

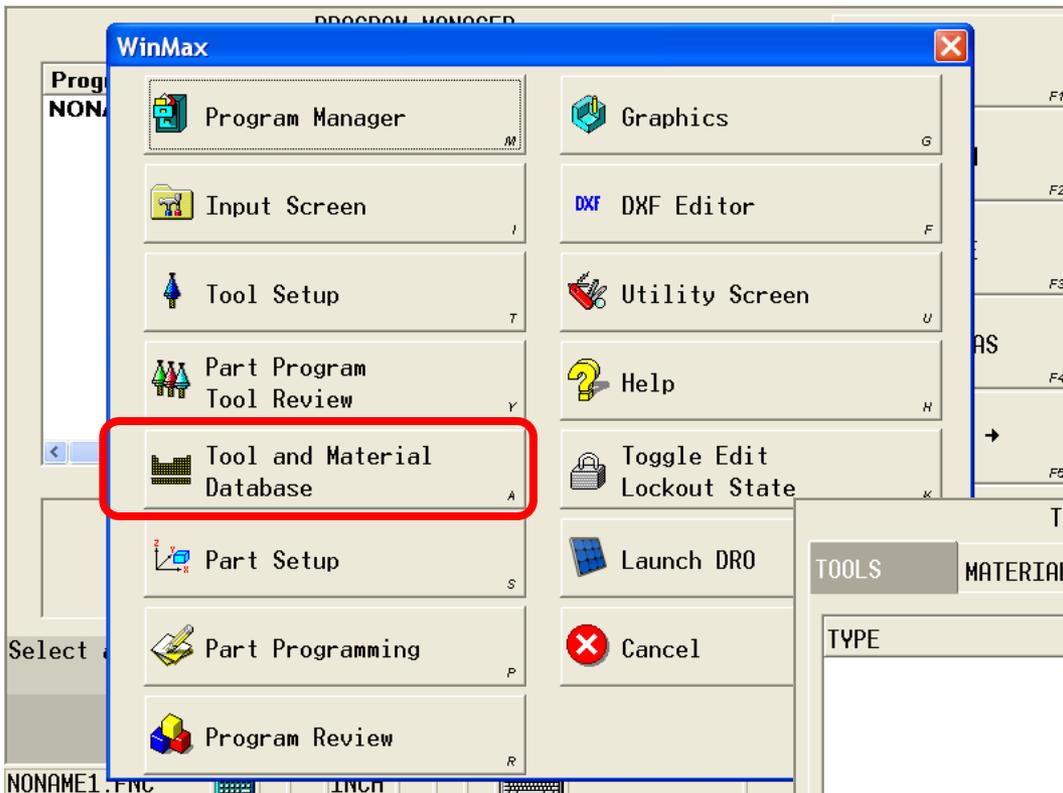
# Tool and Material Data Base

## Quick Guide / Suggested Use

2/6/2015

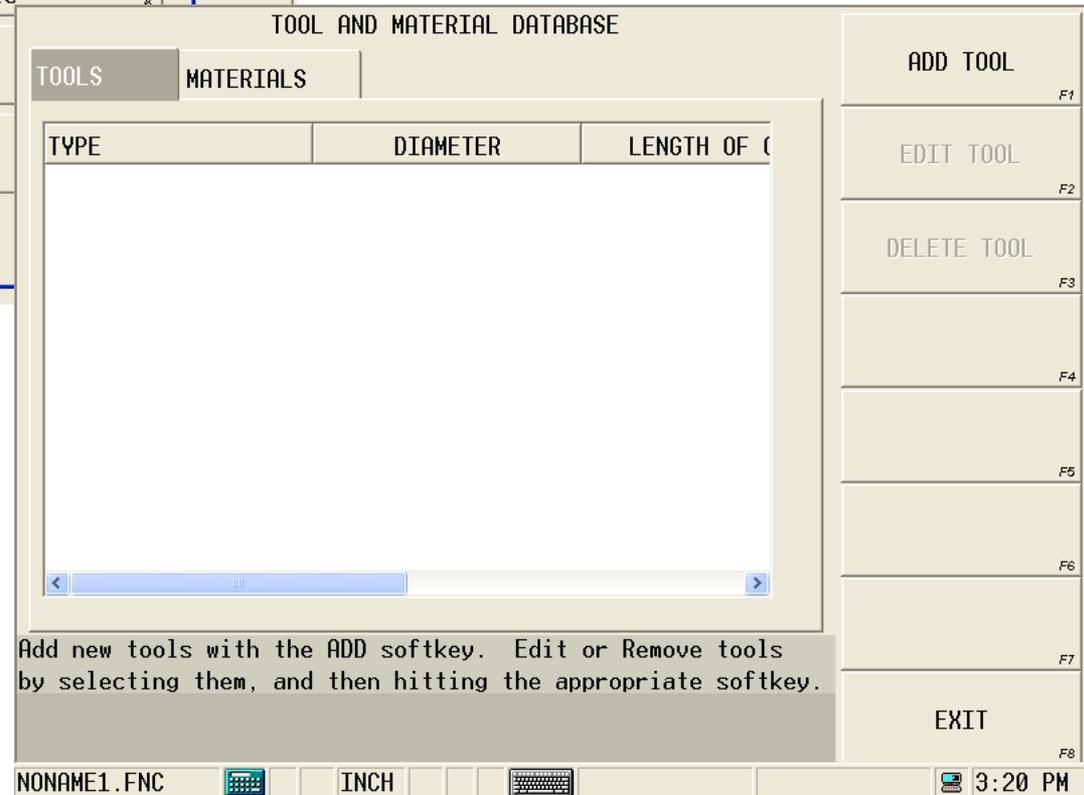
The logo for HURCO, featuring the word "HURCO" in a bold, white, sans-serif font. The letter "O" is stylized with a blue square in