



HRS XYZ Router Post

Thank you for Downloading a **Hawk Ridge Systems** post!

Hawk Ridge Systems has decades of experience developing posts for CAMWorks and SolidWorksCAM.

Overview

This post Supports All 3 axis operations.

1. SWCAM and CAMWorks Probing supported (2020)
2. All post files must be in the same location.

Output file

The extension for this post is .TXT, changeable in the file .PINF. To edit this, open by right clicking on it and selecting "Open With" then select notepad.

Change the line "PostExtension = " to your preferred extension. Do not use a "." before it.

Posting Tab of Machine

Parameter	Value
Program Number	1
Customer Name	*-*ENTER VALUE*-*
Programmer	*-*ENTER VALUE*-*

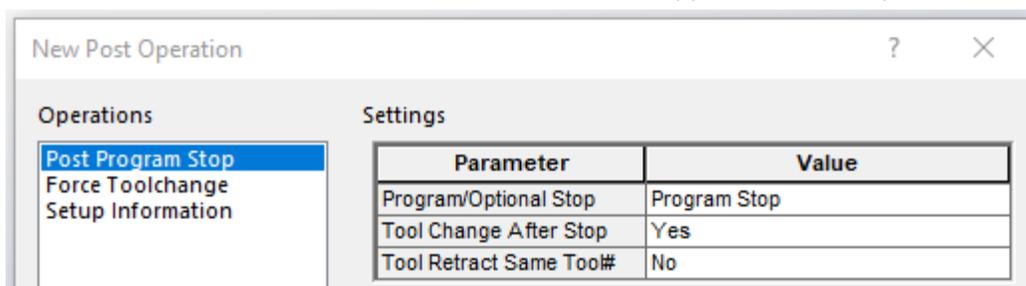
1. Program Number Type= Number, Default=**1**.
This is the Oxxxx number at the top of the program.
2. Customer Name Type=Characters, Default=*_***ENTER VALUE***_*_*
Default or blank will not be posted.
3. Programmer Type=Characters, Default=*_***ENTER VALUE***_*_*
Default or blank will not be posted.

Post Operation Insert

Right click on operations tree and select "Post Operation" from the menu

Post Program Stop

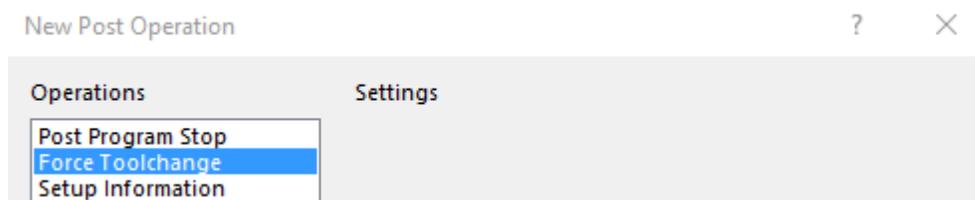
Inserts a Program or Option Stop.



1. Program/Optional Stop **Type=Select, Default=Program Stop**
Select between "Program Stop" (M00) or "Optional Stop" (M01) output.
2. Tool Change After Stop **Type=Select, Default=Yes**
Select "Yes" for a tool change after the stop (M00) or "No" for no tool change.
3. Tool Retract Same Tool# **Type=Select, Default=No**
If "Tool Change After Stop" = "No" then you can output a Z retract or not. If "Tool Change After Stop" = "Yes" then the post will automatically output a Z retract home.

Force Tool Change

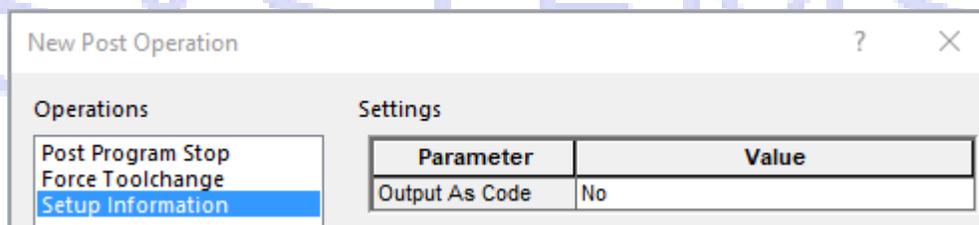
Forces the post to do a tool change. Normally used between two operations using the same tool number.



