

LAZYCAM LATHE - TURNING TOOLS & PARAMETERS

The following can be used as guide to quickly gain some experience about how the parameter settings and tool settings affect generated toolpaths. This will be accomplished by giving a few examples using a very simple DXF imported file. If you don't spend a little time doing this and think you going to hack your way thru the program LOL!.

Have Fun,
RICH

Overview:

Import the file into Lazycam, clean & convert it, then start with the settings shown in this Tutorial. Then play around with different settings one at a time and see the effects. I have made comments on the screen shots and written ones below each figure.

Note: This tutorial is not a how to machine a piece in some proper way, but rather to help you from becoming frustrated and giving up on LAZCAM.

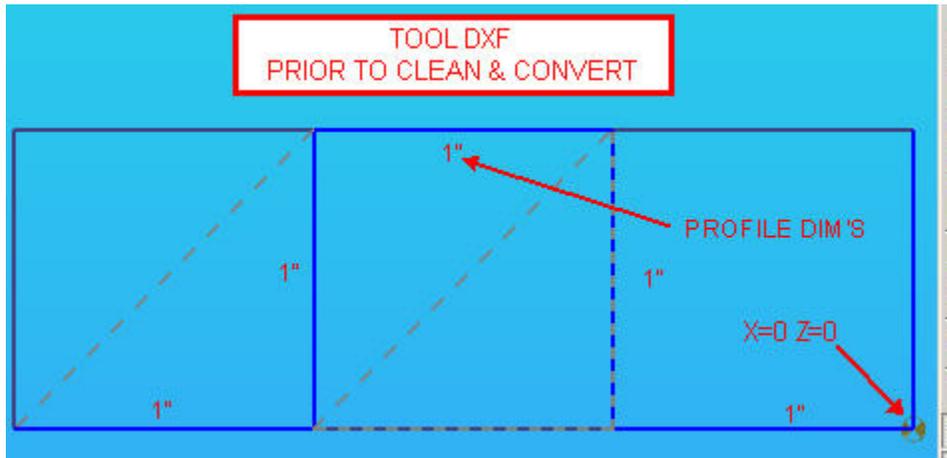


FIGURE -1

This shows what the screen should look like post dxf import of the file. Dimensions and comments were added. The centerline of the profile was deliberately placed on Z=0 so distances about the profile compared to the tool settings and can be easily reviewed (just a comment – it's covered in another figure).

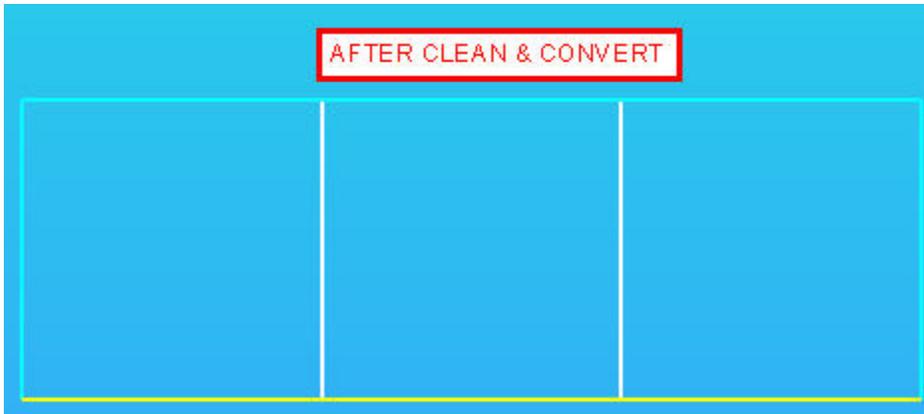


FIGURE -2

This shows result of clean and convert.

Lazcam will need some info on the cutting tools to generate toolpaths.

