



# Acorn Parameter Settings

## Parameter Settings for Centroid Acorn

These parameters are in the *Centroid M-Series Mill Operators Manual*. However, we have isolated them as a separate file, so they are easier for our customers to find.

## 15.3 Machine Parameters (F3–Parms from Configuration)

Machine Parameters P0 - P99									
000	-8.0000	020	72.0000	040	0.0010	060	0.0000	080	0.0000
001	0.0000	021	7.9778	041	0.0100	061	5.0000	081	-1.0000
002	0.0000	022	7.9778	042	0.0000	062	90.0000	082	500.0000
003	0.0000	023	7.9778	043	0.0000	063	128.0000	083	0.0500
004	9.0000	024	7.9778	044	0.0000	064	0.0000	084	3.0000
005	0.0000	025	3.3333	045	0.0000	065	1.0000	085	0.0000
006	0.0000	026	3.3333	046	2.0000	066	1.0000	086	0.0000
007	0.0000	027	3.3333	047	0.0000	067	1.0000	087	24.0000
008	2.0000	028	3.3333	048	0.1000	068	640.0000	088	24.0000
009	0.0000	029	212.0000	049	0.0000	069	1.7500	089	24.0000
010	0.0000	030	260.0000	050	0.0000	070	0.0010	090	24.0000
011	0.0000	031	0.0000	051	0.0000	071	0.0000	091	0.0000
012	10.0000	032	25.0000	052	0.0000	072	0.0000	092	0.0000
013	0.0500	033	1.0000	053	0.0000	073	0.0500	093	0.0000
014	12.0000	034	8000.0000	054	0.0000	074	4.0000	094	1.0000
015	5.0000	035	0.0000	055	0.0000	075	0.0000	095	2.0000
016	10.0000	036	0.0000	056	4.0000	076	0.0000	096	2.0000
017	0.0000	037	3.0000	057	0.0000	077	0.0000	097	2.0000
018	0.0000	038	0.0000	058	0.0000	078	0.0000	098	2.0000
019	0.0000	039	100.0000	059	0.0000	079	0.0000	099	1.0000

E-Stop PLC Bit Number

X Esc      Prev. Table F7      Next Table F8      Save F10

This screen provides you with a method of changing various parameters that are used by the control. Altogether, you have access to 500 parameters spread across 5 tables. Each table gives you access to 100 parameters at a time. You can navigate between tables using the following keys: F7-Previous Table and F8-Next Table. The title at the top tells you which table you are on. If you wish to change a field in the table, use the arrow keys to move the cursor and select the desired field. A short description of the parameter will appear below the table. Type the new value and press ENTER. When you are done editing the fields, press F10-Save to accept any changes you have made and save them. Note that F10-Save is a single operation that will save all changes in every table that you modified. Pressing ESC will discard all changes in every table that were modified and will return to the previous menu [Setup].

**NOTE:** Many machine parameters can also be set with the G10 G-code or by #variable assignment.

### 15.3.1 Bit-mapped parameters

Certain control parameters are defined by bit-mapped values. In order to change these parameters you must understand how bit mapping works. A bit-mapped parameter is stored as a number, representing a 16-bit value in the control. If a certain bit needs to be turned on, that bit's binary value must be added to the parameter value, if the bit needs to be turned off, its binary value must be subtracted from the parameter value. The values for each of the 16 bits can be seen in the table below.

Bit-Mapped Parameter Bits

Bit	0	1	2	3	4	5
Value	$2^0=1$	$2^1=2$	$2^2=4$	$2^3=8$	$2^4=16$	$2^5=32$

Bit	6	7	8	9	10	11
Value	$2^6=64$	$2^7=128$	$2^8=256$	$2^9=512$	$2^{10}=1024$	$2^{11}=2048$

Bit	12	13	14	15
Value	$2^{12}=4096$	$2^{13}=8192$	$2^{14}=16\ 384$	$2^{15}=32\ 768$

To set bit-mapped parameters simply add together the bit values that you need to have enabled.

**Example:**

Parameter Value	Bit number and setting							
	0	1	2	3	4	5	6	7
0	X	X	X	X	X	X	X	X
1	ON	X	X	X	X	X	X	X
$11 = 8 + 2 + 1$	ON	ON	X	ON	X	X	X	X
$24 = 16 + 8$	X	X	X	ON	ON	X	X	X

Parameter Value	Bit number and setting							
	8	9	10	11	12	13	14	15
0	X	X	X	X	X	X	X	X
1	X	X	X	X	X	X	X	X
$11 = 8 + 2 + 1$	X	X	X	X	X	X	X	X
$24 = 16 + 8$	X	X	X	X	X	X	X	X

The following parameters are currently defined:

Parameter	Definition	Default setting
0	E-Stop PLC Bit	0
1	Y jog key orientation	0
2	G-Code Interpretation Control and Slaving Rotary-axis feedrate	0
3	Modal Tool and Height Offset Control	0
4	Remote File Loading Flag & Advanced File Ops	0
5	Suppress Machine Home Setup	0
6	Auto Tool Changer Installed	0
7	Display Colors	0
8	Available Coolant System(s)	2
9	Display Language	0
10	Macro M-Function Control/Probe Stop Handling	0
11	Touch Probe PLC Input	15
12	Touch Probe Tool Number	0
13	Probing Recovery Distance	0.05" / 1.27 mm
14	Fast Probing Rate	10 ipm or 254 mm/min
15	Slow Probing Rate	1 ipm or 25.4 mm/min
16	Probing Search Distance	10" / 254 mm
17	Tool Detector Reference Number	0
18	PLC Input Spindle Inhibitor	0
19	MPG modes	0
20	Ambient Temperature	72° F / 22°C
21-24	Motor Heating Coefficients for axes 1,2,3,4	Refer to text
25-28	Motor Cooling Coefficients for axes 1,2,3,4	Refer to text
29	Warning Temperature	150°F / 65.5°C
30	Limit Temperature	180°F / 82°C
31	Legacy SPIN232 Com Port	0
32	Autonomous Digitizing Offset Files	27
33	Spindle Motor Gear Ratio	1
34	Spindle Encoder Counts/Rev	8,000
35	Spindle Encoder Axis Number	0
36	Rigid Tapping Enable/Disable	0

Parameter	Definition	Default setting
37	Spindle Deceleration Time	10
38	Multi-Axis Max Feedrate	0
39	Feedrate Override Knob Limit	120
40	Basic Jog Increment	0.0001" / .001 mm
41	Rotary-axis Jog Increment	.01°
42	Password for Configuration Menus	0
43	Automatic tool measurement options	0
44	TT1 PLC input number	0
45	WCS Lockout	0
46	Active G-Codes Display	0
48	Grid Digitize Patch Playback Z rapid clearance amount	.1
49-51	Small Arc Feedrate Limiting	0
56	Feedrate Override Display Properties	0
57	Use Generic Load Meter Data from PLC	0
61	High Power Stall Timeout	0.5
62	High Power Stall PID Limit	115
63	High Power Idle PID Multiplier	1.5
64	4th/5th Axis Pairing	0
65-67	Spindle Gear Ratios	1.0
68	Minimum Rigid Tapping Spindle Speed	0
69	Duration For Minimum Spindle Speed	1.0
70	Offset Library Inc/Decrement Amount	.001" / .02 mm
71	Part Setup Detector Height	0
72	Data Recording M-Function Options	0
73	Peck Drill Retract Amount	0.05
74	M-Function executed at bottom of tapping cycle	4
75	Axis Summing Control	0
76	Manual Input Unrestricted Distance	0
77	Manual Input Movement Tolerance	0
78	Spindle Speed Display and Operations	0
79	Paired Axes Re-sync Delay Time	0
80	Voltage Brake Applied Message Frequency	0
81	Air Drill M-Function (executed instead of Z movement in drilling)	-1
82	Spindle Drift Adjustment	0
83	Deep Hole Clearance Amount	0.05
84	M-Function executed at return to initial point of tapping cycle	3

Parameter	Definition	Default setting
85	"Door Open" Interlock PLC bit	0
86	Rapid/Linear vector rate limit	0
87-90	Autotune Ka Performance parameters for axes 1,2,3,4	0
91-94	Axis Properties for axes 1,2,3,4	0
95-98	Autotune / Auto Delay Move Distance for axes 1,2,3,4	2" / 50.8 mm
99	Cutter Compensation Look-ahead	6
100	Intercon comment generation	0
101	Intercon clearance amount	0.1
102	Intercon spindle coolant delay	3.0
103	Intercon corner federate override	50.0
104	Intercon modal line parameters	0
105	Intercon modal arc parameters	0
106	Intercon modal drilling cycle parameters	0
115	Intercon Help	0
116	A-Axis Y Coordinate	0
117	A-Axis Z Coordinate	0
118	B-Axis X Coordinate	0
119	B-Axis Z Coordinate	0
120	Probe stuck clearance amount	0.10" / 2.54 mm
121	Grid digitize prediction minimum Z pullback	0.002" / .0508 mm
122	Grid digitizing deadband move distance	0.0002" / .00508 mm
123	Radial digitizing clearance move	0
130	Z-axis on/off selection	0
131	4th axis on/off selection	0
132-135	Motor Heating Coefficients for axes 5,6,7,8	Refer to text
136	G76 Fine Bore Retract Angle	0
137	Load Meter Filter Size	0
138	DRO Encoder Deadband	0
139	Special Dwell between Moves	0
140	Message log priority level	1
141	Maximum message log lines	100
142	Message log trim amount	100
143	DRO properties (load meters, 4/5 digits, DTG)	0
144	Comparison rounding	0
145	Advanced macro properties (fast branching)	0
146	Feed hold threshold for feed rate override	0

Parameter	Definition	Default setting
147	Number of Messages in Operator Message Window	10
148	Miscellaneous Jogging Options	0
149	Spindle Speed Threshold	1.0
150	Backplot Graphics display options	0
151	Repeatability tolerance for probing	0
154	Touchscreen options	0
155	DSP Probe Installed	0
156-159	Autotune / Auto Delay Move Distance for axes 5,6,7,8	2" / 50.8 mm
160	Enhanced ATC	0
161	ATC Maximum Tool Bins	0
162	Intercon M6 Initial M-Code	0
164	ATC Feature Bit	0
165	Acceleration/Deceleration Options	0
166-169	Axis Properties for axes 5,6,7,8	0
170-177	PLC parameters	0
178	PLC I/O configuration (PLC program specific)	0
179	Lube Pump Operation	0
186	Probe Stuck retry disable	0
187	Hard Stop Homing Power Limit	0
188-199	Aux key functions	0
200-207	OPTIC4 Tach Volts Per RPM	0
208-215	MPU Lash/Screw Comp Acceleration Coefficient	0.125
216	PC Based Lash Compensation on/off	0
217	PC Based Screw Compensation on/off	0
218	USB MPG Options	0
219	Virtual Control Panel Options	0
220-231	Smoothing Parameters	Refer to text
236-239	Motor Cooling Coefficients for axes 5,6,7,8	0
240-241	Rigid Tapping Decel and Stepsize	0
252-255	Autotune Ka Performance parameters for axes 5,6,7,8	0
256	Drive Mode	0
257	TT1 connection detection PLC input	0
258	Velocity/Torque Mode override in Precision mode	0
259	Manual Axis Designation	0
260	Accelerated Graphics Backplot on/off	1
261-262	Precision Mode Standoff (Tolerance Percentage/Delay Time)	0

Parameter	Definition	Default setting
265-266	BISS Encoder Options	0
270	Skew Correction: X Skew offset from Trusted/Intended X	0
271	Skew Correction: Trusted/Intended Y	0
278	Spindle Speed Display Precision	0
284-291	Brake Resistor Wattage for ACDC Drives 1-8 50	
300-307	Drive assignment to Axes 1-8	Refer to text
308-315	Encoder assignment to Axes 1-8	Refer to text
316	Absolute Encoder Bits	0
317	Single Turn Absolute Encoder Bits	0
318	Five Axis Configuration	0
323	MPU11 Encoder Speed Filter	0
324-331	Axis Boxcar size	0
332-335	Encoder error suppression	0
336-339	Motor torque estimation for velocity mode drives	Refer to text
340-347	Precision Mode delay (in milliseconds) for axes 1-8	0
348,351,354	MPG 1,2,3 Encoder Input	15, 0, 0
349,352,355	MPG 1,2,3 Detents per Revolution	100
350,353,356	MPG 1,2,3 Encoder Counts per Revolution	400
357-364	Axis Drive Max RPM	0
365	Drive power-on delay	0.0
366	Probe deceleration multiplier	2.0
367	TT1 deceleration multiplier	2.0
368	Autonomous Digitizing Angle Adjustment	4
369	Tool Check Max Absolute Angle	0
374	ACDC Drive Debug Log Axis Inclusion	0
375	ACDC Drive Debug Log Size	0
376	ACDC Drive Debug Log Collection Type	0
377	ACDC Drive Debug Log Trigger Type	0
378	ACDC Drive Debug Log Trigger Level	0
379	ACDC Drive Debug Log Option	0
392-394	DP-7 parameters	Refer to text
395	Probing setup traverse speed	30 ipm or 762 mm/min
396	Probing setup plunge speed	30 ipm or 762 mm/min
398	Port/block mode	0
399	AD1 arc chord tolerance adjustment	.5
400	Run Menu Cycle Start Enabled	0.0

Parameter	Definition	Default setting
401	Forget Last Job Loaded	0.0
411	Mpg Type	0
900-999	PLC program parameters	

### 15.3.2 Parameter 0 – E-Stop PLC Bit

This parameter specifies the PLC bit to which the physical Emergency Stop switch is connected. It is mainly used for ATC applications that use custom PLC messages. See table below for examples.

