

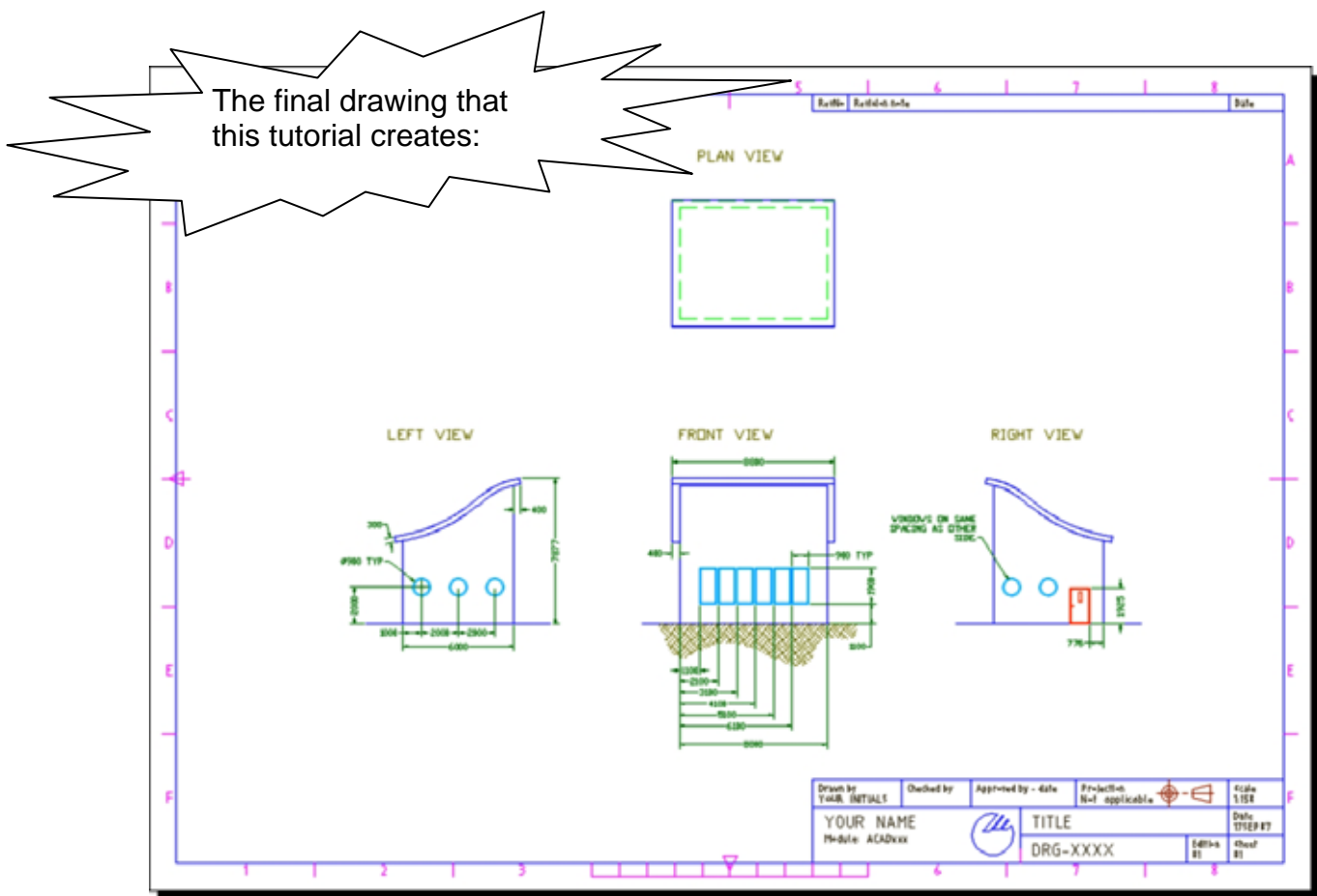
An introduction to 2D AutoCAD

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1 Introduction.

1.1 What is AutoCAD?

AutoCAD is a software package used for draughting and designing almost anything you wish. The software has evolved massively over the years and now offers various 'themed' versions catering for the specific requirements of different engineering disciplines, such as mechanical engineering, civil engineering and architecture.

Basically AutoCAD provides a 3D space (model space) in which you construct or draw your design. It provides drawing layout tools which enable you to prepare and plot conventional 2D drawings of your model. More recent versions also include advanced rendering tools, enabling you to create photo-realistic images of your design.

1.2 The interface.

Standard Windows type command buttons, pull down menus etc. are provided to use AutoCAD.

One strong feature of AutoCAD is the continued support of its **command line**. Any function or command you can give AutoCAD can be given by typing in the appropriate command on the screen. AutoCAD supports legacy commands, from previous versions, so learning to use AutoCAD through the command line means you should be able to use any subsequent or previous releases of the software without getting lost in the menu systems.

This manual will encourage you to use the command line but may also involve use of tool bars and pull down menus.

1.3 Saving your AutoCAD work.

You are responsible for the safe keeping of your work with AutoCAD. Work saved on your network U:\ drive is safe, but the drive is limited in storage capacity. You may wish to use other removable media such as a USB flash stick (pen drive).

Get into the habit of creating a folder for each project.

When you save work, AutoCAD will create a *.dwg file and a *.bak file of your work. The *.bak file is a backup copy of the *.dwg file.

1.4 About the 2D tutorial.

As an introduction to the basics of using AutoCAD in 2D you are presented with this practical tutorial.

You will be introduced to a method of creating a projection drawing of a building in 2D and then presenting that drawing on an A3 formatted sheet. Once you are comfortable with these basic tools you are encouraged to experiment yourself and then to move on to the [3D tutorial](#).

2 AutoCAD - Creating a new drawing.

2.1 Introduction.

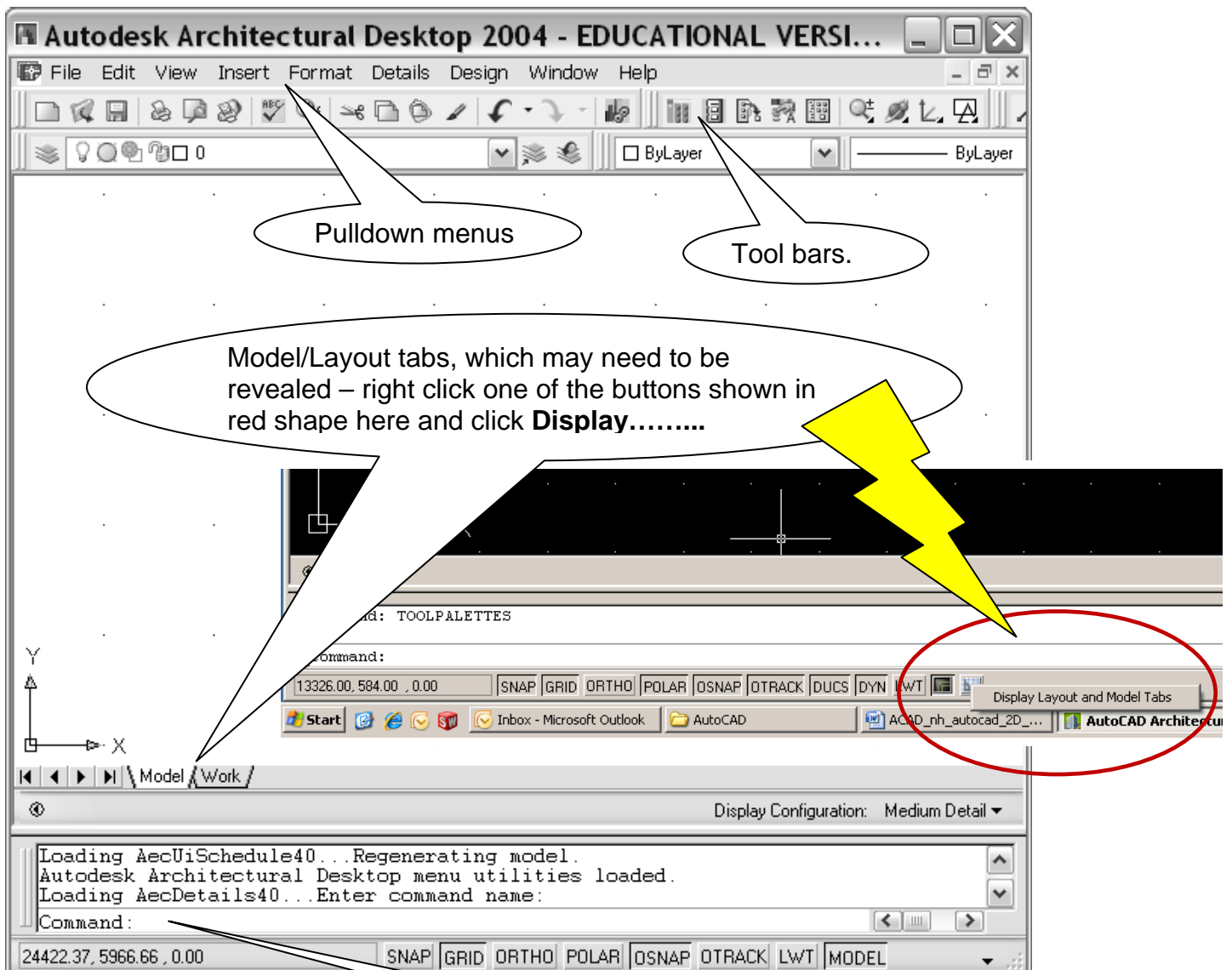
In this section you will use AutoCAD to create and set up a new drawing file. You will also become familiar with the toolbars, menus and command line.

2.2 Activating AutoCAD.

Only certain rooms contain computers with access to AutoCAD. You will find a shortcut installed under the University Software list, under A for AutoDesk. Click the appropriate shortcut and wait for the program to load. Video clips will require use of headphones.

2.3 The AutoCAD display screen.

When AutoCAD has successfully loaded you should have the window similar to that shown below:



2.4 Setting up a new drawing.

New commands used in this section:

close, **new**, **saveas**, **units**, **limits**, **grid**, **snap**, and **save**.

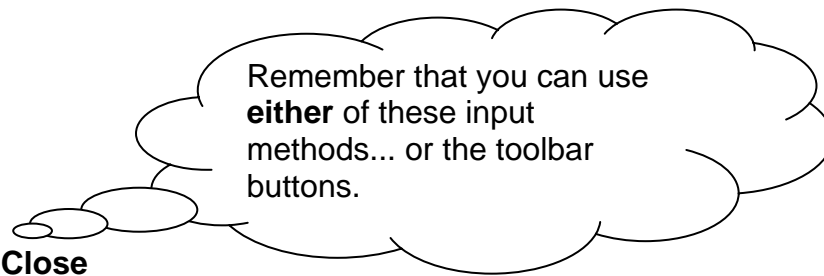
2.4.1 Creating a new drawing file.

To make a space to store your work, create a new folder called **acad_examples**

This folder could be on the U:\ drive or any other suitable location of your choice.

To start working with a fresh drawing, close the drawing file that AutoCAD opened with:

Command: **close**
Pull down menu: **File > Close**



Create a new drawing using the acad.dwt template:

Command: **new** ↵ Enter (Only if command line is available)
Pull down menu: **File > New...**

Select the **acadiso.dwt** template (which is metric) and click **Open**.

Save this drawing to your **acad_examples** folder calling it **template_mm**.

Command: **saveas** ↵ Enter
Pull down menu: **File > Save As...**

You now have a new drawing named and saved. Next job is drawing set up.

2.4.2 Drawing units setup.

The AutoCAD drawing units can be defined by you.

Command: **units** ↵ Enter
Pull down menu: **Format > Drawing Setup...**

You should now see the **Drawing Setup** window. As we will be drawing buildings in the region of thousands of millimeters in length, set the units to **Millimeters**, and the precisions of **Length**, **Area** and **Volume** to **0**.

Click **OK** to enter these changes and **OK** to confirm scale changes, if it appears.

Video demo: Units, limits and grid >

http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/units_limits_grid.wmv

2.4.3 Drawing limits setup.

AutoCAD likes to know the overall size limits of the model. For this example a square area of 10000 mm by 10000 mm will be appropriate. To set this up:

Command: **limits** ↵ Enter
Pull down menu: **Format > Drawing Limits** (then type in command line)

In the command line observe the text in brackets, **<0,0>**. This is the default value which will be entered if you press **return** or... **right mouse button** click then click on **Enter**.

This is acceptable, so enter. Next type in the required x and y co-ordinates of the top right corner of the required drawing limit:

10000,10000 and enter.

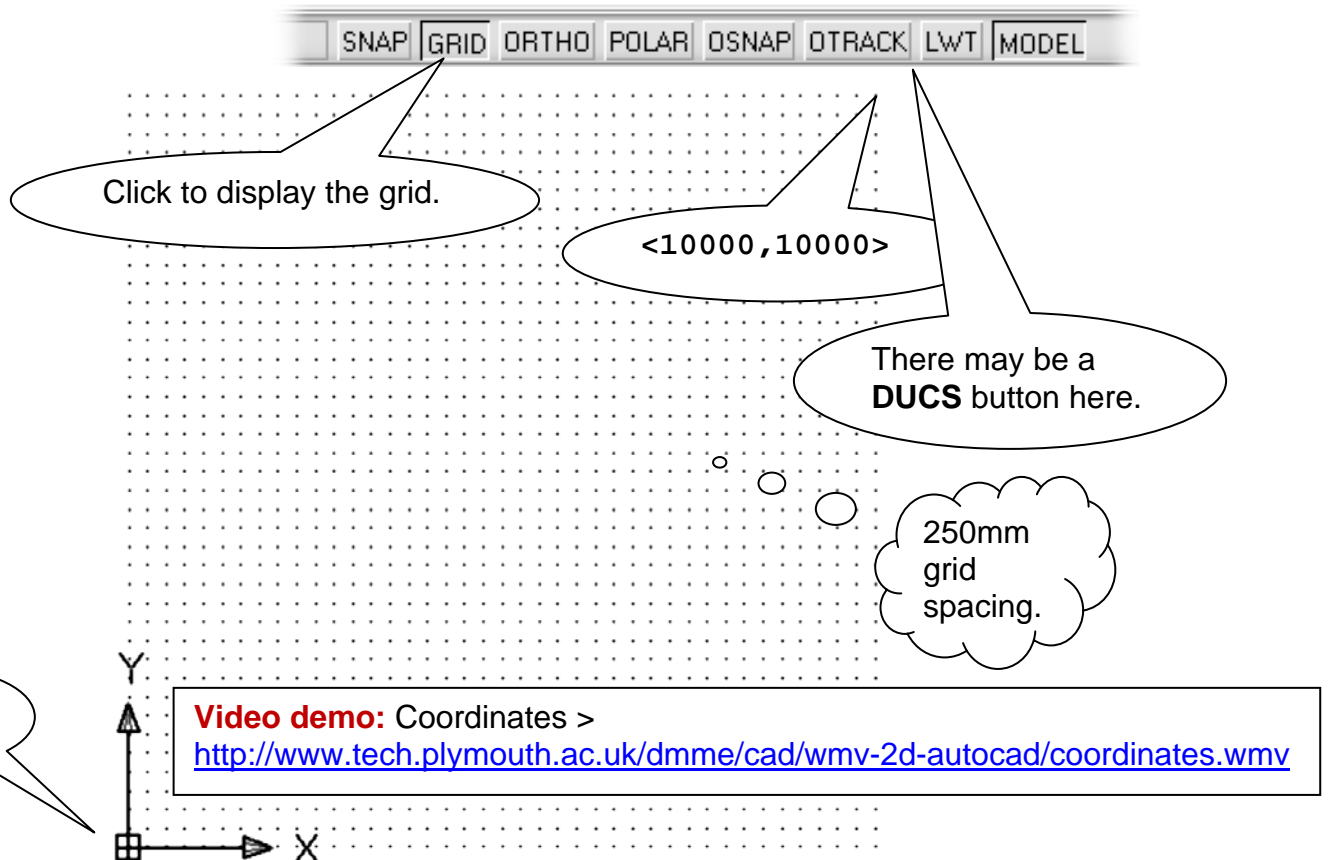
2.4.4 Drawing grid and snap setup.

