



Unleash your Router's 3D Capabilities

How to cut out simple lines in ArtClip3D?

You just installed ArtClip3D and you want to cut something right away to check if it can drive your machine or if it needs a post tune up. This tutorial guides you through the process to cut a simple rectangle and a simple circle.

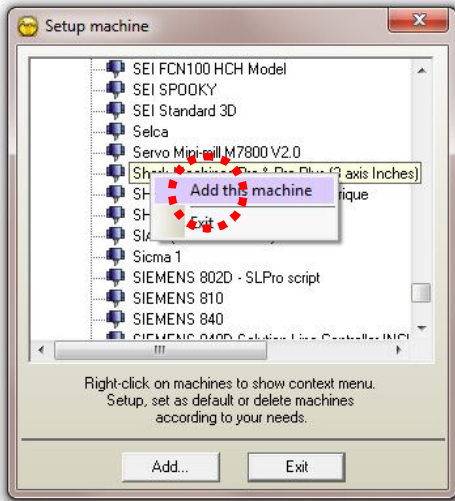
- Opening ArtClip3D



On the Desktop, an ArtClip3D icon is displayed, double click it to start the Software.

At the first opening only, you are requested to install a machine. If you received a postprocessor from your reseller you will need to copy/save it in the POSPROTE directory of ArtClip3D on your C drive (C:\ArtClip3D_V10). Open Windows Explorer to proceed.

If you did not receive a postprocessor with your purchase, then go into the list of machine and pick the machine name corresponding to your CNC. If your machine is not listed then contact your reseller so it provides you with one.

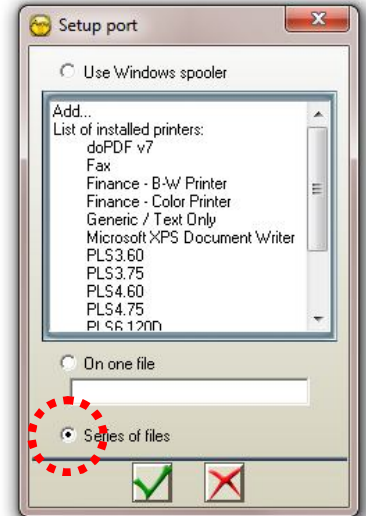


When the machine is shown in the list right click on it to "Add this machine", the Setup port windows pops up to define the type of output (right picture). If your machine runs from the windows spooler chose the first selection and the Generic/Text only.

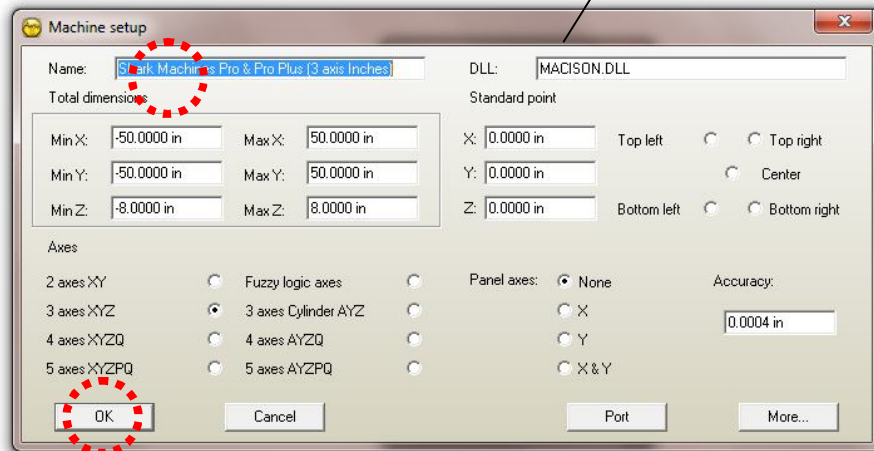
If your machine runs with G-Code files (NC, TAP, CNC, ISO, TXT etc...) select the "Series of Files".

For any other case you may be in select the "On one File".

Validate with the tick sign and the Machine Setup window comes up.



