SHORT SHARP TRAINING (monthly)

issue 0911

Welcome to this issue of the VectorWorks Short Sharp Training (monthly). This manual is designed to work like a user group meeting. There is a main workshop topic, then extended movies showing tips or techniques and an area for beginners.

Workshop Topic
Annotation

With out text and dimensions, your drawings are pretty sad. Annotation is a fundamental part of drawing.

Annotation is more than just putting text and dimensions on the drawing, although I will be covering this to make sure you are doing it correctly, there are note tools, tools to store a database of notes and tools for labeling

Extended Podcast 094

How to use Windoor to show demolished windows.

Extended Podcast 095

Moving dimensions, using the 2D selection tool and using nudge.

Beginnercast014

The Clip Tool is really useful.

Annotation Tools

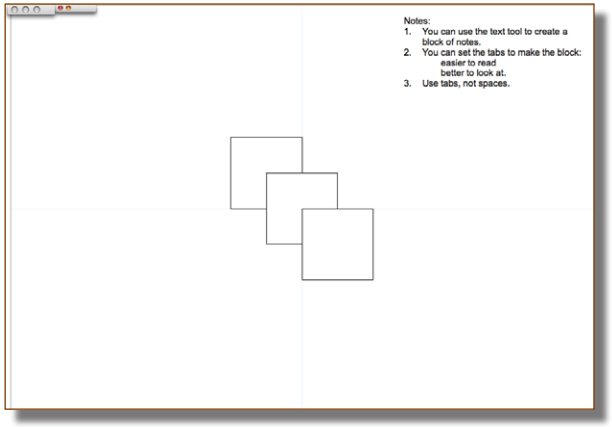
Text

Getting Started

Most drawings require text so it's important that you are able to create and edit text easily.

- Open the **issue0911** file that comes with this manual. If you are using Vectorworks 12, 2008- 2010, open the file from the appropriate folder, and use the file with the correct units (metric or imperial).

When the file opens you should see a 2D drawing with a few rectangles and some text. Do not worry about any bad spelling, we'll fix that later.

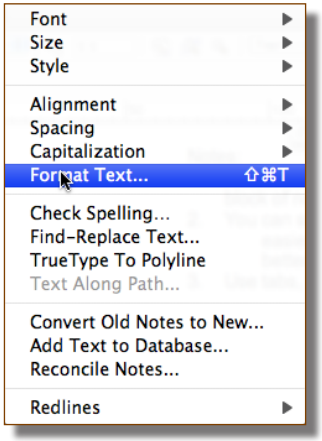


Creating Text

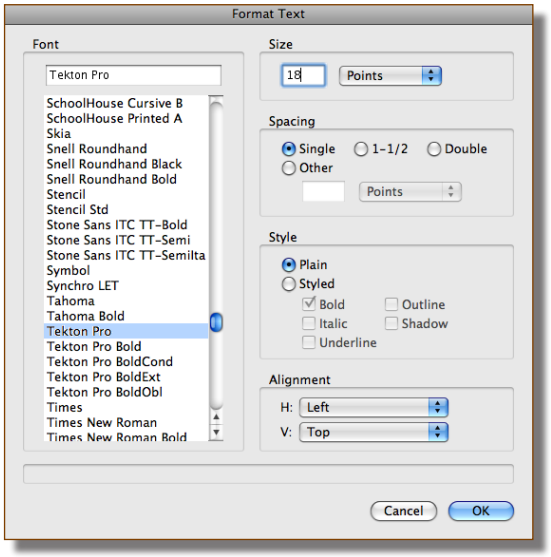
cadmovie401

- Before you start make sure that nothing is selected by double clicking on the 2D Selection tool.
- The easiest method to set the default text properties is to use the **Format Text...** command. Ignore all the other commands for the font, size and so on, this is the quickest way.
- Go to the Menu Bar

- Choose **Text > Format Text...**.

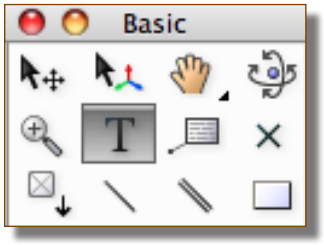


- Set the text options that you want. This is the quickest way because you can set the Font, Size, spacing and so on all in one place.



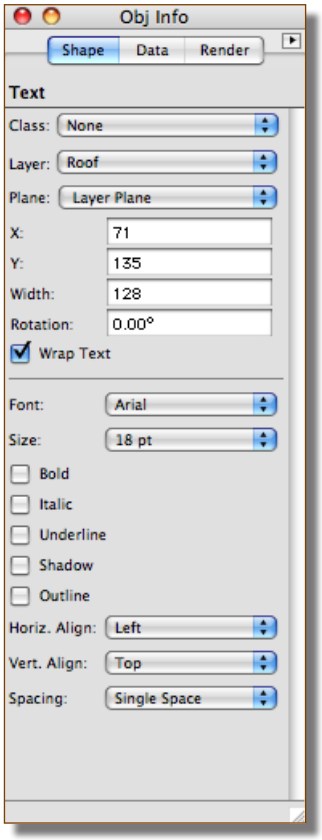
[cadmovie402]

- To create text, select the **Text Tool** from the Basic Tool Palette.



When you create text it can be one line or several lines. To make more than one line of text, use the Return (the middle Enter key for Windows) key to start a new line. To finish a block of text hit the Enter key (the far right hand Enter key on a windows machine).

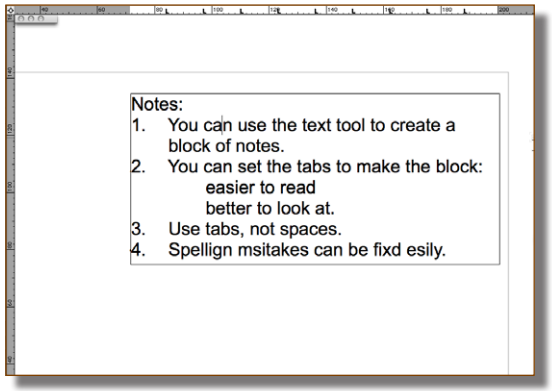
- After text is created you can edit the font, text size, style etc from the Object Info Palette.



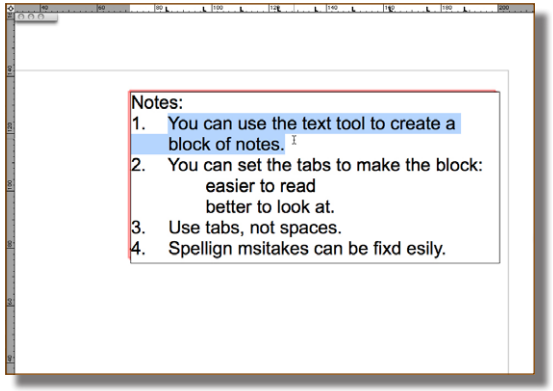
Editing text

cadmovie403

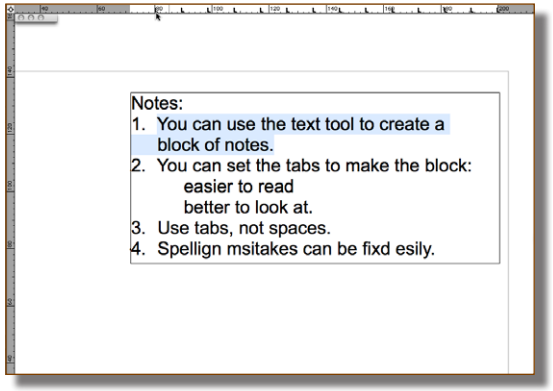
- The easier way to edit text is use the 2D selection tool. Double click on the text to edit it. If you double click on a text block VectorWorks will select the text tool so that you can edit the text.



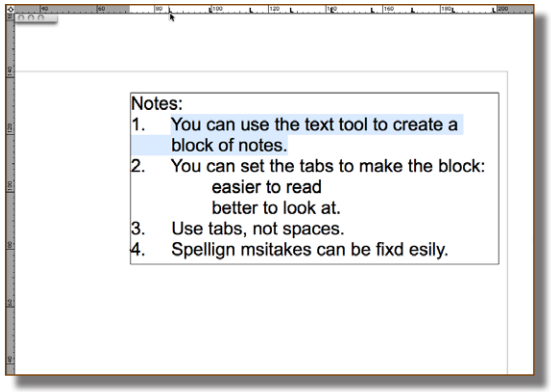
- Click on the text that you want to edit. Highlight parts that you want to change and type the new text.



When you are editing the text block the tabs appear on the rulers at the top of the drawing area. If you have the rulers turned off the tabs appear just above your text block.



- Click and drag to move the tab settings.
- Now you can use tabs to set up blocks of text.
- When you are finished choose the 2D Selection Tool from the Basic Tool palette.



Checking Spelling

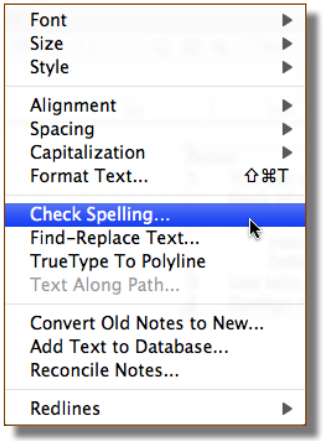
cadmovie404

VectorWorks includes a command to check the spelling in your drawings. This can be used in two ways:

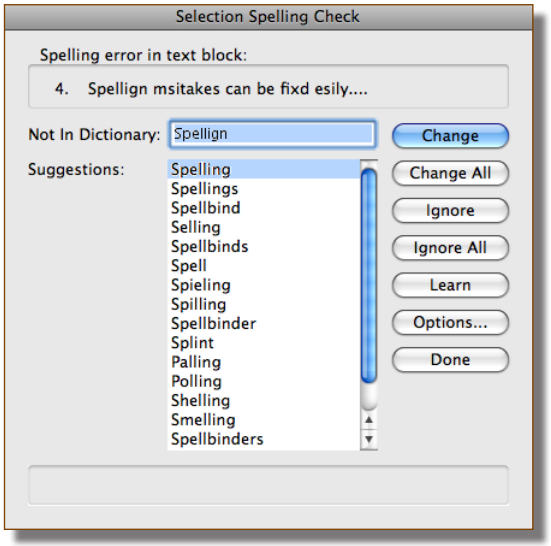
Check selected text:

- Select the block of text that you just edited, the block of notes.
- Go to the Menu bar.
- Choose **Text > Check Spelling...**

VectorWorks will check just the block of text that you have selected.

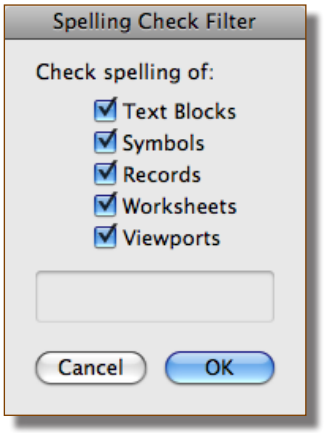


If the word is not in the dictionary then you get this dialog box. If the word is speled correctly, you can choose to add this to the dictionary by clicking on the **Learn** button.



Check all spelling in the Drawing

- Make sure that nothing is selected, by clicking away from everything or by double clicking on the 2D Selection tool.
- Go to the Menu bar.
- Choose **Text > Check Spelling...**
- VectorWorks can check the spelling in the entire file or you can choose to filter what VectorWorks checks.

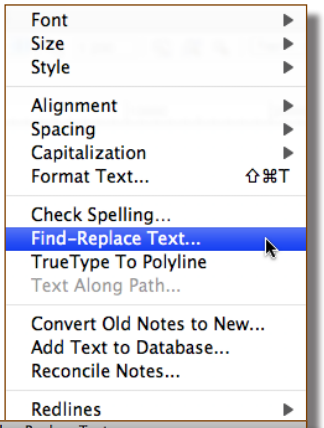


Don't forget that if you choose all text blocks it will check all the text blocks on each layer in the entire file. This could take a long time to complete. You might think that VectorWorks has crashed, but it might still be checking the spelling of stuff in other layers, viewports etc.

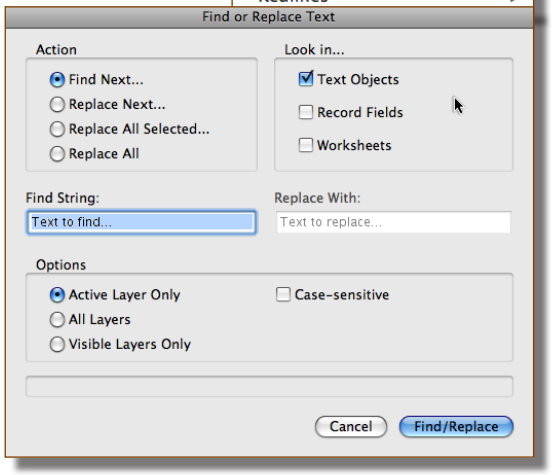
Find/Replace Text

[cadmovie405]

- Go to the Menu Bar.
- Choose **Text > Find-Replace Text...**



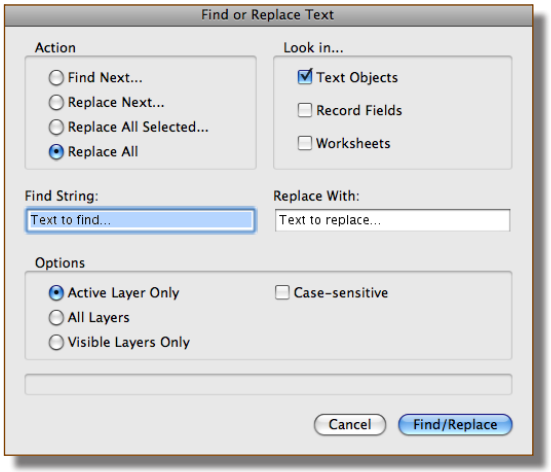
This command can be used to look for a word or a phrase in your file.



You can also use this command to find and replace a word or phrase another word or phrase. It's really useful if the client has changed a finish that you have put all over the drawings, of it the engineer makes a change to the engineering requirements. You can get VectorWorks to change all occurrence of the word or phrase.

- Type in the text that you want to find.
- Type in the text that you want it changed to.
- Click on the Find/Replace Button.

VectorWorks will tell you how many instances were replaced.



Dimensions

Dimension values will depend on how you create the objects. If you create the objects accurately then they will dimension accurately. However if you create the drawing roughly then when you dimension the drawing the dimensions will be accurate to the stuff that you've drawn but the dimensions may not be what you were expecting. Therefore it is important that you create your models accurately and that you are very precise when dimensioning (use Object Snaps wherever possible).

Because we need to be able to snap accurately to the stuff that we are dimensioning, it will help you to zoom in. You will then easily see the drawing that you want to dimension.

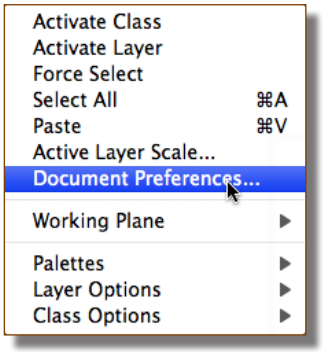
Associative Dimensions

cadmovie406

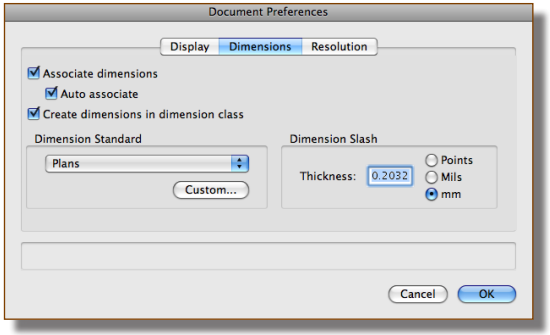
Before you start to dimension, it is important to turn on the Associative Dimensions. This is a document preference that you have to turn on. Associative dimensions are vital, because they link the dimension to the object. When you change the object, the dimensions automatically update.

If you were designing a building or landscape, this would mean that as you move the object, say a door in a wall, the dimensions would automatically update. You don't have to go back and change the dimensions manually. This not only save you hours, it will also save you from errors. Think of associative dimensions as a risk management tool.

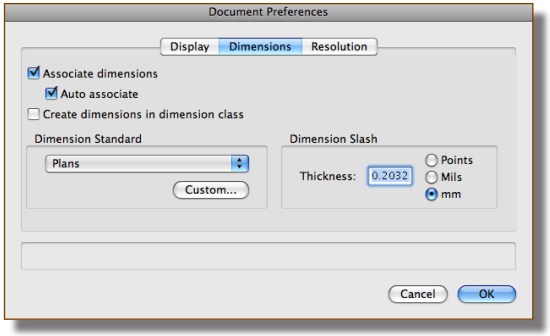
- Move you cursor into the drawing area. Make sure you are away from all the objects in the drawing.
- Right mouse click.
- Choose **Document Preferences...**



- Make sure Associate dimensions has a tick. Also useful is the Auto associate, it makes it faster to associate the dimensions.



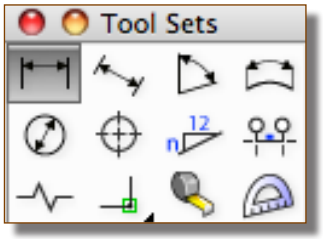
- I usually turn off the option to Create dimensions in the dimension class. This gives me more flexibility when creating dimensions on different classes.



