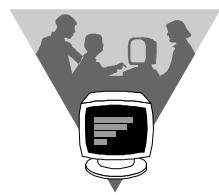




University of Edinburgh



Library Systems Department

Turbo CAD Manual

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1 Document Control

1.1 Amendment History

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1.0	4/3/99	Current and Initial Version (awaiting testing)	KM

1.2 Filename Path

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Section 1 - The File Menu

File|New... or <ctrl>+N

Opens a new drawing file.

File|Open... or <ctrl>+O

Opens a previously saved file.

File|Close

Closes the current file.

File|Save or <ctrl>+S

Saves the current file, providing a filename has already been assigned, if not the user is prompted to enter a filename.

File|Save As...

Prompts the user for a filename (no more than 8 characters), file location and file type (i.e. TurboCAD, AutoCAD, Template etc).

File|Save All

This command allows the user to save all open drawings at the one time, providing they have all been assigned filenames.

File|Libraries...

Displays a list of the available symbol libraries.

File|Summary Info...

The Summary Info command opens the Summary Info dialogue, in which you can record information that will be saved with your drawing file. There are several text fields which can be used to record information about your drawing, these are:

- Title
- Subject
- Author
- Keywords
- Comments
- Preview Graphic - Allows a graphic representation of the drawing to be previewed before opening the file. There 3 choices for the preview graphic:
 - None - No graphic.
 - Bitmap - Low detail.
 - Metafile - High detail.

File|Page Setup...

The Page Setup dialogue provides various features which allow the user to change the way in which the drawing is printed. There are 2 property sheets within the dialogue: *Paper* and *Layout*.

Paper enables the user to determine: paper & drawing sheet size, orientation (i.e. Portrait and Landscape), and several advanced features which can be found by clicking the *Setup* button.

The user may also check (✓) the following boxes for printing purposes:

Print Margins Print Grid Print Crop Marks Print Construction

The controls in the Scale control group let you scale the drawing so that it fits within the drawing sheet or choose a custom scale.

To scale the drawing so that it fits into the drawing sheet, click the *Fit* button. If the *Keep Centred* option is turned on, the drawing will be centred as it is fit into the drawing sheet. Click the *Reset* button at any time to return the scale that was set when you entered the *Page Layout* dialogue.

The *Layout* property sheet gives you direct control of the number of rows and columns (sheets of printer paper arranged vertically and horizontally). It also has functions for setting the printer paper margins, setting the height and width of the drawing sheet in either *World* or *Paper* co-ordinates, and relocating the paper in the drawing area.

File|Print...  or <ctrl>+P

This command will enable the user to print directly to the selected printer and make other selections, such as:

- Print Source - Choose Drawing (complete drawing sheet) or Selection (previously selected object on drawing sheet).
- Print Range - Select from All (print all of the current drawing), View (print a single view of the drawing), Range (print a range of pages, using the "from" and "to" box).
- Print Quality - Select either 600 DPI (high quality) or 300 DPI (lower quality).
- Copies - Number of copies you wish to print.

Print Preview 

Displays how complete page will look before printing.

File|Send

This command is used to send the drawing as an attachment of an e-mail message. This can be done via Microsoft Mail or Microsoft Exchange.

File|1,2,3,4

The numbers 1 to 4 represent the last 4 files that were used in TurboCAD. TurboCAD stores the path which leads to each of these file, which saves time.

File|Exit

Exits from TurboCAD.

Section 2 - The Edit Menu

Edit|Undo or <ctrl>+Z

Undo stores the previous state of the drawing before the most recent action. This enables the user to take “steps” back in the drawing or to rectify a mistake. The *undo* command in the menu can change to represent the most recent action (i.e. *Undo Cut* or *Undo Paste*).

Edit|Redo or <ctrl>+Y

Redo stores the previous *undo* and therefore allows the user to redo the action.

Edit|Cut or <ctrl>+X

The *cut* command cuts the select object out of the drawing sheet and places it into the clipboard, where it can be pasted elsewhere or in another software package (ie Word).

Edit|Copy or <ctrl>+C

This command copies any selected objects into the clipboard, from which they can be pasted elsewhere or in another software package.

Edit|Paste or <ctrl>+V

The *paste* command pastes any objects in the clipboard that have been *cut* or *copied* from a drawing or from another software package.

Edit|Clear

Clears the screen of the following:

- Selection
- All
- Construction
- All Construction

Edit|Select

This command allows the user to select any objects in the drawing area using the mouse to point and click or creating a window (click and drag) over the entity. To de-select an object in the drawing area, the user should move the cursor to any other part of the drawing area and click with the left mouse button.

Edit|Select All or <ctrl>+A

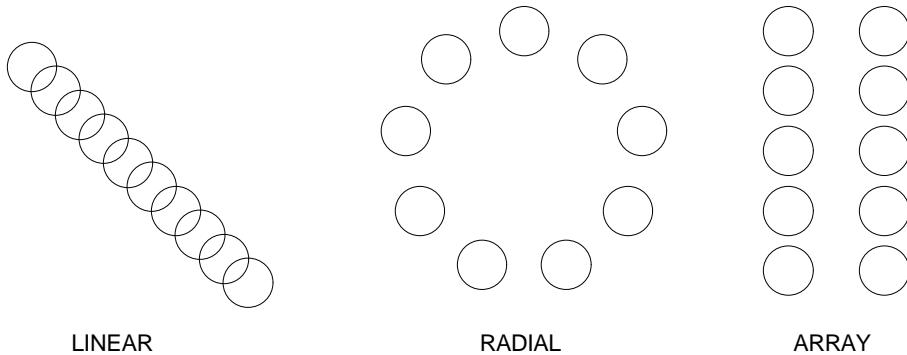
When the user chooses to *select all*, every entity contained in drawing will be selected in one group.

Edit|Select By

This command enables the user to select entities by their common features. The feature can be chosen from:

-
- Entity type - lets you select all the objects in the drawing that belong to specific types (i.e. Arc, Line or text).
 - Layer - selects all objects that exist on a specified layer of the drawing.
 - Attribute - selects all objects that share the same text in their attribute property.
 - Fence - Selects objects by drawing a polygon fence around them.

Edit|Copy Entities



The *copy entities* command allows the user to make single or multiple copies of entities and control their placement (see previous page - LINEAR / RADIAL / ARRAY).

Linear Copy - Makes copies along a line, specifying intervals between each copy and number of copies to make.

Radial Copy - Arrange copies in an arc, again specifying distance and amount of copies.

Array Copy - Copy entity into array of rows and columns specified by the user.

Fit Linear Copy Array Copy & Radial Copy - The user specifies the area for all the copies to fit into and TurboCAD fits them into the area using equal distances etc.

Mirror Copy - Create mirror image of selected entity over a line specified by the user.

Vector Copy - This command makes one copy of the selected entity and allow the user to specify the angle and distance of the copy from the original.

Section 3 - The View Menu

View|Redraw or <f5>

This command allows the user to “clean-up” the screen after editing has been done. This isn’t usually required as TurboCAD will normally automatically clean-up the drawing sheet.

View|Zoom

Zoom In  or <grey+> - Zoom in to drawing one step by position of cursor.

Zoom Out  or <grey-> - Zoom out from drawing one step by position of cursor.

Zoom Window  or <ctrl>+<home> - Click and drag a window over the area required for zoom.

Zoom Extents  or <ctrl>+<backspace> - This command enable the user to zoom to a level which views all objects in the drawing at the one time.

Full View  or <shift>+<backspace> - View the complete drawing sheet.

Printed Size  - View the drawing at its printed size.

Previous View  Return to the last view.

View|Pan to Point <ctrl>+<end>

This commands allows the user to select a point in the drawing to become the centre point of the view.

View|Vector Pan

This command lets the user to select the point on the drawing sheet to move (base point) and then select the required destination.

View|Named View

Once the user has used zoom and pan to arrive at the desired view of the drawing, it is wise to create a *named view* which saves the view in a list to be referred to later.

View|Create View

An alternative to the *named view* command is the *create view* command. Using this command, the user can select any part of the drawing using a click and drag window and the contents of the window can be saved along with the drawing, and given a name.

View|Previous View

This command will return to the previous view.

View|Grid

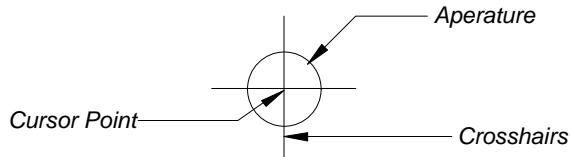
This is a toggle to display or hide the grid (a series of lines, dots, crosses) that divide the drawing sheet geometrically in order to assist with drawing technique (i.e. snapping to grid).

View|Crosshairs

This command also toggles between display or hide. *Crosshairs* enable more precise positioning of the cursor in relation to other graphic objects within the drawing sheet.

