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UNIVERSITY OF NEW SOUTH WALES
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TUTORIAL INTRODUCTIONS TO CAD

AUTOCAD 2000 – A RESIDENTIAL SITE PLAN

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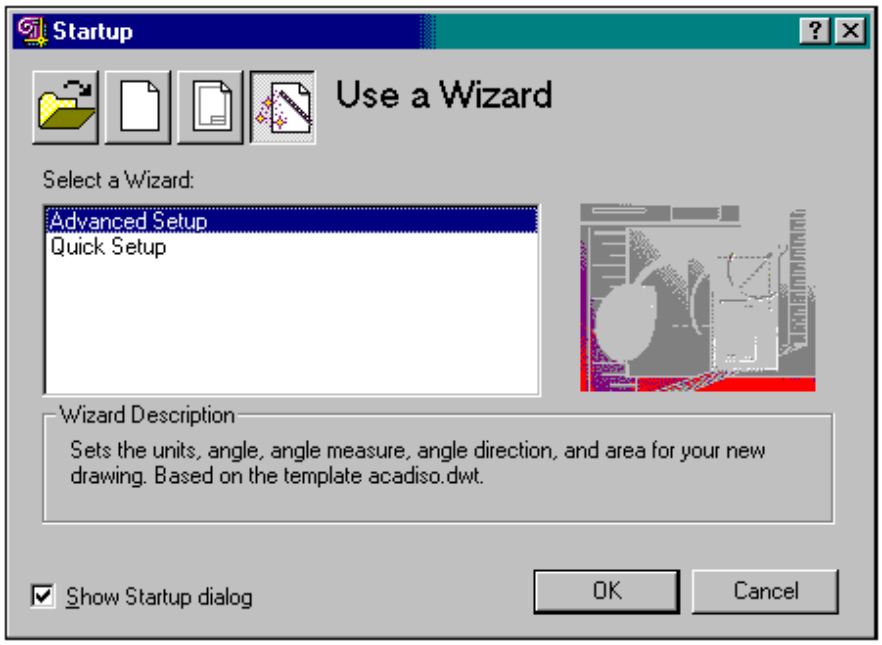
This is the second in a series of tutorial introductions to CAD. The general aim of these tutorial guides is to give you a firm conceptual understanding of CAD, including the scope of techniques available for design and presentation, and strategies for approaching a variety of drawing tasks. This tutorial concentrates on 2D drawing, using a simple residential site plan to explore the range of basic techniques for producing and locating accurate linework. It ignores concepts like layering, but does introduce most drawing and editing tools and the manipulation of object properties. The final drawing is shown at the end of this handout.

These tutorials are written for *AutoCAD 2000* running under the *Windows 95/98* operating environment. Some of the later tutorials make use of enhancement packages that are sold in conjunction with AutoCAD to address specific disciplinary CAD modelling tasks. This tutorial was developed using the Architectural Desktop package, but only uses AutoCAD 2000 features and tools.

GETTING STARTED

Architectural Desktop 2.0

Start up AutoCAD 2000. In the *Startup* dialogue box, use the *Wizard* with the *Advanced Setup* option as shown in the following dialogue box to create a new drawing. This option will walk you through the initial steps necessary to set up a new drawing, showing you each of the standard settings that AutoCAD uses. Some of these will suit us, while others we will wish to change.



Drawing Units

Decimal units with precision 0.

Click the *OK* button to go to the first screen that allows you to set up the units to be used in your drawing. If you select each alternative in turn while watching the small graphic image, you will quickly appreciate why we use the first option labeled *Decimal*. Re-select *Decimal* and set the precision to 0 since it is meaningless to work with building geometry in fractions of a millimetre. Then click the *Next* button to move to the next screen.

Angle Measurements

Decimal angles with precision 0.0

Again, check out each of the options here, particularly noting AutoCAD's interpretation of surveyor's units. In Australian practice, surveyors express bearings in degrees (0-360) measured clockwise from north. We may be tempted to use the second option (degrees, minutes & seconds) but that is only typically used for site bearings, while we will use decimal measures in all other drawing actions. Select Decimal angles with a precision of 0.0 and then click *Next* to continue.

Base Angle

North

This option allows us to review the AutoCAD convention that angles are measured relative to 0 which is assumed to be an *Easterly* direction. This follows the normal computer graphics convention where the positive X-axis is taken as zero. I normally recommend not changing this convention, but since our first task in this drawing is to draw the site boundaries (which are normally read directly off a survey plan) select the *North* option to make angle 0 "up the page". Then select *Next* to continue.

Angle Direction

Clockwise

The normal AutoCAD practice again follows computer graphics convention where the counterclockwise direction is taken as positive. This conforms to the "right-hand-rule" where, if the thumb is pointing in the positive direction along the axis, then the fingers point in the direction of positive rotation. Again, I normally leave this option at the AutoCAD default of counterclockwise, but on this occasion, since surveyors measure boundary bearings clockwise from North, select the *Clockwise* option before clicking *Next* to continue.

Drawing Area (Limits)

60000 x 60000

Since our site is about 45 metres long, it would be sensible to make our drawing area about 60000 mm square. Notice from the example provided that AutoCAD assumes we work in metres, while the Australian convention is to work in millimetres. Set both values to 60000 and then click *Finish*. That completes our drawing setup and AutoCAD displays the normal editing screen.

As you move the mouse around the screen, note the coordinates displayed in the lower left corner: AutoCAD is still only displaying the default drawing area of 420x297. We use the *Zoom All* command to expand the view to "see" the whole drawing area.

Zoom All Tool

[Standard. Zoom All]	Issue the command *Zoom* with the option *All* - check out the *Command Window* to see the way AutoCAD prompts you as you enter this command and then enters the response by itself!! *Remember, the Button Tools can be programmed to execute several commands, including all the necessary responses in order to perform quite complex operations*.

Save the Drawing!!

Click the *Save* button and then navigate to your student folder and save this drawing under an appropriate name.

DRAWING THE SITE BOUNDARY

The final drawing on the back sheet of this handout shows the boundary bearings much as they would be shown on a typical survey plan. The lengths are shown in metres, and the bearings in degrees, minutes and seconds relative to North (which is also assumed, for the sake of this exercise, to be off to the right rather than exactly "up-the-page"). Take a moment to study those

bearings, making sure that you understand each one. Naturally, the value of the bearing depends on which end of the boundary line is considered to be the "start". In this fictitious site I've deliberately mixed that up a little! North on this drawing is actually 23^{o} off towards the right as shown.

Rotating the Drawing Axes

User Coordinate Systems (UCS)

All graphics objects in AutoCAD are modelled in relation to a fixed ***World Coordinate System*** *with a global origin at the bottom left corner of the drawing area and the X-axis pointing to the right and the Y-axis pointing directly up the page.*

At any time, a user can set up a temporary ***User Coordinate System*** *by moving the origin and/or rotating the axes. All data is then entered relative to that temporary axis system, but is converted automatically and stored in World Coordinates.*

The easiest way to draw the site boundaries is to begin by rotating the drawing axes so that the Y-axis ("up-the-page") is pointing in the exact direction of North on the plan. Remember that we have already altered the AutoCAD convention (temporarily) so that "up-the-page" is considered 0^{o} and angles are measured clockwise. Therefore, all we have to do is rotate the axes through 23^{o} while we draw the site boundaries and the north arrow. We do that by temporarily setting up a new User Coordinate System (or UCS).

