

Postability Post Processor Documentation



Rev. Date September 1, 2020



Contents

- Application of this Document 3
 - General Terminology..... 3
 - 5-Axis Mill/Router Machines..... 3
 - Mill-Turn Machines 3
- Unified Post Kernel (UPK) General Settings..... 4
 - Rotary Switches..... 4
 - Machine Orientation and Offset 6
 - Advanced Control Options 8
 - Home Motion 9
 - Misc Control Options 10
 - 5-axis switches/variables 11
 - Switches/variables for Mill-turn posts..... 12
- Miscellaneous Values..... 13
 - General Notes 13
 - Milling 13
 - Miscellaneous Integers 13
 - Miscellaneous Reals..... 14
 - Lathe (Mill-Turn) 16
 - Miscellaneous Integers 16
 - Miscellaneous Reals..... 16
 - Work Offset, Work Coordinate System (WCS), Tool/Construction Planes..... 17
- Rotary Axis 18
 - 3+2 Machining..... 18
 - Rotary Axis Control (Milling Operations) 18
 - Rotary Axis Control (Mill-Turn Operations) 19
 - Multi-Axis 20
 - Transitions between cuts..... 21
- Machine Setup Information..... 22
 - Machine and Control Definition..... 22
 - Rotary Axis 23
 - General Machine Parameters 24
 - Axis Combinations..... 25
 - Arcs..... 26



Application of this Document

This document contains generic information about our Unified Post Kernel (UPK), which applies to mill and mill-turn applications. Additional documentation for specific machine/control combinations is available and can be sent to you. For information regarding our general post setup please see our installation documents which can be found [here](#) under the Installation heading, as well if you have any questions or comments do not hesitate to contact us through [our email](#) listed on our website.

The first two sections, UPK General Settings, and Miscellaneous Values, contain post setup information applicable to many post functions. These sections will reference other, more detailed, sections for further application.

All remaining sections are grouped under common post functions, which will reference back to the post switches and miscellaneous values shown in the first two sections.

The post will come with default settings for each section however these should be changed as necessary to best suit your application

General Terminology

5-Axis Mill/Router Machines

5-Axis machines are made up of three linear axis and two rotary axes. In the UPK two rotary axis are designated as either **rot** (rotary) or **tilt**. The rot axis refers to the primary rotary axis which typically has the most travel. The tilt axis refers to the secondary rotary axis which generally has less travel than the primary rotary axis. An example configuration would be a vertical milling machine with a rotary axis about the Z-axis, labelled C, and another rotary axis about the X-axis, labelled A. In this case the C-axis is considered the rot axis and the A-axis is considered the tilt axis.

Mill-Turn Machines

Mill-Turn machines come in many configurations. For machines with milling capabilities the main and sub spindle are considered the rot (rotary) axis depending on which axis combination is being used for a given operation. If the machine has a rotary axis in the head, typically a B-axis, this axis is considered the tilt axis.





Unified Post Kernel (UPK) General Settings

Rotary Switches

userotlock	0 = Do not output locking M codes 1 = Output locking M codes 2 = Rot&Tilt
userotbrake	0 = Do not output braking M codes 1 = Output braking M codes
usetiltlock	0 = Do not output locking M codes 1 = Output locking M codes
usetiltbrake	0 = Do not output braking M codes 1 = Output braking M codes
lockcomm	0 = Do not output comments 1 = Output comments

These variables are used to control the output of M-codes to lock, brake and unlock the rotary axes if the machine is equipped with this functionality. The post will automatically determine when each of the lock states should be used based on the operation being programmed. Override variables are available to allow the user to force the state of the lock codes. For further details see [Misc Int 7](#) and [Misc Int 8](#)

one_revr	Limit 3+2 rotary positioning? (0 = No, 1 = 0 to 360, 2 = -180 to 180)
one_revtilt	Limit 3+2 tilt positioning? (0 = No, 1 = 0 to 360, 2 = -180 to 180)

The post supports indexing to within certain ranges to keep indexing distances to a minimum and to help make the code output easy to read. Note the selected ranges above will only apply for 3+2 operations or the start of other operation types where the rotary motion is deemed to make sense and can be made safely.

frdegstp	Step limit for rotary feed in deg/min
-----------------	---------------------------------------

Post calculated rotary feed rates will only output if there is a change in calculated feed rate by an increment greater than this number.



rot_feed Use calculated rotary feed values, (**0** = no, **1** = yes)

Setting the value to 1 will have the post calculate and output rotary feed rates as either inverse time or deg/min based on the control definition settings. If set to 0, the post will ignore any control definition settings and avoid calculated rotary feed rates. Instead of outputting the calculated values, the post will simply output the feed rate as programmed in the operation. As a note, most modern machines that have TCPM functionality do not require the post to calculate the rotary feed rates.

maxincrot Maximum incremental rotary motion before unwind or solution flip
maxintilt Maximum incremental tilt motion before unwind or solution flip

These values represent the largest rotary move that can occur in one line of code. Motion that exceeds these limits will result in an unwind or solution flip depending on the specific scenario. Large rotary moves can result in poor cut result and potentially collisions depending on the conditions of the cut. Generally, these values should not exceed 170 degrees.

