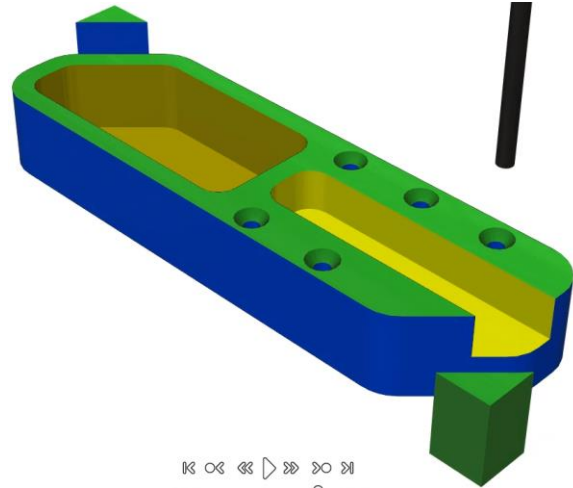


Lesson: Create a Peck Drilling Operation

In this lesson, you'll create an operation to drill a part's holes.

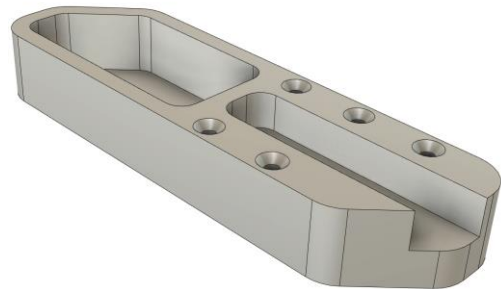
Learning Objectives

- Create a peck drilling toolpath.



The completed exercise

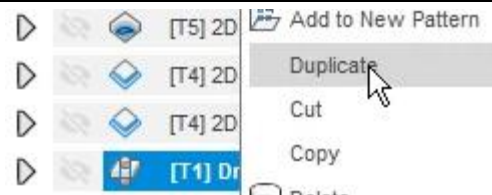
1. Continue with the *Introduction to Milling* file from the previous module.



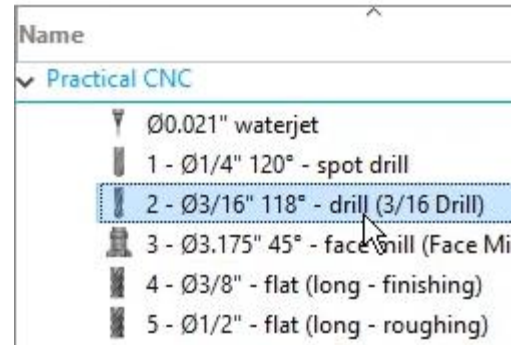
2. Click Inspect> Measure to open the Measure tool. Measure the diameter of a hole and note the measurement.



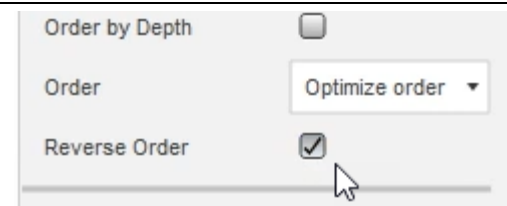
3. Since the holes have already been spot drilled, the first Drill operation can be copied and tweaked to efficiently create the drilling operation. In the Browser, duplicate the Drill1 operation. Edit the new Drill operation.



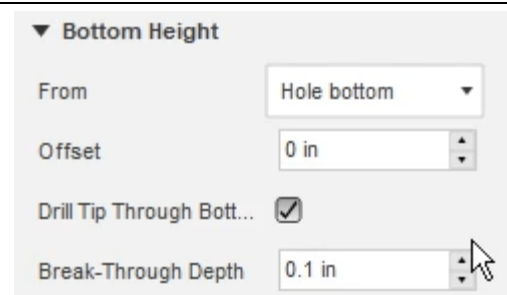
4. Click the dialog's Select to choose the appropriate tool for the operation. Navigate to the Practical CNC library and choose Tool 2. Click OK.



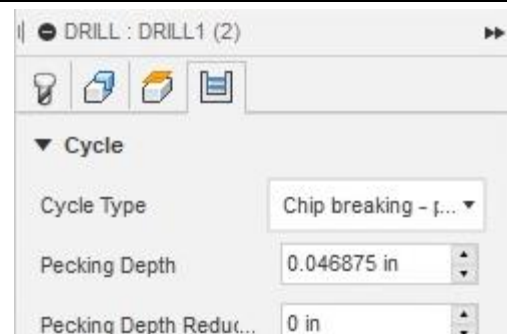
5. Navigate to the Geometry tab and activate the Reverse Order option. Reversing order can increase the operation's efficiency since the table will already be in the last hole's position after the spot drilling operation.



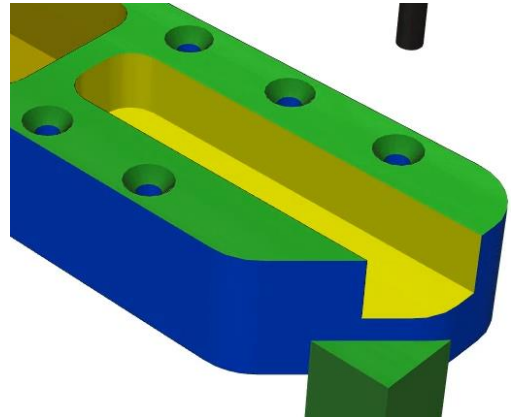
6. Navigate to the Heights tab and choose the Whole bottom option in the Bottom Height section's From menu. Activate the Drill Tip Through Bottom option, then enter **0.1 in** into the Break-Through Depth field to make sure the operation drills all the way through the part.



7. Navigate to the Cycle tab and choose the Chip breaking - partial retract option from the Cycle Type menu. OK the dialog to generate the toolpath.



8. Select Setup1 in the Browser and simulate it by clicking Actions> Simulate. Check the machined part's geometry against the underlying modeled part's geometry.



9. The operations inside the Browser can be renamed to help identify their function. Modify each operation's name to include a short note about the geometry they cut. Save the file and continue to the next module.