PLC Type	ESTOP Input on PLC	Parameter Value
GPIO4D	Input 11	-11
ALLNONE	Input 11	-11
OAK	Input 11	-11
RTK2/3/4	Input 11	-11
PLCIO2	Input 11	-11
DC3IO	Input 11	-11
Servo3IO	Input 1	1

### 15.3.3 Parameter 1 – Y jog key orientation

This parameter is a 3-bit field where bit 0 is not used in the mill software. Bit 1 sets the direction of movement for the Y+ and Y- jog keys and bit 2 will swap the X and Y jog keys. This should always be set to 0 except for very special applications. **NOTE:** PLC program interaction is needed for these features.

Bit	Function Description	Parameter Value
0	Not	Used
1	Flip direction of Y jog keys?	Yes=2, No=0
2	Exchange X-axis and Y-axis jog keys?	Yes=4, No=0

### 15.3.4 Parameter 2 – G-code Interpretation Control and Slaving Rotary axis feedrate

This parameter is a bit field that controls optional interpretation of several G-codes. The following table shows the functions performed by the value entered in this parameter:

Bit	Function Description	Parameter Value
0	Arc centers I, J, K are absolute in G90 mode?	Yes = 1, No = 0
1	Allow Z being specified alone to be sufficient to trigger execution of a canned tapping or drilling cycle to be executed?	Yes = 2, No = 0
2	Interpret dwell times associated with G4, G74, G82, G84, and G89as milliseconds rather than seconds?	Yes = 4, No = 0

3	Slaving rotary-axis feedrate to non-rotary-axis feedrate. Note that this feature has no effect for movement commands handled by smoothing (when parameter 220=1).	Yes = 8, No = 0
4	Selects the center for scale, mirror and rotate. By default the center will be 0,0,0. Add 16 to this parameter to make the center of scale, mirror and rotate the current position.	Yes = 16, No = 0
5	Prevent rotary-only moves (rotary-axis by itself on a line of G-code) from acquiring a remembered slaved rotary feedrate previously set by a previous line of G-code containing a rotary and non-rotary together. Note that this feature has no effect for movement commands handled by Smoothing (when parameter 220=1).	Yes = 32, No = 0

### 15.3.5 Parameter 3 – Modal Tool and Height Offset Control

Bit	Function Description	Parameter Value
0	Tool and Height Offset numbers, T and H, will be reset upon job completion (and not remain modal and active between jobs).	Reset upon job completion = 1, Remain modal between jobs = 0
1	Reference tool position is set to Z home.	Yes = 2 No = 0
2	Tool Height Offset Retention option. This option prevents the current tool height offset from being turned off when the user enters the Tool Height Offset menu. Additionally, this option will cause the H height offset to be set to match the tool number when an Auto Tool Change is performed (via F7 ATC) in the Offset Library.	Turn on Tool Height Offset Retention option = 4, No = 0

### 15.3.6 Parameter 4 – Remote File Loading Flag & Advanced File Ops

This parameter controls the action of the Load Job menu when CNC job files are selected from drives letters higher than C. These drives (i.e. drives D, E, F, etc.) are presumed to be network drives or extra hard drives.

Value	Meaning
0	Job files are not copied or cached. They are run from whichever drives they reside on.
1	Job files are copied to the C drive (c:\cnc\ncfiles) when they are loaded. The local copy is used when the job runs.

2	Turn on file caching. Job files are temporarily cached on the C drive. The cached copy is used while the job is running. The cached copy is deleted when the next job is loaded or when Parameter 4 changes to a 0 or 1. Digitize files are cached as the machine is digitizing. When digitizing is complete, the resulting file is copied to the digitize directory specified in pathm.ini.
4	Set the Advanced File load menu as default for loading files
8	Use Windows-style File Open dialogue

File caching is useful for machines with both a flash card and a hard drive. By caching job files from the hard drive on the flash card, the hard drive is not used while the job is running. As a result, the life of the hard drive is extended and the flash card does not fill up with job files.

### 15.3.7 Parameter 5 – Suppress Machine Home Setup

This parameter controls machine homing upon startup of the control. The following table details the functions controlled by this parameter:

Bit	Function Description	Parameter Value
0	Suppress the requirement to set machine home before running jobs?	Yes = 1, No = 0
1	Display router bit map at homing screen	Yes = 2, No = 0
2	Disable stall detection when the CNC software first starts.	Yes = 4, No = 0

Bit 0 suppresses the requirement to set machine home before running. If bit 0 of Parameter 5 is 0, machine home must be set before jobs may be run. If bit 0 of Parameter 5 is 1, machine home is not requested or required, but Graphing and running of jobs will not work until the Z-axis is homed.

**NOTE:** Parameter 5 Bit 0 is separate from the “Machine Home at Powerup” flag in the Control Configuration Screen. Parameter 5 Bit 0 determines whether you must home the machine; the “Machine Home at Powerup” flag determines how you will home the machine, if you must do so.

### 15.3.8 Parameter 6 – Automatic Tool Changer Installed

This parameter tells the control whether an automatic tool changer installed on your machine. This field affects the action of M6 in your CNC programs. See M6 for more information. It also affects whether the ATC key is present in the Tool Offset Setup and whether to save the last tool change number in the job files.

Value	Meaning
0	Auto Tool Changer NOT Installed
1	Auto Tool Changer Installed

### 15.3.9 Parameter 7 – Display Colors

This parameter determines what combination of colors will be used for display. If you have a color display, set this parameter to 0. If you have a monochrome display (especially a monochrome LCD panel) set this parameter to 1.

### 15.3.10 Parameter 8 – Available Coolant Systems

This parameter is used by Intercon to determine what coolant systems are available on the machine. It should be set as follows:

Value	Meaning
1	Mist Coolant (M7) only
2	Both coolant systems
3	Flood Coolant (M8) only

### 15.3.11 Parameter 9 – Display Language

This parameter determines what language will be used for menus, prompts, and error messages.

Value	Meaning
0	English
1	Spanish
2	French
3	Traditional Chinese
4	Simplified Chinese
5	German
6	Swedish
7	Finnish
8	Portuguese
9	Greek

### 15.3.12 Parameter 10 – Macro M function handling/ Probe Stop Handling

This parameter is a bit field that controls various aspects of M functions. The following table shows the functions performed by the value entered in this parameter. The default value is 0.

Bit	Function Description	Parameter Value
0	Display M & G-codes in M function macros?	Yes = 1, No = 0
1	Step through M function macros in Block Mode?	Yes = 2, No = 0
2	Decelerate to stop on M105 and M106. With decel.set these moves take longer and are slightly less accurate. With immediate stop these moves are faster and more accurate; however the lack of controlled deceleration can cause excessive machine vibration.	Decel = 4, Immediate Stop = 0

3	Move to Z home on M6?	No = 8, Yes = 0
4	(not used) This bit was previously used for controlling the type of deceleration to stop on digitizing and probing moves. This functionality is now controlled by parameters 366 and 367.	Recommended value = 0

### 15.3.13 Parameter 11 - DP4 PLC Input Number and Contact State

This parameter is used for the PLC input number that is used by the DP4 Touch Probe. Allowable range is a single value, +/- 1 to 240 and 50001 to 51312. A Positive number indicates Closed on contact and a negative number indicates Open on contact. A list of default settings for different console types are listed below.

**NOTE:** Changing this parameter can cause damage to your probe. You should contact your Dealer or Local Tech Representative before any modifications are made.

Console Type/Model	Input Number	PLC Type/Model	Input Number
M39	15	DC310	14
M39s	14	RTK3	14
M400	15	Servo310	2
M400s	14	PLC102	15
M15-10	2	15/15	15
M400 ATC (RTK3)	14	RTK2	15
M400 ATC (PLC102)	15	Koyo	ATC 1
MPU11	50769		

### 15.3.14 Parameter 12 – Touch Probe Tool Number

This parameter is the tool number of the DP4 probe. Allowable range is 0 through 200. By default the value is 10. This is used to look up the length offset and tip diameter of the probe in the Tool Offset Library.

### 15.3.15 Parameter 13 – Recovery Distance

This parameter is the distance that the probe moves off a surface after initial contact (only during probing cycles), before returning to the surface to take a recorded reading. For DSP Probes: This parameter is used for failed DSP windows. On a failed window, the DSP probe will retract this distance before retrying.

### 15.3.16 Parameter 14 – Fast Probing Rate

The fast probing rate is used for positioning moves and initial surface detection, and is determined by the machines response time and the permitted probe deflection. The default is 10 in/min. This is a very conservative feedrate, in actual use 20 to 30 in/min is a good value and will not have any detrimental impact on accuracy in most cases.

### 15.3.17 Parameter 15 – Slow Probing Rate

The slow probing rate is used for the final measurement moves. The default setting is 1 in/min. The following are some typical accuracy tolerances with the corresponding value set in parameter 15:

Probing Rate		Accuracy	
0.5 in/min	12.7 mm/min	0.0001 in	.00254 mm
1 in/min	25.4 mm/min	0.0002 in	.00508 mm
1.5 in/min	38.1 mm/min	0.0005 in	.0127 mm
3.5 in/min	88.9 mm/min	0.0010 in	.0254 mm
18 in/min	457.2 mm/min	0.0050 in	.1270 mm

### 15.3.18 Parameter 16 - Maximum Probing Distance

This is the maximum distance that the Boss and Web probing cycles “search” for a surface in a given direction if no travel limits have been entered. The default setting is 10 inches. A larger value should be entered for the boss and web cycles if you are measuring very large features. These settings are conservative measurements and can be used initially for startup purposes. However, they can be changed to accommodate your work.

### 15.3.19 Parameter 17 – Detector Location Return Point

A non-zero value specifies the number of the reference return point (entered into the WCS menu) directly above a permanently mounted TT-1 tool detector. When the Auto function is called up in the tool offset library, the control will position the table to the return point specified by this parameter, and touch the tool off the TT-1 Tool detector.

Entry	Return Point
0	None
1	G28
2	G30
3	G30P3
4	G30P4

A zero indicates that the tool detector is no permanently mounted; automatic tool measurement will be performed without X/Y axis movement.

### 15.3.20 Parameter 18 – PLC Input Spindle Inhibit Parameter

This parameter stores the input for the Spindle Inhibit feature. A negative value must be entered if a “normally closed” probe is to be used with the control. A positive value must be entered if a “normally open” probe is to be used with the control. The absolute value of Parameter 18 will directly reflect the PLC input the Spindle Inhibit is wired to.

When this parameter is set, Digitizing and Probing cycles will not run unless a probe or touch-off block is connected. This parameter is used to prevent the tool or probe from crashing into the table. The default for this parameter is 0, which disables this feature.

PLC Type/Model	Input Number
Allin1DC	50771
MPU11	50771
DC310/DC310B	15
RTK3	15
Servo310/M15DRV1	5
PLC102	10
15/15	10
RTK2	10
Koyo ATC	2

### 15.3.21 Parameter 19 – MPG modes

The MPG is a hand-held device that is used as an alternate way of jogging the machine. This parameter defines the MPG’s mode of operation.

Bit	Function Description	Parameter Value
2	Enable Z axis MPG* – This will allow the z- axis to be moved with the MPG while running a job independent of the x and y axes	Yes = 4, No = 0

\*PLC program interaction is needed for these features. The plc program is in direct control of MPG modes. Z-axis MPG operation is not available with all controls.

### 15.3.22 Parameters 20–30 (also 132–135, 236–239) — Motor Temperature Estimation

These parameters are used for motor temperature estimation. Parameters 20, 29 and 30 correspond respectively to the ambient temperature of the shop, the overheating warning temperature, and the job cancellation temperature, all in degrees Fahrenheit. Parameters 21-24 and 132-135 are the heating coefficients. Parameters 25-28 and 236-239 are the cooling coefficients.

To disable Motor Temperature Estimation for an axis, set its heating and cooling coefficients to 0. For example, to disable Motor Temperature Estimation for axis 1, set Parameter 21 to 0, and set Parameter 25 to 0.

**NOTE:** Temperature estimation only applies to controls operating in Torque mode (i.e. DC brushed systems and CentroidAC systems). MPU11 systems running in Velocity mode (i.e. third party drive systems) do not use this feature, and thus should be disabled (by setting all heating and cooling coefficients to 0).