Tools > New UCS > Z	Use the pull-down menus to issue the *New UCS* command with the option to rotate the axes around the Z-axis (which is an imaginary axis that comes out of the screen).
23 ↵	Type the value of the angle of rotation followed by the *Enter* key to complete the command. Notice that the UCS icon in the lower left corner of the screen has changed to indicate the new axis direction.

We now use the *Polyline* command to draw the four boundary lines. A polyline is an entity that is made up of a series of straight line or arc segments but is treated as a single entity in AutoCAD. If you use the *Line* command, each boundary line will be treated as a separate entity. Although that does not matter particularly, it seems sensible to make them one object.

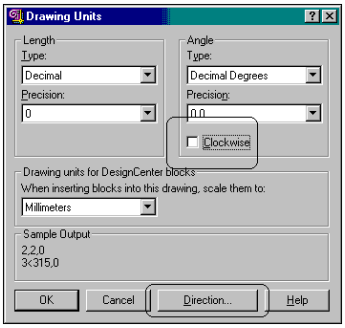

Polyline Tool

[Draw.Polyline]	Select the button illustrated from the *Draw* toolbar … notice the prompt on the Command Line … pick a starting point with the mouse pointer near the bottom of the drawing area about one-third along from the left.

To enter the four boundary lines, we use AutoCAD's coordinate entry technique. In this case the coordinates are *relative* (indicated by the prefix "@") *polar* values. That means that we specify the length and angle of each line (separated by the "<" symbol) relative to its starting point. While we would normally specify angles in decimal form (eg 327.3), it is convenient here to use the "degrees/minutes/seconds" format since we can read that directly off a survey drawing. Notice also that the third entry is a "back bearing". That is, the original bearing was taken from the SE corner of the site whereas we are drawing the line from its other end. Rather than re-calculate what the bearing should be, we simply enter the length as a negative number so that AutoCAD constructs the line in the correct direction. Finally, the last line is drawn using the "close" option (abbreviated to "c"). This ensures that the polyline closes correctly. If we wished to test the surveyor's accuracy, then we could draw the last line (**@-19400<67d**) and measure the distance between the two points!

@37258<327d19'48" ↵
@26654<51d19'48" ↵
@-43928<337d ↵
c ↵

Before proceeding, we need to revert back to AutoCAD's "normal" convention of measuring angles counterclockwise from 0 taken as an easterly direction. We do this because all the other tutorials follow that convention and we may as well get used to it!



Format > Units	Select from the pull-down menus … in the resulting dialogue box (illustrated at left) de-select the *Clockwise* checkbox and then click the *Direction* button to open another dialogue where you can select the *East* option … then click OK to dismiss each dialogue in turn.

Before going any further, we need to turn off some of the AutoCAD snapping tools so that they don't confuse you as you learn the basic techniques first. Look at the row of *Toggle Buttons* on the *Status Line* across the bottom of the AutoCAD Window as illustrated below.

You should simply go along and click each of the first three buttons that are currently "depressed" (that is, turned on) so that each appears as a raised button and therefore is inactive. These are POLAR, OSNAP and OTRACK. Do not turn off the MODEL button at this stage. We will progressively turn each of these back on again as we learn how each one works to make drawing easier.

Drawing the North Arrow

Now, with the axes still rotated through 23^{o}, we can construct the North arrow symbol. We will use some more examples of coordinate entry to illustrate that technique and then have our first taste of using the *Text* tool to write the word "North". We will also introduce *Object Snapping*.

Line Tool

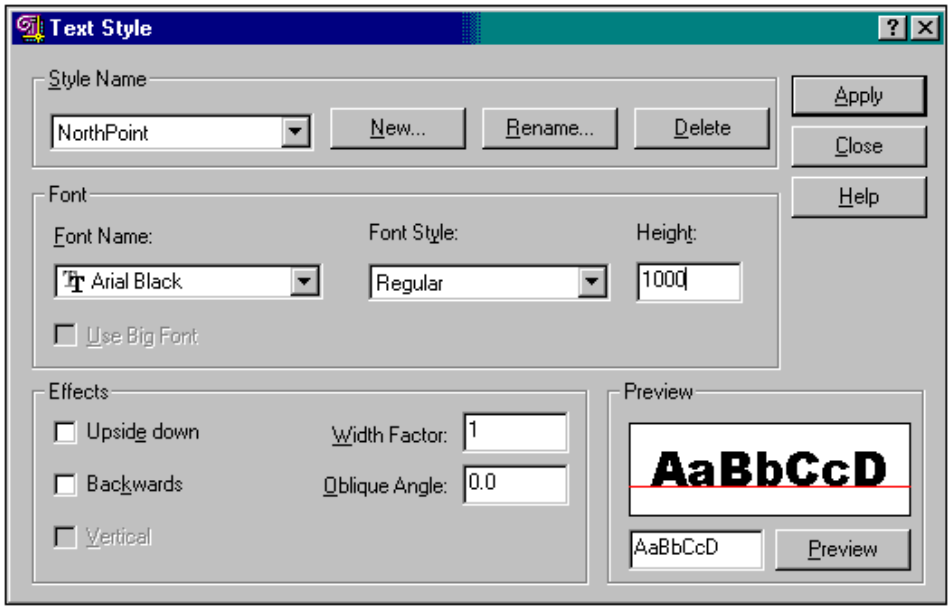
[Draw.Line]	Issue the *Line* command from the *Draw* toolbar … pick a suitable starting point for the lower end of the arrow line.
@0,7000 ↵	Type the coordinates at the prompt in the *Command Window* … this time it is a *relative cartesian* coordinate, telling AutoCAD to draw 0 units in the X-direction and 7000 in the Y-direction relative to the current (rotated) UCS.
@-1300,-1300 ↵	Another relative cartesian coordinate, but using negative values this time to construct the line we want.
[Osnap. Perpendicular]	Select the *Perpendicular Object Snap* mode (illustrated) from the Osnap flyout toolbar (also illustrated) … this may be found in the Standard toolbar below the pull-down menus … if you then move the mouse pointer over the first line we constructed in this sequence a yellow snap symbol will appear indicating that AutoCAD is ready to snap to that line using the *Perpendicular* snap mode, thus creating a line exactly perpendicular to the line selected … click the left mouse button with the snap symbol visible to complete the snap operation.
↵	Press the *Enter* key to complete the command.

Object Snap Flyout on the Standard toolbar …

Perpendicular Snap Mode Tool in the Osnap Flyout shown above …

The next step is to create a new text style to write the word "NORTH". Select *Format > Text Style* from the pull-down menus to bring up the *Text Style Dialogue* box illustrated over the page. Once the dialogue is visible, click the New button to create a new style. Type a name for the style (I called mine "NorthPoint") and press OK. Then select a font from the pull-down list (I used *Arial Black* which is a "thicker" version of the normal *Arial* font and good for headings and the like). Finally, specify the height as **1000**. In AutoCAD text height is always expressed in drawing units relative to the size of real-world objects in your drawing. In selecting the text height, think about the likely scale to be used when plotting the drawing. In this case, we will be plotting at 1:200

scale, so this text will end up being 5 mm (1000/200) high on the drawing. You can always change text height at any time … it is simply a "property" of that text object. Note that you can change other features of the text by applying some quite unique "effects". It turns out that AutoCAD text is more versatile than many of the very sophisticated desktop publishing applications.