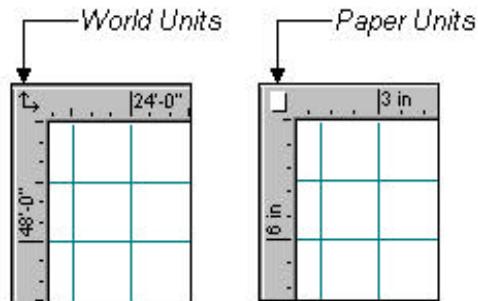
View|Aperture

As above. The aperture is a circle located around the cursor. An entity must be within this circle in order to be snapped to. When both aperture and crosshairs are displayed the cursor appears as follows:



View|Rules

Toggle for display and hide of rules. Rules are located on the left and top edges the screen and show the measurement in "world" and "paper" units. The toggle button for "world" and "paper" unit is located at the intersection between the top and left rules, and is represented as follows:



View|Status Bar

This is a toggle that displays and hides the Status Bar. The Status Bar is displayed along the bottom of the screen. It contains a brief status message, the Co-ordinate Fields, and a zoom percentage box on its right side.



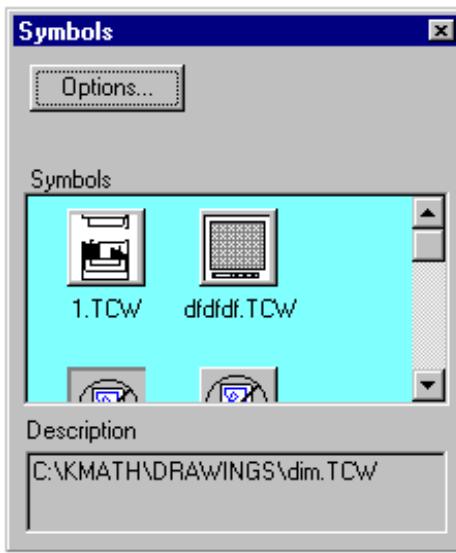
View|Edit Bar

The Edit Bar enables the user to create and edit entities by entering distances, angles, radius and circumference etc directly into the Edit Bar fields. Below is an example of an edit bar when the circle tool is selected.



View|Symbol Library...

The symbol library is a palette which displays libraries (i.e. windows folders) of TurboCAD files. The library displays the symbols as large/small icons or file names. When display method is by icons, the icon will be a small version of the drawing contained in the file linked to it. The symbols can be dragged on to the drawing sheet.



View|Blocks

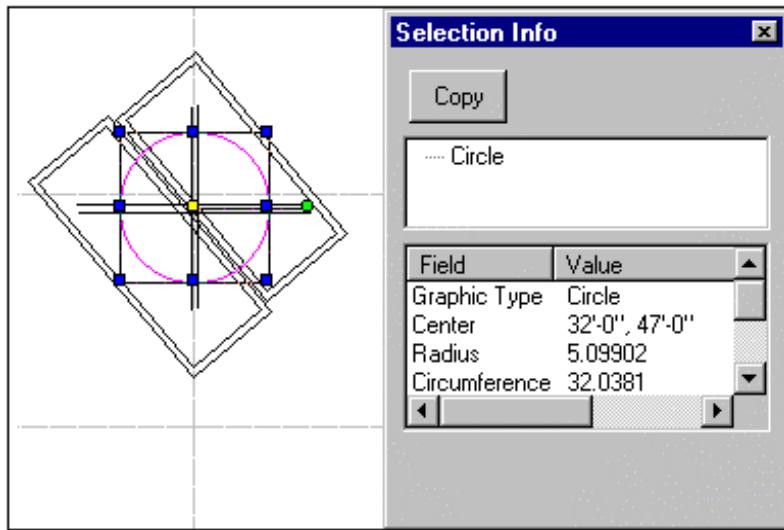
Blocks are similar to group, in that they consist of collection of entities brought together to create one object. Blocks are stored in a block library using the same methods as used in symbol libraries and can also be inserted on to the drawing sheet as well.

View|Toolbars

This command displays or hides the desktop property sheet, which will enable the user to display or hide certain toolbars.

View|Selection Info

Enables the user to display or hide the Selection Info palette. The information contained within the palette relates to the selected object, and informs the user of entity type, dimensions and location in the drawing.



Section 4 - The Insert Menu

Insert|Construction

Construction tools enable the user to put temporary lines and circles on the drawing to be used as guides only.

Angular Line - Construction line at an angle.

Horizontal Line - Horizontal construction line.

Vertical Line - Vertical construction line.

Centre and Point Circle - Circle where user selects centrepoint and perimeter.

Double Point Circle - Circle that passes through two point 180° apart.

Triple Point Circle - Circle that passes through three points.

Insert|Point

The user can choose from several different points: Dot, Star, Square, Cross and Circle. Dot is the only point that does not change size, whatever the zoom level. The dot is therefore particularly useful to use as a reference point in the drawing.

Insert|Line

This command enables the user to draw a single lines, multiple connected lines, polygon etc.

Single Line - Draws single lines.

Multiline - Draws connected irregular lines.

Polygon  - Draw a regular polygon with an arbitrary number of sides.

Irregular Polygon  - Draw a polygon with arbitrary segment lengths and angles.

Rectangle -  Draw rectangle.

Rotated Rectangle  - Draws rectangle by selecting base angle, then rectangle size.

Perpendicular Line  - Draws perpendicular line to an entity line selected by the user.

Parallel Line  - Draws a parallel line to an entity line selected by the user.

Tan Arc Point  - Draw a line tangent to an arc touching the arc's centre point.

Tan to Arc  - Draw a line tangent to an arc, with the lines second endpoint touching the arc.

Tan from Arc  - Draw a line tangent to an arc, with the lines first endpoint touching the arc.

Line 2 Arcs  - Draw a line tangent to two arcs, with each endpoint of the line touching one of the arcs.

Insert|Double Line

This tool is very similar to Insert|Line, except that another line will run parallel to single line. The tools within Insert|Double Line are as follows:

Single  - As with single line.

Multiline  - As with single line.

Polygon  - As with single line.

Irregular Polygon  - As with single line.

Rectangle  - As with single line.

Rotated Rectangle  - As with single line.

Perpendicular  - As with single line.

Parallel  - As with single line.

Insert|Circle/Ellipse

Used for drawing different styles of circles and ellipses.

Centre and Point  - Draw a circle with centrepoint and radius.

Concentric  - Draw a set of circles with the same centrepoint.

Double Point  - Draw a circle by defining two points on its perimeter.

Circle Tan to Arc  - Draw a circle tangent to another circle or arc.

Circle Tan to Line  - Draw a circle tangent to a line.

Triple Point  - Draw a circle by defining three points on the circles perimeter.

Tan to 3 Arcs  - Draw a circle tangent to three existing arcs or circles.

Ellipse  - Draw an ellipse by defining its bounding rectangle.

Rotated Ellipse  - Draw an ellipse rotated at any angle.

Ellipse Fixed Ratio  - An ellipse, with ratio of the major axis length to the minor axis length.

Insert|Arc

This command works using the same methods as the Insert|Circle/Ellipse command. But in addition to the usual steps, the user has to define the start and end angles of the arc.

Centre and Point Arc  - Draws a circular arc by selecting centrepoint and circumference point.

Concentric Arc  - Draws a set of concentric circular arcs with the same centrepoint.

Double Point Arc  - Draws a circular arc by selecting the endpoints of its diameter.

Arc Tan to Arc  - Draws a circular arc tangent to a circle or arc.

Arc Tan to Line  - Draw a circular arc tangent to a line.

Triple Point 1-2-3  - Draw a circular arc by defining (1) its starting point, (2) a point on its perimeter, and (3) its ending point.

Triple Point 1-3-2  - Draw a circular arc by defining (1) its starting point, (3) its ending point, and (2) a point on its perimeter.

Arc Tan to 3 Arcs  - Draws a circular arc tangent to 3 existing arcs or circles.

Elliptical Arc  - Draws an elliptical arc by defining its bounding rectangle.

Rotated Elliptical Arc  - Draws an elliptical arc rotated at any angle .

[Elliptical Fixed Ratio Arc](#)  - Draws an elliptical arc by setting its aspect ratio.

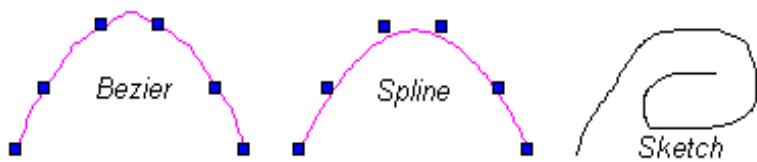
Insert|Curve

This tool allow the user to draw curves using three different methods:

[Bezier](#)  - Will connect two endpoints, whilst using others points in between as control points.

[Spline](#)  - Will connect two endpoint, whilst passing through all other points in between.

[Sketch](#)  - Freehand drawing tool.



