

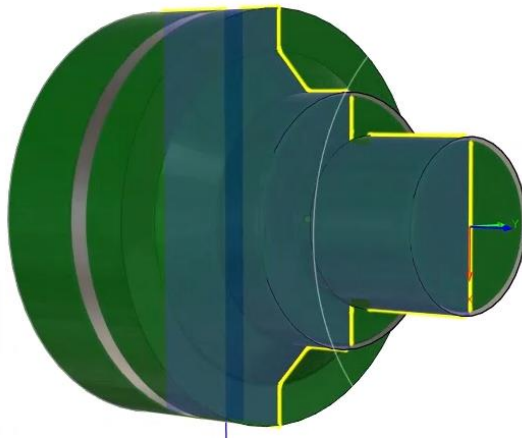
Step-by-step guide

Create lathe toolpaths

Create and customize several different operation types to cut a part's geometry.

Learning objectives:

- Create a profile roughing toolpath.
- Create a profile finishing toolpath.
- Create a groove roughing toolpath.
- Create a turning single groove toolpath.
- Create a chamfer toolpath.
- Create a threading toolpath.
- Create a parting toolpath.



The completed exercise

1. Open the supplied *CAM Lathe Toolpaths.f3d* file.

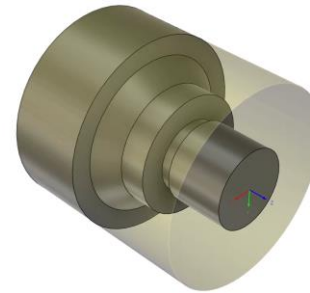


Figure 1. Open the supplied file

2. Navigate to the Manufacture workspace and activate the Toolbar's Turning tab. Click Turning> Turning Face. This operation usually clears the rough stock material from the end of the stock body.