We will use the *Single Line Text* command to write this text. This command is NOT available on the toolbars, but may be accessed through the pull-down menus (*Draw* > *Text* > *Single Line Text*) or as a typed command word. We will use the typed command this time.

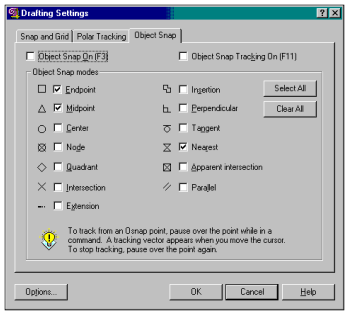
dtext ↵	Issue the *Single Line Text* command (also called *dynamic text*) … AutoCAD prompts for the starting point of the text.
 *Endpoint Object Snap tool* [Osnap. Endpoint]	Use the *Endpoint* snap mode to lock onto the lower end of the arrow line that we created.
[Osnap. Endpoint]	Use the Endpoint snap mode again, this time to indicate the direction of the text … snap onto the other end of the main arrow line … this means that the text will be constructed as shown in the final drawing … AutoCAD next prompts for the text itself …
NORTH ↵	Type the text required and press the *Enter* key.
↵	Press the *Enter* key to complete the command … the next step to move the text slightly so that it is just above the arrow line …
 *Move Tool* [Modify.Move]	Issue the *Move* command (illustrated) from the *Modify* toolbar … notice the prompt says "Select Objects" … click on the text object just created …
↵	Press *Enter* again to complete the object selection process … AutoCAD now prompts you for the first of two points to describe how you want to move the text object … click any position to act as the "base point" of the move … then move the mouse around until the text is neatly positioned in relation to the arrow and then click the left mouse button again.

That completes the arrow. Before moving to the next stage, you need to rotate the axes back to normal. You do that simply by setting the UCS back to "world" by selecting from the pull-down menus: *Tools* > *New UCS* > *World*.

This is a good time to Save your drawing ready for the next stage in this tutorial.

Labeling the Site Boundaries

Adding text to the boundaries is very similar to adding the text to the north arrow. The trick is lining up the text with the boundary line and, again, we can use snap modes to do that. Rather than enter each snap mode as we need it, AutoCAD allows us to set up "running object snaps" which offer themselves automatically as we get near the point. We will try that method now!



Place the mouse pointer over the *OSNAP* toggle button on the *Status Line* and click the right mouse button. In the pop-menu, select *Settings*. That will bring up the dialogue illustrated on the left. Make sure that the three snap modes *Endpoint*, *Midpoint* and *Nearest* have been selected as shown, and then click OK. Now left-click the *OSNAP* toggle button to turn on running object snap. Also note the symbol next to each snap mode in the dialogue box: these are used on the drawing to show when AutoCAD has found a suitable snap point.

Now, set up a new text style. I called mine "BoundaryText" and used the standard AutoCAD text font called *Simplex* (an old favourite!) and set the text height to 400. You can vary that if you wish for your drawing! Note that *Simplex* is one of the native text fonts in AutoCAD. These are vector fonts (no solid fill like the *Windows True Type* fonts) and are quick to draw and useful for general notes and annotation. They are easy to identify because they have a different icon than the *True Type* fonts.

Note that there is no toolbar tool for Single Line Text in AutoCAD as it comes shipped from the factory … the tool in the Draw toolbar is Multiline Text …

Issue the *Single Line Text* command (from the pull-down menus or type the command, **dtext**). Check that AutoCAD is going to use your new style by checking the prompt. If not, use the *Style* option to change the "current style". AutoCAD then prompts for the starting point of the text. Starting with the west boundary, move the mouse around noting that as you get near the ends of the line, the *Endpoint* snap locks in. Right at the midpoint of the line, the *Midpoint* snap will become activated. Otherwise, as you get near any other point on the line, the *Nearest* snap mode will operate. All these snap modes will lock onto the line, so you can pick any you like to lock the starting point on the line. When AutoCAD prompts for the text direction, again use the running object snaps to lock onto the line in the direction you wish the text to follow. When AutoCAD prompts for the text itself, enter the west boundary text as follows:

37.258 m 327%%d19'48"

Note the use of the string "%%d" to represent the degree symbol. This is a special code recognised by AutoCAD as part of its text entry features. There are other similar codes for representing special symbols that you can find in the Help system. Press *Enter* a second time to complete the command.

Now enter the remaining boundary labels as shown on the final drawing at the end of this tutorial.

DRAWING THE CONTOUR LINES

AutoCAD is surprisingly tolerant when supporting the input of coordinate information. We have already seen how coordinates can be typed at the keyboard for accurate data entry. Since contour information is often obtained directly from a Surveyor in electronic form, we are going to demonstrate how you can very easily import such data from any file that contains the data in some

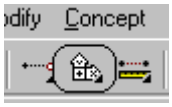
text form. Without actually knowing how this text data might come from a Surveyor, I have simply created a text file that contains the critical points that define the contours for this site. We are going to cut and paste that information into AutoCAD.

Open the file called "ContourPolylines.txt" in the following folder on the Faculty's Server:

Resources on 'Emuwater' (R:)\Samples\benv\CAD Files

In that file you will notice that I have created a list of the points that define each contour. The first point is expressed as a 3D coordinate where the Z-value is the height of the contour (above some local datum point). The remaining points are 2D coordinates. Although this is only a 2D drawing, it seems worthwhile to enter the contours at their correct height relative to each other as this would be useful if we wanted to use this drawing as the basis for a terrain model at some future time. The XY values of all these points are relative to a base point on the site. In this case, that base point is the lower left (SW) corner of the site, but in reality would be whatever datum the surveyor established for the site.

The following steps detail the process, but I begin with a quick overview. We begin by positioning the UCS at the base point and then use the Polyline command to draw the contours. Since polylines are 2D entities in AutoCAD, they are constrained to always lie in the same XY plane. That means that by specifying the height of the first point on the polyline, all the other points are forced to have the same Z-value. We then edit the polyline to fit a smooth curve through those points.



UCS Flyout

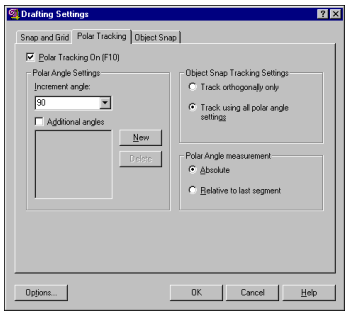

UCS Origin Tool


Polyline tool

[UCS.Origin]	Click the *UCS Origin* tool (illustrated) from the UCS flyout (also illustrated) on the *Standard* toolbar … move the mouse pointer to the SW corner of the site and AutoCAD will snap to that point because running object snap is still selected … the UCS icon will then be positioned at that point as a visual indicator of the location of the origin.
[Draw.Polyline]	Issue the *Polyline* command … AutoCAD will prompt for the Starting Point. Switch to the "ContourPolylines.txt" file (which you probably opened in Notepad) and highlight the full list of coordinates for Polyline 1 … press [**Cntrl-C**] to cut that text (or use the Edit menu) and then return to AutoCAD … be careful to click the mouse in the Command Prompt area so that you have a flashing text cursor on the command line and then press [**Cntrl-V]** to paste the text (*do not use the pull-down menu in this case as it will paste the text into your drawing as a text object!*).
↵	Press the Enter key to complete the command.
Modify > Polyline	Select from the pull-down menus … select the polyline just drawn in response to the prompt … note the long list of editing options that you can apply to a polyline …
f ↵	Select the *Fit* option … AutoCAD fits a smooth curve to the polyline, ensuring that it passes through each point.
↵	Complete the *Polyline Edit* command.


