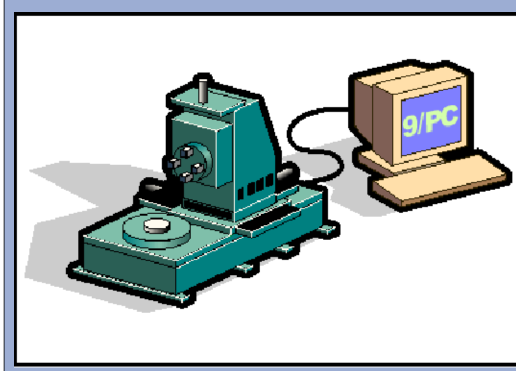
Allen-Bradley

9/PC CNC AMP

Reference Manual



Important User Information

Because of the variety of uses for the products described in this publication, those responsible for the application and use of this control equipment must satisfy themselves that all necessary steps have been taken to assure that each application and use meets all performance and safety requirements, including any applicable laws, regulations, codes and standards.

The illustrations, charts, sample programs and layout examples shown in this guide are intended solely for purposes of example. Since there are many variables and requirements associated with any particular installation, Allen-Bradley does not assume responsibility or liability (to include intellectual property liability) for actual use based upon the examples shown in this publication.

Allen-Bradley publication SGI-1.1, *Safety Guidelines for the Application, Installation, and Maintenance of Solid State Control* (available from your local Allen-Bradley office), describes some important differences between solid-state equipment and electromechanical devices that should be taken into consideration when applying products such as those described in this publication.

Throughout this manual we use notes to make you aware of safety considerations:

ATTENTION: Identifies information about practices or circumstances that can lead to personal injury or death, property damage, or economic loss.

Attention statements help you to:

- identify a hazard
- avoid the hazard
- recognize the consequences

Important: Identifies information that is critical for successful application and understanding of the product.

Summary of Changes

New Information

The following is a list of the larger changes made to this manual since its last printing. Other less significant changes were also made throughout.

- Capability of using two 1394 drives
- Using DAC monitor for multifunction monitoring

Revision Bars

We use revision bars to call your attention to new or revised information. A revision bar appears as a thick black line on the outside edge of the page.

Table of Contents

Chapter 1
Using This Manual

Manual Objective . . . 1-1
An Overview of AMP . . . 1-1
Terms and Conventions . . . 1-2
  Using the Screen Examples . . . 1-3
Attentions and Important Information . . . 1-4
Related Publications . . . 1-4
  Related Publications . . . 1-4

Chapter 2
Using AMP Applications and ODS Utilities

Chapter Overview . . . 2-1
Configuring an ODS Project for 9/PC . . . 2-3
    If you need to change your project information: . . . 2-5
Selecting the AMP Application . . . 2-5
AMP Editor Utility . . . 2-5
Selecting AMP Files . . . 2-6
  Creating a New AMP File . . . 2-6
  Opening an Existing AMP File . . . 2-6
  Copying an AMP File within the Current Project . . . 2-7
  Copying an AMP File from Another Project . . . 2-7
Renaming an AMP File . . . 2-8
Deleting an AMP File . . . 2-8
Selecting a Parameter Group . . . 2-9
Setting Parameter Values . . . 2-10
Quick Edit . . . 2-11
Saving AMP Files . . . 2-11
  Save Option . . . 2-11
  Save As Option . . . 2-12
Recover Backup File . . . 2-12
  Exiting the AMP Editor . . . 2-13
Downloading AMP Files . . . 2-14
  Downloading an AMP File from the Host PC to a 9/PC . . . 2-14
  Downloading Over a Network from a Second PC . . . 2-16
  Downloading from the Serial Port of a Second PC . . . 2-16
    Setting up the 9/PC host (or Server) computer . . . 2-17
    Setting up the ODS (or Client) computer . . . 2-20
    Troubleshooting tips for RAS configuration . . . 2-22
Uploading AMP Files . . . 2-23
  Uploading an AMP File from a 9/PC to the Host PC . . . 2-23
  Uploading an AMP File from a 9/PC Over a Network . . . 2-24
  Uploading an AMP File from the 9/PC Over a Serial Connection . . . 2-25
  Documenting AMP Files . . . 2-25
  Creating a Document File . . . 2-25
  Printing or Displaying an AMP Document File . . . 2-27

Chapter 3
Configuring Axes

Chapter Overview 3-1
Selecting an Axis 3-1
Specifying Axis Names 3-3
Specifying Axis Types 3-4
Specifying Spindles 3-4
Configuring Spindles 3-4
Configuring Axes 3-5
Deleting an Axis 3-6
Copying an Axis 3-7
Configuring a Servo 3-7
Selecting Units 3-8
Selecting Control Type 3-9

Chapter 4
Axis Parameters

Chapter Overview 4-1
Axis Name 4-2
Axis Integrand Name 4-3
Axis Incremental Name 4-5
Diameter Axis Name 4-5
Rollover Value 4-6

Chapter 5
Home Parameters

Chapter Overview 5-1
Homing Concepts 5-2
Manual Homing Using a Home Switch 5-2
Mechanical Considerations 5-4
Homing Axes with Distance-coded Marker Scales 5-5
Manual Homing Sequence in Home Mode 5-6
Manual Homing Sequence in Continuous or Incremental Mode 5-7
Homing Axes with 1326 Motors with Absolute Feedback 5-8
Home Switch Method 5-8
Switchless Homing Method 5-8
Homing Sequence Parameters 5-9
Dir to Move Off Limit Switch 5-9
Home Calibration 5-10
Axis Position After Homing 5-11
Home Speed from Limit Switch 5-12
Automatic Homing Parameters 5-13
Automatic Homing Sequence 5-13
Automatic Homing Sequence (G28) with Distance-coded Markers 5-14
G28 Direction to Home 5-15
G28 Home Speed 5-16
G30 Secondary Home Position 5-17
Machine Position at DCM Scale 0 5-18

Chapter 6
Zone/Overtravel Parameters

Chapter Overview 6-1
Zones 6-3
Programmable Zone Group Axis 6-4
Number of Limit 2,3 Groups 6-6
Limit 2 Max Value 6-7
Function 6-7
Limit 2 Min Value 6-8
Limit 3 Max Value 6-8
Limit 3 Min Value 6-9
Software Overtravel Parameters 6-10
Software Overtravel Used 6-10
Positive Software Overtravel 6-10
Negative Software Overtravel 6-11

Chapter 7
Servo Parameters

Chapter Overview 7-1
Servo Feedback Options 7-2
SERCOS Parameters 7-3
Baud Rate for SERCOS 7-3
Cycle Time for SERCOS 7-4
SERCOS Transmit Level 7-5
General Servo Parameters 7-6
Standard Motor Table Values 7-6
Servo Hardware Type 7-8
Servo Loop Type 7-9
Output Port Number 7-11
Position Loop Parameters 7-13
Servo Position Loop Type 7-14
Lead Screw Thread Pitch 7-17
Reversal Error Compensation 7-18
Excess Error 7-20
Feedrate Suppression Point 7-21
Gain Break Point 7-22
Inposition Band 7-24
Initial Gain of Position Loop 7-25
Position Loop Gain Break Ratio 7-27
Feed Forward Percent 7-28
Position Loop Feedback Port 7-29
Systems Using the CNC Serial Drive 7-29
Position Feedback Type 7-31
Position Feedback Counts/Cycle 7-33
Sign of Position Feedback 7-36
Teeth on Gear for Position Feedback 7-38
Teeth on Lead Screw for Position Feedback 7-40
Analog Servo Pos. Voltage 7-41
Analog Servo Neg. Voltage 7-43
Smaller Period for DCM Scales 7-44

Larger Period for DCM Scales 7-45
Velocity Loop Parameters 7-46
Velocity Loop Feedback Port 7-46
Velocity Feedback Type 7-47
Velocity Feedback Counts/Cycle 7-49
Sign of Velocity Feedback 7-50
Teeth on Motor Gear for Velocity Feedback 7-52
Teeth on Lead Screw for Velocity Feedback 7-53
Velocity Proportional Gain 7-54
Velocity Integral Gain 7-56
VE Integrator Discharge Rate 7-58
Peak Current as a % of RMS 7-60
Analog Systems 7-60
1394 Drives 7-61
Maximum % Rated Torque (-) 7-61
Maximum Percent Rated Torque (+) 7-62
Torque Offset Percentage 7-63
Torque Off set Direction 7-64
Hard Stop Detection Torque 7-65
Hard Stop Holding Torque 7-66
Maximum Servo Acceleration 7-67
Torque Filter Cut off Frequency 7-69
Digital Servo Parameters 7-70
Load Inertia Ratio 7-71
Motor Type 7-72
Number of Poles on Motor 7-73
Maximum Motor Speed 7-74
Motor Rated Current 7-75
ID of Amplifier Rack 7-76
Servo Amplifier Type 7-77
Shunt Resistor Pack 7-78
Spindle Parameters 7-79
Spindle Type for Axis 7-82
No of Teeth on Motor Gear 7-82
No of Teeth on Spindle Shaft 7-83
Friction Parameters 7-84
Threshold for Friction Comp 7-85
Stiction Comp Torque Percent 7-87
Positive Friction Comp Percent 7-88
Negative Friction Comp Percent 7-89

Chapter 8 Jog Parameters

Chapter Overview 8-1
Jog Speeds 8-2
Lowest Jog Speed 8-2
Second Jog Speed 8-3
Third Jog Speed 8-4
Fourth Jog Speed 8-5
Highest Jog Speed 8-5
Jog Increments 8-6
Smallest Jog Increment 8-7
Second Jog Increment 8-8
Third Jog Increment 8-8
Fourth Jog Increment 8-9
Largest Jog Increment 8-10
Jog Retract 8-11
Jog Retract Velocity 8-12
Max Jogs in Retracts 8-13
Traverse Jog Speed 8-14

Chapter 9 Feedrate Parameters

Chapter Overview 9-1
Standard Motion Feedrate Parameters 9-1
Rapid Feedrate for Positioning 9-1
Maximum Cutting Feedrate 9-2
Feedrate Override Parmeters 9-4
Rapid Override in Dry Run 9-4
“ F1” Rapid Override Percent 9-5
External Decel Speed (Cutting) 9-5
External Decel Speed (Posit .) 9-6
Skip Cycle Feedrate Parameters 9-7
Use AMP Skip Feedrate 9-8
Skip Feedrate For G31/G37 9-8
Circular Error Tolerance Limit 9-9

Chapter 10 Acc/Dec Parameters

Chapter Overview 10-1
Acceleration and Deceleration 10-2
Positioning ACC/DEC Mode 10-3
Velocity Step for ACC./DEC 10-7
Linear Acceleration Ramp 10-8
Linear Deceleration Ramp 10-10
S- Curve Acceleration Ramp 10-11
S-Curve Deceleration Ramp 10-13
Manual Delay Constant 10-14
Programmed Delay Constant 10-17
Manual ACC./DEC Mode 10-18
Axis Jerk 10-19
Minimum Programmable Jerk 10-21

Chapter 11
Constant Surface Speed

Chapter Overview . . . 11-1
CSS Parameters . . . 11-2
Default CSS Axis Name . . . 11-2
P1-P9 Constant Surface Speed Axis Name . . . 11-3
CSS Radius During G00 Rapid . . . 11-4
Using Synchronized Spindles with CSS . . . 11-4

Chapter 12
Spindle 1 Parameters

Chapter Overview . . . 12-1
DAC Voltage and Spindle Gear Parameters . . . 12-2
Spindle DAC Output Ramping . . . 12-3
Voltage at Max for Gears 1-8 . . . 12-5
Spindle Deviation Tolerance . . . 12-5
Number of Gears Used . . . 12-7
Minimum and Maximum Spindle Speeds . . . 12-8
Dev. Detection Filter Time . . . 12-9
Spindle Orienting Parameters . . . 12-10
Spindle Reference . . . 12-10
Spindle Marker Calibration . . . 12-11
Orient Speed . . . 12-12
Default Orient Direction . . . 12-13
Default Orient Angle . . . 12-14
Orient Inposition Band . . . 12-14
Gain for Spindle 1 . . . 12-15

Chapter 13
Spindle 2 Parameters

Chapter Overview . . . 13-1
DAC Voltage and Spindle Gear Parameters . . . 13-3
Spindle DAC Output Ramping . . . 13-3
Voltage at Max for Gears 1 - 8 . . . 13-5
Spindle Deviation Tolerance . . . 13-6
Number of Gears Used . . . 13-7
Minimum and Maximum Spindle Speed . . . 13-8
Dev. Detection Filter Time . . . 13-9
Spindle Orienting Parameters . . . 13-10
Spindle Reference . . . 13-11
Spindle Marker Calibration . . . 13-11
Orient Speed . . . 13-13
Default Orient Direction . . . 13-13
Default Orient Angle . . . 13-14
Orient Inposition Band . . . 13-14
Gain for Spindle 2 . . . 13-15

Chapter 14
Spindle Synchronization

Chapter Overview 14-1
Controlling Spindle 14-2
Follower Spindle 14-2
Follower Orientation 14-4
Synch Gain 14-5
Default Position Offset 14-6
Maximum Deviation 14-7
Seek Tolerance 14-7
Seek Timeout 14-8

Chapter 15
Axis Program Format Parameters

Chapter Overview 15-1
Axis 1-8 Word Format 15-1
Zero Suppression and Error Modes 15-2
Leading Zero Suppression Mode 15-3
Trailing Zero Suppression Mode 15-3
Error if Letter Numeric Missing 15-4

Chapter 16
Letter Format Parameters

Chapter Overview 16-1
D-Integer and D-Word Formats 16-3
D:Integer Format 16-3
D: Word Format 16-4
E-Word Format 16-5
E: # THRDS/INCH Word Format 16-5
F-Word Formats 16-6
F: IPM/MMPM Word Format 16-6
F: IPR/MMPR Word Format 16-7
F: V/D Word Format 16-7
H-Integer and H-Word Formats 16-8
H: Integer Format 16-8
H: Word Format 16-9
Lead: Word Format 16-10
P-Word Formats 16-11
P Dwell Type: Integer/Decimal 16-11
P: Integer Format 16-11
Q-Word Formats 16-12
Q: Integer Format 16-12
Q: Word Format 16-13
Q: Thread Marker Angle Shift 16-13
R Words 16-14
R: Word Format 16-14
R: Angle Word Format 16-15
S-Word Letter Formats 16-15
S: CSS Word Format 16-15
S: Orient Angle Word Format 16-16

S: Spindle RPM Word Format 16-17
T-Word Formats 16-17
T: Tool Number Integer Format 16-17
Chamfering/Corner-R Word Letter Formats 16-18
,C/,R Word Format 16-18

Chapter 17
Plane Select Parameters

Chapter Overview 17-1
Plane Select Parneters 17-1
(G17, G18, G19) Primary Axis 1 17-3
(G17, G18, G19) 1st Axis Parallel to 1 17-4
(G17, G18, G19) 2nd Axis Parallel to 1 17-5
(G17, G18, 19) Primary Axis 2 17-6
(G17, G18, G19) 1st Axis Parallel to 2 17-7
(G17, G18, G19) 2nd Axis Parallel to 2 17-8

Chapter 18
Power-up G-code Parameters

Chapter Overview 18-1
PTO G-code 18-2
PTO G-code For Modal Group5 18-2
PTO G-code For Modal Group 6 18-3
PTO G-code For Modal Group1 18-3
PTO Plane Select G-code 18-4
PTO G-code For Modal Group 3 18-5
PTO G-code For Modal Group 8 18-5
PTO G-code For Modal Group 18 18-6
PTO WorkCoordinate 18-7
PTO G-code For Modal Group 4 18-7
PTO G-code for Modal Group 10 18-8
PTO G-code for Modal Group 13 18-8
PTO G-code for Modal Group 20 18-9
PTO G-code for Modal Group 22 18-10
PTO ACC/DEC Mode 18-11
PTO G-code for Modal Group 25 18-12

Chapter 19
Logic Parameters

Chapter Overview 19-1
Logic Only Axis 19-1
Logic Analog Output Parameters 19-2

Chapter 20
Paramacro Parameters

Chapter Overview 20-1
T-,S-, and B-code Parmacro Calls 20-3
T-, S-, B-cod May be a Macro Call 20-3
G-Code Paramacro Calls 20-4
Modality of AMP G-codes 20-5
G-code for Macro Call to #9010 to #9019 20-5
Type of Macro Call to #9010 to #9019 20-6
G-code for Macro Call 1 to 15 20-7
Type of Macro Call 1 to 15 20-9
Program Numbers for Macro Call 1 to 15 20-10
M-code Paramacro Calls 20-11
M-code for Call to #9001 to #9009 or -1 20-11
Calling AMP-defined G-,M-S-,T-, or B-codes 20-12
Func of AMP-defined G/M in MDI 20-13
Func of Called AMP Macro 20-14

Chapter 21
Tool Offset Parameters

Chapter Overview 21-1
Basic Tool Offset Setup Parameters 21-2
T-code Format 21-2
Tool Length Axis 21-3
Maximum Tool Offset 21-4
Tool Geometry Mode 21-4
Tool Wear Mode 21-6
Tool Offset Cancel 21-8
G37 Tool Gauging Cycle Parameters 21-9
Position Tolerance for Skip 1 - 4 21-9
Add to Tool Offset for Skip 1 - 4 21-10
Target Offset for Skip 01 - 04 21-11
Tool Magazine/Turret Parameters 21-12
Number of Tool Pockets 21-13
Tool Table Motion 21-13
Tool Life Monitor Parameters 21-14
Tool Number/Group Boundary 21-14
T-word Programming Method 21-15
Tool Offset Range Verification Parameters 21-16
Maximum Wear Offset Change 21-16
Maximum Geometry Offset Change 21-17
Maximum +/- Wear Offset 21-18
Maximum +/- Geometry Offset 21-19
Maximum +/- Wear Radius 21-20
Maximum +/- Geometry Radius 21-21
Maximum Radius Offset Change 21-21
Maximum +/- Radius Offset 21-22

Chapter 22
Cutter Comp/Tool Tip Radius

Chapter Overview 22-1
Compensation Basic Setup 22-1
Cutter Compensation Type 22-1
Define Offset as Diam. or Rad (Mill Only) 22-3
Minimum Block Generation Length 22-4
Minimum Feed Reduction % 22-5
Maximum Number of Nonmotion Blks 22-6
Corner Override Parameters 22-7
Corner Override Angle 22-8
Corner Override Distance (DTC) 22-8
Corner Override Distance (DFC) 22-9
Corner Override % 22-10
Compensation Error Detection 22-10
Interference Detection 22-11
Reverse Compensated Motion Detection 22-11
M-word to Disable Interference Detection 22-12
M-word to Enable Interference Detection 22-13

Chapter 23
QuickPath Plus Parameters

Chapter Overview 23-1
QuickPath Plus Parameters 23-1
QPP Angle Word 23-1
QPP Angle Word Format 23-2
L: Word Format 23-2

Chapter 24
Fixed Cycles

Chapter Overview 24-1
Milling Fixed Cycle Parameters 24-2
Retract Amount for Peck Drilling 24-2
Fine Boring Shift in Q-word 24-3
Fine Boring Shift Direction G17, G18, and G19 24-4
Cycle Clearance Amount 24-5
Ignore Dwell in Tapping Cycles 24-6
Rapid to Drilling Hole 24-7
Fixed Drilling Axis 24-8
Threading Cycle Parameters 24-9
Pullout Distance, Chamfered Thread 24-9
Pullout Angle,Chamfered Thrd 24-10
Min Infeed in Multi Threading 24-12
Finish Allow in Multi Threading 24-13
Turning Cycle Parameters 24-14
Always Repeat Turning Cycles 24-14
Retract Amount in Grooving 24-15

Chapter 25
Interrupt Paramacros

Chapter Overview 25-1
Interrupt Macro Parameters 25-2
Interrupt Enable M-code 25-2
Interrupt Disable M-code 25-2
Trigger Method for Interrupt 0 - 3 25-3
Interrupt 0 - 3 Routine Call 25-3
Interrupt 0 Service Action 25-5

Chapter 26
Roughing Cycle Parameters

Chapter Overview 26-1
Roughing Cycle Parameters 26-1
Roughing Cycles Retract Amount 26-1
Percent of Cutting Depth 26-2
Perform a Rough-Finish Cut 26-3
Roughing Cycle Threshold Depth 26-4

Chapter 27
Solid–tapping Parameters

Chapter Overview 27-1
Acceleration Time 27-2
Maximum Tapping Speed 27-3

Chapter 28
Cylindrical/Virtual C Parameters

Chapter Overview 28-1
Mill Cylindrical Interpolation Parameters 28-2
Cylindrical Feed Axis Name 28-2
Cylindrical Park Axis Name 28-2
Cylindrical Linear Axis Name 28-3
Cylindrical Rotary Axis Name 28-4
Rotary Center Feed Coordinate 28-5
Rotary Center Park Coordinate 28-5
Lathe Virtual C Parameters 28-6
Virtual C Rotary Axis 28-6
End Face Milling Axis 28-7
End Face Milling Incremental Axis 28-8
End Face Axis Integrand Name 28-8
Axis Along Center Line 28-9
Feed Axis Park Location 28-10
Automatic Home on Virtual C Entry 28-10

Chapter 29
Probing Parameters

Chapter Overview 29-1
Probe Length Compensation 29-1
Probe Radius Compensation 29-2
Probe Transition 29-2
G38, G38.1 Probing Cycle Parameters 29-3
Approach Distance (R) 29-4
Tolerance Band Distance (D) 29-5
Approach Rate (E) 29-6
Probe Rate (F) 29-6

Chapter 30
Adaptive Feed & Depth Parameters

Chapter Overview 30-1
Feed Integral Torque Gain 30-1
Feed Proportional Torque Gain 30-2
Feedrate Acc/Dec Enable 30-3
Adaptive Depth AMP Overview 30-4
  Servo Parameters for the Adaptive Depth Probe 30-5
  Controlling Axis for Probe 30-8
  Controlling Axis Trigger Dir 30-9
  Probe Trigger Tolerance 30-10
  Depth Sensor Travel Limit 30-11
  Adaptive Depth Feedback Source 30-12

Chapter 31
Dual Axis Parameters

Chapter Overview 31-1
  Positioning Dual Group Members Independently 31-2
  Per Axis versus Per Group 31-2
Dual Axis Group 31-3

Chapter 32
Deskew Parameters for Split Axes

Chapter Overview 32-1
Deskew Master Servo Name 32-4
Deskew Slave Servo Name 32-7
Deskew Gain 32-9
Excess Skew Limit 32-10

Chapter 33
Miscellaneous Parameters

Chapter Overview 33-1
General Control Operating Parameters 33-1
Lathe Type 33-2
Mill Type 33-2
Control Reset on E-Stop Reset 33-3
High-speed Input Trigger Point 33-4

Dwell Type . . . 33-4
Threading E-word Definition . . . 33-5
Secondary Auxiliary Function Word . . . 33-6
Block Delete Type . . . 33-6
Reset M&G Codes on M02/M30 . . . 33-7
Reset Coord Offset on M02/M30 . . . 33-8
Cancel Tool Offsets On M02/M30 . . . 33-9
Block Retrace Limit . . . 33-10
Scaling Allowed . . . 33-11
Password Parameters . . . 33-12
Characters of the Password . . . 33-12
Reset G92 Offsets . . . 33-13
System Timing . . . 33-13
System Timing Overview . . . 33-14
Analyzing Your System Timing . . . 33-18
Factors that Affect CNC System Timing . . . 33-18
System Timing Screens . . . 33-19
Setting Your System Timing Parameters . . . 33-20
Fine Foreground Scan Time . . . 33-20
Fine Scans per Logic Scan . . . 33-21
Logic Scan Warning Time . . . 33-21
Procedure for Determining Values . . . 33-22

Chapter 34
Open CNC Interface (OCI) Parameters

Chapter Overview . . . 34-1
Maximum Number of OCI Connections . . . 34-1
CNC-OCI Data Transfer Rate . . . 34-2
Monitor Frequency for Changes . . . 34-2
Watchlist Buffer Size . . . 34-3
OCI Foregrnd Data Buffer Size . . . 34-5
OCI Backgrnd Data Buffer_Size . . . 34-5

Chapter 35
Reserved Custom Parameters

Reserved Custom Parameters . . . 35-1

Chapter 36
Tuning AMP at the Machine

Chapter Overview . . . 36-1
Axis Parameters . . . 36-2
Online Reversal Error Parameters . . . 36-2
Home Calibration Parameters . . . 36-4
Online Axis Calibration Parameters . . . 36-5
Accessing Axis Calibration Parameters . . . 36-6
Entering Axis Calibration Parameters . . . 36-7
Changing Axis Calibration Data . . . 36-9
Inserting Axis Calibration Points . . . 36-10
Deleting Axis Calibration Points . . . 36-10
Calibration Status On/Off . . . 36-10

Online Servo Parameters 36-11
Spindle Parameters 36-12
Backup Data to/from Hard Disk 36-14

Chapter 37
Patch AMP

Patch AMP 37-1
Accessing Patch AMP 37-1
  Patch AMP Search Functions 37-2
  Editing AMP with the Patch AMP Utility 37-3
  Patch AMP Values 37-3
Writing AMP to Flash 37-15

Appendix A
Tuning a Digital or Tachless Analog System

Overview A-1
  The Velocity Loop A-3
AMP for Tuning A-4
Logic Considerations A-4
Connect the Strip Chart Recorder A-5
  Turning the DAC Monitor On A-5
  Monitoring Multiple Functions via DAC Monitor A-6
  Scaling DAC Monitor Output A-7
Create Your Tuning Part Program A-7
Tuning the Drive A-8
  Find the Maximum Velocity Proportional Gain value: A-9
  Tuning for Optimal Acceleration A-10

Appendix B
Integrating a Linear Feedback Device

Overview B-1
Linear Feedback Device Specifications B-1
Configuring the Position Loop with Linear Feedback B-2
Configuring the Velocity Loop with Linear Feedback B-3

Using This Manual

Manual Objective

This manual is designed to assist system installers in programming the Adjustable Machine Parameters (AMP) for the Allen-Bradley 9/PC CNC.

Table 1.A briefly outlines the structure of this manual and how to locate certain information.

Table 1.A
What This Manual Contains

Chapter	Content
2	• a description of the AMP Editor, which is an application program in the Off-line Development System software package (ODS) • instructions on setting up and using ODS software on the 9/PC host computer. For using ODS on a PC connected to the 9/PC host via network or serial port, see the *Offline Development System (ODS) Software User Manual* (pub. no. MCD-5.1)
3	step through process of the initial procedures required to configure axes
4 - 35	a detailed description of the various AMP parameters so they can be programmed to maximize machine performance
36 - 37	• a description of the two ways of modifying the AMP parameters directly at the machine tool's 9/PC host PC: Online AMP and Patch AMP • use these for fine tuning the system after AMP has been downloaded from ODS
App. A	• instructions for tuning digital servo systems
App. B	• instructions for integrating a linear feedback device

An Overview of AMP

The Allen-Bradley 9/PC control lets you customize control installation to nearly any type of mill, transfer line, or lathe machine tool configuration. This is made possible by using Adjustable Machine Parameters (AMP).

AMP is one of the two software utilities that a system installer can use to make the control "fit" the machine. AMP parameters allow you to define (in terms that the control can interpret) the physical details of the machine tool's axes and hardware elements as well as the many other features that are available with the control.

The other utility is for sequential machine logic (SoftLogix 5). SoftLogix 5 is used to handle the sequential program that interfaces the control and the machine tool. It is used to produce a logic program that is executed on the host PC to monitor and command peripheral devices such as tool turrets, part clamps, and coolant systems. Refer to your *SoftLogix 5 Reference Manual* and your *9/PC CNC Logic Reference Manual* for additional information.

AMP lets you:

- define basic parameters such as system resolution, axis types, and programming formats that affect the overall operation of the control

These parameters are global control parameters.

- define individual axis parameters such as soft travel limits, position tolerances, and feedback constants

These are per axis parameters parameters.

- set up an axis calibration table for each axis to adjust for mechanical deficiencies in the machine (On-line AMP only)

Default parameters are present in the control's memory, even if the control's AMP memory and backup copy are accidentally erased. The default values provide a base from which to set up the machine.

We wrote this manual to include mills, transfer line stations, and lathes. The majority of sample screens show a Mill/Lathe application type and a lathe machine type. The parameters available differ depending on the control type you select. The default control type is Lathe.

Transfer line applications use the mill configuration type. The transfer line option is selected using the Mill Type parameter found in the Miscellaneous Parameter group.

Terms and Conventions

To make this manual easier to read and understand, full product names and features are shortened where possible. Here are the shortened terms:

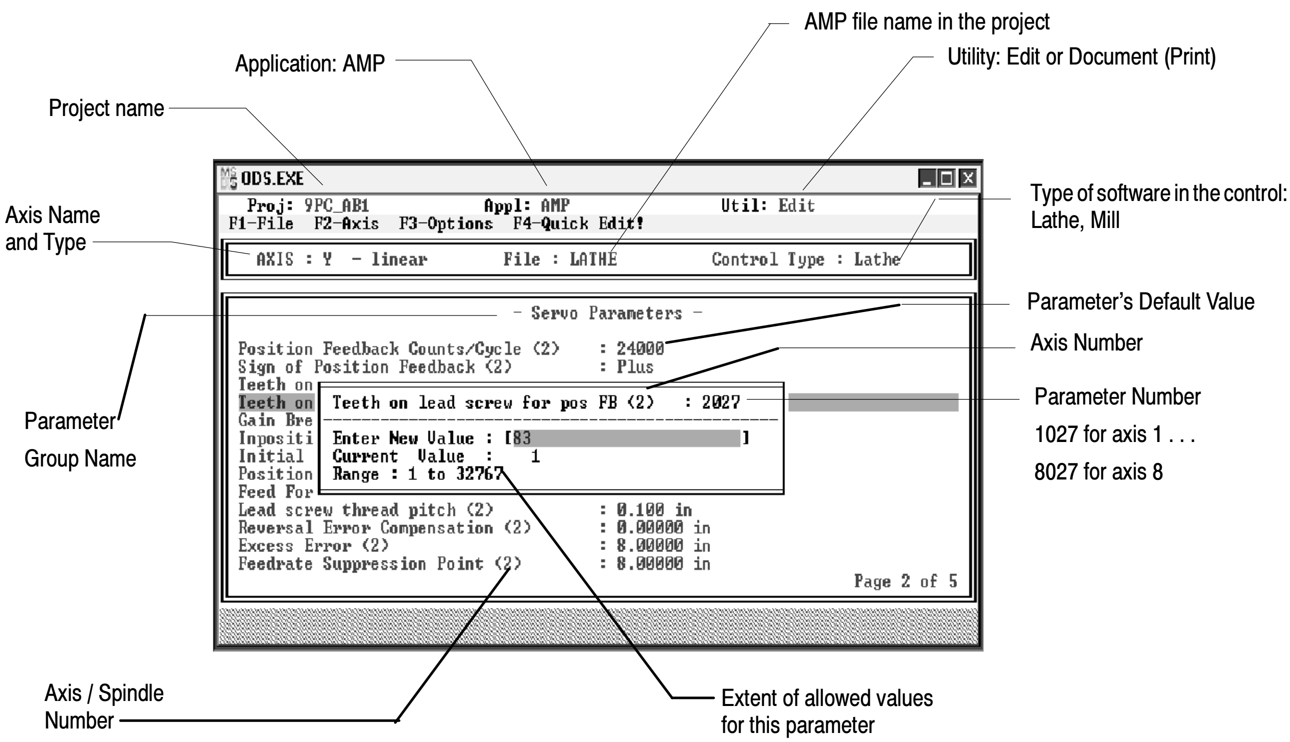
- 9/PC — The PCI card that executes part programs, and plans the axes motion
- AMP — Adjustable Machine Parameters
- BDS -- The "Basic Display Set" which is the visual basic application that makes up the standard 9/PC operator interface screens.
- CNC — Computer Numerical Control, the 9/PC card
- CRT — Cathode Ray Tube (the PCs monitor, or display device)
- Host PC -- The personal computer that contains the 9/PC card, and the I/O card. This PC executes the sequential logic and runs the operator interface software.
- The control -- The host PC plus the 9PC card, I/O card, and logic engine.
- E-Stop — Emergency Stop
- Flash memory — non-volatile, programmable memory resident on the 9/PC.

- I/O — Input/Output
- MDI — Manual Data Input
- ODS — Offline Development System
- RAM — Random Access Memory (volatile during power loss)
- Softkeys — the row of dynamically labeled functions at the bottom of the 9/PC display screen. These are activated by touch screen selection, pointer device click, or function keys.
- System Installer — the company, contractor, or person responsible for integrating this control to the machine
- Workstation — personal computer without a 9/PC, on which the ODS software package is installed

Using the Screen Examples

We show several different types of screens in this manual. Some of the screens will not match what you see on your PC screens. The screens are offered as examples only. Figure 1.1 shows two example screens and defines the terms that they use.

Figure 1.1
Terms You See in the Screen Examples



When you use ODS, some menus or parameters may not be accessible. These menus or parameters are turned off (ghosted) so that they are less illuminated than the rest of the text on the screen.

Attentions and Important Information

Information that is especially important is indicated by the following:

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

Important: Indicates information that is necessary for successful application of the control.

Related Publications

Use this manual along with these:

Related Publications

Publication Number	Document Title	Catalog Number
8520-9.1	9/PC CNC Software and Hardware Installation and Integration Manual	8520-9IM
8520-9.2	9/PC CNC Logic Reference Manual	8520-9LM
8520-9.4	9/PC CNC Lathe Operation and Programming Manual	8520-9LUM
8520-9.5	9/PC CNC Mill Operation and Programming Manual	8520-9MUM

END OF CHAPTER

Using AMP Applications and ODS Utilities

Chapter Overview

This chapter covers the AMP application and the related download and upload applications of the Offline Development System (ODS). Use these applications and their utilities to prepare an AMP file for use by the control.

This chapter contains this information:

Topic:	Page:
Configuring a 9/PC project	2-3
Selecting the AMP Application	2-5
Selecting AMP Files	2-6
Setting Parameter Values	2-10
Saving AMP Files	2-12
Exiting the AMP Editor	2-13
Downloading AMP Files	2-14
Uploading AMP Files	2-23
Documenting AMP Files	2-25

This manual assumes that you have:

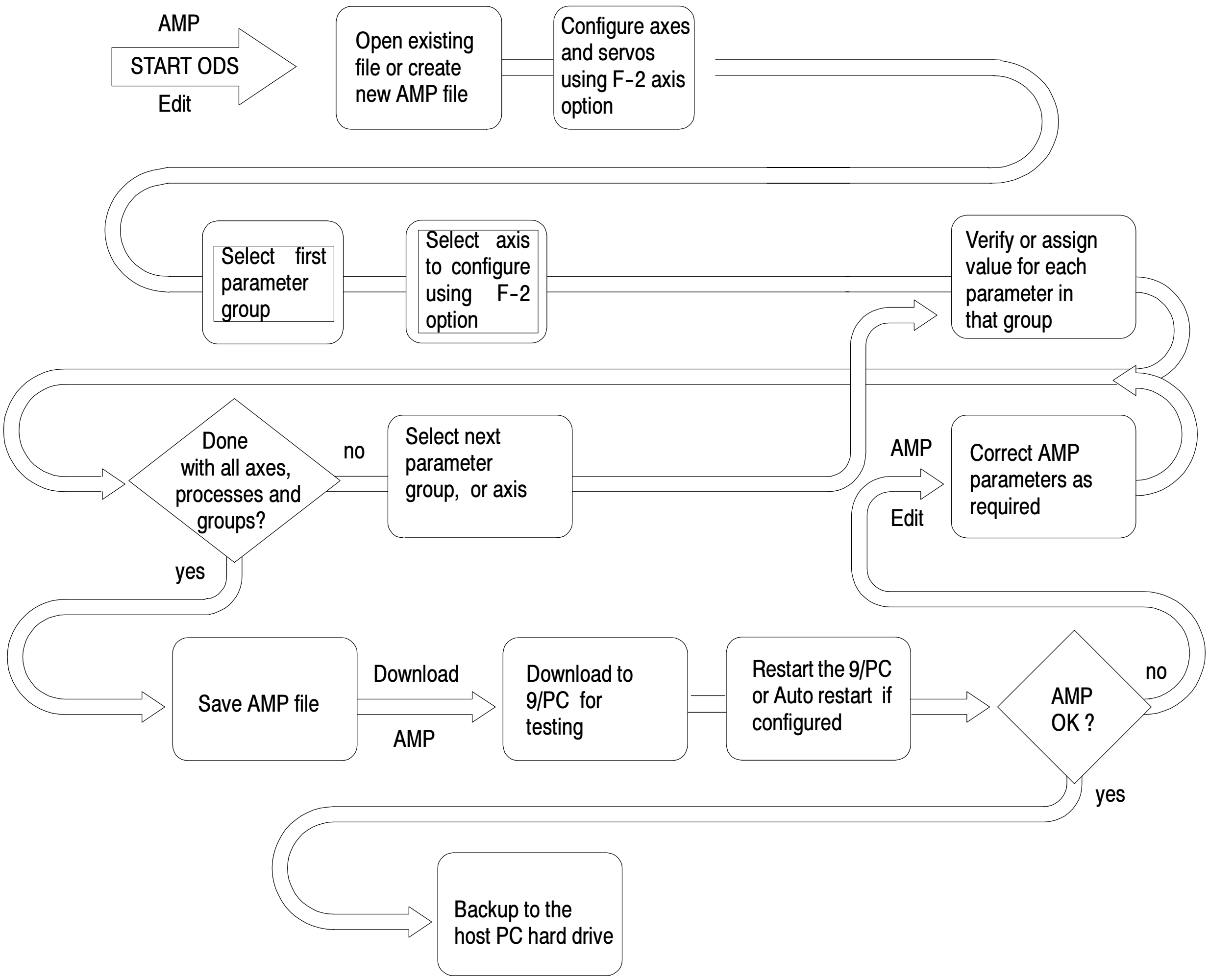
- read the *Offline Development System (ODS) Software User's Manual*, (pub. no. MCD-5.1)
- installed the ODS software in a workstation or on the host PC

Use the AMP editor of ODS to set the value for each parameter that applies to the selected AMP file. This process creates an AMP file for a specific control application.

Use the download application of ODS to download this file of AMP values to the control. The file remains stored in the control. The control uses the information in the file to execute its software according to the needs of the application.

Figure 2.1 shows the process of creating an AMP file.

Figure 2.1
Process of Creating AMP for a Control



Important: Throughout this chapter the term **select** means that you can do either of the following:

- type in the letter indicated on the pull-down menu that corresponds to the desired selection, or
- use the cursor keys to highlight the selected item on the pull-down menu, then press [**ENTER**]

The AMP parameter file is developed using the Offline Development System (ODS). The ODS software provides a full-featured AMP development environment. All AMP parameter values, except those dealing with axis calibration and fine tuning the spindle and servos, can be entered through ODS.

Important: Once AMP editing is completed, back up the project with the functions provided in ODS.

The AMP file developed with ODS is downloaded to the 9/PC. This can be done whether ODS is running on:

- the host PC
- a workstation with a network connection to the host PC
- a workstation connected to the serial port of the host PC

There are two ways to change AMP files after they have been downloaded to the control, without using ODS:

- Patch AMP

  Patch AMP should be restricted for use by qualified users with password access and Allen-Bradley field support personnel. Patch AMP is explained in chapter 37 of this manual.

- Online AMP

  Online AMP is used to modify set of parameters that are often altered after the control is powered up and the servos are energized. Online AMP is covered in Chapter 36 of this manual.

Configuring an ODS Project for 9/PC

Follow these steps to create a new 9/PC project.

1. Start the ODS application by running the file "IMC.bat, using the most convenient windows method for you; ie Double click a shortcut, select "RUN" from the start menu, etc.
2. Select **F2** to pull down the F2-Project menu. Select the **New** option. ODS displays the message **New project** and a directory of existing projects.
3. Enter a name for the new project and press `[Enter]` to continue, or `[Esc]` to cancel. If ODS accepts the project name, it displays a box that asks you to select the control type of the new project.
4. Select 9PC as the control type.

   If you select the **9/PC**, a screen appears allowing you to select the application software type.
5. Select the application type.

   At the initial release, only **Lathe/Mill** application is available. A screen appears, asking you to identify the communication method for the project. After you make this selection, ODS briefly displays the message "Creating Project Sub-Directories".
6. Select **PC to 9/PC** (p) for the communication mode.

   This selection allows ODS to communicate directly to the 9/PC card if ODS is running on the host PC, or over a network to the host PC if this is a network workstation. A selection box appears asking you to identify the host PC name.

7. Enter **"Local"** or computer name.

   This name is used as the destination PC, for any AMP download from this project.

   The default entry "local" indicates to ODS that the 9/PC card is in the same PC that ODS is running in. However, if the 9/PC is in a separate, network-connected PC, you must enter the name of that host PC here. To view the name of the host PC , under the Windows start menu, select: "settings"; "control panel"; Network; "Identification"

Important: In order to allow ODS to communicate in any way with a 9/PC, the workstation (the PC running ODS) must be running Microsoft's Windows NT 4.0 or later as an operating system. In addition, if this is a dual boot PC, NT must have been running when ODS was installed to insure that the proper communications drivers are loaded.

   When you enter the host PC name, a screen appears that asks you to identify the assigned name of the 9/PC card .

8. Enter 9/PC name.

   Enter the topic name for this 9/PC card. This is the same name that you entered when you installed the 9/PC software on the host PC. The default is "CNC_1".

9. Select the appropriate option to determine control software revision.

   You can directly select the revision level, or have ODS read the software revision number from the 9/PC.

   If you select the "read from control" option, the 9/PC card in that PC must have been started and running.

10. Continue entering the project information as described in the *Offline Development System (ODS) Software User Manual* (pub. no. MCD-5.1).

If you need to change your project information:

1. Press **[F2]** to pull down the F2– Project menu.
2. Select the **Change info** option.
3. Step through the project information data, making changes as desired.
4. Press **[Enter]** to retain the current information.
5. When the system asks you to verify your information, select YES **[Y]** to use the updated project information in your current project.

Selecting the AMP Application

Use this procedure to select the AMP application of the Offline Development System

1. Select or create a project (refer to the *Offline Development System (ODS) Software User's Manual*, and the *Configuring an ODS Project for 9/PC* section, located in this chapter).
2. Press **[F3]** to pull down the Applications menu.

3. Press **[A]** to select the AMP application.

AMP Editor Utility

To select the AMP editor utility:

1. Once you have selected the AMP application, press **[F4]** to pull down the Utility menu.

2. Press **[E]** to select the Edit option.

   The workstation displays a screen showing that it is loading the selected utility, then displays the AMP editor menu screen.

   The **F1-File** option is the only option that is enabled.

Selecting AMP Files

Before specifying any parameter values, you must use one of these options to select the desired AMP file:

- Creating a New AMP File
- Opening an Existing AMP File
- Copying an AMP File from within the Current Project
- Copying an AMP File from Another Project

Creating a New AMP File

Use this procedure to create a new AMP file:

1. Press [F1] to pull down the File menu.

2. Press [N] to select the New option.
3. Type in the name of the new AMP file; then press [ENTER].

   A file name can contain up to 8 letters and/or numbers and underscores (_).

   When you press [ENTER], ODS creates the new file and opens it for editing.

Opening an Existing AMP File

To open an existing AMP file:

1. Press [F1] to pull down the F1-File menu.
2. Press [O] to select the Open option.

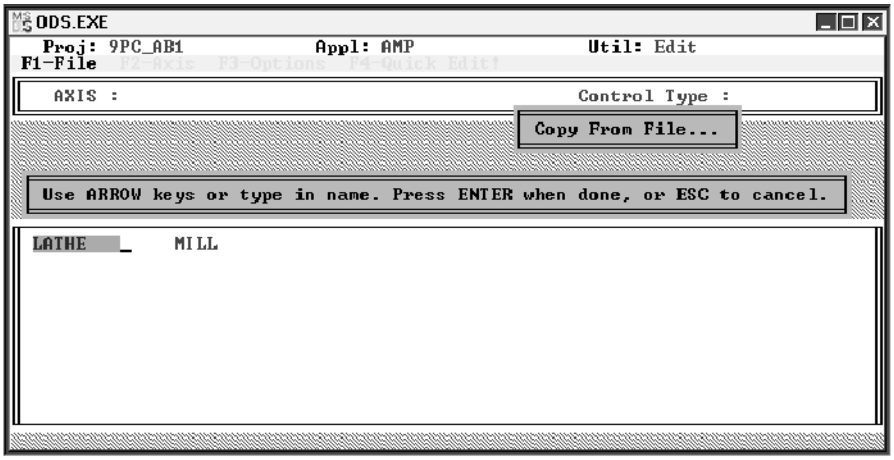
   A menu of existing files appears on the workstation screen.
3. Select the desired file.

   ODS opens the selected file for editing.

Copying an AMP File within the Current Project

To copy an AMP file from within the current project:

1. Press [F1] to pull down the F1-File menu.
2. Press [C] to select the Copy option.



Your screens may differ slightly, depending on the application you use.

3. Select the AMP file to copy.
4. Type in the name for the new file. Press [ENTER].

   A file name can have up to 8 letters and/or numbers. It may not include any of these characters:

   . " / \ [ ] : ; , | < > + =

   ODS copies the selected file and stores it under the specified name.
5. Open the file for editing.

Copying an AMP File from Another Project

To copy an AMP file from a project other than the currently selected project:

1. Press [F1] to pull down the F1-File menu.
2. Press [F] to select the Copy from Project option.
3. Select the project from which an AMP file is to be copied.
4. Select the AMP file that you want to copy.

5. Type in the name for the new file. Press `[Enter]`.

   A file name can have up to 8 letters and/or numbers. It may <u>not</u> include any of these characters:

   `. " / \ [ ] : ; , | < > + =`

   The workstation copies the selected file and stores it in the current project under its new name.

6. Open the file for editing.

Renaming an AMP File

Use the rename option to change the name of an AMP file in the active project. To rename a file:

1. Press `[F1]` to pull down the F1-File menu.
2. Press `[R]` to rename a file.
3. Identify the file to be renamed by Cursoring to the file, or typing in the name of the file and press `[Enter]`.
4. ODS asks you for the new name. Type the new name for the file and press `[Enter]`.

   ODS will rename the file if the new name does not conflict with an existing AMP filename.

Deleting an AMP File

To delete an AMP file from the current file:

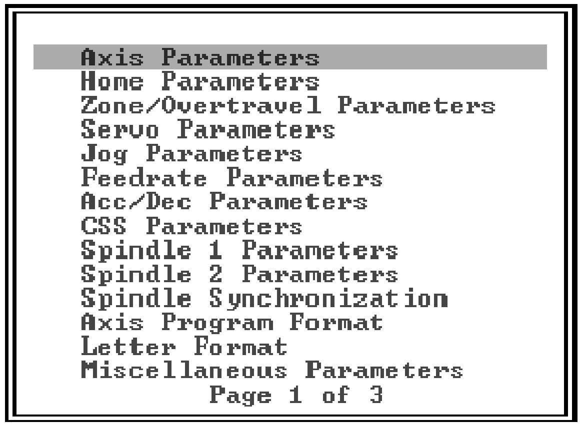
1. Press `[F1]` to pull down the F1-File menu.
2. Press `[D]` to select the Delete option.
3. The workstation displays a directory of existing AMP files for the active project. Cursor to the file (or type in the name) and press `[Enter]`.
4. Select "Yes", or "No" when ODS asks if it is OK to delete the file.

   If you select Yes, the file is deleted and the AMP editor menu screen reappears. If you select No, the workstation returns you to step 3.

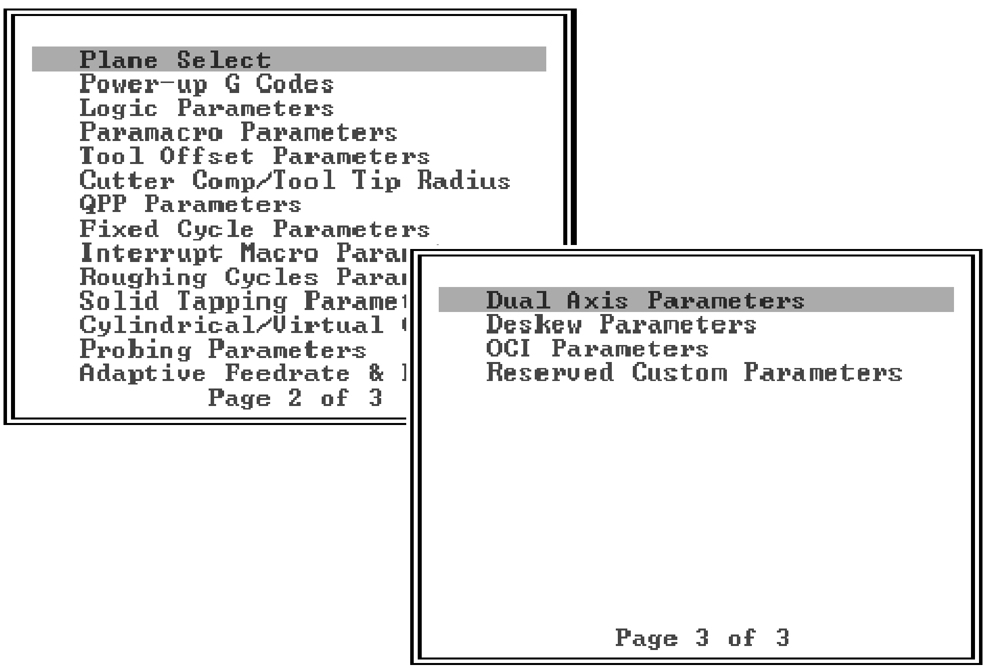
Selecting a Parameter Group

Before you can begin to set AMP parameter values, open a file.

When you open a file, a list box showing page 1 of the AMP parameter groups appears.



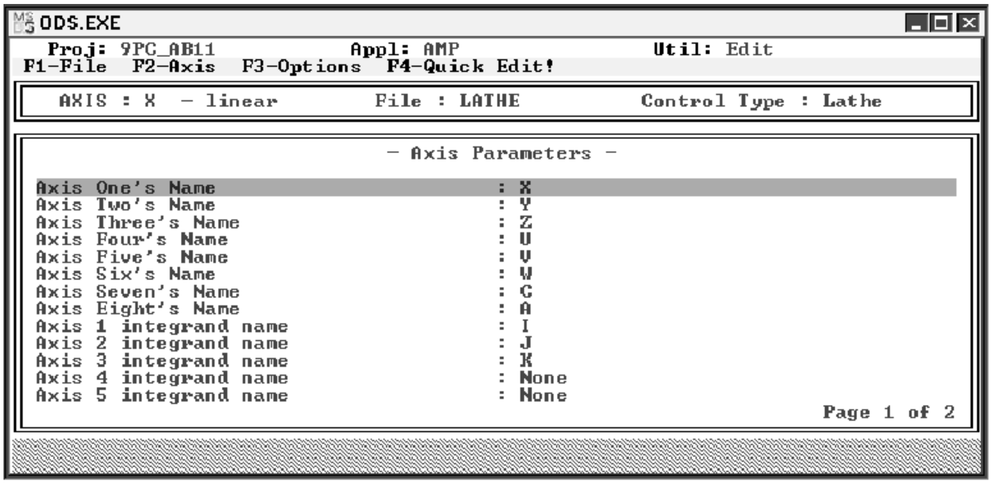
Press the [PgDn] key twice to see page 2 and twice more for page 3.



To select an AMP parameter group:

1. Move the cursor to the desired group, then press [ENTER].

   For example, when you have selected the Axis Parameters group, this screen is displayed:



Setting Parameter Values

To set a parameter value:

1. Select the parameter by cursoring to the parameter you want and press **[Enter]**.

   There are two types of parameters:

   - Parameters that prompt for a numeric value; these require that the numeric value be entered within a certain range. To specify a new value, type in the new numeric value, then press [Enter]
   - Parameters that give a choice of selections from a list of fixed values displayed on the workstation; one of these choices must be selected. To specify a new value, select the desired value, then press [ENTER]

Important: For additional information on a parameter, press and hold the [Alt] key down and then press the [H] key to display a Help message for each parameter.

2. Select or enter a new value.

   The workstation stores the new value and returns to the selected parameter group menu.

3. Press the [ESC] key to return to the group selection screen.

Quick Edit

The software refers to each AMP parameter by an identification number. All AMP parameters have an identification number that allows for Quick Editing. This capability allows you to go directly to the parameter for edits.

To edit a parameter, follow these steps:

1. Press [**F4**] to pull down the Quick Edit menu.

   The following dialog box appears:

2. Type in the number of the parameter to be edited. Press [**ENTER**].

   ODS displays the parameter name and number, and waits for a new value to be entered.

3. Select the new value.

   The workstation stores the new value and returns to the current AMP editor menu.

4. To halt the Quick Edit process, press [**ESC**].

Saving AMP Files

After editing the open AMP file, it must be saved to store the new parameter values. There are two ways to save the file:

- Save — saves the edited file under its original name.
- Save As — saves the edited file under a new name, and retains the original, unedited file under its original name.

Save Option

1. Press [**F1**] to pull down the F1-File menu.
2. Select the Save option.

   The workstation saves the edited file under the original name. The original, unedited version of the file is made into the backup copy.

Important: Use the Save function periodically to update the AMP file during a long editing session. This helps to guard against accidental loss of data due to unforeseen faults, such as a power failure.

Save As Option

To save the original unedited file and the edited file:

1. Press [F1] to pull down the F1-File menu.
2. Select the Save As option; the workstation waits for you to enter the new name for the edited file.
3. Type in the name given to the edited file; then press [ENTER].

   A file name can contain up to 8 letters and/or numbers and underscores (_).

   When you press [ENTER], the workstation saves the edited file under the specified name. The original unedited file remains in memory under its original name.

   Exit the AMP editor after saving the open file.

Recover Backup File

When you save an AMP file, the original file is saved as a backup file, then overwritten with the edited file. To recover the AMP file that was overwritten during the most recent save operation, use the Recover Backup File option.

1. Press [F1] to pull down the F1-File menu.
2. Press [B] to select the Recover Backup File option. The workstation displays a directory of AMP backup files.
3. Select the file you want to recover.

The workstation renames the selected backup file to a source file. If the source file already exists under the selected name, the workstation asks you if it is OK to overwrite the existing file with the backup file.

4. Select Yes or No.

   If you select Yes, the workstation overwrites the existing file with the backup file. If you select No, the workstation aborts the recover operation. The AMP editor menu screen reappears.

Exiting the AMP Editor

You can exit the AMP editor before or after you save the open AMP file. If you did not save the open AMP file, the workstation displays a screen that lists the options of saving or discarding the open file.

To exit the AMP editor:

1. Press [F1] to pull down the File menu.

2. Select the Quit option.

If you have not edited the open file, the workstation terminates the edit session and clears the edit window from the screen. You can select another utility.

If the open file has been edited, the workstation displays this dialog box:

```
        W A R N I N G
   Opened file has not been
           saved.
------------------------------
      Save            (S)
      Save As...      (A)
      Discard         (D)
```

3. Select the desired option.

The Discard option quits the open file without saving any editing changes.

If you select the:	and the Verify option of the F5-Configuration menu is:	Then the workstation:
Discard option	OFF	• discards any changes • closes the open file • exits the AMP editor • returns to the main menu line of ODS
	ON	displays a confirmation screen:

4. On the confirmation screen, you have these choices.

If you want to:	select:	the workstation:
• discard any changes • close the open file	[Y]	• exits the AMP editor • returns to the main menu line of ODS
terminate the quit procedure	[N]	returns to the screen that was displayed before the quit procedure started

Downloading AMP Files

After you edit and save your AMP file, download the it to the 9/PC so that it becomes the currently active AMP file. You can use ODS to download AMP files in these three ways:

- from the local host PC into the 9/PC
- from a separate network-connected ODS workstation that is running Windows NT to the host PC, and into the 9/PC card
- from an ODS workstation running Microsoft Windows NT and Dial-up Networking, connected to the serial port of the host PC running Microsoft's' Remote Access Services (RAS), and into the 9/PC card

In all cases, the information you enter when creating the project is used to direct the download to the correct 9/PC for this project. This includes the host PC name and the topic name of the 9/PC card. This project information can be changed at any time. Refer to *Configuring an ODS Project for 9/PC*, located in this chapter.

Downloading an AMP File from the Host PC to a 9/PC

When you download locally to the 9/PC, no external connections are required. The download is done across the computer's PCI buss.

This procedure loads the selected AMP file into the memory of the 9/PC card and then forces the 9/PC to restart. This restart will either be done manually or automatically, depending upon the 9/PC configuration settings made during installation.

1. Selectd `[F3]` to pull down the Application menu.

This screen is displayed:

2. Select the **Download** application.

3. Press `[F4]` to pull down the Utility menu.

This screen is displayed:

4. Select the **Send AMP params** utility.

   The workstation displays the message:

   ```
   Please place the control in ESTOP to download
   ```

   When you press any key, the workstation displays the list of AMP files that you have created for this project.

5. Select or type in an AMP file name. The workstation displays the message:

   ```
   Waiting for response on the control
   ```

   This message is only displayed while ODS is waiting for the 9/PC to verify that it is in E-STOP. The display of this message is often too brief to even be seen under normal operation.

   The workstation then displays the message:

   ```
   Download in progress
   Percent transferred   xx %
   ```

   ODS displays the percentage of the file that has been transferred.

   When the download process is complete, ODS displays the message:

   ```
   DOWNLOAD COMPLETE
   ```

6. After downloading any new AMP file, the 9/PC must be restarted. This restart will be initiated in one of two ways, depending upon the 9/PC configuration settings that were made.

   - If the **"Autostart After AMP download"** selection was made in the 9/PC configurator, the 9/PC card will be restarted as soon as the download is complete.
   - If this was not the configuration setting made, then you will need to follow the simple procedure discussed in the *9/PC CNC Software and Hardware Installation and Integration Manual* to restart the 9/PC.

   In either case, the 9/PC operator interface or Basic Display Set (BDS) will display the message:

   ```
   CNC was stopped. Restart the CNC using the 9/PC
   configuration manager. Select "exit" or use the
   options menu to reconnect BDS to the CNC.
   ```

   This indicates that BDS must be "reconnected" to the 9/PC card as indicated in the message. Attempting to reconnect BDS without restarting the 9/PC will result in the error message:

   ```
   Connot Link to ABOCISERVER | CNC_1
   ```

   Once the 9/PC has been restarted, the new AMP parameter values are used for operation of the system.

Downloading Over a Network from a Second PC

ODS can download AMP files to the 9/PC over a communications network assuming that both the ODS workstation and the host PC are on the network. The Computer that is running ODS and initiating the download must be running the Windows NT operating system.

When an AMP file is downloaded to the 9/PC, the ODS workstation downloads the file directly to the 9/PC card. This means that the previous AMP file is no longer in use and, unless this older file was backedup, it no longer exists.

The difference between this network download and a local , or host download is in the value entered in the project information for the "computer Name". In the case of network download, the name of the host computer must be entered.

To determine the name of the host computer, perform these steps at the host computer: Under the Windows start menu, select: "settings"; then "control panel"; activate the Network control; then use the "Identification" tab.

If this information is correct, follow the steps in the previous section on downloading an AMP file.

Downloading from the Serial Port of a Second PC

ODS can download the AMP file through the serial port of the workstation to the serial port of the host PC.

Important: These computer configurations are required for correct operation of this capability:

- The computer running ODS and initiating the download must also be running Windows NT.
- The 9/PC host computer must have Remote Access Services (RAS) installed and configured. This allows the host computer to be a server to the ODS (or client) PC.
- Both PCs must have Dial-up Networking installed. This is required in a Windows NT system to be able to communicate remotely over Modem or serial cable.
- Both PCs must have an identical user "account" created to facilitate dial-up networking operation.

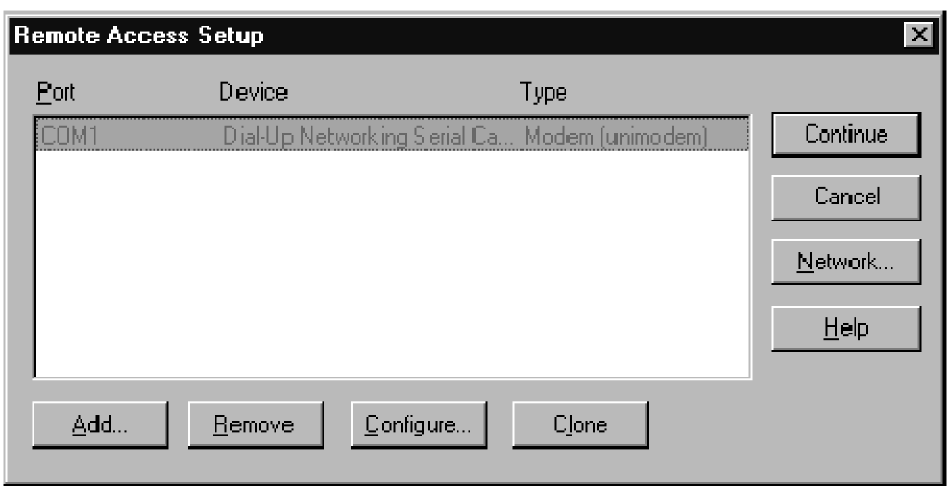
When an AMP file is downloaded to the 9/PC, the ODS workstation downloads the file directly to the 9/PC card. This means that the previous AMP file is no longer in use and if this older file was not backed-up, it no longer exists.

The difference between a download from a serial connection from a second PC and a local , or host download is in the value entered in the project information for the “computer Name”. In this case of serial download, the name of the host PC must be entered.

Setting up the computers needs to be done only one time. Here are the steps necessary to correctly install and configure the communications software on the two PCs. Note that an installation / setup must be done for both the ODS workstation and the Host PC.

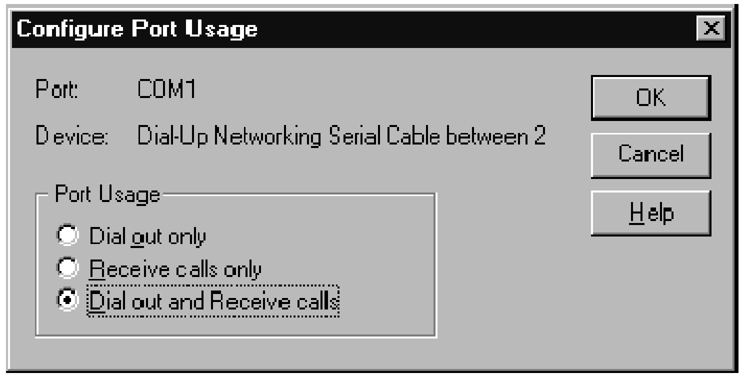
Setting up the 9/PC host (or Server) computer

1. Double-click **My Computer** on the desktop.
2. Double-click **Dial-Up Networking**.
3. Click **Install**
4. Enter **?:\I386** in the “Copy Files From“ text box where “?” is the drive that contains your I386 folder. Probably your CD ROM drive containing the Windows NT CD.
   - If you get the message “no RAS capable devices available” click **YES** next to modem configuration
   - Select “don’t detect, select from list”. When the list box appears, select “standard modem” and “**Dial-Up Serial Cable between 2 PCs”**
5. Click **OK** to add COM1 - Dial-Up Networking Serial Cable between 2 PCs. The Remote Access Setup window appears.



6. Click **Configure** in the Remote Access Setup window

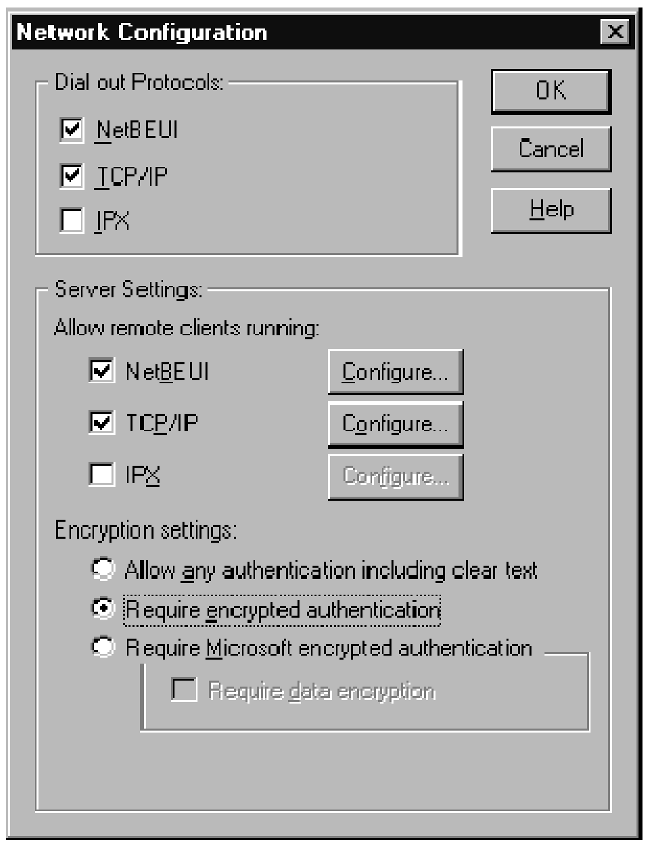
7. Select Dial out and Receive Calls, to allow the Host PC to be both the RAS Client and Server.



8. Click **OK**

9. On the remote Access Setup click **Network. . .**

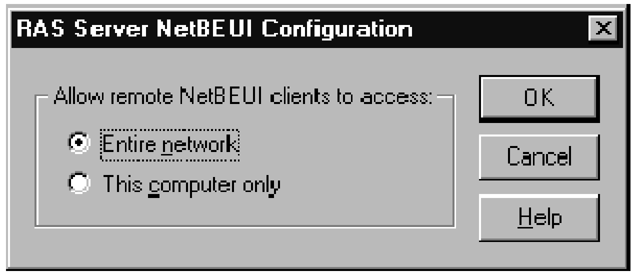
10. Select the following protocols: **TCP/IP** and **NetBEUI**



11. You must select the Encryption setting **Require encrypted authentication**.

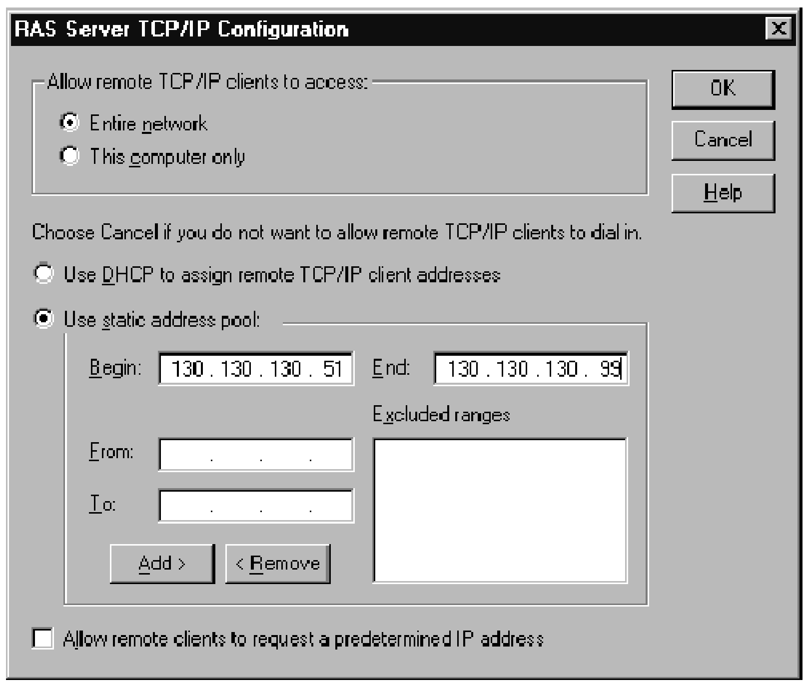
12. Click the **Configure** button next to NetBEUI

13. Select **Entire Network**, and then click **OK.**



You can use TCP/IP, but you must either assign a pool of static IP addresses to be available for the clients or if you have a DHCP server, allow it to assign client IP addresses.

14. Click on the **Configure** button next to TCP/IP to get RAS Server TCP/IP configuration dialog.



15. Check the **Entire Network** button. If you are setting up the ODS (client) computer, go to step 2 of the next section. If you are setting up the 9/PC host (server) computer, go to the next step.

16. Check the **Use static address pool** button, if you are assigning static addresses. Click the **Use DHCP** button and skip the next step if you are using DHCP.

17. Enter your starting IP address in the **Begin** box, and your end IP address in the **End** box.

18. Click **OK**, then click **Continue** to install RAS.

19. Click Close, then reboot the computer when prompted.

20. After logging into Windows NT, select: **Start** > **Programs** > **Administrative tools** > **Remote Access Admin.**

   - These Next Steps assume that you have set up accounts on the subject PC for those users (or administrators) who will be "dialing in" to the server PC.

21. Select **Users** > **Permissions**.

22. Click on the name of the user (or administrator) who you will allow to dial-in to the server. Check the **Grant dial-in permission to user** checkbox.

23. Under Call Back, select **No Call Back**, then click **OK**

24. Select: Server > Start remote access server. In the Server field, the name of the local computer should already be present. If it is not present, enter the computer name preceded by two back slashes. Example: \\YourComputerName. Click **OK** to start the service.

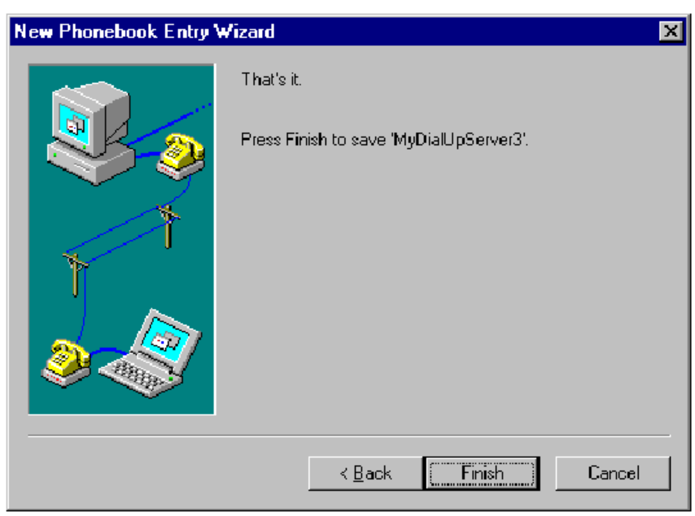
Setting up the ODS (or Client) computer

1. Repeat steps 1 through 16 from the previous section.

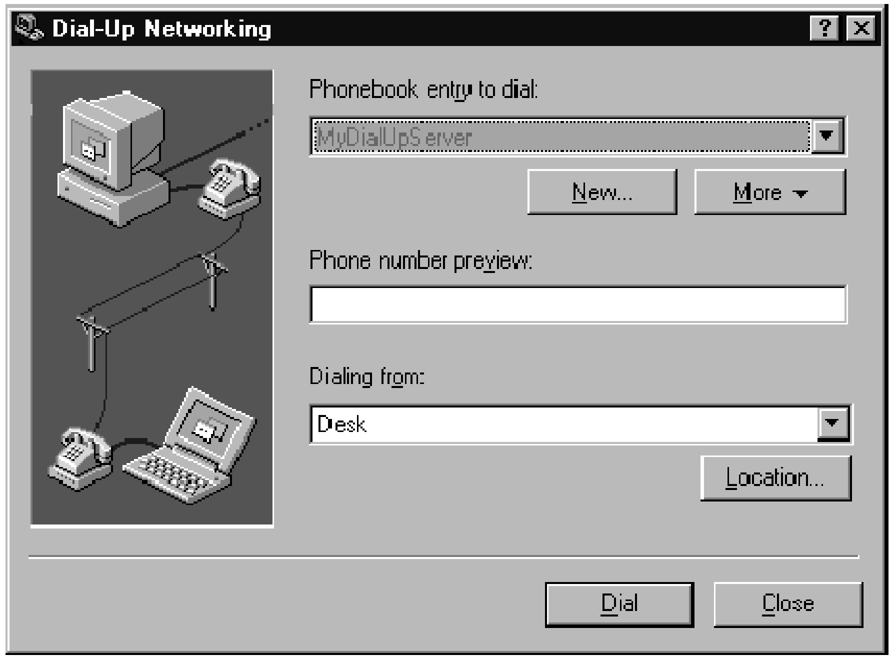
2. After logging in again, double click the **My Computer** icon.

3. Double click **Dial-up Networking**.

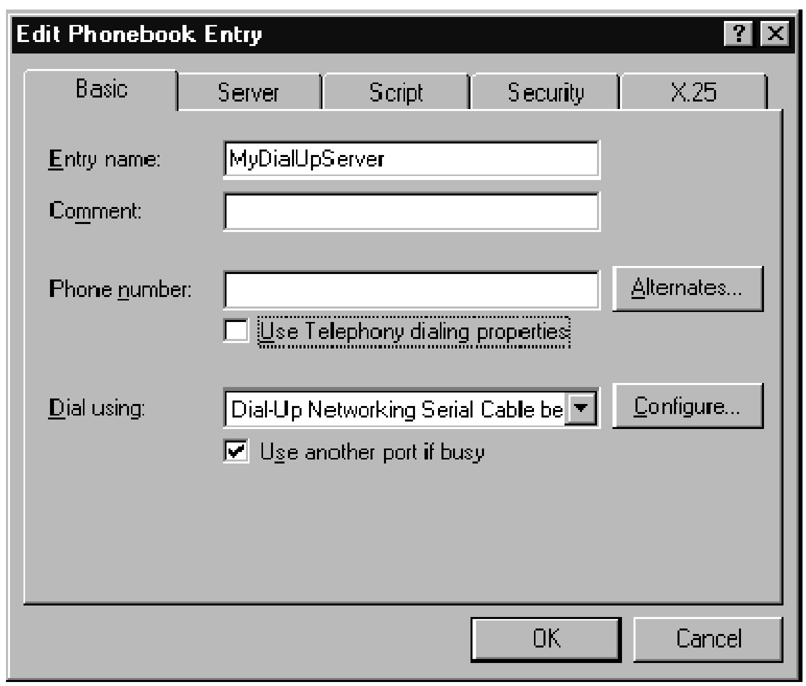
   - If you have not configured any dial-up Phonebook entries before, you will get the new entry Wizard.
   - Click **next**, and select **Plain Text Password**
   - Hit enter (or **Next**) through the rest of the selections until the following screen appears:



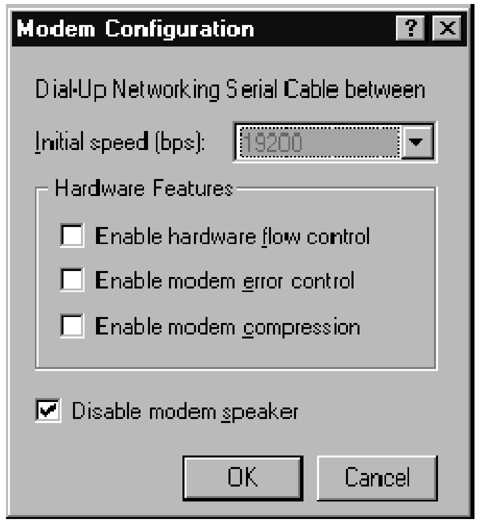
4. Click **More** and “Select Edit Entry and Modem Properties”.



5. Click **Configure** next to “Dial Using”.



The following screen appears:



6. Select a baud rate that is available on this and the server PC.
7. Click **OK** on this window and the "edit phone book" window.
8. Disconnect your EtherNet cable.
9. Start the RAS server on the 9/PC Host (server PC) if you did not already do so in the last section. Do this by selecting **Start > Settings > Control Panel**. Double click the **Services** control panel to view the available services. Locate and highlight the **"Remote Access Server"** , and click **Start** .
10. Click Dial to connect to the 9/PC host (server PC) from your ODS workstation (client PC).

Troubleshooting tips for RAS configuration

- If you receive the message "Error 651" when establishing a connection, the first items to check are:
  - cable pin-outs are correct and have continuity
  - Cable connections are good to the PC serial ports
  - The port you have connected the cable to actually corresponds to the port name you have configured (ex: COM1 vs COM2).
  - Baud rates and other port settings for the configured ports must be the same on the client and Server PCs
  - Baud rates up to 115,200 have been successfully used on short cable lengths.

Uploading AMP Files

The currently active AMP file in the 9/PC can be edited. These files must be uploaded to the ODS workstation before they can be edited. Use the upload application of ODS to upload AMP files in these ways:

- from the 9/PC card to the host PC.
- from the 9/PC card over a network to a second PC
- from the 9/PC card through a serial port to a second PC

Uploading an AMP File from a 9/PC to the Host PC

To upload an AMP file to the host PC from the 9/PC:

1. At the main ODS menu, Press `[F3]` to pull down the Application menu.

   This pull down menu is displayed:

2. Select the **Upload** application.
3. Press `[F4]` to pull down the Utility menu.

   This pull down menu is displayed:

4. Select the **Get AMP params** utility.

   The dialog box is displayed to enter the name of the AMP file that will be uploaded:

5. Type in a new file name for the file that will be uploaded, then press `[ENTER]`.

Important: You must assign a new name to the AMP file that is uploaded.

This message is displayed:

```
Upload in progress
Percent transferred xx  %
```

6. ODS starts uploading a copy of the active AMP file to the Host PC. ODS continuously updates and displays the percentage of the file that has been transferred.

   This message is then displayed:

   `Recovering control image`

ODS recovers the AMP control image and creates an AMP file that you can edit.

When the upload process is complete, ODS displays the message:

`Upload complete`

Press any key to return to the main menu line of ODS.

If you changed any parameter values while using Patch AMP or on-line AMP and these values are out of range, ODS will not be able to recover the control image of the AMP file unless the parameters are corrected.

These selections are displayed:

Continue Checking	(C)
Edit Parameter	(E)
Quit Checking	(Q

Correct the out of range error(s) so that the recovered control image can be created. If you do not correct the errors, the upload process aborts.

If you select the:	ODS:	and:
Continue Checking option	checks additional parameters for out of range errors	• if there are any others errors,ODS displays the previous screen in response to each additional error • if there are no other errors, ODS displays the previous screen in response to the first error
Edit Parameter option	displays a screen listing the legal range options for the current parameter	Select one of the legal range options to eliminate the out of range error, which enables ODS to continue recovering the control image.
Quit Checking option	aborts the recover control image process	• the upload process aborts • ODS returns to the main menu

Uploading an AMP File from a 9/PC Over a Network

ODS can upload the AMP file over a communications network to the 9/PC. This assumes that both the ODS PC and the host PC are on the network. The Computer that is running ODS and initiating the upload must be running the Windows NT operating system.

The difference between this network upload and a local , or host upload is in the value entered in the project information for the "computer Name". In the case of network upload, the name of the host PC must be entered.

To determine the name of the host PC, you must be at that computer, and perform these steps: Under the Windows **Start** menu, select: "**Settings**"; then "**Control panel**"; activate the **Network** control; and select the "**Identification**" Tab

If this project information is correct, follow the steps in the previous section on downloading an AMP file.

Uploading an AMP File from the 9/PC Over a Serial Connection

ODS can upload the AMP file through the serial port of the workstation from the serial port of the host PC. Both PCs must have Dial-up Networking installed and configured.

- The 9/PC host computer must also have Remote Access Services (RAS) setup and running.
- The Computer that is running ODS and initiating the download must be running the Windows NT operating system.

Installation and configuration of Remote access Services (RAS) is discussed in detail in the Serial download section of this chapter.

Documenting AMP Files

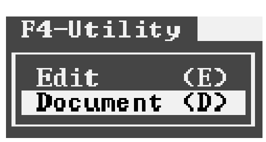
After you create a file by using the AMP editor, it can be documented using the Document utility. This file can be printed or displayed on the workstation.

Creating a Document File

To document an AMP file, use this procedure:

1. Press [**F3**] to pull down the Application menu.
2. Select AMP, then press [**ENTER**].
3. Press [**F4**] to pull down the Utility menu.

   The workstation displays this selection box:



4. Select the Document utility.

The workstation displays a screen showing that it is loading the selected utility, then displays this selection box:

```
A M P   D O C U M E N T O R
-----------------------------
Document File            <D>
Quit                     <Q>
```

5. Select the Document File option.

The workstation displays the directory of AMP files for this project:

6. Select the file you want to document.

The workstation displays this selection box:

```
A M P   D O C U M E N T O R
-----------------------------
Document parameters by :

Axis/Non-Axis Sort        <a>
Parameter Number Sort     <b>
Group Parameter Sort      <c>
Cancel                  <ESC>
```

7. Select (a), (b), (c), or press <ESC> to cancel.

a. The Axis/Non-Axis Sort option creates a file of parameter values sorted by axis parameter and non-axis parameters.

b. The Parameter Number Sort option creates a file of parameter values sorted by their identification numbers.

c. The Group Parameter Sort option creates a file of parameter values sorted by their groups.

The workstation creates the specified document file. When it is finished, the workstation displays these messages:

`DOCUMENTING COMPLETE`

`Press any key to continue.`

Press any key to go to the “Document” or “Quit” selection box:

8. Select another file to document or quit the AMP documenter.

Printing or Displaying an AMP Document File

To print or display a document file that was created with the document utility, use this procedure:

1. Press [F3] to pull down the Application menu.
2. Select the AMP application.
3. Press [F1] to pull down the File menu.

4. Select the Print or Type option.
5. The workstation displays a directory of AMP documents. Select the file to be printed or typed.

If, in step 4, you selected:	the workstation:	and you can select:
Type option	displays the first part of the selected file on the workstation	• to view the next page of the file, press [PgDn] • to view the previous page of the file, press [PgUp]
Print option	outputs the selected file to the printer interfaced with the workstation	

END OF CHAPTER

Configuring Axes

Chapter Overview

This chapter covers the procedures that must be performed to configure the axes of the control. Before setting the axis parameter values, an axis name and axis type must be specified for each axis.

This chapter contains this information:

Topic:	Page:
Selecting an Axis	3-1
Specifying Axis Names	3-3
Specifying Axis Types	3-4
Deleting an Axis	3-6
Copying Axis Parameters to Another Axis	3-7
Configuring a Servo	3-7
Selecting Inch or Metric Units	3-8

Important: Note that:

- you can select any of the options shown above whenever the AMP editor utility is active and an AMP file is open
- additional general axes configuration parameters are discussed in chapter 4 "Axis Parameters"

Important: The order in which you configure axes has a significant impact on several features including:

- Dual Axes – order they are AMPed determines which axis is your master axis

Make sure that you read the section on configuring this feature before continuing with this chapter.

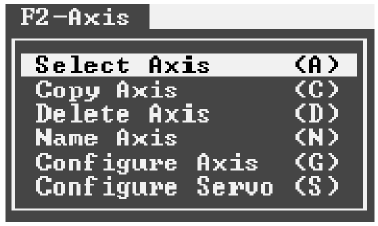
Selecting an Axis

The axes of the control must be configured in AMP through their corresponding AMP parameters. Some AMP parameters are global parameters. The value assigned to these parameters apply to all axes of the control. Other AMP parameters must be set independently for each axis. The Select Axis option of the Axis menu is used to select an axis for AMP configuration. The parameters listed for each axis will vary depending on the selected axis type.

Use this procedure to select an axis:

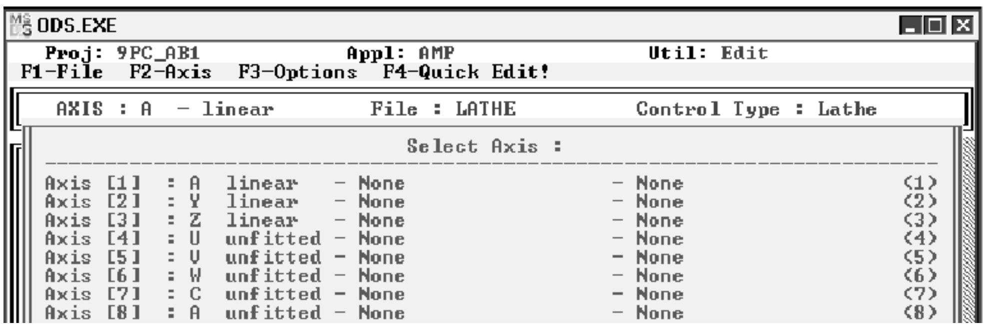
1. Press [**F2**] to pull down the Axis menu.

   The workstation displays this screen:



2. Select the Select Axis option (A).

   The workstation displays this screen:



   The screen displays each axis, and assigns it a number that you can use to select it.

Important: The 9/PC supports the following Maximum number of axes, Servo motors, and spindles. NOTE that at the initial release these quantities are limited to the number of axes, motors, spindles , and total motors supported by a **single 1394 serial drive**:

- up to 8 servo axes
- up to 8 servo motors
- up to 2 open or closed loop spindles
- **a maximum total of 8 spindle and servo motors**

The maximum controllable axis on a particular 9/PC is dependant upon the options selected when the system was ordered. If you use AMP to configure more axes than your system can support, the control will be held in E-stop after the AMP file is downloaded.

3. Select the axis to be configured.

   The workstation displays the selected axis on the third status line of the AMP screen.

Specifying Axis Names

Use the following procedure to specify the names of the axes on the machine. The AMP editor utility must be active and an AMP file must be open. Repeat the procedure for each axis on the machine.

Important: Note that axis names may also be altered using the parameter group called “Axis Parameters” found in chapter 4.

1. Press [F2] to pull down the Axis menu.
2. Select the Name Axis option (N).

   The workstation displays the select axis screen

Important: The axes must be named in a consecutive series; fitted axes cannot follow an unfitted axis. Any fitted axes that follow an unfitted axis are ignored by the control. Any configured spindle must be configured after all fitted axes in the consecutive series of named axes.

3. Select the axis for which a name is to be specified. The axis name selection box appears to allow you to enter the name of the current axis:

```
 Axis One's Name : 250
-------------------------
 A                  (a)
 B                  (b)
 C                  (c)
 U                  (d)
 V                  (e)
 W                  (f)
 X                  (g)
 Y                  (h)
 Z                  (i)
 S                  (j)
 $B                 (k)
 $C                 (l)
 $X                 (m)
 $Y                 (n)
 $Z                 (o)
```

4. Select the new name for the axis.

Typical linear axis names for mills are:

```
X, Y, Z
```

Typical rotary axis names are:

```
A, B, C
```

where the axis of rotation for A, B, and C is X, Y, and Z respectively.

Typical parallel linear axis names for mills are:

```
U, V, W
```

where U, V, and W are parallel to X, Y, and Z respectively.

Typical linear axis names for lathes are: X and Z

The typical rotary axis name for a lathe is: C

where the axis of rotation for C is Z.

Typical parallel linear axis names for lathes are:

```
U, W
```

where U and W are parallel to X and Z respectively.

Important: The axis name S is reserved as the name for spindle axes. Do not use S for linear or rotary axes.

Specifying Axis Types

Use this procedure to specify the axis types of the axes on the machine. Repeat the procedure for each axis on the machine.

Specifying Spindles

Follow these guidelines if you are using one or more spindles in your system:

- Spindles must be configured after you have configured all rotary and linear axes. For example, if you have an application with 3 linear axes, one rotary axis, and one spindle, your axis sequence appears in ODS as:

```
                      Select Axis to Configure :
--------------------------------------------------------------------------
Axis [1]  : A  linear   - SERCOS Interface     - Digital or Digital S  <1>
Axis [2]  : Y  linear   - SERCOS Interface     - Digital or Digital S  <2>
Axis [3]  : Z  linear   - SERCOS Interface     - Digital or Digital S  <3>
Axis [4]  : B  rotary   - SERCOS Interface     - Digital or Digital S  <4>
Axis [5]  : S  spindle  - SERCOS Interface     - Position or Analog S  <5>
Axis [6]  : W  unfitted - None                 - None                  <6>
Axis [7]  : C  unfitted - None                 - None                  <7>
Axis [8]  : A  unfitted - None                 - None                  <8>
```

Axes must be specified in a consecutive series. No fitted (linear or rotary) axes can follow unfitted axes. All spindle axes must follow fitted axes. Any axes that follow unfitted axes are ignored by the control.

Important: When you configure the axes, the axes to be used for the DAC monitor output, or analog output (typically spindles) must be configured last.

Configuring Spindles

If you are using one or more spindles, refer to chapter 7 for information on setting the parameter **Spindle Type for Axis** .

You must configure the spindles in this order:

- the first spindle axis is spindle 1
- the spindle axis that follows the spindle 1 must be configured as spindle 2

Configuring Axes

1. Press **[F2]** to pull down the Axis menu.

2. Select the Configure Axis option (G).

The workstation displays the axes with their currently assigned names.

3. Select the axis to configure.

The workstation displays this selection box:

```
Axis Five's Type : 364
---------------------------
linear              <a>
rotary              <b>
spindle             <c>
unfitted            <d>
```

Possible axis types are:

- **linear** – specifies an axis that moves along a straight line
- **rotary** – specifies an axis that moves in a circle about a fixed center
- **spindle** – specifies a spindle that provides position and velocity feedback

Important: When configuring axes, any motors using a DAC output (typically the spindles) must be configured as the last axes.

- **unfitted** – specifies an axis that is not used in the current configuration of the machine. An unfitted axis can not be moved.

4. Select the axis type.

The workstation assigns the selected type to the selected axis, then returns to the main menu.

The following table lists the available axes with their corresponding AMP identification numbers and default type values.

Axis Number	Parameter Number	Axis Type
1	[360]	linear
2	[361]	linear
3	[362]	linear
4	[363]	unfitted
5	[364]	unfitted
6	[365]	unfitted
7	[366]	unfitted
8	[367]	unfitted

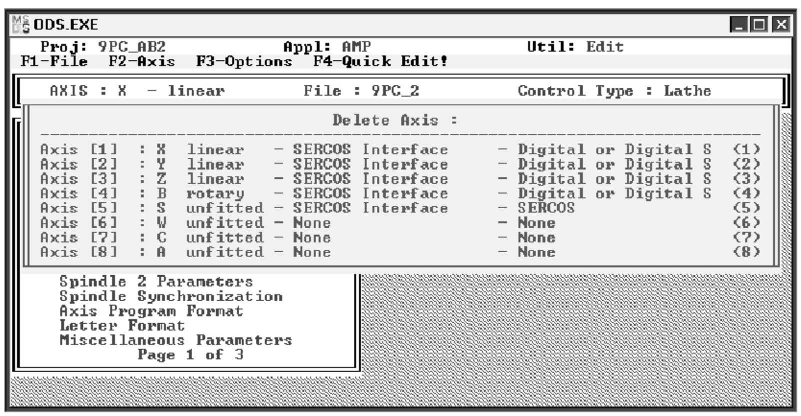
Deleting an Axis

If an unwanted axis is displayed in the axis configuration, it can be deleted with this procedure:

To delete an axis from the open AMP file:

1. Press [F2] to pull down the Axis menu.
2. Select the **Delete** option.

   The workstation displays this screen:



3. Select the axis that you want to delete.

The workstation reclassifies the selected axis as “unfitted” and the parameters change to their default values.

Important: The axes must be named in a consecutive series, no fitted axes may follow an unfitted axis. All spindle axes must follow fitted axes. Any fitted axes that follows an unfitted axis is ignored by the control.

Copying an Axis

The parameter values of an axis can be copied from one axis to another. This is a convenient way to set the parameter values for an axis.

Important: When axis parameters are copied from one axis to the next, all "per axis" parameters are copied from all parameter groups.

To copy an axis to another axis in the open AMP file:

1. Press [F2] to pull down the axis menu.
2. Select the Copy operation.

   ODS displays the axis select screen.
3. Select the axis from which to copy. The workstation displays :

```
linear          File : 9PC_2          Control
         Copy from Axis  1 to Axis ...
```

4. Select the axis that will receive the copy of the axis parameters.

   The workstation copies the parameter values from the selected source axis to the selected destination axis.

Important: The two axes must be of the same type. For example, a linear axis can only be copied to another linear axis.

Configuring a Servo

For each axis, the type of servo needs to be entered. To do this:

1. Configure the axis type as described on page 3-5.
2. Configure the servo by setting the:
   - Servo Hardware Type (control hardware)
   - Servo Loop Type (drive hardware and servo control type of the drive)

   The full servo parameter selections will only become available when both of these parameters are defined as a selection other than "None."

   a. To set the servo hardware type, select "Configure Servo" under the F2-Axis menu. You are prompted to select a servo type. For the 9/PC select SERCOS as the servo hardware type.

Select this Servo Type:	If you have this Servo Hardware:
None	None
SERCOS Interface	A 9/PC CNC

Once you have selected the servo hardware, you are prompted to select the type of servo loop or servo drive hardware that will be connected to the 9/PC.

b. Select a Servo Loop Type.

Servo Loop Type:	Results in:
None	This is the default setting that the control uses when no loop type has been selected. The control stays in E-stop if an axis type has been defined, and no loop type has been defined.
Position or Analog Spindle	Use this selection if the analog output of the 1394 based CNC serial drive is being used (8520-SER). This selection is also used for spindles (excluding digital spindles) and depth probes.
Position/Velocity	The drive will close both the position loop and the velocity loop. Separate devices or the same feedback device can be used to close both the position and velocity loops.
Digital or digital spindle	The drive will close both the position and the velocity loop. Additionally, motor information necessary for proper commutation is also provided for digital drive systems. This must be selected for all 1394 digital drive systems (including 1394 spindles). All servo specific parameters will be set via AMP.
SERCOS	This selection indicates that a SERCOS drive other than the 1394 is being used for this servo. This selection indicates that all servo specific parameters must be set directly at the drive.

Once you have selected the servo hardware type and servo loop type, additional AMP parameters become available to you for configuration in the Servo Parameters Group. The specific parameters available for you to set depends upon the selections you made for Servo hardware type, and Servo Loop Type.

Selecting Units

The control can be configured for display and input in several different types of units. To select units of the open AMP file:

1. Press [F3] to pull down the **Options** menu.

   The workstation displays this screen:

The checked (√) units are the current settings.

2. Select the display and input units.

The workstation returns to the main menu.

Selecting Control Type

Your control type is based on the application type that you selected when you created the project.

Application type	these control types are available:	the default value is:
Mill/Lathe	lathe (A, B, or C)	lathe
	mill (standard or transfer line)	

The control type parameter number is [350].

To select the control type of the open AMP file:

1. Press [F3] to pull down the **Options** menu.

2. Select **Control Type** option (j).

The workstation displays this screen:

```
Control Type : 350
---------------------
Lathe           (a)
Mill            (b)
```

3. Select either Lathe or Mill for Lathe/Mill applications. You may also want to change your lathe type (lathe A, B, or C) or your mill type (standard mill or transfer line mill). Refer to the chapter on **Miscellaneous** parameters.

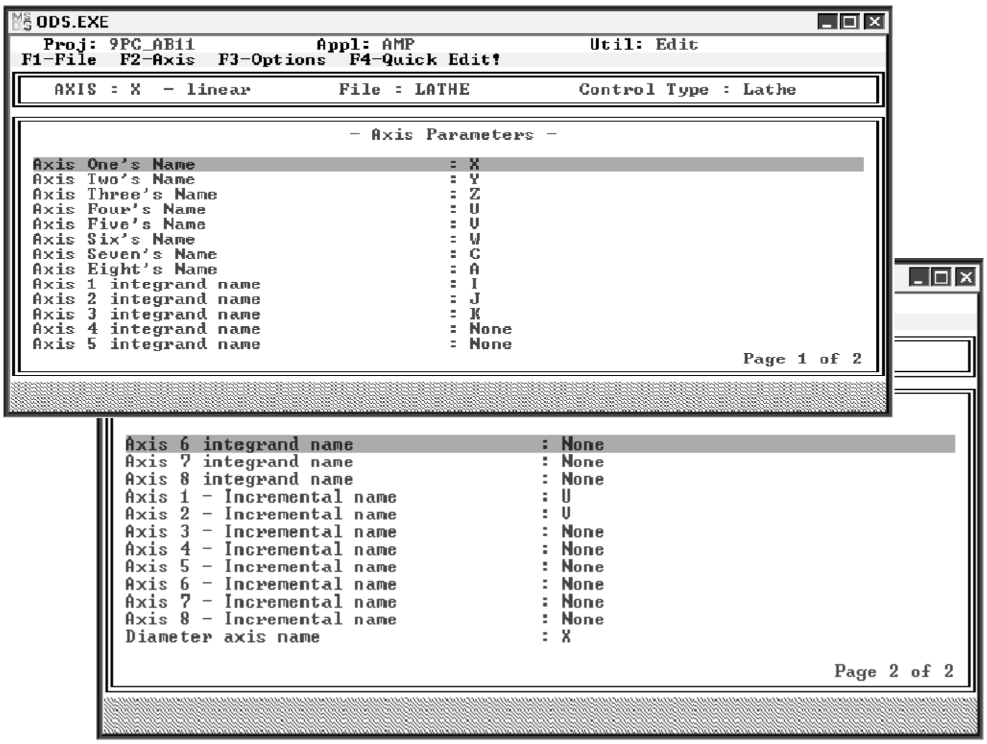
The workstation records the selected control type and returns to the main menu.

END OF CHAPTER

Axis Parameters

Chapter Overview

This chapter covers the parameters used to specify the various axis related parameters. These parameters are accessed by selecting the Axis Parameters group displayed on the main AMP menu screen. When you select the “Axis Parameters” group, these screens or pages are available:

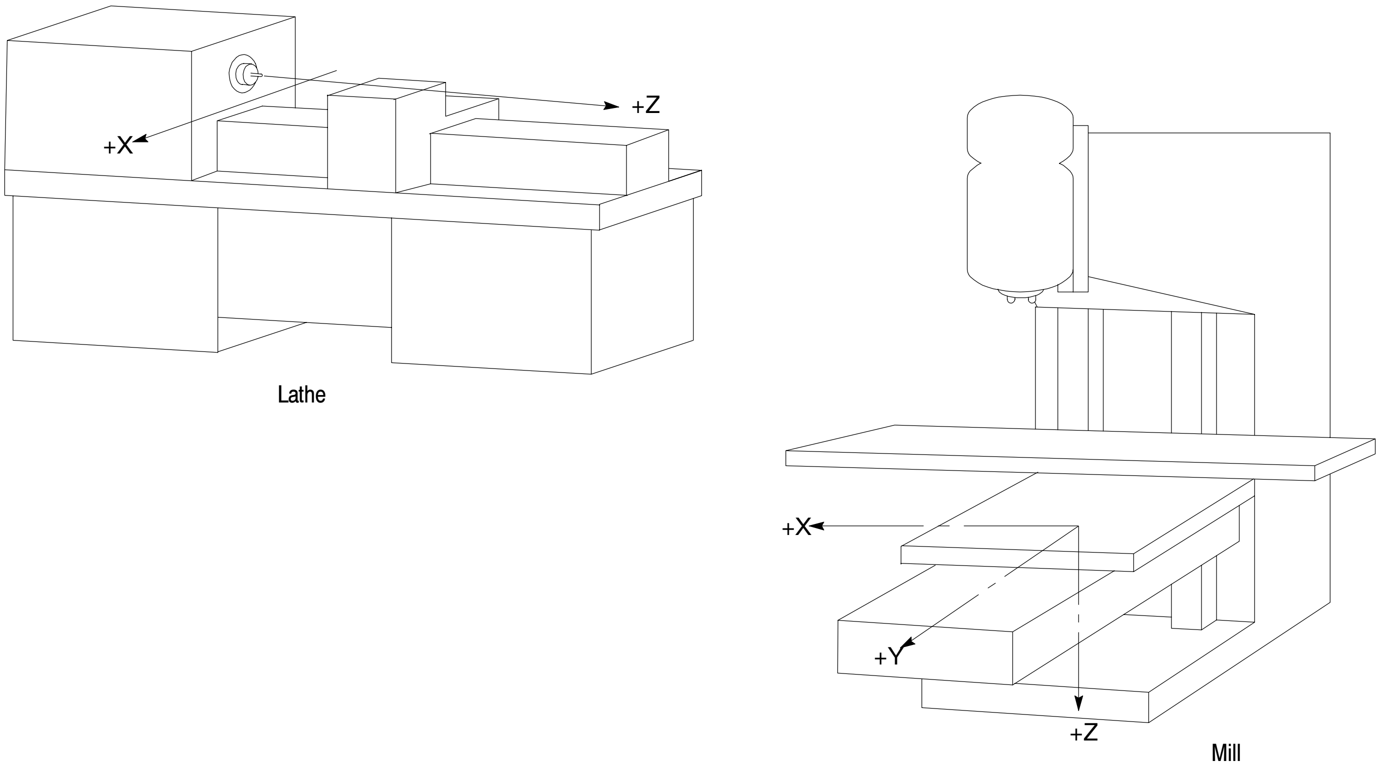


Your screens may differ slightly, depending on your application type. This table shows you where each of the axis parameters is found:

Axis Parameter:	Page:
Axis Name	4-2
Integrand Name	4-3
Incremental Name	4-5
Diameter Axis Name	4-5
Rollover value	4-6

All axes of the control must be assigned an axis name. Typical axes names of a lathe and mill are shown in Figure 4.1.

