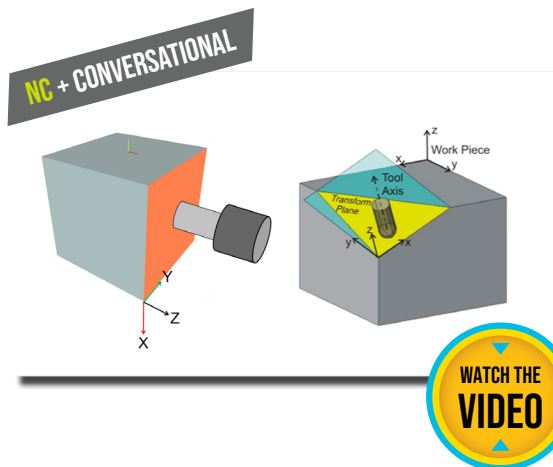


Thanks for checking out the control features that make Hurco CNC machines easy to learn and easy to use. This document highlights the control features for Hurco 5-axis CNC machines (the ones that make the transition from traditional 2D programming to 5-sided programming intuitive).

NOTE: All of the 3-Axis control features are included on our 5-Axis Machines. Download the [3-Axis Control Features packet](#) to learn about these additional features.



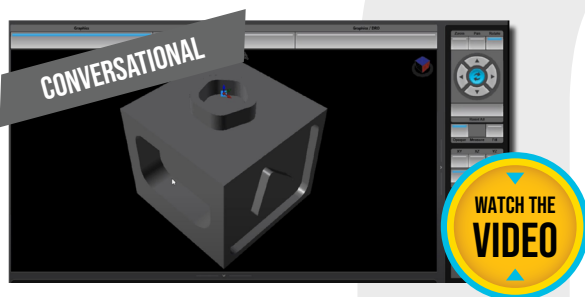
### TRANSFORM PLANE

Description: 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.

### 3D IMPORT

3D Import includes two important features for programming 5-sided parts: 3D DXF and Solid Model Import.



### SOLID MODEL IMPORT

With Solid Model Import you can take a customer's 3D solid model (.stp file) and import it into the control. For 5-sided programming, Solid Model Import is extremely powerful because it automatically creates the transform planes. The Transform Planes are the foundation of how Hurco makes 5-sided programming easy.

For all Hurco CNC machines, the ability to import stp files and select specific features to be machined significantly increases productivity. For example, if you select the floor of a pocket, and then identify the top of the pocket. The pocket depth will automatically be pulled from the model. Then you just pick a tool, verify your speeds and feeds, and you're ready to start making chips.

### Benefits

- » Simplifies programming. Saves time. Preserves accuracy.
- » No need to enter feature dimensions – simply click and cut
- » Transform Planes are created automatically for easy 5-sided conversational programming – no data entry required



### 3D DXF

3D DXF displays all the geometry the CAD system outputs, including splines. For example, you can select the bottom of a contour, and the cutting geometry and Z-axis depths will be automatically input into the conversational block.

### MILL SURFACE BLOCK

Conversational programming has always been a great choice for getting jobs up and running quickly, and for very efficient programming of 2D and 2.5D contours and features. However, machining actual 3-dimensional surfaces required the use of a CAD/CAM software package...that is, until now!

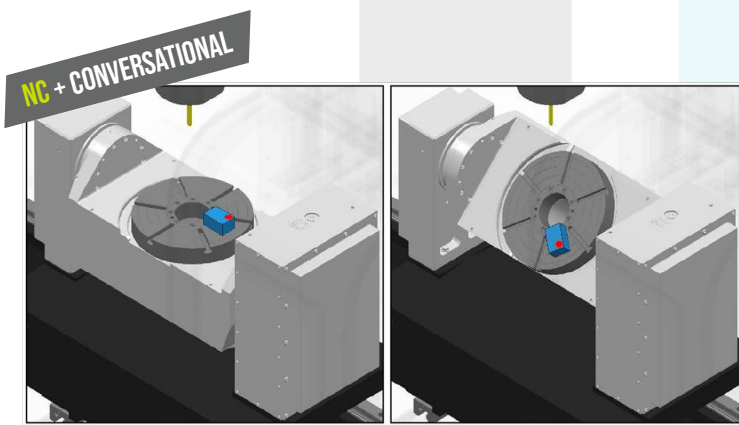
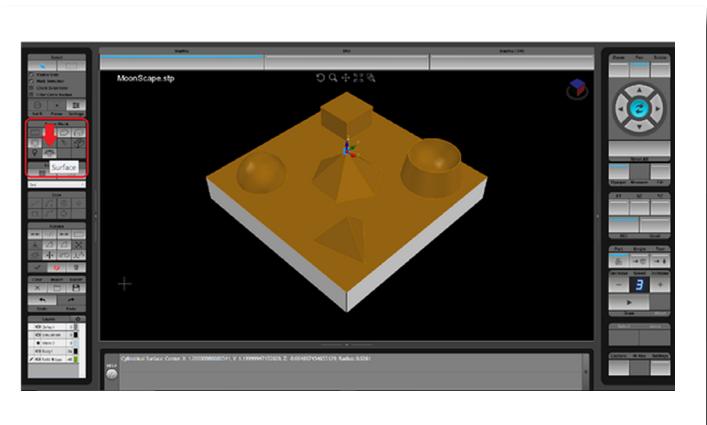
The Mill Surface feature of 3D IMPORT allows the user to program full 3-axis simultaneous toolpaths conversationally, by selecting individual surfaces on a solid model. The conversational block is created automatically, and once the block has been created there are CAD/CAM type of toolpath controls within the block for cutting strategies, stepdown, containment, etc...