Figure 2. Open the Turning Face tool

3. Click the Face dialog's Select to choose an appropriate tool for the operation.

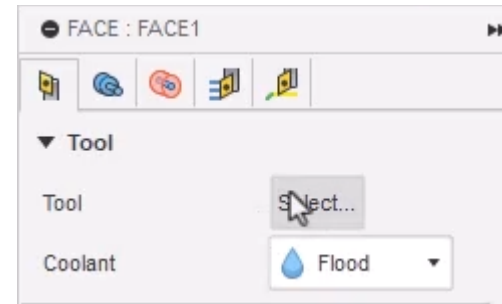


Figure 3. Click Select

4. Navigate to the CAM Lathe Toolpaths tool library and choose Tool 2 shown in the image on the right.

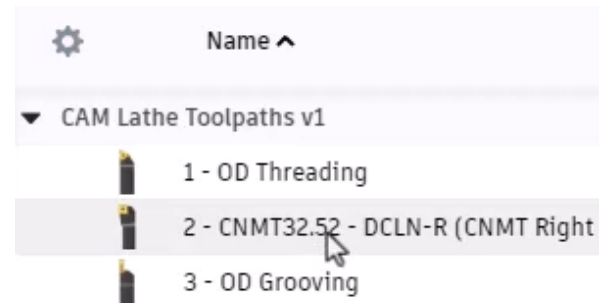


Figure 4. Choose Tool 2

5. Click the Select Tool dialog's Select to finalize the tool selection.

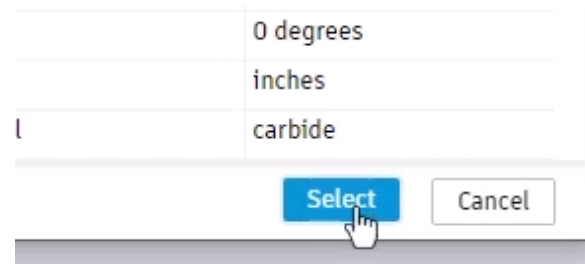


Figure 5. Click Select

6. Inspect the toolpath preview and note the material that the operation will remove. OK the Face dialog.

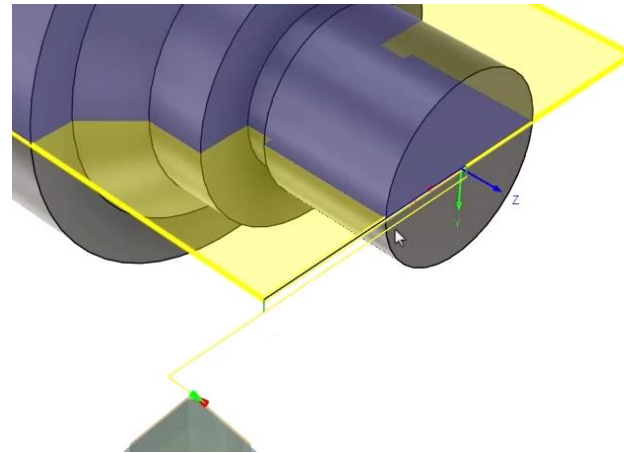


Figure 6. Inspect the toolpath preview

7. Notice the stock changes to show the in process stock. This shows the material removed by the selected operation

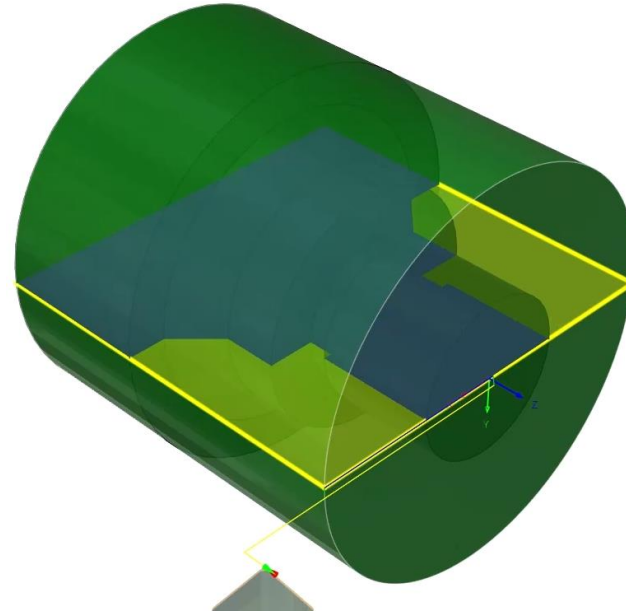


Figure 7. Inspect the in process stock

8. The in process stock option can be toggled using the Display & Settings bar.

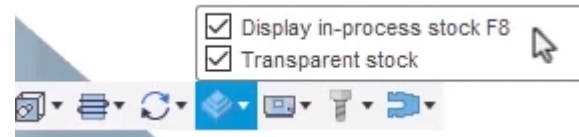


Figure 8. Toggle the in process stock

9. To remove the bulk of the remaining material, click Turning > Turning Profile Roughing.



Figure 9. Create a Turning Profile Roughing operation

10. Navigate to the dialog's Geometry tab and note that you can adjust the planes to focus the operation to a specific area. Drag the Back plane forwards to reduce the area affected by the operation

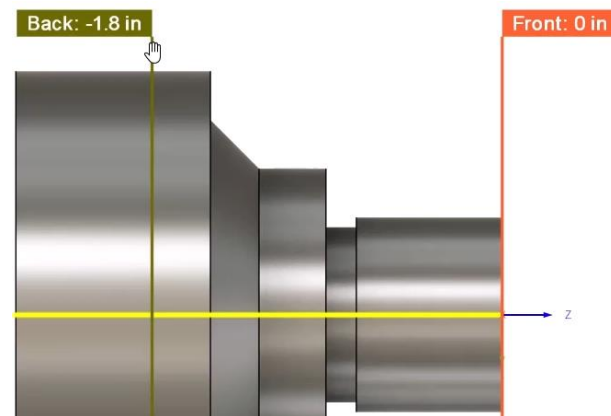


Figure 10. Adjust the Back plane

11. Navigate to the dialog's Radii tab and notice that you can adjust the Clearance, Outer Radius, and Inner Radius. These planes can be adjusted to further limit the area affected by the operation.

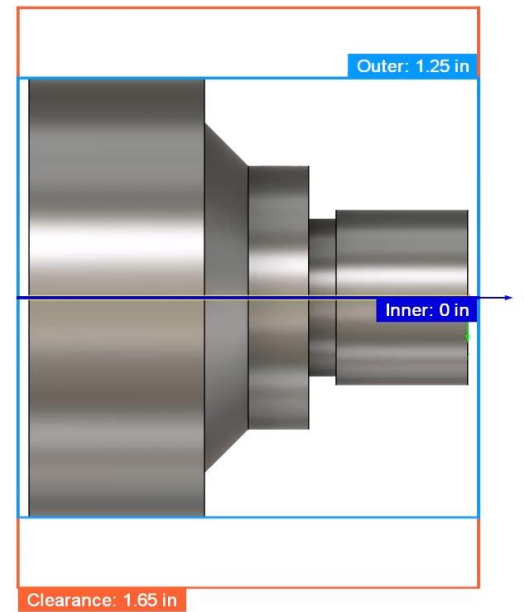


Figure 11. Inspect the radii

12. Continue to the Passes tab and notice that you can change the operation's cycle type, direction, and much more. OK the dialog after you finish exploring the operation's many options.

