

ZTM112 BİLGİSAYAR DESTEKTEKLİ ÇİZİM TEKNİĞİ

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Teknik çizim nedir ?

Bir çizim nesnelerin grafik gösterimidir ve serbest el, mekanik veya bilgisayarlar kullanılarak yapılır.

Teknik Çizim büyük çapta üretilecek ve/veya kolayca birbirleriyle yer değişebilecek parçalar için yeterli alanda karmaşık fikirleri göstermek amacı ile kullanılır.

Geleneksel çizim araçları



- Tahta veya mekanik kalemler.
- Alet setleri
(Pergel ve pusula)
- 45- ve 30/60-derece üçgen cetverller.
- Ölçekli cetveller.
- Eğriler.
- İletki.
- Silgiler.
- Çizim Kağıdı.
- Daire şablonlar.
- İzometrik şablonlar.



Home Insert Annotate Parametric View Manage Output Add-ins A360 Express Tools

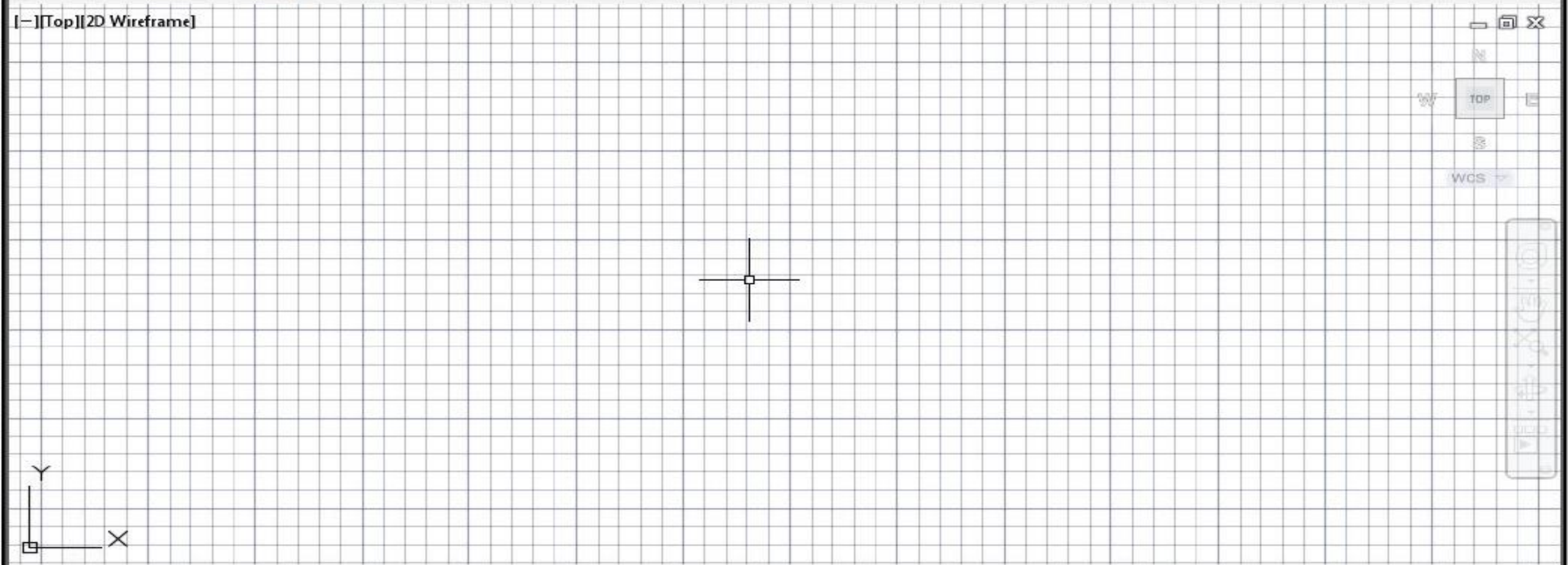
Line Polyline Circle Arc Draw

Modify

Text Dimension Annotation

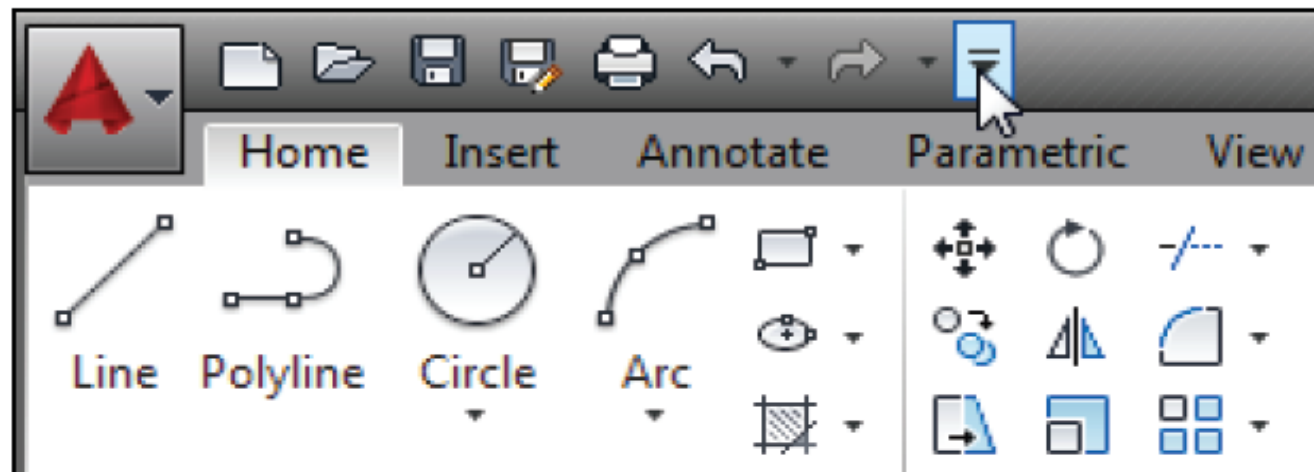
Layers Block Properties Groups Utilities Clipboard View

Start Drawing1*

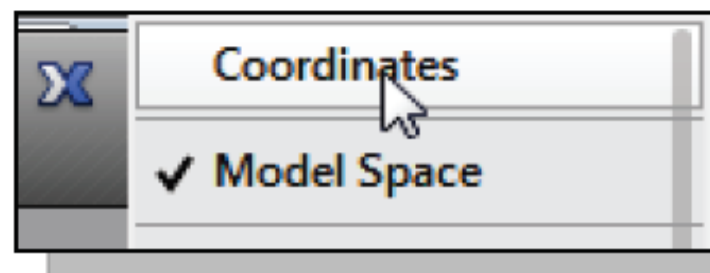
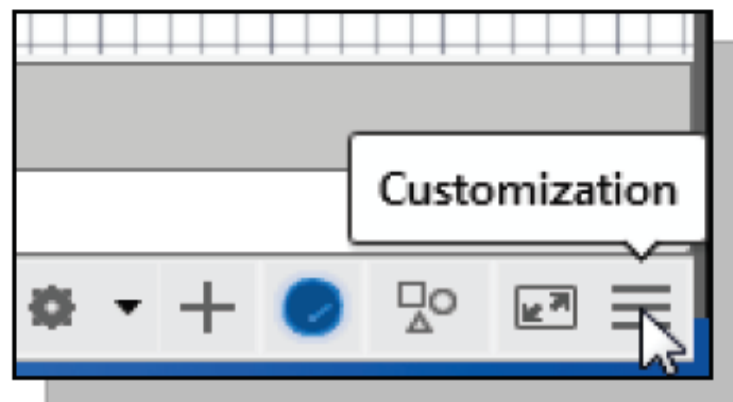


