AutoCAD Tutorial - Part1: Draw the front view of the mechanical part from Tutorial 3.

SECTION A-A

50
28
14
5
R2
R2
15
18

2 x Ø10.2 x 22 depth then
M12 x 1.75 x 6H x 15 depth
A
R6
R32
R20
97
55
13
105
70
55
A
A
A
A
12
30
12
110
12
30
12
2xØ16
32
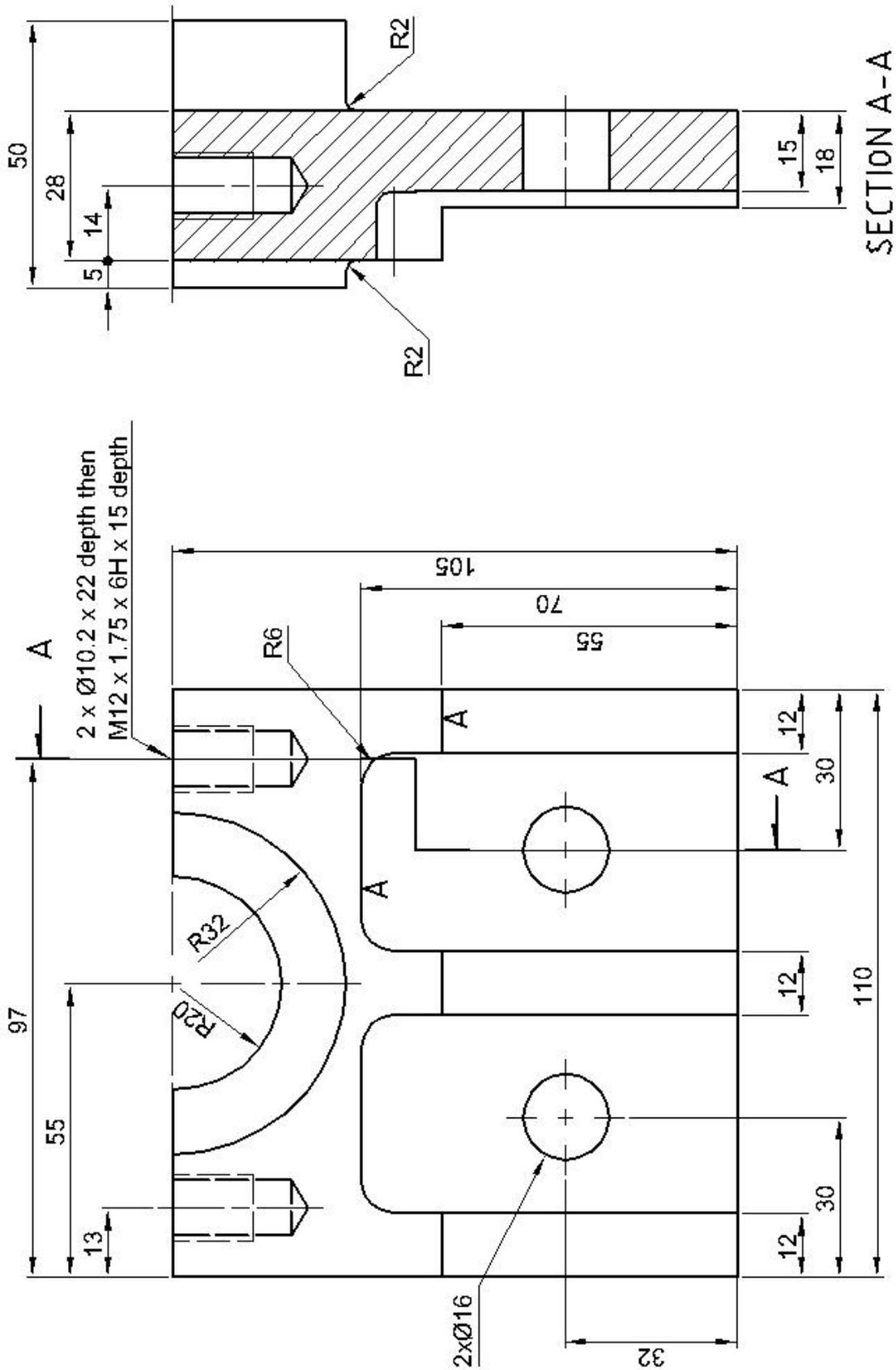
In this tutorial you will learn how to start AutoCAD, make the new drawing, draw the front view of the object shown and how to save a drawing.