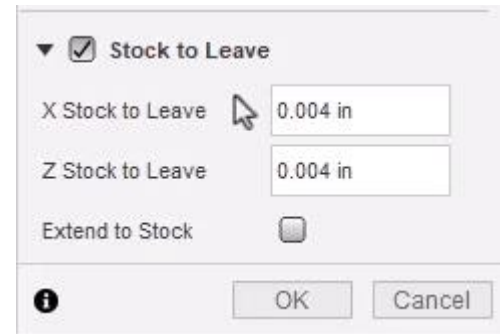


Figure 12. OK the dialog

13. Inspect the toolpath preview and notice the strategy used to rough the part's geometry. The operation leaves 0.004 inches of material on the part because of the operation's Stock to Leave option.

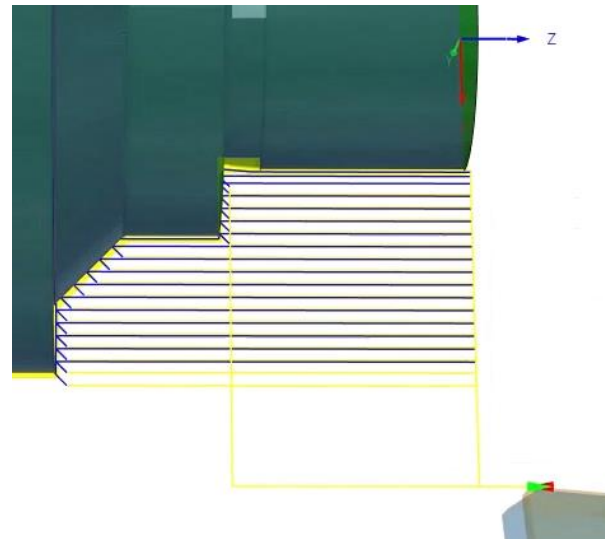


Figure 13. Inspect the toolpath preview

14. To remove the remaining material, create a finishing operation by clicking Turning > Turning Profile Finishing.



Figure 14. Create a Turning Profile Finishing operation

15. Navigate to the dialog's Geometry tab, then drag the Back plane forwards to limit the operation's area.

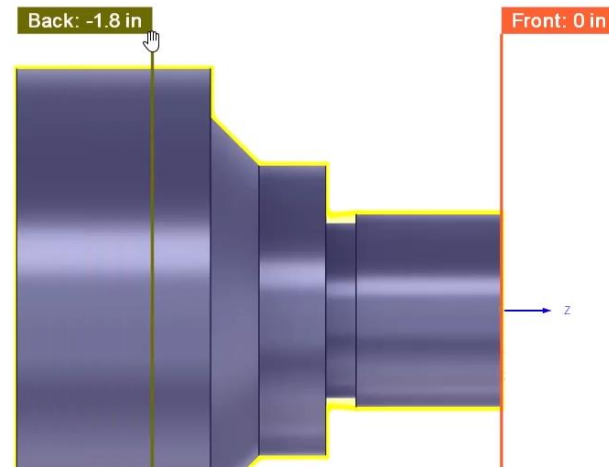


Figure 15. Adjust the Back plane

- 16.** Continue to the dialog's Passes tab and verify that the Stock to Leave option is unchecked. OK the dialog.

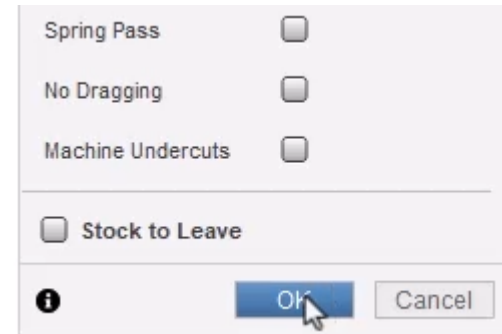


Figure 16. OK the dialog

- 17.** Next, the part's groove needs to be cut. Click Turning> Turning Groove Roughing.

