These tutorials are intended for ABSOLUTE BEGINNERS and are to accompany tutor led training in the Department's Computer Suites, using PC's. (The 'user interface' is different on MAC's but the overall concepts remain the same.) Most users prefer to use a mouse with a left and a right button and a central scrolling wheel.

As a student you are able to download a copy of AutoCAD to your own computer / laptop from here http://www.autodesk.com/education/free-software/all .

You will be offered the latest version ie AutoCAD 2015 but bear in mind this will not look like the version you are learning on. Opt for an earlier version such as 2013 / 2014 as these have all the features you will need and the screen will match that in the hand-outs.

Many people are afraid of AutoCAD- fearing its 'complexity'. However, although it can be used to do amazingly complex things we will be learning the SIMPLE, BASIC, tools that will be enough to produce simple 2D drawings and to print them at scale.

There are many places to get help whilst you are learning –

- Ask a fellow student in the computer suite on floor 11.
- From the internet – there are many different organisations offering free CAD tutorials including :-
- From the software designers' website - https://knowledge.autodesk.com/support/autocad/getting-started/caas/CloudHelp/cloudhelp/2015/ENU/AutoCAD-Core/files/GUID-2AA12FC5-FBB2-4ABE-9024-90D41FEB1AC3-htm.html. It's called the " Hitchhikers Guide to AutoCAD basics".
- From "Lynda" – free tutorials on a massive range of subjects – available via the University by logging into MUSE, clicking the 'View all services" link and searching for 'Lynda' in the alphabetical list
- By asking a member of staff who is familiar with AutoCAD
- By clicking on the 'help' button at the top of the screen.

In these tutorials we will be working our way through the tools necessary to produce the plans for the construction and the planting aspects of your current modules ; to annotate them; and to plot at a chosen scale. We will also show you how AutoCAD is increasingly used in practice.

SESSION ONE

AutoCAD is installed on all the department's computers on Floors 10 and 11 so should be available from the start menu once you've logged in.

It may take a while to get started but when it does we are aiming for your screen to look like this.
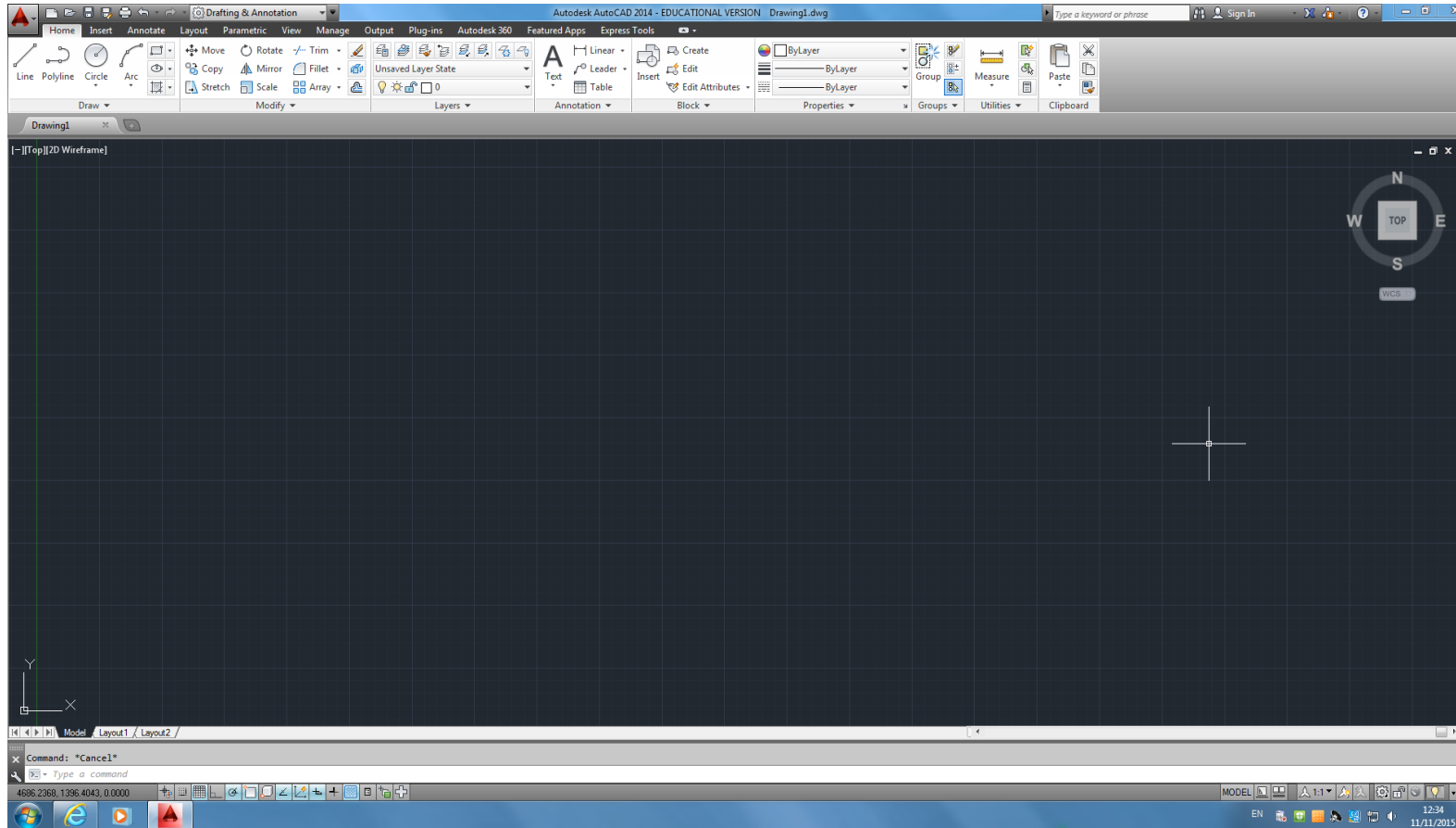

The 'Welcome' screen will be at the front but can be closed in the usual manner – click on the 'x'. If you have and drop down menus floating over the wide black drawing space then close them in the same way until your screen looks like this. If AutoCAD is open but you don't have the same ribbon of tools across the top or the big black space to draw in then go to the next page.

This next procedure is to create a NEW drawing to work on.

Click on the red letter A on the top left of your screen

Click on the 'New' button Your screen should then look like this. This drop-down menu contains a list of different templates built into ACAD that we could use.

