

# 14 - Dimensioning

Dimensioning is always a complex topic in any CAD system because there are so many options and variables to deal with. progeCAD collects all the numerous settings together in the **Dimension Styles**. This way a collection of settings can be named and you can have several predefined sets of settings to pick from.

progeCAD is delivered with a Style called **Standard** which is not suitable for use outside America. But it also provides a Dimension Style called ISO-25 which is set up for metric, but not entirely correctly for Australian usage (the same as ISO-25 in AutoCAD). Mainly, it uses commas for decimal points because it is set up for European usage. There are also a few other details that it is desirable to alter for Australia. I suggest you create a Dimension Style called AUS-MM and such a Style is included on the CD with this course. Another Style is also included called AUS-ARCH, which is the same except that instead of arrows it uses ticks as per common architectural practice.

There is also a fundamental decision to be made as to which rationale of placement of dimensions to adopt – in **ModelSpace** or in **PaperSpace** and hence how to manage scaling. This is quite a confusing choice, and it is important to stick with the one method. This is explained below.

## Dimension Styles & settings

The Dimension Style in use is shown on the Properties toolbar at the top of the screen. It has a dropdown list to select which Style to use. There is also a button to the left of it to go to the Dimension Styles settings dialogs.

Go to the **Dimension** menu, **Dimension Style** and the **Dimension settings dialog** pops up, as seen below.

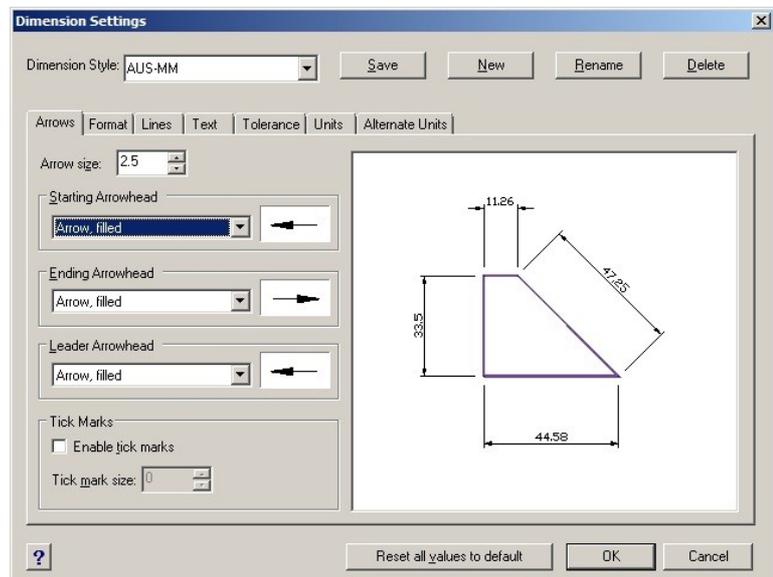
Since Dimension Styles are stored in the current drawing, to be of permanent usefulness new styles and new settings will need to be done in your Template file, as we will describe later. To create a new dimension style, click in the New button in the Styles dialog and then proceed to change whatever needs to be changed. Be sure to click the Save button - not just OK.

The new Style will have inherited the settings from the Style that was shown in the drop-down list. We need to go through them all and alter some of them. The display has Tabs for seven pages of settings. In the following illustrations the settings I suggest for an AUS-MM style are shown.

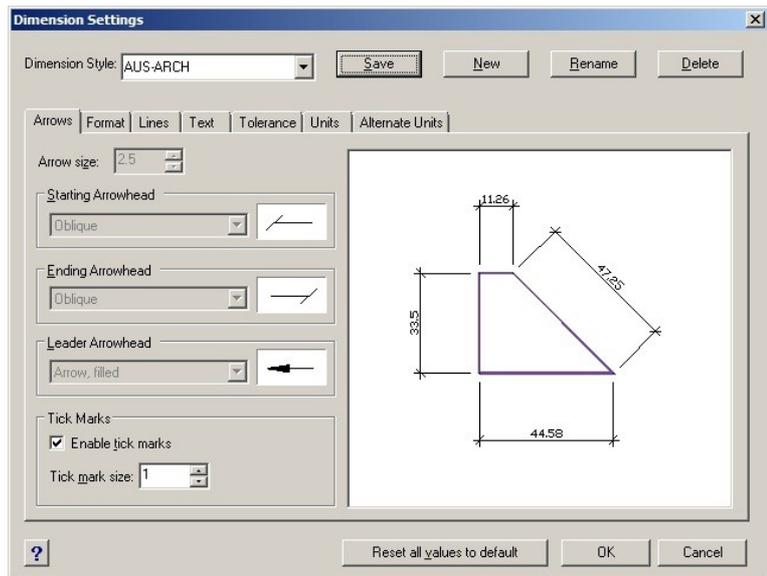
### Arrows tab

This is the initial display, seen above. The numbers for sizes represent printed mm. You can select various arrow styles.