UCS World tool

Repeat that process to draw each contour. Then click the *UCS World* tool (illustrated) to switch back to world coordinate space. To do that you will need to pull down the UCS flyout because each time you use a flyout, AutoCAD places the most recently-used tool at the "head" of the flyout so in this case, it will be the *UCS Origin* tool that we used most recently!



Polar Tracking dialogue …

Polyline Tool

From tool in the Osnap flyout …

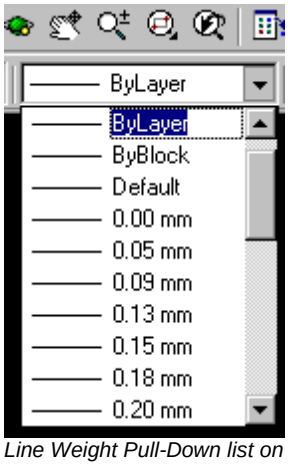
DRAWING THE BUILDING OUTLINES

Next, we will draw the main building outline along with the dotted gable roof lines. This will permit us to explore the use of polar tracking techniques for quick and accurate input of coordinate geometry.

Begin by setting *Endpoint* & *Intersection* as the running object snaps and then right-click on the POLAR toggle key on the Status Line (to open the pop-up menu) and select *Settings* to open the *Polar Tracking* dialogue (as illustrated). Enter values in the dialogue as illustrated, including the *Polar Tracking On* check box. Then click OK.

[Draw.Polyline]	Issue the *Polyline* command from the toolbars … AutoCAD prompts you for a starting point … we assume that we know the position of the SW corner of the building relative to the SW corner of the site (which we will take as the datum point for this site) … we can specify that position using the *From* snap mode as follows …
[Osnap. Snap From]	Select *Snap From* tool (as illustrated) in the *Osnap* flyout … use *Endpoint* running object snap to lock onto the SW corner point of the site …
@100,10000 ↵	Type this relative coordinate at the prompt … this tells AutoCAD to begin the polyline at this position relative to the point selected … now move the mouse until the tracking line is shown heading "right" from the starting point … leave the mouse in that position and type …
6000 ↵	This tells AutoCAD to draw the line this distance along the tracking line … next, position the mouse so the tracking line is "up" from the end of that line and then type …
2000 ↵	This draws the line this distance from the last position along the tracking line … repeat this process to enter each segment of the building outline: **4500** (right); **4000** (down); **7000** (right); and **10000** (up) … this takes us to the NE corner of the building … to draw the north face we want the line to end up in line with the SW corner of the house … to do that we use *Object Snap Tracking* in conjunction with *Polar Tracking* …
[OTRACK]	Click the *OTRACK* toggle button on the *Status Line* … now move the mouse pointer so that it passes over the SW corner of the building … the *Endpoint* snap symbol should appear at that point and as you move away again (moving "up") a tracking line should appear passing through that snap point … as you then move towards the position of the NW corner, two tracking lines should appear (one vertical from the SW corner and the other horizontal from the NE corner) with an *Intersection* symbol at their point of intersection … when you see that intersection symbol, click the left mouse button … the line should then be drawn to the point of intersection of the two tracking lines …
c ↵	Type "c" for close … this will complete the polyline by drawing a line back to the starting point.

That simple outline showed you how to use both *Polar Tracking* and *Object Snap Tracking*. It is important that you understand both techniques, as we will use them all the time from now on!



Line Weight Pull-Down list on right-hand end of the Object Properties toolbar above the drawing area.

In the final plotted drawing, we want this outline to be drawn with a heavier line weight than other objects in the drawing. We can do that by assigning a line weight property to it. First select the polyline by clicking on it with the left mouse button. It should appear as a dotted line with blue "handles" at each corner and at the midpoint of each edge (to show that it has been selected). Next, pull down the list of line weights as illustrated. Scroll down the list until you see 0.70 mm and click on that. The line will now be plotted in that thickness, though it appears normal on the screen. To preview lineweights on the screen, click the LWT toggle on the Status Line. I find the displayed line weights are often exaggerated and it is best not to have them displayed when working with the drawing.

Adding the Roof Form

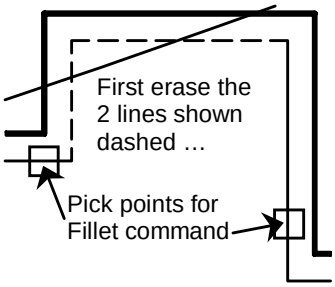
The next step is to draw the roof line offset 450 mm from the edge of the building outline.

	Select the *Offset* tool from the *Modify Toolbar* … AutoCAD prompts for an offset distance …
450 ↵	Enter an offset distance of 450 being the roof overhang … click on the building outline in response to the prompt … then point anywhere outside the polyline to show the direction of the offset … AutoCAD should create a new polyline 450 outside the original one.

Zoom Window tool in the Zoom flyout …

Note that the new offset polyline also has a line weight of 0.70 mm (inherited from the line used to create it). Change its line weight to "By Layer".

The next step is to simplify the roof shape over the entry area at the front of the house (compare your roof outline to the final drawing!). Use the Zoom Window command (tool button illustrated from the Zoom flyout) to zoom in on that part of the drawing. Using the figure on the left, follow these steps …



	Issue the *Explode* command (tool illustrated) and pick the roof polyline just created … this breaks the polyline down into separate line entities.
	Issue the *Erase* command and pick the two lines shown dashed in the figure on the left.
	Issue the *Fillet* command … this command joins two lines with a small fillet arc … note that the fillet radius is initially set to 10, while we want the radius to be 0 so that the two lines join at a point with no fillet arc.
r ↵	Select the *Radius* option to change the fillet radius …
0 ↵	Set the fillet radius to zero.
↵	Press *Enter* again to recall the *Fillet* command … this is a useful shortcut in AutoCAD … pressing *Enter* in response to the *Command* prompt always recalls the most recent command used … pick the two lines to be joined at the positions shown in the diagram
	Use the *Zoom Previous* command to get back to the previous view.
	Use the *Zoom Window* command to zoom in on just the outline of the main building just drawn.

The next step is to draw the roof ridge lines. If you set the *Polar Tracking* angle to 45^{O} and turn on **POLAR**, **OSNAP** and **OTRACK**, then you will find that you can create all those lines (see final drawing) without having to type any values, compute any geometry or use any editing commands. Use only the *Line* command with *Endpoint* snapping to lock onto existing points and the intersection of two tracking lines to position each of the ridge points.

