Anna Gresham School of Landscape Design

CAD for Beginners

CAD 3: Using the Drawing Tools and Blocks

Amended for DraftSight V4 October 2013

INDEX OF TOPICS for CAD 3   **Pages**

SCOPE OF THIS MODULE

This module will teach you how to use the essential CAD drawing tools. Not all tools are covered but once you learn to use the basic ones then using the rest will become intuitive.

ESnap

ESnap stands for Entity Snap. A 'snap' point is a special point on an entity that allows other entities to join onto it. ESnap can be turned off and on using the ESnap button on the Status bar. While snaps are used for most drawing operations there are time when they need to be turned off.

Because CAD is all about accuracy, using snaps is essential. You can preset the default snaps in Options>> User Preferences >> Drafting Options >> EntitySnaps. An easier way is to right-click the **ESnap** button on the **Status** bar.

1. Open DraftSight with any template you like.
2. Right-click on the **ESnap** button then click **Settings**.
3. All snaps are displayed. The ones that are checked are available when ESnap is turned on, and the others not. You can make as many or as few as you like available. The current selection is the 'default' set and is most satisfactory. Click **Cancel** to exit without changes.

- Each snap point has a special indicator to let you know which snap point the cursor is currently over.
- When several snaps are active at once you need to watch for the indicator to ensure you are snapping to the right point.
- The indicator pops up as a magenta-colored shape accompanied with a label once the cursor is in range.
- Snap indicators do not normally showup unless an appropriate tool is selected.
- Use snaps to start a new entity joined to another or to a marker point, end an entity on another entity or on a marker point, move and copy entities to join onto other entities or to a marker point, snap various parts of a complex entity onto other entities or marker points.
- Whenever you need any part of any entity to join onto any part of another entity then use ESnaps.

Getting ready to practice with snaps

1. Using the appropriate tools, draw two lines intersecting to form a cross, a rectangle and a circle - as shown below.
2. Activate **ESnap** (the ESnap button should look pressed in).
3. Select the **Line** tool but do not actually draw anything. Follow all the procedures on the next page to familiarise yourself with the snap indicators and the snap functions.

Center point snap: Hover near the center of the circle. The Center point snap indicator is a magenta circle.

Quadrant point snap: Hover on the outline of the circle at 12, 3, 8 and 9 o'clock positions. These are the 'quadrants' of a circle and the quadrant point snap indicator is a magenta diamond.

Tangent snap: Click to start a line a short distance from the circle. Hover over the circle until you see the Tangent snap indicator, which is a magenta circle with a short line attached to the top. Any line finished on a Tangent snap point will form a tangent with the circle.

IMPORTANT NOTE: Other snap points will also appear as you move the cursor over an entity. Be sure to wait until the right snap indicator is visible before clicking to complete the line.
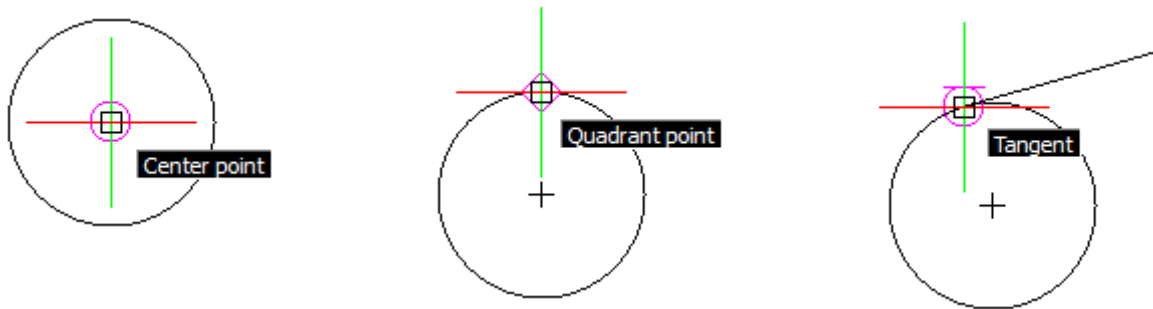
End point snap: Hover over one end of a line, then hover over the corners of the rectangle. End point snap will appear each time. End point snap occurs at the end of any line, whether it be a single line or part of a more complex shape. End point snap is the most used snap. The indicator is a magenta square.

Midpoint snap: This snap occurs at the very middle of a line. Hover over one side of the rectangle at about the middle. The indicator is a magenta triangle.

Intersection snap: This is used to snap an entity onto any intersecting point. Hover over the intersection of the two lines. The indicator is a magenta X.

Perpendicular snap: This will snap the end of a line perpendicular to the entity it joins. The indicator is a magenta 'right-angle' symbol. To try this, start a line a short distance above the rectangle. Move the cursor along one side of the rectangle and wait until you see the right-angle indicator appear. Don't confuse it for any other snap indicator such as mid point. Click while the perpendicular indicator is visible. The new line will be at exact right-angles to the rectangle.
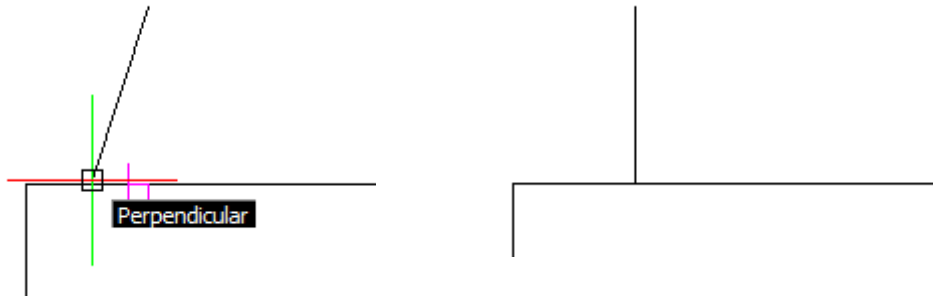
Node snap: This snap works with marker points allowing you to snap to a marker point. To try this, first draw a point (see page 10 of this module). Now hover over the point to see the magenta indicator (an X over a circle).
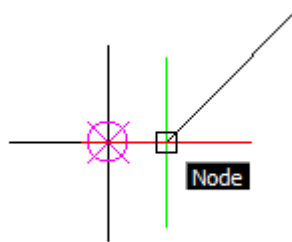
Insert snap: This snap is for text. It allows you to snap a text object at its insertion point. This is useful when you want to line text up with other objects.
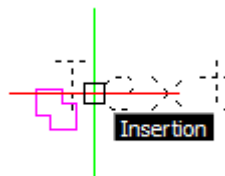
ESSENTIAL DRAWING TOOLS