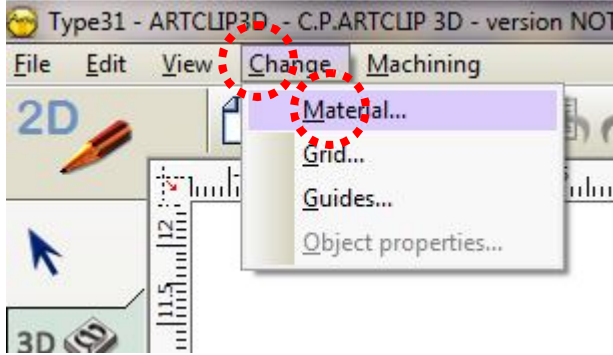
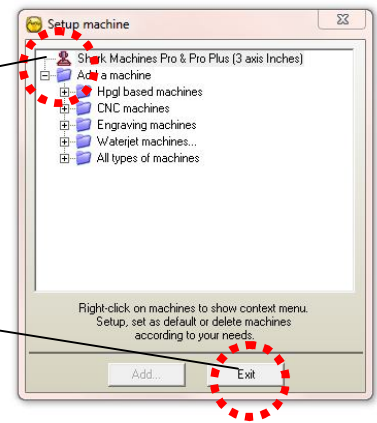
Unless the size would be too small you do not need to change anything in this window. Click the OK button.



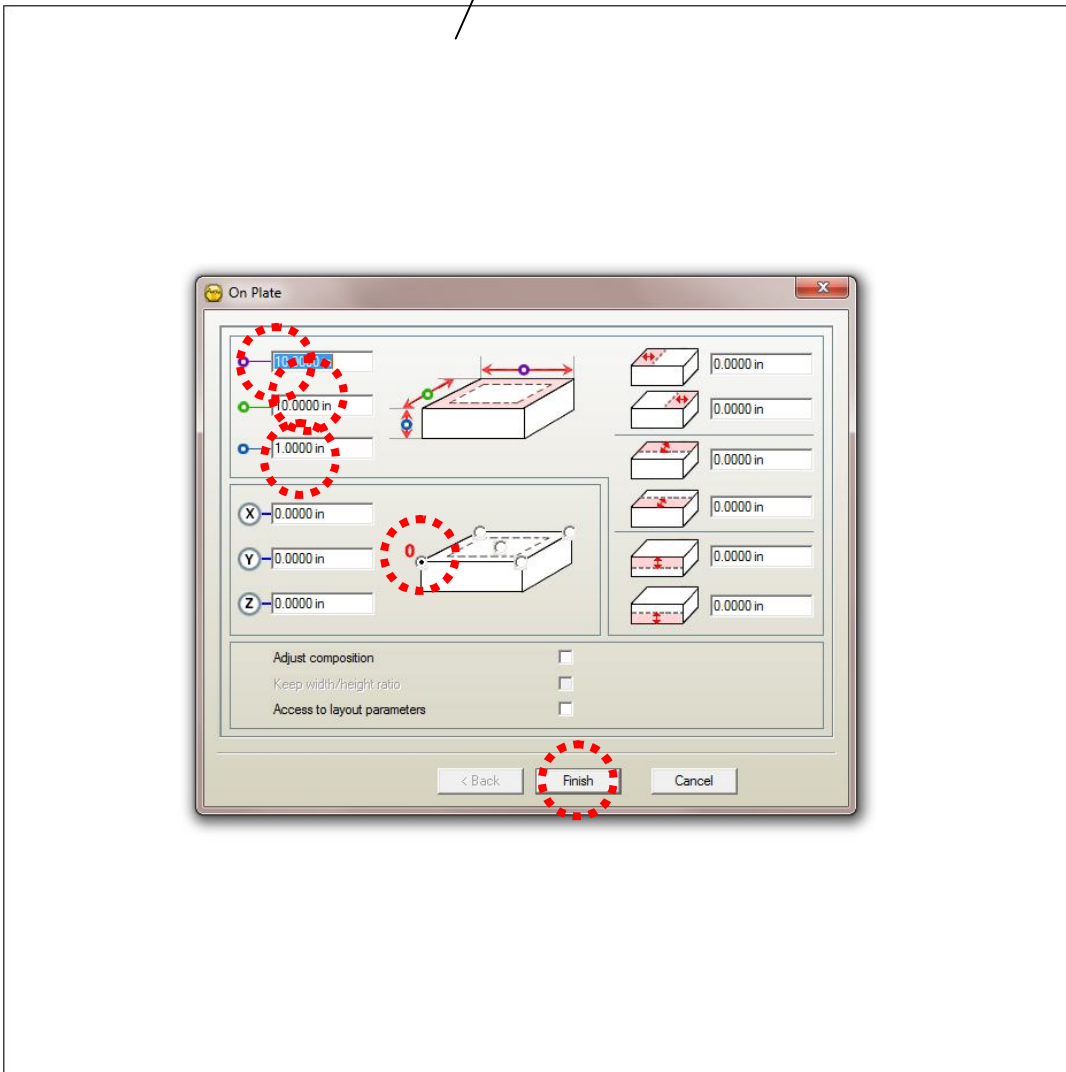
Your machine is now installed in ArtClip3D and your Software is ready to create toolpaths to send to the machine controller.