To help with draughting, a grid can be displayed over the area defined by the limits of the drawing. To define the spacing of the grid:

Command: **grid** ↵ Enter
Pull down menu: **Format > Draughting Settings...**

For this project, let's use a grid spacing of 250 mm. Enter these values, either on the command line or in the Draughting Settings window in the Grid value boxes.

To display the grid, or to activate snap, use the buttons along the bottom of the screen or the command line **grid** command.



To define the snap settings:

Command: **snap** ↵ Enter
Pull down menu: **Format > Draughting Settings...**

Video demo: Snap >
<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/snap.wmv>

For this project make the snap spacing 250 mm, the same as the grid. Enter these values, either on the command line or in the Draughting Settings window in the Snap value boxes.

To zoom the display to view the extremes (extents) of the drawing limits:

Toolbar button:



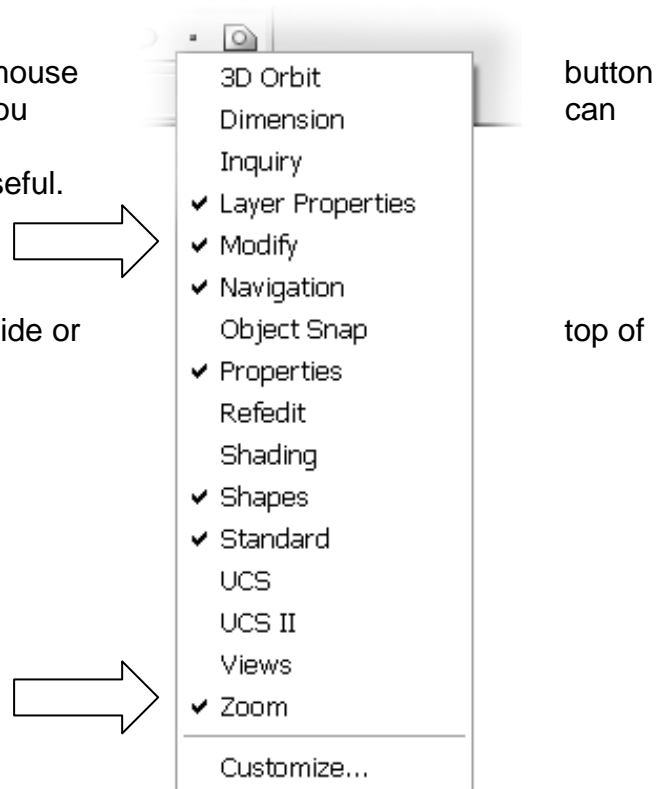
Command: **z** <Return> to run the zoom command, then enter **a** <Return> for zooming to all objects in the drawing.

2.5 Toolbar setup.

Move the pointer over any toolbar and right hand mouse click and select **ADT >** . From this selection box you define which toolbars to display on the screen. Experience has shown that these selections are useful.

Particularly Modify & Zoom.....

You can drag the resulting button windows to the side or the display.



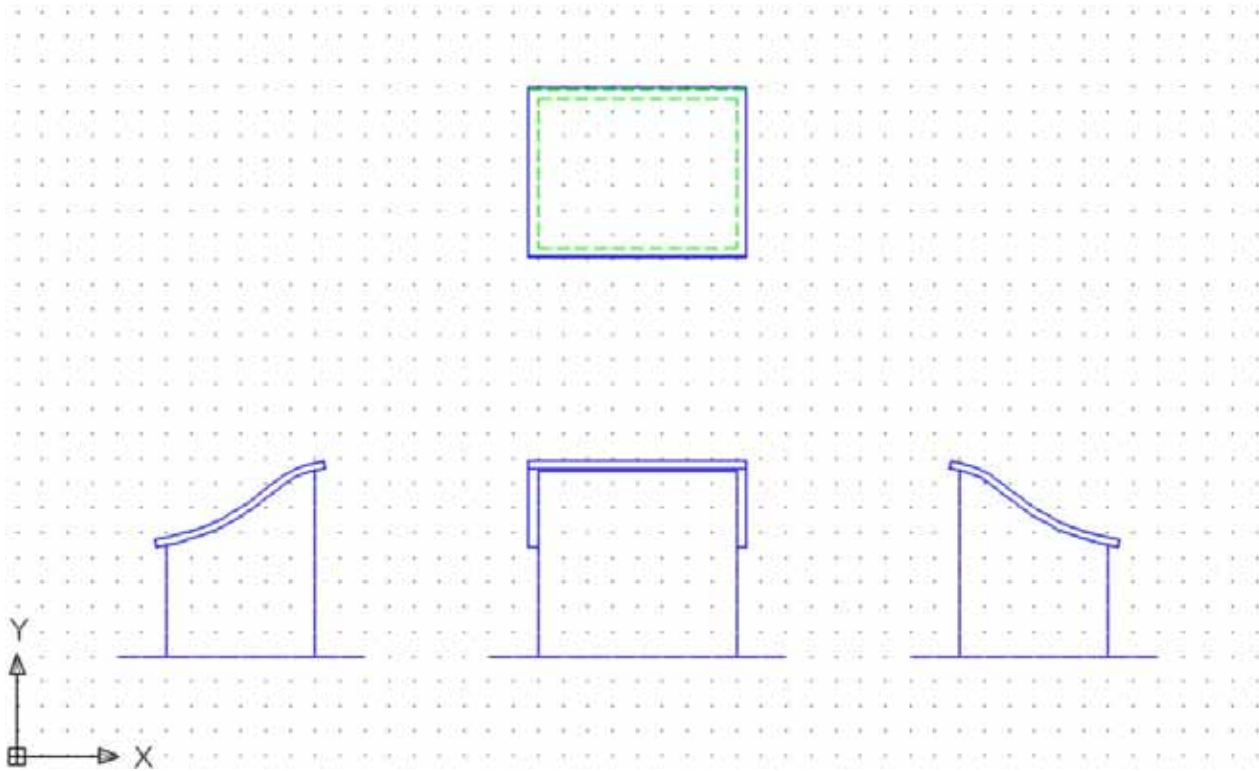
Save the file:

Pull down menu: **File > Save**
Command: **save** ↵ Enter

3 AutoCAD – Lines and layers.

3.1 Introduction.

This section will introduce you to some basic fundamental draughting techniques with AutoCAD by leading you through a practical drawing tutorial. You will work with **entities** and **layers** to create the 2D 3rd angle orthographic projection drawing show below.



3.2 Pre-requisites.

You should have completed section: 2 AutoCAD - Creating a new drawing.

...and have the **template_mm.dwg** file ready to open.

3.3 Setting up the drawing.

New commands used in this section: the **F9** key and **z** (for zoom).

Start AutoCAD and open the **template_mm.dwg** file.

Save the file.

Command: **saveas** ↵ Enter **sports_hall.dwg**

Check the units set up

Command: **units** ↵ Enter **mm, precision 0**

We will assume that for this drawing a rectangular drawing area of 50m by 30m is required. This is set using the drawing limits.

Set the limits.

Command: **limits** ↵ Enter
Accept default, <0,0> ↵ Enter
50000, 30000 ↵ Enter

Set the grid.

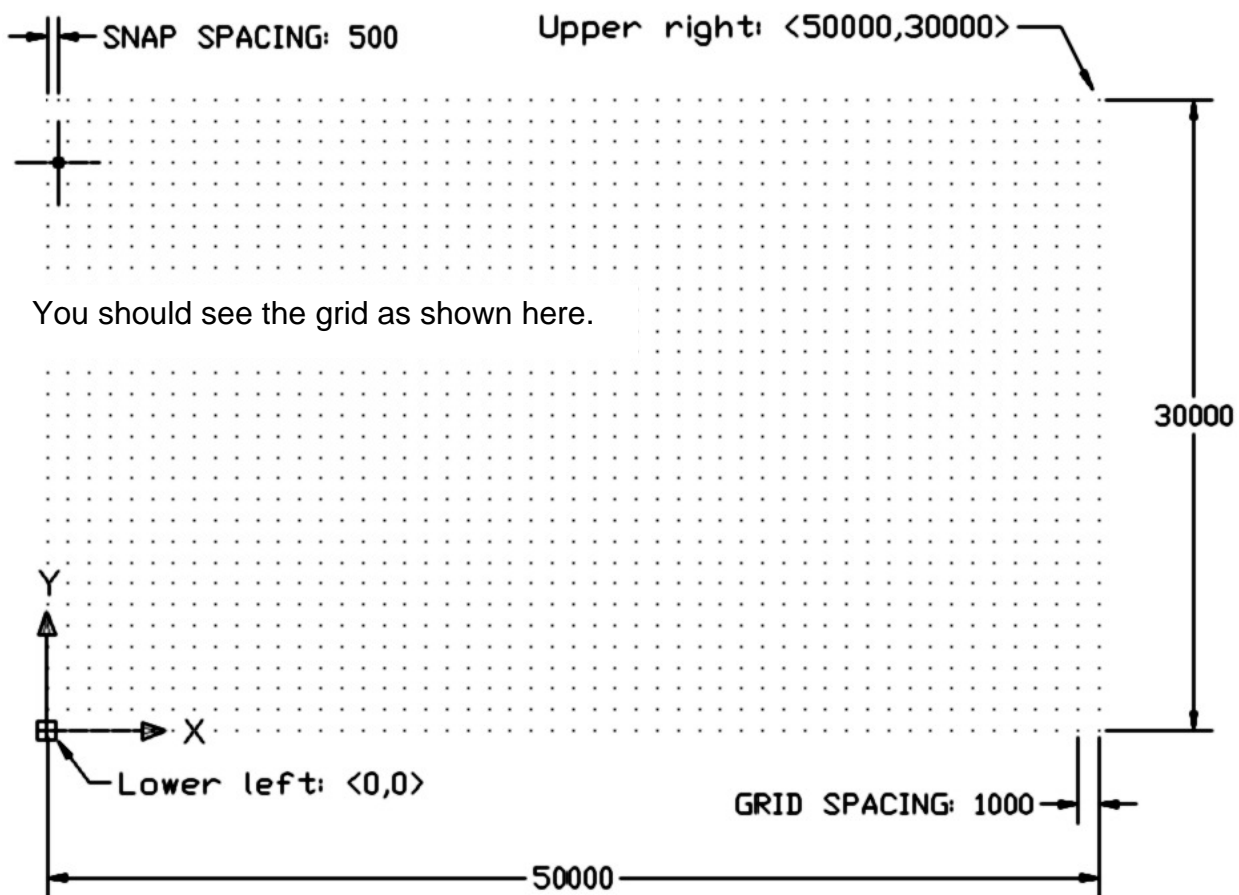
Command: **grid** ↵ Enter
to a spacing of: **1000** ↵ Enter

Set the snap.

Command: **snap** ↵ Enter
to a spacing of: **500** ↵ Enter

Zoom to all objects in drawing:

Command: **z** ↵ Enter
all objects in the drawing **a** ↵ Enter



You should see the grid as shown here.

 The **F9** key:

Move the pointer around the display. Does it appear to jump around in intervals of 500 units? (Hint: Look in the bottom left hand corner to see the display of the pointers current coordinates.)

If it does, your snap function is ON. At any moment in time when you are drawing you can toggle the snap ON or OFF using the **F9** key. Try it. Move the pointer around whilst repeatedly pressing the F9 key.

To stop the line command, simply use the **Esc** key. The **Esc** key will always cancel or finish any running command. This is now a completed line. Relative and absolute coordinates have their use, particularly relative coordinates, when moving objects, etc. There are other ways of specifying positions in AutoCAD. Read on...

3.4.2 Using snap.

New commands used in this section: the **F7** key, **coords**.

We will draw some more lines using the grid and **snap** as guides for positioning the pointer along with absolute coordinates. We will also use the coordinate display in the bottom left of the screen to confirm the position of the pointer.

Press **Esc** just to cancel any commands still running.

Ensure grid is on:	F7 key toggles the grid display.
Zoom to all objects:	Command: z ↵ Enter
	a ↵ Enter

Move pointer around the screen to see if snap is ON or OFF. Use **F9** key to ensure snap is ON. Observe the coordinate display in the bottom left of the screen. The pointer should be jumping in steps of 500 units.

Start drawing the next line:

Command:	line	↵ Enter
Specify these coordinates:	21000,12000	↵ Enter

Now move pointer around, observing the coordinate display.



It should be displaying the absolute x & y coordinates of the pointer.

Just for information:

The **coords** command controls how coordinates are updated on the status line. It can be controlled by setting its value as specified below:

- 0** Coordinate display is updated as you specify points with the pointing device.
- 1** Display of absolute coordinates is updated continuously.
- 2** Display of absolute coordinates is updated continuously, and distance and angle from last point are displayed when a distance or angle is requested.


Back to drawing the line. After the first (start) point, move the pointer around and snap to and click on the position:

21000, 20500

Esc or (right mouse button **Enter**) to finish the line.

Command: **erase** ↵ Enter

Select the splines you have just drawn and ↵ **Enter** (or right mouse button) to the execute erase.

