AutoCAD Tutorials

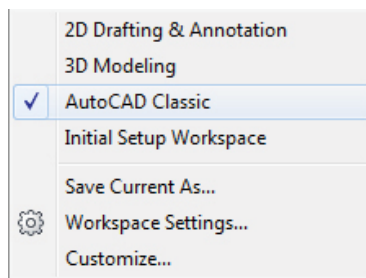## Content

This Tutorial is formatted mainly for AutoCAD 2010/2011 2D Drafting & Annotation Mode.

Important tools/commands are highlighted in red bold.

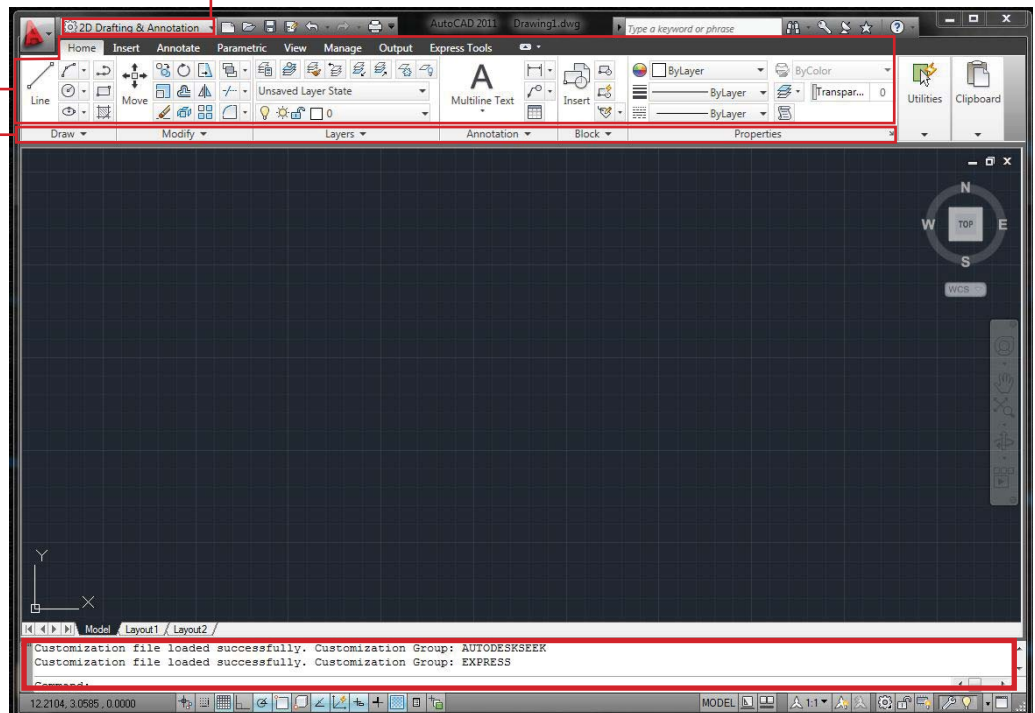Words in parenthesis are type-in commands.

workspace

2D Drafting & Annotation
3D Modeling
✓ AutoCAD Classic
Initial Setup Workspace

Save Current As...
⚙ Workspace Settings...
Customize...

If you are used to the workspace in older version, you can choose "AutoCAD Classic"

Tabs

Panels

enter commands for drafting and other functions here (recommended way to draw because it is faster than clicking iron) This line will also show options within a tool. Please make a habit to constantly check the command line for what you can do with the tool you selected.

When Tabs or Panels disappear sometime, you can right-clicking on the grey part of the menu bar to find them.
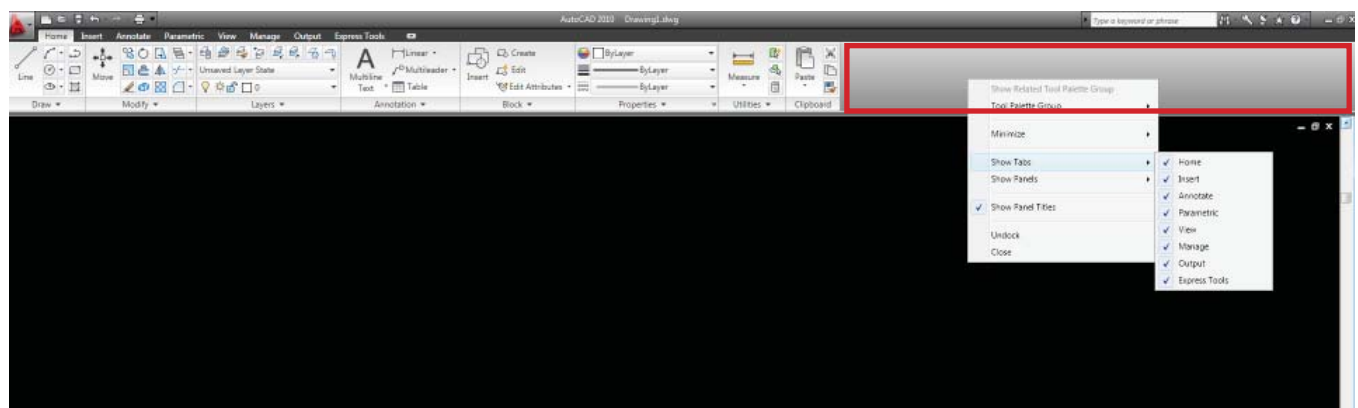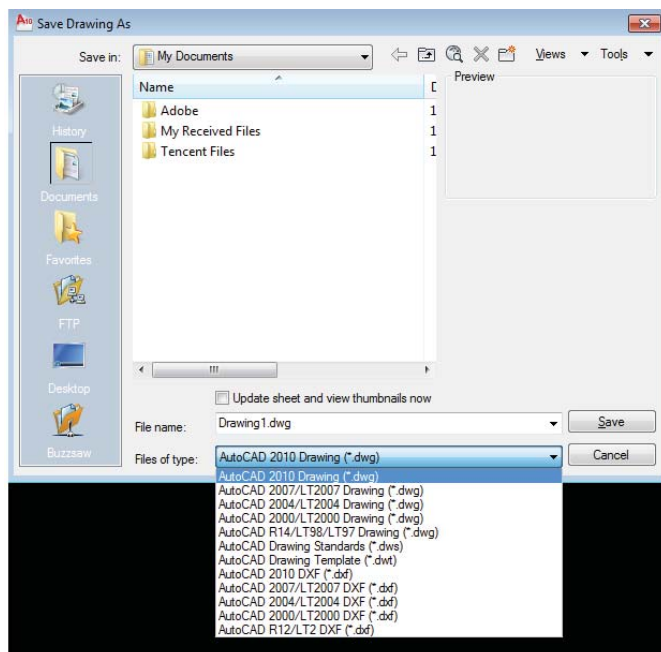
save

When you open AutoCAD, there is an existing sheet called "Drawing1.dwg". You can work on this sheet and then save it by either of these two ways.
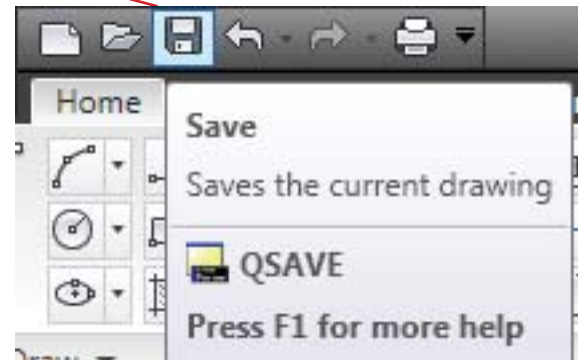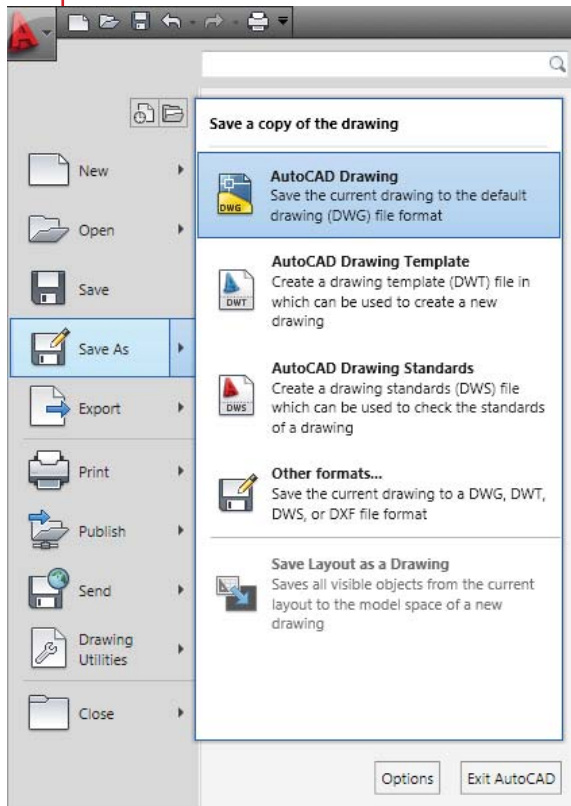

After the first time save, you can click this button or use shortcut Ctrl+S to save your drawing. It is a good habit to save your work frequently in case the software crashes or power off.

The drawing is saved as AutoCAD 2010 Drawing as default. If you want to open it in other softwares(like Sketch Up, Rhino) or lower version AutoCAD, you have to save it as a lower version.

**set up a new sheet**

There are two ways to set up a new sheet
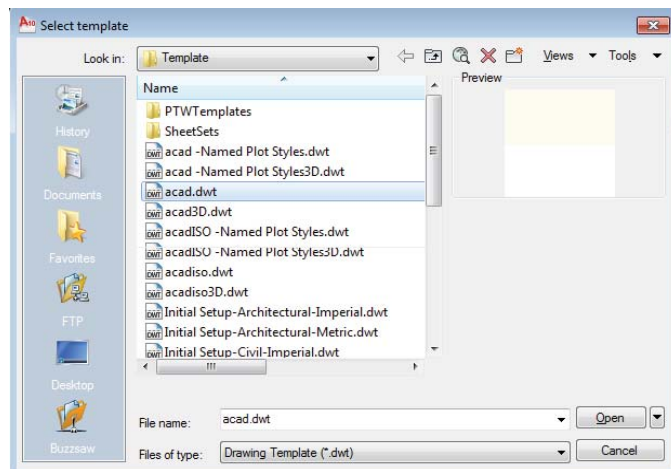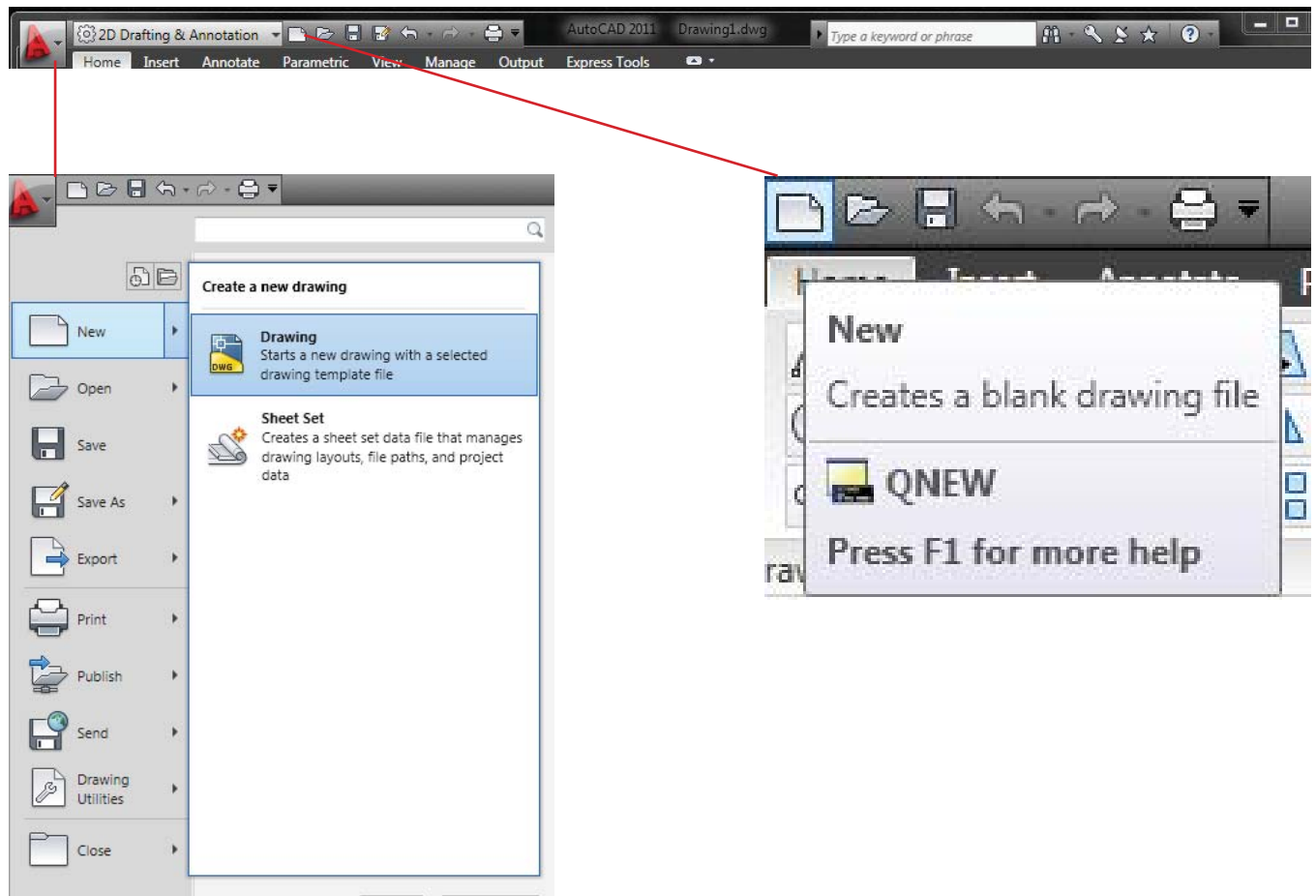
The default format is "acad.dwt". In practice field, you may be required to use a template.

drawing units (units)

You can set the drawing units in **Tools > Units**. As a default, the measurements in the drawing is unitless. For example, when you draw a line with length "10", AutoCAD doesn't know if it is 10 miles or 10 millimeters until you set the unit. Remember to set it before you start drawing, or scale the drawn objects accordingly if you change the unit in the existing drawing.

Architectural (i.e. 1'-3 1/2") and engineering (i.e. 1'-3.50") units are most frequently used in the states. Please note that you have to specify the type of units in Insertion Scale tab when it is set to decimal, scientific, or fractional units.

For example, if you want to use millimeter:
set "Type" in Length window to Decimal > set Insertion Scale to Millimeter
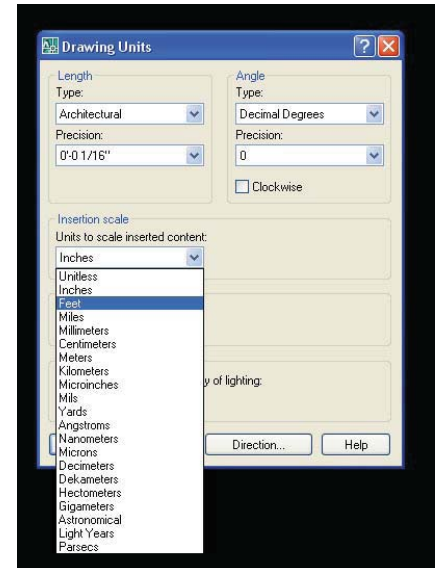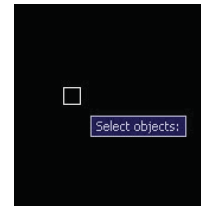
cursors

When you are not in command, the cursor looks like a cross with a square in the middle. When you are in drawing-related commands, the square disappear. When you are in modify-related commands, only the cross disappear. You can escape from a command by hitting **Esc** key.
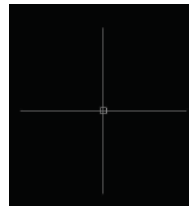
not in command   in drawing command   in modifying command

select / deselect

You can select the objects by clicking on an object or drawing a window around it. **Drawing a window from left to right selects everything that the window crosses. Drawing a window from right to left selects everything that the window contains.** You can deselect objects by doing the same operation while holding down Shift key, or deselect everything by hitting **Esc** key.

general shortcuts

Many general shortcuts are used in AutoCAD.