Command: Type a command

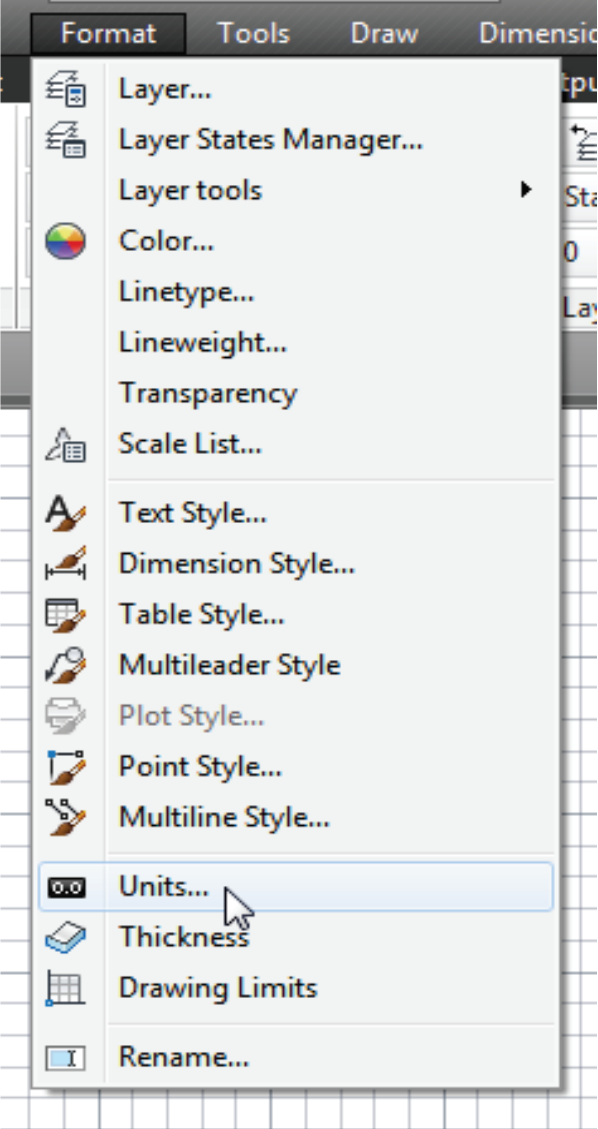
2. If necessary, click on the down-arrow in the *Quick Access* bar and select **Show Menu Bar** to display the **AutoCAD Menu Bar**. The *Menu Bar* provides access to all AutoCAD commands.



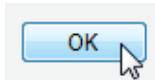
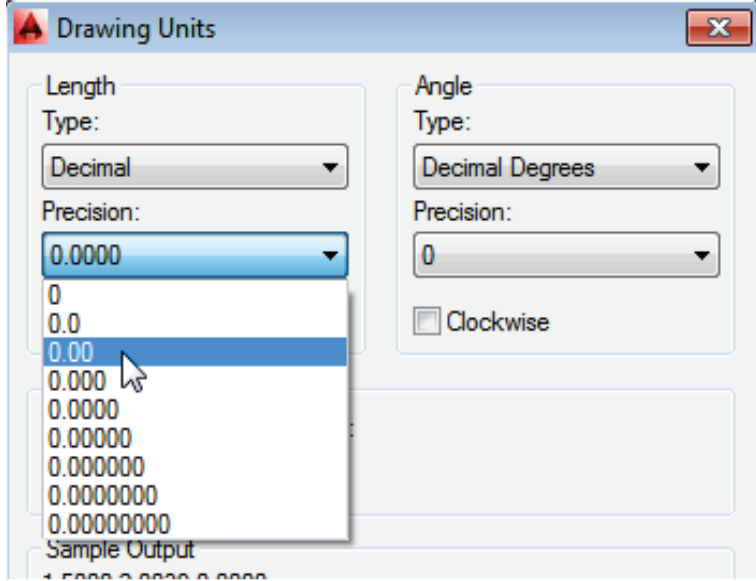
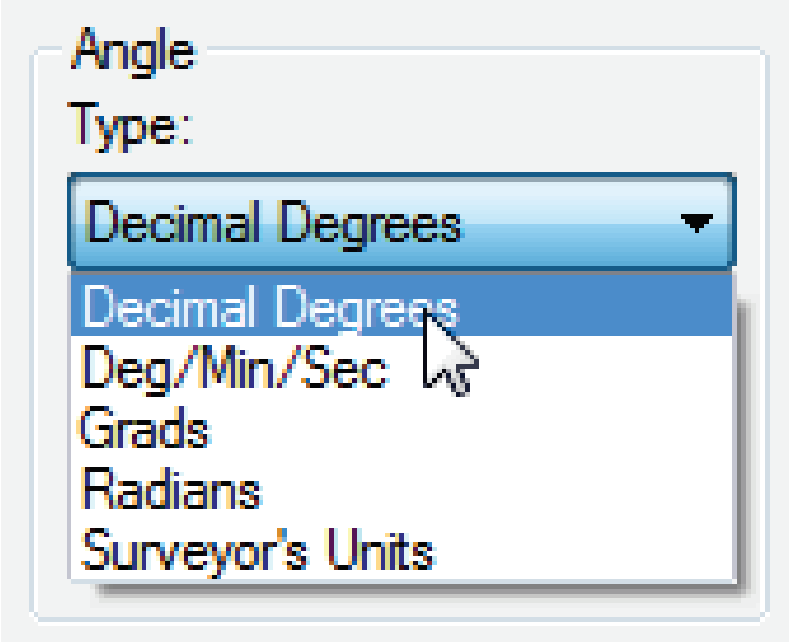
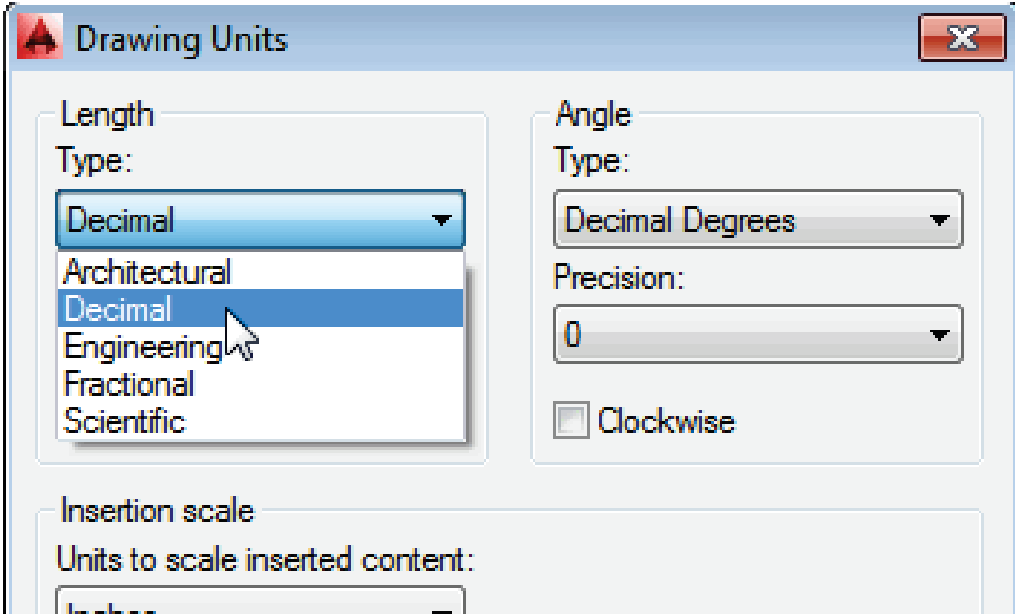
3. To switch on the **AutoCAD Coordinates Display**, use the *Customization option* at the bottom right corner.



