol comprises several different types of components, you could then make use of patterns to separate these components from each other. You can then manipulate the various patterns individually or as groups. It is also a good idea to name these patterns especially if you ever need to count patterns in a drawing.

Layers: You can go one step further in controlling how symbols are created by drawing the various components of the symbol so that they span several layers. This can facilitate the placing of objects on pre-defined layers according to your layering standards.

Summary

If you make judicious use of these concepts when creating symbols, you will improve your productivity immensely in not constantly having to change aspects of symbols. Bear in mind that you can change some settings in the process of inserting symbols into a drawing. For example, you
can inset a symbol \&rotate it or \&scale it or \&mirror it or any combination of these commands.

If you do not have a clear modelspace when creating a symbol, you **MUST** change the scope of the drawing **BEFORE** saving the symbol otherwise all entities in the drawing will be saved as part of the symbol.

In order best to demonstrate the creation, saving and modifying of symbols, you will create an office boardroom suite comprising a table and eight chairs such as illustrated in the diagram below.

**Exercise**

In creating this symbol you will demonstrate the following features in Caddie.

- Drawing lines and rectangles.
- Creating patterns.
- Naming patterns.
- Create and activate new layers.
- Copying and moving objects.
- Rotating and mirroring objects.
- Saving symbols.

**Specifications**

For this example, you will draw the chair on layer 10 and the desk on layer 20. You will use pen type 2 and line style 1 (Continuous) with a thickness of 0.18mm. You will create the chair as one pattern and the desk as a separate pattern. The chair will be named **Office Chair Type 1** and the desk will be named **Office Desk Type 1**.

**Drawing the chair**

- Activate layer 10 by selecting it from the drop-down on the relevant toolbar.
- You want the chair to be a pattern on its own. To achieve this, press [I] on your keyboard to initialise a new pattern. Caddie displays the **New pattern created** message in the command history.
- Select Pen 2 and line style 1 with a width of 0.18mm.
- Select the **Rectang** command from the LINES menu. Draw a rectangle that is 450mm x 350mm.
- Click in the drawing to indicate the first corner of the chair.
- Instead of clicking at a point in the drawing to indicate the diagonally opposite corner,
type in the following: 450,-350 and press [ENTER]. This draws a rectangle 450mm wide by 350mm high.

- Delete the top line by using the Del-Elm command from the LINES menu.
- Use the Arc-2Pt command from the CURVE menu to draw the curve representing the back of the chair. Use a radius of 350mm and select the two endpoints across the element you have just deleted to draw the arc.
- Select the Name command from the PATTERN menu. Type in Office Chair Type 1 as the name of the pattern.

Saving the chair as a symbol

- For convenience you will place the origin of the symbol at the midpoint of the bottom line.
- Select the Save Other | Symbol option from the File menu.
- Caddie prompts you to Indicate the origin for the symbol. Change to Midpoint snap mode and click on this line.
- Browse to the folder where you want to save the symbol if not already there. Type Office Chair Type 1 as the name of the symbol and click on Save or press [ENTER].
- The symbol is now saved for later use.

Drawing the desk

- You want the desk to be a pattern on its own and that it should be located on layer 15. If you try to activate layer 15, you will see that there is currently no layer 15 created yet. This is because Caddie only creates the first 10 layers when you create a new drawing.
- Select the LaySet command from the SCOPE menu to access the layer definitions. Click repetitively on the Create button to create new layers until you see 15 layers.
- Name this layer Furniture - Bulk. Whilst you’re in the layer setup, rename layer 10 to Furniture - Light.
- Click on OK when done.
- Select layer 15 from the drop-down list of layers on the SCOPE toolbar to activate it.
- Press [I] to initialise a new pattern.
- Select Pen type 2 and line style 1 with a width of 0.18mm.
- Select the Rectang command from the LINES menu.
- Click in the drawing away from the chair to indicate the first corner of the desk.
- Type in 2500,-1400 and press [ENTER] to draw the desk.
- You will chamfer the corners of the desk. Select the ChamOfs command from the MODIFY menu and enter the value 75 for both offsets.
- Select the Both option from the popup menu to clean up both corners.
- Caddie prompts you to Indicate the first line. Pick any corner to begin with and click just inside one of the lines making up the corner.
- Caddie prompts you to Indicate the second line. Click just inside the other line forming the corner. Caddie draws the chamfer.
- Repeat steps 13 and 14 for all four corners and select Option End to end the command.
- Select the Offset command from the MODIFY menu. Set up the offset as follows:
  - Trace Method: Inside;
  - Pattern: Current;
  - Offset: One side;
  - No of offsets: 1;
Select pent type 2 and line style 1 (Continuous).
Click on OK when done.
Caddie prompts you to **Indicate the area to trace inside**. Click anywhere inside the desk.
Caddie prompts you to **Indicate the direction of offset**. Once again, click just inside the octagon. Caddie draws the offset.
Select the Auto command from the LINES menu and draw all the segments joining the corners of the octagon. Select Option End to end the command.
Select the Name command from the PATTERNS menu and type: **Office Desk Type 1** for the pattern name and click on OK.