Starting a new drawing sheet.

1. Start DraftSight. Start a **New** drawing with a **1:96** [**1:100**] template.
2. Switch to the Titleblock layer.
3. Draw the sheet border rectangle using the dimensions for an **ARCH C** [**A2**] paper size and **1:96 [1:100]** scale. *Refer to CAD 2 if you have forgotten how to do this.*
4. Click **Zoom Fit** to bring the titleblock into full view.
5. Lock the Titleblock layer by clicking the padlock icon in the layer box, then switch to Layer zero. You are now ready to begin drawing practice.

NOTE: The round brackets show the menu selection for each tool. The image shows the icon as seen on the tool bar.

Distance or Length format

Imperial: 6'-6-1/2" is 6ft 6 and 1/2 inches. Note the use of the short dash instead of a space.

Metric: 5500 is 5500 millimetres (the same as 5.50 metres). Metric users enter all distances in millimetres. No need to add 'mm' after the number.

Line (single line segment)

1. Select the **Line** tool (Draw >> Line). Type **S** then **Enter** to select single line segment.
2. Click to start the line. Remember to have **ESnap** and **Polar** active.
3. Rotate the line through angles 0, 45, 90, 135, 180 etc by moving the cursor and watching for the blue Polar guides.
4. Type the length into the command window, **10'-4"** [**3150**] then press **Enter**. The line will draw automatically. Zoom in to see it if necessary.
5. Click to start a new line, then repeat steps 3 and 4 choosing your own line lengths.
6. When you have enough lines, press **Escape** to release the line tool.

Line (angled)

There is no option offered on the Command line to specify a line angle. Mostly lines will be either horizontal or vertical, at 45 degrees or whatever your Polar settings are. However, if you need to draw a line at a different angle you can.

1. Select the Line tool then click to start the line.
2. Type in the Command window **@** then the *length* then the symbol **<** then the *angle.*
    Example 1:   @5'<20   specifies a 5 ft long line at 20 degrees.
    Example 2: @1200<30 specifies a 1200mm long line at 30 degrees.
3. Press **Enter** to draw the line.

<u>Line</u> (continuous line)

1. Select the **Line** tool (Draw >> Line). Type **C** then **Enter** to select continuous line.
2. Click to start the line. Remember to have **ESnap** and **Polar** active.
3. Set the desired line angle (0, 45, 90, 135, 180 etc) by moving the cursor and watching for the blue Polar guides.
4. Type the length into the command window, **6'-6-1/4"** [**1875**] then press **Enter** once. The line will draw automatically. Remember to pan and zoom as needed.
5. Set the angle for the next line. Remember that it is already connected to the first line.
6. Type in a length then press **Enter**.
7. Repeats steps 5 and 6 until you have drawn enough lines then press **Escape** to release the tool.

<u>Dashed lines</u>

Dashed lines are created by selecting a dashed line style from the **Line Style** list. The spacing of the dashes is determined by the Line Scale setting. Line scale for Entities was specified when you created the templates. But if you feel that the dash spacing is not suitable you can change it. After drawing the dashed line:

1. Click on the line to select it.
2. Change the Line Scale in the Properties palette.
3. Press **Enter** then **Escape** twice.
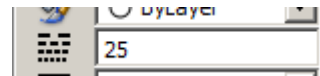
NOTE: Line scale does not always show correctly in screen and when you zoom out the line may appear solid. However, it will not print as a solid line. Judge the scale by the size of the object you have drawn. If the dashes appear to far apart for the size of the object, then reduce the line scale. If they appear too close together then increase the line scale.

<u>Polygon</u> (Draw >> Polygon)

1. Select the **Polygon** tool.
2. Enter the number of sides. In this case enter **8**. *Don't forget to press Enter each time you type or change something in the Command window.*
3. Click where you want the center of the polygon to be.
4. Enter **CO** to specify that you want the polygon sized by distance from <u>center</u> to corner. *Did you notice the options (in blue) in the Command window?*
5. Enter the distance across, in this case **12'** [**3600**] then Enter. The Polygon will draw squarely oriented regardless of the rotation when you pressed Enter. It will measure 24' [7200] from corner to corner.
6. Select the **Polygon** tool
7. Enter the number of sides.
8. Click the center point.
9. Enter **S** to specify that you want the polygon sized by the length of each side.
10. Enter the length of the side, in this case **3'-2"** [**950**] then Enter. The Polygon will draw squarely oriented regardless of the rotation when you pressed Enter.

Spend time practicing drawing polygons by both methods. Also try snapping the center and/or corner onto another entity, say another polygon or line.