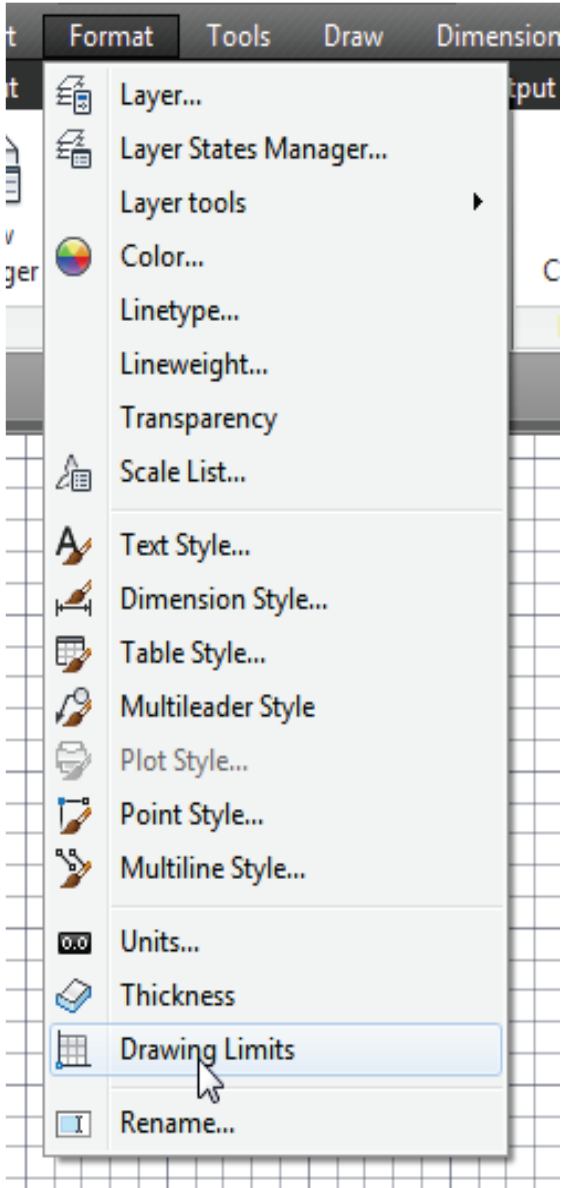
Drawing Units Setup



[Format] → [Units]



Drawing Area Setup

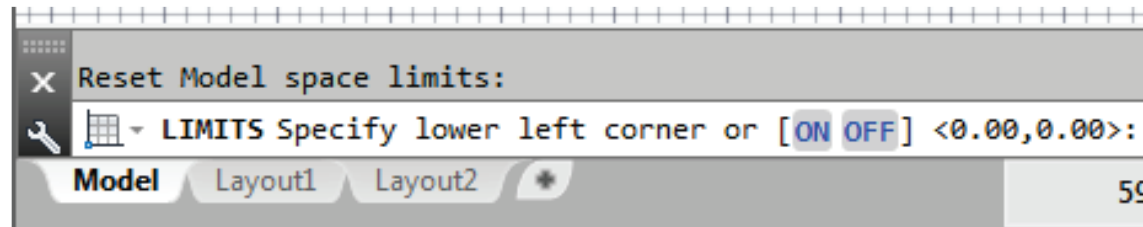


[Format]

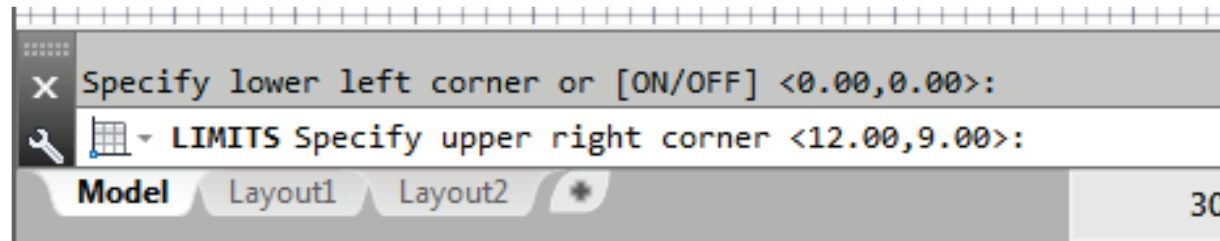


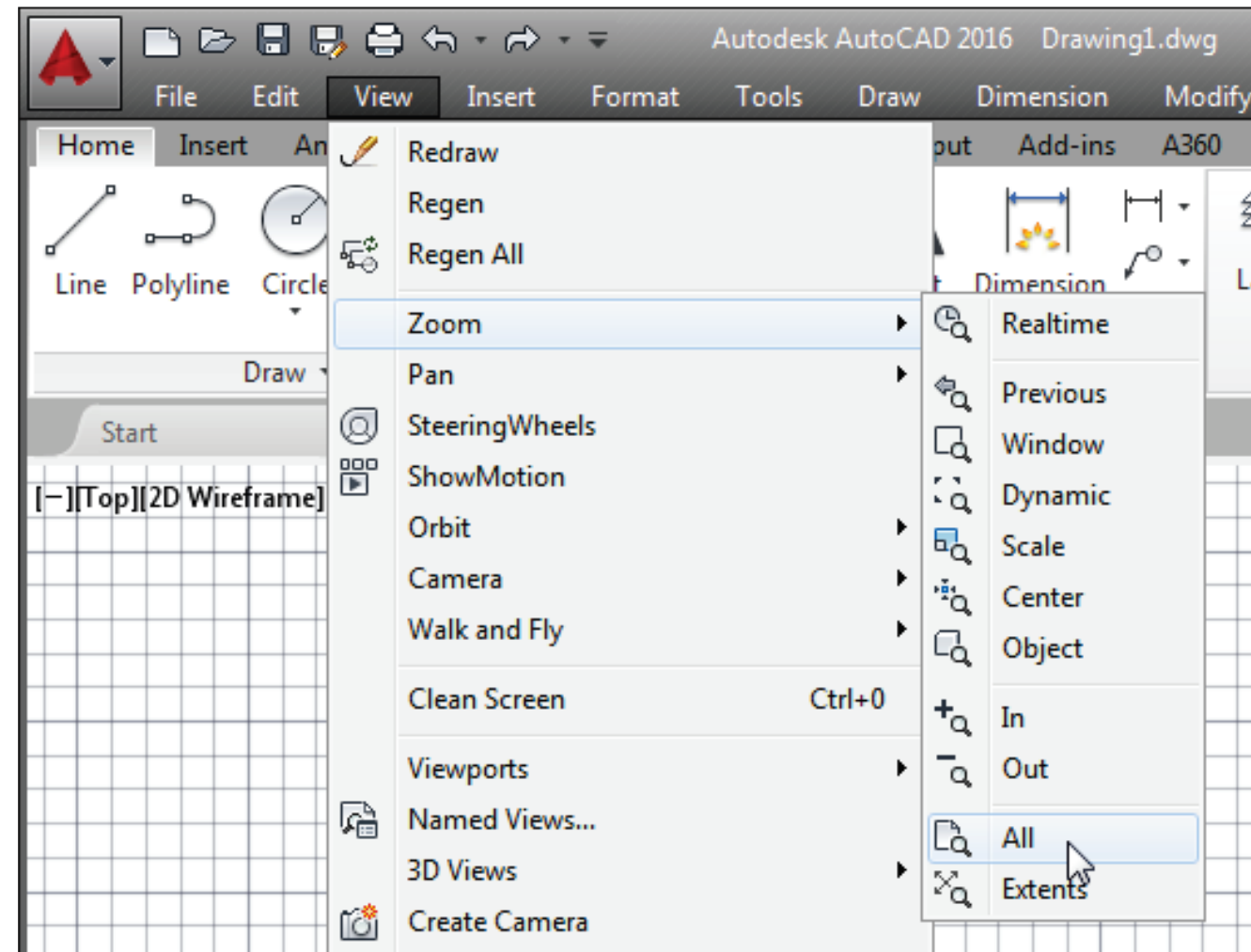
[Drawing Limits]

In the command prompt area, the message “*Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>.*” is displayed. Press the **ENTER** key once to accept the default coordinates **<0.00,0.00>**.



