Postability Post Processor Documentation

Thank you for your purchase of a Postability post processor. Great care is taken in development, testing, and documentation of all of our products. If you ever have any questions or comments, please do not hesitate to contact us.

How to Use This Document

The following document contains generic information about our Unified Post Kernel. This applies to mill and mill-turn applications. We do have additional documentation for specific machine/control combinations. If available, it should have been sent to you along with the initial version of your post.

The first three sections, General Post Setup, UPK General Settings, and Miscellaneous Values, contain post setup information applicable to many post functions. These sections will reference other, more specific, sections for further information.

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

General Post Setup

Locate the Mastercam (mcamx or similar) directory on your computer.

Locate the machine definition (.mmd/.lmd/.rmd/.wmd) and control definition (.control)
Place it in (mcamx or similar)/cnc_machines directory

Locate the post processor files (.pst and .psb)
Place them in the directory based on the product you are using.
(mcamx or similar)/mill directory for use with Mastercam Mill
(mcamx or similar)/lathe directory for use with Mastercam Lathe
(mcamx or similar)/router directory for use with Mastercam Router
(mcamx or similar)/wire directory for use with Mastercam Wire

Unified Post Kernel (UPK) General Settings

Default settings will change between machines and controls. Please open the .pst file and review the settings. Adjust these to settings to best suit your application.

userotlock	0 = Do not output locking M codes, 1 = Output locking M codes
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

See **Misc Int 7** and **Misc Int 8**

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)
wcstype	0 = G92 at start, 1 = G92 at toolchanges, 2 = G54, 3 = Off
force_wcs	Force WCS output at every toolchange?
shiftlocation	0 = Apply tool plane based shifts in world coords before rotation of plane
	1 = Apply with rotation of plane
	2 = Apply tool plane based shifts in plane coords after rotation of plane

See **Work Offset, Work Coordinate System (WCS), T/C Planes**

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

The post will support indexing to within certain ranges to keep indexing distances to a minimum and to help make the code output easy to read.

hel_lead	Special helical move treatment with lead, 0=no, 1=yes
force_dpts	Force XY output on all drilling lines including cycle call?
tool_table	Tool table, 0=no, 1=yes
use_pitch	0 = Use feed for tapping, 1 = Use pitch for tapping
rigid_tap	0 = Floating tap output, 1 = Rigid tap output (suppress spindle output and output M29)
use_gear	Output gear selection code, 0=no, 1=yes

frdegstp	Step limit for rotary feed in deg/min

Post calculated rotary feed rates will only output if they change in value by an increment greater than this number.

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

If set to 0, this will override any control definition selections and avoid calculated rotary feed rates.

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

See **Multi-Axis**

ixtol	Tolerance in deg. for index error
miscerror	Error out if Misc Values button is disabled, 0=no, 1=yes
tseqno	Output sequence number at toolchanges when omitseq = yes
	0=off, 1=seq numbers match toolchange number, 2=seq numbers match tool number
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)
force_output	0 = Modal G code output at toolchanges, 1 = Force output of all G codes
	(see also force_wcs)

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

See **Machine Setup Information**

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

See **Work Offset, Work Coordinate System (WCS), T/C Planes**

tcp	Enable tool center point managment (0=Off,1=On)
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)
tiltcomb	Tilted work plane combination, only applicable when tiltplane active (0=AC,1=BC)
vector	0 = Physical rotary/tilt axis output, 1 = Vector output (5-axis only)
use_home	0 = Use home positions defined below (x_home)
	1 = Use home positions from Mastercam
use_g_inc	0 = Use new addresses for absolute/incremental output
	1 = Use g codes for absolute/incremental output (G90/G91 default)
strrotinc	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

#5-axis switches/variables

pivotdis	Distance from center of tilting axis to gauge line/collet line - mm
offsetdis	Distance from spindle center to center of tilting axis - mm
postcomp	Compensate for tool length in post (0=No,1=Yes)
maxtoolno	Maximum tool number for tool length buffer intialization
vrtltip	Control point is the virtual tool tip (0=No,1=Yes)

#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
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
old_new_sw	Switch old (6T), new (0T+) lathe canned cycle formats, 0=old, 1=new
thdaddress	0 = Use F address for threading, 1 = use E address for threading
css_start_rpm	0 = Start with CSS directly, 1 = Start with RPM then change to CSS
linneglim	#Negative axis stroke limit - mm
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
#Only for tilt about X - 2 = Watch Y axis stroke and change solutions accordingly

Miscellaneous Values

Milling Only

Milling (Mill-turn)