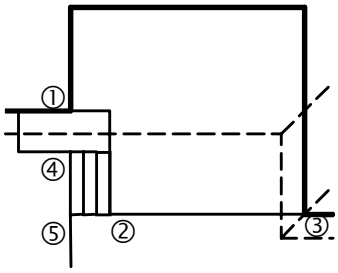
A note about the Global Linetype Scale Factor set in this paragraph …

I have never quite understood how the linetype scale factor works and always find the correct scale factor for each drawing by trial and error. If the scale factor is not set incorrectly, then the dashed lines will be either too short or too long to show on the drawing so the line appears to be continuous. I have found it best to start with a factor of about 10 and increase it upwards until the dashed lines appear correct. The "correct" value for this drawing turned out to be 25.

We now want to make all those lines into dashed lines. Begin by pulling down the list of line types and selecting "Other…". In the resulting dialogue box, click the Load button. This will open a dialogue with a list of pre-defined line types that can be loaded into the drawing. Scroll down the list and look at the range of styles available. For simplicity, I loaded all the line types from "BATTING" through to "ZIG ZAG" at the very end of the list. To do that, first select "BATTING" then scroll to the end of the list and hold down the Shift key while selecting "ZIG ZAG". That will highlight all those styles. Then click OK. Before leaving the *Linetype Manager* dialogue, set the *Global Scale Factor* to 25 (see lower right corner!). This will make your linetypes about the right size on your drawing. Click OK again to close that dialogue box.

Now select all the roof lines just created by left clicking them. When each one is highlighted, simply pull down the list of line types again, scroll down to the "Dashed" type and select it. That will change each of the roof lines to a dashed line as shown on the final drawing.

Adding the Front Porch




Perpendicular Snap tool


Copy tool

The next step is to draw the front porch and steps (as shown illustrated). Study the final drawing first, noting that the porch comes out to line up with the front of the house on the east side and has three steps leading down to the driveway next to a small garden plot adjacent to the house.

Draw a line beginning at point ① in the figure extending 750 to the east and then south to line up with the front of the house using *Polar Tracking* and *Object Snap Tracking*. Draw the third line to snap to the corner of the house as shown at ③. Next, draw the first step beginning at point ②, extending 250 west, 1200 north and finishing using the *Perpendicular* snap mode to link back to the porch line. We will now use multiple copy to draw the last two steps …

[Modify.Copy]	Issue the *Copy* command … pick the three lines just drawn as the top step and then press *Enter* to complete the selection process …
m ↵	Select the *Multiple* option because we want to produce two copies … use *running Endpoint snap* to place the base point at ② and then position the mouse to set up a tracking line to the west of the base point …
250 ↵	To specify the distance along the tracking line to the first copy …
500 ↵	To specify the distance along the tracking line to the second copy.
↵	Press *Enter* to terminate the command.

Draw the rest of the garden bed by drawing the line from the corner of the steps (point ④) 1000 mm west and then back to the house outline using the Perpendicular snap mode. Finally, zoom out a little and draw the driveway line from point ⑤ and then perpendicular to the site boundary.


Parallel Snap tool

To draw the left edge of the driveway, we use the new *Parallel* snap mode. Begin the line at the SW corner of the building and then select the *Parallel* snap mode (illustrated) from the *Snap mode flyout*. Pause the mouse pointer over the west boundary line until the *Parallel snap symbol* appears and then move the mouse back towards the east until a tracking line appears parallel to the side boundary. Using that tracking line, pick a point just short of the south boundary (you can't snap to that boundary in this operation). We will now use the Extend command to extend that line accurately to reach the boundary line.


Extend tool from the Modify toolbar

[Modify.Extend]	Issue the *Extend* command … first pick the boundary line as the *bounding edge* …
↵	Press *Enter* to complete the selection of *boundary edges* … then pick the driveway line we just constructed … AutoCAD should extend it to reach the first *bounding edge* it finds …
↵	Press *Enter* again to complete the command … note that with this command you can select several *bounding edges* and extend several lines to meet those bounding edges.

That completes the main building. You should now apply all the same techniques to draw the second building by yourself. It is actually a granny flat and is the subject of the next tutorial where we draw up an architectural plan of this granny flat!. The SW corner of the granny flat is positioned at **–2000,24500** relative to the SW corner of the site. It's dimensions, working counterclockwise from that point are: 5500 (east); 1200 (north); 3500 (east); 4500 (north); and then west and south to close the shape. Place the roof over it without any end gables as shown on the final drawing.


Save tool ...

This would be a very good time to save your drawing if you have not been doing so regularly!

OUTDOOR ENTERTAINMENT AREA

This section deals with the area around the pool and includes several new drawing techniques. We begin with the pergola.

Drawing the Pergola

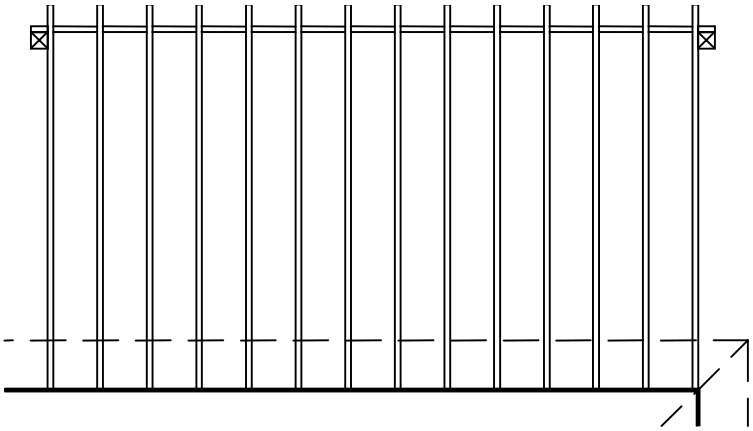
Zoom in on the area around the NE corner of the main house. We have used *Perpendicular* snap a few times now. It would be sensible to add it to your list of running object snaps: *Endpoint*, *Midpoint*, *Intersection* and *Perpendicular*.

Begin by drawing a polyline from the NE corner of the house using *Endpoint*, extending 3500 (north), then 50 (west) and then using *Perpendicular* snap mode to draw back to the face of the house. We then use the *Array* command (for large multiple copies) to duplicate this single joist as follows …


Array copy tool in Modify toolbar

[Modify.Array]	Issue the *Array* command … in response to the prompt, pick the polyline just drawn to represent one joist …
↵	Press *Enter* to complete the selection process.
↵	Press *Enter* again to select the default rectangular array.
↵	Press *Enter* again to select the default of 1 row.
14 ↵	Number of columns in this single row array of copies.
–450 ↵	A negative spacing will generate the copies to the left.

You should now have 14 joists arrayed across the back of the house as in the completed pergola shown below. We now need to add the edge beam and a couple of columns to support the joists (I realise that we probably need at least one more post, but that would spoil the symmetry so I left it out!). We will then trim the edge beam so that it appears to be under the joists.