the center. The logo is set against a black background that tapers to the right.

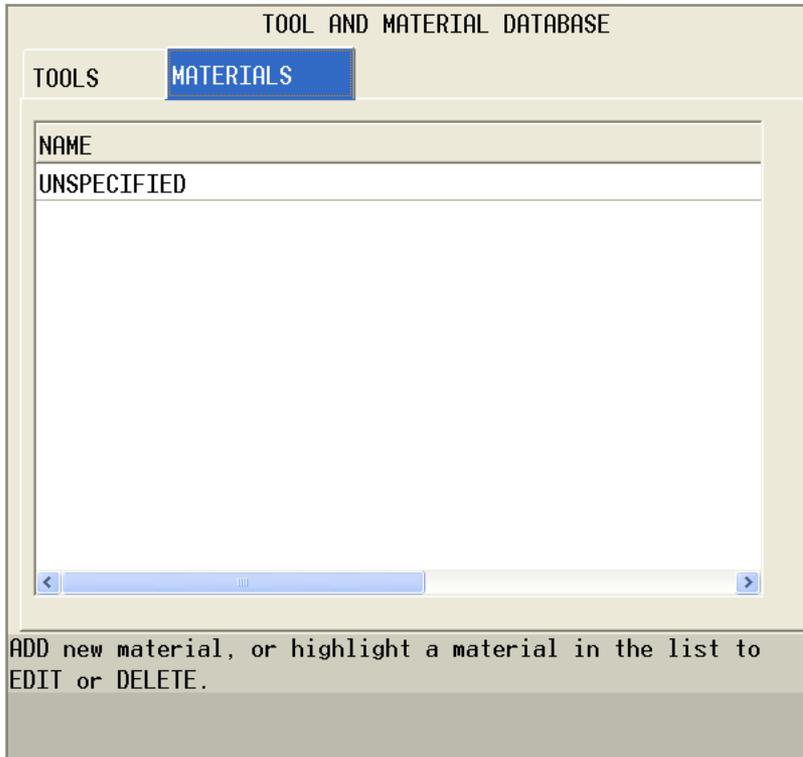
Applications  
*(317) 614-1549*  
Applications@hurco.com



You will find the Tool and material Database by pressing the **Auxiliary** button on your WinMax Control.

