## Introduction To AutoCAD

AutoCAD is a computer-aided design program (CAD) used for 2-D and 3-D design and drafting. AutoCAD is developed and marketed by Autodesk Inc. And was one of the first CAD programs that could be executed on personal computers.

Learning to use a CAD system is like to learning a new language. It is necessary to begin with the basic alphabet and learn how to use it correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary.

In other ward AutoCAD is a computer-aided drafting software program used to create blueprints for buildings, bridges etc.

As known it is impossible to do any work without plan, the same theory applies to the creation of big buildings, amazing architectures or any machine design. We need a perfect plan/blueprint to create those things.

Now a day's technology is growing day by day, so to stay with the time we must run with technology, AutoCAD software helps architects,

## مقدمة

AutoCAD هو برنامج رسم هندسي بمساعدة الحاسوب, يستخدم للرسومات الثنائية والثلاثية الابعاد ويتم تطويره وتسويقه بواسطة شركة Autodesk ويعتبر من البرامج التي يمكن تنصيبها واستخدامها بسهولة على الحاسب الشخصي.

تعلم استخدام برنامج AutoCAD هو شبيه بالتعلم للغة جديدة برمجية كانت ام كلامية, لذلك ومن المهم في البدء التطرق الى اساسيات هذا البرنامج وتعلم استخدامها بشكل صحيح وفعّال من خلال التطبيق العملي على البرنامج وهذا يتطلب تعلم مفاهيم ومهارات جديدة وهي شبيهة بتعلم مفردات اللغة.

يمكن تعريف برنامج AutoCAD بانه برنامج صياغة لرسومات بمساعدة الحاسوب يمكن من خلاله انشاء مخططات للابنية والجسور وغيرها الكثير الخ...

عند تنفيذ اي عمل وكما هو معلوم فمن المستحيل تنفيذه بدون مخطط لهذا العمل, ونفس النظرية تطبق عند عمل الابنية فاننا نحتاج الى مخطط متكامل وملائم لتنفيذ هذه الاعمال.

التكنولوجيا في هذه الايام تتطور شيأ فاشيأ لذلك يجب علينا مواكبة هذا التطور التكنلوجي, برنامج AutoCAD يساعد المصممين المعمارين والمهندسين الخ.. لانشاء تصميم معين ومحاكاة لهذا التصميم قبل التنفيذ حيث ان ذلك يوفر من صرف المال والايدي العاملة والوقت.

designers, engineers, etc. To create designs and test it, before spending money, labour or time.

AutoCAD provides countless methods and tools for producing, viewing, and editing two dimensional drawings and three-dimensional models.

- **AutoCAD Interface**

1. AutoCAD Version Number: Shows what AutoCAD version you are running.
2. Current Open Drawing Name: Shows the name of the drawing you currently have open.
3. Pull down menus: Easy access to AutoCAD commands.
4. UCS Icon: is short for “user coordinate system.
5. Command Line: Used to type in AutoCAD commands from keyboard.
6. Layers: Access AutoCAD layer command.
7. Coordinate readout: used to keep track of cursor location in drawing area.
8. Grey part: When Tabs or Panels disappear sometime, you can right-clicking on the grey part of the menu bar to find them.
9. Status bar: used to turn on and of AutoCAD setting.

يوفر برنامج الاوتوكاد طرق وادوات لا تحصى لانشاء وعرض وتعديل الرسومات الثنائية والثلاثية الابعاد.

- **واجهة الاوتوكاد**

1. سنة اصدار البرنامج : توضح سنة الاصدار للبرنامج الذي تعمل عليه وكما موضح بالشكل 1
2. اسم العمل الحالي ( الرسم الحالي ) : يوضح اسم ملف الرسم الذي تعمل عليه الان.
3. القوائم المنسدلة : للتعامل مع اوامر البرنامج بسهولة.
4. الاحداثيات : وهي مختصر (نظام الاحداثيات للمستخدم) وهي مهمة جدا للتعامل مع البرنامج خصوصا مع الرسم ثلاثي الابعاد.
5. شريط الاوامر : يمكن من خلاله ادخال الاوامر الى البرنامج من خلال لوحة المفاتيح.
6. الطبقات : ويمكن من خلالها ضبط الطبقات التي تستخدم في العمل وذلك لتسهيل الرسم وتمييزه وكما موضح بالشكل 2.
7. قراءات الاحداثيات : توضح وتتبع احداثيات المؤشر في ورقة الرسم.
8. المنطقة الرمادية : لاخفاء او اظهار الاشرطة التى تحتوي على الاوامر وبحسب رغبة المستخدم.
9. شريط الحالة : يستخدم لاطفاء او تشغيل اوامر ضبط واجهة الرسم في البرنامج.