### TOOL CENTER POINT MANAGEMENT (TCPM)

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).



NC ONLY

| TOOL SETUP - NC TOOL OFFSETS |        |         |                 |           |        |
|------------------------------|--------|---------|-----------------|-----------|--------|
| MACHINE                      | PART   |         | SPINDLE         | PAGE UP   |        |
| X                            | 0.0000 | -0.0000 | 0               | P1        |        |
| Y                            | 0.0000 | -0.0000 | FEED (STOPPED)  | PAGE DOWN |        |
| Z                            | 0.0000 | 0.0000  | TOOL IN SPINDLE | P2        |        |
| 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.

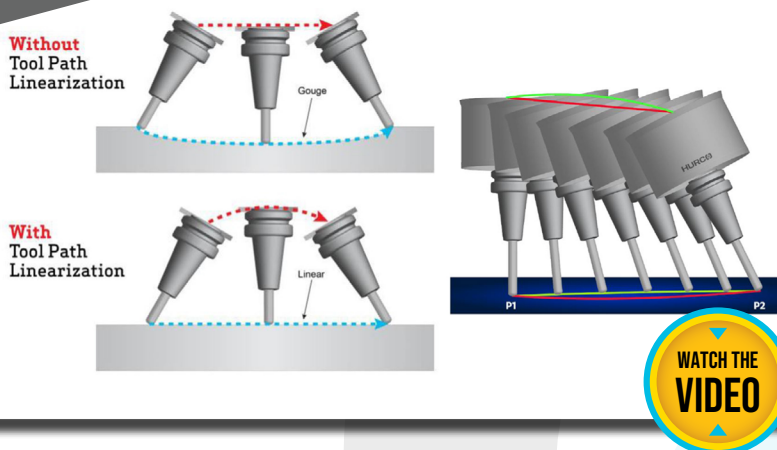
### 3D TOOL COMPENSATION

Description: Eliminates the need to re-post the program when you need to adjust the tool diameter for tool wear or tool substitution when cutting 3D geometry.

#### Benefits

- » Compensates for tool wear.
- » Eliminates idle time caused by tool breakage.
- » Provides flexibility and freedom for tool selection.
- » Supports both ballnose and bullnose tools.

NC ONLY



### TOOL PATH LINEARIZATION

Description: Tool Path Linearization eliminates gouging of the workpiece and removes the looped line segments that may appear on the part due to the XYZBC or AC moves that a CAM system outputs.

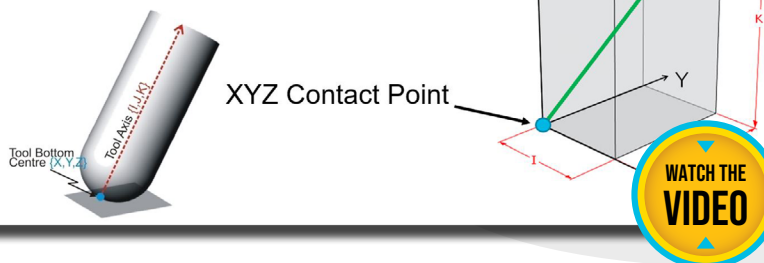
#### Benefits

- » Improves surface finish quality.
- » Generates smaller NC programs.