**Saving the desk as a symbol:**
At this point you want to save the desk as a symbol. The problem, however, is that the chair is in the scope of the drawing so if you just went ahead and saved the desk as a symbol now, you would save the chair as part of the desk, which is not what you want. In order to save the desk only, you have to change the scope of the drawing so that only the desk remains visible on the screen.
Because both the chair and desk were created as individual patterns, you can use the Pattern command from the SCOPE menu to display only the desk.
Select this command from the SCOPE menu now and click on the desk. All other objects are now hidden from view and you can go ahead and save the desk as a symbol.
For convenience you will place the origin of the symbol at the centre of the desk.
Select the Save Other | Symbol option from the Main File Menu.
Caddie prompts you to **Indicate the origin for the symbol**. Change to Midway between two points snap mode and then immediately select the End Point snap mode.
Browse to the folder where you want to save the symbol if not already there. Type **Office Desk Type 1** as the name of the symbol and click on Save or press [ENTER].
The symbol is now saved for later use.
Select the All-Lay command from the SCOPE menu to re-display all objects in the drawing.
Using symbols in a drawing:
Let’s now have a look at how to use these symbols to build yet another larger composite symbol which will be a complete chair/desk office suite. To do this you will proceed as follows:
Retrieve the desk;
Retrieve a chair;
Copy and rotate the chairs around the desk;
Create an owner pattern and add the chairs as Child patterns.

**Retrieving the desk**
Close the drawing you are busy with where you created the two symbols and start a new drawing.
Set the modelspace to A4 at 1:100 scale.
Select the Open Other | Symbol command from the File menu. (You can also use the shortcut key [O]). Select the **Office Desk Type 1** symbol. Make sure that you set the following options:
Auto Scale;
Original Layer;
Angle: 0;
• Click on OK when done.
• If the symbol appears very small, just zoom into your drawing until it is a practical size. (Please note that you are NOT re-scaling the symbol but merely zooming into the drawing so as to render more visible.) Place the desk anywhere in the drawing and select Option End to end the command.
• Let’s say that you want to place the chairs around the desk at 75mm from the edge of the desk with the corner of the first chair at 240mm from the external edge of the desk.
• Select the Parlel command from the CnLIN menu, enter an offset of 75 and press [ENTER]. Click just above the top line of the desk and just to the left of the left-most vertical line as well as to the right of the right-most vertical line.
• Re-select the Parlel command and enter an offset of 365 and press [ENTER]. This time, click just INSIDE the left-most line of the desk to draw a construction line 365mm away from this line. This gives an intersection at the midpoint of the chair where you will place it.

Retrieving the chair

• Once again press O on the keyboard to open a symbol. Select the Office Chair Type 1. Symbol setting the same values as for the desk and click on OK when done.
• The chair will display virtually on the mouse pointer. Change to Construction Intersection snap mode and click at the intersection of the two construction lines you just drew. Select Option End to end the command.
• Since the chair is a pattern, you can now copy and repeat the chair twice to the right. You will use an offset of 785mm for this.
• Select the Trnlst command from the TRANS menu. Set the following options in the dialogue:
  • Selection: Pattern;
  • Translate Type: Repeat;
  • Number (Repeats): 2.
• Click on OK to accept the settings.
• Caddie prompts you to Pick the object to mark its patterns. Click anywhere on the chair.
• Caddie prompts you to Pick start of translation. This is the FROM position. Since you are going to copy the chair a known distance in a horizontal direction, you can just press [ENTER] here. This will snap to the current cursor location.
• Caddie prompts you to Indicate the position for the next copy. Type 785 and press the RIGHT arrow key to indicate a direction to the right. Caddie repeats the pattern twice at a distance of 785mm between each diametric point of the chairs.