If setting up a Style for architectural practice you may want to tick **Enable Tick marks** which disables the arrow settings, and set **Tick mark size** to 2 or similar. See the illustration below

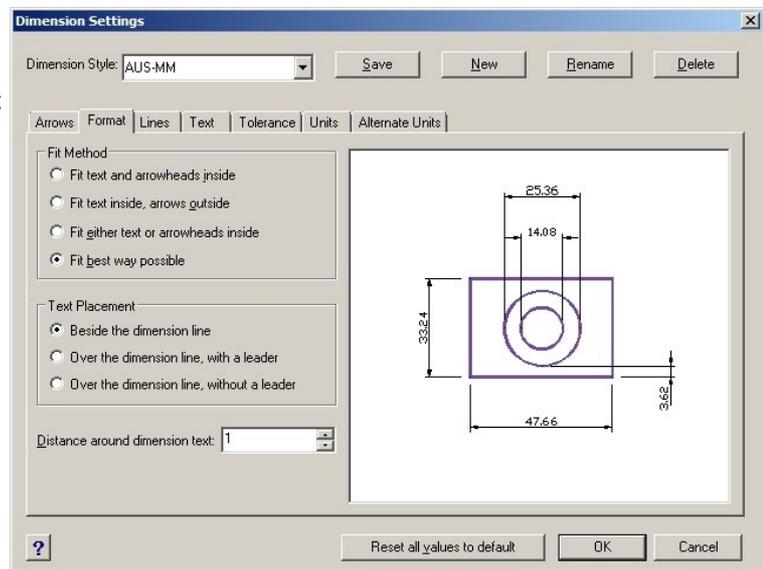


Settings for using Architectural-style tick marks instead of arrows



### Format tab

The only thing needing change here is *Distance around dimension text* and is not critical. I suggest 0.5 or 1.0

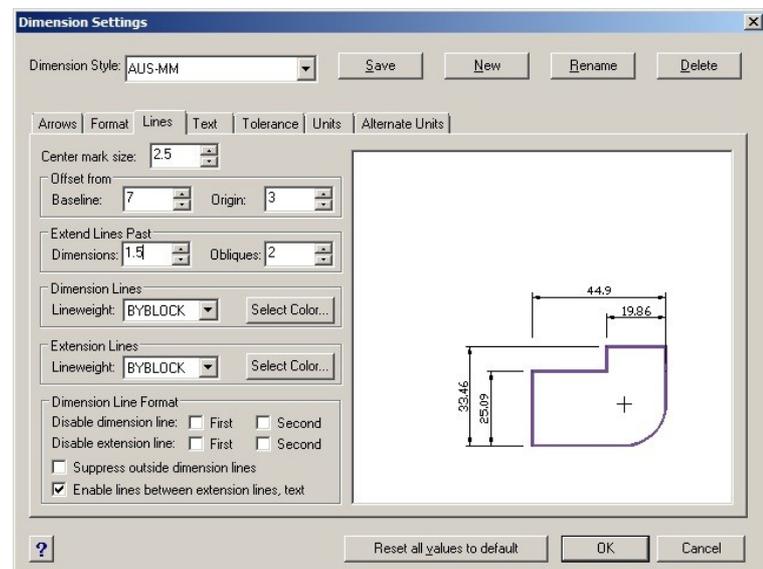


### Lines tab

Set *Extend lines past dimensions* and *obliques = 2* *Offset from origin = 2* or *3* *Offset from baseline = 7*

If using an SHX font for the numeric text, set *Lineweight* here to 0.18 mm and place the dimensions on a layer with *by-layer Lineweight* set to 0.35 mm. This will draw the dim lines thin and the figures with an adequate weight.

If using a TrueType font the text weight is automatically correct as it is part of the font design. Either set lines to 0.18 mm here or place on a layer set to 0.18 mm.



