

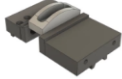
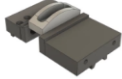
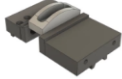
Step-by-step guide

Create setup sheets and NC programs for milling

Learn how to create a setup's NC program, then create setup sheet so that a machine operator can prepare a machine.

Learning objectives:

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

Setup Sheet											
Configuration: Detailed											
Setup Sheet for Program 1001											
▼ Summary <table> <tr> <td> Program comment: Module Series Design document: CAM 3x Toolpaths Programmed v2 Product version: Fusion 360 CAM 2.0.16976 Created: Tuesday, August 15, 2023 15:15:11 Number of setups: 1 </td><td> Number of operations: 3 Number of tools: 2 Tools: T1 T2 Maximum Z: 0.6 in Minimum Z: -1 in </td><td> Maximum feedrate: 159.29 in/min Maximum spindle speed: 7639 rpm Cutting distance: 700.7501 in Rapid distance: 257.3216 in Estimated cycle time: 5m 1s </td><td>  </td></tr> </table>				Program comment: Module Series Design document: CAM 3x Toolpaths Programmed v2 Product version: Fusion 360 CAM 2.0.16976 Created: Tuesday, August 15, 2023 15:15:11 Number of setups: 1	Number of operations: 3 Number of tools: 2 Tools: T1 T2 Maximum Z: 0.6 in Minimum Z: -1 in	Maximum feedrate: 159.29 in/min Maximum spindle speed: 7639 rpm Cutting distance: 700.7501 in Rapid distance: 257.3216 in Estimated cycle time: 5m 1s					
Program comment: Module Series Design document: CAM 3x Toolpaths Programmed v2 Product version: Fusion 360 CAM 2.0.16976 Created: Tuesday, August 15, 2023 15:15:11 Number of setups: 1	Number of operations: 3 Number of tools: 2 Tools: T1 T2 Maximum Z: 0.6 in Minimum Z: -1 in	Maximum feedrate: 159.29 in/min Maximum spindle speed: 7639 rpm Cutting distance: 700.7501 in Rapid distance: 257.3216 in Estimated cycle time: 5m 1s									
▼ Tools <table> <tr> <td> Number: 1 Diameter offset: 1 Length offset: 1 Tool type: Flat end mill Diameter: 0.5 in Length: 1.1 in </td><td> Number of flutes: 3 Tool description: 1/2" Flat Endmill Tool product: Minimum Z: -1 in Maximum feedrate: 126.06 in/min Maximum spindle speed: 7639 rpm </td><td> Cutting distance: 375.4468 in Rapid distance: 247.0271 in Estimated cycle time: 2m 59s Holder description: Maritool CAT40-ER32-2.35 Holder vendor: Maritool Holder product: CAT40-ER32-2.35 </td><td>  </td></tr> <tr> <td> Number: 2 Diameter offset: 2 </td><td> Number of flutes: 3 Tool description: 1/2" Ball Endmill </td><td> Rapid distance: 10.2945 in Estimated cycle time: 2m 3s </td><td>  </td></tr> </table>				Number: 1 Diameter offset: 1 Length offset: 1 Tool type: Flat end mill Diameter: 0.5 in Length: 1.1 in	Number of flutes: 3 Tool description: 1/2" Flat Endmill Tool product: Minimum Z: -1 in Maximum feedrate: 126.06 in/min Maximum spindle speed: 7639 rpm	Cutting distance: 375.4468 in Rapid distance: 247.0271 in Estimated cycle time: 2m 59s Holder description: Maritool CAT40-ER32-2.35 Holder vendor: Maritool Holder product: CAT40-ER32-2.35		Number: 2 Diameter offset: 2	Number of flutes: 3 Tool description: 1/2" Ball Endmill	Rapid distance: 10.2945 in Estimated cycle time: 2m 3s	
Number: 1 Diameter offset: 1 Length offset: 1 Tool type: Flat end mill Diameter: 0.5 in Length: 1.1 in	Number of flutes: 3 Tool description: 1/2" Flat Endmill Tool product: Minimum Z: -1 in Maximum feedrate: 126.06 in/min Maximum spindle speed: 7639 rpm	Cutting distance: 375.4468 in Rapid distance: 247.0271 in Estimated cycle time: 2m 59s Holder description: Maritool CAT40-ER32-2.35 Holder vendor: Maritool Holder product: CAT40-ER32-2.35									
Number: 2 Diameter offset: 2	Number of flutes: 3 Tool description: 1/2" Ball Endmill	Rapid distance: 10.2945 in Estimated cycle time: 2m 3s									

