

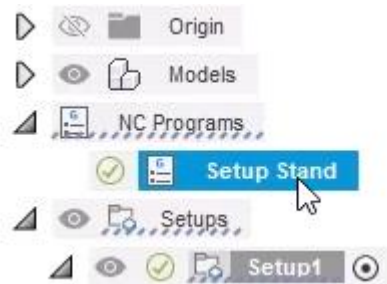
Step-by-step guide

Create an NC program

Selectively convert Fusion setups and/or operations into an NC program.

Learning objectives:

- Create an NC program.



The completed exercise

1. Continue with the file from the previous video or open the supplied *Cell Phone INCH - Simulate.f3z* file.

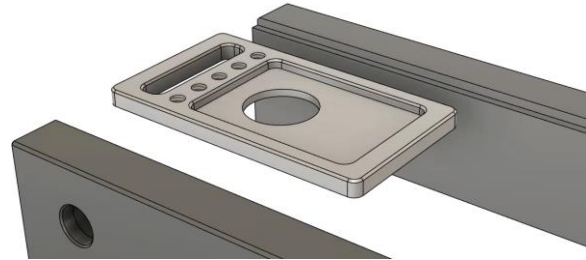


Figure 1. Open the file

2. Now that you've created all the operations needed to machine the part and have verified them in Fusion's simulation, the next step is to create an NC program. Select Setup1.

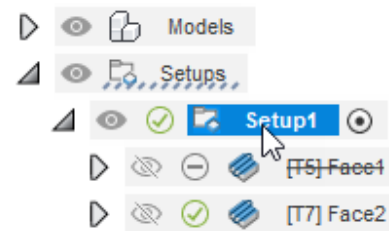


Figure 2. Select the setup

3. Click Setup> Create NC Program.

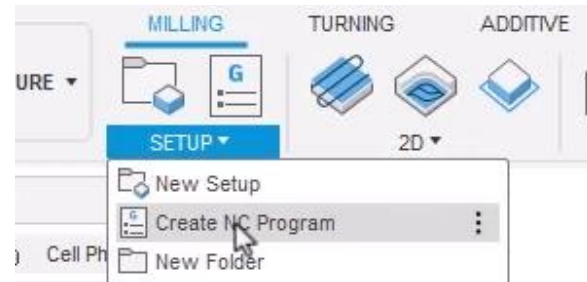


Figure 3. Create a new NC program

4. In the Manufacture workspace, you configured the information and the setup's Post Process tab. This information is automatically imported into this dialog's Program section. You could change the information here to overwrite the setup's information. Use the image on right as a guide for configuring the program's information.

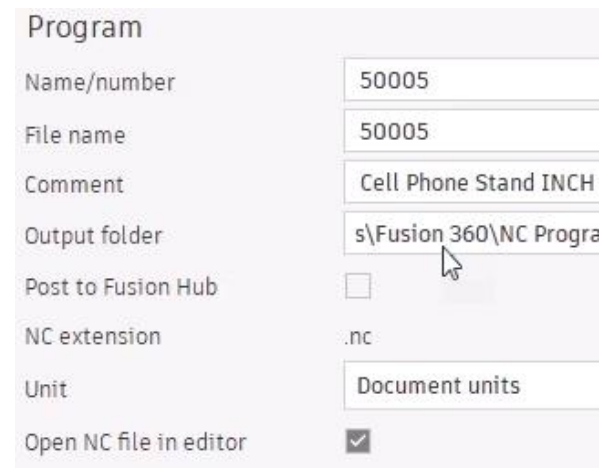
A screenshot of the 'Program' dialog box in the software. It contains the following fields and values: 'Name/number' is '50005', 'File name' is '50005', 'Comment' is 'Cell Phone Stand INCH', 'Output folder' is 's\Fusion 360\NC Progra', 'Post to Fusion Hub' is an unchecked checkbox, 'NC extension' is '.nc', 'Unit' is 'Document units', and 'Open NC file in editor' is a checked checkbox. A mouse cursor is pointing at the 'Output folder' field.