For example:
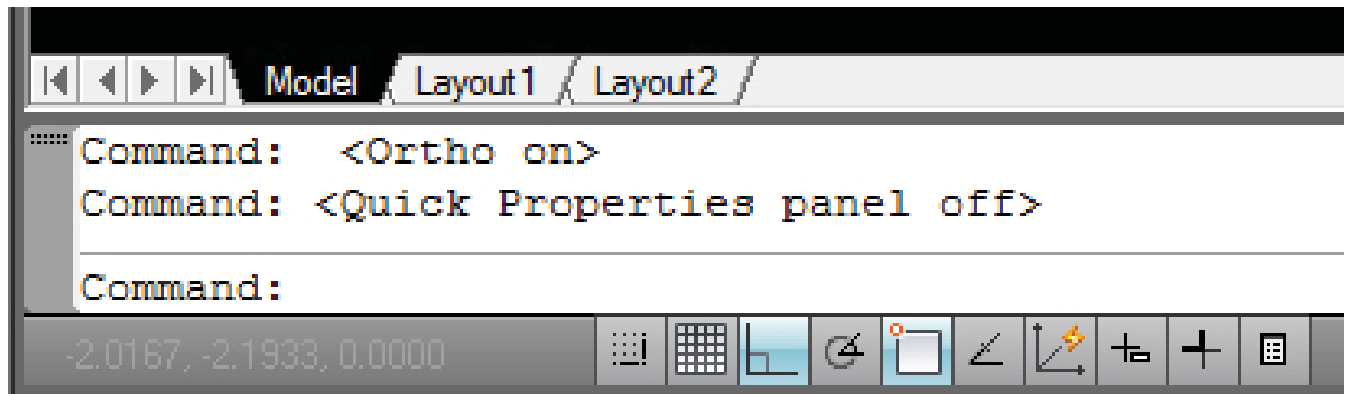Ctrl+S saves the dwg file (qsave)
Ctrl+Z undoes the previous action (u) and Ctrl +Y redose the undone action again (redo)
Ctrl+C copies and Ctrl+V pastes
Ctrl+P opens the plot window (plot)

snap & tracking

There are settings that help you draw accurately. Please pay attention to the tabs below the command line and click on them to turn on/off.

Snap Mode: snaps to increments of the spacing that you specify. (i.e. if you set it to 2 in both x,y direction, it will snap on 2,2:2,4:10,8 etc)

Grid Display: displays grids with the spacing that you specify

Ortho Mode: restricts movements to horizontal and vertical directions.

Polar Tracking: tracks increments of the angle that you specify(i.e. if you set it to 30 degrees, it will track 30,60,90,120 degrees etc.)

Object Snap: snaps to existing objects on the screen. You can also activate specific Object Snap during drawing/inquiry operations by right clicking and selecting snap overrides.
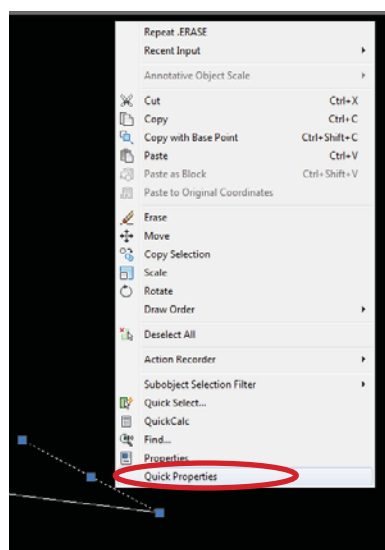
Show/Hide Lineweight: turns on/off the line width on display.It does not affect the line widths on plot.

Quick Properties: displays the properties of selected object(s). Another way to show properties: After select a object/objects, right click and select "Quick Properties". You can change the object(s) properties here(layers,color and linetype.)
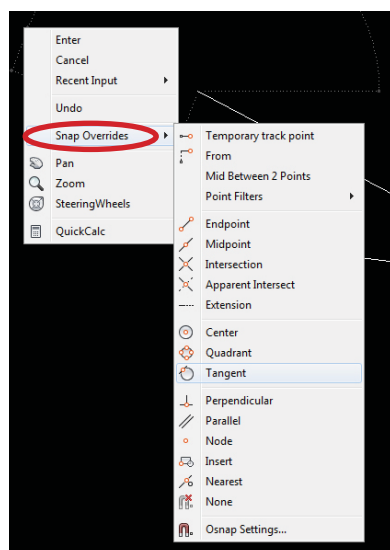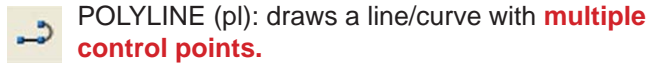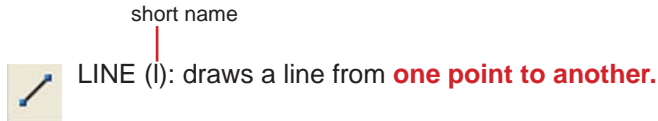

snap overrides

right click
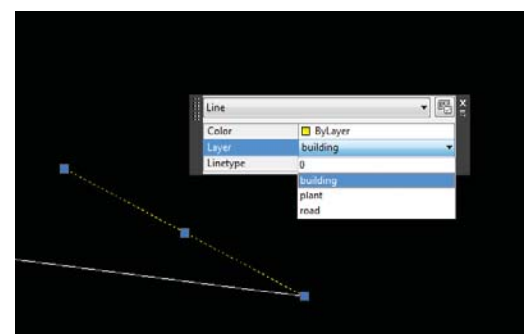
quick properties

Draw

There are two ways to draw

1. type full or short name in command line(increase efficiency)

2. Icon: Home/Draw

short name

LINE (l): draws a line from **one point to another.**

line

POLYLINE (pl): draws a line/curve with **multiple control points.**

Pay attention to the command line while using POLYLINE tool; it will show you variety of options for the next move you can take. Following are some of the examples.

polyline

drawing a polyline

If you hit "**Esc**" while drawing a polyline, the line terminates at the last point you clicked.
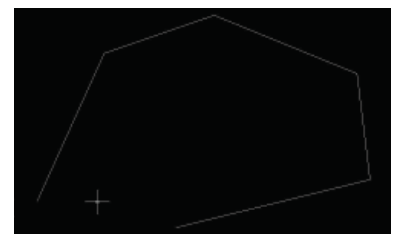

hitting Esc

If you type "**cl**" while drawing a polyline, it draws a segment that connects the starting point and the last point you clicked (the line becomes a closed shape).

typing "cl"

You can also draw an object with arcs using POLYLINE tool.
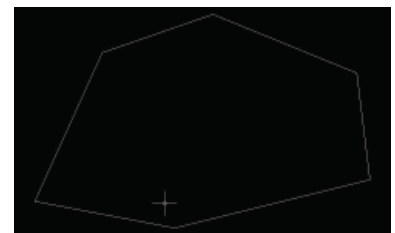
If you type "a" while drawing a polyline, your next segment will be an arc. It will continue to draw arc segments with tangent to the previous arc.
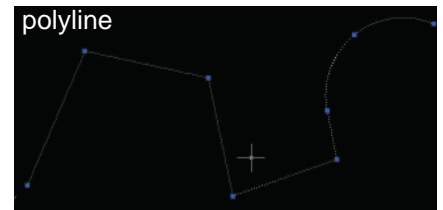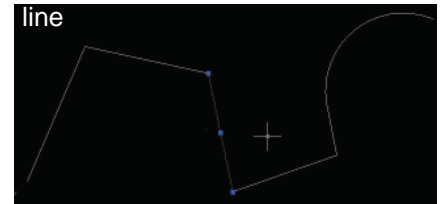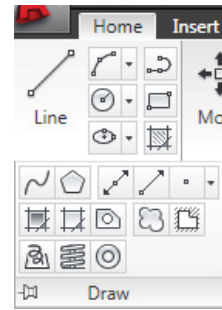
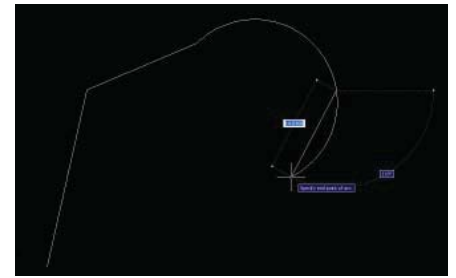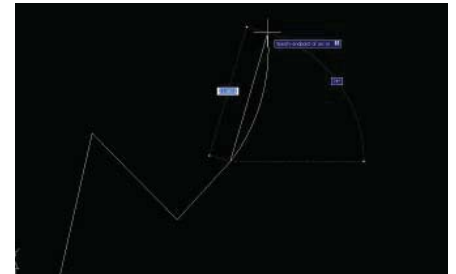If you type "s" while drawing an arc segment, you can define an arc angle by specifying three points that the arc goes through.

If you type "r" while drawing an arc segment, you can define an arc radius.


RECTANGLE (rec): draws arectangle by defining two corners.

Click on the point where you want to place the first corner of a rectangle or type in x,y coordination for precise imput.

If you type "@" after defining the first corner, you can specify the second corner by x,y coordination. For example, type @36,24 for 36" x 24" box.

If you type "d" after the first corner, you can specify the second corner by the x and y distance from the first corner. For example, type "d" > space > 36 > space > 24 for 36" x 24" box.
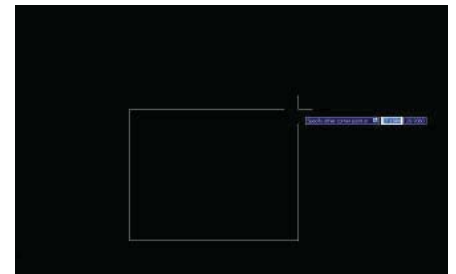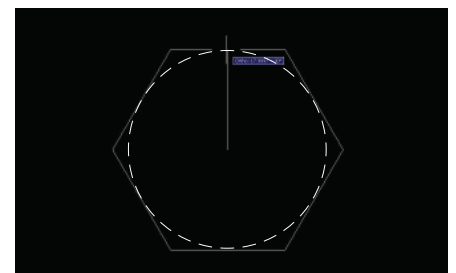

POLYGON: draws a polygon by number of edges.

If you choose to inscribe in circle, it draws a polygon **inside the circle** with the radius you specify.
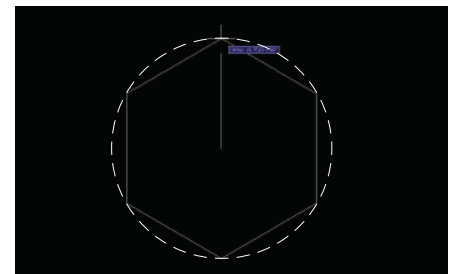
If you choose to circumscribe about circle, it draws a polygon that **contains the circle** with the radius you specify.
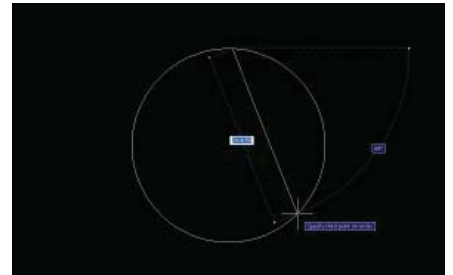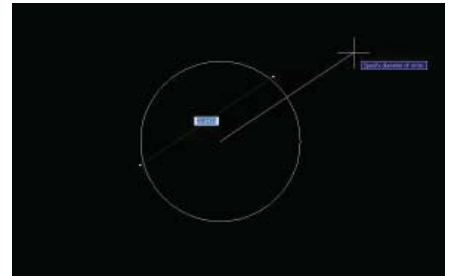
CIRCLE (c): draws a circle.

First click will define center of the circle. define the **radius** by typing it in or clicking.

If you type "d" after the first click, it will draw a circle with the diameter that you specify.

If you type "2p" or "3p" **before** clicking the center, it will draw a circle that goes through the points you specify.


SPLINE (spline): draws a spline connecting multiple points.

Define the points that a spline goes thgough by clicking.

End the tool by hitting Esc (terminates the spline at the last point clicked) or typing "cl" (adds a curve back to the starting point).

Spline has a tendency to increase the file size and may cause complecation when modify later. Excessive use of splines are not recommended especially when sharing a file with other people.
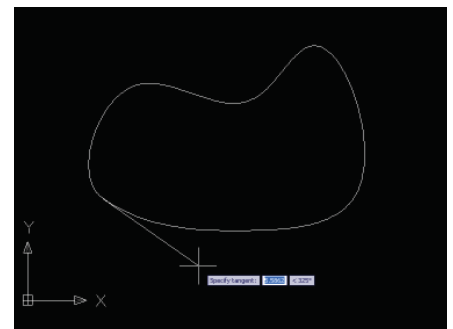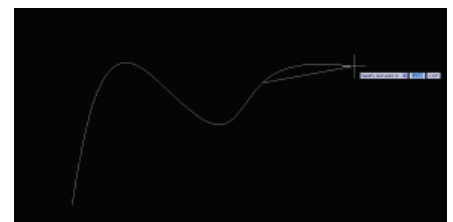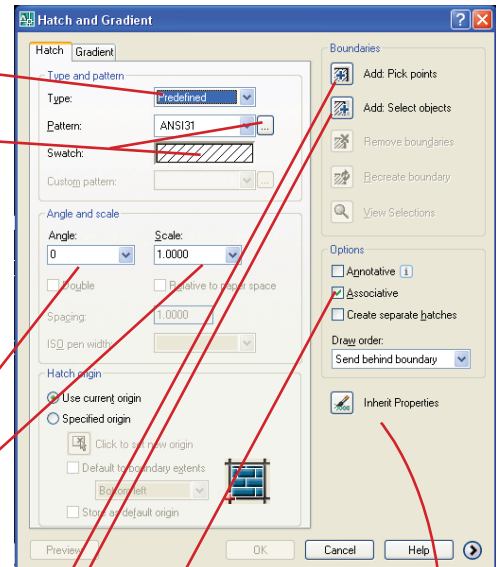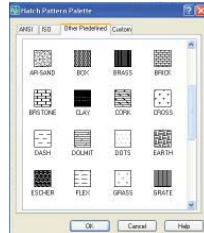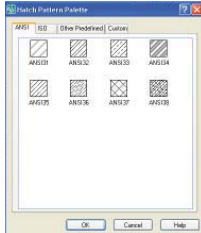
HATCH(h): fills an enclosed area or selected objects with a hatch pattern or fills.

Type: usually predefined

Pattern: choose the pattern from list (pull down) or clicking either of these.

The window below pops up. ISO has basic patterns. Other predefined tab shows commonly used patterns such as BRICK, EARTH, GRAVEL, CONC, SAND...


You can modify the angle and scale of patterns

When hatching a space defined by different objects such as an overlapped zone of 2 circles, select "Add Pick points". The more complex the area is the longer it takes to calculate.   You get an error message when the area is too complicated to calculate. Zooming in/out to the hatching area sometimes helps. You can also maunally manipulate the selection areas by using buttons below: "remove boundaries" and "recreate boundaries"

When hatching a complete object, such as circle, rectangle, or polygon, select "Add select object". If you wish, you can pick multiple objects at one time.
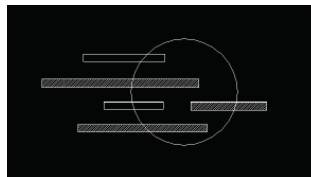

Check "Associative" if there is a possibility that the shape of the hatch may be modified later.

You can copy/paste the exact hatch properties from existing hatches by clicking this.

Once the selections are set, let's check how it looks by clicking "preview" button on the left corner. Modify angle and scale if necessary.

You can also modify the property after you complete the command. Double click the hatch and the property window will pop up. Simply make changes.

Modify ▾

▫ Multiple Points
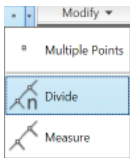
Divide

Measure

DIVIDE (div): Divide an object into certain length or perimeter segment. You can draw from the node or you can place objects along the nodes. i.e. You want to place 5 trees equally on a line.

Select object to divide, enter # of segments.