Starting AutoCAD

Log on to the computer using your university login name and password. Start AutoCAD by double clicking on the AutoCAD icon or through Start/Programs/Main Applications/AutoCAD 2009 menu.

Once AutoCAD has loaded, first cancel AutoCAD Today window by pressing on the X on the top right side of the window. Then move the mouse around until you see a crosshair cursor. The AutoCAD window has a number of important features:

Despite command line interfaces being considered totally archaic the command area in AutoCAD is absolutely vital! One of the key things I'm trying to "get you to do" in these tutorials is to watch the command area! Using AutoCAD is like a conversation and AutoCAD's half of the conversation comes from the text in the command area...

1.   The standard Windows ribbons and panels.
2.   The standard Windows toolbar above the menus, it includes: File-New, File-Open, File-Save, Print and "Find and Replace"(!!).
3.   AutoCAD specific toolbars: Object Properties, Draw and Modify (there may be others...?).
4.   The graphics area - that's the area where you draw - note the scroll bars and the axis label.
5.   View icons - these 'tabs' give access to different view of the current drawing. The "model" tab should be selected at present.
6.   The command area - this small window (by default) has space for three lines of text - this is where you type commands.
7.   The status area, at the bottom of the AutoCAD window, this includes the current cursor position.
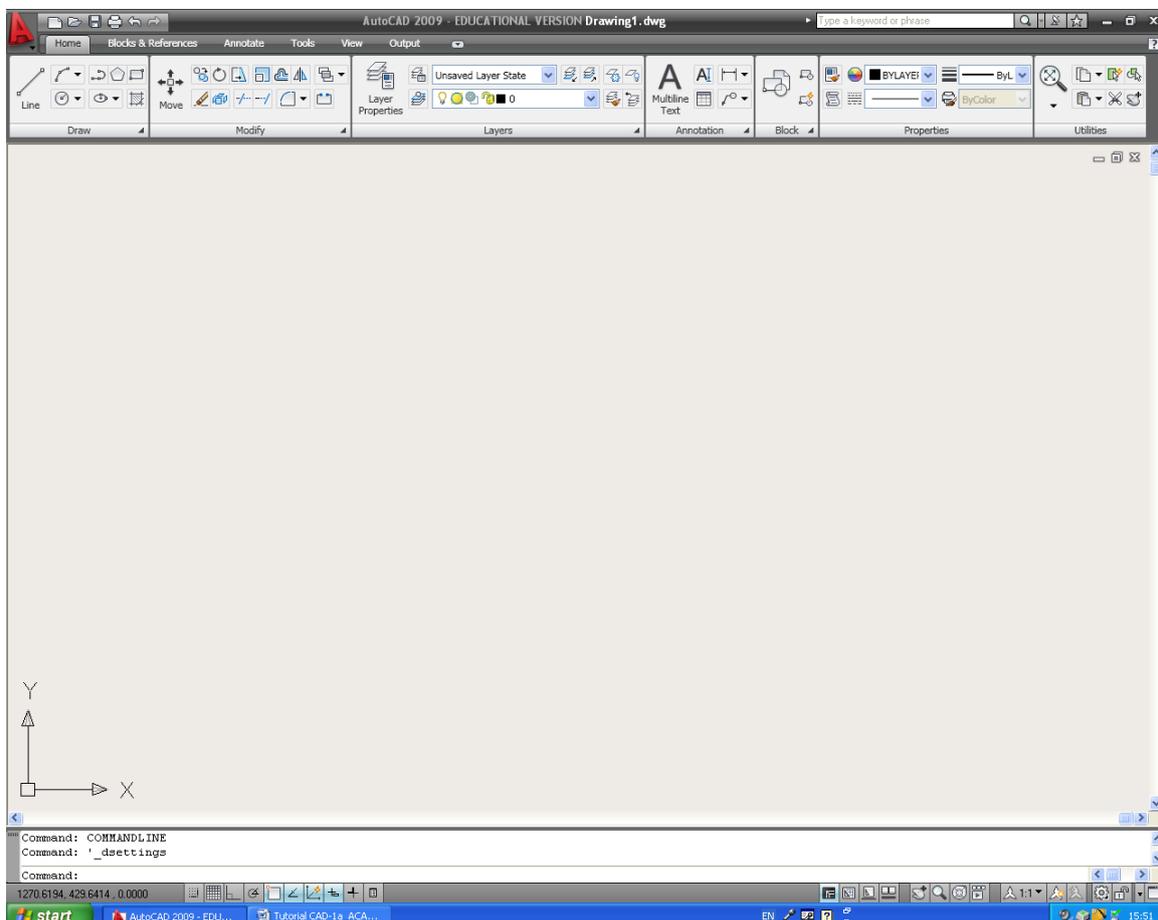
Command Entry