NC ONLY

### Tool Vector

Tool tilt angle & direction away from surface contact point  
G01 X10. Y10. Z10. I0.5 J0.5 K0.707106



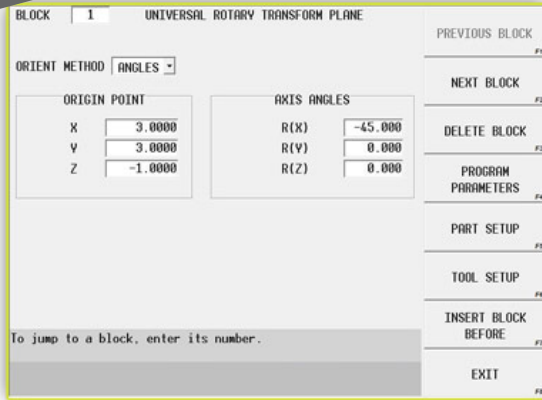
### TOOL VECTOR INPUT

Description: Tool Vector Input allows the control to compute machine angles and positions from a Contact Point, which allows it to calculate the angle the tool tilts in a uniform manner.

#### Benefits

- » Scheduling flexibility [the program can run on any type of 5-axis machine].
- » Simplified post processor.

CONVERSATIONAL ONLY

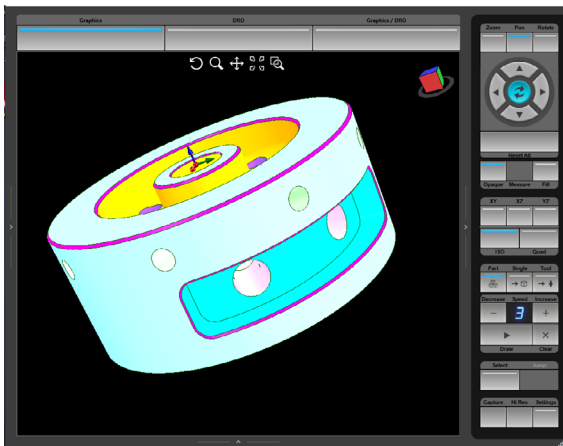


### UNIVERSAL ROTARY

Description: Universal Rotary allows a part program to be used interchangeably on all types of Hurco 5-axis machining centers even if 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.

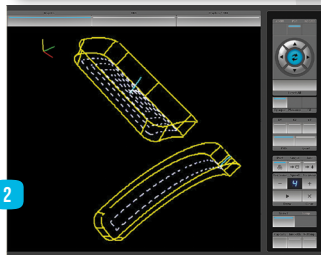
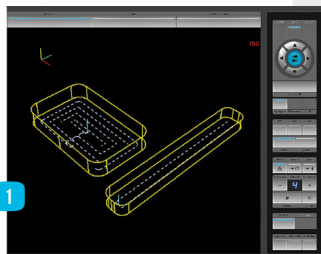
#### Benefit

- » Scheduling flexibility.



### ROTARY PROGRAMMING

WinMax® Rotary Milling allows the user to program part features that wrap around a cylindrical workpiece. Features such as rotary frames, rotary circles, rotary polygons, rotary slots, and rotary lines and arcs contours. Standard Holes features with rotary locations can also be easily programmed using the simple conversational rotary menus.



### 4TH AXIS ROTARY WRAP

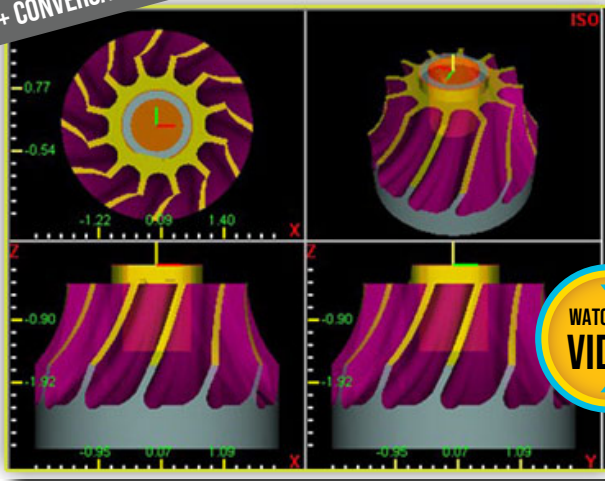
The Hurco 4th Axis Rotary Wrap works like a quick rotary function should work.

1. Simply create the 2D feature, select the Convert to Rotary button on the control's touch-screen menu, and enter the radius of the cylinder.
2. The new verification graphics appear on the screen.

Because we make it easy to convert linear to rotary and back to linear again, you have complete flexibility to quickly make changes to the feature without having to start all over again.

Additionally, 4th Axis Rotary Wrap supports multiple machining strategies, such as pocketing, islands, rotary pattern functions, holes, and tool cutter compensation.

