
User Manual for

SavoyCAD

By Savoy Computing Services Ltd

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Contents

The SavoyCAD System	1
What it can and can't do	1
Manual layout.....	1
Notation conventions	1
Mouse conventions	1
Getting help.....	2
Context sensitive on-line help	2
Overview	3
General	3
Menus	3
Toolbars	3
To switch toolbars on and off (Toolbars dialog) ⁽¹⁴²⁰⁾	3
To move a toolbar	4
To resize the command window	4
Tooltips.....	5
Help.....	5
File menu	7
File handling.....	7
Opening vs Importing files	7
To start a new drawing.....	7
To open a drawing file.....	7
To save a drawing file	8
To save a copy of a file	9
To import a DXF, ATX or SVC file	9
To export a DXF file	9
Last used file list	10
Previewing your printed output (Print Preview screen).....	11
Printing (Plotter / Printer Settings dialog).....	12
Printer / Plotter Settings: Print Settings ⁽¹³¹⁰⁾	13
Printer / Plotter Settings: Title Block ⁽¹³²⁰⁾	15
Printer / Plotter Settings: Notes ⁽¹³⁰⁰⁾	16
Drawing & editing	17
Fundamentals of drawing & editing	17
Selecting / deselecting objects.....	17
Entering values	17
Layers	18
Blocks & Inserts	18
Attributes	18
Layer visibility options	18
Drawing.....	20
To draw a point	20
To draw a line.....	20
To draw a polyline.....	20
To draw a circle.....	21

To draw an arc.....	22
To draw an ellipse	22
To draw text.....	22
To automatically repeat the last command.....	23
To automatically use the last point drawn	23
To explode a polyline.....	23
Editing.....	24
To view the properties of an object (Properties dialog) ⁽¹⁴⁰⁰⁾	24
To view the coordinates of an object (List command)	24
To move an object	25
To copy an object	25
To rotate an object.....	26
To delete an object.....	26
Grip editing	27
GripTips	27
To move an object using grips	27
To resize / reshape an object	27
Blocks & Inserts	28
To create a block	28
To insert a block	28
To explode a block	29

Editing Properties 31

Layers list.....	31
To move an object to another layer	31
To make a layer invisible	31
To stop a layer from being printed.....	32
To make a layer non-editable	32
To change to colour of a layer	32
To change a layer's name	32
To create a new layer	32
Maintain Layers dialog ⁽¹⁰⁰⁰⁾	33
To create a new layer	34
To change the current layer.....	34
To delete a layer	34
Linetypes list.....	35
To change the linetype of an entity	35
To create a new linetype	35
Maintain Linetypes dialog ⁽¹¹⁰⁰⁾	36
To create a new linetype	37
To change the current linetype	37
To delete a linetype	37
Colours list.....	38
To change the colour of an object	38
Line Weights list	39
To change the line weight of an entity.....	39

Changing the view 41

To redraw all entities	41
To zoom in.....	41
To zoom out.....	41
To zoom to a specific area	42
To zoom to the drawing extents	42
To zoom to the drawing limits.....	42
To zoom to the tablet limits.....	42
To go back to the previous zoom window	42
To go forward to the next zoom window.....	42
To pan across the the drawing	43
To rotate the drawing.....	43

To reset any rotation	43
To set drawing limits	43
To clear drawing limits	43
Tools	45
Grid	45
To display a grid.....	45
To change the grid spacing.....	45
To change the grid colour	45
Snap grid.....	45
To snap to the snap grid	45
To change the snap grid origin or spacing.....	46
To change the snap grid colour	46
Orthogonal mode	46
To enable orthogonal mode	46
Object Snap modes (Object Snap Modes dialog) ⁽¹²⁰⁰⁾	47
More on object snap modes.....	47
Measuring distances (Distance command).....	49
Listing entity details (List command).....	50
Preferences (Preferences dialog)	51
Preferences: General ⁽¹⁵⁰⁰⁾	52
Preferences: Point Format ⁽¹⁵³⁰⁾	54
Preferences: Grid / Snap ⁽¹⁵²⁰⁾	55
Preferences: Display ⁽¹⁵¹⁰⁾	56
Preferences: Zoom ⁽¹⁵⁴⁰⁾	57
Digitising drawings	59
Setting up your digitising tablet	59
To set up your digitising tablet	59
Calibrating the tablet	59
To calibrate your tablet ⁽¹⁴¹⁰⁾	59
WinTab driver not found ⁽¹⁶⁰⁰⁾	61
Unable to access digitising tablet ⁽¹⁶¹⁰⁾	61
Calibration error	61
Seeing your reference points	62
Digitising data.....	62
To digitise a point.....	62
To digitise a line	62
To digitise a polyline	63
To digitise a circle	63
To digitise an arc.....	64
To digitise an ellipse	64
Tools to help you.....	64
Snap modes	64
Zoom	65
Glossary of Terms	67
Index	71

The SavoyCAD System

Synonymous with quality SavoyCAD provides a simple, fast and effective tool to create and edit drawings. It is not a replacement for more powerful systems; on the contrary, SavoyCAD's strength lies in its simplicity. SavoyCAD is currently only available with AutoTrack.

What it can and can't do

SavoyCAD can import and export DXF files making it suitable for editing drawings created in other CAD systems. It supports lines, arcs, polylines, lwpolylines, circles, text, blocks, inserts, etc. However, it is not able to interpret all the types of data that may be stored in DXF files and those elements that it cannot interpret will be lost. Therefore, if you are using the DXF transfer feature to link to another CAD system we recommend that you export only new and changed data.

SavoyCAD also includes built-in digitising tools allowing you to electronically "trace" printed drawings. This feature is available on all digitising tablets with WINTAB drivers.

Manual layout

Notation conventions

Throughout this manual key depressions are shown enclosed by angle brackets thus enter <F> means type the letter F and <OR> means type the letters O and R.

Menu options or commands are printed in a **bold** font.

Messages displayed by SavoyCAD are printed in *italics*.

Text that you should type and filenames are shown in a `courier` font.

All screen images and functionality descriptions are for SavoyCAD running on Windows XP. The screen image may vary slightly on other platforms. If functionality is different for other platforms the variations are noted.

Mouse conventions

Windows allows mouse button functionality to be reversed for left handed users. This manual refers to the default, right-handed, functionality. If

you have configured your mouse for left handed operation then for left mouse button read right mouse button and vice versa.

Getting help

If the user manual is not available try one of the alternatives.

Context sensitive on-line help

The extensive context sensitive on-line help mirrors the manual, but has the powerful windows search facilities. To get help click the **Help** button on any dialog or press **F1**.

Help is also available from the drop down menus:-

Select **Help, SavoyCAD Help** to get a structured breakdown of the help text.

Select **Help, About SavoyCAD** to check the program module versions.

Overview

General

The user interface follows conventions adopted by well designed Windows software. At the top of the main window are the toolbars and at the bottom is the command window. All can be displayed or not according to user preference.

SavoyCAD has a multi-document interface (MDI) so you can edit more than one file at a time allowing you to cut and paste data between documents.

Menus

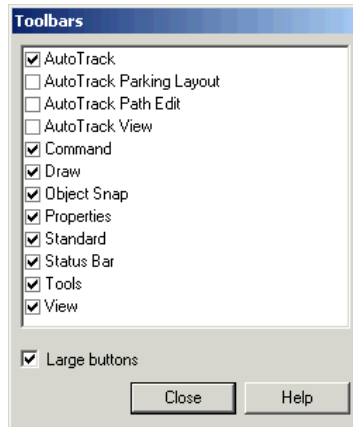
The drop down menus cannot be switched off. Other integrated Savoy programs may appear with self-contained drop down menus to the right of the CAD menus.

Toolbars

Many of the commands listed on the drop down menus are repeated on toolbars. By default these are displayed below the drop down menus.

To switch toolbars on and off (Toolbars dialog) (1420)

1. Select Toolbars from the View menu. The Toolbars dialog will appear.



Toolbars dialog

2. Tick the toolbars that you require and click **OK**.
3. Tick **Large buttons** to display oversized buttons.

To move a toolbar

1. The toolbars are by default docked at the top of the window but can be moved to any location or docked on any window edge. The toolbar positions are saved between sessions.
2. Move the cursor over the toolbar and, holding down the left mouse button, drag the toolbar to the required location.
3. Release the mouse button when the toolbar is in position.
4. To dock a toolbar simply drag it over the window edge that you want to dock it to.
5. To undock a toolbar simply drag it away from the window edge.

To resize the command window

Just like the other toolbars, the command window can be located anywhere on screen. However, unlike the others, it can also be resized. Note that if it is docked then only one dimension can be changed. The toolbar sizes are saved between sessions.

Resizing a docked command window

1. If you only want to change the height or width of a docked window then move the cursor over the free edge until it changes to a double ended arrow.
2. Now holding the left mouse button down drag the window edge to the required size.
3. Release the button when the window is the desired size.

Resizing an undocked command window

1. You can adjust both height and width of an undocked command window by moving the cursor over one of the corners of the command window until it changes to a diagonal double ended arrow.
2. Now holding the left mouse button down drag the corner to the required position.
3. Release the button when the window is the required size.

Tooltips

Reminders for what each button does appear as floating text adjacent to each button as you move the cursor over them. At the same time a reminder appears on the status line at the bottom of the CAD window.

Help

You can get help relating to a given screen by clicking on the help button, pressing **F1**. If you do not have a dialog box open you may also type ? on the command line to display help.

If there is no help specifically related to that screen you will see the contents.

File menu

File handling

SavoyCAD's native file format is the SVC file. However, it can also open or import DXF files from other CAD systems as well as ATX files from AutoTrack and WTP files from WinTrack.

Opening vs Importing files

Files may be opened or imported. They may appear to do the same thing but there is a difference:-

When you open a file, of any type, the contents will appear in a new CAD window.

When you import a file, of any type, the contents will be added to those currently being edited.

Note

SavoyCAD incorporates various features that apply specifically to imported data and allow you display such entities in a single chosen colour or prevent editing or selection of entities. You may therefore wish to import a DXF file rather than open it simply to benefit from these features.

To start a new drawing



New button

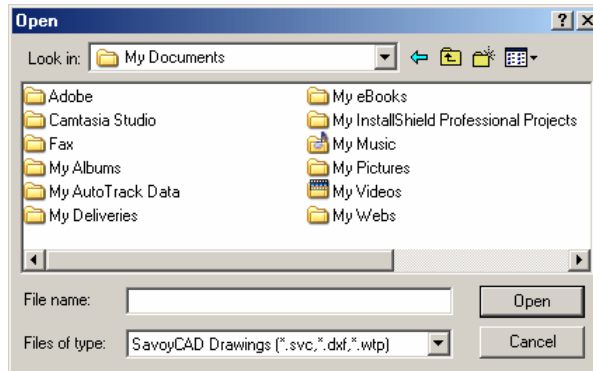
1. Select **New** from the File menu or click the **New** button on the Standard toolbar or type `NEW` on the command line. A new window will be created into which you may import data or draw new entities.

To open a drawing file



Open button

1. Select **Open** from the File menu or click the **Open** button on the Standard toolbar or type `OPEN` on the command line. The Open dialog will appear.



Open dialog

2. Use the file browser to find the file you require and click open to open it. The file will appear in the CAD window. If you already have a drawing loaded SavoyCAD will open a new window.

Remember

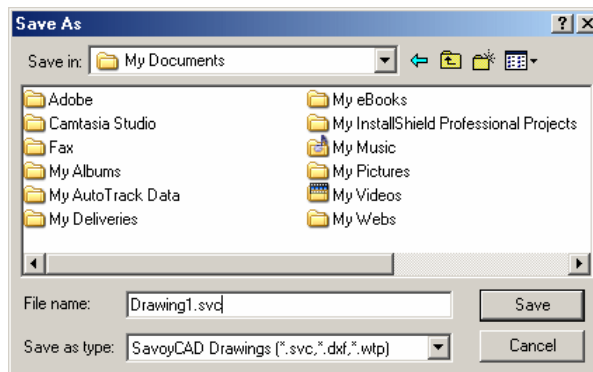
Opening a file, of any type, creates a new drawing window.

To save a drawing file



Save button

1. Select **Save** from the File menu or click the **Save** button on the Standard toolbar or type `SAVE` on the command line.
2. If the drawing was loaded from disk, or had previously been saved, the changes will be saved to the drawing file.
3. If the drawing was not loaded from disk, or had never been saved, the Save As dialog will appear.



Save As dialog

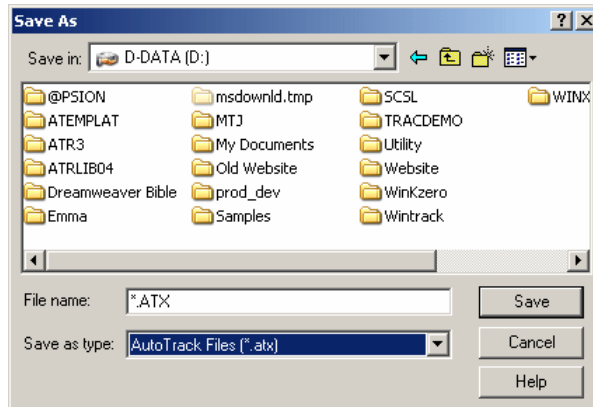
4. Browse to locate the required directory, enter the **File name** and click **Save**.