Typically there are three ways of giving a command!

1. Type the command using the keyboard - the command is displayed in the command area.
2. Select the command from a menu (under the big red A).
3. Select the command's icon from a ribbon panel.

Not all commands are on the Menus and/or toolbars!

When we want you to type a command in the command area the AutoCAD command will be written like:

   type: **QSAVE** ↵

This means : type the text (qsave) and then press the Enter key (or the space-bar).

When we want you to select a command from a menu, it will look like:

   select **File - Save**

This means: click on the 'File' menu and then 'Save' (which should be one of the items on the 'File' menu).

When I want you to pick a command from a toolbar, I'll write:

   select **Save**

AutoCAD also supports common shortcuts like Ctrl-S for Save!!

Here I'm trying to cover all the bases by giving you the command to type, the menu options and showing you what the toolbar icon looks like!

Despite all of the above, I'll probably use a combination of the above like:

   select **File - Save** (or type **QSAVE** ↵)

Create and set up New Drawing

This will create new blank drawing

Go to City Space ME1105 page and Click on **City A3 – V2004Template**. Switch to the model space by clicking on **Model** icon (from the Status area (underneath the command line).

Set up the drawing limits - space in which the drawing will be made

Select: **Format – Drawing Limits**, then type:
   0,0 ↵
   420,297 ↵   these are absolute Cartesian coordinates

This repeats last executed command

Press: **ENTER** to repeat the last command

*This turns drawing limits on, which enables you to draw only within that area. Attempts to draw outside it result only with message ****Outside limits***

Type: **ON** ↵

Show the grid	Select: **GRID** from the Status area or type: **GRID** ↵ and then **ON** ↵
Snap to grid	Select: **SNAP** or type: **SNAP** ↵ and then **ON** ↵ Type: **SNAP** ↵ and then **5** ↵ Turn **SNAP** OFF
Enter ortho mode	Select: **ORTHO** or type: **ORTHO** ↵ and then **ON** ↵
Select current layer	Select **Layer Properties manager** 🗒 or type **LAYER** ↵ Double click on Layer **DRAW** to make it the Current layer (green tick). Close **Layer Properties manager**.

Start drawing the front view of the mechanical part

In this part of the exercise you will make a front view of the part drawn in exercise DrE-3

Select **Rectangle** (or type RECTANG↵), then type:

The text typed is displayed in the command area at the bottom of AutoCAD's window.

100,100↵	starting point is absolute Cartesian coordinates
110,105↵	continues in relative (Incremental) Cartesian coordinates

Hopefully AutoCAD drew a rectangle, which fits comfortably in the AutoCAD graphics area!? If you can't see the rectangle, type z [space] a [space] (zoom all) - this instructs AutoCAD to redraw the view, "zooming out" to show all the graphics on the drawing.

Coordinate Systems

AutoCAD is a three dimensional CAD system, so you can also enter XYZ values instead of the XY values shown here.

When specifying positions you can use Cartesian or Polar Coordinates. Cartesian coordinates are simply a X value, a comma, and a Y value, for example: **100,100**. Polar coordinates are a Distance followed by a < symbol and an angle, for example: **10<25**. Angles are measured in degrees, with 0 = East and 90 = North. Any of these numbers can have decimal values.

Consider relative coordinates simply as distances!

