



Lab I

Introduction to AutoCAD

Objectives

This lab is an introduction to the basics of AutoCAD 2D:

- Command line/Pull down Menu/Toolbar/Shortcuts
- Points/Lines
- Undo/Redo/Erase
- Pick Box: Contained and Crossing
- Coordinate System
- Osnap

1 Introduction

Computer-aided design (CAD), also known as **computer-aided design and drafting (CADD)**, is the use of computer technology for the process of design and design-documentation. AutoCAD is an example of such software.

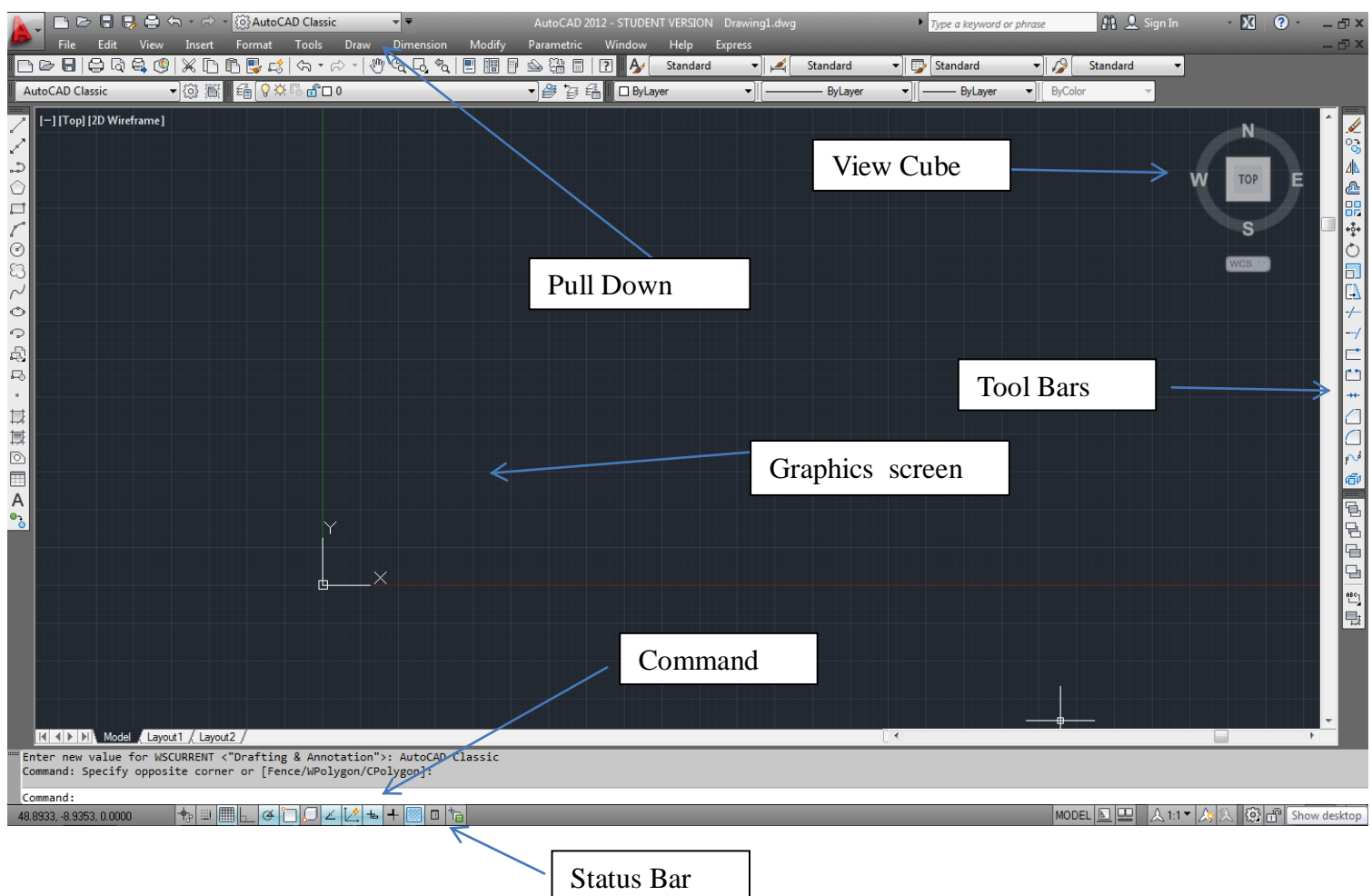
Computer-aided Engineering (CAE) is another kind of software that includes CAD and design analysis engineering tools such as Finite Element Analysis (FEA). Solidworks and proE can be examples of such softwares.