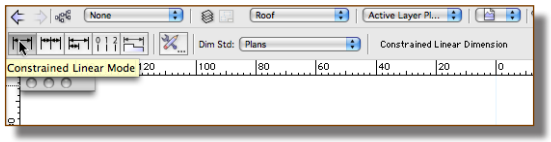
Constrained Linear Dimensions

cadmovie407

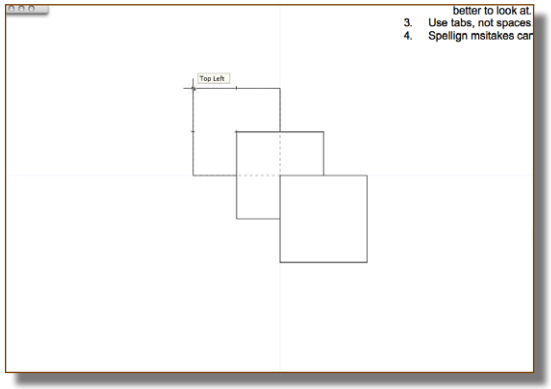
- Zoom in.
- Choose the **Constrained-Linear Dimension** tool from the Dim/Notes Tool Set.



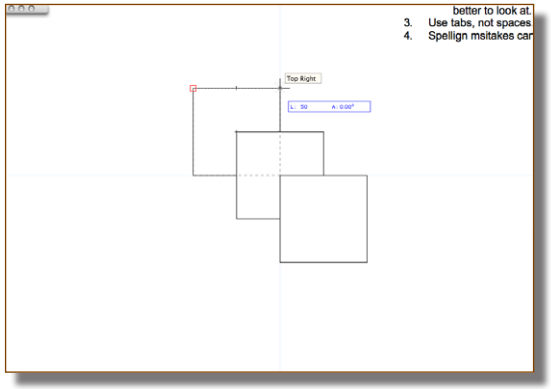
- The Tool bar shows the various options available to you (or modes of use). The first mode is used for dimensioning single items.



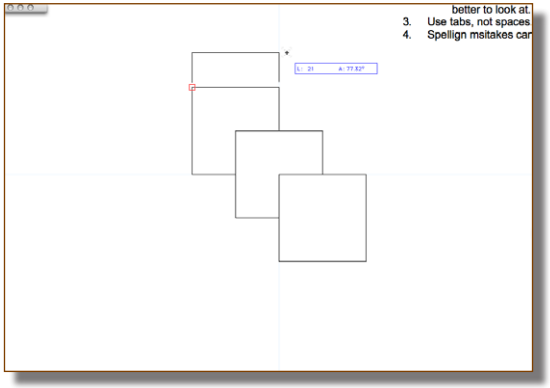
- Move the mouse to the first point that you want to dimension. Make sure that you snap onto the end of an object.



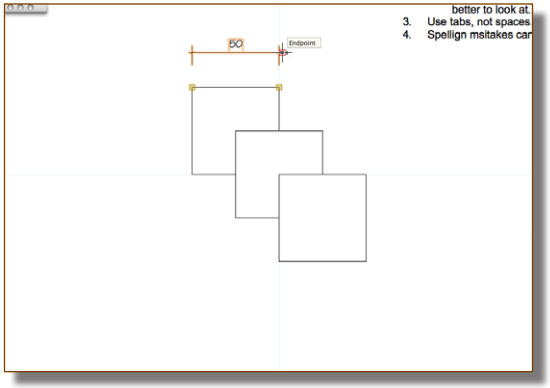
- Move the mouse to the second point that you want to dimension and click once. This sets the distance to dimension.



- Move the mouse up the screen.



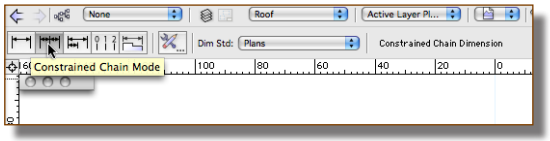
- Click once where you want to place the dimension.



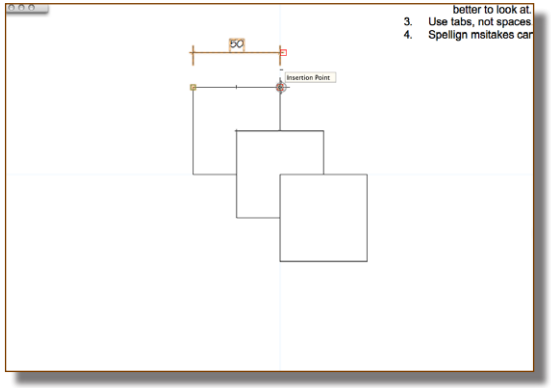
Chain Dimensions

[cadmovie408]

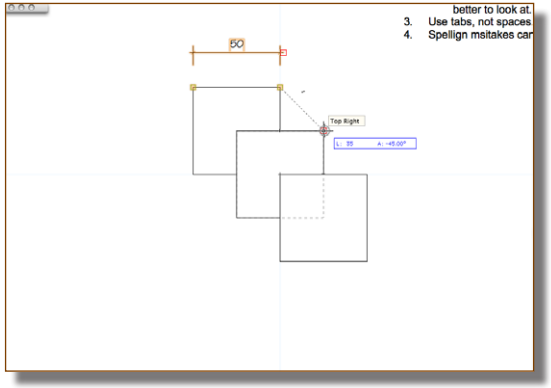
- Go to the Tool bar.
- Select the second mode. This is the mode for chain dimensions. In Vectorworks 2008 and above, chain dimensions stay together as a group after you have made them, making it easier to move them.



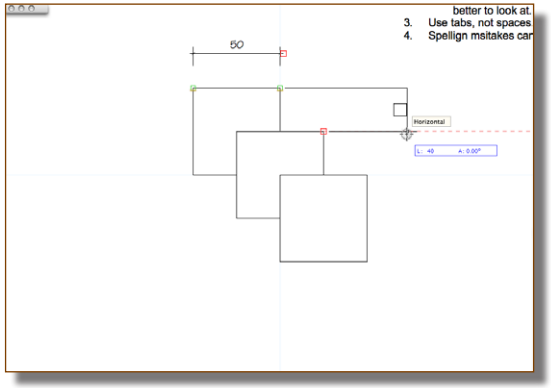
- Move to the first point
- Click to start. Make sure that you snap onto the end of an object.



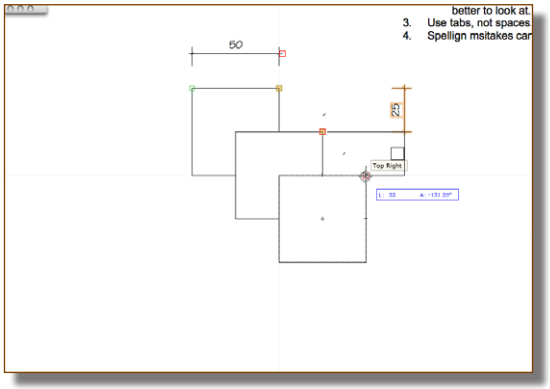
- Move the mouse to the second point.
- Click once.



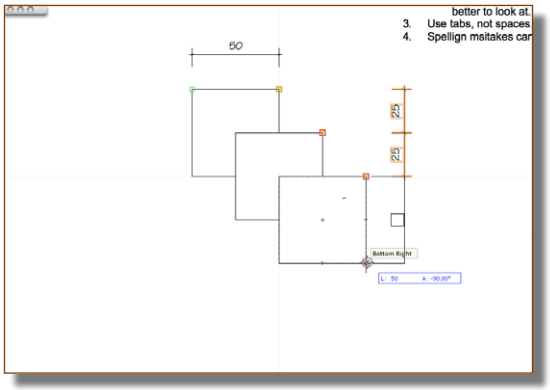
- Move the mouse.
- Click once where you want to place the dimension.



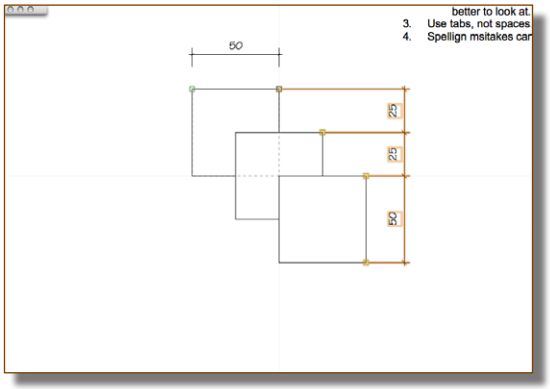
- Move to the next point.
- Click once.



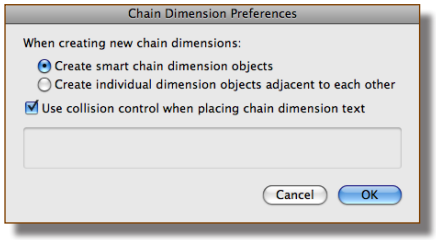
- Keep clicking at each point you want to dimension.



- Double click to stop.



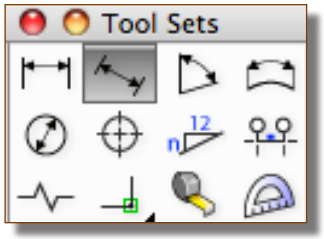
On the tool bar is a preference for chain dimensions. The preferences depend on the version of Vectorworks you are using.



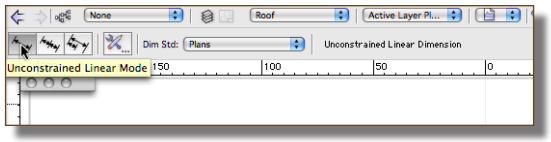
Unconstrained Linear Dimensions

cadmovie409

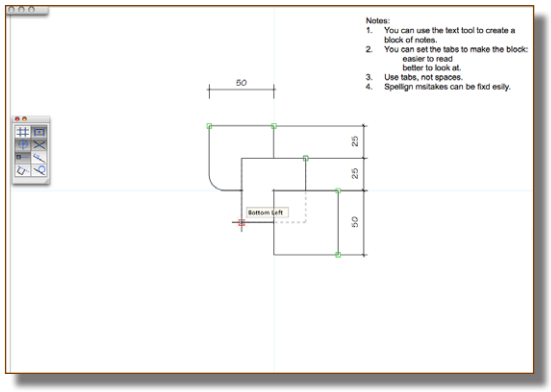
- Go to the Dim/Notes Tool Set.
- Choose the **Unconstrained-Linear Dimension** tool.



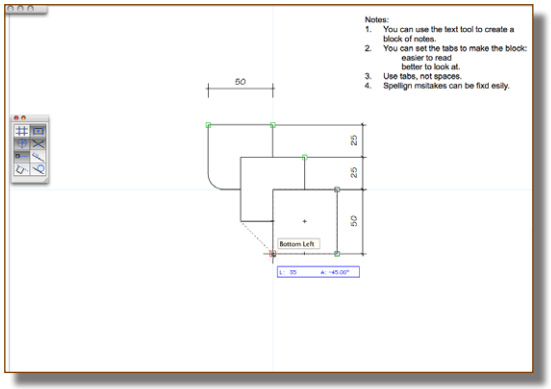
- Go to the Tool bar.
- Choose the first mode. This is used for dimensioning single items.



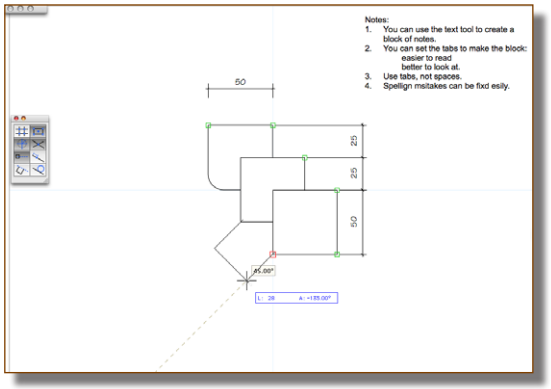
- Move the mouse to the first point that you want to dimension. Make sure that you snap onto the end of an object, or the centre of a circle.
- Click once.



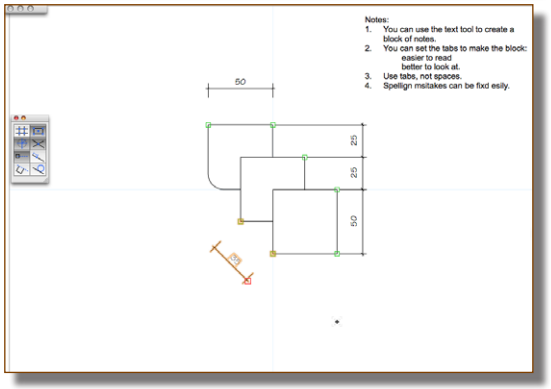
- Move to the next point.
- Click once.



- Move your cursor out to position the dimension line.



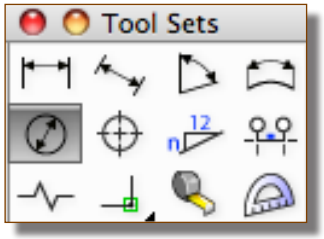
- Click once.



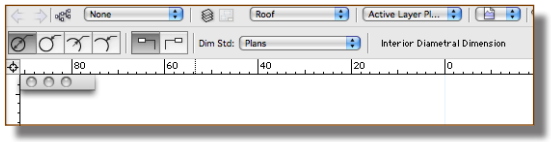
Circular Dimensions

cadmovie410

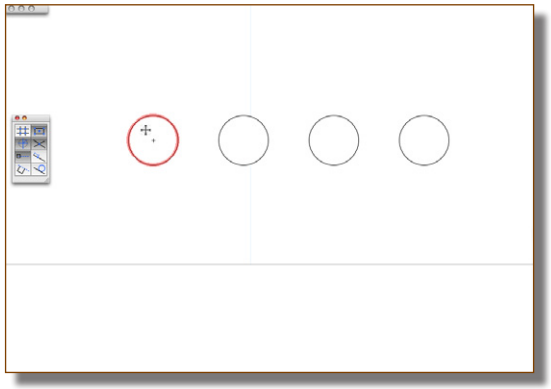
- Go to the Dim/Notes Tool Set.
- Choose the **Radial Dimension** tool.



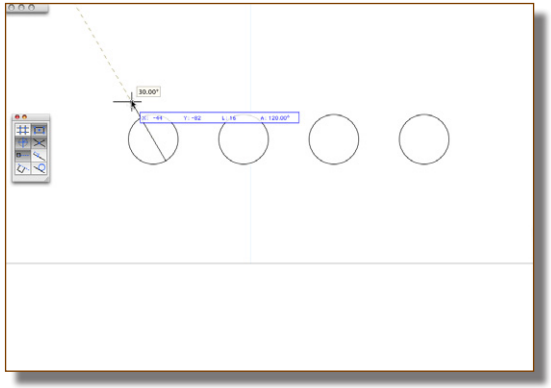
- Go to the Tool Bar. This shows the various options available (or modes of use).
- Click on the first mode, diameter dimensions.



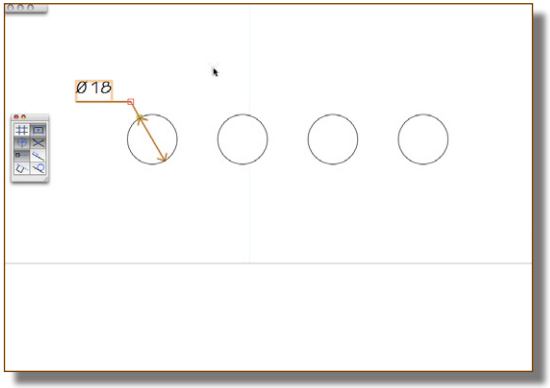
- Click on the arc or circle that you want to dimension.



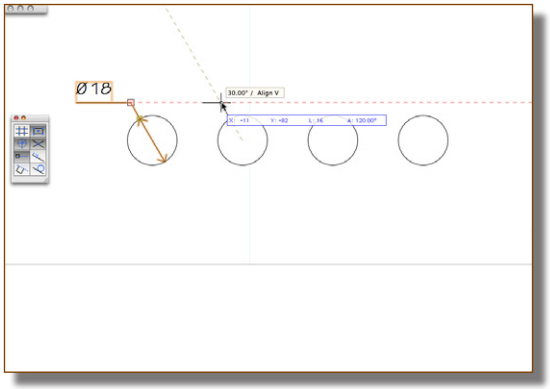
- Move to the place that the want to place the dimension.



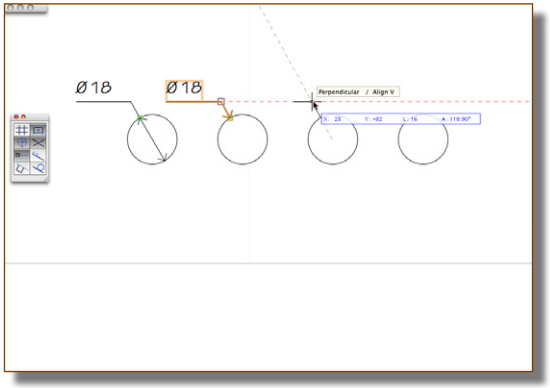
- Click once.



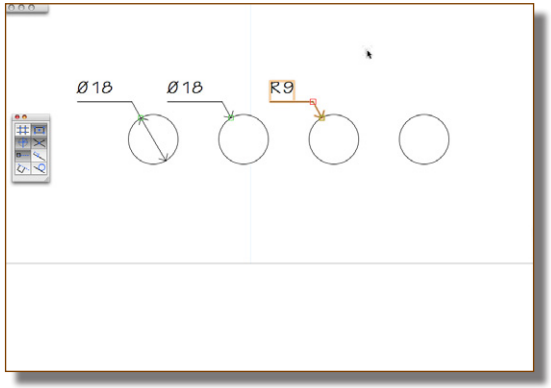
- Try the second mode.