 Whenever prompted to select objects, AutoCAD will expect you to either click on entities in the drawing or enter an extra command on the command line, such as **all**, which selects all objects in the drawing. Once selected, you have to press **enter** to confirm the selection. For more details refer to **select** in the **Help** menu.

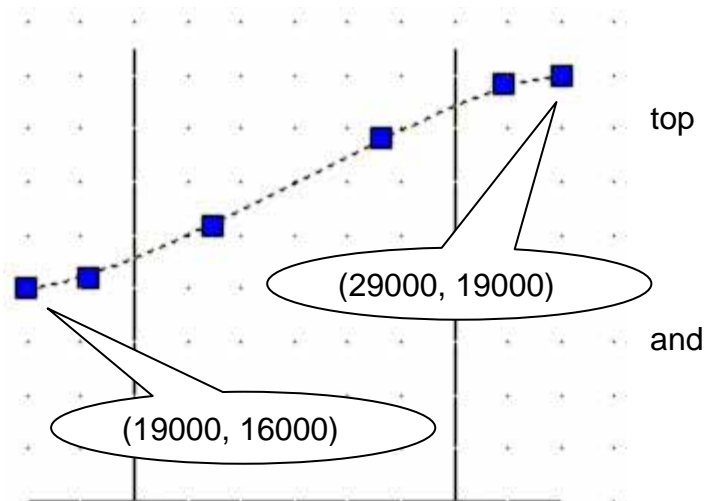
Now draw the proper roof spline, like this:

Using absolute coordinates, make sure the end is at (x, y):

29000, 19000

Click, with snap OFF, to define four intermediate control points as shown here, finally make the lower end point at absolute coordinate (x, y):

19000, 16000



Remember to toggle the snap ON/OFF, as required, using the F9 key.

3.4.4 Trimming.

New commands used in this section: **trim**

Video demo: Trimming & extending >

http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/trim_extend.wmv

Now we will trim the two vertical wall lines back to the roof spline using the **trim** command.

The **trim** command requires **two** groups of selections. The **first** group defines the entities you wish to use to trim up to (the cutting edges), the **second** is the portion of each line you actually wish to be removed, or trimmed.

So for this example we will first select the roof spline as the cutting edge, ↵ Enter that selection, then click on the two upper parts of the vertical lines representing the walls to trim them.

So... command: **trim** ↵ Enter

Click on the roof spline: ↵ Enter or right button

Now click on part of each wall line you wish to remove... and it should trim for you.
Esc to finish.

 Note that often the right mouse button performs the same function as the ↵ **Return** or ↵ **Enter** key.

3.4.5 Offsetting lines.

New commands used in this section: **offset**, **extend**.

The **offset** command is a very useful one to know. It allows us to effectively copy a line and project or translate it a specified distance from the original.

Video demo: Offsetting >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/offset.wmv>

This sports hall project calls for the roof to be exactly 400mm proud of the walls. To draw this we can **offset** the walls by 400mm exactly, and then use these temporary lines as cutting edges to trim the roof spline back to. We erase them once the roof has been trimmed.

So, to offset each wall by 400mm:

Command: **offset** ↵ Enter

The first thing needed is the distance to offset, so enter the value of 400:

400 ↵ Enter

Now simply click on the wall you wish to offset and then click in the area on the side of it to offset to. The new line should appear.

The offset command continues to run, now offsetting anything you select until you press Esc or the right mouse button. Offset the other wall as well and then **Esc**.

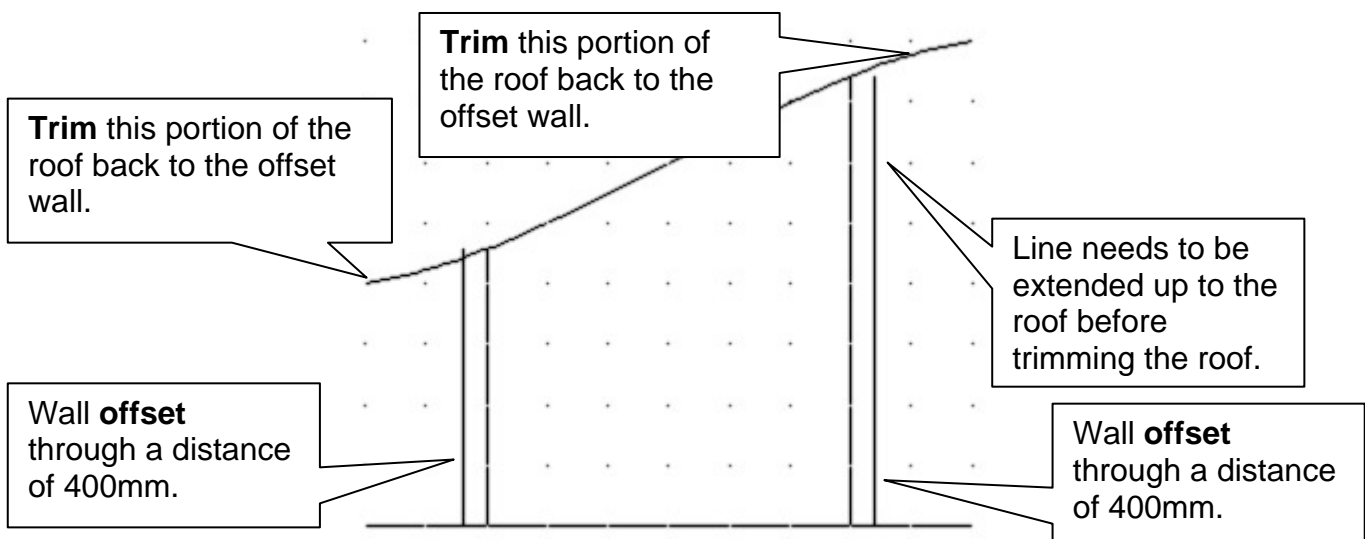


You can offset circles, rectangles, polylines, splines, etc.

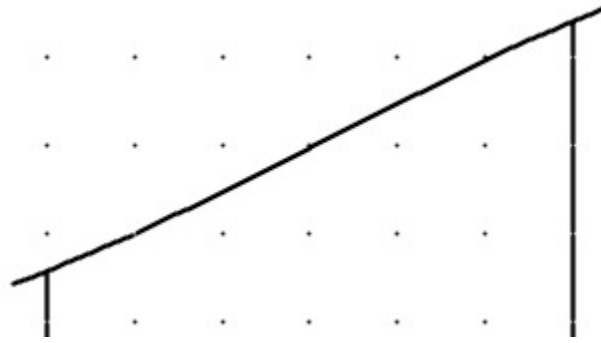
Extending lines:

In order to trim the roof back to these new temporary lines, they have to either touch or intersect the line to be trimmed. To achieve this, the right hand wall line will need to be extended up to the roof. To do this we use the **extend** command, which works just like the trim command. Run the **extend** command and experiment! Remember to select first the line to extend up to, ↵ Enter, then select the lines to extend.

Now use these two new offset lines as cutting edges to **trim** the roof to. Then **erase** them.



You should now have a neat roof, with an overhang of exactly 400mm, like this.



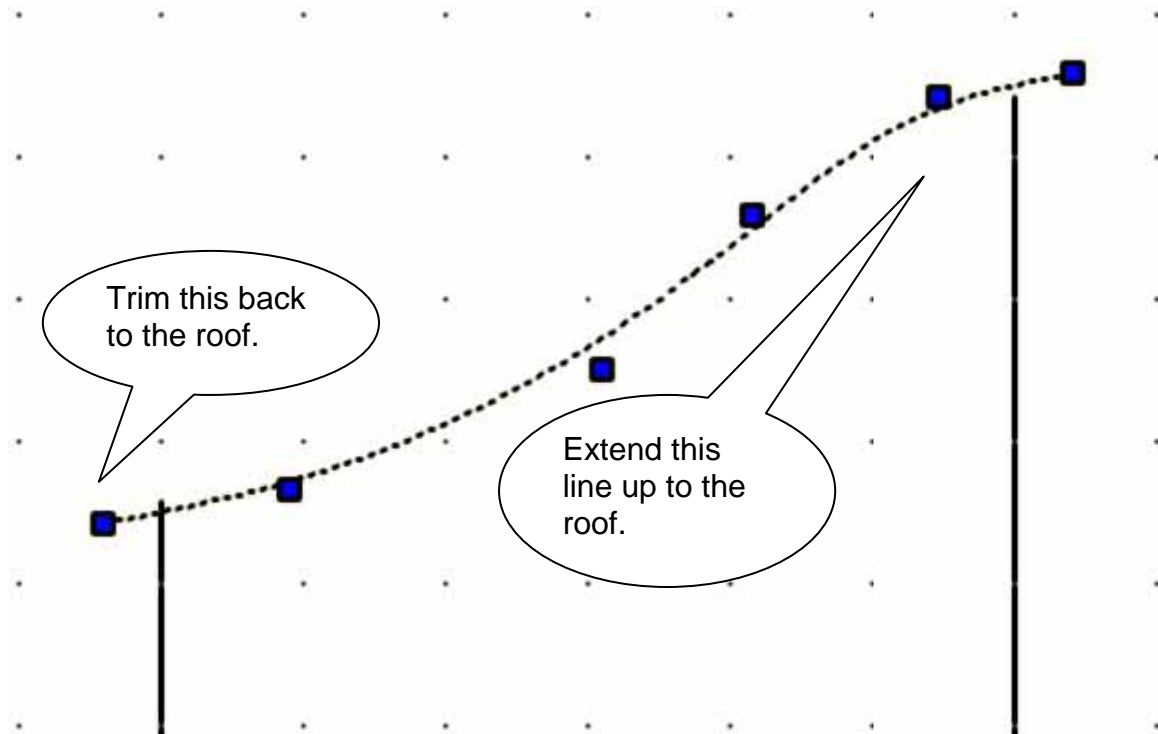
3.4.5.1 Grabs.

All entities you create in AutoCAD have grabs. These are points on geometry which can be easily clicked on and dragged to relocate.

Press **Esc** just to cancel any previous command and click on the roof spline to reveal its grabs..

Give the roof a more interesting shape by moving the control point grabs, the ones between the ends, around, not the end points.

You will need to practice using [trim](#) and extend on the walls to make them match the roof line.



3.4.6 Using object snap.

New commands used in this section: **osnap**, the **F3** key.

So far, when drawing, the pointer position has been controlled using:

- **coordinates**, actually entered in on the command line, both relative and absolute x and y values.
- **snap**, which forces the pointer to jump or snap to preset intervals. (F9 key, remember?)

Another very useful facility for placement of the pointer is called **object snap**. Object snap allows you to snap with total mathematical accuracy to particular features of an object, such as the end point or midpoint of a line.

Video demo: Object snap >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/osnap.wmv>

Object snap can be toggled ON/OFF any time you like using the **F3** key.

Object snap can be set up by running the **osnap** command. Have a look:

Command: **osnap** ↵ Enter

The resulting window is fairly self explanatory. Select all or any of the options, any time you like when drawing.

Now we will develop the roof detail. The roof is 300mm thick, and will be drawn by offsetting the existing roof spline by 300mm. These two splines will then be joined at their ends with straight lines, using object snap.

First... offset the roof spline by 300mm.

Command: **offset** ↵ Enter

The first thing needed is the distance to offset, so enter the value of 300:

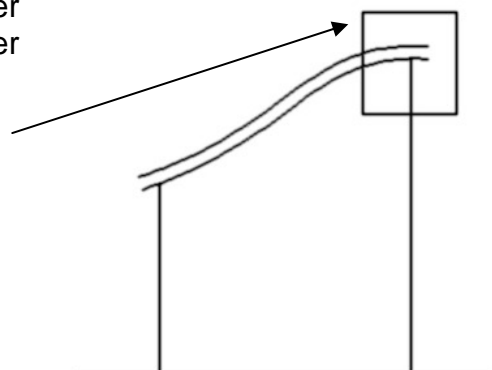
300 ↵ Enter

Now simply click on the roof spline and then click somewhere above it to perform the offset. The new line should appear.

Now is the time to use object snap. We will fill in the end sections of the roof. Firstly zoom in to the area we will be working on by zooming to a window:

Command: **z** ↵ Enter
w ↵ Enter

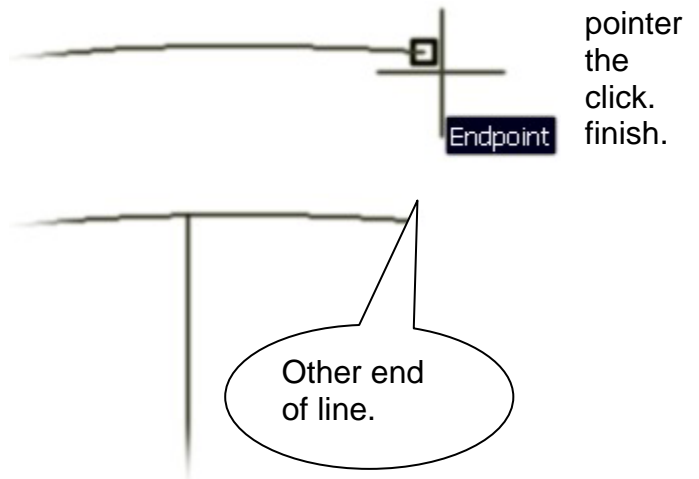
Left mouse click to start dragging a window over the area shown here, left mouse click again to zoom.



Now use **F3** to ensure object snap is on. Then start drawing the line.

Command: **line** ↵ Enter

To define the start point of the line move the pointer over the end of one of the splines and look for **Endpoint** flag to be raised by object snap, then click. Repeat this for the other end of the line. **Esc** to finish.



Draw a similar line for the other side of the roof.

Save the work.

3.4.7 Layers.

New commands used in this section: **la** (layer)

We now have a simple outline of one of the side views of the sports hall. We will use this view as a starting point for creating the other orthographic projection views.

We will create construction lines to help us. We will also learn about use of layers to help control the viewing of our work.

As soon as we start to create construction lines the drawing starts to become messy and hard to read. Layers are, in a way, like sheets of tracing paper. They can actually be turned off so that they cannot be seen. (They can also be locked so that they can be seen but not changed.)

Every type of object should really have its own layer. For example, construction lines would have their own layer, which can be turned on or off as required.

When you create a line, it will be placed on the layer that is current. This line will, by default, assume the **linetype** and **colour** set by the layer. This is the most efficient way of controlling what a line looks like, by the properties of the layer it is on.