For further information on installing machine and add another one please read the tutorial **How to add another machine to run with ArtClip3D?**

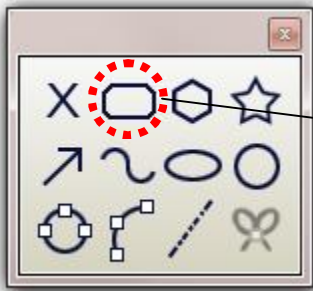
Click the **Exit** button to access the 2D module.





Click then the **Change** Menu and Select the **Material...** to set the size of your piece of material. Here we will use a 10, x 10 x 1 material size. You can enter any material size you have available; you will just need to adapt the size of the vectors accordingly to your material size then.

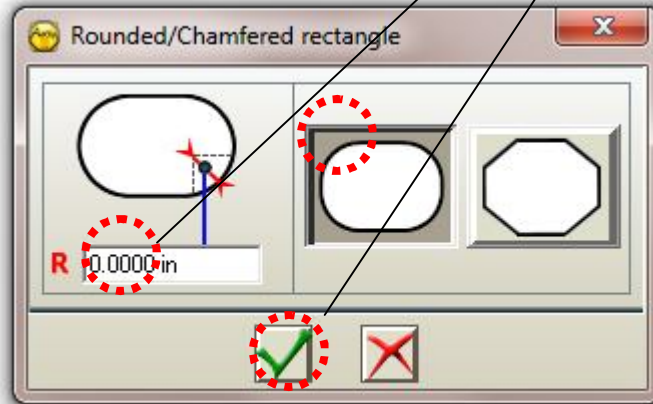


o **Drawing your vector lines**



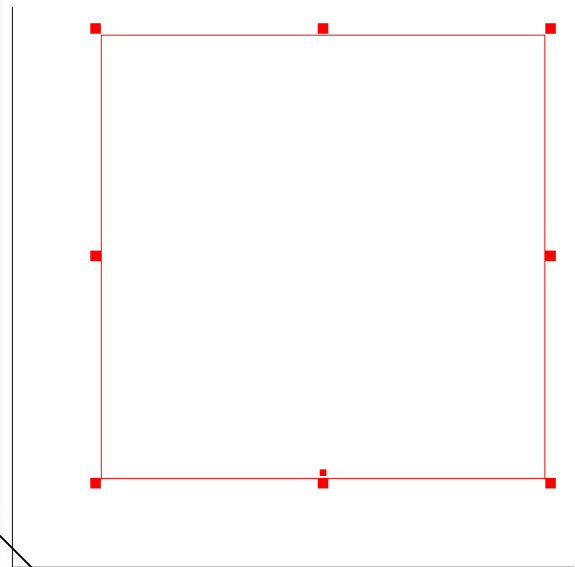
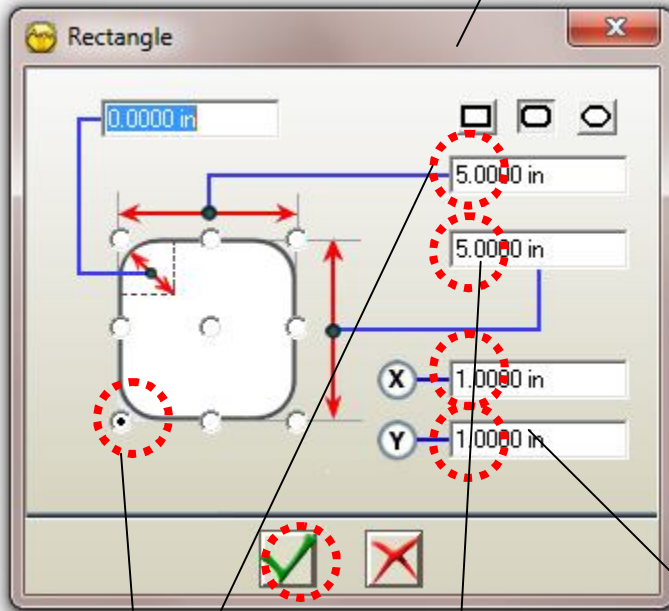
From the **Geometrical Shape** toolbar  select the **Rounded Chamfered Rectangle**  tool by clicking on it.