Warning

Save saves files in the new format to the original filename, including extension. So to avoid overwriting non-SavoyCAD files we recommend that you save the data as an SVC file before making any changes. Thereafter you can safely use Save which will save your data to your SVC format file.

To save a copy of a file

1. Select **Save As** from the File menu or type `SAVEAS` on the command line. The Save As dialog will appear.

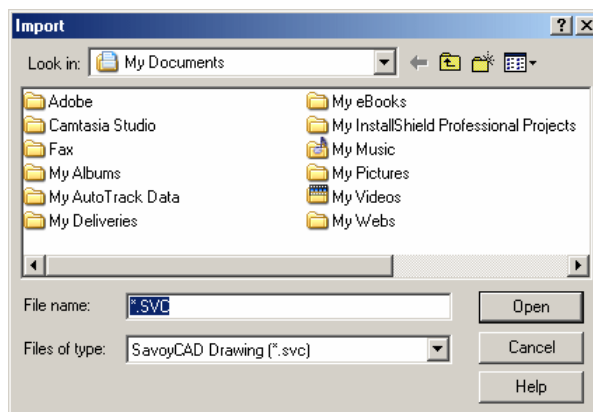


Save As dialog

2. Browse to locate the required directory, enter the **File name** and click **Save**.

To import a DXF, ATX or SVC file

1. Select **Import** from the File menu or type `IMPORT` on the command line. The Import dialog will appear.



Import dialog

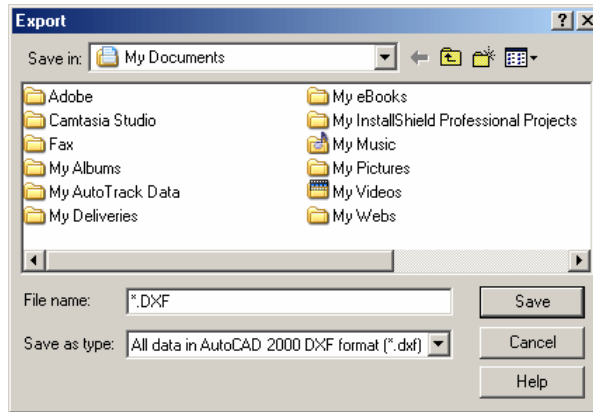
2. Select your required file type from the **Files of type** drop down list.
3. Browse to find your file and click **Open** to open it.

Remember

Importing a file, of any type, adds it to the current drawing.

To export a DXF file

1. Select **Export** from the File menu or type `EXPORT` on the command line. The Export dialog will appear.



Export dialog

2. Select your required file type from the **Save as type** drop down list bearing in mind the following:-

If your CAD system cannot read data in AutoCAD 2000 format then select **AutoCAD R14 DXF** format or AutoCAD R12 format. AutoCAD R12 format is the most portable.

If you loaded a background in DXF format (whether you used the **Open** or **Import** command) then we recommend that you export only changes and new data (i.e. all data either created or edited in SavoyCAD) by selecting the appropriate **New data in ... DXF** file type. The results can then be superimposed on your original drawing in your CAD system.

If you did not import any DXF data or if you want to export all the data then select the appropriate **All data in ... DXF** file type.

Note

The default file type will depend upon your Preferences settings. Be careful to check that you are saving in the correct format in case they have been changed by others.

3. Browse to locate the required directory, enter the **File name** and click **Save**.

Last used file list

1. At the bottom of the File menu is a list of the most recently loaded files (of any type). You can use these to quickly load recently used files.

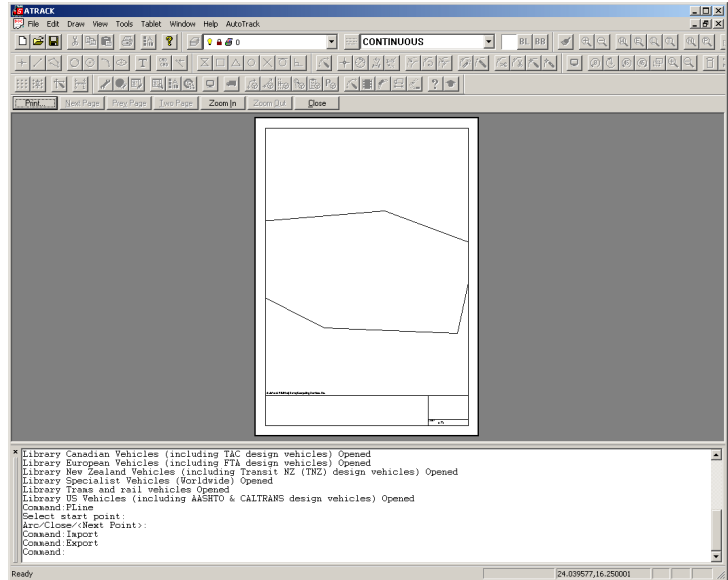
Note

Only opened or saved files appear in the last used file list and accessing a file by selecting it in this way is always treated as opening the file (rather than importing it).

Previewing your printed output (Print Preview screen)

It is normally wise to check how your drawing will appear in printed form especially if you are using expensive paper or ink.

1. Select **Print Preview** from the File menu. The screen will change to show a representation of the output plot.



Print Preview

2. Use the **Zoom In** and **Zoom Out** buttons to examine the output more closely.
3. Use the **Next Page** and **Previous Page** buttons to browse between multi-page output.
4. Click the **Two Page** button to display two pages at once.
5. If you are satisfied click the **Print** button to set print options and proceed.

Tip

If you cannot see anything in the print preview window make sure that you have set the drawing units correctly. Go to Tools, Preferences and select the correct units in the Scale / Units window.

Printing (Plotter / Printer Settings dialog)

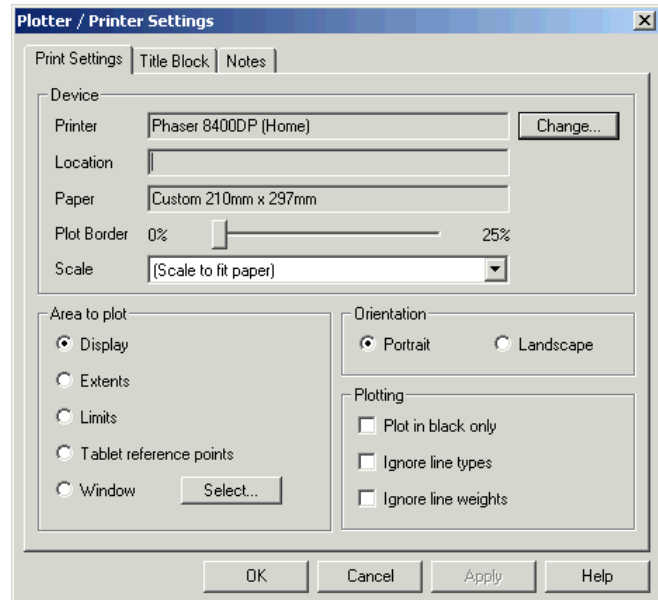


Print button

1. Select **Print** from the File menu or click the **Print** button on the Standard toolbar or type `PRINT` on the command line. The Plotter / Printer Settings dialog will appear.

Note

*This dialog is also accessible by clicking the **Print** button on the Print Preview screen.*

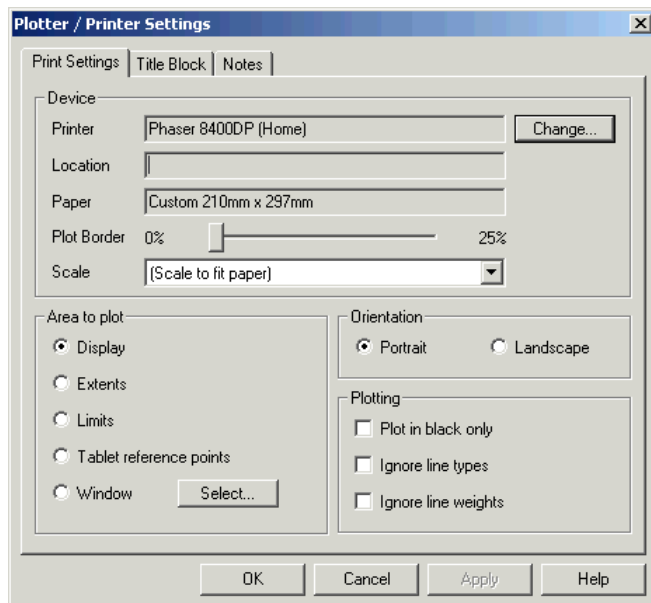


Plotter / Printer Settings dialog

2. Click the appropriate tab to edit or view the settings.
3. Make any changes that you want and click **OK** to print or **Cancel** to abort. See Plotter / Printer Settings dialog.

Printer / Plotter Settings: Print Settings ⁽¹³¹⁰⁾

1. Click on the **Print Settings** tab.



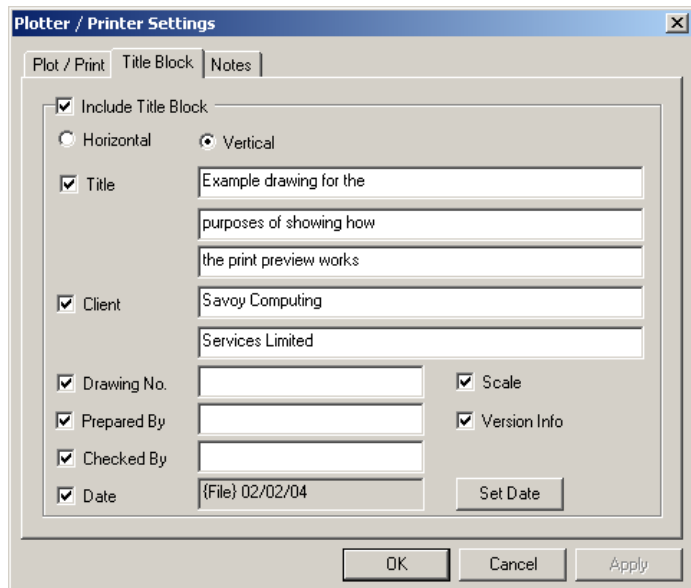
Plotter / Printer Settings dialog: Print Settings tab

4. The default **Printer**, together with its **Location** (if known) and **Paper** size are listed in the top window. Click the **Change** button if you want to change any of these.
 5. The **Plot Border** slider allows you to adjust the width of the printed border.
 6. By default SavoyCAD will scale the output to fit the paper size. However, this means that it is likely to be a non-standard engineering scale. If you want SavoyCAD to select the best standard scale for you then set the **Scale** to **(Auto Select)**. If you want to print to a specific scale then select it from the list.
2. Select the **Area to plot** as follows:-
- Display** to plot everything you can see in the CAD window.
- Extents** to plot the entire drawing.
- Limits** to plot everything within the drawing limits. Refer to [To set drawing limits](#) to learn how to set limits.
- Tablet area** to plot everything that falls within the extents of the three digitizing reference points. If you have not digitised data then this option will default to **Extents**.
- Window** to plot everything within a specific window. Click the **Select** button to specify the window. You may also set this window by selecting **Set Print Area** from the File drop down menu.- 3. Set **Orientation** to **Portrait** or **Landscape** as appropriate for your drawing.
- 7. Tick **Plot in black only** to print the drawing in black.

8. Tick **Ignore line types** to print all lines as continuous.
9. Tick **Ignore line weights** to print all lines as thin as possible.

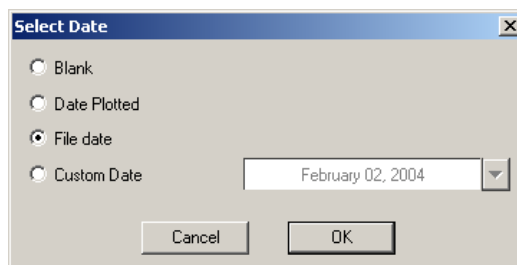
Printer / Plotter Settings: Title Block (1320)

1. Click on the **Title Block** tab.



Plotter / Printer Settings: Title Block tab

2. Tick **Include Title Block** if you want to plot a title block around the output.
10. Select the orientation of the title block. Note that this is different from the paper orientation. A **Horizontal** title block will be drawn across the bottom of the drawing sheet and a **Vertical** title block will be drawn up the right hand side of the sheet.
11. Tick the boxes that you want to appear in the title block. Note that a box will be drawn for those ticked even if there is no text defined. So, for example, you could print a **Checked By** box for someone to sign manually later.
3. By default today's date is entered in the **Date** field on new drawings. To change the date click the **Set Date** button. The Set Date dialog will appear.