Miscellaneous values

Integers

Misc. Integer [1]	0
Abs/Inc, top level [0=Abs,1=Inc]	0
3D Comp (Surface Normal) [0=No,1=Yes]	0
Start Solution [0=-Tilt,1=+Tilt]	0
Multi-ax TCP suppress [0=TCPC,1=No]	0
Unit/Rev Feed [0=No,1=Yes]	0
Rotary [0=Post,1=Lock,2=Brake,3=Off]	0
Tilt [0=Post,1=Lock,2=Brake,3=Off]	0
Face/OD Interpolation [0=Off,1=On]	0
M00 Before Operation [0=No,1=Yes]	0

Reals

HS [1=G08,2=AICC,3=HPCC]	0.0
Accel/Decel Value [0=No Output]	0.0
Misc. Real [3]	0.0
Misc. Real [4]	0.0
Misc. Real [5]	0.0
Misc. Real [6]	0.0
Start Op Retract [0=Axis,1=XY,2=XY/Z,3=N	1.0
Mid Op Retract [0=Axis,1=XY,2=XY/Z,3=No	1.0
Rot Start [1=Value,2=Last,3=Rev,4=Prompt]	0.0
Rot Start [Enter Value or Rev]	0.0

Automatically set to post values when posting

Set to post values | Set to Op Defaults | ✔ | ✖ | ?

Lathe (Mill-turn)

Miscellaneous values

Integers

Misc. Integer [1]	0
Abs/Inc, top level [0=Abs,1=Inc]	0
Misc. Integer [3]	0
Misc. Integer [4]	0
Misc. Integer [5]	0
Misc. Integer [6]	0
Misc. Integer [7]	0
Misc. Integer [8]	0
Misc. Integer [9]	0
M00 Before Operation [0=No,1=Yes]	0

Reals

Misc. Real [1]	0.0
Misc. Real [2]	0.0
Misc. Real [3]	0.0
Misc. Real [4]	0.0
Misc. Real [5]	0.0
Misc. Real [6]	0.0
Start Op Retract [0=Axis,1=XY,2=XY/Z,3=N	1.0
Mid Op Retract [0=Axis,1=XY,2=XY/Z,3=No	1.0
Misc. Real [9]	0.0
Misc. Real [10]	0.0

Automatically set to post values when posting

Set to post values | Set to Op Defaults | ✔ | ✖ | ?

Misc Int 1 -

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 overtravels.

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 Postability 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.

Misc Int 7 - Selection of rotary axis lock/brake code output. If left as 0, the post will make an automatic selection based on the cutting type. If you wish to override this selection, select 1 to lock the rotary axis, 2 to turn the rotary axis brake on, 3 to turn off rotary locking. See the post switches, **userotlock** and **userotbrake**.

Misc Int 8 - Selection of tilt axis lock/brake code output. If left as 0, the post will make an automatic selection based on the cutting type. If you wish to override this selection, select 1 to lock the rotary axis, 2 to turn the rotary axis brake on, 3 to 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.

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 3 -

Misc Real 4 -

Misc Real 5 -

Misc Real 6 -

Misc Real 7 - Mastercam goes through what is called a null tool change in between 2 operations with the same tool and between steps of a transform tool path. Use this selection to manipulate the retract move before the current operation. Retracts are triggered by a set of customizable conditions that may include motion detected in the rotary axes, change in tool plane, angular change in a tilted work plane, change in TCP, change in cut strategy (e.g. planar to rotary substitution), change in work offset, change in toolplane origin, forced M00 before operation, etc.

> 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 called 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 move during the current operation. Retracts are triggered by a set of customizable conditions that may include motion detected in the rotary axes, change in tool plane (Drill 5-axis, Circmill 5-axis), angular change in a tilted work plane, etc.

> 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 axis output where applicable. 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 prevous 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 value (1) or the Rev (2) for the rotary output.

