



Lab II

Basics of AutoCAD (1)

Objectives

In this lab you will learn slightly more advanced features of AutoCAD:

- Drawing Area:
 - Zoom/Pan
 - Regenerate/Redraw
- Drawing Commands:
 - Rectangle
 - Circle
 - Polygon
 - Arc
 - Ellipse

1 Introduction

In the previous lab, we gave a small intro to AutoCAD and we learned the basic commands such as drawing a point and a line. In this lab, slightly more advanced drawing commands will be introduced. “AutoCAD allows you to have access to a large number of commands. A general rule is that you will use 20% of the commands 80% of the time.

The important thing to remember is that AutoCAD will expect you give it information in a very particular order. The most frustrating thing when you begin using this program is that you will try to do something, but AutoCAD will 'not work'. In most cases, it means that you are trying to input information at the wrong time. This is why it is **very important** to be in the habit of looking at the command line.

The command line tells you what information AutoCAD requires to continue. [2]

2 Theoretical Background

The drawing area is the black/grey space where you perform your drawing (also called Model space). This is like a drawing paper; you can zoom and pan as much as you want (till infinity).

If you press F7 you can toggle the grid.

2.1 Drawing Area

2.1.1 Zoom

In order to **zoom in and out you can use the scroll wheel on the mouse** (10% zooming factor). However if you want to have more technical zooming options use the zoom command:

Command	PDM	TB	Shortcut
zoom	View→Zoom (or right-click→zoom)		z

If you type in z enter on the command line, you will get the following:

Specify corner of window, enter a scale factor (nX or nXP), or [[All](#)/[Center](#)/[Dynamic](#)/[Extents](#)/[Previous](#)/[Scale](#)/[Window](#)/[Object](#)] <[real time](#)>:

Note: When you type a command in AutoCAD, you will see the options being displayed in the command window. The options are displayed between brackets []. The default option is displayed between arrowheads <>. If you directly press ENTER the default

option will be chosen, however if you type the capital letter of an option and press ENTER this option will be chosen.


In this case you can:

- 1- **Zoom Window:** With you mouse specify a zooming window. Zooms to display an area specified by two opposite corners of a rectangular window.
- 2- **Zoom Scale:** Enter a scale factor (nX or nXP): Enter a value followed by **x** to specify the scale relative to the current view. For example, entering **.5x** causes each object to be displayed at half its current size on the screen. (Later on in the course we will see the difference between nX and nXP).
- 3- **Zoom All:** If you press ‘a’ ENTER, the zooming will be done such that all the drawings and the global coordinate will be filling the screen.
- 4- **Zoom Extents:** If you press ‘e’ ENTER, the zooming will be done such that all the drawings will be filling the screen. **Double clicking on the scroll wheel will perform a zoom extent.** This can be really helpful in case you got lost in the drawing space.
- 5- **Zoom Object:** If you press ‘o’ ENTER, Zooms to display one or more selected objects as large as possible and in the center of the drawing area. You can select objects before or after you start the ZOOM command.

These are the most important options; you can try the remaining options (Center/Dynamic/Previous/real time) on your own.

2.1.2 Pan

Pan allows moving the paper on which we are drawing and not the objects. Pan can be performed by **clicking the scroll wheel while moving the mouse.**

Command	PDM	TB	Shortcut
pan	View→pan (or right-click→pan)		p

2.1.3 Redraw/Regenerate

From the AutoCAD point of view, each drawing has two parts:

- The important part is the DWG (drawing) file, a highly precise database of objects that is stored on disk. AutoCAD uses high-precision numbers to describe the location of each object.
- The less important part is the part that you interact with — the on-screen display of the drawing. For displaying your drawing on-screen, AutoCAD uses less-precise integer numbers that are faster for the computer to calculate and that work well with screen displays.

You need to understand the distinction between the highly precise DWG file version of your drawing and the less-precise screen version because it's possible for the two to get out of

sync. For example, and as mentioned previously, you can Zoom in/out or Pan as much as you want, till infinity. However if you try to do this you will realize that you will reach a limit. After that limit you have to regenerate in order to keep zooming or panning.

Also, sometimes when you zoom in on a circle for example, you realize that the circle becomes a polygon; you have to regenerate in order to be able to see the circle again.


The **Regenerate** command (regen) performs the synchronization described in this section, while Redraw on the other hand simply repaints the screen without attempting to synchronize with the drawing database. The **Redraw** command was useful in the days of very slow computers and older versions of AutoCAD, which didn't handle the display as effectively, but it's essentially a useless command now.