Computer-aided Manufacturing (CAM) is the use of computer software to control machine tools and related machinery in the manufacturing of workpieces (Computer Numerical Control-CNC machines).

Throughout the entire semester we will be dealing with CAD and specifically we will be learning how to do 2D drafting and 3D modeling on AutoCAD. The focus will be on **AutoCAD 2012**, however you can use any other version.

First, Open AutoCAD 2012. On top of the screen, where you have “Drafting & Annotation”, press the pull down menu, and choose **AutoCAD Classic**. From now on we will always be working with AutoCAD classic view for consistency reasons.

Main Screen:



1.1 Communication

F1	F2	F3	F7	F8	F9	F12	ESC
Help	Command Line	Osnap	Grid	Ortho	Snap	Dyn	Cancel

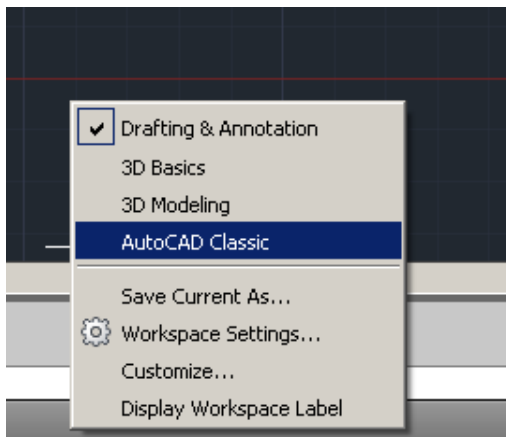
2 Theoretical Background

2.1 Switch workspace layout:

Use the *workspace switching* button at the bottom of the screen to switch to autocad classic.



Workspace switching




2.2 Commands

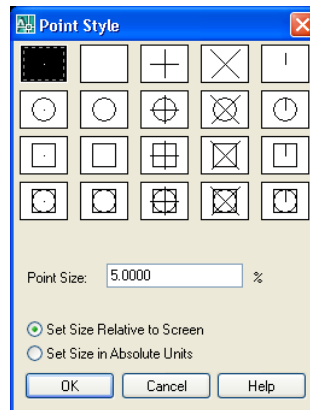
Each command can be called using four different methods:

- Command
- Pull-down menu (PDM)
- Tool bar (TB)
- Shortcut


2.2.1 Point:

Command	PDM	TB	Shortcut
point	Draw → point → single point (→multiple point)		po

Note: To change the style of the point, go to the pull-down menu: Format → Point style



2.2.2 Line:




Command	PDM	TB	Shortcut
line	Draw → line		l

Note: Draw multiple lines that are connected, when you want to close the drawing (i.e. connect the last point to the first one) use the close command:

- Command: [close](#)
- Shortcut: [c](#)

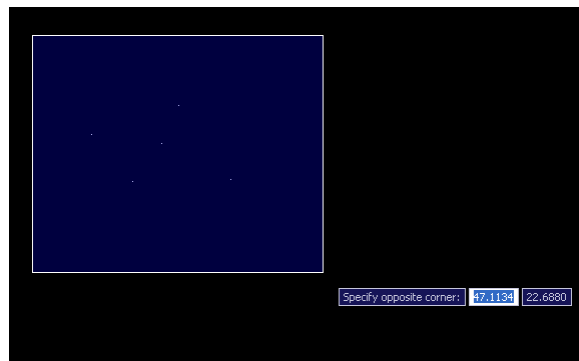
If you want to exit any command you are using, just press [Esc](#)

2.2.3 Undo/Redo/erase:

Command	PDM	TB	Shortcut
undo	edit → undo		Ctrl+z
redo	edit → redo		Ctrl+y
erase	Modify → erase		e

2.3 Object Select: Pick Box

1. Contained (left-to-right): Select what is inside the box only



2. Crossing (right-to-left): Select what is inside and in contact with the box



2.4 Coordinate system

If you want to draw any engineering drawing you need accurate and detailed measurements. For example, the site engineer will be measuring the distance on the AutoCAD map and knowing the scale (example 1:100) he will be able to tell the real distance on site. For this, and to be able to draw accurate drawings we need to work with coordinate systems.

We have two types of referencing: Absolute and relative. Absolute is when we reference with respect to the global coordinates, while relative is when we reference with respect to a certain point.

