

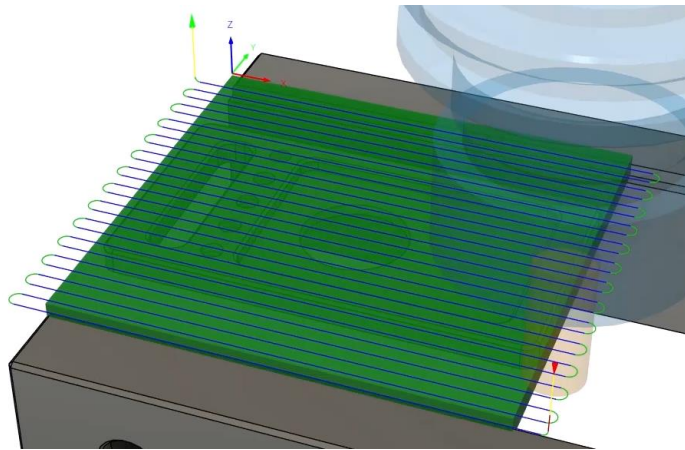
Step-by-step guide

Face stock body

Remove the extra material from the top of a stock body by creating and customizing two different Facing operations.

Learning objectives:

- Create a Facing toolpath with a face mill.
- Create a Facing toolpath with an end mill.



The completed exercise

1. Continue with your file from the previous video or open the supplied *Cell Phone INCH – Face.f3d* file. If you choose to open the supplied file, the link to the external parent files will be broken, so it's better to use your own file if possible.

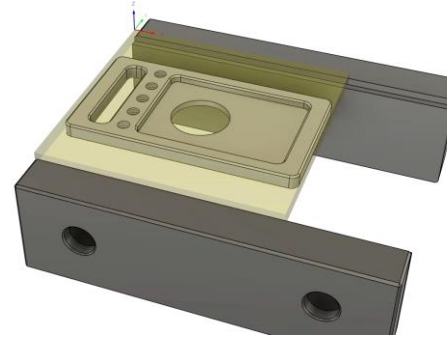


Figure 1. Open the file

2. The small amount of extra stock above the model needs to be removed. The most common method to remove this material is to use a Facing toolpath.