For further details see [Multi-Axis](#)

use_rotmcode Output M-Code for Axis direction (sindx_mcr)
0 = Do not output M code,
1 = Output M code (Only valid when rot_type = 1 instead of sign indicating direction),

use_tiltmcode Output M-Code for Axis direction (sindx_mct)
0 = Do not output M code,
1 = Output M code (Only valid when rot_type = 1 instead of sign indicating direction),

Some machines require an M-code to indicate the direction of rotation (CW or CCW). Note that the corresponding rotary axis must be set to Shortest Distance in the control definition in order for this variable to take effect.





Machine Orientation and Offset

workofs_out -1 = Use output from Mastercam
 0 or any number = Always use this work offset number (e.g. 0 gives G54, 6 gives G54.1 P1)
 -2 = Use 1st offset for main spindle and 2nd offset for sub spindle (mill-turn),

Setting to 0 or any number greater than 0 will result in that work offset being used for all operations regardless of the operation setting. How this value is mapped (eg 0=G54,1=G55,etc) is dependent on how the post has been configured. Setting the value to -1 will use Mastercam values as programmed. Setting to -2 applies only for Mill-Turn machines and will apply the first offset (0) to the main spindle and the second offset (1) to the sub spindle.

wcstype **0** = G92 at start
 1 = G92 at toolchanges
 2 = G54
 3 = Off

Controls the style of wcs output. Setting to 0 will output a G92 style shift at the beginning of the program only, 1 will output a G92 shift with each tool change, 2 will output the traditional G54 style offsets and 3 will remove the output of offsets altogether.

force_wcs Force WCS output at every toolchange, (**0** = no, **1** = yes)

Forces the output of work offsets during every tool change. Note work offsets are automatically output at the start of file, after a retract to the home position or a change in work offsets has been detected.

shiftlocation **0** = Apply tool plane-based shifts in world coordinates before rotation of plane
 1 = Apply with rotation of plane
 2 = Apply tool plane-based shifts in plane coordinates after rotation of plane

Setting this variable controls the location of the plane coordinate shift based on the difference in the work offset origin and the tool plane origin for a given operation. The value should only be set to 1 if the control has a tilted plane function that has been enabled in the post (see **tiltplane**).

ignoreorg Ignore toolplane origin, use WCS origin (**0** = Off, **1** = On)

Setting this value to 1 omits the difference between the tool plane and work offset plane origin and outputs all code relative to the work offset origin. If enabled, the value of **shiftlocation** is irrelevant.

See [Work Offset, Work Coordinate System \(WCS\), T/C Planes](#)



orgshift Shift origin from Mastercam's programmed origin (**0** = Off, **1** = On)

Used to apply a manual shift to coordinates. Users are typically prompted to enter X, Y and Z shifts when enabled. Coordinates are then shifted in the post by the specified amount prior to output.

See [Machine Setup Information](#)



Advanced Control Options

tcp Enable tool center point management (**0** = Off, **1** = On, **2**=Table fixed coordinate system)

Enable if the machine is equipped with TCPM functionality. If in use, no additional transformation is needed for the coordinates output. If not used, the post will need to compensate for the machine's kinematics and rotary motion.

tiltplane Enable tilted work planes
0 = Off
1 = On
2 = On, not used for Z/X tool axes
3 = On, not used for Z tool axis

Determines whether the post will output the tilted plane functionality and map the coordinates accordingly. If set to 2, the tilt plane will only output if the original tool vector is not along the X or Z axis. Setting to 3 will not output if the original tool vector is along the Z-axis.

tiltcomb Tilted work plane combination, only applicable when tiltplane active
(**0** = AC, **1** = BC)

Tilt plane functions are often controlled using spatial angles rather than physical angles. This switch selects which spatial angles are used in tilted planes.

vector **0** = Physical rotary/tilt axis output
1 = Vector output (5-axis only)

Controls whether 5-axis motion is output as vectors or physical angles.





Home Motion

use_home **0** = Use home positions defined by “**x_home**”
 1 = Use home positions from Mastercam

If enabled the home positions will be pulled from the Mastercam operation. If disabled the home positions are set in the post using the values from **x_home_in**, **y_home_in**, **z_home_in**, **x_home_mm**, **y_home_mm**, and **z_home_mm**.

home_style **0** = G90 G53
 1 = G91 G28
 2 = G91 G30
 3 = G90

Set to control the style of the home position call. Note that the values available will vary based on the controller.

use_g_inc **0** = Use new addresses for absolute/incremental output
 1 = Use g codes for absolute/incremental output (G90/G91 default)

Controls whether the post will output a G-code to represent absolute/incremental coordinate output or using a new axis prefix.

strotinc String for incremental output if **use_g_inc = 0**
strtiltinc String for incremental output if **use_g_inc = 0**
strxinc String for incremental output if **use_g_inc = 0**
stryinc String for incremental output if **use_g_inc = 0**
strzinc String for incremental output if **use_g_inc = 0**

These strings are used on machines to control the incremental motion of the specified axis where supported.

x_home_mm : **0.** #X home position – metric (mm)
y_home_mm : **0.** #Y home position – metric (mm)
z_home_mm : **0.** #Z home position – metric (mm)
x_home_in : **0.** #X home position – inch
y_home_in : **0.** #Y home position – inch
z_home_in : **0.** #Z home position – inch

X, Y, and Z-axis home position for use when **use_home** has been set to 0. Note that metric Mastercam files will use the **mm** values and inch Mastercam files will use the **in** values.