Suggested Values for AC Brushless Motors and Drives						
SD Drive		SD3, SD1 750 W motors	SD3, SD1 1, 2 KW motors	SD3, SD1 (finned heatsink) 1, 2, KW motors	SD1 45A (finned heatsink) 3 KW motors	SD1 45A (finned heatsink) 4 KW motors
Parameters	Axes	Values	Values	Values	Values	Values
21–24	1–4	0.23	0.5	0.23	0.23	0.23
132–135	5–8	0.23	0.5	0.23	0.23	0.23
25–28	1–4	12.0	9.0	12.0	12.0	14.5
236–239	5–8	12.0	9.0	12.0	12.0	14.5
20	N/A	72	72	72	72	72
29	N/A	150	150	150	150	150
30	N/A	180	180	180	180	180

Suggested Values for DC Brush Motors and Drives						
Servo Drive		9A Drive, 16 in/lb motors	12A Drive, 29 in/lb motors	15A Drive, 29 in/lb motors	15A Drive, 40 in/lb motors	25A Drive, 40 in/lb motors
Parameters	Axes	Values	Values	Values	Values	Values
21-24	1-4	0.028	0.02	0.027	0.03	0.04
132-135	5-8	0.028	0.02	0.027	0.03	0.04
25-28	1-4	0.68	0.68	0.68	0.68	0.68
236-239	5-8	0.68	0.68	0.68	0.68	0.68
20	N/A	72	72	72	72	72
29	N/A	150	150	150	150	150
30	N/A	180	180	180	180	180

### 15.3.23 Parameter 31 – Legacy SPIN232 Com Port

For values 1-255, CNC11 will attempt to open that COM port and send out spindle commands. CNC11 should be restarted after changing this value from 0 (disabled) to a valid value. Note that the baud rate is assumed to be 19200 to work specifically with a SPIN232.

### 15.3.24 Parameter 32 – Autonomous Digitizing Offset Files

This bitwise parameter specified which files should be output from the autonomous digitizing offset surface calculations. The bits are:

Bit	Function Description	Parameter Value
0	Raw Centerline Data	Yes = 1, No = 0
1	Clean Centerline Data	Yes = 2, No = 0
2	Centerline Flow Splines	Yes = 4, No = 0
3	Legacy Offset Data	Yes = 8, No = 0
4	Spline Offset Data	Yes = 16, No = 0
5	Spline Offset Flow Splines	Yes = 32, No = 0
6	Offset Plus Flow	Yes = 64, No = 0

### 15.3.25 Parameter 33 – Spindle Motor Gear Ratio

The default value for this parameter is 1 and should not be changed unless you have consulted your dealer or local Technical representative!!!

Parameter 33 is used for the gear or belt ratio between the spindle motor and the chuck in high gear range. It should be greater than 1.0 if the motor turns faster than the chuck and less than 1.0 if the chuck turns faster than the motor. **NOTE:** this value applies to high range. The ratio between high range and lower ranges is established by the gear ratio parameters (65-67).

### 15.3.26 Parameter 34 – Spindle Encoder Counts/Rev

This parameter controls the counts revolution for the spindle encoder. Input from the spindle encoder is required for the spindle-slaved movements used in the Rigid Tapping cycles. If the encoder counts up when running CW (M3), the value of this parameter must be positive. If the encoder counts up when running CCW (M4), the value of this parameter must be negative.

### 15.3.27 Parameter 35 – Spindle Encoder Axis Number

Input from a spindle encoder is required for spindle-slaved movements such as those used in the Rigid Tapping cycles. If there is no spindle encoder connected, then this parameter should be set to 0. This parameter specifies the axis number (1 through 8) to which the spindle encoder is assigned. Encoder assignments are specified by parameters 308-315. For example, if you decide to configure the 5th axis as the spindle, and the spindle's encoder is the 1st MPU11 onboard input encoder, then Parameter 35 (this parameter) should be set to 5, and Parameter 312 should be set to 1.

### 15.3.28 Parameter 36 – Rigid Tapping Enable/Disable

This parameter is a bit field that enables or disables Rigid Tapping and its options. Bit Function

Bit	Function Description	Parameter Value
0	Enable Rigid Tapping?	Yes = 1, No = 0
1	Suppress sending "Wait for Index Pulse" during Rigid Tapping?	Yes = 2, No = 0
2	Allow Spindle Override during Rigid Tapping?	Yes = 4, No = 0
3	Use Spindle Off system variable bit? (see note below)	Yes = 8, No = 0 (see note below)
4	What is the Spindle Encoder measuring? i.e. What is the physical mounting location of Spindle Encoder? (see note below)	Spindle Encoder measures rotation of spindle drive motor: value = 16, Spindle Encoder measures rotation of spindle head: value = 0 (see note below)
5	Turn Off Spindle Following While At the Bottom of a Hole. (NOT RECOMMENDED)	Yes = 32, No = 0

Note on Bit 3: This bit enables the spindle off system variable, for mpu11 systems. Most systems will not need to set this bit. Mpu11 systems will not execute custom M5 macro commands during a rigid tap.

Instead the software determines which bit, M3 or M4, to turn off to stop the spindle. Alternatively, setting bit 3 will cause the software to set the spindle off system variable bit, SV PC RIGID TAP SPINDLE OFF. The plc program is then responsible for monitoring that bit and performing all actions in order to turn off the spindle.

Note on Bit 4: On machines where the Spindle Encoder is set up to measure the spindle drive motor rotation (Bit 4 = on, value 16), multi-pass Rigid Tapping and repeated Rigid Tapping down the same hole can only be done in the High Spindle Range. (See Parameters 33 and 65-67 for description of Spindle Gear ranges/ratios.)

### 15.3.29 Parameter 37 – Spindle Deceleration Time ( Rigid Tapping Parameter)

This parameter is used in conjunction with parameter 36 when rigid tapping is enabled. This sets the amount of time required for the spindle to decelerate before it switches direction during a rigid tapping operation.

### 15.3.30 Parameter 38 – Multi-Axis Max Feedrate

This parameter is used to limit the feedrate along all commanded move vectors. This parameter can be used to limit the speed of multi-axis moves on machines that may have enough power to move a single axis rapidly, but starve out of power on 2 or 3 axis rapid moves. A zero in this parameter will disable this feature. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

### 15.3.31 Parameter 39 – Feedrate Override Percentage Limit

This parameter is used for limiting the upper end of the Feedrate Override Knob percentage to a value from 100% to 200%. This parameter can be used to restrict the Feedrate Override Knob effect on machines with maximum rates over 200 in/min. The Feedrate Override Knob percentage is normally allowed to go to 200%. However, on machines with high cutting speeds, if the knob is turned up to 200%, it creates overshoots on corners. If this parameter is set to something like 110, it will stop the Feedrate Override Knob from exceeding 110% and thus causes the overshoots to disappear. The PLC program must check this parameter to actually enforce the limiting of Feedrate Override.

### 15.3.32 Parameter 40 – Basic Jog Increment

This parameter holds the basic jog increment for linear axes (0.0001” or 0.002 mm by default). This value works in conjunction with the x1, x10, and x100 jog keys to determine the amount to move a linear axis during incremental jogging.

### 15.3.33 Parameter 41 – Rotary Axis Jog Increment

This parameter holds the jog increment for rotary axes (0.01° by default). This value works in conjunction with the x1, x10, and x100 jog keys to determine the amount to move a rotary axis during incremental jogging.

### 15.3.34 Parameter 42 – Password for Configuration Menus

This parameter determines the password that the user must enter in order to gain full access to the configuration menus.

Value	Meaning
54.0	No password required for supervisor access; the user is not prompted for a password
ABCD. ABCD	Password is 4 digits represented by “ABCD” Example: for the password to be 1234, set to 1234.1234
Any other number	Password is “137”

### 15.3.35 Parameter 43 – Automatic tool measurement options

This parameter is a bit field that is used to configure properties of the TT1.

Bit	Function	Parameter Value
0	The height of the tool detector (parameter 71) will be subtracted from the measured height of the tool.	Yes = 1; No = 0
1	Which PLC input to use for the Tool Z reference measurement. 0 = Use Touch Probe input in Parameter 11. 2 = Use TT1 input in Parameter 44.	0 = Use Parameter 11. 2 = Use Parameter 44.
2	Remind the operator to plug in the TT1 before doing Batch Tool Measuring.	Yes=4; No = 0

### 15.3.36 Parameter 44 – TT1 PLC input number

This parameter is the input number that the TT1 is wired into on the PLC. If a shared PLC input is used for the TT1 and the DP4 probe, then the value can be left at zero or set to the same value as parameter 11.

**NOTE:** If you are using a different PLC input for the TT1 and DP4 when setting the Z reference in the tool library with the DP4, make sure you don’t use a ruby probe tip. The TT1 is electrical continuity based and the ruby tip is not electrically conductive!

### 15.3.37 Parameter 45 — WCS Lockout

This parameter allows you to “lock out” a WCS from editing if you do not want it’s values to change. Simply add the bit of the WCS you want locked out. For example, if you want to lock out WCS’s 1, 2, and 5, you set the value of parameter 45 to  $2(1-1) + 2(2-1) + 2(5-1) = 1 + 2 + 16 = 19$ .  
**NOTE:** G92 can still set a WCS even if it is locked out.

### 15.3.38 Parameter 46 — Active G-Codes Display

Set this parameter to 0 to always display the currently active G-codes in the bottom left corner of the main screen and MDI screen. Set it to 1 to display the G-codes only in MDI. Set it to 2 to disable the display completely.

### 15.3.39 Parameter 48 – Grid Digitize Patch Playback Z rapid clearance amount

This is the additional Z clearance amount higher than the Z surface level at which the original Grid Digitizing operation was begun. The purpose of this value is to set the recorded starting “rapid to” Z level of a Grid Digitize playback patch. In other words, the Z starting point of the first feedrate plunge move of the recorded patch is equal to the Z coordinate of the start of the original Grid Digitizing operation plus the value of this parameter.

### 15.3.40 Parameters 49–51 — Small Arc Feedrate Limiting

These parameters are intended to control the federate limits of small-radius arc moves.

Parameter	Function Description	Input format
49	Lower arc radius/federate limiting parameter	rrrr.ffff, where rrrr is a radius threshold setting, and .ffff is a federate adjustment setting
50	Upper arc radius/federate limiting parameter	RRRR.FFFF, where RRRR is a radius threshold setting, and .FFFF is a federate adjustment setting
51	Arc Feedrate Limiting mode	2 digit positive number. 1's digit (right digit) = Lower Limiting mode 10's digit (left digit) = Upper Limiting mode

The 2 digits in Parameter 51 can have the following values and associated meanings:

- 0 – Turn off Arc feedrate limiting for the associated radii range (default value)
- 1 – Literal Mode Feedrate limit = FFFF (.FFFF x 10000) for the associated radii range.
- 2 – Fractional Mode Feedrate limit = .FFFF x Programmed feedrate for the associated radii range
- 3 – Proportional Mode Feedrate limit = Arc radius x .FFFF x Programmed feedrate for the associated radii range.

The Lower Arc Limiting mode’s radii range includes any arc radius from 0 up to and including the rrrr setting of Parameter 49. So, if you program a G2 or G3 arc with a radius of rrrr (of Parameter 49) or less, and the Lower Limiting mode in Parameter 51 is turned on, then the arc’s federate will be limited according to how ffff (of Parameter 49) is interpreted. If the Control is set up in Millimeter mode,

then rrrr specifies the number of millimeters. However, if the Control is set up in Inch mode, then rrrr means the number of inches divided by 100.

The Upper Arc Limiting mode’s radii range includes any arc radius larger than the rrrr setting of Parameter 49 up to and including the RRRR setting of Parameter 50. So, if you program a G2 or G3 arc with a radius of RRRR (of Parameter 50) or less, but greater than rrrr (of Parameter 49), and the Upper Limiting mode in Parameter 51 is turned on, then the arc’s federate will be limited according to how FFFF (of Parameter 50) is interpreted. If the Control is set up in Millimeter mode, then rrrr specifies the number of millimeters. However, if the Control is set up in Inch mode, then rrrr means the number of inches divided by 100.

Note that if Parameter 51 contains values other than 00, 01, 02, 03, 10, 11, 12, 13, 20, 21, 22, 23, 30, 31, 32, 33, it will be treated as invalid and will cause the Small Arc Feedrate Limiting feature to be disabled.

Also note that this feature has no effect for arc movement handled by Smoothing (P220=1).

#### Examples: (in millimeters)

Parameter 49 = 5.0050 (rrrr = 5 mm, ffff = 0050)

Parameter 50 = 75.1500 (RRRR = 75 mm, FFFF = 1500)

If Parameter 51 = 31 and you program a G2 arc with a radius of 4 mm at a feedrate of 1000 mm/min, then the actual feedrate of the arc will be lowered down to 50 mm/min.

If Parameter 51 = 23 and you program a G3 arc with a radius of 40 mm at a feedrate of 700 mm/min, then the actual feedrate of the arc will be lowered down to 105 mm/min (=700 x .1500).

If Parameter 51 = 33 and you program a G3 arc with a radius of 72 mm at a feedrate of 1200 mm/min, then the actual feedrate of the arc will remain at 1200 mm/min and will not be modified because it is well within the Feedrate limit of 12960 mm/min (Arc radius x .FFFF x Programmed feedrate = 72 x .1500 x 1200 = 12960).