Rectangle (Draw >> Rectangle)

1. Select the **Rectangle** tool.
2. Click on the sheet to start.
3. Type **D** for 'dimensions' (refer to the Command window) then **Enter**.
4. Following the prompts in the Command window, type in the horizontal dimension **8'** [**2400**] the **Enter**.
5. Type in the vertical dimensions **4'** [**1200**] then Enter. The rectangle will automatically draw.

Draw some more with different dimensions.

- You can also draw rectangles without entering distances. Just click to start, drag, then click to end.
- You can begin and end a rectangle by snapping onto existing entities if ESnap is active.

Polyline (Drawn >> Polyline)

A Polyline is a line that is composed of several line segments. But unlike a 'continuous' Line, the segments form one single line so clicking on one will select all of the other segments as well. While the segments of a Continuous line are joined together, each segment is a separate entity that can be individually selected.

1. Select the **Polyline** tool.
2. Click to start, then click to add more segments. Dimensions are not specified for a Polyline.
3. To end, press **Enter**. This forms an 'open' Polyline.

4. Select the **Polyline** tool.
5. Click to start, then click to make four more lines in a rough circle shape.
6. Enter **C** to 'close' the shape (reminder: watch the Command line for prompts and options at all times). A closed shape is formed.

Arc (center point) (Draw >> Arc)

The arc tool is a very important tool used for plotting site surveys. You must learn to use this tool very well.

1. Select the **Arc** tool.
2. Type **C** then **Enter** (this specifies that you want to draw an arc staring from the center).
3. Click at the center point of the arc.
4. Drag the cursor out to where you want the arc to start. Remember that arcs draw in an anti-clockwise rotation.
5. Enter the radius dimension **6'** [**1800**] then **Enter**.
6. Drag out the arc and click to complete it. Press **Escape** to release the tool. Practice with the arc tool many times with different radius distances.

Arc (appended)

There is another method for drawing arcs but it is not used when plotting site surveys. However, it may be useful for joining arcs together to create scalloped shapes or undulating lines.

1. Select the **Arc** tool.
2. Type **A** and press **Enter** (specifies that you want the arc appended from an existing line or arc).
3. Click on an existing line or arc. ESnap is not required because it will automatically snap to the nearest end of the selected line.
4. Move the cursor to form the arc then click to set it.

Arc (three point)

This method draws an arc through three points. You cannot use this arc to plot a site survey.

1. Select the **Arc** tool.
2. Click to start the arc.
3. Click in two more places to define the points the arc will pass through.

Practice will all three arc tools because they are very useful. Landscape designs often based on curves and arcs so being familiar with all the arc method will ensure that you can draw on paper what you visualize in your mind.

Circle (center point) (Draw >> Circle)

Draw a circle based on the center point.

1. Select the **Circle** tool.
2. Click where you want the center of the circle to be.
3. Type in a radius distance, **7'-2"** [**195**] then press **Enter**. The circle will draw automatically.

Circle (3 point)

Draw a circle that passes through three points.

1. Select the **Circle** tool.
2. Type **3P** then **Enter**.
3. Click to define the first point, then define 2 more points by clicking. The circle will draw automatically through all three points.

Circle (2 point)

Draw a circle by diameter distance.

1. Select the **Circle** tool (watch the Command window).
2. Type **2P** then **Enter .**
3. Click to start the edge of the circle.
4. Enter a diameter dimension or drag to size, then click.

Like arcs, circles are used a lot, so practice all three methods. Remember that all methods can be used with ESnaps.

Ellipse (Draw >> Ellipse)

1. Select the **Ellipse** tool.
2. Click to start.
3. Type in a distance, or drag, then press **Enter** to define the length of the ellipse (the X axis).
4. Rotate and move the cursor in or out from the center to draw the ellipse, then click to finish.

Ellipse Arc (Draw >> Ellipse Arc)

This tool draws an arc based on an ellipse.

1. Select the **Ellipse Arc** tool.
2. Click to start.
3. Type in a distance, or drag, then press **Enter** to define the length of the ellipse (the X axis).
4. Rotate and move the cursor in or out from the center to draw the ellipse, then click to draw the ellipse.
5. Move the cursor and click to define a start point for the arc.
6. Move the cursor again to define the end of the arc then click to finish.

Point (Draw >> Point)

A point is a marker used to temporarily or permanently mark a precise location.

1. Select the **Point** tool.
2. Click where you want the point. Use ESnaps for accurate placement.
3. Click as many times as needed to place multiple points then press **Enter** to release the tool.

NOTE: The shape and size of the point is set in Tools >> Options >> Drawing Settings >> Points. You have already set up Point options in CAD 1 when you created the templates.

<u>Spline</u> (Draw >> Spline)

A spline is a curve that passes through several points. It is used a lot in landscape design to create informal curved paths and planting beds. A spline can be 'open' and create a curvy line, or 'closed' and create a curvy shape.

1. Select the **Spline** tool.
2. Click to start the curved line.
3. Click several more times to create a curvy line. *You use the ESnaps if you want the line to follow a certain predefined path or to connect to existing entities.*
4. When the line is complete press **Enter** (but do not press Escape yet).
5. After you press Enter you will see a 'tail' connected to the start of the line. Move the cursor and the beginning of the curve will change shape. Click when it looks correctly aligned.
6. Align the curve at the other end in the same way and click to finish.

This creates an 'open' spline.

Now draw a 'closed' spline.