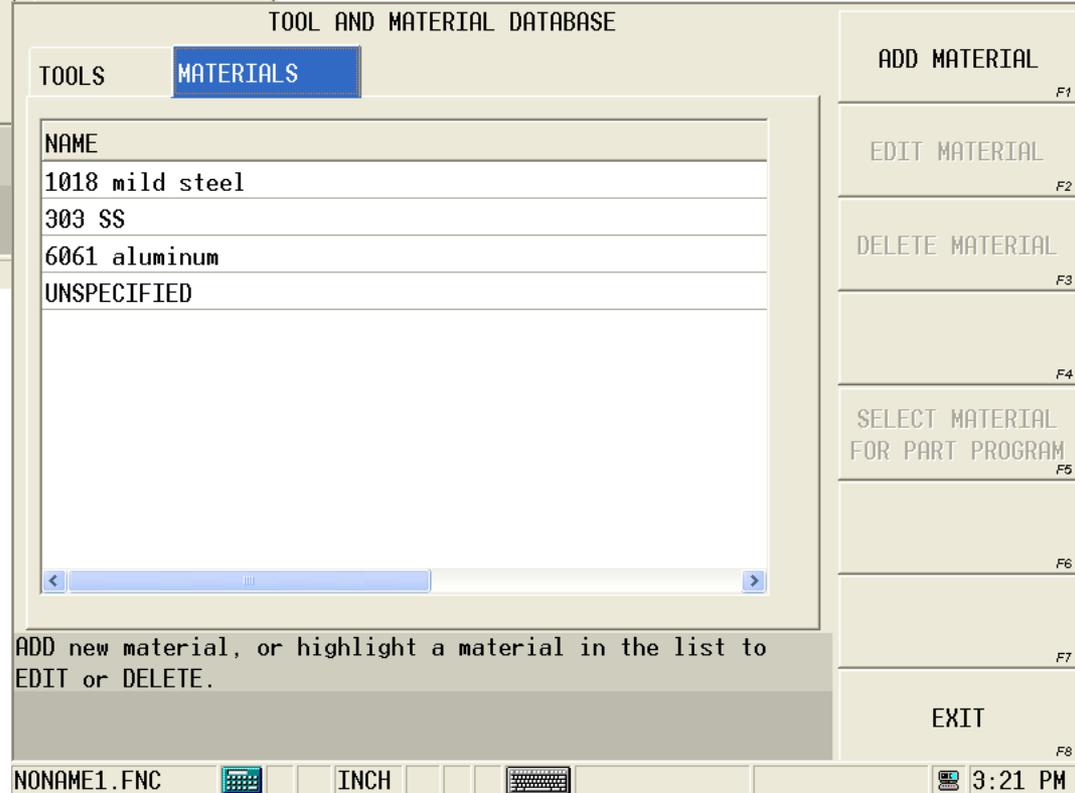
After selecting the Tool and Material Database, you will be taken to a screen similar to below.