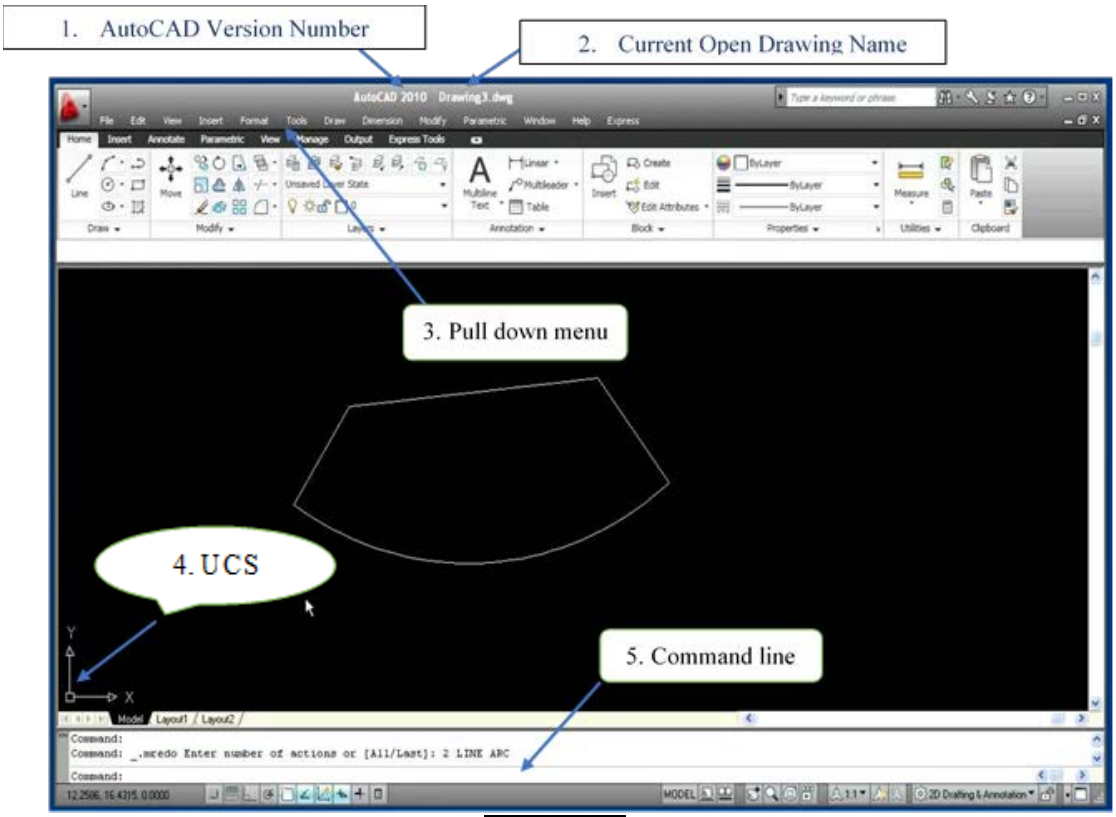



Figure 1

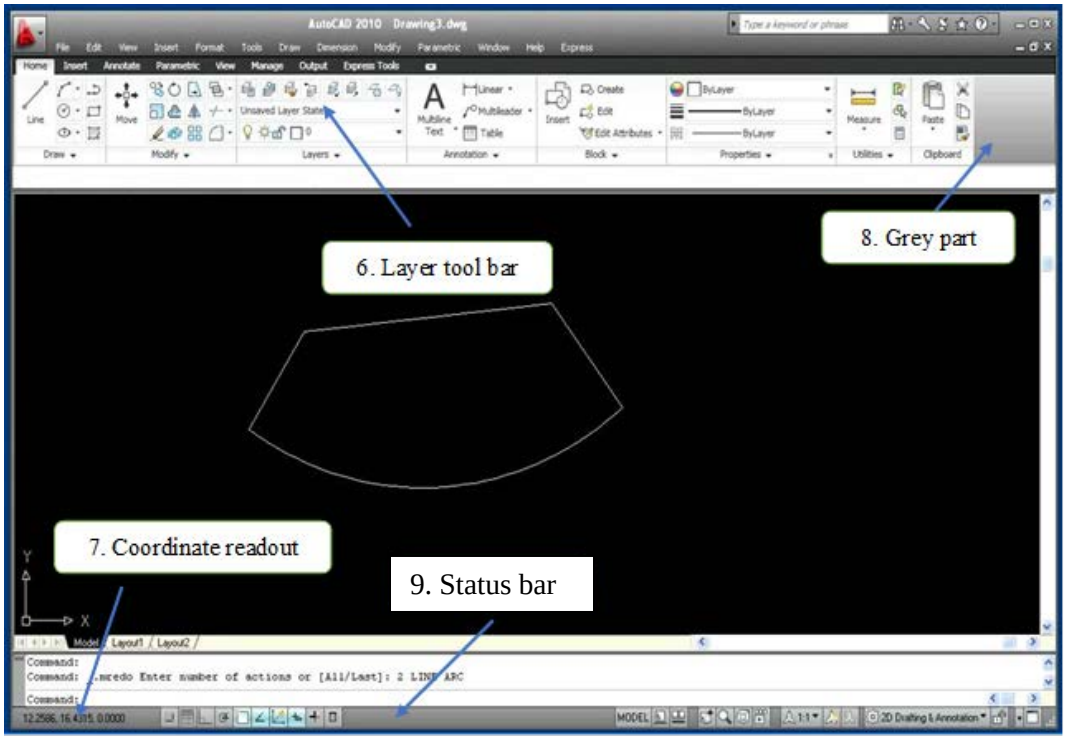



Figure 2

- **Starting Up AutoCAD® 2010**

Select the AutoCAD 2010 option on the Program menu or select the AutoCAD 2010 icon on the Desktop.

- **Create and Save AutoCAD drawing files**
- **Create**

The Figure 3 show the method to create new sheet.

The default format is “acad.dwg”. In practice field.

- **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.

The drawing is saved as AutoCAD 2010 drawing as default. If you want to open it with lower version of AutoCAD, you must save it as a lower version. Figure 4 below illustrate the method of saving.

- **Drawing Units Setup**

Every object we construct in a CAD system is measured in units. We should determine the system of units within the CAD system before creating the first geometric entities.

- **تشغيل البرنامج**

لتشغيل البرنامج اما ان نذهب الى قائمة Start ونضغط على ايقونة البرنامج المتوفر في القائمة او بالضغط بشكل مباشر على ايقونة البرنامج والموجودة بشكل قياسي على سطح المكتب.

- **انشاء وحفظ ملف الرسم**
- **انشاء ملف رسم**

الشكل 3 يوضح طريقة انشاء ملف رسم جديد

عند تشغيل ملف رسم جديد فان الامتداد للرسم هو dwg

- **الحفظ**

عند تشغيل برنامج AutoCAD فانه يتم تشغيل ورقة رسم تحت مسمى Drawing.dwg يمكن العمل على هذه الصفحة وعند الانتهاء يمكننا الحفظ باحد الطرق التالية :

ملف الرسم يحفظ وبشكل قياسي تحت مسمى AutoCAD2010 drawing ولكن من المفضل حفظ ورقة العمل بنسخة ذات اصدار اقدم من النسخة المستخدمة ليتلائم مع جميع نسخ البرنامج القديمة. الشكل 4 يوضح عملية الحفظ.

- ضبط وحدات الرسم

كما هو معلوم فأن اي مخطط نتعامل معه في البرنامج له ابعاد وهذه الابعاد بوحدات معينة لذلك لابد في البدء من ضبط هذه الوحدات قبل البدء برسم المطلوب وكما موضح بالشكل 5.

*برنامج AutoCAD هو برنامج بلا وحدات unitless لذلك فأن ضبط الوحدات في البدء الغرض منها عند تصدير المخطط من برنامج AutoCAD الى البرامج التحليل والتصميم.