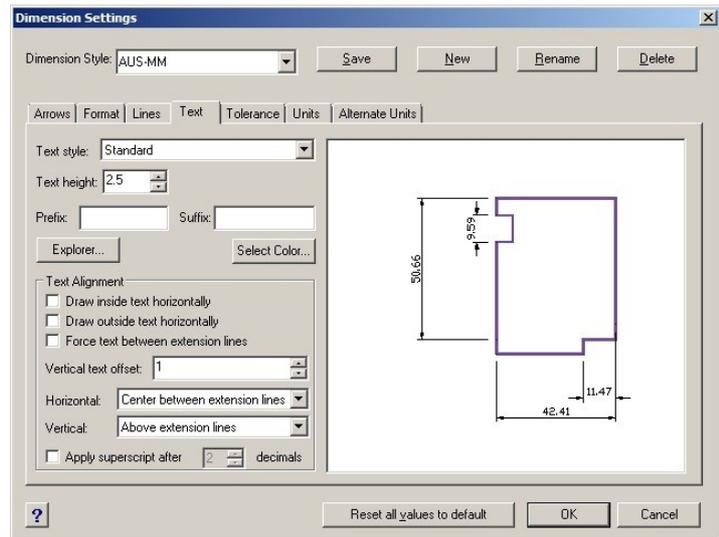
## Text tab

*Text style* can be 'Standard' if you redefine Standard to a good-looking font (see the chapter on Text) or else set it to a Text Style that you have defined.

*Text height* needs to be 2.0, 2.5 or 3.0 as preferred. I use 2.5 mm.

All the *Text alignment* options need to be un-ticked for non-USA practice.

*Vertical text offset* is the text base distance above dim lines, and 1.0 should be adequate, maybe 0.7. USA practice breaks the text into the dim lines and so they use 0. Outside USA we need to set *Vertical* to be 'Above' so that it doesn't break into the dim lines.



## Tolerance tab

These settings only apply for precision work needing tolerance specification. If you do not use tolerances, keep the boxes on this tab un-ticked. If you do use tolerances, I'm sure you will know which options to tick and what values to enter.

## Units tab

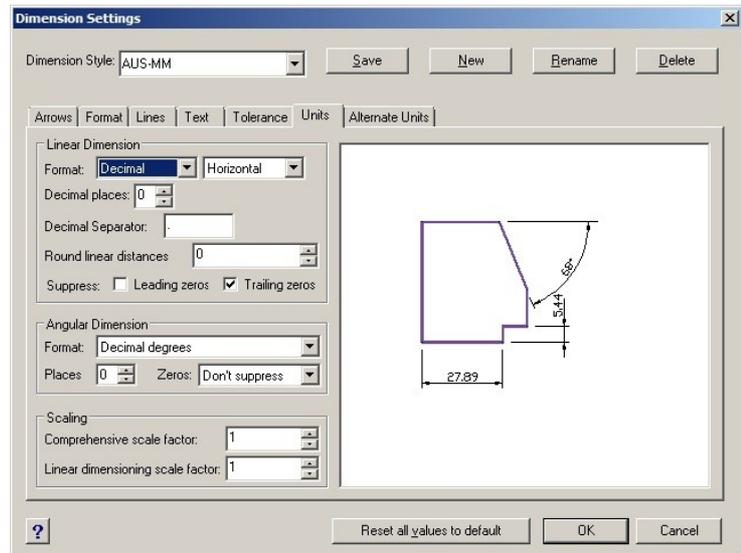
*Decimal places* will be best at 0 for mm dimensions in many fields of work.

Ensure the *decimal separator* is set to a point (full-stop) and not a comma which is the standard throughout continental Europe. (It is very hard to tell which is set, so set a full-stop anyway to be sure.)

The two scale factors should be 1.0. The '*Linear dimensioning scale factor*' should NEVER need to be anything other than 1.0. It is only relevant for drawing enlarged details, which are better done by using the Paperspace Viewport system.

The '*Comprehensive scale factor*' will need to be changed if dimensions are placed in Model Space and a print scale other than 1:1 has to be used. If the printing scale is to be 1:100 then this value will need to be 100.

But if you place dimensions on PaperSpace Layouts, then it will always be 1.0 and so is usually the easier way. Actually, when placing dims in PaperSpace this value will automatically set itself to the Viewport scale.



## Alternate Units tab

This is mainly relevant in countries that use both metric and inch units such as Canada and UK. In Australia and NZ keep *Enable Alternate Units* un-ticked.