Misc Control Options

hel_lead Special helical move treatment with lead, (**0** = no, **1** = yes)

Set to 1 to output helical motion using the lead/pitch.

force_dpts Force XY output on all drilling lines including cycle call? (**0** = no, **1** = yes)

Force the output of drill coordinates for each point in a drilling operation.

tool_table Tool table, (**0** = no, **1** = yes)

Enables the output of the tool table in the header of the program.

use_pitch **0** = Use feed for tapping
 1 = Use pitch for tapping

Controls the output of the feed in tapping canned cycles. When set to 0 the programmed feed will output, when set 1 the pitch of the tap will be used.

rigid_tap **0** = Floating tap output
 1 = Rigid tap output (suppress spindle output and output M29)

Enables the use of rigid tapping if enabled. Depending on the control requirements, the spindle speed and spindle rotation commands may be suppressed.

use_gear Output gear selection code, (**0** = no, **1** = yes)

Enable to output M-codes for gears dependent on the spindle RPM.

ixtol Tolerance in deg. for index error

This value is the tolerance checked against for rotary axis that are only able to index. Any calculated value for a rotary axis limited to indexing that deviates by more than this value will cause an error to be generated. Note the rotary axis in the machine definition needs to be set to position at fixed increments only, and the index increment also needs to be set.

tseqno Output sequence number at toolchanges
 0 = off
 1 = seq numbers match toolchange number
 2 = seq numbers match tool number

When left off, the sequence numbers will output as set in the control definition (either on each line or entirely off). If set to 1, a single sequence number will be output on tool changes, with the number matching the tool change count, if set to 2 the sequence number will instead match the tool number.

force_output **0** = Modal G code output at toolchanges
 1 = Force output of all G codes

Controls whether select modal G-codes are forced out on each tool change. Typical codes forced out are for motion, interpolation plane, absolute/incremental, feed rates and feed rate type.



5-axis switches/variables

pivotdis Distance from center of tilting axis to gauge line/collet line – mm

This is the distance from the collet/gauge line to the center of the tilting axis and is only relevant for machines where the tilt axis is located in the Head and the post needs to compensate for the rotary axis (postcomp = 1).

cleardis Retract clearance along tool vector in 5-axis unwinds – mm

When reaching a limit or large rotary motion and an unwind is required, some operations do not provide a clearance distance for the post to use to retract away from the part. This value is used in the cases a clearance value is not available.

offsetdis Distance from spindle center to center of tilting axis – mm

The perpendicular distance from the centerline of the spindle to the center of the tilting axis. This value only applies on machines with one rotary axis in the table and the other in the head and is only necessary if the post needs to compensate for rotary motion (postcomp = 1).

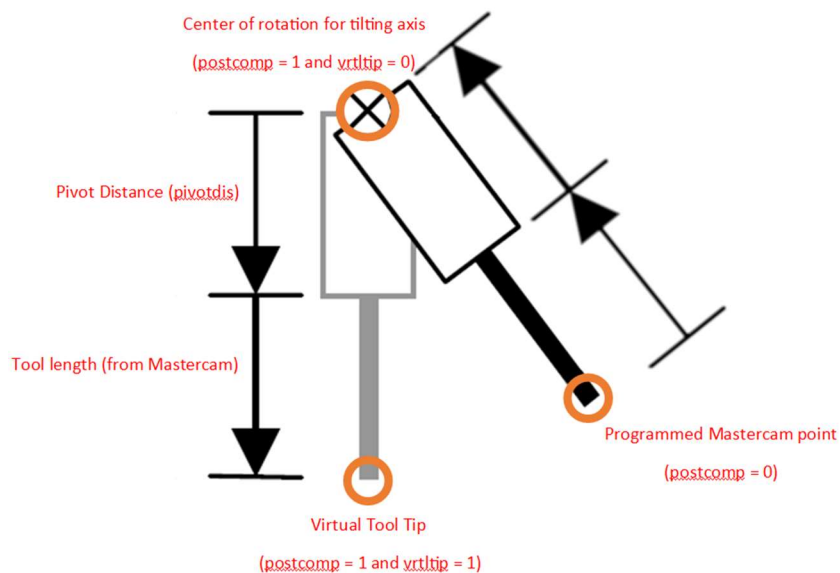
postcomp Compensate for tool length in post (0 = no, 1 = yes)

When enabled, the post will shift coordinates to compensate for the rotary axis position. This is applicable for machines with the tilting rotary axis in the head.

vrlltip Control point is the virtual tool tip (0 = no, 1 = yes)

Applicable only when post compensation is required (postcomp = 1). When enabled, the post will shift coordinates to the virtual tool tip, or the position the tool would be in if the tilting axis was at the zero position.