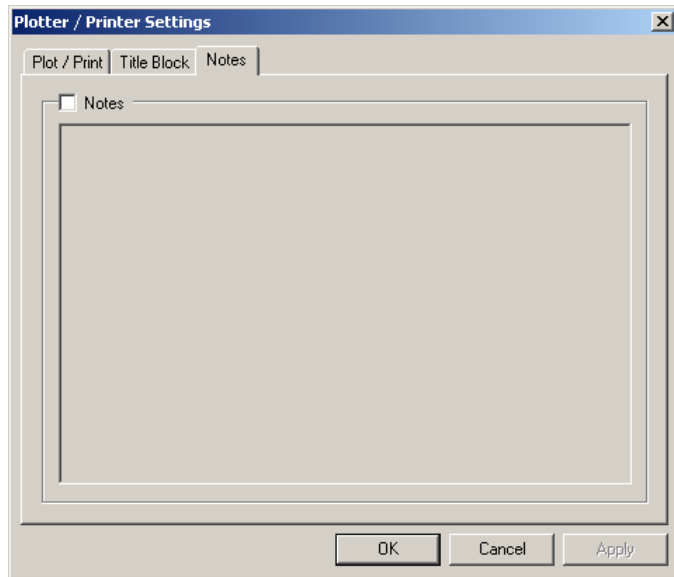


Select Date dialog

4. Select **Blank** to leave the Date field blank.
Select **Date Plotted** to print the date it was printed.
Select **File Date** to print the last changed date of the project file.
Select **Custom Date** if you want to specify a date to be printed. You will then need to select a date by clicking the arrow next to the date field.

Printer / Plotter Settings: Notes ⁽¹³⁰⁰⁾

1. Click on the **Notes** tab.



Plotter / Printer Settings: Notes tab

2. Tick **Notes** if you want to add a notes area to your drawing.
3. Enter the text of the notes in the large text window.

Drawing & editing

Fundamentals of drawing & editing

Selecting / deselecting objects

As you move the mouse over the drawing the object 'closest' to the cursor will be highlighted. If you click the left mouse button while the object is highlighted it will be selected. Selected objects have small squares or 'grips' at key points and are collectively referred to as a selection set. Objects may be added to the selection set by simply selecting them, i.e. no need to hold down the shift or control key.

All selected objects may be deselected by clicking on the background. The selection function is a toggle so an object may be deselected by re-selecting it.

All commands operate on objects already selected.

Entering values

In most cases values may be entered either graphically or numerically. Thus, for example, the radius of a circle may either be defined by selecting a point on screen using the mouse or by typing a value on the command line.

Where you can enter data numerically you will see a prompt on the command line. For example the following command line prompt will appear on starting to draw a line:-

```
Select start point:
```

This is asking for a coordinate which should be entered with commas separating the values, thus:-

```
Select start point: 1,2
```

The prompt will then change to:-

```
Select end point:
```

You may enter points as absolute or relative cartesian coordinates, or as absolute or relative polar coordinates. Absolute cartesian coordinates are entered thus:-

```
Select end point: 3,4
```

To make these relative simply add an @ symbol before the values, thus:

```
Select end point: @3,4
```

Polar coordinates (e.g. 3 units at an angle of 45 degrees) are entered thus:-

Select end point: 3<45

To make these relative, add the @ symbol before the values, thus:-

Select end point: @3<45

Layers

Entities may be drawn on different layers allowing related objects to be manipulated in groups. For example, an engineer might place drainage on one layer, electrical on another and landscaping on another. This would then allow the electrical and landscaping to be made invisible to show just drainage.

Blocks & Inserts

Entities may be grouped together into Blocks which may then be inserted into a drawing as entities in their own right. A block inserted into a drawing is called an Insert. Blocks may themselves contain other inserted blocks (or Inserts).

Blocks may also contain entities on multiple layers. When a block containing entities on multiple layers is inserted into a parent drawing the entities on these layers are placed on the layers with the same names in the parent drawing, if they exist. If any of the layers do not exist in the parent drawing they are created and the sub-entities placed on them.

Attributes

Entities have attributes including line type, line weight and colour which may be specified in three ways,

- Explicitly
- ByBlock
- ByLayer

If you want the attribute to remain the same regardless of how it is used then you should specify that attribute explicitly.

If you want the attribute to be inherited from the properties of the layer on which it is inserted then you should specify **ByLayer**.

If you want the attribute to be inherited from the parent block then you should specify **ByBlock**.

Layer visibility options

Layer visibility can be controlled in one of two ways; either by switching the layer off or by freezing the layer.

Freezing a layer makes all entities on the frozen layer invisible, as well as sub-entities on other layers if their parent is on the frozen layer.

Switching a layer off makes all entities on the frozen layer invisible but continues to display sub-entities if they exist on other non-frozen layers.

Note

Switching a layer off still requires SavoyCAD to consider the visibility of entities within inserted blocks, so even if they turn out to be invisible, this can result in slower redraws, on large drawings, than freezing the layer.

Example

Consider a block called `TREE` containing two layers, `LEAVES` and `BRANCHES`, inserted into a drawing on layer `TREES`. All layers are initially on and thawed.

- If the layer `TREES` is frozen then the trees will all disappear including all entities on the layers `LEAVES` and `BRANCHES`.
- If, instead, the layer `TREES` is switched off then the entities on layers `LEAVES` and `BRANCHES` will continue to be drawn.
- If, instead the layer `LEAVES` is frozen then the leaves will disappear but the branches will remain.
- Finally, if the layer `LEAVES` is switched off then the leaves will disappear but the branches will remain.

Drawing

SavoyCAD can be used to create or edit most simple entities. A couple of features are also available to improve usability. These allow you to automatically repeat the last command and to automatically use the last point drawn. These two options are described below.

To draw a point



Point button

1. Select **Point** from the Draw drop down menu or click **Point** on the Draw toolbar or type `POINT` on the command line. You will be prompted for the point.
2. Either select the **Point** using the mouse or type the coordinates on the command line. The point will appear in your chosen format (which may render it almost invisible!).

Note

You can change the appearance of point entities in the Preferences dialog.

To draw a line



Line button

1. Select **Line** from the Draw drop down menu or click **Line** on the Draw toolbar or type `LINE` on the command line. You will be prompted for the start point.
2. Either select the **Start point** using the mouse or type the coordinates on the command line. The point will be placed and a line displayed connecting it to the cursor position. You will be prompted for the end point.
3. Either select the **End point** using the mouse or type the coordinates on the command line. The line will appear selected.

To draw a polyline



Polyline button

1. Select **Polyline** from the Draw drop down menu or click **Polyline** on the Draw toolbar or type `PLINE` on the command line. You will be prompted for the start point.
2. Either select the **Start point** using the mouse or type the coordinates on the command line. The point will be placed and a line is displayed connecting it to the cursor position. You will be prompted to enter the next point or switch to arcs or close the polyline.
3. If you want to draw another line segment:-
Either select the **Next point** using the mouse or type the coordinates on the command line.

If you want to start an arc:-

Type `A` (for arc) on the command line. The line connecting the last point to the cursor will change to an arc and the command line prompt will change. Now, either select the end point using the mouse or type the coordinates on the

command line.

If you want to join the line back to the first point:-
Type **C** (for close) on the command line. A line will be drawn connecting the last point back to the start.

If you want to revert to drawing line segments:-
Type **L** (for line) on the command line. The arc connecting the last point to the cursor will change to a line and the command line prompt will change. Now, either select the **Next point** using the mouse or type the coordinates on the command line.

4. Repeat step 3 until you have drawn the entire polyline.
5. Click the right mouse button or hit Esc to cancel the polyline command. The polyline will appear selected.

To draw a circle

Circles can be drawn in two ways; by defining three points on the circle or by defining the centre and radius.

To draw a 3-Point Circle

1. Select **Circle: 3 Points** from the Draw drop down menu or click **3-Point Circle** on the Draw toolbar or type **CIRCLE3P** on the command line. You will be prompted for the first point.
2. Either select the **First point** using the mouse or type the coordinates on the command line. At this stage nothing will be visible. You will be prompted for the second point.
3. Either select the **Second point** using the mouse or type the coordinates on the command line. Now you should see a circle drawn through the two defined points which changes radius and position as you move the cursor. You will be prompted for the third point.
4. Either select the **Third point** using the mouse or type the coordinates on the command line. The full circle will now appear selected.



3-Point Circle button

To draw a Centre/Radius Circle

1. Select **Circle: Centre / Radius** from the Draw drop down menu or click **Centre/Radius Circle** on the Draw toolbar or type **CIRCLE** on the command line. You will be prompted for the centre of the circle.
2. Either select the **Centre** using the mouse or type the coordinates on the command line. A circle will appear whose radius changes as you move the mouse. You will be prompted for the radius of the circle.
3. Either select the **Radius** using the mouse or type a value on the command line. The circle will now appear selected.



Centre/Radius button



Arc button

To draw an arc

1. Select **Arc: 3 Points** from the Draw drop down menu or click **Arc** on the Draw toolbar or type `ARC` on the command line. You will be prompted for the start point.
2. Either select the **Start point** using the mouse or type the coordinates on the command line. At this stage nothing will be visible. You will be prompted for the middle point.
3. Either select the **Middle point** using the mouse or type the coordinates on the command line. An arc will appear whose shape changes as you move the mouse. You will be prompted for the end point.
4. Either select the **End point** using the mouse or type the coordinates on the command line. The completed arc will appear selected.



Ellipse button

To draw an ellipse

1. Select **Ellipse** from the Draw drop down menu or click **Ellipse** on the Draw toolbar or type `ELLIPSE` on the command line. You will be prompted for the centre of the ellipse.
2. Either select the **Centre** of the ellipse using the mouse or type the coordinates on the command line. At this stage nothing will be visible. You will be prompted for the major axis point.
3. Either select the **Major axis point** using the mouse or type the coordinates on the command line. At this point an ellipse will appear whose shape changes as you move the cursor. You will be prompted for the minor axis point.
4. Either select the **Minor axis point** using the mouse or type the coordinates on the command line. The completed ellipse will appear selected.

Note

Although the ends of the axes are requested in a particular order (major axis first), you may select a minor axis point that makes it the longer of the two.

To draw text



Text button

1. Select **Text** from the Draw drop down menu or click **Text** on the Draw toolbar or type `TEXT` on the command line. You will be prompted to enter the text you want to add.
2. Type the **Text** on the command line. You will be prompted for the Insertion point.
3. Either select the **Insertion point** using the mouse or type the coordinates on the command line. You will be prompted for the text height.
4. Either select the text **Height** using the mouse or type the height on the command line. You will be prompted for the rotation angle.
5. Either select the **Rotation** using the mouse or type the

angle on the command line. The text will appear.

Note

Angles are measured in degrees anti-clockwise from a zero datum facing East.

To automatically repeat the last command



Auto-repeat Last Draw Command button

1. Select **Repeat Last Draw Command** from the Draw drop down menu or depress the **Repeat Last Draw Command** button on the Draw toolbar or type `REPEAT` on the command line. With this mode enabled the same draw command is invoked every time you click the left mouse button.
2. To cancel simply hit **Esc**.
3. The option is a toggle so to disable the mode simply select **Repeat Last Draw Command** again.

To automatically use the last point drawn



Auto-use Last Point Drawn button

1. Select **Reuse Last Drawn Point** from the Draw drop down menu or depress the **Reuse Last Drawn Point** button on the Draw toolbar or type `REUSE` on the command line. With this mode enabled all the draw commands take the last point drawn as their first input no matter how long ago the last point was drawn.

Note

*If you enable both **Reuse Last Drawn Point** and **Repeat Last Draw Command**, the `Line` command is identical to drawing a *polyline*.*

2. To cancel simply hit **Esc**.
3. The option is a toggle, so to disable the mode simply select **Reuse Last Drawn Point** again.

To explode a polyline

1. Select **Explode** from the Draw drop down menu or type `EXPLODE` on the command line. Alternatively, select the polylines and then select **Explode**. You will be prompted to select the entities to be exploded.
2. Select the polylines and right click to confirm. The polylines will be exploded into their constituent line and arc entities.

Editing

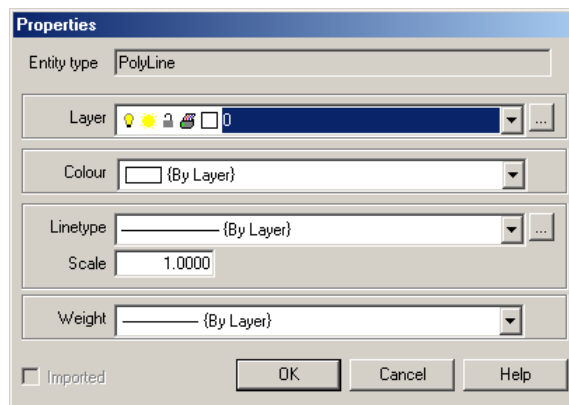
Any drawn entity can be moved, copied, rotated, resized or reshaped and deleted. Although, in the following descriptions we select the object(s) before the command you may select the command first if you prefer.

To view the properties of an object (Properties dialog) ⁽¹⁴⁰⁰⁾



Properties button

1. Select the object(s).
2. Select **Properties** from the Edit drop down menu or click the **Properties** button on the Object Properties toolbar or type `PROPERTIES` on the command line. The Properties dialog will appear.