NC + CONVERSATIONAL



### ADVANCED VERIFICATION GRAPHICS WITH 3D SOLID RENDERING

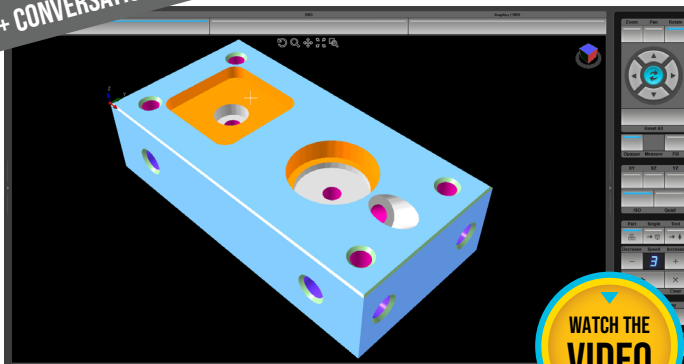
★ ★ ★ CUSTOMER  
FAVORITE

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. You can also view the part in transparent or opaque.

#### Benefits:

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

NC + CONVERSATIONAL



### DATA BLOCK SEARCH

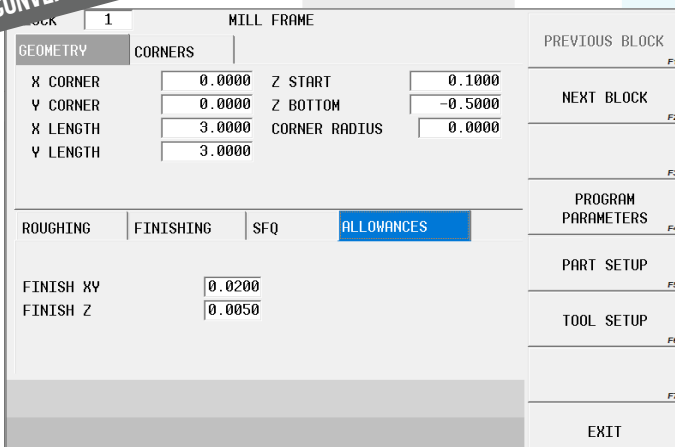
★ ★ ★ CUSTOMER  
FAVORITE

On the graphics screen, touch the feature you need to edit on the graphics screen, and the cursor will jump to the corresponding program section.

#### Benefits:

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

CONVERSATIONAL

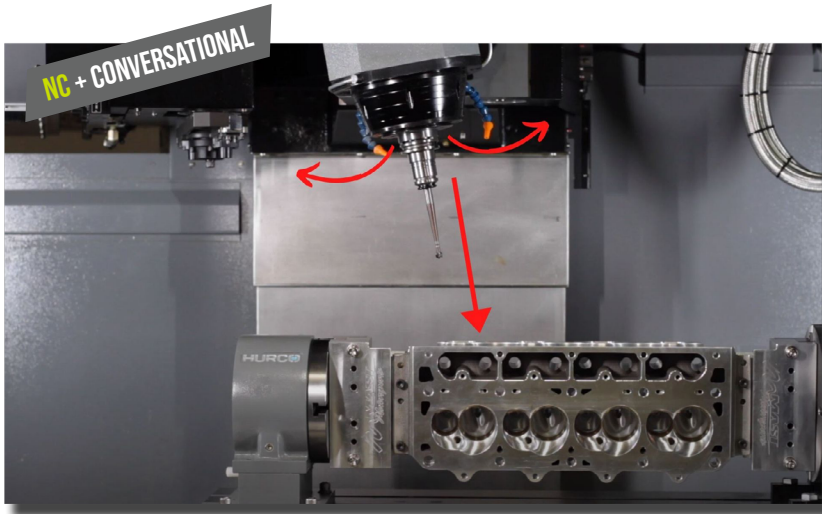


### ROUGHING STOCK ALLOWANCE

Description: The "Allowances" tab lets you leave a specific amount of material for finishing operations (no need to alter the tool diameter or the geometry data of the feature).