The corner options pops up. Enter 0 in the R field corresponding to the Radius so you create straight corners. Validate with the tick sign.



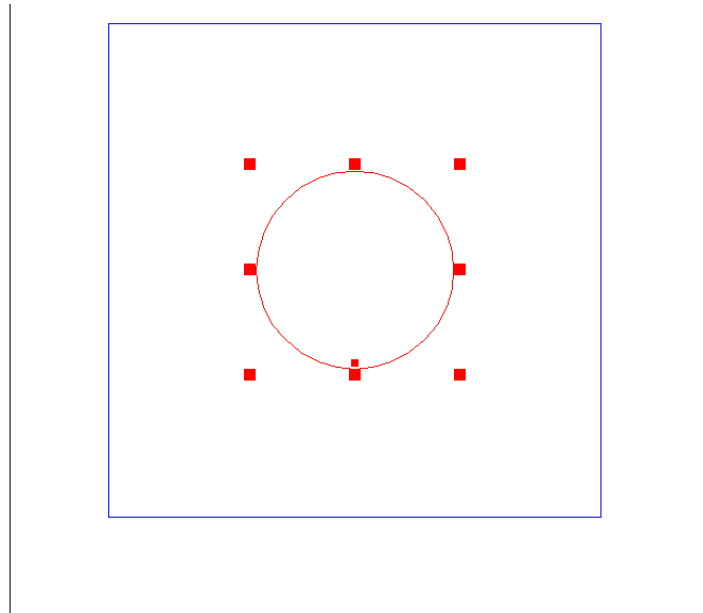
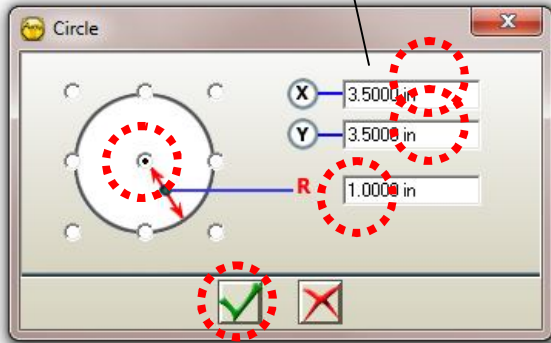
F2

Hold the **F2** Key, the **Rectangle Construction Parameters** window will pop up.