1. Select the **Spline** tool.
2. Click to start the curved shape, then click more times to create the shape you want. Do not click back onto the start of the line to close the shape but stop well short - see bottom left.
3. Type **C** then **Enter**. The shape will close.
4. Use the 'tail' to adjust the curve (see bottom right) then click to finish.
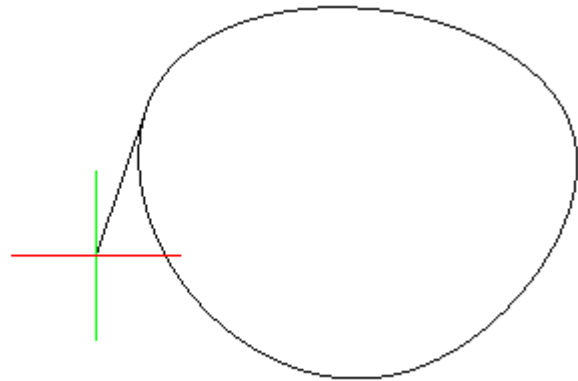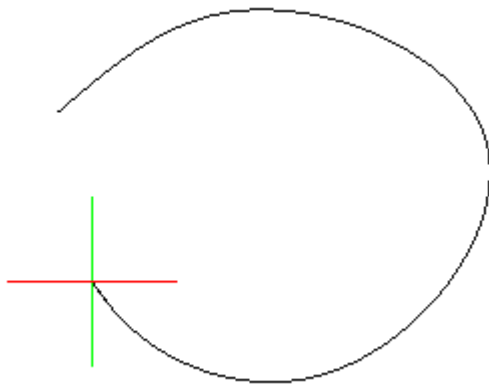
<u>Hatch</u> (Draw >> Hatch)

A 'hatch' is a pattern of lines used to fill a space. Hatch fills can only be applied to closed spaces. At times it is necessary draw a temporary outline to contain a hatch fill. This occurs if the area to be hatched is not closed, or behaves as though it is not closed as is the case with some seemingly closed shapes. The temporary outline is deleted once the hatch is drawn. Although there are several hatching methods, only the most useful one for landscape design will be shown.

1. Using the **Rectangle** tool, draw a rectangle **10' x 12'** [**3000 x 3600**].
2. *Click the* **Zoom Fit** *tool so that the entire drawing sheet fills the drawing area. You don't want to be too close to an area that you are hatching otherwise it is difficult to choose the best hatch scale.*
3. Select the **Hatch** tool and the hatch window will open. Below is a screen shot of the Hatch window. The circled options are the only ones that you will be concerned with for now.
4. Click the button with the **3 dots** to open the Pattern window.
5. Select pattern **ANSI31** then click **OK.**
6. Leave the **Angle** at zero and **Scale** at 1. Click the **Specify points** icon.
7. Click inside the rectangle then press **Enter**. The hatch window will return.
8. Click the preview button. Does the hatch fill look like solid black? If so, press the **Escape** key and you will be returned to the hatch window where you can make adjustments.
9. Type **8'** [**2400**] into the **Scale** box then press the **Preview** button.
10. Now the hatch fill should look fine, so **right-click** to finish
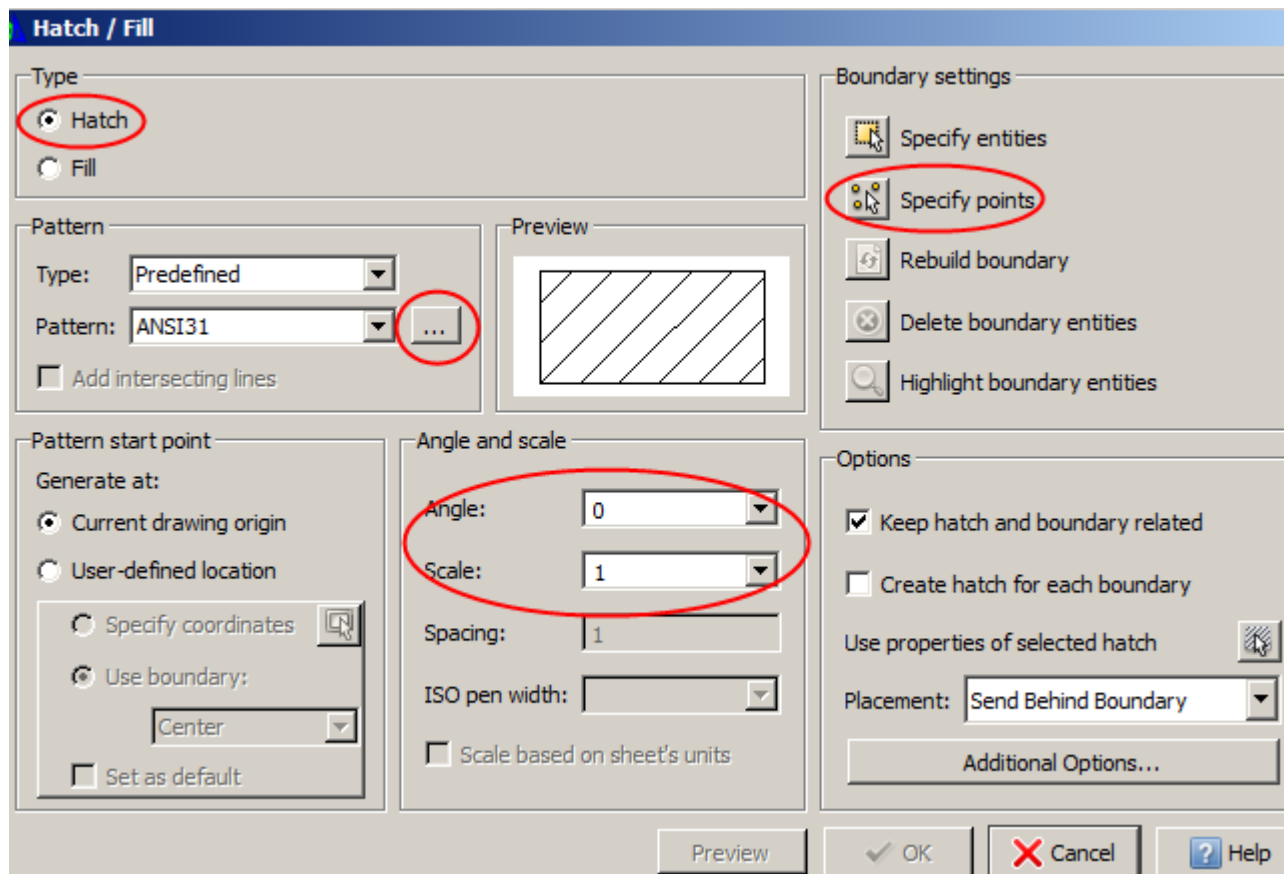
Step 8