Below is a diagram to further illustrate the application of the post based compensation switches:





Switches/variables for Mill-turn posts

vtl **0** = Standard mill-turn programming with Top WCS
1 = VTL mode with Lathe
Z = World Z WCS

yturn **0** = No Y output for turning
1 = Y output for turning

Enabling this variable will output a Y0 value at the start of each turning operation.

xmult0 X axis multiplier for cuttype 0 #Toolplane Positioning
xmult1 X axis multiplier for cuttype 1 #Axis Substitution
xmult2 X axis multiplier for cuttype 2 #Polar Conversion
xmult3 X axis multiplier for cuttype 3 #Simultaneous 4/5 axis
xmult4 X axis multiplier for cuttype 4 #Turning
xmult5 X axis multiplier for cuttype 5 #Face Milling Interpolation on Control
xmult6 X axis multiplier for cuttype 6 #OD Rotary Interpolation on Control

These values are the X-axis multipliers for the different cut types and control whether the X-axis values are output as diameter or radius values.

old_new_sw Switch old (6T), new (0T+) lathe canned cycle formats, (**0** = old, **1** = new)

The post has been setup to support multiple standards of lathe canned cycles, typically seen in Fanuc based controllers. This setting controls which format of canned cycle is output.

thdaddress **0** = Use F address for threading
1 = use E address for threading

Controls the prefix for the feed output for threading operations.

css_start_rpm **0** = Start with CSS directly
1 = Start with RPM then change to CSS

When enabled the post will output an RPM speed based on the programmed CSS and the current X-axis position prior to switching to CSS output. In some posts this switch will also control the conversion back to RPM at the end of operations using CSS.

linneglim Negative axis stroke limit – mm

Set to the negative stroke limit for the X or Y based on the configuration of the machine as indicated through the linstroke setting. The post will prompt the user with an error if the selected crosses this limit.

linstroke Solution selection (mi4) based on stroke limit
0 = Do not watch for linear axis stroke overtravel
1 = Watch X axis stroke and change solutions accordingly
2 = Watch Y axis stroke and change solutions accordingly (Only for tilt about X)

Select which axis to monitor for stroke overtravel based on the limit set in linneglim.





Miscellaneous Values

General Notes

Below are the default miscellaneous values found in the UPK. These values and their use will vary depending on the implementation of your post. Some options are added to further customize your post, while other values may be removed if the functionality is not necessary/available for your specific machine/controller combination. Please contact our support team at team@postability.com if you have questions about the miscellaneous values setup in your post.

Milling

Integers		Reals	
	<input type="text" value="0"/>	HS [1=G08,2=AICC,3=HPCC]	<input type="text" value="0.0"/>
Abs/Inc, top level [0=Abs,1=Inc]	<input type="text" value="0"/>	Accel/Decel Value [0=No Output]	<input type="text" value="0.0"/>
3D Comp (Surface Normal) [0=No,1=Yes]	<input type="text" value="0"/>		<input type="text" value="0.0"/>
Start Solution [0=-Tilt,1=+Tilt]	<input type="text" value="0"/>		<input type="text" value="0.0"/>
	<input type="text" value="0"/>		<input type="text" value="0.0"/>
Unit/Rev Feed [0=No,1=Yes]	<input type="text" value="0"/>		<input type="text" value="0.0"/>
Rotary [0=Post,1=Lock,2=Brake,3=Off]	<input type="text" value="0"/>	Start Op Ret [0=Axis,1=Z,2=Z/XY,3=No]	<input type="text" value="1.0"/>
Tilt [0=Post,1=Lock,2=Brake,3=Off]	<input type="text" value="0"/>	Mid Op Ret [0=Axis,1=Z,2=Z/XY,3=No]	<input type="text" value="1.0"/>
	<input type="text" value="0"/>	Rot Start [1=Value,2=Last,3=Rev,4=Prompt]	<input type="text" value="0.0"/>
M00 Before Operation [0=No,1=Yes]	<input type="text" value="0"/>	Rot Start [Enter Value or Rev]	<input type="text" value="0.0"/>

Automatically set to post values when posting

Miscellaneous Integers

- Misc Int 2** This controls absolute or incremental output in the main program, also see the [use_g_inc](#) switch
- Misc Int 3** Output surface normal vectors for 3D tool wear compensation. For use with 5-axis toolpaths only. Ball nose tools are recommended.
- Misc Int 4** If your machine's configuration allows, you can index the tilt and rotary axes to either of 2 solutions to achieve the current programmed angle of attack on the part. The post will use the selected solution as a starting point but will change to the other solution to avoid large rotary moves and overtravel.



Misc Int 5 (Mill-turn Only) In some situations the user may wish to run multi-axis tool paths on the machine without the use of the TCP function. Details such as feed rate mode, programming point, tool length compensation method, coordinate output may all change with this selection. Be sure to consult with us and your machine tool dealer before attempting this.

Misc Int 6 Change from units per minute to units per rev feed rate output. Leave your programmed feed rate as the standard units/min, the post will convert for you based on the programmed spindle speed.

Misc Int 7 Selection of rotary axis lock/brake code output.
0 = the post will make an automatic selection based on the cutting type
1 = force lock the rotary axis
2 = turn the rotary axis brake on
3 = turn off rotary locking

See the post switches, [userotlock](#) and [userotbrake](#).

Misc Int 8 Selection of tilt axis lock/brake code output.
0 = automatic selection by post based on the cutting type
1 = force lock the tilt axis
2 = turn the tilt axis brake on
3 = turn off rotary locking

See the post switches, [usetiltlock](#) and [usetiltbrake](#).

Misc Int 9 (Mill-turn Only) Selection of face or diameter rotary interpolation functions. Examples of these functions include G12.1 and G07.1.

