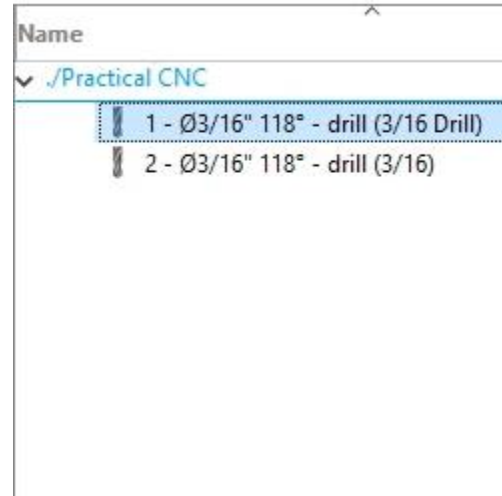


## Lesson: Create a Custom Tool

In this lesson, you'll begin populating a tool library with custom tools.

### Learning Objectives

- Create a custom tool.
- Create a cloud tool library.
- Modify a copied tool's parameters.

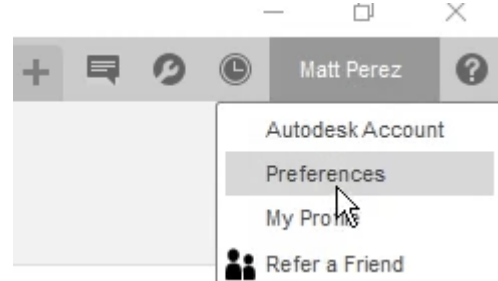


The completed exercise

1. Continue with the *CAM Milling Setup* file from the previous module.

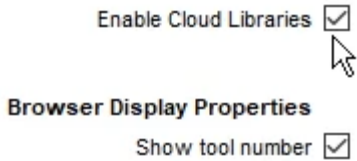
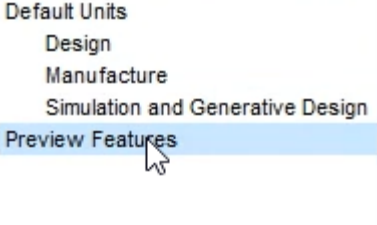


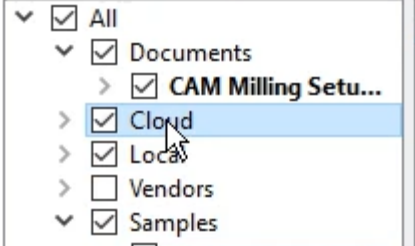


2. Click your name in the screen's upper right corner and choose the Preferences option from menu.

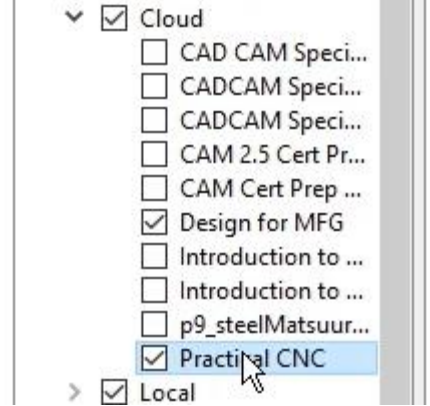


3. Navigate to the General > Manufacture section of the Preferences dialog.

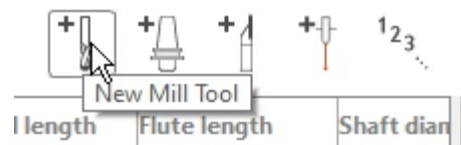


<p>4. Make sure the Enable Cloud Libraries option is activated; this option allows you to save your tool libraries to the cloud so that you can access them on any computer you log into.</p>	
<p>5. Navigate to the dialog's Preview Features section.</p>	
<p>6. Use the menu to show only the Manufacture preview features. Navigate through the results and notice that an updated tool library can be activated. Click OK in the Preferences dialog to accept the changes.</p>	
<p>7. Before operations can be created to cut this model's geometry, a tool library needs to be populated. Click Manage&gt; Tool Library.</p>	
<p>8. In the CAM Tool Library dialog's left column, activate and select the Cloud library. Any tools in this library will be shown in the dialog's main window. Right-click the Cloud library and select the New Tool Library option from the menu.</p>	