The completed exercise

1. Upload and open the supplied *CAM 3x Toolpaths Programmed.f3d* file.

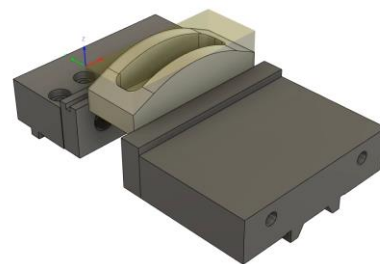


Figure 1. Open the supplied file

2. After all the operations have been created, the information needs to be converted to machine-readable code so that the parts can be cut. Click Setup> Create NC Program.

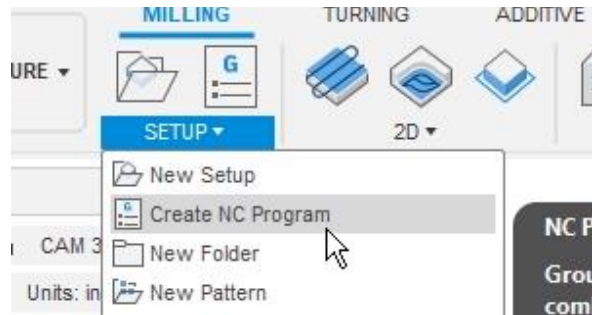


Figure 2. Create a new NC program

3. In the NC Program dialog, you need to select the correct post for your machine. If you don't see the correct option available in the Post menu, click the folder icon to browse for it.

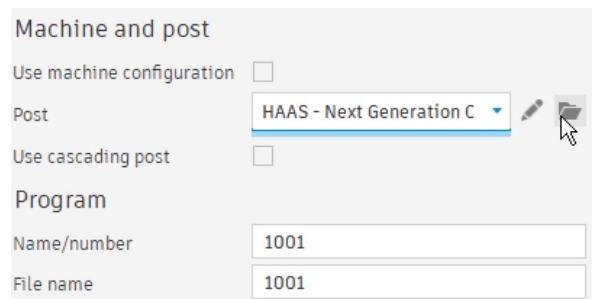


Figure 3. Browse for the correct post

4. Locate and select the Fusion 360 library option in the dialog's left column.

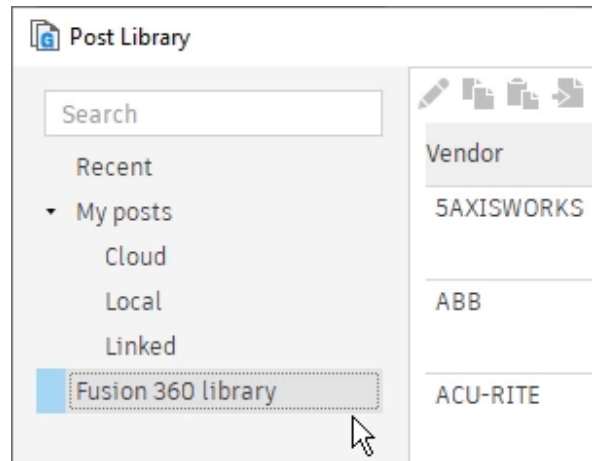


Figure 4. Select the Fusion 360 library

5. Use the options in the Filters tab to filter the posts. Select Milling from the Capabilities section, then choose Haas Automation from the Vendor menu.