All we need is to click on the down arrow next to 'Open' then choose 'Open with no template – Metric'

.
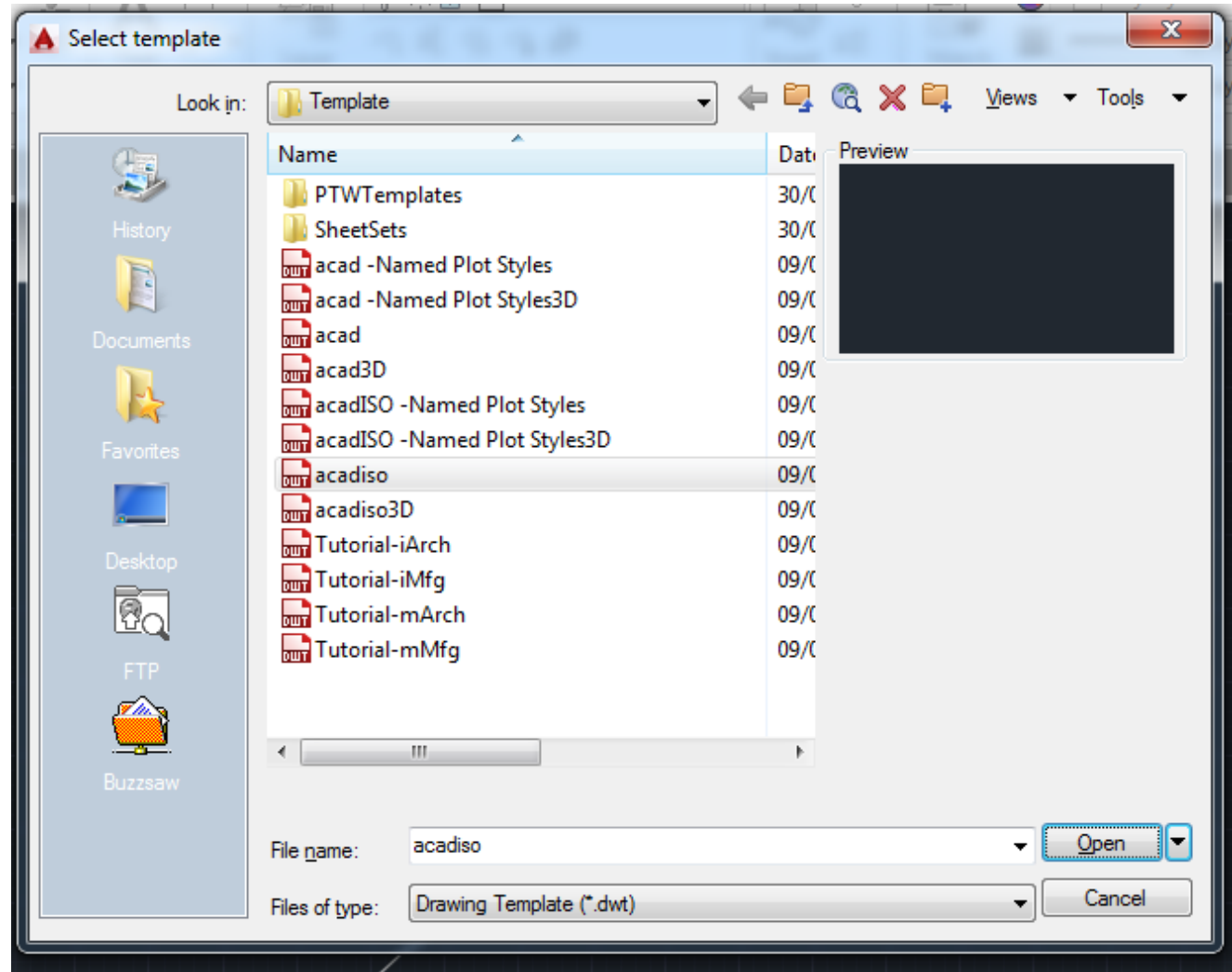
So here is the starting position again.

There is a 'ribbon' across the top with a whole strip of palettes of icons - the tools. If you're not sure what an icon or tool does, you can hover the cursor over it for a few seconds and it will give you information on its function and quick keys for using it. As with most software 'drawing' packages if you click on the small down arrow you get a broader set of options. The first ones we'll use will be in the 'draw' set and the 'modify' set.

In the middle is a dark space into which we will be drawing – this is essentially our infinitely big piece of drawing paper – or, in AutoCAD terms – our 'Model space'

The "Command bar" – vital! Watch this all the time

Ignore

The "Status bar" – controls how you are working

Along the bottom of the screen (ignoring the computer's usual 'tray' of programmes etc) there should be 3 separate other 'active' toolbars.

The Command line is essential - we can type into the space here to start commands and if there are options then or steps in a procedure then this is where they will appear. This is where we "talk" to AutoCAD.

The Status bar controls the 'environment' in which we are working.   Hover your cursor