Work Offset, Work Coordinate System (WCS), T/C Planes

Shifted T planes will output a shift command, either before, with, or after a tilt plane/tcp call. See **shiftlocation**. If you wish to have the post ignore any shifted plane origins see **ignoreorg.**

If you want to change work offsets on the machine, adjust your WCS origin in the View Manager. The easiest way to do this is to take a copy of the plane you want to use (usually Top), shift the origin to the position you wish to use on the machine, then select this as your WCS for the applicable operations.
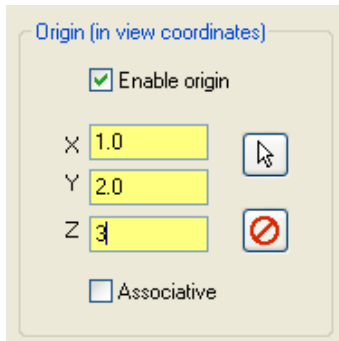
If you just want a new local origin (shift) inside the same work offset, adjust your T/C plane origin for that tool path. This is often used in situations where you want to make the code easier to read for the operator.
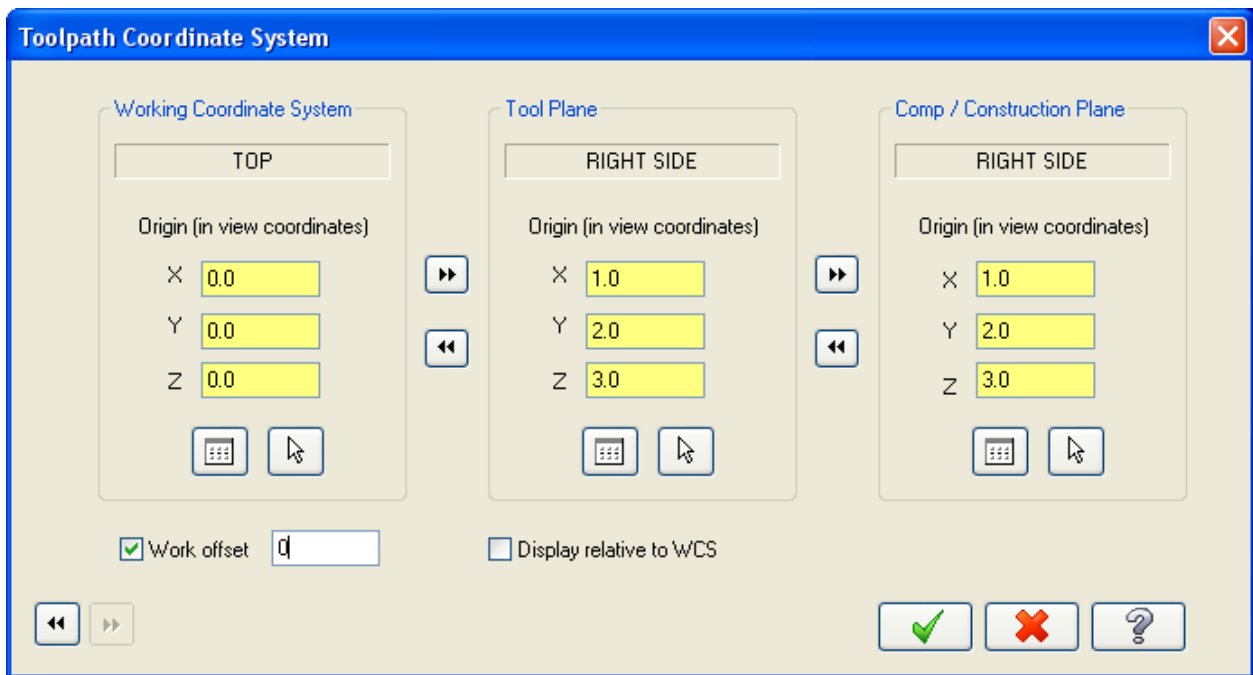
If using work offset values from Mastercam, be sure to set the work offset for each operation. If you leave it as -1, Mastercam will automatically increment the number with each new plane and you will get several work offset values output.