The Finished Pergola

Draw the edge beam as another polyline, beginning @-150,-200 from the NW corner of the west-most joist, making it 6200 long and 50 wide (towards the south). Draw the post on the west side as a 150x150 rectangle, crossed to show that it is a timber post. To duplicate the post, we use mirror copy …


Mirror tool

[Modify. Mirror Copy]	Issue the *Mirror Copy* command … in response to the prompt, pick the rectangle and two crossing lines just drawn to represent the post …
↵	Press *Enter* to complete the selection … AutoCAD prompts for the axis of the mirror … we will show that by picking two points … move the mouse pointer along the edge beam until the Midpoint symbol appears (a yellow triangle) and click that position … then move the mouse down along the vertical tracking line, noting the position of the mirror copy … pick any point along that tracking vector.
↵	Press *Enter* again to accept the default which is NOT to delete the original, so keeping both the original post and its mirrored copy.


Trim tool

[Modify.Trim]	Issue the *Trim* command … AutoCAD prompts you to select the objects to be used as *cutting edges* … in our case that will be all the joists … to pick them in a single action, proceed as follows …
c ↵	Type “c” to indicate we wish to select the objects using a *crossing window* … then pick any two points that form the opposite corners of a rectangle that crosses every joist but does not enclose or cross any other graphic entity … only the joists should become selected and highlighted …
↵	Press *Enter* to complete the selection of *cutting edges* … AutoCAD now prompts you to pick the lines to be trimmed … you need to pick each small segment of line that has to be trimmed away, that is, where each joist crosses the edge beam … it is best to use *Zoom Window* to enlarge an area including 5 joists, pick all those, and then *pan* across to the next 5 joists, trim them and so on … the *Zoom* and *Pan* commands are both “transparent”, meaning they can be executed while another command remains active …


Zoom Window tool


Pan tool

↵	When all the lines have been trimmed, press *Enter* to complete the command.

Drawing the Swimming Pool

We will draw the swimming pool and spa by constructing two rectangles precisely placed relative to the site origin (SW corner). We will then fillet those rectangles to create the rounded corners, offset them to make the ledges inside the pool and then do some editing in the spa area to clean it all up. There are many tricks to be learnt in this process.

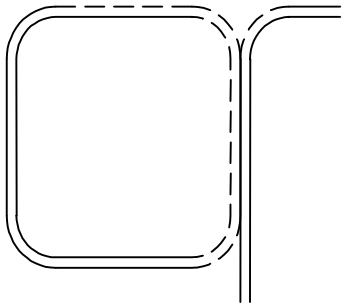

Origin UCS tool


Rectangle tool

[UCS. Origin UCS]	Issue the *Origin UCS* command and use *Endpoint* snap to place the origin at the SW corner of the site …
[Draw. Rectangle]	Issue the *Rectangle* command.
18000,22300 ↵	Enter the starting point of the rectangle as an *absolute coordinate*.
@-4000,7000 ↵	Enter the opposite corner of the rectangle as a *relative coordinate*.
↵	Recall the *Rectangle* command … use *Endpoint* to put its starting point at the NW corner of the first rectangle.
@-1920,-2000↵	Use *relative coordinates* to enter the opposite corner of the spa.
[Modify.Fillet]	Issue the *Fillet* command.
r ↵	Select the *Radius* option …
400 ↵	Set the radius to 400 …
↵	Recall the *Fillet* command …
p ↵	This time, select the *Polyline* option … since the rectangle is actually a closed polyline, we can use this option to fillet all the corners of the polyline in one action … simply click on the large rectangle.


Fillet tool

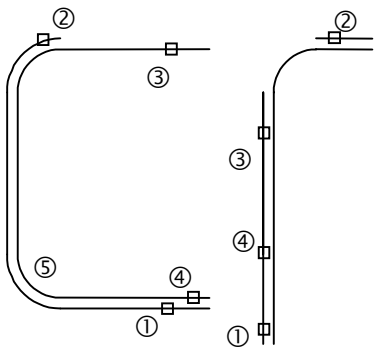
Recall the *Fillet* command a second time and fillet all four corners of the spa rectangle. Next, *offset* each rectangle inward by 80mm. Then use the *Explode* command to convert all four rectangles to separate lines and arcs so that we can edit them. Finally zoom in on the area shown in the next figure in order to clean up the area between the spa and the main pool.



STEP 1

Erase the five fillet arcs and two lines shown in this figure …

The first step (illustrated at left) is to use the erase command to remove the lines and arcs shown dashed in the figure.



The second step (illustrated in the figure on the right) uses the *Fillet* command to clean up the drawing as follows. Begin by using a fillet radius of 400 to fillet the two points labeled ①. Then use a fillet radius of 0 to fillet the line and the arc labeled ②. Finally to fillet the two pairs of lines labeled ③ and ④, we need to first determine the correct fillet radius for these "inside arcs". To do that, select any of the other inside arcs (such as the one marked ⑤) using the left mouse button, and then right click it and pick *Properties* from the pop-up menu. That will open the *Properties*

window showing all the details of that arc including its radius. Set the fillet radius to that value and then fillet the two pairs of lines labeled ③ and ④.

Real-Time Zoom and Pan

Real-Time Zoom tool
This tool allows you to zoom in and out of your drawing dynamically. Simply position the mouse pointer anywhere on the screen and, while holding down the left button, move up to zoom in and down to zoom out.

Real-Time Pan tool
This works in a similar way, allowing you to drag your drawing across the screen while holding down the left button.

*At any time during these operations, you can use the right mouse button to switch between Pan and Zoom or to Exit the command. You can also press [**Esc**] to exit either command.*

Now use the real-time *Zoom* and *Pan* tools (explained in the left margin) to zoom out to a view where you can see the whole of the pool and spa area. Then, using the *Polyline Edit* command (*Modify* > *Polyline*), we will join up the three rounded shapes that form the pool and spa to convert them back into polylines. You will need to use the command three times. Each time, first pick one of the lines or arc that forms the shape. AutoCAD will prompt you saying that the line is not a polyline, but will offer to convert it to one. Answer "yes". Choose the *Join* option in the next prompt and when AutoCAD asks you to select the objects to be joined to the new polyline, simply pick two points below and to the left and above and to the right of the whole rectangle (this is called a selection window). Press *Enter* to complete the selection process and AutoCAD will join as many of the selected objects as possible to form the polyline shape. It doesn't matter if you selected extra objects in your *selection window* because AutoCAD will ignore any objects that do not connect to the polyline it is creating. Finally, press *Enter* to complete the command and then use the command again twice in order to make the other two polylines.

To complete the pool and spa, give the outer polyline shape a line weight of 0.40mm and the two inner shapes a line weight of 0.13mm. Then offset the outer shape by 400 to create an apron around the pool.

Pool Fence and Paving

By now you should be able to construct the pool fence with only a minimal amount of instruction! I will give you the key dimensions and a few specific clues, and leave you to do the rest.