Misc Int 10 This selection is to force an output of a program stop (M00) after the current tool path. If left as 0, the post will output an optional stop (M01) by default.

Miscellaneous Reals

Misc Real 1 High speed machining method selection. If available, please check for methods particular to your machine/control in the add-on manual.
1 = G08 P1 - Advanced look ahead
2 = G05.1 Q1 R - Artificial intelligence contour control
3 = G05 P10000 R - High precision contour control

Misc Real 2 Control of the R coefficient listed in **Misc Real 1**.





Misc Real 7

Mastercam goes through what is known as a null tool change in between two operations using the same tool and also between steps of a transform tool path. Use this selection to manipulate the retract motion before the current operation.

0 = Along the current tool axis if supported by your control, if not the post will change this selection to a 1.

1 = Retract along a single axis. For most machine configurations this will be Z, for some it will be X.

2 = Retract along a single axis (same as selection 1), but then move to home in the other two axes.

3 = Do not retract. This selection will keep the tool at the clearance height.

Misc Real 8

Mastercam goes through what is known as a null tool change in between multiple chains, depth cuts, multi passes, etc. during the middle of a single operation. Use this selection to manipulate the retract motion during the current operation.

0 = Along the current tool axis if supported by your control, if not the post will change this selection to a 1.

1 = Retract along a single axis. For most machine configurations this will be Z, for some it will be X.

2 = Retract along a single axis (same as selection 1), but then move to home in the other two axes.

3 = Do not retract. This selection will keep the tool at the clearance height.

Misc Real 9

Use this selection to manipulate the rotary (rot) axis position at the start of the selected operation. See also [Rotary Axis](#).

1 = Value - Enter the value of the rotary axis. This is a starting value for multi-axis paths and permanent value during a 3+2 tool path. Note the value will be used at the start of every null tool change (new chain, depth cuts, etc).

2 = Last - Look at the rotary axis value of the previous operation and index the rotary axis as close as possible to that location.

3 = Rev - Enter the number of revolutions for the rotary axis to start at.

4 = Prompt - Have the post prompt the user upon posting for a rotary axis value. Note the post will prompt at the start of every null tool change (new chain, depth cuts, etc).

Misc Real 10

Works with **Misc real 9**. Enter the physical value (1) or the rev number (2) for the rotary output.





Lathe (Mill-Turn)

Integers		Reals	
Spindle Synch [0=Off,1=On]	0		0.0
Abs/Inc, top level [0=Abs,1=Inc]	0		0.0
	0		0.0
	0		0.0
	0		0.0
	0		0.0
	0		0.0
	0		0.0
	0		0.0
M00 Before Operation [0=No,1=Yes]	0	Start Op Retract [0=Axis,1=XY,2=XY/Z,3=No]	1.0
		Mid Op Retract [0=Axis,1=XY,2=XY/Z,3=No]	1.0
			0.0
			0.0

Automatically set to post values when posting

Set to post values Set to Op Defaults

Miscellaneous Integers

Misc Int 1 This value is used to enable the synchronization of the main and sub spindle on machines with two spindles.

Misc Int 2 See [Misc Int 2](#) for milling

Misc Int 10 See [Misc Int 10](#) for milling

Miscellaneous Reals

Misc Real 7 See [Misc Real 7](#) for milling

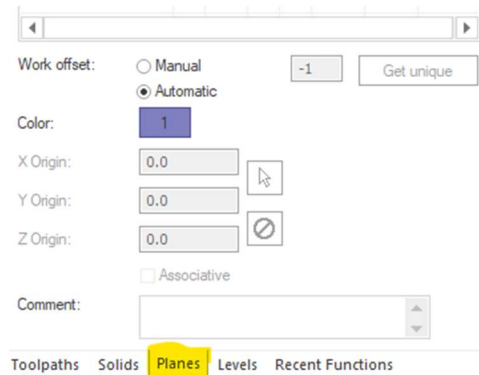
Misc Real 8 See [Misc Real 8](#) for milling



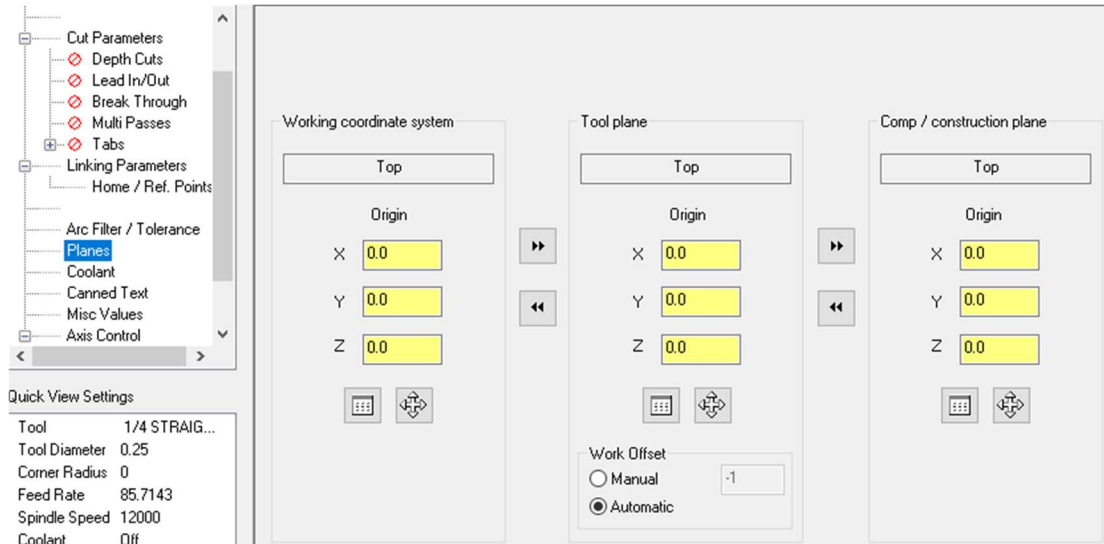
Work Offset, Work Coordinate System (WCS), Tool/Construction Planes