1. No settings. Select and press the Insert button then close.

Setup Information

Will insert setup information (as a comment) or Code (as-typed) in the posted code going to the machine.



1. Output As Code **Type=Select, Default=No**
 - a. Depending on the Output as code selection you will either get comments or code.



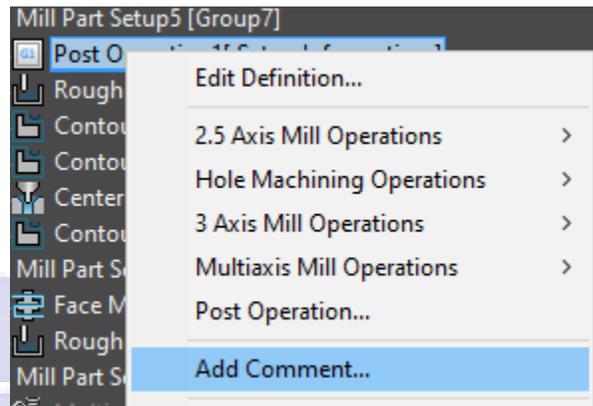
HAWK RIDGE SYSTEMS

Worldwide Post Processor Support and Development

- b. In the operations tree Find the newly created post operation. Normally called 'Post OperationX[Setup Information]'.

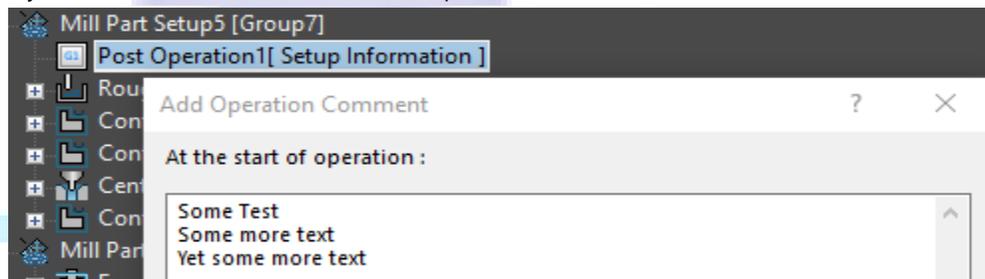
The 'X' represents the number after the post operation.

- c. Right click on 'Post OperationX[Setup Information]' and select 'Add Comment'



- d. In the top box in the add operation window.

Enter your comments or code. For a new line press CTRL + Enter.



How it works:

When you post the system will output the comments or code you entered. This can go anywhere in the tree but most of the time it is at the top before the first operation in a setup.

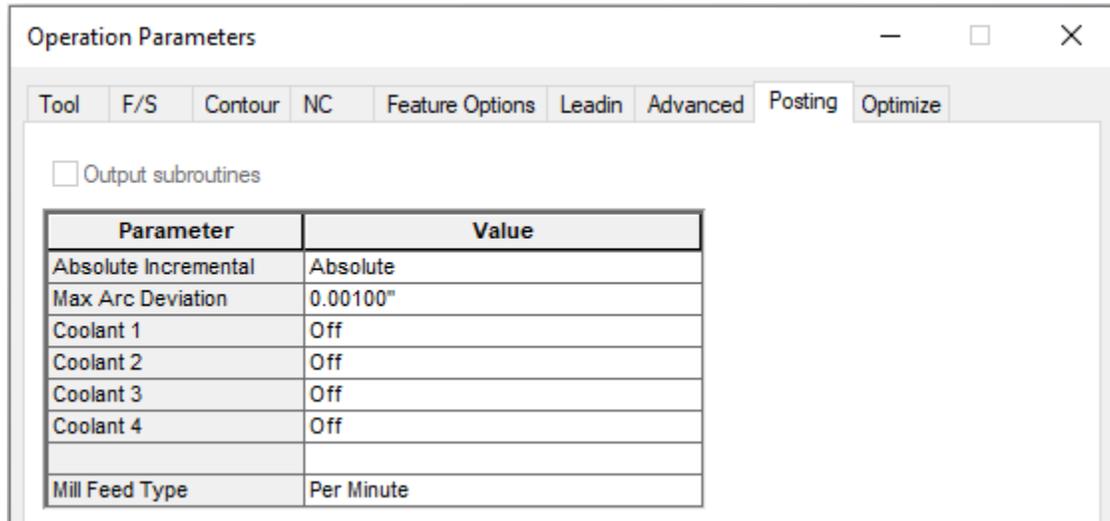
Note: If you want to insert comments to an operation in CAMWorks start at letter 'c' above. Right click on the operation instead of the post operation.

