

Step-by-step guide

Post process toolpaths

Convert an NC program to machine-readable G-Code so that a machine can cut the parts.

Learning objectives:

- Post process toolpaths.
- Review G-Code.

```
14 (Face2)
15 N25 T7 M6
16 N30 S7639 M3
17 N35 G17 G90 G94
18 N40 G54
19 N45 M8
20 N50 G1 X4.4312 Y-4.1 F650.
```

The completed exercise

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

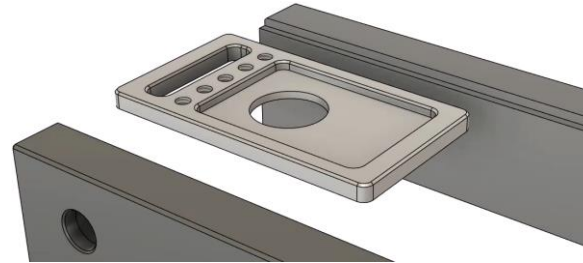


Figure 1. Open the file

2. You can convert your NC programs to machine-readable G-Code by post processing them. Right-click the Setup Stand, then choose Post Process from the menu. In the image on right, notice the NC Program needs to be regenerated. Fusion will automatically regenerate the NC program.

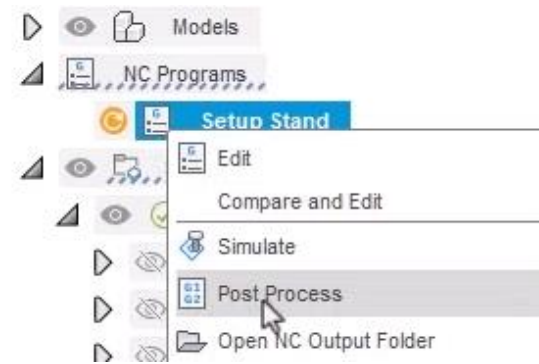


Figure 2. Post process an NC code

3. Fusion uses the selected post to convert all of the NC program's selected toolpaths to machine-readable G-Code. This G-Code will open inside an external code editor app.

```
D:\> Users > Matt Perez > Documents > Fusion 360 > NC Programs > 50005.nc
1  %
2  050005 (Cell Phone Stand INCH)
3  (Using high feed G1 F650. instead of G0.)
4  (T1 D=0.25 CR=0. TAPER=90deg - ZMIN=-0.12 - spot drill)
5  (T2 D=0.201 CR=0. TAPER=118deg - ZMIN=-0.3604 - drill)
6  (T3 D=0.25 CR=0. - ZMIN=-0.35 - right hand tap)
7  (T4 D=0.5 CR=0. TAPER=45deg - ZMIN=-0.225 - chamfer mill)
8  (T6 D=0.25 CR=0. - ZMIN=-0.3 - flat end mill)
9  (T7 D=0.5 CR=0. - ZMIN=-0.28 - flat end mill)
10 N10 G90 G94 G17
11 N15 G20
12 N20 G53 G0 Z0.
13
14 (Face2)
15 N25 T7 M6
16 N30 S7639 M3
17 N35 G17 G90 G94
18 N40 G54
19 N45 M8
```

Figure 3. Inspect the G-Code

4. The program name and number are displayed on Line 2. The comment is displayed on Line 3, and the tools are described on Lines 4 through 9.

```
1  %
2  050005 (Cell Phone Stand IN
3  (Using high feed G1 F650. i
4  (T1 D=0.25 CR=0. TAPER=90de
5  (T2 D=0.201 CR=0. TAPER=118
6  (T3 D=0.25 CR=0. - ZMIN=-0.
```

Figure 4. Notice the name, comment, and tools

5. The first operation's name is on Line 14, the tool is on Line 15, the spindle speed is on Line 16, and the WCS location is on Line 18. Explore the code to make sure everything looks appropriate, then save the file. This file can be sent to the machine that will cut the parts.

```
14 (Face2)
15 N25 T7 M6
16 N30 S7639 M3
17 N35 G17 G90 G94
18 N40 G54
19 N45 M8
20 N50 G1 X4.4312 Y-4.1 F650.
```

Figure 5. Inspect the G-Code