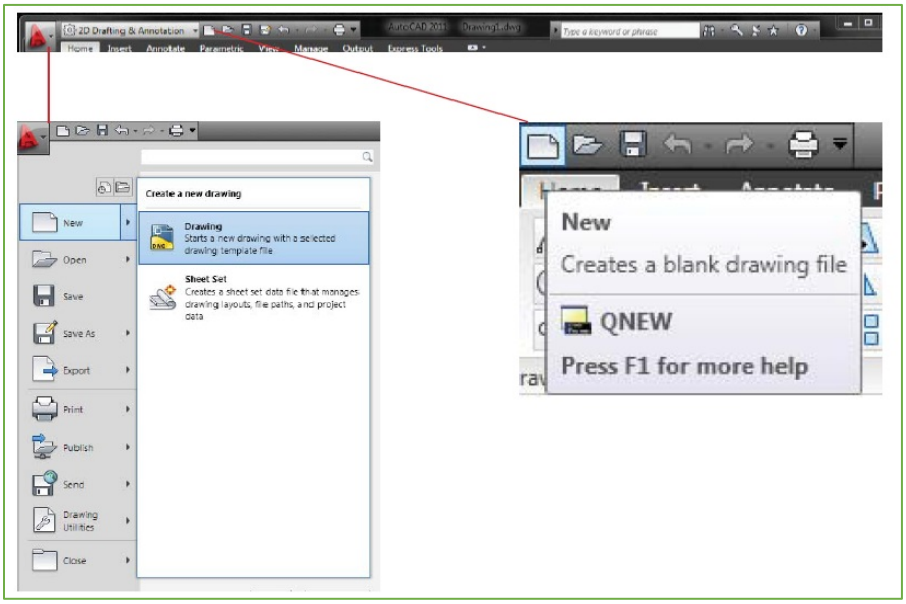



Figure 3

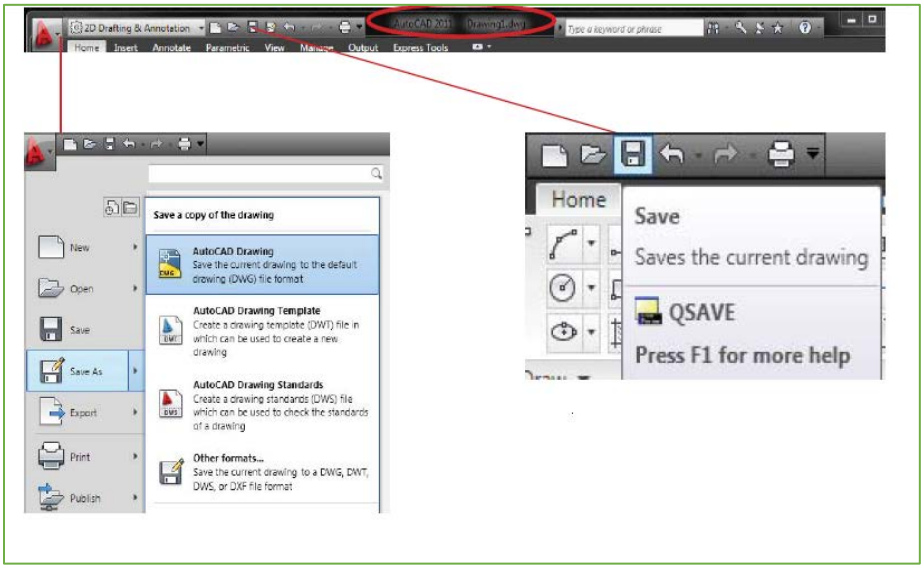
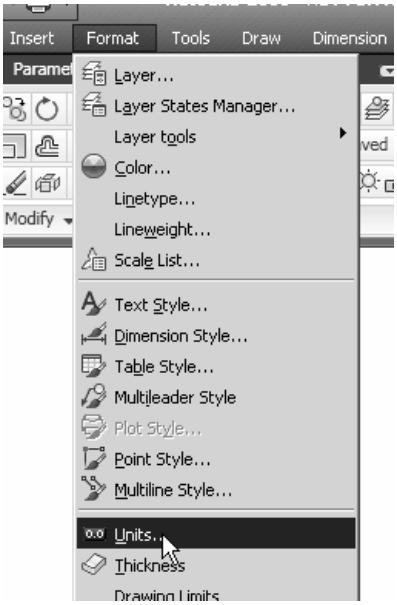



Figure 4



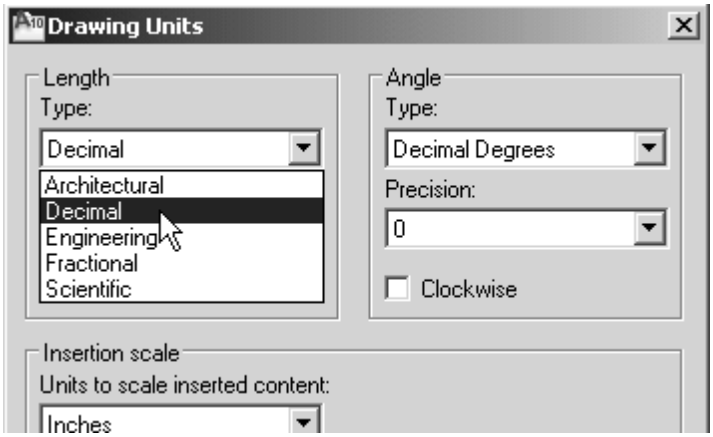




Figure 5

- Commands in AutoCAD

You can start a command by doing one of the following:

1. Select the command from a menu, tool bar.
2. Enter the command name or command alias at the Command line and press ENTER or space bar.

All AutoCAD commands can be typed in at the command line. Many commands also have one or two letter aliases that can also be typed as shortcuts to the commands.

**TIP**: Many AutoCAD commands require you to press **ENTER** to complete the command.

- الاوامر في برنامج AutoCAD

يمكن البدء باحد الاوامر في البرنامج عن طريق عمل الاتي:

1. اختيار الامر من القائمة المنسدلة او من شريط الادوات.
2. ادخال اسم الامر او المختصر للامر عن طريق شريط الاوامر والضغط على مفتاح enter or spacebar من لوحة المفاتيح.

*جميع اوامر AutoCAD يمكن ادخالها عن طريق شريط الاوامر, العديد من الاوامر لها حرف او حرفان مختصر يمكن كتابتها في شريط الاوامر لتنفيذ امر معين.

تلميح : العديد من اوامر AutoCAD تتطلب الضغط على المفتاح enter من لوحة المفاتيح لانهاء العملية.

- **Reissuing the Last Command**

The last used AutoCAD command can be re-entered by one of the following three methods:

1. Press the **ENTER** key on the keyboard
2. Press the Space bar on the keyboard.
3. Click the right mouse button.

- **Status bar**

**The important commands in the status bar are as follow:**

- **Ortho**

Ortho means orthogonal, allows us to draw horizontal or vertical lines quickly and easily with restricted all other angles.

- **Polar**

Polar command used to draw lines at regular angular increments, such as 30,45, or 90 degree or any other angle entered by user.

- **Object Snap**

One of the more important command in AutoCAD, when its on the work become easy and quickly, Figure 6 illustrate the OSnap window.