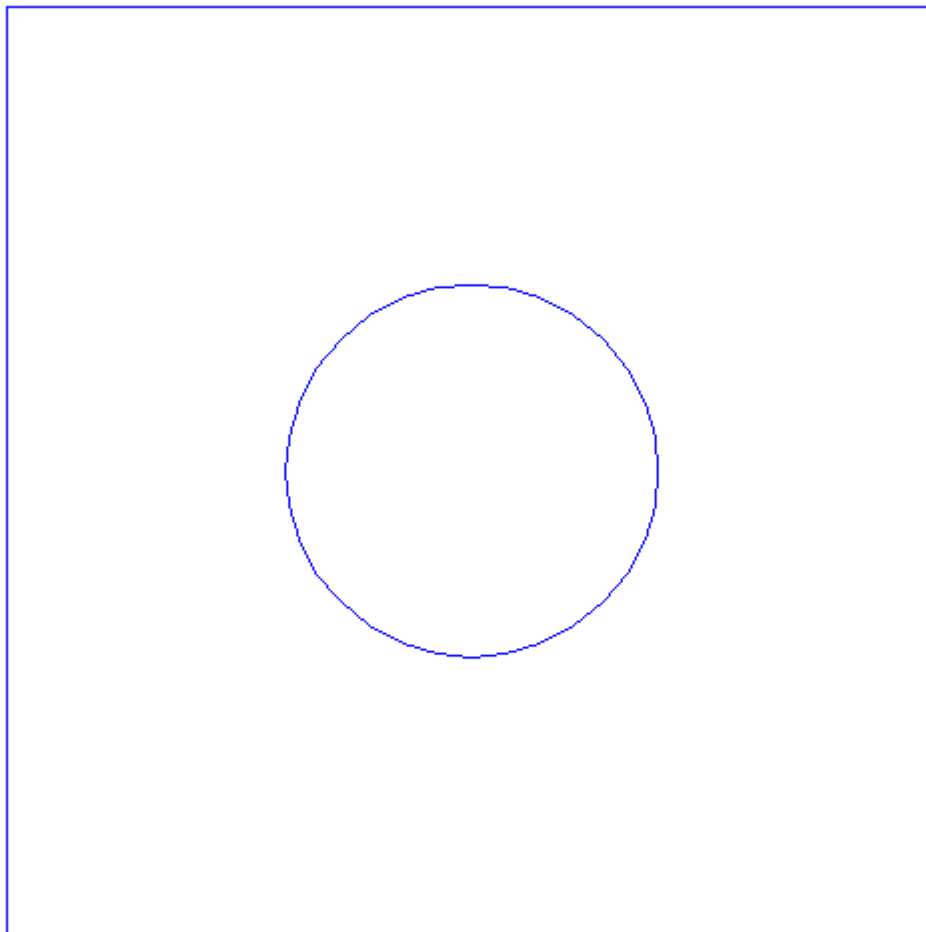


Enter 5 for the Width and 5 for the Height so you draw a square. Enter 1 for X and Y to leave some room from the Machine origin so your tool can run the cut. Finally select the bottom left corner of the object to define the Starting point.

From the **Geometrical Tools** , click the **Circle** tool .
Enter X = 3.500 and Y = 3.500, Radius = 1



You should have 2 vector lines positioned as below and ready to be machined. If your material was set smaller then try to enter smaller value so they fit in your material area.



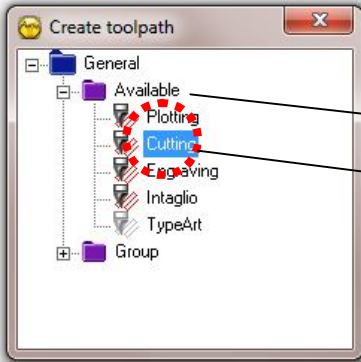
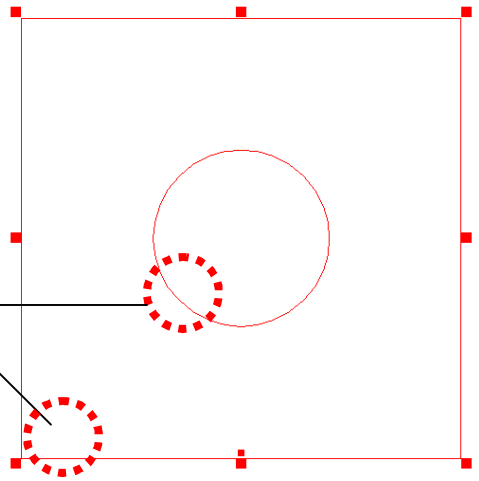
o **Machining your Vectors**



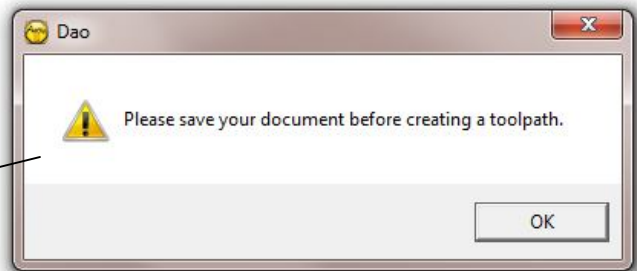
Click the CAM module Tab on the Left hand side of your screen.

Open the **Create Toolpath** window to access the toolpath available .