Insert|Text

This tool allows the user to insert text anywhere on the screen simply by clicking the cursor at the appropriate location, typing the required text and pressing <Enter>. If the user requires to format the text, the View|Toolbars command should be selected and check (✓) the text format

toolbar. The text format toolbar enables the user to change the font, size, attributes (bold, italic, underline) and alignment of the selected text.

Insert|Dimension

These tools are used to display measurements of lines and angles, using leader arrows.

[Horizontal Dimension](#)  - Draw a linear dimension showing horizontal distance.

[Vertical Dimension](#)  - Draw a linear dimension showing vertical distance.

[Parallel Dimension](#)  - Draw a linear dimension showing absolute distance between two points.

[Rotated Dimension](#)  - Draw a linear dimension showing absolute distance between two points, with extension lines rotated to any angle relative to the line you are dimensioning.

[Datum Dimension](#)  - Draw a series of dimensions connected to angled leader lines.

[Baseline Dimension](#)  - Draw a series of parallel linear dimensions sharing a common baseline.

[Continuous Dimension](#)  - Draw a series of parallel linear dimension, each measuring a distance from the end of the previous dimension in the series.

Angular Dimension  - Draw a angular dimension showing the angle formed by two lines, by points on the perimeter of a circle or arc, or the angle formed by any two points relative to a defined vertex.

Radius Dimension  - Draw a dimension indicating the radius of an arc or circle.

Diameter Dimension  - Draw a dimension indicating the diameter of an arc or circle.

Leader Dimension  - Draw a simple line leader with label text.

Insert|File

This tool can insert another chosen TurboCAD or similar file into an open drawing.

Insert|Picture

This tool allows different picture files to be inserted into the drawing sheet, these file can be of various formats, including: .wmf, .bmp or .dib.

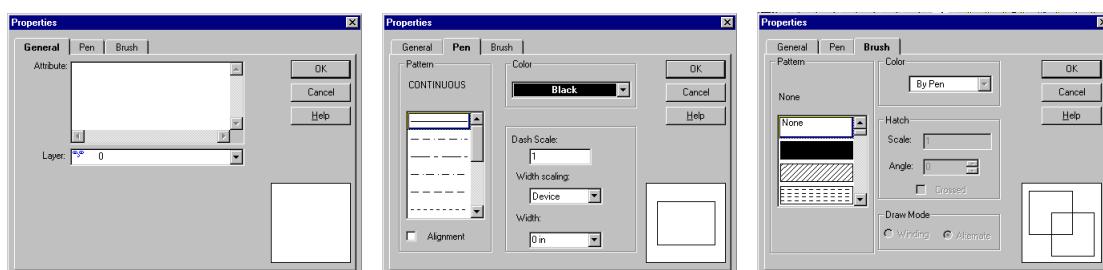
Insert|Object

Enables the user to insert Object Linking and Embedding objects from other applications into the drawing. The object can then be edited, within the TurboCAD drawing sheet using the appropriate applications tools (i.e. Excel).

Section 5 - The Format Menu

Format|Properties

This tool will enable the user to access the three property sheets (General, Pen and Brush) shown below:



Other property sheets may be added to these depending on the tool that is currently selected (i.e. double line, curve, point, text).

General - select drawing layers, edit text attributes.

Pen - edits patterns, colour, dash scale and width scaling.

Brush - edits patterns, colour, hatch scale/angle and draw mode.

Curve - edits number of segments and curve type.

Double Line - edits line separation, reference and line endcaps.

Point - edits point size, type and colour.

Text - edits font, size, style, colour, effects and justification.

Format|Create Hatch

This tool draws a hatched pattern over the selected object. This useful to display in the drawing where a section has be “cut away”.

Format|Create Block

This tool enables the user to define many object on the drawing as a block and save the gathered objects in the block library.

Format|Create Group

Similar to format|create block, the create group tool gathers all the selected objects to create one object.

Format|Explode

Break up an object into its constituent parts.

Format|Bring to Front

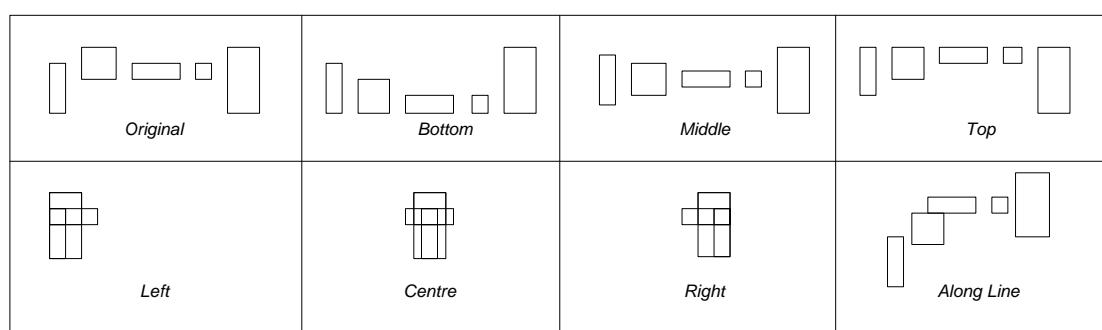
Moves the selected objects in the drawing to the front of the object stack on the appropriate layer.

Format|Send to Back

Moves the selected objects in the drawing to the back of the object stack on the appropriate layer.

Format|Align

The user can align objects in one of the following ways:

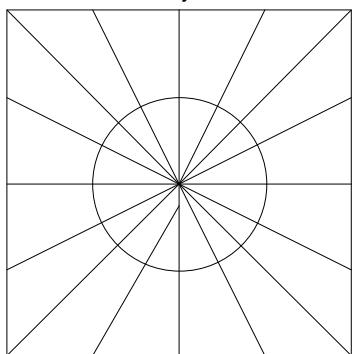


Section 6 - The Modify Menu

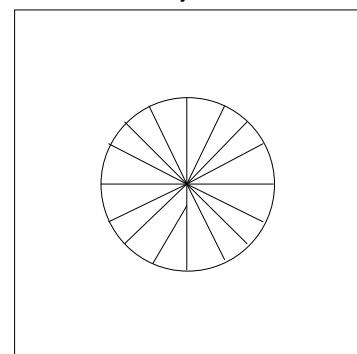
Modify|Object Trim

This tool enables the user to select an object as a cutting edge and then click on other entities around it to trim them, as below.

Before Object Trim



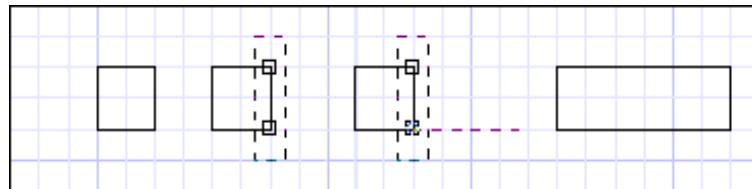
After Object Trim



The circle was used as the cutting edge and then each of the “spoke” areas outside of the circle were clicked on in order to produce the wheel effect.

Modify|Stretch

The stretch facility allow the user to select areas of the entity to be stretched to another point.

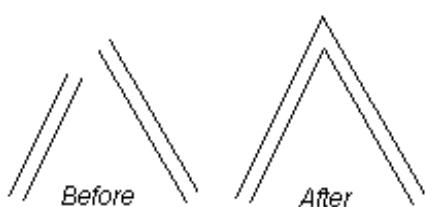


Modify|Split

This will split any given entity into two entities. The user can select for example, a circle and click the first and second cutting position, this will create two entities within the circle which can be edited separately.

Modify|Meet 2 Lines

This command enables the user to select two different entities (as in the example below) and meet their separate endpoints.



Modify|Chamfer

Use the Chamfer tools to join two line segments or double line segments with a flattened corner. You can also use the Chamfer Multiline option to chamfer all the corners of a multiline or polygon. A chamfered corner appears as follows:



When you activate a chamfer tool, the values you can use to calculate the chamfer appear in the edit bar.

The following table describes these values and their meaning. Note that point of intersection means the point where the two lines intersect, or the point where they would intersect if they were extended.

Value	Meaning
Distance A	The distance from the point of intersection to the chamfer on the first line clicked.

Distance B	The distance from the point of intersection to the chamfer on the second line clicked.
Length	The length of the chamfer line.
Angle	The angle of the chamfer line from the first line clicked.

Modify|Line Length

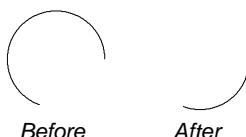
The line length tool will shrink or extend the length of a select line from either endpoint. However, if the line is part of a polygon, multiline or square the other areas of the shape will retain their relation to the modified line.

Modify|Shrink/Extend Line

This tool is particularly useful for more precise work. The user can extend or shrink a line to meet another entity. Instead of dragging the endpoint to the entity, the user only need to select the line (s)he wishes to extend/shrink and then select the entity to extend/shrink towards.