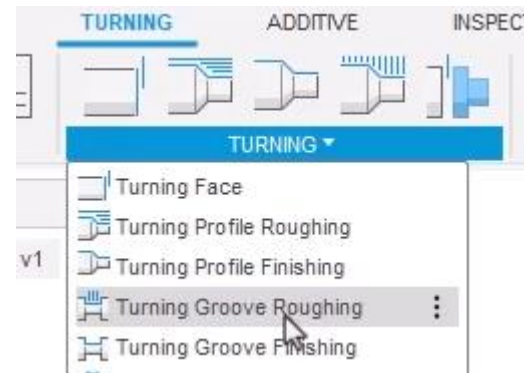


Figure 17. Open the Turning Groove Roughing tool

18. To choose an appropriate tool for this operation, click the Groove Roughing dialog's Select. Navigate to the CAM Lathe Toolpaths library and select Tool 3. Click the Select Tool dialog's Select.

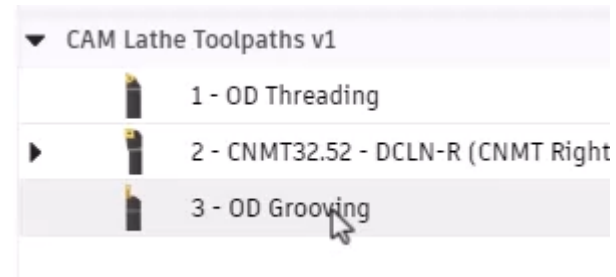


Figure 18. Select the operation's tool

19. OK the Groove Roughing dialog to accept all of the default values. Inspect the toolpath preview and notice that this is not appropriate strategy for cutting the part's groove.

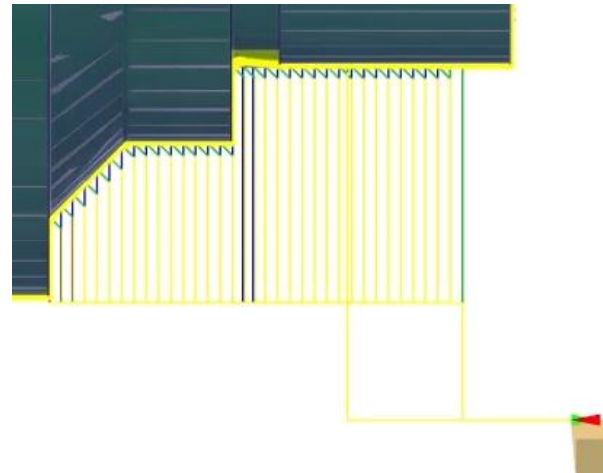


Figure 19. Inspect the toolpath preview

20. Right-click the operation in the Setups folder, right-click it, and choose Delete from the menu.

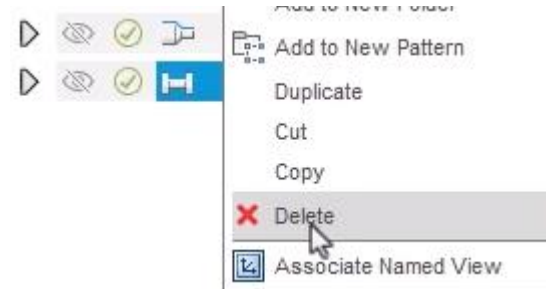


Figure 20. Delete the Groove Roughing operation

21. To try a different operation, click Turning> Turning Single Groove.

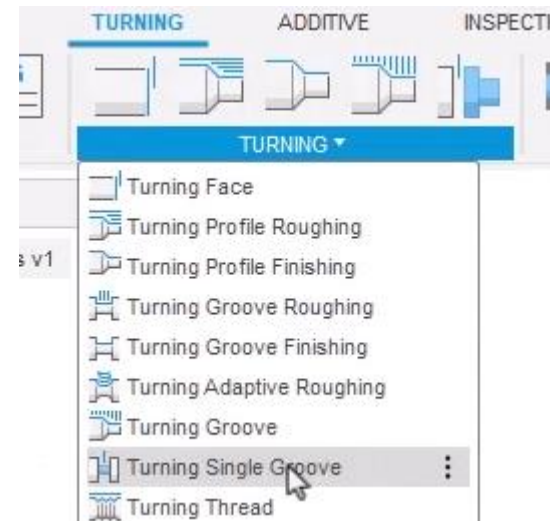


Figure 21. Open the Turning Single Groove tool

22. Click the Single Groove dialog's Select to choose the operation's tool. Choose the same tool you chose in Step 18, then click the Select Tool dialog's Select.