(Read more: [4])

Command	PDM	TB	Shortcut
Regenerate	View→regen		re

2.2 Drawing Commands

2.2.1 Rectangle

Command	PDM	TB	Shortcut
rectangle	draw→rectangle		rec

Example: Draw a rectangle by specifying the first corner point and the opposite corner point, i.e. First corner: #0,0 Opposite corner: @20,10

When you press rec ENTER you will get the following on the command window

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Specify other corner point or [Area/Dimensions/Rotation]: a
```

After specifying the first corner point, we can – instead of specifying the coordinates of the second point – use area of the rectangle. Hence, write “a” in the command line and then press enter. Now you need to input the area (e.g.: 200)

Next you need to specify if you are drawing with respect to the Length (by pressing “l”) or Width (by pressing “w”) of the rectangle.

Note: If length is written between brackets (e.g.: <length>), this means that if you press enter without entering your choice, AutoCad will take the length as the default setting.

```
Enter area of rectangle in current units <100.0000>: 200
Calculate rectangle dimensions based on [Length/Width] <Length>:
```

Finally you need to enter the length or width of the rectangle depending on the choice of the previous step

```
Calculate rectangle dimensions based on [Length/Width] <Length>: 1
Enter rectangle length <10.0000>: 10
```

2.2.2 Circle

Command	PDM	TB	Shortcut
circle	draw → circle		c

```
Command: c
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
```

We have 5 options to draw a circle:

- 1- **Default:** Specifying center point and radius (or diameter):

You need to specify two main features in order to draw a circle:

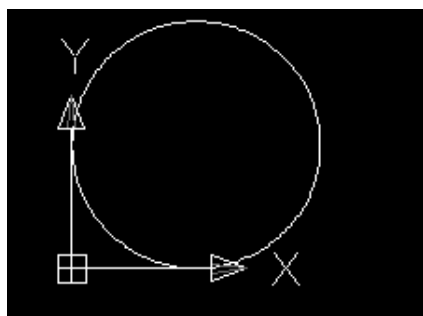
- Center point (e.g.: @10,0)
- Radius (e.g.: 5)

```
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @10,0
Specify radius of circle or [Diameter]: 5
```

Another option would be to input diameter instead of entering the radius. Press “d” for diameter and then press ENTER. Afterwards, input the given diameter.

- 2- **2P:** This option allows drawing a circle by specifying two points, i.e. by giving the diameter.

```
Specify first end point of circle's diameter: 10,0
Specify second end point of circle's diameter: 10,20
```

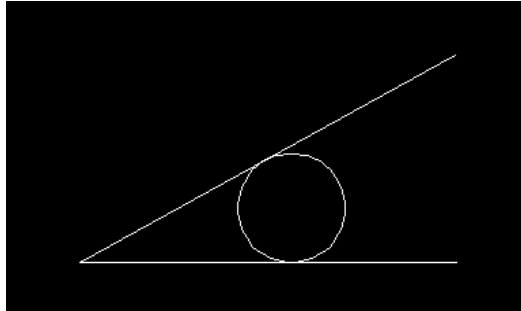


- 3- **3P:** This option allows drawing a circle by specifying three points.

4- **Ttr: Tangent, Tangent, Radius**

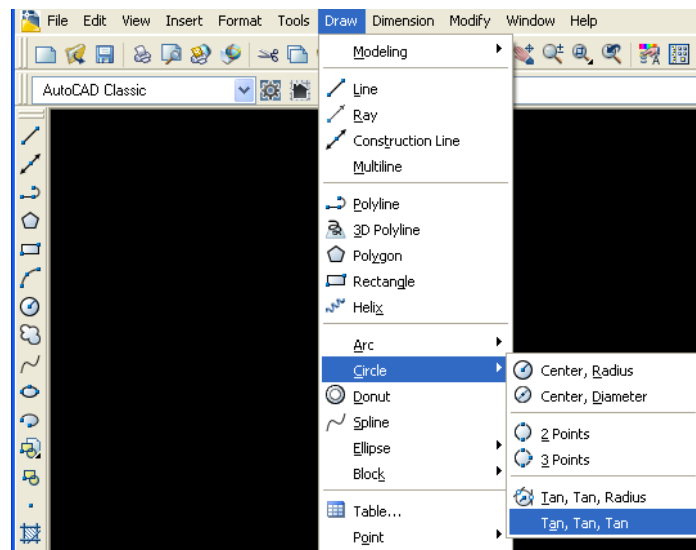
It allows drawing a circle that is tangent to two lines and that has a specific radius.