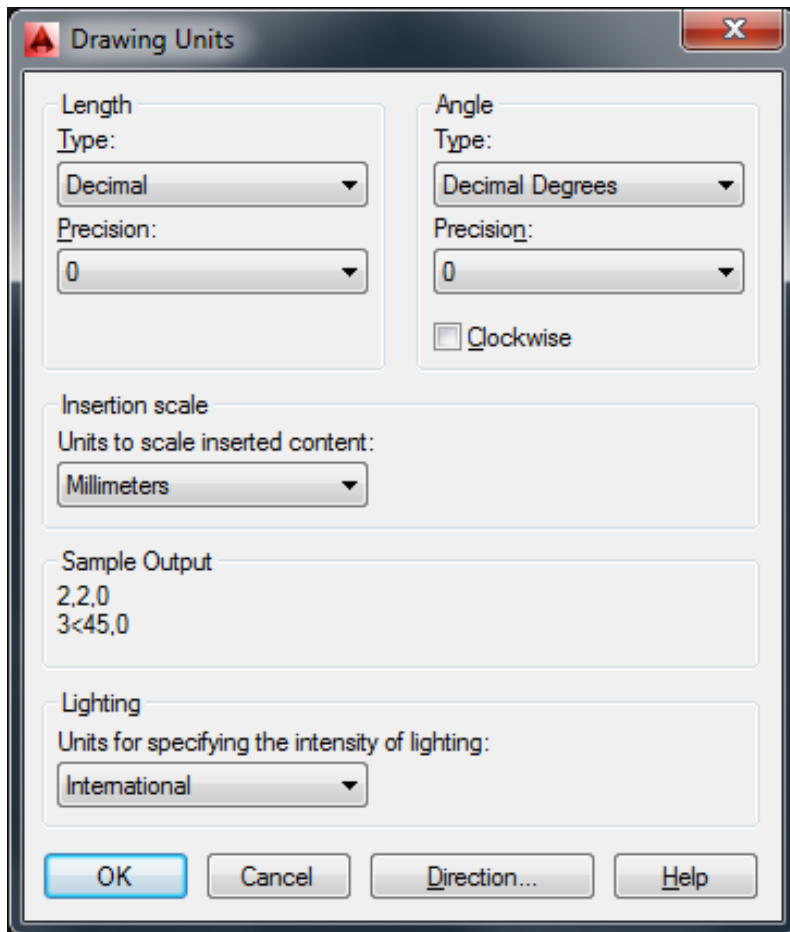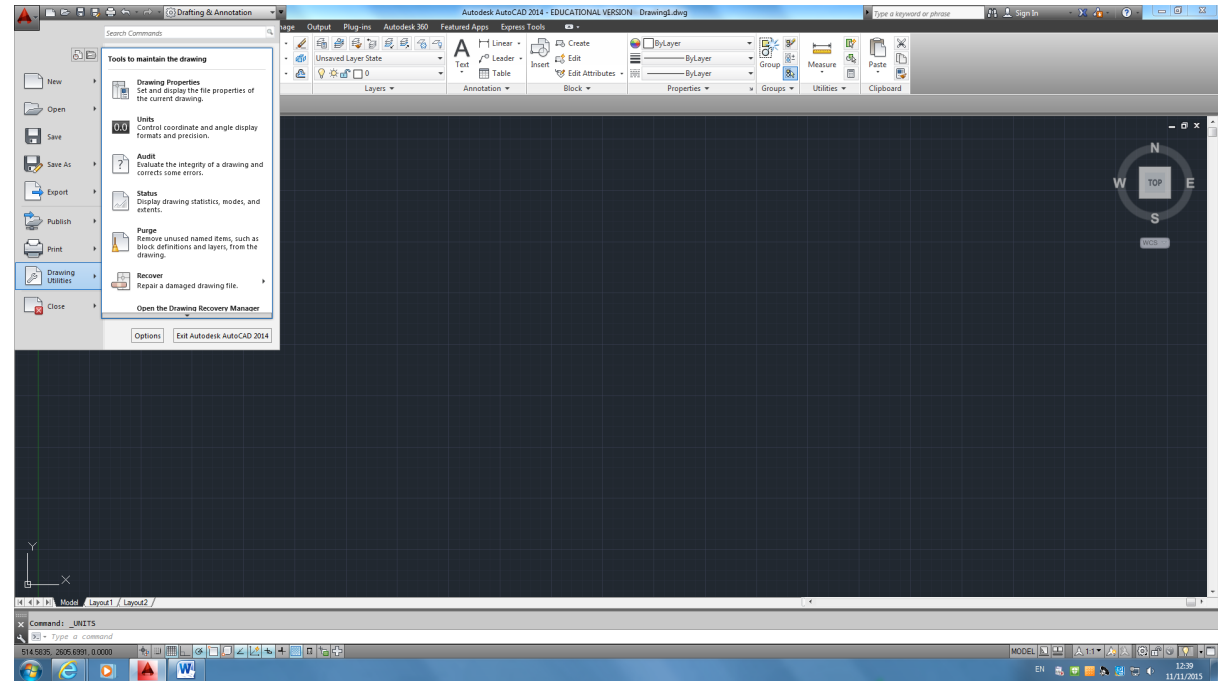 over the icons and it will tell you what each of them are.
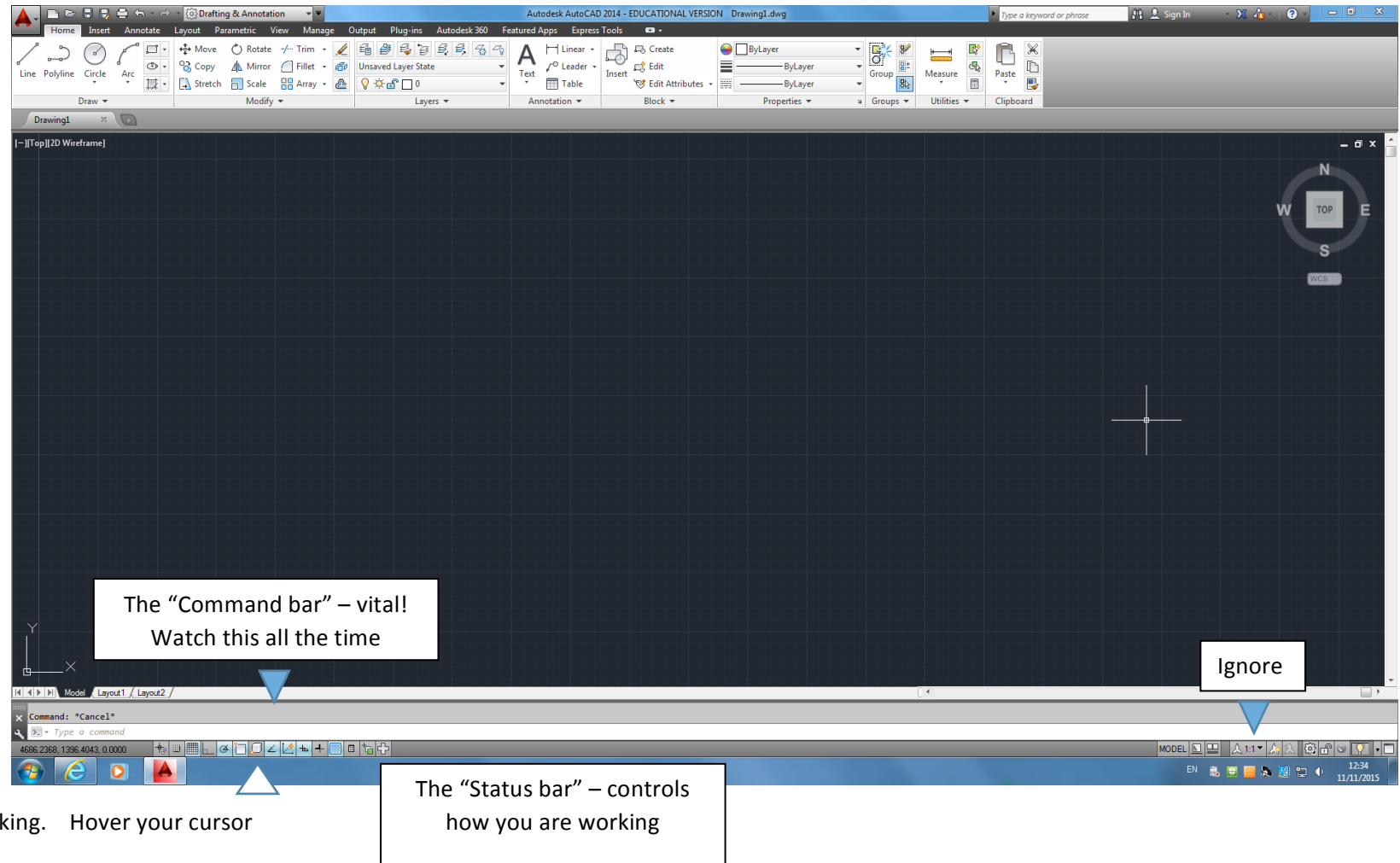
SET THE CORRECT UNITS IN YOUR DRAWING:

This is very important to do as it will affect the dimensions and scale of all your drawings.

Click on the red A -AutoCAD button -and a new list drops down.

Click on "Drawing Utilities" then "Units" and the dropdown menu below will appear.

The default settings should be correct but you need to make sure that the units are set to millimeters by clicking on the "Insertion Scale" pull down menu and choosing millimeters. Also choose "Decimal" and "Decimal Degrees" for the "Length" and "Angle Types" respectively and set the precision to 0.000.