- Try the third mode, this is for placing radial dimensions.



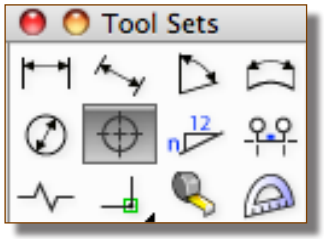
- Try all the modes.



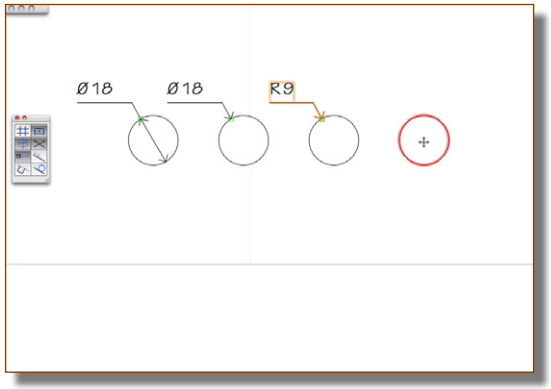
Centre Mark

cadmovie411

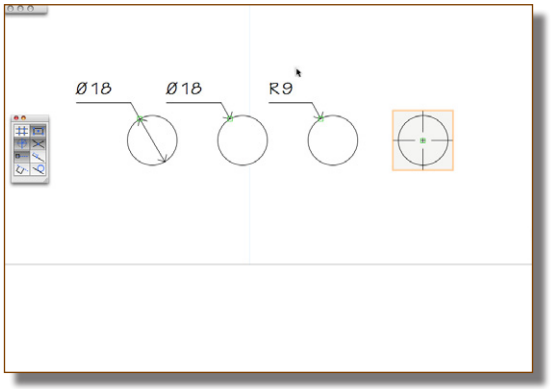
- Go to the Dim/Notes Tool Set.
- Choose the Center Mark tool.



- Move to a circle or arc. If you can place a centre mark on it the object will highlight. This only happens with the more recent versions of Vectorworks.



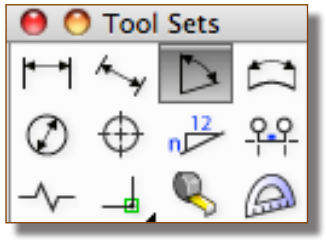
- Click once.



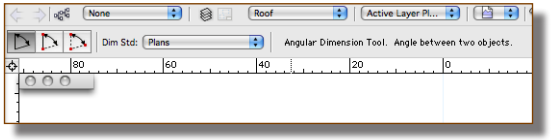
Angular Dimensions

cadmovie412

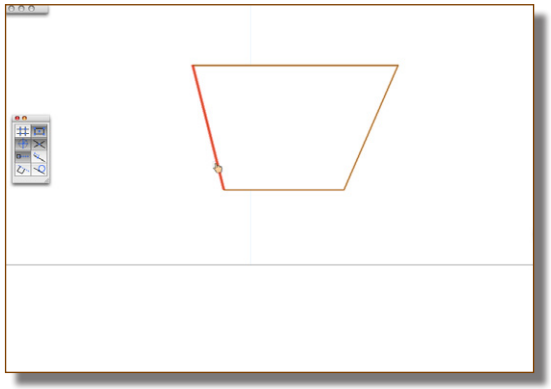
- Go to the Dim/Notes Tool Set.
- Choose the **Angular Dimension** tool.



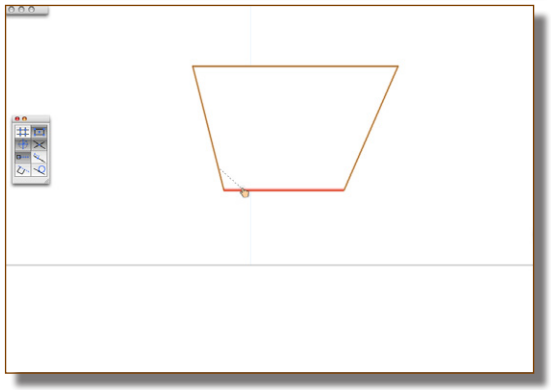
- The Tool Bar shows the various options available to you (or modes of use).
- Click on the first mode. This mode measures the angle between two linear objects.



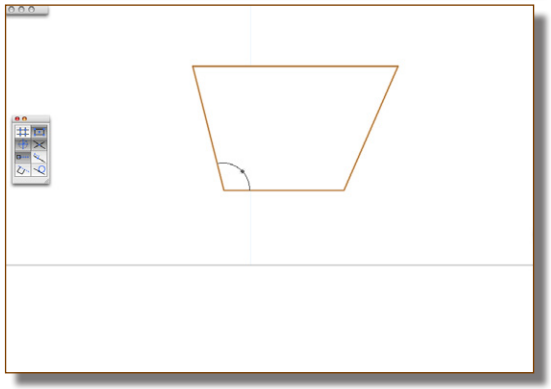
- Click on the first line. Notice that the cursor changes shape. In more recent versions of Vectorworks the line will hightlight.



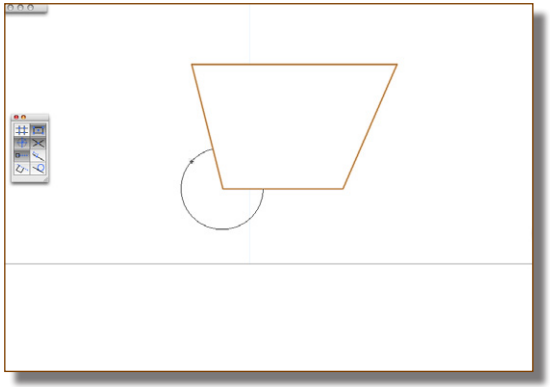
- Click on the second line for the angle. Notice that the cursor changes shape, and in more recent versions of Vectorworks the line will hightlight.



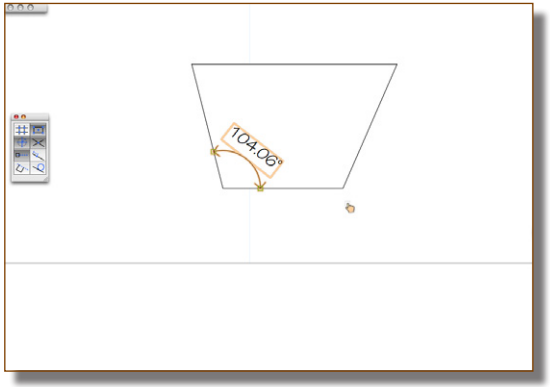
- Move the mouse inside the lines to place the angular dimension inside the lines



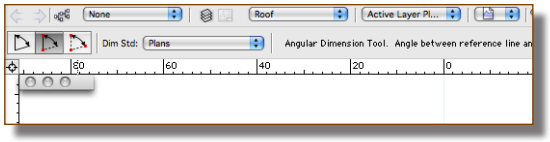
- Move the mouse outside the line to place the angular dimension outside the lines.



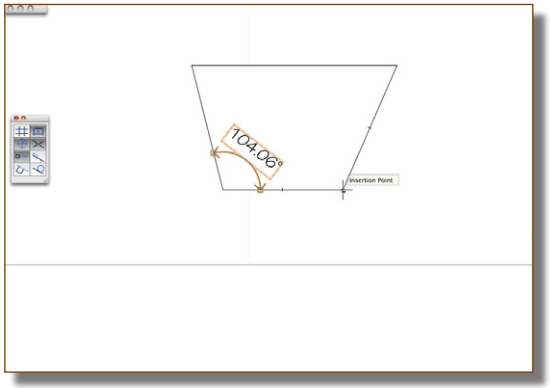
- Click once where you want to place the angle dimension.



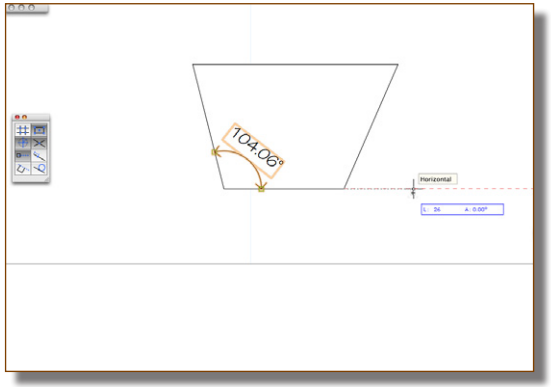
- Go to the Tool Bar.
- Click on the second mode. This places the dimension from a reference line.



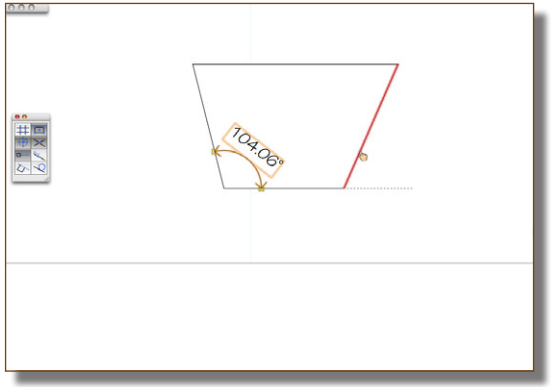
- Click to start the reference line.



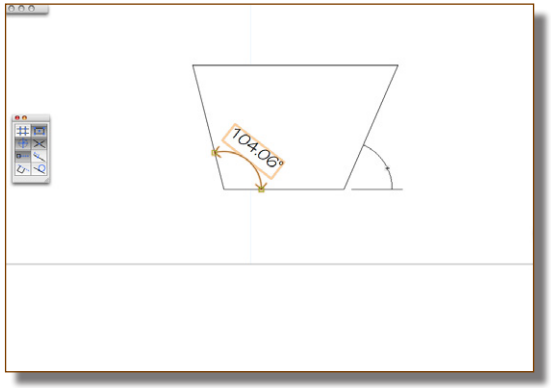
- Click to finish the reference line.



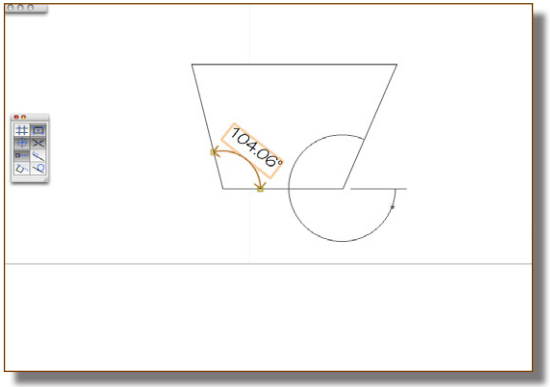
- Click on the second line for the angle. Notice that the cursor changes shape, and in more recent versions of Vectorworks the line will hightlight.



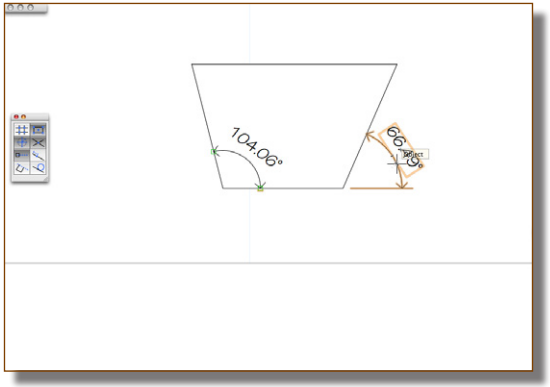
- Move the mouse inside the lines to place the angular dimension inside the lines



- Move the mouse outside the line to place the angular dimension outside the lines.



- Click once where you want to place the angle dimension.

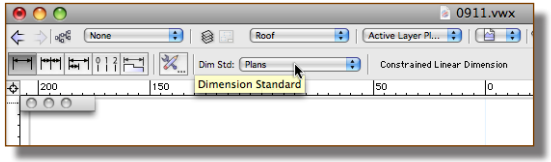


Changing the Dimension Standard

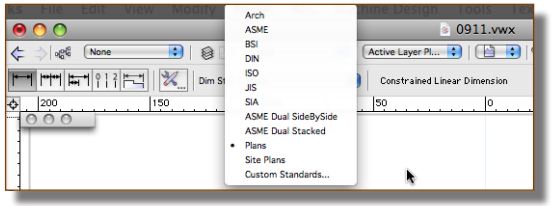
cadmovie413

VectorWorks comes with several built in standards to choose from.

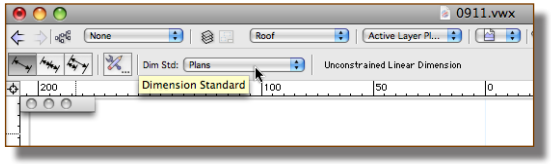
In Vectorworks 2010, you can select the dimension standard from the Tool bar when you have any dimension tool active.



- Click on the pop-up menu.
- Choose the dimension standard you want.

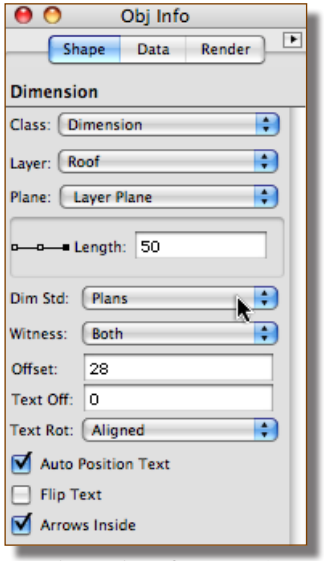


- From now on Vectorworks will use the selected dimension standard.



On all versions of Vectorworks, you can change the dimension standard of a selected dimension on the Object Info palette.

- Select the dimension that you want to change.
- Go to the **Object Info Palette**.
- Click on the **Dim Standard** pop-up.
- Choose the dimension standard that you want to apply to the dimension.
- The dimension will take on the new standard.

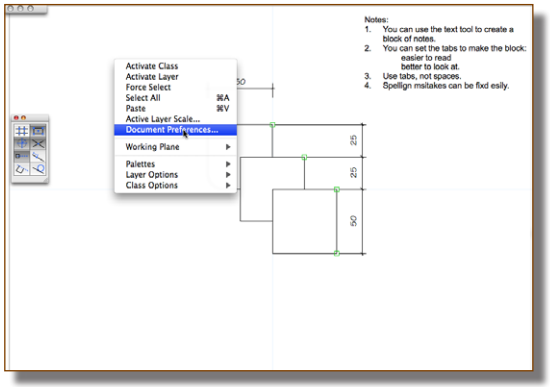


Changing your Active Dimension Standard

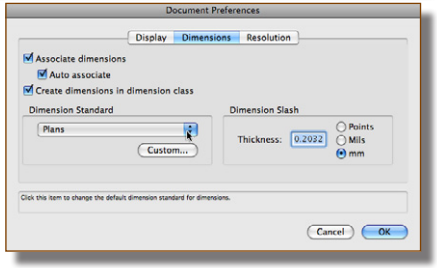
cadmovie414

If you are using Vectorworks 2009 or earlier, you have to use Document Preferences to change the active dimension standard.

- Right mouse click in the drawing area, but not near any objects.
- Choose **Document Preferences...**



- Click on the **Dimensions** tab.
- Click on the dimension standard pop-up menu and choose the dimension standard you want.

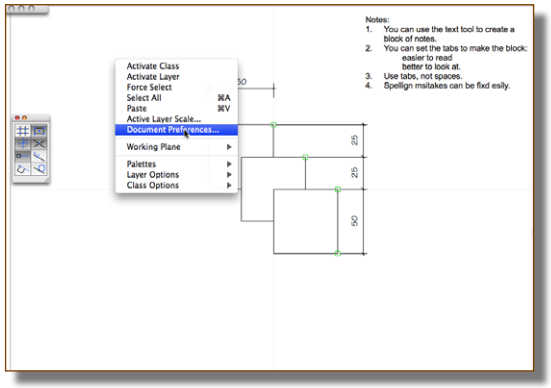


Creating Your Own Dimension Standard

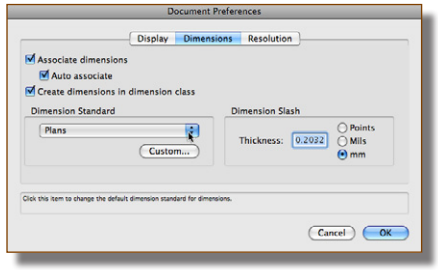
cadmovie415

Creating your own dimension standard allows you set up a dimension standard that suits your company. You might like longer lines, heavier lines and so on.

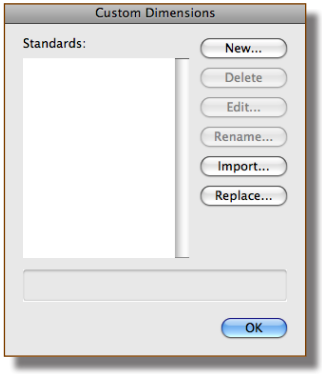
- Right mouse click in the drawing area, but not near any objects.
- Choose **Document Preferences...**



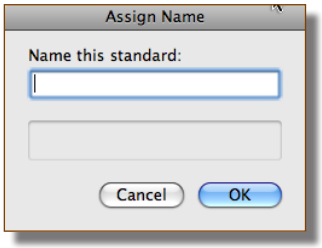
- Click on the **Dimensions** tab.
- Click on the **Custom...** button.



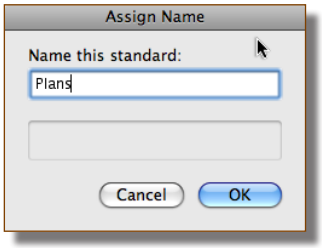
- Click on the **New...** button.



