

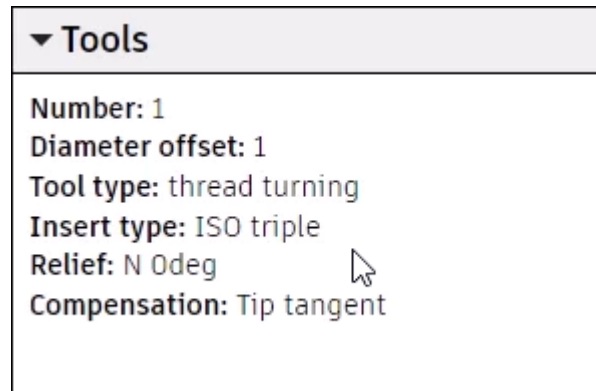
Step-by-step guide

Create setup sheets and NC programs for lathe

Convert digital operations to machine-readable G-Code and create a setup sheet describing the job.

Learning objectives:

- Create an NC program.
- Create a setup sheet.
- Post Process toolpaths.



The completed exercise

1. Open the supplied *CAM Lathe Simulation.f3d* file or continue with your file from the previous module. In the previous modules, this part's operations were created and were verified using simulation.



Figure 1. Open the supplied file

2. Before the part can be cut on a machine, the operations need to be converted to G-Code. To begin this process, click Setup> Create NC Program.