This means that now everything you draw will be in millimeters. The reason this unit is chosen over other, perhaps easier units such as metres is (for now) scaling drawings to print is easier (see Workshop 4)

Millimeters are often used when measuring dimensions on drawings so it's useful to get into the habit of working to them . However, this does take some getting used to - for example if you wanted to draw a 1m line you would have to type 1000 into the command line. A 5m line would be 5000, 20m would be 20000, a 0.1m line would be 100 and so on.

Page 5

DRAWING LINES

To start drawing we need to check on one of the settings on the status bar – the Dynamic Imput function

Those tools that are light blue are 'on'. To turn them on and off we simply (left) click on the icon. Make sure your dynamic imput tool is on.

To begin any task you need to tell AutoCAD by clicking on the tool you want to use in this case 'LINE'   OR type the letter L into the command bar.

Now look at the Command bar – It is asking for the first point ie where to start the line.

Click anywhere on screen and now the command line is asking where is the next point. Move the cursor and click again. It will keep drawing lines. When you want to stop (this is the same for many CAD commands) you can

- press Escape
- press enter
- press space bar which works as 'enter'.
- Right click which produces a new menu and click on 'enter'.

Have a go at this.

However, the best thing about AutoCAD is that we can draw PRECISELY and ACCURATELY and with drawing lines there are 2 methods. After choosing the start point, two small dialogue boxes appear – one bright, one not:-

A) – Line length and angle (as seen on the right)

In the lighter box (usually on the left) type in the **length** of the line you want

Press the tab key (--->)

Now type in the **angle** you want to draw at (Auto CAD tells you what angle the line is currently at)

type the distance
press the Tab button --->
type in the angle
press enter
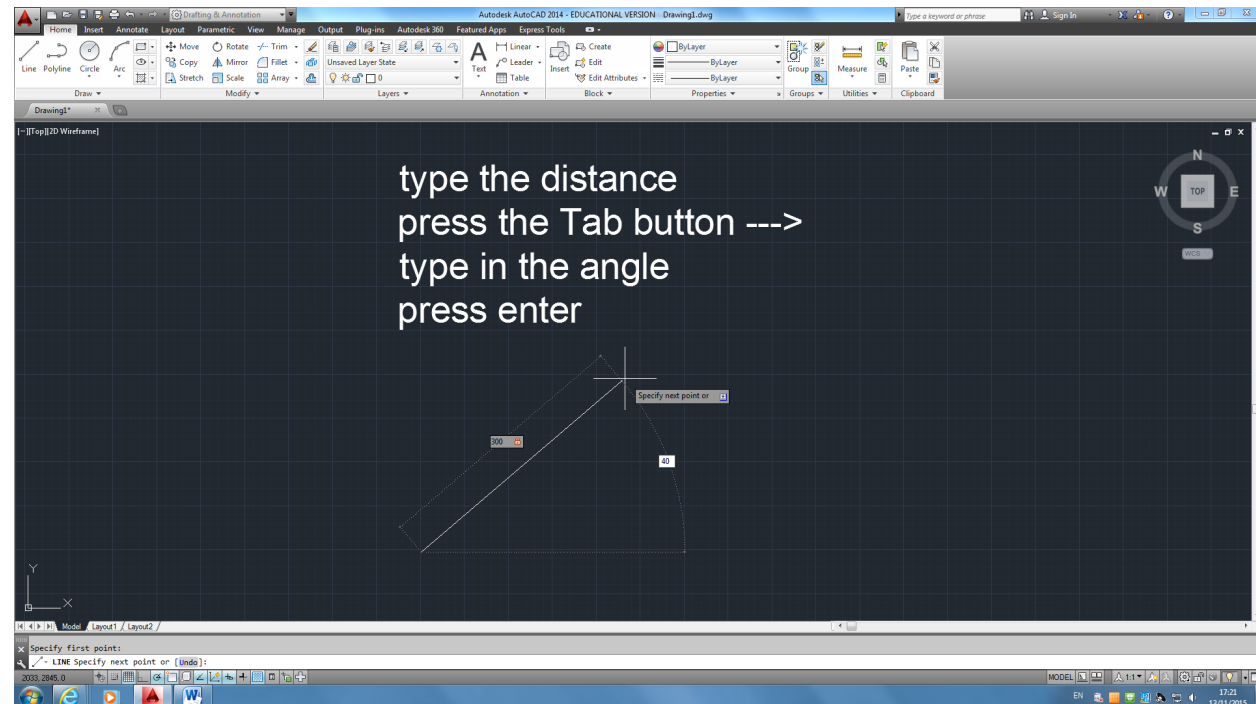
B) – give the 'co-ordinates' of the next point(as seen on the right)

In the lighter box (usually on the left) type in the **horizontal distance** of the point you want the line to go to (the 'eastings')

Press the comma key (,)

Now type in the **vertical distance** you want to draw the line to (the 'northings')

Both these give you complete control of where the line goes to.

Note that the command stays active until you've finished when you press enter / escape.

If, whilst the command is still going you think you've made a mistake then simply type the letter U into the command line and the previous 'action' is undone!

If you've finished the command and want to go back then press CTRL Z. THIS IS THE MOST IMPORTANT PIECE OF INFORMATION SO FAR!!

VIEWING YOUR DRAWING - ZOOMING

You may have already discovered that what you've drawn is either too small to be seen or you can only see a very small part of it. The easiest way to move around your drawing is to use the mouse.
 The picture on the right shows how the mouse buttons function. For zooming and panning the 'wheel ' is the *key* key. Have a go now to get used to how they work.

The other option is to type **Z.** press enter. Look at the command line options. Wherever there is a Capital letter that means we can type just that letter and press Enter to access that command. I usally use E for extents or A for All of my drawing. W for window can be useful when you want to move in to a specific area.
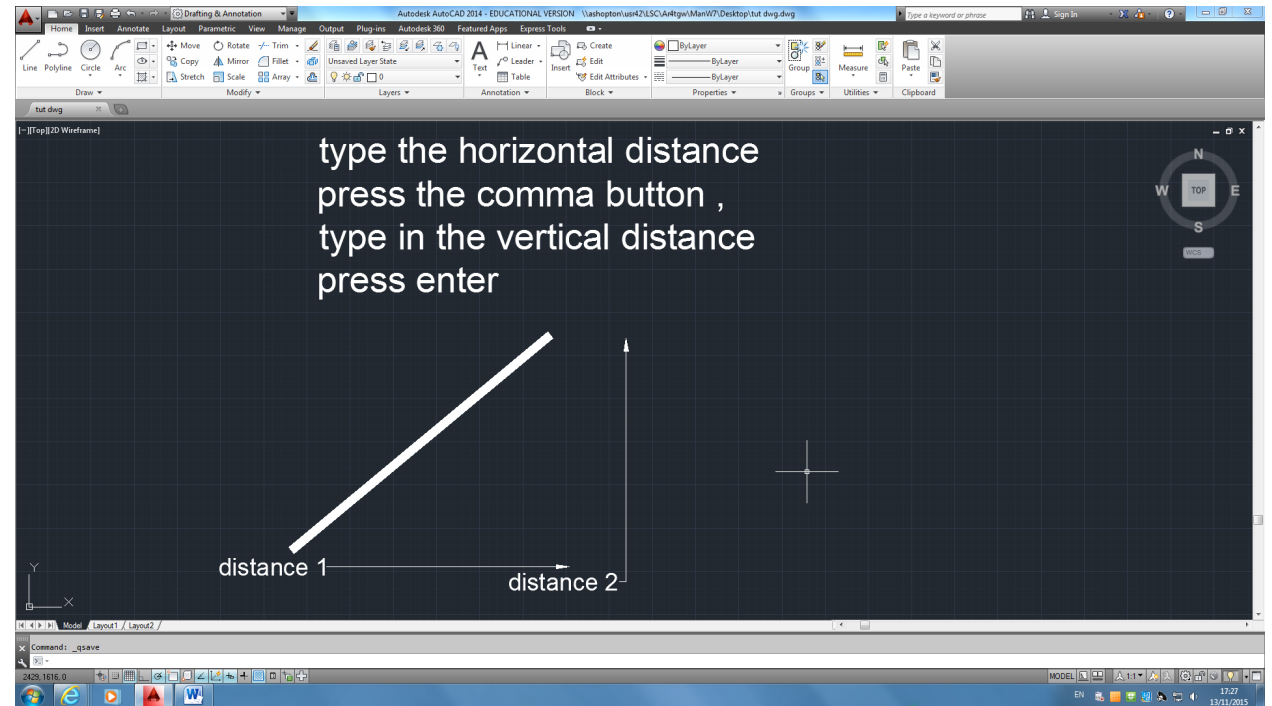
type the horizontal distance
press the comma button ,
type in the vertical distance
press enter