Mirroring the chairs

• Select the Mirror command from the TRANS menu.
• Set the following options:
  • Selection: Rectangle;
  • Axis angle: 0 (This is the angle of the axis of symmetry); Mirror Type: Copy;
  • Un-check the Axis from entity as well as the Axis from 2 points options.
• Click on OK when done.
• Select all three chairs when prompted to draw a rectangle around the area.
• Caddie prompts you to Indicate the point about which to mirror. Change to Midpoint
Working with Symbols

Chapter 22

snap mode and click on any of the vertical sides of the desk. Caddie mirrors the three chairs placing them at the bottom of the desk.

Placing the two side chairs

- Draw a horizontal construction line through the midpoint of one of the sides of the desk. Use the ConX command from the CNLIN menu changing to Midpoint snap mode and clicking an any of the vertical sides of the desk.
- You will use the top left chair to copy it to the side. Select the Trnslt command from the TRANS menu.
- Selection: Pattern;
- Translation Type: Copy;
- Click on OK when done.
- Caddie prompts you to Pick object to mark its patterns. Click anywhere on the top left chair.
- Caddie prompts you to Pick start of translate. Change to midpoint snap mode and click on the bottom line of the chair.
- Select the &Rotate command from the TRANS menu.
- Set the rotation angle to 90 and set the rotation direction to Anticlockwise and click on OK.
- Change to Construction Intersection snap mode and click at the construction intersection just to the left of the desk.
- Select the &Mirror command from the TRANS menu. Set the mirror angle (Angle of axis of symmetry) to 90 and click on OK.
- Click on the construction intersection to the right of the desk to place a copy of the chair here.
- Delete all construction lines by selecting the AllCon command from the DELETE menu.

Creating an owner pattern

- Now that you have created the Office Suite, you will first merge the patterns so as to create a link between all the patterns. You will use the desk as the Owner pattern and the chairs will be Child patterns.
- Select the AddPat command from the PATTERN menu.
- Caddie prompts you to Select the owner pattern. Click anywhere on a line of the desk.
- Caddie prompts you to Select the pattern to add. In succession, click on all the chairs and select Option End to end the command.

Saving the Office Suite as a symbol

- You will now save the office suite as a symbol. Select the Symbol option from the File | Save Other main menu.
- Select a suitable origin for your symbol. Remember the this origin will determine where the symbol will be attached to the mouse when you retrieve it into a drawing so use a little common sense when setting the symbol origin so as to make your life easier.
- Type: Office Suite Type 1 for the name of the symbol and click on Save.

Why use symbols in a drawing?

Now that you have had a taste of how to create symbols and insert them into a drawing, you may wonder as to why the concept of symbols exists and how this applies in practical everyday situations. You might be tempted to say that it is just as easy to draw an object and simply copy it
working with Symbols

around in a drawing or to other drawings. This is true indeed but consider the following facts about symbols.

Once a symbol has been created and saved it can be used in any drawing any number of times.

When symbols are saved on a network drive other people that have access to this drive can make use of your symbol library.

Referenced symbols bring yet another dimension into the picture. These referenced symbols really come into their own when you need to make changes to recurring objects in a drawing on a global scale. By making use of referenced symbols, you only need to make changes to the original referenced symbol and the changes will automatically ripple throughout the drawing when you open it.

Large parts of a drawing can be **Built up** by inserting symbols of objects that are used on a regular basis. This has the effect of reducing drawing time and therefore improve your productivity.

**Differences between symbols and referenced symbols**

Lets now have a closer look and see what the differences between symbols and referenced symbols really are.

**Comparing Symbols and Referenced Symbols**

<table>
<thead>
<tr>
<th>Similarities</th>
<th>Symbol</th>
<th>RefSym</th>
</tr>
</thead>
<tbody>
<tr>
<td>Can be created on its own?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can be created with other entities present?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can be used in any Caddie drawing?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can be transformed in the process of retrieval?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can move and copy once located in a drawing?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can Rotate, Scale and Mirror once located in a drawing?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can print?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Affects print time?</td>
<td>Yes</td>
<td>No</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Differences</th>
<th>Symbol</th>
<th>RefSym</th>
</tr>
</thead>
<tbody>
<tr>
<td>Affects drawing size?</td>
<td>Very much</td>
<td>Very little</td>
</tr>
<tr>
<td>Can modify once located in a drawing?</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Can move from one layer to another?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Can change Pen and line settings?</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Can manipulate entities within objects retrieved?</td>
<td>Yes</td>
<td>No</td>
</tr>
</tbody>
</table>

**Why the difference between symbols and referenced symbols?**

You have already seen how to create symbols and insert them into drawings. As an exercise, we
will use the symbol of the office suite you created earlier. Clear your drawing and reset your modelspace to A4 at 1:100. You are going to insert the office suite, one as a symbol and one as a referenced symbol.