In the command prompt area, the message “*Specify upper right corner <12.00,9.00>.*” is displayed. Press the **ENTER** key again to accept the default coordinates **<12.00,9.00>**.

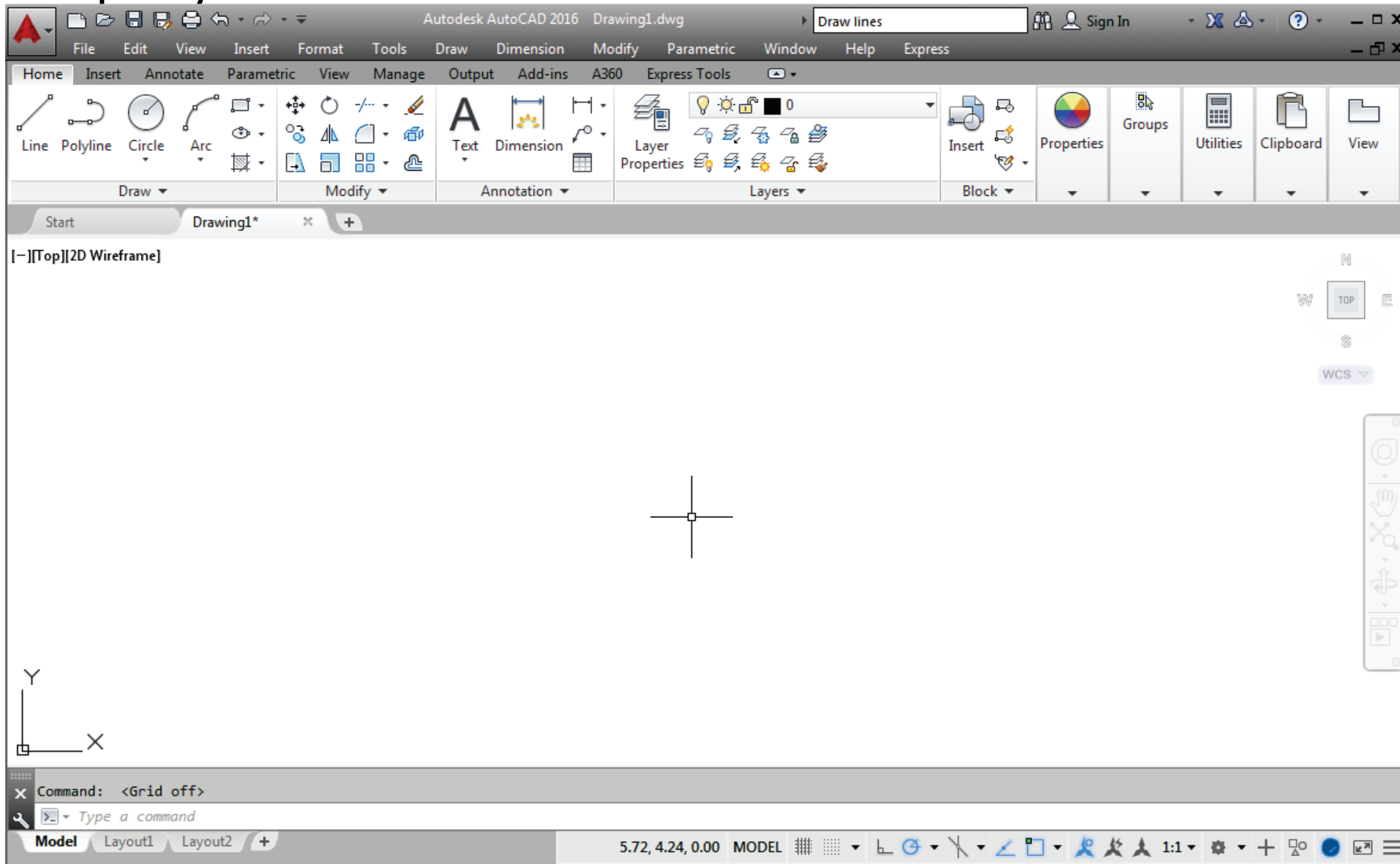




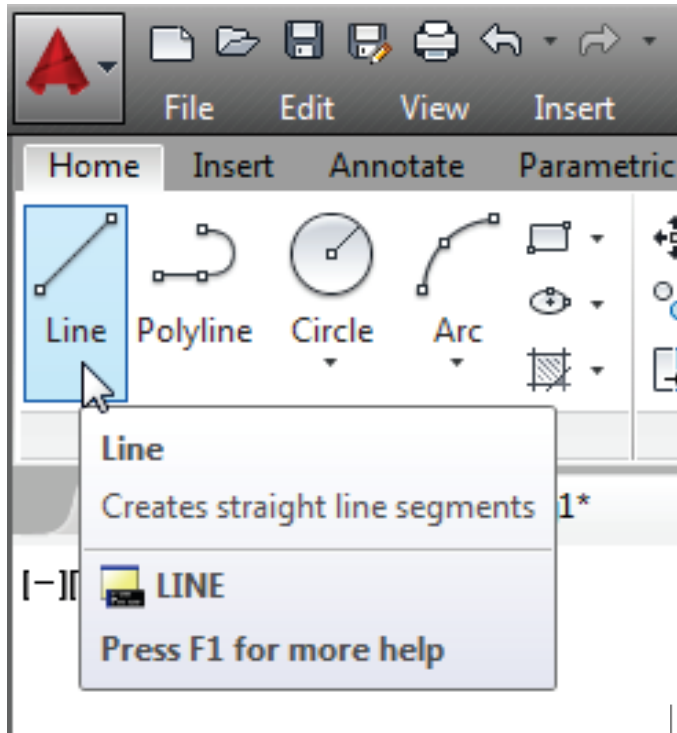
[View] → **[Zoom]** → **[All]**

The **Zoom All** command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

Hit the function key [F7] once to turn **off** the display of the *Grid* lines.