Video demo: Layers >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/layers.wmv>

3.4.7.1 Creating a new layer.

To create a new layer, first open up the layer manager window:

Command: **la** ↵ Enter

From the Layer Manager toolbar click on the **New Layer** button:

Enter the new layer name:

construction_lines

Double click on the colour square to select a **colour**, perhaps a pale green.

In the **Linetype** column double click on the linetype description (it may currently be 'Continuous').

If you cannot see the **PHANTOM2** line type click on the Load button in the Select Linetype window. In the resulting **Load or Reselect Linetypes** window scroll down and select the **PHANTOM2** type. Click on **OK**. This loads the line pattern in to the drawing.

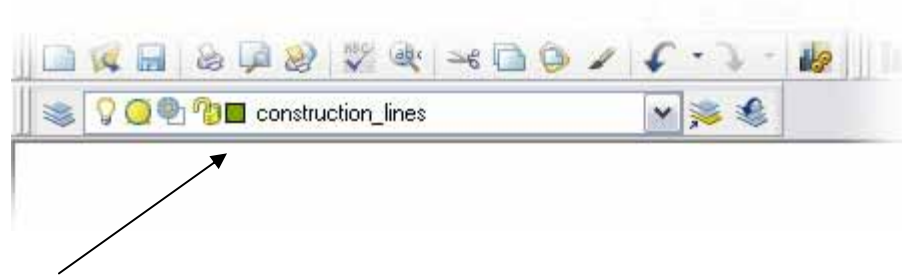
Now select the new linetype for this layer by clicking on **PHANTOM2** in the **Select Linetype** window and click on **OK**.

Note the addition of the new layer in the **Layer Manager** window.

Click on **OK** again to return to drawing.

Before we start creating construction lines, make the **construction_lines** layer current:

In the Layer Properties tool bar, pull down the selection menu and click on the **construction_lines** name. This should be the current 'live' layer.



pull
menu

layer
make it

It should show in the window, indicating that it is the current layer. Anything you draw now will go on to that layer and adopt that layer's properties, such as colour and linetype.

3.4.7.2 Turning a layer off.

In the Layer Properties tool bar, pull down the selection menu and click on the **light bulb** next to the layer you wish to turn off. Click in the display screen somewhere, and the layer will be turned off. If that layer is the current layer you will be told so. If you find you can't see lines you draw at any stage, just check to make sure that the current layer is on!

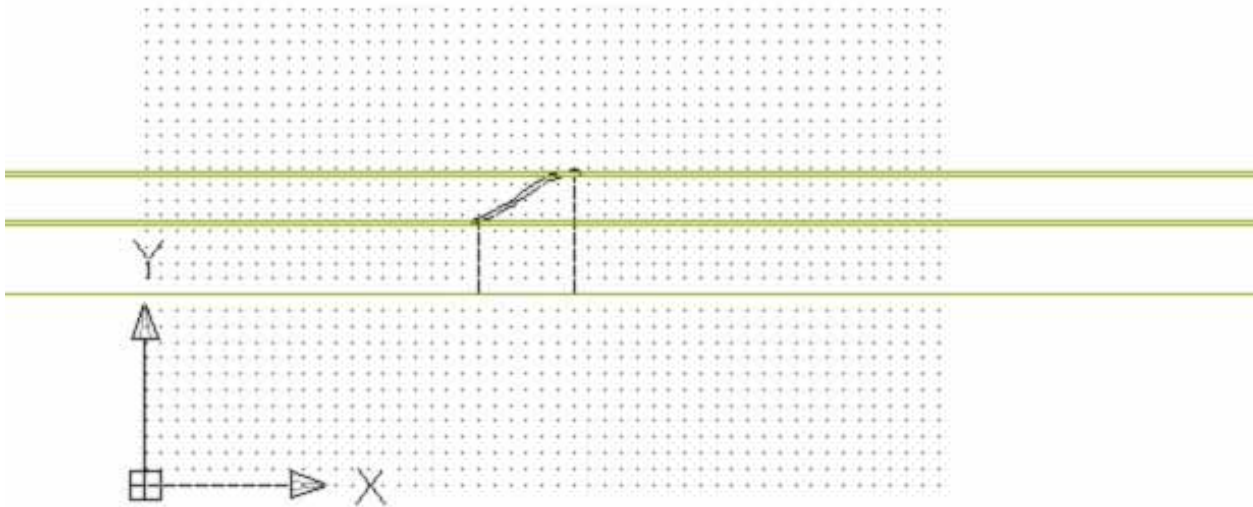
3.4.8 Construction lines and linetype scale.

New commands used in this section: **xline** and **ltscale**


We will now create some construction lines. Construction lines are the same as lines but with infinite length. We use them for projecting locations, to help with building up a drawing. They are created using the **xline** command. Amongst other options, you can create horizontal and vertical lines.

Command: **xline** ↵ Enter
h ↵ Enter

This sets us up for drawing horizontal construction lines. Using object snap, check it's on using F3, and click on roof and ground endpoints to create these lines:



Note the appearance of the lines. They should appear according to how the layer was set up, i.e. in this case be the 'one long and two short' dash PHANTOM2 type linestyle.

 AutoCAD tends to display linetype patterns by scaling them relative to a drawing unit. A solid continuous line will be displayed if the scale of the linetype is too small or too big to be clearly shown. For example, if your drawing units are in mm, the linetype pattern will repeat every couple of mm. If however your drawing units are meters, the linetype pattern will repeat every couple of meters. Also check out the **Scale** tab in the **Format > Drawing Setup...** window, this can affect the ltscale as well.

You can change the scale that all lines are displayed within the drawing using the **LTSCALE** command, which means **Line Type Scale**. The command line is not case sensitive, so either upper or lower case characters can be used. The default value of the LTSCALE is usually 1. Try a linetype scale value of 1000:

Command: **LTSCALE** ↵ Enter
1000 ↵ Enter

Observe how the line appearance may change. Change the value of ltscale again if you like, to suit your needs. This value affects the linetypes of all lines drawn, it is said to be applied 'globally'.

Video demo: Linetype scale >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/ltscale.wmv>

Now we will create the FRONT view of the sports hall.

We will end up drawing the front view using the intersection of construction lines as object snap references.

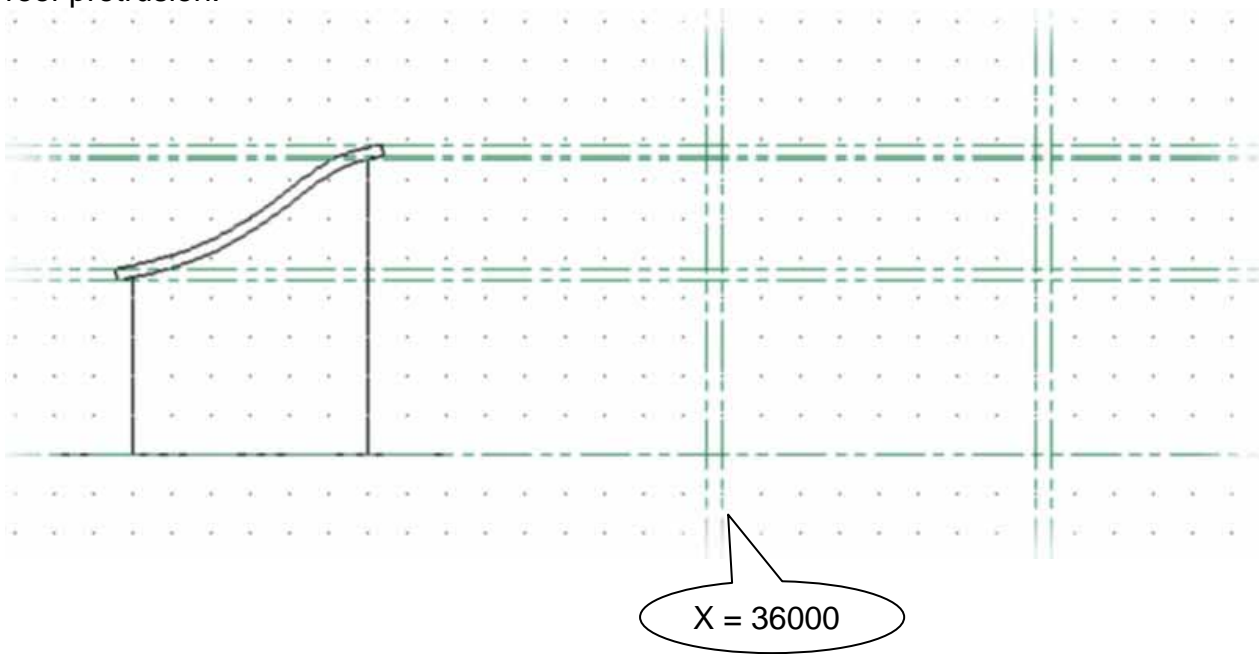
Firstly, use vertical construction lines to define the size and position of the new view. The building incidentally is 8000mm wide.

Command: **xline** ↵ Enter
v ↵ Enter

Use snap, referring to the coordinate display, position the vertical construction line anywhere where $x=36000$.

Esc to cancel, and offset this line 8000mm to the right.

Then **offset** each of these new lines 400mm outside the building to act as guides for the 400mm roof protrusion.



Using these new construction lines, and object snap, we can now start drawing the FRONT view.

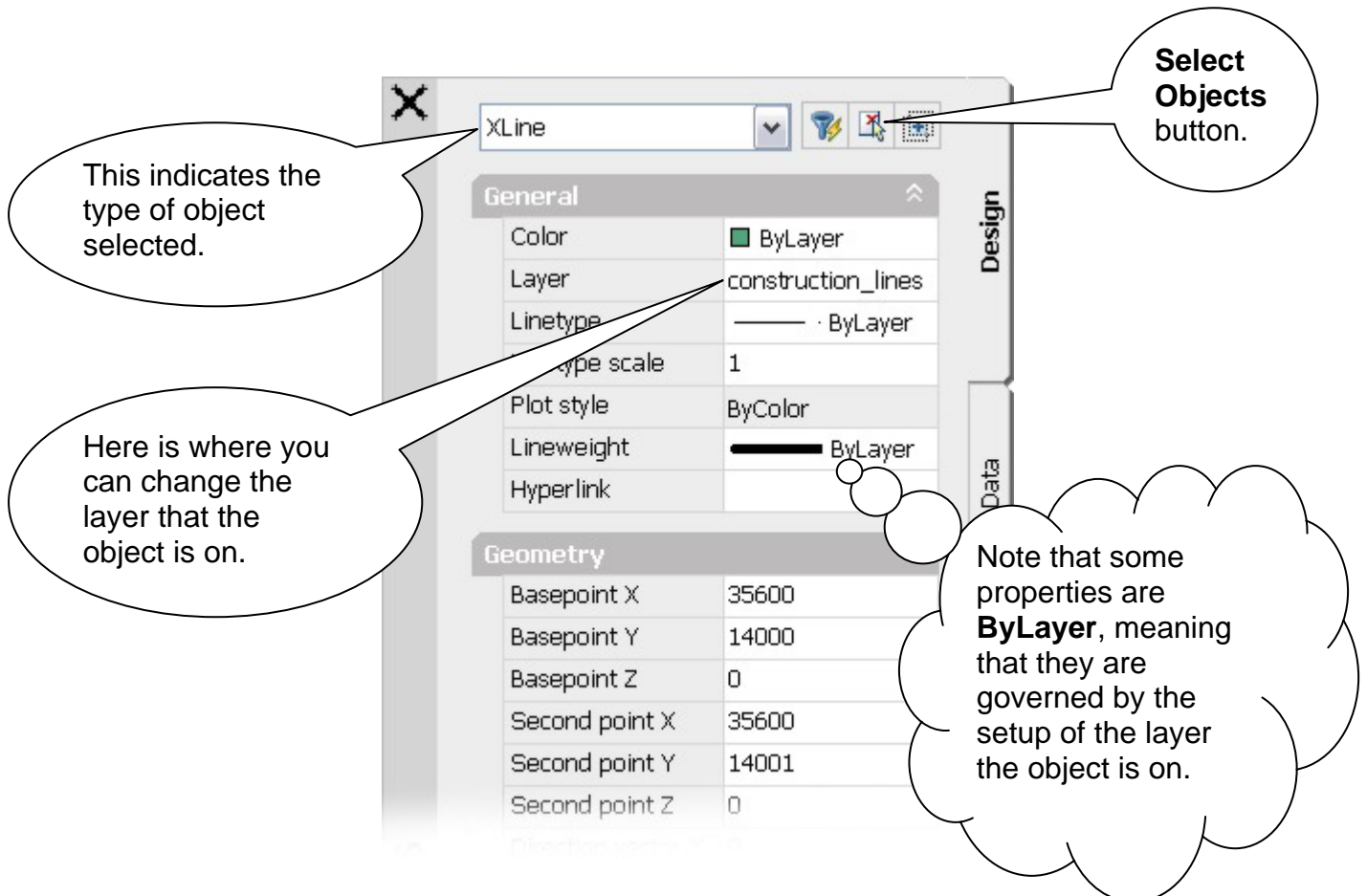
First we need to think about layers. Create a new layer called **outlines** (refer to the section on [layers](#)), make the colour a light blue and ensure the linetype is **continuous**.

3.4.9 Properties of objects and changing layers.


New commands used in this section: **properties**.

An object is anything created in the drawing, such as a line, circle, dimension, text, spline, etc. Each object has its own set of properties. You can change any of these properties whenever you like. To access an objects properties you simply double click on it.

