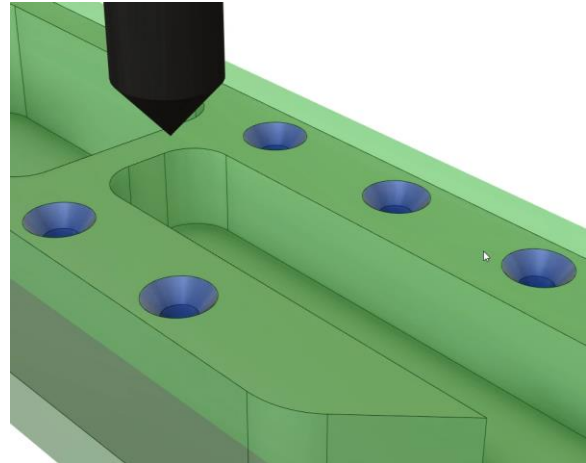


## Lesson: Spot Drilling Holes

In this lesson, you'll spot drill and countersink a part's holes.

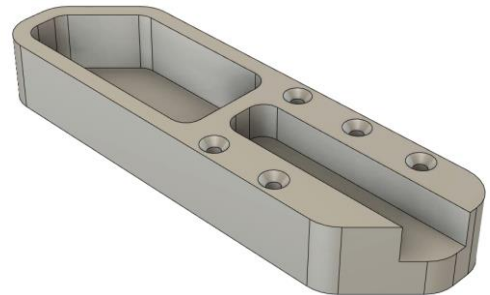
### Learning Objectives

- Modify a tool diameter.
- Create a spot drilling operation.

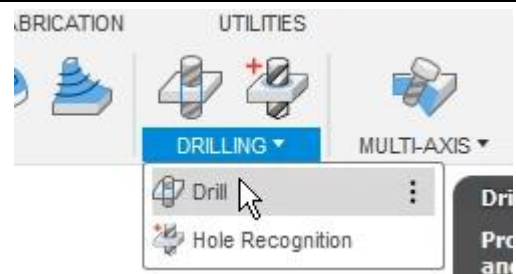


The completed exercise

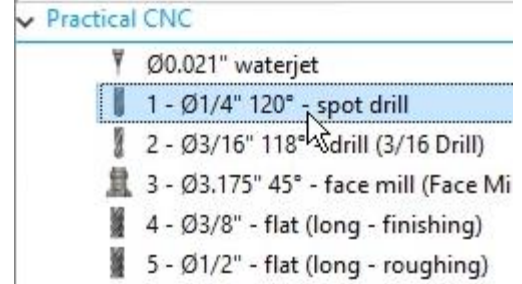
1. Continue with the *Introduction to Milling* file from the previous module. If you had any difficulties with the previous file, upload the supplied *Introduction to Drilling.f3d* file.



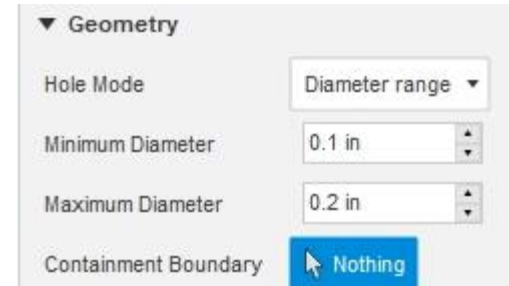
2. The part's holes need to be drilled. Click Drilling> Drill.



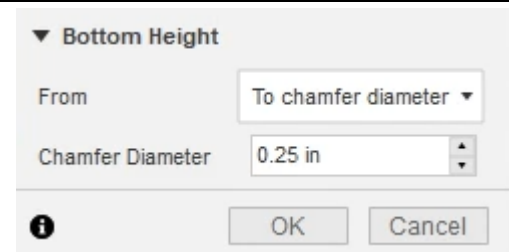
3. To choose an appropriate tool for the operation, click the dialog's Select. Navigate to the Practical CNC library, then choose Tool 1. Click OK.