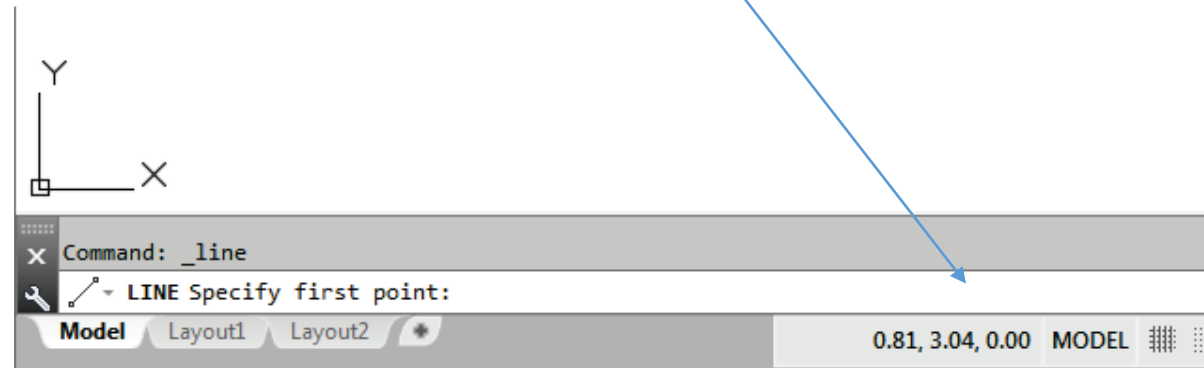
Drawing Lines with the *LINE* Command

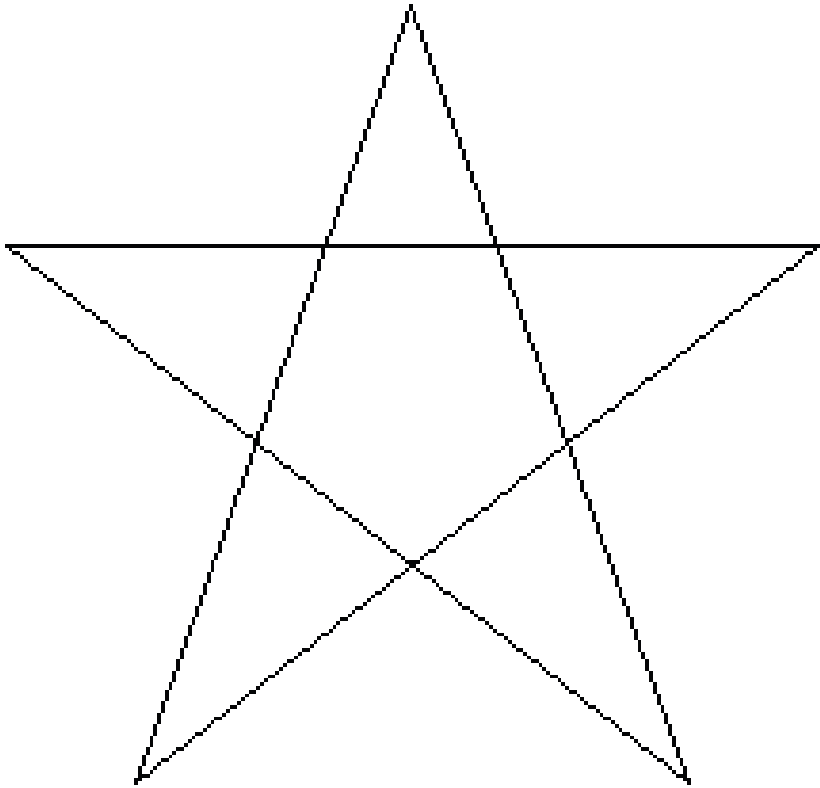


Move the graphics cursor to the first icon in the *Draw* panel. This icon is the **Line** icon. Note that a brief description of the Line command appears next to the cursor.

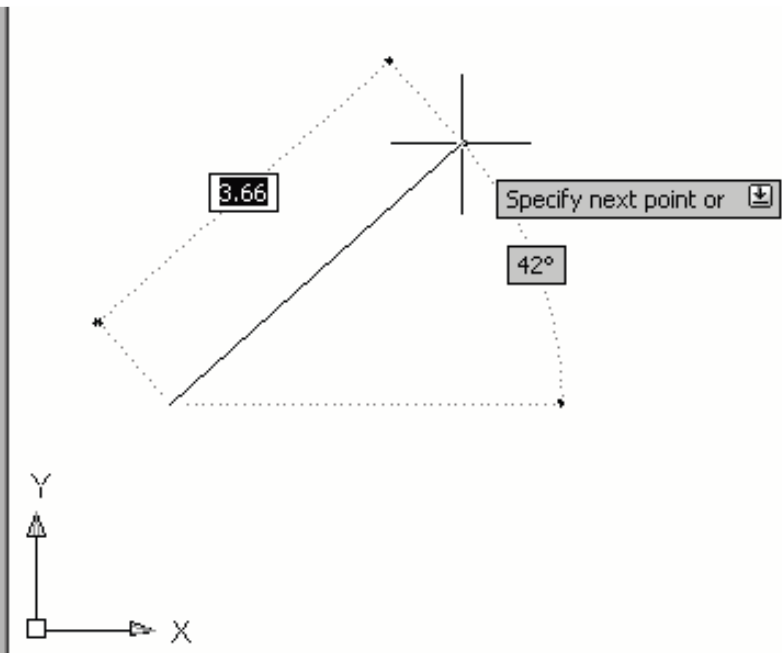
Select the icon by clicking once with the **leftmouse-button**, which will activate the Line command.

Coordinates of the graphics cursor





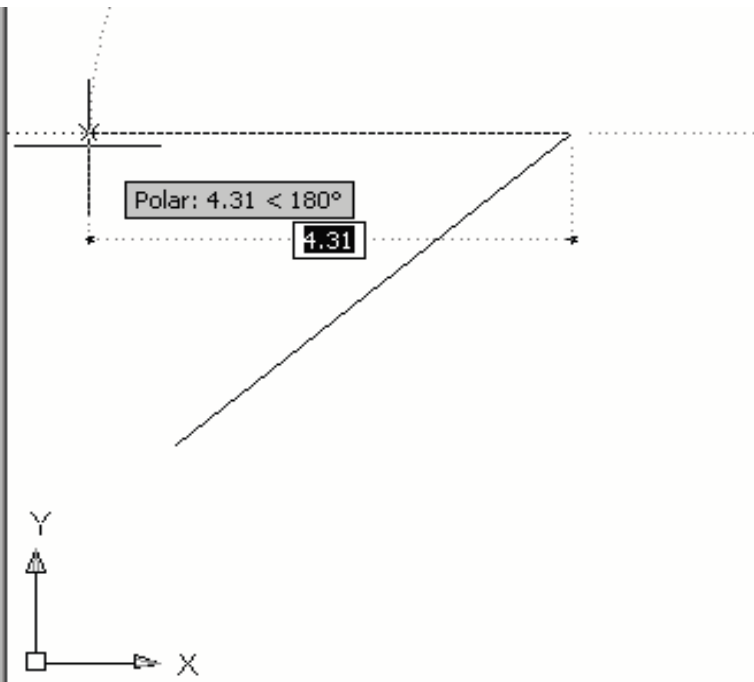
We will create a freehand sketch of a fivepoint star using the Line command. Do not be overly concerned with the actual size or accuracy of your freehand sketch. This exercise is to give you a feel for the **AutoCAD 2016** user interface.



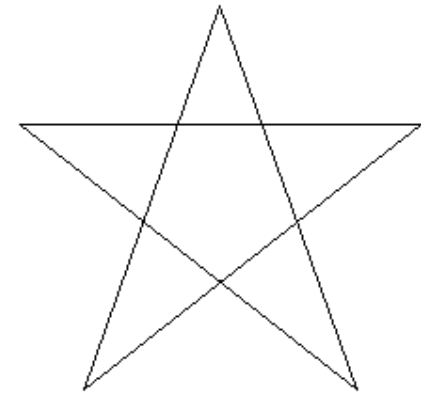
We will start at a location about one-third from the bottom of the graphics window. Left-click once to position the starting point of our first line. This will be *point 1* of our sketch.