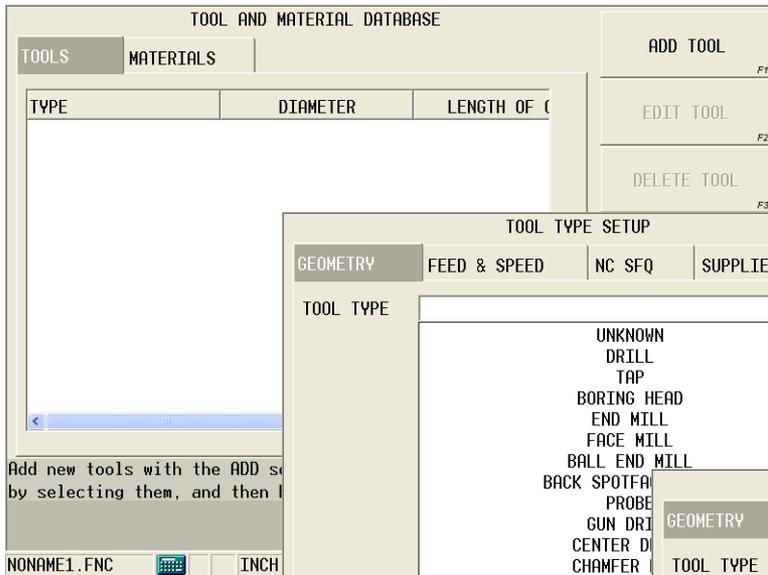




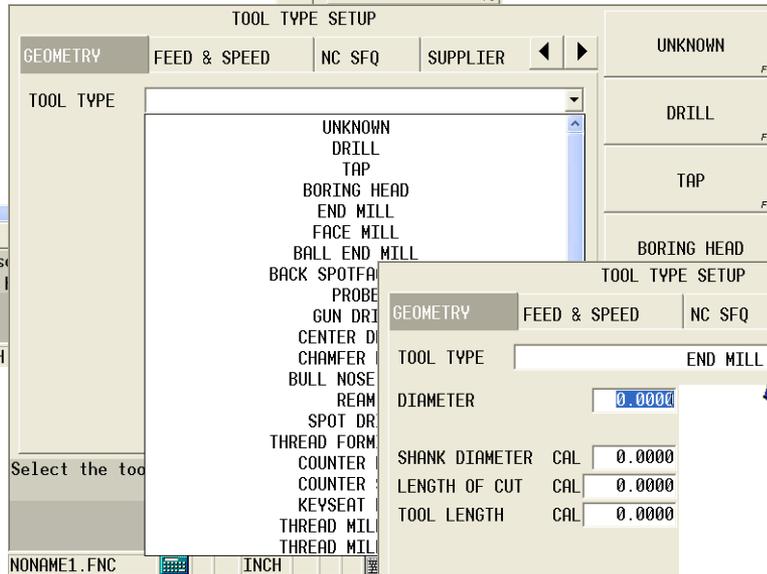
The first step is to setup all the materials that you will typically run in the machine.

To do this select the Materials tab, and select Add Material. You will be just adding the name of the material at this point

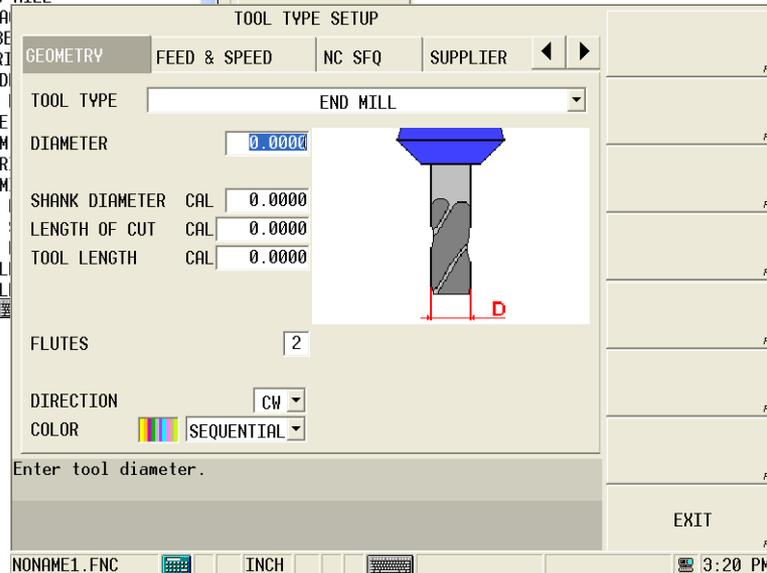




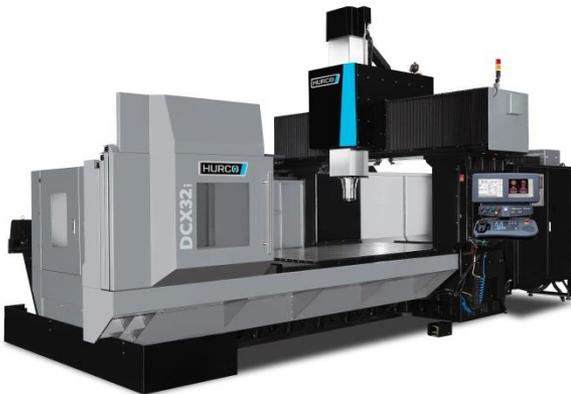
Now go back to the Tool and Material Database, and select the Tools tab, and Add Tool.



Select the tool type you want to create a template for.



To make things easy on yourself, leave the diameter blank. This will be set when we create an actual tool from this template.