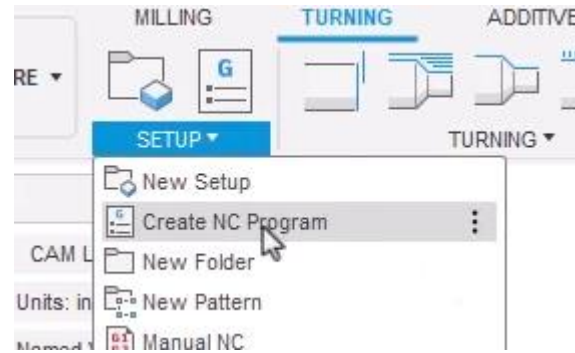


Figure 2. Create a new NC program

3. The first step is to pick the correct machine. Expand the Post drop-down and select Choose from library.

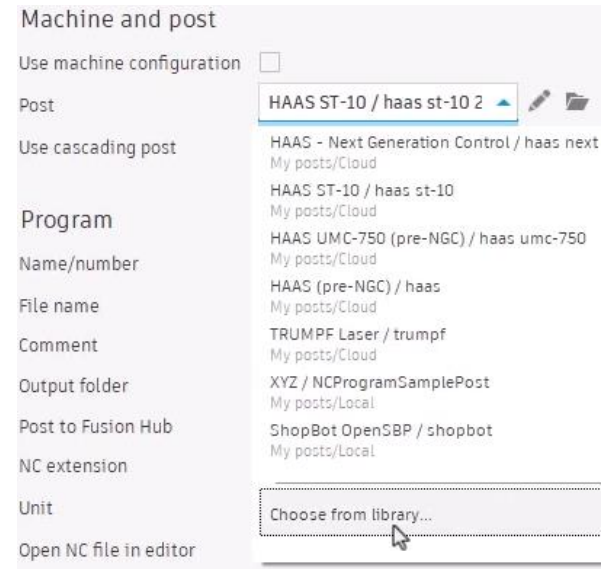


Figure 3. Choose the machine

4. Open the Fusion library by selecting it in the Post Library dialog.

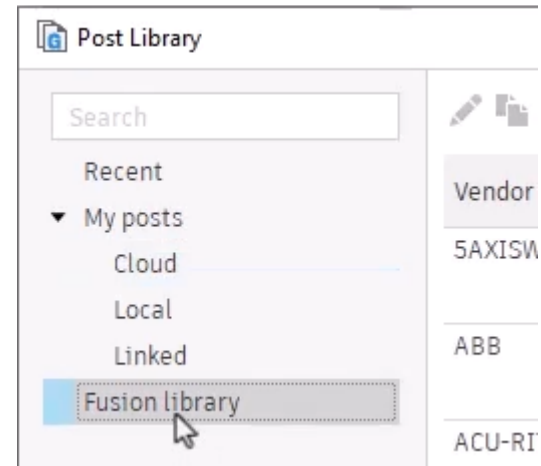


Figure 4. Open the Fusion library

5. Filter the many machine options by selecting the Turning and Haas Automation filters.

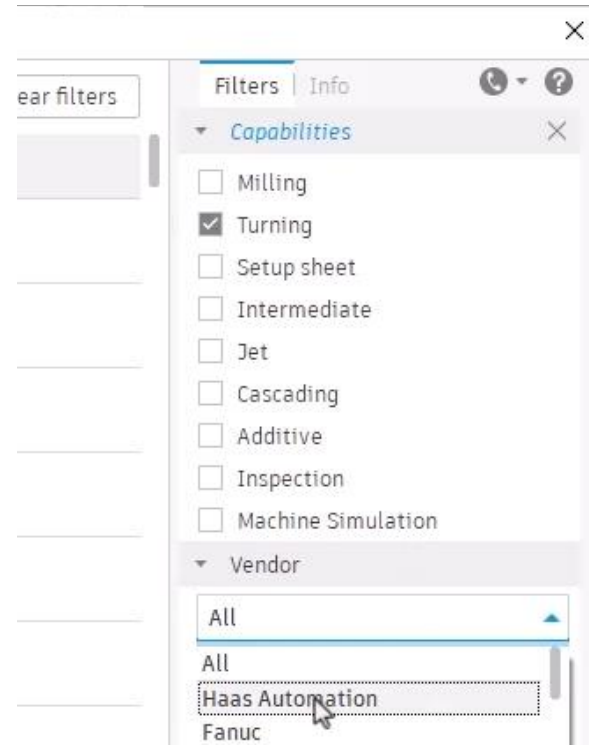


Figure 5. Filter the machines

6. Locate and select the machine you want to use.

Haas Automation	HAAS DS-30Y
Haas Automation	HAAS ST-10
Haas Automation	HAAS ST-10L

Figure 6. Select your machine

7. Click the Post Library dialog's Select.

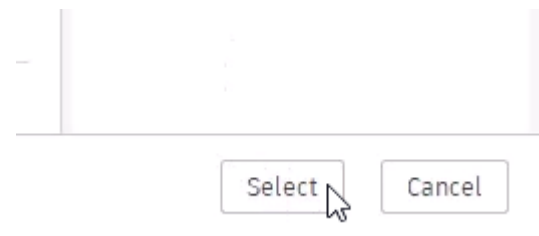


Figure 7. Click Select

8. If you have never selected this machine before, you'll be asked to save the machine. Choose to save it on the Cloud, then click Copy to My Posts.

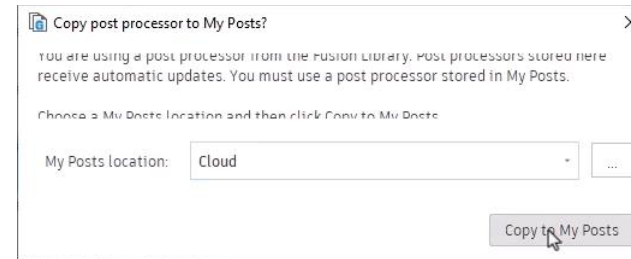


Figure 8. Save the machine

9. The information in the Program section is automatically imported from the setup's information. The NC program can override the setup's information if you change the name, number, or comment.



Figure 9. Update the program's information

10. Enter the name you want to use into the Name box. In this instance, **Setup4** is used to match the design's setup name.