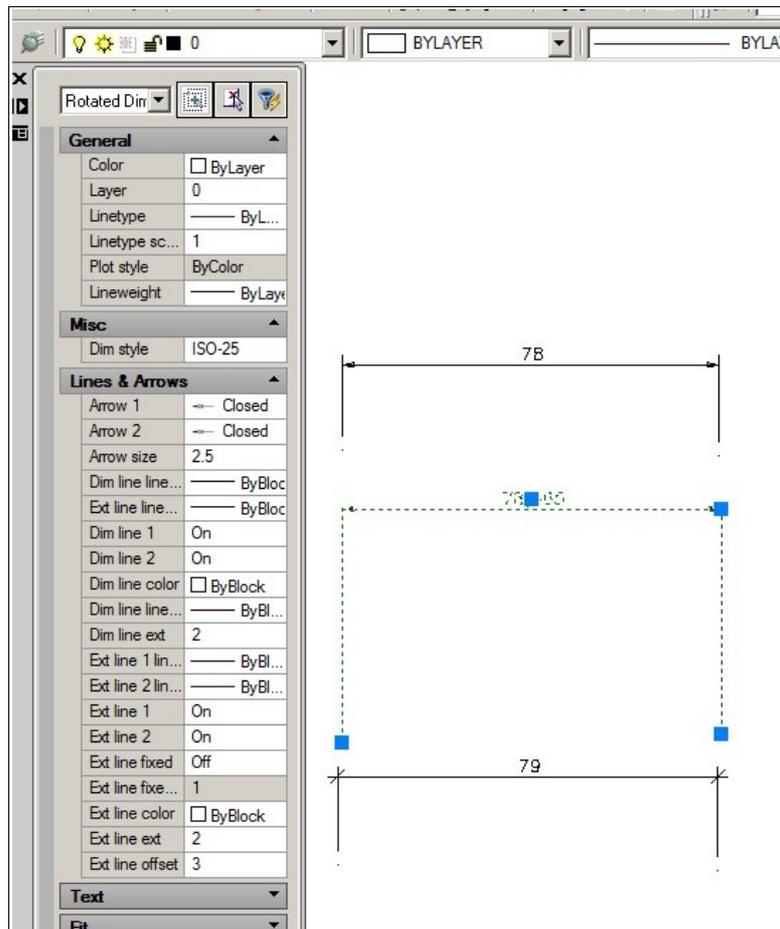
**BEWARE: The "Reset all values to default" buttons on each tab reset ALL settings and values on EVERY tab to the default USA-inch values - so DO NOT CLICK THEM!**

Click OK, and the settings for the named dimension style are saved in the drawing. As we've said, this job needs to be done in your Template drawing so that all your new drawings will already have this complex set of settings already available for use. Making a Template is explained later, as it involves other things too.

## Adjusting Dimension Style values for particular dimensions

The Dimension Styles system has a secondary mode called OVERRIDES. This is intended for applying variants of a style to particular existing dimensions. The idea is you select the dimension to be changed and then the Dim Style Manager. It appears with the Dimension Style box showing -OVERRIDES- However, I recommend you do not use that system, but instead adjust individual dimensions by using the **Property Palette**.

To do it by Properties, select the dimension to be altered. Pop up or slide-out the Property Palette. It will show all the settings for dimensions but organised differently from the Styles Manager tabs, and, rather confusingly, uses different names for the various settings. The Property Palette for a selected dimension is seen here....



You might need to widen the palette to see some of the headings fully,

## Placing dimensions

The dimensioning tools are fairly obvious as far as how to place dimensions on the drawing is concerned and not a lot needs to be said here about it. But the main difficulty is the rationale of which workspace to place the dimensions in, and how to scale them so as to appear correctly when printed.

The basic method is -

1. Select the dimension type to use from the Insert menu, Dimensions, or the Dimensions toolbar if enabled. The most often used is 'Linear', which provides vertical or horizontal dimension lines.
2. Follow the command line prompts, as usual.
3. In most cases, click two points on the drawing using entity-snap for the definition points – the features on the drawing that you want to show the distance between, and then click a point to show where you want the dimension-line placed. It drags the dimension out as you do so.

The above varies a little depending on the type of dimension, eg circles, and whether any options are picked during the process. As always, watch the command prompts. If there is a row of confusing options, they will be seen on the pop-up menu more clearly.