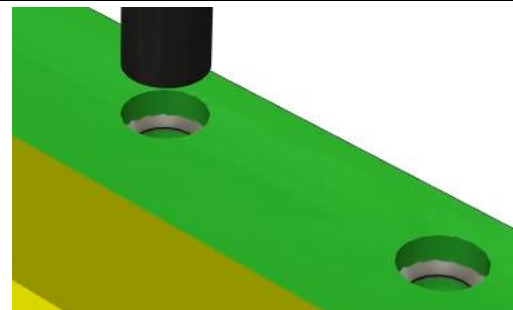
4. Continue to the dialog's Geometry tab, then choose the Diameter range option from the Hole Mode menu. This will automatically identify and select holes within a specified range. Specify a minimum diameter of **0.1 in** and notice the holes are automatically selected in the Canvas.



5. Continue to the Heights tab. Since this operation will only spot drill the holes, the Bottom height needs to be adjusted. Select the To chamfer diameter option in the Bottom Height section's From menu. OK the dialog to generate the toolpath.



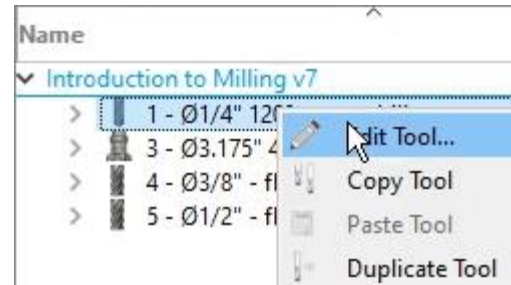
6. Simulate Setup1. To do this, select Setup1 in the Browser, right-click it, then choose Actions> Simulate. Notice that the tool's size does not look right.



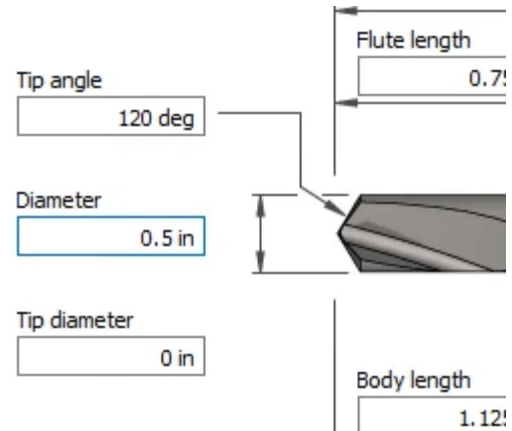
7. Use the Measure tool to measure a countersink's top edge. The spot drill tool can be edited to better suit the part's geometry.



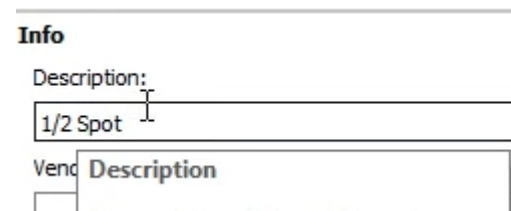
8. Open the tool library by clicking Manage> Tool Library, then navigate to the document's tool library. Select the spot drill and edit it.



9. Navigate to the Cutter tab and notice the diameter is currently set to 0.25 inches. Increase the Diameter value to **0.5 in**.



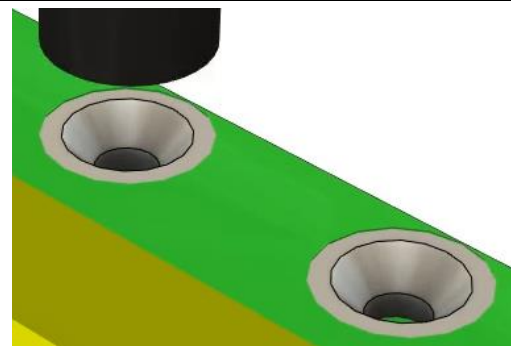
10. Navigate to the General tab and add a description of **1/2 Spot**. OK the dialog to accept the changes. Click Yes in the dialog that warns that operations will be affected by the changes. Close the CAM Tool Library dialog.



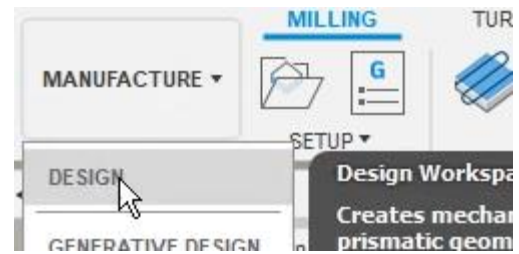
11. In the Browser, a red icon appears next to the operation. To clear this icon, click Actions> Generate. This will update the operation and clear the icon.