Figure 4. Verify the imported information

5. You need to choose the correct post for your specific machine so that the operations will be converted into the correct language.

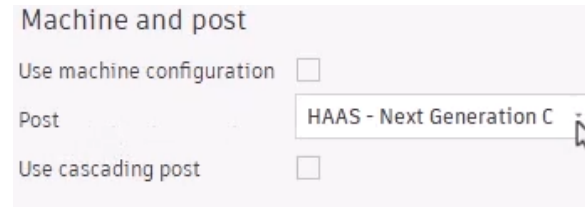


Figure 5. Select your specific machine

6. If the drop-down menu does not list your specific machine or the correct post for your machine, click the Choose from library option.

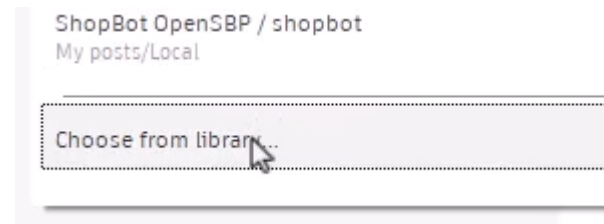


Figure 6. Search for your specific machine

7. Choose the Fusion library in the dialog's left column.

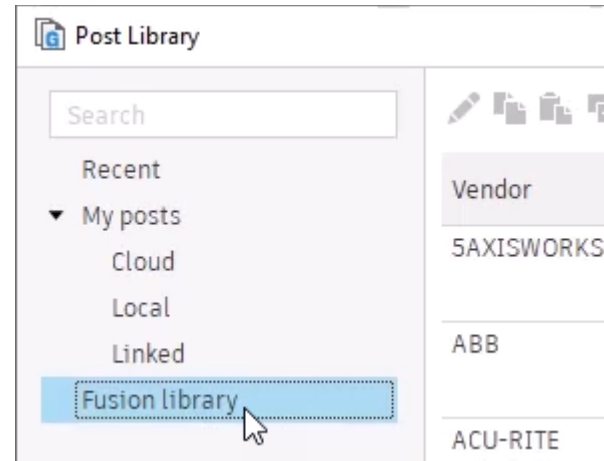


Figure 7. Choose the Fusion library

8. Activate the Milling filter, then choose your machine's vendor from the Vendor drop-down menu.

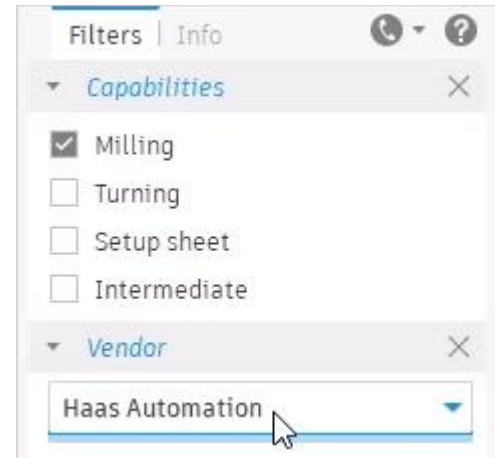


Figure 8. Filter the results

9. Select the correct post from the dialog.

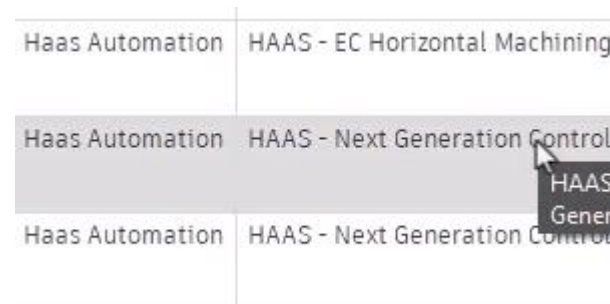


Figure 9. Select the correct post

10. Click the Post Library dialog's Select.

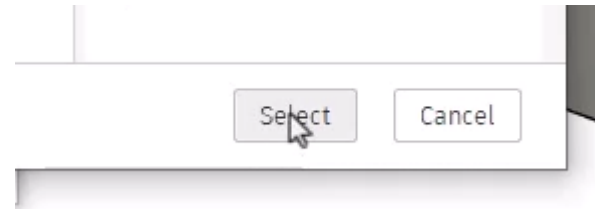


Figure 10. Click Select

11. Choose your posts' save location, then click Copy to My Posts.

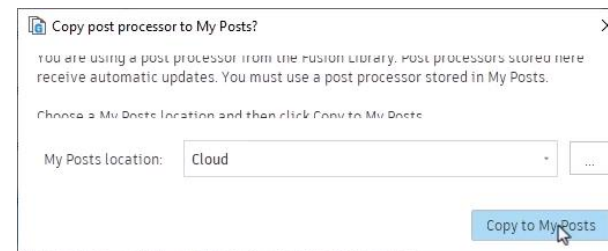


Figure 11. Copy to your posts

12. Enter **Setup Stand** into the NC program's Name box.

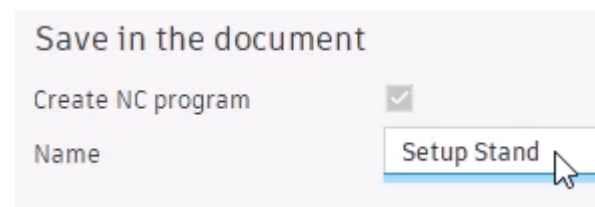


Figure 12. Name the program