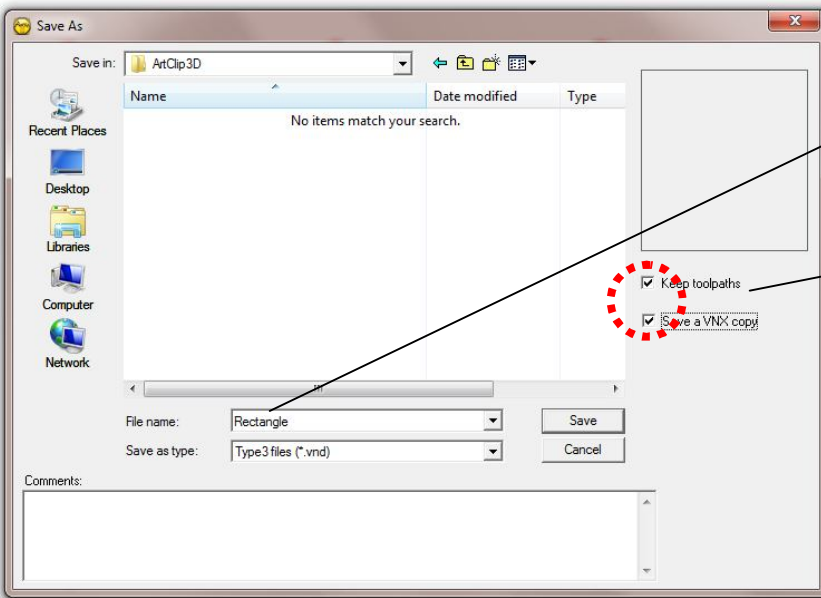
Select your 2 vector lines so they both turn red as the selected color.



Click the Available folder and select the **Cutting** toolpath by double clicking it.



A DAO windows message pops up asking you to save your file. Save it under a specific directory you will know where to find then.

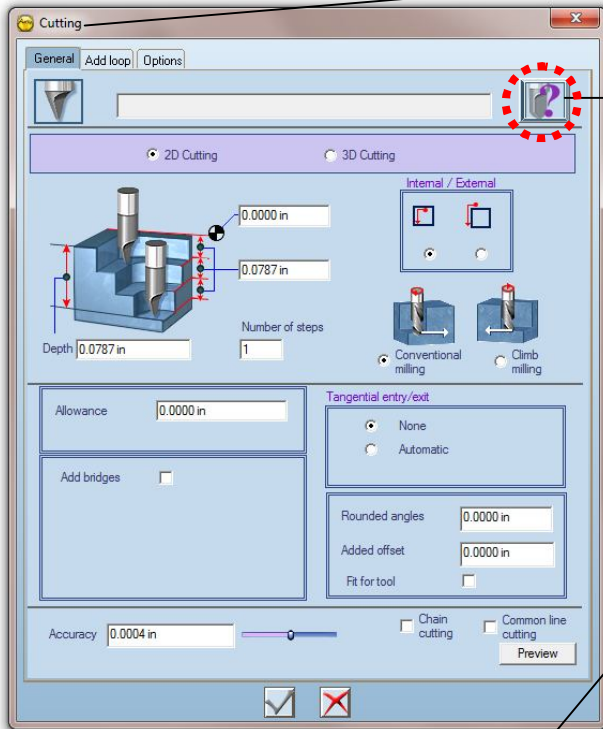


In the Save As window save your file under **Rectangle**.

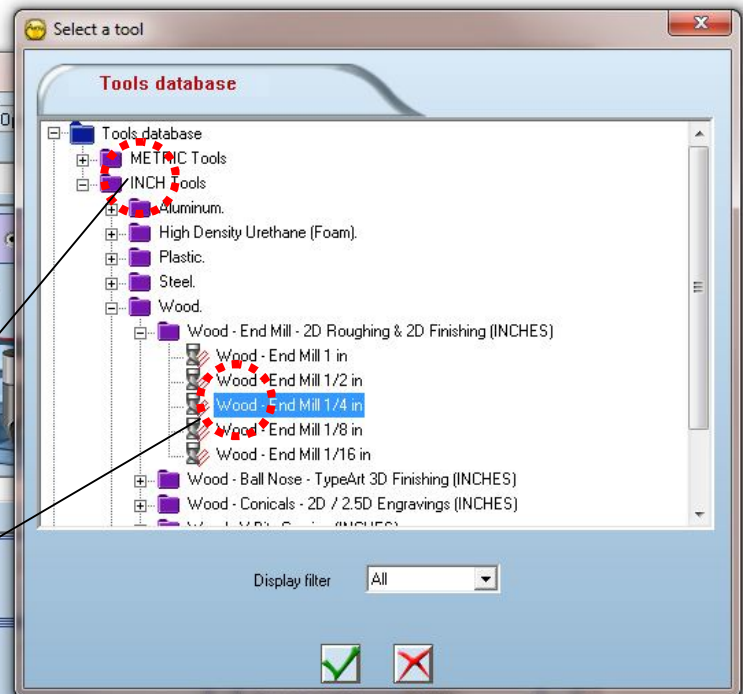
Note the "Keep toolpaths" option allowing to save your files associated with your toolpaths, as well as the "Save a VNX copy" which allow a back up file compatible with all the product line of ArtClip3D.