Figure 4.1
Example of a Lathe, and Mill axis names and orientation



Axis Name

Function

These parameters specify the axis name for each axis of the control. Each axis of a selected control type must be assigned an axis name. These names are used in part programs to reference a specific axis.

Important: The Name Axis option, accessed by pressing the **[F2]** key, can also be used to specify the axis name for a selected axis. Any time the Name Axis option is used, it also changes the value of the parameter that corresponds to the selected axis.

Parameter	Parameter Number
Axis One's Name	[250]
Axis Two's Name	[251]
Axis Three's Name	[252]
Axis Four's Name	[253]
Axis Five's Name	[254]
Axis Six's Name	[255]
Axis Seven's Name	[256]
Axis Eight's Name	[257]

Range

Selection	Result	Selection	Result	Selection	Result
(a)	A	(f)	W	(k)	$B
(b)	B	(g)	X	(l)	$C
(c)	C	(h)	Y	(m)	$X
(d)	U	(i)	Z	(n)	$Y
(e)	V	(j)	S	(o)	$Z

Notes

Each of these parameters must be set independently for its corresponding axis.

We recommend using axis names preceded by a $, only if you must use the same letter for an auxillery / parallel axis to one of the primary axes.

Axis Integrand Name

Function

These parameters specify the axis integrand name for each axis of the control. The numeric value programmed with the axis integrand name is the integrand value for that axis. This value is used by the control to reference a point on the axis used in calculating arc centers, fixed cycle variables, and other similar functions.

Parameter	Parameter Number
Axis 1 Integrand name	[185]
Axis 2 Integrand name	[186]
Axis 3 Integrand name	[187]
Axis 4 Integrand name	[188]
Axis 5 Integrand name	[189]
Axis 6 Integrand name	[190]
Axis 7 Integrand name	[191]
Axis 8 Integrand name	[192]

Range

Selection	Result
(a)	I
(b)	J
(c)	K
(d)	None

Notes

Each of these parameters must be set independently for its corresponding axis.

The axis integrand name is a secondary programming word used to program additional axis data when necessary. For example, the axis integrand name can be used to specify:

- arc center position
- thread lead

Typically, axis integrand names are:

- X axis: I
- Y axis: J
- Z axis: K

Assign integrand names for a dual axis only for the master axis and other axes you intend to decouple from the dual group. When a dual group is decoupled, each slave can use integrand planer functions (such as circular interpolation) only after the dual group is decoupled and the slave axis name has been AMPed as a primary or parallel axis name in the active plane definition. The master axis in the dual group uses the dual groups integrand letter when decoupled.

Important: Axis integrand for parallel axes must be defined with the same integrand word if they are to be used interchangeably for planer functions (such as G02 or G03). For example if X and U are parallel and both are used to define the G17 plane along with some other perpendicular axis, both parallel axes must be defined here with the same integrand word. Parallel axes that are not used in planer functions do not need to use the same integrand.

Important: Non-Planar axes that are defined with the same integrand letter as an axis used in one of the planes must be configured in AMP after the planar axis. For example if G17 is defined as the XY plane and the X integrand is I, an axis, $X for example, that uses the I integrand letter and is not a planar axis must be configured in AMP as an axis following the X axis.

Axis Incremental Name

Function

Important: The axis incremental name parameter is used only for lathes using G-code type A. It is not used for any other lathe types or mill applications.

These parameters specify the incremental name for each axis of the control. The numeric value programmed with these axis words generates an incremental move along the programmed axis.

Parameter	Parameter Number
Axis 1 - Incremental name	[170]
Axis 2 - Incremental name	[171]
Axis 3 - Incremental name	[172]
Axis 4 - Incremental name	[173]
Axis 5 - Incremental name	[174]
Axis 6 - Incremental name	[175]
Axis 7 - Incremental name	[176]
Axis 8 - Incremental name	[177]

Range

Selection	Result
(a)	U
(b)	V
(c)	W
(d)	None

Notes

Set each parameter independently for its corresponding axis.

Diameter Axis Name

Function

This parameter is used only in lathe applications. Use this parameter to specify the diameter axis name. This is the axis that is perpendicular to the spindle. Programming G07 or G08 changes programming between radius and diameter mode for the diameter axis.

Axis	Parameter Number
	Single Process
All	[442]

Range

Selection	Result	Selection	Result
(a)	A	(h)	Y
(b)	B	(i)	Z
(c)	C	(j)	$B
(d)	U	(k)	$C
(e)	V	(l)	$X
(f)	W	(m)	$Y
(g)	X	(n)	$Z

Notes

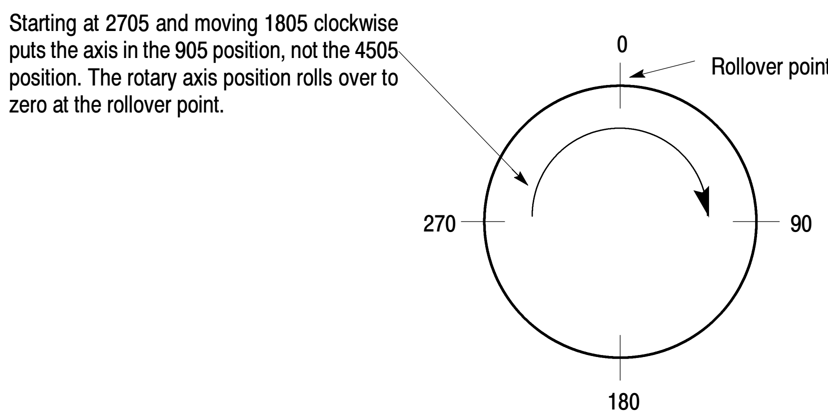
This parameter is a global parameter. The value applies to all axes.

Rollover Value

Function

This parameter is available only when configuring a rotary axis (selected with the F2 option). Typically, for rotary tables, this value represents full rotation of the axis (normally 360 degrees).

Figure 4.2
Rotary Axis Rollover Value (configured to rollover at 360 degrees)



Axis	Parameter Number
1	[1207]
2	[2207]
3	[3207]
4	[4207]
5	[5207]
6	[6207]
7	[7207]
8	[8207]

Range

0 to 9999 degrees

Notes

Assigning a rollover point of 0 degrees disables the rollover feature. When rollover is disabled the rotary axis is no longer sign dependant in absolute mode (programming -90 degrees and +90 degrees will position to the same location since there is no rollover from which to calculate negative angles).

Each of these parameters must be set independently for each rotary axis.

END OF CHAPTER

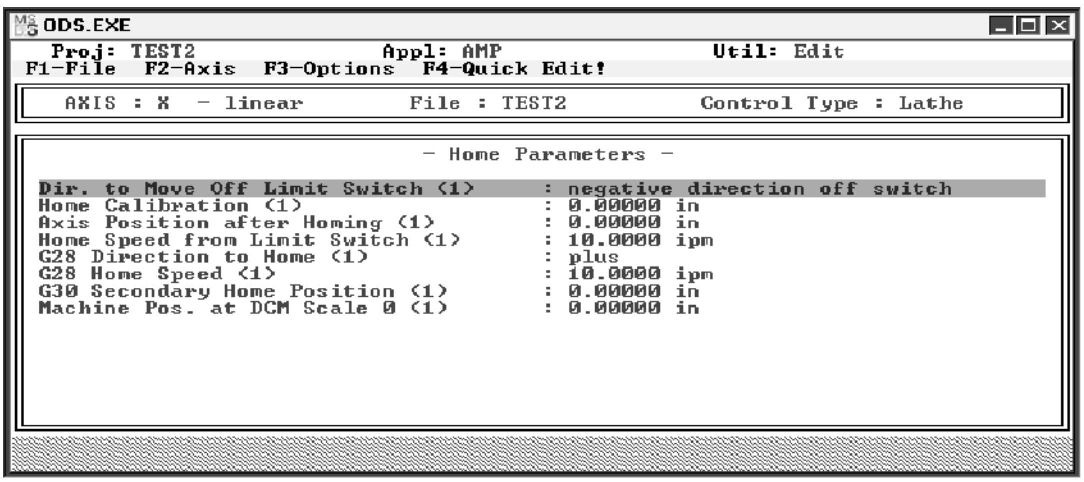
Home Parameters

Chapter Overview

Homing refers to the process of orienting a linear or rotary axis to a specific, repeatable, mechanical position. This chapter covers the AMP parameters that the control uses to home the axes.

If you have configured your axis as open loop, servo off, or servo detached, you do not need to set the parameters in this chapter. Refer to Chapters 7 and 8 for more information on Digital or Analog Servo Parameters.

Access these parameters by selecting the “Home Parameters” group on the AMP main menu screen.



An axis can be homed manually or automatically (G28) as described in your *9/PC CNC Operation and Programming Manual*. The basic homing process for three different types of position feedback is described in the next section of this chapter:

Type of parameters:	Page:
Homing sequence parameters	5-9
Automatic parameters	5-13
Distance coded marker	5-18

Homing Concepts

This section covers the sequencing, operation and strategy used to home, or reference, axes. The considerations for homing depend upon the type of feedback used on the axis. There are three general categories:

- incremental feedback; motor mounted, or axis mounted; that requires a full homing sequence.
- Distance coded marker; absolute feedback that requires motion
- 1326 motor absolute feedback: position set without motion

Manual Homing Using a Home Switch

The following outlines how a typical homing operation functions for axes with incremental feedback. Systems with absolute multi-turn encoders (available on specific versions of 1326 servo motors) that are compatible with the 9/PC and the CNC serial drives do not require a homing sequence after initial integration.

1. Set the Mode Select switch to "Manual," and the Jog Select switch to "Home" (assuming the Standard MTB Panel is being used).

2. Press the appropriate Axis/Direction button to move the axis in the direction of the home limit switch. Speed for this move can be altered manually through use of the jog speed and feedrate override switches.

Once the homing sequence is initiated, the jog speed switch has no effect on the speed of the axis. However, the feedrate override switch still effects homing speed.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

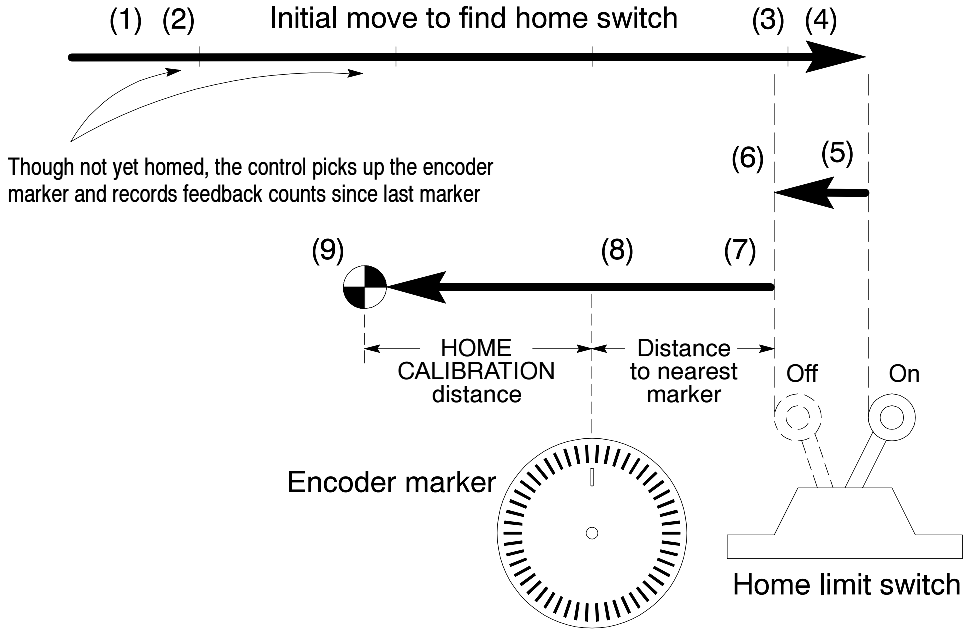
ATTENTION: Some mechanical switches will turn "OFF" at a different location when turned "ON" from different directions. In those cases, it is essential that the home limit switch be approached from the same direction every time the axis is homed. This can be assured by proper switch positioning, logic programming, AMP settings and/or operator instruction.

Important: If the speed selected here is too great, the axis may move "through" the home switch before coming to a stop. If the switch changes from "OFF" to "ON" and then back to "OFF" before the axis can stop, and the **Dir to Move off Limit Switch** is the same as the direction moved to the switch, the control will generate a homing error. This error is "JOGGED HOME TOO FAST:" and includes the name of the violating axis.

3. The axis moves until it trips the home limit switch.
4. The axis is commanded to stop and the control waits until the following error is within the range defined by the AMP **Inposition Band** parameter.
5. The axis then proceeds in the direction entered for the parameter **Dir. to Move off Limit Switch** at the speed entered for the parameter **Home Speed from Limit Switch**. All feedrate overrides are disabled throughout the rest of the homing sequence.
6. The axis continues until it comes off the home limit switch, at which time it is again commanded to stop.
7. The control waits for the axis to stop, until the following error within the range defined by the AMP **Inposition Band** parameter.
8. The control calculates the distance to the home position by summing the distance to the nearest encoder marker plus the distance entered for the **Home Calibration** parameter.
9. The axis moves at **Home Speed from Limit Switch** to the calculated home position (this can be in either direction, depending on the result of the previous calculations).
10. The control defines this position as the home position for the axis, and assigns it the coordinate value entered for the parameter **Axis Position after Homing**.

A typical homing sequence is illustrated in Figure 5.1. This figure assumes that the value entered for the AMP parameter **Dir. to Move Off Limit Switch** is opposite the direction moved to the switch (opposite of **G28 Direction to Home** for automatic homing). It also assumes that the sum of the distance to the nearest marker and the **Home Calibration** distance, result in a move in the same direction as **Dir. to Move Off Limit Switch**.

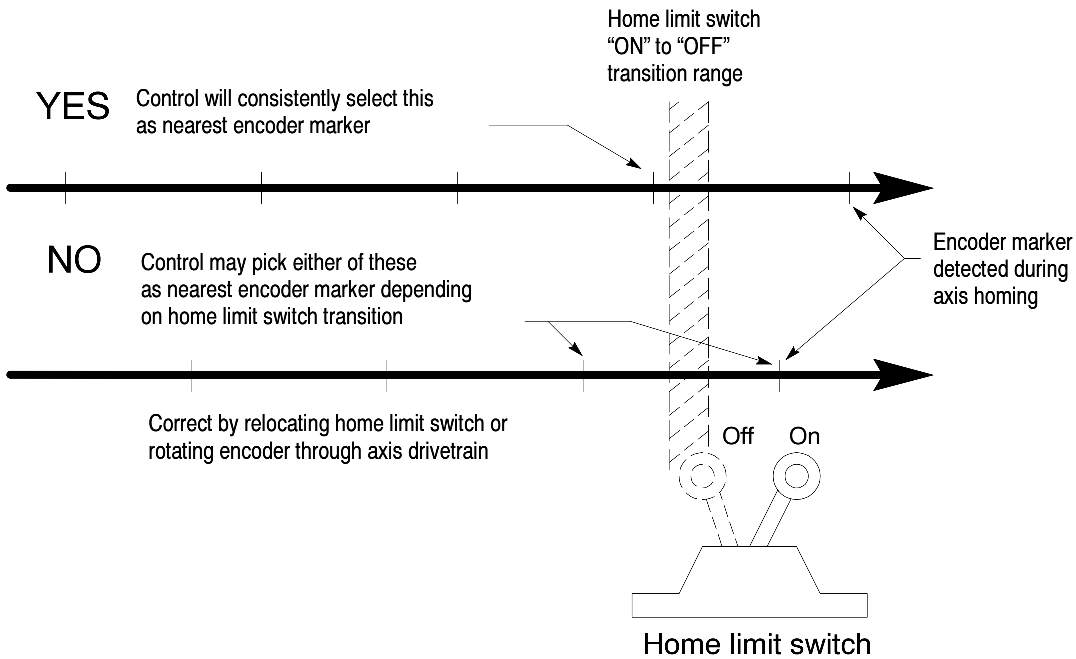
Figure 5.1
Typical Homing Sequence



Mechanical Considerations

ATTENTION: If the encoder marker is exactly 180 degrees away from the point at which the home limit switch transitions from "ON" to "OFF," an undesirable condition is created that could affect homing repeatability. To avoid this condition, we recommend that the encoder or the home limit switch be mounted so that the encoder marker is near or within that switch transition range. Refer to Figure 5.2.

Figure 5.2
Encoder/Home Limit Switch Positioning Considerations



The control allows for a variety of homing hardware configurations and homing procedures. These variations require proper AMP and PAL programming to perform effectively.

Here are some common variations that can be accommodated:

- use a momentary contact home limit switch; if the axis passes the switch, it reverts to the "OFF" state
- hold a home limit switch in the "ON" state whenever the axis is on one side of it; hold it in the "OFF" state when the axis is on the other side
- after tripping the home limit switch, the axis can move in either direction to the home position
- add a positive or negative home calibration distance to the home position

Homing Axes with Distance-coded Marker Scales

Axes with linear scales that incorporate distance-coded markers (DCMs) do not home to a limit switch. Instead, these systems find absolute position when any three consecutive markers are passed on the scale. Since all consecutive markers are a different distance apart, the control can identify which three markers it just passed by identifying how many counts are between these markers.

For manual homing, the operator selects the direction and speed at which these markers are to be found, using the same procedure as the **Manual Homing Sequence Using Home Switch** describes; however, no limit switch is used. Homing is complete when three consecutive markers are passed.

Automatic homing is also available. This allows AMP to select the direction and speed of the G28 homing operation when using G28 Homing of Linear Scales with DCMs.

The following parameters are needed to home linear scales using DCMs:

- G28 Direction to Home (refer to page 5-15)
- G28 Home Speed (refer to page 5-16)
- Machine Pos. at DCM Scale 0 (refer to page 5-18)
- Smaller Period for DCM Scales (refer to page 7-44)
- Larger Period for DCM Scales (refer to page 7-45)

Important: Switchless homing is not applicable to axes with distance coded marker feedback. Attempting to program this feature with a DCM (FW_HMNO set to true) results in the error, “HOMING NOT ALLOWED”.

Manual Homing Sequence in Home Mode

The following outlines a manual homing operation for systems with DCMs when you set the Jog Select Switch to “Home”. Select “Home” if your axis **was never** homed.

1. Assuming you are using the standard MTB Panel, set the Mode Select switch to “Manual”. Once the switch is set, set the Jog Select switch to “Home”.
2. Press the appropriate Axis/Direction button to move the DCM axis across three consecutive markers. Speed for this move can be altered manually through use of the jog speed and feedrate override switches.

   Once the homing sequence is initiated, the jog speed switch has no effect on the axis speed. However, the feedrate override switch still effects speed.

   After the axis passes three consecutive markers, the axis stops. Here, the control waits until the following error is within the range specified in the **Inposition Band** parameter (refer to page 7-24).

Once the axis is within the in position band, the control recalculates the absolute axis position, based on the position *after* the axis crosses the three consecutive markers added to the value in **Machine Pos. at DCM Scale 0**.

`New Abs. Pos. = DCM Physical Abs. Pos. + Machine Pos. at DCM Scale 0`

Important: Use **Machine Pos. at DCM Scale 0** to determine the coordinate value at the actual zero marker on your linear scale. This parameter shifts the scale's machine coordinate system so the value entered becomes the actual value of the zero marker. For more information about this parameter, refer to page 5-18.

Manual Homing Sequence in Continuous or Incremental Mode

The following outlines a manual homing operation for systems with DCMs when you set the Jog Select Switch to "Continuous" or "Incremental". Select "Continuous" or "Incremental" if your axis **has not** been homed.

1. Assuming you are using the standard MTB Panel, set the Mode Select switch to "Manual". Once the switch is set, set the Jog Select switch to "Incremental" or "Continuous".
2. Press the appropriate Axis/Direction button to move the DCM axis across three consecutive markers. When you release the Axis/Direction button, the control automatically determines the absolute position when the output command equals 0.

   When the axis comes to a stop, the control waits until the following error is within the range specified in the **Inposition Band** parameter (refer to page 7-24).

   Once the axis is within inposition, the control recalculates the absolute axis position, based on the position *after* the axis crosses the three consecutive markers added to the value in **Machine Pos. at DCM Scale 0**.

`New Abs. Pos. = DCM Physical Abs. Pos. + Machine Pos. at DCM Scale 0`

Important: Use **Machine Pos. at DCM Scale 0** to determine the coordinate value at the actual zero marker on your linear scale. This parameter shifts the scale's machine coordinate system so the value entered becomes the actual value of the zero marker. For more information about this parameter, refer to page 5-18.

Homing Axes with 1326 Motors with Absolute Feedback

Axes that have the HIPERFACE high-resolution absolute feedback included in the 1326 motors do not require home to a permanent limit switch. In fact, the home operation is not required once the initial machine integration is completed. The initial home operation that is required to initially set the reference position can be accomplished in one of two ways; the home switch method, and the switchless homing method. Both of the methods require coordination with the machine logic program.

Home Switch Method

If using this method, a dedicated home limit switch is mounted to the axis. The operator procedure is identical to that described on page 5-2. Primarily this means that axis motion will occur, as detailed in that section, and the **Axis Position After Homing** value will be aligned with a high-resolution feedback device null.

On any subsequent control resets, or power cycles, the current position of the feedback device is read, and using the stored reading taken during homing, the axis absolute position is determined. This position is displayed as the current absolute position of the axis.

Switchless Homing Method

For this method of homing, the axis is typically moved to a specific location identified by: a visible mark on the slide, a laser reading from a fixed location, or holding the axis against a permanent stop. Once the axis is moved to the required location, the reference operation is initiated as defined by the system integrator. This may be the same sequence as for manual homing, or a different sequence of operator actions.

The difference with switchless homing is that a different set of system logic flags are used to cause the CNC to reference the axis position. These are the **Switchless Homing flags**. When these flags are used, no axis motion occurs. Instead, the AMP Axis position after homing value is set in the absolute position registers for the current location.

For this method of referencing an axis, the home Calibration parameter is not used. That is, the value of that parameter has no effect on the axis absolute reference position.

Homing Sequence Parameters

The parameters defined in this section configure the operation of the homing sequence for each axis. These parameters are significant for both manual and automatic axis homing:

Parameter:	Page:
Dir. to Move Off Limit Switch	5-9
Home Calibration	5-10
Axis Position after Homing	5-11
Home Speed from Limit Switch	5-12

Dir to Move Off Limit Switch

Function

This parameter specifies the direction in which the axis moves after it has tripped the home limit switch.

After the home limit switch is turned "ON," the axis is commanded to stop. It decelerates until it is within the range defined by the AMP **Inposition Band** parameter.

The axis then proceeds in the direction entered for this parameter.

Axis	Parameter Number
(1)	[1002]
(2)	[2002]
(3)	[3002]
(4)	[4002]
(5)	[5002]
(6)	[6002]
(7)	[7002]
(8)	[8002]

Range

Selection	Result
(a)	Negative direction off switch
(b)	Positive direction off switch

Notes

This parameter must be set independently for each axis.

This parameter is not used for axes that use distance coded marker linear feedback devices.

Home Calibration

Function

This parameter specifies the distance from the encoder marker to the desired home position. The control moves this distance plus the distance to the encoder marker after the home limit switch changes from "ON" to "OFF."

Use this parameter to define a precise mechanical home position without adjusting the home limit switch or encoder positions.

Axis	Parameter Number
(1)	[1300]
(2)	[2300]
(3)	[3300]
(4)	[4300]
(5)	[5300]
(6)	[6300]
(7)	[7300]
(8)	[8300]

Range

-999999.99000 to 999999.99000 mm

or

-39370.07835 to 39370.07835 in.

Notes

Use this parameter to compensate for mechanical changes that otherwise would have relocated the home position for an axis.

For example, it may be necessary to move the encoder or home limit switch to avoid the undesirable condition mentioned in the Caution note on page 5-4. Use the **Home Calibration** parameter to compensate for this mechanical change.

As another example, the encoder orientation relative to the axis position may change when repairing gears or gear belts. Consequently the distance that the axis moves off the home limit switch during homing may be different. That difference can be measured and entered as, or added to, the **Home Calibration** value.

This parameter is not used for systems using distance coded marker linear feedback devices. Refer to page 5-18 (**Machine Pos. at DCM Scale 0**) for information correlating axis and scale position.

This parameter must be set independently for each axis.

Axis Position After Homing

Function

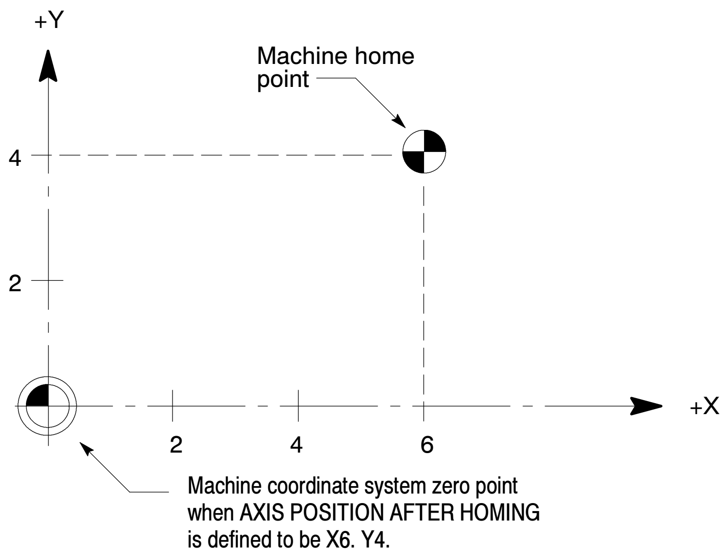
This parameter determines the coordinate value of the axis at its home position.

The home position for an axis can have any legal coordinate value – positive, negative, or zero.

When assigning this value, give consideration to the axis position relative to its range of travel immediately after being homed.

Figure 5.3 shows the results when the **Axis Position after Homing** is 4.0 for the Y axis and 6.0 for the X axis.

Figure 5.3
Defining the Coordinate of the Home Position



Axis	Parameter Number
(1)	[1310]
(2)	[2310]
(3)	[3310]
(4)	[4310]
(5)	[5310]
(6)	[6310]
(7)	[7310]
(8)	[8310]

Range

-2450000.00000 to 2450000.00000 mm

or

-100000.00000 to 100000.00000 in.

Notes

For example, the home position for the lathe axis that moves toward the chuck (typically called the Z axis) is usually at the opposite end of travel from the chuck. A high positive value entered here as the home position coordinate would result in the chuck face having a coordinate value at or near zero.

This parameter is not used for systems that use distance coded marker linear feedback devices.

This parameter must be set independently for each axis.

Home Speed from Limit Switch

Function

This parameter specifies the feedrate at which the axis moves to come off the home limit switch.

This feedrate is used for manual and automatic homing. It is used in coming off the switch and also in moving to the calculated home position.

When a homing operation is being executed, this feedrate cannot be altered by any feedrate overrides.

Axis	Parameter Number
(1)	[1350]
(2)	[2350]
(3)	[3350]
(4)	[4350]
(5)	[5350]
(6)	[6350]
(7)	[7350]
(8)	[8350]

Range

0.0060 to 10160.0000 mmpm

or

0.0002 to 400.0000 ipm

Notes

This parameter must be set independently for each axis.

Automatic Homing Parameters

These parameters are related to automatic homing:

Parameter:	Page:
G28 Direction to Home	5-15
G28 Home Speed	5-16
G30 Secondary Home Position	5-17

Automatic homing is similar to manual homing and is described below.

Automatic Homing Sequence

Use the following instructions if your axis **has not** already been homed:

1. To initiate an automatic homing operation, execute a G28 block, either through the part program, or MDI. In this case, the direction will be determined by the parameter **G28 Direction to Home** and the speed will be determined by the parameter **G28 Home Speed**. If the axis was previously homed, the speed will be determined by the rapid feedrate. All feedrate overrides are disabled throughout an automatic homing sequence.
2. Once the home limit switch is tripped, homing follows the same series of events as described in steps 3 through 10 of the Manual Homing Sequence (refer to page 5-3).

A typical homing sequence is illustrated in Figure 5.1. This figure assumes that the value entered for the AMP parameter **Dir. to Move Off Limit Switch** is opposite of the direction moved to the switch (opposite of **G28 Direction to Home** for automatic homing). It also assumes that the sum of the distance to the nearest marker and the **Home Calibration** distance, result in a move in the same direction as **Dir. to Move Off Limit Switch**.

Automatic Homing Sequence (G28) with Distance-coded Markers

The following outlines automatic machine homing (G28) for a DCM axis that **has not** been previously homed. Only axes that have their axis words programmed in the G28 block are homed. For more information about **Automatic Machine Homing (G28)**, refer to the Axis Motion chapter in your *9/PC CNC Operation and Programming Manual*.

1. Execute a G28 block, either through the part program or MDI. The axis moves at a direction and speed defined in AMP by **G28 Direction to Home** and **G28 Home Speed**, respectively. All feedrate overrides are disabled throughout an automatic homing sequence.

   The axis comes to a stop once the axis crosses three consecutive markers on the DCM scale.

   Important: To determine an absolute position using DCMs, you must encounter at least three consecutive markers. Thus, if the axis position will not accommodate this assumption, the axis must be moved to another position before attempting a homing operation.

2. Once the axis crosses three consecutive markers on the DCM scale, the axis stops. Here, the control waits until the following error is within the range specified in the **Inposition Band** parameter (refer to page 7-24).

   Once the axis is within the inposition band, the control recalculates the absolute axis position, based on the position *after* the axis crosses the three consecutive markers added to the value in **Machine Pos. at DCM Scale 0**.

3. The axis moves to the home position as defined by the parameter **Axis Position After Homing** at a speed defined by the parameter **Home Speed from Limit Switch**.

The following outlines automatic machine homing (G28) for a DCM axis that **has** been previously homed.

1. Execute a G28 block, either through the part program or MDI. The axis moves at a direction determined by the parameter **G28 Direction to Home** and a speed determined by the rapid feedrate. All feedrate overrides are disabled throughout an automatic homing sequence.

   Important: The axis does **not** repeat the homing routine of moving to the limit switches and searching for the encoder marker.

2. The axis moves to machine home via an intermediate point. The control stores this intermediate point specified by the axis word in memory to be used as the point of return for the automatic return **from** machine home operation called out by G28.

   The return operation generates two axis moves both executed at the rapid feedrate: to the intermediate point and to the axis home position.

Important: DCM axis homing must be performed manually or by programming a G28. Attempting to program any motion command other than a G28 will result in the decode error, “MUST HOME AXIS”.

For more information regarding **Automatic Return to Machine Home (G28)**, refer to the Axis Motion chapter in your *9/PC CNC Operation and Programming Manual*.

G28 Direction to Home

Function

This parameter specifies the initial direction the axis moves while searching for the home limit switch during an automatic homing operation (G28).

If the axis has already been homed when the G28 is commanded, this value is not used. Refer to your *9/PC CNC Operation and Programming Manual* for more information.

Important: Special Logic programming or operator instruction may be required to position the axis on the correct side of the home limit switch when a G28 is executed.

Axis	Parameter Number
(1)	[1010]
(2)	[2010]
(3)	[3010]
(4)	[4010]
(5)	[5010]
(6)	[6010]
(7)	[7010]
(8)	[8010]

Range

Selection	Result
(a)	plus
(b)	minus

Notes

This parameter must be set independently for each axis.

For axes using A quad B scale feedback with DCMs: Use **G28 Direction to Home** to specify the initial direction your axis moves to cross three consecutive markers during an automatic homing operation (G28). Refer to page 5-5 for more information about Homing Linear Scales with Distance Coded Markers.

G28 Home Speed

Function

This parameter specifies the feedrate for the axis while searching for the home limit switch during an automatic homing operation (G28).

Once the switch is found the axis moves in the direction determined by the AMP parameter **Dir to Move Off Limit Switch** at a feedrate determined by the AMP parameter **Home Speed from Limit Switch**.

If the axis has already been homed when the G28 is commanded, this value is not used. Refer to your *9/PC CNC Operation and Programming Manual* for more information on automatic homing.

Important: If the speed selected here is too great, the axis may move "through" the home switch before coming to a stop. If the switch changes from "OFF" to "ON" and then back to "OFF" before the axis can stop, and the **Dir to Move off Limit Switch** is the same as **G28 Direction to Home**, the control generates a homing error. This error is "JOGGED HOME TOO FAST: " and includes the name of the violating axis.

Axis	Parameter Number
(1)	[1011]
(2)	[2011]
(3)	[3011]
(4)	[4011]
(5)	[5011]
(6)	[6011]
(7)	[7011]
(8)	[8011]

Range

0.0000 to 10160.0000 mmpm

or

0.0000 to 400.0000 ipm

Notes

This parameter must be set independently for each axis.

For axes using A quad B scale feedback with DCMs: Use **G28 Home Speed** to specify the initial direction your axis moves to cross three consecutive markers during an automatic homing operation (G28). Refer to page 5-5 for more information about Homing Linear Scales with Distance Coded Markers.

G30 Secondary Home Position

Function

This parameter determines the coordinate value to be assigned to a secondary home position for the axis.

The G30 secondary home command provides an alternative home position, often necessary if special attachments or different tool changer are to be used. Refer to your *9/PC CNC Operation and Programming Manual* for more information.

The secondary home position for an axis can have any legal coordinate value; positive, negative, or zero.

Axis	Parameter Number
(1)	[1004]
(2)	[2004]
(3)	[3004]
(4)	[4004]
(5)	[5004]
(6)	[6004]
(7)	[7004]
(8)	[8004]

Range

-2540000.00000 to 2540000.00000 mm

or

-100000.00000 to 100000.00000 in.

Notes

The axis moves to the position indicated by the coordinate value entered here whenever a G30 is executed. If the value entered here is identical to the valued entered for the AMP parameter **Axis Position after Homing**, then the G30 functions as if a G28 were programmed after homing.

This parameter must be set independently for each axis.

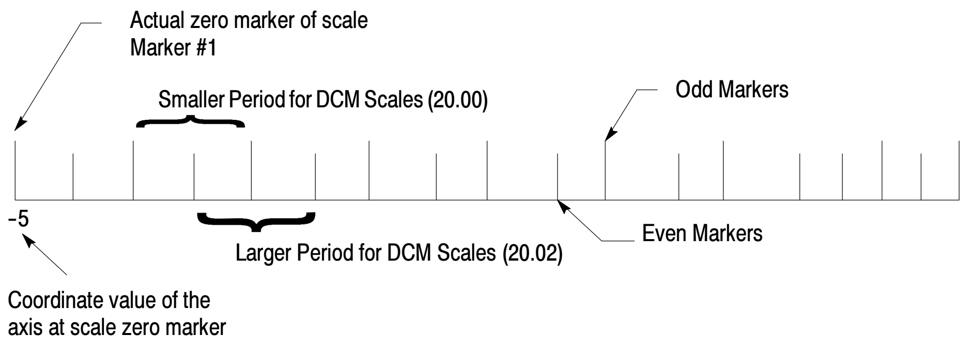
Machine Position at DCM Scale 0

Function

This parameter is only available for axes that use DCMs with a linear scale. See Appendix B for details on integrating a DCM scale.

This parameter is similar to the homing parameter "Axis Position After Homing". The value entered here is used to define the axis' coordinate value at the the actual zero marker on a linear scale. Use this parameter to shift the machine coordinate system on your scale. The value entered here becomes the actual value of the zero marker.

For example, if you want the zero marker on your scale to be displayed and used for programming as the position -5, enter -5 as the machine position at DCM scale 0.



Axis	Parameter Number
(1)	[1502]
(2)	[2502]
(3)	[3502]
(4)	[4502]
(5)	[5502]
(6)	[6502]
(7)	[7502]
(8)	[8502]

Range

-2540000.00000 to 2540000.00000 mm

or

-100000.00000 to 100000.00000 in.

Notes

This parameter must be set independently for each axis that uses DCM scales for the position feedback.

END OF CHAPTER

Zone/Overtravel Parameters

Chapter Overview

Zones and overtravels define areas that restrict the movable range of the cutting tool. The control is equipped to establish two overtravel areas and two programmable zones.

There are two types of overtravel:

- Hardware overtravels -- Established by mounting mechanical limit switches on the movable range of the axes (see documentation prepared by the system installer).
- Software overtravels -- Established in AMP by assigning coordinate values in the machine coordinate system that the axes may not exceed.

There are two types of programmable zones:

- Programmable Zone 2 -- Is established either on the control or in AMP by assigning coordinate values in the machine coordinate system that define an area that the axes may not enter. Programmable zones may be turned on and off in a part program.
- Programmable Zone 3 -- Is established either on the control or in AMP by assigning coordinate values in the machine coordinate system that define an area that the axes may not enter or the axis may not exit (depending on the location of the axis when the zone is activated). These values may also be assigned through programming. Programmable zones may be turned on and off in a part program.

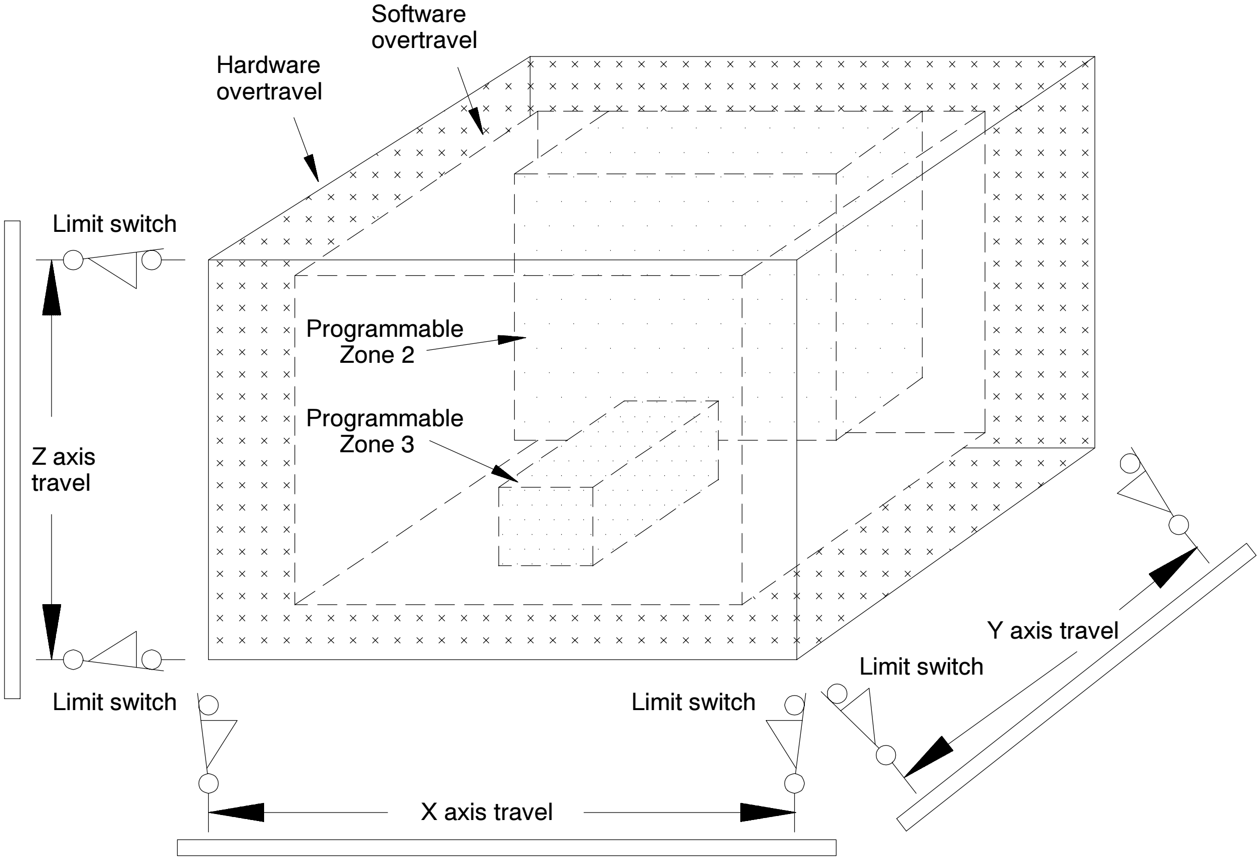
Zones are meant to be less permanent than overtravels and prohibit axis movement into machine attachments. On a lathe, you might set up a zone to protect the lathe chuck. Overtravels are usually used to prohibit axis motion from exceeding the physical travel limits of the machine.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: The programmable zones and software overtravels do not take into consideration the tool length or radius of the tool that might currently be installed in the machine. These zones respect only the axis position in the machine coordinate system.

Figure 6.1 shows a typical configuration of zones and overtravels.

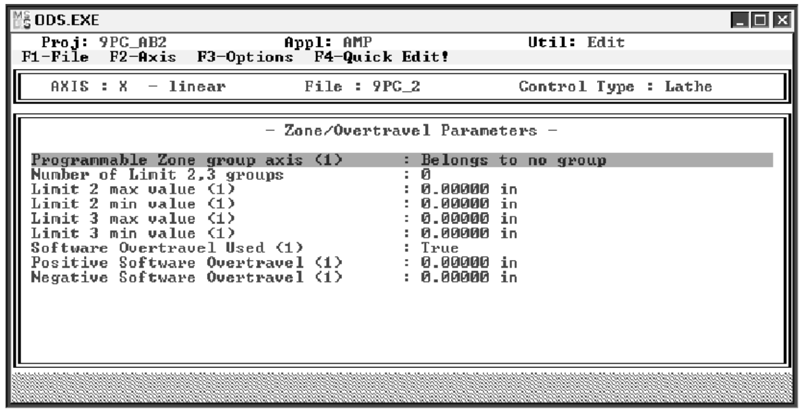
Figure 6.1
Overtravels and Programmable Zones



Refer to these areas:

Parameter:	Page:
programmable zones	6-3
software overtravels	6-10

When you select the "Zone/Overtravel Parameters" group from the main menu in AMP, the workstation displays this screen:

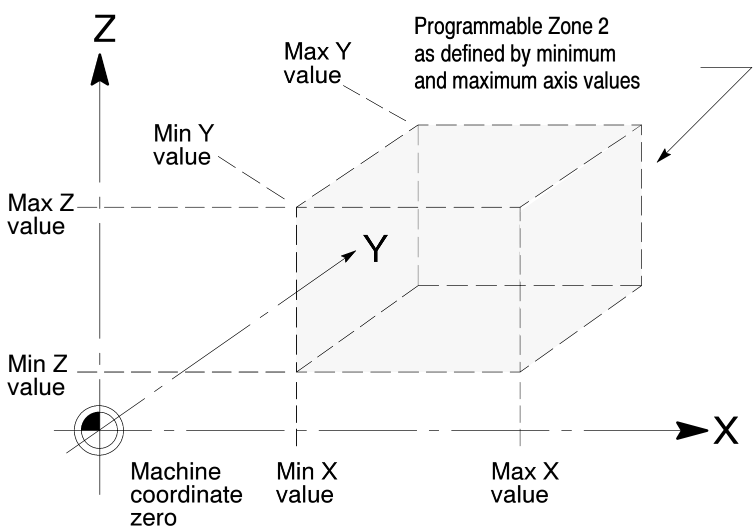


Your screens may differ slightly, depending on your application type.

Zones

The control can have two programmable zones into or out of which the tool is not permitted to move. A typical configuration of zone 2 is shown in Figure 6.2:

Figure 6.2
Programmable Zone 2



Zone 2 defines an exterior zone (tool must stay outside of the zone boundaries). This zone is enabled or disabled by part program commands. The size and location of this zone are determined in AMP or on the control.

Zone 3 defines either an interior zone (tool must stay within zone boundaries) or an exterior zone (tool must stay outside of the zone boundaries). This zone is enabled or disabled by part program commands. The size and location of this zone is determined in AMP, on the control, or through programming commands.

Programmable zones can check:	where:
each axis independently	the tool cannot move past the coordinate value set for an axis
axes simultaneously	the coordinate values set for each axis define an area that the tool may not move into or out of

These subsections offer a discussion on the parameters that are used for the programmable zones.

Subsection:	Page:
Programmable Zone Group Axis	6-4
Number of Limit 2, 3 Groups	6-6
Limit 2 Max Value	6-7
Limit 2 Min Value	6-8
Limit 3 Max Value	6-8
Limit 3 Min Value	6-9

Programmable Zone Group Axis

Function

Use this parameter to assign each axis to a particular programmable zone group. Programmable zones are checked in groups.

If 3 axes are assigned to a group, a cube shaped zone is formed. If only two axes are assigned to that group, then the two sides of the cube that would have been defined by that axis open up all the way to the positive and negative software overtravels for that axis.

Figure 6.1 assumes that axes 1, 2, and 3 have all been assigned to the same zone group.

Figure 6.3 illustrates the results of assigning the Z axis to one group by itself, and the X and Y axes to another group. The result is two 3-dimensional rectangular areas that the tool cannot enter (or cannot exit under zone 3 conditions).

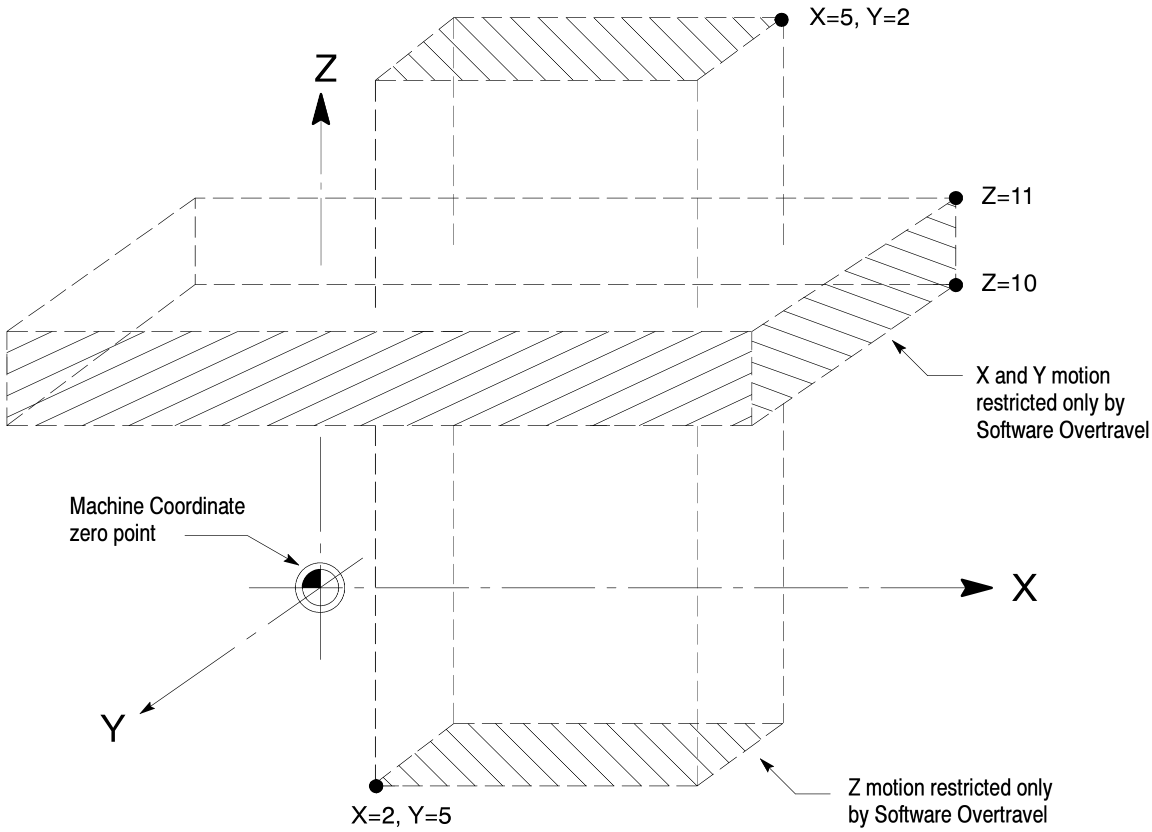
Up to 3 programmable zone groups are available. By assigning an axis to a group, that axis is checked simultaneously with any other axes in that group.

For example, assume that only the Z axis (axis 3) is entered in group 1 and its minimum and maximum values for zone 2 are set at +10 and +11. Axis 3 will not be allowed to reach any position with a coordinate value between 10 and 11 on that axis regardless of the position of the other axes. The other axes minimum and maximum values are determined by their software overtravels.

Expanding on this example, assume that the X axis (axis 1) and Y axis (axis 2) have both been assigned to group 2 and both have minimum and maximum values for zone 2 set at +2 and +5. When zone 2 is active, the tool may not enter the area that is enclosed by the coordinate values defined by the X and Y axes regardless of the position of the Z axis. The minimum and maximum Z axis values are determined by its software overtravels.

Figure 6.3 shows the zone resulting from the conditions described in this example.

Figure 6.3
Zone 2 Programmable Zone Groups



All of the axes may be assigned to one group if it is desired. This results in 3-dimensional zones as shown in Figure 6.1.

Choose from these options for this parameter:

Belongs to no groups – When this is selected, it indicates that zones 2 and 3 are not used for this axis.

Group 1 – If this is selected for an axis, it indicates that its values for programmable zones 2 and 3 are checked simultaneously with any other axis in group 1.

Group 2 – If this is selected for an axis, it indicates that its values for programmable zones 2 and 3 are checked simultaneously with any other axis in group 2.

Group 3 – If this is selected for an axis, it indicates that its values for programmable zones 2 and 3 are checked simultaneously with any other axis in group 3.

Axis	Parameter Number
(1)	[1009]
(2)	[2009]
(3)	[3009]
(4)	[4009]
(5)	[5009]
(6)	[6009]
(7)	[7009]
(8)	[8009]

Range

Selection	Result
(a)	Belongs to no groups
(b)	Group 1
(c)	Group 2
(d)	Group 3

Notes

This parameter must be set independently for each axis.

Number of Limit 2,3 Groups

Function

There are 3 programmable zone groups for the programmable zones (see the parameter **Programmable Zone Group**). By assigning an axis to a group, that axis is checked simultaneously with any other axes in that group.

This parameter tells the system how many zone groups that the machine has to check. The axes are assigned to these groups by using the parameter **Programmable Zone Group**.

If zero is selected for this parameter, it indicates to the control that there are no programmable zone groups, and the programmable zone feature is disabled.

Axis	Parameter Number		
	Single Process	Process 1	Process 2
All	[20]	[20020]	[21020]

Range

0 to 3

Notes

This parameter is global; the value set here applies to all axes.

Limit 2 Max Value

Function

This parameter is used to enter the larger machine coordinate value that determines the location of programmable zone 2. This value may be altered on the control, if desired, by using the programmable zone table. When programmable zone 2 is active, the axis is not permitted to exceed this value.

Axis	Parameter Number
(1)	[1005]
(2)	[2005]
(3)	[3005]
(4)	[4005]
(5)	[5005]
(6)	[6005]
(7)	[7005]
(8)	[8005]

Range

Limit 2 Min value to 999999.99000 mm

or

Limit 2 Min value to 39370.07835 inch

The range of this parameter is dependent on the value set for the parameter "Limit 2 Min Value."

Notes

This parameter must be set independently for each axis.

Limit 2 Min Value

Function

Use this parameter to enter the smaller machine coordinate value that determines the location of programmable zone 2. This value may be altered on the control, if desired, by using the programmable zone table. When programmable zone 2 is active, the axis is not permitted to move to a position that is smaller than this value.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1006]	(5)	[5006]
(2)	[2006]	(6)	[6006}
(3)	[3006]	(7)	[7006]
(4)	[4006]	(8)	[8006]

Range

-999999.99000 mm to Limit 2 max value

or

-39370.07835 inch to Limit 2 max value

The range of this parameter is dependent on the value set for the parameter “Limit 2 Max Value.”

Notes

This parameter must be set independently for each axis.

Limit 3 Max Value

Function

Use this parameter to enter the larger machine coordinate value that determines the location of programmable zone 3. This value may be altered on the control, if desired, by using the programmable zone table or by the proper programming commands. The area defined as programmable zone 3 limits axis motion according to the position of the axis when the zone is activated.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1007]	(5)	[5007]
(2)	[2007]	(6)	[6007]
(3)	[3007]	(7)	[7007]
(4)	[4007]	(8)	[8007]

Range

Limit 3 Min value to 999999.99000 mm

or

Limit 3 Min value to 39370.07835 inch

The range of this parameter is dependent on the value set for the parameter “Limit 3 Min Value.”

Notes

This parameter must be set independently for each axis.

Limit 3 Min Value

Function

Use this parameter to enter the smaller machine coordinate value that determines the location of programmable zone 3. This value may be altered on the control, if desired, by using the programmable zone table or by the proper programming commands. The area defined as programmable zone 3 limits axis motion according to the position of the axis when the zone is activated.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1008]	(5)	[5008]
(2)	[2008]	(6)	[6008]
(3)	[3008]	(7)	[7008]
(4)	[4008]	(8)	[8008]

Range

-999999.99000 mm to Limit 3 max value

or

-39370.07835 inch to Limit 3 max value

The range of this parameter is dependent on the value set for the parameter “Limit 3 Max Value.”

Notes

This parameter must be set independently for each axis.

Software Overtravel Parameters

The software overtravel cannot be canceled on the control. AMP is the only means available to activate, deactivate, and establish the range for the software overtravel. These subsections offer a discussion of parameters used to establish the software overtravel:

Subsection:	Page:
Software Overtravel Used	6-10
Positive Software Overtravel	6-11
Negative Software Overtravel	6-11

Important: Note that the software overtravel is only effective provided that a homing operation has been conducted for that axis. If the axis has not been homed, there is no machine coordinate system from which the software overtravel can be referenced.

Software Overtravel Used

This parameter is used to determine whether the software overtravel is used for a specific machine application. Setting a value of "True" for this parameter causes the software overtravel to be active for that axis. Setting a value of "False" for this parameter disables the software overtravel for that axis.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1020]	(5)	[5020]
(2)	[2020]	(6)	[6020]
(3)	[3020]	(7)	[7020]
(4)	[4020]	(8)	[8020]

Range

Selection	Result
(a)	True
(b)	False

Notes

This parameter must be set independently for each axis.

Positive Software Overtravel

Function

Use this parameter to enter the larger machine coordinate value that determines the location of the software overtravel. Provided that software overtravels are active, the axis is not permitted to exceed this value.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1320]	(5)	[5320]
(2)	[2320]	(6)	[6320]
(3)	[3320]	(7)	[7320]
(4)	[4320]	(8)	[8320]

Range

Negative Software Overtravel to 2540000.00000 mm

or

Negative Software Overtravel to 100000.00000 inch

The range of this parameter is dependent on the value set for the parameter "Negative Software Overtravel."

Notes

This parameter must be set independently for each axis.

Negative Software Overtravel

Function

Use this parameter to enter the smaller machine coordinate value that determines the location of the software overtravel. Provided that the software overtravels are active, the axis is not permitted to move to a position that is smaller than this value.

Axis	Parameter Number	Axis	Parameter Number
(1)	[1330]	(5)	[5330]
(2)	[2330]	(6)	[6330]
(3)	[3330]	(7)	[7330]
(4)	[4330]	(8)	[8330]

Range

-2540000.00000 mm to Positive Software Overtravel

or

-100000.00000 inch to Positive Software Overtravel

The range of this parameter is dependent on the value set for the parameter “Positive Software Overtravel”.

Notes

This parameter must be set independently for each axis.

END OF CHAPTER

Servo Parameters

Chapter Overview

Use the servo parameter group to configure information about the servos controlled by your 9/PC CNC. When you select the "Servo Parameters" group, some or all of the following four screens of parameters are available to you:

Additional servo parameters become available based on your choice of "**Servo Hardware Type**" (SERCOS Interface for your 9/PC control), and the "Servo Loop Type" (None, Position or Analog Spindle, Digital or Digital Spindle, or SERCOS). You can configure the "Servo Hardware Type" and "Servo Loop Type" parameters using the F2 option as discussed on page 3-7 to have the correct parameters displayed for your particular servo system.

Once you selected the **Servo Hardware Type** and **Servo Loop Type**, additional AMP parameters become available to you for configuration in the Servo Parameters Group. Only the parameters that apply to the servo hardware and servo loop types that you selected with the F2 option will appear on these screens. Different parameters appear for different servo configurations. All parameters are discussed in these sections:

These Servo Parameters:	See page:
SERCOS Parameters	7-3
General Servo Parameters	7-6
Position Loop Parameters	7-13
Velocity Loop Parameters	7-46
Digital Parameters	7-70
Spindle Parameters	7-79
Friction Parameters	7-84

Servo Feedback Options

Each servo system contains two loops:

- velocity loop, used to control motor speed vs commanded motor speed
- position loop, used to control axis position vs commanded axis position.

The parameter **Servo Loop Type** has four possible settings. The following table describes what each setting tells the 9/PC about your servo system hardware configuration.

Servo Loop Type Setting	Typical Servo Drive Hardware	Servo System Description
Position or Analog Spindle	1394 CNC serial drive, with this servo, using the analog output as a velocity command to another amplifier.	Using this setting means that a separate drive amplifier with an analog interface is used to directly control the motor velocity. With this method, the velocity feedback is generated by a motor-mounted device and the velocity loop is closed by the motor drive system. Position feedback is generated by an encoder either mounted on the motor shaft or at some other point on the axis and the position loop is closed by the 1394 drive. The AMP values are sent to the drive over the SERCOS connection.
Digital or Digital Spindle	1394 CNC serial drive, with this servo using a 1394 axis module as the amplifier.	This selection must be made when the 1394 axis modules are used for the servo. The 1326 motor internal feedback device must be used to allow motor commutation. An additional feedback device may be added to serve as the position loop device. For this configuration, all of the servo operational settings defined in AMP are sent to the 1394 drive over SERCOS.
SERCOS	This setting indicates that a SERCOS drive other than a CNC serial drive is connected to the 9/PC for control of this axis. This could be a SERCOS spindle or axis drive.	This selection identifies the drive hardware as SERCOS. With this method both the velocity and position loop are closed by the servo drive. No servo operation parameters are sent to the drive from the 9/PC. Using the SERCOS ring, the 9/PC CNC transmits course position commands to the drive. The drive then returns position information to the control. Refer to the drive's documentation for detail on compatible feedback devices, and how to make configuration settings.

SERCOS Parameters

The parameters in this section apply to the configuration of the SERCOS network. Refer to the documentation accompanying your SERCOS drive prior to configuring these parameters to identify your drive's capabilities.

Baud Rate for SERCOS

Function

Use this parameter to select the rate at which the SERCOS ring will operate. The value of this parameter is dictated by your drive hardware. Refer to the documentation accompanying your SERCOS drive to identify the baud rate the drive is capable of handling.

For example if three drives on your ring are capable of 4 MHz and one drive is capable of 2 MHz the ring should be configured at 2 MHz as dictated by the slowest device. The default for this parameter is 2 MHz.

Axis	Parameter Number
All	[97]

Range

Selection	Result
(a)	2 MHz
(b)	4 MHz

Notes

This is not a "per axis" parameter. The value entered here affects all axes.

Cycle Time for SERCOS

Function

Use this parameter to specify the rate at which a SERCOS drive can process data from the 9/PC CNC. This is the servo update rate. The value of this parameter is dictated by your drive hardware. Refer to the documentation accompanying your SERCOS drive to identify the cycle time supported by your drive.

You must set this parameter for the drive with the slowest update time on the SERCOS ring. For example if three drives on your ring are capable of 1 ms and one drive is capable of a 2 ms update rate, the cycle time for all drives on the ring will be slowed to 2 ms.

Important: The minimum SERCOS cycle time must be equal to or less than the value configured for the "System Scan Time". The "System Scan Time" must be an even multiple of this value. For example, if your system scan time is set at 6 ms, valid SERCOS Minimum Cycle Time values are
1, 2, 3, and 6 ms. If your system scan time is set at 8 ms, valid SERCOS Minimum Cycle Time values are 1, 2, 4, and 8 ms.

Important: Maximum SERCOS ring size is indirectly set by this parameter. We recommend not exceeding four servos when set at 1 ms minimum cycle time at a baud rate of 2 MHz. Up to eight servos can be supported when set to 2 ms or slower. Larger 1 ms configurations are possible at 4 MHz.

Axis	Parameter Number
All	[99]

Range

1 to 30 msec

Notes

This is not a "per axis" parameter. The value entered here affects all axes.

SERCOS Transmit Level

Important: This parameter only applies to 1394 CNC Digital Drives and 1394 Analog Interface Modules.

Function

Use this parameter to define the output level of the fiber optic transmitter used by each Allen-Bradley device in the network. By default, the device transmits to the maximum signal.

In most cases and for all Allen-Bradley systems, the default value of the parameter will be suitable to support reliable communication over the network. However, the integrator is provided a high and low setting to vary the transmit power. The capability of lowering the output power may be needed when the receiving device is connected with a short SERCOS cable.

This parameter provides you with the ability to lower the transmitting intensity to avoid overdriving the next device in the ring.

Axis	Parameter Number
(1)	[1504]
(2)	[2504]
(3)	[3504]
(4)	[4504]
(5)	[5504]
(6)	[6504]
(7)	[7504]
(8)	[8504]

Range

Selection	Result
(a)	LOW
(b)	HIGH

Notes

This parameter must be set independently for each servo.

Since this is a per axis parameter and the 1394 is a multiaxis drive, it is important to note that a high setting for any axis on that 1394 chassis results in a high transmit level.

General Servo Parameters

The servo parameters in the sections that follow are general servo parameters. General servo parameters are those servo parameters that must be set regardless of the type of servo hardware and type of servo loop you have selected.

Standard Motor Table Values

Function

This parameter is only used for Allen-Bradley 1394 digital servo systems. Third-party SERCOS users ignore this parameter. Use this parameter to get a starting point for configuring and tuning 1326 digital servo motors.

The control has internal tables that contain data for each of the standard motors. Whenever the control is powered-up, these tables are read to obtain certain values for the standard motors you select in AMP. The appropriate value is then sent to the 1394 Servo drive.

This parameter determines whether or not all of the values for the standard motors configured in AMP are read on power-up.

If this parameter is set to “Yes,” and you are using one of the motors listed in Table 7.B (page 7-55), then all motor parameters are read from the internal tables at power-up .

If this parameter is set to “No,” then the parameters **Motor Rated Current**, **Velocity Proportional Gain**, **Velocity Integral Gain,** and **Number of Poles on Motor** are not read from the internal tables. Instead, the last value entered for these parameters (through either ODS AMP or Patch AMP) prior to turning power off is the active value after power is turned on. Setting this parameter to “No” allows for fine tuning of these key motor parameters for a standard motor. For a procedure on fine tuning a motor, refer to Appendix A of this manual.

Important: This parameter is intended for use with standard motors only at power up. Once you have powered up your system and confirmed wiring, fine tuning of the servo gains is almost always required once the servo shaft is coupled to a load. Using online AMP to adjust the velocity/position loop gains will automatically reset this parameter to No at the control. Once you finish tuning your servos we recommend uploading the AMP from the control to store the new gain values and store this parameter as "No" in the project AMP file.

Axis	Parameter Number
All	[11]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

A typical application of this parameter would be as follows:

1. When you finish setting AMP on the Host PC or ODS workstation, and have selected standard motors wherever they are to be used, enter all appropriate data for the motors, except the parameters **Motor Rated Current**, **Velocity Proportional Gain**, and **Velocity Integral Gain**.
2. Set this parameter to "Yes."
3. Download AMP to the control and test each axis that uses a standard motor.
4. If axis performance is acceptable, leave this parameter set at "Yes".
5. If axis performance is not acceptable, upload AMP to the ODS terminal to view the gain values that the control calculated and entered.
   - **a.** Set this parameter to "No" and modify the gain values as recommended in their parameter descriptions.
   - **b.** Download AMP to the control and test each axis that uses a standard motor.
   - **c.** Restart the control when instructed to do so after downloading or changing AMP. After the restart, the parameters modified in step 5 retain their modified value.

d. Test the axis again, and continue modifying parameters as in step 5.

6. When motor performance is optimized, upload and/or backup the final AMP file (leaving the Standard Motor Table Values parameter set to “No”).

If problems are encountered during fine tuning, the original table values for that particular motor can always be restored by downloading AMP with this parameter set to “yes.” When the control is restarted, the standard motor tables are read and downloaded.

Important: You can use patch AMP to modify this parameter and other gain parameters to simplify tuning. Online AMP screens are also available that allow you to change gain values real time, without a control restart. Adjusting the gain values using online AMP will force this parameter (Standard Motor Tables) to “NO”.

This parameter is global; the value set here applies to all axes.

Servo Hardware Type

Function

We recommend setting this parameter by using the [F2] function key as described on page 3-7, and selecting the “Configure Servo” option. You can however set this parameter directly from the servo parameter screen. This parameter is used to indicate the type of hardware you have purchased for your servo system.

Axis	Parameter Number
(1)	[1030]
(2)	[2030]
(3)	[3030]
(4)	[4030]
(5)	[5030]
(6)	[6030]
(7)	[7030]
(8)	[8030]

Range

Select this Servo Type:	If you Have this Servo Hardware:
(a) None	This is the default value for this parameter. This value should be changed to correctly reflect the servo hardware you purchased.
(b) SERCOS Interface	The 9/PC (SERCOS Interface)

Notes

This parameter must be set independently for each servo.

Once you have configured both the “Servo Hardware Type” and the “Servo Loop Type” additional parameters to configure that specific servo system become available.

Servo Loop Type

Function

You may set this parameter by using the [F2] function key as described on page 3-7, and selecting the “Configure Servo” option, or you can set this parameter directly from the servo parameter screen. This parameter is used to indicate the type of servo loop the drives connected to the 9/PC will use.

Axis	Parameter Number
(1)	[1031]
(2)	[2031]
(3)	[3031]
(4)	[4031]
(5)	[5031]
(6)	[6031]
(7)	[7031]
(8)	[8031]

Range

Servo Loop Type:	Results in:	CNC generates a:
(a) None	Control behaves as if that servo is not controlled.	PTO error
(b) Position or Analog Spindle	Select this as the servo loop type for analog drive systems that have an external tachometer mechanically coupled to the motor shaft. Feedback from this tachometer is typically returned to the servo amplifier to close the velocity loop. Can also be used if the position loop is open or detached.	Position command
(c) Digital or Digital Spindle	The control closes the position and velocity loop. Motor information necessary for proper commutation is provided for digital drive systems. Used for 1394 drive/1326 motor systems.	Digital command (current)
(d) SERCOS	The servo drive closes both the velocity and position loops. The control sends course position commands to the drive and receives position status through the SERCOS ring. Refer to the drive manufacturer's documentation for detail on compatible feedback devices.	Course command

Important: For more detailed information regarding **Servo Loop Type**, refer to page 7-3.

Notes

This parameter must be set independently for each servo.

Once you have configured both the "Servo Hardware Type" and the "Servo Loop Type" additional parameters to configure that specific servo system become available.

Important: When a tachless analog systems detects an E-stop condition, the control sets the torque output command to zero. You need to reduce the motor speed to zero in a safe manner. This can be done through one of these methods:

- Torque Amplifier Dynamic Braking
- Resistor Type Dynamic Braking
- Setting a Motor Brake
- User-defined braking

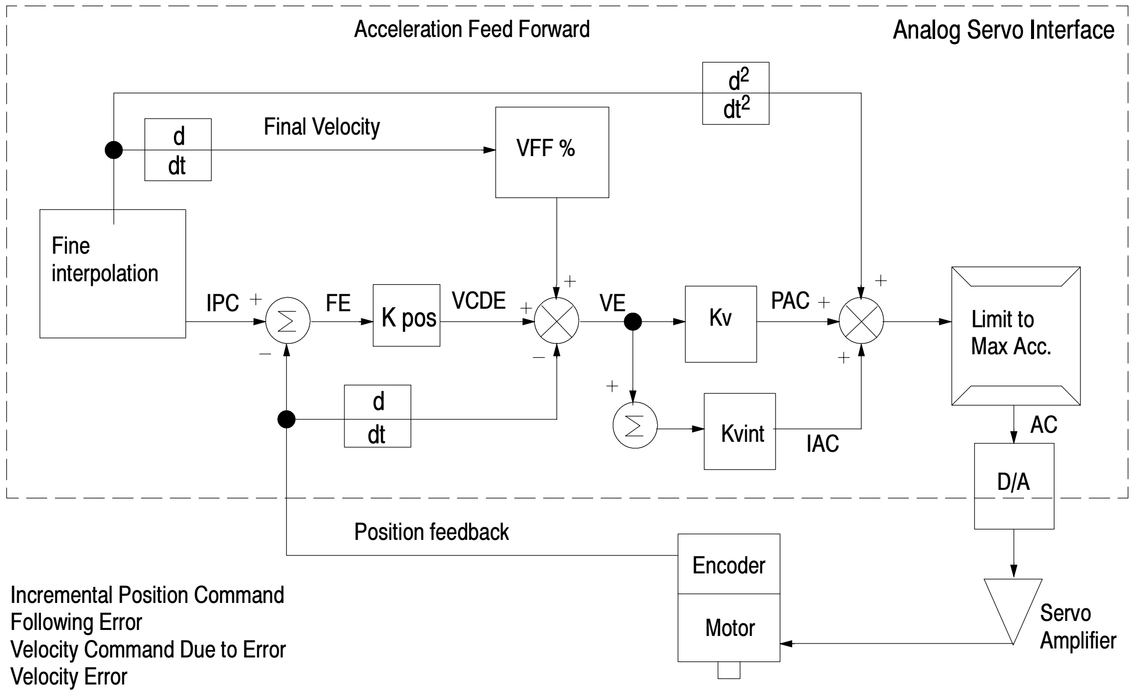
The customer-supplied torque-block amplifier is responsible for both short and long term thermal protection of the amplifier and motor.

The torque command is limited at ±10 volts. You need to scale the peak motor current to this limit.

The parameter **ACC/DEC RAMP** may need to be set to control the speed torque envelope.

This block diagram illustrates tachless operation when the control is in ZFE (zero following error) mode.

Figure 7.4
Block Diagram Tachless Operation (with ZFE)



IPC . . . Incremental Position Command
FE . . . . Following Error
VCDE . . Velocity Command Due to Error
VE . . . . Velocity Error
IAC . . . Integral Term for the Acceleration Command
PAC . . . Proportional Term of the Acceleration Command
AC . . . . Acceleration Command

Output Port Number

Function

For 1394 CNC serial drives, this parameter identifies the specific output for the motor. In making this selection you are also identifying the type of output
(1394 digital axis module or analog output).

If you have this type of axis:	define your (1394) drive amplifier output ports with this parameter:
open-loop analog spindle	Analog Output Connector TB2 or TB3 (serial drive) or Aout1 - Aout6 (analog interface)
closed-loop analog servo	
closed-loop digital servo or digital spindle using 1394 axis	Axis Module 1, 2, 3, or 4 (to associate the axis to an amplifier axis module)

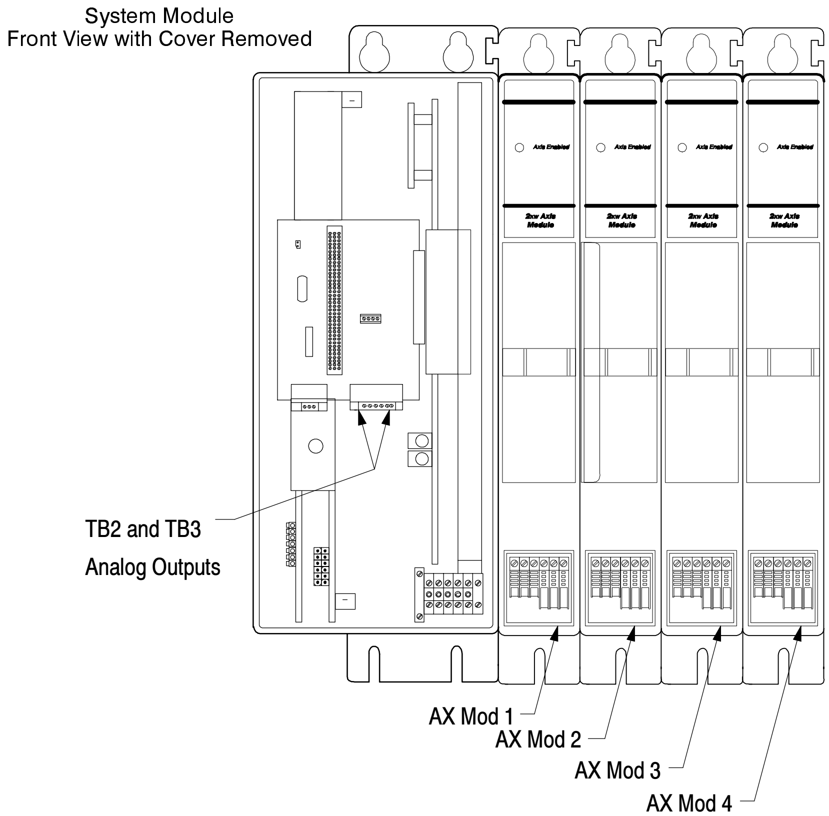
Axis	Parameter Number
(1)	[1540]
(2)	[2540]
(3)	[3540]
(4)	[4540]
(5)	[5540]
(6)	[6540]
(7)	[7540]
(8)	[8540]

Important: This parameter does not apply to third-party SERCOS drives.

Range

Make this selection. . .	. . . if you have this drive/motor connection on the CNC serial drive.
(a) No Output	No output to a drive or motor
(b) Output Axis Mod 1 or Aout1 (1394)	Motor connected to the first axis module or drive connected to AOUT1
(c) Output axis Mod 2 or Aout2 (1394)	Motor connected to the second axis module or drive connected to AOUT2
(d) Output axis Mod 3 or Aout3 (1394)	Motor connected to the third axis module or drive connected to AOUT3
(e) Output axis Mod 4 or Aout4 (1394)	Motor connected to the fourth axis module or drive connected to AOUT4
(f) Analog Out TB2 or Aout5 (1394)	Analog output to a separate servo drive
(g) Analog Out TB3 or Aout6 (1394)	Analog output to a separate servo drive

Figure 7.5
Connectors on the CNC Serial Drive



Notes

This parameter must be set independently for each servo.

Position Loop Parameters

The servo parameters in the sections that follow are available to configure the position loop. These parameters are not available for servos that have the servo loop type configured as "none".

Important: SERCOS systems must set the parameter "Servo Position Loop Type". Other position loop parameters in this section are not used on third-party SERCOS drive systems (drives other than the Allen-Bradley 1394 Serial Drives). Third-party SERCOS drive systems must have the servo loops configured as defined by the drive manufacture. 1394 Serial Drives have this data configured here in AMP and it is downloaded to the drive through the SERCOS ring.

Servo Position Loop Type

Function

There are five different position loop algorithms that you can choose from.

- **Open Loop —** This is used for the spindle or for any motor that the control/drive is not to close the position loop. Typically these motors are connected through one of the ANALOG OUT connectors. Some open-loop motors can be equipped with a position feedback device which provides the control with position information (for features such as IPR or threading) but is not used for closed-loop motion control. The control/drive determines if there is a position feedback device based on your setting of the parameter Position Loop Feedback Port. Refer to Figure 7.6 and Figure 7.7.

  Important: Spindles are always configured as “Open Loop”. The loop type will be changed by the software during a closed-loop orient, solid tapping operation, or any other operation that requires closed-loop spindle operation.

  SERCOS users should make this selection for devices on the ring for which the control will not expect position status information (typically spindle drives).

Figure 7.6
Open-loop Block Diagram (no position feedback)

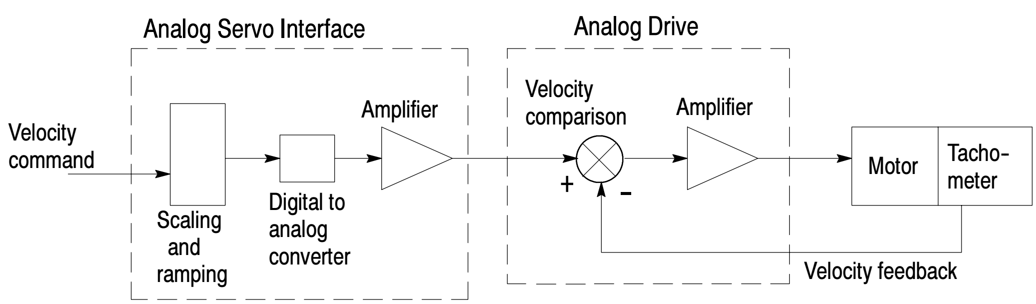
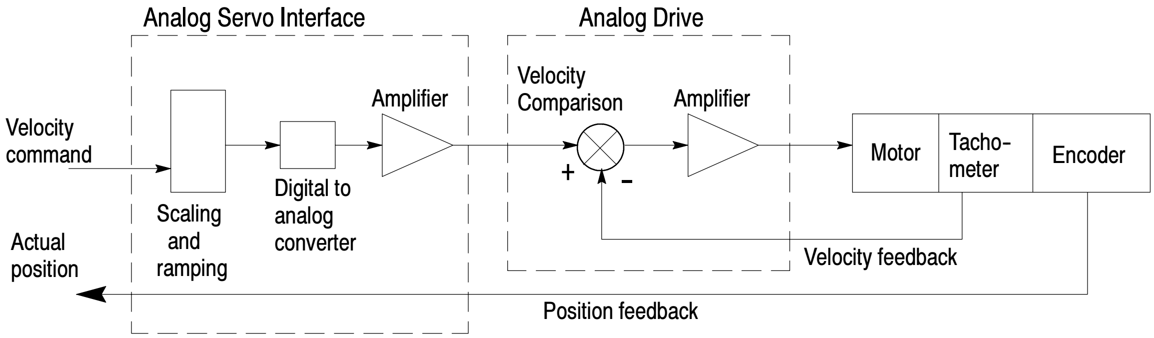


Figure 7.7
Open-loop with Position Feedback Block Diagram



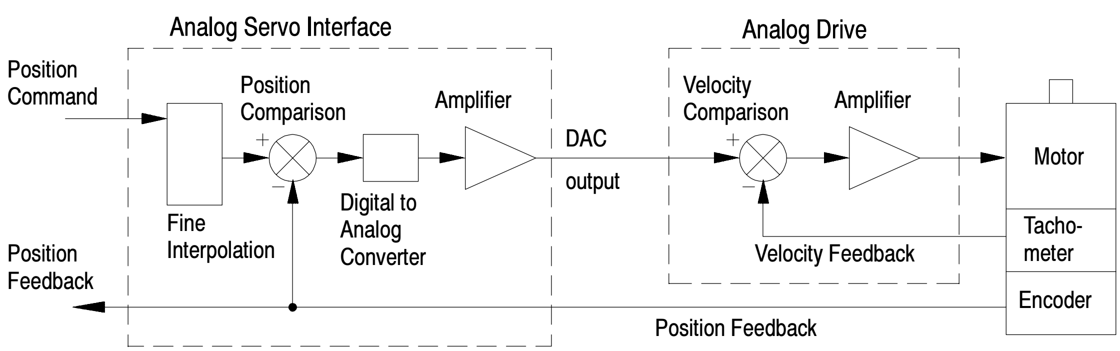
- **Closed Loop —** Use this for any axis that returns position feedback to the control for use in closing the position loop. The axis must be equipped with a position feedback device (e.g., encoder).

  In closed loop, the velocity command sent to the servo drive is proportional to the following error. Following error is the difference between the commanded and the actual axis position. Refer to Figure 7.8.

  SERCOS users should make this selection for devices on the ring that the control expects position status information (most servo systems) from. Remember, SERCOS systems do not close the loop at the control; the loop is closed at the drive. This information is only used to identify current servo position.

Important: A closed-loop axis requires homing. Refer to the homing parameters of chapter 5.

Figure 7.8
Closed-loop Block Diagram



- **ZFE Closed Loop —** Use this for any axis that returns position feedback to the control for use in closing the position loop. The axis must be equipped with a position feedback device (encoder).

ZFE stands for Zero Following Error. Following error is the difference between the commanded position and the actual axis position, and zero following error refers to the control's ability to minimize that difference after reaching the commanded feedrate.

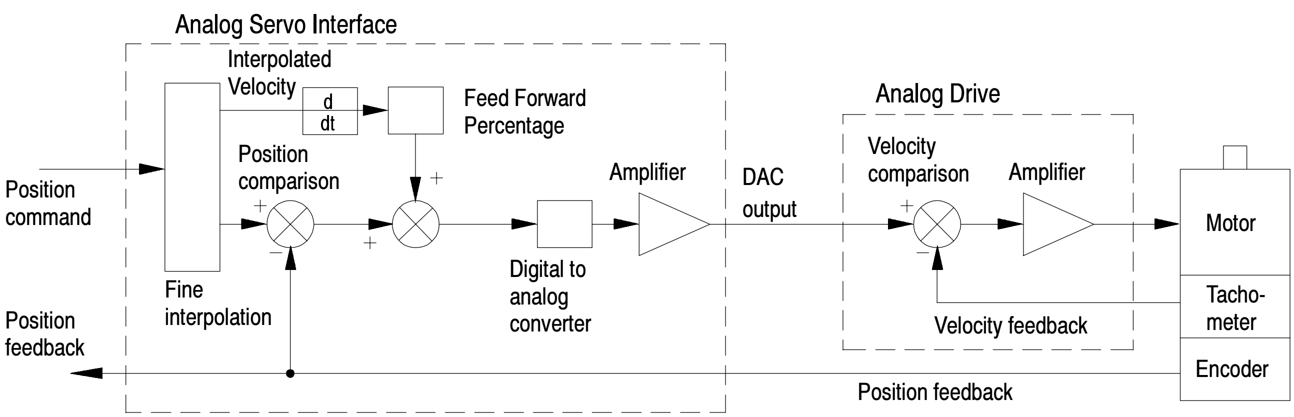
This is accomplished by allowing a percentage of the velocity command from fine interpolation to be summed with the following error to produce the velocity signal. This is referred to as velocity feed-forward. Refer to Figure 7.9.

Advantages to ZFE Closed Loop are
1) improved performance during high precision or high speed applications;
2) faster part program execution because velocity transition times are reduced.

Third-party SERCOS users do not make this selection. ZFE closed-loop is not available on third-party SERCOS drives. However, 1394 Serial Drives are able to run ZFE closed-loop. For more information on ZFE closed-loop, refer to page 7-28.

Important: A closed-loop axis requires homing. Refer to the homing parameters of chapter 5.

Figure 7.9
ZFE Closed-loop Block Diagram



- **Servo Off —** The servo output has been disabled, but feedback can still be monitored. This provides a valuable tool for diagnosing feedback problems. With proper Logic programming, this can also be used to "measure" tool or part position as in digitizing.

  SERCOS users make this selection when motor position status is to be monitored from the drive however no course commands will be issued to provide motor torque. Be aware that drive logic may still be active to issue torque commands to maintain position.

- **Servo Detached —** There is no servo connected to this axis. The servo output and feedback are both disabled. The control will display a series of dashes in the Axis Position screen to indicate a detached axis.

Axis	Parameter Number
(1)	[1510]
(2)	[2510]
(3)	[3510]
(4)	[4510]
(5)	[5510]
(6)	[6510]
(7)	[7510]
(8)	[8510]

Range

Selection	Result if axis is linear or rotary
(a)	Open Loop
(b)	Closed Loop
(c)	ZFE Closed Loop
(d)	Servo Off
(e)	Servo Detached

Notes

This parameter must be set independently for each servo.

The significance of most of the following axis servo parameters is directly related to the selection made here.

Lead Screw Thread Pitch

Function

Enter the pitch of the lead screw for the axis being configured. Pitch refers to the number of inches or millimeters between each thread.

For example, a common lead for machine tools in the U.S. is 5 threads per inch. The lead screw thread pitch in this case would be:

```
 1 inch
----------  =  .20 inch
 5 threads
```

Refer to figure 7.13 for an illustration of the mechanical configuration of a typical axis.

Axis	Parameter Number
(1)	[1590]
(2)	[2590]
(3)	[3590]
(4)	[4590]
(5)	[5590]
(6)	[6590]
(7)	[7590]
(8)	[8590]

Range

0.000 to 254.000 mm

0.000 to 10.000 in.

Notes

This parameter must be set independently for each servo.

Reversal Error Compensation

Function

When an axis is commanded to reverse direction, mechanical play in the machine (through the gearing, ballscrew, etc.) may result in a small amount of motor motion without axis motion. As a result, the feedback device may indicate movement even though the axis has not moved.

The control uses the value entered for the **Reversal Error Compensation** parameter to compensate for this motion. The value entered is added to the beginning of the commanded axis motion.

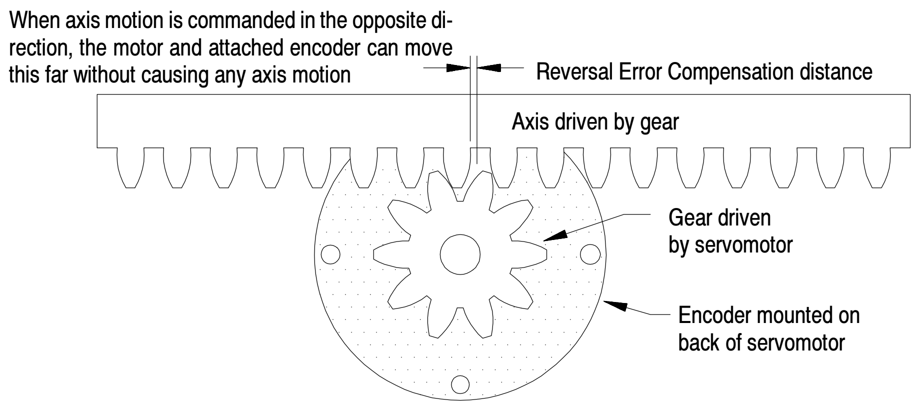
Important: Verify that the reversal error value entered here is not greater than the actual mechanical reversal error. If too large a value is entered, it interferes with proper axis motion.

Important: You can not use Reversal Error Compensation on axes equipped with linear scales. Because the position loop is closed on the slide mounted scale, there is no lost motion (or reversal error) with this feedback configuration.

The sign included with this parameter value is used only to establish the initial reversal error direction as described below.

If the value is entered as:	then it is added to the first move in this direction:	and, from that time on:
negative	positive	any time the axis reverses direction, the Reversal Error Compensation distance is added to the move distance
positive	negative	

Figure 7.10
Example of Reversal Error



Axis	Parameter Number
(1)	[1340]
(2)	[2340]
(3)	[3340]
(4)	[4340]
(5)	[5340]
(6)	[6340]
(7)	[7340]
(8)	[8340]

Range

-9.99999 to 9.99999 mm

or

-0.39366 to 0.39366 in.

Notes

This parameter must be set independently for each axis.

When activated, reversal error acts on all axis moves including manual jogging and homing.

SERCOS Users — if your SERCOS drive has reversal error compensation, do not set this parameter. Doing so will cause the control to compensate for reversal error when the drive is already doing so. 1394 Serial Drives systems should use this parameter for reversal error.

Excess Error

Function

This parameter places an upper limit on following error for a servo. When the magnitude of the following error exceeds the value entered here, the control goes into E-STOP. The error message EXCESS FOLLOWING ERROR appears on the message line of the control.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: This parameter plays a significant role in machine safety, as it can prevent damage and injury resulting from mechanical or electrical failure of a servo.

The smaller the value for this parameter, the sooner the control detects a servo problem and goes into E-Stop. However, too small a value here interferes with normal axis motion.

Important: To avoid getting a Warning: “SERVO AMP FE LIMITS CORRECTED” on power up of the control, you must ensure that the following relationship exists between these AMP values :
Inposition Band < Gain Break Point < Feedrate Suppression Point ≤ Excess Error

Axis	Parameter Number
(1)	[1750]
(2)	[2750]
(3)	[3750]
(4)	[4750]
(5)	[5750]
(6)	[6750]
(7)	[7750]
(8)	[8750]

Range

0 to 214.10000 mm or 0 to 8.42913 in. (linear axes)
0 to 1440.0000 degrees (closed-loop spindles)
0 to 214.00000 degrees (rotary axes)

Notes

This parameter must be set independently for each servo.

A typical value is 110% of the maximum following error expected during operation of an axis. This can be determined by the following equation:

```
                             (maximum rapid speed) (1.1)
Min. Excess Error =    -----------------------------------------
                       (Initial Gain) (1000) (Gain Break Ratio)
```

On closed-loop spindle applications, this parameter is only applied to the servo while the spindle is orienting or tapping.

Feedrate Suppression Point

Function

When the following error exceeds the value entered here, the control automatically reduces the axis feedrate by 50%. Axis feedrates remain reduced by 50% until the following error drops below the value entered here. At that time, the control increases the feedrate to the originally programmed speed.

Important: The value entered here must be less than or equal to the value entered for Excess Error, or a Warning: "SERVO AMP FE LIMITS CORRECTED"occurs.

Axis	Parameter Number
(1)	[1740]
(2)	[2740]
(3)	[3740]
(4)	[4740]
(5)	[5740]
(6)	[6740]
(7)	[7740]
(8)	[8740]

Range

0 to 214.10000 mm

or

0 to 8.42913 in.

Notes

Typically the value entered here is at least 10% less than the value entered for Excess Error. If desired, the Feedrate Suppression Point can be disabled by entering a value equal to the value entered for the Excess Error.

This parameter must be set independently for each axis.

Gain Break Point

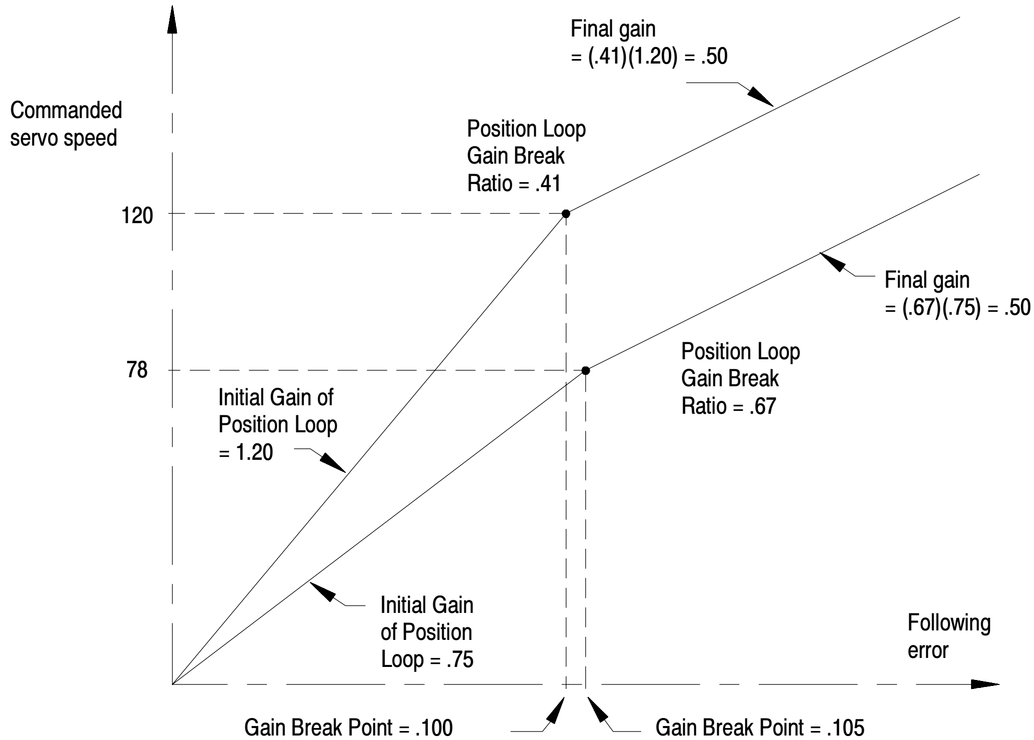
Function

Specifies the amount of following error at which gain is reduced from the Initial Gain value.

Following error increases with servo speed. When the following error exceeds the value entered for this parameter, the gain is reduced on the basis of the **Position Loop Gain Break Ratio**.

Important: The value entered here must be less than the value entered for **Feedrate Suppression Point**, or a Warning: "SERVO AMP FE LIMITS CORRECTED"occurs. This parameter is not used for a ZFE Closed-loop axis, but still must be less than the value entered for **Feedrate Suppression Point**.

Figure 7.11
Example of Interrelationship Between Gain Parameters



Axis	Parameter Number
(1)	[1730]
(2)	[2730]
(3)	[3730]
(4)	[4730]
(5)	[5730]
(6)	[6730]
(7)	[7730]
(8)	[8730]

Range

0 to 214.10000 mm or 0 to 8.42913 in. (linear axes)
0 to 720.00000 degrees (closed-loop spindles)
0 to 214.00000 degrees (rotary axes)

Notes

Higher gain is necessary for contouring accuracy. Reduced gain at higher feedrates helps reduce stress on the axis mechanism.

It is impossible to contour accurately above the gain break point, therefore, it is necessary for the Gain Break to occur at a feedrate above the maximum allowable cutting feedrate. A typical value for gain break point (also the minimum value) can be calculated by:

$$\text{Gain Break Point} = \frac{(\text{maximum contouring speed})\,(.001)}{(\text{Initial Gain of Position Loop})}$$

If the gain break point is below the maximum contouring speed, contours will be inaccurate.

On spindle applications, this parameter is only applied to the servo while the spindle is orienting or tapping.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system and may not be used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Inposition Band

Function

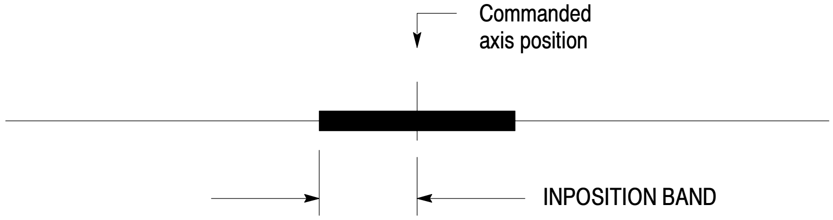
The value entered here sets the furthest distance the axis can be from its commanded destination to be considered in position when the control is in G00 (rapid mode), G09 (exact stop nonmodal), G61 (exact stop modal), or when homing.

This does not affect the final positioning accuracy of the system. It simply determines how close to the endpoint of a move the axis must be before the next move can begin.

Important: The value entered here must be less than the value entered for Gain Break Point or a Warning: “SERVO AMP FE LIMITS CORRECTED” occurs. However, if the value entered here is too small, inherent axis and feedback instability may prevent the axis from ever reaching the commanded position (including home position).

Important: In the event that you have a split axis, both servos must be within their inposition bands before the control determines the axis inposition.

Figure 7.12
Axis In-Position Band



Axis	Parameter Number
(1)	[1735]
(2)	[2735]
(3)	[3735]
(4)	[4735]
(5)	[5735]
(6)	[6735]
(7)	[7735]
(8)	[8735]

Range

0 to 214.10000 mm

or

0 to 8.42913 in.

Notes

This parameter must be set independently for each servo.

Initial Gain of Position Loop

Function

This parameter specifies the servo loop gain at speeds below the gain break speed.

Gain is a function of the drives, servo motors, and machine. For example, many machines are sized with drives and motors so that the axes can run at a gain of 1 (0.001 mm of following error for each 1 mmpm of feedrate) up to the machine's specified maximum cutting speed.

Typical values for the Initial Gain are between 0.5 and 2.0 (most CNC machines use a value of 1).

Smaller values produce a “loose” system (characterized by large, heavy, or underpowered axes). The benefits of smaller gains are improved control over surface finish and reduced stress on the machine.

Larger values produce a “tight” system (characterized by small, light, quick axes). The benefits of higher gains are faster response and increased rigidity.

The relationship between following error and system gain is shown in the following equation:

```
                  Feedrate (mmpm or ipm)
Gain  =   -------------------------------
          Following Error (0.001 mm or in.)
```

Gain is sometimes discussed in units of inverse milliseconds. It is often necessary to know the **Initial Gain of Position Loop** in units of inverse milliseconds when making comparisons to other parameters in this section. To represent the value entered here in inverse milliseconds:

```
                                           60
gain (in inverse milliseconds) = -----------------------
                                   Initial Gain Position Loop
```

Table 7.A
Examples of Initial Gain Values and the Resulting Following Error

Initial Gain	Following Error per mmpm of Velocity	Initial Gain in Inverse msec
0.25	0.0040 mm	240 invrs msec
0.50	0.0020 mm	120 invrs msec
1.00	0.0010 mm	60 invrs msec
2.00	0.0005 mm	30 invrs msec

Axis	Parameter Number
(1)	[1710]
(2)	[2710]
(3)	[3710]
(4)	[4710]
(5)	[5710]
(6)	[6710]
(7)	[7710]
(8)	[8710]

Range

0 to 30.00000

Notes

If the control is closing the velocity loop in addition to the position loop (loop type selected as digital), then the value of this parameter affects the value you enter for the parameter **Velocity Proportional Gain**. Refer to the parameter **Velocity Proportional Gain** for details on how the **Initial Gain of Position** impacts the **Velocity Proportional Gain**.

You cannot specify this parameter for spindles. Use gain/gear parameters [777] through [800] instead.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Position Loop Gain Break Ratio

Function

Specifies the ratio by which the control reduces position loop gain when the following error reaches the value entered for the **Gain Break Point** parameter. A value of one entered here results in no gain break. A value of .5 reduces the gain by half at gain break.

This parameter is not used for a ZFE closed-loop axis.

Axis	Parameter Number
(1)	[1720]
(2)	[2720]
(3)	[3720]
(4)	[4720]
(5)	[5720]
(6)	[6720]
(7)	[7720]
(8)	[8720]

Range

0 to 1.000000

Notes

Refer to Figure 7.11 on page 7-23 for examples.

This parameter must be set independently for each servo.

Feed Forward Percent

Function

This parameter is significant only if the **Servo Position Loop Type** is ZFE Closed Loop. It specifies the percentage that the following error is reduced as compared to simple closed-loop operation.

For example, if 80% is entered here, axis following error (after reaching the commanded feedrate) is approximately 20% of what it would have been if closed-loop was selected as the Servo Position Loop Type.

If you enter a value close to:	then:
0%	axis performance is nearly identical to that of a "Closed Loop" axis instead of a "ZFE Closed Loop" axis
100%	the result is near zero following error when the axis is moving at a constant velocity

Important: If high percentage values are entered here, careful consideration must be given to the Gain and acceleration/deceleration parameters to give the axis stability. Refer to chapter 10 for Acc/Dec parameter information.

ATTENTION: High percentage values entered for this parameter might impose greater stress on axis components. Mechanical discontinuities in the machine may be exaggerated. Life expectancy of some machines may be reduced.

Axis	Parameter Number
(1)	[1680]
(2)	[2680]
(3)	[3680]
(4)	[4680]
(5)	[5680]
(6)	[6680]
(7)	[7680]
(8)	[8680]

Range

0 to 100

Notes

This parameter must be set independently for each axis.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Position Loop Feedback Port

Function

This parameter is used to configure which port will be receiving position feedback.

You may choose to use optional feedback for position data. Optional feedback can be used for:

- a second feedback device to improve accuracy of a positioning axis
- spindle feedback
- analog servo feedback

Systems Using the CNC Serial Drive

There are eight feedback connectors, all of which can be used at one time. The first four connectors (J1 to J4) are reserved for the motor-mounted feedback devices. Motor-mounted feedback is required for the velocity feedback loop. This same motor-mounted feedback device may also be used for position feedback in which case both the position and velocity loop feedback port will be the same (J1 to J4). If you have an optional feedback device returning position feedback, it must be assigned to one of the additional AquadB ports on the system module (J9, J10, J11, or J12).

Axis	Parameter Number
(1)	[1530]
(2)	[2530]
(3)	[3530]
(4)	[4530]
(5)	[5530]
(6)	[6530]
(7)	[7530]
(8)	[8530]

Range

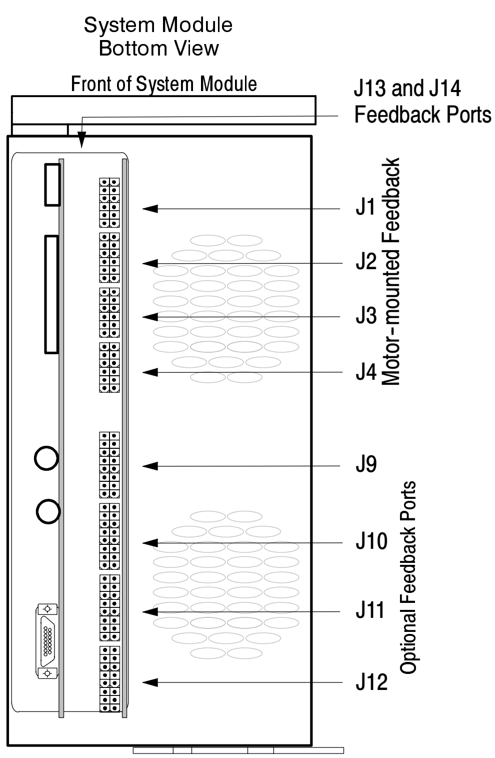
Selection	Result
(a)	No Feedback
(b)	Feedback Connector J1
(c)	Feedback Connector J2
(d)	Feedback Connector J3
(e)	Feedback Connector J4
(f)	Feedback Connector J9
(g)	Feedback Connector J10
(h)	Feedback Connector J11
(i)	Feedback Connector J12
(j)	Feedback Connector J13
(k)	Feedback Connector J14

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Figure 7.13
Connectors on the 1394 CNC Serial Drive



The number of optional feedback ports supported on your 1394 drive system is defined by options installed at the factory. Some ports may not be enabled. The following table shows catalog numbers and the feedback ports enabled by them:

catalog number	8520-2Q	8520-4Q
A quad B (with single, periodic, or distance-coded marker)	J9, J10	J11, J12

Position Feedback Type

Function

This parameter tells the control the type of position feedback device used for the axis.