Properties dialog

3. Change the layer that the object is on by clicking the **Layer** drop down list. If you edit one of the layer attributes or create a new layer then click the **Layers** adjacent to the drop down list. This will display the Maintain Layers dialog.
4. Select the entity colour from the **Colour** drop down list.
5. Select the entity linetype from the **Linetype** drop down list. If you want to edit a linetype or create a new one then click the **Linetypes** button adjacent to the drop down list.
6. Enter the linetype **Scale** to adjust the length of the linetype pattern.

To view the coordinates of an object (List command)

1. Select **List** from the Edit drop down menu or type `LIST` on the command line. You will be prompted to select the objects to be listed.
2. Select the objects using the mouse and right click when you have finished. Alternatively you may select the entities and then select the list command.
3. Detailed information for each selected entity will be displayed in the command window.

To move an object

1. Select **Move** from the Edit drop down menu or type `MOVE` on the command line. You will be prompted to select the objects to be moved.
2. Select the object(s) using the mouse and right click to confirm. You will be prompted for the base point.

Alternatively, you could select the objects first and then select the **Move** command. You will be prompted for the base point.

3. Select the **Base point** using the mouse or type the coordinates on the command line. You will be prompted for the destination point.
4. Select the **Destination point** using the mouse or type the coordinates on the command line. Remember it's relative to the base reference point which is not necessarily the same as the object location. The object will appear in its new location.

Note

The use of a base reference point enables you to move objects using known distances from other locations in the drawing.

5. An alternative way to move individual objects is to select it's green grip and drag it to the new location.

To copy an object

1. Select **Copy** from the Edit drop down menu or type `COPY` on the command line.
2. Select the object(s) using the mouse and right click to confirm. You will be prompted for the base point.

Alternatively, you could select the objects first and then select the **Copy** command. You will be prompted for the base point.

3. Select the **Base point** using the mouse or type the coordinates on the command line. You will be prompted for the destination point.
4. Select the **Destination point** using the mouse or type the coordinates on the command line. Remember it's relative to the base reference point which is not necessarily the same as the object location. The object will appear in its new location.

Note

The use of a base reference point enables you to copy objects using known distances from other locations in the drawing.

To rotate an object

1. Select **Rotate** from the Edit drop down menu or type `ROTATE` on the command line.
2. Select the object(s) using the mouse and right click to confirm. You will be prompted for the base point.

Alternatively, you could select the objects first and then select the Rotate command. You will be prompted for the base point.

3. Select the **Base point** using the mouse or type the coordinates on the command line. You will be prompted for the rotation angle.
4. Select the **Angle** using the mouse or type a value on the command line. The object(s) will appear in their new orientation.

To delete an object

1. Select **Delete** from the Edit drop down menu or type `DELETE` on the command line.
2. Select the object(s) to be deleted using the mouse and right click to confirm. The objects will be deleted immediately.

Alternatively you could select the objects first and then select the Delete command. The objects will be deleted immediately.

3. You can also delete selected objects by pressing the `<Delete>` or `` key on your keyboard.

Grip editing

Most entities can be edited individually using their editing grips. Every entity has a move grip, that may be used to move the object, and all basic entities have other editing grips that may be used to adjust the shape or size of the object.

GripTips

1. By default all grips have associated text showing the coordinates, the type of entity and the use of the grip. This text is called a GripTip. GripTips can be toggled on and off in the [Preferences: Display](#) ⁽¹⁵¹⁰⁾ dialog.

To move an object using grips

1. Select the object to be moved. One or more grips should appear on the object.

Note

If you cannot see them you may have the grip symbol size set too small or the grip colour set to an inappropriate colour. You may change either of these in Preferences, Display.

2. Select the move grip. This is coloured green by default but may have been changed. To check the move grip colour go to Preferences, Display. You will be prompted to move the object.
3. Select the new location for the object using the mouse. The object will appear in the new location.

To resize / reshape an object

1. Select the object to be resized or reshaped. One or more grips should appear on the object.

Note

If you cannot see them you may have the grip symbol size set too small or the grip colour set to an inappropriate colour. You may change either of these in Preferences, Display.

2. Select one of the editing grips to change the shape of the object. These are coloured blue by default but may have been changed. To check the editing grip colour go to Preference, Display.
The change made by moving the grip depends upon the object being edited. Thus:-
On a line it will move the end point.
On a polyline it will move the individual vertex.
On a circle it will change the radius.
On an arc it will change the radius and/or centre.
On the first defined axis of an ellipse it will change the orientation and/or size, and on the second defined axis it will change the major/minor axis proportion.

You cannot edit inserted blocks.

Blocks & Inserts

You may group entities together into Blocks. These blocks may be inserted into your drawing and used just like the basic drawing entities.

To create a block

1. First draw the entities that you want to include in the block.
2. Make sure that no entity selected.
3. Now select **Block** from the Draw drop down menu or type `BLOCK` on the command line. You will be prompted for the block name.
4. Type the **Name** on the command line.

Note

You may not use certain characters in the block name.

5. If a block with this name already exists you will be prompted for another name. Otherwise, you will be prompted for the origin point.
6. Either select the **Origin point** using the mouse or type the coordinates on the command line. This is the point that you will use to position the block when you insert it and the point about which you may rotate it. You will be prompted to select the objects to include in the block.
7. Select the objects for inclusion using the mouse in the normal way.
8. When you have finished selecting objects right click to end the command. The objects will disappear.

To insert a block

1. Select **Insert** from the Draw drop down menu or type `INSERT` on the command line.
2. If there are no blocks currently defined you will be warned. Otherwise, you will be asked to enter the name of the block you want to insert.
3. Type the **Name** on the command line. If you cannot remember the name type `?` to see a list of currently defined blocks and then type the name at the prompt. You will be prompted for the insertion point.
4. Either select the **Insertion point** using the mouse or type the coordinates on the command line. You will be prompted for the X Scale, the scale in the X direction.
5. Either select the **X Scale** using the mouse or type the value on the command line or press return to accept the default of 1. You will be prompted for the Y Scale.
6. Either select the **Y Scale** using the mouse or type the value on the command line or press return to accept the default of 1. You will be prompted for the rotation.

7. Either select the **Rotation** using the mouse or type the value on the command line. The block will appear.

Note

Rotation is measured in degrees anti-clockwise from East.

To explode a block

1. Select **Explode** from the Draw drop down menu or type `EXPLODE` on the command line. You will be prompted to select the object(s) to explode.
2. Select the objects using the mouse and right click to confirm.

Alternatively, you could select the objects first and then select the **Explode** command.

3. The objects will be exploded into their constituent entities.

Note

Any nested blocks inserted into the exploded block will need to be exploded.

Editing Properties

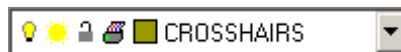
The Properties toolbar contains layer, line type, line weight and colour controls. It is shown here in three rows in the interests of clarity.



Properties toolbar

Layers list

The layers list shows the name and status of the current layer or, if an entity is selected, the layer and status of the selected entity.



Layers list

If you select several entities the Layers list will display the layer name if all the selected entities are on the same layer. Otherwise it will show nothing.

To move an object to another layer

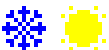
1. Simply select the object(s) to be moved.
2. Select the layer you want to move them to from the Layers drop down list. All the selected entities will be updated immediately.

To make a layer invisible

1. Make sure that no objects are selected and then select the layer that you want to make invisible.
2. Now you have two options, either switch the layer off, or freeze it.
To switch the layer off, click the on/off icon so that it goes out, and vice versa.
To freeze the layer, click the freeze/thaw icon so that it changes to a snowflake, and vice versa.
Refer to [Layer visibility options](#) for an explanation of the



On / Off icons

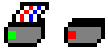


Freeze / Thaw icons

difference.

3. In AutoCAD, frozen layers are not regenerated whereas layers that are switched off are regenerated but not drawn.

To stop a layer from being printed



Plot / No Plot icons

1. Make sure that no objects are selected and then select the layer that you want to make view only.
2. To make the layer view only, click the plot/noplot icon so that the green light changes to red, and vice versa.

To make a layer non-editable



Lock / Unlock icons

1. Make sure that no objects are selected and then select the layer that you want to make non-editable.
2. To lock the layer, click the lock/unlock icon so that it appears closed, and vice versa.

To change to colour of a layer

1. Make sure that no objects are selected and then select the layer whose colour you want to change.
2. Now, simply click on the colour swatch and select the new colour from the palette displayed.

Note

Objects changing the colour of a layer will only affect those objects with colour set to ByLayer.

To change a layer's name



Layers button

1. You can only change a layer's name in the Maintain Layers dialog. Click on the **Layers** button. See [Maintain Layers dialog](#).

To create a new layer



Layers button

1. Select **Layers** from the View menu or click the **Layers** button on the Properties toolbar or type `LAYERS` on the command line. The Maintain Layers dialog will appear. Refer to [Maintain Layers dialog](#).

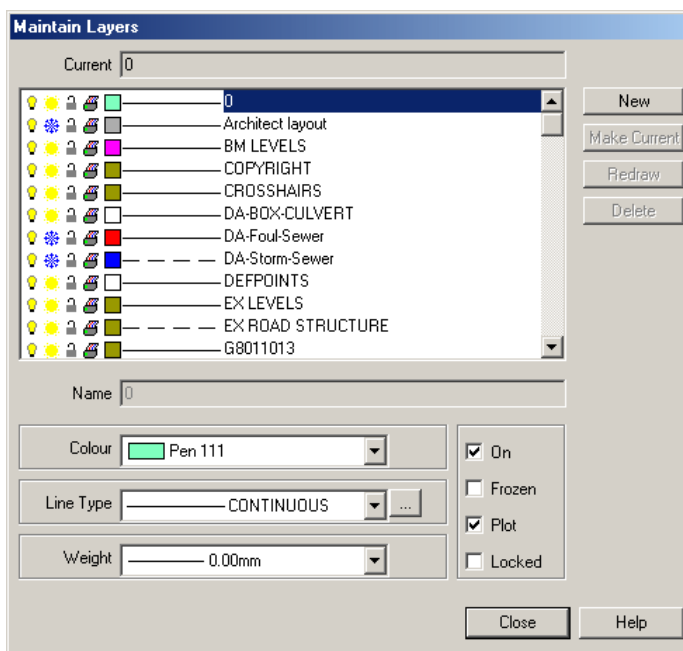
Maintain Layers dialog (1000)

Layers allow you to group entities together and control attributes such as visibility and colour. Each layer has attributes associated with it that are used if you select ByLayer for the entity.



Layers button

1. Select **Layers** from the View menu or click the **Layers** button on the Properties toolbar or type `LAYERS` on the command line. The Maintain Layers dialog will appear.



Maintain Layers dialog

2. The main window lists the layers that currently exist in the drawing. The **Current** layer, the one on which new entities will be drawn by default, is shown in the text box at the top.
3. Highlight the layer that you want to edit.
4. Edit the **Name** as necessary.
5. Select the pen colour from the drop down list. If you want to use a colour that is not listed then select **Custom** and click in the colour swatch to select the colour.
6. Select the required linetype from the drop down list. If you want to create a new linetype then click the **Linetype** button to display the Linetypes dialog. Once the new linetype has been created, select it from the drop down list.
7. Tick **On** to switch the selected layer on.
8. Tick **Frozen** to freeze the selected layer.
9. Tick **Plot** to ensure that all entities on the selected layer plot.
10. Tick **Locked** to prevent accidental changes to entities on the current layer.



Layers button

To create a new layer

1. Select **Layers** from the menu or click the **Layers** button on the Properties toolbar or type `LAYERS` on the command line to display the Maintain Layers dialog.
2. Click the **New** button.
3. A new layer with the default name **NewLayer** will be created.
4. You may edit the name and change the associated colour and linetype using the edit boxes below the list.

To change the current layer



Layers button

1. Select **Layers** from the **Properties** menu or click the **Layers** button on the **View** toolbar or type `LAYERS` on the command line to display the Maintain Layers dialog.
2. Highlight the layer you want to make current and click the **Make Current** button. The name of the layer will appear in the top edit box.

To delete a layer



Layers button

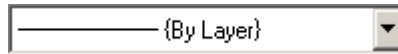
1. Select **Layers** from the **Properties** menu or click the **Layers** button on the **View** toolbar or type `LAYERS` on the command line to display the Maintain Layers dialog.
2. Highlight the layer you want to delete and click the **Delete** button.

Note

You cannot delete the current layer or layers that still contain entities.

Linetypes list

The Lintypes list shows the current lintype or, if an entity is selected, the colour of that entity.



Lintypes list

If the lintype is set to **By Layer** then the indicator will show the lintype of the layer. Likewise if it is set to **By Block** it will show the lintype of the parent block.

If you select several entities the Lintypes list will display the lintype if all the selected entities are set to the same lintype. Otherwise it will show nothing.

Note

*If selected entities are set to **By Layer** then even if the lintypes of the layers are different the lintype will still appear as **By Layer**. However, the lintype indicator will show the lintype of the first entity selected.*

To change the lintype of an entity

1. Select the object(s) whose lintype you want to change.
2. Select the new lintype from the Lintypes list. All the selected entities will be updated immediately.

Select a pen colour from the list to set the lintype explicitly to one of the preset values.

Select **By Layer** if you want to be able to control the lintype of the object by changing the lintype of the layer.

Select **By Block** if you want to be able to change the lintype of the object by changing the lintype of the parent block.