If the tool plane origin differs from the working coordinate system (WCS) origin, the post will output a shift command, either before, with, or after a tilt plane/tcp call based on the value of [shiftlocation](#). If you wish to have the post ignore any difference between the tool plane and WCS origin [ignoreorg](#) can be set to 1.

If you want to change work offsets on the machine, adjust your WCS origin in the Planes manager. The easiest way to do this is to take a copy of the plane you want to use, then shift the origin to the position you wish to use on the machine, then select this as your WCS for the applicable operations.



If you just want a new local origin (shift) inside the same work offset, adjust your Tool/Construction plane origin for that tool path. This is often used in situations where you want to make the code easier to be modified or read by the operator.



If using work offset values from Mastercam, be sure to set the work offset for each operation. If you leave it as -1, Mastercam will automatically increment the number with each new plane and you will get several work offset values output. If you do not wish to use Mastercam's work offset values, see the [workofs_out](#) switch for options.



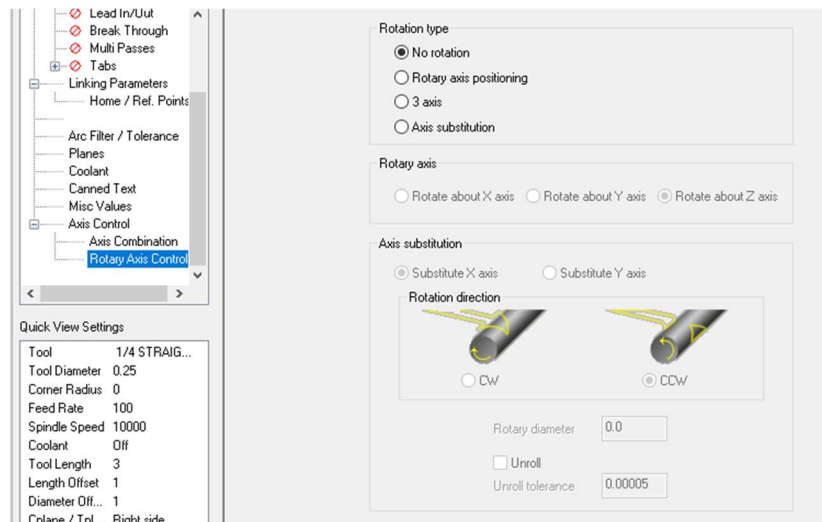
Rotary Axis

The rotary axis behaviour of the post output can be driven by the programmer in many ways. Current tool plane rotation, [Misc real 9](#), [Misc real 10](#), as well as the Rotary Axis button on the tool path parameters tab control the rotary output. See [3+2 Machining](#), [Cross Machining](#), [Face Machining](#), and [Multi-Axis](#).

3+2 Machining

Rotary axis indexing is driven by the orientation of the tool plane relative to the work coordinate system. If you are programming a transform operation and rotating existing operation, ensure the Tool plane method is selected in order to have the post output the rotation.

Rotary Axis Control (Milling Operations)



- No rotation** Uses the programmed tool plane to calculate the rotary axis position
- Rotary axis positioning** Uses the programmed tool plane to calculate the rotary axis position
- 3-Axis** Otherwise referred to as polar interpolation. In this mode the post will calculate the rotary axis position to lock one of the linear axes at zero based on the setup of the rotary axis in the machine definition. Note that operation must be programmed with a tool plane where the tool vector is parallel to the axis of rotation for the primary rotary (rot) axis.
- Axis Substitution** When machining with a tool that is perpendicular to the primary axis of rotation (rot), allows the user to replace motion in one of the linear axes with rotary motion. [Misc Real 9](#) and [Misc Real 10](#) are very useful for axis substitution to control the rotary start locations.



Rotary Axis Control (Mill-Turn Operations)

Rotation type

No rotation


Y axis

C axis

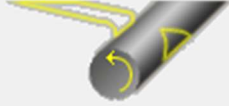
Axis substitution

Axis substitution

Rotation direction



CW



CCW

Rotary diameter

Unroll

Unroll tolerance

- No rotation** Uses the programmed tool plane to calculate the rotary axis position
- Y-Axis** Select Y axis if you would like to drive Y-axis output from the post processor. If your machine definition does not have a Y axis, the post will change this selection to C axis.
- C-Axis** Select C axis if you wish to drive rotary axis motion through the post processor.
- Axis Substitution** Used to output 4-axis simultaneous motion. Select the diameter that you wish to wrap the geometry about and select unroll if your geometry is already 'rolled up'. [Misc Real 9](#) and [Misc Real 10](#) are very useful for axis substitution to control the rotary start locations.