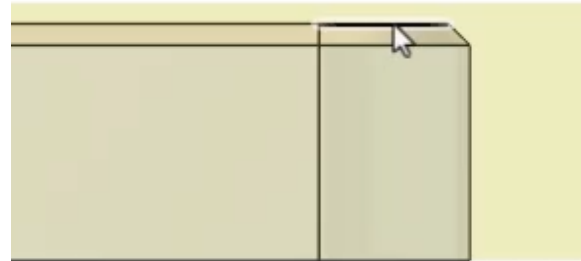


Figure 2. Inspect the stock

3. Click 2D > Face.

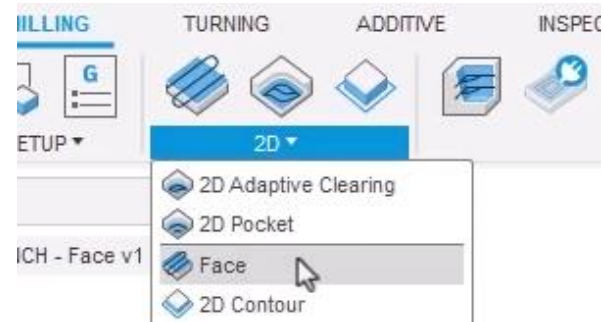


Figure 3. Create a Face operation

4. To choose an appropriate tool for this operation, click the Face dialog's Select.

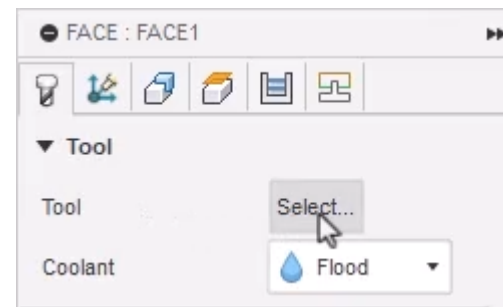


Figure 4. Click Select

5. Inside the Select Tool dialog, navigate to the Learn Cam 90 – INCH tool library.

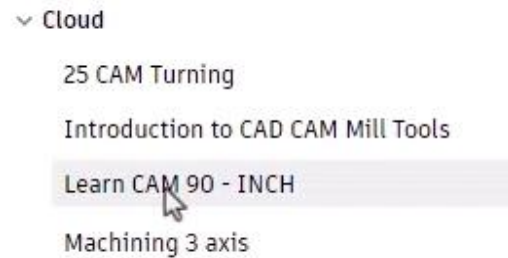


Figure 5. Open the tool library

6. Select the 2" Face Mill from the tool library.

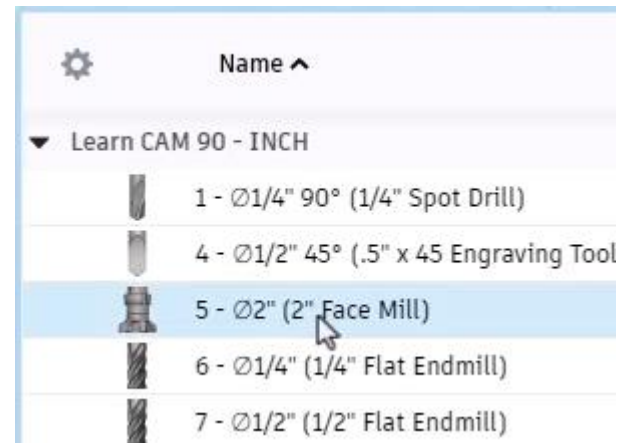


Figure 6. Select the tool

7. Click the Select Tool dialog's Select.

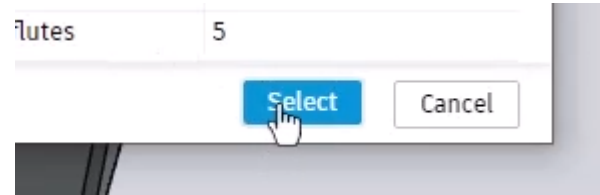


Figure 7. Click Select

8. Continue to the Face dialog's Geometry tab and notice that Fusion automatically selected the stock body's perimeter. This is correct and nothing else needs to be selected.

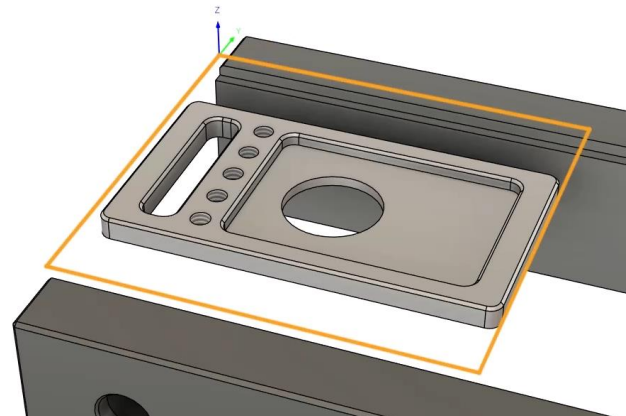


Figure 8. Note the default selection

9. Continue to the Heights tab. This tab allows you to configure when the tool changes from rapid movements to feed and plunge movements, how high the tool will retract before beginning the next cutting pass, the top of the cut, and the bottom of the cut. Understanding the various heights is a critical part of creating a successful toolpath. Explore all the different heights to learn their roles. Make sure the Model top option is selected in the Bottom Height section's From menu.

The screenshot displays the 'HEIGHTS' tab for 'FACE : FACE1'. It contains five sections, each with a 'From' dropdown menu and an 'Offset' text field:

- Clearance Height**: From: Retract height, Offset: 0.4 in
- Retract Height**: From: Stock top, Offset: 0.2 in
- Feed Height**: From: Top height, Offset: 0.2 in
- Top Height**: From: Stock top, Offset: 0 in
- Bottom Height**: From: Model top, Offset: 0 in

Figure 9. Explore the Heights tab

10. Continue to the Passes tab and explore the ways you can customize the toolpath. For instance, you could activate the Multiple Depths option if there is too much material to cut at a single depth.

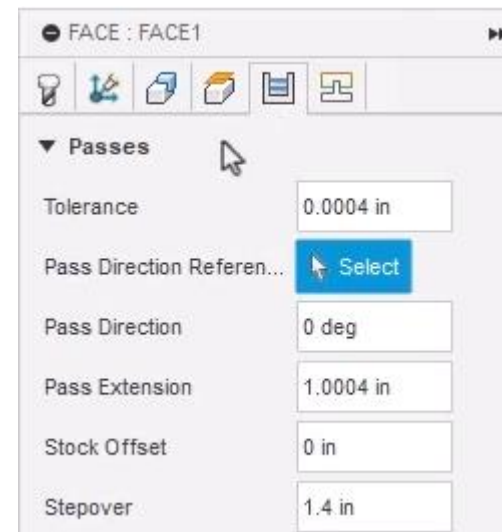


Figure 10. Explore the Passes tab

11. Explore the options in the Linking tab and note how you can customize the transitions between passes and the way toolpaths are linked. Accept all the default values by clicking OK.