We have two types of coordinate systems: Polar and Cartesian. Polar is when we specify the magnitude and angle. Cartesian is when we specify x and y (and z if in 3D) coordinates. The following table summarizes the symbols used in order to indicate which referencing and coordinate system to use:

	Relative	Absolute
Cartesian	@ x,y	# x,y
Polar	@ Mag<Angle	Doesn't exist

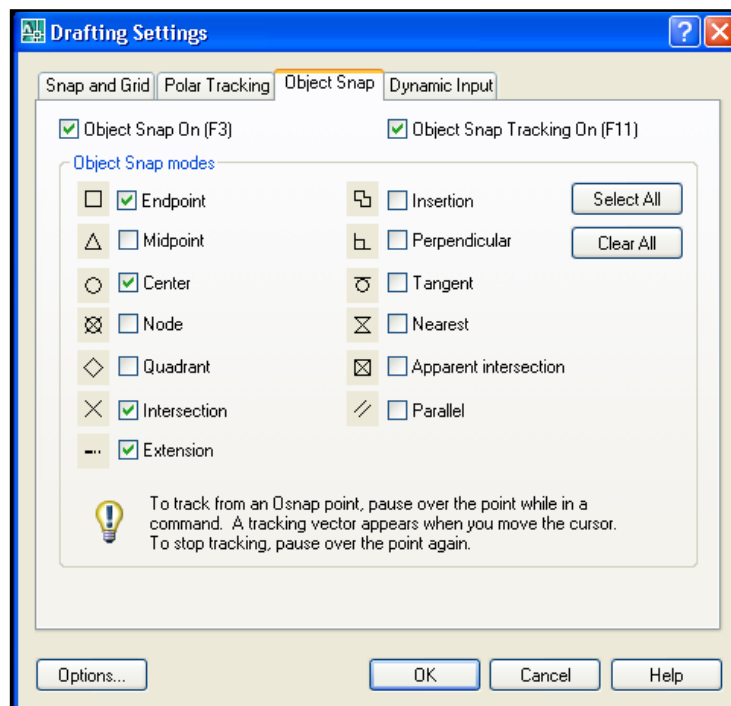
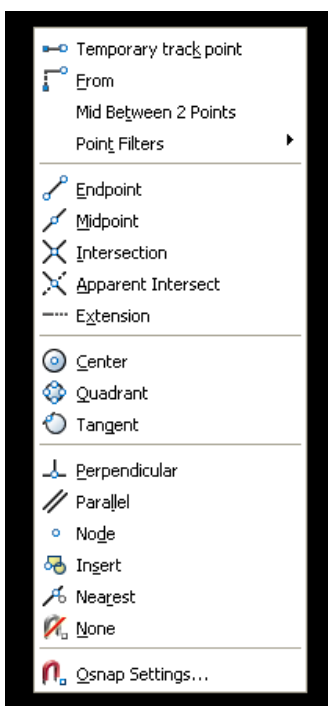
Note: If you press F12 You can toggle Dynamic input (Dyn) on and off (look at the status bar when you do so). This represents the writing in the above figure which helps to see the coordinates of the point, line, etc. If you have Dyn off you can only use Absolute system.

2.5 Object Snap (OSNAP)

Command	PDM	TB	Shortcut
osnap	Shift→right click→Osnap settings or tools→drafting settings		os

While drawing an object, you have the option to let the cursor snap on certain specific points. Suppose you want to draw a line from the center of the circle to the middle of the vertical line you extended earlier. AutoCAD has a feature that makes this very easy. These are the Object Snaps (or Osnaps "Oh-Snaps"). Type **OS <ENTER>**. You will see this dialog box appear.

OSNAP allows you to point out the midpoint, the endpoint, the tangent point, etc. of the objects you are drawing.



You may select whichever points you want to 'snap' on an object.

You can also snap using command lines. For example if you have a line already drawn and you want to snap another line to its midpoint, you can type MID and the snap point will be shown. Here is a list of your options, followed by the command entry to invoke the needed Osnap.

Endpoint - snaps to either the beginning or the end of an object such as a line - **END**

Midpoint - snaps to the exact middle of a line or an arc - **MID**

Center - snaps to the center-point of a circle or arc - **CEN**

Node - snaps to 'nodes' (not covered in this course) - **NOD**

Quadrant - snaps to any of the four quadrants of a circle - **QUA**

Intersection - snaps to the point where two object cross - **INT**

Extension - Snaps to the phantom extension of an arc or line - **EXT**

Insertion - snaps to the insertion point of an object (such as a block or text) - **INS**

Perpendicular - will snap so that the result is perpendicular to line selected - **PER**