- **No Feedback —** The servo has no position feedback.
- **INC Encoder A/B/Z (Z < A) —** for a narrow marker differential incremental encoder (sometimes called a "gated" encoder). Z < A refers to a narrow marker pulse. This means that there is only one period during which the A, B and Z channels can simultaneously be high. Select this feedback type if using Allen-Bradley bulletin 845H encoders. Refer to Figure 7.14 for examples.

  Digital systems only use this selection if using a position feedback device other than the encoder attached to the digital servomotors.
- **INC Encoder A/B/Z (Z > A)** — for a wide marker differential incremental encoder (sometimes called a "non-gated" encoder). Z > A refers to a wide marker pulse. This means that there is more than one period during which the A, B, and Z channels can simultaneously be high. Refer to Figure 7.14 for examples.

  Digital systems only use this selection if using a position feedback device other than the encoder attached to the digital servomotors.
- **A Quad B No Marker —** Use this if your feedback device has no marker (typically a glass slide type device). Be aware that this type of device cannot be homed and therefore cannot establish an absolute position (relative positioning only).
- **A Quad B with One Marker** — Use this if your linear feedback device (typically a glass slide type device) has a marker located near your home limit switch.
- **HIPERFACE Absolute** — Use this selection for 1326 motors attached to a 9/440HR system with absolute high-resolution feedback devices.

- **HIPERFACE Incremental** — Use this selection for 1326 motors attached to a 9/440HR system with incremental high-resolution feedback devices.
- **A Quad B with Distance-coded Marker** — This selection is only available for 9/440HR systems equipped for optional feedback on ports J9 to J12. This feedback device is equipped with multiple markers at progressively increasing distances apart along its length. This allows the CNC to identify absolute axis position whenever two consecutive markers are passed.

Important: Remember that for an axis to home using the encoder marker channel (Z channel), there must be an encoder position in which all three channels (A, B, and Z) are simultaneously high. For some encoders (such as the Allen-Bradley Bulletin 845H Encoders) it may be necessary to swap the B and the B-inverse channels to attain this.

Range

Selection	Result
(a)	No Feedback
(b)	INC Encoder - A/B/Z (Z < A)
(c)	INC Encoder - A/B/Z (Z > A)
(d)	A Quad B No Marker
(e)	A Quad B With One Marker
(f)	1326HR HIPERFACE Absolute (1326 digital systems only)
(g)	1326HR HIPERFACE Incremental (1326 digital systems only)
(h)	A Quad B Distance-coded Marker

Axis	Parameter Number
(1)	[1570]
(2)	[2570]
(3)	[3570]
(4)	[4570]
(5)	[5570]
(6)	[6570]
(7)	[7570]
(8)	[8570]

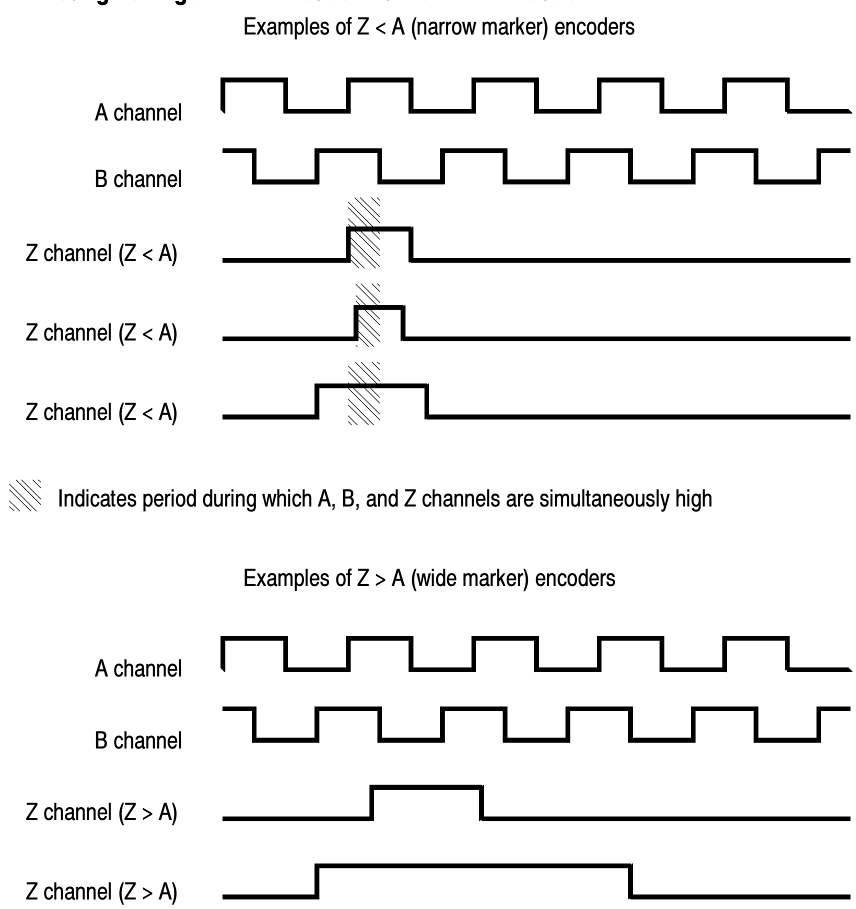
Notes

Important: An application note is available for integrating linear scales to the control. Contact your local Allen-Bradley representative.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Figure 7.14
Distinguishing a Z < A Encoder from a Z > A Encoder



Position Feedback Counts/Cycle

Function

Position Feedback Counts/Cycle specifies the number of counts that are produced by the encoder for each electrical cycle. For encoders with one marker, an electrical cycle is one revolution. For encoders with two markers, an electrical cycle is one half revolution.

Important: For single and no marker linear scales, enter the number of counts expected per lead screw revolution (pitch) to establish the proper number of counts per unit of travel. Use the parameters **Position Feedback Counts/Cycle** and **Lead Screw Thread Pitch** to enter the number of counts returned by the device per revolution of the lead screw. The ratio between **Teeth on Gear for Position FB** and the parameter **Teeth on Lead Screw for Position Feedback** should be one to one. For example if the linear device has 5000 lines per .5 inch enter 5000 as the **Position Feedback Counts/Cycle** and .5 inch as the **Lead Screw Thread Pitch**. Both **Teeth on Gear for Position Feedback** and **Teeth on Lead Screw for Position Feedback** can then be set to one.

Important: For linear scales with distance-coded markers, enter the number of counts for "Smaller Period for DCM Scales". For example, the Heidenhain LS176 with distance-coded markers provides 5 A quad B cycles per scale signal period

```
.02mM/5 = .004mM A quad B cycle
```

Each A quad B cycle produces 4 counts

```
.004/4 = .001 mM per count
```

Enter the number of Counts per Cycle

$$\frac{20mM/Cycle}{.001mM/Count} = 20,000Counts/Cycle$$

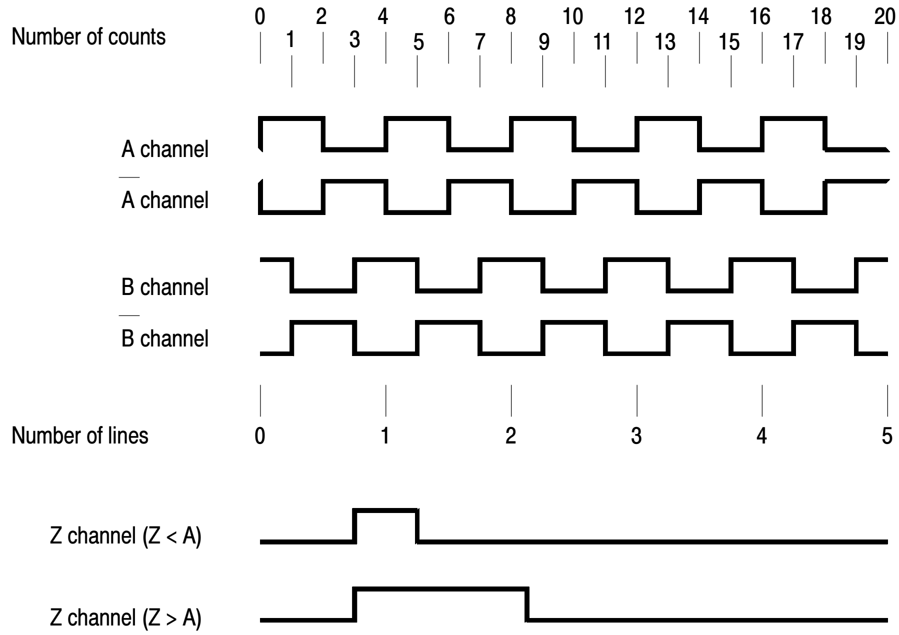
Important: Encoders on spindles must have one marker per revolution and must rotate on a 1:1 ratio with the face of the spindle.

Encoders are generally labeled with the number of **lines** they have per electrical cycle. Since encoders used with the control must have 2 feedback channels (A and B) positioned in quadrature, the number of **counts** produced is actually 4 times the number of **lines.**

```
Position Feedback Counts/Cycle = 4 x lines/elec. cycle
```

Important: If configuring an adaptive depth probe see page 30-6.

Figure 7.15
Encoder Input to the Control



Axis	Parameter Number
(1)	[1575]
(2)	[2575]
(3)	[3575]
(4)	[4575]
(5)	[5575]
(6)	[6575]
(7)	[7575]
(8)	[8575]

Range

4 to 4,194,304

High-resolution Systems	Cnts/Cycle
HR absolute	1,048,576
HR incremental	2,097,152

Important: The high-resolution feedback devices supplied with the 1326 motors that are compatible with the 1394 CNC serial drive, produce the number of counts per revolution that are indicated above. There is a limitation on the number of feedback counts that are supported for all conditions of 4,194,304 counts per inch. If this value is exceeded for your screw pitch/gear box combination your system may be held in E-Stop, causing an error message to be displayed.

Notes

This parameter must be set independently for each servo.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Note higher resolution feedback devices can exceed the controls maximum feedback frequency at higher speeds. Refer to your *9/PC CNC Software and Hardware Installation and Integration Manual* for more information.

Sign of Position Feedback

Function

This parameter allows the installer to reverse the polarity of the feedback without rewiring the encoder.

When the encoder is installed, feedback should count up as the axis is moved in the positive direction, and count down when the axis is moved in the negative direction. This is all relative to axis direction assignment, rotation direction changes through gearing and encoder phasing.

After encoder installation, if it is determined that the feedback is counting up when it should be counting down, or counting down when it should be counting up, it can be corrected simply by changing this parameter to "Minus."

ATTENTION: Axis runaway can result if this parameter is incorrect. Refer to the notes in this section for integration suggestions.

Axis	Parameter Number
(1)	[1595]
(2)	[2595]
(3)	[3595]
(4)	[4595]
(5)	[5595]
(6)	[6595]
(7)	[7595]
(8)	[8595]

Range

Selection	Result
(a)	Plus
(b)	Minus

Notes

This parameter must be set independently for each servo.

SERCOS Users — this parameter should remain set to Plus. Setting this parameter to Minus should only be necessary if your drive is misconfigured and returns negative position information when the axis is positive and positive position information when the axis is negative. Typically this should be corrected on the drive.

After installing and wiring the feedback device, put the control in E-Stop, and set the axis display to monitor feedback. Manually rotate the feedback device, noting axis direction and whether the feedback is counting negative or positive.

If the axis is moving in the direction defined as:	and if the feedback is counting:	then, for the Sign of Position Feedback, enter:
positive	up	PLUS
	down	MINUS
negative	down	PLUS
	up	MINUS

Teeth on Gear for Position Feedback

Function

If you are using the same physical device for both the velocity loop and position loops, then the value entered for this parameter should be the same as the value entered for the parameter **Teeth on Lead Screw for Vel FB**.

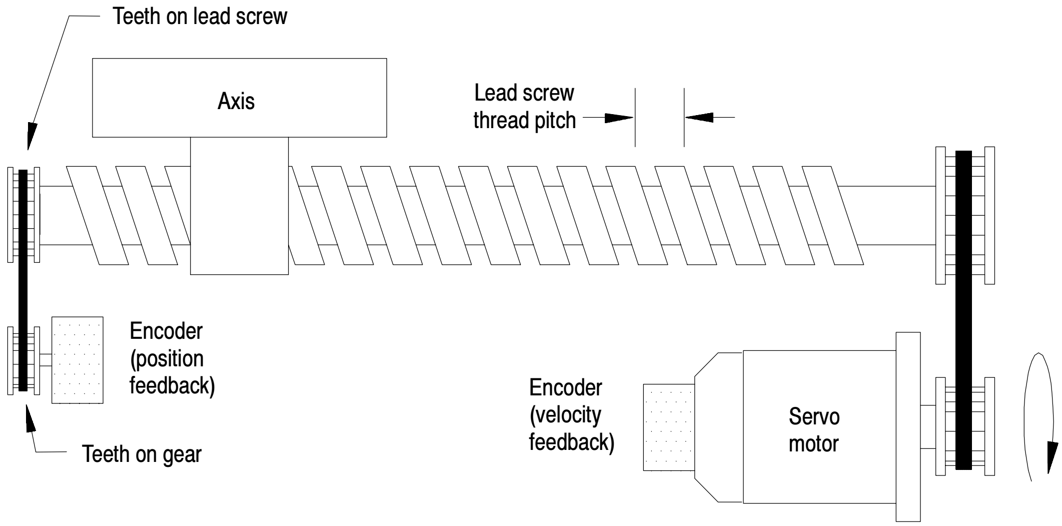
If you are using a nonmotor-mounted position feedback device (connected to a separate feedback port than the velocity feedback), enter the number of teeth on the gear or gearbelt pulley attached to the feedback device. This number is used in conjunction with the parameter **Teeth on Lead Screw for Pos FB** to calculate the gear ratio from the position feedback device to the lead screw (number of lead screw revolutions that occur between position feedback device revolutions). The control then uses this ratio combined with the entered **Lead Screw Thread Pitch** and **Position Feedback Counts/Cycle** to determine the number of feedback counts that occur per revolution of the lead screw.

Important: If configuring a linear position feedback device (excluding distance-coded marker systems) use the parameters **Position Feedback Counts/Cycle** and **Lead Screw Thread Pitch** to enter the number of counts returned by the device per revolution of the lead screw. The ratio between **Teeth on Gear for Position Feedback** and the parameter **Teeth on Lead Screw** should be one to one. For example if the linear device has 5000 lines per 1/2 inch enter 5000 as the **Position Feedback Counts/Cycle** and .5 inch as the **Lead Screw Thread Pitch**. Both **Teeth on Gear for Position Feedback** and **Teeth on Lead Screw for Position Feedback** can then be set to one.

In some cases, there may be a series of gears or gearbelt pulleys that make up a final gear ratio for a servo. In this case, it is necessary to factor this gear ratio into this parameter and the parameter **Teeth on Lead Screw for Pos FB**.

Important: The system automatically reduces the gear ratio to the lowest common denominator. For example, a ratio of 44:40 will be reduced to 11:10 on an AMP upload.

Figure 7.16
Typical Nonmotor-mounted Position Feedback Configuration



Axis	Parameter Number
(1)	[1026]
(2)	[2026]
(3)	[3026]
(4)	[4026]
(5)	[5026]
(6)	[6026]
(7)	[7026]
(8)	[8026]

Range

1 to 32767

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

When configuring any closed-loop axis you must configure two gear ratios for feedback; one for velocity feedback, the other for position feedback. On axes with a single feedback device, the setup and gear ratios should be identical for position and velocity feedback since the same device is being used for both feedback types.

This parameter must be set independently for each axis.

Teeth on Lead Screw for Position Feedback

Function

If you are using the same physical device for both the velocity loop and position loops, then the value entered for this parameter should be the same as the value entered for the parameter **Teeth on Motor Gear For Vel FB**.

If you are using a nonmotor-mounted position feedback device (connected to a different feedback port than the velocity feedback), enter the number of teeth on the gear or gearbelt pulley attached to the lead screw. This number is used in conjunction with the parameter **Teeth on Gear for Position FB** to calculate the gear ratio from the position feedback device to the lead screw (number of lead screw revolutions that occur between position feedback device revolutions). The control then uses this ratio combined with the entered **Lead Screw Thread Pitch** and **Position Feedback Counts/Cycle** to determine the number of feedback counts that occur per unit of axis travel.

Important: If configuring a linear position feedback device (excluding distance-coded marker systems) use the parameters **Position Feedback Counts/Cycle** and **Lead Screw Thread Pitch** to enter the number of counts returned by the device per revolution of the lead screw. The ratio between **Teeth on Gear for Position FB** and the parameter **Teeth on Lead Screw for Position Feedback** should be one to one. For example if the linear device has 5000 lines per 1/2 inch enter 5000 as the **Position Feedback Counts/Cycle** and .5 inch as the **Lead Screw Thread Pitch**. Both **Teeth on Gear for Position Feedback** and **Teeth on Lead Screw for Position Feedback** can then be set to one.

In some cases there may be a series of gears or gearbelt pulleys that make up a final gear ratio for a servo. In this case, it is necessary to factor this gear ratio into this parameter and the parameter **Teeth on Gear for Pos FB**.

Important: The system automatically reduces the gear ratio to the lowest common denominator. For example, a ratio of 44:40 will be reduced to 11:10 on an AMP upload.

Refer to Figure 7.16 for an illustration of typical gearing for position feedback devices.

Axis	Parameter Number
(1)	[1027]
(2)	[2027]
(3)	[3027]
(4)	[4027]
(5)	[5027]
(6)	[6027]
(7)	[7027]
(8)	[8027]

Range

1 to 32767

Notes

SERCOS Users — this parameter is only used when configuring 1394 Serial Drives. It is not used for third-party SERCOS drives.

When configuring any closed-loop axis you must configure two gear ratios for feedback; one for velocity feedback, the other for position feedback. On axes with a single feedback device, the setup and gear ratios should be identical for position and velocity feedback since the same device is being used for both feedback types.

This parameter must be set independently for each servo.

Analog Servo Pos. Voltage

Function

This parameter is only available for analog servo hardware.

This parameter is used to scale the signal sent from the analog servo interface to the servo amplifier. This signal may range from -10.0000 to +10.0000 volts. The lower portion of this range is set with the parameter **Analog Servo Neg. Voltage**. This value must be entered accurately. Use values attained from drive specifications or testing. This value is used for calculation of the servo motors continuous rated current which is used later as a percent of this parameter. If you feel you must limit the analog output voltage to protect your machine, use the parameter Max Rated Torque.

If the servo loop type is configured as:	enter the positive voltage that the servo module must send to the amplifier to:
position loop	maintain rapid axis speed in the positive direction The value entered for this parameter should be below 10 volts. If 10 volts is entered, the control is unable to compensate for any increase in axis load that may occur at rapid speed.

Axis	Parameter Number
(1)	[1019]
(2)	[2019]
(3)	[3019]
(4)	[4019]
(5)	[5019]
(6)	[6019]
(7)	[7019]
(8)	[8019]

Range

-10.0000 to +10.0000 volts dc

Notes

This parameter must be set independently for each servo.

Important: This parameter should have a value that is greater than that of the parameter **Analog Servo Neg. Voltage**.

An example of this parameter would be if a 15 amp drive is combined with a servo motor rated at 9 amps peak current, the servo card should probably never exceed 6 volts to get the drive to output 9 amps to the motor. You would set this parameter to 6 volts. The following equation is not valid for servo systems that use the CNC serial drive to close the velocity loop.

$$\frac{\text{10 V (maximum servo card output) x 9 A (maximum rated motor current)}}{\text{15 A (Drive output at 10V)}} = \text{6 V (Analog Servo Positive Voltage)}$$

Analog Servo Neg. Voltage

Function

This parameter is only available for analog servo hardware.

Use this parameter to scale the signal sent from the analog servo interface to the servo amplifier. This signal may range from -10.0000 to +10.0000 volts. The upper portion of this range is set with the parameter Analog Servo Pos. Voltage. This value must be entered accurately. Use values attained from drive specifications or testing. This value is used for calculation of the servo motors continuous rated current which is used later as a percent of this parameter. If you feel you must limit the analog output voltage to protect your machine, use the parameter Max Rated Torque.

If the servo loop type is configured as:	enter the negative voltage that the servo module must send to the amplifier to:
position loop	maintain rapid axis speed in the negative direction This parameter should be above -10V. If -10V is entered, the control is unable to compensate for any incontinuities that may occur at rapid speed.

Axis	Parameter Number
(1)	[1024]
(2)	[2024]
(3)	[3024]
(4)	[4024]
(5)	[5024]
(6)	[6024]
(7)	[7024]
(8)	[8024]

Range

-10.0000 to +10.0000 volts dc

Notes

This parameter must be set independently for each servo.

Important: This parameter should have a value that is less than that of the parameter Analog Servo Pos. Voltage.

An example of this parameter would be if a 15A drive is combined with a servo motor rated at 9A peak current, the servo card should probably never exceed -6V to get the drive to output 9 amps to the motor. You would set this parameter to -6V. The following equation is not valid for servo systems that use the CNC serial drive to close the velocity loop.

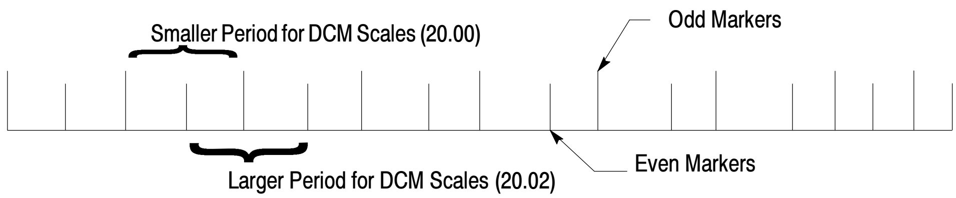
$$\frac{-10\text{ V (maximum servo card output)} \times 9\text{ A (maximum rated motor current)}}{15\text{ A (Drive output at 10V)}} = -6\text{ V (Analog Servo Positive Voltage)}$$

Smaller Period for DCM Scales

Function

This parameter is only used on systems that use a linear scale with distance-coded markers (DCMs). See Appendix B for details on integrating a DCM scale.

The DCM supported by the 9/PC must have equally spaced odd and even markers (i.e., the distance between each odd marker must be the same and the distance between each even marker must be the same). For this parameter, enter the smaller of these two distances (typically the distance between odd markers).



Axis	Parameter Number
(1)	[1500]
(2)	[2500]
(3)	[3500]
(4)	[4500]
(5)	[5500]
(6)	[6500]
(7)	[7500]
(8)	[8500]

Range

.00001 to 1000 mm
.00000 to 39.37008 in.

Notes

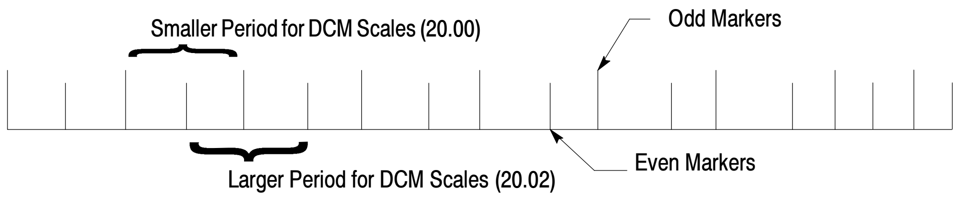
This parameter must be set independently for each servo.

Larger Period for DCM Scales

Function

This parameter is only available for systems that use a linear scale with distance-coded markers (DCMs). Refer to Appendix B for details on integrating a DCM scale.

The DCM supported by the CNC serial drive must have equally spaced odd and even markers (i.e., the distance between each odd marker must be the same and the distance between each even marker must be the same). For this parameter, enter the larger of these two distances (typically the distance between even markers).



Axis	Parameter Number
(1)	[1501]
(2)	[2501]
(3)	[3501]
(4)	[4501]
(5)	[5501]
(6)	[6501]
(7)	[7501]
(8)	[8501]

Range

.00001 to 1000 mm
.00000 to 39.37008 in.

Notes

This parameter must be set independently for each servo.

Velocity Loop Parameters

The servo parameters in the sections that follow are available to configure the velocity loop. The velocity loop is only configured for digital systems.

Important: Digital systems must use the motor-mounted feedback device (that comes preinstalled in the 1326 digital motors) for velocity feedback. This motor-mounted feedback must not be removed from the motor or uncoupled from the motor shaft. Precisely oriented motor-mounted feedback is necessary for accurate motor commutation on digital systems.

Velocity Loop Feedback Port

Function

Select the port connector number through which velocity feedback is sent to the control. You may select the same port as used for **Position Loop Feedback Port** if you do not have a separate position feedback device.

The standard servo feedback configuration uses one feedback device for both positioning and velocity data. If this is the case on your system, this parameter must be set to the same value as the parameter **Position Loop Feedback Port**. Refer to this chapter's overview for an example of a system that uses two feedback devices for the same axis and may require a different port be selected for feedback.

Refer to page 7-30 for connector locations.

CNC Serial Drive Systems
1394 CNC serial drive systems must use the motor-mounted feedback device for velocity feedback (ports J1 to J4). This device MUST be used for velocity feedback. This motor-mounted feedback device can also be used for position feedback or you may use an auxiliary feedback device connected to one of the optional feedback ports .

Axis	Parameter Number
(1)	[1020]
(2)	[2020]
(3)	[3020]
(4)	[4020]
(5)	[5020]
(6)	[6020]
(7)	[7020]
(8)	[8020]

Range

Selection	Result
(a)	No Feedback
(b)	Feedback Connector J1
(c)	Feedback Connector J2
(d)	Feedback Connector J3
(e)	Feedback Connector J4

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Velocity Feedback Type

Function

The standard servo feedback configuration uses one feedback device for both positioning and velocity data. If this is the case on your system, this parameter must be set to the same value as the parameter **Position Feedback Type**. Refer to this chapter's overview for an example of a system that uses two feedback devices for the same servo and may require a different feedback type to be selected here.

Though the analog servo interface does not support a separate feedback device for the velocity loop, you must still configure these velocity loop parameters including **Velocity Loop Feedback Port**, **Velocity Loop Feedback Type**, **Velocity Feedback Counts/Cycle**, and **Sign of Velocity Feedback**. These parameters must be set to the same values as their positioning loop counterparts.

Digital systems must use the motor-mounted feedback device (that comes preinstalled on the 1326 digital motors) for velocity feedback. This high-resolution device can not be removed from the motor or realigned with the motor shaft. This device is used for motor commutation.

This parameter tells the control the type of velocity feedback device used for the axis (typically the same feedback device selected with the parameter **Position Feedback Type**). These options are:

- **No Feedback —** only selected if the velocity loop is closed by an external device (typically the drive for analog systems). This option is not valid for digital systems. Digital motors must always close the velocity loop through the CNC with the encoder/resolver device attached directly to the motor shaft.
- **INC Encoder A/B/Z (Z < A) —** for axes that have a narrow marker differential incremental encoder (sometimes called a "gated" encoder). Z < A refers to a narrow marker pulse. This means that there is only one period during which the A, B and Z channels can simultaneously be high. Select this feedback type if using Allen-Bradley bulletin 845H encoders. Refer to page 7-33 Figure 7.14 for examples.

  Digital systems cannot use this selection.
- **INC Encoder A/B/Z (Z > A)** — for axes that have a wide marker differential incremental encoder (sometimes called a "non-gated" encoder). Z > A refers to a wide marker pulse. This means that there is more than one period during which the A, B, and Z channels can simultaneously be high. Refer to page 7-33, Figure 7.14 for examples.
- **HIPERFACE Absolute** — for 1326 motors attached to a CNC serial drive system with absolute high-resolution feedback devices. The maximum number of feedback counts/cycle for this selection is 1,048,576.
- **HIPERFACE Incremental** — for 1326 motors attached to a CNC serial drive system with incremental high-resolution feedback devices. The maximum number of feedback counts/cycle for this selection is 2,097,152.

Axis	Parameter Number
(1)	[1560]
(2)	[2560]
(3)	[3560]
(4)	[4560]
(5)	[5560]
(6)	[6560]
(7)	[7560]
(8)	[8560]

Range

Selection	Result
(a)	No Feedback
(b)	INC Encoder - A/B/Z (Z < A)
(c)	INC Encoder - A/B/Z (Z > A)
(d)	HIPERFACE Absolute (1326 digital systems only)
(e)	HIPERFACE Incremental (1326 digital systems only)

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each axis.

Velocity Feedback Counts/Cycle

Function

Important: The standard servo feedback configuration uses one feedback device for both positioning and velocity data. If this is the case on your system, this parameter must be set to the same value as the parameter **Position Feedback Counts/Cycle**. Refer to this chapter's overview for an example of a system that uses two feedback devices for the same axis and may require a different value be selected here.

Important: Though the analog servo interface does not support a separate feedback device for the velocity loop, you must still configure these velocity loop parameters including **Velocity Loop Feedback Port**, **Velocity Loop Feedback Type**, **Velocity Feedback Counts/Cycle**, and **Sign of Velocity Feedback**. These parameters must be set to the same values as their positioning loop counterparts.

Velocity Feedback Counts/Cycle specifies the number of counts that are produced by the velocity encoder for each electrical cycle (usually the same number of counts as set with the parameter **Position Feedback Counts/Cycle** as they are typically the same encoder). For encoders with one marker, an electrical cycle is one revolution. For encoders with two markers, an electrical cycle is one half revolution.

Important: Spindle encoders must have only one marker per revolution.

Encoders are generally labeled with the number of **lines** they have per electrical cycle. Since encoders used with the control must have two feedback channels (A and B) positioned in quadrature, the number of **counts** produced is actually 4 times the number of **lines.**

```
Velocity Feedback Counts/Cycle = counts/elec. cycle = 4 x lines/elec.
cycle
```

Axis	Parameter Number
(1)	[1565]
(2)	[2565]
(3)	[3565]
(4)	[4565]
(5)	[5565]
(6)	[6565]
(7)	[7565]
(8)	[8565]

Range

4 to 4,194,304

Notes

This parameter must be set independently for each servo.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Sign of Velocity Feedback

Function

Important: The standard servo feedback configuration uses one feedback device for both positioning and velocity data. If this is the case on your system, this parameter must be set to the same value as the parameter **Sign of Position Feedback**. Refer to this chapter's overview for an example of a system that uses two feedback devices for the same axis and may require a different value be selected here.

This parameter allows the installer to reverse the polarity of the velocity feedback without rewiring the encoder (usually this is the same sign as selected with the parameter **Sign of Position Feedback** as it is typically the same encoder).

If a separate feedback device (other than the encoder used for positioning data) is used to derive velocity data, the sign of this parameter may be different from the **Sign of Position Feedback**. When the encoder, tachometer, or other feedback device is installed, the velocity feedback may count up when it should count down. This is all relative to the desired motor rotation direction as affected by gearing and encoder wiring.

After installation of a separate velocity feedback device, it is determined that this velocity feedback is counting up when it should be counting down, or counting down when it should be counting up, it can be corrected simply by changing this parameter to "Minus." When the sign of this parameter is incorrect the control may enter E-Stop when axis motion takes place.

ATTENTION: Motor runaway can result if this parameter is incorrect.

Axis	Parameter Number
(1)	[1600]
(2)	[2600]
(3)	[3600]
(4)	[4600]
(5)	[5600]
(6)	[6600]
(7)	[7600]
(8)	[8600]

Range

Selection	Result
(a)	Plus
(b)	Minus

Notes

This parameter must be set independently for each servo.

When configuring any closed-loop axis you must configure two gear ratios for feedback; one for velocity feedback, the other for position feedback. On axes with a single feedback device, the setup and gear ratios should be identical for position and velocity feedback since the same device is being used for both feedback types.

SERCOS Users — this parameter should remain set to Plus. Setting this parameter to Minus should only be necessary if your drive is misconfigured and returns negative velocity information when the axis is moving positive and positive velocity information when the axis is moving negative. Typically this should be corrected on the drive.

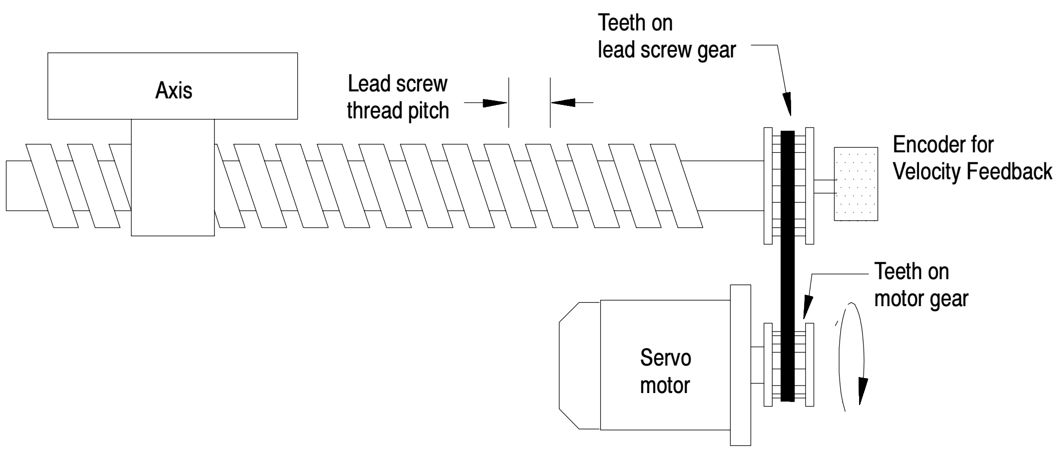
Teeth on Motor Gear for Velocity Feedback

Function

Use this parameter and the parameter **Teeth on Lead Screw for Vel FB** to tell the control the gear ratio between the motor and velocity feedback device. Typically the velocity feedback device is motor mounted. Regardless of where the velocity feedback is mounted, use this parameter in conjunction with Teeth on Lead Screw for Velocity Feedback to identify the number of feedback device rotations that occur per revolution of the ball screw. If the same feedback device is used for position feedback the value entered here must be the same as the value entered for “Teeth on Motor Gear for Position Feedback”.

In some cases, there may be a series of gears or gearbelt pulleys that make up a final gear ratio for velocity feedback. In this case, it is necessary to factor this gear ratio into this parameter and the parameter **Teeth on Lead Screw for Vel FB**.

Figure 7.17
Mechanical Considerations for Teeth and Pitch Parameters



Axis	Parameter Number
(1)	[1013]
(2)	[2013]
(3)	[3013]
(4)	[4013]
(5)	[5013]
(6)	[6013]
(7)	[7013]
(8)	[8013]

Range

1 to 32767

Notes

This parameter must be set independently for each servo.

When configuring any closed-loop axis you must configure two gear ratios for feedback; one for velocity feedback, the other for position feedback. On axes with a single feedback device, the setup and gear ratios should be identical for position and velocity feedback since the same device is being used for both feedback types.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Teeth on Lead Screw for Velocity Feedback

Function

Use this parameter and the parameter **Teeth on Motor Gear for Vel FB** to tell the control the gear ratio between the motor and velocity feedback device. Typically the velocity feedback device is motor mounted. Regardless of where the velocity feedback is mounted, use this parameter in conjunction with Teeth on Motor Gear for Velocity Feedback to identify the number of feedback device rotations that occur per revolution of the ball screw. If the same feedback device is used for position feedback the value entered here must be the same as the value entered for "Teeth on Lead Screw for Position Feedback".

In some cases there may be a series of gears or gearbelt pulleys that make up a final gear ratio. In this case, it is necessary to factor this gear ratio into this parameter and the parameter **Teeth on Motor Gear for Vel FB**.

Refer to Figure 7.17 for an illustration of the mechanical configuration of a typical axis.

Axis	Parameter Number
(1)	[1018]
(2)	[2018]
(3)	[3018]
(4)	[4018]
(5)	[5018]
(6)	[6018]
(7)	[7018]
(8)	[8018]

Range

1 to 32767

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Velocity Proportional Gain

Function

This is also a scaling parameter used to create a portion of the acceleration command for a particular servomotor.

In the control, **Velocity Proportional Gain** is multiplied by the velocity error. **Velocity Integral Gain** is multiplied by the summation of velocity error. These two products are added to form the acceleration command. (Velocity error is the difference between the velocity command and the actual velocity as derived from the feedback device.)

Generally, if set too high, the servo is underdampened and oscillation results. If set too low, the servo is overdampened and requires too much transition time between servo speeds.

For rapid positioning, with the **Positioning Acc/Dec Mode** as exponential, these parameters may affect underdampening/overdampening so that the **Programmed Delay Constant** will have to be adjusted to achieve proper machine performance.

```
(Init. Gain Pos. Loop)(3277) ≤ (Vel. Proportional Gain) ≤ (Init. Gain Pos.
Loop)(10923)
```

Important: On digital systems, if you use a standard motor, a value for this parameter is entered automatically by the control when AMP is downloaded or when the control is powered-up. When parameter **Standard Motor Table Values** is set to Yes, then the control ignores the value entered for this parameter and uses values as shown in Table 7.B.

Important: Once you have powered up your system and confirmed wiring, fine tuning of the servo gains is almost always required once the servo shaft is coupled to a load. Using online AMP to adjust the velocity/position loop gains will automatically reset the **Standard Motor Table Values** parameter to "No" at the control. Once you finish tuning your servos we recommend uploading the AMP from the control to store the new gain values.

Important: The velocity integral gain can be adjusted online using online AMP capabilities of the control. Once this is adjusted, online AMP sets the **Standard Motor Table Values** parameter to "No".

The following tables are provided for reference only.

Table 7.B
Velocity Proportional Gain Values for Standard Motors

MOTOR:LOAD INERTIA RATIO	1326 MOTOR TYPES								
	1326AB B410G B410J	1326AB B420E B420H	1326AB B430E B430G B740E	1326AB B515E B515G	1326AB B520E	1326AB B520F	1326AB B530E	1326AB B720E	1326AB B730E
1 : 0	1000	2000	3000	7500	8738	6000	10000	3600	22000
1 : 1	2000	4000	6000	15000	17460	12000	20000	7200	44000
1 : 2	3000	6000	9000	22500	26214	18000	30000	10800	66000
1 : 3	4000	8000	12000	30000	34952	24000	40000	14400	88000

Generally, if set too high, the axis is underdampened and oscillation results. If set too low, the axis is overdampened and requires too much time to reach position.

Axis	Parameter Number
(1)	[1801]
(2)	[2801]
(3)	[3801]
(4)	[4801
(5)	[5801]
(6)	[6801]
(7)	[7801]
(8)	[8801]

Range

0 to 65535

Notes

This parameter must be set independently for each servo.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

For assistance on determining the optimal velocity proportional gain amount refer to the tuning procedure in Appendix A of this manual.

Velocity Integral Gain

Function

This is a scaling parameter used to create a portion of the acceleration command for a particular servomotor.

In the control, **Velocity Proportional Gain** is multiplied by the velocity error. **Velocity Integral Gain** is multiplied by the summation of velocity error. These two products are added to form the acceleration command. (Velocity error is the difference between the velocity command and the actual velocity as derived from the feedback device.

Generally, if set too high, the servo is underdampened and oscillation results. If set too low, the servo is overdampened and requires too much time to reach speed.

Often the value of this parameter (in inverse milliseconds) is larger than the **Velocity Proportional Gain** (in inverse milliseconds) and smaller than the **Initial Gain of Position Loop** (in inverse milliseconds). It is unusual that an application would require an inverse **Velocity Integral Gain** value equal to or greater than the inverse **Initial Gain of Position Loop**. Note the following equation has constants that convert the parameter values you enter into units of inverse milliseconds.

```
0  <  (Vel. Integral Gain)  <  (Init. Gain Pos. Loop)(1092)
```

Important: On digital systems, if you use a standard motor, the correct value for this parameter is entered automatically by the control when AMP is downloaded or when the control is powered-up. When parameter **Standard Motor Table Values** is set to “Yes”, then the control ignores the value entered for this parameter and uses values as shown in Table 7.C.

Important: The velocity integral gain can be adjusted online using online AMP capability of the control. However, if the parameter **Standard Motor Table Values** is set to “Yes”, then the velocity integral gain is automatically overwritten by the control upon power-up and when AMP is downloaded.

The following tables are provided for reference only.

Table 7.C
Velocity Integral Gain Values for Standard Motors

MOTOR:LOAD INERTIA RATIO	1326 MOTOR TYPES							
	1326AB B410G B410J	1326AB B420E B520H B730E	1326AB B430E B430G	1326AB B515E B515G	1326AB B520E B520F	1326AB B530E	1326AB B720E	1326AB B740E
1 : 0	70	100	150	300	400	500	30	20
1 : 1	140	200	300	600	800	1000	60	40
1 : 2	210	300	450	900	1200	1500	90	60
1 : 3	280	400	600	1200	1600	2000	120	80

If a nonstandard motor is to be used, the initial value for this parameter must be determined through extensive motor testing. For assistance call Allen-Bradley Commercial Engineering at (440) 646-5000.

Generally, if set too high, the axis is underdampened and oscillation results. If set too low, the axis is overdampened and requires too much time to reach speed.

Axis	Parameter Number
(1)	[1800]
(2)	[2800]
(3)	[3800]
(4)	[4800
(5)	[5800]
(6)	[6800]
(7)	[7800]
(8)	[8800]

Range

33 to 32768 (8500 digital systems)
0 to 65535 (other systems)

Notes

This parameter must be set independently for each servo.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

For assistance on determining the optimal velocity proportional gain amount refer to the tuning procedure in Appendix A of this manual.

VE Integrator Discharge Rate

Function

During steady state the VE (Velocity Error) integrator compensates for external forces working on the motor (such as increase or decrease in loads). The VE integrator also plays a role in Acceleration/Deceleration compensating for any error during the velocity transition.

When the axis is accelerating or decelerating, the velocity error integrator compensates for any non-linear response of the servo. Any lag or lead in the servo response is accumulated by the VE integrator and returned in the form of an increased (or decreased) acceleration command.

Typically this accumulated error is returned to the system towards the end of the velocity ramp with little chance of positioning error. However, in some cases, (such as when the servo is coming to a stop while under a heavy load) the integrator may not discharge itself completely before the end of the velocity transition. When this happens, the VE integrator forces a velocity overshoot, and possibly a position overshoot. (This is even more likely when the type of position loop is chosen as “ZFE Closed Loop.”)

This parameter sets the rate at which the VE integrator discharges. A value of 1 indicates that the integrator cancels the error at the same rate it is accumulated. Setting this parameter to a value larger than 1 causes the integrator to cancel the accumulated error faster than it was accumulated. This can improve the performance of the integrator during velocity transitions, though having little to no effect on steady state behavior.

Important: Before adjusting this parameter, both the **Velocity Proportional Gain** and the **Velocity Integral Gain** should have been adjusted to attain the best possible servo performance. This should be done with the default value of one set for the **VE integrator discharge rate**. Once the servo is running properly, test for improved performance by adjusting this parameter.

To adjust this parameter, a small part program should be made using a series of:

- G01 moves making at least two noticeable velocity transitions

  Note that the G01 moves should take place at cutting feedrates.
- G00 moves making at least two noticeable velocity transitions

When this program is executed with the position loop type set to “ZFE Closed Loop” and feed forward set to 100%, changing the **VE Integrator Discharge Rate** has noticeable results during the velocity transitions (especially if a strip chart recorder is used). If the position loop type is set to “Closed Loop,” the effects are more difficult to observe.

Continue increasing the value of the **VE Integrator Discharge Rate**, executing this program after each change. The transitions and steady state movements should remain smooth and stable. When the **VE Integrator Discharge Rate** is set too high, velocity transitions are no longer smooth (steady state conditions most likely are not affected).

Important: If the velocity transitions are not smooth with a value of one set for the **VE Integrator Discharge Rate**, the initial settings for the **Velocity Proportional Gain** and/or **Velocity Integral Gain** are most likely incorrect.

Axis	Parameter Number
(1)	[1023]
(2)	[2023]
(3)	[3023]
(4)	[4023]
(5)	[5023]
(6)	[6023]
(7)	[7023]
(8)	[8023]

Range

1 to 8

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

This parameter must always be entered as an integer value.

Peak Current as a % of RMS

Function

Analog Systems

This parameter is used to specify the servo motors nominal rated current for continuous duty. This parameter is entered as a percentage of the parameter Analog Servo Pos. Voltage or Analog Servo Neg. Voltage (whichever is smaller). Calculate what servo card voltage will yield an amplifier output equal to your motors rated continuous duty current as follows:

$$\frac{\text{(Continuous duty rated motor current}}{\text{(Amplifier output current , A/V)}} = \text{(servo card voltage for continuous duty)}$$

Use the servo card voltage calculated above and determine what percentage it is of the servo card's maximum voltage as set with the parameter Analog Servo Pos. Voltage or Analog Servo Neg. Voltage (whichever is smaller).

$$\frac{\text{(Servo Card Voltage for Max Current)}}{\text{(Servo Card Voltage for Continuous Duty)}} \times 100 = \text{(Value of Peak Current as a \% of RMS)}$$

1394 Drives

Your Allen-Bradley 1394 digital amplifier requires this parameter be set at 300%. Your system may never reach this 300% peak current as it is ultimately limited by your selection of 1394 axis module(s) and 1326 motor(s).

Axis	Parameter Number
(1)	[1660]
(2)	[2660]
(3)	[3660]
(4)	[4660]
(5)	[5660]
(6)	[6660]
(7)	[7660]
(8)	[8660]

Range

Selection	Result
(a)	200%
(b)	300%

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each axis.

Maximum % Rated Torque (-)

Function

Enter the peak torque (as a percentage of the servo's rated continuous current) to be allowed for attaining and maintaining position in the negative travel direction.

Since motor torque is proportional to motor current, the following equation can be used to determine a value to be entered here:

```
 Maximum desired motor current
------------------------------  x 100%
 Motor's maximum rated current
```

Important: This parameter is only used if the velocity loop is closed in by the 1394 drive .

We recommend that the default value of 200% be left here if using a standard 1326 motor with 1394 drive (**Standard Motor Values** is set to "Yes"). When using the standard motor table settings, the 9/PC will override the value you enter here and use a value appropriate for your drive/motor combination.

Axis	Parameter Number
(1)	[1012]
(2)	[2012]
(3)	[3012]
(4)	[4012]
(5)	[5012]
(6)	[6012]
(7)	[7012]
(8)	[8012]

Range

0 to 300%

Notes

This parameter must be set independently for each axis.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This peak torque determines the maximum force an axis can produce during acceleration or deceleration in the negative direction.

Maximum Percent Rated Torque (+)

Function

Enter the peak torque (as a percentage of rated torque) to be allowed for attaining and maintaining position in the positive travel direction.

Since motor torque is proportional to motor current, the following equation can be used to determine a value to be entered here:

```
 Maximum desired motor current
-------------------------------  x 100%
 Motor's maximum rated current
```

Important: This parameter is only used if the velocity loop is closed in by the 1394 drive.

Axis	Parameter Number
(1)	[1670]
(2)	[2670]
(3)	[3670]
(4)	[4670]
(5)	[5670]
(6)	[6670]
(7)	[7670]
(8)	[8670]

Range

0 to 300%

Notes

This parameter must be set independently for each axis.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This peak torque determines the maximum force an axis can produce during acceleration or deceleration in the positive direction.

Torque Offset Percentage

Function

This is used to correct an axis imbalance caused by a constant force applied in one direction on an axis that is not balanced by a force applied in the opposite direction. An example would be a heavy vertical axis subject to gravity load in the downward direction.

Determine what percentage of the motor's maximum torque must be applied in one direction to offset axis loading in the opposite direction. Note that the maximum motor torque is represented by the AMP parameter **Maximum Servo Acceleration**.

For example, entering 1% here when the parameter **Torque Offset Direction** is set to Plus, increases the applied torque in the positive direction and decreases it in the negative direction. If commanded to accelerate in the positive direction at a rate that would normally require 3% of the motor's maximum torque, the control actually provides current for 4%. If the same motion command were executed in the negative direction, the control provides current for 2% of the motor's
maximum torque.

In this example, even when the axis is stationary, the control provides current for 1% of the motor's maximum torque in the positive direction.

Axis	Parameter Number
(1)	[1690]
(2)	[2690]
(3)	[3690]
(4)	[4690]
(5)	[5690]
(6)	[6690]
(7)	[7690]
(8)	[8690]

Range

0 to 100%

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Torque Off set Direction

Function

Enter the direction in which the **Torque Offset Percentage** is to be applied.

For example, if a heavy vertical axis is being pulled in its negative direction by gravity, enter Plus for this parameter. This increases the applied torque in the positive direction by the amount entered for the **Torque Offset Percentage** parameter.

Axis	Parameter Number
(1)	[1700]
(2)	[2700]
(3)	[3700]
(4)	[4700]
(5)	[5700]
(6)	[6700]
(7)	[7700]
(8)	[8700]

Range

Selection	Result
(a)	Plus
(b)	Minus

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Hard Stop Detection Torque

Function

Feed-to-Hard-Stop is only available on mill control types.

Use the Feed-to Hard-Stop feature when an axis is fed up to a fixed mechanical stop that physically halts axis travel. This parameter configures the torque that the servo must exert before the control identifies a hard stop. Enter this torque value as a percentage of the servo's maximum rated torque. Refer to your *9/PC CNC Operation and Programming Manual* for details on programming an axis against a hard stop using G24.

Axis	Parameter Number
(1)	[1151]
(2)	[2151]
(3)	[3151]
(4)	[4151]
(5)	[5151]
(6)	[6151]
(7)	[7151]
(8)	[8151]

Range

1 to 300%
(a percentage of motor rated continuous duty current)

Notes

In order to perform feed to hard stop, you must set the **Servo Loop Type** parameter to digital.

You must properly configure the parameter **Maximum Rated Motor Torque** before attempting to use feed to hard stop.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. The feed to hard stop feature is not compatible with any third-party SERCOS drive systems.

This parameter must be set independently for each axis that uses the feed to hard stop feature.

Hard Stop Holding Torque

Function

Feed to hard stop is only available on mill control types.

Use this parameter to have the servo for an axis that has reached a hard stop maintain a holding torque to keep constant pressure against the hard stop. If all motor torque was removed from an axis that has reached a hard stop some bounce could occur allowing the axis to move away from the mechanical stop. Also once a motor reaches the hard stop it would be impossible for the servo to respond quick enough, without a holding torque, to prevent any instantaneous forces from momentarily forcing the axis away from the hard stop.

This parameter specifies the torque that the servo generates to maintain force against a hard stop once it has been reached. Enter this torque value as a percentage of the servo's maximum rated torque.

Axis	Parameter Number
(1)	[1150]
(2)	[2150]
(3)	[3150]
(4)	[4150]
(5)	[5150]
(6)	[6150]
(7)	[7150]
(8)	[8150]

Range

1 to 100% of motor rated current (continuous duty)

Notes

In order to perform feed to hard stop you must set the **Servo Loop Type** parameter to digital.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. The feed to hard stop feature is not compatible with any third-party SERCOS drive systems.

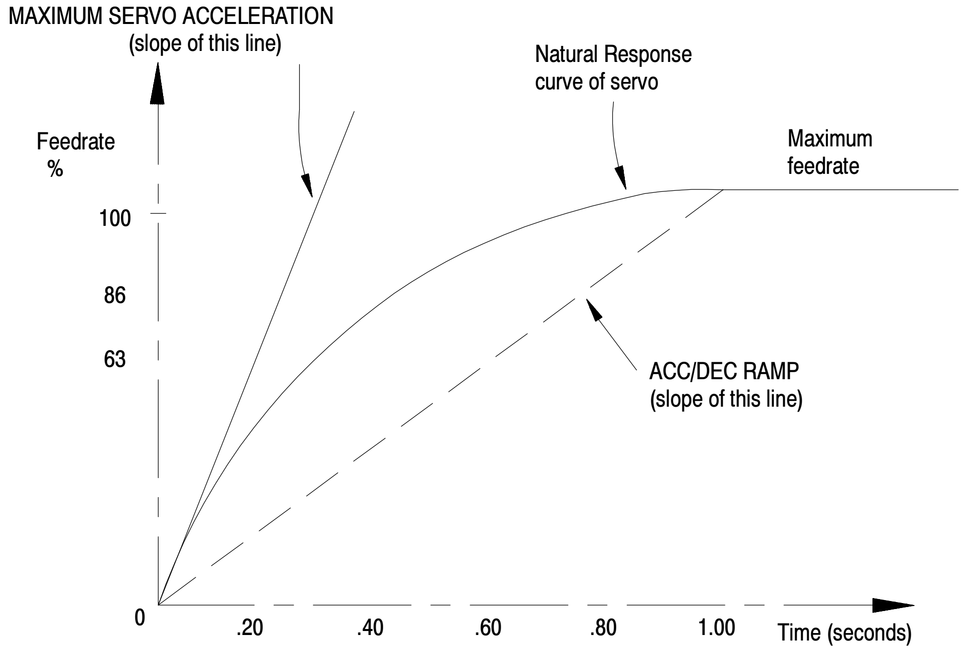
This parameter must be set independently for each axis that uses the feed to hard stop feature.

Maximum Servo Acceleration

Function

Important: This parameter is not available for digital motor configurations.

Figure 7.18
Graph of Typical Motor Maximum Acceleration



Though this value may be roughly estimated using a strip chart recorder, we recommend calculating this value using the following equation:

$$\text{Max. Servo Acc.} = \frac{\text{Maximum Motor Torque}}{\text{Effective Inertia}}$$

where:

Maximum Motor Torque — This is the maximum effective torque this servo motor can output given the amperage limitations of the servo amplifier being used. Maximum motor torque is typically in units of lb/in or newton/cm.

Effective Inertia — This includes the rotary inertia of all items being driven by the servo (such as gears and ball screws) and the effective inertia of the table, turret or other mechanical entity. Part mass should be considered if it is expected to be significant. Also any mechanical advantage given to the servo by motor gearing etc. should be factored into the calculations. Effective Inertia should typically be in units of in.-lb-sec^2 / rad. or Newton-meter-sec^2 / rad.

Axis	Parameter Number
(1)	[1021]
(2)	[2021]
(3)	[3021]
(4)	[4021]
(5)	[5021]
(6)	[6021]
(7)	[7021]
(8)	[8021]

Range

0.00000 to 23622.04701 in/min/sec

or

0.00000 to 599999.99400 mm/min/sec

Notes

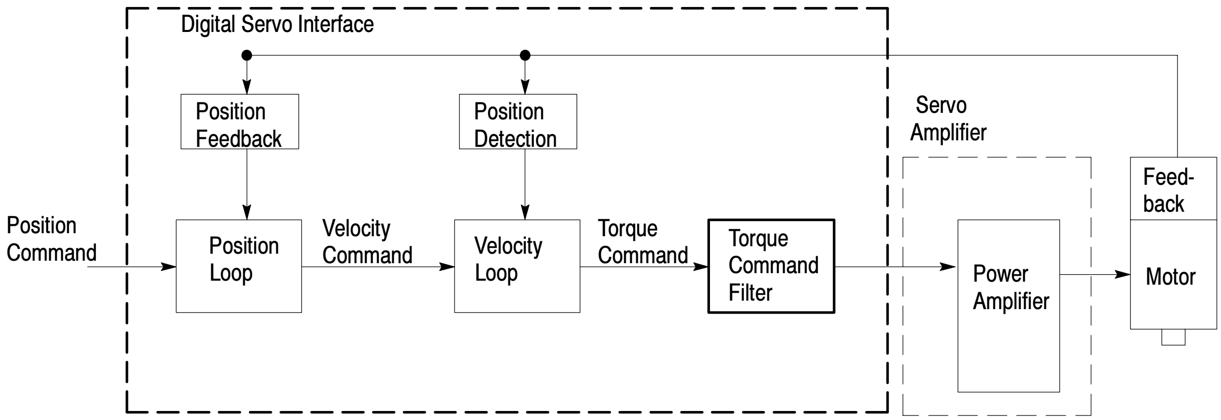
This parameter must be set independently for each servo.

This parameter represents the maximum acceleration of the servo resulting from peak current for the motor/amplifier being used. It must be much larger than the **Acc./Dec Ramp**, often several times larger.

Torque Filter Cut off Frequency

Function

This parameter is used on all digital systems and on analog systems that use the Tachless Software Velocity Loop feature (digital loop type). This filter can be used to remove some oscillations that can occur in your drive systems (typically caused by some mechanical characteristic). If an oscillation exists (and you have eliminated other possible tuning causes) set this torque filter to a point just below the frequency (in Hz) at which the oscillation stops occurring.



Axis	Parameter Number
(1)	[1162]
(2)	[2162]
(3)	[3162]
(4)	[4162]
(5)	[5162]
(6)	[6162]
(7)	[7162]
(8)	[8162]

Range

10 to 10000 Hz

Notes

This parameter must be set independently for each servo. The torque filter can be disabled by setting this parameter as high as possible (10000 Hz) and should be disabled during initial servo tuning.

If you have no method of identifying the frequency of the oscillation, use this procedure to set the torque filter cutoff frequency. First disable this filter by raising the cutoff frequency to a very high number (2000 or 3000). Then keep lowering the cutoff frequency until the oscillation stops occurring.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Digital Servo Parameters

The servo parameters in the sections that follow are available to configure digital systems only. They specify important motor and machine information necessary for proper motor commutation and digital drive operation. These parameters become available when the servo loop type is configured as “digital”.

Load Inertia Ratio

Function

The load to inertia ratio is used in conjunction with the standard motor parameters to choose the initial start up velocity loop gains (note typically the gains must be changed once the servo is connected to a load). The value entered here represents the ratio between the inertia of the motor and the inertia of the load. If a standard motor has been selected, motor inertias can be obtained from the standard motor tables in your *9/PC CNC Software and Hardware Installation and Integration Manual.*

If a nonstandard motor is to be used, motor inertias are generally included with the motor specification literature.

The load inertia can be calculated using conventional equations of mechanics. Calculations should consider drive train mass, and part mass (if it is expected to be significant).

Important: Once you have powered up your system and confirmed wiring, fine tuning of the servo gains is almost always required once the servo shaft is coupled to a load. Using online AMP to adjust the velocity/position loop gains will automatically reset the **Standard Motor Table Values** parameter to "No" at the control. Once you finish tuning your servos we recommend uploading the AMP from the control to store the new gain values and set this parameter to "No".

Axis	Parameter Number
(1)	[1017]
(2)	[2017]
(3)	[3017]
(4)	[4017]
(5)	[5017]
(6)	[6017]
(7)	[7017]
(8)	[8017]

Range

Selection	Result
(a)	1 : 0
(b)	1 : 1
(c)	1 : 2
(d)	1 : 3

Notes

This parameter must be set independently for each servo.

Select the value from the options provided that is closest to the calculated ratio. Generally it is better to round up rather than rounding down if between values.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Motor Type

Function

Select the motor type for the axis being configured by using the Allen-Bradley part numbers shown. Select “nonstandard” if the motor is not on this list. A different range is available (and a different quick edit number) depending on the type of digital servo hardware you have selected.

Axis	Parameter Number
(1)	[1651]
(2)	[2651]
(3)	[3651]
(4)	[4651]
(5)	[5651]
(6)	[6651]
(7)	[7651]
(8)	[8651]

Range

1326 Standard Motors	
Selection	**Result**
(a)	1326AB-B410G
(b)	1326AB-B420E
(c)	1326AB-B430E
(d)	1326AB-B515E
(e)	1326AB-B520E
(f)	1326AB-B530E
(g)	1326AB-B410J
(h)	1326AB-B420H
(i)	1326AB-B430G
(j)	1326AB-B515G
(k)	1326AB-B520F
(l)	1326AB-B720E
(m)	1326AB-B730E
(n)	1326AB-B740C
(o)	Nonstandard

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Number of Poles on Motor

Function

Enter the number of poles on the motor. For both standard and nonstandard digital servomotor(s), the number of poles from the motor specification literature should be entered here.

Important: If using a standard motor, the correct value for this parameter is entered automatically. Altering it here has no effect. Standard 1326 servo motors are 4-pole motors. If using a nonstandard motor, refer to the motors documentation for determining the number of poles.

Axis	Parameter Number
(1)	[1550]
(2)	[2550]
(3)	[3550]
(4)	[4550]
(5)	[5550]
(6)	[6550]
(7)	[7550]
(8)	[8550]

Range

Selection	Result
(a)	2 Poles
(b)	4 Poles
(c)	6 Poles
(d)	8 Poles
(e)	16 Poles
(f)	32 Poles
(g)	No Poles

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each servo.

Maximum Motor Speed

Function

This parameter is only used for servo systems controlled by the 1394 CNC Serial drive.

Enter the maximum motor speed (in RPM) that your motor/drive/load combination allows.

The motor speed entered here should be approximately 10% above what would be required for the axis to achieve its maximum feedrate.

Standard motor rated speeds and maximum speeds are shown in the motor tables of your *9/PC CNC Software and Hardware Installation and Integration Manual*. The correct value for the selected motor should be entered here.

If a nonstandard motor is to be used, motor maximum speeds can be obtained from the motor specification literature.

Axis	Parameter Number
(1)	[1580]
(2)	[2580]
(3)	[3580]
(4)	[4580]
(5)	[5580]
(6)	[6580]
(7)	[7580]
(8)	[8580]

Range

0 to 999999 rpm

Notes

This parameter must be set independently for each servo.

Digital Spindle users must set this parameter to determine the maximum gear one motor speed in addition to configuring the spindle group parameter “Maximum Spindle Speed - Gear 1”.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Motor Rated Current

Function

Enter the motor’s rated current in amps.

Important: If the motor is <u>not</u> a digital servomotor, this parameter is not used. It is recommended that the default value be left here.

Important: If using a standard motor, the correct value for this parameter is entered automatically. Altering it here has no effect. Standard 1326 servo motor rated currents are shown in your 1326 user’s manual.

If a nonstandard motor is to be used, motor rated currents obtained from the motor specification literature should be entered here.

Axis	Parameter Number
(1)	[1016]
(2)	[2016]
(3)	[3016]
(4)	[4016]
(5)	[5016]
(6)	[6016]
(7)	[7016]
(8)	[8016]

Range

0.000000 to 255.000000 Amps

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

This parameter must be set independently for each axis.

ID of Amplifier Rack

Function

SERCOS Systems
Use this parameter to indicate to the control the SERCOS ring address of the drive that you are connecting to this servo. In addition, use the output port number (refer to page 7-11) to associate each axis module amplifier in the particular 1394 rack to a particular 1394 Serial Drive axis module.

Up to eight single-axis drive systems can be installed in the SERCOS ring. The different drives on the SERCOS ring are each given an individual ring address for each servo motor controlled by that drive. Refer to your drive manufacturers documentation for details on setting this address at the drive. Valid addresses range from 0 to 254.

Axis	Parameter Number
(1)	[1160]
(2)	[2160]
(3)	[3160]
(4)	[4160]
(5)	[5160]
(6)	[6160]
(7)	[7160]
(8)	[8160]

Range

1 to 254

Notes

This parameter must be set independently for each servo.

Servo Amplifier Type

Function

This parameter lets you select the 1394 servo amplifier type. This is the parameter used to identify the axis module size.

ATTENTION: Only one motor can be connected to each of the output connectors on a servo amplifier.

Axis	Parameter Number
(1)	[1780]
(2)	[2780]
(3)	[3780]
(4)	[4780]
(5)	[5780]
(6)	[6780]
(7)	[7780]
(8)	[8780]

Range

Selection	Result
(a)	no servo amplifier
(b)	1394-AM03 (2KW module)
(c)	1394-AM04 (3KW module)
(d)	1394-AM07 (5KW module)
(e)	1394-AM50 (10KW module)
(f)	1394-AM75 (15KW module)

Notes

This parameter must be set independently for each servo.

Examples:

Using a nondigital servo amplifier for the motor connected to the DAC output (typically the spindle): Use **a**

Using a 1394 axis module for the motor, select the appropriate axis module size , or catalog number being used for the axis.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Shunt Resistor Pack

Function

Use this parameter to select the shunt resister value for your 22 kW 1394 system module. This value must correspond to the hardware shunt module that it is connected to.

Axis	Parameter Number
(1)	[1220]
(2)	[2220]
(3)	[3220]
(4)	[4220]
(5)	[5220]
(6)	[6220]
(7)	[7220]
(8)	[8220]

Range

Selection	Result
(a)	300 W
(b)	900 W
(c)	1800 W
(d)	3600 W

Notes

This parameter must be set independently for each servo.

Spindle Parameters

Function

The parameters in this section appear only if you have configured the servo as a spindle. A 9/PC system can have up to two spindles. Refer to chapter 3 for information on the **Specifying Axis Types** parameter.

Using a 1326 Motor/1394 Drive Combination as a Spindle

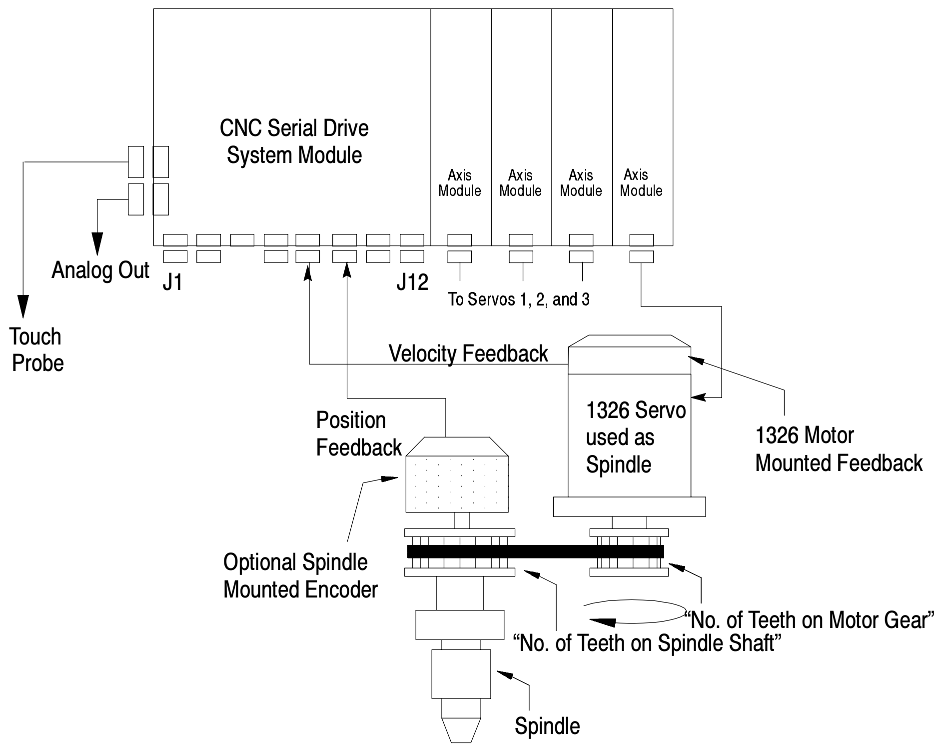
You can configure a 9/PC to use the 1394 drive as a spindle drive. Since the 1326 is a brushless servo motor, motor commutation and the velocity loop must be closed by the 9/PC. The limitation of two spindles still applies even if using this type of spindle configuration (i.e. max 2 spindles).

There are two feedback configuration options for 1394 spindles.

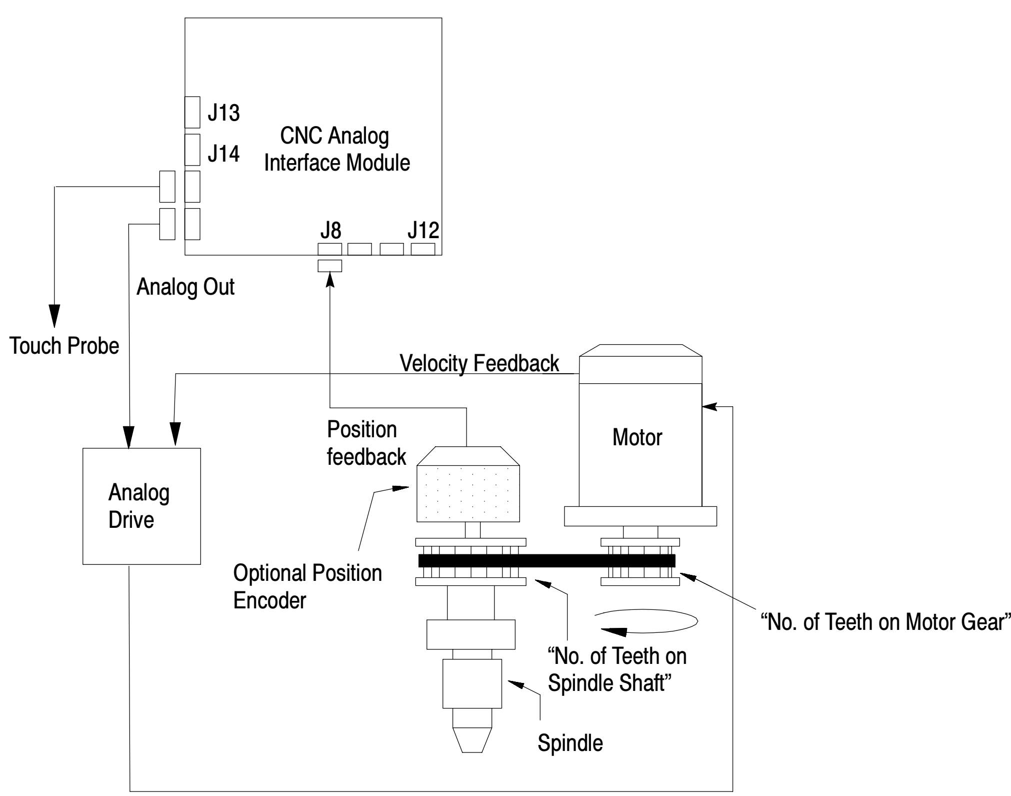
Single-device feedback — This configuration uses the 1326 motor-mounted feedback for motor commutation, and velocity loop closure. With this configuration with only the motor-mounted feedback device, closed-loop spindle operations including solid-tapping, spindle synchronization, virtual C, are not permitted. Spindle orient and threading modes are permitted but operate in the same fashion as an analog spindle with no feedback (simulated feedback is used for threading and “shot-pin” type open-loop orients must be used).

Two-device feedback — This configuration uses the 1326 motor-mounted feedback for motor commutation and velocity loop closure. A second feedback device (single marker encoder) must be installed on the spindle shaft (one to one ratio with the spindle, see the parameter “No of Teeth on Spindle Shaft”) for position loop closure. This configuration allows closed-loop spindle operations for a 1326 motor .

1394 CNC Serial Drive Connections



1394 Analog Interface Module Connections



To configure a 1394 spindle, make the following AMP configuration considerations:

- Select the “Axis Type” as spindle (see chapter 3 for details)
- “Servo Hardware Type” is “SERCOS”
- “Servo Loop Type” is configured as “Digital or Digital Spindle” OR “Position or Analog Spindle”.
- “Servo Position Loop Type” is always Open Loop regardless of the single or two device feedback configurations.
- Use the parameters “No of Teeth on Motor Gear” and “No Teeth on Spindle Shaft” to account for any gearing between the motor shaft and the actual physical machine spindle.
- Use of different gear ranges is not supported when using the 1394 drive as a spindle. The AMP parameter “Number of Gears Used” is forced to one in software. Values entered into AMP for spindle gear ranges other than gear one are ignored.

  Many of the gear parameters for Spindle Gear 1 still impact your 1394 spindle. For example:

  - “Max Tap speed for Spn, Gear 1”
  - Min and “Max Spindle Speed - Gear 1” (though you still need to configure a max motor speed in this servo parameters group)
  - “Gain for Spindle, Gear 1”

  Other spindle gear parameters that do not impact the 1394s operation as a spindle and should not be configured include:

  - “Voltage at Max for Gear 1” (1394 as spindle output current is scaled via “Maximum Rated Torque” servo parameter).
  - “Number of Gears Used” (1394 as spindle only allows gear range 1).

  Other parameters in the Spindle Groups are used assuming supporting hardware is available and the software option for the feature has been purchased. For example without a separate position feedback device installed the closed-loop spindle orient parameters are not used.
- The parameter “Initial Gain of Position Loop” is not used for a 1394 drive configured as a spindle. Use the parameters “Gain for Spindle, Gear 1” to set the position loop gain for solid-tapping, closed-loop spindle, and synchronized spindle orient operations.
- All applicable velocity and position loop parameters must be properly configured. If the same feedback device is used for both loops you must enter the same values for parameters of both loops.
- After your spindle is configured you may still have to tune the servo loops as you would any closed-loop system.

- The 1394 as a spindle is seen on the Online AMP servo parameter tuning page. Parameters on this page an be used to tune the 1326 motor. However the position loop gain on this page does not apply to the 1394 as a spindle. The spindle gain can only be adjusted offline in AMP via the spindle parameter "Gain for Spindle, Gear 1 Param."

Spindle Type for Axis

Function

For each spindle configured, this parameter identifies the type of spindle. You cannot have any spindle axes identified as the same type in one system. If you are using more than one spindle, you must configure the spindles in this order:

- the first spindle is spindle 1
- the spindle that follows spindle 1 must be configured as spindle 2

Axis	Parameter Number
(1)	[1040]
(2)	[2040]
(3)	[3040]
(4)	[4040]
(5)	[5040]
(6)	[6040]
(7)	[7040]
(8)	[8040]

Range

Selection	Result
(a)	None
(b)	spindle 1
(c)	spindle 2

This parameter must be set for each spindle.

No of Teeth on Motor Gear

Function

This parameter is used only for 1394 drives configured to use one of their axis modules as a spindle drive. See page 7-79 for details.

Use the parameters **No of Teeth on Motor Gear** and **No of Teeth on Spindle Shaft** to identify the gear ratio between the motor shaft and the final spindle output. If the motor is directly connected in a one to one ratio with the spindle shaft you must still identify the ratio by entering one for both of these parameters. The ratio established by these parameters is used for spindle speed calculations by the control and to identify the ratio between velocity feedback and spindle position feedback for spindles with two feedback devices.

In some cases, there may be a series of gears or gearbelt pulleys that make up a final gear ratio for a spindle. In this case, it is necessary to calculate this gear ratio so that this parameter and the parameter **No of Teeth on Spindle Shaft** result in the desired ratio.

Axis	Parameter Number
(1)	[1013]
(2)	[2013]
(3)	[3013]
(4)	[4013]
(5)	[5013]
(6)	[6013]
(7)	[7013]
(8)	[8013]

Range

1 to 32767

Notes

This parameter must be set independently for each 1394 spindle.

No of Teeth on Spindle Shaft

Function

This parameter is used only for 1394 drives configured to use one of their axis modules as a spindle drive. See page 7-79 for details.

Use the parameters **No of Teeth on Motor Gear** and **No of Teeth on Spindle Shaft** to identify the gear ratio between the motor shaft and the final spindle output. If the motor is directly connected in a one to one ratio with the spindle shaft you must still identify the ratio by entering one for both of these parameters. The ratio established by these parameters is used for spindle speed calculations by the control and to identify the ratio between velocity feedback and spindle position feedback for spindles with two feedback devices.

In some cases, there may be a series of gears or gearbelt pulleys that make up a final gear ratio for a spindle. In this case, it is necessary to calculate this gear ratio so that this parameter and the parameter **No of Teeth on Motor Gear** result in the desired ratio.

Axis	Parameter Number
(1)	[1018]
(2)	[2018]
(3)	[3018]
(4)	[4018]
(5)	[5018]
(6)	[6018]
(7)	[7018]
(8)	[8018]

Range

1 to 32767

Notes

This parameter must be set independently for each 1394 spindle.

Important: For two device feedback 1394 drives as a spindle configurations, the 2nd feedback device used for closure of the position loop must be a 1:1 ratio to the final spindle rotation. This parameter in that case would be used to define the gear ratio between motor shaft and final spindle output for the velocity loop.

Friction Parameters

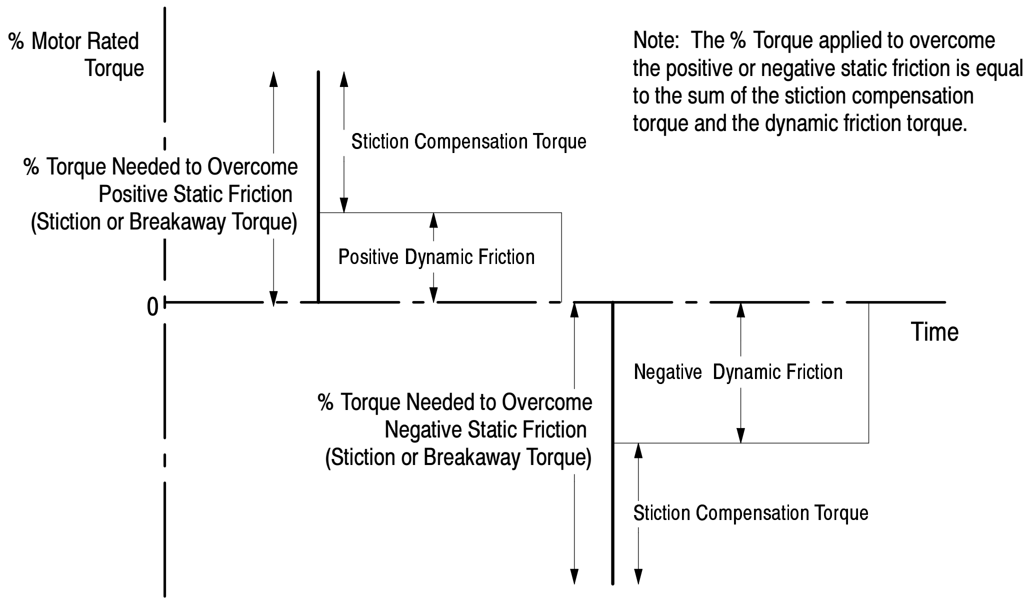
The parameters in this section are typically used on systems where friction is a problem.

- Symptoms of high friction systems can include: position error lag at the beginning of a move
- position overshoot and slow recovery at the end of a move
- position error on a circular quadrant boundaries

These parameters adjust the output torque of the system to overcome the dynamic friction that occurs during a move, or the static friction (stiction) that occurs at the start of a move.

Friction parameters are used on all digital systems and on analog systems that use the Tachless Software Velocity Loop feature (loop type digital).

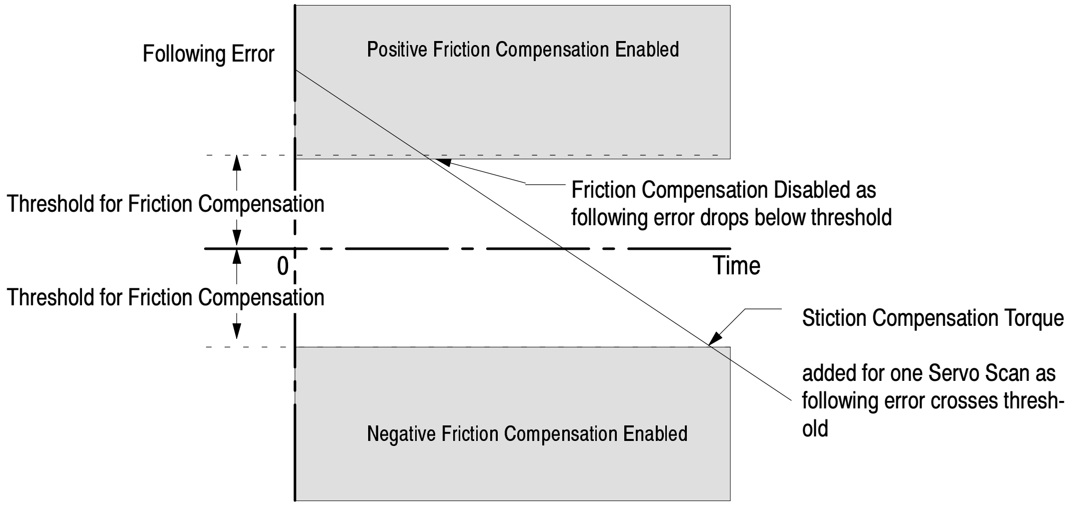
The following figure illustrates a positive and negative move showing the amount of torque to initiate a move is higher than the torque necessary to keep the axis moving.



Threshold for Friction Comp

Function

This parameter specifies the velocity (in terms of feedback counts of following error) at which the control adds the "Stiction Compensation Torque" value to the positive or negative dynamic friction value. When the following error drops below this threshold value, both stiction and friction compensation are zero. When the following error is above this threshold, the positive or negative dynamic friction value is enabled. Additionally when the following error value crosses this following error threshold at the beginning of a move, the additional Stiction Compensation Torque value is added to the dynamic friction value.



The units of this parameter are position feedback counts, and not programming resolution units. Using position feedback counts simplifies the adjustment of the threshold.

The same threshold value will be used in the positive and negative positions. While the following error counts are greater than this threshold, the friction compensation is applied to the torque command. When the magnitude of the following error counts crosses the threshold at the beginning of a move, stiction compensation will be applied on the transition. For example setting this parameter to 10 means that when the following error is between:
 0 and 9 counts — Compensation for friction is zero
 10 counts or higher — Friction compensation is added
 the first iteration that reaches or exceeds 10 — Stiction and Friction is added for one servo iteration

Axis	Parameter Number
(1)	[1810]
(2)	[2810]
(3)	[3810]
(4)	[4810]
(5)	[5810]
(6)	[6810]
(7)	[7810]
(8)	[8810]

Range

0 to 60,000 feedback counts

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Stiction Comp Torque Percent

Function

This parameter specifies any additional torque to be applied to the motor to overcome static friction when the motor starts from rest (as selected with the parameter **Threshold for Friction Compensation**). This torque percent is added to any existing positive or negative friction compensation value for one servo scan only.

Axis	Parameter Number
(1)	[1811]
(2)	[2811]
(3)	[3811]
(4)	[4811]
(5)	[5811]
(6)	[6811]
(7)	[7811]
(8)	[8811]

Range

0 to 100% motor rated continuous torque

Notes

The same percentage of torque is applied to the axis in both directions for static friction. You can not differentiate between positive and negative moves when using stiction compensation.

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Positive Friction Comp Percent

Function

This parameter specifies any additional torque to be applied to the motor, for moves in the positive direction, to overcome dynamic friction once an axis is moving (as selected with the parameter **Threshold for Friction Compensation**).

Axis	Parameter Number
(1)	[1812]
(2)	[2812]
(3)	[3812]
(4)	[4812]
(5)	[5812]
(6)	[6812]
(7)	[7812]
(8)	[8812]

Range

0 to 100% motor rated continuous torque

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

Negative Friction Comp Percent

This parameter specifies any additional torque to be applied to the motor, for moves in the negative direction, to overcome dynamic friction once an axis is moving (as selected with the parameter **Threshold for Friction Compensation**).

Axis	Parameter Number
(1)	[1813]
(2)	[2813]
(3)	[3813]
(4)	[4813]
(5)	[5813]
(6)	[6813]
(7)	[7813]
(8)	[8813]

Range

0 to 100% motor rated continuous torque

Notes

SERCOS Users — this parameter is only used when configuring a 1394 Serial Drive system. It is not used for third-party SERCOS drives.

END OF CHAPTER

Chapter 8

Jog Parameters

Chapter Overview

Jogging covers all axis motion that is controlled by the use of push buttons, switches, or logic initiated motion. Typically these are mounted on or near the MTB panel. This does not include any of the homing operations. Homing operations are discussed in chapter 5.

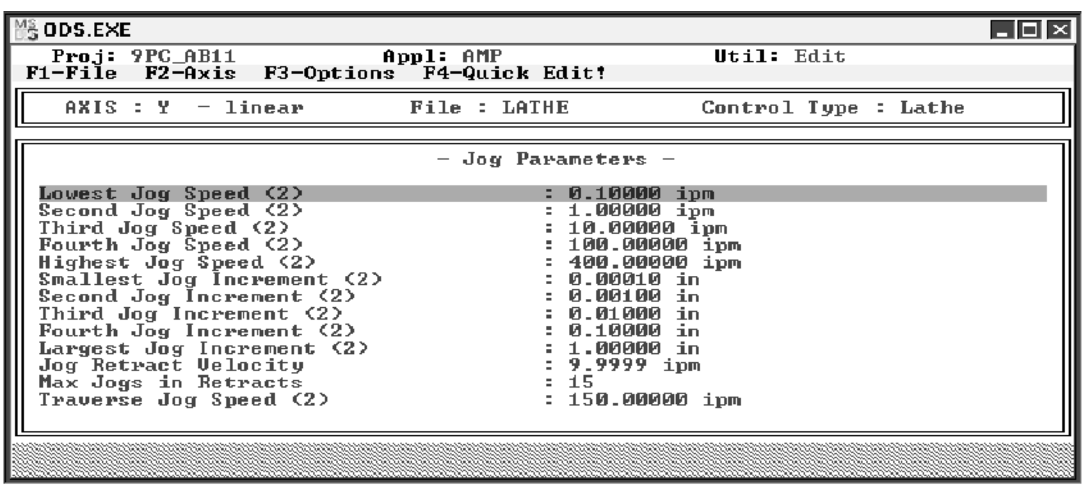
- Continuous jogging refers to holding the jog push button down and letting the selected axis move continuously at the selected jog feedrate.
- Incremental jogs force the selected axis to move a preset incremental distance.
- Jog retract allows manual withdrawal from a complex part and then provides automatic return to the pre-retracted position.

For details on jogging the axes, refer to your *9/PC CNC Opeation and Programming Manual*.

This table lists the parameters that are normally necessary to configure jogging operations:

Parameter:	Page:
Jog Speeds	8-2
Jog Increments	8-6
Jog Retract	8-11

After you select "Jog Parameters" from the main menu, this screen becomes available:



Jog Speeds

There are five Jog Speed parameters used to set the speed at which continuous jogs take place:

Important: Incremental jogs occur at the value assigned to the **Third Jog Speed** parameter.

The value specified for each of the five jog speed parameters is equal to the speed at which a continuous jog move takes place (unless that jog is selected as a "Traverse" jog). These five parameters correspond to the five selections under <SPEED/MULTIPLY> on the MTB panel when a continuous jog takes place. If the continuous jog is selected as a traverse jog, then the <SPEED/MULTIPLY> setting is ignored, and the special traverse jog speed is used.

The jog speed parameters may range between the following values; however, keep in mind that none of the values for these parameters may overlap (the feedrates must increase or stay the same as the higher switch positions are set). This does not apply to the traverse jog speed. This is the acceptable range for jog speeds:

0.00600 to 243840.00000 mmpm.

or

0.00024 to 9600.00000 ipm.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Lowest Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when <SPEED/MULTIPLY> on the MTB panel is set LOW X1.

Axis	Parameter Number
(1)	[1370]
(2)	[2370]
(3)	[3370]
(4)	[4370]
(5)	[5370]
(6)	[6370]
(7)	[7370]
(8)	[8370]

Range

0.00600 mmpm to Second Jog Speed value

or

0.00024 ipm to Second Jog Speed value

The maximum value that may be set for this parameter is dependant on the value set for the parameter “Second Jog Speed.”

Set each parameter independently for each axis.

Notes

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Second Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when <SPEED/MULTIPLY> on the MTB panel is set to the second position.

Axis	Parameter Number
(1)	[1371]
(2)	[2371]
(3)	[3371]
(4)	[4371]
(5)	[5371]
(6)	[6371]
(7)	[7371]
(8)	[8371]

Range

Lowest Jog Speed to Third Jog Speed value

The minimum and maximum values that may be set for this parameter are dependant on the value set for the parameters “Lowest Jog Speed” and “Third Jog Speed.”

Set each parameter independently for each axis.

Notes

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Third Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when <SPEED/MULTIPLY> on the MTB panel is set to MED X100.

Important: Incremental jogs jog at the value assigned to the **Third Jog Speed** parameter.

Axis	Parameter Number
(1)	[1372]
(2)	[2372]
(3)	[3372]
(4)	[4372]
(5)	[5372]
(6)	[6372]
(7)	[7372]
(8)	[8372]

Range

Second Jog Speed to Fourth Jog Speed value

The minimum and maximum values that may be set for this parameter are dependant on the value set for the parameters "Second Jog Speed" and "Fourth Jog Speed."

Set each parameter independently for each axis.

Notes

This is a global parameter. The value set here applies to all axes.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Fourth Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when <SPEED/MULTIPLY> on the MTB panel is set to MEDH X1000.

Axis	Parameter Number
(1)	[1373]
(2)	[2373]
(3)	[3373]
(4)	[4373]
(5)	[5373]
(6)	[6373]
(7)	[7373]
(8)	[8373]

Range

Third Jog Speed to Highest Jog Speed value

The minimum and maximum values that may be set for this parameter are dependant on the value set for the parameters "Third Jog Speed" and "Highest Jog Speed."

Set each parameter independently for each axis.

Notes

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Highest Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when <SPEED/MULTIPLY> on the MTB panel is set to HIGH X10000.

Axis	Parameter Number
(1)	[1374]
(2)	[2374]
(3)	[3374]
(4)	[4374]
(5)	[5374]
(6)	[6374]
(7)	[7374]
(8)	[8374]

Range

Fourth Jog Speed to 243840.00000 mmpm

or

Fourth Jog Speed to 9600.00000 ipm

The minimum and maximum values that may be set for this parameter are dependant on the value set for the parameter's "Fourth Jog Speed" and "System Max."

Set each parameter independently for each axis.

Notes

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Jog Increments

There are five Jog Increment parameters used to set the distance that incremental jogs travel:

Important: Incremental jogs jog at the value assigned to the **Third Jog Speed** parameter. This value may be overridden by the logic program.

The value specified for each of the five jog increment parameters is equal to the distance that an axis travels when an incremental jog takes place. These five parameters correspond to the five positions of <SPEED/MULTIPLY> on the MTB panel when an incremental jog is taking place.

The five parameters may range between the following values; however, keep in mind that none of the values for these parameters may overlap (the distance must increase as the higher switch positions are set). This is the allowable range for jog increments.

0.00010 to 127000.00000 mm.

or

0.00000 to 5000.00000 ipm.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Smallest Jog Increment

Function

The value specified for this parameter is equal to the distance that an incremental jog move travels when <SPEED/MULTIPLY> on the MTB panel is set to LOW X1.

Axis	Parameter Number
(1)	[1360]
(2)	[2360]
(3)	[3360]
(4)	[4360]
(5)	[5360]
(6)	[6360]
(7)	[7360]
(8)	[8360]

Range

0.00010 mm or .0001 degree to Second Jog Increment value

or

0.00000 inch to Second Jog Increment value

The maximum value that may be set for this parameter is dependant on the value set for the parameter “Second Jog Increment.”

Notes

You must set this parameter independently for each axis.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Second Jog Increment

Function

The value specified for this parameter is equal to the distance that an incremental jog move travels when <SPEED/MULTIPLY> on the MTB panel is set to MEDL X10.

Axis	Parameter Number
(1)	[1361]
(2)	[2361]
(3)	[3361]
(4)	[4361]
(5)	[5361]
(6)	[6361]
(7)	[7361]
(8)	[8361]

Range

Smallest Jog Increment to Third Increment value

The minimum and maximum values that may be set for this parameter are dependant on the values set for the parameters “Smallest Jog Increment” and “Third Jog Increment.”

Notes

You must set this parameter independently for each axis.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Third Jog Increment

Function

The value specified for this parameter is equal to the distance that an incremental jog move travels when <SPEED/MULTIPLY> on the MTB panel is set to MED X100.

Axis	Parameter Number
(1)	[1362]
(2)	[2362]
(3)	[3362]
(4)	[4362]
(5)	[5362]
(6)	[6362]
(7)	[7362]
(8)	[8362]

Range

Second Jog Increment to Fourth Jog Increment value

The minimum and maximum values that may be set for this parameter are dependant on the values set for the parameters "Second Jog Increment" and "Fourth Jog Increment."

Notes

You must set this parameter independently for each axis.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Fourth Jog Increment

Function

The value specified for this parameter is equal to the distance that an incremental jog move travels when <SPEED/MULTIPLY> on the MTB panel is set to MEDH X1000.

Axis	Parameter Number
(1)	[1363]
(2)	[2363]
(3)	[3363]
(4)	[4363]
(5)	[5363]
(6)	[6363]
(7)	[7363]
(8)	[8363]

Range

Third Jog Increment to Largest Jog Increment value

The minimum and maximum values that may be set for this parameter are dependant on the values set for the parameters “Third Jog Increment” and “Largest Jog Increment.”

Notes

You must set this parameter independently for each axis.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Largest Jog Increment

Function

The value specified for this parameter is equal to the distance that an incremental jog move travels when <SPEED/MULTIPLY> on the MTB panel is set to HIGH X10000.

Axis	Parameter Number
(1)	[1364]
(2)	[2364]
(3)	[3364]
(4)	[4364]
(5)	[5364]
(6)	[6364]
(7)	[7364]
(8)	[8364]

Range

Fourth Jog Increment to 127000.00000 mm or degrees

or

Fourth Jog Increment to 5000.00000 inch

The minimum value that may be set for this parameter is dependant on the value set for the parameter Fourth Jog Increment.”

Notes

You must set this parameter independently for each axis.

The default setting for <SPEED/MULTIPLY> on the pushbutton MTB panel is LOW X1.

Jog Retract

By pressing the **CYCLE START** button, the jog retract feature lets you:

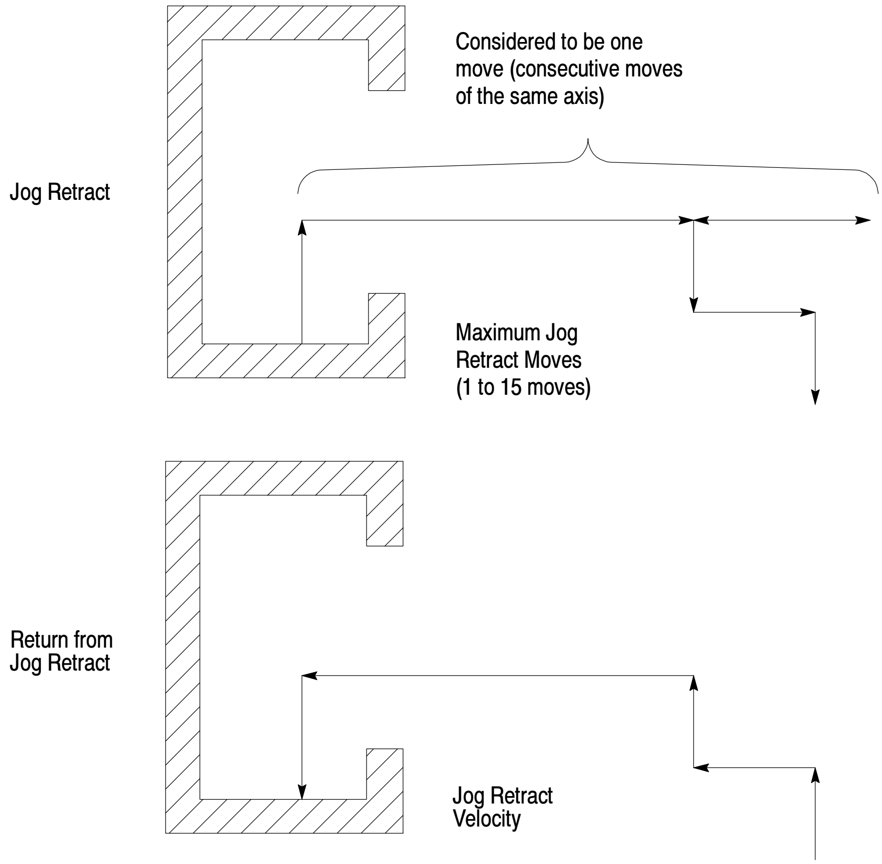
- stop part program execution
- jog the tool to another location
- return the tool to its original position (see Figure 8.1)

The control remembers up to the number of jog moves specified by the parameter **Max Jogs in Retracts** and returns the tool to the part by the same path and at the feedrate specified by the parameter **Jog Retract Velocity**.

Refer to your *9/PC CNC Operation and Programming Manual* for details on using the jog retract feature.

Important: Jog retract moves are performed as normal jogs and are subject to the feedrates and resolutions that that jog type has been assigned.

Figure 8.1
Jog Retract Parameters



Jog Retract Velocity

Function

This parameter specifies the speed at which the control moves the tool when returning from jog retract to the position where jog retract began. A return from jog retract is usually done when <CYCLE START> is pressed, after performing a jog retract.

Axis	Parameter Number
All	[330]

Range

.0060 to 243840.0000 mmpm

or

0.0002 to 9600.0000 ipm

Notes

We recommend that the jog return speed be kept relatively low so that the control can accurately position the tool when it reaches the original position.

This is a global parameter. The value set here applies to all axes.

Max Jogs in Retracts

Function

This parameter specifies the maximum number of jog retract moves that the control can retrace. The value set for this parameter applies to the total number of jog retract moves that may be remembered by the control. This number is the combined moves on all of the axes.

Refer to your *9/PC CNC Operation and Programming Manual* for details on using this feature.

Axis	Parameter Number
	Single Process
All	[331]

Range

1 to 15 moves

Notes

If the tool is jogged more than the specified number of moves, the first move of the return from jog retract is a straight line from the final jog position to the jog position of the last allowed jog retract.

In jog retract, consecutive jogs in the same axis are considered to be one move, i.e., three or four jogs are equal to one jog retract move.

This is a global parameter. The value set here applies to all of the axes.

Traverse Jog Speed

Function

The value specified for this parameter is equal to the speed at which a continuous jog move takes place when the traverse jog speed is selected (typically this is selected by holding down the <TRVRS> button when jogging). This traverse feedrate may be modified by the operator using the <RAPID FEEDRATE OVERRIDE> switch.

Axis	Parameter Number
(1)	[1003]
(2)	[2003]
(3)	[3003]
(4)	[4003]
(5)	[5003]
(6)	[6003]
(7)	[7003]
(8)	[8003]

Range

0.00600 mmpm to 10160.00004 mmpm

or

0.00024 ipm to 400.00000 ipm.

Notes

This parameter must be set independently for each axis.

END OF CHAPTER

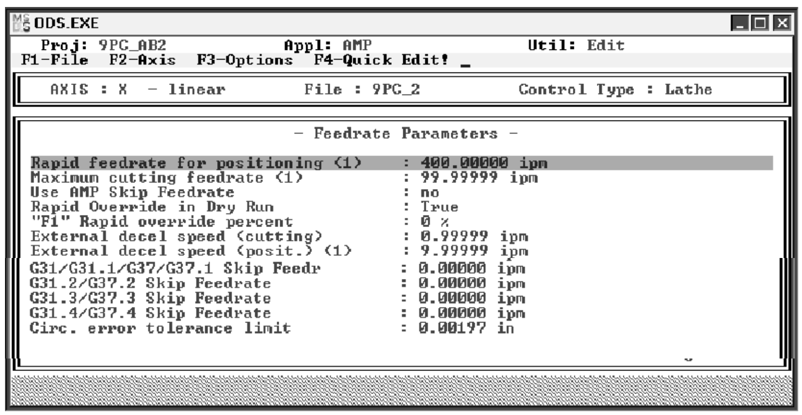
Feedrate Parameters

Chapter Overview

This section describes the parameters that affect programmed feedrates, with the exception of jogging feedrates (which are discussed in chapter 8). We describe these programmed feedrate parameters:

Parameter:	Page:
Standard Motion Feedrate parameters	9-1
Feedrate Override parameters	9-4
Skip, Gauge, and Probing cycles, feedrate parameters	9-7

When you select the “Feedrate Parameters” group from the main menu screen, these parameter screens become available:



Standard Motion Feedrate Parameters

Standard motion feedrate parameters include:

Rapid Feedrate for Positioning

Function

Use this parameter to determine the feedrate that is used by the control for rapid positioning. This mainly includes the feedrate of axis motions when in the G00 mode; however, this feedrate is used for other operations as specified in your *9/PC CNC Operation and Programming Manual*.

Axis	Parameter Number
(1)	[1203]
(2)	[2203]
(3)	[3203]
(4)	[4203]
(5)	[5203]
(6)	[6203]
(7)	[7203]
(8)	[8203]

Range

0.00000 to 243840.00000 mmpm

or

0.00000 to 9600.00000 ipm

Notes

This feedrate may be modified by the operator with the rapid feedrate override switch.

A value for this feedrate is set independently for each axis. Motions that use more than one axis for rapid positioning (including a virtual axis which requires two axes to position) use a feedrate that will not exceed the rapid feedrate for any of the moving axes. Refer to your *9/PC CNC Operation and Programming Manual* for additional information.

