

## 11. DIMENSIONS

Please read the Dimensions section of the TurboCAD Manual as well as this document. Here, I deal only with some topics that need special explanation, and not all types of dimension. First, we need to fully understand the vital issue of scaling of dimensions. Then we will look at organising the numerous option settings.

### Model-related Objects & Paper-related Objects

You need to always recognise that there are two distinct types of objects that can be drawn in CAD. One type is objects that are drawn to represent the real-world objects being described by the drawing. They may be a building, walls, etc, a manufactured object or a bridge, etc. In CAD you draw all those things in Model Space full size. That is, the numbers representing lengths are the values of the actual built objects in the chosen units, usually millimetres anywhere except America.

The other type is objects that do not exist in the real-world where the described objects will be constructed. These are annotations in text and dimensions mainly. They only have any existence on the paper on which the drawing will be printed, and their magnitude is related to the size of that paper. As well as text items and dimensions there may also be title block boxes and sheet borders in this category.

We can describe the 'real' objects as "Model-related" and the text, dimensions and title blocks, etc as "Paper-related".

### Scale issues: in Model or Layout

When we draw to represent real objects of some substantial size we know that we will have to print them to some scale in order to fit them on the paper. So if we are drawing a house we may intend to print it on an A3 sheet at 1/100 scale. We also know that we want general text notes and dimension figures to be say, 3 mm high, and we want dimension arrows to be perhaps 2 mm long, etc. Those sizes for text and dimensions are what we want on the printed paper.

If we draw text and dimension figures on the model drawing of our house at the size we want them to be on the paper, when we scale-down the Model Space drawing at 1/100 to fit on the A3 paper, the text and figures will be scaled down by the same amount and will therefore print as 0.03 mm high text – just about invisible!

So to achieve the desired result, text and dimension figures and dimension arrows etc, need to be drawn 100 times bigger than we want them on paper if we will be printing at a 1/100 scale.

On the other hand, if we draw the text and dimensions in Paper Space, where we always print at a scale of 1:1, we must draw the text and dimensions exactly the size we want them on paper. The view of our Model Space drawing appears in Paper Space through a Viewport and we define that Viewport to have the desired scale, so that our drawing shows up in Paper Space scaled down 1/100 in this case. (See Chapter 8: *Model Space & Paper Space*.)

We can choose to draw text and dimensions in either Model Space or Paper Space, but we have to remember to manually fix the 'up-scaling' of them if placed in Model Space to allow for the print scaling.

The 'up-scaling' of dimensions placed in Model Space is done by the **Dimension Size Scale** property set in the Dimension Properties. It will need to adjust by trial-&-error.

For dimensions placed in Paper Space, **that property must always be set to 1** and then we don't have to worry about scaling.

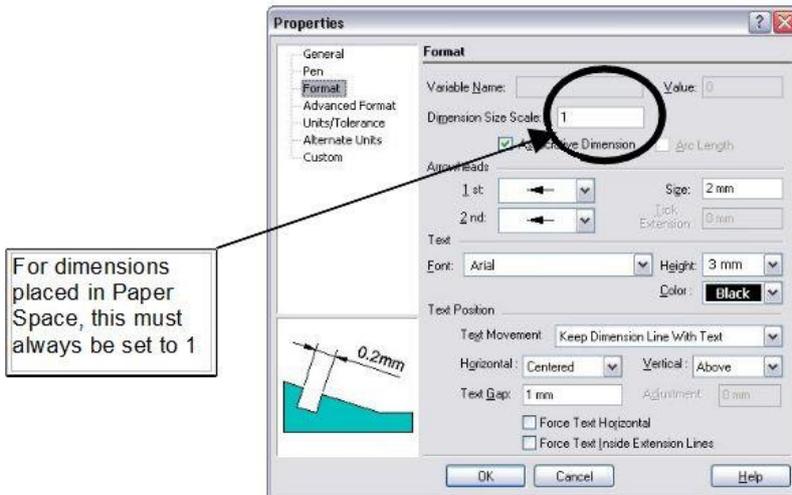


Fig 11- 1 Dimension size scale = 1 for Paper Space

Fig11-2 below shows an object drawn in Model Space and dimensioned in Model Space, with the *Dimension Size Scale* set to the default value of 1, but an intended print scale of 1/20. You can see that the figures are too small to be seen let alone be read, and there appear to be no arrows or gaps between the extension lines and the corners of the object.

I show this erroneous example because it is a very common happening and you need to understand that it is the *Dimension Size Scale* that needs to be fixed.