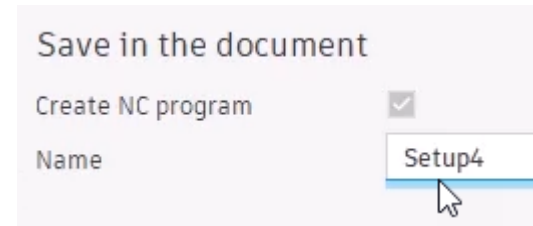


Figure 10. Choose the program's name

11. You can use the options in the Post Properties column to configure your specific machine's capabilities. For example, the chip conveyor, live tooling, and bar features can be toggled.

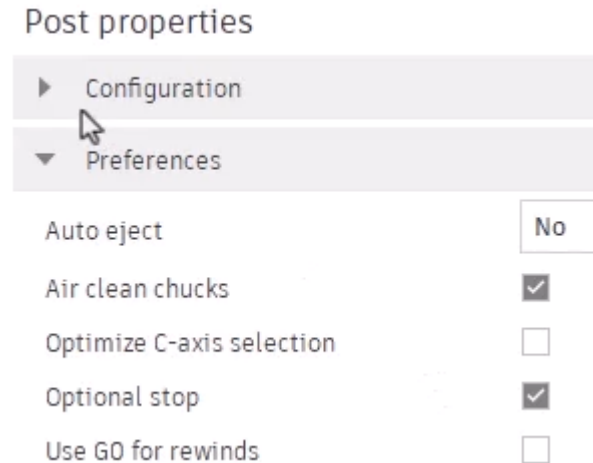


Figure 11. Explore ways to configure your machine

12. Continue to the Operations tab and notice you can choose the operations you want to include or exclude. Select all of the setup's operations by clicking the top level check box. OK the NC Program dialog to save the program.

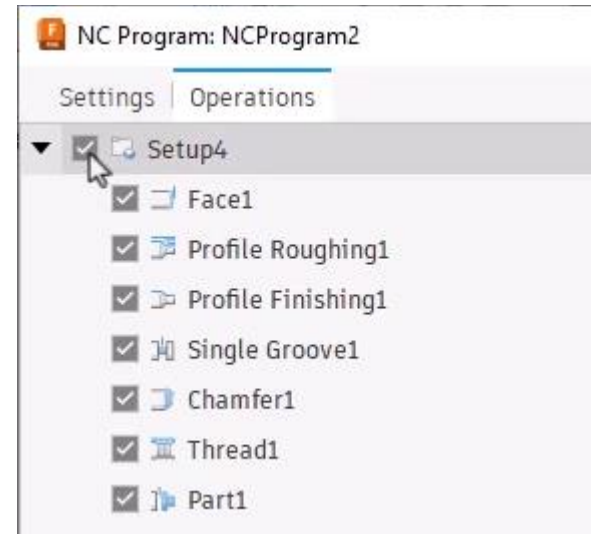


Figure 12. Select all of the operations

13. Right-click the program inside the Browser's NC Programs folder, then select Post Process from the menu.

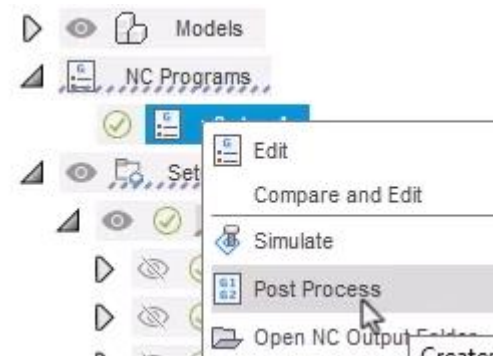


Figure 13. Post process the NC program

14. If the program name conflicts with an existing program, you'll be asked whether you want to overwrite the file.

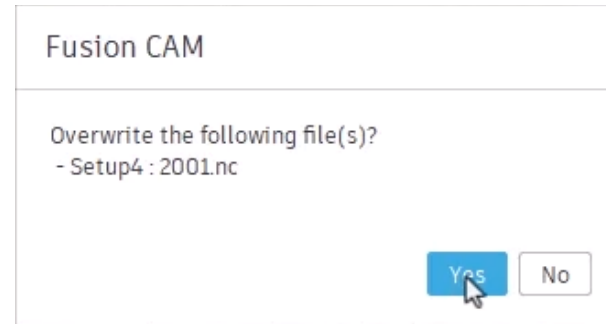


Figure 14. Determine whether you want to overwrite existing files

15. After the G-Code automatically opens in a text editor, explore the code's information. For example, the program name and number are listed on Line 2 and the spindle speed is listed on Line 5.

```
1  %
2  002001 (SIMPLE LATHE PART)
3  G98 G18
4  G20
5  G50 S6000
6  M31
7  G53 G0 X0.
8  G53 Z0.
9
10 (Face1)
```