13. The options in the Post properties column can be used describe your machine's capabilities. For example, you can determine whether your machine has a chip transport, whether the tools will be cycled at startup, whether pecking will be used during tapping operations, and much more. Explore all the options.

Post properties

▼ Configuration	
Use chip transport	<input type="checkbox"/>
Has A-axis rotary	No
Has B-axis rotary	No
Has C-axis rotary	No
Machine model	None
Machine has a tool setting probe arm	<input type="checkbox"/>
▼ Preferences	
Coolant pressure	Default
Fast tool change	<input type="checkbox"/>

Figure 13. Explore the Post properties options

14. Continue to the dialog's Operations tab. You can use this tab to select entire setups or individual operations that will be included in the NC program. Notice that the Face1 operation is suppressed in the left column. This suppressed file does not appear in the right column's operation list even though the operation is checked in the left column. You can choose to deselect specific operations if you want to create an NC program that omits certain operations. For example, you could choose to create a program that excludes all the operations needed to create the five tapped holes. OK the NC Program dialog.

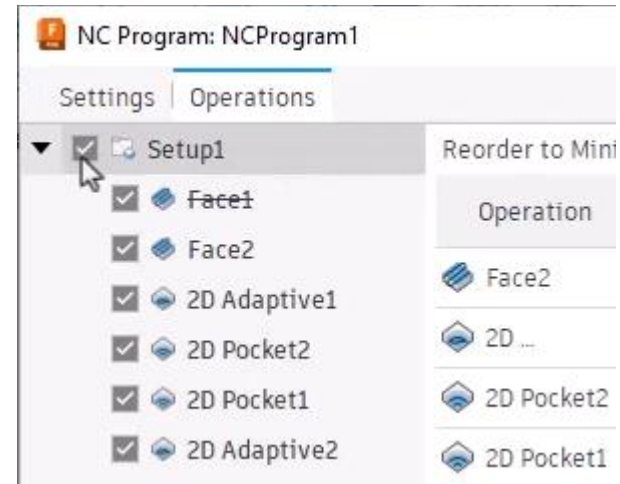


Figure 14. Choose the setups and operations you want to include

15. The Setup Stand NC program is added to the Browser's NC Programs folder. Save the file.



Figure 15. Inspect the Browser