- Insert the Office Suite as a symbol using the File|Open Other|Insert Symbol menu option and place it to the left of the screen.
- Insert the Office Suite as a referenced symbol using the File|Open Other|Insert Referenced Symbol menu option and place it to the right of the screen alongside the symbol. Visually speaking, there is no difference at all between the symbol and referenced symbol. Let's perform some actions on these two and see where the differences really lie.
- Change the pen type to 3 and the line style to 2 (PDASHEDX2). Now, using the Edit command in the LINES menu. Click on any element on the symbol. What happened? Well, the element you clicked on should have changed both its colour and line style. Now click on any element on the referenced symbol. Caddie displays an error message indicating that No entity was found. Indeed? So what is it that you see on the screen making up the referenced symbol?
- Let's try something else. We'll move the end of a line. Select the MveLnE command from the LINES menu. Select any element on the symbol and move the end of the element to a new location. Now try to do the same thing on the referenced symbol. What happens? You get the same error message!
- Let's try something more radical and try to move a symbol. Select the Trnlst command from the TRANS menu. Set the Selection value to Pattern and the Translate type to Move. Click on the symbol to indicate the pattern to move. Remember that the Office Suite pattern is made up of a group of patterns so if you click on one of the chairs, Caddie will display a list of patterns in the group. Select the Office Desk Type 1 pattern so that you include the entire symbol. Now move it to a new location. Try the same on the referenced symbol and again you get the same error message. Re-select the Trnslt command but this time set the Selection option to Rectangle. Now select the referenced symbol and move it to a new location. This time - NO ERRORS!! Success at last!.

Interesting facts

What has actually happened in the above scenarios? A symbol is a group of objects in a drawing as if you had drawn them yourself each time. The only difference is that you instructed Caddie to Fetch an existing drawing and insert it at a specific location in a drawing. A referenced symbol, on the other hand, is not in your drawing at all! As its name implies, Caddie only REFERS to the symbol located on your local or network drive and displays it as a pictorial representation of the symbol. Put another way, Caddie projects an image of the symbol onto the drawing.

- The overhead slide projector comparison
- Imagine, if you will, an overhead slide projector projecting the image of slides onto a screen.
- The projector represents your computer;
- The slide rack in which the slides are stored represents your local or network drive;
- The slides themselves represent the symbols;
- The screen represents the drawing.
- When you switch on the light of the projector and a slide is placed on the display surface of the projector, an image of the slide is projected onto the screen. It is not the slide itself that is placed on the screen. If you wanted to make a change to the
slide, you **CANNOT** make such changes on the screen where the image of the slide is projected. You would have to make the changes on the slide itself.

- Another point is that the image of the slide onto the screen does not in any way add to the weight of the screen because the slide itself is not really there!

### The Caddie drawing

Let's see how this relates to a Caddie drawing.

#### Overhead Project vs Caddie Drawing

<table>
<thead>
<tr>
<th>Step</th>
<th>Overhead Projector</th>
<th>Caddie Refsym</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Create slide</td>
<td>Create refsym</td>
</tr>
<tr>
<td>2</td>
<td>Print slide</td>
<td>Save refsym</td>
</tr>
<tr>
<td>3</td>
<td>Pull screen down</td>
<td>Create new drawing</td>
</tr>
<tr>
<td>4</td>
<td>Switch light on</td>
<td>Insert refsym into drawing</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Step</th>
<th>Overhead Projector</th>
<th>Caddie Refsym</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Switch light off</td>
<td>Close drawing</td>
</tr>
<tr>
<td>2</td>
<td>Remove slide to working surface</td>
<td>Retrieve refsym AS A DRAWING</td>
</tr>
<tr>
<td>3</td>
<td>Modify slide</td>
<td>Modify refsym</td>
</tr>
<tr>
<td>4</td>
<td>Print modified slide</td>
<td>Save modified refsym AS A REFSYM</td>
</tr>
<tr>
<td>5</td>
<td>Switch light on</td>
<td>Retrieve drawing containing refsyms</td>
</tr>
<tr>
<td>6</td>
<td>Changes displayed on screen</td>
<td>Changes displayed in drawing</td>
</tr>
</tbody>
</table>

From the above comparisons a number of points are worth noting. A refsym is not part of a drawing and thus:

- It cannot be modified from within a drawing.
- It does **NOT** add to the size of a drawing.
- Individual entities forming part of the refsym cannot be modified, moved or have their pen and line settings altered.

