

OzCAD Tech. Note 002 — Importing DXF/DWG files into VectorWorks 12

Consider this anecdote from a VectorWorks user:

To be honest, I have more problems translating AutoCAD files from consultants/architects into something usable in AutoCAD than I do with VectorWorks. In my job, I'm basically the 'portal' through which all our project files come through, and every project has one (or two) architectural firms, a surveyor, interior architect, structural, mechanical, electrical and fire protection engineers, landscape architect, etc.

Every one of them sets up their AutoCAD drawing differently, plots it differently, and uses different .ctb files (and always forgets to include them). Some work totally in *Modelspace* (and plot from there) and some work in *Model/Paperspace*. Some scatter floor plans and drawing parts all over the (model) place, some stack them neatly on top of each other (like they should) and use viewport display settings. Some use multiple *Layout Tabs*, some don't. Most draw to scale, some don't. Some plot using *Layout*, some using *Extents*, others using *Display*, etc.

So not all DWG files are created equal! With this in mind here are some tips for making file translation easier. Before we begin, here what you should try to obtain before attempting to import the file:

1. A hard copy or PDF of the original file.

Knowing how a drawing is supposed to look is a sure fire way of discovering if there have been any errors in the translation. It can also reveal things like page size, scale and units.

2. The units that were used (if any).

Drawings are typically done in millimetres or metres, however it is possible to draw in AutoCAD without actually specifying the units that were used. In this case VectorWorks 12.5 will try and guess the units if the import option is set to *Determine Automatically*. Take a look at the guess VectorWorks has made and if it is something like:

1 DXF Unit = 25.4mm

(and the DWG file didn't come from the USA), chances are the guess is wrong. In this case, choose *Metres* or *Millimetres* (based on the information you have about the file) from the menu that says *Determine Automatically*.

3. The AutoCAD version that was used to create the file.

This will enable you to ascertain whether the version of VectorWorks that you have is compatible with the AutoCAD version used to create the file. Currently VectorWorks 12.5 can import up to and including AutoCAD 2006 files. If you can't obtain this information, there is another way to determine the AutoCAD version. Open the AutoCAD file in any text editor and the first six characters will reveal the version. Use the following table to decipher the number:

AC1002 = release 2.5	AC1012 = release 13
AC1003 = release 2.6	AC1014 = release 14
AC1004 = release 9	AC1015 = release 2000/2002
AC1006 = release 10	AC1018 = release 2004/2005/2006
AC1009 = release 11/12	AC1021 = release 2007

4. The .ctb file that defines the line weights/colours.

If you can get the .ctb file then the translation will be more accurate as the line weights will be correctly translated.

Even though AutoCAD can apply true line weights to objects, there are many users who still use the old method of line colours, so the .ctb file is still required.

5. All files that are referenced by the main file. Alternatively, you can ask that Xrefs be bound to the main file.

It is very common for AutoCAD users to reference or link one or more files into their main drawing. These are called *Xrefs* and if you have been sent a DWG file that utilises them, you will also need to get these files so that VectorWorks can import the data from them. If you prefer to get only a single file from your consultant or client, then you can request that they bind the *Xrefs* in with the main DWG file.

Assuming that you have all the required information, DWG files etc., let's look at the rules for importing.

1. Always import into a blank VectorWorks drawing.

Never ever import a DWG file into a working file. Not only is there a risk of bringing in a corrupt object, but you will bring lots of unnecessary classes, symbols and objects into your drawing and have to carry this baggage for the remainder of the project. It is also not possible to scale an imported drawing if there are other objects already in existence.

Instead, open a blank file with the correct units (perhaps a predefined template file), then do the import.

2. Always use the Import Single DXF/DWG File command.

Using this command to import the file will ensure that the *Import DXF DWG Import Options* dialogue appears and act as a reminder that there are things to check and set.

3. Consider converting the file to 2D on import.

If you only require 2D information from the file, consider choosing *2D* from the *Convert Objects To* drop down menu. This will make the drawing more convenient to work with and see things like associative hatches correctly applied. It will also generally do a better job than you can do with the VectorWorks convert commands.

4. Consider the drawing scale.

DWG files do not contain drawing scale information. What VectorWorks does to try and guess the correct scale, is to look at your current page size, then fit the whole DWG file onto that page. Given that DWG files typically have stuff scattered about and that you have not matched the page size in VectorWorks to the DWG file, this is a pretty inaccurate way of ascertaining the drawing scale. Consequently, it is much better for you to click the *Scale...* button and set the *Model Space* scale to what you know the drawing to be.