MEASURE (me): Places point objects or blocks at measured intervals on an object. i.e. You want to place trees every 50 feet.
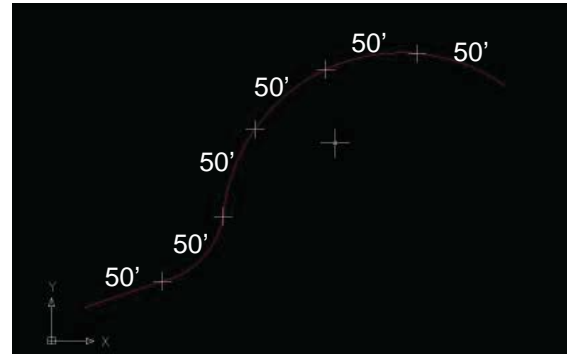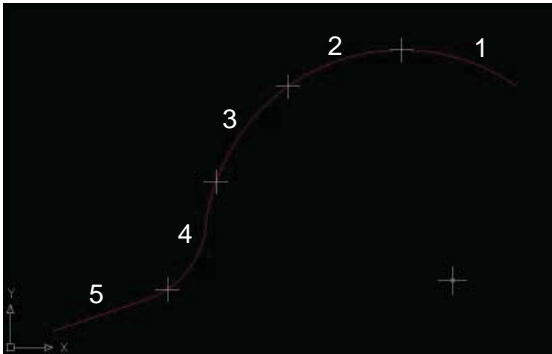
As default, the cross nodes won't show up unless you type "pdmode" and change value to 2 or 3. It is not necessary to show them up. You can snap the nodes by choose "Node" in "Object Snap" setting. If you don't want to plot nodes, do not forgot to change pdmode value back to 0 before you plot.
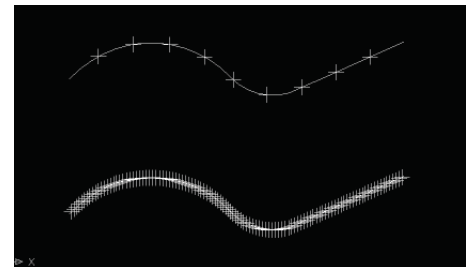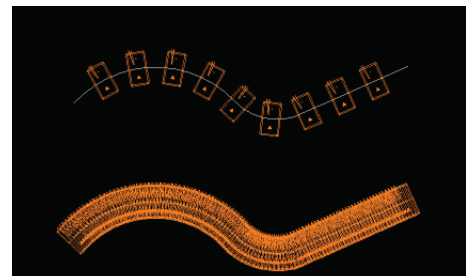

You can directly place blocks by specifying a block instead of #s.

Divide or measure, select the object, (b) for block, type the name of block (must be exact name), define if you want the block to align the curves or not, enter # of segments or length of segments.
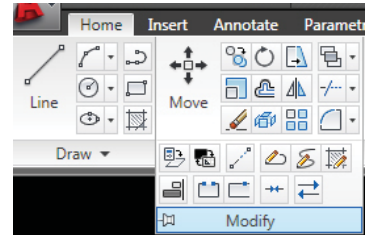
Do not forget to place blocks at the ends.

This may be useful for placing plants, light, etc...

Modify

There are two ways to modify

1. type full or short name in command line(increase efficiency)

2. Icon: Home/Modify

ERASE (e): erases the selected objects. You can get the same effect by selecting objects and hitting Delete key.

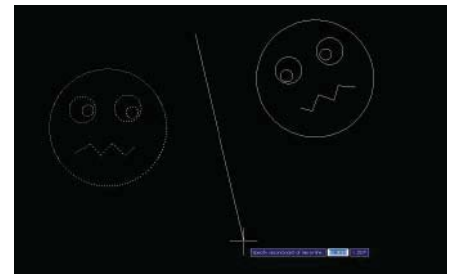COPY (cp): copies the selected objects from one place to another.

Select objects, click/type the base point and the second point that you want to copy objects to.

You can also copy+paste objects by selecting objects and right-clicking on them. It will save objects in clipboard and allows you to copy things from one file to another. Select "copy with basepoint" to copy things between files at precise locations.
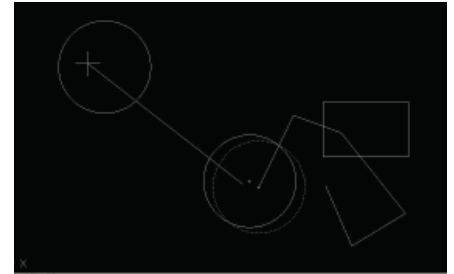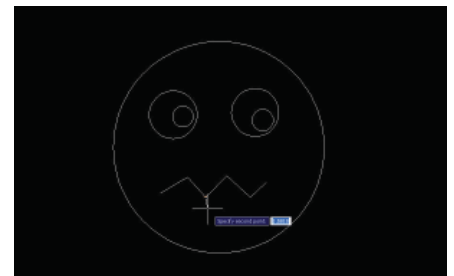

MIRROR (mi): mirrors objects along the line that you define.

It will ask you whether you want to keep the original object or not after defining the mirror line. type "y" for yes or "n" for no.
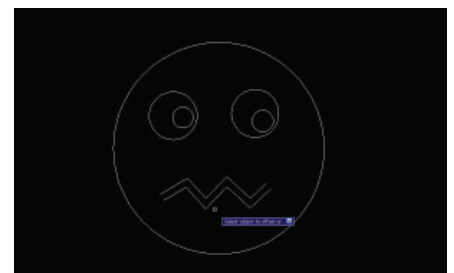

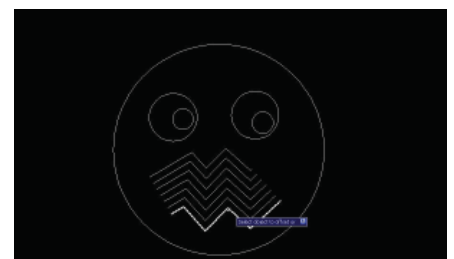OFFSET(o): offsets objects to the distance you specify.

It will ask you the distance to offset objects first. you can specify it by typing the distance in or clicking two points on screen.

After entering the distance, grab an object to offset and click on the side that you want to offset it to.
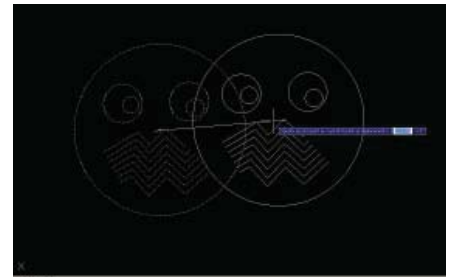

You can offset an object multiple times in the same distance by repeating click.

MOVE (m): moves the selected objects from one place to another.

Select objects, click/type a base point and the second point that you want to move objects to.

**SCALE (sc):** scale objects by the ratio you specify.

select objects, click/type a base point and click/type the scale numerically.

you can also scale objects by the relative distances using "Reference" points.

select objects, click/type a base point, then type "r" for Reference.

click the base point again, then click the second point to define the original distance. click the third point at where you wish the previous point to be after scaling objects.

In this way, it will scale the objects by the distance ratio of base point to the second point : base point to the third point. Position of the base point needs to be constant.
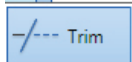
**TRIM (tr):** trim objects along specified objects.
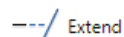
Create objects that you want to use as trim edges.

Click on Trim tool, select the trim edges and hit enter, click on the objects to trim **on the side that you want to erase**.

You can trim multiple objects at once by using selection window or "fence" tool.

To use selection window, draw a window after selecting trim edges instead of directly clicking on objects to trim.

To use fence, type "f" after selecting trim edges and draw a line. It will trim everything that the line crosses.
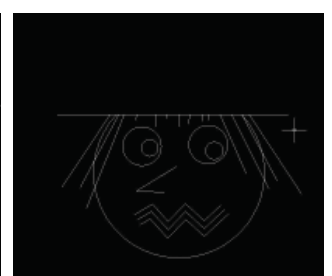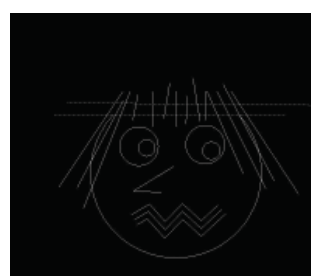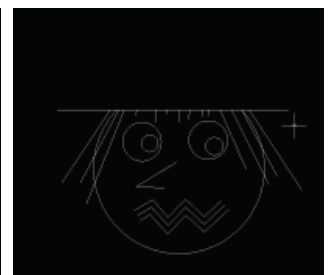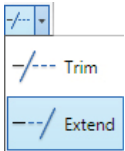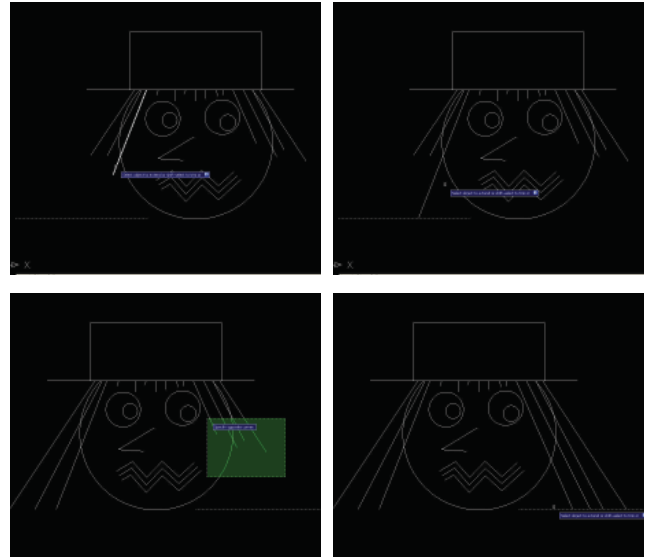
EXTEND (ex): extend objects to reach specified objects.

Basic operation is similar to "trim". Select objects that you want to use as extend edges and hit enter, click on the objects that you want to extend.

You can also extend multiple objects at once by using selection window or "fence" tool. For directions, please see "trim" section.

EXPLODE (x): break selected complex objects such as blocks and polylines down to lines and arcs.