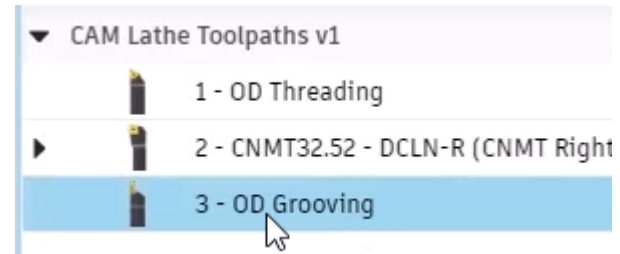


Figure 22. Choose the operation's tool

23. Navigate to the dialog's Geometry tab and choose the edge shown in the image on the right as the Groove Positions selection.

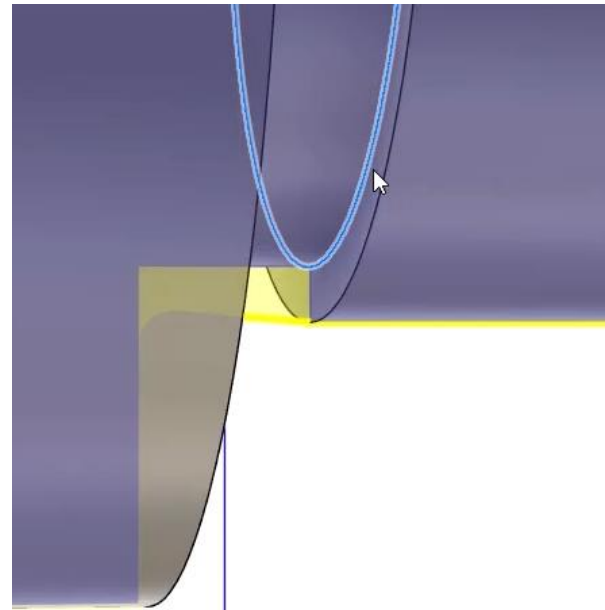


Figure 23. Select the edge

24. Choose the Back option from the Groove Side Alignment menu. This option determines whether the edge you selected in Step 23 will be at the tool's back, middle, or front edge. OK the dialog.

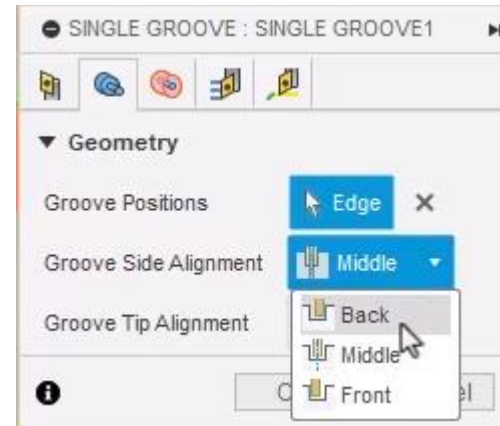


Figure 24. Specify the alignment option

25. Inspect the toolpath preview and notice that the tool correctly cuts the part's groove.

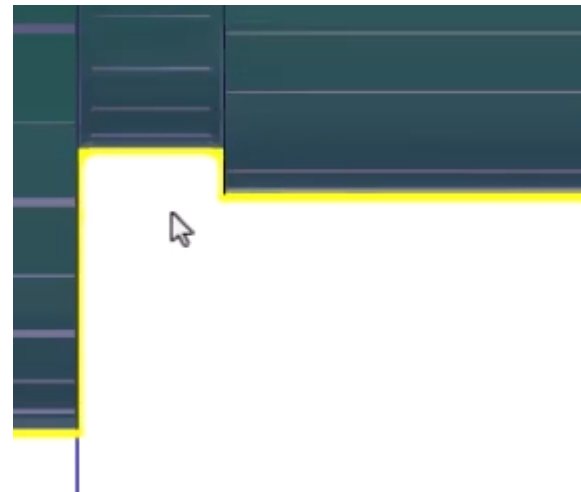


Figure 25. Inspect the toolpath preview

26. Even though small chamfers are not modeled, they need to be added to the machined part. Click Turning> Turning Chamfer.

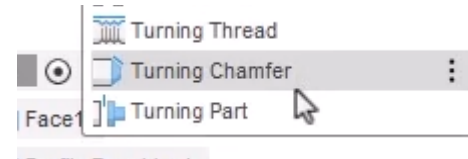


Figure 26. Open the Chamfer tool

27. Click the Chamfer dialog's Select to choose an appropriate tool for the operation. Select Tool 2, then click the Select Tool dialog's Select.