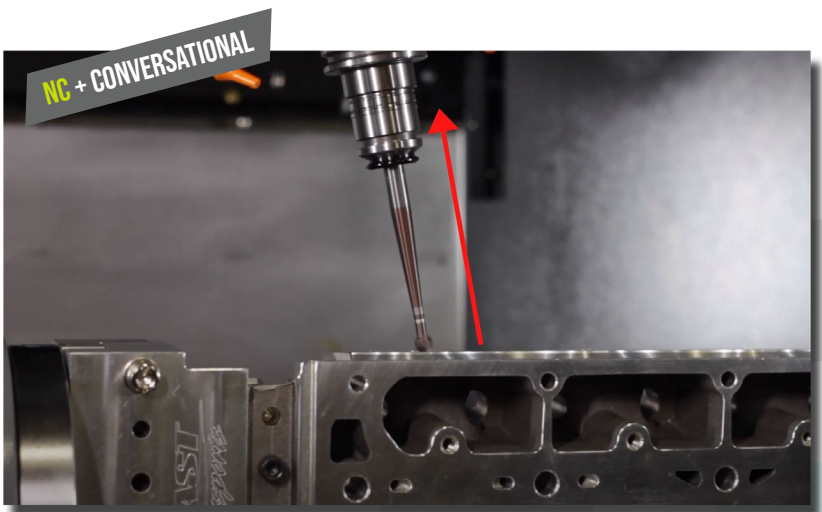
When you enter a roughing allowance and don't use a finish tool, the stock will remain without being machined away.





### AUTOMATIC SAFE REPOSITIONING

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



### TOOL VECTOR RETRACT

The tool retracts along its current angle.

- » Eliminates the need to program retract clearance points.
- » Lets you designate an incremental retract distance.

### 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. This feature is useful when machining parts with lots of holes at different angles.

CONVERSATIONAL ONLY

MILL FRAME

| GEOMETRY                        | CORNERS | PREVIOUS BLOCK |
|---------------------------------|---------|----------------|
| 1 ARC RADIUS 0.1250             |         | F1             |
| 2 ARC RADIUS 0.2500             |         | F2             |
| 3 LINE LENGTH 0.7500 ANGLE 45.0 |         | F3             |
| 4 ARC RADIUS 0.3750             |         | F4             |

ROUGHING FINISHING SFQ ALLOWANCES

TOOL 133 END MILL, dia. 0.3750

MILLING TYPE OUTSIDE

ENABLE BLEND MOVES YES

MILL FEED 91.6 PECK DEPTH 0.1000

SPEED (RPM) 6116 PLUNGE FEED 10.0

To jump to a block, enter its number.

PRESS MANUAL MODE, POWER, AND START CYCLE TO RESTORE POWER.

PROGRAM PARAMETERS PART SETUP TOOL SETUP INSERT BLOCK BEFORE EXIT

### ENHANCED CORNER GEOMETRY

The Enhanced Corner Geometry tab (in the Mill Frame data block) lets you quickly program chamfers or unique corners with different radii. Simply select Line or Arc for each corner, and specify the correct data for the feature.

NC + CONVERSATIONAL

PATTERN LOOP LINEAR

WinMax Mill Help v6.0 July 2012

Contents Index Search Favorites

Type in the keyword to find

3D Linear  
24 station  
3D Arc Data Block  
3D circular interpolation, G02.4 and G03.4  
3D Mold  
Blend Arc  
Contour  
Line  
Parameters  
3D tool geometry compensation, G41.2  
Sawtooth linear interpolation, G43.4  
A-Angle field  
Rotary Lines and Arcs Start Segment  
Rotary Position block  
R-axis  
clamp, M32  
unclamp, M33  
A-Center field  
Rotary Circle block

Loop Linear

This routine repeats a pattern a specified number of times along a line defined in the X-Y plane. Even though the defined line of this pattern is not parallel to the X or Y axes, the original pattern is always milled at its programmed location and orientation does not change with respect to the X and Y axes. Always program a Pattern End data block following a Loop Linear block.

The the  
PRES



### INTERACTIVE HELP SYSTEM

Our help system 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.

Additionally, our robust Interactive Help System includes pictures, diagrams, and videos when helpful.

BONUS: The help system is dynamic – it automatically updates whenever you update your control software.

CONVERSATIONAL ONLY

FEED & SPEED OPTIMIZATION

| AXIS | PART    | CHIP REMOVAL    | STOPPED | SET SPEED |
|------|---------|-----------------|---------|-----------|
| X    | -0.0000 | TOOL IN SPINDLE | 0       | F1        |
| Y    | -0.0000 | SPINDLE         | 0       | F2        |
| Z    | 0.0000  | FEED (STOPPED)  | 0.0     | F3        |

PROGRAM RUN TIME 0:00:00

F(x)|100 R(x)|37 S(x)|100

AXIS LIMIT SWITCHES:

SPINDLE LOAD MONITOR

BLOCK 0

PART COUNT 155

150%  
100%  
50%  
0%

WATCH THE VIDEO

CHIP REMOVAL FORWARD ON/OFF F4

CHIP REMOVAL REVERSE F5

MRS COOLANT F6

TOGGLE OVERRIDE EN F7

WORKLIGHT ON/OFF F8

### FEED & SPEED OPTIMIZATION

The Feed & Speed Optimization menu 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.