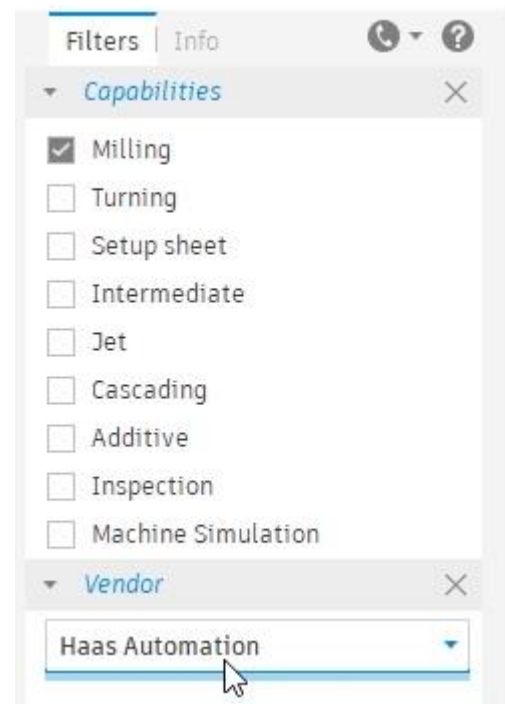


Figure 5. Filter the posts

6. Locate and select the HAAS - Next-Generation Control post, then click the Post Library dialog's Select.

Haas Automation	HAAS - Next Generation Control I
Haas Automation	HAAS - Next Generation Control
Haas Automation	HAAS ST-10

Figure 6. Select the Post

7. Click the dialog's Copy to My Posts.

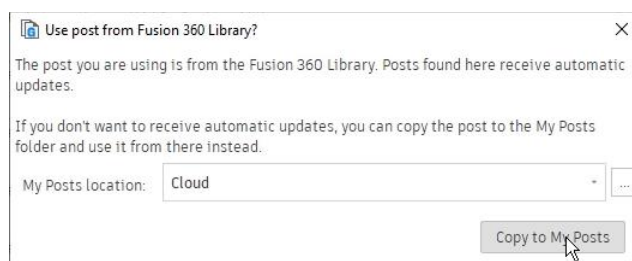


Figure 7. Click Copy to My Posts

8. Select the correct post from the dialog's Post menu, then enter the program's information into the Program section.

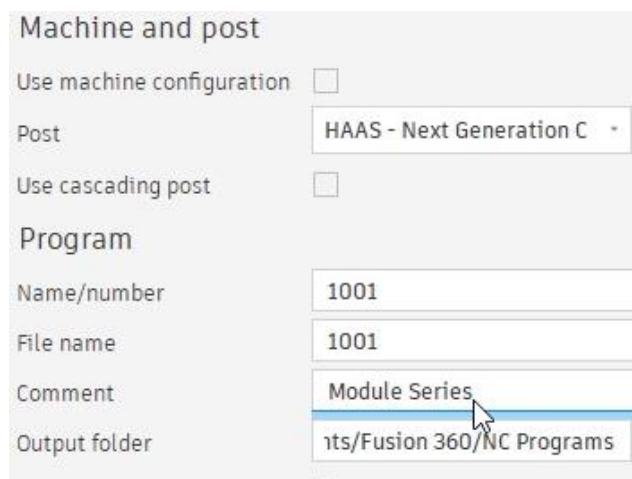


Figure 8. Select the correct post from the menu

9. The options in the Post properties section can be used to describe your machine's capabilities. For instance, you can determine whether or not your machine uses chip transport, a rotary table, a probe arm, and more.

Post properties

Configuration

Preferences

Coolant pressure

Default

Fast tool change

☐

Optionally measure tools at start

☐

Optional stop

☒

Optionally cycle tools at start

☐

Preload tool

☒

Safe start all operations

☐

Figure 9. Note the options in the Post properties section

10. Continue to the dialog's Operations tab. Verify that the operations use the correct work offset and tool. OK the NC Program dialog.