POLYLINE EDIT (pe): There are extensive options to edit polyline. Please pay attention to the command line for the available options and directions.

```
Command:
Command: _pedit

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

Useful options are:

Close: adds a line to close an open polyline.

Join: joins touching multiple polylines into one object

> AutoCAD does not join vertexes when overlapped. Make sure that the lines/arcs that you want to join are touching at an exact point. If you are unsure, move one of the vertexes away and put it back together using "end point" snap.

Edit Vertex: modifies control points.

> After you select Edit Vertex, a little X appears at the end of a polyline. It will move to the next control point every time you hit Enter. You should navigate this X to the control point that you want to modify.
>
> Adding a control point:
> Navigate X to the control point **right before** the segment that you waht to add a point. Type "i" to insert a vertex. click where you want to add a new point. Hit Esc to exit the option.
>
> Dividing a polyline at a control point:
> Navigate X to the point that you want to divide. Type "b" for break. Type "g". Hit Esc to exit the option.
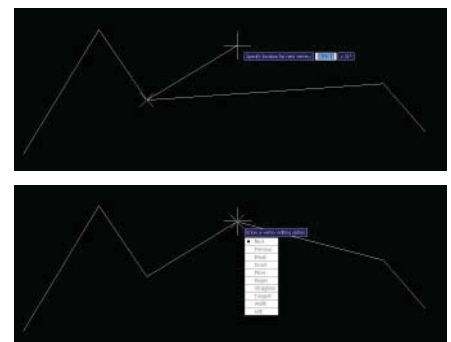
Spline: makes a selected polyline into B-spline curve.

Decurve: makes a curved polyline into lines.

BREAK (br): can make a cut in continuous line.

Choice 1) select an object by clicking where you want to start the break, specify the second point.

Choice 2) select an object, (f) for first point, then click the second point.

When breaking a non-closed shape, the break occurs simply between two points. When breaking a closed shape, the break occurs between two points **counterclockwise**.
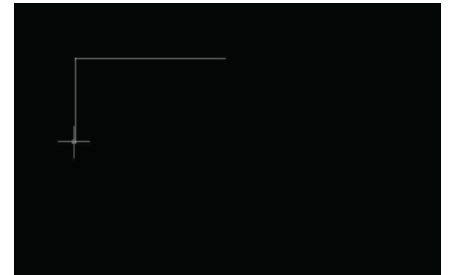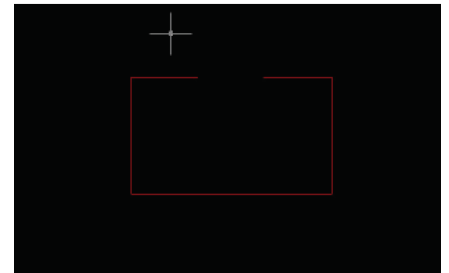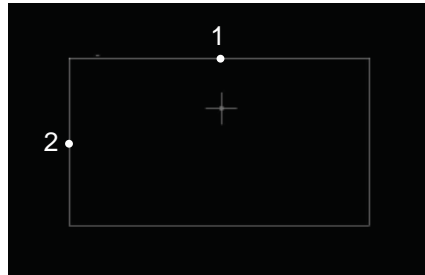
FILLET (fillet): connects two objects with an arc that is tangent to the objects and has a specified radius.

Select radius, type number, select firs object, and then the second one. If you don't enter a number, the two lines will extend to join each other.

You can fillet:
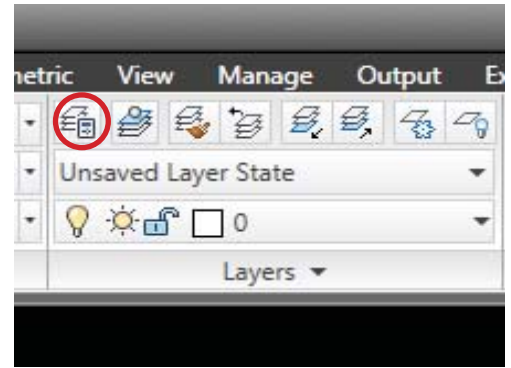 - rectangle   - arcs
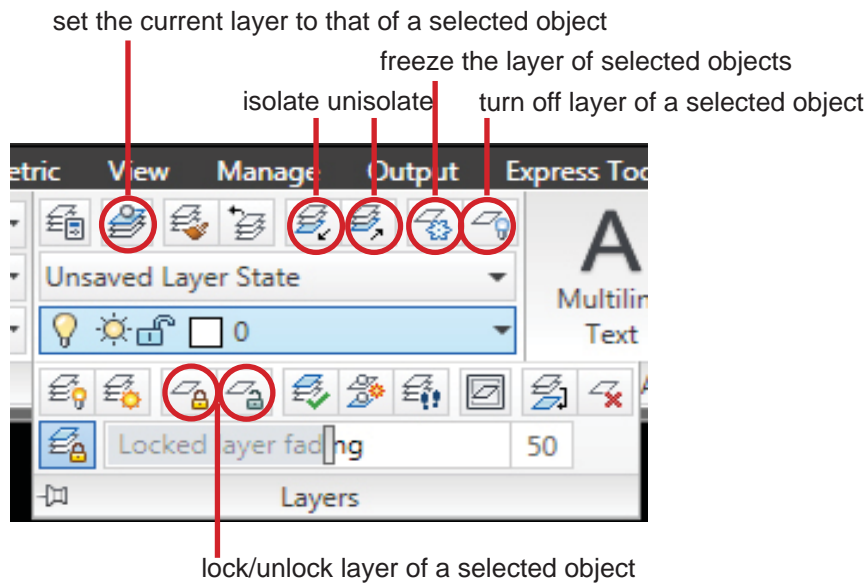
 - circles
 - ellipses
 - different line types
 etc

Layers

AutoCAD uses layer systems like Photoshop and SketchUp to organize a drawing.

Layer palette shows your current layer name and state. To access Layer Manager, click on the highlighted icon or type "layer" on the command

most useful layer icons, especially when you have a lot of layers.

set the current layer to that of a selected object

freeze the layer of selected objects

isolate unisolate   turn off layer of a selected object

lock/unlock layer of a selected object

Please make a habit to name and manage your layers appropriately. Generally, it is better to put the same objects in the same layer. For example, you can name them "column", "plant", "building", "road" and etc. If you manage your layers appropriately, you can lock/turn off/freeze all the layers except for the layers you are working on easily. In this way, you can select the objects you want easily.
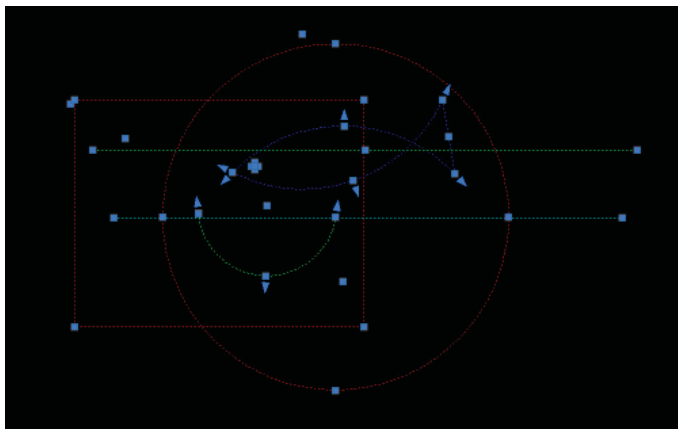
right cross select all objects when all the layers are on
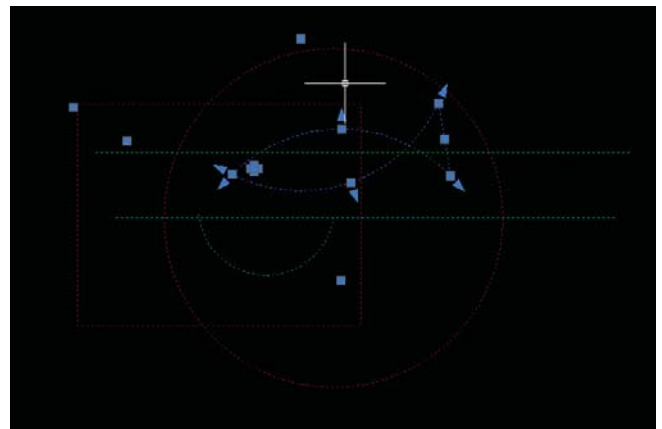
right cross only select the blue fish when other layers are locked

Layer Manager (layer)

turns on/off the layer

turned-off layers become invisible on the screen but geometric information is still in the drawing

makes the selected layer current

freezes/defreezes the layer

frozen layers are invisible and geometric information is unavailable

deletes the selected layer

makes a new layer

locks/unlocks the layer

locked layers are visible but objects are not modifiable

indicates current layer

you can select layer colors by clicking here

changes layer line type (dotted, dashed etc.). You have to load the line type in a different window to be able to use it in the drawing
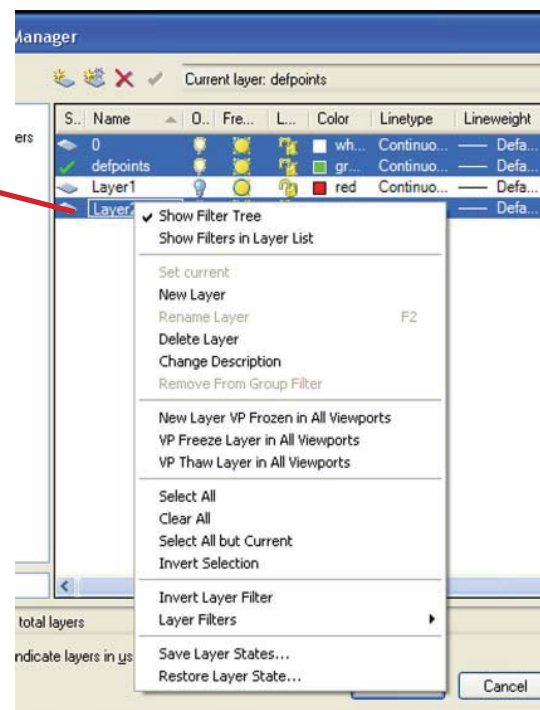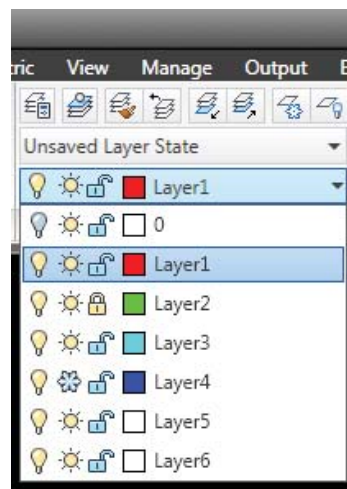
changes line weight

turns on/off on the plot. De-floints layer cannot be plotted by default
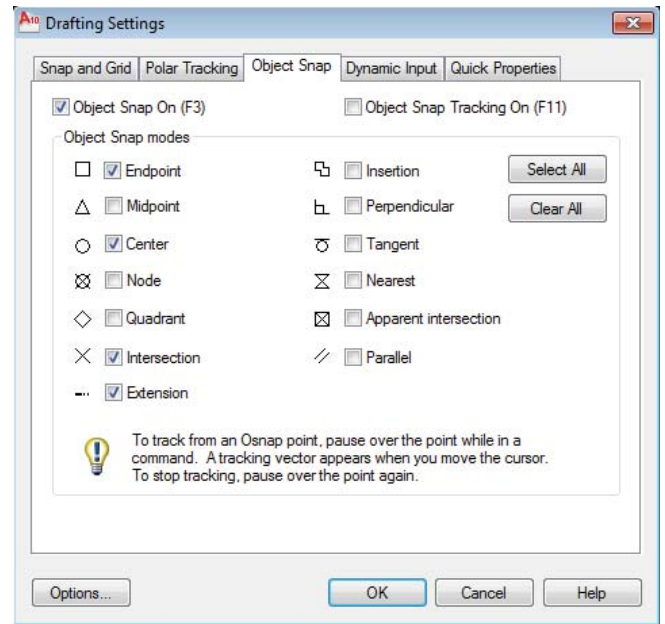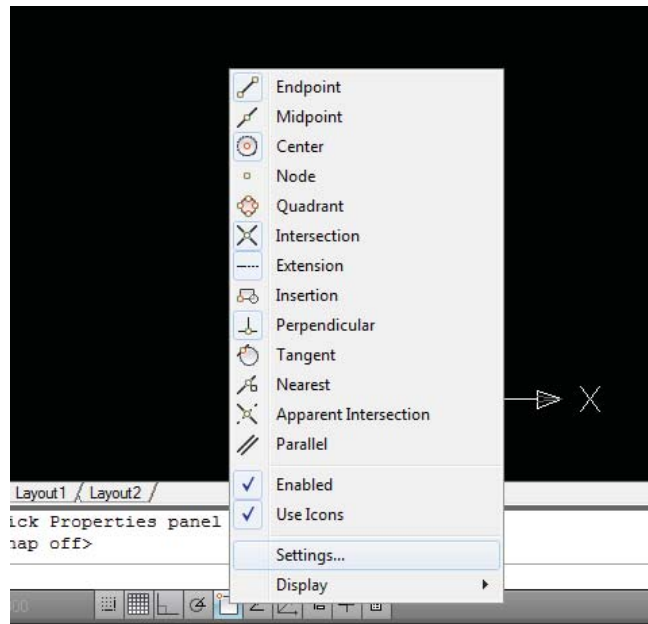

Rightclicking on the layer names gives you more options to manipulate layers such as filters and selections.

Pulling down the layer name on the layer palette also allows you some control on layers.

You can turn off/on, freeze/defreeze and lock/unlock layer easily here.

6

To adjust settings, right click on any snap/tracking tabs. A setting window will pop up.
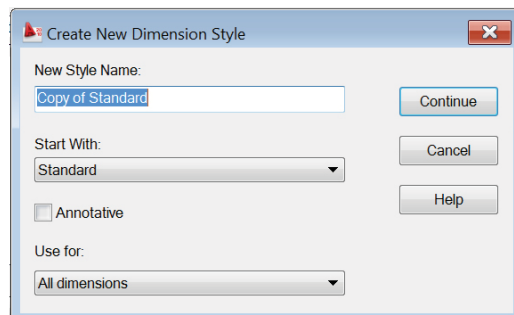
Annotate

Dimension Style Manager

First, set up a new dimension style in Dimension Style Manager by clicking
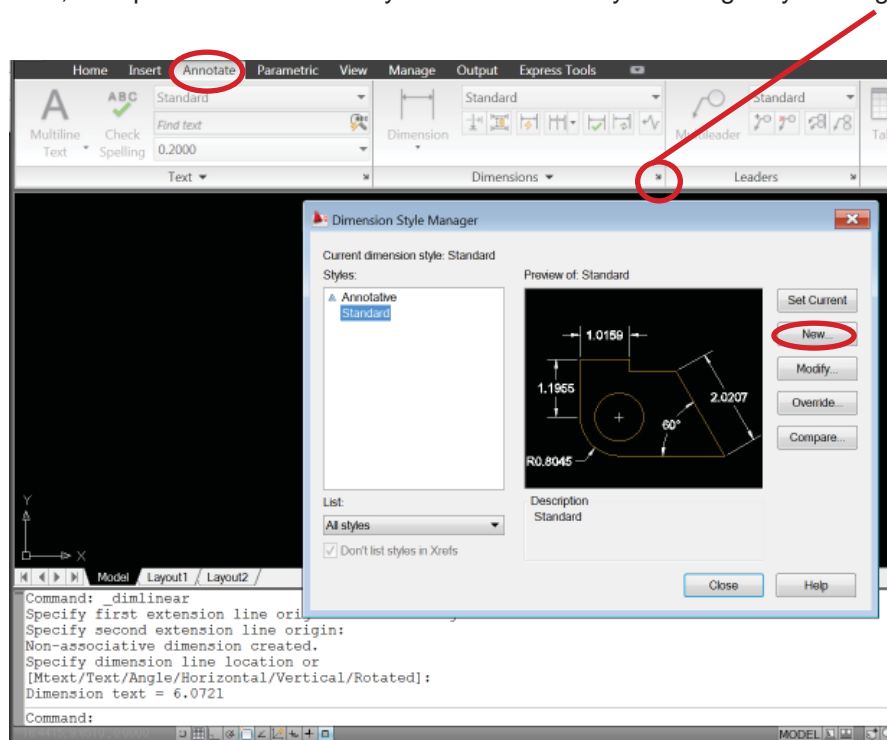
choose new and set up a new style

choose continue and then you can modify the settings. The most important ones are:
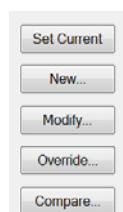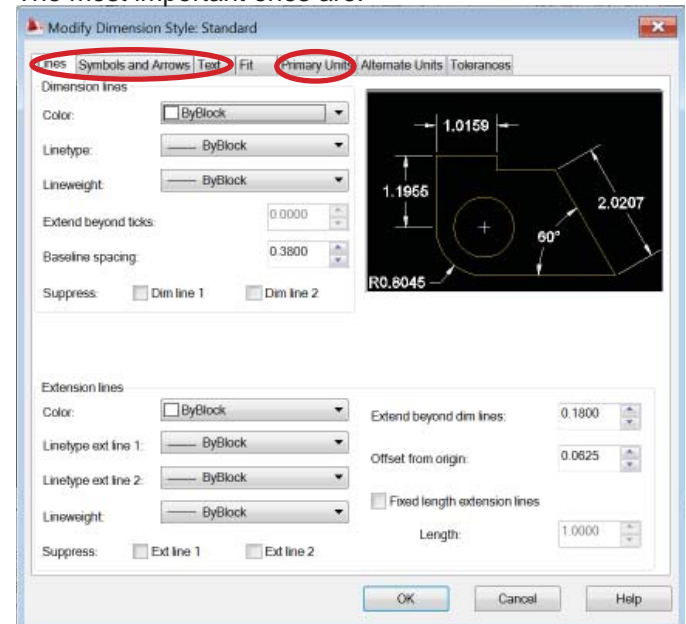
Set the right style as current if there are several by choosing "set current"

Modify dimension styles by choosing "modify"

Dimension

set a new layer for dimension, then use the following tools.

Dimension text override

If the dimension is not correct but it is hard to change the drawing, you can override the text by: select the dimension>right click>select "properties"> "Text"> "Text override"> type the number you want

Block

or

BLOCK: groups and objectifies selected lines as one entity.

Give a name easy to remember