If Parameter 51 = 11 and you program a G2 arc with a radius of 100 mm at a feedrate of 2500 mm/min, then the actual feedrate of the arc will remain unmodified at 2500 mm/min because the arc radius is outside both ranges specified by Parameters 49 and 50, and therefore this feature does not affect such arcs.

#### Examples: (in inches)

Parameter 49 = 50.0050 (rrrr = 1/2 inch, ffff = 0010)

Parameter 50 = 200.1500 (RRRR = 2 inches, FFFF = 1500)

If Parameter 51 = 31 and you program a G2 arc with a radius of 1/4 inch at a feedrate of 100 inches/min, then the actual feedrate of the arc will be lowered down to 10 inches per minute.

If Parameter 51 = 23 and you program a G3 arc with a radius of 1.5 inches at a feedrate of 200 inches/min, then the actual feedrate of the arc will be lowered down to 30 inches/min (=200 x .1500).

If Parameter 51 = 33 and you program a G3 arc with a radius of 1.75 inches at a feedrate of 300 inches/min, then the actual feedrate of the arc will be lowered down to 78.75 inches/min (= Arc radius x.FFFF x Programmed feerate = 1.75 x .1500 x 300).

If Parameter 51 = 11 and you program a G2 arc with a radius of 3 inches at a feedrate of 250 inches/min, then the actual feedrate of the arc will remain unmodified at 250 inches/min because the arc radius is outside both ranges specified by Parameters 49 and 50, and therefore this feature does not affect such arcs.

### 15.3.41 Parameter 56 — Feedrate Override Display Propertiexs / Inverse Time Interpolation

Bits 1 and 2 define how the federate override is displayed in the status window. Bit 3 selects the meaning of inverse time feedrates (F) when inverse time mode (G93) is active. Bit 0 is unused.

Bit	Function	Parameter Value
0	Not used	
1	Display programmed rate not actual	2 = Yes; 0 = No
2	Display a bar meter of percentage	4 = Yes; 0 = No
3	Inverse time meaning of F	8 = F is in inverse seconds, 0 = F is in inverse minutes

### 15.3.42 Parameter 57 — Use Generic Load Meter Data from PLC

If load meters are enabled (via parameter 143) the data being displayed for each axis is normally calculated from its PID output (either real or simulated). However, if any axis has its corresponding bit set in this parameter, then the data going into the load meter will come from the PLC system variables SV METER 1- SV METER 8 (corresponding to axes 1-8). Furthermore, if an axis is a spindle axis (as set by parameter 35) the load meter data will be displayed on the spindle override meter in the Status Window.

Cooperation from the PLC program is required for this feature to work. The PLC program needs to send back a value between 0.0 and 100.0 via floating point system variables SV METER 1 - SV METER 8 (corresponding to axes 1- 8). Note that there is future support for SV METER 9 up to SV METER 16, but for now there is support for up to only 8 axes.

### 15.3.43 Parameters 61 – 62 — Stall Detection Parameters

The M-Series control will detect and report several stall conditions. The low power stall occurs if the control has been applying a specified minimum current for a specified time, and no encoder motion has been detected. This may indicate a loose or severed encoder cable. A high power stall occurs if the control has been applying at least 90% current for a specified time, and no motion greater than

0.0005” has been detected. This may indicate a physical obstruction. Note that this feature will only work with torque mode drives and not velocity mode drives.

Parameter 61 is the time limit, in seconds, for a high power stall. The default is 0.5 seconds. Parameter 62 is the PID output threshold for a high power stall. The default is 115.

### 15.3.44 Parameter 63 — High Power Idle PID Multiplier

This parameter holds the value of a constant used for motor high power idle detection when an axis is not moving and no job is running, but there is power going into the motor to maintain its position. The default value is 1.5. This is intended for early detection of an axis if it’s stopped against some abnormal resistance or not tuned correctly, such that it will probably overheat later.

### 15.3.45 Parameter 64 — Fourth/Fifth Axis Pairing

This feature enables the 4th and 5th axes to be paired together or individually be run in a slaved state with any of the other axes. This is intended to drive 2 screws on opposite sides of a table (probably a router table or gantry system). Set this parameter to 0 (default) to indicate that no other axis is paired with the 4th or 5th axis. In order to pair both the 4th and 5th axes on the same system add the 4th axis value with the 5th axis value. Example: 4th axis paired with the X-axis and 5th axis paired with the Z-axis a value of 49 would be entered into parameter 64. The axes are slaved upon power up but it is still possible to move the paired (4th or 5th) axis independently if the axis is labeled.

**\*NOTE:** You cannot run Autotune on paired axes.

Value	Meaning
0	No Pairing (Default)
1	Pair 4th axis with 1st Axis
2	Pair 4th axis with 2nd Axis
3	Pair 4th axis with 3rd Axis
16	Pair 5th axis with 1st Axis
32	Pair 5th axis with 2nd Axis
48	Pair 5th axis with 3rd Axis
64	Pair 5th axis with 4th Axis

### 15.3.46 Parameters 65 – 67 — Spindle Gear Ratios

These parameters tell the control the gear ratios for a multi-range spindle drive. Up to four speed ranges are supported; high range is the default. Parameters 65-67 specify the gear ratio for each lower range, relative to high range. For example, if the machine is a mill with a dual range spindle, and the spindle in low range turns 1/10 the speed it turns in high range, then parameter 65 should be set to 0.1. Note that these values can be signed +/- . So, if switching from high range to a lower range causes the spindle encoder to count in the opposite direction, then a negative value can be used to compensate for this behavior.

Parameter 65 is the low range gear ratio. Parameter 66 is the medium-low range gear ratio. Parameter 67 is the medium-high range gear ratio.

These parameters work in conjunction with the PLC program, which uses the states of INP63 and INP64 to signal to the CNC software which range is in effect, according to the table below.

Spindle Range				
PLC INPUT	High Range	Medium High Range	Medium Low Range	Low Range
INP63	0	1	1	0
INP64	0	0	1	1

### 15.3.47 Parameter 68 — Minimum Spindle Speed (Rigid Tapping Parameter)

This parameter holds the value that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z axis will land on target, but at the expense of possibly stalling the spindle motor which in turn will cause Z-axis to fall short. If this value is too large, the off target error will increase. The suggested starting value is 640 RPM.

### 15.3.48 Parameter 69 — Duration for Minimum Spindle Speed Mode (Rigid Tapping Parameter)

This is the duration of time, in seconds, that the control will stay at minimum spindle speed. If the number is too small, overshoot will occur. If the number is too large, the user waits longer for the hole to be tapped at the slow speed specified by parameter 68. The suggested starting value is 1.25 seconds.

### 15.3.49 Parameter 70—Offset Library Inc/Decrement Amount

Sets the increment and decrement amount used in the offset library.

### 15.3.50 Parameter 71 — Part Setup Detector Height

If this Parameter is set to a non-zero value, it indicates that the F3 - Auto feature in part setup should be available using the tool detector (TT1) instead of the probe. The value in this parameter is the height of the detector. A value of 0 disables this feature.

When this feature is enabled:

- Probe detection (Parameter 18) is not checked
- The tool number and/or edge finder diameter entered by the operator are used; Parameter 12 is ignored.
- The value from Parameter 71 is added to (or subtracted from, depending on approach direction) the part position.

### 15.3.51 Parameter 72 — Data M Function Options

The setting of this parameter affects the operation of the

data M functions M122, M123, and M124.

Bit	Function Description	Parameter Value
0	Suppress output of axis labels by M122/M124?	Yes = 1, No = 0
1	Insert commas between positions/values with M122/M124?	Yes = 2, No = 0
2	Suppress spaces between positions/values outputted by M122/M123/M124?	Yes = 4, No = 0

### 15.3.52 Parameter 73 — Peck Drill Retract Amount (Canned Cycle Parameter)

This specifies the retract amount used during a G73 peck drilling cycle.

### 15.3.53 Parameter 74 — M-function executed at bottom of tapping cycle (Canned Cycle Parameter)

This specifies the number of the M-function that is executed at the bottom of the G84 tapping cycle (primarily used for reversing the spindle in preparation for pulling out of the tap hole). This also specifies the number of the M-function that is executed after the G74 countertapping cycle is done (returned to the initial point).

### 15.3.54 Parameter 75 — Summing Control

This parameter controls the type of position to be summed (local or machine), and which axes are to be summed together and which axis will bear the effects of the the summing. The parameter can contain up to four digits. The sign of the parameter value and the position and value of each digit has special significance as indicated in the tables below:

The “Summed Axis” is the axis that bears the position sum of itself with the “Axis to Sum with”. The DRO display of the “Summed Axis” will show this summed position. The DRO will display both labels when displaying a summed axis. Furthermore, G-codes that command movement on the “Summed Axis” will have their positions offset by the position of the “Axis to Sum with”.

It is highly recommended that summing be done with Local Coordinates (using the ‘-’ sign in the parameter value). Summing with Machine Coordinates can cause the effective software travel limits to move, thus resulting in physical overtravel or severely handicapping the amount of available travel, due to the fact that software travel limits are defined in terms of Machine Coordinates. Summing with Local Coordinates avoids this problem.

Here are some examples:

Desired Display	Parameter
Axis 3 DRO will display the result of axis 3 and 4 machine coordinate positions summed together.	400
Axis 4 DRO will display the result of axis 4 and 3 local coordinate positions summed together. Machine coordinate positions are not affected.	-3000

Axis 3 DRO will display the result of axis 3 and 4 local coordinate positions summed together. Axis 4 DRO display will be disabled.	-6400
Axis 4 DRO will display the result of axis 4 and 3 local coordinate positions summed together. Axis 3 DRO display will be disabled.	-3600
Axis 3 DRO will display the result of axis 3 and 4 local coordinate positions summed together. Axis 4 will be displayed only if it moves.	-7400

### 15.3.55 Parameter 76 — Manual Input Unrestricted Distance

This parameter is intended to be used with Z-axis summing. It defines the maximum distance from the summed axis start of travel in which manual movements can occur without causing a fault. Use a negative value to specify a distance from the minus travel limit, a positive value for a distance from the plus travel limit.

When used with manual drilling, for example, setting this parameter will allow the operator to keep a hand on the quill at all times and even begin pulling on the quill in anticipation of a programmed stop.

Setting this value to zero will cause a fault if there is any manual movement.

To completely disable manual movement restrictions, set this parameter to a value exceeding the total travel of the summed axis.

Minimum = -99999.9999, maximum = 9999.9999, default = 0, typical = +/- 1.0 inch or +/- 20.0 mm

### 15.3.56 Parameter 77 — Manual Input Movement Tolerance

This parameter specifies the manual movement tolerance while a job is running. It is intended for use with a quill locking mechanism. It allows the lock to distort and/or slip a small amount when under stress. If the quill moves more than the given tolerance, the job will stop with a fault. A typical setting for Parameter 77 is 0.005 inches.

### 15.3.57 Parameter 78 — Spindle Speed Display and Operations

Bit 0 (value = 1) specifies how the spindle speed is determined and displayed in the CNC software status window. When turned on (value = 1), the spindle speed is determined by reading the encoder feedback from the axis specified according to parameter 35. Which has the number of encoder counts/revolution specified in parameter 34. When turned off (value = 0), the displayed speed is not measured; the speed is calculated based upon the programmed speed, spindle override adjustment, and gear range.

Bit 1 (value = 2) allows the control to prorate the programmed feedrate to be proportional to the spindle speed if the spindle speed ever slows down below the spindle speed threshold percent as set by parameter 149.

Bit 2 (value = 4) will turn on the “Spindle up-to-speed” function. The active modal spindle speed S at the point where the most recent M3 or M4 is invoked sets the target spindle speed for this function. This function is invoked on the first feed-per-minute move (such as G1/G2 G3) following the aforementioned M3 or M4. If the actual measured spindle speed at this moment is below the spindle speed threshold percent (parameter 149) of target spindle speed, this function will pause the job at this point until the spindle speed gets up to at least this spindle speed threshold percent level.

Bit	Function	Value
0	Display actual spindle speed	Yes = 1, No = 0
1	Slave feedrate to spindle speed	Yes = 2, No = 0
2	Wait for spindle to get up to speed on feed-perminute moves	Yes = 4, No = 0

### 15.3.58 Parameter 79 — Paired Axes Re-sync Delay Time (milliseconds)

This parameter specifies the length of time (in milliseconds) that the control will wait before re-syncing paired axes. If this parameter is a negative value, the control will prompt the operator to press Cycle Start before re-syncing.

### 15.3.59 Parameter 80 — Voltage Brake Message Frequency

This parameter specifies the number of times the “450 Voltage brake applied” message has to occur before we show it in the message window and message log. A value of 0 or 1 will display the message for every instance that it occurs.

### 15.3.60 Parameter 81 — Air Drill M-function (Canned Cycle Parameter)

P81 (when not equal to -1.0) specifies the M-function to be called in place of Z-axis movement during a G81 drilling cycle.

### 15.3.61 Parameter 82 — Spindle Drift Adjustment (Rigid Tapping Parameter)

This value is the number of degrees that the spindle will take to coast to a stop, when the spindle is turned off at the minimum spindle speed specified by parameter 68.