### SELECT SURFACE 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 part program.

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

NC + CONVERSATIONAL

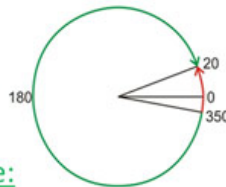
### Shortest Angular Traverse

- 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, which saves time (and frustration of witnessing inefficiency in action)!

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.

### NOTES:

---

---

---

---

---

---

---

---

---

---





### AUTO 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.

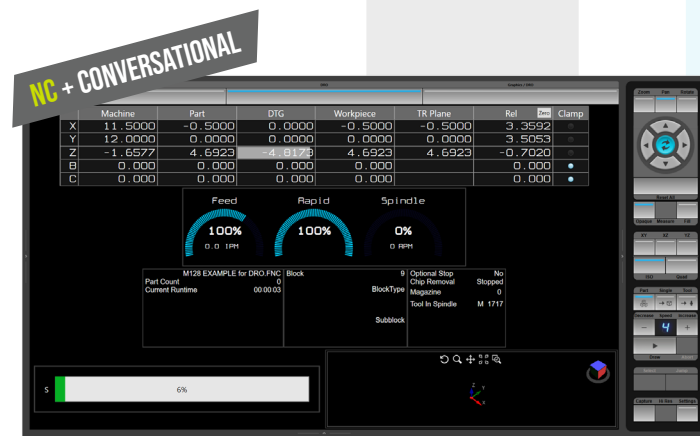


### ESTIMATED RUN TIME

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.
- » Bonus: Automatically checks for errors.



### RELATIVE POSITION DRO (DIGITAL READOUT)

Description: The Relative Position DRO gives you the power and flexibility to display the data that is relevant. Different circumstances call for different data to be displayed. Therefore, the DRO screen that is visible can be selected by the operator by pressing one of the three tabs at the top of the graphics screen. The DRO display also has a “relative” display that can be zeroed-out whenever needed by simply touching one button.

NC + CONVERSATIONAL

PART AND TOOL SETTINGS

GENERAL | MATCHING | LIFE MONITORING

ENABLE TOOL LIFE MONITORING  YES

MAX TOOL CUTTING TIME EXCEEDED

DISPLAY WARNING MESSAGE  YES

MARK TOOL DEFECTIVE  YES

RESET CUTTING TIME ON TOOL DATA CHANGE

TYPE  AUTOMATIC

DIAMETER +/- 0.0025  PROMPT

LENGTH +/- 0.0030  PROMPT

EXIT

### TOOL LIFE MANAGEMENT

The ability to manage tools by either tolerance or cut time is a real benefit for long running parts or for lights-out machining.

If a tool is determined to be “defective” because it has reached the end of its machining life, you simply program a sister-tool to automatically take its place whenever the original tool is called in the program. The sister-tool can also be managed by time or tolerance, and have its own sister-tool if needed.

NC + CONVERSATIONAL

| LINE | COORD | VALUE  | PART | COORD | VALUE  |
|------|-------|--------|------|-------|--------|
| X    |       | 0.0000 | B    |       | 0.000  |
| Y    |       | 0.0000 | C    |       | 0.000  |
| Z    |       | 0.0000 |      |       | -0.000 |

TOOL IN SPINDLE 0

TOOL 1 END MILL, dia. 0.7500

EDIT PARAMETERS  TOOL CAL LENGTH 0.0000

MAIN | LENGTH | DIAMETER

LENGTH OFFSET X 0.0000

LENGTH OFFSET Y 0.0000

STORE RESULT AS BOTH

OFFSET NUMBER BOTH

SISTER TOOL 0

Select the destination of the probed value.

PRESS MANUAL MODE, POWER, AND START CYCLE TO RESTORE POWER.

TOOL LENGTH

TOOL LENGTH OFFSET

BOTH

EXIT

### MULTIPLE PROBING RESULTS

When probing tools, the operator chooses where the results are stored: on the Tool Setup page, in the Tool Length and Diameter Table, or BOTH.

#### Benefits

- » Peace of mind to know the results are being utilized where necessary and for any programming convention (Conversational or NC).
- » Speed
- » Efficiency (Eliminates Redundancy)

NC + CONVERSATIONAL

ROTARY AXES PARAMETERS

A AXIS

ROTARY CENTERLINE Y 10.2355

ROTARY CENTERLINE Z -23.7796

AUTOMATIC CENTERLINE CALCULATION  YES

MACHINE COORDINATE RELATIVE  YES

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.
- » Identifies rotary axis alignment issues.
- » Easy to set up and execute.