This is your choice to activate them if you want to keep them in the same folder as the VND file.

When the file has been saved then automatically the 2D cutting window comes up on screen.

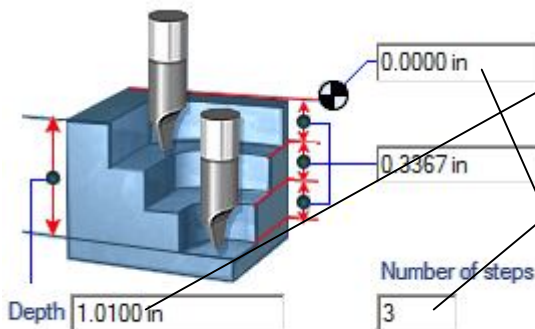


The first thing to do in this window is to set the tool you are going to use with your machine to cut the 2 vector lines you drew earlier.



In the **INCH Tools** section you will need to expand with the + sign on the left the **Wood.** group and select the tool named **Wood - End Mill ¼ in.**

If none of the tool in the database is in your possession you might want to customize a tool in the tool database corresponding to what you have available. Please read the **How to define a tool in the tool database?** tutorial to have a complete overview on adding your own tools.



The material size was defined at 1", the Depth will then be set at 1.0100 in to insure the tool cut beyond the thickness a little bit.

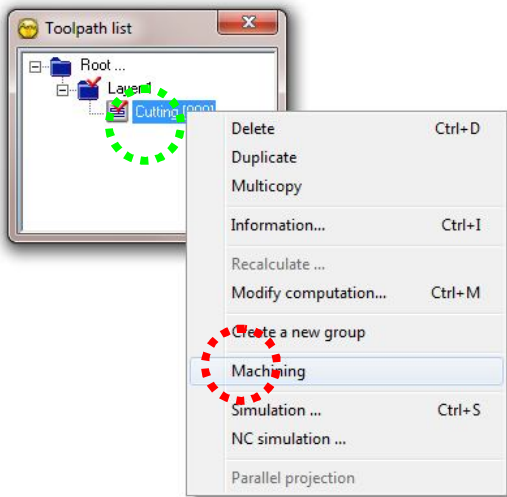
The Number of steps can be set at 3 or more but not less according to the machine and tool capability.

Note the 0 position at the top of the material this is where the machine will make the tool touch at Z=0

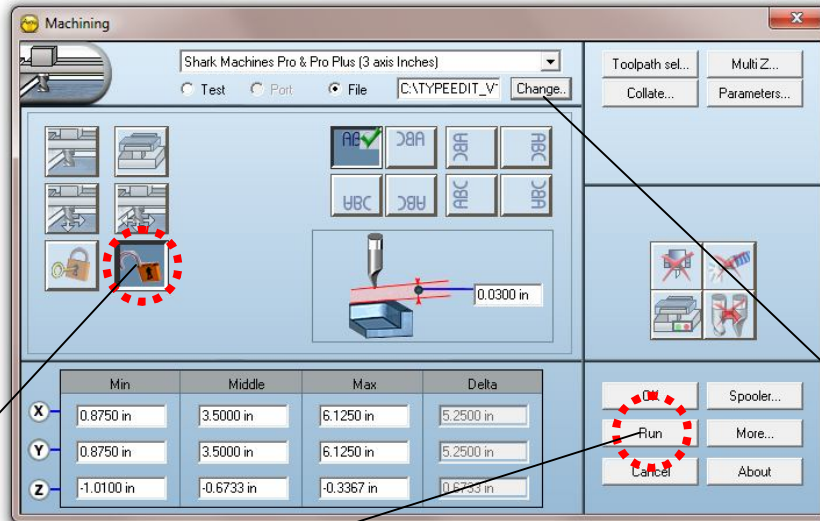
All the other parameters can stay as they are just validate with the tick sign.

A first toolpath is being computed and added to the Layer 1 root as a Cutting toolpath ready to be sent to your machine.





Right click the Cutting toolpath and select the **Machining** command.



Open the lock and click on the **Run** button, your file is being output where you saved your vnd file earlier. Open it with Notepad or with your CNC controller interface to look at it.

