

How to use Carbide 3D® Carbide Create 4.62 Pro Version for Shark Pro Plus HD

Carbide 3D® Carbide Create 4.62 has a Pro option available. In the Pro version there are several post processors available.

1. Generic gcode
2. GRBL
3. Carbide 3D® Nomad® Projects
4. Carbide 3D® Shapeoko®

This document is a how to use the Generic gcode or the GRBL post processors for a Shark Pro Plus HD CNC Router. Discussed will be how to edit the two types of gcode for use with the Shark Pro Plus HD. These instructions may work with other Shark CNC Routers but I have not tested it with any other machine except the Shark Pro Plus HD system.

The Shark does not have a Computer Aided Design (CAD) program that is made by the Original Equipment Maker (OEM). You must use freeware, open source or commercial software for the Shark machines. The most common of the commercial programs are the Vetric products. You can use Fusion 360 and others but Vetric is designed for wood carving where the other commercial products are designed around designing general products and not specifically wood carving. Carbide 3D® makes Carbide Create to use with their CNC Routers and has added the post processors listed above. Carbide Create makes gcode geared toward their Nomad and Shapeoko® machines and is free for the non pro version. You can get a 2 week trial of the Pro version and at some point they will start charging for the Pro version. The pricing and other details for Carbide Create Pro have yet to be determined and will be announced by Carbide 3D® at a later date.

The following commands are the most commonly used gcode for the Shark. The propriety Shark controller does not recognize all gcode commands. There may be other commands that work but researching on forums and other sources these do work.

Shark G and M codes known to work

- G00 Fast Move
- G01 Linear interpolation (Move to)
- G02 Circular Interpolation CW (Move to)
- G03 Circular Interpolation CCW (Move to)
- G20 Set English Units

G21 Set Metric Units

G64 Cutting Mode G64 P0.01 Allows variance of 0.01 inches in path (see G20 and G21 for units)

G90 Absolute Programming of XYZ Coordinates (For Shark user setting XY&Z zero before starting job)

M03S18000 Spindle On CW (Not required and is ignored) S Sets spindle speed to 18000 RPM (Not required and is ignored)

F100.0 Feed rate 100 inches/min or Milometers (MM)/minute depending on G20 and G21 Settings

M02 Program End

The gcode file is basically made up of 3 parts of the file.

1. The preamble containing comments, setting of units, setting of system coordinates (origin), positioning of router. (G90, G20, G21, G64, G00, M03 and F for setting speed during movements) Comments are usually in parenthesis and are ignored by CNC Controller.
2. Cutting commands, router movements to cut using Feeds and Speeds (F&S), Depth of Cut (DOC), step over of cutting bits, safe retract during moves of router. These things come from the CAD program and are set by the user during job setup. (G00, G01, G02, G03 and F for setting speed during movements) The above settings are already in the gcode output.
3. Post actions to move router to safe space after cutting project and signals end of job to controller of your CNC router. (G00, G01, M02)

There are no separators for the three parts but are a continuous file that is read by the controller line by line and executed. The file is plain text and human readable. When using applications to edit the gcode file be sure to only save in the text format. Word processors may add unseen characters to the file and cause the CNC controller to not read the file or reject it. The Carbide Create applications saves gcode files with a .nc file extension. The Shark CNC Controller requires a .tap file extension to be read. The files output by Carbide Create must be edited to remove g and m commands which the Shark does not use and saved or renamed with the .tap extension.

The following files represent the output from Carbide Create for Carbide 3D® Shapeoko®, Generic Gcode and GRBL post processors. The files only contain the pre and post commands with the cutting instructions left out for clarity. The actual file is a simple v carve with a Carbide 3D® #302 60 degree vee bit. The file represents a cut out of a fairy in profile with little detail only the outline of the subject. The actual subject of the file is unimportant. The 3 main parts of the gcode files are listed below.

Carbide Create Carbide 3D® Shapeoko® Post Processor applied

```
(Design File: C:/Users/gdon_2003/Desktop/cnc_router_04032020/Projects_Templates/
keepsake_box_lid_fairy_outline/keepsake_box_lid_fairy_outline.c2d)
(stockMin:-4.000in, -2.000in, -0.250in)
(stockMax:4.000in, 2.000in, 0.000in)
(STOCK/BLOCK,8.000, 4.000, 0.250,4.000, 2.000, 0.250)
G90
G20
(Move to safe Z to avoid workholding)
G53G0Z-0.197
(ks_box_fairy_outline_Shapeoko®_pp_302)
M05
(TOOL/MILL,0.001, 0.000, 0.394, 30.00)
M6T302
M03S18000
(PREPOSITION FOR RAPID PLUNGE)
G0X-1.689Y0.627
Z0.106
```