- **اعادة استخدام اخر امر**

يمكن اعادة استخدام اخر امر مستخدم في AutoCAD بعد الانتهاء منه من خلال عدة طرق وهي كالاتي:

1. الضغط على مفتاح enter من لوحة المفاتيح.
2. او الضغط على مفتاح Space bar من لوحة المفاتيح.
3. او الضغط على الزر الايمن للماوس واختيار اخر امر اعادة الاستخدام.

- **شريط الحالة**

**اهم الاوامر في شريط الحالة يمكن توضيحها كالاتي:**

- **Ortho**

ان هذه الكلمة تعني " متعامد " يمكننا هذا الامر عند تفعيله من رسم خطوط عمودية او افقية بشكل سريع وسهل مع تقييد باقي الزوايا, يمكن تفعيله بالضغط عليه او الضغط على F8 من لوحة المفاتيح.

- **Polar**

يستخدم هذا الامر لرسم الخطوط بالاعتماد على الزوايا التي تظهر اثناء عملية رسم الخط ويمكن تثبيت الزيادة في الزوايا بحسب رغبة المستخدم, ويمكن تفعيله اما بالضغط عليه او الضغط على مفتاح F10 من لوحة المفاتيح.

- **Object Snap**

يعتبر من اهم الاوامر في البرنامج حيث يسهل عملية الرسم عند تفعيله بشكل ملحوظ. الشكل 6 يوضح نافذة الامر و من المفضل تفعيل جميع الخيارات داخل هذه النافذة.

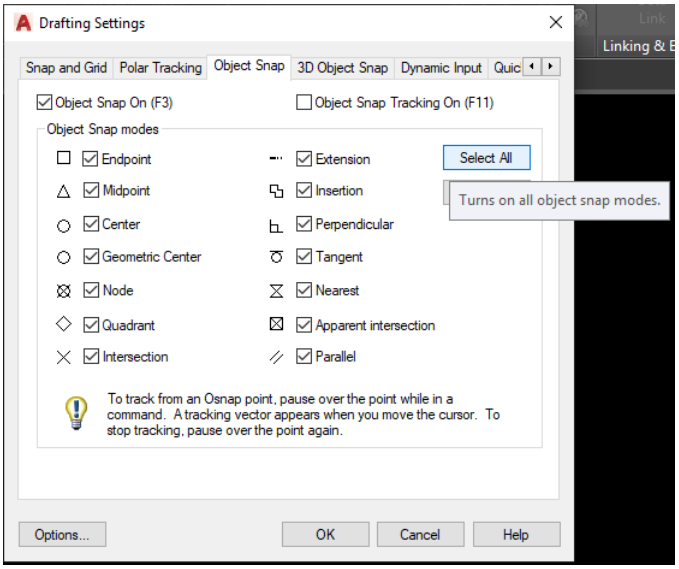


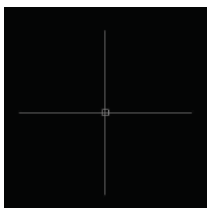
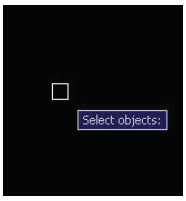

Figure 6

- **Model**

Model Space used to construct blueprints while layout space used to print mode.

- **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.

| | | |
|---|---|---|
|  |  |   |
| Without command | Drawing command | Modified command |

- **Model**

تستخدم صفحة model لرسم المخططات بينما تستخدم صفحة layout للطباعة.

- **المؤشر**

للمؤشر في برنامج AutoCAD عدة حالات تعتمد على الاوامر المستخدمة مثلا اذا كان البرنامج بدون امر فان المؤشر يكون على شكل + ويتوسطه مربع, اما اذا كان امر الرسم مفعل فان المؤشر يكون على شكل + بدون وجود المربع في الوسط, اما اذا كان الامر من اوامر التعديل فان المؤشر يكون على شكل مربع فقط.

*حالات المؤشر مهمة جدا للتعامل مع البرنامج حيث لايمكن ادخال امر ما ولم ننتهي من الامر السابق اذا كان الامر رسم او امر تعديل يجب الانتباه الى حالة المؤشر عند الرسم.

*لانهاء امر ما في البرنامج نضغط على ESC من لوحة المفاتيح.

- **select / deselect**

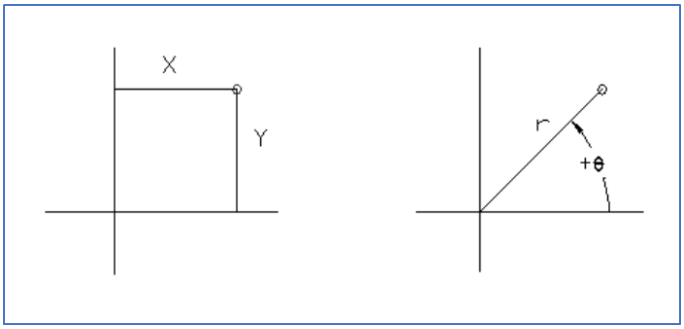
You can select the objects by clicking on an object or drawing a window around it. Drawing a window from left to right (window selection) selects everything that the window contains. Drawing a window from right to left (cross selection) selects everything that the window crosses. You can deselect objects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

- **الاختيار**

يمكن اختيار اي عنصر في الرسم بواسطة الضغط عليه او فتح نافذة بواسطة الماوس حوله, فتح نافذة من اليسار الى اليمن فان البرنامج يختار كل شيء داخل هذه النافذة فقط اما في حال فتح النافذة من اليمين الى اليسار فان البرنامج سوف يختار كل عنصر داخل هذه النافذة بالاضافة الى العناصر المتقاطعة معها.

- **Cartesian and Polar Coordinate Systems**

In a two-dimensional space, a point can be represented using different coordinate systems. The point can be located, using a Cartesian coordinate system, as X and Y units away from the origin. The same point can also be located using the polar coordinate system, as r and θ units away from the origin.




For planar geometry, the polar coordinate system is very useful for certain applications. In the polar coordinate system, points are defined in terms of a radial distance, r, from the origin and an angle θ between the direction of r and the positive X-axis. The default system for measuring angles in AutoCAD 2010 defines positive angular values as counter-clockwise from the positive X-axis.

- **Absolute and Relative Coordinates**

AutoCAD 2010 also allows us to use absolute and relative coordinates to quickly construct objects. Absolute coordinate values are measured from the current coordinate system's

- **نظام الاحداثيات في برنامج AutoCAD**

يمكن تمثيل النقاط في الرسم ثنائي الابعاد في برنامج AutoCAD بنظام الاحداثيات, حيث يمكن تمثيل النقطة اما بوسطة نظام الاحداثيات الديكارتي X , Y محسوب من نقطة الاصل, او يمكن تمثيلها بواسطة نظام الاحداثيات القطبي r , θ محسوب من نقطة الاصل.

