

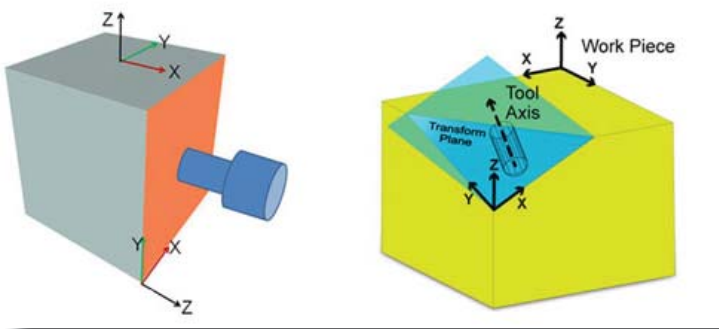
NC + Conversational

Tool Center Point Management

Description: Eliminates the need to account for the machining center's centerlines of rotation. Instead, you simply program from part zero. Tool Center Point Management allows you to position the part anywhere on the table.

Benefits

- » Faster setup.
- » Less complex post processor (NC).
- » Save time reposting the program (NC).



NC + Conversational

Transform Plane

Description: Transform Plan essentially changes programming on a 5-axis machining center back to 2.5D programming that you would use on your 3-axis machine. You don't need to worry about the tilting or rotating. You define your part origin one time, and Transform Plane manages all of the other part origin locations.

- » Transform Plane makes the transition to 5-sided easy. Simply establish the initial part origin and program the first side of the part. Then, tell the control the axis angle(s) to the next side.
- » Program the features on that side of the part and tell the control the axis angle(s) to the next side.
- » Follow the same steps for each side of the part, and Transform Plane figures out the tilting and rotating required.

TOOL SETUP - NC TOOL OFFSETS					
MACHINE	PART		SPINDLE		
X	0.0000	-0.0000			0
Y	0.0000	-0.0000	FEED (STOPPED)		0.0
Z	0.0000	0.0000	TOOL IN SPINDLE		0

CORNER RADIUS OFFSETS					
1	0.0625	9	0.0000	17	0.0000
2	0.3000	10	0.0000	18	0.0000
3	0.0000	11	0.0000	19	0.0000
4	0.0000	12	0.0000	20	0.0000
5	0.0000	13	0.0000	21	0.0000
6	0.0000	14	0.0000	22	0.0000
7	0.0000	15	0.0000	23	0.0000
8	0.0000	16	0.0000	24	0.0000

Enter tool corner radius offset.
These offsets are used for cutter compensation.

NC Only

3-D Tool Compensation

Description: Eliminates the need to repost the program in order to adjust the tool diameter due to tool wear or tool substitution.

Benefits

- » Compensates for tool wear.
- » Eliminates idle time caused by tool breakage.
- » Provides flexibility and freedom for tool selection.

Shortest Angular Traverse

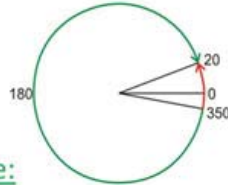
NC + Conversational

- With Shortest angular traverse:

Initial Position	Commanded Position	Angular distance traverse
350°	20°	+30°

- Without Shortest angular traverse:

Initial Position	Commanded Position	Angular distance traverse
350°	20°	-330°



Shortest Angular Traverse

Shortest Angular Traverse takes the fastest path to reach the target position.

For example, if you position the rotary axis at 350 degrees and then command the next position to 20 degrees, the distance will be 30 degrees when Shortest Angular Traverse is activated versus 330 degrees when it is deactivated.

Without Tool Path Linearization



With Tool Path Linearization



NC Only

Tool Path Linearization

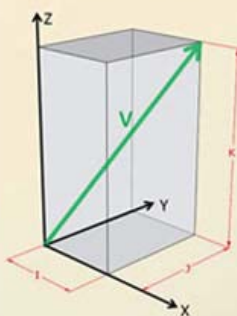
Tool Path Linearization eliminates gouging of the workpiece and removes the looped line segments on the part that are formed from the XYZBC or AC moves that a CAM system uses.

Benefits:

- » Improves surface finish quality.
- » Smaller NC programs.

Tool tilt angle & direction away from surface contact point

- G01 X10. Y10. Z10. I0.5 J0.5 K0.707106



NC Only

Tool Vector Input

Tool Vector Input allows the control to compute machine angles and positions, and calculates the angle the tool is going to tilt from the contact point of the surface.