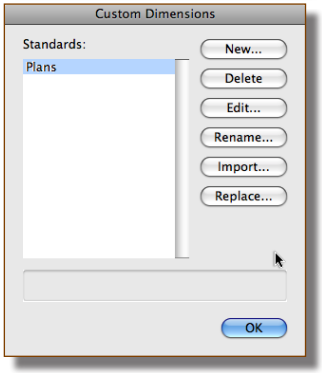
- Name the Dimension Standard.



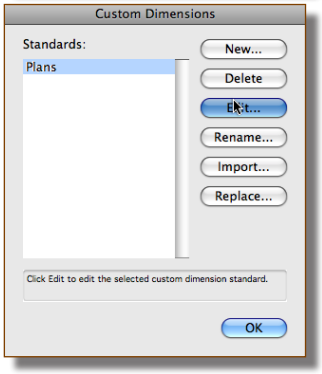
- Click on the OK button.



- Your dimension standard appears in the list.

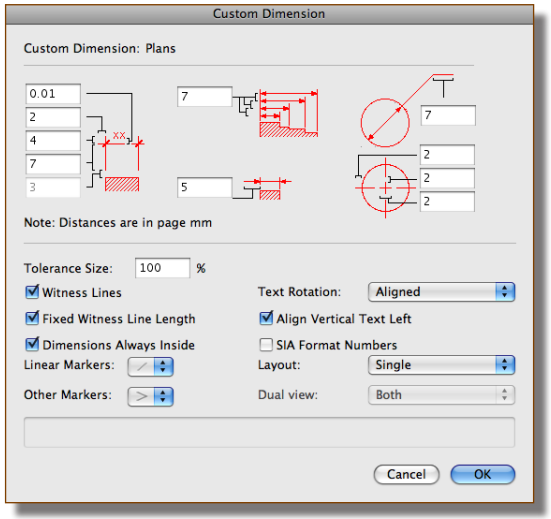


- Select your dimension standard.
- Click on the **Edit...** button.

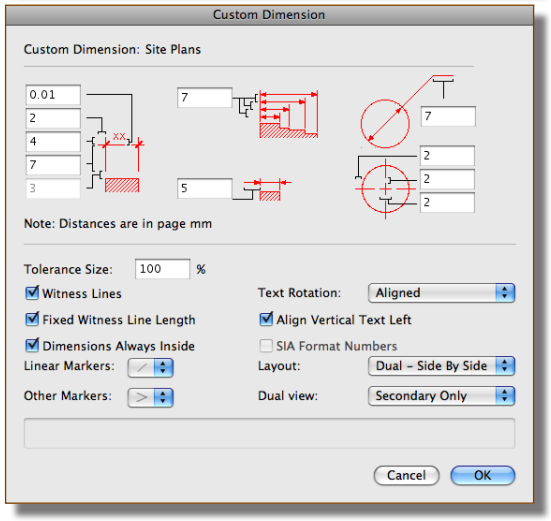


This dialog box controls the dimension standard.

- Set these up to suit the way you want your dimensions to appear.
- Click on the **OK** button.



You can set up a dimension standard to use primary or secondary units for the dimensioning.



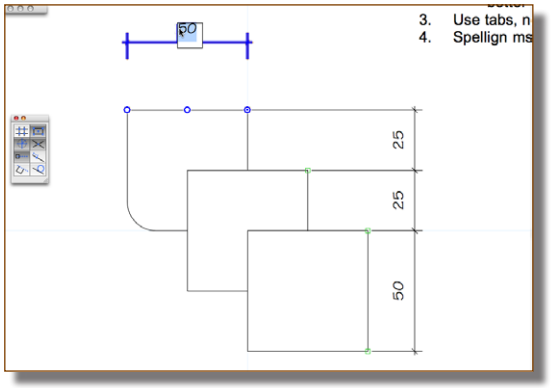
- When you are back at the Document Preferences you have to choose your dimension standard from the Dimension Standard pop-up menu. Now VectorWorks will use your custom dimension standard.

Interactive Dimensions

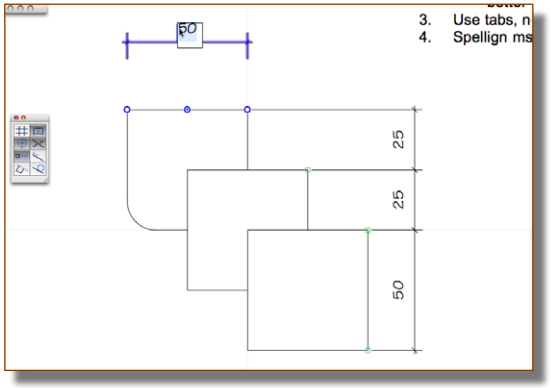
cadmovie416

In Vectorworks 2010, we have to ability to edit the size of an object by editing the dimension. This only works if you use associative dimensions, so make sure you have read the section on associative dimensions.

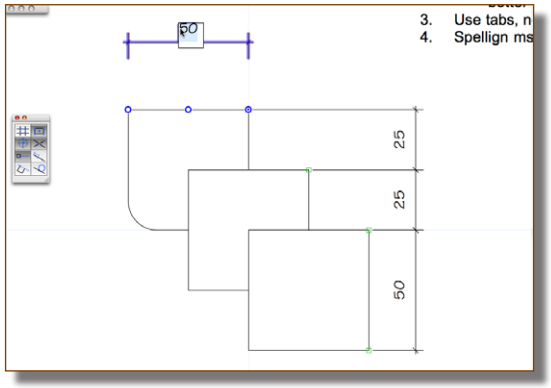
- Double click on a dimension. You will notice the dimension number highlights, like text to be edited.
- Notice there are blue dots at the end of the dimension witness lines, and one in the middle. These set the anchor point.



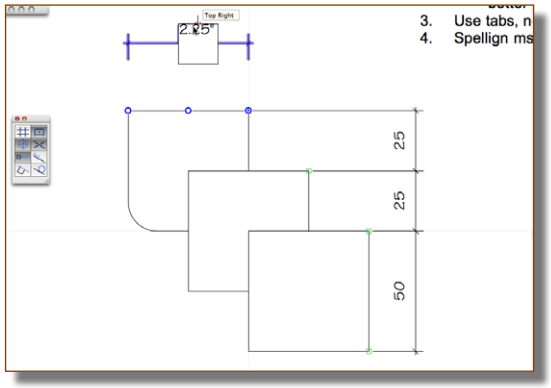
- Hit the **tab** key to change the anchor point to the next dot.



- Hit the tab key again.

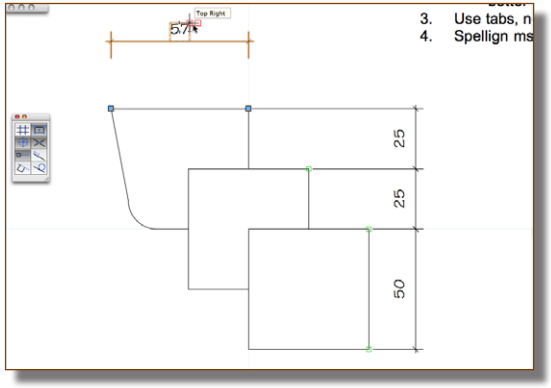


- Type in the new size.
- Hit the enter or return key.



- The associated object changes.

You can make this change on the Object Info palette as well.

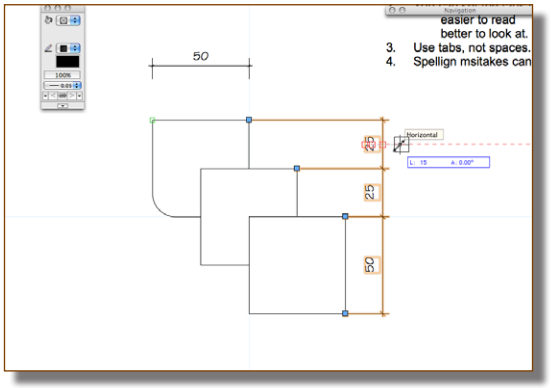


Moving a Dimension Line

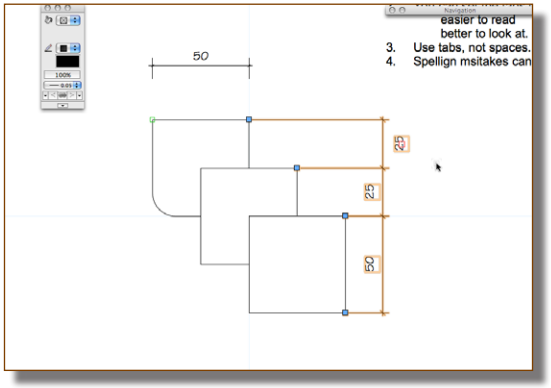
cadmovie417

Dimensions can be modified easily. To move the position of a dimension line, to change its location in relation to the drawing

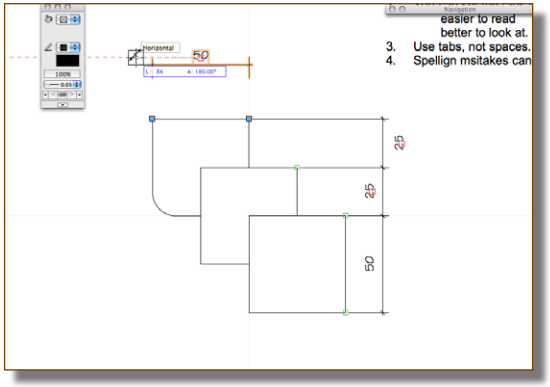
- Go to the Basic tool palette.
- Choose the 2D Selection Tool.
- Move your cursor to the line part of the dimension that you want to move. Your cursor will change to a reshape cursor when you are near the dimension line.



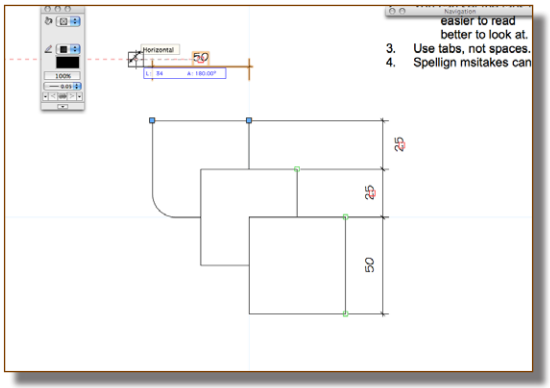
- Click and hold your mouse button. Drag the mouse but do not let go of the mouse button.
- Drag the dimension to where you want it. When you are happy about its position, let go of the mouse button.



- Click and hold your mouse button. Drag the mouse but do not let go of the mouse button.



- Drag the dimension to where you want it. When you are happy about its position, let go of the mouse button.

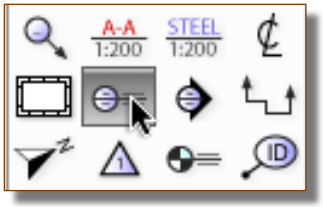


Drawing Label

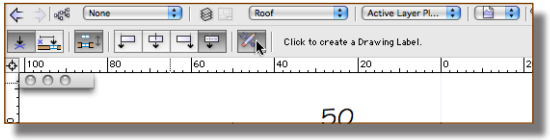
cadmovie418

The drawing label tool is used to place labels on the drawing. They can be used in 2D drawings, like I'm going to show you and they can use used in ViewPorts. The Drawing Label is scale-intelligent, that is it can detect the scale of the layer that you put the object on, so regardless of the layer the Drawing Labels are always the same size and they can put the correct scale under the label for you. If you use the drawing label in a viewport it will also read the viewport name and use the viewport name for the label text.

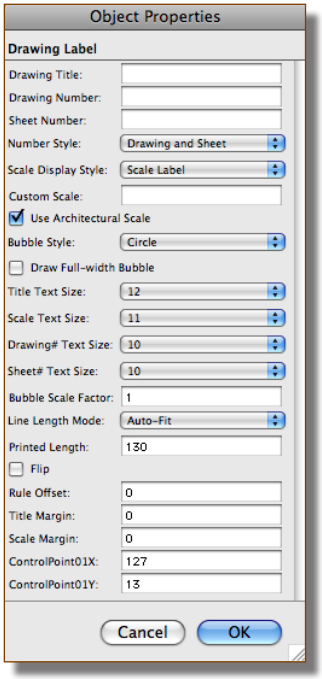
- Go to the Dims/Notes Tool set.
- Select the **Drawing Label** tool.



- Go to the mode bar.
- Click on the Preferences button.

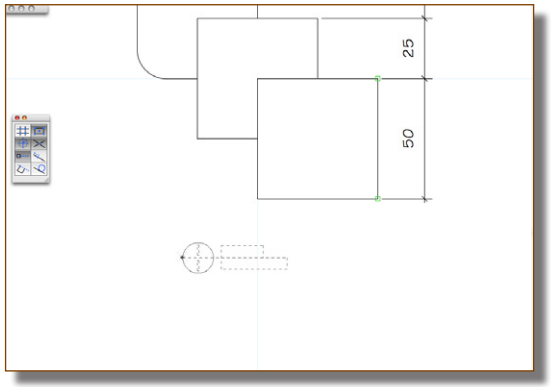


- Set Preferences for the Drawing Label. If you set the preferences before you use the Drawing Label then each label that you create in this file will have the size, text size and configuration that you want (rather than placing each Drawing Label and then changing the preferences to suit you).

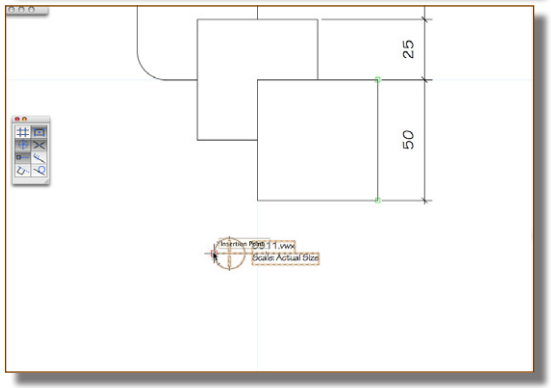


If you set the preferences the way that you want for the Drawing Label in your default drawing then every new project that you start will have the Drawing Label with the correct preferences and you won't have to change the preferences again.

- Move to the location for the left side of the drawing label.
- Click once.

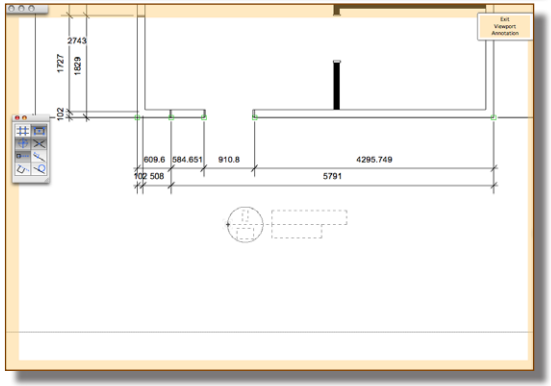


- Move to the right.
- Click once.
- Place one drawing label under the plan view, then

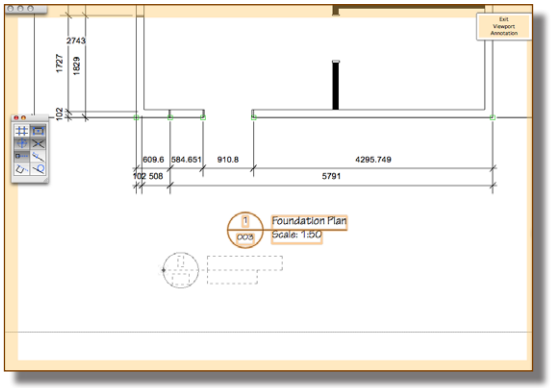


Drawing labels work well inside a viewport.

- Right mouse click on a viewport.
- Choose Edit Annotations
- Double click to place the drawing label.



The drawing label will pick up the viewport name or description and the viewport scale.

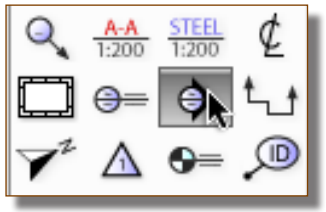


Reference Marker

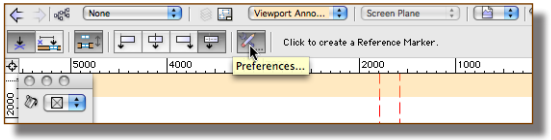
cadmovie419

The Reference Marker is used to place a detail reference, section reference or an elevation reference. Like the Drawing label the tool is scale-aware so regardless what the layer scale is the object always appears the same size.

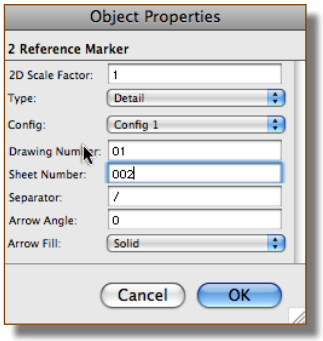
- Go to the Dims/Notes tool set.
- Select the **Reference Marker** tool from the.



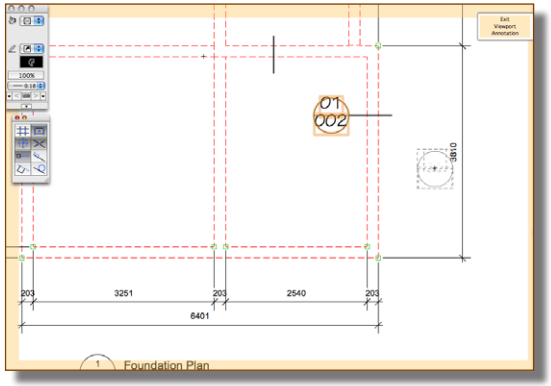
- Go to the Tool Bar.
- Click on the **Preferences**... button.