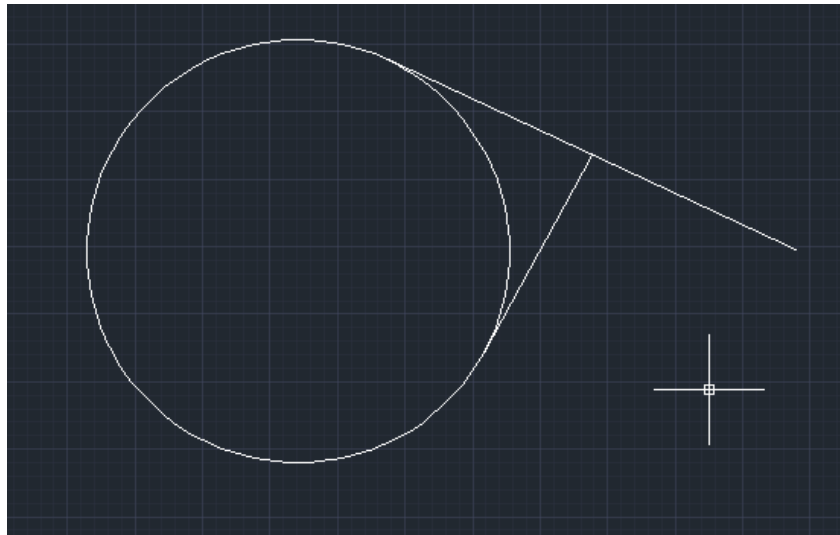
Tangent - snaps to create a line tangent to a circle or arc - **TAN**

Nearest - will find the closest point an object and snap to that point - **NEA**

Parallel -Snaps parallel to a specified line - **PAR**

Example:

Choosing osnap for tangent and midpoint, you are able to draw the following figure.



3 Exercises

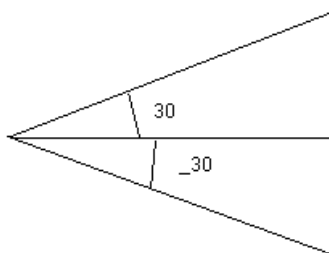
3.1 Exercise 1:

Draw the following rectangle using lines, and using: 1- Absolute Cartesian system and 2- Relative Cartesian system:



3.2 Exercise 2:

Draw the following image using Relative Polar coordinate system:



All lines are of length 10.

3.3 Exercise 3:

To check lengths, use the *di* command.

Note: If you write distance or *di* in your command line, you can measure the distance between two points by just selecting the first point and then the second point. You will get the below output.

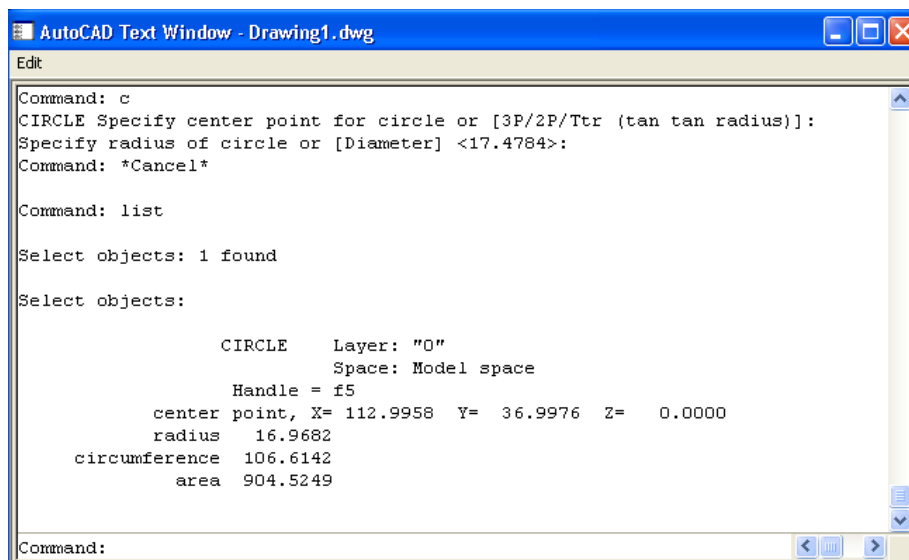
```
Command: di
DIST
Specify first point:
Specify second point or [Multiple points]: *Cancel*
Command:
DIST
Specify first point:
Specify second point or [Multiple points]:
Distance = 18.0278, Angle in XY Plane = 326, Angle from XY Plane = 0
Delta X = 15.0000, Delta Y = -10.0000, Delta Z = 0.0000
Command:
```