### 15.3.62 Parameter 83 — Deep Hole clearance amount (Canned Cycle Parameter)

Parameter 83 specifies the clearance amount used during a G83 deep hole drilling cycle.

### 15.3.63 Parameter 84 — M function executed at a return to initial point of tapping cycle (Canned Cycle Parameter)

This specifies the number of the M-function that is executed after the G84 tapping cycle is done (returned to the initial

point). This also specifies the number of the M-function that is executed at the bottom of the G74 countertapping cycle (to reverse the spindle in preparation for pulling out of the countertap hole).

### 15.3.64 Parameter 85 — “Door Open” Interlock PLC Bit

This parameter provides a way for a system integrator to implement a safety interlock that limits rate of movement when the doors are open. This parameter specifies the PLC bit number and PLC bit polarity that indicates the “door open” condition. If the specified PLC bit is in the specified “door open” condition, then all normal movement commands will be limited to the slow jog rate (as specified in the Jog Parameters menu in Machine Configuration). Polarity of the “door open” condition is specified thuswise: a positive number indicates that the “door open” condition occurs when the specified PLC bit is On, and a negative number indicates that the “door open” condition occurs when the specified PLC bit is Off. If this parameter is set to 0 (the default value), then this feature is disabled, and no checking for a “door open” condition is done. Note that this feature has no effect for movement commands handled by Smoothing (P220=1). Also note that PLC program interaction is needed for this feature to work.

### 15.3.65 Parameter 86 — Rapid/Linear vector rate limit

This parameter controls the feature that imposes a limit on the number of rapid and/or linear moves per second to the value specified in this parameter. If the value of this parameter is more than 0, Rapid and/or Linear moves will be combined to prevent the aforementioned limit from being exceeded. This parameter is used for testing purposes and should be set to 0 to disable this feature. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

### 15.3.66 Parameters 87 – 90 (and also 252 – 255) — Autotune Ka Performance parameters

These parameters are used by autotune. Increasing the value will increase the Ka used by autotune which when used will increase the PID used during acceleration. The default value is 0. The maximum value is 50 and the minimum value is 0. Values for axes 1–4 are specified in parameters 87–90. Values for axes 5-8 are specified in parameters 252–255.

### 15.3.67 Parameters 91 – 94 (and also 166 – 169) — Axis Properties

These parameters may be used to set various axis properties. Properties for axes 1 – 4 are specified in parameters 91 – 94. Properties for axes 5 – 8 are specified in parameters 166 – 169.

Bit	Function Description	Parameter Value
0	Rotary/Linear Axis Selection Rotary	Axis = 1, Linear Axis = 0
1	Rotary Display Mode	Wrap Around = 2, Show Rotations = 0
2	Suppress direction check when doing Tool Check?	Don't Check = 4, Check = 0
3	Suppress park function?	Don't Park = 8, Park = 0
4	NOT USED ON MILL	Recommended bit value is 0
5	Linear Display of Rotary Axis	Linear Display = 32, Default Rotary = 0
6	4th Axis works like Z axis	Yes = 64, No = 0
7	NOT USED ON MILL	Recommended bit value is 0
8	Axis is triangular rotary?	Yes = 256, No = 0
9	Hide axis display from DRO	Yes = 512, No = 0
10	Triangular rotary axis type	Articulated Head = 1024, Tilt Table = 0
11	Rotation Center is parallel to X?	Yes = 2048, No = 0
12	NOT USED	Recommended bit value is 0
13	NOT USED	Recommended bit value is 0
14	Enable TWCS for Articulated Head machines	Yes = 16384, No = 0

**Bit 0:** Turning this bit on will cause the DRO display for the affected axis to be displayed in degrees. Also this information is used by Intercon to make rotary axis support available (by setting parameter 94 to 1, indicating that the fourth axis is rotary). This bit is also used when performing inch/mm conversions: values for a rotary axis will not be converted since they are assumed to be in degrees regardless of the system of linear units.

**Bit 1:** This bit has no effect unless Bit 0 (mentioned above) is turned on. When this bit is turned on, a “Wrap Around” display is shown on the DRO. A “Wrap Around” Rotary Display is a display in degrees without the number of rotations shown. If this bit is turned off, the number of rotations away from 0 degrees will be shown alongside the degree display.

**Bit 2:** This bit will only affect the Z-axis. It controls whether or not a direction check will be performed when the Tool Check button is pressed. If this bit is turned on, direction checking is turned off, and thus, there is a possibility for the Z-axis to move downward unexpectedly, depending on the Z value of Return Point #1 (G28). Therefore, it is best in most cases to leave this bit turned off to allow direction checking to be turned on (value = 0).

**Bit 3:** Setting this bit prevents jF1<sub>z</sub> (Park) in the Shutdown menu from parking this axis.

**Bit 5:** This setting overrides only the DRO display options for an axis that has bit 0 set (including the Rotary Display Mode – bit 1) so that the display does not reflect a degree symbol or any indication of the number of rotations, but appears as a linear axis.

**Bit 6:** This bit only works for Parameter 94 (4th axis). Setting this bit will cause the 4th axis to respond to Z-axis only commands just like the Z-axis, for example issuing an M25 with this bit set will cause the Z and 4th axes to go the home (G28) position.

**Bit 8:** This setting is used in conjunction with bit 10. It only works for parameter 166 (5th axis). Setting this bit on will identify this axis as a Triangular Rotary, which may either be an Articulated Head axis, or a Tilt Table axis.

**Bit 9:** This setting will hide the affected axis from the DRO display. Note that this does not prevent such an axis from being commanded to move.

**Bit 10:** In order for this setting to work, bit 8 must be turned on. This setting has meaning only for parameter 166 (5th axis). Setting this bit on will identify this axis as the controller of the angle of articulation on an Articulated Head machine. If this bit is not set, then this axis will be identified as the controller of the tilt angle on a Tilt Table machine.

**Bit 11:** In order for this setting to work, bit 0 must be turned on (i.e. axis is rotary). Also, this setting only affects axes 4 and higher. This setting indicates that this rotary axis is mounted in such a way so that its rotation center line is parallel to the X axis (axis 1). The rotation center line's machine coordinate location is then defined by parameters 116 and 117. This setting enables backplot to display rotary movements encircling the X axis and also gives the needed machine geometry information to G93.1 for it to properly calculate the tool tip feedrate in 3D space in such a configuration.

**Bit 14:** This bit (in conjunction with the CSR unlock) enables the TWCS (Transformed WCS) feature for Articulated Head machines. When this feature is enabled, each individual WCS can be independently set up as either transformed or non-transformed using the TWCS=Yes/No setting in the WCS configuration screen (See TWCS in Chapter 4). However, enabling this feature bit will always cause automatic B/5th axis transformation on certain features regardless of the TWCS=Yes/No setting for each WCS. Such features are: Tool Check, M25, Digitizing, and the movements done in the Probing Cycles.

### **15.3.68 Parameters 95 – 98 (and also 156 – 159) — Autotune Move Distance/Auto Delay Calculation Move Distance**

These parameters hold the maximum distance that the control will move each axis in either direction from the starting point when either Autotune or Auto Delay Calculation is run. The default value for these parameters is 2.0 inches. Values for axes 1 – 4 are specified in parameters 95 – 98. Values for axes 5 – 8 are specified in parameters 156 – 159.

### **15.3.69 Parameter 99 — Cutter Compensation Look-ahead**

This parameter sets the default number of line or arc events for the G-code interpreter to scan ahead when Cutter Compensation (G41 or G42) is active. Values of 1 to 99 are allowed for this parameter and default is 1.

### **15.3.70 Parameters 100 – 106 and 115 — Intercon parameters**

These parameters are some of the Intercon setup parameters. See Chapter 10 for more information about these parameters. Changing values will change Intercon settings and may affect the output of the G-code program if it is reposted.

### **15.3.71 Parameter 116 — A-Axis Y Coordinate**

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Y coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

### **15.3.72 Parameter 117 — A-Axis Z Coordinate**

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Z coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

### **15.3.73 Parameter 118 — B-Axis X Coordinate**

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the X coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

### **15.3.74 Parameter 119 — B-Axis Z Coordinate**

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Z coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

### **15.3.75 Parameter 120 — Probe Stuck Clearance Amount**

This parameter specifies the distance that digitizing or probing functions will move to try to clear a stuck probe condition. A stuck probe condition exists when the probe detects a point and then moves away but the probe input has not changed. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

### **15.3.76 Parameter 121 — Grid digitize prediction minimum Z pullback**

This parameter specifies the minimum distance the Z-axis will move upward when pulling back from a surface. The digitizing function attempts to predict the slope of a part surface because time is saved when the Z-axis does not have to travel upward to the starting Z depth for every digitized

point. When probe contact is made traversing in the XY plane, this parameter specifies the minimum distance the Z-axis moves upward before attempting another XY plane move. Smaller values are better when the surface being digitized has smooth curves. Larger values are better for surfaces that have steep walls. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

### 15.3.77 Parameter 122 — Grid digitizing deadband move distance

This parameter specifies a deadband distance used for internal calculations when doing a clearance move. It is recommended that this parameter should not be changed from its default value.

### 15.3.78 Parameter 123 — Radial Clearance Move

This parameter only applies to radial digitizing and determines what type of positioning move the digitizing probe will make should it encounter an unexpected probe contact with the surface of the part during Radial Digitizing. Unexpected probe contact is defined as probe contact occurring while the probe is traversing towards the user defined center point.

With Parameter 123 set to 0: When the probe encounters an unexpected probe contact, the digitizing program stops data collection. The control then prompts the operator to jog the probe to a clear position. This can be any place inside the digitizing radius and above the part, that the probe stylus has a clear path to the defined center position. To restart data collection press Cycle Start. The probe moves in the XY plane from the position the operator placed it at, to the center position defined in the radial setup menu. After reaching the center position, the probe will feed down to the Zaxis position it was at when the data collection was interrupted. The digitizing run will resume with the probe approaching from the defined center position.

With Parameter 123 set to 1: When the probe encounters an unexpected probe contact, it will automatically move (with probe detection turned off) to the maximum Z height, then moves the X and Y-axis to the defined center position. The probe will then move to the Z position it was at when the unexpected contact occurred. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

With Parameter 123 set to 2: When the probe encounters an unexpected probe contact, it will automatically move back to the defined center position (with probe detection turned off), at its present Z height. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

Settings 1 and 2 should only be used with extreme caution because probe detection during some positioning moves is turned off, and damage to the probe or work piece could occur!

### 15.3.79 Parameter 130, 131 — 3rd/ 4th axis on/off selection

These parameters control the display of the 3rd and 4th axes, respectively. The tens digit of the parameter value specifies the label of the affected axis when it is enabled, with values 1-9 corresponding to axis labels ABCUVWXYZ. The ones digit specifies the label of the axis when it is disabled, with 0.0 meaning the axis is not switchable, 1.0 meaning it turns off (N), a 2.0 meaning manual (M), and a 3.0 meaning 2-axis with manual Z (@). Parameter 130 also supports additional modes depending upon the value of the hundreds digit. See the chart below for valid values for the Hundred's digit of Parameter 130. Note that Parameter 131 does not support the Hundred's digit. When P130/P131 is configured for axis switching, the Setup menu displays function keys F5/F6 to switch the axes.

**Example 1:** A value of 192 in parameter 130 will toggle the 3rd axis between Z and M and power off all axes. The 1 sets bit one to power all axes off, the 9 enables the 3rd axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

**Example 2:** A value of 392 in parameter 130 will toggle the 3rd axis label between Z and M and power off all axes and receive its positions from the 4th axis encoder input. The 3 sets bits two and one to power off all axes and use the 4th encoder input as a scale input, the 9 enables the 3rd axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

**Example 3:** A value of 61 in parameter 131 will toggle the 4th axis between W and N. The 6 enables the 4th axis to "W" and the 2 changes the axis label to "N" when toggled with the F6 key in the Setup menu. This will turn the 4th axis on and off.

Hundred's Digit (Parameter 130 only)	Function Description	Value of the Hundred's Digit
Bit1	Axis motor power when switching to two-axis mode.	1 = power all axes off, 0 = power 3rd off only
Bit2	Use 4th encoder input for scale input	2 = Use 4th encoder input, 0 = no manual input
Bit3	Use 5th encoder input for scale input	4 = Use 5th encoder input, 0 = no manual input

Enabled Axis:	A	B	C	U	V	W	X	Y	Z
Ten's Digit	1	2	3	4	4	6	7	8	9

Disabled Axis:	N	M	@
One's Digit	1	2	3

### 15.3.80 Parameter 128 — Dry Run PLC Bit

When non-zero, this parameter specifies the PLC bit which is checked when starting a job to determine if Dry Run mode is in effect. When in Dry Run mode, all non-thread/tap moves are converted to feed/minute at the Dry Run Feedrate (parameter 129). When in Dry Run mode, the Feedrate word in the status window is drawn in reverse video (blue letters on a yellow background).

### 15.3.81 Parameter 129 — Dry Run Feedrate

Specifies the feedrate to use for Dry Run mode.