Figure 15. Explore the G-Code

- 16.** Continue to explore the code and notice that the operation's name is displayed. You can customize this operation's name in the Browser to help organize the G-Code. The tool information and WCS location are listed on Lines 15 and 16. Close the G-Code after you finish exploring it.

```
10 (Face1)
11 T202
12 G99
13 G18
14 G97
15 S677 M3
16 G54
17 M8
18 G0 Z0.1969
```

Figure 16. Continue to explore the G-Code

- 17.** Notice that the NC program now has a warning icon next to it.



Figure 17. Notice the warning icon

- 18.** Click the icon to learn more information about the warning. In this instance, the setup's WCS offset is set as 0 which defaults to using the G54 location. Close the warning dialog.

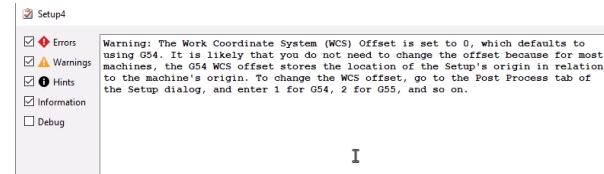


Figure 18. Learn more about the warning

- 19.** Double-click the setup to edit it.



Figure 19. Edit the setup

20. Navigate to the Post Process tab and customize the information. Enter **2002** into the Program Name/Number box, then enter **Simple Lathe Part** into the Program Comment box. Enter **1** into the WCS Offset box to clear the warning icon, then OK the dialog.

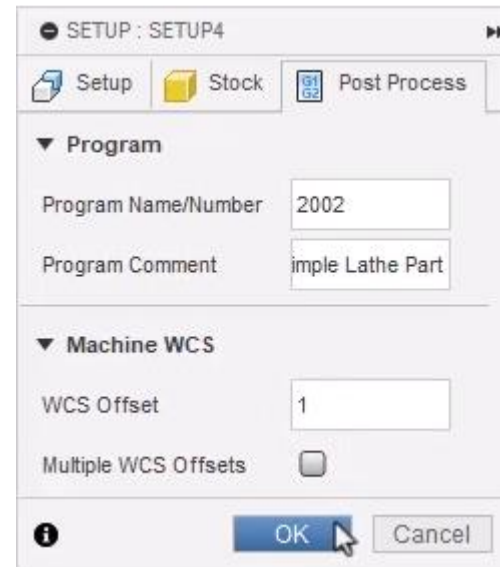


Figure 20. Customize the setup's information

21. The icon next to the NC program indicates that it is out of date and needs to be rebuilt. Select the NC program.



Figure 21. Notice the new icon

22. Click Actions> Generate to rebuild the selected program.

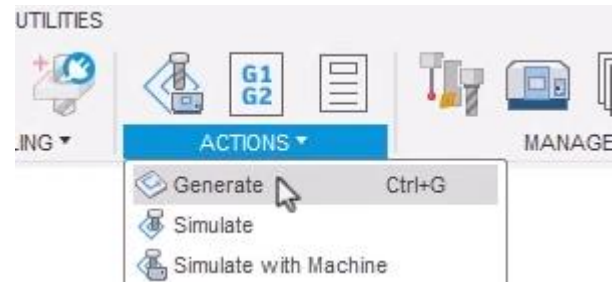


Figure 22. Rebuild the program

23. Edit the NC program and notice that this dialog's information is overriding the information you entered in Step 20. Update the Name/number and File name to **2002**, then OK the dialog.