AutoCAD defaults to use **"relative coordinates"** to specify a position relative to the current position, for example: **5.6,-3.4** and **16.32<62**. To specify a particular position we need to use **"absolute coordinates"** by preceding the coordinate with #, eg **#5.6,-3.4** and **#16.32<62**.

Draw some lines using Line command

*Instead of **LINE**, you can also type: **L**↵ this is an "alias" - which is a short-cut.*

We will now draw some more graphics and then save and exit AutoCAD.

Type **LINE** ↵ or select ✏ from the draw menu

Type **112,100** ↵
Type **0,70** ↵
Type **37,0** ↵
Type **0,-70** ↵

Remember: If you press ↵ after you finish a command, the command is re-issued.

Press ↵ to terminate the command, and then press ↵ again to re-start the command!

If you type symbol @ before

Here is an alternative way to specify **"relative coordinates"**.

the coordinate entry this specifies relative position to the previous last position of the cursor

Type: **100,155**↵ for the line start, then type **@12,0**↵ for the relative coordinate definition.
Terminate line command by pressing ↵.

Snap Modes

A simple way to turn Object Snap ON or OFF, is to click on "OSNAP" in the status Area. To see the various snap options "right-click" on "OSNAP" (in the status area) and select "Settings...".

It is often useful to be able to draw something from (for example) the end of another shape. AutoCAD has a large selection of "snap modes" for this purpose. The most commonly used snap modes are "Endpoint" (which snaps to the end of the selected graphics entity) and "Intersection" (which snaps to the intersection of two graphics entities).

To get AutoCAD to display the Object Snap Modes toolbar, select "Tools","Toolbars", "Autocad","Object Snap".

The object snap modes can either be typed or they can be selected from the standard toolbar ⬚or from the snap toolbar:

To display the "Drafting Settings" dialog, click on the "horse-shoe" shaped icon on the end of the Object Snap Toolbar.


Make fillet in the corner ⌐

From the Modify panel Select **FILLET** icon ⌐ or type **F**↵ or **FILLET**↵.
Type **R**↵ and then type **6**↵. Note the question Autocad is asking you in Command line.
Select the second vertical line from the left and the horizontal line on top of it. AutoCAD will make fillet in that corner.

Repeat the fillet command

Press ↵ to repeat the last command and select the next vertical line on right and the line on top.

*You should have this shape on the screen*
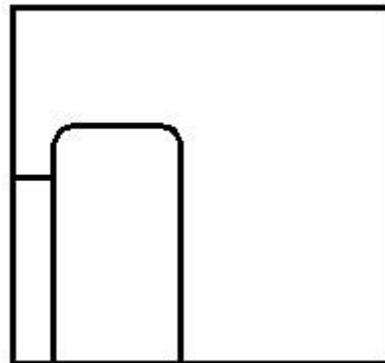
Use mirror command ⧗

Select **MIRROR** icon ⧗ from the modify panel or type **MI**↵ or **MIRROR**↵
Position the cursor on the left side of this drawing underneath the top horizontal line. Left click to mark the first corner of the selection box. Position the cursor somewhere in the bottom middle part of this drawing underneath the bottom horisontal line. Left click to mark the second corner. This will select objects that have to be mirrored. Press ↵.

To be sure that you have selected the mid point of the line you should look for an orange triangle on mid point of the line.

To mirror these objects around the mid line, from the Object snap toolbar select 'Snap to Midpoint' and then select the top horizontal line of the drawing. Repeat that for the second point on the mirror line (midpoint of lower line). Type **N**↵ (don't delete the old objects).