Reorder to Minimize Tool Changes ☐

Operation	Instance	Setup	Work Offset
Adaptive1	1/1	Setup1	0
2D Pocket1	1/1	Setup1	0
Scallop1	1/1	Setup1	0

Figure 10. Verify that the operations are correct

11. Notice that the new NC program is added to the Browser's NC Programs section.

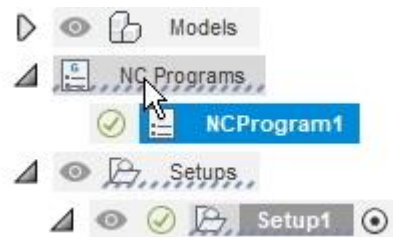


Figure 11. Inspect the Browser

12. If you modify any of the setup's operations, this NC program will be out of date. Modify one of the operations and notice that the NC program has an orange out of date icon.

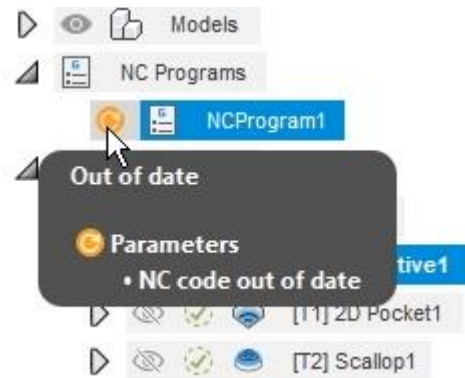


Figure 12. Modify one of the operations

13. Note that you can simulate the NC program to verify that it is correct.

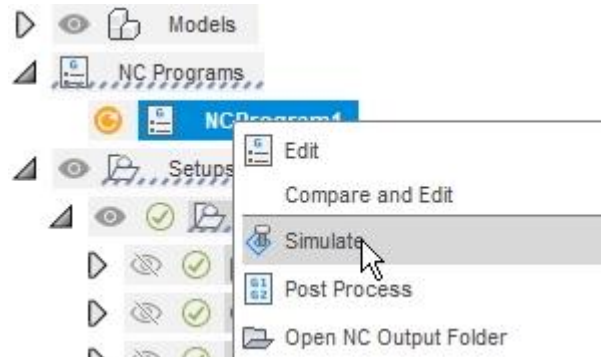


Figure 13. Note the option to simulate

14. Right-click the NC program and choose Machining Time from the menu.

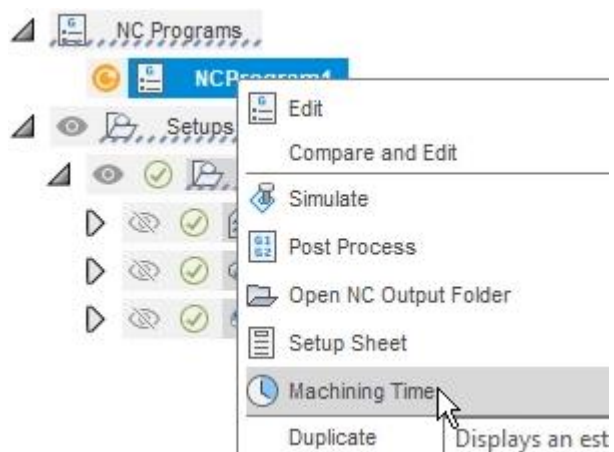


Figure 14. Inspect the NC program's machining time

15. Explore the dialog's information. For instance, the entire setup only has 13 seconds of rapid movements. Close the dialog after you finish exploring information.

MACHINING TIME	
Feed scale (%)	95
Rapid feed (in/min)	1181.1
Tool change time (s)	15
Feed distance	700.75 in
Total feed time	0:05:17
Rapid distance	257.322 in
Total rapid time	0:00:13
Tool changes	1
Total tool change time	0:00:15
Machining time	0:05:45