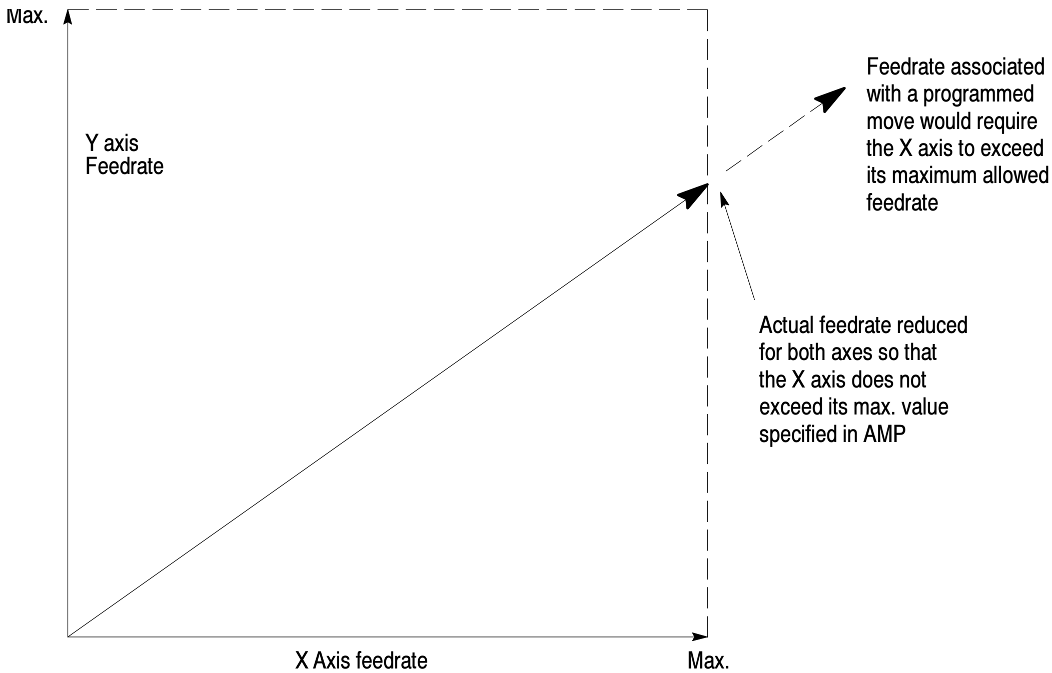
Maximum Cutting Feedrate

Function

This parameter specifies the maximum feedrate that the axis uses when moving at a programmed feedrate. This is also referred to as a "feedrate clamp" because the feedrate is, in effect, clamped at this maximum cutting feedrate. When an attempt is made to exceed the maximum cutting feedrate through programming or using the feedrate override switch, the feedrate is "clamped" at the maximum cutting feedrate. This parameter does not affect rapid feedrates (G00). The maximum cutting feedrate can not be larger than the value set for Rapid Feedrate for Positioning parameter.

In a contouring move involving more than one axis, the control limits the vector feedrate so that the maximum cutting feedrates of all axes involved are not exceeded.

Figure 9.1
Effect of Maximum Cutting Feedrate



Refer to your *9/PC CNC Operation and Programming Manual* for additional information about axis motion.

Axis	Parameter Number
(1)	[1201]
(2)	[2201]
(3)	[3201]
(4)	[4201]
(5)	[5201]
(6)	[6201]
(7)	[7201]
(8)	[8201]

Range

0.00000 ipm or mmpm to Rapid Feedrate for Positioning

Notes

This parameter must be set independently for each axis.

Feedrate Override Parmeters

Feedrate Override parameters include:

Rapid Override in Dry Run

Function

This parameter determines whether the Rapid Feedrate Override switch affects the Feedrate when Dry Run is active. Note that Dry Run replaces all feedrates programmed with an F-word in a program with the Maximum cutting feedrate. Rapid moves remain at the rapid feedrate. Refer to chapter 8 of your *9/PC CNC Operation and Programming Manual* for additional information on Dry Run mode.

True – for this parameter means that **<RAPID FEEDRATE OVERRIDE>** affects the feedrate for rapid moves when a program is executing in Dry Run. The programmed feedrates are overridden using **<FEEDRATE OVERRIDE>**.

False – for this parameter means that the Rapid Feedrate Override switch is ignored when a program is executing in Dry Run mode. Both rapid and programmed feedrates are overridden using **<FEEDRATE OVERRIDE>** on the MTB panel.

Axis	Parameter Number
All	[66]

Range

Selection	Result
(a)	True
(b)	False

Notes

This is a global parameter. The value entered here applies to all axes.

“ F1” Rapid Override Percent

Function

Use this parameter to set a value for the first position of the Rapid Feedrate Override switch (labeled F1 on the MTB panel). Enter a percent value for this parameter. The value for the other three switch positions is determined in PAL. The axis travels in normal rapid positioning (G00) mode at the Rapid Feedrate for Positioning multiplied by the percent value entered here when the Rapid Feedrate override switch is set to F1.

Axis	Parameter Number
All	[67]

Range

0 to 100%

Notes

When a value of 0% is set for this parameter, the Rapid Feedrate for Positioning is reduced to 0 and, in effect, places the control in “Feedhold.” No axis motion takes place.

When the control powers up F1 is the default feedrate override on a push-button MTB panel.

This parameter in no way relates to the single-digit “Feedrate for F1” parameter.

This is a global parameter. The value set here applies to all axes.

External Decel Speed (Cutting)

Function

Enter the feedrate at which all axes decelerate to when an external deceleration request is sent from the Logic program during a cutting move (G01, G02, or G03 active).

This feature requires coordination with the Logic Program and is usually set up to operate as follows:

An axis travel switch is placed at a point where you want to slow down axis travel. Usually this is somewhere near the hardware overtravel. When the switch is tripped, the logic flag “External Deceleration Request” is set TRUE, and the control decelerates all axes to the feedrate entered for this parameter.

As long as the logic flag remains TRUE, all axis motion in any direction will be restricted to the feedrate entered here. Refer to your *9/PC CNC Logic Reference Manual* for more information.

Jogging moves are not affected by this parameter.

This is not a per-axis parameter. The speed selected here is applied to all axes.

Axis	Parameter Number
	Single Process
All	[408]

Range

0.00000 to 10160.00004 mmpm

or

0.00000 to 400.00000 ipm

Notes

This is a global parameter. The value set here applies to all axes.

External Decel Speed (Posit .)

Function

Enter the feedrate to which the axis is to decelerate when an external deceleration request is sent from the Logic Program during a positioning move (G00 active).

This feature requires coordination with the Logic Program and is usually set up to operate as follows:

An axis travel switch is placed at a point where you want to slow down axis travel. Usually this is some place near the hardware overtravel. When the switch is tripped, the Logic flag “External Deceleration Request” is set TRUE, and the control decelerates each axis to the feedrate entered for this parameter.

As long as the logic flag remains TRUE, the motion of each axis in any direction is restricted to the feedrate entered here. Refer to your *9/PC CNC Logic Reference Manual* for more information.

Jogging moves are not affected by this parameter.

Axis	Parameter Number
(1)	[1200]
(2)	[2200]
(3)	[3200]
(4)	[4200]
(5)	[5200]
(6)	[6200]
(7)	[7200]
(8)	[8200]

Range

0.000000 to 10160.00004 mmpm

or

0.00000 to 400.00000 ipm

Notes

This parameter must be set independently for each axis.

Skip Cycle Feedrate Parameters

Skip Cycle Feedrate Parameters include:

These parameters apply to the Skip cycles that are executed when one of these G-codes is programmed:

G31	G37
G31.1	G37.1
G31.2	G37.2
G31.3	G37.3
G31.4	G37.4

Refer to your *9/PC CNC Operation and Programming Manual* for additional information on skip cycle feedrates.

Use AMP Skip Feedrate

Function

Use this parameter to determine whether the Skip cycles, which are executed when a G31 - G31.4 or G37 - G37.4 is programmed, use the current programmed feedrate or the feedrate entered as an AMP parameter.

NO – entered as a value for this parameter causes the control to use the current programmed feedrate. The feedrate entered as an AMP parameter is ignored.

YES – entered as a value for this parameter causes the control to use the AMP feedrate for the Skip cycle. The programmed feedrate is ignored during skip cycle execution.

Axis	Parameter Number
	Single Process
All	[4]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This parameter is a global parameter. The value applies to all axes.

Skip Feedrate For G31/G37

Function

Use these parameters to establish the feedrate that the skip cycles use when executed. The feedrate set here for these parameters is used only if the parameter Use AMP Skip Feedrate is “yes.” If that parameter is “no,” then these parameters are ignored. Refer to your *9/PC CNC Operation and Programming Manual* for details on skip cycle feedrates.

Enter a feedrate for this parameter in units of inches-per-minute or millimeters-per-minute.

Important: Note that the G31 and G31.1 are functionally identical as are the G37 and G37.1 codes. This means that they use the same feedrate for execution, and there is only one feedrate parameter for both pairs.

Parameter	Parameter Number
	Single Process
G31/G31.1/G37/G37.1 Skip Feedrate	[5]
G31.2/G37.2 Skip Feedrate	[2]
G31.3/G37.3 Skip Feedrate	[6]
G31.4/G37.4 Skip Feedrate	[7]

Range

0.00000 to 243840.00000 mmpm

or

0.00000 to 9600.00000 ipm

Notes

This is a global parameter. The value entered here applies to all axes.

Circular Error Tolerance Limit

Function

Use this parameter to enter the amount of inaccuracy permitted when programming the creation of an arc. The control takes the distance from the start-point of the arc to the center-point, and the distance from the end-point of the arc to the center-point, and determines the difference between these two distances.

If the difference between these two lengths is greater than the value entered for this parameter, the control generates an error.

This error is normally generated when using I, J, K to program the arc center. When using an R to program the arc center, the only time this error may occur is if the distance between the start-point and end-point of the arc is more than twice the programmed radius.

Axis	Parameter Number
	Single Process
All	[411]

Range

0.00000 to 25.40000 mm

or

0.00000 to 1.00000 inch

Notes

This is a global parameter. The value set here applies to all axes.

END OF CHAPTER

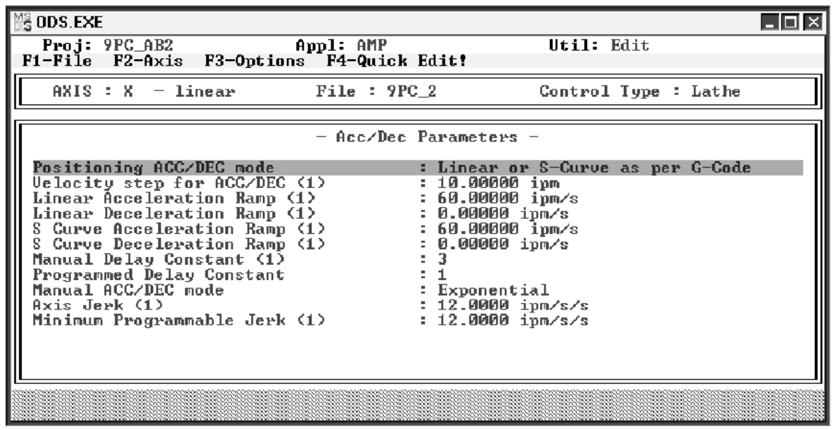
Acc/Dec Parameters

Chapter Overview

This chapter covers the acceleration and deceleration AMP parameters for the axes controlled by the 9/PC.

Use acceleration and deceleration (Acc./Dec) parameters to control the method and rate of speed change when an axis is starting or stopping.

When you select the "Acc./Dec Parameters" group from the main menu screen in the AMP application, the workstation displays this screen:



This chapter covers these Acc./Dec parameters:

Parameter:	Page:
Positioning Acc./Dec Mode	10-3
Velocity Step for Acc./Dec	10-7
Linear Acceleration Ramp	10-8
Linear Deceleration Ramp	10-10
S-Curve Acceleration Ramp	10-11
S-Curve Deceleration Ramp	10-13
Manual Delay Constant	10-14
Programmed Delay Constant	10-17
Manual Acc./Dec Mode	10-18
Axis Jerk	10-19
Minimum Programmable Jerk	10-21

Acceleration and Deceleration

The parameters in these subsections cover linear and exponential acceleration for the control. Some are global parameters (they affect all axes) and some can be entered independently for each axis.

Proper setting of these parameters is necessary for accurate, efficient, and safe machine performance.

In general, acceleration and deceleration times need to be increased when attempting to operate a machine with high gain and minimum following error. If not, the machine may frequently shut down due to feedback errors when the axes start and stop.

Calculating your Linear Acc./Dec Ramp
Assuming your initial gain of position loop has been selected, configuration of the Linear Acc./Dec ramp is a straight forward calculation.

The position loop gain sets the overall response time for the servo. For a given gain, the time required to go from one steady state velocity to another steady state velocity is a constant:

Position Loop Gain	Approximate Response Time
1/2 IPM/mil	240 msec
1 IPM/mil	120 msec
2 IPM/mil	60 msec
3 IPM/mil	30 msec
4 IPM/mil	15 msec

The approximate Response Time (in the table on page 10-2) is inversely proportional to the position loop gain. You can calculate it if it does not appear in the table above by applying the proportion. Simply divide 120 msec by your position loop gain.

$$\text{Approximate Response Time} = \frac{\text{120 msec.}}{\text{Position Loop Gain}}$$

Applying an Acc./Dec ramp that is steeper than the actual response curve of the servo is the same as applying a step function with no Acc./Dec ramp. For the Acc./Dec ramp to have any effect on the operation of the servo the ramp must be below the response curve of the servo. If you selected Closed Loop for the position loop type, the approximate response time of your servo, and the maximum contouring speed, the correct Acc./Dec ramp can be calculated directly:

$$\text{Linear Acc./Dec Ramp} = \frac{\text{Maximum Contouring Speed (IPM)}}{\text{Approximate Response Time (msec)}}$$

Important: If the selected position loop type is Zero Following Error, you must use the Rapid Traverse instead of Maximum Contouring Speed.

This calculation results in a ramp expressed in inches per minute per millisecond. Apply the appropriate conversion factors to convert it into the units you have selected to enter via AMP. This ramp results in a linear response throughout the contouring range for accurate contouring moves. It will also allow the response to curve at speeds above the contouring range.

Positioning ACC/DEC Mode

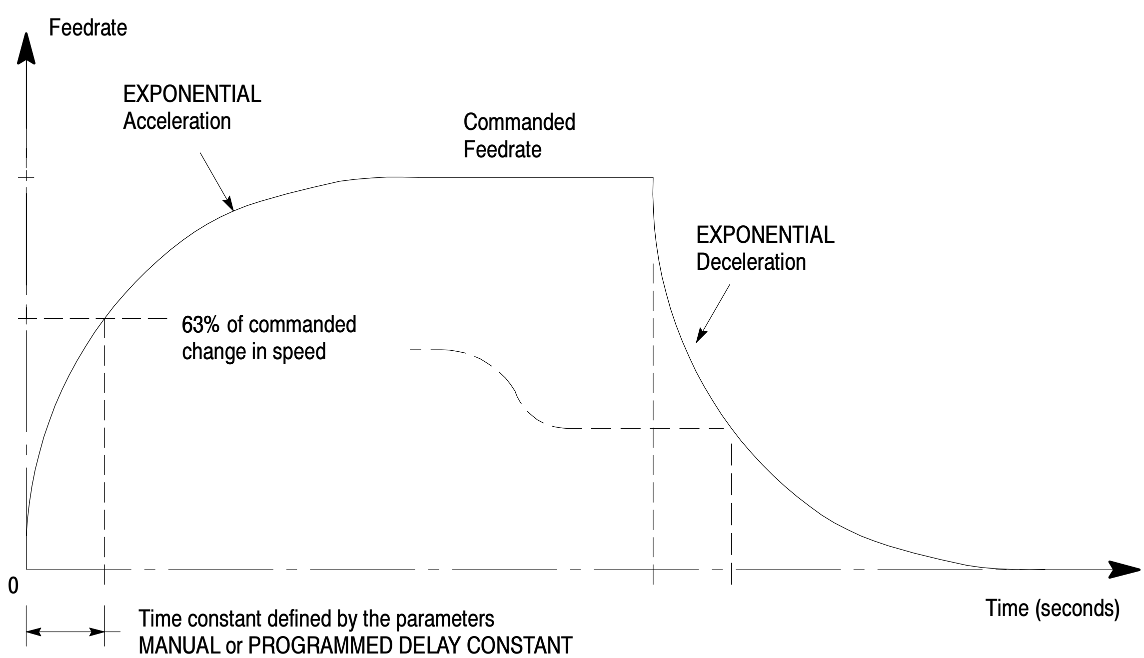
Function

Use this parameter in conjunction with Acc./Dec G-codes (G47, 47.1, and 47.9) to select the type of acceleration or deceleration to use when in positioning (G00) and exact stop modes. It does not affect the type of acceleration or deceleration used during axis jogging or contouring or cutting moves (G01, G02, G03).

This is not a per-axis parameter. The Acc./Dec mode selected here is applied to all axes.

- **Exponential Acc./Dec –** If exponential Acc./Dec is selected for this parameter, the axis velocity (during positioning moves) increases or decreases at an exponential rate determined by the **Programmed Delay Constant** parameter. If Exponential is selected, then Exponential Acc./Dec will be performed during positioning, regardless of the active Acc./Dec mode (G47.x).

Figure 10.1
Exponential Acc./Dec



Important: Rapid moves that are made for a lathe control type when performing a compound turning routine do not use this parameter. Rapid moves for compound turning routines always use exponential Acc./Dec regardless of the setting of this parameter. Be aware that if you choose linear Acc./Dec for this parameter, and if you are going to use compound turning routines, you must still configure the axes to allow rapid moves in exponential Acc/Dec. This requires that the setting of the parameter for the Programmed Delay Constant must be valid with the feedrate chosen as the rapid feedrate for each axis. If improperly configured, the control may generate a SERVO AMPLIFIER FAULT error when a rapid move is executed in a compound turning routine.

- **Linear or S-Curve Acc./Dec as Per G-Code -** Use this parameter to select the type of acc./dec mode (Linear or S-Curve) you want to perform. Activating **G47** enables Linear Acc/Dec. Activating **G47.1** enables S-Curve Acc./Dec. S-Curve Acc./Dec is only applicable to positioning moves in Exact Stop mode.

Figure 10.2
Linear Acc./Dec

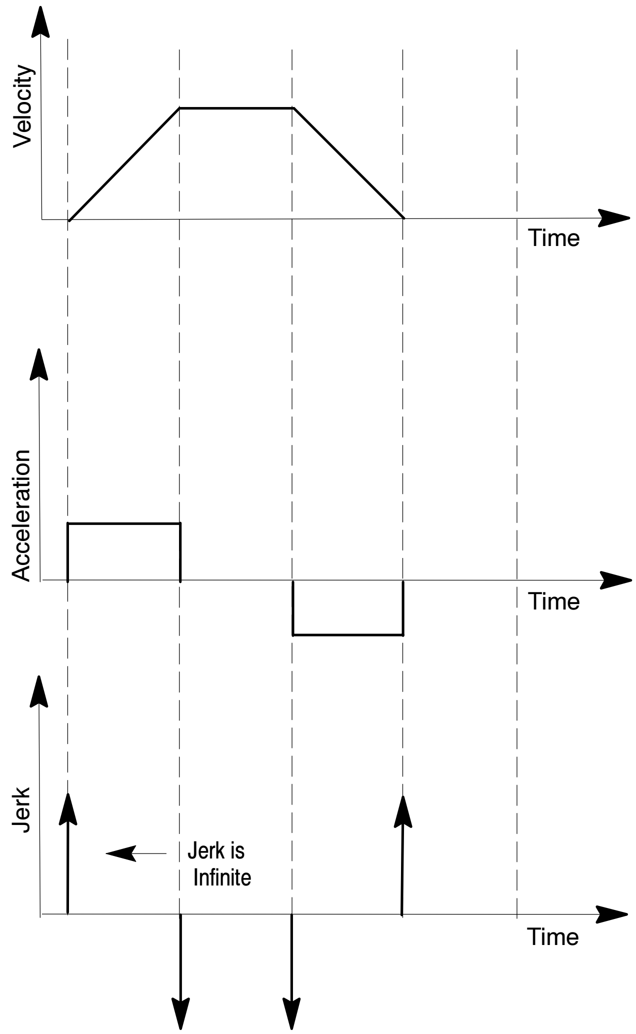
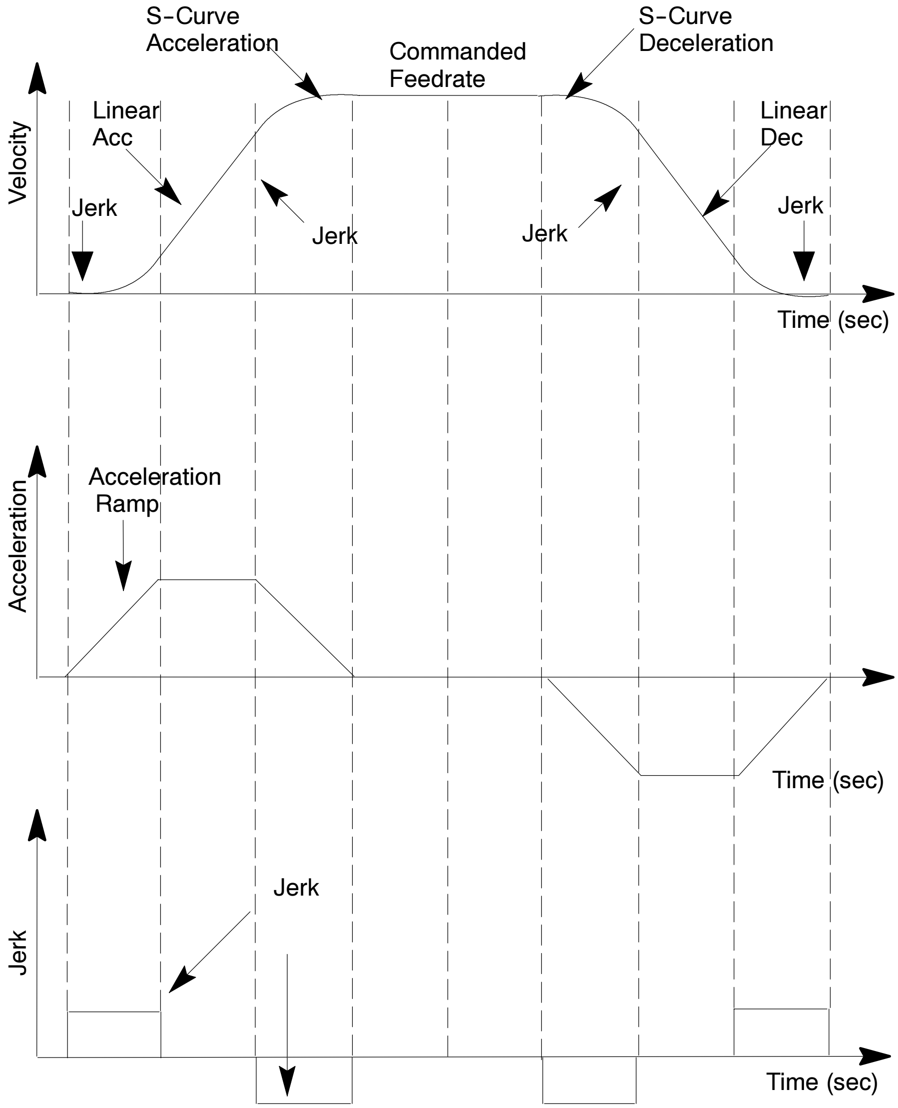


Figure 10.3
S-Curve Acc./Dec



Refer to the following table to determine the type of acc./dec mode that will be used in positioning (G00) or exact stop (G61) mode when this parameter is set as linear or S-Curve:

If you want this acc./dec type:	Select this acc./dec mode:
Linear	G47
S-Curve for positioning and exact stop	G47.1
Acc./Dec disabled	G47.9

Axis	Parameter Number
	Single Process
All	[402]

Range

Selection	Result
(a)	Exponential
(b)	Linear or S-Curve as per G-code

Notes

This is a global parameter. The value set here applies to all axes.

Axis jogging moves use either linear or exponential Acc./Dec as defined in AMP via Manual Acc./Dec (refer to page 10-18 for more information about Manual Acc./Dec). Axis cutting moves use Linear or S-Curve Acc/Dec.

Velocity Step for ACC./DEC

Function

If two adjacent motion blocks in a program require different axis feedrates, Acc./Dec may be necessary to smooth the speed transition. The value entered for this parameter specifies the maximum change in feedrate that will be executed without any Acc./Dec activated.

For example, entering 50 mmpm here, means that any feedrate changes equal to or less than 50 mmpm will be executed as a step, with no Acc/Dec. Acc./dec ramping will be used for any feedrate transitions greater than 50 mmpm.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: Entering too large a value here may result in damage to the part, tooling, or machine. Large changes in feedrate should always be made gradually when using Acc/Dec.

For most axis configurations, this value should be less than or equal to the value entered for the **Acc./Dec Ramp** parameter.

Axis	Parameter Number
(1)	[1204]
(2)	[2204]
(3)	[3204]
(4)	[4204]
(5)	[5204]
(6)	[6204]
(7)	[7204]
(8)	[8204]

Range

0.00000 to 10160.00004 mmpm

or

0.00000 to 400.00000 ipm.

Notes

This parameter must be set independently for each axis.

Linear Acceleration Ramp

Function

Use this parameter to select the acceleration rate for each axis during Linear Acc./Dec Mode (G47). A 0.0 value in this mode disables Acc./Dec for the applicable axis. This ramp value is used per axis for manual motion when the **Manual Acc./Dec** parameter is set to Linear.

Important: If you attempt to run the axis with a high gain (refer to the **Initial Gain** parameter in chapter 7), higher values are generally required for this parameter to avoid getting the "EXCESS FOLLOWING ERROR" message.

Axis	Parameter Number
(1)	[1210]
(2)	[2210]
(3)	[3210]
(4)	[4210]
(5)	[5210]
(6)	[6210]
(7)	[7210]
(8)	[8210]

Range

0.00000 to 20,000.00000 mm/sec/sec

or

0.00000 to 787.4015 inch/sec/sec

or

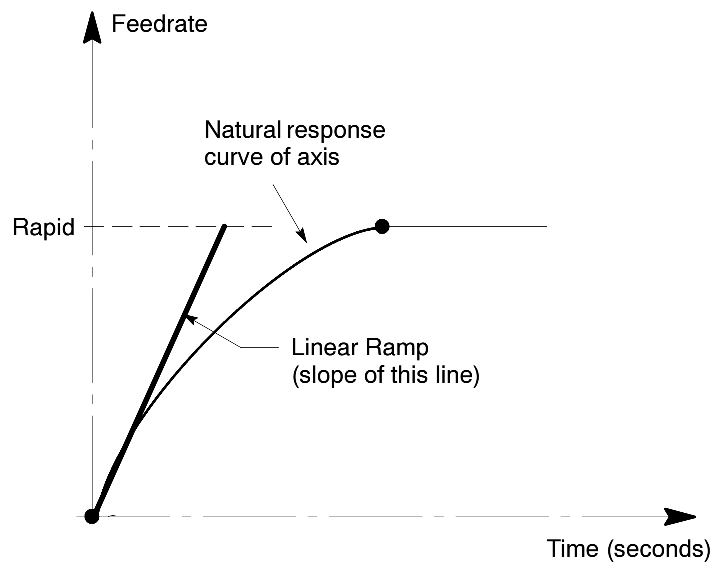
0.00000 to 20,000.00000 deg/sec/sec

Notes

This parameter must be set independently for each axis.

Important: If the **Type of Feedback** parameter (chapter 7) is set for "ZFE Closed Loop" and the Feed Forward Percentage is set relatively high, then it is important that the axis acceleration and deceleration ramping closely match the "natural response" of the axis.

Figure 10.4
Matching Linear Acc. Ramp to the Natural Response of the Axis



Linear Deceleration Ramp

Function

Use this parameter to select the deceleration rate for each axis during Linear Acc./Dec Mode (G47). A 0.0 value in this parameter forces the system to replace this value with the value set for **Linear Acceleration Ramp**.

Important: If you attempt to run the axis with a high gain (refer to the **Initial Gain** parameter in chapter 7), higher values are generally required for this parameter to avoid getting the "EXCESS FOLLOWING ERROR" message.

Axis	Parameter Number
(1)	[1211]
(2)	[2211]
(3)	[3211]
(4)	[4211]
(5)	[5211]
(6)	[6211]
(7)	[7211]
(8)	[8211]

Range

0.00000 to 20,000.00000 mm/sec/sec

or

0.00000 to 787.4015 inch/sec/sec

or

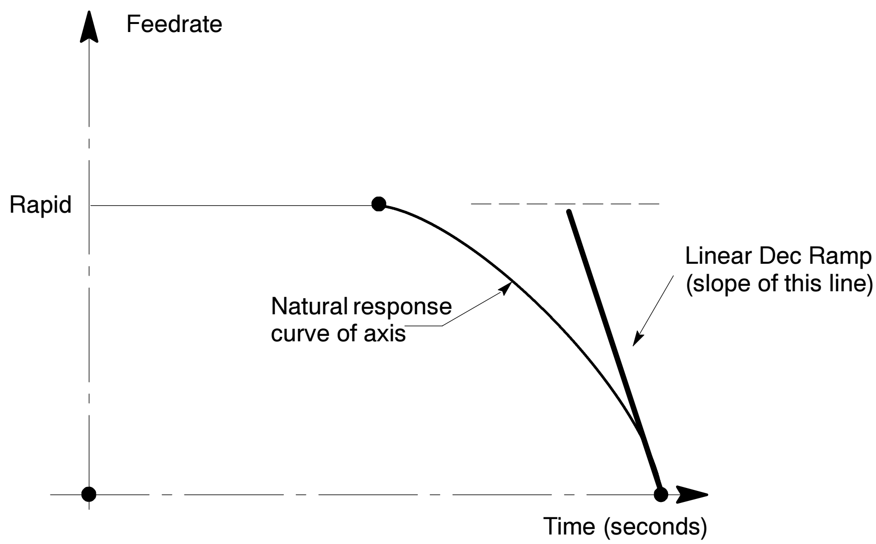
0.00000 to 20,000.00000 deg/sec/sec

Notes

This parameter must be set independently for each axis.

Important: If the **Type of Feedback** parameter (chapter 7) is set for "ZFE Closed Loop" and the Feed Forward Percentage is set relatively high, then it is important that the axis acceleration and deceleration ramping closely match the "natural response" of the axis.

Figure 10.5
Matching Linear Dec Ramp to the Natural Response of the Axis



S- Curve Acceleration Ramp

Function

Use this parameter while S-Curve Acc./Dec (G47.1) is active to select the desired acceleration rate. A 0.0 value in this parameter disables Acc./Dec for the applicable axis.

If a large value is entered here, the axis accelerates rapidly. A small value results in a more gradual acceleration.

Important: If you attempt to run the axis with a high gain (refer to the **Initial Gain** parameter in chapter 7), higher values are generally required for this parameter to avoid getting the "EXCESS FOLLOWING ERROR" message.

Important: This parameter is not for use with manual jogged motion.

Axis	Parameter Number
(1)	[1212]
(2)	[2212]
(3)	[3212]
(4)	[4212]
(5)	[5212]
(6)	[6212]
(7)	[7212]
(8)	[8212]

Range

0.00000 to 20,000.00000 mm/sec/sec

or

0.00000 to 20,000.00000 deg/sec/sec

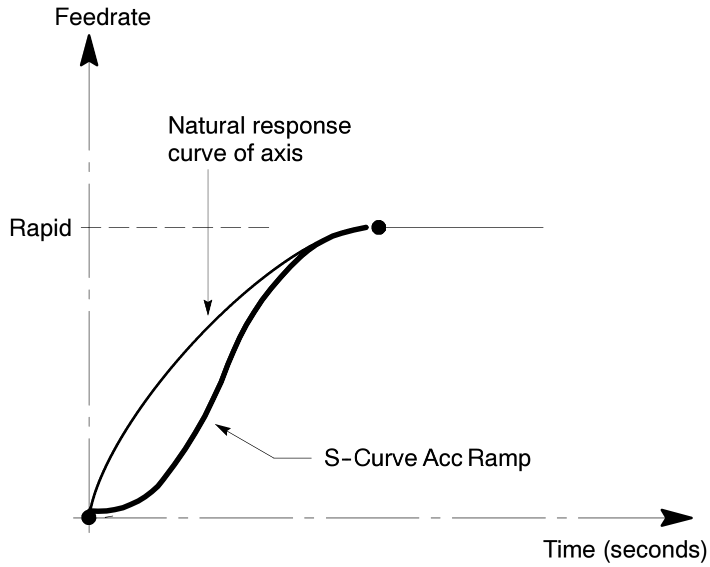
Notes

This parameter must be set independently for each axis.

Important: If the **Servo Position Loop Type** parameter (chapter 7) is set for "ZFE Closed Loop" and the Feed Forward Percentage is set relatively high, then it is important that the axis acceleration and deceleration ramping closely match the "natural response" of the axis.

This is a simplified procedure for determining the **Acc. Ramp** value in cases where it is critical to axis performance:

Figure 10.6
Matching S-Curve Acc. Ramp to the Natural Response of the Axis



S-Curve Deceleration Ramp

Function

This parameter allows you to select the deceleration rate for each axis during S-curve Acc./Dec Mode (G47.1). A 0.0 value in this parameter forces the system to replace this value with the value set for **S-Curve Acceleration Ramp**.

If a large value is entered here, the axis decelerates rapidly. A small value results in a more gradual deceleration.

Important: If you attempt to run the axis with a high gain (refer to the **Initial Gain** parameter in chapter 7), higher values are generally required for this parameter to avoid getting the "EXCESS FOLLOWING ERROR" message.

Important: This parameter is not for use with manual jogged motion.

Axis	Parameter Number
(1)	[1213]
(2)	[2213]
(3)	[3213]
(4)	[4213]
(5)	[5213]
(6)	[6213]
(7)	[7213]
(8)	[8213]

Range

0.00000 to 20,000.00000 mm/sec/sec

or

0.00000 to 20,000.00000 deg/sec/sec

Notes

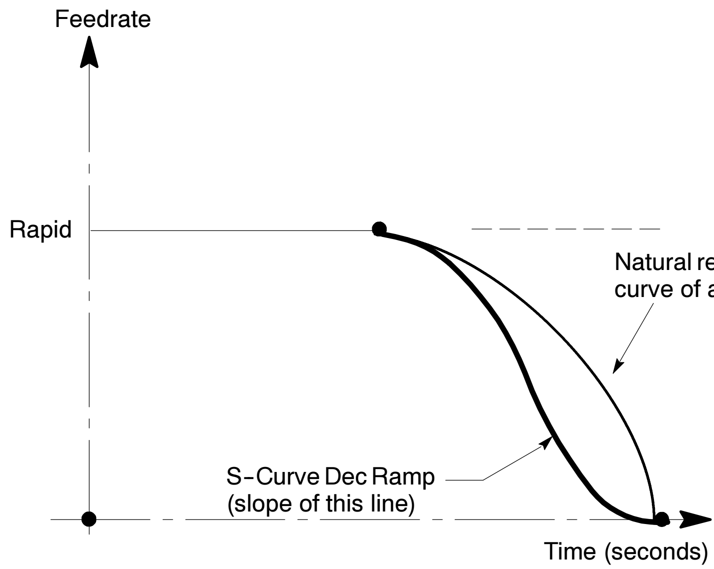
This parameter must be set independently for each axis.

Important: If the **Servo Position Loop Type** parameter (chapter 7) is set for "ZFE Closed Loop" and the Feed Forward Percentage is set relatively high, then it is important that the axis acceleration and deceleration ramping closely match the "natural response" of the axis.

This is a simplified procedure for determining the **Dec Ramp** value in cases where it is critical to axis performance:

Important: The deceleration ramp value may be slightly higher than the acceleration rate due to friction.

Figure 10.7
Matching S-Curve Dec Ramp to the Natural Response of the Axis



Manual Delay Constant

Function

This parameter determines the acceleration or deceleration time for all manual moves when using exponential manual acc./dec mode. Refer to page 10-18 for more information about about Manual Acc./Dec Mode.

"Manual" moves refers to manual jog, incremental jog and handwheel jog moves. These moves always use exponential Acc./Dec when jog mode is exponential.

The value entered here indicates the number of "coarse iterations" that make up an Acc./Dec time constant. A coarse iteration is one of the control's software timing cycles and is equal to one system scan time. The Acc./Dec time constant represents the amount of time required to accelerate or decelerate 63% of the way to the commanded speed (refer to Figure 10.1).

For all practical purposes, acceleration or deceleration is completed after five time constants. Therefore, the total acceleration or deceleration time can be determined by multiplying the value entered here first by 20 milliseconds, and then by 5.

Axis	Parameter Number
(1)	[1000]
(2)	[2000]
(3)	[3000]
(4)	[4000]
(5)	[5000]
(6)	[6000]
(7)	[7000]
(8)	[8000]

Range

1 to 100

Notes

This parameter must be set independently for each axis.

This equation can be used to approximate axis speed at any time during exponential axis acceleration:

$$\text{Speed} = (S_{in}) + (S_{com} - S_{in})(1 - e^{-x})$$

where :

S_{in} = Initial axis speed

S_{com} = Commanded axis speed

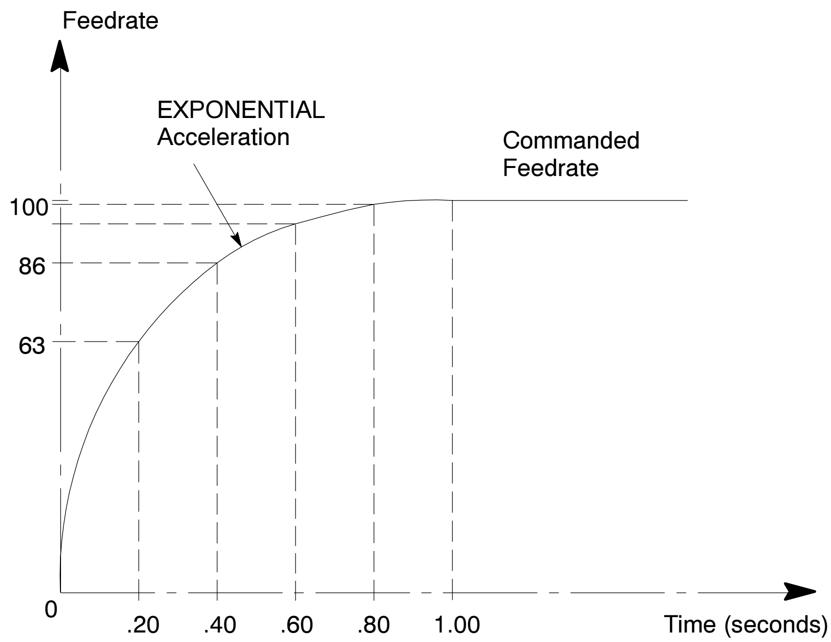
e = the natural antilogarithm

$$x = \frac{(1000/\text{system scan time})(\text{elapsed time in seconds})}{(\text{Manual Delay Constant})}$$

For example, Figure 10.8 shows axis speeds at various times during an exponential acceleration. This example assumes an initial speed of zero, a commanded speed of 100 ipm, a system scan time of 20 ms, and a **Manual Delay Constant** of 10.

After .2 seconds (one time constant), the axis is moving at a speed of 63 ipm. After .4 seconds, the axis is moving 86 ipm.

Figure 10.8
Exponential Acceleration Example



The following equation can be used to approximate axis speed at any time during exponential axis deceleration:

$$Speed = (S_{in}) - (S_{in} - S_{com})(1 - e^{-x})$$

where :

S_{in} = Initial axis speed

S_{com} = Commanded axis speed

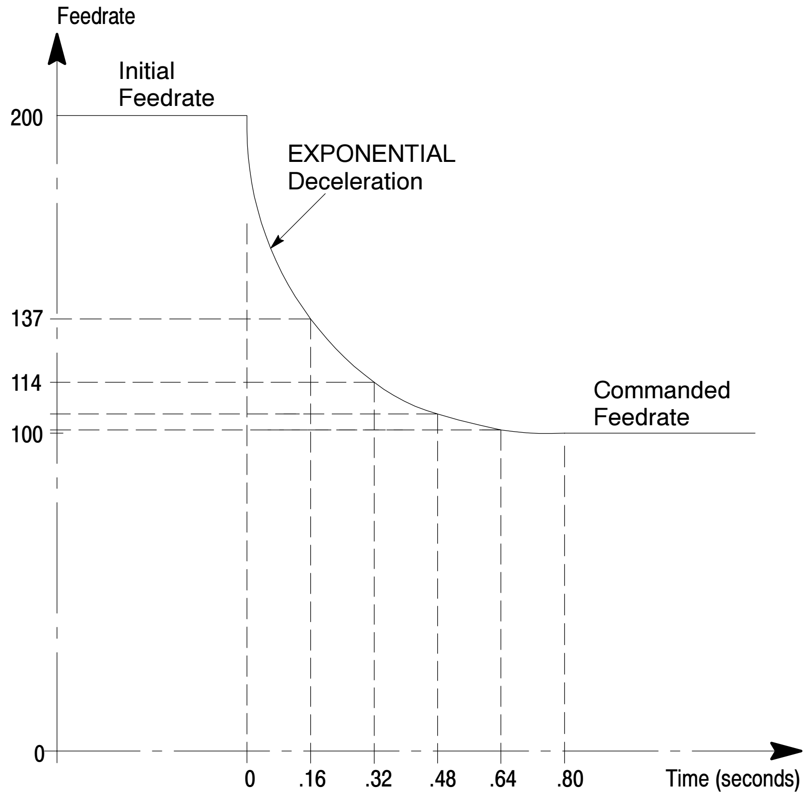
e = the natural antilogarithm

$$x = \frac{(1000/\text{system scan time})(\text{elapsed time in seconds})}{(\text{Manual Delay Constant})}$$

As an another example, Figure 10.9 shows axis speeds at various times during an exponential deceleration. This example assumes an initial speed of 200 ipm, a commanded speed of 100 ipm, a system scan time of 20 ms, and a **Manual Delay Constant** of 8.

After .16 seconds (one time constant), the axis has decelerated from 200 ipm to 137 ipm. After .32 seconds, the axis has decelerated to 114 ipm.

Figure 10.9
Exponential Deceleration Example



Programmed Delay Constant

Function

This parameter determines the total acceleration or deceleration time whenever Exponential Acc./Dec is being used during positioning moves. This includes MDI moves and programmed moves executed when G00 mode is active.

This parameter is significant only when the parameter **Positioning Acc./Dec Mode** is set for "Exponential."

The value entered here indicates the number of "coarse iterations" that make up an Acc./Dec time constant. A coarse iteration is one of the control's software timing cycles and is equal to one system scan time. The Acc./Dec time constant represents the amount of time required to accelerate or decelerate 63% of the way to the commanded speed (refer to Figure 10.1).

For all practical purposes, acceleration or deceleration is completed after five time constants. Therefore, the total acceleration or deceleration time can be determined by multiplying the value entered here first by the system scan time, and then by 5.

Refer to the notes for the parameter **Manual Delay Constant** for more application information and examples. The equations provided there can be applied here if the variable **Manual Delay Constant** is replaced with **Programmed Delay Constant.**

This is not a per axis parameter. The delay constant selected here is applied to all axes.

Axis	Parameter Number
	Single Process
All	[8]

Range

1 to 100

Notes

This is a global parameter. The value set here applies to all axes.

Manual ACC./DEC Mode

Function

This parameter allows you to select the Acc./Dec type (exponential or linear) when you are performing manual motion on your control (e.g., continuous and incremental jogging).

If you select "linear", the parameter uses ramps AMPed for programmed moves for each axis. The Acc./Dec motion applies to **continuous manual**, **continuous Logic axis mover**, and **incremental manual jogs**. For continuous, the jog feedrate will be accelerated from zero to the requested feedrate, when the request for a jog is initiated. The jog will start to decelerate at the AMPed ramp rate when the request to jog is released.

Selecting "Linear" causes your system to always use the downloaded AMP linear ramp settings, regardless of changes to your part program.

The linear acc./dec ramps for incremental manual jogs will be accelerated from zero to the requested feedrate when a jog is initiated.

For both continuous and incremental motion, there will be no acc./dec when an overtravel is hit; all commands will be stopped immediately. It will still move past the overtravel by the amount of the following error.

Axis	Parameter Number
	Single Process
All	[399]

Range

Selection	Result
(a)	Exponential
(b)	Linear

Notes

This is a global parameter; the value set here applies to all axes.

Manual Acc./Dec does not affect acceleration or deceleration during positioning (G00) or contouring moves (G01, G02, or G03).

Refer to the table below to determine the type of acc./dec performed for manual motion.

Motion Type	Always Uses Exponential Acc./Dec	Configurable in AMP by System Installer
Homing	✔	
Logic-axis Home Type	✔	
Manual continuous motion*		✔
Manual incremental motion*		✔
Logic axis mover (Types 1 through 6 are continuous, incremental, and absolute)*		✔

Axis Jerk

Function

Use this parameter to select the maximum jerk for each axis used in a G48.5 block. This parameter is only used when in S-Curve Acc./Dec (G47.1) mode and when in one of the following modes: Positioning (G00) and Exact Stop (G09/nonmodal and G61/modal). Programming G48.5 values that are out of range generates the decode error, "RAMP/JERK OUT OF RANGE".

Axis Jerk is also used at PTO to allocate the per axis S-Curve filter size buffer, with a minimum size of 2 iterations (0.04 sec) and a maximum size of 60 iterations (1.20 sec).

Axis	Parameter Number
(1)	[1214]
(2)	[2214]
(3)	[3214]
(4)	[4214]
(5)	[5214]
(6)	[6214]
(7)	[7214]
(8)	[8214]

Range

0.00000 to 1,000,000.0000 mm/sec/sec/sec

or

0.00000 to 1,000,000.0000 deg/sec/sec/sec

Notes

This parameter must be set independently for each axis.

Use this equation to calculate the Axis Jerk value:

$$T = \frac{R}{J} \quad \text{and}$$

$$J_{Max} = \frac{R}{T_{Min}} \quad \text{where JMax = 2 scans (0.04 s) and}$$

$$J_{Min} = \frac{R}{T_{Max}} \quad \text{where } J_{Min} = 60 \text{ scans (1.20 s)}$$

Where :	Is :
J	Axis jerk value
R	Ramp value
T	S-Curve time per system scan rate[1,2]

[1]Time is in seconds.
[2]Maximum S-Curve time = 1.20 s (60 scans at 20 ms scan rate)
Minimum S-Curve time = 0.04 s (2 scans at 20 ms scan rate)

Refer to the table below for examples of resulting jerk values, given **R**, **T**, and the system scan rate:

Ramp Value	S-Curve Time per System Scan Rate	Coarse Foreground Scan Time	Axis Jerk Value
1,000 ipm/s	0.04 s	20 ms	25,000 ipm/s/s (max)
	1.20 s		833 ipm/s/s (min.)
	0.16 s		6,250 ipm/s/s (example per machine parameters)
2,500 ipm/s	0.028 s	14 ms	89,286 ipm/s/s (max)
	0.84 s		2,976 ipm/s/s (min.)
	0.112 s		22,321 ipm/s/s (example per machine parameters)

Minimum Programmable Jerk

Function

Use this parameter when in S-Curve Acc./Dec (G47.1) mode to specify the minimum allowable jerk value that the part program can set for each axis used in a G48.5 block. Programming G48.5 values that are out of range generates the decode error, "RAMP/JERK OUT OF RANGE".

Minimum Programmable Jerk is also used at PTO to allocate the S-Curve filter size buffer, with a maximum size of 60 iterations (1.20 sec) and a minimum size of 2 iterations (0.04 sec).

To calculate the Minimum Programmable Jerk value:

1. Determine the maximum acceleration value and the appropriate units (e.g., g) using **S-Curve Acceleration** parameter. Refer to page 10-11 for more information about S-Curve acceleration.
2. Determine the coarse iteration value in milliseconds using the **System Scan Time** parameter. Refer to page 33-15 for more information about system scan time.
3. Determine the maximum linear velocity (i.e., rapid feedrate) and the appropriate units (e.g., IPM) using the **Rapid Feedrate for Positioning** parameter. Refer to page 9-1 for more information about rapid feedrate for positioning.
4. Enter the values from steps 1 to 3 into the spreadsheet located on the MC Electronic Bulletin Board (440-646-3963). If you do not have access to the Bulletin Board, the system will prompt you with further instructions.

   Once these values are entered, Excel will return the minimum and maximum programmable jerk values. To manually calculate the minimum programmable jerk value, refer to page 10-22.

Axis	Parameter Number
(1)	[1215]
(2)	[2215]
(3)	[3215]
(4)	[4215]
(5)	[5215]
(6)	[6215]
(7)	[7215]
(8)	[8215]

Range

0.00000 to 1,000,000.0000 mm/sec/sec/sec

or

0.00000 to 1,000,000.0000 deg/sec/sec/sec

Notes

This parameter must be set independently for each axis.

Changing the Rapid Feedrate or Acceleration Ramp affects the Minimum Programmable Jerk value.

To manually calculate the Minimum Programmable Jerk value, use this equation:

Use this equation to calculate the Axis Jerk value:

If $T = \frac{R}{J}$ then

$J_{Max} = \frac{R}{T_{Min}}$ where JMax = 2 scans (0.04 s) and

$J_{Min} = \frac{R}{T_{Max}}$ where J_{Min} = 60 scans (1.20 s)

Where :	Is :
J	Axis jerk value
R	Ramp value
T	S-Curve time per system scan rate[1,2]

[1]Time is in seconds.
[2]Maximum S-Curve time = 1.20 s (2 scans at 20 ms scan rate)
Minimum S-Curve time = 0.04 s (60 scans at 20 ms scan rate)

Refer to the table below for examples of resulting jerk values, given **R**, and **T**, and the system scan rate:

Ramp Value	S-Curve Time per System Scan Rate	Coarse Foreground Scan Time	Axis Jerk Value
1,000 ipm/s	0.04 s	20 ms	25,000 ipm/s/s (max)
	1.20 s		833 ipm/s/s (min.)
	0.16 s		6,250 ipm/s/s (example per machine parameters)
2,500 ipm/s	0.028 s	14 ms	89,286 ipm/s/s (max)
	0.84 s		2,976 ipm/s/s (min.)
	0.112 s		22,321 ipm/s/s (example per machine parameters)

END OF CHAPTER

Constant Surface Speed

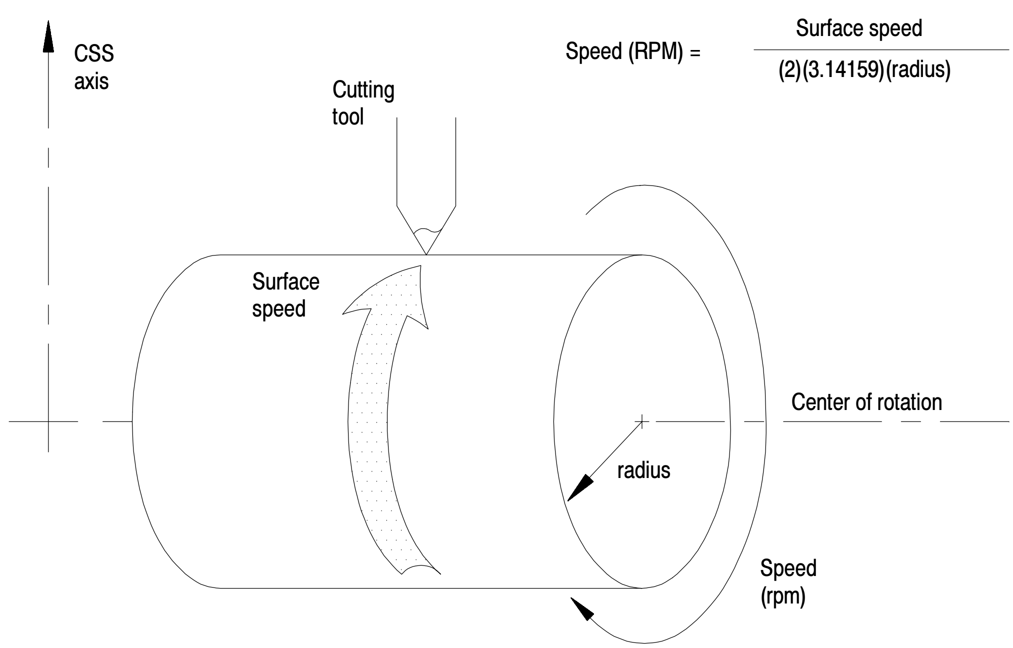
Chapter Overview

As a cutting tool's distance from the center of a rotating part changes, the surface speed also changes. Use the constant surface speed feature (CSS) to maintain a tool's cutting speed on a rotating part regardless of the diameter of the part. Refer to your programming and operation manual for more information.

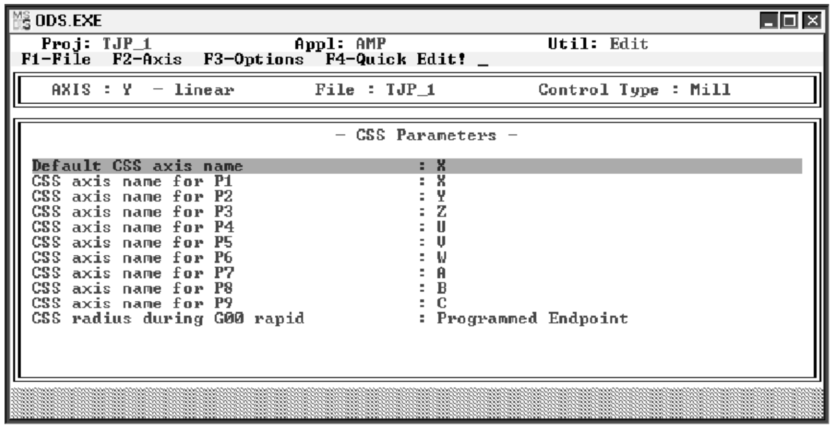
The CSS feature is activated in a program by a G96 code. A surface speed is programmed with an S-word. The CSS feature maintains a constant surface speed by varying the spindle RPM as the cutting tool's distance from the center of rotation changes.

CSS helps to produce a uniform machine finish for a rotating part.

Figure 11.1
Constant Surface Speed



When you select the "CSS Parameters" group from the main menu, the workstation displays this screen:



Your screens may differ slightly, depending on your application type.

CSS Parameters

Use these parameters to configure the CSS feature:

Parameter:	Page:
Default CSS axis name	11-2
CSS axis name for P1 - P9	11-3
CSS radius during G00 rapid	11-4
Using synchronized spindles with CSS	11-4

Default CSS Axis Name

Function

Use this parameter to establish the default CSS axis. Typically you would select an axis that is perpendicular to the center line of the rotating part. The axis named here is used if a programmable CSS axis (P1-P9) is not programmed in the G96 block.

If an axis perpendicular to the line of a rotating part is selected here, motion along that axis would change the cutting tool's distance from the part's center of rotation. The control then would alter the spindle RPM to maintain the programmed surface speed.

Axis	Parameter Number
	Single Process
All	[413]

Range

Selection	Result	Selection	Result
(a)	A	(h)	Y
(b)	B	(i)	Z
(c)	C	(j)	$B
(d)	U	(k)	$C
(e)	V	(l)	$X
(f)	W	(m)	$Y
(g)	X	(n)	$Z

Notes

This is a global parameter. The value set here applies to all axes.

P1-P9 Constant Surface Speed Axis Name

Function

Use these parameters to establish the programmable CSS axes. These axes become the CSS axis when their corresponding P-word is programmed in a G96 block. This provides a quick way to change the CSS axis from the default axis to a different axis. Refer to your programming and operation manual for more information.

Axis	Parameter Number
	Single Process
CSS axis name for P1	[414]
CSS axis name for P2	[415]
CSS axis name for P3	[416]
CSS axis name for P4	[417]
CSS axis name for P5	[418]
CSS axis name for P6	[419]
CSS axis name for P7	[420]
CSS axis name for P8	[421]

Range

Selection	Result	Selection	Result
(a)	A	(h)	Y
(b)	B	(i)	Z
(c)	C	(j)	$B
(d)	U	(k)	$C
(e)	V	(l)	$X
(f)	W	(m)	$Y
(g)	X	(n)	$Z

Notes

This is a global parameter. The value set here applies to all axes.

CSS Radius During G00 Rapid

Function

When a rapid move (or many consecutive rapid moves) takes place if CSS is active, undesirable results may occur when the rotating part attempts to increase or decrease RPM rapidly as the tool moves. This parameter allows the option of the spindle speed being calculated based on the programmed end-point instead of the continuously changing position of the CSS axis.

Programmed End Point – When you select Programmed End Point, this causes any RPM changes to be calculated based on the end-point of a rapid move.

Momentary Position – When you select Momentary Position, this causes any RPM changes that take place for rapid moves to be executed as the axis moves. This is the same as though the move was a regular axis move using the programmed feedrate.

Axis	Parameter Number
	Single Process
All	[854]

Range

Selection	Result
(a)	Programmed End Point
(b)	Momentary Position

Notes

This is a global parameter. The value set here applies to all axes.

This parameter has no effect on moves that are executed at the programmed feedrate.

Using Synchronized Spindles with CSS

Any changes that occur as a result of constant surface speed are delayed while the spindles attempt to synchronize, until synchronization is achieved. For more information about synchronized spindles, refer to your operation and programming manual.

END OF CHAPTER

Spindle 1 Parameters

Chapter Overview

Use these parameters to define spindle operation for spindle 1. These parameters must be set separately for each spindle.

These parameters include:

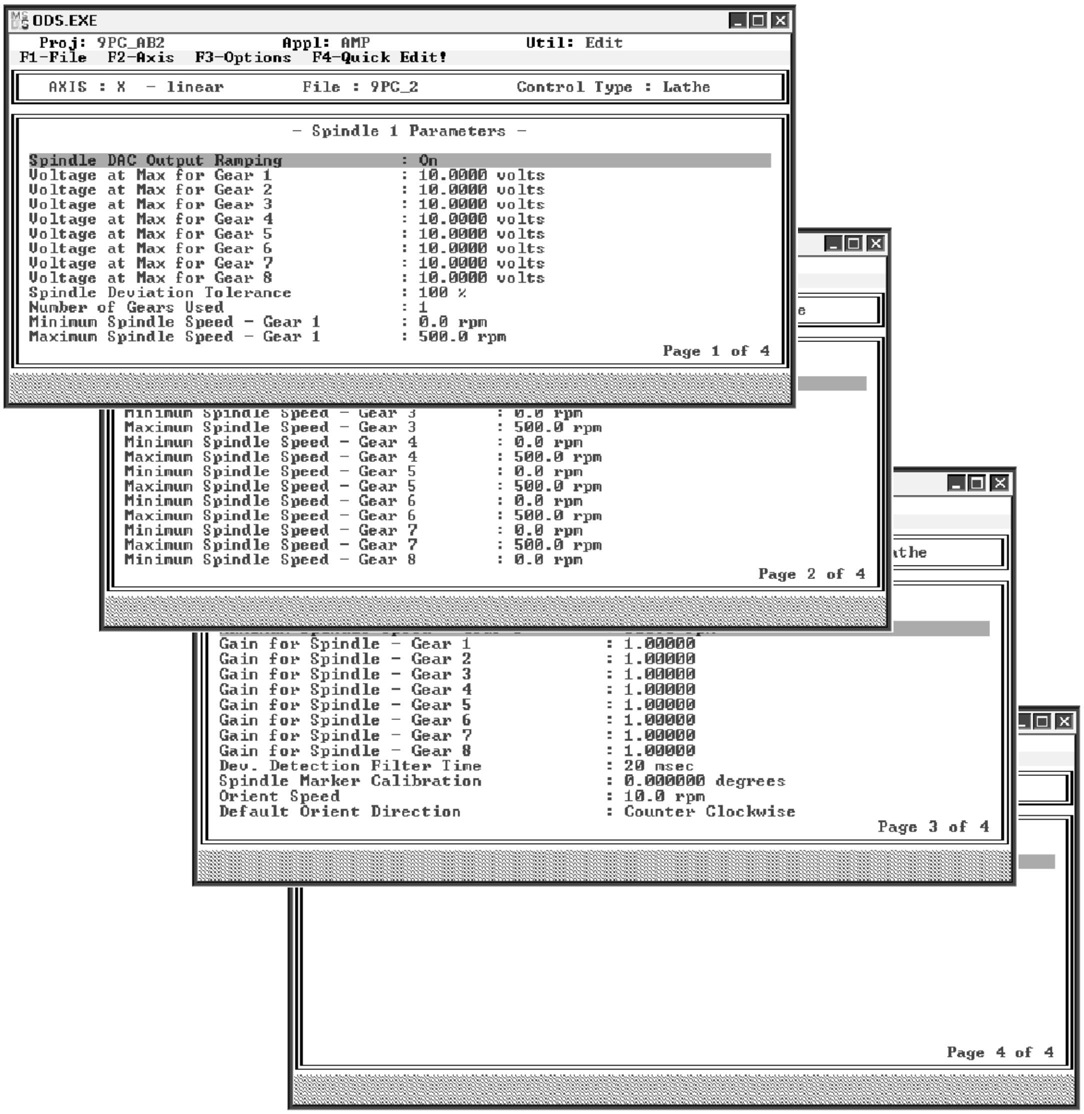
- Spindle DAC Output Ramping
- Voltage at Max for Gear 1-8
- Spindle Deviation Tolerance
- Number of Gears Used
- Minimum Spindle Speed - Gear 1-8
- Maximum Spindle Speed - Gear 1-8
- Dev. Detection Filter Time
- Spindle Marker Calibration
- Orient Speed
- Default Orient Direction
- Default Orient Angle
- Orient Inposition Band

Important: Note that if the spindle is to also be used with virtual C or Cylindrical Interpolation you must set the parameters in chapter 26.

Only the controlling spindle may be used with virtual C or Cylindrical Interpolation. The default controlling spindle is the spindle that is assigned first or the spindle selected with a G12 code.

Important: Many of the parameters in this section are not used when using a 1394 drive/1326 motor combination as a spindle. Refer to page 7–79 for details on this special spindle configuration.

To edit the spindle parameters, select “Spindle 1 Parameters” from the first page of the main menu screen. The workstation displays these four screens:



DAC Voltage and Spindle Gear Parameters

These parameters deal with the DAC output voltage and spindle gears. “DAC” stands for “Digital to Analog Converter.” This is in reference to analog outputs from the 1394 CNC serial drive. The output port used is determined by the axis configured as a spindle with the F2 Axis option and the servo parameter “Output Port Number.”

For direct SERCOS connected spindles, these values must be set with the spindle drive configuration tool.

Important: If the spindle has multiple gear ranges and encoder feedback is being used, the encoder must have a 1 : 1 rotation ratio to the spindle. The encoder must make one revolution per revolution of the spindle and only provide 1 marker pulse per revolution, regardless of which gear is selected.

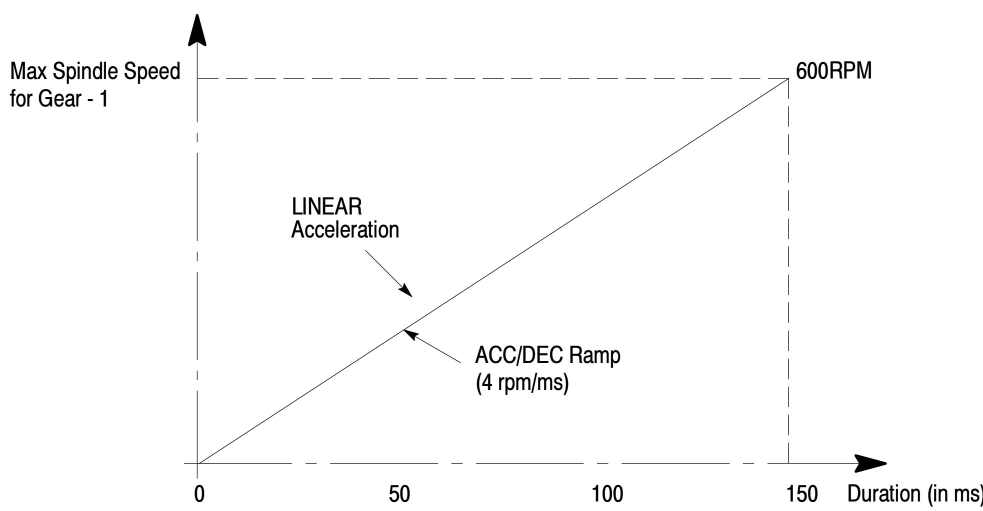
This DAC output provides for an analog voltage output within the range of -10 to +10 V DC for use as a command signal to the spindle drive. This voltage range is limited by the parameter **Voltage at Max...** for each spindle gear.

Spindle DAC Output Ramping

Function

This parameter determines whether the DAC output is a gradual voltage ramp or a voltage step. For this parameter:

If you select:	then:
ON	any change in DAC output voltages is made as a ramp-up or ramp-down. The actual spindle ramp duration is selected using the **Dev. Detection Filter Time** parameter (if your system does not have the solid tapping option). The actual spindle ramp duration is selected using the **ACC. time for Spindle 1, Gear __** if your system has the solid tapping feature (this is used if your system has the solid tapping option and does not mean you must be using the solid tapping feature to get the ramp). The max speed for the ramp is determined from your setting of MAX. Spindle Speed-Gear __ . Thus a different ramp is created for each gear range.
OFF	changes in DAC output voltages are made immediately as one step



If your system:	Your ramp duration is set with:	Your RPM max over that duration is set with
has the solid tapping option	ACC. time for Spindle 1, Gear __ (in solid tapping parameter group)	Max Tapping Speed - Gear __ (in solid tapping parameter group)
does not have the solid tapping option	Spindle DAC Output Ramping (in spindle parameter group)	Max Spindle Speed - Gear __ (in spindle parameter group)

Spindle	Parameter Number
1	[803]

Range

Selection	Result
(a)	On
(b)	Off

Notes

The following notes assume you have chosen yes for this parameter enabling spindle ramping:

Open Loop Spindles
During normal open loop operation any change in commanded spindle speed is ramped. Any time that the commanded spindle speed changes from the current speed, the ramp is added to or subtracted from the current spindle speed each coarse iteration until the commanded spindle speed is reached. Spindle speed changes in both RPM mode and CSS mode are ramped. Changes in spindle direction (from logic or programming M03/M04) are also ramped.

Important: If using an open loop spindle orient (Shot Pin Orient), when the logic flag to complete the orient becomes true, the spindle command is zeroed immediately (no ramping occurs). For this type of orient your orient speed should be configured slow enough to allow this spindle stop without ramping. Any acceleration/deceleration of the spindle to reach the initial orient speed is done using the spindle ramp.

Closed-loop Spindle Orients
The initial acceleration or deceleration of the spindle to the orient speed is ramped. Once the orient velocity is reached, the spindle's position loop is closed and no further spindle ramping is performed. The AMPed orient speed should be slow enough that no problems occur when the spindle loop is closed. The gain for the spindle position loop should be chosen such that the natural exponential velocity profile of the position loop can be accommodated by the drive.

E-Stop
On transition into the Emergency Stop state, all spindle DAC output is zeroed immediately with no spindle ramping.

This parameter must be set for each spindle axis.

Voltage at Max for Gears 1-8

Function

This parameter is used to set the maximum DAC output voltage for each gear range of the spindle.

The maximum spindle RPM for each gear range is defined by the parameter **Maximum Spindle Speed**. It is necessary to determine what DAC output voltage is required to attain this maximum speed.

This information may be provided with the spindle drive. It can also be determined through testing. By connecting a battery box to the drive and a tachometer to the spindle, the command voltages required to produce specific spindle speeds in each gear can be plotted. These same "command voltages" are sent to the drive from the DAC output when the battery box is replaced with the proper cabling.

Spindle	Parameter	Parameter Number
1	Voltage at Max for Gear 1	[820]
	Voltage at Max for Gear 2	[821]
	Voltage at Max for Gear 3	[822]
	Voltage at Max for Gear 4	[823]
	Voltage at Max for Gear 5	[824]
	Voltage at Max for Gear 6	[825]
	Voltage at Max for Gear 7	[826]
	Voltage at Max for Gear 8	[827]

Range

-10.0000 to 10.0000 volts

Notes

This parameter must be set for each spindle axis.

Spindle Deviation Tolerance

Function

Specifies the percentage by which the actual spindle speed can deviate from the anticipated spindle speed before the control sets the logic flag "Spindle Speed Deviation Excessive" to "TRUE."

Corrective action by the control is limited to setting this logic flag. Therefore, this parameter requires proper logic programming to be effective. Refer to the notes below for more information about properly programming this logic flag.

Important: For this parameter to have significance (and not be ignored):

- the spindle must have position feedback
- the software option for spindle speed deviation detection must be installed

Spindle	Parameter Number
1	[861]

Range

0 to 100%

Notes

If the spindle has a feedback device, the control monitors the feedback and determines a velocity.

The "anticipated spindle speed" mentioned under FUNCTION above is the result of simulated feedback. The control simulates an acceleration / deceleration ramp for the spindle based on the parameter **Dev. Detection Filter Time**. The actual spindle velocity (from the spindle feedback) is then constantly compared to this simulated feedback.

For example, assume 20% is entered for this parameter and the commanded spindle speed (after considering spindle speed override) is 1000 RPM. As the spindle accelerates, its actual speed must remain within 20% of the speed anticipated through simulated feedback. If not, the control advises the logic program by setting the logic flag "Spindle Speed Deviation Excessive" to "TRUE."

This flag also is set "TRUE" if the spindle speed remains below 800 RPM or above 1200 RPM after the spindle should have reached a steady state speed of 1000 RPM.

Generally the logic program is written to display a warning message whenever the control sets the Spindle Speed Deviation Excessive flag to "TRUE." The logic program may also include a timer that forces more aggressive action if the flag remains true for an extended period.

If the “Spindle Speed Deviation Excessive” flag remains “TRUE” for too long or appears too often, there are a number of changes that can prevent this:

1. The value for this parameter can be increased.
2. The parameter Dev. Detection Filter Time can be increased or decreased to more closely match the actual spindle speed transition time.
3. The logic program can enhance a decision on spindle speed deviation by using the “spindle up to speed” signal (if available) from the spindle drive.
4. Your logic program can be written to ignore the “Spindle Speed Deviation Excessive” flag during certain spindle speed transitions.

Refer to your *9/PC CNC Logic Reference Manual* for more information.

This parameter must be set for each spindle axis.

Number of Gears Used

Function

This parameter specifies the number of available spindle gear ranges for a specific machine application. The control allows for 8 spindle gear ranges.

Important: If the machine does not have multiple spindle gear ranges, enter 1 for this parameter.

Spindle	Parameter Number
1	[860]

Range

1 to 8

Notes

Actual spindle gear changing is implemented through the machine logic program. When logic sets the gear mode request flag to “automatic,” the control uses this parameter and the other AMP gear parameters to determine the correct gear for the commanded spindle speed. It then requests gear changes from logic as needed. Refer to your *9/PC CNC Logic Reference Manual* for more information.

This parameter must be set for each spindle axis.

Minimum and Maximum Spindle Speeds

Function

You must enter the lowest and highest spindle RPM for each gear range to optimize a machine's performance. Overlapping ranges are permitted. The maximum value for a specific gear must be greater than the minimum value for that gear.

Gear change operations are controlled by the logic program with gear change logic flags. Refer to your *9/PC CNC Logic Reference Manual* for more information.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: If the spindle is not capable of attaining the speed entered for a particular gear range, program execution may halt and/or an error message results (as determined by the control's logic program).

Values must be entered for all gear ranges specified by the parameter **Number of Gears Used**. If any of the gears used has no value or an illegal value entered here, the control assumes that no gears are available and never requests a gear change.

Spindle	Parameter	Parameter Number	Spindle	Parameter	Parameter Number
1	Min. Spindle Speed - Gear 1	[900]	1	Max. Spindle Speed - Gear 1	[910]
	Min. Spindle Speed - Gear 2	[901]		Max. Spindle Speed - Gear 2	[911]
	Min. Spindle Speed - Gear 3	[902]		Max. Spindle Speed - Gear 3	[912]
	Min. Spindle Speed - Gear 4	[903]		Max. Spindle Speed - Gear 4	[913]
	Min. Spindle Speed - Gear 5	[904]		Max. Spindle Speed - Gear 5	[914]
	Min. Spindle Speed - Gear 6	[905]		Max. Spindle Speed - Gear 6	[915]
	Min. Spindle Speed - Gear 7	[906]		Max. Spindle Speed - Gear 7	[916]
	Min. Spindle Speed - Gear 8	[907]		Max. Spindle Speed - Gear 8	[917]

Range

Min. Spindle Speed:	0 to Max. Spindle Speed (RPM)
Max. Spindle Speed:	Min. Spindle Speed to 99999.9(RPM)

Notes

This parameter must be set for each spindle axis.

Digital Spindle users must set this parameter (for gear 1 only) in addition to setting the servo parameter "Maximum Motor Speed" to define valid motor speeds.

Dev. Detection Filter Time

Function

If spindle DAC output ramping is off:
This parameter specifies the spindle acceleration / deceleration time to be used by the control only for simulated feedback. Simulated feedback is explained with the parameter **Spindle Deviation Tolerance**.

Since the value entered here is used in simulating any change in speed (regardless of whether it's from 0 to 20 RPM, from 10 to 2000 RPM, or from 500 to 2000 RPM), it is recommended that an average time be entered.

For example, if it takes 20 msec for a small change in spindle speed (such as from 0 to 5 RPM) and 160 msec for the maximum change in spindle speed (such as from 0 to 4000 RPM), a good value to enter here would be the average transition time, 90 msec.

Refer to the discussion for the parameter Spindle Deviation Tolerance.

If spindle DAC output ramping is on:
This parameter specifies:

- the spindle acceleration /declaration time to be used by the control for simulated feedback. Simulated feedback is explained with the parameter **Spindle Deviation Tolerance**. If you have the solid tapping feature option, enter a value for this parameter that closely matches the spindle acc./dec curve established by the solid tapping parameters (refer to page 12-3).
- if your system does not have the solid tapping feature this parameter specifies the duration of the spindle ramp (refer to page 12-3). This is also the same value the control uses for simulated feedback.

Spindle	Parameter Number
1	[70]

Range

20 to 400 msec

Notes

Simulated feedback allows the control to perform “feed per revolution” moves on machines that do not have position feedback from the spindle.

If the spindle has position feedback and the software option for spindle speed deviation detection has been installed, simulated feedback is also used for spindle speed deviation detection.

This parameter must be set for each spindle axis.

Spindle Orienting Parameters

These parameters are relevant only if your spindle provides position feedback and is capable of orienting to a specific position.

Important: It is possible to perform coarse spindle orients without position feedback through proper logic programming and the use of mechanical switches mounted on the spindle mechanism. These parameters are not relevant for such an application.

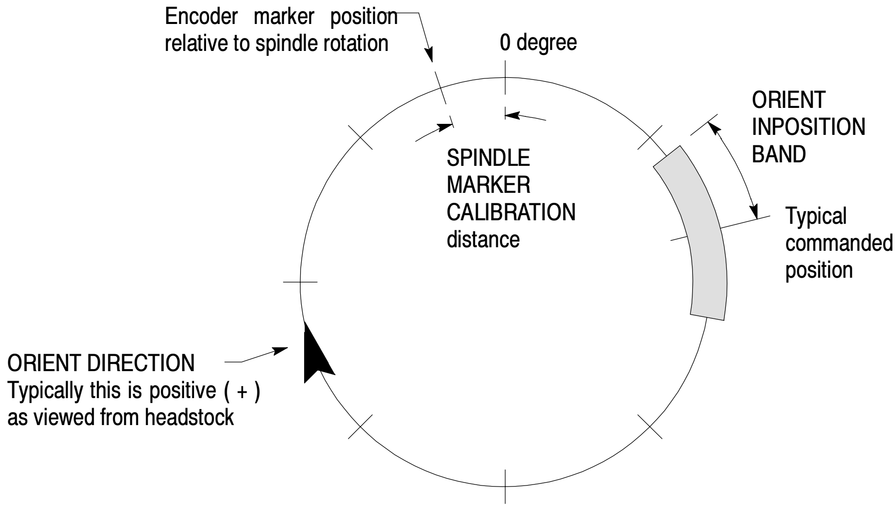
The spindle orient feature (Figure 12.1) lets the control stop the spindle at either a predefined angle set in AMP or a programmed angle. The spindle orient feature is discussed in the Lathe and Mill Operation and Programming Manuals.

Spindle Reference

Closed loop spindle orients require a marker be found on the spindle encoder before the orient is performed. If the spindle has not found a marker before an orient is requested (M19), a homing operation will be initiated. (Note that spindle homing may also be forced when the spindle enters the virtual C mode as discussed in chapter 28.) Spindle homing consists of the spindle decelerating to the orient spindle RPM, and finding the encoder marker (or null position). Once this position is found, the spindle moves on to the spindle marker calibration point that determines the location of the zero point of the spindle relative to the encoder marker position. After this location is reached, the spindle moves to the location requested in the M19 block. This homing operation is typically only required once after every power cycle.

Important: Note that the use of a spindle with feedback must have only one encoder marker per revolution of the spindle. If more than one encoder marker is present, inconsistent homing results depending on which marker is found first when the spindle homing operation is activated.

Figure 12.1
Parameters for Spindle Orient



Spindle Marker Calibration

Function

Use this parameter to define a precise mechanical zero position for the spindle without requiring a mechanical adjustment of the encoder mounting on the spindle shaft. When the spindle is homing, the control is looking for the location of the encoder marker. After this marker is found, the spindle continues on moving the angular distance specified with this parameter. When this move is completed, the control calls its new rotary position angle zero for all future orient and virtual C operations.

Specify the rotary distance (in degrees) from the encoder marker (or null) to the "zero" angle position desired for the spindle. This distance defines the location from the encoder null to the spindle orientation position that is to be defined as home (orientation zero). If the location of the encoder null position is the same as the position to be defined as spindle home, enter a value of zero for this parameter.

The sign associated with this parameter simply allows this angular distance to be specified as a positive or negative value. This sign has no effect on the actual direction of rotation used to position to the calibration angle. This direction of rotation is the same direction used to find the encoder marker. Note that positive angles are typically measured from the encoder null in the clockwise direction (M03 spindle direction).

Spindle	Parameter Number
1	[857]

Range

-360.000000 to 360.000000 degrees

Notes

This parameter can also be used to compensate for mechanical changes that otherwise would have relocated the zero position for the spindle. For example, the encoder orientation relative to the spindle position may change when repairing gears or gear belts. Consequently the distance that the axis moves from the encoder marker during homing may be different. That difference can be measured and entered as, or added to, the **Spindle Marker Calibration** value.

Important: Note that the use of a spindle with feedback must have only one encoder marker per revolution of the spindle. If more than one encoder marker is present, inconsistent homing results depending on which marker is found first when the spindle homing operation is activated.

This parameter must be set for each spindle axis.

Orient Speed

Function

This parameter determines the spindle speed in RPM that the control uses when positioning the spindle during a spindle orient (M19).

Refer to your programming and operation manuals for more information on spindle orient.

Spindle	Parameter Number
1	[858]

Range

0.0 to 99999.9 RPM

Notes

The orient speed is usually a very low value to assure that the spindle moves accurately to the orient position.

Executing an M19 may require you to force a gear change through logic to accommodate the spindle speed entered here. Automatic gear selection through logic will only select gears based on the last entered programmed spindle speed.

This parameter must be set for each spindle.

Default Orient Direction

Function

This parameter specifies the direction the spindle rotates during:

- any open loop spindle orient
- any closed loop spindle orient that takes place before the spindle has found a marker.

Spindle is:	Encoder Marker is:	The Orient is Performed in the Direction:
closed loop	already found	that would yield the shortest path to the orient position
closed loop	not yet found	as specified with this parameter
open loop	N/A	as specified with this parameter

If a closed loop spindle is already turning when an orient request is made, the spindle will decelerate (or accelerate) to the AMP defined orient spindle speed and orient to the requested location (this parameter has no affect on the spindle direction). Closed loop spindles only use this parameter if it has not yet seen a marker. Typically this is only after a power up cycle before the spindle has been moved. If there is no S word in the closed loop spindle orient block (M19), the spindle rotates to the position defined by the **Default Orient Angle** parameter.

Open loop spindles always use this parameter to determine spindle direction for an orient operation. The spindle will orient to the specified location in open loop orient position in the direction specified with this parameter (even if the spindle is already rotating in the opposite direction).

The direction entered here overrides the setting of the logic flag “Spindle Direction” (FW_SPNDI) during execution of an M19. FW_SPNDI usually checks the spindle direction switch or button on the MTB panel to determine spindle direction.

Refer to your programming and operation manuals for more complete information on spindle orient.

Spindle	Parameter Number
1	[851]

Range

Selection	Result
(a)	Clockwise
(b)	Counter Clockwise

Notes

This parameter must be set for each spindle.

The servo parameters Excess Error and Gain Break Point apply to closed loop spindles while they are orienting or tapping.

Default Orient Angle

Function

This parameter determines the absolute angle that the spindle rotates to during a spindle orient (M19) if no angle is programmed in the M19 block.

Refer to your programming and operation manuals for more information on spindle orient.

Spindle	Parameter Number
1	[862]

Range

0.000000 to 360.000000 degrees

Notes

If the M19 block is programmed with an S-word, this parameter is ignored and the spindle rotates to the programmed angle.

This parameter must be set for each spindle axis.

Orient Inposition Band

Function

This parameter determines a positioning range (in degrees) that the spindle must be within for the control to consider the spindle "in position."

When the spindle position is within this range, the control closes the positioning loop (even though spindles are always configured as "Open Loop") to move and hold the spindle at the commanded position.

The control also sets a logic flag indicating that the spindle is in position. Therefore, this parameter requires proper logic programming to be fully effective. Refer to the notes below for more information about properly programming this logic flag.

Important: If too small a value is entered here, the “Spindle Orientation Complete” flag in logic may never be set “TRUE.” Allowances must be made for positioning accuracy and inherent instabilities.

Refer to your *9/PC CNC Operation and Programming Manual* for more information on spindle orient.

Spindle	Parameter Number
1	[859]

Range

0.000000 to 360.000000 degrees

Notes

The value entered here sets the farthest distance (in either the positive or negative directions) that the spindle can be from its programmed destination to be considered “in position” for orient operations.

This does not affect the final positioning accuracy of the system. It simply determines how close to the commanded position the spindle must be before the logic flag “Spindle Orientation Complete” is set “TRUE.”

This parameter must be set for each spindle axis.

Gain for Spindle 1

Function

Enter the gain in units of RPM per .001 revolutions for this spindle in this gear.

Parameter	Parameter Number
Gain for Spindle 1, Gear 1	[777]
Gain for Spindle 1, Gear 2	[778]
Gain for Spindle 1, Gear 3	[779]
Gain for Spindle 1, Gear 4	[780]
Gain for Spindle 1, Gear 5	[781]
Gain for Spindle 1, Gear 6	[782]
Gain for Spindle 1, Gear 7	[783]
Gain for Spindle 1, Gear 8	[784]

Range

0.00000 to 100.00000

Notes

You must set this parameter for each gear you use of each spindle.

Use these parameters for spindle orient gain and gain during solid tapping. For solid tapping, the lower of the two gains (tapping axis and spindle) is applied to both motions.

You can use these parameters to shorten overall machining time by allowing the spindle to be oriented in any of 8 different gear ranges.

END OF CHAPTER

Chapter 13

Spindle 2 Parameters

Chapter Overview

Use these parameters to define spindle operation for Spindle 2. Each spindle is considered a separate axis. These parameters must be set separately for each spindle.

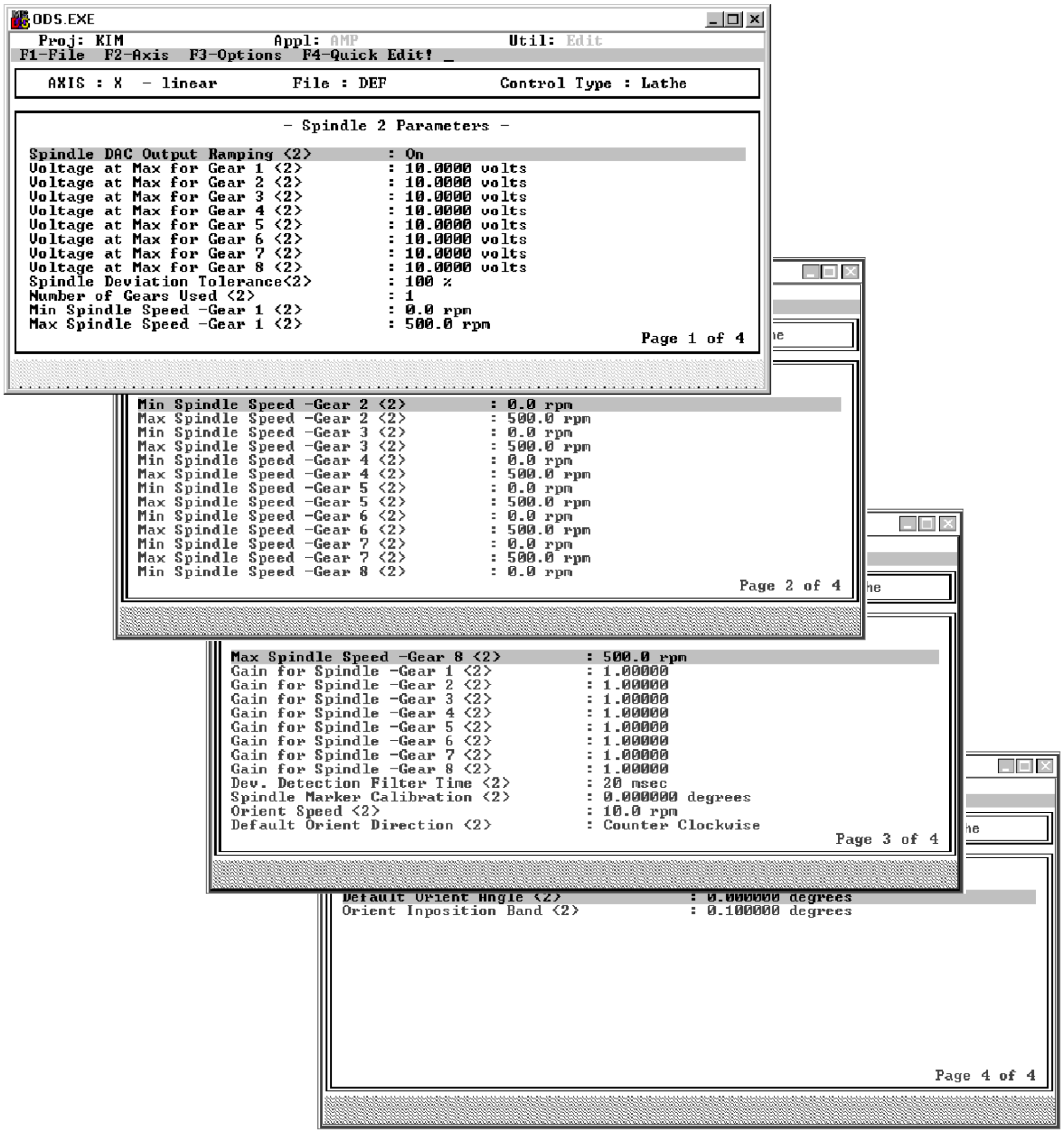
These parameters include:

- DAC Voltage and Spindle Gear Parameters
- Spindle DAC Output Ramping
- Voltage at Max for Gears 1-8
- Spindle Deviation Tolerance
- Number of Gears Used
- Minimum Spindle Speed - Gear 1-8
- Maximum Spindle Speed - Gear 1-8
- Deviation Detection Filter Time
- Spindle Orienting Parameters
- Spindle Marker Calibration
- Orient Speed
- Default Orient Direction
- Default Orient Angle
- Orient Imposition Band

Important: Only Spindle 1 may be used in conjunction with Virtual C or Cylindrical Interpolation.

Important: Many of the parameters in this section are not used when using a 1394 drive/1326 motor combination as a spindle. Refer to page 7–79 for details on this special spindle configuration.

To edit the spindle parameters, select “Spindle 2 Parameters” from the first page of the main menu screen. The workstation displays four screens similar to the spindle 1 parameters:



DAC Voltage and Spindle Gear Parameters

These parameters deal with the DAC output voltage and spindle gears. "DAC" stands for "Digital to Analog Converter." This is in reference to analog outputs from the 1394 CNC serial drive. The output port used is determined by the axis configured as a spindle with the F2 Axis option and the servo parameter "Output Port Number."

Important: If the spindle has multiple gear ranges and encoder feedback is being used, the encoder must somehow be driven directly by the spindle itself. The encoder must make one revolution per revolution of the spindle and only provide 1 marker pulse per revolution, regardless of which gear is selected.

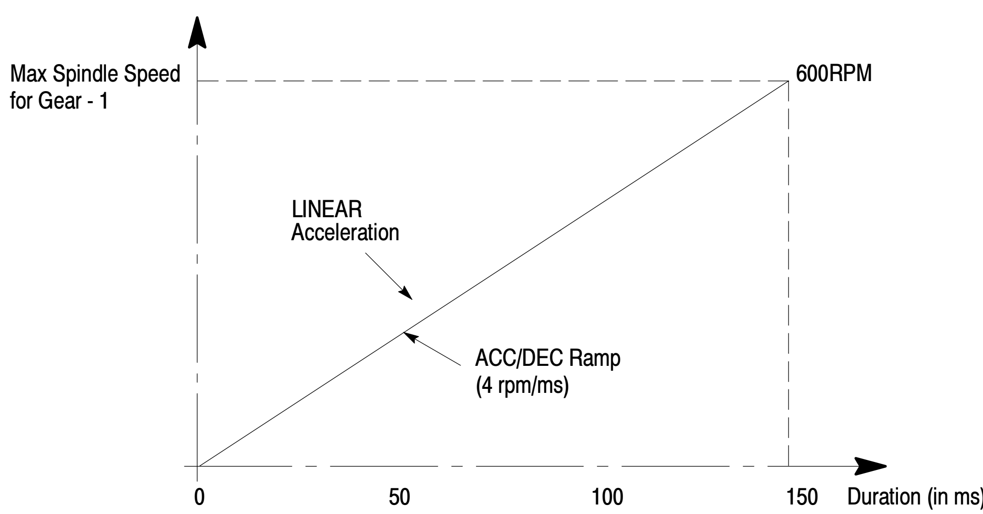
This DAC output provides for an analog voltage output within the range of -10 to +10V dc for use as a command signal to the spindle drive. This voltage range is limited by the parameter **Voltage at Max...** for each spindle gear.

Spindle DAC Output Ramping

Function

This parameter determines whether the DAC output is a gradual voltage ramp or a voltage step. For this parameter:

If you select:	then:
ON	any change in DAC output voltages is made as a ramp-up or ramp-down. The actual spindle ramp duration is selected using the **Dev. Detection Filter Time** parameter (if your system does not have the solid tapping option). The actual spindle ramp duration is selected using the **ACC. time for Spindle 2, Gear __** if your system has the solid tapping feature (this is used if your system has the solid tapping option and does not mean you must be using the solid tapping feature to get the ramp). The max speed for the ramp is determined from your setting of MAX. Spindle Speed-Gear __ . Thus a different ramp is created for each gear range.
OFF	changes in DAC output voltages are made immediately as one step



If your system:	Your ramp duration is set with:	Your RPM max over that duration is set with
has the solid tapping option	ACC. time for Spindle 2, Gear __ (in solid tapping parameter group)	Max Tapping Speed - Gear __ (in solid tapping parameter group)
does not have the solid tapping option	Spindle DAC Output Ramping (in spindle parameter group)	Max Spindle Speed - Gear __ (in spindle parameter group)

Spindle	Parameter Number
2	[804]

Range

Selection	Result
(a)	On
(b)	Off

Notes

The following notes assume you have chosen yes for this parameter enabling spindle ramping:

Open Loop Spindles
During normal open loop operation any change in commanded spindle speed is ramped. Any time that the commanded spindle speed changes from the current speed, the ramp is added to or subtracted from the current spindle speed each coarse iteration until the commanded spindle speed is reached. Spindle speed changes in both RPM mode and CSS mode are ramped. Changes in spindle direction (from logic or programming M03/M04) are also ramped.

Important: If using an open loop spindle orient (Shot Pin Orient), when the logic flag to complete the orient becomes true, the spindle command is zeroed immediately (no ramping occurs). For this type of orient your orient speed should be configured slow enough to allow this spindle stop without ramping. Any acceleration/deceleration of the spindle to reach the initial orient speed is done using the spindle ramp.

Closed Loop Spindle Orients
The initial acceleration or deceleration of the spindle to the orient speed is ramped. Once the orient velocity is reached, the spindle's position loop is closed and no further spindle ramping is performed. The AMPed orient speed should be slow enough that no problems occur when the spindle loop is closed. The gain for the spindle position loop should be chosen such that the natural exponential velocity profile of the position loop can be accommodated by the drive.

E-Stop
On transition into the Emergency Stop state, all spindle DAC output is zeroed immediately with no spindle ramping.

This parameter must be set for each spindle axis.

Voltage at Max for Gears 1 - 8

Function

This parameter is used to set the maximum DAC output voltage for each gear range of the spindle.

The maximum spindle RPM for each gear range is defined by the parameter **Maximum Spindle Speed**. It is necessary to determine what DAC output voltage is required to attain this maximum speed.

This information may be provided with the spindle drive. It can also be determined through testing. By connecting a battery box to the drive and a tachometer to the spindle, the command voltages required to produce specific spindle speeds in each gear can be plotted. These same "command voltages" is sent to the drive from the DAC output when the battery box is replaced with the proper cabling.

Parameter	Parameter Number
Voltage at Max for Gear 1	[830]
Voltage at Max for Gear 2	[831]
Voltage at Max for Gear 3	[832]
Voltage at Max for Gear 4	[833]
Voltage at Max for Gear 5	[834]
Voltage at Max for Gear 6	[835]
Voltage at Max for Gear 7	[836]
Voltage at Max for Gear 8	[837]

Range

-10.0000 to 10.0000 volts

Notes

This parameter must be set for each spindle axis.

Spindle Deviation Tolerance

Function

Specifies the percentage by which the actual spindle speed can deviate from the anticipated spindle speed before the control sets the flag "Spindle Speed Deviation Excessive" to "TRUE."

Corrective action by the control is limited to setting this logic flag. Therefore, this parameter requires proper logic programming to be effective. Refer to the notes below for more information about properly programming this parameter.

Important: For this parameter to have significance (and not be ignored):

- the spindle must have position feedback
- the software option for spindle speed deviation detection must be installed

Parameter Number
[867]

Range

0 to 100%

Notes

If the spindle has a feedback device, the control monitors the feedback and determines a velocity.

The "anticipated spindle speed" mentioned under FUNCTION above is the result of simulated feedback. The control simulates an acceleration / deceleration ramp for the spindle based on the parameter Dev. Detection Filter Time. The actual spindle velocity (from the spindle feedback) is then constantly compared to this simulated feedback.

For example, assume 20% is entered for this parameter and the commanded spindle speed (after considering spindle speed override) is 1000 RPM. As the spindle accelerates, its actual speed must remain within 20% of the speed anticipated through simulated feedback. If not, the control advises the logic program by setting the logic flag "Spindle Speed Deviation Excessive" to "TRUE."

This flag also is set "TRUE" if the spindle speed remains below 800 RPM or above 1200 RPM after the spindle should have reached a steady state speed of 1000 RPM.

Generally the logic program is written to display a warning message whenever the control sets the Spindle Speed Deviation Excessive flag to "TRUE." The logic program may also include a timer that forces more aggressive action if the flag remains true for an extended period.

If the "Spindle Speed Deviation Excessive" flag remains "TRUE" for too long or appears too often, there are a number of changes that can prevent this:

1. The value for this parameter can be increased.
2. The parameter Dev. Detection Filter Time can be increased or decreased to more closely match the actual spindle speed transition time.
3. The logic program can enhance a decision on spindle speed deviation by using the "spindle up to speed" signal (if available) from the spindle drive.
4. The logic program can be written to ignore the "Spindle Speed Deviation Excessive" flag during certain spindle speed transitions.

Refer to your *9/PC CNC Logic Reference Manual* for more information about the spindle speed deviation flag.

This parameter must be set for each spindle axis.

Number of Gears Used

Function

This parameter specifies the number of available spindle gear ranges for a specific machine application. The control allows for 8 spindle gear ranges.

Important: If the machine does not have multiple spindle gear ranges, enter 1 for this parameter.

Parameter Number
[866]

Range

1 to 8

Notes

Actual spindle gear changing is implemented through logic. Logic sets the gear mode request flag to "automatic," the control uses this parameter and the other AMP gear parameters to determine the correct gear for the commanded spindle speed. It then requests gear changes from logic as needed. Refer to your *9/PC CNC Logic Reference Manual* for more information.

This parameter must be set for each spindle axis.

Minimum and Maximum Spindle Speed

Function

It is necessary to enter the lowest and highest spindle RPM for each gear range to optimize a machine's performance. Overlapping ranges are permitted. The maximum value for a specific gear must be greater than the minimum value for that gear.

Gear change operations are controlled by the logic program with gear change logic flags. Refer to your *9/PC CNC Logic Reference Manual* for more information.

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: If the spindle is not capable of attaining the speed entered for a particular gear range, program execution may halt and/or an error message results (as determined by the control's logic program).

Values must be entered for all gear ranges specified by the parameter **Number of Gears Used**. If any of the gears used has no value or an illegal value entered here, the control assumes that no gears are available and never requests a gear change.

Parameter	Parameter Number	Parameter	Parameter Number
Min. Spindle Speed - Gear 1	[920]	Max. Spindle Speed - Gear 1	[930]
Min. Spindle Speed - Gear 2	[921]	Max. Spindle Speed - Gear 2	[931]
Min. Spindle Speed - Gear 3	[922]	Max. Spindle Speed - Gear 3	[932]
Min. Spindle Speed - Gear 4	[923]	Max. Spindle Speed - Gear 4	[933]
Min. Spindle Speed - Gear 5	[924]	Max. Spindle Speed - Gear 5	[934]
Min. Spindle Speed - Gear 6	[925]	Max. Spindle Speed - Gear 6	[935]
Min. Spindle Speed - Gear 7	[926]	Max. Spindle Speed - Gear 7	[936]
Min. Spindle Speed - Gear 8	[927]	Max. Spindle Speed - Gear 8	[937]

Range

Min. Spindle Speed:	0 to Max. Spindle Speed (RPM)
Max. Spindle Speed:	Min. Spindle Speed to 99999.9(RPM)

Notes

This parameter must be set for each spindle axis.

Digital Spindle users must set this parameter (for gear 1 only) in addition to setting the servo parameter “Maximum Motor Speed” to define valid motor speeds.

Dev. Detection Filter Time

Function

If spindle DAC output ramping is off:
This parameter specifies the spindle acceleration/deceleration time to be used by the control only for simulated feedback. Simulated feedback is explained with the parameter **Spindle Deviation Tolerance**.

Since the value entered here is used in simulating any change in speed (regardless of whether it’s from 0 to 20 RPM, from 10 to 2000 RPM, or from 500 to 2000 RPM), it is recommended that an average time be entered.

For example, if it takes 20 msec for a small change in spindle speed (such as from 0 to 5 RPM) and 160 msec for the maximum change in spindle speed (such as from 0 to 4000 RPM), a good value to enter here would be the average transition time, 90 msec.

Refer to the discussion for the parameter Spindle Deviation Tolerance.

If spindle DAC output ramping is on:
This parameter specifies:

- the spindle acceleration /declaration time to be used by the control for simulated feedback. Simulated feedback is explained with the parameter **Spindle Deviation Tolerance**. If you have the solid tapping feature option, enter a value for this parameter that closely matches the spindle acc/dec curve established by the solid tapping parameters (see page 12-3).
- if your system does not have the solid tapping feature this parameter specifies the duration of the spindle ramp (refer to page 12-3). This is also the same value the control uses for simulated feedback.

Spindle	Parameter Number
2	[875]

Range

20 to 400 msec

Notes

Simulated feedback allows the control to perform “feed per revolution” moves on machines that do not have position feedback from the spindle.

If the spindle has position feedback and the software option for spindle speed deviation detection has been installed, simulated feedback is also used for spindle speed deviation detection.

This parameter must be set for each spindle axis.

Spindle Orienting Parameters

These parameters are relevant only if your spindle provides position feedback and is capable of orienting to a specific position.

Important: It is possible to perform coarse spindle orients without position feedback through proper logic programming and the use of mechanical switches mounted on the spindle mechanism. These parameters are not relevant for such an application.

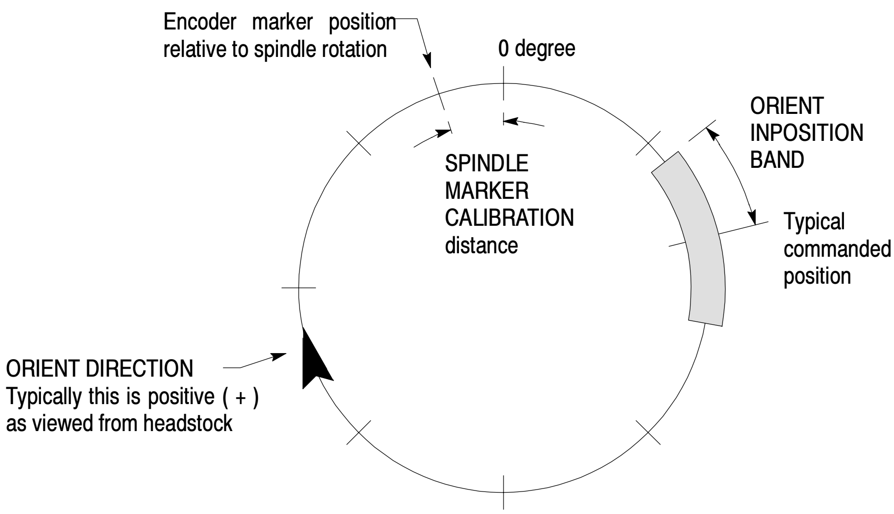
The spindle orient feature (Figure 9.1) lets the control stop the spindle at either a predefined angle set in AMP or a programmed angle. The spindle orient feature is discussed in your *9/PC CNC Operation and Programming Manual*.