Follow these two steps shown in the figure to select objects for mirror and to define a mirror line.
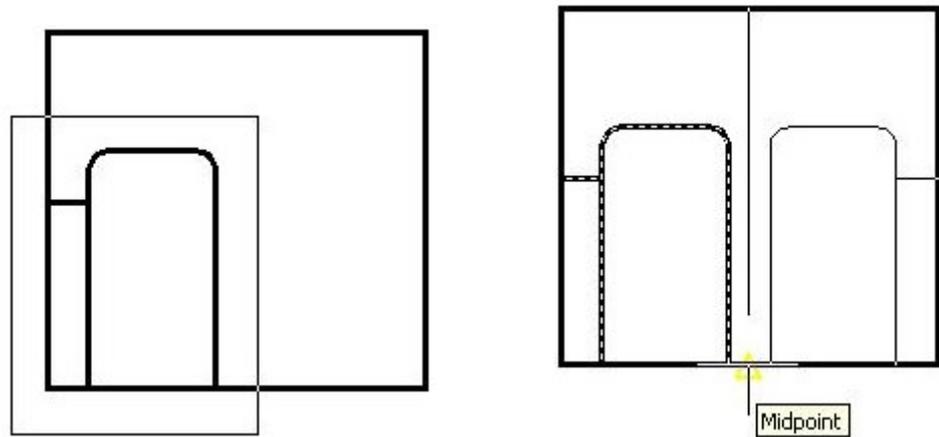
Click **OSNAP** (object snap) and **OTRACK** (object track) to turn them on. Type **L↵**, then position cursors on the intersection and them move to the right and click to select the intersecting point. (Look for orange X).

Draw the line in the middle
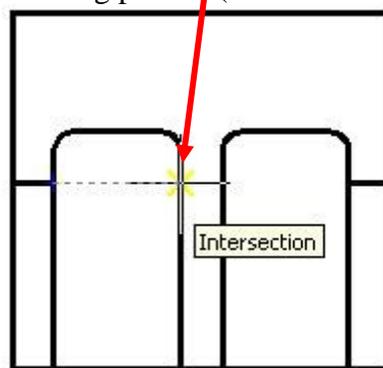
Type **12,0↵**. Press ENTER to cancel the command

Saving and Closing a Drawing

Note that if you type SAVE ↵ , you actually get "Save As"!!

The quickest way to save is to press Ctrl-S - this is the same as "qsave".

Select the Save icon from the standard toolbar. The drawing has not been saved before, so AutoCAD will display the SAVE AS dialog box, select the appropriate Drive and Directory (for example: U:\My Documents), type the drawing name: **CADTutorial1**, and then select **"OK"**.
If AutoCAD 2009 Today window appears, turn it off by pressing on X in the top right corner.

Starting AutoCAD with an Existing Drawing

To load a drawing, start Windows Explorer and select the appropriate Drive and Directory. Once Explorer is showing the correct directory then double-click on your **CADTutorial1.dwg** drawing.
If AutoCAD 2009 Today window appears, turn it off by pressing on X in the top right corner.

If you can't find your drawing (in Windows Explorer) then press F5 (function key 5), this tells Explorer to update the directory display; if you still can't find your drawing then perhaps you saved the drawing in some other directory - load AutoCAD and then select the File menu, at the bottom of the File menu is a list of recently opened drawings, select your drawing from the list.

Draw an Arc

Select **Arc**, then **Center, Start, Angle** (or type ARC↵ , then type **C↵**) to select the centre of the arc. Select the **midpoint snap mode** (or type: MID↵) and select the top line of the drawn object.
Specify start point of the arc: **-20,0↵**, then type **180↵** (or **A↵** for angle and type

180 ˌ ꟾ - check what the command line is asking you for) to define the angle.

Use offset command 🖰

Select: **OFFSET** button or type **OFFSET** ↵
Then type **12**↵ for the offset distance
Position the crosshair over the last drawn arc and click the left button to select.
Move the crosshair to the outside of the arc and click the left button again.
Press **ENTER** (↵) to terminate the command.

Trim icon ⵂ
When picking lines to be trimmed, the part of the line that you select is important. In the example in the following figure, if you pick to the left of the selected arc (rather than to the line inside the arc), then the line will be trimmed from the arc to the boundary line.

Select: **TRIM** button from the edit toolbar or type **TR**↵ or **TRIM** ↵
Select the smaller arc you drew above. Press ↵ to terminate object selection.
To select object to trim, position the crosshair on the top line in between the ends of the smaller arc and click the left button to select.
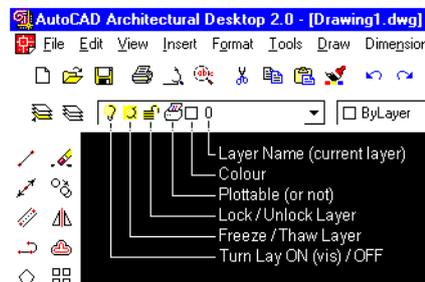Press **ENTER** (↵) to terminate the command.