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 rotation of the plane. So, if you program a transform by tool plane you will get rotary axis index output. The same applies if you manually rotate a tool plane.
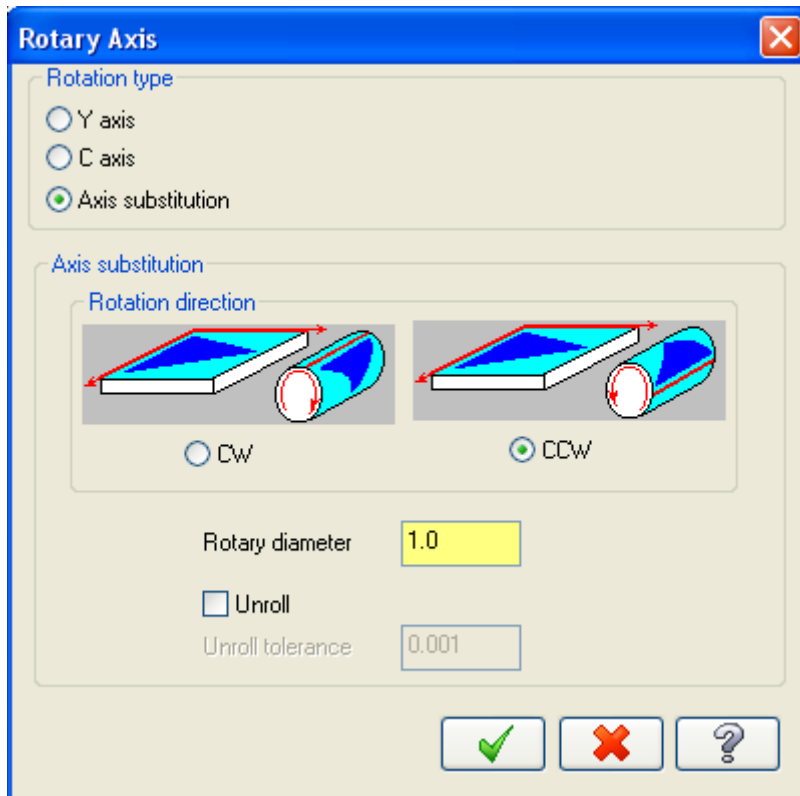
Cross Machining (Machining from the side or outer diameter of the part)

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. Leaving the small check box next to the rotary axis button (on the tool path parameters page) unchecked is equivalent to selecting Y axis.
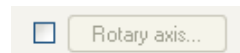
Select C axis if you wish to drive rotary axis motion through the post processor.

Axis Substitution is 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.

Face Machining (Machining from the top or positive Z-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. Leaving the small check box next to the rotary axis button (on the tool path parameters page) unchecked is equivalent to selecting Y axis.
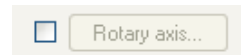
Select C axis if you wish to drive rotary axis motion through the post processor for polar machining. Cutting to the centre of rotation is a very dangerous practice but if you do, the post will output the previously output rotary axis value.

Multi-Axis

Multi-axis cuts can be achieved by either of two solutions. To select the starting solution use **misc Int 4**. The post processor may change to the other solution to avoid machine limits or large rotary/tilt axis motions.

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. A large motion is defined by the switches **maxincrot** and **maxinctilt**. If you are a running a TCP type function, you may be able to set these values higher depending on the application. Postability does not recommend increasing these values, please do this at your own risk.

You can select 'yes' to have the machine retract, index the rotary and tilt axis to the next cutting location, approach, then carry on cutting. This will skip the cutting motion from the first point to the second, so, if possible, try to adjust your tool path to avoid this. Retract motion is governed by your **Misc Real 8** selection. You can also select 'no' to have the machine carry on without a retract.

Singularities in multi-axis machining occur when you have a tool vector pointing along the axis of rotation of your rotary axis. This is most often directly along the Z axis of a machine. 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 you can make. 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 a slight tilting of the part will help.

During multi-axis operations, any rapid indexing of a rotary or tilt axis will 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.

Indexing during advanced multi-axis will not occur in the same manner. Here the post will produce high feed rotary/tilt indexing motion but will not cancel or recall tcp modes. This is because advanced multi-

axis breaks the indexing up in such a way that indexing around a part occurs in very small increments. Motion is converted to high feed to avoid many 'jerky' rapid motions in succession.

Machine Setup Information

The Postability 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 any dynamic datum tracking functions that your machine. 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 with at least one rotary or tilt axis holding the part. 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 it will work. If you run another part and it's in a different location, you only have to relocate the datum on the machine.

Use the **orgshift** switch to have the post prompt for a shift of all coordinates from Mastercam's WCS origin. So, you can program 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, find the difference between the programmed WCS origin location and that machining origin, go back to Mastercam and enter this difference when you post out.

Machine Definition

Every post processor is setup with a working machine definition. It is important to note that the posts read this machine definition during setup. Please make any changes with extreme caution.

Posts read the following sections of the machine definition. Please be sure they are accurate.

Rotary Axis

General Machine Parameters

Axis Combinations

Home Positions

Home position values can be used in work offset coordinates, machine coordinates, or reference return coordinates.

use_home   0 = Use home positions defined below (x_home)
                1 = Use home positions from Mastercam

x_home_mm : 0.   #X home position
y_home_mm : 0.   #Y home position
z_home_mm : 0.   #Z home position

x_home_in : 0.   #X home position
y_home_in : 0.   #Y home position
z_home_in : 0.   #Z home position

Arcs

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

Transitions between cuts

Misc Reals 7 and 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 has the ability to 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.