```
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: t
Specify point on object for first tangent of circle:
```

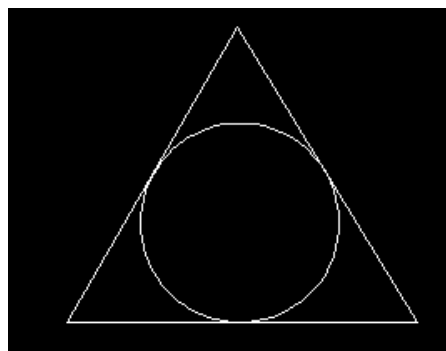


5- **Ttt: Tangent, Tangent, Tangent**


This feature is only available from the pull down menu



It allows drawing a circle that is tangent to three lines (e.g.: draw a circle inside a triangle).



2.2.3 Polygon

Command	PDM	TB	Shortcut
polygon	draw → polygon		pol

Steps to drawing a polygon:

- a. Define the number of sides (for example 6 sides)

```
Command: _polygon Enter number of sides <4>: 6
```

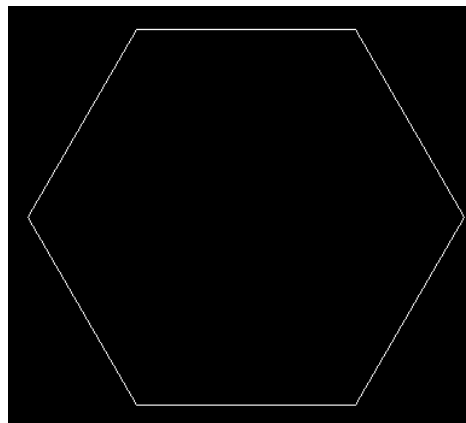
- b. Define the center point or the edge

Note: To use the edge instead of the center press “e” and then enter

```
Specify center of polygon or [Edge]: e
```

Then you need to specify first and second endpoint of edge

e.g.: @0<0 (pt.1) @10<0 (pt.2)

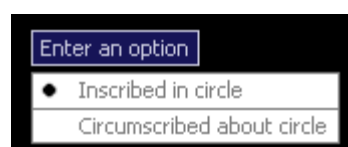


- c. If you define the center point (e.g.: @0,0), you need to specify whether the polygon is inscribed in a circle or circumscribed about a circle.

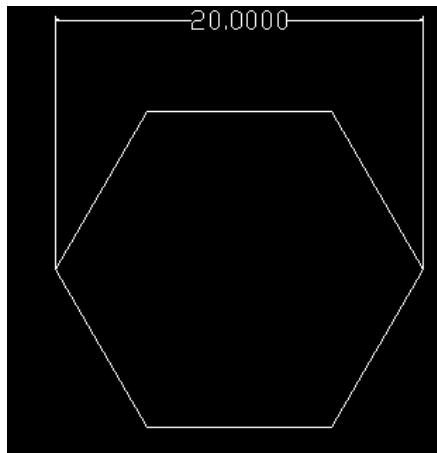
```
Specify center of polygon or [Edge]: @0,0
Enter an option [Inscribed in circle/Circumscribed about circle] <I>:
```

Hence, you can input “c” for circumscribed or press enter for inscribed.

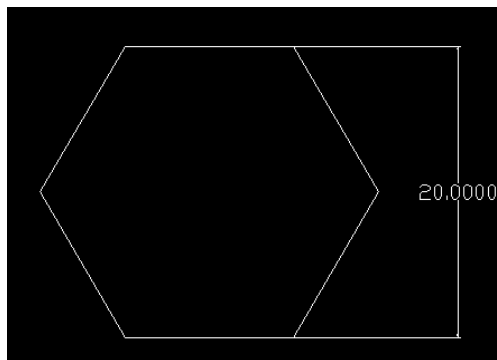
Note: You can choose the option directly on the graphic screen