### Advantages of symbols and referenced symbols

Apart from the reasons stated above, a very important factor to bear in mind when using referenced symbols is that they MUST be available to Caddie when opening a drawing containing refsyms. Whenever you open a drawing that contains refsyms, Caddie scans the location of the refsyms on your local or network drive. If it does not find the refsym at the expected location, you will not see any instances of these refsyms in the drawing. This can happen for a number of reasons.

- If the refsyms are located on a network drive, you may not be logged in to the network.
you may have lost the connection to the network. Contact your network administrator to restore your connection.

- The refsym may have been renamed. Rename the refsym to its original name when you saved it.
- The refsym may have been deleted. Use the backup copy of the refsym and copy it into the original location.

**Drawing exchange when using referenced symbols**

Since Caddie requires access to refsyms when opening drawings, you need to be extremely careful exchanging drawings with others either within your own organisation or when giving drawings to outside parties. There are two possible scenarios to handle such a situation.

- You must furnish the persons copies of all the refsyms in the drawing.
- Explode all the refsyms in the drawing then save it under a different name so as not to overwrite the original drawing. This drawing can then be given to another person without having to furnish the original refsyms. Exploding a refsym has the effect of converting it to actual elements in a drawing. The analogy with the overhead projector would be that you would physically take the overhead slide and glue it to the screen. It is now part of the screen and you can switch the overhead projector off and remove it and still, the slide would be part of the screen.

**Deciding when to use symbols versus referenced symbols**

There is a fine dividing line when deciding whether to use a symbol as opposed to a refsym. The best guide as to making the right decision is as follows. If you know or suspect that, during the course of developing a project, an object is likely to undergo design changes, use a refsym. This way, you only need to make one change to the original refsym and these changes will automatically be reflected in all drawing referring to this refsym.

- If a refsym is inserted more than once, editing the symbol will change all occurrences of this refsym throughout the entire drawing.
- If a refsym is used on more than one project, you must think about using different refsyms for each project since if the refsym changes for one then it will change wherever it is used in any other drawing.

**Managing referenced symbols**

To access this feature, select **File | Drawing Utilities** and then select the **Manage Referenced Symbols** option. Caddie displays the **Reference Symbol Manager** dialogue. Caddie lists all occurrences of referenced symbols in the drawing.

**Manage referenced symbol options**

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Insert</td>
<td>Insert a new reference symbol into a drawing. Caddie will prompt you to locate the reference symbol to insert into the drawing.</td>
</tr>
<tr>
<td>Remove</td>
<td>Caddie will remove ALL occurrences of the selected reference symbol.</td>
</tr>
<tr>
<td>Hide</td>
<td>Hide the selected reference symbol from view. Note that this will also cause all hidden referenced symbols NOT to print.</td>
</tr>
<tr>
<td>Show</td>
<td>Show the selected reference symbol.</td>
</tr>
</tbody>
</table>
### Chapter 22

#### Working with Symbols

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Merge</td>
<td>Merge the selected reference symbol into the drawing. Caddie will create a block out of the merged referenced symbol.</td>
</tr>
<tr>
<td>Merge/Explode</td>
<td>Merge the selected reference symbol into the drawing. Caddie will convert all entities of the referenced symbol to drawing elements.</td>
</tr>
<tr>
<td>Replace</td>
<td>Replace the selected referenced symbol with another referenced symbol. Caddie will prompt you to locate the referenced symbol to replace the selected one with.</td>
</tr>
<tr>
<td>Print list</td>
<td>Caddie will generate a text file with reference symbol information. This information will include the path and name of the reference symbol as well as its X, Y and Z coordinate location in the drawing.</td>
</tr>
<tr>
<td>Reload</td>
<td>Reload the selected reference symbol. This refreshes the selected symbol if changes have been made to it since loading the drawing.</td>
</tr>
</tbody>
</table>

### Conclusion

All-in-all you can see that making use of symbols and referenced symbols can dramatically improve your productivity. It is strongly suggested, therefore, that you get into the habit right from the start in making use of symbols.

**A motto to remember:** If you need to draw it more than once, save it as a symbol or referenced symbol!