

HAAS UMC-750 NX Postprocessor

User Guide

Version 1.0.0
2021-08-08



By
James

NX CAD/CAM Automation Service

I. Machine Specification

TRAVELS	S.A.E	METRIC
X Axis	30.0 in	762 mm
Y Axis	20.0 in	508 mm
Z Axis	20.0 in	508 mm

B AXIS – TILT	S.A.E	METRIC
Travel	120 ° to 35- °	120 ° to 35- °

C AXIS – ROTATION	S.A.E	METRIC
Travel	360 °	360 °

Dynamic Work Offsets + Tool Center Point Control = Making 5-Axis Easier

Standard On All UMC Series Universal Machining Centers



WITH DWO/TCPC THERE'S NO NEED TO:

- X Place the part in the exact same location every time
- X Repost the program when the fixture changes
- X Repost the program if the job moves to another Haas machine

DWO/TCPC CHANGES EVERYTHING:

- ✓ Save time on programming and setup
- ✓ Reduce fixturing costs
- ✓ Increase your number of multi-axis operators
- ✓ Increase your productivity and profits

- NX Version 1899
- Post Builder 1899
- Unit: Inch

II. Work Dynamic Offset

G254 - Dynamic Work Offset (DWO) (Group 23)

G254 Dynamic Work Offset (DWO) is similar to TCPC, except that it is designed for use with 3+1 or 3+2 positioning, not for simultaneous 4- or 5-axis machining. If the program does not make use of the tilt and rotary Axes, there is no need to use DWO.



CAUTION:

The B-Axis value of the work offset you use with G254 MUST be zero.

With DWO, you no longer need to set the workpiece in the exact position as programmed in the CAM system. DWO applies the appropriate offsets to account for the differences between the programmed workpiece location and the actual workpiece location. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

The control knows the centers of rotation for the rotary table (MRZP) and the location of the workpiece (active work offset). This data is used to produce the desired machine motion from the original CAM-generated program. Therefore, it is recommended that G254 be invoked after the desired work offset is commanded, and after any rotational command to position the 4th and 5th axes.

After G254 is invoked, you must specify an X, Y, and Z Axis position before a cutting command, even if it recalls the current position. The program should specify the X and Y Axis position in one block and the Z Axis in a separate block.



CAUTION:

Before rotary motion, use a G53 Non-Modal Machine Coordinate motion command to safely retract the tool from the workpiece and allow clearance for the rotary motion. After the rotary motion finishes, specify an X-, Y-, and Z-Axis position before a cutting command, even if it recalls the current position. The program should specify the X- and Y-Axis position in one block and the Z-Axis position in a separate block.

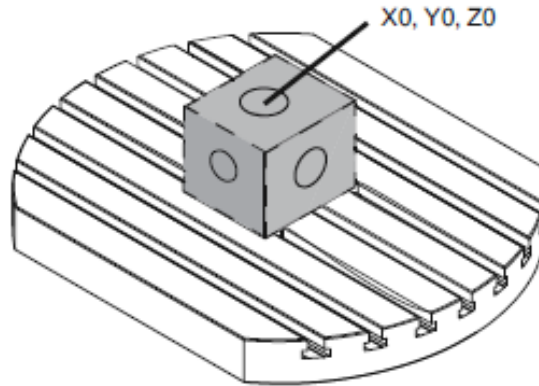


CAUTION:

Be sure to cancel G254 with G255 when your program does simultaneous 4- or 5-axis machining.

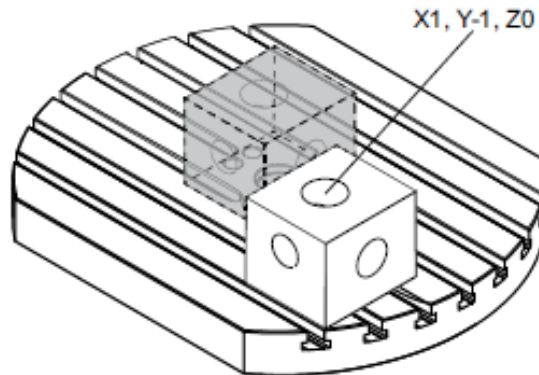
***** You can set the work offset at anywhere!!!**

F7.50: Original Programmed Position



In Figure F7.51, the actual workpiece is not located in this programmed position. The center of the workpiece is actually located at X_1, Y_1, Z_0 , and is defined as G54.

F7.51: Center at G54, DWO Off



DWO is invoked in Figure F7.52. The control knows the centers of rotation for the rotary table (MRZP), and the location of the workpiece (active work offset G54). The control uses this data to apply the appropriate offset adjustments to make sure that the proper toolpath is applied to the workpiece, as intended by the CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

III. TCPC-G234 Tool Center Point Control

G234 Tool Center Point Control (TCPC) is a software feature in the Haas CNC control that allows a machine to correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by a CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different.

The Haas CNC control combines the known centers of rotation for the rotary table (MRZP) and the location of the workpiece (e.g., active work offset G54) into a coordinate system. TCPC makes sure that this coordinate system remains fixed relative to the table; when the rotary axes rotate, the linear coordinate system rotates with them. Like any other work setup, the workpiece must have a work offset applied to it. This tells the Haas CNC control where the workpiece is located on the machine table.

The conceptual example and illustrations in this section represent a line segment from a full 4- or 5-axis program.

These key presses and program codes impact G234:

- G234 invokes TCPC and cancels G43.
- When using tool length compensation, either G43 or G234 must be active. G43 and G234 cannot be active at the same time.
- G234 cancels the previous H-code. An H-code must therefore be placed on the same block as G234.
- G234 cannot be used at the same time as G254 (DWO).

G234 (TCPC) is intended for simultaneous 4- and 5-axis contouring programs. An active work offset (G54, G55, etc.) is required to use G234.

Notes:

- The TCPC will be automatically turned on if it is a simultaneous 4/5 axis machining process.
- When TCPC is on, dynamic offset G254 will be disabled.

For more information

Please contact **NCCAS** (NX CAD/CAM Automation Services)

support@nxcadcam.com

THANKS!