```
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: I  
Specify radius of circle: 10
```



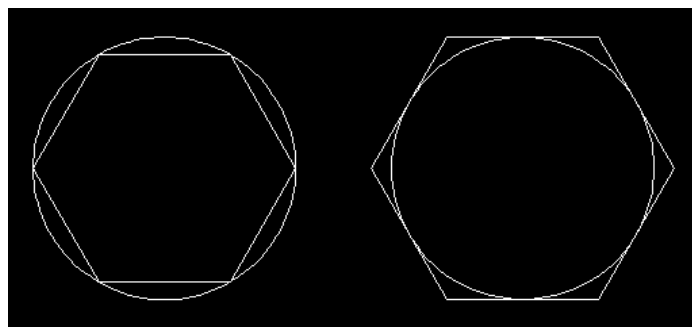
Inscribed in a circle of radius 10




Circumscribed about a circle of radius 10

Note:

- If the polygon is inscribed in a circle of radius “x”, then the length of the polygon from left to right is “2x”.
- If the polygon is circumscribed about a circle of radius “x”, then the length of the polygon from top to bottom is “2x”.



2.2.4 Arc

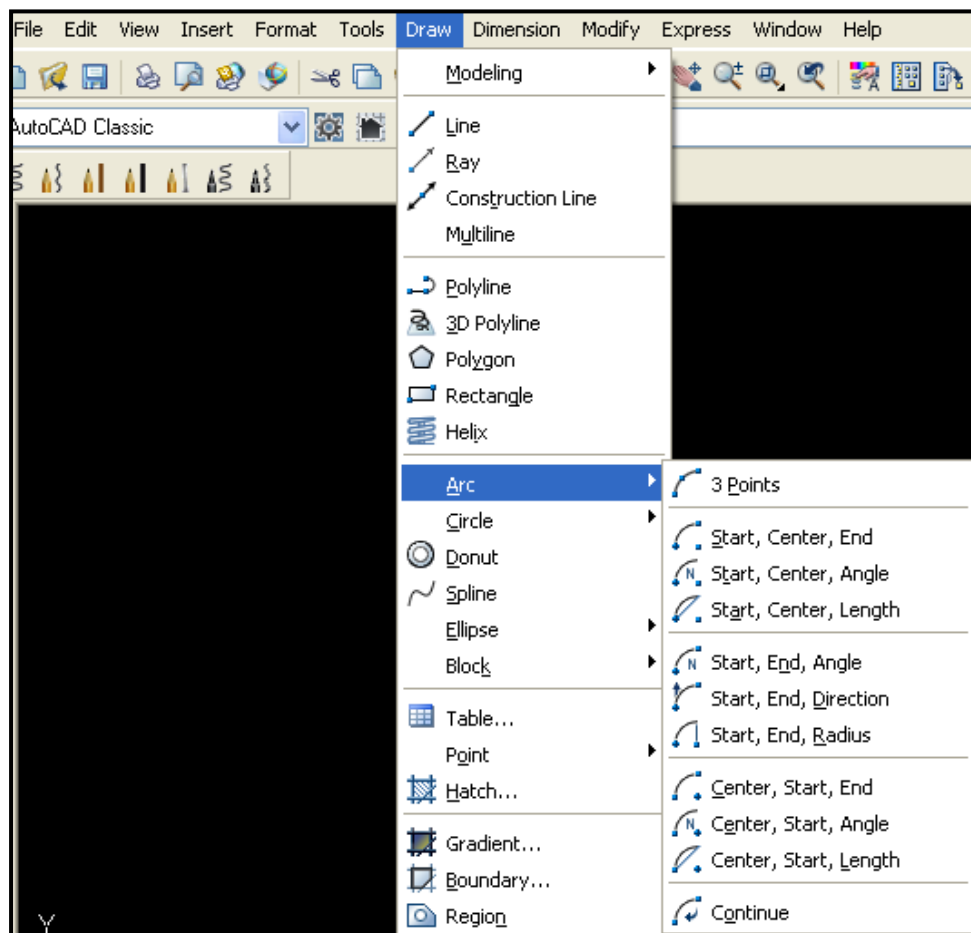
Command	PDM	TB	Shortcut
arc	Draw → arc		a

In order to check all the options available for drawing an arc, open the pull-down menu. Pull down menu: Draw → Arc

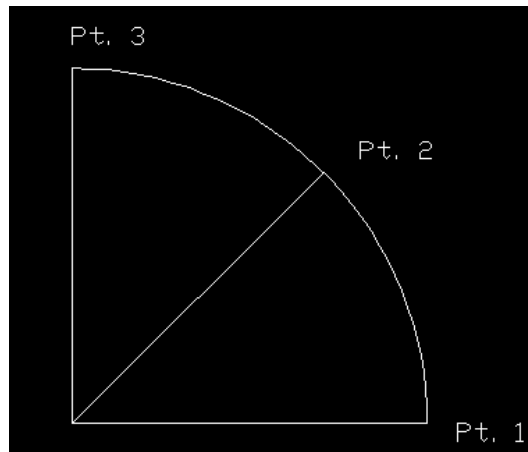
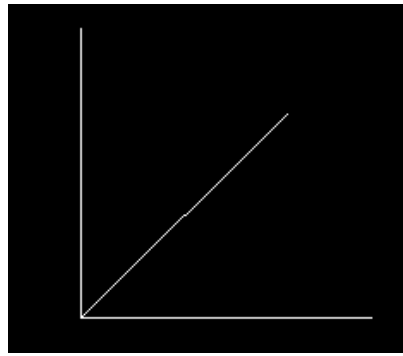
Note:

- The arc command includes several approaches to draw an arc such as “3 points” (draw an arc by specifying three points on it), “Start, Center, End” (draw an arc by specifying its start point, its center point, and its end point).
- In AutoCAD, the arcs are drawn counterclockwise

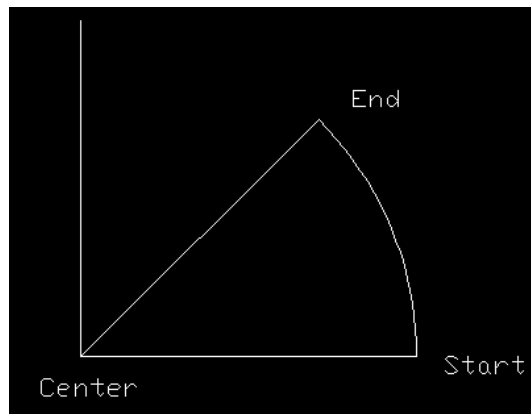
If you use the command line to draw an arc, for each point you will be given to choose whether this is a start, center or end point.