*In this step we are just creating a **template**, that will later get a Surface Speed and Chip Load added to it. When we create an actual tool based off the template, we will give the tool an actual diameter. Once a tool is created, the diameter will then be used to calculate the actual speeds and feeds.*

TOOL TYPE SETUP

GEOMETRY FEED & SPEED NC SFQ SUPPLIER

TOOL END MILL, dia. 0.0000

WORK MATERIAL  
 UNSPECIFIED  
 6061 aluminum  
 303 SS

ROUGHING FINISHING

SURFACE SPEED 1000 SPEED 0  
 CHIPLOAD 0.004000 FEED CAL 0.0  
 MAX DEPTH 0.0000 PECK DEPTH 0.0000  
 COOLANT PRIMARY PLUNGE FEED 0.0

ROUGHING F1  
 FINISHING F2  
 F3  
 F4  
 F5

Once you have your tool type selected, go into the Feed & Speed tab.

Fill in the Surface Speed and Chipload for each material that you previously setup

Select the operation the parameters are applicable to.

NONAME1.FNC INCH



TOOL TYPE SETUP

GEOMETRY FEED & SPEED NC SFQ SUPPLIER

TOOL END MILL, dia. 0.0000

WORK MATERIAL  
 303 SS  
 1018 mild steel

ROUGHING FINISHING

SURFACE SPEED 400 SPEED  
 CHIPLOAD 0.003000 FEED CAL 0.0  
 MAX DEPTH 0.0000 PECK DEPTH 0.0000  
 COOLANT PRIMARY PLUNGE FEED 0.0

F1  
 F2  
 F3  
 F4  
 F5  
 F6  
 F7  
 F8  
 EXIT

Enter speed in RPM.

NONAME1.FNC INCH 3:22 PM

TOOL TYPE SETUP

GEOMETRY FEED & SPEED NC SFQ SUPPLIER

TOOL TYPE DRILL

DIAMETER 0.0000

SHANK DIAMETER CAL 0.0000

LENGTH OF CUT CAL 0.0000

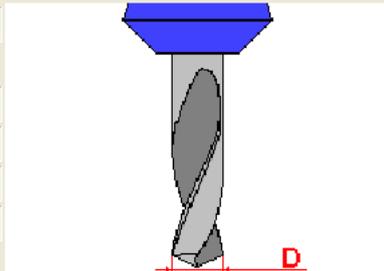
TOOL LENGTH CAL 0.0000

DRILL ANGLE 0.0

FLUTES 2

DIRECTION CW

COLOR SEQUENTIAL



F1  
F2  
F3  
F4  
F5

Create a Tool Template for each type of tool that you will run in the machine.

Enter tool diameter.

NONAME1.FNC INCH



TOOL TEMPLATE LIST

NO.	TYPE	DIAMETER	DESCRIPTION
	END MILL	0.0000	END MILL, dia. 0.0000
	DRILL	0.0000	DRILL, dia. 0.0000
	FACE MILL	0.0000	FACE MILL, dia. 0.0000
	TAP		TAP, M0 x 0
	BALL END MILL	0.0000	BALL END MILL, dia. 0.0000

CREATE TOOL USING TEMPLATE F1

EDIT TOOL TEMPLATE F2

CREATE NEW TOOL F3

UP F4

DOWN F5

EXIT F6

Create a tool with defaults copied from a selected tool template in the list, or create a new tool.

NONAME1.FNC INCH 3:24 PM

MANUAL

TOOL MANAGEMENT

To add a new tool based off the template, press Manual Mode on the control, then choose Tool Management.

Choose the Manual Tab, and select Insert Tool

Choose Create Tool Using Template

INSERT TOOL

CREATE TOOL USING TEMPLATE

MACHINE	PART	CHIP REMOVAL
X	9.5675	-0.9408
Y	7.9729	0.1679
Z	0.0000	20.2971
A (c)	0.000	-0.000
C (c)	0.000	-0.000

STOPPED	TOOL MANAGEMENT
0	F1
0	MANUAL FUNCTION SETUP
0.0	F2

SPINDLE AUTO MANUAL

NO.	TOOL
1	BALL END MILL, dia. 0.2362
2	BALL END MILL, dia. 0.3750
3	END MILL, dia. 0.5000
4	TAPER RADIUS END MILL, dia. 0.3750
5	BALL END MILL, dia. 0.2500
6	BALL END MILL, dia. 0.1250
7	FACE MILL, dia.
13	END MILL, dia.