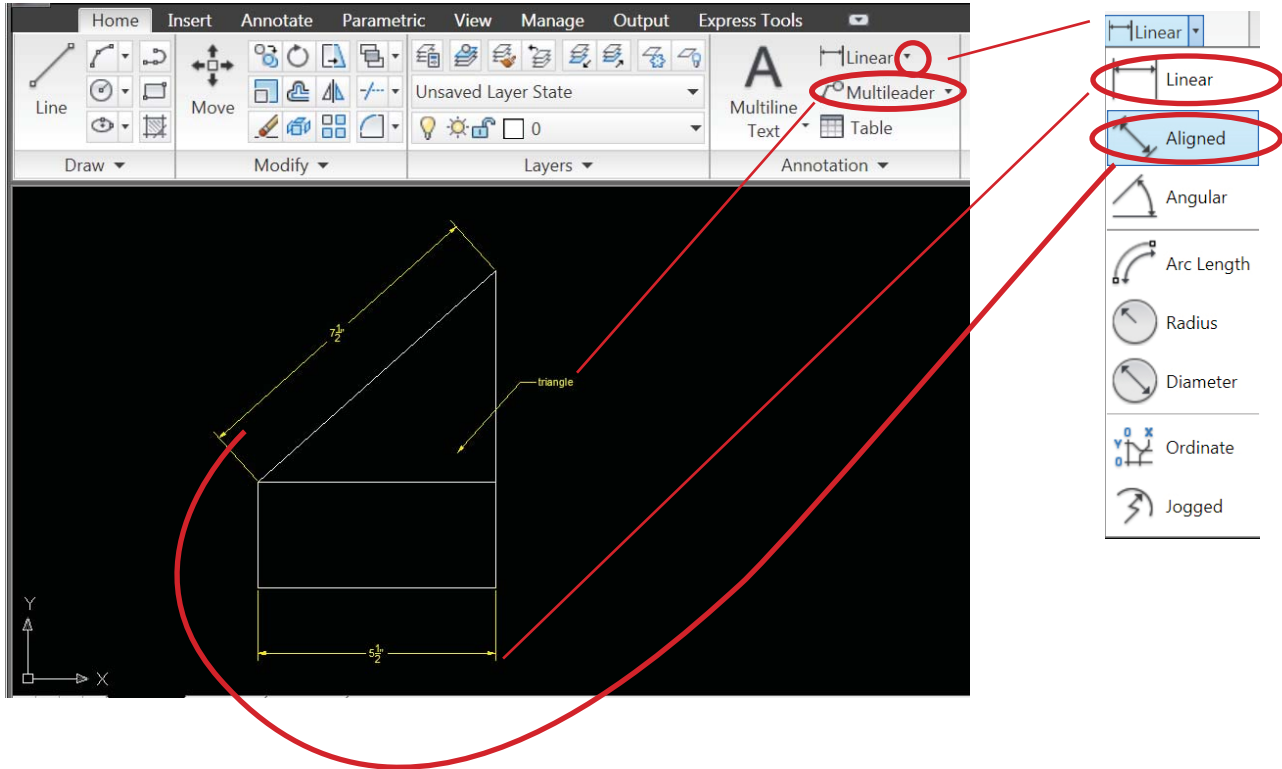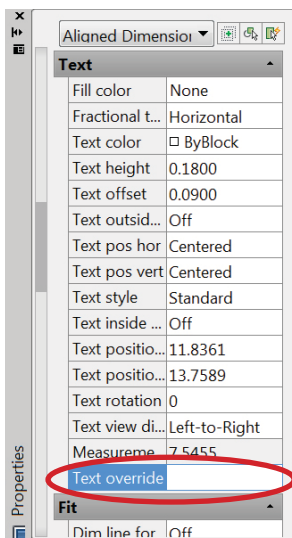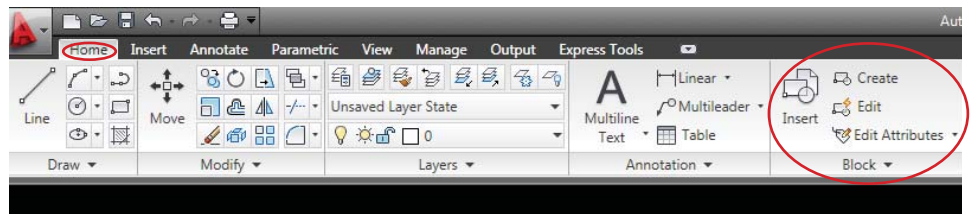Draw/move lines into layer 0 (or create block in new sheet, then later import the dwg file itself)
select them, **block (b)**, pick base point by clicking pick point, later this point is used as insertion point.

Make sure convert to block is checked

Once the block is saved, you can insert from the saved location
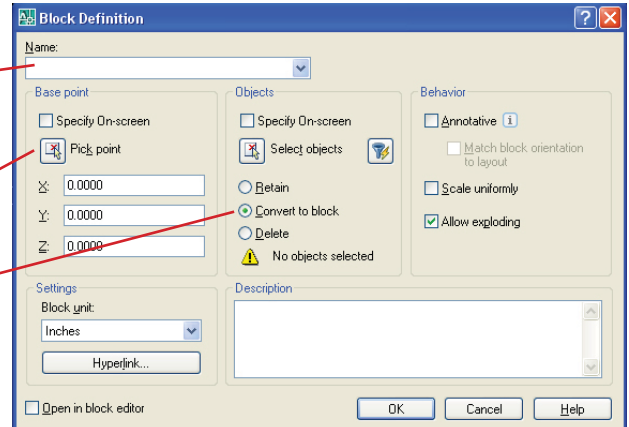
You can modify a block in "block editor"
Or explode, modify, then re-create the block.

BCOUNT: count the number of objects made by the same block. The computer will ask "select objects" after you type "bcount". i.e. You can simplily count the number of the blocks named "Tree-6ft" by selecting any "Tree-6ft" block. If you type"all" and press "Enter" button, it will count all the blocks.

Properties
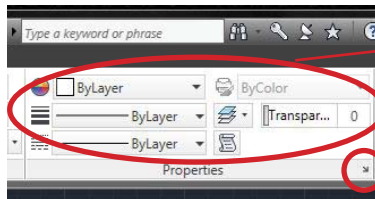
You can change main properties. Auto CAD 2011 adds "Transparency" feature here

Detailed properties list will pop up once right click
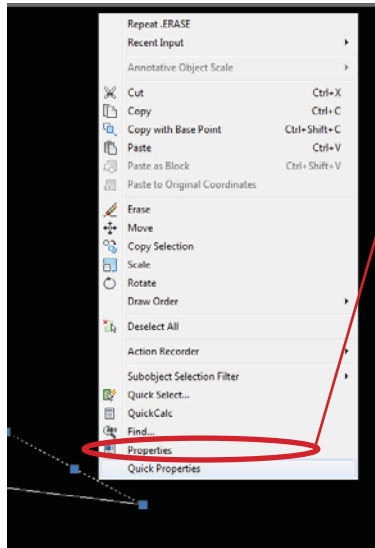
Detailed properties list will also pop up if you select a object/objects, right click and then select Properties.

Properties can be overridden in this list
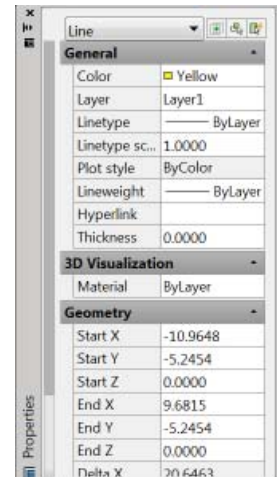
LIST (li): pops up a window with information of the selected geometry.

List, select object.

When you want to know the length of an arc or polyline, you should use either list of property.

MEASURE
distance(di): measure the distance between two points. click the first point and then the second one.
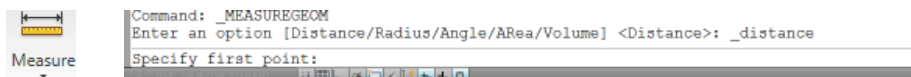
FLATTEN : when you can't measure the distance correctly, check the elevation or z-value in list or property. If that is a file that does not require to have 3 dimensional information, create 2D file by flattening. The command is also useful to export sections and perspectives as 2D drawings.

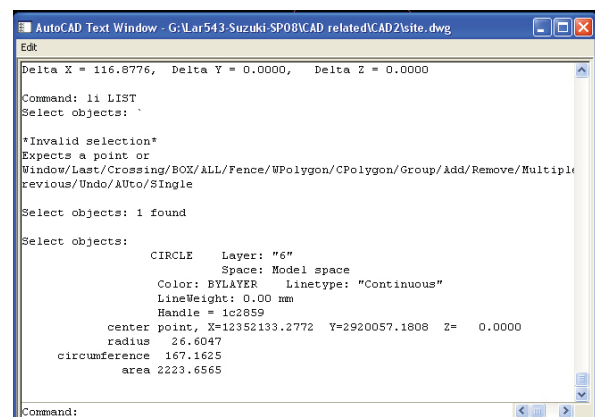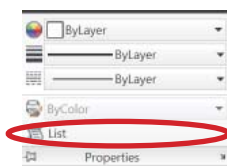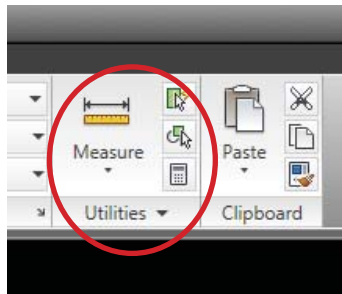 QUICKSELECT (qselect): allows you to select by different property values.

This is a useful tool expecially when modifying a large amount of segments or small scale objects dispersed in a large area with particular property.

For example, you want to erase all blocks named "tree2" but not other tree blocks:

- Select "Entire drawing" on the top tab.

- Select "Block" on Object type tab.

- Select "Name" on Properties

- Change "Value" to "tree2" and click on OK button.

- It should bring you back to the drawing screen with all "tree2" blocks highlighted. Hit Delete to erase them.
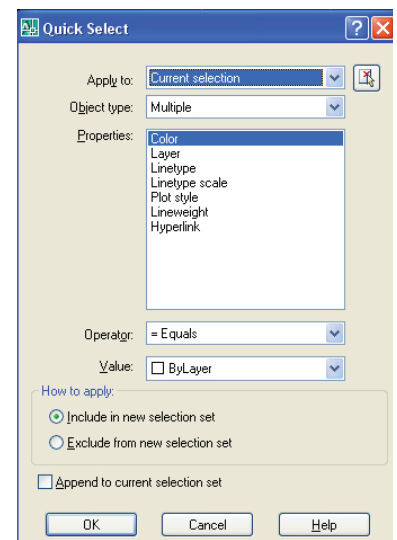
Clipboard
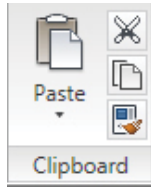
MATCH PROPERTY (ma   ): applies the properties of a selected object to other objects (layer, color, line type...).

Matchprop, select the source object - the object you want to copy from, click the lines/objects that you want to paste the property.

View

zoom (z)

There are three ways to zoom:

1.The middle wheel on the mouse allows you to easily zoom in and out.

2.type zoom(z in short) in command line.
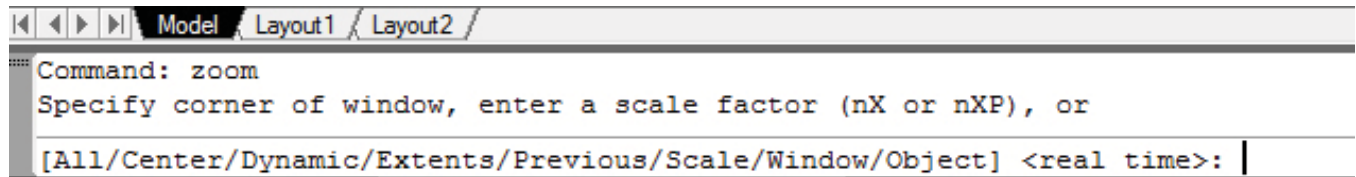
3. View/Extents
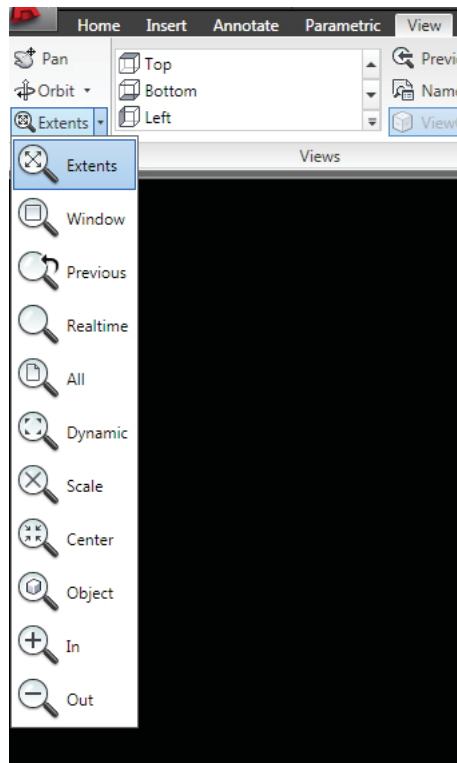
(type "E" or "e" if use command line) allows you to zoom in/out to the whole area that you have any objects

(type "W" or "w" if use command line) allows you to draw a window around the area you want to zoom in
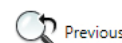
(type "P" or "p" if use command line) go back to last zoom area
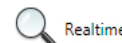
the same function as middle wheel on the mouse

(type "A" or "a" if use command line) allows you to zoom out to the entire drawings are
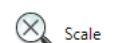
(type "S" or "s" if use command line) allows you to zoom in/out in certain scale
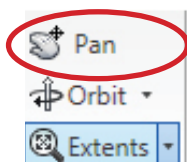
PAN(p) move the view

Insert reference
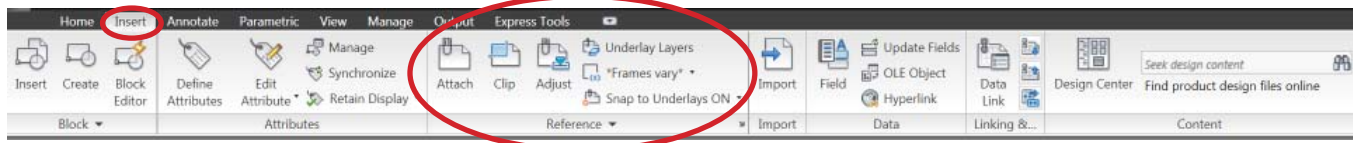
You can insert files in CAD, like image, other CAD drawing, excel form, pdf and etc.

ATTACH: Click attach and choose the file you want to insert. Make sure you choose the right file format.
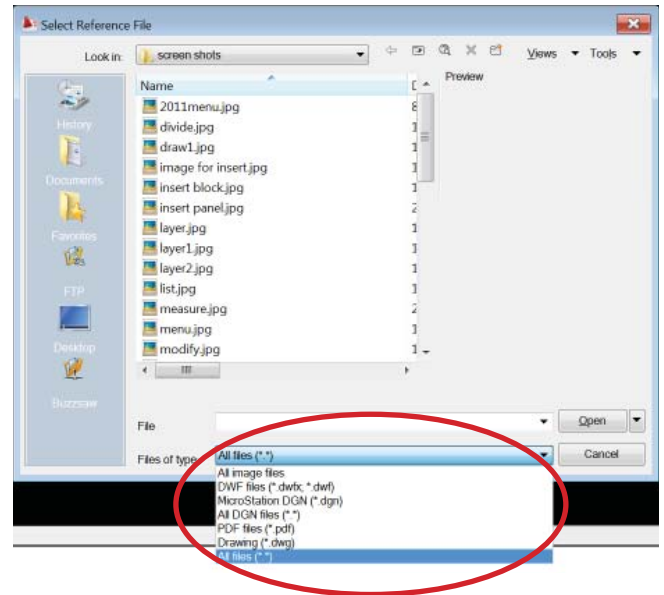
insert image

You can trace images, like hand sketch, after you insert them in CAD.

Create new layer "image" and set it as the current layer. Insert the image in this layer.

If you know the scale of the image, type value in Scale
i.e.1"=10' drawing in the dwg with "inch"unit setting - 120 (10 x 12)
1"=200'drawing - 2400 (200 x 12)

If you don't know the scale of the image, guess and type an appropriate scale or check specify on-screen.

Keep "Specify on screen" checked unless you know the exact values of x and y coordinates.
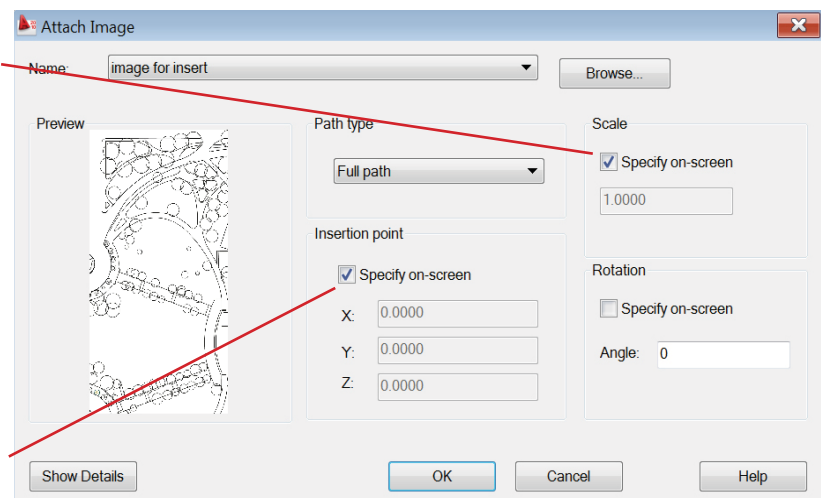
Select the image, and the adjust options will show up at the top.
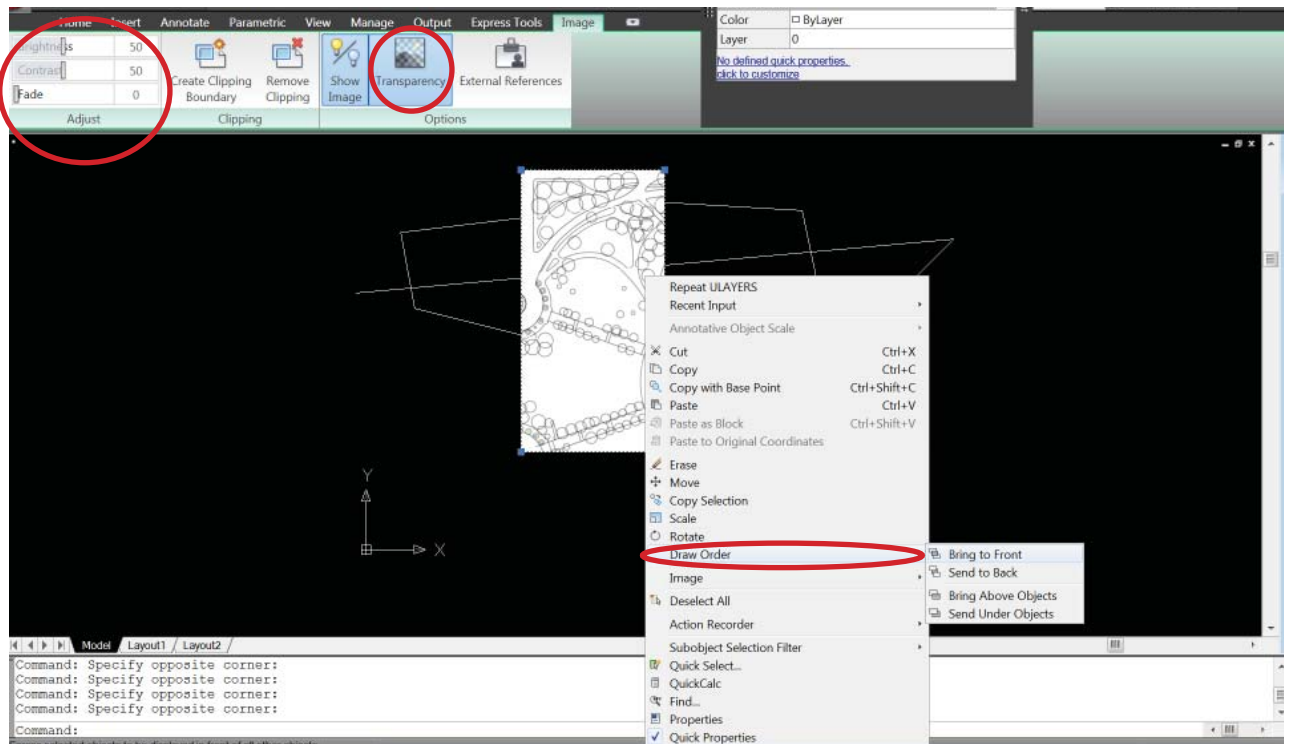
If the image is over other lines that you want to see, adjust the draw order by selecting the image, right clicking Draw Order > Send to Back

ALIGN: Align the image to the existing
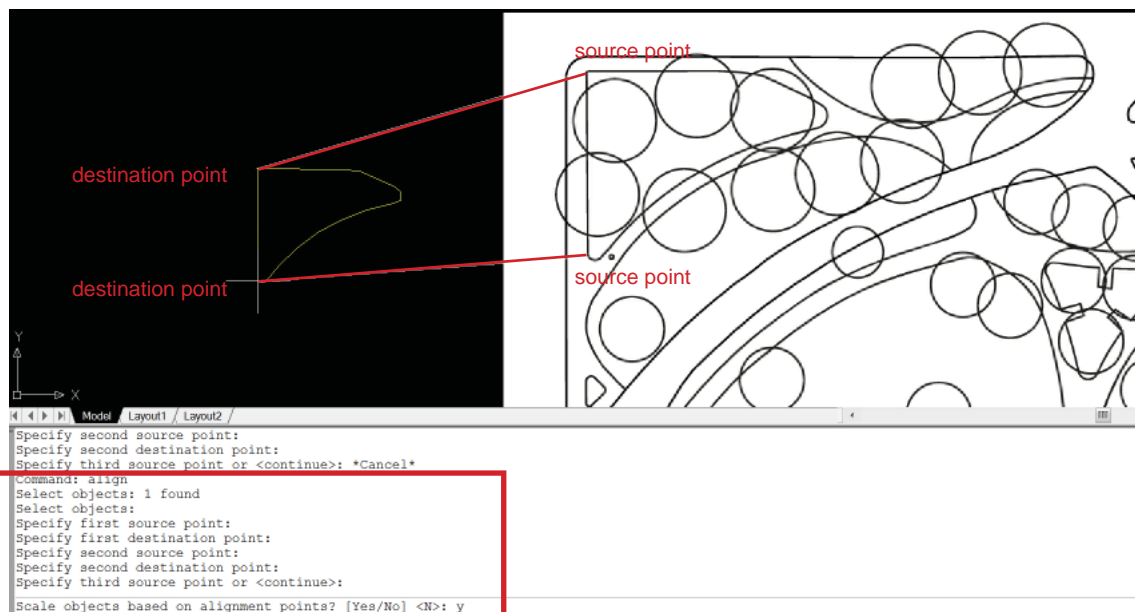
For third point, normally just press enter.
Whether scale object based on alignment points is based on case by case.
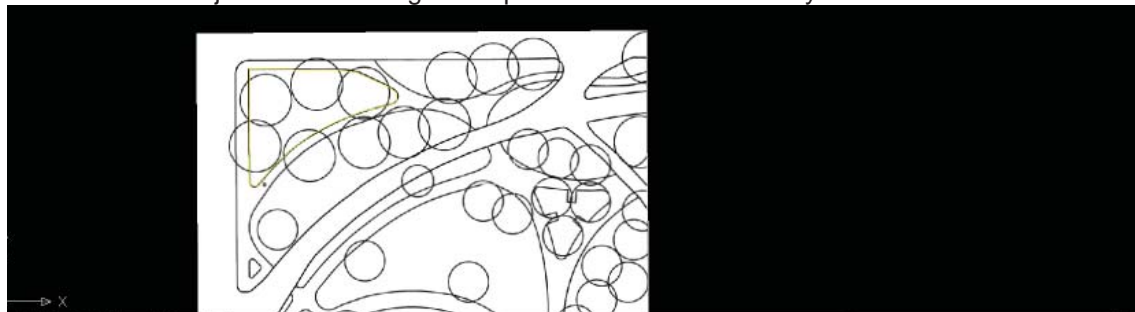
SCALE(sc)

You can scale the image if you know a dimension of an object.

i.e. You know the length of the yellow edge is 2. You can scale the entire image based on it.

press enter after finishing selecting

If you know the scale, type it here. If you know a dimension of an object in your image, type R

click one end of the edge and then the other

type the new length