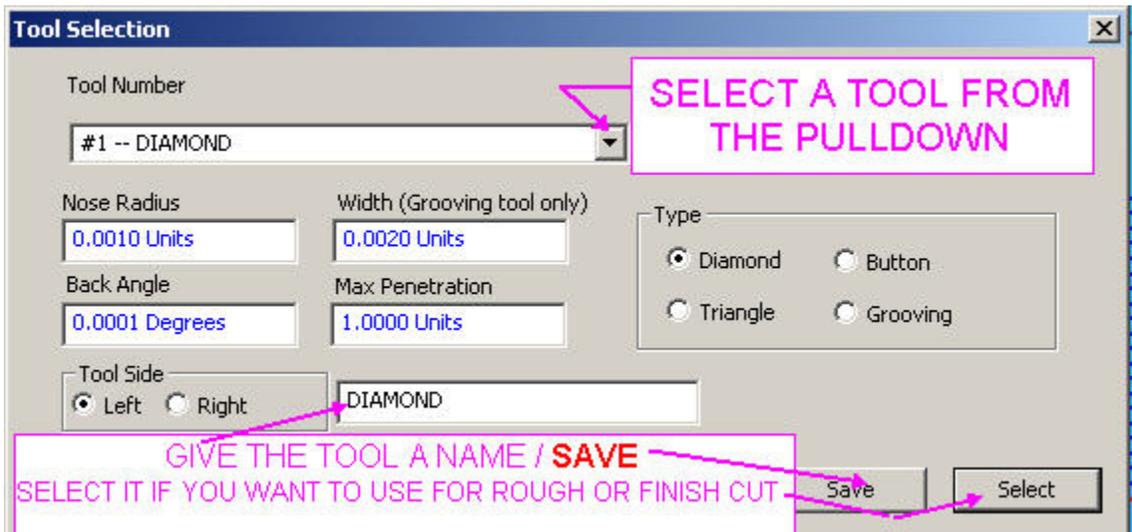


FIGURE -3

This is a display of a tool I created. Here's how:

1. Select number from the pulldown of tool numbers
2. Give it a name in the box as shown "DIAMOND"
3. Add the settings as shown / duplicate what is shown on the screen
4. **SAVE** by clicking the box

When the rough or finish boxes are selected later on you can just open the pulldown and it should be there with the settings you defined. The intent here is just to get a very simple point tool so you can easily compare tool location to the profile, how penetration and back angle will affect displayed tool paths (this is covered in later figures).

Here is a screen shot of generated tool paths using the parameter settings and tool settings shown in the screen.

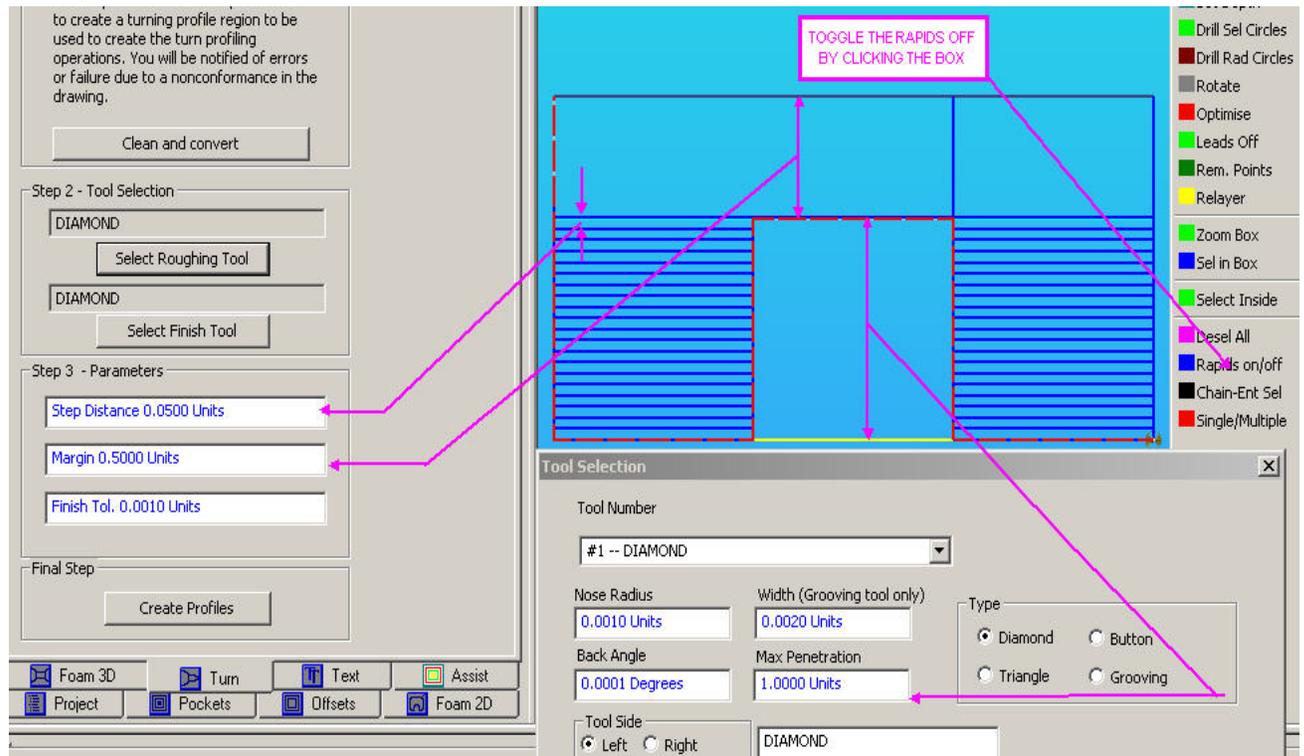


FIGURE -4

I added the tool selection pullout onto the screen capture.