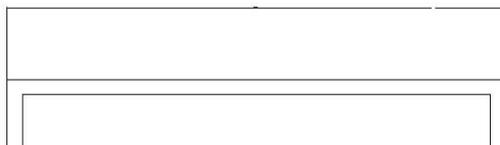


Fig 11-2 Erroneous dim scale in Model Space

Fig 11-3 shows the same drawing seen in Paper Space, with a Viewport set to a scale of 1/20, and with two dimensions. The top one was drawn in Model Space as above but with the *Dimension Size Scale* set (correctly) to 20. The side dimension was drawn on Paper Space with *Dimension Size Scale* set to 1. As you can see the result is that they both look the same. The Viewport Scale is set to 1/20

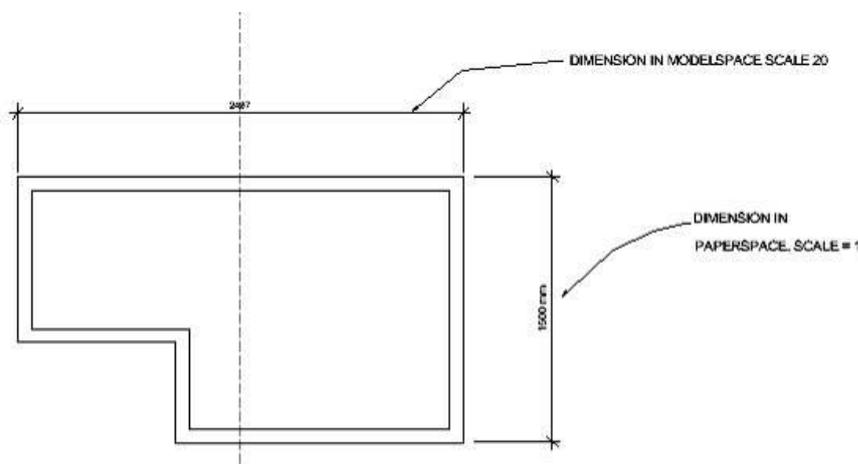


Fig 11-3 Correct dim scaling on Model and Paper Spaces

There are several advantages with dimensioning in Paper Space. Mainly you don't have to think about up-scaling. You can still use Snaps to snap to the corners of the object – you don't lose that facility. The

main downside of dimensioning in Paper Space is that although you can snap onto the objects that were drawn in Model Space, if you change the positioning of the View, like panning it, the dimensions will not move with the object outlines. However, it's easy enough to delete and redraw the dimensions, and TurboCAD makes it not too easy to accidentally shift the view seen in the Viewport. I prefer to dimension in Paper Space. Try both and decide which to adopt or take my word for it and use Paper Space.

**WARNING:** The Paperspace view must be exactly 'square-on' to the intended XY plane otherwise dimension values will be measured incorrectly. 'Square-on' normally happens automatically, but it can get upset if you have mixed up the 3D view direction somehow. Check that you are operating in the 3D Top view in Model Space. That is the Plan view that is worked on in 2D.

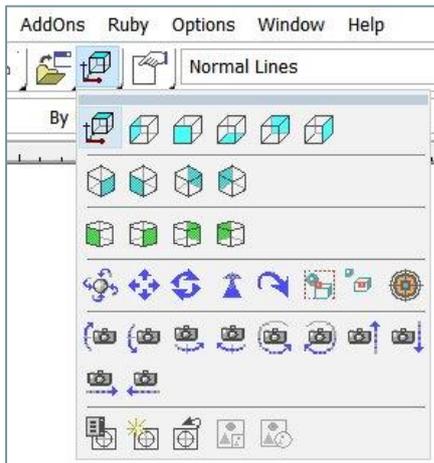


Fig 11-3a 3D Top View selection

The above panel is slightly different pre-2017 but the wanted icon is recognisably the same.

## Dimension Settings

The Dimension property settings determine the style of the dimension appearance. You should set your preferred Dimension Properties as the default before drawing any dimensions, preferably in the Template drawing from which you create new blank drawings. Then you can change the Properties for selected dimensions if desired. That might be to alter the precision of the figures, to add tolerances for example.

To set the default Dimension Properties, right-click any of the Dimension draw tools and pick Properties. A multi-tabbed dialog pops up. See Fig 11-4 The main tab is **Format**. Here you can define the type of arrows and the placement of the figures relative to the dimension lines. This is also where the *Dimension Size Scale* is set.

All the values you enter for dimension features throughout these settings, are the mm values you want to see when printed on paper. If the *Dimension Size Scale* value is set to other than 1, all the many options sizes get multiplied by that scale.

Text height must be set in mm for Model Space. In Paper Space you can use points but I suggest it's better to use mm throughout, including when setting up ordinary Text objects also.