```
Command: sc SCALE
Select objects: Specify opposite corner: 1 found
Select objects:
Specify base point:
Specify scale factor or [Copy/Reference] <0.6000>: r
Specify reference length <2.0000>:  Specify second point:
Specify new length or [Points] <1.0000>: 2

Command:
```

insert dwg.file- external reference (xref)

less useful at school but very important when you collaborate with other people in the office

When there is a file that you want to view in your drawing and are not intending to modify them, use external reference.

*Usually you are not supposed to modify files that you recieved from others.

* before you start modifying and xrefing any existing documents (especially from external sources), it is a good idea to clean the files by purge and audit.

PURGE (pu): Removes unused named items, such as block definitions and layers, from the drawing
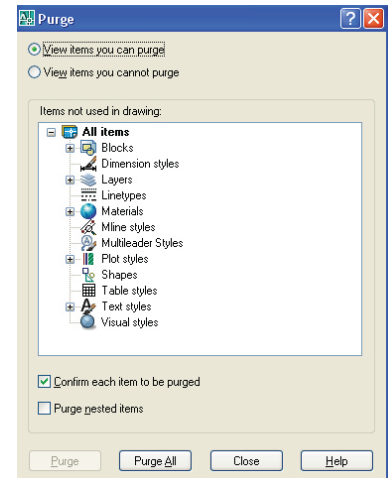
purge, click "yes to all" (in most cases, you want to purge everything)

AUDIT (au): Evaluates the integrity of a drawing and corrects some errors

audit, (y) for fixing errors automaticlaly

Attach vs overlay: generally choose overlay, unless you are submitting the files themselves to clients.

Attach = the attached xref will be carried to the next tier of files
i.e. if you xref a file that has attached xref, you will see both xrefs

Overlay = the overlaid xref will not be displayed in the next tier of files.

* it is commonly thought that the information itself is not carried by overlay. However, that is not always true. Sometimes overlaid information is carried as ghost and start cross referencing each other. This could cause slowing down the file and possible damage.
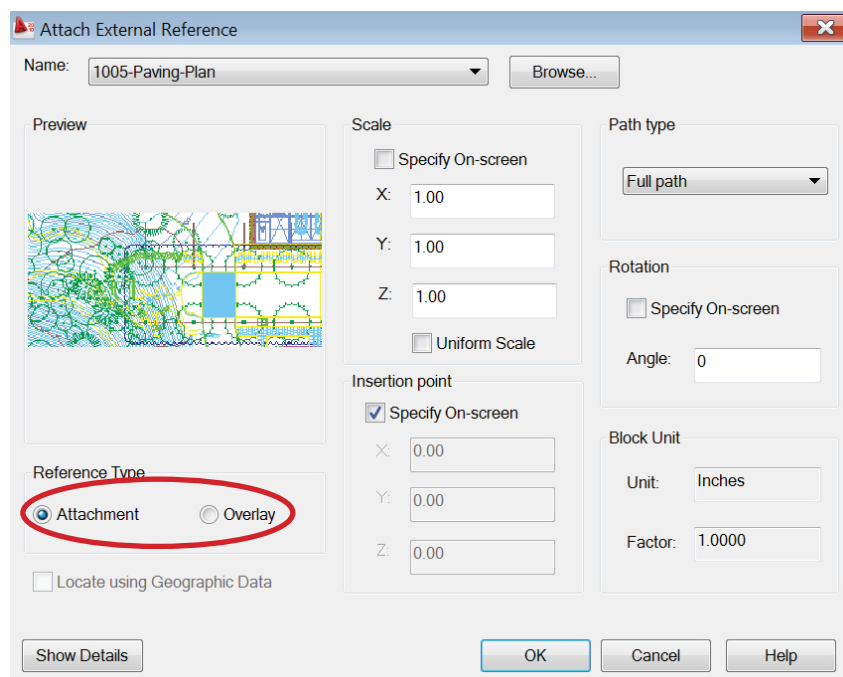
Browse the file, choose attach or overlay, choose insertion point , scale, rotation, in the case, it is a plan that has same ucs (origin 0,0) and drawn to full (1:1) scale, leave boxes unchecked.

If you are xrefing a file that is not created with same ucs, either pick the insertion point, or move the file and align after you attach it.
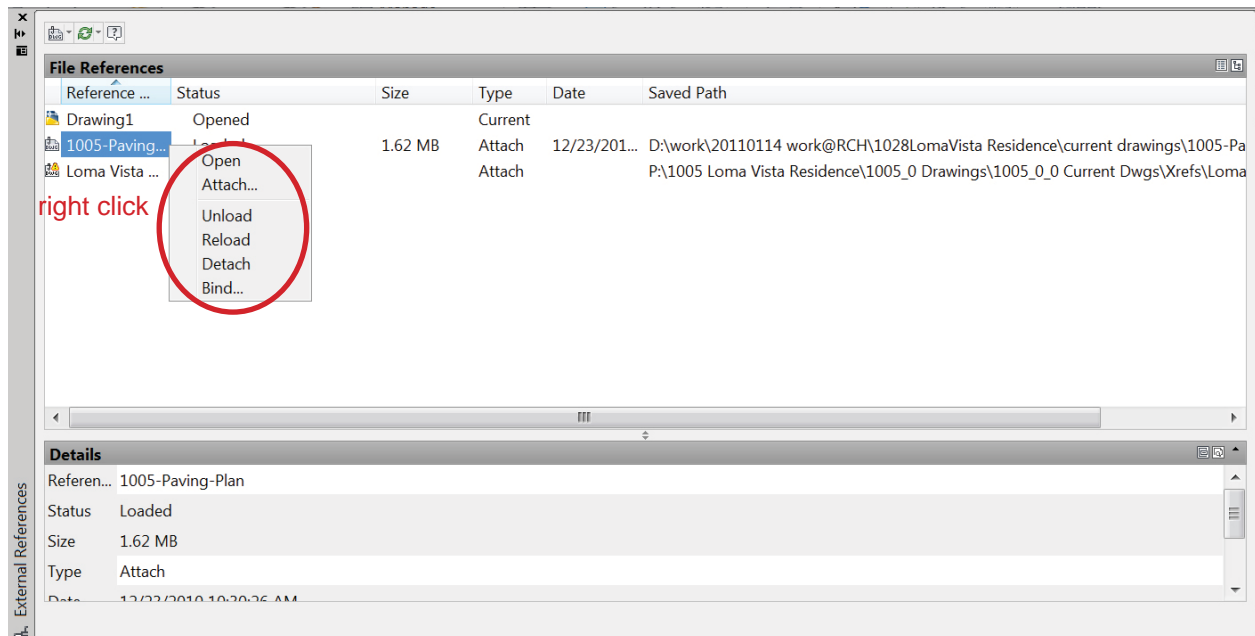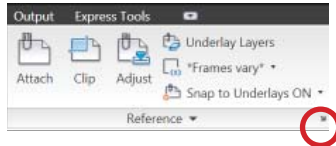
* If you do not see anything, zoom extend and find out their location.
When it does not match, make sure your ucs is "world"

xreference manager

Unload vs **Detach**: Detaching delete the xref. Unloading just makes xref disappear from the screen.

Modify xreference

When you want to modify an xref file through the working file:
- refedit (or double click), save the change.

> This option is not the most stable way to edit an xref.   Unless you want to do a few very simple things, open the original drawing, edit and save it, then reload the xref in your working drawing.

When you want to copy objects from an xref drawing to your current drawing:
- Nested object copy (ncopy), select objects, enter, click the paste destination.

> * there are limits to the number of objects to select at a time

We are going to cut a simple section, ground line and a structure.

The basic idea is same as hand drawing.
- Make a section cut
- Extend the height

For example, on the right drawing, there is a 100' tall tower on the top of a mound. There are 10' contour lines.

Draw a section cut line.

Make lines from the intersection of the section cut and each contour line & structure line by typing the numbers (this drawing unit is in ichi = # should be multiplied by 12).

You can either connect lines where the reference lines are, or make a copy of lines, move them to the clear area, then connect them.
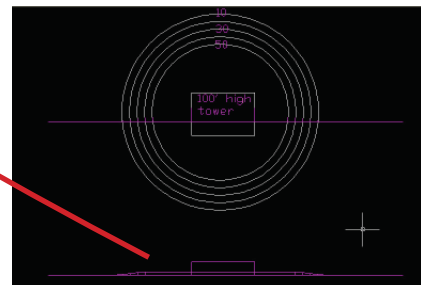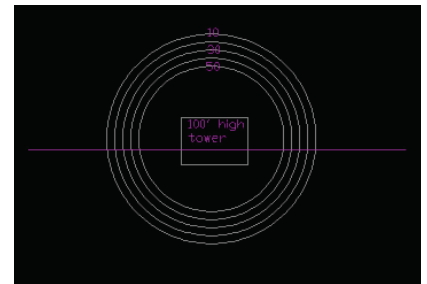
Now, let's practice on Carr's hill plan.
Open Carr's Hill Base file and zoom into Payton hall parking lot area.

Let's create a new layer called "Section" and make the layer current
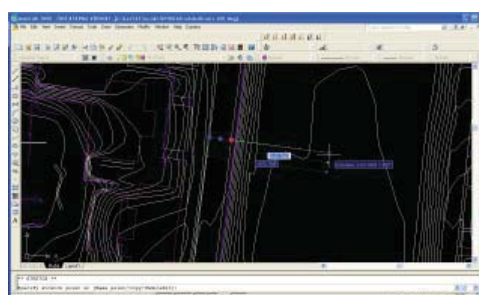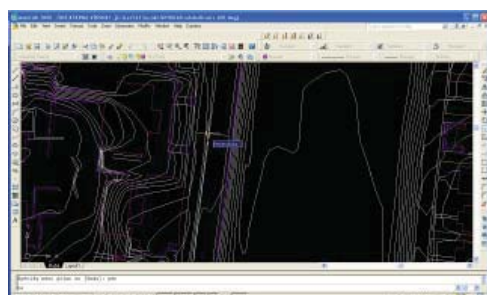
<click layer manager or (layer), click new layer icon, name the layer>

* If you are creating a section/elevation that includes multiple objects, you may want to create a layer for each object and color code them for your reference.
i.e. structure layer = red, trellis layer = blue, tree layer = green

Draw a line close to mid point of the football field, perpendicular to the street.
<line (L), click the first point, perpendicular (per), click the other side of the street>

Extend the line 200' (2400") to the both sides.
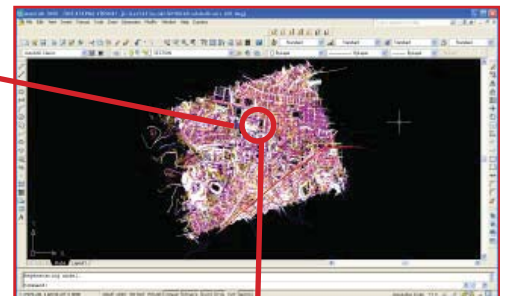
When the plan is angled like this case, you can either:

1) Rotate UCS (User Coordination System) accoding to the section line.

or

2) Make the section cut as is and rotate the reference lines later.
(This works when you can't change UCS for some reason, and the section is short and simple)

After completing the section, you should change UCS back to default (World), especially when sharing a file with someone else.

1)
Rotating the view

We are going to rotate the view based on the section line. We use **UCS** command.
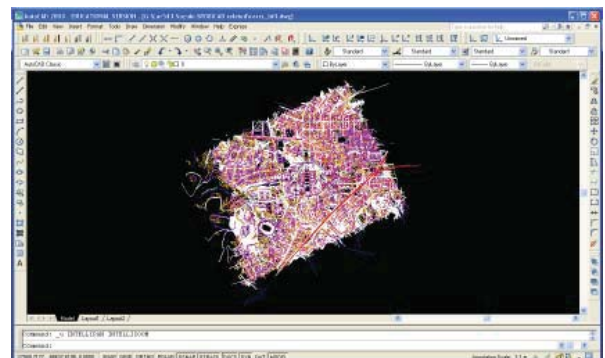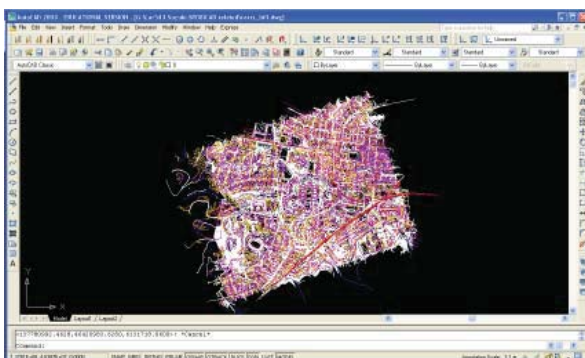
There are 2 steps

1. Creating new UCS
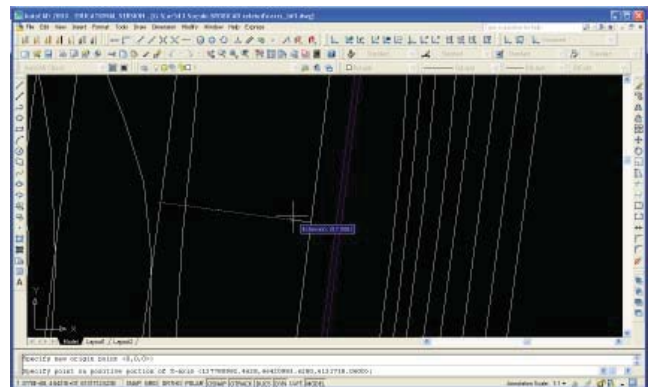<type (UCS), then (e) to define XY cordinatnts by an element you want to align the X axis to, in this case your section line>

2. Changing the view according to the new UCS
< type (plan), choose current ucs or just hit Enter>

You can create section reference lines using **ortho snap**, you can save the file specificly for section or you can change the UCS back to **"world"** when you are done.

2)
Now we can try the way without changing the UCS.

There are multiple ways to do this. Drawing perpendicular reference lines to the section cut is one way.