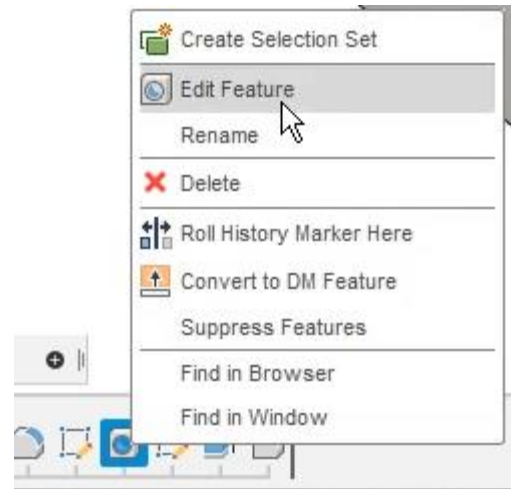
12. Select Setup1 in the Browser and simulate it. The tool's diameter looks more appropriate but now the tool is plunging too far. End the simulation by clicking Close.



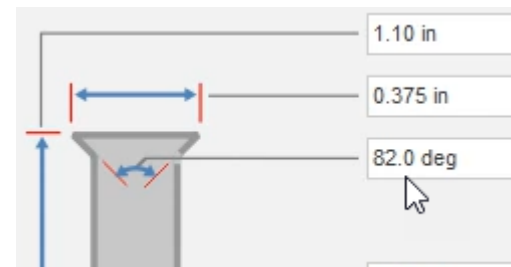
13. Navigate to the Design workspace.



14. Edit the Hole operation that created the countersinks.



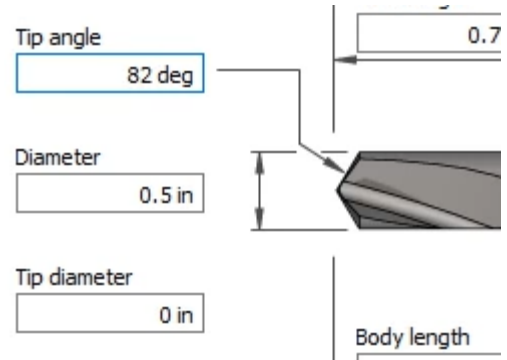
15. The countersink feature's angle is set to 82°. Click the dialog's Cancel.



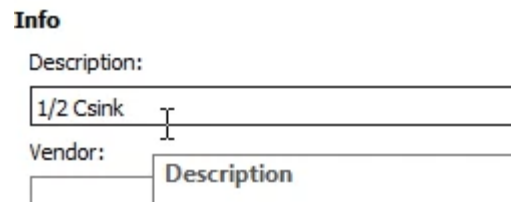
16. Return to the Manufacture workspace and open the tool library. Edit the spot drill again.



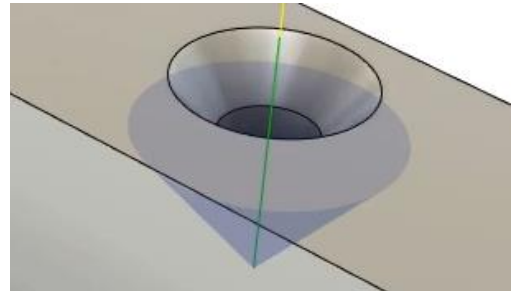
17. Decrease the tool's Tip angle to **82°**.



18. Navigate to the General tab and update the Description to **1/2 Csink** to indicate that it is a countersink tool. OK the dialog to accept the changes, click Yes in the small dialog, then close the CAM Tool Library dialog. Regenerate the operation by pressing Ctrl+G on the keyboard.



19. Notice that the toolpath preview makes the countersink depth look too low.



20. Edit the operation and navigate to the Heights tab. Choose the To chamfer width option in the From menu, then OK the dialog.



21. In the Browser, select the Face and the Drill operations and simulate them. This new cut geometry matches the modeled geometry. Save the file and continue to the next module.