This is the object that you should have now
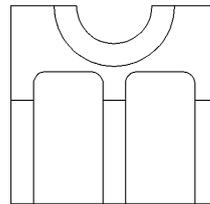
Layers

Most CAD systems have some kind of overlay concept. AutoCAD uses **layers**. Layers are used to separate and structure drawings; layers can be turned on and off (for example to vary the amount of detail in a drawing), and can have linetypes associated with them....

When using AutoCAD, the graphics appears on the current layer, so be careful that the current layer is correct. Look in the **Layers** panel: the current layer and its colour is displayed there (*see previous figure*).

Layers icon 🗐
*The **current layer** is the layer to which new graphics is added. The properties command can be used to change a graphics object's layer (or linetype or colour)*

Select the **Layer Properties** icon or select **Format - Layers...**, AutoCAD will display the "Layer Properties Manager" dialog. Select the layer with name **CENTRE** then double click to make this the current layer.
Close the layer property manager by pressing OK.

Draw centrelines

Draw some lines   Select **LINE** icon or type **L**↵, select the centre of the arc as first point and pull down to pass the bigger arc. Terminate command with ↵.
Note: *You have selected a circle centre if a small orange circle appears there.*
Press ↵ to repeat line command. Draw line by selecting ends of the smaller arc. Terminate command with ↵.
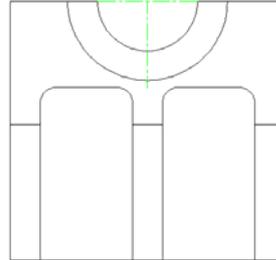
Draw more lines   Type **L**↵ an then:
113,207↵
0,-30↵ and ↵ to terminate the line command
Type ↵ to repeat the line command, then
120,132↵
20,0↵ and ↵ to terminate the line command
Type ↵ to repeat the line command, then
130,122↵
0,20 and ↵ to terminate the line command
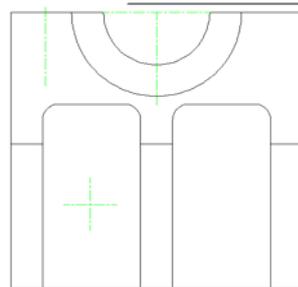
Draw threaded hole with hidden lines

Select the **LAYERS** icon (or type **LA**↵), then press select **New Layer** icon to create new layer**.**
Type **SECTION** for the name of the layer. Press on the field **continuous** in order to change the linetype for the layer. Select **LOAD** option and scroll down until you find linetype **PHANTOM**. Select it and press OK. Press on "color" field in the same lane to change layer colour and select green.

Create new layer and change it to a current layer
Layers 📑

Select layer **HID**. Double click to make it current layer.

Close the Layer Properties dialogue.

Type **L**↵, and then give coordinates:
107.9,205↵
0,-22↵
Draw lines   **10.2,0**↵
0,22↵, then press ↵ to terminate line command.
Press ↵ to reactivate line command and then type in following coordinates:
107, 205↵

0,-15↵
12,0↵
0,15↵, then press ↵ to terminate line command.
Press ↵ to reactivate line command and then type in following coordinates:
Select the bottom left corner of the hidden hole first drawn in the previous set of commands or type: **107.9, 183**↵.

Then type **10<-30**↵.
Press ↵ to terminate line command.

Use mirror command Press **MIRROR** icon or type **MI**↵.
Type **L** for last drawn object, or select the line that you have just drawn.
Press ↵ to terminate object selection.
Select the point on the top of the centreline and another point on the bottom of the centreline. Press **N** for 'do not delete original object.