distance 1   distance 2

Scrolling Zooms In & Out
Click & Drag Pans (Move Left/Right, Up/Down)
Double-Click Zoom Extends (View All)   End Command
&
Show Menu
Select

This second technique is the one used to draw rectangles.

Select the **RECTANGLE TOOL.**

Choose the starting point then either click to choose the opposite corner or type in the co-ordinates of the next corner using the dynamic imput options as shown

OSNAP -- One of the tools to make your life easier in AutoCAD is OSnap – or Object snap. The icon for this is on the Status bar - a square with one corner highlighted. What happens is that whilst you are trying to draw AutoCAD can 'snap' to specific points on what you've already drawn eg the end of anther line; the intersection of 2 lines; the middle of a circle etc..

This can be switched on and off as with Dynamic Imput so make sure it's on. Now right click on the icon; select 'Settings' and you should have the menu on the right. Select (toggle on) all those I've selected then click OK.

Now to test how these work - draw a rectangle. Press Enter and the same command is repeated (this also works with all tools) Hold your cursor over the rectangle at different points and you will see little green 'highlights' – these are points to which you can automatically start the next element of your drawing. Practice using line and rectangle with the new OSnap settings.

'Apparent intersection' is a little tricky but is useful and is worth practicing. Eg to draw a line which ends directly in line with another line – choose the start point then when prompted to choose the second point 'hover' your cursor over the end of the line to be lined up wiith then move your cursor to where you roughly want it to be – you'll see a green dotted line which means AutoCAD is lining you new line up with the old. Try it. This works equally well with Rectangles and Circles as well as lines.
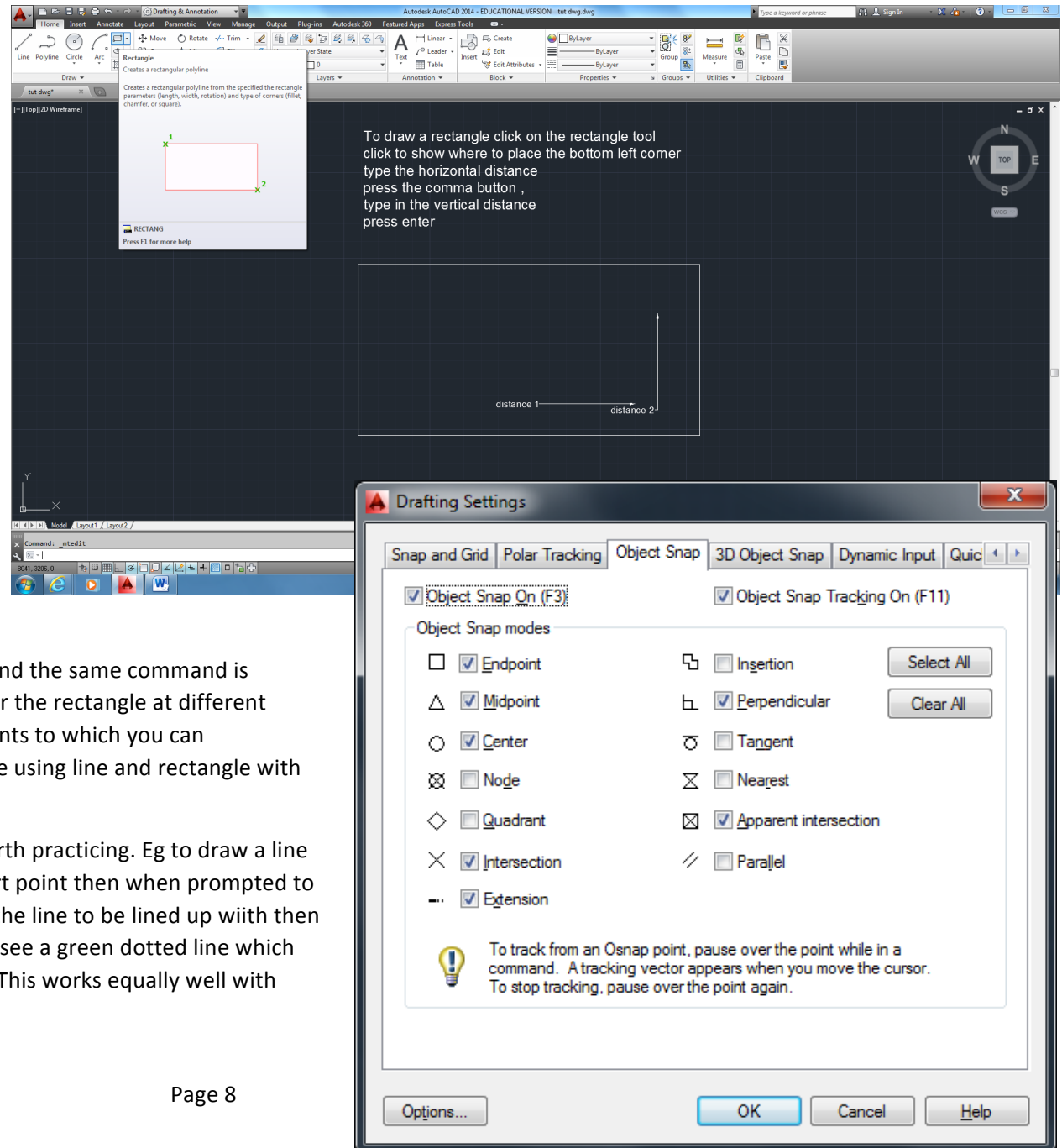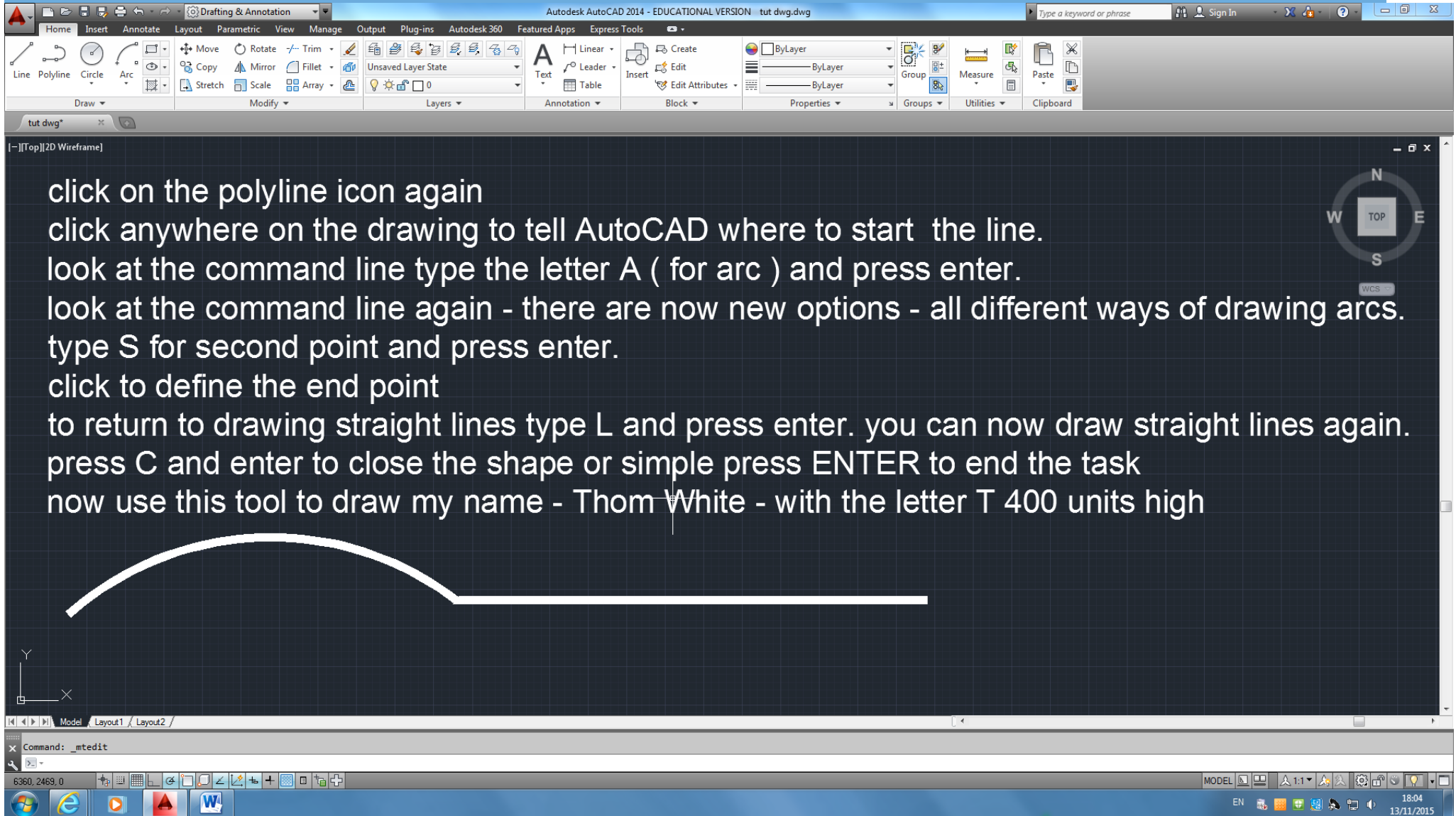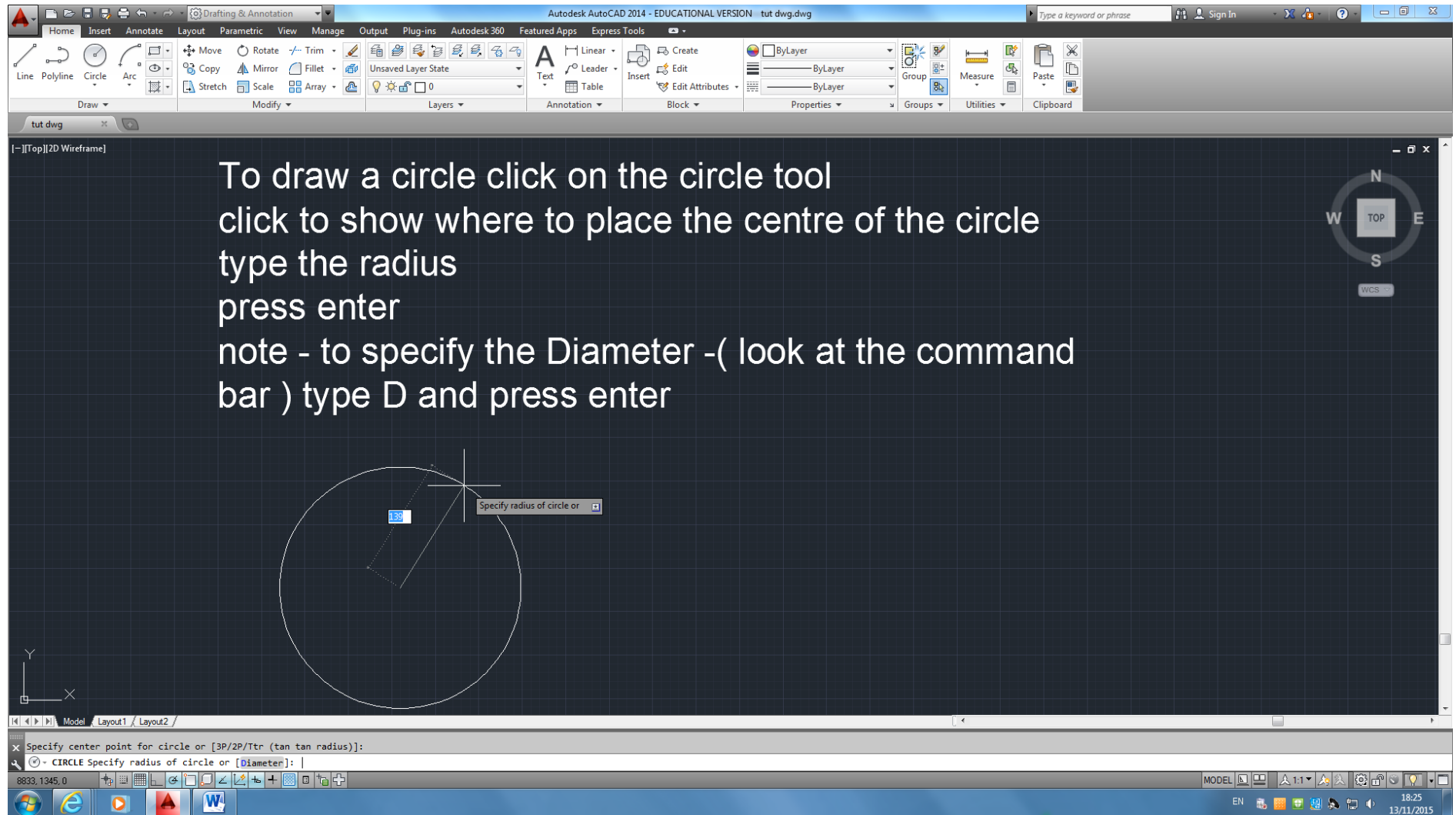
POLYLINES