Figure 15. Inspect the dialog's information

16. Right-click the NC program and choose Post Process from the menu.

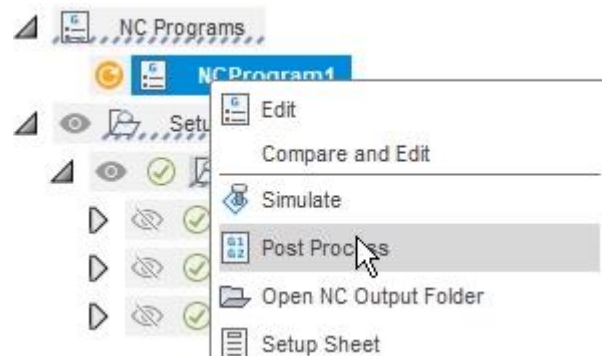


Figure 16. Post process the NC program

17. The NC program is converted to G-Code and opens in a text editor window. Notice that the program's name and comment are listed on Line 2. The tools are described on Lines 4 and 5. The first operation's name is shown on Line 11 and Line 15 specifies the G54 coordinate system. Close the text editor window after you finish exploring the G-Code.

```

1  %
2  001001 (Module Series)
3  (Using high feed G1 F650. i
4  (T1 D=0.5 CR=0. - ZMIN=-1.
5  (T2 D=0.5 CR=0.25 - ZMIN=-0
6  N10 G90 G94 G17
7  N15 G20
8  N20 M31
9  N25 G53 G0 Z0.
10
11  (Adaptive1)
12  N30 T1 M6
13  N35 S7639 M3
14  N40 G17 G90 G94
15  N45 G54
16  N50 M8

```

Figure 17. Inspect the G-Code

18. Fusion 360 can create a setup sheet to help a machine operator prepare a machine for the job. Right-click the NC Code and select Setup Sheet from the menu.

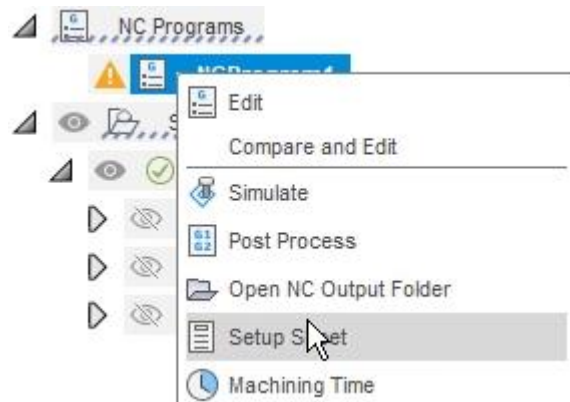


Figure 18. Create a setup sheet

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

Publish setup sheet: NCPro

Name:

1001 Setup Sheet

Location:

23 Module Series > 10 CAM 2.5 and 3x

Figure 19. Save the setup sheet

20. The setup sheet will open in a new dialog.

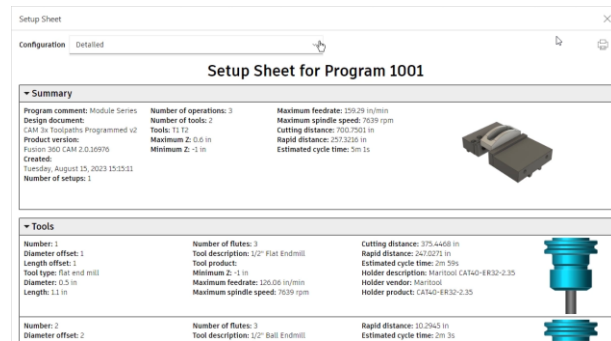


Figure 20. Explore the setup sheet

21. Explore the Summary section and notice that you can learn things like the number of operations, number of tools, maximum feed rate, estimated cycle time, and much more.

▼ Summary	
Program comment: Module Series	Number of operations: 3
Design document: CAM 3x Toolpaths Programmed v2	Number of tools: 2
Product version: Fusion 360 CAM 2.0.16976	Tools: T1 T2
Created: Tuesday, August 15, 2023 15:15:11	Maximum Z: 0.6 in
Number of setups: 1	Minimum Z: -1 in

Figure 21. Inspect the Summary section

22. Inspect the Tools section and notice that each tool is thoroughly described.

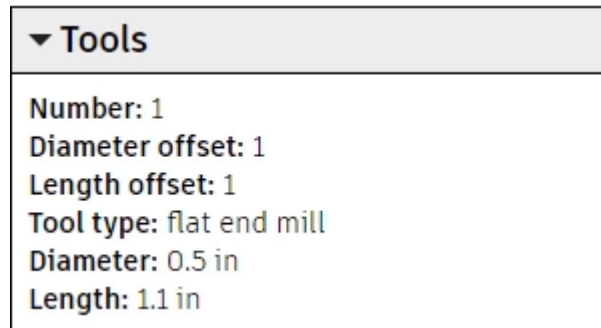


Figure 22. Export the Tools section

23. The Operations section describes each of the setup's operations. For instance, you can learn the estimated cycle time, whether the operation uses coolant, the operation's tool, and much more.