Spindle Reference

Spindle homing is performed automatically when the control enters spindle orient mode (M19). (Note that spindle homing may also be forced when the spindle enters the virtual C mode as discussed in chapter 28.) Spindle homing consists of the spindle decelerating to the orient spindle RPM, and finding the encoder marker (or null position). Once this position is found, the spindle moves on to the spindle marker calibration point that determines the location of the zero point of the spindle relative to the encoder marker position. After this location is reached, the spindle moves to the location requested in the M19 block.

Important: Note that the use of a spindle with feedback must have only one encoder marker per revolution of the spindle. If more than one encoder marker is present, inconsistent homing results depending on which marker is found first when the spindle homing operation is activated.

Figure 13.1
Parameters for Spindle Orient



Spindle Marker Calibration

Function

Use this parameter to define a precise mechanical zero position for the spindle without requiring a mechanical adjustment of the encoder mounting on the spindle shaft. When the spindle is homing, the control is looking for the location of the encoder marker. After this marker is found, the spindle continues on moving the angular distance specified with this parameter. When this move is completed, the control calls its new rotary position angle zero for all future orient and virtual C operations.

Specify the rotary distance (in degrees) from the encoder marker (or null) to the "zero" angle position desired for the spindle. This distance defines the location from the encoder null to the spindle orientation position that is to be defined as home (orientation zero). If the location of the encoder null position is the same as the position to be defined as spindle home, enter a value of zero for this parameter.

The sign associated with this parameter simply allows this angular distance to be specified as a positive or negative value. This sign has no effect on the actual direction of rotation used to position to the calibration angle. This direction of rotation is the same direction used to find the encoder marker. Note that positive angles are typically measured from the encoder null in the clockwise direction (M03 spindle direction).

Parameter Number
[863]

Range

-360.000000 to 360.000000 degrees

Notes

This parameter can also be used to compensate for mechanical changes that otherwise would have relocated the zero position for the spindle. For example, the encoder orientation relative to the spindle position may change when repairing gears or gear belts. Consequently the distance that the axis moves from the encoder marker during homing may be different. That difference can be measured and entered as (or added to) the **Spindle Marker Calibration** value.

Important: Note that the use of a spindle with feedback must have only one encoder marker per revolution of the spindle. If more than one encoder marker is present, inconsistent homing results depending on which marker is found first when the spindle homing operation is activated.

This parameter must be set for each spindle axis.

Orient Speed

Function

This parameter determines the spindle speed in RPM that the control uses when positioning the spindle during a spindle orient (M19).

Refer to your *9/PC CNC Operation and Programming Manual* for more information on spindle orient.

Parameter Number
[864]

Range

0.0 to 99999.9 RPM

Notes

The orient speed is usually a very low value to assure that the spindle moves accurately to the orient position.

Executing an M19 may force a gear change to a different gear range to accommodate the speed entered here.

This parameter must be set for each spindle axis.

Default Orient Direction

Function

This parameter specifies the direction the spindle rotates during a spindle orient (M19).

When an M19 is executed in the part program or through MDI, the spindle rotates in the direction entered here to the position defined by the S word in the M19 block (if there is no S word in the block, the spindle rotates to the position defined by the **Default Orient Angle** parameter).

The direction entered here overrides the setting of the logic flag "Spindle Direction" during execution of an M19. This flag usually checks the spindle direction switch on the MTB panel to determine spindle direction.

Refer to your *9/PC CNC Operation and Programming Manual* for more information on spindle orient.

Parameter Number
[852]

Range

Selection	Result
(a)	Clockwise
(b)	Counter Clockwise

Notes

This parameter must be set for each spindle axis.

Default Orient Angle

Function

This parameter determines the absolute angle that the spindle rotates to during a spindle orient (M19) if no angle is programmed in the M19 block.

Refer to your *9/PC CNC Operation and Programming Manual* for more information on spindle orient.

Parameter Number
[868]

Range

0.000000 to 360.000000 degrees

Notes

If the M19 block is programmed with an S word, this parameter is ignored and the spindle rotates to the programmed angle.

This parameter must be set for each spindle axis.

Orient Inposition Band

Function

This parameter determines a positioning range (in degrees) that the spindle must be within for the control to consider the spindle "in position."

When the spindle position is within this range, the control closes the positioning loop (even though spindles are always configured as "Open Loop") to move and hold the spindle at the commanded position.

The control also sets a logic flag indicating that the spindle is in position. Therefore, this parameter requires proper logic programming to be fully effective. Refer to the notes below for more information on properly programming this parameter.

Important: If too small a value is entered here, the "Spindle Orientation Complete" flag in logic may never be set "TRUE." Allowances must be made for positioning accuracy and inherent instabilities.

Refer to your *9/PC CNC Operation and Programming Manual* for more information on spindle orient.

Parameter Number
[865]

Range

0.000000 to 360.000000 degrees

Notes

The value entered here sets the farthest distance (in either the positive or negative directions) that the spindle can be from its programmed destination to be considered "in position" for orient operations.

This does not affect the final positioning accuracy of the system. It simply determines how close to the commanded position the spindle must be before the logic flag "Spindle Orientation Complete" is set to "TRUE."

This parameter must be set for each spindle axis.

Gain for Spindle 2

Function

Enter the gain in units of RPM per .001 revolutions for this spindle in this gear.

Parameter	Parameter Number
Gain for Spindle 2 - Gear 1	[785]
Gain for Spindle 2 - Gear 2	[786]
Gain for Spindle 2 - Gear 3	[787]
Gain for Spindle 2 - Gear 4	[788]
Gain for Spindle 2 - Gear 5	[789]
Gain for Spindle 2 - Gear 6	[790]
Gain for Spindle 2 - Gear 7	[791]
Gain for Spindle 2 - Gear 8	[792]

Range

0.00000 to 100.00000

Notes

You must set this parameter for each gear you use of each spindle.

Use these parameters for spindle orient gain and gain during solid tapping. For solid tapping, the lower of the two gains (tapping axis and spindle) is applied to both motions.

You can use these parameters to shorten overall machining time by allowing the spindle to be oriented in any of 8 different gear ranges.

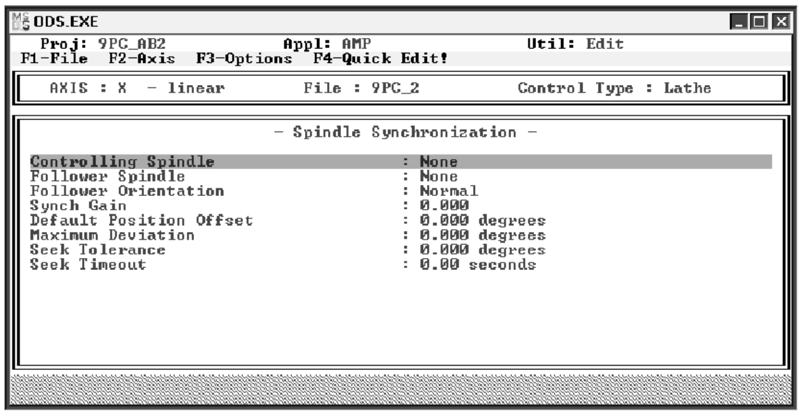
END OF CHAPTER

Spindle Synchronization

Chapter Overview

Use the spindle synchronization parameters to synchronize velocity or position and velocity between two spindles with feedback using your 9/PC control.

The feedback ratio between the two spindles must be a simple ratio with either side no greater than 10 (e.g., 1:1 or 10:7). If either side of the ratio exceeds 10 (e.g., 11:1 or 5:12), an error will occur.



Select the "Spindle Synchronization" screen from the first page of the main menu to access the synchronization parameters.

This table lists the parameters that are necessary to configure spindle synchronization:

Parameter:	Page:
Controlling Spindle	14-2
Follower Spindle	14-2
Follower Orientation	14-4
Synch Gain	14-5
Default Position Offset	14-6
Maximum Deviation	14-7
Seek Tolerance	14-7
Seek Timeout	14-8

Important: Make sure you take into consideration other parameters that will be used in conjunction with the synchronized spindle parameter
(e.g., Gain for Spindles and Excess Error).

Controlling Spindle

Function

This parameter determines the controlling spindle for a synchronized pair. Use the spindle number to define this parameter. Spindle numbers are identified by the Spindle Type for Axis parameter (refer to page 7-82 for more information about this parameter).

During synchronization, commands to the controlling spindle (i.e., programmed spindle speed and direction) are mimicked by the follower spindle.

Parameter Number
[590]

Range

0 to 3

Selection	Result
(a)	None
(b)	Spindle 1
(c)	Spindle 2

Notes

This is a global parameter. The value set here applies to all axes.

Important: When configuring your spindles in AMP, make sure they are given separate, definitive spindle parameters. If one spindle is defined as the controlling spindle as well as the follower spindle, the control will assume you are trying to synchronize a spindle to itself and you will receive the error: SYNCH SPINDLES MISCONFIGURED.

Follower Spindle

Function

This parameter determines the follower spindle for a synchronized pair. Use the spindle number to define this parameter. Spindle numbers are identified by the Spindle Type for Axis parameter (refer to page 7-82 for more information about this parameter).

During synchronization, commands to the controlling spindle (i.e., programmed spindle speed and direction) are mimicked by the follower spindle.

Important: When configuring your spindles, make sure they are given separate, definitive spindle parameters. If one spindle is defined as the controlling spindle as well as the follower spindle, the control will assume you are trying to synchronize a spindle to itself and you will receive the error: SYNCH SPINDLES MISCONFIGURED.

Parameter Number
[591]

Range

0 to 3

Selection	Result
(a)	None
(b)	Spindle 1
(c)	Spindle 2

Notes

This is a global parameter. The value set here applies to both spindles.

To accommodate for the difference in gear ranges for the controlling and follower spindles, AMP takes the lowest maximum and the highest minimum gear ranges of the follower spindle to create the available gear range. While the follower spindle is ramping to the required spindle speed, a warning will appear, warning you that you have bypassed the minimum or maximum RPM.

The following example shows the value at which the follower spindle ramps to when:

- No overlap occurs between the controlling and follower spindles' gear ranges
- The controlling spindle has a higher gear range than the follower spindle
- The controlling spindle has a lower gear range than the follower spindle

Example 14.1
Valid Gear Ranges for Synchronized Spindles

Controlling Spindle Gear Range (RPM)	Follower Spindle Gear Range (RPM)	Requested Spindle Speed (RPM)	Valid Programmed Spindle Speeds (RPM)	Spindles will Synchronize at (RPM):
1000 to 3000	100 to 300	1500	None	N/A
1000 to 3000	800 to 1500	1800	1000 to 1500	1500
1000 to 3000	1800 to 3200	1500	1800 to 3000	1800

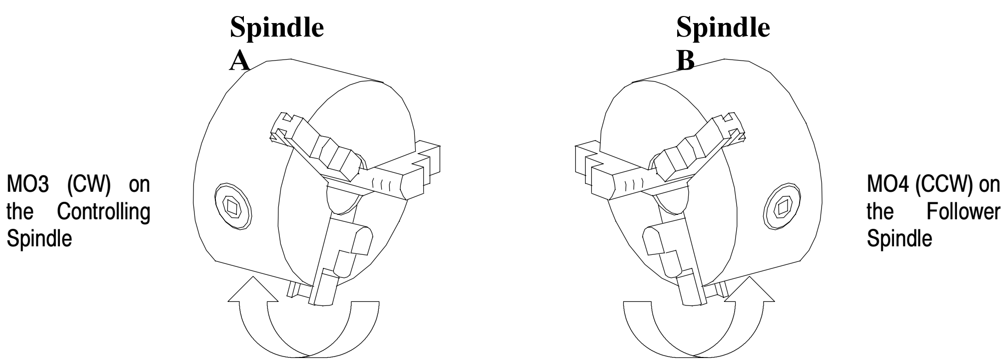
If the external load is on the follower spindle, the controlling spindle will not try to adjust to following spindle's velocity or velocity and position during synchronization. This action returns a BR_SYNE error, identified by logic. Refer to your *9/PC Logic Reference Manual* for more information on the BR_SYNE flag.

Follower Orientation

Function

This parameter determines whether reversed rotation is required on the follower spindle to synchronize with the controlling spindle. This parameter also determines if orient angles are reversed (after the zero points are designated on both spindle markers, 45 degrees on controlling = 315 degrees on follower). For information on configuring the zero point, refer to page 14-6.

When configuring Normal follower orientation, an M03 (CW) returns an M03. When configuring Reversed follower orientation, an M03 (CW) returns an M04 (CCW). In the figure below, the spindles are using Reversed follower orientation, where spindle A is programmed with an M03, while spindle B is programmed with an M04.



Parameter Number
[592]

Range

Normal and Reversed

Selection	Result
(a)	Normal
(b)	Reversed

Notes

This is a global parameter. The value set here applies to both spindles.

Synch Gain

Function

The servos of a synchronized spindle pair are always given the same commands. Thus, a well-balanced, synchronized spindle pair with equally sized motors that operate on fairly symmetric loads may not need to use this parameter. The normal spindle positioning loops should adequately respond to keep the difference in following error between two spindles to a minimum.

However, in some cases, the normal servo loop may allow too great a difference in following error between spindles when motors are sized differently or one servo is under a higher load than another. When the normal spindle positioning loops are not sufficient to compensate for this difference in following error, use the Synch Gain parameter.

When synchronization occurs, the control detects a difference in following error between the two servos. When using Synch Gain, this difference in the following error is controlled by increasing or decreasing the gain of the follower spindle. This causes the control to make the follower spindle's response match more closely with that of the controlling spindle.

The follower spindle's gain is modified proportionally to the difference in following errors between the controlling and follower spindles (= synchronization error). The ratio of this multiplier is controlled by this parameter. Here, a value of 0 disables any gain modification of the follower spindle. The higher the value set here, the larger the gain modification for the same difference in following error.

This parameter determines the amount of positional spindle gain necessary to attempt to correct positional discrepancy once synchronization is active.

Parameter Number
[593]

Range

0.000 to 1.000

Notes

This is a global parameter. The value set here applies to both spindles.

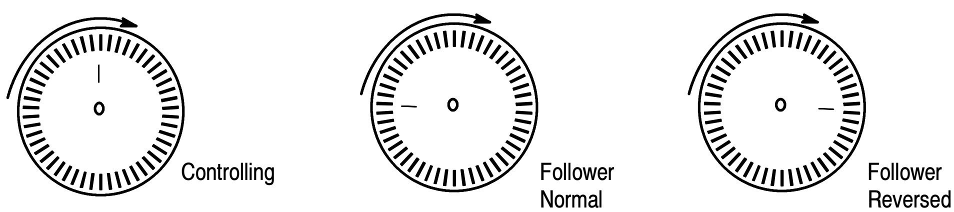
Default Position Offset

Function

This parameter denotes how much the controlling spindle's marker leads the follower spindle's zero position during positional synchronization. The zero point is determined by the marker and Spindle Marker Calibration (page 12-11, or 13-11). If a positional value is programmed in the part program block, this default position offset value is ignored. This parameter will also apply to any spindle orients that take place while positional synchronization is active. For example, if a default position offset parameter = 60.0, when the controlling spindle is oriented to 80 degrees, the follower spindle should be at 20 degrees if the synchronization direction is normal, or -20 if the synchronization direction is reversed.

Figure 14.1
Aligning Controlling and Follower Encoder Markers

In this example, the controlling spindle marker is positioned at 0. The follower spindle's zero position is offset 90 degrees from the controlling spindle's position in normal and reversed positions.



Parameter Number
[594]

Range

0.000 to 360.000 degrees

Notes

This is a global parameter. The value set here applies to both spindles.

Threading with synchronization active will use the controlling spindle marker.

Maximum Deviation

Function

This parameter determines the maximum amount of positional deviation allowed between synchronized spindle markers **after** synchronization is achieved. When this maximum deviation is exceeded, the logic flag, BR_SYNE is set.

Parameter Number
[595]

Range

0.000 to 360.000 degrees

Notes

This is a global parameter. The value set here applies to both spindles.

Maximum synchronization accuracy will depend on following error stability when synchronization is achieved.

Seek Tolerance

Function

This parameter determines the maximum amount of positional deviation allowed between synchronized spindle markers **before** synchronization is achieved.

Parameter Number
[596]

Range

0.000 to 360.000 degrees

Notes

This is a global parameter. The value set here applies to both spindles.

The Spindle Deviation Tolerance parameter in spindles 1, and 2 also effects how closely the spindles are synchronized.

Seek Timeout

Function

This parameter sets the time frame within which synchronization should occur. If spindle synchronization is not successful within the time specified, an error message is posted and the logic flag, BW_SYNTO is set. Refer to your *9/PC Logic Reference Manual* for more information about the BW_SYNTO flag.

Parameter Number
[597]

Range

0.00 to 300.00 seconds

Notes

This is a global parameter. The value set here applies to both spindles.

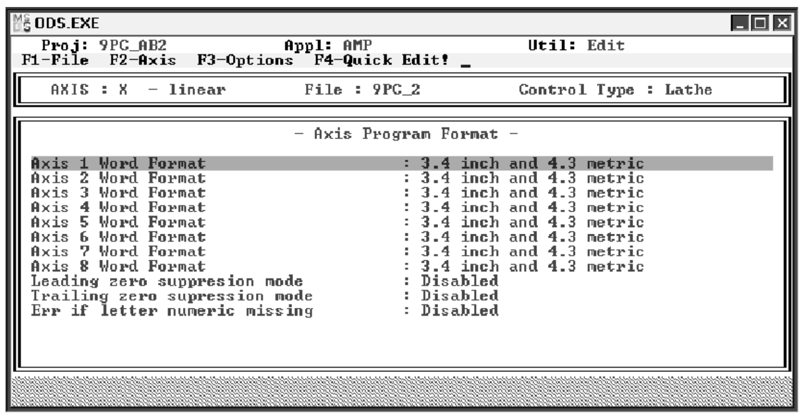
END OF CHAPTER

Axis Program Format Parameters

Chapter Overview

This chapter covers the Axis Program Format parameters. These parameters specify the word formats that must be used when specifying axis position and/or distance in part programs. The specified format limits the number of digits that may be entered to the left and to the right of the decimal point when entering an axis word. There are additional parameters in this group that can be set to determine zero suppression and error checking if a letter or number is missing.

The workstation displays this screen after you select the "Axis Program Format" group:



The parameter values shown above are the default values of these parameters. These values specify that:

When the control is in:	No. of digits that may be entered to the left of the decimal point:	No. of digits that may be entered to the right of the decimal point:
Inch Mode	3	4
Metric Mode	4	3

Axis 1-8 Word Format

Function

This parameter specifies the word format of the word used to specify the axis position and distance of axes 1-8 . It specifies the maximum number of digits that can be entered to the left and to the right of the decimal point when the control is configured for inch or metric modes.

Axis	Parameter Number
1	[150]
2	[151]
3	[152]
4	[153]
5	[154]
6	[155]
7	[156]
8	[157]

Range

Selection	Result	Selection	Result
(a)	3.4 inch and 4.3 metric	(i)	2.6 inch and 3.5 metric
(b)	3.5 inch and 4.4 metric	(j)	3.3 inch and 4.2 metric
(c)	4.3 inch and 5.2 metric	(k)	4.2 inch and 5.1 metric
(d)	4.4 inch and 5.3 metric	(l)	5.2 inch and 6.1 metric
(e)	5.3 inch and 6.2 metric	(m)	3.4 inch and metric (rotary)
(f)	2.3 inch and 3.2 metric	(n)	3.3 inch and metric (rotary)
(g)	2.4 inch and 3.3 metric	(o)	3.2 inch and metric (rotary)
(h)	2.5 inch and 3.4 metric	(p)	3.1 inch and metric (rotary)

Notes

You must set a word format parameter for each axis.

Zero Suppression and Error Modes

The zero suppression mode parameters are covered in the following sections:

Parameter:	Page:
Leading Zero Suppression Mode	15-3
Trailing Zero Suppression Mode	15-3
Error if Letter Numeric Missing	15-4

Leading Zero Suppression Mode

Function

This parameter specifies whether leading zero suppression mode is active when a programmed word is decoded by the control. It should be disabled when decimal point programming is being performed. When this parameter is enabled, the control "right-justifies" the programmed word according to the word format of the programmed word.

Trailing zeros must be programmed to make sure that each word has its required number of digits to the left and the right of the decimal point. This number includes any suppressed leading zeros.

For example, if this parameter is enabled and the X-axis word format is 4.3, then X01 is decoded as X0000.001 and X2790 is decoded as X0002.790.

Axis	Parameter Number
All	[404]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This is a global parameter; the value set here applies to all axes and processes.

Important: Leading zero suppression mode and trailing zero suppression mode cannot be enabled at the same time.

Trailing Zero Suppression Mode

Function

This parameter specifies whether trailing zero suppression mode is active when a programmed word is decoded by the control. It should be disabled when decimal point programming is being performed. When this parameter is enabled, the control "left-justifies" the programmed word according to the word format of the programmed word.

Leading zeros must be programmed to make sure that each word has its required number of digits to the left and the right of the decimal point. This number includes any suppressed trailing zeros.

For example, if this parameter is enabled and the X-axis word format is 4.3, then X01 is decoded as X0100.000 and X279 is decoded as X2790.000.

Axis	Parameter Number
All	[407]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This is a global parameter; the value set here applies to all axes and processes.

Important: Trailing zero suppression mode and leading zero suppression mode cannot be enabled at the same time.

Error if Letter Numeric Missing

Function

This parameter specifies whether a part program format error is generated if a letter is programmed without a succeeding numeric digit (for example, programming the block XYZ; would generate an error). When this parameter is disabled, the control assumes a value of zero for any letter programmed without a numeric digit.

Axis	Parameter Number
All	[406]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This is a global parameter; the value set here applies to all axes and processes.

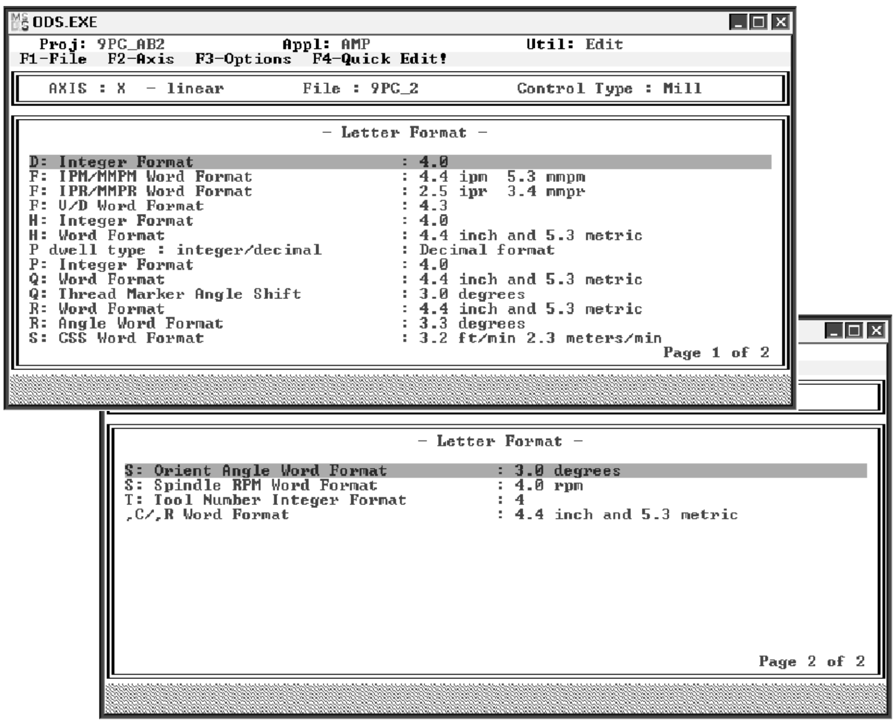
END OF CHAPTER

Letter Format Parameters

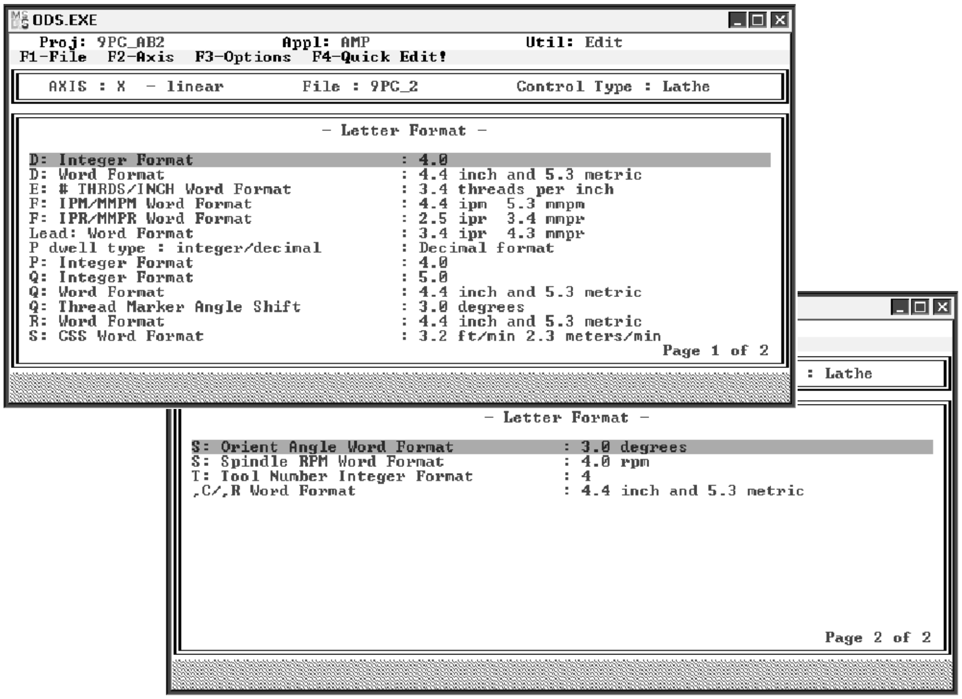
Chapter Overview

This chapter covers the Letter Format parameters that specify the word formats that must be used when entering D–, E–, F–, H–, Lead, P–, Q–, R–, S–, and T–words in part programs. These parameters specify the number of digits that may be entered to the left and to the right of the decimal point (program resolution) for a word in a part program.

When you select the “Letter Format” group and the control type “Mill,” the workstation displays these screens:



When you select the “Letter Format” group and the control type “Lathe,” the workstation displays these screens:



These parameters are described on these pages:

Parameter:	Page:
D Integer and Word Formats	16-3
E Word Format	16-5
F Word Formats	16-6
H Integer and Word Formats	16-8
Lead Word Format	16-10
P Word Format	16-11
Q Integer and Word Formats	16-12
R Word Formats	16-14
S Word Formats	16-15
T Word Format	16-17
Chamfering/Corner-R Word Letter Formats	16-18

D-Integer and D-Word Formats

The following sections cover the D-word integer and word letter format parameters:

D:Integer Format

Function

This parameter specifies the maximum number of digits that are permitted to the left of the decimal point in the D-word integer format. Since this parameter has an integer format, the number of digits to the right of the decimal point will always be zero.

Use this parameter when programming a D-word that corresponds to a tool diameter (or radius) offset number. Use D-words during cutter compensation and any other time that data for tool diameter (or radius) is necessary. When programmed, the control uses the D-word to call information from the tool geometry and tool wear tables for tool diameter (or radius).

Parameter	Parameter Number
D: Integer Format	[460]

Range

Selection	Result
(a)	1.0
(b)	2.0
(c)	3.0
(d)	4.0
(e)	5.0

Notes

These are global parameters; the values entered here apply to all axes and processes.

D: Word Format

Function

This parameter is set for lathe controls only. For Lathe fixed cycles, it refers to the D-word programmed in these compound turning routines:

G Code System			Description
A	B	C	
G71	G71	G73	O.D. & I.D. Roughing Routine
G72	G72	G74	Rough Facing Routine
G73	G73	G75	Casting/Forging Roughing Cycle
G76	G76	G78	Multi-Pass Threading Cycle

The D-word is used to define a depth of cut or incremental shift in these cycles (refer to the specific cycle description in your programming and operation manual).

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the D-word format when programming in inch or metric modes.

Parameter	Parameter Number
D: Word Format	[461]

Range

Selection	Result	Selection	Result
(a)	3.4 inch and 4.3 metric	(g)	2.4 inch and 3.3 metric
(b)	3.5 inch and 4.4 metric	(h)	2.5 inch and 3.4 metric
(c)	4.3 inch and 5.2 metric	(i)	2.6 inch and 3.5 metric
(d)	4.4 inch and 5.3 metric	(j)	3.3 inch and 4.2 metric
(e)	5.3 inch and 6.2 metric	(k)	4.2 inch and 5.1 metric
(f)	2.3 inch and 3.2 metric	(l)	5.2 inch and 6.1 metric

Notes

This is a global parameter; it applies to all axes and processes.

E-Word Format

The E-word letter format parameter is covered in E: # THRDS/INCH Word Format in the following section.

E: # THRDS/INCH Word Format

Function

This parameter is used for lathe controls and grinder applications only. It corresponds to the E-word that is used to program the thread lead in one of these threading methods:

Lathe G-Code System			Description
A	B	C	
G32	G32	G33	Constant lead thread-cutting mode
G33	G33	G34	Variable lead thread-cutting mode
G92	G78	G21	Single Pass Threading Cycle
G76	G76	G78	Multi-Pass Threading Cycle

E-words correspond to the thread lead (in number of threads per inch) along the axis with the largest programmed distance to travel to make the thread cut.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the E-word format.

Parameter	Parameter Number
E: #THRDS/INCH Word Format	[464]

Range

Selection	Result	Selection	Result
(a)	3.4 threads per inch	(g)	2.4 threads per inch
(b)	3.5 threads per inch	(h)	2.5 threads per inch
(c)	4.3 threads per inch	(i)	2.6 threads per inch
(d)	4.4 threads per inch	(j)	3.3 threads per inch
(e)	5.3 threads per inch	(k)	4.2 threads per inch
(f)	2.3 threads per inch	(l)	5.2 threads per inch

Notes

This parameter is a global parameter; it applies to all axes and processes.

This parameter is not used for decoding the E-word when E is being used to program the reciprocation feedrate, or when E is used in a G38 block. In these cases, the current F-word format is used for decoding the E-word.

F-Word Formats

The F-word format parameters are covered in the following three sections:

F: IPM/MMPM Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the F-word format. This parameter specifies the format of an F-word that is used to program a linear feedrate.

Parameter	Parameter Number
F: IPM/MMPM	[467]

Range

Selection	Result	Selection	Result
(a)	4.4 ipm 5.3 mmpm	(i)	3.2 ipm 4.1 mmpm
(b)	3.4 ipm 4.3 mmpm	(j)	2.2 ipm 3.1 mmpm
(c)	2.4 ipm 3.3 mmpm	(k)	6.1 ipm 7.0 mmpm
(d)	4.3 ipm 5.2 mmpm	(l)	5.1 ipm 6.0 mmpm
(e)	3.3 ipm 4.2 mmpm	(m)	4.1 ipm 5.0 mmpm
(f)	2.3 ipm 3.2 mmpm	(n)	3.1 ipm 4.0 mmpm
(g)	5.2 ipm 6.1 mmpm	(o)	2.1 ipm 3.0 mmpm
(h)	4.2 ipm 5.1 mmpm		

F: IPR/MMPR Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the F-word format. This parameter specifies the format of an F-word that is used to program a feedrate in inches or millimeters per revolution.

Parameter	Parameter Number
F: IPR/MMPR Word Format	[470]

Range

Selection	Result
(a)	2.5 IPR 3.4 MMPR
(b)	1.5 IPR 2.4 MMPR
(c)	3.4 IPR 4.3 MMPR
(d)	2.4 IPR 3.3 MMPR
(e)	1.4 IPR 2.3 MMPR
(f)	4.3 IPR 5.2 MMPR
(g)	3.3 IPR 4.2 MMPR
(h)	2.3 IPR 3.2 MMPR
(i)	1.3 IPR 2.2 MMPR

Notes

This parameter is a global parameters; it applies to all axes.

F: V/D Word Format

Function

This parameter is used for mill controls only. It specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the F-word format. This parameter specifies the format of an F-word that is used to program a feedrate in inverse time feed mode (minutes per move).

Parameter	Parameter Number
F: V/D Word Format	[471]

Range

Selection	Result
(a)	5.3
(b)	5.2
(c)	5.1
(d)	5.0
(e)	4.3
(f)	4.2
(g)	4.1
(h)	4.0
(i)	3.3
(j)	3.2
(k)	3.1
(l)	3.0
(m)	2.3
(n)	2.2
(o)	2.1
(p)	2.0

Notes

This parameter is a global parameter; it applies to all axes.

H-Integer and H-Word Formats

The H-integer and word letter format parameters are covered in the following two sections:

H: Integer Format

Function

This parameter is used for mill controls only. It is used for an H-word that programs a tool length offset number. The H-word is used during tool length offset programming and any other time that data for tool length is necessary. When programmed, the control uses the H-word to call information from the tool geometry and tool wear tables for tool length.

This parameter specifies the maximum number of digits that are permitted to the left of the decimal point in the H-word format. The number of digits to the right of the decimal point will always be zero since this parameter has an integer format.

Parameter	Parameter Number
H: Integer Format	[473]

Range

Selection	Result
(a)	1.0
(b)	2.0
(c)	3.0
(d)	4.0
(e)	5.0

Notes

This parameter is a global parameter; it applies to all axes.

H: Word Format

Function

This parameter is used for mill controls only. It corresponds to the H-word used in the G38 hole probing feature. In this feature, the H-word is used to program the expected diameter of the hole.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the H-word format when programming in inch or metric modes.

Parameter	Parameter Number
H: Word Format	[474]

Range

Selection	Result
(a)	3.4 inch and 4.3 metric
(b)	3.5 inch and 4.4 metric
(c)	4.3 inch and 5.2 metric
(d)	4.4 inch and 5.3 metric
(e)	5.3 inch and 6.2 metric
(f)	2.3 inch and 3.2 metric
(g)	2.4 inch and 3.3 metric
(h)	2.5 inch and 3.4 metric
(i)	2.6 inch and 3.5 metric
(j)	3.3 inch and 4.2 metric
(k)	4.2 inch and 5.1 metric
(l)	5.2 inch and 6.1 metric

Notes

This parameter is a global parameter; it applies to all axes.

Lead: Word Format

Function

This parameter is used for lathe controls only. It refers to the F-word that is used to program the thread lead in one of these threading methods:

Lathe G-Code System			Description
A	B	C	
G32	G32	G33	Constant lead thread-cutting mode
G33	G33	G34	Variable lead thread-cutting mode
G92	G78	G21	Single Pass Threading Cycle
G76	G76	G78	Multi-Pass Threading Cycle

This F-word represents the thread lead (in inches or millimeters of axis travel per spindle revolution) along the axis with the largest programmed distance to travel to make the thread cut.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the F-word format when in inch or metric modes.

Parameter	Parameter Number
Lead: Word Format	[477]

Range

Selection	Result
(a)	2.5 IPR 3.4 MMPR
(b)	1.5 IPR 2.4 MMPR
(c)	3.4 IPR 4.3 MMPR
(d)	2.4 IPR 3.3 MMPR
(e)	1.4 IPR 2.3 MMPR
(f)	4.3 IPR 5.2 MMPR
(g)	3.3 IPR 4.2 MMPR
(h)	2.3 IPR 3.2 MMPR
(i)	1.3 IPR 2.2 MMPR

Notes

This parameter is a global parameter; it applies to all axes.

P-Word Formats

Review the following sections for information about the P-word letter format parameters:

P Dwell Type: Integer/Decimal

Function

This parameter specifies whether the dwell time should be interpreted as in integer format or as in decimal point format when programming P-words.

Axis	Parameter Number
All	[563]

Range

Selection	Result
(a)	Decimal format
(b)	Integer format

Notes

This parameter is a global parameter; it applies to all axes.

P: Integer Format

Function

This parameter specifies the integer format of the P-word that is used to call subprograms, paramacro programs or dressing programs.

This parameter specifies the maximum number of digits that are permitted to the left of the decimal point in the P-word format. The number of digits to the right of the decimal point will always be zero since this parameter has an integer format.

Parameter	Parameter Number
P: Integer Format	[482]

Range

Selection	Result
(a)	1.0
(b)	2.0
(c)	3.0
(d)	4.0
(e)	5.0

Notes

This parameter is a global parameter; it applies to all axes.

Q-Word Formats

The next three sections cover Q-word letter format parameters:

Q: Integer Format

Function

This parameter is used for lathe controls only, and refers to the Q-word that is used in the compound turning routines called by these G-codes.

G Code System			Description
A	B	C	
G70	G70	G72	O.D. & I.D. Finishing Cycle
G71	G71	G73	O.D. & I.D. Roughing Cycle
G72	G72	G74	Rough Facing Cycle
G73	G73	G75	Casting/Forging roughing Cycle

Use this parameter for the sequence number of roughing and finishing cycles. It specifies the maximum number of digits that are permitted to the left of the decimal point in the Q-word format. The number of digits to the right of the decimal point will always be zero since this parameter has an integer format.

Parameter	Parameter Number
Q: Integer Format	[483]

Range

Selection	Result
(a)	1.0
(b)	2.0
(c)	3.0
(d)	4.0
(e)	5.0

Notes

This is a global parameter. The value entered here applies to all axes.

Q: Word Format

Function

This parameter is used for the milling cycles of mill controls, the drilling cycles of lathe controls and the grinding cycles of grinder controls. A Q-word in a cycle block is used to program various information depending on the cycle. Generally it is used to program either a shift amount for cycles that use a spindle orient or to program an infeed amount for cycles that reach total cutting depth in steps.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the Q-word format when programming in inch or metric modes.

Parameter	Parameter Number
Q: Word Format	[484]

Range

Selection	Result
(a)	3.4 inch and 4.3 metric
(b)	3.5 inch and 4.4 metric
(c)	4.3 inch and 5.2 metric
(d)	4.4 inch and 5.3 metric
(e)	5.3 inch and 6.2 metric
(f)	2.3 inch and 3.2 metric
(g)	2.4 inch and 3.3 metric
(h)	2.5 inch and 3.4 metric
(i)	2.6 inch and 3.5 metric
(j)	3.3 inch and 4.2 metric
(k)	4.2 inch and 5.1 metric
(l)	5.2 inch and 6.1 metric

Notes

This parameter is a global parameter; it applies to all axes.

Q: Thread Marker Angle Shift

Function

This parameter specifies the Q-word for threading blocks. It provides a relative value for the start offset angle of the thread. Its primary use is in cutting multi-start threads.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the Q-word format.

Parameter	Parameter Number
Q: Thread Marker Angle Shift	[485]

Range

Selection	Result
(a)	3.0 degrees
(b)	3.1 degrees
(c)	3.2 degrees

Notes

These are global parameters. The values entered here apply to all axes.

R Words

For information about the R-word letter format parameters, review the next two sections:

R: Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the R-word format. An R-word is typically used to program a radius.

Parameter	Parameter Number
R: Word Format	[488]

Range

Selection	Result
(a)	3.4 inch and 4.3 metric
(b)	3.5 inch and 4.4 metric
(c)	4.3 inch and 5.2 metric
(d)	4.4 inch and 5.3 metric
(e)	5.3 inch and 6.2 metric
(f)	2.3 inch and 3.2 metric
(g)	2.4 inch and 3.3 metric
(h)	2.5 inch and 3.4 metric
(i)	2.6 inch and 3.5 metric
(j)	3.3 inch and 4.2 metric
(k)	4.2 inch and 5.1 metric
(l)	5.2 inch and 6.1 metric

Notes

This parameter is a global parameter; it applies to all axes.

R: Angle Word Format

Function

This parameter is used for mill controls only. It refers to the R-word that is used when programming a coordinate system rotation (G68). This R-word specifies the angle of rotation (entered in units of degrees) at which the coordinate system is to be rotated.

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the R-word format.

Parameter	Parameter Number
R: Angle Word Format	[489]

Range

Selection	Result
(a)	3.1 degrees
(b)	3.2 degrees
(c)	3.3 degrees
(d)	3.4 degrees

Notes

This parameter is a global parameter; it applies to all axes.

Part rotation cannot be performed if the reciprocation axis is in the currently active phase.

S-Word Letter Formats

The S-word letter format parameters are covered in next few sections:

S: CSS Word Format

Function

This parameter specifies the format of the S-word that is used to program a cutting speed in units of feet/minute or meters/minute, and for grinders, feet/second or meters/second. An S-word programs a cutting speed when the constant surface speed feature (CSS) is active. This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the S-word format.

Parameter	Parameter Number
S: CSS Word Format	[492]

Range

Selection	Result for mill/lathe	Result for grinder
(a)	4.2 ft./min. 3.3 meters/min.	4.2 ft./time 3.3 meters/time
(b)	3.2 ft./min. 2.3 meters/min.	3.2 ft./time 2.3 meters/time
(c)	4.1 ft./min. 3.2 meters/min.	4.1 ft./time 3.2 meters/time
(d)	3.1 ft./min. 2.2 meters/min.	3.1 ft./time 2.2 meters/time

Notes

This parameter is a global parameter; it applies to all axes.

This parameter sets only the number of digits to the left and to the right of the decimal point.

S: Orient Angle Word Format

Function

This parameter specifies the format of an S-word that is used to program an angle when using the spindle orient feature (M19). This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the S-word format.

Parameter	Parameter Number
S: Orient Angle Word Format	[493]

Range

Selection	Result
(a)	3.0 degrees
(b)	3.1 degrees
(c)	3.2 degrees

Notes

This parameter is a global parameter; it applies to all axes.

S: Spindle RPM Word Format

Function

This parameter specifies the format of the S-word that is used to program a spindle speed in units of RPM (revolution per minute). This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point in the S-word format.

Parameter	Parameter Number
S: Spindle RPM Word Format	[494]

Range

Selection	Result
(a)	5.0 RPM
(b)	4.0 RPM
(c)	3.0 RPM
(d)	2.0 RPM
(e)	5.1 RPM
(f)	4.1 RPM
(g)	3.1 RPM
(h)	2.1 RPM
(i)	4.2 RPM
(j)	3.2 RPM
(k)	2.2 RPM

Notes

This parameter is a global parameter; it applies to all axes.

T-Word Formats

The T-word letter format parameter is covered in T: Tool Number Integer Format on page 16-17.

T: Tool Number Integer Format

Function

This parameter specifies the integer format of a T-word that is used to program a tool number. This parameter specifies the maximum number of digits that are permitted to the left of the decimal point in the T-word format. The number of digits to the right of the decimal point will always be zero since this parameter has an integer format.

For lathe controls, this T-word is used to call a tool number, wear offset number, and geometry offset number. For mill controls, this T-word is used to call a tool number only.

Parameter	Parameter Number
T: Tool Number Integer Format	[497]

Range

Selection	Result
(a)	1
(b)	2
(c)	3
(d)	4
(e)	5
(f)	6

Notes

This parameter is a global parameter; it applies to all axes .

Chamfering/Corner-R Word Letter Formats

The chamfer and radius letter format parameter is covered in ,C/,R Word Format on page 16-18.

,C/,R Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point when entering Chamfering/Cornering words (,C and ,R) while in inch or metric modes.

Parameter	Parameter Number
,C/,R Word Format	[515]

Range

Selection	Result
(a)	3.4 inch and 4.3 metric
(b)	3.5 inch and 4.4 metric
(c)	4.3 inch and 5.2 metric
(d)	4.4 inch and 5.3 metric
(e)	5.3 inch and 6.2 metric
(f)	2.3 inch and 3.2 metric
(g)	2.4 inch and 3.3 metric
(h)	2.5 inch and 3.4 metric
(i)	2.6 inch and 3.5 metric
(j)	3.3 inch and 4.2 metric
(k)	4.2 inch and 5.1 metric
(l)	5.2 inch and 6.1 metric

Notes

This parameter is a global parameter; it applies to all axes.

END OF CHAPTER

Plane Select Parameters

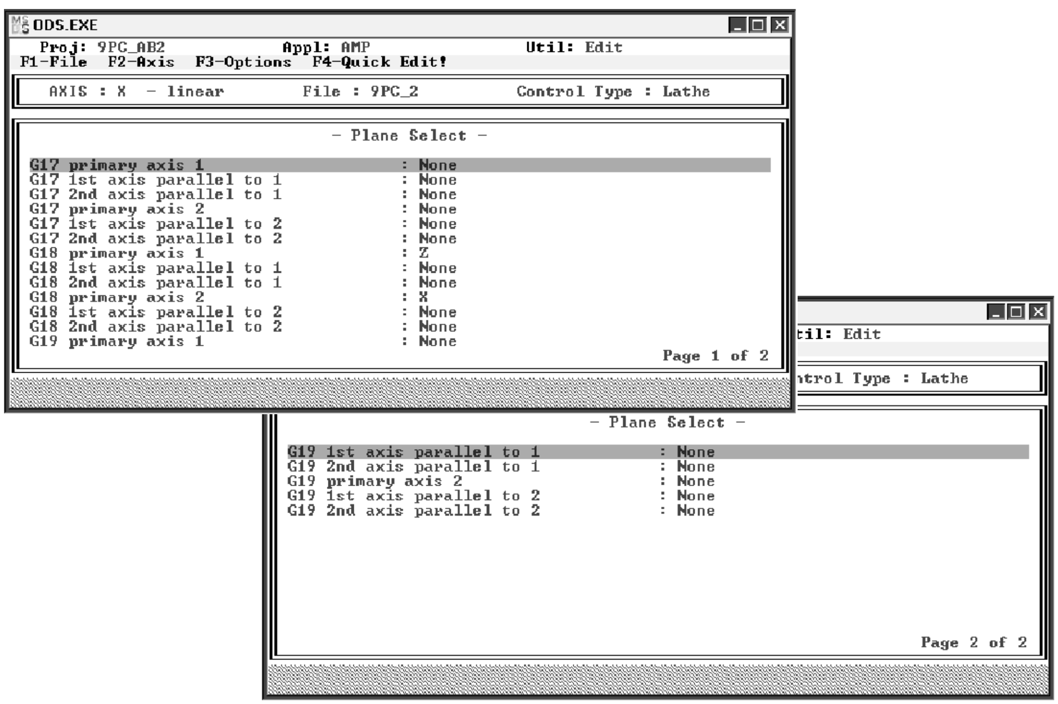
Chapter Overview

The control has a number of features that operate in specific planes. For that reason, it is frequently necessary to change the active plane by using a G17, G18, or G19. These G-codes may be used to activate 3 primary planes. These G-codes may also be used to activate up to four parallel planes to each of the primary planes.

Important: These parameters use the axis names that were assigned in chapter 3. These axis names should have been determined before any of the parameters in this chapter are set.

When using parallel planes, only two axes may be selected as parallel to each of the primary axes in the primary plane.

When you select “Plane Select” from the main menu screen, these screens become available:



Plane Select Parneters

These subsections describe the parameters that are used to determine the planes selected on the control when the G17, G18, and G19 codes are programmed.

These parameters are available for plane selection. Note that the parameters for G17, G18, and G19 have been combined in these subsections. This is because the parameters are basically identical, with the exception of the G-code that calls the plane.

Parameter:	Page:
(G17, G18, G19) Primary Axis 1	17-3
(G17, G18, G19) 1st Axis Parallel to 1	17-4
(G17, G18, G19) 2nd Axis Parallel to 1	17-5
(G17, G18, G19) Primary Axis 2	17-6
(G17, G18, G19) 1st Axis Parallel to 2	17-7
(G17, G18, G19) 2nd Axis Parallel to 2	17-8

ATTENTION: Indicates circumstances or practices that can lead to personal injury as well as damage to the control, the machine, or other equipment.

ATTENTION: Most cycles on the control are plane dependent. Care should be taken when defining your coordinate planes. Keep in mind that the axis you define as the 1st axis in the plane is the abscissa, and the 2nd axis defining the plane is the ordinate. The order these axes are assigned is significant to the operation of fixed cycles, i.e., ZX plane is not the same as the XZ plane.

If you intend to match your control's operation identically to the description in the user's manual, you must use these primary plane definitions:

Plane Select Parameters	Mill/Lathe Applications	
	Mill	Lathe
G17 Primary Axis 1	X	none
G17 Primary Axis 2	Y	none
G18 Primary Axis 1	Z	Z
G18 Primary Axis 2	X	X
G19 Primary Axis 1	Y	none
G19 Primary Axis 2	Z	none

Changing the preceding definitions can have a dramatic change on operation. For example, switching a lathe from the standard ZX plane to the reversed XZ plane causes:

- G02 circular interpolation clockwise and G03 circular interpolation counterclockwise to reverse. G02 becomes circular interpolation counterclockwise and G03 becomes circular interpolation clockwise.

- fixed cycle operation rotates 90 degrees. For example, on a lathe, a normal straight threading operation that cuts threads in the Z axis rotates to cut face threads along the X axis. The Z axis move becomes the infeed into the thread.
- other plane dependent features to reverse their operation.

(G17, G18, G19) Primary Axis 1

Function

Use this parameter to determine the first primary axis that is used to make up the plane called by G17, G18, or G19. This parameter sets which axis is first in the plane. For example, if the G17 plane is to be set as the XY plane, "G17 primary axis 1" should be set as X. If the G17 plane is to be set as the YX plane, "G17 primary axis 1" should be set as Y.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that the plane that is selected with that G-code is not used.

Parameter	Parameter Number	
	Mill	Lathe
G17 primary axis 1	[541]	[570]
G18 primary axis 1	[547]	[547]
G19 primary axis 1	[553]	[576]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

(G17, G18, G19) 1st Axis Parallel to 1

Function

Use this parameter to determine an axis that is parallel to primary axis 1 for this G-code. If no parallel axes are used, this parameter is set as "None."

Up to 4 parallel axes may be set for a select plane. Up to two parallel axes for each primary axis may be set. These parameters set the first of the two possible axes that are parallel to the first primary axis in the G17, G18, or G19 planes.

A parallel plane is selected in a part program by entering the name of the axis that is parallel to the primary axis in the G-code block that calls the plane. For example, if the G17 primary plane is the XY plane and U is determined with this parameter to be parallel to the primary axis 1, programming

```
G17 U1;
```

would make the UY plane active.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that there is no parallel axis to that primary axis in the plane selected with that G-code.

Parameter	Parameter Number	
	Mill	Lathe
G17 1st axis parallel to 1	[542]	[571]
G18 1st axis parallel to 1	[548]	[548]
G19 1st axis parallel to 1	[554]	[577]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

Important: If you are performing planar functions on parallel axes, both the parallel axis and the primary axis must be defined to use the same integrand word.

(G17, G18, G19) 2nd Axis Parallel to 1

Function

Use this parameter to determine an axis that is parallel to primary axis 1 for this G-code. If no primary axes are used, this parameter is set as "None."

Up to 4 parallel axes can be set for a select plane. Up to two parallel axes for each primary axis can be set. These parameters set the second of the two possible axes that are parallel to the first primary axis in the G17, G18, or G19 planes.

A parallel plane is selected in a part program by entering the name of the axis that is parallel to the primary axis in the G-code block that calls the plane. For example, if the G17 primary plane is the XY plane, and V is determined with this parameter to be parallel to the primary axis 1, programming

```
G17 V1;
```

would make the VY plane active.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that there are not two axes parallel to that primary axis in the plane selected with that G-code.

Important: Do not use this parameter if a value has not been set for the parameter that assigns the 1st axis parallel to a primary axis.

Parameter	Parameter Number	
	Mill	Lathe
G17 2nd axis parallel to 1	[543]	[572]
G18 2nd axis parallel to 1	[549]	[549]
G19 2nd axis parallel to 1	[555]	[578]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

Important: If you are performing planar functions on parallel axes, both the parallel axis and the primary axis must be defined to use the same integrand word.

(G17, G18, 19) Primary Axis 2

Function

This parameter is used to determine the second primary axis that is used to make up the plane called by G17, G18, or G19. This parameter sets which axis is second in the plane. For example, if the G17 plane is to be set as the XY plane, the parameter "G17 primary axis 2" should be set as Y. If the G17 plane is to be set as the YX plane, the parameter "G17 primary axis 2" should be set as X.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that the plane that is selected with that G-code is not used.

Parameter	Parameter Number	
	Mill	Lathe
G17 primary axis 2	[544]	[573]
G18 primary axis 2	[550]	[550]
G19 primary axis 2	[556]	[579]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

(G17, G18, G19) 1st Axis Parallel to 2

Function

Use this parameter to determine an axis that is parallel to primary axis 2 for this G-code. If no parallel axes are used, this parameter is set as "None."

Up to 4 parallel axes may be set for a select plane. Up to two parallel axes for each primary axis may be set. These parameters set the first of the two possible axes that are parallel to the second primary axis in the G17, G18, or G19 planes.

A parallel plane is selected in a part program by entering the name of the axis that is parallel to the primary axis in the G-code block that calls the plane. For example, if the G17 primary plane is the XY plane and U is determined with this parameter to be parallel to the primary axis 2, programming

```
G17 U1;
```

would make the XU plane active.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that there is no parallel axis to that primary axis in the plane selected with that G-code.

Parameter	Parameter Number	
	Mill	Lathe
G17 1st axis parallel to 2	[545]	[574]
G18 1st axis parallel to 2	[551]	[551]
G19 1st axis parallel to 2	[557]	[580]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

Important: If you are performing planar functions on parallel axes, both the parallel axis and the primary axis must be defined to use the same integrand word.

(G17, G18, G19) 2nd Axis Parallel to 2

Function

Use this parameter to determine an axis that is parallel to primary axis 2 for this G-code. If no parallel axes are used, this parameter is set as "None."

Up to 4 parallel axes may be set for a select plane. Up to two parallel axes for each primary axis may be set. These parameters set the second of the two possible axes that are parallel to the second primary axis in the G17, G18, or G19 planes.

A parallel plane is selected in a part program by entering the name of the axis that is parallel to the primary axis in the G-code block that calls the plane. For example, if the G17 primary plane is the XY plane and V is set with this parameter to be parallel to the primary axis 2, programming

```
G17 V1;
```

would make the XV plane active.

A value for this parameter may be chosen from these allowable axis names: A, B, C, U, V, W, X, Y, Z, $B, $C, $X, $Y, or $Z. A value of "None" may also be selected indicating that there are not two axes parallel to that primary axis in the plane selected with that G-code.

Important: Do not use this parameter if a value has not been set for the parameter that assigns the 1st axis parallel to a primary axis.

Parameter	Parameter Number	
	Mill	Lathe
G17 2nd axis parallel to 2	[546]	[575]
G18 2nd axis parallel to 2	[552]	[552]
G19 2nd axis parallel to 2	[558]	[581]

Range

Selection	Results	Selection	Results
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

This is a global parameter; the value set here applies to all axes.

Important: If you are performing planar functions on parallel axes, both the parallel axis and the primary axis must be defined to use the same integrand word.

END OF CHAPTER

Power-up G-code Parameters

Chapter Overview

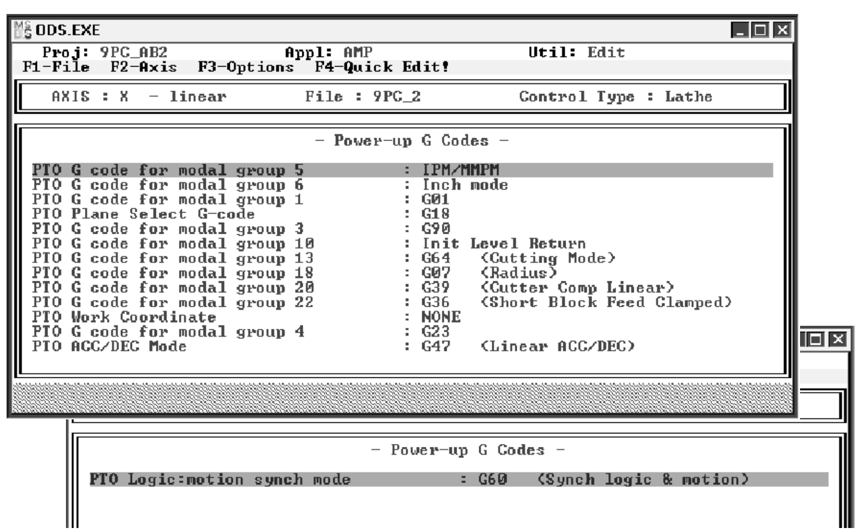
This chapter covers the Power-up G-code parameters. These parameters specify which G-code is the default G-code for a specific modal G-code group. These default G-codes become effective when the control is powered up or when a control reset is performed. A control reset does not change inch/metric mode from its last programmed state. The parameter for the inch/metric PTO state (Group 6) is only applied at power up.

They may also become effective when the control executes an end-of-program (M02 or M30) or when an E-STOP reset is executed. This depends on the values of AMP parameters **Reset M- and G-codes on M02/M30** and **Control Reset on E-Stop Reset** found in chapter 36.

Any other G-code group not discussed here is not configurable. The default G-code for other G-code groups is fixed and may not be changed. The default G-codes may be displayed on the control by selecting the **{G CODE STATUS}** screen. Refer to your programming and operation manual for more information.

Important: The operating G-code for a specific group may be altered at any time on the control with program commands. These override the default G-codes until one of the operations that reactivates the default G-codes is performed.

Access these parameters by selecting the "Power-up G Codes" parameter group displayed on the main AMP menu screen. When you select the "Power-up G Codes" parameter group, the workstation displays this screen:



Your screens may differ slightly, depending on your application.

PTO G-code

These parameters are covered on these pages:

Parameter:	Page:
PTO G-code for modal group 5	18-2
PTO G-code for modal group 6	18-3
PTO G-code for modal group 1	18-3
PTO plane select G-code	18-4
PTO G-code for modal group 3	18-5
PTO G-code for modal group 8	18-6
PTO G-code for modal group 18	18-6
PTO work coordinate	18-7
PTO G-code for modal group 4	18-7
PTO G-code for modal group 10	18-8
PTO G-code for modal group 13	18-8
PTO G-code for modal group 20	18-9
PTO G-code for modal group 22	18-10
PTO ACC/DEC mode	18-11
PTO G-code for modal group 25	18-12

For details on specific G-codes and their meanings, refer to one of your programming and operation manuals.

PTO G-code For Modal Group5

Function

Use this parameter to select the default G-code for modal group 5. The selected G-code specifies whether the control operates under feedrate-per-minute or feedrate-per-revolution modes at power-up or reset (mill or grinder control types allow the selection of inverse time feed mode using a G93 in a program).

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
N/A	N/A	N/A	G93	Feedrates programmed in inverse time feed
G98	G94	G94	G94	Feedrates programmed in feed-per-minute
G99	G95	G95	G95	Feedrates programmed in feed-per-revolution

Axis	Parameter Number
	Single Process
All	[525]

Range

Selection	Result
(a)	IPM/MMPM
(b)	IPR/MMPR

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code For Modal Group 6

Function

This parameter selects the default G-code for modal group 6. The selected G-code specifies whether the control operates under inch or metric mode at power -up only. A control reset will not re-establish this mode. The last programmed state remains in effect.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G20	G20	G70	G20	Inch system
G21	G21	G71	G21	Metric system

Axis	Parameter Number
	Single Process
All	[526]

Range

Selection	Result
(a)	Inch mode
(b)	Metric mode

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code For Modal Group1

Function

This parameter selects the default G-code for modal group 1. The selected G-code specifies whether the control operates under G-code G00 (rapid positioning) or G01 (linear interpolation) at power-up or reset.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G00	G00	G00	G00	Rapid Positioning
G01	G01	G01	G01	Linear Interpolation

Axis	Parameter Number
	Single Process
All	[521]

Range

Selection	Result
(a)	G00
(b)	G01

Notes

This is a global parameter. The value set here applies to all axes.

PTO Plane Select G-code

Function

This parameter selects the default G-code for modal group 2. The selected G-code (G17, G18, G19) specifies the primary machining plane of the control at power-up or reset.

Axis	Number	Control Type
All	[522]	Mill
	[564]	Lathe

Range

Selection	Result
(a)	G17
(b)	G18
(c)	G19

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code For Modal Group 3

Function

This parameter selects the default G-code for modal group 3. The selected G-code specifies whether the control operates under G-code G90 (Absolute) or G91 (Incremental) positioning at power-up or reset.

Important: When control type "Lathe" and G-code system A is selected, this parameter is ignored. Absolute or incremental moves are determined by the axis word that is used to program the move.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
---	G90	G90	G90	Absolute Mode
---	G91	G91	G91	Incremental Mode

Axis	Parameter Number
	Single Process
All	[523]

Range

Selection	Result
(a)	G90
(b)	G91

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code For Modal Group 8

Function

This parameter selects the default G-code for modal group 8. The selected G-code specifies whether the control operates under G43, G44, or G49 at power-up or reset. This parameter is used for mill and surface grinder applications only.

G-code	Active Mode
G43	Tool Length Offset, Plus
G44	Tool Length Offset, Minus
G49	Tool Length Offset Cancel

Axis	Parameter Number
	Single Process
All	[528]

Range

Selection	Result
(a)	G49 (offset cancel)
(b)	G43 (+ offset)
(c)	G44 (- offset)

PTO G-code For Modal Group 18

Function

This parameter selects the default G-code for modal group 18. The selected G-code specifies whether the control operates under G07 or G08 at power-up or reset. This parameter is used for lathe and cylindrical grinder applications only.

Lathe A	Lathe B	Lathe C	Active Mode
G07	G07	G07	Programming using radius values
G08	G08	G08	Programming using diameter values

Axis	Parameter Number
	Single Process
All	[538]

Range

Selection	Result
(a)	G07 (radius)
(b)	G08 (diameter)

Notes

This is a global parameter. The value set here applies to all axes.

PTO WorkCoordinate

Function

This parameter selects the default work coordinate system. It determines the work coordinate system that is activated at power-up, control reset, or when a G92.1 is programmed. The default work coordinate system may also be be activated when the control executes an end-of-program block (M02/M30) if the the value of parameter **Reset Coord Offsets on M02/M30** is "yes."

Axis	Parameter Number
	Single Process
All	[15]

Range

Selection	Result	Selection	Result
(a)	None	(f)	G58
(b)	G54	(g)	G59.1
(c)	G55	(h)	G59.2
(d)	G56	(i)	G59.3
(e)	G57		

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code For Modal Group 4

Function

Use this parameter to selects the default G-code for modal group 4. The G-code you select specifies which Programmable Zone will automatically activate upon power up or reset.

Axis	Parameter Number
	Single Process
All	[524]

Range

Selection	Result
(a)	G22 - Zones 2 & 3 Active
(b)	G22.1 - Zone 2 Inactive, Zone 3 Active
(c)	G23 - Zones 2 & 3 Inactive
(d)	G23.1 - Zone 2 Active, Zone 3 Inactive

Notes

This is a global parameter. The value set here applies to all axes.

PTO G-code for Modal Group 10

Function

Use this parameter to select the default G-code for modal group 10. You can use this parameter for lathe/mill applications only; however, this parameter is not active on lathe A.

The selected G-code specifies whether the control operates under G98 Init Level Return or under G99 R Point Level Return at power-up or reset.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
---	G98	G98	G98	Init Level Return
---	G99	G99	G99	R Point Level Return

Axis	Parameter Number
	Single Process
All	[530]

Range

Selection	Result
(a)	Init Level Return
(b)	R Point Level Return

Notes

This is a global parameter; the value set here applies to all axes.

PTO G-code for Modal Group 13

Function

Use this parameter to select the default G-code for modal group 13. You can use this parameter for lathe/mill applications.

The selected G-code specifies under which mode the control operates at power-up or reset:

- G61 Exact Stop
- G62 Auto Corner Override
- G63 Tapping
- G64 Cutting

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G61	G61	G61	G61	Exact Stop Mode
G62	G62	G62	G62	Auto Corner Override Mode
G63	G63	G63	G63	Tapping Mode
G64	G64	G64	G64	Cutting Mode

Axis	Parameter Number
	Single Process
All	[533]

Range

Selection	Result
(a)	G61 (Exact Stop Mode)
(b)	G62 (Auto Corner Override Mode)
(c)	G63 (Tapping Mode)
(d)	G64 (Cutting Mode)

Notes

This is a global parameter; the value set here applies to all axes.

PTO G-code for Modal Group 20

Function

Use this parameter to select the default G-code for modal group 20. You can use this parameter for lathe/mill applications.

The selected G-code specifies whether the control operates under G39 Cutter Comp Linear or G39.1 Cutter Comp Rounding at power-up or reset.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G39	G39	G39	G39	Cutter Comp Linear
G39.1	G39.1	G39.1	G39.1	Cutter Comp Rounding

Axis	Parameter Number
	Single Process
All	[519]

Range

Selection	Result
(a)	G39 (Cutter Comp Linear)
(b)	G39.1 (Cutter Comp Rounding)

Notes

This is a global parameter; the value set here applies to all axes.

PTO G-code for Modal Group 22

Function

Use this parameter to select the default G-code for modal group 22. You can use this parameter for lathe/mill applications.

The selected G-code specifies whether the control operates under G36 Short Block Feed Clamped or G36.1 Short Block Full Feed at power-up or reset.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G36	G36	G36	G36	Short Block Feed Clamped
G36.1	G36.1	G36.1	G36.1	Short Block Full Feed

Axis	Parameter Number		
	Single Process	Process 1	Process 2
All	[520]	[20520]	[21520]

Range

Selection	Result
(a)	G36 (Short Block Feed Clamped)
(b)	G36.1 (Short Block Full Feed)

Notes

This is a global parameter; the value set here applies to all axes.

PTO ACC/DEC Mode

Function

Use this parameter to select the default G-code for modal group 24. The selected G-code specifies the type of acc/dec mode (Linear, S-Curve, or Acc/Dec Disabled) is active.

Lathe A	Lathe B	Lathe C	Mill Control	Active Mode
G47	G47	G47	G47	Linear ACC/DEC
G47.1	G47.1	G47.1	G47.1	S-Curve ACC/DEC
G47.9	G47.9	G47.9	G47.9	ACC/DEC Disabled

Axis	Parameter Number		
	Single Process	Process 1	Process 2
All	[540]	[20540]	[21540]

Range

Selection	Result
(a)	G47 (Linear ACC/DEC)
(b)	G47.1 (S-Curve ACC/DEC)
(c)	G47.9 (ACC/DEC Disabled)

Notes

This is a global parameter; the value set here applies to all axes.

G47.9 is only programmable in AMP, not on the control via a part program block.

PTO G-code for Modal Group 25

Function

Use this parameter to select the default G-code for modal group 25. The selected G-code specifies the type of logic to Motion and part program synchronization that is active at system reset and power turn-on.

Axis	Parameter Number
	Single Process
All	[559]

Range

Selection	Result
(a)	G60 (Synchronous Logic and Motion)
(b)	G60.1 (Asynchronous Logic and Motion))
(c)	G60.2 (Autosynchronous Logic and Motion)

Notes

This is a global parameter; the value set here applies to all axes.

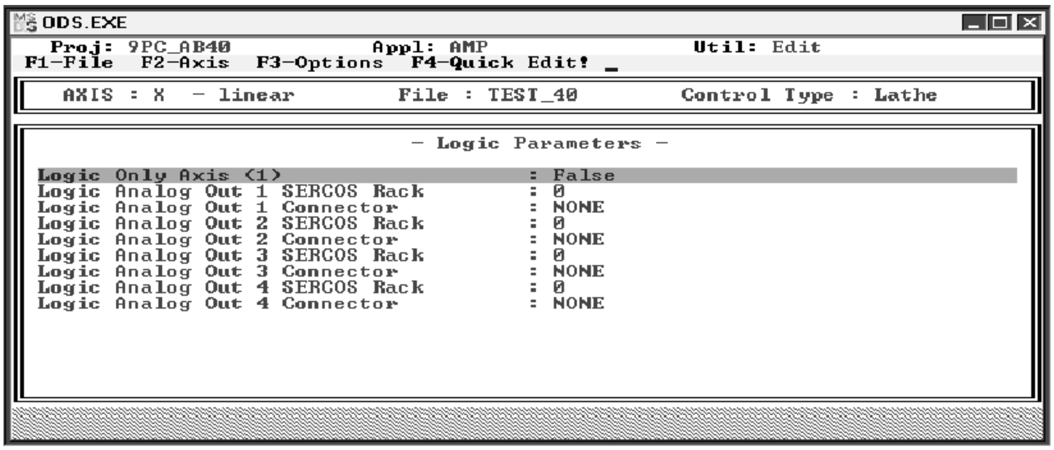
END OF CHAPTER

Logic Parameters

Chapter Overview

This chapter covers the logic-related AMP parameters. Access these parameters by selecting the **Logic Parameters** group from the AMP main menu.

The workstation displays this screen when the **Logic Parameters** group is selected:



Logic Only Axis

Function

This parameter defines whether an axis is controlled by logic only. The logic has the option to control any configured axis; however, axes selected with this parameter (logic-only axes) can be controlled only by logic. This means the operator and part programmer have no means of positioning, jogging, homing, etc. logic-only axes, except those methods provided by the system installer with the logic program.

When this parameter is entered as:	the current axis being configured:	and:
True	is controlled entirely by the logic program	This axis does not respond to the normal part program requests or jog requests unless the logic program was written to allow these requests to be acknowledged.
False	is a normal programmable axis	This means the axis responds to part program requests and jog requests as normal. The axis can still be controlled by logic if the logic program requests that axis.

Axis	Parameter Number
(1)	[1029]
(2)	[2029]
(3)	[3029]
(4)	[4029]
(5)	[5029]
(6)	[6029]
(7)	[7029]
(8)	[8029]

Range

Selection	Result
(a)	True
(b)	False

Notes

This parameter must be set independently for each axis.

Logic Analog Output Parameters

Function

Use the logic analog output parameters to assign an analog output port on a SERCOS connected 1394 CNC Serial Drive to one of the analog output control flags (BW_AOUT00 - BW_AOUT03). Logic can control up to four analog output ports. Only one analog output port can be assigned to each logic analog output port control parameter. Refer to your *9/PC CNC Logic Reference Manual* for more details about assigning an analog output port to a logic analog output port control parameter.

Logic-controlled analog output ports must not be defined in AMP as an analog output port assigned to a fitted axis or a spindle.

Parameter	Parameter Number
Logic Analog Out 1 SERCOS Rack	[501]
Logic Analog Out 1 Connector	[502]
Logic Analog Out 2 SERCOS Rack	[503]
Logic Analog Out 2 Connector	[504]
Logic Analog Out 3 SERCOS Rack	[505]
Logic Analog Out 3 Connector	[506]
Logic Analog Out 4 SERCOS Rack	[507]
Logic Analog Out 4 Connector	[508]

For each logic analog output port, you must set two parameters: the SERCOS Rack parameter and the Connector parameter. Refer to the information below to determine the settings for each parameter type.

Range

Range for SERCOS Rack parameters:

1 to 254 corresponding to the desired 1394 CNC serial drive system module SERCOS address.

Range for the connector parameters:

Selection	Result
(a)	None
(b)	Analog Out TB2 or Aout5 (1394)
(c)	Analog Out TB3 or Aout6 (1394)

Notes

Each analog output port controlled by logic must not be defined in AMP as an analog output port assigned to a fitted axis or a spindle. Only one analog output port can be assigned to a logic parameter.

The total number of servos plus the number of analog output ports assigned may not exceed the maximum number of available servo loops.

END OF CHAPTER

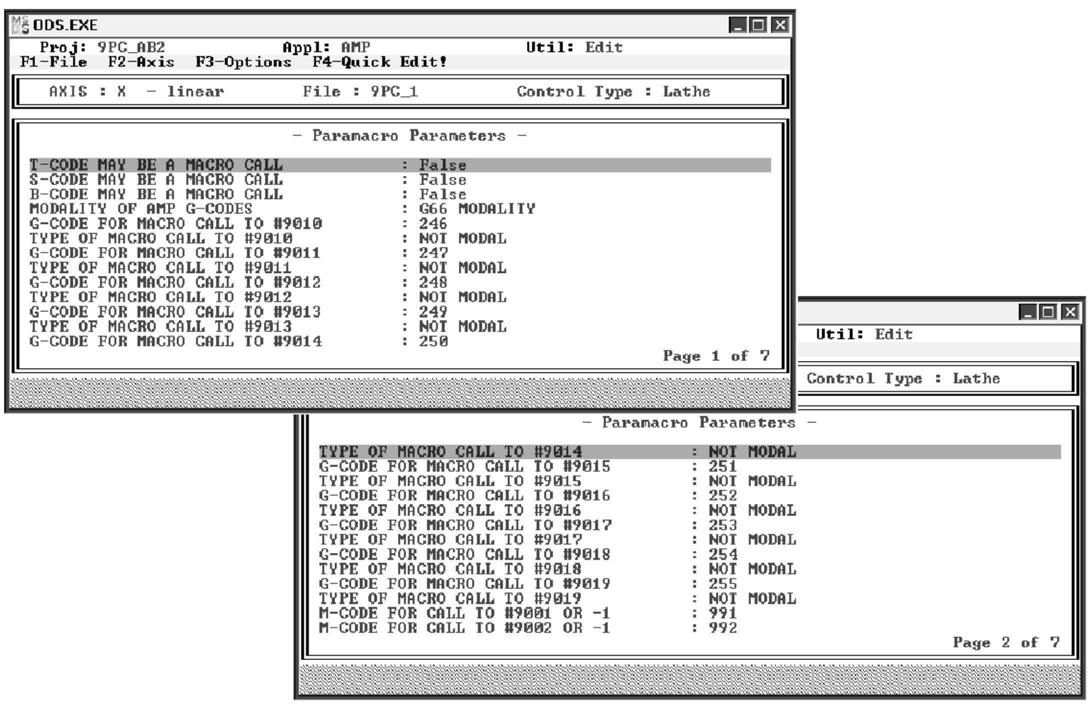
Paramacro Parameters

Chapter Overview

This section describes the parameters that are available for the paramacro feature. The parameters are broken into these major sections:

Section:	Page:
T, S, and B code paramacro calls	20-3
G-code paramacro calls	20-4
M-code paramacro calls	20-11
Calling AMP-defined G, M, S, T, or B codes	20-12

After you select "Paramacro Parameters" from the main menu screen, the workstation displays these screens, showing the available paramacro parameters:



Refer to your programming and operation manual for details on the paramacro features.

T-,S-, and B-code Parmacro Calls

This subsection describes the T, S, B – CODE MAY BE A MACRO CALL parameter that is used for T–, S–, and B–code AMP-assigned paramacro calls.

The paramacro program names that are called by a T–, S–, or B–words are:

T–word calls the paramacro program named 9000.

S–word calls the paramacro program named 9029.

B–word calls the paramacro program named 9028.

The value of a T–, S–, or B–word that calls a paramacro is determined by the value assigned to a corresponding paramacro system parameter.

T word assigned using system parameter #149

S word assigned using system parameter #147

B word assigned using system parameter #146

Important: #149, #147, and #146 refer to paramacro system parameters. These are not AMP parameters and are not set in AMP. Refer to your mill or lathe programming and operation manuals for more information.

T–, S–, B–cod May be a Macro Call

Function

Use these parameters to determine whether a paramacro program can be called in a program with a T–, S–, or B–word.

If the parameter is:	then:
True	that word may be used to call a paramacro
False	that word cannot be used to call a paramacro

Parameter	Parameter Number
T-CODE MAY BE A MACRO CALL	[33]
S-CODE MAY BE A MACRO CALL	[34]
B-CODE MAY BE A MACRO CALL	[35]

Range

Selection	Result
(a)	True
(b)	False

Notes

This is a global parameter; the value set here applies to all axes.

G-Code Paramacro Calls

This feature lets you define your own G-codes to be used in a part program. When executed, these G-codes call a specific paramacro program that you have previously written and stored in control memory.

There are two types of AMP-defined G-codes: Type I and Type II. These two types of G-codes are functionally identical. The only differences are:

- **Type I –** This type of AMP-defined G-code is restricted to calling specific program numbers ranging from 9010 through 9019. G-code numbers defined here to call a macro program must be integers; no decimal points are allowed in the G-code.
- **Type II –** This type of AMP-defined G-code allows selectable program numbers ranging from 1 through 8999. G-code numbers defined here to call a macro program can optionally contain a decimal point (for example, G120.1).

When creating a G-code to call a paramacro program, you must define:

Define	Using Type 1 Parameter	Using Type 2 Parameter
What G-code is to be used	G-code for Macro Call to 9010 – 9019 Select from: 1 – 255	G-code for Macro Call 1 – 15 Select from: 1.0 – 255.9
If G-code is modal or non-modal	Type of Macro Call to 9010 – 9019 Select from: not used/modal/not modal	Type of Macro Call 1 – 15 Select from: not used/modal/not modal
If modal, select a modal type [1] (G66 or G66.1)	Modality of AMP G-Codes	Modality of AMP G-Codes
What part program number	not configurable; fixed at 9010 through 9019	configurable; select from: 1 – 8999; Program Number for Macro Call 1 – 15

[1] Nonmodal macro calls are always executed in the same fashion as a G65 macro call

Modality of AMP G-codes

Function

Use this parameter for Type I and II AMP-defined G-codes. AMP-defined G-code paramacro calls are either modal or non-modal. Use this parameter for modal macro calls only. Whether an AMP-defined G-code paramacro call is modal or non-modal is determined by using the parameter **TYPE OF MACRO CALL TO 9010-9019** for Type I or **TYPE OF MACRO CALL 1-15** for Type II.

Modal macro calls mean that the paramacro is automatically called in other blocks until canceled (similar to the fixed cycles). There are two different forms of modal paramacro calls. Use this parameter to determine whether the modal AMP-defined G-code paramacro calls use the modal form that is associated with the G66 or the G66.1 paramacro calls. Refer to your mill or lathe programming and operation manuals for more information.

Axis	Parameter Number
All	[36]

Range

Selection	Result
(a)	G66 MODALITY
(b)	G66.1 MODALITY

Notes

This is a global parameter; the value set here applies to all axes and all AMP-assigned G-code macro calls selected as modal for both Type I
and II.

G-code for Macro Call to #9010 to #9019

Function

Use these parameters to determine the Type I AMP-defined G-codes used to call paramacros program numbers 9010 to 9019. Specify the number of the G-code that is used to call a specific paramacro number. This G-code must be an integer (no decimals allowed).

Parameter	Parameter Number
G-CODE FOR MACRO CALL TO #9010	[37]
G-CODE FOR MACRO CALL TO #9011	[39]
G-CODE FOR MACRO CALL TO #9012	[41]
G-CODE FOR MACRO CALL TO #9013	[43]
G-CODE FOR MACRO CALL TO #9014	[45]
G-CODE FOR MACRO CALL TO #9015	[47]
G-CODE FOR MACRO CALL TO #9016	[49]
G-CODE FOR MACRO CALL TO #9017	[51]
G-CODE FOR MACRO CALL TO #9018	[53]
G-CODE FOR MACRO CALL TO #9019	[55]

For example, if **G-CODE FOR MACRO CALL TO #9018** is set at a value of 100, programming a G100 in a program will call the paramacro program in control memory named 9018.

Range

1 to 255

Notes

This is a global parameter; the value set here applies to all axes.

You cannot set this parameter equal to the value of G65; G66; G66.1; G70; G71, G72, G73, G74, G75 (lathe only); and G89.1 and G89.2 (mill only). If G65, G66, or G66.1 is assigned as an AMP-defined G-code, it is ignored, and the system-defined G65, G66, or G66.1 operations will take precedence. Other system-defined G-codes can be reassigned using this parameter; however, doing so can disable their normal system-defined function.

Type of Macro Call to #9010 to #9019

Function

Use these parameters, #9010 to #9019, to determine whether a Type I AMP-assigned G-code paramacro is modal or non-modal. This parameter may also disable a paramacro from being called by a G-code if it is set at “Not Used.”

Parameter	Parameter Number
TYPE OF MACRO CALL TO #9010	[38]
TYPE OF MACRO CALL TO #9011	[40]
TYPE OF MACRO CALL TO #9012	[42]
TYPE OF MACRO CALL TO #9013	[44]
TYPE OF MACRO CALL TO #9014	[46]
TYPE OF MACRO CALL TO #9015	[48]
TYPE OF MACRO CALL TO #9016	[50]
TYPE OF MACRO CALL TO #9017	[52]
TYPE OF MACRO CALL TO #9018	[54]
TYPE OF MACRO CALL TO #9019	[56]

If this parameter is set at:	this means:
NOT USED	the control may not call this macro number with an AMP-assigned G-code macro call
NOT MODAL	the paramacro called is not a modal paramacro and is executed as if called by a G65
MODAL	the paramacro called is modal and is executed as if called by either a G66 or G66.1 paramacro call (as determined by the parameter MODALITY OF AMP G-CODES on page 20-11)

Range

Selection	Result
(a)	NOT USED
(b)	NOT MODAL
(c)	MODAL

Notes

This is a global parameter; the value set here applies to all axes.

G-code for Macro Call 1 to 15

Function

Use these parameters to define the Type II AMP-defined G-code used to call your paramacro program. Up to 15 different G-codes may be defined to call 15 different paramacro programs. Specify the number of the G-code that is used to call a specific paramacro program. These G-codes can contain up to one decimal place.

Parameter	Parameter Number
G-CODE FOR MACRO CALL 1	[700]
G-CODE FOR MACRO CALL 2	[703]
G-CODE FOR MACRO CALL 3	[706]
G-CODE FOR MACRO CALL 4	[709]
G-CODE FOR MACRO CALL 5	[712]
G-CODE FOR MACRO CALL 6	[715]
G-CODE FOR MACRO CALL 7	[718]
G-CODE FOR MACRO CALL 8	[721]
G-CODE FOR MACRO CALL 9	[724]
G-CODE FOR MACRO CALL 10	[727]
G-CODE FOR MACRO CALL 11	[730]
G-CODE FOR MACRO CALL 12	[733]
G-CODE FOR MACRO CALL 13	[736]
G-CODE FOR MACRO CALL 14	[739]
G-CODE FOR MACRO CALL 15	[742]

For example, assume that the parameter **G-CODE FOR MACRO CALL 1** is set at a value of 231.6 and the parameter **PROGRAM NUMBER FOR MACRO CALL 1** is defined as program number 100. Programming a G231.6 in a program calls the paramacro program named 100 from control memory. If, however, the **TYPE OF MACRO CALL** is set to NOT USED, any values assigned to the parameter are ignored.

Assigning a decimal value of 0 gets the same result as not having a decimal-pointed value, e.g., a G231.0 is functionally the same as a G231.

Range

1.0 to 255.9

Notes

You cannot set this parameter equal to the value of G65; G66; G66.1; G70; G71, G72, G73, G74, G75 (lathe only); and G89.1 and G89.2 (mill only). If G65, G66, or G66.1 is assigned as an AMP-defined G-code, it is ignored, and the system-defined G65, G66, or G66.1 operations will take precedence. Other system-defined G-codes can be reassigned using this parameter; however, doing so can disable their normal system-defined function.

For example, assigning the value of G31 to this parameter in a main level program will have the following effects:

- G31 (External Skip Function 1) can no longer be programmed. In effect, the feature is no longer available in a main program.
- Whatever paramacro is now assigned to this G-code will be executed as defined in AMP.

In a nested macro, an AMP-defined G-code can work as a system-defined G-code or work as a macro call as determined with the parameter **Func of Called AMP Macro**.

Type of Macro Call 1 to 15

Function

Use these parameters to determine whether a Type II paramacro called by an AMP-assigned G-code is modal or non-modal. These parameters can also be used to disable a paramacro from being called by a G-code when set to "Not Used."

Parameter	Parameter Number
TYPE OF MACRO CALL 1	[701]
TYPE OF MACRO CALL 2	[704]
TYPE OF MACRO CALL 3	[707]
TYPE OF MACRO CALL 4	[710]
TYPE OF MACRO CALL 5	[713]
TYPE OF MACRO CALL 6	[716]
TYPE OF MACRO CALL 7	[719]
TYPE OF MACRO CALL 8	[722]
TYPE OF MACRO CALL 9	[725]
TYPE OF MACRO CALL 10	[728]
TYPE OF MACRO CALL 11	[731]
TYPE OF MACRO CALL 12	[734]
TYPE OF MACRO CALL 13	[737]
TYPE OF MACRO CALL 14	[740]
TYPE OF MACRO CALL 15	[743]

Range

Selection	Result
(a)	NOT USED
(b)	NOT MODAL
(c)	MODAL

If this parameter is set at::	this means:
NOT USED	this AMP-assigned G-code macro call is disabled and is not used. Any values assigned to the parameters G-CODE FOR MACRO CALL and PROGRAM NUMBER FOR MACRO CALL are ignored.
NOT MODAL	the G-code paramacro called with this is executed as non-modal (executed as a G65).
MODAL	the G-code paramacro call is executed as a modal G-code. Modal type is determined by the parameter MODALITY OF AMP G-CODES

Important: These G-code types must be set sequentially. If any of the 15 G-code types are set to "NOT USED," the control assumes the rest of the macro calls in the group of 15 are also "NOT USED." For example, if TYPE OF MACRO CALL 6 is set to "NOT USED," the control assumes TYPE OF MACRO CALL 7 - 15 are also "NOT USED" regardless of how 7 - 15 are set.

This is a global parameter; the value set here applies to all axes.

Program Numbers for Macro Call 1 to 15

Function

Use these parameters to define the program number called by the Type II paramacro call. The program name entered here must be stored in memory at the time the G-code macro call is executed.

The program you call with a G-code macro must have a 5-digit numeric program name preceded by the letter O. When defining a program name using this parameter, the O is not specified.

Parameter	Parameter Number
PROGRAM NUMBER FOR MACRO CALL 1	[702]
PROGRAM NUMBER FOR MACRO CALL 2	[705]
PROGRAM NUMBER FOR MACRO CALL 3	[708]
PROGRAM NUMBER FOR MACRO CALL 4	[711]
PROGRAM NUMBER FOR MACRO CALL 5	[714]
PROGRAM NUMBER FOR MACRO CALL 6	[717]
PROGRAM NUMBER FOR MACRO CALL 7	[720]
PROGRAM NUMBER FOR MACRO CALL 8	[723]
PROGRAM NUMBER FOR MACRO CALL 9	[726]
PROGRAM NUMBER FOR MACRO CALL 10	[729]
PROGRAM NUMBER FOR MACRO CALL 11	[732]
PROGRAM NUMBER FOR MACRO CALL 12	[735]
PROGRAM NUMBER FOR MACRO CALL 13	[738]
PROGRAM NUMBER FOR MACRO CALL 14	[741]
PROGRAM NUMBER FOR MACRO CALL 15	[744]

Range

1 to 8999

Notes

This is a global parameter; the value set here applies to all axes.

M-code Paramacro Calls

This section describes the AMP parameters that are used when M-codes are required to call a paramacro. AMP-defined M-code paramacro calls may call only nonmodal paramacros (executed similarly to a G65 paramacro call). Refer to your mill or lathe 9/PC Operation and Programming Manual for more information.

M-code for Call to #9001 to #9009 or -1

Function

These parameters are used to determine the M-codes used to call paramacro programs. Specify the number of the M-code that is used to call a specific paramacro program number for these parameters.

Parameter	Parameter Number
M-CODE FOR CALL TO #9001 OR -1	[57]
M-CODE FOR CALL TO #9002 OR -1	[58]
M-CODE FOR CALL TO #9003 OR -1	[59]
M-CODE FOR CALL TO #9004 OR -1	[60]
M-CODE FOR CALL TO #9005 OR -1	[61]
M-CODE FOR CALL TO #9006 OR -1	[62]
M-CODE FOR CALL TO #9007 OR -1	[63]
M-CODE FOR CALL TO #9008 OR -1	[64]
M-CODE FOR CALL TO #9009 OR -1	[65]

For example, if **M-CODE FOR MACRO CALL TO #9009** is set at a value of 100, programming a M100 in a program calls the paramacro number 9009.

Assigning a value of -1 to this parameter disables the M-code macro call to this paramacro program number. Disabling this M-code macro call can increase system performance.

Range

-1 to 999

Notes

This is a global parameter; the value set here applies to all axes

Calling AMP-defined G-,M-S-,T-, or B-codes

Use these AMP parameters to modify how the control reacts when a macro call is made by a G-, M-, T-, S-, or B-code. Specifically they are intended to give more definition to the operation of the codes when conflicts occur between other features and these G-, M-, T-, S-, or B-codes. These parameters give more control over the nesting of some of these macro calls and allow these calls to be prohibited in MDI.

Func of AMP-defined G/M in MDI

Function

This parameter impacts the use of AMP-defined G- and M-codes that call a paramacro program (see previous parameters in this chapter) while in MDI mode. The two options for this parameter are:

- Works as a macro call – When "works as a macro call" is selected, G- or M-codes that are assigned in AMP to call a paramacro will call that paramacro. This is, of course, provided that the macro being called is a valid paramacro number and no nesting violations occur (refer to the parameter **Func of called AMP macro**). Any other operations that would normally be performed by that G- or M-code (as defined by the control as a standard code, logic, or some other AMP feature) are ignored and are not performed.
- Works as the system defined code – When "works as the system defined code" is selected, any attempt to call an AMP-defined G- or M-code macro call in the MDI modify is ignored. If that G- or M-code is defined by some other feature (either the control as a standard code, logic, or some other AMP feature), it is executed as defined by that feature. If the G- or M-code is not defined by one of these features, the system generates an error indicating that an invalid code has been programmed (even if that code is defined as a paramacro call).

Parameter	Parameter Number
Func of AMP-defined G/M in MDI	[12]

Range

Selection	Result
(a)	works as a macro call
(b)	works as the system-defined code

Notes

This is a global parameter; the value set here applies to all axes.

Note that T–, S–, and B–codes that are defined to call paramacros always call the paramacro, and ignore the system-defined function in MDI unless a nesting violation has occurred as shown in Table 21.A and Table 21.B. When a nesting violation occurs, the control executes the system-defined function if one is defined. A system-defined function is any G- or M-code that is inherently defined on the control, defined in AMP, or in the logic ladder.

Func of Called AMP Macro

Function

This parameter impacts the nesting of AMP-defined G-, M-, T-, S-, or B-codes that call a paramacro program (see previous parameters in this chapter). The two options for this parameter are:

- Works as a macro call - When "works as a macro call" is selected, G-, M-, T-, S-, or B-code macro calls that are nested and called by other G-, M-, T-, S-, or B-code macro calls allow nesting as shown in Table 21.A.

Table 21.A
Works as a Macro Call

CALLING-PROGRAM	TYPE OF MACRO NESTED [1]			
	G65, G66,or G66.1	AMP-G	AMP-M	AMP-T S or B
G65, G66 or G66.1	Yes	Yes	Yes	Yes
AMP-G-code	Yes	No	Yes	Yes
AMP-M-code	Yes	Yes	No	No
AMP-T, S or B code	Yes	yes	No	No

[1] What Yes/No means:
Yes -- the macro type across the top row may be called from the macro type down the left column.
No -- the macro type across the top row may **not** be called from the macro type down the left column. When this nesting is attempted, the control executes any other operation that would normally be performed by that G, M, T, S, or B code (as defined by the control as a standard code, logic, or some other AMP feature), and the paramacro call normally made by that code is not performed.

- Works as the system defined code - When "works as the system defined code" is selected, G, M, T, S, or B code macro calls that are nested and called by other G, M, T, S, or B code macro calls allow nesting as shown in Table 21.B.

Table 21.B
Works as the System-defined Code

CALLING-PROGRAM	TYPE OF MACRO NESTED [1]			
	G65, G66,or G66.1	AMP-G	AMP-M	AMP-T S or B
G65, G66 or G66.1	Yes	Yes	Yes	Yes
AMP G-code	Yes	No	No	No
AMP M-code	Yes	No	No	No
AMP-T, S or B code	Yes	No	No	No

[1] What Yes/No means:
Yes -- the macro type across the top row may be called from the macro type down the left column.
No -- the macro type across the top row may **not** be called from the macro type down the left column. When this nesting is attempted, the control executes any other operation that would normally be performed by that G, M, T, S, or B code (as defined by the control as a standard code, logic, or some other AMP feature); and the paramacro call normally made by that code is not performed.

Important: feature); and the macro call is not made. If no other function is found that uses that G-, M-, T-, S-, or B-code, the control generates an error.

Parameter	Parameter Number
Func of called AMP macro	[13]

Range

Selection	Result
(a)	works as a macro call
(b)	works as the system-defined code

Notes

This is a global parameter; the value set here applies to all axes.

END OF CHAPTER

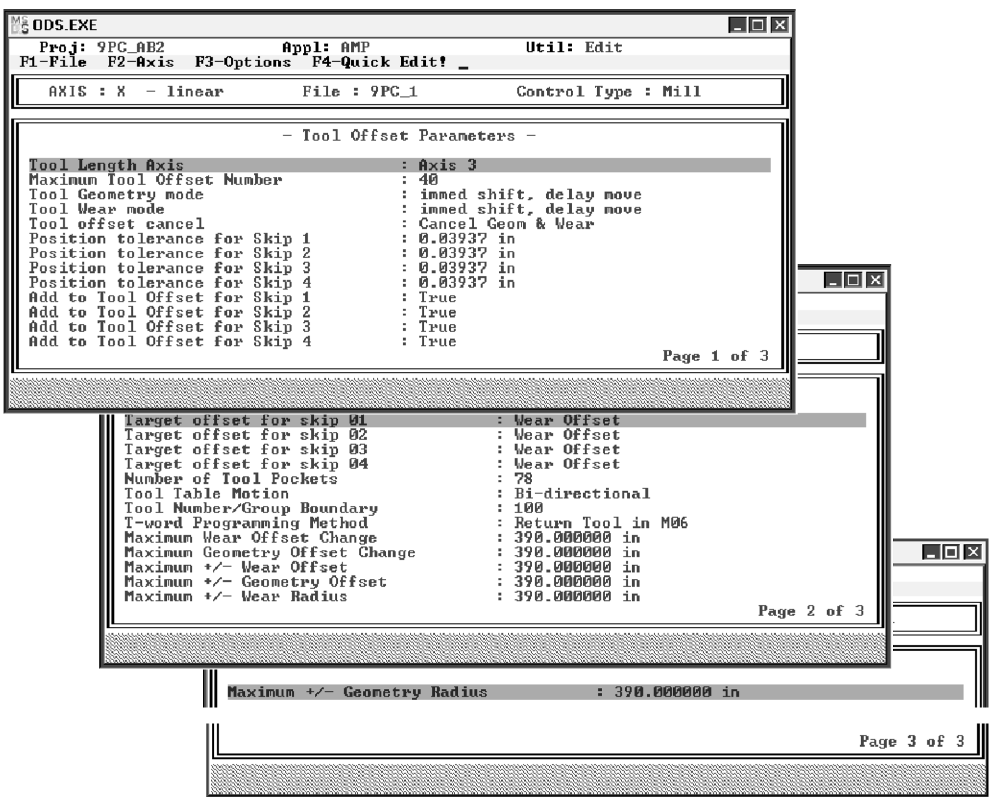
Tool Offset Parameters

Chapter Overview

Modern machining processes usually require a machine that is capable of selecting different tools. Typically tools are mounted in a turret or chuck and assigned tool numbers. These sections describe the parameters that are used to set up tool offsets. Tool offsets let the programmer basically ignore tool mounting in the part program. Tool offset parameters are broken into these sections:

Parameter:	Page:
Basic Tool Offset Setup Parameters	21-2
G37 Tool gauging Cycle Parameters	21-9
Tool Magazine/Turret Parameters	21-12
Tool Life Monitor Parameters	21-14

When you select “Tool Offset Parameters” from the main menu screen, these screens become available:



Important: If the selected control type is “Lathe,” the parameter Tool Length Axis is replaced by the parameter T-code Format, and the parameter T-word “Programming Method” is not available.

Basic Tool Offset Setup Parameters

These subsections describe the basic parameters that should be set to properly use the tool offset feature:

- T-code Format
- Tool Length Axis
- Maximum Tool Offset Number
- Tool Geometry Mode
- Tool Wear Mode
- Tool Offset Cancel

T-code Format

Function

Important: Use this parameter for lathe types only. The T-word format is not configurable for mill versions.

Use this parameter to determine the format for a T-word in a program. This parameter determines which digits of the T-word call the tool number, tool wear number, and tool geometry number.

Refer to your programming and operation manual for more information.

This table gives the different format types.

* Format Type	Wear Offset #	Geometry Offset #
(1) 1 digit Geom + Wear	last digit	same as wear #
(2) 2 digit Geom + Wear	last two digits	same as wear #
(3) 3 digit Geom + Wear	last three digits	same as wear #
(4) 1 digit Wear	last digit	same as tool #
(5) 2 digit Wear	last two digits	same as tool #
(6) 3 digit Wear	last three digits	same as tool #

For the above table, any digits in the T-word that are to the left of the wear offset number are used as the tool number. The geometry number is either the same as the tool number or the same as the wear number.

Axis	Parameter Number
	Single Process
All	[205]

Range

Selection	Results	Selection	Results
(a)	1 digit Geom + Wear	(d)	1 digit Wear
(b)	2 digit Geom + Wear	(e)	2 digit Wear
(c)	3 digit Geom + Wear	(f)	3 digit Wear

Notes

This is a global parameter; the value set here applies to all axes.

This parameter is not available for mill applications.

Tool Length Axis

Function

This parameter is for mill versions only. The tool length axis (or axes) for lathes is determined by axis words that are programmed in the block with the T-word. If, for a lathe version, a T-word is programmed with no axis words, then all axes that have a tool length value assigned to them are offset.

This parameter determines what axis is offset by the tool length values on mill versions only. The tool length values are entered in the tool offset tables (geometry and wear) and are called by an H-word in a program.

Axis	Parameter Number
	Single Process
All	[200]

Range

Selection	Results
(a)	Axis 1
(b)	Axis 2
(c)	Axis 3
(d)	Axis 4
(e)	Axis 5
(f)	Axis 6
(g)	Axis 7
(h)	Axis 8

Notes

The part programer can temporarily override the axis selected here by programming a G43.1. Refer to your programming and operation manual for more information.

This is a global parameter; the value set here applies to all axes.

Maximum Tool Offset

Function

Use this parameter to determine the maximum number of tool offsets that may be entered in the tool offset tables. Any program that calls an offset number greater than this parameter generates an error or may be used as a tool life group number. The control does not allow the entry of a tool offset number in the offset tables that is greater than this parameter.

Axis	Parameter Number
	Single Process
All	[201]

Range

40 to 200 tool offsets in lathe or mill applications .

Notes

This is a global parameter; the value set here applies to all axes.

Tool Geometry Mode

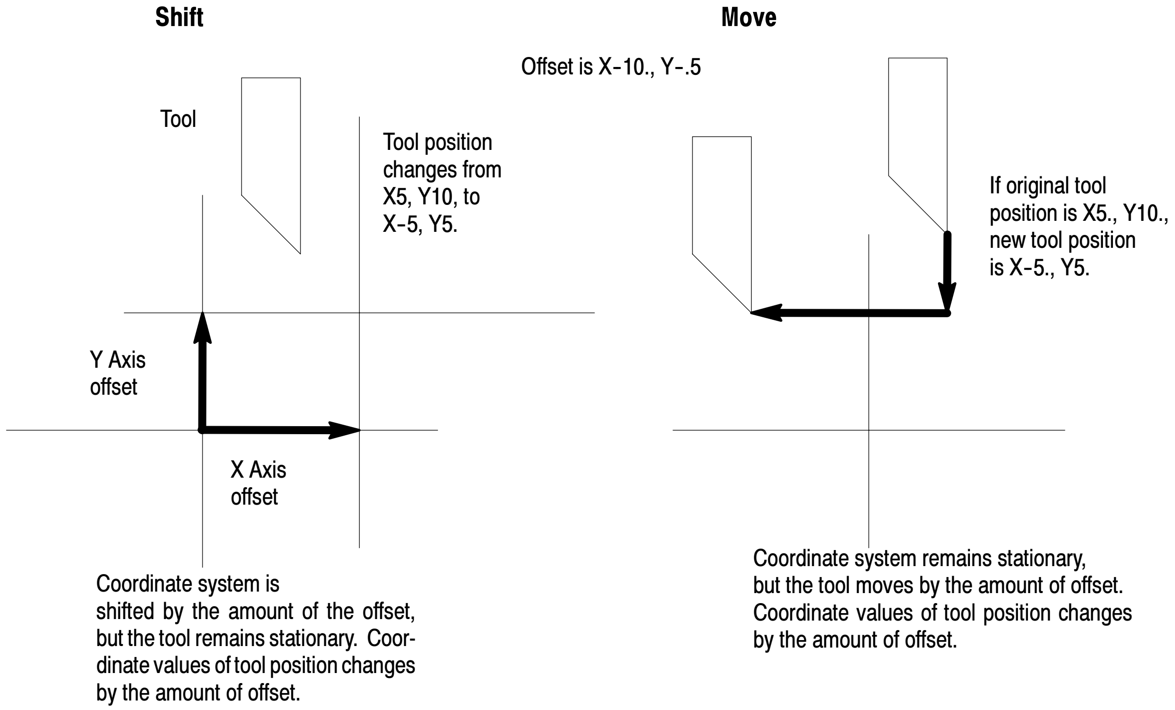
Function

Use this feature to determine how the geometry tool offset takes place. This parameter determines when the work coordinate system is shifted, and when the axis position is returned to the programmed position after the coordinate system shifts.