Step 10

More about hatching on the next page.

The procedure for hatching is to (1) select the hatch pattern, (2) select the area to be hatched, (3) preview it, (4) make adjustments to scale and angle, then (5) finalize it. It seems complicate at first but becomes quite automatic with practice.

Hatch Scale

Hatch scale refers to the spacing between the lines that make up the pattern, not to actual drawing scale. In the example on the previous page you may have noticed that the scale for metric users was 25 times bigger than the scale for Imperial users yet the appearance of the finished hatch fill is the same. This has something to do with how the hatch fills were originally created and that 1 inch is roughly equal to 25 millimetres. However, the size of the number is not important just as long as you remember that the larger the scale the more open the pattern will appear. You simply increase or reduce the scale until the pattern looks as you want it to. Hatch scale is related to drawing scale. A good starting scale for hatch fills when the drawing scale is 1:96 [1:100] is **8'** [**2400**]. See how it looks and adjust as necessary.

The Scale box will retain the last used scale so you may not have to alter it every time.

Angle

The angle box lets you rotate the pattern within the shape by typing in a specific angle. So a hatch fill that is horizontal to start with can be made vertical by typing 90 (for 90 degrees) into the Angle box, or made diagonal by typing 45 in the angle box.

1. Draw another rectangle the same as before.
2. Select the **Hatch** tool.
3. Click the **...** Button then select the ANSI hatch fill called **ANSI32**.
4. Type **8'** [**2400**] in the Scale box the click the **Specify points** icon.
5. Click inside the rectangle then press **Enter**.
6. Click the **Preview** button. The pattern should like the one shown right.
7. Press **Escape** to return to the Hatch window.
8. Type **45** into the **Angle** box then click **Preview**. Now the hatch should look like the lower picture.
9. Right-click to finalize the hatch and release the tool.
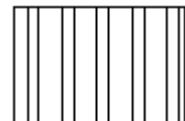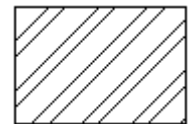
You can go through the 'Preview and Escape' sequence as many times as you like until the hatch fill is adjusted to appear exactly as you want.

Hatching is very important so practice this a lot.

NOTE: Hatch fills should ideally be drawn on the Hatch layer but the actual outline should be on Layer 0. So the sequence would be:

1. Draw the outline on Layer 0 then switch to the Hatch layer
2. Draw the hatch fill, deselect pressing the **Escape** key, then return to Layer 0.

Revision Cloud tools

The revision cloud is useful for indicating large areas of planting easily and quickly. I find the Freehand one the best but all are useful. They take a bit of practice to master but are really quite simple.

There are four revision cloud tools.

Cloud setting ✎
Rectangular ▭
Elliptical ◯
Freehand ✧
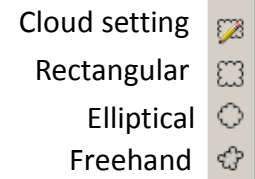
Cloud Setting

This is where you set up the cloud options. The setting most used is the Radius setting. This sets the size of each arc. For best appearance the arcs should be quite large. It is best to try out various radius options to see what looks best to the specific application.

1.   Click the **Cloud Setting** tool.
2.   Select the **Shape Option** you want then type a radius into the Radius box.
       Start with 1ft [300mm] but you might want to change it once it is drawn. Click **OK**.
3.   If you selected the Rectangular option, click in the drawing to start, move the cursor to the opposite corner of the rectangle then click to end.
4.   If you selected the Elliptical option, click in the drawing to start, drag the cursor out to form the long axis of the ellipse then click, then move the cursor to form the desired oval shape. Click to end.
5.   If you selected the Freehand option, click to begin the shape, click various points to define the shape then press Enter to finish. The shape will close automatically when you press enter so the last click does not have to be on top of the start of the line.

The Radius setting will not stay for all shapes until you change it again.

Rectangular Cloud

See step 3 above.
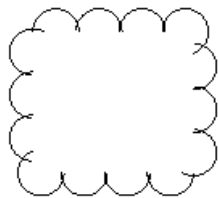
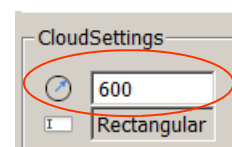Elliptical cloud

See step 4 above.

Freehand Cloud

See step 5 above.

Changing the radius of an existing cloud

1.   Click the cloud to select it.
2.   In the Properties Palette, type in a new radius then press Enter. The radius of the cloud will change.
3.   Press Escape twice to deselect the cloud.