MOVE TOOL TO SPINDLE

INSERT TOOL

TOOL TEMPLATE LIST

NO.	TYPE	DIAMETER	DESCRIPTION
	BALL END MILL	0.0000	BALL END MILL, dia. 0.0000
	TAP		TAP, M0 x 0
	FACE MILL	0.0000	FACE MILL, dia. 0.0000
	DRILL	0.0000	DRILL, dia. 0.0000
	END MILL	0.0000	END MILL, dia. 0.0000

EDIT TOOL TEMPLATE

CREATE NEW TOOL

UP

DOWN

EXIT

AXIS LIMIT SWITCHES:

NONAME1.FNC INCH

NONAME1.FNC

Create a tool with defaults copied from a selected tool template in the list, or create a new tool.  
EMERGENCY STOP HAS BEEN DEPRESSED.

NONAME1.HWM INCH

9:31 AM



TOOL SETUP

MACHINE	PART	A	B
X 9.5675	-0.9408	0.000	-0.000
Y 7.9729	0.1679	0.000	-0.000
Z 0.0000	20.2971	TOOL IN SPINDLE 0	

DELETE TOOL F1

PART SETUP F2

PART PROGRAMMING F3

TOOL OFFSETS F4

TOOL HOME F5

TOOL NUMBER 8 LOCATION Manual

TOOL TYPE END MILL

DIAMETER 0.0000 SPEED CW 0

COOLANT NONE

SURFACE SPEED 0

TOOL CAL LENGTH 0.0000 FEED/FLUTE 0.000000

TOUCH-OFF DEVICE 1 GAUGE FLUTES

FEED CUTTING TIME DIAMETER WEAR

Provide the tool's number.

NONAME1.FNC INCH

The control will create the tool, and give it the next available tool number it sees as being available.

Enter the Tool Diameter

TOOL SETUP

MACHINE	PART	A	B
X 9.5675	-0.9408	0.000	-0.000
Y 7.9729	0.1679	0.000	-0.000
Z 0.0000	20.2971	TOOL IN SPINDLE 0	

DELETE TOOL F1

PART SETUP F2

PART PROGRAMMING F3

TOOL OFFSETS F4

TOOL HOME F5

TOOL NUMBER 8 LOCATION Manual

TOOL TYPE END MILL

DIAMETER 0.7500 SPEED CW 0

COOLANT NONE

SURFACE SPEED 0

TOOL CAL LENGTH 0.0000 FEED/FLUTE 0.000000

TOUCH-OFF DEVICE 1 GAUGE FLUTES 2

FEED CAL 0.0 CUTTING TIME 0

DIAMETER WEAR 0.0000

Enter or store the tool calibration value.  
'P' designator indicates values set by probing.

NONAME1.FNC INCH 3:25 PM



To verify that the tool was created from the template, select More, and then choose Advanced Tool Settings.

If you select one of the materials, you should see the speeds and feeds have been calculated off the tool diameter entered

TOOL SETUP

MACHINE	PART	A		
X 9.5675	-0.9408	0.000	-0.000	
Y 7.9729	0.1679	0.000	-0.000	
Z 0.0000	20.2971	TOOL IN SPINDLE	0	

TOOL NUMBER 8 LOCATION Manual

TOOL TYPE END MILL

DIAMETER 0.7500 SPEED CW 0

COOLANT NONE

SURFACE SPEED 0

TOOL CAL LENGTH 0.0000 FEED/FLUTE 0.000000

TOUCH-OFF DEVICE I GAUGE FLUTES 2

FEED CAL 0.0

CUTTING TIME 0

DIAMETER WEAR 0.0000

DELETE TOOL F1

PART SETUP F2

PART PROGRAMMING F3

TOOL OFFSETS F4

TOOL HOME F5

SET LENGTH USING TOUCH-OFF DEVICE F7

MORE →

Enter or store the tool calibration value.  
'P' designator indicates values set by probing.