To create a new lintype

1. Select **Lintypes** from the View menu or click the **Lintypes** button on the Properties toolbar or type `LINETYPES` on the command line. The Maintain Lintypes dialog will appear. Refer to [Maintain Lintypes dialog](#).



Lintypes button

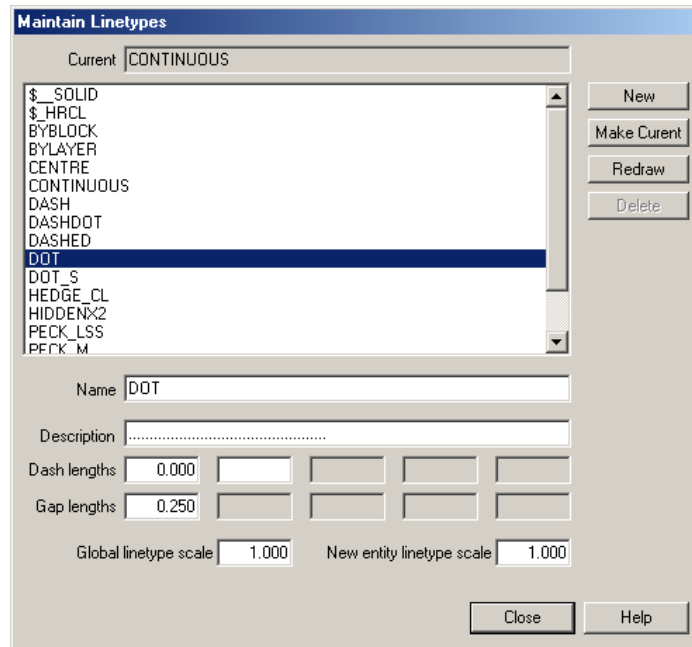
Maintain Linetypes dialog (1100)

Linetypes can be created or edited in the Linetypes dialog.



Linetypes button

1. Select **Linetypes** from the View menu or click the **Linetypes** button on the Properties toolbar or type `LINETYPES` on the command line. The Maintain Linetypes dialog will appear.



Maintain Linetypes dialog

2. The layers that exist in the current drawing are listed in the main window. Above that the **Current** linetype is displayed. New entities are drawn in the current linetype.
3. The **Global Linetype Scale** allows you to adjust the overall length of the pattern of all linetypes in the current drawing. You might use this to scale all your linetypes from millimetres to metres.
4. The **Entity Linetype Scale** allows you to adjust the linetype scale for each entity.
5. Below the main window are listed the details of the highlighted linetype, i.e. the one under the cursor.
6. The **Name** is used to select the linetype and appears in the linetype drop down list.
7. The **Description** is free format and users may enter either a text description or, as is often the case, a visual reminder using standard keyboard characters.
8. Below these fields are fields for the **Dash Lengths** and **Gap Lengths**. The line format starts with the first dash length followed by the first gap length, then the second dash length, and so on.



Linetypes button

To create a new linetype

1. Select **Linetypes** from the View menu or click the **Linetypes** button on the Properties toolbar or type `LINETYPES` on the command line. The Maintain Linetypes dialog will appear.
2. Click the **New** button. A new linetype with a default name `NewLineType` will be added to the list.
3. Edit the **Name** to reflect the style of the line. Try to make this name self explanatory and not too long.
4. You may enter a **Description** if you wish. This may either be a simple text description or a more visual description using keyboard characters, thus:-
 — · · — · · —
5. Enter the first **Dash length** in drawing units followed by the first **Gap length**. With just one dash and one gap of the same lengths the line will appear as a simple dashed line similar to this:-
 — — — —
6. Enter the second **Dash length** and **Gap length** to create a more complex linetype such as this:-
 — — — — —
7. Enter further values to generate even more complex linetypes but make sure that you always end on a gap length.



Linetypes button

To change the current linetype

1. Select **Linetypes** from the **Properties** menu or click the **Linetypes** button on the **View** toolbar or type `LINETYPES` on the command line to display the Maintain Linetypes dialog.
2. Highlight the linetype you want to make current and click the **Make Current** button. The name of the linetype will appear in the top edit box.



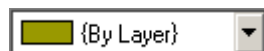
Linetypes button

To delete a linetype

1. Select **Linetypes** from the **Properties** menu or click the **Linetypes** button on the **View** toolbar or type `LINETYPES` on the command line to display the Maintain Linetypes dialog.
2. Highlight the linetype you want to delete and click the **Delete** button.

Colours list

The colours list shows the current colour, or, if an entity is selected, the colour of that entity.



Colours list

If the colour is set to **By Layer** then the colour swatch will indicate the colour of the layer. Likewise if it is set to **By Block** it will indicate the colour of the parent block.

If you select several entities the Colours list will display the colour if all the selected entities are set to the same colour. Otherwise it will show nothing.

Note

*If selected entities are set to **By Layer** then even if the colours of the layers are different the colour will still appear as **By Layer**. However, the colour swatch will indicate the colour of the first entity selected.*

To change the colour of an object

1. Select the object(s) whose linetype you want to change.
2. Select the new linetype from the Linetypes list. All the selected entities will be updated immediately.

Select a pen colour from the list to set the colour explicitly to one of the 255 preset colours.

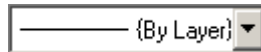
Select **Custom** if you want to use a colour that is not listed. Then click on the colour swatch to select the colour.

Select **By Layer** if you want to be able to control the colour of the object by changing the colour of the layer.

Select **By Block** if you want to be able to change the colour of the object by changing the colour of the parent block.

Line Weights list

The Line Weights list shows the current colour, or, if an entity is selected, the line weight of that entity.



Line Weights list

If the line weight is set to **By Layer** then the colour swatch will indicate the colour of the layer. Likewise if it is set to **By Block** it will indicate the colour of the parent block.

If you select several entities the Line Weights list will display the line weight if all the selected entities are set to the same line weight. Otherwise it will show nothing.

Note

*If selected entities are set to **By Layer** then even if the line weights of the layers are different the colour will still appear as **By Layer**. However, the line weight indicator will show the weight of the first entity selected.*

To change the line weight of an entity

1. Select the object(s) whose line weight you want to change.
2. Select the new line weight from the Line Weights list. All the selected entities will be updated immediately.

Select a line weight from the list to set the line weight explicitly to one of the preset values.

Select **By Layer** if you want to be able to control the line weight of the object by changing the line weight of the layer.

Select **By Block** if you want to be able to change the line weight of the object by changing the line weight of the parent block.

Changing the view

The view of the drawing can be rotated, magnified and panned to make it easy to make changes. All the view adjustments are available on the View drop down menu or on the View toolbar.



View toolbar

To redraw all entities



Redraw button

1. Select **Redraw** from the View drop down menu or click the **Redraw** button on the View toolbar or type `REDRAW` on the command line.

To zoom in



Zoom In button

1. Select **Zoom, In** from the View drop down menu or click the **Zoom In** button on the View toolbar or type `ZOOM IN` (or the abbreviation `z i`) on the command line. A window with the same proportions as the CAD window will appear. This represents what you will see with that level of zoom. You will be prompted for the centre point.
2. Use the scroll wheel on your mouse (if it has one) to adjust the size of the window and move the cursor to adjust the location of the zoom window.
3. Select the **Centre point** using the mouse or type the coordinates on the command line. The CAD window will be redraw to show the new display extents.

To zoom out



Zoom Out button

1. Select **Zoom, Out** from the View drop down menu or click the **Zoom Out** button on the View toolbar or type `ZOOM OUT` (or the abbreviation `z o`) on the command line. The zoom level will halve, i.e. the visible length and width displayed will double.



Zoom Window button

To zoom to a specific area

1. Select **Zoom, Window** from the View drop down menu or click the **Zoom Window** button on the View toolbar or type `ZOOM WINDOW` (or the abbreviation `Z W`) on the command line. You will be prompted for the first corner point.
2. Select the **First corner point** using the mouse or type the coordinates on the command line. A rectangular dotted window will appear whose corners are the point just specified and the cursor. This represents what you will see with that level of zoom. You will be prompted for the second corner point.
3. Select the **Second corner point** using the mouse or type the coordinates on the command line. The window will be redrawn to show the new display extents.



Zoom Extents button

To zoom to the drawing extents

1. Select **Zoom, Extents** from the View drop down menu or click the **Zoom Extents** button on the View toolbar or type `ZOOM EXTENTS` (or the abbreviation `Z E`) on the command line.



Zoom Limits button

To zoom to the drawing limits

1. Select **Zoom, Limits** from the View drop down menu or click the **Zoom Limits** button on the View toolbar or type `ZOOM LIMITS` (or the abbreviation `Z L`) on the command line.



Zoom Tablet button

To zoom to the tablet limits

1. Select **Zoom, Tablet** from the View drop down menu or click the **Zoom Tablet** button on the View toolbar or type `ZOOM TABLET` (or the abbreviation `Z T`) on the command line.



Zoom Previous button

To go back to the previous zoom window

1. Select **Zoom, Previous** from the View drop down menu or click the **Zoom Previous** button on the View toolbar or type `ZOOM PREVIOUS` (or the abbreviation `Z P`) on the command line. The previous view will be redisplayed if there is one. The command can be used repeatedly.



Zoom Next button

To go forward to the next zoom window

1. Select **Zoom, Next** from the View drop down menu or click the **Zoom Next** button on the View toolbar or type `ZOOM NEXT` (or the abbreviation `Z N`) on the command line. The next view will be redisplayed if there is one. The command can be used repeatedly.



Pan button

To pan across the the drawing

1. Select **Pan** from the View drop down menu or click the **Pan** button on the View toolbar or type `PAN` on the command line. You will be prompted to enter the base point.
2. Select the **Base point** using the mouse or type the coordinates on the command line. A dotted window of the same size and shape as your CAD window will appear. This represents what you will see with that degree of pan. You will be prompted for the destination point.
3. Select the **Destination point** using the mouse or type the coordinates on the command line. The window will be redrawn to show the new display extents.



Rotate button

To rotate the drawing

1. Select **Rotate** from the View drop down menu or click the **Rotate** button on the View toolbar or type `VIEWROTATE` on the command line. A line will appear connecting the centre of the screen to the cursor pointer.
2. Move the mouse to adjust the required rotation angle and click to confirm. Note that the zero datum is horizontally to the right, i.e. facing East, and positive rotation is anti-clockwise.



Reset Rotation button

To reset any rotation

1. Select **Reset Rotation** from the View drop down menu or click the **Reset Rotation** button on the View toolbar or type `VIEWROTATERESET` on the command line. Any drawing rotation will be reset to zero.

To set drawing limits

1. Select **Set Limits** from the View drop down menu.
2. Use the mouse to select a window representing the drawing limits.
3. The drawing limits may be used when printing your drawing. See [Printer / Plotter Settings](#).

To clear drawing limits

1. Select **Clear Limits** from the View drop down menu.

Tools

This chapter covers the various tools built into SavoyCAD to make the drawing and editing process easier.

Grid

A rectangular grid can assist in aligning objects and in estimating distances.



Show Grid button

To display a grid

1. Select **Show Grid** from the Tools drop down menu or click on the **Show Grid** button on the Tools toolbar or type `GRID` on the command line. A rectangular grid pattern will appear over the CAD window. If no grid appears the grid spacing may be set too small or too large. If the spacing is set too large then the grid markers may be outside the displayed area. If it is set too small then SavoyCAD does not display it.
2. The command is a toggle so select it again to switch it off. The toolbar button remains down if the grid is switched on.

To change the grid spacing

Refer to [Preferences: Grid / Snap](#).

To change the grid colour

Refer to [Preferences: Display](#).

Snap grid

It is sometimes useful, when drawing repeating elements, to be able to snap to a grid pattern. The snap grid allows this and is independent of the normal grid described above. This command is also quite separate from the object snap modes described elsewhere.

To snap to the snap grid



Snap to Grid button

1. Select **Snap to Grid** from the Tools drop down menu or click on the **Snap to Grid** button on the Tools toolbar or type `SNAP` on the command line. The cursor will now snap to the invisible snap grid.

Tip

Although the snap grid is invisible you can of course set the grid spacing and origin to match the snap grid.

2. The command is a toggle so select it again to switch it off.
The toolbar button remains down if snap is switched on.

To change the snap grid origin or spacing

Refer to [Preferences: Grid / Snap](#).

To change the snap grid colour

Refer to [Preferences: Display](#).

Orthogonal mode

Orthogonal mode is useful when you need to draw precisely vertical and / or horizontal lines at a arbitrary positions.

To enable orthogonal mode

1. Select **Orthogonal Mode** from the Tools drop down menu or click on the **Orthogonal Mode** button on the Tools toolbar or type `ORTHOGONAL` on the command line.
2. The command is a toggle so click it again to switch it off.