DIMENSION TOOLS

While most distances can be measured from a drawing using a scale ruler, sometimes it is necessary to write distances directly onto the drawing. The dimension tools are used for this.

The Dimension toolbar may be open and floating, or docked to your interface, or you can access the tools through the Dimension menu. **Macs:** The Dimensions toolbar will be on the Tool Matrix.

Make sure to use the ESnaps when applying dimensions otherwise the resulting distances will not be accurate.

Open a **1:96** [**1:100**] template. Before you start practicing with the dimension tools draw a few shapes such as a line, rectangle, triangle and a circle. Size is not important but keep them roughly 4' to 8' [1200 to 2400] long/across.
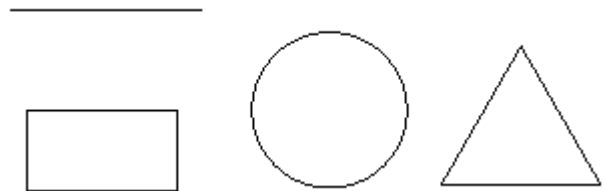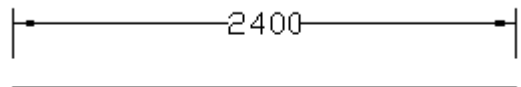
NOTE: All the templates you created and saved in the last module are already formatted for dimensions so no further dimension set up is necessary..

Linear (Dimension >> Linear)

1.   Select the **Linear** dimension tool.
2.   Snap to the left end of the straight line, then snap to the right end.
3.   Move the cursor up or down to position the dimension then click to finish.

Aligned (Dimension >> Aligned)

1.   Select the Aligned dimension tool.
2.   Click on the top corner of the triangle, then click on the lower right corner.
3.   Move the cursor in or out to position the dimension then click to finish.

NOTE: Use the Aligned dimension when you need to show the length of angled lines such as shown below left. The dimension on the right is incorrect because the
Linear dimension tool was used.
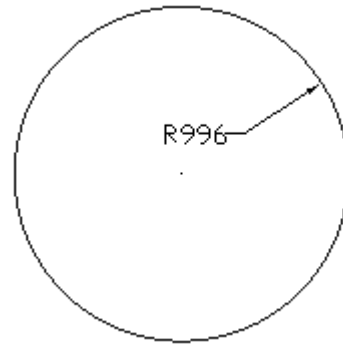
wrong

correct

2317

1158
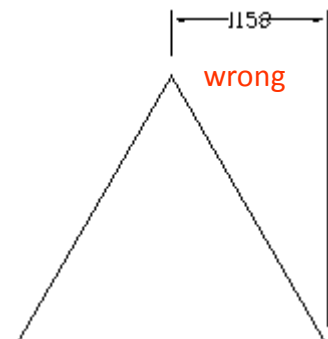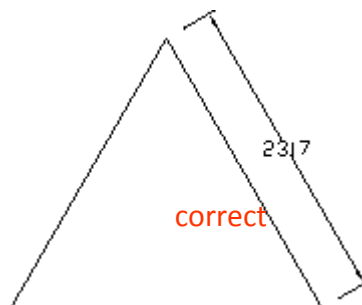
Radius (Dimension >> Radius)

The 'radius' is the distance from the center of the circle to the edge.

1.   Select the **Radius** dimension tool.
2.   Click on the edge of the circle.
3.   Position the dimension then click to finish.

You can place the dimension inside or outside of the circle.

Diameter (Dimension >> Diameter)

The 'diameter' is the distance across the entire circle at the widest point.

1.   Select the **Diameter** dimension tool.
2.   Click on the edge of the circle.
3.   Position the dimension then click to finish.

The radius and diameter dimensions appear the same at first glance but you will notice that the radius distance is preceded by the letter 'R' and that the diameter distance is preceded by a symbol that denotes 'diameter'.
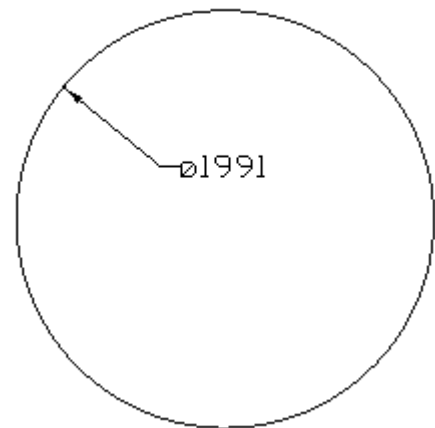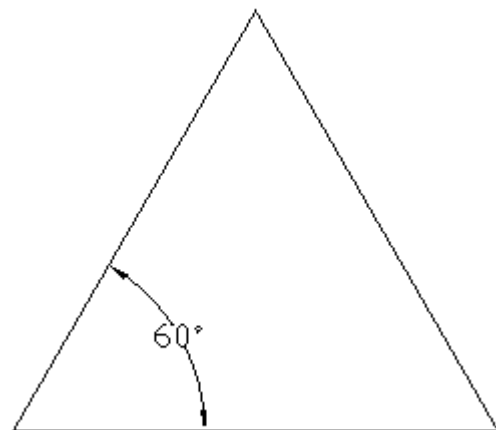
To prevent confusion it is best to dimension circles only with the Radius tool and not use the Diameter tool.

Angular (Dimension >> Angular)

This dimension tool shows the angle between two lines.

1.   Select the Angular dimension tool.
2.   Click on each of two adjoining lines of the triangle.
3.   Position the dimension then click to finish.