Modify|Arc Complement

The arc complement tool is used to reverse the appearance of an arc (i.e reverse the angle over 360°). For example, if an arc was



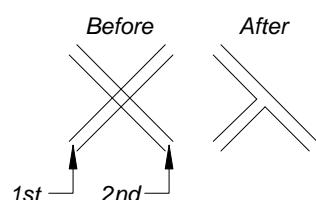
Modify|Fillet

Use the fillet tool to create a similar effect as the modify|chamfer tool, except that it will appear as a rounded edge. The user selects the radius of the fillet and the to lines to be filleted. The final result would be as follows:



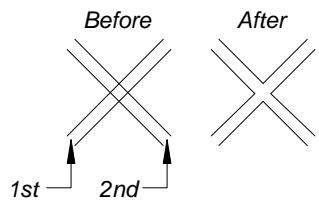
Modify|T-Meet 2 Double Lines

Use this tool to create a "T" intersection with two double lines. The first line clicked will form the "stem" of the "T" and the next line clicked will represent the "Bar" of the "T". In the example below, the 1st and 2nd indicates the mouse click movements.



Modify|Intersect 2 Double Lines

This tool will clean-up the "X" area of an intersection. To do this, the user should click on the first and second lines of the intersection. The example below shows the result.



Section 7 - The Mode Menu

Modes|Snaps

The tool allows the user to set the desired snap mode (i.e. the area in which (s)he requires the cursor point to snap to). Some of the snap modes can be combined with others, the following is a list of all available snap modes:

No Snap - Turns off all snap modes.

Snap to Vertex - Snaps to nearest end of line segment, endpoint of arc or corner of a polygon or multiline.

Snap to Nearest Point - Snaps to the nearest point of any entity.

Snap to Midpoint  - Snaps to the midpoint of a line.

Snap to Arc Centre  - Snaps to the centre of a circle or arc.

Snap to Quadrant Point  - Snaps to nearest quadrant point (at 0, 90, 180 or 270°) on the circumference of a circle or arc.

Snap to Intersection  - Snaps to the nearest intersection of two lines.

Snap to Grid  - Snap to the nearest grid point.

Ortho  - Constrain the angles of lines; the default settings constrain them to 0, 90, 180, or 270 degrees.

Magnetic Point  - Increase visual feedback, the cursor will be gravitate towards the snap point.

Modes|Coord System

The co-ordinate system menu commands enable you to change between absolute, relative, and polar co-ordinates, and to relocate the origin of the relative and polar co-ordinate systems.:

Absolute  - Change to absolute Cartesian co-ordinates, in which you define points by their distance from a fixed (absolute) point of origin.

Relative  - Change to the relative co-ordinate system, in which you define points by their X and Y distance from the Relative Origin point.

Polar  - Change to the polar co-ordinate system, in which you define points by their distance and angle from the Relative Origin point.

Relocate Origin  - Change the position of the Relative Origin point.

Section 8 - The Tools Menu

Tools|Program Setup

The program setup dialogue contains five property sheets for controlling TurboCAD program settings, these are as follows:

General - Specify template, read-only file access, user information, snap aperture, zoom factor, save, and advanced settings.

Desktop - Display and hide user-interface controls, control desktop colours and paper display.

Groups and Blocks - Set options for naming groups and blocks.

File Location - Specify directories for autosave, backup, template, and script files.

Colour Palette - Add new colours to the palette, modify existing colours, and delete colours from the palette.

Note: The above property sheets can be accessed directly from the tools menu. They are indented within the menu under the program setup command.

Tools|Drawing Setup

The drawing setup dialogue also contains five property sheets which are saved along with the drawing file, these are as follows:

Display - Set options for optimising redraw speed.

Grid - Choose the grid type and other grid options.

Advanced Grid - Control the major and minor grid and grid origin.

Symbol Libraries - View and maintain symbol libraries in current drawing or template.

Units and Scale - Set the unit of measure, the drawing scale, and related options.

Advanced Units - Set paper, text, and line width units.

Angle - Choose direction, units, and base angle for angular measurements.

Scripts - View and maintain scripts in current drawing or template.

Layers - Create new layers, modify existing layers, and delete layers.

Note: The above property sheets can be accessed directly from the tools menu. They are indented within the menu under the program set-up command.

Section 9 - The Window Menu

Window|Cascade

This tool allows windows to overlap in a manner that the title bars will still be displayed.

Window|Tile

This tool allows the open windows to be displayed all the same size and side by side in a tile formation.

Window|Arrange Icon

Arranges the minimised icons along the bottom of the screen.

Window|Close All

Closes all open windows.

Changing the Active Window

Makes a chosen drawing active.

Section 10 - The Help Menu

Help|TurboCAD Help Topics 

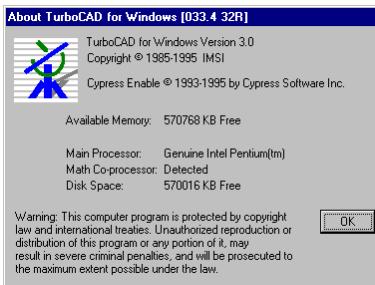
This command displays a dialogue with a contents list of help topics for TurboCAD.

Help|Tip of the Day

TurboCAD help has a small database of tips for using TurboCAD. A tip can be shown on startup by checking the box marked "show tips at startup".

Help|Tutorial

Within the TurboCAD package there are several tutorial sessions and a quick tour which can be accessed within the help|tutorial menu.



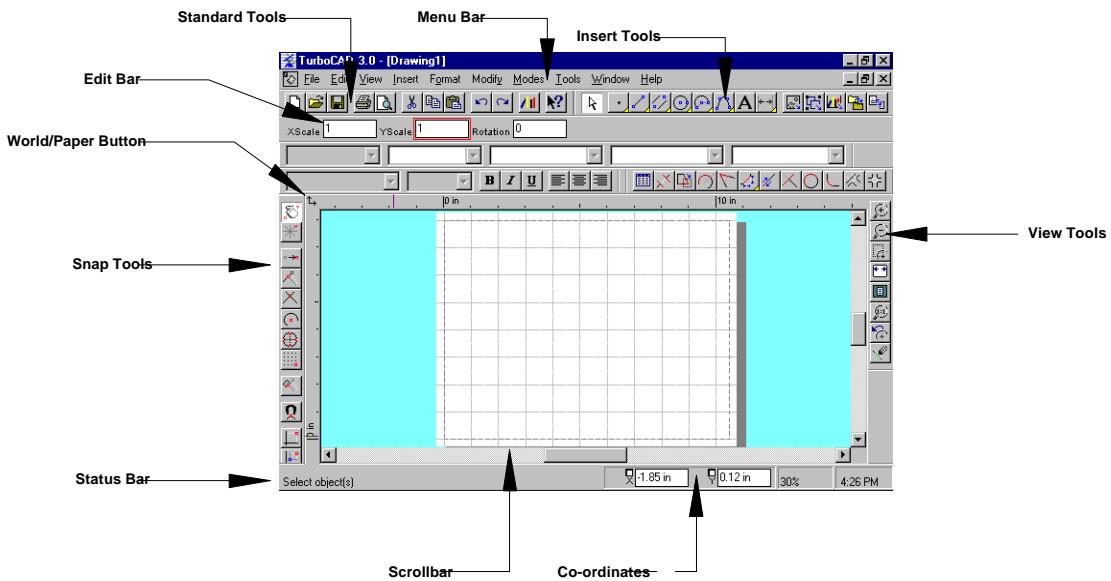
Tutorial No.1 BASIC DRAWING TECHNIQUES

Starting TurboCAD

To start TurboCAD double-click on the  TurboCAD 3.0 icon within the IMSI folder. This will start the TurboCAD application and then present you with a list of templates to choose from in the *Create from Template* dialogue. Choose the template marked Tutorial1 (default).

The Desktop

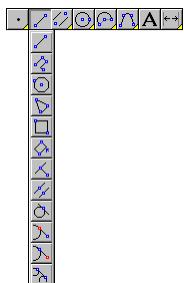
Once you have select the correct template from the dialogue, the TurboCAD desktop will appear in front of you. This will consist of several menus, tools, bars and other normal TurboCAD features. You will also see a blank drawing area for you to begin. The desktop appears as below:



Using Flyout Toolbars

Fly out toolbars are sets of buttons which flyout when you hold the mouse button down whilst the cursor is over a tool button. A tool that contains further “fly out” tools is marked with a small yellow triangle at the bottom right of the button. Try for yourself:

- [1] Place the mouse cursor over the single line tool .
- [2] Hold down the left mouse button and the following will appear allowing you to select a further tool.



Drawing

Now we can now begin our first drawing.