Axis	Parameter Number
	Single Process
All	[202]

Figure 21.1 defines the difference between “shift” and “move.”

Figure 21.1
Shift vs. Move



immed shift – When this parameter is set for "immed shift," the work coordinate system shifts when the control executes the block that activates, deactivates, or changes the tool length value.

delay shift – When this parameter is set for "delay shift," the work coordinate system does not shift until an axis motion takes place using the axis that the tool length offset applies to. This includes the block that activates, deactivates, or changes the tool length value.

immed move – When the control executes a tool length offset, the position of the cutting tool must change in the machine coordinate system and must reflect no change in the tool position in the work coordinate system. This requires tool motion to take place. When this parameter is set for "immed move," the tool moves to its offset position when the control executes the block that activates, deactivates, or changes the tool length value.

delay move – When the control executes a tool length offset, the position of the cutting tool must change in the machine coordinate system and must reflect no change in tool position in the work coordinate system. This requires tool motion to take place. When this parameter is set for "delay move," the tool motion does not take place until an axis motion takes place using the axis that the tool length offset applies to. This includes the block that activates, deactivates, or changes the tool length value.

Range

Selection	Results
(a)	immed shift, immed move
(b)	immed shift, delay move
(c)	delay shift, delay move

Notes

This is a global parameter; the value set here applies to all axes. This parameter does not affect the operation of the cutter compensation or the TTRC features in any way.

Important: It is incompatible to select a geometry offset mode of "immed shift, immed move" when the wear offset mode is configured as "immed shift, delay move" or a geometry offset mode of "immed shift, delay move" and a wear offset mode of "immed, shift immed move". An error message (INCOMPATIBLE TOOL ACTIVATION MODES) is displayed and the control is held in E-Stop when the wear and geometry activation mode is incompatible.

Tool Wear Mode

Function

Use this feature to determine how the wear tool offset takes place. This parameter determines when the work coordinate system is shifted, and when the axis position is returned to the programmed position after the coordinate system shifts.

Axis	Parameter Number
	Single Process
All	[203]

immed shift – When this parameter is set for "immed shift," the work coordinate system shifts when the control executes the block that activates, deactivates, or changes the tool length value.

delay shift – When this parameter is set for "delay shift," the work coordinate system does not shift until an axis motion takes place using the axis that the tool length offset applies to. This includes the block that activates, deactivates, or changes the tool length value.

immed move – When the control executes a tool length offset, the position of the cutting tool must change in the machine coordinate system and must reflect no change in the tool position in the work coordinate system. This requires tool motion to take place. When this parameter is set for “immed move,” the tool moves to its offset position when the control executes the block that activates, deactivates, or changes the tool length value.

delay move – When the control executes a tool length offset, the position of the cutting tool must change in the machine coordinate system and must reflect no change in tool position in the work coordinate system. This requires tool motion to take place. When this parameter is set for “delay move,” the tool motion does not take place until an axis motion takes place using the axis that the tool length offset applies to. This includes the block that activates, deactivates, or changes the tool length value.

Refer to Figure 21.1 for a definition of the difference between shift and move.

Range

Selection	Results
(a)	immed shift, immed move
(b)	immed shift, delay move
(c)	delay shift, delay move

Notes

This is a global parameter; the value set here applies to all axes. This parameter does not affect the operation of the cutter compensation or the TTRC features in any way.

Important: It is incompatible to select a geometry offset mode of “immed shift, immed move” when the wear offset mode is configured as “immed shift, delay move” or a geometry offset mode of “immed shift, delay move” and a wear offset mode of “immed, shift immed move”. An error message (INCOMPATIBLE TOOL ACTIVATION MODES) is displayed and the control is held in E-Stop when the wear and geometry activation mode is incompatible.

This parameter is not available for grinder applications.

Tool Offset Cancel

Function

Use this parameter to determine what tool offsets are canceled when a control reset is performed. These offsets can still be activated, canceled, or replaced by the programmer at any time. You can configure your system to perform a control reset whenever an E-Stop reset is performed using the parameter Control Reset on E-Stop Reset (page 33-3).

Axis	Parameter Number
	Single Process
All	[204]

Do Not Cancel - When "do not cancel" is selected for this parameter, the control does not cancel the offset values when a control reset is executed.

Cancel Wear Only - When "cancel wear only" is selected for this parameter, the control cancels only the wear offset when a control reset is executed. Any geometry offset that is active is not canceled.

Cancel Geom & Wear - When "cancel geom & wear" is selected for this parameter, the control cancels both geometry and wear tool offset data when a control reset is executed.

Cancel Geometry Offset - when you select "cancel geometry offset" for this parameter, the control cancels both length offset and radius/orientation offset when a control reset is executed.

Use the AMP parameter Cancel Tool Offsets on M02/M30 (page 33-9) to determine if tool offsets are canceled at the end of a part program.

Range

Application type:	Selection	Results
Mill/Lathe	(a)	Do Not Cancel
	(b)	Cancel Wear Only
	(c)	Cancel Geom & Wear

Notes

This is a global parameter; the value set here applies to all axes.

G37 Tool Gauging Cycle Parameters

These subsections describe the parameters that are available for execution of the G37 tool gauging features. The feedrate for execution of the G37 tool gauging features are discussed in chapter 10.

The G37s tool gauging feature modifies the values in the tool offset tables. There are five G37s used to activate four gauging operations. They are:

G37, G37.1	Skip 1
G37.2	Skip 2
G37.3	Skip 3
G37.4	Skip 4

Note that the G37 and G37.1 codes are functionally the same. Refer to your mill or lathe programming and operation manuals for more information.

The parameters in this section for G37 cycles are:

- Position Tolerance for Skip 1 – 4
- Add to Tool Offset for Skip 1 – 4
- Target Offset for Skip 01 –04

Position Tolerance for Skip 1 – 4

Function

Use these parameters to determine the position tolerance for the four different skip cycles called by the G37 codes. This position tolerance is used to set a range ± from the nominal end-point that is programmed in the G37 block. The skip signal must be received by Logic when the tool is within this position tolerance range for the control to modify the offset tables.

If the skip signal is received before the tool enters the position tolerance range, the remaining motion of the G37 block is aborted, and the control continues on to the next block. If the skip signal is not received by the time the tool exits the position tolerance range, the control continues on to the next block. In either case, no modification of the tool offset table is performed, and an error is generated though program execution is not interrupted.

Parameter	Parameter Number
	Single Process
Position tolerance for Skip 1	[206]
Position tolerance for Skip 2	[207]
Position tolerance for Skip 3	[208]
Position tolerance for Skip 4	[209]

Range

0.00000 to 100.00000 mm

or

0.00000 to 3.93701 in.

Notes

These are global parameters; the value set here applies to all axes.

Add to Tool Offset for Skip 1 - 4

Function

Typically, when one of the G37 cycles is executed, the control moves toward a programmed position. A signal is sent to Logic from a device when contact is made between the tool and the device. The control then subtracts the actual position of the tool when the signal is sent to Logic from the programmed end-point of the move.

Use these parameters to determine whether the value generated as described above is added to or replaces the current value in the table for tool length (or radius value for mill versions when some axis other than the tool length axis is used).

Parameter	Parameter Number
	Single Process
Add to Tool Offset for Skip 1	[210]
Add to Tool Offset for Skip 2	[211]
Add to Tool Offset for Skip 3	[212]
Add to Tool Offset for Skip 4	[213]

True - When this parameter is true, the value generated by the G37 cycle is added to the offset data selected in the parameter **Target Offset for Skip**.

False – When this parameter is false, the value generated by the G37 cycle replaces the offset data in the table selected in the parameter **Target Offset for Skip**.

Range

Selection	Results
(a)	True
(b)	False

Notes

These are global parameters; the value set here applies to all axes.

Target Offset for Skip 01 - 04

Function

Use this parameter to determine what offset table is modified by the G37 skip cycles. A choice of the wear table or the geometry table is provided.

For lathe controls, the value always modifies the tool length data for the axis programmed in the G37 block.

For mill controls, the value modifies the tool length data if the axis programmed in the G37 block is the tool length axis. If some axis other than the tool length axis is programmed in the G37 block, the tool radius value is modified in the table.

Selecting this parameter as "WEAR_OFFSET" causes the wear offset table to be altered after execution of a G37 block. Selecting this parameter as "GEOM_OFFSET" causes the geometry offset table to be altered after execution of a G37 block.

Parameter	Parameter Number
	Single Process
Target Offset for Skip 01	[214]
Target Offset for Skip 02	[215]
Target Offset for Skip 03	[216]
Target Offset for Skip 04	[217]

Range

Selection	Results
(a)	Wear Offset
(b)	Geom Offset

Notes

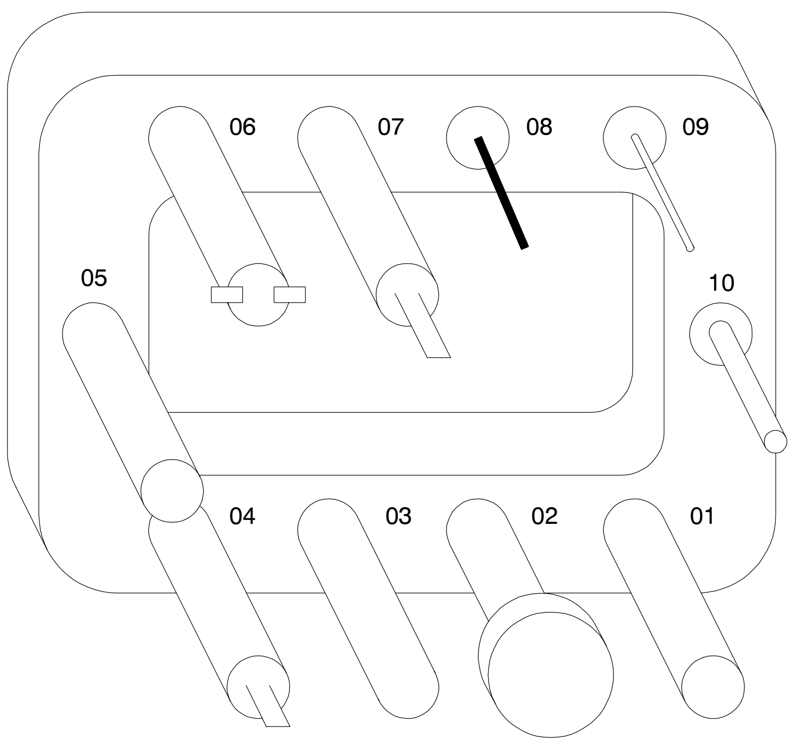
These are global parameters; the value set here applies to all axes.

Tool Magazine/Turret Parameters

This section describes the parameters that are used to configure a tool magazine or turret or other tool changing device. When a T-word is programmed that requires a tool change, these parameters are used to set the number of different tool positions and the direction of rotation of the tool turret and magazine. Figure 21.2 shows a typical mill tool magazine.

Important: These parameters are used only in conjunction with the random tool software option.

Figure 21.2
Typical Mill Tool Magazine



Use these parameters to configure the tool magazine/turret:

Number of Tool Pockets

Function

Use this parameter to determine the maximum number of tools that can be selected. These tool are arranged in a magazine, turret, or some other tool changing device. Enter the number of tool pockets (sometimes called tool pots) that a specific system contains. For example, in the typical mill tool magazine shown in Figure 21.2, this parameter would be set to a value of 10, the number of tool pockets.

Axis	Parameter Number
	Single Process
All	[128]

Range

1 to 200 different tool pockets

Notes

This is a global parameter; the value set here applies to all axes.

Tool Table Motion

Function

Use this parameter to determine the direction in which the tool magazine, turret, or other tool changing device can be rotated. This parameter allows for these direction options:

Plus only – If this parameter is set as "Plus only," it indicates that the tool changer may be rotated only in the plus direction when indexing a tool.

Minus only – If this parameter is set as "Minus only," it indicates that the tool changer may be rotated only in the minus direction when indexing a tool.

Bidirectional – If this parameter is set as bidirectional, it indicates that the tool changer can be rotated in the plus and minus directions when indexing a tool. If this is the case, the control selects the direction that brings the programmed tool into position the fastest.

Axis	Parameter Number
	Single Process
All	[129]

Range

Selection	Results
(a)	Plus only
(b)	Minus only
(c)	Bi-directional

Notes

This is a global parameter; the value set here applies to all axes.

Tool Life Monitor Parameters

Use this section to set the parameters for the tool life monitor feature. Typically, tools are classified into various groups, with the tool life for each group specified. This feature accumulates the use of each tool in the groups and compares this to the expected tool life. When the tool life is exceeded for a tool, a different tool in the group is then selected.

Use these parameters to configure the tool life feature:

- Tool Number/Group Boundary
- T-word Programming Method

Tool Number/Group Boundary

Function

Use this parameter to determine what T-words are used as ordinary tool words or used as tool life groups. Any T-word that is programmed equal to or larger than this parameter is considered to be a tool life group number. The group number is the value of the T-word minus the value set for “Tool Number/Group Boundary.” For example if this parameter is set at 20 programming T21 would call a tool from tool group 1.

Any T-word that is programmed smaller than this parameter is considered to be an ordinary tool word, and no tool life management is performed. For example, if this parameter is set at 20, programming T19 would simply call tool number 19.

Axis	Parameter Number
	Single Process
All	[131]

Range

0 to 9799

Notes

This is a global parameter; the value set here applies to all axes.

T-word Programming Method

Function

This parameter determines how a T-word may be programmed in conjunction with an M06 code. There are 4 methods available:

Method	Description
Return tool in M06	With this method, the T-word to be activated is programmed in a block that does not contain an M06. The T-word is stored until some later block that contains an M06. When the M06 is executed, the currently active tool is replaced with the last tool number programmed with a T-word. With this method, the tool number (or group number) that is being replaced as the active tool must be programmed in the block that contains the M06 command. If the M06 block does not contain the previously active tool number/group number, or if the wrong number is programmed with the M06, the control will generate an error.
Next tool in T-word	This method is identical to the "Return tool in M06" method with the exception that the block containing an M06 cannot contain a T-word. The new tool/group number is programmed in some block before the M06 block. It is not necessary to program the previously active tool number/group number in the M06 block.
M06 Required	This method defines that a tool is only activated in an M06 block. A T-Word that is programmed by itself becomes the next tool activated at an M06 block. Programming an M06 by itself activates the next tool. If a T-word is programmed in an M06 block, that T-word is used as the active tool and any other unactivated T-word is discarded.
Activate Tool in T-word	For this method, no M06 needs to be programmed to change tools. When the T-word is executed, a tool change occurs. It is not necessary to program the previously active tool number/group number in the M06 block.

Axis	Parameter Number
	Single Process
All	[132]

Range

Selection	Results
(a)	Return Tool in M06
(b)	Next Tool in T-word
(c)	M06 required
(d)	Activate Tool in T-word

Notes

This is a global parameter; the value set here applies to all axes. This is a mill only parameter.

Tool Offset Range Verification Parameters

These subsections describe the parameters used for tool offset range verification. These parameters check:

- the maximum values entering the tool offset tables
- the maximum change that can occur in either table

Maximum Wear Offset Change

Function

Use this parameter to specify the amount that a Wear Offset may change between successive entries.

If the operator exceeds the amount set by the system installer in AMP, the change is not allowed. The control generates the error message "OFFSET EXCEEDS MAX CHANGE."

Important: The control does not perform the verification if the value, old or new, is zero, nor does it check G10 data-setting codes.

Refer to your programming and operation manual for more information.

Axis	Parameter Number	
	Mill	Lathe
All	[244] (single process) [20244] (process 1) [21244] (process 2)	—
(1)		[1410]
(2)		[2410]
(3)		[3410]
(4)		[4410]
(5)		[5410]
(6)		[6410]
(7)		[7410]
(8)		[8410]

Range

0.000000 to 390.000000 in./9906.0000 mm

Notes

For *mill* applications, this parameter is global; the value set here applies to all axes.

For *lathe* applications, set this parameter independently for each axis.

Maximum Geometry Offset Change

Function

Use this parameter to specify the amount that a Geometry Offset may change between successive entries.

If the operator exceeds the amount set by the system installer in AMP, the change is not allowed. The control generates the error message "OFFSET EXCEEDS MAX CHANGE."

Important: The control does not perform the verification if the value, old or new, is zero, nor does it check G10 data-setting codes.

Refer to your programming and operation manual for more information.

Axis	Parameter Number	
	Mill	Lathe
All	[245] (single process) [20245] (process 1) [21245] (process 2	—
(1)		[1411]
(2)		[2411]
(3)		[3411]
(4)		[4411]
(5)		[5411]
(6)		[6411]
(7)		[7411]
(8)		[8411]

Range

0.000000 to 390.000000 in./9906.0000 mm

0.000000 to 10000.000000 degrees

Notes

For *mill* applications, this parameter is global; the value set here applies to all axes.

For *lathe* applications, set this parameter independently for each axis.

Maximum +/- Wear Offset

Function

Use this parameter to specify the maximum value that a Wear Offset may contain. This value represents the absolute maximum value per table for all tool offsets in that table.

If the operator enters:	then:
a positive number greater than the maximum value	the control generates the error message:
a negative number less than the negative of the maximum value	"OFFSET EXCEEDS MAX VALUE"

Refer to your programming and operation manual for more information.

Axis	Parameter Number	
	Mill	Lathe
All	[246](single process) [20246] (process 1) [21246] (process 2	—
(1)		[1412]
(2)		[2412]
(3)		[3412]
(4)		[4412]
(5)		[5412]
(6)		[6412]
(7)		[7412]
(8)		[8412]

Range

0.000000 to 390.000000 in./9906.0000 mm

Notes

For *mill* applications, this parameter is global; the value set here applies to all axes.

For *lathe* applications, set this parameter independently for each axis.

Maximum +/- Geometry Offset

Use this parameter to specify the maximum value that a Geometry Offset may contain. This value represents the absolute maximum value per table for all tool offsets in that table.

If the operator enters:	then:
a positive number greater than the maximum value	the control generates the error message: "OFFSET EXCEEDS MAX VALUE"
a negative number less than the negative of the maximum value	

Refer to your programming and operation manual for more information.

Axis	Parameter Number	
	Mill	Lathe
All	247 (single process) [20247] (process 1) [21247] (process 2	—
(1)		[1413]
(2)		[2413]
(3)		[3413]
(4)		[4413]
(5)		[5413]
(6)		[6413]
(7)		[7413]
(8)		[8413]

Range

0.000000 to 390.000000 in./9906.0000 mm

0.000000 to 10000.000000 degrees

Notes

For *mill* applications, this parameter is global; the value set here applies to all axes.

For *lathe* applications, set this parameter independently for each axis.

Maximum +/- Wear Radius

Function

Use this parameter to specify the maximum value that a Wear Radius Offset may contain. This value represents the absolute maximum value per table for all tool offsets in that table.

If the operator enters:	then:
a positive number greater than the maximum value	the control generates the error message:
a negative number less than the negative of the maximum value	“OFFSET EXCEEDS MAX VALUE”

Refer to your programming and operation manual for more information.

Axis	Parameter Number
	Single Process
All	[248]

Range

0.000000 to 390.000000 in./9906.0000 mm

Notes

For *mill* and *lathe* applications, this parameter is global; the value set here applies to all axes.

Maximum +/- Geometry Radius

Function

Use this parameter to specify the maximum value that a Geometry Radius Offset may contain. This value represents the absolute maximum value per table for all tool offsets in that table.

If the operator enters:	then:
a positive number greater than the maximum value	the control generates the error message: “OFFSET EXCEEDS MAX VALUE”
a negative number less than the negative of the maximum value	

Refer to your programming and operation manual for more information.

Axis	Parameter Number
	Single Process
All	[249]

Range

0.000000 to 390.000000 in./9906.0000 mm

Notes

For *mill* and *lathe* applications, this parameter is global; the value set here applies to all axes.

Maximum Radius Offset Change

Function

Use this parameter to specify the amount that a Radius Offset may change between successive entries.

If the operator exceeds the amount set by the system installer in AMP, the change is not allowed. The control generates the error message “OFFSET EXCEEDS MAX CHANGE.”

Important: The control does not perform the verification if the value, old or new, is zero, nor does it check G10 data-setting codes.

Refer to your programming and operation manual for more information.

Axis	Parameter Number
All	[245]

Range

0.000000 to 390.000000 in./9906.0000 mm

Maximum +/- Radius Offset

Function

Use this parameter to specify the maximum value that a Radius Offset may contain. This value represents the absolute maximum value per table for all tool offsets in that table.

If the operator enters:	then:
a positive number greater than the maximum value	the control generates the error message: “OFFSET EXCEEDS MAX VALUE”
a negative number less than the negative of the maximum value	

Refer to your programming and operation manual for more information.

Axis	Parameter Number
All	[244]

Range

0.000000 to 390.000000 in./9906.0000 mm

END OF CHAPTER

Cutter Comp/Tool Tip Radius

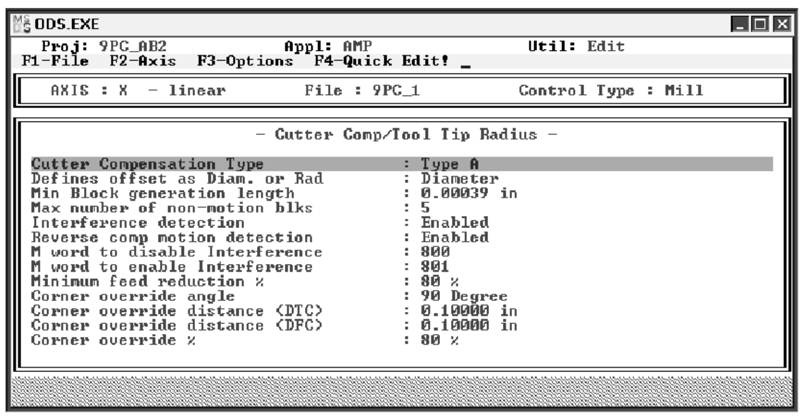
Chapter Overview

This chapter describes the Cutter Compensation and Tool Tip Radius Compensation (TTRC) parameters. This lets the part programmer program the path the tool follows in terms of the tool's center or gauge point without regard to the diameter or tool-tip radius.

The Cutter Compensation and TTRC parameters are broken into two sections:

Parameter:	Page:
Compensation Basic Setup	22-1
Compensation Error Detection	22-10

When you select "Cutter Comp/Tool Tip Radius" from the main menu screen, this screen becomes available:



If the application type is:	and the control type is:	these parameters do not appear:
Mill/Lathe	Lathe	Defines offset as Diam. or Rad

Compensation Basic Setup

This set of parameters determines how the basic cutter compensation and TTRC features operate.

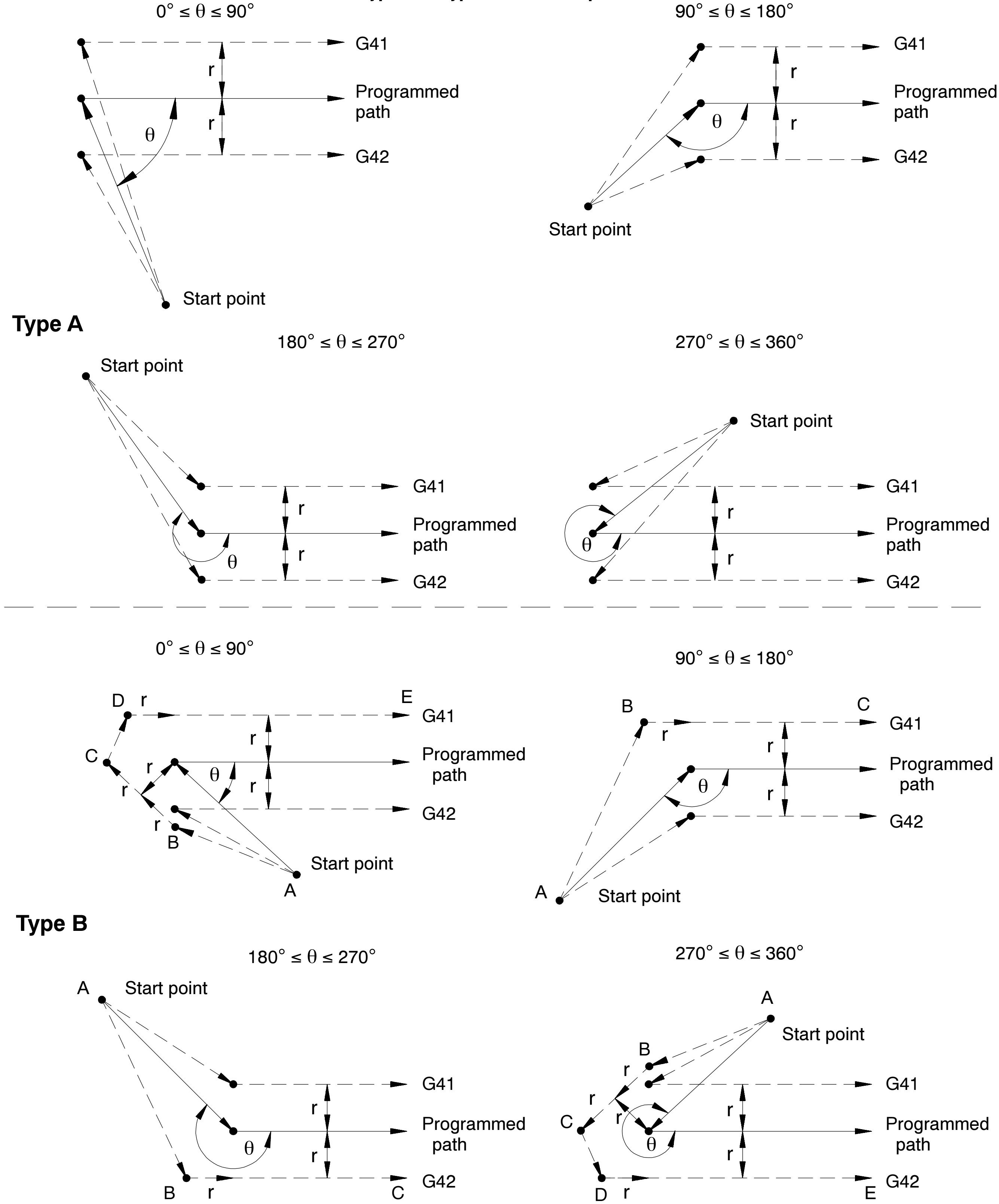
Cutter Compensation Type

Function

Use this parameter to determine the cutter compensation type. There are two types of cutter compensation: type A and type B. The main difference between these two types is the entry move into compensation.

Figure 22.1 compares typical "straight line to straight line" entry moves for types A and B compensation.

Figure 22.1
Type A vs Type B Cutter Compensation



Axis	Parameter Number
All	[24]

Range

Selection	Result
(a)	Type A
(b)	Type B

Notes

This is a global parameter. The value set here applies to all axes.

Define Offset as Diam. or Rad (Mill Only)

Function

This parameter allows for flexibility when entering the tool offset data into the offset tables. This parameter determines if the values in the offset table for the tool cutter compensation data are radius values or diameter values. If they are entered as diameter values, set this parameter at "diameter"; if they are radius values, set this parameter at "radius."

Axis	Parameter Number
All	[25]

Range

Selection	Result
(a)	Diameter
(b)	Radius

Notes

If this parameter is set at "diameter," the control uses half of the value entered in the table as the compensation data for wear and geometry.

This is a global parameter. The value set here applies to all axes.

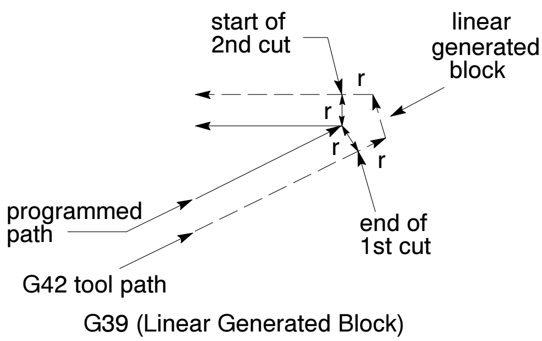
Minimum Block Generation Length

Function

During cutter compensation, use this parameter to determine the minimum length that a linear generated block may be. Note that these generated blocks, as shown in Figure 22.2, are not created by the programmer; they are blocks that the control generates to correctly compensate a programmed path without over-cutting.

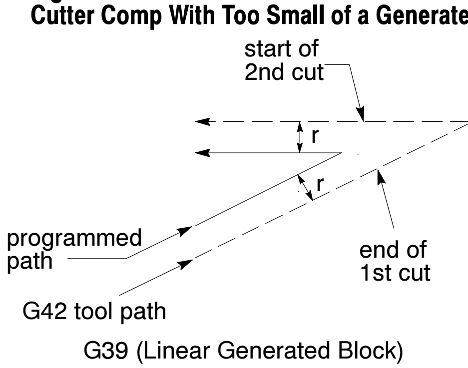
Use this parameter only in G39 Linear Generated Block cases; it is not used for G39.1 Circular Generated Block cases.

Figure 22.2
Cutter Comp Generated Blocks



If a generated block length is smaller than the value set for this parameter, the control ignores the generated block. It is not executed and the control continues to the intersection of the compensated paths. For example, if a **Minimum Block Length** of .75 inches is set in AMP, and if the generated block length is 0.5, the generated tool path is ignored as shown in Figure 22.3.

Figure 22.3
Cutter Comp With Too Small of a Generated Block



Axis	Parameter Number
All	[27]

Range

0.00010 to 9.99900 mm

or

0.00000 to .39366 inch

Notes

This is a global parameter. The value set here applies to all axes and processes.

Minimum Feed Reduction %

Function

This parameter limits the amount of automatic feedrate reduction that occurs under certain compensation conditions.

When cutting along the inside of a circular path with cutter compensation active, the actual compensated path covers less distance than the programmed path. As a result, the control automatically reduces the feedrate as much as necessary to maintain a constant cutting feedrate (tool tip relative to point of contact on the workpiece).

This parameter sets a minimum to that feedrate reduction. The value entered here is the minimum percentage of the programmed feedrate that is allowed.

If 100% is entered here, it effectively disables automatic feedrate reduction during cutter compensation.

This is not a per-axis parameter. The value selected here is applied to all axes .

Axis	Parameter Number
All	[405]

Range

0 to 100%

Notes

Consider the following example:

As the tool approaches an inside arc with compensation active at a feedrate of 200 mmpm, the control determines that the feedrate must be reduced to 120 mmpm to maintain a constant cutting feedrate. If this parameter is set to 50%, the 120 mmpm would be allowed.

If this parameter is set to 80%, the 200 mmpm feedrate of this example could not be reduced below 160 mmpm. The 120 mmpm feedrate would not be allowed, and the feedrate would instead be reduced to 160 mmpm until the arc cut was completed.

The minimum feedrate can be affected by the feedrate override switch setting. If the feedrate override is set to 80%, the feedrate of the above example would be 180 mmpm. If the **Minimum Feed Reduction** parameter is set to 80%, the control could reduce the feedrate to 80% of 180 mmpm or 144 mmpm.

Maximum Number of Nonmotion Blks

Function

The control is always looking ahead to the next motion block to determine the actual tool path taken for a motion block in compensation. If the next program block is not a motion block, the control continues to scan ahead for a motion block until it either detects one or until the allowable number of non-motion blocks as set with this parameter is reached.

Axis	Parameter Number
All	[29]

Range

0 to 25 blocks

Notes

This is a global parameter. The value set here applies to all axes and processes.

Corner Override Parameters

The Automatic Corner Override feature compensates for increased spindle loading when cutting inside corners with cutter compensation active.

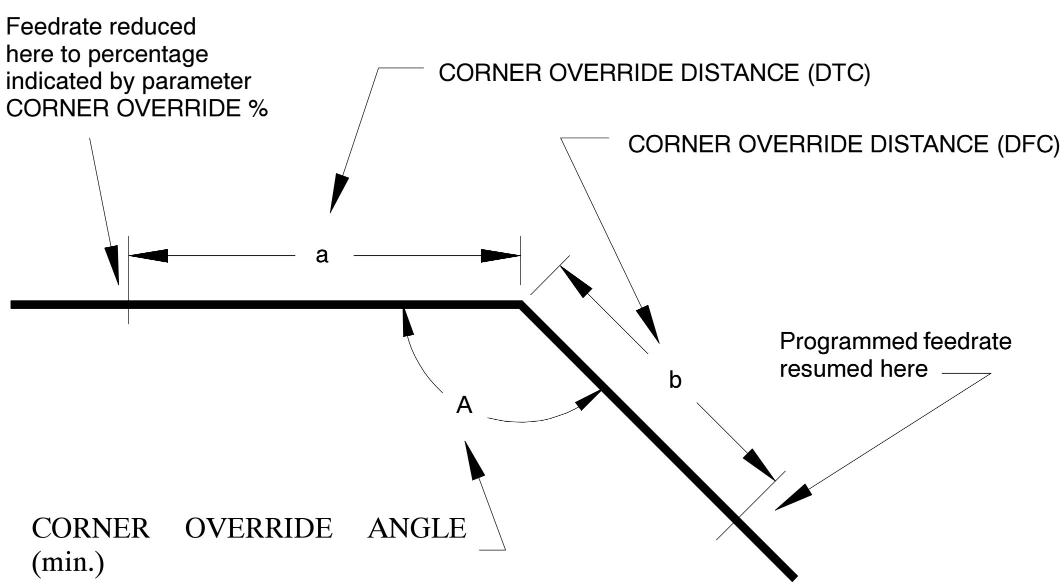
Since the relative speed between the part and the cutting tool edge increases when cutting inside corners, a more uniform finish and reduced tool loading results if the feedrate is reduced.

The feedrate is reduced to the percentage entered for the AMP parameter **Corner Override %**, when these conditions are met:

- Cutter Compensation (G41/G42) is active
- Automatic Corner Override (G62) is active
- the corner angle (or angle between arc tangent lines) is equal to or greater than the value entered for the AMP parameter **Corner Override Angle**
- the tool has reached the point defined by the AMP parameter **Corner Override Distance (DTC)**
- the tool has not reached the point defined by the AMP parameter **Corner Override Distance (DFC)**

This set of parameters determines how the corner override feature operates.

Figure 22.4
Automatic Corner Override



Corner Override Angle

Function

Enter the minimum angle between two consecutive programmed moves (or angle between tangent lines if moves are arcs) required before Automatic Corner Override can be activated.

Axis	Parameter Number
All	[28]

Range

0 to 180 degrees

Notes

The parameters for the override percent and the distance from the corner that the override is effective from are set under the "Acc/Dec Parameters" screen.

This is a global parameter. The value set here applies to all axes and processes.

Corner Override Distance (DTC)

Function

Enter the vector distance from the end-point of a move that creates a corner, back to the point on that move at which Automatic Corner Override is to be activated.

(DTC stands for Distance To Corner)

Refer to the discussion of the AMP parameter **Min Angle for Corner Override** for details.

This is not a per-axis parameter. The distance selected here is applied to all axes.

Axis	Parameter Number
All	[409]

Range

0.00000 to 9999.99999 mm

or

0.00000 to 393.70079 in

Notes

This parameter is a global parameter. The value set here applies to all axes.

Corner Override Distance (DFC)

Function

Enter the vector distance from the end-point of a move that creates a corner, to the point on the next move at which Automatic Corner Override is to be deactivated.

(DFC stands for Distance From Corner)

Refer to the discussion of the AMP parameter **Min Angle for Corner Override** for details.

This is not a per-axis parameter. The distance selected here is applied to all axes.

Axis	Parameter Number
All	[410]

Range

0.00000 to 9999.99999 mm

or

0.00000 to 393.70079 in

Notes

This is a global parameter. The value set here applies to all axes.

Corner Override %

Function

Enter the percentage to which the current feedrate is to be reduced when Automatic Corner Override has been activated.

Refer to the discussion of the AMP parameter **Min Angle for Corner Override** for a discussion of the overall operation of Corner Override.

This is not a per-axis parameter. The percentage selected here is applied to all axes .

Axis	Parameter Number
All	[412]

Range

0 to 100 %

An Example of Corner Override

If 60% is entered here and the currently active feedrate is 100 mmpm, it is reduced to 60 mmpm when Automatic Corner Override is activated.

The feedrate can be affected by the feedrate override switch setting. If the feedrate override was set to 80%, the active feedrate of the above example would be 80 mmpm. If the **Corner Override %** parameter was set to 60%, the control would reduce the feedrate to 60% of 80 mmpm (48 mmpm) during automatic corner override.

Notes

This is a global parameter. The value set here applies to all axes.

Compensation Error Detection

This section describes the parameters that enable the control's error detection for compensation. Error detection can be disabled or enabled for all blocks, or temporarily enabled or disabled for a specific program block, by using the parameters listed in the following subsections.

Compensation errors are described in detail in Chapter 15 of your mill or lathe programming and operation manual.

Interference Detection

Function

Use this parameter to activate or deactivate the Interference Detection feature for cutter compensation. An interference error is generated when an intersection occurs among any three consecutive compensated tool paths. This includes generated motion blocks.

Enabled – When this parameter is set as enabled, the control generates an error and halts program execution whenever interference occurs.

Disabled – When this parameter is set as disabled, the control ignores any interference errors and allows the program to execute.

Axis	Parameter Number
All	[93]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This is a global parameter. The value set here applies to all axes.

Reverse Compensated Motion Detection

Function

Use this parameter to activate or deactivate the Reverse Compensated Motion Detection feature for cutter compensation. A reverse motion error is generated when any two consecutive compensated tool paths reverse direction 180 degrees. This includes generated motion blocks.

Enabled – When this parameter is set as enabled, the control generates an error and halts program execution whenever a reverse motion error occurs.

Disabled – When this parameter is set as disabled, the control ignores any reverse motion errors and allows the program to execute.

Axis	Parameter Number
All	[94]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This is a global parameter. The value set here applies to all axes.

M-word to Disable Interference Detection

Function

Use this parameter when the parameter for Interference Detection is set as enabled. If Interference Detection is enabled, it is possible to temporarily disable interference detection by programming the M-word set with this parameter in a block. Interference detection is disabled only for the block that contains this M-word; any following blocks have interference detection enabled unless they also contain this M-word.

Axis	Parameter Number
All	[95]

Range

-1 to 999

Notes

This is a nonmodal M-word.

This is a global parameter. The value set here applies to all axes.

M-word to Enable Interference Detection

Function

Use this parameter when the parameter for Interference Detection is set as disabled. If Interference Detection is disabled, it is possible to temporarily enable interference detection by programming the M-word set with this parameter in a block. Interference detection is enabled only for the block that contains this M-word; any following blocks have interference detection disabled unless they also contain this M-word.

Axis	Parameter Number
All	[96]

Range

-1 to 999

Notes

This is a nonmodal M-word.

This is a global parameter. The value set here applies to all axes.

Refer to your programming and operation manual for more information.

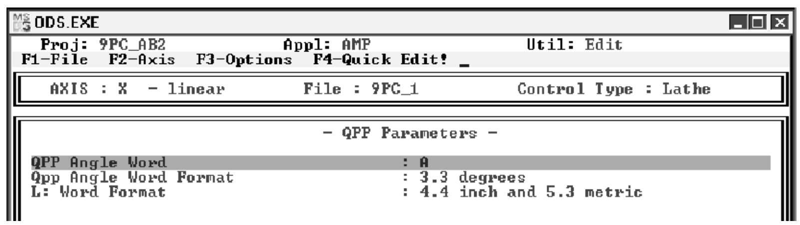
END OF CHAPTER

QuickPath Plus Parameters

Chapter Overview

The QuickPath Plus feature provides a convenient programming method that simplifies programming of the control. Use this feature to create a part program from a part drawing through the use of common drawing dimensions such as angles, chamfer sizes, fillet radius, or line segment length. Refer to your mill or lathe programming and operation manuals for more information.

The workstation displays this screen after you select the "QPP Parameters" group:



QuickPath Plus Parameters

These pages cover the parameters that are available for the QuickPath Plus feature:

Parameter:	Page:
QPP Angle Word	23-1
QPP Angle Word Format	23-2
L: Word Format	23-2

QPP Angle Word

Function

This parameter lets you select the letter that programs an angle for QuickPath Plus programming.

Axis	Parameter Number
All	[348]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	,A

Notes

This is a global parameter; the value set here applies to all axes.

The selection ",A" can be used if all other selections are being used as axis words.

QPP Angle Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point for an angle word used in QuickPath Plus programming.

Parameter	Parameter Number
QPP Angle Word Format	[512]

Range

Selection	Result
(a)	3.1 degrees
(b)	3.2 degrees
(c)	3.3 degrees
(d)	3.4 degrees

Notes

This is a global parameter; the value set here applies to all axes.

L: Word Format

Function

This parameter specifies the maximum number of digits that are permitted to the left and to the right of the decimal point for an L-word used in QuickPath Plus programming in inch or metric modes. Use an L-word in QuickPath Plus programming to program the length of a line.

Parameter	Parameter Number
L: Word Format	[500]

Range

Selection	Result	Selection	Result
(a)	3.4 inch and 4.3 metric	(g)	2.4 inch and 3.3 metric
(b)	3.5 inch and 4.4 metric	(h)	2.5 inch and 3.4 metric
(c)	4.3 inch and 5.2 metric	(i)	2.6 inch and 3.5 metric
(d)	4.4 inch and 5.3 metric	(j)	3.3 inch and 4.2 metric
(e)	5.3 inch and 6.2 metric	(k)	4.2 inch and 5.1 metric
(f)	2.3 inch and 3.2 metric	(l)	5.2 inch and 6.1 metric

Notes

This is a global parameter; the value set here applies to all axes .

END OF CHAPTER

Fixed Cycles

Chapter Overview

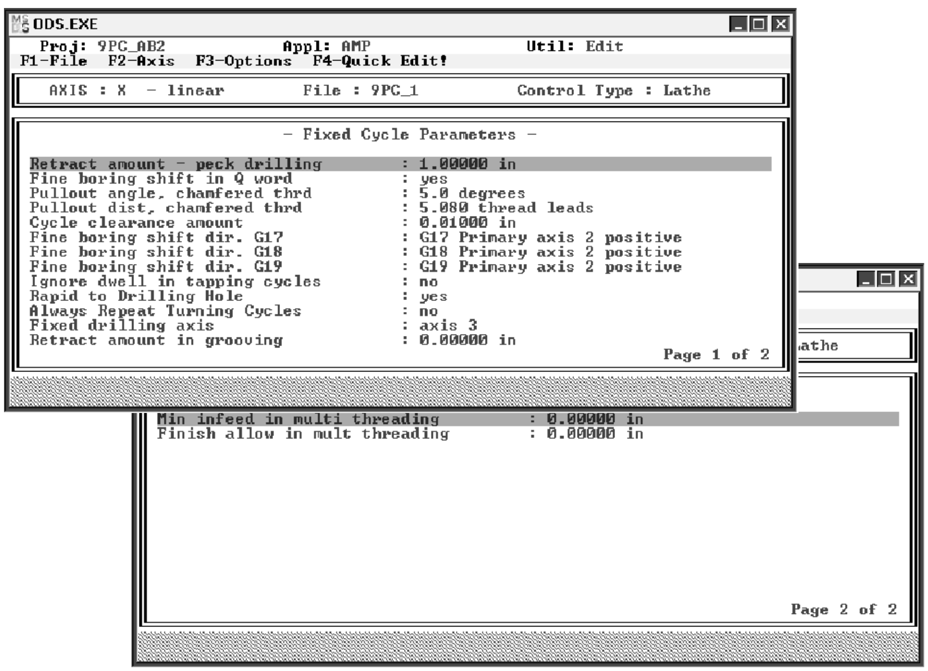
This chapter describes the available milling and turning fixed-cycle AMP parameters.

The fixed-cycle parameters are separated into these four sections:

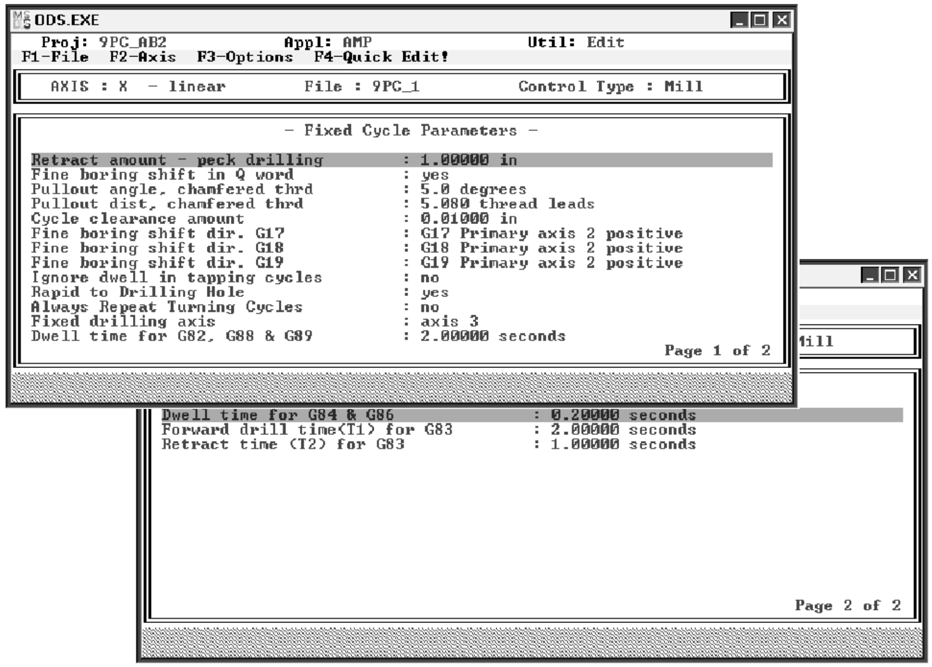
Parameter:	Page:
Milling fixed-cycle parameters	24-2
Threading cycle parameters	24-9
Turning cycle parameters	24-14

All of these parameters may also be changed on the control with the fixed cycle parameters table.

When you select the “Fixed Cycle Parameters” group and the control type “Lathe,” the workstation displays these screens:



When you select control type “Mill,” the workstation displays these screens:



Milling Fixed Cycle Parameters

The following sections describe the parameters that are used for the milling fixed cycles. Milling fixed cycles typically include the control's drilling, tapping, and boring canned cycles.

Retract Amount for Peck Drilling

Function

G73 mill control types.

G83.1 lathe control types.

Use this parameter to set the retraction amount for the intermittent high speed drilling cycle. This parameter assigns a value "d" representing the distance that the cutting tool retracts after each infeed amount in the drilling operation. For this cycle, the tool cuts to the programmed depth in steps. The tool feeds into the part a distance programmed with the Q-word and then retracts out of the part the distance assigned with this parameter "d."

Axis	Parameter Number
	Single Process
All	[71]

Range

0.00000 to 99999.00000 mm

or

0.00000 to 3936.96850 inch

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control with the fixed-cycle parameter table.

Fine Boring Shift in Q-word

Function

G76 mill control types.

G86.1 lathe control types.

This parameter specifies whether the distance of a cutting tool shift during a fine boring cycle is determined by a programmed Q-word on an AMP assigned axis or if the distance is set with I, J, K, words in the currently active plane.

YES – Setting a value of "YES" for this parameter allows the programming of a Q-word to specify the distance of the shift in the block that calls the fine boring cycle. The axis and direction that the shift uses is determined by the AMP parameters discussed in section titled "Fine Boring Shift Direction."

NO – Setting a value of "NO" for this parameter causes the value set for the axis and direction of shift using the parameters discussed in the Fine Boring Shift Direction section (page 24-4) to be ignored. It is necessary to program the shift amount, direction, and axes for the shift, using I, J, and K words in the block that calls the fine boring cycle.

Axis	Parameter Number
All	[72]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control with the fixed-cycle parameter table.

Fine Boring Shift Direction G17, G18, and G19

Function

G76 mill control types.

G86.1 lathe control types.

These parameters determine the axis and direction that the cutting tool shifts when executing a fine boring cycle. For these parameters to be effective, it is necessary to program a shift amount using a Q-word (the parameter **Fine Boring Shift in Q-word** must be “YES”).

There are 3 parameters available to determine the shift direction and axis for the fine boring cycle. The parameter that is used to select the direction and axis used for the shift is determined by the active plane (G17, G18, or G19) when the fine boring cycle is programmed. For example, if the fine boring cycle is programmed in the G17 plane, the control uses the axis and direction for the shift that is selected with parameter number [77] (single process).

Select the primary axis number in that plane, and the direction to travel on that primary axis. This determines the axis and direction that a Q-word shifts the cutting tool.

Parameter	Parameter Number
	Single Process
Fine Boring shift direction G17	[77]
Fine Boring shift direction G18	[78]
Fine Boring shift direction G19	[79]

Range

Selection	Result
(a)	(G17, G18, G19) Primary axis 1 positive
(b)	(G17, G18, G19) Primary axis 1 negative
(c)	(G17, G18, G19) Primary axis 2 positive
(d)	(G17, G18, G19) Primary axis 2 negative

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameters table.

Cycle Clearance Amount

Function

This parameter sets the clearance amount for the G83 peck drilling cycle, G88 pocket/post/hemisphere cycles, and G89 irregular pocket cycles.

G83 Drilling cycle (mill and lathe control types)
Use this parameter to set the clearance amount “d” for this cycle. This clearance amount is a buffer that prevents the tool from hitting the bottom of the previous infeed when the tool plunges at rapid feedrate. The control feeds the drilling tool at the rapid feedrate to this clearance amount “d” that is above the depth of the previous infeed.

G88.x Pockets,Posts, and Hemispheres (mill control types)
This parameter sets the clearance amount for any cycle called with a G88.x. This value represents the amount the tool is raised form the pre-cycle position during the move of the tool to the center of the specified pocket/post. This value also represents the amount the tool is raised when returning to the center point between each level in the pocket/post cutting operation.

G89.x Irregular Pockets (mill control types)
Important: The Irregular Pocket Milling Cycles feature (G89.1 and G89.2) is only available prior to release 12.xx. Any attempt to program a G89.1 or G89.2 in release 12.xx or later will result in the error message, “Illegal G-code”.

This parameter sets the clearance amount “e” for irregular pocket cycles called with a G89.x. This value represents the amount the tool is raised when returning to the center point between each level of the pocket cutting operation.

Axis	Parameter Number
All	[76]

Range

0.00000 to 99999.00000 mm

or

0.00000 to 3936.96850 inch

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameters table.

Ignore Dwell in Tapping Cycles

Function

G84 and G74 for mill control types.

G84 and G84.1 for lathe control types.

This parameter determines whether the control allows a dwell at hole bottom during the execution of either the tapping cycle or left hand tapping cycle. Dwells are normally programmed in the cycle block with a P-word and may be in either seconds or spindle revolutions.

YES – Setting a value of "YES" for this parameter causes the control to ignore any P-word that is programmed for a dwell in a tapping or left hand tapping cycle block. No dwell is permitted during the actual tapping operation; however, dwells may still be programmed using a G04 code if desired.

NO – Setting a value of "NO" for this parameter allows the control to recognize any P-word that is programmed for a dwell in a tapping or left hand tapping cycle block.

Axis	Parameter Number
All	[80]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

This parameter does not affect normal dwells that are programmed with a G04. Only dwells that are programmed in a milling or drilling cycle are affected.

Rapid to Drilling Hole

Function

All milling cycles, lathe and mill controls.

This parameter determines the speed at which the cutting tool will be positioned above the point where the milling cycle is to be performed. The cutting tool is always positioned to this point on a linear path.

YES – Setting a value of "YES" for this parameter causes the control to use the rapid positioning feedrate (G00) to position the cutting tool above the point where the milling cycle is to be performed. This is independent of the currently active positioning mode (G00, G01, G02, G03, etc.).

NO – Setting a value of "NO" for this parameter allows the programmer to select the rate at which the cutting tool cutting tool is to be positioned above the point where the milling cycle is to be performed. If the control is in G00 mode when the cycle is programmed, the tool is positioned by using the rapid feedrate. If the control is in any mode other than G00 (for example G01, G02, G03), the cutting tool is positioned on a linear path (G01 mode) above the workpiece at the currently active programmed feedrate.

Axis	Parameter Number
All	[81]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes. This parameter also applies to all fixed milling cycles.

This parameter may also be changed on the control by using the fixed-cycle parameters table.

Fixed Drilling Axis

Function

All milling and drilling cycles, lathe and mill controls.

This parameter determines the axis on which drilling operations are performed. This is the axis that the tool is actually feeding into the part. Other axes are used to position the cutting tool above the location for the milling operation. If an axis is selected here for the fixed drilling axis (any value other than none), the control uses this axis as the hole-machining axis regardless of the currently active plane.

Selecting None for a Drilling Axis
Selecting none for this parameter allows the part programmer to select the drilling axis using his choice of plane. When this parameter is set to none, the drilling axis is always the axis configured as perpendicular to the active plane. An axis is determined to be perpendicular to a plane if it is in some other defined plane. For example assume the following plane definition:

G17 (XY), G18 (ZX), G19 (YZ).

With the above plane definition, when the part programmer activates the G17 plane, the Z axis is determined to be the drilling axis since it is configured as perpendicular to the G17 plane (i.e. it is not in the active plane and is in the two planes perpendicular to the active plane). None is usually selected for mill controls to allow the programmer to select the hole machining axis by using plane select. Lathe controls typically require a fixed drilling axis.

Axis	Parameter Number
	Single Process
All	[26]

Range

Selection	Result
(a)	none
(b)	Axis 1
(c)	Axis 2
(d)	Axis 3
(e)	Axis 4
(f)	Axis 5
(g)	Axis 6
(h)	Axis 7
(i)	Axis 8

Notes

This is a global parameter; the value set here applies to all axes.

Threading Cycle Parameters

The following sections describe the parameters that are used for the threading cycles. These threading-cycle parameters are applicable only on lathes. The threading cycle parameters in this section are used for the single pass threading cycle, and the multipass threading cycle. No AMP parameters are available here for the G33 and G34 single pass threading modes.

Pullout Distance, Chamfered Thread

Function

Use this parameter for lathe controls when cutting either the single pass or multiple pass threading cycles. The G-codes for these cycles are:

G Code System	Single Pass	Multiple Pass
A	G92	G76
B	G78	G76
C	G21	G78

This parameter determines the pullout distance "r" for the threading chamfer feature.

If a value has been set for this and the "**Pullout Angle, Chamfered Thrd**" parameters, and the Logic flag BW_FCPULL has been set true, then the control automatically cuts a chamfer at the end of each threading pass during the operation of the threading cycles. This also lets you abort a threading pass and cut a chamfer out of the thread when the <CYCLE STOP> button is pressed.

Refer to your programming and operation manual for details on threading chamfer.

Enter in this parameter the number of threads that are to be chamfered. This determines the length of the chamfer as a function of the thread lead.

The resolution of this parameter is in 1/10th of a thread increments. The angle for this chamfer is determined by the **Pullout Angle, Chamfered Thread** parameter. Refer to Figure 24.1 for an application example.

Axis	Parameter Number
All	[74]

Range

0.0 to 50.0 threads

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

To disable the threading chamfer features enter a value of zero for this parameter.

Pullout Angle,Chamfered Thrd

Function

Use this parameter for lathe controls when cutting either the single pass or multiple pass threading cycles. The G-codes for these cycles are:

G Code System	Single Pass	Multiple Pass
A	G92	G76
B	G78	G76
C	G21	G78

This parameter determines the pullout angle “a” for the threading chamfer feature.

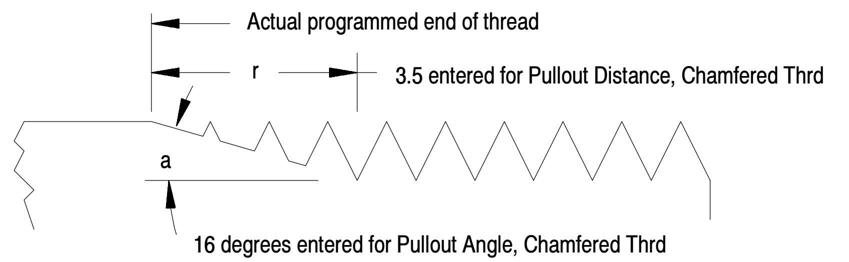
If a value has been set for this and the "**Pullout Distance, Chamfered Thrd**" parameters, and the Logic flag BW_FCPULL has been set true, then the control automatically cuts a chamfer at the end of each threading pass during the operation of the threading cycles. This also lets you abort a threading pass and cut a chamfer out of the thread when the <CYCLE STOP> button is pressed.

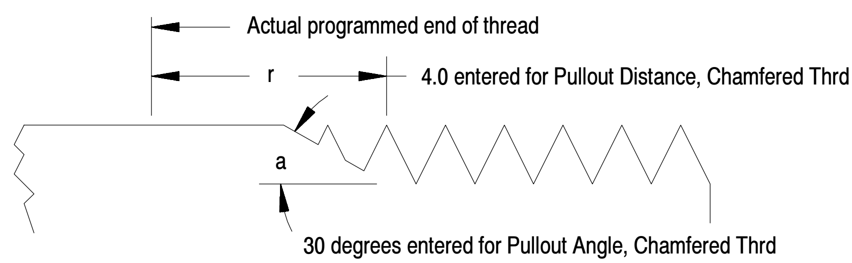
Refer to the lathe operation and programming manual for details on threading chamfer.

For this parameter, enter in degrees the angle of the chamfer. The length of the chamfer is determined by using the "**Pullout Distance, Chamfered Thread**" parameter. See Figure 24.1 for examples of Pullout Distance and Pullout Angle.

This angle is always measured from the axis parallel to the spindle center-line regardless of whether parallel or tapered threads are being cut. The angle direction, clockwise or counterclockwise, is always such that it will increase the root diameter of the thread.

Figure 24.1
Pullout Distance and Angle





Axis	Parameter Number
All	[73]

Range

0.0 to 89.0 degrees

Notes

This is a global parameter; the value set here applies to all axes and processes.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

To disable the threading retract and threading chamfer features, enter a value of zero for this parameter.

Min Infeed in Multi Threading

Function

This parameter is for lathe controls that use multiple pass threading cycle, G-codes:

System	G Code
A	G76
B	G76
C	G78

This parameter enables the setting of the smallest amount of material that may be cut during the multiple pass threading cycle. The control automatically generates the infeed for each cutting pass for this cycle.

If the generated depth of cut is smaller than the value entered for this parameter, the control will infeed an amount equal to the parameter value.

Axis	Parameter Number
All	[111]

Range

0.00000 to 9999.90000 mm

or

0.00000 to 393.69685 inch

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

The control will not infeed the cutting tool less than the value set for this parameter when executing a multi-pass threading cycle. The control looks ahead as many blocks as is necessary to prevent its having to execute a block that is less than this value. The depth of other cuts can be made smaller to compensate for later cuts.

Finish Allow in Multi Threading

Function

This parameter is for lathe controls that use the multiple pass threading cycle G-codes:

System	G Code
A	G76
B	G76
C	G78

This parameter determines whether the multiple-pass threading cycle is going to make a finishing pass and, if so, determines the depth of cut for this finishing pass.

If a value other than zero is entered for this parameter, the control will make a finishing pass when executing a multiple-pass threading cycle. The amount of material removed by this finishing pass is equal to the value set with this parameter. This is done by forcing the next-to-the-last pass in the cycle to leave an amount of material, equal to this parameter, above the actual programmed thread depth. The finish pass is then made cutting at the actual programmed thread depth. This parameter is always an unsigned radius value.

Axis	Parameter Number
All	[112]

Range

0.00000 to 9999.90000 mm

or

0.00000 to 393.69685 inch

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameters table.

Turning Cycle Parameters

The following sections describe the parameters that are used for turning fixed cycles. Turning fixed cycles typically include the control rough-turning, face-turning, and grooving cycles.

Always Repeat Turning Cycles

Function

All single-pass turning cycles (excluding threading and grooving) lathe control types.

Normally, turning cycles only repeat after execution of a block that contains axis motion that changes the depth of cut. This parameter is used to cause the turning cycle to be repeated after any block is executed when the cycle is active.

YES – Setting a value of "YES" for this parameter causes the control to execute the single-pass turning cycle after execution of any program block when the cycle is active.

NO – Setting a value of "NO" for this parameter causes the control to execute the single-pass turning cycle only after execution of a program block that generates axis motion in the direction of the depth of cut for the cycle.

Axis	Parameter Number
All	[82]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes. This parameter also applies to all fixed turning cycles.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

Retract Amount in Grooving

Function

This parameter is used for lathe controls only.

This parameter is used for the grooving cycles called by these G-codes:

G Code System	Grooving in X	Grooving in Z
A	G75	G74
B	G75	G74
C	G76	G76

This parameter determines the retraction amount "e" for the grooving cycles. Normally for these cycles, the control attains the programmed depth of cut in steps. It infeeds a programmed amount and retracts an amount "e." This cut repeats until the control has reached the programmed depth. Enter the distance of retraction "e" for the grooving cycles.

Axis	Parameter Number
All	[113]

Range

0.00000 to 9999.90000 mm

or

0.00000 to 393.69685 inch

Notes

This is a global parameter; the value set here applies to all axes.

This parameter may also be changed on the control by using the fixed-cycle parameter table.

END OF CHAPTER

Chapter 25

Interrupt Paramacros

Chapter Overview

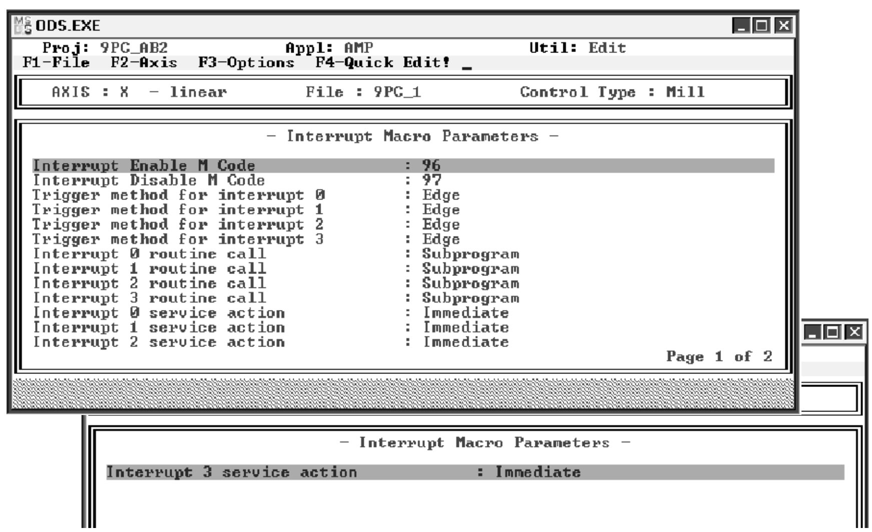
This chapter discusses the AMP parameters that you can set for the interrupt paramacro feature, which is used to interrupt normal program execution in response to some external signal, and allow execution of a paramacro or subprogram. After execution of the paramacro or subprogram, the control automatically returns to the interrupted program.

There are five types of interrupt macros available:

- Interrupt 0
- Interrupt 1
- Interrupt 2
- Interrupt 3

Refer to your programming and operation manual for more information.

These screens become available when you select the “Interrupt Macro Parameters” group from the main AMP menu:



Your screens may differ slightly, depending on your application type.

Interrupt Macro Parameters

Refer to the following subsections for information about the AMP parameters used for the interrupt macro feature.

Interrupt Enable M-code

Function

This parameter determines the M-code that is used to enable the interrupt macro feature. This M-code is modal with the M-code that is determined with the parameter “Interrupt Disable M-code”. When the interrupt enabled mode is activated with this M-code, the control allows execution of an interrupt macro.

Axis	Parameter Number
All	[114]

Range

10 to 97

Notes

This is a global parameter. The value set here applies to all axes.

Interrupt Disable M-code

Function

This parameter determines the M-code that is used to disable the interrupt macro feature. This M-code is modal with the one that is determined by the parameter “Interrupt Enable M-code”. When the interrupt disabled mode is activated with this M-code, the control does not allow execution of an interrupt macro.

Axis	Parameter Number
All	[115]

Range

10 to 97

Notes

This is a global parameter. The value set here applies to all axes and processes.

Trigger Method for Interrupt 0 – 3

Function

This parameter is typically used as a form of safety feature to help prevent an interrupt paramacro from executing immediately when the enabled mode becomes effective. There are two options for this parameter:

Edge – When a trigger method of edge is selected, the control recognizes a request for an interrupt macro only if the switch that calls the interrupt macro makes a transition from the false to a true state. If the switch that calls an interrupt macro is inadvertently left on when the interrupt macro is enabled, the control will not recognize an interrupt macro call unless the switch is turned off and then on again.

Level – When a trigger method of level is selected, the control only recognizes whether the switch that calls the interrupt macro is true or false. If this switch is inadvertently left on when the interrupt macro is enabled, the control will recognize the interrupt macro request and execute it in the same block that enables the interrupt macro.

Parameter	Parameter Number
Trigger method for interrupt 0	[116]
Trigger method for interrupt 1	[117]
Trigger method for interrupt 2	[118]
Trigger method for interrupt 3	[119]

Range

Selection	Result
(a)	Edge
(b)	Level

Notes

This is a global parameter. The value set here applies to all axes.

Interrupt 0 – 3 Routine Call

Function

This parameter determines the type of execution that the interrupt paramacro call will use. Interrupt paramacros can be called as either a subprogram or a nonmodal paramacro.

Subprogram – When subprogram is selected for this parameter, the control executes the interrupt macro as if a subprogram call (M98) were being used. This adds as one of the four nesting levels of subprograms. The interrupt macro is executed by using the same provisions as a subprogram call.

Macro – When macro is selected for this parameter, the control executes the interrupt macro as if a non-modal paramacro call (G65) were being used. This adds as one of the four nesting levels of the paramacros. The interrupt macro is executed with the same provisions as a G65 paramacro call.

The key difference a between macro type and a subprogram type of call (aside from maximum nesting level considerations) is that the subprogram call uses the same local parameter assignments as the calling program. The macro type of call gets its own set of local parameter assignments.

Parameter	Parameter Number
Interrupt 0 routine call	[120]
Interrupt 1 routine call	[121]
Interrupt 2 routine call	[122]
Interrupt 3 routine call	[123]

Range

Selection	Result
(a)	Subprogram
(b)	Macro

Notes

This is a global parameter. The value set here applies to all axes.

Interrupt 0 Service Action

Function

This parameter determines when program execution will be interrupted by an interrupt paramacro signal that is received when the control is executing a program block. When the control is not executing a block (Cycle Suspend or in between blocks in Single Block mode), the interrupt macro is always executed when the signal is received. There are two available methods for the point of interruption during program block execution. They are:

Immediate – When immediate is selected for this parameter, the control will halt program execution regardless of the current status of a program block, and execute the interrupt macro. When the interrupt macro is completed, the control will return execution to the point in the block where execution was halted.

Delay – When delay is selected for this parameter, the control will wait to halt program execution until the currently executing block is completed. The interrupt macro will then be executed in between blocks. When the interrupt macro is completed, the control will return execution to the beginning of the next block.

Parameter	Parameter Number
Interrupt 0 service action	[124]
Interrupt 1 service action	[125]
Interrupt 2 service action	[126]
Interrupt 3 service action	[127]

Range

Selection	Result
(a)	Immediate
(b)	Delayed

Notes

This is a global parameter. The value set here applies to all axes and processes.

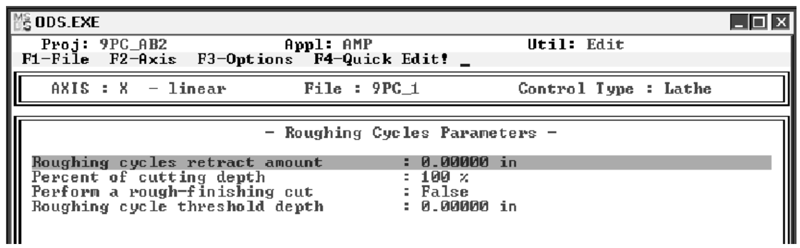
END OF CHAPTER

Roughing Cycle Parameters

Chapter Overview

This chapter offers a discussion on the roughing cycle parameters (sometimes referred to as compound turning routines). Roughing cycles are only available for lathe control types.

The following screens become available when “Roughing Cycle Parameters” is selected from the main menu screen:



Important: Roughing cycles are not available for mill control types. If an attempt is made to access this parameter group when a mill control type has been selected, the message “No Parameters For Current Configuration” is displayed on the work station. It is necessary to select a lathe control type to set any parameters in this parameter group.

Roughing Cycle Parameters

Roughing cycle parameters are set for the cycles called by these G-codes:

Description	G code System A	G code System B	G code System C
O.D. and I.D finishing Cycle	G70	G70	G72
O.D and I.D Roughing Cycle	G71	G71	G73
Rough Facing Routine	G72	G72	G74
Casting/Forging Roughing Cycle	G73	G73	G75

Refer to your programming and operation manual for more information on these G-codes.

Roughing Cycles Retract Amount

Function

This parameter only applies to the O.D and I.D Roughing routine and the Rough Facing routine. This parameter sets the default value of “R” for the roughing cycles. “R” is the distance that the cutting tool is retracted after each rough cut is made across the parts contour.

The value set here with this parameter is used only when no R–word is programmed in the block that defines the parameters for the roughing/facing routines. If an R word is programmed in the program block, then the value set with this parameter is ignored.

Axis	Parameter Number
All	[21]

Range

0.00000 in to 393.69685 in

or

0.00000 mm to 9999.90000 mm

Notes

This parameter is a global parameter. The value set here applies to all axes.

Percent of Cutting Depth

Function

This parameter only applies to the O.D and I.D Roughing Cycle and the Rough Facing cycle (G71 and G72 lathe G code system C). This parameter is used as an override for a programmed depth of cut. Typically the programmer determines the depth of each cut for each pass with these cycles by programming a "D" word in the calling block. Setting this parameter at 100% allows the full depth of cut as programmed with a D word to be made.

When a value other than 100% is entered for this parameter, then that percentage, times the programmed "D" word, is used as the depth of cut for each pass of these cycles. Typically this is done when the material being cut is changed or if a different quality finish is desired, and it is not convenient to go into the program and alter the value of "D."

Axis	Parameter Number
All	[22]

Range

0 to 255%

Notes

This parameter is a global parameter. The value set here applies to all axes and processes.

Perform a Rough-Finish Cut

Function

This parameter applies only to the O.D and I.D Roughing Cycle and the Rough Facing cycle. This parameter is used to force the control to make a finishing pass that cuts parallel to the workpiece's exact contour. Typically this finishing pass is defined with the I and/or K words in the calling block. This finishing pass is made at the same feedrate as the rough contouring routine used.

Axis	Parameter Number
All	[23]

Important: This parameter is not related to the finishing cut made by the O.D. and I.D. finishing cycle. If an amount of material is to be left on the workpiece to be removed later by a O.D and I.D. finishing cycle, the finishing pass discussed here will not affect this material. This parameter is related to the finishing pass that is made immediately after the basic contour of the part has been roughed out with a O.D. and I.D. Roughing Cycle or a Rough Facing Routine.

When this parameter is set:	And if:	Then:
TRUE	an I and/or K parameter is programmed in the calling block, then the amount of material removed on this last pass is equal to the I and/or K amounts	the control always cuts a finishing pass as the last pass of the O.D and I.D Roughing Cycle or the Rough Facing cycle
	no I and/or K is programmed in the calling block, then the finishing pass is made along the workpiece contour (removing whatever material is left by the roughing operation) + or – the amount programmed for an O.D. and I.D. finishing cycle (programmed with a U and/or W if any)	
FALSE	the I and/or K parameters are programmed in the calling block	this is the only time that a finishing pass is made during these cycles
	no I and/or K parameters are programmed	no finishing pass is made

Range

Selection	Result
(a)	True
(b)	False

Notes

This parameter is a global parameter. The value set here applies to all axes and processes.

Roughing Cycle Threshold Depth

Function

This parameter applies to the O.D and I.D Roughing Cycle, and the Rough Facing cycle (G71 and G72 lathe G-code system C). Use this parameter to establish the least amount of material that can be removed in a single pass.

Typically the programmer determines the depth of each pass in these cycles by programming a “D” word in the calling block. If the depth of pass generated by this D word is smaller than the value assigned with this parameter, the control will increase the depth of cut up to the value assigned here. If the last finishing pass is calculated to be smaller than this value, the control will decrease prior roughing cuts to allow enough material to remain for the final pass to be as large as the value of
this parameter.

Axis	Parameter Number
All	[83]

Range

0.00000 in to 393.69685 in

or

0.00000 mm to 9999.90000 mm

Notes

This parameter is a global parameter. The value set here applies to all axes.

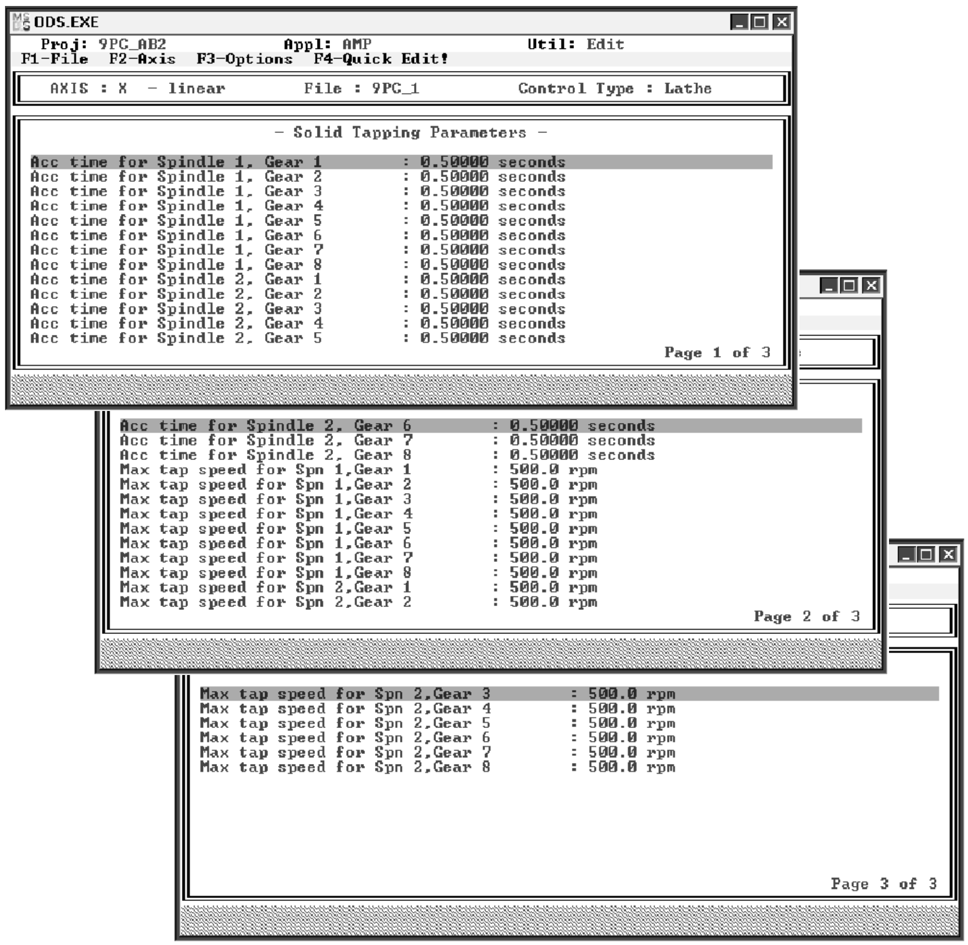
END OF CHAPTER

Solid-tapping Parameters

Chapter Overview

Solid-tapping parameters let you specify AMP ramps and maximum tapping speeds for each gear of each spindle. The two types of solid-tapping parameters are acceleration time and maximum tapping speed.

When you select the "Solid-tapping Parameters" group from the main menu in AMP, the workstation displays these screens:



Important: For solid-tapping, if the AMPed tapping axis is defined as a zero following error (ZFE) position loop type, then during solid-tapping, the spindle will be forced to be the same loop type and have the same percent feed forward as the tapping axis.

Acceleration Time

Function

Enter the time in seconds required for the spindle to accelerate from 0 rpm to the maximum solid-tapping speed in this gear.

For example, enter 0.125 if it takes 125 msec to accelerate from 0 rpm to the maximum solid-tapping speed in this gear.

Parameter	Parameter Number
Acc time for Spindle 1, Gear 1	[882]
Acc time for Spindle 1, Gear 2	[883]
Acc time for Spindle 1, Gear 3	[884]
Acc time for Spindle 1, Gear 4	[885]
Acc time for Spindle 1, Gear 5	[886]
Acc time for Spindle 1, Gear 6	[887]
Acc time for Spindle 1, Gear 7	[888]
Acc time for Spindle 1, Gear 8	[889]
Acc time for Spindle 2, Gear 1	[812]
Acc time for Spindle 2, Gear 2	[813]
Acc time for Spindle 2, Gear 3	[814]
Acc time for Spindle 2, Gear 4	[815]
Acc time for Spindle 2, Gear 5	[816]
Acc time for Spindle 2, Gear 6	[817]
Acc time for Spindle 2, Gear 7	[818]
Acc time for Spindle 2, Gear 8	[819]

Range

0.00000 to 1000.00000 seconds

Notes

Values must be entered for all gear ranges specified by the parameter *Number of Gears Used*. If any of the gears used has no value or an illegal value entered here, the control assumes that no gears are available and never requests a gear change.

Use these parameters for spindle orient acceleration and acceleration during solid-tapping. For solid-tapping, the lower of the two accelerations (tapping axis and spindle) is applied to both motions.

Maximum Tapping Speed

Function

Enter the maximum speed in rpm at which solid-tapping can take place for this gear of this spindle.

Parameter	Parameter Number
Max tap speed for Spn 1, Gear 1	[753]
Max tap speed for Spn 1, Gear 2	[754]
Max tap speed for Spn 1, Gear 3	[755]
Max tap speed for Spn 1, Gear 4	[756]
Max tap speed for Spn 1, Gear 5	[757]
Max tap speed for Spn 1, Gear 6	[758]
Max tap speed for Spn 1, Gear 7	[759]
Max tap speed for Spn 1, Gear 8	[760]
Max tap speed for Spn 2, Gear 1	[761]
Max tap speed for Spn 2, Gear 2	[762]
Max tap speed for Spn 2, Gear 3	[763]
Max tap speed for Spn 2, Gear 4	[764]
Max tap speed for Spn 2, Gear 5	[765]
Max tap speed for Spn 2, Gear 6	[766]
Max tap speed for Spn 2, Gear 7	[767]
Max tap speed for Spn 2, Gear 8	[768]

Range

0.0 to Max. Spindle Speed for that gear for that spindle (rpm)

Notes

Values must be entered for all gear ranges specified by the parameter *Number of Gears Used*. If any of the gears used has no value or an illegal value entered here, the control assumes that no gears are available and never requests a gear change.

Important: This parameter also sets the spindle ramp duration when solid-tapping is not being used. Refer to page 12-3.

END OF CHAPTER

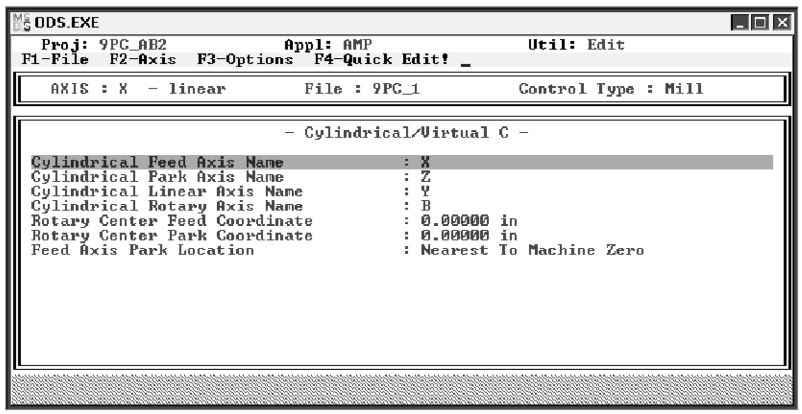
Cylindrical/Virtual C Parameters

Chapter Overview

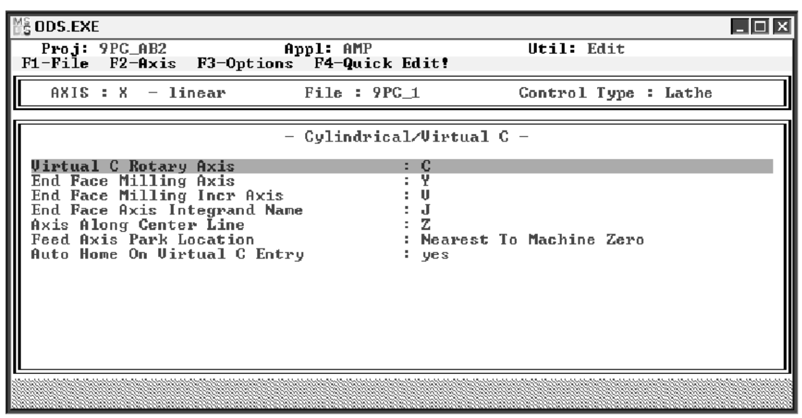
These parameters specify the configuration of the axes during cylindrical interpolation operations on a mill or Virtual C operations on a lathe. Refer to your programming and operation manual for more information on these features. If you are using the multispindle feature, the Virtual C parameters apply only to Spindle 1.

The workstation displays these screens when you select the "Cylindrical/Virtual C Parameters" group from the main menu screen:

These parameters are displayed when you select control type "MILL"



These parameters are displayed when you select control type "Lathe"



Mill Cylindrical Interpolation Parameters

The mill cylindrical interpolation parameters are covered in the following subsections.

Refer to your programming and operation manual for details on the cylindrical interpolation feature.

Cylindrical Feed Axis Name

Function

This parameter specifies the name of the machine axis that will be the feed axis during cylindrical interpolation. The mill tool's position along the feed axis determines the depth of the contour cuts during cylindrical interpolation.

Important: The machine axis named as the feed axis must be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
Cylindrical Feed Axis Name	[233]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for mill controls only.

Cylindrical Park Axis Name

Function

This parameter specifies the name of the machine axis that will be the park axis during cylindrical interpolation. The park axis is used to align the mill tool with the center-line of the rotary axis.

Important: The machine axis named as the park axis must be an axis in the current machine configuration or configured as none.

When cylindrical interpolation is activated, the control moves the mill tool along the park axis to the axis position specified by the **Rotary Center Park Coordinate** parameter.

Once the tool is positioned at the park coordinate of the park axis, the park axis is locked at its current position. The control will not allow any additional park axis commands during cylindrical interpolation. This prevents the tool from being moved off the center-line of the rotary axis during cylindrical interpolation.

Parameter	Parameter Number
	Single Process
Cylindrical Park Axis Name	[232]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for mill controls only.

Cylindrical Linear Axis Name

Function

This parameter specifies the name of the machine axis that will be the cylindrical linear axis during cylindrical interpolation. The linear axis and the rotary axis will be the two axes of the circular plane during cylindrical interpolation. Linear axis motions are interpolated with rotary axis motions to cut contours on the cylindrical workpiece.

Important: The machine axis named as the linear axis must be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
Cylindrical Linear Axis Name	[230]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for mill controls only.

Cylindrical Rotary Axis Name

Function

This parameter specifies the name of the machine axis that will be the cylindrical rotary axis during cylindrical interpolation. The rotary axis and the linear axis will be the two axes of the circular plane during cylindrical interpolation. Rotary axis motions are interpolated with linear axis motions to cut contours on the cylindrical workpiece.

Important: The machine axis named as the rotary axis must be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
Cylindrical Rotary Axis Name	[231]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for mill controls only. F

Rotary Center Feed Coordinate

Function

This parameter specifies the cylindrical interpolation feed axis coordinate that corresponds to the center of the cylindrical interpolation rotary axis. This coordinate is used to align the feed axis with the center-line of the rotary axis.

Important: The center of the cylindrical interpolation rotary axis must coincide with the center of the cylindrical workpiece.

When cylindrical interpolation is activated, the control moves the mill tool along the feed axis to the feed axis coordinate specified by this parameter. This aligns the feed axis with the center-line of the rotary axis. The control then moves the tool along the feed axis to the radius specified by the R parameter in the G16.1 block.

Parameter	Parameter Number
	Single Process
Rotary Center Feed Coordinate	[235]

Range

-100000.00000 to 100000.00000 in.

or

-2540000.00000 to 2540000.00000 mm

Notes

This parameter is used for mill controls only.

Rotary Center Park Coordinate

Function

This parameter specifies the cylindrical interpolation park axis coordinate that corresponds to the center of the cylindrical interpolation rotary axis. This coordinate is used to align the park axis with the center-line of the rotary axis.

Important: The center of the cylindrical interpolation rotary axis must coincide with the center of the cylindrical workpiece.

When cylindrical interpolation is activated, the control moves the tool along the park axis to the park axis coordinate specified by this parameter. This aligns the park axis with the center-line of the rotary axis. The park axis is then "parked" by the control.

Parameter	Parameter Number
	Single Process
Rotary Center Park Coordinate	[234]

Range

-100000.00000 to 100000.00000 in.

or

-2540000.00000 to 2540000.00000 mm

Notes

This parameter is used for mill controls only.

Lathe Virtual C Parameters

The lathe Virtual C parameters are covered in the following subsections.

Virtual C Rotary Axis

Function

This parameter specifies the name of the machine axis that will be Virtual C rotary axis during Virtual C cylindrical interpolation. When Virtual C cylindrical interpolation is activated, the rotary axis commands are sent to the axis specified by this parameter.

Important: If the axis name specified by this parameter is **not** an axis in the current machine configuration, the rotary axis commands will be sent to the spindle.

Parameter	Parameter Number
	Single Process
Virtual C Rotary Axis	[236]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

End Face Milling Axis

Function

This parameter specifies the name of the machine axis that will be end face milling axis during Virtual C end face milling. This axis will be perpendicular to the diameter and feed axes during Virtual C end face milling.

Important: The machine axis named as the end face milling axis must **not** be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
End Face Milling Axis	[238]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for lathe controls only.

End Face Milling Incremental Axis

Function

Important: The **End Face Milling Incremental Axis** parameter is used only for lathe type A.

This parameter specifies the name of the machine axis that will be end face milling incremental axis during Virtual C end face milling. This incremental axis will be perpendicular to the diameter and feed axes during Virtual C end face milling.

Important: The machine axis named as the end face milling incremental axis must **not** be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
End Face Milling Incremental Axis	[239]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for lathe controls only.

End Face Axis Integrand Name

Function

This parameter specifies the integrand name of the end face milling axis during Virtual C end face milling. The end face axis integrand name is used while programming in modes where an integrand letter is expected.

The numeric value programmed with the end face axis integrand name is the integrand value for the end face milling axis. This value is used by the control to reference a point on the end face milling axis that is used in calculating arc centers, fixed cycle variables, and other similar functions.

Parameter	Parameter Number
	Single Process
End Face Axis Integrand Name	[240]

Range

Selection	Result
(a)	I
(b)	J
(c)	K
(d)	None

Notes

This parameter is used for lathe controls only.

Axis Along Center Line

Function

This parameter specifies the name of the machine axis that is parallel to the spindle center-line during Virtual C cylindrical interpolation. This axis is the linear axis during Virtual C cylindrical interpolation and the feed axis during Virtual C end face milling.

Important: The machine axis named as the axis along the center line must be an axis in the current machine configuration.

Parameter	Parameter Number
	Single Process
Axis Along Center Line	[237]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This parameter is used for lathe controls only.

Feed Axis Park Location

Function

This parameter specifies whether the tool is positioned along the feed axis nearest to machine zero or farthest from machine zero of the cylindrical workpiece. This parameter provides the option of cutting the contour on either side of the cylindrical workpiece.

Parameter	Parameter Number
	Single Process
Feed Axis Park Location	[242]

Range

Selection	Result
(a)	Nearest To Machine Zero
(b)	Farthest From Machine Zero

Notes

This parameter is used for mill and lathe controls.

Automatic Home on Virtual C Entry

Function

Important: This parameter and the discussion included here assumes that the lathe spindle has been defined to be the Virtual C axis for your machine. If this is not the case, ignore the discussion here and set this parameter to "NO."

This parameter specifies whether an automatic spindle homing operation is to be performed by the control when the Virtual C feature is activated. Just before the first move is performed in the Virtual C mode, the control sets the current spindle location to angle zero.

For some applications the location of this zero point is not significant and may be defined as any location in the 360 degree rotation of the spindle. However, for applications requiring that the contour be cut at a specific orientation of the part, it is necessary that this zero point be accurately defined and repeatable.

For orientation dependent applications it is recommended that a spindle homing operation be performed prior to entering the Virtual C mode.

One method of performing a spindle home operation is to use spindle orient (M19). Spindle homing is performed automatically when the control enters spindle orient mode. This method requires that the part program, logic program, or operator execute a spindle orient (M19S0) immediately prior to entering Virtual C mode.

Instead, this parameter can be used to force a spindle home operation each time Virtual C mode is entered. By setting this parameter to "YES," the control will automatically perform a spindle home operation and locate the spindle to zero. This homing operation will occur immediately before the first move is performed inVirtual C mode.

Parameter	Parameter Number
	Single Process
Automatic Home on Virtual C Entry	[243]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This parameter is used for lathe controls only.

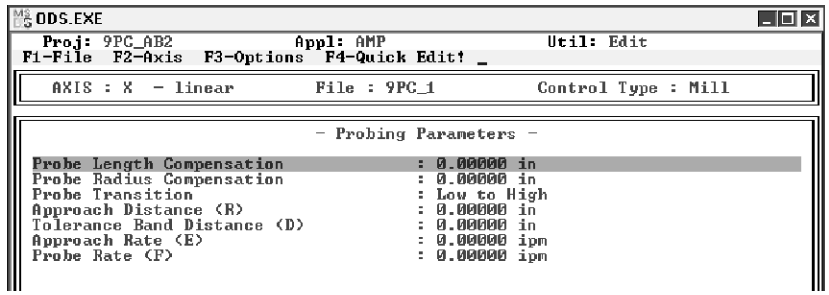
END OF CHAPTER

Probing Parameters

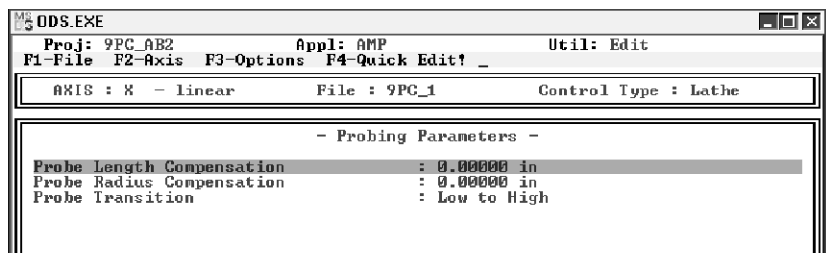
Chapter Overview

The workstation displays this screen when the “Probing Parameters” group is selected:

When you select control type “MILL” these parameters are displayed:



When you select control type “Lathe” these parameters are displayed :



If the application type is:	and the control type is:	these parameters do not appear:
Mill/Lathe	Lathe	• Approach Distance (R) • Tolerance Band Distance (D) • Approach Rate (E) • Probe Rate (F)

Adaptive depth probe is discussed on page 30-1.

Probe Length Compensation

Function

The value entered for this parameter is the probe length used for probe length compensation in G37’s cycles. Probe length is the distance from the tool Gauge Point on the tool holder to the center of the probe radius, as measured only along the axis that the probe extends.

For example, if the probe extends along the Z axis, but is mounted off center from the tool Gauge Point, the Probe Length Compensation amount is only the amount that it extends along the Z axis.

Axis	Parameter Number
	Single Process
All	[600]

Range

0.00000 to 1270.00000 mm, or

0.00000 to 50.00000 in.

This is a per process parameter. The value set here applies to all axes.

Probe Radius Compensation

Function

The value entered for this parameter is the probe radius used for probe radius compensation in G31's, G37's, and G38's cycles. Probe radius refers to the radius of the probe tip. If the documentation for your probe lists a tip diameter, enter half of that value here.

Axis	Parameter Number
	Single Process
All	[601]

Range

0.00000 to 254.00000 mm or

0.00000 to 10.00000 in.

This is a per process parameter. The value set here applies to all axes.

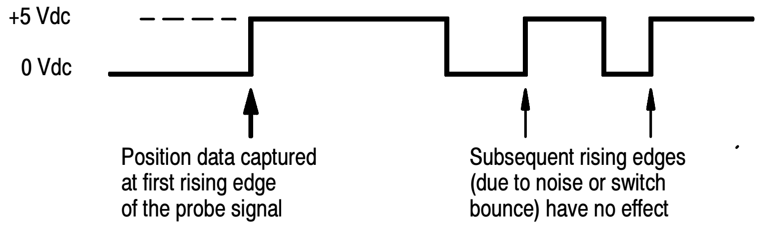
Probe Transition

Function

This parameter is used to determine when the control recognizes a signal from a probe to execute one of the skip or probing functions. The control recognizes a probe trigger when the probe signal is turned on or off.

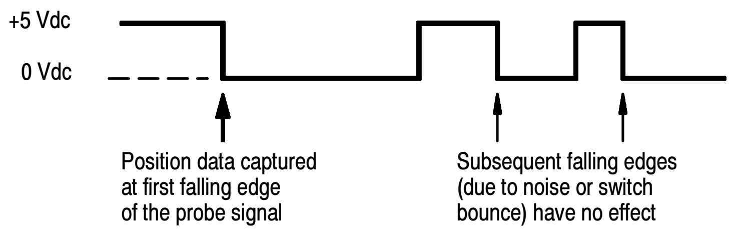
Low to High – When this parameter is set at "Low to High," the control recognizes a probe signal when the probe turns on (rising edge of the signal).

Figure 29.1
Low to High Probe Transition



High to Low – When this parameter is set at "High to Low," the control recognizes a probe signal when the probe turns off (falling edge of the signal).

Figure 29.2
High to Low Probe Transition



Axis	Parameter Number
All	[602]

Range

Selection	Result
(a)	Low to High
(b)	High to Low

Notes

This is a global parameter. The value set here applies to all axes and processes. All processes must use the same probe transition type.

G38, G38.1 Probing Cycle Parameters

This and the following sections discuss the parameters that are used with the G38 hole probing and G38.1 part rotation probing cycles. These parameters correspond directly to a G38 or G38.1 block parameter entered in the part program. For example, the value for the probe approach distance can be entered as under the **Approach Distance "R"** parameter here, or it can be entered as an "R" value in the G38 block. A value entered in the part program block will override the value entered in AMP.

Typically the G38 hole probing cycle is used to measure the diameter of a hole in a part by using a touch probe. The G38.1 part rotation probing cycle is used to measure (with a touch probe) the amount that a part is out of parallel with a selected axis. These cycles are discussed in detail in your programming and operation manual. Their operation is also very Logic-dependant. Refer to documentation prepared by the system installer for more information.

Important: There are also two skip functions (G31s and G37s) that may use a probe for tool measurement or other functions. The AMP parameters for these cycles are discussed in chapter 21 of this manual.

These parameters are for mill applications only.

Approach Distance (R)

Function

This parameter is used by both the G38 and G38.1 probing cycles.

Enter the distance from the start-point of the probing cycle to a point where the feedrate is to be slowed. The feedrate should be slowed at a reasonable distance from the expected end-point of the probing move to allow a more accurate measurement of the axis position when the probe is fired. At this point, the feedrate will slow from the **Approach Feedrate "E"** to the **Probing Feedrate "F"**.

This parameter may also be entered directly in the probing cycle block of the part program with an "R" word. If the "R" word is present in the probing cycle block, the value entered here is ignored.

Axis	Parameter Number
	Single Process
All	[603]

Range

0.00000 to 99999.00000 mm

or

0.00000 to 3936.96850 in.

Notes

This is a per process parameter. The value set here applies to all axes.

This parameter may also be changed on the control by using the fixed cycle parameter table.

This parameter is for mill applications only.

Tolerance Band Distance (D)

Function

This parameter is used by both the G38 and G38.1 probing cycles.

The value entered for this parameter defines a tolerance band on either side of the coordinate of the expected firing point (as entered in the probing cycle block) for the probe. Enter a value for this parameter "D" that defines a tolerance distance on either side of the expected firing point of the probe. This value is added to and subtracted from the expected firing point of the probe making the band width twice this value. This parameter is an unsigned value.

If the probe does not fire within the tolerance band defined by this parameter, a PROBE ERROR occurs.

This parameter may also be entered directly in the probing cycle block of the part program with a "D" word. If the "D" word is present in the probing cycle block, the value entered here is ignored.

Axis	Parameter Number
	Single Process
All	[604]

Range

0.00000 to 99999.00000 mm

or

0.00000 to 3936.96850 in.

Notes

This is a per process parameter. The value set here applies to all axes.

This parameter may also be changed on the control by using the fixed cycle parameter table.

This parameter is for mill applications only.

Approach Rate (E)

Function

This parameter is used by both the G38 and G38.1 probing cycles.

The value entered for this parameter defines a feedrate at which the probe is to approach the location specified by the “R” parameter. This feedrate may be relatively high to allow for a faster cycle time. After the probe reaches the point specified by the “R” parameter, the feedrate will slow to that rate specified by the “F” parameter.

This parameter may also be entered directly in the probing cycle block of the part program with an “E” word. If the “E” word is present in the probing cycle block, the value entered here is ignored.

Axis	Parameter Number
	Single Process
All	[605]

Range

0.00000 to 5999940.00000 mmpm

or

0.00000 to 236218.11024 ipm

Notes

This parameter may also be changed on the control by using the fixed cycle parameter table.

This parameter is for mill applications only.

Probe Rate (F)

Function

This parameter is used by both the G38 and G38.1 probing cycles.

The value entered for this parameter defines a feedrate at which the probe is to be moved after passing the point defined by the “R” parameter. This feedrate should be relatively low to allow for a more accurate measurement of the coordinate position when the probe is fired.

This parameter may also be entered directly in the probing cycle block of the part program with an “F” word. If the “F” word is present in the probing cycle block, the value entered here is ignored.

Axis	Parameter Number
	Single Process
All	[606]

Range

0.00000 to 5999940.00000 mmpm

or

0.00000 to 236218.11024 ipm

Notes

This parameter may also be changed on the control by using the fixed cycle parameter table.

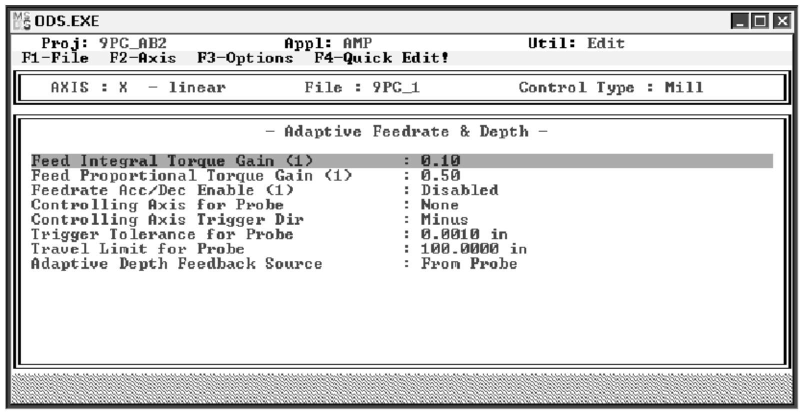
This parameter is for mill applications only.

END OF CHAPTER

Adaptive Feed & Depth Parameters

Chapter Overview

When you select the “Adaptive Feed & Depth” group the workstation displays this screen:



The Adaptive Feed & Depth parameter group is only available for mill control types. These parameters are described on these pages:

Parameter:	Page:
Feed Integral Torque Gain	30-1
Feed Proportional Torque Gain	30-2
Feedrate Acc/Dec Enable	30-3
Controlling Axis for Probe	30-8
Controlling Axis Trigger Dir.	30-9
Trigger Tolerance for Probe	30-10
Travel Limit for Probe	30-11
Adaptive depth feedback source	30-12

Feed Integral Torque Gain

Function

This parameter specifies the integral torque gain for an axis move that is performed in adaptive feed mode. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

This parameter (in conjunction with the parameter Feed Proportional Torque Gain) is used to determine the net change to the 9/PC feedrate command. The feedrate override is modified in order to try to maintain a constant torque. The desired torque is specified in the program block.

This parameter specifies the ratio of % change in feedrate to the existing integrated torque error. When you increase the value of the Feed Integral Torque Gain, it will cause the integrator to have a greater effect on the adaptive feedrate. The integrator helps smooth out the response to instantaneous changes in torque error smoothing out the 9/PC feedrate commands.