Make note of the following:

1. TOGGLE the rapids on / off by clicking the box to get what is displayed
2. Step distance is your cut depth / how far each tool movement will be for cutting
3. Margin is will be how far the tool will retract (consider it a clearance)
4. Finish defines the last profile cut that will be made with the finish tool
5. The 1" penetration in the tools settings allows the tool to go to the bottom of the desired cut. Since the tool is only 0.001 " radius and the back angle is almost zero, it's a point tool which can almost go to a sharp corner. Also since the tool is .002 wide and if we were to look at the tool path start and end locations there should be .002 to the tool centerline from the profile.

You can post the code into MACH and look at the gcode but it's easier to just to use the entity info which can be found clicking on the project box.
Here is a screen shot.

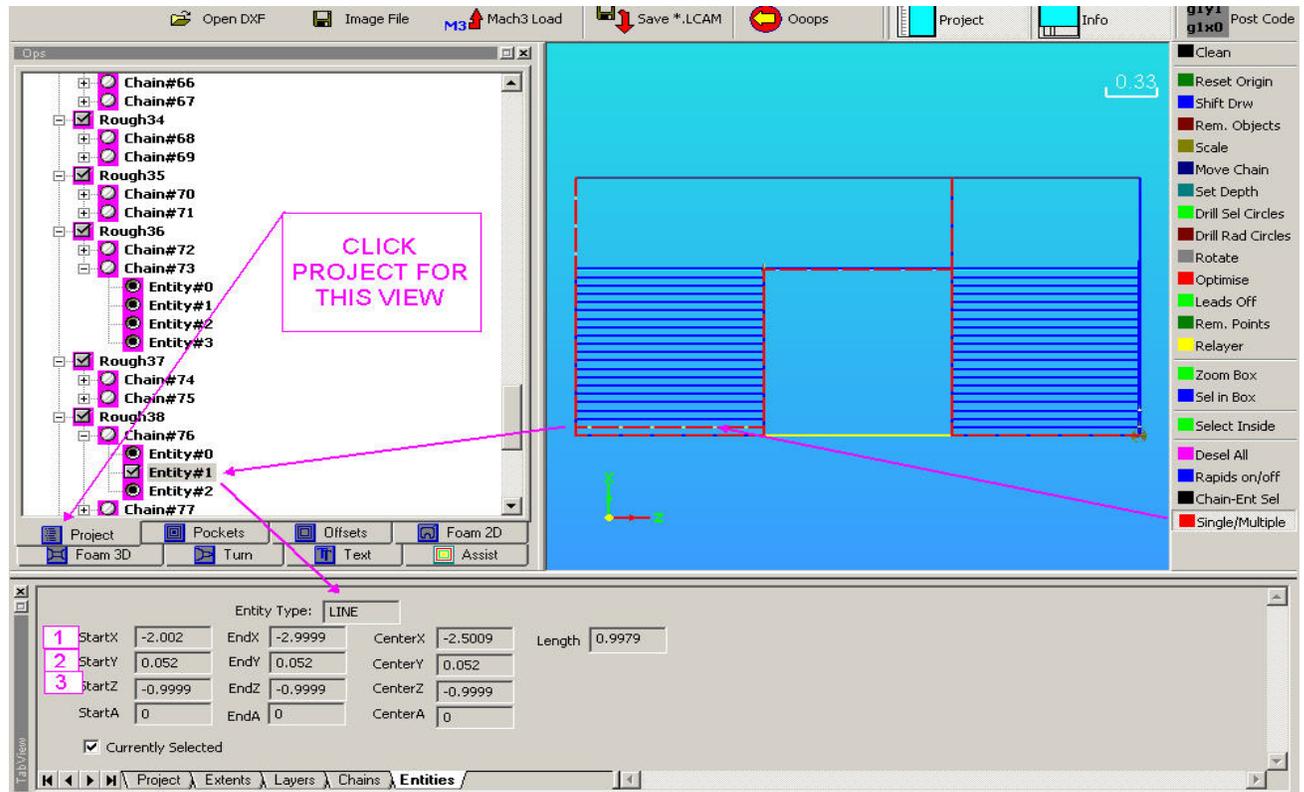


FIGURE – 5

Click the single / multiple box, use your mouse to select a toolpath and it will highlight, the entity will shown by the check mark, and below information on it is provided.

