Some Helpful Hints on working with dxf.

It is best to open a dxf in a CAD program.

Drawing programs, such as Corel or Sketchup, either will not open a dxf, not open a dxf well, or not allow proper manipulation of the dxf. Also CAM programs, such as ArtCAM or Vectric, will usually open a dxf BUT are an extremely time consuming and difficult way to manipulate a dxf. They also have a very limited work area and some of my dxf are simply too large to completely fit into the space provided by certain CAM programs.

So, it is best to open a dxf in a CAD program.

Once you open the dxf in a CAD program, you will see that I have tried to lay out most of these dxf into easily printable “pages”. This is to assist both builders that will be cutting with a scrollsaw, and/or the builders that will be cutting with a cnc or laser ~ of course, if you are not scrollsawing the full size patterns, printing is usually not necessary since you will be loading the drawings into a CAM program, but either way a printed page is still a nice thing to be able to refer back to.

However, all builders will want to print out the Arbors and Spacers page to get the exact sizes of the metal parts.

I recommend using DeltaCAD since it is an easy CAD program to learn with great tutorials. It is a powerful CAD program that has everything we need, and very little that we don’t. It’s also an inexpensive program, and best of all, they offer a free, 45 day trial. You can get this free download and print from my dxf by going to [www.deltacad.com](http://www.deltacad.com)

Here is how to use DeltaCAD to print a whole page, or just a single part…

Select the "view", and with that first box on the left, select the area you want to work with ~ this can be the entire page, or a single part.

Then with the "select" choose the page or part that you would like to print.

With an area selected go up to "file" and select "print".  Tell it to print.  Go to "region", "set to last select", and print scale of 1.

Hit "ok" and "ok" and it will print for you. Be sure that your printer is also set to 1:1. There is usually a one inch square somewhere on the drawing inside the dxf to check to see if your printer is printing at the correct size.

CAM

If I am going to import a part into my CAM program (I use VCarve Pro from Vectric), I measure the height and width of the part, or of the combined parts, so that I know what size to set the work area in the CAM program. I then set the work area in the CAM program, import the part(s), and center the parts in the work area.

Next it is important to remove any superfluous parts of the drawing ~ things that you don’t want your cnc or laser to cut, such as the PD - if the wheel and/or pinion have one. The PD is that circle around the wheel or pinion half way down the teeth. The PD is only an alignment line. When the PD of the wheel and the PD of the pinion just touch each other, the centers between the two parts are correct. Some of my wheels and pinions are not shown with their PD, and some are. If there is a PD, remove it now because it will be difficult to remove it after the next step.

Next we want to close any open vectors. Select the drawing that you are working with in the CAM program, and click on the “Join Open Vectors”, or “Close Vectors” option and reset the “Tolerance” to 0.02, then click “Join” and then “Close”, and your drawing is now ready to create the toolpath.