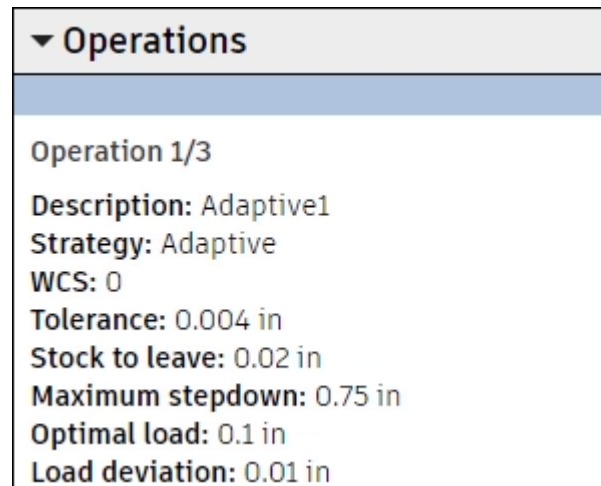


Figure 23. Explore the Operations section

24. The dialog's Configuration menu can be used to customize the sheet's information. For instance, the Tools only option will only display the Tools section.

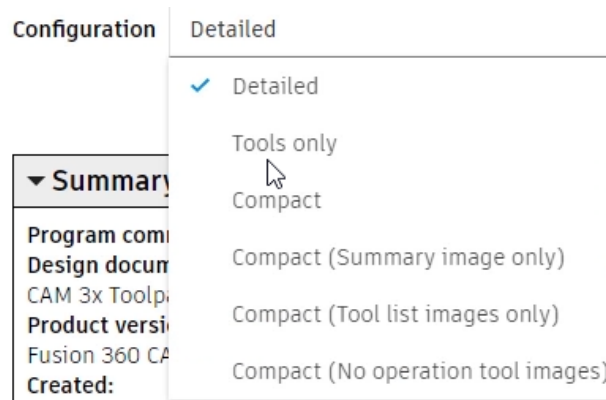


Figure 24. Notice the Configuration menu

25. The setup sheet can be printed by clicking the printer icon in the sheet's top right corner. Close the setup sheet after you finish exploring it.

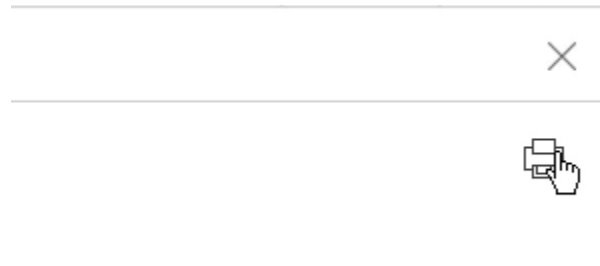


Figure 25. Notice that you can print the sheet

26. You can also create a setup sheet using the setup. Right-click Setup1 and choose Setup Sheet from the menu.

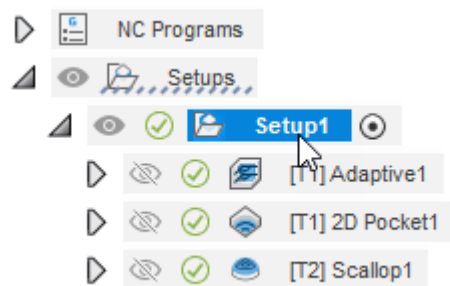


Figure 26. Use the setup to create a setup sheet

27. Choose the location to save the setup sheet.

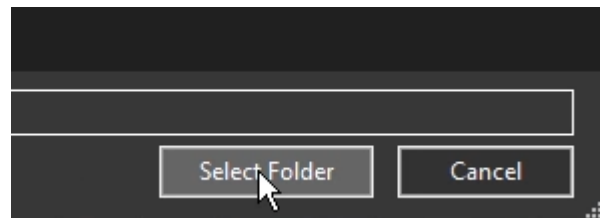


Figure 27. Choose the setup sheet's save location

28. Open the HTML setup sheet and notice that most of the information is the same as the first setup sheet you created. Close the setup sheet after you finish exploring it.