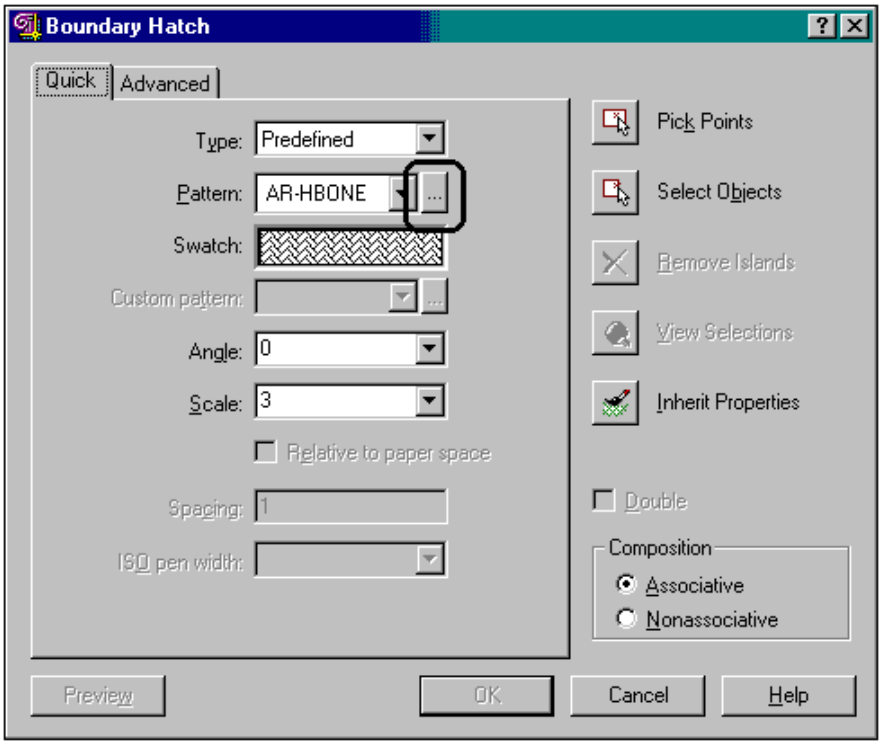
The fence should be drawn as a polyline beginning 6500 west from the NE corner of the main house. It extends 1200 north, then 3000 west, then follows a gentle arc to a position about 13000 at an angle of 80^{o} before turning east and extending perpendicular to the property line. When drawing a polyline, you can switch between arc and line segments by using the *Arc* and *Line* options respectively. To force the arc to begin in the northerly direction, use the *Direction* option and then the polar tracking line to specify the starting direction of the arc before entering its endpoint using a relative polar coordinate.

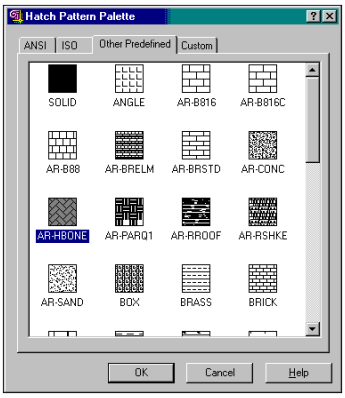
Offset the fence line by 150 on either side to create a 300 strip under the fence. Then set the line type of the fence to something suitable (I used *Fenceline2*) and the lineweight of the two adjacent lines to 0.13 so that they fade out a little in the final plotted drawing.

Now add a similar fence between the house and the boundary line about 1000 south of the NE corner of the house. He pool and spa are now fully enclosed so we can add the pavement hatching.

Boundary Hatch tool

Adding the pavement hatching is remarkably simple once the process has been explained. AutoCAD has a very smart boundary hatching tool that allows you to fill any bounded shape with a hatch pattern simply by picking a point within that bounding shape. AutoCAD searches outwards from the pick point to identify any complex shape. Furthermore, it allows you to pick several bounding shapes to fill with the one hatch object. That is useful in this case because we want the hatching to go not only right round the pool, but also to appear between the pergola timbers and under the roof overhang. AutoCAD includes a wide range of hatch patterns that are pre-defined, as well as the ability to create custom patterns from a mixture of cross-hatch lines of any linetype at any angle. You will pick up some of these options by looking closely at the dialogue boxes as we go through this process. ***Begin by clicking the Boundary Hatch tool (illustrated) and study the resulting dialogue box***.

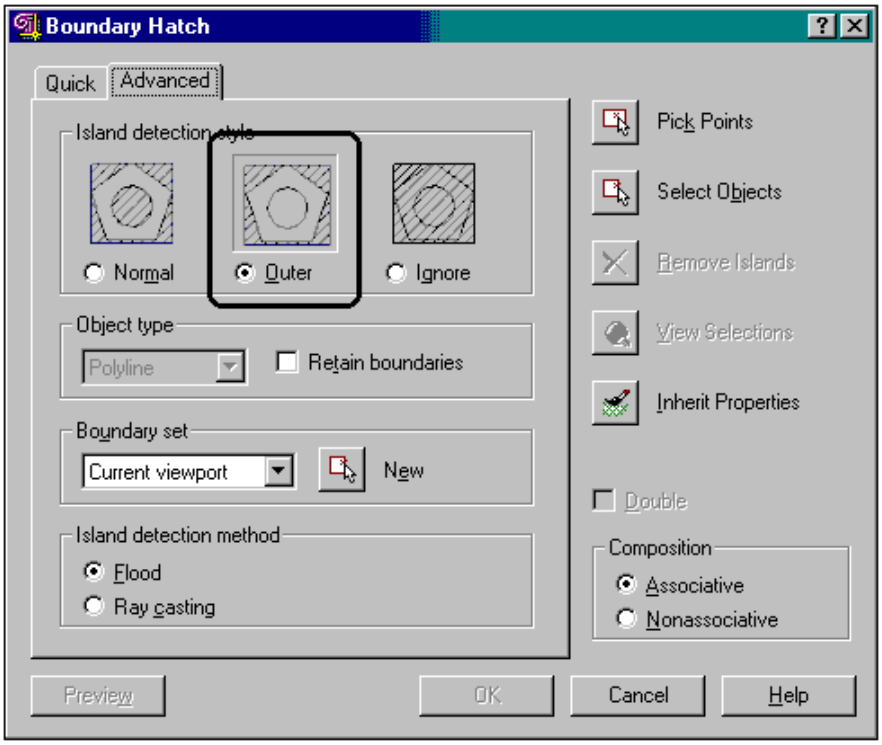




Sample from the Hatch pattern palette available in AutoCAD.

The figure (above) shows the boundary hatch dialogue box with the basic hatching options selected. I have nominated to use a *Pre-defined* hatch pattern. I selected *Pattern* "AR-HBONE" which I chose by clicking the browse button (circled) next to the *Pattern* list box. The browse button opens the dialogue illustrated in the left margin, allowing you to browse through the range of pre-defined patterns available in the various libraries that come packaged with AutoCAD. Note that you can add new patterns to the library set, but that goes way beyond this introductory tutorial. Notice that I have set the scale to 3.

Once you have set the basic options, switch to the *Advanced Tag* as illustrated below.



On the *Advanced Tag*, select the *Outer* option as indicated. This tells AutoCAD to only hatch around "islands" rather than to hatch the inner parts of islands which is the *Normal* option.

Now switch back to the *Quick* tag and then click the *Pick Points* button (top right corner of the dialogue). This returns you to the drawing (temporarily) so that you can pick points within the boundary you wish to hatch. Pick any point around the pool first, and then pick points between every joist in the pergola and along under the eaves line so that you include every boundary. When you think you have it right, press *Enter* to return to the dialogue box. Next click the *Preview* button to see what the hatching looks like. When ready, press *Enter* again to return to the dialogue. If you missed any areas, or you wish to try a different scale factor, set those things and then preview the hatching again. Repeat that process until you are happy with the result and then click *OK* to draw the hatching.