Probably the tool you will use most.
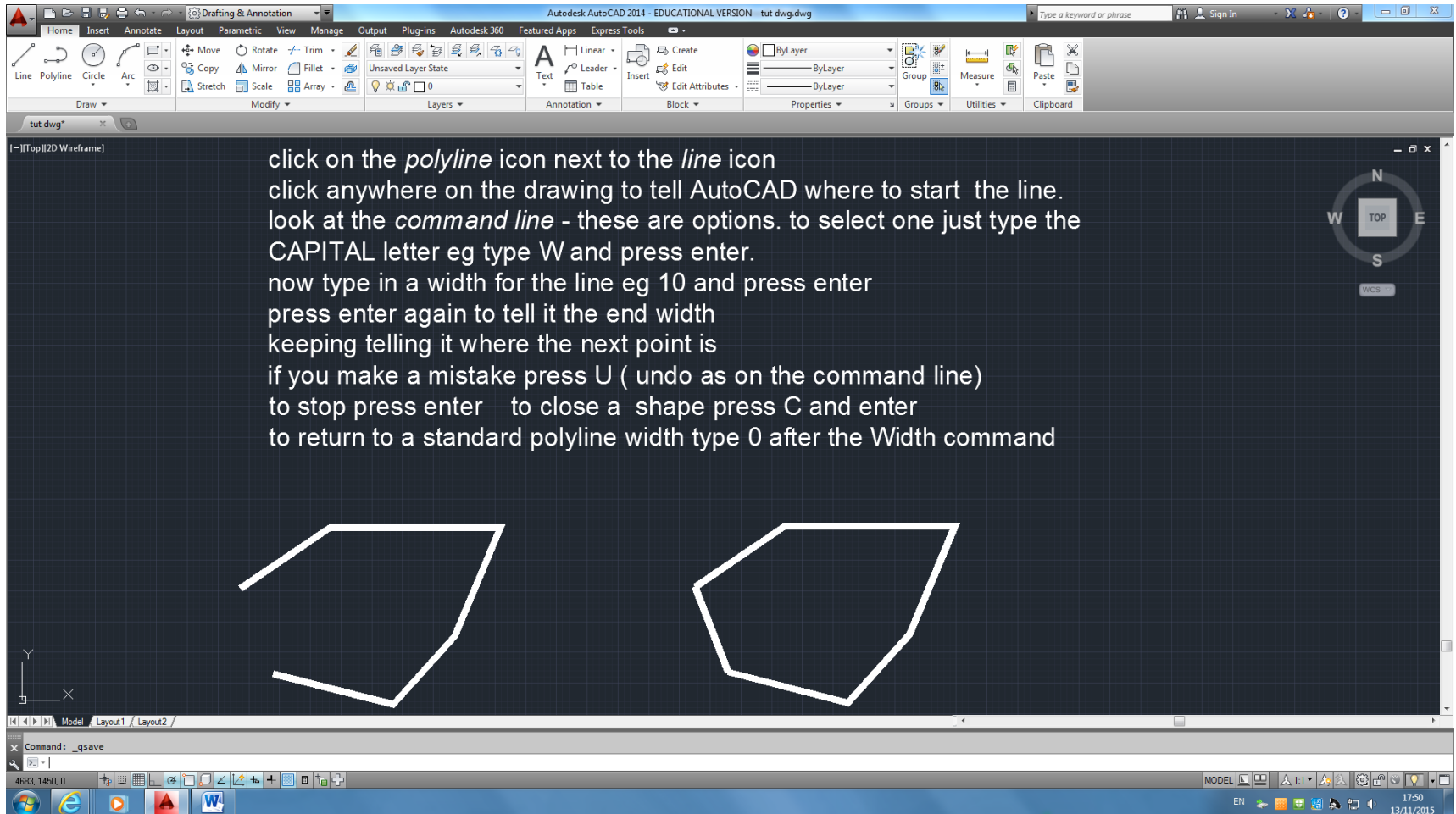
click on the polyline icon again

click anywhere on the drawing to tell AutoCAD where to start the line.

look at the command line type the letter A (for arc) and press enter.

look at the command line again - there are now new options - all different ways of drawing arcs.

type S for second point and press enter.

click to define the end point

to return to drawing straight lines type L and press enter. you can now draw straight lines again.

press C and enter to close the shape or simple press ENTER to end the task

now use this tool to draw my name - Thom White - with the letter T 400 units high

CIRCLES

To draw a circle click on the circle tool
click to show where to place the centre of the circle
type the radius
press enter
note - to specify the Diameter -(look at the command
bar) type D and press enter

Try drawing a couple of rectangles and using OSnap attach a few circles to them.

MODIFYING TOOLS
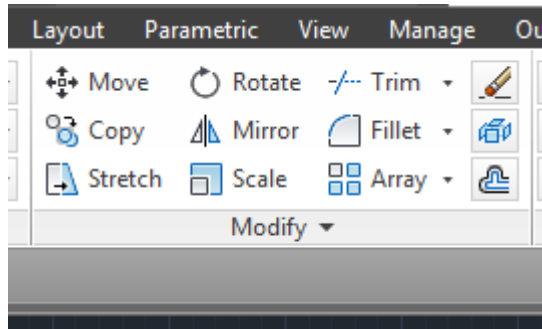
OFFSET

allows   you to create a copy of a line or object at a (chosen) fixed distance - on either side of the original