- [1] Check that the no snap button  is highlighted before commencing.
- [2] Select the Single Line tool  from the Insert toolbar (alternatively, select Insert|Line|Single from the menu bar).
- [3] Now, using the mouse, place the cursor anywhere in the drawing area of the desktop. Click once on the left mouse button where you desire to start your line. This will define the starting point.

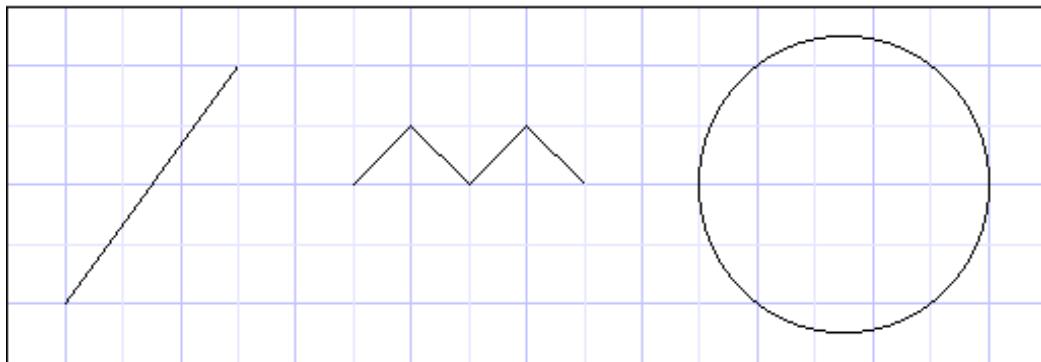
-
- [4] As you move the cursor about the drawing area, you will see a line connects the starting point of your line with the cursor. When you find the area you wish to place your finishing point, click once again on the left mouse button and you have a complete line.

Now draw a multiline.

- [1] Highlight the Multiline button  or select Insert|Line|Multiline from the menu bar.
- [2] Click on the left mouse button to select starting point and then select a 2nd, 3rd and 4th point elsewhere in the drawing area. You can complete your multiline by pressing the <enter> key after the last point is selected or you can click the right mouse button on the drawing to reveal the “local” menu and choose Finish.

Drawing a circle.

- [1] To complete the drawing, we will draw a circle. First, select the Circle Centre & Point tool  in the Insert toolbar (alternatively select Insert|Circle/Ellipse|Centre and Point).
- [2] Now select the starting point (centrepoint of the circle) and move the cursor. As you move the cursor around the drawing area, you will notice that the circle will shrink and extend according to your movements. TurboCAD is waiting for you to select your finishing point which will be the circumference point, click the mouse button where required.
- [3] You have now completed your first drawing, it may look something like the example below:



Making Mistakes

Should you make any mistakes, you can take a step back in your drawing by clicking the undo button  or selecting Edit|Undo. Repeating this action will take you back through each action made. However, should you change your mind after you have selected Undo, you can reverse that decision by clicking the Redo button  Edit|Redo. Sometimes, when we make movements within TurboCAD, the drawing sheet is left marked by previous entities which are normally automatically “cleaned up”. To clear these marks, the user can select the Redraw button  Edit|Redraw.

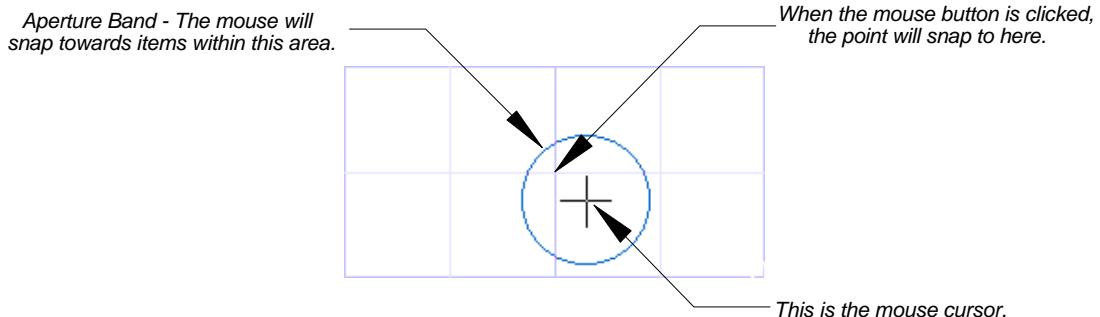
Note: It's always a good idea to give your drawing a filename as soon as you start! It is also good practice to save your work frequently (i.e. every 10 minutes).

Clearing the Drawing Sheet

Before we begin the next exercise, we can clear the drawing sheet. To do this, select Edit|Clear>All.

Using Snap Modes

Snap mode enables you to draw with more precise movements. “Snap” means that the cursor is attracted to certain pre-selected objects on the drawing sheet (i.e. gridline or other drawing entities). The cursor will “snap” towards the chosen object as long as it is within the aperture line. The example below shows how the aperture band works:



This example shows the mouse cursor being positioned near to the grid intersection. When the mouse button is clicked in Snap-to-Grid mode the cursor point will snap to the intersection of the gridlines.

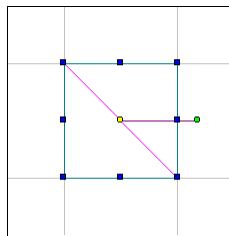
Draw a Line Using Snap-To-Grid Mode

- [1] Select the Snap-To-Grid tool  and the Single Line tool .
- [2] Place the mouse cursor near an intersection of two gridlines and click the left mouse button. The first point of the line will appear at the intersection between the two gridlines (as in the previous example).

Deleting an Object

You can delete any object on the drawing space at any time. We can try this with the line you have just drawn.

- [1] Choose the Select Tool  from the toolbar, this allow you to click on any object for modification purposes.
- [2] Click on the line you have just drawn in the previous exercise. The line should now be surround by a “selection box” which means modification can begin. The following displays how the line will now look:



- [3] Now press the <delete> key - it disappears.

Remember: You can remove the Snap to Grid tool by pressing it again, so that it is not highlighted.

Coordinate Fields

This exercise lets you take advantage of the precise drawing functions available within the TurboCAD desktop.

- [1] Select the single line tool .

-
- [2] Now, either click in the coordinate field box x or press **<ctrl>+<R>** to activate the x box. The co-ordinate boxes are position in the bottom left section of the desktop.
- [3] Now enter **3 m** and press **<enter>** to accept the new value for X Co-ordinate.
- [4] Press the **<tab>** key to move to the Y box and again type **3 m**.
- [5] The first point of your line will be selected and a moving line appears waiting for the second point to be allocated on the drawing sheet **DON'T SELECT THE SECOND POINT YET!**



The Edit Bar

Having selected the first point of the line in our previous exercise, its now time to create the complete line using the edit bar. The edit bar is located at the top left section of the desktop.

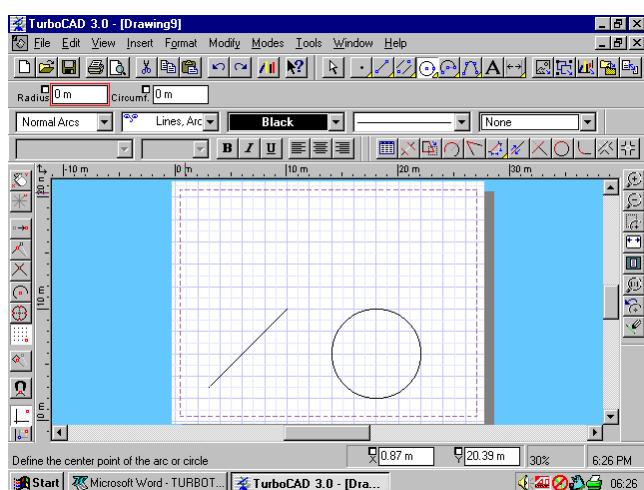


- [1] Either click on the edit bar or press **<ctrl>+<E>** to activate the edit bar.
- [2] Enter **10 m** into the length box and **45** into the angle box. This will draw a 10 m line at a 45° angle from the starting point. Your line is now complete.

Drawing a circle with the Edit Bar

We can also draw other objects using the edit bar.

- [1] Select the circle - centre and point tool .
- [2] Activate the Co-ordinate field box and enter the following **x=18 m** and **y=6 m**.
- [3] Now once again activate the Edit Bar and you will notice that the field names have changed from Length and Angle to Radius and Circumference. Enter **4 m** into the Radius field and notice that the Circumference has changed automatically to **25.13 m**, press **<enter>** to accept the values. You can also enter a value in the Circumference box and the radius will automatically change to suit. Your drawing should now look like this.