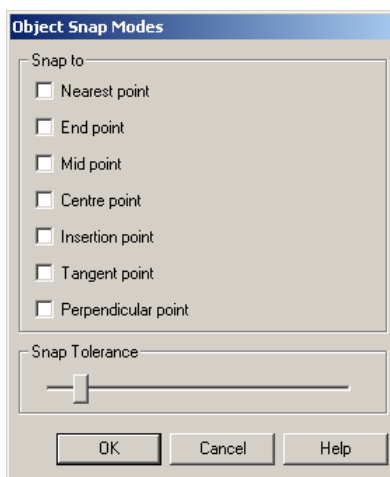
Orthogonal Mode button

Object Snap modes (Object Snap Modes dialog) ⁽¹²⁰⁰⁾

Snap modes make drawing objects easier by selecting points when you get within a preset distance or tolerance. The types of points (or snap modes) that you may use include the centres of circles, the ends of lines or arcs, and tangent points. You can select which, if any, snap modes you want to activate and can change them at any time.

The snap modes are controlled by the Object Snap Modes dialog and / or the Snap toolbar.

1. Select **Object Snap Modes** from the Tools drop down menu. The Object Snap Modes dialog will appear.



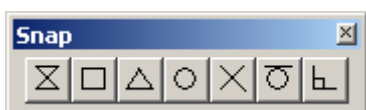
Object Snap Modes dialog

2. Select the modes that you want active.

Tip

The quickest and easiest way to switch snap modes on and off is to depress the appropriate button on the Object Snap toolbar. If the button is down the snap mode is active and vice versa.

3. You can adjust the snapping distance using the **Snap Tolerance** slider. Move the slider to the right to make the snap window larger, i.e. the cursor will snap to a point from further away.
4. Click **OK** to confirm your changes and close the dialog or **Cancel** to abort and close the dialog.



Object Snap toolbar

More on object snap modes

There is an order of precedence for the snap modes so that for example a nearest point will be snapped to before a perpendicular point. Note that the snap point will be offered for an entity when the cursor is within the snap tolerance distance of a visible part of that entity and not when it is close to the snap point itself. Thus, for example, a centre point snap will be offered when the cursor is close to the perimeter of a circle and not when it approaches the actual centre point.



Snap to Nearest Point button

Nearest Point

The cursor snaps to the nearest point on the nearest entity. Applies to all entities.



Snap to End Point button

End Point

Cursor snaps to the nearest end of the entity. Applies only to lines, polylines and arcs.



Snap to Midpoint button

Mid Point

Cursor snaps to the mid point on the entity. Applies only to lines, polylines and arcs.



Snap to Centre button

Centre Point

Cursor snaps to the centre of the entity. Applies only to Circles, Arcs and Ellipses. Note that the centre point will be snapped when the cursor is close enough to any visible part of one of these entities not when it is close to the centre point itself.



Snap to Intersection button

Intersection Point

Cursor snaps to the intersection between two entities. Applies to intersections between all entities.



Snap to Tangent button

Tangent Point

Cursor snaps to the point on the entity at which a line drawn from the current point would intersect tangentially. Applies only to circles, arcs and ellipses.

Note

This snap mode can only be used when the other end of the entity is fixed.



Snap to Perpendicular button

Perpendicular Point

The cursor snaps to the point on the entity at which a line drawn from the current point would intersect at right angles. Applies to all entities. If the perpendicular point occurs on the extension of a line no snap point will be offered.

Note

This snap mode can only be used when the other end of the entity is fixed.

Measuring distances (Distance command)



Distance button

1. Select **Distance** from the Tools drop down menu or click the **Distance** button on the Tools toolbar or type `DISTANCE` on the command line. You will be prompted to select the start point that you wish to measure from.
2. Select the **Start point** using the mouse or type the coordinates on the command line. You will be prompted to select the end point that you wish to measure to.
3. Select the **End point** using the mouse or type the coordinates on the command line. The distance between the two points will appear similar to the following:-

```
Distance = 46.505105m
Distance = 46.505105
DX = -45.674044
DY = 8.752515
```

The first value, **Distance**, is the absolute distance between the two points “as the crow flies” in real world units. This value is calculated using the **Drawing Scale** set in **Preferences, General**.

The second value for **Distance** is again the absolute distance between the two points but expressed in drawing units. Unless you are drawing to a scale other than 1:1 these two values will be the same.

The third value, **DX**, is the difference between the X ordinates of the start point and the end point.

The fourth value, **DY**, is the difference between the Y ordinates of the start point and end point.

Listing entity details (List command)

1. Select **List** from the Tools drop down menu or type `LIST` on the command line. You will be prompted to select the objects you wish to list.
2. Select the object(s) and right click to confirm.

Alternatively, you could select the objects first and then select the **List** command.

3. Each entity will be listed in a format similar to the following:-

```
Entity           : Line
Layer            : HT_HOUSES
Linetype         : CONTINUOUS
Linetype Scale  : 1.00
Colour (RGB)    : 255,0,0
Closed          : No
Start Point     : 56.3400,23.3220,0.0000
End Point       : 23.3420,12.5670,0.0000
```

The first five of these always appear. The remainder are specific to the entity type.

4. Block inserts are listed as follows:-

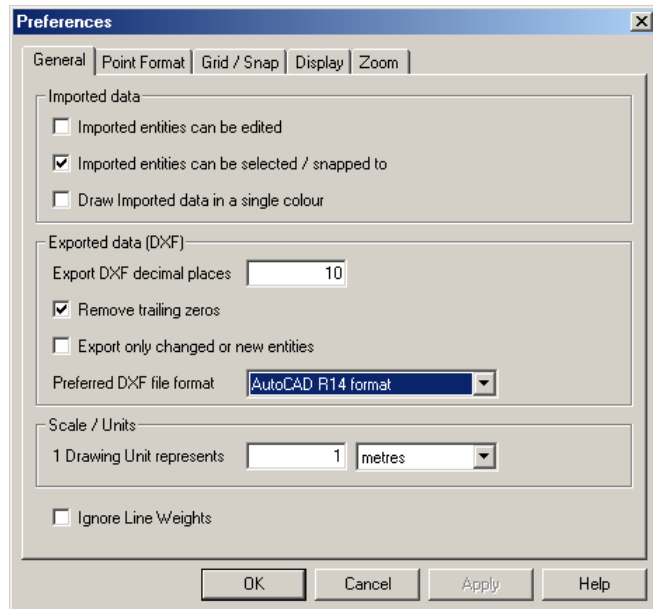
```
Entity           : Insert
Layer            : HT_DRIVES_HATCH
Linetype         : DASHED
Linetype Scale  : 1.00
Colour (RGB)    : 255,0,0
Block           : W123D65X
Insert Point    : 12.4567,43.7865,0.000
Scale X         : -1.0000
Scale Y         : 1.0000
Scale Z         : 1.0000
XY Rotation     : 34.7898
```

Notice in the above listing that the **Scale X** value of -1 indicates that the block was flipped or mirrored about the X-axis upon insertion.

Preferences (Preferences dialog)

You can customise the way SavoyCAD works by changing your preferences.

1. Select **Preferences** from the Tools drop down menu. The Preferences dialog will appear.

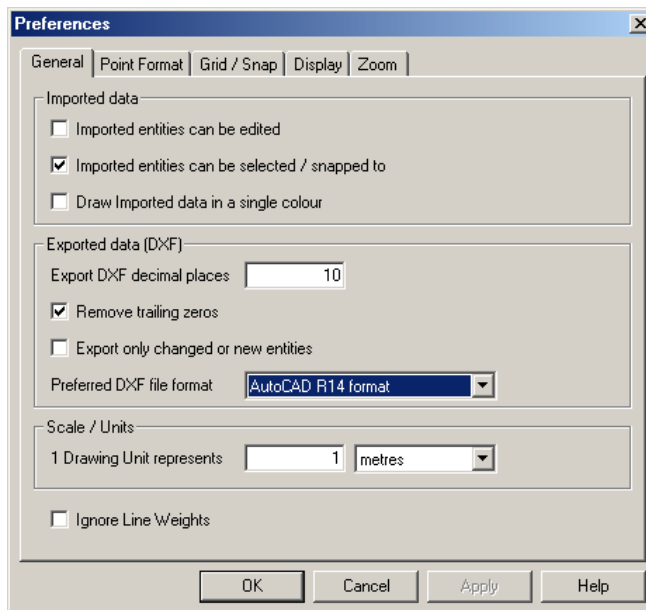


Preferences dialog

2. Click the appropriate tab to edit or view your preferences.
3. Edit the data as required and click **OK** to confirm when you are satisfied. Click **Cancel** to abort all changes and close the Preferences dialog.
4. The **Display** and **Zoom** settings are saved between sessions and may be reset to defaults by clicking the Reset button. The other settings are only saved for the duration of the session.

Preferences: General ⁽¹⁵⁰⁰⁾

1. Click on the **General** tab.



Preferences dialog: General tab

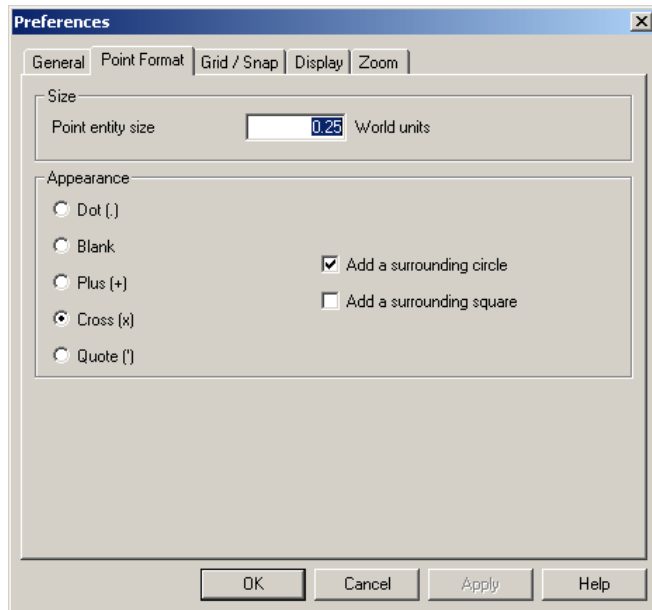
12. Tick **Allow imported entities to be edited** if you want to be able to edit entities in imported DXF files.
13. Tick **Allow imported entities to be selected / snapped to** if you want to be able to select or snap to imported entities.
14. It is sometimes useful to be able to distinguish between imported data (of any type, DXF, WTP or ATX) and newly drawn entities. By default imported entities are drawn in the respective colours but you may force them all to appear in single colour by ticking **Draw imported entities in a single colour**. You may then set the colour on the Display tab.
15. When a DXF file is saved the entity coordinates and angles are saved by default to 16 decimal places. You can reduce the **DXF Decimal places** in order to reduce the file size but we recommend that you do not reduce the figure below 8. We do not recommend reducing this figure if the DXF file is the only format saved since you are irretrievably losing accuracy.
16. DXF files are text format so you may be able to considerably reduce the file size by not saving trailing zeros on decimal numbers. To do this tick **Remove trailing zeros**.
17. Select your **Preferred DXF file format** bearing in mind that AutoCAD R14 format does not support line weights and AutoCAD R12 format does not support the ellipse entity (they are converted to polylines) or the lightweight polyline entity. AutoCAD R12 format should be the most portable format. If you are working with Unicode data then you should use the AutoCAD 2007 format which is fully Unicode compatible.
18. SavoyCAD is unable to interpret all data held in DXF files from all sources system. The data it cannot understand it ignores and so it is lost. We therefore recommend that, if

you have imported DXF data from another source, you tick **Export only changed and new data** to save only the new or changed data.

19. SavoyCAD needs to know the units you are using in order to show distances correctly and to plot output to a specified scale such as 1:200. You will probably draw 1:1 by setting 1 drawing unit equivalent to 1 real world unit (for example, metre). However, you may from time to time need to read DXF files that have been generated at some other scale. In these cases you will need to set **1 Drawing Unit represents** to the appropriate number of real world units.
20. By default SavoyCAD draws entity attributes, including line weights, as accurately as possible. If you would rather all lines were drawn in the thinnest line tick **Ignore line weights**.

Preferences: Point Format (1530)

1. Click on the **Point Format** tab.



Preferences dialog: Point Format tab

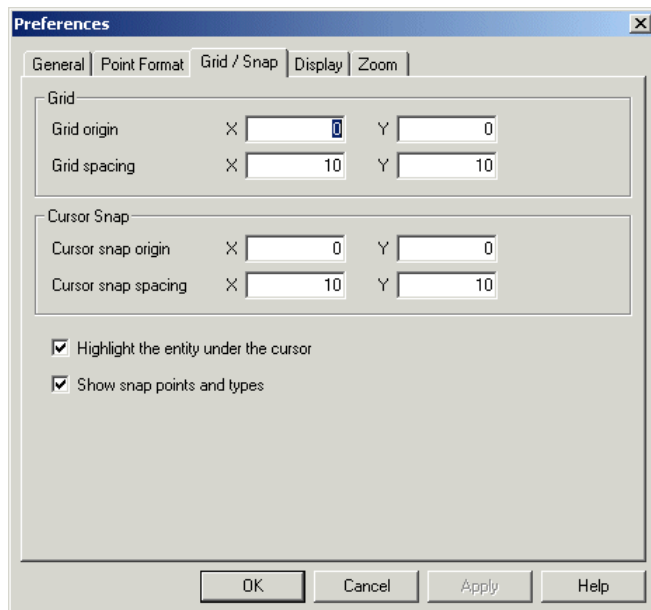
2. By default point entities are drawn 0.25 units wide and high based upon the presumed use of metres. If you are working in millimetres or inches you may find this too small and wish to set **Point size** to, say, 250 millimetres (or, say, 10 inches).
3. Point entities are drawn by default as a cross surrounded by a circle. You may select an alternative format with optional surrounding square and or circle.