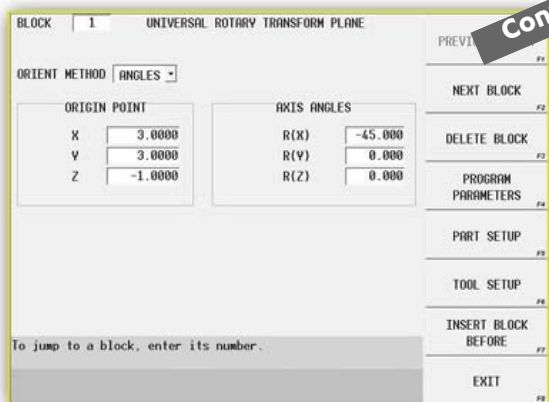
Benefits:

- » Scheduling flexibility (the program can run on another 5-axis machine with a different axis configuration)
- » Simplified post processor.



Automatic Safe Repositioning

Invented by Hurco. Automatic Safe Repositioning (ASR) tells the machine to retract along the vector and override the out-of-limits protocol. No stoppage. No error message. The software is smart enough to retract, move to Z0 along the X-limit, re-orient the tool, move to the retract plane, move above the plunge point, and plunge to the target along the tool vector.



Conversational Only

Universal Rotary

Universal Rotary allows a part program to be used interchangeably on both types of Hurco 5-axis machining centers even though they have different configurations. You simply program how the tool rotates into position on the part, and the control instructs the machine how to orient the axis to cut the feature.



Tool Vector Retract

The tool retracts along the tool tilt angle and the tool vector angle becomes the Z axis.

- » Allows the tool to retract along the current tool vector.
- » Eliminates the need to program retract clearance points.
- » An incremental retract distance can also be designated (M140 L2.0).

NC Only

Tool Vector Canned Cycles (TVCC)

NC canned cycles can be modified to execute along the current tool vector at a 3D point without having to define a full Transform Plane (G68.2). The G08.2 ASR Command can be used to re-orient the tool to a new tool vector after which the canned cycle can be commanded to execute along the new tool vector:

G08.2 X_Y_Z_I_J_K (ASR move to start point and tool vector orientation)

G81 X_Y_Z_I_R_[P_Q_L_K_F_S_](e.g. Drill along current tool vector)

Syntax

G_ X_Y_Z_I_R_

G_	Canned cycle G-code
X_Y_Z	3D position of hole top
I_	Incremental depth of hole from 3D position of hole top, along (inverse of) current tool vector. Normally a positive number to drill into part.
R_	Optional. Incremental retract distance from 3D hole top, along current tool vector. Normally a positive number. Default is 0 when not specified.
[P_Q_L_K_F_S_]	Optional canned cycle parameters are supported

Tool Vector Canned Cycles

The Tool Vector Canned Cycles make it easier to program in CAM. You can drill or tap along angles without Transform Plane.

NC + Conversational

ROTARY AXES PARAMETERS

A AXIS

ROTARY CENTERLINE Y

ROTARY CENTERLINE Z

AUTOMATIC CENTERLINE CALCULATION

MACHINE COORDINATE RELATIVE

Enter the machine coordinate for the center of rotation.

A AXIS

B AXIS

C AXIS

STORE MACHINE POSITION

EXIT

Rotary Axes Centerline Probing

Rotary Axes Centerline Probing automatically measures the centerlines of the rotary axes.

Benefits:

- » Generates accuracy report.
- » Determines rotary axis alignment issues.
- » Easy to setup and execute.



True Interrupt Cycle

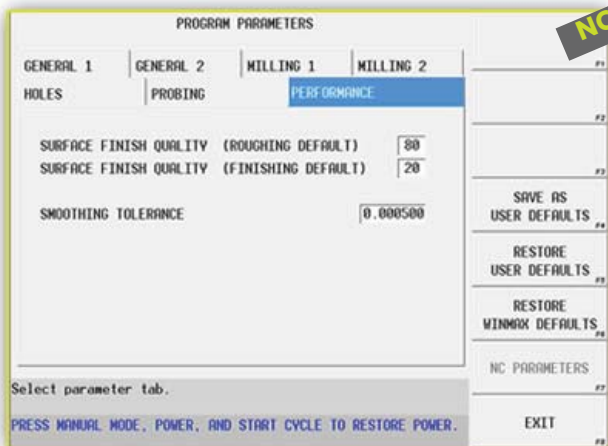
No need to teach the control the path the tool takes when retracting and returning to the part –the Hurco control does it for you.

When you press the Interrupt button, the spindle stops cutting, the coolant shuts off, and the tool automatically retracts to Z home. You can jog the machine in any direction to check the part or change tool inserts. Then, simply press two buttons and the cycle automatically resumes right where it left off—at the speed you choose.