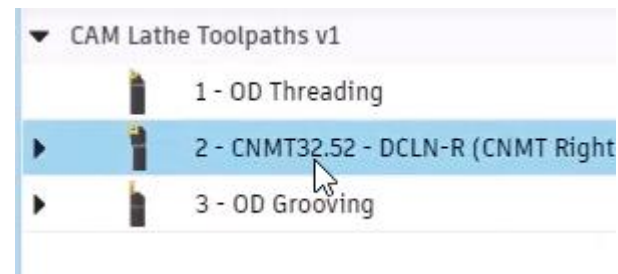


Figure 27. Choose the operation's tool

28. Navigate to the Geometry tab and select the three edges shown in the image on the right.

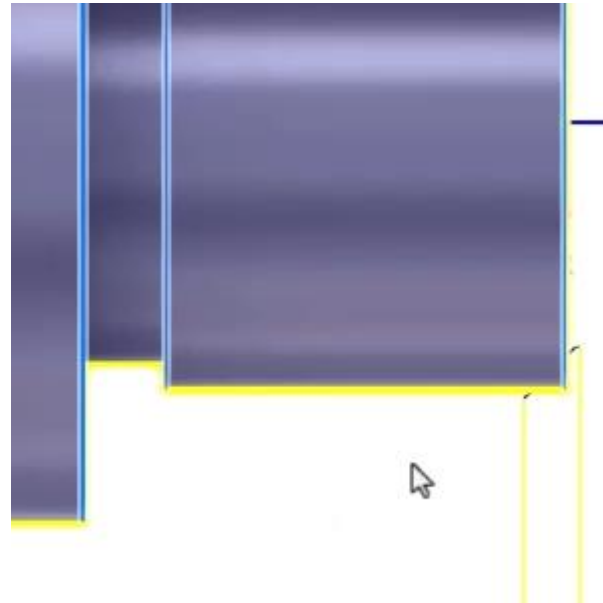


Figure 28. Select the edges you want to chamfer

29. OK the Chamfer dialog and notice that fusion opens an Error dialog. The selected tool cannot cut the middle edge without a collision.

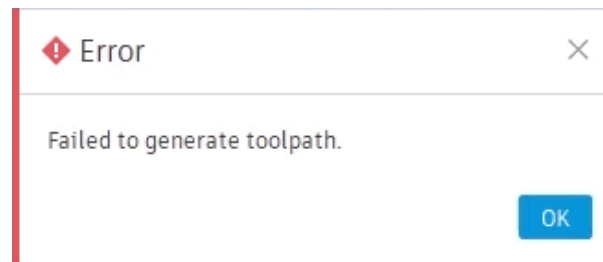


Figure 29. OK the dialog and note the error

- 30.** Right-click the Chamfer operation in the Browser, then choose Edit from the menu.



Figure 30. Edit the Chamfer operation

- 31.** Navigate to the Geometry tab and click the middle edge to remove it from the selection.

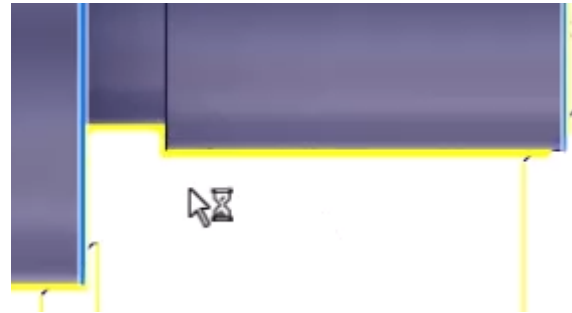


Figure 31. Remove the problematic edge

32. Navigate to the Passes tab, verify the chamfer's parameters, then OK the dialog.

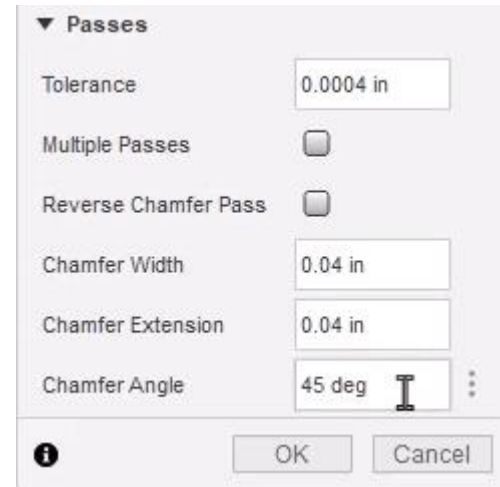


Figure 32. OK the dialog

33. Inspect the toolpath preview and notice that the Chamfer operation revealed the underlying modeled part. This has happened because the chamfers you just created were not included on the 3D model.