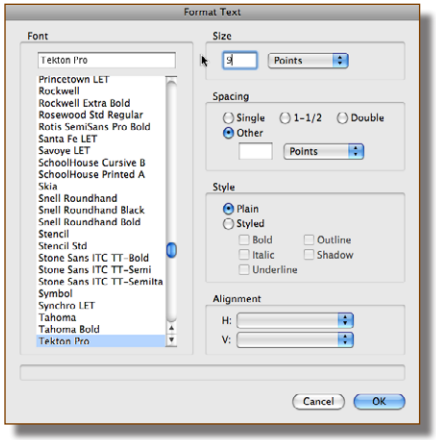
- If you set the preferences before you use the **Reference Marker** then each time you use this tool you will get the same type of object, same size and so on.



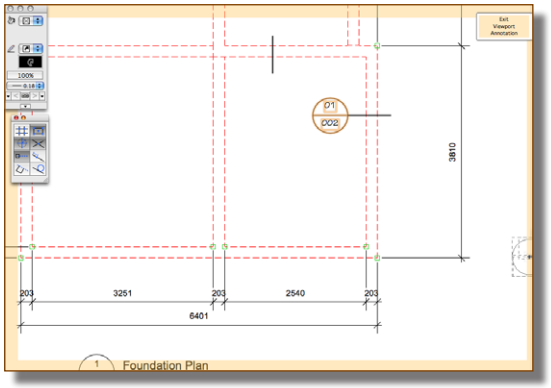
- Once you have placed the object you can edit it to suit using the Object Info Palette.



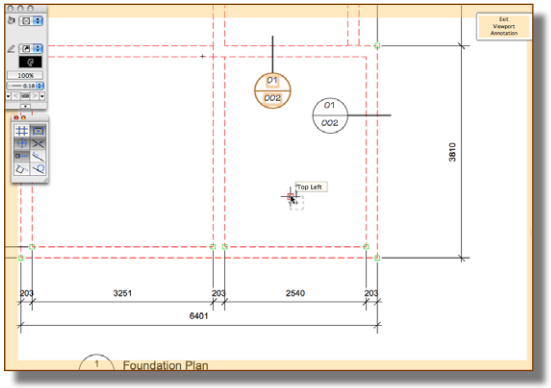
- The font size on this object is controlled from the Text Menus, so use the Format Text command to set the font size before you start.



- Use this object for sections, elevations and detail references. For sections and details you can use the Object Info Palette to rotate the arrow without rotating the object.



I use this object along with a heavy line for detail references.

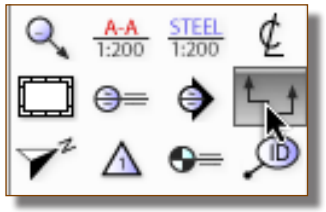


Section-Elevation Marker

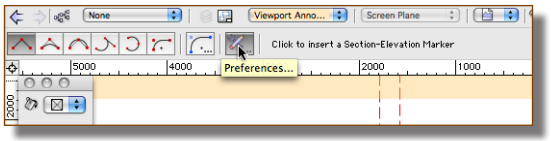
cadmovie420

The Section-Elevation marker is used to place the section or elevation line on your plans.

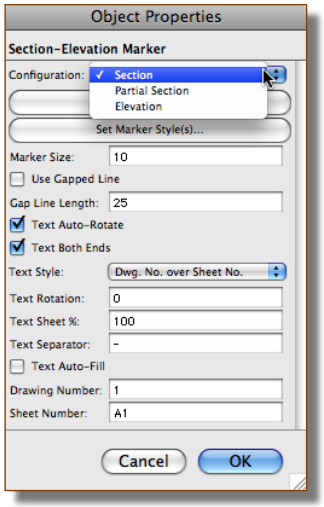
- Go to the Dims/Notes Tool set.
- Select the **Section-Elevation Marker** tool.



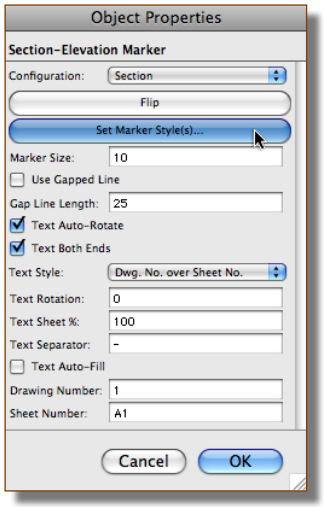
- Go to the Tool Bar.
- Click on the **Preferences** button.



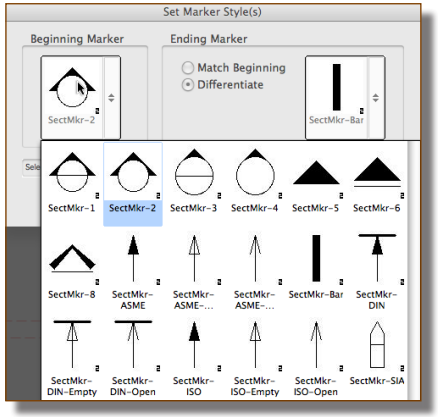
If you set the preferences before you use the **Section-Elevation Marker** then each time you use this tool you will get the same type of object, same size and so on.



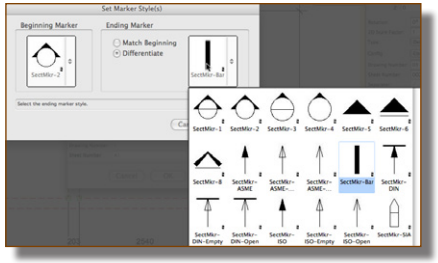
- Click on the Set Marker Symbol.. Button. Use this to choose the symbols for each end of the section marker.



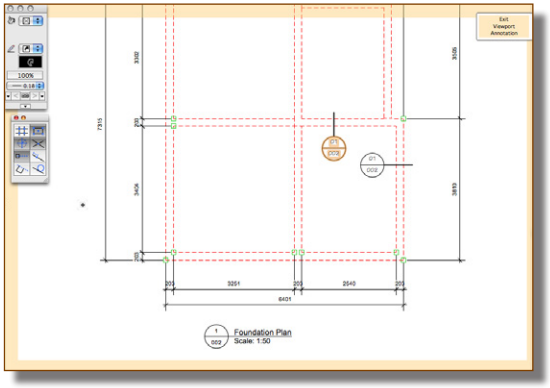
- Click on the left hand area to choose the start symbol.



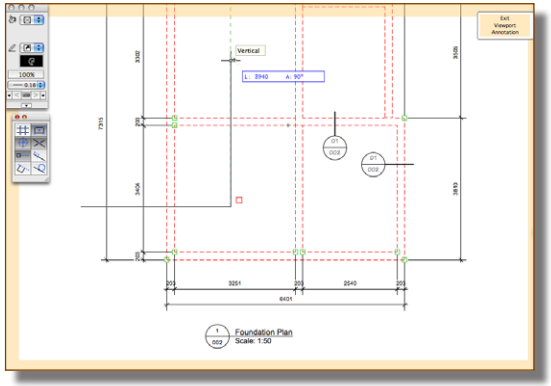
- Click on the right hand area to choose the end symbol.



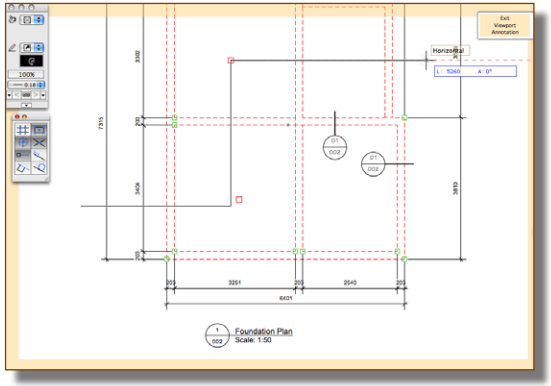
- Click to start the **Section-Elevation Marker.**



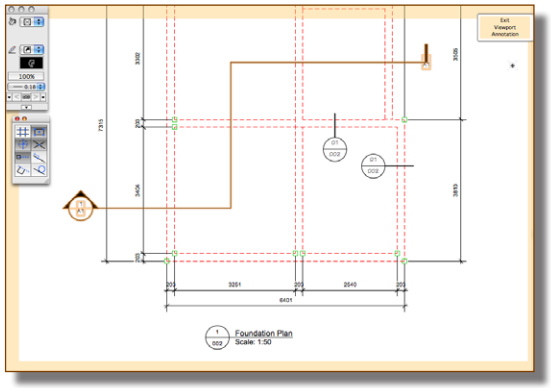
- Click where you what to change direction, this will be perpendicular to the first line.



- Click to change direction again.



- Double click to finish.
- Check the Object Info palette, you can make changes to the Section-Elevation marker after you have placed it.

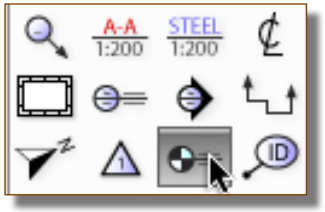


Elevation Benchmark

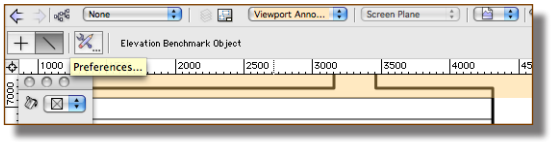
cadmovie421

The Elevation Benchmark tool is really useful for placing a benchmark on your elevations with the correct height on it. Then if you move the benchmark the height can be set to automatically update to the new elevation.

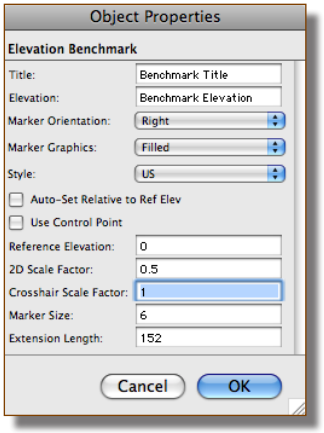
- Go to the **Dims/Notes** toolset.
- Choose the **Elevation Benchmark** tool.



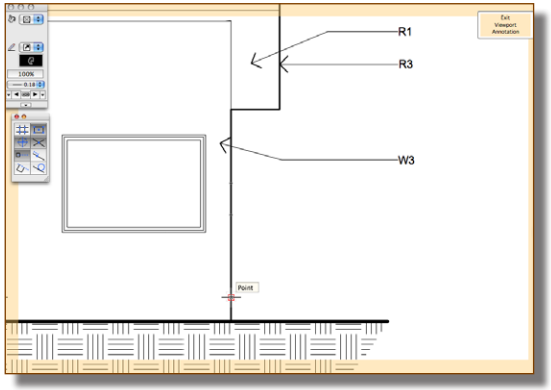
- Go to the Tool Bar.
- Click on the **Preferences** for the Elevation Benchmark tool.



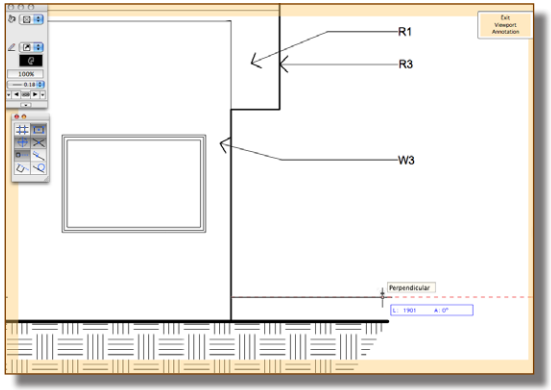
- Fill in the preferences that you want. You will have to play with this before you get all the settings you want.
- Click on the **OK** button.



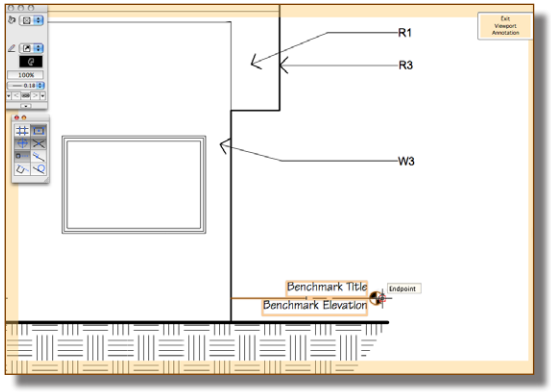
- Click to start the benchmark.



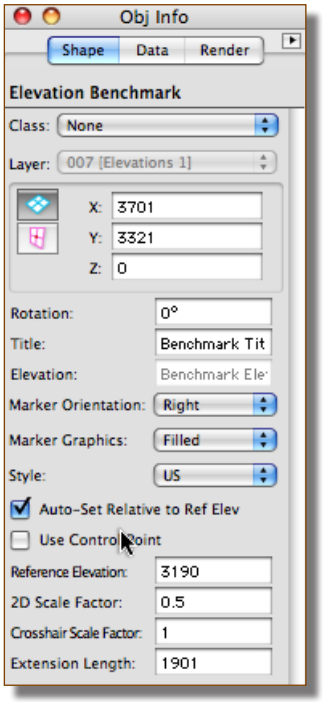
- Move the cursor across to the left.



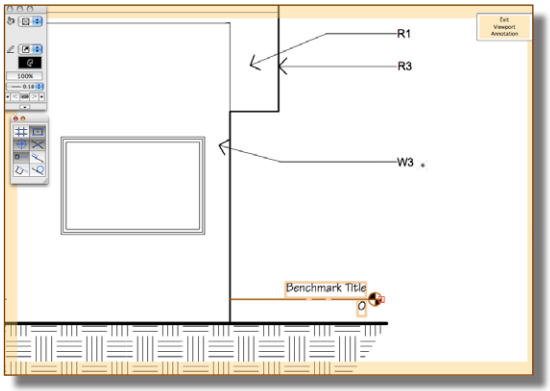
- When the benchmark is as long as you want, click again. The height of the text is set buy the Default Text Settings. Change the text size with the Format Text command from the Text menu.



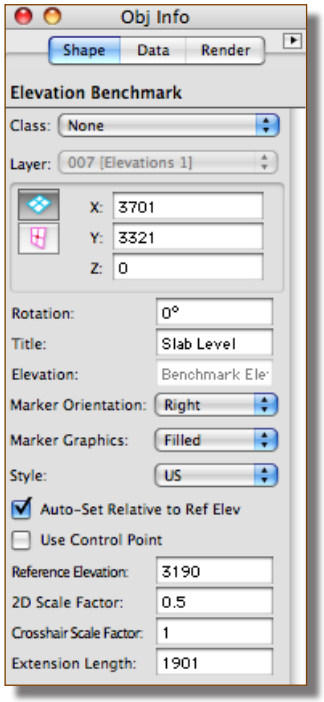
- Go to the **Object Info** palette.
- Turn on the option to **Auto-Set Relative to Ref Elev**. This can be used to make the Elevation Benchmark automatically update when you move it.



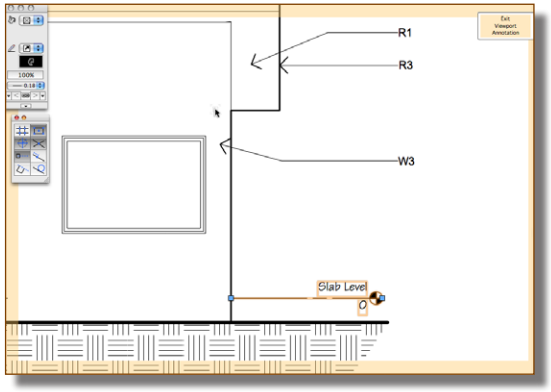
- Set the Reference Elevation on the Object Info palette so the Elevation Benchmark displays the correct height for you.



- Go to the Object Info palette.
- Edit the Title of the Elevation Benchmark.

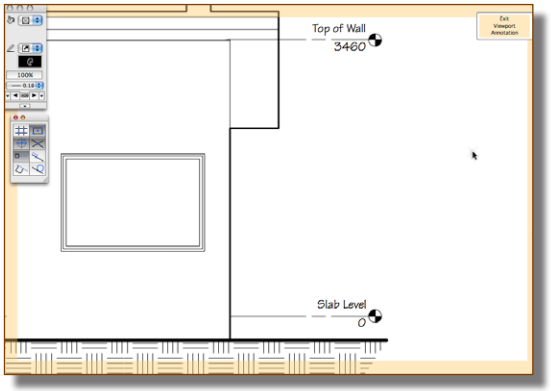


- Select the Elevation Benchmark.



- Drag a copy of the Elevation Benchmark to a new position, and it shows the correct level.

Beware, if use this in a viewport and you move the viewport up or down the screen, the Elevation Benchmark will change.

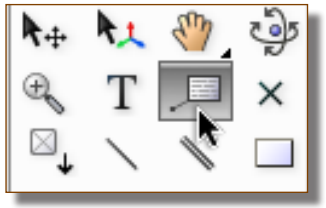


Callout tool

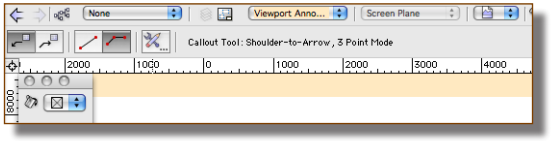
cadmovie422

The Callout tool is used to place notes on the drawing with arrows pointing to the object that the notes refer to. The callout tool can be used to place a note that you type in, it can be used to place a note from a database of notes, and it can use used to place keynotes.