NONAME1.FNC INCH

TOOL SETUP

MACHINE	PART	A		
X 9.5675	-0.9408	0.000	-0.000	
Y 7.9729	0.1679	0.000	-0.000	
Z 0.0000	20.2971	TOOL IN SPINDLE	0	

TOOL NUMBER 8 LOCATION Manual

TOOL TYPE END MILL

DIAMETER 0.7500 SPEED CW 0

COOLANT NONE

SURFACE SPEED 0

TOOL CAL LENGTH 0.0000 FEED/FLUTE 0.000000

TOUCH-OFF DEVICE I GAUGE FLUTES 2

FEED CAL 0.0

CUTTING TIME 0

DIAMETER WEAR 0.0000

ADVANCED TOOL SETTINGS F1

CHANGE TOOL NUMBER F2

TOOL PROBING F3

PROGRAM PARAMETERS F4

PART PROGRAM TOOL REVIEW F5

Provide the tool's number.

NONAME1.FNC INCH

TOOL TYPE SETUP

GEOMETRY FEED & SPEED NC SFQ SUPPLIER

TOOL 8 END MILL, dia. 0.7500

WORK MATERIAL

UNSPECIFIED

6061 aluminum

202 SS

ROUGHING FINISHING

SURFACE SPEED 1000 SPEED CAL 5093

CHIPLOAD 0.004000 FEED CAL 40.7

MAX DEPTH 0.0000 PECK DEPTH 0.0000

COOLANT PRIMARY PLUNGE FEED 0.0

COPY DATA TO ALL MATERIALS F3

UP F6

DOWN F6

EXIT F8

Select the material that you want to associate with the parameters below.

NONAME1.HWM INCH 10:56 AM



TOOL SETUP

	MACHINE	PART			
X	9.5675	-0.9408	A	0.000	-0.000
Y	7.9729	0.1679	C	0.000	-0.000
Z	0.0000	20.2971	TOOL IN SPINDLE	0	

TOOL NUMBER	8	LOCATION	Manual
TOOL TYPE	END MILL		
DIAMETER	0.7500	SPEED	CW 0
		COOLANT	NONE
TOOL CAL LENGTH	0.0000	SURFACE SPEED	0
TOUCH-OFF DEVICE	1 GAUGE	FEED/FLUTE	0.000000
		FLUTES	2
		FEED	CAL
		CUTTING TIME	
		DIAMETER WEAR	0.0

Provide the tool's number.

- ADVANCED TOOL SETTINGS F1
- CHANGE TOOL NUMBER F2
- TOOL PROBING F3
- PROGRAM PARAMETERS F4
- PART PROGRAM TOOL REVIEW F5

If you wish to change the number assigned to the tool, choose Change Tool Number below Advanced Tool Settings.

You can choose if you want to change the tool number or copy the tool. Typically you want to Change Tool Number to prevent duplicate tools.

NONAME1.FNC INCH



CHANGE TOOL NUMBER

CURRENT TOOL DESCRIPTION

8 END MILL, dia. 0.7500

NEW TOOL NUMBER

8

Enter the new tool number and select the desired operation.

CHANGE TOOL NUMBER <small>F1</small>
COPY TOOL <small>F2</small>
<small>F3</small>
<small>F4</small>
<small>F5</small>
<small>F6</small>
<small>F7</small>
EXIT <small>F8</small>

NONAME1.FNC INCH 3:26 PM

You will repeat the previous steps for each tool that you want to create from the template. By using the template, you are able to enter the speeds and feeds for each material only once. Because the feeds and speeds were entered as a Surface Speed and Chip Load, the actual speeds and feeds will be calculated once the tool diameter is entered.