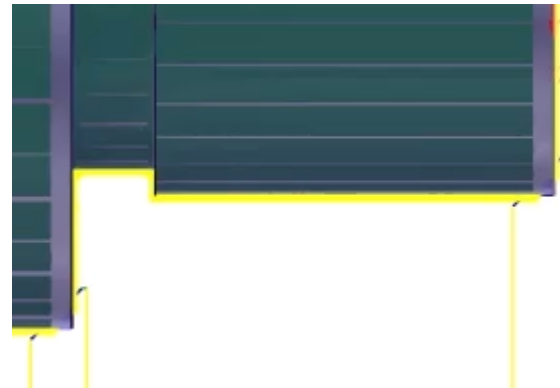


Figure 33. Inspect the toolpath preview

34. Next, threads need to be cut into the part. Once again, the threads are not included on the 3D model. Click Turning> Turning Thread.

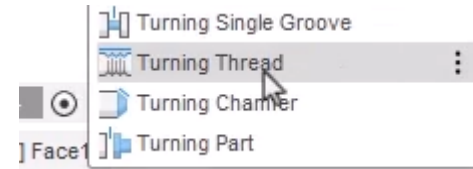


Figure 34. Open the Thread tool

35. Click the Thread dialog's Select to choose an appropriate tool for the operation. Navigate to the CAM Lathe Toolpaths library and choose Tool 1. Click the Select Tool dialog's Select.

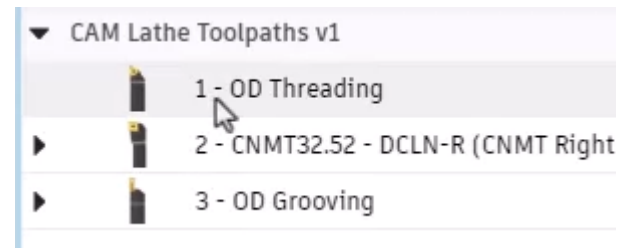


Figure 35. Choose the operation's tool

36. Navigate to the Geometry tab and choose the face shown in the image on the right as the Thread Faces selection.

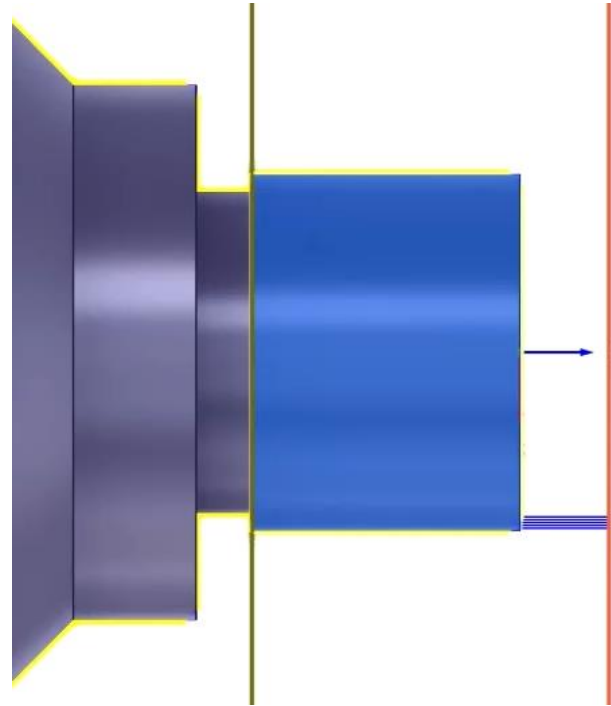


Figure 36. Select the face

- 37.** Drag the Back plane backwards so the threading tool can cut slightly past the selected face. OK the dialog.

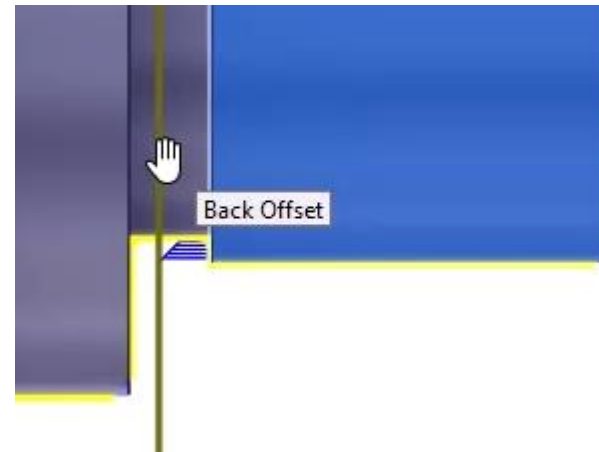


Figure 37. Adjust the Back plane

- 38.** Inspect the toolpath preview and notice that you cannot see threads that the operation created. You will not be able to see the threads until you simulate the setup.