Save tool

That almost completes the outdoor entertainment area. There is one more pavement line to add between the base of the Granny Flat steps and the pool fence. Add that line into your drawing and then save it before proceeding.

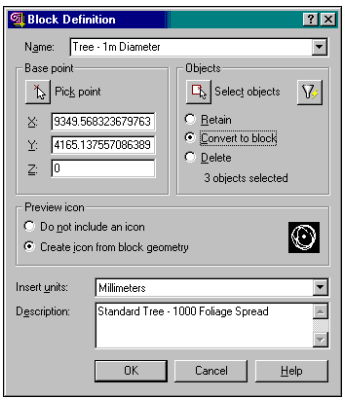
ADDING THE TREES

For this exercise, we are going to create one tree that has a canopy spread of about 1 metre. We will then save that tree symbol as a block and then insert instances of that block at a variety of scales and rotations to represent the different trees on this site. By making the block 1 meter is size, we can easily scale it up to suit any size tree that we wish to represent. I would suggest to you that that is a good strategy for representing objects like trees in a landscape drawing.


Spline tool.

Begin by going to any clear area of your drawing and constructing a circle that is exactly 1000 in diameter. Zoom in on that circle and place a second circle in the centre of the first with a diameter of 100 to represent the trunk. (If your circle is drawn as a polygon, type in the **viewres** command, press *Enter* for the first prompt and **500** for the second … this command effectively tells AutoCAD to represent circles as true circles.) Then use the *Spline* tool (illustrated at left) to draw two or three spline curves to represent the foliage of the tree. You will find it necessary to turn off all running snap and tracking modes while you do this "sketching". You can model your tree on the one in the tutorial, or vary it in any way you wish! Use the 1000 diameter circle to guide you with the size of your foliage. When you are satisfied with your tree, delete the original 1000 diameter circle. We are now ready to convert the tree into a block.


Make Block tool



[Draw. Make Block]	Issue the *Make Block* command … in the resulting dialogue (illustrated), set the name of the block and put a brief description down the bottom … then click the *Pick Point* button …
[Osnap. Centre]	Activate the Centre snap mode and click on the small circle at the centre of the tree … this defines the insertion point for the block when this object is placed into a drawing … you will be then be returned to the dialogue box … next, click the *Select Objects* button …
w ↵	Type "w" to create a window selection box and then pick two points that define a rectangle that encloses the whole of your tree.
↵	Press *Enter* to terminate the command and return to the dialogue box … click *OK* to complete the process.

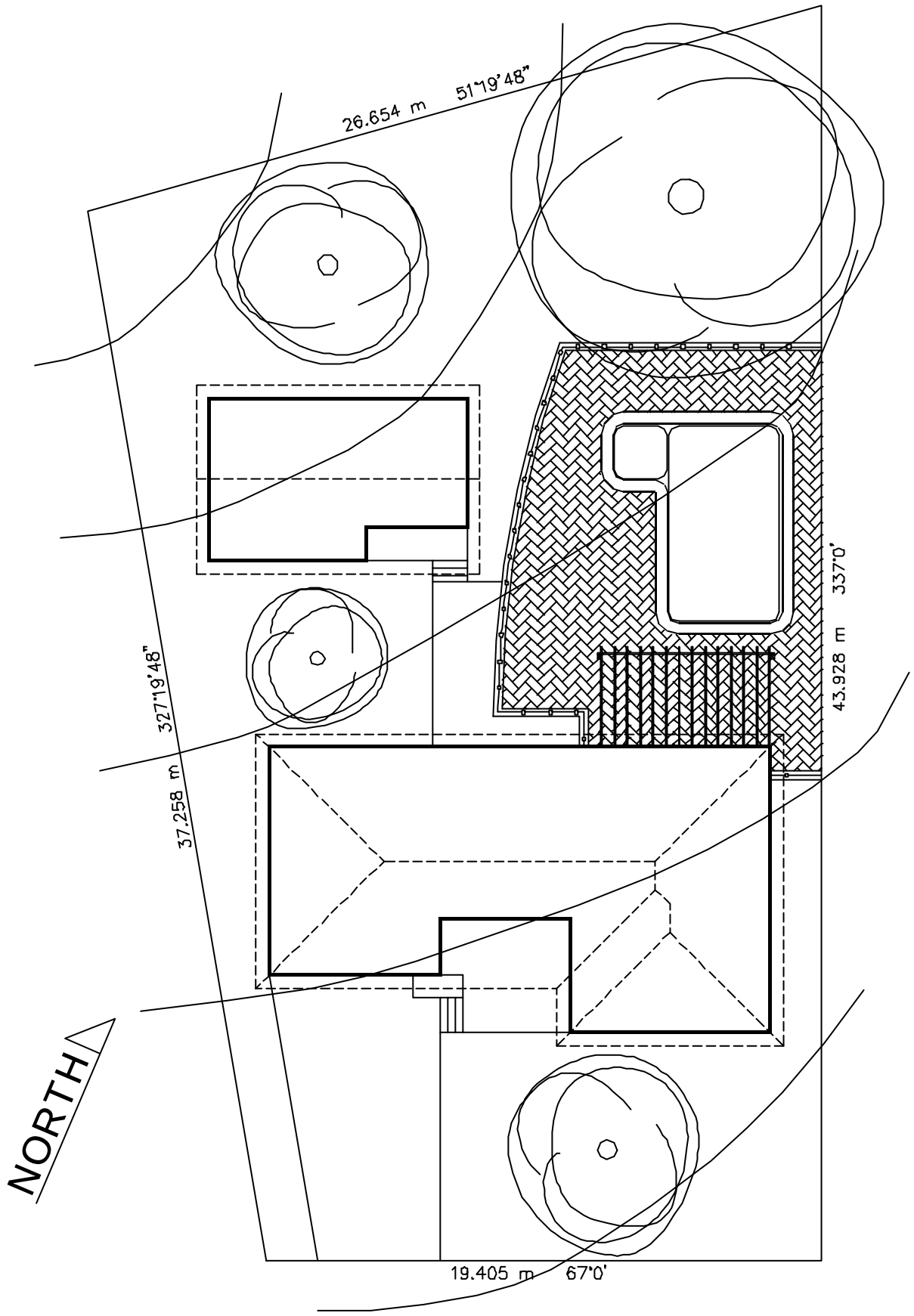
By default, AutoCAD leaves the block on the drawing. In this case, we don't actually want a 1 metre tree, so just delete the block by picking it and clicking the *Erase* tool. Then zoom out so that you can see the whole site.

Insert Block tool

I now want you to complete the drawing by using the *Insert Block* tool (illustrated) to place the following trees on the site at the locations specified relative to a datum point at the SW corner of the site. You will need to follow the prompts carefully and to rotate each tree as you enter it so that they do not all look too obviously the same!

6.8 m canopy at	**11900, 3900**
12.6 m canopy at	**14650, 37250**
7.2m canopy at	**2150, 34860**
5.0m canopy at	**1790,21100**

Done!!



Residential Site Plan

Drawn by: Jim Plume 23/3/00

Scale: 1:200