To create perpendicular lines easily, turn the **polar tracking** snap on and open the setting by right clicking on it.

In "Polar Tracking" tab, "Polar Angle Measurement" section, click on "Relative to last segment". The guideline perpendicular to the section cut will appear.
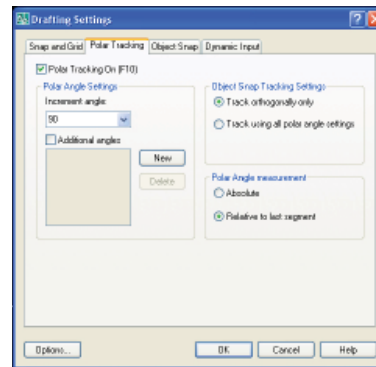
Draw perpendicular lines from the section cut intersections. When OSNAP intersection does not appear where it looks 2 lines intersect, try **Apparent Intersection**. This is because some lines have z values (elevation) and 2 lines are not exactly intersecting. Apparent Intersection selects intersected objects on the plan view.
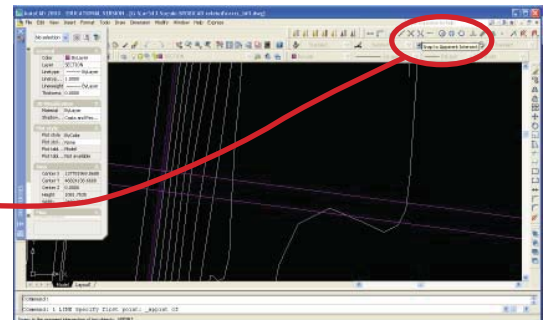
<line (L), click Apparent Intersection or OSNAP apparent intersection on, click the first point and start a perpendicular line, move the cursor on the top of the second line, type in the height of the reference line>
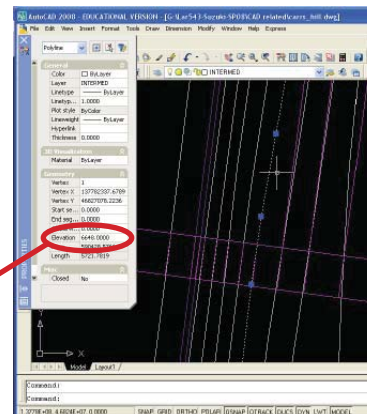

Extend the lines to the appropriate height.

In this example, we do not have contour lines labeled. Let's check the elevation in the property box and use that number.

Grab the end point at the section cut. Move it up and find the point that says extension. Where it shows "extension", lock the point and type #s. When the "extension" does not show up, it usually helps to put the cursor over the original endpoint.

For fence, add 40" on the top of the elevation, for the structure add 240" on the top of elevation.
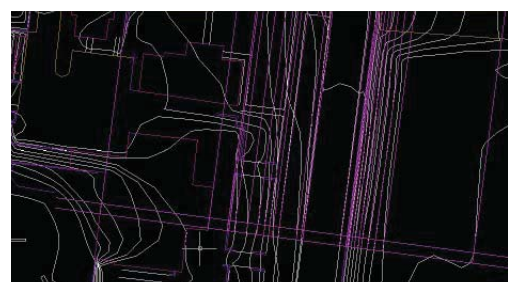
When you are done with making lines, let's copy them and paste where there are no other lines.

In order to select only the section line, you can use the command layer isolate (layiso).

<type (layiso) or [ ] layers/layer isolate, select the lies on the layers that you want to isolate, enter>

Now you can only select the lines/objects on the isolated layers. You can change the other layers off/freezed/locked.

<select the lines, copy, paste above>

Now rotate the lines.

<Turn on Ortho, darw a horizontal reference line from one end of section cut line, select all the section related lines except the reference line, (ro) for rotate, base point is the shared endpoint, (r) for reference, click the shared endpoint, then the other end of the line you want to move, click the the other endpoint of the reference line>

Now you connect the lines.
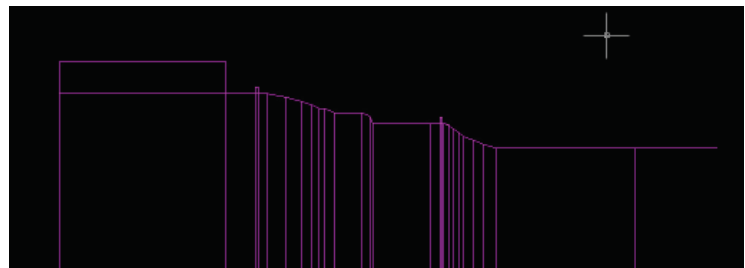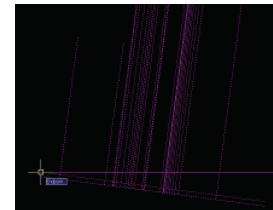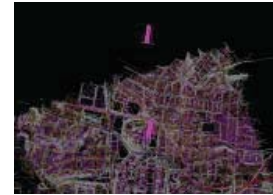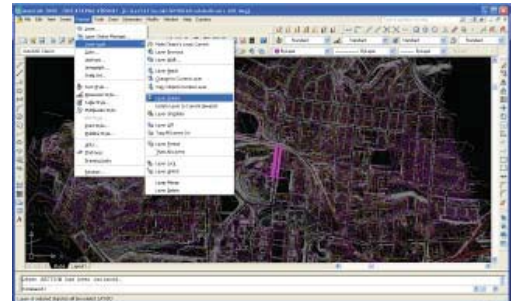
Xclip:

You can "clip" xrefs to display only parts of them. It is useful especially when you want to make a partial section so that your section is not too far away from the cut line on the plan.

Type "xclip" and enter
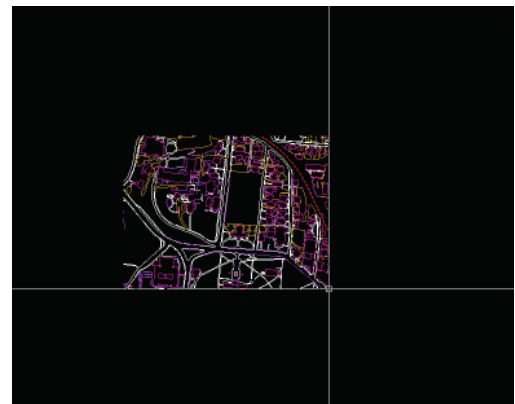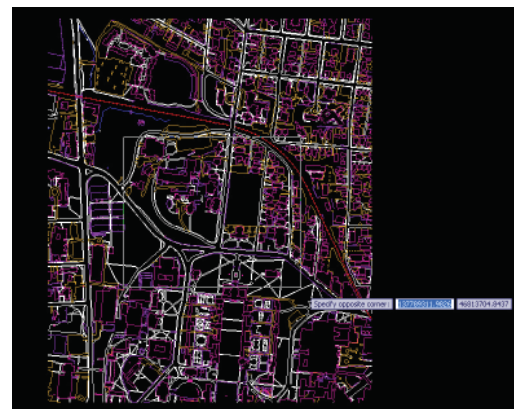
Click on the object on xref that you want to clip then enter

Type "n" for New Boundary

Type "s" and select the clipping boundary if you already have it drawn in CAD

OR

Type "r" for drawing a rectangle clipping boundary or "p" for polygonal

Area calculation

Create a new layer and give a clear name like "take off."

Make polygons of the areas that you want to know.

If it is one connected polygon area, you can simply select the polygon and "list."
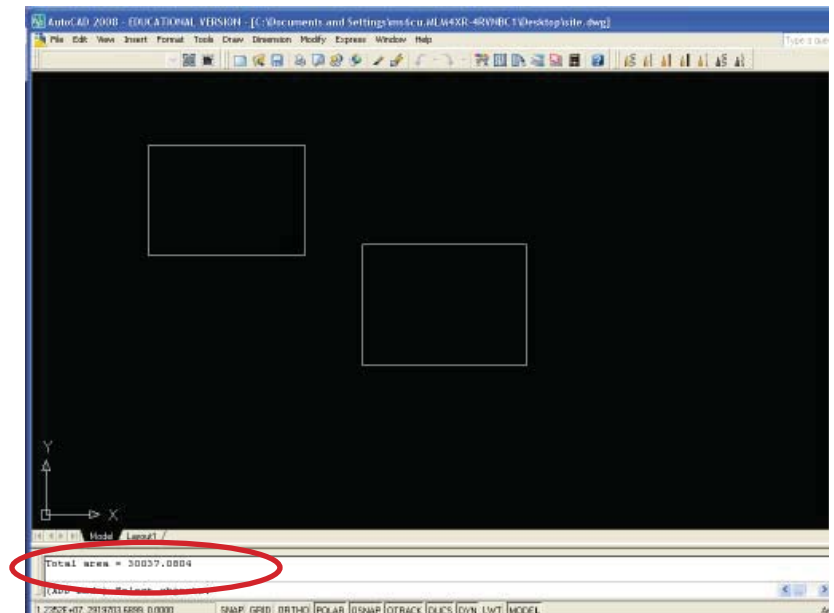
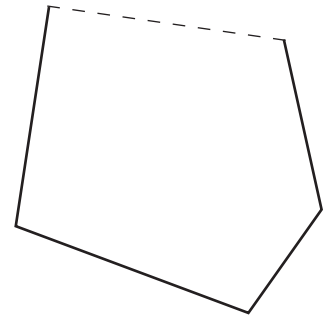If there are multiple polygons, you can use the command "area."

AREA (aa)

"a" for add, "o" for object, select polygons that you want to sum up.

Total area is shown in the command line.
You can subtract the areas by typing "s" and select shapes to subtract.

When you select an open object, CAD will automatically bound the area connecting the start and end points with a straight line.

Plot

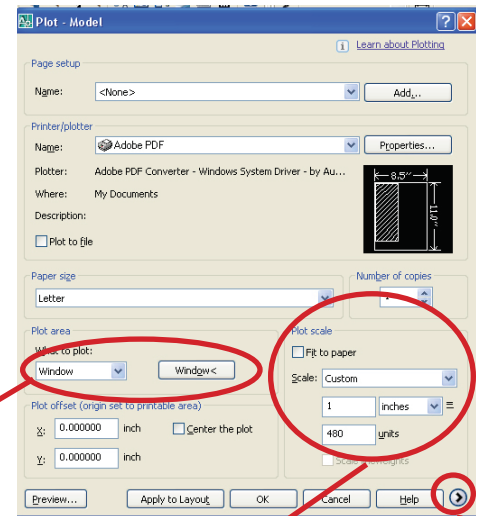You can plot either from the model space and layout space.

Plotting from the model space is used for quick not-to-scale reference printing or in-house basemaps.

Plotting from model space:

While you are in the model space, go to print, the window on the right pops up.

Choose paper size.

Select "Window" and draw a rectangle around where you want to plot.

Check "Fit to paper" if the scale is not important. If you want to print in scale, put the appropriate number in the box.
i.e. 1" = 40', 40 x 12 = 480
you type 480

By clicking the arrow on the bottom right, you get more options.

For plot style, you should select appropriate pen settings for the project. The CAD default is "acad".

CTB files translate the line colors on your drawing to line widths, types and tones using index color system. Firms often have their own standard settings, typically color 1 (red) being the thinnest line and lines becomes wider as the color number goes up.
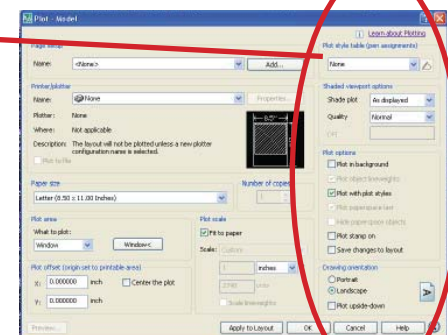
You can also create or edit the CTB file by clicking the pen symbol on the right top.

Some machines at A-school do not load CTB files. If that is the case and you want a monotone print, create your own CTB file or change all object colors to monotone and assign line width manually by individual object or layer in your drawing.
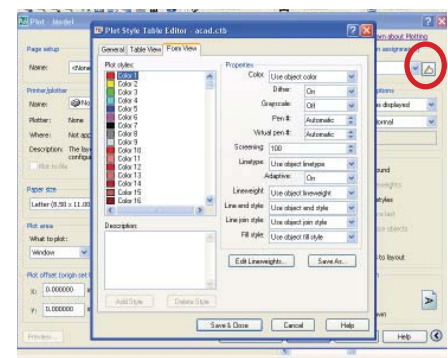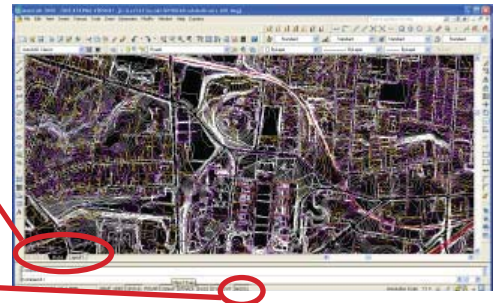
Plotting from layout:

Layout: You can change model space and layout space by clicking the tab on the bottom of display.

Layout is a sheet setting. Layout has 2 different spaces;
1) model space, 2) paper space
You can check which space you are in and change the status by clicking the tab on the fat bottom.
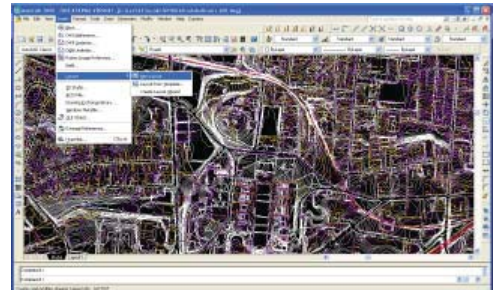
Make a new layout

insert>layout>new
or
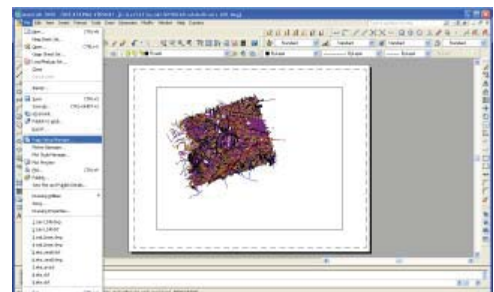right click the bottom tab "layout 1"