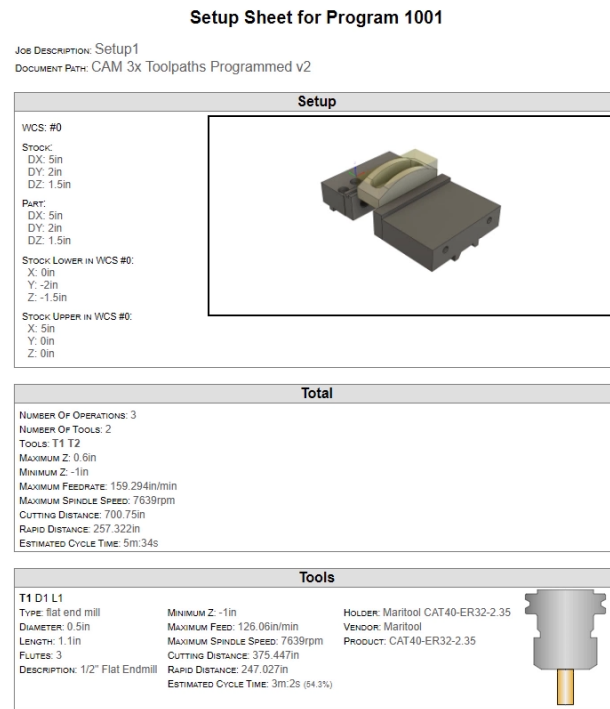


Figure 28. Explore the new setup sheet

29. Expand the Actions group's menu and notice that you can post process a setup, create a setup sheet, or check the Machining Time information. Make sure Setup1 is selected in the Browser and choose Actions> Post Process.