Arc Length (Dimension >> Arc Length)

This dimension shoes the actual length of an arc. It does not work on circles, just arcs. Begin by drawing an arc.

1.   Select the **Arc Length** dimension tool.
2.   Click on the arc.
3.   Position the dimension then click to finish.

The dimensions discussed represent all the most useful dimensions. If you want to learn more about dimensions and dimension setups you should refer to the user Help.
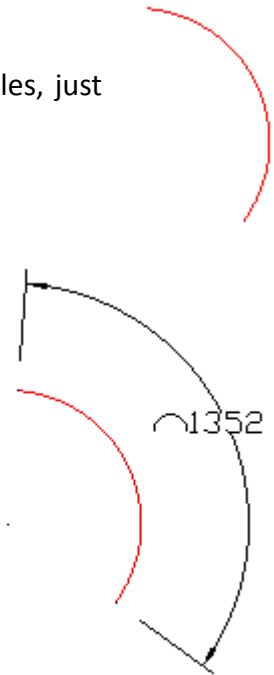
Leader tool

Technically not a dimension tool but it comes under the same group, this tool is used to make the arrowed line pointing a line of text to an object.

1.   Click the **Leader** tool on the Dimension tool bar or click **Dimension** >> **Leader** from the Menu bar.
2.   To make neat leaders, turn on **Polar**. Turn off Ortho and Esnap.
        Leader segments should be either horizontal, vertical or 45 degrees.
        A two-part leader is often used.
3.   To make a two-part leader:
        - click to start the line, draw the first segment out at 45 degrees, then click
        - draw out the horizontal segment, click then press Escape to finish.
4.   To make a one part leader:
        - click to start the line, draw out a horizontal line, click then press Escape to finish.
5.   Use the **Simple Note** tool to write the label then move it into place.
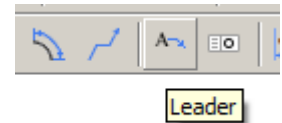
NOTE: The arrow size is set up in the template and is the same as the settings for other types of dimensions. You have already set these up when you created the templates.
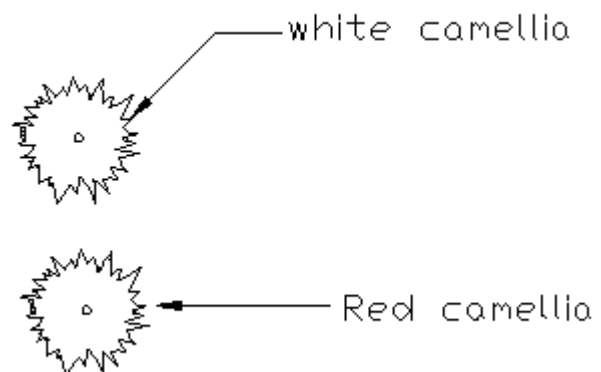
BLOCKS

A 'block' (sometimes called a symbol in other CAD programs) is a complex object that can be stored in a file and used again and again to save repetition when drawing. The most commonly used blocks are those that represent plants. Other commonly used blocks represent landscape features and others are used to improve presentation of a drawing. One block you are already familiar with is the 'titleblock' that you created in CAD 2.

DraftSight does not come with a library of useful blocks so a comprehensive block library has been included with your study material. If you look inside the Additional Material folder you will find a folder called My Block Library. To make the blocks easier to access it is recommended that you copy this folder into the DraftSight folder.

Follow these instructions. Shut down DraftSight before you begin.

Copying the Block Library into the DraftSight folder. **Windows**

1. Open the downloaded folder called **Part 1**.
2. Find the folder called **My Block Library** and **COPY** it onto **Desktop**. You can now close the **Part 1** folder.
3. Click the **Start** button, then click on **Computer**.
4. Double-click on **Local Disk (C):**
5. Double-click on **ProgramData** (not Program Files) - see below if you can't find it.
6. Double-click on **Dassault Systemes**.
7. Double-click on **DraftSight**.
8. The DraftSight folder should now be open. Copy or drag the **My Block Library** folder into the **DraftSight** folder.

ProgramData file: If the ProgramData folder is not visible then it is 'hidden' on your computer. Refer to the Additional Materials folder and find the document **Unhiding files and folders.pdf**. Follow the instructions then go back and start from step one above again.

The DraftSight folder should already contain a folder called Examples and one called My Titleblocks. Don't worry if the last file in the list shown right is not there or if it has a different name.

Now close the DraftSight folder and go back to Desktop. If it is still on Desktop, you can now delete the folder called My Block Library.

Name ▲
📁 Examples
📁 My Block Library
📁 My Titleblocks
📄 ProxyServerConfig.xml
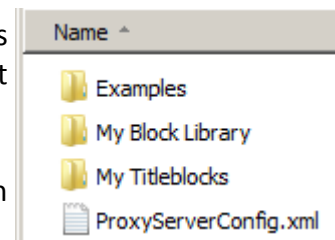
Copying the Block Library into the DraftSight folder. **Macs**

1. Follow steps 1 and 2 above.
2. Open your **Documents** folder.
3. Open the **DraftSight** folder. You created this in CAD 2.
4. Drag or copy **My Block Library** into the **DraftSight** folder.
5. Delete the unwanted My Block Library folder from Desktop.

<u>Using blocks</u>

Blocks are mini drawing files that contain just one block. Blocks are located via the **Insert** menu.

Start DraftSight and open a NEW 1:96 [1:100] drawing template.