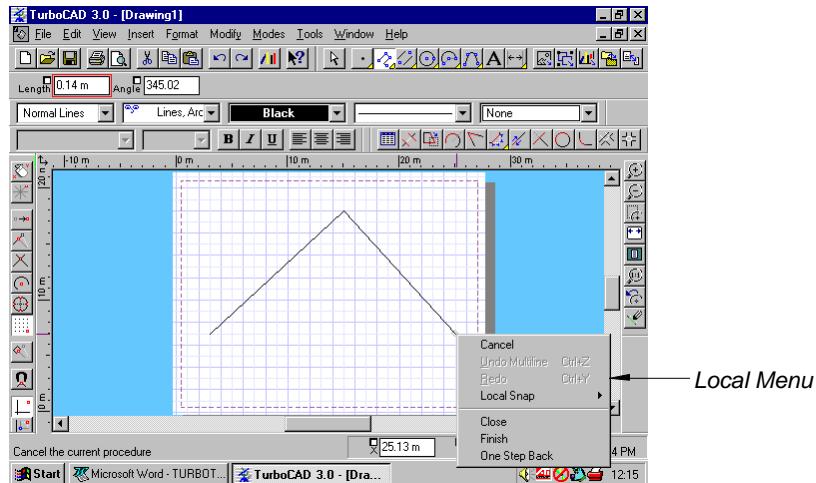


The Local Menu

Before we commence the next exercise, clear the previous objects on your drawing area using the Edit|Clear|All command.

The object of this exercise is to learn more about local menus.

- [1] Make sure the Snap-to-grid tool  is selected.
- [2] Now select the Multiline tool  from the appropriate “fly-out” bar. The multiline tool enables you to draw a continuous line selecting different points in its course.
- [3] On the drawing area, click a point at the bottom left, then select another point in the middle and now select a third point in the bottom right of the drawing area.
- [4] At this stage, click the right mouse button and the local menu should appear as follows:



- [5] From the local menu, choose finish to complete the object. You can select as many points as you wish for the multiline before choosing finish.
- [6] You may have noticed that there are other options in the local menu, for example Close. The Close command enables you to close the starting point and the end point, creating one object. Try going through steps 1 to 4 again and this time select close instead of finish. The result should be a triangle.
- [7] The options available are:

Cancel - Cancel the multiline.

Undo Multiline - This option is only available once the multiline has been finished/closed.

Redo - This option is only available after Undo Multiline has removed the multiline.

Local Snap - This command allows a snap mode for a “one-off” movement.

One Step Back - Step back to the last point selected on the multiline.

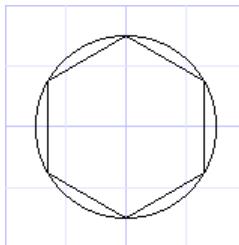
Note: The local menu may not always contain the same options, it depends entirely on what tool is being used.

Using the Local Menu for “One-Off” Snap Modes

As you will have seen in the previous exercise, the local menu contains a sub-menu for snap modes. There are various snap modes available and when selected in a local menu, they will only be used for one action and then return to the snap mode selected from the snaps toolbar on the left of the screen. This is particularly useful when the user wishes to draw using the snap-to-grid mode all the time, but requires to snap to an intersection for one movement only.

We can practice using these snap modes from the local menu in the following exercise:

-
- [1] Clear all the previous work using **Edit|Clear|All**.
- [2] Make sure that the **Snap-to-Grid** tool is **not** highlighted.
- [3] Choose the **Circle Centre & Point** tool .
- [4] Click a point at the centre of the drawing area and click the second point of your circle anywhere else in the drawing area to give you a circle.
- [5] Now, we can choose the **Zoom Extents** tool  from the right side of the desktop. Alternatively, you can select this command from the **View|Zoom|Extents** menu.
- [6] The zoom extents tool allows you to zoom in as far as possible, but still allowing you to view all of your drawing. The circle should now take up the whole viewing area.
- [7] We are now going to use the local menu snap modes to draw a polygon precisely inside the circle. Select the **Insert|Line|Polygon** tool  and place the cursor in the drawing area. Click the right mouse button to display the local menu.
- [8] Select **Local Snap|Arc Centre** and click anywhere on the perimeter of your circle. The first point of the polygon should appear exactly in the middle of the circle.
- [9] Click on the right mouse button once again and select **Local Snap|Quadrant Point** and click near the top of your circle. The second point should have snapped exactly 90° at the top of the circle, giving a precisely drawn polygon inside a circle. Your drawing should now look like this.

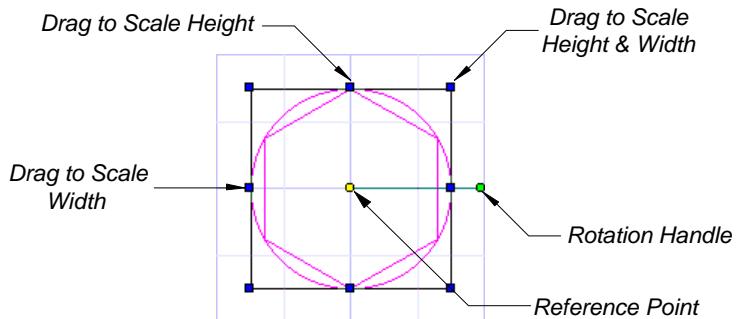


Selecting an Object

You can select an object using the **Select Tool** .

- [1] Click on the select tool - a white arrow cursor appears.
- [2] Click on any part of the polygon, a selection box should then enclose it.
- [3] Now hold down the **<shift>** key and click on the circle, it is now highlighted as well.

Your circle and polygon should now look like this:



The above explains the purpose of each component of the selection box.

Note: You can also select more than one object by using the mouse and dragging a window over all objects.

Moving a Selection

- [1] Click on the yellow reference point in the middle of the selection you have just made.
- [2] Now you will see a square with a four arrow cursor in the middle.
-

-
- [3] Use the cursor to place your selection elsewhere.

Rotating a Selection

Having select the circle and polygon, we can now rotate them as with any object selected in this way.

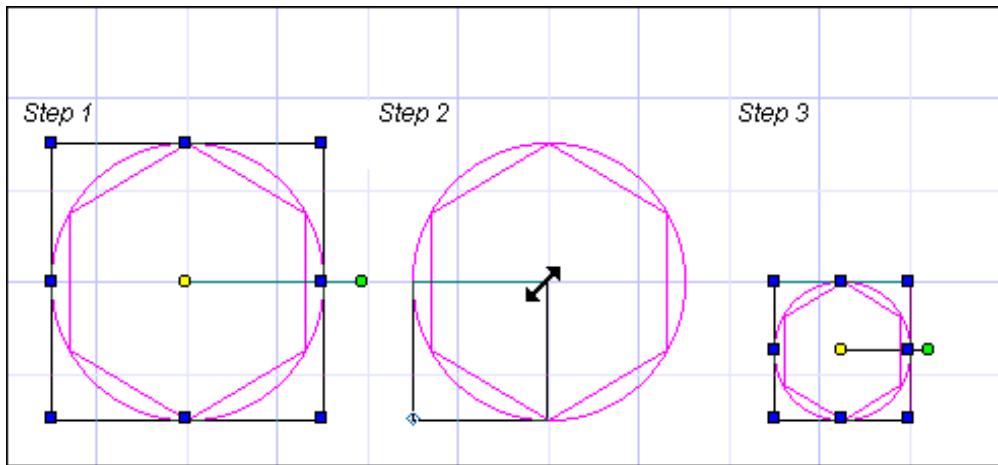
- [1] Place the cursor over the the green rotation handle.
- [2] Hold down the left mouse button and move the handle clockwise or anti-clockwise towards the desired point of rotation.
- [3] Alternatively, you may enter a value into the Rotation box of the edit bar.



Scaling the Selection

The selection can be scaled using the manual (mouse) method or the edit bar.

- [1] Try experimenting by clicking the top right pixel (drag to scale - height and width) of the selection box and dragging in an upward right direction, releasing the mouse button when you are satisfied with the new size.
- [2] You can scale the selection by height and width separately using the drag to scale width and drag to scale height pixels (see diagram on previous page).
- [3] When using the edit bar to scale, select the object you wish to scale and enter values in the X Scale and Y Scale boxes. A minus figure can be entered for reducing the selection size. An example of this exercise is shown below:



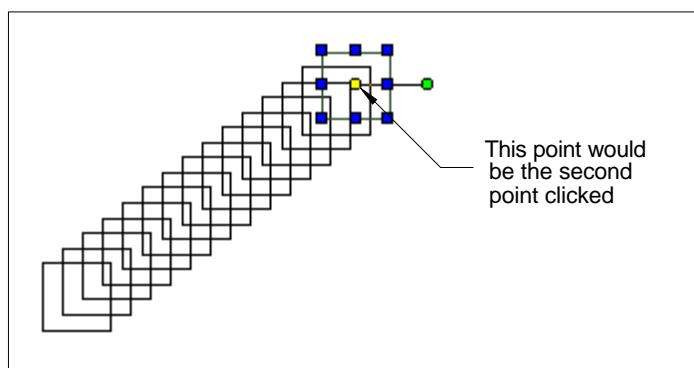
Editing The Selection

As with most windows applications, TurboCAD allows you to copy, cut and paste. There are also a few special copy commands, we shall try a linear copy as follows:

- [1] First, select the rectangle tool .
- [2] Now enter the following value in the co-ordinate bar **X=1 m** and **Y=1 m**.
- [3] You can now either click and drag a rectangle from this point approximately **3 m x 3 m** or you can click on the Edit Bar boxes: Size A & B and enter the value **3 m** in both the boxes to produce a box.
- [4] Now, select the square as the object that you are going to copy, then select from the menu **Edit|Copy Entities|Fit Linear**. The edit bar will appear as follows:

XScale	1	YScale	1	Rotation	0	XSize	5.45 m	YSize	5.81 m	Sets	2
--------	---	--------	---	----------	---	-------	--------	-------	--------	------	---

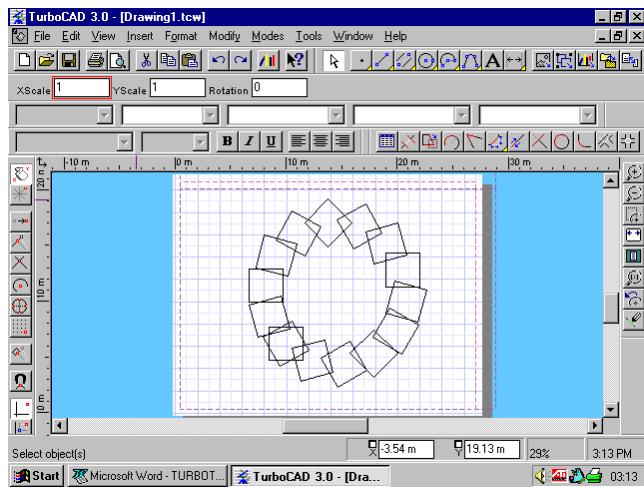
- [5] Click in the Sets boxes of the Edit Bar and type **15 DON'T PRESS ENTER AFTER THIS.**
- [6] Move the cursor on to the drawing area and click once to activate it.
- [7] You will now see that a dashed line follows the cursor round the screen, click on a point at the centre of your drawing area.
- [8] You now see that Turbo CAD has copied original square and reproduced it 14 times to make 15 sets and has fitted the sets equally between the middle point of the first square and the second point that you clicked in the drawing area. The second point that was clicked in the drawing area represent the middle of the last square (see below).



Radial Copying

We can also copy the square using Edit|Copy Entities|Fit Radial.

- [1] Select Edit|Clear|All to clear your previous drawing from the drawing sheet.
- [2] Now repeat steps 1 to 3 of the previous exercise (editing the selection).
- [3] Highlight the square that you have created and select Edit|Copy Entities|Fit Radial.
- [4] Now we shall enter the co-ordinates for the radius, press **<ctrl>+R** to activate the coordinate bar and type the value **7 m** into both the X and Y boxes and the press **<Enter>**.
- [5] We then need to activate the edit bar, press **<ctrl>+E**.
- [6] Do not enter any values into the Xscale, Yscale or Rotation boxes, instead click on the Sets box.
- [7] Type **15** in the Sets box and press **<tab>** to move on to the next box.
- [8] Type **360** in the Angle box and press **<tab>** to move on to the next box.
- [9] Type **15** in the +Rot box and then press **<enter>** to enter all these values. Your copy is finished and there are once again 15 sets displayed in a radial format and rotated at 15° after each copy.
- [10] You will notice that part of the drawing is not on the drawing sheet, we can rectify this by dragging a window over all the squares, then click the yellow reference point at the centre of the selection and move to a position at the centre of the drawing sheet.
- [11] Your desktop should now look as follows:

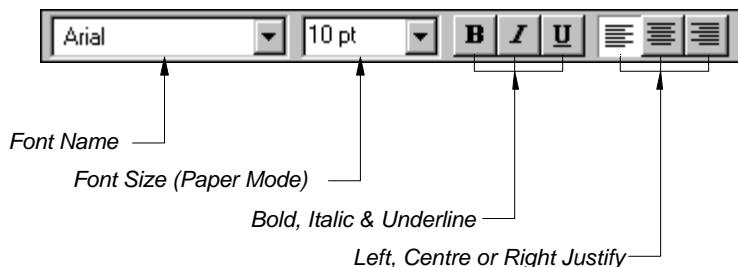


Adding Text To Your Drawing

You will often wish add text to the different objects that appear on your drawing, TurboCAD provides a text tool which allows you to format text using much the same methods as you may have used in wordprocessing or spreadsheet packages.

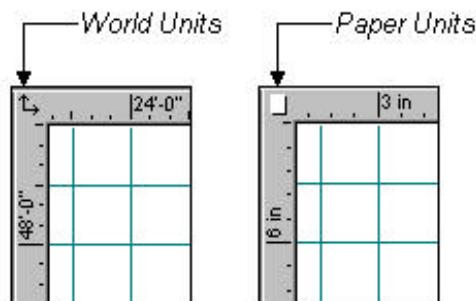
To add text to the drawing created in the previous exercise:

- [1] Click on the Insert|Text tool  and you will notice that a text format bar (displayed below) will appear under the edit bar.



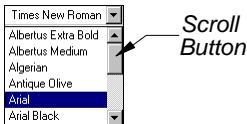
The text format bar allows you to define what the text shall look like before you begin or to make changes to a selection of text that you have highlighted.

- [2] Ensure that the world/paper button located at the intersection of the two rulers has paper highlighted, i.e.



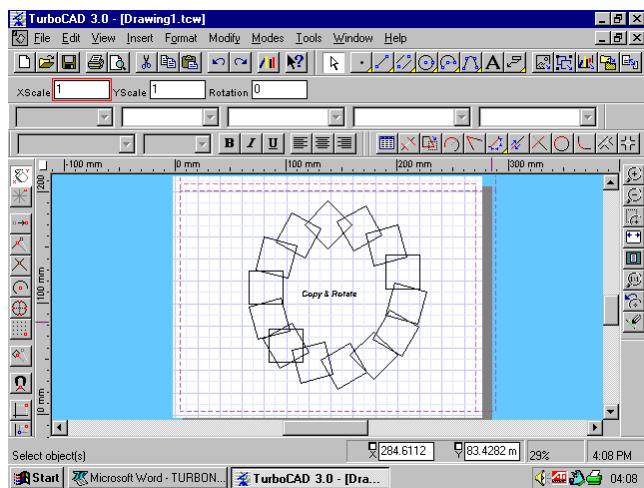
- [3] Now click somewhere inside the “circle” of the drawing that you created previously. The cursor should appear flashing. If you can't see the cursor flashing properly, try pressing the plus button on the numer area of your keyboard - this will allow you to zoom in to the text.

-
- [4] Now click on the scroll button to reveal the available fonts and select the font *Arial*.



- [5] Also click and change the font size to 28 pt.
- [6] Now highlight the Bold and Italic buttons **B I U**.
- [7] Type **Copy & Rotate** and press <enter>.
- [8] Your item of text may be out of the centre of the drawing or overlapping other areas. If you wish to move it, then select and reposition in the centre using the method explained earlier in **Moving a Selection**.
- [9] Your drawing should now look like the one on the follow page.

Remember: You can see the full drawing sheet by clicking the Zoom|Full View button .



Printing Your Drawing

There are many advanced methods of printing in TurboCAD which allow you over tiled areas etc. For the moment we will try a simple print procedure in order to print out our drawing.

- [1] Select Print... from the file menu and the print dialogue box will appear.
- [2] In the Print Range area of the print dialogue, click on the *All* option as this should print the complete drawing.
- [3] Now check the Fit on 1 Page option at the bottom of the dialogue.
- [4] Click on Print Setup... and select a printer name from the pull down menu option.
- [5] Now select A4 from the paper size pull down menu.
- [6] Click on the landscape orientation before clicking on OK to return to the print dialogue.
- [7] Once you are satisfied that the correct options have been selected, click on the OK button and your drawing should begin to print.

Tutorial No.2 **DRAWING A ROOM PLAN**

In the last tutorial we learned how to operate some of the basic tools concerned with TurboCAD. In this tutorial we shall look at how to:

- Open the plan drawing template.
- Create a room plan.
- Create furniture as symbols.

Opening The Plan Template

You will remember that in the last tutorial we used the template - *Tutorial1*. In this tutorial we will activate the template called *RoomPlan*. This template has been created as fast and easy basis to begin drawing plans of any kind. The *RoomPlan* templates is set out as follows:

	
Drawing Title	
Department	
Date	
Drawing Number	
Filename	
Drawer Name	
LEGEND	
Scale:	
Signature:	

As you can see from the above display, the details of your drawing shall appear on the right side of the template.

Drawing Title - explains what type of drawing you have done (i.e. Wiring, Fire Alarms etc).

Department/Level - this shows the area (i.e. Systems, Ground Floor).

Date - the date the drawing was last modified.