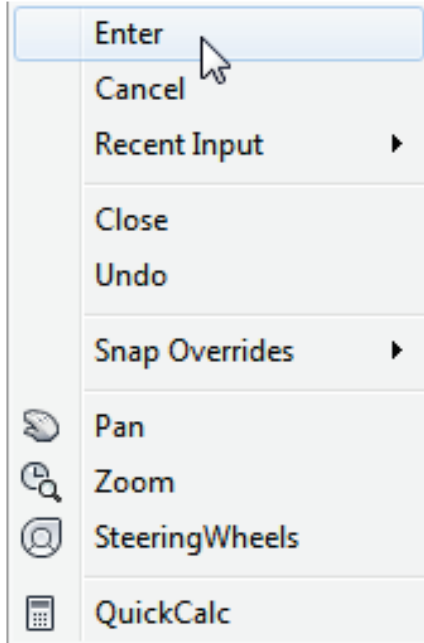
Next, move the cursor upward and toward the right side of *point 1*

1. Notice the rubber-band line that follows the graphics cursor in the graphics window. Left-click again (*point 2*) and we have created the first line of our sketch.



Move the cursor to the left of *point 2* and create a horizontal line about the same length as the first line on the screen.





Notice that the Line command remains activated even after we connected the last segment of the line to the starting point (*point 1*) of our sketch. Inside the graphics window, **click once** with the **right-mouse-button** and a popup menu appears on the screen.

Select **Enter** with the left-mouse-button to end the Line command. (This is equivalent to hitting the [ENTER] key on the keyboard.)

Move the cursor near *point 2* and *point 3*, and estimate the length of the horizontal line by watching the displayed coordinates for each point.

Visual Reference

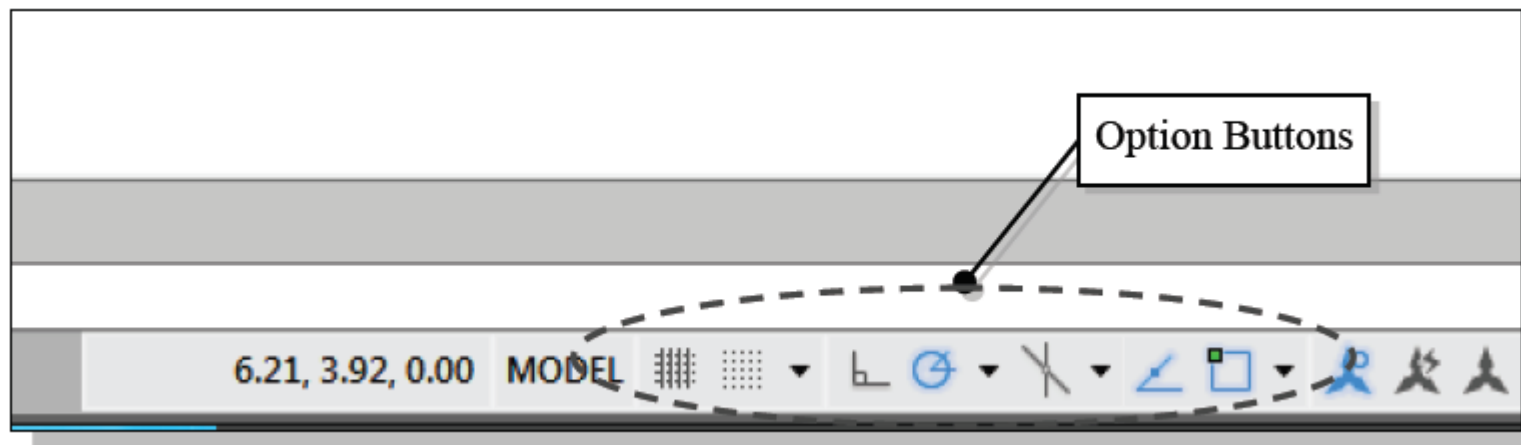
The method we just used to create the freehand sketch is known as the **interactive method**, where we use the cursor to specify locations on the screen.

This method is perhaps the fastest way to specify locations on the screen.

However, it is rather difficult to try to create a line of a specific length by watching the displayed coordinates. It would be helpful to know what one inch or one meter looks like on the screen while we are creating entities.

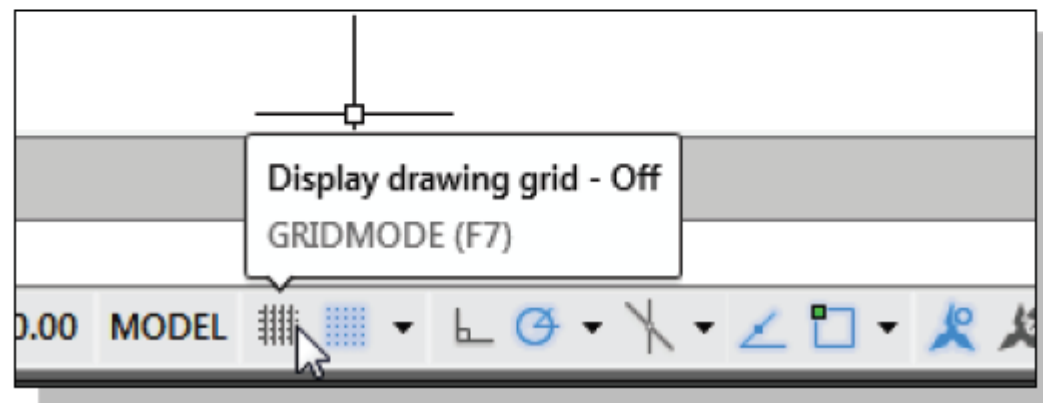
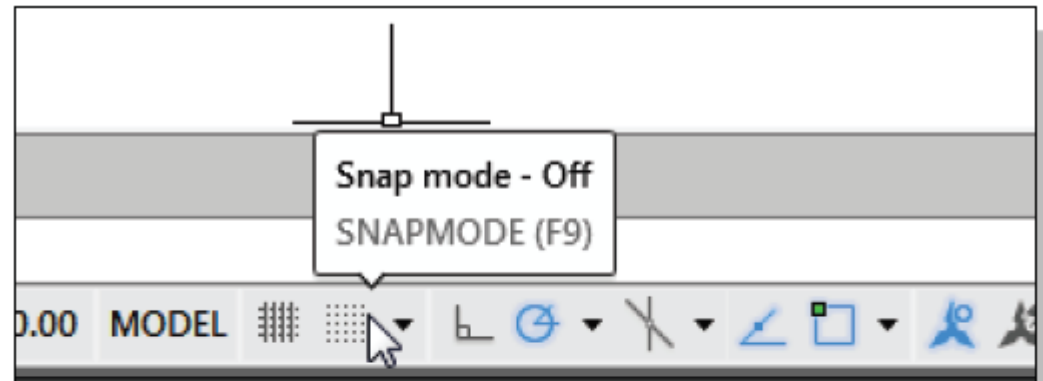
AutoCAD 2016 provides us with many tools to aid the construction of our designs. For example, the **GRID** and **SNAP MODE** options can be used to get a visual reference as to the size of objects and learn to restrict the movement of the cursor to a set increment on the screen.

The **GRID** and **SNAP MODE** options can be turned *ON* or *OFF* through the *Status Bar*. The *Status Bar* area is located at the bottom left of the AutoCAD drawing screen, next to the cursor coordinates.