Output of the 'di' command

3.4 Exercise 4:

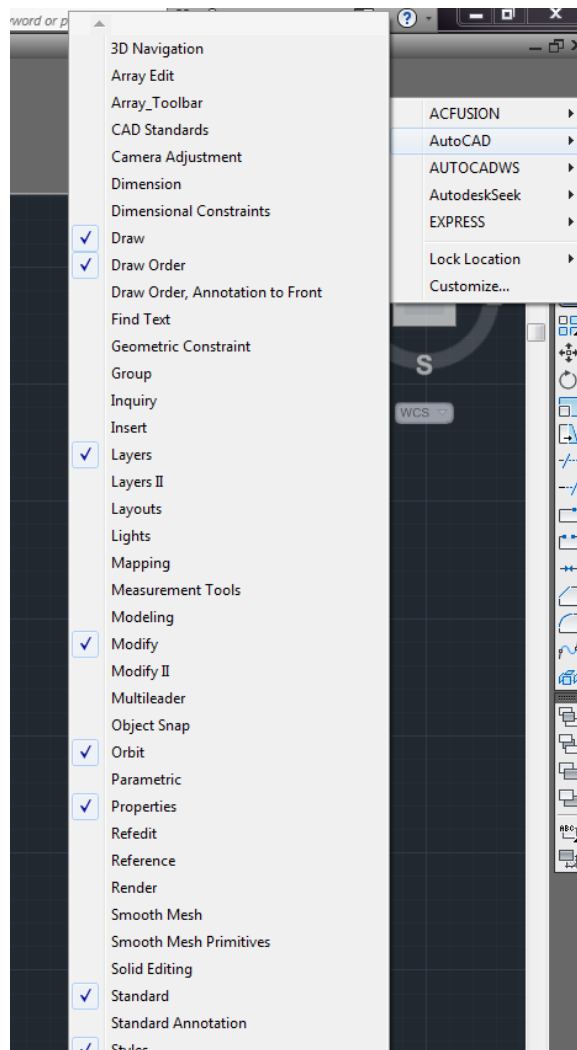
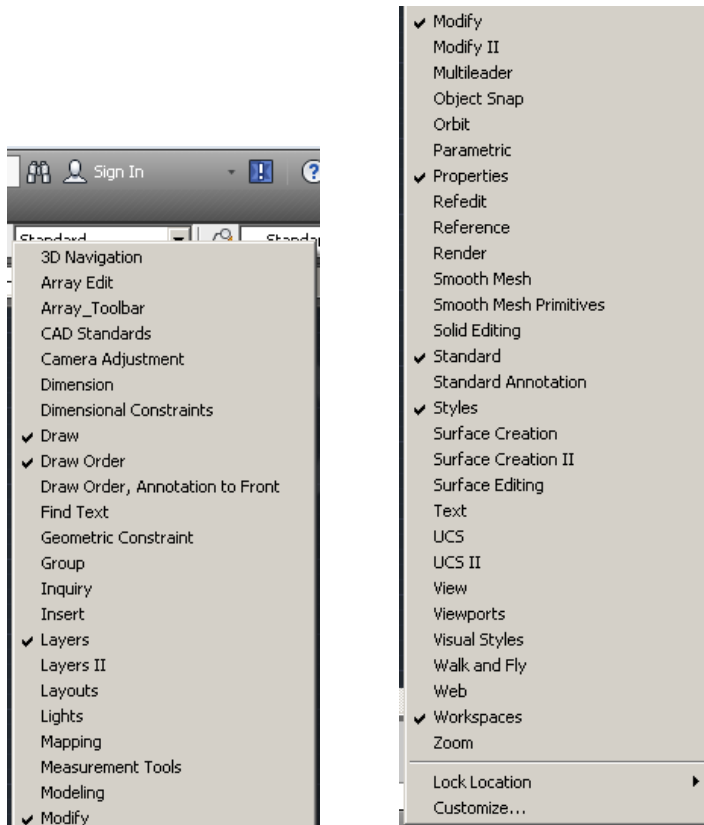
To check object information, use the *li* command.

Note: If you have an object drawn on your graphic screen, for example a circle, you can use the list command “*li*” to get information about your object.



3.5 Exercise 5:

To add a toolbar right click on any available toolbar to see a list of all the toolbars in AutoCAD and select *dimension*; or right click on the upper right side near the layer toolbar and choose AutoCAD then select the toolbar that you want to add.



References

- [1] www.autodesk.com
- [2] <http://www.we-r-here.com/cad/tutorials/>
- [3] <http://www.cadtutor.net/tutorials/autocad/>