1. Click **Insert** >> **Block** to open the Insert Block window.
2. Click the **Browse** button. This is how you locate the required block library.
3. Find **My Block Library** - it will be inside the **DraftSight** folder.
4. Open the **My Block Library** folder. You will see a list of sub-library folders.
5. Open the **Plants planview** folder. What you should now see is a long list of block files (partly shown on right)
6. Click (do not double-click) on **Fern 1** and you will see a preview in the right-hand preview pane. **Macs:** Preview not available at this time, unfortunately.
7. Using the **down-arrow** key to scroll down the list.
8. When you get to **Shrub 02**, click **Open**.
9. Back in the Insert Block window, you can set up options before the block is placed into the drawing. In the **Position** section, select **Specify Later**. This option allows you to manually place the block in the desired location.
10. In the **Scale** section, select **Apply Uniform Scale**. This means that the block will retain the original X and Y (width and height) ratio and will not become elongated.
11. If you use Imperial units leave the X scale at 1. This retains the original size of the block. If you use **metric** units change the **X** scale to **25**. This converts the block to metric size and is necessary for metric users (*This is because the original blocks were created with Imperial units and if you don't specify a 25 scale then the block will be 25 times smaller than it should be*).
12. Click **OK** then click on the drawing sheet to place the block. Press **Escape**.
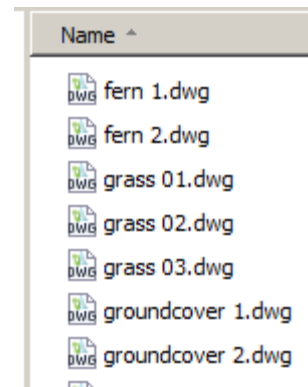
If you want more of the same block then use the Copy tool instead of using the Insert Block function.

The My Block Library holds seven sub-libraries and you should be able to find a block for most purposes. But if not, you can make more quite easily.

<u>Grouping Entities</u>

Separate entities can be 'grouped' so they behave as one single entity by using the Make Block tool. Use this tool whenever you want to combine entities together, such as the entities that make up the sheet border and title block.

1. Select all of the entities that are to be part of the same group (block).
2. Follow steps 5 to 8 on page 20.
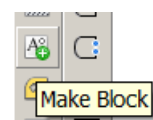
Changing the size of a block

If you specify the block scale as described on the last page, the blocks will come into the drawing at a real size. The block Shrub 2 will be about 4' [1200] across. Perennials and trees, cars and people, etc will all arrive into the drawing at a realistic size. You can check the size of a block with the dimension tools if you are not sure.

If you wanted to increase or reduce the size of any block it is best to do it once it is in the drawing. To do this you should use the Scale tool (see CAD 4).

However, you can also change the scale by specifying a different value in the block Scale setup. For instance, in the previous exercise you entered 1 [25] for the Scale to insert the block at the original size. If you specify 2 [50] for the block scale then it would arrive in the drawing at twice the original size.

TIP: Since the setup for one block insertion remains for the next one until changed, it is best to change the size after the block has been placed in the drawing so that you won't accidentally bring in subsequent blocks at the wrong size. Even so, you should always check the block scale.

Properties of blocks

The properties of a block (line color, style and weight) can be changed in exactly the same way as for other entities.

Altering a block

Block can be altered but you must explode them first. Use the shrub block for practice.

1. Click on the shrub block then click the **Explode** tool. The block will be exploded into its component entities.
2. Draw a selection rectangle over the lower part of the exploded shrub.
3. The enclosed entities of the shrub will be selected. Press the **Delete** key. Some of the shrub will now be gone but be careful because the remainder is a whole bunch of little bits.
4. Use the selection rectangle method to select the entire remaining shrub.
5. Select the **Make Block** tool. Type in a name for the block. Make sure the **Convert to Block** option is selected.
6. Click the **Select in graphics area** icon on the left.
7. Click in the middle of the shrub (or the lower left corner if you prefer) to specify the block insertion point.
8. Click **OK**. Nothing further will happen but if you click on the shrub now it will move and behave like a block.

Saving a block to the block library

New or amended blocks can be saved to the block library. Any compound object can be converted to a block following steps 4 to 8 on the previous page. You can also refer to CAD 2.

Once a block has been created in the drawing you can save it to the library. For this exercise use the half shrub block you made earlier.

1.   Click **File** >> **Export** >> **Export Drawing**.
2.   In the **Source** section select the **Block** option.
3.   From the drop down list select the block you wish to save. Do you remember what you named it? If not, use the preview to the right of the window to help you select the right block.
4.   Click the **Browse** button.
5.   Find and open the **My Block Library** folder, then open the **Plants planview** folder.
6.   Click the **Save** button then **OK**.

Now you can check if the block has been saved into the library.

1.   Click **Insert** >> **Block**.
2.   Find the **Plants planview** folder and insert the new block as you did earlier.

Metric users: You will notice that the new block is coming into the drawing at 25 times bigger than before. Because this block was created with a metric setup the 25 times scaling is not required when you use it. Therefore enter 1 for the Scale instead of 25. Try the procedure again with a scale of 1.

TIP: If you amend blocks in a metric environment it is advisable to add "m" to the name of the block to denote a metric block. Then you will know to specify a scale of 1 instead of 25 when you insert it into a drawing.

If you are confident about exploding, making and saving blocks you could convert all the blocks to metric. Time-consuming but it maybe a worthwhile way to occupy yourself during the slow season. The procedure would be:

1.   Insert a block.
2.   Explode it.
3.   Make into a block again with a new name (such as 'shrub 2 m').
4.   Save it in the block library.

Adding a new block library folder

If you want to make your own saved blocks it might be better to save them in your own special block library. You can add as many folders as you like inside the DraftSight folder.