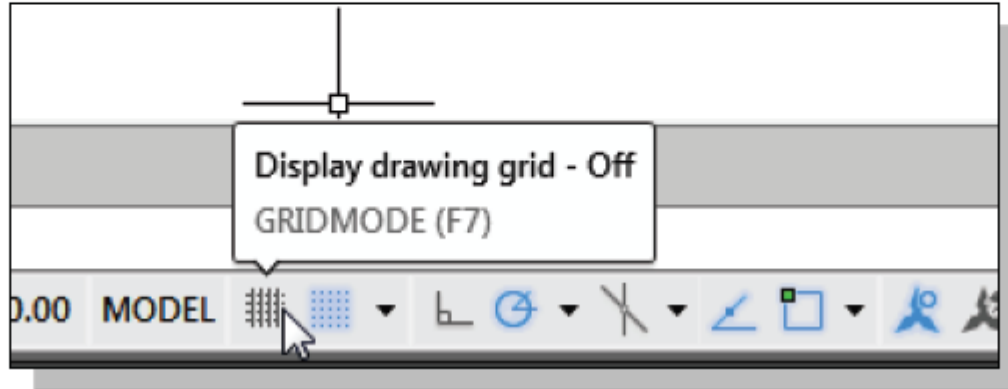
The second button in the *Status Bar* is the *SNAP MODE* option and the third button is the *GRID DISPLAY* option. Note that the buttons in the *Status Bar* area serve two functions:

(1) the status of the specific option, and (2) as toggle switches that can be used to turn these special options *ON* and *OFF*. When the corresponding button is *highlighted*, the specific option is turned *ON*. Using the buttons is a quick and easy way to make changes to these *drawing aid* options. The buttons in the *Status Bar* can also be switched on and off in the middle of another command.

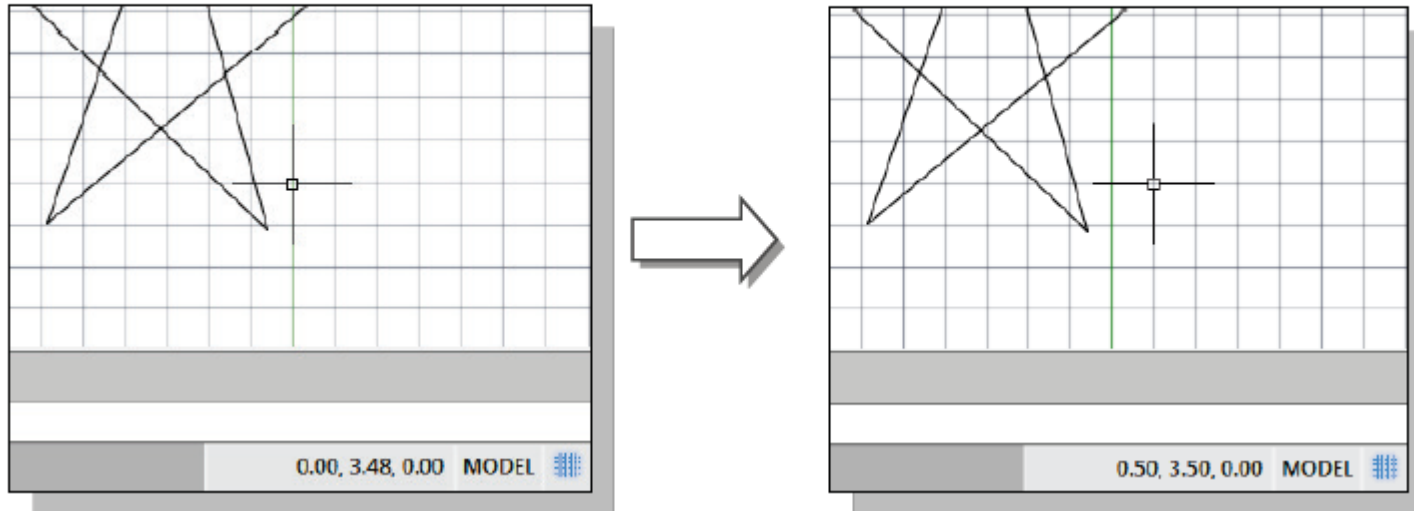


GRID ON

1. Left-click the **GRID** button in the *Status Bar* to turn **ON** the *GRID DISPLAY* option. (Notice in the command prompt area, the message “<Grid on>” is also displayed.)



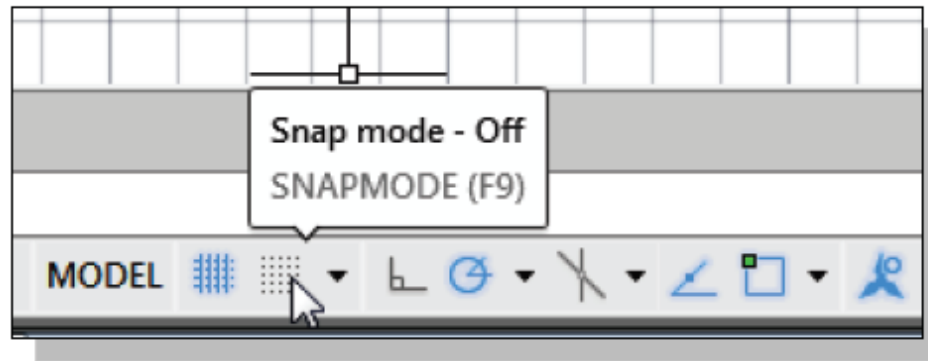
2. Move the cursor inside the graphics window, and estimate the distance in between the grid lines by watching the coordinates displayed at the bottom of the screen.



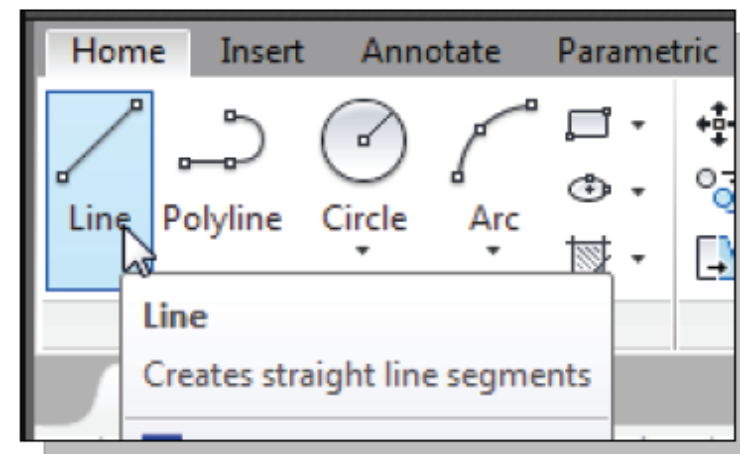
- ✓ The *GRID* option creates a pattern of lines that extends over an area on the screen.
- ✓ Using the grid is similar to placing a sheet of grid paper under a drawing.
- ✓ The grid helps you align objects and visualize the distance between them. The grid is not displayed in the plotted drawing.