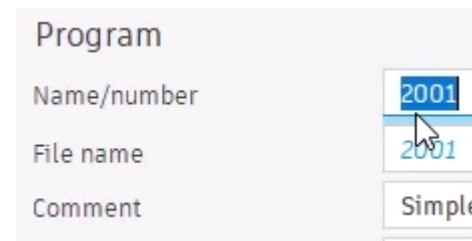


Figure 23. Update the program's information

24. Update the existing program by right-clicking it and choosing Post Process from the menu.

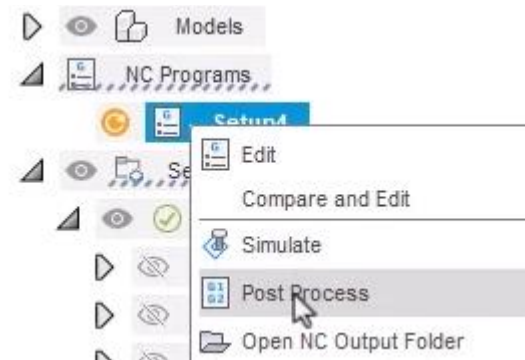


Figure 24. Post process the program

25. Verify that the program's information is updated. The new program name and description is displayed on Line 2, and the G54 WCS location is listed on Line 16. Close the G-Code.

```
1  %  
2  002002 (SIMPLE LATHE PART)  
3  G98 G18  
4  G20  
5  G50 S6000  
6  M31  
7  G53 G0 X0.  
8  G53 Z0.  
9  
10 (Face1)  
11 T202  
12 G99  
13 G18  
14 G97  
15 S677 M3  
16 G54  
17 M8
```

Figure 25. Inspect the G-Code

26. Notice that the program's icon has changed to a green checkmark. Right-click the program and choose Setup Sheet from the menu.

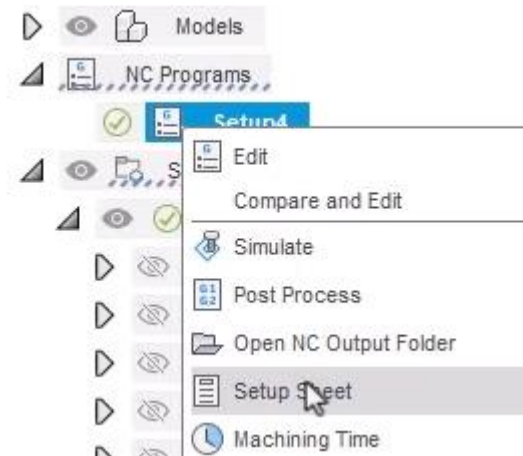


Figure 26. Create the program's setup sheet

27. Choose the sheet's name and save location, then click the dialog's Save.

Publish setup sheet: Setup4

Name:

2002

Location:

23 Module Series > 11 CAM Lathe > 2025 Update

Figure 27. Save the setup sheet

28. Explore the setup sheet after it opens. The options inside the Configuration menu can be used to customize the types and amount of information displayed inside the setup sheet.

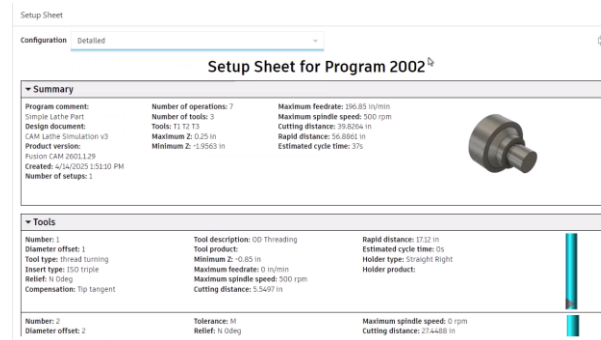


Figure 28. Explore the setup sheet

29. The information inside the Summary section can be used to determine the design's comment, number of setups, number of tools, maximum feed rate, spindle speed, and more.