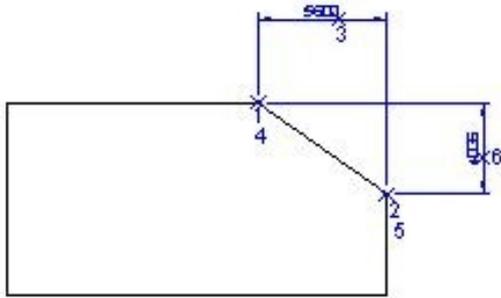
### ***Some notes on certain common types of dimensions –***

**Linear** – (Would be better called Orthogonal) This places dimension lines that are either horizontal or else vertical. You pick two definition points which may be at an angle relative to each other. Then whether a vertical or

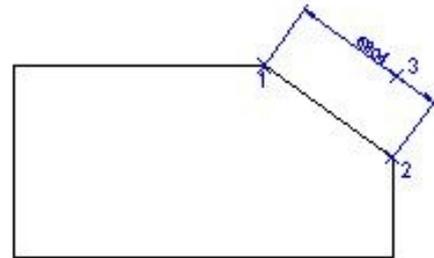
horizontal dimension is applied depends on whether you drag the dimension out upwards (or downwards) or else sideways for the 3<sup>rd</sup> point that defines the dimension position.

To place both vertical & horizontal components of a slope, use the same definition points for two Linear dimensions and place one above and one to the side as shown below.

**Aligned** – The Linear option, described above, lets you pick definition points at an angle to each other but always places the dimension orthogonally. To make the dimension line follow the same angular relationship as the definition points, use the *Aligned* type of dimension.

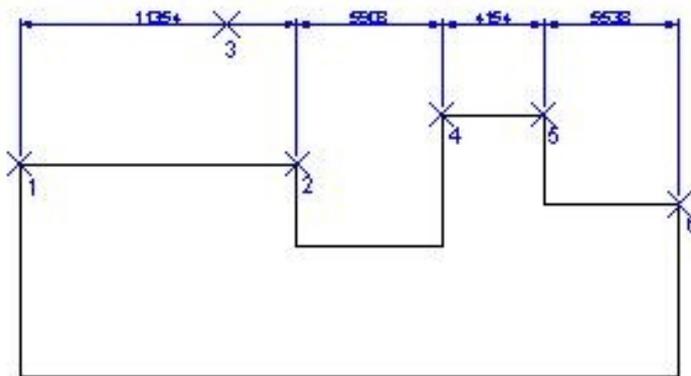


2 Linear dimensions - pick points 1-3 then 4-6 for 2nd dimension



Aligned dimension

**Continue** – This continues a Linear dimension as a string of abutting dimensions without drawing duplicate intermediate extension lines. It automatically places dimension line in alignment with the last linear dimension.



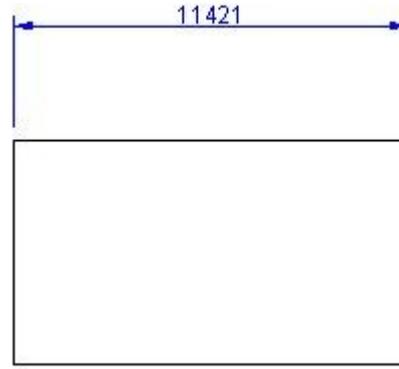
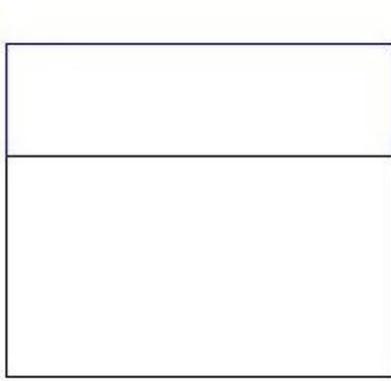
Linear dimension points 1,2,3 . Then use Continue & pick 4,5,6

Therefore, use Continue right after placing the left-most or right-most or top-most or bottom-most linear dimension of an intended string of dimensions. It will continue on from the extension line that was picked last, so if you place the first linear dimension at the left end, pick the first point as the left-most definition point and the second as the right one. Conversely if you want to draw the first dimension at the right end, pick the right-hand definition point first and then the Continue dimensions will grow from the left extension line. But progeCAD lets you pick which existing dimension to continue from, if you press Enter at the first prompt. The big X marks here indicate where the placement clicks were made.