Cutting commands not shown for clarity

```
G0X-0.717Y-0.452
G1Z-0.001F12.0
X-0.711Y-0.462Z-0.008F35.0
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M02
```

Carbide Create Carbide 3D® Generic Gcode Post Processor applied

```
G90
G20
G53Z-0.197
(ks_box_fa...)
M05
M0 ;T302
M03S18000
(PREPOSITI...)
G0X-1.689Y0.627
Z0.106
```

Cutting commands removed for clarity

```
G0X-0.717Y-0.452
G1Z-0.001F12.0
X-0.711Y-0.462Z-0.008F35.0
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M05
M02
```

Carbide Create Carbide 3D® GRBL Post Processor applied

```
G90
G20
G53Z-0.197
(ks_box_fa...)
M05
M0 ;T302
M03S18000
(PREPOSITI...)
G0X-1.689Y0.627
Z0.106
G1Z-0.001F12.0
```

Cutting commands removed for clarity

```
G0X-0.717Y-0.452
G1Z-0.001F12.0
X-0.711Y-0.462Z-0.008F35.0
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M05
M02
```

The following is how to edit the Carbide Create GRBL post processor file to work with the Shark Pro Plus HD CNC Router

G90 Comment: Leave this line in.
G20 Comment: Leave this line in.
G53Z-0.197 Comment: This line must be removed. This refers to the machine
 coordinates found during the homing of the Shapeoko® and are not
 related to the user set XYZ zero coordinates)
(ks_box_fa...) Comment: This is only a comment and can be left or removed.
M05 Comment: This line must be removed
M0 ;T302 Comment: This line must be removed
M03S18000 Comment: This line must be removed
(PREPOSITI...) Comment: This is only a comment and can be left or removed.
G0X-1.689Y0.627 Comment: The next three lines are the pre positioning of router.
Z0.106
G1Z-0.001F12.0

Cutting commands removed for clarity of example, do not edit cutting lines.

G0X-0.717Y-0.452
G1Z-0.001F12.0 Comment: Leave this line in. The next 8 lines are a
 continuation of this line.
X-0.711Y-0.462Z-0.008F35.0
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M05 Comment: Remove this line.
M02 Comment: Leave this line in. End of program.

File edited from GRBL Post Processor

```
G90
G20
(ks_box_fa...)
(PREPOSITI...)
G0X-1.689Y0.627
Z0.106
G1Z-0.001F12.0
```

Cutting commands removed for clarity of example, do not edit cutting lines.

```
G0X-0.717Y-0.452
G1Z-0.001F12.0
X-0.711Y-0.462Z-0.008F35.0
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M02
```

The following is how to edit the Carbide Create Generic Gcode post processor file to work with the Shark Pro Plus HD CNC Router

G90 Comment: Leave this line in.
G20 Comment: Leave this line in.
G53Z-0.197 Comment: This line must be removed. This refers to the machine coordinates found during the homing of the Shapeoko® and are not related to the user set XYZ zero coordinates)

(ks_box_fa...) Comment: This is only a comment and can be left or removed.
M05 Comment: This line must be removed
M0 ;T302 Comment: This line must be removed
M03S18000 Comment: This line must be removed
(PREPOSITI...) Comment: This is only a comment and can be left or removed.
G0X-1.689Y0.627 Comment: Leave this line in.
Z0.106

Cutting commands removed for clarity

G0X-0.717Y-0.452 Comment: Leave this line in.
G1Z-0.001F12.0 Comment: Leave this line in.
X-0.711Y-0.462Z-0.008F35.0 Comment: The next 8 lines are a continuation of previous line.
X-0.697Y-0.485Z-0.013
X-0.674Y-0.523Z-0.017
X-0.651Y-0.554Z-0.020
X-0.623Y-0.581Z-0.017
X-0.583Y-0.614Z-0.009
X-0.555Y-0.636Z-0.000
Z0.106
M05 Comment: This line must be removed.
M02 Comment: Leave this line in. End of program.

File edited from Generic Gcode Post Processor

```
G90  
G20  
(ks_box_fa...)  
(PREPOSITI...)  
G0X-1.689Y0.627  
Z0.106
```

Cutting commands removed for clarity

```
G0X-0.717Y-0.452  
G1Z-0.001F12.0  
X-0.711Y-0.462Z-0.008F35.0  
X-0.697Y-0.485Z-0.013  
X-0.674Y-0.523Z-0.017  
X-0.651Y-0.554Z-0.020  
X-0.623Y-0.581Z-0.017  
X-0.583Y-0.614Z-0.009  
X-0.555Y-0.636Z-0.000  
Z0.106  
M02
```