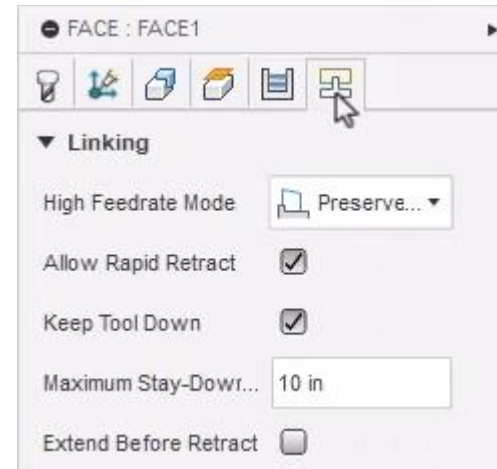


Figure 11. OK the dialog

12. Inspect the Facing toolpath and notice the toolpath has several line segment colors. The red arrow indicates where the tool begins, the yellow line segment indicates a rapid movement, the green segment is a lead-in or lead-out movement, the blue segment is a cutting movement, and the green arrow indicates where the toolpath finishes. If there were any red line segments, those would be ramp movements. Note that this 2" face mill is wide enough to cut the entire stock body in only three passes.

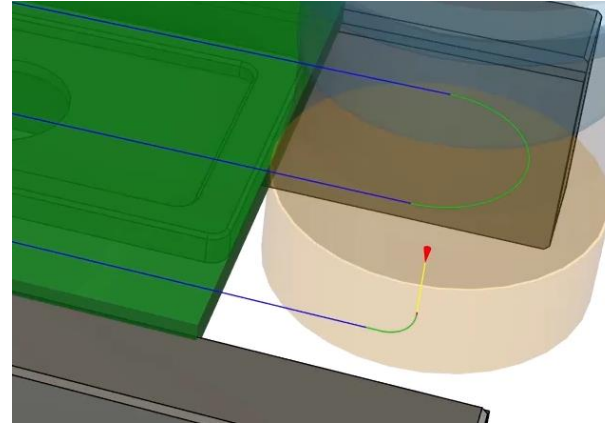


Figure 12. Inspect the toolpath

13. The green body you see in Step 12's image is the in-process stock body. You can toggle the in-process stock using the Navigation bar at the bottom of the screen.



Figure 13. Note how to toggle the in-process stock

14. Many machines cannot support a 2" face mill so smaller tools need to be used. Before re-creating this Face operation with a smaller tool, suppress the current Face operation: right-click it and choose Suppress from the menu.

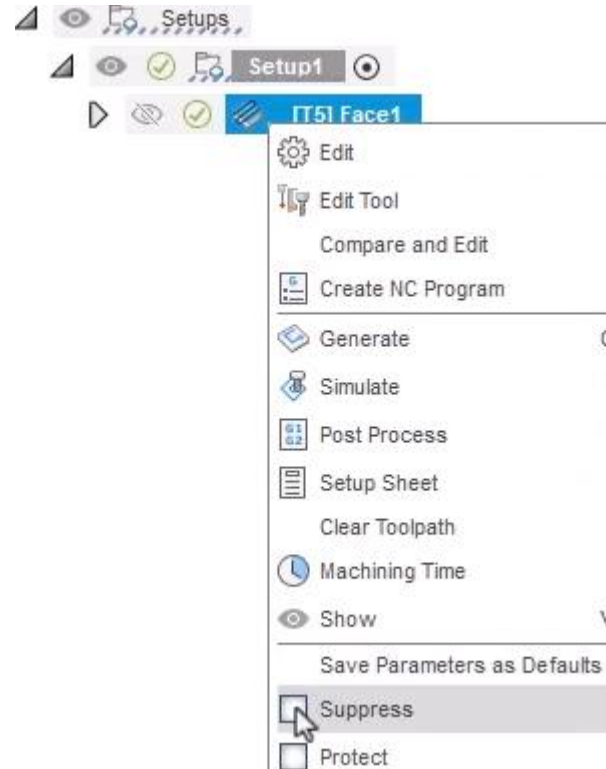


Figure 14. Suppress the Face operation

15. Click Yes to suppress the operation.

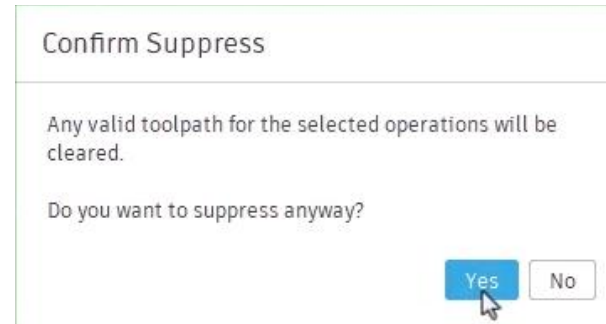


Figure 15. Click Yes

16. Notice the operation is still in the Browser but the name has a strikethrough effect. Suppressing an operation will exclude it when you post process the NC code.