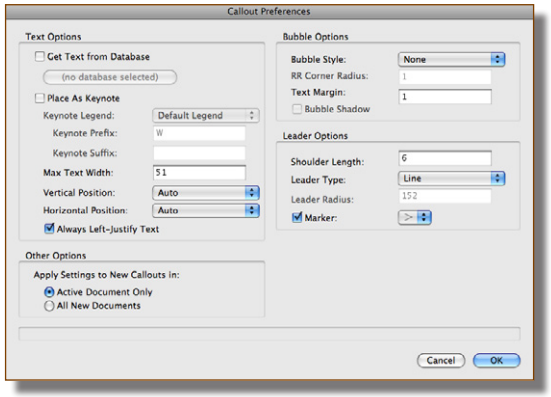
- Go to the Basic tool palette.
- Select the **Callout Tool** from the Basic Tool palette.



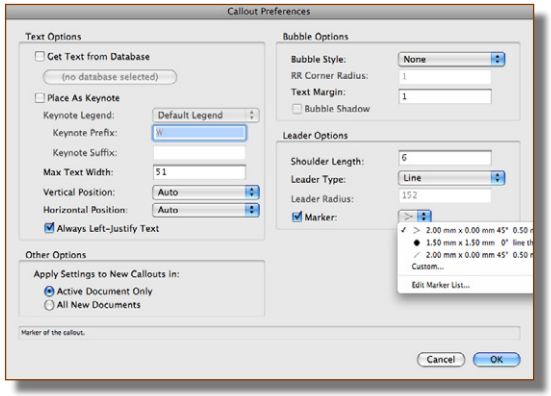
- Go to the Tool Bar.
- Click on the **first** mode and the **fourth** mode.
- Click on the Preferences button.



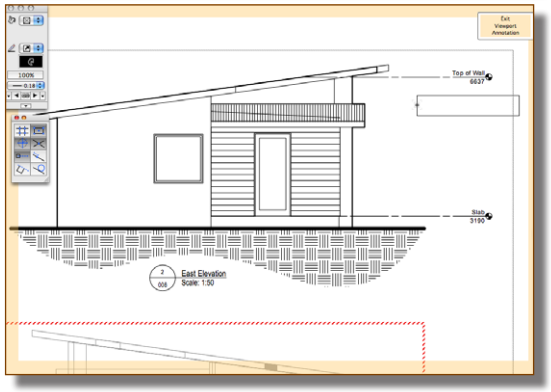
- These are your prefernces for the callout. The way the preferences are set in this image, Vectorworks will place the callout note directly on the screen, and you can Click once for the position of the text.



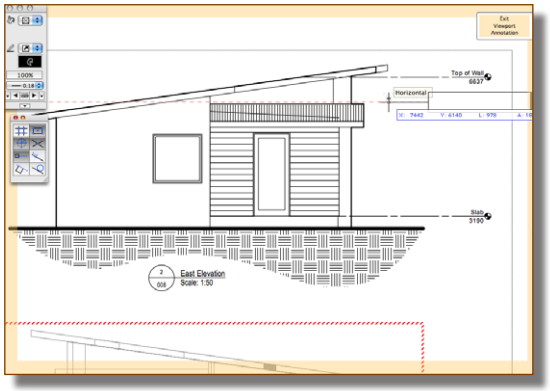
- I prefer to set the marker, using one of the already set styles that I have customized.
- Click on the **OK** button.



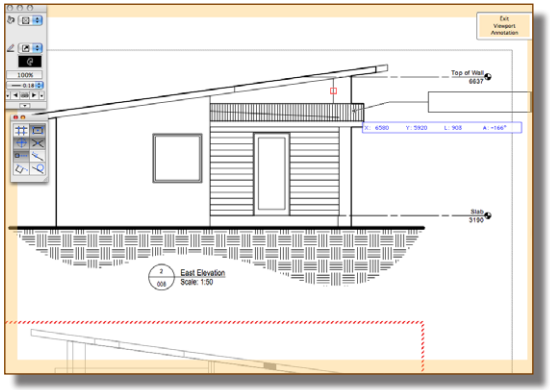
- Click once for the start of the block of text.



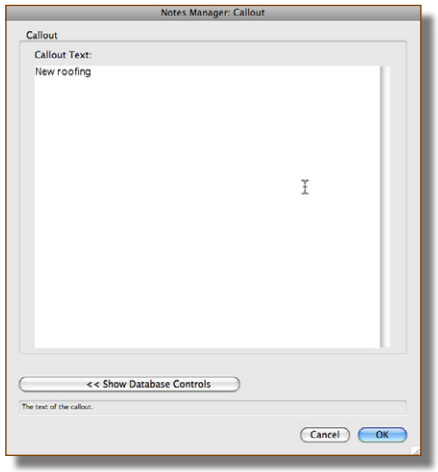
- Click once for the start of the angled line (VectorWorks calls this the shoulder).



- The last click is where you want the note to point to.



- Enter the text of the note in the dialog box.
- Click on the **OK** Button, or hit the Enter key (far right hand side of the keyboard).
- Completed Callout is placed on the drawing.

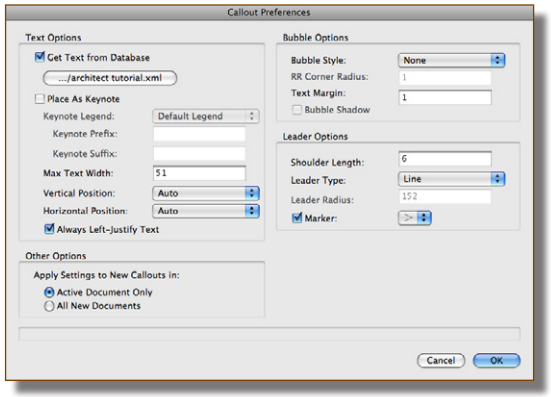


Any Callout note can be edited using the Object Info Palette by clicking on the Edit Note button, or by double clicking on the callout with the 2D Selection tool.

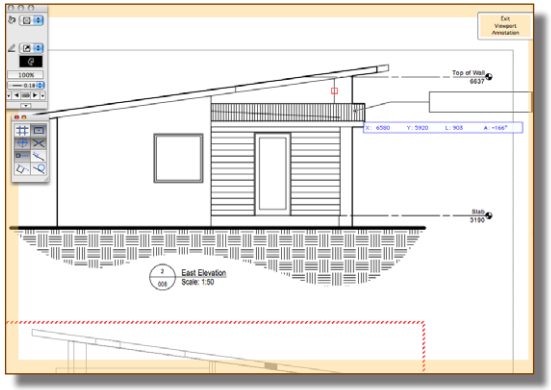
[cadmovie423]

You can use the callout tool like this if you want, but you are losing a lot of power. The callout works really well if you get the text from a database. This allows you to set up database of standard notes that you can use on job after job. You can even set up an office database so that several people can look at the same database and share notes.

- Go to the Tool Bar.
- Click on the **Callout Tool Preferences** button.
- Choose to **Get Text From Database**.
- The button under Get Text from Database is used to find your note database. This can be on a local drive or on a network computer.
- Click on the **OK** button.

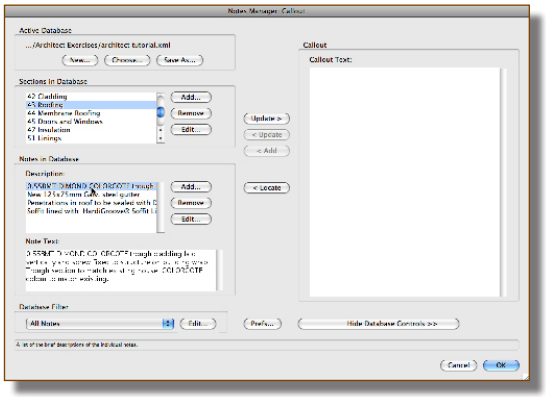


- Place the callout.

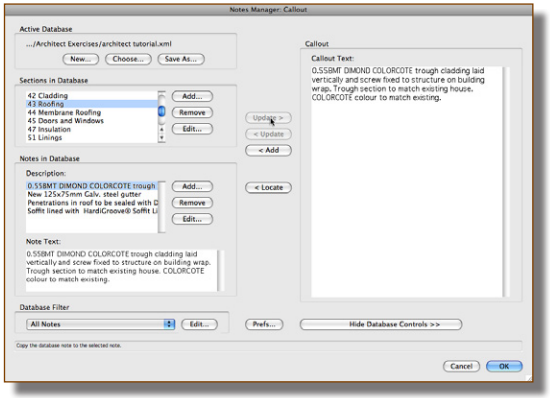


- When you get text from a database you get this dialog box where you choose the actual note that you want to place. You might notice that this database if different from the one that you have. I have created my own sections and notes here so that it's easy for me to find the notes I want.

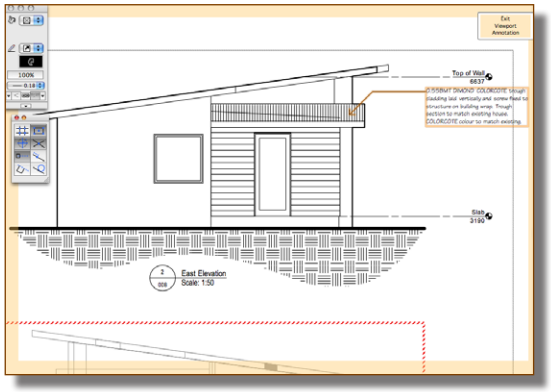
- Choose the Section.
- Choose the note.



- Click on the **Update >** button.
- Click on the **OK** button.



- VectorWorks places the note for you.

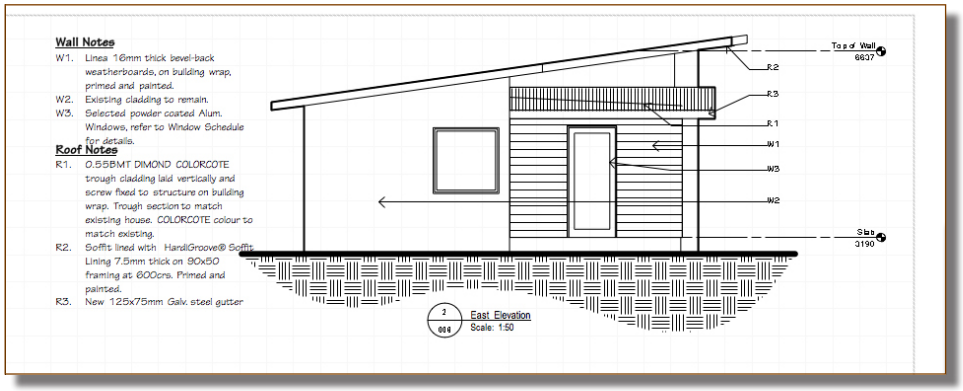


You can edit the callout on the Object Info Palette to change the notes, the length of the note and so on.

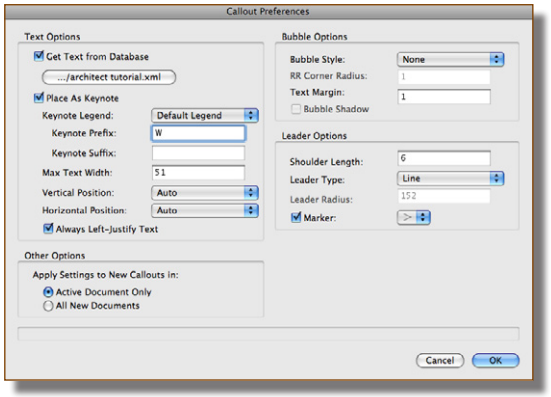
There is another powerful way to use the callout with text from a database. You can tell Vectorworks to place a keynote. A keynote is where you repeat the same key (W1) and have a block of notes on the draw that have the detailed expansion of the W1 note. This allows you to have several notes in a block and then have many instances of the keynotes on the drawing.

[cadmovie424]

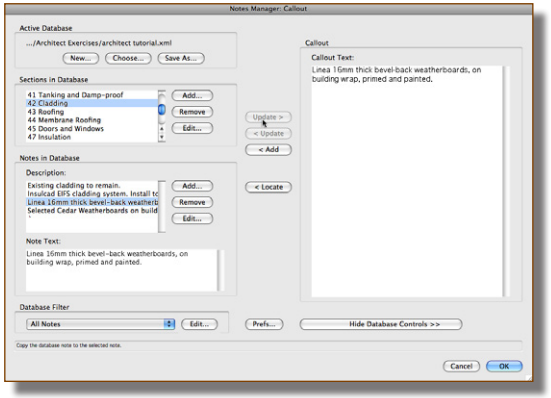
This works really well inside viewports.



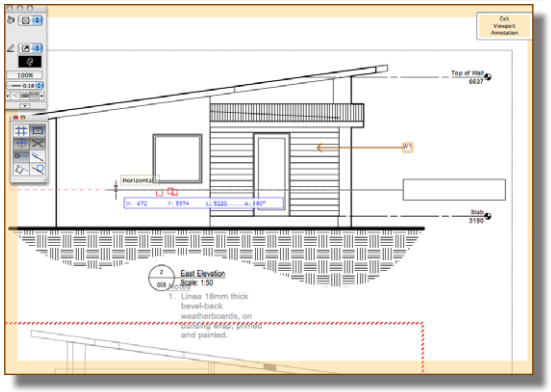
- Enter the Annotation of a viewport.
- Go to the Tool Bar.
- Click on the **Callout Tool Preferences** button.
- Choose to **Get Text From Database**.
- Choose **Place As Keynote**.
- Click on the **OK** button.



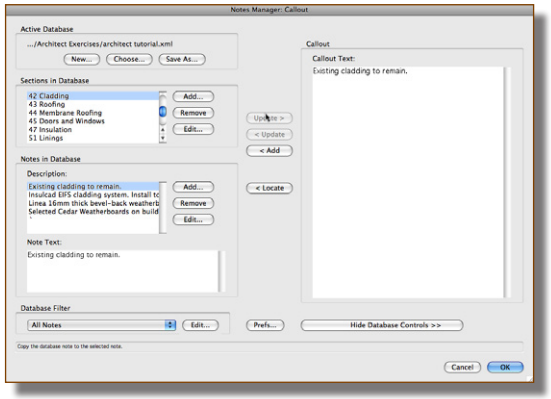
- Place the callout.
- Choose the Section.
- Choose the note.
- Click on the **Update >** button.
- Click on the **OK** button.



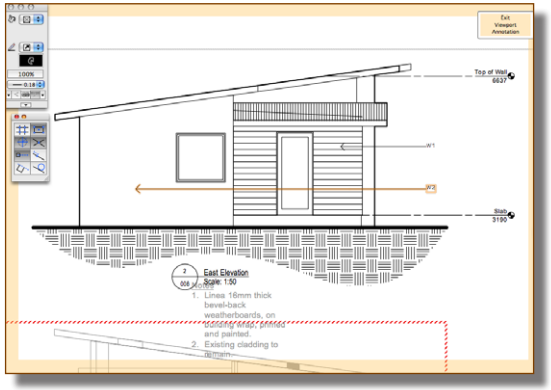
- Place another callout.



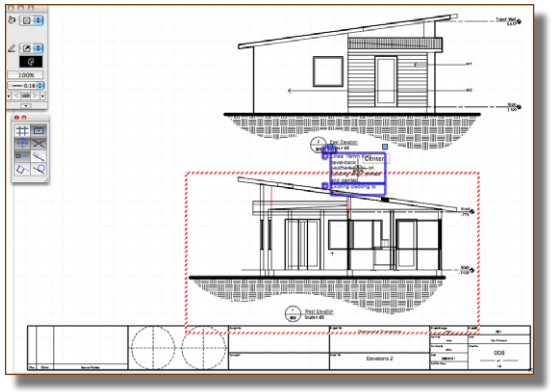
- Choose the Section.
- Choose the note.
- Click on the **Update >** button.
- Click on the **OK** button.



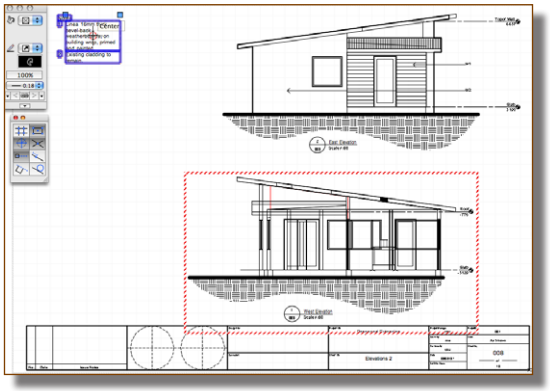
- Exit the Viewport.



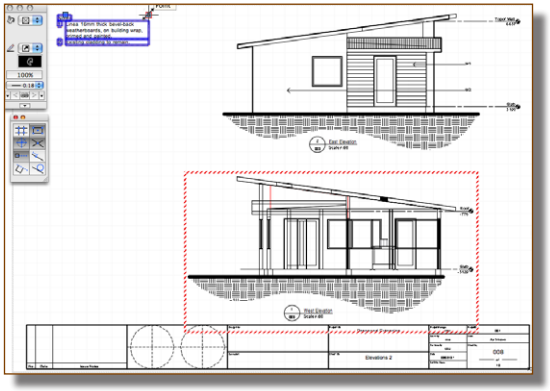
- Now you can see the block of notes outside the viewport.