Double click on one of the construction lines. You should get a **properties** window like this one:



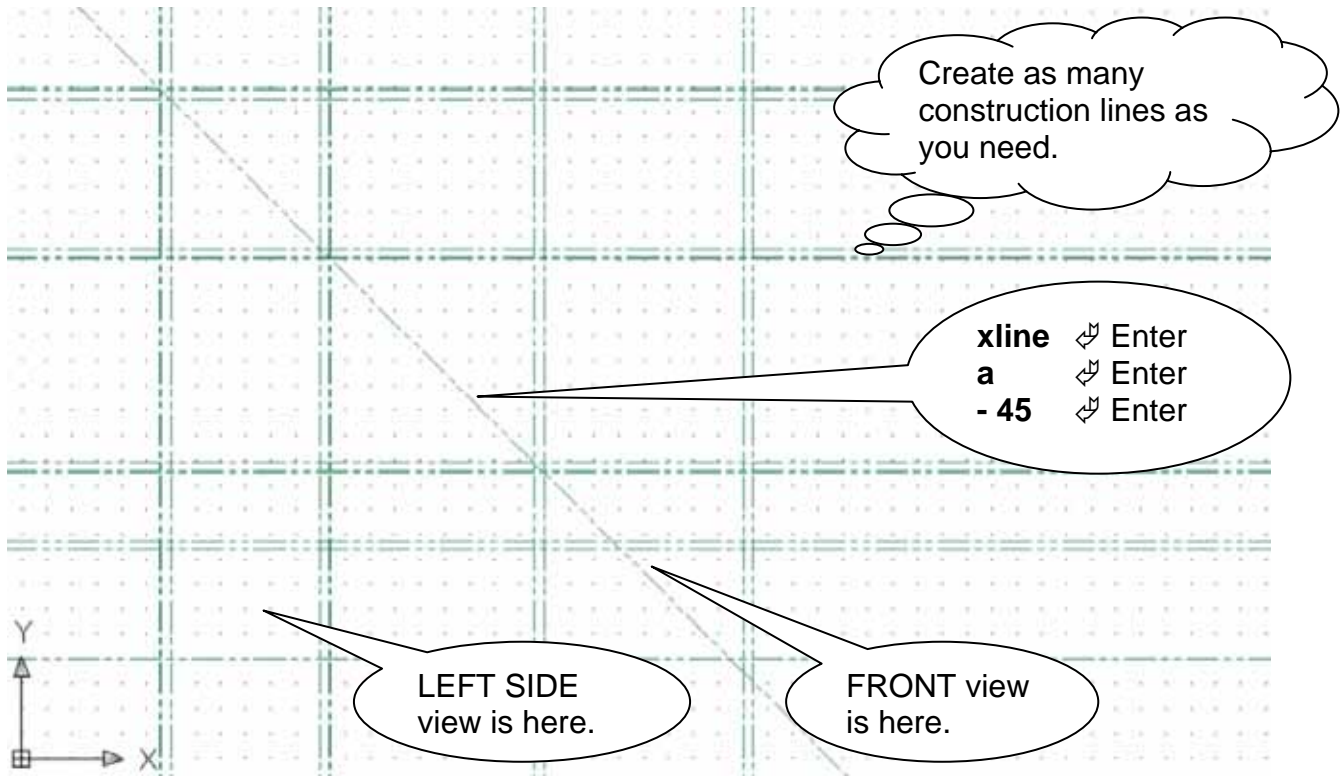
Now click on the **Select Objects** button and drag a window top left to bottom right over the original side view lines. ↵Enter to finalise your selection. The properties window should now indicate that 7 lines are selected. Change the layer that these lines reside upon by clicking on the **0** in the layer window and selecting the **outlines** layer.

 Dragging the selection window from left to right only selects objects **inside** the window perimeter. Dragging the selection window from right to left selects objects which the perimeter **intersects** with as well as those inside.

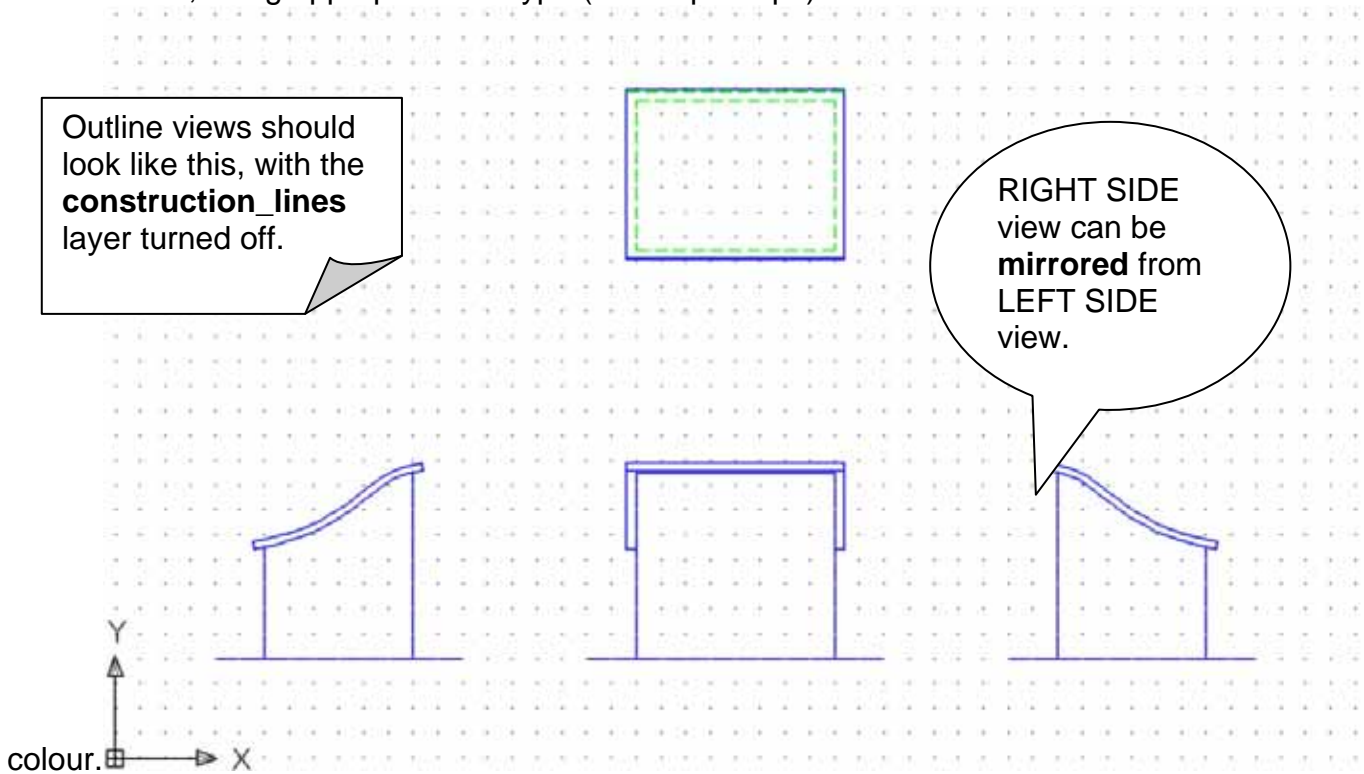
These lines should now be on the outlines layer and they should have changed colour to dark blue.

Click on the **X** to close the properties window.

Create the other views, PLAN and RIGHT VIEW, in 3rd angle orthographic projection. Make use of construction lines and the commands used so far.



Add hidden detail where appropriate; remember to create a layer called **hidden_detail** for the hidden detail, using appropriate linetype (DASH perhaps) and



Always think about how you can save time when drawing. For example, at this stage the left and right views are simply mirror images of each other, so rather than drawing the right view, simply use the **mirror** command to reflect it. With **snap** ON it is easy to select the first and second points down the middle of the FRONT view for the mirror line.

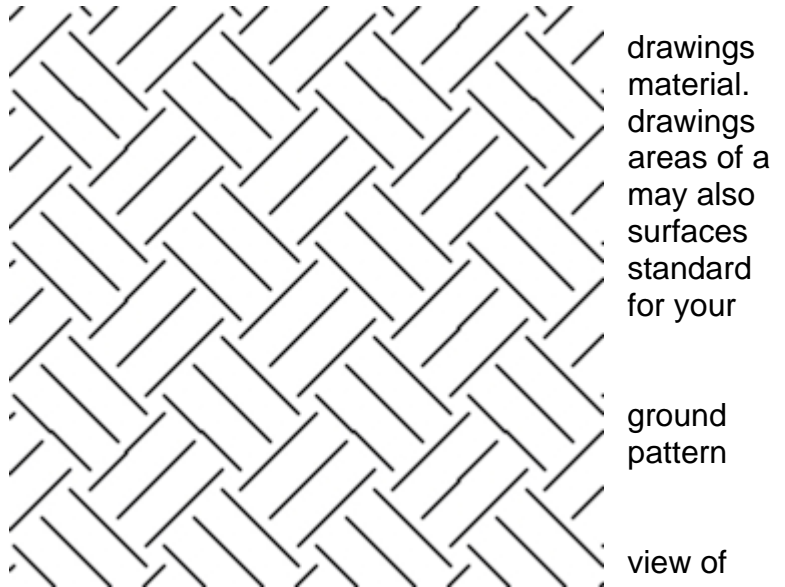
3.5 Hatching.

New commands used in this section: **bhatch**

Hatching is usually applied in construction to specify the use of a certain type of material. Hatching is usually applied to detail drawings to clarify cross sections of complicated structure or assembly of parts. Hatching may also be used on layouts to describe how surfaces are finished. For guidance on the use of hatching patterns refer to the course notes module and of course BS 8888.

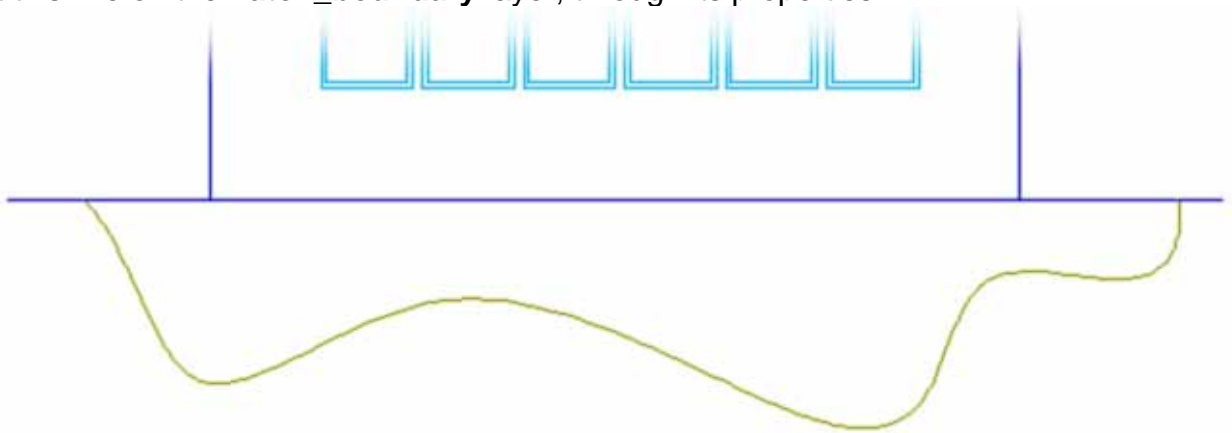
For this example we will indicate that the ground this building is on is earth. The hatching for earth is as shown here:

We will create a hatch region on the front view of the drawing.



Create two new layers, calling them **hatch** and **hatch_boundary**. Give them colours of your choice.

Draw a spline, using object snap to snap to the existing ground line, to define the boundary of the hatch. Put this line on the **hatch_boundary** layer, through its properties



perhaps?

Use the **bhatch** command to create a hatch pattern.

Command: **bhatch** ↵ Enter

In the resulting **Boundary Hatch and Fill** window click on the browse button for the pattern and select the earth pattern. Set the angle to **45°**. Experiment with the scale setting, **100** to **1000** perhaps. Either click on the **Add pick points** button and click inside the region you want hatched or use the **Add select objects** button and click on the lines you want to use to form the boundary of the hatching. Click on **preview** and follow the command line instructions. The hatch pattern should fill the spline area. Put the hatch pattern object on the **hatch** layer and turn off the **hatch_boundary** layer.

Video demo: Hatching >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/bhatch.wmv>

4 Repetitive objects.

New commands used in this section: **block**, **insert**, **wblock**

Reusable objects in AutoCAD are called **blocks**. Once defined they are embedded in the drawing file, and will always be there for future use. When a block is **inserted** it can be scaled in x, y or both, rotated or exploded (exploding breaks a complex object down into its individual elements such as lines and arcs).

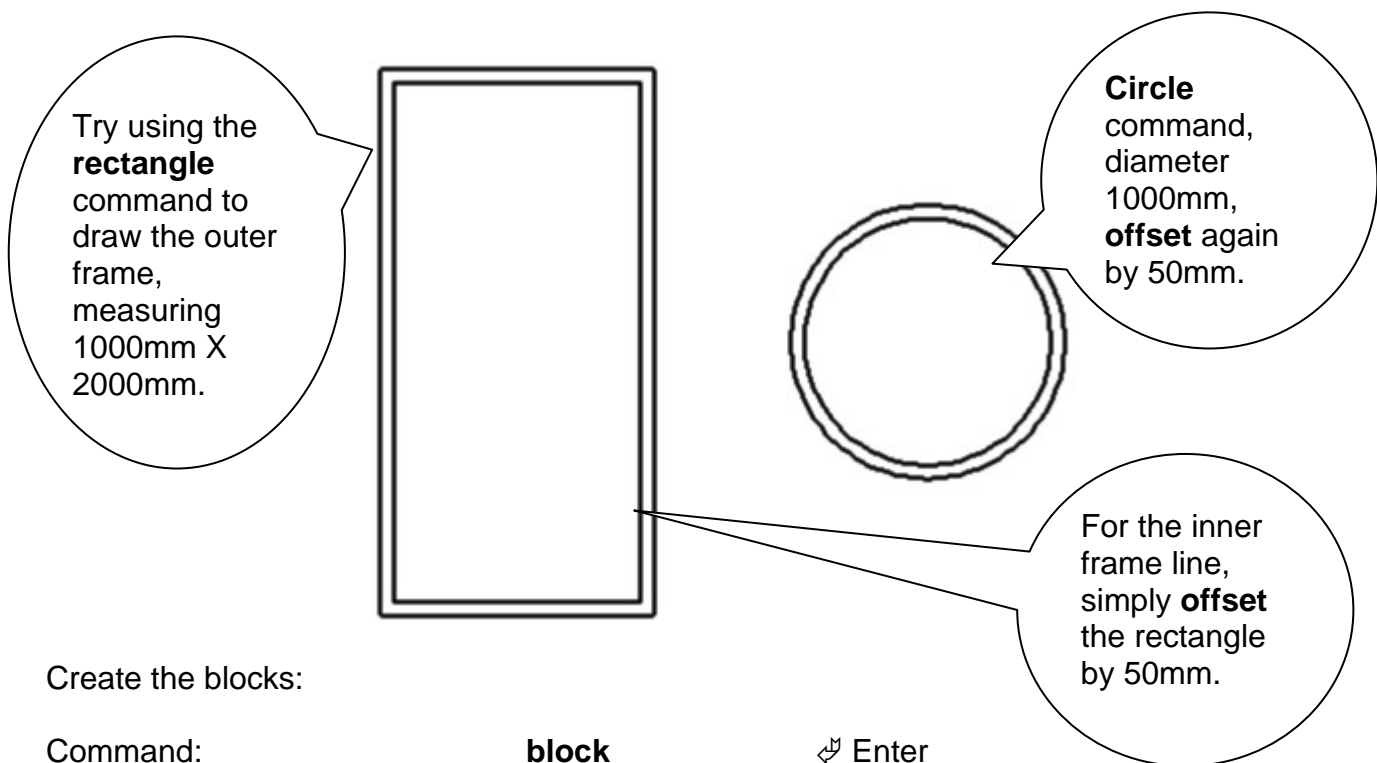
Two things to think about when creating blocks:

- Blocks created on **layer 0** will assume the layer properties any layer they are inserted on at a later date.
- Blocks require an insertion point, which is used as the base point reference when inserting.