يعتبر نظام الاحداثيات القطبي الاكثر فائدة وذلك لسهولة استخدامه في التطبيقات حيث يمكن تعريف النقطة بواسطة الطول القطبي r وزاوية الدوران من الافق (محور x ).

- **الاحداثيات المطلقة والاحداثيات النسبية**

يتعامل برنامج AutoCAD مع الاحداثيات المطلقة والاحداثيات النسبية لتسهيل وتسريع العمل. الاحداثيات المطلقة: اي نقطة يتم رسمها بواسطة هذه الاحداثيات يتم الرجوع الى نقطة الاصل 0 , 0 والحساب من عند هذه النقطة الى النقطة المراد رسمها بالنسبة للمحاور X , Y اما الرسم بواسطة الاحداثيات النسبية لا يشترط فيها الرجوع الى نقطة الاصل حيث تعتبر النقطة الاخيرة هي نقطة الاصل, يمكن التحويل بين الاحداثيات المطلقة الى الاحداثيات النسبية بواسطة كتابة الرمز @ قبل الاحداثي المراد رسمه.

origin point. Relative coordinate values are specified in relation to previous coordinates.

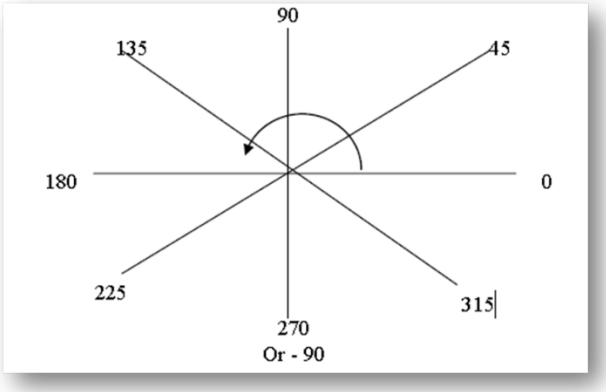
In AutoCAD 2010, the absolute coordinates and the relative coordinates can be used in conjunction with the Cartesian and polar coordinate systems. By default, AutoCAD expects us to enter values in absolute Cartesian coordinates, distances measured from the current coordinate system's origin point. We can switch to using the relative coordinates by using the @ symbol. The @ symbol is used as the relative coordinates specifier, which means that we can specify the position of a point in relation to the previous point.

- **Limits**

This command used to make border to the work sheet.
access the command by:
Menu bar⇨ format ⇨Drawing limits.
Specify lower left corner or [on / off] <0.0000,0.0000>:
Specify upper right corner <420,297>:

- **Angles in AutoCAD Programs**

Direction of angle in AutoCAD by default is counter clockwise like in figure below:




- **Method of Entering Coordinate-Defining Positions**

In AutoCAD, there are methods for specifying the locations of points when we create planar geometric entities.

- **Interactive method**: Use the cursor to select on the screen.
- **Absolute coordinates** (Format: X,Y): Type the X and Y coordinates to locate the point on the current coordinate system relative to the origin.
- **Relative rectangular coordinates** (Format: @X,Y): Type the X and Y coordinates relative to the last point.
- **Relative polar coordinates** (Format: @Distance<angle): Type a distance and angle relative to the last point.

1. **(Absolute Cartesian Coordinate system). Example: 5,7**
2. **(Relative Cartesian Coordinate system). Example: @10,3**
3. **(Absolute Polar Coordinate system). Example :8<60**
4. **(Relative Polar Coordinate system). Example: @7<30**

***Example**:*

1. **Draw Line using Absolute Cartesian Coordinate System**

Command line: Line ↵
*The following message in command line will appear
Specify first Point:3,3 ↵ Specify next Point: 10,7 ↵
*The length of line calculated from original point (0,0)

2. **Draw Line using Relative Cartesian Coordinate System**

Command line: Line ↵
Specify first Point:7,4 ↵ Specify next Point: @3,2 ↵
*The length of line calculated from point (7,4)

3. **Draw Line using Absolute Polar Coordinate System**

In this system the line can be drawn by the point of coordinate and angle.
Command line: Line ↵
Specify first Point:7,5 ↵
Specify next Point: 7.6 <66.80
The length of line calculated from original point (0,0) using Pythagorean triangle as following:

$L = \sqrt{x^2 + y^2}$

$L = \sqrt{3^2 + 7^2}$

L = 7.6

The angle is calculated as following:

$\tan^{-1}\frac{3}{7} = 66.8$

4. **Draw Line using Relative Polar Coordinate System**

Command line: Line ↵
Specify first Point:10,10 ↵
Specify next Point: @4.47<26.56 ↵
The length of line calculated from point (10,10) using Pythagorean triangle as following:

$L = \sqrt{x^2 + y^2}$

$L = \sqrt{4^2 + 2^2}$

L = 4.47

The angle is calculated as following:

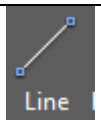
$\tan^{-1}\frac{2}{4} = 26.56$

**Note**: the symbol ↵ refers to **enter key** on the key board.

- **Draw Commands in AutoCAD**

1. **Line**

line in **AutoCAD** can be drawn as follow:

| | |
|---|---|
| Command Line | Line or L ↵ |
| Menu Bar | Draw → Line |
| Draw Bar |  |

When select line command, programme requires specify first point, display in command line:

1. Specify first Point:
2. Specify next point or [Undo]:
3. Specify next point or[Close/Undo]:

**Note:**

- **close, C**↵ : this option close series of lines, connect first point with last point by line.
- **Undo, U**↵ : enter U for undo. AutoCAD back up one segment, undoing it so that you can recreate it.

2. **Rectangle**

| | |
|---|---|
| Command Line | Rectangle or Rec ↵ |
| Menu Bar | Draw → Rectangle |
| Draw Bar |  |

When enter rectangle command, the program require specify first corner:
Specify first corner point or [ Chamfer/ Elevation / Fillet / Thickness / Width]:
Specify other corner Point:

1. **Chamfer**, **C**↵ : the chamfer command enables to place a chamfer at the rectangle corner

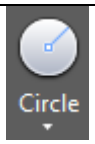
Specify first chamfer distance for rectangles<0.000>: in x direction
Specify second chamfer distance for rectangles<0.000>: in y direction

2. **Fillet**, **F**↵: the fillet command creates both fillets and rounds on any combination of two lines.