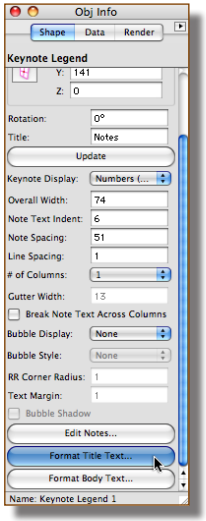
- Go to the **Basic** toolset.
- Select the **2D Selection** tool. Move the note block to the location you want.



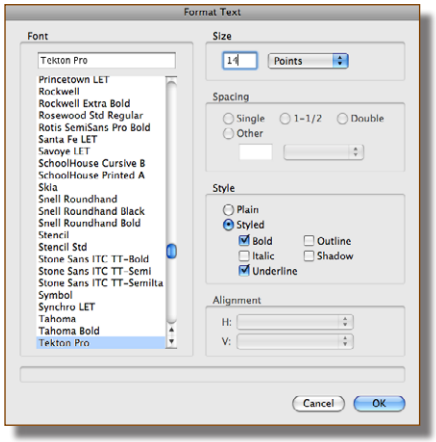
- Use the **2D Selection** tool to move the right-hand handle. This controls the width of the text block.



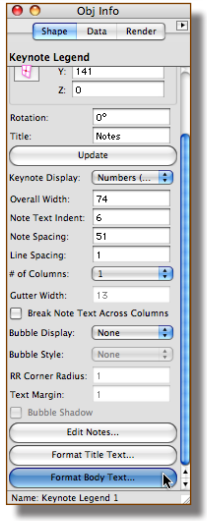
- Go to the **Object Info** palette.
- Click on the **Format Title Text...** button.



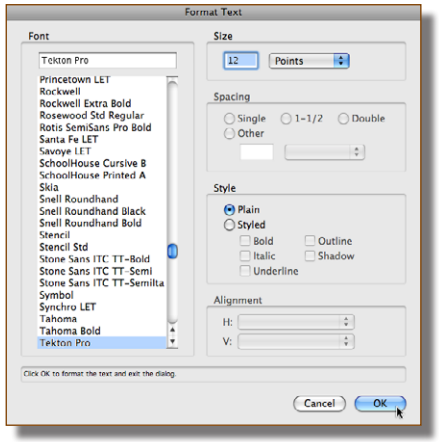
- Set the text formatting options.
- Click on the **OK** button.



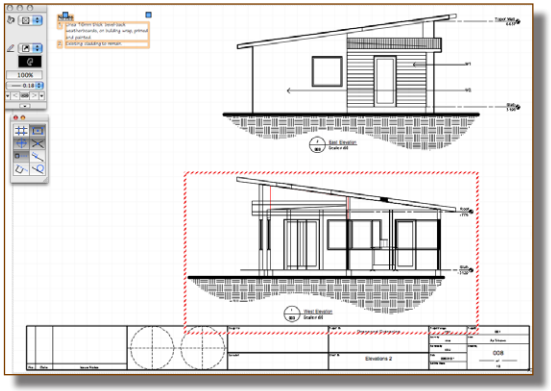
- Go to the **Object Info** palette.
- Click on the **Format Body Text...** button.



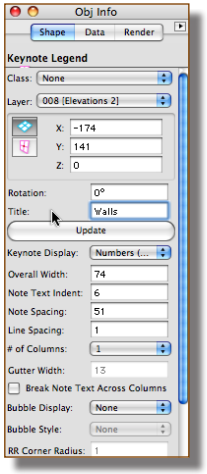
- Set the text formatting options.
- Click on the **OK** button.



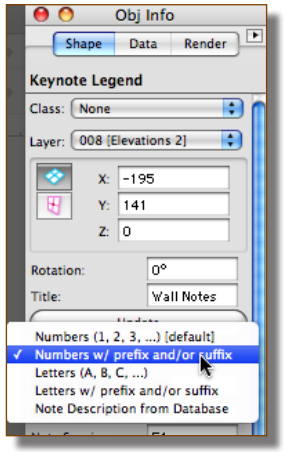
You will see the text block (keynote legend) update.



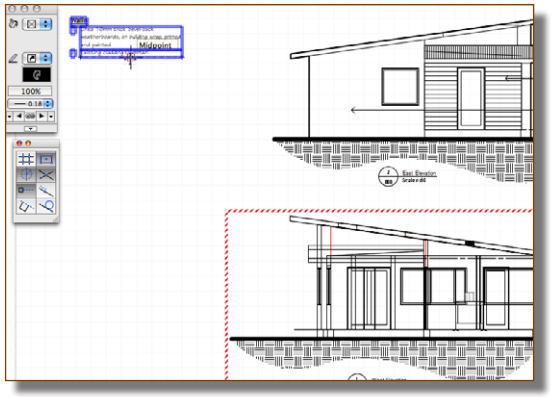
- Go to the **Object Info** palette.
- Edit the title to be more descriptive.
- This will update the title of your text block (keynote legend), and it will allow you to choose which block of notes your keynote belongs to.



- If you want to see the prefix on the keynotes, you have to turn this on with the Object Info palette.

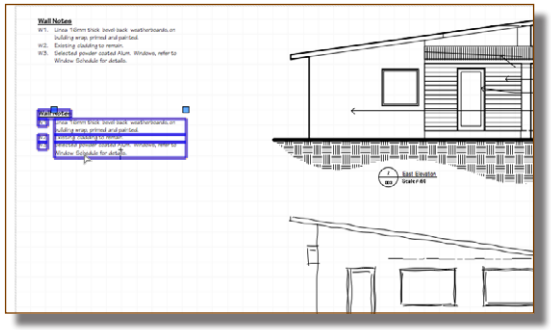


- Notice the Title on the Keynote legend has changed.

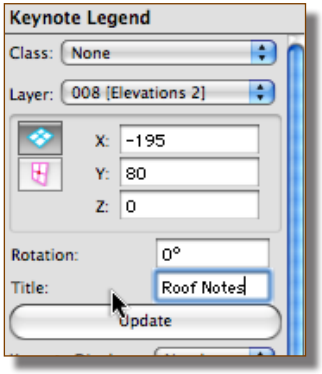


[cadmovie425]

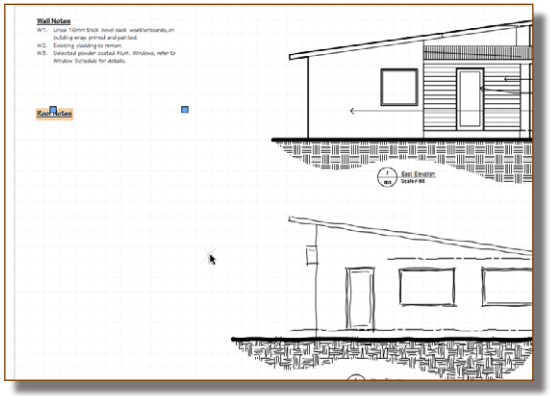
- Copy the block of notes to a new location.



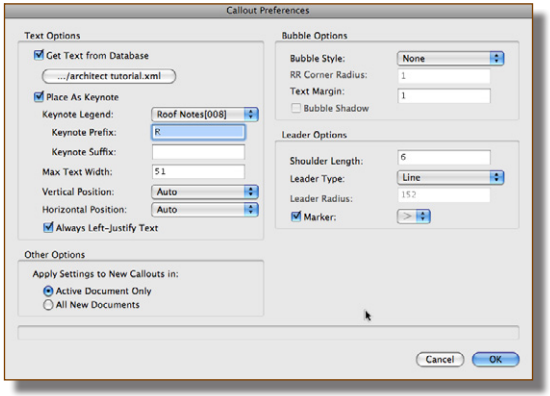
- Go to the Object Info palette.
- Edit the title.



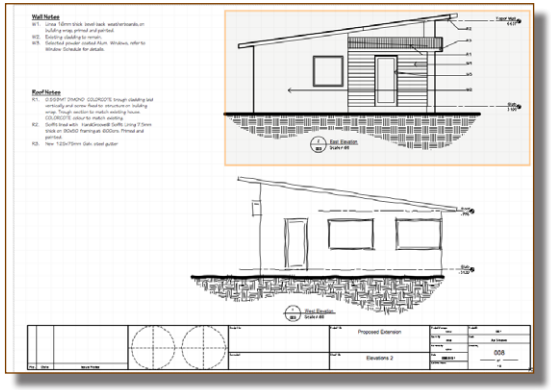
- The title of the Keynote updates.



- Go to the callout preferences.
- Choose the Keynote Legend from the pop-up menu.
- Enter the Prefix you want to use.



- Enter the viewport annotations and add all the keynotes you want.



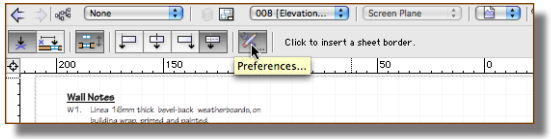
Now you can set up as many Keynote Legends as you want.

Standard Title Blocks (Drawing Border)

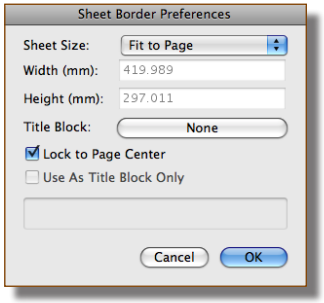
cadmovie426

- Go to the **Dims/Notes** Tool Set.
- Click once on the **Drawing Border** tool.

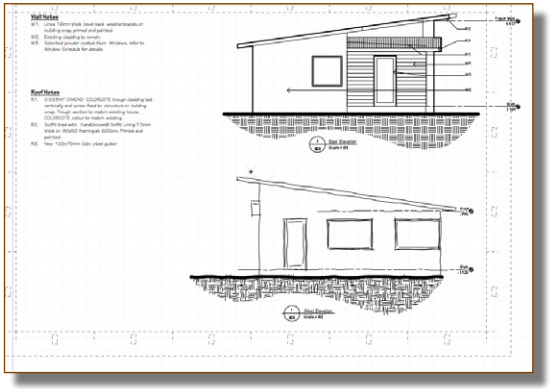
- Go to the **Tool Bar.**
- Click on the **Preferences...** button.



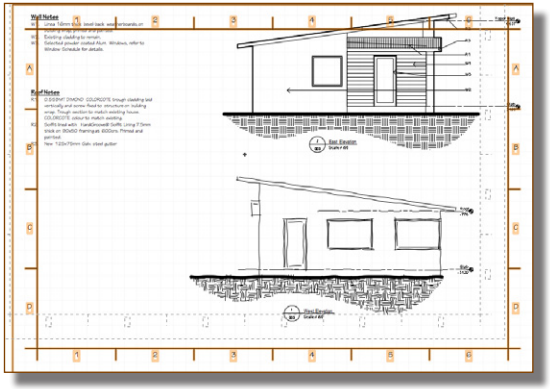
- Choose the option for fit to page size. You have already set up the page size for this drawing, so it's easier to use this rather than try to work out the printed page size.



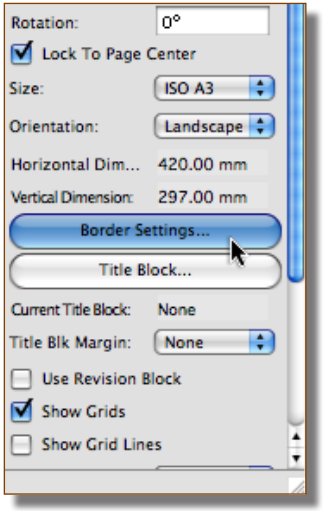
- Bring your cursor into the drawing area.



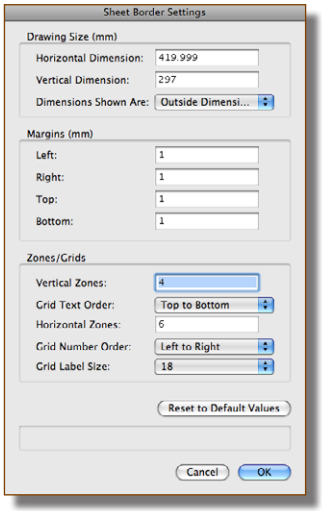
- Double click in the drawing area. VectorWorks will place the drawing border to fit the page.



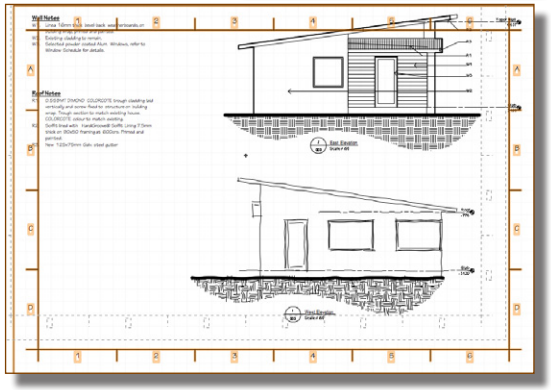
- Go to the Object Info Palette.
- Click on the **Border Settings...** button.



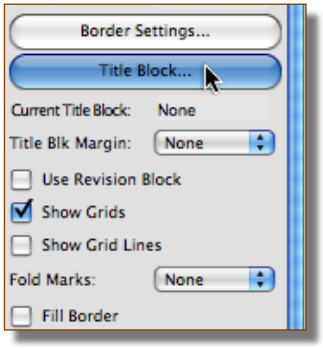
- Change the border settings to suit your drawing style. I've used 1mm, if you are using imperial, try 1/32".
- Click on the **OK** button.



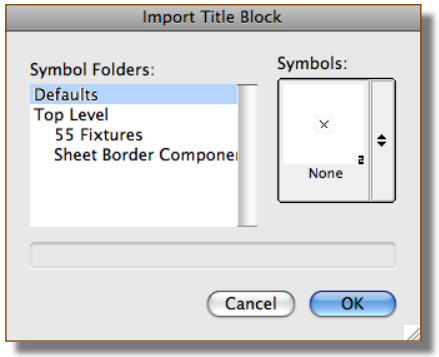
- VectorWorks changes the border to suit the settings.



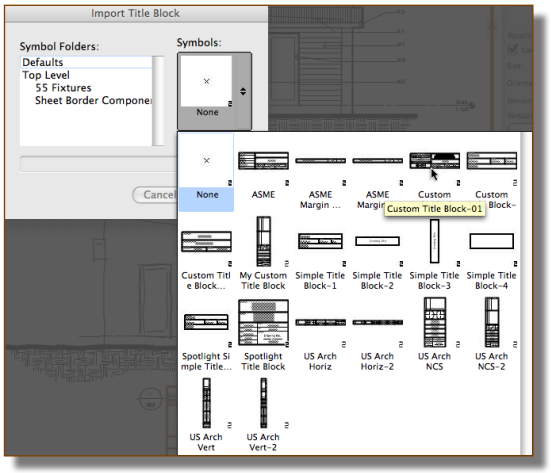
- Click on the **Title Block...** button.



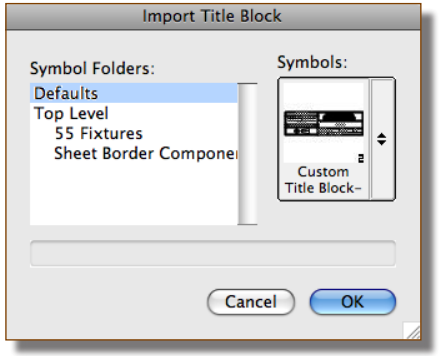
- This allows you to choose a sample title block to use as the basis of your title block.



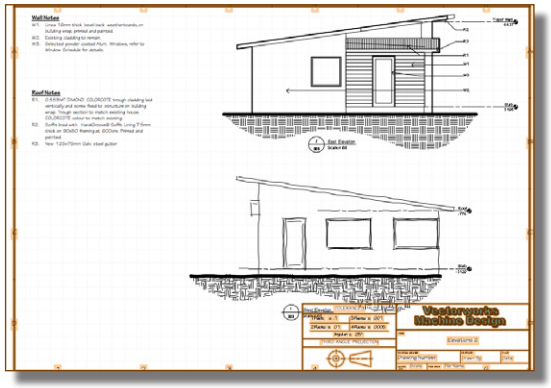
- All these sample title blocks are stored in the VectorWorks folder and they can be edited to suit your company.



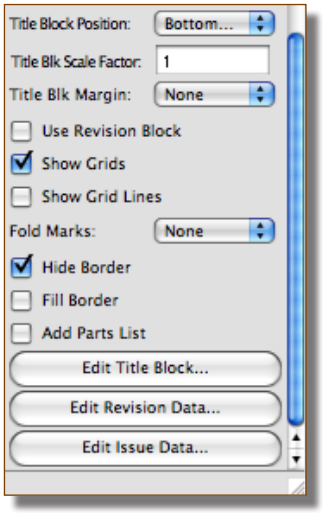
- Choose the **Custom Title Block -01**.



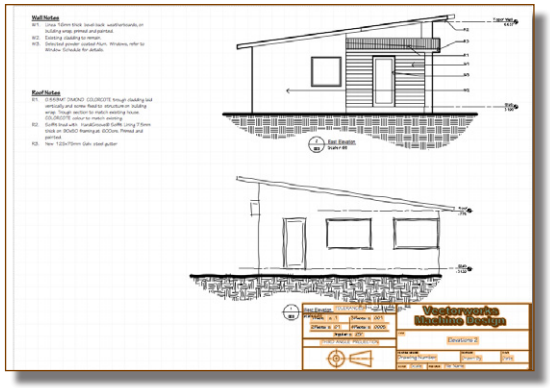
- You can see your title block on the the drawing. You can change the scale of the title block by using the Object Info Palette.



