Building a Hurco 5-Axis Post Processor

Author: Michael Cope - November, 2012

Provided Example IGES Model:



- The provided program examples (*5 Axis Simultaneous.FNC*; *and 3+2.FNC*) both reference the features on the supplied .IGES file (pictured above).
- The **3+2.FNC** program uses G68.2 Transform Planes to drill the 4 angled holes, and to position and machine the J-shaped slot on the right side of the part.
- The **5** Axis Simultaneous.FNC program shows an example of simultaneous machining of the red projected square with the tool tilted normal to the surface of the sphere.
- The simultaneous example uses Tool Center Point Management (TCPM) and is programmed using IJK tool vectors.

Miscellaneous Post Processor Considerations

Program Header:

% :0001 M31 (Rotary Axes Encoder Reset) M126 (Shortest Angular Traverse) G0 G20 G40 G80 G54 G90 (Traditional Safety Line) M140 (Retract Along Tool Vector) G53 Z0. G0 X0. Y0. A0 . C0.

- Make sure to output an M31 to reset the rotary axis encoder to the current machine position. Failure to do this can cause the rotary axis to "unwind" when commanded to return to a position of zero degrees.
- Shortest Angular Traverse (M126) will force the control to take the shortest path to the commanded rotary position regardless of negative or positive. This can be cancelled using an M127 for special cases in which a particular rotary direction is necessary.
- DO NOT call a G17 or other plane designation in the safety line. This can cause problems when programming in 5-axis...especially with Transform Planes.
- M140 will cause the tool to retract along the current tool vector to the machine limit position. The tool can be commanded to retract a specified distance by the addition of an "L".
 <u>Example</u>: M140 L3.0
- When retracting the Z axis to the home position, it is recommended that a G53 Z0 be used (machine coordinate system). The traditional G91 G28 Z0 will work, but the post must be configured to output the G90 immediately after to return the control to absolute.

Transform Plane - G68.2:

G68.2 X0. Y0. Z0. A-45. C225. (Transform Plane Data)

- G68.2 also enables Tool Center Point Management.
- G68.2 does NOT command any machine movement rotary axis machine commands must be output on a separate line of code, or by use of the ASR command (G08.2).
- G69 cancels the Transform Plane.
- The XYZ data in a G68.2 is used to reposition the origin point for the transform plane if desired, and all following XYZ data will reference this new zero location. This reference position is relative to the original workpiece zero...unless another transform plane was active when the G68.2 was called.
 <u>Note</u>: G68.2's can be stacked, but a separate G69 is required to cancel each active transform plane. They will be cancelled in the reverse order they were called the last one called will be cancelled first. When stacking G68.2, each additional transform plane will be relative to the previous workplane.

- The rotary information (A, B, C) (I, J, K) (U, V, W) is used to determine the angle of the new workplane.
- Rotation angles in the G68.2 are commanded using ISO standard rotation conventions.
 <u>Example</u>: rotations to the front or right side of a block are positive movements; rotations to the back or left side of a block are negative movements; counter-clockwise rotations (CCW) around the Z-axis are positive rotations; clockwise (CW) rotations around the Z-axis are negative rotations.
- NOTE: actual machine rotation commands can be either ISO standard or NON-ISO standard depending on the parameter setting within the control.

To verify the current machine setting follow the steps below:

- Press the AUXILIRY button on the control panel
- Select the UTILITY SCREEN icon from the pup-up menu
- Select the USER PREFERENCES softkey on the screen
- Select the MORE softkey on the screen
- Select the ROTARY AXES PARAMETERS softkey on the screen
- <u>Recommended Settings</u>:
 - ISO Standard = YES
 - Tilt Axis Preference = NEGATIVE