To offset click on the offset tool

The command line wants to know how far to offset "through"

type a distance press enter

click on the line or object you want to offset then
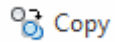
click on the side you want to offset

you can carry on offsetting either the original or different objects as many times as you like

press enter when you want to end the command

using the 'Multiple' option after specifying the distance allows you to keep clicking outside the original object until you have as many as you like
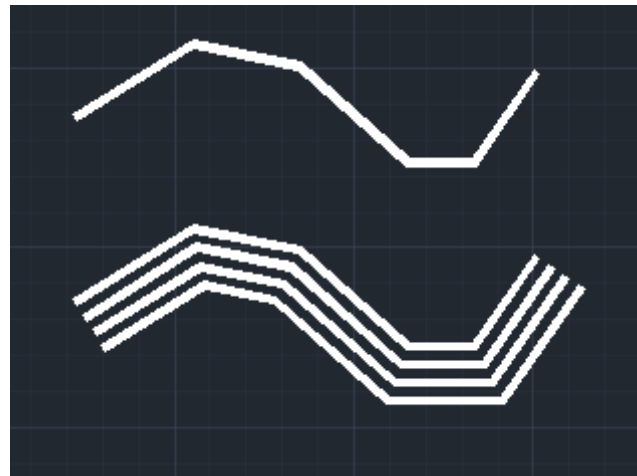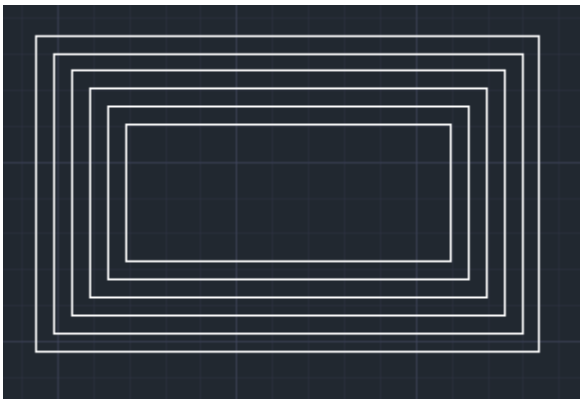
COPY   🔲 Copy

click on the COPY icon or type CO and press enter
select the object / line / text to copy by clicking or using the selection
rectangle/marquee - you can select as many as you like. if you select
something by mistake then simply type U and enter and it is deselected
press enter when you have them all
select a 'base point' - this is the point around which the abject will be   'pasted'
eg the centre of a circle or corner of a rectangle.
select the point at which you want to paste the copy – this is where OSnap   really is
useful!
this can be repeated until you press enter.
NOTE - you can also use CTRL C and CTRL V to copy and paste as in most   other
programmes.
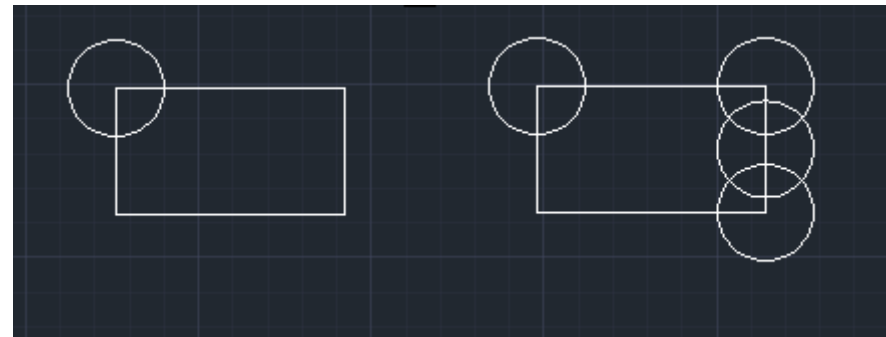
MOVE   ✛ Move

Similar process to copy
click on the MOVE icon or type M and press enter
select the object / line / text to move by clicking or using the selection
rectangle/marquee - you can select as many as you like - press enter to   confirm
your selection.
select the basepoint
select the next point
the command is then over
NOTE Once selected you can click and hold down the cursor and 'drag' the   object
to where you want it – less accurate but useful for quick moving of an object.
Don't click on any of the square, highlighted 'handles' as this will only move the handle.
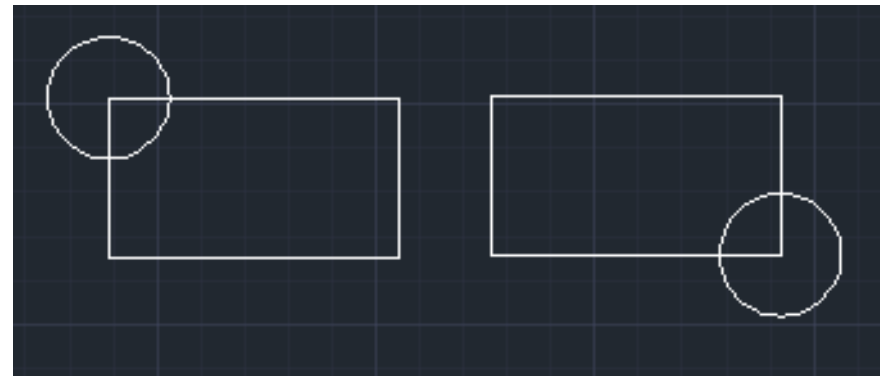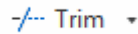
TRIM

-/-- Trim ▾

click on   the TRIM icon or type TR and press enter
select the object / line to select the line to which you want the other objects cut back to (
you can select as many as you like) press enter to confirm your selection.
select the objects or lines to be cut - again as many as you want
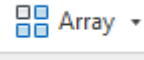press enter to end the command
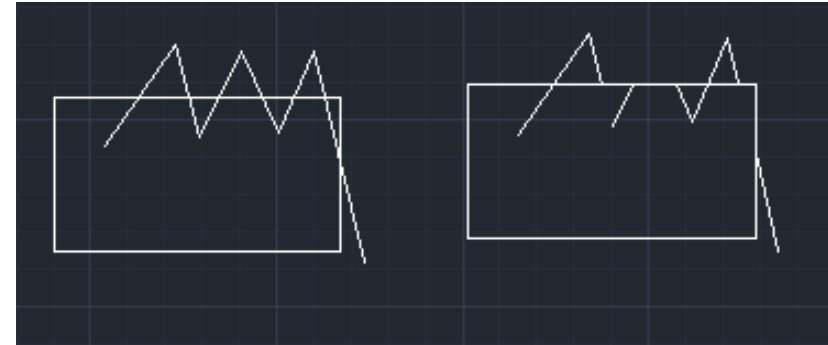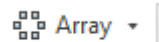
ARRAY – POLAR   Array ▾
Takes a copy of   an object and repeats in in an array of your choosing.
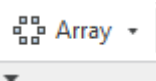There are 3 options - In this session we are looking at "POLAR ARRAY"
click on the small down arrow next to the ARRAY icon   Array ▾
a choice of 3 options appears –

click on POLAR ARRAY   Array ▾

select the object / line to be arrayed (you can select as many as you like)
press enter to confirm your selection.
Select the base point around which the selected object will be arrayed
The ribbon of tools at the top of your screen then changes - (see right)
there are a number of different options. The easiest is to change the number

of items you want arrayed - eg change the '6' to 12 and press enter
once you have finished making changes to the options click on "Close Array"
You need to play around with this tool in order to learn.
NOTE - An array behaves like a single object. If you want to alter any aspect of
it - eg Trimming you will need to "Explode"   it into its component
parts.
Follow the on screen instructions for explode.

SAVING YOUR WORK:
Having created your drawing and worked in it, it is very important to save it
and then continually save it as you work. It is also recommended that you
save the file to a variety of sources to ensure you have back ups.
Saving is simple- Ctrl S. Then choose a location to save the work . It is suggested for the purpose of these workshops that you create a "Autocad" Folder in your
university account hard drive and save all work in that, so you know where it is.