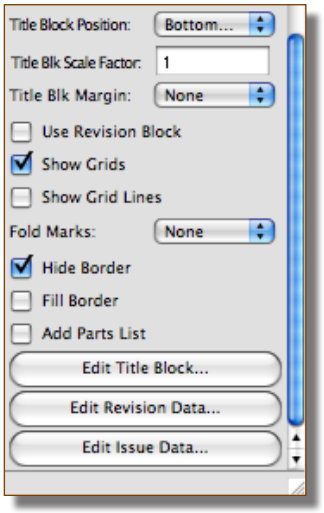
- Go to the Object Info palette.
- Click on the Hide Border option.



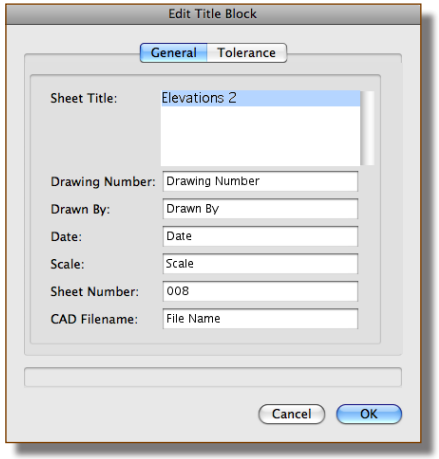
- This will give you a title block with no border.



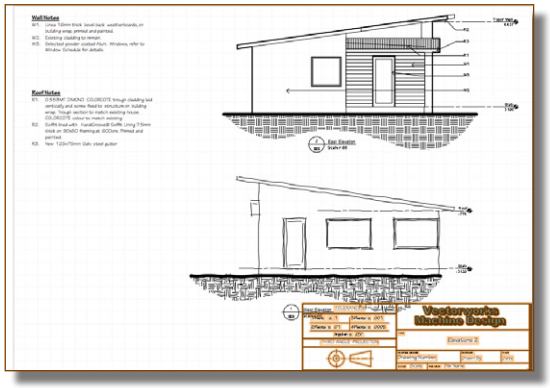
- Go to the Object Info palette.
- Click on the **Edit Title Block...** button on the Object Info Palette to edit the text on the title block.



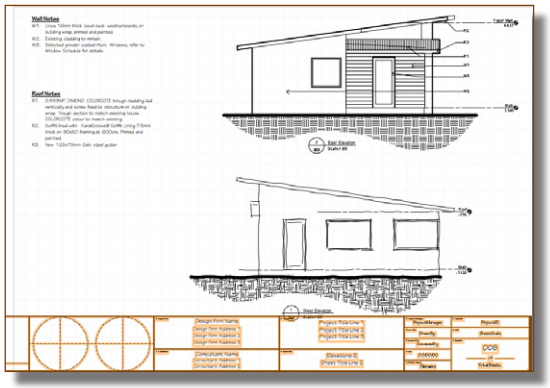
- Fill in the drawing name, drawing number and so on. You might notice that with Vectorworks 2010 the Sheet title is filled in for you
- Click on the **OK** button.



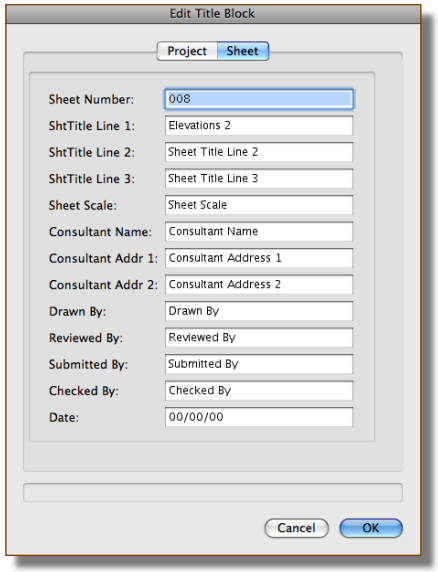
- VectorWorks fills in your title block.



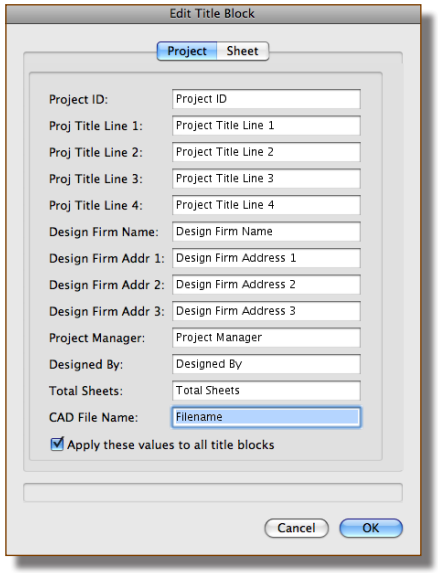
- There are other title blocks available.



- The other title blocks have more information on them, and a different dialog box.
- This tab shows all the information that relates to the Sheet.



- This tabs shows the information that relates to the entire project. When you fill in the information here, it can be applied to all the title blocks. There is an option at the bottom to make this happen.



Custom Title Blocks

If you want to use a standard title block, that is OK, but most people don't want to use a standard title block, they want a title block like the one that they use at the moment, the title block that they have designed.

The drawing border uses a symbol from one of the special files in your VectorWorks folder. If you edit one of these symbols then all your drawing borders can use this symbol. The files are located in your VectorWorks folder:

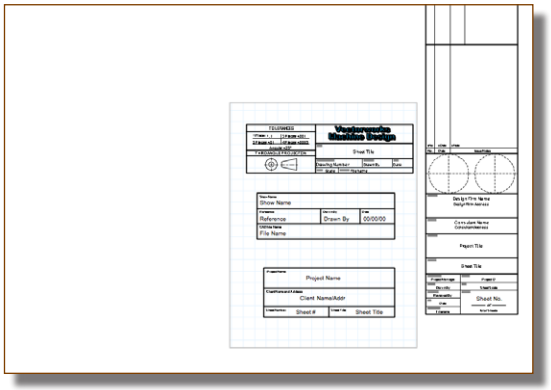
On a Macintosh the file is in the folder:
Applications: VectorWorks: Libraries: Defaults: Drawing Border - Title Blocks:

Or
Applications: VectorWorks: Libraries: Defaults: Sheet Border - Title Blocks:

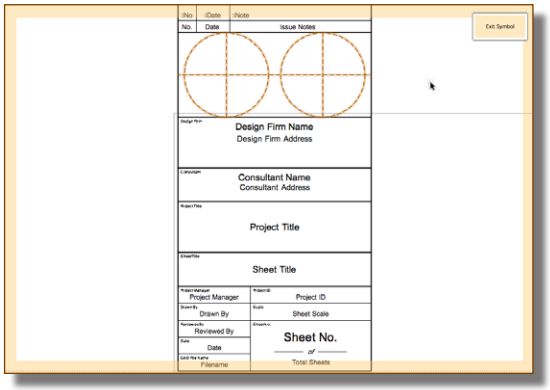
On a Windows Machine store the file in:
Program Files\ VectorWorks\ Libraries\ Defaults\ Drawing Border - Title Blocks

Or
Program Files\ VectorWorks\ Libraries\ Defaults\ Sheet Border - Title Blocks

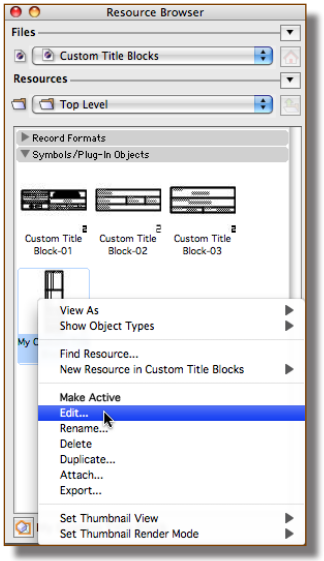
- Open on of the files from the Drawing Border folder or Sheet Border folder.



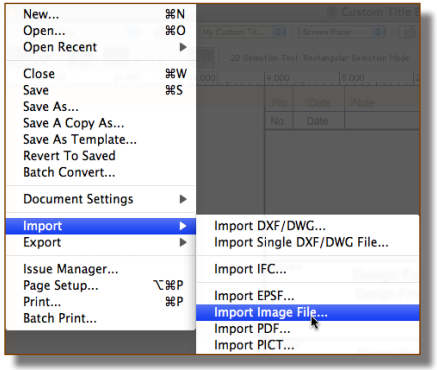
- Delete the parts of the title block that you don't want.



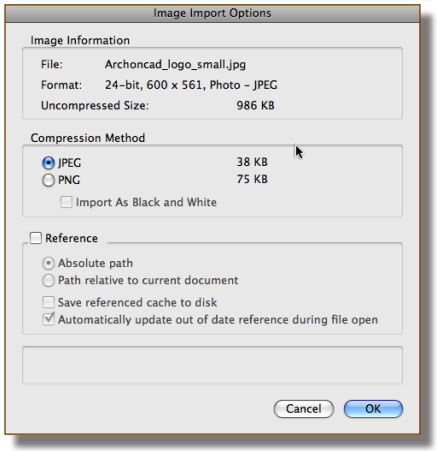
- Go to the Resource Browser.
- Locate the title block symbol you want to edit.
- Right mouse click on the title block to edit it.
- Choose to Edit the 2D part.



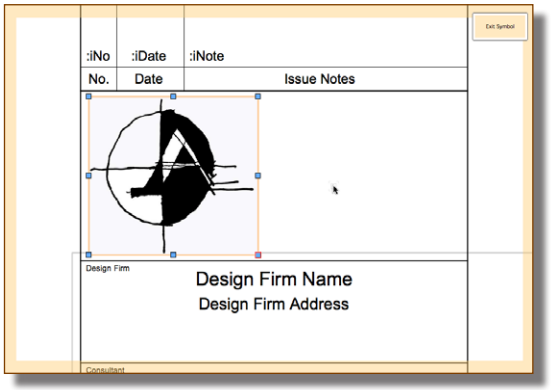
- If you want to use your company logo, use the import image command from the File menu.



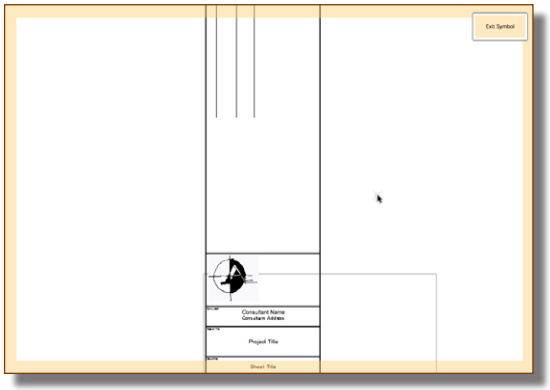
- Import your graphic logo, do not reference it.



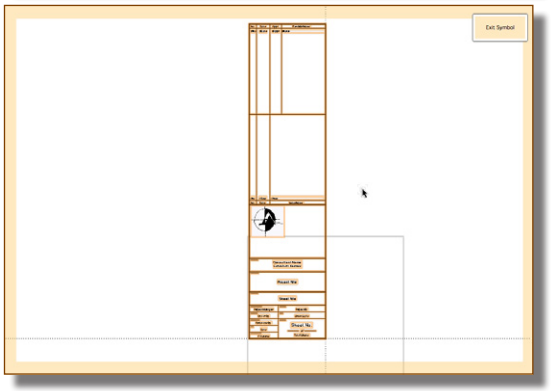
- Re-size the logo, add your company information, address and phone number



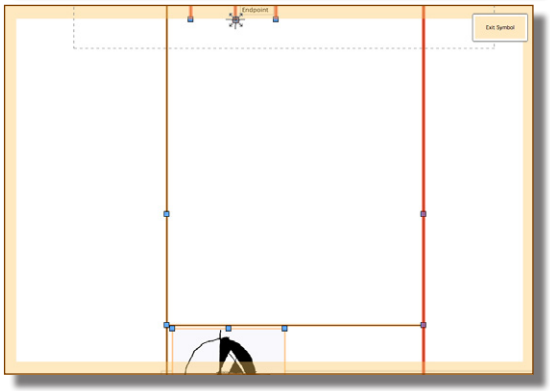
- Delete the parts you no longer want.



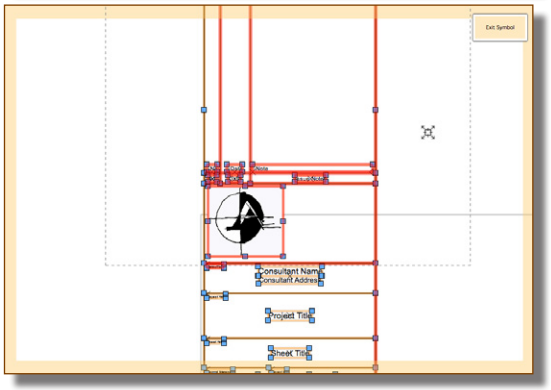
- Stretch the title block outline to suit the revised title.



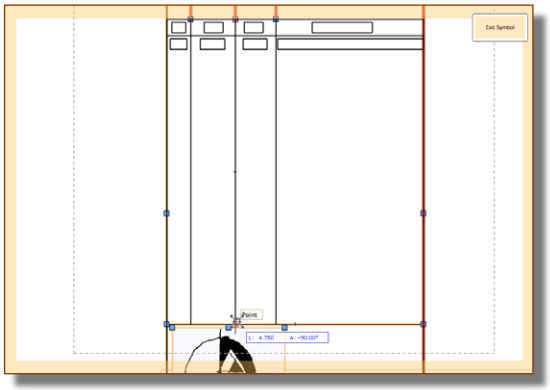
- Use the 2D Reshape tool to stretch parts.



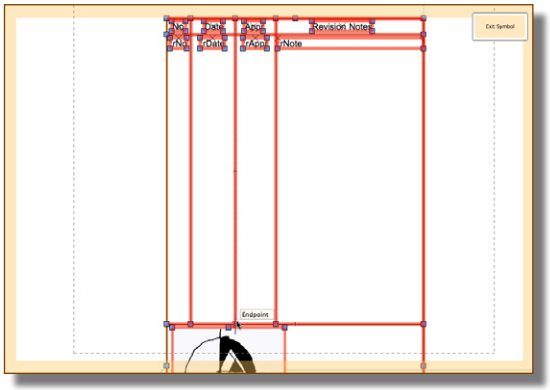
- Use the 2D Reshape tool to stretch parts.



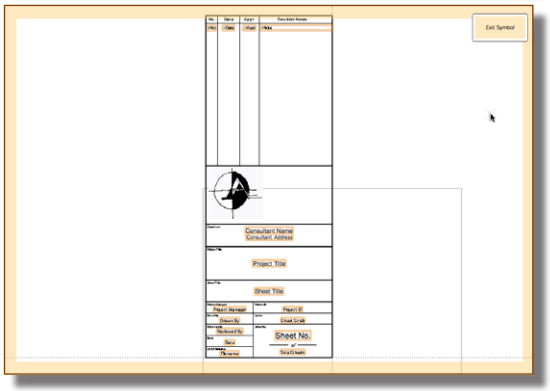
- Use the 2D Reshape tool to move parts



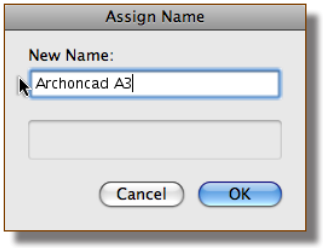
•. Get everything the way you want.



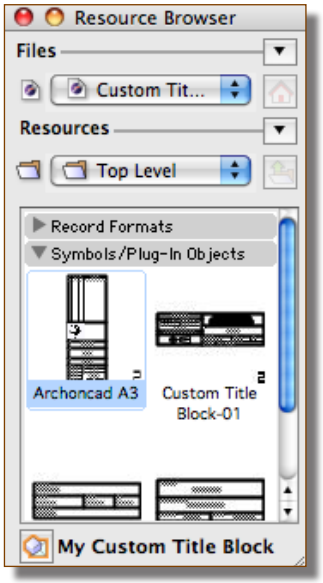
- Do not delete the part Of text shown selected. these are the bits that connect to the dialog box. If you delete these parts, they are really hard to re-connect.
- Exit the symbol. Use the **Exit Symbol** button at the top right hand side of the screen.



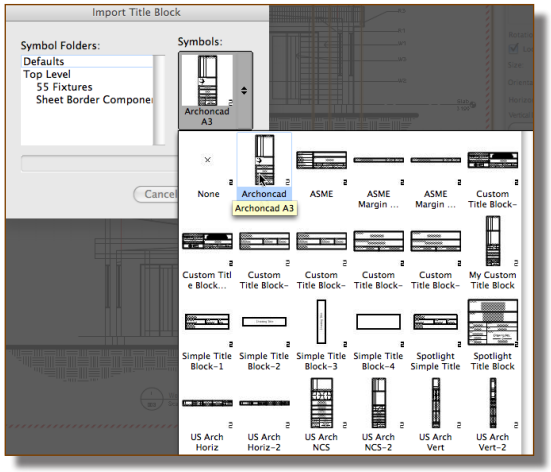
- Save the file.
- Rename the symbols so you can find them easily.



- Any new files can use this title block straight away.
- When you use the drawing border tool again you can choose your custom title blocks as if they were VectorWorks standard title blocks.



- Use the Object Info Palette to change the title block, click on the Title Block... button.
- You should see your new title block in the preview pane on the right hand side.
- The drawing border will show your new title block.



The ideal way to deal with this is to edit the folder that contains the default title block symbols to hide all the files that you don't need. Edit the symbols to suit you. Don't delete any of the text on the symbols that you want.