### 15.3.82 Parameters 132–135 — Motor Heating Coefficients for axes 5-8

See parameters 20-30 for more information.

### 15.3.83 Parameters 136 — G76 Fine Bore Retract Angle

0-360 Degrees. A setting of 0 = Retract in Y+ direction.

### 15.3.84 Parameters 137 — Load Meter Filter Size

This controls the number of samples used in calculating an average output for the load meter display, as a way of smoothing it out.

### 15.3.85 Parameters 138 — DRO Encoder Deadband

This controls the deadband amount (in encoder counts) used by the anti-flicker DRO filter.

### 15.3.86 Parameter 139 — Special Dwell between Moves

This parameter turns on and specifies the amount dwell time between moves, with exceptions in special cases. This parameter is similar to specifying a G61 (Modal Decel and Stop) at the beginning of a CNC program, except that dwelling will not happen under certain conditions: (1) No dwell will happen between 2 arcs. (2) No dwell will happen if before the move that contains Z movement.

### 15.3.87 Parameter 140 — Message log priority level

This parameter controls the messages that are written to the message log, which can be accessed through the F9 - Logs function in the Utilities menu. See Chapter 15 for the list of numbered messages. Message logging can be disabled by setting this parameter to -1. The recommended log level is 4.

Value	Which numbered messages are logged
-1	None
1	Numbered messages 0-299 and 400-499 – The most serious faults.
4	Numbered messages 0-299 and 400 and higher – The most serious faults and medium severity errors.
9	All numbered messages

### 15.3.88 Parameter 141 — Maximum message log lines

This parameter is the number of lines that will be kept in the message log. If this parameter is set to 10,000, for example, the newest 10,000 messages will be retained. The CNC software will delete the oldest messages, trimming the log file to the given number of lines at startup and periodically while the CNC software is in an idle state. Parameter 142 controls the frequency of the log cleanup.

### 15.3.89 Parameter 142 — Message log trim amount

This parameter is the number of additional lines above the minimum that can be added to the log before it is reduced to the minimum size. Setting this parameter to a lower value will cause the log file to be trimmed to its minimum size more often. The higher the value, the less often the log will be trimmed. The speed of the disk drive and total size of the log file at the time it is trimmed will determine how long the log cleanup takes. Under most circumstances, using 10,000 and 1,000 for parameters 141 and 142 will provide a reasonable and useful log size with no noticeable effects on performance. If parameters 141 and 142 are set to excessively high values, the message “Trimming excess lines from log file” will be presented. This message will appear at startup and very infrequently when the CNC software is idle. Normal operation can proceed after the message disappears. If the delay is unacceptable, reduce the values of parameters 141 and 142.

### 15.3.90 Parameter 143 — DRO Properties (load meters, 4/5 digits, Distance To Go)

This parameter controls the display of the axis load meters and 4/5 digit DRO precision.

Bit	Function Description	Parameter Value
0	Enable Load Meters	Enable = 1, Disable = 0
1	Load Meter Outline	Enable = 2, Disable = 0
2	DRO 4/5 Digit Precision	5 digits = 4, 4 digits = 0
3	Mini DRO (Distance to Go)	Enable = 8, Disable = 0

Add the values of the desired properties. For example, use a value of 3 to display load meters with outlines. The value 11 will display load meters, outlines and the mini-DRO. The axis load meters will be colored green for values that are up to 70% of maximum power output, yellow for values between 70% and 90%, and red for values between 90% and 100%. The axis load meters appear below the DRO for each axis (see Chapter 1).

### 15.3.91 Parameter 144 — Comparison Rounding

This parameter determines the built in rounding for the comparison operators ('EQ', 'NE', 'LT', 'GT', etc.) in expressions. Rounding of comparison arguments is

necessary due to extremely small errors that are part of every floating-point calculation. The result of such errors is that two floating-point values are rarely exactly equal. The value of parameter 144 represents the precision of comparison in places after the decimal point. If the parameter is set to 9.0, for example, then comparison operators will declare two numbers that differ in value by less than 0.000000005 as being equal. The value 0.0 is a special value that turns comparison rounding off. When comparison rounding is off, it is up to the G code programmer to build the precision into conditional statements, for example IF ABS[#A - #B] LT 0.00005 THEN GOTO 100. When comparison rounding is off, the “EQ” usually returns “false”. If parameter 144 is set to 9, the programmer can shorten the previous example to IF #A EQ #B THEN GOTO 100.

### 15.3.92 Parameter 145—Advanced Macro Properties (Fast Branching)

This parameter turns fast branching on (1) and off (0). The other bits of this parameter are reserved for future use. If fast branching is disabled, the CNC software searches forward in the program for the first matching block number and resumes searching, if necessary, from the top of the program. For this reason, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is significantly large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled, the CNC software remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as the CNC software searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

### 15.3.93 Parameter 146 — Feed Hold Threshold for Feed Rate Override

This parameter sets the lowest value permitted as the feed rate override percentage before feed hold is engaged. Feed hold will be released when the override percentage is greater than this value.

### 15.3.94 Parameter 147—Number of Status Messages to keep in Operator Message Window

The Operator Message Window is the box of scrolling status messages that appears in the upper right corner of the Main Screen. The number of remembered status messages can be adjusted by this parameter.

### 15.3.95 Parameter 148 — Miscellaneous Jogging Options

This parameter enables and/or disables certain optional modes of jogging.

Bit	Function Description	Parameter Value
0	Unused	Should be set to 0
1	Prohibit Keyboard Jogging	Prohibit Keyboard Jogging = 2 Keyboard Jogging allowed = 0

**NOTE:** With this parameter set to zero, you need to set parameter 170 to enable keyboard jogging.

### 15.3.96 Parameter 149 — Spindle Speed Threshold

This parameter defines the spindle speed threshold percent for the “Slave feedrate to spindle speed” function and the “Spindle up-to speed” function, both of which are enabled and disabled via Parameter 78. It is specified as a percentage of the programmed spindle speed. For example a value of 0.8 means 80 percent of the programmed spindle speed. See Parameter 78 for more details.

### 15.3.97 Parameter 150 — Backplot Graphics display options

This parameter controls the various options related to backplot graphics.

Bit	Function Description	Parameter Value
0	Sets Run Time Graphics option default to ON	Enable = 1, Disable = 0
1	Displays CSR positions in graphing	Enable = 2, Disable = 0
2	Display A and B rotations for 5 axis machines	Disable = 4, Enable = 0
3	Display Skew Correction	Enable = 8, Disable = 0
4	Display Lash/Screw Compensation	Enable = 16, Disable = 0

### 15.3.98 Parameter 151 — Repeatability tolerance for probing and radial digitizing.

Default is 0, repeatability check disabled. When disabled, only one measurement per point is taken in the probing cycles and radial digitizing. When enabled, a minimum of two measurements are taken per point and the difference (if any) is then compared to the repeatability tolerance as set in Parameter 151. If the difference is less than or equal to parameter 151 the point is stored and probing continues. If the difference is greater than parameter 151, 2 more measurements are taken and the process repeated up to a maximum of 10 times. In probing cycles, if the repeatability tolerance cannot be met the cycle is cancelled and an error message generated. In radial digitizing, the point is discarded and digitizing continues without interruption.

### 15.3.99 Parameter 153 — Probe Protection

A value of 1.0 will enable probe detection. Probe protection will cause motion to stop if the probe is tripped while moving under the following G-codes: G0, G1, G2, G3, G28, G29, G30, G53, G73, G76, G81, G82, G83, G85, G89, G173, G176, G181, G182, G183, G185, and G189. Probe protection is also in effect when using these M-codes: M25, M91, M92, M105, M106, and M128.

### 15.3.100 Parameter 154 — Touchscreen Options

If = 1 a half-width ESC key is displayed in most menus so there is a means of exiting the menu without using the keyboard. Most of the menus have been reworked to more fully support touch.

### 15.3.101 Parameter 155 — Probe Type

This parameter specifies the type of probe being used.

Value	Meaning
0	Standard Mechanical probe
1	DSP probe
2	DP-7 probe

### 15.3.102 Parameters 156–159 — Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5-8

See parameters 95 - 98 for more information.

### 15.3.103 Parameter 160 — Enhanced ATC

This parameter controls enhanced automatic tool changer (ATC) options. A value of 1 indicates a nonrandom type of ATC (carousel ATC) and a value of 2 indicates a random type ATC. A value of 0 disables enhanced ATC features. A warning is displayed when attempting to enable enhanced ATC features as these features work in conjunction with specific PLC programs. The enhanced ATC option has the following characteristics:

The beginning of an M6, whether it be a customized mfunc6. mac routine or not, flags the job file, setting the ATC error flag field to 1.

The end of an M6, whether customized or not, performs the following:

- The ATC error flag is set to zero.
- The tool number displayed on the screen is updated and this value is saved in the cncm.job file.
- The tool library bin fields are updated in this manner:

If there was a valid tool in the spindle at the start of the M6, then the tool library bin field for this tool will be updated with either the “putback” field for that tool (if nonrandom type) or the current ATC carousel position (for random type). For both random and nonrandom types, the “putback” field is set to 0. The “putback” field is an internal field for each tool in the tool library. It can be displayed by using the cncfcgutil utility with the -dt

option to display the tool library.

For nonrandom types, the new tool now in the spindle will have its “putback” field updated to the current ATC carousel position.

For both random and nonrandom types, the new tool now in the spindle has the bin field set to 0.

The current ATC carousel position is constantly monitored. When there is a change, the ATC bin field in the cncm.job file is updated and the file is saved. The ATC carousel position is read from the PLC system variable SV ATC CAROUSEL POSITION, which should be written by the PLC program. At the start of running a job, to include MDI mode, the ATC error field is checked. If this field is 1, then a warning message is displayed with a prompt to either clear the fault by entering a ‘Y’ or canceling the job by pressing some other key.

A tool change is not performed if the requested tool is already in the spindle.

An M107 command sends the bin number for the specified tool number, not the tool number

For random types, tool changes in Intercon are posted as a tool change (Tnn M6) followed by a pre-fetch command for the next tool in the program (Tn2 M107). This allows the PLC program to rotate the tool carousel to the next tool while a job continues with the current tool.

For random types, a job search for a tool number will look for lines of the form Tnn M6, i.e., the search bypasses lines of the form Tnn M107, which are just pre-fetch commands.

The tool library allows editing of the bin fields to specify which carousel bin number the tools are stored in.

### 15.3.104 Parameter 161 — ATC Maximum Tool Bins

This parameter sets the number of tool changer bins (carousel positions) used with the enhanced ATC option described above. PLC programs are responsible for reading this value. The tool library interface uses this parameter to validate bin fields and perform initialization of the bin fields.

### 15.3.105 Parameters 162—Intercon M6 Initial M-Code

This parameter affects how Intercon programs post M& G-codes for a tool change operation. When set to a non-zero value, Intercon will post out an M-code at the start of a tool change and an M5 command after the Txx M6 command. This parameter should be set to 17 so that Intercon will post an M17 code at the start of a tool change. The M17 command turns off spindle and coolant and starts the spindle orientation process.

### 15.3.106 Parameters 164 — ATC Feature Bit

This parameter controls specific features of the ATC system. It should be set to 1.0 to enable the ATC Reset feature, which will appear as the F6 - ATC Reset function key in the tool library menu. This parameter only works with ATC3 plc programs.

### 15.3.107 Parameters 165—Acceleration/Deceleration Options

This is a bit field parameter which modifies certain details of axis acceleration and deceleration when an axis stops moving, changes direction, or starts moving. The Jog Parameters screen in the Machine Configuration set the original DeadStart values for each axis. This parameter allows you to modify these DeadStart settings under certain conditions. Note that if both Bits 0 and 1 are turned on (value = 1+2 = 3), the effect is cumulative, i.e. the net effect will be that 1/2 DeadStart value will be used when a slave axis stops or starts up from a stop. Likewise, if both Bits 2 and 3 are turned on, the effect will be cumulative also. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

Bit	Function Description	Parameter Value
0	Use 1/4 DeadStart value for a slave axis that stops or starts from a stop	Enable = 1, Disable = 0
1	Use 2 x DeadStart value for a slave axis that stops or starts from a stop	Enable = 2, Disable = 0
2	Use 1/4 DeadStart value for a slave axis that reverses	Enable = 4, Disable = 0
3	Use 2 x DeadStart value for a slave axis that reverses	Enable = 8, Disable = 0
4	Limit the feedrate along the path of G2 or G3 arc moves such that the feedrate will be uniformly limited to the lesser of the maximum rate of the 2 axes involved in the circular motion.	Enable = 16, Disable = 0

### 15.3.108 Parameters 166–169 — Axis Properties for axes 5–8

See parameters 91–94 for more information.

### 15.3.109 Parameters 170–179 — PLC Parameters

These parameters are especially reserved as a space for data which is to be sent to the PLC. Parameters 177, 178, 179 have been standardized for specific applications. Parameter 177 is used for trouble shooting purposes only.