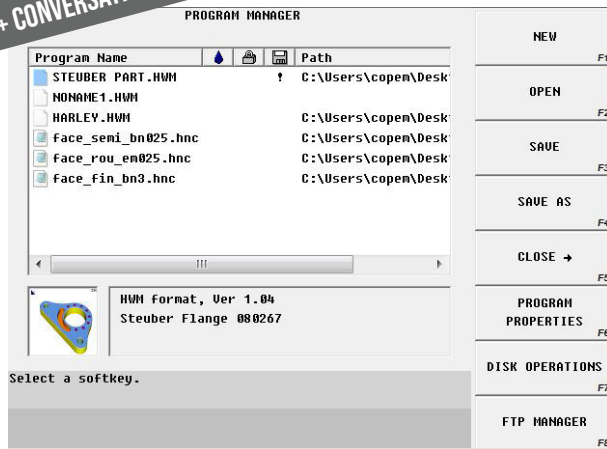
NC + CONVERSATIONAL



### 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 (no need to add additional code or cut air).

NC + CONVERSATIONAL

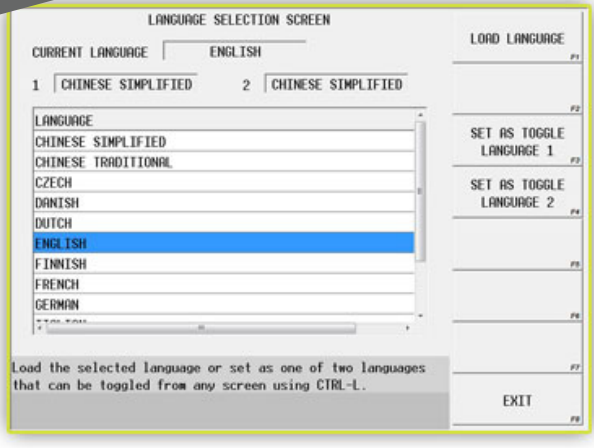


### EXTENSIVE PROGRAM MANAGER

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

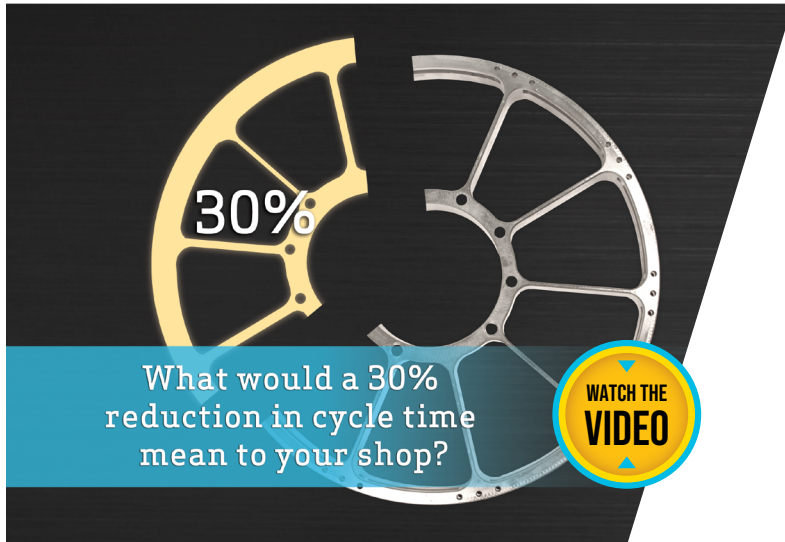
- » Part Program Preview includes a thumbnail graphic of the part program.
- » Extended file naming—no weird nomenclature required.
- » Ability to write protect part programs, add notes, and define material.

NC + CONVERSATIONAL



### FOREIGN LANGUAGE TOGGLE

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.