Simultaneous Movement:

```
T03 M6
S1200 M3 M8
G0 X0.Y0. (XY Initial Position)
Z5.0 (Initial Z Axis Position)
M128 (Tool Center Point Management ON)
G8.2 X-1.0934 Y-1.0941 Z3.5643 I-0.4893582 J-0.4902342 K0.7212482 (ASR Target Position)
G43.4 (Toolpath Linearization)
M13 (C Axis Unclamp)
M33 (A Axis Unclamp)
G01 X-1.0934 Y-1.0941 Z3.5643 I-0.4893582 J-0.4902342 K0.7212482 F50.
X-0.8487 Y-0.849 Z3.2037 I-0.4893582 J-0.4902342 K0.7212482
~
~
\sim
X-0.75 Y-0.75 Z3.0607 I-0.4893582 J-0.4902342 K0.7212482 F200.
X-0.7471 Z3.0627 I-0.4877059 J-0.4906040 K0.7221154
X-0.7441 Z3.0647 I-0.4860530 J-0.4909710 K0.7229800
X-0.7412 Y-0.7499 Z3.0667 I-0.4844140 J-0.4913580 K0.7238165
~
M129 (Tool Center Point Management Cancel)
G0 M140 (Retract Along Tool Vector)
G53 Z0.
M31 (Rotary Axis Encoder Reset)
```

G53 A0. C0. M30

- It is a good idea to position the XY & Z axes to an initial point location prior to invoking the M128 TCPM command. This will help eliminate out of travel limit errors when running the program.
- M128 forces all preceding XYZ data to reference the un-rotated workpiece coordinate system.
 <u>Example</u>: if the part is rotated -90 degrees around the A-axis (toward the operator), a commanded position of Z-1.0" will still be located along the backside of the part in the Y-axis machine direction...but one inch down from what would actually be the top of the part if the part was not rotated. The physical or corresponding rotary angle position will determine the tool vector for the move in the above example the tool would be perpendicular to the backside of the part.
- Simultaneous moves can also be programmed using ABC rotary angles instead of IJK tool vectors if desired.
- IJK tool vector tokens are not modal and must be output for every line.
- Tool vectors should be output to 6 decimal places. Outputting to only 4 is not sufficient and can cause erratic movement or poor surface finishes.
 <u>NOTE</u>: tool vector IJK tokens have no unit of measurement therefore they should remain unchanged when switching between inch and metric. It is recommended that this function be tested by posting a process in both inch and metric verify the tool vector data in the program did not change.
- M129 cancels TCPM.

G8.2 - ASR (Automatic Safe Repositioning):

G8.2 X-1.0934 Y-1.0941 Z3.5643 I-0.489358 J-0.490234 K0.721248 (ASR Target Position with tool tilted, or not perpendicular to active workplane)

(OR)

G8.2 X0. Y-0. Z4.0472 IO JO K1 (ASR Target Position with tool perpendicular to active workplane)

ASR is a command buffer that occurs internally within the control, and causes the machine to
reposition automatically to a specified target point – without allowing the machine to overtravel in
any axes. The post should be triggered to output the necessary G08.2 whenever a reposition of the
tool-tip is necessary. The post-processor would simply output a line of NC code with the G08.2,
followed by the desired XYZ position and necessary tool vector or rotary angle data.

The ASR command causes the machine to find the most optimized path to the new target point by creeping along the machine travel limits, without the operator or post having to designate these moves in the program code.

• The rotary angles on the G8.2 line should be output using IJK vectors - to align the tool with the active work coordinate. If rotary ABC axis commands are used, and an offset is applied to the tilting axis in the part setup, the axis will tilt correctly in one direction but mis-position by the same amount in the other direction.

G43.4 - Toolpath Linearization:

- Toolpath Linearization eliminates gouging of the workpiece, and removes the looped line segments on the part that are formed from the XYZBC or AC moves that a CAM system outputs.
- Without linearization, only the starting and ending points of a move are controlled...whatever happens in between is a blind rotation.
- With linearization the tool-tip "attached itself to the workpiece" and in the example below, the Zaxis would move with the rotation – creating a linear movement between the start and endpoints of the move.