SNAP MODE ON

1. Left-click the **SNAP MODE** button in the *Status Bar* to turn **ON** the *SNAP* option.

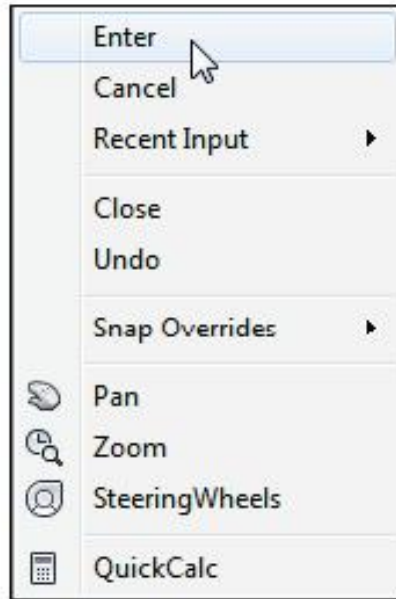
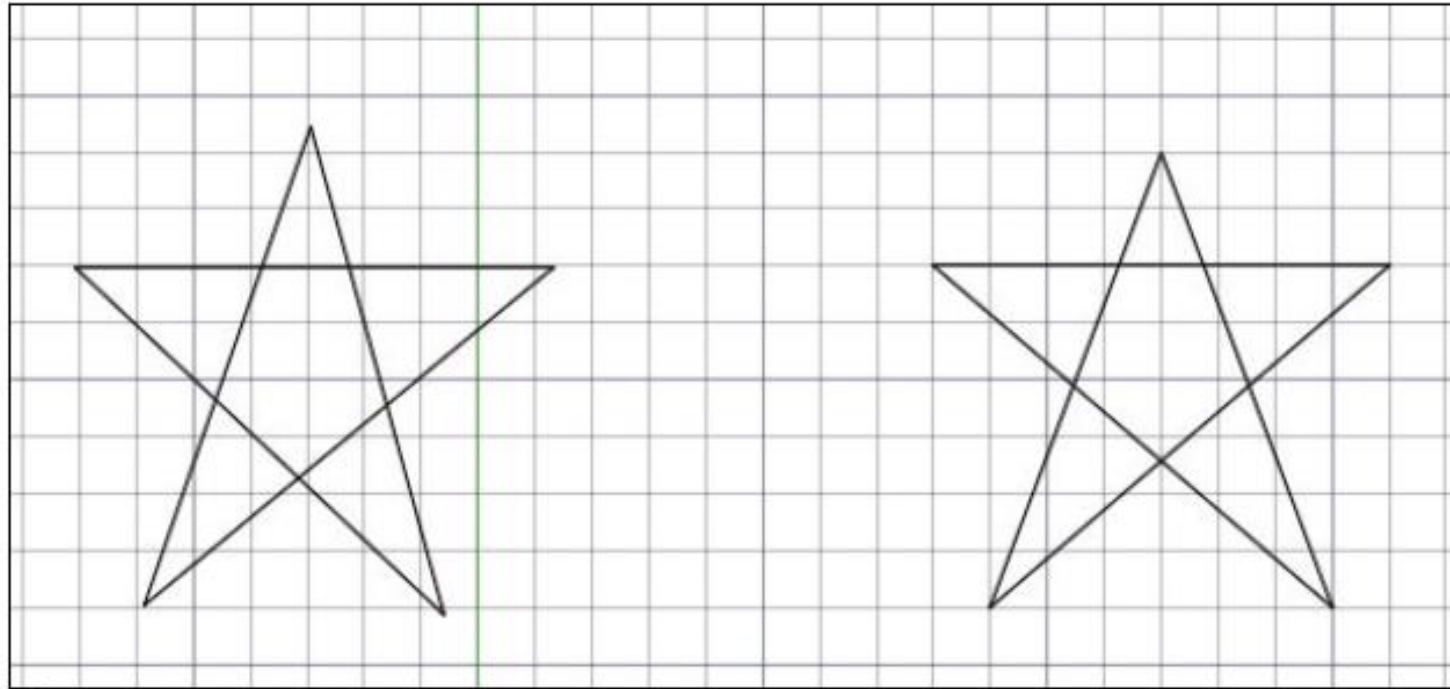


2. Move the cursor inside the graphics window, and move the cursor diagonally on the screen. Observe the movement of the cursor and watch the *coordinates display* at the bottom of the screen.
- The *SNAP* option controls an invisible rectangular grid that restricts cursor movement to specified intervals. When *SNAP* mode is on, the screen cursor and all input coordinates are snapped to the nearest point on the grid. The default snap interval is 0.5 inches and aligned to the grid points on the screen.



3. Click on the **Line** icon in the *Draw* toolbar. In the command prompt area, the message “*_line Specify first point:*” is displayed.

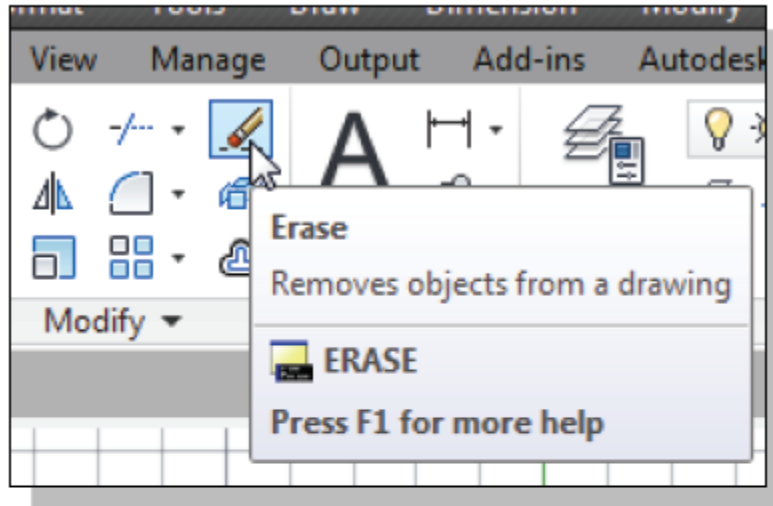
4. On your own, create another sketch of the five-point star with the *GRID* and *SNAP* options switched *ON*.



5. Use the **right-mouse-button** and select **Enter** in the popup menu to end the **Line** command if you have not done so.

Silme Komutu (Using the *ERASE* Command):

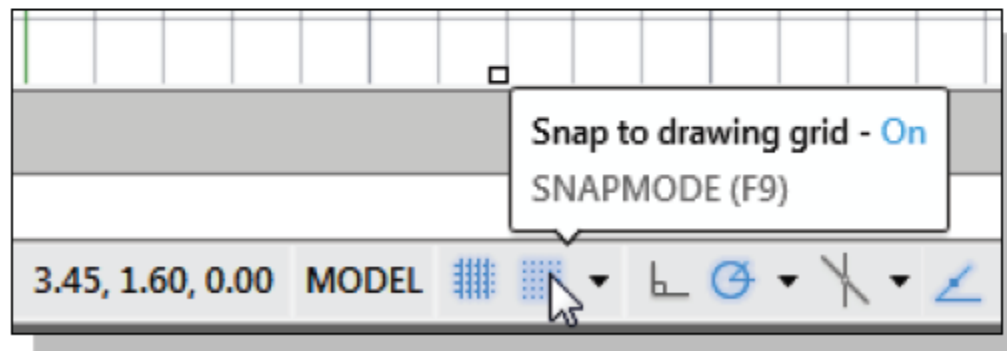
- ❖ One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will erase two of the lines using the **Erase** command.



1. Pick **Erase** in the *Modify* toolbar. (The icon is a picture of an eraser at the end of a pencil.) The message “*Select objects*” is displayed in the command prompt area and AutoCAD awaits us to select the objects to erase.

- ✓ Seçim iptali için SHIFT e basılı tutulması gerekiyor (To **deselect** an object from the selection set, hold down the [**SHIFT**] key and select the object again).

2. Left-click the **SNAP MODE** button on the *Status Bar* to turn **OFF** the **SNAP MODE** option so that we can more easily move the cursor on top of objects. We can toggle the *Status Bar* options **ON** or **OFF** in the middle of another command.



3. Select any two lines on the screen; the selected lines are highlighted as shown in the figure below.

✓ **Mauseun sađ tuřuna tıklanarak silme iřlemi gerekleřtirilir**

4. **Right-mouse-click** once to accept the selections. The selected two lines are erased.