**Example:**

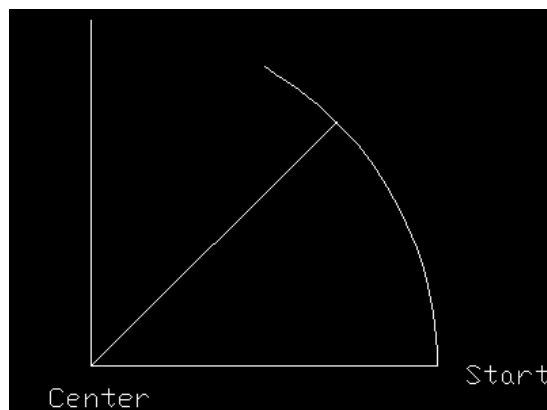
Draw three lines originating from the same point, having the same length (10) but at different angles (0° , 45° , 90°). Then, try the different arc options:



3 points




Start, Center, End



Start, Center, Angle (60°)

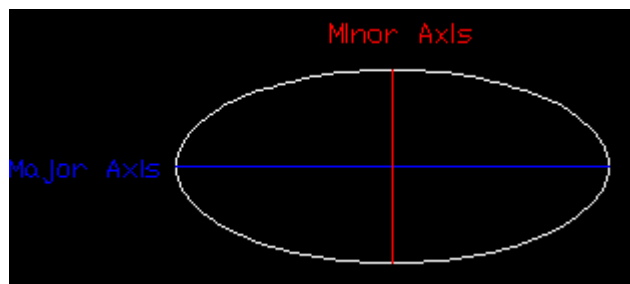
Note: When using the “Start, Center, Length” note that the length is equal to $2\pi R$.

2.2.5 Ellipse

Command	PDM	TB	Shortcut
ellipse	Draw→ellipse		el

Any ellipse is defined using two axes: a major one and a minor one. Therefore, in order to draw an ellipse we need to:

- Specify the two endpoints of an axis, in addition to the distance from the first axis to the other axis.
- Or, specify the center of the ellipse, one endpoint of an axis, and the distance to the other axis.



3 Exercises

3.1 Exercise 1: Zoom (Window / Extent)

Practice the zoom window and zoom extents (explained in the previous section)

References

- [1] www.autodesk.com
- [2] <http://www.we-r-here.com/cad/tutorials/>
- [3] <http://www.cadtutor.net/tutorials/autocad/>
- [4] <http://www.dummies.com/how-to/content/degenerating-and-regenerating-in-autocad.html#ixzz1PKoh5xjf>