Operation Post Tab

Double click on an operation or RBM (right button mouse) on the operation and select 'Edit Definition'

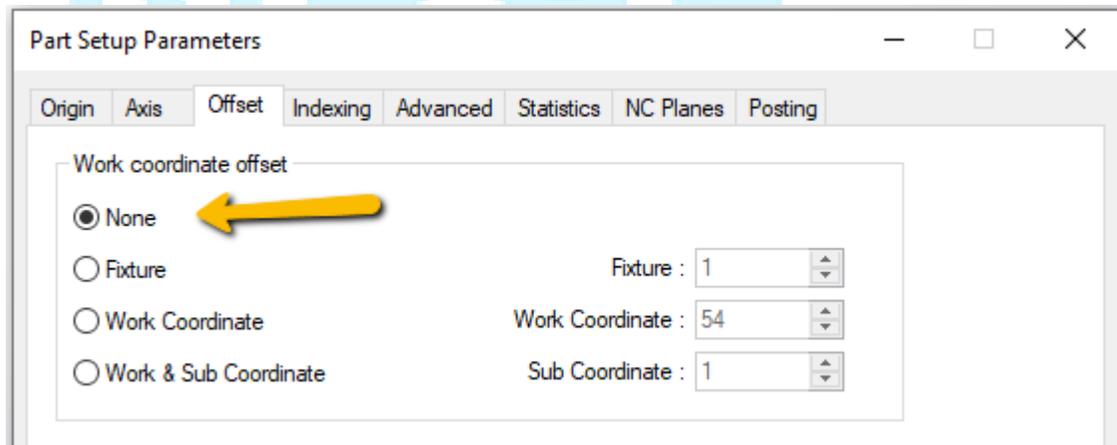


Mill Operation

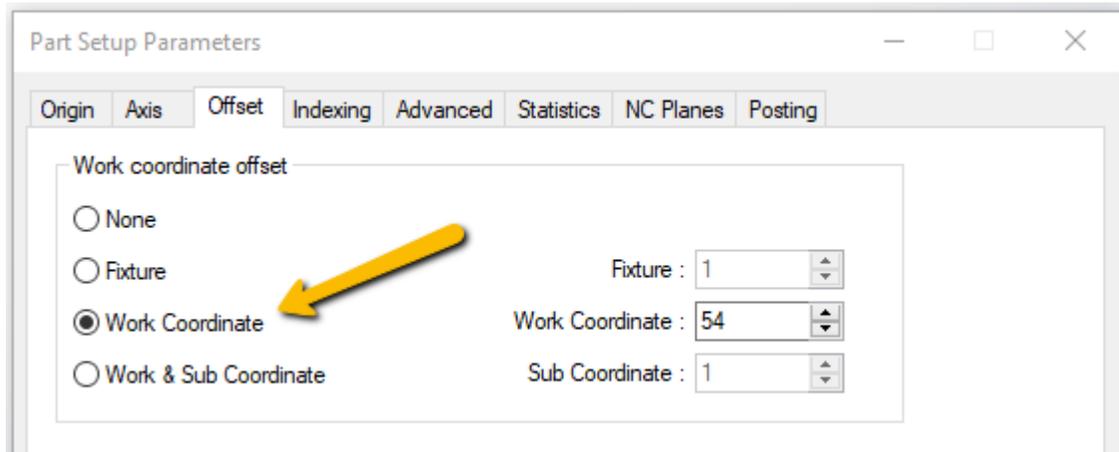


1. Absolute Incremental Type=Select, Default=**Absolute**
All Z moves are Absolute regardless of selection.
2. Max Arc Deviation Type=Decimal, Default=**.001"**
If the Arc needs to be broken into line segments this is the deviation for the breaking of the arc.
3. *Coolant 1 Type=Select, Default=**Off**
4. *Coolant 2~4 Type=Select, Default=**Off**
5. Mill Feed Type Type=Select, Default=**Per Minute**
Feed of per minute (IPM/MMPM) or per revolution (IPR/MMPR)

Part Setup Parameters



When Offset is set to "None" the post will not output G54-59(Work coordinate offset), G43(Tool length compensation), G49(Tool length compensation cancel) or H(Length offset tool value).