### 15.3.110 Parameter 170 — Enable Keyboard Jogging and set Feedrate over ride Control

This PLC parameter is used to enable keyboard jogging and determine whether jog panel or keyboard feedrate over ride is used. To enable keyboard jogging set parameter 148 to zero and this parameter to a 1.

Bit	Function Parameter	Value
0	Enables Keyboard jogging	Enable = 1, Disable = 0
1	Only looks at Feedrate over ride from Jog panel	Enable = 2, Disable = 0
2	Only looks at Feedrate over ride from keyboard	Enable = 4, Disable = 0

### 15.3.111 Parameter 178 — PLC I/O configuration

This parameter can be use to set switch types from NC to NO and some other options. Each Bit corresponds to a different function. All values are to be added to the current setting. For example, if you need to switch the low lube input to normally open add 1 to this parameter.

**NOTE:** This parameter works only with specific PLC programs. The PLC program installed in the control MAY NOT be mapped as indicated below. These parameters should only be changed by a qualified technician. The example given below is intended for reference only:

Bit	Function	Default state	Opposite State
0	Lube Fault	Closed = OK	Add 1
1	Spindle Fault	Closed = Fault	Add 2

### 15.3.112 Parameter 179 — Lube Pump Operation

This parameter can be configured to control a variety of lube pumps. The value is formatted as MMMSS, MMM for minutes and SS for seconds. Below is a table of some examples.

Type of Pump	MMM	SS	Operation
Mechanical/CAM	0	0	179=0 Power is on when machine is running a job or in MDI Mode
Electronic "Lube First"	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Electronic "Lube Last"	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Direct Controlled Pump	30	15	179=3015 Waits for 30 min of job or MDI time, then applies power for 15 seconds.

### 15.3.113 Parameters 186 — Probe Stuck retry disable

This parameter is used to disable retries when a probe is detected to be in a "stuck" condition. A probe "stuck" condition occurs during a probing move when a probe's red light (LED) stays on even after the probe has moved clear of the contact surface. The control can sometimes detect this condition and go through a series of corrective moves to "reseat" the probe and retry the probing move. If this parameter is set to a non-zero value, then the control will not do this corrective action nor attempt another probing move. If this parameter is set to 0, then the control will go through the corrective action and retry the probing move up to 5 times.

### 15.3.114 Parameters 187 — Hard Stop Homing

This parameter is used when homing off hard stops. The value set in this parameter determines the amount of current sent to the motor while homing. Value range is 0-32000; typical value for a DC system is 16000. Note that this feature does not work with velocity mode drives.

### 15.3.115 Parameters 188–199 — Aux Key Functions

These parameters are used to assign a function to aux keys 1-12 (i.e. P188 = Aux1 . . . P199 = Aux12). The following is the list of possible functions that can be executed when an aux key is pressed.

For example, if you wanted Aux4 to call up the “One Shot - Circular Pocket “, you would set parameter 191 to 7.

The Input Axis Position functions must be used with the Set ABS/INC Zero functions. After entering the desired value at the input field provided by the Input Axis Position function, press an aux key assigned either the function Set ABS Zero or Set INC Zero.

\* m is the number of the M code to execute. For example, if the parameter value is set to 7211, the file mfunc72.mac will be loaded and executed when the Aux key was pressed. Custom overlays with the keys that represent these functions are available; contact your dealer for pricing.

### 15.3.116 Parameters 200–207 — OPTIC 4 Tach Volts Per 1000 RPM

These parameters control the digital Tach output on the Optic4 boards. They are used on drives like old Fanuc velocity mode drives that require a tach input. The value put here is the volts/1000 RPM off of the motor. A negative value can be entered to invert the tach voltage compared to the encoder count derived velocity direction from the encoder.

### 15.3.117 Parameters 208–215 — MPU-based Lash/Screw Compensation Acceleration Coefficient

These parameters control the speed of the Lash and/or Screw Compensation for axes 1-8. The lash will be taken up with acceleration equal to the coefficient multiplied by the acceleration rate for the axis. A value of zero would effectively disable MPU-based Lash and/or MPU-based Screw Compensation. \*NOTE: These coefficients are not used by PC-Based Lash nor PC-based Screw Compensation

### 15.3.118 Parameters 216 — PC Based Lash Compensation on/off

This parameter controls which Lash Compensation Algorithm to use. The default value of 0 is recommended because it allows lash compensation to occur during any kind of motion. If PC Based lash is used then only during an MDI or programmed move (but not during jogging) will lash compensation be applied.

Function	Value
Use MPU-Based Lash Compensation	0
Use PC-Based Lash Compensation	1

\*NOTE: Lash/Screw Compensation Acceleration Coefficients (parameters 208-215) are not used by PC-Based Lash Compensation.

### 15.3.119 Parameters 217 — PC Based Screw Compensation on/off

This parameter controls which Screw Compensation Algorithm to use. The default value of 0 is recommended because it allows screw compensation to occur during any kind of motion.

Function	Value
Use MPU-Based Screw Compensation	0
Use PC-Based Screw Compensation	1

### 15.3.120 Parameter 218 — USB MPG Options

A non-zero value specifies that a Wireless USB MPG is being used. When set, the CNC12 software will load the driver (MpgClient.exe) used to communicate with the Wireless USB MPG. The PLC program may also use this parameter to determine the type of MPG connected to the system. A restart of CNC software is required for changes to this parameter to take effect. Typical values are seven (P218 = 7) for a three axis system and 15 (P218 = 15) for a four axis system.

Notes: 1.) also Set P348 = 15 to tell CNC12 to look for an MPG. 2.) Wireless USB MPG support is a software unlock.

### 15.3.121 Parameter 219 — Virtual Control Panel Options

When set to 1.0, CNC software will launch the Virtual Control Panel (VCP) at start up. The VCP is an on-screen equivalent of an actual jog panel that allows the use of mouse clicks (or touches on a touch sensitive screen) to control the same things as the real jog panel. Use of the VCP requires support from the PLC program.

### 15.3.122 Parameters 220–231 — Smoothing Parameters

These parameters are used for controlling the behavior of the Smoothing feature used during feed per minute moves. In particular, parameter 220 turns Smoothing on or off. When Smoothing is turned on, extreme care must be practiced to ensure that the rest of the Smoothing parameters are set to reasonable values, or else damage to the machine may result. For further Smoothing information, please see the sections “Smoothing Configuration Parameters” and “Smoothing Setup Menu” later in this chapter.

### 15.3.123 Parameters 236–239 — Motor Cooling Coefficients for axes 5-8

See parameters 20-30 for more information.

### 15.3.124 Parameter 240 — Rigid Tapping Accel Rate Distance

Default is 0.1000

Setting to -1 for servo motors this will disable the parameters 240, 241. This will introduce banging at the higher spindle

speeds as before. For stepper motors do not set to a -1 because this will cause loss of axis position.

This parameter is used for servo motors to eliminate the banging at the start and end of tapping and threading.

This is the distance to get this axis up to speed to sink with the marker pulse on the spindle.

For servo-motors this allows for higher spindle speeds to eliminate the banging. The value that may be used is 1/2 to 2 times the pitch. With higher spindle speeds this value will need to be increased if banging is heard.

This value can be set to cover a range of different pitches of taps and threads.

Also, the clearance amount should be increased to allow enough distance to get the axis up to speed to be synced with the spindle.

For stepper-motors this will allow for larger values to help with the possibility of loss of axis positions.

### 15.3.125 Parameter 241 — Rigid Tapping Rotational Step Size (Degrees)

Default value: 10.00000 Minimum value: 10.00000

This is the degrees of rotation that the pitch is changed to get its axis synced up with the spindle to help eliminate the banging at the start and end of the tap and thread. A smaller value is preferred. With high RPM and a coarse pitch thread this may have to be increased to eliminate the banging.

### 15.3.126 Parameters 252–255 — Autotune Ka Performance parameters for axes 5-8

See parameters 87–90 for more information.

### 15.3.127 Parameters 256 — Drive Mode

This parameter indicates to the control software what mode the drives are operating under. It also controls the availability and behavior of the F5 Tune key in the PID Menu based on the drive mode.

Drive mode	Value
Torque mode. The autotune feature is enabled, accessible via F5 in PID Menu	0
Velocity mode. Autotune feature is disabled. Autotune does not work with velocity mode drives and therefore this parameter should be set to 1 to prevent access to Autotune on such machines with these drives.	1
Precision mode. This enables the Precision Mode delay parameters 340–347. This also enables the Auto Delay Calculation feature, accessible via F5 in PID Menu, which is used to automatically calculate these parameters.	2

### 15.3.128 Parameter 257 — TT1 connection detection PLC input

This parameter stores the input for the TT1 connection detection feature. The spindle inhibit parameter (parameter 18) must be set (non-zero) for this feature to work. The

default for this parameter is 0, which disables this feature. When this parameter is set (non-zero), the Tool Measuring cycle will not run unless a TT1 or a probe is connected. A negative value must be entered if a “normally closed” input is to be used with the control. A positive value must be entered if a “normally open” input is to be used with the control. The absolute value of this parameter will directly reflect the PLC input the TT1 connection detect is wired to.

### 15.3.129 Parameter 258 — Velocity/Torque Mode override in Precision mode

This is an axis bitfield where setting a bit to ‘on’ allows the corresponding axis to run as if it were in velocity or torque mode, but prone to servo mismatch errors. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth.

### 15.3.130 Parameter 259 — Manual Axis Designation

This parameter is a bit field that designates an axis as a manual axis regardless of its label. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth. (Note that there is another way to designate an axis as a manual axis, which is to set its label to “M” or “@”.)

### 15.3.131 Parameter 260 — Accelerated Graphics Backplot

This parameter controls the user interface that is presented when F8 – Graph is pressed. See Chapter 3 for more information about Accelerated Graphics Backplot.

Function	Value
Use the original Graphics Backplot	-1
Use the new Accelerated Graphics Backplot	1

### 15.3.132 Parameter 261 — Precision Mode Standoff Tolerance Percentage

Indicates the percentage of the motor encoder counts per revolution for which standoff error must exceed before any correction is made. Setting P261 to zero disables standoff error correction. Suggested setting is = 0.01

### 15.3.133 Parameter 262 — Precision Mode Standoff Delay Time

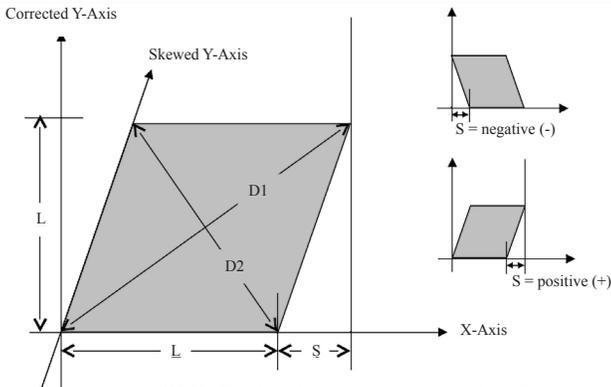
Time in seconds that an axis must be continuously at rest (no change in expected position) before a correction is made. Suggested setting is 0.1

### 15.3.134 Parameter 270–271 — XY Skew Correction

These parameters work together to correct XY position skew, which can occur if the X axis is not exactly perpendicular to the Y axis (or vice versa). To turn on XY skew correction, use the chart and follow the skew measurement procedure described below. To turn off XY skew correction, set both parameters to 0.

Parameter	Symbol	Description
270	S	+/- X skew deviation from perfect X and perfect Y
271	L	Y length over which to apply a position correction to counteract a skew of amount S

Skew Measurement Procedure: Program a sufficiently large L x L square and cut it on a scrap piece of material using the machine in question. Put this “square” piece against a true square corner and measure the skew S. If the square is leaning to the left, then S is negative; if it is leaning to the right, then S is positive. Set Parameter 270=S and Parameter 271=L.



(NOTE: The skew shown above is exaggerated for illustration purposes. Actual skew usually is not visually detectable unless it is measured.)

If you have trouble directly measuring S, you can calculate it by measuring the diagonals D1 and D2, and then using the following formula:  $S = (D1^2 - D2^2)/(4L)$

### 15.3.135 Parameter 278 — Spindle Speed Display Precision

This sets the number of digits after the decimal point to display on the Spindle Speed display in the Status Window. A setting of 0 means to show whole number spindle speeds.

### 15.3.136 Parameter 284–291 — Brake Resistor Wattage for ACDC Drives 1–8

These parameters specify the brake resistor wattage which default to the minimum internal resistor value. If CNC11 detects that the estimated brake wattage exceeds these parameter settings, then a “470 axis(drive)brake wattage exceeded” message is reported in the status window. These warnings may be written at most twice a minute. User variables #27201-#27208 can be used in an M- or G-code program to return the estimated brake wattage as reported by the ACDC drive to MPU11. Note that these are drive numbers, and match the LED indicator on the ACDC drive, but do not necessarily match the axes as displayed in the DRO due to the flexibility of drive mapping.

### 15.3.137 Parameter 300–307 — Drive assignment to Axes 1-8

These parameters control to what physical drive the commands for motion are sent. Parameter 300 assigns a physical drive to axis 1, parameter 301 assigns a physical

drive to axis 2, and so on. The values for these parameters can be set to any value from 1-25 based on the table below. These parameters must be set before attempting to move motors. Note that if you change any of these values, the machine must be powered down and restarted for the changes to take effect. Contact your dealer before changing these values.