Recovery Restart

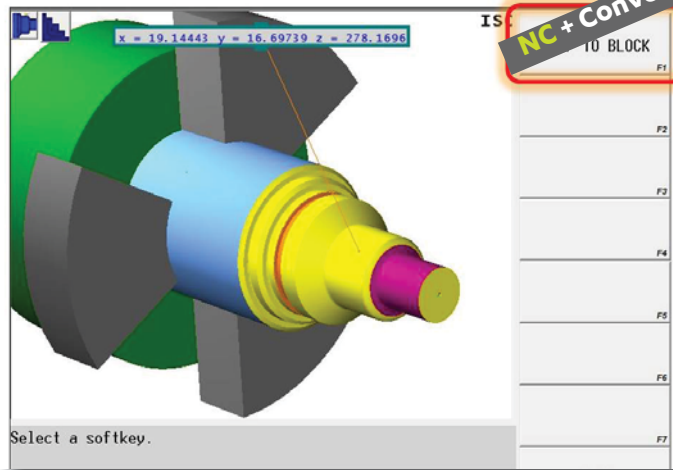
After stopping a cycle, Mid-Program Restart makes it easy to restart where you left off. You can change a broken tool without the typical frustration because you can get right back to the correct point in the program without the need to write additional code or cut air.



SelectSurface Finish Quality (SFQ)

Tired of supposedly smooth arcs looking like orange peels? What about vibration lines appearing all over your part? SFQ is an easy way to control surface finish quality from within your conversational or NC part program.

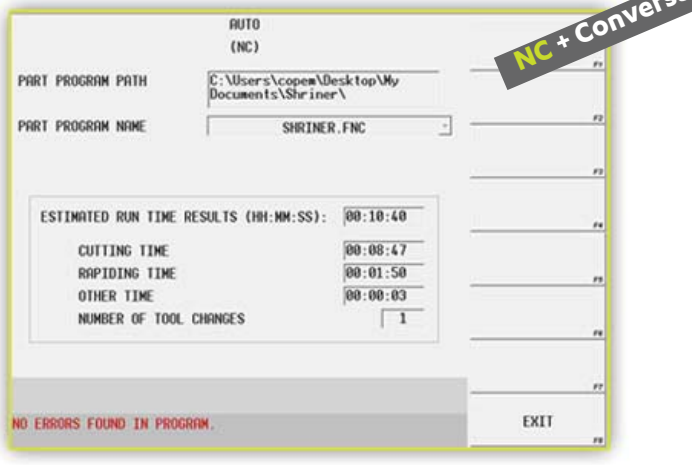
You select the desired surface finish machine parameter and the software automatically adjusts servo gains, cornering tables, acceleration and jerk parameters to give you the best cycle time.



Data Block Search

Touch the feature you need to edit on the graphics screen and the corresponding code appears automatically.

Benefits: Simplifies the tedious task of searching for a data block or line of code during the editing process.

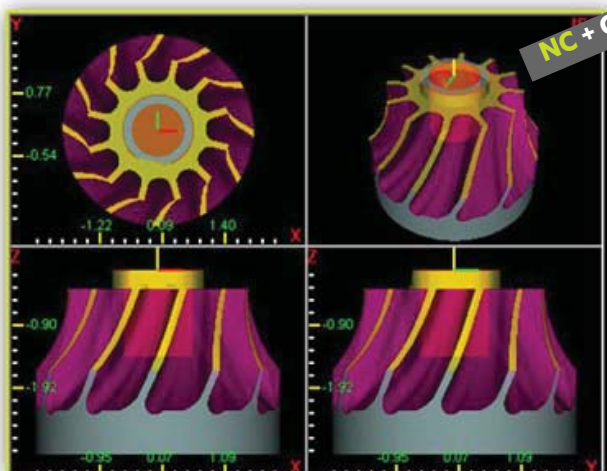


Estimated Run Time

Provides more accurate computation of run time than estimations from a CAM system because our computation takes into account algorithms from the machine's motion control system.

Benefits:

- » Accurately quote jobs at the control.
- » Gives you peace of mind and better scheduling capabilities.



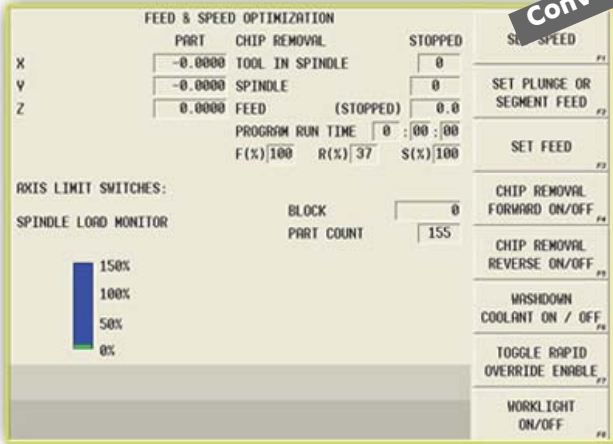
Advanced Verification Graphics with 3D Solid Rendering

Graphics feature with solid rendering of the tool path, including dynamic rotation, tool cut simulation, dynamic view manipulation, and more! You can view the rendered part from any angle and you won't have to redraw it. Every peck level can be optionally displayed to see as much detail as you need.