Multi-Axis

Depending on the limits of your machine, for a cut requiring rotary motion, two rotary axis solutions may exist that will result in the correct positioning of the tool to the part. For example on a machine with a C-axis rotating about world Z mounted on an A-axis rotating about world X, A90 C0 and A-90 C180 will result in the same tool to part orientation. Note that the linear axis coordinates required to perform the cut at these position may vary depending on the machine kinematics and controller options. The initial rotary axis solution for an operation can be selected through the use of [Misc Int 4](#). The post processor may change to the other solution to avoid machine limits or large rotary/tilt axis motions throughout the operation.

If the post processor encounters a large rotary/tilt axis motion and cannot change solutions to avoid it, the post will prompt the user for a course of action, typically a retract and reposition. A large rotary move is defined based on the values of [maxincrot](#) and [maxinctilt](#) for the rotary (rot) and tilt axis respectively. These limits are set based on the machine kinematics and controller options available to avoid causing part gouges and potentially even crashes. If you are a running a TCP type function, you may be able to set these values higher depending on the application. **We do not recommend increasing these values, do so at your own risk.**

When prompted, selecting 'yes' will have the machine retract, index the rotary and tilt axis to the next cutting location, approach, then continue cutting. If the retract is caused by the operation reaching one of the rotary axis limits and there are two possible solutions for the given position, the post will reengage the part at the previous point and continue cutting. If the retract is caused for any other reason, the post will skip the cutting motion from the point before the retract to the next point. If possible, try to adjust your tool path to avoid either case to achieve the best cut quality. The retract motion carried out here is governed by the [Misc Real 8](#) selection. If you select 'no' when prompted, for large rotary moves the post will skip the retract and simply continue the cut.

Singularities in multi-axis machining occur when you have a tool vector pointing along the axis of rotation of your rotary (rot) axis. For a vertical milling machine, this is most often directly along the Z axis of the machine. In this scenario the rotary axis can be positioned in any orientation to achieve the correct tool to part orientation. If a singularity occurs at the first point in a tool path, [Misc real 9](#) and [Misc real 10](#) control the initial index of the rotary axis. If a singularity occurs during a multi-axis tool path, the post processor will keep the previously output rotary axis value. When you leave the singularity (tool vector no longer along axis of rotation) you will often find that a large rotary or tilt axis move will occur.

If you are experiencing difficulties with large rotary/tilt motions and singularities, there are many adjustments that can be made. Try to break the tool path up at these locations (often these are singularity points). Try to change the tool constraint method to tilt the tool away from the singularity (this is difficult during swarf machining). Add data points to your tool path using tolerance, point generators, and angle generators. Adjust the part setup on the machine (often even a slight tilting of the part will help).

Depending on the kinematic configuration of the machine, during multi-axis operations, rapid indexing of a rotary or tilt axis may result in the output of a tool length/tcp cancellation (if



applicable), followed by an index, then recall of the tool length/tcp. This is for safety as large indexes while in a tcp type mode can produce dangerous machine motion.

There are some advanced multi-axis features that have been added to help with indexing of the rotary axes during an operation. The safety zone feature is used to avoid unnecessary retracts when indexing, instead keeping the tool at a safe location, and making small rotations to reposition for the next cut position. Similar to the safety zone, multi-axis linking can be used to achieve similar motion in between operations and reducing program run time.

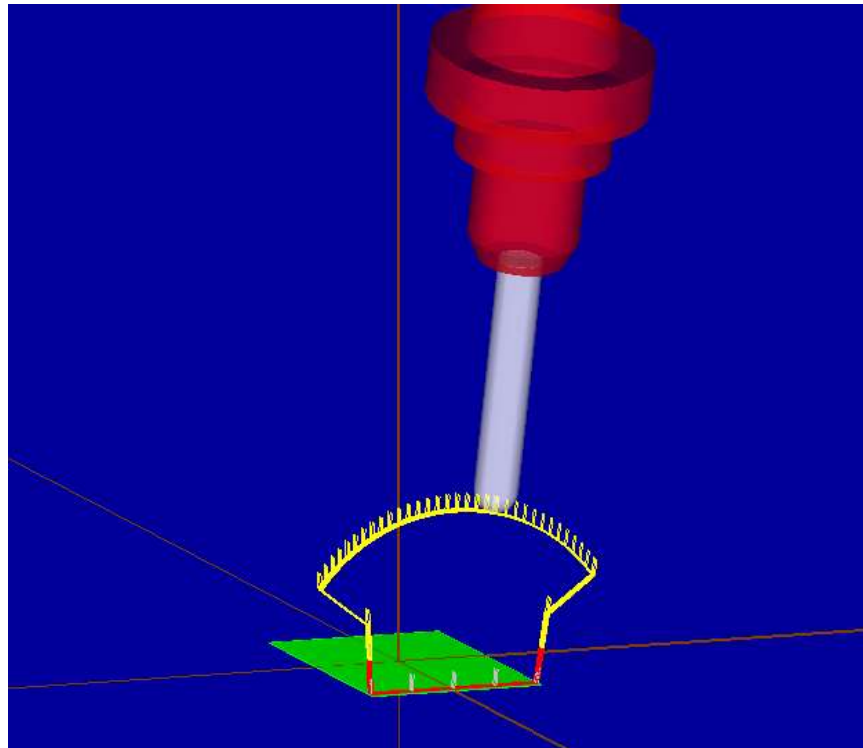
Transitions between cuts

Misc Real 7 and **Misc Real 8** control motion between cuts at null tool changes.

pretdec_strt (transitions at start of operations with same tool and transforms) and **pretdec_mid** (transition in the middle of an operations) are the two postblocks that make decisions about whether or not a retract and recall of modal values is required. There are many criteria shown here. This is an area that is very easily edited by the end user.