(More detail on setting Dimension options is given under Making a Template.)

Always use the Associative Dimensions option. For dimensioning conventions used almost everywhere other than America, un-tick *Force Text Horizontal*, and set *Text Position, Vertical* to 'Above'. The rest are up to your liking or local standards. The values shown are my preferences for use in Australia and most metric countries. For continental Europe a comma is used for the decimal market instead of a dot.

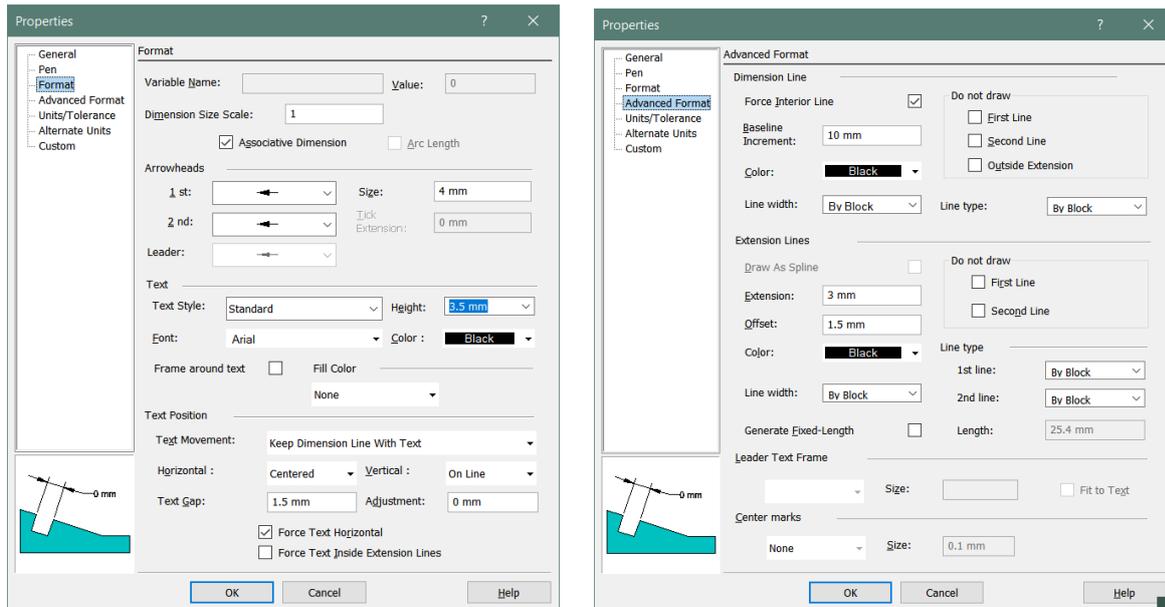


Fig 11-4 Dimensions Properties: Format & Advanced Format pages, as in TC 2017

If you use the 'tick' style instead of arrow heads, as preferred by architects, drop down the arrow list, select Tick and set .Size to 2 and *Tick Extension* to 2. It also needs Extension Line Extension set to 2.

**On the Advanced Format tab**, you probably only need to adjust Extension Line Extension and Offset.

*Extension* is the overshoot of the extension line past the dimension line. 2 mm is usually fine.

*Offset* is the size of the gap between the extension line and the feature of the object that you snapped onto. I think it needs to be at least 2 or maybe 3 mm rather than the default 1.5 mm.

The *Draw As Spline* option under *Extension Line* becomes available if you selected the Leader Dimension Type before Properties. It makes the leader lines on text notes with a pointer be drawn as a curve instead of straight line segments. That's intended to make them more distinct from the object outlines.

On the **Units/Tolerance** tab you can set the precision of the figured dimension value, and set up tolerancing format if you use that in your work.

We don't need **Alternate Units** in 100% metric countries, so ensure it is turned off on that tab page. Users in Canada and Britain might need it.

All the above settings should preferably be set up in creating a custom Template drawing from which you will base all new drawings. That is explained in Chapter 11: *Creating Your Own Template*, and the above settings are explained again there, plus more.

## Editing settings of individual dimensions

To alter certain features of particular dimensions after they have been placed, select the desired dimension or dimensions, right-click and pick Properties. You will get the same set of dialogs as above. Change whatever is needed, click OK and those changes will be applied only to the dimensions that you selected.

You can also manually override the automatically measured dimension figure, by typing what you want in the Properties *General* page. But that is usually to be avoided of course, as per normal drafting practice for avoiding out-of-scale drawings.