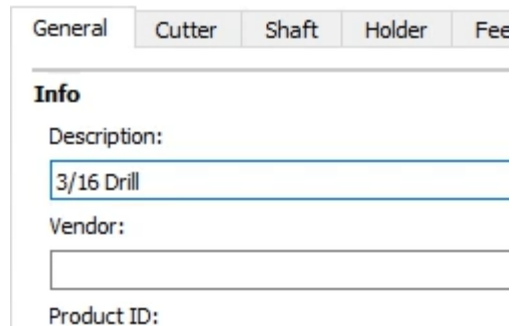
9. Choose to name the new library Practical CNC. Activate this library and select it. Notice that there are no tools shown in the dialog's main window because this library is currently empty.



10. A new custom tool can be created by clicking New Mill Tool in the dialog's upper right corner.



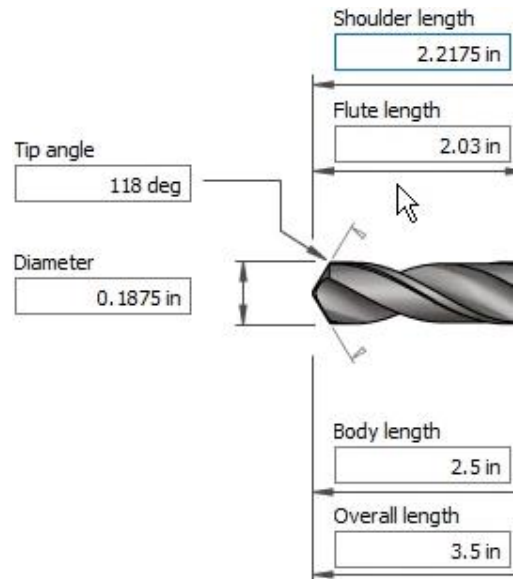
11. Navigate to the Library dialog's General tab and enter **3/16 Drill** into the Description field.



12. Continue to the Cutter tab. Choose the Drill Option from the Type menu, then enter **2** into the Number of Flutes field. Also choose the Ti coated option from the Material menu.



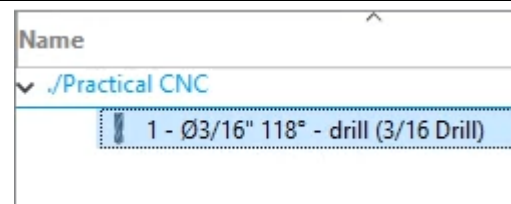
13. In the Geometry section, enter **0.1875** inches into the Shaft Diameter field and the Diameter field. Enter **3.5** into the Overall Length field and **2.5** into the Body length field. Enter **2.03** inches into the Flute length field. These dimensions should come from measurements you take from existing tools or from a vendor's website. These numbers should not be estimated or imagined. These dimensions will be used to create the operations that will cut the model's geometry. Precise measurements are needed if the operations will be created correctly.



14. Continue to the Shaft and Holder tabs and note the information on them. Don't make any changes. Continue to the Feed & Speed tab. Reduce the Spindle speed value to **4000 RPM**, then enter **18 in/min** into the Plunge feed rate field. Notice that the Feed per revolution value automatically updates because of the change in the Plunge feed rate field. Click OK in the Library dialog to create the custom tool.