For this drawing we will draw a couple of windows, make a block out of each and then insert multiple instances of them.

4.1 Creating blocks.

Make sure that layer **0** is the current layer. Draw these two windows, anywhere in the drawing, they will be removed later.



Create the blocks:


Command: **block** ↵ Enter

In the **Block Definition** window enter the name of the block: **window_rectangular**

Click on the **Select Objects** button and select the 'rectangular window' objects. In the Base Point section click on the **Pick point** button and using snap or object snap click on the bottom left of the rectangular window. Click the OK button and the block is defined. Repeat this for the round window, name the block **window_round** and use the center as the base point.

4.2 Inserting blocks.

We will insert these new window blocks using the **insert** command.

 Because our two new blocks were created on layer **0**, when inserted they will adopt the properties of the **current** layer.

So, create a new [layer](#) for the windows, give it a blue colour and make it current.

Run the insert command:

Command: **insert** ↵ Enter

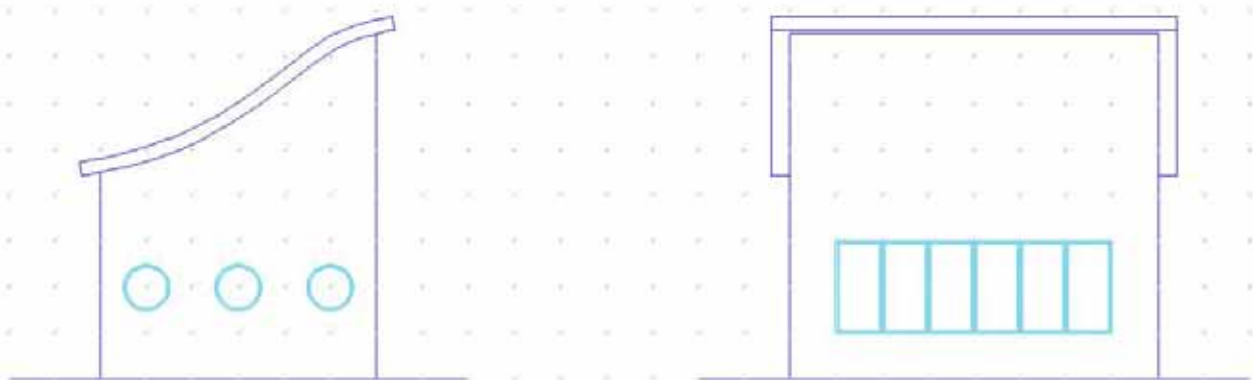
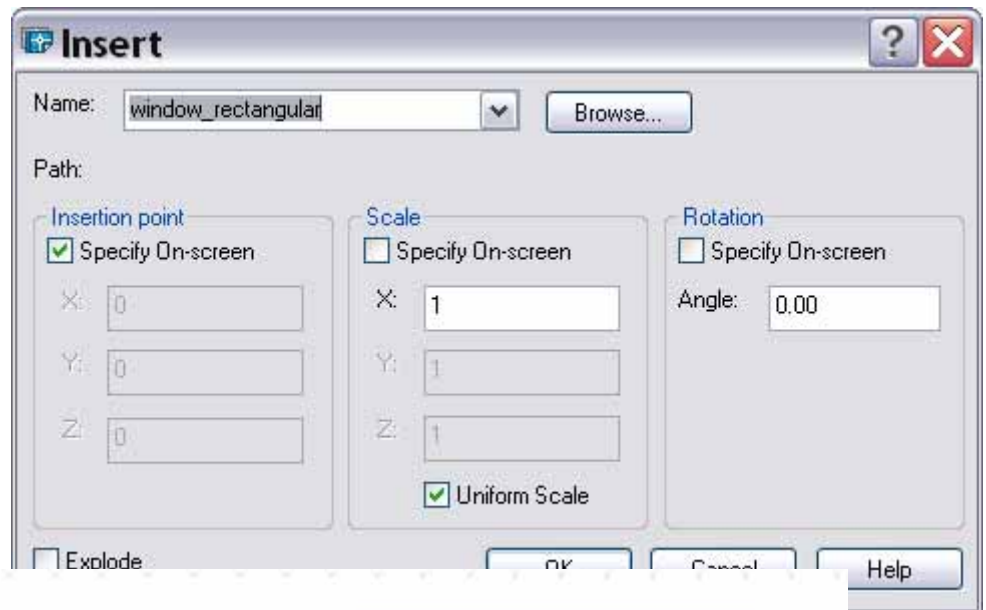
In the **Insert** window the following settings are made and click **OK**.

Then place the block in the drawing as required, perhaps like below.



For multiple insertions you could use the copy command (with the 'm for multiple' option) rather than inserting the block each time.

Complete the other views as appropriate.



For more practice draw a door on layer 0, 2200mm high. Make it into a block called door, and insert somewhere appropriate. (Don't forget to create a new layer for doors.)

4.3 Using blocks in other drawings.

You can insert any external AutoCAD drawing as a block into your current drawing. This, incidentally, is how the boarder is placed on the layout paper in section 6. You can also turn any block in your current drawing into an external separate drawing file. You do this using the **wblock** command (write block).

If you wish to experiment with this command:

Command: **wblock** ↵ Enter

...and experiment with the settings.

4.4 Re-defining blocks.

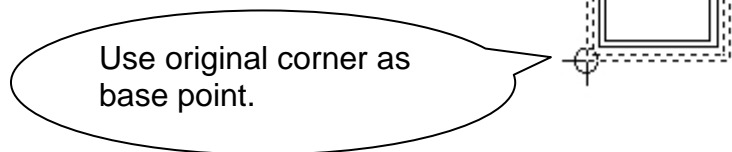
If blocks are not exploded, they are still defined by the block definition saved in the drawing. If you change this block by re-defining it, all instances of it will automatically change as well.

If, for example, you wanted to change the rectangular window design in the above example, you simply repeat the **block** creation process using the same name for the block, like this.....

Make layer **0** current. Redraw the window by inserting the original **window_rectangular** block. Explode it and offsetting the rectangles by 100mm, making a smaller window. Erase the original rectangles.

Use the block command to re-define the **window_rectangular** block. Use the original corner as the base point.

When the block has been re-defined your drawing should show the new smaller windows.



5 Dimensioning and annotation.

New commands used in this section: **dimstyle**

We now have a simple layout drawing, showing the Sports Hall design using FRONT, LEFT SIDE, RIGHT SIDE & PLAN views created using 3rd angle orthographic projection. Now we will add more information to the drawing such as basic overall dimensions and positions of various features such as windows and doors, and descriptions of the views.

5.1 Dimension style and drawing standards.

Dimensioning in AutoCAD is straight forward if you follow a consistent procedure for setting things up to begin with. The appearance or **style** of dimensions is highly configurable in AutoCAD. The style that **should** be used is usually specified in the drawing standards required by the client or organisation the drawings are being created for. They will either specify their own standards or internationally recognised standards defined by organisations such as BSI, ISO or ANSI.

All drawing conventions in these notes are based on:

BS 8888 Technical product documentation (TPD)

BS 1192 Construction drawing practice

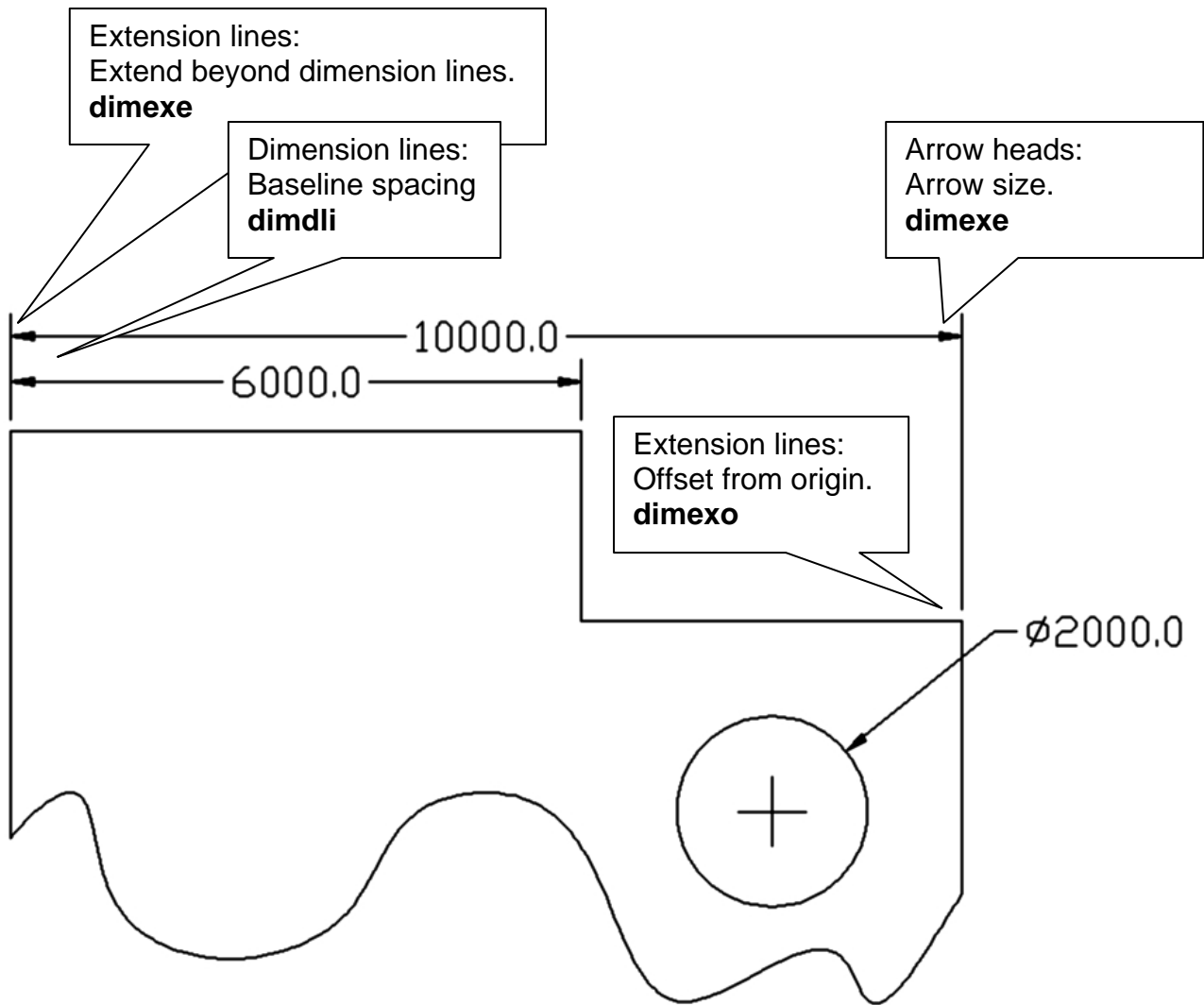
If no standards are referred to when drawing, the two standards above are recommended.

5.2 Creating a new dimension style.

We will use the default 'standard' style, modify it slightly and then save it so it can be used later on.

5.2.1 The features of dimensions.

Standard dimensions have particular features which are all configurable in AutoCAD. Each has its own value which can be set in the **dimstyle** manager window, as you will be shown next. These are some of the features along with the system variable in bold.



5.2.2 Defining a new dimension style.

To set up the dimension style use the **dimstyle** command:

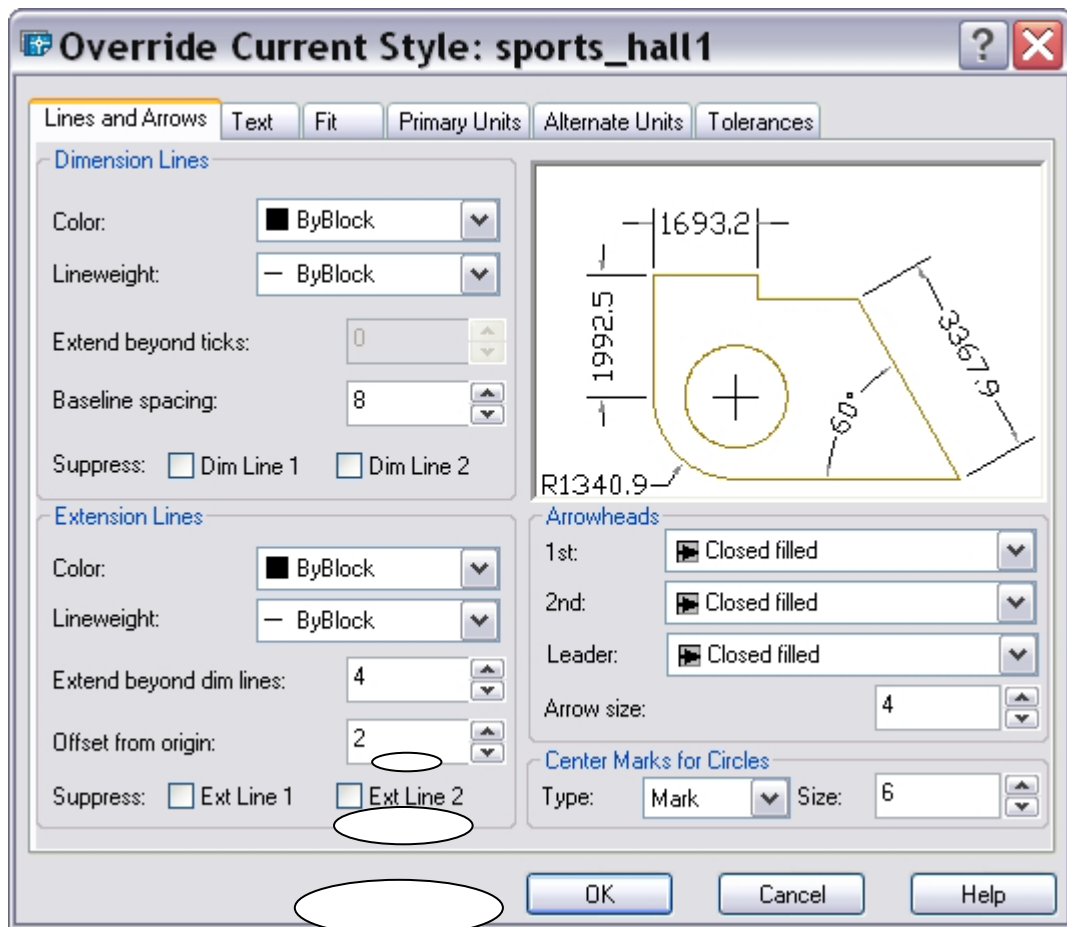
Command: **dimstyle** ↵ Enter

Note that this uses the 'standard' style. We will now create a new style:

Click on the **New...** button and in the window enter a name for the style such as sports_hall1.