Drawing Number - a reference number (i.e. UNI-WIR-001/A).

Filename - The TurboCAD filename for the drawing, including full extension (i.e. 1stfloor.tcw
.tcw is the filename extension).

Remember: *The first part of the filename can be no more than 8 characters long (not including extension).*

Drawer Name - Name of the person(s) who created the drawing.

Legend - This area explains what symbols on your drawing mean (i.e. ☎ could mean a telephone box on your drawing).

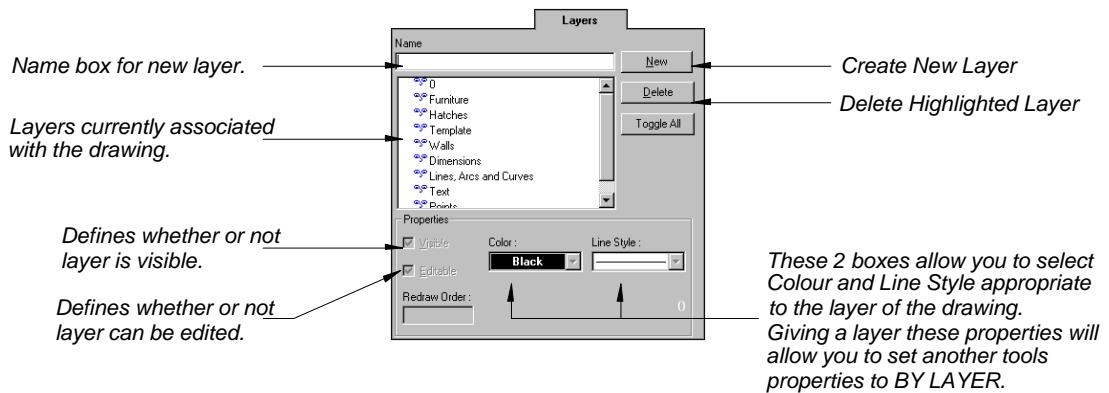
Scale - Scale of the drawing and units of measurement (i.e. 1:100 - 1cm=1metre).

Signature - Signature of drawer.

Creating Layers

Layers in your drawing are virtual levels, which are similar to acetate sheets in conventional drawing methods. Where the traditional drawing person would place acetate sheets, over each other to produce a complete drawing, we can also do this using layers. To create a new layer, do the following

- [1] Select Tools|Drawing Setup|Layers and you will see the following dialogue displayed:

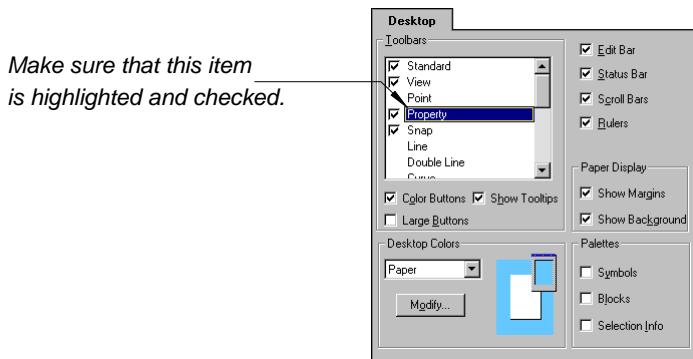


- [2] Now enter the name you wish to call the layer, for this exercise we shall be creating an office, so call the first layer "walls" and click on New and then OK.
 [3] You have now created your first layer on which you shall draw the walls, doors and stairs of the building plan.

Viewing the Available Layers within the Drawing

There are now several layers within your drawing including the one that you have just created, to view these:

- [1] Select View|Toolbars.... and you will see the following dialogue:



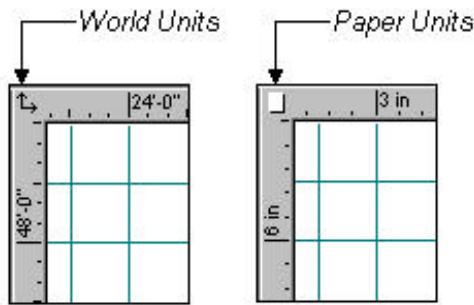
- [2] Click on *Property* in the Toolbars section of the dialogue and click *OK*. The toolbar appears on the drawing sheet or another area of the desktop.

Remember: You can move the toolbar to any position by clicking on it and dragging to new position.

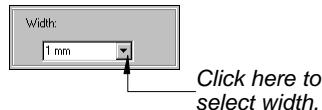
Creating an Office Plan

Now that we have opened the *RoomPlan* template and selected a new layer (walls), we shall begin drawing the first stage:

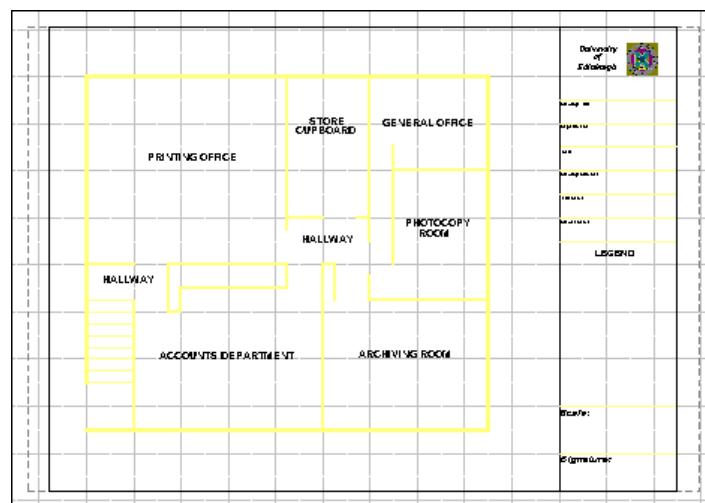
- [1] Before we start, select the snap-to-grid mode . Ensure that the world/paper button located at the intersection of the two rulers has the world units symbol highlighted.



- [2] RIGHT-Click on the insert rectangle tool to access the *Properties* dialogue for that particular tool.
- [3] Click on the *Pen* section to reveal pattern, colour and dash scale etc.
- [4] Now click on the downward arrow to the right of the width box and select **1mm**.



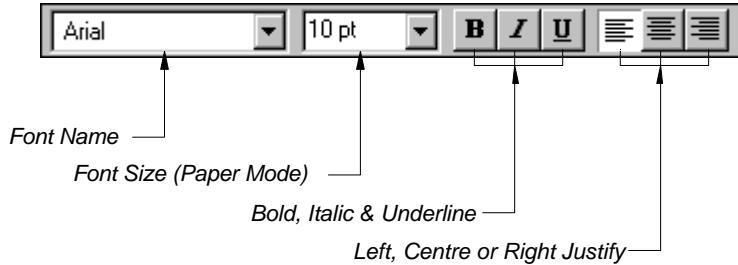
- [4] As in Tutorial 1, we shall define the first point of our wall using the co-ordinate bar situated at the bottom right of the screen. Press **<ctrl>+<R>** and the x axis box should be highlighted - type in the following values **x=2 m** and **y=2 m** and this will begin your line at the bottom left section of the screen.
- [5] Now change to the Edit Bar by pressing **<ctrl>+<E>** and enter **Size A=19 m** and **Size B=17 m** and then press enter.
- [6] You should now see a large box in the centre of your drawing which will represent the outer wall of the imaginary building. We can now continue with the construction of the internal walls. Click on the insert single line button to highlight it and then right click on the same button to display the properties. Repeat steps 3 and 4 and select **0.7mm** in the width box.
- [7] Now with the single line tool you can begin drawing your own building/floor layout of internal wall (at this stage there is no need to use scales in your drawing as it is only a practice session). Set your drawing out similar to the one below:



Drawing Details

In order to enter text into the drawing detail boxes (right area of the drawing sheet) you should:

-
- [1] When in entering text click on the world/paper icon and highlight paper. This allows you to see the font size in a familiar way.
- [2] Click on the text tool  to insert text into your drawing.
- [3] Now click the cursor on any area of the drawing details, i.e. the drawing title (the flashing cursor indicates your desired text insertion point).
- [4] You should see the text edit bar displayed on the toolbar area of the desktop.



- [5] You should check that your text bar displays the following:
- [6] Font Name = **Arial**
- [7] Font Size (Paper Mode) = **14 pt**
- [8] Bold, Italic, Underline = **Off**
- [9] Alignment = **Left**

Remember: You can change these settings if you wish.

- [10] You can begin typing your drawing title. Once you have typed the drawing title, press the <enter> button.
- [11] If the text is not exactly where you wish it to be, highlight it and move it around. To move the text to more exact position you should de-select the snap-to-grid  button first. Your finished details should look something like this:



Leave the legend for the moment as it will be explained in more detail later in the tutorial.

Creating Block

A block is an object that is used frequently on a drawing to represent an object. For example, on an electrical drawing, a socket may be represented by a block that looks something like this :- 