Note

Any changes to the point format settings applies to all points in the current drawing including those in imported data and those in block inserts.

Preferences: Grid / Snap ⁽¹⁵²⁰⁾

1. Click on the Grid / Snap tab.



Preferences dialog: Grid / Snap tab

2. A grid can optionally be displayed.
3. Set the **Grid Origin** to the real world coordinates at which you want the grid to start. Grid lines will be drawn either side of this point.
4. Set the **Grid Spacing** to the required horizontal (**X**) and vertical (**Y**) spacing.
5. In addition to the grid a snap grid may be displayed. This is invisible but you may still adjust the origin and spacing.
6. Set the **Cursor Snap Origin** to the real world coordinates at which you want the grid to start. Invisible snap points will be spaced either side of this point.
7. Set the **Cursor Snap Spacing** to the required horizontal (**X**) and vertical (**Y**) spacing.

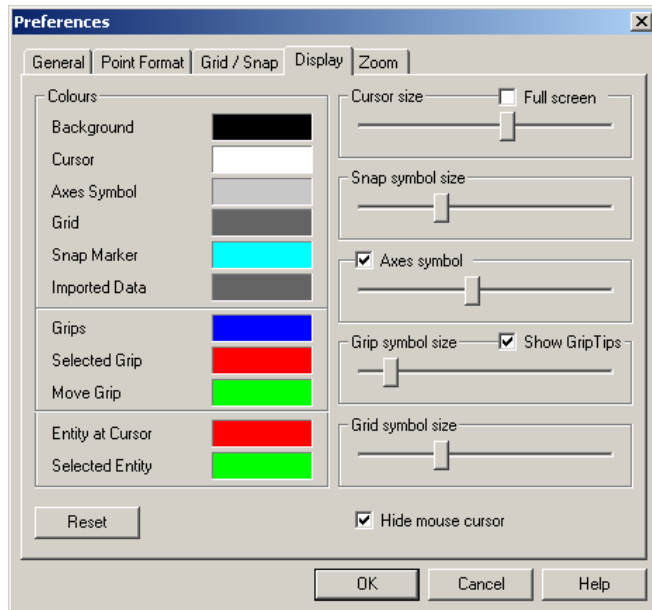
Note

The snap feature is independent of the object snap feature.

21. By default SavoyCAD highlights the entities as the cursor passes over them. If you are working with a large drawing or lots of paths SavoyCAD may run slower and you may therefore want to disable this option by unticking **Highlight the entity under the cursor**.
22. If any snap modes are enabled SavoyCAD indicates possible snap points and types with small icons. If you are working with a large drawing or lots of paths SavoyCAD may run slower and you may therefore want to disable this option by unticking **Show snap points and types**.

Preferences: Display ⁽¹⁵¹⁰⁾

1. Click on the **Display** tab.



Preferences dialog: Display tab

2. You can change the display colours to anything you wish. To change a colour, simply click on the colour swatch. Colours may be set for the Background, Cursor, Cursor Box, Snap Marker, Grip, Move Grip, Selected Grip, Grid, Axes Symbol and Imported Data.

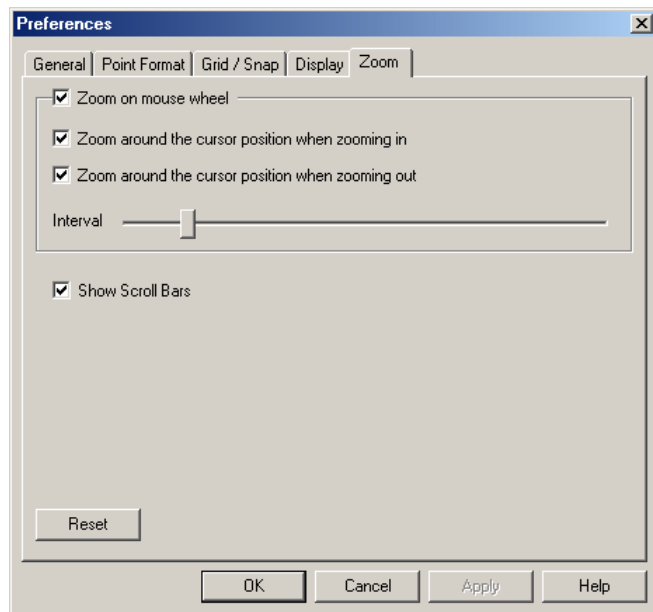
Note

If you set conflicting colours such that for example the cursor is the same colour as the background the foreground colour will be changed to ensure that it is visible.

3. Tick **Axes symbol** to display a coordinate axes symbol in the bottom left corner of the window. Change the size of the symbol using the slider.
4. Change the size of the cursor crosshairs using the **Cursor size** slider. If you tick **Full screen** the crosshairs will be shown extending to the edges of the CAD window.
5. Change the size of the entity editing grips using the **Grip Symbol Size** slider control.
6. To enable the display of GripTips (text prompts that appear as you hover over a grip) tick **GripTips**.
7. Change the size of the symbol displayed when the cursor snaps to point using the **Snap Symbol Size** slider control.
8. Change the size of the grid cross hairs using the **Grid Symbol Size** slider control.
9. Tick **Hide mouse cursor** to hide the arrow pointer while the cursor is over the drawing window.
10. Click **Reset** to reset all values to the Savoy defaults.
11. Click **OK** to confirm changes or **Cancel** to abort all changes and close the dialog.

Preferences: Zoom (1540)

1. Tick on the **Zoom** tab.



Preferences: Zoom tab

23. If you have a mouse with a scroll wheel then you may want to tick **Zoom on mouse wheel** to zoom in and out by turning the mouse wheel.

By default this zooms in and out on the centre of the CAD window. If you tick **Centre on cursor when zooming in** the zoom in will be centred on the cursor position when the wheel is turned. This allows for fast movement around the drawing without the need to pan the drawing. Unsurprisingly, **Centre on cursor when zooming out** works the same for zooming out.

Change the increment using the **Zoom on mouse wheel** slider control. This adjusts the zoom factor for each click of the mouse wheel.

24. Tick **Show scroll bars** to display scroll bars at the right and bottom of the CAD window when you zoom such that part of the drawing is not visible. They allow you to pan sections of the drawing that are off-screen back into view.

Note

Displaying scroll bars requires more calculation so if you are finding that there is a time lag every time you draw an entity, perhaps because you are running on a slow computer or working with a very large drawing, then you may wish to try switching off Scroll bars.

2. Click **Reset** to reset all values to the Savoy defaults.

Digitising drawings

SavoyCAD has built in support for digitizing tablets allowing you to electronically “trace” drawings and save them in digital form.

Setting up your digitising tablet

To set up your digitising tablet

1. Make sure that the tablet that you wish to use has WinTab support.
2. Follow the instructions provided with your digitising tablet to connect it to your computer.
3. Install the necessary drivers including the WinTab driver.
4. Follow the manufacturers instructions to test the digitising tablet before trying to use it with SavoyCAD.

Calibrating the tablet

Before you can use your digitising tablet you need to calibrate it. This process locates the drawing relative to the tablet and allows SavoyCAD to calculate the real world coordinates of the points you subsequently pick as you trace your drawing.

The calibration process requires you to identify three points on your drawing whose coordinates you know accurately. Ideally these points should be part of the base drawing, e.g. setting out data, in case the drawing has shrunk or stretched. They should also be as far apart as possible and preferably form a triangle.

If there are no coordinated points available on the base drawing but you know the scale of the drawing, then you could plot three points to the appropriate scale. Note however, that if the drawing has shrunk or stretched your plotted points will not be accurate and distances (e.g. road widths) in your digitized drawing may be greater or smaller than they should be.

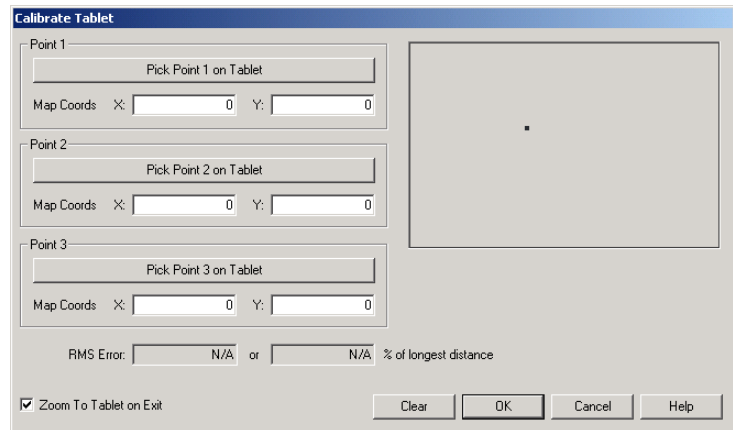
To calibrate your tablet ⁽¹⁴¹⁰⁾

1. Run SavoyCAD.
2. Select **Calibrate** from the Tablet drop down menu.

If you do not have WinTab drivers installed then you will be warned. Refer to [WinTab driver not found](#).

If SavoyCAD cannot access the digitizing tablet then you will be warned. Refer to [Unable to access digitising tablet](#).

Otherwise, the Calibrate Tablet dialog will appear.



Calibrate Tablet dialog

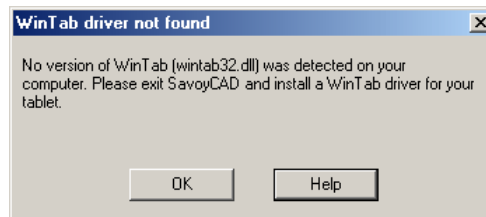
- Using the mouse click **Pick Point 1 on Tablet**. The text will change to Picking Point 1 on Tablet.
- Now, using the digitising tablet puck, carefully select the first of your reference points. The text on the button will change to Repick Point 1 on Tablet. If you wish to repick the point at any time simply click the button again and reselect the point.
- Enter the real world coordinates of point 1 in the **Map Coords X** and **Map Coords Y** boxes. Notice that the point is been plotted in red in the window to the right of the buttons.
- Repeat this process with points 2 and 3 which will be plotted in the window in green and blue respectively.
- As soon as you have picked three distinct points the **RMS Error** will appear at the bottom of the dialog. Also displayed is the error expressed as a percentage of the longest distance. The smaller this value is the more accurately you have positioned the drawing. If you are not happy with the accuracy try re-picking the points or selecting different points. For more information on how to interpret the RMS error refer to [Calibration error](#).
- If you tick **Zoom to tablet reference point on exit** SavoyCAD will zoom to the extents of your three reference points when you close this dialog. This allows you to see subsequently digitised data without having to pan and zoom first.
- Once you have calibrated the tablet you may wish to plot them to the same scale as your original drawing and overlay it as a double check.

Note

If you move the drawing, remember to recalibrate the tablet before digitising any more data.

WinTab driver not found ⁽¹⁶⁰⁰⁾

1. If SavoyCAD cannot find your WinTab driver, the WinTab driver not found dialog will appear.

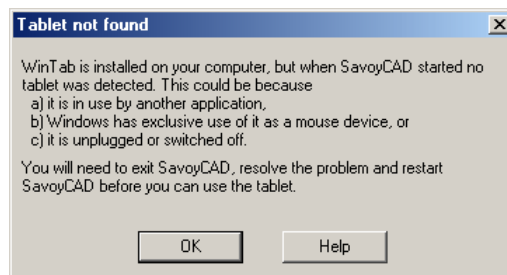


WinTab driver not found dialog

2. Check that you have installed the WinTab device driver supplied with your tablet.
3. Check that the WinTab driver supports the version of Windows that you are running. If it does not then try going to the manufacturers website and searching for a driver that does.

Unable to access digitising tablet ⁽¹⁶¹⁰⁾

1. If SavoyCAD is unable to find a digitizing tablet connected to your computer, the Tablet not found dialog will appear.



Digitising tablet not found dialog

2. Check that the tablet is plugged into the computer and switched on.
3. Test the tablet if possible according to the manufacturers instructions.

Calibration error

SavoyCAD only really requires two points to be able to position the drawing. The third point is requested for confirmation and in order to calculate likely error. The error at each point is calculated by first back-calculating point 1 from points 2 and 3, point 2 from points 1 and 3 and finally point 3 from points 1 and 2. The distance between the calculated coordinates and the actual coordinates is then calculated for each point. The RMS error is then the square root of the mean of the squares of these distances. It is best expressed thus:-

$$\text{RMS Error} = \text{SQRT}((\delta 1^2 + \delta 2^2 + \delta 3^2) / 3)$$

Where:-

- $\delta 1$ is error at Point 1
- $\delta 2$ is error at Point 2
- $\delta 3$ is error at Point 3

The error will normally be significantly less than 1%. However, the calibration calculation assumes equal distortion in the X and Y directions, so if your drawing is stretched in one direction only you may get higher errors.