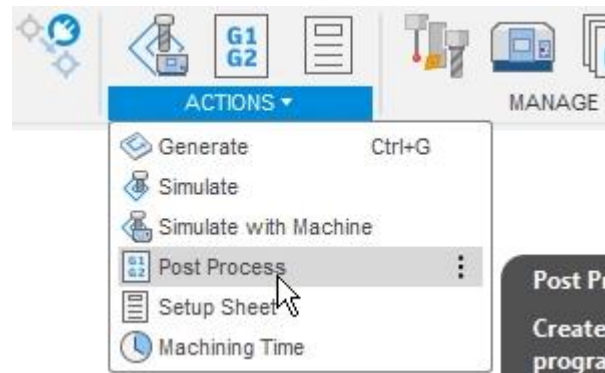


Figure 29. Notice the Actions group's options

30. Click the folder icon next to the Post menu to browse for a new post.

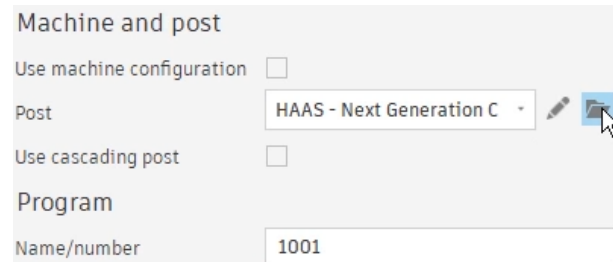


Figure 30. Browse for a new post

31. Use the dialog's left column to navigate to the Fusion 360 library.

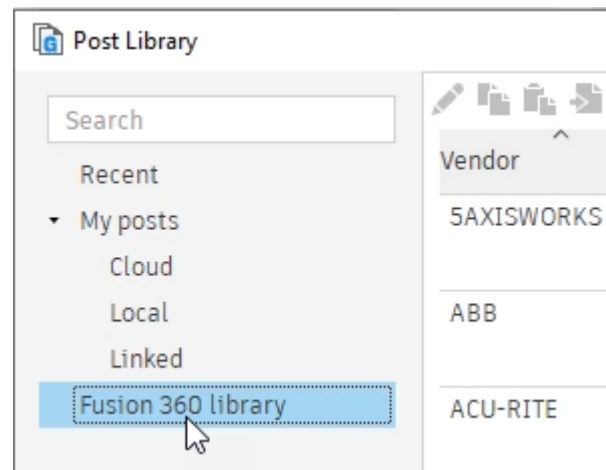


Figure 31. Open the Fusion 360 library

32. In the Filters tab, activate the Setup sheet option inside the Capabilities section.

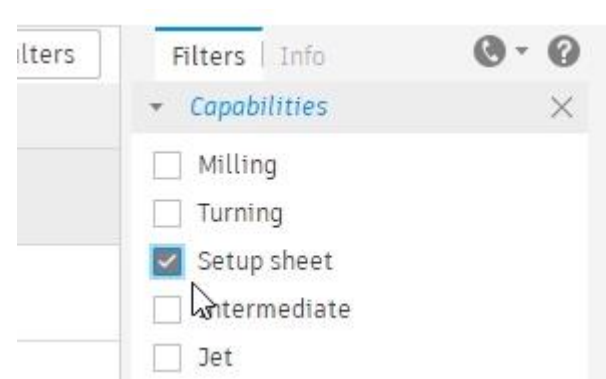
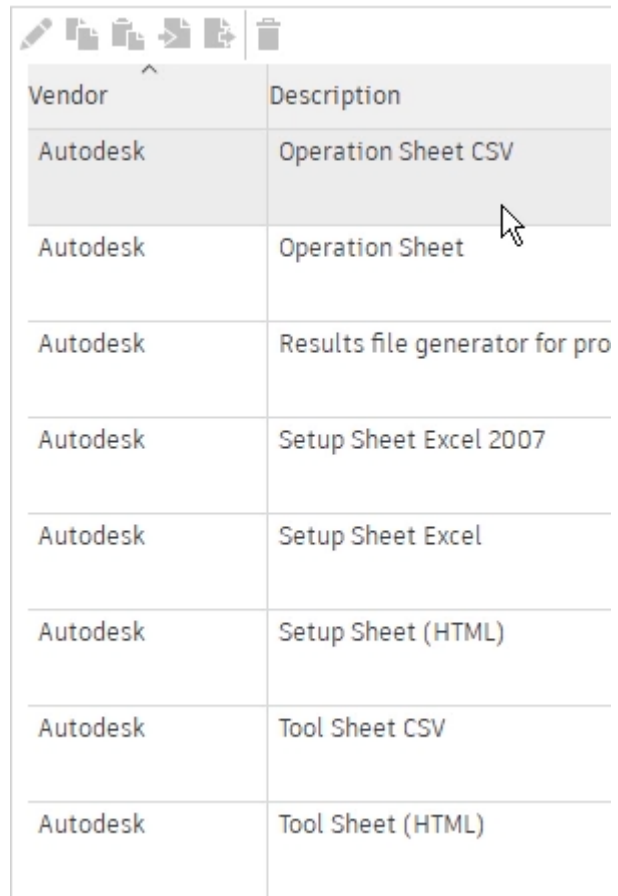


Figure 32. Filter the posts

33. Notice that you can export the setup sheet using several different file types. Cancel the Post Library dialog, then cancel the NC Program dialog. Save the design.



Vendor	Description
Autodesk	Operation Sheet CSV
Autodesk	Operation Sheet
Autodesk	Results file generator for pro
Autodesk	Setup Sheet Excel 2007
Autodesk	Setup Sheet Excel
Autodesk	Setup Sheet (HTML)
Autodesk	Tool Sheet CSV
Autodesk	Tool Sheet (HTML)

Figure 33. Notice the setup sheet's file type options