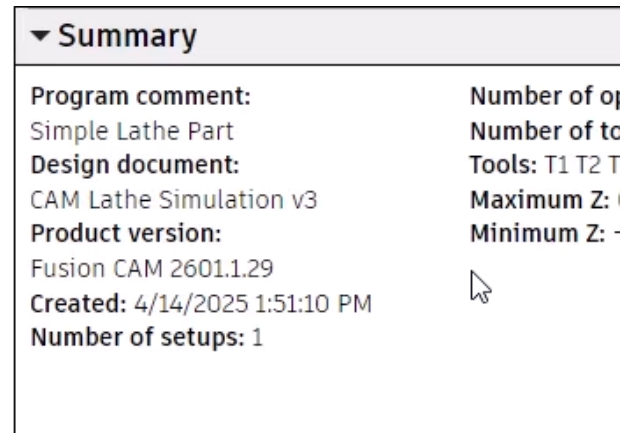


Figure 29. Inspect the sheet's summary

30. The Tools section gives you detailed information about each tool needed for this job.

Tools

Number: 1

Diameter offset: 1

Tool type: thread turning

Insert type: ISO triple

Relief: N 0deg

Compensation: Tip tangent

Figure 30. Inspect the Tools section

31. The Setup section describes the WCS location, the stock dimensions, and more.

Setup

Description: Setup4

WCS: 1

Stock:

x: 2.5 in

y: 2.5 in

z: 2.5 in

Part:

x: 2.5 in

y: 2.5 in

z: 2.5 in

Figure 31. Inspect the Setup section

32. The Operations section lists important details about each operation. For example, whether or not the coolant needs to be turned on, the operation's tool, the feed rate, and much more.

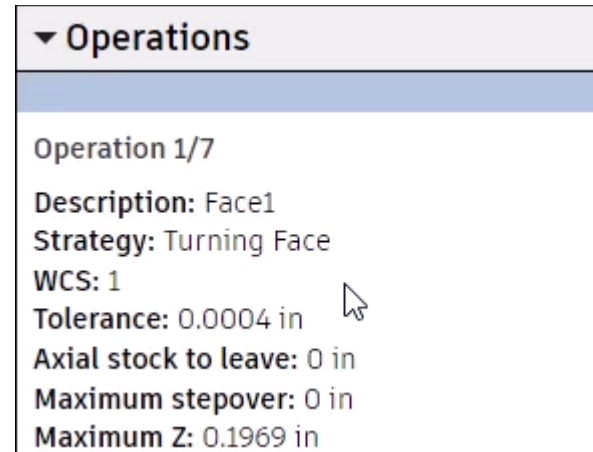


Figure 32. Inspect the Operations section

33. You can print a paper copy of the setup sheet by clicking the printer icon in the top right corner. This setup sheet can be a valuable tool to help a machine operator properly prepare for the job. Close the setup sheet after you finish exploring its information.



Figure 33. Print the setup sheet

- 34.** The default save location for the setup sheet is inside the Fusion project. Anybody who has access to the project can access the setup sheet. Save the design.

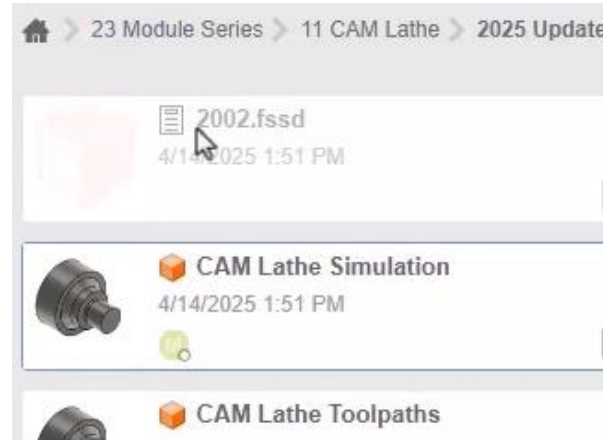


Figure 34. Note the setup sheet location