15. Continue to the Post-Processor tab. The tool is added as Tool 1 into the Practical CNC library.



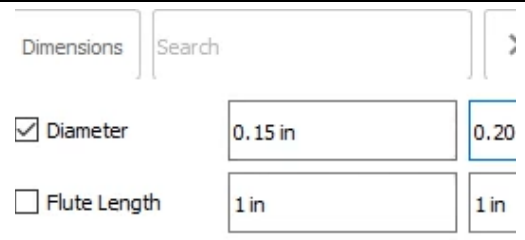
16. In the CAM Tool Library dialog's left column, activate and select the Samples library. This library has many tools and could take a long time to search. The buttons at the top of the dialog can be used to filter the results. Click Operation and choose the Hole option from the menu. Click OK. The Samples library will be filtered to show only the tools used to make holes.



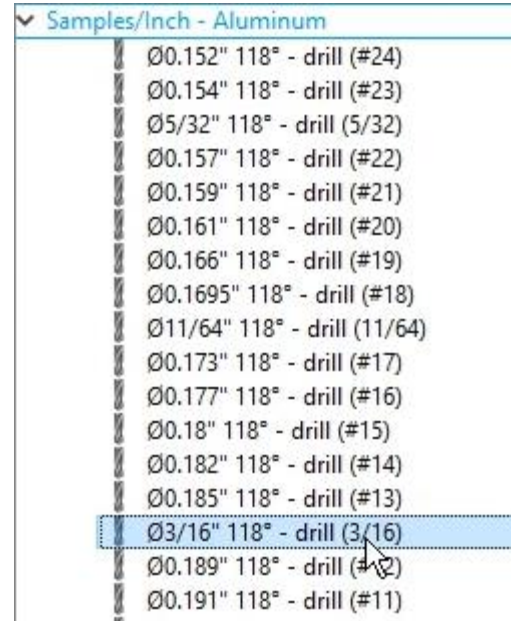
17. Click the Type filter and clear the current selections by clicking Clear at the bottom of the menu. Select the Drill option, then click on it.



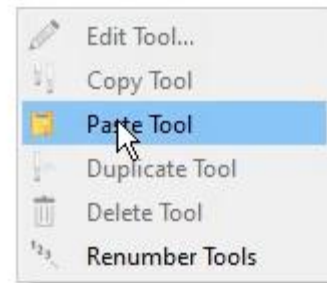
18. Click the Dimensions filter and activate the Diameter option. Specify a range of **0.15** inches to **0.2** inches, then click OK to show only the drills that fall within this range. The tools that match the selected criteria are much easier to search.



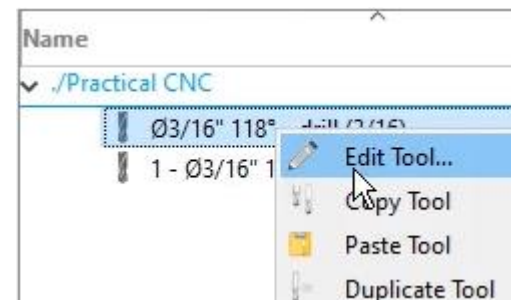
19. Choose the 3/16" drill from the Samples/Inch – Aluminum library. Right-click this tool and choose the Copy Tool option from the menu.



20. Navigate back to the Practical CNC library and paste the tool into the library. Right-click in the library and choose the Paste Tool option from the menu.



21. The tool needs to have a tool number associated with it. Select the tool, right-click it, and edit it by choosing the Edit Tool option from the menu.



22. Navigate to the Library dialog's Post-Processor tab, then enter 2 into the Number field.

NC	
Number:	Comment:
2	
Length offset:	Coolant:
2	Flood

**23.** Continue to the Feed & Speed tab and reduce the Spindle speed value to **4000 rpm**. Also reduce the Plunge feed rate to **18 in/min**. Click OK to accept the changes.

Spindle speed

4000 rpm

Surface speed

196.35 ft/min

Plunge feedrate

18 in/min

**24.** The tool is updated in the Practical CNC library as Tool 2. Close the CAM Tool Library dialog by clicking the red X button in the upper right corner. The file does not need to be saved because the changes are automatically saved in the club library. Continue to the next module.

✓ /Practical CNC

1 - Ø3/16" 118° - drill (3/16 Drill)

2 - Ø3/16" 118° - drill (3/16)