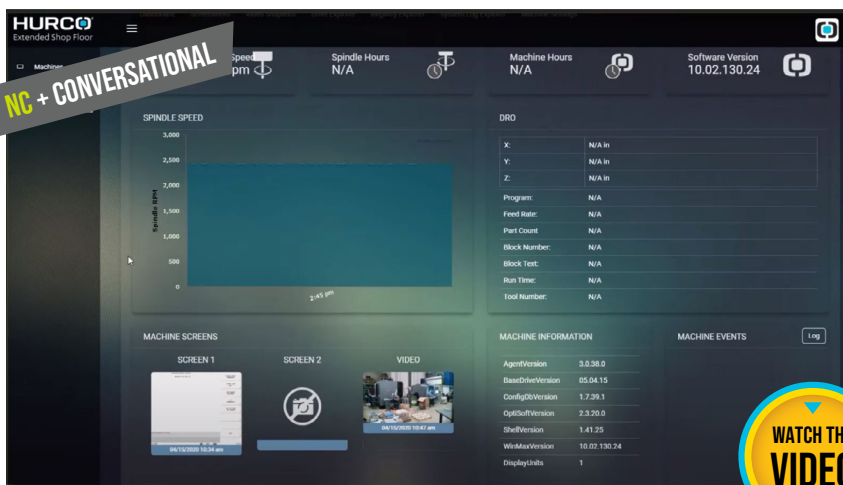


### ULTIMOTION®

UltiMotion® is an entirely new way to approach motion planning. UltiMotion is a tighter, smoother, control system that can dramatically improve cycle times and surface finish through better handling of the machine mechanics and dynamics.

#### Benefits

- » Cornering velocity is 2.5 times faster than conventional motion
- » Machine jerk is reduced by 50%
- » Look-ahead is dynamic and optimized instead of a fixed number of blocks that is used in traditional motion systems



### ULTIMONITOR

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

### NOTES:

---

---

---

---

---

---

---

---

---

---



### CUSTOMER FAVORITES

- AdaptiPath™
- Dynamic Variable Look Ahead capable of up to 10,000 blocks (with UltiMotion®)
- Advanced Verification Graphics with Data Block Search
- Estimated Run Time
- Inch/Metric Toggle
- UltiMotion®
- Mill Frame with Enhanced Corner Geometry
- NC Work Offsets with Conversational Programming
- Relative Position DRO
- Roughing Stock Allowance
- WinMax Desktop Complete



### PROGRAMMING

- 3D Import
- 3D DXF
- DXF Scaling
- 4th Axis Rotary Wrap
- 99 Work Offsets (NC)
- 99 Tool Offsets (NC)
- AutoCalc
- Autosave
- Blend Arcs
- Canned Cycle Blocks
- Chamfers
- Comment Block
- Concurrent Programming
- Context Sensitive Help
- Drill/Dwell Cycles (Chip Break + Peck)
- Drilling and Boring with Inserted Drill
- DXF Transfer
- Frame Mill
- Helical Interpolation
- Hole Operations – Tap, Drill, Center Drill, Dwell and Ream
- Inch/Metric Programming
- Indexer Routine
- Industry Standard NC (ISNC)
- Language Toggle
- Lines & Arcs
- Linear Repeat
- Mirror Image
- M-Code Auxiliary Functions
- NC/Conversational Merge
- NC Editor
- NC Macro Package (NCMP)
- NC Probing Cycles
- NC Productivity Package (NCPP)
- Pattern (Scaling, Rotation, Translation)
- On-screen User Prompt
- Optional Stop

- Parts Counter
- Part Zero Shift
- Peck Mill
- Program Manager Functions
- Program Parameters
- Program Review with Cut/Copy/Paste
- Programmable Safety Zones
- Rectangular Repeat
- Rigid Tap
- Roughing Stock Allowance
- Select Surface Finish Quality (SFQ)
- Serial Number (Letter & Part Serialization)
- Slots
- Speed and Feed Edit while Running
- Swept Surface with 3D Mold
- Thread Milling
- Tool Setup and Review with Graphics
- Tool & Material Library
- Tool Change Optimization
- TrueType® Lettering Package
- UltiPocket with Helical Ramp Entry

### VERIFICATION

- 3D Solid Rendering
- Automatic Error Check
- Graphics Display (Tool Path, Solids, Projection in 3 Planes, Isometric)
- Graphics Error Verification
- Graphics Scaling
- Graphics Zoom
- Real Time Tool Simulation
- Wire frame graphics of part geometry with zoom. Includes error verification

### OPERATIONAL

- 128GB Solid State Hard Drive
- 2.7 GHz Dual Core Processor
- 4GB RAM Memory
- Fast Draw Graphics Engine
- Font Magnification
- Job List
- Multiple Options to Store Tool
- Probing Results
- Stick Lettering
- Tool Life Management
- TrueType® Fonts Along a Contour
- User Assigned Tool Pockets
- Auto Interrupt Cycle
- Automatic Tool Home
- Control and Machine Diagnostics
- Coolant Select (Dual)
- Distance To Go
- Feed Hold Button
- Feedrate Override
- Spindle Load Monitor
- UltiMonitor®

### 5-AXIS

- 3D Tool Compensation
- Automatic Safe Repositioning
- Rotary Axes Centerline Probing
- Shortest Angular Traverse
- Tool Center Point Management
- Tool Path Linearization
- Transform Plane
- Tool Vector Canned Cycles
- Tool Vector Input & Retract
- Universal Rotary