## Types of Dimensions

I am not going to explain all the types of dimensions here. They are detailed very well in the TurboCAD manual and on-line help. Here are notes on certain items that I think need a bit more explanation than in the Manual:

Always use Snaps for the reference points – but you will probably need the ‘*use cursor position if no snap point*’ option in order to be able to pick the location for the dimension line.

### ORTHOGONAL

This is the most commonly used dimension type. It places dimensions that are either vertical or horizontal even though the reference points may be diagonally placed relative to each other. You can use Orthogonal twice on the same two reference points, once dragging the dimension position out to the side to make it vertical, and the second time dragging down or up to make it horizontal.

### PARALLEL -V- ROTATED

Parallel makes the dimension parallel to the two reference points; Rotated asks what angle you want the dim line to be. Watch its prompts carefully.

### LEADER

You can set different options for arrows on Leader type than for other types of dimensions – you will probably want an arrow regardless of what you use for dimensions. Right-click the Leader tool button and set as shown here, or similar: Note if you use ticks, you will need to select Leader dims and change to arrow and On-line.

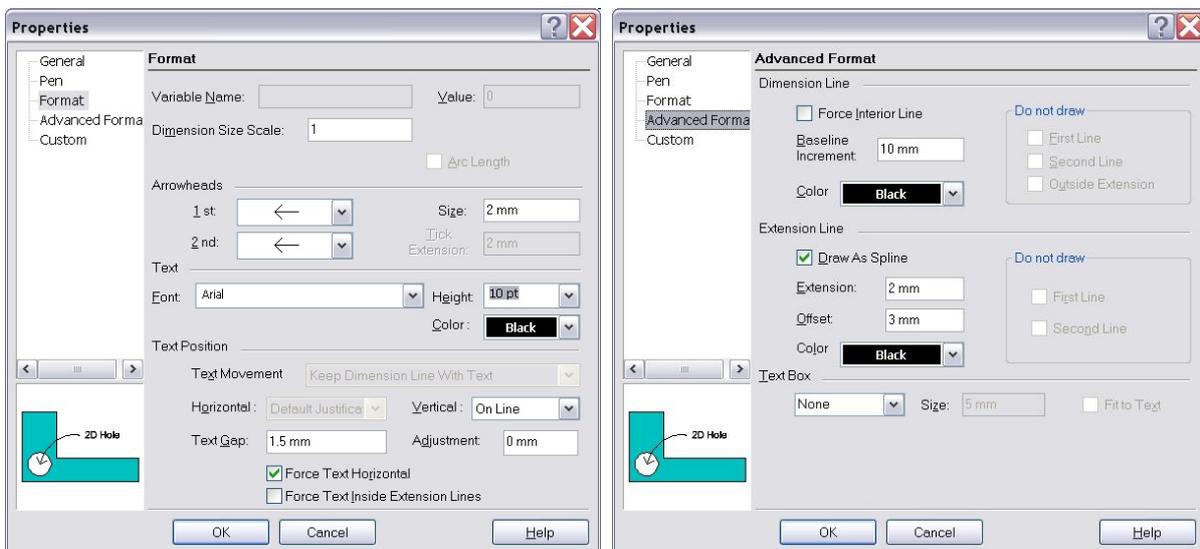


Fig 11- 4 Setting Leader format (Click Properties when a leader is selected)

Dimensioning conventions everywhere except America generally use the ‘Above’ option for dimension text, to place it above the dimension line. TurboCAD defaults to USA practice: ‘on-line’ to place the figures into a break in the line. But for leaders, you need to pick the ‘on-line’ option as shown in the Format dialog, unless you want the note to be in effect underlined as seen in the second example in Fig 11-6.

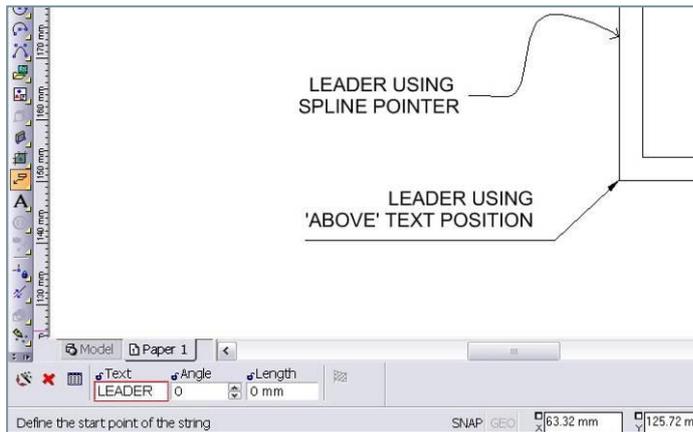


Fig 11-5 Leader format examples

**Steps for placing leader text:**

This is not really intuitive, so note the steps:

1. Type in the text for the leader in the Inspector panel. This only allows a single short line of text. For multi-line, edit it afterwards by selecting the leader, right-click, *Properties*, *General* tab as shown below. You can also add line breaks into it with Enter.
2. Pick the 1<sup>st</sup> point – which is to be the point of the arrow.
3. Pick additional points along the leader as desired, if any.
4. Double-click or pick the Finish flag in the Inspector panel to end the placement.