Since the profile is just 1” lengths in the X and Z and the tool is .002” wide, note the following:

- 1.The back of the tool is .001” away from the profile. We left .001” for a finish cut.
The face ends .0001” away from the end of the profile.
- 2.The y starts at .050” margin + .001” tool radius + .001” allow for finish cut =.052”
(this is the X movement in the gcode)

So if you play around with different settings you can go in and check for specific toolpath information and compare relative to tool parameter changes, should you so desire.

So I decided to make a change to the tool used above by changing the settings to reduce penetration and change the back angle. Additionally the step was changed to 0.1” and wanted to see the effects.

Here is a screen shot.

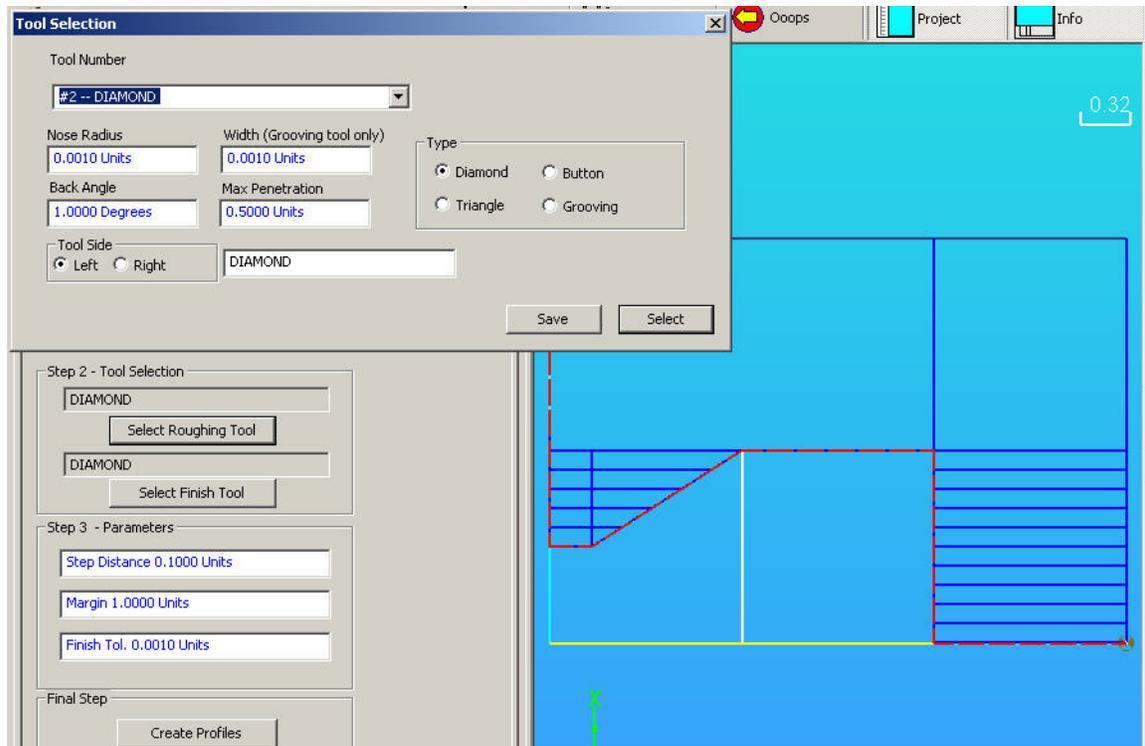


FIGURE – 6

You can do this very quickly and if something happens just go out and come back into Lazcam and start over as the parameters and tool settings last used are saved.

SUMMARY:

- 1.import dxf
- 2.clean and convert
- 3.select roughing tool (or create / modify one)
- 4.select finishing tool (or create / modify one)
- 5.create profiles
- 6.use the OOPS to go back
- 7.reselect roughing and finishing tool / change paramters
8. see the effects

The above can be done very quickly, over and over again. Now you're an expert on your tool settings and parameters.

REMEMBER THIS SUPPOSED TO BE FUN! RICH