Figure 16. Inspect the Browser

17. Click 2D > Face, then notice the new Face operation defaults to the tool you used in the previous operation. Click Select to choose a smaller tool for the operation.

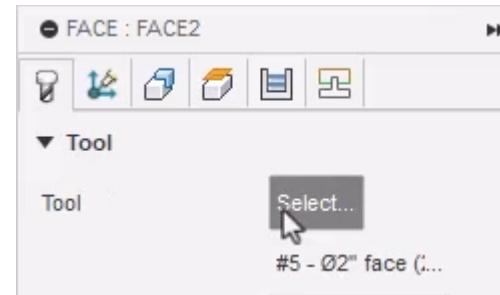


Figure 17. Create a new Face operation

18. Navigate to the Learn Cam 90 – INCH tool library and select Tool 7. This 1/2" tool has a much smaller diameter than the 2" tool you selected in Step 6.

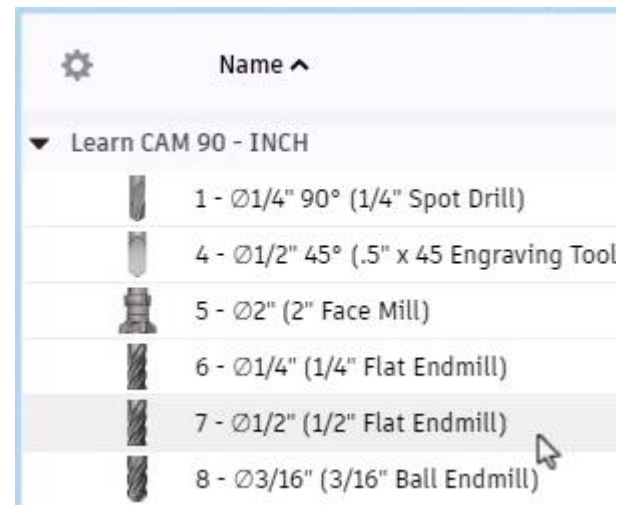


Figure 18. Select the operation's tool

19. Select the Aluminum – Finishing cutting preset from the dialog's Cutting data section. This cutting preset has a higher rpm than the default preset.

Cutting data	Spindle speed
Default preset	5000 rpm
Aluminum - Slotting	7639 rpm
Aluminum - Roughing	7639 rpm
Aluminum - Finishing	7639 rpm

Figure 19. Choose the cutting preset

20. Click the Select Tool dialog's Select.

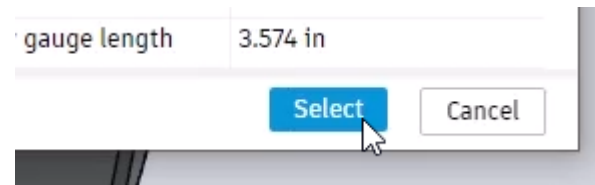


Figure 20. Click Select

21. Verify that all of the operation's settings are correct, then OK the dialog.



Figure 21. OK the dialog

22. Inspect the toolpath preview and notice that the smaller tool needs 27 passes to face the part. Save the project.

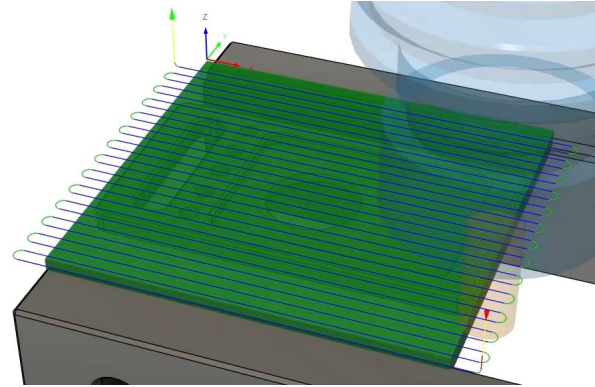


Figure 22. Inspect the toolpath preview