Cut off exceeding parts of lines. Use **TRIM** command or press TRIM icon ⌐. Select both lines you previously made and press↵. Click on the overhanging parts of these two lines to cut them off. Terminate the command with ↵.
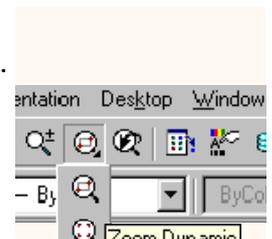
This is the result of the previously taken action
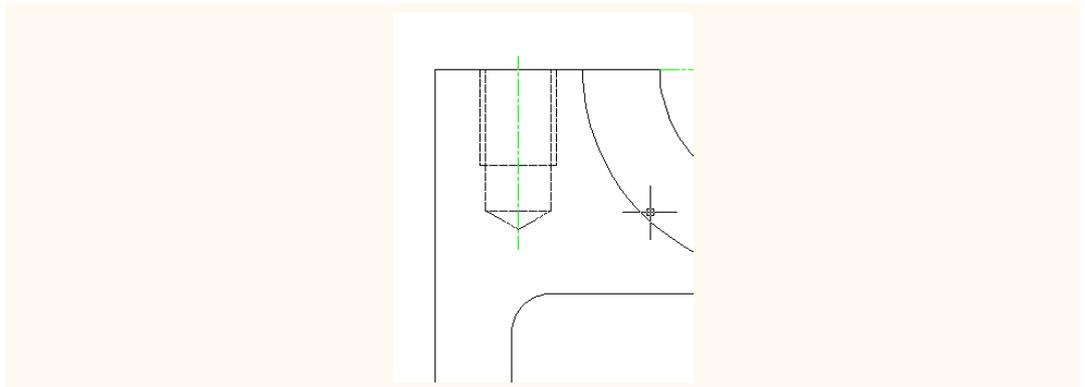
Zoom Dynamic

If you don't like the dynamic zoom command, there's always the scroll bars...

One of the most powerful of the zoom options is zoom-dynamic, it allows for zooming and panning at the same time. Type **Z**↵ **D**↵ or select the command from the zoom flyout (flydown?) shown in the figure.

"REGEN" can be abbreviated to "RE"...

Move the zoom window (the cursor!) over the lower half of the "bay" and press ↵. If the arc has turned into a series of straight lines then type **REGEN**↵ to re-generate the view.

The Array command

The array command ⊞ is used to copy one or more objects in either a rectangular or circular pattern. When a circular pattern is used the objects can be rotated; it can be an incredibly powerful command

Select **Modify - Array**, or type **ARRAY** ↵ or **AR** ↵. AutoCAD will open the Array dialog box.
Select rectangular array. Type in **1** ROW and **2** COLUMNS.
Set Column offset to **84** mm.
Click on Select objects icon.
Use mouse to select the previously drawn hidden hole and the centre line. Press ↵ to terminate object selection. Press **PREVIEW** button. If you are happy with the result press ↵. Otherwise press Escape key to edit until you are satisfied with the result.
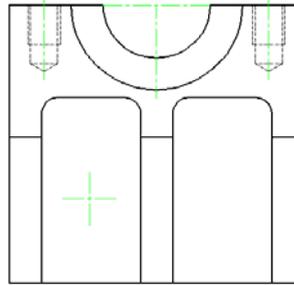
Make circle and copy it

Change current layer

Open Layer manager, select layer **DRAW** and make it current layer.

Circle can be drawn in many different ways. It can be specified by centre point and radius or diameter. It can be drawn on two or three point. Circle can also be drawn to be tangential to two objects.

Select **CIRCLE** icon from the draw toolbar or type **C** ↵ or **CIRCLE** ↵. Select the intersection between two centrelines in the left bottom corner of the drawing and then type **D** ↵ for diameter and **16** ↵ to specify the diameter of the hole.

Copy the circle and centre lines by use of copy command

Select **COPY** icon from the Modify panel or type **CO** ↵ or **COPY** ↵. Select objects to be copied: circle and centrelines. Press ↵ to terminate object selection. As the base point specify the centre of the circle. Type **50,0** ↵ to define the distance and the direction for copy command. Press ↵ to end Copy command.
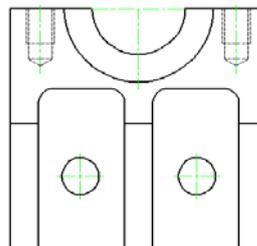
Finishing up

That's all for tutorial one! Save your drawing (press **Ctrl-S**) and then exit AutoCAD (select **File - Exit**).

Remember to log off before leaving.