There are some instances during multi-axis tool paths where the post is not sent through a null tool change. It will handle the transition in the same manner with 1 exception.

Advanced multi-axis can control the tool transitions on the linking page in such a manner that machine axis retracts are not required. Instead the post will output rapid moves converted to high feed and leave tool length, tcp, and high-speed functions active. Be sure your machine definition general machine parameters maximum feed rates are set correctly.





Machine Setup Information

The UPK provides several tools to help with machine setup, and programming to work with that setup. The words datum and origin are used interchangeably in this section.

First, the posts will support dynamic datum tracking functions. This is when you setup a datum location on a part that moves along with the part in space. Dynamic datum tracking type functions are applicable to all machine where the workpiece is mounted to at least one rotary axis. The benefit of this type of function is that you can setup a part on the machine, have the datum located anywhere on the part (and anywhere in machine space) and the control will compensate for the parts location. It takes the pressure off needing to setup each individual part in the exact same location every time.

Use the [orgshift](#) switch to have the post prompt for a shift of all coordinates from Mastercam's WCS origin. This allows for programming a part in Mastercam with a WCS origin located anywhere. After the part is setup on the machine, locate the position for your machining origin and find the difference between the programmed WCS origin location and that machining origin. Then go back to Mastercam and enter this difference when you post out.

Machine and Control Definition

Every post processor is setup with a working machine and control definition. It is important to note that the posts reads both the machine and control definition during the posting process. Please make any changes with extreme caution. The following sections outline the areas of the machine and control definition that are read by the post processor, please ensure they are accurate.

Figures below use arbitrary values DO NOT COPY



Rotary Axis

The post reads the parameters from each of the rotary axis pages to define the kinematics for the given machine.

Machine Component Manager - Rotary Axis

Name: C Axis

Parameters | Geometry | Position/Orientation On Machine

Machine coordinate: C | Axis label (absolute coordinates): C | Axis label (incremental coordinates): C

Define macro-driven axis feed rate in axis motion events

World coordinate system
Axis of rotation: O deg. position: 0 deg. position: 0.0 deg.

Direction: CCW CW

Center of Rotation
X: 0.0 | Y: 0.0 | Z: 0.0

Tilt Angle... Defined angle (from 0 deg.): 0.0 deg.

Travel Limits (from 0 deg.)
Minimum travel limit: 0.0 deg.
Maximum travel limit: 360.0 deg.
Minimum reposition angle: 0.0 deg.
Maximum reposition angle: 360.0 deg.

Maximum Feed Rate
From Machine
Set from machine general parameters
Maximum feed rate: 36000.0 deg./min.

Break Motion
 Break rotary motion
 Use chordal deviation
Maximum rotary move: 6.283185 deg.

Fixed/Continuous Positioning
 Axis positions to fixed angles only
Index increment: 0.0
 Axis supports continuous positioning
Continuous Axis Type
 Signed continuous
 Signed direction, absolute angle (0-360 deg.)
 Shortest direction, absolute angle (0-360 deg.)

✓ ✗ ?



General Machine Parameters

The maximum and minimum feed rates are used by the post in situations where rapid motion is converted to high feed linear motion.

General Machine Parameters X

Coolant commands	Tool/material libraries	Machine dynamics	Cplane/Tplane
Op. feed rate limits, axis motion		Axis feed rate limits	

Toolpath Feed Rate Limits

	inch		mm
	Convert to mm/min.		Convert to in./min.
Minimum feed/min.:	0.0001 in./min.		0.001 mm/min.
Maximum feed/min.:	500.0 in./min.		12500.0 mm/min.

Minimum feed/rev.:	0.0001 in./rev.		0.001 mm/rev.
Maximum feed/rev.:	100.0 in./rev.		1000.0 mm/rev.

Inverse Time Feed Rate Limits

	inch		mm
	Convert to min./mm		Convert to min./in.
Minimum inverse feed rate:	1000.0 min.		100.0 min.
Maximum inverse feed rate:	0.0001 min.		0.001 min.

5 Axis Rotary Motion

Break rotary motion as a combined angle

Maximum rotary angle before break is required:



Axis Combinations

The axis combination specifies which axes defined in the machine definition are to be used for a given operation. Multiple axis combinations may exist for a given machine, but the post will only drive the axes that are used in the axis combination selected for a given operation.

Machine Axis Combinations ×

Axis Combinations

Name	Axes	Components
Default (1)	Y, X, Z, C, B	Machine Table, Tool Spindle

Components Used

Reference Points...
Home Position...



Arcs

Arc output can be setup in the control definition. We **recommend** the arc error checks shown in the screen shot below.