## Scaling of Dimensions

If the placed dimension appears to lack any arrow heads or figured text or any gaps to the drawing objects, then you have placed it at the wrong display scale – much too small – probably at the default value of 1 when it ought to be 100 or more. See the illustration below left.

This is the major hassle with dimensioning. If the opposite circumstance happens (much less frequent) and the arrow heads and text are gigantic, then the dimension display scale is much too large.



Dimension placed on Model with dim scale set to the default 1 which is much too small in this context – no visible arrows, figures or gaps.  
 (The drawing in Model Space was intended to be printed at 1/200)

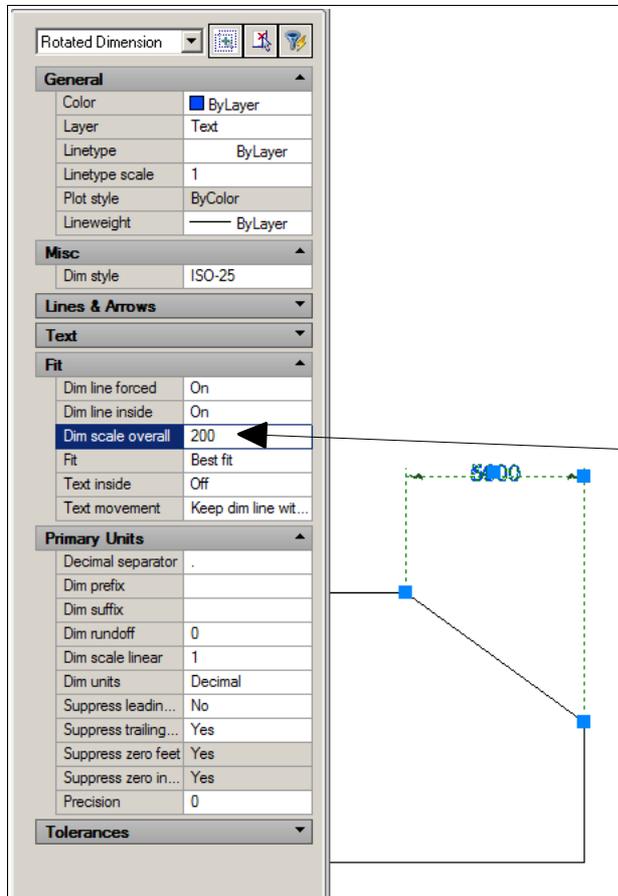
The same dimension placed on the same model with the “Comprehensive scale factor” for the dimensions set to 200, to suit the printing scale of 1/200 in this case.

The display scaling is set in the Dimension Style – see “Comprehensive scale factor”. But the manner of how to set it is the question, and where the difficult choices arise.

There are 2 possible methods for placing and scaling dimensions You will need to decide which to adopt.

1. **Place dimensions in Model Space** and manually set the *Comprehensive scaling factor* in Dimension Style (units tab), or adjust it using the Properties Panel. It needs to be the inverse of the intended plot scale. That is - if the scale will be 1/200 set the scale factor to 200, or for 1/4 set to 4.
2. **Place dimensions in PaperSpace**, snapping to Model Space objects, and keep the *Comprehensive scaling factor* in the Dimension Style set always to 1, since the dimensions are being placed on the Paper regardless of the scale of the Viewport, and prints from a PaperSpace Layout are always done at 1:1 scale.

I recommend using Method 2.



This book refers to progeCAD 2010 where Method 2 works reliably. Some other versions of IntelliCAD (v6.4) have a serious defect that makes placing dimensions in PaperSpace impractical.

If you really do want to put dimensions in Model Space drawing (Method 1), and hence have to adjust the dimension scale, the best way is from the **Properties Panel**.

To do this, with no command active select the desired dimension in ModelSpace, so that it gets grips on it. Then right-click, pick *Properties* from the popup menu and the *Properties Palette* appears as at left.

Go to the **Fit** tab and be careful to change '**Dim scale overall**' and NOT '**Dim scale linear**' lower down.

Alternatively, go to the Dim Styles Manager before placing dimensions and set Comprehensive scaling factor to the inverse of the intended print scale.

**Never** change the individual dimension settings (e.g. arrow size, text height etc) to fix scaling issues. Always change this '**Dim Scale Overall**' (called *Comprehensive scaling factor in Style Manager*). It multiplies all the individual settings by this factor.