Feed Integral Torque Gain Axis number	Parameter Number
Axis (1)	[1152]
Axis (2)	[2152]
Axis (3)	[3152]
Axis (4)	[4152]
Axis (5)	[5152]
Axis (6)	[6152]
Axis (7)	[7152]
Axis (8)	[8152]

Range

0.1 to 2.0

Notes

This parameter must be set independently for each axis.

Feed Proportional Torque Gain

Function

This parameter specifies the proportional torque gain for an axis move that is performed in adaptive feed mode. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

This parameter (in conjunction with the parameter Feed Integral Torque Gain) is used to determine the net change to the 9/PC feedrate command. The feedrate is modified in order to try to maintain a constant torque. The desired torque is specified in the G25 program block.

This parameter specifies the ratio of % change in feedrate to the instantaneous torque error. When you increase the value of the Feed Proportional Torque Gain, it will cause the instantaneous torque error to have a greater effect on the adaptive feedrate. This parameter is typically less than the Feed Integral Torque Gain parameter.

Feed Proportional Torque Gain Axis number	Parameter Number
Axis (1)	[1153]
Axis (2)	[2153]
Axis (3)	[3153]
Axis (4)	[4153]
Axis (5)	[5153]
Axis (6)	[6153]
Axis (7)	[7153]
Axis (8)	[8153]

Range

0.1 to 2.0

Notes

This parameter must be set independently for each axis.

The adaptive feed mode can be programmed on any closed loop axis except an axis that positions more than one servo (such as dual or deskew axes).

If you are going to use the adaptive feed feature on an analog system the system must be configured to run in tachless operation. Refer to page 7-9 for details on configuring a tachless velocity loop on an analog system.

Feedrate Acc/Dec Enable

Function

The feedrate Acc/Dec Enable parameter is used to enable Acc/Dec for an axis that is in adaptive feed mode. Adaptive feed mode is used to cause the servo to maintain a constant torque while cutting by varying the axis feedrate. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

Acc/Dec on an adaptive feed axis is used to smooth out the shock that could occur to an axis from the rapid application of increasing/decreasing torque by the servo.

If you select:	it:
enable	causes the programmed torque to be ramped as the control ramps the feedrate using linear Acc/Dec. On shorter moves, that do not extend beyond the Acc/Dec ramps, the programmed torque may never be reached by the servo. This Acc/Dec ramping is applied over top of the control's already modified feedrate command being used to obtain the programmed torque.
disable	causes the control to issue non-accelerate/decelerated feedrate commands and may in some cases cause rough or choppy machine motion as sudden acceleration or deceleration occurs.

Feedrate Acc/Dec Enable Axis number	Parameter Number
Axis (1)	[1154]
Axis (2)	[2154]
Axis (3)	[3154]
Axis (4)	[4154]
Axis (5)	[5154]
Axis (6)	[6154]
Axis (7)	[7154]
Axis (8)	[8154]

Range

Selection	Result
(a)	Enabled
(b)	Disabled

Notes

This parameter must be set independently for each axis.

Adaptive Depth AMP Overview

The adaptive depth feature uses an adaptive depth probe that is attached to an axis on the control. Typically a probe returns feedback very similar to an A quad B incremental encoder. The control then uses this feedback to determine the actual position of the axis the probe is attached to relative to some surface that the probe is deflecting from (typically the part surface).

The adaptive depth probe is wired into a feedback port just as any other incremental encoder feedback device. Refer to your *9/PC Installation and Integration Manual* for information on wiring feedback devices and specifications for valid adaptive depth probes.

This section discusses AMP parameters that impact how the adaptive depth probe operates. They are:

Parameter name:	Page:	Selects the:
Controlling Axis for Probe	30-8	axis that the adaptive depth probe is attached to.
Direction of Probe Trip	30-9	Defines the direction from zero that indicates probe deflection into the part.
Probe Trigger Tolerance	30-10	amount of probe deflection necessary for the control to assume contact with the part has been made.
Depth sensor travel limit	30-11	maximum distance the probe will be allowed to deflect.
Adaptive Depth Feedback Source	30-12	feedback device for positioning while the probe is in contact with the part. Select from either the controlling axis feedback, or the probe feedback.

In addition to these AMP parameters, you must also configure the adaptive depth probe as a feedback device in the servo parameters group (analog or digital).

Do this by first creating an axis. If you intend to use the feedback from the probe to control positioning (only available after the probe has tripped), many of the controlling axis' parameters must also be copied to the adaptive depth probe. This is not a real physical axis, it is only the name of the adaptive depth probe and is for configuration purposes only. This axis must be configured as having no output port. It will have a feedback port only. You can mount the probe on any real physical axis connected to the same 1394 amplifier as the adaptive depth probe. The axis you mount the probe on is selected with the Controlling Axis for Probe parameter and the controlling axis name and its associated integrand letter is used when programming the G26 block.

Configure an adaptive depth probe by first creating a normal linear axis using the F2 [Configure Axis] option. Name the axis using F2 [Name Axis] option. This adaptive depth axis must be configured after all physical axes including any deskew axis slaves but before any spindles.

Servo Parameters for the Adaptive Depth Probe

In the servo parameter groups the following servo parameters must be configured as described in the table below for the adaptive depth probe axis. The adaptive depth probe axis is not the same axis as the controlling axis configured on page 30-8.

Important: Other servo parameters not listed in this table must be set identically to the adaptive depth controlling axis (the axis the probe is physically riding on). We recommend copying the controlling axis into the adaptive depth axis (using the [F2] Copy Axis Option) before setting any other servo parameters for the adaptive depth probe.

Use of gear ranges on the probe position feedback parameters is not supported. The probe feedback is designed to be a device mounted directly on the machine slide, in parallel with axis motion. No gearing can be present between the axis motion and probe feedback.

Important: You can not configure an adaptive depth probe in conjunction with an auxiliary feedback device (additional encoder for position loop). An axis with the adaptive depth probe can only have the primary feedback device and the adaptive depth probe.

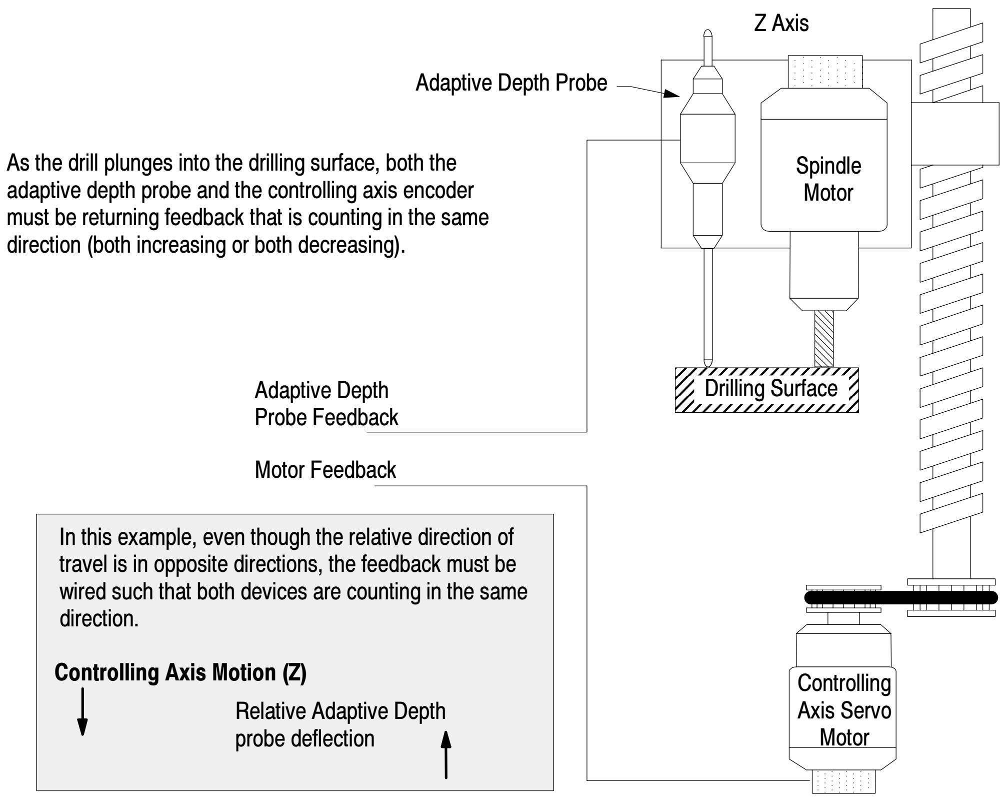
After you have copied the adaptive depth controlling axis into the adaptive depth axis make the following additional settings:

Servo Parameter:	For Adaptive Depth Probe:
Servo Position Loop Type	Select “Depth Probe” for the loop type.
Position Loop Feedback Port	Select the port the adaptive depth probe is connected to. This port must be on the same servo module as the probe controlling axis.
Position Loop Feedback Type	Refer to the specifications of your depth probe. Typically depth probes are configured as INC Encoder A/B/Z (Z<A).
Position Feedback Counts/Rev	Enter the number of counts your depth probe produces per inch or per mm (depending on the units you are using as selected with the F3 option this may be counts per foot, inch, meter, centimeter, or millimeter). If the resolution on your probe is not an even number of counts per inch (or mm), compensate for this using the parameter Lead Screw Pitch. The resolution of the adaptive depth probe must be close to or higher than the axis encoder resolution it is paired with when the adaptive depth probe is selected to close the position loop. If the resolution of the probe is too coarse relative to the axis it is paired with, the control generates an error doesn’t come out of E-Stop.
Lead screw thread pitch	Enter the dimension used in the denominator of the parameter Number of Position Feedback Counts/Rev. For example if you entered 2000 counts/1in. this parameter must be set at 1.000 inch. If you entered 3000counts/.5inch this parameter must be set at 0.5000 inch.
Sign of Position Feedback	After you have finished installing/configuring your probe, check the axis monitor page on the control’s CRT to determine which direction the probe is counting when deflecting. The probe must count in the same direction as the setting of **Controlling Axis Trigger Dir**. If the probe counts in the wrong direction, rather than re-wiring the probe feedback, you can simply reverse the sign of this parameter. For more information on setting this parameter, see the section after this table called “Setting Trigger Direction and Feedback Direction”

Output Port Number	Must be set as “No Output”.
Teeth on motor gear for pos. FB	Must be set at “1”.
Teeth on lead screw for pos. FB	Must be set at “1”.

Assigning Trigger Direction and Feedback Direction

If you are using an adaptive depth probe, it is very important to make sure that the probe’s position register (on the axis monitor page) is counting in the same direction as the probe’s trigger direction. For example, if the parameter **Controlling Axis Trigger Dir** is negative, then the probe position register must count down as the probe is depressed.



Use the monitor pages (as discussed in your *9/PC CNC Software and Hardware Installation and Integration Manual*) to see the direction the feedback is counting. You can reverse the feedback on the adaptive depth probe either by rewiring the probe, or using the AMP parameter “Sign of Position Feedback”.

Controlling Axis for Probe

This parameter is used with the adaptive depth feature. Refer to your *9/PC CNC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

This parameter specifies the name of the real, physical axis that is positioning the adaptive depth probe (the actual axis the probe is riding on). This is not the name of the adaptive depth axis used to configure servo parameters. You will use this axis name when programming the G26 block. You can mount the probe on any real physical axis that is connected to the same servo card as the depth probe. Many of the servo parameters for this axis must be copied into the adaptive depth axis (refer to page 30-5).

Enter the name you previously chose with the F2 option for this axis.

Axis	Parameter Number
	Single Process
All	[633]

Range

Selection	Result	Selection	Result
(a)	A	(i)	Z
(b)	B	(k)	$B
(c)	C	(l)	$C
(d)	U	(m)	$X
(e)	V	(n)	$Y
(f)	W	(o)	$Z
(g)	X	(p)	none
(h)	Y		

Notes

This parameter applies to the adaptive depth probe and selects what axis is programmed in a G26 block.

Important: The direction the adaptive depth probe feedback counts must be the same direction that the adaptive depth controlling axis encoder is counting when the axis is moving in the same direction. See page 30-7 for details.

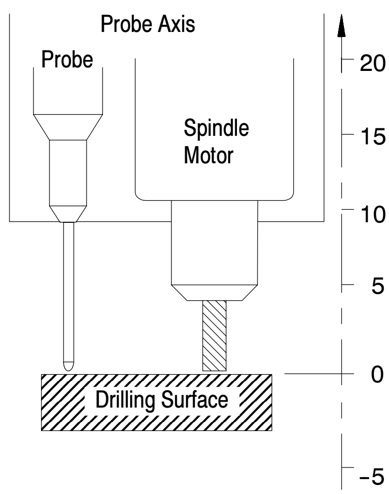
Controlling Axis Trigger Dir

Function

This parameter is used with the adaptive depth feature. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

After homing the probe (either automatically after power turn on or through the logic) the control needs to know which direction from that zero point is into the part. Since the probe can be depressed a significant amount when at zero, the control has no way of knowing which direction of probe travel is into the part.

Use this parameter to determine the direction from the established home position of the probe that indicates the adaptive depth controlling axis is moving into the part. This is the direction programmed in the G26 block that would move the tool further into the part. If moving the controlling axis from Z10 to Z5 would move the tool into the part enter a value of minus for this parameter. If moving the controlling axis from Z5 to Z10 would move the tool into the part enter a value of plus for this parameter.



As the drill plunges into the drilling surface, the controlling axis' position registers are decreasing (Z10 to Z5). Enter minus for this example.

Axis	Parameter Number
	Single Process
All	[634]

Range

Selection	Result
(a)	Positive (counting up)
(b)	Negative (counting down)

Notes

This parameter applies to the adaptive depth probe.

Probe Trigger Tolerance

Function

This parameter is used with the adaptive depth feature. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

With an adaptive depth probe occasional misfires can occur as some erroneous probe deflection is detected when the adaptive depth axis accelerates/decelerates or simply from axis vibration. This is typically dependant on mechanical features like probe sensitivity, machine configuration and rigidity, etc.

To avoid these probe misfires this parameter allows you to enter a dead band. This dead band is the amount of deflection the probe can return to the control without registering a probe trip.

When probe deflection is less then the value entered for this parameter, the control ignores the probe deflection.

When probe deflection is greater than or equal to the value entered for this parameter, the control considers the probe tripped. Any deflection that has occurred, up until the probe trip is recognized, is added as probe deflection from the point at which the probe tripped.

Axis	Parameter Number
	Single Process
All	[630]

Range

0 to 2540 mm

Notes

This parameter applies to the adaptive depth axis.

Excessively large values for this trigger tolerance can decrease your axis speed when searching for the part surface with the adaptive depth probe. The control will attempt to keep the axis speed slow enough to leave enough time to decelerate after the probe has tripped to reach final position. For example if your probe tolerance for this parameter was 20mm and you programmed a G26 integrand amount of 21mm, this move can be very slow since the axis only has 1 mm within which to decelerate once the probe trips.

Depth Sensor Travel Limit

Function

This parameter is used with the adaptive depth feature. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

Most adaptive depth probes have a travel limit that is less than the travel limits of the axis that is positioning the probe. This parameter is used to define the maximum amount of probe deflection allowed. It is an incremental value that is relative to where you configure your probe zero point.

Important: This depth sensor travel limit is not an absolute value. As such, the overall probe deflection allowed is dependent on where the operator determines the probes' zero point to be. Make sure this parameter is small enough to compensate for any offset the operator may enter when the probe zero point is established.

When your total probe deflection reaches the value entered for this parameter an error is displayed on the CRT and the control enters cycle stop. This error can be monitored in the Logic using the BR_ADPSTA flag. You can monitor the adaptive depth probe position using the axis monitor screens (refer to your *9/PC CNC Software and Hardware Installation and Integration Manual*).

Axis	Parameter Number
	Single Process
All	[631]

Range

0 to 2540 mm

Notes

This parameter applies to the adaptive depth probe.

Adaptive Depth Feedback Source

Function

This parameter is used with the adaptive depth feature. Refer to your *9/PC Mill Operation and Programming Manual* for details on this feature. This parameter is only available on mill control types.

Often the accuracy of the adaptive depth probe resolution is higher then the resolution of the axis encoder. In these cases it may be desirable to allow the adaptive depth probe to supply position feedback and close the axis' position loop after the probe has tripped.

Selecting the **From Axis** option with this parameter will cause the servo position loop to be closed by the adaptive depth controlling axis encoder for all moves. G26 adaptive depth moves will use probe feedback only to determine the actual depth of the hole. Positioning accuracy is limited to the resolution of the feedback for the adaptive depth controlling axis.

Selecting the **From Probe** option with this parameter will cause the servo position loop to be closed by the adaptive depth probe (once it has fired) when performing an adaptive depth move (G26). All other non G26 moves will use normal adaptive depth axis encoder feedback. The From Probe option allows you to take advantage of any positioning accuracy gained by the adaptive depth probe.

Axis	Parameter Number
	Single Process
All	[632]

Range

Selection	Result
(a)	From Probe
(b)	From Axis

Notes

This parameter applies to the axis performing the adaptive depth operation.

When the From Probe option is selected programming of the G26 block hole depth (axis integrand word) can be done using the resolution of the probe. Keep in mind the axis letter/integrand programing resolution is determined in the Axis Program Format Parameter group. The integrand word uses the same format as defined for the axis word. You should configure the axis word format to the resolution of the adaptive depth probe. This allows you to use the adaptive depth probes greater resolution and truncates the extra resolution when programmed for non-adaptive depth moves.

END OF CHAPTER

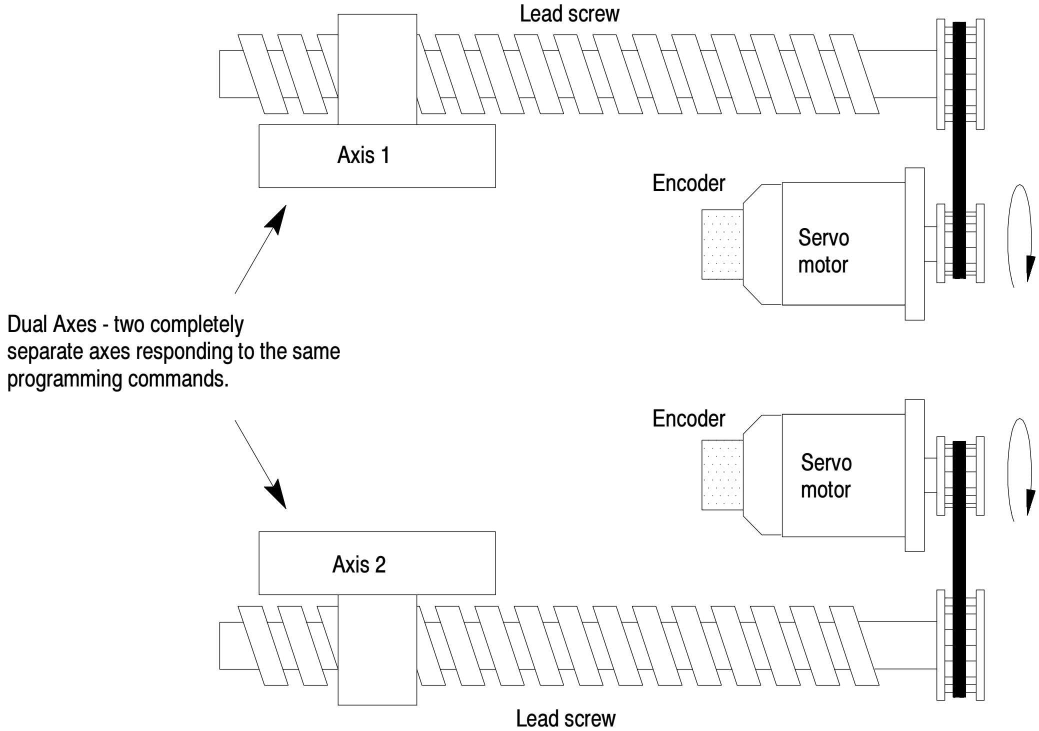
Dual Axis Parameters

Chapter Overview

This section describes the parameters used to configure dual axes. The dual axes feature lets the part programmer simultaneously control multiple axes while programming commands for only one. This feature is especially useful for gantry mills with dual cutting heads and other machines running with parallel cutting tools.

Implementing the dual axis feature can require significant logic modification as well as the AMP parameters discussed in this chapter.

Figure 31.1
Dual Axis Configuration



The control can support five dual axis groups. A dual axis group consists of two or more grouped axes coupled through AMP and commanded by a master axis name. The master axis name is used by the part programmer or operator when commanding the dual axis group in part programs or for jog moves.

Each axis that makes up a dual axis group is controlled by a separate positioning command from the servo module. This dual group command is based on the move generated by the control when the master axis is commanded to a position. Each axis in the group must still have its own position and velocity feedback device which is monitored and used to correct any following error independently for each axis.

Positioning Dual Group Members Independently

If desired, certain axes in a dual group may be disabled (or **parked**). Parked axes are not positioned when a request is made to move the dual axis group. Parking an axis in a dual group is accomplished through the logic. Refer to your *9/PC Logic Reference Manual* for more information.

If it is necessary to position members of a dual axis individually without parking any axes in the group, the logic axis mover may be used. This feature ignores any link between axes in a dual group. Even axes that are parked may be positioned using the logic axis mover.

Per Axis versus Per Group

All axes that make up a dual axis reach end-point at the same time. This requires that all axes that make up a dual axis group share the same feedrate parameters, acc/dec ramps, and other axes specific data for the group. All axes in a dual axis group take on the characteristics of the worst case axis. For example, the slowest "maximum cutting feedrate" of one of the axes is used as the "maximum cutting feedrate" for all axes in the dual group. This is the case even if the worst case axis is parked at the time of the motion.

This same worst case characteristic remains when the dual group is decoupled. When decoupled the individual axes that made up the dual group are still bounded by the slowest axis:

- Acc/Dec Ramp
- Velocity Step
- Rapid Feedrate
- Max Cutting Feedrate

A warning message is generated at power up to indicate which axes have these above features limited.

If the dual group is to be the diameter axis (selected with G07 or G08 lathe controls only), the master axis must be configured as the diameter axis with the parameter **Diameter Axis Name** discussed in chapter 4. This will cause all axes in the dual group to be diameter axes even when decoupled. The same logic applies to the parameter that configures the **Default CSS Axis Name** discussed in chapter 13 and items such as **Drilling Axis** for cycles. Assigning a dual axis member that is not the master axis of the group to one of these parameters will have no affect on any axis in the dual group even after that slave axis is decoupled.

When you select the "Dual Axis Parameters" group, the workstation displays this screen:

```
Dual Axis Group (2) : 2101
------------------------------
independent axis        (a)
group 1                 (b)
group 2                 (c)
group 3                 (d)
group 4                 (e)
group 5                 (f)
```

Dual Axis Group

Function

This parameter is used to logically couple axes that should move when the group master axis name is programmed. The number of axes allowed in a dual group can range from a minimum of two to a maximum of nine.

Assign all axes that are to be commanded in dual group 1 as group 1 axes. Assign all axes that are to be commanded in dual group 2 as group 2 axes. You can assign axes in this manner for up to five dual axis groups. Select the axis to configure using the F2 function key. Any axis that is not to be commanded as part of a dual axis group should remain at the default value of this parameter, "independent axis". You can choose from the following parameter values.

- independent axis – Selects the axis being configured as an independent axis. It is not positioned as a part of any dual axis group. This is the default condition, and all axes that are not part of a dual axis group should be configured as independent axes.
- group 1 to 5 – This assigns the axis currently being configured as a member of dual group 1, 2, 3, 4, or 5, depending on your selection. Any positioning commands (that reference the dual group master) command this axis if it is not parked (refer to your *9/PC Logic Reference Manual*). Each group must be configured in order. For instance if you want to use three sets of dual axes, you have to configure groups 1, 2, and 3 in order, starting with group 1.

The first axis in your system that is configured in a dual group is that dual groups master axis. For example if you assign axis 3, 5, and 6 as members of the same dual group, axis 3 will be the group's master axis. You can not assign a two digit axis name as the master axis in a dual group ($B, $C, $X, $Y, $Z can not be the first member).

Axis	Parameter Number
1	[1101]
2	[2101]
3	[3101]
4	[4101]
5	[5101]
6	[6101]
7	[7101]
8	[8101]

Range

Selection	Result
(a)	independent axis
(b)	group 1
(c)	group 2
(d)	group 3
(e)	group 4
(f)	group 5

Notes

When defining integrand names for a dual axis, typically only the master axis should have an integrand defined. Other axes assigned as members of the dual group typically should have their integrand name configured as "none."

If you want to decouple a dual axis group in a dual processing system, and you want to perform planar functions that require and integrand word, then you must assign an integrand letter and plane for that axis.

Important: A group may have all linear or all rotary axes. A linear and a rotary axis may not be part of the same dual axis group.

Only the master axis name is valid when defining a plane in the **Plane Select** parameter group. No other dual axes group members can be used in a plane definition.

If axes are assigned as members of a dual axis group, the group must have a valid Dual Axis Master name configured. If no master axis name is configured, the control will not come out of E-Stop.

You should already have configured your axis types (linear or rotary) before attempting to set this parameter. If after you have assigned an axis to a dual group and the axis type for that axis is changed, this parameter automatically resets to the default condition of independent axis for that axis.

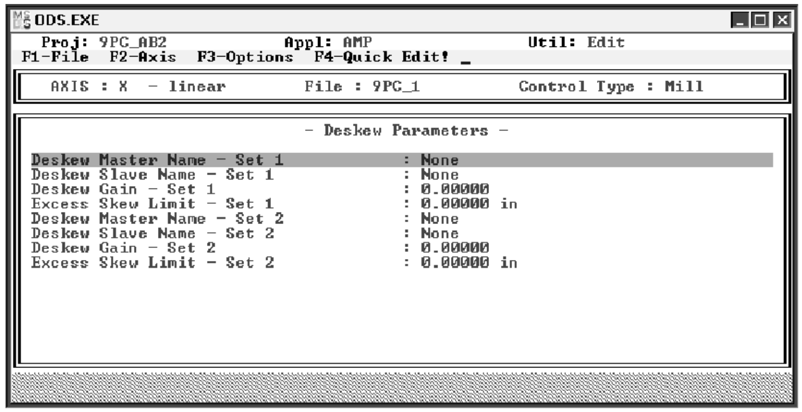
This parameter must be set independently for each axis.

END OF CHAPTER

Deskew Parameters for Split Axes

Chapter Overview

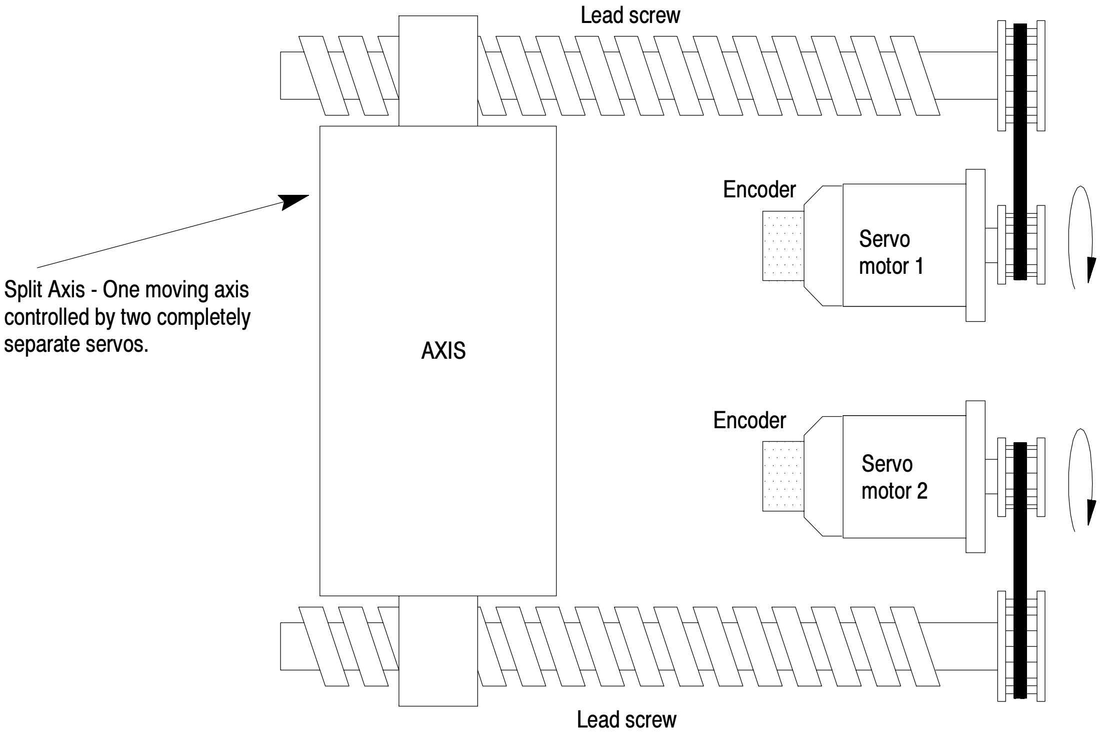
When you select the “Deskew Parameters” group, the workstation displays this screen:



Use this feature to control a single axis positioned by two servo motors.   This is called a split axis. One of the servos is configured to be a “master” while the other is selected as a “slave.” The servo module automatically maintains the position of the slaved servo by monitoring the difference between the following errors of the slave and master servos.

Your control supports the use of two split axis pairs in your 9/PC system.

Figure 32.1
Split Axis Configuration



Both servos that comprise a split axis must respond to the same part program and jog commands. When a split axis is configured using deskew, the two servos that make up the split axis are accelerated, positioned, decelerated, and stopped at the same time (except at E-Stop reset and when homing).

There are two cases in which the servos of a split axis do not receive the same motion commands:

- Homing

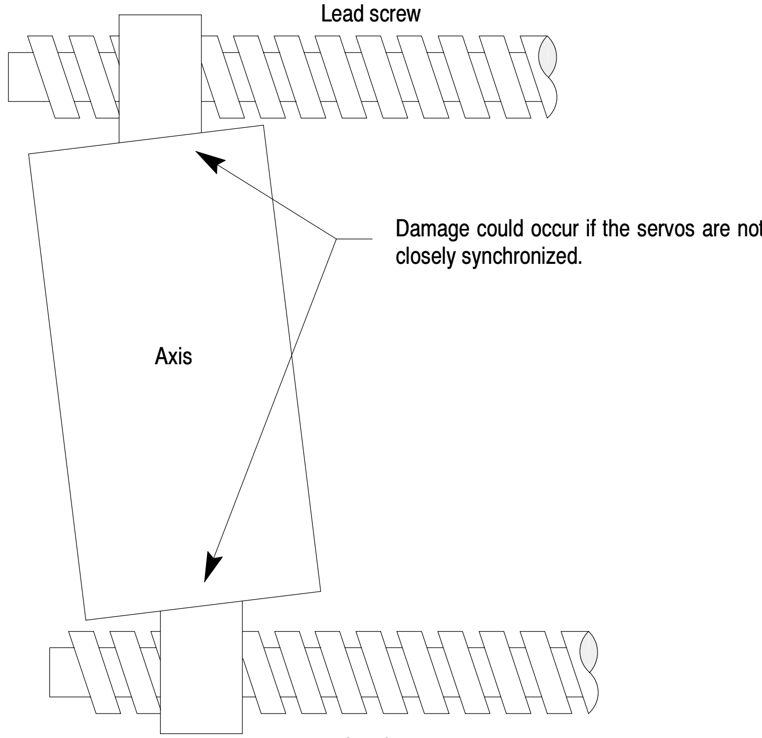
  When homing the split axis, a single home limit switch is used. The distance from that limit switch to the home position is dependent on the closest marker position and the value entered for **Home Calibration**. The difference between the two home calibration values and the direction and distance to the closest marker are independent for both axes. This means they can receive different positioning commands to reach the home position.

- E-Stop Reset

Important: The slave servo of the deskewed pair moves independently to compensate for any skew that may exist between the master and slave servo when E-Stop is reset. This eliminates any skew that may have occurred from servo drift during E-Stop or any skew that may have existed before entering E-Stop. This move occurs at the Medium/Low jog speed.

ATTENTION: The Deskew feature is used to control the skew when two servos drive one axis. Skew occurs when one servo's position is not the same as the other servo in the pair. If too great a skew occurs, damage may result to the axis and its drive components due to mechanical binding of the drive system.

Figure 32.2
Split Axis with Large Skew



ATTENTION: Whenever two servos are configured to drive one axis, you must use the deskew feature to control servo misalignment. Failure to consider possible servo misalignment may result in damage to drive components.

Each split axis can be driven by a maximum of two servo motors. The control supports two split axis pairs in a 9/PC system.

Aside from the parameters in this deskew group, you must configure the servos that drive the split axis as normal independent servos, including:

- separate feedback devices
- feedback directions
- loop type
- axis name

Each servo axis has its own port configured for command, position, and velocity. When configuring a split axis, keep the following in mind:

If	then the slave	and the master servo
One or more spindles are configured in AMP	must be AMP'd as the highest axis numbers in each process before spindles. Spindles must be the highest axis numbers used on a system.	can be configured as any axis before the slave axis.
No spindles are configured	must be AMP'd as the highest axis number used in the process	can be configured as any axis before the slave axis

Each servo in the split axis can be configured with independent:

- **Reversal Error** values
- **Axis Calibration** points
- **Home Calibration** values

Many parameters, such as Acc/Dec, feedrates, etc., are shared between the master servo and slave servo. These shared parameters are listed in this chapter. (**Deskew Master Servo Name).**

Deskew Master Servo Name

Function

This parameter selects the master servo and determines the axis name you program when commanding the split axis. Specifying this axis name in a part program causes the master servo selected here and the slave servo (selected later) to move the split axis to the specified position at the specified feedrate.

Many parameters that control axis motion and response are attained from the values configured for the master servo. The slave servo shares the values of these parameters with the master servo:

- All Homing Parameters (except **Home Calibration**)
- All Zones/Overtravel Parameters
- All Jog Parameters including jog speeds and jog increments
- All Feedrate parameters
- All Acc/Dec Parameters
- All Constant Surface Speed parameters
- All Cutter Comp/Tool Tip Radius parameters
- All Axis Program format Parameters
- All Tool Offset Parameters
- All Fixed Cycles parameters
- All Roughing Cycle Parameters
- All Cylindrical/Virtual C Parameters
- the master servo(s) status in regard to the PAL controlled features, including:
  - Servo Off
  - Servo Detach
  - Axis Inhibit
  - Axis Clamp

  These are all automatically copied from the master servo to the slave servo. The machine Logic cannot select different states between the master and slave servos for these features.

Any of the above parameters that have been configured for the slave servo are ignored and replaced with the master servo's values.

Independent AMP parameters must still be configured for the following:

- All Digital Servo Parameters
- All Analog Servo Parameters
- All Plane Select Parameters must not contain a slave servo name

Important: The master servo selected here <u>must be assigned as any axis that precedes the slave axis.</u> The deskew slave axis must always be the highest number axis in the process that precedes the spindle(s). For example, if configuring a three-axis system with no spindle and the X axis is driven by two servo motors, this configuration could be used:

Axis Number (servo)	Axis Name (name of servo)	Configuration
1	**X**	**Configured as Deskew Master Servo Name**
2	Y	Normal Servo Configuration
3	Z	Normal Servo Configuration
4	U	Configured as Deskew Slave Servo Name

The above configuration would result in servos 1 and 4 being part of a split axis. The name used for the split axis is X. Both the X and U servos would respond to any programmed or jog requests for the X axis. X and U would not be available for independent positioning. Common axis parameters AMPed for the U axis (such as acc/dec ramps, feedrates, etc.) would be ignored and replaced with the values entered for the master axis X (refer to the parameter **Deskew Master Servo Name** for a listing of slave servo parameters that are ignored).

Deskew	Axis	Parameter Number
		Single Process
Set 1	All	[660]
Set 2	All	[640]

Range

Selection	Result	Selection	Result
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

If you intend to use the Gain Break feature, both the **Gain Break Point** parameter and the **Position Loop Gain Break Ratio** parameter must be set to identical values for both master and slave servos.

The Software Overtravels and Programmable Zones for a slave servo must be configured as “not used.” All software overtravel and programmable zone information for a split axis are taken from the master servo.

A different **Feedrate Suppression Point** may be configured for both master and slave servo. If the following error of either exceeds its feedrate suppression point, the feedrate for both master and slave servo is reduced by 50%.

This parameter is a global parameter. The value set here applies to all axes.

Deskew Slave Servo Name

Function

Select a slave servo for your split axis with this parameter. Only one slave and one master servo can be configured to position a split axis.

Important: The slave servo selected here must always be the last AMP'd axis that precedes the spindle(s). For example, if configuring a three-axis system with one spindle and the X axis is driven by two servo motors, this configuration could be used:

Axis Number (servo)	Axis Name (name of servo)	Configuration
1	X	Configured as Deskew Master Servo Name
2	Y	Normal Servo Configuration
3	Z	Normal Servo Configuration
4	U	Configured as Deskew Slave Servo Name
5	C	Configured as Spindle

This configuration would result in servos 1 and 4 being part of a split axis. The name used for the split axis is X. Both the X and U servos would respond to any programmed or jog requests for the X axis. X and U would not be available for independent positioning. Common axis parameters AMP'd for the U axis (such as acc/dec ramps, feedrates, etc.) would be ignored and replaced with the values entered for the master axis X (refer to the parameter **Deskew Master Servo Name** for a listing of slave servo parameters that are ignored).

Deskew	Axis	Parameter Number
		Single Process
Set 1	All	[661]
Set 2	All	[641]

Range

Selection	Result	Selection	Result
(a)	A	(i)	Z
(b)	B	(j)	$B
(c)	C	(k)	$C
(d)	U	(l)	$X
(e)	V	(m)	$Y
(f)	W	(n)	$Z
(g)	X	(o)	none
(h)	Y		

Notes

When defining integrand names for a split axis, only the master servo should have an integrand defined. The slave axis selected with this parameter should have its Integrand Name configured as "none."

Important: Both linear or rotary axes may be configured as a split axes with deskew. However, all servos that make up a split axis must be configured as the same axis type. A linear and a rotary axis cannot be part of the same split axis.

Plane definitions cannot contain slave axes names. Only the master axis name is valid when defining a plane in the Plane Select group.

The Software Overtravels and Programmable Zones for a slave servo must be configured as "not used." All software overtravel and programmable zone information for both servos in a split axis pair is taken from the master servo.

The control's axis position displays, graphics screens, etc., do not contain any reference to the slave servo name. To the operator, only the master servo name is displayed representing the position of the split axis. However, for the purpose of integration and troubleshooting, the slave servo does appear in online AMP for the axis parameter features including Reversal Error, Home Calibration, Axis Calibration, and Servo Parameters. Refer to chapter 36 in this manual for details.

In addition to the above displays, the slave servo is also visible on the axis monitor feature discussed in the 9/PC Integration and Installation Manual, publication 8520-9.1. On this screen, in place of the Spindle DAC Command display, a Skew display is available for the slave servo. This display shows the difference in following error between the master and slave servos.

This parameter is a global parameter. The value set here applies to all axes.

Deskew Gain

Function

The servos of a split axis are always given the same positioning and contouring commands.

A well balanced split axis with equally sized motors operating on fairly symmetric loads may not need to use this parameter. The normal deskew velocity and positioning loops should adequately respond to keep any skew to a minimum.

In some cases, however, the normal servo loop may allow too much skew (motors are sized differently or one servo is under a higher load than another). When the normal deskew positioning and velocity loops are not sufficient to compensate for this difference in following error, the **Deskew Gain** parameter should be used.

When a skew exists, the control detects a difference in following error between the two servos. When using the **Deskew Gain** parameter, this difference in the following error is controlled by increasing or decreasing the gain of the slave servo. The net effect of this is that the control can make the slave servo's response more closely match that of the master servo.

The slave servo's gain is modified proportionally to the difference in following errors between the master and slave servos (= skew error). The ratio of this multiplier is controlled by this parameter. A value of 0 here disables any gain modification of the slave servo. The higher the value set here, the larger the gain modification for the same difference in following error.

Deskew	Axis	Parameter Number
		Single Process
Set 1	All	[662]
Set 2	All	[642]

Range

0.00000 to 10.00000

Notes

It is important that the slave servo be capable of matching the master servos response.

If:	And:	Then:
too great of a difference exists between the slave and master servo's response (i.e., greatly different loads or very different motors)	the slave servo's best possible response (under load) is slower than the master servo's response	the slave servo never matches the response of the master servo no matter how much the slave servo's gain is modified
	slave servo's response is faster (under load) than the master servo's response	the slave servo gain can be modified to slow down the slave servos response and make it match the master servo's response

When designing your split axis, try to match servo motors and motor loads as closely as possible. If this is not possible, make sure that the servo with the slowest response (under normal operating load) is configured as the master servo.

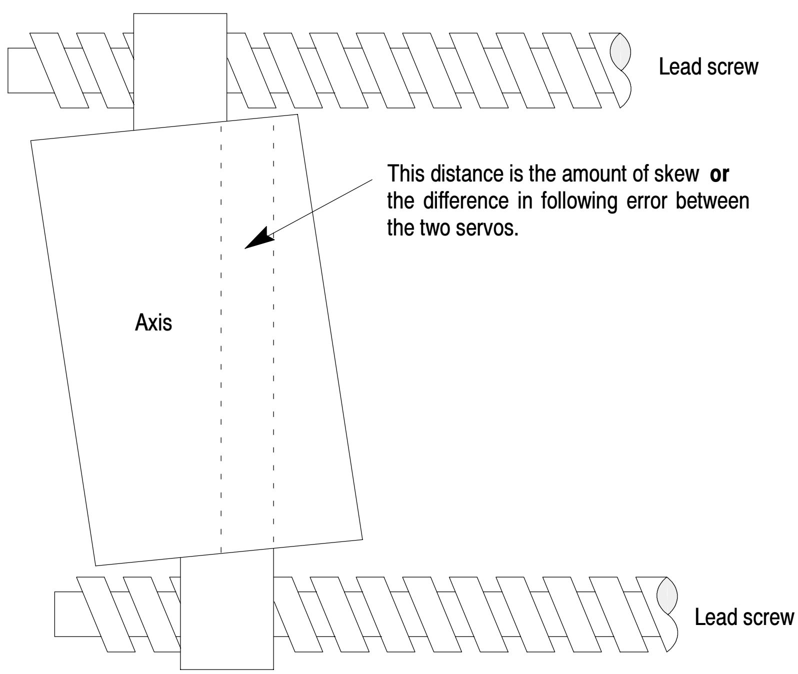
This parameter is a global parameter. The value set here applies to all axes.

Excess Skew Limit

Function

This parameter specifies the maximum difference in following error allowable between servos of a split axis. This distance is actually the maximum skew limit.

Figure 32.3
Split Axis Skew Distance



Enter for this parameter a value equal to the greatest amount of skew (difference in following error) that is acceptable between the servos of your split axis. When the difference in following error between the two servos exceeds the value set for this parameter, the control is forced into E-Stop.

ATTENTION: Make sure the value set here is small enough so that when a skew of the axis occurs, the control enters E-Stop before damage to the machine occurs.

Deskew	Axis	Parameter Number
		Single Process
Set 1	All	[663]
Set 2	All	[643]

Range

0.00000 to 214.00000 mm

or

0.00000 to 8.42520 in.

Notes

In the event that excess skew does occur, the control automatically cancels any skew amount when E-Stop is reset. When E-Stop is reset, the slave servo is moved to the same position as the master servo. Any difference in following error is canceled.

This parameter is a global parameter. The value set here applies to all axes.

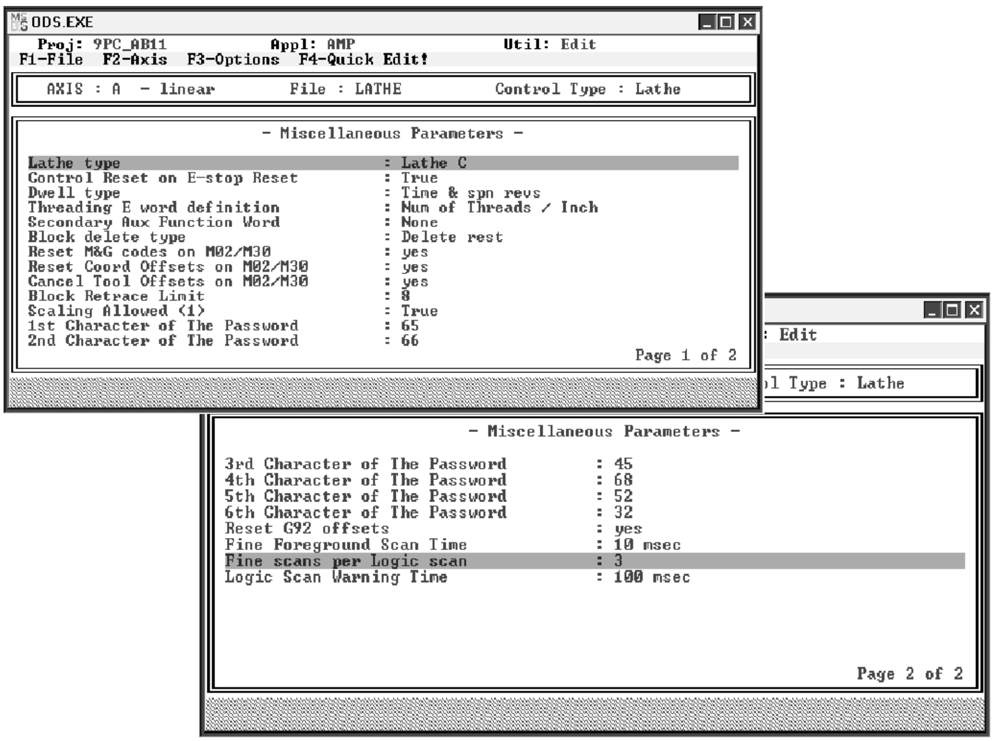
END OF CHAPTER

Miscellaneous Parameters

Chapter Overview

This chapter describes miscellaneous parameters that should be set. These miscellaneous parameters do not in any way refer to the miscellaneous functions that are called using an M- or B-code.

The workstation displays these screens when the **Miscellaneous Parameters** group is selected:



The above screens may differ slightly depending on the control type you select.

Important: When the selected control type is "Mill," the Lathe type parameter is replaced with the Mill type parameter.

General Control Operating Parameters

Use these parameters as general control operating parameters, which are used to configure very basic control operations, such as E-Stop Reset, probe trigger signals, and logic interaction with the 9/PC.

Lathe Type

Function

This parameter specifies the lathe type for which the control is configured. The selected lathe type determines the G-code system (A, B, or C) that the control uses. Refer to your *9/PC CNC Operation and Programming Manual* for additional information on lathe G-code systems.

Axis	Parameter Number
All	[518]

Range

Selection	Result
(a)	Lathe A
(b)	Lathe B
(c)	Lathe C

Notes

This is a global parameter; the value set here applies to all axes.

Lathe type A uses U, V, W as axis names for incremental axes. This reduces the number of axis names available on Lathe Type A so that the maximum number of axes allowed is 6 axes.

Mill Type

Function

This parameter is available when your [F3] option – Control Type has been selected as a mill. Use this parameter to indicate to the control that you are configuring a standard Mill or a transfer line control. This will enable transfer line specific features. The default for this parameter is a standard mill. This parameter should be set as a standard mill for the 9/PC.

Axis	Parameter Number
All	[517]

Range

Selection	Result
(a)	Standard
(b)	Transfer Line

Notes

This is a global parameter; the value set here applies to all axes.

Control Reset on E-Stop Reset

Function

This parameter forces the control to perform a control reset when the control is brought out of E-Stop by performing an E-Stop Reset operation. Typically, E-Stop reset is performed by pressing the **<E-STOP RESET>** key on the MTB panel.

A control reset performs these functions:

- all modal G- and M-codes are reset to the default condition (see the PTO parameters)
- all nonmodal G- and M-codes are reset to their normal condition at power-up
- all coordinate system offsets and rotations return to their normal condition at power-up
- any MDI command is discarded
- any active program is reset to the first block of the program
- any user's password that is active is deactivated, and it is necessary to re-enter the password

TRUE – Setting this parameter as true causes the control to execute a control reset when E-Stop is reset with the E-Stop reset feature.

FALSE – Setting this parameter as false will not cause the control to execute a control reset when E-Stop is reset with the E-Stop reset feature. A control reset may still be performed in the normal method.

Axis	Parameter Number
All	[68]

Range

Selection	Result
(a)	True
(b)	False

Notes

This is a global parameter; the value set here applies to all axes.

High-speed Input Trigger Point

Function

Use this parameter to determine when the control recognizes a high-speed trigger signal from some external device to execute one of the skip or probing functions. The control recognizes a high-speed trigger signal when the signal is turned on or off.

Rising Edge – When this parameter is set at "Rising Edge," the control recognizes a high-speed input signal when the probe turns on (rising edge of the signal).

Falling Edge – When this parameter is set at "Falling Edge," the control recognizes a high speed input signal when the probe turns off (falling edge of the signal).

Axis	Parameter Number
All	[3]

Range

Selection	Result
(a)	Rising edge
(b)	Falling edge

Notes

This is a global parameter; the value set here applies to all axes.

Dwell Type

Function

Use this parameter to force the control to execute only dwells (G04) in units of seconds.

Time & Spn Revs – When this parameter is selected as "Time & Spn Revs," a dwell may be programmed in second or in spindle revolutions. Dwells that are programmed in G94 mode are in units of seconds, and dwells that are programmed in G95 mode are in units of spindle revolutions.

Time – When this parameter is selected as "Time," a dwell is always programmed in units of seconds regardless of the G94 or G95 mode.

Axis	Parameter Number
All	[400]

Range

Selection	Result
(a)	Time only
(b)	Time & spindle revs

Notes

This is a global parameter; the value set here applies to all axes.

This parameter does not effect dwells that are executed in fixed cycles which can be either in units of seconds or spindle revolutions.

Threading E-word Definition

Function

This is a lathe or grinder control parameter. It determines the units that are used for the thread lead when programming an E- or F-word in a threading block. This parameter is set for all threading blocks.

Use this parameter with these G-codes:

	G-Code System		
	Lathe		
Description	A	B	C
Single Pass	G32	G33	G33
Variable Lead Single Pass	G34	G34	G34
Single Pass Cycle	G92	G78	G21
Multi-Pass Cycle	G76	G76	G78

Inch/Rev – When this parameter is set as Inch/Rev, the control reads an E- or F-word in a threading cycle block as the number of inches the axes travels per revolution of the spindle.

Num of Threads/Inch – When this parameter is set as Num of Threads/Inch, the control reads an E- or F-word in a threading cycle block as the number of threads to be cut for every inch of axis motion.

Axis	Parameter Number
All	[401]

Range

Selection	Result
(a)	Inch / Rev
(b)	Num of Threads / Inch

Notes

This is a global parameter; the value set here applies to all axes.

Secondary Auxiliary Function Word

Function

This parameter specifies the letter that is used to call the second auxiliary function which is commonly used when the number of M-codes is not sufficient for the available number of miscellaneous functions. The system installer determines what second auxiliary function words call what auxiliary functions in the logic program.

Axis	Parameter Number
All	[443]

Range

Selection	Result	Selection	Result
(a)	A	(f)	W
(b)	B	(g)	X
(c)	C	(h)	Y
(d)	U	(i)	Z
(e)	V	(j)	none

Notes

This is a global parameter; the value entered here applies to all axes.

Block Delete Type

Function

This parameter determines the operation of the block delete feature programmed in a block with a / code. There are nine block delete characters available ranging from /1 (the same as programming / only) to /9. Use this feature to let the operator delete either part or all of a program block at will by selecting a block delete number as active.

This parameter determines whether the entire block is deleted when the proper block delete switch is on, or if only the information in the block to the right of the block delete character.

Delete rest – When this parameter is set as delete rest, the control deletes any portion of the block that is to the right of the block delete character.

Delete whole – When this parameter is set as delete whole, the control deletes the entire program block regardless of the location of the block delete character in the program block.

Axis	Parameter Number
All	[403]

Range

Selection	Result
(a)	Delete rest
(b)	Delete whole

Notes

This is a global parameter; the value set here applies to all axes.

Reset M&G Codes on M02/M30

Function

This parameter determines how the control handles M- and G-codes when the control executes an end of program command (M02 or M30).

Yes – When this parameter is set as yes, the control sets all M- and G-codes back to their default values when the control executes an end of program command. Modal M- and G-codes default back to their power-up condition, and nonmodal G-codes are reset to their default values.

No – When this parameter is set as no, the condition of the M- and G-codes are retained when the control executes an end-of-program command. All modal M- and G-codes remain at their present value, and nonmodal M- and G-codes remain at their present value.

Axis	Parameter Number
All	[539]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes.

Reset Coord Offset on M02/M30

Function

Use this parameter to determine if the control resets to the default work coordinate system when an end-of-program (M02 or M30) block is read.

Yes – Setting a value of Yes for this parameter causes the control to cancel all coordinate system offsets that may be active except the "External Offset" (as if a G92.1 was executed). This also reactivates the work coordinate system that is set as the default work coordinate system using the parameter called **PTO Work Coordinate** discussed in chapter 13. When the end of program code is read by the control, the active coordinate system becomes either G54-G59 or none as set using the PTO Work Coordinate parameter plus any "External offset" value that has been entered on the control.

No – Setting a value of No for this parameter causes the control to keep the currently active work coordinate system and its active offsets active. M02 and M30 do not have any effect on the coordinate system. The next program that is executed adopts the coordinate system with its offsets.

Axis	Parameter Number
All	[84]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes.

Important: Setting a value of No for this parameter may cause undesired results if the program being executed contains a G52 coordinate offset. If this is the case, the coordinate system gets offset by the incremental amount in the G52 block each time the program gets executed. This problem does not occur when a G92 offset is programmed, since this offset absorbs any other active G92 offset. We recommend that if a G52 offset is used in the program, it be cancelled before the M02 or M30 program block using a G92.1 code.

Cancel Tool Offsets On M02/M30

Function

Use this parameter to determine if the control cancels the active tool length offsets when an M02 or M30 end of program block is executed.

Yes – Selecting a value of Yes for this parameter causes the control to cancel all tool length offsets that are active when an M02 or M30 block is executed.

No – Setting a value of No for this parameter causes the control to keep the currently active tool length offsets active even after an M02 or M30 code.

ATTENTION: If this parameter is set as Yes, tool length offsets are cancelled when an M02 or M30 is executed. At that point, any active programmable zones will restrict axis motion based on the position of the tool gauge point rather than the tool tip. This may cause an undesired machine “crash” with some fixture that would normally have been protected by a programmable zone.

Axis	Parameter Number
All	[85]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes.

This parameter has no effect on the cutter compensation and TTRC features. Cutter compensation and TTRC are always canceled when a M02 or M30 code is read by the control.

Block Retrace Limit

Function

Use this parameter to determine the maximum number of motion blocks that may be retraced using the block retrace feature. The block retrace feature is used to reverse execution of an active program. The control executes blocks in reverse order and reverse direction up to the maximum number of retractable blocks set with this parameter. Press the **<CYCLE START>** button to return the program to normal forward execution.

Important: There are restrictions as to the type of blocks that may be retraced. Refer to your *9/PC CNC Operation and Programming Manual* for more information.

Axis	Parameter Number		
	Single Process	Process 1	Process 2
All	[9]	[20009]	[21009]

Range

1 to 15 blocks (refer to Notes)

Notes

It is important to keep the value of block retrace limit as small as possible. The number of retrace blocks is directly affected by the number of look ahead blocks that the control has available for processing. The control will delete retrace blocks as needed to maintain its minimum number of look ahead blocks. Keeping this value small helps increase the control's efficiency by always leaving enough memory available for block look ahead. Refer to your *9/PC CNC Operation and Programming Manual* more information.

Important: Some system configurations will have less than 15 retrace blocks available to them. The exact number depends on your hardware configuration, the number of axes on the system, the options activated, and the revision of the system executive. If you configure more retrace blocks than are available at any given time on your system, the error message "A RETRACE BUFFER WAS DELETED" is displayed while a part program is executing. This error message indicates that the control has deleted one (or more) of the block retrace buffers and can no longer retrace as many part program blocks as you originally specified in AMP.

Possible solutions include:

- alter your part program by removing or combining consecutive nonmotion blocks that occur during cutter compensation, QPP, or Corner/Chamfer blocks.
- reduce the Cutter Compensation Maximum Non-Motion Block Limit in AMP (note this may change your compensation contour).
- lower the AMPed number of retrace blocks available.
- purchase more RAM.

Contact Allen-Bradley customer service for assistance on determining the proper solution for your system.

This is a global parameter; the value set here applies to all axes.

Scaling Allowed

Function

Use this parameter to specify whether scaling is allowed on a specified axis. When this parameter is set "TRUE" for an axis, scaling may be applied to that axis. When this parameter is set "FALSE " for an axis, scaling may not be applied to that axis. Use this parameter only if the scaling option is installed.

Refer to your *9/PC CNC Operation and Programming Manual* for more information.

Axis	Parameter Number
(1)	[1100]
(2)	[2100]
(3)	[3100]
(4)	[4100]
(5)	[5100]
(6)	[6100]
(7)	[7100]
(8)	[8100]

Range

Selection	Result
(a)	True
(b)	False

Notes

This parameter must be set independently for each axis.

Password Parameters

Individual user passwords are assigned on the control to protect different features of the control from an unauthorized operator. The password names and functions assigned to passwords are all determined on the control. Refer to your *9/PC CNC Operation and Programming Manual* for more information.

These AMP parameters determine the system's master password. This master password allows access to all control features, including patch AMP and other password levels. The master password should not be given the everyday-user since significant damage to the control may be done by inexperienced users.

Characters of the Password

Function

Use these AMP parameters to determine the system's master password that allows access to all of the control's features, including patch AMP and other password levels.

Enter the ASCII character value for each digit of the 6-digit master password. The default value of the master password is "AB-D4." Note that the sixth character for the default password is ASCII character 32. Character 32 is the ASCII space character; therefore, there's no need to enter a sixth character for the default password.

Parameter	Parameter Number
1st Character of The Password	[101]
2nd Character of The Password	[102]
3rd Character of The Password	[103]
4th Character of The Password	[104]
5th Character of The Password	[105]
6th Character of The Password	[106]

Range

32 to 95 ASCII character value

Reset G92 Offsets

Function

Use this parameter to determine if G92 and Set Zero offsets will clear when you perform a control reset. If you choose Yes, G92 and Set Zero Offsets will clear by a control reset, a program command, or by power up. If you choose No, G92 and Set Zero Offsets will clear only by a program command or by power up.

Axis	Parameter Number		
	Single Process	Process 1	Process 2
All	[91]	[20091]	[21091]

Range

Selection	Result
(a)	Yes
(b)	No

Notes

This is a global parameter; the value set here applies to all axes. On dual processing controls you must set a value for each processes independently.

System Timing

This section covers the information you need to be able to set the system timing parameters for the 9/PC. Three parameters affect the way that the CNC schedules time critical motion tasks and foreground machine logic execution:

- Fine Foreground Scan Time
- Fine Scans per Logic Scan
- Logic Scan Warning Time

In order to correctly set the values for these parameters for your system, overview information on system timing is discussed. Also covered in this section are the system timing displays provided with the 9/PC system. These displays allow you to view the timing of the operating system and provide a means to see the effect of changes you make to these parameters.

The major parts of this section cover these topics:

System Timing topic	Page:
System timing overview	33-14
Analyzing your system timing	33-18
Setting system timing parameters	33-20

System Timing Overview

This section covers the scheduling of software tasks by the 9/PC. This scheduling concerns: servo control; logic synchronization; part program block activation and block decode, as well as an overview of the scheduling that the host PC does for the machine logic program.

There are certain software tasks that must be executed at a very repeatable rate or frequency on a CNC control. These are collectively referred to as foreground tasks. In the 9/PC there are two foreground frequencies for two different sets of tasks.

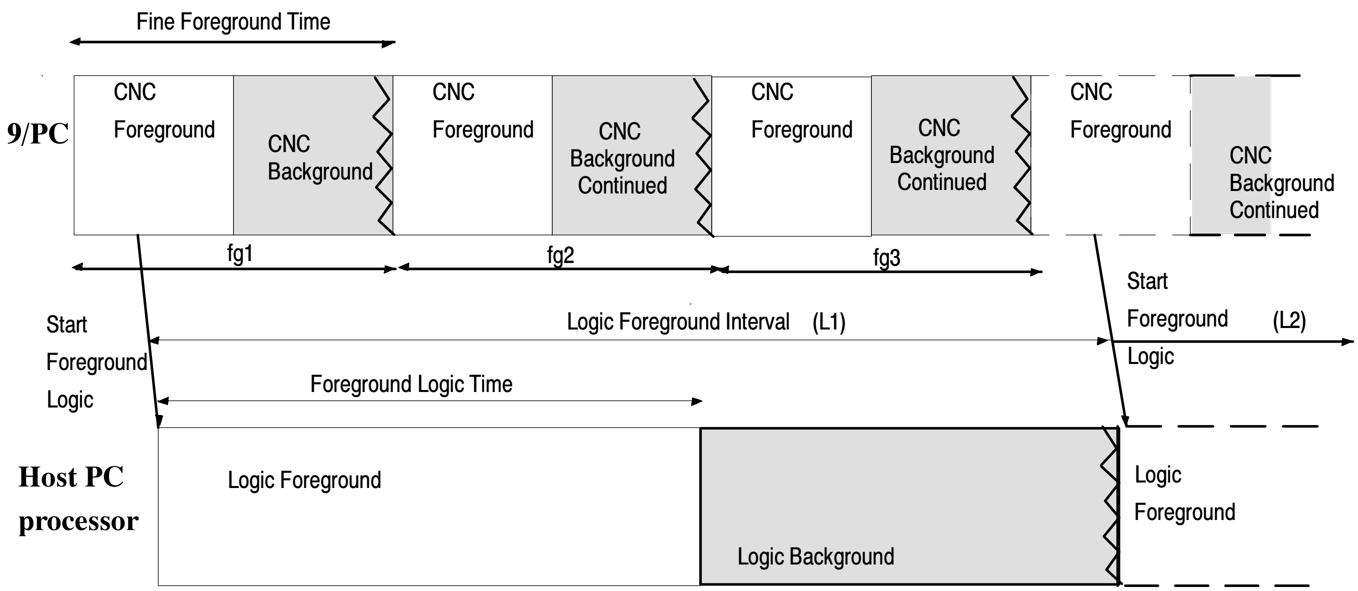
Fine foreground time: This frequency, and the associated tasks that are executed at this frequency make up the basis of the servo control of the system. The system will interrupt other lower priority tasks (system background tasks) at this time interval.

Logic foreground interval: This time interval is the time the foreground logic program is triggered to run. This event is triggered by the 9/PC at a frequency that you determine, by the setting of two AMP parameters: fine foreground time and fine scans per logic scan. To calculate the logic foreground interval, multiply the fine foreground time by the fine scans per logic scan. The 9/PC will call foreground logic to execute once for every specified number of fine scans. The foreground logic that executes on the host PC should also complete within this time interval for correct execution. For more information about the logic foreground interval parameter, refer to page 33-21.

The duration of the foreground logic that runs on the host PC is also shown on the following figure. This value is displayed on the system timing screens (discussed later in this chapter). This time is dependent on the amount of foreground logic you have written in your logic program.

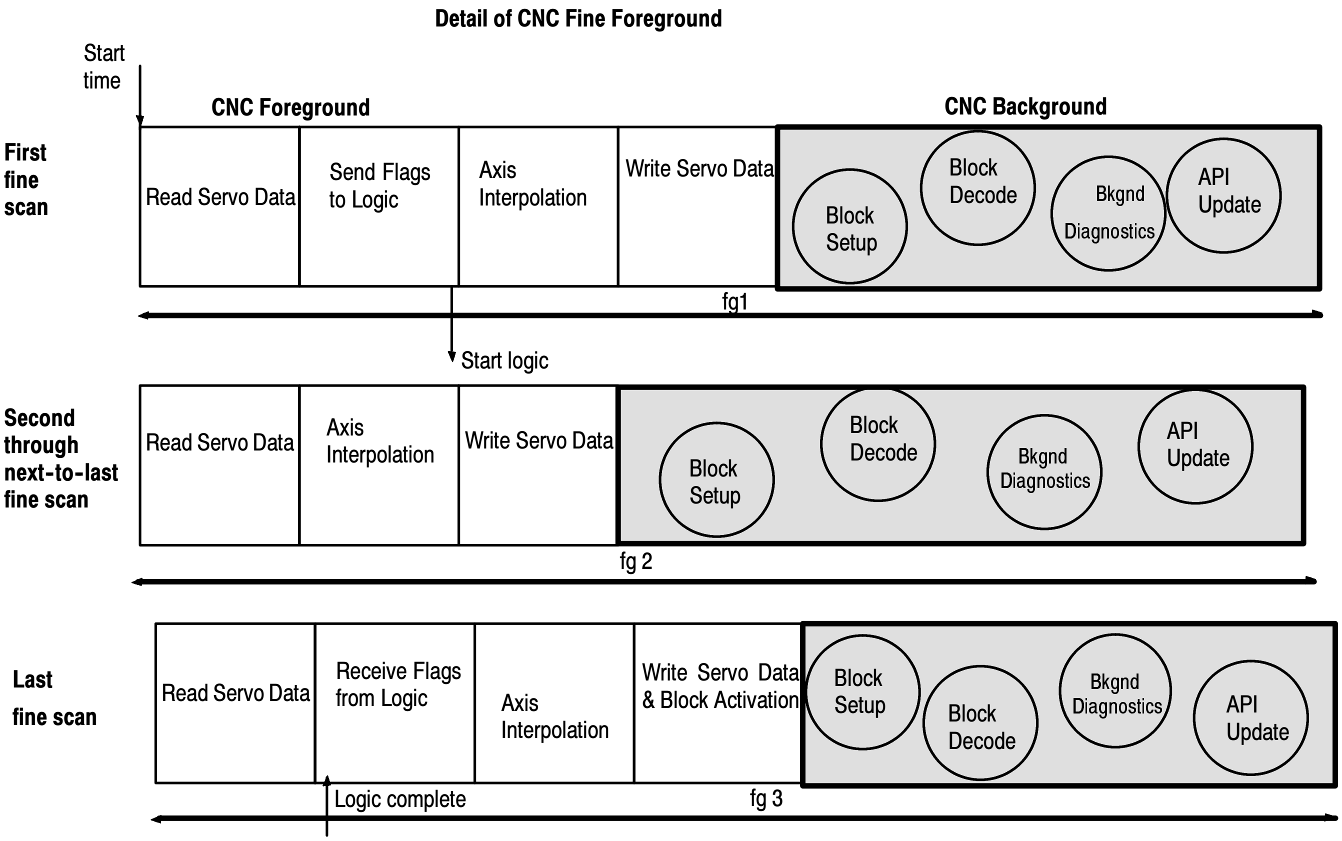
This is a graphical representation of the relationship between CNC **fine foreground time** and **logic foreground interval.**

9/PC to Logic Engine Timing
With fine scans per logic scan set to 3

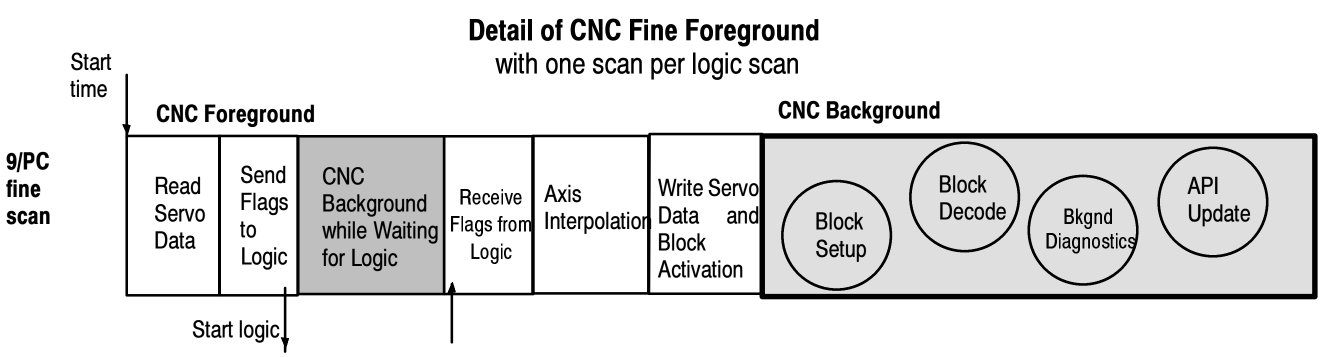


Within the CNC **fine foreground time** several tasks are executed. The order that these tasks are executed in can be important, as you analyze the system timing.

The specific tasks that make up each execution of the CNC fine foreground, can vary from scan to scan if you have selected more than one fine scan per logic scan. The following figure shows these specific tasks for a typical configuration of three fine scans per logic scan.



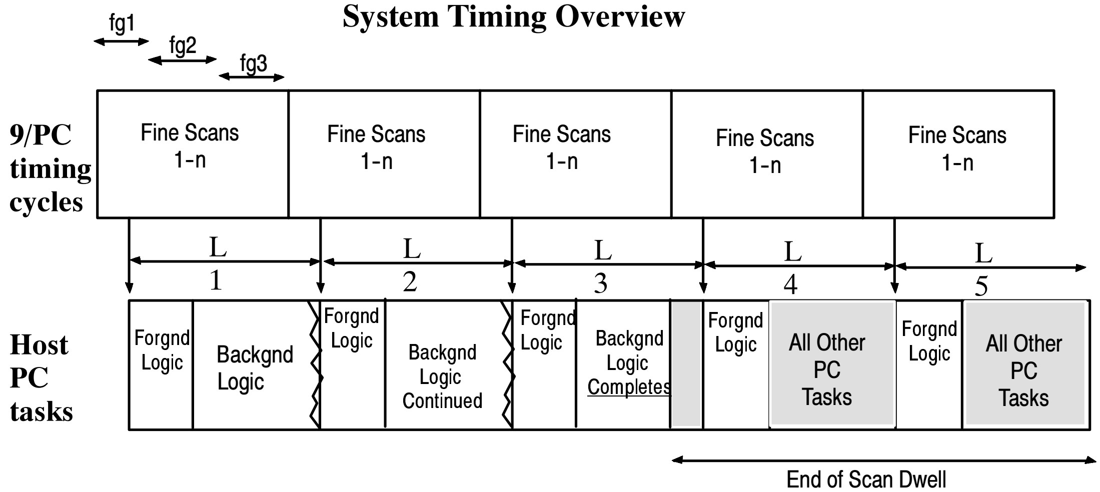
If you have set **one** fine scan per logic scan, this view of fine scan tasks changes somewhat. All of the CNC fine foreground tasks that are run during a logic scan must be executed in the single fine foreground time. The following figure shows the task execution for the setting of one fine foreground scan per logic scan.



One final area to discuss regarding system timing covers the overall picture of system timing. It is especially important to understand how the complete logic program is executed in the host PC's time line. This Host PC task is not actually controlled by any 9/PC AMP parameters, but it is important to understand this operation as you set the two 9/PC AMP timing parameters.

As discussed earlier, the 9/PC interrupts the many tasks that the host PC is performing and requests that the foreground logic be executed. This is done once for every configured number (n) of fine foreground scans.

System Timing Overview



Where fg = fine foreground scan time
and L = logic foreground interval

This figure shows that the background logic runs until completion, using all of the PC's available CPU time except for the foreground logic execution that is triggered by the 9/PC. Only the SoftLogix 5 operational parameter **End of Scan Dwell** will give the PC time to run the many other tasks that will need CPU time. Examples of these other tasks are: I/O scan update, CNC display update, communication with the 9/PC, operating system tasks, other PC-based software.

The setting of the "end of scan dwell" is discussed in more detail in the *9/PC CNC Logic Reference Manual.*

Logic Scan Warning Time: This parameter lets you specify the time (in ms) that must be exceeded before the control is forced into a Cycle Stop as a result of missing a reply from logic.

Important: If you are operating in asynchronous mode, the control is forced into a cycle stop in the event that a reply from logic is missed for a length of time (in ms) that exceeds the logic scan warning time. In this case, a "LOGIC OVERLAP WARNING" message displays.

If you are operating in synchronous mode, the control is forced into E-Stop if logic is missed on the first or last iteration (prelude and postlude) of the block, regardless of the setting of the logic scan warning time. In this case, a "LOGIC OVERLAP ERROR" message displays. If logic is missed on any other iteration for a length of time that exceeds the value of the logic scan warning time, the control is forced into a Cycle Stop.

Analyzing Your System Timing

This section describes the factors that effect your system timing during machine operation, and the CNC displays that are provided to allow you to see the real-time effect of the AMP system timing settings that you make.

Factors that Affect CNC System Timing

The reason to look at the system timing is to use the information to make adjustments to the two system timing AMP parameters. Although you can look at the system timing displays at any time, and under any system operating conditions, you can get the most meaningful information by operating the system under "worst case" conditions. This means the conditions that will have the biggest impact on the system timing, by using the greatest amount of processing time.

Here are the conditions that have an effect on the system timing information that is shown on the system timing displays.

Condition	Affects the duration (timing) of these tasks in the system
Your number of configured axes	Servo Read, Servo Write, Axis Control
Your number of logic foreground elements	Logic Foreground
Your maximum number of axes in motion	Interpolation (in order of importance)
Circular or helical type of interpolation (G02,G03)	
Cutter compensation	
Probing	
Homing	

Your maximum number of characters in the block	Part Program Decode
Paramacro expressions	
Qpp programing statements	
Corner rounding and champfering	

These conditions or factors should be varied in an analysis of the system timing to determine the conditions that have the greatest impact for your application.

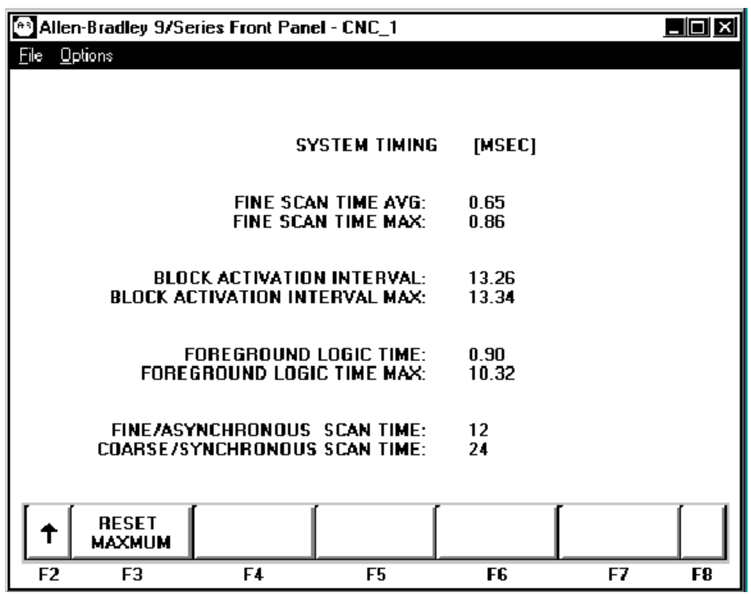
System Timing Screens

The CNC displays (BDS) include the following screens showing system timing information. This screen is accessed starting at the main, top softkeys level and making the following softkeys choices:

System Support → Right Arrow → System Timing

[F7] [F8] [F5]

A description of each value is discussed below.



- FINE SCAN TIME AVG.: The 9/PC fine foreground execution time averaged over the number of “fine scans per logic scan” specified in AMP.
- FINE SCAN TIME MAX: The maximum time of any single fine foreground scan since the maximums were reset. This value can be reset with the indicated softkeys. If this value exceeds the value permitted by the Logic Scan Warning Time parameter, the system will generate a “LOGIC OVERLAP WARNING” message.

- BLOCK ACTIVATION INTERVAL: This value updates whether part program blocks are running or not. It represents the time from the start of a foreground logic scan to the block activation period. This time represents the limit of the minimum block cycle time and is dependant upon the mode that you are running in (i.e., synchronous or asynchronous). This value must not be greater than the "coarse scan time" or data starvation will occur.

  In synchronous mode: if the value of fine scans per logic scan is greater than 1, you will find that this number is closer to the value in logic foreground interval.

  In asynchronous mode: if the value of fine scans per logic scan is greater than 1, you will find that this number should be closer to the value in fine scan time (Coarse/Synchronous Scan Time).

  Synchronous/asynchronous mode has no effect on the value of fine scans per logic scan if it is equal to 1 (Fine/Asysnchronous Scan Time).
- BLOCK ACTIVATION INTERVAL MAX: This is the highest value of block activation interval that has been recorded since the RESET MAXMUM key has been used to reset these values.
- FOREGROUND LOGIC TIME: This value is the actual time that the Host PC took to execute the foreground logic from the time the 9/PC initiated the start of that task. Also included in the time to transfer the SoftLogix 5 flags back to the 9/PC.
- FOREGROUND LOGIC TIME MAX: Again, this is the maximum value since all maximums have been reset with the reset softkey.
- FINE/ASYNCHRONOUS SCAN TIME: This is a display of the value you set in AMP of the fine foreground scan time.
- COARSE/SYNCHRONOUS SCAN TIME: This is a display of the logic foreground time the system is attempting to perform to, based on the values you set in AMP for fine foreground scan time, times the number of fine scans per logic scan.

Setting Your System Timing Parameters

This section identifies the specific parameters used and a brief strategy of how to determine these values.

Fine Foreground Scan Time

Parameter Number
626

Range

5 to 40 ms

Notes

To determine the minimum allowable fine foreground scan time, add 7 ms to 1 ms times the **maximum** number of axes on either drive. For example:

If drive 1 has 4 axes and drive 2 has 3 axes:
7 ms + (1 ms x 4) = 11 ms

As another example:

If drive 1 has 1 axis and drive 2 has 6 axes:
7 ms + (1 ms x 6) = 13 ms

Fine Scans per Logic Scan

Parameter Number
627

Range

1 to 8 fine scans per logic scan

Logic Scan Warning Time

Parameter Number
628

Range

5 to 200 ms

Notes

In the event that missed scan periods do not effect the performance of your system, you may set this value to greater than the logic foreground interval; however, whenever you miss a scan period, note that the message “LOGIC OVERLAP WARNING” displays. If this condition is not corrected within the amount of time specified by this parameter, your system also enters into Cycle Stop. In the event that your foreground logic execution time exceeds 300 ms (maximum), your system automatically enters into E-Stop and the control will have to be restarted.

When you restart your control, the messages “LOGIC OVERLAP ERROR” and “LOGIC NOT RESPONDING - CNC STOPPING” can be found in the 9/PC Error Log with a time stamp indicating when the errors occurred. Similar error messages may also be found in the Windows Event Log. For more information, refer to the Important note on page 33-18.

Procedure for Determining Values

The following steps provide an outline of the steps you should take to determine values for these three system timing parameters. In general you will initially be setting large values to give the system sufficient time to execute all of the tasks, then taking readings from the System Timing Screen that will guide you in setting final values for these parameters. **Example values are shown in bold and are dependant on the operational mode (i.e., asynchronous or synchronous)**.

1. Set the AMP parameters as follows:

- fine foreground scan time to a fairly large number (20-40 ms).

  Ex: 36 ms

- the number of fine scans per logic scan **must** be set to 1 for this procedure.

  Ex: 1 scans

- logic scan warning time to a number that is greater than or equal to the logic foreground interval (= fine foreground time x fine scans per logic scan). For more detailed information, refer to the **Notes** section.

  Ex: ≥ 36 ms

2. After resetting maximums, run a test part program with several linear and circular motion blocks with all of the machine axis programmed. You should also activate cutter compensation to provide complete information.

3. Next, record the values for the:

- Fine scan time maximum **Ex: 4.3 ms**
- Block activation interval maximum **Ex: 5.8 ms**
- Foreground logic time maximum **Ex: 2.8 ms**

The new AMP fine scan time that you will set will be equal to twice the “fine scan time maximum”. **Ex: 2 x 4.3 ms = 8.6 = 9 ms**

4. Next, determine the larger of the following two values:

- Sum of the “foreground logic time maximum” and the “Fine Scan Time Maximum” **Ex: 2.8 ms + 4.3 ms = 7.1 ms**

- Sum of the “Block activation interval max” and the “Fine Scan Time Maximum” **Ex: 5.8 ms + 4.3 ms = 10.1 ms**

5. The larger of these two values is then used to calculate the “fine scans per logic scan”. This is a simple calculation that takes the larger value from step 4, divided by the AMPed fine scan time determined in step 3.

The result is then rounded up to the nearest integer, and this is the “Fine Scans per Logic Scan” value to be entered in AMP.

Ex: 10.1/9 = 1.12 = 2 fine scans

6. Finally, set the Logic Scan Warning Time to a value greater than or equal to the Logic Foreground Interval (18 ms). You can calculate this value by multiplying the “fine foreground scan time” by the “fine scans per logic scan”. If you set this value greater than the recommended time, make sure that the value is based on your system’s ability to withstand missing data that may be obtained from the execution of the foreground logic. For more information, refer to the Important note on page 33-18.

END OF CHAPTER

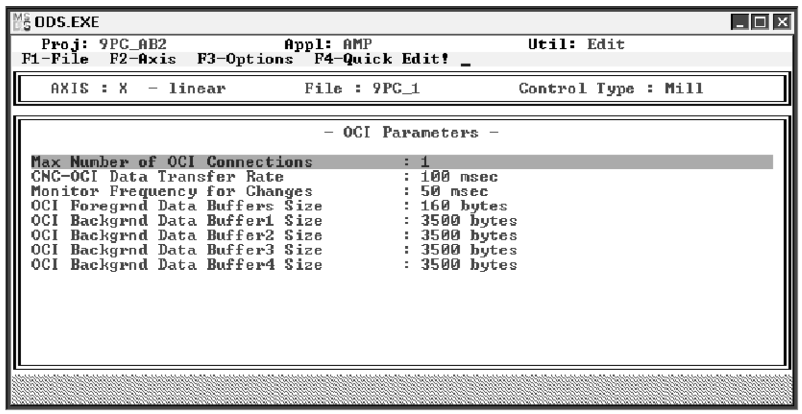
Chapter 34

Open CNC Interface (OCI) Parameters

Chapter Overview

This chapter describes the parameters used to configure your 9/PC for use with the Open CNC interface (OCI) API or Data Server. This interface allows access to the 9/PC resident data from any DDE/OPC-compliant application.

The workstation displays these screens when the “OCI Parameters” group is selected:



The above screens may differ slightly depending on the control you use.

Maximum Number of OCI Connections

Function

The use of this parameter will be significant at a subsequent release of the 9/PC. At this release, the 9/PC can support a connection to only one CNC OCI data server.

Axis	Parameter Number
All	[332]

Range

1 to 4 OCI stations

This is a global parameter; the value set here applies to all axes and processes.

CNC-OCI Data Transfer Rate

Function

Use this parameter to specify the frequency at which the 9/PC card passes updated information to the OCI data server. This parameter specifies the shortest amount of time that can pass between control updates to the OCI server. The actual time between updates may be greater than the minimum time configured here if higher priority tasks are being executed by the 9/PC processor or in the event that there is no new changed data to transmit. Data item values on the watchlist are only transmitted when its value changes.

Selecting a longer minimum duration will reduce traffic on your PC buss. Selecting a shorter minimum duration will increase OCI performance for time critical OCI applications.

Axis	Parameter Number
All	[333]

Range

20 to 3000 ms

Notes

This is a global parameter; the value set here applies to all axes.

We recommend keeping this parameter equal to or greater than the "Monitor Frequency for Changes" parameter. Following this recommendation will allow the system to buffer up changes and send larger packets of data each transmit cycle improving system and network performance.

Monitor Frequency for Changes

Function

Use this parameter to specify the frequency at which the control checks data for updated information. The application program running on the PC (for example the AB OCI Basic Display Set) requests data from the OCI data server as either automatic (updates are automatically sent from the CNC whenever a data item changes) or manual (updates are only obtained when the item is initially requested). Both automatic and manual items are placed on a watchlist at the control which checks this data for changes and sends updates to the PC when a change occurs (manual items are deleted from the watchlist after one data transfer). This parameter specifies the minimum amount of time that can pass before the control will check the items placed on its watchlist for a value change.

Selecting a longer minimum monitor frequency improves control performance and keeps 9/PC processor overhead low. Selecting a shorter minimum monitor frequency allows time critical data changes to be identified more rapidly to your OCI application.

Axis	Parameter Number
All	[334]

Range

20 to 3000 ms

Notes

This is a global parameter; the value set here applies to all axes.

The actual time between checks may be greater than the minimum time configured here because of some other higher priority task being executed by the 9/PC processor. Any time the actual monitor frequency is greater than the the value configured with this parameter for three scans in a row, the Logic flag BW_OCIEN indicates an error. For most applications this error is not common and can occur at different times during execution without causing any noticeable performance issues. In the event that many of these errors are recorded without an equal number of successful scans the control will display the error message "OCI WatchList Task Overlap" and you should raise this scan time.

Watchlist Buffer Size

The following parameters allow the configuration of the watchlist buffer sizes. One parameter is available to select the foreground watchlist buffer size for all connected OCI stations. Separate parameters are available for each background watchlist created by connected OCI stations.

Important: We recommend not changing the defaults on any of these watchlist buffer sizes. The default values for these watchlists are acceptable for most applications including the BDS.

At power up the control does a memory allocation. Space for two watchlists (one foreground and one background) is reserved for the OCI server. The more memory that is allocated for watchlist space, the less memory that is available for other CNC functions. The result is the control will begin to remove setup buffers for part program execution when the watchlist buffer size is allocated as too large. The available amount of memory on any given system depends on the processor, features enabled, and memory upgrade options you have purchased for the machine.

When the control has insufficient memory to create at least five setup buffers, the control is held in E-Stop. When this occurs only watchlist #1 is created and the error message “Not Enough Setup Buffers” is returned.

Change the size of these watchlists when your OCI application program requires a larger watchlist size or you suspect machine performance is suffering because the watchlist size is larger than necessary for your application. To help you identify how large your watchlist must be for your application use the following table in conjunction with appendix A of your OCI API developers reference manual. Appendix A of the API developers reference manual will help you identify the data types of items used in your application:

This Data Type	Watchlist Bytes Used:
Overhead per item	28 bytes/item
All Command Requests	0
BOOL	1
BYTE	1
SINT	1
USINT	1
INT	2
UINT	2
DINT	4
UDINT	4
LINT	8
STRNG	128
LREAL	8
DATE	8
TIMEOFDAY	8
LTIME	8
STRUCT	8
REAL	4

The overhead of 28 bytes/item applies to each data item in the watchlist.

$$watchlist = \sum_{(i=1)}^{(APIItems)} overhead_i + [(numarrayindices)_i \times (bytes/DataType)_i]$$

Any item placed on a watchlist remains on the watchlist until removed by your application or heart beat timeout occurs (refer to **JBoxDestroyInactiveParts** in your *9/Series OCI API Developer’s Guide*).

OCI Foregrnd Data Buffer Size

Function

Before changing this parameter make sure you have read and understand the watchlist concepts outlined on page 34-3.

Use this parameter to specify the size (in bytes) of the foreground watchlists for the 9/PC card. All foreground watchlists created are the same size as specified with this parameter. The foreground watchlist is typically very small as most applications do not use foreground watchlist items. Refer to your *9/Series OCI API Developer's Guide* for a listing of the data items that are foreground and their data types.

Axis	Parameter Number
All	[335]

Range

50 to 2000 bytes

Notes

This is a global parameter; the value set here applies to all axes.

Calculating the necessary foreground watchlist size is the same calculation as a background watchlist (see page 34-4)

OCI Backgrnd Data Buffer_Size

Function

Before changing this parameter make sure you have read and understand the watchlist concepts outlined on page 34-3.

There are four "OCI Backgrnd Buffer_Size" parameters. Parameters 2, 3, and 4 are not used at this time.

Axis	Parameter Number	Parameter Name
All	[336]	OCI Backgrnd Data Buffer1 Size
All	[337]	OCI Backgrnd Data Buffer2 Size
All	[338]	OCI Backgrnd Data Buffer3 Size
All	[339]	OCI Backgrnd Data Buffer4 Size

Range

50 to 10000 bytes

Notes

This is a global parameter; the value set here applies to all axes.

END OF CHAPTER

Chapter 35

Reserved Custom Parameters

Reserved Custom Parameters

Parameters found in this group of AMP parameters are for use with Allen-Bradley Custom Software that may have been ordered for your specific machine. It is not necessary to alter any of these parameters unless one of the custom software packages has been purchased.

If you have purchased a Custom Software Package, a detailed description of these parameters should have been provided in a separate document that accompanies this software. Refer to this document for details on any parameters in this group. If this document is not available, contact the system installer that you purchased the specific machine from or your Allen-Bradley sales representative.

END OF CHAPTER

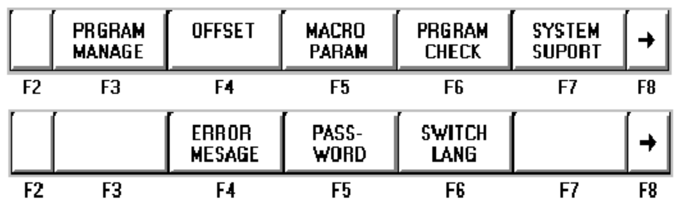
Tuning AMP at the Machine

Chapter Overview

This chapter covers the Adjustable Machine Parameters (AMP) that can be tuned at the machine. These parameters are tuned using the softkeys on the Basic Display Set human interface application. Use this procedure to access the online AMP parameter softkeys:

1. Press the **{SYSTEM SUPORT}** softkey.

(softkey level 1)

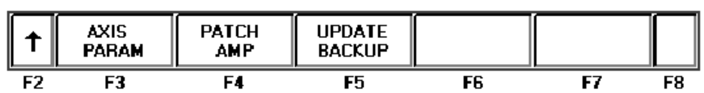


2. Press the **{AMP}** softkey.

(softkey level 2)

The control display the following softkeys:

(softkey level 3)



The functions of these softkeys are covered on these pages:

Softkey:	Page:
{AXIS PARAM}	36-2
{STORE BACKUP}	36-14

Important: Refer to chapter 37 for information on the functions of the **{PATCH AMP}** softkey.

Axis Parameters

One of the softkeys displayed after accessing the online AMP softkeys is the **{AXIS PARAM}** softkey. Pressing this softkey displays these softkeys:

Softkey:	Page:
{REVERS ERROR}	36-2
{HOME CALIB}	36-4
{AXIS CALIB}	36-6
{SERVO PARAM}	36-11
{SPNDL PARAM}	36-13

These softkeys and their functions are covered in the following sections.

Online Reversal Error Parameters

The online reversal error parameters are used to tune the reversal error compensation value of each axis. Reversal error compensation specifies the distance the control must add to the commanded axis motion to compensate for reversal error when the axis reverses direction. Refer to chapters 7 for additional information on reversal error compensation.

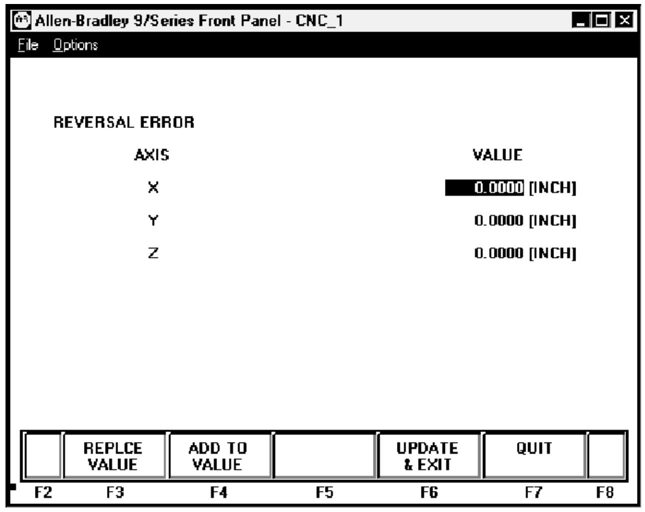
Important: You can not use Reversal Error Compensation on axes equiped with linear scales. Because the position loop is closed on the slide mounted scale, there is no lost motion (or reversal error) with this feedback configuration.

Use this procedure to access the online reversal error compensation parameters.

1. Place the control in E-Stop.
2. Press the **{REVERS ERROR}** softkey.

(softkey level 4)

When the **{REVERS ERROR}** softkey is pressed, this screen is displayed on the Basic Display Set application screens :



3. Use the up or down cursor keys to highlight the axis reversal error value that is to be changed.

   To replace the current reversal error compensation data with new data:

   - cursor to the value you want to change
   - press the **{REPLCE VALUE}** softkey
   - key-in the new data
   - press **[ENTER]**

   To add to the current reversal error compensation data:

   - cursor to the value you want to change
   - press the **{ADD TO VALUE}** softkey
   - key-in the new data
   - press the **[ENTER]** key

Important: If a parameter value that is out of range is entered, the error message "PARAM VALUE OUT OF RANGE" is displayed on row 0 (zero) of the display. If an invalid value, such as a float value (decimal number), is entered where an integer is expected, the error message "INVALID INPUT VALUE" is displayed in row 0 (zero).

4. Press the **{UPDATE & EXIT}** softkey to save the modified reversal error compensation data only on the 9/PC card. The control then returns to softkey level 4.

   Press the **[QUIT]** softkey to return to softkey level 4 without saving any changes.

(softkey level 4)

Home Calibration Parameters

The online home calibration parameters are used to tune the home calibration value of each axis. Home calibration specifies the distance from the encoder marker to the desired home position. The control moves the home calibration distance plus the distance to the encoder marker after the home limit switch changes from “ON” to “OFF”. Refer to chapter 5 for additional information on home calibration.

Use this procedure to access the online home calibration parameters.

1. Place the control in E-Stop.
2. Press the **{AXIS PARAM}** softkey.

(softkey level 3)

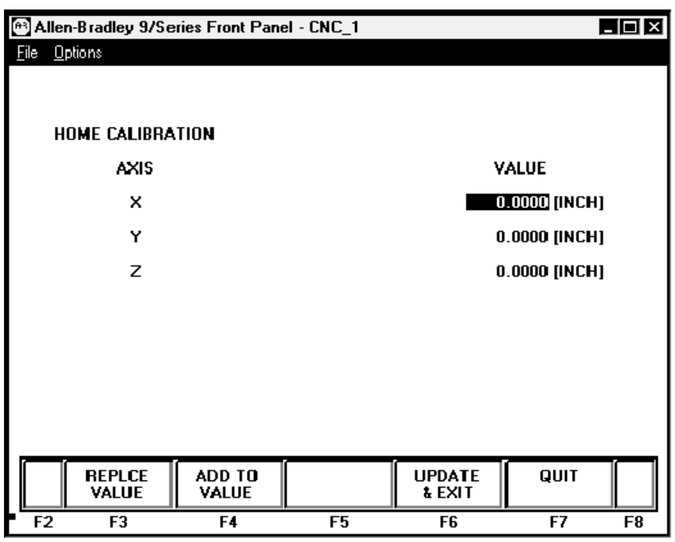
↑	AXIS PARAM	PATCH AMP	UPDATE BACKUP			
F2	F3	F4	F5	F6	F7	F8

3. Press the **{HOME CALIB}** softkey.

(softkey level 4)

↑	REVERS ERROR	HOME CALIB	AXIS CALIB	SERVO PARAM	SPNDL PARAM	→
F2	F3	F4	F5	F6	F7	F8

When the **{HOME CALIB}** softkey is pressed, this screen is displayed on the operator panel CRT:



4. Use the up or down cursor keys to highlight the home calibration value that is to be changed.

   To replace the current home calibration data with new data:

   - press the **{REPLCE VALUE}** softkey
   - cursor to the position you want
   - key-in the new data
   - press **[ENTER]**

To add to the current home calibration data:

- press the **{ADD TO VALUE}** softkey
- cursor to the position you want
- key-in the new data
- press **[ENTER]**

Important: If a parameter value that is out of range is entered, the error message “PARAM VALUE OUT OF RANGE” appears in row 0 (zero) of the screen. If an invalid value such as a float value (decimal number)is entered where an integer is expected, the error message “INVALID INPUT VALUE” appears in row 0 (zero).

5. Press the **{UPDATE & EXIT}** softkey to save the modified home calibration data only on the 9/PC card. The control then returns to softkey level 4.

Press the **{QUIT}** softkey to return to softkey level 4 without saving any changes.

(softkey level 4)

Online Axis Calibration Parameters

The online axis calibration parameters specify the commanded position and the actual position or the difference between the commanded position and the actual position of a machine axis.

Once any axis calibration parameter value is altered, the axis is marked as unhomed. Homing operations are disabled while the axis values
are updated.

Modified Axis Calibration Parameter values for an axis become active once the axis has been homed (assuming the axis has this feature).

If no axis data has been entered, the control displays the axis calibration screen shown in Figure 36.1. If axis data has previously been entered for an axis, the control displays the axis calibration screen shown in Figure 36.2.

Important: To avoid the accidental loss of your axis calibration data you should backup the axis calibration points to the host PC’s hard disk (page 36-14). You can then restore these calibration points whenever necessary.

Accessing Axis Calibration Parameters

Use this procedure to access the axis calibration parameters:

1. Press the **{AXIS PARAM}** softkey.

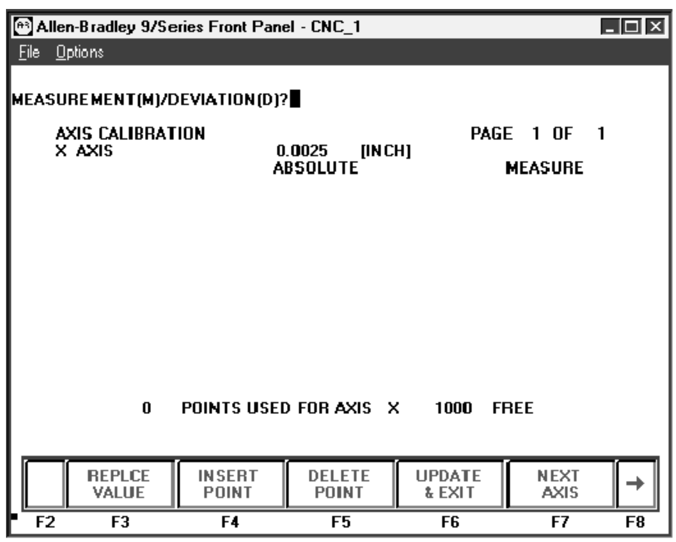
(softkey level 3)

2. Press the **{AXIS CALIB}** softkey.

(softkey level 4)

Once the **{AXIS CALIB}** softkey is pressed, the control displays the following screens. The first screen only appears when no data has been previously entered.

Figure 36.1
Initial Online Axis Calibration Screen



Initially the control prompts you to select whether inputs should be handled as measurements or deviations. If measurement is selected, the user input is the actual reading. If deviation is selected, the user input is the difference between the control reading and the
actual reading.

3. To specify that the input data is the actual reading, type "M" and press the **[ENTER]** key or just press the **[ENTER]** key at the prompt:

"MEASUREMENT(M)/DEVIATION(D)?".

For Dual Processing controls, the status bar indicates which process is being displayed. Press the **[PROC SELECT]** key to change the active process. You can only access the axes in the currently selected process.

4. To specify that the input data is the difference, type "D" and press the **[ENTER]** key at the prompt. The message on row 5 of the Axis Calibration screen, "MEASURE," changes to "DEVIATION."

   The control prompts the user to select whether measurement starts at the most negative or positive position on the axis. To start at the most positive position on the axis, type "+" and press the **[ENTER]** key at the prompt: "STARTS AT THE MOST +/-?" or just press the **[ENTER]** key. To start at the most negative position on the axis, type "-" and press the **[ENTER]** key at the prompt.

Important: The input data type and the start measurement point are asked only when there is no data for the axis. If the user wants to change either of the two, all the data for the axis must be deleted by deleting all measurement points with the **{DELETE POINT}** softkey.

Entering Axis Calibration Parameters

Use the following steps to enter the Axis Calibration data:

1. Move the axis to the most negative position on the axis if this position has been selected as the start point. Otherwise, move the axis to the most positive position on that axis.
2. Press the **{INSERT POINT}** softkey. The control reading appears on row 7 in the ABSOLUTE column.

   If the difference option has been selected, 0 is automatically displayed in the DEVIATION column for the first point entered for the axis.

   If the difference was not selected, the value displayed for the control reading (ABSOLUTE column) and the actual reading (MEASURE column) are the same.
3. Move the axis to the next position and press the **{INSERT POINT}** softkey.

Important: The maximum interval between consecutive axis calibration points is 84 inches (2133.6 mm).

Important: The maximum calibration value that you should enter can not exceed 2% (two percent) of the interval distance. Exceeding this limit can result in small, but unexpected motion of the axis.

If the first point is the most negative position on the axis, the control reading is displayed on the following line in the ABSOLUTE column.

If the first point inserted was not the most negative position, the first point is shifted down by one line and the control reading of the current position will be displayed on the first row in the ABSOLUTE column.

4. Press the **{REPLCE VALUE}** key and enter the difference between the control reading and the actual axis position, then press the **[ENTER]** key. The difference cannot be greater than 2% of the distance between this point and the previous one. If it is the message “PARAMETER OUT OF RANGE” appears.