Drive Number Assignment	Drive Type and Location	Description
1,2,3,4,5,6,7,8	Drive Bus Channel 1-8	Drive types are DC3IOB, DC1, ACSINGLE, OPTIC4, and OPTICDIRECT. Other drive types may be added in the future.
9, 10, 11, 12	GPIO4D/RTK4 Drive Out 1-4	Drive output is implemented as outputs on the PLC bus, which are updated at 4000hz.
13	RTK4 Drive Out 5	5th output for RTK4 drive
14-16	Reserved	
17, 18, 19, 20	Legacy DC 1-4	Examples of Legacy DC drives are: QUADDRV1, SERV01, M15DRV1, DC3IO, SERV0LV, SERV03IO, DCSINGLE.
21,22,23,24,25	Legacy AC 1-5	Examples of Legacy AC drives are: SD1, and SD3.

### 15.3.138 Parameter 308–315 — Encoder assignment to Axes 1-8

These parameters control to which encoder the axis should look for feedback. Parameter 308 assigns an encoder to axis 1, parameter 309 assigns an encoder to axis 2, and so on. The values for these parameters can be set to any value from 1-15 based on the table below. These parameters before motors can move.

Note that if you change any of these values, the machine must be powered down and restarted for the changes to take effect.

Encoder Number	Location	Description
1	MPU11 onboard encoder 1	Encoder inputs on the MPU11.
2	MPU11 onboard encoder 2	
3	MPU11 onboard encoder 3	
4	MPU11 onboard encoder 4	
5	MPU11 onboard encoder 5	
6	MPU11 onboard encoder 6	

7	Drive Bus Channel encoder 1	Encoder inputs on Drive bus devices.
8	Drive Bus Channel encoder 2	
9	Drive Bus Channel encoder 3	
10	Drive Bus Channel encoder 4	
11	Drive Bus Channel encoder 5	
12	Drive Bus Channel encoder 6	
13	Drive Bus Channel encoder 7	
14	Drive Bus Channel encoder 8	
15	MPU11 onboard MPG encoder	MPG connector with no index pulse
16 – 21	Encoder Expansion 1 – 6	Encoder inputs on EncExp

**NOTE:** Here are some examples:

- One DC3IOB would occupy 3 Drive Bus Channel encoder locations numbered 7,8,9.
- Two DC3IOB's chained together would occupy 6 Drive Bus Channel encoder locations numbered 7,8,9,10,11,12.
- An OPTICDIRECT occupies 1 Drive Bus Channel encoder location.
- Chaining 8 OPTICDIRECTs together would occupy 8 encoder locations numbered 7-14.
- Legacy AC drives (such as SD1 or SD3) occupy 6 encoder locations numbered 7-12.
- Note that every Drive Bus device takes up a Drive Bus encoder location even if there is no encoder going to the drive.

**15.3.139 Parameter 316 — Absolute Encoder Bits**

This is a bitfield parameter that is used to mark an encoder as being absolute. Note that it is the encoder number or index (see parameters 308-315 above) that is specified as being absolute, not the drive number. For example, if the X, Y, and Z axes had absolute encoders and parameters 308, 309, and 310 were set to the values 7, 8, and 9, then this parameter would be set to 448 (26 + 27 + 28).

**15.3.140 Parameter 317 — Single Turn Absolute Encoder Bits**

This is a bitfield parameter that is used to mark an absolute encoder as being single turn. Note that it is the encoder number or index that is specified as being absolute, not the drive number. This parameter applies to and has an effect for a scale encoder connected to a rotary axis. A motor encoder that is marked as absolute in parameter 316 should not be marked as single turn in this parameter.

**15.3.141 Parameter 318 — Five Axis Configuration**

This parameter specifies configuration settings specific to five axis systems. On five axis systems that have the fifth axis (usually labeled B) as a rotary axis that rotates around a line parallel to the Y axis, i.e., an articulated head, values

1-8 are used to specify the drive to which the scale encoder is connected. A non-zero value also signifies that the fifth axis is a “straight rotary” (it is not a “triangular rotary” type).

**15.3.142 Parameter 323 — MPU11 Encoder Speed Filter**

This is an axis bitfield where setting a bit to ‘on’ selects the low speed filters for the corresponding axis. As a general guideline, an axis’s bit should be set unless that axis refers to a 3rd party drive. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth.

**15.3.143 Parameter 324–331 — Axis Boxcar Size**

These parameters set the maximum sample size of the boxcar filter for each axis. For most applications these values should be set to 0 (default). For applications where the motor drives are too responsive or jumpy, these boxcar filters act as a way to smooth the PID output. They average the PID output by the entered boxcar size thus reducing PID spikes. Eg. A value of 4 would add the last 4 PID values and then divide by 4 for before output. A value of 5 would add the last 5 PID values then divide by 5 before output. Note that during PID averaging, the Error Sum is not zeroed during direction reversal. Conversely, a value of 0 disables the boxcar filter for that axis and also zeroes the Error Sum on Direction Reversal. Note also that as a byproduct of averaging by 1, a boxcar value of 1 may be used to produce a true PID output while not zeroing the Error Sum on direction reversal.

**15.3.144 Parameters 332–335 — Encoder error suppression**

These parameters control suppression of various types of encoder errors on a per encoder basis. These parameters are bitfields by encoder index, NOT axis index. The mpu11 has 15 encoder indexes. For example to disable encoder faults for Encoder #5 on the mpu11, enter a 16 into the parameters 332 and 334. To suppress encoder faults and error messages for all 8 axes, enter 255 into each of the paramters mentioned in the following table.

Parameter	Function
332	Suppress encoder differential faults
333	Suppress encoder differential error messages
334	Suppress encoder quadrature faults
335	Suppress encoder quadrature error messages

**15.3.145 Parameters 336–339 — Motor torque estimation for velocity mode drives**

These parameters are intended to be used with velocity mode drives in order to faciliate a more accurate display of the axis load meter bars shown under each position in the main DRO display. If P336 = 0, then this feature is disabled and the normal PID output is displayed by the axis load meter bars. This feature is enabled if P336 is non-zero.

Parameter	Symbol	Function
336	G	Overall gain setting (0 = disable Motor torque estimation)
337	Ga	Absolute error gain
338	Gs	Error sum gain
339	Gd	Delta error gain

#### Technical details:

The axis meter bar value (V) is then calculated as:  $V = \text{abs}(100.0 * G * ((Ea * Ga + Es * Gs + Ed * Gd) / \text{integration limit}))$ , where Ea is the absolute error, Es is the error sum, and Ed is the delta error from the PID algorithm and the integration limit is from the "Limit" value set in the PID Config screen. This value V is then bound to the range 0–100.

#### 15.3.146 Parameters 340–347 — Precision Mode delay (in milliseconds) for axes 1-8

These parameters are used for synchronizing individual precision mode drives with different delays. The MPU11 will use these values and compensate for the differences in the delays. These parameters have up to a 0.25 millisecond resolution.

#### 15.3.147 Parameter 348, 351, and 354 — MPG/ Handwheel Encoder Input 1, 2, and 3

The encoder input for the MPG or handwheel. (1-15) See the encoder chart above. **NOTE:** PLC program interaction is needed to enable an MPG or handwheel.

#### 15.3.148 Parameter 349, 352, and 355 — MPG/ Handwheel Detents per Revolution 1, 2, and 3

This value is the number of clicks (detents) per revolution. It is the number of divisions or markings on the mpg or handwheel. Moving the mpg or handwheel one detent or division will cause the motor to move one jog increment (depending on the multiplier x1, x10, x100, etc). **NOTE:** PLC program interaction is needed to enable an MPG or handwheel.

#### 15.3.149 Parameter 350, 353, and 356 — MPG/ Handwheel Encoder Counts per Revolution 1, 2, and 3

This value is the number of counts generated per rotation of the mpg or handwheel. **NOTE:** PLC program interaction is needed to enable an MPG or handwheel.

#### 15.3.150 Parameters 357–364 — Axis Drive Max RPM for Axes 1-8

These parameters allow you to set the drive/motor max rate capability (in RPMs) for use by the PID algorithm for the calculation of the axis KV1 contribution. This value is independent from the axis Max Rate setting in the Jog Parameters menu, which is used by the control software. However, for those axes whose corresponding parameters are set to 0 (the default) the the PID algorithm will use

the axis Max Rate setting in the Jog Parameters for the calculation of the axis KV1 contribution. These parameters are intended for 3rd party velocity mode drives that have a different max rate setting than that of the control software.

#### 15.3.151 Parameters 365 — Drive power-on delay

This specifies the number of milliseconds that the MPU11 will wait between the moment that drive power first comes on and the start of commanded motion. However, this does not work for the case of turning off a single axis using M93, moving a different axis, and then moving the powered off axis. The default value is 0 which means no delay.

#### 15.3.152 Parameters 366 – 367 — Probe / TT1 deceleration multiplier

Parameter	Function
366	Probe deceleration multiplier: This factor adjusts the deceleration rate of probing moves coming to a stop due o probe hit.
367	TT1 deceleration multiplier: This factor adjusts the deceleration rate of TT1 tool measuring moves coming to a stop due to tool touch detect.

The normal axis acceleration rate (configured by dividing the Max Rate in the Jog Parameters screen by the Accel Time in the PID Config screen) is multiplied by the value of these parameters to determine the actual decelerations used for each situation. A value higher than 1 will cause a more abrupt deceleration than the normal axis configuration. A value below 1 will cause a gentler deceleration.

#### 15.3.153 Parameter 368 — Autonomous Digitizing Angle Adjustment

This parameter determines by what angle adjustments will be made to avoid shanking during Autonomous Digitizing **NOTE:** This parameter applies only to performance racing users.

#### 15.3.154 Parameter 369 — Tool Check Max Absolute Angle

This parameter is used in five-axis systems that have articulated heads (a rotary axis usually labeled as B) to limit the angle at which the tool check can be performed. If the absolute value of the B axis exceeds this value, a warning is displayed. This parameter also specifies a limit for the tilt angle used in five axis autonomous digitizing routines.

#### 15.3.155 Parameters 374–379 — ACDC Drive Debug Log Settings

These parameters are used by support technicians and should be left at value 0.

#### 15.3.156 Parameters 392–394 — DP-7 parameters

These are parameters specific to the DP-7 probe and are used only if parameter 155 = 2.

Parameter	Function
392	DP-7 Pullback Distance: The distance the probe moves off of the surface after a probing move.
393	DP-7 Pullback Feedrate: The feedrate for the pullback move.
394	DP-7 Measuring Feedrate: The feedrate for the slow measuring move.

### 15.3.157 Parameters 395 — Probing Setup Traverse Speed

This sets the probing traverse feedrate for the macro-based probing cycles on engine block systems.

### 15.3.158 Parameters 396 — Probing Setup Plunge Speed

This sets the probing plunge feedrate for the macro-based probing cycles on engine block systems.

### 15.3.159 Parameters 398 — Port/Block mode

This determines the current mode of Port/Block systems and is set by the Port/Block menu. This parameter should not be manually modified.

Value	Mode
0	3-Axis
1	Engine Block
2	Tilt Table

### 15.3.160 Parameters 399 — AD1 arc chord tolerance adjustment

This parameter adjusts the precision of AD1 arcs. When Smoothing is turned off (P220 = 0) arc moves (such as G2 and G3) are generated as a string of many small linear moves that are used to closely approximate the programmed arc. These small linear moves are called arc chords. These arc chords straddle each side of the theoretical true arc path, but their distance (in encoder counts) from the path is limited by what value is set in this parameter. The default value is .5, meaning that by default the arc chord never strays away from theoretical true arc by more than 1/2 encoder count.

### 15.3.161 Parameter 400 — Run Menu Cycle Start Enabled

Set this value to zero to disable the CYCLE START button in the Run Menu (F4 from Main Menu). For any other value, CYCLE START will be enabled on the Run Menu.

### 15.3.162 Parameter 401 — Forget last job loaded

Setting this parameter to a value of 1.0 will cause the last job loaded to be forgotten when CNC software is started and replaced with the name “no job loaded.cnc”.

### 15.3.163 Parameter 403 — Disable Keyboard Jogging Legend

This parameter determines whether or not the Keyboard Jogging Legend is launched with an Alt+j press. A value of 1.0 will prevent it from launching.

### 15.3.164 Parameter 411 — MPG Type

Set this parameter to select the type of MPG connected according to the table below.

MPG type	Value
Non-USB MPG	0
USB CWP-4	0
USB WMPG-4	1

### 15.3.165 Parameters 900–999 — PLC program parameters

These parameters are used as a way of communicating floating point values to a PLC program. The meanings of these parameters depends on how a PLC program uses them and can vary from one machine to another. One suggested use of these parameters is as a set of configuration values. The values of these parameters are saved upon modification (via a menu or CNC job) and will be retain their values even after shutdown and restart of the control software.

All remaining parameters are reserved for further expansion.

Thank you,  
Sherline Products Inc.