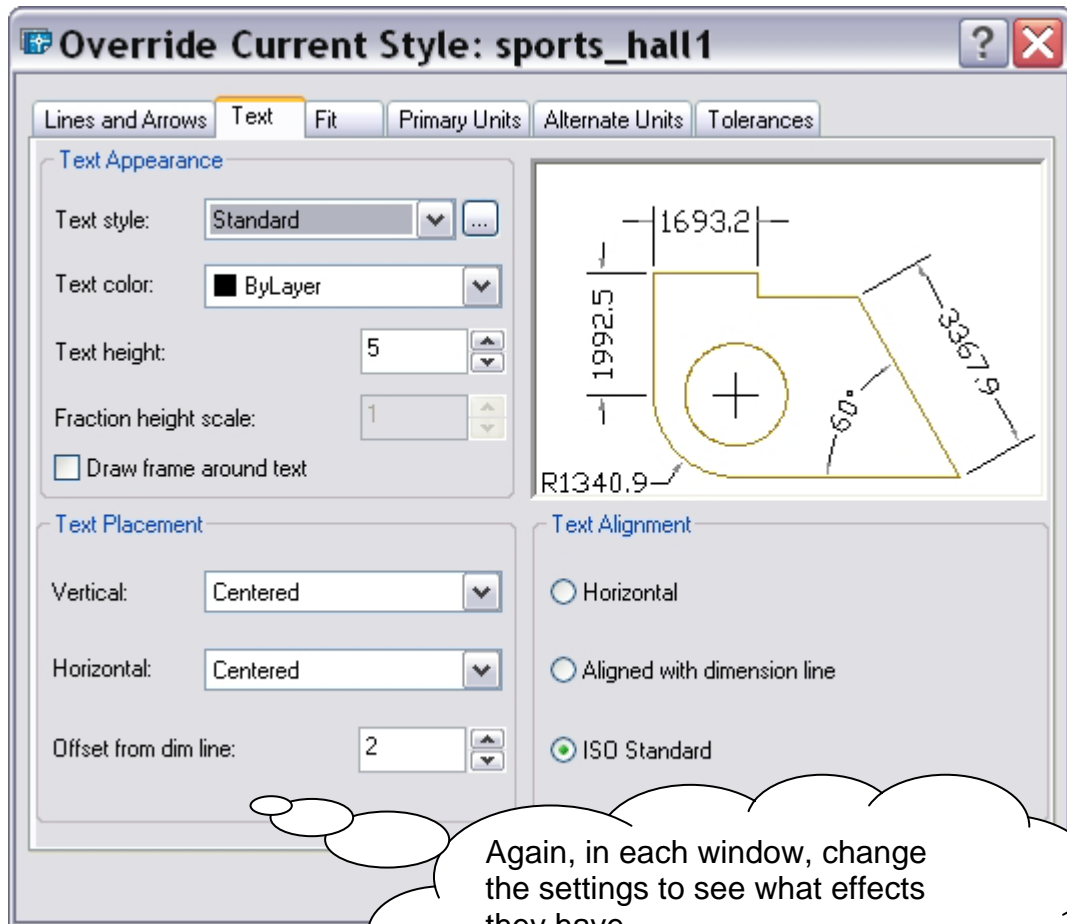
Have a look at the current set up, click on the **Lines and arrows** and **Text** tabs. Note that the sizes of the various features of the dimensions are all too small for our drawing. We will define the sizes of the various dimension features in order to conform to recommendations from BS 8888.

On the **Lines and Arrows** tab:



Change all of the settings to see what effects they have. Then set them as shown here. Note that the tabs in this window may be structured slightly differently on your version of the software, but the same parameters will be there somewhere, look for them.

On the **Text** tab:



Again, in each window, change the settings to see what effects they have.

Then set them as shown here.



On the **Fit** tab:

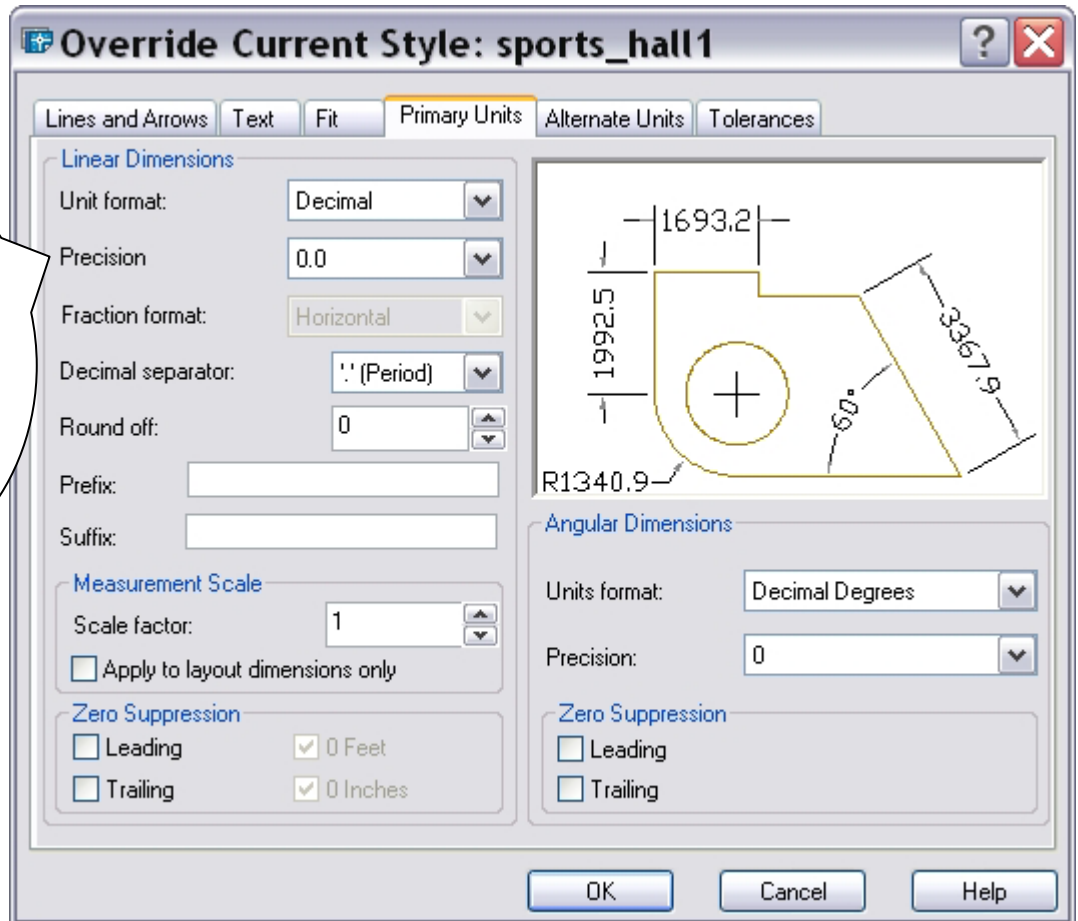
Overall scale is important.

As we are dimensioning in model space, we will have to apply an overall scale to magnify the dimensions.

Dimscale command also sets this value.

On the **Primary Units** tab:

Precision is also important, and its setting would depend upon overall size of objects drawn.

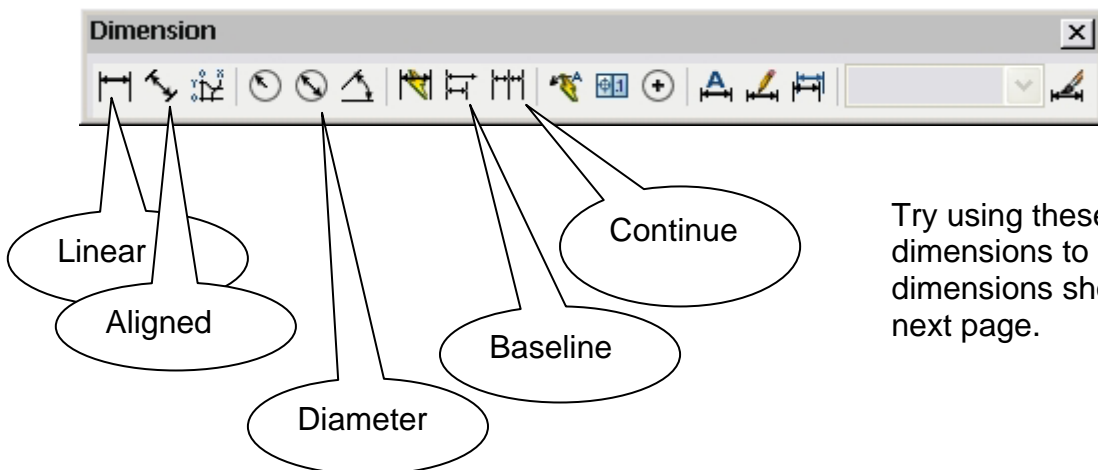


Once these settings have been defined, click on **OK** and set this new style to current and close.

5.3 Creating dimensions.

Create a new layer for the dimensions called **dimensions** and colour it **yellow**. Make this new layer the current layer.

Bring up the dimension toolbar by right mouse button clicking in an empty area of the top toolbar and select **ADT > Dimension**.



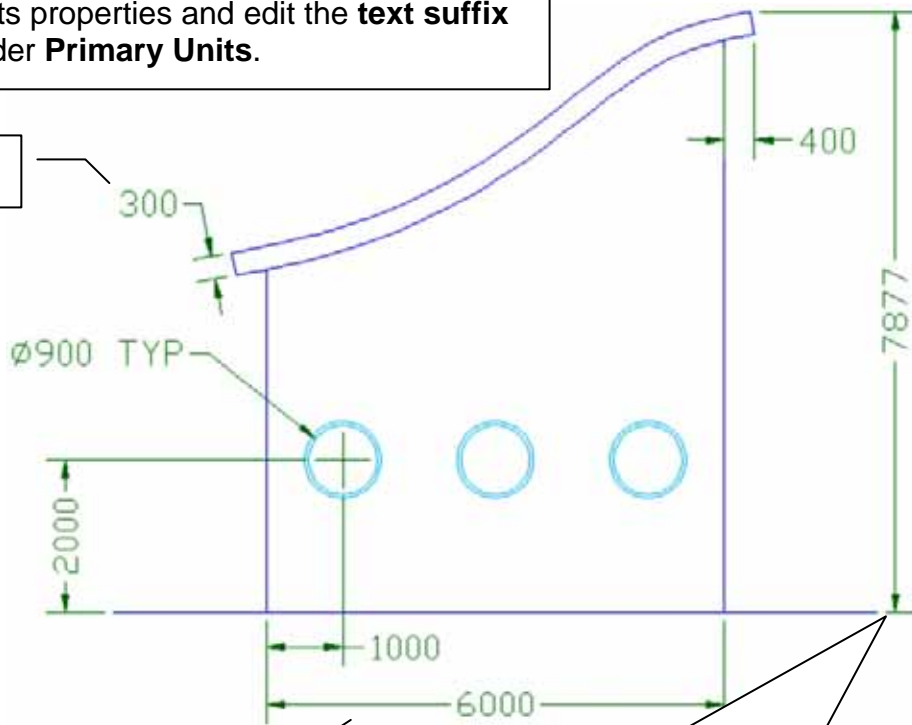
Try using these types of dimensions to create the dimensions shown on the next page.

Add these dimensions to a side view. (Your dimensions should appear yellow.)

Diameter

To add the text suffix TYP (which means typical), simply double click on the dimension to open its properties and edit the **text suffix** entry under **Primary Units**.

Aligned

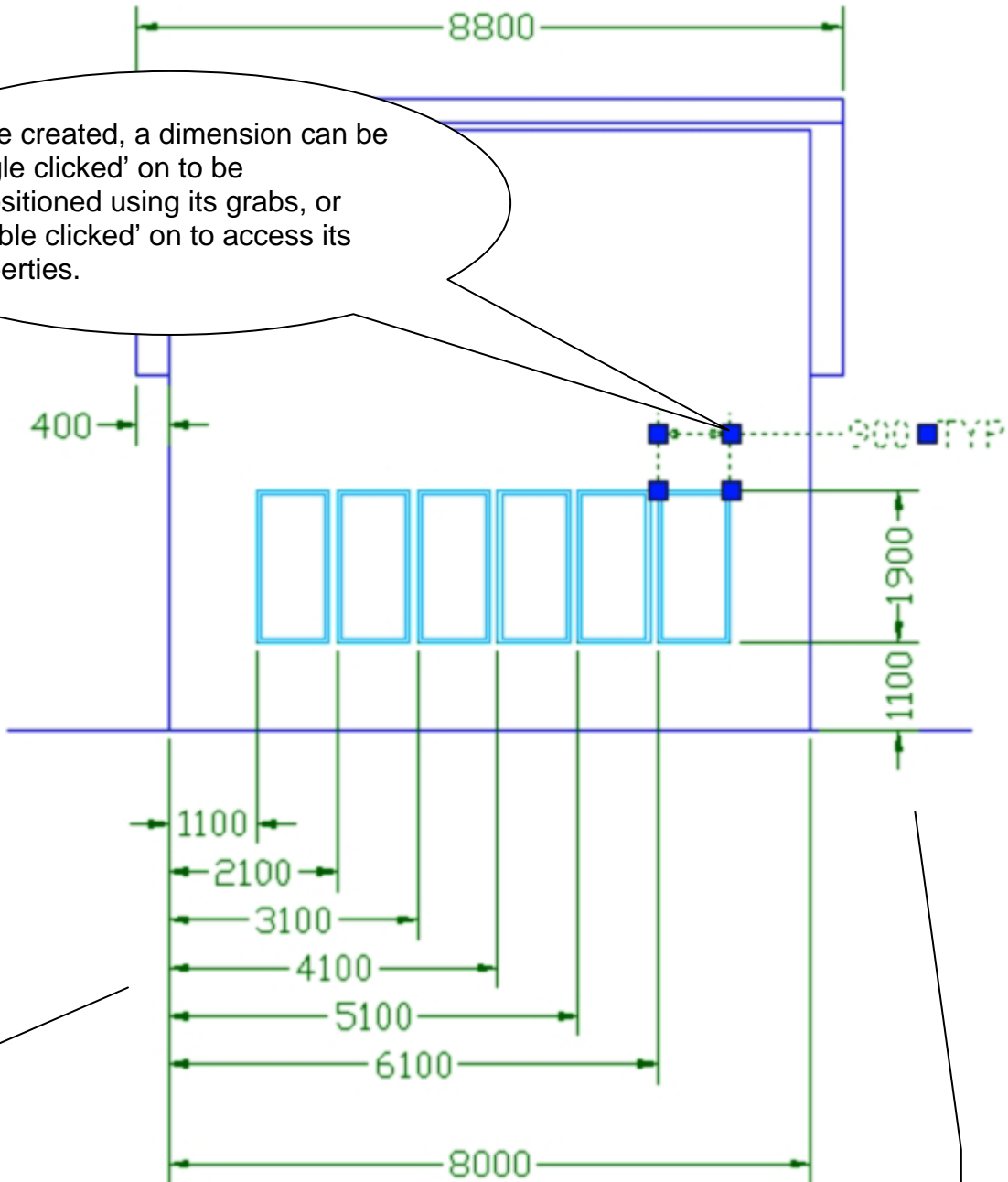


All other dimensions on this view are Linear

Dimension extension lines **should** have a gap or offset from their point of measurement or origin. This is to avoid confusion between actual drawing geometry and dimension lines. So... be careful what you select using object snap.