Important: Put a negative sign on the difference if the actual axis position is on the negative side of the absolute axis position.

5. Enter the actual reading using the operator panel keyboard, then press the **[ENTER]** key. The actual reading is displayed in the MEASURE column.

6. Repeat steps 3 and 4 until the entire axis has been calibrated.

7. To store data, press the **{UPDATE & EXIT}** softkey.

   After the entire axis has been calibrated, assuming the difference and most negative position have been selected for the input data type and the start point, the display will look like the Axis Calibration Screen shown in Figure 36.2.

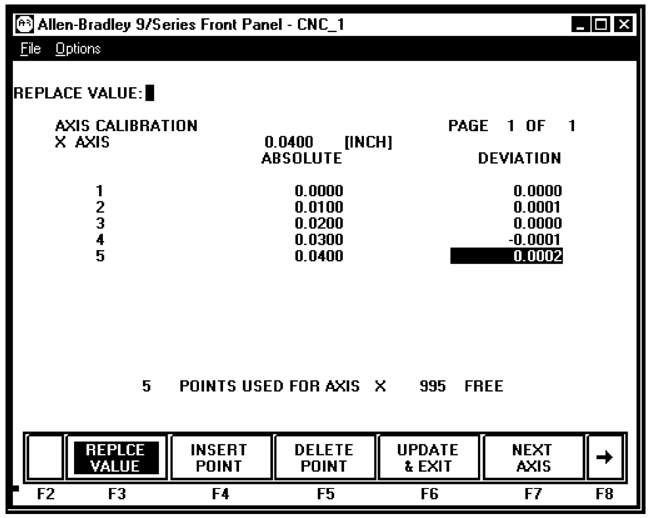
Important: If the displayed axis was a roll-over axis, which would be indicated by the “ROLL OVER” display to the right side of the axis name on row 3, the net amount of correction values for the axis must be zero for one complete revolution. If a control operator tries to leave the screen of a roll-over axis by pressing the **{UPDATE & EXIT}** softkey, or the **{NEXT AXIS}** softkey when the net amount for the axis is not equal to zero, the error message: “NET CORRECTION IS NOT ZERO” is displayed in row 0.

8. Press the **{EXEC}** softkey to leave the screen with the error, or press the **{CANCEL}** or the {↑} softkey to stay in the screen.

Important: If the rotary axis does not have a net amount of zero calibration, the control will still allow this calibration to be applied to the axis.

If data has been previously entered, the control displays a screen that list the axis calibration points. This screen is a typical axis calibration screen.

Figure 36.2
Typical Online Axis Calibration Screen



Important: An axis may have more than one screen of measurement points assigned to it. If there is more than one page, pressing the down cursor while holding the **[SHIFT]** key down displays the second page.

Changing Axis Calibration Data

Use this procedure to change the axis calibration data shown in Figure 36.2.

1. When the Axis Calibration screen is opened, data of the first axis of the control is displayed. Data of other axes can be displayed by pressing the **{NEXT AXIS}** softkey.
2. Either a new measurement point can be entered or an existing measurement point can be replaced for the displayed axis:

Inserting Axis Calibration Points

To insert a measurement point:

1. Use the cursor keys to select the position where a measurement point is to be inserted, then press the **{INSERT POINT}** softkey (see Figure 36.2).

Important: The number of measurement points to be assigned to any one axis may vary as long as the total number for all axes does not exceed 1001. If the operator tries to insert the 1002nd measurement point to the table, the error message "MEASUREMENT POINT OVERFLOW" is displayed in row 0 of the screen.

2. To replace stored travel data with new data, move the cursor keys to the data to be changed, then key-in the new data and press the **{REPLCE VALUE}** softkey.

Deleting Axis Calibration Points

The following procedure can be used to delete a current measurement point from the displayed table:

1. Use the cursor keys to highlight to the measurement point to be deleted.
2. Press the **{DELETE POINT}** softkey. The control deletes the highlighted point, then shifts all the points that were below the deleted point up one line.

To store data, press the **{UPDATE & EXIT}** softkey.

Calibration Status On/Off

Axis calibration is turned on whenever an axis is homed. When the axis calibration is on for an axis the **{CAL STATUS}** softkey is shown in reverse video.

It is often helpful to turn axis calibration off when inserting a calibration point. With calibration off, you can more accurately determine the axis position without accounting for previously entered calibration points. Press the **{CAL STATUS}** softkey to turn axis calibration off for the selected axis. Once turned off, you must home the axis again to turn axis calibration back on.

(softkey level 4)

F2	F3	F4	F5	F6	F7	F8
	REPLCE VALUE	INSERT POINT	DELETE POINT	UPDATE & EXIT	NEXT AXIS	→

F2	F3	F4	F5	F6	F7	F8
	CAL STATUS	DELETE ALL				→

Online Servo Parameters

The online servo parameters are used to tune specific servo parameters. Refer to chapter 7 for additional information on these parameters.

Important: This capability is available only on 1394 CNC Serial drives. The parameters for any other drive must be set with the Drive manufacturers configuration tool.

Use this procedure to access the online servo parameters:

1. Press the **{AXIS PARAM}** softkey.

(softkey level 3)

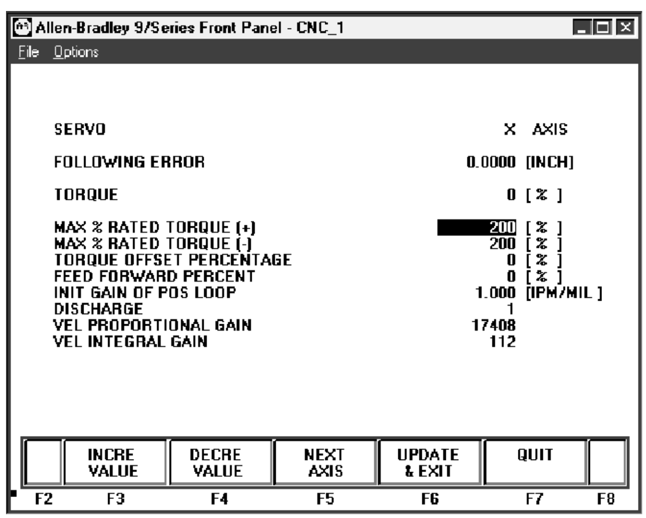
F2	F3	F4	F5	F6	F7	F8
↑	AXIS PARAM	PATCH AMP	UPDATE BACKUP			

2. Press the **{SERVO PARAM}** softkey.

(softkey level 4)

F2	F3	F4	F5	F6	F7	F8
↑	REVERS ERROR	HOME CALIB	AXIS CALIB	SERVO PARAM	SPNDL PARAM	→

The control displays this screen when the **{SERVO PARAM}** softkey is pressed:



When this screen is selected, the block cursor is placed on the **MAX % RATED TORQUE (+)** line (row 7).

Important: Although the value of the following error is displayed on row 5 for every axis, it cannot be altered; it is displayed only for user reference.

3. Use the up or down cursor keys to highlight the servo parameter that is to be changed.

   To increase or decrease the value of the highlighted servo parameter, press the **{INCRE VALUE}** or the **{DECRE VALUE}** softkey respectively. Hold the key down to change the value faster.

Important: If a servo parameter value reaches the maximum or minimum range value of the parameter being changed, the value stops increasing or decreasing.

4. To modify values for another axis, press the **{NEXT AXIS}** softkey. The softkey will toggle through all the axes.

5. Press the **{UPDATE & EXIT}** softkey to save the modified servo parameter values only on the 9/PC card. The control then returns to softkey level 4.

   If you have the parameter **Standard Motor Table Values** set to yes, and you edit either the Velocity Proportional Gain or Velocity Integrator Gain, pressing the **{UPDATE & EXIT}** softkey will save your edits and change the value of **Standard Motor Table Values** to no.

   Press the **{QUIT}** softkey to return to softkey level 4 without saving any changes.

Spindle Parameters

The online spindle parameters are used to tune specific spindle parameters. These parameters specify the gear range, the maximum voltage, and the servo outputs to the spindle motor. These values must be within the range of –10.000 through 10.000 (volts). Refer to chapter 9 for additional information on these parameters.

Use this procedure to access the online spindle parameters:

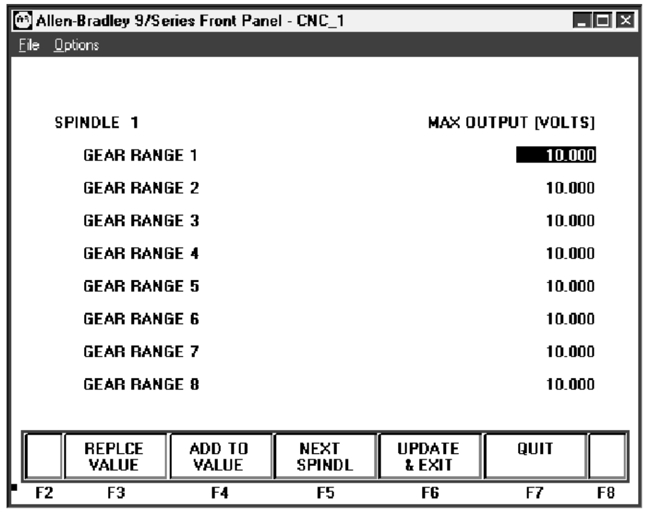
1. Press the **{AXIS PARAM}** softkey.

(softkey level 4)

2. Press the **{SPNDL PARAM}** softkey.

(softkey level 4)

When the **{SPNDL PARAM}** softkey is pressed, this screen is displayed on the operator panel CRT:



When the control displays this screen, the cursor will be shown on the value GEAR RANGE 1 in the MAX OUTPUT column.

3. Use the up or down cursor keys to highlight the spindle parameter value that is to be changed.

   To replace the current spindle parameter value:

   - press the **{REPLCE VALUE}** softkey
   - cursor to the position you want
   - key-in the new value
   - press **[ENTER]**

   To add to the current spindle parameter value:

   - press the **{ADD TO VALUE}** softkey
   - cursor to the position you want
   - key-in the new value
   - press **[ENTER]**

4. When you have configured more than one spindle, the **{NEXT SPINDL}** prompt highlights. Use this softkey to toggle through the various spindles you have configured.

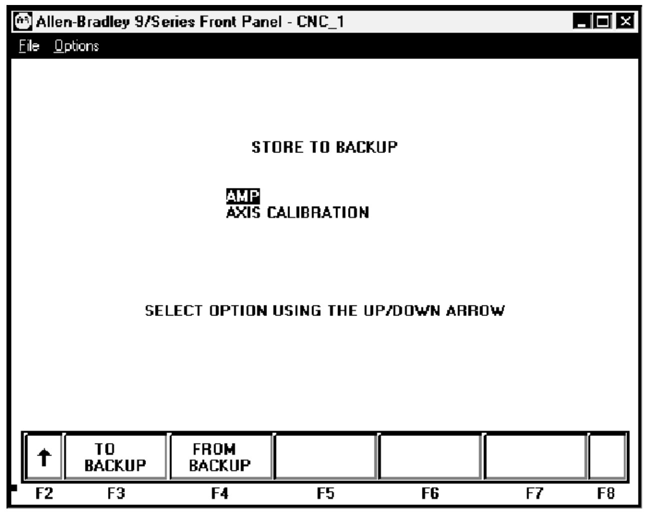
5. Press the **{UPDATE & EXIT}** softkey to save the modified spindle parameter values on the 9/PC card. The control then returns to softkey level 4.

   Press the **{QUIT}** softkey to return to softkey level 4 without saving any changes.

Backup Data to/from Hard Disk

One of the softkeys displayed after accessing the online AMP softkeys is the **{UPDATE BACKUP}** softkey. This softkey is used to backup the copy of AMP or axis calibration data that is on the 9/PC card to or from the host PC's hard disk.

The control displays this screen when you press the **{UPDATE BACKUP}** softkey:



Use the up or down cursor keys to highlight the type of information that you need to back up to or from the PC's hard disk. Only one of these sets of data can be backed up at a time.

To backup information TO the host PC hard disk from the 9/PC 's memory, press the **{TO BACKUP}** softkey. The selected information, AMP or axis calibration data, is copied from the control's memory to a file on the hard disk.

To backup information FROM the PC's hard disk the to the control's memory, press the **{FROM BACKUP}** softkey. The selected information, AMP or axis calibration data, is copied to the 9/PC card.

The control displays the message "PLEASE RESTART the 9/PC". The control must be turned off and then on again after the Backup information is stored to the 9PC.

END OF CHAPTER

Patch AMP

Patch AMP

Use of the Patch AMP utility requires access to the highest user level password, plus a special second password.

ATTENTION: Altering AMP through the Patch AMP utility bypasses the system software that checks AMP parameters for their range of values. You can enter illegal and/or dangerous values that will not generate an error. Illegal or dangerous values can cause undesirable changes to a machine responding to motion commands. This can result in damage to the part or the machine, or injury to an operator.

Accessing Patch AMP

The Patch AMP utility is softkey selectable; but due to the danger associated with changing AMP data without any cross-reference checking, the utility is "double" password protected. Only the highest level user will see the **{PATCH AMP}** softkey, and another password must be entered before that softkey will function.

Important: AMP is typically entered by the system installer and should not require changes once the control is installed and the machine is running. Most users should not be permitted access to AMP parameters.

The Patch AMP utility is used to modify the Adjustable Machine Parameters (AMP) directly on the control. This gives the machine tool builder the flexibility to change AMP without installing or running ODS.

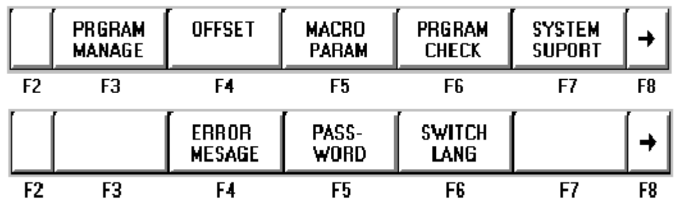
When AMP is modified using this utility, no cross-reference checking is performed. This means that almost any value can be assigned to any available parameter regardless of the consequences.

Patch AMP contains a limited number of the total available AMP parameters and is not intended as a substitute for ODS. Its intent is to allow minor changes to be made to an existing AMP that was previously downloaded to the control from ODS.

Assuming that the control is powered up and the highest access password has been entered, use the following procedure to get to the **{PATCH AMP}** softkey:

1. Press the **{SYSTEM SUPORT}** softkey.

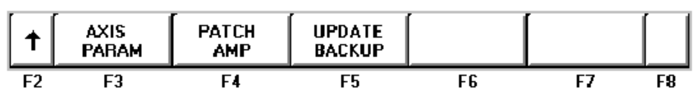
```
(softkey level 1)
```



2. Press the **{AMP}** softkey.

```
(softkey level 2)
```

3. Press the **{PATCH AMP}** softkey. The control will prompt you for an additional password to access the patch AMP utility.



Important: If the **{PATCH AMP}** softkey is not visible on this softkey level, the currently active access level does not permit the use of this utility. Return to softkey level 1, press the **{PASSWORD}** softkey, and enter the highest level password (access level 0). This password is assigned using the AMP parameters discussed in chapter 33 for the first - sixth character of the password.

4. The patch AMP password is the word **LOCK**. Type in **LOCK** using the keyboard and press the **[ENTER]** key. The control will display the patch AMP utility screen.

   The AMP parameter QuickEdit number is in the left column. The value of that parameter is shown in the right column of the screen.

Patch AMP Search Functions

Use the search utility to quickly find the AMP parameter you need to edit. The following procedure outlines how to use the search utility:

1. Press the **{SEARCH NUMBER}** softkey.
2. Using the keyboard, type in the parameter number to search for and press the **[ENTER]** key. The control will display the selected parameter in reverse video.

If you enter a parameter number that can not be edited using patch AMP, the control displays the error message "PARAMETER NUMBER NOT FOUND". Not all of the AMP parameter available can be edited using patch AMP. If you need to edit a parameter not in patch AMP you must use the ODS software.

Editing AMP with the Patch AMP Utility

The control must be placed in E-Stop before you can modify patch AMP parameters. Use this procedure to edit parameters using the Patch AMP utility:

1. Using the cursor keys or the search utility place the cursor on the AMP parameter to be edited. The selected parameter is shown in reverse video.
2. Press the **{REPLCE VALUE}** softkey
3. Type in the new value of the parameter to be edited using the keyboard. Pressing the **[ENTER]** key will change the parameters value on the CRT.
4. Repeat the above procedure until you have finished changing all the parameters you need to modify. When you are done, press the **{UPDATE & EXIT}** softkey. This will make the corrections to the controls AMP stored in RAM.

You will be prompted to power down the control and power it back up. When power is reapplied to the control the new AMP values become active. After your machine is running the way you want it to and all patch AMP edits are completed, you should use the backup utility to write the AMP to the PC hard disk (refer to page 37-15).

The control displays a message in the Basic Display Set application indicating AMP HAS BEEN MODIFIED BY PATCH AMP. This message will appear at every power up. Clear the message using the **[CAN]** key on the keyboard.

After you complete all of your AMP changes and your machine is running properly, we recommend you upload AMP to ODS. Once you have uploaded AMP to ODS you must: (1) edit and save AMP, and (2) download that AMP file again; to permanently remove the AMP HAS BEEN MODIFIED BY PATCH AMP message. Refer to Chapter 2 for details on uploading and downloading AMP.

Patch AMP Values

AMP values are configured using their QuickEdit numbers in patch AMP. The patch AMP values are always numeric values.

Important: All patch AMP values are in millimeters or degrees. Units of time can be milliseconds, seconds, or minutes (as shown in the following table).

Use the Patch AMP Value in the table below to make the correlation between the numeric value and the AMP description. Shaded parameters are used for most applications. An L next to the parameter number indicates lathe only.

Four-digit AMP parameter numbers are per axis. The first digit being the axis number.

Parameter	Parameter Name	Default Value	Patch AMP Value
11	Standard Motor Table Values	yes	0 - No 1 - Yes
21 L	Roughing cycles retract amount	0.00000 mm	0.00000 to 9999.90000 mm
22 L	Percent of cutting depth	100%	0 to 255%
23 L	Perform a rough-finishing cut	False	0 - False 1 - True
30	Paramacro Parameter Init Value	Zero	0 - Zero 1 - Vacant
32	PORT # FOR PARAMACRO EXTER-NALS	PORT B	0 - PORT B 1 - PORT A
66	Rapid Override in Dry Run	True	0 - False 1 - True
70	Dev. Detection Filter Time, spindle 1	20 msec	20 to 400 msec
71	Retract amount - peck drilling	25.40000 mm	0.00000 to 99999.00000 mm
73	Pullout angle, chamfered thrd	5.0 degrees	0.0 to 89.0 degrees
74	Pullout dist, chamfered thrd	5.080 thread leads	0.000 to 50.000 thread leads
76	Cycle clearance amount	0.25400 mm	0.00000 to 99999.00000 mm
83 L	Roughing cycle threshold depth	0.00000 in	0.00000 to 393.69685 inch
111 L	Min infeed in multi threading	0.00000 mm	0.00000 to 9999.90000 mm
112 L	Finish allow in mult threading	0.00000 mm	0.00000 to 9999.90000 mm
113 L	Retract amount in grooving	0.00000 mm	0.00000 to 9999.90000 mm
204	Tool offset cancel	Cancel Geom & Wear	0 - Do not Cancel 1 - Cancel Wear Only 2 - Cancel Geom & Wear
206	Position tolerance for Skip 1	1.00000 mm	0.00000 to 100.00000 mm
207	Position tolerance for Skip 2	1.00000 mm	0.00000 to 100.00000 mm
208	Position tolerance for Skip 3	1.00000 mm	0.00000 to 100.00000 mm
209	Position tolerance for Skip 4	1.00000 mm	0.00000 to 100.00000 mm
399	Manual ACC/DEC mode	Exponential	0 - Exponential 1 - Linear
402	Positioning ACC/DEC mode	Linear	0 - Exponential 1 - Linear or S-Curve as per G-Code

Parameter	Parameter Name	Default Value	Patch AMP Value
405	Minimum feed reduction %	80 %	0 to 100 %
408	External decel speed (cutting)	0.42333 mm/sec	0.00000 to 169.33333 mm/sec
409	Corner override distance (DTC)	2.54000 mm	0.00000 to 9999.99999 mm
410	Corner override distance (DFC)	2.54000 mm	0.00000 to 9999.99999 mm
590	Primary Spindle	None	0 to 2
591	Follower Spindle	None	0 to 2
592	Synchronized Spindle Direction	Normal	0 to 1
593	Synch Gain	0.0	0.000 to 1.000
594	Default Position Offset	0 degrees	0.000 to 360.000 degrees
595	Maximum Deviation	0 degrees	0.000 to 360.000 degrees
596	Seek Tolerance	None	0.000 to 360.000 degrees
597	Seek Timeout	300.000 sec	0.000 to 300.000 sec
600	Probe Length Compensation	0.00000 mm	0.00000 to 1270.00000 mm
601	Probe Radius Compensation	0.00000 mm	0.00000 to 254.00000 mm
603	Approach Distance (R)	0.00000 mm	0.00000 to 99999.00000 mm
604	Tolerance Band Distance (D)	0.00000 mm	0.00000 to 99999.00000 mm
605	Approach Rate (E)	0.00000 mm/sec	0.00000 to 99999.00000 mm/sec
606	Probe Rate (F)	0.00000 mm/sec	0.00000 to 99999.00000 mm/sec
630	Probe Trigger Tolerance	0.0254 mm	0.0000 to 2540.0000 mm
631	Depth Sensor Travel Limit	2540.0000 mm	0.0000 to 2540.0000 mm
640	Deskew Master Name - Set 2	None	0 - None 65 - A 66 - B 67 - C 85 - U 86 - V 87 - W 88 - X 89 - Y 90 - Z
641	Deskew Slave Name - Set 2	None	0 - None 65 - A 66 - B 67 - C 85 - U 86 - V 87 - W 88 - X 89 - Y 90 - Z
642	Deskew Gain - Set 2	0.00000	0.00000 to 10.00000
643	Excess Skew Limit - Set 2	0.00000 mm	0.00000 to 214.10000 mm

Parameter	Parameter Name	Default Value	Patch AMP Value
660	Deskew Master Name - Set 1	None	0 - None 65 - A 66 - B 67 - C 85 - U 86 - V 87 - W 88 - X 89 - Y 90 - Z 92 - $B 93 - $C 94 - $X 95 - $Y 96 - $Z
661	Deskew Slave Name - Set 1	None	0 - None 65 - A 66 - B 67 - C 85 - U 86 - V 87 - W 88 - X 89 - Y 90 - Z 92 - $B 93 - $C 94 - $X 95 - $Y 96 - $Z
662	Deskew Gain - Set 1	0.00000	0.00000 to 10.00000
663	Excess Skew Limit - Set 1	0.00000 mm	0.00000 to 214.10000 mm
753	Max Tap Speed for Spindle 1, Gear 1	500.0 rpm	0.0 to 500.0 rpm
754	Max Tap Speed for Spindle 1, Gear 2	500.0 rpm	0.0 to 500.0 rpm
755	Max Tap Speed for Spindle 1, Gear 3	500.0 rpm	0.0 to 500.0 rpm
756	Max Tap Speed for Spindle 1, Gear 4	500.0 rpm	0.0 to 500.0 rpm
757	Max Tap Speed for Spindle 1, Gear 5	500.0 rpm	0.0 to 500.0 rpm
758	Max Tap Speed for Spindle 1, Gear 6	500.0 rpm	0.0 to 500.0 rpm
759	Max Tap Speed for Spindle 1, Gear 7	500.0 rpm	0.0 to 500.0 rpm
760	Max Tap Speed for Spindle 1, Gear 8	500.0 rpm	0.0 to 500.0 rpm
761	Max Tap Speed for Spindle 2, Gear 1	500.0 rpm	0.0 to 500.0 rpm

Parameter	Parameter Name	Default Value	Patch AMP Value
762	Max Tap Speed for Spindle 2, Gear 2	500.0 rpm	0.0 to 500.0 rpm
763	Max Tap Speed for Spindle 2, Gear 3	500.0 rpm	0.0 to 500.0 rpm
764	Max Tap Speed for Spindle 2, Gear 4	500.0 rpm	0.0 to 500.0 rpm
765	Max Tap Speed for Spindle 2, Gear 5	500.0 rpm	0.0 to 500.0 rpm
766	Max Tap Speed for Spindle 2, Gear 6	500.0 rpm	0.0 to 500.0 rpm
767	Max Tap Speed for Spindle 2, Gear 7	500.0 rpm	0.0 to 500.0 rpm
768	Max Tap Speed for Spindle 2, Gear 8	500.0 rpm	0.0 to 500.0 rpm
777	Gain for Spindle 1 - Gear 1	1.00000	0.00000 to 100.00000
778	Gain for Spindle 1 - Gear 2	1.00000	0.00000 to 100.00000
779	Gain for Spindle 1 - Gear 3	1.00000	0.00000 to 100.00000
780	Gain for Spindle 1 - Gear 4	1.00000	0.00000 to 100.00000
781	Gain for Spindle 1 - Gear 5	1.00000	0.00000 to 100.00000
782	Gain for Spindle 1 - Gear 6	1.00000	0.00000 to 100.00000
783	Gain for Spindle 1 - Gear 7	1.00000	0.00000 to 100.00000
784	Gain for Spindle 1 - Gear 8	1.00000	0.00000 to 100.00000
785	Gain for Spindle 2 - Gear 1	1.00000	0.00000 to 100.00000
786	Gain for Spindle 2 - Gear 2	1.00000	0.00000 to 100.00000
787	Gain for Spindle 2 - Gear 3	1.00000	0.00000 to 100.00000
788	Gain for Spindle 2 - Gear 4	1.00000	0.00000 to 100.00000
789	Gain for Spindle 2 - Gear 5	1.00000	0.00000 to 100.00000
790	Gain for Spindle 2 - Gear 6	1.00000	0.00000 to 100.00000
791	Gain for Spindle 2 - Gear 7	1.00000	0.00000 to 100.00000
792	Gain for Spindle 2 - Gear 8	1.00000	0.00000 to 100.00000
803	Spindle DAC Output Ramping (spindle 1)	Off	0 - Off 1 - On
804	Spindle DAC Output Ramping (spindle 2)	Off	0 - Off 1 - On
805	Spindle DAC Output Ramping (spindle 3)	Off	0 - Off 1 - On
812	Acceleration Time for Spindle 2, Gear 1	0.50000 sec	0.00000 to 1000.00000 sec
813	Acceleration Time for Spindle 2, Gear 2	0.50000 sec	0.00000 to 1000.00000 sec
814	Accelerating Time for Spindle 2, Gear 3	0.50000 sec	0.00000 to 1000.00000 sec

Parameter	Parameter Name	Default Value	Patch AMP Value
815	Acceleration Time for Spindle 2, Gear 4	0.50000 sec	0.00000 to 1000.00000 sec
816	Acceleration Time for Spindle 2, Gear 5	0.50000 sec	0.00000 to 1000.00000 sec
817	Acceleration Time for Spindle 2, Gear 6	0.50000 sec	0.00000 to 1000.00000 sec
818	Acceleration Time for Spindle 2, Gear 7	0.50000 sec	0.00000 to 1000.00000 sec
819	Acceleration Time for Spindle 2, Gear 8	0.50000 sec	0.00000 to 1000.00000 sec
820	Voltage at Max for Spindle 1, Gear 1	10.0000 volts	-10.0000 to 10.0000 volts
821	Voltage at Max for Spindle 1, Gear 2	10.0000 volts	-10.0000 to 10.0000 volts
822	Voltage at Max for Spindle 1, Gear 3	10.0000 volts	-10.0000 to 10.0000 volts
823	Voltage at Max for Spindle 1, Gear 4	10.0000 volts	-10.0000 to 10.0000 volts
824	Voltage at Max for Spindle 1, Gear 5	10.0000 volts	-10.0000 to 10.0000 volts
825	Voltage at Max for Spindle 1, Gear 6	10.0000 volts	-10.0000 to 10.0000 volts
826	Voltage at Max for Spindle 1, Gear 7	10.0000 volts	-10.0000 to 10.0000 volts
827	Voltage at Max for Spindle 1, Gear 8	10.0000 volts	-10.0000 to 10.0000 volts
830	Voltage at Max for Spindle 2, Gear 1	10.0000 volts	-10.0000 to 10.0000 volts
831	Voltage at Max for Spindle 2, Gear 2	10.0000 volts	-10.0000 to 10.0000 volts
832	Voltage at Max for Spindle 2, Gear 3	10.0000 volts	-10.0000 to 10.0000 volts
833	Voltage at Max for Spindle 2, Gear 4	10.0000 volts	-10.0000 to 10.0000 volts
834	Voltage at Max for Spindle 2, Gear 5	10.0000 volts	-10.0000 to 10.0000 volts
835	Voltage at Max for Spindle 2, Gear 6	10.0000 volts	-10.0000 to 10.0000 volts
836	Voltage at Max for Spindle 2, Gear 7	10.0000 volts	-10.0000 to 10.0000 volts
837	Voltage at Max for Spindle 2, Gear 8	10.0000 volts	-10.0000 to 10.0000 volts
851	Default Orient Direction, Spindle 1	Counterclockwise	1 - Counterclockwise 2 - Clockwise
852	Default Orient Direction, Spindle 2	Counterclockwise	1 - Counterclockwise 2 - Clockwise

Parameter	Parameter Name	Default Value	Patch AMP Value
857	Spindle Marker Calibration, Spindle 1	0.000000 degrees	–360.000000 to 360.000000 degrees
858	Orient Speed, spindle 1	10.0 rpm	0.0 to 99999.9 rpm
859	Orient Inposition Band, spindle 1	0.100000 degrees	0.000000 to 360.000000 degrees
860	Number of Gears Used, spindle 1	1	0 to 8 gears
861	Spindle Deviation Tolerance, Spindle 1	100 %	0 to 100 %
862	Default Orient Angle, spindle 1	0.000000 degrees	0.000000 to 360.000000 degrees
863	Spindle Marker Calibration, Spindle 2	0.000000 degrees	–360.000000 to 360.000000 degrees
864	Orient Speed, spindle 2	10.0 rpm	0.0 to 99999.9 rpm
865	Orient Inposition Band, spindle 2	0.100000 degrees	0.000000 to 360.000000 degrees
866	Number of Gears Used, spindle 2	1	0 to 8 gears
867	Spindle Deviation Tolerance, Spindle 2	100 %	0 to 100 %
868	Default Orient Angle, spindle 2	0.000000 degrees	0.000000 to 360.000000 degrees
875	Dev. Detection Filter Time, spindle 2	20 msec	20 to 400 msec
882	Acceleration Time for Spindle 1, Gear 1	0.50000 sec	0.00000 to 1000.00000 sec
883	Acceleration Time for Spindle 1, Gear 2	0.50000 sec	0.00000 to 1000.00000 sec
884	Accelerating Time for Spindle 1, Gear 3	0.50000 sec	0.00000 to 1000.00000 sec
885	Acceleration Time for Spindle 1, Gear 4	0.50000 sec	0.00000 to 1000.00000 sec
886	Acceleration Time for Spindle 1, Gear 5	0.50000 sec	0.00000 to 1000.00000 sec
887	Acceleration Time for Spindle 1, Gear 6	0.50000 sec	0.00000 to 1000.00000 sec
888	Acceleration Time for Spindle 1, Gear 7	0.50000 sec	0.00000 to 1000.00000 sec
889	Acceleration Time for Spindle 1, Gear 8	0.50000 sec	0.00000 to 1000.00000 sec
900	Minimum Spindle 1 Speed – Gear 1	0.0 rpm	0.0 to 500.0 rpm
901	Minimum Spindle 1 Speed – Gear 2	0.0 rpm	0.0 to 500.0 rpm
902	Minimum Spindle 1 Speed – Gear 3	0.0 rpm	0.0 to 500.0 rpm
903	Minimum Spindle 1 Speed – Gear 4	0.0 rpm	0.0 to 500.0 rpm
904	Minimum Spindle 1 Speed – Gear 5	0.0 rpm	0.0 to 500.0 rpm
905	Minimum Spindle 1 Speed – Gear 6	0.0 rpm	0.0 to 500.0 rpm

Parameter	Parameter Name	Default Value	Patch AMP Value
906	Minimum Spindle 1 Speed - Gear 7	0.0 rpm	0.0 to 500.0 rpm
907	Minimum Spindle 1 Speed - Gear 8	0.0 rpm	0.0 to 500.0 rpm
910	Maximum Spindle 1 Speed - Gear 1	500.0 rpm	0.0 to 99999.9 rpm
911	Maximum Spindle 1 Speed - Gear 2	500.0 rpm	0.0 to 99999.9 rpm
912	Maximum Spindle 1 Speed - Gear 3	500.0 rpm	0.0 to 99999.9 rpm
913	Maximum Spindle 1 Speed - Gear 4	500.0 rpm	0.0 to 99999.9 rpm
914	Maximum Spindle 1 Speed - Gear 5	500.0 rpm	0.0 to 99999.9 rpm
915	Maximum Spindle 1 Speed - Gear 6	500.0 rpm	0.0 to 99999.9 rpm
916	Maximum Spindle 1 Speed - Gear 7	500.0 rpm	0.0 to 99999.9 rpm
917	Maximum Spindle 1 Speed - Gear 8	500.0 rpm	0.0 to 99999.9 rpm
920	Minimum Spindle 2 Speed - Gear 1	0.0 rpm	0.0 to 500.0 rpm
921	Minimum Spindle 2 Speed - Gear 2	0.0 rpm	0.0 to 500.0 rpm
922	Minimum Spindle 2 Speed - Gear 3	0.0 rpm	0.0 to 500.0 rpm
923	Minimum Spindle 2 Speed - Gear 4	0.0 rpm	0.0 to 500.0 rpm
924	Minimum Spindle 2 Speed - Gear 5	0.0 rpm	0.0 to 500.0 rpm
925	Minimum Spindle 2 Speed - Gear 6	0.0 rpm	0.0 to 500.0 rpm
926	Minimum Spindle 2 Speed - Gear 7	0.0 rpm	0.0 to 500.0 rpm
927	Minimum Spindle 2 Speed - Gear 8	0.0 rpm	0.0 to 500.0 rpm
930	Maximum Spindle 2 Speed - Gear 1	500.0 rpm	0.0 to 99999.9 rpm
931	Maximum Spindle 2 Speed - Gear 2	500.0 rpm	0.0 to 99999.9 rpm
932	Maximum Spindle 2 Speed - Gear 3	500.0 rpm	0.0 to 99999.9 rpm
933	Maximum Spindle 2 Speed - Gear 4	500.0 rpm	0.0 to 99999.9 rpm
934	Maximum Spindle 2 Speed - Gear 5	500.0 rpm	0.0 to 99999.9 rpm

Parameter	Parameter Name	Default Value	Patch AMP Value
935	Maximum Spindle 2 Speed - Gear 6	500.0 rpm	0.0 to 99999.9 rpm
936	Maximum Spindle 2 Speed - Gear 7	500.0 rpm	0.0 to 99999.9 rpm
937	Maximum Spindle 2 Speed - Gear 8	500.0 rpm	0.0 to 99999.9 rpm
1002 2002 3002	Dir. to Move Off Limit Switch	negative direction off switch	-1 - negative direction off switch 1 - positive direction off switch
1012 2012 3012	Max % rated torque (-)	200 %	0 to 200%
1013 2013 3013	Teeth on motor gear for vel FB	1	1 to 32,767 teeth
1017 2017 3017	Load inertia ratio	1 : 0	0 - 1:0 1 - 1:1 2 - 1:2 3 - 1:3
1018 2018 3018	Teeth on lead screw for vel FB	1	1 to 32,767
1019 2019 3019	Analog Servo Pos. Voltage	10.0000 volts	-10.0000 to 10.0000 volts
1020 2020 3020	Software Overtravel Used	False	0 - False 1 - True
1021 2021 3021	Maximum Servo Acceleration	1989.66667 mm/s/s	0.00000 to 9999.99990 mm/s/s
1023 2023 3023	Ve Integrator Discharge Rate	1	1 to 8
1024 2024 3024	Analog Servo Neg. Voltage	-10.0000 volts	-10.0000 to 10.0000 volts
1026 2026 3026	Teeth on gear for pos. FB	1	1 to 32,767 teeth
1027 2027 3027	Teeth on lead screw for pos FB	1	1 to 32,767 teeth
1150 2150 3150	Hard Stop Holding Torque	10%	1 to 100% (nominal rated torque)
1151 2151 3151	Hard Stop Detection Torque	90%	1 to 300% (nominal rated torque)

Parameter	Parameter Name	Default Value	Patch AMP Value
1152 2152 3152	Feed Integral Torque Gain (for adaptive depth)	0.500000	0.100000 to 2.000000
1153 2153 3153	Feed Proportional Torque Gain (for adaptive depth)	0.100000	0.100000 to 2.000000
1154 2154 3154	Feedrate Acc/Dec Enable (for adaptive depth)	Enabled	0 - Disabled 1 - Enabled
1162 2162 3162	Torque Filter Cutoff Frequency	200 Hz	10 to 10000 Hz
1200 2200 3200	External decel speed (posit.)	4.23333 mm/sec	0.00000 to 400.00000 mm/sec
1201 2201 3201	Maximum cutting feedrate	42.33333 mm/sec	0.00000 to 169.33333 mm/sec
1202 2202 3202	ACC/DEC ramp	27.12674 mm/s/s	0.00000 to 9999.99999 mm/s/s
1203 2203 3203	Rapid feedrate for positioning	169.33333 mm/sec	0.00000 to 4064.00000 mm/sec
1204 2204 3204	Velocity step for ACC/DEC	4.23333 mm/sec	0.00000 to 169.33333 mm/sec
1210 2210 3210	Linear Acceleration Ramp	25.4 mm/sec/sec	0.00000 to 20,000.00000 mm/sec/sec
1211 2211 3211	Linear Deceleration Ramp	25.4 mm/sec/sec	0.00000 to 20,000.00000 mm/sec/sec
1212 2212 3212	S-Curve Acceleration Ramp	25.4 mm/sec/sec	0.00000 to 20,000.00000 mm/sec/sec
1213 2213 3213	S-Curve Deceleration Ramp	25.4 mm/sec/sec	0.00000 to 20,000.00000 mm/sec/sec
1214 2214 3214	Axis Jerk	5.08 mm/sec/sec/sec	0.00000 to 1,000,000.0000 mm/sec/sec/sec

Parameter	Parameter Name	Default Value	Patch AMP Value
1215 2215 3215	Minimum Programmable Jerk	5.08 mm/sec/sec/sec	0.0000 to 1,000,000.0000 mm/sec/sec/sec
1300 2300 3300	Home Calibration	0.00000 mm	-999999.99000 to 999999.99000 mm
1310 2310 3310	Axis Position after Homing	0.00000 mm	-2540000.00000 to 2540000.00000 mm
1320 2320 3320	Positive Software Overtravel	0.00000 mm	-2540000.00000 to 2540000.00000 mm
1330 2330 3330	Negative Software Overtravel	0.00000 mm	-2540000.00000 to 2540000.00000 mm
1340 2340 3340	Reversal Error Compensation	0.00000 mm	-9.99900 to 9.99900 mm
1350 2350 3350	Home Speed from Limit Switch	4.2333 mm/sec	0.0001 to 169.3333 mm/sec
1504 2504 3504	SERCOS Transmit Level	High	0 - Low 1 - High
1510 2510 3510	Servo Position Loop Type	Closed Loop	0 - Open Loop for Analog Only 1 - Closed Loop 2 - ZFE Closed Loop 3 - Servo Off 4 - Servo Detached 5 - Depth Probe
1560 2560 3560	Velocity Feedback Type	Hiperface Incremental	0 - No Feedback 3 - INC Encoder - A/B/Z (Z<A) 4 - INC Encoder - A/B/Z (Z>A) 9 - Hiperface Absolute 10- Hiperface Incremental
1565 2565 3565	Velocity Feedback Counts/Cycle	2000	4 to 4,194,304
1570 2570 3570	Position Feedback Type	Hiperface Incremental	0 - No Feedback 3 - INC Encoder - A/B/Z (Z<A) 4 - INC Encoder - A/B/Z (Z>A) 9 - Hiperface Absolute 10- Hiperface incremental 11- distance coded marker
1575 2575 3575	Position Feedback Counts/Cycle	2000	4 to 4,194,304

Parameter	Parameter Name	Default Value	Patch AMP Value
1580 2580 3580	Maximum Motor Speed	2200 rpm	0 to 99,999 rpm
1590 2590 3590	Lead screw thread pitch	2.540 mm	0.000 to 254.000 mm
1595 2595 3595	Sign of Position Feedback	Plus	1 - Plus -1 - Minus
1600 2600 3600	Sign of Velocity Feedback	Plus	1 - Plus -1 - Minus
1660 2660 3660	Peak Current as a % of RMS	200%	200 - 200% 300 - 300%
1670 2670 3670	Max % rated torque (+)	200 %	0 to 200%
1680 2680 3680	Feed Forward Percent	0 %	0 to 100%
1690 2690 3690	Torque Offset Percentage	0 %	0 to 100%
1700 2700 3700	Torque Offset Direction	Minus	1 - Plus -1 - Minus
1710 2710 3710	Initial Gain of Position Loop	1.00000	0.00000 to 30.00000
1720 2720 3720	Position Loop Gain Break Ratio	1.000000	0.000000 to 1.000000
1730 2730 3730	Gain Break Point	203.20000 mm	0.00000 to 214.10000 mm
1735 2735 3735	Inposition Band	0.02540 mm	0.00000 to 214.10000 mm
1740 2740 3740	Feedrate Suppression Point	203.20000 mm	0.00000 to 214.10000 mm
1750 2750 3750	Excess Error	203.200000 mm	0.00000 to 214.10000 mm
1755 2755 3755	For internal use only (do not adjust)	7	1 to 32,767

Parameter	Parameter Name	Default Value	Patch AMP Value
1780 2780 3780	Servo Amplifier Type	No Servo Amplifier	14 - 1394 - AM03 (2 kW module) 15 - 1394 - AM03 (3 kW module) 16 - 1394 - AM07 (5 kW module) 17 - 1394 - AM75 (10 KW module) 18 - 1394 - AM 50 (15 KW module)
1800 2800 3800	Velocity Integral Gain	82	33 to 32,768
1801 2801 3801	Velocity proportional gain	17408	0 to 65,536

Writing AMP to Flash

When you modify the AMP parameters using patch AMP you are editing a copy of AMP that is in memory local to the 9/PC card. Once you have completed all of your modifications to AMP you should backup AMP to the host CP's hard disk.

To save an additional copy of your AMP changes in a format that can be read and edited in ODS, we recommend uploading your final AMP configuration by uploading to ODS .

END OF CHAPTER

Tuning a Digital or Tachless Analog System

Overview

Digital and analog tachless servo systems differ from traditional drive systems in that the CNC controls the velocity loop, which is closed by the 1394 CNC Serial Drive or the 1394 Analog Interface Module. Traditional systems typically use the drive amplifier to close the velocity loop while the CNC controls the position loop. The digital and analog tachless servo system is designed to:

- reduce integration costs by not requiring an external tachometer on the servo motor
- give the 9/PC CNC control of the velocity loop (necessary for adaptive feed and feed to hard stop features) and provide better servo control.

You can either close the velocity and position loops using the same encoder device (attached to the motor shaft) or you can close the velocity loop with an encoder on the motor shaft and use some other A quad B device to close the position loop. The velocity loop must always be closed by a feedback device mounted directly on the motor shaft.

Important: Currently, (up to Release 2) the 9/PC does not support the Servo Position Loop Type “Position/Velocity” (i.e., tachless analog).

ATTENTION: For safe performance, your servo drive system uses normal regenerative breaking for 2 seconds after transitions into E-Stop. However, we recommend incorporating dynamic breaking circuitry into your servo drive system for any failure that might prevent regenerative breaking (e.g., control voltage loss, servo watchdog, overtemperature modes).

Use the following AMP parameters to configure your drive system:

Table A.A
Velocity Loop Parameters

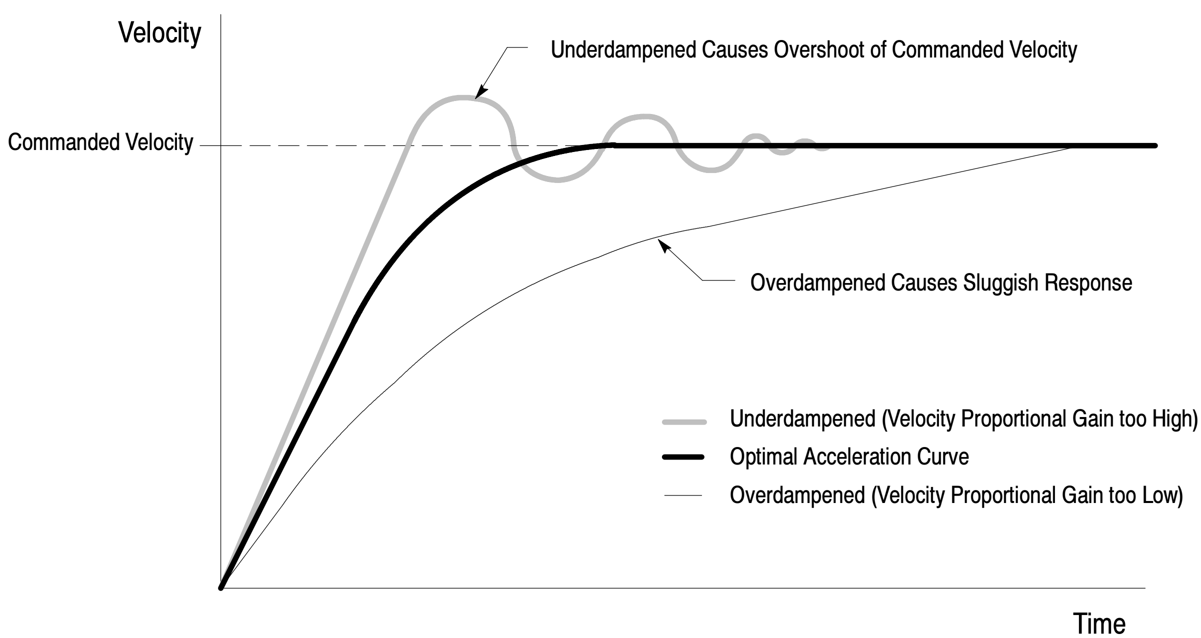
AMP Parameter	Description	For Tuning Procedure:	Page
Servo Loop Type	Selects the type of servo loop the axis will be using	Set to final value: Digital or Position/Velocity	7-9
Velocity Loop Feedback Port	Indicates what connector the velocity feedback will be connected to. Select the same port as the position loop feedback port if the same feedback device is to be used for the velocity and position loops.	Set to final value	7-46

AMP Parameter	Description	For Tuning Procedure:	Page
Velocity Feedback Type	Indicates the type of feedback device attached to the velocity feedback port. Select the same device as the position loop feedback type if the same feedback device is to be used for the velocity and position loops.	Set to final value	7-47
Velocity Feedback Counts/Cycle	Indicates the number of counts per revolution generated by the velocity feedback device. Select the same number of counts as the position loop feedback Counts/Rev if the same feedback device is to be used for the velocity and position loops.	Set to final value	7-49
Sign of Velocity Feedback	Indicates the direction you have wired feedback counts. As the servo moves positive, counts must be increasing.	Set to final value	7-50
Maximum Servo Acceleration	Set this parameter to indicate to the control the maximum acceleration for the drive system.	Set to final value	7-67
Velocity Integral Gain	Use this parameter to tune the velocity loop using the procedure described later in this appendix.	Set to zero using Patch AMP Look at defaults from Table 7.C for digital systems to get an approximation of your final value.	7-56
Velocity Proportional Gain	Use this parameter to tune the velocity loop using the procedure described later in this appendix.	Set to 1500 Look at defaults from Table 7.B for digital systems to get an approximation of your final value.	7-54
VE Integrator Discharge Rate	This parameter is used when a large variation in load may occur on a moving axis. It reduces velocity overshoot and flattens the servo response curve.	Set to 1 (one)	7-58
Torque Offset Percentage	Use this parameter when there is a continuous static load in one direction. This parameter tells the control the percentage of the servo's maximum torque which must be applied to compensate for this load.	Set to 0 (zero)	7-63
Torque Offset Direction	Use this parameter when there is a continuous static load in one direction. This parameter tells the control the direction to apply the torque offset percentage.	Set to Plus	7-64
Max % rated torque (-)	Indicates the maximum torque limit for the servo in the negative direction.	Set to final value	7-61
Max % rated torque (+)	Indicates the maximum torque limit for the servo in the positive direction.	Set to final value	7-62

AMP Parameter	Description	For Tuning Procedure:	Page
Analog Servo Pos. Voltage	Indicates the servo motor's maximum rated current in the positive direction relative to amplifier output (what voltage the servo chassis should signal the amplifier to reach maximum motor rated current).	Set to final value	7-41
Analog Servo Neg. Voltage	Indicates the servo motor's maximum rated current in the negative direction relative to amplifier output (what voltage the servo chassis should signal the amplifier to reach maximum motor rated current).	Set to final value	7-43
Peak Current as a % of RMS	Indicates the percent of the servo motors nominal rated current (continuous duty) the Analog Servo Pos. Voltage or Analog Servo Neg. Voltage generate (which ever is higher).	Set to final value	7-60

The Velocity Loop

The Optimal Acceleration Curve on the following chart is your desired servo response. This curve is determined by your machine dynamics and the amplifier/motor combination you have selected.



The tuning procedure discussed here covers how to adjust the velocity proportional gain and the velocity integral gain to fine tune your servo drive system. Before you can begin adjusting these parameters, you must complete the following tasks.

Important: This tuning procedure assumes you have already successfully completed the system start up procedure described in chapter 4 of your *9/PC CNC Software and Hardware Installation and Integration Manual*.

AMP for Tuning

Download AMP with the following consideration:

- Configure all velocity loop parameters as listed in Table A.A. You should be able to set most of the parameters to their final values (such as number of counts per rev and velocity feedback port).
- Configure your linear Acc/Dec Curve for an acceleration/deceleration ramp at a slope close to (but slightly less than) your expected maximum servo acceleration slope (refer to appendix B for assistance). Your entire Acc/Dec curve must be entirely below the curve of the selected position loop gain throughout the operating speed range.
- Unfit the last two analog/out DAC ports on each 1394 servo chassis in your system. Typically these are used as spindle ports. This is necessary to avoid conflicts with the DAC monitor feature discussed later.
- Configure the following spindle gear range parameters:
  Voltage at Max for Gear 1 – set to 10V
  Max Spindle Speed Gear 1 – Set to 3000 RPM
  The settings for gear range one scale the output of the DAC monitor. You may need to adjust the RPM value later depending on the feedback resolution for the axis you are tuning.

Logic Considerations

Servos must be homed before they can be programmed and tuned. This can prove inconvenient since homing an untuned servo can be dangerous and damaging to hardware.

This homing requirement can be avoided through logic. The flag FW_HMNO is available which causes the control to believe the selected axis has already been homed. Write logic to set the bit of FW_HMNO that corresponds to the axes being tuned. Refer to your *9/PC CNC Logic Reference Manual* for details on using the switchless homing feature.

ATTENTION: Using the switchless homing feature can cause invalid/inaccurate software overtravels and programmable zones. Confirm that the hardware overtravel switches are functioning properly. Make sure that the axis' position when the tuning part program is executed does not violate any of these areas.

Connect the Strip Chart Recorder

This tuning procedure assumes you are using a strip chart recorder (an oscilloscope or other device may be used, however, we recommend it has either printing or storage capability for comparison of curves). Since this is a tachless servo system you must connect the strip chart recorder to the analog connector and use the DAC monitor feature.

The DAC monitor feature is used to direct velocity or following error information for a selected servo to the analog connectors. The servo you are tuning must be on the same servo chassis as the analog connector used for the strip chart recorder.

ATTENTION: If a spindle (or other device) is configured and attached to the Analog Out port, make sure that the drive is disabled and will be in a safe state throughout the tuning procedure. We recommend disconnecting the wiring to this device from the Analog Out port when using the DAC monitor.

Attach the strip chart recorder to the Analog Out port. Refer to your *9/PC CNC Software and Hardware Installation and Integration Manual* and your strip chart recorder documentation for polarity considerations.

Turning the DAC Monitor On

The DAC monitor utility can only be enabled through a feature called Patch AMP. Patch AMP allows you to alter some AMP values from the control without having to use the ODS AMP editor and download utilities. The parameters to turn the DAC monitor utility on are not available to the AMP editor in ODS.

For details on using the Patch AMP feature, refer to page 37-1. These Patch AMP parameters are used to configure the DAC monitor feature:

DAC Monitor Parameters	Function
# 86	Enables the DAC monitor on TB2 on the 1394 CNC Serial Drive or AOUT5 on the 1394 Analog Interface Module.
# 820	Sets the maximum full scale voltage output for #86.
# 910	Scales the DAC output for #86.
# 87	Enables the DAC monitor on TB3 on the 1394 CNC Serial Drive or AOUT6 on the 1394 Analog Interface Module.
# 830	Sets the maximum full scale voltage output for #87.
# 930	Scales the DAC output for #87.

Monitoring Multiple Functions via DAC Monitor

DAC monitor enables you to monitor up to two functions simultaneously. These two functions may either be on the same 1394 serial drive or on different 1394 serial drives, depending on the selected axes for each function. Select these functions by setting parameters 86 and 87. Refer to the table on page A-5 regarding the function of these parameters.

To configure DAC monitoring for multiple functions simultaneously, use the following calculations:

#86 = Function code + AMPed axis number of the *first* function to be DAC monitored.
#87 = Function code + AMPed axis number of the *second* function to be DAC monitored.

Table A.B
DAC Monitor Function Codes for #86 and #87

Name	Function Code
Velocity Feedback (VFBx)	100
Following Error (FEx)	200*
Velocity Integrator (VINTx)	300
Velocity Error (VRx)	400
Coarse IPCs (CIPCx)	500*
Final Velocity (VFx)	600

*Applies to Servo Loop Type “Position/Analog Spindle”

For example, to simultaneously monitor following error on TB2 and coarse IPCs on TB3 for the fourth AMPed, axis one would set

P86 = 204 and P87 = 504

Make sure the chart recorder is connected to TB2 and TB3 on the proper 1394 serial drive where the fourth AMPed axis is located to get the outputs. TB2 and TB3 on the other 1394 serial drive are not available for this example, since a maximum of two functions may be used.

Important: If an invalid parameter is entered on either P86 or P87, the system will be held in E-Stop displaying an error message.

Scaling DAC Monitor Output

DAC Monitor allows you to scale the output voltage to make the results visible on the chart recorder. You may scale the output by setting parameters #820, #830, #910, and #920 in Patch AMP. Refer to Table A.C for example values that may provide acceptable output voltages for monitoring the listed parameters. Although the following data shows values that may provide acceptable output, keep in mind that AMP values such as **Pitch**, **Feedback Resolution**, and **Program Formats** affect the outcome.

Important: Parameters #820 and #910 can be used to scale DAC monitoring output. However, we recommend setting #820 (and #830) to 10V and varying parameter #910 (and #930) for accurate scaling. Note that for the following example, parameters #820 set to 10V.

Table A.C
Example Values for DAC Monitor Parameter #910 or #930

DAC Monitor Function Code (#86 or #87)	Pitch (in.)	
	0.1	0.5
101 = Velocity Feedback (VFBx)	5000	1000
201 = Following Error (FEx)	99999.9	20,000
301 = Velocity Integrator (VINTx)	10,000	2000
401 = Velocity Error (VRx)	50	20
501 = Coarse IPCs (CIPCx)	99999.9	20,000
601 = Final Velocity (VFx)	10,000	2000

Important: If the example values do not work, increase the value to reduce the output voltage and decrease the value to increase the output voltage.

When you press the update and exit softkey in Patch AMP, the control will ask you to cycle power. You may also select `{Start 9/PC}` or `{Stop 9/PC}` via the **9/PC Configuration Manager**. Once you turn power back on, the control will display a message indicating the DAC monitor has been enabled, what it is monitoring, and the axis name being monitored.

Create Your Tuning Part Program

A tuning part program is not necessary but can be helpful when tuning your system. This part program should perform a repetitive move that reverses direction in a repeating loop. The feedrate for this move should be equal to your machine's maximum cutting feedrate. Refer to your *9/PC CNC Operation and Programming Manual* for details on creating a part program.

The example part program below repeats its moves six times using paramacro looping. If your system does not have paramacros, just repeating the same motion blocks will have the same effect. This example assumes that the servo being tuned positions the X axis and that the maximum cutting feedrate is 1000 IPM. You may need to adjust the axis name, range of motion, and feedrate in this example for your application.

Example Tuning Part Program

```
N00001 G20 G94 G91 G94;
N00002 #100=0;
N00003 G01 X5 F1000;
N00004 G04 P1
N00005 X-5;
N00006 G04 P1;
N00007 #100=#100+1;
N00008 If [#100 LT 6] GOTO 3;
N00009 M02;
```

Tuning the Drive

This procedure assumes you have AMPed your system as described above, have the DAC monitor on and outputting velocity feedback for the axis you are monitoring, and have your strip chart recorder connected.

You should have your velocity integral gain set to zero and your proportional gain set at around 1500 (you can adjust these using online AMP refer to page 36-11). Tuning is performed on a cutting move (G01) at the maximum cutting feedrate. You should not actually be cutting material for this procedure. A G01 cutting move is used so that the control uses linear acceleration for tuning instead of the exponential acceleration that is used for rapid moves.

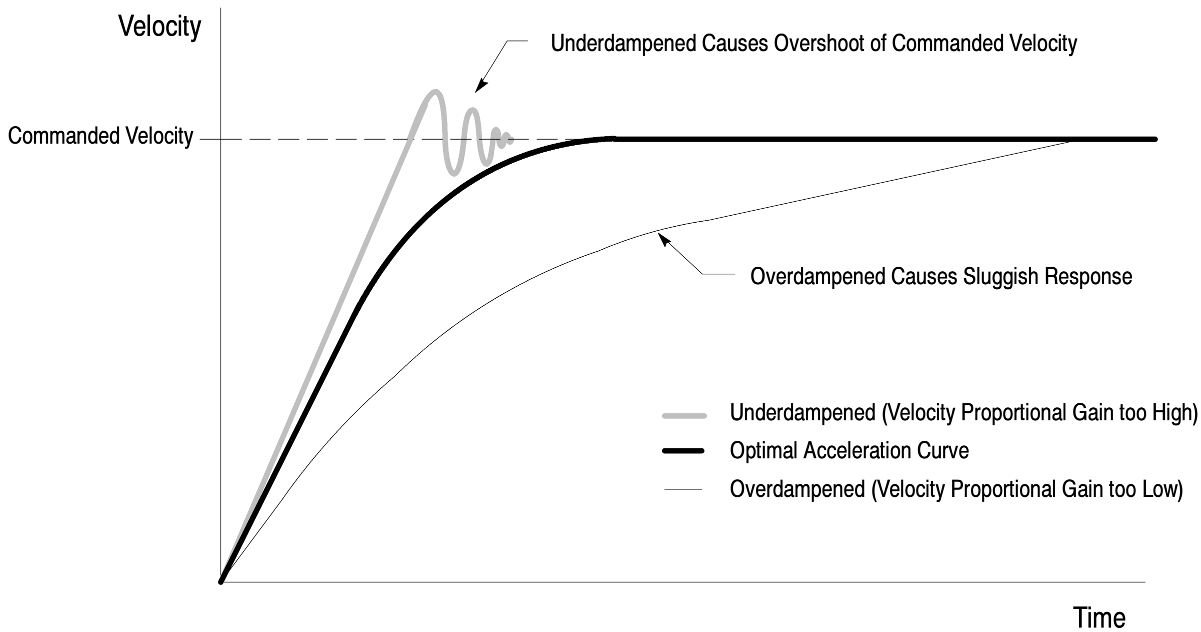
We also recommend that for the purpose of tuning you disable the Torque Filter Cutoff Frequency that you may have enabled using AMP (set it as high as possible to disable) and set the VE Integrator Discharge rate to 1. You can add these features once the optimal acceleration curve is found.

You can tune the velocity loop with the position loop open or closed. We recommend tuning with the position loop closed since better servo performance can be obtained in this fashion.

Find the Maximum Velocity Proportional Gain value:

1. Activate and execute your tuning part program and record the servos velocity response with the strip chart recorder.

Figure A.1
Acceleration Curve for Tuning



With the Velocity Proportional Gain set as discussed in Table A.A, most systems will be overdampened as shown in Figure A.1.

Important: If your system strip chart record is either off the scale or too small to easily read, you can scale the output to the recorder using Patch AMP parameters #910 or #930 (refer to page A-5 for a description). By raising the value of this parameter you will scale down the velocity output. By lowering the value of this parameter you will scale up the velocity output.

ATTENTION: Remember these are the spindle gear range scale factors. Be sure to restore them before reconnecting your spindle.

2. Adjust the velocity proportional gain (refer to page 36-11) and change the velocity proportional gain for the axis you are tuning. If the acceleration curve appears overdampened, raise the value of the velocity proportional gain. If the acceleration curve appears underdampened lower the velocity proportional gain.

3. Rerun your tuning part program. Continue adjusting the velocity proportional gain until the axis just becomes underdampened (small sharp velocity overshoot occurs over the command velocity).

   Smaller lighter systems may become very unstable. Chatter can occur instead of the smooth overshoot curve shown in Figure A.1 which is typical of larger systems. With no integral gain, it is doubtful you will see the optional acceleration curve on most systems.

4. Lower the velocity proportional gain to the highest value possible at which only a slight velocity overshoot (or chatter) occurs. This is your maximum velocity proportional gain value.

Tuning for Optimal Acceleration

Now that you know the maximum value for velocity proportional gain, you can begin fine tuning the drive for optimal performance as follows:

5. Raise the velocity integral gain (refer to page 36-11). Up to this point, the velocity integral gain was set at its minimum value.

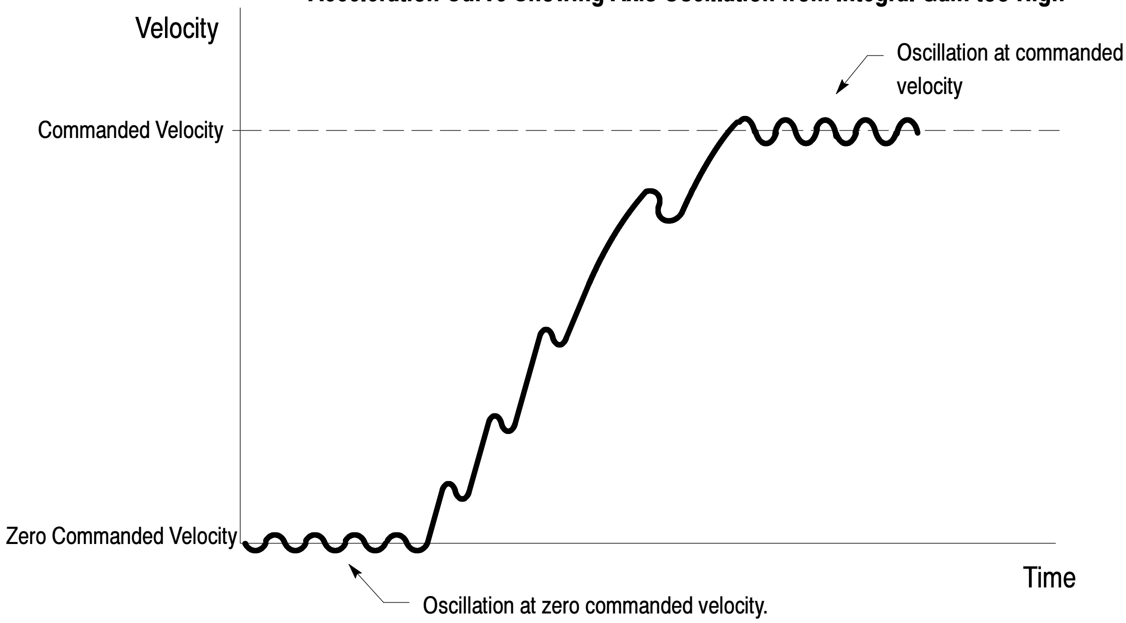
6. Lower the velocity proportional gain until the sharp overshoot occurs again.

   Continue raising the velocity integral gain by small increments. Each time you raise the integral gain you should create a velocity overshoot that can be detected on your strip chart recorder. Remove the created velocity overshoot by lowering the velocity proportional gain (strip chart recording performance as described previously while running your tuning part program). Typical final velocity integral values are between 5 and 10 times smaller than the velocity proportional gain value.

   Continue raising velocity integral gain and lowering velocity proportional gain until an oscillation occurs that can not be stabilized by adjusting the velocity proportional gain. This “bounded” oscillation will appear different than the velocity overshoot shown previously. This will be a continuous harmonic oscillation either when the servo reaches speed or when the servo is at rest.

   This bounded oscillation is a result of the integral gain being too high. You should first notice the oscillation during motion at the commanded velocity. If you continue raising the velocity integral gain you will see oscillation when the axis is at rest (no commanded velocity).

Figure A.2
Acceleration Curve Showing Axis Oscillation from Integral Gain too High



7. Once you have found the value of velocity integral gain at which the axis just starts to oscillate, lower its value until the oscillation just stops. Adjust the velocity proportional gain to a point just below where over shoot occurs. This will be your optimal values for that axis and will give you the best servo performance.

8. Upload or manually enter these values into your ODS AMP file for backup.

9. Repeat this procedure until all axes are tuned.

10. Remove the strip chart recorder and turn the DAC monitor feature off in Patch AMP.

11. Reconfigure and attach any devices (typically spindles) that are to be connected to the analog out ports (DAC) in AMP and reset the spindle gear range parameters. Reset your ACC/DEC ramp for the axis to an acceleration curve value below the optimal curve you just generated with your strip chart recorder.

12. Remove the FW_HMNO flag from logic if you enabled it for tuning.

END OF APPENDIX

Integrating a Linear Feedback Device

Overview

Two of the more common positioning feedback types supported by the 9/PC are rotary encoders and linear scales. This appendix is designed as an overview to help you get the necessary information into the system to make your linear scale work.

The 9/Series is designed to support these three major styles of linear scales:

- A quad B with no marker – since this type of feedback device has no marker, homing to a reliable zero point is not possible. This device should only be used on applications where relative (incremental) positioning is all that is required.
- A quad B with one marker – this type of device has one marker located at or near a mechanical home limit switch. It is important that no more than one marker be observed with this type of feedback device.
- A quad B with distance coded markers – this selection is only available for 9/440HR systems. This feedback device is equipped with multiple markers at progressively increasing distances along its length. This allows the CNC to identify absolute axis position whenever two consecutive markers are passed.

These linear feedback types can be used to close the position loop on both digital and analog servos. On digital systems the velocity loop must always be closed by the factory installed rotary feedback device mounted directly on the motor shaft.

Linear Feedback Device Specifications

Input frequency, load, voltage, etc... must be compliant with those specifications given for your specific 9/Series hardware as defined in your 9/Series integration manual for optional feedback devices. The following linear feedback devices have been tested and are compatible as 9/Series optional feedback:

- Sony Magnescale (model GF-45E with broad-type detector MD10-FR)
- Heidenhain (model 704 with external interpolation and digitizing model EXE 602 D/5-F)
- Heidenhain Distance Coded Marker Systems (model LS176)

The feedback from the scale should produce an A quad B signal that matches the required signature given in chapter 7 of this document. Quadrature error should be less than 27 degrees. Scales with single-ended inputs are not supported.

Configuring the Position Loop with Linear Feedback

To properly configure the position loop you must identify to the control the type of feedback device expected, and the number of feedback counts that should be expected per unit of axis travel.

The following AMP parameters have special configurations required to configure your liner scale. The remaining AMP parameters in chapter 7 may or may not apply to your servo setup depending on your specific application.

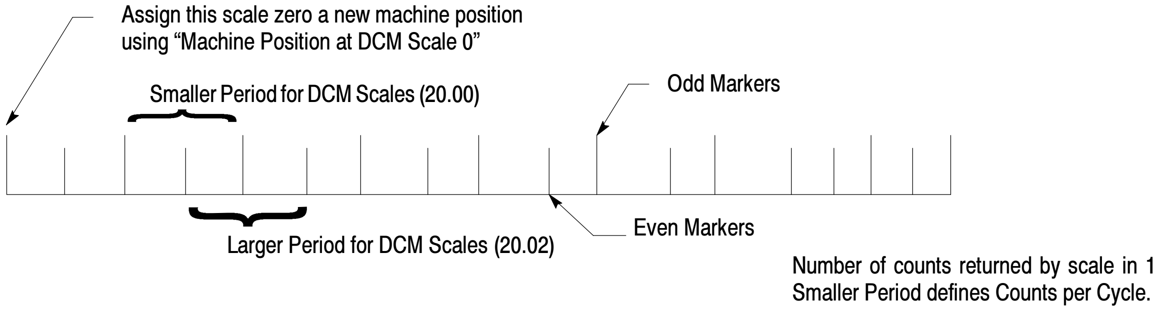
Table B.1
Position Loop Parameters

AMP Parameter	Description	Value for Heidenhain Distance-coded Marker*
Position Feedback Type	Selects the type of position feedback device used. Three types of linear scale hardware is supported. Select between A Quad B No Marker, A Quad B One Marker, or Distance Coded Marker.	Distance-coded Marker
Lead Screw Thread Pitch	**A Quad B No or One Marker**: The lead screw thread pitch defines the distance of one electrical cycle for these feedback devices (see Counts per Cycle above). This is necessary because there is no actual physical marker series to define the cycle. **Distance Coded Marker:** The entered lead screw thread pitch is not used for position loop calculations when configuring distance coded markers.	(enter actual lead screw thread pitch for velocity loop)
Position Feedback Counts/Cycle	**A Quad B No or One Marker:** The entered Lead Screw Thread Pitch defines the distance of one cycle for these feedback devices. Enter the number of feedback counts expected over the thread lead distance. For example if you entered a one inch lead screw thread pitch you would enter the number of counts expected from the scale over that one inch of travel. If the number of counts expected is greater than the allowable number of counts simply enter a smaller lead screw thread pitch. If you are forced to enter a lead screw thread pitch other than the actual screw pitch, you may need to compensate when configuring gear ratios for the velocity loop. **Distance Coded Marker:** The number of feedback counts expected for the distance equal to the entered Smaller Periodic Distance.	20,000
Smaller Period for DCM Scales	This parameter is used for scales with distance coded markers only. The distance coded marker supported by the 9/Series must have equally spaced odd and even markers. The distance between every odd marker must be the same. The distance between every even marker must be the same. For this parameter enter the smaller of these two distances.	20 mM
Larger Period for DCM Scales	This parameter is used for scales with distance coded markers only. The distance coded marker supported by the 9/Series must have equally spaced odd and even markers. The distance between every odd marker must be the same. The distance between every even marker must be the same. For this parameter enter the larger of these two distances.	20.02 mM
Machine Pos. at DCM Scale 0	This parameter is used for scales with distance coded markers only. This parameter is similar to the homing parameter “Axis Position After Homing”. The value entered here is used to redefine the control’s interpretation of the actual zero marker on the scale. Use this parameter to shift the machine coordinate system on your scale. The value entered here becomes the actual value of the zero marker.	0
Teeth on Gear for Pos. FB	This parameter should be set to one. Entering something other than one will change the final Counts per Cycle calculation based on the ratio between this parameter and “Teeth on Lead Screw for Pos. FB”.	1

AMP Parameter	Description	Value for Heidenhain Distance-coded Marker*
Teeth on Lead Screw for Pos. FB	This parameter should be set to one. Entering something other than one will change the final Counts per Cycle calculation based on the ratio between this parameter and "Teeth on Gear for Pos. FB".	1

* Assumes Heidenhain model LS176 with:
Grating period = 20μM (i.e., 20,000 cts/cycle)
Small period = 20mM
Large period = 20.02mM

Figure B.3
Configuring a Scale with Distance Coded Markers



Configuring the Velocity Loop with Linear Feedback

To properly configure the velocity loop you must identify to the control the type of feedback device expected, and the number of feedback counts that should be expected per unit of axis travel. We strongly recommend you use motor mounted feedback to close the velocity loop and do not attempt to use a linear feedback device for this loop. Closing the velocity loop with something other than motor mounted feedback can cause undesirable results from any unmonitored backlash that may exist in the axis drive gearing.

Velocity feedback is used relative to the rotation of the lead screw. Any gearing or belts that exists between lead screw rotation and the velocity feedback device must be entered as "Teeth on Lead Screw" and "Teeth on Gear for Velocity Feedback".

Important: If you choose to use the same linear feedback device to close the velocity loop that you used to close the position loop (not recommended as discussed above) the parameters for the velocity loop must match the position loop parameters.

END OF APPENDIX

Symbols

,C/,R Word Format, 16-18

(G17, G18, G19) 1st Axis Parallel to 1, 17-4

(G17, G18, G19) 1st Axis Parallel to 2, 17-7

(G17, G18, G19) 2nd Axis Parallel to 1, 17-5

(G17, G18, G19) 2nd Axis Parallel to 2, 17-8

(G17, G18, G19) Primary Axis 1, 17-3

(G17, G18, G19) Primary Axis 2, 17-6

Numbers

1326 Motors as Spindle, 7-79

1394 Drive as Spindle, 7-79

9/PC
  Name, 2-4
  Software revision, 2-4

A

A Retrace Buffer Was Deleted, 33-11

Acc/Dec, for adaptive feed, 30-3

Acceleration and Deceleration, 10-2

Acceleration, Maximum Analog Servo, 7-66, 7-67

Access Word, for patch AMP, 37-2

Adaptive Depth Axis Name, 30-8

Adaptive Depth Dead Band, 30-10

Adaptive Depth Probe
  as feedback, 30-12
  travel limit, 30-11
  trip direction, 30-9

Adaptive Feed and Depth, 30-1

Adaptive Feedrate
  feedrate Acc/Dec enable, 30-3
  torque integral gain, 30-1
  torque proportional gain, 30-2

Add to Tool Offset for Skip 1-4, 21-10

Always Repeat Turning Cycles, 24-14

AMP Editor Utility, 2-5

AMP File
  Copying
    from another project, 2-7
    from current project, 2-7
  Creating, 2-6
  Selection, 2-6

AMP G-Codes
  Modality (Type I, Type II), 20-5
  Paramacro Calls, 20-4

Analog Output Port, 7-11

Analog Servo Neg. Voltage, 7-43

Analog Servo Pos. Voltage, 7-41

Application Selection, 2-5

Approach Distance (R), 29-4

Approach Rate (E), 29-6

Automatic Home on Virtual C Entry, 28-10

Automatic Homing (G28), Distance Coded Marker, 5-14

Automatic Homing Parameters, 5-13

Automatic Homing Sequence, 5-13

Axis
  Copying, 3-7
  Deleting, 3-6
  Selecting, 3-1

Axis Along Center Line, 28-9

Axis Calibration On/Off, 36-10

Axis calibration parameters, 36-5

Axis Jerk, 10-19

Axis Name, adaptive feed probe, 30-8

Axis Names, Specifying, 3-3

Axis Parameters, 36-2
  Overview, 4-1
  Rollover Value, 4-6

Axis Position After Homing, 5-11

Axis Program
  Error if Letter/Numeric Missing, 15-4
  Error Modes, 15-2, 15-3

Format Parameters, 15-1
Word Format, Axis 1, 15-1
Zero Suppression, 15-2
Leading Zero, 15-3
Trailing Zero, 15-3

Axis Types, Specifying, 3-4

B

Backing up your AMP, 37-15

Backup EEPROM, 36-14

Block Delete Type, 33-6

Block Retrace Limit, 33-10

Boring, Shift in Q-word, 24-3

Boring Cycle, Shift Direction, 24-4

Break Point, Gain, (Analog), 7-22

C

Calibration on/off, axis, 36-10

Calibration, axis parameters, 36-5

Cancel Tool Offsets on M02/M30, 21-8

Chamfering/Corner-R Word Letter Formats, 16-18

Character of The Password, 33-12

Circular Error Tolerance Limit, 9-9

Closed Loop, Analog, 7-15

CNC-OCI Data Tranfer Rate, 34-2

Computer Name, 2-4, 2-16, 2-25

9/440, as spindle, 7-79

Configure, project, 2-3

Control Operating Parameters, General, 33-1

Control Reset on E-Stop Reset, 33-3

Control Type Selection, 3-9

Controlling Spindle, 14-2

Copying, Axis, 3-7

Copying AMP File
From another project, 2-7
From current project, 2-7

Corner Override, 22-7

Creating
Document file, 2-25
New AMP File, 2-6

CSS Parameters, 11-2

CSS Radius During G00 Rapid, 11-4

Current as a % of RMS, Peak, (Digital), 7-60

Cutter Compensation
Basic Setup, 22-1
Corner Override
Angle, 22-8
Distance From Corner (DFC), 22-8, 22-9
Distance To Corner (DTC), 22-8
Percentage (%), 22-10
Defining Offsets, (Mill Only), 22-3
Error Detection, 22-10
Generated Blocks, 22-4
Interference Detection, 22-11
M Word Interference Detection
Disable, 22-12
Enable, 22-13
Maximum Number of Non-Motion Blocks, 22-6
Minimum Feed Reduction %, 22-5
Overview, 22-1
Reverse Compensated Motion Detection, 22-11
Type A, Type B, 22-1

Cycle Clearance Amount, 24-5

Cycle Time for SERCOS Board 1, 7-4

Cycles
Clerance Amount, 24-5
Fine Boring Shift Direction, 24-4
Fine Boring Shift in Q-word, 24-3
Fixed Drilling Axis, 24-8
Ignore Dwell in Tapping Cycles, 24-6
Parameters, 24-2
Rapid to Drilling Hole, 24-7
Retract Amount for Peck Drilling, 24-2
Threading Pullout Distance, 24-9

Cylindrical Feed Axis Name, 28-2

Cylindrical Interpolation
cylindrical feed axis name, 28-2
cylindrical linear axis name, 28-3
cylindrical park axis name, 28-2
cylindrical rotary axis name, 28-4
feed axis park location, 28-10
rotary center feed coordinate, 28-5

rotary center park coordinate, 28-5
Cylindrical Linear Axis Name, 28-3
Cylindrical Park Axis Name, 28-2
Cylindrical Rotary Axis Name, 28-4

D

D: Integer Format, 16-3, 30-12
D: Word Format, 16-4
D-Integers and D-Words, 16-3
DAC Voltage and Spindle Gear Parameters, 12-2, 13-3
Dead Band, adaptive depth, 30-10
Default CSS Axis Name, 11-2
Default Orient Angle, 12-14, 13-14
Default Orient Direction, 12-13, 13-13
Default Position Offset, 14-6
Deleting, Axis, 3-6
Deleting an AMP file, 2-8
Depth Probe, 30-1
Depth Sensor Dead Band, 30-10
Depth Sensor Travel Limit, 30-11
Deskew Gain, 32-9
Deskew Master Servo Name, 32-4
Deskew Parameters for Split Axes, 32-1
Deskew Slave Servo Name, 32-7
Diameter Axis Name (lathes), 4-5
Digital Servo Parameters, 7-70
  Servo Feedback, 7-2
Digital Spindle Drive, 7-79
Direction of Probe Trip, 30-9
Direction to Move Off Limit Switch, 5-9
Distance Coded Marker
  Larger Period for DCM Scales, 7-45
  Machine Position at DCM Scale 0, 5-18
  Smaller Period for DCM Scales, 7-44
Distance Coded Markers
  Automatic Homing (G28), 5-14
  Homing Linear Scales, 5-5
  Manual Homing, 5-5
Document File
  Creating, 2-25
  Displaying, 2-27
  Printing, 2-27
Download, 2-3
  AMP files, 2-14
  Local host PC, 2-14
  network , 2-16
  RAS software, 2-16
  Serial port, 2-16
Drilling, Retract Amount for Peck Cycle, 24-2
Drilling Axis, Fixed, 24-8
Drilling, Cycle Clerance Amount, 24-5
Drilling, Rapid to Hole, 24-7
Drive Signal (Analog Module)
  Negative Voltage, 7-43
  Positive Voltage, 7-41
Dual Axis Group, 31-3
Dual Axis Parameters, 31-1
  Dual Axis Slave, 31-3
Dual Processing, with Deskew Axes, 32-5
Dwell Type, 33-4
Dynamic Friction Compensation, 7-84

E

E: # THRDS/INCH Word Format, 16-5
E-Words, 16-5
E-STOP reset on a Split Axis, 32-3
Editor Utility, 2-5
End Face Axis Integrand Name, 28-8
End Face Milling Axis, 28-7
End Face Milling Incremental Axis, 28-8
Error if Letter Numeric Missing, 15-4
Excess Skew Limit, 32-10
Exiting the AMP Editor, 2-13
External Decel Speed (Cutting), 9-6
External Hardware Tach Loop (Analog), 7-10

F

F: IPM/MMPM Word Format, 16-6

F: IPR/MMPR Word Format, 16-7

F: V/D Word Format, 16-7

F-Words, 16-6

F1 Rapid Override Percent, 9-5

Feed Axis Park Location, 28-10

Feed Integral Torque Gain, 30-1

Feed Proportional Torque Gain, 30-2

Feedback
- direction for adaptive depth probe, 30-9
- from adaptive depth probe, 30-12

Feedback Parameters, Servo
- Position Loop Feedback Port, (Analog), 7-29
- Sign of Position Feedback, (Analog), 7-36
- Velocity Loop Feedback Port, (Digital), 7-46

Feedrate Acc/Dec Enable, 30-3

Feedrate Override Parameters, 9-4

Feedrate Suppression Point, (Analog), 7-21

Feedrate, Clamp, 9-2

Feedrate, Rapid, 9-1

Feedrates, Adaptive Feed, 30-1

Filter, torque filter, 7-69

Fine Boring Shift Direction, 24-4

Fine Boring Shift in Q Word, 24-3

Finish Allow in Mult Threading, 24-13

Fixed Drilling Axis, 24-8

Flash Memory for AMP, 37-15

Follower Orientation, 14-4

Follower Spindle, 14-2

Fourth Jog Increment, 8-9

Fourth Jog Speed, 8-5

Friction Compensation
- Negative Percentage, 7-89
- Positive Percentage, 7-88
- Threshold, 7-85

Friction Parameters, 7-84
- Negative Friction Compensation Percent, 7-89
- Positive Friction Compensation Percent, 7-88
- Stiction Compensation Torque Percent, 7-87
- Threshold for Friction Compensation, 7-85

G

G-codes, G28, 5-14

G-Code for Macro Call, to #9010 to #9019 (Type I), 20-5

G28 Direction to Home, 5-15

G28 Home Speed, 5-16

G30 Secondary Home Position, 5-17

G31, G37 Probing Parameters, 29-1

G38, G38.1 Probing Cycle Parameters, 29-3

Gain
- integral torque, 30-1
- proportional torque, 30-2

Gain Break Parameters (analog), 7-23

Gain Break Point, (Analog), 7-22

Gain Break Ratio, Position Loop, (Analog), 7-27

Gain for spindle 1, 12-15

Gain for Spindle 2, 13-15

Gain in inverse milliseconds (analog), 7-26

Gain of Position Loop, Initial, (Analog), 7-25

Gain parameters, online, 36-11

Gear Ratios
- Position Feedback, (Analog Servo), 7-38, 7-40
- Velocity Feedback, (Analog Servo), 7-53

General Servo Parameters, 7-6

H

H: Integer Format, 16-8

H: Word Format, 16-9

H-Integers and H-Words, 16-8

Help, 2-10

Hemisphere, Cycle Clerance Amount, 24-5

High Speed Input Trigger Point, 33-4

Highest Jog Speed, 8-5

Home Calibration, 5-10

Home Calibration Parameters, 36-4

Home Speed From Limit Switch, 5-12

Homing
  absolute feedback, 5-8
  using distance coded markers, 5-5

homing, switchless, 5-8

Homing a Split Axis, 32-2

Homing Parameters
  Automatic, 5-13
  Distance Coded Marker, 5-5
  Manual Homing Sequence, 5-2
  Mechanical Considerations, 5-4
  Overview, 5-1

I

Ignore Dwell in Tapping Cycles, 24-6

Incremental Axis Name, 4-5

Inertia, as used for acceleration (analog), 7-68

Initial Gain of Position Loop, (Analog), 7-25

Initigral gain of position loop, online, 36-11

Inposition Band, (Analog), 7-24

Integral Torque Gain, 30-1

Integrand, for parallel axes, 4-4

Integrand Name for Axis, 4-3

Integrator Discharge Rate, Velocity, (Analog), 7-58

Interrupt 0 - 3 Routine Call, 25-3

Interrupt Disable M Code, 25-2

Interrupt Enable M Code, 25-2

Interrupt O Service Action, 25-5

Interrupt Program Parameters, 25-2

Irregular Pockets, Cycle Clearance Amount, 24-5

J

Jog Increments, 8-6

Jog Retract, 8-11

Jog Retract Velocity, 8-12

Jog Speeds, 8-2

L

L: Word Format, 23-2

Larger Period for DCM Scales, 7-45

Largest Jog Increments, 8-10

Lathe Type, 33-2

Lead: Word Format, 16-10

Limit 2 Min Value, 6-8

Limit Switch, 5-9

Limits of Travel, for adaptive depth probe, 30-11

Linear Acc/DEC Acceleration, 10-8

Linear Acc/Dec Deceleration, 10-10

Linear Feedback
  Configuring position loop with, B-2
  Configuring velocity loop with, B-3

Linear Feedback Device, Specifications, B-1

Linear Scales, 7-33

Local Host, 2-4

Lowest Jog Speed, 8-2

M

M-Code for Call, #9001 to #9009 or -1, 20-11

Macro Call
  1 to 15 (Type II), Program Numbers for, 20-10

to #9010 to 9019 (Type I)
G-Codes For, 20-5
Type of, 20-6

Manual Delay Constant, 10-14

Manual Homing, Distance Coded Markers, 5-5

Max Jogs In Retracts, 8-13

Maximum % Rated Torque (+), (Analog), 7-62

Maximum % Rated Torque (-), (Analog), 7-61

Maximum +/- Geometry Offset, 21-19

Maximum +/- Geometry Radius, 21-21

Maximum +/- Radius Offset, 21-22

Maximum +/- Wear Offset, 21-18

Maximum Deviation, 14-7

Maximum Geometry Offset Change, 21-17

Maximum Motor Speed, (Digital), 7-74

Maximum Number of OCI Connections, 34-1

Maximum Radius Offset Change, 21-21

Maximum Servo Acceleration, (Analog), 7-67

Maximum Tool Offset Number, 21-4

Maximum Wear Offset Change, 21-16

Mill Type, transfer line, 33-2

Milling Fixed Cycle Parameters, 24-2

Min Infeed in Multi Threading, 24-12

Minimum Block Generated Length, 22-4

Minimum Programmable Jerk, 10-21

Monitor Frequency for Changes, 34-2

Motion Feedrate Parameters, 9-1

Motor Rated Current, (Digital), 7-75

Motor Table Values, Standard, (Digital), 7-6

Motor, Number of poles on, (Digital), 7-73

N

Name, adaptive depth axis, 30-8

Name Axis, 4-2

Negative Friction Compensation Percent, 7-89

Negative Software Overtravel, 6-11

No of Teeth on Motor Gear, (Digital Spindle), 7-82

No of Teeth on Spindle Shaft, (Digital Spindle), 7-83

Number of Gears Used, 12-7, 13-7

O

OCI Backgrnd Data Buffer __Size, 34-5

OCI Foregrnd Data Buffer Size, 34-5

OCI Parameters
CNC-OCI Data Transfer Rate, 34-2
Maximum Number of OCI Connections, 34-1
Monitor Frequency for Changes, 34-2
OCI Backgrnd Data Buffer_Size, 34-5
OCI Foregrnd Data Buffer Size, 34-5
Watchlist Buffer Size, 34-3

Offset Percentage, Torque, (Analog), 7-63

Offsets, reset G92 and set zero, 33-13

Online AMP, 2-3

Online Axis Calibration Parameters, 36-5

Online Servo Parameters, 36-11

Operating System, 2-4

Orient Inposition Band, 12-14, 13-14

Orient Speed, 12-12, 13-13

Output Port Number, (Analog), 7-11

Overtravel Parameters, Software, 6-10

Overview of AMP, 1-1
Related Publications, 1-4

P

P dwell type : integer/decimal, 16-11

P: Integer Format, 16-11

P-Words, 16-11

P1-P9 Constant Surface Speed Axis Name, 11-3

Parallel Axes, for plane definition, 17-4, 17-5, 17-7, 17-8

Parallel Axis Integrands, 4-4

Paramacro, Overview, 20-1

Paramacro Calls
- G-Code, 20-4
- M-Code, 20-11
- T, S, and B Codes, 20-3
  - Determination, 20-3

Parameter Group, Selecting, 2-9

Password, for patch AMP, 37-2

Password Parameters, 33-12

Patch AMP, 2-3, 37-1
- search functions, 37-2
- values, 37-3

Patch AMP, Accessing, 37-1

Peak Current as a % of RMS, (Digital), 7-60

Peck Cycle, Retract Amount for, 24-2

Percent of cutting depth, 26-2

Perform a rough-finishing cut, 26-3

Plane Definition, parallel axes, 17-4, 17-5, 17-7, 17-8

Pocket, Cycle Clerance Amount, 24-5

Position Loop, Configuring with linear feedback, B-2

Position Loop Feedback Port, (Analog), 7-29

Position Loop Gain Break Ratio, (Analog), 7-27

Position Loop Parameters, 7-13

Position Tolerance For Skip 1 - 4, 21-9

Positioning ACC/DEC Mode, 10-3

Positive Friction Compensation, 7-88

Posts, Cycle Clerance Amount, 24-5

Power Turn On G Code Parameters, 18-2

Probe Length Compensation, 29-1

Probe Radius Compensation, 29-2

Probe Rate (F), 29-6

Probe Transition, 29-2

Probe Trip Direction, 30-9

Probing
- Adaptive Depth Probe, 30-10
- direction of adaptive depth probe trip, 30-9
- using adaptive depth feedback, 30-12

Probing Parameters, 29-1
- Approach Distance (R), 29-4
- Approach Rate (E), 29-6
- Hole and Part Rotation Parameters, 29-3
- Probe Length Compensation, 29-1
- Probe Radius Compensation, 29-2
- Probe Rate (F), 29-6
- Probe Transition, 29-2
- Tolerance Band Distance (D), 29-5

Program Numbers for Macro Calls, 1 to 15 (Type II), 20-10

Programmable Zone Group Axis, 6-4

Programmable Zones, 6-1

Programmed Delay Constant, 10-17, 10-18

Project Information, 2-5

Proportional Torque Gain, 30-2

PTO ACC/DEC Mode, 18-11

PTO G Code For Modal Group 1, 18-3

PTO G Code For Modal Group 5, 18-2

PTO G Code For Modal Group 6, 18-3

PTO G-Code Modal Group 18 (Lathe only), 18-6

PTO G-Code Modal Group 10, 18-8

PTO G-Code Modal Group 13, 18-8

PTO G-Code Modal Group 20, 18-9

PTO G-Code Modal Group 22, 18-10

PTO G-Code Modal Group 24, 18-11

PTO G-Code Modal Group 25, 18-12

PTO G-Code Modal Group 4 , 18-7

PTO G-Code Modal Group 8 (Mill only), 18-5

PTO Plane Select G-Code (Mill only), 18-4

PTO Synch Mode, 18-12

PTO Work Coordinate, 18-7

Pullout Angle, Chamfered Thrd, 24-10

Pullout Distance, Chamfered Thread, 24-9

Q

Q: Integer Format, 16-12

Q: Thread Marker Angle Shift, 16-13

Q: Word Format, 16-13

QPP Angle Word, 23-1

Qpp Angle Word Format, 23-2

Quick Edit, Using, 2-11

QuickPath Plus Parameters, 23-1

R

R: Angle Word Format, 16-15

R: Word Format, 16-14

R-Words, 16-14

Radius Offset, Maximum +/-, 21-22

Radius Offset, Maximum Change, 21-21

Rapid Feedrate For Positioning, 9-1

Rapid Override in Dry Run, 9-4

Rapid to Drilling Hole, 24-7

RAS, 2-17

Recover Backup File, 2-12

Remote Access Services, 2-17

Renaming an AMP File, 2-8

Reserved Custom Parameters, 35-1

Reset Coord offset on M02/M30, 33-8

Reset G92 Offsets, 33-13

Reset M&G Codes On M02/M30, 33-7

Retrace Buffer Deleted, 33-11

Retract Amount for Peck Drilling, 24-2

Retract Amount in Grooving, 24-15

Reversal Error Compensation, (Analog), 7-18

Online Reversal Error Parameters, 36-2

Rollover Value, 4-6

Rotary Axis, Rollover Point, 4-6

Rotary Center Feed Coordinate, 28-5

Rotary Center Park Coordinate, 28-5

Roughing Cycle Parameters, 26-1

Roughing cycle threshold depth, 26-4

Roughing Cycles Retract Amount, 26-1

S

S: CSS Word Format, 16-15

S: Orient Angle Word Format, 16-16

S: Spindle RPM Word Format, 16-17

S-Curve Acc/Dec Deceleration, 10-13

Save As, 2-11

Saving AMP Files, 2-11

Second Jog Increments, 8-8

Second Jog Speed, 8-3

Secondary Auxiliary Function Word, 33-6

Seek Timeout, 14-8

Seek Tolerance, 14-7

Selecting a Parameter Group, 2-9

SERCOS
- Baud Rate for SERCOS Board 1, 7-3
- Cycle Time for SERCOS Board 1, 7-4
- SERCOS Transmit Level, 7-5
- Transmit Level, 7-5

SERCOS Parameters, 7-3

SERCOS Transmit Level, 7-5

Servo Acceleration, Maximum, (Analog), 7-67

Servo Amplifier Type, (Digital), 7-77

Servo Detached, (Analog), 7-17

Servo Feedback, 7-2

Servo Off, (Analog), 7-16

Servo Parameters, 37-11, 37-12, 37-13, 37-14, 37-15
- Digital, 7-70
- Friction, 7-84
- General, 7-6
- online, 36-11
- Position Loop, 7-13
- Spindle, 7-79

Servo Position Loop Type, (Analog), 7-14

Shift Direction, for fine boring, 24-4

Shift, Q word in Fine Boring, 24-3

Sign of Position Feedback, 7-36

Sign of Velocity Feedback, (Digital), 7-50

Skip Cycle Feedrate Parameters, 9-7

Skip Feedrate For G31/G37, 9-8

Smaller Period for DCM Scales, 7-44

Smallest Jog Increment, 8-7

Software Overtravel Parameters, 6-10

Software Overtravel Used, 6-10

Special Software Parameters, 35-1

Specifications, Linear Feedback Device, B-1

Specifying
- Axis Names, 3-3
- Axis Types, 3-4

Spindle 1 Gain, 12-15

Spindle 2 Gain, 13-15

Spindle DAC Output Ramping, 12-3, 13-3

Spindle Deviation Tolerance, 12-5, 13-6

Spindle Marker Calibration, 12-11, 13-11

Spindle Orienting Parameters, 12-10, 13-10

Spindle Parameters, 7-79, 36-12

Spindle Type for Axis, analog, 7-82

Spindles
- Configuring more than one, 3-4, 3-5
- No of Teeth On Motor Gear, (Digital Spindle), 7-82
- No of Teeth On Spindle Shaft, (Digital Spindle), 7-83
- number, 7-82
- Specifying Type, 3-4
- using a 1326 motor, 7-79

Split Axis Parameters, 32-1
- Deskew Gain, 32-9
- Deskew Master Servo Name, 32-4
- Deskew Slave Servo Name, 32-7
- Excess Skew Limit, 32-10
- Parameters copied from master to slave, 32-5

Stabalizing servos, online parameters, 36-11

Standard Motor Table Values, (Digital), 7-6

Static Friction Compensation, 7-84

Stiction Compensation, Torque Percent, 7-87

Storing AMP, 37-15

Suppression Point, Feedrate, (Analog), 7-21

switchless homing, 5-8

Synch Gain, 14-5

Synchronized Spindle Parameters
- Controlling Spindle, 14-2
- Default Position Offset, 14-6
- Follower Orientation, 14-4
- Follower Spindle, 14-2
- Maximum Deviation, 14-7
- Seek Timeout, 14-8
- Seek Tolerance, 14-7
- Synch Gain, 14-5

System Timing, 33-13
- analyzing , 33-18
- Overview, 33-14
- Setting Parameters, 33-20

T

T: Tool Number Integer Format, 16-17

T-word Programming Method, 21-15

Tapping Cycle, Ignore Dwell, 24-6

Target Offset For Skip 01 - 04, 21-11

Teeth on gear for pos FB, 7-38

Teeth on Lead Screw for pos FB, (Analog), 7-40

Teeth on Lead Screw for vel FB, (Analog), 7-53

Terms and Conventions, 1-2

Third Jog Increment, 8-8

Third Jog Speed, 8-4

Threading, pullout distance, 24-9

Threading Cycle Parameters, 24-9

Threading E Word Definition, 33-5

Threshold for Friction Compensation, 7-85

Timing, System, see System Timing, 33-13

Tolerance Band Distance (D), 29-5

Tool Gauging Cycle Parameters, G37, 21-9

Tool Geometry Mode, 21-4

Tool Length Axis, 21-3

Tool Offset Cancel, 21-8

Tool Offset Range Verification Parameters, 21-16

Tool Offset Setup Parameters, 21-2

Tool Table Motion, 21-13

Tool Tip Radius Compensation (TTRC), Generated Blocks, 22-4

Tool Wear Mode, 21-6

Torque, torque filter cutoff frequency, 7-69

Torque (+), Maximum % Rated, (Analog), 7-62

Torque control, using Acc/Dec with, 30-3

Torque Control (G25), Adaptive Feed, 30-1

Torque Filter Cutoff Frequency, analog servo, 7-69

Torque Offset Percentage, (Analog), 7-63

Trailing Zero Suppression Mode, 15-3

Transfer Line Selection, mill type, 33-2

Travel Limit, adaptive depth probe, 30-11

Traverse Jog Speed, 8-14

Trigger Method for Interrupt 0 - 3, 25-3

Turning Cycle Parameters, 24-14

Type of Macro Call, to #9010 to #9019 (Type I), 20-6

U

Units, Selecting, 3-8

Upload
- Host PC, 2-23
- Network, 2-24
- RAS, 2-25
- Serial port, 2-25

Uploading AMP Files, 2-23

Use AMP Skip Feedrate, 9-8

Use Probe Feedback for Servos, 30-12

V

Values, for patch AMP, 37-3

Ve Integrator Discharge Rate, (Analog), 7-58

Velocity Error Integrator Discharge Rate, (Analog), 7-58

Velocity Feedback Type, (Digital), 7-47

Velocity Loop, Configuring with linear feedback, B-3

Velocity Loop (Analog), Closed by Amplifier, 7-10

Velocity Loop Feedback Port, (Digital), 7-46

Velocity Proportional Gain
- (Digital), 7-54
- online, 36-11

Velocity Step For ACC/DEC, 10-7

Virtual C Parameters
- automatic home on virtual C entry, 28-10
- axis along center line, 28-9
- end face axis integrand name, 28-8
- end face milling axis, 28-7
- end face milling incremental axis, 28-8
- feed axis park location, 28-10
- virtual C rotary axis, 28-6

Virtual C Rotary Axis, 28-6

Voltage at Max for Gears 1-8, 12-5, 13-5

W

Watchlist Buffer Size, 34-3

Z

Zero Suppression
- Error Modes, 15-2
- Leading Zero, 15-3
- Trailing Zero, 15-3

ZFE Closed Loop, Analog, 7-15

Zones, Programmable, 6-3

Rockwell Automation

Allen-Bradley

Allen-Bradley, a Rockwell Automation Business, has been helping its customers improve productivity and quality for more than 90 years. We design, manufacture and support a broad range of automation products worldwide. They include logic processors, power and motion control devices, operator interfaces, sensors and a variety of software. Rockwell is one of the world's leading technology companies.

Worldwide representation.

Argentina • Australia • Austria • Bahrain • Belgium • Brazil • Bulgaria • Canada • Chile • China, PRC • Colombia • Costa Rica • Croatia • Cyprus • Czech Republic • Denmark • Ecuador • Egypt • El Salvador • Finland • France • Germany • Greece • Guatemala • Honduras • Hong Kong • Hungary • Iceland • India • Indonesia • Ireland • Israel • Italy • Jamaica • Japan • Jordan • Korea • Kuwait • Lebanon • Malaysia • Mexico • Netherlands • New Zealand • Norway • Pakistan • Peru • Philippines • Poland • Portugal • Puerto Rico • Qatar • Romania • Russia-CIS • Saudi Arabia • Singapore • Slovakia • Slovenia • South Africa, Republic • Spain • Sweden • Switzerland • Taiwan • Thailand • Turkey • United Arab Emirates • United Kingdom • United States • Uruguay • Venezuela • Yugoslavia

Allen-Bradley Headquarters, 1201 South Second Street, Milwaukee, WI 53204 USA, Tel: (1) 414 382-2000 Fax: (1) 414 382-4444

Publication 8520-9.3 - April 2000

PN 176828