5. Always clean up the imported file.

Once the DWG file is successfully imported yet isolated from your working file, you have the opportunity to clean it up. This might involve:

- Deleting or merging classes, which is easy with multiple selection in the *Organization* dialogue. Select multiple classes, click the *Delete* button, then choose the class for the objects to go into. Alternatively, you can also delete the objects.
- Removing unneeded *Sheet Layers* or *Viewports*, consultant's title blocks, drawing notes, unneeded geometry, etc.

- Converting whole floor plans that are symbols to groups to make their geometry easier to access (choose *Don't convert sub-objects to groups* in the *Options* dialogue).
- Purging unused objects using the *Purge Unused Objects* command in the *Tools* menu. This can dramatically reduce file size and reduce clutter in resource lists such as symbols and hatches.
- If the origin in the file is unimportant, *Select All* then reposition the objects to a convenient location on the page.

Once the file is cleaned up and ready for use, there are two ways you can transfer this information to your working file.

1. Using Copy and paste.

This is the simplest way to transfer data. Start by preparing the destination layer in your working file, giving it an appropriate name and the same layer scale as the imported drawing. In the cleaned up file, *Select All* then *Copy*. Switch to your working file then choose *Paste in Place*, or right click and choose *Paste* to centre the copied objects in a specific location.

2. Using the Workgroup Reference Command.

This command allows you to transfer one or more layers from one file (often called the *Source* file) to another file (often called the *Destination* file), then update the information in the *Destination* file if or when changes are made to the *Source* file. You can also break or remove a reference, but choose to keep the referenced information, thereby transferring it from one file to another. *Workgroup Referencing* (WGR) is particularly useful when a DWG file you have imported is likely to be updated during a project. In this case you can reimport and clean up a new version of the file into your *Source* file, then update the WGR to have the newer information displayed in your *Destination* file.

So what happens when you take all these precautions and the imported information is still the wrong size? As mentioned previously, this can be the result of an incorrect or absent setting in AutoCAD, so the first thing to do is to establish what the error is.

To do this you need to measure something in the drawing of a known size. If there is a scale bar, great. If not, then try to find a dimensioned object. I find drawing a line is the easiest way to measure something. Once done, you can read or copy the length from the *Obj Info* palette (click the Polar Coordinates button to get the readout in *Length/Angle* format).

Now, let's say you know something should measure 3565mm, but it scales at 3.565mm. From this you can immediately see that the drawing is 1000x too small, so there was a mismatch of units in the import dialogue. It was incorrectly set to:

$$1 \text{ DXF Unit} = 1\text{mm}$$

when it should have been set to:

$$1 \text{ DXF Unit} = 1000\text{mm}$$

In other words, the drawing was done in metres but imported as millimetres. The simplest solution is to just do the import again into a fresh file. If you get a factor like 24.5, 39.37, 0.03937, then it is a metric/Imperial issue and again just a mismatch in the units setting in the import dialogue.

If the known measurement scales at something completely arbitrary however, then a different strategy is needed. Say the known measurement is 3565mm, but scales at 287.3mm (make your measurement more accurate by temporarily increasing the

number of decimal points in the *Units* dialogue). The drawing needs to be scaled by the following factor:

$$3565 / 287.3$$

(always put the known size over the measured size). You can do this by deselecting everything then choosing *Scale Objects* from the *Modify* menu and entering exactly the same text:

$$3565 / 287.3$$

then clicking *OK*. VectorWorks will do the maths for you. If you have one or more objects selected, then you will have to check the *Entire Drawing* box yourself. If you do this, **make sure you remember to uncheck it** if you come back later to scale a single object.

Now repeat the measurement of the known object. If it scales correctly the drawing is fixed. If it is still slightly off you can repeat the process, but remember that drawings are not always drawn with accuracy, so measure a few different lines or objects. I have seen survey drawings where line lengths do not correspond to written dimensions so the moral is to never assume anything.

Of course the required accuracy of an imported drawing depends on its use. If you are simply overlaying some plants for a landscape drawing, then accuracy to 0.1mm is probably not that critical. If you are doing 1:1 setout drawings for complex machinery, then accuracy is more paramount.

The last piece of advice I have is to record what you did for each consultant or client that you received DWG files from. This way you have a much better chance of repeating a successful workflow when receiving drawings from them in the future.