Warning

You may get apparent high degrees of accuracy even when making considerable errors in picking the points as long as the triangle formed by the picked points forms a scale version of the true triangle. We therefore recommend that you view the RMS error as a guide rather than as an absolute criteria.

Seeing your reference points

You may, at some point after you have calibrated the tablet, need to remind yourself which points you used. To do this:-

1. Select **Show tablet points** from the View drop down menu. The last three digitizing tablet reference points used will be plotted on your drawing.

Digitising data

The same commands are used to trace a drawing as are used to draw entities from scratch but using the tablet puck instead of the mouse. So for example:-

To digitise a point

1. Select **Point** from the Draw drop down menu or click **Point** on the Draw toolbar or type `POINT` on the command line. You will be prompted for the point.
2. Pick the **Point** using the digitising puck. The point will appear in your chosen format (which may render it almost invisible!).

Note

You can change the appearance of point entities in the Preferences dialog.

To digitise a line

1. Select **Line** from the Draw drop down menu or click **Line** on the Draw toolbar or type `LINE` on the command line. You will be prompted for the start point.
2. Pick the first point of the line using the digitising puck. The point will be placed and a line displayed connecting it to the cursor position. You will be prompted for the end point.
3. Pick the end point using the puck. The line will appear selected.



Point button



Line button



Polyline button

To digitise a polyline

1. Select Polyline from the Draw drop down menu or, using the mouse, click on the **Polyline** button on the Draw toolbar or type `PLINE` on the command line.
2. Now, using the digitising tablet puck pick the first point on the line. Button 1 on the puck is usually the select button but, if in doubt, refer to your tablet's instructions.
3. Move the cursor to the next point and pick that.
4. If you want to trace an arc, type `A` (for arc) on the command line. The line connecting the last point to the cursor will change to an arc and the command line prompt will change. Now, using the puck, pick the end point.

If you want to revert to line segments, type `L` (for line) on the command line. The arc connecting the last point to the cursor will change to a line and the command line prompt will change. Pick the next point using the puck.

5. Repeat the process until you have traced the whole line.
6. You can close the line by typing `CL` on the command line or finish by clicking button 2 or the confirm button.

To digitise a circle

Circles can be drawn in two ways; by defining three points on the circle or by defining the centre and radius.

To digitise a 3-Point Circle

1. Select **Circle: 3 Points** from the Draw drop down menu or click **3-Point Circle** on the Draw toolbar or type `CIRCLE3P` on the command line. You will be prompted for the first point.
2. Pick the first point on the circle using the digitising puck. At this stage nothing will be visible. You will be prompted for the second point.
3. Pick the second point on the circumference of the circle using the puck. Now you should see a circle drawn through the two defined points which changes radius and position as you move the cursor. You will be prompted for the third point.
4. Pick the third, and final, point using the puck. The full circle will now appear selected.

To digitise a Centre/Radius Circle

1. Select **Circle: Centre / Radius** from the Draw drop down menu or click **Centre/Radius Circle** on the Draw toolbar or type `CIRCLE` on the command line. You will be prompted for the centre of the circle.
2. Pick the centre point using the digitising puck. A circle will appear whose radius changes as you move the mouse. You will be prompted for the radius of the circle.
3. Pick a point on the circumference of the circle using the



3-Point Circle button



Centre/Radius button

puck. The circle will now appear selected.

To digitise an arc



Arc button

1. Select **Arc: 3 Points** from the Draw drop down menu or click **Arc** on the Draw toolbar or type `ARC` on the command line. You will be prompted for the start point.
2. Pick the start point of the arc using the digitising puck. At this stage nothing will be visible. You will be prompted for the middle point.
3. Pick the middle point using the puck. An arc will appear whose shape changes as you move the mouse. You will be prompted for the end point.
4. Finally, pick the end point of the arc using the puck. The completed arc will appear selected.

To digitise an ellipse

It may not be easy to accurately digitise an ellipse unless you can fairly precisely identify the ends of the major and minor axes.



Ellipse button

1. Select **Ellipse** from the Draw drop down menu or click **Ellipse** on the Draw toolbar or type `ELLIPSE` on the command line. You will be prompted for the centre of the ellipse.
2. Pick the centre of the ellipse using the digitising puck. At this stage nothing will be visible. You will be prompted for the major axis point.
3. Pick one end of the major axis using the puck. At this point an ellipse will appear whose shape changes as you move the cursor. You will be prompted for the minor axis point.
4. Pick one end of the minor axis using the puck. The completed ellipse will appear selected.

Note

Although the ends of the axes are requested in a particular order (major axis first), you may select a minor axis point that makes it the longer of the two.

Tools to help you

Digitised drawings can appear very disjointed, especially if you need to move the map and recalibrate the tablet. However, some of the features in SavoyCAD may be used to eliminate some of these problems.

Snap modes

If you have to stop and restart digitising a line you can end up with a gap. You can prevent these kinds of discontinuities by using snap to end. You may other snap modes useful. For example, snap to centre may be used if you need to digitise circles concentric circles. Simply toggle the appropriate snap mode and move the digitising puck close to the required point. Refer to [Object Snap modes \(Object Snap Modes dialog\)](#) for how to set snap modes.

Zoom

Zoom may be used at any time to magnify an area and make it easier to see the data you are digitising. Just use Zoom Previous to return to the previous zoom level.

Glossary of Terms

Absolute coordinates

Coordinates expressed as values independent of any other coordinates.

ATX file

The native file format of AutoTrack.

Axes Symbol

A symbol that indicates the direction of the X and Y coordinate axes.

Cursor

A crosshairs marker similar to the standard Windows arrow cursor that indicates the current position of the mouse.

Cursor Box

The square surrounding the cursor crosshairs.

Digitising tablet

A device used to electronically trace printed graphics into digital form. They range in size from A4 up to A0.

Docked

A window or toolbar is said to be docked when it appears to be attached to the frame of the parent window. Objects may be docked against any side of the window frame.

DXF

Stands for Drawing eXchange Format. A file format developed by Autodesk but now used by most CAD systems to exchange data.

Entity

In the context of SavoyCAD and entity is a visible or sometimes invisible object that forms part of a drawing.

Extents

The furthest point of any entity in each direction. Zoom Extents to view everything in your drawing.

Explode

Reduce an object (e.g. a block or a polyline) to its constituent entities.

Floating

A window or toolbar is said to be floating when it does not appear attached to the frame of parent window.

Frozen

A term used in layer management. See Frozen layer.

Frozen layer

All entities on a Frozen layer are invisible. The same would be true if the layer were Off.

Grid

A rectilinear array of crosses at specified horizontal and vertical intervals superimposed on the drawing.

Grip

Small squares that appear at each edit point on an entity when it is selected. Grips can be selected and moved using the mouse.

Import

To add data and entities from an external file into the current drawing.

Imported Data

Drawing entities that have been loaded into the current drawing using the Import command.

Insert point

The datum point in a block that is used when insertion it into a drawing.

Insertion point

The point at which a block or text is inserted.

Layer

A means of grouping objects together. The properties of layers (e.g. visibility) may be controlled independently of the objects they include.

Locked

A term used in layer management. See Locked layer.

Locked layer

The entities on a locked layer may not be edited.

Major axis point

The two points (one at each end) at the intersection of the major axis of an ellipse and the ellipse itself.

Minor axis point

The two points (one at each end) at the intersection of the minor axis of an ellipse and the ellipse itself.

Move Grip

A move grip is the grip used to move an entity. Every entity has a move grip.

Object Snap

The ability of the cursor to move to an object that is within a certain distance without the user having to precisely position it.

Polyline

A line made up of any number of line and arc segments.

Puck

The input device connected to a digitizing tablet. Similar to a mouse but with crosshairs to enable accurate selection of points.

Relative coordinates

Coordinates expressed as distances from the last point by prefixing the ordinate(s) with the @ symbol.

Scroll bar

Scroll bars appear at the side and / or bottom of the CAD window if you zoom in such that part of the drawing is not visible. They allow you to pan the window horizontally and vertically.

Selected Entity

An entity that has been selected for editing. Selected entities appear in a special colour and always have at least a move grip visible.

Selected Grip

A grip that has been selected by moving the mouse over it and clicking.

Snap

The ability of the cursor moves to a given position when it is within a certain distance without the user having to precisely position it.

Snap Grid

A rectilinear grid to which the cursor may optionally snap. The Snap Grid is not the same as the Grid.

Snap Symbol

A symbol that appears at a potential snap point when the cursor is within a specified distance.

Snap Point

The point that the cursor snaps to when it gets within a certain distance.

SVC file

The native file format of SavoyCAD.

Toolbar

A toolbar is a row of buttons that provide shortcuts to menu commands.

WinTab

WinTab is a device driver standard for graphical input devices. The WinTab driver for a particular device allows all WinTab compliant applications to access the device.

WTP file

The native file format of WinTrack.

Index

A

Attributes 18

B

Blocks & Inserts 18, 28

C

Calibrating the tablet 59

Calibration error 61

Centre Point 48

Colours list 38

Context sensitive on-line help 2

D

Digitising data 62

Drawing 20

E

Editing 24

End Point 48

Entering values 17

Example 19

F

File handling 7

Fundamentals of drawing & editing 17

G

General 3

Getting help 2

Glossary of Terms 67

Grid 45

Grip editing 27

GripTips 27

H

Help 5

I

Intersection Point 48

L

Last used file list 10

Layer visibility options 18

Layers 18

Layers list 31

Line Weights list 39

Linetypes list 35

Listing entity details (List command) 50

M

Maintain Layers dialog (1000) 33

Maintain Linetypes dialog (1100) 36

Manual layout 1

Measuring distances (Distance command) 49

Menus 3

Mid Point 48

More on object snap modes 47

Mouse conventions 1

N

Nearest Point 48

Notation conventions 1

O

Object Snap modes (Object Snap Modes dialog)
(1200) 47

Opening vs Importing files 7

Orthogonal mode 46

P

Perpendicular Point 48

Preferences (Preferences dialog) 51

Preferences: Display (1510) 56

Preferences: General (1500) 52

Preferences: Grid / Snap (1520) 55

Preferences: Point Format (1530) 54

Preferences: Zoom (1540) 57

Previewing your printed output (Print Preview
screen) 11

Printer / Plotter Settings: Notes (1300) 16

Printer / Plotter Settings: Print Settings (1310) 13

Printer / Plotter Settings: Title Block (1320) 15

Printing (Plotter / Printer Settings dialog) 12

S

Seeing your reference points 62

Selecting / deselecting objects 17

Setting up your digitising tablet 59

Snap grid 45

Snap modes 64

T

- Tangent Point 48
- To automatically repeat the last command 23
- To automatically use the last point drawn 23
- To calibrate your tablet (1410) 59
- To change a layer's name 32
- To change the colour of an object 38
- To change the current layer 34
- To change the current linetype 37
- To change the grid colour 45
- To change the grid spacing 45
- To change the line weight of an entity 39
- To change the linetype of an entity 35
- To change the snap grid colour 46
- To change the snap grid origin or spacing 46
- To change to colour of a layer 32
- To clear drawing limits 43
- To copy an object 25
- To create a block 28
- To create a new layer 32, 34
- To create a new linetype 35, 37
- To delete a layer 34
- To delete a linetype 37
- To delete an object 26
- To digitise a 3-Point Circle 63
- To digitise a Centre/Radius Circle 63
- To digitise a circle 63
- To digitise a line 62
- To digitise a point 62
- To digitise a polyline 63
- To digitise an arc 64
- To digitise an ellipse 64
- To display a grid 45
- To draw a 3-Point Circle 21
- To draw a Centre/Radius Circle 21
- To draw a circle 21
- To draw a line 20
- To draw a point 20
- To draw a polyline 20
- To draw an arc 22
- To draw an ellipse 22
- To draw text 22
- To enable orthogonal mode 46
- To explode a block 29
- To explode a polyline 23
- To export a DXF file 9
- To go back to the previous zoom window 42
- To go forward to the next zoom window 42
- To import a DXF, ATX or SVC file 9
- To insert a block 28
- To make a layer invisible 31
- To make a layer non-editable 32
- To move a toolbar 4
- To move an object 25
- To move an object to another layer 31
- To move an object using grips 27
- To open a drawing file 7
- To pan across the the drawing 43
- To redraw all entities 41
- To reset any rotation 43

- To resize / reshape an object 27
- To resize the command window 4
- To rotate an object 26
- To rotate the drawing 43
- To save a copy of a file 9
- To save a drawing file 8
- To set drawing limits 43
- To set up your digitising tablet 59
- To snap to the snap grid 45
- To start a new drawing 7
- To stop a layer from being printed 32
- To switch toolbars on and off (Toolbars dialog) (1420) 3
- To view the coordinates of an object (List command) 24
- To view the properties of an object (Properties dialog) (1400) 24
- To zoom in 41
- To zoom out 41
- To zoom to a specific area 42
- To zoom to the drawing extents 42
- To zoom to the drawing limits 42
- To zoom to the tablet limits 42
- Toolbars 3
- Tools to help you 64
- Tooltips 5

U

- Unable to access digitising tablet (1610) 61

W

- What it can and can't do 1
- WinTab driver not found (1600) 61

Z

- Zoom 65