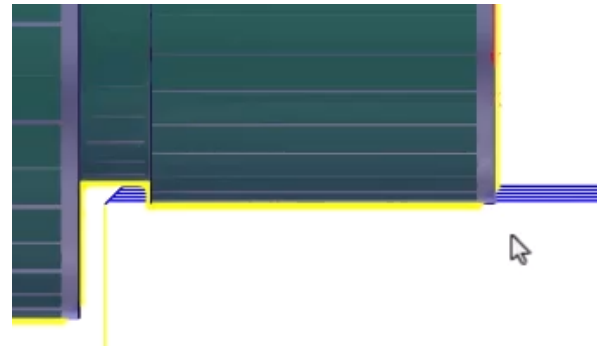


Figure 38. Inspect the toolpath preview

39. After all the operations are complete, the part needs to be cut away from the stock body. Click Turning > Turning Part.

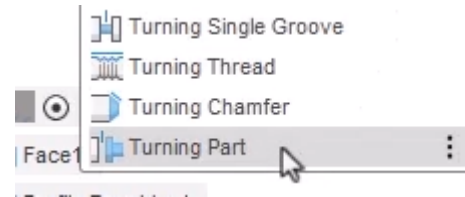


Figure 39. Open the Part tool

40. Click Select to choose an appropriate tool for this operation. Navigate to the CAM Lathe Toolpaths library and choose Tool 3. Click the Select Tool dialog's Select.

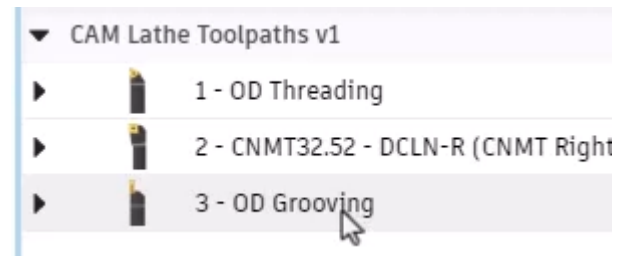


Figure 40. Select the operation's tool

41. Navigate to the Geometry tab, then click and drag the Back plane forwards -1.8 inches. This will move the parting line away from the stock's back face. OK the dialog.

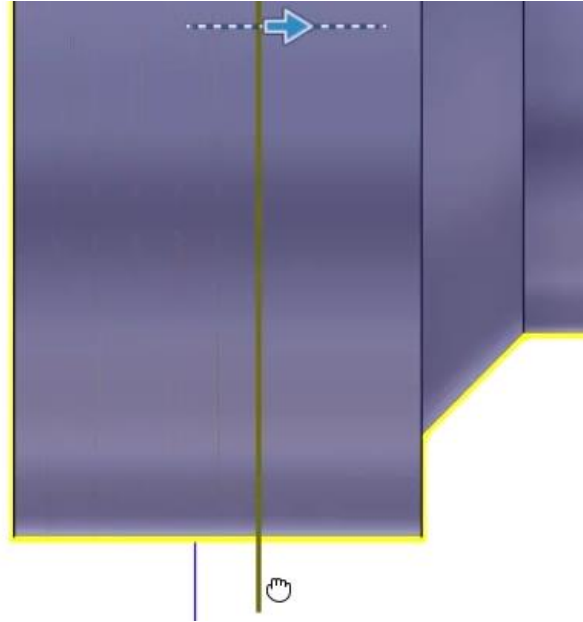


Figure 41. Move the Back plane

42. Once again, the operation appears to have removed too much material. This is misleading because of the way the part was modeled. Save the design.

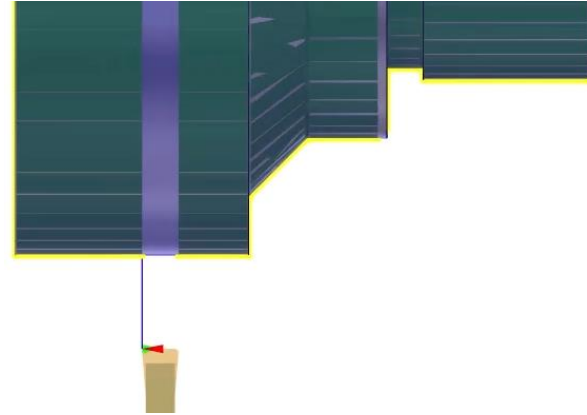


Figure 42. Inspect the toolpath preview