Specify fillet radius for rectangles<0.000>:

### 3. Circle

There are many methods to draw circle in AutoCAD as follow:

| | |
|---|---|
| Command Line | Circle or C ↵ |
| Menu Bar | Draw → Circle |
| Draw Bar |   |

When select circle command the programme display the following message in the command line:

1. Specify center point for circle or [3P/2P/Ttr(tan tan radius)]:
2. Specify radius of circle or [ Diameter]: D
3. Specify diameter of circle<current default>:

Note:

- **3P(3 point):**the circle passes through all three points specified.
- **2P(2 point)** .the two points specify the location and diameter.
- **Ttr(tan tan radius)** .specify two objects for the circle to be tangent to, then specify the radius.
- **TTT** (Draw Circle (Tan Tan Tan).
- **Arc**

Arc in AutoCAD can be drawn by several methods:

| | |
|---|---|
| Command Line | Arc or a ↵ |
| Menu Bar | Draw → Arc |
| Draw Bar |   |

The following message will appear in command line:
Specify start point of arc or [CEnter]:
Specify second point of arc or [CEnter/ End]:
Specify endpoint of arc
*Specify endpoint of arc (hold Ctrl to switch direction) or [**Angle**/**chord Length**]:
**Note**:
**Angle**: draw arc using angle
**Chord length**: draw arc using chord length

## 4. Ellipse

| | |
|---|---|
| Command Line | Ellipse or el ↵ |
| Menu Bar | Draw → Ellipse |
| Draw Bar |  |

There are three methods to draw ellipse in AutoCAD:

1. **Axis end (Specify one axis and the end of second)**

Command : ellipse or el ↵
Specify axis end point of ellipse or [Arc / Centre]:
Specify other end point of axis:
Specify distance to other axis or [Rotation]:

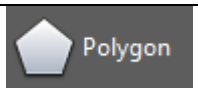
2. **Centre (Specify the centre and the end of each axis)**

Specify axis end point of ellipse or [Arc / Center]: C
Specify center of ellipse:
Specify endpoint of axis:
Specify distance to other axis or [rotation]:

3. **Arc (create an ellipse arc)**

Specify axis endpoint of ellipse or [Arc / Center]: A
Specify axis endpoint of elliptical arc or [center]:
Specify other endpoint of axis:
Specify distance to other axis or [rotation]:
Specify start angle or [parameter]:
Specify end angle or [parameter / included angle**]:**

## 5. Polygon

| | |
|---|---|
| Command Line | Polygon or Pol↵ |
| Menu Bar | Draw → Polygon |
| Draw Bar |  |

When Enter polygon command, the following message will appear:
Enter number of sides< >:
Specify center point or [edge]:

If the Edge option chosen, the program require the following:
Specify first endpoint of edge:
Specify second endpoint of edge:

If Center point option chosen, the program requires the radius of circle:
Enter an option [Inscribed in circle/Circumscribed about circle]< >:

Note:
**I** ↵ Inscribed option : polygon will be drawn outside the circle
**C** ↵ Circumscribed option : polygon will be drawn inside the circle

When determine one of the options the program display the message:
Specify radius of circle:

**Remark:**
the circle is not display in the drawing.

**Donut**

The donut command allows to create thick – walled or solid circles, known in AutoCAD as donuts.

| | |
|---|---|
| Command Line | Donut or do ↵ |
| Menu Bar | Draw → Donut |
| Draw Bar |  |

When select donut command the following message will appear
Specify inside diameter of donut:
Specify outside diameter of donut:
Specify center of donut or <exit>:

**Construction Line**

Construction lines, line can extend to infinite such as xline can be used to create construction and reference line and for trimming

| | |
|---|---|
| Command Line | xl ↵ |
| Menu Bar | Draw → Construction lines |
| Draw Bar |  |

When select this command the following message will appear in the command line:
Specify a point or [Hor/ Ver /Ang / Bisect / Offset]
The Xline command provides several options that allow creating specific types of construction lines.

**Options:**

**Hor:** the horizontal option forces all the lines to be exactly horizontal.

**Ver**: the vertical option forces all the lines to be exactly vertical.

**Ang**: the angle option use to create construction lines at angles other than 0 or 90 degrees.

**Bisect:** this option draws the xline at an angle between two selected points.

**Offset**: offsetting a line means creating the line at a specific distance from another line.

## Hatch

Fill an enclosed area or selected object by a hatch pattern or fill. Access the command by:

| | |
|---|---|
| Command Line | hatch or h ↵ |
| Menu Bar | Draw ⟶ hatch |
| Draw Bar |  |

The following message will appear:

Select objects or [picK internal point/Undo/seTtings]

### ❖ Modify commands in AutoCAD

## Erase

Removes objects from the drawing.

| | |
|---|---|
| Command Line | Erase or e ↵ |
| Menu Bar | Modify ⟶ Erase |
| Draw Bar |  |

When use the command, the program require about objects needs to delete by display message:

**Select Objects**:

The cursor change to small square selected objects by three ways

1. Pick the objects. this default option is used for selecting one object at a time.
2. crossing window. all objects within and crossing through a crossing window are selected. (drag the cursor from right to the left).
3. window. only objects completely within the window are selected. not objects that cross through or outside the window. (drag the cursor from left to the right).

## Move

This command uses to move objects to new locations.

| | |
|---|---|
| Command Line | Move or m ↵ |
| Menu Bar | Modify ⟶ Move |
| Draw Bar |  |

When use move command, the program require the following:

Select Objects:

Specify base point or displacement:

Specify second point of displacement:

**Copy**

Copies object a specified distance in a specified direction.

| | |
|---|---|
| Command Line | Copy or co or cp ↵ |
| Menu Bar | Modify ⟶ copy |
| Draw Bar |  |

When use copy command the program require the following:
Select Objects:
Specify base point or displacement or [ Multiple]:
Specify second point or displacement:

**Offset**

This command used to create concentric circle, parallel lines and parallel curves by determine distance.

| | |
|---|---|
| Command Line | offset or o ↵ |
| Menu Bar | Modify ⟶ offset |
| Draw Bar |  |

When use offset command the program require parallel distance.
Specify offset distance or [through]:
Select object to offset or <exit>:
Specify point on side to offset:

**Mirror**

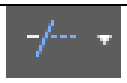
This command used to create mirrored copy of selected object.

| | |
|---|---|
| Command Line | mirror or mi ↵ |
| Menu Bar | Modify ⟶ mirror |
| Draw Bar |  |

The program requires the following:
Select objects:
Specify first point of mirror line:
Specify second point of mirror line:
Delete source object? [yes/no]:

## Trim

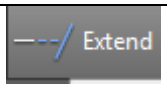
This command used to trim objects to meet the edge of another object.

| | |
|---|---|
| Command Line | trim or tr ↵ |
| Menu Bar | Modify ⟶ trim |
| Draw Bar |  |

When enter trim command, the following message will appear:
Select objects or <select all>:
Select cutting edges:
[Fence/Crossing/Project/Edge/eRase/Undo]:

## Extend

Extends objects to meet the edge of other objects.