Next add some dimensions to the front view using the **linear** and then **baseline** and **continue** dimension types.

Once created, a dimension can be 'single clicked' on to be repositioned using its grabs, or 'double clicked' on to access its properties.



Baseline dimensions need a linear dimension to start them off.

Continue dimensions also need a linear dimension to start them off.

5.4 Annotation.

It is often necessary to add text to a drawing. In this section we will give each view a description by adding text, using the **mtext** command.

Firstly **create** and **make current** a layer called text.

We will now create a title for the front view. Run the **mtext** command:

Command: **mtext** ↵ Enter

The **mtext** command requires that you define a window in which the text will appear. Follow prompts on the command line, dragging a window over where you want the text to appear.

Examine the **Text Properties** window. Note how you can select the font type and size, and how you can enter the text in the window. Experiment with these settings.



Remember that BS 8888 requires that all characters in a drawing must be legible and **consistent**, with consideration being given to the possibility of drawing reductions and poorer quality reproductions being made.

No particular style is required, but characters should all be consistent on the same drawing, which means **using only one font type**. Capital letters are preferred to lower case ones.

Size of lettering, according to BS 8888, is given as a minimum height, relating to drawing size, as shown below:

Application	Drawing sheet size	Min. character height (in mm)
Drawing numbers	A0, A1, A2 & A3	7
Titles, etc.	A4	5
Dimensions & Notes.	A0	3.5
	A1, A2, A3 & A4	2.5

These sizes refer to the actual sizes on the printed paper copies of the drawing. As we are working to full scale in model space we will need to use a text height of something like 500mm. As the drawing model will all be reduced to fit on to an A3 sheet later on, this text should be about the right size.

Remember also that the [properties](#) of the text can be changed after you have created it.

Once you are familiar with the **mtext** command, create the other view titles, as shown on the [target drawing](#) on page 3.

6 AutoCAD – Presenting your drawing.

6.1 Introduction.

Once you have completed your drawing work in model space you usually need to present the information. It should be laid out on correctly formatted drawing sheets, with title blocks, boarders, company logos, etc. AutoCAD provides this function through the use of layouts.

Layouts can be thought of as sheets of drawing paper which can be added to the drawing file, as many as required. (AutoCAD files have default layout tabs next to the model tab at the bottom of the display.)

On a layout sheet AutoCAD shows you the model through a view port. You can create and size as many view ports as you like. Each view port can offer a different view point, pan and zoom set up, giving you total freedom to create any view you like.

The layout also allows you draw straight on to the paper itself, for displaying a title block and boarder for example.

In this section you will create a new layout and set up an A3 drawing of the Sports_hall drawing using a standard boarder format.

Video demo: Model space & paper space >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/paperspace.wmv>

6.2 Pre-requisites.

You must have completed the previous sections or have a drawing in model space.

You must have the drawing boarder file to hand, **UoP_A3.dwg**, downloaded from the module resource web page.

6.3 How to prepare a blank layout.

Open up the **Sports_hall.dwg** file.

You may need to display these tabs first: Right click over the **Model** or **Layout** buttons in lower toolbar area and select **Display Layout and Model tabs**. See [Section 2.3](#).

Right Mouse Button click the **Work** or **Layout1** layout tab next to the **Model** tab, bottom left of the display. Select **New Layout**. To activate it click on the new layout tab and then Right Mouse Button click on it and **Rename** it **A3plan**.

Right Mouse Button click on the new tab again and in **Page Setup Manager...** **Modify** set paper size to **A3** and make sure the plotting scale is set to 1:1, then click **OK**.

You will now be looking at a default viewport showing what you have drawn in the model space.

[Erase](#) this viewport. You now have a blank A3 sheet with no viewports.

6.4 Setting up the boarder.

Start creating the presentation of your drawing on this layout by setting up the boarder.

Insert the A3 boarder **UoP_A3.dwg** file as a block, **Insert > Block > Browse...** (for the downloaded **UoP_A3.dwg** file). Examine the layers now present in the drawing. You should see new layers brought in by the boarder file, prefixed with the paper size, A3 in this case.

Using the **explode** command explode this boarder block. This enables you to edit the text, etc.

Turn off the appropriate 1st/3rd angle projection layer, these two layers control the display of the projection symbol.

6.5 Setting up a viewport.

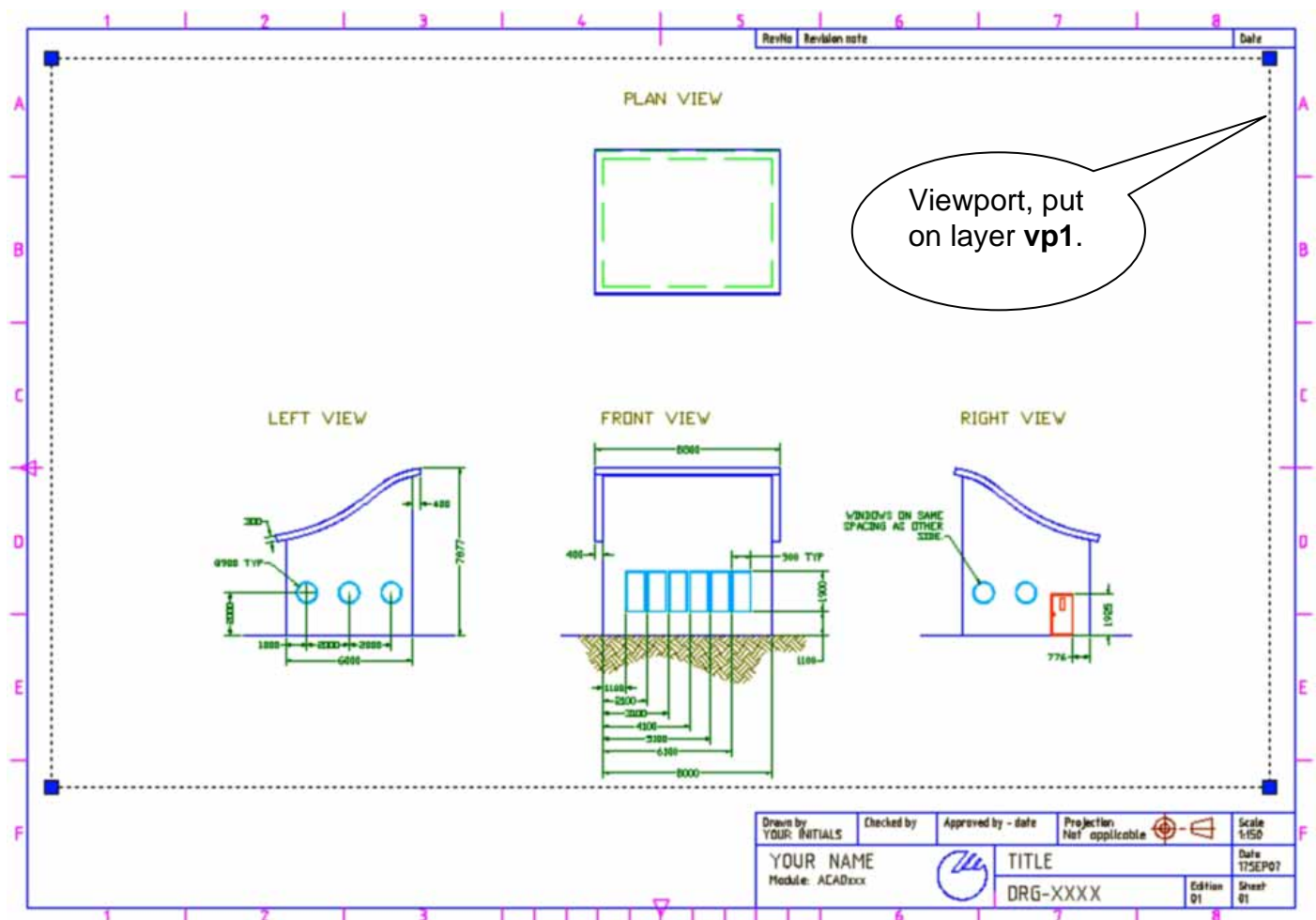
Now you will create a single viewport and re-size it to fit inside the boarder format.

Create and make current a new layer called **vp1** in the usual way. Create a new viewport by doing this.....

Pull down menu: **View > Viewports > 1 viewport**
Command line: **VPORTS**

In the display drag a rectangle on the layout to size to area of the paper you wish to use.

You should now be viewing the model from the layout.



You are viewing the layout in **Paper Space**. To modify how the viewport shows the model, change the mode to **Model Space**. Either click on the **Model/Paper** button, bottom of screen, use the **mSPACE** or **pspace** command in the command line, or double click inside the viewport for **Model Space** or outside the viewport for **Paper Space**.

In model space you will see a heavy boarder around the viewport and the model space coordinate system icon. You can now use the usual pan and zoom features to control how the viewport displays the model, but only if you set the **Display locked** property to **No**. Once you are happy with the view go back to paper space.

 To set the scale that the viewport uses to view the model:

Single click to select the viewport boarder. Activate its properties dialogue box either by clicking the properties toolbar button, or 'right mouse button' > properties or by running the **properties** command from the command line.

Under the **Misc** header, set the scale you wish to use from the pull down menu of the **Standard** scale. Alternatively you could enter a **Custom** scale or use the **ZOOM** command with the **Scale** option. Check out the Help for this.

To lock this view, and therefore the scale setting, simply select **Yes** for **Display locked**.

To stop displaying the viewport boarder simply turn off the **vp1** layer.

A viewport, remember, is an entity, just like a line or a circle is. It has its own properties and you have now placed it on its own layer. You can set up as many different viewports as you wish.

Edit the text in the title block appropriately and check that the projection symbol indicates the correct system of orthographic projection. (See course notes.)

Save your work.

Note that when working on a layout sheet, in AutoCAD, you may find that linetypes are not displayed properly. This may be because their size is being calculated relative to the paper size, not the model size. So for this example you may need to change the [LTSCALE](#) to something like **5** to see the dashed lines used to show hidden detail.

You may also note that any text added to the layout uses sizes relating to paper size, not model size.

6.6 Plotting.

Here are some tips for plotting your prepared layout sheets.

- It is probably best to plot in monochrome. Right click the layout tab for the sheet you wish to plot. To select a monochrome pen style:

Page Setup Manager > Modify > Plot Style Table > monochrome.ctb
....or similar

- Always perform a **Preview** before sending to the plotter.
- Check the appearance of the lines carefully.
- Make sure that the **line styles** display correctly.
- Make sure you have used appropriate **line weights**. In the Layer manager I recommend setting the geometry outline line weights to something like 0.3 to 0.5mm, and all centerlines and dimensions to 0.1 to 0.2mm. See the course notes for guidance on this.

7 Finally.....

With these basic draughting CAD skills you should now be able to create engineering drawings, fully dimensioned, annotated and presented on a standard paper size, using standard border/title blocks. You should be able to control objects using layers and change the properties of each object using its own properties, create repetitive objects using blocks and perform various editing functions such as copying, trimming, etc.

Using the AutoCAD help system you should now have a look at creating **polylines**, **multilines**, **arcs**, etc. You should also examine some more editing commands such as **scale**, **stretch**, **rotate** and **array**.

When you are familiar with these commands you could move on to the [3D tutorial](#).

8 Summary of commands & video demonstrations:

8.1 A quick summary of the basic commands used in this tutorial.

Tool bars are also used. Right click in an empty toolbar area to configure.

Command line:	Key shortcut:	Pull down menu:	
CLOSE		File >	Close
UNITS		Format >	Drawing Setup...
LIMITS		Format >	Drawing Limits
GRID	F7	Format >	Drafting Settings...
SNAP	F9	Format >	Drafting Settings...
ZOOM	Z		
	Esc		
SAVEAS		File >	Save as...
LINE		Draw >	Line
COORDS			
SPLINE		Draw >	Spline
ERASE		Modify >	Erase
TRIM		Modify >	Trim
OFFSET		Modify >	Offset
EXTEND		Modify >	Extend
OSNAP	F3		
LA			
XLINE		Draw >	Construction Line
LTSCALE			
PROPERTIES		Modify >	Properties
MOVE		Modify >	Move
BHATCH		Draw >	Hatch
BLOCK		Draw >	Block > Make
INSERT		Insert >	Block...
DIMSTYLE		Format >	Dimension Style...
MTEXT		Draw >	Text
VPORTS		View >	Viewports

8.2 Video demonstrations.

List of video demonstrations, to be used in conjunction with the tutorial:

Units, limits and grid >

http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/units_limits_grid.wmv

Coordinates >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/coordinates.wmv>

Snap >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/snap.wmv>

Absolute & relative coordinates >

http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/abs-rel_coordinates.wmv

Splines >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/splines.wmv>

Trimming & extending >

http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/trim_extend.wmv

Offsetting >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/offset.wmv>

Object snap >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/osnap.wmv>

Layers >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/layers.wmv>

Linetype scale >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/ltscale.wmv>

Hatching >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/bhatch.wmv>

Model space & paper space >

<http://www.tech.plymouth.ac.uk/dmme/cad/wmv-2d-autocad/paperspace.wmv>