Benefits:

- » Reduces scrap.
- » Reduces programming time.
- » Quickly proves out the part program.

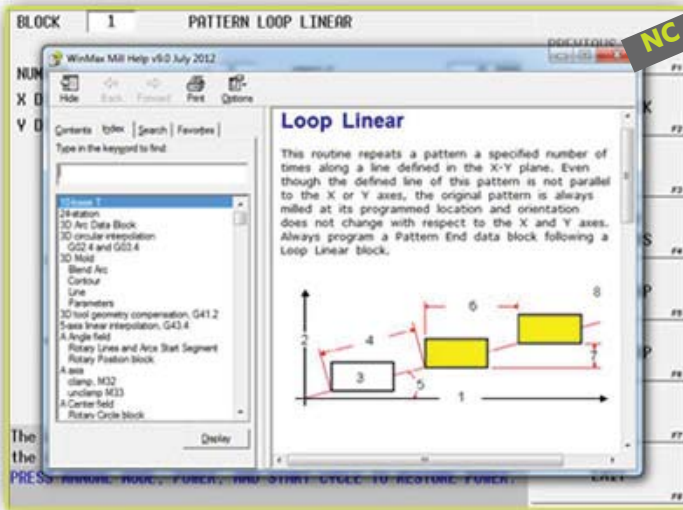
Conversational Only



Feed + Speed Optimization

We understand how you work. The Feed + Speed Optimization of our control lets you save your adjustments instead of forcing you to manually enter the new values into the part program. Using your expertise, just find the sweet spot with the override knobs and store the optimal settings with the press of a button.

NC + Conversational



Interactive Help System

Our help system, which includes pictures, diagrams, and videos when helpful, takes you to the appropriate section based on where you are in the control. No searching. No frustration. Just answers.

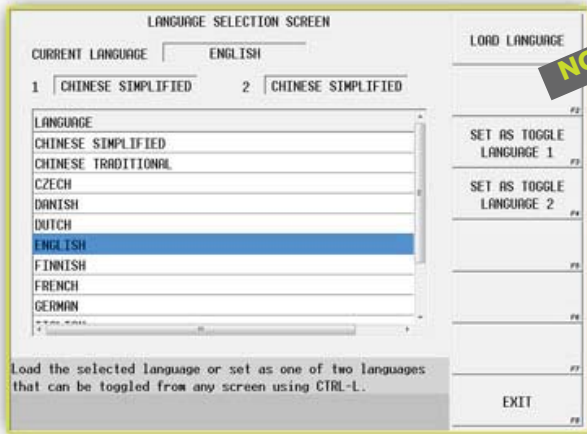
Includes G and M code lookup table.

NC + Conversational



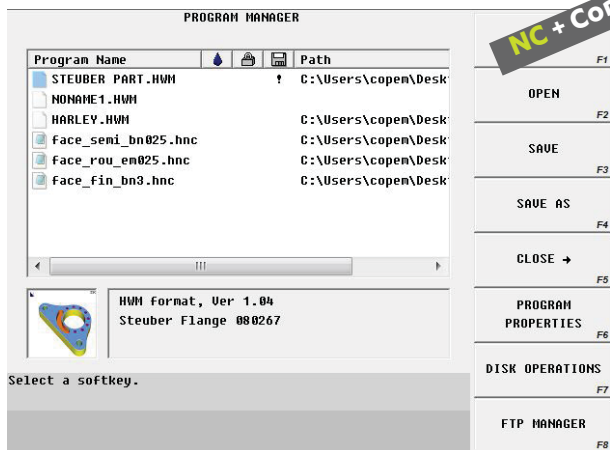
UltiMonitor

UltiMonitor combines powerful networking capabilities with remote monitoring capabilities. You can remotely monitor your Hurco equipment from your web browser via control views and live video (camera included).



Foreign Language Toggle check

Choose two languages from the exhaustive list, and the control allows you to switch back and forth during both programming and machining mode without having to reboot the control.



Extensive Program Manager

The Hurco program manager saves pertinent information and includes a user-friendly interface to make accessing the information you need painless.

Part Program Preview includes a thumbnail graphic of the part program.

Extended file naming—no weird nomenclature required.

Ability to write protect part programs, add notes, and define material.