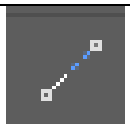
| | |
|---|---|
| Command Line | extend or ex ↵ |
| Menu Bar | Modify ⟶ Extend |
| Draw Bar |  |

- To extend objects, first select the boundaries, then press enter and select the objects that you want to extend.

When select extend command the programme will appear the following message:
Select objects or <select all>:
[Fence/Crossing/Project/Edge/Undo]:

## Lengthen

Change the length of the object.

| | |
|---|---|
| Command Line | lengthen or len ↵ |
| Menu Bar | Modify ⟶ Lengthen |
| Draw Bar |  |

The following message appear:
Select an object to measure or [DElta/Percent/Total/DYnamic] <Total>:

**Options:**

**DElta:** in this option determine increment or decrement in length, then click the object, the object is lengthening in determined value.

**Percent:** in this option determine length percentage to original length. For example: 110 mean increment 10%, 90 mean decrements 10 %.

**Total:** determine all length value to object.
**DYnamic:** in this option using mouse to increment or decrement object to new location.

## Stretch

This command stretches selected object from selected side.

| | |
|---|---|
| Command Line | stretch or s ↵ |
| Menu Bar | Modify ⟶ Stretch |
| Draw Bar |  |

The following message appear:
Select objects to stretch by crossing window or crossing polygon:
Specify base point or displacement:
Specify second point of displacement:

## Rotate

This command uses to rotate sleeted objects around point by determining angle.

| | |
|---|---|
| Command Line | rotate or ro ↵ |
| Menu Bar | Modify ⟶ rotate |
| Draw Bar |  |

When use the command, the program will appear the following message:
Select objects:
Specify base point:
Specify rotation angle or [Reference]:

## Scale

Enlarges or reduce selected objects, keeping the proportions of the object the same after scaling.

| | |
|---|---|
| Command Line | scale or sc ↵ |
| Menu Bar | Modify ⟶ scale |
| Draw Bar |  |

When select this command the following message will appear:
Select objects:
Specify base point:
Specify scale factor or [Reference]:

**Break**

Breaks the selected object between two points.

| | |
|---|---|
| Command Line | Break or br ↵ |
| Menu Bar | Modify → Break |
| Draw Bar |  |

When select this object the programme appears the following message:
Select Object:
Specify second break point or [First Point]: f
Specify First break point:
Specify second break point:

**Explode**

Breaks a compound object into its component objects.

| | |
|---|---|
| Command Line | Explode or expl↵ |
| Menu Bar | Modify → Explode |
| Draw Bar |  |

The following message will appear:
Select objects:

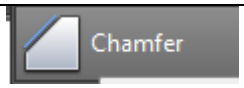
**Fillet**

Rounds and fillets the edges of objects.

| | |
|---|---|
| Command Line | Fillet or f ↵ |
| Menu Bar | Modify → Fillet |
| Draw Bar |  |

**Chamfer**

Bevels the edges of objects.
This command used to change two objects meeting to meet by line.

| | |
|---|---|
| Command Line | Chamfer or cha ↵ |
| Menu Bar | Modify → Chamfer |
| Draw Bar |  |

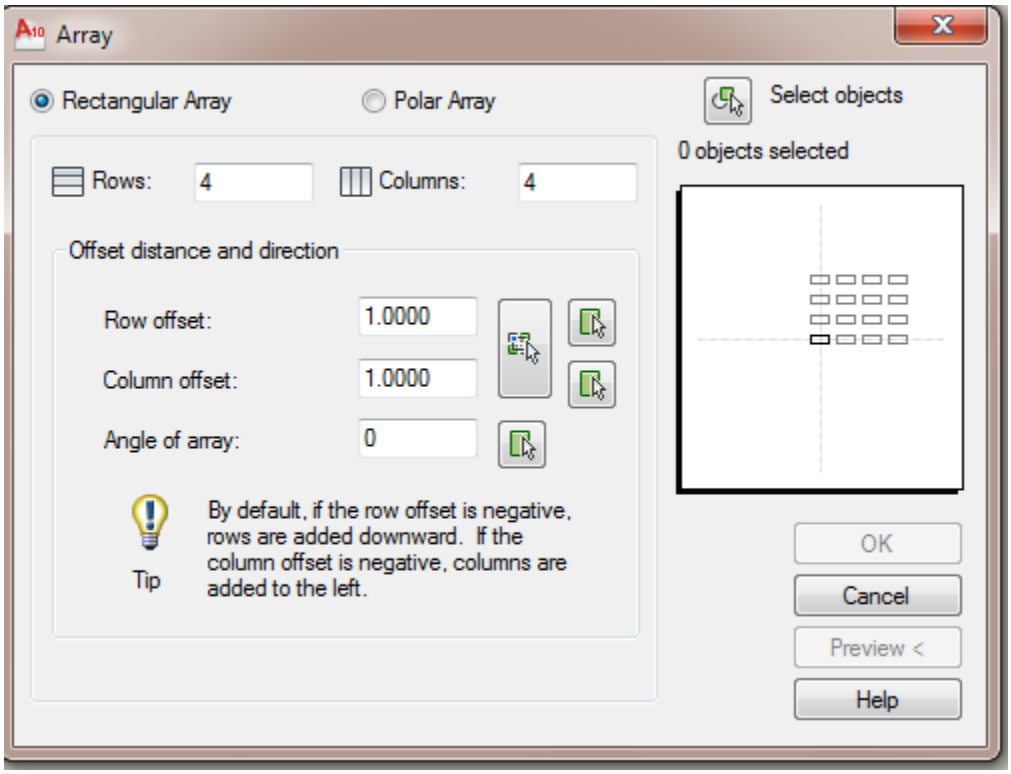
## Array

Distributes object copies into any combination of rows, columns, and levels.