## Formulas

**Surface Ft. Per Minute** =  $(\text{RPM} \times 3.14 \times \text{Tool Dia.}) / 12$

**Chip Load** =  $\text{Feed in IMP} / (\text{RPM} \times \text{Number of Teeth})$

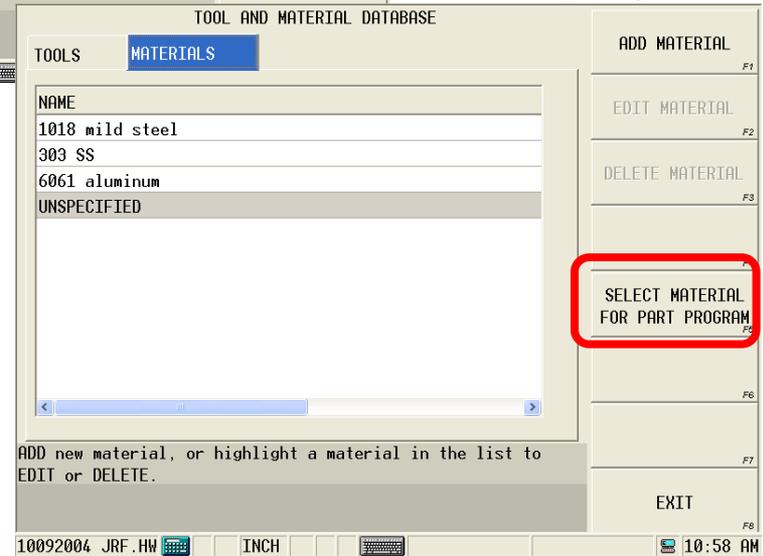
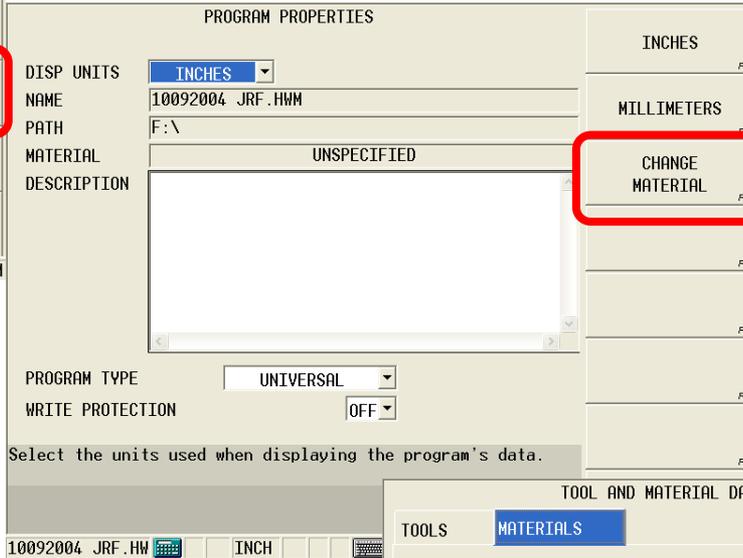
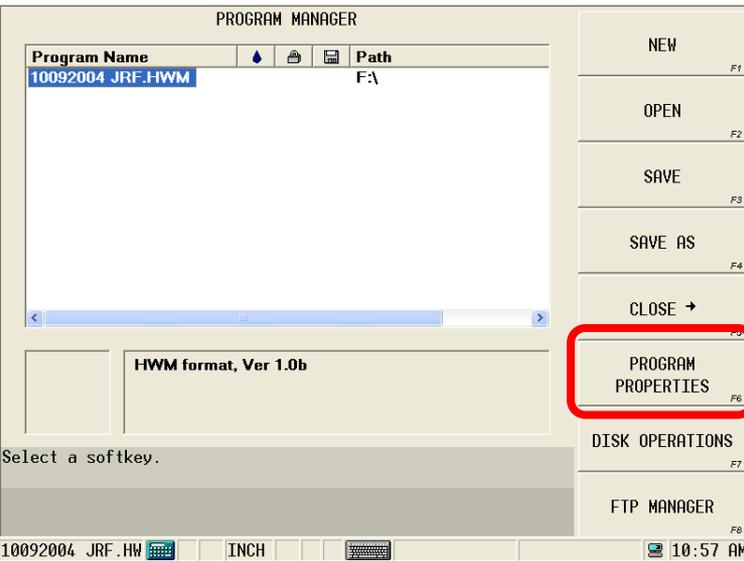
**Feed in IPM** =  $\text{Chip Load} \times \text{RPM} \times \text{Number of Teeth}$

**RPM** =  $(12 \times \text{SFPM}) / (3.14 \times \text{Tool Dia.})$



To select a material type for a program select the program in Program Manager. Then select Program Properties.

Select Change Material, highlight the material needed, and then choose Select Material for Part Program. Feeds and speed for that material will now be used in the program



If you have any questions about this or any other application please don't hesitate to contact us.

Prepared by

Jason Falk

[falkj@hurco.com](mailto:falkj@hurco.com)

Desk- 317-298-2614

Hurco Applications Department

[Applications@hurco.com](mailto:Applications@hurco.com)

317-614-1549

Thank You !