Name the new layout "11x17" - because that will be our page size

Go to the layout "11x17"

Open the "page setup manager"

Select "11x17" and click modify

Same as plotting from the model space. Select "acad" for this exercise.

Select appropriate printer and modify the propertoes by click the property button and go into property windows, if necessary.

For this exercise, choose Adobe PDF.

Choose Tabloid/11x17
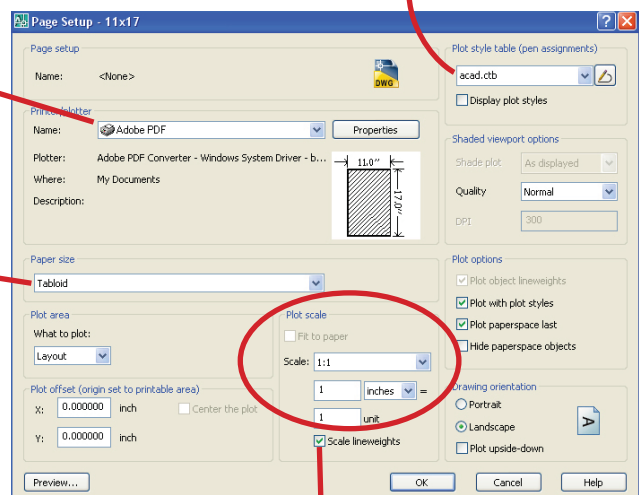
Make sure that the plot scale is correct.
If your dwg units are in inch, the default is right
1"=1unit

If your dwg units are in feet, that would be
12 inches = 1 unit

If your dwg units are in meter, you should choose millimeter instead of inch, then type

1000 millimeter = 1 unit

In metric CAD file, layout/plot space is alwasy in mil-limeter. Model space units vary.

Check "scale lineweight" box when you are plotting in smaller scale or size, so that the lines are not too thick.

Layout uses a frame called "viewport" to display the model space. A sheet can have multiple viewports and you can turn on/off certain layers per viewport.

Let's delete this existing viewport.

Create a layer called "mview" in the layer manager.

We want to create a new viewport.

View>Viewports>New Viewport
<(mv) for mview, (f) for fit, enter>

Fit automatically gives you the maximum fit for the sheet. You can adjust the rectabgle afterwords too. If you want to be able to create non rectalinear shape, choose (p) for polygonal and start drawing the box manually.

By double clicking inside of viewport, you can go into the model space and modify the objects in the model space. Double clicking outside of the viewport or clicking the tab on the bottom put you back into the paper space.
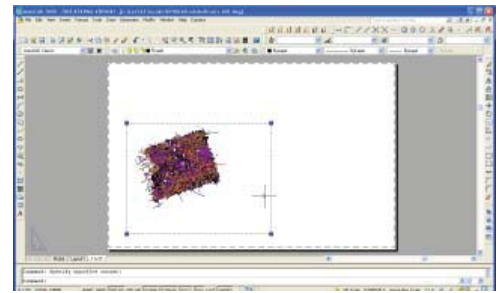
Scale:

To scale the model to plot, you want to know approximate scale that you want to print.

In this case, I want to plot 1"=200' (200 x 12 = 2400")

Double click and go into the model space through the viewport, <zoom (z), type (1/2400xp)>

* "xp" refers to "in paper space"

Adjust the location (when you do this, do not touch the middle wheel and change the scale...)

Get out to the paper space by double clicking outside of the viewport and plot!

* You can always preview too. If the print does not come out with the right orientation, although the preview is correct. Usually it is the property setting of the printer, rather than the page set up of the CAD.
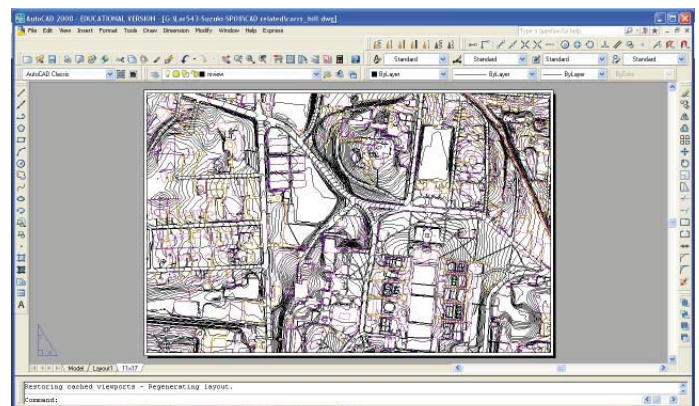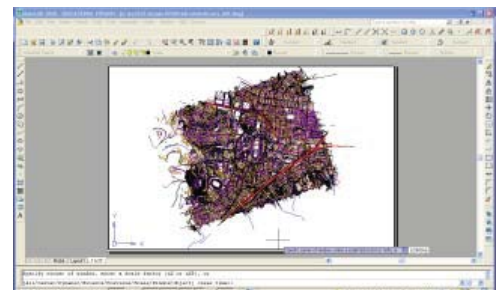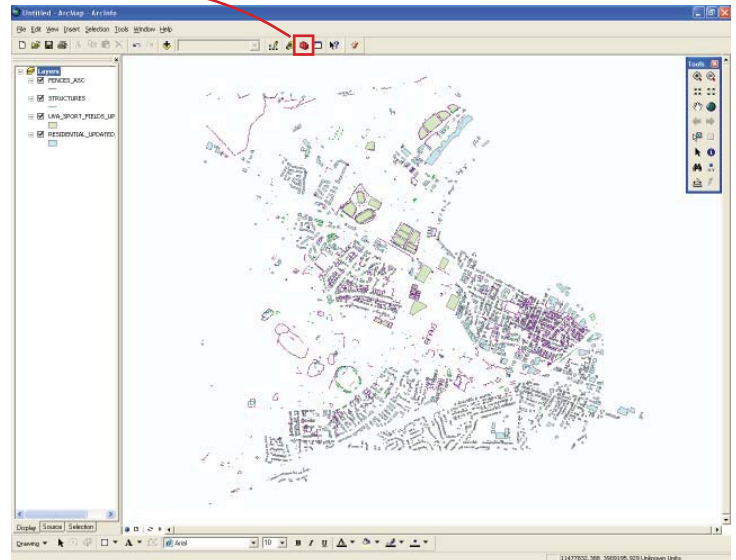
Import & export

   **import from GIS to CAD**

In ArcMap (version 9x or up)

Open Arc Toolbox

Select Export to CAD

Conversion Tool > To CAD > Export to CAD

* When you can't find the tool you need, you can keyword search by clicking the search tab on the bottom of toolbox window. When you see the appropriate choice, you can either double click to open the tool or click "locate" to see where the tool is in the original view, "favorite" tab.
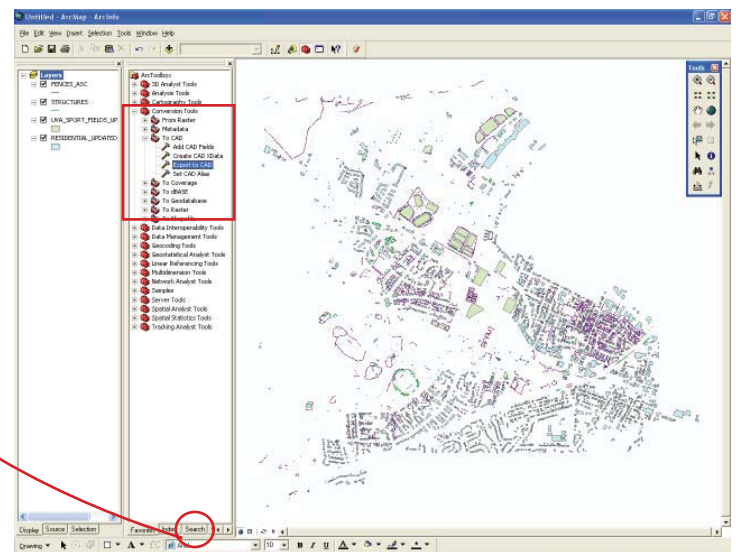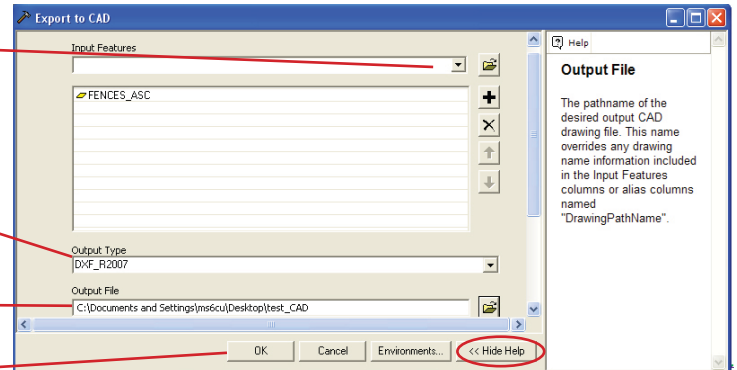
Choose the layers that you want to export from the pull down bar on the top.

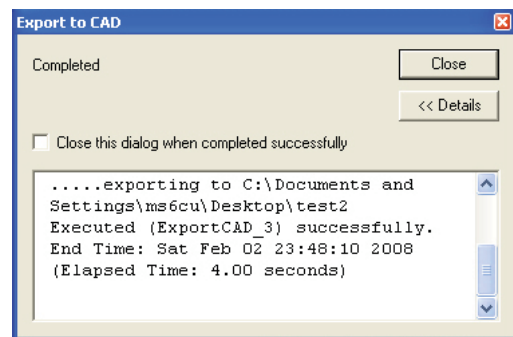Select appropriate version of DWG

Specify the location and name of the file (may require you to type the extension ".dwg"

Press OK

* Show Help button offers brief explanations on the right.

Confirm completion and open the exported dwg file in AutoCAD.

In AutoCAD

REMEMBER, you normally have to rescale the drawing x12 because the unit of GIS data is usually in feet, but the default unit of CAD is usually in Inch.

Re-Scaling the drawing

(Scale) > (Select) >(All) > enter > specify appropriate base point by clicking the point or typing the value such as (0,0) > (12) for scale factor.

Zoom extent to see the whole drawing.
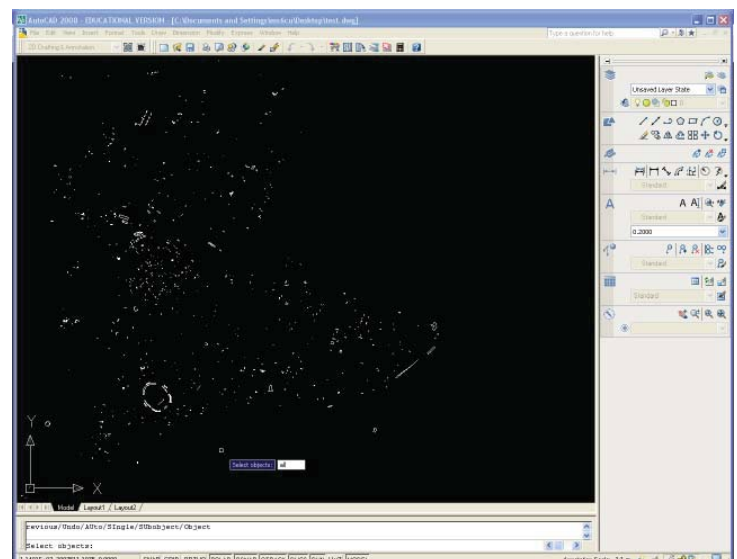
export from CAD to Sketch Up

preparation
Turn off all unnecessary layers such as dimensions, furniture,etc.
Save a new file to import into SketchUp. Don't use your original CAD files
You will need to know the CAD drawing units: inches, feet or meter.
If the objects in the drawing don't need to have Z value, flatten all by using
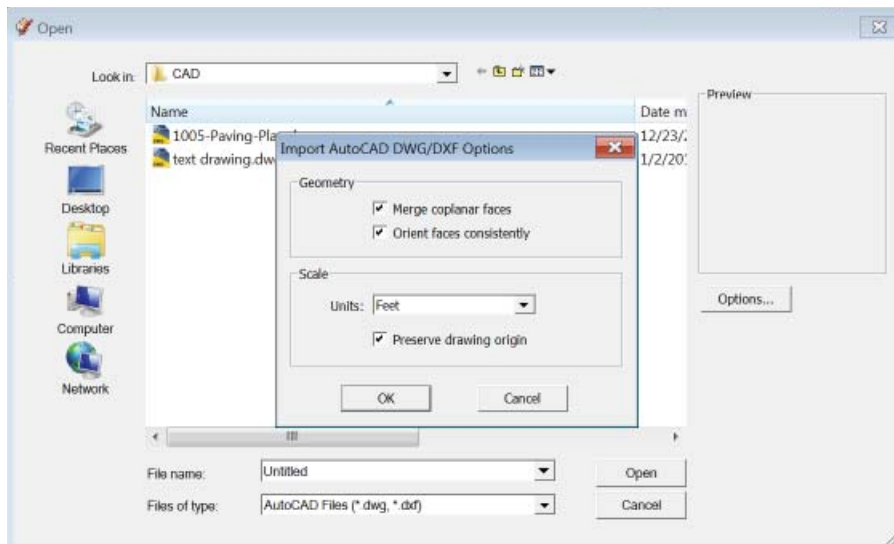"flatten" command incase some objects has wrong Z value.
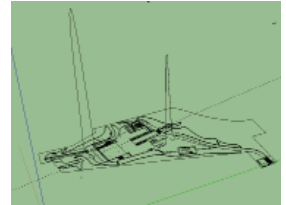
wrong Z value

import
File>Import>choose dwg. file,click "Options"
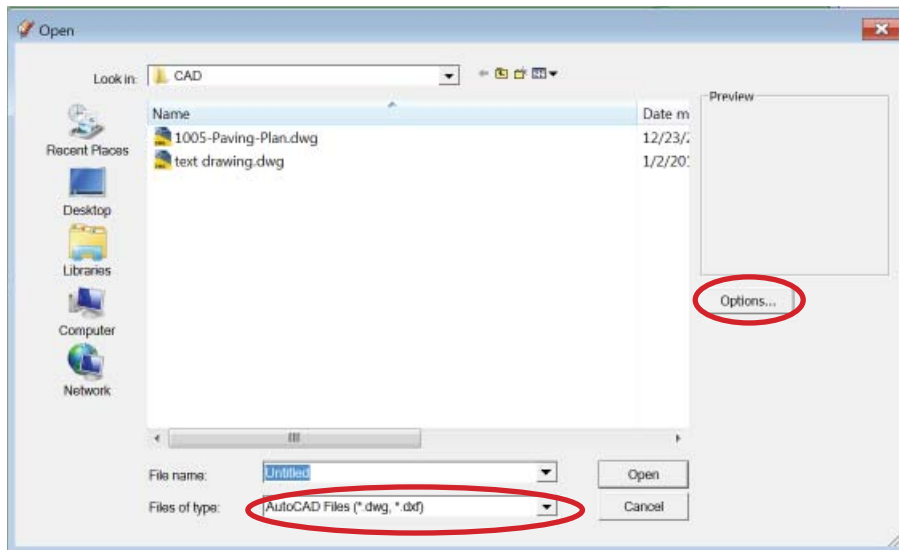
Check the three boxes and match the Units with your CAD drawing.

Check a object's length you know by "type measure tool" to make sure the scale is correct.


If your CAD version is too high to be compatible with your Sketch Up, you wont' be able to import it to Sketch Up successfully. If so, you need to save the CAD drawing as a lower version drawing.

There are two ways to use the CAD drawing.

1. Use the CAD drawing as a reference
select all the layers except for Layer0> click "-" button> check "move contents to default layer">set up a new layer by "+" button>put the drawing into the new layer
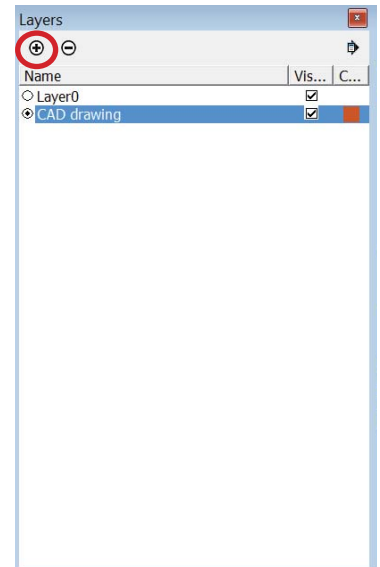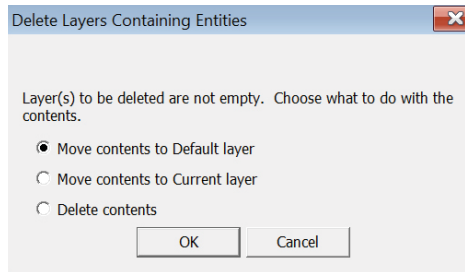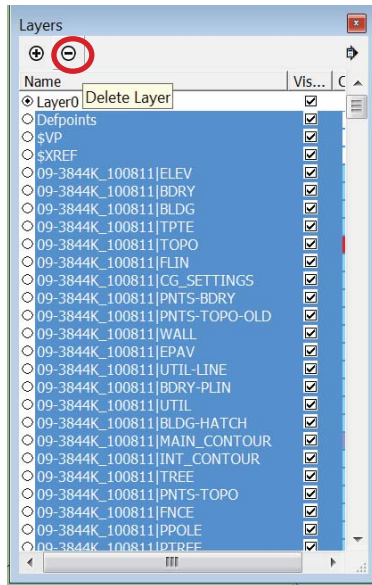
2. Create surface in the CAD drawing's layers, especially when the shape is hard to trace in Sketch Up

You need to close the shape in CAD.It is better to make the outline as a Polyline by command "pedit(pe)". In this way, you can double click to enter the group and trace one edge in order to make the surface.

This method is useful when you only import one layer or a few layers. So you don't need to trace them again.If you import a lot of layers at one time, using Method 1 will avoid to mass the Sketch up drawing up.