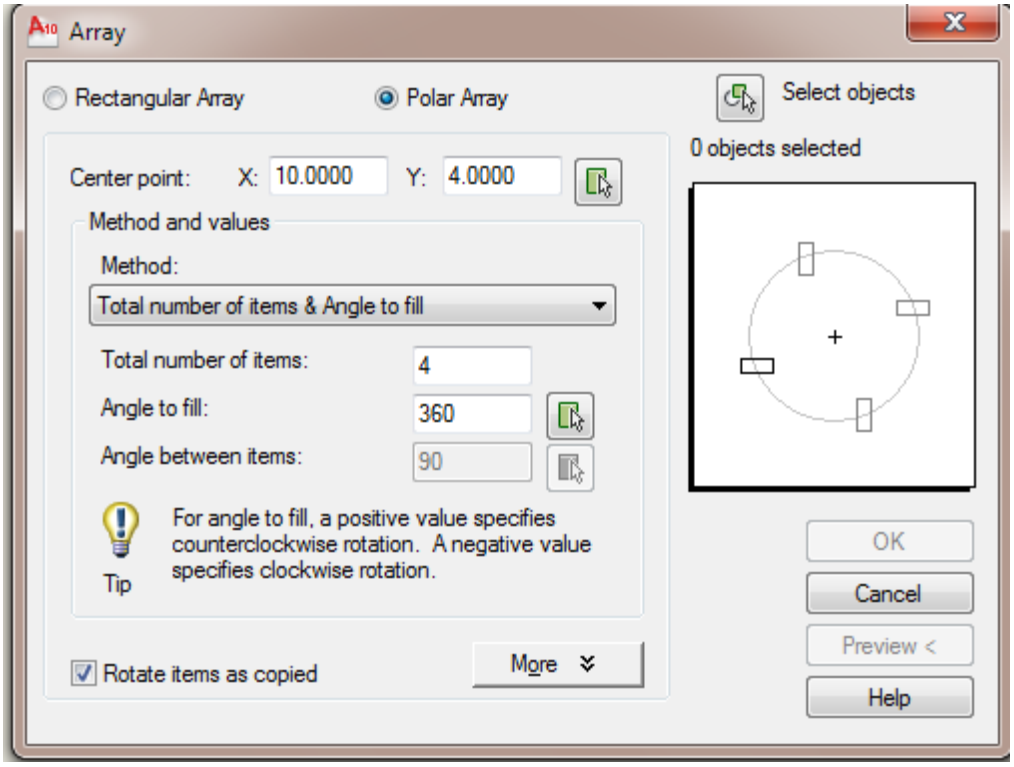
This command used for creating multiple copies of object in the pattern, can create rectangular array or polar array.

| | |
|---|---|
| Command Line | Array or ar ↵ |
| Menu Bar | Modify ⟶ Array |
| Draw Bar |  |

In choice the rectangular array following message box appear. Determine number of rows and number of columns, the distance between rows, the distance between columns, and array direction.



In choice the Polar array following message box appear. determine total number of items, angle to fill, angle between items.



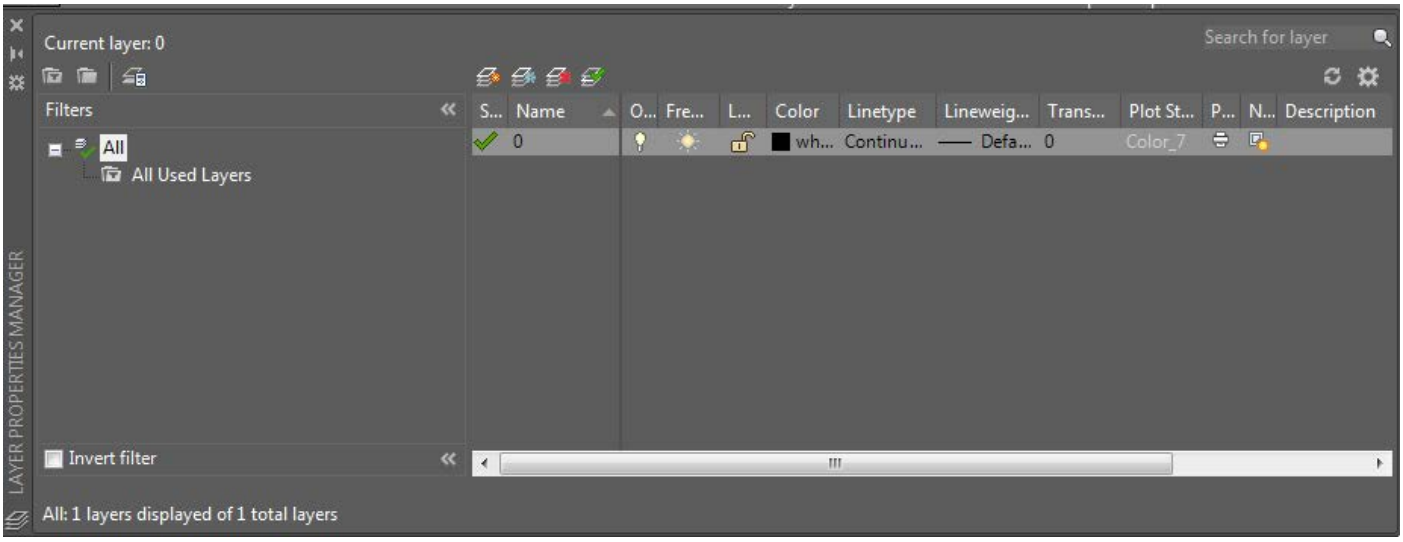

## Layers

The way to gain complete layer control is through the layer Properties Manager.

| | |
|---|---|
| Command Line | Layers or la |
| Tool Bar | Format → Layers |
| Draw Bar |  |

**Steps of Create Layers:**

- Click  on the left of properties bar. The following dialog box appear.




- Click **(New), new** raw adding to existing layers. we saw always there was first layer it is zero layer which created by AutoCAD Automatically when open new drawing. Appeared in new raw name of new layer we can change it to objective name to easily access to layer we need. such as wall, column, steel, ground.
- To determine layer color click on black square under address (**color)** in new raw color window appear select from it appropriate color.
- To determine line style click on **(continuous**) in new raw under address **(line type**), window appear, select from it appropriate line type. click on **(load**) to appear more line types.
- To determine line weight used for drawing objects click **(default**) under address (**line weight**) then select weight we need from the window that appear.
- Repeat the operation for other layers then click (**ok)**.

**Layers Managements**

**Select drawing layer to be activate**.

1. Click on manager Layer List.
2. Select layer from it.
3. Click on drawing board. the selected layer be the active layer .note that object color and line type and line weight all must be assistant to**( By layer**).

**Move drawing object from layer to another.**

1. Select drawing object.
2. Click on manager Layer List to open it.
3. Select layer want to move the object to it.
4. Click on ESC twice to undo select object, we saw change the layer color and line to color of selected layer.

**Hide objects belong to determine drawing layer.**

1. Click on manager Layer List to open it.
2. Click on yellow light for layer wants to hid it by put it to off then click on enter we obtain the drawing without the selected layer.

**lock objects belong to determine layer.**

1. Click on manager Layer List to open it.
2. Click on lock button for layer wants to lock it then click enter, the layer is locked.

(the locked layer it is the layer that can’t change or modified to it unless we open it by click the lock button again.

**change colour of drawing layer.**

1. Click on properties bar, the dialog box appear for control layers and lines types.
2. Click on colour, colour control box appear.
3. Select appropriate colour.
4. Click **ok** in two open boxes to close them we saw all the objects belong to the layer their colour are change to new colour.