When Offset is set to "Work Coordinate" G54-59(Work coordinate offset), G43(Tool length compensation), G49(Tool length compensation cancel) and H(Length offset tool value) will be output in normal fashion. See below.

```

(DRILL1)
(F SCREW MACH DRILL)
( TOOL ID      :F CNC DRILL )
N5 G0 G91 G28 X0 Y0 Z0
N10 G0 G17 G20 G40 G80 G49
N15 M6 T2
N20 M3 S7000
N25 G0 G90 G54 X0.3750 Y4.6250
N30 G43 Z1.0000 H2
N35 G83 G98 X0.3750 Y4.6250 R0.1 Z-0.8032 Q0.05 F18.1234
N40 Y0.3750
N45 X5.6750
N50 Y4.6250
N55 G80 Z1.0000
N60 G0 G91 G28 Z0
N65 G49 H0
N70 G0 G91 G28 X0 Y0
N75 M30
    
```



External Configuration

Warning: This needs to be edited before posting or your output may not run your machine.

Externally configurable post processor output options are found in the file '.CNF'. These are self-explanatory in nature.

General output to the machine will define items and how you prefer to see them in the code.

CNF file contents

```

*-----
* The settings in this file will override the settings of the post processor.
* Lines started with an asterisk (*) are comments and are ignored.
* This file will only affect the post processor (*.CTL file) with the same name.
* Not all post processors support CNF files.
*-----
*           -*-*- General output to the machine -*-*-
*-----
*   * CNF_TLIST_CNFG=0 then no tool list will be present.
*   *
*   * CNF_TLIST_CNFG=1 then a tool list will be present.
*   *
CNF_TLIST_CNFG=1
*-----
*   * CNF_LINE_NUM_CNFG=0 then no line number.
*   *
*   * CNF_LINE_NUM_CNFG=1 then line numbers on every line.
*   *
*   * CNF_LINE_NUM_CNFG=2 then Line Numbers at Tool change by tool #. (N7,N10,N1).
*   *
*   * CNF_LINE_NUM_CNFG=3 then Line Numbers at Tool change incrementally. (N1,N2,N3).
*   *
CNF_LINE_NUM_CNFG=1
*-----
*   * CNF_TC_POS_CNFG=1 then G53 Z at operation end and G53 Z then Y at program end.
*   *
*   * CNF_TC_POS_CNFG=2 then G91 G28 Z0 at operation end and G91 G28 Z0 then
*   * G91 G28 X0 Y0 at program end.
*   *
*   * CNF_TC_POS_CNFG=3 then nothing
*   *
CNF_TC_POS_CNFG=2
*-----

```



HAWK RIDGE SYSTEMS

Worldwide Post Processor Support and Development

* If above (TC_POSITION_CNFG) equals 1

* G53 End of Tape Y position

CNF_Y_EOT_POS=0.0

*

* G53 Z retract position

CNF_Z_RET_POS=0.0

*-----

* Determines if the drill X and Y positions need to be in the cycle line.

*

* CNF_DRILL_XY_CNFG = 0 - X&Y's are not in the cycle line.

* CNF_DRILL_XY_CNFG = 1 - X&Y's are in the cycle line.

*

CNF_DRILL_XY_CNFG=1

*-----

End User Agreement

The software/macro/customization is supplied "**as is**" and all use is at your own risk. *Hawk Ridge Systems* disclaims all warranties of any kind, either express or implied, as to the features and functionality. Neither this agreement nor any documentation furnished under it is intended to neither express or imply any warranty that the operation of the software will be error-free nor cause undesirable results on files that it is ran against. Under no circumstances shall *Hawk Ridge Systems* be liable to any user for direct, indirect, incidental, consequential, special, or exemplary damages, arising from or relating to this agreement, the software, or users use or misuse of the software or any other services provided by *Hawk Ridge Systems*. Such limitation of liability shall apply whether the damages arise from the use or misuse of the software or any other services supplied by *Hawk Ridge Systems* (including such damages incurred by third parties), or errors of the software. Use of the software/macro/customization constitutes acceptance to this agreement.