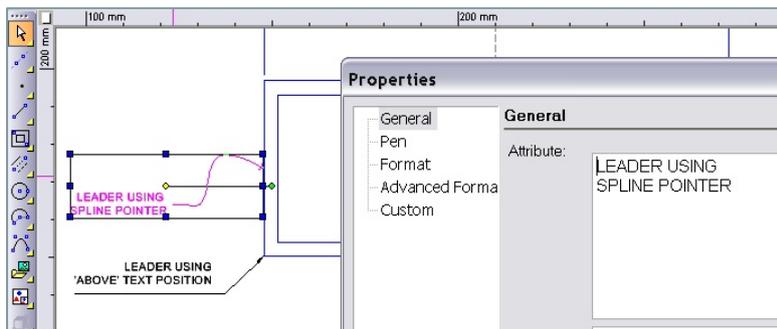


Fig 11-6 Editing Leader text

**TO EDIT LEADER TEXT –**

Click the leader item, which selects both text and leader line, and use the Properties panel, General tab, 'Attribute' as seen in Fig 11-7.

**Centre Marks**

TurboCAD 2017 introduced new facilities for placing centre marks on arc or circle objects.

To place a Centre Mark click the new Centre Mark button on the Draw (or 'Left') toolbar as seen in Fig 11-8. It might appear initially as a Dot. If so, left-click & hold so that the flyout appears, as shown, and pick the centre Mark icon from that. That will then become the icon shown in the toolbar.

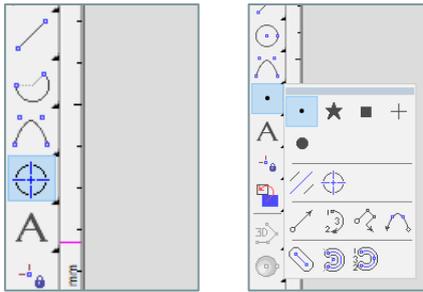


Fig 11-8 Centre Mark button & toolbar flyout

To adjust the layout of the mark, select the placed mark, right-click and pick Properties. The panel seen in Fig 11-9 will pop up. If you slide it so that you can see your placed Centre Mark as well as the Properties panel, you can see the effects of changing the various settings before clicking OK.

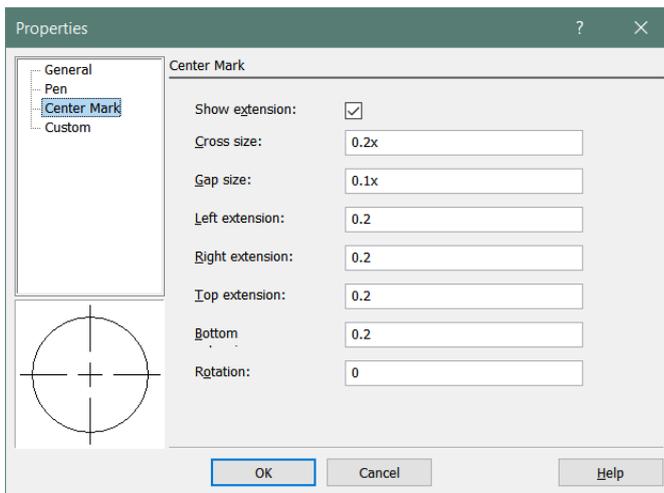


Fig 11-9 Centre Mark Properties

If you want the centre-lines to project beyond the object, as in the thumbnail, the four 'extension' settings need to be set to about 3.0.

In TurboCAD 2017, I find the centre mark facility a bit confusing, although it is a better facility than before. The problem is the old Centre mark settings still appear on the Dimension Properties settings but do not function. I think they are there to cope with older TurboCAD drawing files where the old Centre mark feature was used. But I haven't found any old drawings that can test that theory!

If you select an already placed Centre Mark, right-click and pick Properties, the new Centre Mark Properties panel appears instead of the Dimensions Properties.

## Dimension Styles

TurboCAD's Styles system lets you define several sets of settings for differing dimensions usage, such as for architectural and engineering practices, or for certain types of dimension such as Leader. See Chapter 15.